Bricsys®

# BricsCAD V21

## Документация по продукту



Bricsys®

# Содержание


1. Что нового 31
1.1 Overview 31
1.1.1 Release notes 31
1.1.2 Diagnostics and Usage Data Collection 31
1.2 User interface 32
1.2.1 BricsCAD Launcher 32
1.2.2 Commands 32
1.2.3 Ribbon 33
1.2.4 Toolbars 33
1.2.5 Menus 33
1.2.6 Layouts 33
1.2.7 Entity Snaps 34
1.3 Productivity 35
1.3.1 Move with guide curves 35
1.3.2 Copy in a linear array 35
1.3.3 Undo per entity 36
1.3.4 Manipulator 36
1.3.5 Nearest Distance 36
1.3.6 Layers 37
1.3.7 Linetypes 39
1.3.8 Parameters & Constraints 39
1.3.9 Blocks 42
1.3.10 Drawing Optimizations 46
1.3.11 Object linking and embedding 48
1.3.12 Images 48
1.3.13 Hyperlinks 48
1.4 Drafting 49
1.4.1 Combine Text to Mtext. 50
1.4.2 Arc Aligned Text 50
1.4.3 Find and Replace Text 50
1.4.4 Field 50
1.5 Modeling 51
1.5.1 Dynamic UCS 51
1.5.2 Direct modeling 51
1.5.3 Broken views 53
1.5.4 3D Entity snaps 53
1.5.5 3D Polyline 53
1.5.6 Parametric blockify 54
1.5.7 3D display 54
1.5.8 Solids and meshes 54
1.5.9 Associative data 56
1.5.10 View Twist 56
1.5.11 Animation Editor 56
1.5.12 Point clouds 56
1.6 Civil tools 60
1.6.1 Import/Export 60
1.6.2 Civil Entity Properties 60
1.6.3 Civil settings 61
1.6.4 TIN Surfaces 61

# Содержание


1.6.5 Alignments 63
1.6.6 Corridors 64
1.6.7 Gradings 65
1.6.8 TIN Volume Surfaces 65
1.6.9 API Samples 65
1.7 Performance and compatibility 65
1.7.1 Selection Preview 65
1.7.2 Xref Loading 65
1.7.3 Multiuser Version Control (BETA) 66
1.7.4 DGN Export 67
1.7.5 3D DWF Output 67
1.7.6 User Profile Manager 68
1.7.7 Start Menu Shortcuts 68
2. Установка и лицензирование 69
2.1 Установка **BricsCAD®** 69
2.1.1 Минимальные системные требования и рекомендации 69
2.1.2 Загрузка **BricsCAD®** 72
2.1.3 Обзор процедуры установки 72
2.1.4 Ознакомительный режим 74
2.2 Активация и лицензирование **BricsCAD®** 75
2.2.1 Обзор процедуры лицензирования 75
2.2.2 BricsCAD® and AutoCAD® feature comparison 77
2.2.3 Ознакомительная версия: Активация через Интернет 102
2.2.4 Ознакомительная версия: Ручная активация 102
2.2.5 Однопользовательская или корпоративная лицензия: активация через Интернет 104
2.2.6 Одиночная или корпоративная лицензия: автономная (ручная) активация 105
2.2.7 Использование сетевой лицензии на компьютере пользователя 107
2.3 Настройка сервера сетевых лицензий 108
2.3.1 Сервер Сетевой лицензии 108
2.3.2 Краткое руководство по использованию сетевой лицензии 108
2.3.3 Установка сервера сетевых лицензий для Windows 109
2.3.4 Установка сервера сетевых лицензий для Linux 110
2.3.5 Сетевая лицензия: активация через Интернет 111
2.3.6 Сетевая лицензия: ручная активация (без подключения к Интернет) 111
2.3.7 Администрирование сервера сетевой лицензии 112
2.3.8 Дополнительные настройки конфигурации 114
2.4 Управление лицензиями 116
2.4.1 Изменение лицензии **BricsCAD®** 116
2.4.2 Деактивация или аннулирование лицензии **BricsCAD®** 116
2.4.3 Перемещение лицензии на другой компьютер 119
2.4.4 Перенос сетевой лицензии на другой сервер 119
2.4.5 Пул лицензий 119
2.4.6 Отчет об использовании сетевой лицензии 123
2.5 Использование сетевой лицензии 124
2.5.1 Перемещение лицензии (роуминг) 124
2.5.2 Настройки прокси на клиентском компьютере 126
3. Начало работы с BricsCAD® 128
3.1 Запуск **BricsCAD®** 128
3.1.1 Окно запуска BricsCAD 128

# Содержание


3.2 Интерфейс 129
3.2.1 Закладка Начало работы 129
3.2.2 Закладки чертежей 129
3.2.3 Выбор рабочего пространства 130
3.2.4 Графическая область 130
3.2.5 Командная строка 132
3.2.6 Меню 132
3.2.7 Панели инструментов 132
3.2.8 Лента 133
3.2.9 Основы работы с квадро-меню 133
3.2.10 Контекстное меню правой кнопки мыши 135
3.2.11 Строка состояния 135
3.2.12 Панели 136
3.2.13 Полноэкранный режим 136
3.2.14 Виджет 3D видов 137
3.3 Работа в **BricsCAD®** 137
3.3.1 Создание чертежа 137
3.3.2 Template drawings 137
3.3.3 Параметры создания объектов 138
3.3.4 Обеспечение точности построений 138
3.3.5 Executing commands 139
3.3.6 View control 140
3.3.7 Выбор объектов 141
3.3.8 Using the Manipulator 142
3.3.9 Панель Структура 143
4. Пользовательский интерфейс 145
4.1 The BricsCAD application window 145
4.1.1 The color theme 145
4.1.2 Menu bar 146
4.1.3 The ribbon 146
4.1.4 Document tabs 146
4.1.5 Model and layout tabs 146
4.1.6 Scroll bars 146
4.1.7 The Command line 146
4.1.8 The status bar 146
4.1.9 Hiding a selection of user interface elements 147
4.2 The Command line 147
4.2.1 Setting the Command line state 147
4.2.2 Launching a command in the Command line 148
4.2.3 Setting the Command line properties 148
4.3 The Status bar 148
4.3.1 The Status bar fields 148
4.3.2 1. Status field 148
4.3.3 2. Field List button 149
4.3.4 Cursor coordinate values 149
4.3.5 Current Layer 149
4.3.6 Current Color 149
4.3.7 Current Linetype 149
4.3.8 Current Text Style 149

# Содержание


4.3.9 Current Dimension Style 150
4.3.10 **Current Work Space** 150
4.3.11 Snap 150
4.3.12 Grid 150
4.3.13 Ortho 150
4.3.14 Polar 150
4.3.15 Esnap 150
4.3.16 Strack 150
4.3.17 Lineweight (LWT) 151
4.3.18 Paper/Model 151
4.3.19 Annotation Scale 151
4.3.20 Autoscale 151
4.3.21 Tablet 151
4.3.22 Viewport Lock (VPLOCK) 151
4.3.23 Dynamic UCS (DUCS) 152
4.3.24 Dynamic Input (DYN) 152
4.3.25 Quad 152
4.3.26 Rollover tips (RT) 152
4.3.27 Hotkey Assistant (HKA) 152
4.3.28 Lock User Interface (LOCKUI) 153
4.3.29 GIS Coordinate System 153
4.3.30 Tray Icon Items 153
4.4 Document tabs 153
4.5 Toolbars 153
4.5.1 To set the icon size 153
4.5.2 Toolbar Flyouts 153
4.5.3 Opening a toolbar 153
4.6 The Quad cursor menu 154
4.6.1 Activating the Quad 154
4.6.2 Setting the Quad display options 154
4.6.3 Editing the Quad settings 154
4.6.4 Using the Quad 155
4.6.5 Selecting multiple entities 156
4.6.6 Using the Quad to create entities 156
4.7 Rollover tips 157
4.8 Prompt menus 157
4.8.1 Controlling the display of prompt menus 157
4.9 Keyboard shortcuts 158
4.10 Digitizing tablet 162
4.10.1 Using a digitizing tablet in BricsCAD 162
4.10.2 Downloading the tablet overlay and the CUI files 162
4.10.3 Loading the overlay CUI file 163
4.10.4 Switching the tablet ON / OFF 163
4.10.5 Configuring the tablet 163
4.10.6 Calibrating the tablet 164
4.11 Panels 165
4.11.1 Opening or closing a panel 166
4.11.2 Available panels 166
4.11.3 Setting the stack type 168

# Содержание


4.11.4 Moving a panel to the other icon tab strip 170
4.11.5 Animation editor panel 171
4.11.6 Attachments panel 173
4.11.7 BIM BCF panel 175
4.11.8 BIM compositions panel 176
4.11.9 BIM profiles panel 178
4.11.10 BIM project browser panel 180
4.11.11 BOM manager panel 183
4.11.12 Bricsys 24/7 panel 190
4.11.13 Command line panel 192
4.11.14 Content browser panel 193
4.11.15 Details panel 195
4.11.16 Drawing customizations panel 197
4.11.17 Layers panel 204
4.11.18 Library panel 210
4.11.19 Mechanical browser panel 213
4.11.20 Parameters and constraints panel 217
4.11.21 Properties panel 219
4.11.22 Render materials panel 226
4.11.23 Report panel 228
4.11.24 Sheet sets panel 230
4.11.25 Structure panel 233
4.11.26 Tips panel 235
4.11.27 Tool palettes panel 236
4.12 Dialog boxes 237
4.12.1 About BricsCAD dialog box 237
4.12.2 Add block to library dialog box 238
4.12.3 Add button item dialog box 240
4.12.4 Add menu item dialog box 241
4.12.5 Add-in manager dialog box 243
4.12.6 Add plot style table dialog box 244
4.12.7 Add scale dialog box 244
4.12.8 Allowed deviations from center axis dialog box 245
4.12.9 Annotation Object Scale dialog box 246
4.12.10 Array dialog box 247
4.12.11 Attach external reference 251
4.12.12 Attach PDF underlay dialog box 253
4.12.13 Attach raster image dialog box 256
4.12.14 Attribute editor dialog box 258
4.12.15 Auto Match dialog box 260
4.12.16 Background dialog box 263
4.12.17 BIM BCF panel - Custom login dialog box 269
4.12.18 BIM project info dialog box 272
4.12.19 BIM properties dialog box 276
4.12.20 Bimify dialog box 278
4.12.21 Blade - BricsCAD LISP advanced development environment dialog box 279
4.12.22 Block attribute manager dialog box 280
4.12.23 Block definition dialog box 281
4.12.24 Boundary dialog box 284
4.12.25 Bricsys license manager dialog box 285

# Содержание


4.12.26 Calculator dialog box 289
4.12.27 Change dictionaries dialog box 291
4.12.28 Change Icon dialog box 293
4.12.29 Choose window style dialog box 294
4.12.30 Classify As Bim Element dialog box 295
4.12.31 Column filter dialog box 306
4.12.32 Combine Duplicate Block Definitions dialog box 308
4.12.33 Communicator info dialog box 308
4.12.34 Compare two models dialog box 309
4.12.35 Component Tool Properties 312
4.12.36 Compositions dialog box 313
4.12.37 Configure Settings dialog box 316
4.12.38 Configure structure tree dialog box 317
4.12.39 Confirm deleting selected cache items 321
4.12.40 Create detail dialog box 322
4.12.41 Create new drawing dialog box 323
4.12.42 Create new sheet set dialog box 329
4.12.43 Create or edit block definition dialog box 331
4.12.44 Create Schedule dialog box 332
4.12.45 Create section/elevation dialog box 333
4.12.46 Current Plot Style dialog box 335
4.12.47 Customization groups dialog box 336
4.12.48 Customize dialog box 337
4.12.49 Customize (Tool Palettes) dialog box 345
4.12.50 Custom paper sizes dialog box 346
4.12.51 Custom properties dialog box 348
4.12.52 DataLink edit dialog box 348
4.12.53 Datalink manager dialog box 350
4.12.54 Default scale list dialog box 351
4.12.55 Define attribute dialog box 352
4.12.56 Delete duplicate entities dialog box 354
4.12.57 Drawing explorer dialog box 356
4.12.58 Drawing explorer options dialog box 362
4.12.59 Drawing properties 362
4.12.60 Drawing view presets dialog box 365
4.12.61 Edit attributes dialog box 366
4.12.62 Edit custom properties dialog box 368
4.12.63 Edit hyperlink dialog box 368
4.12.64 Edit scale dialog box 369
4.12.65 Edit scale list dialog box 370
4.12.66 Entity grouping dialog box 371
4.12.67 eTransmit dialog box 373
4.12.68 Extract attributes dialog box 377
4.12.69 Field dialog box 378
4.12.70 Filter Configurator dialog box 385
4.12.71 Find and replace dialog box 386
4.12.72 Flatshot dialog box 388
4.12.73 Format dialog box 390
4.12.74 Geographic location dialog box 391
4.12.75 Geometric tolerance dialog box 392

# Содержание


4.12.76 Hatch and gradient dialog box 394
4.12.77 Hatch Edit dialog box 399
4.12.78 Import Civil 3D dialog box 400
4.12.79 Import page setups dialog box 401
4.12.80 Import sheet set dialog box 402
4.12.81 Insert block dialog box 403
4.12.82 Insert object dialog box 405
4.12.83 Insert table dialog box 407
4.12.84 Layer Property Filter dialog box 409
4.12.85 Layer settings dialog box 411
4.12.86 Layer translator dialog box 412
4.12.87 Layout manager dialog box 413
4.12.88 Links dialog box 415
4.12.89 Load application files dialog box 416
4.12.90 Load linetypes dialog box 417
4.12.91 Load Multiline Styles dialog box 418
4.12.92 Login dialog box 418
4.12.93 Manage Customizations dialog box 419
4.12.94 Manage visibility 421
4.12.95 Map Connect dialog box 422
4.12.96 Match Properties dialog box 424
4.12.97 Motion path animation 427
4.12.98 Navigate Settings dialog box 429
4.12.99 New Layer Name dialog box 430
4.12.100 New Multileader Style dialog box 431
4.12.101 New Multiline Style dialog box 431
4.12.102 New Page Setup dialog box 432
4.12.103 New Sheet(s) dialog box 433
4.12.104 Optimize 2D lines dialog box 435
4.12.105 Page setup dialog box 436
4.12.106 Paste special dialog box 438
4.12.107 Physical materials dialog box 441
4.12.108 Plot stamp dialog box 442
4.12.109 Plot style table editor dialog box 443
4.12.110 Plotter Configuration Editor dialog box 444
4.12.111 Pre-process point cloud data file(s) dialog box 446
4.12.112 Print dialog box 446
4.12.113 Profiles dialog box 452
4.12.114 Project setup dialog box 454
4.12.115 Proxy information dialog box 455
4.12.116 Publish dialog box 456
4.12.117 QLeader Settings dialog box 460
4.12.118 Reassociate entity data dialog box 462
4.12.119 Reference Edit dialog box 463
4.12.120 Render dialog box 465
4.12.121 Run BricsCAD VBA macro dialog box 466
4.12.122 Save block definition dialog box 467
4.12.123 Save detail dialog box 468
4.12.124 Save workspace dialog box 470
4.12.125 Security dialog box 470

# Содержание


4.12.126 Security options dialog box 472
4.12.127 Select color dialog box 472
4.12.128 Select Custom Arrow Block dialog box 475
4.12.129 Select items to delete from cache dialog box 475
4.12.130 Select item to attach to this DWG 476
4.12.131 Settings dialog box 477
4.12.132 Set viewpoint dialog box 479
4.12.133 Sheet Properties dialog box 482
4.12.134 Sheet selection sets dialog box 483
4.12.135 Sheetset setup dialog box 483
4.12.136 Spatial Locations Manager dialog box 486
4.12.137 Spell checking 488
4.12.138 Standard open file dialog box 490
4.12.139 Standard save file dialog box 492
4.12.140 Tags dialog box 493
4.12.141 Tool image dialog box 494
4.12.142 Tool Properties dialog box 495
4.12.143 Upload to Bricsys 24/7 dialog box 496
4.12.144 User coordinate system dialog box 497
4.12.145 User profile manger dialog box 498
4.12.146 VBA manager dialog box 498
4.12.147 View Options dialog box 501
4.12.148 Visual settings dialog box 502
4.12.149 Wall detection options dialog box 502
4.12.150 Wizard page dialog box 504
4.12.151 Working sets dialog box 510
4.12.152 Write block to file dialog box 512
5. Управление чертежами 514
5.1 Creating a drawing 514
5.1.1 Specifying the template folder 514
5.1.2 Setting the default template drawing 514
5.1.3 Using the Start tab 514
5.1.4 Creating a drawing using the default template drawing 514
5.1.5 Creating a drawing using the New command 515
5.1.6 Setting the Plot style policy 515
5.1.7 Creating a drawing using a template drawing 515
5.1.8 Creating a drawing using a wizard 516
5.2 Working in the Drawing Explorer dialog box 516
5.2.1 Adding a drawing folder 516
5.2.2 Opening a drawing 516
5.2.3 Logging on to Bricsys 24/7 517
5.2.4 Opening a drawing from a Bricsys 24/7 project 517
5.2.5 Copying definitions between drawings 517
5.3 Drawing properties 518
5.3.1 Opening the Drawing Properties dialog 518
5.3.2 Defining default properties 518
5.3.3 Creating custom properties 518
5.4 Drawing security options 518
5.4.1 Setting the password 518

# Содержание


5.4.2 Removing the password 519
5.5 Opening and saving drawings 519
5.5.1 Using drag-and-drop to open a drawing 522
5.6 Getting drawing information 523
5.6.1 General drawing information 523
5.6.2 Information about entities 523
5.7 Comparing drawings 524
5.7.1 Comparison system variables 524
5.7.2 Executing a comparison session 524
5.7.3 Analyzing the comparison in the Structure panel 525
5.7.4 Merging the compared drawings 525
5.8 Repairing a drawing 526
5.8.1 Analyzing the integrity of the current drawing 526
5.8.2 Repairing a drawing 526
5.9 Projects 526
5.9.1 Accessing the Project settings 527
5.9.2 Creating projects 527
5.9.3 Adding search paths to a project 527
5.9.4 Changing the order of the search paths 527
5.9.5 Assigning a project to the current drawing 527
6. Точность построений 529
6.1 Orthogonal mode 529
6.1.1 Toggling Orthogonal mode 529
6.2 Unit settings 529
6.2.1 System variables 530
6.2.2 Entering angles 531
6.3 Coordinate reference system 531
6.3.1 CoordinateReferenceSystems 532
6.3.2 Datums 535
6.3.3 Ellipsoids 536
6.3.4 Transformations 537
6.3.5 ProjectionCode and Methods 538
6.4 2D constraints 538
6.4.1 Commands and toolbars 540
6.5 Using dimensional constraints 543
6.5.1 Editing dimensional constraints 543
6.5.2 Setting the visibility of dimensional constraints 544
6.5.3 Using expressions to define dimensional constraints 545
6.6 Using geometric constraints 548
6.6.1 Creating geometric constraints 548
6.6.2 Displaying constraint bars 549
6.6.3 Controlling the position of a constraint bar 550
6.6.4 Working with constraint bars 550
6.6.5 Deleting constraints 551
6.6.6 Some examples 551
6.7 Dynamic dimensions 552
6.7.1 Defining the Dynamic Dimensions Settings 552
6.7.2 Using dynamic dimensions when creating entities 555
6.7.3 Using dynamic dimensions to grip-edit entities 556

# Содержание


6.7.4 Using dynamic dimensions to measure entities 556
6.7.5 Using dynamic dimensions with tracking points 557
6.8 Nearest distance 557
6.8.1 Using Nearest Distance 558
6.9 Coordinate input 561
6.9.1 Relative coordinates 561
6.9.2 Absolute coordinates 562
6.9.3 Working with Cartesian coordinates 562
6.9.4 Working with cylindrical coordinates 562
6.9.5 Working with spherical coordinates 563
6.10 Snap and grid 564
6.10.1 Displaying the Snap and grid settings 565
6.10.2 Synchronizing snap and grid spacing 565
6.10.3 Using drawing limits 565
6.10.4 Using isometric snap 565
6.10.5 Setting the snap angle 566
6.11 Adaptive grid snap 566
6.11.1 Using adaptive grid snap with the Manipulator 566
6.12 Nudge 569
6.12.1 Using Nudge 569
6.13 2D entity snaps 569
6.13.1 Defining the 2D Entity Snap system variables 570
6.13.2 2D Entity Snap options 570
6.13.3 Entity Snap Modes 571
6.13.4 Setting the Entity snaps 573
6.13.5 Working with multiple Entity Snap modes 574
6.13.6 Snapping to the extension of two entities 574
6.13.7 Using the Middle of 2 points snap mode 575
6.13.8 Using the Parallel entity snap mode 575
6.14 3D entity snaps 575
6.14.1 Setting the 3D Entity Snap modes 576
6.15 Polar tracking 576
6.15.1 Defining the tracking settings 577
6.15.2 Locking a tracking line 577
6.15.3 Using temporary tracking points 578
6.16 Entity snap tracking 578
6.16.1 Locking a tracking line 579
6.16.2 Using Entity Snap Tracking 579
6.17 User coordinate systems 580
6.17.1 Controlling the display of the coordinate system icon 580
6.17.2 Defining a UCS 581
6.17.3 Defining and saving a User Coordinate System 582
6.17.4 Restoring a saved UCS 582
6.17.5 Restoring the WCS 582
6.18 Dynamic UCS 582
6.18.1 Toggling the dynamic UCS 583
6.18.2 Using the Dynamic UCS 583
6.19 Measuring 586
6.19.1 Measuring distances 587

# Содержание


6.19.2 Measuring lengths 588
6.19.3 Measuring areas 588
6.20 Entity snaps overview 589
6.20.1 Apparent intersection entity snap 589
6.20.2 Center entity snap 589
6.20.3 Center of a Face 3D entity snap 590
6.20.4 Объектная привязка Конечная точка 590
6.20.5 Extension entity snap 591
6.20.6 Geometric center entity snap 591
6.20.7 Insertion entity snap 591
6.20.8 Объектная привязка Пересечение 592
6.20.9 Объектная привязка 3D пересечение 592
6.20.10 Knot 3D entity snap 593
6.20.11 Midpoint entity snap 593
6.20.12 Midpoint 3D entity snap 594
6.20.13 Nearest entity snap 594
6.20.14 Nearest 3D entity snap 594
6.20.15 Node entity snap 594
6.20.16 Parallel entity snap 595
6.20.17 Perpendicular entity snap 595
6.20.18 Perpendicular on a face 3D entity snap 596
6.20.19 Quadrant entity snap 596
6.20.20 Tangent entity snap 596
6.20.21 Vertex 3D entity snap 597
7. Просмотр чертежа 598
7.1 View commands overview 598
7.2 Layer utilities 600
7.3 Hide and isolate entities 601
7.4 Redrawing and regenerating a drawing 601
7.5 Panning 602
7.5.1 Using the pan command 602
7.6 Zooming 602
7.6.1 Real-time zooming 602
7.6.2 Zooming to extents 603
7.6.3 Dynamic zooming 603
7.7 View manipulation using the mouse and keyboard 603
7.8 View rotation 605
7.8.1 Rotating a view freely 606
7.8.2 Using Preset Viewpoints 606
7.9 The LookFrom control 607
7.9.1 Context menu options 607
7.9.2 LookFrom control settings 607
7.9.3 Setting the LookFrom mode 607
7.9.4 Using the LookFrom control in Isometric mode 608
7.9.5 Using the LookFrom control in Twist mode 608
7.9.6 View transition options 609
7.10 Layer states 609
7.10.1 Saving the current layer state 609
7.10.2 Editing layer states 609

# Содержание


7.10.3 Restoring layer states 610
7.10.4 Copying layer states between open drawings 610
7.10.5 Exporting layer states 610
7.10.6 Importing layer states 610
7.11 Named views 611
7.11.1 Exploring views 611
7.11.2 Saving a view 611
7.11.3 Restoring a view 611
7.11.4 Editing the view background 611
7.12 Visual styles 612
7.12.1 Setting the current visual style 612
7.13 Model space and paper space 613
7.13.1 Switching between model space and paper space 613
7.13.2 Using the layout tabs 614
7.14 Model space viewports 614
7.14.1 Working with multiple viewports 614
7.14.2 Joining adjacent viewports 614
7.15 Paper space viewports 615
7.15.1 Displaying the paper sheet and the printable area 615
7.16 Creating paper space viewports 616
7.16.1 Creating an array of layout viewports 616
7.16.2 Defining the viewport properties 616
7.17 Editing paper space viewports 617
7.17.1 Setting the layer properties in a paper space viewport 617
7.17.2 Clipping a viewport 618
7.17.3 Removing the clipping boundary of a viewport 618
7.17.4 Rotating the display in a paper space viewport 618
7.17.5 Aligning viewports 618
7.17.6 Aligning model space and paper space points 619
7.18 Layouts 620
7.18.1 Creating a new layout 620
7.18.2 Copying layouts 620
7.18.3 Arranging the layout tabs 621
7.18.4 Importing layouts 621
7.18.5 Renaming a layout 621
7.18.6 Deleting a layout 621
7.18.7 Using the Layout Manager 622
7.19 Generated drawing views 622
7.19.1 Standard drawing views 623
7.19.2 Generating section views 625
7.19.3 Special views 626
7.19.4 Updating drawing views 626
7.19.5 Exporting views to model space 626
7.19.6 Customizing drawing views 627
8. 2D черчение 628
8.1 Создание объектов 628
8.1.1 Основные операции при создании новых объектов 628
8.2 Параметры создания объектов 628
8.3 Работа со слоями 629

# Содержание


8.3.1 Текущий слой 629
8.3.2 Состояния слоев 630
8.4 Entity linetype 630
8.4.1 Setting the current linetype 631
8.4.2 Adding a new linetype 632
8.4.3 Setting the entity linetype scale 632
8.5 Цвет объекта 632
8.5.1 Индексные цвета 632
8.5.2 Реальные цвета 633
8.5.3 Установка текущего цвета объекта 633
8.6 Толщина линии 634
8.6.1 Системные переменные 634
8.6.2 Толщина линий при печати 635
8.7 Fill mode 635
8.7.1 Setting Fill Mode 635
8.8 Прозрачность 635
8.8.1 Печать с прозрачностью 636
8.9 Editing entity properties 636
8.9.1 Editing in the Properties panel 636
8.9.2 Editing with Rollover Tips 636
8.10 Drawing linear entities 636
8.11 Отрезки 637
8.11.1 Основные операции 638
8.11.2 Укажите угол и длину 638
8.11.3 Указание угла относительно другого линейного объекта 638
8.12 Лучи 639
8.12.1 Основные операции 639
8.13 Прямые 639
8.13.1 Основные операции 640
8.14 Полилинии 640
8.14.1 Основные операции 641
8.14.2 Создание контурной полилинии 642
8.15 Мультилинии 642
8.15.1 Основные операции 643
8.16 Drawing splines 643
8.16.1 General procedure 644
8.17 Эскизы 644
8.17.1 Основные операции 644
8.18 Окружности 645
8.18.1 Основные операции 645
8.18.2 Построение окружности, касательной к трем объектам 646
8.18.3 Редактирование окружности 646
8.19 Drawing arcs 646
8.19.1 General procedure 647
8.19.2 Editing an arc 647
8.20 Эллипсы 647
8.20.1 Основной метод построения эллипса 648
8.20.2 Основной метод построения эллиптической дуги 648

# Содержание


8.20.3 Редактирование эллипса или эллиптической дуги 648
8.21 Прямоугольники 649
8.21.1 Основные операции 649
8.22 Многоугольники 650
8.22.1 Основные операции 650
8.23 Drawing traces 651
8.23.1 General procedure 651
8.24 Кольца 652
8.24.1 Основные операции 652
8.25 Drawing solids 653
8.25.1 General procedure 653
8.26 Drawing revision clouds 654
8.26.1 General procedure 654
8.27 Drawing points 655
8.27.1 General procedure 655
8.28 Drawing wipeouts 656
8.28.1 General procedure 656
8.28.2 To create a wipeout from a polyline 656
8.29 Спирали 657
8.29.1 Основные процедуры при построении 2D спирали 657
8.29.2 Основные процедуры при построении 3D спирали 658
8.29.3 Редактирование спирали 658
8.30 3D полилинии 659
8.30.1 Основные операции 659
8.30.2 Редактирование 3D полилинии 660
8.31 Creating regions 660
8.31.1 To create regions 660
8.32 Creating hatches and gradient fills 660
8.32.1 General procedure 661
8.32.2 To define hatch properties 661
8.32.3 To define gradient fill properties 662
8.32.4 To define the boundary 663
8.32.5 Set the options 664
8.33 Editing hatches and gradient fills 664
8.33.1 To edit a hatch or gradient fill in the properties panel 664
8.33.2 To edit a hatch or gradient fill in a dialog box 664
8.33.3 To edit a hatch or gradient fill using the Quad 665
9. 3D моделирование 666
9.1 Direct modeling overview 666
9.2 Navigate in 3D 667
9.2.1 To control the Perspective property 667
9.2.2 Mouse and keyboard navigation options 667
9.2.3 The LookFrom navigation tool 667
9.3 Direct modeling commands 668
9.3.1 Creation commands: 668
9.3.2 Modification commands: 668
9.3.3 Helper commands: 669
9.4 Direct modeling operations 671

# Содержание


9.4.1 Controlling the coordinate system 671
9.4.2 Selecting geometry 671
9.4.3 Moving faces to control dimensions 672
9.4.4 Making holes 674
9.4.5 Deleting features 675
9.4.6 Removing edges between coplanar faces 676
9.4.7 Copying a Feature 676
9.4.8 Deformable modeling 677
9.5 Design intent recognition 678
9.6 Working with sections 681
9.6.1 Difference between Live Section and Clip Display 682
9.6.2 Align a section plane to a face 682
9.6.3 To create a vertical section plane 682
9.6.4 To create a horizontal section plane 683
9.6.5 To create a vertical jogged section plane 684
9.6.6 To create a horizontal jogged section plane 684
9.6.7 To create an orthographic section plane 685
9.6.8 To modify the size and position of a section plane 686
9.6.9 To save a section 688
9.7 Create a TIN Surface 688
9.7.1 Commands 688
9.7.2 Overview 688
9.7.3 Setting the TIN Surface preferences 689
9.7.4 Properties of a TIN Surface 690
9.7.5 Procedure: Creating a TIN Surface from a selection set of points and polylines 691
9.7.6 Procedure: Creating a TIN Surface from an imported points file (.txt or .csv) 692
9.7.7 Procedure: Creating a TIN Surface of a selected area using Clip polygon 692
9.7.8 Procedure: Creating a TIN Surface by placing points 693
9.7.9 Procedure: Creating a TIN Surface from Faces 693
9.7.10 Procedure: Creating a TIN Surface from a Point Cloud 694
9.7.11 Procedure: Creating a TIN Surface from a Civil 3D surface 694
9.7.12 About TIN Assign Image 694
9.7.13 About TIN Project 695
9.8 TIN Surface Edit 696
9.9 TIN Surface Direct Modeling 696
9.9.1 Commands 696
9.9.2 Overview 696
9.9.3 About TINEXTRACT 696
9.9.4 Procedure: Creating a solid from a TIN surface using option Elevation 696
9.9.5 Procedure: Creating Points from a TIN surface 697
9.9.6 Procedure: Creating Faces from a TIN surface 697
9.9.7 Procedure: Creating a Contours from a TIN surface 697
9.9.8 Procedure: Creating a Border from a TIN surface 698
9.9.9 About TIN Merge 698
9.9.10 Merging TIN Surfaces 698
9.9.11 About TIN Modify 699
9.9.12 Procedure: Adding deformations on a TIN Surface using Radius 699
9.9.13 Procedure: Adding deformations on a TIN Surface using specify Contour 700
9.9.14 Procedure: Adding deformations on a TIN Surface using select Entity 701

# Содержание


9.9.15 Procedure: Adding deformations on a TIN Surface using draw Polygon 702
9.9.16 Procedure: Smoothening a TIN Surface 703
9.10 Analyzing TIN Surfaces 704
9.10.1 Commands 704
9.10.2 Overview 704
9.10.3 About TIN Volume 704
9.10.4 Procedure: Creating a TIN Volume 704
9.10.5 Creating a TIN Volume using a comparison TIN Surface 705
9.10.6 Creating a TIN Volume using Elevation 705
9.10.7 About TIN Water Drop 706
9.10.8 Procedure: Creating a water drop path 706
9.11 Working with Gradings 707
9.11.1 Commands 707
9.11.2 Overview 707
9.11.3 Procedure: Grading a surface to create a pond 708
9.11.4 Displaying the properties of a graded surface 709
9.11.5 Procedure: Grading a surface to create a platform 711
9.11.6 About Balance Grading Volumes 713
9.11.7 Procedure: Adjusting the elevation of a grading to balance cut and fill volumes 713
9.12 Create and edit alignments 714
9.12.1 Commands 714
9.12.2 Overview 714
9.12.3 About ALIGNMENT 714
9.12.4 Procedure: Creating an Alignment 715
9.12.5 Procedure: Creating a Horizontal Alignment without 3D alignments 716
9.12.6 About Vertical Alignment View 716
9.12.7 Procedure: Creating a Vertical Alignment View 717
9.12.8 Procedure: Creating a Vertical Alignment 718
9.12.9 Procedure: Creating an Alignment from Civil 3D drawing 719
9.12.10 About ALIGNMENTEDIT 719
9.12.11 Procedure: Continuing a horizontal alignment with ALIGNMENTEDIT 719
9.12.12 Procedure: Adding PI to an existing horizontal alignment 721
9.12.13 Procedure: Removing PI to an existing horizontal alignment 721
9.12.14 Procedure: Adding additional TIN Surface to Vertical alignment view 722
9.13 Modeling a Corridor 722
9.13.1 Commands 722
9.13.2 Overview 723
9.13.3 About Corridor 723
9.13.4 About Corridor Template 723
9.13.5 About Corridor Template Element 723
9.13.6 About Corridor Extract 723
9.13.7 Procedure: Creating a template element of kerb 723
9.13.8 Procedure: Creating a template of road cross-section with asphalt layer and left and right kerb 725
9.13.9 Procedure: Creating a road corridor 727
9.13.10 About Corridor Edit 728
9.13.11 Procedure: Editing a corridor 728
9.13.12 Displaying the properties of a Corridor 730

# Содержание


10. Параметрическое моделирование 734
10.1 Parametric modeling overview 734
10.1.1 Global and local parameters 734
10.1.2 Defining an expression for a parameter 734
10.1.3 Defining dependent parameters 735
10.1.4 Parameters of associative arrays 735
10.1.5 Parameters of mechanical components 735
10.1.6 Parameter properties 736
10.1.7 3D constraints properties 736
10.1.8 Context menus 737
10.1.9 The Parameters and Constraints panel 738
10.2 Operations and functions in expressions 739
10.2.1 Operators 739
10.2.2 Expressions 739
10.3 3D constraints 741
10.3.1 Variational Direct Modeling 741
10.3.2 Geometry-driven constraints 742
10.3.3 3D constraints and direct modeling 742
10.4 Suppression feature 743
10.4.1 Creating a suppression feature 743
10.4.2 Managing suppression features 743
10.4.3 Setting a condition for a suppression feature 744
10.5 Working with design tables 744
10.5.1 Creating a design table from a .csv file 744
10.5.2 Direct lookup in a design table 745
10.5.3 Reverse lookup in a design table 745
10.5.4 Adding a new configuration to a design table 745
10.5.5 Creating an empty design table 746
10.5.6 Using a parametric component with design tables 746
10.5.7 Other actions with design tables 746
11. Облака точек 747
11.1 Working with Point cloud 747
11.1.1 Commands 747
11.1.2 Pre-processing 747
11.1.3 Alignment 747
11.1.4 Bubble Viewer 747
11.1.5 Entity snaps 749
11.1.6 Export 751
11.1.7 Floor Detection 751
11.1.8 Point cloud projection 751
11.1.9 Planar Fit 752
11.1.10 In bubble view 752
11.1.11 In model space 752
12. Блоки, атрибуты, внешние ссылки и компоненты 754
12.1 Создание блоков 754
12.1.1 Создание блока 754
12.1.2 Сохранение блока в отдельный файл 755
12.2 Using Blockify 756
12.2.1 Blockify from a selected input 756

# Содержание


12.2.2 Existing block definitions 757
12.2.3 Combining duplicate block definitions 757
12.3 Редактирование блоков 757
12.4 Вставка блоков 757
12.4.1 Использование команды INSERT 759
12.4.2 Вставка блока с помощью Проводника по чертежам 759
12.4.3 Вставка блока с выравниванием 760
12.4.4 Вставка блоков в виде прямоугольного массива 761
12.5 Расчленение блоков 762
12.5.1 Чтобы расчленить блок 762
12.5.2 Настройка свойства Расчленяемый 762
12.6 Параметрические 2D блоки 763
12.6.1 Основные процедуры при создании параметрических 2D блоков 763
12.6.2 Вставка параметрических блоков 763
12.6.3 Настройка значений параметров 763
12.7 Dynamic blocks 763
12.7.1 Editing dynamic blocks 764
12.7.2 Grip editing dynamic blocks 764
12.8 Определение атрибутов 765
12.8.1 Основные процедуры при создании атрибутов 765
12.8.2 Определение атрибута 765
12.8.3 Использование свойств подшивки в основной надписи 767
12.9 Работа с атрибутами 768
12.9.1 Основные операции при работе с атрибутами 768
12.10 Редактирование определений атрибутов 769
12.11 Редактирование атрибутов блока 769
12.12 Вставка блоков с атрибутами 769
12.12.1 Вставка блока с атрибутами 770
12.13 Извлечение данных из атрибутов 770
12.13.1 Извлечение данных из атрибутов 772
12.14 Управление внешними ссылками 772
12.14.1 Отключение внешней ссылки 773
12.14.2 Выгрузка внешней ссылки 773
12.14.3 Перезагрузка внешней ссылки 773
12.15 Подключение внешней ссылки 773
12.15.1 Чтобы подключить внешнюю ссылку 773
12.16 Working with external references 774
12.17 Редактирование блоков и внешних ссылок 775
12.17.1 Редактирование внешних ссылок 775
12.18 Работа с группами 777
12.18.1 Создание группы 777
12.18.2 Редактирование группы 778
12.18.3 Разгруппировка объектов 778
12.18.4 Изменение порядка объектов группы 779
12.19 Работа с подосновами 779
12.19.1 Управление кешированием подосновы PDF 780
12.19.2 Подключение подосновы PDF 781
12.19.3 Вставка подосновы PDF 782

# Содержание


12.19.4 Управление отображением подосновы PDF 782
12.19.5 Настройка отображения слоев подосновы PDF 782
12.19.6 Обрезка подосновы PDF 783
12.19.7 Отображение отсеченной части подосновы PDF 783
12.19.8 Удаление контура обрезки 783
12.19.9 Настройка параметров отображения подосновы PDF 784
12.20 Components and component inserts 784
12.20.1 External and local components 784
12.20.2 Converting components 785
12.20.3 Parametric Components 785
12.20.4 To edit the parameters of a component insert 786
12.20.5 To replace components 787
12.20.6 Components, blocks or external references? 787
12.20.7 Converting blocks and external references 787
12.20.8 Component-based features 787
12.20.9 Assign physical materials to components 788
13. Управление изображениями 789
13.1 Managing images 789
13.1.1 Attaching an image 789
13.1.2 Loading and unloading images 790
13.1.3 Detaching images 790
13.2 Image utilities 791
13.2.1 Clipping an image 791
13.2.2 Modifying the Image Frame setting 791
13.2.3 Setting the image transparency 792
13.2.4 Setting the display properties of an image 792
14. Аннотация 794
14.1 Dimensions 794
14.1.1 Обзор инструментов для создания размеров 794
14.1.2 Ассоциативные размеры 797
14.1.3 Параметры размеров 797
14.1.4 Working with dimension styles 798
14.1.5 Using the dimension style tools 801
14.1.6 Creating linear dimensions 802
14.1.7 Создание угловых размеров 804
14.1.8 Создание диаметральных и радиальных размеров 806
14.1.9 Centerlines and center marks 807
14.1.10 Creating ordinate dimensions 809
14.1.11 Editing dimensions 810
14.1.12 Создание выносок с комментариями 812
14.1.13 Creating multileaders 813
14.1.14 Добавление геометрических допусков 816
14.2 Texts 817
14.2.1 Text system variables 817
14.2.2 Работа со стилями текста 819
14.2.3 Создание текста 821
14.2.4 Редактирование текста 827
14.2.5 Exploding text 828
14.2.6 Поиск и замена текста 829

# Содержание


14.2.7 Проверка правописания 830
14.2.8 Working with fields 833
14.2.9 Работа с таблицами 835
14.3 Annotative Entities 841
14.3.1 Аннотативные объекты 841
14.3.2 Annotation scales 844
14.3.3 Аннотативные текстовые объекты 845
14.3.4 Аннотативные штриховки и градиентные заливки 845
14.3.5 Annotative blocks 846
14.3.6 Аннотативные размеры 847
14.3.7 Аннотативные выноски 847
15. Настройка 849
15.1 Startup options 849
15.2 Пользовательские файлы BricsCAD 850
15.2.1 Диспетчер пользовательских файлов BricsCAD 851
15.3 Loading and unloading CUI files 852
15.3.1 Loading a main CUI file 852
15.3.2 Loading a partial CUI file 852
15.3.3 Unloading a partial CUI file 852
15.3.4 Managing customization groups 853
15.4 Creating a partial CUI file 853
15.5 Managing toolboxes 853
15.5.1 Creating a toolbox in a menu group 853
15.5.2 Creating a tool in a toolbox 854
15.6 Creating toolbars 854
15.6.1 To add a flyout to a toolbar 855
15.7 Creating menus 856
15.7.1 Creating a main menu 856
15.7.2 Adding a submenu to a main menu 856
15.7.3 Adding a submenu to a submenu 857
15.8 Creating context menus 857
15.8.1 Creating shortcut menus 858
15.9 Creating keyboard shortcuts 858
15.10 Customizing the Quad 859
15.10.1 Creating a quad tab 859
15.10.2 Copying a quad tab 859
15.10.3 Setting the Entity Filter property of a tool 860
15.11 Adding tools to user interface controls 860
15.12 Настройка ленты 861
15.12.1 Создание закладки ленты 861
15.12.2 Создание панели ленты 861
15.12.3 Добавление панели ленты на закладку 862
15.12.4 Добавление строки на панель 862
15.12.5 Добавление элементов в строку 862
15.12.6 Определение свойств элемента 863
15.12.7 Изменение положения элементов 863
15.12.8 Удаление объектов 863
15.13 Tool palettes 863

# Содержание


15.13.1 Opening the Tool Palettes panel 863
15.13.2 Setting the view options 863
15.13.3 Customizing tool palettes 864
15.13.4 Opening the **Customize** dialog box 864
15.13.5 Importing a tool palette 864
15.13.6 Creating a palette group 864
15.13.7 Creating a palette subgroup 864
15.13.8 Adding a palette to a group 864
15.13.9 Removing a palette from a group 864
15.13.10 Setting a palette group current 864
15.13.11 Deleting a palette group 865
15.13.12 Renaming a tool palette 865
15.13.13 Deleting a tool palette 865
15.13.14 Managing tools 865
15.13.15 Creating a new tool palette 865
15.13.16 Adding items to a tool palette 866
15.13.17 Adding hatch patterns 866
15.13.18 Adding blocks 866
15.13.19 Adding blocks from the current drawing 866
15.13.20 Adding blocks from any drawing 866
15.13.21 Adding a drawing as a block 867
15.13.22 Adding all blocks from multiple drawings 867
15.13.23 Adding commands 867
15.14 Command aliases 867
15.14.1 Editing the alias file in a text editor 868
15.14.2 Using the Customize dialog to edit the alias file 868
15.15 User profiles 869
15.15.1 Content of a user profile 869
15.15.2 The User Profile Manager 870
15.15.3 Starting BricsCAD with a specific user profile 870
15.16 Workspaces 871
15.16.1 Managing workspaces 872
15.16.2 Setting the current workspace 872
15.16.3 Creating a workspace 872
15.16.4 Editing a workspace 873
15.16.5 Deleting a workspace 874
15.16.6 Importing a workspace 874
15.17 Express tools 874
15.17.1 About Express tools 874
15.17.2 Adding Express tools to bricsCAD 875
15.17.3 Unloading the Express tools 875
15.17.4 Reloading the Express tools 875
15.18 Lisp 876
15.18.1 BricsCAD Lisp 876
15.18.2 Drawing Setup Lisp Routines for BricsCAD 877
16. Редактирование объектов 878
16.1 Modifying commands overview 878
16.2 Entity modification settings 880
16.3 Match properties 880

# Содержание


16.3.1 Copying properties between entities 880
16.3.2 Copying properties between entities with BIM data 881
16.4 Grips editing 881
16.4.1 Selecting multiple grips 882
16.4.2 Grip Edit Command Options 882
16.4.3 The grips editing context menu 883
16.4.4 Stretching entities using grips 883
16.4.5 Moving entities using grips 883
16.4.6 Copying entities using grips 884
16.4.7 Mirroring entities using grips 884
16.4.8 Rotating entities using grips 885
16.4.9 Scaling entities using grips 885
16.5 Selection methods 886
16.5.1 Composing a selection set before (pre-pick) 886
16.5.2 Composing a selection set afterwards (post-pick) 887
16.5.3 Selecting overlapping entities 888
16.5.4 Selecting subentities 888
16.6 Quick select 888
16.6.1 To create a selection set 888
16.7 Draw order 889
16.7.1 Using Draw Order tools 890
16.8 The manipulator 890
16.8.1 Adaptive snap step 890
16.8.2 The manipulator settings 891
16.8.3 The manipulator layout 891
16.8.4 The location and orientation of the manipulator 892
16.8.5 Displaying the manipulator 893
16.8.6 Moving the manipulator 893
16.8.7 Rotating the manipulator 894
16.8.8 Reorienting the manipulator 894
16.8.9 Aligning the manipulator with a coordinate system 894
16.8.10 Aligning the manipulator with the face of a solid 894
16.9 Moving entities 894
16.9.1 Moving entities in a drawing 895
16.9.2 Moving entities between drawings 895
16.9.3 Moving a selection as a block 896
16.10 Rotating entities 896
16.10.1 General procedure 897
16.10.2 Rotating a selection set relative to a base angle 897
16.10.3 Rotating in 3D 898
16.11 Mirroring entities 898
16.11.1 Mirroring entities along a line 898
16.11.2 Mirroring entities along a plane 899
16.12 Aligning entities 900
16.12.1 Aligning in 2D 900
16.12.2 Aligning in 3D 901
16.13 Changing entities 903
16.13.1 Changing text entities 903
16.13.2 Relocating blocks 903

# Содержание


16.14 Copying entities 904
16.14.1 General procedure 904
16.14.2 Using the Repeat option 905
16.14.3 Using the Array option 905
16.14.4 Copying a selection as a block 906
16.14.5 Copying entities between drawings 906
16.15 Making parallel copies 906
16.15.1 Making a parallel copy at a specified distance 907
16.15.2 Making a parallel copy through a point 907
16.16 Arraying entities 908
16.16.1 Creating a static rectangular array 908
16.16.2 Creating a static polar array 909
16.16.3 Creating a static 3D rectangular array 909
16.16.4 Creating a static 3D polar array 911
16.17 Extending entities 912
16.17.1 Extending entities 912
16.18 Trimming entities 913
16.18.1 Trimming entities 913
16.19 Lengthening entities 914
16.19.1 Changing the length dynamically 914
16.20 Selection and grip settings 914
16.20.1 Selection settings 914
16.20.2 Grips settings 918
16.21 Stretching entities 919
16.21.1 Stretching entities 919
16.22 Scaling entities 920
16.22.1 Scaling a selection set 920
16.22.2 Scaling a selection set using a reference scale 920
16.23 Breaking entities 921
16.23.1 To break an entity 921
16.24 Joining entities 922
16.24.1 General procedure 922
16.24.2 Joining two arcs 922
16.25 Chamfering entities 923
16.25.1 Chamfering using the distance-distance method 923
16.25.2 Chamfering using the length-angle method 924
16.25.3 Chamfering all vertices of a polyline 924
16.25.4 Removing a chamfer 924
16.26 Filetting entities 925
16.26.1 Filleting two entities or polyline segments 925
16.26.2 Filleting all vertices of a polyline 925
16.26.3 Filleting two parallel lines 926
16.27 Editing polylines 926
16.27.1 Launching the PEDIT command 926
16.27.2 Converting an entity into a polyline 927
16.27.3 Fitting a curve to a polyline 927
16.27.4 Tapering a polyline uniformly along its length 927
16.27.5 Reversing the direction of a polyline 928
16.27.6 Opening or closing a polyline 928

# Содержание


16.27.7 Applying a uniform width to a polyline 928
16.27.8 Changing the width of a polyline segment 928
16.27.9 Setting the linetype mode 929
16.27.10 Using the edit polyline segments tools in the Quad 929
16.28 Exploding entities 929
16.28.1 Exploding entities 930
16.29 Measuring entities 930
16.29.1 Measuring an entity using points 930
16.29.2 Placing blocks a specified interval along an entity 930
16.30 Dividing entities 931
16.30.1 Dividing an entity using points 931
16.30.2 Dividing an entity using blocks 932
17. Печать 933
17.1 General procedure for printing 933
17.1.1 Opening the Print dialog box 934
17.1.2 Choosing the paper size 934
17.1.3 Choosing the paper unit 934
17.1.4 Choosing the paper orientation 934
17.1.5 Defining the plot offset 934
17.1.6 Defining the plot scale 935
17.1.7 Setting the plot options 936
17.1.8 Setting the shade plot mode 937
17.1.9 Defining the plot stamp 937
17.1.10 Specifying the number of copies 938
17.1.11 Previewing the plot output 938
17.2 Plotter configuration 939
17.2.1 Configuring the built-in PDF printer 939
17.2.2 Creating a plotter configuration 940
17.2.3 Editing a plotter configuration 941
17.2.4 Assigning a plotter configuration 941
17.3 Plot style tables 942
17.3.1 Opening the Plot Style Table editor 942
17.3.2 Assigning a plot style table 945
17.4 The plot styles manager 945
17.4.1 Launching the plot styles manager 946
17.4.2 Creating a new plot style table 946
17.4.3 Creating a new plot style table in the Print dialog box 946
17.4.4 Editing a plot style table 947
17.4.5 Editing a plot style table in the Print dialog box 947
17.5 Batch plot 947
18. Визуализация 948
18.1 Using TwinMotion 948
18.1.1 Installing Twinmotion 948
18.1.2 Exporting your model 949
18.1.3 Importing your model in Twinmotion 950
18.2 Rendering overview 950
18.3 Rendering preferences 951
18.3.1 Setting the render preferences 951
18.4 Rendering tools 952

# Содержание


18.5 Render materials 954
18.5.1 Materials glossary 954
18.5.2 Defining render materials 956
18.5.3 Assigning render materials 956
18.5.4 Adding a render material to the drawing 956
18.5.5 Assigning a render material to entities 956
18.5.6 Removing a render material 957
18.5.7 Assigning a material to a layer 957
18.5.8 Adding the Redway material library 957
18.5.9 Adding your own materials 958
18.6 Lights 959
18.6.1 Lighting glossary 959
18.6.2 System variables 959
18.6.3 General properties 960
18.6.4 Photometric light properties 960
18.6.5 Geometry properties 961
18.6.6 Attenuation properties 961
18.6.7 Rendered shadow details 962
18.6.8 Spotlight hotspot and falloff properties 962
18.6.9 Managing Lights 963
18.6.10 Defining a light 963
18.6.11 Editing lights 964
18.6.12 Defining the sun properties 964
18.6.13 Defining the geographic location 964
18.7 Creating a rendered image 964
18.7.1 Rendering a viewport 965
18.7.2 Rendering in a window 965
18.7.3 Saving a rendering to a file 965
19. Обратная связь 966
19.1 Diagnostics and usage data collection 966
19.1.1 Stopping the process 968
19.1.2 Restarting the process 968
19.1.3 Removing your data from the database 968
20. BricsCAD BIM 969
20.1 Что нового 969
20.1.1 Overview 969
20.1.2 Modeling techniques 970
20.1.3 Building data 977
20.1.4 Project collaboration 986
20.1.5 Design documentation 987
20.1.6 Point clouds 996
20.2 Начало работы 999
20.2.1 Creating a BIM model 999
20.2.2 Exploring the interface 1000
20.2.3 Quad cursor 1016
20.2.4 Viewing entities 1021
20.2.5 Selecting entities 1024
20.2.6 Level of detail 1037
20.3 Методы моделирования 1038

# Содержание


20.3.1 BIM linear solid 1038
20.3.2 BIM window create 1044
20.3.3 The library panel 1051
20.3.4 Connecting solids 1056
20.3.5 Creating a custom window 1062
20.3.6 Creating roofs 1068
20.3.7 Creating slabs 1084
20.3.8 Creating walls 1086
20.3.9 Curtain wall 1097
20.3.10 DmExtrude 1105
20.3.11 DmPushPull 1112
20.3.12 Editing grids 1113
20.3.13 Inserting and editing windows and doors 1115
20.3.14 Mass modeling with quickbuilding 1123
20.3.15 MEP modeling 1130
20.3.16 MEP flow connection points 1141
20.3.17 Offset 1146
20.3.18 Splitting solids 1148
20.3.19 Stair tool 1150
20.3.20 Structural steel 1160
20.3.21 Using BIM quickdraw 1178
20.3.22 Using drag 1190
20.3.23 Using parametrize 1193
20.3.24 Working with parameters and constraints 1202
20.4 Данные здания 1213
20.4.1 Analytical model 1213
20.4.2 AutoComplete BIM data 1225
20.4.3 Attaching compositions 1231
20.4.4 BIM classify 1235
20.4.5 BIM compositions 1239
20.4.6 BIM data structure 1246
20.4.7 BIM ply editing 1248
20.4.8 BIM project info 1256
20.4.9 Using Python scripts (beta) 1260
20.4.10 Classification codes 1264
20.4.11 Creating and editing BIM spaces 1268
20.4.12 Custom properties 1270
20.4.13 Customizing BIM quantities 1274
20.4.14 Load bearing direction for slabs 1279
20.4.15 Physical materials 1281
20.4.16 Project browser 1288
20.4.17 Spatial locations 1297
20.4.18 Structure browser 1301
20.5 Средства проектирования 1307
20.5.1 Using bimify 1307
20.5.2 Using propagate 1313
20.5.3 Propagate planar 1322
20.5.4 Propagate linear 1325
20.5.5 Propagate pattern 1327

# Содержание


20.5.6 Propagate edges 1330
20.5.7 Propagate corner 1333
20.5.8 Parametrize details 1336
20.5.9 Details panel 1351
20.5.10 Rhino/Grasshopper integration 1359
20.6 Совместная работа над проектом 1375
20.6.1 BIM collaboration format (BCF) panel 1375
20.6.2 Bricsys 24/7 panel 1381
20.6.3 IFC import and export in BricsCAD 1385
20.6.4 Import and underlay Autodesk® Revit® files in BricsCAD 1389
20.6.5 Importing Autodesk® Revit® families in BricsCAD 1397
20.6.6 Interference checking 1401
20.6.7 Using Xrefs 1403
20.7 Проектная документация 1408
20.7.1 Annotations in 3D 1408
20.7.2 BIM sections 1414
20.7.3 BIM tags 1424
20.7.4 Create drawing types with drawing customization 1438
20.7.5 Define detailed section 1458
20.7.6 Define reflected ceiling plan 1461
20.7.7 Detailing with BimPatch 1463
20.7.8 Editing data extraction definition (.dxd) files 1467
20.7.9 Editing generated drawings 1471
20.7.10 Elevation hatch 1474
20.7.11 Enscape integration 1477
20.7.12 Generating Drawings 1484
20.7.13 Working with Point Clouds in BIM 1490
20.7.14 Interior elevations 1498
20.7.15 Schedules 1503
20.7.16 Spot elevation level 1516
20.7.17 Templates for generated drawings 1529
21. BricsCAD Mechanical 1540
21.1 Что нового 1540
21.1.1 Обзор 1540
21.1.2 Диспетчер спецификаций 1541
21.1.3 Библиотеки стандартных изделий 1544
21.1.4 Параметры и зависимости 1545
21.1.5 Стандартные детали 1545
21.1.6 Листовой металл 1545
21.2 Начало работы 1548
21.2.1 Рабочее пространство Mechanical 1548
21.2.2 Mechanical browser panel 1548
21.2.3 Сравнение 3D моделей 1552
21.3 Моделирование деталей 1555
21.3.1 Элемент Резьба 1555
21.3.2 Методы проектирования деталей 1556
21.3.3 Sketching 1557
21.3.4 Solid Modeling 1557
21.3.5 Поверхностное моделирование 1560

# Содержание


21.3.6 Моделирование произвольных форм 1561
21.4 Моделирование сборок 1562
21.4.1 Восходящее и нисходящее проектирование сборок 1562
21.4.2 Команды и панели инструментов 1564
21.4.3 Свойства компонентов 1570
21.4.4 Массовые характеристики сборки 1579
21.4.5 Standard Mechanical Components 1581
21.4.6 Piping in Assembly Design 1586
21.4.7 Массивы компонентов 1588
21.4.8 Свойства массива 1590
21.4.9 Покомпонентные виды 1593
21.4.10 Диспетчер спецификаций 1604
21.4.11 Спецификация 1641
21.4.12 Параметрическая 3D анимация 1665
21.5 Листовой металл 1680
21.5.1 Методы проектирования из листового металла 1680
21.5.2 Команды и панели инструментов 1683
21.5.3 Элементы деталей из листового металла 1692
21.5.4 Цветовое выделение элементов 1698
21.5.5 Обозреватель конструкции для листового металла 1698
21.5.6 Свойства деталей из листового металла и их элементов 1705
21.5.7 K-фактор 1723
21.5.8 Таблица гибов 1727
21.5.9 Параметры листового металла 1733
21.5.10 Creating Flanges 1736
21.5.11 Редактирование стенок из листового металла 1751
21.5.12 Working with Bends and Junctions 1763
21.5.13 Working with Hems 1774
21.5.14 Работа с шип-пазами 1779
21.5.15 Working with T-tabs and T-junctions 1784
21.5.16 Working with Bevels 1787
21.5.17 Creating and Splitting Lofted Bends 1791
21.5.18 Bend Relief and Corner Relief Feature Operations 1794
21.5.19 Работа с элементами формы и ребрами жесткости 1799
21.5.20 Работа с отверстиями 1826
21.5.21 Converting to Sheet Metal and Repairing 1830
21.5.22 Unfolding Sheet Metal Parts and Exporting to CAM 1833
21.5.23 Ассоциативные развертки 3D тел 1838
21.5.24 Smart Features 1847
21.6 Генерация чертежей 1851
21.6.1 Drawing Views 1851
21.6.2 Аннотации чертежа 1853
22. Communicator для BricsCAD 1854
22.1 BricsCAD Communicator 1854
22.1.1 Supported export file formats 1854
22.1.2 Supported import file formats 1855
22.2 BricsCAD Communicator settings 1856
22.2.1 Import settings 1856
22.2.2 Product structure 1856

# Содержание


22.2.3 Translate colors 1857
22.2.4 Product and manufacturing information 1857
22.2.5 Perform stitching 1857
22.2.6 Repair model on import 1857
22.2.7 Perform simplification 1858
22.2.8 Hidden parts 1858
22.2.9 Creo options 1858
22.2.10 IGES options 1858
22.2.11 Export settings 1860
22.2.12 Product structure 1860
22.2.13 Hidden parts 1860
22.2.14 STEP options 1860
23. BricsCAD Shape 1862
23.1 Introduction to BricsCAD Shape 1862
23.1.1 What is BricsCAD Shape 1862
23.1.2 BricsCAD Shape lets you create without limits 1862
23.1.3 How is BricsCAD Shape different from other tools? 1862
23.1.4 BricsCAD Shape accelerates your BIM workflow 1862
23.1.5 Stop sketching. Start shaping! 1862
23.2 BricsCAD Shape user interface 1863
23.3 Basic concepts of modeling in BricsCAD Shape 1863
23.3.1 Selection methods 1863
23.3.2 Quad 1865
23.3.3 Manipulator 1865
23.4 Navigation in BricsCAD Shape 1867
23.4.1 Panning 1867
23.4.2 Rotating a view 1867
23.4.3 Zoom in/out 1868
23.4.4 Look From control 1868
23.5 Components in BricsCAD Shape 1868
23.5.1 To insert a component 1869
23.5.2 To insert a window in the face of a solid 1869
23.6 Materials in BricsCAD Shape 1870
23.7 Layers in BricsCAD Shape 1872
23.7.1 Turn on/off 1873
23.7.2 Lock/unlock 1873
23.7.3 Create/delete 1874
23.7.4 Edit 1874
23.8 Working with steel profiles 1874
23.8.1 Creating a steel profile 1874
23.8.2 Connecting Profiles 1878
23.8.3 Assigning a different profile 1878
23.8.4 Assigning a custom profile 1879
23.9 Visualization in BricsCAD Shape 1880
23.9.1 Visual styles panel 1880
23.9.2 Hide/Isolate/Show entities 1881
23.9.3 Sections 1882
23.10 Solids in BricsCAD Shape 1883
23.10.1 Drawing a box 1883

# Содержание


23.10.2 Manipulate a solid face 1884
23.10.3 Manipulating a solid edge 1885
23.10.4 Extrude solids 1887
23.11 Creating walls in BricsCAD Shape 1887
23.11.1 About the Polysolid tool 1887
23.11.2 Procedure: Create a simple wall 1887
23.11.3 Procedure: Using an existing 2D plan to create walls 1888
23.11.4 Procedure: Creating L-connections between two walls 1889
23.11.5 Procedure: Creating T-connections between two elements 1891
23.11.6 Procedure: Creating a curved wall 1892
23.11.7 Procedure: Connecting multiple faces of the wall to the roof slab 1893
23.12 Quickdraw in BricsCAD Shape 1894
23.12.1 About the Quickdraw tool 1894
23.12.2 The magnetic snapping feature 1894
23.12.3 Procedure: Creating an L-shaped house 1895
23.12.4 Procedure: using Quickdraw starting from a 2D layout 1897
23.13 Creating windows and doors in BricsCAD Shape 1898
23.13.1 Procedure: Pick windows or doors from the library 1899
23.13.2 Procedure: Create a custom window 1899
23.14 Creating roofs in BricsCAD Shape 1900

# 1. Что нового

## 1.1 Overview

BricsCAD® V21 offers a plethora of new and improved tools to help maximize your productivity.

This section gives an overview of all the new features and improvements in BricsCAD® **V21 Core and Civil tools**:

User interface 32

Productivity 35

Drafting 49

Modeling 51

Civil tools 60

Performance and compatibility 65

For information about the new features and improvements in BricsCAD® **V21 BIM**, go to the What's new in BricsCAD® V21 BIM section. It contains the following articles:

Overview 969

Modeling techniques 970

Building data 977

Project collaboration 986

Design documentation 987

Point clouds 996

For information about the new features and improvements in BricsCAD® **V21 Mechanical** tools, go to the What's new in BricsCAD® V21 Mechanical section. It contains the following articles:

Обзор 1540

Диспетчер спецификаций 1541

Библиотеки стандартных изделий 1544

Параметры и зависимости 1545

Стандартные детали 1545

Листовой металл 1545

### 1.1.1 Release notes

Click here for an overview of the release notes.

### 1.1.2 Diagnostics and Usage Data Collection

Many of the improvements were inspired by requests and suggestions from you, BricsCAD® users. At Bricsys, we continue to expand our methods of gathering user feedback with new data collection tools.

When you launch BricsCAD® for the second time, a dialog box displays, inviting you to participate in the Diagnostics and Usage Data Collection program. The data collection process enables you to share

anonymous diagnostic and usage data with Bricsys to help improve BricsCAD®. You can withdraw your consent and have your data removed from our databases at any time.

## 1.2 User interface

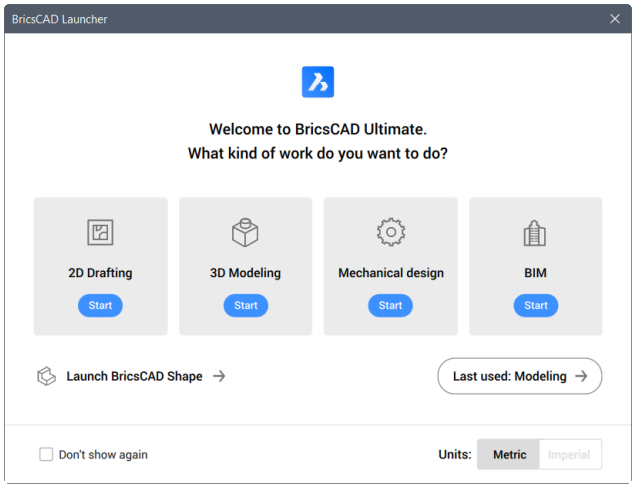
BricsCAD® V21 offers a variety of user interface enhancements to help increase your productivity without disrupting your existing workflow.

### 1.2.1 BricsCAD Launcher

The redesigned BricsCAD Launcher enables you to launch BricsCAD® in an environment that is most relevant for your design tasks and license level.

The available environment options depend on your license level. You can change the license level using the new SETLICENSELEVEL command which controls the RUNASLEVEL system variable. If you are using a trial license, you may choose any environment. Otherwise, you may choose any environment supported by the license level you purchased. For example, an Ultimate license supports all environments. Whereas, a Lite license supports only the 2D Drafting environment.

The new Units option enables you to set the measurement units (MEASUREINIT system variable) that determine which hatch pattern and linetype definition files to use. This control also specifies the drawing units for new drawings created with no template file.




### 1.2.2 Commands

The Command window now supports clickable command options enabling you to select options at the Command line without having to type at the keyboard.



The COMMANDS command allows you to look up available commands. In BricsCAD® V21 it's updated to list only the commands that are available with your current license level.

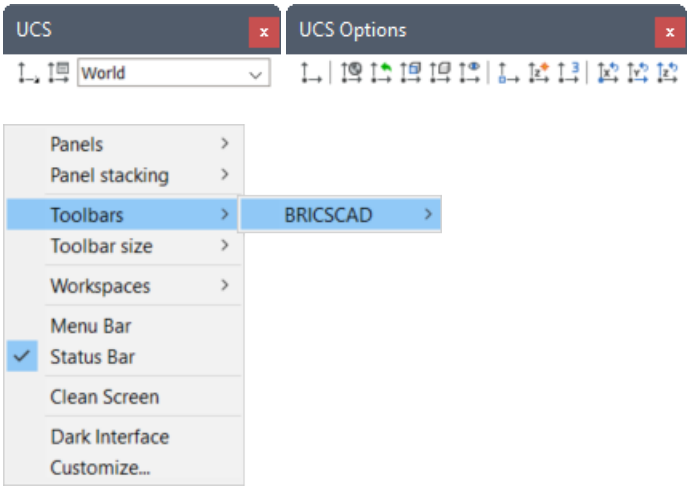
### 1.2.3 Ribbon

The ribbon enables you to scroll through ribbon tabs using the mouse wheel. Simply place the cursor anywhere on the ribbon and roll the wheel.

The RIBBONTOOLSIZE system variable sets the size of the tool buttons in the ribbon. The options are **Small**, **Large** and **Extra Large**.

### 1.2.4 Toolbars

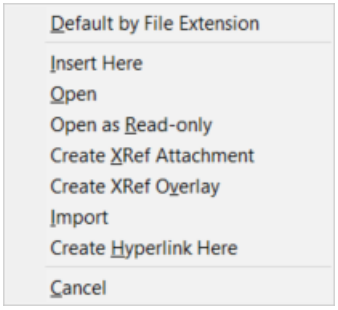
New UCS and UCS Options toolbars offer easy access to UCS commands and options. You an access these toolbars by right-clicking on a toolbar, a panel, or the ribbon and expanding the **Toolbars** > **BricsCAD** menu.




The TOOLBUTTONSIZE system variable applies to toolbars only, no longer to ribbon tool buttons (see above).

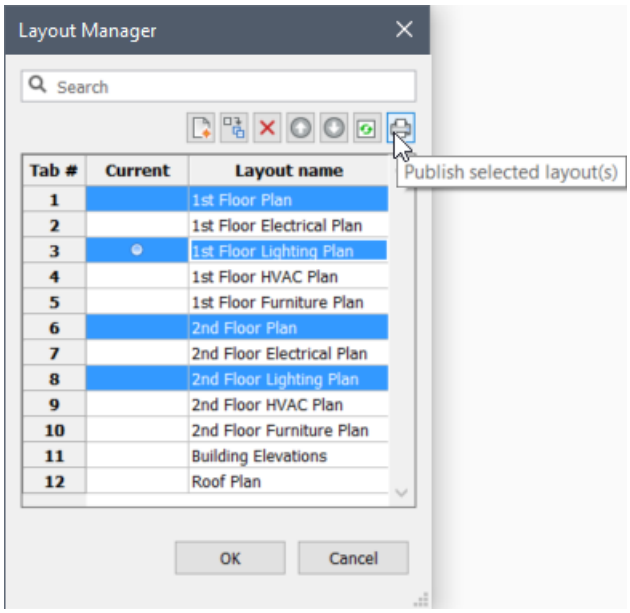
### 1.2.5 Menus

The menu that displays when you drag and drop a dwg file into BricsCAD®, using the right mouse button, offers a new option to **Open as Read-only**.

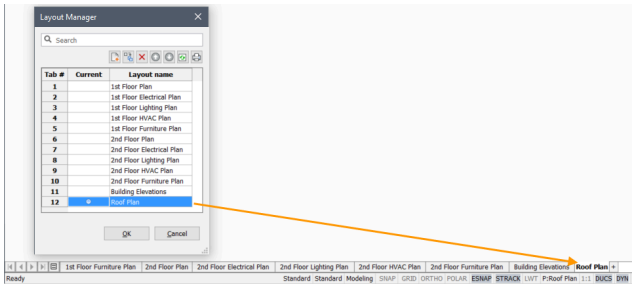



### 1.2.6 Layouts

The Layout Manager is extended to enable publishing of multiple layouts.

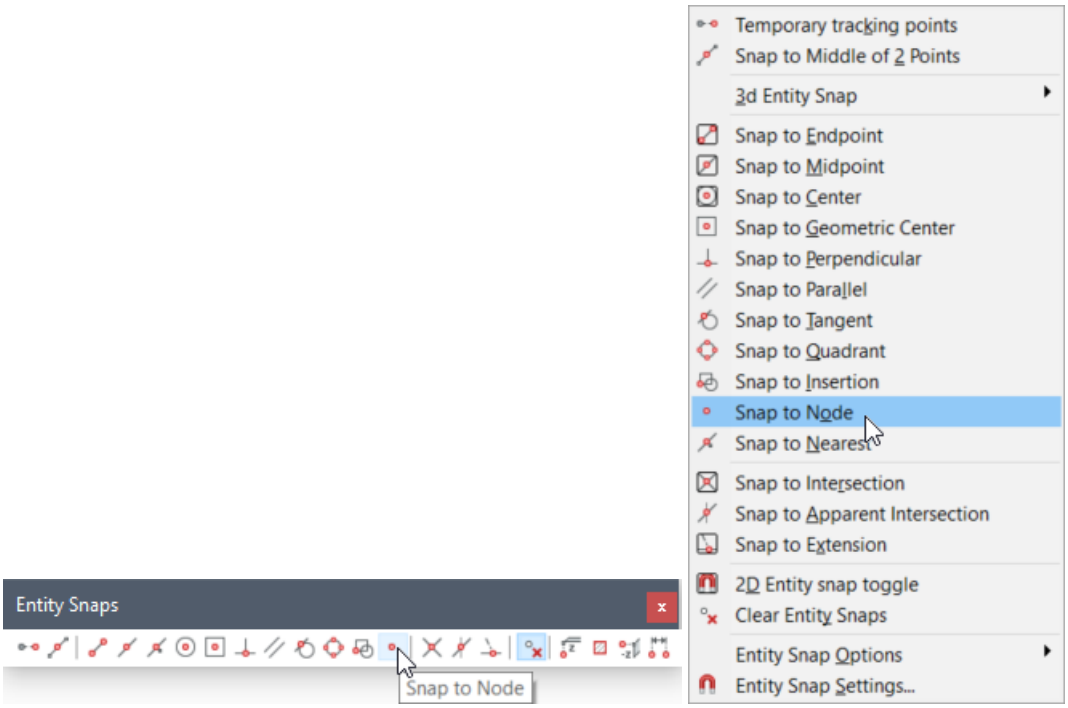



When you set a layout current in the Layout Manager, the active layout tab automatically scrolls to remain visible in the layout tab list.



### 1.2.7 Entity Snaps

The **Snap to Point** option in the Entity Snaps toolbar and menu is renamed to **Snap to Node**.

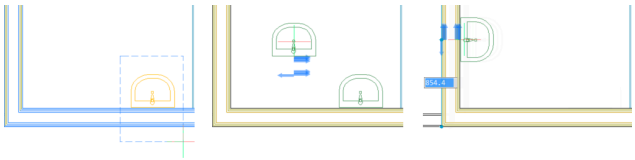



## 1.3 Productivity

### 1.3.1 Move with guide curves

The new MOVEGUIDED command enables you to move selected entities using reference curves.

When you launch the MOVEGUIDED command and select the entitiesyou want to move, entities that fall completelywithin the selection windowwill bemoved. Entities crossing the selection windowareused as reference curves.Blue vectors on theselectedentitiesrepresent the reference curves.As you move the selected entities near geometry that matches the reference curves, they automatically snap into place. Relevant geometry in the new location is automatically trimmed and geometry in the original location is automatically healed. This significantly reduces the amount of manual editing required!

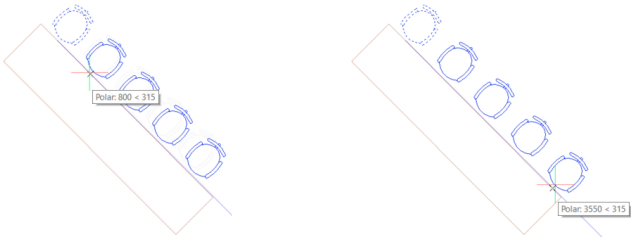


For more flexibility, you can first select the entities you want to move and then launch the MOVEGUIDED command. The pre-selected entities are used for the Move selection set and you are prompted to select entities to use as reference curves.

### 1.3.2 Copy in a linear array

The COPY command now includes a new **Array** option enabling you to copy selected entities in a linear array. The new Array option is available after you specify the base point for the copy operation.

You specify the number of copies to array then pick a point to specify the distance and angle between the first two sets of entities or use the **Fit** option to specify the distance and angle between the first and last sets of entities.



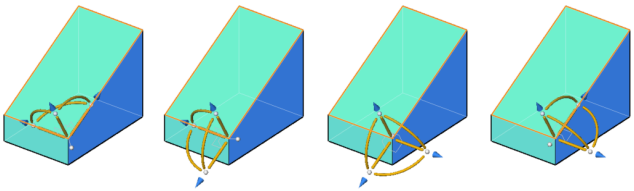

### 1.3.3 Undo per entity

A new History control in the Properties panel enables you to undo editing operations for a selected entity. The History property only displays when you select a single entity. Open the History drop-down list to undo the entity back to one of the previous steps without undoing all the other commands and view operations that have been executed since.



### 1.3.4 Manipulator

Press the SHIFT key to rotate the Manipulator 90 degrees about its normal axis.

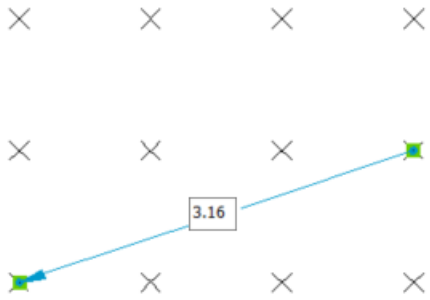


The Copy and Repeat options in the Manipulator are now visible, clickable, and localized.



### 1.3.5 Nearest Distance

The NEARESTDISTANCE system variable offers improved performance and behavior including support for Point entities.

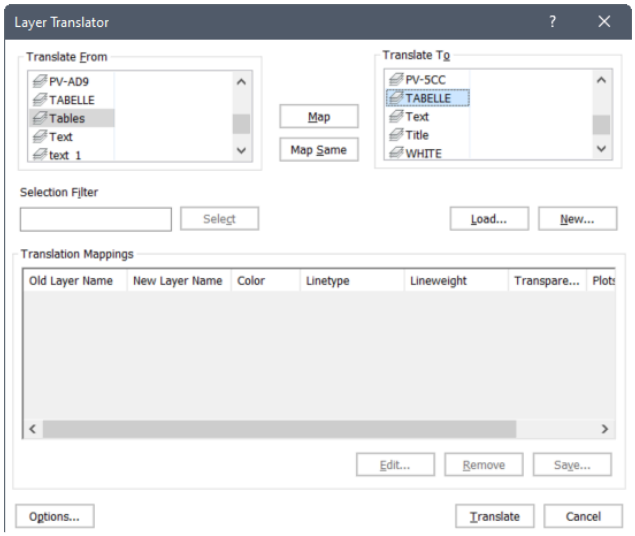


### 1.3.6 Layers

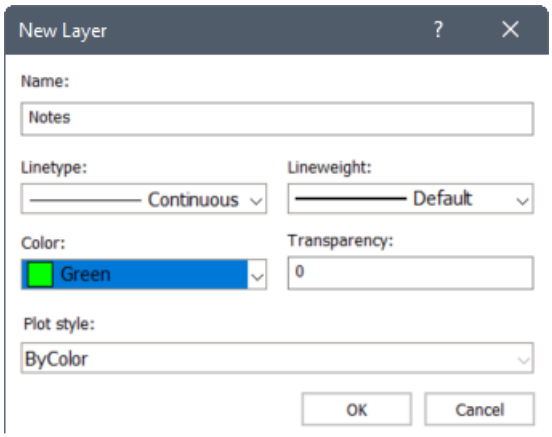
BricsCAD® V21 offers several layer enhancements to help increase your productivity.

**Layer Translator**

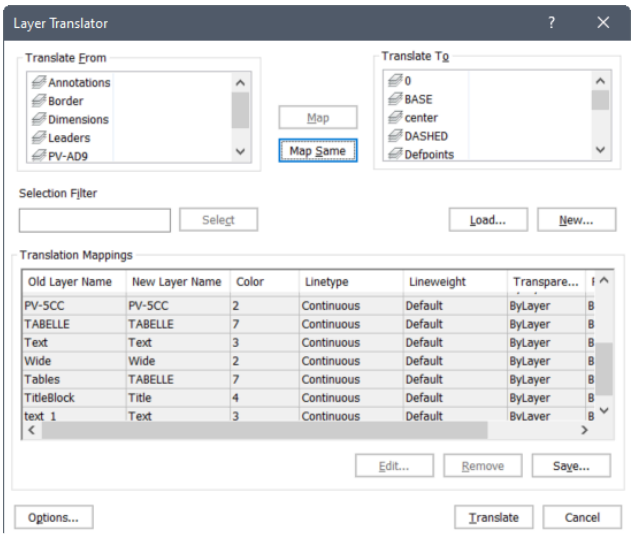
The new LAYTRANS command enables you quickly apply layer names and properties from another drawing file to the current drawing. When you launch the LAYTRANS command, the Layer Translator dialog displays. All the layer names in the current drawing are displayed in the Translate From list. You can specify the layers to which you want them to map by loading layer information from an existing DWG, DWS, or DWT file.




You can also create new layers. Simply enter the layer name and properties to which you want to map existing layer.

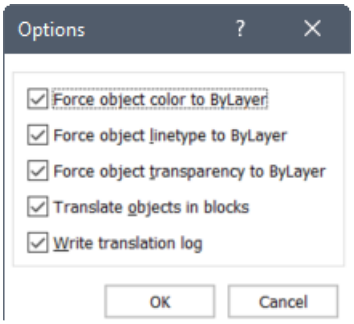



You can select one or more layers from the current drawing, the list on the left, to map to a layer from the list on the right. If you choose **Map Same**, any layer names in the current drawing with corresponding names in the list on the right will inherit the properties from the list on the right. The layer mappings you select are displayed in the Translation Mappings list.




Regardless of how you add layers to the Translation Mappings list, you can modify their mapping properties. Double-click on a layer or choose **Edit** to open the Edit Layer dialog box with the same options at the New Layer dialog. You can also remove layers from the Translation Mappings list or save the mapping list to use again in the future.

The Options dialog offers additional controls for layer mappings.




**Set by Layer**

The new SETBYLAYER and -SETBYLAYER commands enable you quickly change property overrides of selected entities to ByLayer.

Use the new SETBYLAYERMODE system variable to control which properties are affected by the SETBYLAYER commands. The value is stored as a bitcode using the sum of the values of all selected options.

- 1: Color
- 2: Linetype
- 4: Lineweight
- 8: Material
- 16: Plot style
- 32: ByBlock
- 64: Blocks

- 128: Transparency

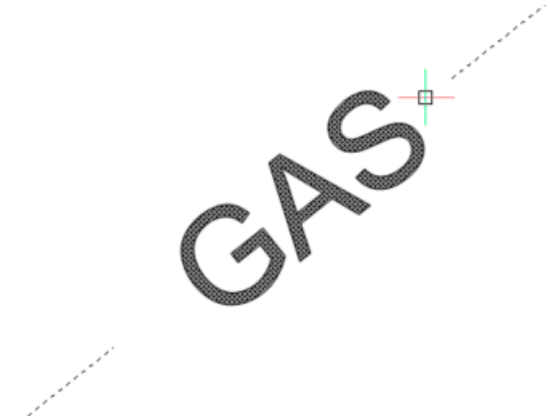
**Layer properties by Viewport**

The VPLAYER command is enhanced in 21 allowing you to select entities on layers for which you want to apply viewport overrides. After selecting which layer property you want to modify for the viewport, you can type the name of the layer or press enter to specify the layer by selecting an entity on that layer.




### 1.3.7 Linetypes

BricsCAD® now supports selection of linetype gaps with the addition of the LTGAPSELECTION system variable. With LTGAPSELECTION enabled, entities are highlighted and selected even when you pick in the gap.

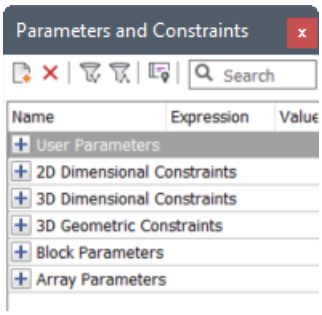



### 1.3.8 Parameters & Constraints

BricsCAD® V21 offers powerful enhancements for creating parametric drawings and block definitions.

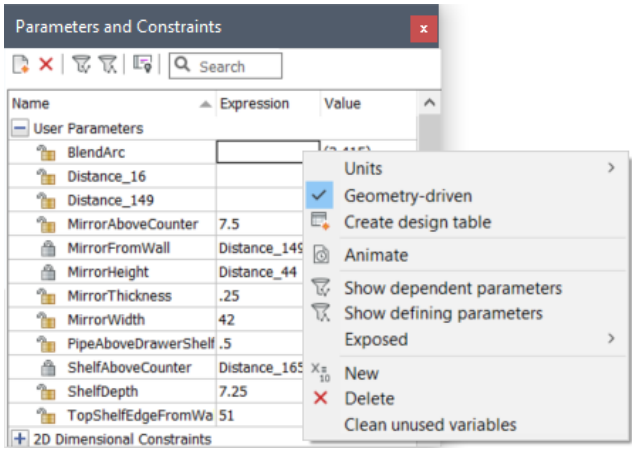
**Parameters & constraints panel**

All BricsCAD® license levels now support 2D parameters and constraints. View and modify parameters and constraints in the Parameters and constraints panel. The panel is extended to display 3D geometric constraints, 2D dimensional constraints, parametric blocks, and arrays.




The right-click menu for parameters and constraints enables you to specify if parameters are geometry-driven. Expressions for such parameters cannot be edited (you can switch geometry-driven status off to

edit them) – instead they are automatically computed from 3D geometry via 3D dimensional constraints that depend, directly or indirectly, on geometry-driven parameters. Simple arithmetic expressions for such 3D dimensional constraints are solved in dynamics when you change your geometry with direct editing tools such as DMPUSHPULL or DMMOVE/DMROTATE.

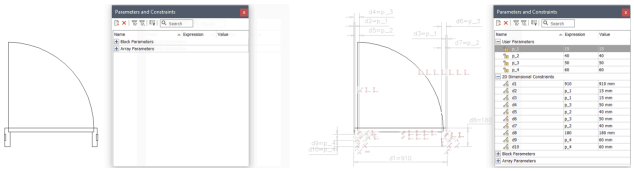



The right-click menu also enables you to control if a parameter is exposed for editing when inserted as a block.

If you open a 3D parametric model with a BricsCAD® Lite license, 3D constraints are shown in magenta. You can view and edit their expressions with BricsCAD® Lite. However, the 3D model only reflects the changes after it's opened with a BricsCAD® Pro license or above.

**Automatic constrain**

The new AUTOCONSTRAIN command enables you to automatically apply 2D geometric and dimensional constraints to selected 2D geometry. BricsCAD® fully constraints the geometry, ensuring not to over-constrain it. The geometric and dimensional constraints are displayed on the geometry. You can view, add, remove, and modify the parameters and constraints in the Parameters and Constraints panel.

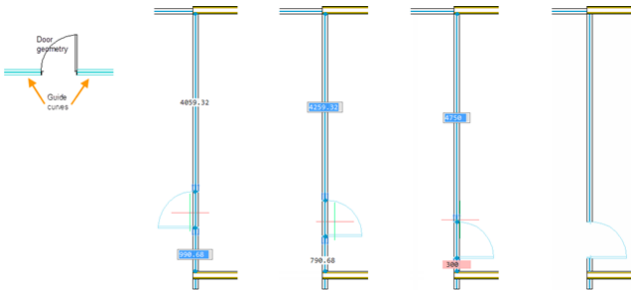


**Reference curves**

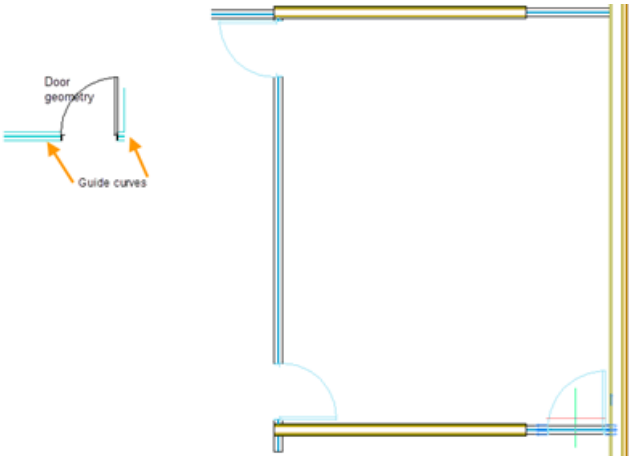
The new REFERENCECURVES command enables you to specify reference geometry for blocks. When you select geometry to use for reference, BricsCAD® creates a REFERENCE_CURVES layer (if it doesn't already exist) and adds the selected entities to that layer.

With reference curves specified in a drawing or block definition, you can automatically align the block or drawing to relevant geometry when you insert it. The number of reference curves and the distance between them determines with which geometry it can align. As the cursor approaches relevant geometry, the block can automatically flip, offering multiple insertion options. Distances between the ends of the

relevant geometry and block are displayed, enabling you to enter specific values if you wish. And, if the reference curves include gaps, relevant geometry is automatically trimmed to produce matching gaps.



You can also use reference curves to automatically align with corners. The following example includes 4 parallel reference curves and a single corner reference curve enabling the door block to align with geometry that matches the four parallel lines and single perpendicular line.



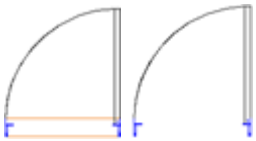

**Visibility states**

The new VISIBILITYSTATES command enables you to define visibility parameters and states. You can create multiple visibility parameters in a drawing or block definition. And, you can create multiple visibility states in each visibility parameter. When you insert the drawing or block, you can change the visibility of each instance. For example, you might create a door block with two visibility parameters, SwingAngle and Header.

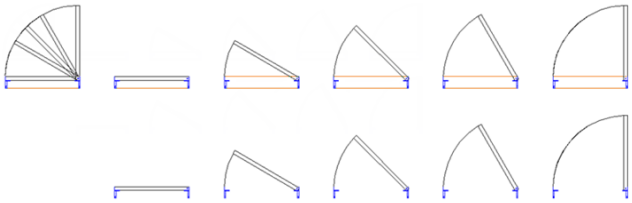
The SwingAngle parameter includes 5 visibility states: Closed, 30, 45, 60, and 90.



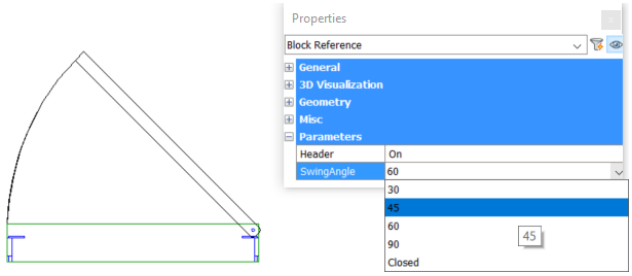
The Header parameter includes two visibility states: **On** and **Off**.



You create all the geometry for each of the visibility states and then select the appropriate geometry to display in each state of each parameter. Geometry that doesn't belong to a visibility state is always displayed. This door example enables 10 variations of block insertions.

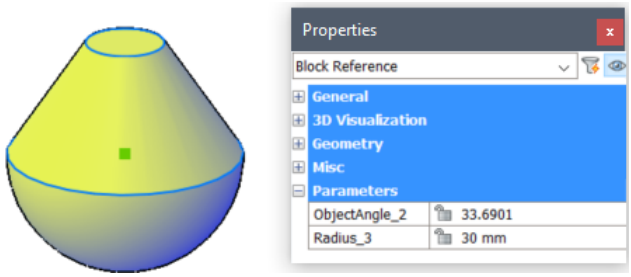


You can easily change the visibility states for a selected block insertion using the Properties panel.




**3D Radius constraints**

The DMRADIUS3D command enables you to create geometry-driven 3D radius constraints to measure the radius of entities inside blocks.

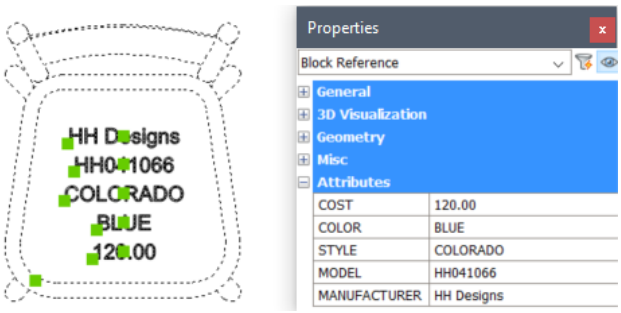



## 1.3.9 Blocks

BricsCAD® V21 offers many enhancements to increase your productivity when working with blocks.

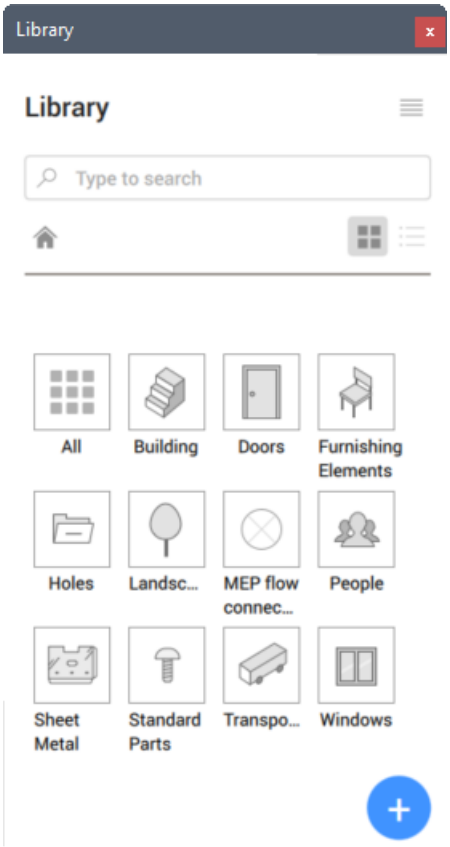
**Block attribute properties**

You can now access block attribute properties in the Properties panel. This allows you to view and edit multiple attributes at the same time. Press the Ctrl key to select one or more attributes within block instances. Open the Properties panel to display the Attributes entity type with all the relevant attribute properties.

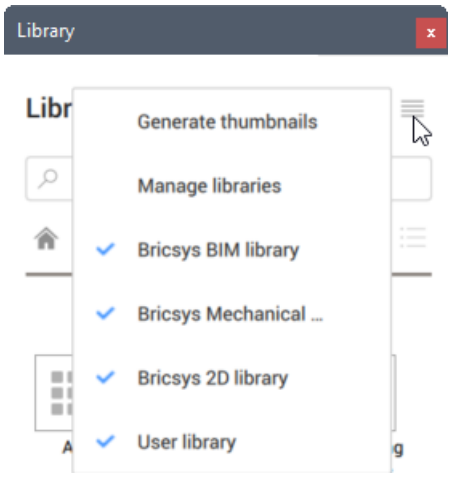



**Library panel**

The new Library panel offers a central location to access 2D and 3D block libraries. It replaces both the Components and the 2D Parametric Blocks panels from previous releases.

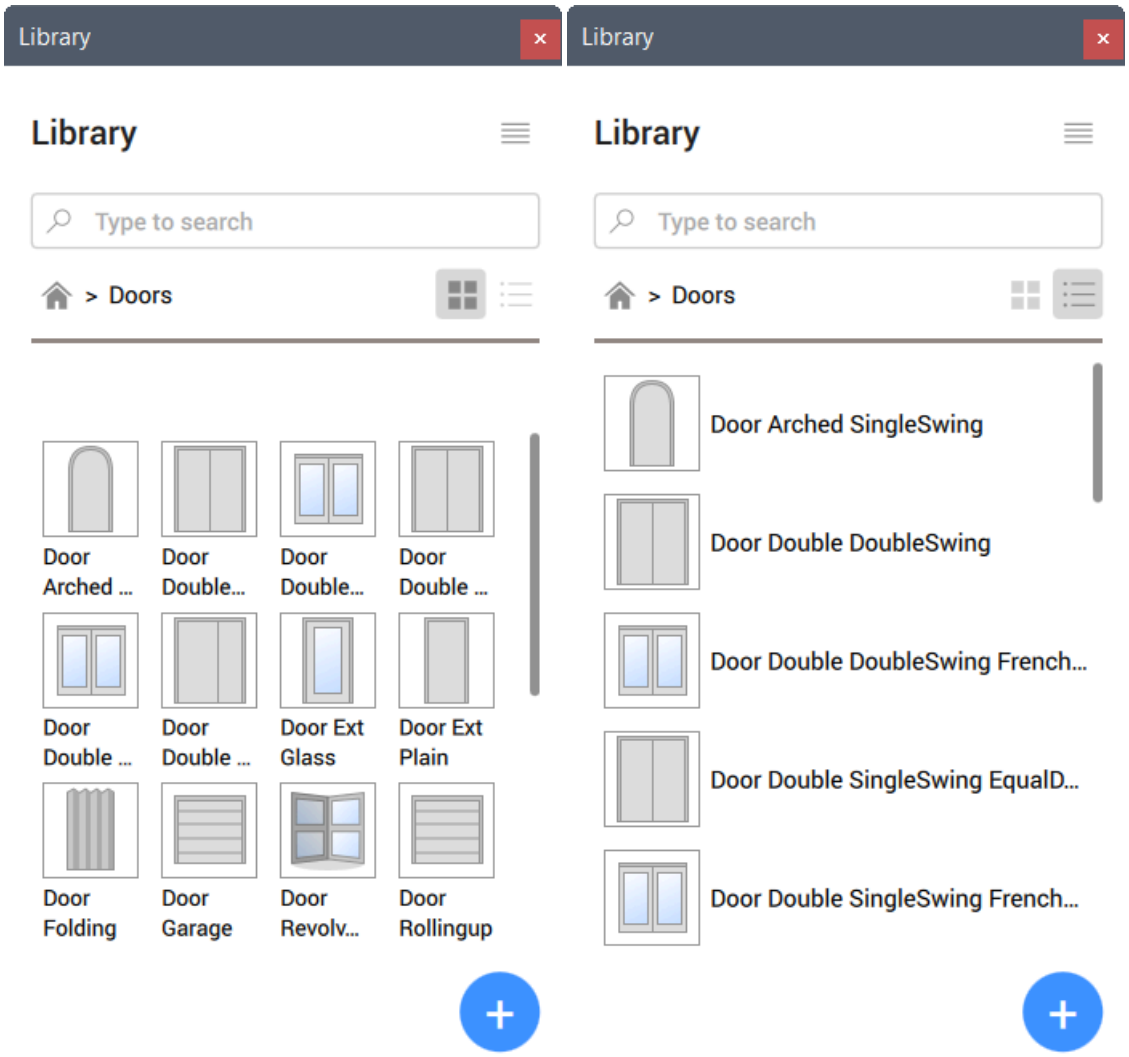



A menu in the upper right corner allows you to control which content is displayed in the Library panel. It may include pre-defined content from Bricsys libraries as well as the User Defined library with your own content.

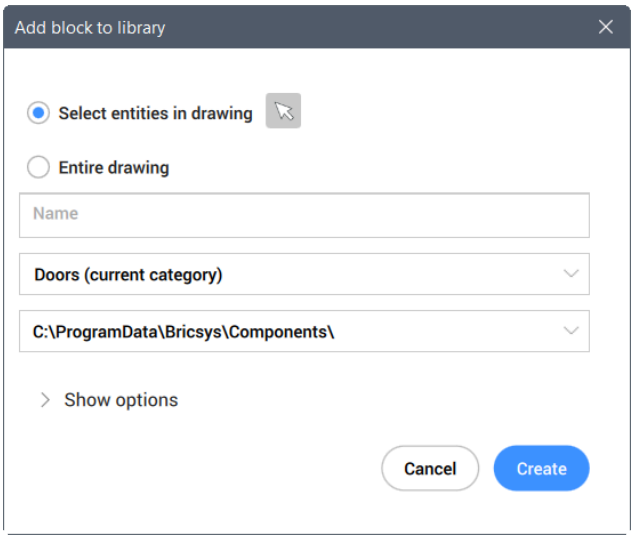



You can specify where BricsCAD® looks for library content by selecting the Manage libraries option. It offers quick access to the COMPONENTSPATH setting so you can view and modify library folder locations. Another menu option automatically generates thumbnail images for any that are missing.

You can search for content by typing key words in the Search box or expand a category to see all the content within that category. And you can display the content in a grid or list view.




You can easily add new block definitions to the library using the Add button at the bottom of the Library panel. Select specific entities from the drawing or use the entire drawing.




**Block icons**

The new BLOCKICON command creates or updates thumbnail images that enable you to preview blocks with various tools including Drawing Explorer and the Library panel.

**Non-uniformly scaled blocks**

The EXPLODE command and the XPLODE command now honor the EXPLMODE system variable when exploding non-uniformly scaled blocks.

**Insert**

The INSERT command is enhanced to support parametric blocks. Block definitions containing parametric entities with exposed parameters can now be parametrically changed after insertion by the INSERT Command. To insert parametric BIM and Mechanical components, use the BMINSERT command.

### 1.3.10 Drawing Optimizations

A new Drawing optimizations ribbon panel offers tools to clean up and optimize your drawings.

**Simplify**

The new SIMPLIFY command reduces the number of vertices of polylines and hatch boundaries without changing their general shape. Options allow you to switch between simplifying and smoothening. Additional options enable you to control the amount of deviation from the original entity. Simplified entities are easier to manipulate and can significantly reduce file size.

**Purge**

The PURGE command is enhanced so the Batch, All, and Orphaned data options remove more items from the drawing.

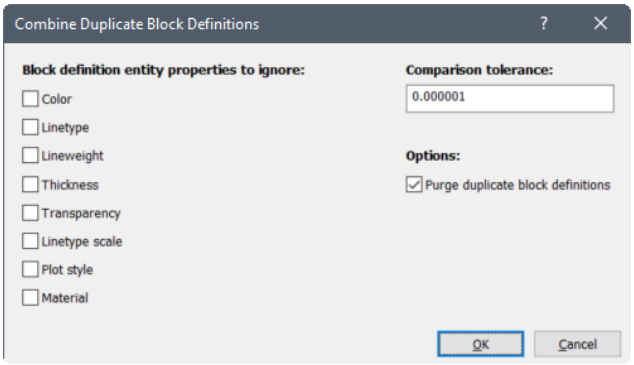
**Audit**

The AUDIT command is enhanced to clean up duplicate ACIS attributes. A combination of AUDIT and CLEANUNUSEDVARIABLES now cleans a drawing from multiple orphan parameters.

**Delete Duplicate Entities**

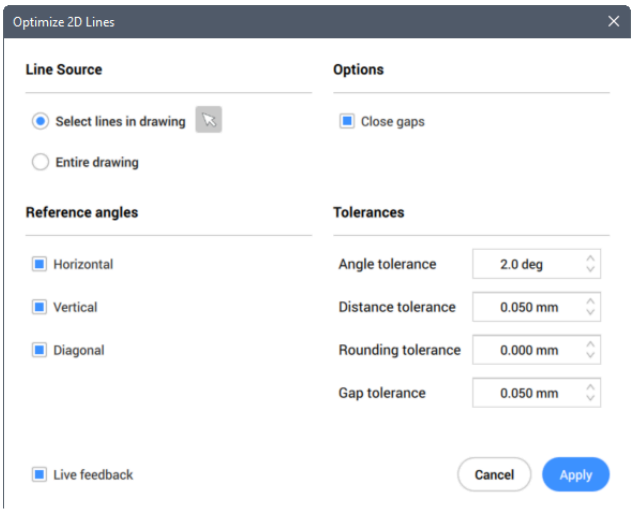
The OVERKILL command enables you to choose if you want to delete duplicate entities or move them to a specified layer. You can specify the layer name using the new OVERKILLLAYER setting or leave the default name, Duplicate Entities.

A new option in the OVERKILL command allows you to combine duplicate block definitions. Choosing this option displays the Combine duplicate blocks dialog box where you can specify any block properties you want to ignore. You can specify how closely the geometry must match by entering a comparison tolerance. If BricsCAD® finds more than one block definition whose geometry meets the tolerance and properties criteria, it uses the most recent definition to replace any instances of the older definitions. The older definitions become unused and you can enable the Purge duplicate blocks option to automatically purge them.

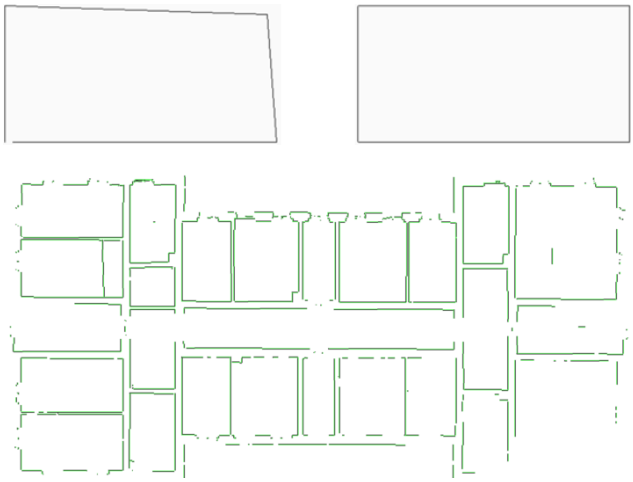



### Optimize

The new OPTIMIZE command helps you clean up 2D lines in your drawings. It displays the Optimize 2D Lines dialog box where you can control which lines are affected. Specify which lines BricsCAD® should consider for optimization. You can select specific line entities or have BricsCAD® analyze and optimize all line entities in the drawing. It can find and fix lines that are within a specified angle tolerance relative to horizontal, vertical, or 45 degrees, in the current UCS. An additional option automatically closes gaps within a specified tolerance.

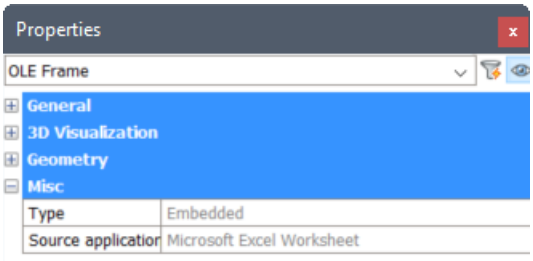


The Optimize tool is useful for cleaning up any 2D lines. For example, you can use it to quickly clean up line geometry extracted from point clouds.



### 1.3.11 Object linking and embedding

The Properties panel for OLE objects is enhanced to provide additional information for the selected OLE entity. The additional information may include Type, such as Embedded or Static, Plot quality, and Source application.

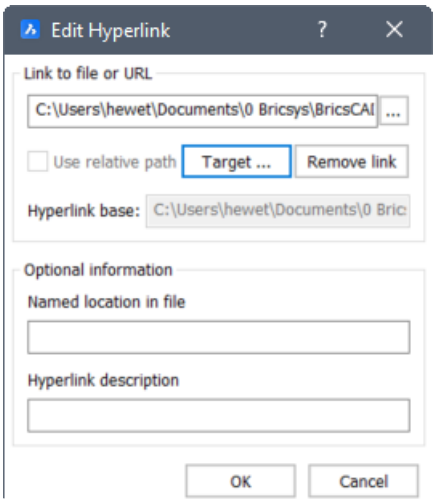



### 1.3.12 Images

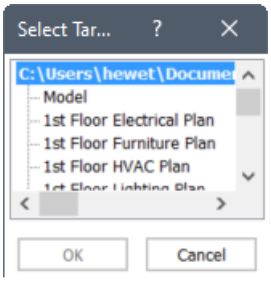
Image manipulation is enhanced in BricsCAD® V21. When you manipulate an attached image using grips and other editing tools the image content displays and updates as you edit the image frame. This enables you to position, align, and scale the image using the image content for reference. The selected image temporarily displays above any overlapping objects as you edit it.

### 1.3.13 Hyperlinks

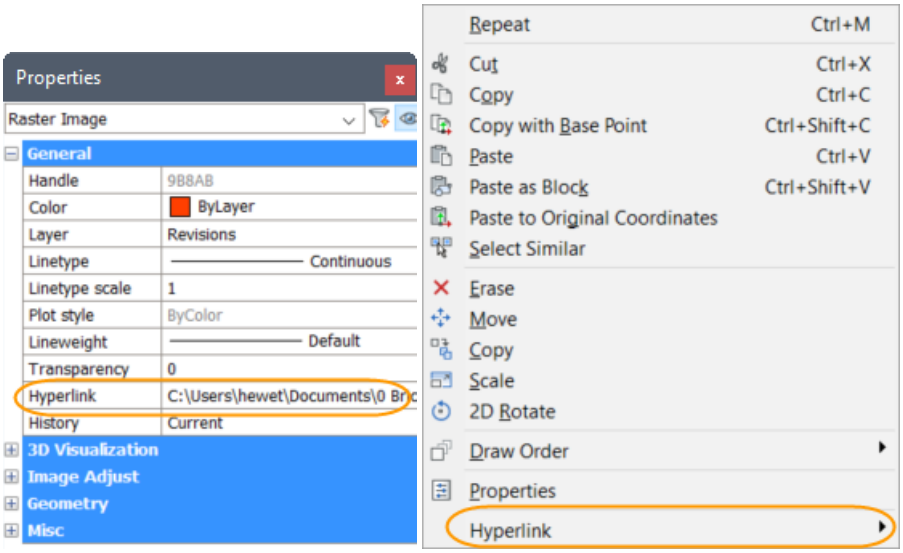
The Edit hyperlink dialog box now enables you to link to layouts and views inside a drawing. After selecting the drawing to which you want to link, choose **Target** to open the Select Target View dialog box.




The Select target view dialog box displays a list of all named views and layouts in the linked drawing.




To access a hyperlink, double-click on the hyperlink in the Properties panel for the selected entity or press and hold the right mouse button and choose **Hyperlink** from the menu.

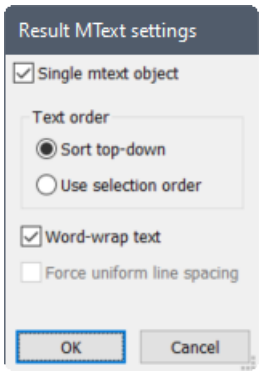



## 1.4 Drafting

BricsCAD® V21 offers drafting enhancements to help increase your 2D drawing productivity.

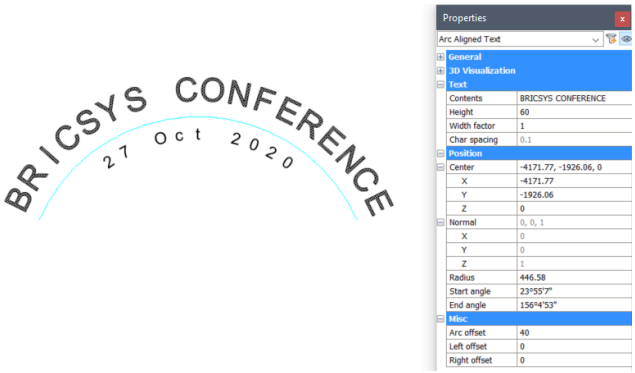
### 1.4.1 Combine Text to Mtext.

The new TXT2MTXT command (alias COMBINETEXT) enables you to combine single- and multi-line text into a single MText entity. You can control the behavior for combining text in the Result MText Settings dialog box.



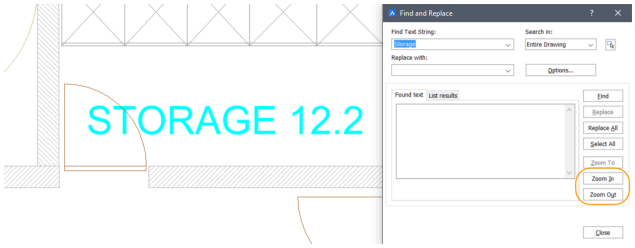

### 1.4.2 Arc Aligned Text

The Properties panel enables you to view and edit properties for selected Arc Aligned Text entities.



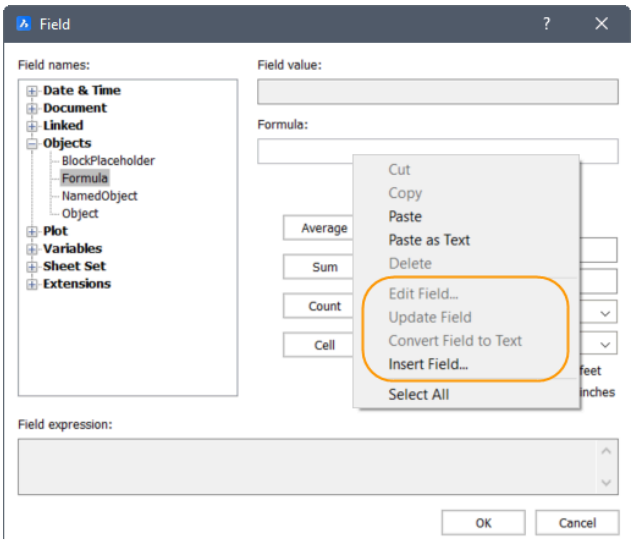

### 1.4.3 Find and Replace Text

The Find and Replace Text dialog box includes new buttons **Zoom In** and **Zoom Out**buttons. These enable you to take a closer look at the text within the context of the drawing without having to end the Find and Replace operation.




### 1.4.4 Field

The Field dialog box supports nested formula fields. When you right-click in the Formula field of the Field dialog box, new options enable you to Insert, Edit and Update a field. You can also convert a field to text.




## 1.5 Modeling

BricsCAD® V21 offers many enhancements to 3D modeling tools to help simplify and automate the modeling process.

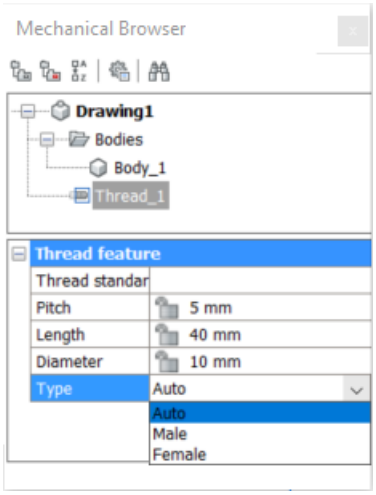
### 1.5.1 Dynamic UCS

Dynamic UCS is now supported by the PLAN command. This allows you to select a face for the current UCS and immediately display it in plan view.

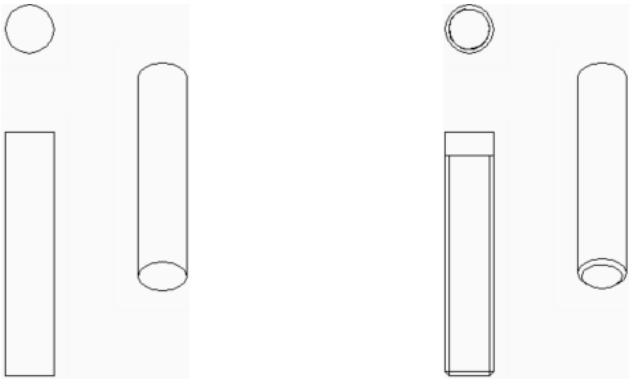
### 1.5.2 Direct modeling

#### Threads

The new DMTHREAD command enables you to add a standard parametric thread to a cylindrical face of your 3D solid. You can edit the properties of the thread using the Mechanical Browser.

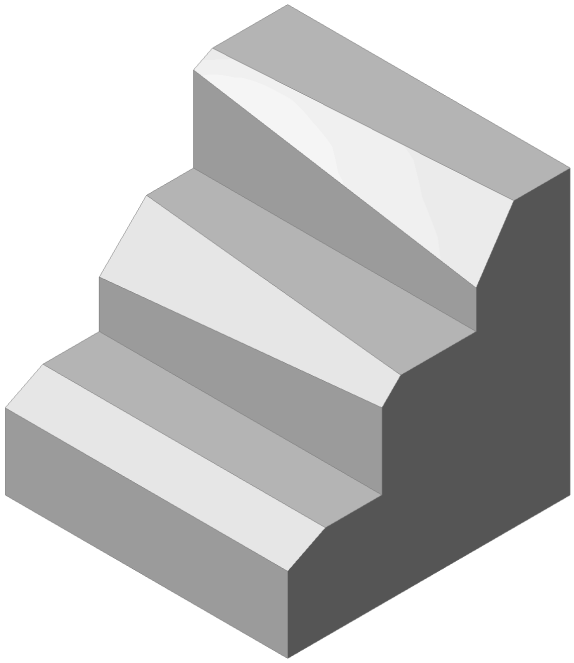


Drawing views created with the VIEWBASE command, the VIEWPROJ command or the VIEWSECTION command represent the threads following drafting standards. The following image show views of a cylinder before and after adding threads.



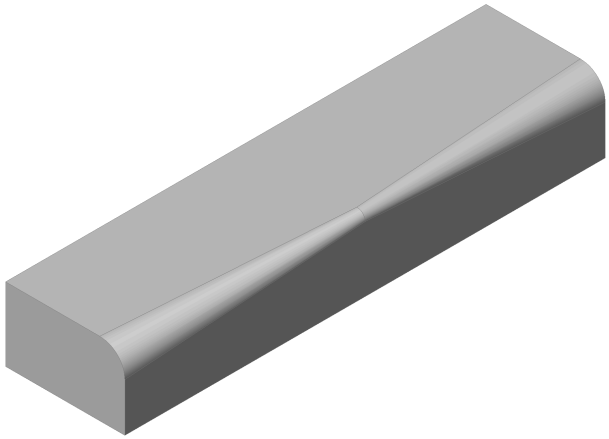
**Chamfers**

The DMCHAMFER command offers more flexibility with new asymmetrical, angular, variable symmetrical, variable asymmetrical and variable angular modes.



**Fillets**

The DMFILLET command offers more flexibility with a new variable radius mode.



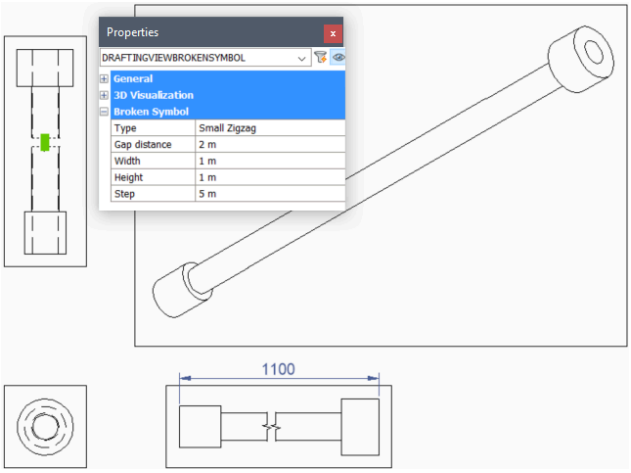
**Push Pull**

The DMPUSHPULL command includes an option to specify subtract mode (on/off), which is also controlled by DMPUSHPULLSUBTRACT system variable.

It also allows you to switch the starting point of the dynamic dimensions on cylindrical faces. Press the Tab key to switch between radial and offset dimensions.

### 1.5.3 Broken views

The new VIEWBREAK command enables you to break an associative drawing view of your 3D model created with the VIEWBASE command, the VIEWPROJ command, or the VIEWSECTION command and automatically include break symbols. You can change the appearance of a selected broken view symbol using the Properties panel. The VIEWBREAK command correctly supports dimensions you put on the drawing view before breaking. They remain associative with your 3D model and the drawing view.

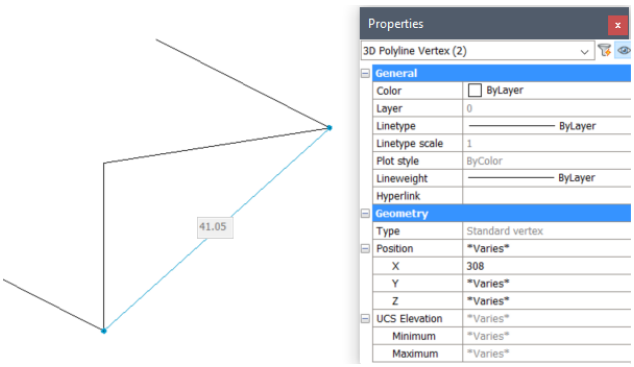



### 1.5.4 3D Entity snaps

Easily toggle 3D entity snaps on and off by pressing the F4 function key.

### 1.5.5 3D Polyline

The vertices of 3D polylines now support sub-entity selection, the Properties panel, and nearest distance.




Additional enhancements to 3D Polylines enable you to apply 3D constraints to their vertices and edges.

### 1.5.6 Parametric blockify

The new PARAMETRICBLOCKIFY command enables you to quickly convert model entities to parametric blocks. Select a parametric block or parametrized entities as input geometry to compare against other geometry in the drawing. Matching geometry is automatically converted to parametric blocks.

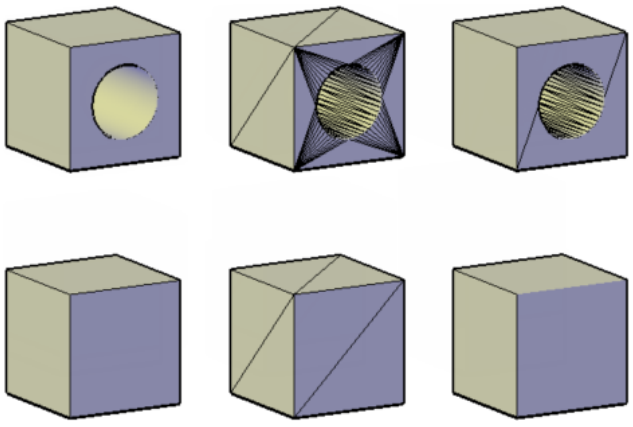
### 1.5.7 3D display

Material glossiness has been adjusted to ensure entity details are distinguishable, even with glossy materials. The glossy effect is less intense for light materials and more intense for dark materials.

### 1.5.8 Solids and meshes

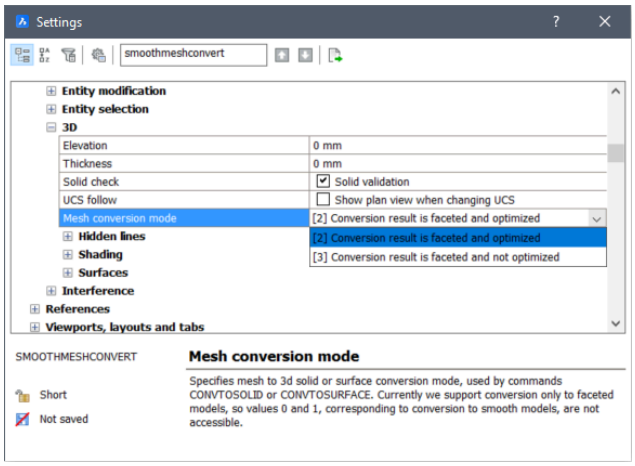
**Convert to mesh**

The CONVTOMESH command is optimized to triangulate planar faces only when necessary and with as few faces as possible. As an example, the original solids on the left are converted to meshes in BricsCAD® V20 (center) and BricsCAD® V21 (right).

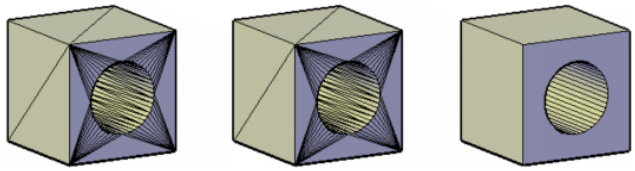


**Convert to solid**

The CONVTOSOLID command enables you to convert a mesh to an optimized 3D solid based on the new SMOOTHMESHCONVERT system variable.

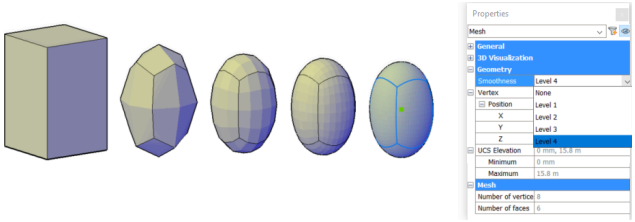



When set to optimized, coplanar and adjacent faces are merged into a single face. As an example, the original mesh on the left is converted to a solid in BricsCAD® V20 (center) and BricsCAD® V21 (right).

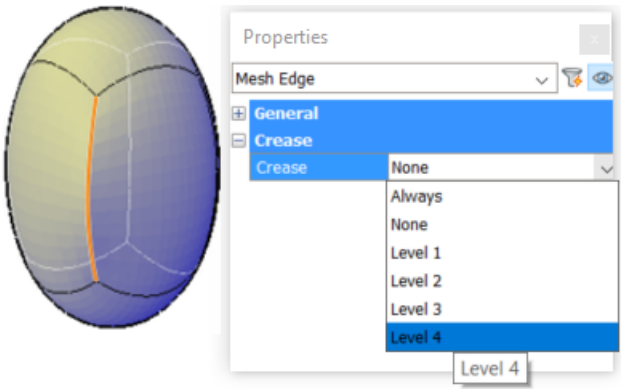


**Mesh properties**

The Properties panel offers access to additional properties for selected mesh entities. Mesh entities now include Mesh and Geometry properties including the ability to specify smoothness.



The Properties panel also supports mesh sub-entities including faces, edges and vertices enabling you to apply different levels of creasing.



### 1.5.9 Associative data

The new GENERATEASSOCATTRS setting controls generation of associative data on 3D solids during modeling. When set to On, 3D solids contain associative data from the moment of creation. This allows BIMSECTIONUPDATE command and VIEWBASE command commands to produce drawings for which dimensions and tags can be automatically updated after the 3D model changes.

### 1.5.10 View Twist

The Twist option is added to the -VIEW command.

### 1.5.11 Animation Editor

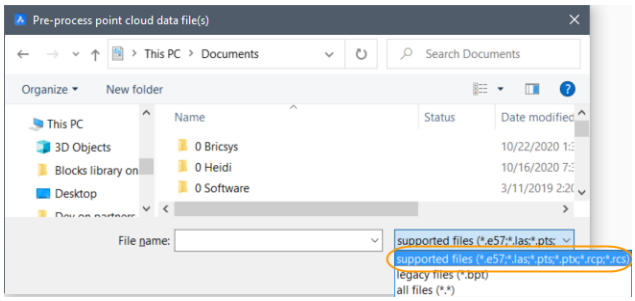
The Animation Editor is enhanced to support the animating of exploded views in model space.

### 1.5.12 Point clouds

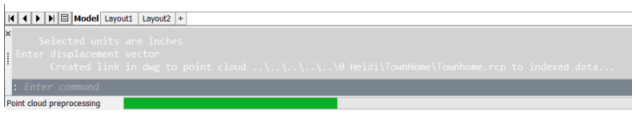
Point cloud functionality is significantly faster and more powerful in BricsCAD® V21.

**Pre-processing**

The E57 file format is added to the supported files for pre-processing.



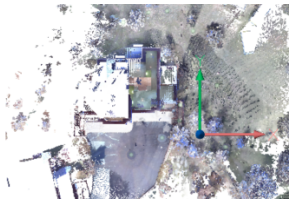

When you attach a point cloud that requires pre-processing, it converts 5 to 8 times faster in BricsCAD® V21 than in V20. A progress meter at the bottom of the display shows the conversion progress.




**Alignment**

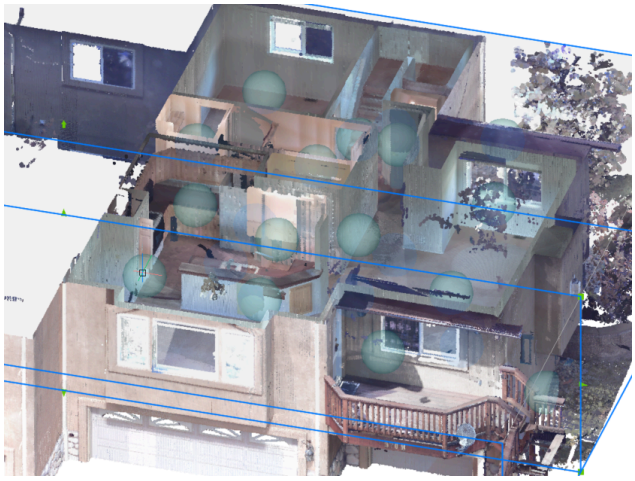
The new POINTCLOUDALIGN command automatically rotates a point cloud to optimally align it with the X and Y axis. To determine the best alignment it can analyze the entire point cloud or you can specify the most relevant area.





**Bubble Viewer**

The point cloud displays bubbles at all the scan locations. It's at those locations where you will experience the most realistic visual representations.



Double-click one of the bubbles in model space to open the Bubble viewer. You can press the middle mouse button and move the mouse to view the point cloud in any direction from that scan location.





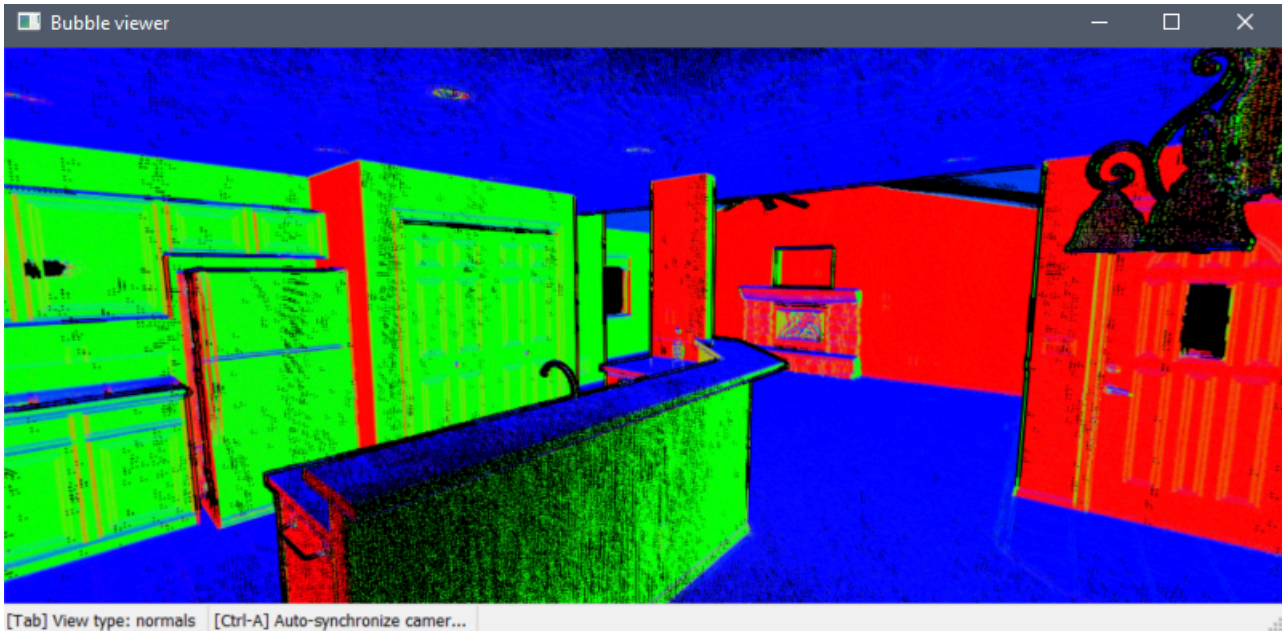


Press the Tab key to cycle between three different visual modes.

The first mode displays the points as their actual colors or in grayscale, depending how the data was scanned.



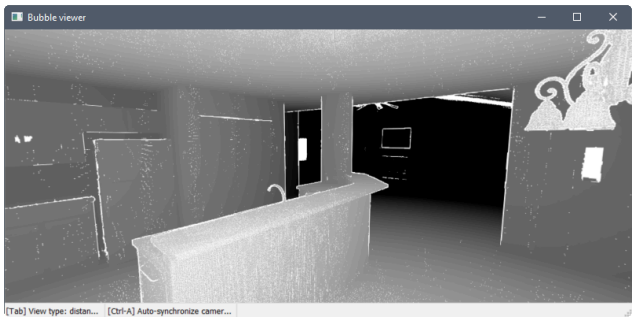

The second mode displays the points as red, green, or blue according to their normal vectors. The colors correspond to the UCS axes.







The third mode displays the points from light to dark as the distance from the scan location increases.



You can easily sync the drawing view to match the Bubble viewer by pressing Ctrl A.



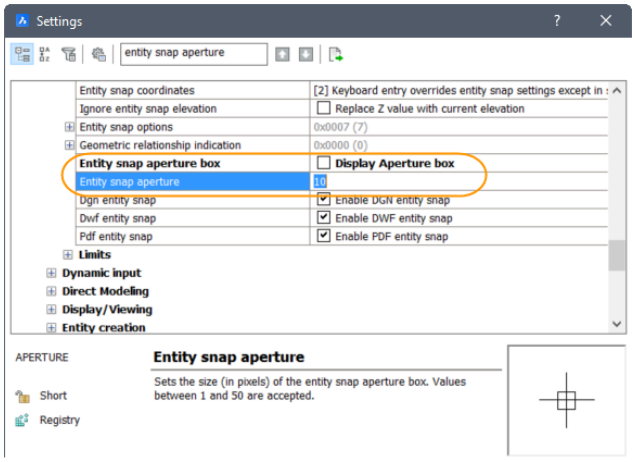
**Entity snaps**

The new Point Cloud Closest Point entity snap significantly improves your ability to select relevant point cloud points. It uses an imaginary cylinder from the current viewpoint toward the cursor.

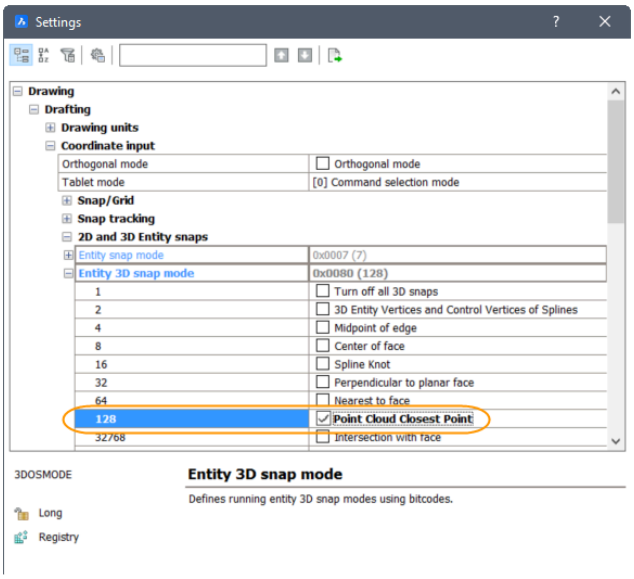




The radius of the imaginary cylinder is defined by the Entity snap aperture box setting.




Enable the new Point Cloud Closest Point entity snap along with other 3D entity snaps in the Entity Snap menus, toolbar, and settings.




**Export**

The new POINTCLOUDEXPORT command allows you to export a cropped selection of a point cloud to a .pts file.

## 1.6 Civil tools

BricsCAD® V21 offers many advanced tools for linear infrastructure modeling.

### 1.6.1 Import/Export

New and enhanced tools for importing and exporting civil data help accelerate collaboration and design processes.

**Civil 3D Data**

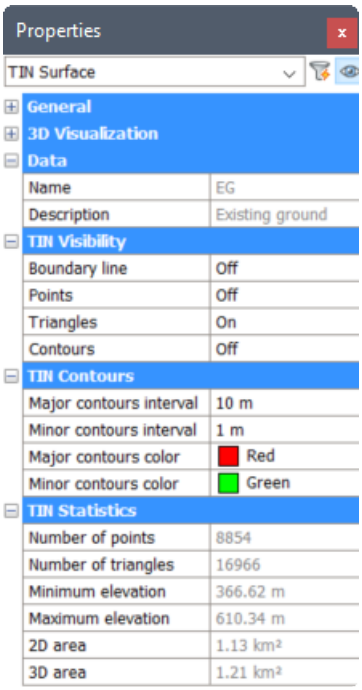
The new CIVIL3DIMPORT command enables you to create multiple civil entities in your BricsCAD® drawing, from an external Autodesk® Civil 3D drawing, in one operation.

**LandXML Data**

The LANDXMLIMPORT command and LANDXMLEXPORT command tools offer more flexibility. They support new alignment curves including spiral-curve-spiral combinations and parabolas for vertical alignments. Imported vertical alignments automatically create 3D Alignments. And, when importing a surface with breaklines, a new option allows you to specify whether the breaklines should be drawn as 3D polylines or not drawn at all.

### 1.6.2 Civil Entity Properties

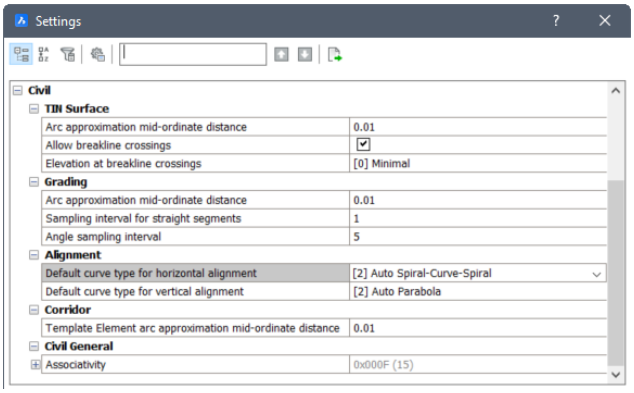
All Civil entities now include Name and Description properties in the Properties panel. Name and description are in the read-only mode in the Properties panel. They can be edited with the API.




Selected sub-entities of (Civil) custom entities are also displayed in the Properties panel.

### 1.6.3 Civil settings

A new Civil node is included in the Settings dialog box for easy access to Civil settings.




### 1.6.4 TIN Surfaces

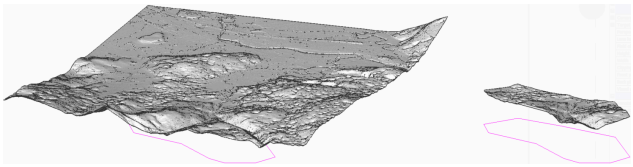
New and enhanced tools for creating and editing TIN surfaces provide more flexibility and a faster workflow.

**Associativity**

Points and blocks are dynamically linked to the TIN surface. If a TIN surface is created from points or blocks in the drawing and some points/blocks are moved, the TIN surface automatically updates to reflect the changes.

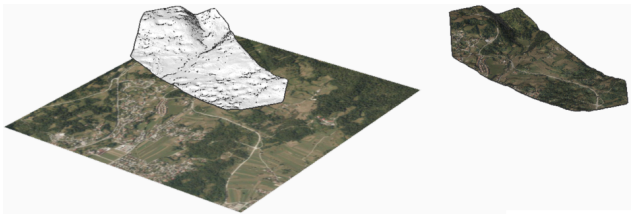
**Clipping**

A new option in the TIN command and TINEDIT command enables you to clip a TIN surface according to a given polyline boundary.



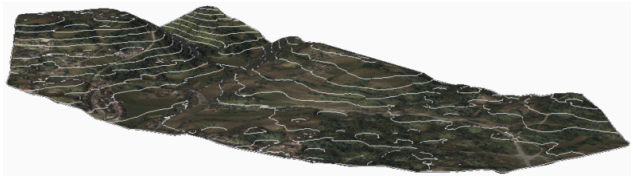
**Raster Images**

The new TINASSIGNIMAGE command enables you to assign a raster image as a TIN Surface material to create a realistic site model.



**Extract**

The TINEXTRACT command enables you to extract faces, points, contours, and the border from a TIN surface, in addition to the previous ability to extract a mesh or solid.



You may also extract a TIN surface from a Grading or TIN Volume Surface using the TINEXTRACT command.

**Project**

The new TINPROJECT command enables you to project point-based entities including points, blocks, and text or linear entities, including lines, polylines, and circles, to a TIN Surface.





**Water Path**

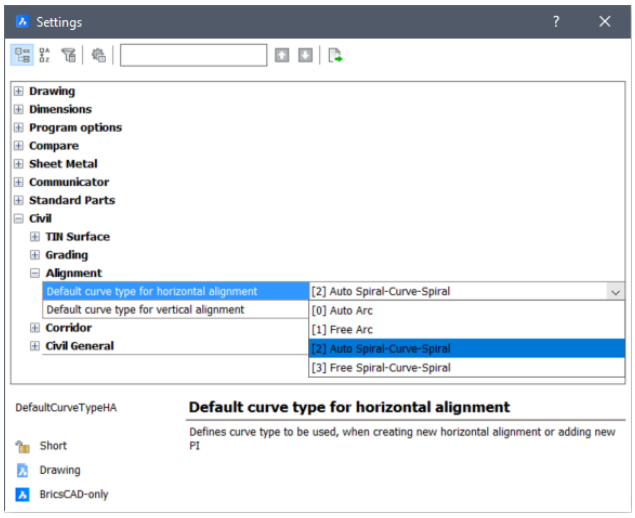
The new TINWATERDROP command enables you to draw a water drop's path from a point on a TIN Surface.

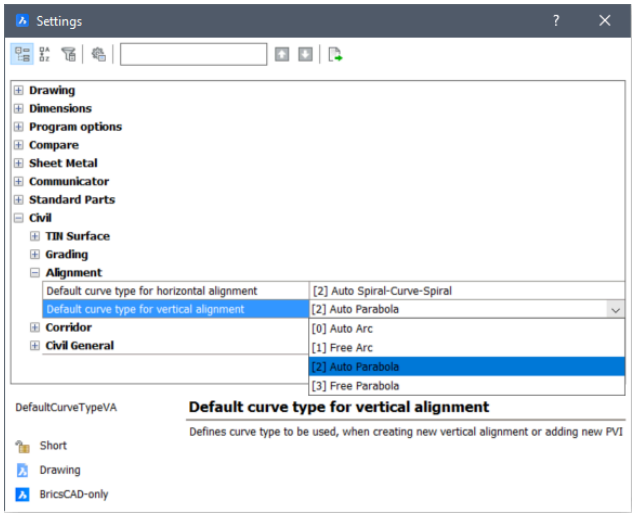


### 1.6.5 Alignments

The workflow for creating alignments is simplified while still offering powerful functionality and additional flexibility.

Horizontal and vertical alignment geometry is enhanced. Horizontal alignments now support spirals and vertical alignments now support parabolic curves. The new DEFAULTCURVETYPEHA and DEFAULTCURVETYPEVA alignment settings enable you to specify the default curve type for creating new horizontal and vertical alignments.



The Properties panel displays properties for Spiral-Curve-Spiral or parabola elements.

### 1.6.6 Corridors

Powerful corridor tools help you create a 3D model of linear infrastructure defined by a template and 3D alignment.

The new CORRIDORTEMPLATEELEMENT command enables you to create a corridor template element from closed polylines.



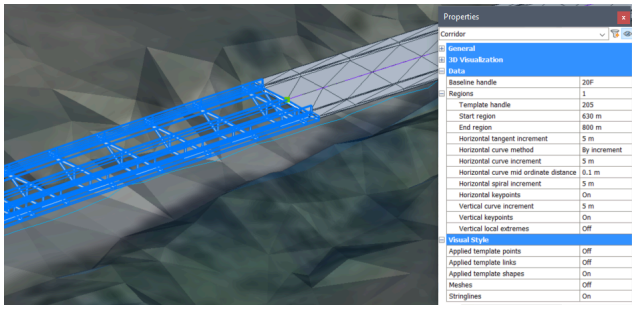
Then use the new CORRIDORTEMPLATE command to create a corridor template from template elements.



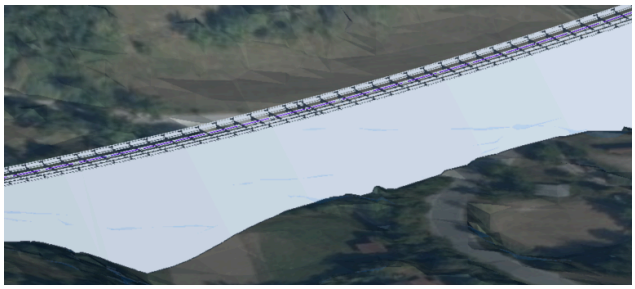
Finally, use the new CORRIDOR command to create a corridor from a corridor template, 3D alignment and/or TIN Surface. You can create multiple regions using different corridor templates. And, you can add or remove regions using the new CORRIDOREDIT command.



Use the Properties panel to modify data and visual style properties of the selected region.




The new CORRIDOREXTRACT command enables you to extract a 3D solid, 3D mesh, or 3D polyline from a corridor to use for various purposes. For example, you can extract polylines from a corridor to use as edges for the grading. If you grip edit the alignment, the corridor and its associated geometry, such as the grading, automatically follows.



### 1.6.7 Gradings

The new GRADINGBALANCE command enables you to balance grading cut and fill volumes.

### 1.6.8 TIN Volume Surfaces

The ARRAYASSOCIATIVITY system variable includes a new control for TIN Volume Surface associativity. TIN volume surfaces automatically rebuild when their source surfaces change.

### 1.6.9 API Samples

The following API samples are provided: Horizontal alignment report, Color surface triangles by elevation, Color surface triangles by slope.

## 1.7 Performance and compatibility

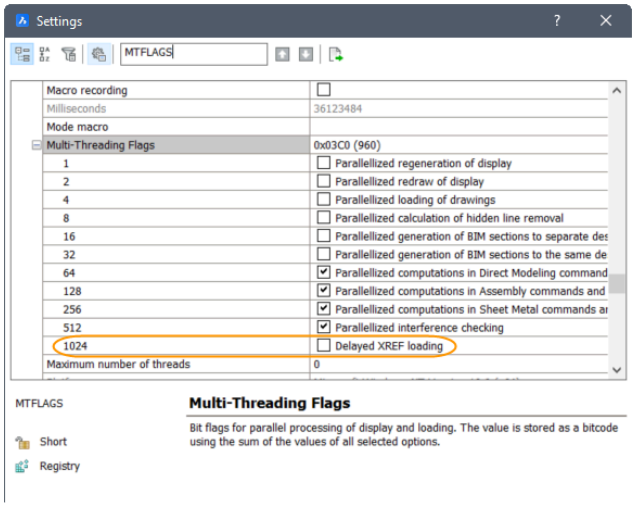
BricsCAD® V21 offers new and enhanced tools to improve performance and compatibility as you work individually or in a collaborative environment.

### 1.7.1 Selection Preview

Selection preview highlighting is deactivated during Pan operations, resulting in a smoother user-experience.

### 1.7.2 Xref Loading

A new MTFLAG is added to delay the loading of Xrefs to idle time.



### 1.7.3 Multiuser Version Control (BETA)

BricsCAD® V21 introduces new multi-user version control functionality as a beta technology preview. This functionality enables multiple users to work on the same drawings, simultaneously. Drawings are stored in the cloud using Bricsys 24/7 and checked in and out from each user's local machine.

**Access**

Since this is Beta functionality, you must request access by submitting a support request.

1 Log into your Bricsys account.

2 Choose New Support Request.

3 Open the BricsCAD® menu and choose **BricsCAD** > **Version Control**.

4 In the Subject field enter: Access to VERSIONCONTROL Beta.

5 Fill in the rest of the fields as needed.

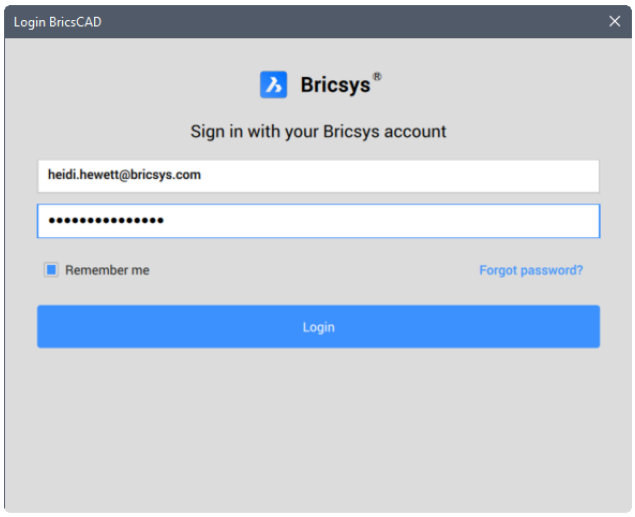
6 Choose Send Support Request.

**VERSIONCONTROL Command**

The new VERSIONCONTROL command offers the following options:

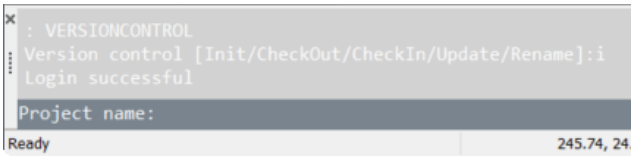
- Init
- Checkout
- Checkin
- Update
- Rename

The first time you launch VERSIONCONTROL in a new BricsCAD® session and choose one of the options, it prompts you to log into your Bricsys account.




If your account has not been granted access to the beta functionality, it directs you to a web page with instructions to request access.

If your account has access to the beta functionality, your successful login is indicated in the Command window and it prompts you to enter the name of the project on which you want to work.




**Init**

The **Init** option enables you to create a new multi-user project. Enter a new project name, keeping in mind project names are case-sensitive. If you enter a name that already exists, it prompts you to enter a different name.

**Checkout**

The **Checkout** option enables you to check out a drawing from your cloud-based projects. You must specify from which project you want to check out a drawing. Enter the project name, keeping in mind project names are case-sensitive. If you enter a project name that doesn't exist, BricsCAD® displays a list of valid project names and prompts you to reenter the project name.

After entering a valid project name, you must specify a version control folder for that project. The version control folder is the location on your local machine where your drawings are stored and synced with the cloud. The default location is *drive:\Users\username\Documents\Bricsys247\projectname*. However, you can specify any location. If the location doesn't exist, it's created and set under version control. If the location already exists it's set under version control. If the location is already under version control, you are prompted to specify a different location.

**Checkin**

The **Checkin** option enables you to check in drawings from your local, version control folder to the cloud-based project. You can check in all files in the version control folder or you can commit modified and untracked files.

After specifying what to check in, you can enter a relevant Check In message.

**Update**

The **Update** option enables you to sync project drawings from your local folder with the cloud.

**Rename**

The **Rename** option enables you to rename a drawing that's under version control.

### 1.7.4 DGN Export

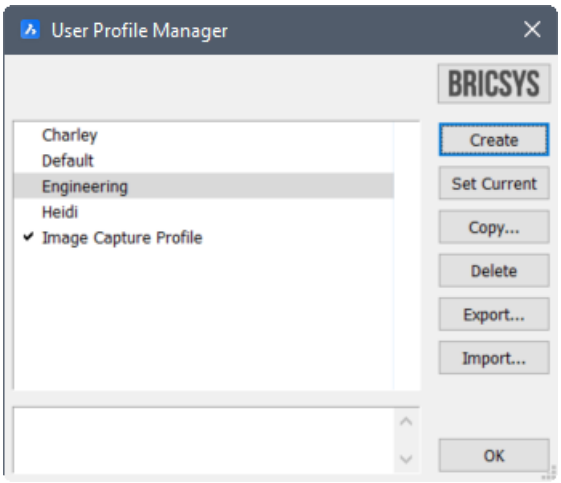
A new DGNEXPORT command enables you to export the current drawing to DGN file format.

### 1.7.5 3D DWF Output

The 3DDWF command enables you to create 3D DWF or 3D DWFx files based on the value of the new DWFFORMAT system variable.

### 1.7.6 User Profile Manager

The User Profile Manager is now more powerful. It can now delete or replace existing user profiles, including the current one. Deleting the current profile doesn't completely remove it, but it does reset all settings to factory defaults.




### 1.7.7 Start Menu Shortcuts

Start menu shortcuts now set the program folder instead of personal documents folder as the initial working directory. This change solves problems with installation under a system account where personal folders are not available.

# 2. Установка и лицензирование

## 2.1 Установка BricsCAD®

### 2.1.1 Минимальные системные требования и рекомендации

Системные требования и рекомендации для ПК с Microsoft Windows и Linux на базе Intel и AMD, а также для компьютеров Apple Mac на базе Intel.

| | | |
|---|---|---|
| **Поддерживаемые операционные системы** | • Windows 10 (x64)<br>• Windows 8.1 (x64)<br>• Ubuntu V18.04 и выше<br>• openSUSE - сборки после апреля 2018 г.<br>• Fedora - сборки после апреля 2018 г.<br>• Mac OS X 10.13 или более новая | ***Прим.:*** 10 января 2023 года Microsoft завершит программу расширенной поддержки Windows 8.1.<br><br>***Прим.:*** Mac OS X в настоящее время поддерживается только оборудованием на базе Intel. |
| **Устаревшие операционные системы** | • Windows Server 2008 R2<br>• Windows Server 2008 (с SP2 и выше)<br>• Windows 7 (x64) | ***Прим.:*** Мы не рекомендуем запускать критически важное программное обеспечение САПР в неподдерживаемых операционных системах, поскольку ваш компьютер может стать более уязвимым для угроз безопасности.<br><br>• 14 января 2020 года Microsoft прекратила поддержку Windows Server 2008 и 2008 R2.<br>• 14 января 2020 года Microsoft завершила программу расширенной поддержки Windows 7. |

| | | |
|---|---|---|
| **Минимальный процессор:**<br>• Intel® Core™ i5<br>• AMD Ryzen™ 5<br>• 2,5 ГГц или выше | **Рекомендуемый процессор:**<br>• Intel® Core™ i7<br>• Intel® Core™ i9<br>• AMD Ryzen™ 7<br>• AMD Ryzen™ 9<br>• 3.0 ГГц или выше | Самые лучшие результаты **BricsCAD®** показывает при использовании ЦП с наивысшим рейтингом однопоточной производительности.<br>Дополнительные ядра ЦП также могут дать прирост производительности, так как **BricsCAD®** поддерживает концепцию «вспомогательных потоков» для многих задач, связанных с данными и вычислениями.<br>**Примечание о ЦП Intel® Xeon® и AMD Ryzen™ Threadripper™** : эти процессоры верхнего уровня для рабочих станций поддерживают большую внутреннюю кэш-память, больше каналов ввода-вывода PCIe® и многоканальные архитектуры с высокоскоростной памятью. Помогут ли они **BricsCAD®** работать быстрее? Возможно. Вам нужно будет провести тестирование в конкретной среде, чтобы определить, оправдана ли дополнительная стоимость этих систем. |
| **Минимальная системная память:**<br>8 GB | **Рекомендуемая системная память:**<br>• 16 GB<br>• 32 GB | Для максимальной производительности определите конфигурацию памяти вашего ПК и убедитесь, что каждый канал памяти на материнской плате заполнен хотя бы одним модулем памяти (SIMM/DIMM).<br>***Прим.:*** количество доступных каналов памяти может не коррелировать 1:1 с количеством разъемов ОЗУ на материнской плате вашего ПК. Есть вопросы? Перед покупкой системной памяти обратитесь к поставщику оборудования. |

| | | |
|---|---|---|
| **Минимальное дисковое пространство:**<br>2 ГБ для **BricsCAD®** **Ultimate** | **Рекомендуемое дисковое пространство:**<br>(По мере необходимости) | Мы рекомендуем использовать в качестве системных дисков как для **BricsCAD®**, так и для Windows, твердотельные накопители (SSD). При необходимости, чертежи и вспомогательные файлы могут находиться на более медленных механических жестких дисках.<br>***Прим.:*** твердотельные диски могут работать в 10 раз быстрее при чтении и в 20 раз быстрее при записи данных по сравнению с жестким диском среднего класса (например, Seagate Barracuda 5.4k RPM, SATA 6.0). |
| **Минимальная графическая система:**<br>• Экран True Color с разрешением 1920 x 1080 и видеокарта с 1 ГБ видеопамяти<br>• Экраны Apple Mac | **Рекомендуемая графическая система:**<br>• Несколько экранов True Color с разрешением 3840 x 2160 (4k), графическая карта PCIe® с графическим процессором и 4 ГБ видеопамяти<br>• Экраны Apple Mac Retina (4K и 5K) | В ОС Microsoft Windows поддерживается большинство видеокарт® и графических процессоров NVIDIA, AMD и Intel.<br>В ОС Linux поддерживается большинство видеокарт и графических процессоров NVIDIA и AMD.<br>В системах Mac поддерживается собственное графическое оборудование Apple.<br>Безкаркасные режимы визуализации в **BricsCAD®** основаны на технологии RedSDK от Redway3D.®Для максимальной производительности RedSDK в этих режимах требуется графический процессор.<br>**Примечание:** Мы рекомендуем вам установить последние версии драйверов, доступных для вашего графического оборудования, в соответствии с рекомендациями производителя.<br>**Примечание для пользователей Linux:** Аппаратное ускорение 3D графики в системах Linux НЕ поддерживается:<br>• Intel® Графические чипсеты/карты Intel<br>• Ноутбуки с двумя графическими адаптерами.<br>Но последние разработки драйверов NVIDIA в этой области показывают многообещающие перспективы.<br>Дополнительные сведения о поддерживаемом графическом оборудовании для **BricsCAD®** см. в разделе Набор микросхем графического процессора Redway3D. |

### 2.1.2 Загрузка BricsCAD®

#### 2.1.2.1 Загрузка последней версии BricsCAD®

1 Откройте сайт Bricsys, выполните регистрацию и войдите в свою учетную запись.

2 Нажмите **Загрузить BricsCAD**.

3 Выберите платформу, язык и версию (32- или 64-разрядная).

4 Установите флажок **Я согласен с условиями использования**.

5 Нажмите кнопку **Загрузить**.

#### 2.1.2.2 Загрузка предыдущих версий BricsCAD®

1 Откройте сайт Bricsys, выполните регистрацию и войдите в свою учетную запись.

2 Нажмите **Загрузить BricsCAD**.

3 Нажмите ссылку **Предыдущие версии**.

4 Выберите платформу и язык.

5 Выберите версию **BricsCAD®**.

6 Выберите версию (32- или 64-разрядная).

7 Установите флажок **Я согласен с условиями использования**.

8 Нажмите кнопку **Загрузить**.

### 2.1.3 Обзор процедуры установки

- **BricsCAD®** поставляется в виде одного загружаемого файла для всех уровней лицензий, включая **BricsCAD® Shape**.
- При первой установке новой версии **BricsCAD®** вы можете использовать 30-дневный ознакомительный режим.
- Каждая новая основная версия устанавливается отдельно и может использоваться совместно с предыдущими основными версиями BricsCAD.

#### 2.1.3.1 Интерактивная установка

При обычной (неавтоматической) установке пользователь указывает необходимые для установки данные в диалоговых окнах программы-установщика.

Пользовательский интерфейс программы-установщика Windows Installer запрашивает целевую систему и выводит на экран Мастер установки программного обеспечения, который позволяет пользователю изменить различные параметры установки.

#### 2.1.3.2 Автоматическая установка

При автоматической установке пользовательский интерфейс Мастера установки отключен, запросы на ввод данных для установки не отображаются и никакие действия от пользователя не требуются.

### 2.1.3.3 Интерактивная установка

1 Щелкните дважды на файле дистрибутива.

2 Нажмите **Далее**.

3 Подтвердите принятие условий **Лицензионного соглашения**.

4 Нажмите **Далее**.

5 Выполните одно из следующих действий:

- Подтвердите использование папки установки по умолчанию.
- Нажмите кнопку **Изменить...** и выберите другую папку установки.

6 Нажмите **Далее**.

7 Для начала процесса установки нажмите кнопку **Установить**.

8 Выберите, следует ли добавлять ярлык на рабочий стол и/или отображать примечания к устанавливаемой версии программы после завершения процесса установки.

9 Нажмите **Далее**.

10 Нажмите кнопку **Завершить** для завершения процесса установки.

Result: **BricsCAD®** установлен.

### 2.1.3.4 Автоматическая установка (только для Windows)

Для выполнения автоматической установки вам необходимо иметь права Администратора.

1 Откройте окно **PowerShell** или **командную строку** и с помощью команды cd измените путь к текущему каталогу на путь к каталогу, в котором находится .msi файл установщика **BricsCAD®** .

***Прим.:*** В Windows 10 вы можете получить доступ к окну **PowerShell** с помощью контекстного меню, которое отображается при одновременном нажатии клавиши Shift и щелчке правой кнопкой мыши на любой папке в окне **Проводник**. Это позволяет открыть окно **PowerShell** в текущем окне без перехода в другой каталог.

2 Введите **msiexec /i**, затем выполните одно из следующих действий:

- Введите имя файла установщика **BricsCAD®** (например, BricsCAD-V20.2.09-1-ru_RU(x64).msi.
- Перетяните файл установщика **BricsCAD®** из окна Проводника Windows в командную строку.

3 Введите в командной строке **/qn**.

4 (при необходимости) Чтобы не добавлять ярлык BricsCAD на рабочий стол, дополнительно введите: **ADDDESKTOPSHORTCUT=""**.

5 (при необходимости) Чтобы после завершения установки не отображался перечень последних изменений, дополнительно введите: **SHOWRELEASENOTES=""**.

6 При необходимости, дополнительно введите: **APPLICATIONFOLDER="ваш_каталог"**, чтобы установить **BricsCAD®** в другой каталог, отличающийся от каталога по умолчанию.

7 (при необходимости) Чтобы указать каталог для расположения файла лицензии при

наличии определения какого-либо из свойств **BRXLICENSEKEY**, **BRXLICENSEFILE** или **BRXLICENSESERVER**, дополнительно введите: **BRXLICENSEDESTFILE="path_to_lic_file"**.

По умолчанию используется путь **APPLICATIONFOLDER\BricsCAD.lic**.

8 (при необходимости) Чтобы выполнить активацию лицензии с ее сохранением в каталоге, определенном значением переменной **BRXLICENSEDESTFILE**, дополнительно введите **BRXLICENSEKEY="ваш_лицензионный_ключ"**.

9 (при необходимости) Чтобы скопировать файл лицензии в каталог, определенный значением переменной **BRXLICENSEDESTFILE**, дополнительно введите **BRXLICENSEFILE="ваш_лицензионный_файл"**.

10 При необходимости, дополнительно введите **BRXLICENSESERVER="имя_сервера"**, или **BRXLICENSESERVER="порт@имя_сервера"**(*), чтобы создать лицензионный файл в папке, определенной значением переменной **BRXLICENSEDESTFILE**.

(*) Если ваш сервер лицензий использует порт, отличный от порта 5053, используемого по умолчанию.

11 (при необходимости, только для 32-разрядной версии) Чтобы не устанавливать VBA, дополнительно введите: **INSTALLVBA="0"** or **INSTALLVBA=""**.

Пример ввода в командной строке: **msiexec /i "BricsCAD-Vxx.x.xx-x-ru_RU(x64).msi" /qn ADDDESKTOPSHORTCUT="" SHOWRELEASENOTES=""**.

12 Нажмите клавишу **Enter** для запуска установки.

***Прим.:***

- Для получения информации об опциях введите в командной строке: **msiexec /?**.
- Дополнительная информация об использовании команды **msiexec** доступна на сайте Microsoft.

### 2.1.4 Ознакомительный режим

**BricsCAD**® может быть использован в ознакомительном режиме в течение 30 дней. По окончании этого периода для дальнейшей работы необходимо выполнить активацию коммерческой лицензии.

#### 2.1.4.1 Запуск BricsCAD в ознакомительном режиме

Каждый раз при запуске **BricsCAD**® в ознакомительном режиме отображается диалоговое окно **Запуск BricsCAD** с указанием количества дней, оставшихся до окончания ознакомительного периода.

Выполните одно из следующих действий:

- Нажмите кнопку **Продолжить** для запуска **BricsCAD**® в ознакомительном режиме.
- Нажмите кнопку **Лицензия...** для активации Однопользовательской или Корпоративной лицензии или использования Сетевой лицензии.

#### 2.1.4.2 Установка уровня лицензии

Ознакомительный режим позволяет использовать все функции **BricsCAD®**, которые доступны для лицензии **BricsCAD® Ultimate**.

Команда SETLICENSELEVEL позволяет выбрать более низкий уровень лицензии.

## 2.2 Активация и лицензирование BricsCAD®

### 2.2.1 Обзор процедуры лицензирования

**BricsCAD®** предлагает различные варианты лицензирования.

#### 2.2.1.1 Уровни лицензии

Для **BricsCAD®** существует 5 уровней лицензии:

- **BricsCAD® Lite**
- **BricsCAD® Pro**
- **BricsCAD® BIM**
- **BricsCAD® Mechanical**
- **BricsCAD® Ultimate**

##### 1 BricsCAD® Lite

Если вы занимаетесь 2D проектированием, вам может подойти лицензия **BricsCAD® Lite**. В отличие от **AutoCAD LT®**, эта версия позволяет выполнять просмотр, перемещение, копирование и зеркальное отображение 3D моделей. Редактирование и создание новых 3D моделей недоступно. **BricsCAD® Lite** позволяет использовать LISP приложения. LISP позволяет пользователям **BricsCAD®** загружать настраиваемые процедуры или небольшие приложения, которые будут выполняться внутри **BricsCAD®**.

##### 2 BricsCAD® Pro

**BricsCAD® Pro** включает прямое 3D моделирование с полной системой 3D зависимостей, визуализацию с использованием материалов и освещения, а также обеспечивает доступ к сторонним приложениям.

##### 3 BricsCAD® BIM

**BricsCAD® BIM** включает все возможности **BricsCAD® Pro**. Вы можете использовать свои текущие навыки САПР, чтобы в рекордно короткие сроки плавно перейти к созданию реальных BIM моделей. Один продукт, один процесс проектирования, один стандартный формат DWG.

#### 4 BricsCAD® Mechanical

Помимо всех функций **BricsCAD® Pro**, лицензия **BricsCAD® Mechanical** предлагает полный набор инструментов для машиностроительного проектирования в формате .dwg., включая разработку изделий из листового металла.

#### 5 BricsCAD® Ultimate

**BricsCAD® Ultimate** объединяет возможности **BricsCAD® BIM** и **BricsCAD® Mechanical**.

### 2.2.1.2 Типы лицензий

Для **BricsCAD®** доступны 2 типа лицензий:

- Постоянная лицензия.
- Лицензия на аренду.

#### 1 Постоянная лицензия

При покупке постоянной лицензии вы становитесь владельцем программного обеспечения **BricsCAD®** и можете использовать его в любой операционной системе - Microsoft Windows, macOS или Linux.

Вам предлагается 2 варианта обслуживания вашей лицензии:

- Пакет Maintenance, в который входит обновление до следующей основной версии и приоритетная поддержка.
- Приобретение новых обновлений по мере необходимости.

#### 2 Лицензия на аренду

При выборе варианта аренды вы получаете доступ к последней версии **BricsCAD®** в течение года. Этот вид лицензии включает в себя приоритетную поддержку и все обновления, которые выпускаются в течение времени действия вашей подписки.

### 2.2.1.3 Дополнительные варианты лицензирования

Дополнительные варианты лицензирования включают академические, однопользовательские, сетевые и корпоративные лицензии.

#### 1 Академическая лицензия

Студентам, преподавателям и учебным заведениям предоставляется бесплатный доступ ко всему программному обеспечению **BricsCAD®** в течение 12 месяцев. Для получения академической лицензии необходимо зарегистрироваться как студент/преподаватель или учебное заведение и предоставить необходимую информацию, подтверждающую академический статус. Вы можете продлевать академическую лицензию ежегодно на протяжении всего периода обучения, используя данные своего студенческого билета.

### 2 Однопользовательская лицензия

Однопользовательское лицензионное соглашение рассчитано на одного пользователя. Каждая однопользовательская лицензия позволяет выполнить две активации. Вы можете установить приложение на два компьютера, например, на рабочем месте в офисе и на ноутбуке для использования в полевых условиях. Но одновременно может использоваться только одно рабочее место. Однопользовательское лицензирование доступно как для постоянной лицензии, так и для аренды. Академические лицензии для студентов всегда однопользовательские.

### 3 Сетевая лицензия

Соглашение о сетевой лицензии позволяет нескольким пользователям одновременно получать доступ к **BricsCAD®** с использованием локальной сети. Количество доступных сетевых лицензий определяет количество пользователей, которые могут одновременно использовать программное обеспечение. Сетевое лицензирование доступно как для постоянной лицензии, так и для аренды и включает в себя годовую лицензию на использование пакета Maintenance. Сетевое лицензирование также доступно для учебных заведений, использующих академическую лицензию.

### 4 Корпоративная лицензия

Соглашение о корпоративном лицензировании позволяет использовать программное обеспечение **BricsCAD®** в рамках определенной пользовательской базы. Вы можете установить программное обеспечение, используя один лицензионный ключ, действительный для каждого пользователя. Корпоративное лицензирование доступно для постоянных лицензий и аренды и включает в себя годовую лицензию на использование пакета Maintenance. Для академической лицензии этот тип лицензирования используется по умолчанию.

## 2.2.2 BricsCAD® and AutoCAD® feature comparison

BricsCAD® versus AutoCAD® feature comparison.

| Drawings | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Native DWG file format (thru AutoCAD 2021) | √ | √ | √ | √ | √ | √ | √ |

| Drawings | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Password Protection | √ | √ | √ | √ | √ | o | o |
| eTransmit | √ | √ | √ | √ | √ | √ | √ |
| Sheet Set Manager | √ | √ | √ | √ | √ | √ | √ |
| Geographic Location | √ | √ | √ | √ | √ | √ | √ |
| Drawing Compare | √ | √ | √ | √ | √ | √ | √ |
| Cloud Collaboration | √ | √ | √ | √ | √ | √ | √ |

| Interface | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Dark and Light Color Themes | √ | √ | √ | √ | √ | √ | √ |

| Interface | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Industry standard commands, aliases, and variables | √ | √ | √ | √ | √ | √ | √ |
| Command Line (Autocomplete, Midstring, Clickable options) | √ | √ | √ | √ | √ | √ | √ |
| Start tab | √ | √ | √ | √ | √ | √ | √ |
| File (Drawing) tabs | √ | √ | √ | √ | √ | √ | √ |
| Model and Layout tabs | √ | √ | √ | √ | √ | √ | √ |
| Layout Manager | √ | √ | √ | √ | √ | o | o |
| Menu bar | √ | √ | √ | √ | √ | √ | √ |
| Toolbars | √ | √ | √ | √ | √ | √ | √ |
| Ribbon | √ | √ | √ | √ | √ | √ | √ |
| Tool Palettes | √ | √ | √ | √ | √ | √ | √ |

| Interface | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Contextual Right-click Menus | √ | √ | √ | √ | √ | √ | √ |
| Contextual Quad Menu | √ | √ | √ | √ | √ | o | o |
| Status Bar | √ | √ | √ | √ | √ | √ | √ |
| Workspaces | √ | √ | √ | √ | √ | √ | √ |
| Searchable Settings (Variables) Dialog Box | √ | √ | √ | √ | √ | o | o |
| Lockable Interface | √ | √ | √ | √ | √ | √ | √ |
| Clean Screen Mode | √ | √ | √ | √ | √ | √ | √ |
| Drawing Explorer | √ | √ | √ | √ | √ | o | o |
| Customizable User Interface (CUI) | √ | √ | √ | √ | √ | √ | √ |

| Interface | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Structure Browser | √ | √ | √ | √ | √ | o | o |

| Selection, snap & track | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Selection Filtering (QSELECT) | √ | √ | √ | √ | √ | √ | √ |
| Add Selected | √ | √ | √ | √ | √ | √ | √ |
| Select Similar | √ | √ | √ | √ | √ | √ | √ |
| Selection Cycling | √ | √ | √ | √ | √ | √ | √ |
| Entity Snaps (Osnaps) | √ | √ | √ | √ | √ | √ | √ |
| Polar Tracking | √ | √ | √ | √ | √ | √ | √ |
| Zintersection 3D snap | √ | √ | √ | √ | √ | o | o |
| Nearest Distance | √ | √ | √ | √ | √ | o | o |

| Selection, snap & track | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Adaptive Gridsnap | √ | √ | √ | √ | √ | o | o |
| Real-time boundary detection | √ | √ | √ | √ | √ | o | o |
| 2D Dynamic UCS | √ | √ | √ | √ | √ | o | o |
| 3D Dynamic UCS | √ | √ | √ | √ | √ | o | √ |

| Drawing tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Line, Polyline, 3D Polyline, Spline, Ray, Xline | √ | √ | √ | √ | √ | √ | √ |
| Multiline and Multiline Styles | √ | √ | √ | √ | √ | o | √ |

| Drawing tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Circle, Arc, Ellipse, Elliptical Arc | √ | √ | √ | √ | √ | √ | √ |
| Polygon, Rectangle, Region, Donut, Point | √ | √ | √ | √ | √ | √ | √ |
| Helix | √ | √ | √ | √ | √ | o | √ |
| 2D Parameters, Constraints | √ | √ | √ | √ | √ | o | √ |
| 2D Autoconstrain | √ | √ | √ | √ | √ | 2 | 2 |

| Annotation tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Dimensions, Dimension Styles | √ | √ | √ | √ | √ | √ | √ |

| Annotation tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Multi-leaders, Multi-leader Styles | √ | √ | √ | √ | √ | √ | √ |
| Single and Multi-line text, Text Styles, Text Editor | √ | √ | √ | √ | √ | √ | √ |
| Hatches, Gradients, Hatch/ Gradient Editor | √ | √ | √ | √ | √ | √ | √ |
| Wipeouts | √ | √ | √ | √ | √ | √ | √ |
| Revision Clouds | √ | √ | √ | √ | √ | √ | √ |
| Attributes, Attribute Editor | √ | √ | √ | √ | √ | √ | √ |
| Tables, Table Styles | √ | √ | √ | √ | √ | √ | √ |
| Fields | √ | √ | √ | √ | √ | √ | √ |
| Date Extraction | √ | √ | √ | √ | √ | o | √ |

| Annotation tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Data Linking | √ | √ | √ | √ | √ | √ | √ |
| Annotation Scaling | √ | √ | √ | √ | √ | √ | √ |
| Spell Check | √ | √ | √ | √ | √ | √ | √ |

| Editing tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Move, Copy, Rotate, Scale, Stretch, Mirror, Align | √ | √ | √ | √ | √ | √ | √ |
| Trim, Extend, Lengthen, Break, Join | √ | √ | √ | √ | √ | √ | √ |
| Fillet, Chamfer | √ | √ | √ | √ | √ | √ | √ |
| CopyGuided | √ | √ | √ | √ | √ | o | o |

| Editing tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| MoveGuided | √ | √ | √ | √ | √ | o | o |
| Grip Edit | √ | √ | √ | √ | √ | √ | √ |
| Properties panel | √ | √ | √ | √ | √ | √ | √ |
| Manipulator (3D Gizmo) | √ | √ | √ | √ | √ | o | √ |
| Drawordder | √ | √ | √ | √ | √ | √ | √ |
| Undo Per Entity (Property History) | √ | √ | √ | √ | √ | o | o |
| Optimize | √ | √ | √ | √ | √ | o | o |
| Simplify | √ | √ | √ | √ | √ | o | o |

| Reference tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Blocks, Block Editor | √ | √ | √ | √ | √ | √ | √ |

| Reference tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Dynamic Blocks | 1 | 1 | 1 | 1 | 1 | √ | √ |
| Block Face Camera | √ | √ | √ | √ | √ | o | o |
| Blockify | √ | √ | √ | √ | √ | o | o |
| Associative Arrays | √ | √ | √ | √ | √ | √ | √ |
| Dwg (Xref) Attach, Clip, Bind, Edit | √ | √ | √ | √ | √ | √ | √ |
| PDF Attach, Clip | √ | √ | √ | √ | √ | √ | √ |
| Image Attach, Clip, Adjust, Transparency | √ | √ | √ | √ | √ | √ | √ |
| Point Clouds Colormap | √ | √ | √ | √ | √ | o | √ |

| Visualization tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Layers, Layer States | √ | √ | √ | √ | √ | √ | √ |
| Transparency | √ | √ | √ | √ | √ | √ | √ |
| Visual Styles | √ | √ | √ | √ | √ | o | √ |
| LookFrom (ViewCube™) | √ | √ | √ | √ | √ | o | √ |
| Match Perspective | √ | √ | √ | √ | √ | o | o |
| Lights | o | √ | √ | √ | √ | o | √ |
| Sun Properties | o | √ | √ | √ | √ | o | √ |
| Materials | o | √ | √ | √ | √ | o | √ |
| Render | o | √ | √ | √ | √ | o | √ |
| Animation Editor | o | √ | √ | √ | √ | o | o |
| Walkthrough navigation | o | √ | √ | √ | √ | o | √ |

| Visualization tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| ANIPATH command | o | √ | √ | √ | √ | o | √ |

| Modeling tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| ACIS Modeling | o | √ | √ | √ | √ | o | √ |
| Solid and Surface Modeling | o | √ | √ | √ | √ | o | √ |
| Solid Primitives | o | √ | √ | √ | √ | o | √ |
| Mesh Primitives | o | √ | √ | √ | √ | o | √ |
| Polysolid | o | √ | √ | √ | √ | o | √ |
| Union, Subtract, Intersect, Interfere | o | √ | √ | √ | √ | o | √ |

| Modeling tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Extrude, Revolve, Loft, Sweep, Thicken | o | √ | √ | √ | √ | o | √ |
| Twist | o | √ | √ | √ | √ | o | o |
| Fillet, Chamfer | o | √ | √ | √ | √ | o | √ |
| Slice | o | √ | √ | √ | √ | o | √ |
| Multi-slice | o | √ | √ | √ | √ | o | o |
| T Connect, L Connect, Connect with Nearest | o | √ | √ | √ | √ | o | o |
| Extract Edges | o | √ | √ | √ | √ | o | √ |
| Direct modeling | o | √ | √ | √ | √ | o | √ |
| Deformable (Freeform) Modeling | o | √ | √ | √ | √ | o | √ |
| Audit, Simplify | √ | √ | √ | √ | √ | o | √ |

| Modeling tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Stitch | o | √ | √ | √ | √ | o | √ |
| 3D Parameters, Constraints | o | √ | √ | √ | √ | o | √ |
| 3D Autoparametrize | o | √ | √ | √ | √ | o | o |
| 3D Blockify and ParametricBlockify | o | √ | √ | √ | √ | o | o |
| 3D Parametric Design Tables | o | √ | √ | √ | √ | o | o |
| Section planes | o | √ | √ | √ | √ | o | √ |
| Drawing Views | o | √ | √ | √ | √ | o | √ |
| 3D Threads | o | √ | √ | √ | √ | o | o |
| Point Cloud Align | o | √ | √ | √ | √ | o | o |

| Modeling tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Point Cloud Bubble Viewer | o | √ | √ | √ | √ | o | o |

| Civil tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| TIN Surface Modeling | o | √ | √ | √ | √ | o | o |
| TIN Volume Surface | o | √ | √ | √ | √ | o | o |
| Grading | o | √ | √ | √ | √ | o | o |
| Alignments | o | √ | √ | √ | √ | o | o |
| Corridors | o | √ | √ | √ | √ | o | o |
| Civil 3D Object Enabler | √ | √ | √ | √ | √ | o | √ |
| Architectural Desktop Object Enabler | √ | √ | √ | √ | √ | o | √ |

| Civil tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| CADWorx Object Enabler | √ | √ | √ | √ | √ | o | √ |

| Architectural tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| BIM features | o | o | √ | o | √ | o | √ |
| Structural Modeling Toolset | o | o | √ | o | √ | o | √ |
| Project Database | o | o | √ | o | √ | o | √ |
| Library Database | o | o | √ | o | √ | o | √ |
| Site, Building and Stories | o | o | √ | o | √ | o | √ |
| Windows and Doors | o | o | √ | o | √ | o | √ |

| Architectural tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Fully Parametric BIM Component Creation | o | o | √ | o | √ | o | o |
| 3D Spaces | o | o | √ | o | √ | o | √ |
| Quickdraw | o | o | √ | o | √ | o | o |
| Quickbuilding | o | o | √ | o | √ | o | o |
| Point Cloud Floor Detection | o | o | √ | o | √ | o | o |
| Point Cloud Projection | o | o | √ | o | √ | o | o |
| Point Cloud Geometry Detection | o | o | √ | o | √ | o | o |

| Architectural tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Point Cloud FitPlanar (geometry detection) | o | o | √ | o | √ | o | o |
| Propagate | o | o | √ | o | √ | o | o |
| Custom BIM Properties | o | o | √ | o | √ | o | o |
| Drawing Customizations | o | o | √ | o | √ | o | o |
| Analytical Model | o | o | √ | o | √ | o | o |
| Python Scripts (Beta) | o | o | √ | o | √ | o | o |

| Mechanical tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Mechanical Browser | o | √ | √ | √ | √ | o | o |

| Mechanical tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Parts Library | √ | √ | √ | √ | √ | o | o |
| Piping | o | √ | √ | √ | √ | o | o |
| Assembly Modeling | o | o | o | √ | √ | o | o |
| Exploded Views | o | o | o | √ | √ | o | o |
| Kinematic Analysis | o | o | o | √ | √ | o | o |
| Bill of Materials | o | o | o | √ | √ | o | o |
| Sheet Metal Design | o | o | o | √ | √ | o | o |
| 3D Solids to Sheet Metal Parts Conversion | o | o | o | √ | √ | o | o |
| Corner Relief for Adjacent Edge Flanges | o | o | o | √ | √ | o | o |
| Lofted Bends | o | o | o | √ | √ | o | o |

| Mechanical tools | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Coloring | o | o | o | √ | √ | o | o |
| Diagnostic for Invalid Sheet Metal Features | o | o | o | √ | √ | o | o |
| Batch Processing | o | o | o | √ | √ | o | o |

| Import, export, output | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Print/ Plot | √ | √ | √ | √ | √ | √ | √ |
| Publish/ Batch Plot | √ | √ | √ | √ | √ | √ | √ |
| CTB & STB Table Files | √ | √ | √ | √ | √ | √ | √ |
| PC3 Printer Configuration Files | √ | √ | √ | √ | √ | √ | √ |

| Import, export, output | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Named Page Setups | √ | √ | √ | √ | √ | √ | √ |
| PDF Import, Export | √ | √ | √ | √ | √ | √ | √ |
| DAE Import, Export | √ | √ | √ | √ | √ | o | o |
| IFC Import, Export | o | o | √ | o | √ | o | √ |
| 3DM Import, Export | o | √ | √ | √ | √ | o | o |
| LandXML Import, Export | o | √ | √ | √ | √ | o | o |
| RFA Import | o | o | √ | o | √ | o | o |
| SKP Import | o | √ | √ | √ | √ | o | o |
| DGN Import | √ | √ | √ | √ | √ | √ | √ |
| RVT Import | o | o | √ | o | √ | o | o |
| DWF Export | √ | √ | √ | √ | √ | √ | √ |

| Import, export, output | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| SVG Export | √ | √ | √ | √ | √ | o | o |
| FBX Export | √ | √ | √ | √ | √ | o | o |
| STL Export | √ | √ | √ | √ | √ | o | √ |
| Point Cloud Export | o | √ | √ | √ | √ | o | o |
| Additional formats available with BricsCAD Communicator | o | √ | √ | √ | √ | o | o |

| Programming | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| ActiveX | √ | √ | √ | √ | √ | o | √ |
| Entity data editor | √ | √ | √ | √ | √ | o | o |
| Script Recorder | √ | √ | √ | √ | √ | o | √ |

| Programming | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Full LISP Support (vl, vlr, vla and vlax) | √ | √ | √ | √ | √ | o | √ |
| Solutions Development System (SDS/ ADS) | √ | √ | √ | √ | √ | o | √ |
| DCL engine | √ | √ | √ | √ | √ | o | √ |
| COM API | o | √ | √ | √ | √ | o | √ |
| Visual Basic for Applications (VBA) | o | √ | √ | √ | √ | o | o |
| VBA DVB projects support | o | √ | √ | √ | √ | o | o |
| .NET | o | √ | √ | √ | √ | o | √ |
| BRX (ARX) | o | √ | √ | √ | √ | o | √ |
| TX (Teigha eXtensions) | √ | √ | √ | √ | √ | o | o |

| Programming | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| CUI menu files support, with Diesel expressions | √ | √ | √ | √ | √ | √ | √ |

| Licensing | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Multi-platform Licenses | √ | √ | √ | √ | √ | o | o |
| Perpetual | √ | √ | √ | √ | √ | o | o |
| Perpetual + Maintenance | √ | √ | √ | √ | √ | o | o |
| Subscription | √ | √ | √ | √ | √ | √ | √ |
| Network Multi-user | √ | √ | √ | √ | √ | o | o |
| Volume (Perpetual) | √ | √ | √ | √ | √ | o | o |

| Licensing | | | | | | | |
|---|---|---|---|---|---|---|---|
| | BricsCAD® | | | | | AutoCAD® | |
| | Lite | Pro | BIM | Mechanical | Ultimate | LT | Full |
| Academic | √ | √ | √ | √ | √ | √ | √ |

(1) You can manipulate dynamic block insertions but you cannot create dynamic block definitions.

(2) You can automatically apply geometric constraints but you cannot automatically apply a combination of geometric and dimensional constraints.

### 2.2.3 Ознакомительная версия: Активация через Интернет

Подготовка: Для активации лицензии **BricsCAD®** необходимо наличие подключения к Интернет.

1 Запустите **BricsCAD®**.

2 Нажмите кнопку **Запуск демо** в диалоговом окне **Запуск BricsCAD**.

Result: Ознакомительная лицензия будет активирована через Интернет.

### 2.2.4 Ознакомительная версия: Ручная активация

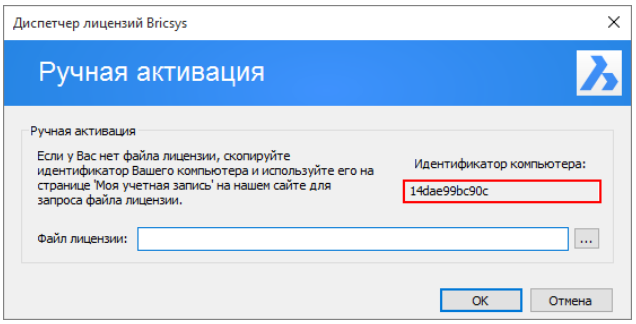
В том случае, если компьютер не подключен к сети Интернет, лицензия может быть активирована вручную с помощью другого компьютера, имеющего подключение к Интернет.

1 Запустите **BricsCAD®**.

2 Нажмите кнопку **Постоянная лицензия** в диалоговом окне Запуск BricsCAD.

3 Нажмите **Активировать вручную...** в диалоговом окне **Диспетчер лицензий Bricsys**.

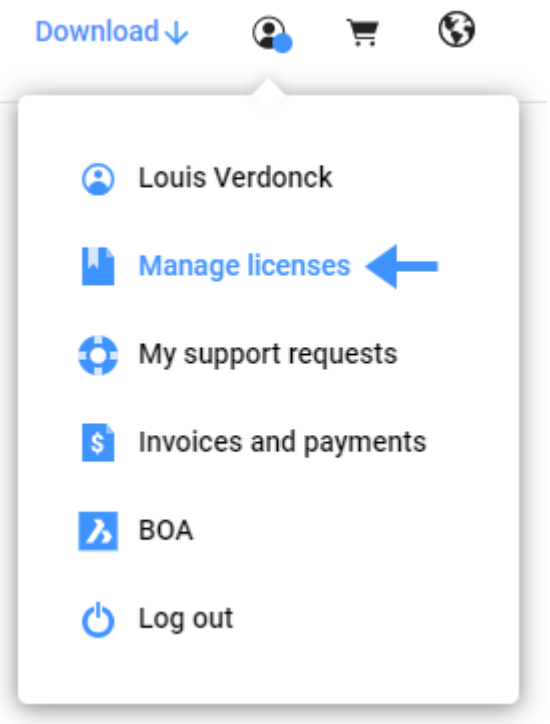
Result: Откроется диалоговое окно **Ручная активация**.




4 Скопируйте содержимое поля **Computer Hostid**.

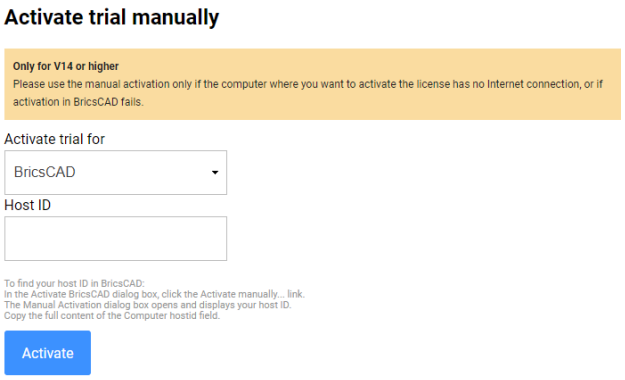
5 Войдите на сайт Bricsys с компьютера, подключенного к Интернету.

6 Щелкните на значке **Моя учетная запись** ( ) и в открывшемся меню выберите **Управление лицензиями**.




7 Выберите **Активировать BricsCAD вручную**.

8 В поле **Host ID** вставьте идентификатор компьютера, на который устанавливается **BricsCAD®**, и нажмите кнопку **Активировать**.



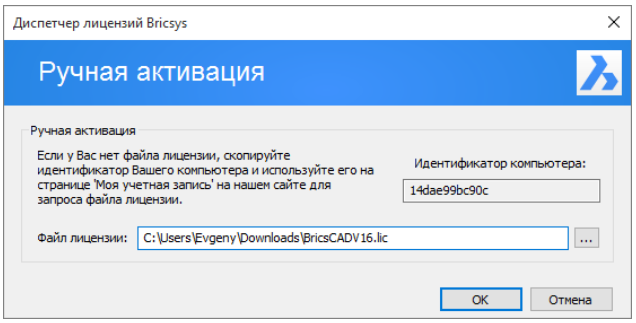

Result: Файл лицензии будет сгенерирован и предоставлен вам для загрузки.

9 Нажмите кнопку **Загрузить файл лицензии** .

Result: Файл лицензии будет загружен в каталог загрузки, используемый вами по умолчанию.

10 В диалоговом окне **Ручная активация** нажмите кнопку **обзора** ( ... ).

11 Откройте папку, в которую вы скопировали файл лицензии, и выберите этот файл.




12 Нажмите кнопку **OK**.

Result: **BricsCAD®** запущен.

### 2.2.5 Однопользовательская или корпоративная лицензия: активация через Интернет

Подготовка: Для активации лицензии **BricsCAD®** необходимо наличие подключения к Интернет.

1 Запустите **BricsCAD®**.

***Прим.:*** После завершения ознакомительного периода нажмите **Активировать лицензию** в диалоговом окне **Запуск BricsCAD** и перейдите к шагу 5.

2 Запустите команду LICENSEMANAGER.

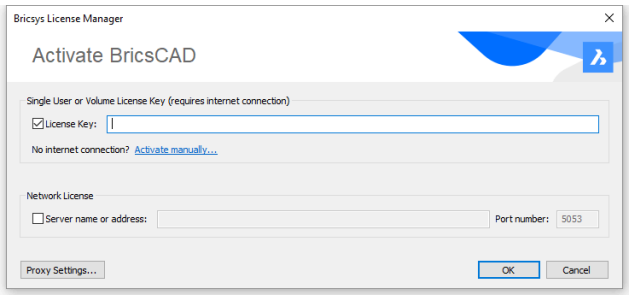
Result: Откроется диалоговое окно Диспетчер лицензий Bricsys.

3 Выберите **Управление лицензиями**.

Result: Откроется диалоговое окно **Информация о лицензии**.

4 Нажмите кнопку **Изменить...**.

Result: Откроется диалоговое окно **Диспетчер лицензий Bricsys** > **Активация BricsCAD**:




5 Введите ваш лицензионный ключ в поле **Лицензионный ключ**.

6 Нажмите кнопку **OK**.

Result:

- Ваш лицензионный ключ буде зарегистрирован через Интернет.
- При успешной активации откроется диалоговое окно.

7 Прочтите текст в диалоговом окне и нажмите кнопку **Я ознакомлен с условиями повторной активации**.

### 2.2.6 Одиночная или корпоративная лицензия: автономная (ручная) активация

В том случае, если компьютер не подключен к сети Интернет, лицензия может быть активирована вручную с помощью другого компьютера, имеющего подключение к Интернет.

***Прим.:*** Деактивация лицензии, активированной вручную, невозможна.

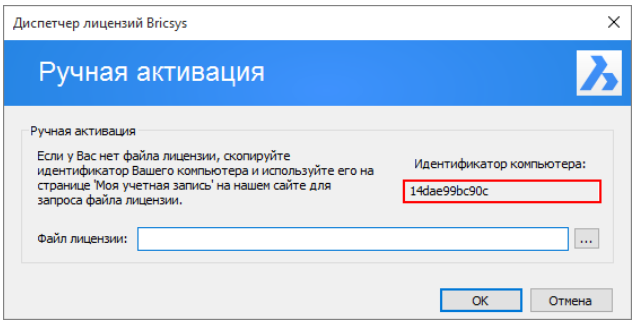
1 Запустите **BricsCAD®**.

2 Выполните одно из следующих действий:

- При первом запуске **BricsCAD®** перейдите к следующему шагу.
- Если вы запускаете **BricsCAD®** в ознакомительном режиме, нажмите **Ввести лицензию...** в диалоговом окне **Бесплатная ознакомительная версия**.
- Если вы заменяете или обновляете существующую лицензию, запустите команду LICENSEMANAGER. Откроется диалоговое окно Диспетчер лицензий Bricsys. Выберите **Управление лицензиями** и в открывшемся окне Информация о лицензии нажмите кнопку **Изменить...**.

3 Нажмите **Ручная активация...** в открывшемся окне **Активация BricsCAD**.

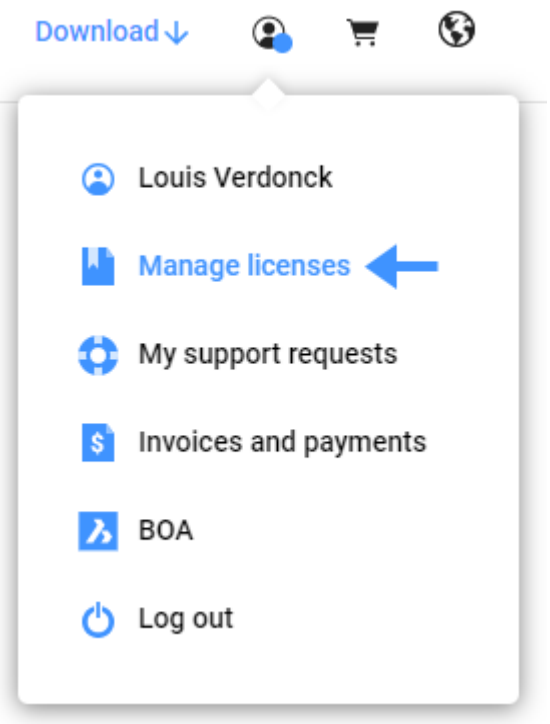
Result: Откроется диалоговое окно **Ручная активация**.




4 Скопируйте содержимое поля **Computer Hostid**.

5 Войдите на сайт Bricsys с компьютера, подключенного к Интернету.
Убедитесь, что вы вошли в систему с учетной записью Bricsys, содержащей ваши лицензии.

6 Щелкните на значке **Моя учетная запись** ( ) и в открывшемся меню выберите **Управление лицензиями**.




7 Наведите указатель мыши на лицензионный ключ, который хотите активировать, и щелкните на значке карандаша (✎).




Result: Отобразится **Информация о лицензии**.

8 Нажмите **Активация вручную**.

9 В поле **Host ID** вставьте идентификатор компьютера, на который устанавливается **BricsCAD®**.

10 Нажмите кнопку **Активировать**.

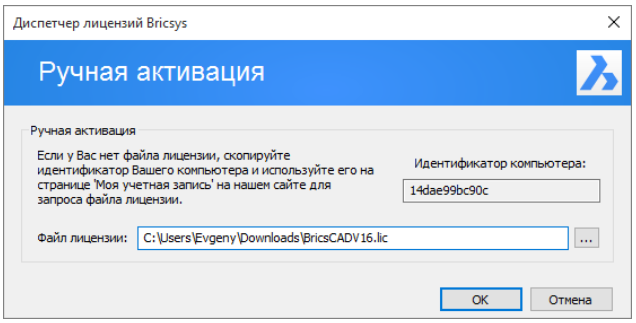
Result: Файл лицензии будет сгенерирован и предоставлен вам для загрузки.

11 Нажмите кнопку **Загрузить файл лицензии** .

Result: Файл лицензии **BricsCADVxx.lic** будет загружен в папку загрузки, используемую вами по умолчанию.

12 Скопируйте файл лицензии на компьютер, на который устанавливается **BricsCAD®**.

13 В диалоговом окне **Ручная активация** нажмите кнопку обзора (...). Откройте папку, в которую вы скопировали файл лицензии **BricsCADVxx.lic** и выберите этот файл.




14 Нажмите кнопку **OK**.

Result: **BricsCAD®** запущен.

### 2.2.7 Использование сетевой лицензии на компьютере пользователя

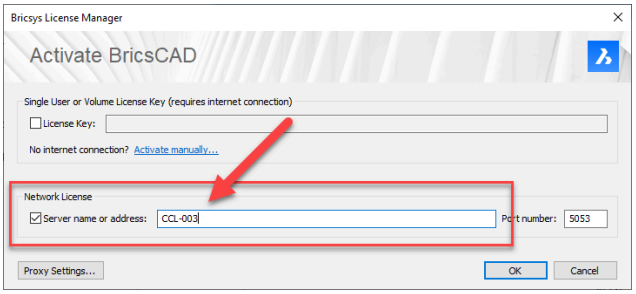
Сразу после активации Сетевой лицензии на сервере лицензий сетевая лицензия может быть использована на клиентских компьютерах.

1 Выполните одно из следующих действий:

- При первом запуске **BricsCAD®** нажмите кнопку **Постоянная лицензия** в диалоговом окне Запуск BricsCAD.
- Если вы заменяете или обновляете существующую лицензию, запустите команду LICENSEMANAGER. Откроется диалоговое окно Диспетчер лицензий Bricsys. Выберите **Управление лицензиями** и в открывшемся окне Информация о лицензии нажмите кнопку **Изменить...**.

2 В открывшемся диалоговом окне **Activate BricsCAD** выберите опцию **Сетевая лицензия**.

3 В поле **Имя или адрес сервера** введите имя или IP-адрес сервера лицензий.




4 При необходимости в поле **Порт** введите номер порта.

5 Если Ваш компьютер подключен к серверу лицензий через прокси-сервер, нажмите кнопку **Параметры прокси-сервера** для настройки конфигурации прокси-сервера сетевой лицензии.

6 Нажмите кнопку **OK**.

## 2.3 Настройка сервера сетевых лицензий

### 2.3.1 Сервер Сетевой лицензии

Чтобы сделать сетевую лицензию **BricsCAD®** доступной на клиентском компьютере, администратор сети должен сначала установить сервер сетевых лицензий от **Reprise Software**.

**Bricsys Network License Manager** - это отдельная программа, устанавливающая сервер. После запуска сервера он может быть настроен с помощью web-интерфейса.

Как для запуска **BricsCAD®** необходим файл лицензии, так и для работы сервера лицензий необходим файл Сетевой лицензии, который может быть активирован через Интернет или вручную.

Сетевая лицензия может быть активирована только один раз с получением привязки к конкретному серверу.

Сразу после установки сервера лицензий пользователь может начать использовать Сетевую лицензию **BricsCAD®** на клиентском компьютере.

***Прим.:*** Reprise Software не поставляет сервер сетевых лицензий для платформы Mac. Но имеется возможность установки серверов сетевых лицензий для Windows или Linux для управления лицензиями на компьютерах-клиентах с платформой Mac.

### 2.3.2 Краткое руководство по использованию сетевой лицензии

Прежде всего, убедитесь, что администратор сети/IT участвует в процессе.

#### 2.3.2.1 1. Действия для сервера

1 Загрузите Bricsys Network License Manager и установите его с настройками по умолчанию:

https://www.bricsys.com/bricscad/tools/Bricsys-NetworkLicenseManager.msi

*Информация, связанная с политиками безопасности сервера* :

после установки файл RLM.exe должен иметь разрешение на использование 3 портов: **5053** для лицензии, **5054** для веб-интерфейса, **динамически выделяемый порт** Windows для сервера ISV, который может быть установлен в качестве фиксированного после активации, как описано в статье Дополнительные конфигурации.

2 После отображения диалогового окна активации, активируйте лицензионный ключ из вашей учетной записи Bricsys.

*Информация, связанная с политиками безопасности сервера* :

приложение-активатор *C:\Program Files (x86)\Bricsys\Bricsys Network License Manager\actNetworkLicense.exe* и доступ к **license.bricsys.com/actpro** порт **80**

3 Страница администрирования RLM license manager доступна по адресу **localhost:5054**

#### 2.3.2.2 2. Действия для компьютера-клиента

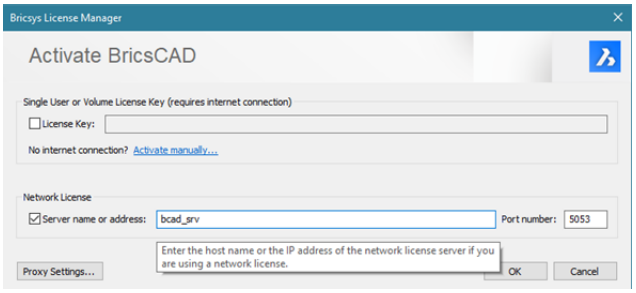
Клиентский компьютер должен:

- Находиться в одной **локальной сети** с компьютером RLM (физически или через VPN-соединение).
- Иметь возможность связи с компьютером RLM через порт **5053** по протоколу **TCP.**

1 Установите **BricsCAD®** с настройками по умолчанию.

2 Запустите **BricsCAD®** и активируйте сетевую лицензию: <servername>/<ip number> и <port>

(в этом, возможно, не будет необходимости, так как **BricsCAD®** при запуске выполняет проверку сети на наличие сервера лицензий)




### 2.3.3 Установка сервера сетевых лицензий для Windows

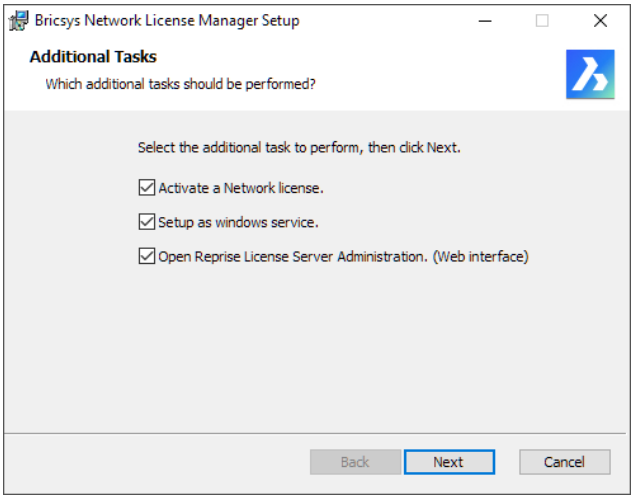
#### 2.3.3.1 Загрузка Bricsys Network License Manager

Загрузите Bricsys Network License Manager (Windows) по следующей ссылке:

https://www.bricsys.com/bricscad/tools/Bricsys-NetworkLicenseManager.msi

#### 2.3.3.2 Новая установка Bricsys Network License Manager

В диалоговом окне **Additional Tasks** (Дополнительные задачи) установите флажки для всех задач.




- **Activate a Network license (Активировать сетевую лицензию)**

При выборе этой опции сразу после установки открывается диалоговое окно **Bricsys Network License Manager** с предложением активировать сетевую лицензию. Вы также можете открыть окно **Bricsys Network License Manager** позже с помощью ярлыка License activation (Активация лицензии), доступного в меню Пуск.

- **Setup as a Windows service (Установка в качестве службы Windows)**

  При выборе этой опции сервер лицензий будет запускаться автоматически в виде службы при каждом запуске Windows. Вы можете сделать это позже с помощью следующей команды в командной строке: rlm.exe-dlog rlm.log-install_service

- **Open Reprise License Server Administration (Открыть веб-интерфейс Reprise License Server Administration)**

  После установки откроется веб-интерфейс администрирования сервера лицензий Reprise. Этот интерфейс используется для управления сетевой лицензией. Вы также можете открыть веб-интерфейс Reprise License Server Administration с помощью ярлыка License Server Administration в меню Пуск.

#### 2.3.3.3 Обновление Bricsys Network License Manager

1 Загрузите и установите Bricsys Network License Manager на компьютере, который будет использоваться в качестве сервера сетевой лицензии.

2 В диалоговом окне **Additional Tasks** (Дополнительные задачи) удалите флажки для всех задач.

***Прим.:*** В процессе установки будет выполнен перезапуск сервера лицензий.

#### 2.3.3.4 Установка с помощью командной строки

Вы также можете выполнить запуск Network License Manager с помощью командного окна.

По умолчанию при установке с помощью командной строки опции **Activate a Network license**, **Setup as Windows service** и **Open Reprise License Server Administration** (Web interface) отключены. Для включения этих опций используйте соответствующие параметры ACTIVATELICENSE, INSTALLSERVICE и OPENWEBINTERFACE.

Например команда **msiexec /qb /i Bricsys-NetworkLicenseManager.msi ACTIVATELICENSE=1 INSTALLSERVICE=1** покажет диалоговое окно активации и выполнит установку в качестве службы Windows без отображения веб-интерфейса.

***Прим.:***

- Для получения информации об опциях введите в командной строке **msiexec /?**.
- Дополнительная информация об использовании команды **msiexec** доступна на сайте Microsoft.

### 2.3.4 Установка сервера сетевых лицензий для Linux

Подготовка: Загрузите и установите Bricsys Network License Manager на компьютере, который будет использоваться в качестве сервера сетевой лицензии.

1 Загрузка Bricsys Network License Manager (Linux)

2 Извлеките содержимое архива **Bricsys-NetworkLicenseManager.tgz** в любую папку (командная

строка: tar xvzf Bricsys-NetworkLicenseManager.tgz).

3 Запустите приложение RLM в этой папке (командная строка в этой папке: ./rlm).

4 Откройте страницу, расположенную по адресу **http://localhost:5054**.

5 Нажмите кнопку **System Info**. Скопируйте идентификатор сети (ethernet hostid) и используйте его для активации Вашей лицензии на странице **Моя учетная запись** на сайте Bricsys.

6 После загрузки файла лицензии с сайта Bricsys сохраните его в том же каталоге, куда ранее был извлечен файл Bricsys-NetworkLicenseManager.tgz.

7 Нажмите кнопку **Reread/Restart Servers**.

8 Нажмите кнопку **Status**b для просмотра состояния сервера и на открывшейся странице в таблице нажмите кнопку **bricsys** для просмотра лицензий Bricsys.

9 Нажмите кнопку **RLM Manual...** для получения подробной информации о порядке настройки конфигурации сервера лицензий, включая автоматический запуск rlm при загрузке сервера.

Result: Сразу после активации Сетевой лицензии и запуска сервера он будет готов к раздаче лицензий. После выполнения этой процедуры Сетевая лицензия **BricsCAD**® будет доступна для использования на компьютере пользователя.

### 2.3.5 Сетевая лицензия: активация через Интернет

Для активации лицензии **BricsCAD**® необходимо наличие подключения к Интернет.

1 Запустите программу **License activation** с помощью соответствующего ярлыка в меню **Пуск**.

*C:\Program Files (x86)\Bricsys\Bricsys Network License Manager\actNetworkLicense.exe*

Откроется диалоговое окно **Bricsys License Manager - Activate Network License**.

2 В поле **License Key** введите ключ вашей сетевой лицензии и нажмите кнопку **OK**.

3 Щелкните на ярлыке **License server administration** в меню **Пуск**. Откроется окно **Reprise License Server Administration**.

4 Нажмите кнопку **Reread/Restart Servers**.

### 2.3.6 Сетевая лицензия: ручная активация (без подключения к Интернет)

В том случае, если компьютер не подключен к сети Интернет, лицензия может быть активирована вручную с помощью другого компьютера, имеющего подключение к Интернет.

***Прим.:*** Деактивация лицензии, активированной вручную, невозможна.

1 Запустите программу **License activation** с помощью соответствующего ярлыка в меню **Пуск**.

*C:\Program Files (x86)\Bricsys\Bricsys Network License Manager\actNetworkLicense.exe*

Откроется диалоговое окно **Bricsys License Manager - Activate Network License**.

2 Скопируйте содержимое поля **Computer Hostid**.

3 Войдите на сайт Bricsys с компьютера, подключенного к Интернету.

4 Щелкните на значке **Моя учетная запись** ( ) и в открывшемся меню выберите **Управление лицензиями**.

Убедитесь, что вы вошли на сайт Bricsys с учетной записью вашей компании, имеющей Сетевую лицензию. При необходимости обратитесь к диспетчеру учетных записей Bricsys.

5 Наведите указатель мыши на лицензионный ключ, который хотите активировать, и щелкните на значке карандаша (✎).

Отобразится **Информация о лицензии**.

6 Нажмите **Активация вручную**.

7 В поле **Host ID** вставьте идентификатор компьютера, на который устанавливается **BricsCAD®**.

8 Нажмите кнопку **Активировать**.

Файл лицензии будет сгенерирован и предоставлен вам для загрузки.

9 Нажмите кнопку **Загрузить файл лицензии** .

Файл лицензии **BricsCAD.lic** будет загружен в каталог загрузки, используемый вами по умолчанию.

10 Скопируйте файл лицензии на сервер в каталог установки **Bricsys Network License Manager** (по умолчанию *C:\Program Files (x86)\Bricsys\Bricsys Network License Manager*).

11 Выполните процедуру **reread/restart servers** с помощью соответствующей кнопки в окне RLM: localhost:5454, чтобы Reprise license manager использовал .lic файл.

### 2.3.7 Администрирование сервера сетевой лицензии

После запуска сервера сетевой лицензии (RLM License Server) он может быть сконфигурирован с помощью web-интерфейса Reprise License Server Administration.

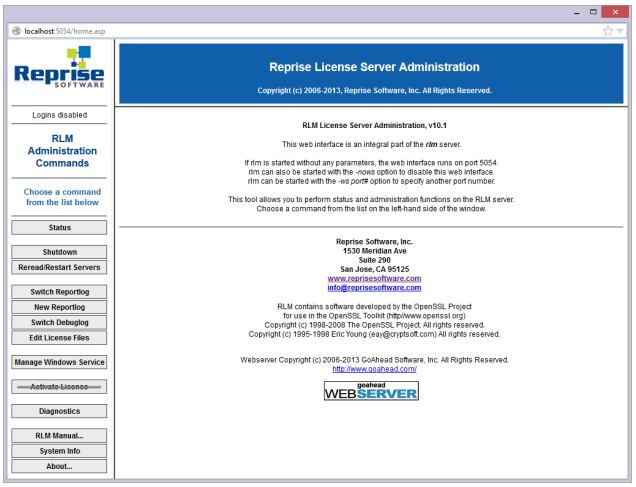
#### 2.3.7.1 Открытие веб-интерфейса

1 Если сервер сетевой лицензии еще не запущен, щелкните на ярлыке **License Server** в меню Пуск (в Windows Vista или более новых версиях: Щелчок правой кнопкой мыши > Запуск от имени администратора).

Result: Откроется окно консоли.

2 Откройте страницу, расположенную по адресу **http://localhost:5054**.

Result: Откроется web-интерфейс Reprise License Server Administration.




***Прим.:***

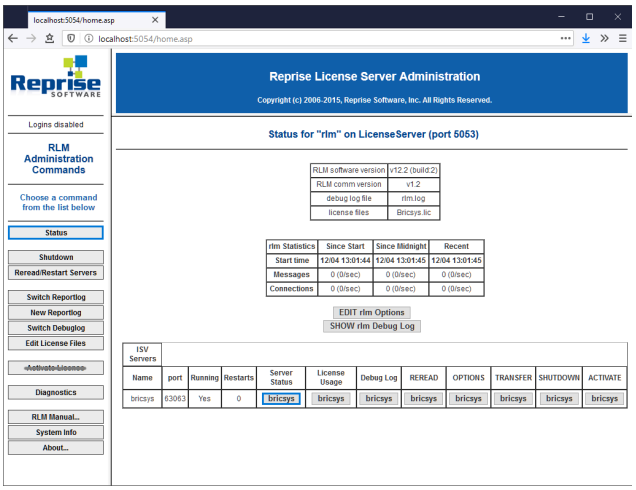
- Не используйте кнопку **Activate License**!
- Для просмотра всех опций нажмите кнопку **RLM Manual...** или откройте Руководство по администрированию RLM лицензии.

### 2.3.7.2 Проверка состояния лицензии и сервера

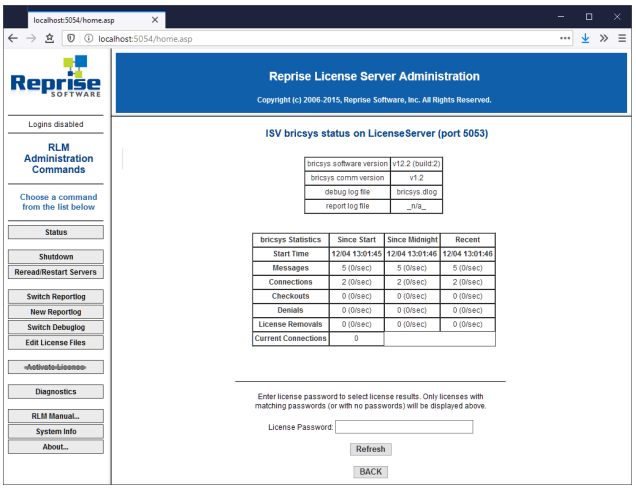
Вы можете проверить состояние всех сетевых лицензий и сервера.

1 Нажмите кнопку **Status**.

Result: Откроется страница **Status for "rlm" on [LicenseServer] (port 5053)**.




2 Для просмотра Сетевой лицензии Bricsys нажмите кнопку **bricsys** в столбце **Server Status** таблицы ISV Servers.

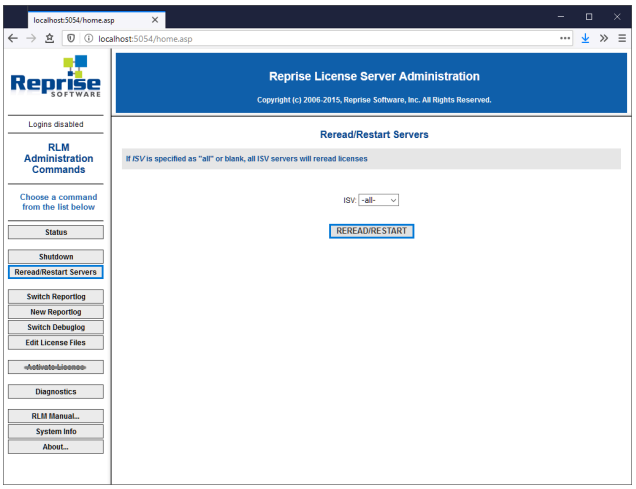



### 2.3.7.3 Перезагрузка сервера

После выполнения активации Сетевой лицензии при запущенном сервере необходимо выполнить перезагрузку сервера.

1 Нажмите кнопку **Reread/Restart Servers**.

Result: Откроется страница **Reread/Restart Servers**.




2 Нажмите кнопку **Reread/Restart Servers**.

Result: Будет выполнен перезапуск сервера с повторным считыванием сетевой лицензии.

### 2.3.8 Дополнительные настройки конфигурации

Сразу после установки сервера лицензий при попытке подключения к серверу в **BricsCAD®** может появиться сообщение об ошибке 17.

Возможной причиной этого может быть антивирус, брандмауэр или прокси-сервер.

### 2.3.8.1 Антивирус

Если на сервере работает антивирусное программное обеспечение, которое блокирует программу License Server (**rlm.exe** для Windows, **rlm** для Linux/Mac), необходимо разблокировать службу RLM в настройках антивирусной программы.

Для разблокирования служб используются средства настройки конфигурации антивирусной программы.

### 2.3.8.2 Брандмауэр

При наличии брандмауэра, установленного между сервером и **BricsCAD®**, необходимо в настройках брандмауэра открыть два порта: порт 5053 и порт, назначенный для сервера лицензий.

Номер порта назначается автоматически при запуске License Server. Чтобы открыть этот порт в брандмауэре, он должен быть вначале установлен в файле Bricsys.lic на сервере.

1 Откройте web-интерфейс License Server Administration по адресу **http://localhost:5054**.

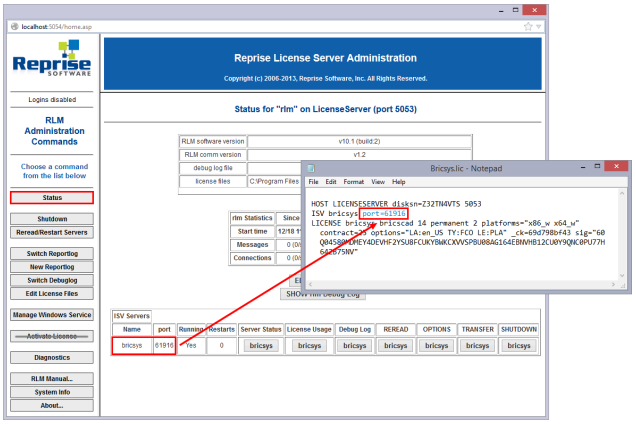
2 Нажмите кнопку **Status**. Откроется страница **Status for "rlm" on [LicenseServer] (port 5053)**.

3 Укажите номер порта для прокси-сервера. Номер порта отображается во втором столбце.

4 Скопируйте этот номер.

5 Откройте файл Bricsys.lic в текстовом редакторе (Notepad, Gedit, ...). По умолчанию этот файл находится в каталоге C:\Program Files (x86)\Bricsys\Bricsys Network License Manager.

Вы можете исправить номер порта путем изменения строки ISV bricsys в виде: ISV bricsys port=[номер порта],где текст [номер порта] должен быть заменен на значение номера порта, скопированное из таблицы ISV Servers.




6 В брандмауэре откройте оба порта: 5053 и добавленный в файл Bricsys.lic.

7 Закройте и перезагрузите программу License Server.

***Прим.:*** Если это не помогает решить проблему, добавьте программу License Server (rlm.exe для Windows, rlm для Linux/Mac) в список исключений брандмауэра и выполните перезапуск License Server.

#### 2.3.8.3 Прокси-сервер

Если между **BricsCAD®** и сервером лицензий используется прокси-сервер, необходимо выполнить настройку его конфигурации в **BricsCAD®**.

См. настройку параметров прокси-сервера для использования Сетевой лицензии.

## 2.4 Управление лицензиями

### 2.4.1 Изменение лицензии BricsCAD®

Используйте эту процедуру для обновления или замены лицензии.

1 Запустите команду LICENSEMANAGER.

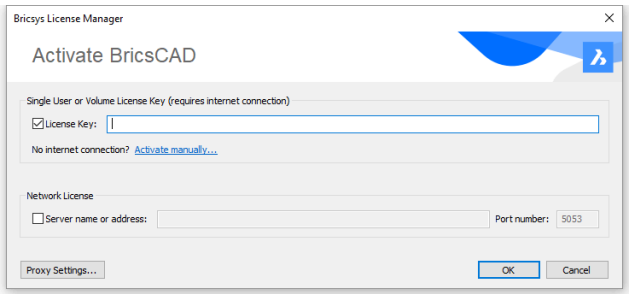
Result: Откроется диалоговое окно Диспетчер лицензий Bricsys.

2 Выберите **Управление лицензиями**.

Result: Откроется диалоговое окно **Информация о лицензии**.

3 Нажмите кнопку **Изменить...**.

Result: Откроется диалоговое окно **Диспетчер лицензий Bricsys** - **Активация BricsCAD**.




4 Введите ваш лицензионный ключ в поле **Лицензионный ключ**.

5 Нажмите кнопку **ОК**.

Result:

- Ваш лицензионный ключ буде зарегистрирован через Интернет.
- При успешной активации откроется диалоговое окно.

6 Прочтите текст в диалоговом окне и нажмите кнопку **Я ознакомлен с условиями повторной активации**.

### 2.4.2 Деактивация или аннулирование лицензии BricsCAD®

Используйте эту процедуру для:

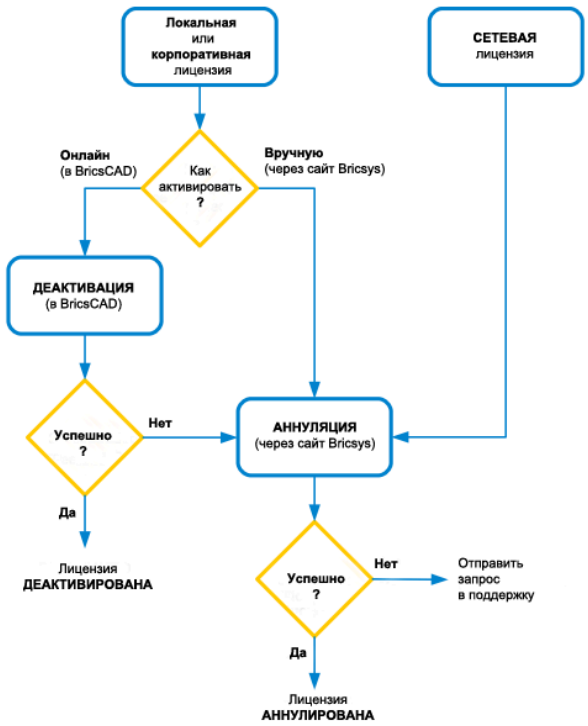
- Перемещения лицензии на другой компьютер.
- Полного удаления **BricsCAD®** из компьютера.
- Повторной активации **BricsCAD®** после замены жесткого диска, изменения его разделов или обновления операционной системы.

Для деактивации лицензии **BricsCAD**® требуется подключение к Интернет. Ручная деактивация лицензии с использованием другого компьютера, подключенного к Интернет, невозможна.

Вы не можете деактивировать:

- Одиночные и корпоративные лицензии, активированные вручную.
- Сетевые лицензии.
- Ознакомительные лицензии.

Ниже приведена схема деактивации лицензии **BricsCAD**®. Если деактивация лицензии **BricsCAD**® невозможна, вы можете аннулировать активацию.



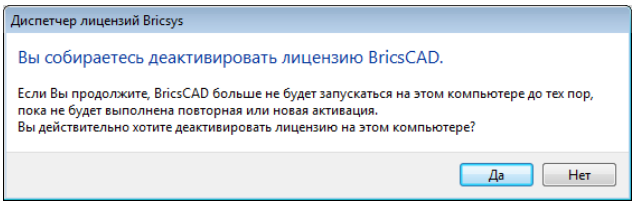

### 2.4.2.1 Деактивация лицензии

1 Запустите команду LICENSEMANAGER. Выберите опцию **Управление лицензиями** в открывшемся диалоговом окне **Диспетчер лицензий Bricsys**.

Result: Откроется диалоговое окно **Информация о лицензии**.

2 Если ваш компьютер подключен к серверу лицензий через прокси-сервер, нажмите кнопку **Параметры прокси-сервера** для настройки конфигурации прокси-сервера.

3 Нажмите **Деактивировать...**



4 В открывшемся окне нажмите кнопку **Да** для подтверждения деактивации лицензии.

Result: Лицензия будет деактивирована и станет доступной для активации на другом компьютере.

### 2.4.2.2 Аннулирование активации

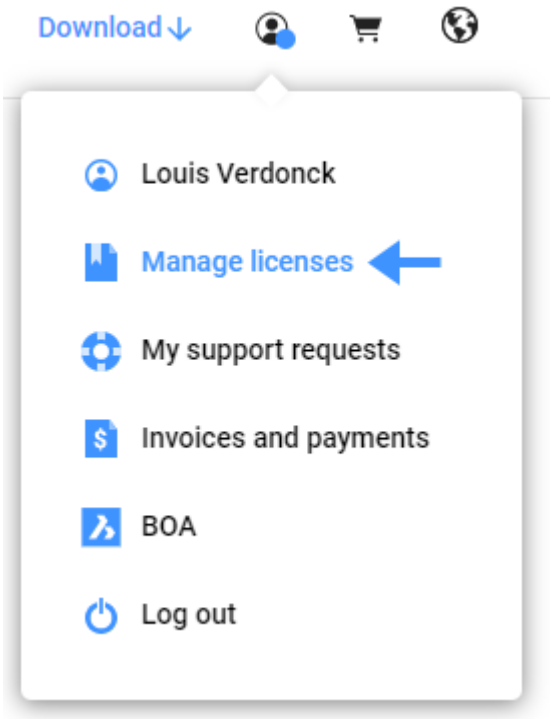
Аннулирование активации применятся только в том случае, если деактивация лицензии невозможна.

Используйте эту процедуру для:

- Деактивации сетевой лицензии, одиночной или корпоративной лицензии, активированной вручную.
- Повторной активации **BricsCAD®** после повреждения жесткого диска.
- Повторной активации **BricsCAD®** после замены жесткого диска, изменения его разделов или обновления операционной системы.
- Перемещения лицензии на другой компьютер.
- Полного удаления **BricsCAD®** из компьютера.

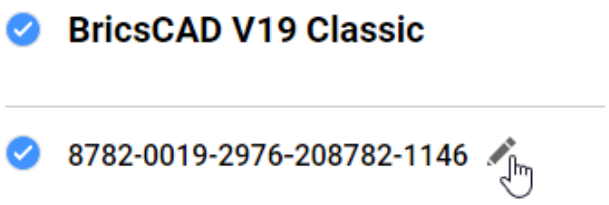
1 Войдите на сайт Bricsys, используя свою учетную запись.
Убедитесь, что Вы вошли на сайт с той учетной записью Bricsys, на которую были получены Ваши лицензии.

2 Щелкните на значке **Моя учетная запись** ( ) и в открывшемся меню выберите **Управление лицензиями**.




3 Наведите указатель мыши на лицензионный ключ, активацию которого необходимо

аннулировать, и щелкните на значке карандаша ( ).




4 Прокрутите страницу вниз и щелкните значок **X** чтобы аннулировать активацию лицензии.




### 2.4.3 Перемещение лицензии на другой компьютер

Вся информация о ваших лицензиях доступна на сайте Bricsys (требуется регистрация): https://www.bricsys.com/protected/account/licenses.

1 Выполните деактивацию лицензии на старом компьютере.

2 Запустите **BricsCAD®** на новом компьютере.

3 Активируйте лицензию на новом компьютере.

### 2.4.4 Перенос сетевой лицензии на другой сервер

Выполните следующие действия:

1 Аннулируйте активацию лицензии.

2 Активируйте лицензию на новом сервере.

3 Закройте менеджер лицензий на старом сервере и удалите файл лицензии на старом сервере.

***Прим.:*** Количество аннуляций лицензии ограничено.

### 2.4.5 Пул лицензий

#### 2.4.5.1 Редактирование параметров

Вы можете редактировать параметры двумя способами:

- Редактирование файла bricsys.opt, расположенного в папке установки Network License Manager.
- Использование веб-интерфейса Network License Manager.
  - Перейдите по адресу *http:/ localhost:5054* на сервере лицензий.
  - Нажмите кнопку **Status**.
  - Щелкните **bricsys** в столбце OPTIONS строки bricsys ISV.

После редактирования выполните перезапуск сервера лицензий.

### 2.4.5.2 Часто используемые параметры

Группы могут использоваться для определения списка имен пользователей, имен хостов или IP-адресов.

Это имя позже можно будет использовать при назначении правил для этой группы.

| | | Синтаксис | Пример: |
|---|---|---|---|
| GROUP | Определяет группу имен пользователей. | GROUP имя_группы список_имен_пользователей | GROUP engineers userA userB userC |
| HOST_GROUP | Определяет группу имен хостов. | HOST_GROUP имя_группы список_имен_хостов | HOST_GROUP firstFloor machineA machineB machineC |
| INTERNET_GROUP | Определяет группу IP-адресов. | INTERNET_GROUP имя_группы список_IP_адресов | INTERNET_GROUP networkGhent 192.168.1. * 192.168.2.* В IP-адресах можно использовать подстановочный знак (*). |
| RESERVE | Зарезервируйте количество лицензий для определенного пользователя или группы пользователей. | RESERVE число продукт пользователь\| хост\|группа\| группа_хостов\| интернет\| интернет_группа\| проект кто [id = nnn] | RESERVE 20 bricscad GROUP engineers |
| MAX | Ограничение количества лицензий, доступных определенному пользователю или группе пользователей. | MAX число продукт пользователь\| хост\|группа\| группа_хостов\| интернет\| интернет_группа\| проект кто [id = nnn] | MAX 5 bricscad HOST_GROUP firstFloor |

| | | Синтаксис | Пример: |
|---|---|---|---|
| INCLUDE | Предоставлять доступ к лицензии на продукт только определенному пользователю или группе пользователей. Любой, кто не указан в строке INCLUDE, не имеет права использовать продукт. | INCLUDE продукт пользователь\| хост\|группа\| группа_хостов\| интернет\| интернет_группа\| проект кто [id = nnn] | INCLUDE bricscad GROUP bricscadusers |
| EXCLUDE | Запретить доступ к лицензии на продукт конкретному пользователю или группе пользователей. | EXCLUDE продукт пользователь\| хост\|группа\| группа_хостов\| интернет\| интернет_группа\| проект кто [id = nnn] | EXCLUDE bricscad INTERNET_GROUP 1.1.*.* |

Дополнительные сведения о параметрах вы можете найти в руководстве пользователя RLM в разделе **The ISV Options File**.

#### 2.4.5.3 Идентификация файла лицензии

Начиная с версии V15, новая функция в диспетчере лицензий Reprise позволяет добавлять в файл лицензии поле _id.

С помощью этого идентификатора _id вы можете определить каждый файл лицензий как отдельный пул лицензий.

При указании параметра [id = nnn] для добавленного OPTION этот OPTION будет иметь эффект только для соответствующего файла лицензии.

Пример использования: у вас есть 2 лицензии, одна на 10 рабочих мест для **BricsCAD® Lite**, а другая на 10 рабочих мест для **BricsCAD® Pro**, и вам необходимо определить, какие пользователи и какой уровень лицензии могут использовать.

**ФАЙЛЫ ЛИЦЕНЗИЙ**

Lite License

HOST host_name ethernet_mac 5053

ISV bricsys

LICENSE bricsys bricscad 15 permanent 10 share=uh

contract=1 options="LA:xx_XX TY:FCO LE:PRO"

_ck=6a06665467 sig=""

HOST host_name ethernet_mac 5053

ISV bricsys

LICENSE bricsys bricscad 15 permanent 10 share=uh

contract=1 options="LA:xx_XX TY:FCO LE:PRO" _id=1

_ck=6a06665467 sig=""

Pro License

HOST host_name ethernet_mac 5053

ISV bricsys

LICENSE bricsys bricscad 15 permanent 10 share=uh

contract=1 options="LA:xx_XX TY:FCO LE:PLA"

_ck=6a06665467 sig=""

HOST host_name ethernet_mac 5053

ISV bricsys

LICENSE bricsys bricscad 15 permanent 10 share=uh

contract=1 options="LA:xx_XX TY:FCO LE:PLA" _id=2

_ck=6a06665467 sig=""

Лицензия Lite определяется идентификатором id=1, а лицензия Pro - идентификатором id=2.

**ПАРАМЕТРЫ**

Если теперь, например, вы хотите, чтобы только инженеры вашей компании могли использовать версию Pro, это можно сделать, добавив следующие параметры.

GROUP engineers userA userB userC

INCLUDE bricscad GROUP engineers id=2

Когда использовать

При необходимости настройки пула лицензий.

Больше информации: http://www.reprisesoftware.com/RLM_Enduser.html

### 2.4.5.4 Создание пула лицензий с использованием RLM_PROJECT

Описанное ниже относится к случаям, когда на одном сервере RLM активируются сетевые лицензии нескольких уровней и пользователи могут решить, какой уровень лицензии необходим ему при активации **BricsCAD®**.

Пользователь может получить определенный уровень лицензии **BricsCAD®** на основе переменной среды RLM_PROJECT, которая должна соответствовать параметрам сервера ISV. Лицензионные места будут зарезервированы IDentifer на сервере ISV для определенных пользователей PROJECT.

***Прим.:*** Приведенные ниже действия должны выполняться сотрудником ИТ-отдела.

**1. Действия с сервером RLM**

1 Добавьте идентификатор _id =<число> в файлы лицензий непосредственно перед строкой _ck = xxxxxxx:

……contract=999-9999-9999-8888 options="LA:xx_XX TY:NFR LE:PRO _id=1 _ck=4b07efdd90….

……contract=999-9999-9999-9999 options="LA:xx_XX TY:NFR LE:LIT _id=2 _ck=4b07efdd90…..

2 Добавьте параметр RESERVE для PROJECT и идентификаторы лицензий для сервера ISV для всех рабочих мест:

RESERVE 5 bricscad PROJECT bc_pro id=1

RESERVE 5 bricscad PROJECT bc_lite id=2

**2. Действия пользователя BricsCAD®**

1 Создайте сценарий, который устанавливает временную переменную среды RLM_PROJECT на пользовательском компьютере, что позволит BricsCAD взаимодействовать с пулом лицензий PROJECT сервера RLM.

2 Выполнение определенного сценария позволит BricsCAD получить необходимый уровень лицензии.

Это могут быть файлы *.CMD (командная строка):

- @echo off
  set RLM_PROJECT=bc_lite
  call "C:\Program Files\Bricsys\BricsCAD V21 en_US\bricscad.exe"
- @echo off
  set RLM_PROJECT=bc_pro
  call "C:\Program Files\Bricsys\BricsCAD V21 en_US\bricscad.exe"
- или *.PS1 (powershell) файлы:
  $env:RLM_PROJECT = "bc_pro"
  Start-Process -FilePath "C:\Program Files\Bricsys\BricsCAD V21 en_US\bricscad.exe"
- $env:RLM_PROJECT = "bc_lite"
  Start-Process -FilePath "C:\Program Files\Bricsys\BricsCAD V21 en_US\bricscad.exe"

Поскольку все лицензионные места RESERVE в параметрах RLM, **BricsCAD®** не сможет получить лицензию, если переменная RLM_PROJECT не существует или если ее значение не соответствует параметрам сервера ISV.

Увеличение количества доступных мест более 5 или уменьшение количества рабочих мест RESERVE менее 5 позволит некоторым лицензиям быть свободно доступными.

RLM_PROJECT также работает с другими параметрами (EXCLUDE/INCLUDE/…), поэтому существует множество других конфигураций.

### 2.4.6 Отчет об использовании сетевой лицензии

Вы можете выполнить проверку использования сетевой лицензии следующими способами:

- Из веб-интерфейса RLM

перейдите по адресу localhost: 5054 / home.asp> Status> License Usage> bricsys

- Из файла "bricsys.dlog", который содержит информацию IN/OUT. Этот файл находится в каталоге *C:\Program Files (x86)\Bricsys\Bricsys Network License Manager*
- Параметр REPORTLOG. Использование этого параметра описано на страницах 77 и 103 руководства RLM:

  http://www.reprisesoftware.com/RLM_License_Administration.pdf

  Его необходимо добавить в файл **bricsys.opt** следующим образом:

  REPORTLOG +D:\ваш_путь\имя_файла.log

  Выполните повторно операцию Reread/Restart Servers.

  Знак + означает, что к файлу добавляются новые данные, в противном случае файл будет перезаписан.
- RLM License Administration Bundle

  https://reprisesoftware.com/admin/software-licensing-downloads.php
- JTB FlexReport

  https://jtbworld.com/jtb-flexreport

## 2.5 Использование сетевой лицензии

### 2.5.1 Перемещение лицензии (роуминг)

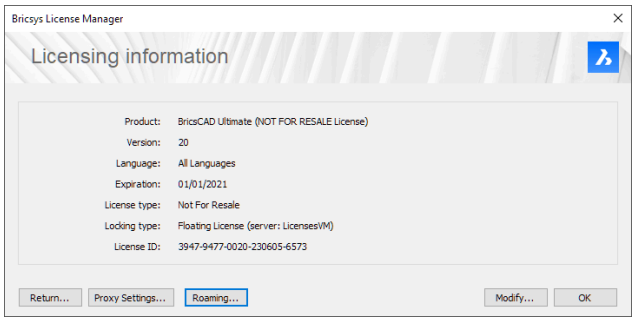
Сетевая лицензия позволяет использовать откреплямую лицензию для тех случаев, когда ваша система временно отключается от сети. Такая лицензия может быть использована в течение определенного количества дней, указанного при установке лицензии, и автоматически возвращается в исходное состояние после завершения этого периода. При необходимости действие откреплямой лицензии может быть завершено раньше указанного срока.

Файл(ы) лицензий на сервере сетевой лицензии содержит параметр *max_roam_count.*Этот параметр определяет общее количество рабочих мест сетевой лицензии по умолчанию.

#### 2.5.1.1 Перемещение сетевой лицензии на компьютер пользователя

Сетевая лицензия может быть перемещена для использования **BricsCAD®** без подключения к серверу лицензий (например, для работы дома или за пределами офиса).

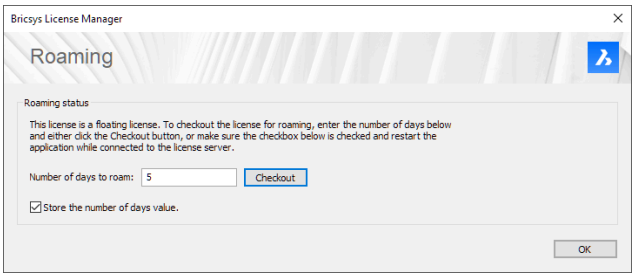
Если сетевая лицензия используется на компьютере пользователя, в диалоговом окне Информация о лицензии отображается кнопка **Роуминг...**




1 Если Ваш компьютер подключен к серверу лицензий через прокси-сервер, нажмите кнопку **Параметры прокси-сервера** для настройки конфигурации прокси-сервера сетевой лицензии.

2 Нажмите кнопку **Роуминг...**.

Result: Откроется диалоговое окно **Роуминг**.




3 Введите количество дней для роуминга лицензии и нажмите кнопку **Checkout**.

4 Выберите опцию сохранения значения количества дней, если Вам необходимо автоматически обновлять период роуминга при каждом подключении Вашего компьютера к серверу.

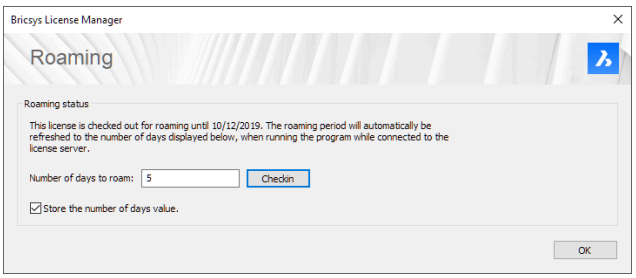
5 Перезапустите **BricsCAD®** для применения изменений.

### 2.5.1.2 Возврат перемещенной лицензии на сервер

Если Вы подключены к серверу, перемещенная лицензия может быть возвращена серверу.

1 Нажмите кнопку **Роуминг...** в диалоговом окне Информация о лицензии.

Result: Откроется диалоговое окно **Роуминг**.




2 Нажмите кнопку **Регистрация**.

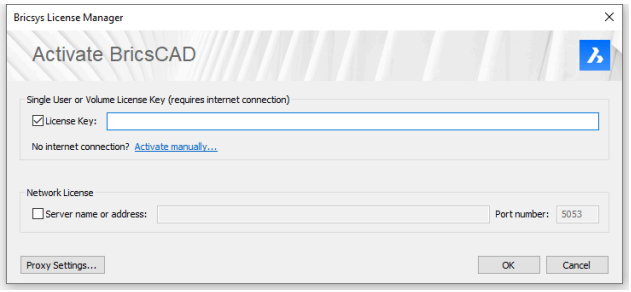
3 Перезапустите **BricsCAD®** для применения изменений.

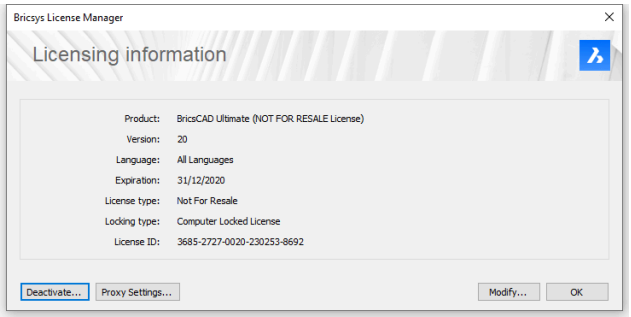
### 2.5.2 Настройки прокси на клиентском компьютере

Подготовка: Некоторые компьютеры подключаются к локальной сети или к Интернет через прокси-сервер. Для таких компьютеров перед выполнением активации или деактивации лицензии необходимо выполнить настройку конфигурации прокси-сервера.

#### 2.5.2.1 Настройка параметров прокси-сервера

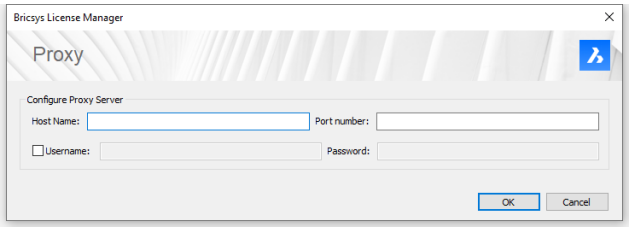
1 Нажмите кнопку **Параметры прокси-сервера...** в диалоговом окне **Активация BricsCAD** или **Информация о лицензии**.




Result: Откроется диалоговое окно **Прокси-сервер**.

2 Заполните необходимые поля в диалоговом окне **Параметры прокси-сервера**.




3 Нажмите кнопку **ОК** для продолжения.

Result:

***Прим.:*** Настройки конфигурации прокси-сервера действуют только для текущей сессии **BricsCAD®**. При необходимости активации или деактивации лицензии в другой сессии потребуется повторная настройка параметров прокси-сервера.

#### 2.5.2.2 Настройка параметров прокси-сервера для использования Сетевой лицензии

Подготовка: Использование Сетевой лицензии требует подключения **BricsCAD®** к серверу лицензии при каждом открытии сессии. Поэтому конфигурирование прокси-сервера должно выполняться постоянно.

На компьютере с **BricsCAD**® выполните следующие действия:

1 Создайте переменную **HTTP_PROXY**: HTTP_PROXY=[СерверЛицензии]:[НомерПорта]

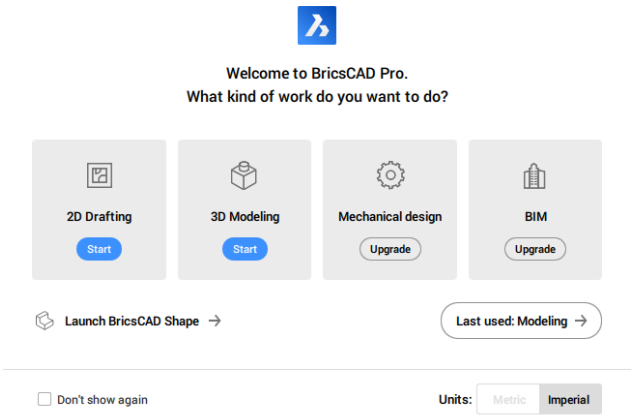
2 Если прокси-сервер поддерживает обычную аутентификацию, создайте переменную **HTTP_PROXY_CREDENTIALS**: HTTP_PROXY_CREDENTIALS=[ИмяПользователя]:[Пароль]

# 3. Начало работы с BricsCAD®

## 3.1 Запуск BricsCAD®

### 3.1.1 Окно запуска BricsCAD

Если для системной переменной GETSTARTED установлено значение ON, при запуске **BricsCAD®** отображается диалоговое окно Запуск BricsCAD, которое позволяет выбрать тип рабочего пространства для текущего сеанса. Доступность разных типов рабочего пространства определяется уровнем лицензии.




- **2D Черчение**: запуск с рабочим пространством 2D чертеж. Требуется лицензия **BricsCAD® Lite** или выше.
- **3D Моделирование**: запуск с рабочим пространством 3D модель. Требуется лицензия **BricsCAD® Pro** или выше.
- **Mechanical**: запуск с рабочим пространством Mechanical. Требуется лицензия **BricsCAD® Mechanical** или **BricsCAD® Ultimate**.
- **BIM**: запуск с рабочим пространством BIM. Требуется лицензия **BricsCAD® BIM** или **BricsCAD® Ultimate**.
- **Запустить BricsCAD Shape**: запуск **BricsCAD® Shape**. Лицензия не требуется.
- **Последнее**: запуск BricsCAD с последним из использованных рабочих пространств.
- **Не показывать это окно**: установка для переменной GETSTARTED значения 0. Чтобы восстановить отображение диалогового окна Запуск BricsCAD, введите в командной строке **getstarted** и установите для этой переменной значение **1 (ON)**.
- **Единицы**: установка единиц измерения для шаблона чертежа по умолчанию. Определяет значение системной переменной BASEFILE.

***Прим.:*** Бесплатный инструмент для твердотельного моделирования **BricsCAD® Shape** встроен в дистрибутив **BricsCAD®**. После завершения 30-дневного ознакомительного периода **BricsCAD®**

**Ultimate** вам предоставляется бессрочная лицензия **BricsCAD® Shape**. Удаление **BricsCAD®** при этом не требуется.

## 3.2 Интерфейс

### 3.2.1 Закладка Начало работы

Закладка **Начало работы** отображается в верхней части графической области окна BricsCAD перед закладками открытых чертежей.

С помощью закладки **Начало работы** вы можете:

- **Открыть последний чертеж**

  Чтобы открыть один из последних чертежей, щелкните на эскизном изображении содержимого выбранного файла.

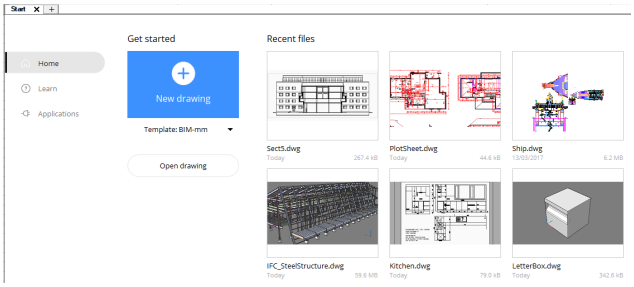
  Количество последних файлов, доступных на закладке Начало работы, определяется значением системной переменной RECENTFILES.

- **Создать новый чертеж**

  Щелкните на синем прямоугольнике с надписью **Создать чертеж**, чтобы создать новый чертеж с использованием текущего шаблона. При необходимости раскройте выпадающий список **шаблонов чертежа**, чтобы выбрать другой шаблон для создания нового чертежа.

- **Открыть чертеж**

  Отображает диалоговое окно **Открыть чертеж**.




Закладка **Начало работы** открывается по умолчанию. Если вы случайно закрыли эту закладку, вы можете снова открыть ее с помощью сочетания клавиш **Ctrl + Home** или команды GOTOSTART.

***Прим.:*** При использовании сочетания клавиш **Ctrl+Home** необходимо выключить режим Num Lock.

***Прим.:*** Для **включения/выключения** отображения закладки **Начало работы** используется системная переменная SHOWDOCTABS.

### 3.2.2 Закладки чертежей

Для каждого открытого чертежа в верхней части графической области отображается закладка с именем этого чертежа.

Закладки чертежей позволяют:

- Быстро переключаться между всеми открытыми чертежами.
- Создавать новый чертеж нажатием кнопки **+**, расположенной справа от последней закладки.
- Закрывать чертеж щелчком на значке **X**, расположенном на закладке этого чертежа.

### 3.2.3 Выбор рабочего пространства

По умолчанию доступны следующие рабочие пространства:

- **2D чертеж**: используется лента.
- **2D чертеж (панели)**: используются меню и панели инструментов.
- **3D модель**: используется лента; необходима лицензия **BricsCAD® Pro** и выше.
- **3D модель (панели)**: используются меню и панели инструментов; необходима лицензия **BricsCAD® Pro** и выше.
- **Mechanical**: необходима лицензия **BricsCAD® Mechanical** или **BricsCAD® Ultimate**.
- **BIM**: необходима лицензия **BricsCAD® BIM** или **BricsCAD® Ultimate**.

Текущее рабочее пространство определяет конфигурацию и общую компоновку рабочего окна **BricsCAD®**.

Рабочее пространство определяет состав:

- Меню
- Панели инструментов
- Панели
- Лента
- Основы работы с квадро-меню

Для выбора текущего рабочего пространства вы можете использовать:

- Диалоговое окно Запуск BricsCAD.
- Поле Рабочее пространство в строке состояния.
- Панель Рабочие пространства.
- Команду WORKSPACE.

### 3.2.4 Графическая область

Работу с чертежом Вы начинаете в рабочем пространстве, которое называется **Пространство модели**. Пространство модели - это область, в которой вы создаете 2D и 3D объекты в Мировой системе координат (МСК) или Пользовательской системе координат (ПСК).**BricsCAD®** позволяет также использовать дополнительное рабочее пространство, которое называется **Пространство листа**. Каждый лист в пространстве листа представляет собой компоновку вашего чертежа на листе бумаги. Для одного чертежа вы можете создать до 255 листов.

Для работы в пространстве модели используется закладка **Модель**. Для переключения на пространство листа используются **закладки листов**. Закладки модели и листов расположены в нижней части графического области.

### 3.2.4.1 Пространство модели

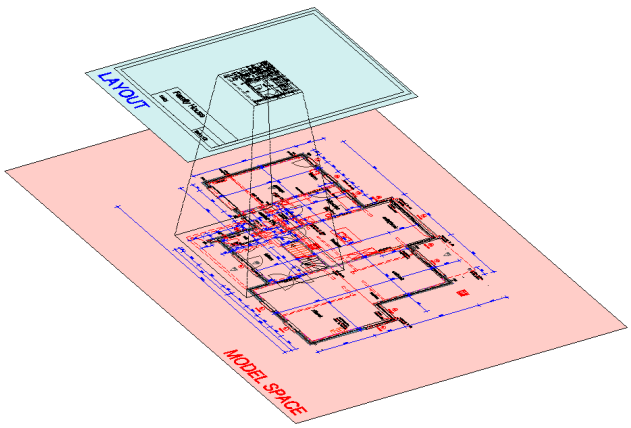
Пространство модели состоит из одного вида, который заполняет всю графическую область окна BricsCAD. При необходимости вы можете создать дополнительные виды (видовые окна), которые будут отображать различные виды вашего чертежа или 3D модели. Для управления видовыми окнами используется команда VPORTS .

Видовые окна располагаются в виде плиток. Одновременно вы можете работать только с одним из видовых окон, которое называется текущим. При этом обновление всех остальных видовых окон выполняется автоматически. Щелчок внутри видового окна делает его текущим.

### 3.2.4.2 Пространство листа

Пространство листа представляет собой компоновку вашего чертежа на листе бумаги. В этой рабочей области вы можете создавать и располагать различные виды вашего чертежа подобно тому, как выполняется чертеж на листе бумаги, а также добавлять аннотации, рамки, штампы, титульные надписи и другие выводимые на печать объекты, которые нет необходимости отображать в пространстве модели.

Каждый чертеж имеет по крайней мере один **Лист**. Каждый лист состоит из одного или нескольких видовых окон. Такие видовые окна можно рассматривать как окна вида вашего чертежа в пространстве модели.




Видовые окна листа могут располагаться в любой области пространства листа. Каждое видовое окно имеет свой собственный масштаб и настройки видимости слоев. Все видовые окна одного листа могут быть выведены на печать одновременно.

Для управления видовыми окнами листа используется команда MVIEW.

Использование пространства листа предоставляет ряд преимуществ:

- Создание нескольких видовых окон для печати различных видов. Каждый из этих видов может отображать часть чертежа с использованием одного или различных масштабов.

- Создание нескольких листов для печати одного и того же чертежа с различными настройками печати, такими как ширина пера, конфигурация принтера, вес линий, масштаб чертежа и другие.
- Добавление объектов, которые не связаны непосредственно с пространством модели, таких как аннотации, рамки, штампы, титульные надписи, и другие.

### 3.2.5 Командная строка

Командная строка - это закрепляемая панель, которая по умолчанию располагается в нижней части графической области.

Командная строка обеспечивает быстрый и эффективный запуск команд и установку значений системных переменных и пользовательских настроек.

Командная строка **BricsCAD®** используется для ввода данных и отображения результатов выполнения таких команд, как DIST, AREA или LIST.

При скрытии командной строки для ввода команд и их параметров используется Строка состояния.

### 3.2.6 Меню

По умолчанию строка меню скрыта во всех рабочих пространствах.

Содержимое меню зависит от текущего рабочего пространства.

#### 3.2.6.1 Управление отображение строки меню

1 Установите курсор над панелью инструментов или лентой.

2 Щелкните правой кнопкой мыши и в открывшемся контекстном меню выберите **Строка меню**.

### 3.2.7 Панели инструментов

По умолчанию панели инструментов используются только в рабочих пространствах 2D чертеж (панели) и 3D модель (панели), но могут применяться и в любом другом рабочем пространстве.

- Вы можете использовать для кнопок панелей инструментов мелкие значки (16 x 16), крупные значки (32 x 32) или огромные значки (64x64).
- Для некоторых инструментов доступны выпадающие панели. На кнопках таких инструментов имеется маленькая черная стрелка. Выпадающая панель объединяет группу связанных инструментов. Чтобы получить доступ к инструменту выпадающей панели, щелкните на кнопке с черной стрелкой левой кнопкой мыши и, удерживая ее нажатой, выберите на открывшейся панели необходимый инструмент. Последний выбранный инструмент остается видимым после свертывания выпадающей панели.
- Вы можете открыть или закрыть панель инструментов в любое время.

### 3.2.8 Лента

Лента представляет собой набор панелей инструментов, которые отображаются при выборе соответствующей закладки и содержат сгруппированные по функциям кнопки инструментов и другие элементы управления.

Содержимое ленты зависит от текущего рабочего пространства.

В рабочих пространствах 2D чертеж, 3D модель, Mechanical и BIM лента отображается по умолчанию.

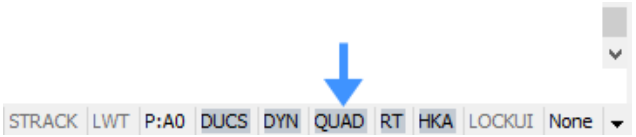
Для включения/выключения отображения ленты используются команды RIBBON и RIBBONCLOSE.

### 3.2.9 Основы работы с квадро-меню

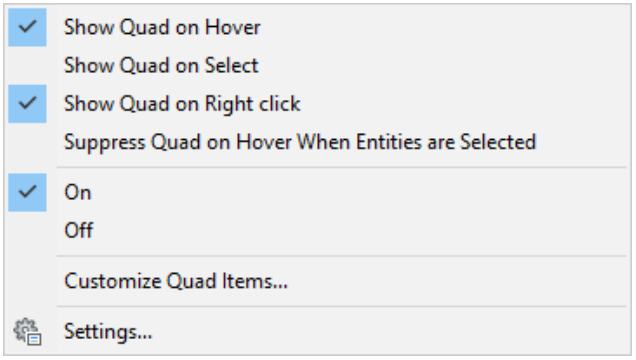
**Квадро-меню** - это элемент интерфейса BricsCAD, который используется для запуска команд и отображается непосредственно под курсором мыши.

При перемещении курсора по экрану или выборе объектов квадро-меню определяет тип объекта и предоставляет пользователю набор команд, которые могут быть использованы для работы с этим объектом или набором выбранных объектов. При отсутствии выбранных объектов вы можете использовать квадро-меню для создания объектов, вставки блоков, подключения подложек или открытия диалоговых окон **Параметры** и **Проводник по чертежам**.

- Чтобы включить/выключить отображение **квадро-меню**, щелкните на поле КВАДРО в строке состояния, которая находится в нижней части окна **BricsCAD®**.




- Чтобы выполнить настройку параметров квадро-меню, щелкните правой кнопкой на поле **КВАДРО** в строке состояния и в открывшемся контекстном меню выберите пункт Параметры.

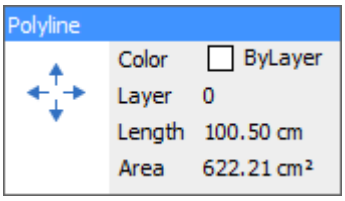



#### 3.2.9.1 Основные процедуры при редактировании объектов с помощью квадро-меню

1 Подведите курсор к объекту. При выделении объекта квадро-меню отображает последний использованный инструмент и тип выделенного объекта.

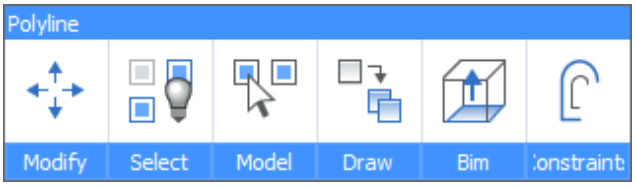



Если значение переменной ROLLOVERTIPS = ON, в квадро-меню будут отображаться свойства выделенного объекта.

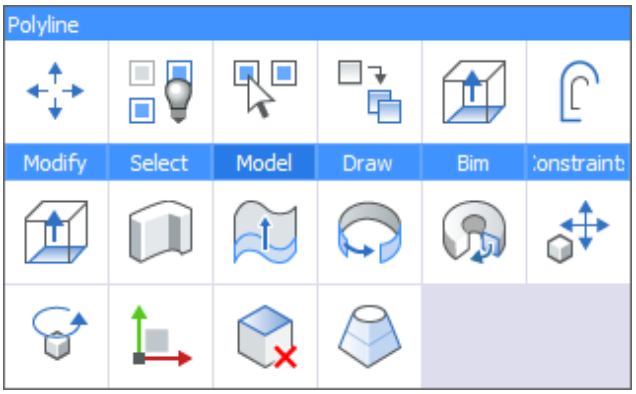



2 Переместите курсор на квадро-меню для отображения набора последних команд, использованных для объектов данного типа.

Внизу будут отображаться закладки категорий команд.




3 Чтобы раскрыть группу команд, поместите курсор над закладкой этой группы.




4 Для запуска команды нажмите на кнопку, соответствующую этой команде.

### 3.2.9.2 Использование квадро-меню без выбора объектов

Убедитесь, что в настройках параметров квадро-меню включен параметр **Отображать квадро-меню по щелчку правой кнопкой мыши**.

1 Выполните одно из следующих действий:

- Переместите курсор в пустую область окна чертежа и щелкните правой кнопкой мыши.
- Если курсор находится над объектом, для отображения квадро-меню без выбора объектов нажмите и удерживайте правую кнопку мыши.

Откроется **квадро-меню** со значками последних использованных команд.




2 Выполните одно из следующих действий:

- Выберите одну из последних использованных команд.
- Переместите курсор на закладку категории команд и выберите необходимую команду.

3 Выполните запуск выбранной команды.

### 3.2.10 Контекстное меню правой кнопки мыши

Контекстное меню представляет собой список команд, который отображается рядом с курсором при щелчке правой кнопкой мыши.

При использовании квадро-меню для отображения контекстного меню необходимо более длительное удерживание правой кнопки мыши.

Содержимое контекстного меню зависит от текущего объекта или набора выбранных объектов.

Если квадро-меню не используется и объекты не выбраны, щелчок правой кнопкой мыши повторно запускает последнюю команду. Для отображения контекстного меню в этом случае также необходимо более длительное удерживание правой кнопки мыши.

### 3.2.11 Строка состояния

Строка состояния располагается в нижней части графической области.

- При подведении курсора к полям строки состояния отображаются соответствующие подсказки.
- При закрытом окне командной строки строка состояния используется для ввода команд и выбора их опций.
- Строка состояния предоставляет возможность просмотра и редактирования текущих параметров создания объекта: слой, цвет, тип линии.

  ***Прим.:*** Примечание: По умолчанию эти поля отключены во всех рабочих пространствах.
- С помощью строки состояния вы также можете выполнить просмотр и выбор текущего размерного стиля и стиля текста.
- Кроме того, строка состояния позволяет включить/выключить шаговую и объектную привязку, вспомогательную сетку, объектную трассировку и другие вспомогательные средства черчения с отображением их текущего состояния.
- ...

Если щелкнуть небольшую черную кнопку со стрелкой у правого края строки состояния, отобразится список всех доступных для отображения полей.

### 3.2.12 Панели

Панели и закрепляемые диалоговые окна.

Доступные панели:

- Панель Редактор анимации
- Вложения
- BIM BCF
- Композиты BIM
- Профили BIM
- Диспетчер проекта BIM
- Диспетчер спецификаций
- Bricsys 24/7
- Командная строка
- Обозреватель чертежей
- Подробности
- Стили чертежа
- Слои
- Библиотека
- Обозреватель конструкции
- Параметры и зависимости
- Свойства
- Материалы для визуализации
- Отчет
- Лента
- Подшивки
- Структура
- Ассистент
- Инструментальные палитры

### 3.2.13 Полноэкранный режим

При использовании полноэкранного режима вы можете максимизировать область черчения, скрыв такие элементы пользовательского интерфейса, как строка меню, лента, панели инструментов, панели, командная строка, строка состояния и закладки документов.

В этом режиме для запуска команд вы можете использовать квадро-меню.

Для включения/выключения полноэкранного режима используйте сочетание клавиш Ctrl+0.

***Прим.:*** Не используйте клавиши цифрового блока.

***Прим.:*** На клавиатурах с раскладкой AZERTY не нажимайте клавишу Shift при нажатой клавише 0.

### 3.2.14 Виджет 3D видов

Виджет 3D видов позволяет управлять отображением вашего рисунка. Вы можете выбрать один из нескольких предустановленных видов, включающих как ортогональные, так и изометрические виды.



По умолчанию **виджет 3D видов** отображается в правом верхнем углу графической области.

Поведение **виджета 3D видов** определяется значениями нескольких системных переменных и пользовательских настроек.

## 3.3 Работа в BricsCAD®

### 3.3.1 Создание чертежа

В BricsCAD доступны следующие способы создания новых чертежей:

- С использованием закладки **Начало работы**.
- С использованием команды QNEW.
- С использованием команды NEW.
- С использованием команды NEWWIZ.
- Двойной щелчок на *.dwt файле в окне Проводник Windows.

***Прим.:*** Новый чертеж всегда создается как копия шаблонного чертежа. Текущий файл шаблона определяется значением системной переменной BASEFILE.

### 3.3.2 Template drawings

A template drawing is a drawing file with file extension DWT.

It is used to create a new drawing that is a copy of the template drawing. Any drawing with extension DWT is considered a template drawing. The BricsCAD® template drawings are located in the template folder(s). Use the TEMPLATEFOLDER command to open the folder. If you save your customized template drawings in the template folder, they are available in the Start tab to create new drawings. Using properly prepared templates is essential for a productive use of BricsCAD®.

A template drawing does not contain entities in model space, but contains settings according to the specific drawing and project requirements:

- Units: imperial or metric and units display.
- Layers.
- Dimension and leader styles.
- Text styles.
- Table styles.
- Multiple layouts, each with title block, paper size, printer, and plot style.
- Standard blocks for notes, tables, symbols, and other entities.

### 3.3.3 Параметры создания объектов

Новые объекты создаются на текущем слое, с использованием текущих параметров создания объекта.

- Цвет
- Тип линии
- Масштаб типа линии
- Толщина линии
- Прозрачность

Для определения параметров создания объектов вы можете использовать раздел **Общие** на Панели свойств.

При необходимости вы можете установить для свойств объекта значение **ПоСлою**. Это означает, что в качестве свойств этого объекта будут использоваться соответствующие параметры текущего слоя.

Для настройки параметров слоя вы можете использовать раздел Слои диалогового окна Проводник по чертежам.

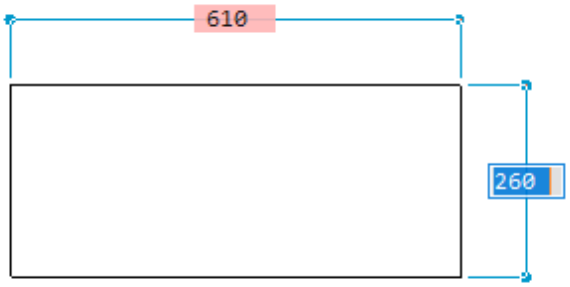
### 3.3.4 Обеспечение точности построений

Новые объекты создаются в плоскости XY текущей системы координат. По умолчанию это Мировая система координат (МСК). Все размеры в файле DWG выражаются в единицах чертежа. Системная переменная INSUNITS (Единицы измерения при вставке) устанавливает значение 1 единицы измерения.

Для новых чертежей значение INSUNITS определяется в шаблоне чертежа, который был выбран для создания нового чертежа. **BricsCAD®** предлагает набор шаблонов чертежей с использованием десятичных и английских единиц измерения.

В **BricsCAD®** доступны следующие средства обеспечения точности построений:

- Объектная 2D привязка: позволяет выбрать конечную или среднюю точку, точку центра, точку пересечения или другую характерную точку 2D объекта. С помощью объектной привязки вы также можете вычерчивать объекты, касательные, параллельные или перпендикулярные к существующим объектам чертежа.
- Объектная 3D привязка: позволяет использовать при создании и редактировании геометрии объектов характерные точки 3D объектов.
- Динамические размеры. При **включенном** режиме динамических размеров вы можете вводить значения расстояний и углов в соответствующих полях динамических размеров. Если отображается несколько полей динамических размеров, например при построении прямоугольника, для переключения на следующее поле нажмите клавишу TAB. После ввода значения и нажатия клавиши Tab поле введенного вами размера изменяет свой цвет на красный. Это означает, что введенное значение зафиксировано.




- Полярная трассировка позволяет выполнять построения с точным соблюдением заданных углов по отношению к осям (X, Y, Z) текущей системы координат.
- Объектная трассировка представляет собой комбинацию объектной привязки и полярной трассировки.
- Ввод координат позволяет вводить координаты точек X, Y и Z при указании их положения.
- Шаговая привязка и вспомогательная сетка. Вспомогательная сетка представляет собой набор равноудаленных линий или точек, которые служат для визуального отображения расстояний. Шаговая привязка с возможностью привязки курсора к точкам вспомогательной сетки позволяет выполнять построения с заданным единичным интервалом.
- Границы чертежа позволяют ограничить область чертежа, что предотвращает создание объекта за пределами этого чертежа.
- Орто-режим ограничивает перемещение курсора направлениями, параллельными осям X и Y текущей системы координат.
- Пользовательские системы координат. Вы можете определить именованные пользовательские системы координат (ПСК), расположенные в любом месте трехмерного пространства. Вы можете сохранять именованные ПСК и использовать их в любое время.
- Прямой ввод расстояния позволяет выполнять ввод длины, радиуса, диаметра и вектора перемещения непосредственно в командной строке.
- Функция Наименьшее расстояние позволяет динамически просматривать и изменять расстояние между двумя выбранными объектами или подобъектами.
- Манипулятор - это универсальный инструмент, который позволяет быстро перемещать , вращать , масштабировать и отражать выбранные объекты.

### 3.3.5 Executing commands

In BricsCAD® you can launch commands in the following ways:

- Type the command in the Command Line.
- Choose the command in a menu.
- Click the command tool button in a toolbar.
- Click the command icon in the Ribbon.

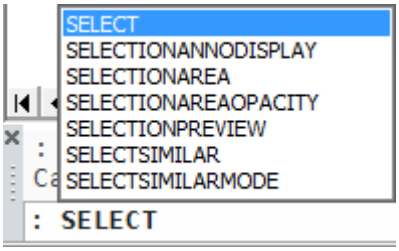
- Choose the command in the Quad.

#### 3.3.5.1 Typing a command in the Command line

If you know the command name, you can enter it in the Command line. When the command line is closed, keyboard entry and command prompts display in the status bar.

If the **Enable** flag of the AUTOCOMPLETEMODE system variable is set, command names are automatically completed when you type.

All commands and/or system variables and/or user preferences that contain the current character string display in the suggestion list.




Do one of the following:

- Press Enter if the currently highlighted command displays.
- Click a command in the suggestion list.

You can launch some commands with a command alias. For example, you can launch the LINE command by entering L in the command prompt.

#### 3.3.5.2 Choosing the command options

Many BricsCAD® commands provide multiple options. These options appear in the Command line or status bar (when the Command line is closed) and optionally also in a prompt menu.

Do one of the following:

- Press Enter to choose the default option. The default option is located at the end of the command prompt between arrow brackets (<>).
- Click the option in the Command line.
- Click the option in the prompt menu.
- Type the capitalized letter of the option and press Enter or right-click.

### 3.3.6 View control

In BricsCAD® you can control the display of your drawing in various ways:

- By using the mouse and keyboard
- By using View commands
- By using the Look From widget

The most frequently used view manipulations are:

| Action | Mouse | Command | Look From |
|---|---|---|---|
| View Extents | Double-click the scroll wheel | **Zoom** > **Extents** | Clicking a trigger zone rotates the view and zooms to the extents of the drawing. |
| Zoom | Scroll wheel | **Zoom** > **InZoom** > **Out** | |
| Pan | Hold down the scroll wheel<br>***Прим.:*** When the MBUTTONPAN system variable = ON | Pan | |
| Restore Plan View (Top View) | | **Plan** > **Current UCS** | Click the center trigger zone |

### 3.3.7 Выбор объектов

Такие команды управления объектами, как MOVE , COPY и MIRROR, предлагают вам выбрать один или несколько объектов. Если для переменной PICKFIRST установлено значение ON, вы можете создать набор выбранных объектов перед запуском применяемой к ним команды.

**BricsCAD**® предлагает несколько методов выбора объектов. Вы можете комбинировать различные методы выбора и создавать наборы выбранных объектов.

#### 3.3.7.1 Выбор объектов по одному

1 Подведите курсор к объекту.

Result: Объект будет выделен.

Рядом с курсором отобразится квадро-меню со значком последней из использованных команд и набором свойств выделенного объекта.

2 Выполните одно из следующих действий:

- Щелкните левой кнопкой мыши, чтобы выбрать выделенный объект.
- Несколько раз нажмите клавишу TAB, чтобы выделить скрытые объекты. После выделения необходимого объекта щелкните левой кнопкой мыши для выбора этого объекта.

3 Повторите предыдущие шаги для выбора других объектов или запустите команду.

#### 3.3.7.2 Выбор нескольких объектов

Для одновременного выбора нескольких объектов в **BricsCAD**® используется рамка выбора:

1 Укажите щелчком левой кнопки мыши угол прямоугольной рамки выбора.

2 Выполните одно из следующих действий:

- Переместите курсор вправо. На экране отобразится голубой прямоугольник. **Окружность внутри прямоугольника** на значке, который отображается рядом с курсором при создании рамки выбора, указывает, что будут выбраны все объекты, полностью лежащие в пределах прямоугольной рамки выбора.
- Переместите курсор влево. На экране отобразится зеленый прямоугольник. **Окружность, пересекающая прямоугольник** на значке, который отображается рядом с курсором при создании рамки выбора, указывает, что будут выбраны объекты, полностью лежащие в пределах прямоугольной рамки выбора и объекты, пересекаемые этой рамкой.

3 Укажите щелчком левой кнопки мыши противоположный угол прямоугольной рамки выбора.

### 3.3.7.3 Удаление объектов из набора выбранных объектов

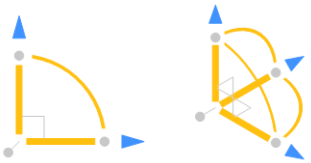
Чтобы удалить объекты из набора выбранных объектов, нажмите клавишу Shift и, удерживая ее, выберите объекты для удаления.

Чтобы отменить выбор объектов, нажмите клавишу Esc.

## 3.3.8 Using the Manipulator

The Manipulator is a versatile grip tool to swiftly move, rotate, scale and mirror selected entities.

If you hold down the Ctrl key during one of these actions, the selected entities are copied.

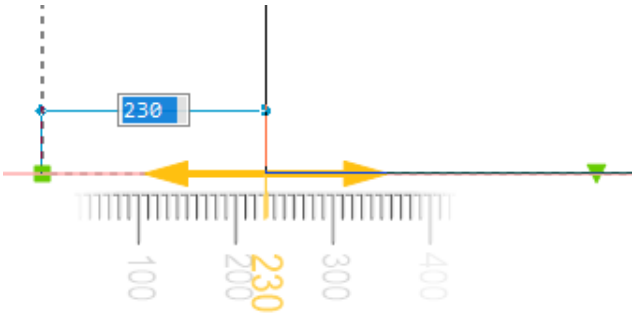


By default, the Manipulator displays when selecting an entity by a long click (keeping the left mouse button down a bit longer).

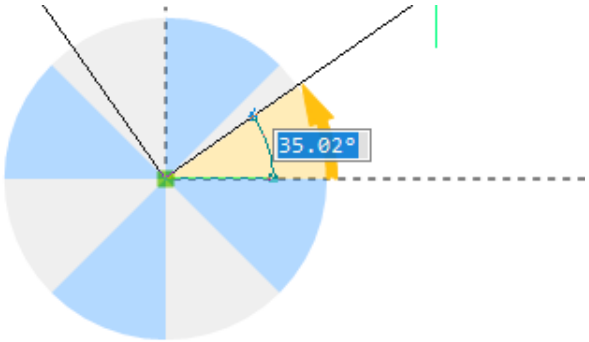
The anchor point of the Manipulator is located at the closest grip point, with respect to the selection point.

Place the cursor over one of the trigger zones, to start an action:

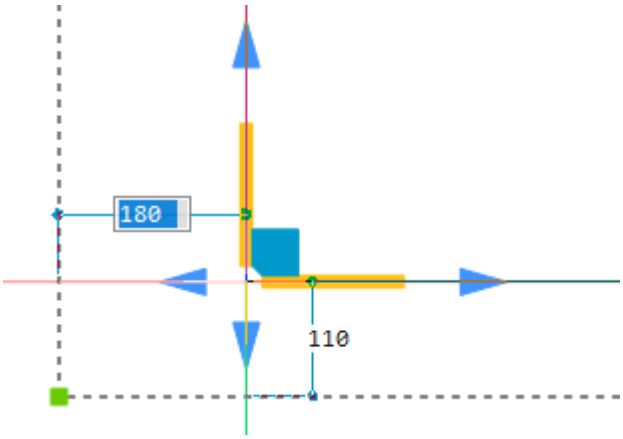
- Axes, to move along the axis:



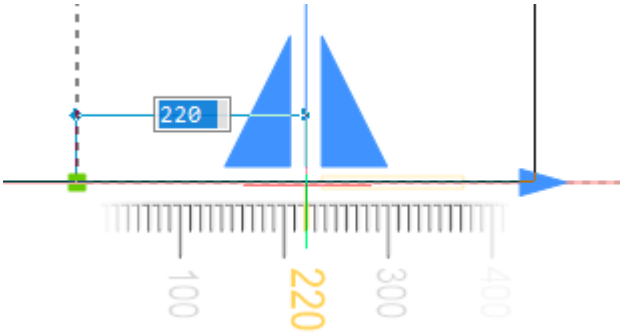

- Arcs, to rotate:




- Plane swatches (= squares at the anchor point), to move freely in the selected plane (XY, XZ or YZ):




- Arrow heads, to mirror:




***Прим.:*** The step size of the ruler depends on the zoom factor. Zoom in to decrease the step size, zoom out to increase.

### 3.3.9 Панель Структура

Панель Структура отображает структуру чертежа в виде дерева.

**BricsCAD**® предлагает три конфигурации дерева структуры (CST):

- **По умолчанию** - отображение структуры на основе слоев.
- **BIM** - отображение структуры на основе объектов BIM.
- **Mechanical** - отображение структуры на основе элементов.

Для выбора необходимой конфигурации нажмите кнопку Меню ().

На панели Структура вы можете:

- Развернуть узловой элемент структуры, щелкнув на значке (**+**).
- Свернуть узловой элемент структуры, щелкнув на значке (**-**).
- Развернуть все подузлы выбранного узлового элемента, щелкнув на нем правой кнопкой мыши и выбрав в открывшемся контекстном меню **Развернуть все** .

  При использовании этой опции для всего чертежа будут развернуты все узловые элементы структуры выбранного чертежа.
- Свернуть все подузлы выбранного узлового элемента, щелкнув на нем правой кнопкой мыши и выбрав в открывшемся контекстном меню **Свернуть все** .

  При использовании этой опции для всего чертежа будут свернуты все узловые элементы структуры выбранного чертежа.
- Выбрать объект на чертеже, щелкнув на соответствующем ему элементе панели Структура.

  ***Прим.:*** Это зависит от параметров **Дерева структуры**. Чтобы настроить дерево структуры, щелкните правой кнопкой мыши на панели **Структура** > **Настроить** > **Параметры**.

  Для создания набора выбранных объектов используйте клавишу Ctrl.
- Открыть контекстное меню, щелкнув правой кнопки мыши. Контекстное меню содержит следующие опции:
  - **Отобразить**: отображение ранее скрытых выбранных объектов.
  - **Скрыть**: скрытие выбранных объектов.
  - **Изолировать**: изоляция выбранных объектов и скрытие остальных объектов.
  - **Показать**: масштабирование с отображением выбранных объектов.

***Прим.:*** Выберите чертежа. Щелкните на нем правой кнопкой мыши и в открывшемся контекстном меню выберите **Показать**, чтобы отобразить весь чертеж, или выберите инструмент **Показать объекты** на закладке квадро-меню **Общие** при отсутствии выбранных объектов.

# 4. Пользовательский интерфейс

## 4.1 The BricsCAD application window

The components of the BricsCAD application window are:

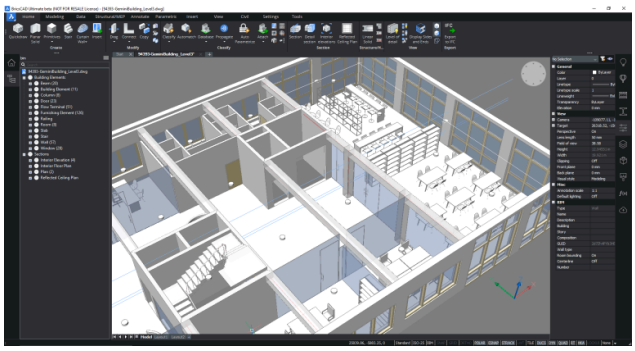
- Menu bar
- Toolbars
- Ribbon Document tabs
- Model space viewports
- Paper space viewports
- Model and Layout tabs
- Dockable panels
- Command line
- Status Bar

The layout of the BricsCAD application window is controlled by the current Workspace.

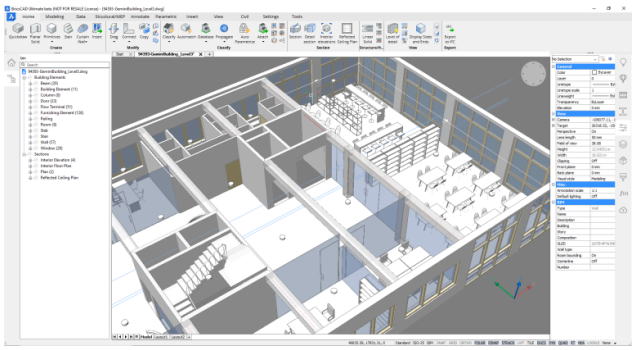
### 4.1.1 The color theme

The COLORTHEME system variable sets the color theme for the ribbon, panels, dialog boxes, etc. , You can choose between a dark or a light color theme.

Dark color theme (COLORTHEME = 0) :



Light color theme (COLORTHEME = 1):



To set the color theme, do one of the following:

- Type **colortheme** in the command line and choose1for light or0for dark.
- Right-click on a toolbar or ribbon panel and select**Dark Interface**in the context menu.

### 4.1.2 Menu bar

The menu bar sits at the top of the BricsCAD application window.

Right click a toolbar, ribbon or panel, then choose Menu Bar in the context menu to toggle the display of the menu bar on/off.

The status of the menu bar is saved through the MENUBAR system variable.

### 4.1.3 The ribbon

To display or hide the ribbon, do one of the following:

- Execute theRIBBONcommand or the RIBBONHIDE command.
- Right click a toolbar, a panel or an empty space in the Layout or Drawing tab area, then choose **Panels** and click **Ribbon**in the context menu.

The size of the tool buttons in the Ribbon is controlled through the RIBBONTOOLSIZE system variable. You can choose between small, large or extra-large buttons.

### 4.1.4 Document tabs

Document tabs sit at the top of the graphical screen by default. They allow you to easily switch between open drawings and to access the Start tab.

The display of the document tabs is controlled through the SHOWDOCTABS system variable.

### 4.1.5 Model and layout tabs

At the bottom of the graphic screen sit the Model and one or more Layout tabs which allow you to easily switch between model space and paper space.

### 4.1.6 Scroll bars

Scrollbars are off by default.

To toggle scroll bars on/off:

- Hold down the Shift key and press the**F4**function key.
- Choose**Scroll Bars**in the**View**menu.
- Launch the SCROLLBAR command.

### 4.1.7 The Command line

Execute the COMMANDLINE command to display the Command line.

Double click the left hand side of the status bar to toggle the display of the Command line.

### 4.1.8 The status bar

The Status bar sits at the bottom of the BricsCAD application window.

Right click a toolbar, ribbon or panel, then choose **Status Bar** in the context menu to toggle the display of the status bar on/off.

### 4.1.9 Hiding a selection of user interface elements

TheCLEANSCREENONcommand allows you to hide a selection of user interface elements simultaneously.

The CLEANSCREENOPTIONS system variable defines which user interface elements are hidden.

The **Ctrl+0** keyboard shortcut toggles the cleanscreen state.

On AZERTY-keyboards:

- Do not use the numeric keypad.
- Do not press the Shift-key when hitting the number key.

## 4.2 The Command line

To display the Command line, do one of the following:

- Launch theCOMMANDLINEcommand.
- Use the Shift + F2 keyboard shortcut.
- Move the cursor over a docked toolbar, the ribbon or a panel icon, then right-click and choose**Panels** > **Command line** in the context menu.
- Double-click the far left side of the**Status** bar.

### 4.2.1 Setting the Command line state

The Command line can either be docked or floating.

By default, the Command line is docked at the bottom of the BricsCAD application window:




To modify the height of the Command line:

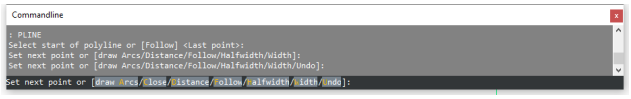
1 Place the cursor over the top edge of the Command line.

2 Press and hold the left mouse button when a double arrow appears.

3 Move the mouse.

Click the**Close**(x) button to close the Command line.

Move the cursor over the dotted left hand side, then press and hold the left mouse button to make the Command line floating:




When the Command line is floating, set the value of theCMDLINEUSENEWFRAME (command line use new floating frame) system variable = 1 to minimize the Command line to a single line. When executing a command, a number of prompt lines are displayed temporarily and transparently on the graphic screen:




The number of prompt lines is set by the CLIPROMPTLINES system variable.

Click the up arrow at the right hand side to display the **Prompt History** window.

When you move the Command line mini-frame near the bottom of the graphic screen, it will stick there.

***Прим.:***

- You can browse the command history with the up arrow and down arrow keys of the keyboard. Press Enter to restart a command.
- When the Command line is closed, command options and keyboard entries are displayed in the **Status** bar.
- Press the F2 function key to toggle the**Command History**window.

### 4.2.2 Launching a command in the Command line

If the **Enable** flag of theAUTOCOMPLETEMODEsystem variable is set, command entries are completed automatically as you type.

All commands and/or system variables and/or user preferences that contain the current character string are displayed in the**Suggestion**list.

Do one of the following:

- Press Enter to choose the highlighted command.
- Use the down or up arrow key to choose a command, then press Enter.
- Click a command in the list.
- Complete the command name, then press Enter.

### 4.2.3 Setting the Command line properties

1 Open the**Settings**dialog box.

2 Go to**Program Options** > **User Preferences** > **Command Line**.

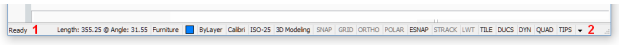
## 4.3 The Status bar

To display the **Status** bar, do one of the following:

- Move the cursor over a docked toolbar, the ribbon or a panel icon, then right-click and choose **Status Bar**in the context menu.
- Press the **Shift + F3** keyboard shortcut.

The **Status** bar closes if it was open, and vice versa.

### 4.3.1 The Status bar fields




### 4.3.2 1. Status field

- Displays the status of the software.

- When the cursor is in a menu or on a toolbar, it gives a brief description of the tool or menu item under the cursor.
- When the command window is closed, it displays the command prompts, tool options and keyboard entry.
- Double-click the status field to open or close the **Command line**.

### 4.3.3 2. Field List button

Click the arrow button (▼) to display a list of available **Status** bar fields. Click an item in the list to toggle the display of the corresponding field in the **Status** bar:

### 4.3.4 Cursor coordinate values

Displays the coordinates of the current cursor position (depending on the value of the COORDS system variable).

- Click the field to cycle the **Coordinate** setting.

  The sequence is: **Geographic** - **Relative** - **Absolute** - **Off**.
- Right-click and choose an option in the context menu.

### 4.3.5 Current Layer

Displays the name of the current layer.

- Click to open the **Layer Explorer**.
- Right-click to select the current layer in the context menu.
- Right-click and choose **Properties** in the context menu to open the **Drawing Explorer – Layers** dialog box.

### 4.3.6 Current Color

Displays the current color.

- Click to open the **Select Color** dialog box.
- Right-click to select one of the basic colors in the context menu.
- Right-click and choose **Select Color...** to select a color in the **Select Color** dialog box.

### 4.3.7 Current Linetype

Displays the name of the current linetype.

- Click to open the **Drawing Explorer – Linetypes** dialog box.
- Right-click to select the current linetype in the context menu.

### 4.3.8 Current Text Style

Displays the name of the current text style.

- Click to open the **Drawing Explorer - Text Styles** dialog box.
- Right-click to select the current text style in the context menu.

### 4.3.9 Current Dimension Style

Displays the name of the current dimension style.

- Click to open the **Drawing Explorer - Dimension Styles** dialog box.
- Right-click to select the current dimension style in the context menu.

### 4.3.10 Current Work Space

Displays the name of the current workspace.

- Click to set the value of the WSCURRENT system variable in the command line.
- Right-click and choose the current workspace in the context menu.
- Choose **Customize** in the context menu to open the **Customize - Workspaces** dialog box.

### 4.3.11 Snap

Indicates the status of the SNAPMODE system variable.

- Click the field to toggle SNAPMODE **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the **Snap and Grid** settings.

### 4.3.12 Grid

Indicates the status of the GRIDMODE system variable.

- Click to toggle the display of grid in the current viewport.
- Right-click and choose **Settings** in the context menu to edit the **Snap and Grid** settings.

### 4.3.13 Ortho

Indicates the status of the ORTHOMODE system variable. Turning on **Ortho** automatically disables **Polar**.

- Click to toggle ORTHOMODE **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the ORTHOMODE system variable.

### 4.3.14 Polar

Toggles the **Polar Tracking** option of the AUTOSNAP system variable.

- Click to toggle Polar Tracking **On**/**Off**. Turning on **Polar** automatically disables **Ortho**.
- Right-click and choose **Settings** in the context menu to edit the **Polar Tracking** settings.

### 4.3.15 Esnap

Indicates the status of the **Turn of all snaps** option of the OSMODE system variable.

- Click to toggle Entity Snaps **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the **Entity Snaps** settings.

### 4.3.16 Strack

Indicates the status of the **Entity Snap Tracking** option of the AUTOSNAP system variable.

- Click to toggle Entity Snap Tracking **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the **Snap Tracking** settings.

### 4.3.17 Lineweight (LWT)

Indicates the status of the LWDISPLAY system variable.

- Click to toggle the display of line weights **On**/**Off**.
- Right-click and choose **Settings** to edit the **Lineweights** settings.

### 4.3.18 Paper/Model

Allows you to choose between model space and paper space.

- When working in **Model Space**, the field reads:

  **Tile** (= Model space with tiled viewports).

  Click to switch to the most recently opened paper space layout.

  Right-click to choose a paper space layout.
- When working in a layout, the field reads:
- **M:Layout**: Model space with floating viewports

  or

  **P:Layout**: Paper space

  Click to toggle between paper space and model space with floating viewports.
- Either in Model Space or Paper Space, right-click and choose:
  - **Paper Space**: Switch to paper space.
  - **Model Space**: Switch to model space with floating viewports.
  - **Model**: Switch to model space with tiled viewports.
  - **Layout**: Switch to the selected paper space layout.

### 4.3.19 Annotation Scale

Displays the value of the CANNOSCALE (= current annotation scale) system variable. Right-click to display a context menu:

- Select an annotation scale in the list.
- **Custom**: launches the SCALELISTEDIT command.
- **Hide Xref scales**: lists annotation scales of the current drawing only.

### 4.3.20 Autoscale

Indicates the status of the ANNOAUTOSCALE system variable. When **On**, annotative entities are synchronized with the current annotation scale. Click the field to toggle ANNOAUTOSCALE **On**/**Off**.

### 4.3.21 Tablet

Initializes the use of a drawing tablet.

### 4.3.22 Viewport Lock (VPLOCK)

Displays the status of **Display Locked** property of the current viewport or the selected viewport(s). Click the VPLOCK field to toggle the property **On**/**Off**.

The VPLOCK field only appears when one or more paper space viewports are selected or when a viewport is active (Model space with floating viewports).

### 4.3.23 Dynamic UCS (DUCS)

Indicates the status of the UCSDETECT system variable.

- Click to toggle the UCSDETECT system variable **On**/**Off**.
- Right-click to select the supported entity types in the context menu:
  - **3D solid faces**: 3D solid faces and regions.
  - **Other entities**: 2D linear entities

### 4.3.24 Dynamic Input (DYN)

Indicates the status of the DYNMODE system variable.

- Click to toggle the display of Dynamic Dimensions **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the **Dynamic Dimensions** settings.

### 4.3.25 Quad

Indicates the status of the QUADDISPLAY system variable.

- Click to toggle the display of the Quad **On**/**Off**.
- Right-click and choose an option in the context menu.
  - **Show Quad on Hover**.
  - **Show Quad on Select**.
  - **Show Quad on Right Click**.
  - **Suppress Quad on Hover When Entities are Selected**.
  - **On**/**Off**
  - **Customize Quad Items…** : Displays the **Customize / Quad** dialog box.
  - **Settings…** : Displays the **Settings** dialog box.

### 4.3.26 Rollover tips (RT)

Indicates the value of the ROLLOVERTIPS system variable.

- Click to toggle the display of rollover tips.
- Right-click and choose **Customize…** in the context menu to open the **Customize / Properties** dialog box. Here you can define which properties are displayed in the **Rollover Tips** for each entity type.

### 4.3.27 Hotkey Assistant (HKA)

Indicates the value of the HOTKEYASSISTANT system variable. When switched **On**, an assistant widget at the bottom of the screen reveals options available by pressing the Ctrl key.

- Click to toggle the display of the HKA widget **On**/**Off**.
- Right-click to display a context menu:
  - **On** or **Off** to set the display of the HKA widget.
  - **Configure**: Displays the **Hotkey Assistant Configuration** dialog. Select the commands for

which you want the HKA widget to display when HOTKEYASSISTANT = ON.

### 4.3.28 Lock User Interface (LOCKUI)

Allows you to lock the location and size of toolbars and dockable panels.

- Click to toggle the lock **On**/**Off**.
- Right-click to display a context menu.
  - Select the UI items to lock: toolbars, panels or both.
  - Choose **Settings...** to set the value of the LOCKUI system variable in the **Settings** dialog box.

### 4.3.29 GIS Coordinate System

Displays the name of the current GIS coordinate system.

### 4.3.30 Tray Icon Items

(not implemented) Display icons instead of text for the **Status** bar fields.

## 4.4 Document tabs

## 4.5 Toolbars

The availability of a command depends on the license level (**Shape** - **Lite** - **Pro** - **Mechanical** - **BIM** - **Ultimate**) as indicated in the command articles in the Command Reference.

### 4.5.1 To set the icon size

You can choose between small (16 x 16), large (32 x 32) or extra large (64 x 64) icons.

The icon size is saved in the TOOLBUTTONSIZE system variable in the current Workspace.

- Move the cursor over a toolbar, then right-click.
- Choose **Toolbar size**, then select either **Small Icons**, **Large Icons** or **Extra Large Icons**.

### 4.5.2 Toolbar Flyouts

On some tool buttons, a small black arrow at the bottom right corner of the icon indicates that a flyout is available. A flyout holds a group of related tools. Press and hold the left mouse button to expand the flyout and choose one of the flyout tools. The tool that was last chosen remains visible in the collapsed toolbar.

Each of the flyouts can be opened as a separate toolbar.

To tear off a flyout toolbar:

1 Press and hold the left mouse button to drag the visible icon of a flyout toolbar into the drawing area.

2 Release the left mouse button to place the toolbar.

### 4.5.3 Opening a toolbar

1 In **Windows**: Hover over a toolbar or a blank portion of the toolbar area, then right-click.

In **Linux**/**Mac**: Hover over a blank portion of the menu bar, status bar or toolbar area, then right-click.

A context menu appears.

2 In the context menu, choose **Toolbars**, then choose the Main CUI file (e.g. BRICSCAD) or one of the

Partial CUI files (if any).

A list of available toolbars in the selected CUI file is displayed.

Toolbars that are already open, are ticked.

If the list does not fit on the screen, go to **More** at the bottom of the context menu.

Some related toolbars are grouped in a flyout menu, e.g. 3D.

3 Click a toolbar on the context menu.

4 (option) To dock a toolbar, drag the toolbar to the top, bottom, left or right edge of the BricsCAD application window.

***Прим.:***

- If you select a toolbar that was already open, it will be closed.
- To prevent a toolbar from docking, press and hold the Ctrl key while dragging the toolbar.
- Drag a toolbar by the dotted grip at its left edge (when docked horizontally) or top edge (when docked vertically) to undock.
- To dock a toolbar vertically in **Linux**, the **Position** property of the toolbar must be set to **Left** or **Right**.
- The LOCKUI system variable allows you to lock the location and size of toolbars and dockable panels.
- Click the **LOCKUI** field in the **Status** bar to toggle the locking of the user interface.

## 4.6 The Quad cursor menu

The Quad cursor menu, aka the Quad is a context sensitive graphic menu which allows you to easily edit, select, show or hide and create entities. The content of the Quad is defined in the workspace and is fully customizable.

### 4.6.1 Activating the Quad

Activation of the Quad is controlled by theQUADDISPLAYsystem variable.

To toggle the Quad On/Off, do one of the following:

- Click theQUADfield in the**Status** bar.
- Press the**F12**function key.

### 4.6.2 Setting the Quad display options

Right-click the **Quad** field in the **Status** bar, then choose an option in the context menu.

- **Show Quad on Hover**.
- **Show Quad on Select**.
- **Show Quad on Right Click**: displays the 'No Selection' Quad which allows you to create entities.
- **Suppress Quad on Hover When Entities are Selected**.

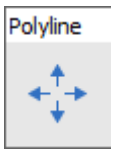
### 4.6.3 Editing the Quad settings

Right-click the**QUAD**field in the**Status** bar, then choose**Settings**in the context menu.

The**Settings**dialog box opens, showing the**Quad settings** screen.

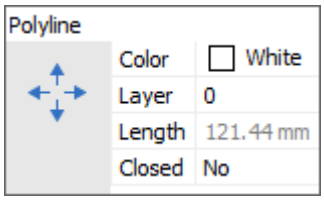
### 4.6.4 Using the Quad

When the cursor hovers over an entity, the entity is highlighted and the Quad displays the most recently used command with this entity type.




If multiple entities are under the cursor, repeatedly press the Tab key to highlight all entities one by one.

IfRollover Tips(RT) = On, the properties of the entity under the cursor are displayed next to the default command icon in the Quad.

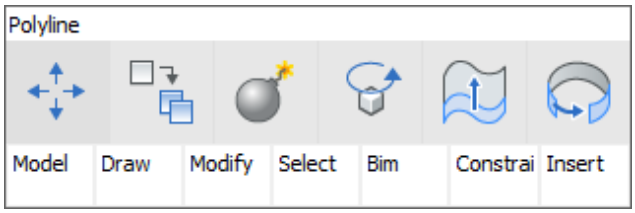



**Launching the default command**

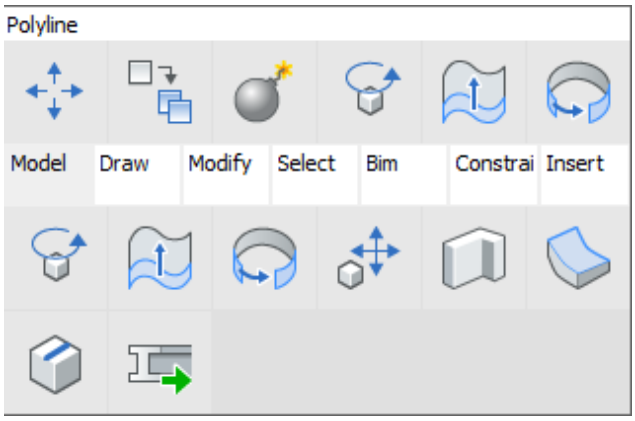
- If QUADCOMMANDLAUNCH = 1, right-click without entering the Quad to launch the command.
- If QUADCOMMANDLAUNCH = 0, click the command button.

**Launching a different command**

Move the cursor to the default command to expand the Quad: the 6 most recently used commands that apply to the selected entity are displayed. The number of icons that appear is set by the QUADWIDTH system variable (default = 6).




Other commands that apply to the selected entity sit in a number of tabs. Move the cursor over a tab to expand its content.




Click an icon to launch the command.

***Прим.:*** Click the title bar of the Quad to go back and forth between the Quad commands and the Rollover Tips.

### 4.6.5 Selecting multiple entities

1 Move the cursor over the first entity, then click when the entity is highlighted.

1 The entity is selected and remains highlighted.

2 Move the cursor to another entity, then click when the entity highlights.

3 The entity is added to the selection set and remains highlighted.

4 (option) Repeat step 2 to select more entities.

5 (option) Move the cursor to a selected entity, then press and hold the Shift key and click.

6 The entity is removed from the selection set.

***Прим.:***

- When the Quad is active, the value of theSELECTIONPREVIEW system variableis neglected unless a command prompts for entity selection.
- When working with solids, theSELECTIONMODESsystem variable controls which subentities are detected: edges, faces and/or boundaries.
- Holding down the Ctrl key inverts the current selection mode.
- Enable detection of 3D solid edges
- Enable detection of 3D solid faces
- Enable boundary detection
- Repeatedly press the Tab key to highlight all (sub)entities under the cursor one by one.

### 4.6.6 Using the Quad to create entities

If flag 4 (= Display the Quad on right-click) of theQUADDISPLAYsystem variable is set, entity creation commands are available in the Quad (No Selection Quad). Right-click the**Quad**field in the**Status** barand check the option in the context menu.

When the cursor is on an entity, hold down the right mouse button longer to display the**No Selection**Quad.

The available entity creation commands in the Quad are:

- The most recently used entity creation commands.
- The number of available commands depends on theQUADWIDTH system variable.
- A number of command group tabs, depending on the currentworkspace.

## 4.7 Rollover tips

The Rollover Tips feature displays a selection of entity properties in a popup box near the cursor when hovering over an entity or when an entity is selected. When multiple entities are selected, only their shared properties are displayed.

Which properties are available can be defined per entity type in the **Customize / Properties** dialog box.

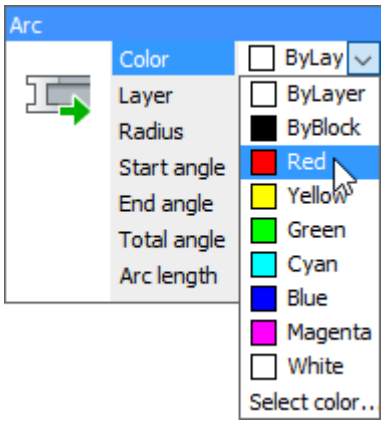
The availability of Rollover Tips is controlled through the ROLLOVERTIPS system variable.

Click the **RT** field in the **Status** bar to toggle Rollover Tips On/Off.

Rollover tips are displayed when:

- The Quad = On.
- The Quad = Off, and SELECTIONPREVIEW system variable = 1 or 3.

Select a property to edit it, much like you would do in the **Properties** panel.




When the Quad is On, you can go back and forth between the Quad commands and the Rollover Tips by clicking the title bar of the Quad/Rollover Tips feature.

## 4.8 Prompt menus

When executing a command, BricsCAD prompts you in the Command line or the Status bar. Optionally, command options are also available in a prompt menu.

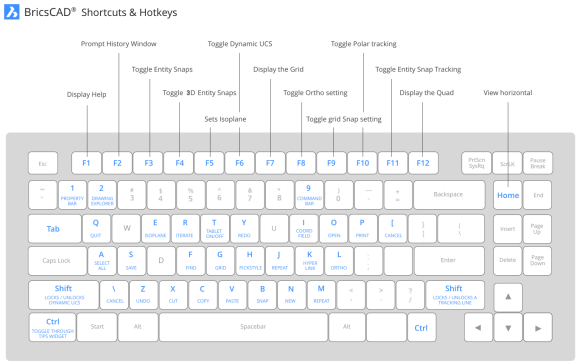
### 4.8.1 Controlling the display of prompt menus

Whether prompt menus are displayed and where they are located is controlled through thePROMPTMENU system variable.

The PROMPTMENUFLAGS system variable controls the content and behavior of prompt menus.

***Прим.:*** These system variables are saved in the registry and therefore apply to all drawings.

## 4.9 Keyboard shortcuts




Keyboard shortcuts can be a great performance enhancer, especially when toggling settings.

The following keyboard shortcuts are available:

| Keyboard | Description |
|---|---|
| F1 | Opens the BricsCAD Help.<br>Executes the HELP command. |
| F2 | Toggles the display of the **Prompt History** window.<br>See the GRAPHSCR command. |
| F3 | Toggles the entity snaps ON/OFF.<br>See the OSMODE setting. |
| F4 | Toggles the 3D entity snaps ON/OFF.<br>See the 3DOSMODE setting. |
| F5 | Sets the isometric plane; toggles the value of the SNAPISOPAIR setting.<br>Applies only when the SNAPSTYL setting = 1. |
| F6 | Toggles the Dynamic UCS feature. |
| F7 | Toggles the display of the grid (see the GRIDMODE setting). |
| F8 | Toggles the ORTHOMODE setting ON/OFF.<br>Disables bitcode 8 of the AUTOSNAP setting, if set. |
| F9 | Toggles Snap ON/OFF (see the SNAP command). |

| Keyboard | Description |
|---|---|
| F10 | Toggles the Polar Tracking setting. Sets the ORTHOMODE setting OFF, if ON.<br>Toggles bitcode 8 of the AUTOSNAP setting ON/OFF. |
| F11 | Toggles the Entity Snap Tracking setting.<br>Toggles bitcode 16 of the AUTOSNAP setting ON/OFF. |
| F12 | Toggles the display of the Quad cursor menu.<br>See the QUADDISPLAY setting. |
| Alt + F8 | Launches the VBARUN command. |
| Alt + F11 | Launches the VBAIDE command. |
| Ctrl + 1 | ***Прим.:*** Do not use the numeric keypad. On AZERTY-keyboards: do not use the Shift key when pressing the number key.<br>Toggles the Properties panel. |
| Ctrl + 2 | ***Прим.:*** Do not use the numeric keypad. On AZERTY-keyboards: do not use the Shift key when pressing the number key.<br>Opens the Drawing Explorer dialog box. |
| Ctrl + 9 | ***Прим.:*** Do not use the numeric keypad. On AZERTY-keyboards: do not use the Shift key when pressing the number key.<br>Toggles the display of the Command line.<br>Executes the COMMANDLINE command or the COMMANDLINEHIDE command. |
| Ctrl + 0 | ***Прим.:*** Do not use the numeric keypad. On AZERTY-keyboards: do not use the Shift key when pressing the number key.<br>Toggles the display of user interface entities as specified by the CLEANSCREENOPTIONS setting.<br>Executes the CLEANSCREENON command or the CLEANSCREENOFF command. |
| Ctrl + A | Selects all entities. |
| Ctrl + B | Toggles Snap ON/OFF (see the SNAP command). |
| Ctrl + C | Copies the selection to the clipboard (COPYCLIP command). |

| Keyboard | Description |
|---|---|
| Ctrl + Shift + C | Copies the selection along with a base point (COPYBASE command). |
| Ctrl + E | Sets the isometric plane; iterates the value of the SNAPISOPAIR setting. Applies only when the SNAPSTYL setting = 1. |
| Ctrl + F | Launches the FIND command. |
| Ctrl + G | Toggles the display of the grid (GRIDMODE setting). |
| Ctrl + H | Toggles the value of the PICKSTYLE setting, which controls the selection of groups and associative hatches. |
| Ctrl + I | Sets the readout of the coordinate field in the Status bar (COORDS setting). |
| Ctrl + J | Repeats the previously issued command. |
| Ctrl + K | Attach a hyperlink to an object or modify an existing hyperlink (HYPERLINK command). |
| Ctrl + L | Toggles the ORTHO setting. |
| Ctrl + M | Repeats the previously issued command. |
| Ctrl + N | Creates a new drawing (NEW command). |
| Ctrl + O | Opens an existing drawing (OPEN command). |
| Ctrl + P | Prints the current drawing (PRINT command). |
| Ctrl + Shift + P | Toggles the display of the Properties panel (PROPERTIES command and PROPERTIESCLOSE command). |
| Ctrl + Q | Quits the application; prompts to save changes (QUIT command). |
| Ctrl + R | Iterates through viewports. |
| Ctrl + S | Saves the current drawing (QSAVE command). |
| Ctrl + T | Turns the tablet on/off (TABLET command). |
| Ctrl + V | Pastes the contents of the Clipboard. (PASTECLIP command). |

| Keyboard | Description |
|---|---|
| Ctrl + Alt + V | Pastes the contents of the Clipboard in a specified format (PASTESPEC command). |
| Ctrl + Shift + V | Pastes the contents of the Clipboard as a block (PASTEBLOCK command). |
| Ctrl + X | Cuts the selection and puts it on the clipboard (CUTCLIP command). |
| Ctrl + Y | Redoes the last action undone (REDO command). |
| Ctrl + Z | Undoes the last action (UNDO command). |
| Ctrl + [ | Cancels the running command. |
| Ctrl + \ | Cancels the running command. |
| Ctrl + Home | Creates or opens the Start tab (GOTOSTART command). |
| Shift + F2 | Toggles the display of the Command line (COMMANDLINE command and COMMANDLINEHIDE command). |
| Shift + F3 | Turns the Status bar on/off (SCROLLBAR command). |
| Shift + F4 | Turns the Scroll Bars on/off (SCROLLBAR command). |
| Shift + F8 | Launches the Visual Basic Project Manager (VBAMAN command). |
| Shift + F11 | Launches the VBA COM Add-In Manager (ADDINMAN command). |
| PgUp | Moves the view up. |
| PgDn | Moves the view down. |
| Shift + Left Arrow | Moves the view to the left. |
| Shift + Right Arrow | Moves the view to the right. |

| Keyboard | Description |
|---|---|
| Shift + Up Arrow | Moves the view up. |
| Shift + Down Arrow | Moves the view down. |

## 4.10 Digitizing tablet

A digitizing tablet, also called a graphics tablet, a graphics pad or drawing tablet, is a computer input device that allows you to hand-draw images and graphics, similar to the way one draws images with a pencil and paper. In a CAD program a digitizing tablet can be used to trace paper drawings into a drawing file or to launch drawing commands from a digitizing tablet overlay.

The TABLET command configures and calibrates a digitizing tablet and toggles the tablet mode.

This command requires that Wintab32.dll from Wacom Technology, Corp. be installed on the computer. Download Wintab32.dll from the internet, then place the file in your system directory: C:\Windows \System32.

### 4.10.1 Using a digitizing tablet in BricsCAD

In order to prepare BricsCAD for the use of a digitizing tablet, do the following:

- Install the appropriate driver for your tablet.
- Make sure Wintab32.dll is present in C:\Windows\System32.
- Download the tablet overlay and CUI files.
- Print the BricsCAD tablet overlay.
- Load the overlay CUI file.
- Start BricsCAD, then initialize the tablet.
- Configure the tablet: part of the tablet surface is used as the menu area, another part of it is used as the screen pointing area.
- Calibrate the tablet: a calibrated tablet can be used to trace the geometry of a paper drawing or image into a Bricscad drawing.

### 4.10.2 Downloading the tablet overlay and the CUI files

Click here to download the file Tablet.zip.

https://www.bricsys.com/bricscad/tools/Tablet.zip

The ZIP file contains the following files:

- **overlay.png**: an image of the tablet overlay
- **overlay(A3).pdf**: to print the tablet overlay on a A3 paper size.
- **overlay(cm).dwg**: a DWG in which the **overlay.png** is attached as an image.

The drawing contains two layouts to print the overlay using decimal units.

- **overlay(inch).dwg**: a DWG in which the **overlay.png** is attached as an image.

  The drawing contains two layouts to print the overlay using imperial units.
- **tablet.cui**: a partial cui file to be used with a tablet.
- **tablet(acadLike).cui**: an AutoCAD-like partial cui file to be used with a tablet.

### 4.10.3 Loading the overlay CUI file

1 Extract the overlay CUI files to the **Support** folder of the **roamable root** folder (see BricsCAD User Files).

2 Load **Tablet.cui** as a partial CUI. The next time you start BricsCAD, **Tablet.cui** will be loaded automatically.

***Прим.:***

- Please make sure the **overlay.png** image file is extracted to the same folder as the DWG's.
- It may be necessary to repeat the above procedure after upgrading BricsCAD.

### 4.10.4 Switching the tablet ON / OFF

Do one of the following:

- Click the **TABLET** field in the Status bar.
- Press the **F4** function key.
- Launch the TABLET command.

### 4.10.5 Configuring the tablet

1 Launch the TABLET command.

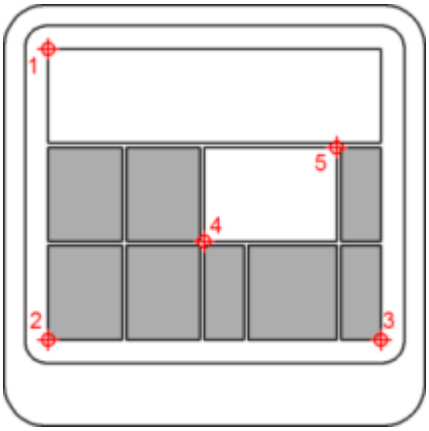
2 Choose the **Configure** option.

3 **You are prompted**: Do you want to align the tablet overlay? Yes/No/<No>.

Type **Y** in the Command line and press Enter or choose **Yes** in the prompt menu.

**You are prompted**: Digitize upper left corner of the overlay.

4 Click on the mark in the upper left corner (1) of the digitizing tablet overlay.



**You are prompted**: Digitize the lower left corner of the overlay.

5 Click on the mark in the lower left corner (2) of the digitizing tablet overlay.

6 **You are prompted**: Digitize the lower right corner of the overlay.

7 Click on the mark in the lower right corner (3) of the digitizing tablet overlay.

8 **You are prompted**: Digitize the lower left corner of the screen pointing area.

9 Click on the mark at position (4) on the of the digitizing tablet overlay.

10 **You are prompted**: Digitize the upper right corner of the screen pointing area.

11 Click on the mark at position (5) on the of the digitizing tablet overlay.

12 **You are prompted**: Tablet configured.

### 4.10.6 Calibrating the tablet

To calibrate the digitizing tablet you must specify at least two points. The more additional points you specify, the more accurate the digitizing process. Additional points are particularly recommended when digitizing a non-orthogonal image, such as an aerial photograph.

1 Launch the TABLET command.

2 Choose the **Calibrate** option.

3 **You are prompted**: Digitize point #1:

Click a point on the digitizing tablet.

**You are prompted**: Enter coordinates for point #1:

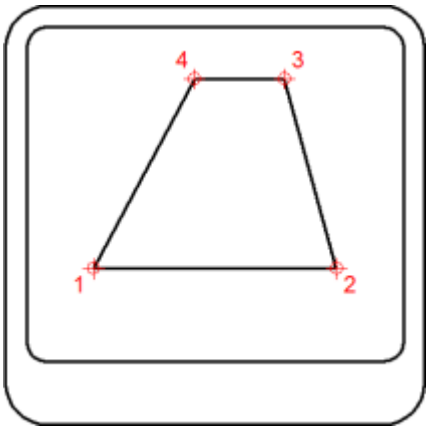
4 Do one of the following:
- Click a point in the BricsCAD drawing window.
- Enter the coordinates (X,Y) in the Command line.

5 Repeat step 3 to specify the following calibration point.

6 **You are prompted**: Digitize point #3 (or ENTER to end):

7 Do one of the following:
- Repeat step 4 to specify an additional calibration point. (You can specify up to 10 calibration points.)
- Press Enter to conclude the calibration process.

8 BricsCAD responds:
- When 2 calibration points are specified: Tablet calibrated.
- When 3 or more than 4 calibration points are specified: Select transformation type Orthogonal/Affine:
- When 4 calibration points are specified: Select transformation type Orthogonal/Affine/Projective:




9 Select the transformation type.

10 Transformation refers to the calculation of the points on the screen that correspond to points you digitize on the tablet.



- **Orthogonal** (left): To be used with accurate paper drawings. Orthogonal transformation maintains all angles and preserves relative distances. If only two calibration points are specified, orthogonal transformation is generated automatically.
- **Affine** (middle): Maintains parallel lines, but not necessarily the angles between intersecting lines.
- **Projective** (right): Does not maintain parallel lines, nor angles.

## 4.11 Panels

Panels are interactive UI elements that can remain open while you use other commands. They can be docked or floating and they can be stacked. And you can control the amount of data displayed on the panel. The location and size of the panel depends on the current workspace or its previous appearance.

You can dock panels at the top, bottom, left or right within the BricsCAD application window. They can be docked alongside other panels or toolbars but are always constrained by the size of the application window. The drawing area, within the BricsCAD application window, automatically reduces to accommodate the docked panels. You can move a docked panel using the grip bar, resize it with the stretch icon, and close it with the close icon.

Floating panels can be placed anywhere on any available monitor. They can be moved and resized beyond the boundaries of the BricsCAD application window. If a floating panel overlaps the drawing area, it floats above it, allowing the drawing area to maintain its size. You can move a floating panel using the title bar, resize it with the stretch icons, and close it with the close icon. Regardless of where a floating panel is

located, the stretch icon appears when you pass the cursor over any edge of the panel. The title bar and close icons are always on the top edge of a floating panel.

Multiple panels can be combined into a panel stack to conserve screen space. Each panel stack can be docked or floating, offering the same functionality as individually docked or floating panels. You can display panel stacks with tabs or icons.

By default, panels are docked at the left or right hand side of the screen as defined in the **Panels** node of a workspace in the**Workspaces** tab of the Customizedialog box:

- **Panels | Stack (LDOCK)** selects the panels stacked a the left hand side of the screen.
- **Panels | Stack (RDOCK)** selects the panels stacked a the right hand side of the screen.

The stack type is defined in the**Workspaces** tab of the Customizedialog box through the **Stack Type** property of a workspace.

The possible stack types are:

- Fixed resizable panel set
- Collapsible panel set
- Flyout panel set

### 4.11.1 Opening or closing a panel

1 Move the cursor over a toolbar or ribbon panel, then right click.

A context menu displays.

2 Place the cursor over **Panels**.

A list of available panels displays.

Panels that are currently open are checked.

3 Click a panel in the menu to toggle its display.

- Some panels can be opened using a command as indicated in the table below.
- The -TOOLPANELcommand allows to show, hide or toggle the display of panels at the Command line.

### 4.11.2 Available panels

| Icon | Title | Command |
|---|---|---|
|  | Animation editor panel 171 | |
|  | Attachments panel 173 | |
|  | BIM BCF panel 175 | |
|  | BIM compositions panel 176 | |

| Icon | Title | Command |
|---|---|---|
|  | BIM profiles panel 178 | |
|  | BIM project browser panel 180 | |
|  | BOM manager panel 183 | |
|  | Bricsys 24/7 panel 190 | |
| | Command line panel 192 | |
|  | Content browser panel 193 | |
|  | Details panel 195 | |
|  | Drawing customizations panel 197 | |
|  | Layers panel 204 | |
|  | Library panel 210 | |
|  | Mechanical browser panel 213 | |
|  | Parameters and constraints panel 217 | |
|  | Properties panel 219 | |
|  | Render materials panel 226 | |
|  | Report panel 228 | |
| | Ribbon | |
|  | Sheet sets panel 230 | |
|  | Structure panel 233 | |

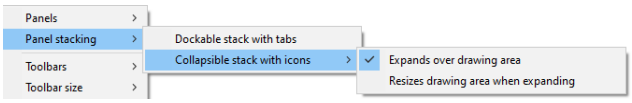
| Icon | Title | Command |
|---|---|---|
| | Tips panel 235 | |
| | Tool palettes panel 236 | Tool palettes 863 |

### 4.11.3 Setting the stack type

1 Move the cursor over a toolbar or panel, then right click.

A context menu displays.

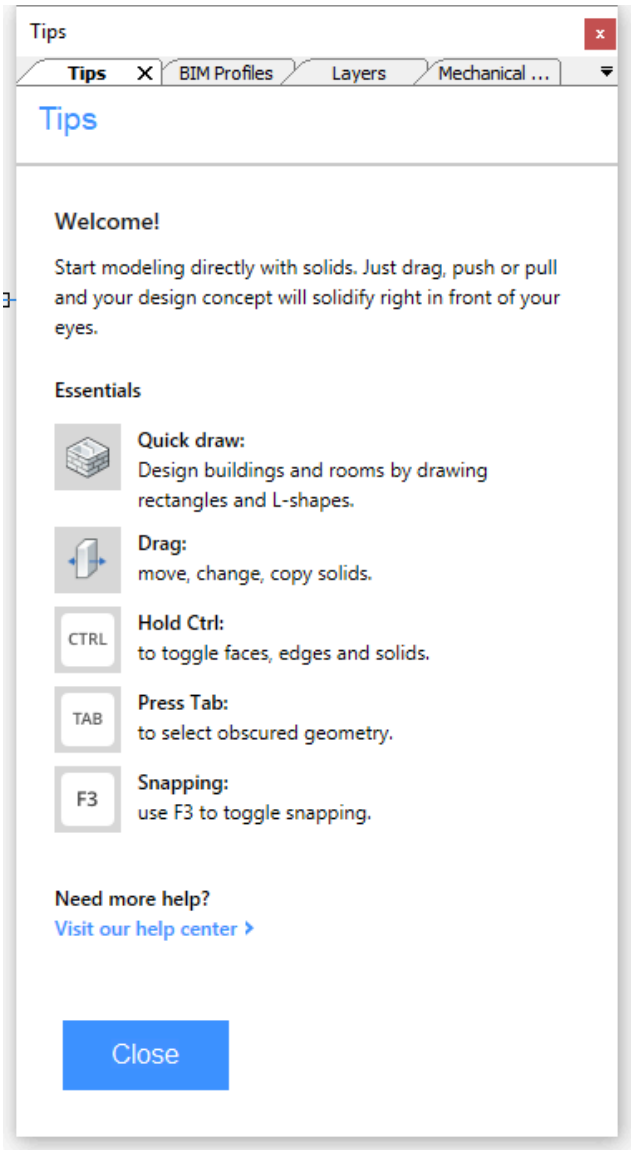
2 Choose **Panel stacking**.




3 Select the desired stack type.

**Dockable stack with tabs**

Panels are docked in a tabbed panel.

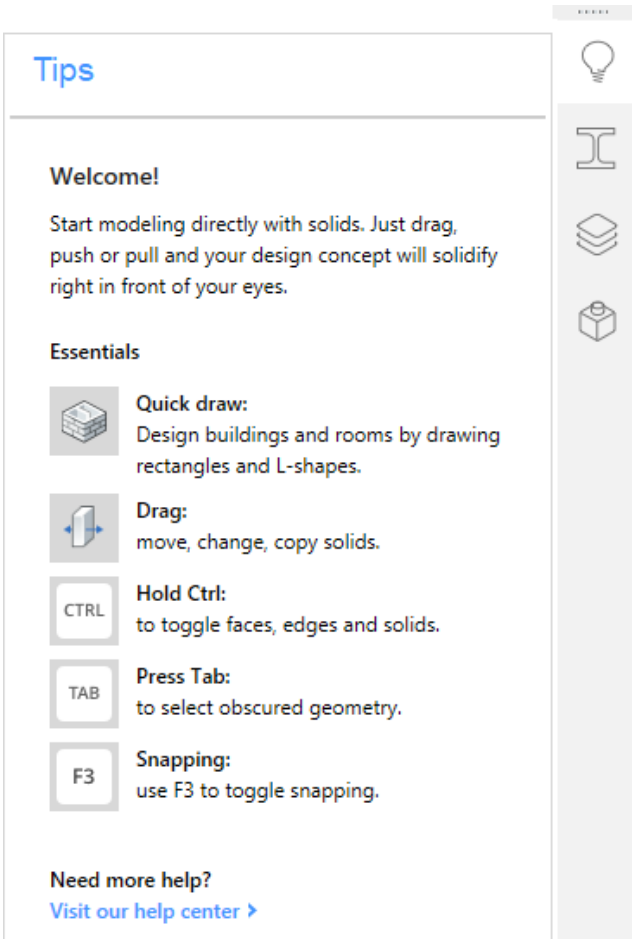



- Click a tab to display the panel.
- Press and hold the left mouse button to drag a tab to re-arrange the tab order.
- Double click a tab to make the panel floating.
- Double click the title bar of floating panel to dock it at its previous position.

**Collapsible stack with flyouts**

Panels are docked in an icon strip at the left and/or right hand side of the screen.

- **Expands over drawing area**: flyout panels overlap the drawing area, thus hiding part of the drawing.
- **Resizes drawing area**: flyout panels narrow the drawing area.

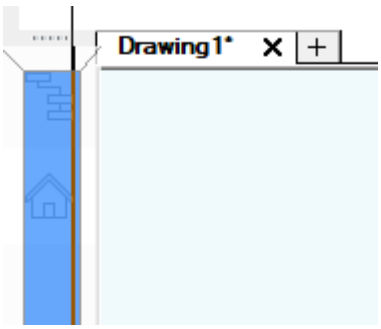



- Click an icon to expand a panel.
- To collapse an expanded panel click its icon again.
- Press and hold the left mouse button to drag an icon to re-arrange the icon order.
- Double click the icon to make the panel floating.
- Double click the title bar of a floating panel to dock it at its previous position.

### 4.11.4 Moving a panel to the other icon tab strip

1 Drag the icon to the other icon strip.

The docking location selector displays. The blue zone indicates the selected docking position.

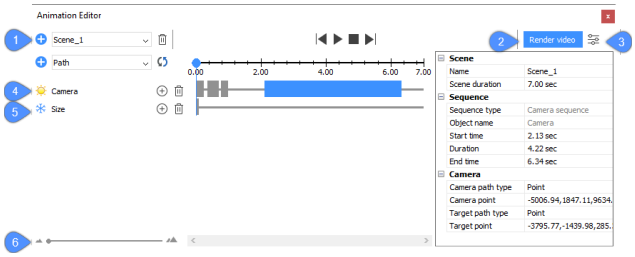


2 Move the cursor over the center docking location then release the left mouse button.

***Прим.:*** TheLOCKUIsystem variable controls whether the current position of the panels is locked or not. Press and hold the Ctrl-key to override. Click the**LOCKUI**field in theStatus Barto toggle the locking of the user interface.

### 4.11.5 Animation editor panel

The Animation Editor Panel allows you to create schedules for multi-parametric animations with complex camera movements and render them as 2D videos.




1 Add scene

2 Render video

3 Configurator

4 Disable timeline

5 Enable timeline

6 Slider

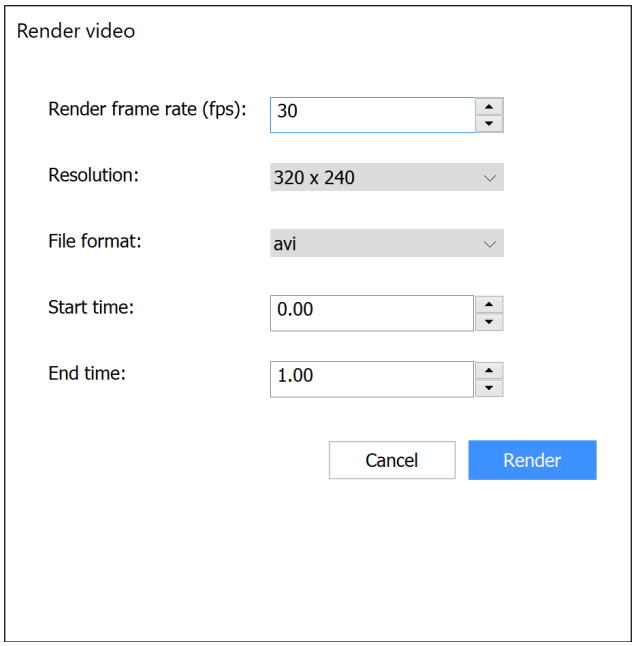
#### 4.11.5.1 Add scene

Scene name.

Opens the Add Scene dialog box name to insert the scene name and its duration (s).

#### 4.11.5.2 Render video

Opens the Render video dialog box to set the render options:




#### 4.11.5.3 Render frame rate (fps)

Sets the framerate of the video.

#### 4.11.5.4 Resolution

Sets the resolution of the video by selecting it from the drop-down list of supported resolutions.

#### 4.11.5.5 File format

Sets the format of the output video. You can select a file format from the drop-down list:

- avi (AVI Animation)
- mpg (MPEG Animation)
- wmv (WMV Animation)

#### 4.11.5.6 Start time

Sets the time position of the animation to start the render.

#### 4.11.5.7 End time

Sets the time position of the animation to end the render.

After pressing the Render button, the Save Video as dialog box opens to allow you to save the rendered video.

#### 4.11.5.8 Configurator

Toggles the configurator panel.

#### 4.11.5.9 Disable timeline

Disables timeline effects.

### 4.11.5.10 Enable timeline

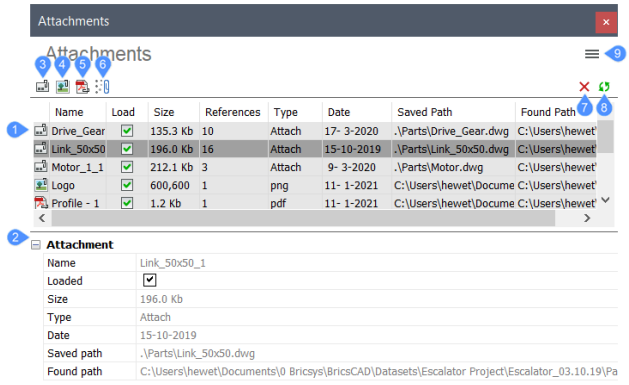
Enable timeline effects.

### 4.11.5.11 Slider

Move the slider to change the scale of the timeline.

## 4.11.6 Attachments panel

The Attachments panel offers a central location for viewing and managing attached drawings, images, PDF files, and point clouds.




1 Attachment list

2 Properties

3 Attach DWG

4 Attach Image

5 Attach PDF

6 Attach Point Cloud

7 Detach

8 Refresh

9 Menu

### 4.11.6.1 Attachment list

Displays a list of attached filesand their properties. Column headers describe theproperties of each attachment. You may sort theattachment listin order or reverse orderforanyof the properties byclickingonce or twice onthecolumn header.A right-click menuon the column headers enables you to customize them. A right-click menu on the attachments offers additional tools.

### 4.11.6.2 Name

Specifies the attachment name. By default, the attachment name is the same as the file name. To rename an attachment, double-click the name and enter the new name. Renaming an attachment does not change the file name.

#### 4.11.6.3 Load

Specifies the load status of the attachments. Attachments with the Load property enabled are loaded and displayed in the drawing editor. Attachments with the Load property disabled maintain their attachment information, such as scale and location, but are not loaded or displayed in the drawing editor.

#### 4.11.6.4 Size

Specifies the file size of the attachments.

#### 4.11.6.5 References

Specifies how many times the attachments are referenced in the current drawing.

#### 4.11.6.6 Type

Specifies the attachment type. Click the attachment type to toggle between Attach and Overlay.

#### 4.11.6.7 Date

Specifies the date the attached files were last saved.

#### 4.11.6.8 Saved Path

Specifies the path, including location and file name, of the attachments. To change the saved path, click it and enter the new path or select the Browse button to display the Select reference file dialog box. The Browse button is only displayed after you click on the saved path.

#### 4.11.6.9 Found Path

Specifies the path, including location and file name, of the attachments. The Found path may or may not be the same as the Saved path.

#### 4.11.6.10 Properties

Displays the properties of the selected attachment in a row format. These are the same properties displayed in the Attachments list with a column format.

#### 4.11.6.11 Attach DWG

Displays the Open file to attach dialog box like theXREFcommand.

#### 4.11.6.12 Attach Image

Displays the Select image file dialog box like theIMAGEATTACHcommand.

#### 4.11.6.13 Attach PDF

Displays the Select PDF underlay file dialog box like thePDFATTACHcommand.

#### 4.11.6.14 Attach Point Cloud

Displays the Preprocessed Point Cloud Data File dialog box like thePOINTCLOUDATTACHcommand.

#### 4.11.6.15 Detach

Detaches the selected attachment from the drawing.

#### 4.11.6.16 Refresh

Updates the display.

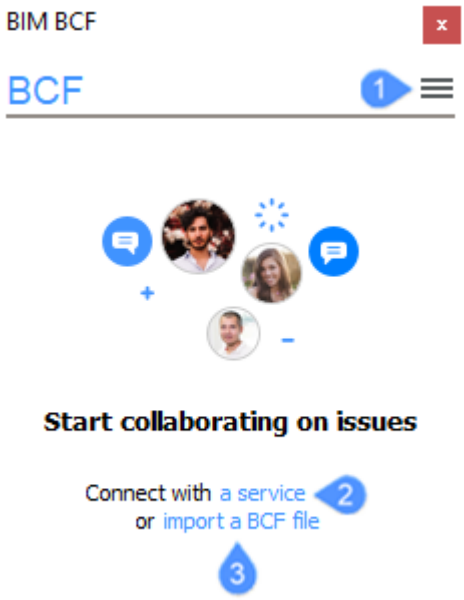
#### 4.11.6.17 Menu

**Grid View**
Displays the attachments in a detailed grid view.

**Tree View**
Displays the attachments in a structured tree view.

### 4.11.7 BIM BCF panel

The BIM BCF panel allows you to link information such as a comment, a screenshot, a list of involved objects and a camera viewport to an issue.




1 Menu

2 Connect with a service

3 Import a BCF file

#### 4.11.7.1 Menu

Opens the main menu of BIM BCF panel.

**Import BCF file**
Allows to import BIM Collaboration Format Files (*.bcf,*.bcfzip).

**Log in with custom service**
Allows you to connect to a cloud service to start collaborating on issues in the drawings.

**Log in with custom service**
Allows you to connect to a cloud service to start collaborating on issues in the drawings.

**Refresh**
Makes sure you see the latest updates in case you are working on the cloud.

**Clear sources**
Disconnects you with the cloud service, in case you are online, or closes the .bcf file, in case you are working on an imported .bcf file.

#### 4.11.7.2 Connect with a service

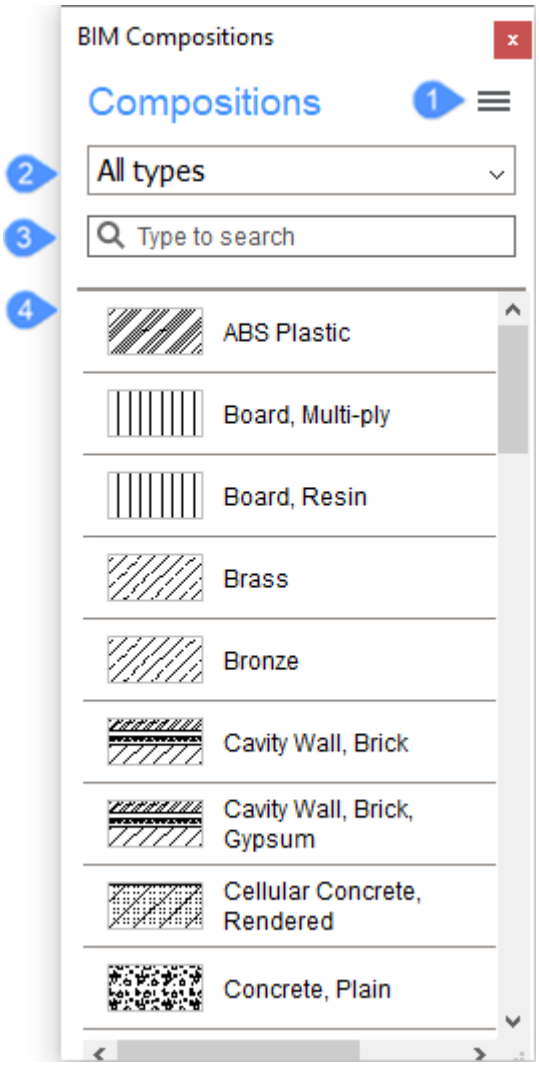
Opens the Custom log in dialog box.

#### 4.11.7.3 Import a BCF file

Displays the Select a BIM Collaboration Format file to import dialog box.

### 4.11.8 BIM compositions panel

The BIM composition panel allows you to assign compositions to any solid in a BIM model.




1 Menu

2 Compositions types list

3 Search

4 Available compositions

### 4.11.8.1 Menu

**Open material dialog...**
Opens the Physical Material dialog box that allows you to create, modify and delete physical materials.

**Open composition dialog...**
Opens the Compositions dialog box that allows you to create, modify and delete compositions.

**Open project dialog...**
Opens the BIM Project Info dialog box that allows you to control the location, properties, and content of the Project and Library databases.

**Show only compositions in project**
Displays only the compositions that are used in the project.

#### 4.11.8.2 Compositions types list

Displays the compositions of the selected category (all types, generic, roof, slab, wall) in the Library database.

#### 4.11.8.3 Search

Filters the list of compositions. Only compositions that contain the current search term are displayed.

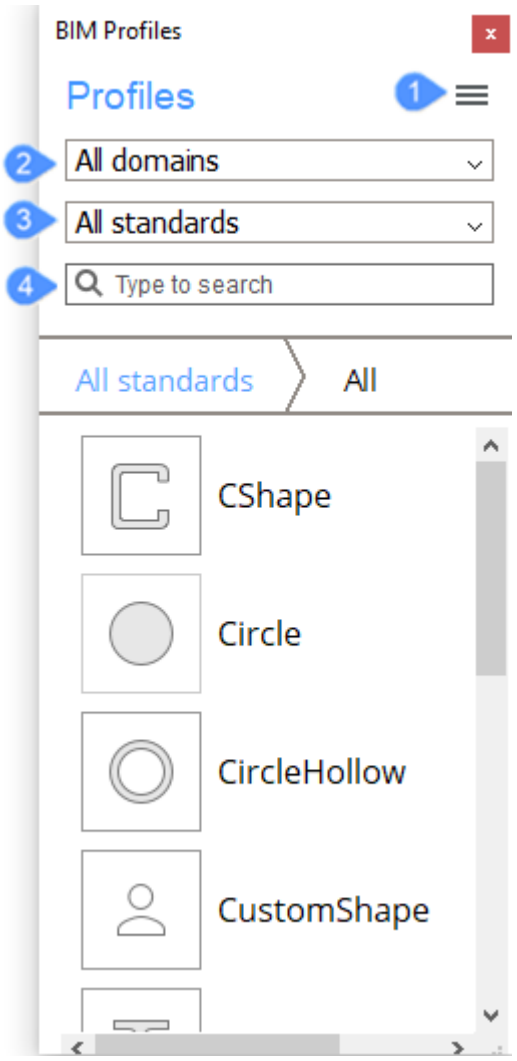
#### 4.11.8.4 Available compositions

Displays all the available compositions.

When you double-click on a composition the Compositions dialog box will open.

### 4.11.9 BIM profiles panel

The BIM Profiles panel offers a central location to access standard and custom profiles.




1 Menu

2 Domains

3 Standards

4 Search bar

### 4.11.9.1 Menu

**Open material dialog...**
Opens the Physical Material dialog box that allows you to create, modify and delete physical materials.

**Open profile dialog...**
Opens the Profiles dialog box that allows you to create, modify and delete profiles

**Open project dialog...**
Opens the BIM Project Info dialog box that allows you to control the location, properties, and content of the Project and Library databases.

**Show only profiles in project**
Displays only the profiles that are used in the project.

#### 4.11.9.2 Domains

Filter profiles based on the domain.

#### 4.11.9.3 Standards

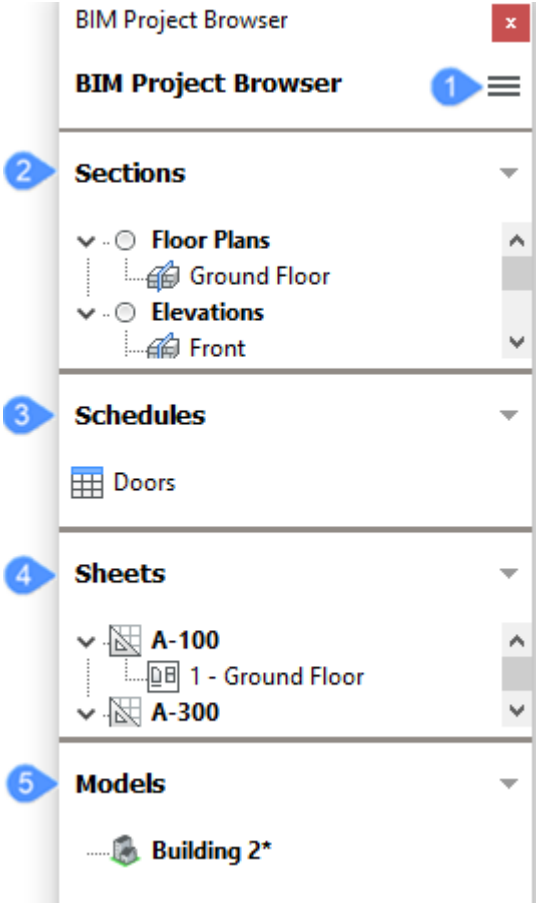
Filter profiles based on the country standard.

#### 4.11.9.4 Search bar

Search for specific profiles using the search bar.

### 4.11.10 BIM project browser panel

The BIM project browser panel offers a central location to manage your BIM projects.




1 Menu

2 Sections

3 Schedules

4 Sheets

5 Models

### 4.11.10.1 Menu

**Refresh**
Refreshes the BIM Project Browser panel to match changes in the model.

**Project Setup**
Opens the Project Setup dialog box that gives a complete overview of the project's files, in different categories.

**Sheetset Setup**
Opens the Sheetset Setup dialog box that allows to set up the complete set of sheets required to fit all bimsections in the project.

**Enable Background Update**
Calculates section results automatically, without interrupting the user workflow.

### 4.11.10.2 Sections

Displays a list of project sections organized by type: floor plans and elevations. You can expand a section type to see its sections.

A context menu displays when you right-click on a section:

- **Section Properties...**: opens the Section Properties dialog box where you can see and edit the properties of a section, such as the name and the result path.
- **Select Section Entity**: selects the section entity in the project model drawing.
- **Rename**: renames the section entity.
- **Update**: recalculates the section result.
- **Display Section Result**: displays the section result in the corresponding sheet and zooms in on its viewport. If not already open, the drawing is opened.

### 4.11.10.3 Schedules

Displays a list of project schedules.

A context menu displays when you right-click on the Schedules section.

- **Add Schedule (Wizard)...**: opens the Wizard Page dialog box that allows you to create a schedule based on a new or an existing data extraction definition.
- **Add Schedule (From dxd)...**: opens the Create Schedule dialog box that allows you to quickly add a BIM schedule, without the creation of a new .dxd file.

A context menu displays when you right-click on a schedule.

- **Schedule Properties...**: opens the Schedule Properties dialog box that allows you to modify the properties of a schedule.
- **Create/Update Table**: creates a sheet with the schedule or updates the sheet.
- **Create/Update CSV**: creates a CSV file with the schedule information and saves it in the folder where the model file is located or updates the CSV file if it already exists.
- **Display Schedule**: displays the schedule in the resulting drawing. If not already open, the drawing is opened.

- **Edit**: opens the Wizard Page dialog box that allows you to edit the schedule.
- **Remove...**: displays the Remove dialog box that allows you to choose between **Schedule Entity** and **Sheet View/Result** and removes the schedule.

#### 4.11.10.4 Sheets

Displays a list of project sheets. You can expand a sheet name to see its sheet views.

A context menu appears when you right-click on the Sheets section:

- **Sheetset Setup...**: opens the Sheetset Setup dialog box that allows you to create a layout from scratch.
- **Add Subset**: adds a subset in the Sheets tab.
- **Add Sheet...**: adds a new sheet in a subset in the Sheets tab.
- **Sheetset Options...**: opens the Sheet sets tab in the Settings dialog box that allows you to modify the sheet sets options.

A context menu displays when you right-click on a sheet name:

- **Sheet Properties...**: opens the Sheet Properties dialog box that allows you to modify the properties of a sheet.
- **Open Sheet**: opens the corresponding sheet.
- **Remove...**: opens the Remove dialog box that allows you to choose between **Result Drawing** and **Sheet** and removes the sheet.

A context menu displays when you right-click on a sheet view:

- **View Properties...**: opens the Sheet View Properties dialog box that allows you to modify the properties of a sheet view.
- **Display View**: displays the corresponding result and zooms in on its viewport.
- **Update**: updates the sheet view. It executes the BIMSECTIONUPDATE command for the related BIM section.

A context menu displays when you right-click on a subset name:

- **Subset Properties...**: opens the Subset Properties dialog box that allows you to modify the properties of a subset.

#### 4.11.10.5 Models

Displays a list of project models.

A context menu displays when you right-click on a model with the following options:

- **Model Properties...**: opens the Model Properties dialog box that allows you to see more information about the model.
- **Close Drawing**: for a currently open model,it closes the model.
- **Open Drawing**: for a currently closed model, it opens the model.
- **Remove from Model List**: removes the drawing from the project model drawings.

### 4.11.11 BOM manager panel

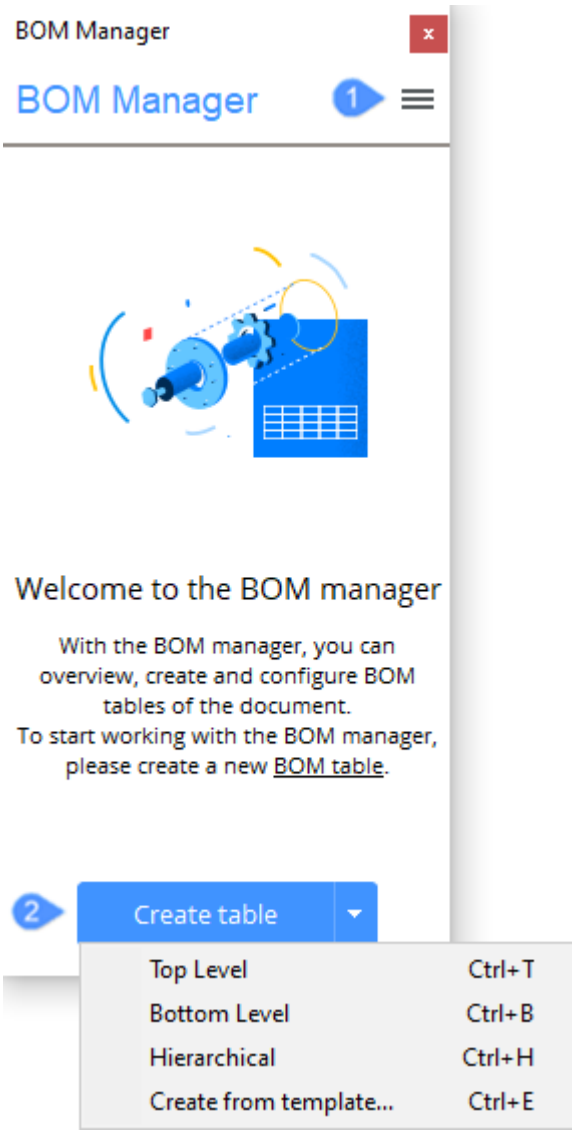
The BOM Manager panel allows you to browse, create and edit bills of materials (BOMs) in the current drawing.

The configuration of a bill of materials can be saved as a template file and reused later in other drawings.

With the Bill of Materials Manager, you can:

- Browse existing bills of materials in the drawing.
- Create a new bill of materials using either a standard type (such as top-level, bottom-level, hierarchical) or an existing BOM template.
- Edit bills of materials:
- Add or remove columns (includes formula columns).
- Move selected columns to new positions.
- Rename the columns.
- Configure how the BOM table is sorted.
- Configure the footer value for each individual column.
- Configure the units and how they are displayed for each column.
- Configure the width of each column with column properties.
- Configure the visibility of each individual column.
- Filter the components with a filter expression.
- Change the title of both the title row and the footer row.
- Search for components in the table.
- Highlight and select component instances.
- Save the BOM configuration as a template file.
- Load a BOM table configuration from a template file.
- Place the BOM table in a drawing.
- Delete BOM tables.

A Mechanical or Ultimate license is required to modify or create bills of materials. However, the manager is available with a Pro license.




1 Menu

2 Create a table

### 4.11.11.1 Menu

Opens the main menu.

**New BOM**
Opens a submenu, which includes:

- **Copy the current table**: adds a new BOM table using the template of the current table.

**BOM tables in document**
opens a submenu.

- **Go to first table**: switches to the first BOM table in the document.
- **Go to previous table**: Switches to the previous BOM table in the document.
- **Go to next table**: Switches to the next BOM table in the document.

- **Go to last table**: Switches to the last BOM table in the document.
- **<Name of the BOM>**: Switches to the corresponding BOM table in the document.

**Place/Update BOM table**
Starts the interactive placement of the table in the drawing. If the table is already in the drawing, this tool will update the table.

**Export to CSV**
Saves the content of the current table to a CSV file.

**Replace from template…**
Replaces the current BOM configuration with the one from the template.

**Save as template…**
Specify the location of the BOM template file.

**Delete current BOM**
Deletes the current BOM table from the document.

**Set filter**
Opens the Filter Configuration dialog box, which allows you to specify the filter configuration.

**Selection set**
Opens a submenu.

- **Entire model**: The BOM table will be created for the entire model.
- **Subassembly**: The BOM table will be created for a selected instance of a subassembly.
- **Drawing view**: The BOM table will be created for the parts selected for the corresponding drawing view, regardless of their visibility in the viewport.
- **Layout viewport**: The BOM table will be created for the selected layout viewport.
- **Custom selection**: The BOM table will be created only for the selected parts.
- **Show selection**: Shows the current selection for the BOM table.
- **Edit**: Allows you to edit the selection.

**Refresh**
Updates the current BOM and the list of all BOM tables in the drawing.

**Insert column after**
Inserts a column from the **Column selector** after the selected column.

**Insert column before**
Inserts a column from the **Column selector** before the selected column.

***Прим.:*** If no column is selected, a new column is added after the last column in the table.

**Insert formula column**
Opens a formula editor, which allows you to configure a formula column.

**Insert template column**
Opens a template editor, which allows you to configure a template column.

**Edit evaluated column**
Allows you to edit the selected formula or template column.

**Delete column**
Removes the selected column from the table.

**Move column left**
Moves the selected column(s) one column to the left.

**Move column right**
Moves the selected column(s) one column to the right.

**Make column visible**
Shows the selected column in the associated table in the drawing.

**Make column invisible**
Removes the selected column from the associated table. Invisible columns remain visible in the BOM manager but are marked with a diagonal hatch pattern in the background.

**Associated table**
Opens a submenu.

- **Zoom to**: Zooms to the table. If necessary, sets the table layout as the current one.
- **Select**: Selects the table. If necessary, sets the table layout as the current one.
- **Delete**: Deletes the table from the document.
- **Detach**: Detaches the table. The table in the document is no longer updated when the BOM table is modified.

**Footer**
Opens a submenu, for each of the selected columns.

- **No footer**: Removes the footer.
- **Sum**: Displays the sum in the footer.
- **Average value**: Displays the average in the footer.
- **Minimal value:**: Displays the minimum value in the footer.
- **Maximal value:**: displays the maximum value in the footer.

**Sorting**
Opens a submenu.

- **Automatic order**:Sorts the table automatically**.**
- **Natural order**: Sort the table by the natural order of component instances in the document.
- **Ascending order**: Sorts the selected columns in ascending order.
- **Descending order**: Sorts the selected columns in descending order.

**Aggregate function**
Opens a submenu.

- **No function**: Removes the aggregate function.
- **Sum**: Displays the sum of all values in the group.
- **Average value**: Displays the average of all values in the group.
- **Minimal value**: Displays the minimum of all values in the group.
- **Maximal value**: Displays the maximum of all values in the group.

- **Concatenation**: displays a concatenation of all values in the group in ascending order, separated by a specified delimiter.
- **Concatenation with count**: Displays a concatenation of all values in the group with their corresponding number of encounters in ascending order. Value and count pairs are separated by a specified delimiter.

**Disable/Enable warnings**
Controls the display of warning messages during the creation or update of BOM tables.

### 4.11.11.2 Create a table

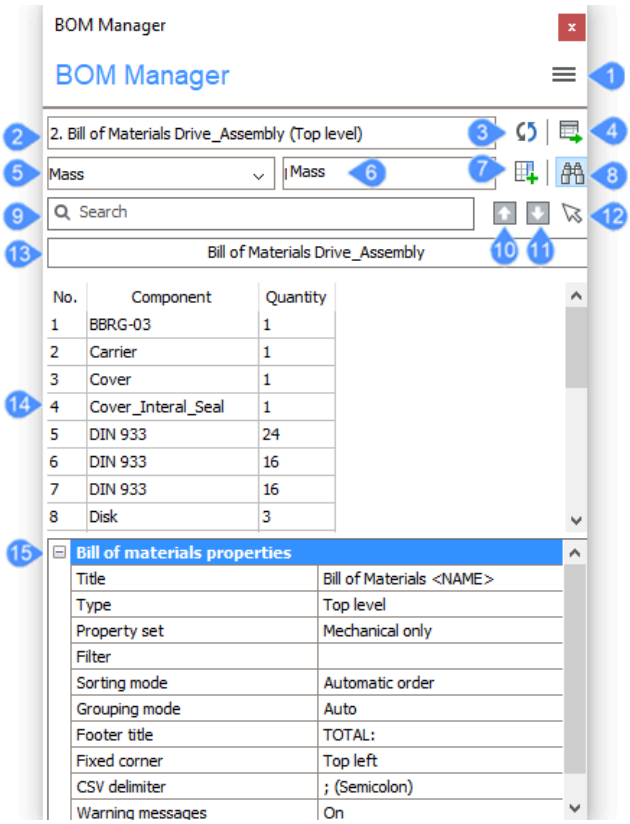
**Top level**
Creates a new Top level BOM.

**Bottom level**
Creates a new Bottom level BOM.

**Hierarchical**
Creates a new Hierarchical BOM.

**Create from template**
Opens a dialog box, which allows you to select a BOM Table template (*.BOM) file.

To start working with the BOM manager, you will have to create a new BOM table.




1 Menu

2 BOM Selector

3 Refresh

4 Place/update table

5 Category selector

6 Column selector

7 Insert column after

8 Find

9 Search

10 Find previous

11 Find next

12 Select all found

13 BOM title

14 BOM table

15 BOM and Column Properties

**BOM Selector**
Selects the current BOM table. The selection is stored in the drawing. When the drawing is closed and reopened, the selected BOM is restored in the manager. When selected, the table is updated in the current state of the drawing.

**Refresh**
Updates both the current BOM and the list of all BOM tables in the drawing.

**Place/update table**
Starts the interactive placement of a table in a drawing. If the table has already been placed in the drawing, it updates it. All edit operations automatically update the associated table in the drawing.

**Category selector**
Select the category of the properties that are displayed in the column selector. The category can be added to the current BOM pressing button no. 7.

**Column selector**
Select the column to add to the current BOM.
You can type here the formula for a formula field. To add a formula, type an expression starting with ”=”. To add a template expression type “$”, to refer columns, use <>. For example, to refer to the Mass column you will type: “=<Mass>”.
When you add a formula field and press button no. 7 the Formula column_1 dialog box will appear.

**Insert column after**
If a column is selected, it will insert the column from the Column selector after the selected column in the table.
If no columns are selected, the new column will be inserted after the last column in the table.

**Find**
Toggles the search field on and off. When you toggle on this button, buttons no. 9, 10, 11 and 12 will appear.
If you click on the BOM Manager panel, you can toggle the search field using **CTRL+F**.

**Search**
Type a string to search and press Enter; wildcards are supported.

**Find previous**
Selects the previous match on the BOM table.
You can also use **Shift+F3**.

**Find next**
Selects the next match in the BOM table.
You can also use **F3**.

**Select all found**
Selects all the items searched in the search field.

**BOM title**
Displays the title of BOM table. You can type the name you want instead of <NAME>. Press Enter or exit the field to apply the changes.

**BOM table**
Displays the content of the BOM table.

Right-click a column label offers additional tools:

- **Add template column from selected**: opens a template editor to configure a new template column. The selected columns will be automatically added to the template expression.
- **Column role**: opens a submenu.
  - **Number**: the column will be used as a source of numbers by the bmBalloon command.
  - **Name**: the column will be used as a source of names by the bmBalloon command.
  - **Quantity**: the column will be used as a source of quantities by the bmBalloon command.
  - **Regular**: the column will have no specific role.
- **Format**: opens the Format dialog box, which allows you to specify the format for the selected column(s).
- **Set column filter**: opens the Column Filter dialog box, which allows you to configure the column filter.
- **Fit column(s)**: fits the width of the selected column(s) to their content.
- **Fit all columns**: fits the width of all columns to their content.

  ***Прим.:*** The column width automatically fits the content unless the width has been changed manually.

Right-click on cells offers additional tools:

- **Zoom to**: zooms to the select part(s) in the model.
- **Select**: selects the part(s). Replaces an existing selection.
- **Add to Selection**: adds part(s) to the selection.
- **Remove from Selection**: removes the parts from the selection.
- **Remove highlighting**: removes highlighting.

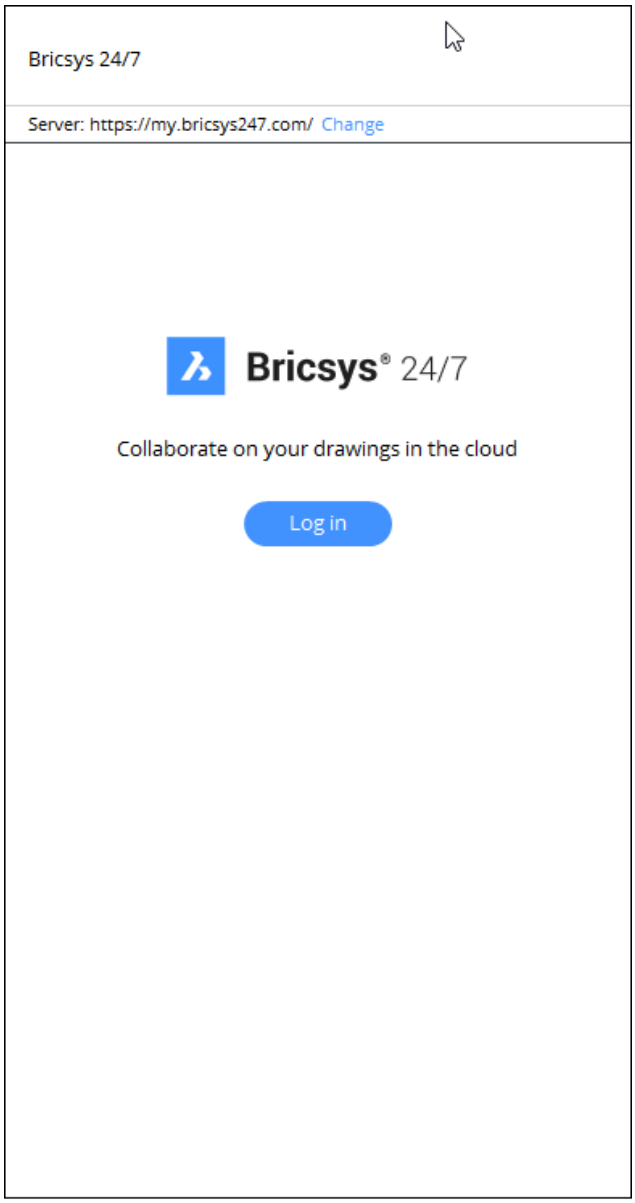
**BOM Properties**
Displays the BOM and columns Properties. The columns properties will appear after you insert a column on the BOM table.

### 4.11.12 Bricsys 24/7 panel

The Bricsys 24/7 Panel helps team members to find and share their drawings.

Bricsys 24/7 makes project management, a collaboration between project teams, cloud computing security, and task automation possible. A big advantage of this cloud-based service is that BricsCAD is directly linked to it by the Bricsys 24/7 Panel.

If you are logged out of your 24/7 account a screen will display:




1 Enter your Email address and Password and press the 'Log in' button.

2 (Option) Tick the**Remember Me**checkbox to store the username and password.

3 Once logged on, you will see an overview of your 'in the cloud' Projects, Folders, and Documents.

#### 4.11.12.1 Project

The first thing you see on the 24/7 Panel when logged on, are all the projects you are assigned to.

#### 4.11.12.2 Project Overview

At the top of the panel you see your project. You can go back to the overview of projects by clicking on it.

#### 4.11.12.3 Folder

When you open a project, a list of folders is shown.

#### 4.11.12.4 Folder Overview

At the top of the panel you see the folder you're in. You can go back to the overview of by clicking it.

#### 4.11.12.5 Documents

In a folder you find documents. On the right-hand icons are displayed:

The profile picture or the initials from the author of the document.

The document is in the cloud.

Someone is currently working on the document.

The document is currently uploading.

The document is locked by someone, another person cannot change it.

Right-click on a folder to get a menu with 2 possible actions:

- Download files not locally available.
- Download out of date locally unmodified files.

Right-click on a document to get a menu with possible document actions

- Lock on server
- Unlock on server
- Upload local copy as a revision
- Download latest revision

### 4.11.13 Command line panel

The Command line panel allows you to launch commands or modify system variables by typing the command or variable name.



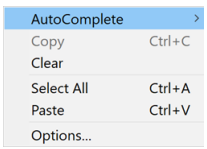

1 Command line

2 Command history

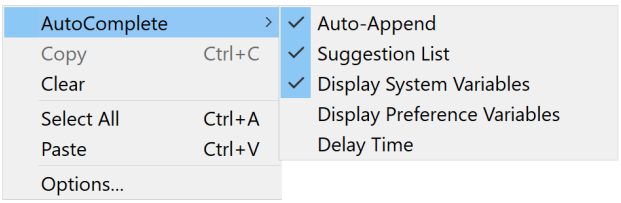
#### 4.11.13.1 Command line

Displays the current command prompt. If no command is active, it prompts you to Enter Command. If a command is active, it displays prompts and options relevant to the current command. Prompt shortcuts are highlighted. You may enter the shortcut characters in the Command line or click on them with the left mouse button.

A right-click menu offers relevant tools.



**AutoComplete**
Hover over the AutoComplete option. Another context menu displays:




- If **Auto-Append** is checked, the entry is automatically completed if only one possible command is left.
- If **Suggestion List** is checked, the list of possible commands is displayed when typing in the command bar.
- If **Display System Variables** is checked, the system variables are included in the suggestion list.
- The **Delay Time** option sets the time in seconds before automated keyboard features display.

**Copy**
This option copies the selected text to the clipboard.

**Clear**
This option clears complete prompt history.

**Select All**
This option selects all the completed content of the prompt history.

**Paste**
This option pastes the text from the clipboard to the Command line.

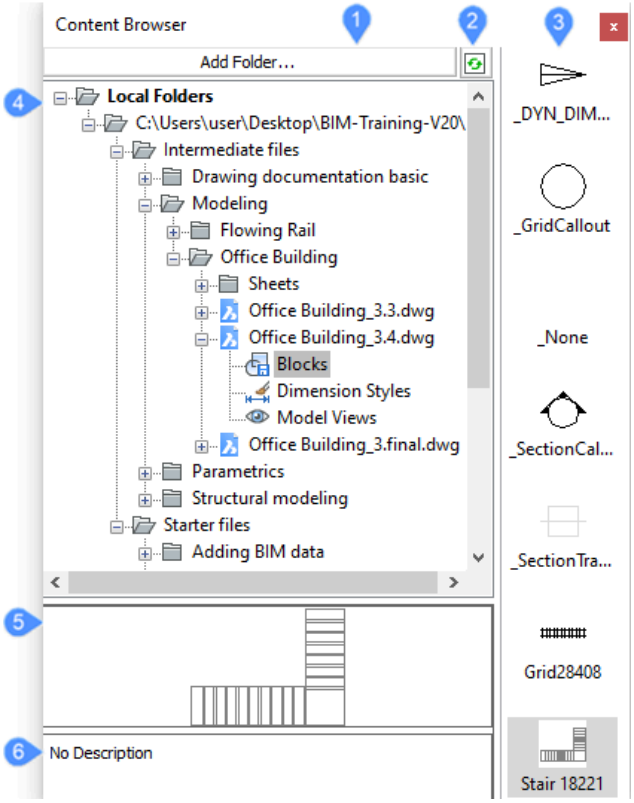
**Options**
This option displays the Settings dialog box at the Command line section.

### 4.11.13.2 Command History

Displays the history of the Command line input for the current BricsCAD session.

## 4.11.14 Content browser panel

The Content Browser panel offers you a quick and convenient way to open drawings, manage your block libraries, copy dimension styles, place views for your entire .dwg files stored on your computer.




1 Add Folder...

2 Refresh

3 Definitions in the selected drawing

4 Local Folders tree

5 Definitions' preview

6 Description

### 4.11.14.1 Add folder...

Displays the Choose a folder dialog box.

### 4.11.14.2 Refresh

Select a drawing and press the Refresh button to expand its content (Blocks, Dimension Styles, Model Views).

### 4.11.14.3 Definitions in the selected drawing

Select Blocks/Dimension styles/Model views to see the definitions used within the drawing.

You can drag & drop a block onto the current drawing and an Insert Block dialog box will appear.

If you drag & drop a dimension style to your drawing a Copy / Paste dialog box will appear.

#### 4.11.14.4 Local Folders tree

Displays the content of a drawing/project. Right-click menus enable you to add or remove folders from within the Folders tree.

#### 4.11.14.5 Definitions' preview

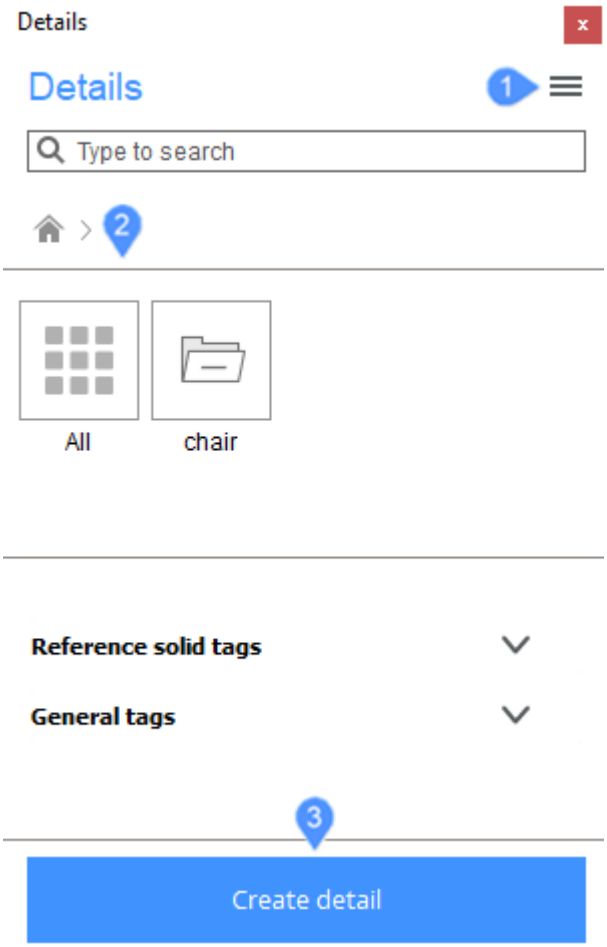
Shows a preview of your drawing/block/dimension style.

#### 4.11.14.6 Description

Displays a description of your selected definition.

### 4.11.15 Details panel

The Details panel allows you to view details, add tags to details, propagate and adjust details.




1 Menu

2 Details library

3 Create detail

### 4.11.15.1 Menu

**Grid View**
Displays the details in a grid view.

**List View**
Displays the details in a list view.

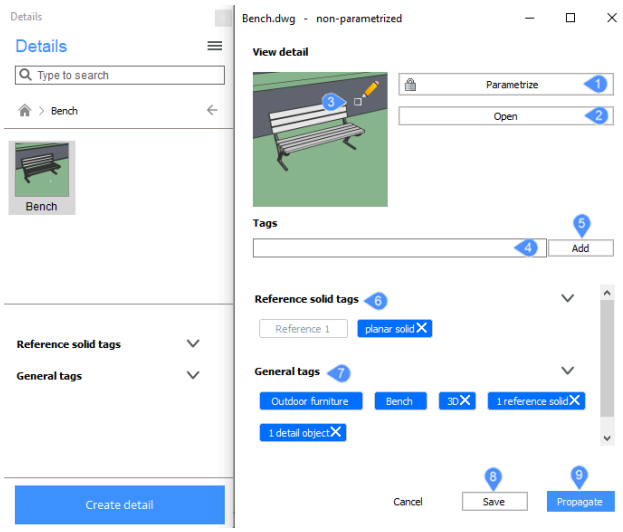
**Generate thumbnails**
Loads the new details in your library panel.

**Manage libraries**
Opens the Settings dialog box to change the details directory path.

### 4.11.15.2 Details library

You can click on the category of the detail you want to view. The details assigned to that category will be displayed.

When you click on a detail, the below dialog box will appear.



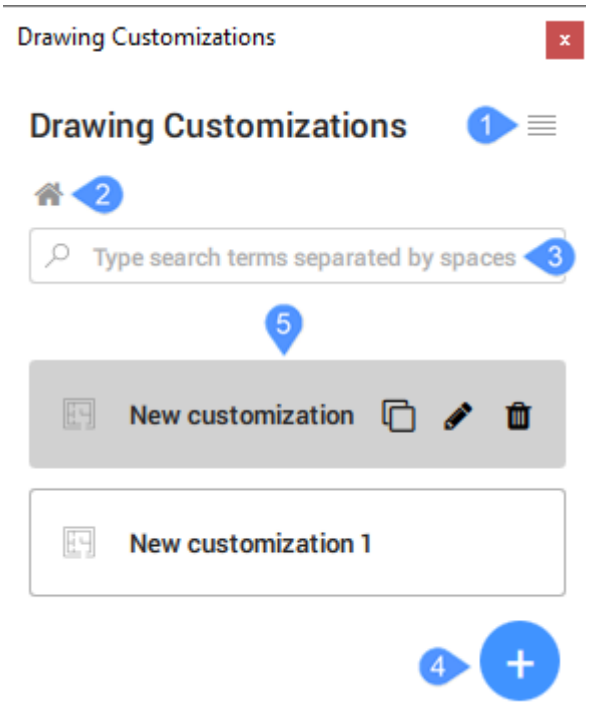

1 Parametrize
2 Open
3 Edit detail
4 Tags
5 Add tag
6 Reference solid tags
7 General tags
8 Save
9 Propagate

### 4.11.15.3 Create detail

Opens the Create detail dialog box.

## 4.11.16 Drawing customizations panel

The Drawing Customizations panel allows you to create custom templates for drawings.




1 Menu

2 Home

3 Search

4 Add customization

5 Customizations List

### 4.11.16.1 Menu

**Refresh**
Refreshes the Drawing Customizations panel.

**Manage libraries**
Opens the Settings dialog box to change the section settings search path.

**Load customizations**
Allows you to choose a folder that contains customizations.

### 4.11.16.2 Home

Moves the cursor to the main user interface page of the panel.

### 4.11.16.3 Search

Searches all the drawing customizations that matches the words you enter in the search box.

### 4.11.16.4 Add customization

Creates a new template.

You can create a template by clicking on the button. By default, the template is named **New Customization** with the text highlighted. After you have added a relevant name for the template, press Enter to save and apply the changes. The inserted customizations will be sorted alphabetically automatically. Use the scroll button to navigate through the list.

### 4.11.16.5 Customizations List

Displays all the available drawing customizations.

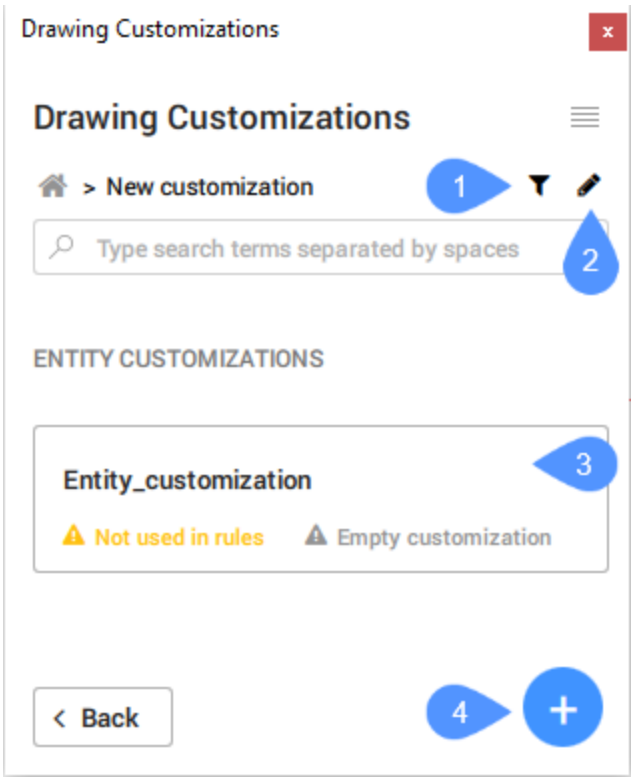
**Duplicate**
Creates a copy of your selected drawing customization.

**Rename**
Edits the name while the text is being highlighted. Press Enter to save and apply the changes.

**Delete**
Removes the drawing customization. The Removing a drawing customization dialog box appears.

**Entity customizations**
Entity Customizations refer to the visibility settings of how you would like to have your elements in your model drawing to look like.
The **Entity customizations list** is empty for a newly created customization.

To create an entity customization left-click on the desired template, then click on the button. By default, the template is named **New Customization** with the text highlighted. Rename it while the text is being highlighted. Press Enter to save and apply the changes.
After creating your desired entity customization templates, click on the individual tabs to modify their respective settings.




1 To rule definitions

2 To style definitions

3 Entity customizations list

4 Add customization

**To rule definitions**

You can create a rule pressing the button. After creating the rule, the button offers you another 3 options:

- Entity customization
- Sub rule
- Filter

When creating an entity customization, you can add a sub rule for a rule already added.

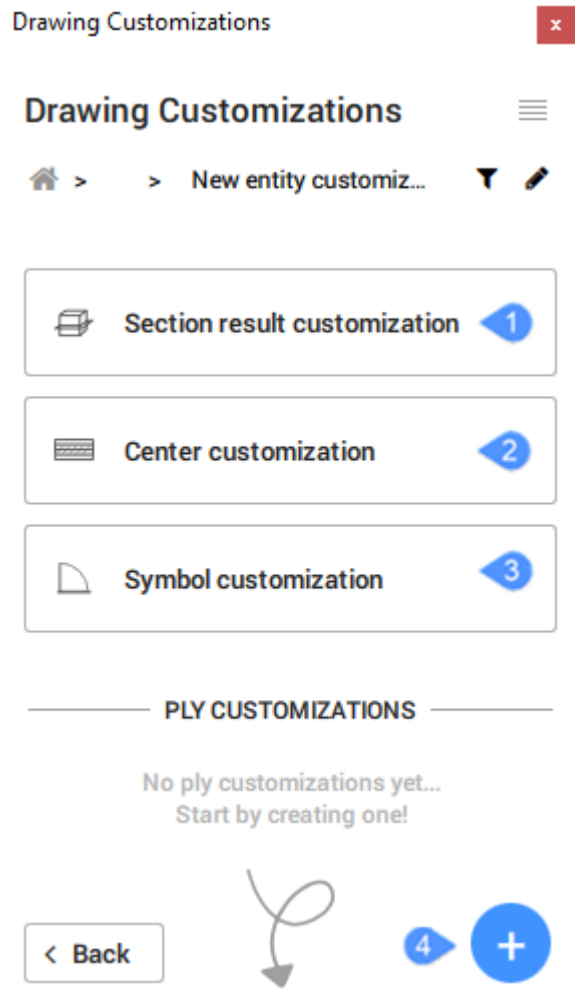
**To style definitions**

To add a style definition, press the button.
When selecting a color, the Select Color dialog box will appear.
When selecting a line type and pressing Load…, the Load Linetypes dialog box will appear.

**Entity customizations list**

Displays the list of entity customizations.

**Add customization**




1 Section result customization

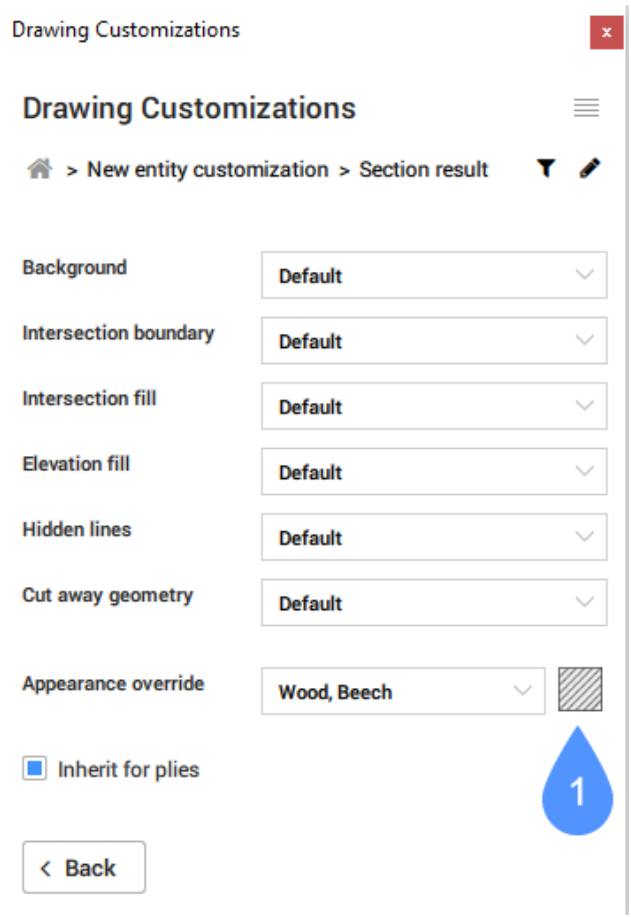
2 Center customization

3 Symbol customization

4 Add ply customization

**Section result customization**

Determines how the different pieces of geometry, generated by sectioning the entity with a section plane, are displayed.

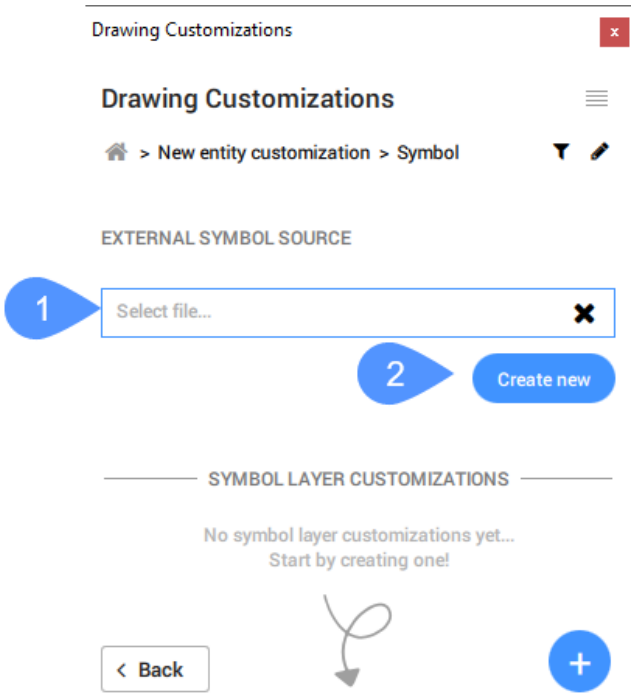



1 Physical Materials

**Physical Materials**
Opens the Physical Materials dialog box.

**Center customization**
Determines the appearance of the center of planar entities (center plane) or linear entities (center line).

**Symbol customization**
Determines the appearance of 2D symbols that are added for the entity and controls the source of those symbols.

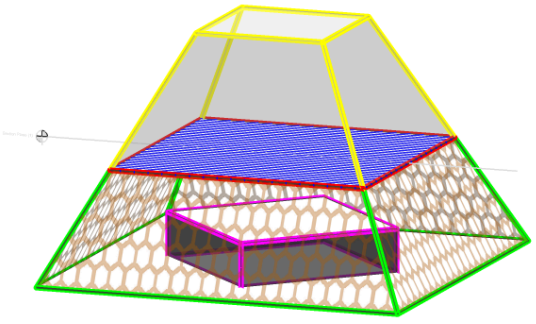



1 Select file...

2 Create new

**Select file…**
Opens the Set external symbol source dialog box.

**Create new**
Opens the New external symbol source dialog box.
The section result customization and center customization display a series of controls for which you can choose values from the drop-down list.



The diagram above illustrates how you could control the visibility and customization settings of an entity with the Drawing Customization tool. Notice that the section plane is aligned with the Blue square with Red outline.

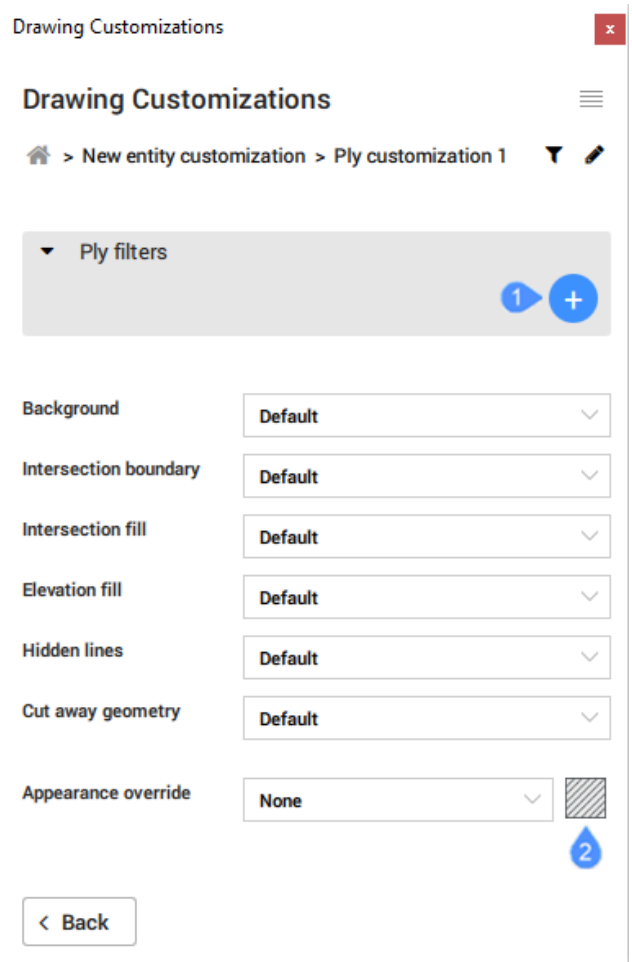
- Green outline represents **Background**, which is essentially the elevation outline of the cut object.
- Red outline represents **Intersection Boundary**, which traces over the cut area.

- Blue hatch represents **Intersection Fill**, as it shows the area of being cut by the section plane.
- Orange hexagons hatch represents **Elevation Fill**, which refers to the area bound by the elevation outline (see Background).
- Pink outline represents **Hidden Lines**, as it is below the blue hatch.
- Yellow outline represents **Cut away geometry**, as it is above the section plane.

**Appearance Override** allows you to specify how an entity's hatch appearances (section and elevation) would appear, in place of the default hatch patterns already defined within the Physical Materials library. When you create any new Physical Material entry in the "In Project" category as part of the Drawing Customization template, your desired configurations are independent of the drawing project and stored in the template instead. You can later apply this template to other project models as you wish.

### Add ply customization

To add a ply customization, press the button.




1 Add default function

2 Physical Materials

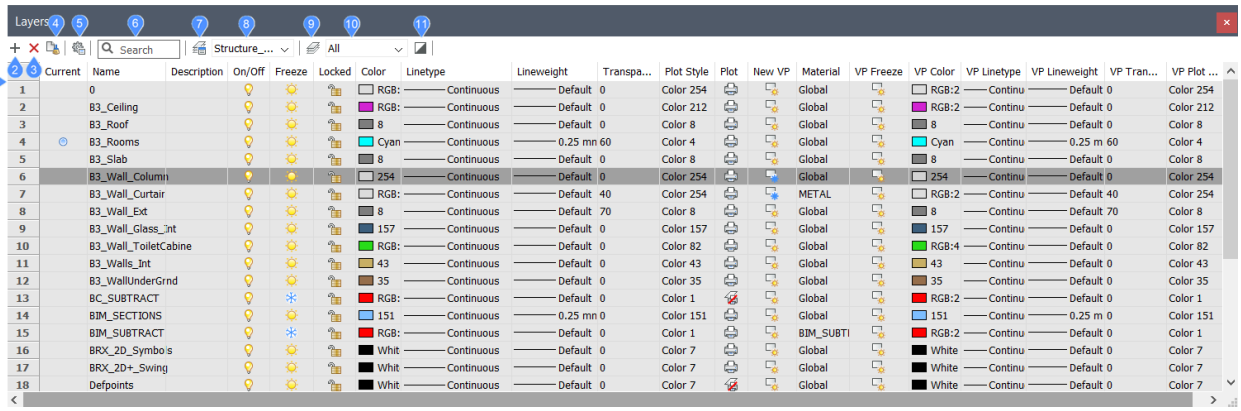
### Add default function

Adds a default function for the ply customization.

**Physical Materials**
Opens the Physical Materials dialog box.

### 4.11.17 Layers panel

The Layers panel allows you to edit layer names and properties in the current drawing, as well as create, delete, edit, apply layers, layers states and layer filters.




1 Layer list

2 New layer

3 Delete

4 Purge

5 Settings

6 Search

7 Layer States Explorer

8 Layer State Control

9 Layers Explorer

10 Filter used

11 Invert filter

#### 4.11.17.1 Layer list

Displays a list of layers and their properties. The list may include all layers in the current drawing or a subset of those layers based on selected layer states or filters. Column headers describe the properties of each layer. You may sort the layer list in order or reverse order for any of the properties by clicking once or twice on the column header.

A right-click menu on the column headers allows you to customize them. You can enable or disable the columns and restore the column positions.

A context menu displays when you right click menu on a layer name.

**Merge to**
Merges the selected layers to a target layer.

***Прим.:*** Merged layers will be purged from the drawing.

**Remove From Group:**
Removes the layer from group.

**Select All:**
Selects all available layers. You can also use the keyboard shortcut Ctrl+A.

**Remove Viewport Overrides:**
Removes overrides from properties that were set differently for viewports.

- **From Selected Layers**: Removes the overrides from the selected layers.
- **From All Layers**: Removes the overrides from all layers.
- **For Current Viewport**: Removes overrides from the current viewport.
- **For All Viewports**: Removes overrides from all viewports in the drawing

**Isolate Selected Layers:**
Isolates the selected layers. All other layers are either locked or turned off.

### 4.11.17.2 Current

Specifies the current working layer. Only one layer can be current at a time.

### 4.11.17.3 Name

Specifies the layer name. To rename a layer, double-click the layer name or select Rename from the right-click menu.

### 4.11.17.4 Description (optional)

Specifies the content or purpose of the layer. To create or edit a description, select one or more layers and then double-click the description field for one of the selected layers. The text you enter is applied to all selected layers.

### 4.11.17.5 On/Off

Specifies the On/Off status of the layers. Entities on layers are turned off are not visible in the drawing. However, they are processed during operations that require a regeneration of the drawing. Turning a layer on and off does not require the drawing to regenerate. Therefore, this toggle is most useful when you want entities on a layer to be hidden for only a short time while you edit the drawing. To change the status, select one or more layers and then click a lightbulb icon for one of the selected layers. The status is applied to all selected layers.

To change the status, select one or more layers and then click a light bulb icon:

- **On**: Displays entities that are on that layer.
- **Off**: Hides entities that are on that layer.

#### 4.11.17.6 Freeze

Specifies the Thaw/Freeze status of the layers. Entities on frozen layers are not visible in the drawing and are not processed during operations that require a regeneration of the drawing. This can save processing time if the layer includes many entities. However, the process of freezing and thawing a layer requires the drawing to regenerate. Therefore, this toggle is most useful when you want entities on a layer to be hidden for an extended time while you edit the drawing. To change the status, select one or more layers, and then click a sun/snowflake icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw**: Displays entities that are on that layer.
- **Freeze**: Hides entities that are on that layer.

#### 4.11.17.7 Locked

Specifies the lock status of the layers. Entities on locked layers cannot be edited. They remain visible; however, they are dimmed based on the LAYLOCKFADECTL System Variable. To change the status, select one or more layers, and then click a padlock icon for one of the selected layers. The status is applied to all selected layers.

- **Unlocked**: Allows entities on that layer to be edited.
- **Locked**: Prevents entities on that layer from being edited.

#### 4.11.17.8 Color

Specifies the color of the layers. To change the color, select one or more layers and then click the current color for one of the selected layers. The Select Color dialog box is displayed. The color you select is applied to all selected layers.

#### 4.11.17.9 Linetype

Specifies the linetype of the layers. To change the linetype, select one or more layers and then double-click the linetype for one of the selected layers. If the linetype you want is not displayed in the drop-down list, choose Load to display the Load Linetypes dialog box. The linetype you select is applied to all selected layers.

#### 4.11.17.1 Lineweight 0

Specifies the Lineweight of the layers. To change the lineweight, select one or more layers and then double-click the lineweight for one of the selected layers. The lineweight you select is applied to all selected layers.

#### 4.11.17.1 Transparency 1

Specifies the transparency of the layers. To change the amount of transparency, select one or more layers and then double-click the transparency for one of the selected layers. You can enter a value between 0

(fully opaque) and 90 (almost fully transparent) or select a predefined value from the drop-down list. The value you specify is applied to all selected layers.

The TRANSPARENCYDISPLAY System Variable must be turned on to experience transparency.

#### 4.11.17.1 Plot style

Specifies the plot style of the layers. If plot style mode (PSTYLEMODE system variable) for the current drawing is set to color dependent plot styles, the layers' plot styles use the layers' colors and are read-only. If plot style mode for the current drawing is set to named plot styles, you can change the plot style. To specify a different named plot style, select one or more layers and then click the current plot style for one of the selected layers. The Select Plot Style dialog box is displayed. The plot style you select is applied to all selected layers.

#### 4.11.17.1 Plot

Specifies the print status of the layers. Entities on No Print layers remain visible in the drawing but are not printed. To change the status, select one or more layers, and then click the printer icon for one of the selected layers. The status is applied to all selected layers.

- **Print**: Prints the layer.
- **No Print**: Does not print the layer.

Some layers created by BricsCAD®, such as Defpoints, are set to No Print by default and cannot be changed.

#### 4.11.17.1 New VP

Specifies the default Thaw/Freeze status of layers for new layout viewports. To change the status, select one or more layers and then click the sun/snowflake icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw**: Displays entities that are on that layer when a new viewport is created.
- **Freeze**: Hides entities that are on that layer when a new viewport is created.

#### 4.11.17.1 Material

Specifies the material of the layers. To change the material, select one or more layers and then double-click the material for one of the selected layers. The material you select is applied to all selected layouts. If the material you want is not displayed in the drop-down list, you can add it to the current drawing using the MATERIALS command.

#### 4.11.17.1 VP Freeze

**6**

Specifies the Thaw/Freeze status of layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space Thaw/Freeze status. To change the status, select one or more layers and then click the sun/snowflake icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw**: Displays entities that are on that layer in the current layout or viewport.
- **Freeze**: entities that are on that layer in the current layout or viewport.

#### 4.11.17.1 VP Color

**7**

Specifies the color of layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space color.

To change the color, select one or more layers and then click the current vp color for one of the selected layers. The Select Color dialog box is displayed. The color you select is applied to all selected layers.

#### 4.11.17.1 VP Linetype

**8**

Specifies the linetype of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space linetype.

To change the linetype, select one or more layers and then double-click the vp linetype for one of the selected layers. If the linetype you want is not displayed in the drop-down list, choose Load to display the Load Linetypes dialog box. The linetype you select is applied to all selected layers.

#### 4.11.17.1 VP Lineweight

**9**

Specifies the Lineweight of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space lineweight.

To change the lineweight, select one or more layers and then double-click the vp lineweight for one of the selected layers. The lineweight you select is applied to all selected layers.

#### 4.11.17.2 VP Transparency

**0**

Specifies the transparency of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space transparency.

To change the amount of transparency, select one or more layers and then double-click the transparency for one of the selected layers. You can enter a value between 0 (fully opaque) and 90 (almost fully transparent) or select a predefined value from the drop-down list. The value you specify is applied to all selected layers.

The TRANSPARENCYDISPLAY system variable must be turned on to experience transparency.

#### 4.11.17.2 VP Plotstyle

**1**

Specifies the plot style of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space plot style. If plot style mode (PSTYLEMODE system variable) for the current drawing is set to color dependent plot styles, the layers' plot styles use the layers' colors and are read-only.

If plot style mode for the current drawing is set to named plot styles, you can change the plot style. To specify a different named plot style, select one or more layers and then click the current plot style for one of the selected layers. The Select Plot Style dialog box is displayed. The plot style you select is applied to all selected layers.

#### 4.11.17.2 New layer

**2**

Creates new layer with the generic name **NewLayer1**. The new layer that is created copies the properties of the currently selected layer.

#### 4.11.17.2 Delete

**3**

Deletes layers, except for the layer with the name **0** and any layer with entities, which cannot be deleted.

#### 4.11.17.2 Purge

**4**

Purges layers from the current drawing, except for the following ones, which cannot be purged:

- The layer 0.
- The current layer.
- Any layer with entities.

#### 4.11.17.2 Settings

**5**

Displays the Layer Settings dialog box.

#### 4.11.17.2 Layer States Explorer

**6**

Displays the Drawing Explorer dialog box with the Layer States category selected.

#### 4.11.17.2 Layer State Control

**7**

Displays the current layer state if there are any defined in the drawing. Click the down-arrow button to choose a different layer state.

#### 4.11.17.2 Layers Explorer

**8**

Displays the Drawing Explorer dialog box with the Layers category selected.

#### 4.11.17.2 Filter used

**9**

Displays the current layer filter if there are any defined in the drawing. Click the down-arrow button to choose a different layer filter. Some filters are automatically created by BricsCAD®: All, All Used Layers.

#### 4.11.17.3 Invert filter

**0**

Inverts the content of the current filter so that the dialog box displays all layers not included in the filter.

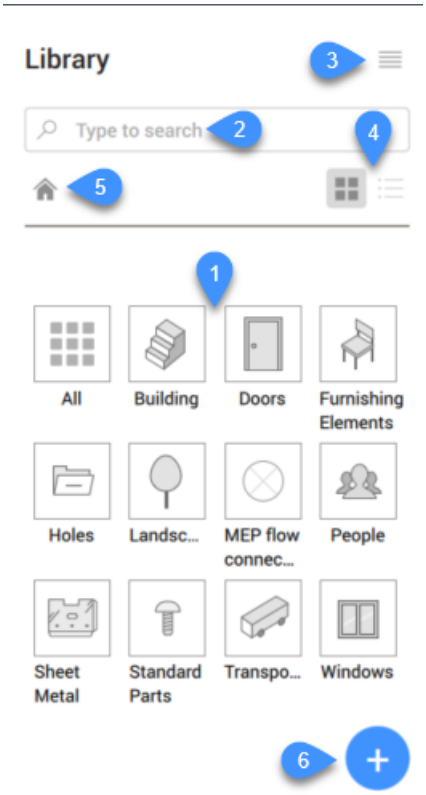
#### 4.11.17.3 Search

**1**

Allows you to search for specific layer names. The list is shortened to display only the layers that contain the entered sequence of characters, regardless of where they are located in the layer name.

### 4.11.18 Library panel

The Library panel offers a central location to access 2D and 3D block libraries.




1 Block library

2 Search

3 Menu

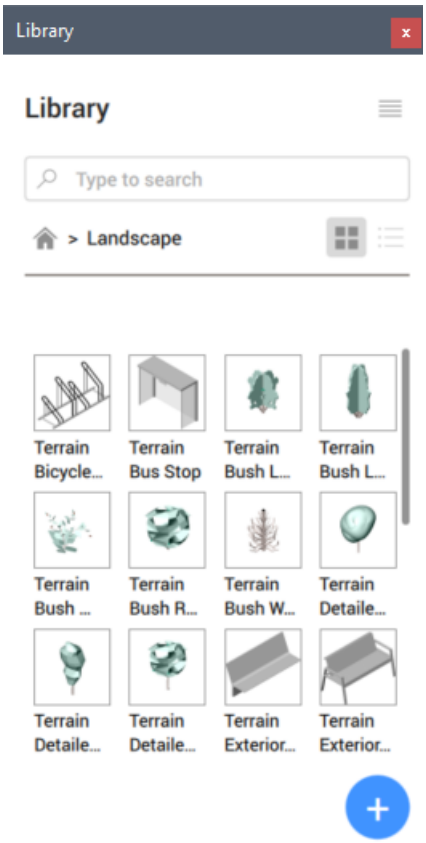
4 View

5 Home

6 Add

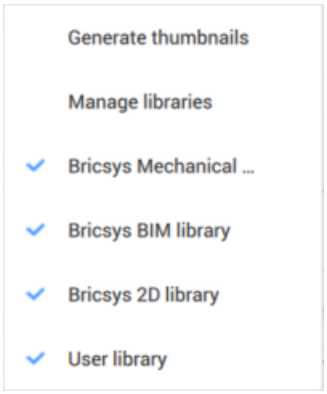
### 4.11.18.1 Block library

The block library is organized by categories. Select a category to view its blocks. Categories and blocks may be user-defined or pre-defined by BricsCAD®. You can scroll through the categories and blocks using the mouse wheel or the scroll bar. The availability of pre-defined blocks depends on your license level.




### 4.11.18.2 Menu

The Library panel menu enables you to control what is displayed in the block library.




**Generate thumbnails**
Generates or updates the thumbnail images of all blocks.

**Manage libraries**
Opens the Settings dialog box to change the library directory path.

**Bricsys BIM library**
When checked, the panel displays BIM components. The default path for BIM components is: *C:\Program Files\Bricsys\BricsCAD V21 en_US\UserDataCache\Support\en_US\Bim\Components*..

**Bricsys Mechanical library**
When checked, the panel displays Mechanical components. The default path for Mechanical components is: *C:\Program Files\Bricsys\BricsCAD V21 en_US\UserDataCache\Support\en_US\DesignLibrary*.

**User library**
When checked, the panel displays the user defined components. The path for user library is defined by the COMPONENTSPATH system variable.

#### 4.11.18.3 Search

Searches the library for words you enter in the search box.

#### 4.11.18.4 View

Toggles between grid and list views of the block content.

#### 4.11.18.5 Home

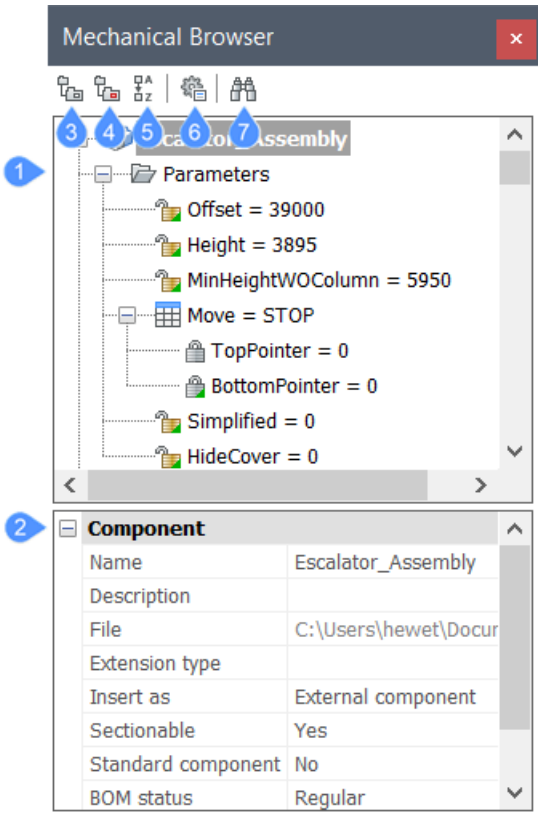
Returns the block library to the home screen.

#### 4.11.18.6 Add

Opens the Add Block to Library dialog box.

### 4.11.19 Mechanical browser panel

The Mechanical browser panel offers a central location to view and modify parametric properties of mechanical parts and assemblies.




1 Parameter list

2 Properties

3 Group by entity

4 Group by type

5 Sort

6 Settings

7 Show search

### 4.11.19.1 Parameter list

Displays a list of all the parameters and constraints in the drawing. They are organized in expandable categories which may include user parameters,2Ddimensional constraints, 3D dimensional constraints, 3D geometric constraints,block parameters and arrayparameters.

A context menu displays when right-click the main component name:

**Update**
Updates the hierarchy of mechanical components for the current drawing in case referenced drawing files of sub-components have been modified.

**Visual style > All by Viewport**
Applies the current visual style to all components in the assembly.

**Insert standard hardware**
Inserts a standard hardware part as a mechanical component in the current drawing.

**Switch all to local**
Converts all external components in the model to local components.

**Switch all to external…**
Switches all internal components to external components.

**Select all**
Selects all components with the same definition.

**Select same**
Selects all components with the same name and the same parameter values.

**Highlight all**
Highlights all components with the same definition.

**Highlight same**
Highlight all components with the same name and the same parameter values.

**Create exploded view**
Creates a block with an exploded representation of the current.

**Dissolve**
The selected feature is removed from the part, but it will keep its geometry. However, design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

**Delete**
This is the analogue of the SMDELETE command. In this case the feature is removed from the browser and geometry is changed depending upon the type of the feature.

**Disable**
The feature will remain in the Mechanical Browser, but the part will behave like the feature was dissolved. It is beneficial in contrast with dissolve, that feature still updates on geometry changes and you can turn it on when you want, avoiding recreating it or recognizing by SMCONVERT.

Right-clickmenusoffer additional toolsfor components:

**Open**
Opens the referenced drawing.

**Open a copy**
Opens a copy of a component insert as a new drawing.

**Update**
Reloads all referenced components from external files and updates BOM tables.

**Replace…**
Replaces a component insert.

***Прим.:*** Replacing a local insert turns it into an external insert.

**Set material to component**
Opens the Physical Materials dialog box, which allows you to assign a physical material to the component.

**BOM status**
Controls the appearance of the component in BOM tables.

Right-clickmenusoffer additional toolsfor parameters:

**Geometry-driven**
If checked, it makes the parameter geometry-driven.

**Create design table**
Creates a design table to drive parametric block parameters.

**Animate**
Animates models by means of parameters.

**Link to parameter**
Links subcomponent parameter to the main level parameter.

### 4.11.19.2 Properties

Specifies the properties of the selected item.

### 4.11.19.3 Group by entity

Groups 3D constraints by entity. Expand an entity to view its associated constraints.

### 4.11.19.4 Group by type

Groups 3D constraints by type. Expand a constraint to view the affected entities.

### 4.11.19.5 Sort Alphabetically

Lists mechanical components and 3D constraints alphabetically. Otherwise, they are listed in the order they are added to the assembly.

### 4.11.19.6 Settings

**Expressions of constraints**
Controls whether the numeric value or the assigned parameter name displays.

**Components parameters**
Expressions at sub-components control the visual representation of sub-component parameters: numeric value or assigned parameter name.

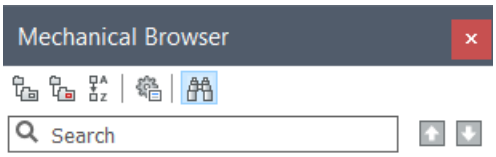
**Expressions of components parameters**
Sub-component parameters control the visibility of sub-component parameters.

**Sub-components of standard parts**
Parameters at properties add a parameters section for the selected instance to the Mechanical browser properties

### 4.11.19.7 Show search

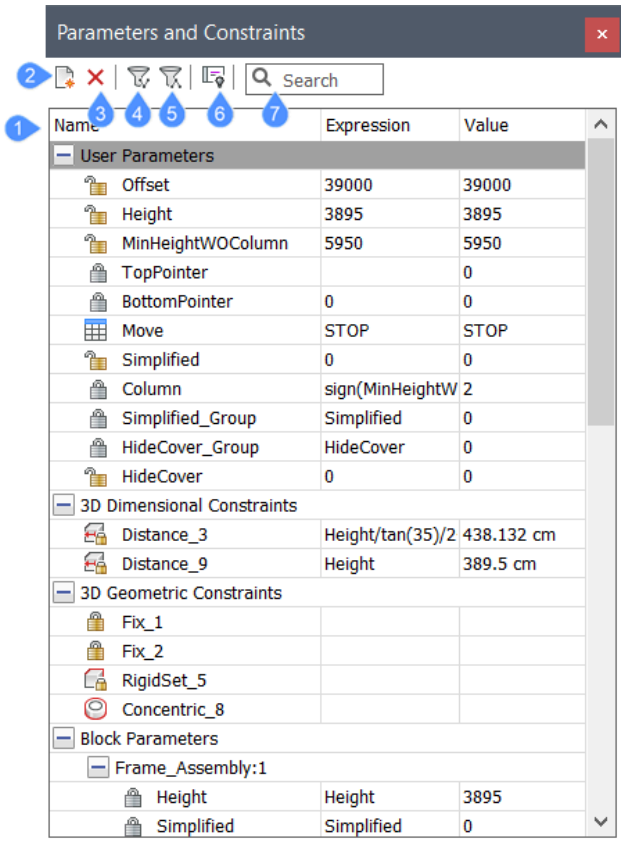
Toggles the search field and associated tools on and off. You can also press CTRL+F to toggle Search.

**Search field**
Searches the parameter list for the character string you enter. The arrow tools to the right of the search field enable you find the next or previous occurrence of the text string.




### 4.11.20 Parameters and constraints panel

The Parameters and Constraints panel offers a central location to view and modify parameters and constraints in the current drawing.




1 Parameter list

2 Properties

3 Group by entity

4 Group by type

5 Sort

6 Settings

7 Show search

#### 4.11.20.1 Parameter list

Displays a list of all the parameters and constraints in the drawing. They are organized in expandable categories which may include user parameters, 2D dimensional constraints, 3D dimensional constraints,

3D geometric constraints, block parameters and array parameters. Right-click menus offer additional tools for some parameters and constraints:

**Units**
Sets the dimension of the parameter. You can choose between: none, linear, square and cubic.

**Geometry-driven**
If checked, it makes the parameter geometry-driven.

**Create design table**
Creates a design table to drive parametric block parameters.

**Animate**
Animates parameter value within some range.

**Exposed**
Defines whether the parameter is visible and can be modified when the component is inserted in an assembly

**Clean unused variables**
Erases the unused variables.

**Link to parameter**
Links subcomponent parameter to the main level parameter.

#### 4.11.20.2 Name

Specifies the name of the parameter.

#### 4.11.20.3 Expression

Specifies the mathematical expression of the parameter.

#### 4.11.20.4 Value

Specifies the value resulting from the parameter's expression.

#### 4.11.20.5 New user parameter

Inserts a new user parameter with default values for the Expression and Value.

#### 4.11.20.6 Delete a user parameter

Removes selected parameter and constraints from the drawing.

#### 4.11.20.7 Show dependent parameters

Toggles the display of dependent parameters on or off.

#### 4.11.20.8 Show defining parameters

Toggles the display of defining parameters on or off.

#### 4.11.20.9 Hide geometrical constraints

Toggles the display of geometrical constraints on or off.

### 4.11.20.1 Search

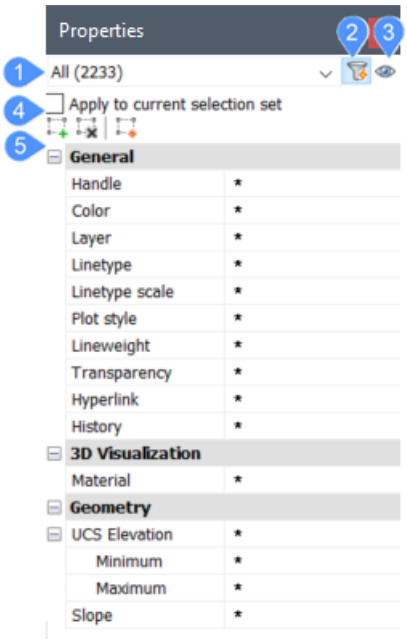
**0**

Filters the parameter list to display only the parameter names that match the character string you enter.

## 4.11.21 Properties panel

ThePproperties panel offers a central location to access drawing, viewport, and entity properties.

It can be used in either Properties mode or Quick Select mode.




1 Entities list

2 Properties/Quick Select toggle

3 Property Preview toggle

4 Selection tools (Quick Select mode only)

5 Properties list

**Entities list**

Displays the type and number of selected entities.

**Properties/Quick Select toggle**

Switches between Properties and Quick Select mode.

**Property Preview toggle**

Turns the PROPERTYPREVIEW setting on or off.

**Selection tools**

Creates a selection set based on properties. These tools are only available in Quick Select mode.

**Properties list**

Displays relevant property categories based on the current mode and selection set. You can expand and collapse categories by clicking on the + and – icons.

### 4.11.21.1 Properties mode

In Properties mode, the Properties panel enables you to view and modify drawing, viewport, and entity properties. Drawing and viewport properties are accessible when the current selection set is empty. Entity properties are accessible when the current selection set has at least one entity.

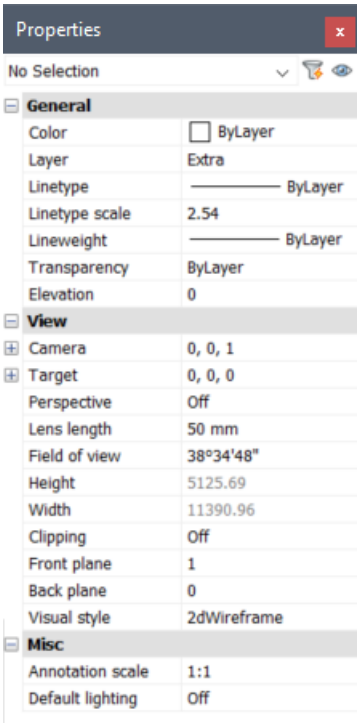
**Entities list**

When no entities are selected in Properties mode, the entities list shows No Selection and the properties list displays drawing and viewport properties.

When entities are selected in Properties mode, the entities list describes the type and number of entities in the selection set. You can filter the selection set by choosing a type of entity from the drop-down list. The types of entities in the selection set determine which properties are accessible.

**Properties list (no selection set)**

The drawing and viewport properties list, available when no entities are selected, displays General, View, and Misc properties.




**General Properties**

In Properties mode, when no entities are selected, General properties describe the current entity settings. View and modify current entity settings to specify the properties that apply to new entities as they are created. Changing the current entity settings in the Properties list has the same effect as changing the relevant system variables.

**Color**

Describes the current color (CECOLOR system variable). You can specify a new color from the drop-down list or choose Select color to access additional colors from the Select Color dialog box.

**Layer**

Describes the current layer (CLAYER system variable). You can specify a new layer from the drop-down list.

**Linetype**
Describes the current linetype (CELTYPE system variable). You can specify a new linetype from the drop-down list or choose Load to access additional linetypes from the Load Linetypes dialog box.

**Linetype Scale**
Describes the current linetype scale (CELTSCALE system variable). You can enter a new linetype scale.

**Lineweight**
Describes the current lineweight (CELWEIGHT system variable). You can specify a new lineweight from the drop-down list.

**Transparency**
Describes the current transparency level (CETRANSPARENCY system variable). You can specify a new transparency level from the drop-down list or enter a value.

**Elevation**
Describes the elevation relevant to the current UCS (ELEVATION system variable). You can enter a new elevation.

**View Properties**
When no entities are selected, View properties describe the current view and relevant view settings.

**Camera**
Describes the absolute coordinates of the camera in the current UCS, based on the cartesian coordinate system (X,Y,Z). These values automatically update when you orbit or change the viewpoint in other ways.

**Target**
Describes the absolute coordinates of the camera target in the current UCS, based on the cartesian coordinate system (X,Y,Z). These values automatically update when you orbit or change the viewpoint in other ways.

**Perspective**
Describes the current perspective setting (PERSPECTIVE system variable). You can turn Perspective on or off from the drop-down list.

**Lens Length**
Describes the current lens length (LENSLENGTH system variable). You can enter a new lens length.

**Field of View**
Describes the current angle for the field of view. You can enter a new angle for the field of view.

**Height (read only)**
Describes the height of the current view.

**Width (read only)**
Describes the width of the current view.

**Clipping**
Describes the clipping mode for the current view. You can specify a new clipping mode from the drop-down list. When clipping is turned on, parts of the view before the front clipping plane and/or behind the back clipping plane are hidden. This property is read only for layouts.

**Front Plane**
Describes the location of the front clipping plane. This property is only available when the Clipping property is turned on. This property is read only for layouts.

### Back Plane

Describes the location of the back clipping plane. This property is only available when the Clipping property is turned on. This property is read only for layouts.

### Visual Style

Describes the current visual style of the drawing. You can specify a new visual style from the drop-down list. This property is read only for layouts.

### Misc Properties

When no entities are selected, Misc properties describe the current annotation scale and default lighting status.

### Annotation Scale

Describes the current annotation scale (CANNOSCALE system variable). You can specify a new annotation scale from the drop-down list.

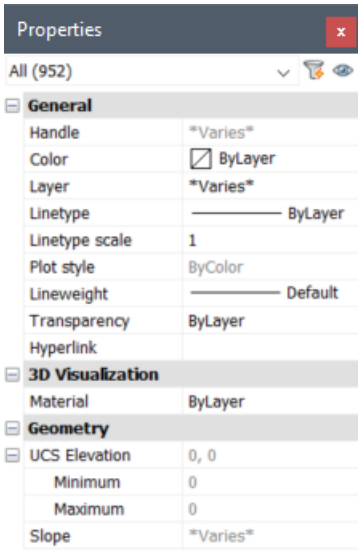
### Default Lighting

Describes the default lighting status (DEFAULTLIGHTING system variable). You can turn Default Lighting on or off from the drop-down list.

### Properties list (current selection set)

The Properties list only displays properties that are valid for all the selected or filtered entities. The more types of entities you select, the fewer properties are available on the Properties list. Filtering the entity list enables you to view and modify properties for multiple entities of the same type with minimal effort. When two or more entities are selected, the properties show their common values. Properties that differ between the selected entities are shown as *varies*.

All entities display General, 3D Visualization (Model space only), and Geometry properties. Additional properties may display for certain types of entities.




### General Properties

In Properties mode, when entities are selected, General properties describe properties that are applied to the selected entities and are valid for almost any type of entity.

### Handle (read only)

Describes the unique hexadecimal value by which the selected entity is identified in the DWG database.

**Color**
Describes the color of the selected entity. You can specify a new color from the drop-down list or choose Select color to access additional colors from the Select Color dialog box.

**Layer**
Describes the layer of the selected entity. You can specify a new layer from the drop-down list.

**Linetype**
Describes the linetype of the selected entity. You can specify a new linetype from the drop-down list or choose Load to access additional linetypes from the Load Linetypes dialog box.

**Linetype Scale**
Describes the linetype scale of the selected entity. You can enter a new linetype scale.

**Plot Style**
Describes the plot style of the selected entity. You can specify a new plotstle from the drop-down list. Note that this property is only available when plot styles are enabled. Also see the PlotStyle command

**Lineweight**
Describes the lineweight of the selected entity. You can specify a new lineweight from the drop-down list.

**Transparency**
Describes the transparency level of the selected entity. You can specify a new transparency level from the drop-down list or enter a value.

**Hyperlink**
Describes the URL of the selected entity. You can enter a new URL or choose the Browse button to access the Edit Hyperlink dialog box.

**History**
Enables you to undo editing operations for a selected entity. You can specify a previous step from the drop-down list to undo the entity back to one of the previous steps without undoing all the other commands and view operations that have been executed since. The History property only displays when the selection set includes only one entity.

**Thickness**
Describes the thickness of the selected entity. You can specify a new thickness by entering a value or selecting the picker icon and picking two points in the drawing.

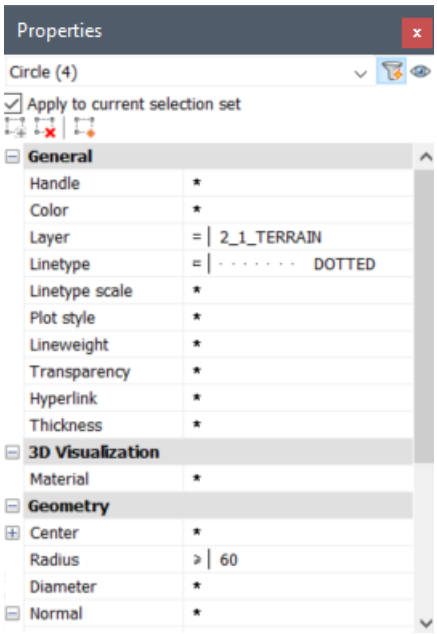
**3D Visualization Properties**
3D Visualization properties describe the materials assigned to the selected entities.

**Material**
Describes the current material of the selected entities. You can specify a new material from the drop-list.

**Geometry Properties**
Geometry properties describe the geometric characteristics of the selected entities. These properties can vary for different types of entities. See Drawing Entities for more information.

### 4.11.21.2 Quick Select mode

In Quick Select mode, the Properties panel enables you to create a selection set based on entity properties. It includes selection tools and a operators.




### Selection tools

Create a selection set based on entity properties.

### Apply to current selection set

This option is only valid if there's an existing selection set. When enabled, the Entities list is limited to the entities in the selection set. When disabled, the Entities list includes all entities in the current viewport or layout.

### Add to current selection set

Adds entities to the current selection set based on the entity types and property values you specify in the Entities and Properties lists. This option is unavailable when the Apply to current selection set is enabled since all possible entities are already included in the current selection set.

### Remove from current selection set

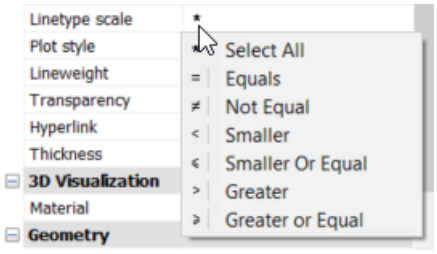
Removes entities from the current selection set based on the entity types and property values you specify in the Entities and Properties lists. This option is unavailable when the Apply to current selection set is disabled since entities can only be removed from an existing selection set.

### Add to new selection set

Creates a new selection set based on the entity types and property values you specify in the Entities and Properties lists.

### Operators

A list of operators enables you to filter properties based on values. You can click on the current operator to display the list and change it to a different operator.



**Select All**
Does not filter the entity list based on the property.

**Equals**
Filters the entity list to include only those entities whose property equals the specified value.

**Not Equal**
Filters the entity list to include only those entities whose property does not equal the specified value.

**Smaller**
Filters the entity list to include only those entities whose property is less than the specified value.

**Smaller Or Equal**
Filters the entity list to include only those entities whose property is less than or equal to the specified value.

**Greater**
Filters the entity list to include only those entities whose property is greater than the specified value.

**Greater or Equal**
Filters the entity list to include only those entities whose property is greater than or equal to the specified value.

**Properties list**
The properties list displays entity properties on which you can filter. The available properties depend on the types of entities in the drawing and whether the option to Apply to current selection set is enabled. General, 3D Visualization (Model space only), and Geometry properties are available for all entities. Additional properties may display for certain types of entities.

**General Properties**
In Quick Select mode, General properties enable you to filter drawing entities based on their general properties.

**Handle**
Enables you to filter entities based on the unique hexadecimal value by which the entity is identified in the DWG database.

**Color**
Enables you to filter entities based on the color of the entity. You can specify a color from the drop-down list or choose Select color to access additional colors from the Select Color dialog box.

**Layer**
Enables you to filter entities based on the layer of the entity. You can specify a layer from the drop-down list.

**Linetype**
Enables you to filter entities based on the linetype of the entity. You can specify a linetype from the drop-down list or choose Load to access additional linetypes from the Load Linetypes dialog box.

**Linetype Scale**
Enables you to filter entities based on the linetype scale of the entity. You can enter a linetype scale.

**Plot Style**
Enables you to filter entities based on the plot style of the entity. You can specify a plotstle from the drop-down list.

***Прим.:*** This property is only available when plot styles are enabled.

**Lineweight**
Enables you to filter entities based on the lineweight of the entity. You can specify a lineweight from the drop-down list.

**Transparency**
Enables you to filter entities based on the transparency level of the entity. You can specify a transparency level from the drop-down list or enter a value.

**Hyperlink**
Enables you to filter entities based on the URL of the entity. You can enter a URL or choose the Browse button to access the Edit Hyperlink dialog box.

**Thickness**
Enables you to filter entities based on the thickness of the entity. You can specify a thickness by entering a value or selecting the picker icon and picking two points in the drawing.

**3D Visualization Properties**
In Quick Select mode, 3D Visualization properties enable you to filter drawing entities based on their 3D visualization properties.

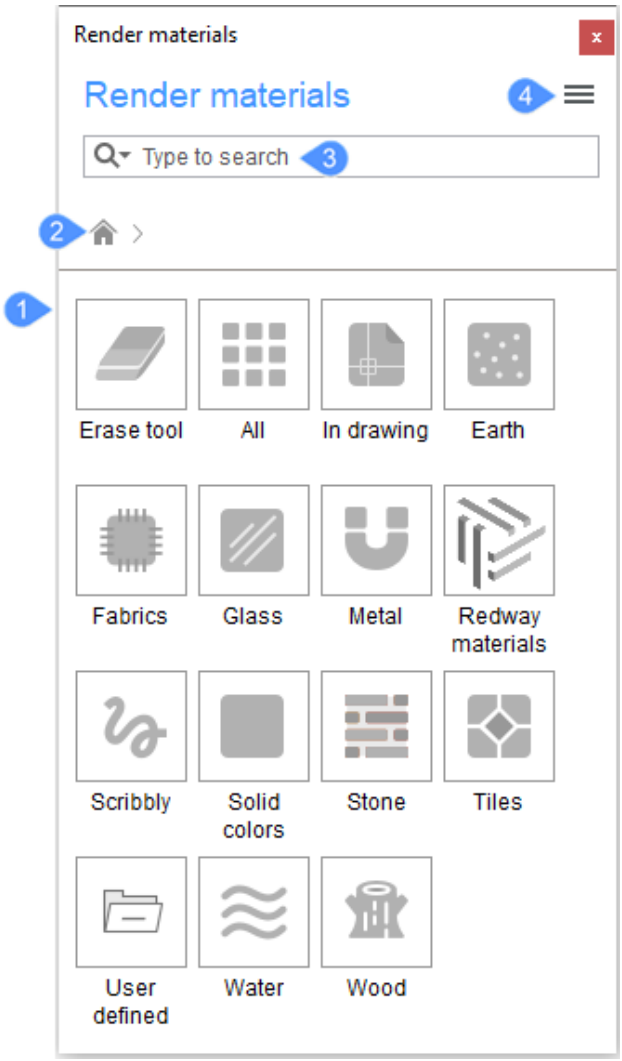
**Material**
Enables you to filter entities based on the material of the entity. You can specify a material from the drop-list.

**Geometry Properties**
In Quick Select mode, Geometry properties enable you to filter drawing entities based on their geometric properties. These properties can vary for different types of entities.

### 4.11.22 Render materials panel

The Render Materials panel offers access to materials used for renderings.

Material definitions are saved in the drawing. Each drawing contains the Global material. It is not possible to delete or rename the Global material, however you can edit the properties of the Global material.




1 Material library

2 Home

3 Search

4 Menu

### 4.11.22.1 Material library

The material library is organized by categories. You can select a material from a category and then apply it to an entity using the brush that appears. You can scroll through the materials using the mouse wheel or the scroll bar.

***Прим.:*** Right-click on a material to add it to library, to remove it from drawing or to add it to drawing.

### 4.11.22.2 Home

Returns the material library to the home screen.

#### 4.11.22.3 Search

Searches the library for words you enter in the search box.

#### 4.11.22.4 Menu

**Grid View**
Displays the materials in a grid view.

**List View**
Displays the materials in a list view.

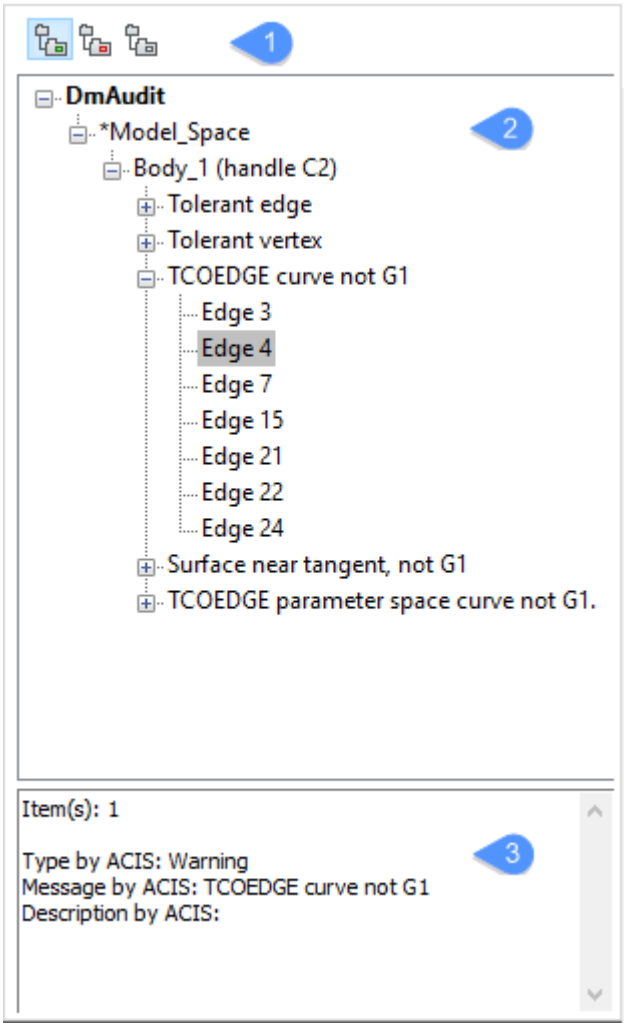
**Manage libraries**
Opens the Settings dialog box to change the render materials directory path.

**Get more materials**
Opens the Render Materials documentation in a browser, where you can download the complete Render Material Library.

### 4.11.23 Report panel

The Report panel offers feedback for relevant commands including DMAUDIT, DMSTITCH, and all sheet metal (SM*) commands.

The appearance of the Report panel is determined by the REPORTPANELMODE system variable.




1 Display grouping

2 Report output

3 Detailed items output

### 4.11.23.1 Display grouping

Specifies how information is grouped for the report.

- Sort by Structure/Message/Subentity
- Group by Message/Structure/Subentity
- Group by Structure/Subentity/Message

### 4.11.23.2 Report output

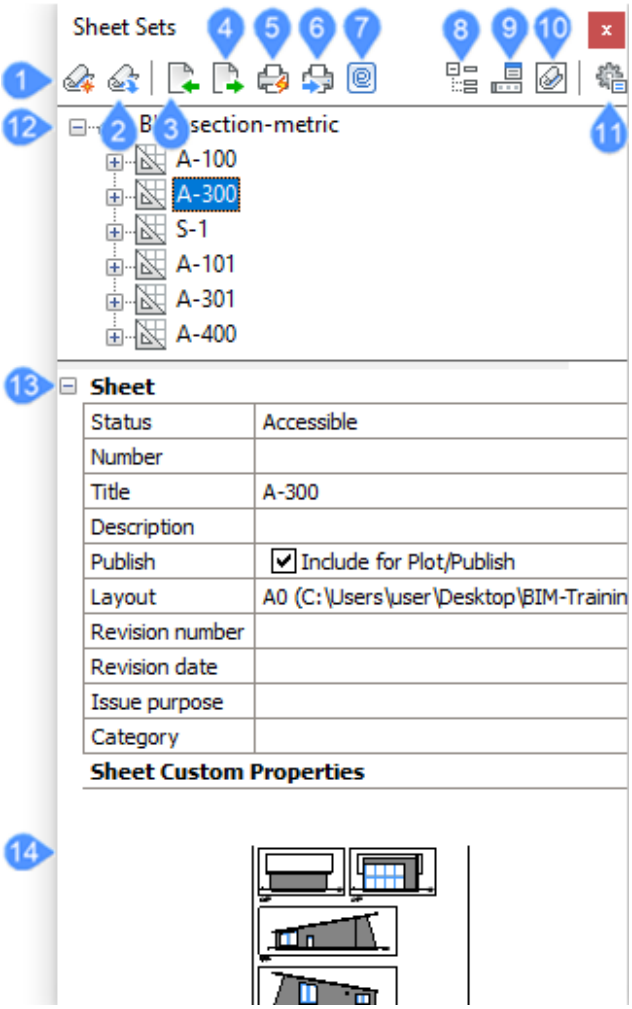
Displays command output.

### 4.11.23.3 Detailed items output

Shows detailed command output for selected item in the report output.

### 4.11.24 Sheet sets panel

The Sheet Sets panel offers a central location to create and manage sheet sets.




1 New sheet set...

2 Open sheet set...

3 Import from XML...

4 Export to XML...

5 Plot

6 Publish...

7 eTransmit...

8 Sheet selections sets...

9 Custom properties...

10 View categories

11 Sheet set options...

12 Sheets

13 Sheet properties

14 Sheet preview

#### 4.11.24.1 New sheet set...

Opens the Create New Sheet Set dialog box.

#### 4.11.24.2 Open sheet set...

Opens the Select a Sheet Set File dialog box.

#### 4.11.24.3 Import from XML...

Allows you to import an XML file that was created in BricsCAD® with **Export to XML...**. Opens the Import Sheet Set dialog box.

#### 4.11.24.4 Export to XML...

Exports a sheet set in XML format. Opens the Export sheet set as dialog box.

#### 4.11.24.5 Plot

Plots the currently selected sheet set(s), subset(s) or sheet(s).

#### 4.11.24.6 Publish...

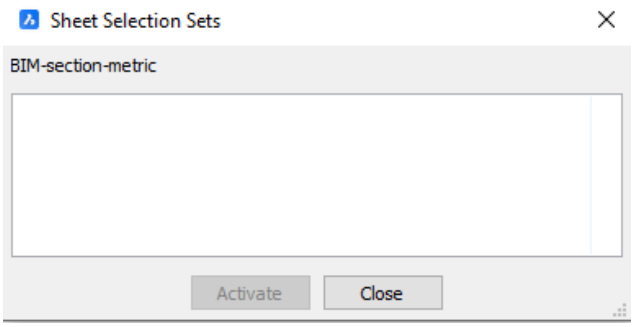
Publishes the currently selected sheet set(s), subset(s) or sheet(s). Opens the Publish dialog box.

#### 4.11.24.7 eTransmit...

Launches the eTransmit command. The currently selected sheet set(s), subset(s) or sheet(s) are included in the eTransmit procedure. Opens the eTransmit dialog box.

#### 4.11.24.8 Sheet selections sets...

Opens the Sheet selection sets dialog box.




#### 4.11.24.9 Custom properties...

Allows you to add custom properties for sheet sets and sheets. Opens the Custom properties dialog box.

#### 4.11.24.1 View categories

**0**

Expands and collapses the currently loaded sheet sets.

#### 4.11.24.1 Sheet set options...

**1**

Opens the Sheet Sets options in the Settings dialog box.

#### 4.11.24.1 Sheets

**2**

Displays the opened and created sheet sets.

Right-clickmenusoffer additional tools:

**Renumber**
Opens the Renumber Sheet Set dialog box, which allows you to renumber all sheets in the sheet set.

**New Sheet**
Displays the New Sheet(s) dialog box, which allows to add new sheets to the current sheet set.

**New Subset**
Adds a new subset to the current sheet set.

**Insert Sheet List Table**
Inserts the sheet list table into the current drawing.

**Open**
Opens the drawing with the layout of the selected sheet.

**Open read-only**
Opens the drawing with the layout of the selected sheet in read-only mode to protect it

***Прим.:*** You can double-click on a sheet set or a subset to expand/collapse. Double-click on a sheet to open it.

#### 4.11.24.1 Sheet properties

**3**

Displays the properties and variables of the selected sheet set, subset or sheet.

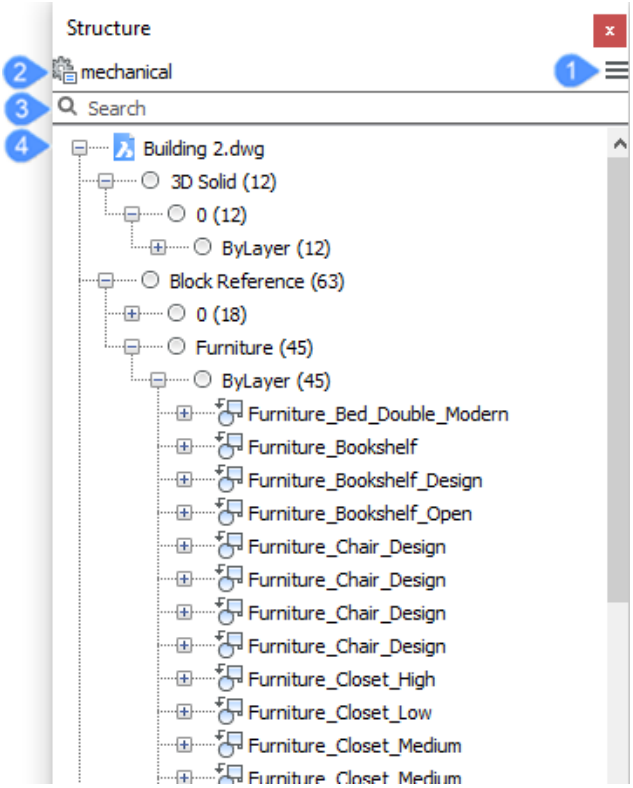
#### 4.11.24.1 Sheet preview

**4**

Displays a thumbnail of the currently selected sheet set without opening it.

### 4.11.25 Structure panel

The Structure panel enables you to view and select drawing content from a structured tree.




1 Menu

2 Configure Structure Tree

3 Search

4 Content tree

### 4.11.25.1 Menu

Displays the Structure panel main menu.

**bim**
Displays a BIM entity-based structure.

**default**
Displays a layer-based structure.

**mechanical**
Displays an entity-based structure.

**Configure structure tree**
Opens the Configure Structure Tree dialog box, which allows you to configure the structure tree.

**Select Configuration…**
Allows you to load Structure Tree Configuration Files (*.cst).

### 4.11.25.2 Configure Structure Tree

Displays the Configure structure tree dialog box.

#### 4.11.25.3 Search

Searches the structure tree for character strings that match what you enter.

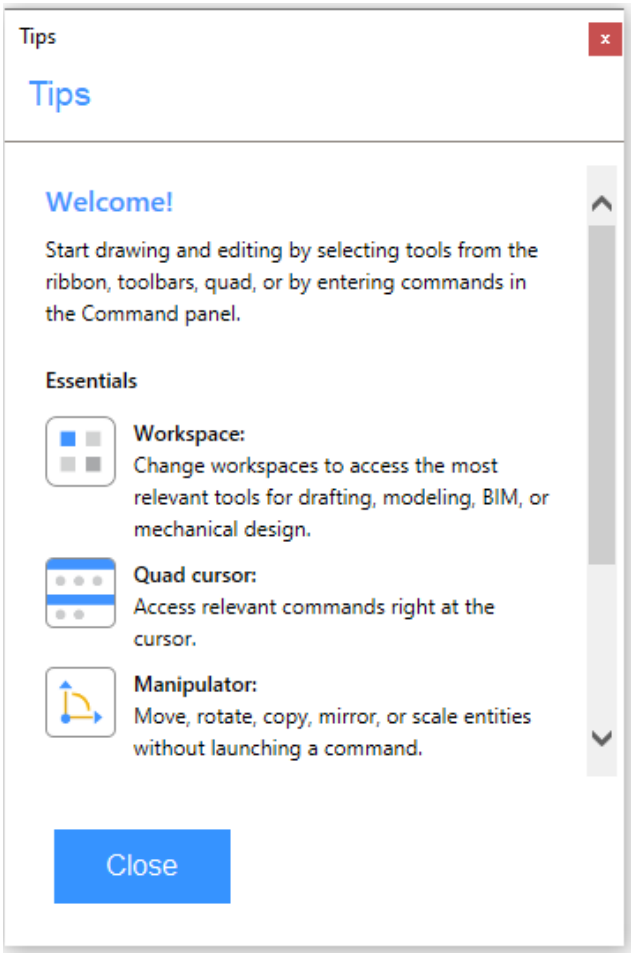
#### 4.11.25.4 Content tree

Displays a structured tree view of the drawing content. The configuration of the structure tree is defined in a .CST file and specified by the STRUCTURETREECONFIG system variable. The location of CST files is specified by the SRCHPATH system variable. You can create custom CST files to meet your needs.

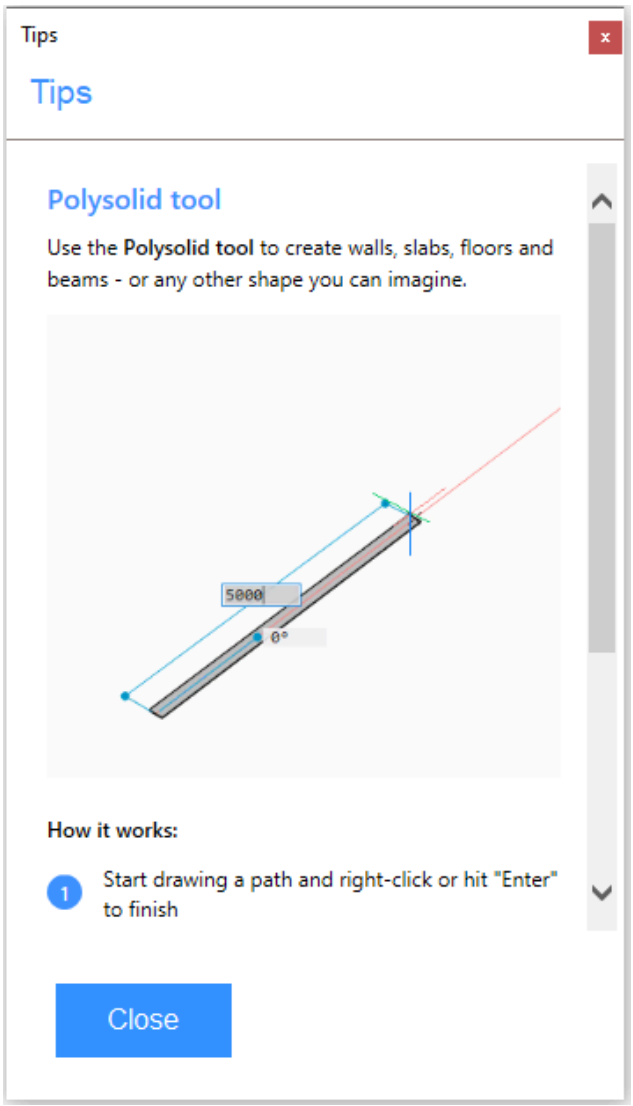
A right-click menu on a tree node or on an entity offers relevant tools.

### 4.11.26 Tips panel

The Tips panel offers relevant advice based on the context in which you are working.




When you start certain commands, the panel displays a short animation and additional information about the command.




### 4.11.27 Tool palettes panel

The Tool palettes panel offers a central location to access blocks, hatches, and command tools.

It includes default tabs with sample tools. You can easily create and customize your own tabs and tools using the right-click menu.

Right-clickmenusoffer additional tools:

**Update tool image:**
Updates tool image for hatches and blocks.

**Specify Image…:**
Allows you to specify another image for the tool.

***Прим.:*** To restore the original tool image, right-click the tool and choose **Remove Image** in the context menu.

**Properties…:**
Opens the Tool Properties dialog box, which allows you to edit a tool.

**View Options…:**
Opens the View options dialog box, which allows you to edit the display of the tool icons.

**Add Text:**
Adds a text separator.

**Add Separator:**
Adds a line separator.

**Customize Palettes…:**
Opens the Customize dialog box, which allows you to customize the palettes. You can import and export palettes or create, import and export palettes groups.

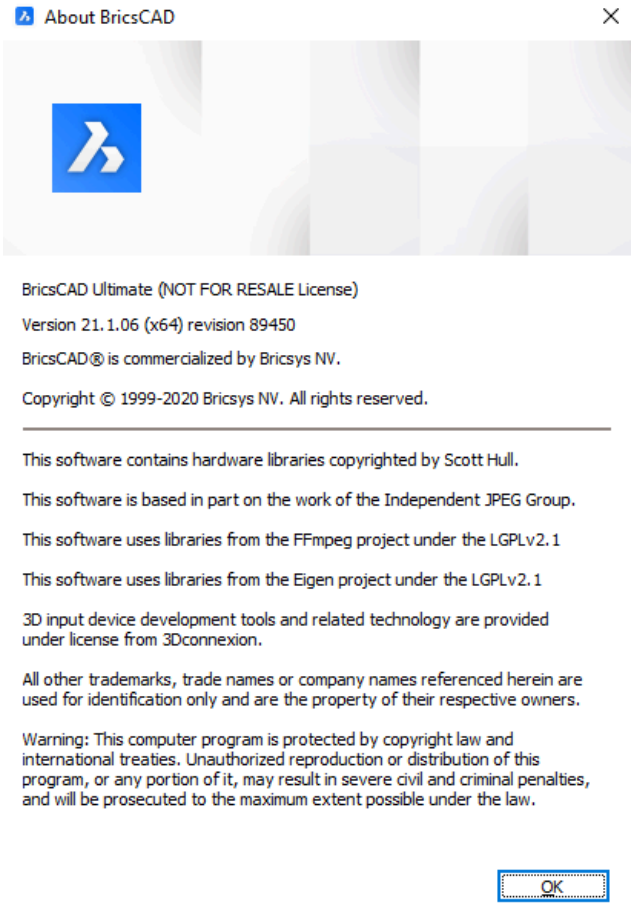
**Add Tool…:**
Opens the Customize dialog box, which allows you to add a new tool.

## 4.12 Dialog boxes

### 4.12.1 About BricsCAD dialog box

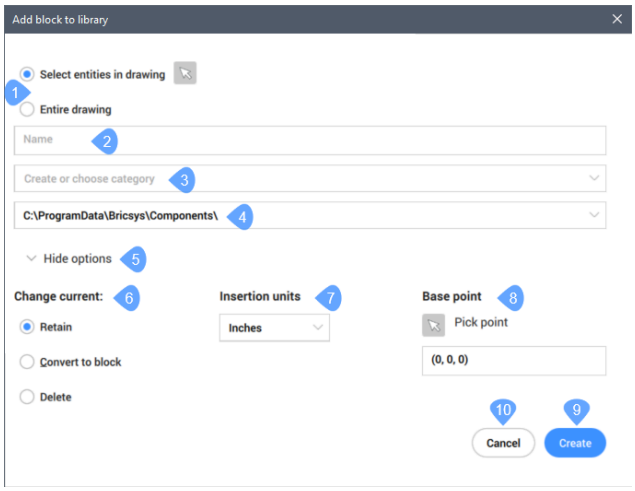
The About BricsCAD dialog box displays copyright and product information.

The product information includes the license type, version and revision number.



### 4.12.2 Add block to library dialog box

The Add block to library dialog box creates a block definition and adds it to the Library.




1 Selection set

2 Name

3 Category

4 Location

5 Options

6 Change current

7 Insertion units

8 Base point

9 Create

10 Cancel

#### 4.12.2.1 Selection set

Specifies which entities are included in the block definition. You can select entities in the drawing or use the entire drawing.

#### 4.12.2.2 Name

Specifies the name of the block.

#### 4.12.2.3 Category

Specifies a category for the block. You can choose from a list of existing categories or enter a new category name.

#### 4.12.2.4 Location

Specifies the location where the block dwg will be saved. Available locations are controlled by the Library directory path setting (COMPONENTSPATH system variable).

#### 4.12.2.5 Show/Hide Options

Shows or hides additional options. The available options depend on the selection set.

#### 4.12.2.6 Change current

Species what happens to selected entities after the block is created. You can choose to retain the individual entities, convert them to a block insertion using the new definition, or delete them. This option is only available if you define the selection set by choosing to Select entities in drawing.

#### 4.12.2.7 Insertion units

Specifies the units for the block insertion. You can choose from the following list of standard units:

- Inches
- Feet
- Miles
- Millimeters
- Centimeters
- Meters
- Kilometers
- Microinches
- Mils
- Yards
- Angstroms
- Microns
- Decimeters
- Dekameters
- Hectometers
- Gigameters
- Astronomical Units
- Light-years
- Parsecs
- US Survey Feet
- Us Survey Inch
- US Survey Yard
- US Survey Mile
- Base point

Specifies the base point for block definition. You can pick a point in the drawing or enter X,Y,Z coordinates. This option is only available if you define the selection set by choosing to Select entities in drawing.

#### 4.12.2.8 Create

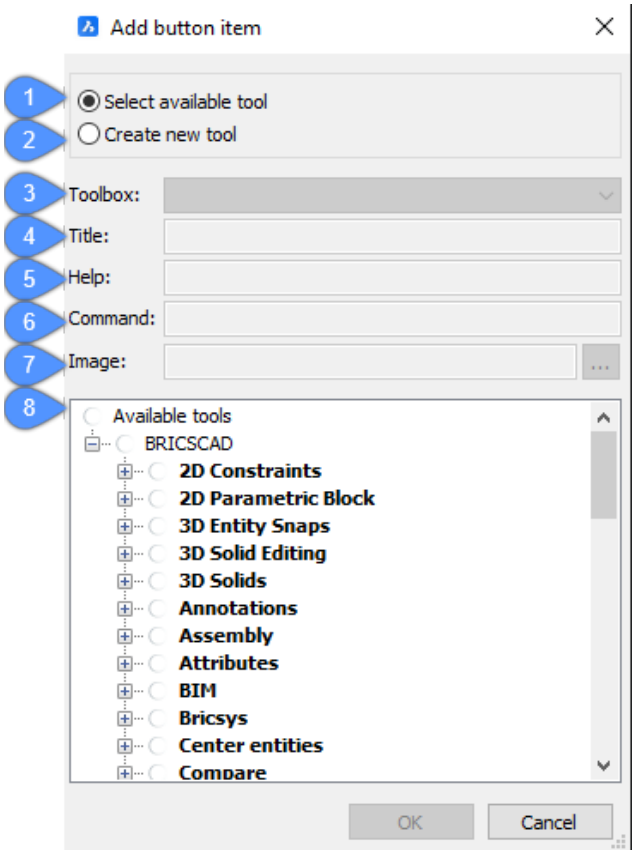
Creates a DWG file for the block definition in the specified location, adds the block to the library panel under the specified category, and closes the dialog box.

#### 4.12.2.9 Cancel

Closes the dialog box without creating the block definition.

### 4.12.3 Add button item dialog box

The Add button item dialog box allows you to add a button item by selecting an available tool or by creating a new tool.




1 Select available tool

2 Create new tool

3 Toolbox

4 Title

5 Help

6 Command

7 Image

8 Available tools

#### 4.12.3.1 Select available tool

Assigns an existing command to thenewbutton or button group. If this option is selected, the other options are grayed out, except for Available tools.

#### 4.12.3.2 Create new tool

Assigns a new command/macro to the newbutton or button group. If this option is selected, Available toolsisgrayed out.

#### 4.12.3.3 Toolbox

Specifies the toolbox to which to add the new command. You can choose a toolboxinthedrop-downlist.

#### 4.12.3.4 Title

Specifies the name of the new tool.

#### 4.12.3.5 Help

Specifies the help string displayed on the status bar.

#### 4.12.3.6 Command

Specifies the command(s) or macros.

#### 4.12.3.7 Image

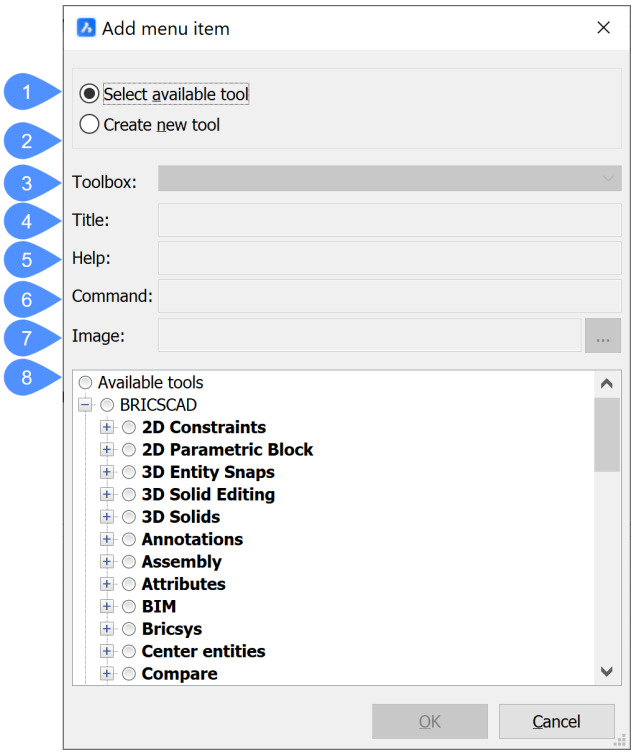
Specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image.

#### 4.12.3.8 Available tools

Selects an existing tool.

### 4.12.4 Add menu item dialog box

The Add button item dialog box allows you to add a menu item by selecting an available tool or by creating a new tool.




1 Select available tool

2 Create new tool

3 Toolbox

4 Title

5 Help

6 Command

7 Image

8 Available tools

#### 4.12.4.1 Select available tool

Assigns an existing command to thenewbutton or button group. If this option is selected, the other options are grayed out, except for Available tools.

#### 4.12.4.2 Create new tool

Assigns a new command/macro to the newbutton or button group. If this option is selected, Available toolsisgrayed out.

#### 4.12.4.3 Toolbox

Specifies the toolbox to which to add the new command. You can choose a toolboxinthedrop-downlist.

#### 4.12.4.4 Title

Specifies the name of the new tool.

#### 4.12.4.5 Help

Specifies the help string displayed on the status bar.

#### 4.12.4.6 Command

Specifies the command(s) or macros.

#### 4.12.4.7 Image

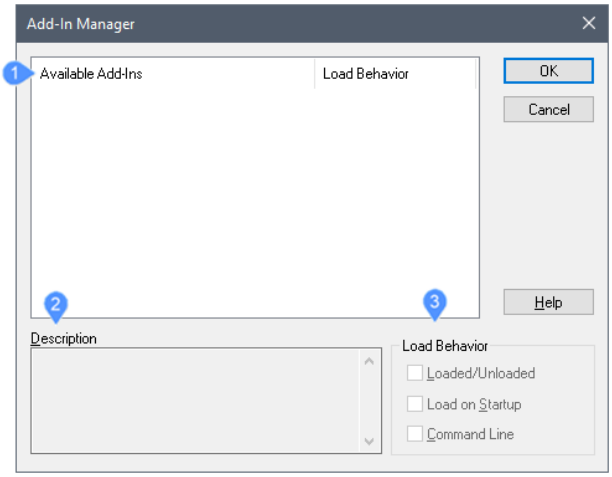
Specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image.

#### 4.12.4.8 Available tools

Selects an existing tool.

### 4.12.5 Add-in manager dialog box

The Add-In Manager dialog box controls how Visual Basic for Applications COM (Common Object Model) modules are loaded. You can either toggle the modules between loaded and unloaded or start modules when BricsCAD starts up.




1 Available Add-ins

2 Description

3 Load Behavior

#### 4.12.5.1 Available Add-ins

Lists the names and status of VBA COM modules available to BricsCAD. "VBA COM" is short for Visual Basic for Applications Common Object Model.

#### 4.12.5.2 Description

Describes the selected VBA COM module, if any.

#### 4.12.5.3 Load Behavior

Determines how modules are loaded.

**Loaded/Unloaded**
Toggles the module between loaded and unloaded.

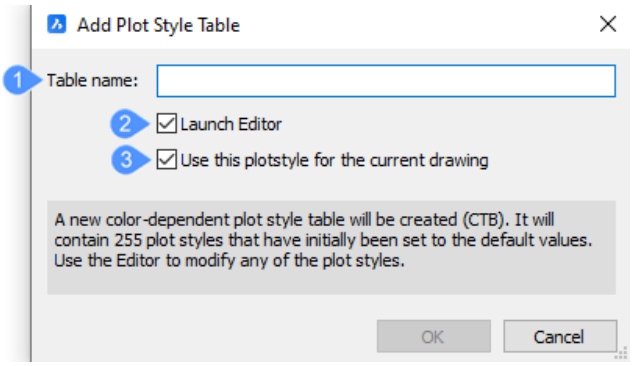
**Load on Startup**
Loads modules when BricsCAD starts up.

**Command line**
Loads modules through the Command line.

### 4.12.6 Add plot style table dialog box

The Add plot style table dialog box allows you to create new plot styles.

If the drawing uses named plot styles, the wizard creates a new named plot style table (STB file). If the current drawing does not use named plot styles, the wizard creates a new color-dependent table (CTB file).




1 Table name

2 Launch Editor

3 Use this Plotstyle for the current drawing

#### 4.12.6.1 Table name

Enter the name you want to give to the plot style table.

#### 4.12.6.2 Launch Editor

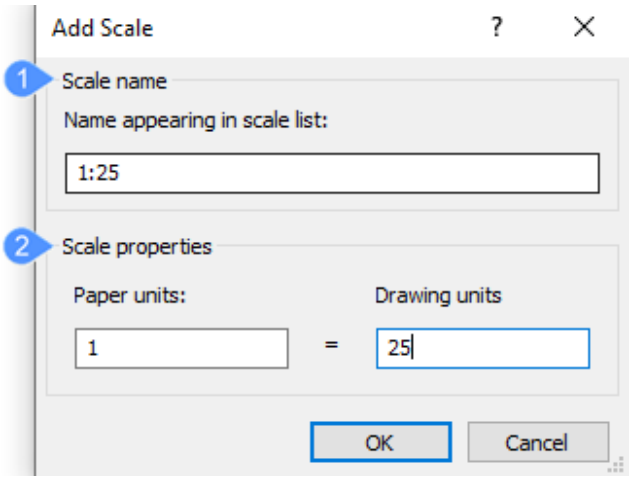
Opens the Plot style table editor dialog box after you enter the name and press the OK button.

#### 4.12.6.3 Use this Plotstyle for the current drawing

If you do not check the Launch Editor option, it saves and sets the table name you have entered as current plot style table in the current layout.

### 4.12.7 Add scale dialog box

The Add Scale dialog box allows you to add a new scale factor to the list.




1 Scale name

2 Scale properties

### 4.12.7.1 Scale name

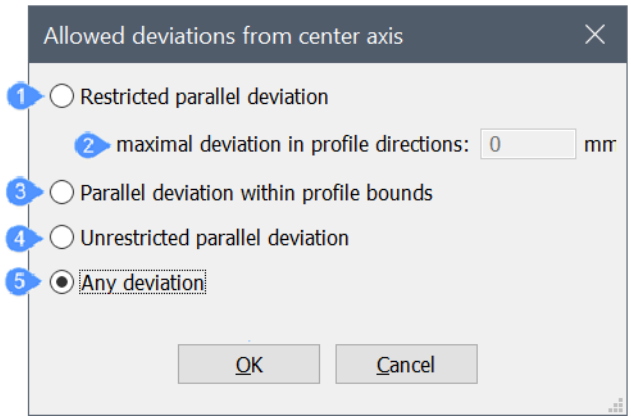
Specifies the scale name as it will appear in the scale list.

### 4.12.7.2 Scale properties

Specifies the scale factor in paper (size when plotted) and drawing units.

## 4.12.8 Allowed deviations from center axis dialog box

The Allowed Deviation from Center Axis dialog box allows you to set the level of freedom used when generating an analytical model.




1 Restricted parallel deviation

2 Maximal deviation in profile directions

3 Parallel deviation within bounds

4 Unrestricted parallel deviation

5 Any deviation

#### 4.12.8.1 Restricted parallel deviation

Axes remain parallel to center axis and within the specified distance from the center axis in both profile directions.

#### 4.12.8.2 Maximal deviation in profile directions

Set the maximal deviation in profile directions.

#### 4.12.8.3 Parallel deviation within bounds

Axis remain parallel to center axis and do not exceed profile bounds.

#### 4.12.8.4 Unrestricted parallel deviation

Axes remain parallel to center axis, but can be placed outside the profile bounds if is useful to avoid node splits or rigid links.

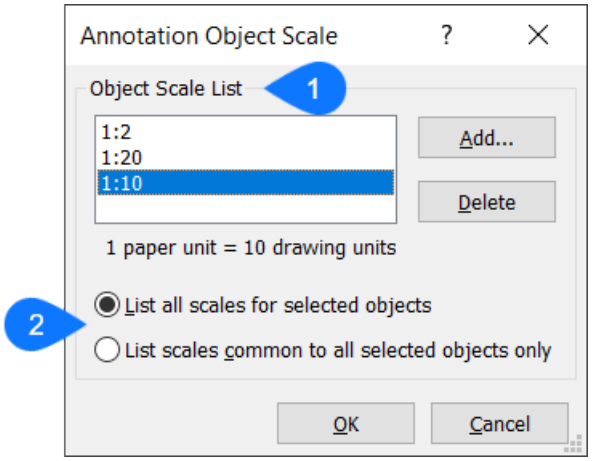
#### 4.12.8.5 Any deviation

The main axes remain parallel to the center axis, but tilted axis (with respect to vertical and main horizontal directions) can be allowed to deviate from parallel if useful to avoid node splits and rigid links. Other axes can move outside of profile bounds.

Click **OK** to open the Visual settings dialog box.

### 4.12.9 Annotation Object Scale dialog box

The Annotation Object Scale dialog box allows you to add or remove scale factors used with annotative entities.




1 Object Scale List

2 Options

#### 4.12.9.1 Object Scale List

Shows all the scales supported by the selected annotative objects.

**Add**

Adds annotation scale factors to the selected entities.

**Delete**
Removes annotation scales from the selected entities. Choose **No** in the **Annotative** field on the Properties Bar to remove the annotative property.

***Прим.:*** This option does not work if there is just one scale factor attached to the entities.

### 4.12.9.2 Options

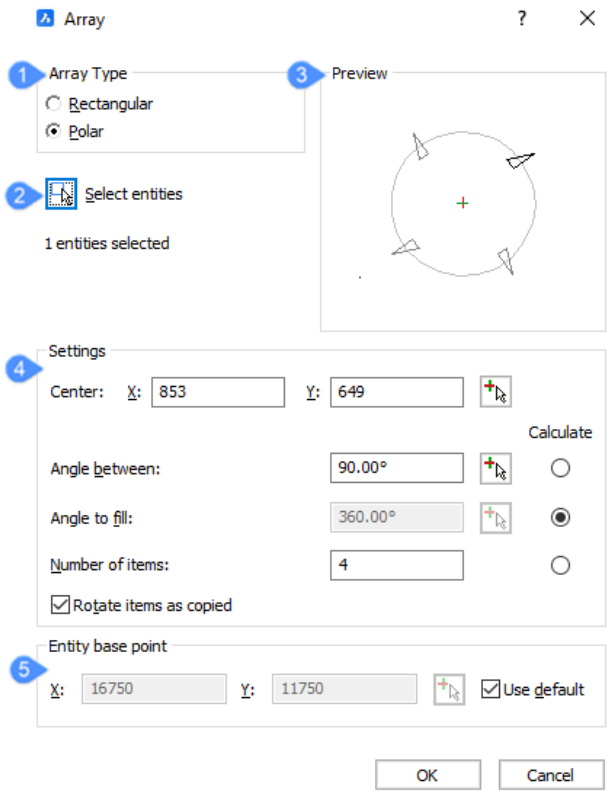
**List all scales for selected objects**
Lists all the scale factors.

**List scales common to all selected objects only**
Lists only the scale factor shared by the selected entities.

***Прим.:*** This option is meaningless when only one entity is selected.

## 4.12.10 Array dialog box

The Array dialog box allows you to create a 2D rectangular or polar array of entities.




1 Array Type
2 Select entities
3 Preview
4 Settings
5 Entity base point (Polar only)

### 4.12.10.1 Array Type

Specifies whether a rectangular or polar array will be created.

### 4.12.10.2 Select entities

Temporarily closes the Array dialog box and prompts you to select entities. Press Enter to complete your select and return to the Array dialog box.

### 4.12.10.3 Preview

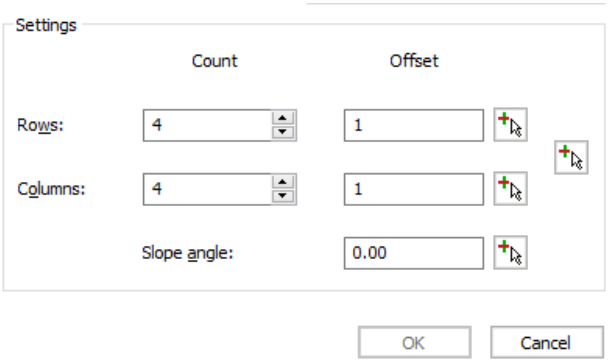
Displays a preview of the array based on the specified settings.

### 4.12.10.4 Settings

Specifies the number and location of items in the array. The available settings differ based on the array type.

**Rectangular**

Copies the selected entities in a rectangular pattern.




**Count**

Specifies the number of rows and columns in the array.

**Offset**

Specifies the distance between rows and columns. You can type a distance in the Offset fields or click the buttons to pick the offset distance in the drawing.

**Offset field**

Enables you to type distances for the offsets. Positive distances add rows along the positive Y axis and add columns along the positive X axis. Negative distances add rows along the negative Y axis and add columns along the negative X axis.

**Offset buttons**

Enables you to specify the offset distance by picking two points in the drawing. Clicking one of these buttons temporarily closes the Array dialog box and prompts you to pick points.

**Pick Row Offset**

Specifies the distance between rows. A temporary line displays between the two points you pick. The length of the line determines the offset distance. The angle of the line determines if rows are added along the positive or negative Y axis.

### Pick Column Offset

Specifies the distance between columns. A temporary line displays between the two points you pick. The length of the line determines the offset distance. The angle of the line determines if columns are added along the positive or negative X axis.

### Pick Both Offsets

Specifies the distance between rows and the distance between columns. A temporary rectangle displays between the two points you pick. The length of the rectangle determines the distance between columns and the width of rectangle determines the distance between rows.

### Slope Angle

Specifies the angle of the array from the X axis. You can type an angle in the Angle field or click the button to pick an angle in the drawing.

### Angle field

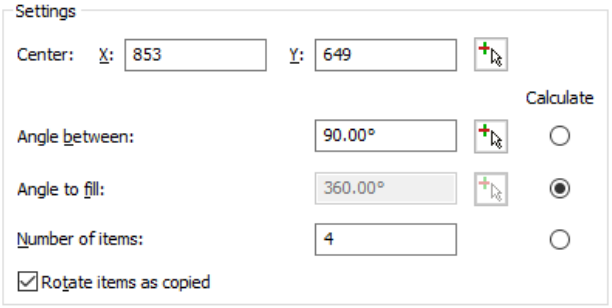
Enables you to type an angle for the slope.

### Pick Angle of Array button

Enables you to specify the angle of the array by picking two points in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick points.

### Polar

Copies the selected entities in a circular pattern.




### Center

Specifies the center point of the polar array. You can type coordinates in the X and Y fields or click the button to pick the center in the drawing.

### X field

Enables you to type a coordinate to place the center of the array along the X axis.

### Y field

Enables you to type a coordinate to place the center of the array along the Y axis.

### Pick Center Point button

Enables you to specify the center of the array by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point.

### Angle between

Specifies the angle between each item in the polar array. You can type an angle in the field, click the button to pick an angle in the drawing, or choose the Calculate option.

### Angle field

Enables you to type a value to specify the angle between each item of the array.

**Pick Angle Between Items button**
Enables you to specify the angle between each time by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point. A temporary line displays between the center point of the array and the point you pick. The angle of the line determines the angle between items.

**Calculate**
Enables you to specify the angle between each item based on the Angle to fill and number of items. Choosing the Calculate radio button disables the Angle between options and enables the Angle to fill options.

**Angle to fill**
Specifies the fill angle and direction for a polar array. You can type an angle in the field, click the button to pick an angle in the drawing, or choose the Calculate option.

**Angle field**
Enables you to type a value to specify the fill angle of the array. Enter 360 for a full circle of copies. Enter a smaller number to draw a partial polar array. Positive values draw the array counterclockwise and negative values draw the array clockwise.

**Pick Angle to Fill button**
Enables you to specify the fill angle of the array by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point. A temporary line displays between the center point of the array and the point you pick. The angle of the line determines the fill angle.

**Calculate**
Enables you to specify the fill angle based on the angle between items and the number of items. Choosing the Calculate radio button disables the Angle to fill options and enables the Angle between options.

**Number of items**
Specifies the number of items in the polar array. You can type a number in the field or choose the Calculate option.

**Number field**
Enables you to enter the number of items to include in the array.

**Calculate**
Enables you to specify the number of items based on the angle between items and the fill angle. Choosing the Calculate radio button disables the Number of items options and enables the Angle Between and Angle to Fill options.

**Rotate items as copied**
Specifies whether array items are rotated when they are copied to form the polar array.

- No: does not rotate items; all copies have the same orientation as the original item.
- Yes: rotates copies.

### 4.12.10.5 Entity base point (Polar only)

Specifies the base point of items in a polar array.



**X field**
Enables you to type a coordinate to specify the basepoint of array items along the X axis.

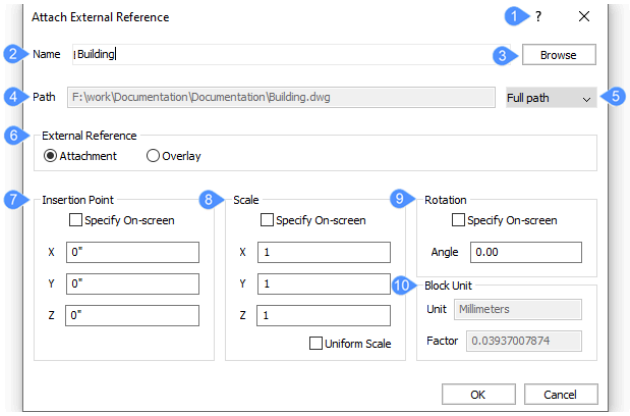
**Y field**
Enables you to type a coordinate to specify the basepoint of array items along the Y axis.

**Pick base point button**
Enables you to specify the base point of array items by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point.

**Use Default**
Uses the default basepoint for array entities. Choosing the Use default option disables the other entity base point options.

### 4.12.11 Attach external reference

The Attach External Reference dialog box allows you to attach externally - referenced drawings to the current drawing.




1 Help
2 Name
3 Browse
4 Path
5 Path type
6 External reference
7 Insertion Point
8 Scale
9 Rotation
10 Block Unit

#### 4.12.11.1 Help

Opens the Bricsys Help article about the XATTACH command.

#### 4.12.11.2 Name

Specifies the name of the DWG file attached to the current drawing.

#### 4.12.11.3 Browse

Opens the Select Reference File dialog box to allow you to choose a different *.dwg file

#### 4.12.11.4 Path

Displays the path of the drawing file.

#### 4.12.11.5 Path type

Determines how much of the path is stored:

**Full path**
Stores the drive and folder names of the DWG file as an absolute reference, such as *c:\cad\dwg \filename.dwg*

**Relative path**
Stores the path from the parent drawing's location to the referenced drawing's location, such as *.. \filename.dwg*. The .. refers to the folder above the current one. The drawing must be saved before you can use this option.

**No path**
Strips the drive and folder names, leaving just the DWG file name, such as *filename.dwg*.

#### 4.12.11.6 External reference

Specifies how the external reference is to be attached.

**Attachment**
Attaches the xref and nested xrefs.

**Overlay**
Attaches only the first level of xref.

#### 4.12.11.7 Insertion Point

Specifies the location of the xref's lower-left corner:

**Specify On-screen**
Determines how you specify the insertion point.

- On: specify the insertion point in the drawing after the dialog box is closed.
- Off: specify the insertion point in the dialog box using the X, Y, and Z fields.

**X, Y, or Z**
Specifies the x, y, and/or z coordinates for the xref's insertion point. Use 0,0,0 to insert the xref at the drawing's origin.

#### 4.12.11.8 Scale

Specifies the size of the xref.

**Specify On-screen**
Determines how you specify the scale factors.

- On: specify the scale factors in the drawing after the dialog box is closed.
- Off: specify the scale factors in the dialog box using the X, Y, and Z fields.

**X, Y, or Z**
Specifies the x, y, and/or z scale factors of the xref in the x, y, and z directions. Use 1,1,1 to keep the xref at its original size.

**Uniform Scale**
Makes the y and z scale factors equal to x.

#### 4.12.11.9 Rotation

Specifies the rotation angle of the xref.

**Specify On-screen**
Determines how you specify the rotation angle.

- On: specify the angle in the drawing after the dialog box is closed.
- Off: specify the angle in the dialog box using the Angle field.

**Angle**
Specifies the rotation angle about the insertion point. Positive angles rotate the xref counterclockwise. Negative angles rotate the xref clockwise. Use 0 to keep the xref at its original orientation.

#### 4.12.11.1 Block Unit

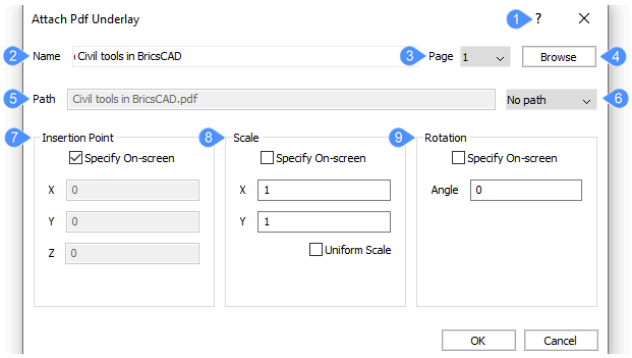
**0**
Specifies the units with which the drawing will be inserted, usually a form of Imperial or metric units, such as inches or mm:

**Unit**
Reports the INSUNITS setting of the drawing being attached.

**Factor**
Reports the calculated scaling factor with respect to the INSUNITS setting of the drawing being attached and the current drawing.

### 4.12.12 Attach PDF underlay dialog box

The Attach PDF underlay dialog box allows you to attach a PDF file as an underlay into the current drawing.




1 Help

2 Name

3 Page

4 Browse

5 Path

6 Path type

7 Insertion Point

8 Scale

9 Rotation

### 4.12.12.1 Help

Opens the Bricsys Help article about the -PDFATTACH command.

### 4.12.12.2 Name

Displays the file name of the PDF file to attach. When more than one PDF file has been attached to the drawing, their names are listed in the drop-down list.

### 4.12.12.3 Page

Allows you to display the page you want from the whole PDF file. You can change the number of the page from the drop-down arrow list.

### 4.12.12.4 Browse

Opens the Select PDF underlay file dialog box from where you can select another PDF file than the one already selected.

### 4.12.12.5 Path

Displays the path of the file according to the path type you choose.

### 4.12.12.6 Path type

Choose the way the program remembers the path to the PDF file. You can select from the Path drop-down list one of the following options:

**Full path**
Stores the drive and the folder(s) name to the PDF file as an absolute reference such as: C:\Users\user \PDFATTACH\filename.pdf

**Relative path**
Stores the part of the path to PDF file's folder, such as .\filename.pdf. This option does not work until the drawing has been saved at least once. (..\ means one folder higher)

**No path**
Strips the drive and folder names, leaving just the PDF file name, such as filename.pdf.

### 4.12.12.7 Insertion Point

Specifies the location of the underlay's lower-left corner.

**Specify On-screen**
Determines how the insertion point is specified.

- On: specify the insertion point in the drawing after the dialog box is closed.
- Off: specify the insertion point in the dialog box, using the X, Y and Z fields.

**X, Y, Z**
Specifies the x, y, z coordinates for the PDF underlay's insertion point. Use 0,0,0 to insert the underlay at the drawing's origin.

### 4.12.12.8 Scale

Specifies the size of the PDF underlay:

**Specify On-screen**
Determines how the scale factors are specified:

- On: specify the scale factor in the drawing after the dialog box is closed.
- Off: specify the scale factor in the dialog box using the X and Y fields.

**X,Y**
Specifies the scale factors along the X and Y axes.

**Uniform scale**
Makes the Y scale factor equal to X.

### 4.12.12.9 Rotation

Specifies the rotation angle of the PDF underlay:

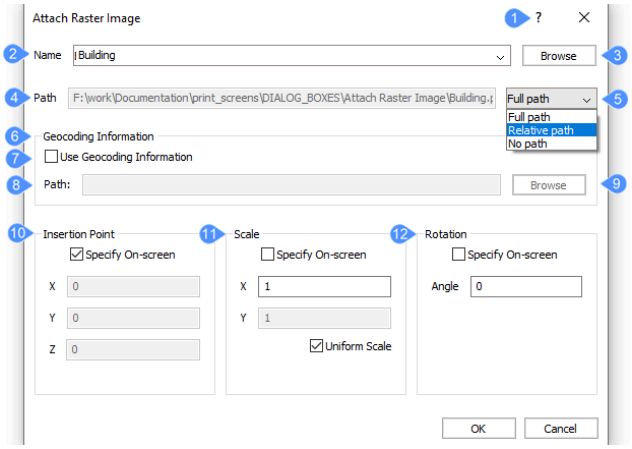
**Specify On-screen**
Determines how the rotation angle is specified:

- On: specify the rotation angle in the drawing after the dialog box is closed.
- Off: specify the rotation angle in the dialog in Angle field.

**Angle**
Specifies the rotation angle about the insertion point. Positive angles rotate the underlay counterclockwise. Negative angles rotate the image clockwise. Use 0 to keep the underlay at its original orientation.

### 4.12.13 Attach raster image dialog box

The Attach image raster dialog box allows you to attach raster images to the drawing.




1 Help

2 Name

3 Browse

4 Path

5 Path type

6 Geocoding Information

7 Use Geocoding Information

8 Path

9 Browse (image position file)

10 Insertion Point

11 Scale

12 Rotation

#### 4.12.13.1 Help

Opens the Bricsys Help article about the IMAGEATTACH command.

#### 4.12.13.2 Name

Specifies the name of the file to attach.

#### 4.12.13.3 Browse

Opens the Select Image File dialog box to allow you to choose a different raster file.

#### 4.12.13.4 Path

Displays the path of the image file.

### 4.12.13.5 Path type

Specifies how much of the path is stored.

**Full path**
Stores the full path to the image file as an absolute reference such as: *C:\foldername\imagename.jpg*

**Relative path**
Stores the path from the drawing's location to the image's location. For example: *..\foldername \imagename.jpg*. The .. refers to the folder above the current one. The drawing must be saved before you can use this option.

**No path**
Strips the drive and folder names, leaving just the image file name such as *imagename.jpg*.

### 4.12.13.6 Geocoding Information

Position files or world files have an extension that depends on what kind of image file they are associated with.

**Use Geocoding Information**
Toggles use of a position file, which specifies the size, location and rotation of the image file.

- Yes: uses a position file
- No: does not use a position file.

**Path**
Displays the path of the position file.

**Browse (image position file)**
Opens the Select Image Position File dialog box.

### 4.12.13.7 Insertion Point

Specifies the location of the image's lower left corner.

**Specify On-screen**
Determines how you specify the insertion point.

- On: specify the insertion point in the drawing after the dialog box is closed.
- Off: specify the insertion point in the dialog box using the X, Y and Z fields.

**X,Y,Z**
Specifies the x, y, and z coordinates for the image's insertion point.

### 4.12.13.8 Scale

Specifies the size of the image.

**Specify On-screen**
Determines how you specify the scale factor.

- On: specify the scale factors in the drawing after the dialog box is closed.
- Off: specify the scale factors in the dialog box using the X and Y fields.

**XY**
Specifies the scale factors along the X and Y axes.

**Uniform scale**
Makes the Y scale factor equal to X.

### 4.12.13.9 Rotation

Specifies the rotation angle of the image.

**Specify On-screen**
Determines how you specify the rotation angle.

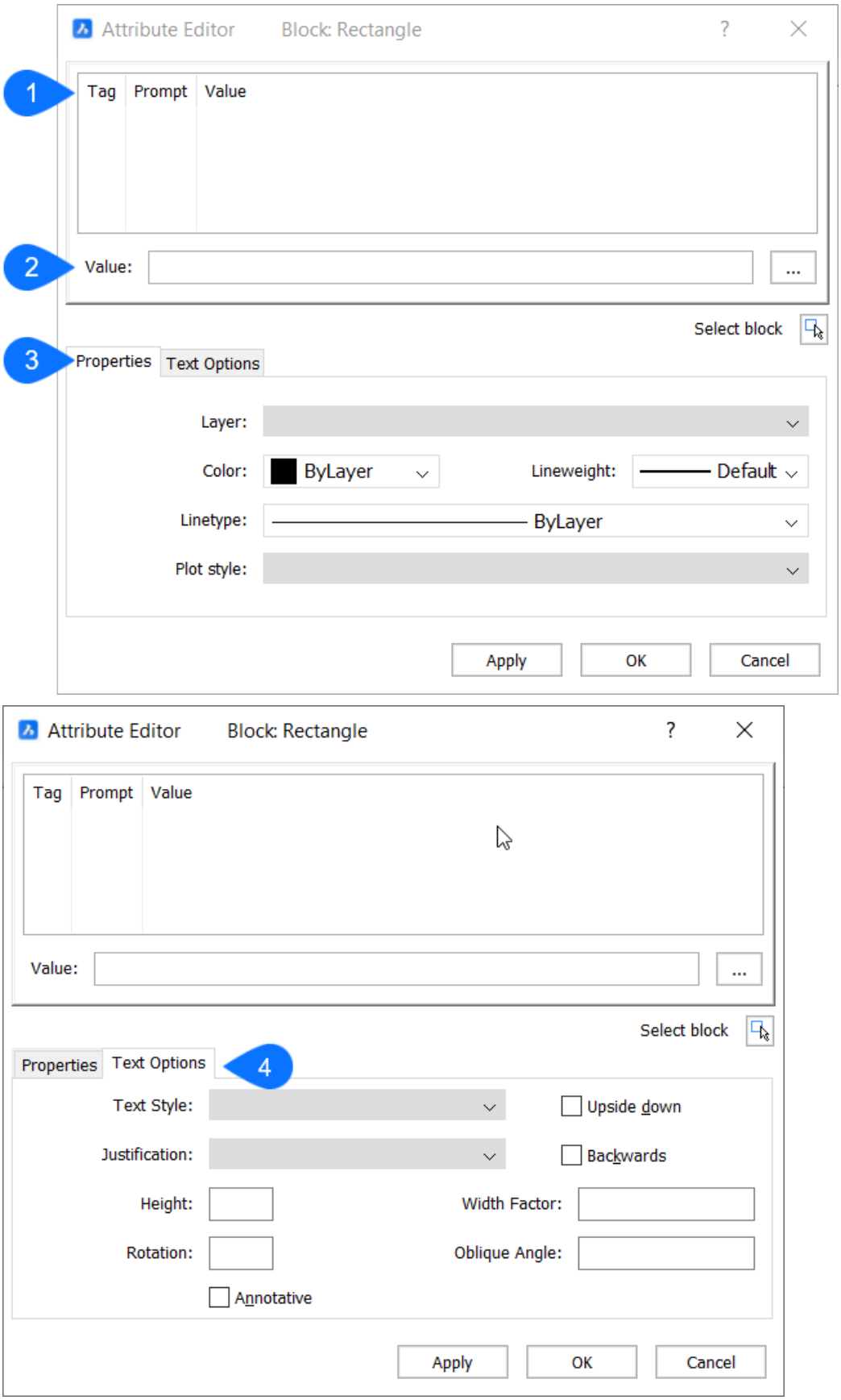
- On: specify the rotation angle in the drawing after the dialog box is closed.
- Off: specify the angle in the dialog box using the Angle field.

### 4.12.13.1 Angle

**0**
Specifies the rotation angle of the image about the insertion point. Positive angles rotate the image counterclockwise. Negative angles rotate the image clockwise. Use 0 to keep the image at its original orientation.

## 4.12.14 Attribute editor dialog box

The attribute Editor dialog box allows you to edit attributes within blocks.




1 Overview list

2 Value

3 Properties

4 Text Options

#### 4.12.14.1 Overview list

Displays an overview of all attributes used in the block.

#### 4.12.14.2 Value

Displays the currently selected attribute from the list. A new value can be typed.

#### 4.12.14.3 Properties

Specifies properties of the selected attribute.

**Layer**
Specifies the layer of the selected attribute.

**Color**
Specifies the color of the selected attribute.

**Linetype**
Specifies the linetype of the selected attribute.

**Plot style**
Specifies the plotstyle of the selected attribute.

#### 4.12.14.4 Text Options

**Text style**
Specifies the font of the text in the selected attribute.

**Justification**
Specifies the alignment of the text in the selected attribute.

**Upside down**
When checked, text is displayed upside down.

**Backwards**
When checked, text is displayed backwards.

**Height**
Specifies the height of the text in the selected attribute.

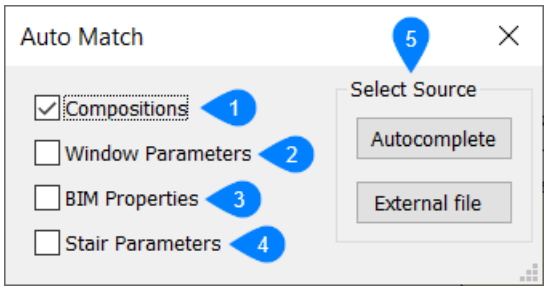
**Rotation**
Specifies the rotation angle of the text in the selected attribute.

**Width factor**
Specifies the width of the text in the selected attribute.

**Oblique angle**
Specifies the oblique angle of the text in the selected attribute.

### 4.12.15 Auto Match dialog box

The Auto Match dialog box defines which characteristics of example entities are matched and lets you select a source.




1 Compositions

2 Window Parameters

3 BIM Properties

4 Stair Parameters

5 Select Source

### 4.12.15.1 Compositions

When checked on, the BIM properties of entities will be copied.

Entities without a composition will be matched with the other compositions in your drawing, based on type, spatial location and orientation of the entity, as well as on the 'Is external'-property.

If all entities have compositions, you can overwrite some by choosing a source entity. AutoMatch will then copy this source's composition to the rest of the valid entities.

### 4.12.15.2 Window Parameters

When checked on, the parameters of the windows will be copied.

If there are unchanged windows in your drawing, AutoMatch will match the properties based on width, height, type, spatial location and orientation of the windows and type and composition of the entities they are in.

If all windows have changed parameters, you can overwrite some by choosing a source window. AutoMatch will then copy this source's parameters to the rest of the valid windows.

***Прим.:*** Some window parameters will never be copied along.

- W (= width of the window)
- H (= height of the window)

### 4.12.15.3 BIM Properties

When checked on, the BIM properties of entities will be copied.

If there are entities with unchanged properties in your drawing, AutoMatch will match the properties based on BIM type, composition and spatial location of the other entities in your drawing.

If all entities have a changed property, you can overwrite it by choosing a source entity. AutoMatch will then copy this source's property to the rest of the valid entities.

Some general BIM properties will never be copied along:

- GUID

- Composition
- Profile
- SpatialLocation
- IsExternal

Some beam and column related BIM properties will never be copied along:

- eomHash
- athString
- XAxis
- YAxis
- ZAxis
- ProfileName
- EndPoint
- StartPoint
- Valid
- InstancePropertySets

Some window and door related BIM properties will never be copied along:

- Name
- SubtractAllSolids
- Translucency
- OverallWidth
- OverallHeight

Some slab related BIM properties will never be copied along:

- RoomBounding

Some room related BIM properties will never be copied along:

- Number
- RoomArea
- RoomRepresentation

Some stair related BIM properties will never be copied along:

- StairType
- NumberOfRisers
- NumberOfTreads

#### 4.12.15.4 Stair Parameters

When checked on, the parameters of a stair will be copied.

If there are unchanged stairs in your drawing, AutoMatch will match the properties based on number of treads, number of risers, height, width, length and type of the stair.

If all stairs have changed parameters, you can overwrite some by choosing a source stair. AutoMatch will then copy this source's parameters to the rest of the valid stairs.

***Прим.:*** Some stair parameters will never be copied along:

- Height_Staircase
- Stair_Width
- Length_Staircase

#### 4.12.15.5 Select Source

**Autocomplete**
Matches all chosen options for entities in the open drawing with similar entities in the open drawing.

**External file**
Opens the Select file to insert dialog box. Here you can choose a file to match all chosen options for entities in the chosen drawing with similar entities in the open drawing.

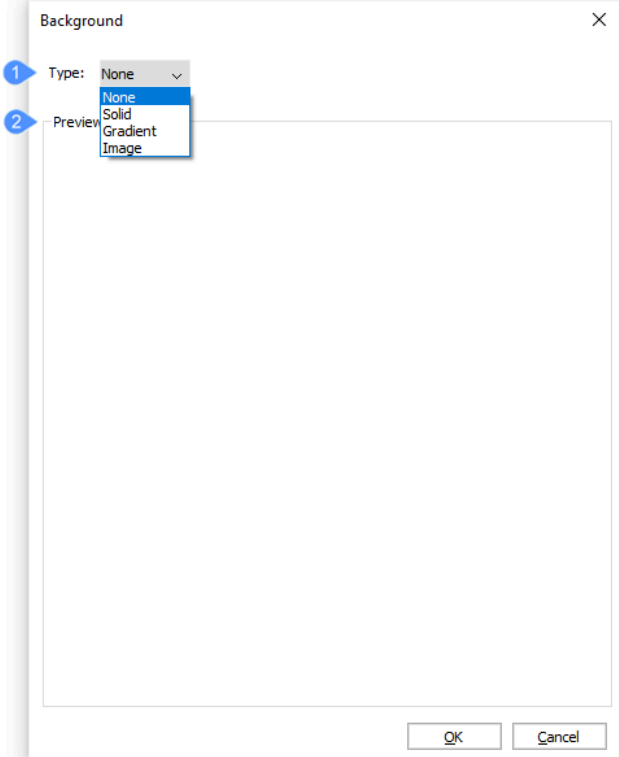
### 4.12.16 Background dialog box

The Background dialog box enables you to apply a background to the current viewport. You can choose from 4 types: None, Solid, Gradient, and Image.

***Прим.:*** Set Backgrounds = On in Visual Styles in Drawing Explorer dialog box to apply the changes you may make in Background dialog box. This option is available in all visual styles, except 2DWIREFRAME.

#### 4.12.16.1 None

When background type is set to None, the Background dialog box includes 2 areas:




1 Type

2 Preview

**Type**
Specifies the background type for the current viewport. When the background type is None, default colors are applied to the current viewport.

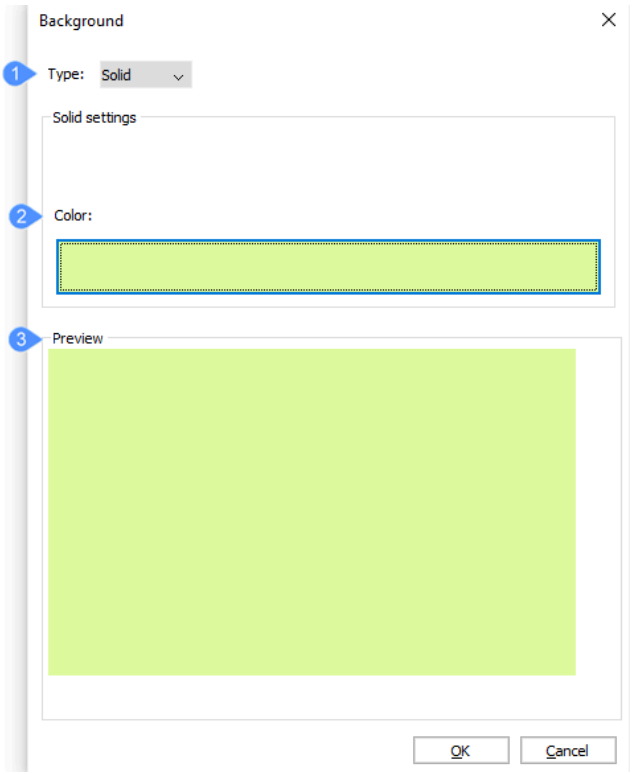
**Preview**
Displays a preview of the background. When set to None, the background depends on the PERSPECTIVE variable:

- **Off**: background is the color specified by the BKGCOLOR variable, most often black or white.
- **On**: two-color gradient that simulates the ground with the sky; the colors are specified by variables that begin with Gradient-, such as GRADIENTCOLORTOP, GRADIENTCOLORMIDDLE and GRADIENTCOLORBOTTOM.

***Прим.:*** After changing the variables' values, rerun the GRADIENTBKGON command.

### 4.12.16.2 Solid

When the background type is set to Solid, the Background dialog box includes 3 areas:




1 Type

2 Solid settings

3 Preview

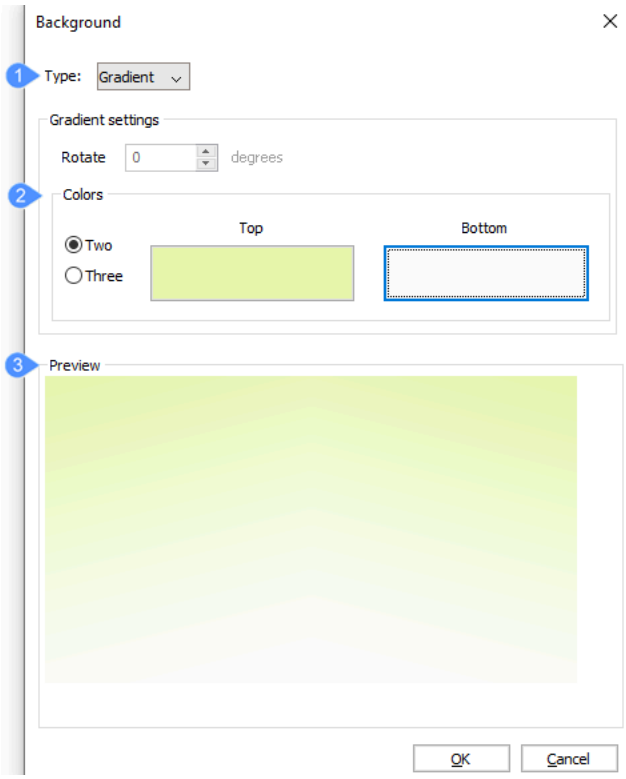
**Type**
Specifies the background type for the current viewport. When the background type is Solid, a single color is applied to the current viewport.

**Solid settings**
Specifies the current color for the solid background. You can click on the color swatch to display the Select Color dialog box and specify a different color.

**Preview**
Displays a preview of the background using the color you selected.

### 4.12.16.3 Gradient

When the background type is set to Gradient, the Background dialog box includes 3 areas:




1 Type

2 Gradient settings

3 Preview

**Type**
Specifies the background type for the current viewport. When the background type is Gradient, a gradient between 2 or 3 colors is applied to the current viewport.

**Gradient settings**
Specifies the rotation and colors for the gradient background.

**Rotate field**
Specifies the rotation angle of the gradient background. You can type an angle or use the arrows to increase or decrease the angle.

**Colors**
Specifies the colors for the gradient background.

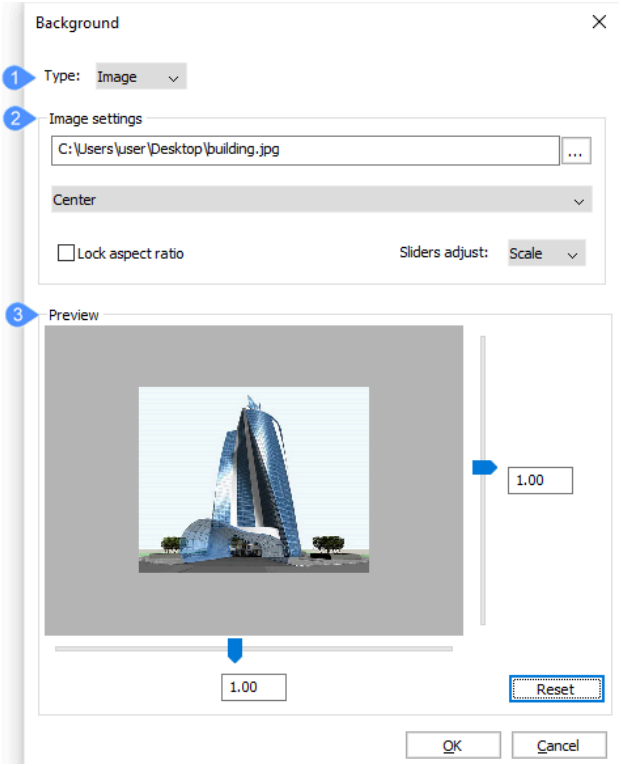
**Two**
Applies a two-color gradient to the background. When you select this option, the dialog box displays two color swatches. One is for the top color and one is for the bottom color. You can click on the color swatch to display the Select Color dialog box and specify a different color.

**Three**
Applies a three-color gradient to the background. When you select this option, the dialog box displays three color swatches. One is for the top color, one is for the middle color, and one is for the bottom color. You can click on the color swatch to display the Select Color dialog box and specify a different color.

**Preview**
Displays a preview of the background using the colors you have selected.

### 4.12.16.4 Image

When the background type is set to Image, the Background dialog box includes 3 areas:




1 Type

2 Image settings

3 Preview

**Type**
Specifies the background type for the current viewport. When the background type is Image, a raster image is applied to the current viewport.

**Image settings**
Specifies the file and position for the image background.

**Path field**
Specifies the name and location of the image file. You can enter the path by typing or choose the browse button to open the Select Image File dialog box.

**Position**
Specifies the position of the image within the current viewport. Options include Center, Stretch and Tile.

**Center**
Centers the image in the viewport.



**Stretch**
Stretches small images to fit the size of the viewport.



**Tile**
Tiles the image as needed to fill the viewport.



**Lock aspect ratio**
Specifies whether the aspect ratio of the image is maintained when the image is scaled. This option is only available when the Sliders Adjust option is set to **Scale**.

- **ON**: maintains the image's aspect ratio; the x and y sliders move together.
- **OFF**: allows the image to be distorted; the x and y sliders move independently of each other.

**Sliders adjust**
Specifies the function of the 2 slides that display in the Preview pane.

- **Offset**: moves the image in the viewport.
- **Scale**: resizes the image; allows you to make the image the same size as the viewport.

**Preview**
Displays a preview of the background using the image file and options you specify.

**X and Y sliders**
Adjusts the offset and/or the scale of the image in the viewport. Which one is adjusted, offset or scale, is determined by the Slides adjust setting.

- **X**: moves the slider left and right. The X field updates accordingly. You can also type a value into the X field.
- **Y**: moves the slider up and down. The Y field updates accordingly. You can also type a value into the Y field.

**Reset**
Resets the offset to a default value of 0 and resets the scale to a default value of 1.

### 4.12.17 BIM BCF panel - Custom login dialog box

The BIM BCF Custom Log In dialog box allows you to connect to a cloud service to start collaborating on issues in the drawings.

**Source URL**

It is generally easier to use cloud services to manage BCF issues. If you wish to connect to these services from within BricsCAD BIM, you will need an account of these services first. This allows you to create issues and manage them in real-time from anywhere.

***Прим.:*** It is not possible tocreateissues within BricsCAD BIM. The BCF panel allows you to connect to these services from within BricsCAD BIM, log into your account and add comments, screenshots or change statuses of existing issues.

To connect to a cloud service, insert in the Source URL field one of the following Cloud services:

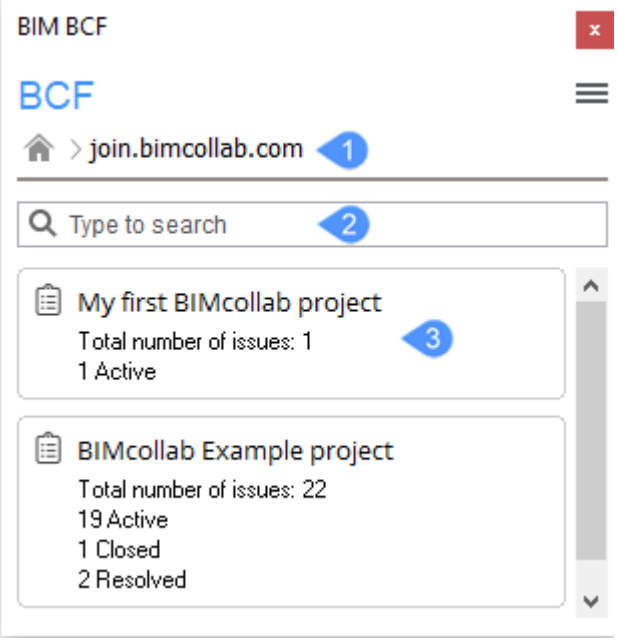
| Cloud Service | Source URL |
|---|---|
| BIMcollab | <space name>.bimcollab.com * |
| BimSync | bcf.bimsync.com |
| BIMtrack | Bcfrestapi.bimtrackapp.co |

***Прим.:*** * The Source URL for BIMcollab includes the name of the space you bought. If using a free account, use join.bimcollab.com.

### 4.12.17.1 Log in

Connects you to the cloud service. You are asked to insert the e-mail address and the password to connect.

After you successfully log in, you can close the browser window and start collaboration on issues in BricsCAD.




1 Source URL

2 Search

3 Project list

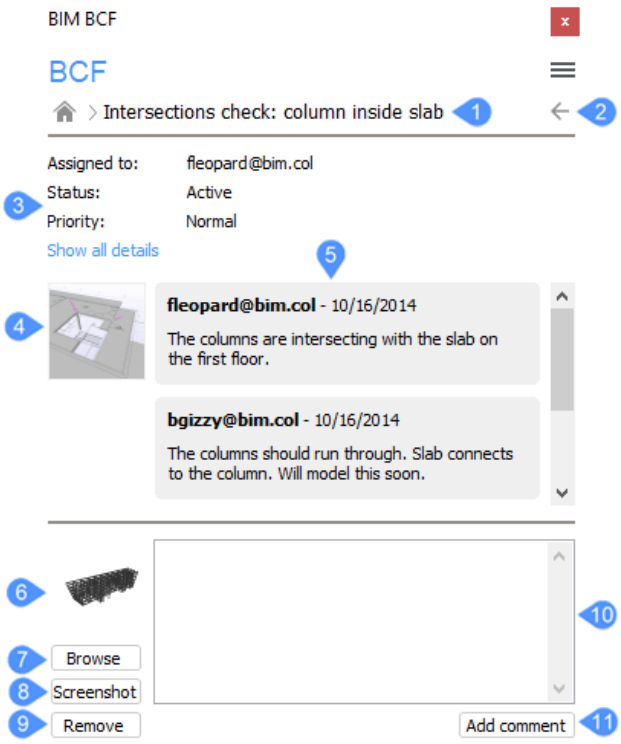
**Source URL**
Displays the source URL.

**Search**
Displays only the projects filtered by the word you type in the search bar.

**Project list**
Displays the list of projects that you are involved in.

### 4.12.17.2 Project list

After you left-click the project will open it and show a list of the issues. When you left-click on an issue, this dialog box will open.




1 Issue name

2 Issue overview

3 Details

4 Thumbnail image

5 Comments

6 Preview

7 Browse

8 Screenshot

9 Remove

10 Comment

11 Add comment

**Issue name**
Displays the name of the opened issue.

**Issue overview**
Returns you to the issue overview.

**Details**
Displays detailed information about the issue, such as status, author and creation date.

**Thumbnail image**
If the comment has an associated camera position, then clicking the thumbnail image will take the camera in the current drawing to this position.

This is useful if you want to know where exactly the issue is, and you do not want to manually search through the model.

**Comments**
Displays the comments on the issue.

**Preview**
Displays a preview of the issue after you press the button no. 8.

**Browse**
Opens the Select a viewpoint for this comment dialog box.

**Screenshot**
Insert a screenshot from model space in the preview field (no. 6).

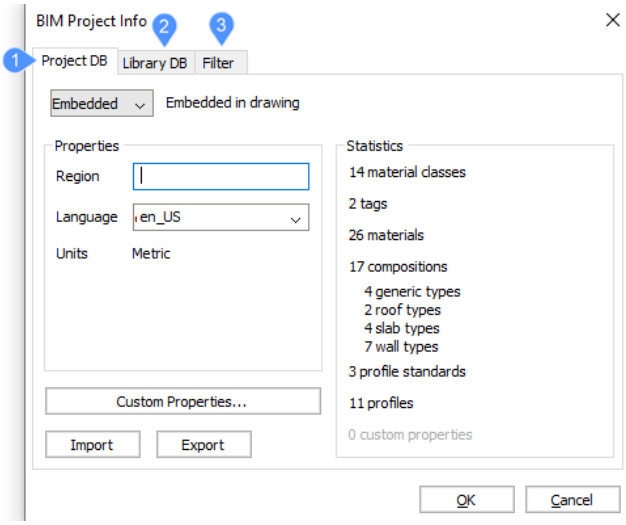
**Remove**
Removes the screenshot preview.

**Comment**
Allows you to insert a comment for an issue.

**Add comment**
Adds the comment you have inserted in field no. 10 to an existing issue.

### 4.12.18 BIM project info dialog box

The BIM Project Info dialog box allows you to set up your project and library database of your BIM models, to control the location, properties and content of the Project and Library databases.



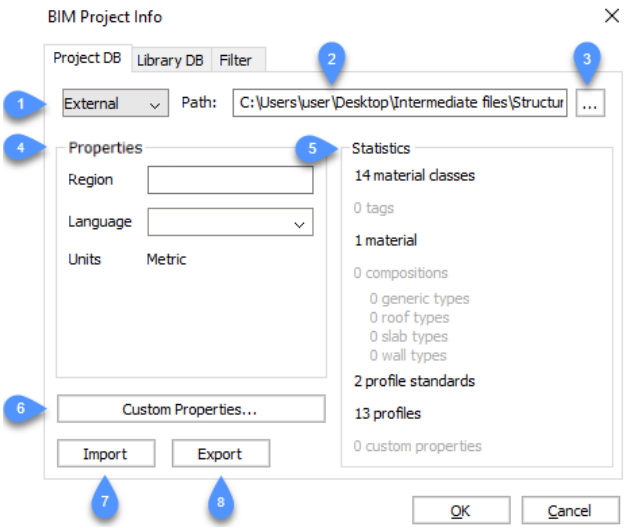

1 Project DB

2 Library DB

3 Filter

#### 4.12.18.1 Project DB

Shows the BIM project info.




1 Project database drop-down list

2 Path

3 Browse

4 Properties

5 Statistics

6 Custom properties...

7 Import

8 Export

**Project database drop-down list**
Choose between Embedded or External.

**Embedded**
Saves the project library in the drawing file.

**External**
Saves the project library in a BIM database file (.bsyslib).

***Прим.:*** The external database keeps the database independent from the current drawing. This is especially useful if you want to share the library between multiple drawings.

When switching from External to Embedded the content of the external library is copied to the embedded project library.

**Path**
Displays the path where the project library can be found.

**Browse**
Opens the Select or create a Bricsys Library dialog box.

**Properties**
Region, Language and Units are the properties of the BIM project.

**Region**
Fill out the region field.

**Language**
You can choose one of the language options from the drop-down list.

**Units**
Displays the system of measurement used in the BIM project (metric, imperial).

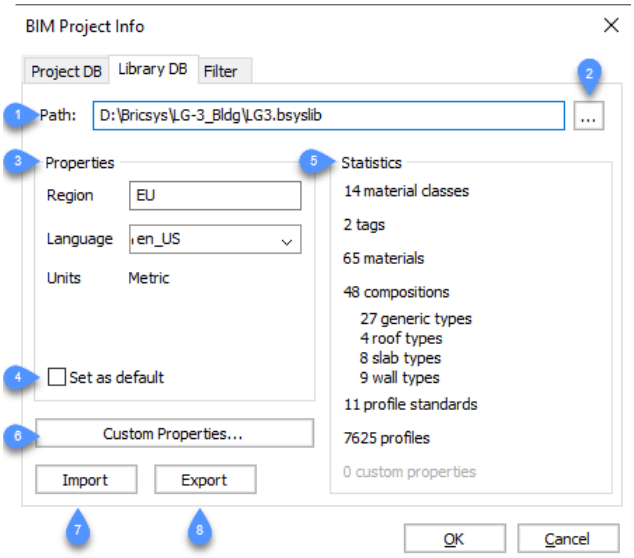
**Statistics**
Shows the location, properties, and the content of the current project library of the BIM model.

**Custom properties…**
Adds custom properties to materials and compositions. Opens the Edit Custom Properties dialog box.

**Import**
Imports the selected *.xml library files. Opens the Select xml/csv file to import dialog box.

**Export**
Saves .xml library files in the defined location. Opens the Select location to save xml file dialog box.

### 4.12.18.2 Library DB




1 Path

2 Browse

3 Properties

4 Set as default

5 Statistics

6 Custom Properties…

7 Import

8 Export

**Path**
Displays the path where you the project library can be found.

**Browse**
Opens the Select or create a Bricsys Library dialog box.

**Properties**
Region, Language and Units are the properties of the project.

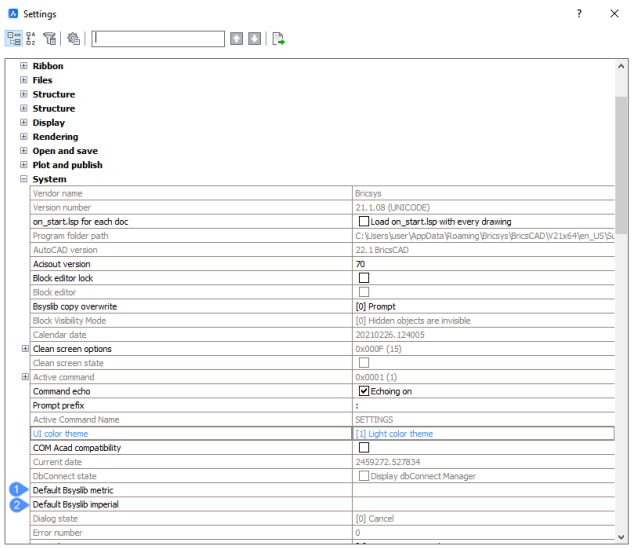
**Region**
Fill out the region field.

**Language**
You can choose one of the language options from the drop-down list.

**Units**
Displays the system of measurement used in the BIM project (metric, imperial).

**Set as default**
This option allows you to set the selected database as default library database for next projects.

***Прим.:*** You can edit the default library settings in the Settings dialog box under Program options/ System.




**Statistics**
Shows the location, properties, and the content of the current project library of the BIM model.

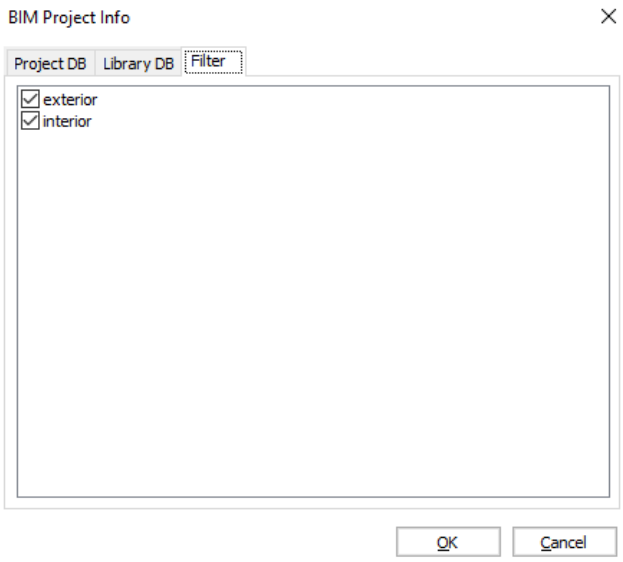
**Custom Properties…**
Adds custom properties to materials and compositions. Opens the Edit Custom Properties dialog box.

**Import**
Imports the selected *.xml library files. Opens the Select xml/csv file to import dialog box.

**Export**
Saves .xml library files in the defined location. Opens the Select location to save xml file dialog box.

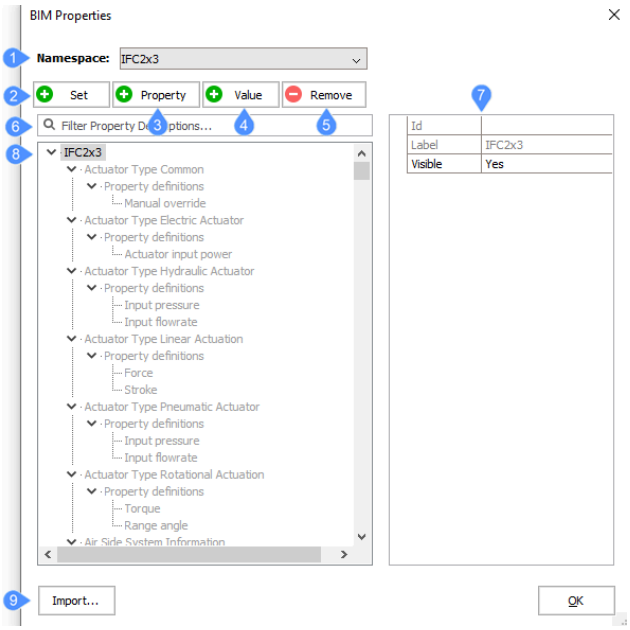
### 4.12.18.3 Filter

Check the tags you want to use in the filter.




## 4.12.19 BIM properties dialog box

The BIM Properties dialog box allows you to create, edit and delete property definitions and to organize properties in property sets.




1 Namespace

2 Add Set

3 Add Property

4 Add Value

5 Remove

6 Filter

7 Properties

8 Properties Tree

9 Import

#### 4.12.19.1 Namespace

By default, 3 namespaces are available:

- **IFC2X3**: shows the IFC2x3 properties; is read-only. This namespace contains properties that are defined by the IFC2X3 schema.
- **Quantity**: shows the quantities properties; is read-only.
- **User**: allows you to create user defined properties.

It is also possible to import namespaces, using the **Import** button.

- **Classification System**: allows you to organize your BIM models with standard classification codes that are used in the construction industry.

#### 4.12.19.2 Add Set

Creates a new property set.

To create a custom property set, first select the user option in the namespace drop-down list, then click the button.

Do the following:

- Type a name in the **Id** field.
- Type a name in the **Label** field.

  The Label name appears in the properties tree. If the **Label** field is empty, the **Id** name is displayed. If you type in the **Label** field, the **Id** field will not be displayed any more. The Id must be unique.
- Per Instance: Allows to enable/disable user-defined properties, on a per-entity basis. When set, the BIM/Entity property sets property is available in the Properties panel when a BIM entity is selected.

#### 4.12.19.3 Add Property

Creates a new property set.

#### 4.12.19.4 Add Value

Creates a new value definition.

#### 4.12.19.5 Remove

Removes the selected property set, property definition or value definition.

#### 4.12.19.6 Filter

Displays only the properties of which the Id and Label contain the search string typed. The properties tree is filtered as you type.

#### 4.12.19.7 Properties

- **Id**: name of the userspace. This field is not editable.
- **Label**: the label of the namespace is used in the properties tree. If the Label field is empty, then the Id is used instead.
- **Visible**: defines whether the namespace properties are visible in the Properties panel.

#### 4.12.19.8 Properties Tree

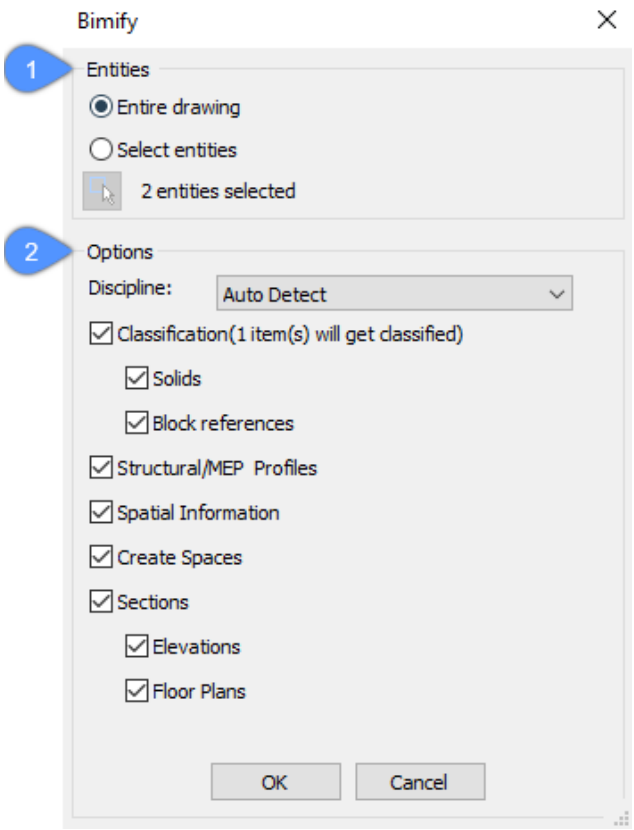
Displays the property sets, property definitions and value definitions.

#### 4.12.19.9 Import

Opens the Select XML file to import dialog box.

### 4.12.20 Bimify dialog box

The Bimify dialog box allows you to analyze the model and to run an automatic classification and a spatial location assignment for the entire model.




1 Entities

### 4.12.20.1 Entities

Specifies if the BIMIFY command needs to be run for all entities in the drawing or just several selected entities.

### 4.12.20.2 Options

**Discipline**
Determines to what discipline the model belongs to in order to improve the automatic classification.

- **Auto Detect**: an AI algorithm determines the discipline automatically.
- **Architectural**: the model is classified as an architectural model.
- **MEP**: the model is classified as a MEP model.
- **Mixed Model**: the model cannot be assigned to 1 discipline and is perceived as a mixed model.

**Classification**
Defines the classification of solids and blocks.

- **Solids**: makes sure that 3D solids are classified as walls, slabs, etc.
- **Blocks**: makes sure that blocks are classified as windows, doors, etc.

**Structural/MEP profiles**
Assigns columns, beams, members or flow segments profiles that are available in BricsCAD BIM profile panel.

***Прим.:*** If no match can be found for a profile element in the standard library, the *BIMIFY* command creates a new custom profile in the project's library.

**Spatial information**
Assigns spatial locations (buildings and floors).

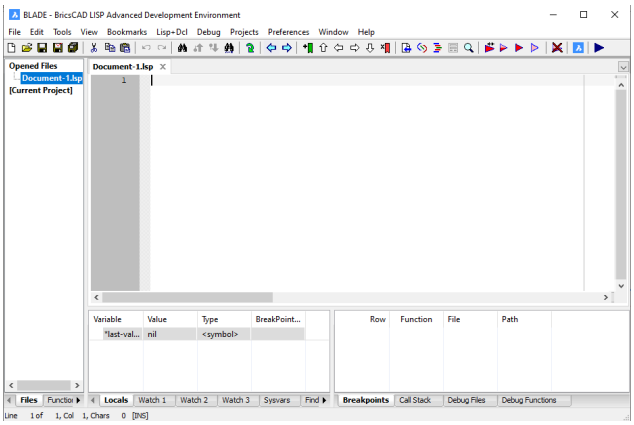
**Create Spaces**
Detects external walls and sets the property Wall Common/is External = ON.

**Sections**
Creates BIM Section entities.

- **Elevations**: creates 4 elevations (Front, Back, Left and Right).
- **Floor Plans**: creates a plan section for each floor.

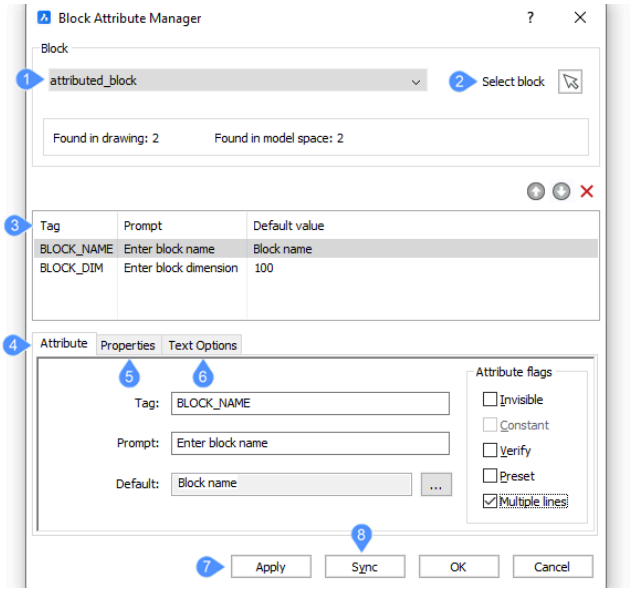
## 4.12.21 Blade - BricsCAD LISP advanced development environment dialog box

The Blade - BricsCAD LISP Advanced Development Environment dialog box allows you to edit and debug LISP applications.




## 4.12.22 Block attribute manager dialog box

The Block attribute manager dialog box allows you to edit all aspects of attributes in a block definition and then optionally apply the changes to all blocks of the same name in the drawing.




1 Block name

2 Select block

3 Attribute list

4 Attribute tab

5 Properties tab

6 Text Options Tab

7 Apply

8 Sync

### 4.12.22.1 Block name

Select a block from the drawing whose attributes you want to edit.

You can select a block from the drop-down list.

#### 4.12.22.2 Select block

Allows you to select a block in the drawing.

#### 4.12.22.3 Attribute list

Select an attribute from the list. It includes the tag, the prompt and the default value.

: moves the attribute up the list.

: moves the attribute down the list.

: deletes the attribute up the block.

#### 4.12.22.4 Attribute tab

Allows you to edit the definition of the selected attribute.

There are 5 attribute flags options:

- **Invisible**: hides the attributes from view; they are not displayed and not printed. Hidden attributes can, however, be displayed with the *ATTDISP* command.
- **Constant**: specifies a default value that the user cannot change.
- **Verify**: forces the user to enter the value a second time; this helps to ensure the value is entered correctly.
- **Preset**: inserts attributes without prompting the user; the attributes can be changed later with the ATTEDIT command.
- **Multiple lines**: when checked, allows the use of multiline text.

#### 4.12.22.5 Properties tab

The**Properties**tab allows you to edit the properties of the selected attribute.

#### 4.12.22.6 Text Options Tab

The**Text Options**tab allows you to edit the text properties of the selected attribute.

#### 4.12.22.7 Apply

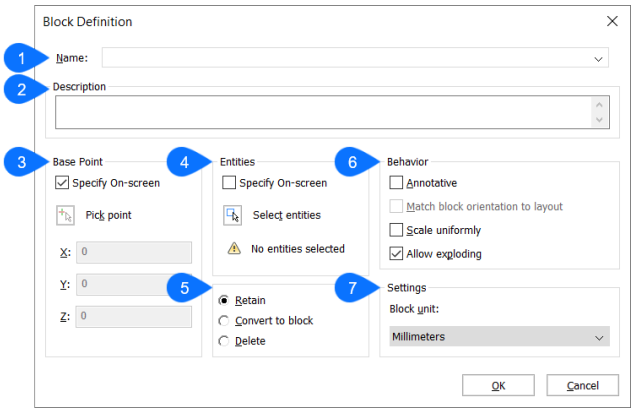
Applies the changes to the block definition, and keeps the dialog box open; choose another block whose attributes you wish to edit.

#### 4.12.22.8 Sync

Applies the changes to all instances of the same block in the drawing.

### 4.12.23 Block definition dialog box

The block definition dialog box allows you to create a new block definition.




1 Name

2 Description

3 Base Point

4 Entities

5 Block Options

6 Behavior

7 Settings

### 4.12.23.1 Name

Specifies the name of the block.

### 4.12.23.2 Description

Provides a description of the block (Optional).

### 4.12.23.3 Base Point

Modifies the properties of the base point of the block definition. This is the point at which the block will be inserted with the INSERT command.

**Specify On-screen**
Allows you to define the base point in the model space after you click the **OK** button, if the box is checked.

**Pick point**
Allows you to pick a point in the drawing for the block's base point. You can also enter the X,Y,Z coordinates in the drawing.

**X/Y/Z**
Defines the coordinates where the block should be inserted. The Z coordinate is usually optional.

### 4.12.23.4 Entities

Selects the entities that make up the block.

**Specify On-screen**
Allows you to select the entities in the model space after you click the **OK** button, if the box is checked.

***Прим.:*** Optionally, include Reference Curves in the selection, which allows to automatically align a block during insertion.

**Select entities**
Selects one or more entities in the drawing.

### 4.12.23.5 Block Options

Defines how the entities are turned into a block.

***Прим.:*** The Convert to block option is the most efficient one.

**Retain**
Selected entities are retained as entities, and the block definition is created in the drawing.

**Convert to block**
Selected entities are converted to a block.

**Delete**
Selected entities are deleted after the block definition is created.

### 4.12.23.6 Behavior

**Annotative**
Sets the annotative property of the block. This kind of block should be created when the annotation scale in model or paper space is 1:1. By being annotative, the block automatically scales itself according to the current annotation scale factor.
Choose whether you want the block to follow annotative scaling:

- **On** – the block will scale itself to whatever annotative scale is in effect. The Scale Uniformly option is grayed out (unavailable).
- **Off** - the block follows the scale factor you give the block during the *Insert* command.

**Match block orientation to layout**
Determines if annotative blocks match the orientation of the layout.

- **On**: annotative blocks display upright, regardless of the orientation of the viewport.
- **Off**: annotative blocks match the orientation of the viewport.

**Scale uniformly**
Determines if blocks can be scaled non-uniformly. This option is not available to annotatively-scaled blocks.

- **On** - X, Y, and Z scale factors of blocks are the same. This prevents blocks from being distorted.
- **Off** - blocks can be inserted with different X, Y, and Z scale factors. This is useful for objects that can have different dimensions, such as differently sized table tops.

**Allow exploding**
Determines if users can explode the block after it is inserted. When a block is exploded, it loses its block status, and the individual entities can be edited.

***Совет:*** To edit the entities of an unexploded block, use the *BEDIT* command.

- **On** - block can be exploded after being inserted, with the Explode command

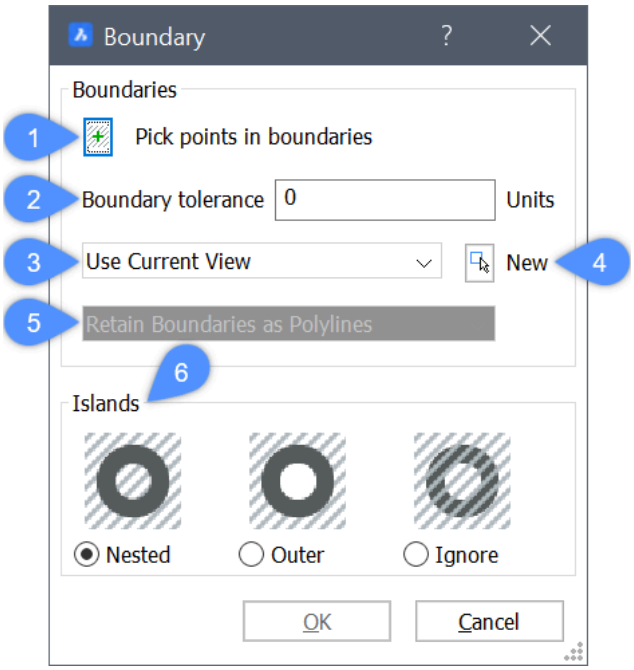
- **Off** - blocks cannot be exploded; you can change this property with the *EXPBLOCKS* command, Blocks section.

### 4.12.23.7 Settings

**Block unit**
Scales the block correctly when the block definition is inserted in a drawing whose units are different from the drawing in which the block was created. More specifically, in which the INSUNITS variable differs.

## 4.12.24 Boundary dialog box

The Boundary dialog box allows you to create closed polylines, defined by surrounding entities.




1 Pick points in boundary
2 Boundary tolerance
3 Boundary set
4 New
5 Retain Boundaries as Polylines
6 Islands

### 4.12.24.1 Pick points in boundary

It allows you to specify a point inside an enclosed area for which you want to create a boundary. It temporarily dismisses the Boundary dialog box.

### 4.12.24.2 Boundary tolerance

Specify the largest gap that can exist in the boundary for BricsCAD to consider the area closed.

***Прим.:*** When boundary tolerance is 0, no gaps can exist.

### 4.12.24.3 Boundary set

Specify where BricsCAD should search for entities that make up the boundary.

- **Use Current View**: searches all entities in the current viewport.
- **Use Boundary Set**: searches only the current selection set for possible boundaries.

### 4.12.24.4 New

It allows you to create a new selection set of entities that make up the boundary. It temporarily dismisses the Boundary dialog box.

### 4.12.24.5 Retain Boundaries as Polylines [Read only]

Indicates how boundaries are retained.

### 4.12.24.6 Islands

Specify island detection. An island is an enclosed area inside a boundary.

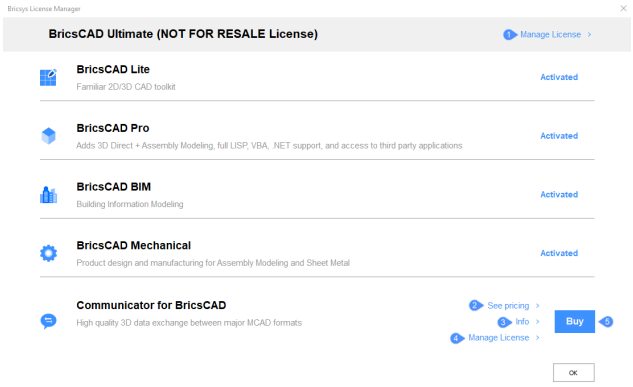
**Nested**
Treats every island as a boundary.

**Outer**
Creates a boundary only of the outermost entities.

**Ignore**
Ignores the innermost areas. A boundary is created between the outermost area and islands.

## 4.12.25 Bricsys license manager dialog box

The Bricsys License Manager dialog box allows you to activate/deactivate the software licenses for BricsCAD and Communicator.




1 Manage License

2 See pricing

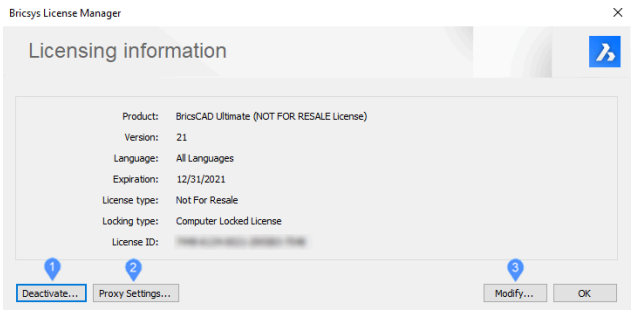
3 Info

4 Manage License for Communicator

5 Buy

### 4.12.25.1 Manage License

Opens the Bricsys License Manager dialog box.



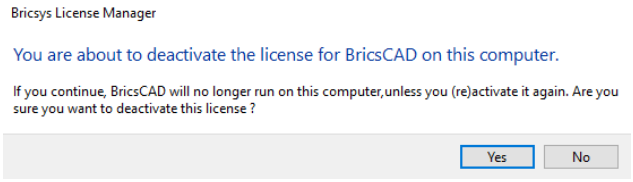

1 Deactivate

2 Proxy Settings...

3 Modify...

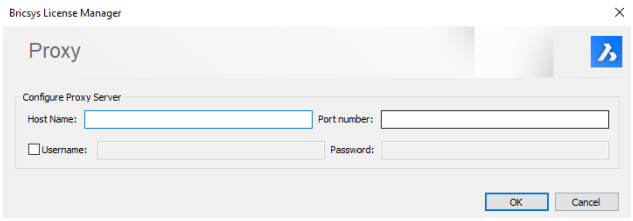
**Deactivate**

Opens the Bricsys License Manager dialog box for confirming deactivation.



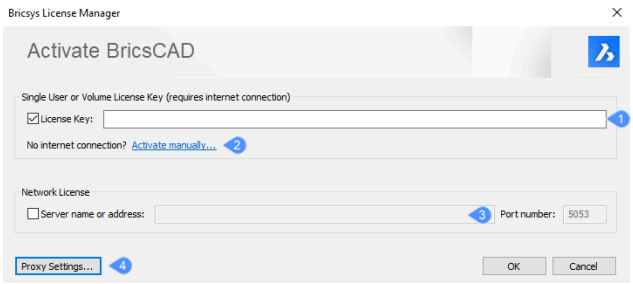

**Proxy Settings...**

Opens the Bricsys License Manager dialog box where you can configure the proxy server.




**Modify...**

Opens the Bricsys License Manager dialog box for activating BricsCAD where you can insert your license key, your network license or activate it manually.




1 License Key

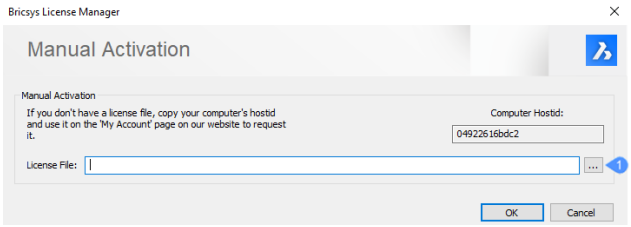
2 Activate manually…

3 Network License

4 Proxy Settings…

**License Key**
If you are connected to the internet, you can enter in this field a single user or volume license key.

**Activate manually…**
Opens the Bricsys License Manager dialog box for Manual Activation, where you can select the license file. The .lic file can be found in this path: C:\ProgramData\Bricsys\BricsCADV21.lic




1 Select license file

**Select license file**
Opens the Select license file dialog box.

**Network License**
If you are using a network license, enter the host name or the IP address of the network license server.

**Proxy Settings…**
Opens the Bricsys License Manager dialog box to configure the proxy server.

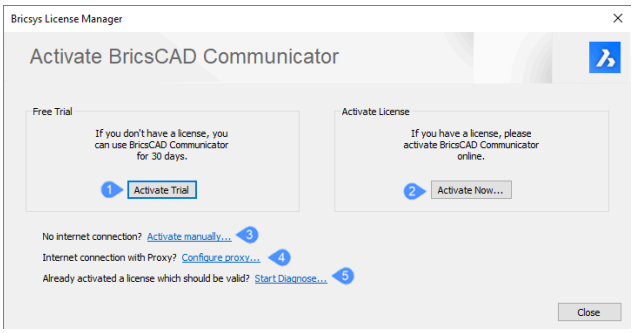
### 4.12.25.2 See pricing

Opens the Bricsys official website (Bricsys), where you can see the prices for Communicator and 1 year/3 years/lifetime BricsCAD subscription.

### 4.12.25.3 Info

Opens the Communicator Info dialog box.

### 4.12.25.4 Manage License for Communicator




1 Activate Trial

2 Activate Now

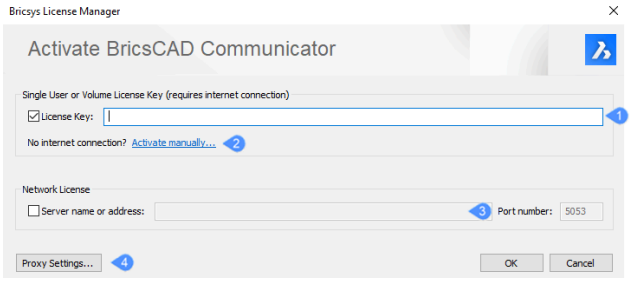
3 Activate manually...

4 Configure proxy...

5 Start Diagnose...

**Activate Trial**
Activates the 30 days free trial of BricsCAD Communicator.

**Activate Now**
Opens the Activate BricsCAD Communicator dialog box.




1 License Key

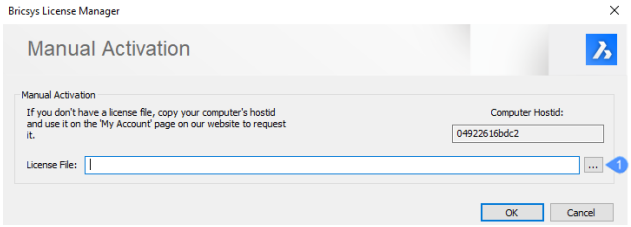
2 Activate manually

3 Network License

4 Proxy settings

**License Key**
If you are connected to the internet, you can enter in this field a single user or volume license key.

**Activate manually…**
Opens the Bricsys License Manager dialog box for Manual Activation, where you can select the license file. The .lic file can be found in this path: C:\ProgramData\Bricsys\BricsCADV21.lic



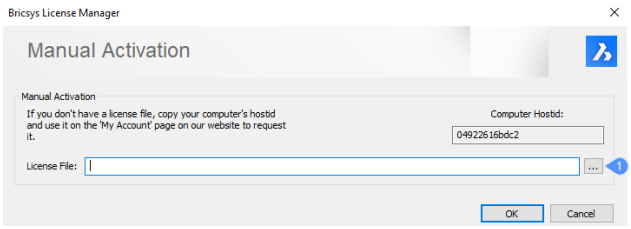

1 Select license file

**Select license file**
Opens the Select license file dialog box.

**Network License**
If you are using a network license, enter the host name or the IP address of the network license server.

**Proxy Settings…**
Opens the Bricsys License Manager dialog box to configure the proxy server.

**Activate manually…**
Opens the Bricsys License Manager dialog box for activating BricsCAD Communicator.



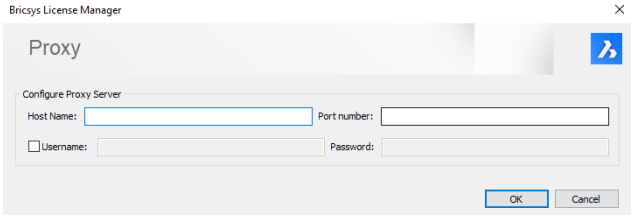

1 Select license file

**Select license file**

Opens the Select license file dialog box.

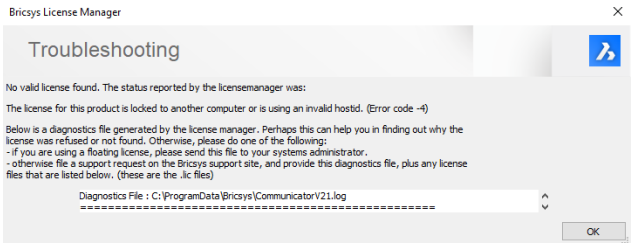
**Configure proxy…**

Opens the Bricsys License Manager dialog box where you can configure the proxy server.




**Start Diagnose…**

Opens the Bricsys License Manager dialog box for Troubleshooting.



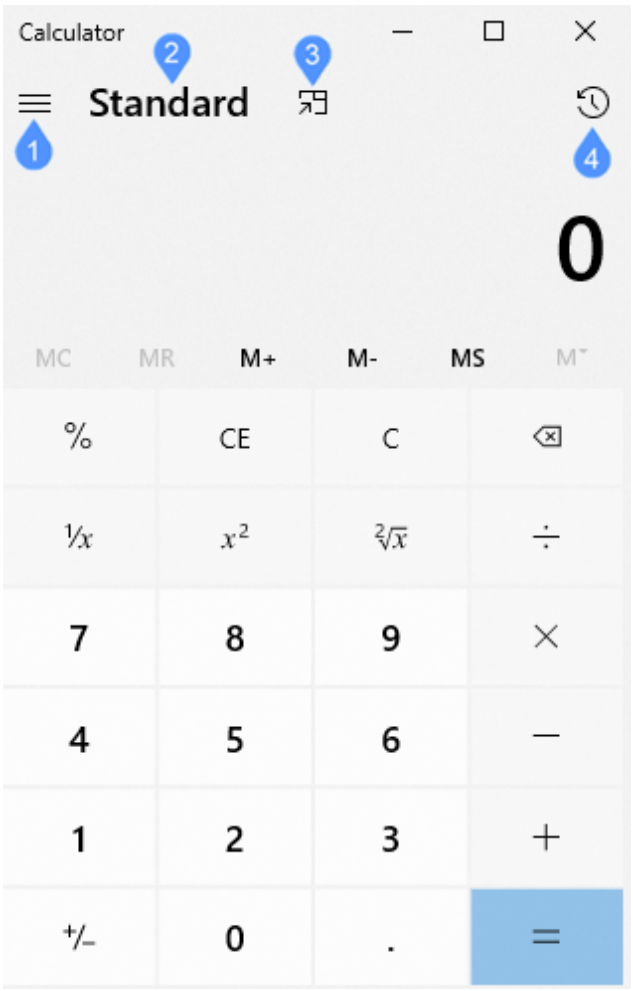

### 4.12.25.5 Buy

Opens the Bricsys official website (Bricsys) where you can buy or subscribe to BricsCAD.

## 4.12.26 Calculator dialog box

The Calculator dialog box displays the operating system's software calculator.

When you are using the Standard view, it includes 4 areas.




1 View

2 View's name

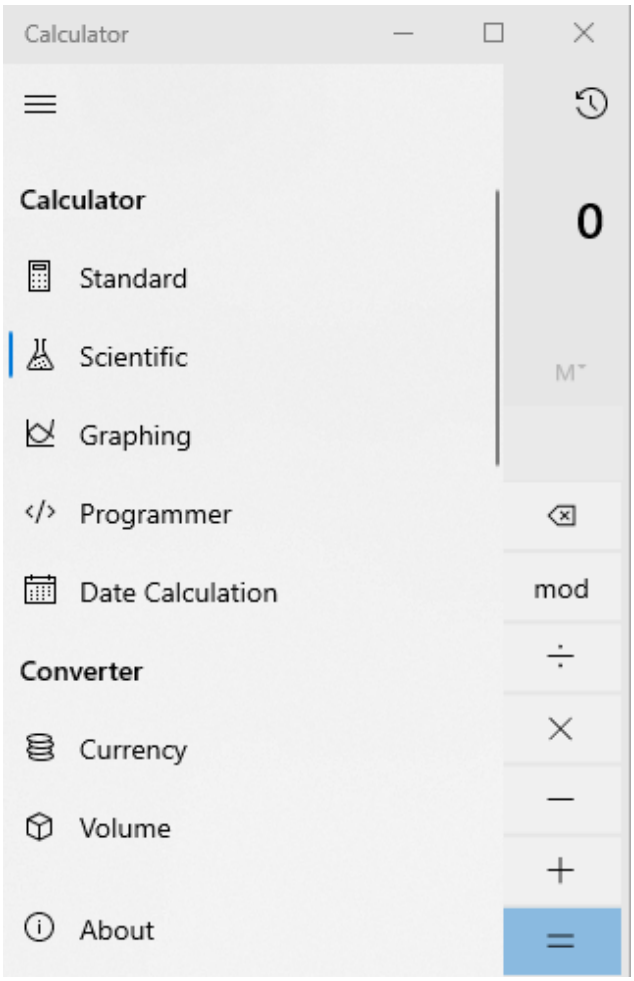
3 Keep on top

4 History

### 4.12.26.1 View

Displays the possible views you can choose from.

It has 2 categories to choose from: **Calculator** (Standard, Scientific, Graphic, Programmer, Date Calculation) and **Converter** (Currency, Volume, Length, Weight and Mass, Temperature, Energy, Area, Speed, Time, Power, Data, Pressure, Angle).




#### 4.12.26.2 View's name

Displays the current view name.

#### 4.12.26.3 Keep on top

You can also use the keyboard shortcut ALT + Up to access this function.

Note that this function is available only for the Standard view.

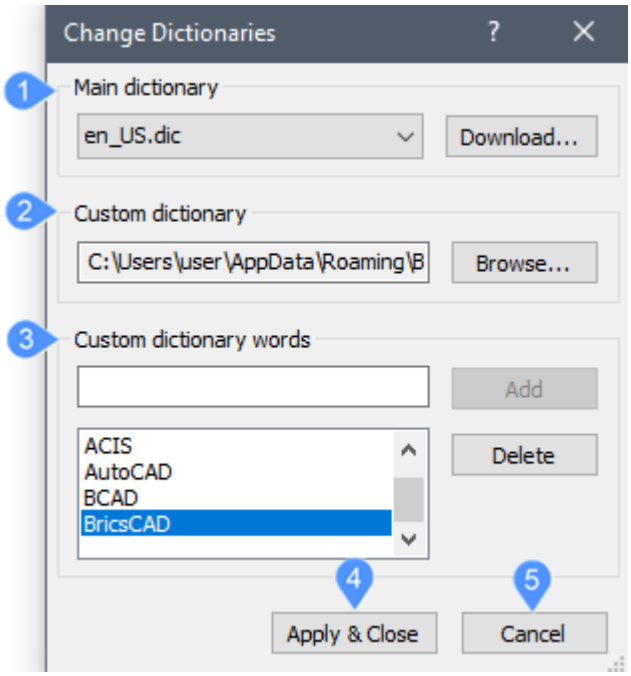
#### 4.12.26.4 History

Displays the history of your calculations done in the current session.

***Прим.:*** This function is available only for the Standard and Scientific calculator.

### 4.12.27 Change dictionaries dialog box

The Change dictionaries dialog box allows you to change dictionaries.




1 Main dictionary

2 Custom dictionary

3 Custom dictionary words

4 Apply & Close

5 Cancel

### 4.12.27.1 Help

Opens the Bricsys Help article for the SPELL command.

### 4.12.27.2 Main dictionary

Specifies the main dictionary to use for spell checking.

**File name**
Specify the dictionary file to use as the main dictionary. Available files are displayed in the drop-down list.

**Download**
Download additional dictionaries from OpenOffice.org. Opens the Web browser at: https://extensions.openoffice.org/dictionaries

### 4.12.27.3 Custom dictionary

Specifies a custom dictionary to use for spell checking. Use custom dictionaries for discipline-specific words, such as medical or mechanical terms.

**File location**
Specifies the file name and location of a custom dictionary to use in addition to the main dictionary.

**Browse**
Open the Select Custom Dictionary dialog box to select a custom dictionary file.

### 4.12.27.4 Custom dictionary words

Specifies words to be included in the custom dictionary.

**Word field**
Allows you to enter a word to be included in the custom dictionary.

**Add**
Add the word you entered in the word field to the custom dictionary.

**Words list**
Displays all the words included in the custom dictionary.

**Delete**
Delete a selected word from the custom dictionary.

### 4.12.27.5 Apply & Close

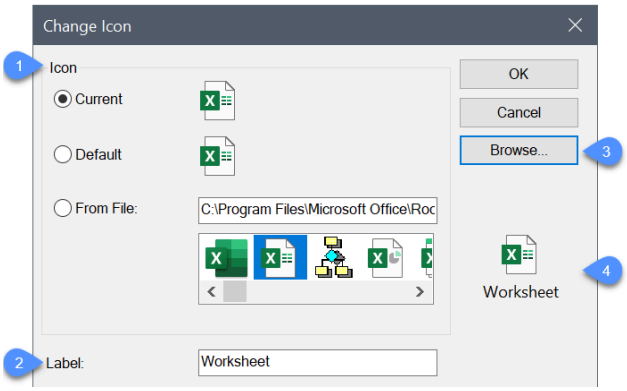
Apply your changes and close the dialog box. The custom dictionary file is only updated with added or deleted words after you choose Apply and Close.

### 4.12.27.6 Cancel

Close the dialog box without applying changes.

### 4.12.28 Change Icon dialog box

The Change Icon dialog box allows you to change the icon of an OLE Frame entity before inserting it into the drawing. The icon is related to the document type and it is defined by the source applications.




1 Icon

2 Label

3 Buttons

4 Preview

### 4.12.28.1 Icon

Specifies an icon to represent an embedded or linked document.

**Current**
Displays the current icon.

**Default**
Displays the default icon.

**From File**
Choose another icon.

### 4.12.28.2 Label

Optionally edit the label text.

### 4.12.28.3 Buttons

The Buttons area contains OK, Cancel and the Browse button.

**OK**
Click OK to accept the changes and to return to the previous dialog box (Insert Object).

**Cancel**
Click Cancel to discard the changes and to return to the previous dialog box (Insert Object).

**Browse**
Displays the Browse dialog box (a standard file selection dialog box), in which you can select an icon from a file.

### 4.12.28.4 Preview

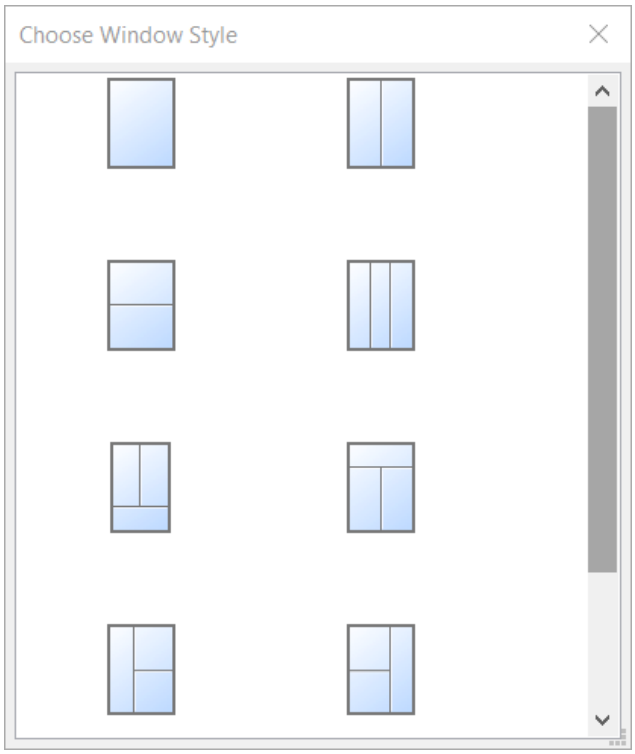
Displays the icon preview.

## 4.12.29 Choose window style dialog box

The Choose Window Style dialog box allows you to specify the style of a window.

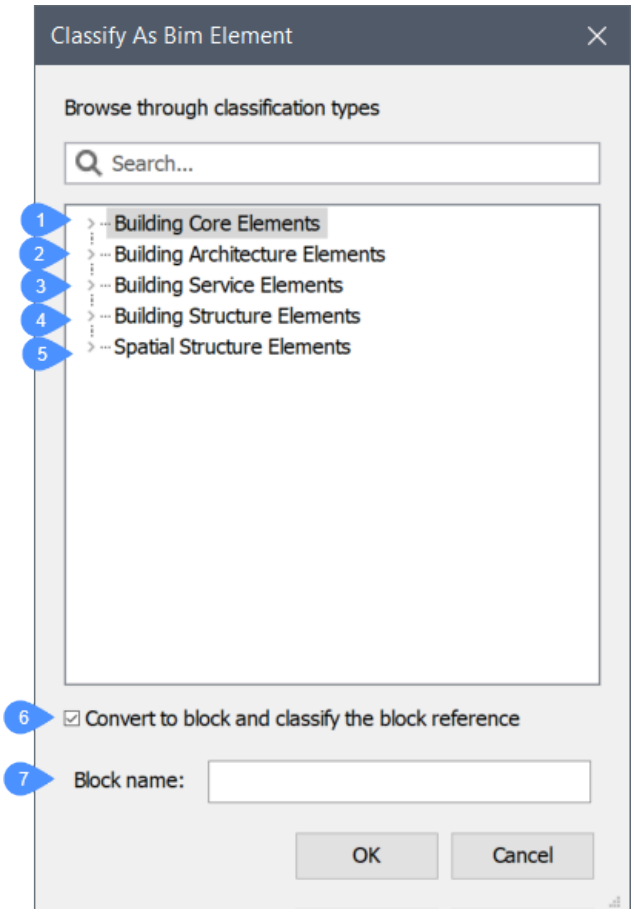
A series of parameters is created, which can be edited in the Parameters section of the Properties panel.

Rectangular windows and openings have a W (Width) and H (Height) parameter.




### 4.12.30 Classify As Bim Element dialog box

The **Classify As Bim Element** dialog box specifies the types of classification or converts the selection to a block and classifies the block reference.




1 Building Core Elements

2 Building Architecture Elements

3 Building Service Elements

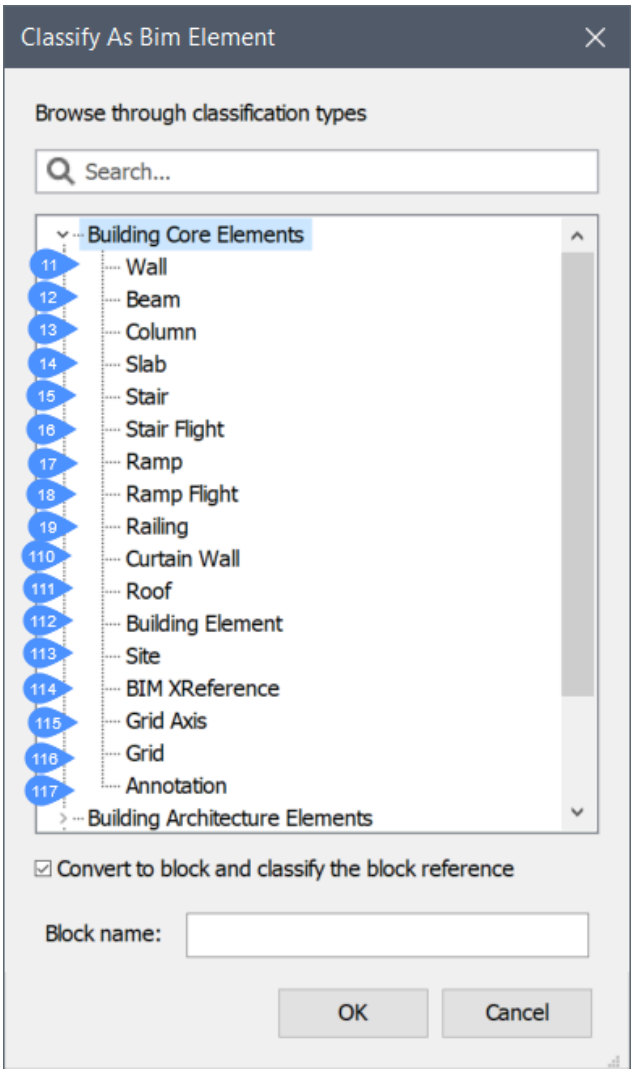
4 Building Structure Elements

5 Spatial Structure Elements

6 Convert to block and classify the block reference

7 Block name

### 4.12.30.1 Building Core Elements




**11.** Wall

**12.** Beam

**13.** Column

**14.** Slab

**15.** Stair

**16.** Stair Flight

**17.** Ramp

**18.** Ramp Flight

**19.** Railing

**110.** Curtain Wall

**111.** Roof

**112.** Building Element

**113.** Site

**114.** BIM XReference

**115.** Grid Axis

**116.** Grid

**117.** Annotation

**Wall**
Vertical construction that subdivides spaces.
Example: separation wall, bearing wall, interior wall.

**Beam**
Structural member that carries loads between or beyond points of support, usually narrow in relation to its length and (nearly) horizontal.
Example: I-beam, L-beam, T-beam, double T-beam.

**Column**
Structural member of slender form, usually vertical, that transmits to its base the forces, primarily in compression, that are applied to it.
Example: Doric column, Ionic column, Corinthian column.

**Slab**
Component of the construction that normally encloses a space vertically. The slab may provide the lower support (floor) or upper construction (roof slab) in any space in a building.
Example: floor slab, roof slab.

**Stair**
Construction comprising a succession of horizontal stages (steps or landings) that make it possible to pass on foot to other levels.
Example: stairs, escalator, spiral staircases interrupted with landings.

**Stair Flight**
Assembly of building components in a single "run" of stair steps (not interrupted by a landing). The stair steps and any stringers are included in this object.
Example: stairs, escalator, spiral staircases NOT interrupted with landings.

**Ramp**
Inclined way or floor joining 2 surfaces at different levels.
Example: Slope/Ramp interrupted with landings.

**Ramp Flight**
Inclined slab segment, normally providing a human circulation link between 2 landings, floors or slabs at different elevations.
Example: Slope/Ramp NOT interrupted with landings.

**Railing**
The railing is a frame assembly adjacent to human circulation spaces and at some space boundaries where it is used instead of walls or to complement walls. It is designed to aid humans, either as an optional physical support, or to prevent injury by falling.
Example: handrail, balustrade, grab bar, guard rail.

**Curtain Wall**
Exterior wall of a building which is an assembly of components, hung from the edge of the floor/roof structure rather than bearing on a floor.

**Roof**
Construction enclosing the building from above.
Example: flat roof, gable roof, gambrel roof, hip roof.

**Building Element**
The building element comprises all elements that are primarily part of the construction of a building.
Example: structural and space separating system like walls, beams, doors.

**Site**
A defined area of land, possibly covered with water, on which the project construction is to be completed.

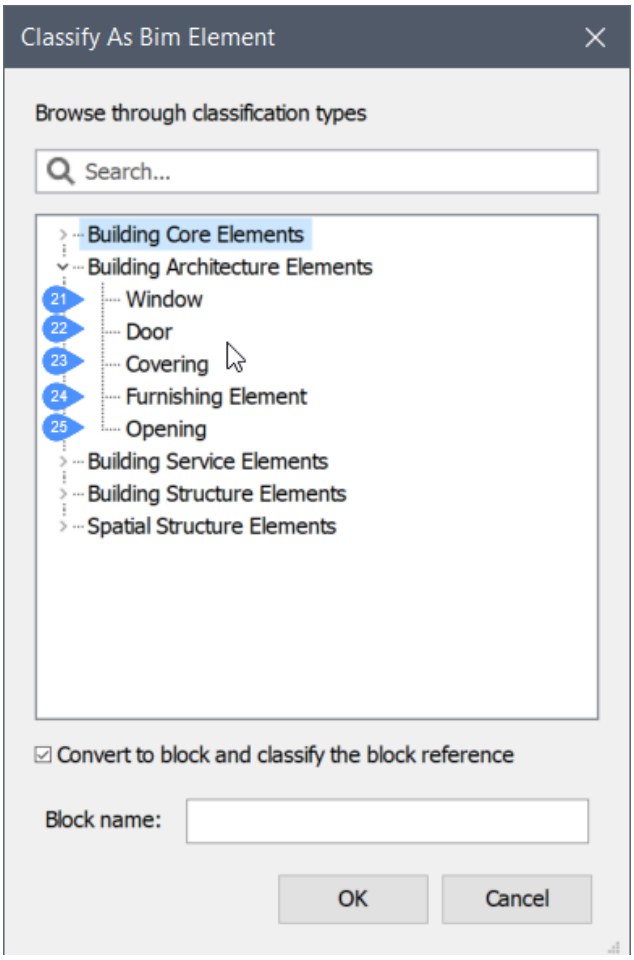
**BIM XReference**
The identification of information that is not explicitly represented in the current model or in the project database. Such information may be contained in classifications, documents or libraries within the external source.

**Grid Axis**
An individual axis that is defined in the context of a design grid. The axis definition is based on a curve of dimensionality 2.
Example: X-axis, Y-axis, Z-axis.

**Grid**
A planar design grid defined in 3D space used as an aid in locating structural and design elements.
Example: rectangular grid, radial grid, triangular grid.

**Annotation**
A graphical representation within the geometric context of a project, that adds a note or meaning to the objects of the project.
Example: additional line drawings, text, dimensioning, hatching, ...

### 4.12.30.2 Building Architecture Elements




**21.** Window

**22.** Door

**23.** Covering

**24.** Furnishing Element

**25.** Opening

**Window**

Construction for closing a (near) vertical opening in a wall or pitched roof that will admit light and may admit fresh air. The window is a building element that is predominately used to provide natural light and fresh air. A window consists of a lining and one or several panels.

Example: skylights, light domes, swinging, pivoting, sliding, or revolving panels and fixed panels.

**Door**

Construction for closing an opening. The door is a building element that is predominately used to provide controlled access for people and goods. A door consists of a lining and one or several panels.

Example: hinged, pivoted, sliding, revolving, folding doors.

**Covering**

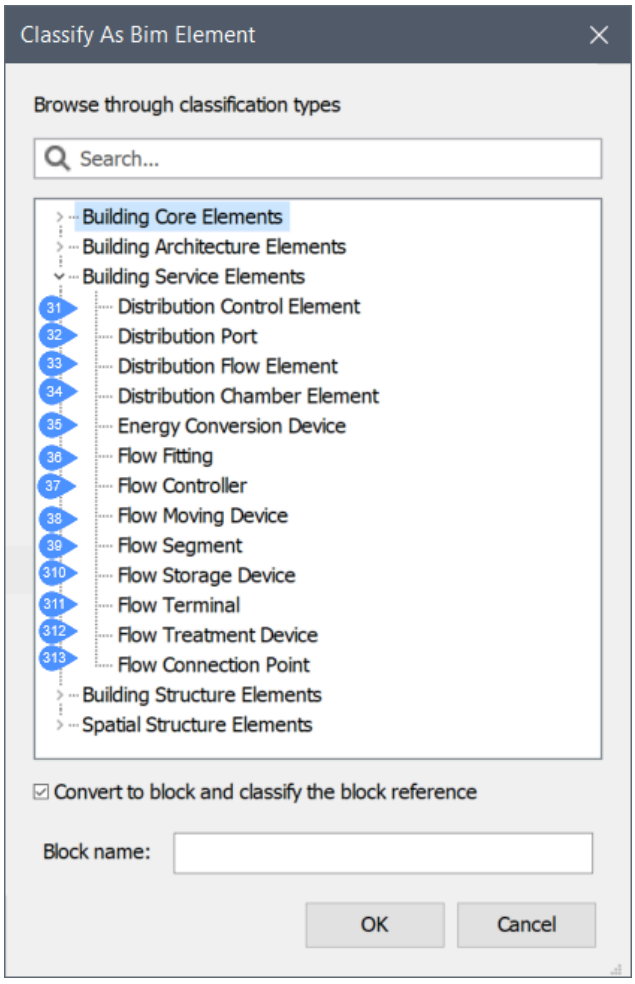
A covering is an element which covers some part of another element and is fully dependent on that other element.

Example: wall claddings, floorings, suspended ceilings, finish trim, base molding.

**Furnishing Element**
Generalization of all furniture related objects.
Example: closet, table, chair, bed.

**Opening**
Represents a void within any element that has physical manifestation.
Example: opening for a door, opening for a window.

### 4.12.30.3 Building Service Elements




**31.** Distribution Control Element

**32.** Distribution Port

**33.** Distribution Flow Element

**34.** Distribution Chamber Element

**35.** Energy Conversion Device

**36.** Flow Fitting

**37.** Flow Controller

**38.** Flow Moving Device

**39.** Flow Segment

**310.** Flow Storage Device

**311.** Flow Terminal

**312.** Flow Treatment Device

**313.** Flow Connection Point

**Distribution Control Element**

The occurrence elements of a building automation control system that are used to impart control over elements of a distribution system. It senses elements and measures changes in the controlled variable such as temperature, humidity, pressure, or flow.

Example: thermostat

**Distribution Port**

The occurrence of a specialized port for use within the context of distribution elements.

Example: A gas-powered hot water heater may have 3 ports: GAS, DOMESTICCOLDWATER, and DOMESTICHOTWATER. The heater is a member of 2 systems (GAS and DOMESTICCOLDWATER) and hosts 1 system (DOMESTICHOTWATER) at the corresponding port.

**Distribution Flow Element**

The occurrence elements of a distribution system that facilitate the distribution of energy or matter, such as air, water or power.

Example: ducts, pipes, wires, fittings

**Distribution Chamber Element**

A place at which distribution systems and their constituent elements may be inspected or through which they may travel.

Example: sump, trench

**Energy Conversion Device**

The occurrence of a device used to perform energy conversion or heat transfer and typically participates in a flow distribution system.

Example: boiler, chiller, or a cooling coil

**Flow Fitting**

The occurrence of a junction or transition in a flow distribution system.

Example: elbow, tee, a junction box in an electrical distribution system

**Flow Controller**

The occurrence of elements of a distribution system that are used to regulate flow through a distribution system.

Example: damper, valve, switch, relay

**Flow Moving Device**

The occurrence of an apparatus used to distribute, circulate or perform conveyance of fluids, including liquids and gases, and typically participates in a flow distribution system.

Example: pump, fan

**Flow Segment**

A flow segment is a section of a distribution system that typically has only 2 ports.

Example: section of a duct, pipe, conduit, etc.

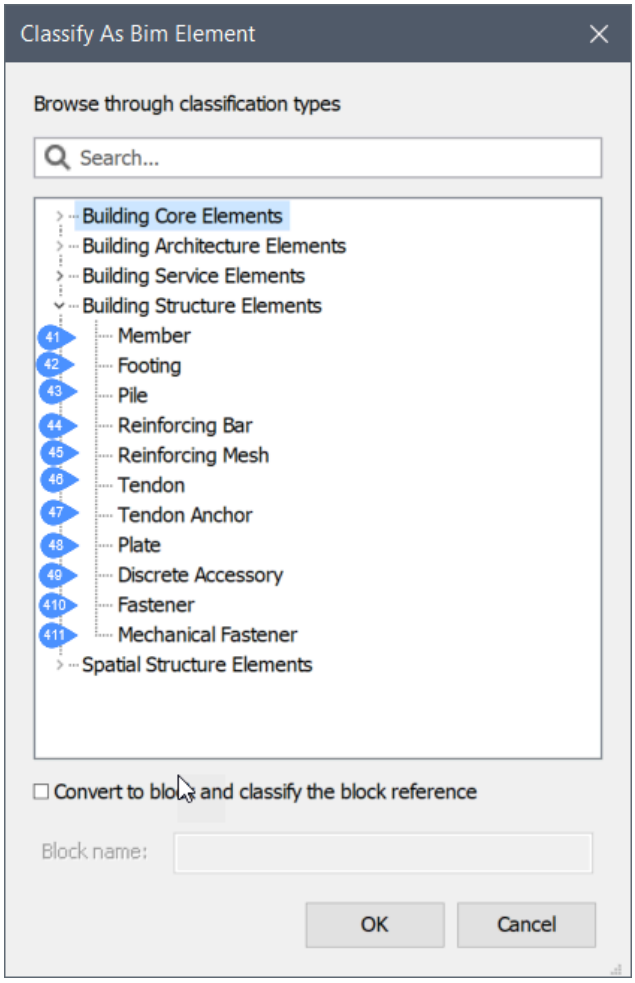
**Flow Storage Device**
Device used for the temporary storage of a fluid such as a liquid or a gas, the voltage potential induced by the induced electron flow.
Example: tank, battery

**Flow Terminal**
The occurrence of a permanently attached element that acts as a terminus or beginning of a distribution system. A terminal is typically a point at which a system interfaces with an external environment.
Example: air outlet, drain, water closet, sink

**Flow Treatment Device**
Device used to change the physical properties of the medium, flow treatment types (or the instantiable subtypes) may be exchanged without being already assigned to occurrences.
Example: air, oil or water filter, duct silencer

**Flow Connection Point**
The connection point is used to describe the geometric constraints that facilitate the physical connection of 2 objects at a point.

### 4.12.30.4 Building Structure Elements




**41.** Member

**42.** Footing

**43.** Pile

**44.** Reinforcing Bar

**45.** Reinforcing Mesh

**46.** Tendon

**47.** Tendon Anchor

**48.** Plate

**49.** Discrete Accessory

**410.** Fastener

**411.** Mechanical Fastener

**Member**

Structural member designed to carry loads between or beyond points of support. It is not required to be load bearing. The orientation of the member (being horizontal, vertical or sloped) is not relevant to its definition.

Example: diagonal element of a truss construction

**Footing**

A part of the foundation of a structure that spreads and transmits the load directly to the soil.

Example: wall footing, stepped footing, sloped footing, raft footing

**Pile**

A slender timber, concrete or steel structural element, driven, jetted or otherwise embedded on end in the ground for the purpose of supporting a load. A pile is also characterized as deep foundation, where the loads are transferred to deeper subsurface layers.

Example: pile foundation

**Reinforcing Bar**

A steel bar, usually with manufactured deformations in the surface, used in concrete and masonry construction to provide additional strength.

Example: reinforced concrete

**Reinforcing Mesh**

A series of longitudinal and transverse wires or bars of various gauges, arranged at right angles to each other and welded at all points of intersection.

Example: reinforced concrete floor slab

**Tendon**

A steel element used to impart pre-stress to concrete when the element is tensioned.

Example: wire, cable, bar, rod, strand

**Tendon Anchor**

A tendon anchor is the end connection for tendons in pre-stressed or post-tensioned concrete.

**Plate**

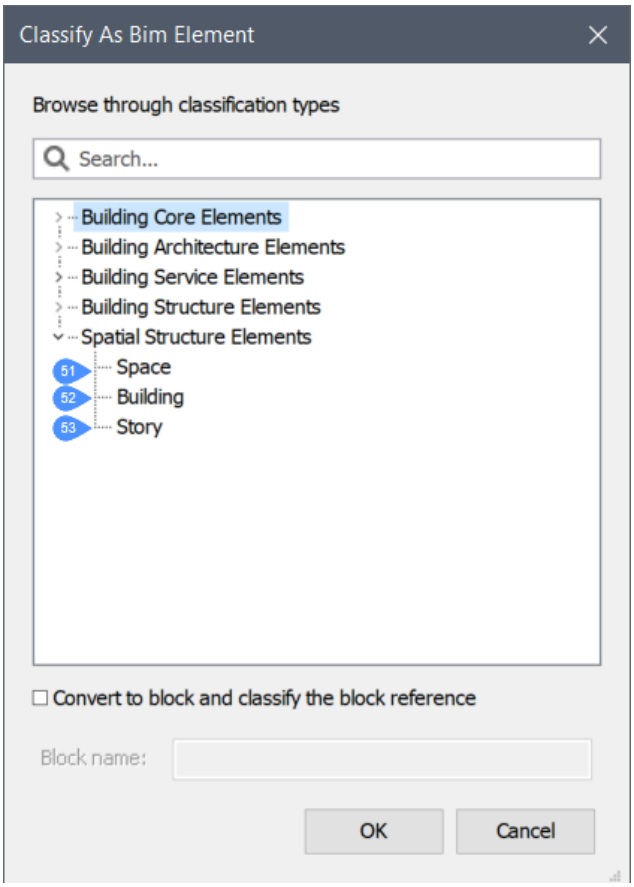
A Plate is a planar and often flat part with constant thickness. A plate can be a structural part carrying loads between or beyond points of support, however it is not required to be load bearing. The location of the plate (horizontal, vertical or sloped) is not relevant to its definition.

**Discrete Accessory**
A discrete accessory is a representation of different kinds of accessories included in or added to elements.
Example: visible steel corbel (corbel system made from steel components protruding from the master element), corner fixing plate (fixing plate attached to the corner of an element), steel plate (steel plate used as an accessory in a joint)

**Fastener**
Representations of fixing parts which are used as fasteners to connect or join elements with other elements.
Example: jointing mortar, glue, weld

**Mechanical Fastener**
Fasteners connecting building elements mechanically.
Example: bolt, screw, nail, nut, washer, rivet

### 4.12.30.5 Spatial Structure Elements




**51.** Space

**52.** Building

**53.** Story

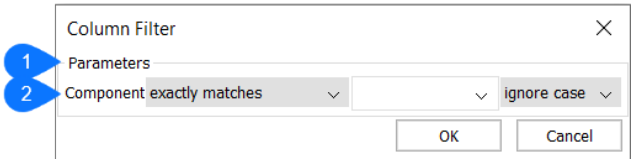
**Space**
BIM Space entities from enclosed boundaries (3D solids or linear 2D entities).

**Building**
Buildings in the model.

**Story**
Story of the building / spatial elements.

### 4.12.31 Column filter dialog box

The Column Filter dialog box configures the column filter.




1 Parameters

2 Column

#### 4.12.31.1 Parameters

Defines the parameters to use in the column filter.

#### 4.12.31.2 Column type

Displays the name of the column you put a filter on. Selects the type of filter that should be applied to this column and configures it.

***Прим.:*** The list of possible filters depends on the column content type.

***Прим.:*** For all text filters, you can also specify whether they should be case-sensitive or case-insensitive.

**Always accepted**
All column values are accepted.

**Equal to**
Value should be equal to the target one.

***Прим.:*** Appears with numerical columns.

**Not equal to**
Value should be not equal to the target one.

***Прим.:*** Appears with numerical columns.

**Less than**
Value should be less than the target one.

***Прим.:*** Appears with numerical columns.

**Greater than**
Value should be greater than the target one.

***Прим.:*** Appears with numerical columns.

**Less than or equal to**
Value should be less than or equal to the target one.

***Прим.:*** Appears with numerical columns.

**Between**
Value should be in the given range, both ends excluded.

***Прим.:*** Appears with numerical columns.

**Not in range**
Value should not be in the given range, both ends excluded.

***Прим.:*** Appears with numerical columns.

**Exactly matches**
The content of the column should exactly match the target one.

***Прим.:*** Appears with columns containing text.

**Does not match**
The content of the column should be different from the target one.

***Прим.:*** Appears with columns containing text.

**Starts with**
The content of the column should start with the given text.

***Прим.:*** Appears with columns containing text.

**Does not start with**
The content of the column should not start with the given text.

***Прим.:*** Appears with columns containing text.

**Ends with**
The content of the column should end with the given text.

***Прим.:*** Appears with columns containing text.

**Does not end with**
The content of the column should not end with the given text.

***Прим.:*** Appears with columns containing text.

**Contains**
The content of the column should contain the given text.

***Прим.:*** Appears with columns containing text.

**Does not contain**
The content of the column should not contain the given text.

***Прим.:*** Appears with columns containing text.

**Matches pattern**
The content of the column should match the given wildcard template.

***Прим.:*** Appears with columns containing text.

**Does not match pattern**
The content of the column should not match the given wildcard template.

***Прим.:*** Appears with columns containing text.

**Included in**
The content of the column should be in the given set of values.

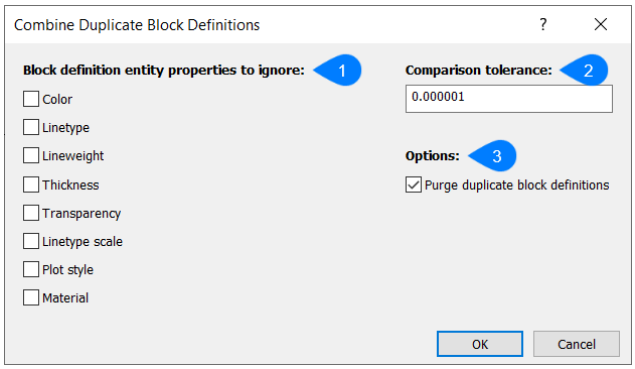
***Прим.:*** Appears with columns containing text and numerical columns.

**Not included in**
The content of the column should not be in the given set of values.

***Прим.:*** Appears with columns containing text and numerical columns.

### 4.12.32 Combine Duplicate Block Definitions dialog box

The Combine Duplicate Block Definitions dialog box allows you to search the drawing for duplicate block definitions and replace all such block instances by the most recent of these duplicates.




1 Block definition entity properties to ignore

2 Comparison tolerance

3 Options

#### 4.12.32.1 Block definition entity properties to ignore

Specifies the block properties to ignore when searching for duplicate block definitions.

#### 4.12.32.2 Comparison tolerance

Sets the tolerance for overlapping blocks. If set to 0 (zero), entities must match completely before being evaluated in the overkill process.

#### 4.12.32.3 Options

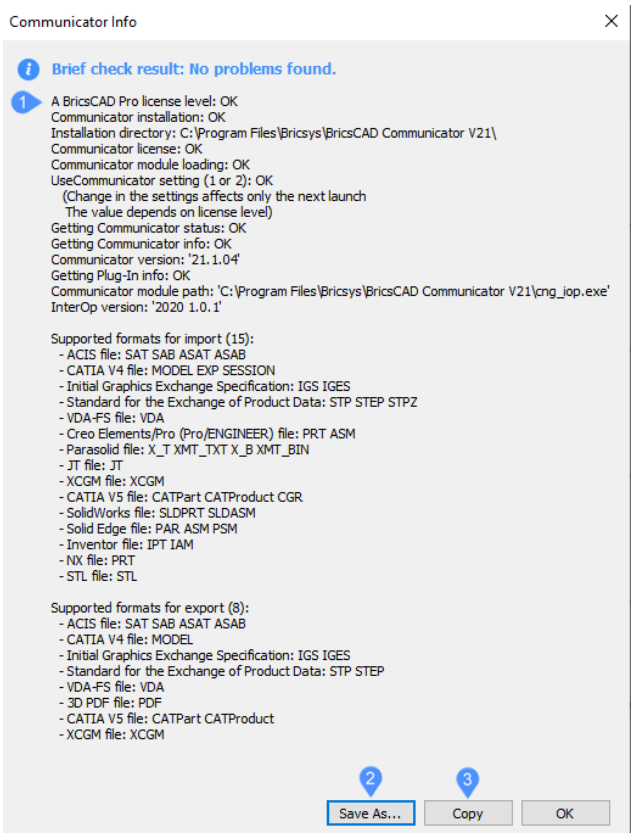
**Purge duplicate block definitions**

Deletes duplicate block definitions.

### 4.12.33 Communicator info dialog box

The Communicator Info dialog box checks whether the Communicator add-on is correctly installed and provides a diagnostic report.

The command is available on Windows only.




1 Diagnostic

2 Save As...

3 Copy

### 4.12.33.1 Diagnostic

Displays a diagnostic report for the Communicator add-on installation, supported formats to import and to export.

### 4.12.33.2 Save As...

Opens the Save Communicator Info As dialog box.

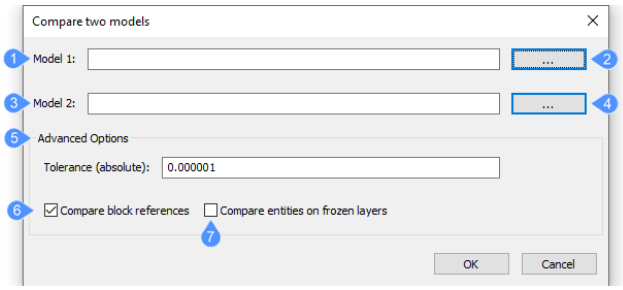
### 4.12.33.3 Copy

Copies the text displayed in this dialog box to clipboard.

## 4.12.34 Compare two models dialog box

The Compare Two Models dialog box allows you to perform a geometric comparison of solids and surfaces between two drawing files.

The result is shown in a new drawing and reported in the Mechanical Browser panel.




1 Model 1

2 Select 1st drawing

3 Model 2

4 Select 2nd drawing

5 Advanced Options

6 Compare block reference

7 Compare entities on frozen layers

### 4.12.34.1 Model 1

Specifies the file name of the first drawing to compare.

### 4.12.34.2 Select 1st drawing

Opens the Select first drawing for comparison dialog box to choose the drawing you want.

### 4.12.34.3 Model 2

Specifies the file name of the second drawing to compare.

### 4.12.34.4 Select 2nd drawing

Opens the Select second drawing for comparison dialog box to choose the drawing you want.

### 4.12.34.5 Advanced Options

The tolerance (absolute) expresses the threshold value for geometry comparison: determines how close two models must be to be considered unchanged.

### 4.12.34.6 Compare block reference

Determines if the content of blocks should be compared:

- **Yes** - compares solids and surfaces in blocks.
- **No** - ignores solids and surfaces in blocks.

This command works with regular blocks and inserts of arbitrary depth, which contain solids or surfaces, including assemblies that use block structures.

### 4.12.34.7 Compare entities on frozen layers

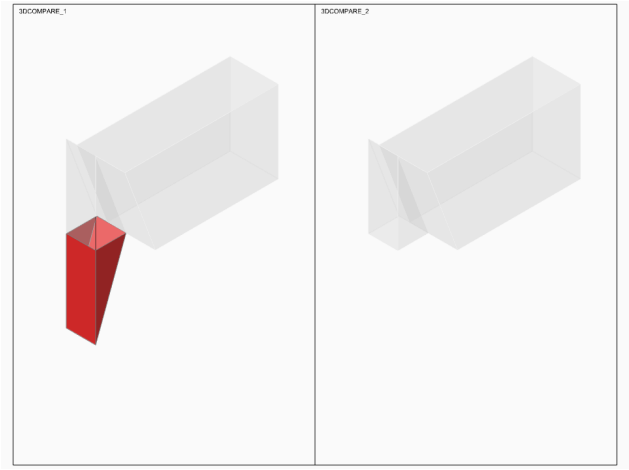
Determines if entities on frozen layers should be compared:

- **Yes** - solids and surfaces on frozen layers are compared.
- **No** - solids and surfaces on frozen layers are ignored.

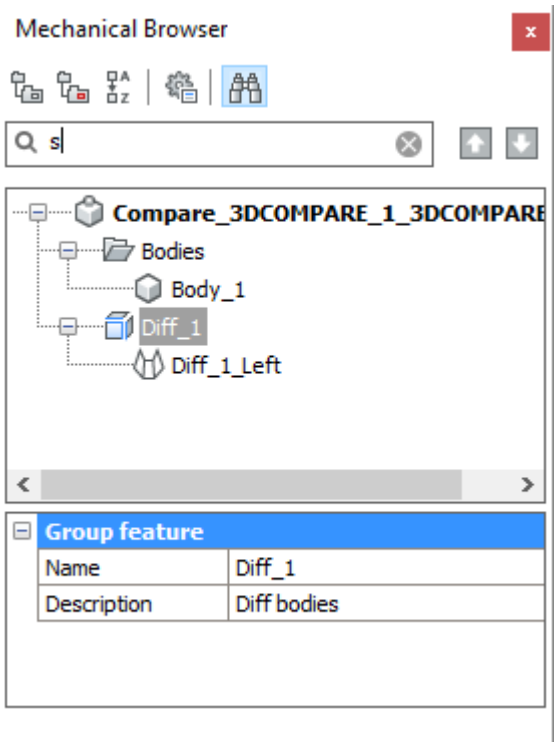
The value of the *3DCOMPAREMODE* system variable controls the display in the viewports of the Comparison layout; in the Settings dialog box, search for "3dcomparemode".

The selected files are attached to a new drawing that is named Compare_<File_1>_<File_2>.dwg

In this drawing, a layout named “Comparison” is created automatically with two viewports, one for each drawing, illustrating the differences between the drawing using colors.

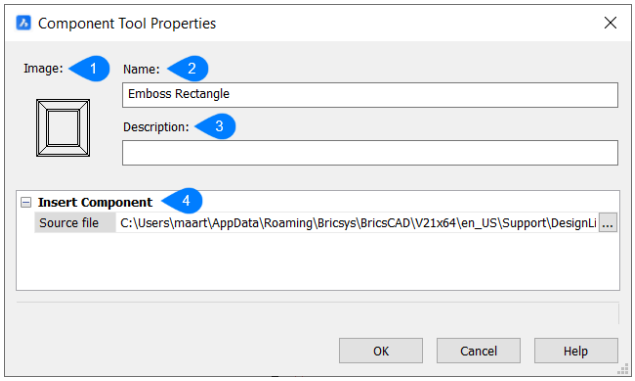



At the same time, the Mechanical Browser panel opens automatically and lists the differences between the 2 drawings.




### 4.12.35 Component Tool Properties

The Component Tool Properties dialog box allows you to view and change the basic properties of components in the tool palettes panel.




1 Image

2 Name

3 Description

4 Insert Component

#### 4.12.35.1 Image

Displays the image that is used as the icon for the selected component in the tool palettes panel.

### 4.12.35.2 Name

Change the name of the selected component.

### 4.12.35.3 Description

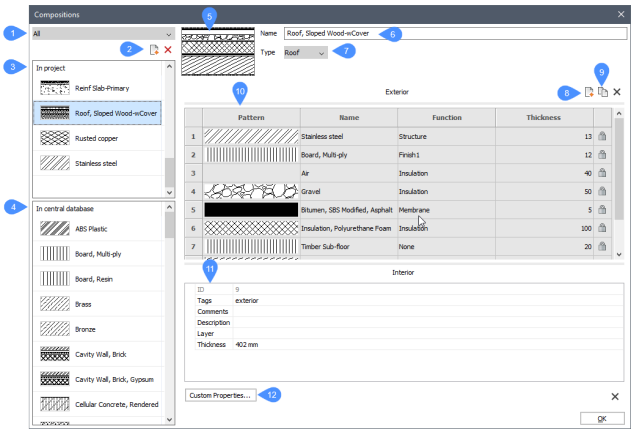
Provide an optional description of the component.

### 4.12.35.4 Insert Component

The browse icon opens the Choose a File dialog box to allow you to select the source file of the component.

## 4.12.36 Compositions dialog box

The Compositions dialog box allows you to create, modify and delete compositions.




1 Composition category selection

2 New composition

3 In project

4 In library

5 Preview pane

6 Name

7 Type

8 Add ply

9 Duplicate ply

10 Structure grid

11 Properties grid

12 Custom properties

### 4.12.36.1 Composition category selection

Composition categories are used to filter the list of the compositions.

Click the drop-down button and choose one of the available categories: All, Generic, Roof, Slab, and Wall.

### 4.12.36.2 New composition

Click this button to create a new composition.

### 4.12.36.3 In project

Displays the compositions of the selected category in the Project database.

Right-click on a composition to add a new composition, duplicate, delete or copy to library.

***Прим.:*** Compositions that are used in the project cannot be deleted from the project database.

### 4.12.36.4 In library

Displays the compositions of the selected category in the Library database.

Right-click on a composition to add a new composition, duplicate, delete or copy to project.

### 4.12.36.5 Preview pane

Displays the preview of the selected composition.

### 4.12.36.6 Name

Displays the selected composition name. You can change the default name.

### 4.12.36.7 Type

Displays the category for the selected composition. You can click the drop-down button and select a new category. The available categories are: Generic, Roof, Slab, and Wall.

### 4.12.36.8 Add ply

Opens the Physical Materials dialog box which allows you to choose a material from the library or project database.

### 4.12.36.9 Duplicate ply

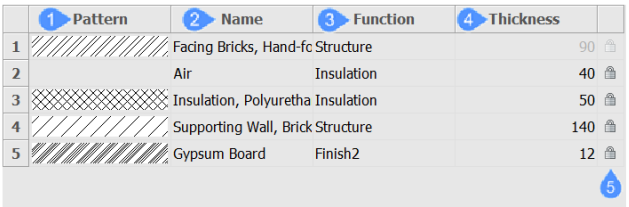
Inserts a copy of the selected ply.

### 4.12.36.1 Structure grid

**0**

Displays the material and thickness of the composition plies. The ply on top (exterior) is applied to the reference of the solid.

The plies of a composition are ordered from Exterior (top) to Interior (bottom). You can drag the ply number on the desired position to change the ply order.




1 Pattern

2 Name

3 Function

4 Thickness

5 Lock/Unlock thickness

**Pattern**
Displays a preview of the ply. You can double click on the Pattern field and change its appearance from the Physical Materials dialog box.

**Name**
Displays the name of the ply. You can double click on the Name field and change it from the Physical Materials dialog box.

**Function**
You can select a function from the drop-down list. The available functions are: None, Structure, Substrate, Insulation, Finish1, Finish2, and Membrane.

**Thickness**
Click on the Thickness field to modify the thickness of a ply.

***Прим.:*** You can only modify the ply thickness if the Variable Thickness property in the Physical Materials dialog box is set to Yes.

**Lock/Unlock thickness**
You can click the Lock/Unlock button to change its status.

***Прим.:*** Only one ply in a composition can have an unlocked thickness. As a result, the total thickness of a composition can be:

- **Fixed**: all plies have locked thicknesses.
- **Minimal**: the composition contains at least two plies and one ply has an unlocked thickness. The minimum thickness is equal to the sum of the thicknesses of the locked plies.
- **Free**: the composition contains a single ply that has an unlocked thickness.

***Прим.:*** The thickness of a ply can only be unlocked if the Variable Thickness property in the Physical Materials dialog box is set to Yes.

### 4.12.36.1 Properties grid

**1**
Displays the properties of the selected composition like tags, comments, description, layer, thickness and custom.

**Tags**
Select the Tags field and type new tags, separated by commas. Tags are case sensitive.
Click the Browse button next to the tags field to open the Tags dialog box.

**Comments**
Optionally, you can add comments to the selected composition.

**Description**
Optionally, you can add a description to the selected composition.

**Layer**
Optionally, you can assign a layer from the drop-down list to the selected composition.

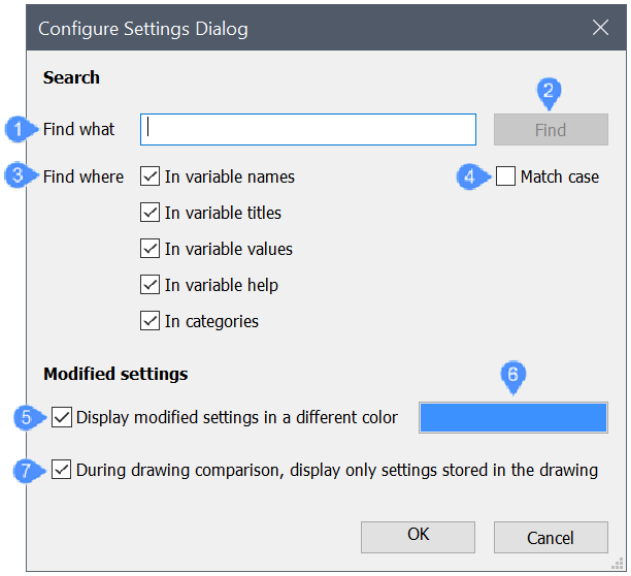
**Thickness**
Displays the overall thickness of a composition, which is equal to the sum of the thicknesses of the composing plies.

### 4.12.36.1 Custom properties

**2**
Opens the Select/Create a Custom Property dialog box. You can add custom properties for compositions.

## 4.12.37 Configure Settings dialog box

The Configure Settings dialog box allows you to control how BricsCAD searches for variables and how the differences are handled.




1 Find what

2 Find

3 Find where

4 Match case

5 Display modified settings in a different color

6 Select color

7 During drawing comparison, display only settings stored in the drawing

### 4.12.37.1 Find what

Type the name of the setting you are looking for.

### 4.12.37.2 Find

Searches in the Settings dialog box for the word you have typed.

#### 4.12.37.3 Find where

Determines which part of the Settings dialog box should be searched.

**In variable names**
Thick this checkbox to search in variable names.

**In variable titles**
Thick this checkbox to search in variable titles.

**In variable values**
Thick this checkbox to search in variable values.

**In variable help**
Thick this checkbox to search in variable help.

**In categories**
Thick this checkbox to search in categories.

#### 4.12.37.4 Match case

Thick this checkbox to specify that the capitalization of the searched phrase must match that of the words in the Settings dialog box.

#### 4.12.37.5 Display modified settings in a different color

Thick this checkbox to see the modified settings in a different color.

#### 4.12.37.6 Select color

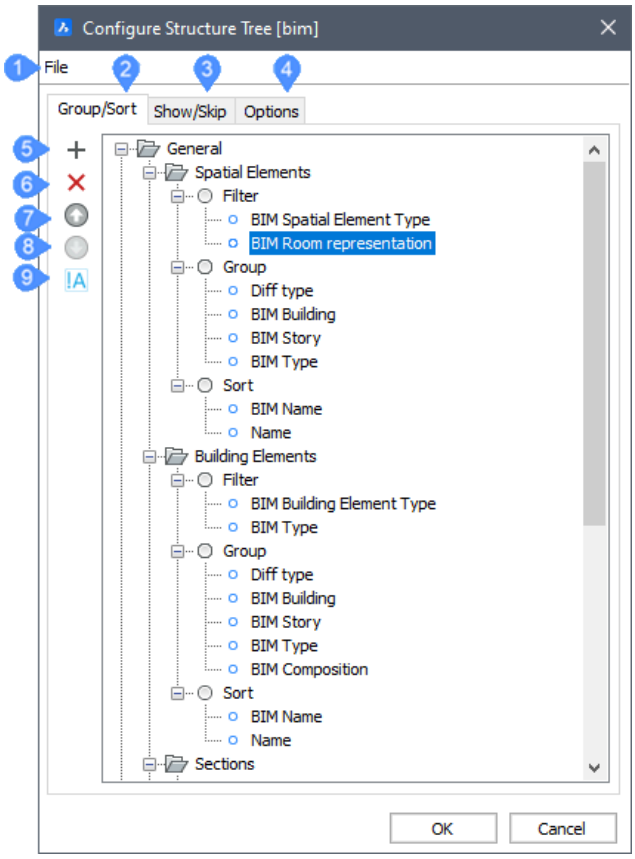
Click this button to select another color.

#### 4.12.37.7 During drawing comparison, display only settings stored in the drawing

Thick this checkbox to see the differences for settings stored in the current drawing and another drawing that you open by running the *DWGCOMPARE* command.

### 4.12.38 Configure structure tree dialog box

The Configure structure tree dialog box allows you to define the tree structure of the Structural panel.




1 File

2 Group/Sort Tab

3 Show/Skip

4 Options

5 Add

6 Delete

7 Move Up

8 Move Down

9 Toggle negation

### 4.12.38.1 File

Click the File menu in the dialog box and choose one of the available options on the list:

- Open...: opens the Select Structure Tree Configuration File dialog box to select another *.cst configuration file.
- Save: saves the current configuration and keep the file dialog open.
- Save as: saves the current configuration under a different name and keep the file dialog open.

### 4.12.38.2 Group/Sort Tab

**Add (+) (5)**
Adds a new rule.

- Select an existing rule, then click the Add icon (+) or right-click and choose Add Rule in the context menu.
- The new rule is added below the selected rule.
- Select the newly created Rule, then click the name or right-click and choose Rename in the context menu and type a new name for the rule.
- Select the Filter node of the rule, then click the Add icon (+) or right-click and choose Add filter property in the context menu.
- Select the Group node of the rule, then click the Add icon (+) or right-click and choose Add Grouping Property in the context menu.
- Select the Sort node of the rule, then click the Add icon (+) or right-click and choose Add sorting property in the context menu.

The Select Property dialog box displays.

- Select a property, then double-click or click the OK button.
- Optionally, type a search string in the Search field, then click the Up (↑) or Down (↓) arrow buttons to look up the properties containing the search string.
- Repeat the previous step to add more filter / grouping / sorting properties.

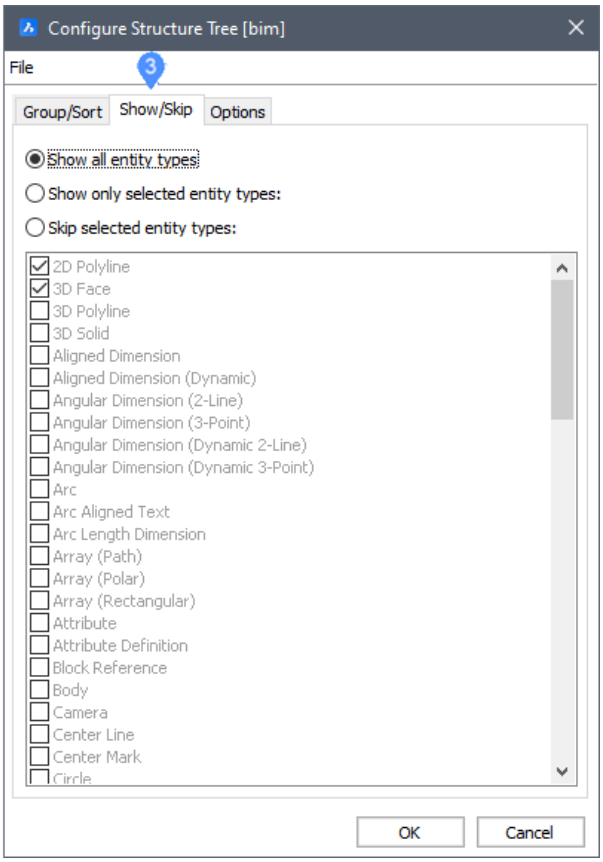
**Delete (×) (6)**
Deletes a selected rule or property.

**Move Up (↑) (7)**
Moves Up a selected rule or property.

**Move Down (↓) (8)**
Moves Down a selected rule or property.

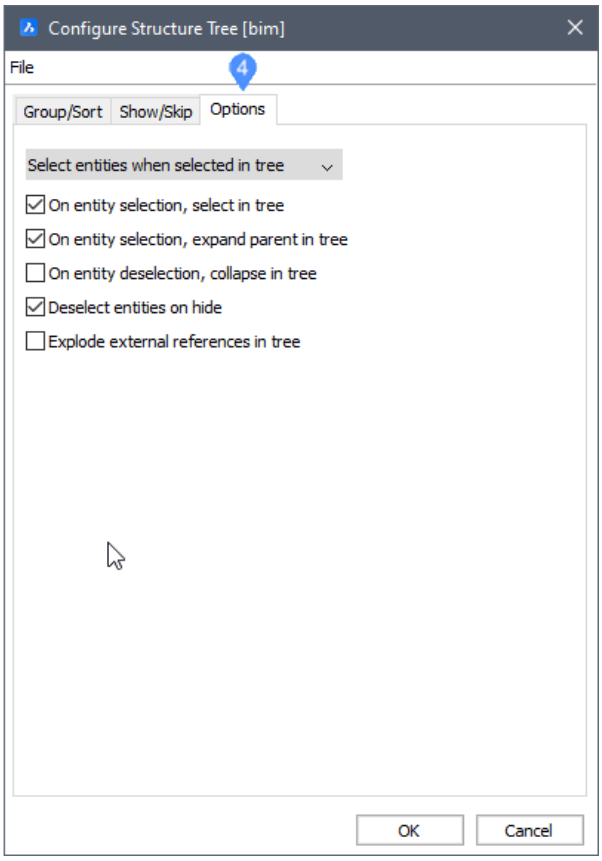
**Toggle negation (9)**
Toggle negation only for filter properties.

### 4.12.38.3 Show/Skip Tab




Select an option and then select the entity types to show or to exclude.

### 4.12.38.4 Options Tab

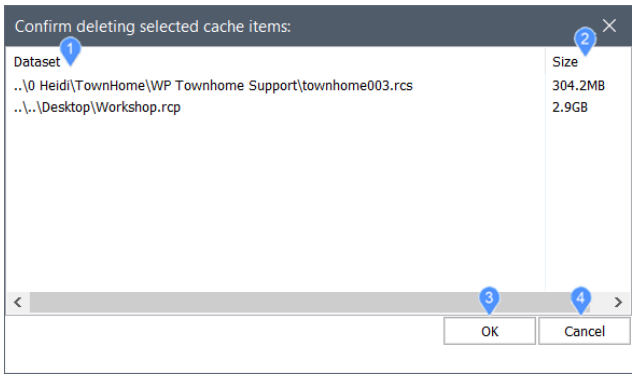



- Choose what happens in the model when entities are selected in the structure tree.
- Select what happens in the structure tree when an entity is selected/deselected in the model.

### 4.12.39 Confirm deleting selected cache items

TheConfirm deleting selected cache items dialog box enables you to review a list of cached point cloud files that were selected to be deleted.

This gives you a final opportunity to cancel the operation before permanently removing them from the cache.




1 Dataset

2 Size

3 OK

4 Cancel

### 4.12.39.1 Dataset

Displays a list of cached point cloud files that you selected to delete from the cache. It includes their names and locations.

### 4.12.39.2 Size

Displays the amount of cache used for each of the selected point cloud files.

### 4.12.39.3 OK

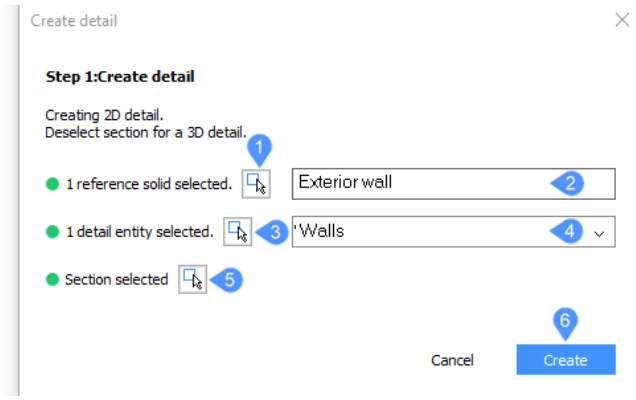
Deletes the point cloud files from the cache. Any drawings that reference these files will be unable to find them.

### 4.12.39.4 Cancel

Closes the dialog box without deleting point cloud files from the cache.

## 4.12.40 Create detail dialog box

The Create Detail dialog box allows you to create details and save them in the library.




1 Pick reference solids

2 Detail name

3 Pick detail entities

4 Create or choose category

5 Pick section

6 Create

#### 4.12.40.1 Pick reference solids

Select reference solids, press Enter to display the dialog box and continue the procedure. These will not be copied but used to match other locations.

#### 4.12.40.2 Detail name

Type the name you want to give to the detail.

#### 4.12.40.3 Pick detail entities

Select detail objects (such as solids, block references, faces and edges) to be copied.

#### 4.12.40.4 Create or choose category

Select a category in the drop-down list or type a name to create a new category.

#### 4.12.40.5 Pick section

Select a BIM section.

Detail categories are subfolders of the Details folder, which is defined by the DETAILSPATH system variable. By default, this is *C:\ProgramData\Bricsys\Details*.

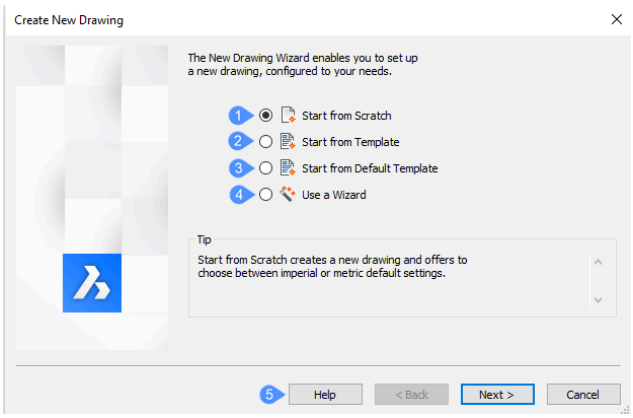
#### 4.12.40.6 Create

Accept/Reject the changes or inflate first.

Displays the Save Detail dialog box, where you will add tags to the details in order to find it easier in the Details panel.

### 4.12.41 Create new drawing dialog box

The Create New Drawing dialog box allows you to create new drawings by several methods with the New Drawing Wizard.




1 Start from Scratch

2 Start from Template

3 Start from Default Template

4 Use a Wizard

5 Help

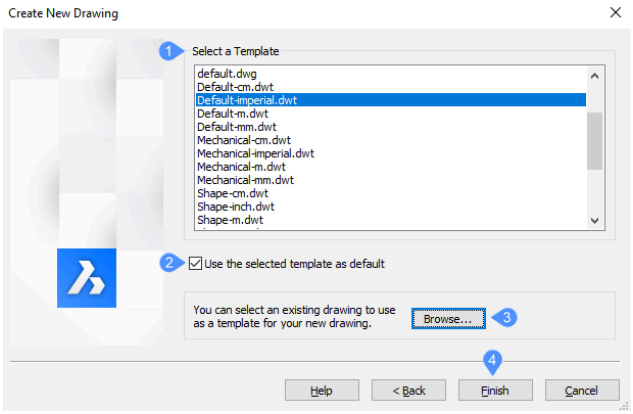
### 4.12.41.1 Start from Scratch

Starts the new drawing using the default template file specified by the BASEFILE system variable. Click the **Next** button and choose a measurement system:

- **Imperial (feet and inches)** - new drawing is based on the default-imperial.dwt file and so uses Imperial measurements.
- **Metric** - new drawing is based on the default-mm.dwt file and so uses metric measurements.

After you click the **Finish** button, the program opens a new drawing whose content depends on the default template file.

### 4.12.41.2 Start from Template




1 Select a Template

2 Use default template

3 Browse...

4 Finish

**Select a Template**

Allows you to choose a file to act as the template. From the Template list, choose one of the DWT file names. These files are stored in the folder specified by the TemplatePath variable; the default location is *C:\Users\useid\AppData\Local\Bricsys\BricsCAD\V19x64\en_US\Templates*.

**Use default template**

Turn on **Use the selected template as default** option when you want to use this file as the template from now on. If so, then the program updates the value of the BaseFile variable. After you click the **Finish** button, the program opens a new drawing whose content depends on the template file you have selected.

**Browse...**

Opens the Select a template file dialog box from where you can select the following kind of file type for a template: DWG - drawing files, DWT - template files, DXF - drawing interchange files.

**Finish**

Click the Finish button after selecting a Template from the list or from the dialog box.

### 4.12.41.3 Start from Default Template

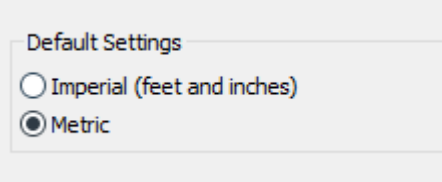
Starts new drawings with the default template defined by the BaseFile variable. After you click the **Finish** button, the program opens a new drawing whose content depends on the template file you have selected.

### 4.12.41.4 Use a Wizard

Starts the new drawing by stepping through a wizard that guides you along the way:

1 Click the **Next** button.

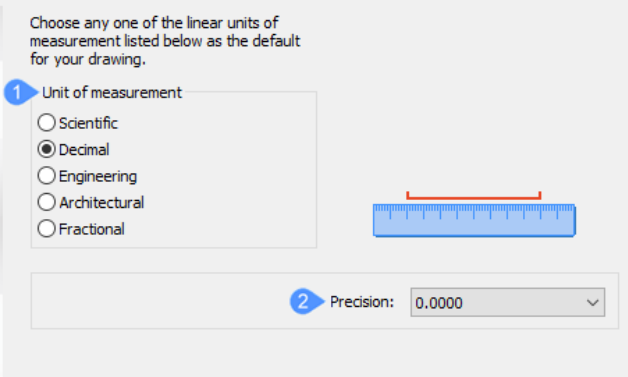
2 Choose a measurement system:




- Imperial (feet and inches) - new drawing is based on the default-imperial.dwt file and so uses Imperial measurements.
- Metric - new drawing is based on the default-mm.dwt file and so uses metric measurements.

3 Click the **Next** button.

4 Select a linear unit of measurement and display precision.




a Unit of Measurement

b Precision

**Unit of Measurement**

Choose one from the list. If you are unsure, choose Decimal; you can always change the units any time later with the UNITS command.

| Unit Name | Measures | Example Display |
|---|---|---|
| Architectural | Feet, inches, fractional inches | 4'-6 1/16" |
| Engineering | Feet, inches, decimal inches | 4'-6.0625" |
| Fractional | Inches, fractional inches | 54 1/6" |
| Decimal | (default) Units and decimal units | 2128.4449 |
| Scientific | Units, decimal units, exponents | 2.1284E+03 |

**Precision**

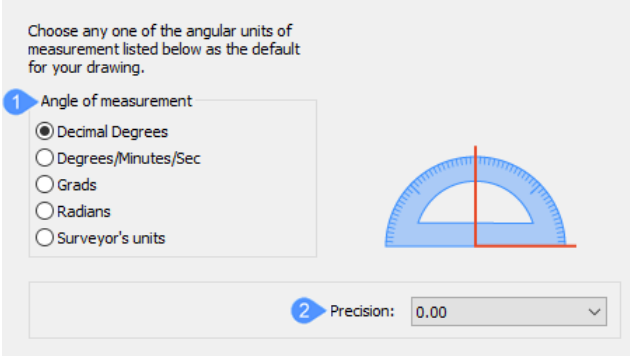
Click the drop-down list and choose a level of precision. If you are unsure, choose the defaults, as you can always change the precision any time later with the Units command.

***Прим.:*** This is a display precision: the program makes calculations internally always to eight decimal places.

***Прим.:***

1 Click the **Next** button.

2 Select an angular unit of measurement and display precision.




a Angle of measurement

b Precision

**Angle of Measurement**

Choose one form the list. If you are unsure, choose Decimal/Degrees; you can always change the units any time later with the Units command.

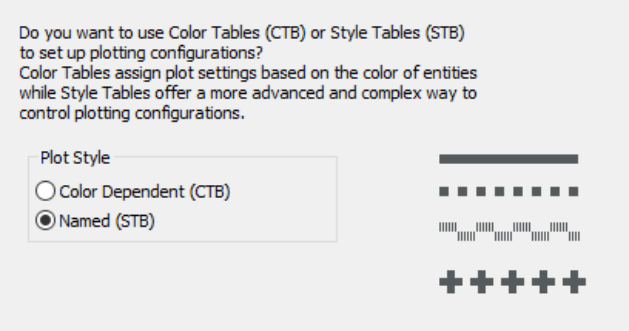
| Angle Name | Measures In | Example |
|---|---|---|
| Decimal degrees | Degrees and decimals | 12.3456 |
| Degrees/Minutes/Seconds | Degrees,minutes,seconds,decimal secs | 12d34'56" |
| Grads (g) | Grads and decimals | 123.45g |
| Radians (r) | Radians and decimals | 1.23r |
| Surveyor's Units | North/South degrees towards East/West | N12d34'56"E |

**Precision**

Click the drop-down list and choose a level of precision. If you are unsure, choose the defaults, as you can always change the precision any time later with the Units command.

1 Click the **Next** button.

2 Determine which kinds of plot style the drawing should employ:

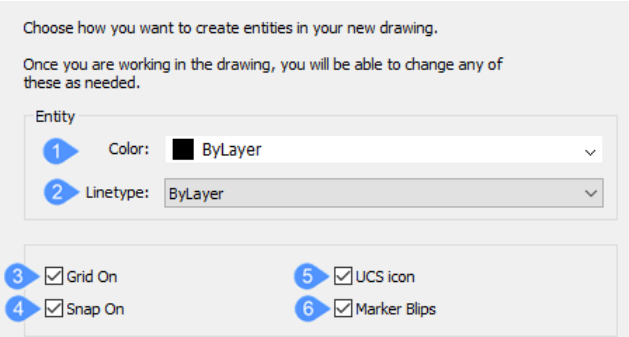



- **Color Dependent (CTB)** - these plot styles are simpler to employ, but less flexible.
- **Named (STB)** - plot styles are more flexible, but complex, because they control every aspect of every entity property during plotting.

If you are unsure, choose Color-dependent.

3 Click the **Next** button.

4 Choose the default working color, the linetype and the visual aids you want to be displayed by the drawing:




a Color
b Linetype
c Grid On
d Snap On
e UCS icon
f Marker Blips

**Color**

You can choose any color from the drop-down list. However, this is not recommended. Choose ByLayer so that layers control the colors of entities. See COLOR and LAYER commands.

**Linetype**

You can choose any linetype from the drop-down list, this again is not recommended. Choose ByLayer, for the same reason. See LINETYPE command.

**Grid On**

When on, displays a grid of dots (or lines) that helps you see the unit spacing, as well as drawing extents, such as every 1m; see the GRID command. It is recommended you turn this on.

**Snap On**
When on, sets the movement increment of the cursor, such as every 0.5m; see SNAP command. It is recommended you turn this on.

**UCS icon**
When on, displays the UCS icon for showing the orientation of the x, y, and z axes; see the UCSICON command. It is recommended that you turn this on for 3D drawings, but turn it off for 2D ones.

**Marker Blips**
When on, draws a tiny + each time you click in the drawing; see the BLIPMODE command. It is recommended you leave this one off, as it can turn the screen messy.

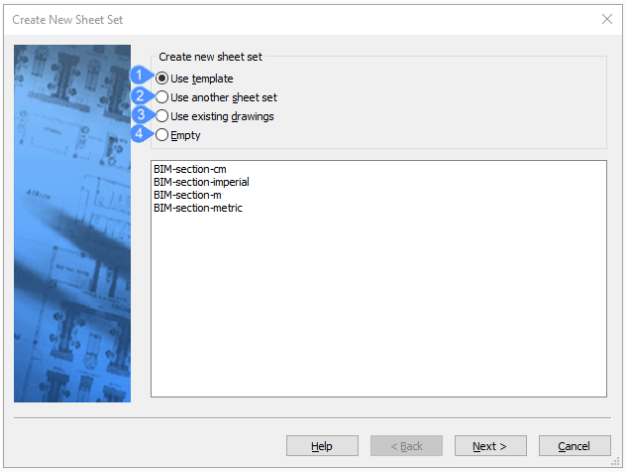
After you click the **Finish** button, the program opens a new drawing whose content depends on the default template file and the options you choose during the wizard.

### 4.12.41.5 Help

Opens the Bricsys Help article about the NEWWIZ command.

## 4.12.42 Create new sheet set dialog box

The Create New Sheet Set dialog box provides 4 methods to create a new sheet set.




1 Use template

2 Use another sheet set

3 Use existing drawings

4 Empty

### 4.12.42.1 Use template

**Use template**
Create a new sheet set from a template.

Sheet set templates are stored as DST files in the folder set by the SheetSetTemplatePath preference (default path is *C:\Users\<UserName>\AppData\Local\Bricsys\BricsCAD\V[Version]\en_US\Templates*)

1 Select a template sheet set from the list shown in the dialog box.

2 Click **Next**.

**Save sheet set file as**

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

**Description**

1 Optional, type a description for the sheet set file.

2 Click **Next**.

**Sheet Set Preview**

Review the summary of the sheet set that will be created, and then click Finish to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

### 4.12.42.2 Use another sheet set

Creates a new sheet set from the settings in an existing sheet set.

**Select sheetset to copy**

1 Click **Browse** and select an existing sheetset file from Select a Sheet Set file dialog box.

2 Click **Next**.

**Save sheet set file as**

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

***Важное замечание:*** You can not progress until you select a folder that stores the new file.

**Description**

1 Optional, type a description for the sheet set file.

2 Click **Next**.

**Sheet Set Preview**

Review the summary of the sheet set that will be created, and then click Finish to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

### 4.12.42.3 Use existing drawings

Create a new sheet set from an existing set of drawings.

1 Select **Use existing drawings**.

2 Click **Next**.

**Select Folder**

1 Select a folder from the Choose a folder dialog box where the new sheet set will be stored.

***Прим.:*** Notice that the dialog box fills with names of drawings and their layouts.

2 Choose among the options (it is recommended to keep both options turned on).

**Create subsets from folder structure**

Uses the folders in the main folder to determine how the sheet set is structured.

**Prefix sheet titles with drawing file names**

1 Add the drawing file's name to the start of the sheet set titles, which makes it easier to track the source of the files.

2 Click **Next**.

**Save sheet set file as**

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

***Важное замечание:*** You can not progress until you select a folder.

**Description**

Optional, type a description for the sheet set file.

**Default drawing template for new sheets**

1 Select a template sheet set from the Select Drawing Template dialog box.

2 Click **Next**.

**Sheet Set Preview**

Review the summary of the sheet set that will be created, and then click Finish to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

#### 4.12.42.4 Empty

Creates a new sheet set from scratch.

1 Select **Empty**.

2 Click **Next**.

**Save sheet set file as**

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

***Важное замечание:*** You can not progress until you select a folder.

**Description**

Optionally, type a description for the sheet set file.

**Default drawing template for new sheets**

1 Select a template sheet set from the *Select Drawing Template* dialog box.

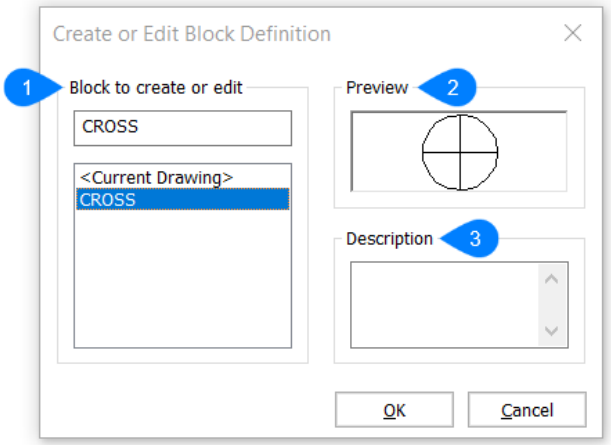
2 Click **Next**.

**Sheet Set Preview**

Review the summary of the sheet set that will be created, and then click Finish to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

### 4.12.43 Create or edit block definition dialog box

The Create or Edit Block Definition dialog box allows you to select an existing block definition to edit, or to enter a new name to create a new block definition.




1 Block to create or edit

2 Preview

3 Description

### 4.12.43.1 Block to create or edit

Specifies the name of the block you want to create or edit. To edit a block, you can select a block from the list below with the blocks available in the drawing.

### 4.12.43.2 Preview

Provides a preview of the selected block.

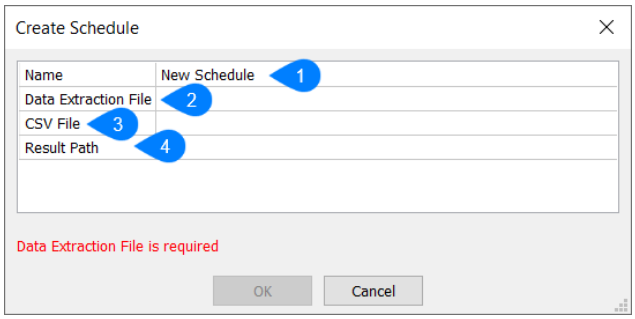
### 4.12.43.3 Description

Displays the description of the selected block.

Click **OK**, to open the Block Editor.

## 4.12.44 Create Schedule dialog box

Create a new schedule in the project browser.




1 Name

2 Data Extraction File

3 CSV File

4 Result Path

### 4.12.44.1 Name

Specifies the name of the new schedule.

### 4.12.44.2 Data Extraction File

Selects the data extraction file that will be used. Opens the Select Data Extraction Definition file dialog box where you can select a file.

### 4.12.44.3 CSV File

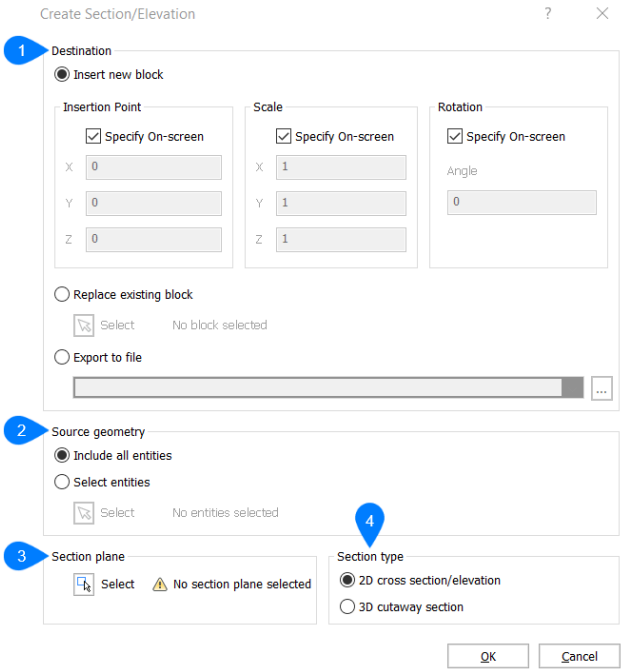
Defines the CSV file that will be used. Opens the Select Delimiter-separated format file dialog box where you can select a file.

### 4.12.44.4 Result Path

Defines the path where the resulting drawing will be saved. Opens the Select Drawing file dialog box where you can select a drawing to save the result.

## 4.12.45 Create section/elevation dialog box

The Create Section/Elevation dialog box saves section planes as blocks inserted in the current drawing or saved to disk as DWG files.




1 Destination

2 Source geometry

3 Section plane

4 Section type

### 4.12.45.1 Destination

Defines where the block will be placed in the drawing.

**Insert new block**
Inserts the created section as a new block in the drawing. The insertion point, scale and rotation can be specified.

**Insertion point**
Allows to specify the insertion point in the model space after you click the OK button, if the Specify on-screen button is checked.

**X/Y/Z**
Defines the coordinates where the block should be inserted, only available if the Specify on-screen button is checked off.

**Scale**
Allows to specify the scale factor in the model space after you click the OK button, if the Specify on-screen button is checked.

**X/Y/Z**
Defines the individual X,Y,Z direction scale factors, only available if the Specify on-screen button is checked off.

**Rotation**
Allows to specify the rotation angle in the model space after you click the OK button, if the Specify on-screen button is checked.

**Angle**
Defines the rotation angle of the inserted section/elevation.

**Replace existing block**
Replaces an existing block in the drawing. This block will be replaced by the created section

**Select**
Allows to select the block to replace in the model space.

**Export to file**
Opens the Select export file dialog box. Here you can select a file to export the created section to.

### 4.12.45.2 Source geometry

Specifies the entities that will be included from the drawing.

**Include all entities**
Selects all 3D geometry in the drawing (includes 3D solids, 3D surfaces, polyface meshes, and 3D faces).

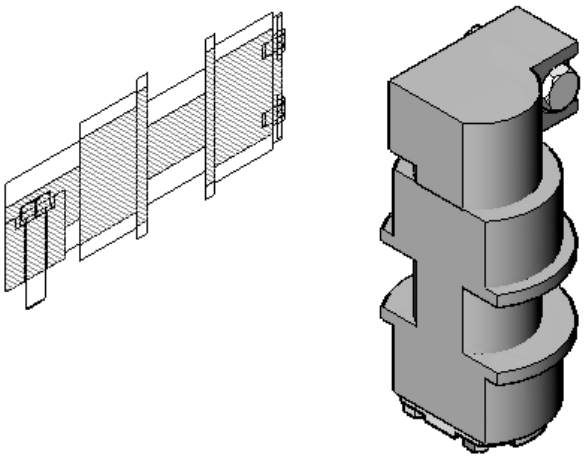
**Select entities**
Allows to select the entities to be included in the model space.

### 4.12.45.3 Section plane

Selects the section plane to be converted to a block.

### 4.12.45.4 Section type

Determines the kind of section plane block to create.

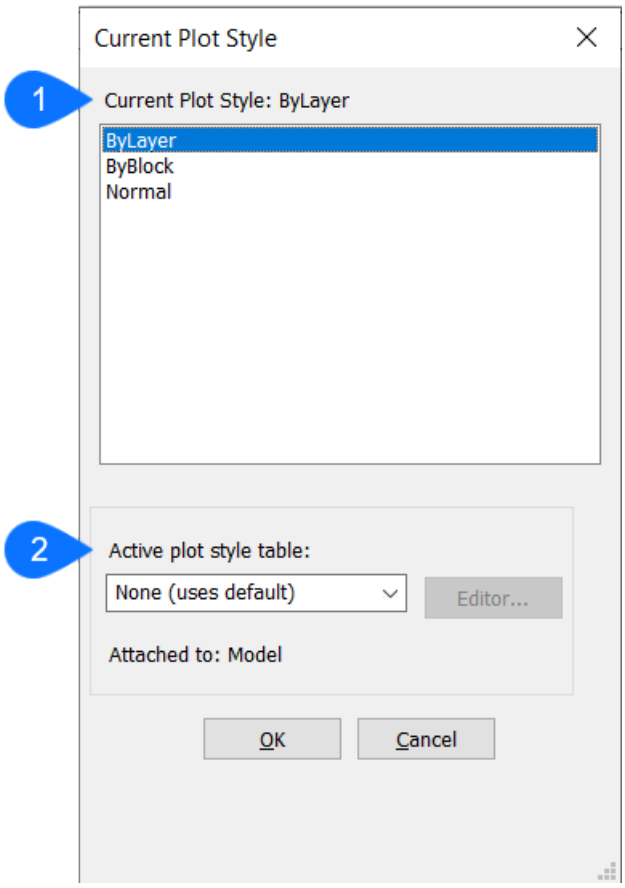


**2D cross section/elevation**
Uses the 2D section settings defined by the SectionPlaneSettings dialog box to create the block.

**3D cutaway section**
Uses the 3D section settings defined by the SectionPlaneSettings dialog box to create the block.

### 4.12.46 Current Plot Style dialog box

Sets the current plot style.




1 Current Plot Style

2 Active plot style table

### 4.12.46.1 Current Plot Style

Sets the current plot style for the drawing. You can choose between:

- **ByLayer**: uses the plot style of the layer of the entity.
- **ByBlock**: uses the plot style of the block the entity is assigned to.
- **Normal**: uses the normal plot style. The default properties of the entity are used.

### 4.12.46.2 Active plot style table

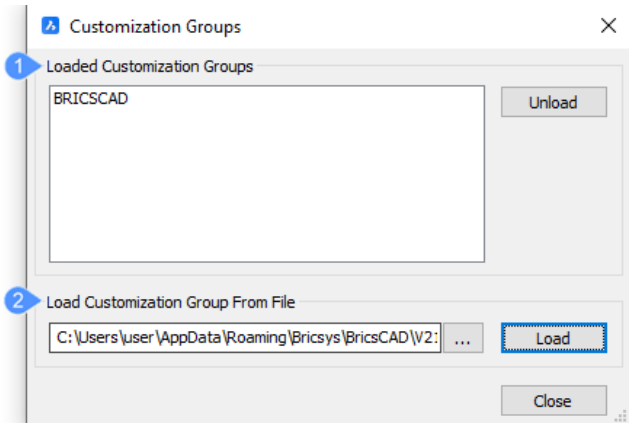
Lists the names of STB files (short for "style table"). To edit the chosen plot style table, click 'Editor...'. The Plot Style Table Editor dialog box will be displayed.

## 4.12.47 Customization groups dialog box

The **Customizations Groups** Dialog Box allows you to load or to unload a partial CUI file.

When you install BricsCAD the default CUI files are copied to the **Support** folder of your Roamable root folder. (e.g. C:\Users\<user_name>\AppData\Roaming\Bricsys\Bricscad\Vxx.x\en_US\Support).

When you launch BricsCAD the first time the default.cui file is loaded automatically.




1 Loaded Customization Groups

2 Load Customization Group from File

### 4.12.47.1 Loaded Customization Groups

A list with the names of the customization files already loaded is displayed.

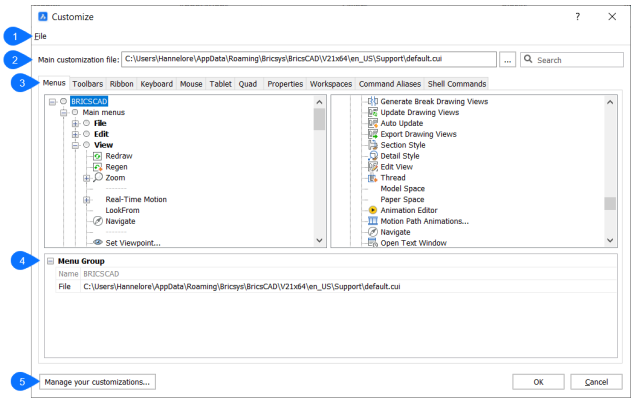
A CUI file can be unloaded by selecting it from the list and clicking the **Unload** button.

### 4.12.47.2 Load Customization Group from File

Click the browse button to select a partial CUI file and click the **Load** button to load it.

## 4.12.48 Customize dialog box

The Customize dialog box allows you to customize the user interface of BricsCAD.




1 File

2 Main customization file

3 Tab options

4 Menu Group

5 Manage your customizations

### 4.12.48.1 File

Shows a menu to select customization files or import workspaces.

**Load main CUI file...**
Opens the Select main CUI file dialog box to select a customization file. See the CUILOAD command.

**Save main CUI file as...**
Opens the Save main CUI file as dialog box to save the current main CUI file.

***Прим.:*** You can save the main CUI file under a different name to copy the adapted main CUI file.

**Load Partial CUI file...**
Opens the Choose a customization file dialog box to open a partial CUI file.

***Прим.:*** See chapter **Loading a partial CUI file** in **Loading and unloading CUI files** for a detailed explanation about the partial CUI files.

**Create new partial CUI file...**
Opens the Create a Customization File dialog box to create an empty CUI file.

**Import workspace(s)…**
Opens the Choose a Customization File dialog box to load one or more workspaces from another CUI file.

### 4.12.48.2 Main customization file

Specifies the CUI file that defines customization of the menu, toolbar, and other elements of the user interface.

***Прим.:*** Aliases and shell commands are defined in a PGP file.

**Browse**
Opens the Select main CUI file dialog box to load a different main CUI file.

**Search**
Searches the Available Tools list for command names.

### 4.12.48.3 Tab options

**Menus**
Adds, edits, moves and deletes menu items.
When right clicking a menu, menu item or submenu in the Menus tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

***Прим.:*** If you select Append item or Insert item from the context menu, the Add menu item dialog box displays.

**Menu Group/Menu/Menu Item/Submenu**
Displays the options of the selected menu, submenu, menu item or separator.

**Title**
Specifies the name that is displayed by the menu, menu item or submenu.

***Прим.:*** You can prefix a letter with '&' to create an Alt-key shortcut. For example: &Line.

**ID**
Unique identifier for each Menu item. (The ID is assigned by BricsCAD).

**Alias**
Defines the aliases of the menu, menu item or submenu. The browse icon opens the ‘Edit Aliases’ dialog box where you can edit, create or delete aliases.

**Diesel**
Specifies the Diesel code to be employed by the menu, menu item or submenu.

**Tool ID**
Identifies the menu item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

**Help**
Specifies the help string displayed on the status bar.

**Command**
Specifies the command(s) or macros.

**Image**
Specifies the image to display for the menu, menu item or submenu. The browse icon opens the Tool Image dialog box. There are five options to select an image.

**Toolbars**
Adds, Moves and deletes toolbar items.
When right clicking a toolbar, tool, control, flyout or separator in the Toolbars tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

***Прим.:*** If you select Append tool or Insert tool from the context menu, the Add tool dialog box displays. For options description see the Add menu item dialog box.

**Menu Group/Toolbar (Button)**
Displays the options of the selected toolbar, tool, control, flyout or separator.

**Title**
Specifies the name that is displayed by the tooltip.

**ID**
Unique identifier for each Menu item. (The ID is assigned by BricsCAD).

**Alias**
Defines the aliases of the selected item. The browse icon opens the ‘Edit Aliases’ dialog box where you can edit, create or delete aliases.

**Position**

**Default Display**
Determines whether the selected item will be added to the workspace.

**Rows**
Specifies the number of rows for an undocked toolbar.

**Xval Yval**
Specifies the distance in pixels measured from the top left corner of the screen to the toolbar. It applies to undocked toolbars.
The values are taken from the X,Y options of the toolbar as they are set in the Workspace tab.

**Diesel**
Specifies the Diesel code to be employed by the selected item.

**Tool ID**
Identifies the toolbar item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

**Help**
Specifies the help string displayed on the status bar.

**Command**
Specifies the command(s) or macros.

**Image**
Specifies the image to display for the toolbar button. The browse icon opens the Tool Image dialog box. There are five options to select an image.

**Ribbon**
Manages the ribbons and/or adds panels to a ribbon tab.
When right clicking a ribbon tab or panel in the Ribbon tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

***Прим.:*** If you select Add launcher, Insert toggle button or Insert command button from the context menu, the Add ribbon dialog launcher dialog box, Add ribbon toggle button or add ribbon command button displays.

- Select available tool: assigns an existing command to the new item. If this option is selected, the other options are grayed out, except for Available tools
- Create new tool: assigns a new command/macro to the new item. If this option is selected, Available tools is grayed out.
- Toolbox: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop down list
- Title : specifies the name of the new tool
- Help: specifies the help string displayed on the status bar
- Command: specifies the command(s) or macros
- Image: specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image
- Available tools: selects an existing tool.

**Menu Group/Ribbon Tab/Ribbon Panel Reference**
Displays the options of the selected ribbon tab or panel.

**ID**
Identifies the item in the CUI file. (The ID is assigned by BricsCAD).

**Collapse**
Allows to choose if the panel will collapse automatically or if it will never collapse.

**Label**
Specifies the name of the ribbon tab or panel.

**Title**
Specifies the name that is displayed by the selected item.

**Key Tip**

**Keyboard**
Assigns different keyboard shortcuts to different commands.
When right clicking a keyboard shortcut in the Keyboard tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

***Прим.:*** If you select Append shortcut or Insert shortcut from the context menu, the Add keyboard shortcut dialog box displays.

- Select available tool: assigns an existing command to the new keyboard shortcut. If this option is selected, the other options are grayed out, except for Available tools
- Create new tool: assigns a new command/macro to the shortcut. If this option is selected, Available tools is grayed out.
- Toolbox: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop-down list.
- Title: specifies the name of the new tool
- Help: specifies the help string displayed on the status bar
- Command: specifies the command(s) or macros
- Image: specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image
- Available tools: selects an existing tool.

**Menu Group/Keyboard Shortcut**
Displays the options of the selected toolbar, tool, control, flyout or separator.

**Key**
Specifies the used keyboard shortcut.

**Tool ID**
Identifies the keyboard item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

**Help**
Specifies the help string displayed on the status bar, even though help strings are not displayed by keyboard shortcuts.

**Command**
Specifies the command(s) to associate with the keyboard shortcut.

**Image**
Specifies the image to associate with the command, even though images are not used with keyboard shortcuts.

**Mouse**
Modifies the different actions that are connected to the mouse buttons.
When right clicking a button or button group in the Mouse tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

***Прим.:*** If you select Append button or Insert button from the context menu, the Add button item dialog box displays.

**Alias**
Defines the aliases of the selected item. The browse icon opens the ‘Edit Aliases’ dialog box where you can edit, create or delete aliases.

**Button**
Specifies the used mouse button.

**Tool ID**
Identifies the menu item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

**Title**
Specifies the name that is displayed by the mouse button.

**Help**
Specifies the help string displayed on the status bar.

**Command**
Specifies the command(s) or macros.

**Image**
Specifies the image to display for the toolbar button. The browse icon opens the Tool Image dialog box. There are five options to select an image.

**Tablet**
Customizes the tablet options.
When right clicking a button or button group in the Tablet tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

***Прим.:*** When you choose Append button or Insert button from the context menu, the Add button item dialog box displays.

**Menu Group/Button Group**

**Alias**
Defines the aliases of the selected item. The browse icon opens the ‘Edit Aliases’ dialog box where you can edit, create or delete aliases.

**Quad**
Adds commands to the quad tab or move commands to other tabs.
When right clicking a quad tab or quad button in the Quad tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

***Прим.:*** If you select Append quad button or Insert from the context menu, the Add quad button dialog box displays.

- Select available tool: assigns an existing command to the new quad button. If this option is selected, the other options are grayed out, except for Available tools
- Create new tool: assigns a new command/macro to the new quad button. If this option is selected, Available tools is greyed out
- Toolbox: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop-down list
- Title: specifies the name of the new tool
- Help: specifies the help string displayed on the status bar

- Command: specifies the command(s) or macros
- Image: specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image
- Available tools: selects an existing tool

**ID**
Identifies the Quad item in the CUI file. (The ID is assigned by BricsCAD).

**Title**
Specifies the name that is displayed by the quad button.

**Help**
Specifies the help string displayed on the status bar.

**Command**
Specifies the command(s) or macros related to the quad button.

**Image**
Specifies the image to display for the toolbar button. The browse icon opens the Tool Image dialog box. There are five options to select an image.

**Entity Filter**
Specifies which items will be displayed in the Quad, depending on the type of the entity.

**Properties**
Edits which properties are shown for different kinds of entities.

***Прим.:*** These properties will only be shown when the ROLLOVERTIPS system variable is ON.

**Rollover**
Specifies which properties should be displayed.

**Workspaces**
Controls which menu tabs are visible in different workspaces.
When right clicking an item in the Workspace tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

**Command Aliases**
Customize aliases and add them to different commands.
When right clicking an item in the Command Aliases tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

***Прим.:*** If you select Add alias or Edit alias from the context menu, the Add alias or edit alias dialog box displays where you can add or edit an alias.

**Alias**
Specifies the name of the alias, a command abbreviation.

***Прим.:*** The list of Aliases and commands will be ordered alphabetically according to the aliases when you click on **Alias**.

**Command**
Specifies the command to which the alias is assigned.

***Прим.:*** The list of Aliases and commands will be ordered alphabetically according to the commands when you click on **Command**.

**Add**
Adds an alias.

**Edit**
Edits the selected alias.

**Delete**
Deletes the selected alias.

**Commands**
Lists all the available commands to add an alias to.

**Shell Commands**
Create and edit shell commands. Shell commands run programs external to BricsCAD.
When right clicking an item in the Command Aliases tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

***Прим.:*** If you select Add shell command or Edit shell command from the context menu, the Add shell command or edit shell command dialog box displays where you can add or edit a shell command.

**Alias**
Specifies the name of the shell command; this is the command you enter at the ' : ' prompt.

**Shell Command**
Specifies the operating system command to execute, or program to run.
To run a program, use the Start prefix, as in Start Notepad.exe.

**Flags**
Determines how the shell command operates:

- Don't wait: BricsCAD immediately returns to the command prompt without waiting for the application to finish
- Minimized: the application runs minimized
- Hidden: the application does not appear on the screen
- Quotes: the shell command uses quotations when the command string uses spaces, such as *C:\cad programs\file name.exe*

**Prompt**
Specifies the prompt to display in the Command line, for example: "Enter file to edit:"

**Add**
Adds a shell command through a dialog box.

**Edit**
Edits the shell command through a dialog box.

**Delete**
Deletes the selected shell command. Gives a warning where you can confirm you to delete the selected item.

**Options Within the Context Menu**

**Append …**
Adds the mentioned item below the last item.

**Insert ...**
Adds the mentioned item above the selected item.

**Add …**
Adds the mentioned item in alphabetical order.

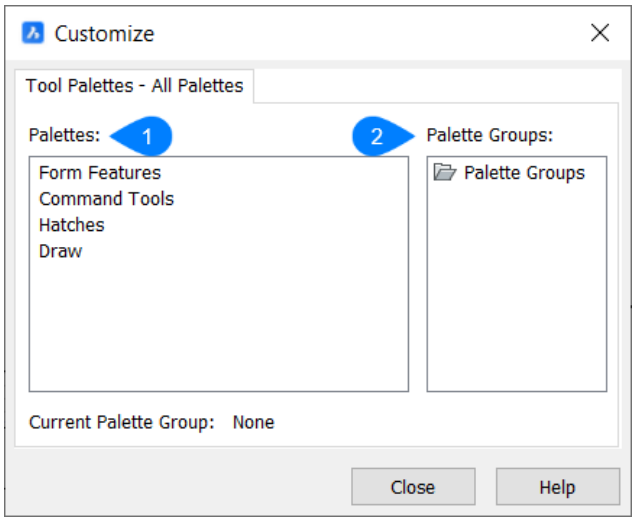
**Delete ...**
Deletes the menu, submenu, item or separator selected item. Gives a warning where you can confirm you to delete the selected item.

### 4.12.48.4 Manage your customizations

Displays the Manage Customizations dialog box for confirming and reversing changes made to the user interface.

## 4.12.49 Customize (Tool Palettes) dialog box

The Customize (Tool Palettes) dialog box allows you to customize your own tabs and tools to use in the tool palettes panel.




1 Palettes

2 Palette Groups

### 4.12.49.1 Palettes

Displays a list of all palettes available in the tool palettes panel. By right clicking on one of the palettes, several options may be available in the context menu.

**Rename**
Renames the Palette by entering the new name.

**New Palette**
Adds a new Palette to the list with a generic name that can be changed right away.

**Delete**
Deletes the selected palette

**Export**
Exports a palette through the Export Palette dialog box.

**Import**
Imports a palette through the Import Palette dialog box.

### 4.12.49.2 Palette Groups

Displays the loaded palette groups. By right clicking on one of the groups, several options may be available in the context menu.

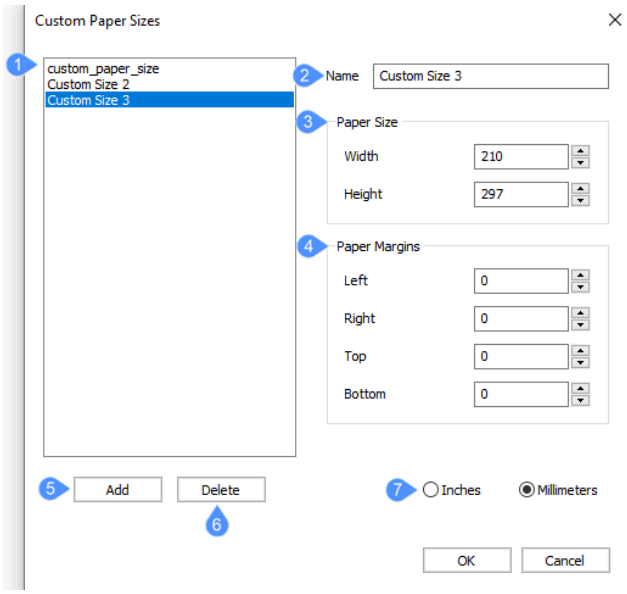
**New group**
Adds a new sub-group to the list with a generic name that can be changed right away.

***Прим.:*** For the additional sub group, the available options in the context menu are the same as those for palettes.

**Import**
Imports a palette through the Import Palette Group dialog box.

## 4.12.50 Custom paper sizes dialog box

The Custom Paper Sizes Dialog Box allows you to create custom paper sizes when Print as PDF.pc3 is selected in the Printer/Plotter Configuration list.




1 Custom Paper Sizes list

2 Name

3 Paper Size

4 Paper Margins

5 Add

6 Delete

7 Units

### 4.12.50.1 Custom Paper Sizes list

Displays the list of all custom paper sizes created by the user.

### 4.12.50.2 Name

Allows you to name the custom paper size.

### 4.12.50.3 Paper Size

You can insert the width and the height of the page in the appropriate fields.

### 4.12.50.4 Paper Margins

You can set the paper margins by entering the desired values in the fields. Only positive values are allowed.

### 4.12.50.5 Add

Allows you to create the new custom paper size and adds it to the list.

### 4.12.50.6 Delete

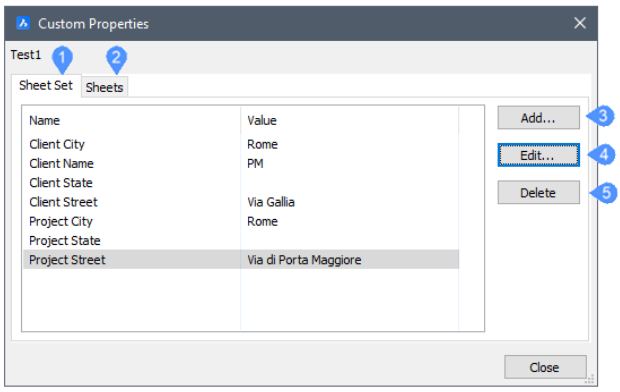
Allows you to select a custom paper size from the list and delete it. You can delete one item at a time.

### 4.12.50.7 Units

You can choose whether to use inches or millimeters for your custom paper size.

## 4.12.51 Custom properties dialog box

The **Custom Properties** dialog box allows you add custom properties for sheet set and sheets.




1 Sheet Set Tab

2 Sheets Tab

3 Add

4 Edit

5 Delete

### 4.12.51.1 Sheet Set Tab

Manages the custom sheet set properties.

### 4.12.51.2 Sheets Tab

Manages the custom sheets properties.

### 4.12.51.3 Add

Adds a custom property. Displays the **Add Custom Property** dialog box in which you add a property name and its initial value.

### 4.12.51.4 Edit

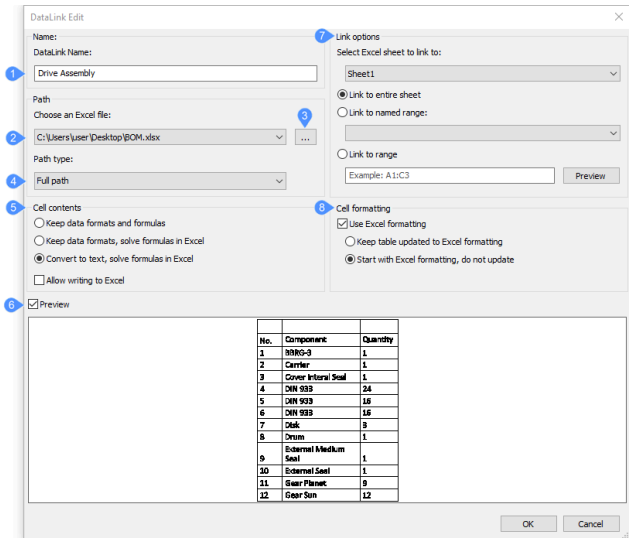
Allows to edit the value of a selected custom property. Displays the **Custom Sheet Property** or **Custom Sheet Set Property** dialog box in which you edit the name of the property.

### 4.12.51.5 Delete

Deletes a selected custom property.

## 4.12.52 DataLink edit dialog box

The **DataLink Edit** dialog box allows you to link data from an Excel spreadsheet directly into drawing tables for bi-directional updating.




1 Name

2 Path

3 Browse

4 Path type

5 Cell contents

6 Preview

7 Link options

8 Cell formatting

### 4.12.52.1 Name

You can type a name in the **DataLink Name** field.

### 4.12.52.2 Path

Displays you the path where your Excel file is located.

### 4.12.52.3 Browse

Allows you to select a data file. *.xls, *.xlsx and *.csv files are accepted.

### 4.12.52.4 Path type

Allows you to choose a path type option:

- **Full Path**: BricsCAD searches for the data file in the specified folder only.
- **Relative Path**: the data file is searched for in a subfolder of the drawing file folder.
- **No Path**: the data file is searched for in the drawing file folder only.

### 4.12.52.5 Cell contents

- **Keep data formats and formulas**: imports data with formulas and supported data formats.

- **Keep data formats, solve formulas in Excel**: imports data formats. Data is calculated from formulas in Excel.
- **Convert to text, solve formulas in Excel**: imports data as text with data calculated from formulas in Excel.
- **Allow writing to Excel**: if checked, the DATALINKUPDATE command uploads changes to linked data in your drawing to the original external spreadsheet.

#### 4.12.52.6 Preview

If this button is selected, a preview of the data file is displayed.

#### 4.12.52.7 Link options

Allows you to select Excel sheet to link to:

- **Link to the entire sheet**: all cells that contain data are linked.
- **Linked to named range**: links to a named range of cells. Click the button to select a named range.
- **Link to range**: specify a range of cells. Click the **Preview** button to display the selected range.

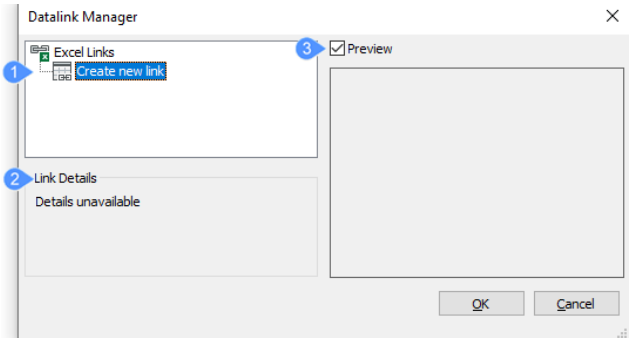
#### 4.12.52.8 Cell formatting

You must select the **Use Excel formatting** option to be able to select one of the following options:

- **Keep table updated to Excel formatting**: updates any changed formatting when the DATALINKUPDATE command is executed.
- **Start with Excel formatting. Do not update**: imports the formatting specified in the original data file. The DATALINKUPDATE command does not affect any changes made to the cell formatting.

### 4.12.53 Datalink manager dialog box

The **Datalink Manager** dialog box links data from an Excel spreadsheet directly into drawing tables for bi-directional updating.




1 Create new link

2 Link details

3 Preview

### 4.12.53.1 Create new link

Opens the **DataLink Edit** dialog box.

### 4.12.53.2 Link details

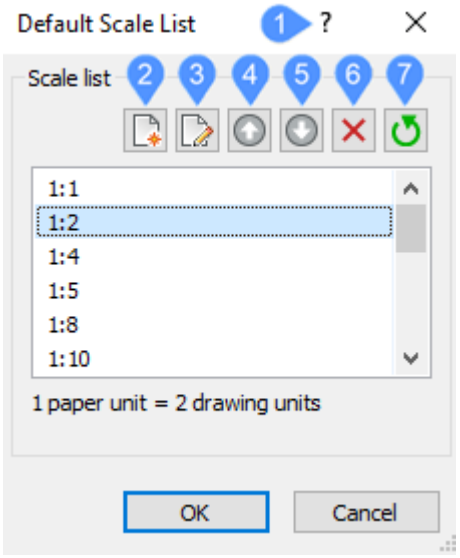
Displays link details.

### 4.12.53.3 Preview

Display a preview of the datalink.

## 4.12.54 Default scale list dialog box

The **Default Scale List** dialog box allows you to edit the list of scale factors displayed by commands, such as PRINT and PAGESETUP and by annotative scale factors.




1 Command reference

2 Add new scale

3 Edit scale

4 Move up

5 Move down

6 Delete

7 Reset

### 4.12.54.1 Command reference

Opens the Bricsys Help article about the DEFAULTSCALELIST command.

### 4.12.54.2 Add new scale

Allows you to add a new scale factor to the list. Opens the **Add Scale** dialog box.

### 4.12.54.3 Edit scale

Allows you to edit the factors of an existing scale. Opens the **Edit Scale** dialog box.

### 4.12.54.4 Move up

Moves the selected scale factor up the list.

### 4.12.54.5 Move down

Moves the selected scale factor down the list.

### 4.12.54.6 Delete

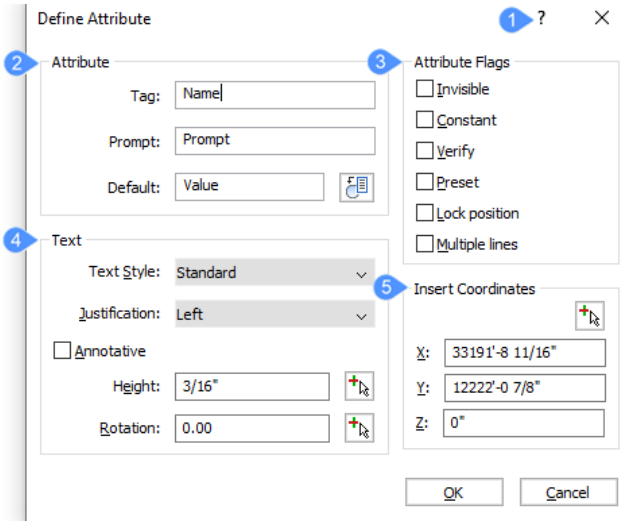
Deletes the selected scale factor(s). Press the Ctrl key to select and delete more than one factor at a time. Scale 1:1 cannot be deleted.

### 4.12.54.7 Reset

Returns the list to its original condition, adding back in the deleted factors and removing those that were added, but are not being used.

## 4.12.55 Define attribute dialog box

The **Define Attribute** Dialog Box defines the options for the attribute values.




1 Command reference

2 Attribute options

3 Attribute Flags options

4 Text options

5 Insert Coordinates options

### 4.12.55.1 Command reference

Opens the Bricsys Help article about the DEFAULTSCALELIST command.

### 4.12.55.2 Attribute options

- **Tag**

  Specify the name of the attribute. This is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers and punctuation.

- **Prompt**

  Specify the user prompt. This is displayed at the command prompt when the attribute is later inserted in the drawing. You can leave this field blank. BricsCAD then uses the tag as the prompt during attribute insertion.

- **Default**

  Specify the default value. This is displayed in angle brackets, like <360>. As with other defaults, the user only needs to press Enter to accept this value.

  Click the **Insert Field** icon to assign a field value to the attribute (see the FIELD command).

### 4.12.55.3 Attribute Flags options

- **Invisible**

  Hide the attribute from view; they are not displayed and not printed. Hidden attributes can, however, be displayed with the ATTDISP command.

- **Constant**

  Specify a default value that the user cannot change.

- **Verify**

  Forces to enter the value a second time; this helps to ensure that the value is entered correctly.

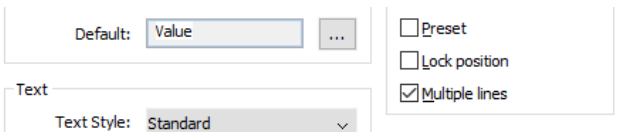
- **Preset**

  Inserts attributes without prompting the user; the attributes can be changes later with the ATTEDIT command.

- **Lock position**
  - On: locks the position of the attribute within the block reference.
  - Off: unlocked attributes can be moved relative to the rest of the block using grip editing.

- **Multiple lines**
  - On: allows the use of multiline text.
  - Off: limits to a single line.

  When the **Multiple lines** box is checked, the button icon displays next to the dimmed **Default** text field.




When you click the button, the **Define Attribute** dialog box temporarily closes to let you define a multiline text entity.

The MTEXT command launches. Type the text for the attribute, then click the **OK** button on the **Text Formatting** toolbar to return to the **Attribute Definition** dialog box.

#### 4.12.55.4 Text options

- **Text style**

  Specify the text style for the attribute text. Choose from one of the styles defined in the drawing. (You can create additional text styles with the STYLE command.)

- **Justification**

  Specify the justification of the attribute text, such as **Center** or **Fit**.

- **Height**

  Specify the height of the attribute text, unless overridden by the text style. Enter a value, or specify two points in the drawing by clicking the button. The last option dismisses the dialog box temporarily.

- **Rotation**

  Specify the rotation angle of the rotation text. Enter a value, or specify two points in the drawing by clicking the button. The last option dismisses the dialog box temporarily.

- **Annotative**

  Specify the **Annotative** property of the attribute text.

#### 4.12.55.5 Insert Coordinates options

- **Select insertion point**

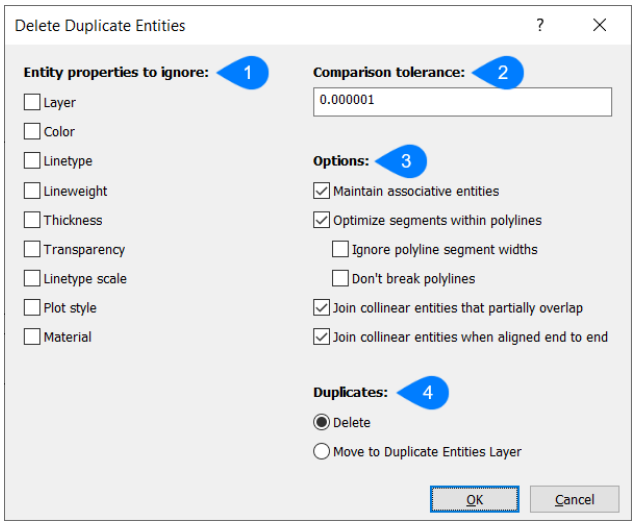
  Specify the insertion point directly in model.

- **X/Y/Z**

  Specify the coordinates of the attribute's insertion point.

### 4.12.56 Delete duplicate entities dialog box

The Delete Duplicate Entities dialog box allows you to delete duplicate entities and overlapping lines, arcs, and polylines, and unifies partly overlapping or contiguous ones.




1 Entity properties to ignore

2 Comparison tolerance

3 Options

4 Duplicates

### 4.12.56.1 Entity properties to ignore

Selected properties are ignored when comparing duplicate or overlapping entities.

### 4.12.56.2 Comparison tolerance

Sets the tolerance for the comparison process. If set to 0 (zero), entities must match completely before being evaluated in the overkill process.

### 4.12.56.3 Options

Determines how special cases are handled.

**Maintain associative entities**
Associative entities are not processed.

**Optimize segments within polylines**
Evaluates segments of selected polylines individually, duplicate vertices and segments are removed.

- **Ignore polyline segment widths**: ignores the Width property of overlapping polyline segments.
- **Don't break polylines**: polylines entities are kept intact.

**Join collinear entities that partially overlap**
Unifies partially overlapping entities into a single entity.

**Join collinear entities when aligned end to end**
Unifies collinear lines and polylines having coinciding endpoints into a single entity or segment.

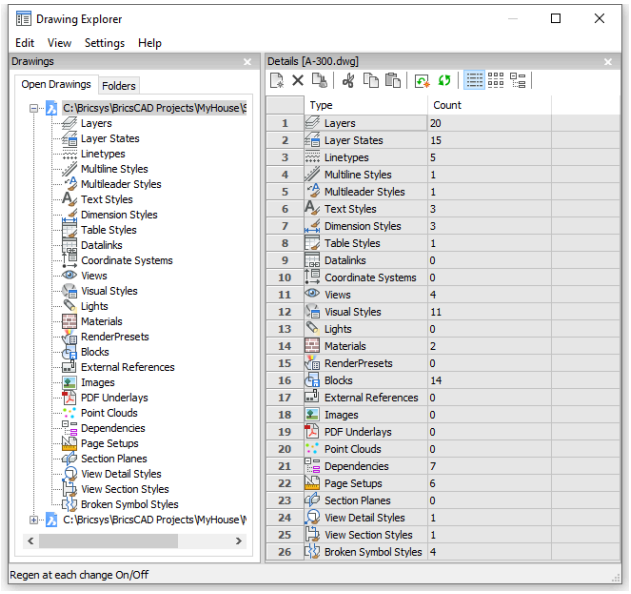
### 4.12.56.4 Duplicates

Determines what to do with duplicates: either delete them or move them to a Duplicate Entities layer.

## 4.12.57 Drawing explorer dialog box

### 4.12.57.1 Opening the Drawing Explorer dialog box

- Launch the EXPLORER command.
- Launch a definition category command, such as LAYERS or EXPBLOCKS.
- Choose a definition category in the **Tools** > **Drawing Explorer** menu.
- Choose **Drawing Explorer** in the **General** tab of the No Selection Quad.




| Categories | Description |
|---|---|
| Layers | Reports the number of layers defined in the selected drawing. Click to explore the layers; see the LAYER command. |
| Layer States | Reports the number of layer states defined in the selected drawing. Click to explore the layer states; see the LAYERSTATE command. |
| Linetypes | Reports the number of linetypes loaded into the selected drawing. Click to explore the linetypes; see the LINETYPE command. |
| Multiline Styles | Reports the number of multiline styles defined in the selected drawing. Click to explore multiline styles; see the MLSTYLE command. |
| Multileader Styles | Reports the number of multileader styles defined in the selected drawing. Click to explore multileader styles; see the MLEADERSTYLE command. |
| Text Styles | Reports the number of text styles defined in the selected drawing. Click to explore text styles; see the STYLE command. |

| Categories | Description |
|---|---|
| Dimension Styles | Reports the number of dimension styles defined in the selected drawing. Click to explore dimension styles; see the DIMSTYLE command. |
| Table Styles | Reports the number of table styles defined in the selected drawing. Click to explore table styles; see the TABLESTYLE command. |
| Data Links | Reports the number of data in the drawing linked to external spreadsheet files. Click to explore data links; see the DATALINK command. |
| Coordinate Systems | Reports the number of named user coordinate systems (UCS) defined in the selected drawing. Click to explore coordinate systems; see the EXPUCS command. |
| Views | Reports the number of named views in the selected drawing. Click to explore saved views; see the VIEW command. |
| Visual Styles | Reports the number of visual styles defined in the selected drawing. Click to explore visual styles; see the VISUALSTYLES command. |
| Lights | Reports the number of lights placed in the selected drawing. Click to explore lights; see the LIGHTLIST command. |
| Materials | Reports the number of materials defined in the selected drawing. Click to explore materials; see the MATERIALS command. |
| Render Presets | Reports the number of render presets defined in the drawing. Click to explore render presets; see the RENDERPRESETS command. |
| Blocks | Reports the number of blocks defined in the selected drawing. Click to explore blocks; see the EXPBLOCKS command. |
| External References | Reports the number of external references (xrefs) attached to the selected drawing. Click to explore xrefs; see the XREF command. |
| Images | Reports the number of images attached to the selected drawing. Click to explore the images; see the IMAGE command. |
| PDF Underlays | Reports the number of PDF underlays attached to the selected drawing. Click to explore PDF underlays; see the PDF command. |
| Point Clouds | Reports the number of point clouds attached to the selected drawing. Click to explore point cloud; see the POINTCLOUD command. |

| Categories | Description |
|---|---|
| Dependencies | Reports the number of external files that the selected drawing depends on. Click to explore dependencies; see the ETRANSMIT command. |
| Page Setups | Reports the number of the page setups attached to the selected drawing. Click to explore page setups; see the PAGESETUP command. |
| Section Planes | Reports the number of section planes defined in the selected drawing. Click to explore the section planes; see the SECTIONPLANESETTINGS command. |
| View Detail Styles | Reports the number of view detail styles defined in the selected drawing. Click to explore the view detail styles; see the VIEWDETAILSTYLE command. |
| View Section Styles | Reports the number of view section styles defined in the selected drawing. Click to explore the view section styles; see the VIEWSECTIONSTYLE command. |
| Broken Symbol Styles | Reports the number of available styles for the VIEWBREAK command. Click to explore the broken symbol styles. |

### 4.12.57.2 The Drawing Explorer layout

The **Drawing Explorer** window consists of a number of sub-windows or panes.

You can select which panes you want to have open or closed in the **View** menu. BricsCAD remembers the display settings for each category.

| Item | Description |
|---|---|
| Drawing | Displays the **Drawing** pane.<br>• **Open Drawings**: a list of all drawings that are currently open.<br>• **Folders**:<br>- **Local Folders**: your favorite local drawing folders.<br>- **Bricsys 24/7**: your drawing folders on Bricsys 24/7. |
| Details | Displays the **Details** pane.<br>Shows the details of the selected drawing or the details of the selected category. |
| Preview | Displays the **Preview** pane.<br>Shows a preview of the selected drawing, layer, block, view, dimension style, table style, XRef, image or material. |
| Editor | Displays the **Editor** pane.<br>Allows you to define or edit the properties of the selected category. |

| Item | Description |
|---|---|
| Status bar | Displays the Status bar at the bottom of the **Drawing Explorer** dialog box. |

| | View options for the Details pane |
|---|---|
| Details | Displays the details of definitions in a grid. |
| Icons | Displays definitions as an array of icons; applies only to blocks and external references. |
| Tree | Displays definitions in tree form; applies only to layer states, data links, dimension styles, external references, images, PDF underlays and dependencies. |

| | |
|---|---|
| Regen | Updates the drawing display to reflect changes made in this dialog box. |
| Regen at each change | Toggles the automatic updating of the display at each change. |

| | |
|---|---|
| Hide Xref symbols | Do not display definitions in Xrefs. |

***Прим.:***

- If you click a definition category of an open drawing in the **Drawings** sub-window when the **Details** sub-window is closed, the **Details** window is re-opened, showing the content of the selected category.
- To close a sub-window, click the close button () in its title bar.
- To restore the default layout, choose **Restore Default Explorer Layout** in the **Settings** menu.

### 4.12.57.3 The Settings menu

| Item | Description |
|---|---|
| Restore Explorer Default Layout | Restores the layout of the Drawing Explorer to the default layout for the selected category. When you makes changes to the layout of the Drawing Explorer, the program remembers them. This option resets all your changes. |

| Item | Description |
|---|---|
| Options | Applies to Blocks and External References only.<br>Specifies default settings for inserting blocks. See the EXPBLOCK command. |

### 4.12.57.4 The Details toolbar

| Icon | Name | Shortcut | Description |
|---|---|---|---|
|  | New | Ctrl + N | Creates a new definition with a generic name, such as NewLayer 1 or NewStyle1. Use the Rename button to rename it. |
|  | Delete | Ctrl + D | Erases the selected definition from the drawing. When the item is in use, such as a layer with entities, you have to option to retain the entities from the layer to be erased.<br>Some items cannot be erased, such as layer 0, Global material, visual style 2D Wireframe, … |
|  | Purge | Ctrl + P | Purges unused definitions from the drawing. Some definitions cannot be erased, such as layer 0 and text style Standard. See the PURGE command. |
|  | Cut | Ctrl + X | Copies the selected definition(s) to the Clipboard, and then deletes it from the drawing following the same procedure as the Delete tool. |
|  | Copy | Ctrl + C | Copies the definition(s) to the Clipboard for pasting elsewhere in the Drawing Explorer. |
|  | Paste | Ctrl + V | Pastes the definition(s) from the Clipboard, into the Drawing Explorer. When the item on which the Clipboard content will be pasted is in use, such as a layer with entities, you have options:<br>• **Copy and Replace** - replaces the existing item with the pasted one<br>• **Don't Copy** - cancels the copy operation<br>• **Copy, but keep both** - pastes the Clipboard content with an incremented name, usually suffixed by a 1<br>• **Cancel** - also cancels the operation |
|  | Details View | | Displays the details of definitions in a grid. |
|  | Icon View | | Displays definitions as an array of icons; applies only to blocks and external references. |

| Icon | Name | Shortcut | Description |
|---|---|---|---|
|  | Tree View | | Displays definitions in tree form; applies only to layer states, data links, dimension styles, external references, images, PDF underlays and dependencies. |
|  | Regen | | Updates the drawing display to reflect changes made in this dialog box. |
|  | Toggle Regen | | Toggles the automatic updating of the display at each change. |

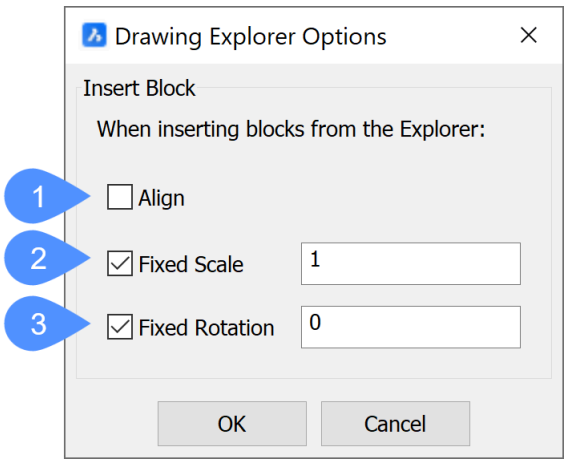
### 4.12.57.5 The Context menu

Select a definition, then right-click.

| Item | Shortcut | Description |
|---|---|---|
| New | Ctrl + N | Creates a new definition as a copy of the selected definition. |
| Delete | Ctrl + D | Deletes the selected definitions. Definitions that are in use cannot be deleted. |
| Purge | Ctrl + P | Deletes unused definitions in the current selection. |
| Cut | Ctrl + X | Cuts the selected definitions to the clipboard. |
| Copy | Ctrl + C | Copies the selected definitions to the clipboard. |
| Paste | Ctrl + V | Pastes definitions from the clipboard. |
| Rename | F2 | Renames the selected definition; alternatively, click on the name twice, slowly, to rename it. |
| Select All | Ctrl + A | Selects all definitions in the current category. |
| Invert Selection | | Inverts the selected definition(s), e.g. select a few layers, then use this to select all other layers. |
| Current  | | Makes the definition current, such as making the selected layer the current (working) layer. |

### 4.12.58 Drawing explorer options dialog box

The **Drawing Explorer Options** dialog box allows you to align or set the scale and the rotation of the inserted blocks.




1 Align

2 Fixed Scale

3 Fixed Rotation

#### 4.12.58.1 Align

Tick the checkbox to align blocks with the snapped entity.

#### 4.12.58.2 Fixed Scale

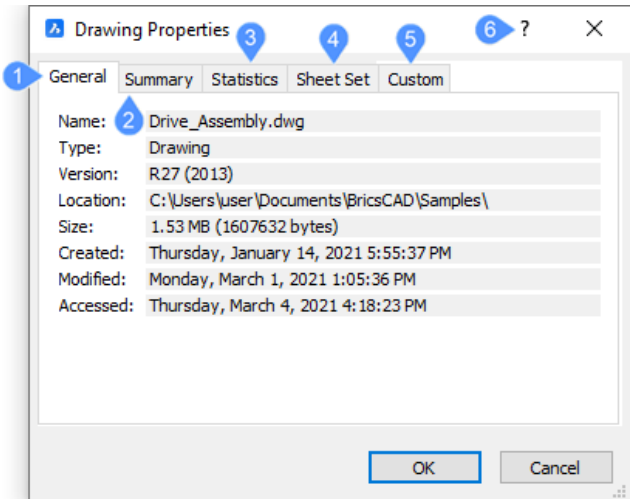
Tick the checkbox to specify the scale factor to be used for all block insertions.

#### 4.12.58.3 Fixed Rotation

Tick the checkbox to specify the rotation angle to be used for all block insertions.

### 4.12.59 Drawing properties

The **Drawing Properties** dialog box allows you to see general information about the drawing and user-defined properties.




1 General

2 Summary

3 Statistics

4 Sheet Set

5 Custom

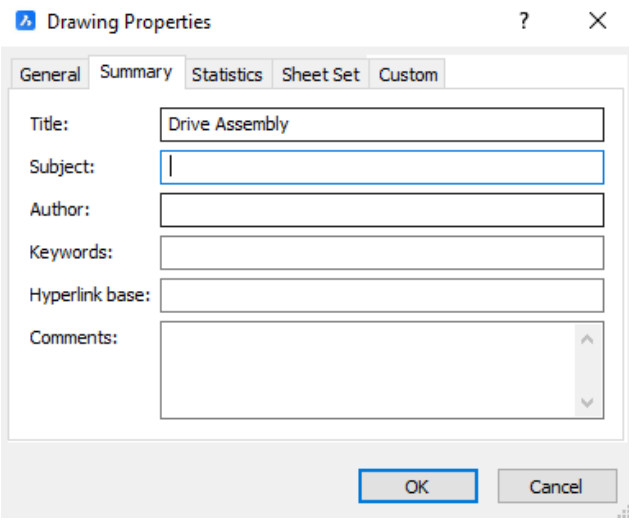
6 Command reference

### 4.12.59.1 General

Displays the general properties of the drawing, such as creation date and time.

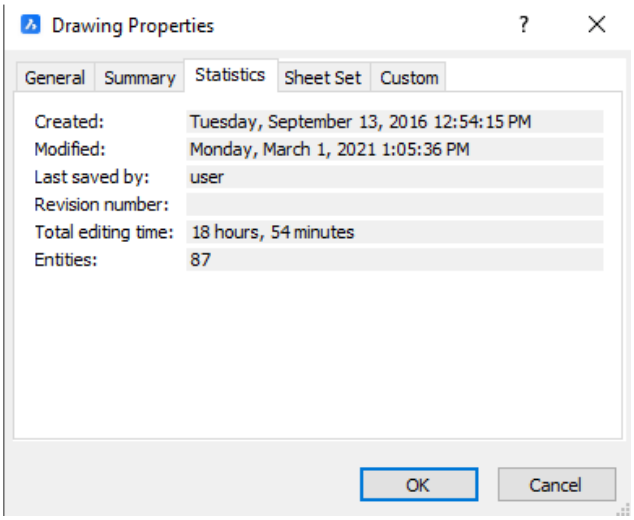
### 4.12.59.2 Summary

Lets you define the summary properties of the drawing, such as subject and keywords. This data can be searched for by EDMS (engineering data management software) to find drawings in large collections.



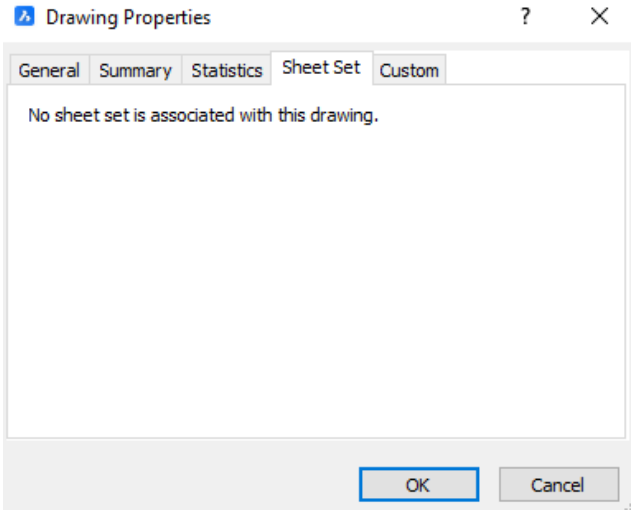
### 4.12.59.3 Statistics

Displays statistical information about the drawing, such as when it was created, modified.




### 4.12.59.4 Sheet Set

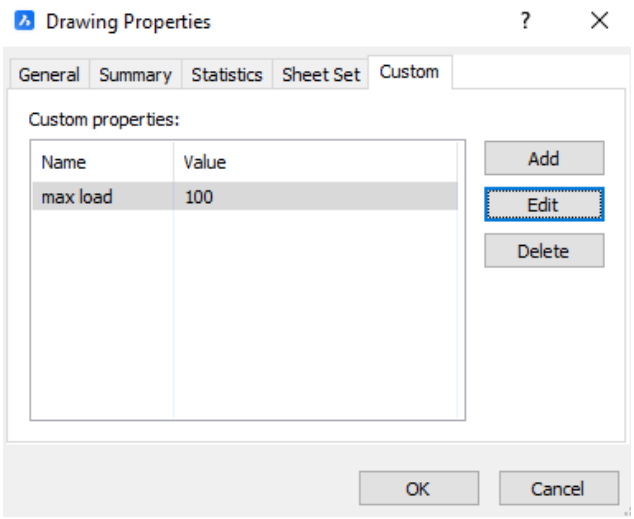
Displays sheet set data when the drawing is associated with a sheet set.




### 4.12.59.5 Custom

Displays the custom properties of the drawing. Click the **Add** button to add a new property or the **Edit** button to edit a property.

The Custom Property dialog box appears when clicking the **Add** and **Edit** buttons.

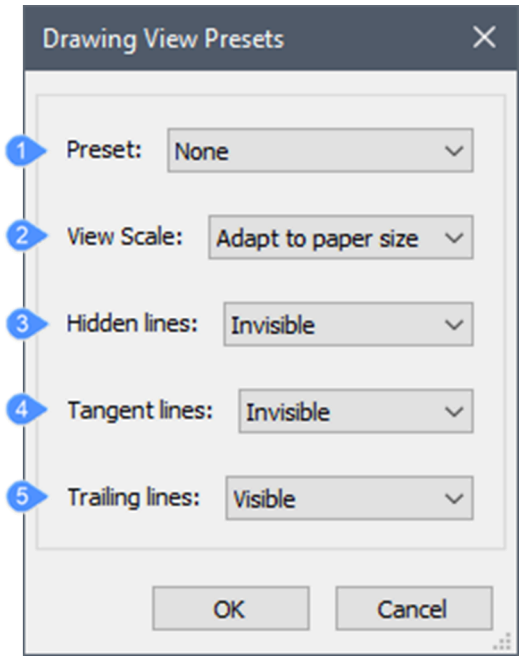



### 4.12.59.6 Command reference

Opens the Bricsys Help article about the DWGPROPS command.

## 4.12.60 Drawing view presets dialog box

Specifies the types of generated drawings and their placement in the layout.




1 Preset

2 View Scale

3 Hidden lines

4 Tangent lines

5 Trailing lines

#### 4.12.60.1 Preset

Select a preset view arrangement:

- **None**: No preset affects the views generated.
- **Architectural Views**: Generates:
  - Five standard views: **Front**, **Right**, **Back**, **Left** (used for elevations) and **Top** views.
  - Three section views: two horizontal sections (for plans) and 1 vertical section.
- **Mechanical Views**: Generates:
  - Three standard views: **Front**, **Top** and **Left** (first angle projection) or **Right** (third angle projection).

The layout of the mechanical views is controlled by the projection type: first or third angle. The current projection type is saved in the PROJECTIONTYPE system variable.

#### 4.12.60.2 View Scale

Sets the default scale factor for the drawing views from drop-down list. The scale list can be edited by the SCALELISTEDIT command.

#### 4.12.60.3 Hidden lines

Sets the default setting for hidden lines in drawing views.

#### 4.12.60.4 Tangent lines

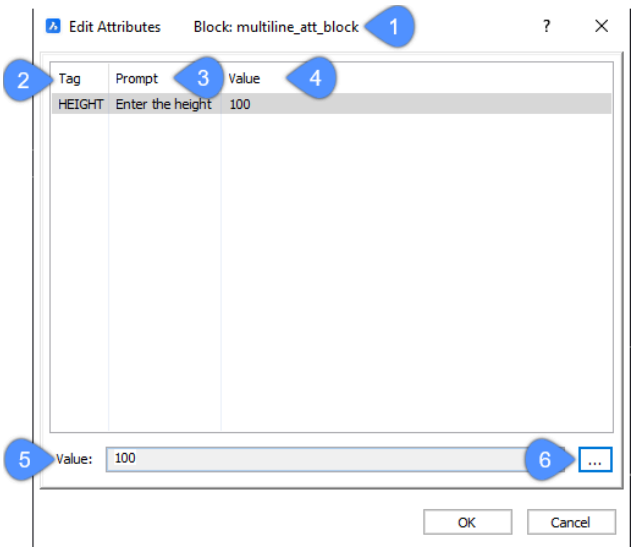
Sets the default for **Tangent Lines** in drawing views.

#### 4.12.60.5 Trailing lines

Sets the default for **Trailing Lines** in drawing views.

### 4.12.61 Edit attributes dialog box

The Edit Attributes dialog box allows you to edit the values of attributes.




1 Block name

2 Tag

3 Prompt

4 Value

5 Value

6 Text Formatting

#### 4.12.61.1 Block name

Displays the name of the selected block.

#### 4.12.61.2 Tag

Specifies the name of the attribute.

***Прим.:*** This is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers, and punctuation.

#### 4.12.61.3 Prompt

Specifies the user prompt.

***Прим.:*** This is displayed at the command prompt when the attribute is later inserted in the drawing. You can leave this field blank, BricsCAD uses the tag as the prompt during attribute insertion.

#### 4.12.61.4 Value

Displays the value that belongs to a certain attribute.

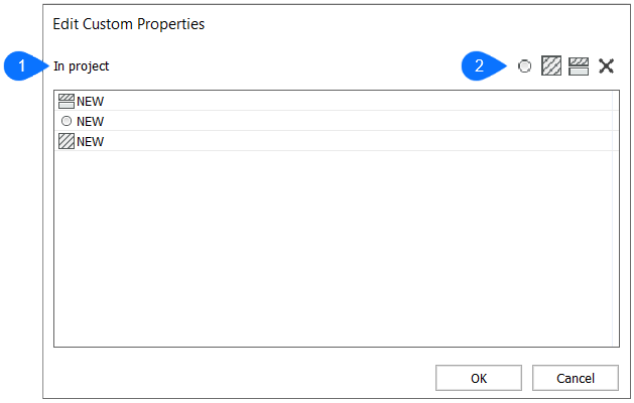
#### 4.12.61.5 ... Text Formatting

Changes the value of the attribute text.

***Прим.:*** To be able to use this option, the Multiple lines attribute flag must be selected, in the Define Attribute dialog box, when defining an attribute.

### 4.12.62 Edit custom properties dialog box

The Edit Custom Properties dialog box allows you to add custom properties to materials and compositions.




1 In project

2 Add Generic/ Material/Composition/Delete properties

#### 4.12.62.1 In project

Displays the custom properties available in the project.

#### 4.12.62.2 Add/Delete Properties

**Add Generic Custom Property**
Creates a new generic custom property, which can be assigned to building elements in the project.

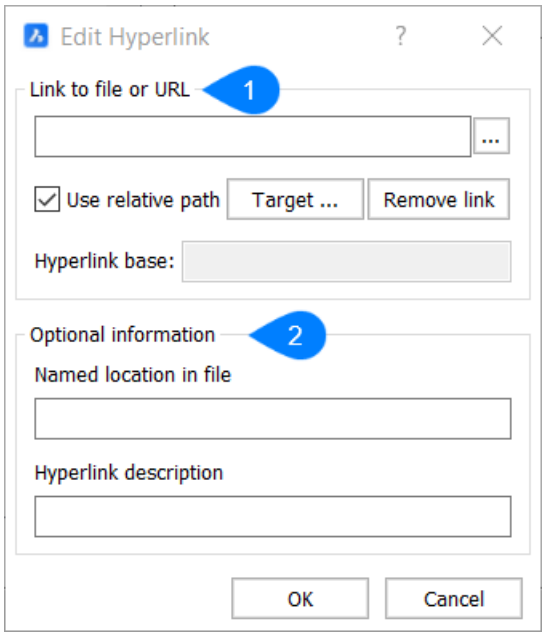
**Add Material Custom Property**
Creates a new material custom property, which can be assigned to materials in the project.

**Add Composition Custom Property**
Creates a new composition custom property, which can be assigned to compositions in the project.

**Delete**
Deletes the selected property.

### 4.12.63 Edit hyperlink dialog box

The Edit Hyperlink dialog box allows you to change the hyperlink to a file or URL.




1 Link to file or URL

2 Optional information

### 4.12.63.1 Link to file or URL

Specifies the URL of the file either by entering the URL or file path or through the Select File dialog box by pressing the Browse button.

**Use relative path**
Specifies a relative path by selecting the target file.

***Прим.:*** To remove the link, press **Remove** link.

**Hyperlink base**
Displays the hyperlink base path.

### 4.12.63.2 Optional information

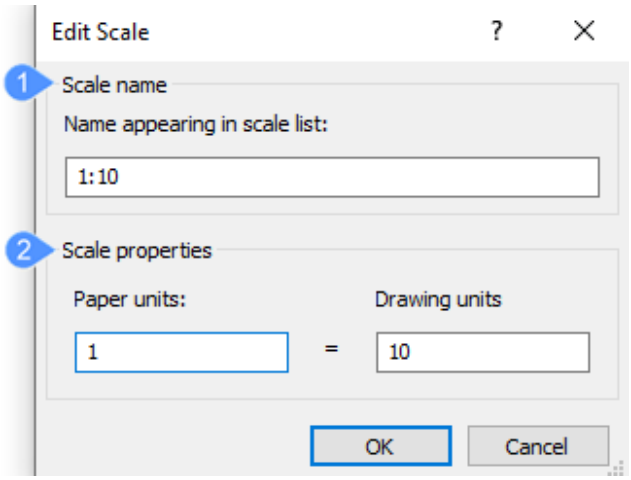
Specifies additional information

**Named location file**
Specifies a bookmark name. The # character specifies locations (bookmarks) within files or Web pages.

**Hyperlink description**
Adds an additional description of the hyperlink.

## 4.12.64 Edit scale dialog box

The **Edit Scale** dialog box allows you to edit the name and the scale properties of a scale factor.




1 Scale name

2 Scale properties

### 4.12.64.1 Scale name

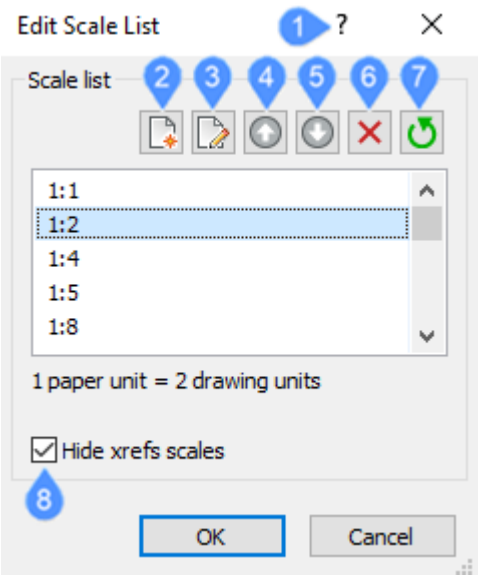
Edit the name by typing the new name.

### 4.12.64.2 Scale properties

Change the scale factor in paper (size when plotted) and/or drawing units.

## 4.12.65 Edit scale list dialog box

The **Edit Scale List** dialog box allows you to add, edit and delete scales.




1 Command reference

2 Add a new scale

3 Edit scale

4 Move up

5 Move down

6 Delete the selected scale

7 Reset

8 Hide xrefs scales

#### 4.12.65.1 Command reference

Opens the Bricsys Help article about the SCALELISTEDIT command.

#### 4.12.65.2 Add a new scale

Opens the **Add scale** dialog box.

#### 4.12.65.3 Edit scale

Opens the **Edit Scale** dialog box.

#### 4.12.65.4 Move up

Moves the selected scale up the list.

#### 4.12.65.5 Move down

Moves the selected scale down the list.

#### 4.12.65.6 Delete the selected scale

Deletes the selected scale. Press the CTRL key to select and delete more than one scale at a time.

Note that the scale 1:1 cannot be deleted.

#### 4.12.65.7 Reset

Returns the list to its original condition, adding back in the deleted scales and removing all added but unused scales.

#### 4.12.65.8 Hide xrefs scales

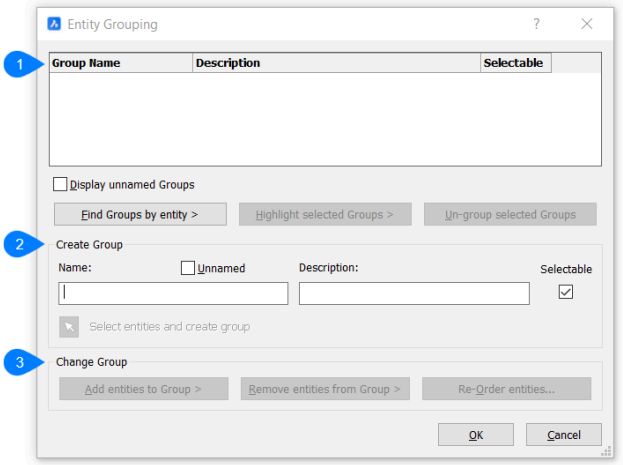
Determines if the scale factors from xrefs are displayed in the list:

- **Yes**: scale factors from externally referenced drawings are not shown in the list.
- **No**: scales in the attached xrefs not in the current drawing are listed here.

Toggles the HIDEXREFSCALES variable.

### 4.12.66 Entity grouping dialog box

The Entity grouping dialog box allows you to create, modify and delete named groups of entities.




1 Existing groups

2 Create Group

3 Change Group

### 4.12.66.1 Existing groups

Lists all existing groups with the Group Name and a short description.

**Display unnamed Groups**
Toggles the display of unnamed groups.

**Find Groups by entity**
Reports the name(s) of groups when entities in the drawing are selected.

**Highlight selected Groups**
Highlights all entities that belong to the selected group.

**Ungroup selected Groups**
Removes group status from selected entities.

### 4.12.66.2 Create Group

**Name**
Specifies the name of a group.

**Unnamed**
Toggles whether the group has a name.

**Description**
Adds an optional description of the group.

**Selectable**
Determines how groups are selected when the PICKSTYLE system variable is set to 1 or 3.

**Select entities**
Selects entities in the workspace which should be part of the group.

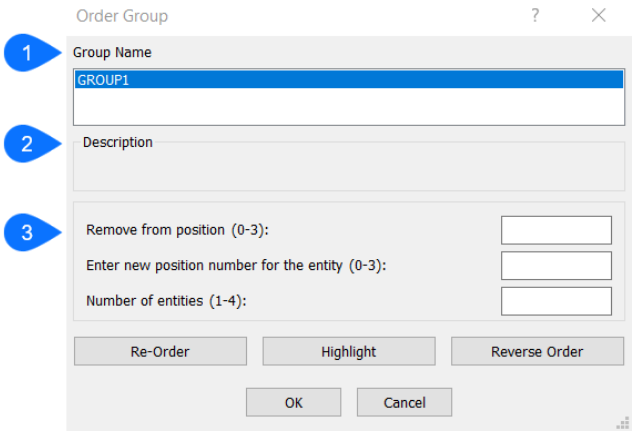
### 4.12.66.3 Change Group

**Add entities to group**
Adds entities to the selected group.

**Remove entities from group**
Removes entities from the selected group.

**Re-Order entities**
Changes the order of entities in groups trough the Order Group dialog box.

- Options in the Order Group dialog box

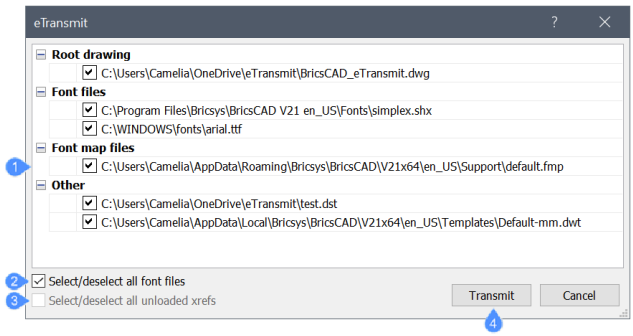



a Group Name
b Description
c Options

- **Group Name**: lists the names of named and unnamed groups.
- **Remove from position**: specifies the position number of the entity to be reordered.
- **Enter new position number for the entity**: specifies the new position number for the group.
- **Number of entities**: specifies the range of entities to reorder.
- **Re-Order**: applies the reordering dictated by the previous three fields.
- **Highlight**: highlights entities in the group; dismisses the Order Group dialog box.
- **Reverse Order**: reverses the order of entities in the group.

## 4.12.67 eTransmit dialog box

The eTransmit dialog box creates a package consisting of the drawing file and all its dependencies, such as external references, images, font files, plot configuration files, plot style tables, and font map files.




1 Available files

2 Select/deselect all font files

3 Select/deselect all unloaded xrefs

4 Transmit

### 4.12.67.1 Available files

Select the files to be included in the eTransmit package (usually all of them).

### 4.12.67.2 Select/deselect all font files

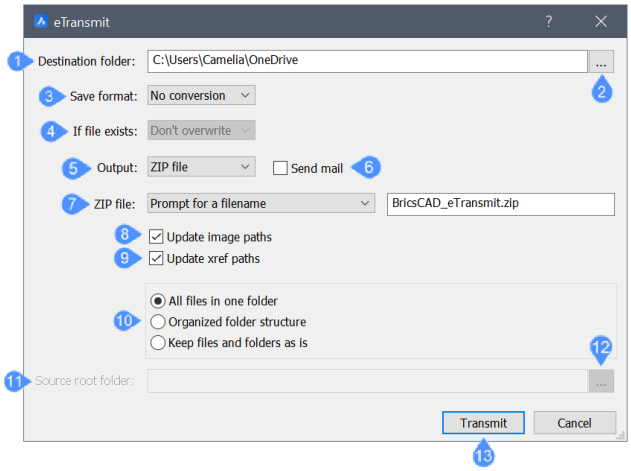
Tick the checkbox to automatically select all font files.

### 4.12.67.3 Select/deselect all unloaded xrefs

Tick the checkbox to automatically select all xref files.

### 4.12.67.4 Transmit

Opens the following dialog box:




1 Destination folder

2 Browse (destination folder)

3 Save format

4 If file exists

5 Output

6 Send mail

7 ZIP file

8 Update image paths

9 Update xref paths

10 Path options

11 Source root folder

12 Browse (root folder)

13 Transmit

**Destination folder**

Specifies the output folder for the eTransmit package. If the destination folder does not exist, you are prompted to create the folder.

**Browse (destination folder)**

Opens the Select the destination folder dialog box to allow you to choose the destination folder for the transmitted package.

**Save format**

Specifies the format of the drawing files in the eTransmit package.

**If file exists**

Specifies what to do if a file with the same name already exists in the output folder.

This option is available when Folder is chosen as output.

Select an option from the drop-down list:

- **Overwrite**: replaces the existing files with new ones.

- **Don't overwrite**: all files are transmitted, except files with the same file name as an existing file in the folder.
- **Abort**: the operation is aborted and no files are saved if a file with the same name already exists in the destination folder.

**Output**
Specifies whether to create a ZIP file or copy the files to the output folder.

**Send mail**
Tick the checkbox to send a mail with the zipped eTransmit package in attachment using your default mail application.
This option is available when ZIP is chosen as output.

**ZIP file**
Specifies what to do when a ZIP file with the same name already exists in the output folder.
Choose an option from the drop-down list:

- **Prompt for a file name**: opens a dialog box where you can choose a name and a path.
- **Overwrite if necessary**: replaces the existing file with the new one if a file with the same name already exists.
- **Increment name if necessary**: adds a suffix to the file name.

**Update image paths**
Tick the checkbox to update the saved paths for image files according to the new location of the transmitted files.

**Update xref paths**
Tick the checkbox to update the saved paths to externally-referenced files according to the new location of the transmitted files.

**Path options**
All files in one folder
All files in the eTransmit package are copied to the destination folder.

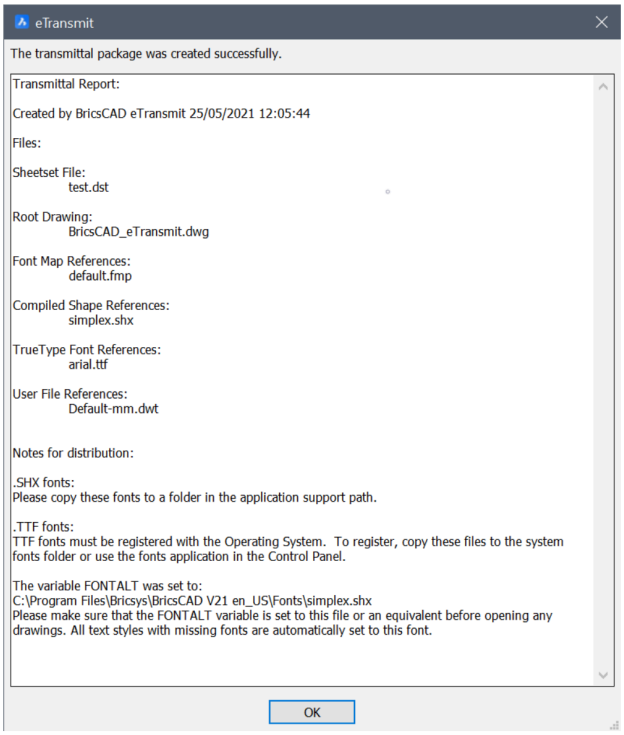
**Organized folder structure**
Creates a hierarchical folder tree with subfolders for all drawing file dependencies.

**Keep files and folders as is**
Keeps the folder structure of all files in the eTransmit package. This procedure allows you to easily install the package on another computer using the folder structure of the source system.

**Source root folder**
Specifies the source root folder for relative paths of drawing-dependent files, such as xrefs or image files.
This option is available when **Organized folder structure** is checked.

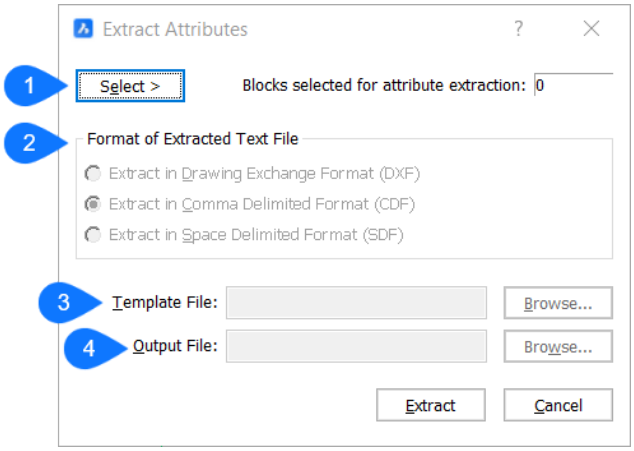
**Browse (root folder)**
Opens the Select the source root folder dialog box to allow you to choose a root folder.

**Transmit**
Transmits the files according to the selected options and displays a report of the transmission. The report contains a complete list of all files included and notes for distribution.




### 4.12.68 Extract attributes dialog box

The Extract Attributes dialog box allows you to copy data from attributes to a text file.




1 Select

2 Format of Extracted Text File

3 Template File

4 Output File

#### 4.12.68.1 Select

Specifies the set of blocks with attributes to copy data from.

***Прим.:*** If blocks without attributes are selected, BricsCAD ignores them.

### 4.12.68.2 Format of Extracted Text File

Specifies the format of the Extracted text file. Several formats are available:

- DXF: drawing Exchange Format
- CDF: extract in Comma Delimited Format
- SDF: space Delimited Format

### 4.12.68.3 Template File

Specifies the location path of the template file by entering the path or through the Attribute Extract Template file dialog box after pressing the Browse button.

### 4.12.68.4 Output File

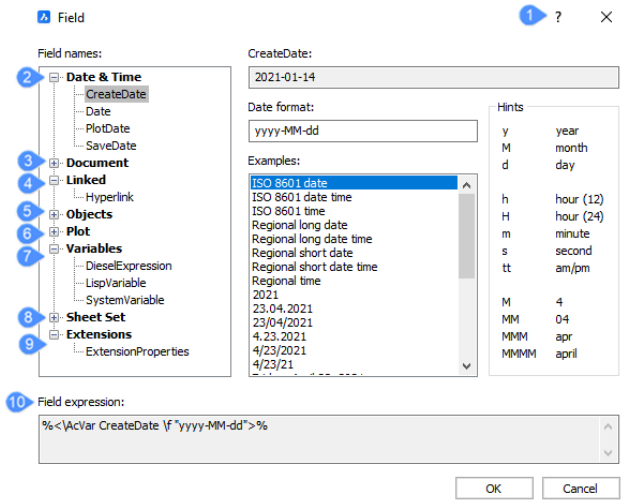
Specifies the location path of the output file by entering the path or through the Attribute Extract Output file dialog box after pressing the Browse button.

### 4.12.68.5 Extract

Exports the attribute values to the output file.

## 4.12.69 Field dialog box

The **Field** dialog box allows you to insert a field in a multiline text, table cells, attributes, and so on.




1 Command reference

2 Date & Time

3 Document

4 Linked

5 Objects

6 Plot

7 Variables

8 Sheet Set

9 Extensions

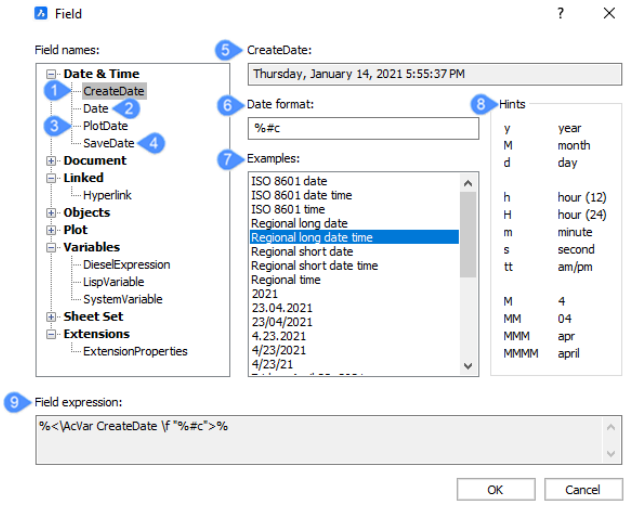
10 Field expression

### 4.12.69.1 Command reference

Opens the Bricsys Help article about the FIELD command.

### 4.12.69.2 Date & Time

For the **Date & Time** category, only the field number 1 changes based on the selected item.




1 CreateDate

2 Date

3 PlotDate

4 SaveDate

5 CreateDate

6 Date format

7 Examples

8 Hints

9 Field expression

**CreateDate**
Displays a random date.

**Date**
Displays the current date.

**PlotDate**
Displays the date the drawing was last plotted.

**SaveDate**
Displays the date the drawing was last saved.

**Date format**
Select a date format in the list or type a format in this field.

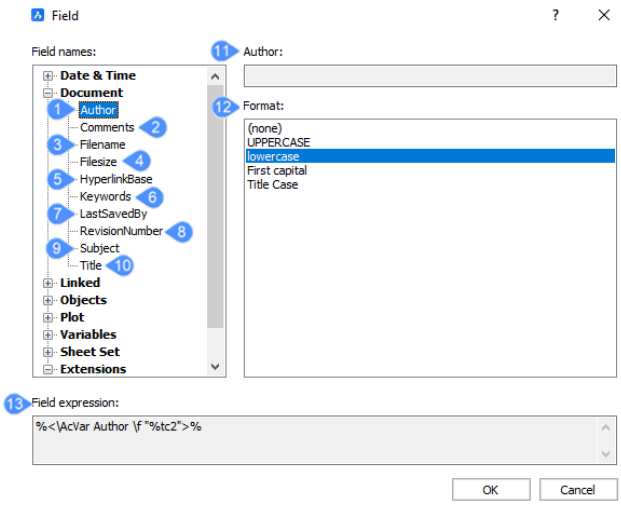
**Examples**
Displays date formats examples.

**Hints**
Explains the meaning of digits used in the date format acronym/abbreviation definition.

**Field expression**
Displays the field expression of the selected date. An operand in an expression can be a DisplayPropertyName enclosed in arrow brackets (<>).

### 4.12.69.3 Document

It includes 10 categories and 3 areas, excepting the Filename (no. 3) category that will be documented separately.

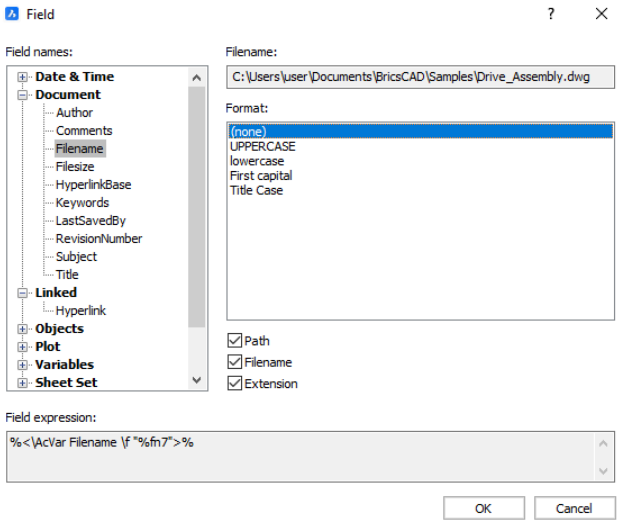



1 Author
2 Comments
3 Filename
4 Filesize
5 HyperlinkBase
6 Keywords
7 LastSavedBy
8 RevisionNumber
9 Subject
10 Title
11 Author
12 Format
13 Field expression

### Author
Displays the author of the document.

### Comments
Displays the comments of the document.

### Filename
Shows the filename. It can include the path and the extension of the file.




### Filesize
Displays the file size expressed in Bytes, Kilobytes and Megabytes.

### HyperLinkBase
Displays the default path for relative hyperlink in drawings.

### Keywords
Displays the keywords for the opened file.

### LastSavedBy
Displays the name of the person who have last saved the file.

### RevisionNumber
Displays the drawing revision number.

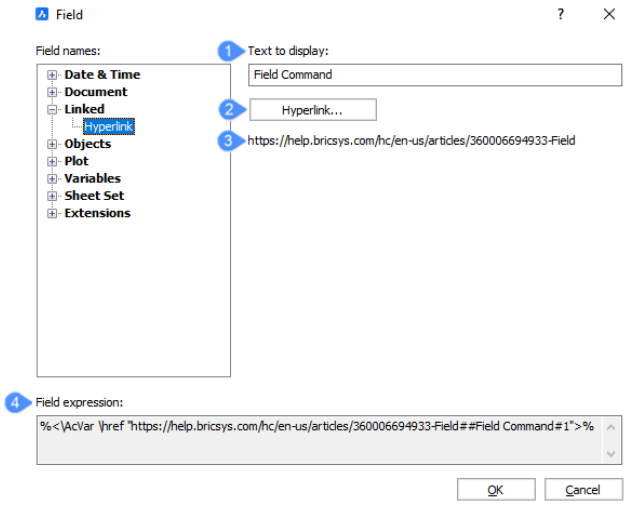
### Subject
Displays the subject of the drawing.

### Title
Displays the title of the drawing.

### Format
Displays the format of the selected category.

### Field expression
Displays the field expression of the selected date. An operand in an expression can be a DisplayPropertyName enclosed in arrow brackets (<>).

### 4.12.69.4 Linked




1 Text to display

2 Hyperlink

3 URL

4 Field expression

**Text to display**
You can insert in this field the text you want to be displayed for this hyperlink.

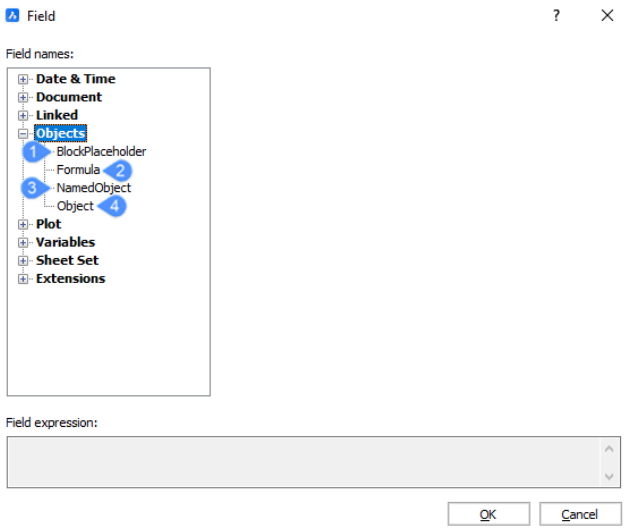
**Hyperlink**
Opens the **Edit Hyperlink** dialog box.

**URL**
Displays the URL.

**Field expression**
Displays the field expression of the selected date. An operand in an expression can be a DisplayPropertyName enclosed in arrow brackets (<>).

### 4.12.69.5 Objects

It includes 4 categories.




1 BlockPlaceHolder

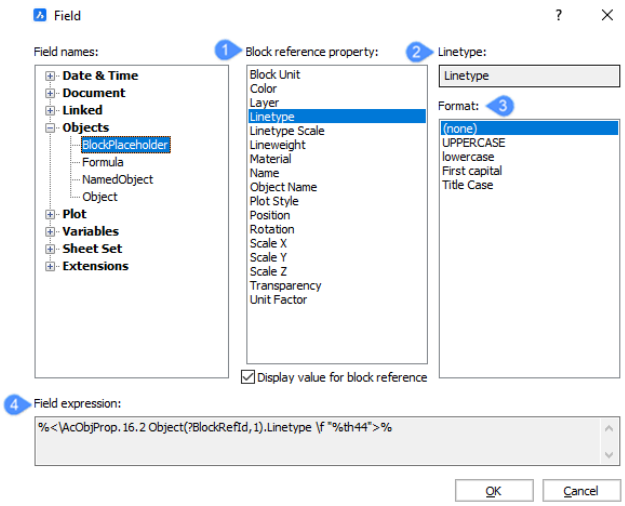
2 Formula

3 NamedObject

4 Object

**BlockPlaceHolder**

See the ATTDEF command to create block property fields in an attribute. When such attribute is included in a block definition, the field displays the current value of the block property.




1 Block reference property

2 Block name

3 Format

4 Field expression

- **Block reference property**: displays the block reference property.
- **Block name**: displays the name of the block.
- **Format**: displays the name of the block in different letter cases.

- **Field expression**: displays the field expression of the selected date. An operand in an expression can be a DisplayPropertyName enclosed in arrow brackets (<>).

**Formula**
Creates a formula field. You can use the values of table cells in a formula field. Click the **Average**, **Sum**, **Count** or **Cell** button. The **Field** dialog closes to let you select cells in a table in the drawing.

**NamedObject**
Creates a field that shows the current name of a named entity.

- Click the **Named object type** field, then select the object type in the list.

  All named objects of the selected type are listed in the **Name list** box.
- Select a name in the list.
- Choose a format.
- Click the **OK** button to place the field.

**Object**
Creates a field that shows a property of a selected entity.

- Click the select button, next to the **Object type** field.

  The **Field** dialog temporarily closes.
- Select an entity in the drawing.
- The properties of the selected entity are listed in the Property list.
- Select a property in the list.
- Choose a format.
- Click the **OK** button to place the field.

### 4.12.69.6 Plot

Plot fields apply to paper space layouts.

Allows to create fields which show the current print settings of a layout.

### 4.12.69.7 Variables

Allows to create fields which show the current value of Diesel Expressions, Lisp Variables and System Variables.

### 4.12.69.8 Sheet Set

Allows to create fields which show default and custom sheet set properties.

### 4.12.69.9 Extensions

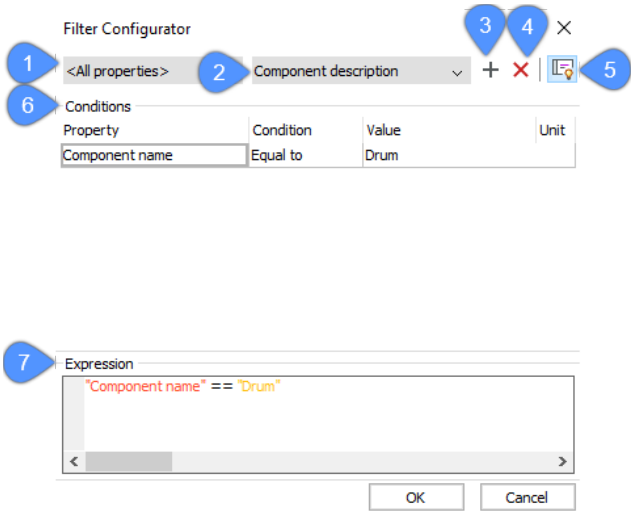
Allows to create fields showing properties of entities created by extensions, such as BIM or Mechanical.

### 4.12.69.1 Field expression 0

Displays the expression of the field.

### 4.12.70 Filter Configurator dialog box

The Filter Configurator dialog box allows you to specify the conditions to filter components in the BOM table.




1 Columns

2 Properties

3 Add

4 Delete

5 Expression

6 Conditions

7 Expression Editor

#### 4.12.70.1 Columns

Selects columns to apply the filter to.

#### 4.12.70.2 Properties

Specifies the available properties that can be used to apply the filter.

***Прим.:*** When specified the options above, the condition can be added by pressing the plus sign.

**Filter conditions**

Displays the conditions were added. This includes the properties, it's condition, value and unit.

#### 4.12.70.3 Add

Adds a condition for the selected property.

#### 4.12.70.4 Delete

Deletes the selected condition(s) from the filter.

### 4.12.70.5 Expression

Shows or hides the expression editor.

### 4.12.70.6 Conditions

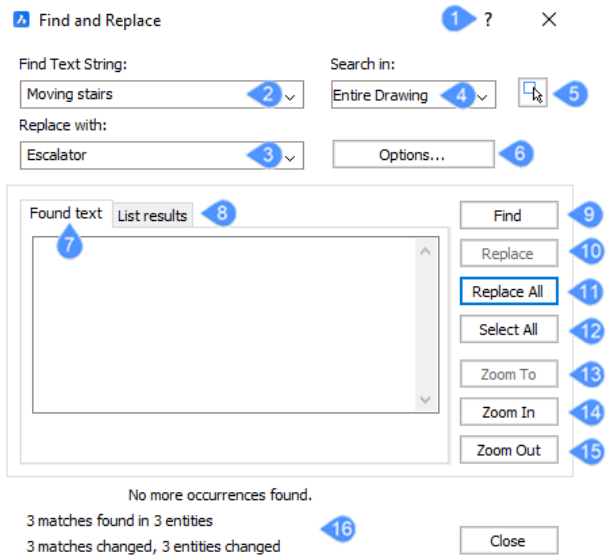
Displays all the added conditions that will be used to filter the content of the BOM table.

### 4.12.70.7 Expression Editor

Displays the Expression Editor where you can configure and preview the filtering expression.

## 4.12.71 Find and replace dialog box

The **Find and Replace** dialog box allows you to search for text in blocks, attributes, dimensions and hyperlinks, and optionally to replace it with another text string.




1 Command reference

2 Find Text

3 Replace With

4 Search In

5 Select Entities

6 Options

7 Found text

8 List results

9 Find

10 Replace

11 Replace All

12 Select All

13 Zoom To

14 Zoom In

15 Zoom Out

16 Search report

#### 4.12.71.1 Command reference

Opens the Bricsys Help article about the FIND command.

#### 4.12.71.2 Find Text

Specifies the text to be located.

#### 4.12.71.3 Replace With

Specifies the text with which to replace the located text. Do not fill in this field when only searching for text.

#### 4.12.71.4 Search In

Specifies the range of search:

- **Entire Drawing**: searches all text in the drawing.
- **Current Layout**: searches only among the selected entities.
- **Current Selection**: searches only in the current selection that can be made using button no. 5.

#### 4.12.71.5 Select Entities

Dismisses the dialog box so that you can select the entities to search. After choosing one or more entities, press ENTER to return to the dialog box.

#### 4.12.71.6 Options

Opens the **Find and Replace Options** dialog box that allows you to specify the kinds of text entities to search.

#### 4.12.71.7 Found text

Displays the text string found.

#### 4.12.71.8 List results

Displays the text string found, its location and the object type.

#### 4.12.71.9 Find

Finds the next occurrence of the text string.

Shortcut Keystroke: CTRL+F.

#### 4.12.71.1 Replace
#### 0

Replaces the found text string with the replacement string.

Shortcut Keystroke: CTRL+R.

#### 4.12.71.1 Replace All

**1**

Replaces all occurrences of the text string with the replacement string.

Shortcut Keystroke: CTRL+A.

#### 4.12.71.1 Select All

**2**

Selects all occurrences of the text string.

#### 4.12.71.1 Zoom To

**3**

Zooms to the part of the drawing containing the found text string.

#### 4.12.71.1 Zoom In

**4**

Zooms in on the center of the window by a factor of 2.

#### 4.12.71.1 Zoom Out

**5**

Zooms out from the center of the window by a factor of ½.

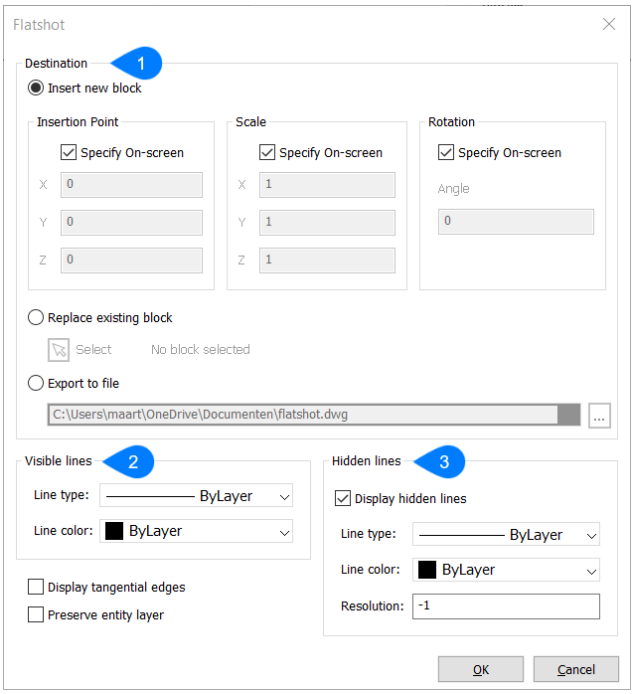
#### 4.12.71.1 Search report

**6**

Displays the number of matches found in the selected entities, but also the number of matches and entities changed after replacement.

### 4.12.72 Flatshot dialog box

The Flatshot dialog box allows you to create a flattened, hidden line representation of 3D solids, 3D surfaces, and polyface meshes as a block or a new drawing.




1 Destination

2 Visible lines

3 Hidden lines

### 4.12.72.1 Destination

Determines where the block should be placed. Several options are available

**Insert new block**
Inserts the flatshot block as a new block in the current drawing by specifying some characteristics

**Replace existing block**
Replaces a block already inserted in the drawing.

**Export to file**
Saves the flatshot block as a dwg-file trough the Select Export File dialog box.

### 4.12.72.2 Visible lines

Sets the linetype and color for the visible lines. Choose the linetype and color from the drop lists.

***Прим.:*** A new linetype can be loaded into the drawing trough the Load Linetypes dialog box by selecting Load. A new line color can be selected trough the Select Color dialog box by selecting Select color.

### 4.12.72.3 Hidden lines

Toggles the display of hidden lines and sets their properties.

**Display hidden lines**
Toggles the display of hidden lines.

**Resolution**
Resolution specifies the short entity that will be represented by a hidden line. Enter a negative value so that the program will determine this automatically. The range is 0.01 to 0.000000000001.

### 4.12.72.4 Display tangential edges

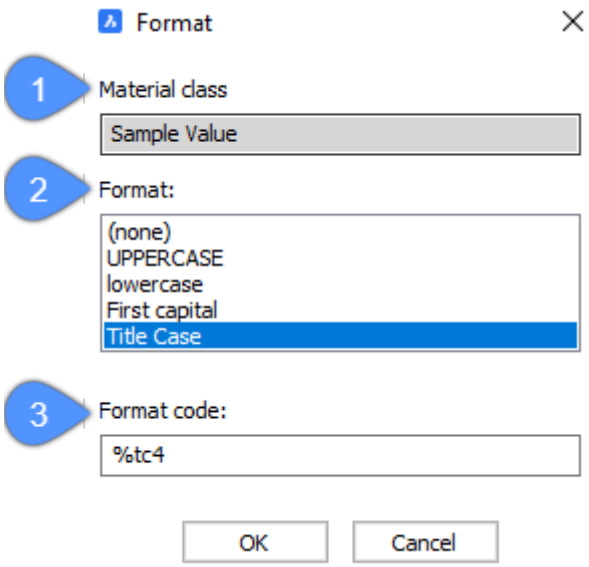
Toggles the display of tangential edges. A tangential edge is the imaginary transition line between two tangent faces.

### 4.12.72.5 Preserve entity layer

When on, maintains the layers of entities in the block.

## 4.12.73 Format dialog box

The Format dialog box allows you to specify the format of text strings in the specified column of a BOM table.




1 Material class

2 Format

3 Format code

### 4.12.73.1 Component

Specifies a component.

### 4.12.73.2 Format

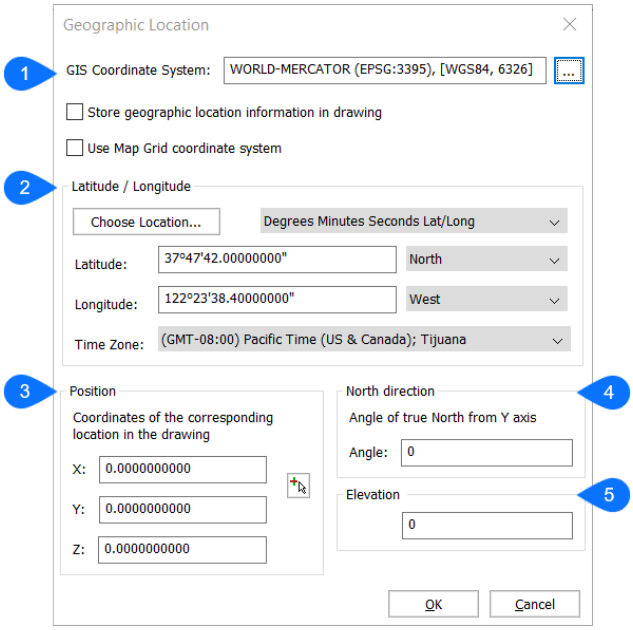
Specifies the desired format for the selected component from the list.

### 4.12.73.3 Format code

Changes depending on the format type you choose for all the materials in the BOM table.

### 4.12.74 Geographic location dialog box

The Geographic Location dialog box allows you to set the geographic location of the drawing by applying the longitude and latitude to a point in the drawing.




1 GIS Coordinate System

2 Latitude / Longitude

3 Position

4 North direction

5 Elevation

#### 4.12.74.1 GIS Coordinate System

Reports the GIS system being used by the drawing.

***Прим.:*** Click the browse icon to change the system.

**Store geographic location information in drawing**
When checked, a red dot indicates the position in the drawing.

**Use Map Grid coordinate system**
Uses the location data from the GIS Coordinate System.

#### 4.12.74.2 Latitude / Longitude

**Choose Location**
Specifies the graphic location trough the Choose Geographic Location dialog box.

**Degrees Minutes Seconds Lat/Long**
Displays the location in a Degrees, Minutes and Seconds format.

**Decimal Lat/Long**
Displays the location in a decimal format.

**Latitude**
Sets the altitude. The valid range is 0 to 90.

**Longitude**
Sets the longitude. The valid range is 0 to 180.

**Time zone**
Specifies the time zone from the Time Zone drop list.

***Прим.:*** This setting is saved to the TIMEZONE system variable.

### 4.12.74.3 Position

Specifies the position of the location in the drawing by entering coordinate values.

***Прим.:*** You can also select a specific point in the workspace to set the position.

### 4.12.74.4 North direction

Specifies the angle of the sun from north, in the context of the world coordinate system.

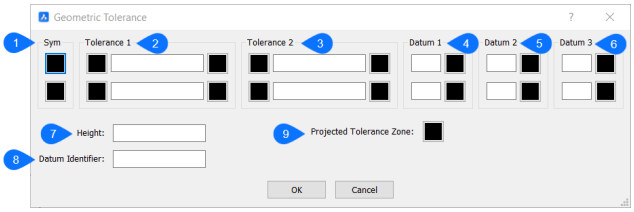
***Прим.:*** The setting is saved to the NORTHDIRECTION system variable.

### 4.12.74.5 Elevation

Specifies an elevation for the long-lat point. The value can be positive or negative.

## 4.12.75 Geometric tolerance dialog box

The Geometric Tolerance dialog box allows you to place tolerance symbols in drawings.




1 Symbol

2 Tolerance 1

3 Tolerance 2

4 Datum 1

5 Datum 2

6 Datum 3

### 4.12.75.1 Symbol

Specifies a tolerance symbol trough the Symbol dialog box.

### 4.12.75.2 Tolerance 1 & 2

Specifies the tolerance specifications (diameter, value and material condition).

**Diameter**
Toggles the diameter symbol.

**Value**
Specifies the tolerance value.

**Material condition**
Specifies the material condition trough the Material condition dialog box.

### 4.12.75.3 Datum 1, 2 & 3

Specifies the datum reference (value and material condition).

**Value**
Specifies the datum value.

**Material condition**
Specifies the material condition trough the Material condition dialog box.

**Height**
Specifies the height of the tolerance symbols.

**Datum Identifier**
Specifies datum identifier, such as Datum A.

**Projected Tolerance Zone**
Toggles the projected tolerance zone symbol.

**Tolerance Symbols**

| Symbol | Characteristic | Type |
|---|---|---|
|  | Position | Location |
|  | Concentricity or Coaxiality | Location |
|  | Symmetry | Location |
|  | Parallelism | Orientation |
|  | Perpendicularity | Orientation |
|  | Angularity | Orientation |
|  | Cylindricity | Form |

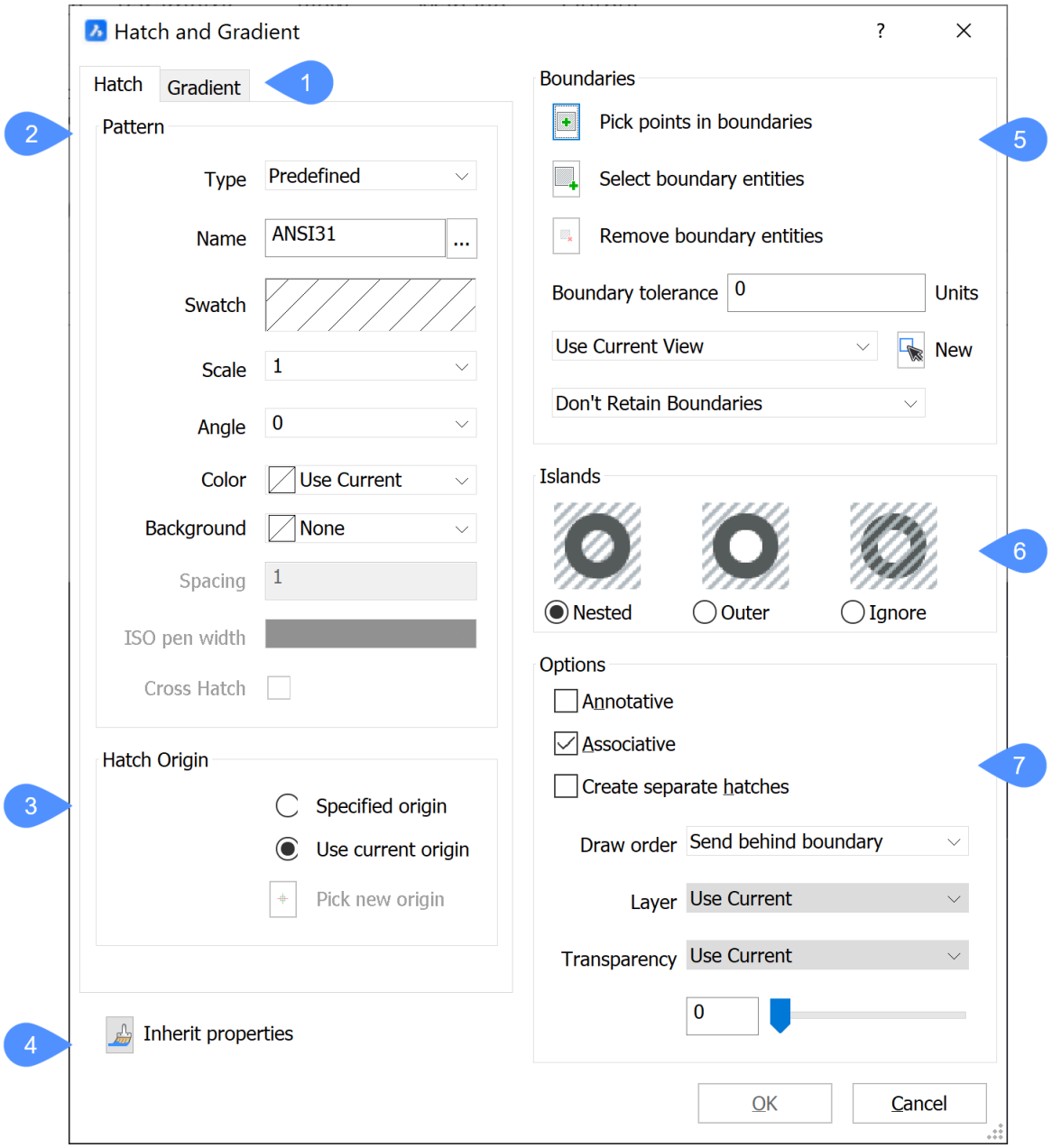
| Symbol | Characteristic | Type |
|---|---|---|
|  | Flatness | Form |
|  | Circularity or Roundness | Form |
|  | Straightness | Form |
|  | Profile of a surface | Profile |
|  | Profile of a line | Profile |
|  | Circular Run-out | Run-out |
|  | Total Run-out | Run-out |

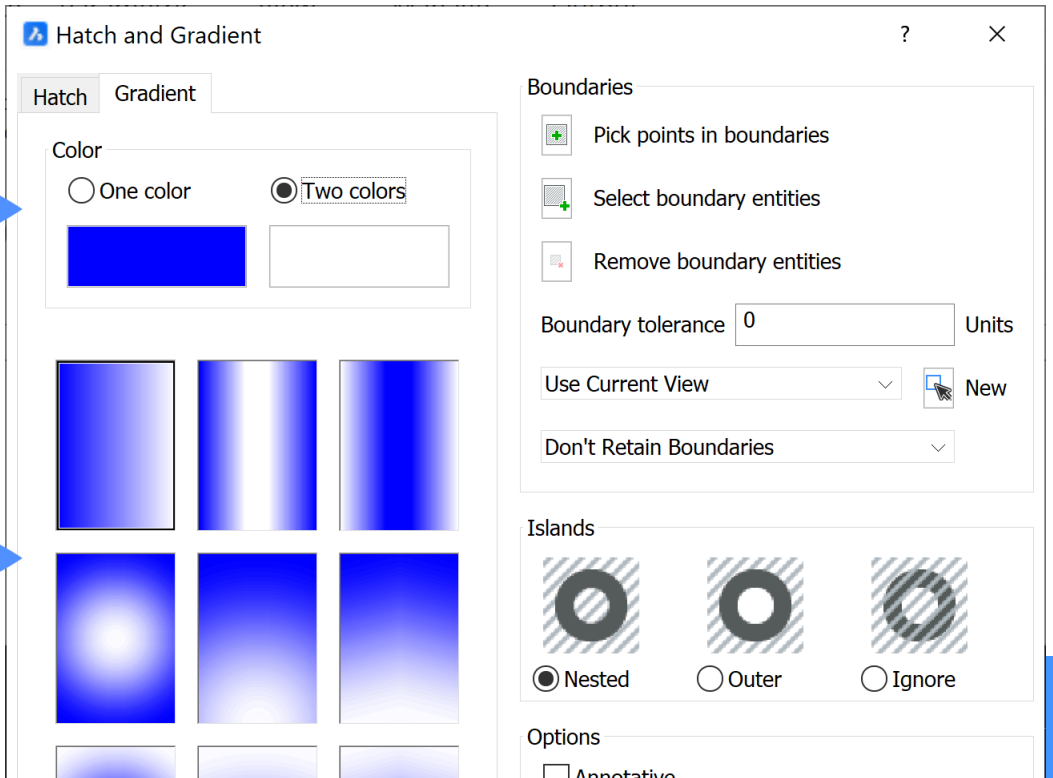
**Material Condition Symbols**

| Symbol | Definition |
|---|---|
|  | At maximum material condition (MMC), a feature contains the maximum amount of material stated in the limits. |
|  | At least material condition (LMC), a feature contains the minimum amount of material stated in the limits. |
|  | Regardless of feature size (RFS) indicates that the feature can be any size within the stated limits. |

### 4.12.76 Hatch and gradient dialog box

The Hatch and gradient dialog box allows you to fill closed 2D areas with repeating patterns or solid colors.


Hatch and Gradient
Hatch
Gradient
1
2
Pattern
Type
Predefined
Name
ANSI31
Swatch
Scale
1
Angle
0
Color
Use Current
Background
None
Spacing
1
ISO pen width
Cross Hatch
3
Hatch Origin
Specified origin
Use current origin
Pick new origin
4
Inherit properties
Boundaries
5
Pick points in boundaries
Select boundary entities
Remove boundary entities
Boundary tolerance
0
Units
Use Current View
New
Don't Retain Boundaries
Islands
6
Nested
Outer
Ignore
Options
7
Annotative
Associative
Create separate hatches
Draw order
Send behind boundary
Layer
Use Current
Transparency
Use Current
0
OK
Cancel


Hatch and Gradient
Hatch
Gradient
Color
8
One color
Two colors
9
Boundaries
Pick points in boundaries
Select boundary entities
Remove boundary entities
Boundary tolerance
0
Units
Use Current View
New
Don't Retain Boundaries
Islands
Nested
Outer
Ignore
Options

2 Hatch Pattern

3 Hatch origin

4 Inherit properties

5 Boundaries

6 Islands

7 Options

8 Gradient Color

9 Gradient pattern

10 Gradient Orientation

### 4.12.76.1 Pattern

Specifies the hatch pattern properties.

**Type**
Specifies the type of hatch.

**User defined**
Constructs the pattern from the user-defined parameters of Angle, Spacing, Color, Background and Cross Hatch.

**Predefined**
Uses the patterns defined in the iso.pat (metric units) or default.pat (imperial units) pattern definition files provided with the program.

**Custom**
Uses a pattern defined by *.pat files (limited to one pattern definition per file) created by users. The program searches for *.pat files in the folders that are defined in the SRCHPATH system variable.

**Name**
Specifies the name of the predefined hatch pattern by one of these methods. Click the Browse button to display the Hatch Pattern Palette dialog box, to choose a pattern.

***Прим.:*** This option is unavailable for the User-defined patterns.

**Swatch**
Displays the Hatch Pattern Palette dialog box.

***Прим.:*** This option is unavailable for the User-defined patterns.

**Scale**
Sets the scale factor of the pattern. To use an enlarged hatch pattern, enter a value bigger than 1.

***Прим.:*** The scale factor of hatch patterns is generally the same as that for text and linetypes.

**Angle**
Sets the angle of the pattern.

***Прим.:*** This angle is measured relative to the positive x-axis of the current UCS.

**Color**
Sets the color of the hatch pattern lines.

***Прим.:*** For the Use Color option, the current color specified by the CECOLOR system variable is used.

**Background**
Sets the color of the background of the hatch.

**Spacing**
Specifies the distance between hatch lines.

***Прим.:*** This option is available for only user defined patterns.

**ISO pen width**
Specifies the lineweight.

***Прим.:*** This option is available only for ISO hatch patterns.

**Cross Hatch**
Determines whether the pattern is crossed (repeated at 90 degrees to the original).

***Прим.:*** This option is available only for user defined patterns.

### 4.12.76.2 Hatch origin

Specifies the origin of the hatch. Either the current or a new specified origin can be chosen.

### 4.12.76.3 Inherit properties

Copies the properties of another existing hatch pattern to use with this pattern.

### 4.12.76.4 Boundaries

Specifies the boundary conditions of the hatch.

**Pick points in boundaries**
Specifies the closed areas (boundaries) in which to place the pattern.

***Прим.:*** You cannot pick an area that already contains a hatch pattern, or areas that are not closed (whose gap is larger than the value specified by Boundary Tolerance).

**Select boundary entities**
Selects the entities that make up the hatching boundary to constrain the extent of the pattern.

***Прим.:*** This option allows you to hatch closed areas, including ones that already contain a hatch. It hatches open areas whose gap is less than the value specified by Boundary Tolerance.

**Remove boundary entities**
Remove entities from the detected boundary set.

**Boundary tolerance**
Specifies the largest gap the program ignores when hatching a boundary that is not fully closed.

- 0 - (Default): the tolerance is set by the application, based on the current view size. When zoomed in closely, boundary detection will fail; when zooming out further so the contour 'looks' closed, the boundary will become detected
- Any Value : defines the maximum gap in drawing units.

***Прим.:*** The value is saved in the HPGAPTOL system variable.

**Use current view / boundary set**
Specifies where the program should search for entities that make up the boundary.

**New**
Creates a new selection set of entities that make up the pattern boundary.

***Прим.:*** The next step is to click the Pick Points in Boundaries button to choose the hatch area.

**Retain boundaries**
Determines what happens to boundaries.

**Don't Retain Boundaries**
Removes the boundary after the hatch is created.

**Retain Boundaries as Polylines**
Retains boundaries and turns them into polylines.

**Retain Boundaries as Regions**
Retains boundaries and turns them into regions (Pro or Platinum versions only).

### 4.12.76.5 Islands

Specifies how BricsCAD will respond when other boundaries are present within the closed hatch boundary.

**Nested**
When a closed hatch boundary contains other boundaries, BricsCAD hatches alternating areas.

**Outer**
When a closed hatch boundary contains other boundaries, BricsCAD hatches only the outermost area.

**Ignore**
When a closed hatch boundary contains other boundaries, BricsCAD hatches all interior areas, as if they were not present.

### 4.12.76.6 Options

**Annotative**
Toggles the Annotative property of the hatch. When turned on, BricsCAD applies the current annotative scale.

***Прим.:*** When this option is turned on, the Associative option is unavailable.

**Associative**
Toggles the associativity of hatch patterns: when the boundary is changed, the pattern automatically updates itself.

***Прим.:*** This option is not available for annotative hatches.

**Create separate hatches**
Toggles how multiple hatches are treated. If turned on, BricsCAD creates a separate hatch entity for each boundary in the selection set.

**Draw order**
Specifies where the pattern should be placed relative to overlapping entities.

**Layer**
Specifies the layer on which the hatch should be placed.

**Transparency**
Set the transparency property of the hatch.

***Прим.:*** For the Use current option, the transparency value, defined by the CETRANSPARENCY system variable, is applied.

### 4.12.76.7 Gradient Color

Specifies the gradient color. The gradient can either exist of one color to create a solid fill hatch or exist out of 2 colors.

### 4.12.76.8 Gradient pattern

Specifies the pattern of the gradient.

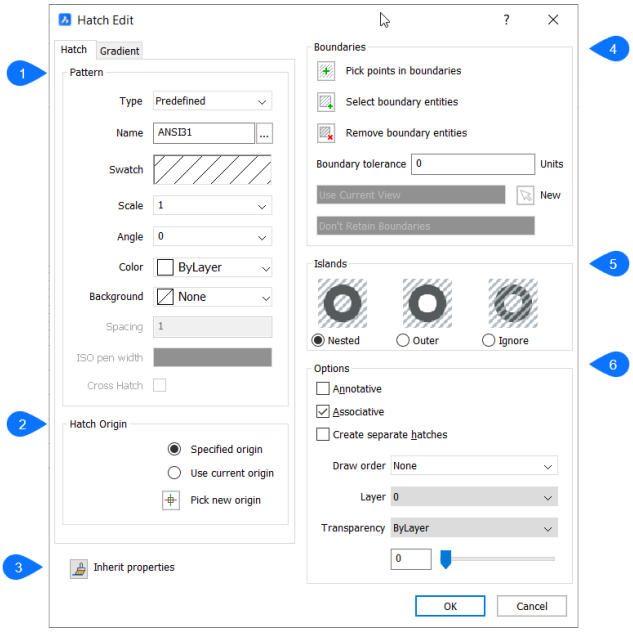
### 4.12.76.9 Gradient Orientation

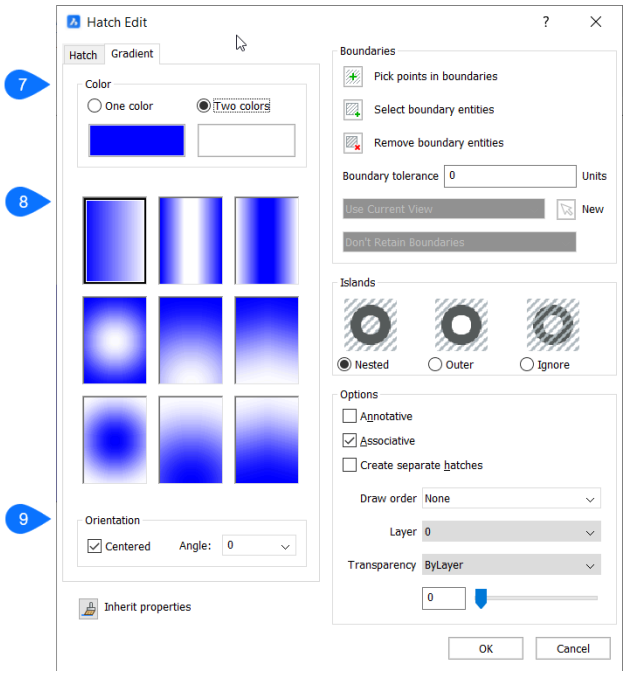
Specifies how the fill is placed in the boundary.

- Centered: centers gradient fill in the area to be filled; when off, starts the fill from the upper left of the boundary.
- Angle: select an angle of the gradient fill, relative to the current UCS; choose from increments on 15 degrees, or else enter any other value for the angle.

## 4.12.77 Hatch Edit dialog box

The Hatch Edit dialog box allows you to edit an already existing hatch in the drawing. All options resemble the options of the Hatch and Gradient dialog box.






1 Pattern

2 Hatch origin

3 Inherit properties

4 Boundaries

5 Islands

6 Options

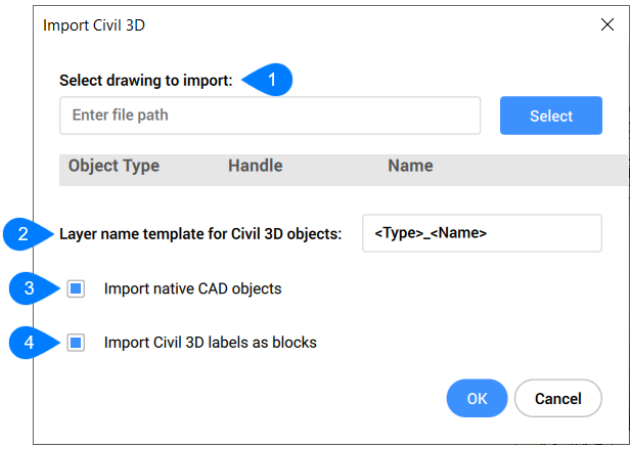
7 Color

8 Pattern

9 Orientation

### 4.12.78 Import Civil 3D dialog box

The Import Civil 3D dialog box allows you to create BricsCAD civil entities from an Autodesk® Civil 3D drawing.




1 Select drawing to import

2 Layer name template for Civil 3D objects

3 Import native CAD objects

4 Import Civil 3D labels as blocks

#### 4.12.78.1 Select drawing to import

Specifies the drawing to import by clicking the Select button, which opens the Open File dialog box.

#### 4.12.78.2 Layer name template for Civil 3D objects

Edits the template that is used for the layer name of all Civil 3D objects.

#### 4.12.78.3 Import native CAD objects

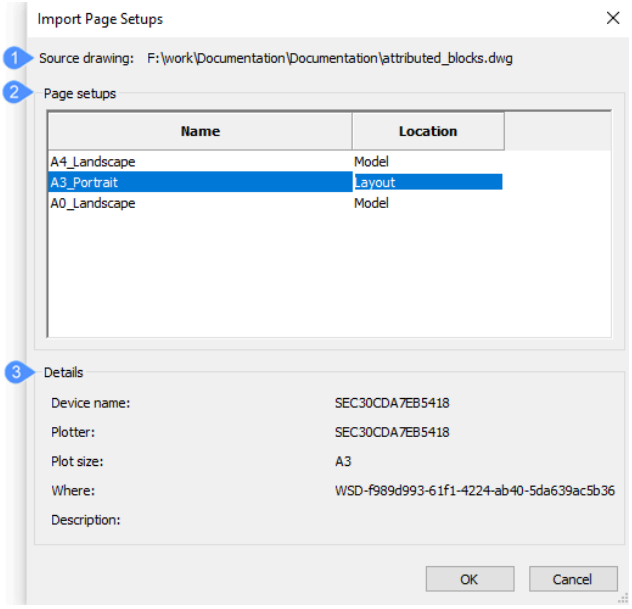
Toggles the option to also import native CAD objects.

#### 4.12.78.4 Import Civil 3D labels as blocks

Toggles the option to import all Civil 3D labels as blocks.

### 4.12.79 Import page setups dialog box

The **Import Page Setups** dialog box allows you to select one or more page setups (using the CTRL key) from the list and import them into the current drawing.




1 Source Drawing

2 Page setups list

3 Details

### 4.12.79.1 Source Drawing

Displays the source of the drawing used to import the page setup(s).

### 4.12.79.2 Page setups list

Specifies the name and the location of each page setup from the list:

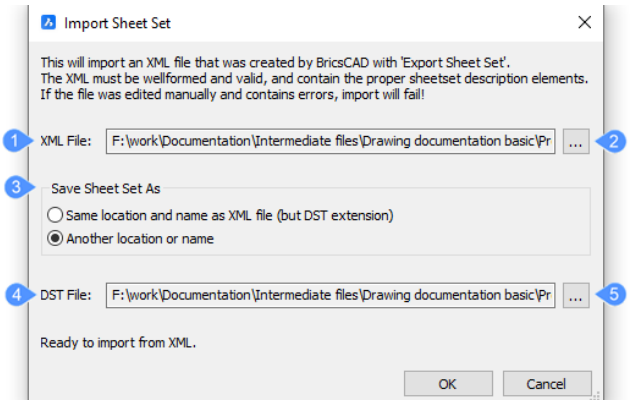
- **Name**: specifies the names of page setups in the drawing. When the drawing contains no page setups, then the dialog box is blank.
- **Location**: specifies the layout and model locations of page setups.

### 4.12.79.3 Details

Includes information about the device name, plotter, plot size, location and a description field.

## 4.12.80 Import sheet set dialog box

The **Import Sheet Set** dialog box allows you to import a sheet set file that has been created using the **Export Sheet Set As** dialog box.




1 XML File path

2 Select XML File

3 Save Sheet Set As

4 DST File path

5 Save Sheet Set As

### 4.12.80.1 XML File path

Displays the path of the *.xml file after selecting it using the **Select XML File** button.

### 4.12.80.2 Select XML File

Opens the **Select XML File** dialog box.

### 4.12.80.3 Save Sheet Set As

Choose whether to save the imported XML file as DST file using:

- The same location and name as XML file. Sections named 4 and 5 are unavailable.
- Another location or name. For the second option, click the button number 5.

### 4.12.80.4 DST File path

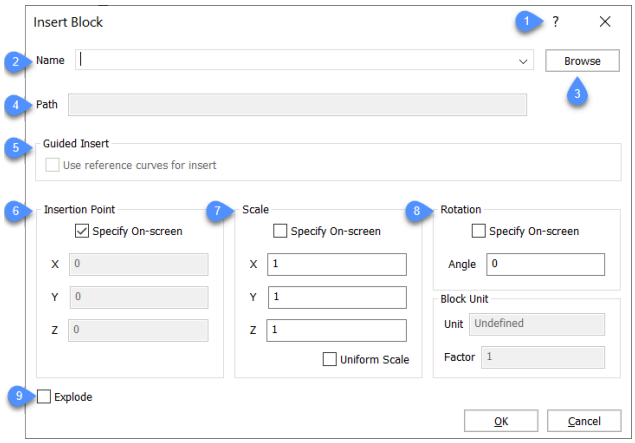
When you choose to save the imported XML file as DST file to a new location or name is displayed the path of this file.

### 4.12.80.5 Save Sheet Set As

Opens the **Save Sheet Set As** dialog box.

## 4.12.81 Insert block dialog box

The **Insert Block** dialog box allows you to insert blocks into the current drawing. You can also insert entire DWG and DXF files as blocks into the drawing.




***Прим.:*** If the block contains attributes, then you are prompted to enter values for the attributes in the Command line or through **Edit Attributes** dialog box, depending on the value of the value of the ATTDIA (attribute dialog) variable.

1 Command reference

2 Name

3 Browse

4 Path

5 Guided Insert

6 Insertion Point

7 Scale

8 Rotation

9 Explode

### 4.12.81.1 Command reference

Opens the Bricsys Help article about the INSERT command.

### 4.12.81.2 Name

Specifies the name of the block, whose definition exists in the drawing, or is a DWG or DFX file on the computer or network.

### 4.12.81.3 Browse

Selects a DWG or DXF file from your computer or network. Opens the **Select Drawing File** dialog box.

### 4.12.81.4 Path

Displays the path to the block, if the block was opened from a DWG or DXF file.

#### 4.12.81.5 Guided Insert

When a dwg is selected that contains guide curves, the **Guided Insert** option becomes selectable. It is used for the insertion of parametric blocks.

- **Use reference curves for insert**: Specifies if a reference should be used for insert or not.

#### 4.12.81.6 Insertion Point

Specifies the insertion point of the block in the drawing.

***Прим.:*** Insertion points are typically chosen at the lower-left corner of the block, in the center of the block, or at entity in the block that represents a connection point.

#### 4.12.81.7 Scale

Scales the block:

- Values larger than 1 make the block larger.
- 1 inserts the block at actual size.
- Values smaller than 1 make the block smaller.
- Values less than zero flip the block, like mirroring it.
- **X, Y, Z**: Defines the scaling of the block:
  - X specifies the scale factor along the x axis. Enter a negative value to mirror the block about the y axis.
  - Y specifies the scale factor along the y axis. Enter a negative value to mirror the block about the x axis.
  - Z specifies the scale factor along the z axis.
- **Uniform Scale**: Toggles whether the same scale factor is used for all axes.

#### 4.12.81.8 Rotation

Specifies the rotation angle of the block about its insertion point, starting with the x-axis as 0 degrees.

***Прим.:*** Positive angles rotate the block counter-clockwise and negative angles rotate the block clockwise.

#### 4.12.81.9 Block unit

Controls the automatic scaling of the block with respect to the INSUNITS system variable of the current drawing.

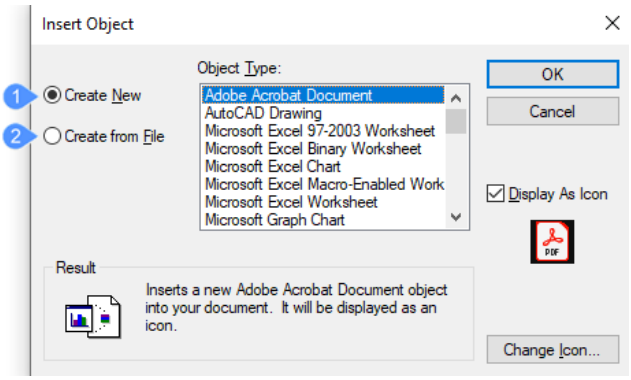
#### 4.12.81.1 Explode

**0**

Toggles whether the block is inserted exploded.

### 4.12.82 Insert object dialog box

The Insert Object dialog box allows you to insert a linked or embedded document in the current drawing.

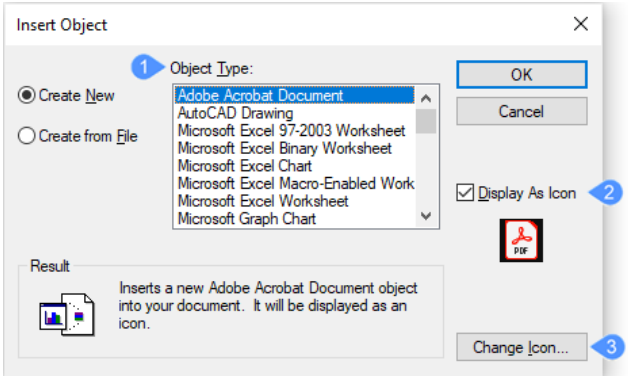



1 Create new Object

2 Create from File

***Прим.:*** The object types listed varies according the software installed on your computer.

### 4.12.82.1 Create new Object




1 Display as icon

2 Change icon

**Display as icon**
Displays the object as an icon. The icon is related to the document type and is defined by source application.

**Change icon**
Opens the Change Icon dialog box.

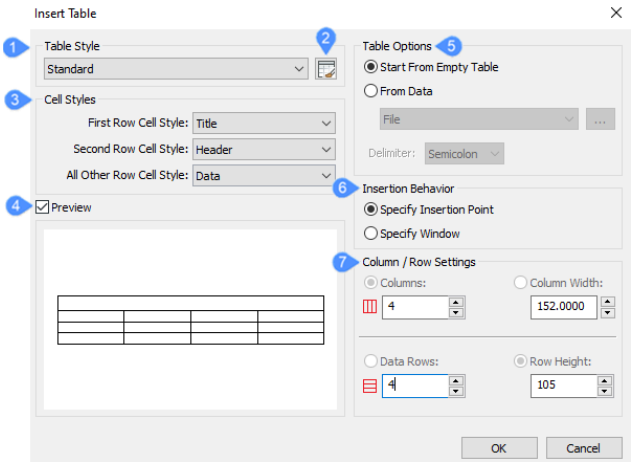
### 4.12.82.2 Create from File

**Link**
Toggles linkage to the source document:

- **On**: links the object in the drawing to the source file, so that when the source file is changed, the linked object also changes.
- **Off**: does link not the object.

### 4.12.83 Insert table dialog box

The **Insert table** dialog box allows you to create a table entity in the drawing either from scratch or from data in an external file or a data link.




1 Table Style

2 Table Style Explorer

3 Cell Styles

4 Preview

5 Table Options

6 Insertion Behavior

7 Column / Row settings

#### 4.12.83.1 Table Style

Chooses an existing table style from the drop-down list.

#### 4.12.83.2 Table Style Explorer

Allows you to create or edit table styles. Opens the **Table Styles** from the **Drawing Explorer** dialog box.

#### 4.12.83.3 Cell Styles

A table style can have different sub-styles for the first two rows named **Title**, **Header** and **Data**:

- **First Row Cell Style**: chooses a sub-style from the drop-down list, which by default is **Title**. A **Title** row is the topmost row of the table.
- **Second Row Cell Style**: chooses a sub-style from the drop-down list, which by default is **Header**. A **Header** row is the first row of a column.
- **All Other Row Cell Style**: chooses a sub-style from the drop-down list, which by default is **Data**. A **Data** row is all other rows in the table.

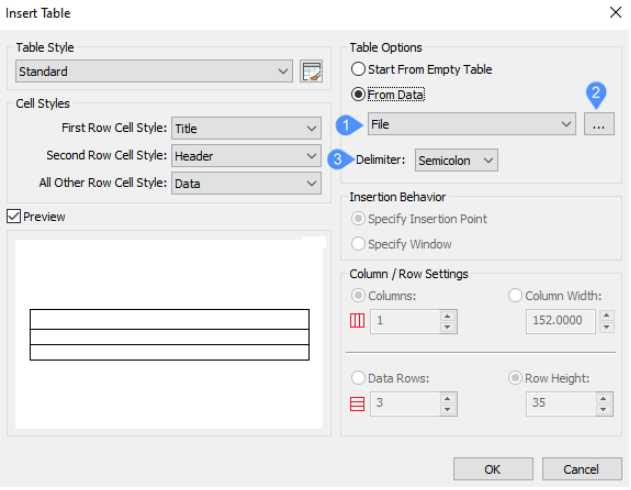
### 4.12.83.4 Preview

Displays a preview of the table depending on the options and settings you set in this dialog box.

### 4.12.83.5 Table Options

Determines the source of the new table:

- **Start from Empty Table**: creates a new, empty table with the parameters listed below.
- **From Data**: reads data from a file or a datalink.




a File
b Open
c Delimeter

- **File**

  Read data for the new table from a file.

- **Open**

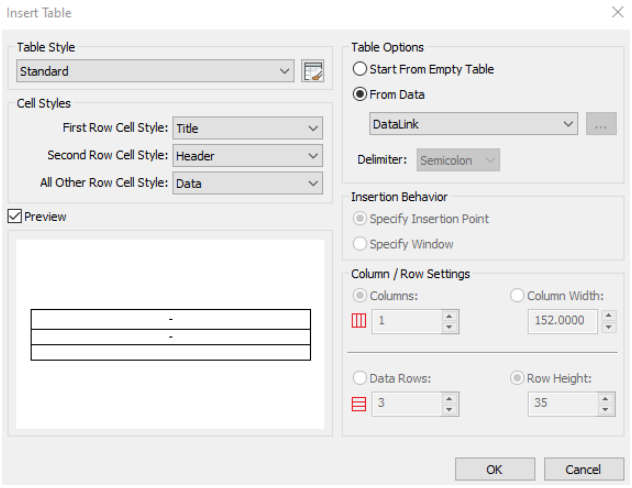
  Displays the **Open** dialog box from where you will select the file.

- **Delimeter**

  Specifies the delimiter character used to separate fields in CSV data files. Select from the drop-down list the delimiter you want to use.

  ***Прим.:*** Available only for the **File** option.

- **DataLink**




When you select to read data from a datalink the **Datalink Manager** dialog box opens.

### 4.12.83.6 Insertion Behavior

(This option is available only when you create a new empty table.)

- **Specify Insertion Point**: specifies where the upper-left corner of the table will be inserted in the drawing.
- **Specify Window**: specifies the area that will be used by the table, this is useful when you need the table to fit a specific are, such as in a revision table or title block.

***Прим.:*** As you move the mouse, a fictive table dynamically resizes itself.

### 4.12.83.7 Column / Row settings

(This option is available only when you create a new empty table.)

Specifies the number of columns or rows and the cell width or height.

Choose one of:

- **Columns**: specifies the number of columns in the table
- **Column Width**: specifies the initial width of all columns

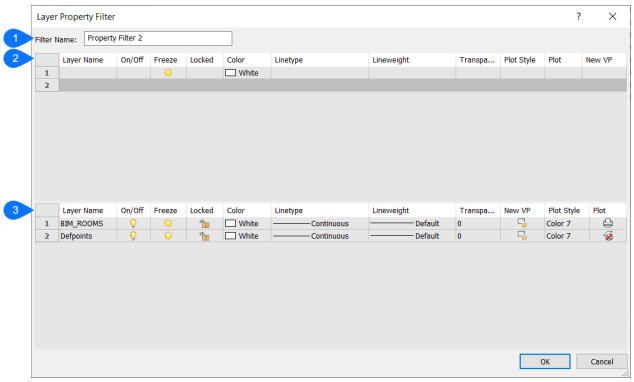
And choose one of:

- **Data Rows**: specifies the number of data rows in the table; the title and header rows are added automatically by the program
- **Row Height**: specifies the initial height of all rows.

The number of columns and rows and size of cells can be adjusted later, after the table is inserted.

## 4.12.84 Layer Property Filter dialog box

The Layer Property Filter Dialog Box allows you to create a new layer filter definition, based on the properties of the layer.




1 Filter Name

2 Filter Properties

3 Filtered Layers

### 4.12.84.1 Filter Name

Sets the name of the filter.

### 4.12.84.2 Filter Properties

Sets the properties to filter the layers. Layers that fulfill all indicated properties of the filter will be added to the layer filter.

**Layer Name**

Filters the layers based on their name. You can use wild cards to specify more than one layer, for example Doors-* for Doors-Exterior and Doors-Interior.

- # - matches any digit. For example, ## matches 12 and 34 but not 5 or 123 (ASCII code: 35).
- @ - matches any alphabetic character, so not digits or symbols. For example, @@@@@ matches all 5-letter layer names. (ASCII code: 64).
- . - matches non-alphanumeric characters, so symbols but not letters or numbers. For example, .* matches all layer names starting with a non-alphanumeric character such as _SITEPLAN and +1_LEVEL (ASCII code: 46).
- * - matches any string, including letters, numbers, and characters. For example, A* matches all layer names starting with the letter A (ASCII code: 42).
- ? - matches any single character. For example, ?AB* matches L**AB**ORATORY and C**AB**ARET but not ST**AB**LE, because STABLE begins with two letters and ? specifies a single letter (ASCII code: 63).
- ~ - negative match, in which any string but the pattern is returned. For example, ~*A* matches all layer names not containing the letter A (ASCII code: 126).
- [ ] - Matches any one of the characters enclosed inside the square brackets. For example, LA[YB] matches LA**Y**and LA**B**(ASCII code: 91 93).
- [~] - Specifies any one of the characters not found in the square brackets. For example, LA[~YB] matches L**AD**but not L**AY**or L**AB** (ASCII code: 91 126 93).

- [-] - Specifies a range for a single character in the layer name. For example, LAY-[A-E] matches any combination of LAY- and A through E, such as LAY-**A**... LAY-**E**but not LAY-F (ASCII code: 91 45 93).
- ‘ - Matches the next string exactly to find a layer name that contains a wild-card character when it is prefixed by the reverse quote character. Use ` to search for layer names that include characters like ? and ~. For example, `~LAY* matches ~LAY-1,~LAYER-1, and ~LAY-B (ASCII code: 96).

***Прим.:*** ASCII code: if a character is not available on the keyboard, press and hold the Alt key and then type the ASCII code on the numeric keypad.

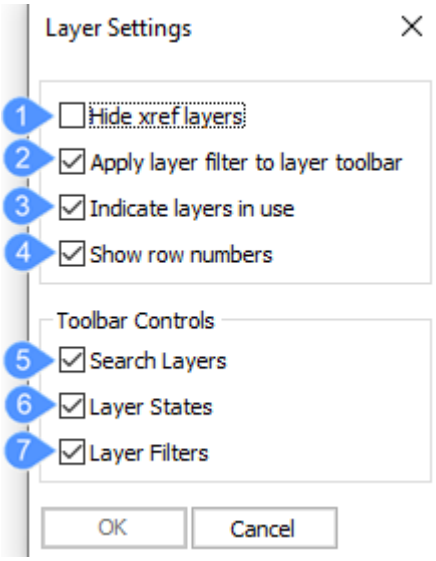
**Properties**
Specifies which layers to include according to their properties, such as freeze and color.

### 4.12.84.3 Filtered Layers

Displays the layers that fulfill the requirements of the filter.

## 4.12.85 Layer settings dialog box

The **Layer Settings** dialog box allows you to apply settings on the **Layers** Panel.




1 Hide xref layers

2 Apply layer filter to layer toolbar

3 Indicate layers in use

4 Show row numbers

5 Search Layers

6 Layer States

7 Layer Filters

### 4.12.85.1 Hide xref layers

When on, hide layers that came from externally referenced drawings.

### 4.12.85.2 Apply layer filter to layer toolbar

When on, shortens the list of layers displayed, according to the conditions of the filter. When off, displays all layer names.

### 4.12.85.3 Indicate layers in use

When on, displays icons in the **Current** column that indicate whether the layer has entities assigned to it. Unused layers can be purged.

### 4.12.85.4 Show row numbers

Toggles the display of the number at the start of each row.

### 4.12.85.5 Search Layers

Toggles the display of the **Search** field on and off.

### 4.12.85.6 Layer States

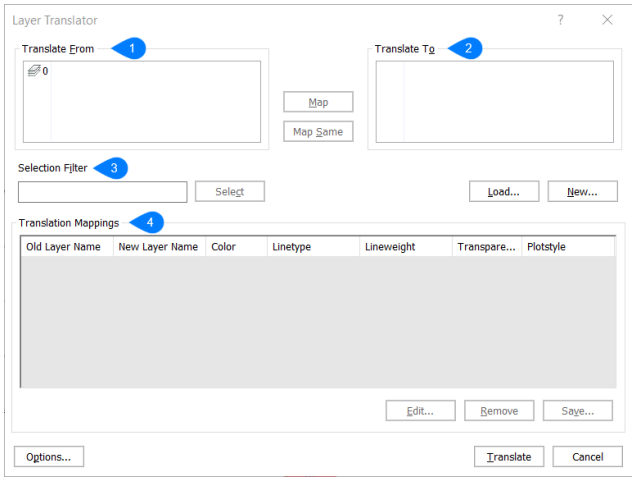
Toggles the display of the **Layer States** drop-down list.

### 4.12.85.7 Layer Filters

Toggles the display of the **Layer Filters** drop-down list.

## 4.12.86 Layer translator dialog box

The Layer Translator dialog box allows you to apply layer names and properties from another drawing to the current drawing.




1 Translate from

2 Translate to

3 Selection filter

4 Translation mappings

### 4.12.86.1 Translate from

Lists all the layer names in the current drawing.

***Прим.:*** You can specify the layers to which you want them to map by loading layer information from an existing DWG, DWS, or DWT file.

***Прим.:*** New layers can be created by simply entering the layer name and properties to which you want to map existing layer.

### 4.12.86.2 Translate to

Specifies which layer you want the chosen layer(s) to map to.

***Прим.:*** Select one or more layers from the current drawing, the list on the left, to map to a layer from the list on the right. If you choose Map Same any layer names in the current drawing with corresponding names in the list on the right will inherit the properties from the list on the right.

### 4.12.86.3 Translation mappings

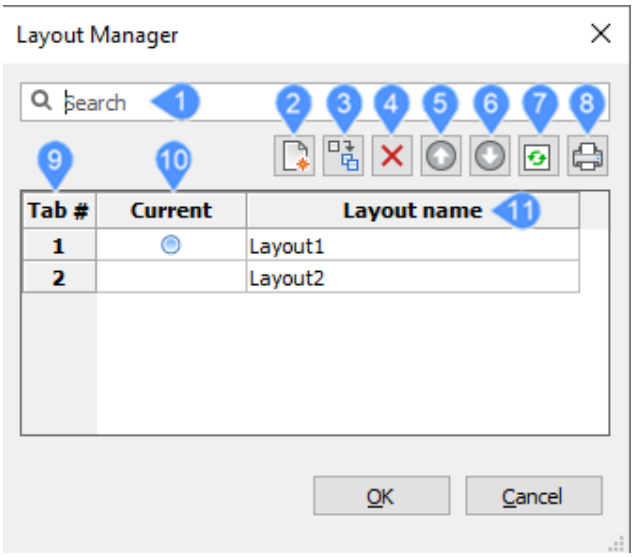
Displays an overview of which layer and its properties are translated to another layer.

### 4.12.86.4 Options

Displays the Options dialog box, which offers additional controls for layer mapping.

## 4.12.87 Layout manager dialog box

The Layout Manager dialog box allows you to create, copy and delete layouts.




1 Search

2 Add new layout

3 Copy selected layout

4 Delete

5 Move up

6 Move down

7 Clear selection

8 Publish

9 Tab

10 Current

11 Layout name

#### 4.12.87.1 Search

Searches for a layout name and displays only the searched layout. This is useful when a drawing contains many layouts.

#### 4.12.87.2 Add new layout

Creates a new layout.

#### 4.12.87.3 Copy selected layout

Makes a copy of the selected layouts and adds them to the end of the list.

#### 4.12.87.4 Delete

Deletes the selected layouts.

#### 4.12.87.5 Move up

Moves the selected layouts up the list.

#### 4.12.87.6 Move down

Moves the selected layouts down the list.

#### 4.12.87.7 Clear selection

Removes highlighting from layout names, thereby unselecting them.

#### 4.12.87.8 Publish

Opens the **Publish** dialog box.

#### 4.12.87.9 Tab

Displays the number of the layout.

#### 4.12.87.1 Current

**0**

Click in the **Current** column to make the layout current.

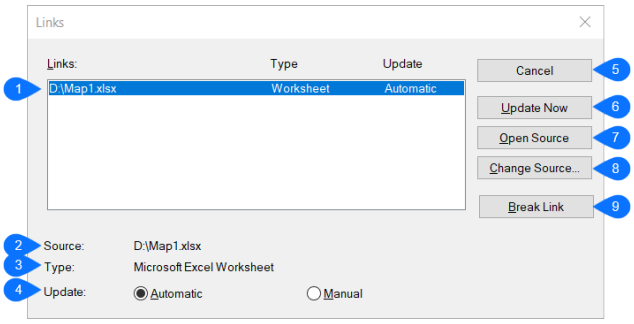
#### 4.12.87.1 Layout name

**1**

Displays the name of the layout.

### 4.12.88 Links dialog box

The Links dialog box allows you to edit the links of OLE entities that were embedded or linked to drawings.




1 Links

2 Source

3 Type

4 Update

5 Cancel

6 Update Now

7 Open Source

8 Change Source

9 Break Link

#### 4.12.88.1 Links

Provides a list of all links loaded in the file.

#### 4.12.88.2 Source

Reports the path to the source file name.

#### 4.12.88.3 Type

Reports the file type.

#### 4.12.88.4 Update

Determines how the OLE entity in the drawing is updated when the source changes.

#### 4.12.88.5 Cancel

Cancels the linking operation.

#### 4.12.88.6 Update Now

Updates linked OLE entities.

#### 4.12.88.7 Open Source

Opens the source file in the source application. This allows you to edit the source file.

### 4.12.88.8 Change Source

Changes the source for the linked OLE entity.

### 4.12.88.9 Break Link

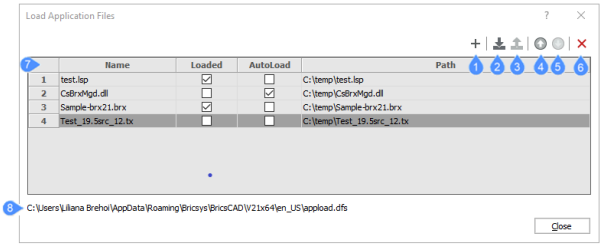
Breaks the link with the source file.

***Прим.:*** When the link is broken, the OLE entity in the drawing no longer changes when the source file changes.

## 4.12.89 Load application files dialog box

The **Load Application Files** dialog box allows you to load and unload the following applications type to run inside BricsCAD:

- LSP - LISP applications from BricsCAD and other CAD programs.
- DES - DES LISP applications.
- BRX - BricsCAD runtime extension applications.
- TX - Teigha extension applications, from Open Design Alliance.
- ARX - AutoCAD runtime extension applications from Autodesk.
- DBX - DBX applications.
- DRX - Drawing runtime extension applications.
- DVD - Visual Basic applications.
- MNL - LISP expressions for menu files.




1 Add application file

2 Load selected application

3 Unload selected application

4 Move application upward

5 Move application downward

6 Remove selected application

7 Application Files List

8 Application File Path

### 4.12.89.1 Add application file

Opens the **Select Application Files** dialog box.

### 4.12.89.2 Load selected application

Loads the selected application(s) into BricsCAD. Only compatible applications with BricsCAD will be loaded.

### 4.12.89.3 Unload selected application

Unloads the selected application(s) from BricsCAD. The LISP applications have to be removed in order to be unloaded in the current BricsCAD session.

### 4.12.89.4 Move application upward

Moves the selected application(s) up in the list.

### 4.12.89.5 Move application downward

Moves the selected application(s) down in the list.

### 4.12.89.6 Remove selected application

Removes the selected application(s) from the list.

### 4.12.89.7 Application Files List

Displays a list with application files that are available for load, unload, move up and down or remove.

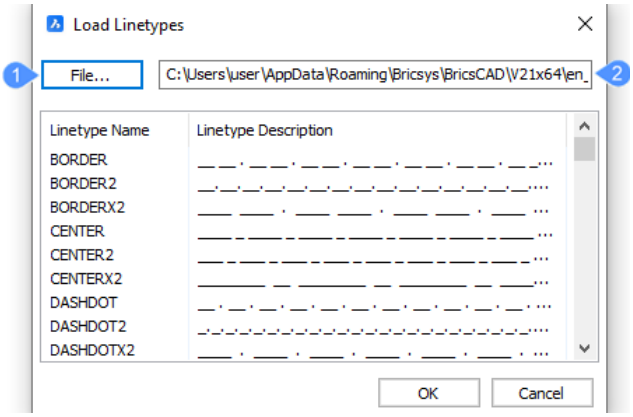
- **Name**: specifies the name and the extension of the application file from the list.
- **Loaded**: specifies if a file is loaded or unloaded in the current BricsCAD session.
- **Autoload**: specifies if a file will be loaded automatically in the next BricsCAD session.
- **Path**: displays the location folder of the added files.

### 4.12.89.8 Application File Path

Displays the location folder for appload.dfs file that contains all applications added, using the **Select Application Files** dialog box. You can open the file using Notepad.

## 4.12.90 Load linetypes dialog box

The **Load Linetypes** dialog box allows you to load additional linetype definitions.




1 File

2 File's path

#### 4.12.90.1 File

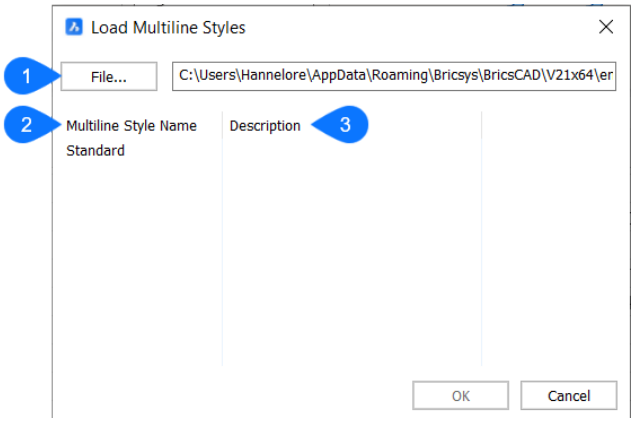
Opens the **Choose a Linetype File** dialog box.

#### 4.12.90.2 File's path

Displays the path of the selected *.lin (linetype) file.

### 4.12.91 Load Multiline Styles dialog box

The Load Multiline Style dialog box allows you to load multiline styles from a multiline (*mln) file.




1 File

2 Multiline Style Name

3 Description

#### 4.12.91.1 File

Displays the Choose a Multiline Style File dialog box where you can select a multiline style file to open.

#### 4.12.91.2 Multiline Style Name

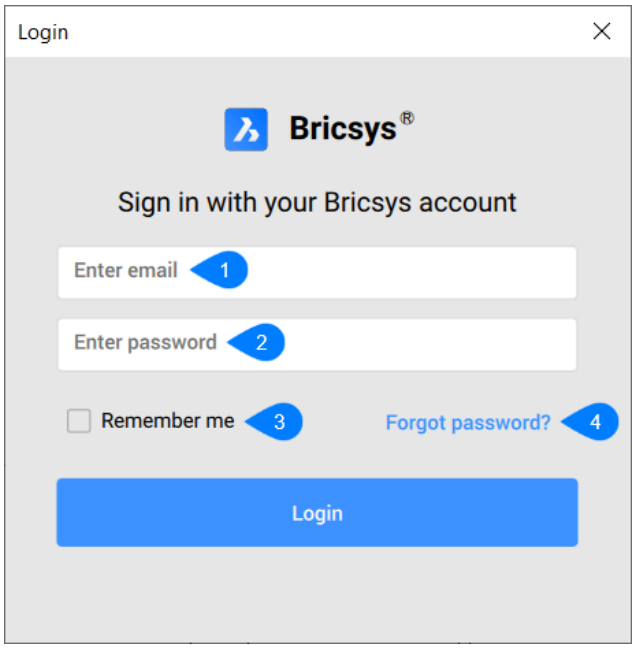
Displays the names of the available multiline style files in the drawing. You can select a multiline style file from this list.

#### 4.12.91.3 Description

Displays a description of the multiline style files, if available.

### 4.12.92 Login dialog box

The Login dialog box allows you to login to BricsCAD with your personal Bricsys account.




1 Enter email

2 Enter password

3 Remember me

4 Forgot password?

#### 4.12.92.1 Enter email

Specifies the email address connected to your Bricsys account.

#### 4.12.92.2 Enter password

Specifies the password.

#### 4.12.92.3 Remember me

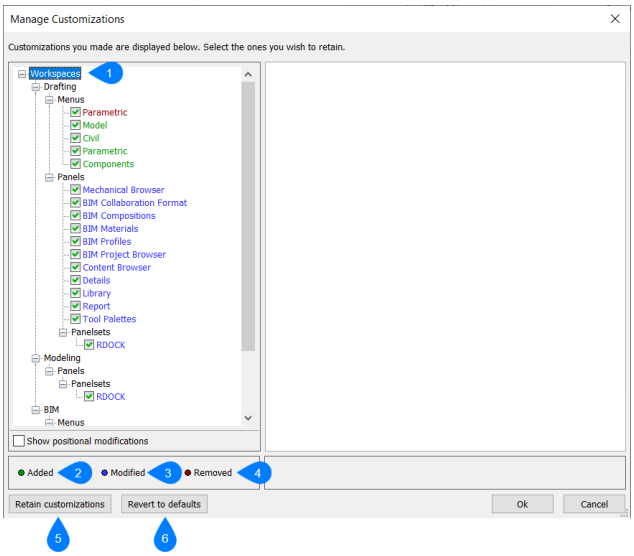
Check this box to keep being logged in.

#### 4.12.92.4 Forgot password?

Redirects you to an internet browser to set up a new password.

### 4.12.93 Manage Customizations dialog box

The Manage Customizations dialog box provides a list of all customizations for confirming and reversing changes made to the user interface.




1 Customizations

2 Added

3 Modified

4 Removed

5 Retain customizations

6 Revert to defaults

### 4.12.93.1 Customizations

Lists all added, modified or removed customizations.

***Прим.:*** You can check a customization to retain the modification in the CUI.

### 4.12.93.2 Added

Items you have created display in green.

### 4.12.93.3 Modified

Items you have modified display in blue.

### 4.12.93.4 Removed

Items you have removed display in red.

### 4.12.93.5 Retain customizations

Keeps your local default CUI as it is.

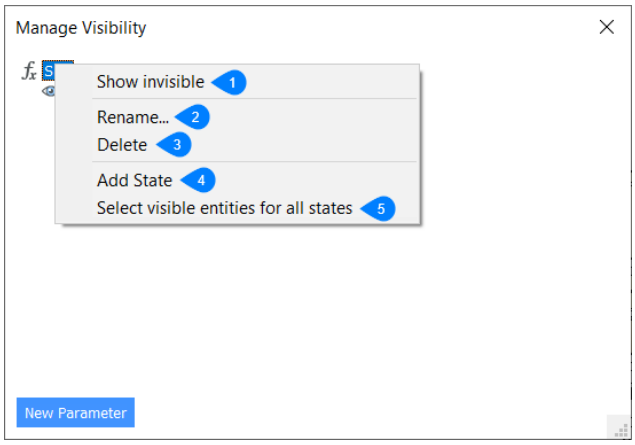
### 4.12.93.6 Revert to defaults

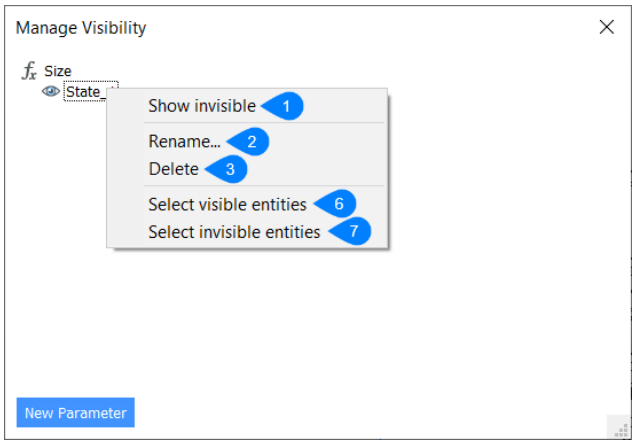
Reverts your local default CUI to the default one installed by BricsCAD in:

*C:\Program Files\Bricsys\BricsCAD V21 en_US\UserDataCache\Support\en_US\default.cui.*

### 4.12.94 Manage visibility

The Manage Visibility dialog box displays a list of visibility states in the drawing.




1 Show invisible

2 Rename

3 Delete

4 Add State

5 Select visible entities for all states

6 Select visible entities

7 Select invisible entities

#### 4.12.94.1 Show invisible

Shows all invisible entities grayed-out.

#### 4.12.94.2 Rename

Displays the Rename dialog box. Type a name to replace the existing name.

### 4.12.94.3 Delete

Deletes the parameter and its visibility states.

### 4.12.94.4 Add State

Displays the Rename dialog box. Type a name for the new state to replace the default name.

### 4.12.94.5 Select visible entities for all states

First choose Show invisible, then select the entities to remove all visibility states from the selected parameter.

### 4.12.94.6 Select visible entities

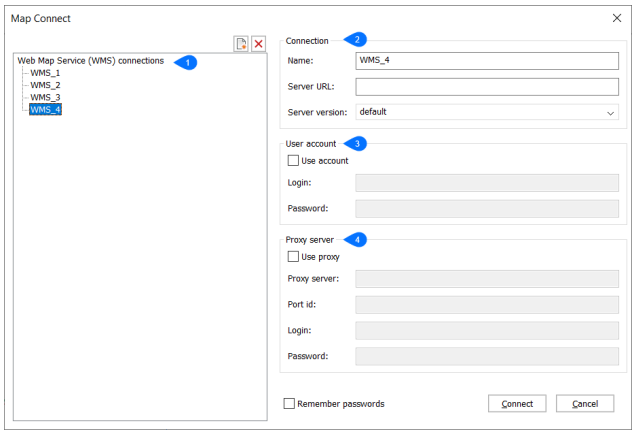
Visualizes all entities that are selected.

### 4.12.94.7 Select invisible entities

Hides all entities that are selected.

## 4.12.95 Map Connect dialog box

The Map Connectdialogboxallowsyoutoset up a connection with a Web Map Service.




1 WMS connections

2 Connection

3 User account

4 Proxy server

### 4.12.95.1 WMS connections

Displays a list of the different possible Web Map Service connections.

### 4.12.95.2 Connection

Allows you to create a Web Map Service (WMS) connection.

**Name**

Specifies the name for the new connection.

**Server URL**
Specifies the URL of the server you want to connect to.

**Server version**
Specifies the version of the server or else select one from the drop-down list.

### 4.12.95.3 User account

Allows you to log in to your account.

**Use account**
Toggles whether the login or password is used when making the connection.

**Login**
Specifies the login information.

**Password**
Specifies the password of your account.

### 4.12.95.4 Proxy server

Allows you to enter the information about a proxy server.

**Use proxy**
Toggles whether proxy is used to connect to the Web Map Server.

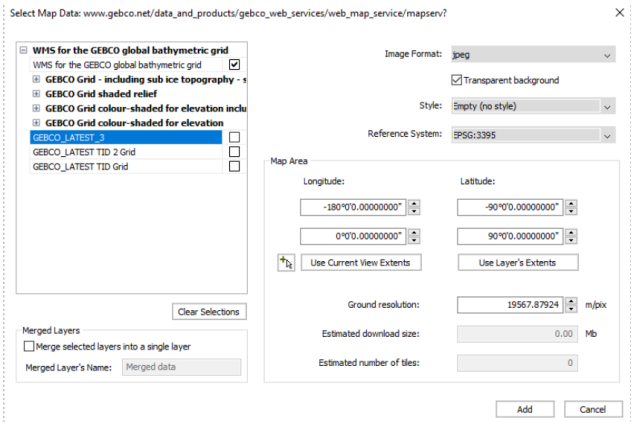
**Proxy server**
Specifies the URL of the proxy server.

**Port id**
Specifies the address of the proxy server.

**Login**
Specifies the login of the proxy server.

**Password**
Specifies the password of the proxy server.

**Remember passwords**
Check this box to keep being logged in. When is set to Off the program does not remember your passwords for greater security.
After connecting to a Web Map Server, the following dialog box is displayed:



Do the following:

1 Select one or more data layers.

2 Specify the desired ground resolution and the preferred maximum dimensions of a tile.

3 An estimate of the download size displays in the Estimated download size field.

4 Press the **Add** button.

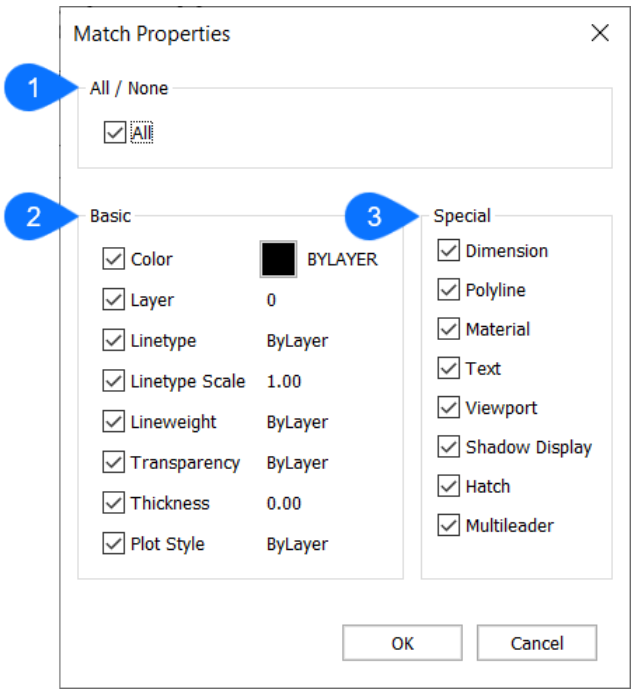
The program stores downloaded map tiles as regular image files in the folder in which the drawing is saved.

The definition of the supported Coordinate Reference Systems (CRS) is stored in the new geodatabase.xml file located in the folder pointed to by the ROAMABLEROOTPREFIX variable, such as *C:\Users\John\AppData\Roaming\Bricsys\BricsCAD\V21x64\en_US\Support*

The geodatabase.xml files lists cities and their WGS84 geo coordinates used by the **Choose Location** function of the Geographic Location dialog box.

### 4.12.96 Match Properties dialog box

The Match Properties dialog box allows you to select which properties you want to copy from one entity to one or more other entities.




1 All/None

2 Basic

3 Special

#### 4.12.96.1 All/None

When checked on, all properties are automatically selected. When checked off, all entities are deselected.

### 4.12.96.2 Basic

Selects the basic entity properties to copy. On the right side, the entity properties of the source entity are displayed.

**Color**
When checked on, the color of the source entity will be copied to the selected entities. The color of the source entity is displayed on the right.

**Layer**
When checked on, the layer of the source entity will be copied to the selected entities. The layer of the source entity is displayed on the right.

**Linetype**
When checked on, the linetype of the source entity will be copied to the selected entities. The linetype of the source entity is displayed on the right.

**Linetype Scale**
When checked on, the linetype scale of the source entity will be copied to the selected entities. The linetype scale of the source entity is displayed on the right.

**Lineweight**
When checked on, the lineweight of the source entity will be copied to the selected entities. The lineweight of the source entity is displayed on the right.

**Transparency**
When checked on, the transparency of the source entity will be copied to the selected entities. The transparency of the source entity is displayed on the right.

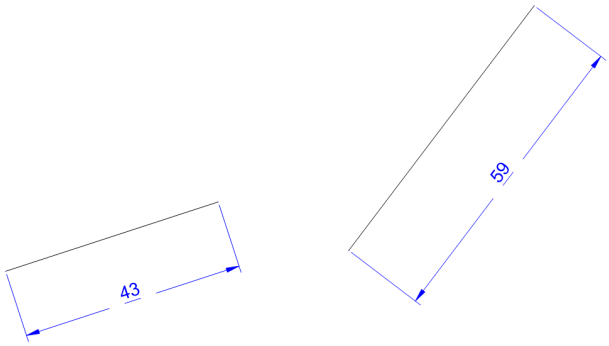
**Thickness**
When checked on, the thickness of the source entity will be copied to the selected entities. The thickness of the source entity is displayed on the right.

**Plot Style**
When checked on, the plot style of the source entity will be copied to the selected entities. The plot style of the source entity is displayed on the right.

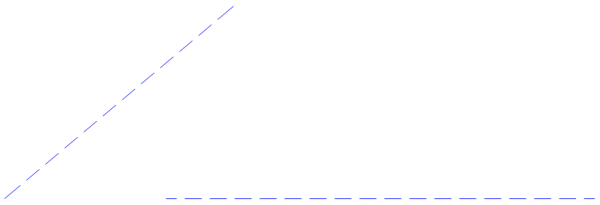
### 4.12.96.3 Special

Selects other less basic properties to copy.

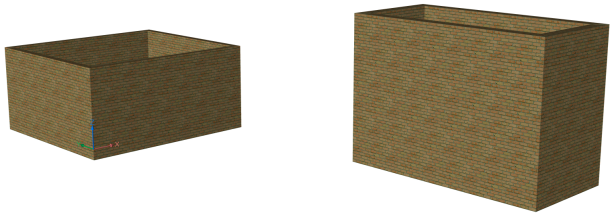
**Dimension**
When checked on, the dimension style and annotative properties of dimensions, leaders and tolerances is copied.



**Polyline**
When checked on, the width and linetype generation settings are copied. Other settings are ignored, such as taper and spline.



**Material**
When checked on, the material properties are copied. Adds or removes materials, depending on whether the source entities has a material definition or not.



**Text**
When checked on, text style and annotative property of text and mtext is copied. Other properties, like the color, are not copied.



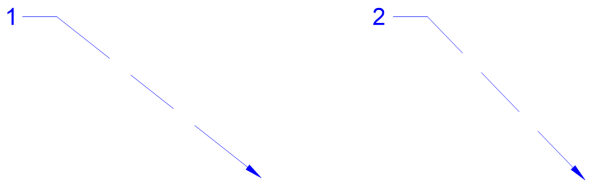

**Viewport**
When checked on, the viewport-specific properties are copied, such as on or off, snap and grid status, and scale factor. Other properties, such as clipping or frozen layer states, are not copied.

**Shadow Display**
When checked on, the shadows of the source entity are copied.

**Hatch**
When checked on, all hatch properties and annotative scaling are copied.

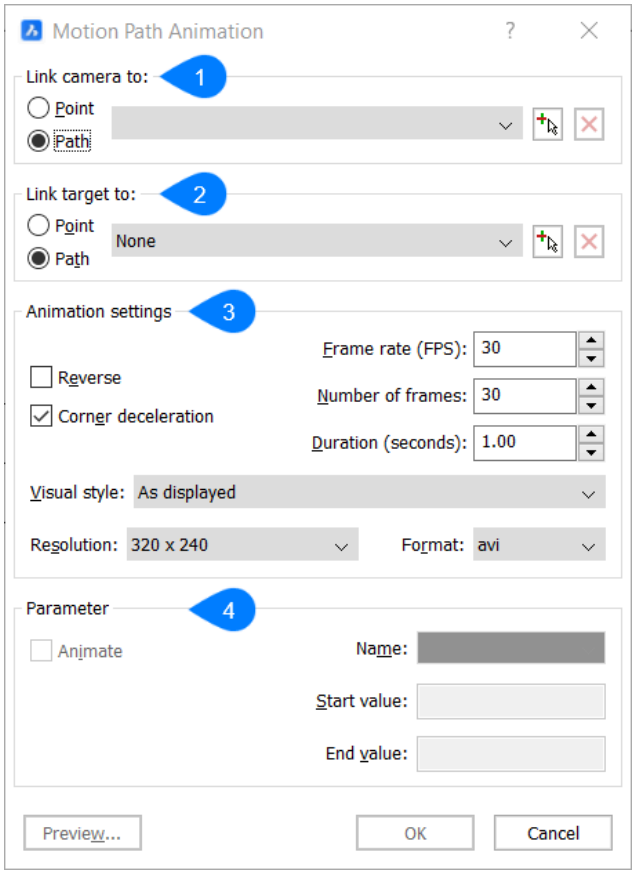


**Multileader**
When checked on, the multileader style and annotative scaling is copied.




## 4.12.97 Motion path animation

The Motion Path Animation dialog box allows you to animate a camera moving along a path or panning in a 3D model and saves it to a movie file.




1 Link camera to

2 Link target to

3 Animation settings

4 Parameter

### 4.12.97.1 Link camera to

Defines the location of the camera by selecting a point or a path in the model space.

***Прим.:*** After selecting a point or path, the option to name the entity is available.

**Point**
Links a camera to a static point in the drawing.

**Path**
Links a camera to a linear entity in the drawing.

### 4.12.97.2 Link target to

Defines the target point of the camera by selecting a point or path in the model space.

***Прим.:*** After selecting a point or path, the option to name the entity is available.

***Прим.:*** If the camera is linked to a point, the target must be linked to a path. If the camera is linked to a path, the target should be linked to either a point or a path.

**Point**
Links the target to a static point in the drawing.

**Path**
Link the target to a linear entity in the drawing.

### 4.12.97.3 Animation settings

Allows to control the output of the movie file.

**Frame rate**
Defines the number of frames per second. When the frame rate is changed, the number of frames will be adjusted according to the duration.

**Number of frames**
Specifies the total number of frames in the movie. When the number of frames is changed, the duration will be changed accordingly.

**Duration**
Specifies the duration of the movie in seconds. When the duration is changed, the number of frames will be changed accordingly.

**Reverse**
Reverses the sequence of the frames.

**Corner deceleration**
Slows down the camera at corners in the animation path.

**Visual style**
Defines the visual style for the movie. Click the button, then select a visual style in the list. All visual styles in the current drawing are available.

**Resolution**
Sets the width x height resolution of the movie. Click the button to select a resolution.
Available resolutions are: 160x120, 320x240, 640x480, 800x600, 1024x768, 1280x1024, 1600x1024, 1920x1080, 1920x1200.

**Format**
Specifies the file format of the movie. Available formats are: avi, mpg and wmv.

### 4.12.97.4 Parameter

Allows to animate the change of a parameter.

**Animate**
Check the box to create realistic movies with moving mechanisms.

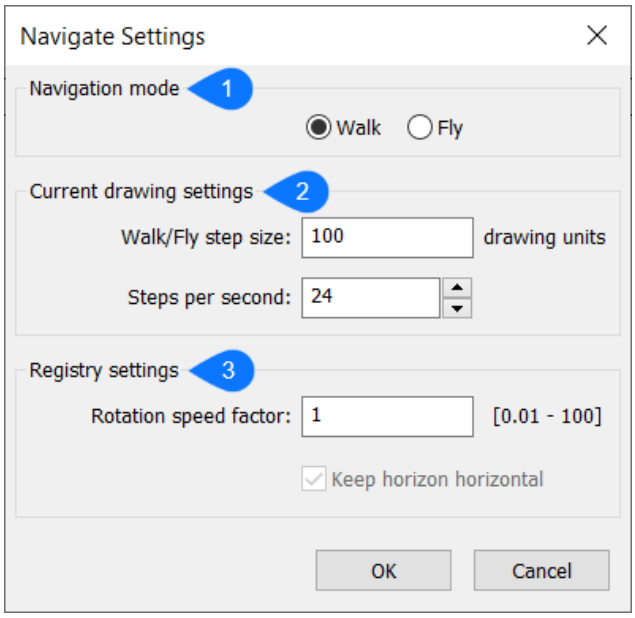
**Name**
Specify the name of the movie.

**Start value**
Specifies a start frame for the selected parameter.

**End value**
The parameter step is set to the Start-End range divided by the number of frames. On each frame, the parameter value should be incremented with the step.

## 4.12.98 Navigate Settings dialog box

The Navigate Settings Dialog box allows you to adjust the navigation settings in a drawing.




1 Navigation mode

2 Current drawing settings

3 Registry settings

### 4.12.98.1 Navigation mode

Toggles between Walk and Fly modes:

- Flight mode: permits movements in all three dimensions.
- Walk mode: restricts movement to the xy-plane.

### 4.12.98.2 Current drawing settings

Sets the settings in the current drawing, and so will be different in other drawings.

**Walk/Fly step size**
Specifies the distance the view travels with each press of the key. The distance is measured in drawing units.

**Steps per second**
Specifies how quickly the view changes when a travel key is held down.

### 4.12.98.3 Registry settings

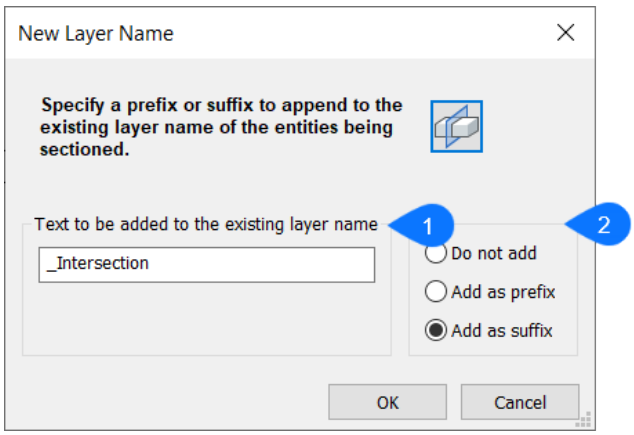
Settings which are saved with the registry, they will be the same in other drawings.

**Rotation speed factor**
Specifies how quickly the view rotates.

**Keep horizon horizontal**
When toggled on, the flight level is kept. This option is grayed-out for Walk mode.

## 4.12.99 New Layer Name dialog box

The New Name dialog box allows you to specify the text which is added to the name of the layer when new layers are created as a copy from the existing layer for section planes.




1 Text to be added to the existing layer name

2 Manner to add the text

### 4.12.99.1 Text to be added to the existing layer name

Specifies the text that will be added to the existing layer name.

### 4.12.99.2 Manner to add the text

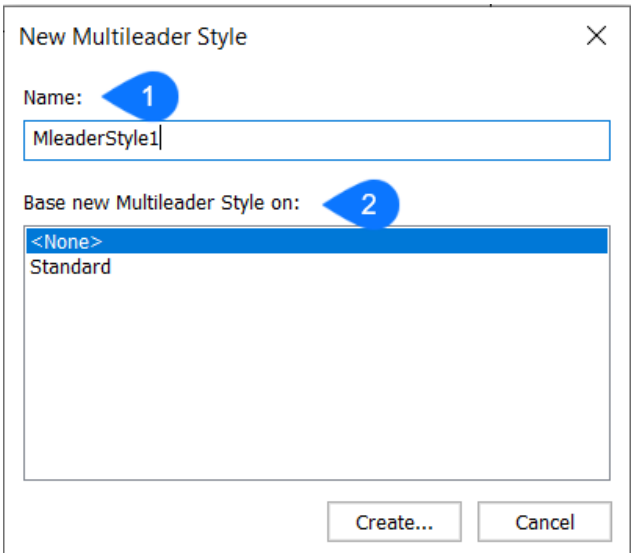
**Do not add**
No text will be added to name of the layer when a copy is created.

**Add as prefix**
The specified text will be added as a prefix to the existing name.

**Add as suffix**
The specified text will be added as a suffix to the existing name.

### 4.12.100 New Multileader Style dialog box

The New Multileader Style dialog box allows you to create a new multileader style.




1 Name

2 Base new Multileader Style on

#### 4.12.100. Name

**1**

Sets the name of the new multileader style.

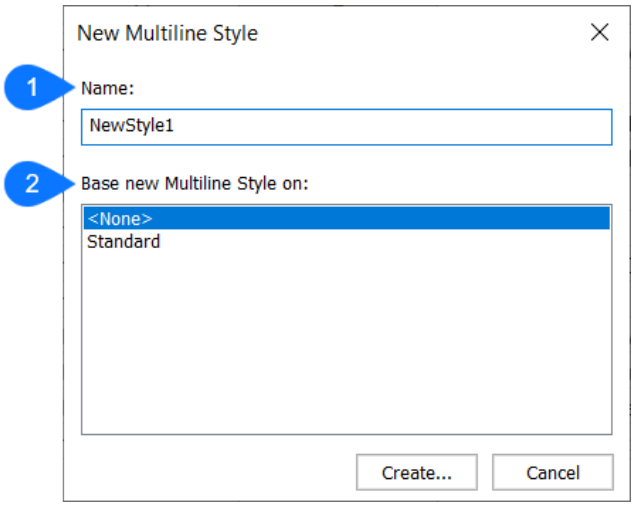
#### 4.12.100. Base new Multileader Style on

**2**

Selects an existing style to create the new one as a copy of it. To start from scratch, select <None>.

### 4.12.101 New Multiline Style dialog box

The New Multiline Style dialog box allows you to create a new multiline style definition.




1 Name

2 Base new Multiline Style on

### 4.12.101. Name

**1**

Sets the name of the new multiline style.

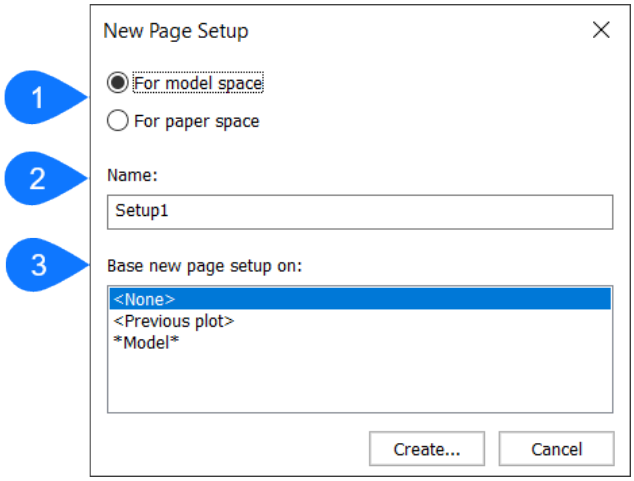
### 4.12.101. Base new Multiline Style on

**2**

Selects an existing style to create the new one as a copy of it. To start from scratch, select <None>.

### 4.12.102 New Page Setup dialog box

The New Page Setup dialog box allows you to create a new page setup for the paper space or model space.




1 Model space or Paper space

2 Name

3 Base new page setup on

### 4.12.102. Model space or Paper space

**1**

Sets the space to create a new page setup to the model space or paper space.

### 4.12.102. Name

**2**

Sets the name of the new page setup.

### 4.12.102. Base new page setup on

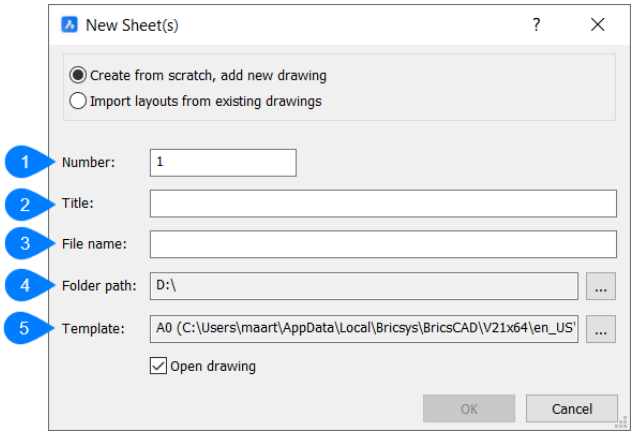
**3**

Selects an existing page setup to create a new one as a copy of the selected one. To start from scratch, select <None>. To use the previous plot, select <Previous plot>.

### 4.12.103 New Sheet(s) dialog box

The New sheets dialog box allows you to create (a) new sheet(s) within an existing sheet set.

When starting from scratch, the dialog box includes 5 areas. By selecting the Import layouts from existing drawings option, the dialog box includes 4 areas.



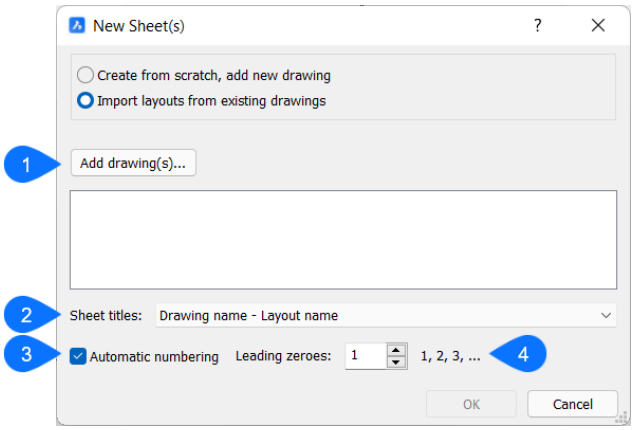

1 Number

2 Title

3 File name

4 Folder path

5 Template




1 Add drawing(s)...

2 Sheet titles

3 Automatic numbering

4 Leading zeroes

### 4.12.103. Create from scratch

**1**

Allows you to create a new sheet from scratch

- **Number**: defines the number of sheets.
- **Title**: defines the title of the new sheet(s).
- **File name**: defines the name of the file in which the new sheet(s) will be saved.
- **Folder path**: defines the location of the file.
- **Template**: choose the template of the new sheet(s).

### 4.12.103. Import layout

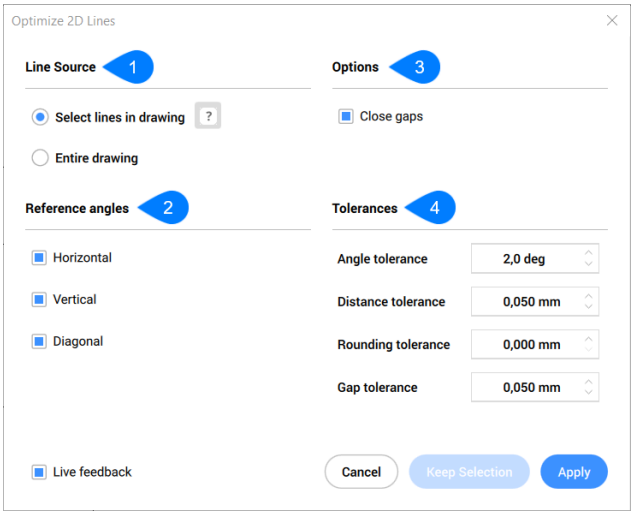
**2**

Allows you to create a new sheet with properties of an existing layout

- **Add drawing(s)...**: selects drawings to import a layout from trough the Select Drawing(s) dialog box.
- **Sheet titles**: specifies how the selected drawing should be listed by selecting one of the options in the drop-down list.
    - **Drawing name**: displays the drawing name of the selected drawings.
    - **Drawing name- Layout name**: displays the drawing name and the layout name of the selected drawings.
    - **Layout name**: displays the layout name of the selected drawings.
- **Automatic numbering**: numbers the layouts automatically.
- **Leading zeroes**: specifies how the Number value of new sheets is formatted by adding zeroes at the start of a sheet number.

### 4.12.104 Optimize 2D lines dialog box

The Optimize 2D Lines dialog box allows you to correct inaccuracies in drawings, such as small gaps between lines or near vertical, horizontal and diagonal lines. It includes 4 areas.




1 Line Source

2 Reference angles

3 Options

4 Tolerances

#### 4.12.104. Line Source

**1**

Specifies the lines in the drawing to work with.

**Select lines in drawing**

Selects line entities in drawing trough the model space. Press enter when done.

**Entire drawing**

All entities in the drawing are taken into account.

#### 4.12.104. Reference angles

**2**

Specifies the angles for which lines will be corrected.

**Horizontal**

Corrects near horizontal lines according to the angle tolerance.

**Vertical**

Corrects near vertical lines according to the angle tolerance.

**Diagonal**

Corrects near 45° lines according to the angle tolerance.

**Live feedback**

Highlights the lines which match the selected options.

### 4.12.104. Options

**3**

**Close gaps**

Closes small gaps between line entities when checked.

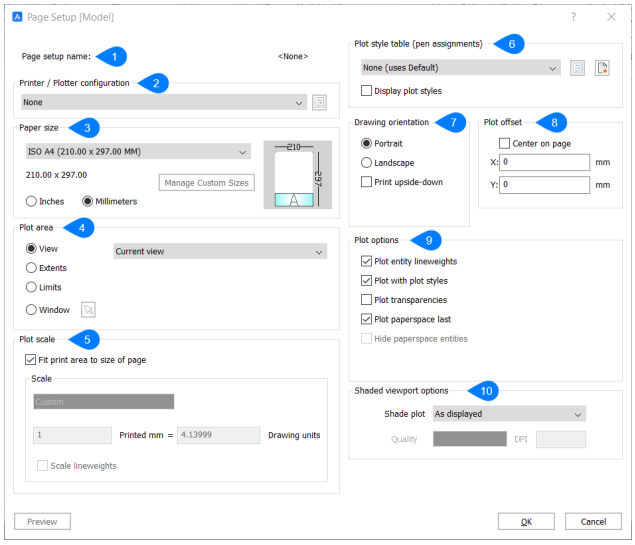
### 4.12.104. Tolerances

**4**

Specifies the tolerance value.

### 4.12.105 Page setup dialog box

The Page Setup dialog box allows you to edit the page setup of the current layout or model space in a dialog box.




1 Page setup name

2 Printer / Plotter configuration

3 Paper size

4 Plot area

5 Plot scale

6 Plot style table

7 Drawing orientation

8 Plot offset

9 Plot options

10 Shaded viewport options

### 4.12.105. Page setup name

**1**

Displays the name of the current setup.

### 4.12.105. Printer / Plotter configuration
**2**

Selects the printer or plotter.

### 4.12.105. Paper size
**3**

Specifies the size of paper by selecting a standard size from the drop-down list. These are sizes supported by the printer.

***Прим.:*** The size can be displayed in inches or millimeters by checking one of the two options.

### 4.12.105. Plot area
**4**

Specifies which area of the drawing to print.

**View**
Prints the current view or a named view. Choose a view from the drop-down list.

**Extents**
Prints the extents of the drawing, which ensures every visible entity is printed.

***Прим.:*** Entities on frozen layers are not taken into account when calculating the extents.

**Limits**
Prints the limits of the drawing, as specified by the Limits command.

**Window**
Prints a rectangular area of the drawing.

### 4.12.105. Plot scale
**5**

Scales the drawing to fit to the paper.

**Fit print area to size of paper**
Calculates the scale automatically, taking into account the print area of the drawing.

**Scale**
Specifies the scale factor to use for plotting.

**Scale lineweights**
Scales lineweights with respect to the plot scale.

### 4.12.105. Plot style table
**6**

Specifies the plot style table to use for the printed output, which assigns properties to "pens," colors, and entities.

### 4.12.105. Drawing orientation
**7**

Specifies the orientation of the drawing on rectangular paper.

**Portrait**
The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.

**Landscape**
The drawing or layout x-axis is aligned with the longest edge of the selected paper size.

**Print upside down**
Prints the drawing upside-down.

### 4.12.105. Plot offset 8

Specifies the offset distance for the print.

***Прим.:*** To align the print in the center of the page, check the Center on page option.

**Center on page**
Centers the print on the page, taking into account margins.

### 4.12.105. Plot options 9

**Plot entity lineweights**
Toggles the use of lineweights.

**Plot with plot styles**
Toggles the use of plot styles.

**Plot transparencies**
Plots entities who have been assigned the transparency property, either by entity or by layer.

**Plot paperspace last**
Specifies the printing order.

***Прим.:*** This option is only available when printing layouts.

**Hide paperspace entities**
Removes hidden lines from 3D entities in paper space when checked.

***Прим.:*** This option is disabled when printing model space.

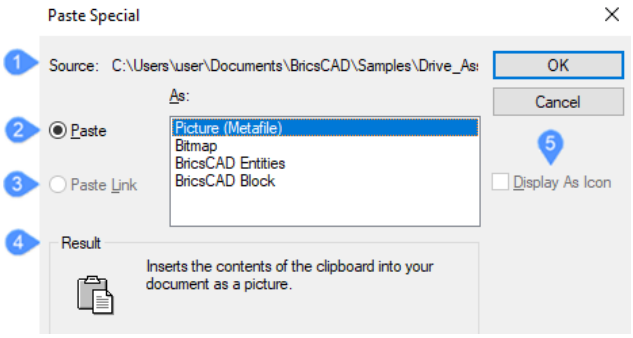
### 4.12.105. Shaded viewport options 10

Override the visual style of the current view when printing model space.

**Shade plot**
Choose a shade style to plot the drawing with.

### 4.12.106 Paste special dialog box

The **Paste Special** dialog box allows you the select the type of object to paste from the Clipboard into the drawing. This command is available on Windows only. Shortcut Keystroke: CTRL+ALT+V.




1 Source

2 Paste As

3 Paste Link

4 Result

5 Display As Icon

### 4.12.106. Source

**1**

Displays the path where the drawing/entity/block can be found.

### 4.12.106. Paste As

**2**

Chooses the format in which to paste the entities. This list varies, depending on the content of the Clipboard. When the Clipboard contains data copied from a BricsCAD drawing, then the following formats are available:

- **Picture Metafile**: entities are converted to WMF (Windows Metafile) format image and then pasted as an OLE (Object Linking and Embedding) image.
- **Bitmap**: Entities are converted to a BMP format image and then pasted as an OLE image.
- **BricsCAD Entities**: pastes as individual entities or table entries including layers, linetypes, etc.
- **BricsCAD Block**: inserts the contents of the clipboard into your document as a block.

**Paste As BricsCAD Entities**

Pastes CAD entities into the drawing as BricsCAD entities.

**Select insertion point or [Rotate/Scale/Mirror]:** - *(enter an option)*

1 **Select insertion point** – *(specifies a point in the drawing by picking a point or entering X, Y coordinates)*

2 **Rotate** *-prompts you:*

**Specify rotation angle** – *(enter an angle by which to rotate the entities)*

3 ***Scale*** *-prompts you:*

**Specify scale factor for XYZ axes** – *(enter a scale factor)*

4 ***Mirror*** – *prompts you:*

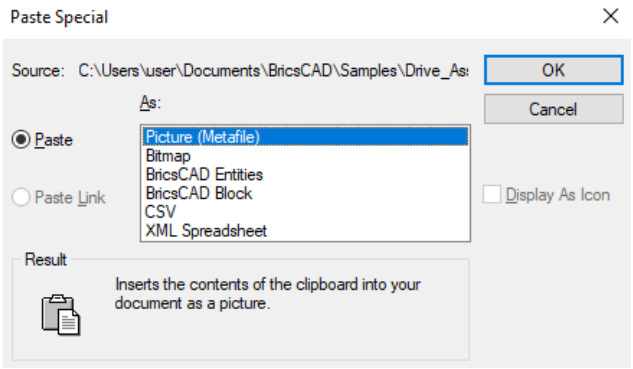
***Select mirroring control point*** – *(Pick a second point for the mirror line)*

The insertion point is the first point for the mirror line. When you enter 0,0 for the insertion point, the entities are pasted to their original coordinates.

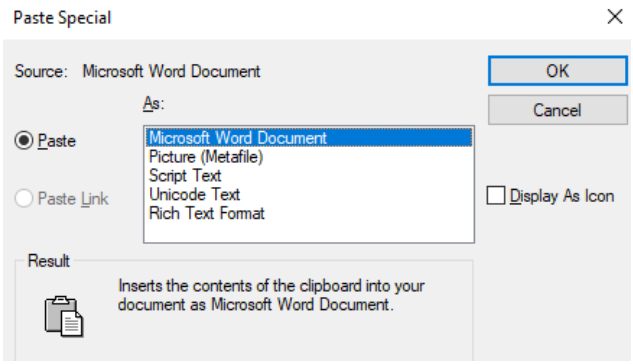
**Paste As BricsCAD Block**

Opens the **Insert Block** dialog box.

Here is the same dialog box, but it has some new options as the entities are copied from an Excel spreadsheet.




If you copy a text string from outside of BricsCAD, there are some new options as shown in the picture below:




### 4.12.106. Paste Link

### 3

This option is not available for CAD entities. Pastes the entities in the drawing as an OLE object and then creates a link to the originating application.

### 4.12.106. Result

### 4

Displays a short description of the selection option you want to use.

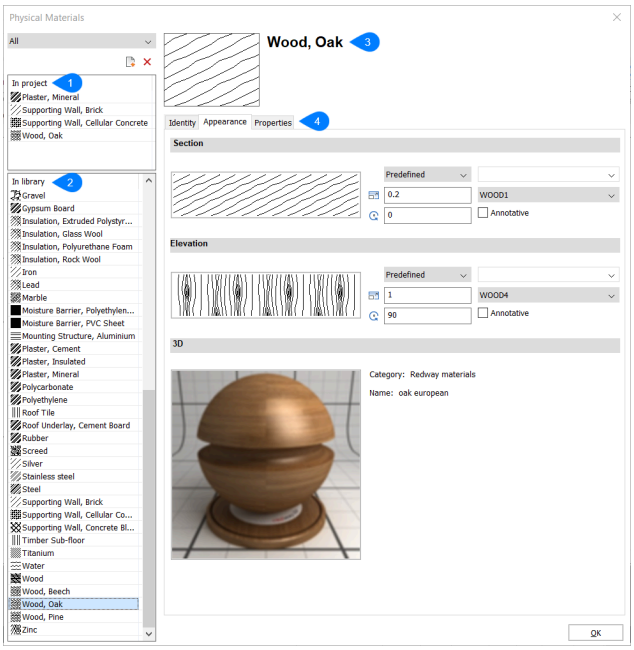
### 4.12.106. Display As Icon

**5**

This option is not available for CAD entities. Displays the pasted entities as an icon identifying their source application.

### 4.12.107 Physical materials dialog box

The Physical Materials dialog box opens the Physical Materials library in the current project and the referenced library.




1 In project

2 In library

3 Material name

4 Material specification tabs

### 4.12.107. In project

**1**

Displays all materials which are currently loaded in the project.

### 4.12.107. In library

**2**

Displays all materials which are loaded in the library.

### 4.12.107. Material name

**3**

Displays the name of the selected material.

### 4.12.107. Material specification tabs

4

**Material specification tabs**

Lists the broader specifications of the selected material into three tabs.

**Identity**

- **Name**: specifies the name of the material.
- **Class**: specifies the material class the material belongs to.
- **Description**: displays a short description of the material.

**Appearance**

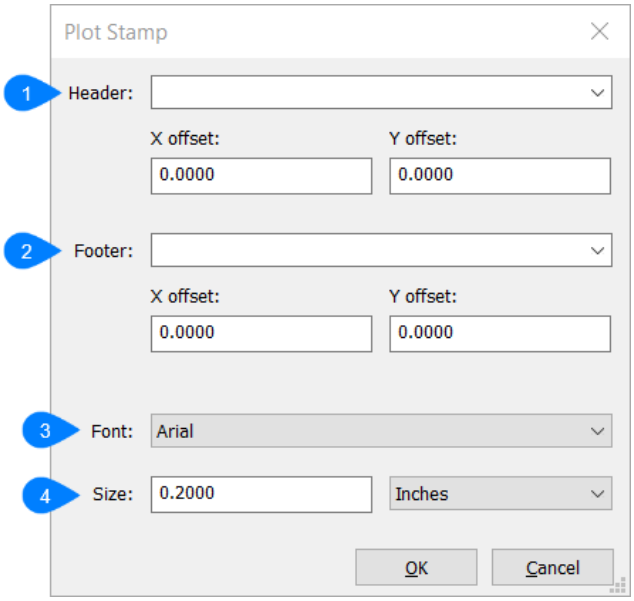
- **Section**: specifies the hatch pattern used to display the material in a section view.
- **Elevation**: specifies the hatch pattern used to display the material in an elevation view.
- **3D**: shows the three dimensional texture of the material that will be used in all 3D views.

**Properties**

Specifies additional properties of the material such as the density.

### 4.12.108 Plot stamp dialog box

The Plot Stamp dialog box allows you to add information about drawings and prints to the edge of the plotted drawing.




1 Header

2 Footer

3 Font

4 Size

### 4.12.108. Header

**1**

Appears at the top of the plot. A meta-phrase for the header can be chosen by clicking the drop-down list or by specifying the X and Y offsets.

### 4.12.108. Footer

**2**

Appears at the bottom of the plot. A meta-phrase for the footer can be chosen by clicking the drop-down list or by specifying the X and Y offsets.

### 4.12.108. Font

**3**

Specifies the font used for the plotstamp.

### 4.12.108. Size

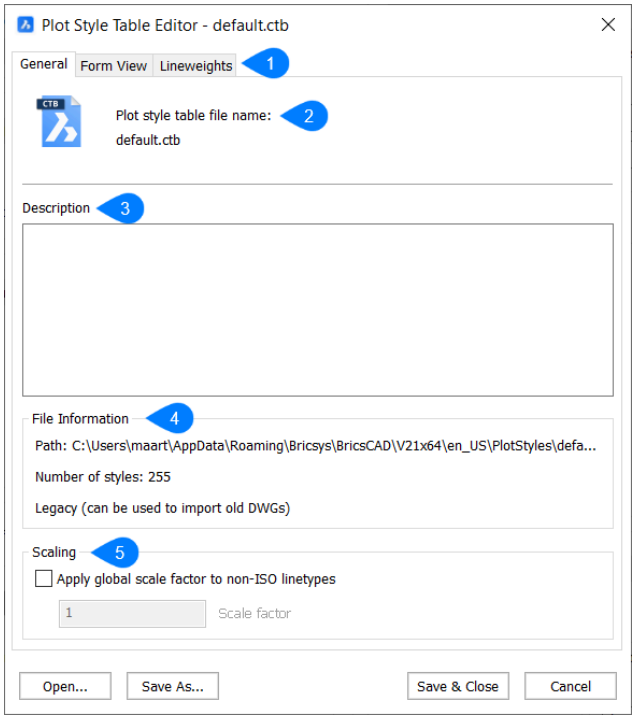
**4**

Specifies the height of the text.

- Inches: specifies the height of the text in inches.
- Millimeters: specifies the height of the text in millimeters.

### 4.12.109 Plot style table editor dialog box

The Plot Style Table Editor displays the plot styles contained in the specified plot style table.




1 Editor tabs

2 Plot style table file name

3 Description

4 File Information

5 Scaling

### 4.12.109. Editor tabs

**1**

The Plot Style Table Editor includes the following tabs.

**General tab**

The General tab lists general information about the plot style table.

**Form View tab**

The Form View tab lists the plot styles contained in the plot style table and their settings. Plot styles are style overrides for your drawings that occur during plotting.

**Lineweights tab**

The Lineweights tab lists the available lineweights.

### 4.12.109. Plot style table file name

**2**

Displays the name of the plot style table file you are editing.

### 4.12.109. Description

**3**

Displays a description you want to include about the plot style table.

### 4.12.109. File Information

**4**

Displays information about path, number of plot styles and the version number of the Plot Style Table Editor.

### 4.12.109. Scaling

**5**

**Apply global scale factor to non-ISO linetypes**

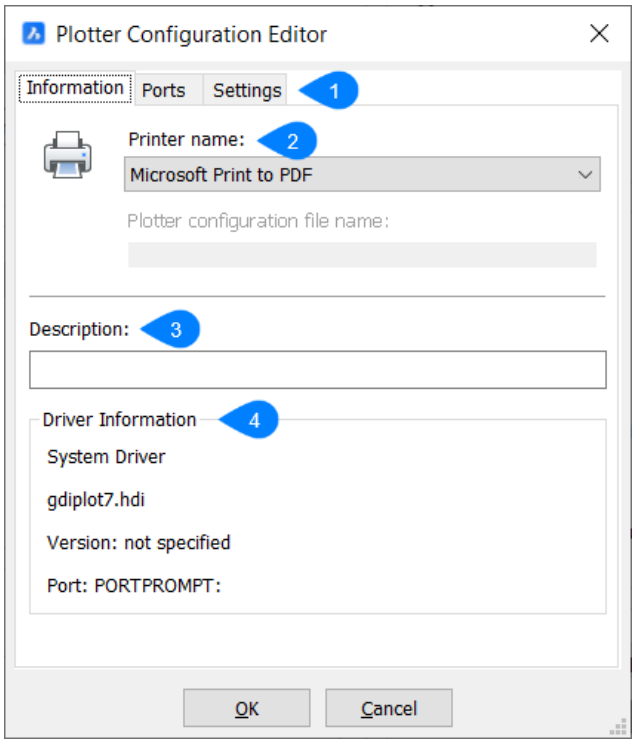
Specifies whether or not you want to scale all non-ISO linetypes in the plot styles of entities controlled by the plot style table.

**Scale factor**

Specifies the factor by which to scale non-ISO linetypes and fill patterns.

### 4.12.110 Plotter Configuration Editor dialog box

The Plotter Configuration Editor dialog box allows you to create and edit customized parameter PC3 files for printers and other output devices.




1 Editor tabs

2 Printer name

3 Description

4 Driver Information

### 4.12.110. Editor tabs

**1**

**Information**

Gives basic information about the plotter.

**Ports**

Gives options on how the file will be printed.

**Settings**

Provides additional settings to alter. The settings available depend on the capabilities of the printer selected.

### 4.12.110. Printer name

**2**

Lists the names of printers supported by the operating system.

### 4.12.110. Description

**3**

Describes the plotter configuration. It is displayed by tooltips.

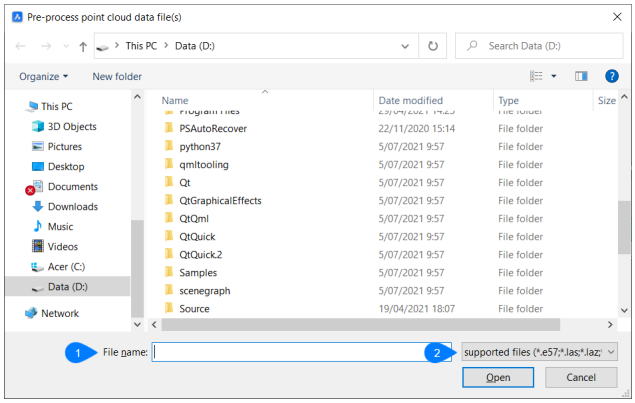
### 4.12.110. Driver Information

**4**

Displays basic information about the driver.

### 4.12.111 Pre-process point cloud data file(s) dialog box

The Pre-process Point Cloud Data File's dialog box allows you to attach point cloud files (that were previously converted to BricsCAD's BPT format) to the current drawing.




1 File Name

2 Supported files

### 4.12.111. File Name

**1**

Specifies the name of the point cloud data file you want to attach.

***Прим.:*** A manual search can be done in the window above.

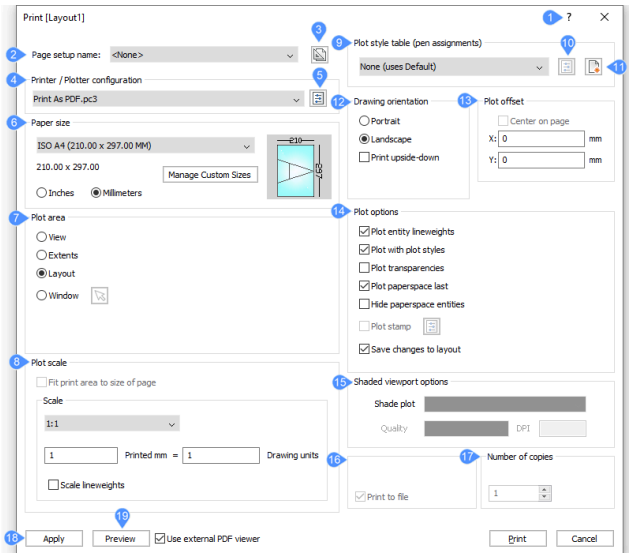
### 4.12.111. Supported files

**2**

Specifies the file types you will be able to choose from.

### 4.12.112 Print dialog box

The **Print** dialog box allows you to print and preview drawings to plotters and files.




1 Command reference

2 Page setup name

3 Create page setup

4 Printer/Plotter configuration

5 Edit plotter configuration

6 Paper size

7 Plot area

8 Plot scale

9 Plot style table

10 Edit Plot Style

11 Create New Plot Style

12 Drawing orientation

13 Plot offset

14 Plot options

15 Shaded viewport options

16 Print to file

17 Number of copies

18 Apply

19 Preview

### 4.12.112. Command reference

**1**

Opens the Bricsys Help article about the PRINT command.

### 4.12.112. Page setup name 2

You can select plot settings used previously from the drop-down list:

- **<None>**: Uses options saved in the current layout or model space.
- **<Previous Plot>**: Uses options that were saved from the last time when this dialog box was used.
- Names of other page setups stored in the drawing.

### 4.12.112. Create page setup 3

Opens the **Create Page Setup** dialog box that allows you to create new page setups. See the PAGESETUP command.

### 4.12.112. Printer/Plotter configuration 4

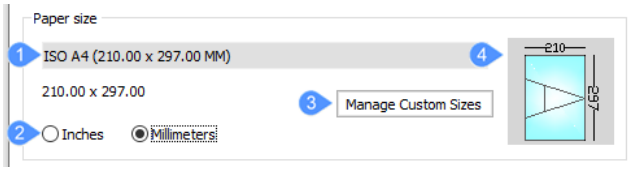
Allows you to select from the drop-down list the printer or plotter.

The program works with any output device installed on the system, including networked printers, print to file, faxes, and PostScript devices -- as well as predefined printer parameters stored in PC3 files.

### 4.12.112. Edit plotter configuration 5

Opens the **Plotter Configuration Editor** dialog box for customizing the printer's parameters and creating PC3files. See the PLOTTERMANAGER command.

### 4.12.112. Paper size 6




1 Standard Paper Sizes list

2 Units

3 Manage Custom Sizes

4 Preview

**Standard Paper Sizes list**

Specifies the size of paper. You can select standard sizes listed by the drop-down list; these are sizes supported by the printer.

While printers may appear to support many different sizes, you should select only the size of paper that is in the printer.

**Units**

- **Inches**: Use imperial units for measurements involving printing.
- **Millimeters**: Use metric units for measurements involving printing.

### Manage Custom Sizes

***Прим.:*** This option is only available for the PDF output.

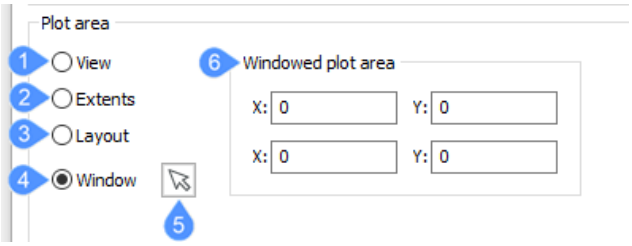
Manages custom paper sizes when Print as PDF.pc3 is selected in the **Printer/Plotter Configuration** list. The **Custom Paper Sizes** dialog box opens where you can create your own paper size.

### Preview

This section indicates the size, position and orientation of the current plot area on the selected paper size.

## 4.12.112. Plot area

**7**




1 View

2 Extents

3 Layout

4 Window

5 Select area to be printed

6 Windowed plot area

### View

Prints the current view or a named view. Choose a view from the drop-down list. Use the VIEW command to create named views.

### Extents

Prints the extents of the drawing, which ensures every visible entity is printed. Entities on frozen layers are not considered when calculating the extents.

### Layout

Prints the current layout.

### Window

Prints a rectangular area of the drawing.

- You define the rectangular area by entering the x, y coordinates or by clicking the **Select Area to be Printed** button.
- The x, y coordinates resulting from the selection or from entering in the Command line are added to the **Windows Plot Area** boxes. You can edit the values in a convenient way.

### 4.12.112. Plot scale 8

Scales the drawing to fit it to the paper:

- **Fit print area to the size of paper**: Scale is calculated automatically by the program, considering the print area of the drawing. When this option is turned on, you cannot specify the scale factor:
  - **Printable area** is the size of the paper, minus the margins.
  - **Margins** are the strips along the four edges that the printer uses for handling the paper.
- **Scale**: Allows you to specify the scale factor to use for plotting; choose a scale factor from the drop-down list or choose the **Custom** option that will allow you to enter your own scale factors in **Printed inches/mm** and **Drawing units** fields. The scale factor shown can be edited with the SCALELISTEDIT command.
  - For scale factors like 1:5, the drawing is printed smaller.
  - For scale factors like 5:1, the drawing is printed larger.
- **Scale lineweights**: When on, lineweights are scaled with respect to the plot scale.

### 4.12.112. Plot style table 9

Specifies the plot style table to use for the printed output, which assigns properties to pens, colors, and entities.

- When the drawing does not use plot styles, then only CTB (color-based table) files are listed.
- When the drawing uses plot styles, then only STB (style-based table) files are listed.

When you switch from **None** to a named plot style, BricsCAD prompts you:

Assign plot style table to all layouts?

- **Yes**: Assigns the CTB or STB plot style file to all layouts.
- **No**: Assigns the plot style file to the current layout only.

### 4.12.112. Edit Plot Style 10

Opens the **Plot Style Table Editor** dialog box. See the PLOTSTYLE command. This button is available only when the drawing uses STB plot styles.

### 4.12.112. Create New Plot Style 11

Opens the **Add Plot Style Table** dialog box to create new plot styles. See the STYLESMANAGER command.

- When the drawing does not use named plot styles, then the wizard creates new color-dependent tables (CTB files).
- If the drawing uses named plot styles, then the wizard creates new named plot style tables (STB files).

### 4.12.112. Drawing orientation
**12**

Specifies the orientation of the drawing on rectangular paper:

- **Portrait**: The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.
- **Landscape**: The drawing or layout x-axis is aligned with the longest edge of the selected paper size.
- **Print upside down**: Prints the drawing upside-down. This is useful when paper with a drawing border is loaded backwards in the printer.

### 4.12.112. Plot offset
**13**

Specifies the offset distance for the print.

- **Center on Page**: Centers the print on the page, considering margins. Together with Fit Print Area to Size of Page, this option is excellent for prints where scale is unimportant.
- **X** and **Y**: Enter positive or negative distances to move the drawing in the x and/or y distances:
  - **Positive values**: Move the drawing up and to the right.
  - **Negative values**: Move the drawing down and to the left.

The lower left corner of the drawing is moved by the specified distance. This is useful when the paper has a title block area that might interfere with the drawing.

### 4.12.112. Plot options
**14**

- **Plot entity lineweights**: Toggles the use of lineweights:
  - On: Mimics the lineweights assigned to the drawing in the print.
  - Off: Ignores lineweights.
- **Plot with Plot Styles**: Toggles the use of plot styles:
  - On: Uses plot styles to determine the look of the printed drawing and overrides lineweight settings.
  - Off: Does not use plot styles.
- **Plot Transparencies**: Plots entities who have been assigned the Transparency property, either by-entity or by layer.
- **Plot Paperspace Last**: Specifies the printing order:
  - On: Prints model space entities, followed by paper space entities.
  - Off: Prints paper space entities first, followed by entities in model space.
- **Hide Paperspace Entities**: When on, removes hidden lines from 3D entities in paper space. This option is disabled when printing model space.
- **Plot Stamp**: Toggles use of the plot stamp; click the Edit Plot Stamp button to change plot stamp data. See the PLOTSTAMP command. This option is unavailable when the Printer/Plotter configuration is set to *Print As PDF.pc3*.
  - On: Applies plot stamp data to the print.

- Off: Does not apply plot stamps.

- **Save Changes to Layout**: Determines if the options you changed in this dialog box are saved; the next time you use this dialog box, you can select "Layout" from the Use Plot Settings From drop-down list.
  - On: Saves changes made to this dialog box with the layout.
  - Off: Does not save changes.

### 4.12.112. Shaded viewport options 15

Override the visual style of the current view when printing model space:

Choose from the drop-down list a shade style to plot the drawing with.

This option is disabled when printing a paper space layout. The **Shade Plot** mode of a paper space viewport is defined in the **Shade Plot** property of the viewport. The quality of the **Rendered** option is defined through the current render preset; see the RENDERPRESETS command.

### 4.12.112. Print to file 16

When checked, redirects the print output to a *.plt file, which can be processed by certain types of software.

### 4.12.112. Number of copies 17

Specifies the number of copies to print. Enter a number or click the buttons to change the value.

### 4.12.112. Apply 18

Applies the changes you made to this dialog box. The changes are remembered the next time you use this command. This is like using the PAGESETUP command.

### 4.12.112. Preview 19

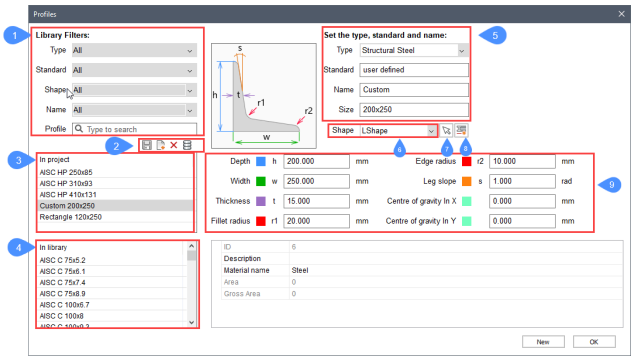
Displays a preview of the print. In the preview window, click the **Print Settings** button to return to this dialog box. See the PREVIEW command.

***Прим.:*** On Mac or Linux, printing will always print to PDF. So, a PDF document will be generated that then still needs to be sent to the printer for physical printing.

On MAC and Linux there is a relevant PAGESETUP command.

### 4.12.113 Profiles dialog box

Opens the profiles library in the current project and the referenced library.




1 Library Filters

2 Tools

3 In Project

4 In Library

5 Custom fields

6 Preselect shape

7 Pick profile in model

8 Set profile offset

9 Properties of the profile

### 4.12.113. Library Filters

**1**

Allows to filter the profiles list by type, standard, shape or name.

- **Type**: The **Type** options are: **All**, **Generic**, **HVAC**, **Piping** or **Structural Steel**.
- **Standard**: The Localized standard options are: **All**, **AISC** (American Institute of Steel Construction), **BS** (British Steel), **EURO** (European standard steel profiles) and **User Defined**.
- **Shape**: The **Shape** options are: **All**, **Circle Hollow**, **I Shape**, **L Shape**, **Rectangle**, **Rectangle Hollow**, **T Shape** and **U Shape**. Depending on the selected standard, some shape options are not available.
- **Name**: Filters the profiles by a selected character string.
- **Profile**: Select a character string. Only profiles that contain the selected string are listed.

### 4.12.113. Tools

**2**

- **Save Profile**: Save the currently edited profile.
- **New Profile**: Create a new profile.
- **Delete Profile**: Delete the selected profile.
- **Project and Library Information**: Opens the BIM **Project Info** dialog box.

#### 4.12.113. In Project

**3**

Lists the available profiles used in the current project with respect with above filters.

#### 4.12.113. In Library

**4**

Lists the available profiles with respect with above filters.

#### 4.12.113. Custom fields

**5**

Lists the characteristics of the chosen profile or in case of creating a new profile, sets the type, standard and name for it.

#### 4.12.113. Preselect shape

**6**

Select a shape from the drop-down list.

#### 4.12.113. Pick profile in model

**7**

Select a closed 2D entity or the boundary entities of an enclosed area.

#### 4.12.113. Set profile offset

**8**

Sets the profile offset.

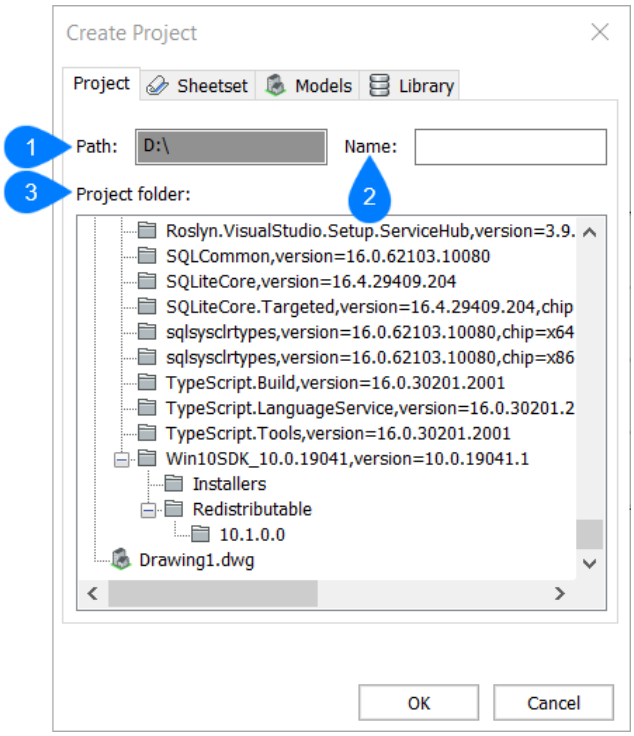
#### 4.12.113. Properties of the profile

**9**

Shows the properties of the selected profile or sets the properties of a new profile.

### 4.12.114 Project setup dialog box

The Project Setup dialog box allows you to create a new project and choose the file location.




1 Path

2 Name

3 Project folder

### 4.12.114. Path

**1**

Specifies the location of the project by selecting the parental folder.

### 4.12.114. Name

**2**

Creates a new project file by entering a new name.

***Прим.:*** By default, the name is that of the parental folder.

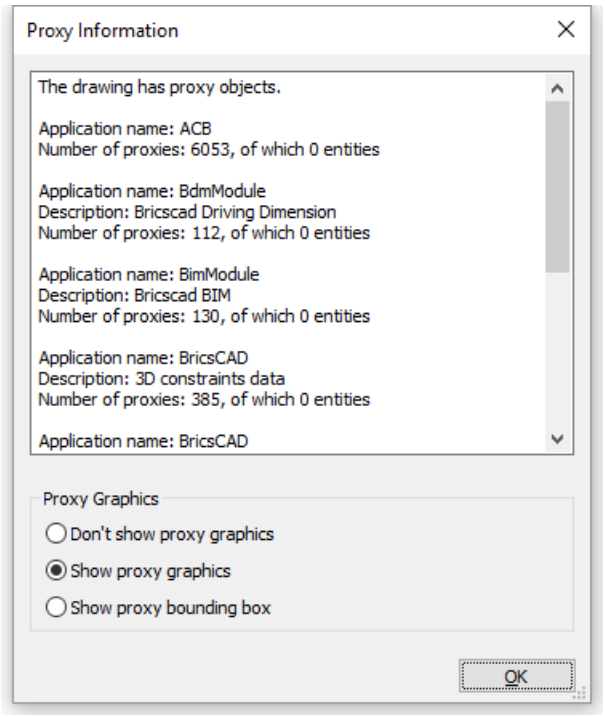
### 4.12.114. Project folder

**3**

Specifies the folder in which the project will be saved by selecting the destination folder.

### 4.12.115 Proxy information dialog box

The **Proxy Information** dialog box displays information about proxy entities and toggles their display in the current drawing.




### 4.12.115. Proxy Graphics

**1**

Toggles the display of proxy graphics in the drawing:

- **Don't show proxy graphics**: turns off the display of proxy graphics.
- **Show proxy graphics**: turns on the display of proxy graphics
- **Show proxy bounding box**: displays a rectangle or cube instead of the proxy.

### 4.12.116 Publish dialog box

The Publish dialog box allows you to send one or more drawings, layouts and sheets to the printer or export them in PDF format. This is useful for batch printing of books of drawings.

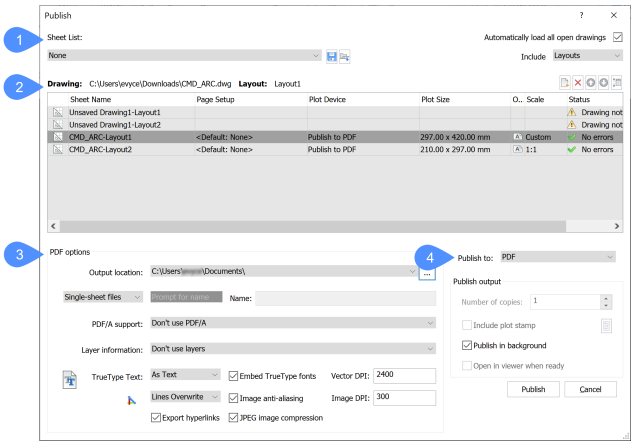
To print a batch of drawings when BricsCAD starts, write in the Terminal the following command

bricscad.exe /pl <"the path of your *.dwg file"> <"the path of your *.dsd file">

For example:

bricscad.exe /pl "C:\Users\user\Desktop\doc.dwg" "C:\Users\user\Desktop\doc.dsd"

Also, in the Page Setups section of the Drawing Explorer dialog box set the printer you use in the Device name.




1 Sheet List

2 Drawing

3 PDF options

4 Publish to

## 4.12.116. Sheet List

### 1

Displays a list of sheets. If you have a *.dsd file (drawing sheets for publishing), you can select it and the Load Sheet List dialog box will appear.

#### Save Sheet List

Saves the current sheet list to a *.dsd (Drawing Set Description) file. Opens the Save Sheet List dialog box.

#### Load Sheet List

Loads a saved sheet list. Opens the Load Sheet List dialog box. If the current list is not saved yet, you are prompted to save it.

If you are trying to load a file with an existing name, after you press the Open button in the Load Sheet List dialog box you are prompted to replace or append the sheets to the list.

#### Automatically load all open drawings

Determines how open drawings are handled:

- **On**(checked): adds all model and/or paper space layouts to the list of sheets to be plotted
- **Off**(unchecked): adds the model and/or paper space layouts of the current drawing only

This option also sets the value of the PUBLISHALLSHEETS system variable.

#### Include

Determines which sheets must be included:

- **Model**: includes only model tabs of open drawings.
- **Layouts**: includes only layout tabs of open drawings.
- **Model & Layouts**: includes both model and layout tabs of open drawings.

### 4.12.116. Drawing

### 2

**Layout**

**Page Setup**
Specifies the Page Setup name of each sheet. Click the Page Setup name, then click the down arrow to select a different page setup or choose **Import…** to import page setups of another drawing. This will open the Import Pagesetups dialog box.

**Plot Device**
Displays the name of the plot device, as defined in the Page Setup dialog box of the sheet.

### 4.12.116. PDF options

### 3

**Output location**
Displays the path where the PDF will be saved. Opens the Choose a folder dialog box to change the path.

**File**
Select an option:

- **Single – sheet files**: each layout is published to a separate *.pdf. The file names are: <drawing>_<layout>.pdf
- **One Multi – sheet file**: layouts are published to a single file. You can choose to specify a name or to be prompted for a name when the publishing procedure starts.

**Name options**
Select an option:

- **Prompt name**: after you press the Publish button you are prompted to insert the name you want for your *.pdf in the Specify PDF file name dialog box.
- **Specify name**: the field for Name becomes available. Insert here the name you want for your *.pdf.
- **Generate name**: the *.pdf will have the same name as the drawing.

**Name**
Allows you to type the name you want for your *.pdf. This field is available only for the **Specify name** option.

**PDF/A Support**
Allows you to select from the drop-down list one of the options.

**Layer information**

- **Don't use layers**: no layers are exported.
- **Use all layers with visible entities**: only layers which are current ON and Thawed are exported.
- **Use layers with entities, including OFF and FROZEN layers**: all layers containing entities are exported.

**TrueType Text**
Select an option:

- **As text**: ensures that the text in the PDF looks exactly as in the drawing.
- **As geometry**: converting to geometry increases the file size and pixelation might occur when viewed at a high zoom level.

**Embed TrueType fonts**
If the font is not embedded, a substitute font might be used by the PDF viewer.

***Прим.:*** Licensed fonts cannot be embedded.

**Vector DPI**
Controls the resolution of vector graphics and gradients.

**Merge control**
Determine whether overlapping lines are merged or lie on top of each other.
Select an option:

- **Lines Overwrite**: the lines on top overwrite the lines below.
- **Lines Merge**: the colors of crossing lines are blended into a new color.

**Image anti-aliasing**
If the resolution of an image is too low to match the Image DPI setting, then the raster image is anti-aliased (generate pixels artificially) to make the raster image look more smoothly. Without that setting an image with a too low resolution is embedded "as-is".

**Image DPI**
Defines the resolution of the resulting image in the PDF for raster images. This setting allows you to lower the resolution of the original raster image to avoid including a gigabyte raster image in the PDF. For printing the resolution should match the output resolution of the printer. For on-screen viewing high resolutions (2400 dpi +) are required.

**Export hyperlinks**
If checked, it allows you to export hyperlinks in the *.pdf.

**JPEG image compression**
This setting applies to all included raster images in the PDF, both raster images and rendered views. It applies a JPEG compression to these images to reduce the PDF file size. When switched off, the images are embedded as raw bitmaps. The JPEG compression is lossy compared to the raw bitmap introducing small imperfections.

### 4.12.116. Publish to

### 4

***Прим.:*** When **Publish to** is set to **Plotter named in page setup**, the PDF options from this dialog box disappear.

**Number of copies**
Sets the number of copies for each sheet to be printed.

**Include plot stamp**
If this option is checked, it adds a plot stamp.

**Plot Stamp**
Opens the Plot Stamp dialog box.

**Publish in background**
Determines if the sheet is published in the background:

- **Yes**: sheets are published in the background, which takes longer to produce the prints, but allows you to continue to work in the program.

- **No**: sheets are published in the foreground, which is faster, but prevents you from working with the program until the print job is completed.

The BACKGROUNDPLOT system variable controls whether documents are published in the foreground or background.

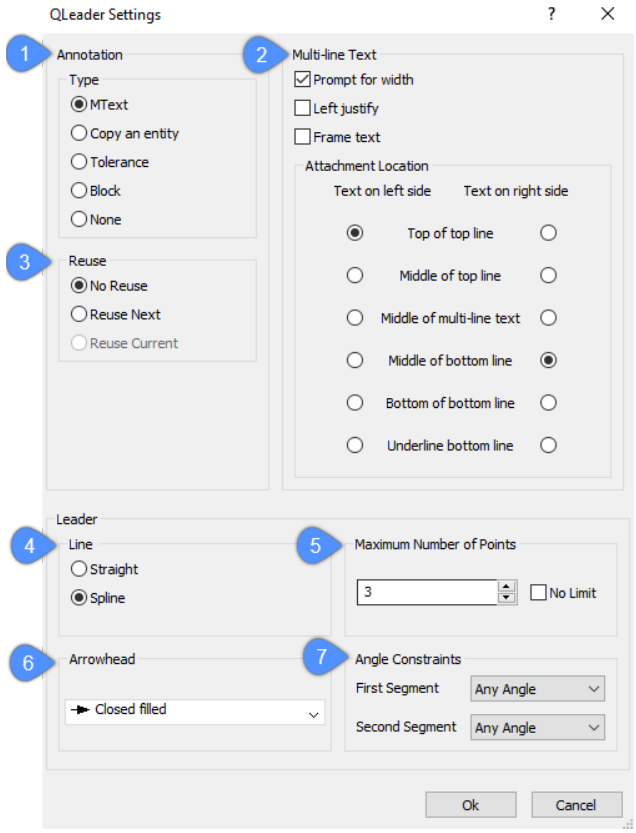
**Open in viewer when ready**

Determines if the sheet set will be opened in the viewer when it is ready.

***Прим.:*** This option is not available when **Publish to** is set to **Plotter named in page setup**, only when **Publish to** is set to **PDF**.

### 4.12.117 QLeader Settings dialog box

The **Qleader Settings** dialog box allows you to specify the properties of leaders.




1 Annotation

2 Multi-line Text

3 Reuse

4 Leader Line

5 Maximum Number of Points

6 Arrowhead

7 Angle Constraints

### 4.12.117. Annotation

**1**

Specifies the type of annotation:

- **Mtext**: Text is placed as the leader line's annotation; a leader can have one or more lines of text.

  ***Прим.:*** Although the mtext editor does not appear when entering text (as with the MLeader command), the mtext editor does appear when editing the annotation (edit the text by double-clicking it).
- **Copy of an entity**: Uses any other entity (than the leader itself) in the drawing as the annotation.
- **Tolerance**: Uses tolerance notation for the annotation; displays the **Geometric Tolerance** dialog box. See the TOLERANCE command.
- **Block**: Uses a block from the drawing or an external block as the annotation. The prompt displayed is identical to that of the -INSERT command.
- **None**: No annotation is used.

### 4.12.117. Multi-line Text

**2**

***Прим.:*** This option is only available when the annotation type is MText.

Specifies some multi-line text options:

- **Prompt for width**: Toggles the use of the 'Specify text width' prompt:
  - On: Displays the 'Specify text width' prompt and so wraps the text to the specified width.
  - Off: Disables the prompt.
- **Left justify**: Determines the justification of the text:
  - On: Forces the text to be left-justified always.
  - Off: The text is left or right-justified, depending on the orientation of the landing line.
- **Frame text**: Toggles the use of a frame:
  - On: Places a rectangle around the text.
  - Off: No frame.
- **Attachment location**: Determines the relative location of where the mtext is attached to the landing line.

### 4.12.117. Reuse

**3**

Specifies how to reuse the annotation:

- **No reuse**: Does not reuse the previous annotation text; you will be prompted to enter new text each time you use this command.
- **Reuse next**: You will be prompted to enter new text, which is then reused for subsequent leader(s).
- **Reuse current**: The current text is reused for each new leader, and you will not be prompted to enter new text.

### 4.12.117. Leader Line 4

Specifies the style of leader line:

- **Straight**: Draws the leader lines as straight segments.
- **Spline**: Draws the leader line as a smooth continuous spline.

### 4.12.117. Maximum Number of Points 5

Specifies the maximum number of vertices for the leader, before it stops drawing them automatically:

- Enter a number; the minimum number is 2.
- **No limit**: Keeps drawing leader segments until you press Enter at the 'To point' prompt.

### 4.12.117. Arrowhead 6

Specifies the type of arrow at the start of the leader; choose from the drop-down list:

- Choose an arrowhead type.
- **None**: No arrow is drawn.
- **Select arrow**: Choose a block from those in the current drawing; displays the **Select Custom Arrow Block** dialog box.

To create a custom arrow, draw it to your liking, and then use the BLOCK command to turn it into a block.

### 4.12.117. Angle Constraints 7

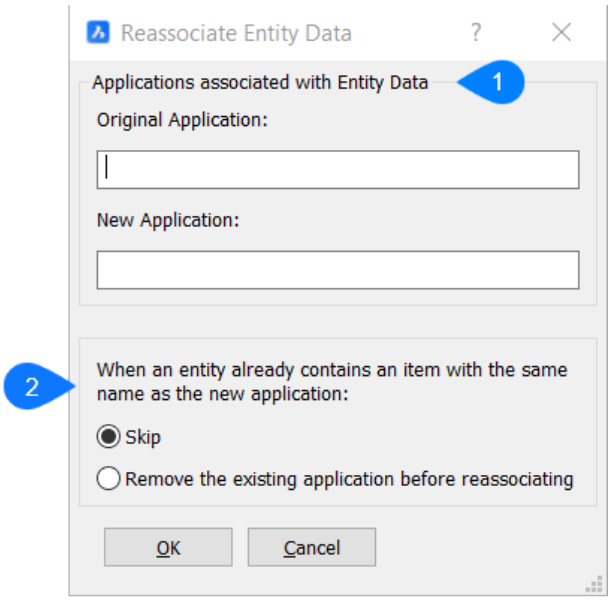
Set an angle to constrain the first and second leader segments. The angles are increments; for instance, choose 15 allows you to draw the leader at 0, 15, 30, 45, and so on degrees.

Click the drop-down list, then choose an option:

- **Any Angle**: No constraint; hold down the Shift key to draw the leader at 90 degrees.
- **Horizontal**: Forces the leader line to be drawn at 90 degrees only.
- **90°**: Draws leader lines at 90-degree increments.
- **45°**: Draws leader lines at 45-degree increments.
- **30°**: Draws leader lines at 30-degree increments.
- **15°**: Draws leader lines at 15-degree increments.

### 4.12.118 Reassociate entity data dialog box

The Reassociate Entity Data dialog box allows you to associate extended entity data with a specific application.




1 Applications associated with Entity Data

2 Options

### 4.12.118. Applications associated with Entity Data

**1**

**Original application**

Specifies the name of the original application.

**New application**

Specifies the name of the new application. Entities will be reassociated with this application name.

### 4.12.118. Options

**2**

**Skip**

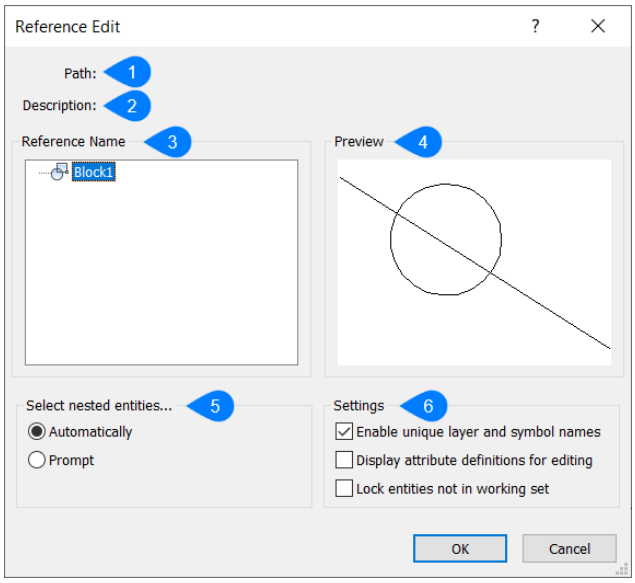
Skips entities that already contains an item with the same name as the new application.

**Remove the existing application before reassociating**

Removes the existing application before reassociating.

### 4.12.119 Reference Edit dialog box

The Reference Edit Dialog Box allows you to edit block references and externally-referenced drawings.




1 Path

2 Description

3 Reference Name

4 Preview

5 Select nested entities

6 Settings

### 4.12.119. Path

**1**

Displays the name of the path.

### 4.12.119. Description

**2**

Displays the description of the path.

### 4.12.119. Reference Name

**3**

Lists the name of the block and nested blocks, if any.

### 4.12.119. Preview

**4**

Displays a preview of the current reference.

### 4.12.119. Select nested entities

**5**

Determines how nested blocks/xrefs are handled.

**Automatically**

Includes nested entities automatically.

**Prompt**
Includes nested entities by selecting them after the dialog box is closed by pressing OK.

### 4.12.119. Settings

**6**
Determines how entities are treated during editing.

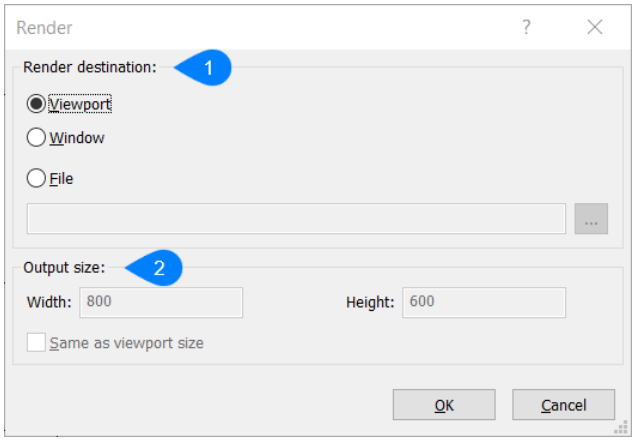
**Enable unique layer and symbol names**
Toggles the use of $0$ in layer and other names.

**Display attribute definitions for editing**
Toggles the inclusion of attributes.

**Lock objects not in working set**
Toggles locking of entities not being edited.

### 4.12.120 Render dialog box

The Render dialog box allows you to generate a photo-realistic rendering using materials and lights.




1 Render destination

2 Output size

### 4.12.120. Render destination

**1**
Specifies where the render will appear.

**Viewport**
Generates the render in the current viewport.

**Window**
Generates the render in a new window.

**File**
Saves the render to a file. Selecting the three dots opens the render output file dialog box.

### 4.12.120. Output size

**2**

Specifies the size of the rendering, in pixels. Larger sizes provide more detail but take longer to render.

**Width**

Specifies the width of the rendering, in pixels.

**Height**

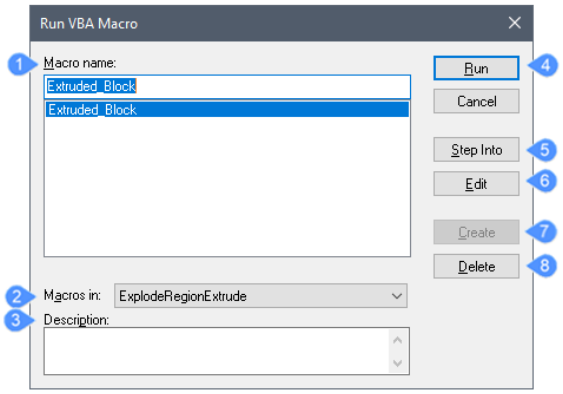
Specifies the height of the rendering, in pixels.

**Same as viewport size**

Matches the output size or the size of the output window to the current viewport size.

### 4.12.121 Run BricsCAD VBA macro dialog box

The **Run VBA Macro** dialog box runs, creates, edits, and deletes VBA macros.




1 Macro name

2 Macros in

3 Description

4 Run

5 Step Into

6 Edit

7 Create

8 Delete

### 4.12.121. Macro name

**1**

Lists the names of macros found in the project file. Choose one to run.

### 4.12.121. Macros in

**2**

Lists the names of projects loaded into the current drawing. Choose one from the drop-down list.

#### 4.12.121. Description

**3**

Describes the macro. You can edit this text and save it pressing the **Close** button.

#### 4.12.121. Run

**4**

Runs the selected macro.

#### 4.12.121. Step Into

**5**

Opens the integrated development environment window. The macro is in debug mode, which means that it runs one line of code at a time.

See the VBAIDE command.

#### 4.12.121. Edit

**6**

Opens the macro in the integrated development environment window. The macro is in editing mode, which allows you to change the code.

See the VBAIDE command.

#### 4.12.121. Create

**7**

Creates new macros; follow these steps:

1 In the **Macro Name** field, enter a name for the new macro.

2 Click **Create**.

3 Notice the integrated development environment window. Enter the VBA code.

See the VBAIDE command.

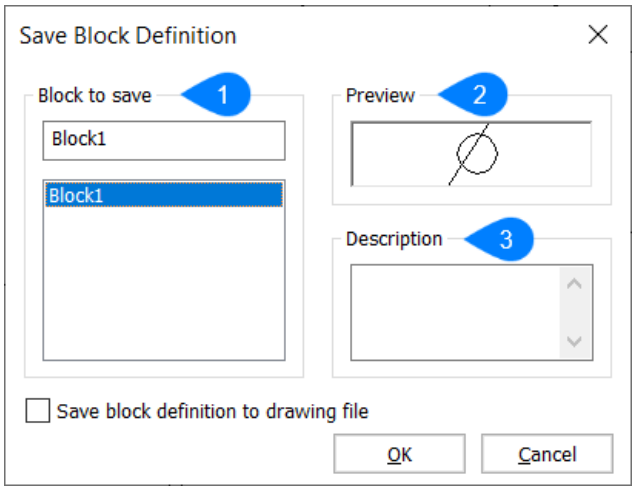
#### 4.12.121. Delete

**8**

Deletes the macro. A dialog asks to confirm the action.

### 4.12.122 Save block definition dialog box

The Save Block Definition dialog box allows you to copy the current block definition to a new name.




1 Block to save

2 Preview

3 Description

### 4.12.122. Block to save

**1**

Specifies a new name for a selected block to make a copy of.

### 4.12.122. Preview

**2**

Displays a preview of the selected block.

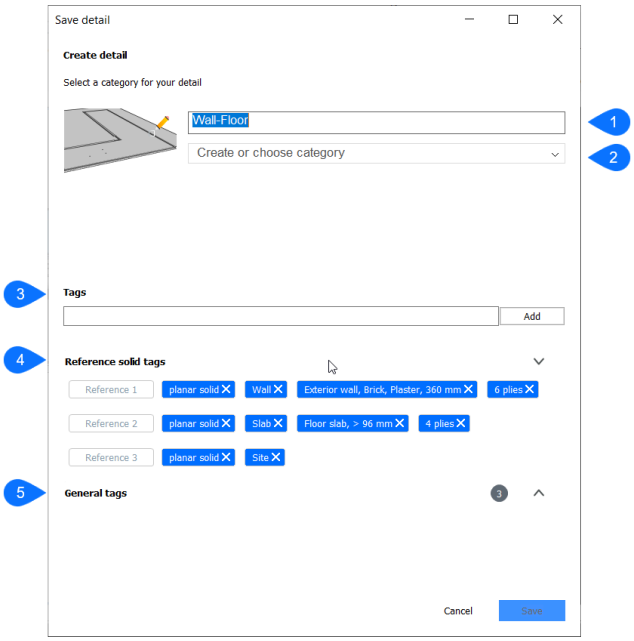
### 4.12.122. Description

**3**

Displays the description of the selected block, if it has one.

### 4.12.123 Save detail dialog box

The Save Detail dialog box allows you to save the selected elements as a detail.




1 Name

2 Category

3 Tags

4 Reference solid tags

5 General tags

### 4.12.123. Name

**1**

Sets the name of the new BIM detail.

### 4.12.123. Category

**2**

Sets the category of the new BIM detail.

### 4.12.123. Tags

**3**

Adds tags to the detail so it can be found easily when searching for this specific detail.

### 4.12.123. Reference solid tags

**4**

Displays tags that specify the detail reference entities.

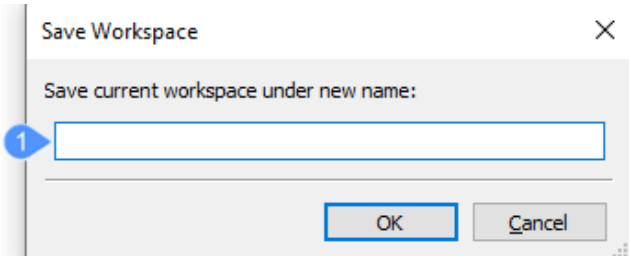
### 4.12.123. General tags

**5**

Displays tags that specify the detail in general.

### 4.12.124 Save workspace dialog box

The **Save Workspace** dialog box allows you save the current workspace under a different name, creating a new workspace.




1 Save current workspace under new name

#### 4.12.124. Save current workspace under new name

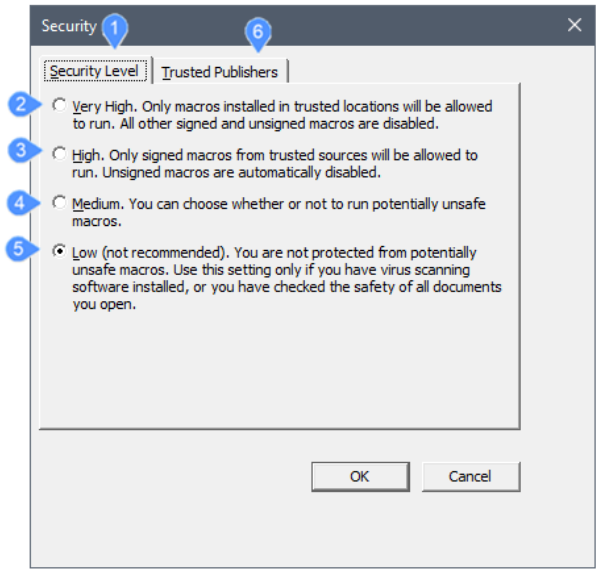
**1**

Specifies the name for the new workspace. Press **OK** to apply the changes.

You can access it later through the **Workspaces** toolbar or from the status bar.

### 4.12.125 Security dialog box

The Security dialog box sets the security level to Low, Medium, High or Very High to determine whether VBA macros are allowed to run automatically.




1 Security Level

2 Very High

3 High

4 Medium

5 Low

6 Trusted Publisher

#### 4.12.125. Very High

**1**

Only VBA macros installed in trusted locations will be allowed to run. All other signed or unsigned macros are disabled.

#### 4.12.125. High

**2**

Only signed macros from trusted sources will be allowed to run. Unsigned macros are automatically disabled.

#### 4.12.125. Medium

**3**

BricsCAD displays a dialog box when a VBA macro runs.

The dialog box allows you to choose whether or not to run the macro.

#### 4.12.125. Low

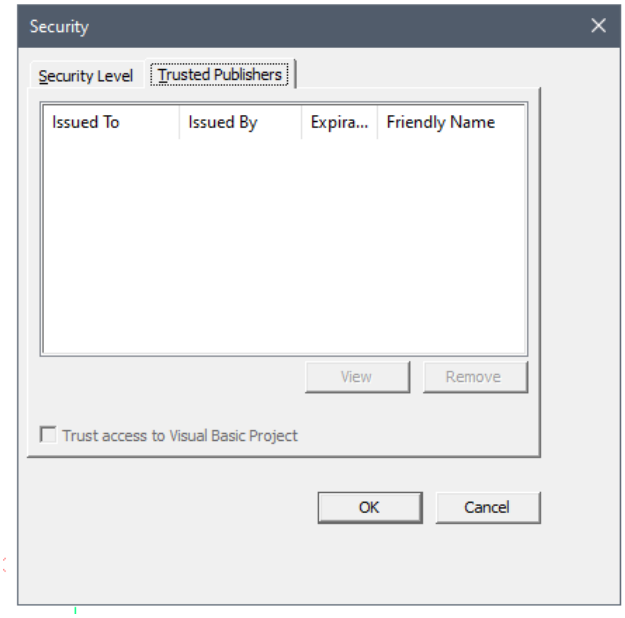
**4**

All VBA macros are run.

#### 4.12.125. Trusted Sources Option

**5**

Lists the names of VBA macros that can be trusted. These are the only macros that will run when security is set to High.




**Remove**

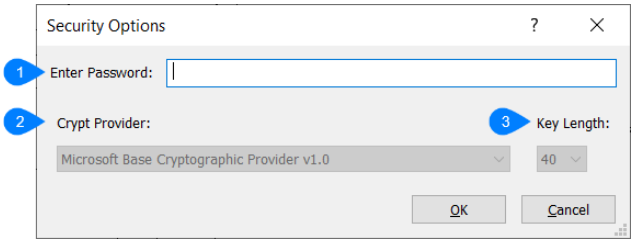
Removes the selected VBA macro from the list.

**Trust access to Visual Basic Project**

Allows access to the VBA Project.

### 4.12.126 Security options dialog box

The Security Options dialog box allows you to set a password to protect your drawing from unauthorized access.




1 Enter Password

2 Crypt Provider

3 Key Length

#### 4.12.126. Enter Password

**1**

Specify a password by entering one or more characters.

#### 4.12.126. Crypt Provider

**2**

Specifies the encryption level for drawing properties. Choose an encryption provider supplied by the operating system.

#### 4.12.126. Key Length

**3**

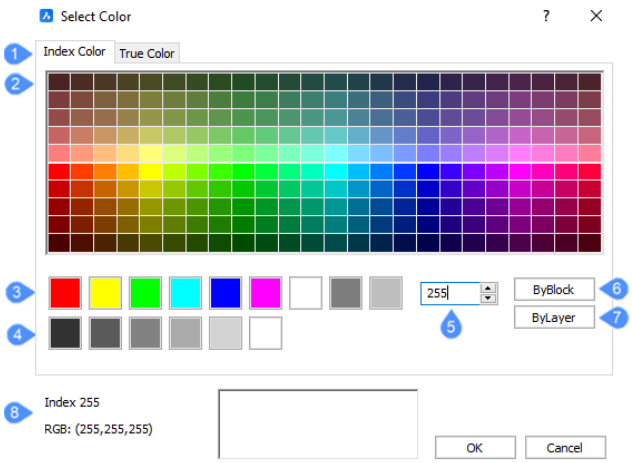
Select a length for the encryption key. The longer the key, the higher the level of protection is, because it takes longer to break the encrypted password.

### 4.12.127 Select color dialog box

The Select Color dialog box allows you to select a color for layers, grids, dimensions, background etc. It is composed from two tabs: **Index Color** tab and **True Color** tab.

#### 4.12.127. Index Color tab

**1**




1 Index Color tab

2 Colors

3 Basic colors

4 Shades of gray

5 Color index

6 ByBlock

7 ByLayer

8 Color sample

**Colors**
Displays the other 240 colors.

**Basic colors**
Displays the basic colors.

**Shades of gray**
Displays the shades of grays.

**Color index**
Displays the index of the selected color. You can insert an index and see the preview of the color at field number 8.

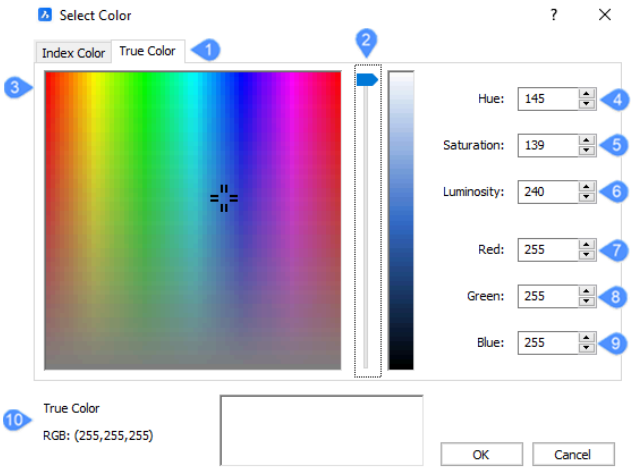
**ByBlock**
Sets the color to ByBlock in which entities take on the color defined by their block.

**ByLayer**
Sets the color to ByLayer in which entities take on the color defined by their layer's property.

**Color sample**
Displays a preview for the selected color and the RGB color value.

### 4.12.127. True Color Tab

**2**




1 True Color Tab

2 Luminosity

3 Color

4 Hue

5 Saturation

6 Luminosity

7 Red

8 Green

9 Blue

10 Color sample

**Luminosity**
Chooses the luminosity of the color.

**Color**
Chooses hue (left and right) and saturation (up and down).

**Hue**
Specifies the color's hue. It is ranging from red through green, blue, and pink.
It ranges from 0 to 240.

**Saturation**
Specifies the intensity of the color, ranges from 0 to 240. Saturation is the intensity of the color, where 0 = gray and 240 = full color.

**Luminosity**
Specifies the brightness of the color, ranges from 0 to 240. Luminosity is the lightness or darkness, where 0 = black and 240 = white. Hue, saturation and luminosity work together, while red, green and blue work separately.

**Red**
Specifies the amount of red in the color, ranges from 0 (no red) to 255 (full red).

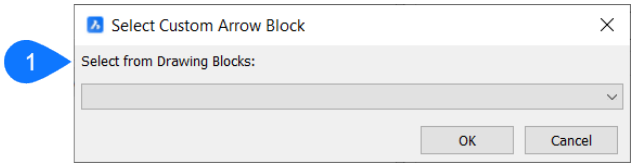
**Green**
Specifies the amount of green in the color, ranges from 0 (no green) to 255 (full green).

**Blue**
Specifies the amount of blue in the color, ranges from 0 (no blue) to 255 (full blue).

**Color sample**
Displays a preview for the selected color and the RGB color value.

### 4.12.128 Select Custom Arrow Block dialog box

The Select Custom Arrow Block dialog box allows you to save the selected elements as a detail.



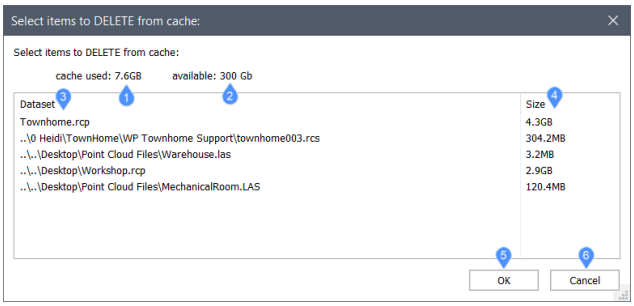

1 Select from Drawing Blocks

### 4.12.128. Select from Drawing Blocks

**1**
Specifies one of the blocks in the drawing to use as arrowhead.

### 4.12.129 Select items to delete from cache dialog box

TheSelect Items To Delete from Cache dialog box enables you to delete point cloud files that are stored in the cache.




1 Cache used

2 Available

3 Dataset

4 Size

5 OK

6 Cancel

### 4.12.129. Cache used

**1**
Specifies the total amount of used cache.

### 4.12.129. Available

**2**

Specifies the total amount of available cache. The available cache is controlled by the POINTCLOUDCACHESIZE system variable.

### 4.12.129. Dataset

**3**

Displays a list of cached point cloud files including their names and locations. You can select point cloud files from the list to delete them from the cache.

### 4.12.129. Size

**4**

Displays the amount of cache used for each point cloud file.

### 4.12.129. OK

**5**

Displays the Confirm deleting selected cache items dialog box where you can review the list of point clouds to be deleted.

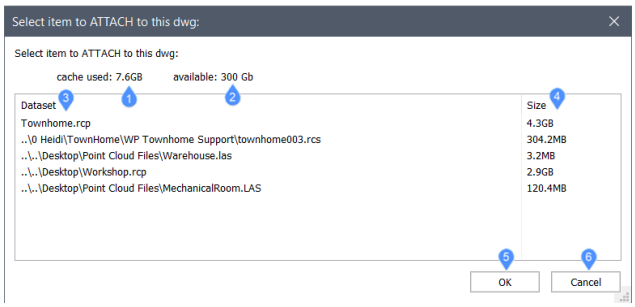
### 4.12.129. Cancel

**6**

Closes the dialog box without deleting point cloud files from the cache.

### 4.12.130 Select item to attach to this DWG

TheSelect item to attach to this DWG dialog box enables you to attach point cloud files that have already been processed and stored in the cache.




1 Cache used

2 Available

3 Dataset

4 Size

5 OK

6 Cancel

### 4.12.130. Cache used

**1**

Specifies the total amount of used cache.

### 4.12.130. Available

**2**

Specifies the total amount of available cache. The available cache is controlled by the POINTCLOUDCACHESIZE system variable.

### 4.12.130. Dataset

**3**

Displays a list of cached point cloud files including their names and locations. You can select a cached file from the list to attach to the current drawing.

### 4.12.130. Size

**4**

Displays the amount of cache used for each point cloud file.

### 4.12.130. OK

**5**

Displays the Attach point cloud dialog box where you can specify how to attach the selected point cloud file to the current drawing.

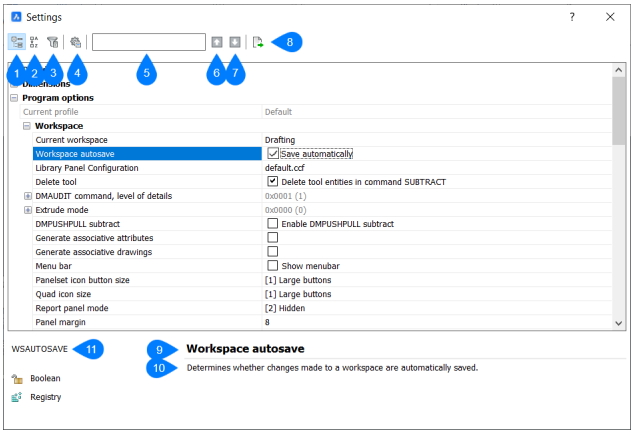
### 4.12.130. Cancel

**6**

Closes the dialog box without attaching a point cloud file.

## 4.12.131 Settings dialog box

The Settings dialog box allows you to change the values of system variables and user preferences.




1 Categorized

2 Alphabetic

3 Show differences

4 Dialog configurations

5 Search bar

6 Next

7 Previous

8 Export

9 Name

10 Description

11 Properties

### 4.12.131. Categorized

**1**

Lists variables in related groups.

### 4.12.131. Alphabetic

**2**

Lists variables in related groups.

### 4.12.131. Show differences

**3**

Lists variables whose values are changed from the initial (default) values.

### 4.12.131. Dialog configurations

**4**

Searches for variables, configures how the search is carried out, and specifies how differences are handled. The Configure Settings dialog box opens when clicking the icon.

### 4.12.131. Search bar

**5**

Lists variables trough a real-time search field when entering the first few letters of their names or descriptions.

### 4.12.131. Next

**6**

Jumps to the next candidate that matches the search phrase.

### 4.12.131. Previous

**7**

Jumps to the previous candidate that matches the search phrase.

### 4.12.131. Export

**8**

Opens the Export Settings dialog box to allow you to save all settings and values to a CSV file.

### 4.12.131. Name

**9**

Displays the name of the selected system variable.

### 4.12.131. Description

**10**

Displays a brief description of the selected system variable.

### 4.12.131. Properties

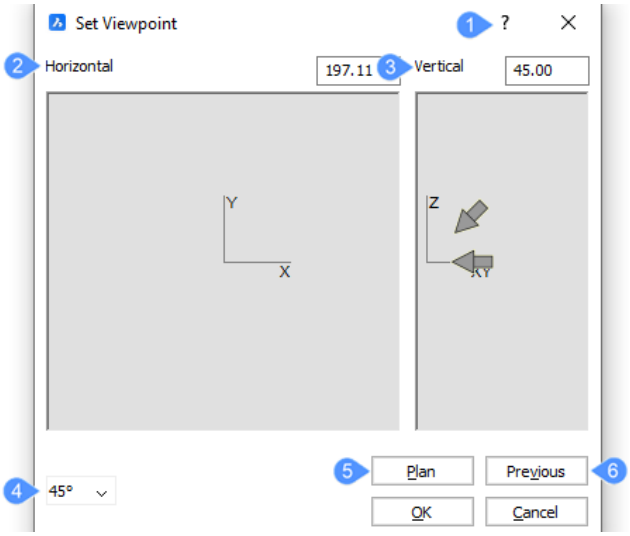
**11**

Lists several basic properties/characteristics of the selected system variable.

### 4.12.132 Set viewpoint dialog box

The Set Viewpoint dialog box allows you to set the current 3D viewport or plan view.

***Прим.:*** Turn off the perspective mode (view PERSPECTIVE command), because this command is not available in the Perspective mode.




1 Command reference

2 Horizontal

3 Vertical

4 Angle Preset

5 Plan

6 Previous

### 4.12.132. Command reference

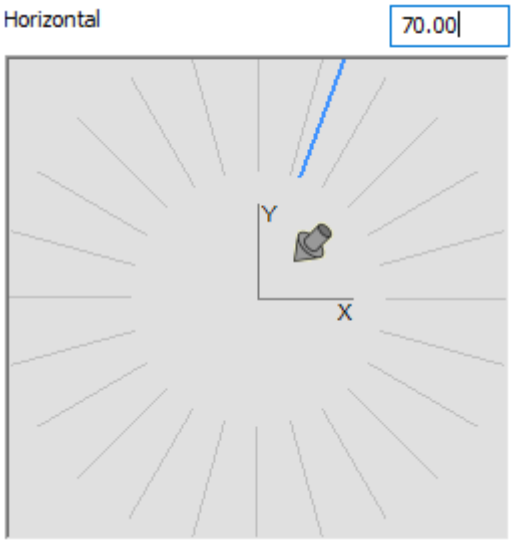
**1**

Opens the Bricsys Help article about the DDVPOINT command.

### 4.12.132. Horizontal

**2**

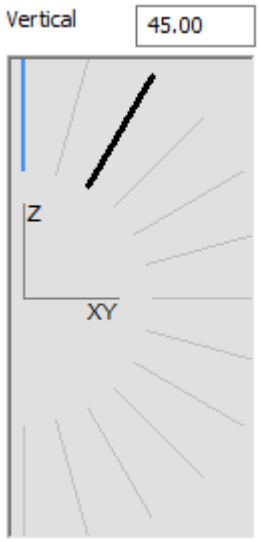
Specifies the horizontal angle from the X - axis. Click a preset angle or type a value in the **Horizontal** field. Click at 0°, 45°, 90°, 135°, 180°, 225°, 270° or 315° in the inner circle. The shape of the arrow indicates whether the view is horizontal or at an angle.



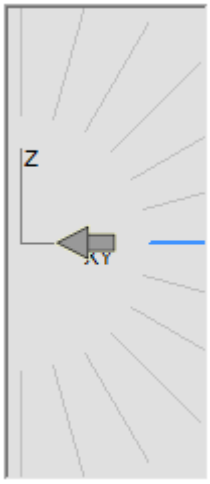

### 4.12.132. Vertical

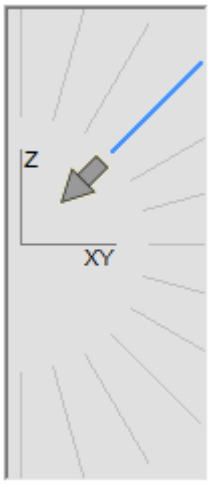
**3**

Specifies the vertical angle from the XY plane. Click a preset angle, or else type a value in the Vertical field.

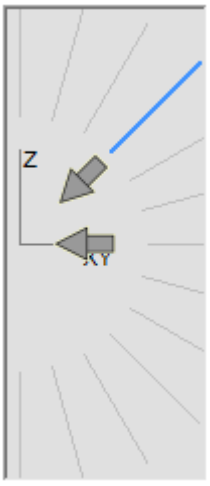


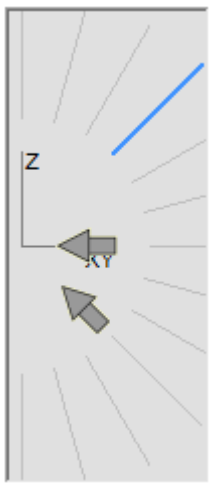

Click in the inner half circle. (1) View direction is horizontal. (2) View direction is 45° down (not available if PresetAngle = 45°).

(1) 


(2) 


View direction is 45° down, except for 0°, 90°, 180° and 270° horizontal angles, which generate orthographic views: front, back, left and right. (3) First select the horizontal arrow, then (4) click the 45° down arrow.

(3) 


(4) 


### 4.12.132. Angle Preset

**4**

Sets markers at 5, 15, or 45 degrees; default = 45.

### 4.12.132. Plan

**5**

Reverts the 3D viewpoint to 2D plan view.

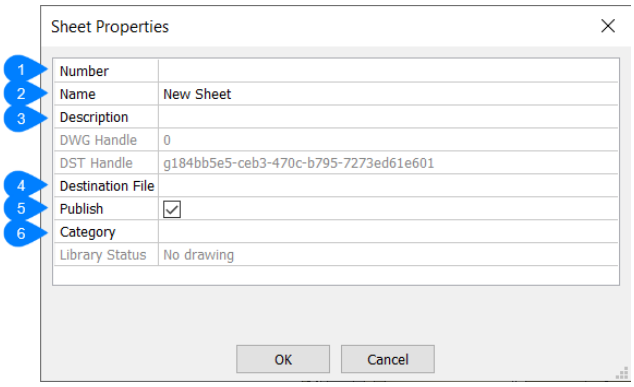
### 4.12.132. Previous

**6**

Reverts to the previous viewpoint.

### 4.12.133 Sheet Properties dialog box

The Sheet Properties dialog box allows you to view and edit properties of a sheet.




1 Number

2 Name

3 Description

4 Destination File

5 Publish

6 Category

#### 4.12.133. Number

**1**

Specifies the number of the selected sheet.

#### 4.12.133. Name

**2**

Specifies the name of the selected sheet.

#### 4.12.133. Description

**3**

Specifies a short description of the selected sheet.

#### 4.12.133. Destination File

**4**

Specifies the file in which the sheet will be saved.

#### 4.12.133. Publish

**5**

Toggles the option to publish the currently selected sheet.

#### 4.12.133. Category

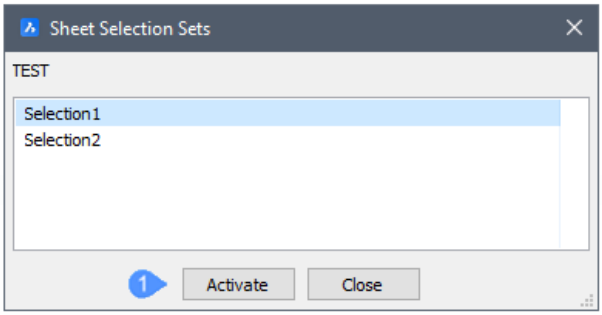
**6**

Specifies the category of the selected sheet.

### 4.12.134 Sheet selection sets dialog box

The Sheet Selection Sets dialog box allows you to manage the sheet selections.

To create a sheet selection, right click when multiple sheets are selected to open the **Sheet Selection** context menu and **Save Sheet** selection with a name.




1 Activate

### 4.12.134. Activate

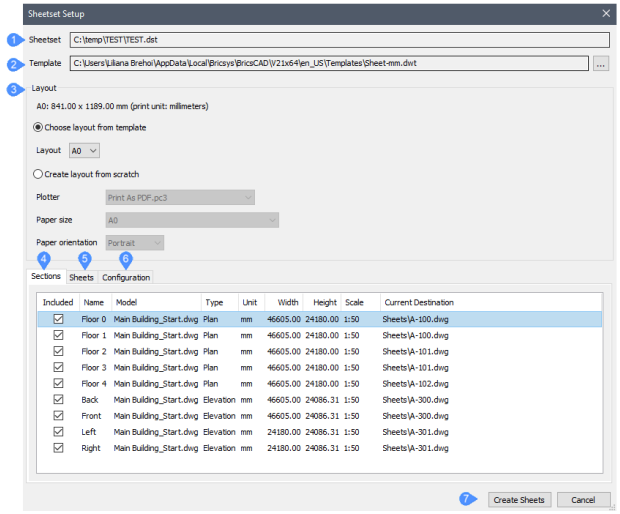
**1**

Restores the selected selection sheet set.

Right click a selection sheet set to display a context menu.

### 4.12.135 Sheetset setup dialog box

The **Sheetset Setup** dialog box provides the settings that are used for plotting and publishing.




1 Sheetset

2 Template

3 Layout

4 Sections Tab

5 Sheets Tab

6 Configuration Tab

7 Create Sheets

### 4.12.135. Sheetset

**1**

Shows the path of the sheetset data file (.dst) that stores the associations and information that define a sheet set.

### 4.12.135. Template

**2**

Shows the path of the current template that contains the page setups for all new sheets.

Click **Browse** () to change the template.

### 4.12.135. Layout

**3**

Selects which layout (different paper sizes) the section results will be generated.

- **Choose layout from template**: Allows to select a predefined layout from a drop-down list.
- **Create layout from scratch**: Allows to set:
  - **Plotter**: Choose from the drop-down list with the available plotters.
  - **Paper size**: Choose the paper size from the drop-down list.
  - **Paper orientation**: Choose the paper orientation from the drop-down list that opened.

### 4.12.135. Sections Tab

**4**

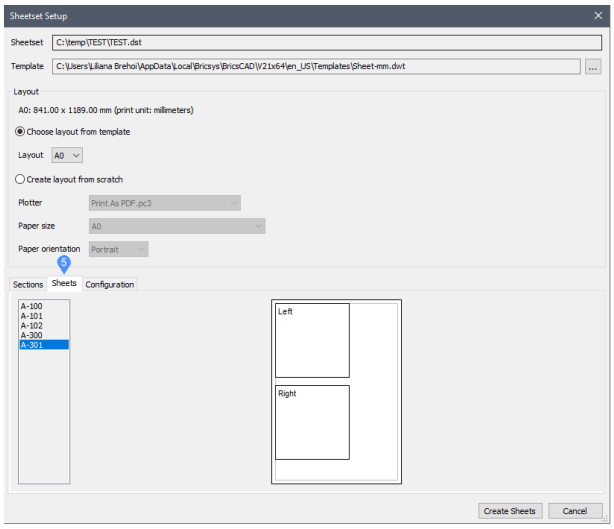
Lists all sections. For each section with an empty **Current Destination** field, a viewport and a sheet view is generated.

- **Included**: Specifies which sections you want to be generated.
- **Name**: Displays the name of the sections.
- **Model**: Displays the name of the model from the sections area.
- **Type**: Displays the section type.
- **Unit**: Displays the drawings unit.
- **Width**: Displays the maximum model width dimension.
- **Height**: Displays the maximum model height dimension.
- **Scale**: Displays the scale for the section sheet. It may be changed pressing right click and selecting a new scale form the drop-down list.
- **Current Destination**: Displays the destination .dwg file of the current section.

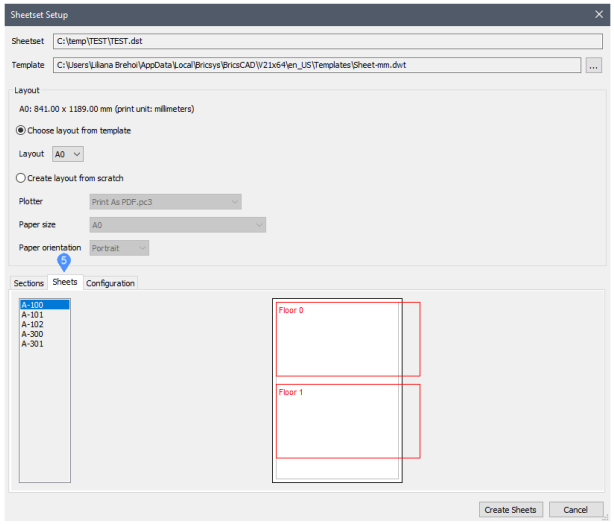
### 4.12.135. Sheets Tab

**5**

Shows how the viewports will be displayed in the layout.

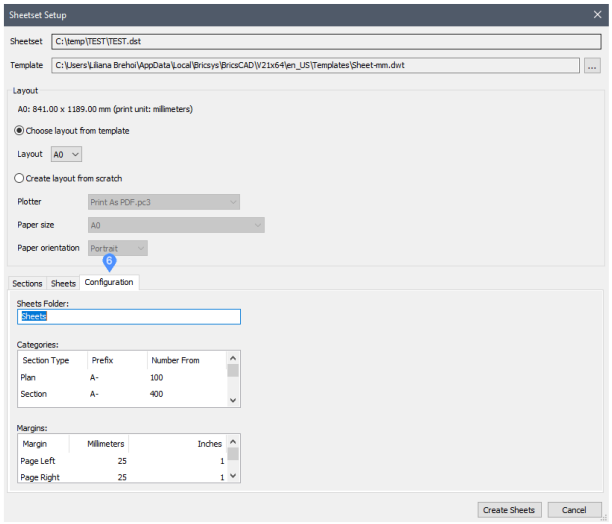



***Прим.:*** When the viewport is larger than the layout, it displays in red.



### 4.12.135. Configuration Tab

**6**




Slowly double-click (or click and hit Space/Enter) to change the properties:

- Under **Sheets Folder** you may change the name of the folder where the sheets will be stored.
- Under **Categories** you can change the **Prefix** and **Number From** for each Section Type.
- Under **Margins** you can set the margins of the page.

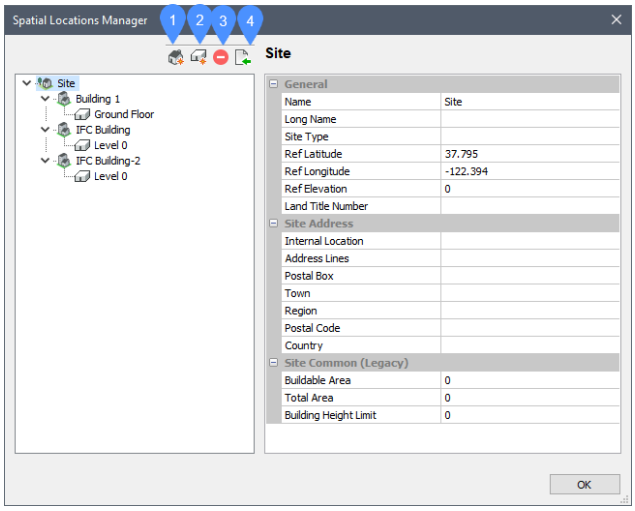
### 4.12.135. Create Sheets

**7**

The sheets are generated.

### 4.12.136 Spatial Locations Manager dialog box

The Spatial Locations Manager dialog box allows you to create and edit sites, buildings and stories.




1 New Building

2 New Story

3 Delete

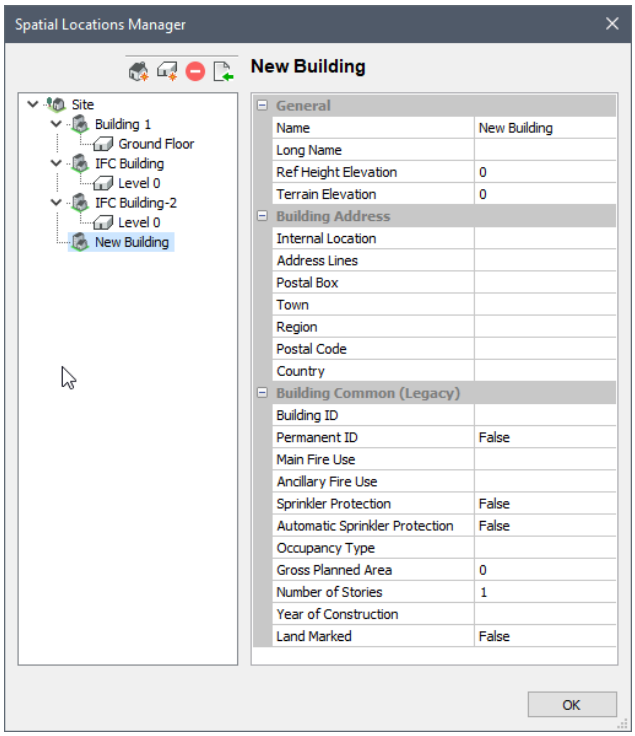
4 Import

### 4.12.136. New Building

**1**

Adds a new building to the model.

Fill out the properties grid:

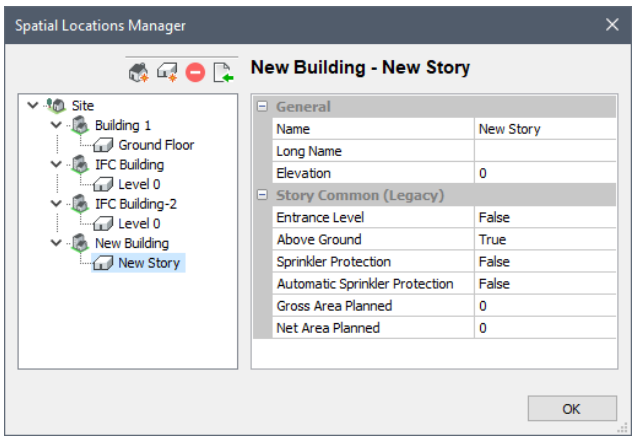



### 4.12.136. New Story

**2**

Adds a new story in the selected building.

The properties grid is completed.



### 4.12.136. Delete

**3**

Deletes the selected building and story.

### 4.12.136. Import

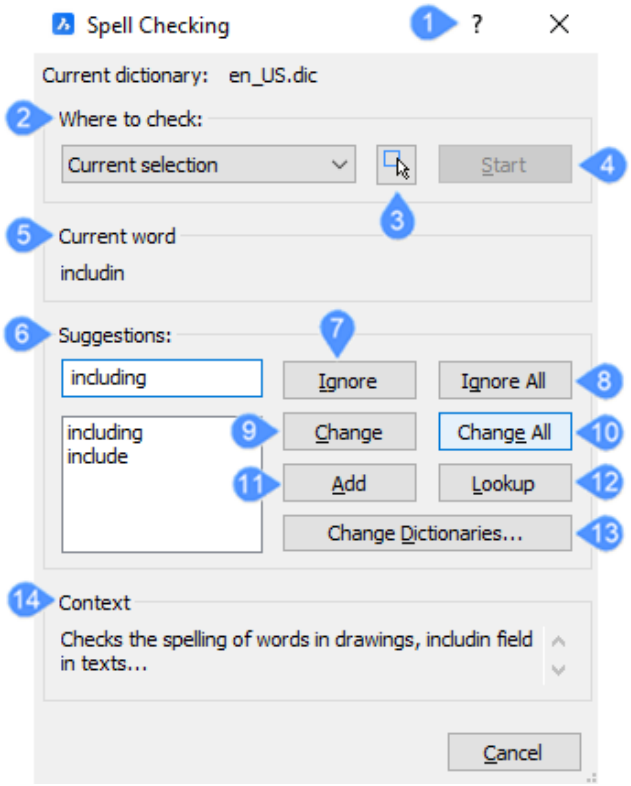
**4**

Imports spatial locations from a text file.

Displays a Select File to Insert dialog box from which will open the *.txt spatial location file.

### 4.12.137 Spell checking

The **Spell Checking** dialog box allows you to check the spelling of words in drawings, including fields in texts, multiline texts, leader, multileader, table and block attributes.




1 Command reference

2 Where to check

3 Select entities

4 Start

5 Current word

6 Suggestions

7 Ignore

8 Ignore All

9 Change

10 Change All

11 Add

12 Lookup

13 Change Dictionaries…

14 Context

### 4.12.137. Command reference

**1**

Opens the Bricsys Help article about the SPELL command.

### 4.12.137. Where to check

**2**

Checks text in:

- **Entire drawing**: Checks all text in the drawing.
- **Selected entities**: Checks text in selected entities. Click the **Select Entities** button to choose text.

### 4.12.137. Select entities

**3**

Select the text to be checked. Choose one or more pieces of text and press Enter to return to the dialog box.

### 4.12.137. Start

**4**

Starts checking the spelling of words.

### 4.12.137. Current word

**5**

Displays the misspelled word.

### 4.12.137. Suggestions

**6**

Lists the suggested replacements for misspelled words. Choose a replacement word and then click **Change** or **Change All**.

### 4.12.137. Ignore

**7**

Ignores the spelling of the word.

### 4.12.137. Ignore All

**8**

Ignores all instances of the word.

#### 4.12.137. Change
**9**

Changes the word to the one selected from the **Suggestion** list.

#### 4.12.137. Change All
**10**

Changes all instances of the word to the one selected from the **Suggestion** list.

#### 4.12.137. Add
**11**

Adds the word to the word list.

#### 4.12.137. Lookup
**12**

Provides additional alternative spellings for the word selected in the **Suggestions** list.

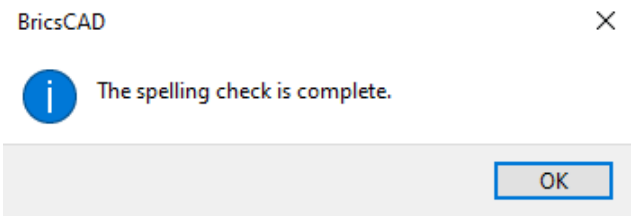
#### 4.12.137. Change Dictionaries...
**13**

Changes the dictionary used for spell checking. Displays the **Change Dictionaries** dialog box.

#### 4.12.137. Context
**14**

Displays the context in which the misspelled word appears.

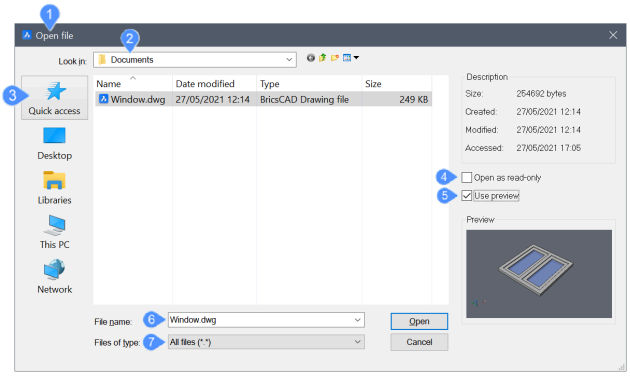
When the spelling check is completed, the following message box appears:




### 4.12.138 Standard open file dialog box

The Standard open file dialog box allows you to open or import a file.

This dialog box can be accessed using multiple commands.




1 Dialog name

2 Look in

3 Quick access

4 Open as read-only

5 Use preview

6 File name

7 Files of type

### 4.12.138. Dialog name

**1**

Displays the dialog name depending on the command used.

### 4.12.138. Look in

**2**

Allows you to specify the location of your file. To change it to a different disk drive or folder, click the arrow and select another disk drive or folder from the drop-down list.

To scroll the list, press the F4 key.

### 4.12.138. Quick access

**3**

The Quick access is available if USESTANDARDOPENFILEDIALOG = 1.

You can populate the Quick access with folders that you want to be always available, not only with folders that have being used recently. The folders shown in Quick access can be configured in Windows Explorer.

The Quick access folders are available during both open and save operations.

### 4.12.138. Open as read-only

**4**

Tick the checkbox to open the file as read-only. Consequently, you do not have permission to write new information or edit the data in the file.

### 4.12.138. Use preview

**5**

Tick the checkbox to view a part of the file before opening.

### 4.12.138. File name

**6**

Specifies the name of the file.

### 4.12.138. Files of type

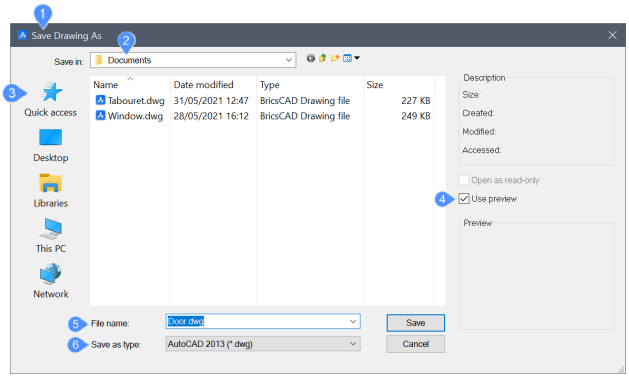
**7**

Displays a list of valid file types. You can select a file type to filter the files list so that it only shows files matching that type.

### 4.12.139 Standard save file dialog box

The Standard Save File dialog box allows you to save or export a file.

This dialog box can be accessed using multiple commands.




1 Dialog name

2 Save in

3 Quick access

4 Use preview

5 File name

6 Files of type

### 4.12.139. Dialog name

**1**

Displays the dialog name depending on the command used.

### 4.12.139. Save in

**2**

Allows you to specify the location of your file. To change it to a different disk drive or folder, click the arrow and select another disk drive or folder from the drop-down list.

To scroll the list, press the F4 key.

### 4.12.139. Quick access

**3**

The Quick Access is available if USESTANDARDOPENFILEDIALOG = 1.

You can populate Quick access with folders that you want to be always available, not only for folders that have being used recently. The folders shown in Quick access can be configured in Windows Explorer.

Quick Access folders are available during both open and save operations.

### 4.12.139. Use preview

**4**

Tick the checkbox to see a part of the file before saving.

***Прим.:*** A preview is available when selecting an existing file to replace it.

### 4.12.139. File name

**5**

Specifies the name of the file. You can accept the predefined name or you can specify a different name.

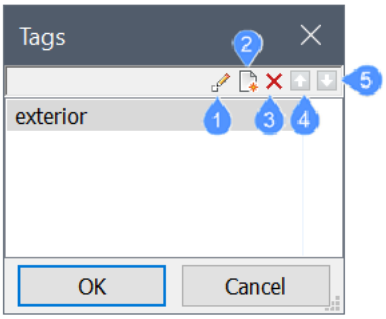
### 4.12.139. Files of type

**6**

Displays a list of valid file types. You can select a file type to filter the files list so that it only shows files matching that type.

### 4.12.140 Tags dialog box

The Tags dialog box allows you to add or edit compositions and physical materials tags.




1 Edit

2 New

3 Delete

4 Move up

5 Move down

### 4.12.140. Edit

**1**

Edits the name of the selected tag.

### 4.12.140. New

**2**

Adds a new tag at the bottom of the list.

### 4.12.140. Delete

**3**

Deletes the selected tag.

### 4.12.140. Move up

**4**

Moves up the selected tag.

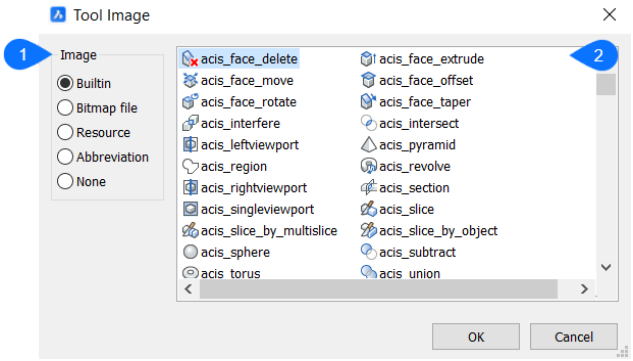
### 4.12.140. Move down

**5**

Moves down the selected tag.

## 4.12.141 Tool image dialog box

The Tool Image dialog box allows you to specify the image of tools.




1 Image

2 List of tool images

### 4.12.141. Image

**1**

Specifies the way to set the tool images.

**Builtin**

Specifies the tool image from one of the builtin tool images from the list.

**Bitmap File**

Specifies the tool image from a bitmap file.

**Resource**

Specifies the resource of the tool image.

**Abbreviation**

Specifies an abbreviation for the tool instead of an image.

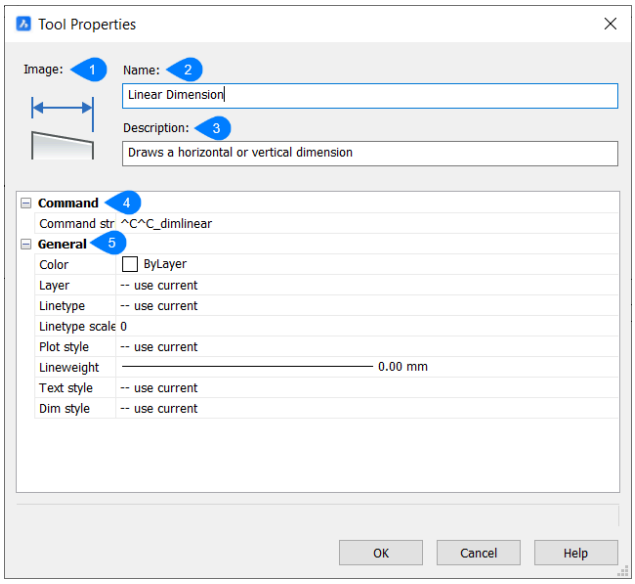
**None**
No image or abbreviation is used for the tool.

### 4.12.141. List of tools

**2**

Displays a list of built in tool images when the **Builtin** option is selected.

### 4.12.142 Tool Properties dialog box




1 Image

2 Name

3 Description

4 Command String

5 General

### 4.12.142. Image

**1**

Displays an image of the selected tool.

### 4.12.142. Name

**2**

Specifies the name of the selected tool.

### 4.12.142. Description

**3**

Specifies the description of the selected tool.

### 4.12.142. Command String

**4**

Specifies the macro to run when you click the icon; you can use any macro characters used for menus and toolbar buttons.

### 4.12.142. General

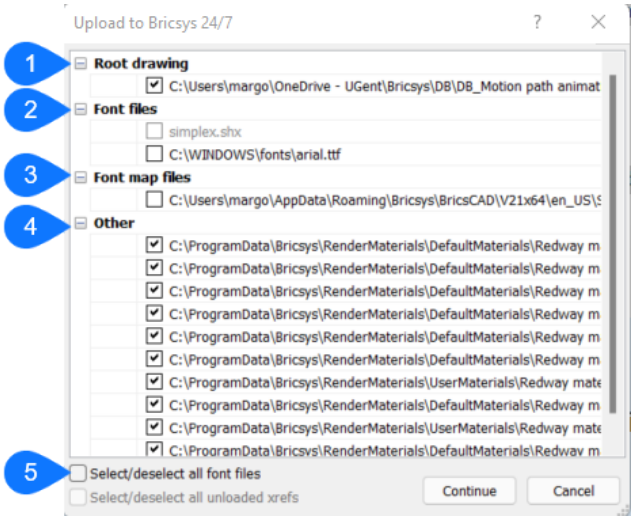
**5**

Specifies the properties of the entity; the properties vary, depending on the tool.

### 4.12.143 Upload to Bricsys 24/7 dialog box

The Upload to Bricsys 24/7 dialog box allows you to upload packages of drawing files and it dependencies to Bricsys 24/7

***Прим.:*** The project needs to be saved before uploading it to Bricsys 24/7.




1 Root drawing

2 Font files

3 Font map files

4 Other

5 Select/deselect all font files

### 4.12.143. Root drawing

**1**

Displays a list of the current drawing files.

### 4.12.143. Font files

**2**

Displays a list of the currently used font files.

### 4.12.143. Font map files

**3**

Displays a list of the currently used font map files.

### 4.12.143. Other

**4**

Displays a list of other used files such as render material images.

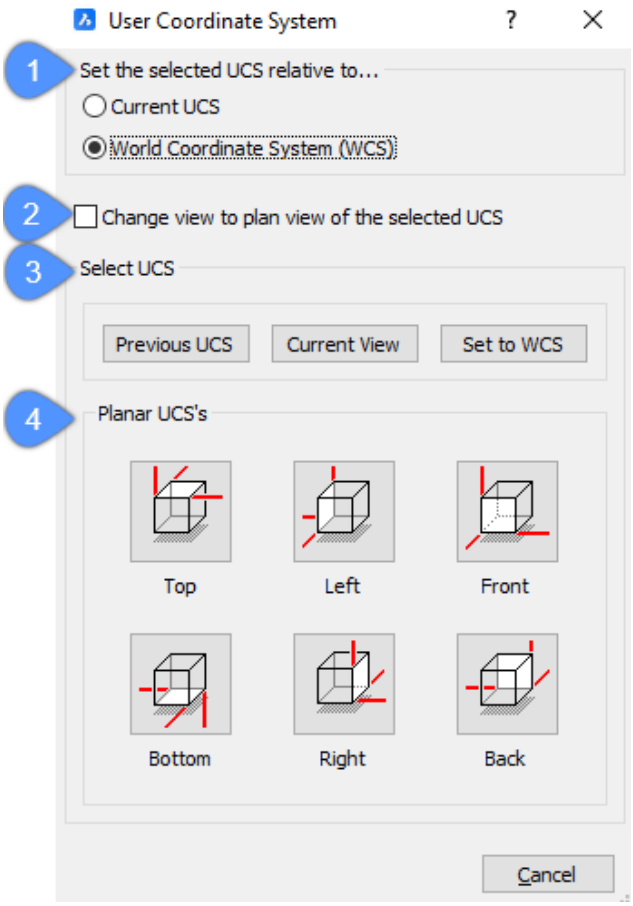
### 4.12.143. Select/deselect all font files

**5**

When checked, all font files and font map files are checked if this was not the case before. When unchecked, all font files are unchecked if there were any checked before. Pressing the **Continue** button will ask you to sign in Bricsys 24/7 to complete the uploading process.

### 4.12.144 User coordinate system dialog box

The **User Coordinate System** dialog box allows you to set the UCS to a specified viewpoint.




1 Set the selected UCS relative to...

2 Change view to plan view of the selected UCS

3 Select UCS

### 4.12.144. Set the selected UCS relative to...

**1**

Specifies how the new UCS is set:

- **Current UCS**: Sets the new UCS relative to the current UCS.
- **World Coordinate System**: Sets the new UCS relative to the WCS. World Coordinate System is the default coordinate system when no UCS is set and it cannot be changed.

### 4.12.144. Change view to plan view of the selected UCS

**2**

Toggles whether the plan view is displayed when switching UCSES:

- **Yes**: plan view of the new UCS is displayed.
- **No**: the viewpoint does not change.

### 4.12.144. Select UCS

**3**

Sets the UCS to one of the following coordinate system. Clicking one of these buttons sets the UCS and then immediately exits the dialog box.

- **Previous UCS**: Changes the UCS to the previous UCS.
- **Current View**: Changes the UCS to match the current viewpoint.
- **Set to UCS**: Changes the UCS to match the WCS.

### 4.12.144. Planar UCS's

**4**

Sets the UCS to one of the following standard orthographic views. After you click one of the options, the program sets the UCS and then exits the dialog box.

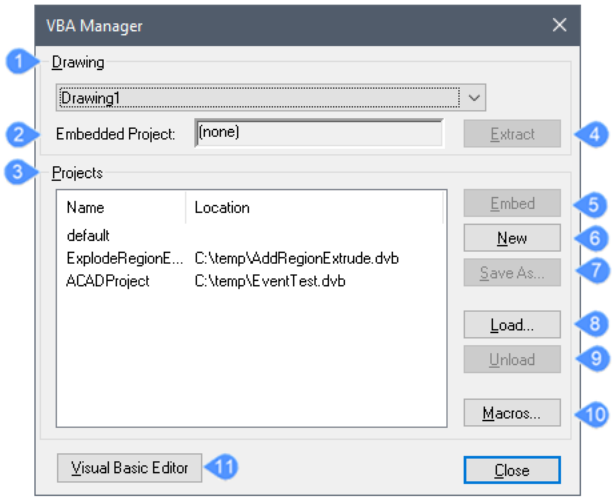
- **Top**: Changes the UCS to match the top view.
- **Left**: Changes the UCS to match the left view.
- **Front**: Changes the UCS to match the front view.
- **Bottom**: Changes the UCS to match the bottom view.
- **Right**: Changes the UCS to match the right view.
- **Back**: Changes the UCS to match the back view.

### 4.12.145 User profile manger dialog box

The User Profile Manager dialog box allows you to set the current user profile, to create, copy, delete, import, and export profiles.

### 4.12.146 VBA manager dialog box

The VBA Manager dialog box allows you to manage VBA projects.




1 Drawing

2 Embedded Project

3 Projects

4 Extract

5 Embed

6 New

7 Save As

8 Load

9 Unload

10 Macros

11 Visual Basic Editor

### 4.12.146. Drawing

**1**

Displays the names of drawings currently open in BricsCAD:

Choose a drawing name from the drop-down list; the embedded project is listed in the Embedded Project text box, if any.

### 4.12.146. Embedded Project

**2**

Displays the name of the embedded project, if any.

### 4.12.146. Projects

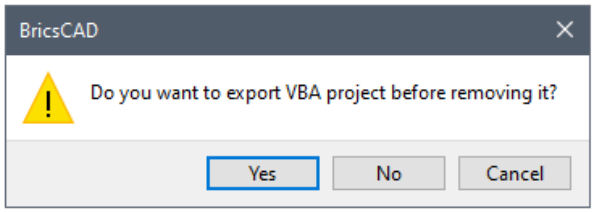
**3**

**Name**

Displays the names of projects loaded into BricsCAD at this time.

**Location**
Displays the drive and folder names of DVB and VBI files.

### 4.12.146. Extract
**4**
Removes the project from the current drawing; displays dialog box:




**Yes**: displays the Save As dialog box:

Name the project and click **Save**.

**No**: removes the project from the drawing without saving it as a DVB file.

**Cancel**: cancels the command; the project remains in the drawing.

### 4.12.146. Embed
**5**
Embeds a project in the current drawing. If the project is a new one, and has not yet been saved, this option displays the *Save As* dialog box.

Each drawing can contain just one embedded project; after one project is embedded, the Embed button becomes unavailable. To change embedded projects, use the Extract option to remove the current project.

### 4.12.146. New
**6**
Creates a new VBA project; gives the generic name VbaProject . Use the Save As option to rename and save the DVB or VBI project file.

### 4.12.146. Save As
**7**
Saves the selected project by another name. Displays the *Save As* dialog box.

### 4.12.146. Load
**8**
Loads DVB and VBI project files; displays the *Open* dialog box.

### 4.12.146. Unload
**9**
Unloads the selected project without a warning.

### 4.12.146. Macros

**10**

Displays the *Run BricsCAD VBA Macro* dialog box.

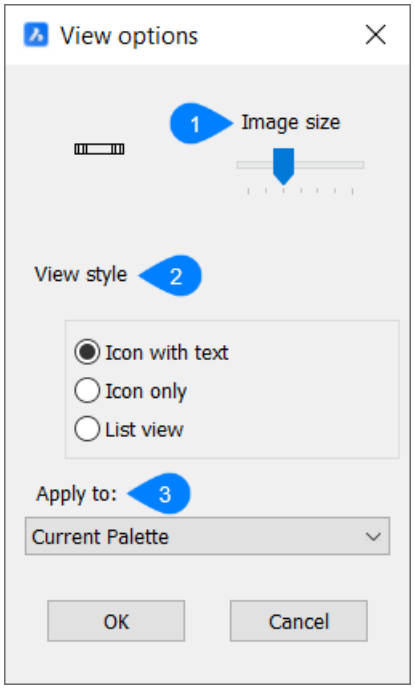
### 4.12.146. Visual Basic Editor

**11**

Opens the VBA integrated development environment.

### 4.12.147 View Options dialog box

The View Options Dialog Box allows you to customize how the tools in the tool palettes panel are shown.




1 Image size

2 View style

3 Apply to

### 4.12.147. Image size

**1**

Sets the size of the displayed icons by sliding the indication arrow.

### 4.12.147. View style

**2**

Specifies how the tools are shown in the palette.

**Icon with text**

Displays the tools in a grid view with their icon and name.

**Icon only**

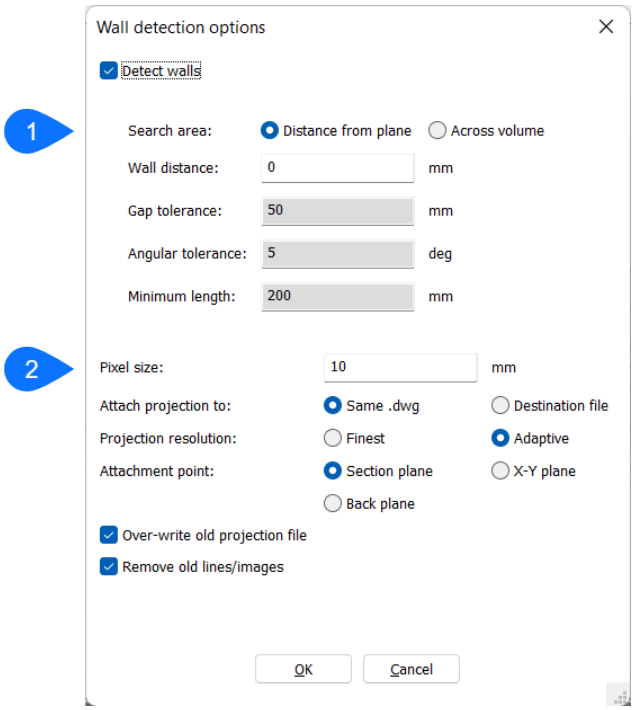
Displays the tools in a grid view with their icon only.

**List view**
Displays the tools in a list view with their icon and name next to it.

### 4.12.147. Apply to
**3**
The changes made can either be applied to the current palette only, or to all palettes.

### 4.12.148 Visual settings dialog box

### 4.12.149 Wall detection options dialog box




1 Parameters that control 2D line drawing

2 Parameters that control the generated raster image

### 4.12.149. Parameters that control 2D line drawing
**1**
**Detect walls**
When checked, lines representing walls will be generated with the raster image. When unchecked, only a raster image is created.

**Search area**

- **Distance from plane**: detects walls in a given radius from a plane. This option is often used to generate vertical sections.
- **Across volume**: detects all walls in the point clouds. This option is often used to generate floor plans.

**Wall distance**
Controls the search area for wall detection.

***Прим.:*** This option is only available when 'Distance from plane' is selected.

**Gap tolerance**
Controls the search area for the algorithm to draw lines. Gaps smaller than the specified value will be closed.

**Angular tolerance**
Controls the angular tolerance of the generated lines. A high tolerance will result in pronounced right angles. This is useful for perpendicular plans.
Generated lines with a smaller angle than the specified value will be merged.

**Minimum length**
Controls the minimum length of a generated line. Lines with a smaller length than the specified value will not be generated.

### 4.12.149. Parameters that control the generated raster image 2

**Pixel size**
Controls the pixel size of the generated image.

***Прим.:*** the pixel size is in model units. Higher pixel size results in a lower resolution, resulting in a faster process.

**Attach projection to**
Determines where the resulting image is generated.

**Same.dwg**
The projection is made in the same file

**New destination**
Creates a new .dwg file to make the projection.

***Прим.:*** You can overwrite a previous file or create a new one.

**Projection resolution**

**Finest**
Produces a sharp and detailed background picture.

**Adaptive**
Produces a less detailed and softer background image.

**Attach point**
Determines the insertion point for the generated image.

- In-place: places the lines and image in the same location and orientation as the sectionplane.
- X-Y plane: aligns the section with the X_Y axis.

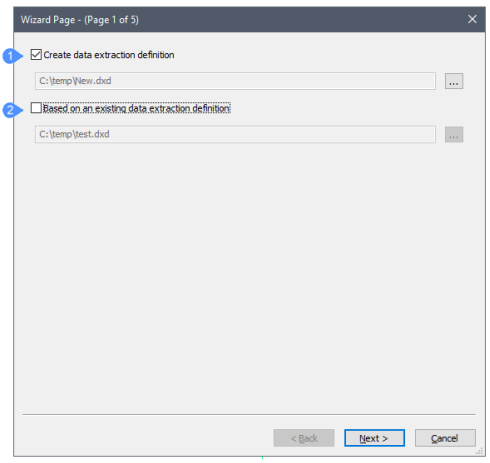
  ***Прим.:*** The section can be edited in Top view as a regular 2D drawing. This is practical for vertical sections.

**Over-write old projection file**
When checked, the existing projection file will be overwritten.

### 4.12.150 Wizard page dialog box

The Wizard Page dialog box guides you to export entity properties, block attributes, and drawing information to a table or to an external data file.

### 4.12.150. Wizard Page 1 Dialog Box 1




1 Create data extraction definition

2 Based on an existing data extraction definition

**Create data extraction definition**

Creates a new .dxd (Data eXtraction Definition) file:

1 Tick the checkbox.

2 Click the Browse button.

3 Select a folder in the Save Data Extraction file dialog box.

4 Enter a name in the File name field.

5 Click the Save button.

**Based on an existing data extraction definition**

Uses an existing .dxd file as the template for the new one to perform the same type of extraction in a different drawing or to edit the data extraction definition:

1 Tick the checkbox.

2 Click the Browse button.

3 In the Open Data Extraction Definition file dialog box, select a .dxd file.

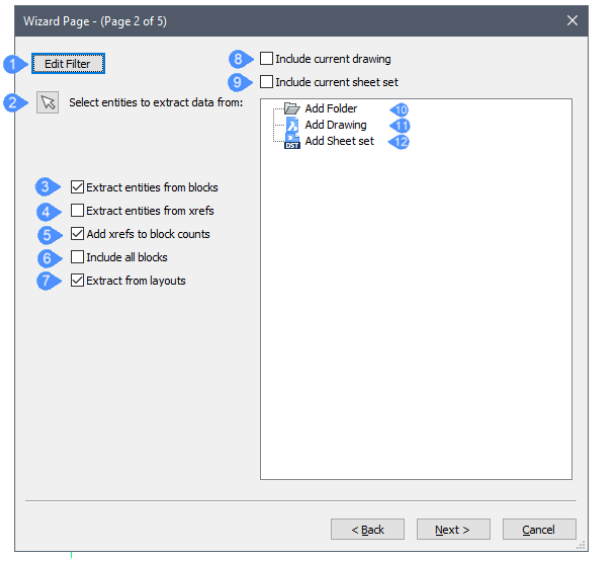
4 Click the Open button.

Newly created custom *.dxd files can be used to create schedules in the Project Browser.

.dxd files can be edited in a text editor such as MS Notepad.

Click **Next** for the next page.

## 4.12.150. Wizard Page 2 Dialog Box 2




1 Edit filter

2 Select entities to extract data from

3 Extract entities from blocks

4 Extract entities from Xrefs

5 Add Xrefs to block counts

6 Include all blocks

7 Extract from layouts

8 Include current drawing

9 Include current sheet set

10 Add Folder

11 Add Drawing

12 Add Sheet Set

Guides to select entities for Data Extraction and apply filters for them before it is extracted in order to have less information to sort later.

The DXEVAL system variable controls when the update notification occurs in the drawings where changes affect the extracted data.

**Edit filter**
Creates or edits a Filter.

**Select entities to extract data from**
Dismisses the dialog box temporarily so that you can select entities in the drawing; enter **Ctrl+A** to select all entities in the drawing. Press **Enter** or right-click to stop selecting entities.

**Extract entities from blocks**
Includes entities nested in blocks.

**Extract entities from Xrefs**
Includes entities in externally referenced drawings (Xref files).

**Add Xrefs to block counts**
Counts selected Xrefs as blocks.

**Include all blocks**
Includes all block entities.

**Extract from layouts**
Includes entities from paper space of the drawing.

**Include current drawing**
Adds the current drawing from which to extract data.

**Include current sheet set**
Adds the current sheet set from which to extract data.

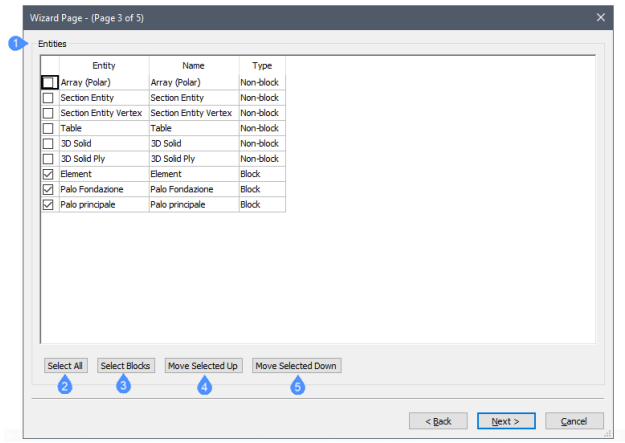
**Add Folder**
Adds a folder of drawing from which to extract data.
Double-click to open the *Folder Options* dialog box.

- **Folder**: click the **Browse** button, then select a folder in *Choose a Folder* dialog box.
- **Options**
  - **Include subfolders**: tick the checkbox to include subfolders.
  - **Use a wildcard to filter drawings to be included**: tick the checkbox, then type a text string in the filter field. For example, House*.dwg includes all files starting with 'House'.

**Add Drawing**
Adds a drawing from which to extract data with a double click.

**Add Sheet Set**
Adds a sheet set from which to extract data with a double click.
Click **Next** for the next page.

### 4.12.150. Wizard Page 3 Dialog Box 3




1 Entities

2 Select All

3 Select Blocks

4 Move Selected Up

5 Move Selected Down

Controls which entity types are included or not in the data extraction procedure.

Select multiple items:

- To select more than one item in a row: select an item, and then hold down the **Shift** key while clicking additional items.
- To select multiple items: hold down the **Ctrl** key while selecting multiple entities.
- To unselect items, click them again while holding down the **Ctrl** key.

To sort the list, click the title in the column heading. Click again to sort in reverse order.

**Entities**

- **Entity**: lists the entity types and blocks in the selection, according to the options set in the *Data Extraction* dialog box.
- **Name**: entity names. Accept the default names or type a customized name. The entity type name displays in the default Name field in the data extraction export.
- **Type**: reports the type of the selected entity: Non-Block, Block or Block with attributes.

**Select All**

Selects all items for data extraction.

**Select Blocks**

Selects only blocks in the list, because they usually contain attributes.

**Move Selected Up**

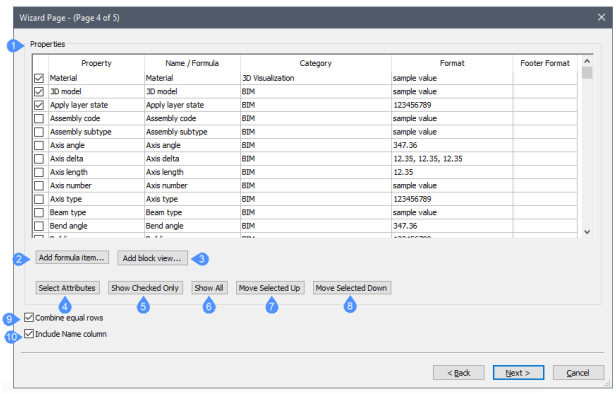
Move the selected item(s) up the list.

**Move Selected Down**

Move the selected item(s) down the list.

Click **Next** for the next page.

### 4.12.150. Wizard Page 4 Dialog Box 4




1 Properties

2 Add formula item

3 Add block view

4 Select Attributes

5 Show checked only

6 Show All

7 Move Selected Up

8 Move Selected Down

9 Combine equal rows

10 Include Name column

Select the properties to be included in the data extraction file: tick the checkbox to check/uncheck a property. Use the same techniques to select more than one item as described in Wizard Page 3 Dialog Box.

To sort the list, click the title in the column heading. Click again to sort in reverse order.

**Properties**

- **Property**: lists all properties of the selected entities.
- **Name /Formula**: lists the name of the selected entities or the formula. Accept the default names or type a customized name. The property names display in the column headings in the data extraction export.
- **Category**: reports the category of the selected entities.
- **Format**: formats the item in the export file. Click to display the dialog box. The layout of the dialog box depends on the selected item, just like the Format section of the Field dialog.
- **Footer Format**: selects from the drop-down list opened with right mouse button the Footer settings:

**Add formula item**

Adds a formula item.

**Add block view**
Adds BlockViewProperty as Block view category.

**Select Attributes**
Select only items that contain attributes.

**Show checked only**
Show only selected items, hiding those that are not selected.

**Show All**
Shows all items, whether checked or not.

**Move Selected Up**
Move the selected item(s) up the list.

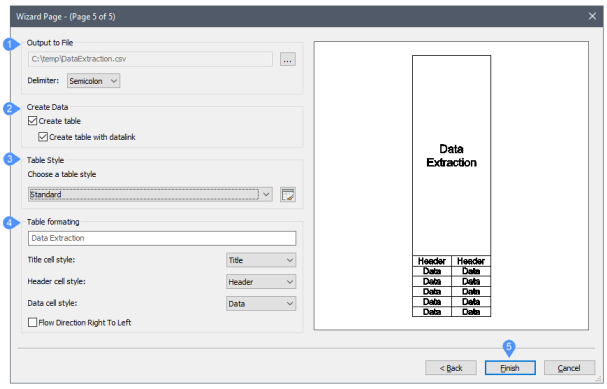
**Move Selected Down**
Move the selected item(s) down the list.

**Combine equal rows**
Combine rows that contain the same entities.

**Include Name column**
Toggles the display of the Name column in the exported data.

Click **Next** for the next page.

### 4.12.150. Wizard Page 5 Dialog Box 5




1 Output to File

2 Create Data

3 Table Style

4 Table formatting

5 Finish

**Output to File**
Exports the data in a delimiter separated format (.csv):
Click the **Browse** button, give the data file a name, and then click **Save**. If you select an existing .csv file, you will be prompted to overwrite it.

**Delimiter**: specifies the separator between fields of data:

- Semicolon
- Comma

  ***Прим.:*** Multiply composition names contain a comma, therefore do not select comma as the delimiter character in case multiply compositions are processed.
- Space
- Tab

**Create Data**
Exports the data in a Table:

- Create table: creates a static table.
- Create table with datalink: creates a dynamic table. Executing the DataLinkUpdate command synchronizes the data in the table with the model. Save the model first. You will be prompted to save the *.dxd definition file with the table.

**Table Style**
Select a table style from the drop-down list.
Click the **Table Style** icon; edit an existing table style or create a new one.

**Table formatting**
Type a title in the Title field.

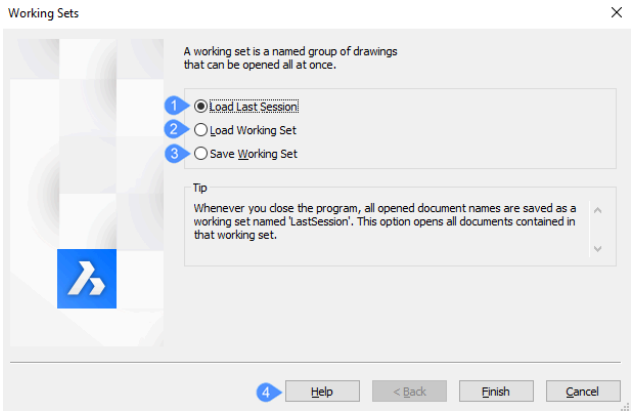
- Title cell style: sets the title cell style.
- Header cell style: sets the Header cell style.
- Data cell style: sets the Data cell style.
- Flow Direction Right to Left: indicates the flow direction from right to left.

**Finish**
Depending on the selected output:

- Extracts the data and saves them to the .csv file. The file can be opened in a spreadsheet or database program.
- Prompts you to specify an insertion point for the table(s).

### 4.12.151 Working sets dialog box

The **Working Sets** dialog box allows you to create and load sets of drawing files, making it convenient to load and entire group of drawings at once.




1 Load Last Session

2 Load Working Set

3 Save Working Set

4 Help

### 4.12.151. Load Last Session

**1**

Opens all drawings that where open when the last BricsCAD session was closed.

After you click the **Finish** button, BricsCAD opens all the drawings that were open the last time you closed the program.

### 4.12.151. Load Working Set

**2**

Click the **Next** button and choose the name of a workset from the **Select a Workset** list. Click the **Finish** button. The program opens all the drawings that are part of the workset.

### 4.12.151. Save Working Set

**3**

Saves drawings currently open by name as a working set. Click the **Next** button.

When you click on the **Save** button, the **Save workset** dialog box opens. Click the **Finish** button to close the dialog box.

***Прим.:*** Working sets are saved in the registry and so they are defined differently for each user profile. See the PROFILEMANAGER command to learn more about user profiles.

***Прим.:*** Delete removes a named working set. Redefine (overwrites) an existing named working set.

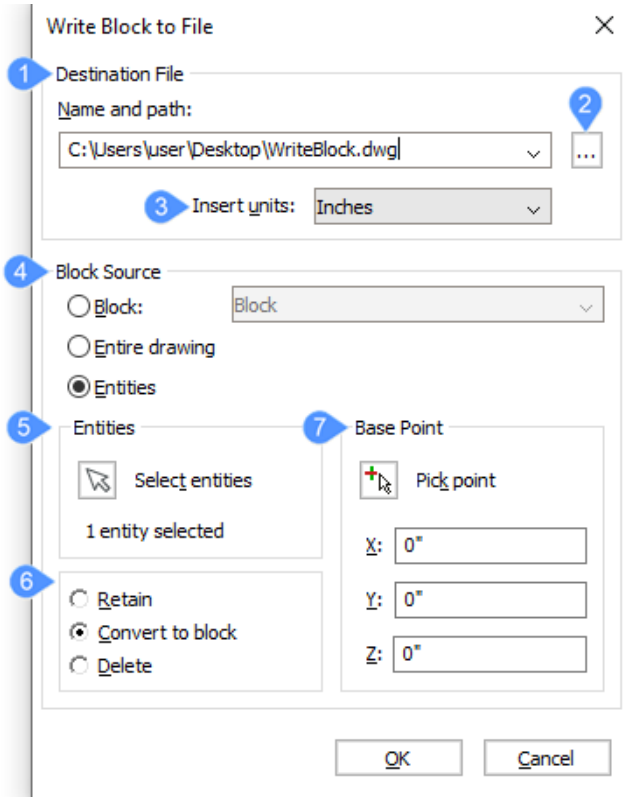
### 4.12.151. Help

**4**

Opens the Bricsys Help article about WORKSETS command.

### 4.12.152 Write block to file dialog box

The **Write Block to File** dialog box allows you to write blocks and other drawing parts to a separate drawing file.




1 Destination file

2 Save Block

3 Insert units

4 Block Source

5 Entities

6 Options

7 Base Point

#### 4.12.152. Destination file

**1**

Specifies what the file should be named and where it should be saved.

- **Name and path**: Specify the default path and file name (new block.dwg).

#### 4.12.152. Save Block

**2**

Changes the path and file name. Opens the **Save Block** dialog box.

### 4.12.152. Insert units
3

Specifies the units to be used by the exported entities when they are later inserted into other drawings.

You can choose the measurement unit from the drop-down list.

Units are saved in the INSUNITS variable.

### 4.12.152. Block Source
4

Selects the part of the drawing to save to a new block:

- **Block**: Saves a block from the drawing. This option is unavailable when no blocks exist in the drawing. When this option is selected, the other options become unavailable. You can choose from the drop-down list the name of a block you want to save.
- **Entire drawing**: Saves the entire drawing to the file. When this option is selected, all other options become unavailable.
- **Entities**: Saves entities that you select to the file.

### 4.12.152. Entities
5

When the **Block Source** is set to **Entities**, you are prompted at the Command line to choose one or more entities using any selection method.

### 4.12.152. Options
6

Specifies what to do with the selected entities after you click **OK** to close the dialog box:

- **Retain**: Keeps the selected entities as they are; this is like copying the selection to a new drawing.
- **Convert to block**: Converts the selected entities to a block in the current drawing.
- **Delete**: Deletes the entities you had selected in the drawing; this is like moving the selection to a new drawing.

  ***Прим.:*** If the **Delete** option was set, the OOPS command brings back the erased block entities.

### 4.12.152. Base Point
7

Specifies the origin point of the saved entities in the new file. You can pick a point in the drawing or specify the X, Y, Z coordinates.

# 5. Управление чертежами

## 5.1 Creating a drawing

A new drawing is always created as a copy of a template drawing. The default template is saved through the BASEFILE system variable.

In BricsCAD you can create new drawings:

- In the **Start** tab.
- Using the QNEW command.
- Using the NEW command.
- Using the NEWWIZ command.
- Double-clicking a *.dwt file in a Windows Explorer window.

### 5.1.1 Specifying the template folder

1 Open the **Settings** dialog box.

2 Under **Program Options** > **Files** > **Templates**, select **Template Path** (TEMPLATEPATH system variable).

3 Click the **Browse** button.

4 On the **Browse for folder** dialog box, select a folder, then click the **OK** button.

### 5.1.2 Setting the default template drawing

1 Open the **Settings** dialog box.

2 Under **Program Options** > **Files** > **Templates**, select the BASEFILE system variable.

3 Click the **Browse** button.

4 On the **Choose a File** dialog box, select a drawing file (*.dwg) or drawing template (*.dwt), then click the **Open** button.

### 5.1.3 Using the Start tab

1 Click the **Start** tab.

Launch the GOTOSTART command to open the **Start** tab if necessary.

The name of the current drawing template is displayed beneath the **New Drawing** button.

2 Do the following:

- (option) Click the template list* button, to choose a different template drawing.
- Click the **New Drawing** button.

* All template drawings in the **Template** folder matching the current workspace (Drafting, Modeling, Mechanical or BIM), are listed here. Choose **More...** to see the full content of the **Template** folder.

### 5.1.4 Creating a drawing using the default template drawing

**First method**:

Click the + button at the right hand side of the document tabs.

**Second method**:

Launch the QNEW command.

**Third method**:

1 Launch the NEWWIZ command.

2 On the **Create New Drawing** dialog, choose **Start from Default template**, then click the **Next** button.

***Прим.:*** If the default template drawing is not specified, the **Create New Drawing** wizard is launched depending on the current value of the STARTUP system variable, otherwise the **Select Template** dialog box appears.

### 5.1.5 Creating a drawing using the New command

1 Launch the NEW command.

2 On the **Create New Drawing** dialog box, choose **Start from Scratch**, then click the **Next** button.

3 Choose between either **Imperial** or **Metric** default settings.

Click the **Finish** button to create the drawing.

***Прим.:*** Using the above procedure, it depends on the status of the BASEFILE and PSTYLEPOLICY variables whether **CTB** (Color table) or **STB** (Named plot style table) will be used to set up plotter configurations in the newly created drawing.

| BASEFILE | PSTYLEPOLICY | CTB or STB? |
|---|---|---|
| not specified | OFF | STB |
| not specified | ON | CTB |
| specified | ON or OFF | as defined in the template drawing |

### 5.1.6 Setting the Plot style policy

1 Open the **Settings** dialog box.

2 Under **Program Options** > **Plot and publish**, select **Plot style policy** (PSTYLEPOLICY system variable).

3 Do one of the following:

- Choose **Associate an entity's plot style with its color** for Color dependent (CTB) plot styles.
- Choose **No association between color and plot style** for Named (STB) plot styles.

### 5.1.7 Creating a drawing using a template drawing

1 Launch the NEWWIZ command.

2 On the **Create New Drawing** dialog box choose **Start from template**, then click the **Next** button.

3 Do one of the following:

- Choose a template drawing in the **Select a template list**.

  By default this is the content of the **Template** folder.

- Click the **Browse...** button to select a template drawing.

  You can choose between ***.dwg** and ***.dwt** file types.

### 5.1.8 Creating a drawing using a wizard

1 Launch the NEWWIZ command.

2 On the **Create New Drawing** dialog box, choose **Use a Wizard**, then click the **Next** button.

3 Choose between either **Imperial** or **Metric** default settings, then click the **Next** button.

4 Select the **Unit of measurement** and the display **Precision**, then click the **Next** button.

5 Select the **Angle of measurement** and the display **Precision**, then click the **Next** button.

6 Choose between **Color Dependent** plot styles (CTB) or **Named** plot styles (STB), then click the **Next** button.

7 Select the default **Color** and **Linetype**.

8 Define the **Grid**, **Snap**, **UCS icon** and **Marker Blips** settings.

9 Click the **Finish** button to create the drawing.

(option) Choose **Save As..** in the **File** menu to save the drawing as a new template in the **Templates** folder.

## 5.2 Working in the Drawing Explorer dialog box

### 5.2.1 Adding a drawing folder

1 In the **Drawings** > **Folders** sub-window of the **Drawing Explorer** main window, click the **Add local folder...** button.

The **Browse For Folder** dialog box appears.

2 Do one of the following:

- To add an existing folder: select the folder you want to add.
- To create a new folder: click the **Make New folder** button.

3 Click the **OK** button.

The folder is added.

***Прим.:***

- Drawings in the subfolders of the selected folder can be opened too.
- The parent folder(s) of the selected folder are greyed out in the drawing folder tree. Drawings in such folders cannot be opened.

### 5.2.2 Opening a drawing

To open a drawing in one of your favorite folders:

1 In the **Drawings** > **Folders** sub-window, open the folder of the drawing.

2 Select a drawing.

3 A preview is shown in the **Preview** sub-window.

4 Do one of the following:
   - Double-click the drawing.
   - Right-click and choose **Open** in the context menu.

### 5.2.3 Logging on to Bricsys 24/7

1 In the **Drawings** pane, select **Folders**.

2 Click **Logon to Bricsys 24/7**.

3 Fill out your username and password.

4 When logged in, all of your projects on Bricsys 24/7 are listed.

### 5.2.4 Opening a drawing from a Bricsys 24/7 project

1 Expand the project, then select the drawing folder.

2 Select the drawing.

3 A preview is shown in the **Preview** sub-window.

4 Right-click the drawing, then choose **Download** in the context menu.

A **Save File As** dialog box appears.

5 Select a folder and optionally rename the drawing, then click the **Save** button.

The drawing is downloaded and opened in the current BricsCAD session.

### 5.2.5 Copying definitions between drawings

1 Open both the source and the target drawing.

2 Launch the **Drawing Explorer** dialog box.

3 In the source drawing, select the category you want to copy items from, e.g. **Blocks**.

4 (option) Adjust the view option of the **Details** pane, e.g. **Icon View** button ( ).

5 Select the definition(s) you want to copy, e.g. a series of blocks.

6 Press and hold the Ctrl key to select multiple items.

7 Do one of the following:
   - Press and hold the left mouse button to drag the selection set to the **Open Drawings** section of the Drawing Explorer.

     Release the left mouse button when on the target drawing name.

     The selected items are copied to the target drawing.
   - Right-click and choose **Copy** in the context menu.

     Select the same definition category in the target drawing, then right-click and choose **Paste** in the context menu.

***Прим.:***

- You will be prompted to overwrite items of the same name.
- Drag-and-drop is not supported for some definition categories, e.g. MultiLeader Styles, Visual styles, Render Presets and Page Setups.

## 5.3 Drawing properties

The DWGPROPS command opens the **Drawing Properties** dialog box, showing the general information and user defined properties stored within a drawing.

The **Drawing Properties** dialog box includes:

- General information: drawing data
- Summary: editable default properties
- Statistics: history data
- Sheetset: info about the associated sheetset
- User defined custom properties

Drawing properties can be referenced in fields.

### 5.3.1 Opening the Drawing Properties dialog

Do one of the following:

- Choose **Drawing Properties...** in the **File** menu.
- Execute the DWGPROPS command.

### 5.3.2 Defining default properties

1 On the **Drawing Properties** dialog box, click the **Summary** tab page.

2 Fill out the property fields.

3 Click the **OK** button.

### 5.3.3 Creating custom properties

1 On the **Drawing Properties** dialog box, click the **Custom** tab page.

2 Click the **Add** button.

The **Custom Property** dialog appears.

3 Type a name in the **Name** field.

The custom property name is listed under **Document** in the **Field** dialog.

4 Type a value in the **Value** field.

The value is displayed where the custom property is inserted as a field.

## 5.4 Drawing security options

The SECURITYOPTIONS command allows you to protect your drawings with a password. Password protected drawings cannot be opened without entering the password first.

### 5.4.1 Setting the password

1 Do one of the following:

- Choose **Security Options...** in the **File** | **Drawing Utilities** menu.
- Execute the SECURITYOPTIONS command.

The **Security options** dialog box opens.

2 Type the password in the **Enter Password** field.

3 Choose a crypt provider from the **Crypt Provider** list.

4 Set the **Key Length** to **40**, **48** or **56**.

5 Click the **OK** button or press Enter.

The **Security options** dialog box closes.

The **Password** dialog opens.

6 Type the password in the **Confirm Password** field.

7 Click the **OK** button or press Enter.

The password is defined.

### 5.4.2 Removing the password

1 Do one of the following:

- Choose **Security Options...** in the **File** | **Drawing Utilities** menu.
- Execute the SECURITYOPTIONS command.

2 The **Security options** dialog box opens.

3 Empty the **Enter Password** field.

4 Click the **OK** button or press Enter.

A message box appears to confirm that the password is removed.

5 Click the **OK** button or press Enter to close the message box.

## 5.5 Opening and saving drawings

The way BricsCAD opens and saves drawings is controlled through a series of system variables.

| Name | Title | Description |
|---|---|---|
| | Display full path in title | Controls whether the full path of a drawing displays in the title bar, or the file name only. |
| | Drawings path | Specifies an additional folder in the file dialog for theOPENandSAVEAScommands.<br>On the Windows platform: the fifth folder in the**Open File** dialog's **Places** bar. |
| - | First, Second, Third and Fourth folder | Sets the folders 1 through 4 in the**Places**bar of the non-standard**Open File**dialog (Windows platform only). |
| | Thumbnail preview image size | Sets the size of the thumbnail image in the Preview on file dialogs. |

| Name | Title | Description |
|---|---|---|
| | Index Control | Determines whether layer and/or spatial indexes are created and saved.<br>• The layer index lists all entities in the drawing and what layer they are on.<br>• The spatial index stores the location of all the entities in 3D space.<br>The indexes are used to load Xrefs if demand loading is enabled. If a layer index exists, entities on layers which are Off or Frozen are not loaded. If a spatial index exists, entities in clipped Xrefs which are outside the clipping boundary are not loaded.<br>The demand loading of Xrefs is controlled through the XLOADCTL system variable).<br>Creating indexes slows down the saving of a drawing, but increases the speed of handling Xrefs. Therefore create indexes only for drawings that will be used as an external reference.<br>0 = No indexes. |
| | Xref load control | Controls demand loading of entities in external references.<br>If demand loading is turned on you will experience a performance increase:<br>• When clipping external references which are saved with a spatial index. Entities in the external reference within the clipped area are loaded only. If the clipping area is modified, more or less entities are loaded from the reference drawing.<br>• When freezing layers in external references which are saved with a layer index, entities in the external reference on thawed layers are loaded only. If xref-dependent layers are thawed more entities are loaded. |
| | Multi-Threading flags | Bitcode 1024: Delayed XREF loading.<br>Delays the loading of Xrefs to idle time. |
| | Incremental save backup | Controls the creation of a backup files (BAK). Improves the speed of incremental saves if switched off, especially for large drawings. |

| Name | Title | Description |
|---|---|---|
| | Save percent | This system variable controls how often BricsCAD performs a full save. A full save reorganizes all the data in the drawing database and creates a compact file. A partial save stores all changes at the end of the drawing data base. A full save takes longer to be executed than a partial save. |
| | Recent File List | Specifies the number of files shown in the**Recent Files**flyout of the**File**menu and on the **Start** tab. |
| | Proxy graphics | Determines whether images of proxy entities(1) are saved in the drawing. If switched off, a bounding box displays instead. |
| | Proxy notice | Displays a notice when you open a drawing containing proxy entities (1) created by an application that is not present. |
| | Proxy show | Controls how proxy entities (1) display in a drawing. |
| | Proxy web search | Determines whether BricsCAD checks for object enablers. |
| | Save format | Sets the default dwg version to save a drawing. |
| | Save file path | Specifies the path and folder where automatic saves (.SV$) and temporary files are stored. TheSAVEFILEFOLDERcommand opens the SAVEFILEPATH location in the system file browser. |
| | Save time interval | Sets the interval in minutes for automatic saves. If set to zero, automatic saves are turned off. Automatic saves are created with a.SV$extension in the folder defined by the SAVEFILEPATH system variable. |
| | Enable macros | Defines whether macros are enabled when loading a VBA-project. |

| Name | Title | Description |
|---|---|---|
| | Ignore all but first bit of DBMOD(2) for autosave | If this user preference is switched on, no auto-save file will be created for files that are viewed only. |
| | Ignore all but first bit of DBMOD(2) for close | If this user preference is switched on, you can view a file, then close it without getting a request to save the file. |

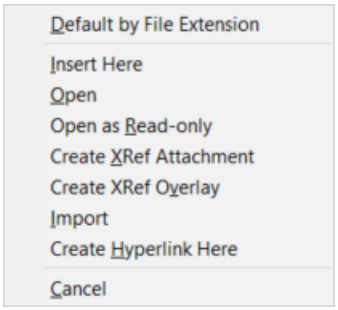
(1) Proxy entities are custom entities which are created by an add-on application. If the add-on is currently not loaded or not installed, such entities cannot be displayed properly.

(2) DBMOD: Stores the status of the drawing modification as a bit-code as the sum of the following values: 1 = Object database modified, 4 = Database variable modified, 8 = Window modified, 16 = View modified, 32 = Field modified.

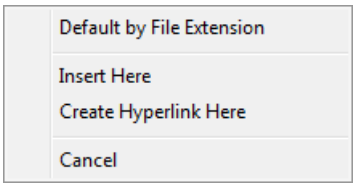
### 5.5.1 Using drag-and-drop to open a drawing

You can open a drawing by dragging it from a File dialog box onto the BricsCAD application window. Select the file, then hold down the left mouse button and move the cursor to the BricsCAD application window and release the left mouse button to load the drawing. When you drag a file holding down the right mouse button, a context menu displays:

When a drawing file is selected:




When another file type is selected:




Choose**Default by File Extension**to:

- Open a drawing file.
- Attach a PDF.
- Attach a raster image file.

- Insert a file as an OLE object (e.g. an Excel spreadsheet).

Choose**Insert Here**to:

- Insert a drawing file as a block in the current drawing.
- Attach a PDF.
- Attach a raster image file.
- Insert the file as an OLE object.

Choose**Open**to open a drawing file .

Choose **Open as Read-only** to open a drawing file and preserve the original from being overwritten. You can only save such drawing under a different name.

Choose**Create Xref**or **Create Xref Overlay** to attach a drawing file as an external reference to the current drawing.

- **Attachment**: if the current drawing is attached as an xref to another drawing, an attachment becomes a nested xref.
- **Overlay**: an overlay is ignored when the drawing to which it is attached is then attached as an xref to another drawing.

Choose **Import** to import the geometry in the current drawing.

Choose**Create Hyperlink Here**to create an hyperlink to the selected file.

***Прим.:***

- Drawing file extensions are: .DWG, .DXF or .DWT
- When multiple files are selected, the files are processed one after the other.

## 5.6 Getting drawing information

You can retrieve two types of information from a drawing:

- General information about the drawing.
- Information about the entities in the drawing.

### 5.6.1 General drawing information

The DWGPROPS command opens the **Drawing Properties** dialog box, showing the general information and user defined properties stored with a drawing.

The SETVAR command displays or changes the values of system variables (short for "set variables").

The STATUS command reports the status of the drawing: drawing name, the status of the most important system variables and the total number of entities in the drawing.

The TIME command reports on time in the drawing: current time, creation time, last update, total editing time and user elapsed time.

### 5.6.2 Information about entities

The STATUS command reports the total number of entities in the drawing.

Using **Quick Select**, you can see the number of entities for each category.

The AREA command finds the area and perimeter (or length) of 2D entities.

The MASSPROP command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for "mass properties").

The ID command reports the X, Y, Z coordinates of picked points.

The LIST command lists the properties of selected entities.

## 5.7 Comparing drawings

The DWGCOMPARE command compares a selected drawing with the current drawing during a comparison session.

The ENDCOMPARE command concludes a comparison session.

The KEEPME command allows you to merge the compared drawings visually.

### 5.7.1 Comparison system variables

| Name | Title | Description |
|---|---|---|
| CMPCLRMISS | Compare missing entities display color | Sets the color for displaying entities which are missing in the other drawing.<br>The default color is Red. |
| CMPCLRMOD1 | Compare modified entities display color 1 | Sets the color for displaying modified entities in the current drawing.<br>The default color is index color 253. |
| CMPCLRMOD2 | Compare modified entities display color 2 | Sets the color for displaying modified entities in the other drawing.<br>The default color is Yellow. |
| CMPCLRNEW | Compare new entities display color | Sets the color for displaying entities which are new in the other drawing.<br>The default color is Green. |
| CMPDIFFLIMIT | Maximal number of output entities | Sets the limit for entities comparison. The comparison will be aborted after reaching the difference limit.<br>Values between 1 and 10 000 000 are accepted. |

### 5.7.2 Executing a comparison session

1 Launch the DWGCOMPARE command.

1 You are prompted: CMPDIFFLIMIT = 10000000. Select the file to compare with or [Limit]:

2 (option) Choose the **Limit** option to set a new value for entities comparison.

3 Press Enter to display a **File** dialog box.

4 Select the drawing to be compared with the current drawing, then click the **Open** button or double-click the drawing name.

The differences between the two drawings are displayed using the **Compare Display Colors**.

The **Compare** toolbar appears.

5 To check the **Diff Type** of an entity, do one of the following:
- If **Rollover Tips** (RT) are on, hover over the entity.
- Select the entity and check the **Diff type** field in the **Properties** panel.

6 Click the **Endcompare** tool button on the **Compare** toolbar to conclude the comparison session.

### 5.7.3 Analyzing the comparison in the Structure panel

1 Load the **Default.cst** configuration in the **Structure** panel.

2 Expand the **Comparison** node.

Four subnodes are shown: **Missing**, **Modified (Original)**, **New** and **Modified (Compared)**.

The number of entities in each subnode is indicated between brackets.

3 Do the following:
- Click a subnode to select the corresponding entities in the drawing.
- Expand a subnode, then click an entity to highlight the corresponding entity in the drawing.
- Press and hold the Ctrl key to select multiple entities.
- Right-click, then choose **Zoom** in the context menu to zoom onto the selected entity or entities.

***Прим.:*** When selecting entities in the **Structure** panel, entities are either selected or highlighted in the drawing, or tree selection is ignored. (See the **Structure** panel).

### 5.7.4 Merging the compared drawings

1 **Missing entities** only exist in the current drawing.
- Select the entities you want to keep, then click the **Keep Me** tool button on the **Compare** toolbar.
- Select the entities you don't want to keep, then click the **Delete** tool button.

2 **Modified entities** exist in both drawings.
- In the **Modified (Original)** node: select the entities you want to keep, then click the **Keep Me** tool button on the **Compare** toolbar.

  The corresponding entities in the **Modified (Compared)** node are removed.
- In the **Modified (Compared)** node: select the entities you want to keep, then click the **Keep Me** tool button on the **Compare** toolbar.

3 **New entities** exist in the compared drawing only.
- Select the entities you want to keep, then click the **Keep Me** tool button on the **Compare** toolbar.
- Select the entities you don't want to keep, then click the **Delete** tool button.

When the merge is complete, the **Comparison** node in the **Structure** panel disappears. A message appears in the Command line: No more differences to show. Session is closed.

## 5.8 Repairing a drawing

TheAUDIT command analyzes the integrity of the current drawing and optionally tries to fix errors.

TheRECOVER command repairs a damaged drawing, then opens the drawing.

### 5.8.1 Analyzing the integrity of the current drawing

1 Launch the AUDIT command.

Prompts you:Fix any errors detected? Yes/<No>

2 Do one of the following:

- Choose**Yes**in the context menu or type**Y**, then press Enter to analyze the integrity of the current drawing and fix the errors detected.
- Choose**No**in the context menu or type**N**, then press Enter to analyze the integrity of the current drawing.

The drawing is analyzed.

3 (option) Press F2 to open the**Prompt History**window in order to read the audit report:

- # objects audited
- Total errors found during audit #, fixed #.

***Прим.:***

- Set theAUDITCTLsystem variable to 1, to letAUDIT create an ASCII file describing the detected problems and the action taken. This report, with the .adt file extension, is placed in the same directory as the current drawing .
- Use theRECOVER command to repair the errors thatAUDIT cannot fix.

### 5.8.2 Repairing a drawing

1 Do one of the following:

- Choose**Recover**in the**File** > **Drawing Utilities**menu.
- Launch the RECOVER command.

The**Open Drawing**dialog window displays.

2 Select the drawing, then press the**Open**button on the**Open Drawing**dialog window.

The drawing is processed and - if the repair succeeds - opened.

3 (option) Press F2 to open the**Prompt History**window in order to read the recover report.

## 5.9 Projects

Projects are especially useful to manage Xrefs and images when exchanging drawings. If external references (Xrefs) and images are not found in the saved path, the project search paths are used to find the external references and images.

Projects are saved in the registry and contain one or more search paths.

### 5.9.1 Accessing the Project settings

1 Launch the SETTINGS command.

1 The **Settings** dialog box opens.

2 Go to **Program Options** > **Files** > **Projects**.

### 5.9.2 Creating projects

1 Edit the PROJECTSEARCHPATHS system variable.

2 Click the **Browse** button at the right hand side of the **Project search paths** settings field.

The **Project Settings** dialog box appears.

3 To create the first project:

- Click the **Add Project or Path** button on the **Project Settings** dialog.

  A new project is added.

- Type a name to replace the **NewProject1** default name.

4 To create additional projects:

- Click **Projects** (on top of the projects tree) or select an existing project.
- Click the **Add Project or Path** button.

  A new project is added.

- Type a name to replace the **NewProject1** default name.

### 5.9.3 Adding search paths to a project

1 Edit the PROJECTSEARCHPATHS system variable.

2 Select the project in the **Projects** tree.

3 Click the **Add Project or Path** button.

A path is added to the project.

4 Type a name for the new path.

5 Click the **Browse** button.

The **Choose a Path** dialog box appears.

6 Browse to a folder, then click the **Select Folder** button.

7 Repeat the previous steps to add more search paths to a project.

### 5.9.4 Changing the order of the search paths

1 Edit the PROJECTSEARCHPATHS system variable.

2 Expand the project in the **Projects** tree.

3 Select a search path, then click the up or down buttons to change the order of the search paths.

### 5.9.5 Assigning a project to the current drawing

1 Edit the PROJECTNAME system variable.

2 Type a project name in the settings field.

# 6. Точность построений

## 6.1 Orthogonal mode

When creating or manipulating entities, Orthogonal mode restricts the movement of the cursor to be parallel to the X-axis or the Y-axis of the currentcoordinate system.

Lines are restricted to 0 degrees, 90 degrees, 180 degrees, or 270 degrees. When drawing, the rubber-banding line follows either the horizontal or vertical axis, depending on which axis is nearest to the cursor.

***Прим.:***

- When you enable the Isometric snap style, cursor movement is restricted to orthogonal equivalents within the current isometric plane.
- You can rotate the Orthogonal Mode axes using the SNAPANG system variable.
- Holding down the Shift key reverses the Orthogonal Mode setting.
- Orthogonal Mode is overruled by entity snaps.
- Turning on Ortho Mode automatically disables Polar Tracking and vice versa.

### 6.1.1 Toggling Orthogonal mode

Do one of the following:

- Click theORTHOfield in theStatus Bar.
- Press theF8function key.
- Hold down the Shift key.
- Launch the ORTHO command.

## 6.2 Unit settings

The UNITS command sets units of linear and angular measurements through the **Settings** dialog box. The -UNITS command sets units of linear and angular measurements in the Command line.

The INSUNITS (Insertion Units) system variable defines a drawing units value for automatic scaling when inserting blocks or attaching Xrefs.

Any length, area or volume property can be displayed using the unit(s) preferred by the user.

The unit defined by INSUNITS is used as the base unit for any requested unit conversions. If INSUNITS = 0 (undefined), the PROPUNITS mechanism is disabled because the base drawing unit is undefined.

***Прим.:***

- When LUNITS = 3 (Engineering) or 4 (Architectural), property values will be formatted in imperial notation.
- Automatic unit conversion during length input can be used when bitcode 1 of PROPUNITS (Format length properties) is set. E.g. when INSUNITS is set to "mm", entering "2m" in a length property field will result in entering [2000.0000 mm]. Entering "1ft" will give [304.8000 mm].

### 6.2.1 System variables

| Name | Title | Description |
|---|---|---|
| INSUNITS | Insertion Units | Defines the value of 1 Drawing Unit (DU), which is used for automatic scaling when inserting or attaching blocks, images or Xrefs.<br>When both INSUNITS and PROPUNITS are ON; **Length**, **Area**, **Volume** and/or **Inertia** properties will be formatted with a dedicated unit. |
| MEASUREMENT | Measurement | Sets drawing units as **Imperial** or **Metric**. The MEASUREMENT system variable controls whether ANSI or ISO hatch pattern and linetype files are used. |
| PROPUNITS | Property Units | Defines which property values are automatically formatted with a dedicated unit when INSUNITS is not zero. The value is stored as a bitcode, using the sum of the values of all selected options. |
| LUNITS | Linear Units Type | Sets the linear unit type: **Scientific**, **Decimal**, **Engineering**, **Architectural** or **Fractional**. |
| LUPREC | Linear Unit Precision | Sets the number of decimal places displayed for linear units. |
| AREAPREC | Area Precision | Sets the number of decimal places displayed for areas when the **Area** bit of PROPUNITS is ON.<br>If set negative, the value of LUPREC is used. |
| VOLUMEPREC | Volume Precision | Sets the number of decimal places displayed for volumes when the **Volume** bit of PROPUNITS is ON.<br>If set negative, the value of LUPREC is used. |
| LENGTHUNITS | Length Units | Defines a list of units used for displaying lengths when the **Length** bit of PROPUNITS is ON.<br>If empty, all lengths are displayed in the current drawing unit, as defined by INSUNITS.<br>If multiple units are selected, the unit that fits best is used. |
| AREAUNITS | Area Units | Defines a list of units used for displaying areas when the **Area** bit of PROPUNITS is ON.<br>If empty, all lengths are displayed in the current drawing unit, as defined by INSUNITS.<br>If multiple units are selected, the unit that fits best is used. |

| Name | Title | Description |
|---|---|---|
| VOLUMEUNITS | Volume Units | Defines a list of units used for displaying volumes when the **Volume** bit of PROPUNITS is ON.<br>If empty, all lengths are displayed in the current drawing unit, as defined by INSUNITS.<br>If multiple units are selected, the unit that fits best is used. |
| UNITMODE | Unit Mode | Specifies how Imperial units are displayed. |

### 6.2.2 Entering angles

The AUNITS system variable defines the angular unit type.

The ANGDIR system variable sets the positive angle direction, counter clockwise (default) or clockwise. Angles are measured with respect to direction of the positive X-axis of the current coordinate system.

You can enter angles in any of 5 formats:

- Decimal degrees : 37.456 (AUNITS = 0)
- Degrees / Minutes / Seconds : 37°27'22" or 37d27'22" (AUNITS = 1)
- Gradians : 41.6178g (AUNITS = 2)
- Radians : 0.6537r (AUNITS = 3)
- Surveyor's Units : N52°32'38"E or N52d32'38"E (AUNITS = 4)
- Surveyor's units are measured from the North (N) or the South (S) to the East (E) or to the West (W).

## 6.3 Coordinate reference system

A Coordinate System is a set of mathematical rules for specifying how coordinates are to be assigned to points.

The coordinate system is unrelated to the Earth. A Coordinate Reference System (CRS) is a coordinate system related to the Earth through a Datum. A Geodetic Datum is a model of the Earth that specifies a reference surface (Ellipsoid or Spheroid).

A projected coordinate reference system is the result of the application of a map projection to a geographic coordinate reference system. A map projection is a type of coordinate conversion. It uses an identified method with specific formulas and a set of parameters specific to that coordinate conversion method.

Coordinates may be changed from one CRS to another through the coordinate operation. Two types of coordinate operation may be distinguished:

- **coordinate conversion**, where no change of datum is involved and the parameters are chosen and thus error free.
- **coordinate transformation**, where the target CRS is based on a different datum to the source CRS. Transformation parameters are empirically determined and thus subject to measurement errors. (Geodetic, Seven parameters, Molodenksy...)

Geographic datasets are defined in **geodatabase.xml**that is placed in *<User folder>\AppData\Roaming \Bricsys\BricsCAD\<RELEASE>\en_US\Support for Windows*. The content of this file is composed of the 5 sections:

1 CoordinateReferenceSystems

2 Datums

3 Ellipsoids

4 Transformations

5 ProjectionCode and Methods

### 6.3.1 CoordinateReferenceSystems

This section is the main XML node where all Coordinate Reference Systems are defined. It has many sub-nodes called**CRS**, each XML node must define a unique projected coordinate reference system and the necessary projected parameters.

The supported CRS projection types are described in the**ProjectionCode**and**Methods**sections.

**CRS**node attributes:

Follow Table 1 of OGP Geomatics Guidance Note No 7, part 2

| Attribute name | Description | Units |
|---|---|---|
| epsg | Unique EPSG database id. | Integer |
| name | Human-readable name of CRS, Example: "WGS 84 -- WGS84 - World Geodetic System 1984, used in GPS". | String |
| codeSpace | Owner of CRS. It is option. | String |
| x,y | Orientation of projected direction. Possible values:<br>• "Westing"<br>• "Easting"<br>• "Southing"<br>• "Northing"<br>Directions MUST be defined. | String |
| xAxis, yAxis | Orientation of geographic direction. Possible values:<br>• "Latitude"<br>• "Longitude" | String |
| Greenwich | Greenwich Meridian relation. | Degree |

| Attribute name | Description | Units |
|---|---|---|
| proj | Projection method of CRS. Possible values:<br>• "LL", "LatLon", "LonLat" is a Geodetic Latitude-Longitude.<br>• "MercA", "Mercator_1SP" is a Mercator (variant A).<br>• "MercB", "Mercator_2SP" is a Mercator (variant B).<br>• "MercC" is Mercator (variant C).<br>• "MercSp" is a Mercator (Spherical).<br>• "MercPv" is a Popular Visualisation Pseudo Mercator.<br>• "TMerc" is a Transverse Mercator.<br>• "TMercSO" is a Transverse Mercator South Orientated.<br>• "LCC_1SP" is a Lambert Conic Conformal 1SP.<br>• "LCC_2SP" is a Lambert Conic Conformal 2SP.<br>• "LCC_2SP_BE" is a Lambert Conic Conformal 2SP Belgium. | String |
| AngleSG | Angle from Rectified to Skew Grid. | Degree |
| Azimuth | Azimuth of initial line. | String |
| FE | Easting at false origin, False easting. | Float-Point value in CRS units |
| EC | Easting at projection center. | Float-Point value in CRS units |
| FN | Northing at false origin, False northing. | Float-Point value in CRS units |
| FC | Northing at projection center. | Float-Point value in CRS units |
| LonI | The longitude of the western limit of the first zone of a Transverse Mercator zoned grid system. | Degree |
| Lat | Latitude of natural origin, Latitude of standard parallel. It depends on Projection method. | Degree |
| Lat1 | Latitude of 1st standard parallel. | Degree |

| Attribute name | Description | Units |
|---|---|---|
| Lat2 | Latitude of 2nd standard parallel. | Degree |
| LatF | Latitude of false origin. | Degree |
| LatC | Latitude of projection center. | Degree |
| Lon | Longitude of natural origin, Longitude of origin. | Degree |
| LonF | Longitude of false origin. | Degree |
| LonC | Longitude of projection center. | Degree |
| SF | Scale factor at natural origin. | Float-Point value, unity |
| SFIL | Scale factor on initial line. | Float-Point value, unity |
| SFPSP | Scale factor on pseudo standard parallel. | Float-Point value, unity |
| W | Zone width in longitude. | Degree |
| zone | Zoned Grid System. | Degree |
| units | Units of conversion result from geographic coordinates to projected. Example: "Meter", unit "Degree" means no conversion, Geographic Coordinate System. | String |

Example:common attributes for all map projections based on EPSG:31468.

```
<CRS epsg="31468" codeSpace="OGP" name="DHDN / 3-degree Gauss-Kruger zone 4"
 y="Easting" x="Northing" Greenwich="0" proj="TMerc" Lon="12" Lat="0" SF="1"
 FE="4500000" FN="0" zone="4" units="Meter">
```

For each node**CRS**there should be at least one sub-node**Datum**that refers via**epsg**or**alias**attributes to sub-node**Datum**in the**Datums**section. The sub-node**Datum**MUST have the**id**attribute of a unique CRS name. At least one of the epsg code and the alias of Datum must be valid.

| Attribute name | Description | Units |
|---|---|---|
| epsg | Unique EPSG database id. Example: "4326". | Integer |
| alias | Unique Datum name. Example: "WGS 84". | String |

| Attribute name | Description | Units |
|---|---|---|
| id | Unique CRS name: combination of Projected method and Datum. Example: "WORLD-MERCATOR". It related to CS-MAP, AutoCAD names. | String |
| pjcode | Indicate code for the coordinate reference system projection method types. Corresponded to AutoCAD projection codes.<br>For example, 3 - Transverse Mercator, 44 - UTM, 45 - Transverse Mercator of Snyder formulation and etc. | Integer |

Example:there are unique CRS definitions according to Datums insideCRSnode:

```
<Datum epsg="6314" alias="DHDN/3" id="DHDN/3.Gauss3d-4" pjcode="3" />
<Datum epsg="6314" alias="DHDN/2" id="DHDN/2.Gauss3d-4" pjcode="3" />
<Datum epsg="6314" alias="DHDN" id="DHDN.Gauss3d-4" pjcode="3" />
```

### 6.3.2 Datums

This section is a main node where all Datums are stored. Datum is a combination of the Earth model (Ellipsoid or Spheroid) and the transformation method to model WGS84. A**Datum**node specifies common parameters and contains at least one inner sub-nodes**Datum**exactly one sub-node**Ellipsoid**. Each inner sub-node**Datum**should define a unique name within the**id**attribute and may specify transformation parameters to WGS84 model via sub-node**Transformation**that refers via**epsg**or**alias**attributes to a sub-node**Transformation**in the**Transformations**section.

| Attribute name | Description | Units |
|---|---|---|
| alias | Unique id. Example: "DHDN/3". It is related to CS-MAP, AutoCAD names. | String |
| epsg | Unique EPSG database id. Example: "6314". | Integer |
| name | Human-readable name of Datum. Example: "Deutsches Hauptdreiecksnetz". | String |
| codeSpace | Owner of CRS. It is optional. | String |

Example:

```
<Datums>
<Datum epsg="6314" codeSpace="OGP" name="Deutsches Hauptdreiecksnetz ">
<Datum id="DHDN/3">
<Transformation epsg="1777" alias="DHDN/3_to_WGS84" />
</Datum>
```

```
<Datum id="DHDN/2" name="Deprecated - Replaced by DHDN/3">
<Transformation epsg="1777" alias="DHDN/2_to_WGS84" />
</Datum>
<Datum id="DHDN" name="Deprecated - Replaced by DHDN/2">
<Transformation epsg="1673" alias="DHDN_to_WGS84" />
</Datum>
<Ellipsoid epsg="7004" alias="BESSEL" />
</Datum>
â€¦
</Datums>
```

### 6.3.3 Ellipsoids

The Ellipsoids section is the main section where models of the Earth are defined. Each **Ellipsoid** node should define an identification by a unique name (**alias**) and a semi-major axis (**a**) and one of the following attributes at least: flattering (**f**), semi-minor axis (**b**) or eccentricity (**e**).

| Attribute name | Description | Units |
|---|---|---|
| alias | Unique id. Example: "WGS84". It related to CS-MAP, AutoCAD names. | String |
| epsg | Unique EPSG database id. Example: "7030". | Integer |
| name | Human-readable name of Datum, Example: "WGS 84". | String |
| codeSpace | Owner of CRS. It is optional. | String |
| a | Length of the semi-major axis of the ellipsoid, the radius of the equator. | Float-Point value, strongly in Meter |
| b | Length of the semi-minor axis of the ellipsoid, the distance along the ellipsoid axis between equator and pole. | Float-Point value, strongly in Meter |
| f | Flattening | Float-Point value, unity |
| e | Eccentricity | Float-Point value, unity |

Example:

```
<Ellipsoid epsg="7008" alias="CLRK66" name="Clarke 1866, Benoit Ratio"
 a="6378206.4000000004" b="6356583.7999999998" f="294.9786982139"
 e="0.0822718542" />
```

### 6.3.4 Transformations

This section is a main node where all transformation methods are defined. Supported Transformation methods are described in the**ProjectionCode**and**Methods**sections.

| Attribute name | Description | Units |
|---|---|---|
| alias | Unique id. Example: "DHDN_to_WGS84". It related to CS-MAP, AutoCAD names. | String |
| epsg | Unique EPSG database id. Example: "1673". | Integer |
| codeSpace | Owner of Transformation. It is optional. | String |
| source | Source Datum. | String |
| target | Target Datum. | String |
| use | Transformation method. Supported transformations using geocentric methods:<br>• Geocentric translations<br>• Four/Six/Seven Parameter Transformation<br>• Similarity Transformation<br>• Position Vector transformation<br>• Coordinate Frame rotation | String |
| method | Methods of building of rotation matrix, if applicable, "PVT" is Position Vector Transformation, "CFR" is Coordinate Frame Rotation. | String |
| tX | X-axis translation. | Meter |
| tY | Y-axis translation. | Meter |
| tZ | Z-axis translation. | Meter |
| rX | X-axis rotation. | Degree |
| rY | Y-axis rotation. | Degree |
| rZ | Z-axis rotation. | Degree |
| dS | Scale difference. | Unity |

| Attribute name | Description | Units |
|---|---|---|
| xp | Coordinate 1 of evaluation point. | Meter |
| yp | Coordinate 2 of evaluation point. | Meter |
| zp | Coordinate 3 of evaluation point. | Meter |
| dtX | Rate of change of X-axis translation. | Unity |
| dtY | Rate of change of Y-axis translation. | Unity |
| dtZ | Rate of change of Z-axis translation. | Unity |
| drX | Rate of change of X-axis rotation. | Unity |
| drY | Rate of change of Y-axis rotation. | Unity |
| drZ | Rate of change of Z-axis rotation. | Unity |
| ddS | Rate of change of scale difference. | Unity |
| t0 | The reference epoch for time-dependent parameters. | Float point value |
| fallback | Alias of transformation from target datum to WGS84 datum. | String |

Example:

```
<Transformation epsg="1679" alias="Pulkovo42/2_to_WGS84" src="Pulkovo42/2"
 trd="WGS84" use="Param7" method="CFR" tx="-40.595" ty="-18.55" tz="-69.339"
 ds="-4.299" rx="-2.508" ry="-1.832" rz="2.611" accuracy="9" />
```

### 6.3.5 ProjectionCode and Methods

These sections describe the**ProjectionCode**and**Methods**that are mapped to AutoCAD definitions of projections and transformations types. They are used to store the definitions in an XML file of coordinate reference system definition inside the**AcDbGeoData**object that represents a geographic location.

## 6.4 2D constraints

Parametric constraints are used to maintain relationships and control dimensions of 2D geometry.

There are two types of constraints:

- **Geometrical**constraints control relationships between entities.

- **Dimensional**constraints are used to control the dimensions of an entity, such as length, angle, radius or diameter.

A dimensional constraint can use a value or a formula, with other dimension constraints or parameters. Dimensional constraints can be edited in the Parameters and Constraints panel and in the Mechanical Browser panel.

***Прим.:***

- It is recommended to first apply geometric constraints, then dimensional constraints.
- ThePARAMETERCOPYMODEsystem variable controls how constraints and related parameters are processed when constrained entities are copied.
- The PARAMETRIZE2D command automatically creates geometrical and dimensional parameters.

When using constraints a drawing can be in one of three states:

- **Underconstrained**: some entities are constrained. It might be impossible to apply some changes using modification commands and procedures.
- **Fully constrained**: all possible and relevant geometric and dimensional constraints are applied to the design geometry. The drawing can be modified only by changing the values of the dimensional constraints.
- **Overconstrained**: one or more constraints violate or contradicts other constraints.

Parametric constraints allow to:

- Maintain geometric design intent.
- Make multiple versions of a design by applying different values to dimensional constrains.

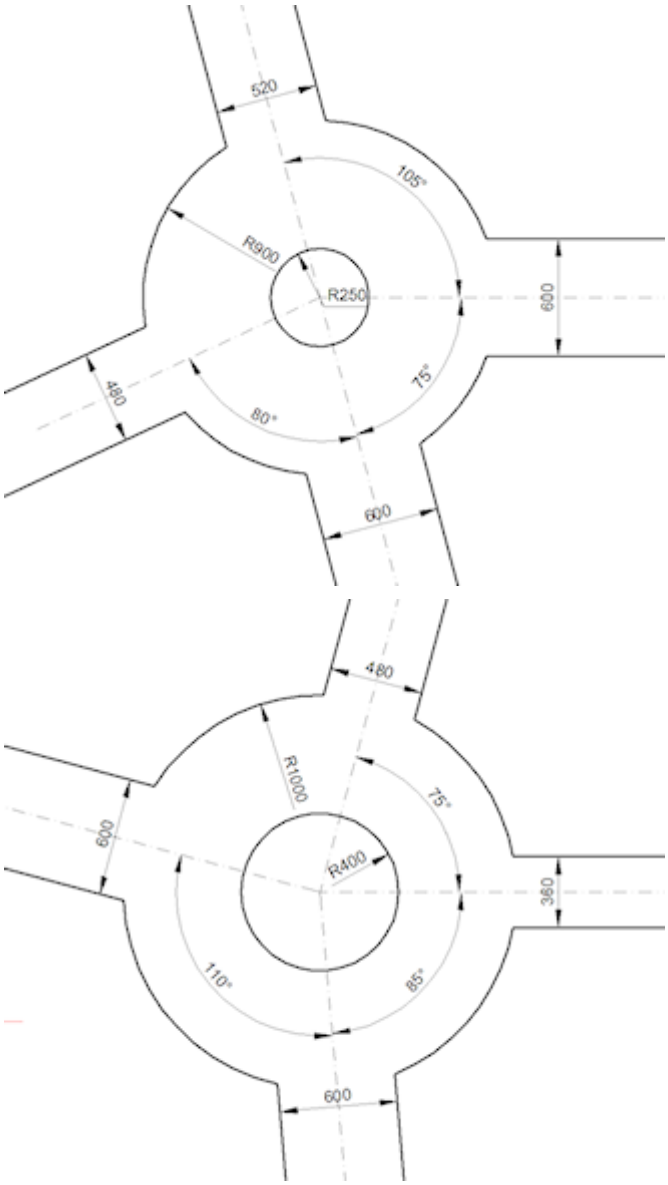



**Рисунок 1:** Two versions of the same fully constrained drawing representing a roundabout

### 6.4.1 Commands and toolbars

2D parametric constraints tools are located on the**2D Constraints**toolbar and the **Parametric** ribbon tab.

Tools to control the display of constraint bars and geometrical constraints are located on two flyout toolbars: **Dimensional Constraint**and **Geometric Constraint**.

| Command | Description |
|---|---|
| | Allows to set dimensional constraints to be displayed or hidden. |

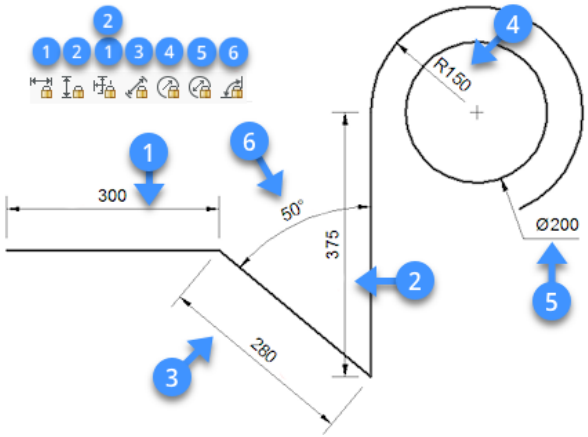
| Command | Description |
|---|---|
| | Applies geometric relationships between entities and/or valid constraint points on entities. |
| | Constrains points and entities at a fixed position. |
| | Applies a coincident geometrical constraint to two points or constraints a point to an entity. |
| | Constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide. |
| | Forces lines or polyline segments to be collinear. |
| | Forces two lines or linear polyline segments to be parallel to each other. |
| | Constrains two lines or polyline segments to lie perpendicularly to each other. |
| | Constrains two entities to maintain a point of tangency to each other or their extensions.<br>The following entities are accepted: lines, polyline segments, circles, arcs, ellipses or elliptical arcs. |
| | Constrains lines, linear polyline segments or pairs of points to lie parallel to the X-axis of the current coordinate system. |
| | Constrains lines, linear polyline segments or pairs of points to lie parallel to the Y-axis of the current coordinate system. |
| | Forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline. |
| | Constrains two entities or points to lie symmetrically with respect to a selected line. |
| | Constrains arcs and circles to the same radius, or lines and polyline segments to the same length. |

| Command | Description |
|---|---|
| | Controls the display of the constraint bar of entities.<br>Constraint bars are always hidden when opening a drawing. |
| | Applies a dimensional constraint to an entity or between constraint points on entities.<br>Converts associative dimensions to dynamic dimensions. |
| | Constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system. |
| | Constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system. |
| | Constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system. |
| | Constrains the distance between two points or the length of a line or polyline segment. |
| | Constrains the radius of a circle, an arc or an arc polyline segment. |
| | Constrains the diameter of a circle, an arc or an arc polyline segment. |
| | Constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities. |
| | Converts an associative dimension to a dimensional constraint. |
| | Removes all dimensional and geometrical constraints from an entity. |

## 6.5 Using dimensional constraints

Dimensional constraints control the dimensions of 2D entities.

- Horizontal distance (1) (DCLINEAR and DCHORIZONTAL commands).
- Vertical distance (2) (DCLINEAR and DCVERTICAL commands).
- Distance between points or the length of lines and polyline segments (3) (DCALIGNED command).
- Radius of arcs, circles and arc polyline segments (4) (DCRADIUS command).
- Diameter of arcs, circles and arc polyline segments (5) (DCDIAMETER command).
- Angle between entities or between points and entities (6) (DCANGULAR command).




***Прим.:*** Dimensional constraints are created using the current dimension style.

### 6.5.1 Editing dimensional constraints

Dimensional constraints can be edited:

- In the Properties panel.
- In theMechanical browser panel.
- In the Parameters and constraints panel.
- Using theDDEDITcommand.

**Editing dimensional constraints in the Properties panel**

1 Select the dimensional constraint.

The properties of the entity display in the Properties panel.

2 Type a new value orexpressionin the**Expression**field, then press Enter.

The readout of the**Value**field changes accordingly.

3 (option) Edit the**Name**field.

4 (option) Edit the**Description**field.

5 (option) Edit the**Text rotation**field.

6 Press the Esc key to unselect.

***Прим.:*** Select multiple dimensional constraints simultaneously to set their value equally.

**Editing dimensional constraints in the Mechanical browser panel**

1 If not already open, open theMechanical Browser panel.

2 Select a name of the dimensional constraint in the Parameters list.

The properties the selected parameter display in bottom part of theMechanical browser panel.

3 (option) Select the**Name**field, then type a new name.

4 (option) Select the**Expression**field, then type a new value or expression.

5 (option) Select the**Description**field, then type a new description.

6 (option) Set the**Exposed**field, which specifies whether the parameter is exposed when the drawing is inserted in another drawing usingthe BMINSERT command.

The options are:

- **Auto** : the parameter is exposed only if it does not depend on other parameters.
- **ON** = the parameter is always exposed.
- **OFF** = the parameter is never exposed.

7 (option) Set the**Units**property: linear, planar or volume.

Select the field, then click the down-arrow button and choose an option.

**Using the DDEDIT command**

1 Do one of the following:

- Double click the dimensional constraint.
- Launch the DDEDIT command.

You are prompted: _DDEDIT

The **Edit Text** dialog box displays.

2 Type a new value in the**Text**field, then press Enter or click the**OK** button.

3 (option) Click another dimensional constraint and repeat the previous step.

4 Do one of the following:

- Keep editing more dimensional constraints.
- Press Enter or right click to stop.

All edited dimensional constraints are updated simultaneously.

### 6.5.2 Setting the visibility of dimensional constraints

Dimensional constraints are hidden when a drawing is closed and reopened.

Tools to control the display of dimensional constraints are located on:

- The **Dimensional Constraint** toolbar.
- the **Parametric ribbon** tab.
- the **Show/Hide 2D Constraints** fly-out menu of the **Parametric** menu.

These tools execute the DCDISPLAY command using a specific option.

The options are:

- Show/Hide: prompts you to select the entities.
- Show All: show all dimensional constraints in the drawing.
- Hide All: hide all dimensional constraints in the drawing.

***Прим.:***

- TheDYNCONSTRAINTMODEsystem variable controls the visibility of hidden dimensional constraints when selecting entities:
  - ON: Display dimensional constraints when entities are selected.
  - OFF: Dimensional constraints remain hidden when entities are selected.
- When hovering over an entity with a dimensional constraint, the blue constraint glyph ( ) displays if theSELECTIONPREVIEWsystem variable is set to 1 or 3 or when the Quad is active.

### 6.5.3 Using expressions to define dimensional constraints

Dimensional constraints are named automatically upon creation.

The naming rules are as follows:

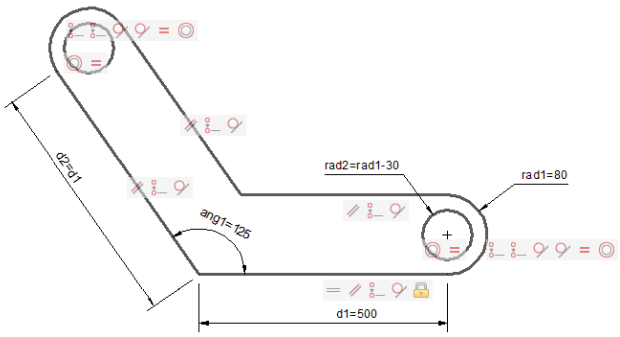
- distance: d1, d2, d3, ...
- radius: rad1, rad2, rad3, ...
- diameter: dia1, dia2, dia3, ..
- angular: ang1, ang2, ang3, ..

The name of a constraint can be used as a parameter in a mathematical expression to define another dimensional constraint.

The dimensions of the bracket below are controlled through the**rad1**radius,**d1**distance and**ang1**angular dimensional constraints.

The radius of the hole (**rad2**) and the length of the second leg (**d2**) are defined by expressions.

Geometrical constraints force the width of the second leg and the hole in it to be equal to the first leg.




**Using operators in expressions**

The following operators can be used in expressions:

| Operator | Description |
|---|---|
| + | Addition |
| - | Subtraction or Negative |
| * | Multiplication |
| / | Division |
| ^ | Exponentiation |
| % | Modulo or Remainder operator<br>The expression "5 % 2" would evaluate to 1, because 5 divided by 2 leaves a quotient of 2 and a remainder of 1. |

Expressions are evaluated according to the standard mathematical rules of precedence:

1 Expressions within brackets; innermost sets first.

2 Standard operations order:
   a exponent
   b multiplication and division
   c addition and subtraction

3 Operators of equal precedence from left to right.

**Using functions in expressions**

The following functions can be used in expressions:

| Function | Syntax |
|---|---|
| Cosine | cos(expression) |
| Sine | sin(expression) |
| Tangent | tan(expression) |
| Arc cosine | acos(expression) |
| Arc sine | asin(expression) |
| Arc tangent | atan(expression) |
| Hyperbolic cosine | cosh(expression) |

| Function | Syntax |
|---|---|
| Hyperbolic sine | sinh(expression) |
| Hyperbolic tangent | tanh(expression) |
| Arc hyperbolic cosine | acosh(expression) |
| Arc hyperbolic sine | asinh(expression) |
| Arc hyperbolic tangent | atanh(expression) |
| Square root | sqrt(expression) |
| Signum function (-1,0,1) | sign(expression) |
| Round to nearest integer | round(expression) |
| Truncate decimal | trunc(expression) |
| Round down | floor(expression) |
| Round up | ceil(expression) |
| Absolute value | abs(expression) |
| Largest element in array | max(expression1;expression2)* |
| Smallest element in array | min(expression1;expression2)* |
| Degrees to radians | d2r(expression) |
| Radians to degrees | r2d(expression) |
| Logarithm, base e | ln(expression) |
| Logarithm, base | 10 log(expression) |
| Exponent, base e | exp(expression) |
| Exponent, base 10 | exp10(expression) |

| Function | Syntax |
|---|---|
| Power function | pow(expression1;expression2) 1 |
| Random decimal, 0-1 | Random |

* Use the list separator character as defined on your system: , (comma) or ; (semicolon).

**Creating a new parameter**

1 Right click **Parameters** in the Mechanical Browser panel.

A context menu displays.

2 Select **Add new parameter** in the context menu.

A new parameter is created.

3 Edit the properties of the new parameter.

See the **Editing dimensional constraints in the Mechanical browser panel** procedure above.

***Прим.:***

- The constants **Pi=3.14...** and **e=2.72...** can be used in expressions. The constants names are not allowed to be used as a parameter or constraint name.
- The CLEANUNUSEDVARIABLES command purges parameters that are not used in constraint expressions or linked to dimensions.

## 6.6 Using geometric constraints

Geometric constraints:

- Specify a geometric relation between 2 entities (coincident, concentric, collinear, parallel, perpendicular, tangent, smooth, symmetric, equal).
- Specify a fixed angle (horizontal, vertical).
- Specify a fixed location (fix).

When a geometric constrained is applied to an entity:

- The position of the entity is adjusted according to the applied constraint.
- An icon displays next to the entity, to indicate the applied constraint. If multiple geometric constraints are applied, the icons are joined in a constraint bar.

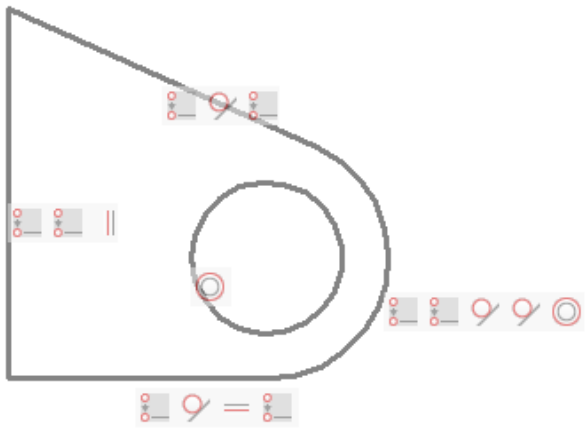
### 6.6.1 Creating geometric constraints

Tools to create dimensional constraints are located in:

- The **2D Constraints** toolbar.
- The **Parametric ribbon** tab.
- The **2D Constraints** fly-out menu of the **Parametric** menu.

In the image below:

- The endpoints of the three lines and the arc are joined by coincident constraints.
- The midpoint of the circle and the arc are made concentric.
- Two lines are forced to be tangent to the arc.
- One line has a vertical constraint (= parallel to the Y-axis of the current coordinate system), one line has a horizontal constraint (= parallel to the X-axis of the current coordinate system).



### 6.6.2 Displaying constraint bars

Constraint bars are hidden when a drawing is closed and reopened.

Tools to control the display of constraint bars are located on:

- The **Geometric Constraint** toolbar.
- The **Parametric** ribbon tab.
- The **Show/Hide 2D Constraints** fly-out menu of the **Parametric** menu.

These tools execute the CONSTRAINTBAR command using a specific option.

The options are:

- Show/Hide: prompts you to select the entities.
- Show All: show all dimensional constraints in the drawing.
- Hide All hide all dimensional constraints in the drawing.

***Прим.:***

- The value of theCONSTRAINTBARDISPLAYsystem variable controls the visibility of geometric constraints:
  - 1: Display constraint bars when geometrical constraints are added.
  - 2: Display hidden dimensional constraints when the constrained entities are selected.
- When hovering over an entity with a geometric constraint, the blue constraint glyph ( ) displays if theSELECTIONPREVIEWsystem variable is set to 1 or 3 or when the Quad is active.

### 6.6.3 Controlling the position of a constraint bar

By default constraint glyph bars are created close to the midpoint of the entity and are kept at that relative position when the entity position changes. You can drag the constraint bar to a different location. This new relative position is then maintained until the**Reset**option of theCONSTRAINTBARcommand restores the default position of the constraint bar.

**Relocating a constraint bar**

1 Place the cursor on the constraint bar.

2 Press and hold the left mouse button to move the constraint bar.

3 Release the left mouse button at the desired location.

**Restoring the default position of constraint bars**

1 Do one of the following:

- Click the**Show/Hide Geometric Constraint**tool.
- Launch the CONSTRAINTBAR command.

Prompts you:Select Entities.

2 Select the entities, then right click or press Enter.

BricsCAD® reports the number of selected entities.

Prompts you: Select option for constraints [Show/Hide/Reset] <Show>:

3 Do one of the following:

- Choose**Reset**in the context menu.
- Type**R**in the Command line, then press Enter.

### 6.6.4 Working with constraint bars

**Controlling a constraint**

1 Move the cursor over a constraint icon.

2 A tooltip displays, indicating the constraint type.

3 The associated entity (or entities) highlights.

4 The corresponding icon on the constraint bar of the associated entity highlights.

**Deleting a constraint**

1 Move the cursor over the constraint icon in the constraint bar.

2 Right click, then click**Delete**.

3 The constraint is deleted and the icon is removed from the constraint bar and from the constraint bar of the associated entity.

**Hiding the constraint bar of an entity**

Move the cursor of the constraint bar, then click the**Close**(x) button.

### 6.6.5 Deleting constraints

To delete all constraints from a selection set:

1 Do one of the following:

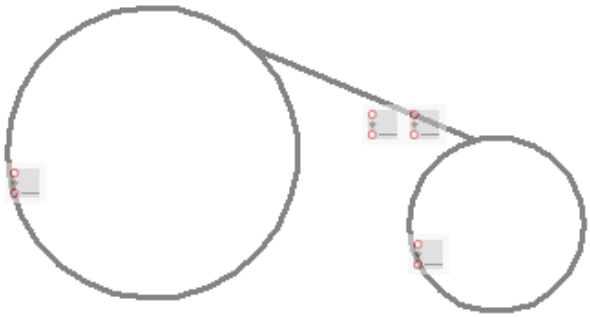
- Click the**Delete 2D Constraints**tool.
- Launch the DELCONSTRAINT command.

You are prompted: Select entities to delete all constraints:

2 Selectthe entities you want to delete the constraints from.

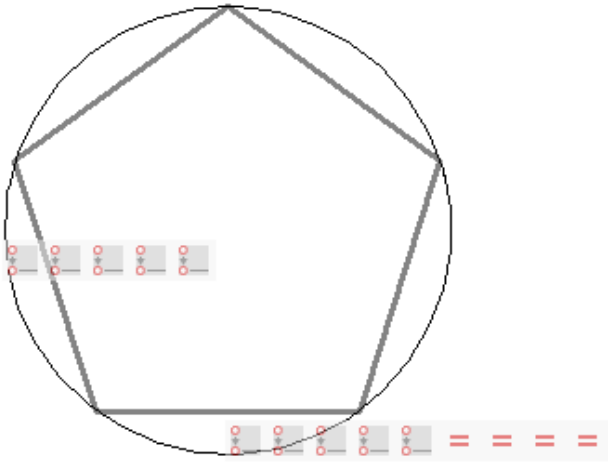
3 Right click to stop selecting entities and delete all constraints from the selection set.

### 6.6.6 Some examples

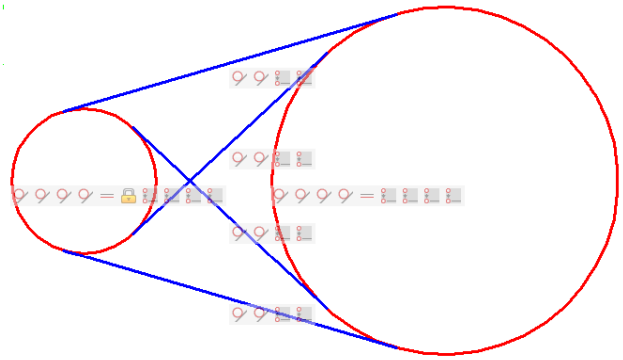


Coincident constraints between the endpoint of a line and two circles.

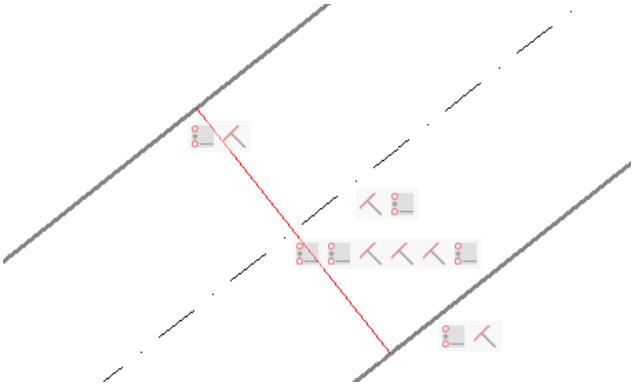
If the endpoints or center points of entities already coincide, the**Autoconstraint**option of theGCOINCIDENTcommand automatically applies coincident constraints to such points.



By applying coincident constraints between the pentagon vertices and the circle, and equal constraints between one side and the four other sides, the circle radius defines the size of the pentagon.



Coincident constraints between the endpoints of the tangent lines and the circles prevent the tangent lines to extend beyond the tangent points.



Coincident constraints are used to:

- Force the endpoints of the red line to lie on the bold lines.
- Force the midpoint of the red line to lie on the dash-dot line.

The bold lines and the dash-dot line have a perpendicular constraint with the red line. As a result the dash-dot line always lies centered between the two bold lines.

## 6.7 Dynamic dimensions

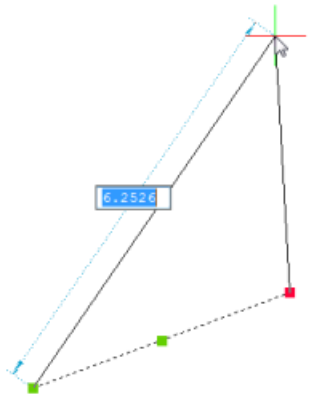
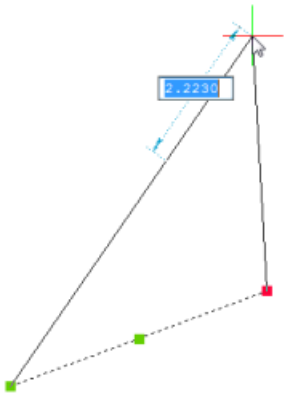
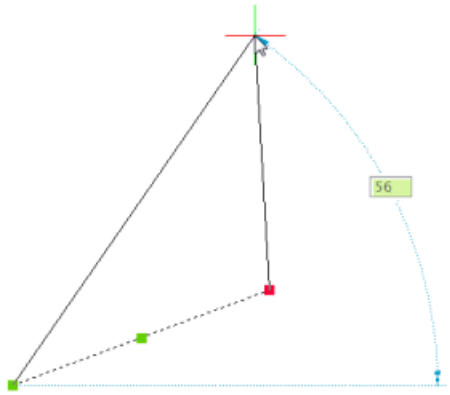
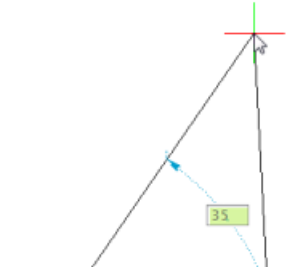
Dynamic dimensions provide an interface at the cursor position to specify the length and angle of the linear entity being created or grip-edited. As such, dynamic dimensions help you to focus on the drawing area.

The behavior and appearance of dynamic dimensions is controlled through a series of system variables and user preferences.

### 6.7.1 Defining the Dynamic Dimensions Settings

Right click the**DYN**field in theStatus Bar, then choose**Settings**in the context menu. The**Dynamic input**settings in the**Settings**dialog display.

| Name | Title | Description |
| --- | --- | --- |
| DYNMODE | Dynamic input mode | Controls whether and when dynamic dimensions display:<br>• Editable dynamic dimensions: when creating or manipulating entities.<br>• Tracking dynamic dimensions: when using entity snap tracking. |

| Name | Title | Description |
| --- | --- | --- |
| DYNDIGRIP | Show dynamic dimensions | Defines which dynamic dimensions display when creating or grip editing entities, depending on the value of the DYNDIVIS system variable<br>Resulting length: total length of the line or polyline segment<br><br>Extended length: incremental length of the line<br><br>Absolute angle: angle relative to the X-axis of the current UCS<br><br>Relative angle: angle relative to the original angle of the linear entity<br> |

| Name | Title | Description |
|---|---|---|
| DYNDIVIS | Dynamic dimension visibility | Defines the number of dynamic dimensions that display when creating or grip editing entities. |
| DYNPICOORDS | Default mode for dynamic coordinates input | Specifies the default mode in which coordinates are entered during dynamic input: Relative or Absolute. |

***Прим.:***

- The appearance of dynamic dimensions (color, linetype, position and transparencey) is controlled by a set of user preferences: DynDimColorHover, DynDimColorHot, DynDimDistance, DynDimLinetype, DynImputTransparence.
- If bitcode 16 (= format dynamic dimensions) of thePROPUNITSsystem variable is set, dynamic dimensions are formatted using a dedicated unit.
- Automatic unit conversion during length input can be used when bitcode 1 ofPROPUNITS(Format length properties) is set. For example, whenINSUNITSis set to **mm**, entering **2m** in a length property field will result in entering [2000.0000 mm]. Entering **1ft** gives [304.8000 mm]. If no unit is specified in the entry field, the unit as specified byINSUNITSis used.

### 6.7.2 Using dynamic dimensions when creating entities

1 Launch the command to create a 2D entity (e.g. LINE andDYNDIVIS= 1 or 2).

2 Specify the first point, then move the cursor to display the dynamic dimensions.

TheLengthdynamic dimension highlights.

3 Do one of the following:

- Type the desired length of the line, then press the TAB key to jump to theAngledynamic dimension.

  TheLengthdynamic dimension is locked and displays in red and theAngledynamic dimension is selected.
- Type the desired length of the line, then click to create the line at the angle defined by the cursor position.
- Press he TAB key to jump to theAngledynamic dimension.

4 Do one of the following:

- Type the desired angle, then press the Enter key to create the line.
- Type the desired angle, then press the TAB key to return to theLengthdynamic dimension.
- Press the Enter key to create the line at the angle defined by the cursor position.
- Press the TAB key to return to theLengthdynamic dimension.

***Прим.:***

- IfDYNDIVIS= 0, only one dynamic dimension displays. Press the TAB key to display the next field.
- If you type a value in the**Length**field, and add a comma (,) the content of the**Length**field is copied to the Command line and the @-character is placed in front automatically, which allows you to specify the next point usingrelative coordinateswith respect to the previous point.

### 6.7.3 Using dynamic dimensions to grip-edit entities

1 Select the entity or entities.

All grips on the selected entities display.

2 Click the grip you want to edit, then move the cursor.

Depending on the value of theDYNDIVISsystem variable one, two or all dynamic dimensions display simultaneously .

The first dynamic dimension highlights.

3 Do one of the following:

- Type the desired value for the currently highlighted dynamic dimension, then press the TAB key to jump to the next dynamic dimension.

  The dynamic dimension is locked and displays in red.
- Type the desired value for the currently highlighted dynamic dimension, then press Enter to stop.
- Press the TAB key to jump to the next dynamic dimension.

4 Repeat step 3 or press Enter to stop.

### 6.7.4 Using dynamic dimensions to measure entities

The following procedure applies to the 2DWireframe visual style only.

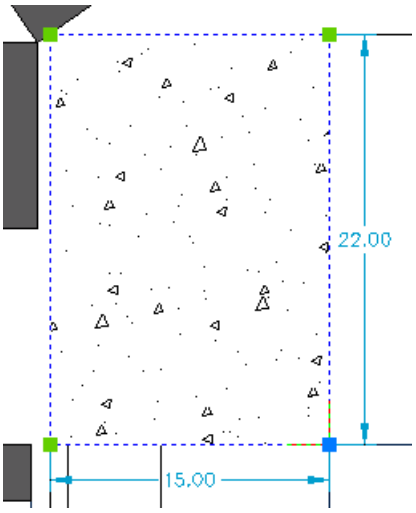
1 Select the entity or entities.

All grips on the selected entities display.

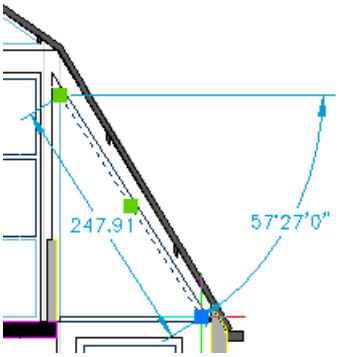
2 Hover the cursor over a grip.

The grip highlights and all dimensions related to the selected grip display.

Dynamic dimensions used to measure a rectangle




Dynamic dimensions used to measure a line




3 (option) Click to start editing the selected grip.

### 6.7.5 Using dynamic dimensions with tracking points

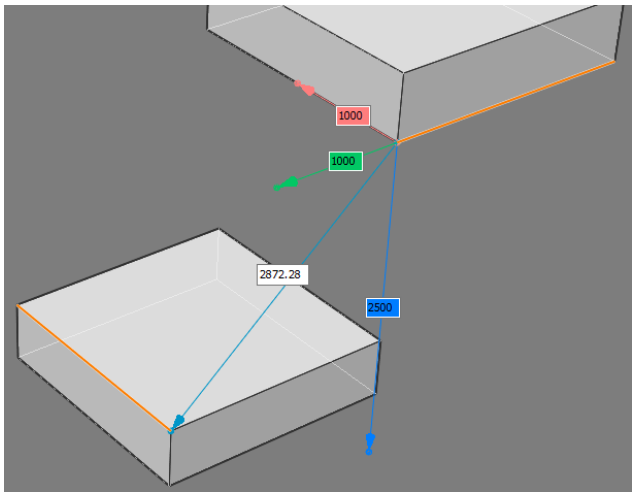
If the**Tracking dynamic dimensions**option of theDYNMODEsystem variable is set, dynamic dimensions display whenentity snap tracking pointsortemporary tracking pointsare acquired.

## 6.8 Nearest distance

The Nearest Distance feature allows to dynamically view and modify the distance between two selected entities or subentities. In addition, the aligned nearest distances along the X-, Y- and Z-axes of the WCS display.

Double clicking a Nearest Distance field:

- Toggles the display of the X-, Y- and Z-axis aligned distance fields.
- Zooms out to fully display the selected entities if necessary.




TheNEARESTDISTANCEsystem variable controls the display of the Nearest Distance feature.

### 6.8.1 Using Nearest Distance

1 Select the first entity or subentity*.

2 Select the second entity or subentity*.

The nearest distance displays.

The arrowhead points to the second entity.

3 (option) Double click a distance field to zoom out to fully display the selected (sub)entities and/or toggle the display of the aligned nearest distances along the X-, Y- and Z-axes.

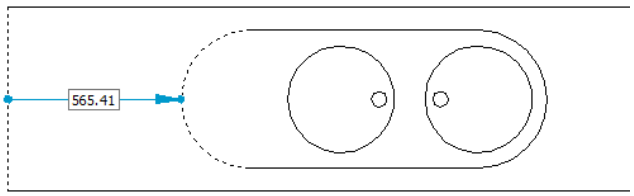
4 (option) Click a distance field and type a value.

The second entity is moved.

* Press and hold the Ctrl-key to select a subentity.

You can use the Nearest Distance feature to:

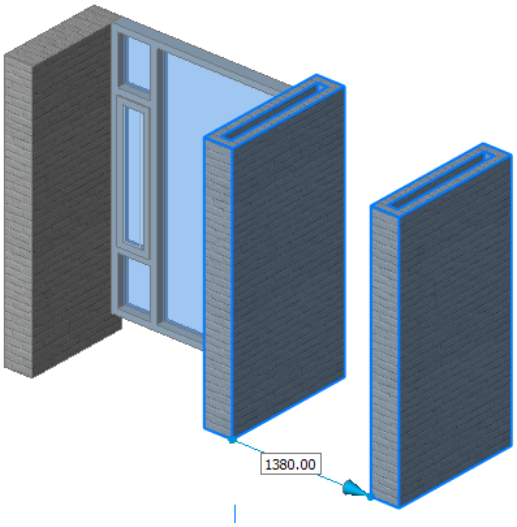
- Measure the distance between two entities or subentities.
- Adjust the distance between a (sub)entity and a block.



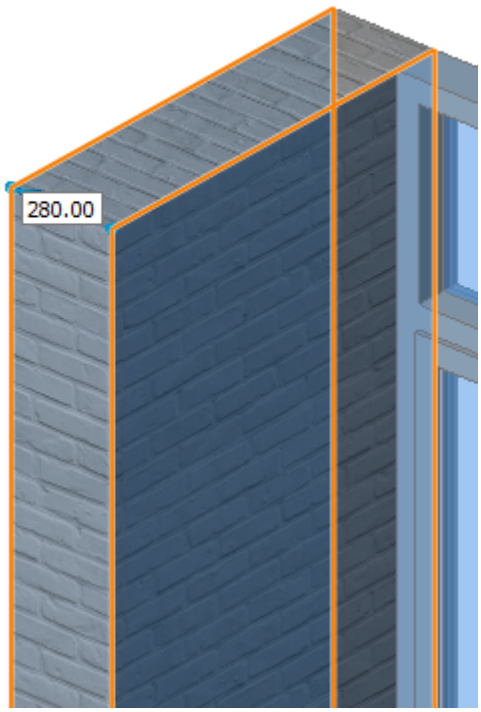

- Modify the dimensions of a rectangle.



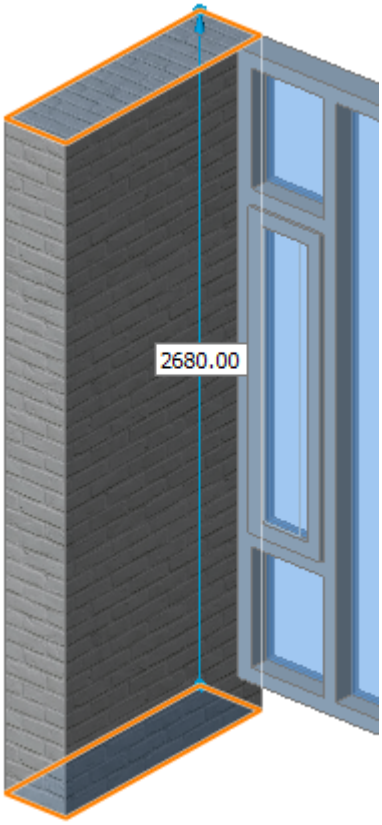

- Adjust the distance between 3D solids.




- Edit the width of a 3D solid.




- Measure and edit the height of 3D solid.




## 6.9 Coordinate input

When you create entities in a drawing, they are located in relation to the underlying Cartesian coordinate system of the drawing. A drawing has a fixed coordinate system called the**World Coordinate System**(WCS).

You can also define arbitrary coordinate systems which are calleduser coordinate systems(UCS) and can be located anywhere in the WCS and oriented in any direction.

To specify points and distances using the keyboard you can use the following formats:

- Cartesian coordinates: x,y,z
- Cylindrical coordinates: R<alpha,z
- Spherical coordinates: R<alpha<beta

### 6.9.1 Relative coordinates

When you type the @-character in front of the entry, the coordinates are calculated with respect to the previous point. This technique is called**Relative Coordinates**.

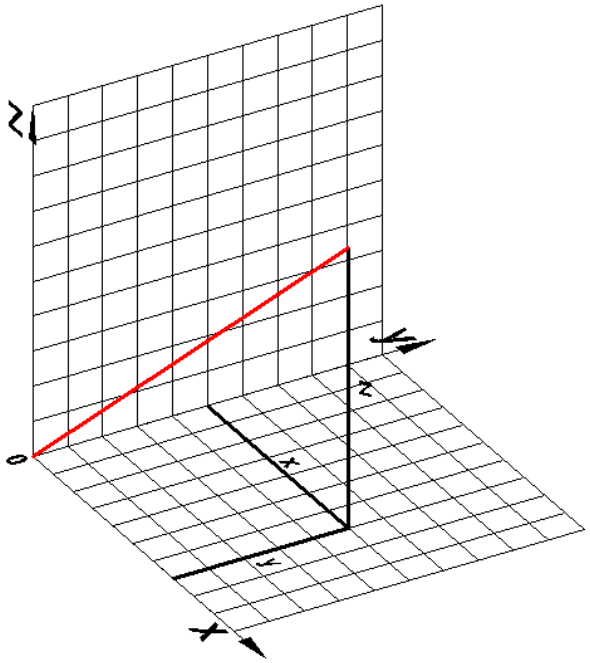
WhenDynamic Dimensionsare active and you type a value in the**Length**field and add a comma (,) the content of the**Length**field is copied to the command line and the @-character is placed in front automatically, which allows you to specify the next point using relative coordinates with respect to the previous point.

### 6.9.2 Absolute coordinates

When a user defined coordinate system is active you can enter absolute coordinates (World coordinates) if you place an asterisk (*) in front. For example, *0,0 refers to the origin of the WCS (World Coordinate System).

### 6.9.3 Working with Cartesian coordinates

The Cartesian coordinate system uses three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes. A point is defined by its distances to the yz-, xz- and xy- planes. These distances are called the xyz-coordinates of a point.




- Type the x-, y- and z- coordinates separated by commas to enter the absolute Cartesian coordinates of a point.
- When the z-coordinate is omitted, the point is placed in the xy-plane (Z = 0).
- Type the @-character in front of the entry (@x,y), to specify the coordinates with respect to the previous point. This technique is called**Relative Cartesian coordinates**.

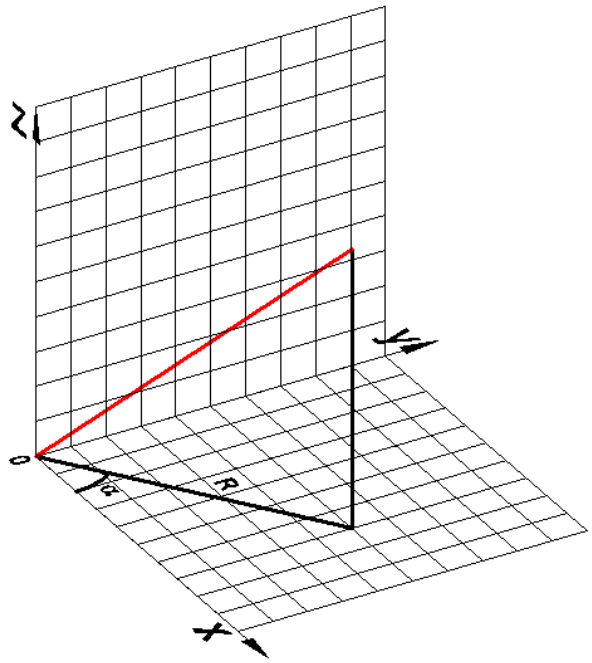
### 6.9.4 Working with cylindrical coordinates

A cylindrical coordinate system uses three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes.

A point is defined using the following format: R<alpha, z.

- R = distance to the origin in the xy-plane
- <alpha = the angle between R and the x-axis (positive angles are measured counter clockwise)

- z = the height above the xy-plane.
- When the z-coordinate is omitted, cylindrical coordinates are referred to as polar coordinates.
- Type the @-character in front of the entry to calculate the coordinates with respect to the previous point.

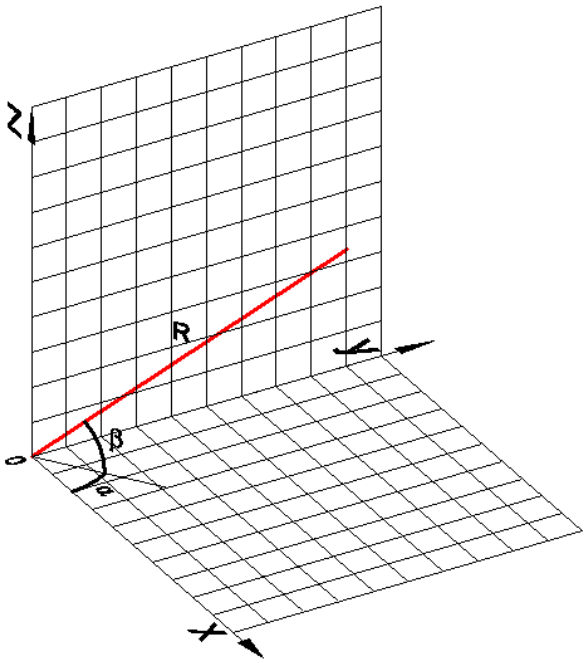



### 6.9.5 Working with spherical coordinates

In a cylindrical coordinate system we use three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes.

A point is defined using the following format: R<alpha<beta

- R = distance from the origin
- <alpha = angle in the xy-plane (positive angles are measured counter clockwise)
- <beta = angle measured from the xy-plane (positive angles are measured counter clockwise, above the xy-plane)

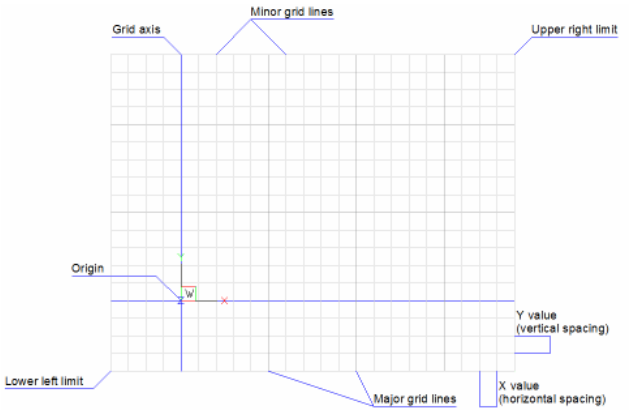



## 6.10 Snap and grid

Grid and snap help you to draw fast and accurately.

A grid is a set of evenly spaced lines or dots that serve as a visual distance reference.

TheGRIDSTYLEsystem variable controls whethergrid lines or dotsdisplay. TheGRIDDISPLAYsystem variable controls thedisplay of the grid. The grid can also indicate how far the drawing limits extend.




The snap feature creates a set of evenly spaced, invisible magnetic points,which make the crosshairs move in even increments. Both grid and snap are like the intersection points of the lines on a piece of grid paper. Grid points are for visual reference only and they do not print. Snap constrains the points that you can pick with the mouse.

- Click theGRIDtool ( ) on the Settings toolbar or theGRID field in theStatus Bar, or pressCtrl+Gto toggle grid display on/off.

- Click theSNAPtool () on the Settings toolbar or theSNAP field in theStatus Bar, or pressCtrl+Bto toggle snap on/off. The setting is saved through the SNAPMODE system variable.

  If SNAPTYPE = 2,adaptive grid snapis used and SNAPMODE is denied.
- BothSNAPandGRIDcan be set differently in eachviewport.
- TheSnapoption of theGRIDcommand equals the grid spacing to the current snap spacing.
- Grid lines or dots do not print.

### 6.10.1 Displaying the Snap and grid settings

Do one of the following:

- In theStatus Bar, right click on theSNAPor GRID field, then chooseSettingsin the context menu.
- In theSettingsdialog go to: **Drawing** > **Drafting** > **Coordinate Input** > **Snap** > **Grid**.

### 6.10.2 Synchronizing snap and grid spacing

1 Launch the GRID command.

  Prompts you:

  Grid is on. Enter grid spacing or [grid OFf/Snap/Aspect] <10>:

2 Type**s**, then press Enter.

  The grid spacing matches the snap spacing.

3 (Option) Use the Aspect option of the SNAP command to adjust the spacing.

***Прим.:*** The grid spacing remains in sync with the snap spacing until you set the grid spacing differently using theGRID command or in the Settings dialog box.

### 6.10.3 Using drawing limits

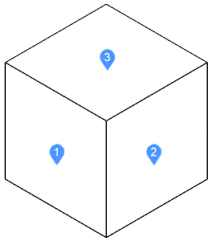
- Execute the LIMITScommand to set the drawing limits.
- Click the**Drawing Limits**item () in the Settings menu to toggle theLIMCHECKsystem variable on/off.

  If LIMCHECK = ON, it is not possible to draw outside the drawing limits.
- The**Display beyond LIMITS area**option of theGRIDDISPLAYsystem variable defines whether or not the grid displays beyond the drawing limits.

### 6.10.4 Using isometric snap

Isometric grid allows to create two-dimensional isometric drawings, Which simulate a three-dimensional view on a two-dimensional plane. Do not confuse isometric drawings with three-dimensional drawings.

The isometric option always uses one of three preset planes, which are denoted as**Left** (1),**Right** (2)and**Top** (3). You cannot alter the arrangement of these planes. The three isometric axes are 30 degrees, 90 degrees, and 150 degrees.




Do the following:

- Set SNAPSTYL = 1 (isometric Snap).
- Set GRIDSTYLE = 1 (dotted grid).
- Set SNAPISOPAIR: 0 = Left, 1 = Top, 2 = Right.

  Press the F5 function key to cycle the isometric planes.
- Set ORTHOMODE = On to constrain the movement of the crosshairs to the current isometric plane.
- Make sure SNAPANG = 0.

### 6.10.5 Setting the snap angle

You can rotate a dotted grid using the SNAPANG system variable (ifSNAPANG is different from 0, grid lines do not display).

1 Display the**Snap/Grid**settings in theSettingsdialog.

2 Select SNAPANG system variable.

3 Type a new value in the setting field.

4 Close the Settings dialog.

***Прим.:*** You can also set theSnap Angleby typing**snapang**in the Command line. When in a command, type**'snapang**(with an apostrophe in front), to set theSnap Anglevariable transparently (= without interrupting the running command).

## 6.11 Adaptive grid snap

The adaptive grid snap feature lets you create, edit,nudgeandmanipulateentities using concise values without keyboard entry.

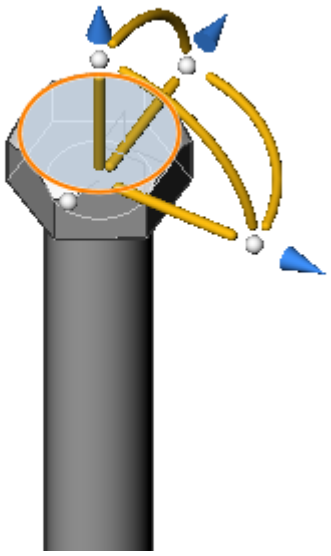
The snap step size is adapted according to the current screen zoom factor. The adaptive grid snap is active ifSNAPTYPE = 2. This system variable overrides theSNAPMODE system variable.

The AdaptiveGridStepSize user preference controls the smallest available step size; default = 4 pixels.

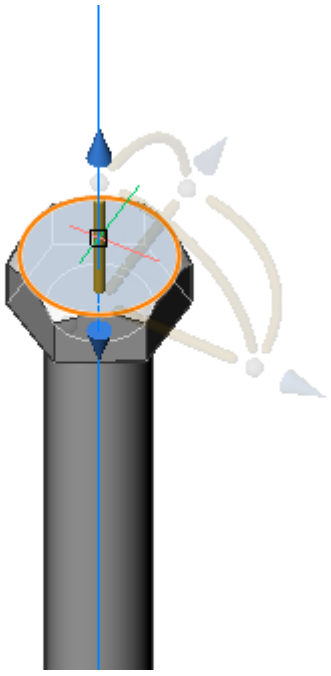
### 6.11.1 Using adaptive grid snap with the Manipulator

1 Move the cursor over an entity or subentity, then hold down the left mouse button a little bit longer to select the entity (= long left click).

The Manipulator widget displays:



2 Move the cursor over the axis in the direction you want to move the entity. The selected axis highlights; the other parts of the Manipulator are faded.



3 Click to select the axis, then move the cursor.

The Ruler displays.

The current displacement distance displays in a dynamic field.

The grid snap resolution automatically adapts according to the zoom factor.

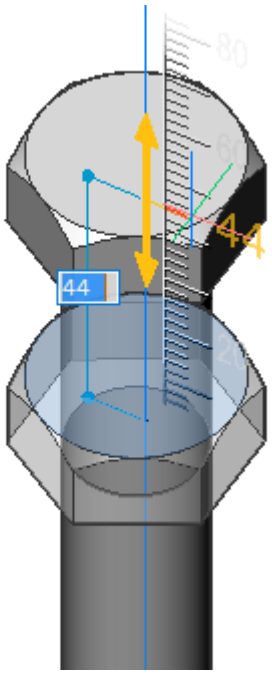
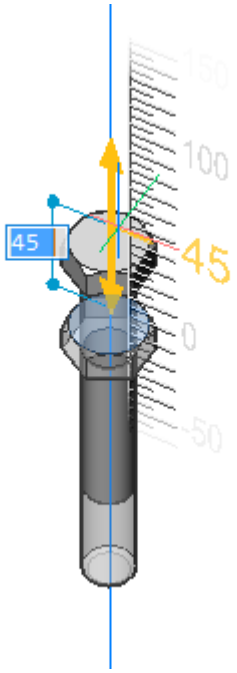



**Рисунок 2:** Ruler divisions at 2 mm

**Рисунок 3:** Ruler divisions at 5 mm




4 (option) Zoom in or out to adjust the dynamic grid.

5 Do one of the following:
   - Click to accept the current value in the Ruler.
   - Type a value in the dynamic field, then press Enter.

## 6.12 Nudge

The Nudge drawing aid allows you to move selected entities or subentities in small increments along the X, Y, or Z axis using the keyboard.

### 6.12.1 Using Nudge

1 Select one or more entities or subentities (for example polyline segments or 3D solid faces).

2 Do the following to move the selection:

- Along the X axis of the current coordinate system:

  Hold down the Ctrl key, then use the left/right arrow keys.

  The Command line reads: NUDGE: offset along the X axis of the UCS: xx*
- Along the Y axis of the current coordinate system:

  Hold down the Ctrl key, then use the up/down arrow keys.

  The Command line reads: NUDGE: offset along the X axis of the UCS: xx*
- Along the Z-axis of the current coordinate system:

  Hold down the Shift key, then use the up/down arrow keys.

  The Command line reads: NUDGE: offset along the Z axis of the UCS: xx*

* The step size used by the NUDGE functionality is based on theadaptive grid snapresolution. Zoom in/out to decrease/increase the step size.

## 6.13 2D entity snaps

Entity snaps enable you to quickly select exact geometric points on existing entities without having to know the exact coordinates of those points.

With entity snaps, you can select the end point of a line or arc, the center point of a circle, the intersection of any two entities, or any other geometrically significant position. You can also use entity snaps to draw entities that are tangent or perpendicular to an existing entity. You can use entity snaps any time you need to specify a point.

You can work with entity snaps in one of two ways:

- Enable a running entity snap that remains in effect until you turn it off by choosing an entity snap when no other command is active.
- Enable a one-time entity snap for a single selection by choosing an entity snap when another command is active. You can also use a one-time entity snap to override a running entity snap.

When using entity snaps, the program recognizes only visible entities or visible portions of entities. You cannot snap to entities on layers that have been turned off or to the blank portions of dashed lines.

If theAPBOXsystem variable is on, the entity Snap Aperture Box is added to the crosshairs when one or more entity snaps are active.

When you move the cross hairs, the program snaps to the snap point closest to the center of the Snap Aperture Box. The**Snap Marker**indicates the current snap point.

TheDRAGSNAPsystem variable controls the snap behavior while dragging, providing an increased WYSIWIG experience. DRAGSNAP controls whether rubber band dynamics are displayed at the current cursor location or at the current entity snap location. DRAGSNAP applies to all modify commands that display dynamics, such as COPY, PASTECLIP, PASTEBLOCK, MOVE, ROTATE, MIRROR, SCALE and STRETCH.

- DRAGSNAP = 0 (Default): dragged entities display at the cursor location.
- DRAGSNAP = 1: dragged entities display at the current entity snap location.

TheOSOPTIONSsystem variable suppresses entity snaps on certain entity types. The value is stored as a bitcode using the sum of the values of the selected options.

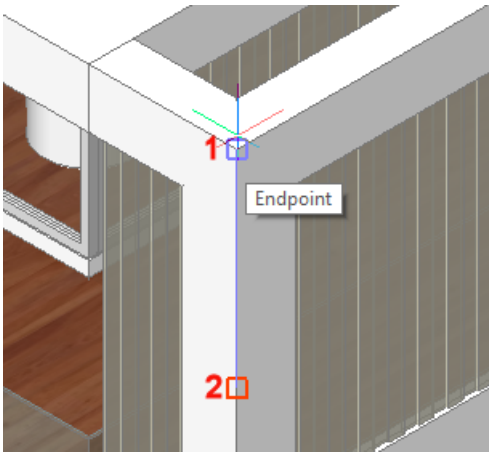
- 1: Entity snapping is disabled for hatch entities.
- 2: Entity snapping is disabled for geometry with negative Z values when using a dynamic UCS.
- 4: Entity snapping is disabled for endpoints of dimension extension lines.

***Прим.:*** Press theTABkey, without moving the mouse, to cycle through all possible entity snaps available at the current cursor position.

### 6.13.1 Defining the 2D Entity Snap system variables

Right click the ESNAP field in the Status bar then choose settings in the context menu. The Settings dialog box displays with the 2D and 3D entity snaps settings expanded.

### 6.13.2 2D Entity Snap options

| Icon | Description |
| --- | --- |
|  | Sets the OSNAPZ system variable to Ignore entity snap elevation, replacing the Z coordinate of the snapped point with the current value of the ELEVATION system variable.<br><br>The 3d snap marker (1) displays the identified 3d point, the snap marker (2) displays at the current Elevation height. |
|  | Disables the Ignore hatches option of the OSOPTIONS system variable, thus allowing to snap to hatches. |
|  | Disables the Entity snap to negative-z option of the OSOPTIONS system variable, thus allowing entity snaps with a negative z-value when aDynamic UCSis active. |
|  | Disables the Ignore endpoints of dimension extension lines option of the OSOPTIONS system variable, thus allowing entity snaps to dimension extension lines. |

### 6.13.3 Entity Snap Modes

| Name | Icon | Marker | Context menu shortcut* | Transparent shortcut** | Description |
|---|---|---|---|---|---|
| TK |  |  | | tk | Use temporary tracking points. Prompts you to identify temporary points; press Enter to accept the last temporary point. |
| MTP |  | | 2 | mtp | Snaps to a point in the middle between two points. Prompts you to identify two points. |
| ENDPOINT |  |  | E | end | Snaps to the nearest endpoint of an entity or polyline segment. |
| MIDPOINT |  |  | M | mid | Snaps to the midpoint of an entity or polyline segment. |
| NEAREST |  |  | N | nea | Snaps to the nearest point on an entity. |
| CENTER |  |  | C | cen | Snaps to the center point of an arc, circle, polygon, ellipse or elliptical arc. |
| GEOMETRIC CENTER |  |  | G | gce | Snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids. |
| PERPENDICULAR |  |  | P | per | Snaps to the perpendicular point of another entity. You can snap to an arc, circle, ellipse, line, polyline, infinite line, ray, spline or edge of a plane to form a perpendicular alignment with that entity or with an extension of that entity. |
| PARALLEL |  |  | L | par | Displays a tracking line parallel to the selected entity. |
| TANGENT |  |  | T | tan | Snaps to the point on an arc, ellipse, spline or circle that, when connected to the previous point, forms a line tangent to that entity. |

| Name | Icon | Marker | Context menu shortcut* | Transparent shortcut** | Description |
|---|---|---|---|---|---|
| QUADRANT |  |  | Q | qua | Snaps to the closest quadrant of an arc, circle, ellipse, or elliptical arc. |
| INSERTION |  |  | I | ins | Snaps to the insertion point of an attribute, block or text entity. |
| NODE |  |  | O | poi<br>nod | Snaps to a point entity. |
| INTERSECTION |  |  | R | int | Snaps to the intersection of any combination of entities. |
| APPARENT INTERSECTION |  |  | A | app | Snaps to the apparent intersection in the current view of two entities that do not intersect in three-dimensional space.<br>Snaps also to the intersection of any combination of entities. |
| EXTENSION |  |  | X | ext | Snaps to the extension of an entity or to the intersection of the extension of two entities. |
| NONE |  | | Y | non | Turns off all entity snap modes. |

* Type this letter to quickly select the entity snap mode in the right click context menu.

** Type the transparent shortcut in the Command line when a command prompts for a point.

Other running entity snaps are temporarily disabled.

### 6.13.4 Setting the Entity snaps

Do one of the following:

- Press and hold the Shift key, then right click and select the Entity Snap mode in the context menu.

  The icons of the currently active Entity Snap modes are outlined.
- Click the buttons on the Entity Snaps toolbar.

  The buttons of the currently active entity snap modes are pressed.

When no command is active, the above procedures toggle the Entity Snap modes on/off.

When a command is active, the above procedures set a 'one shot' override of the current Entity Snap modes.

Alternatively, you can type a single letter shortcut as indicated in the table below.

To toggle theEntity Snapson/off:

- Click theESNAPfield in theStatus Bar.
- Press the**F3**function key.

### 6.13.5 Working with multiple Entity Snap modes

1 Move the cursor to the entity you want to snap.

One of the active Entity Snap modes markers display.

2 Do not move the mouse and press the TAB key.

The entity the snap point is on highlights.

3 Do one of the following:

- Click to accept the snap point.
- Press the TAB key.

The next possible Entity Snap mode marker displays.

The entity the snap point is on highlights.

4 (option) Repeat step 3 until the right snap point is found.

### 6.13.6 Snapping to the extension of two entities

1 If not already on, turn on the**Extension**Entity Snap mode.

2 Launch an entity creation command, for example LINE.

3 Move the cursor over the endpoint of the first entity (1).

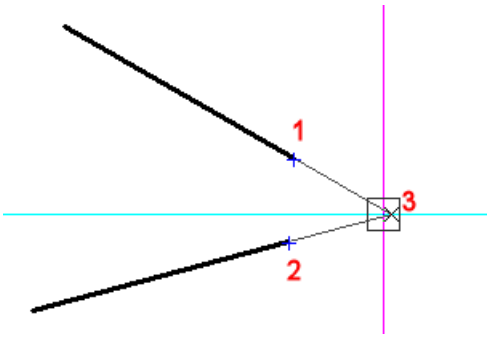
A small red cross indicates the entity is marked for extension.

4 Move the cursor over the endpoint of the second entity (2).

A small red cross indicates the entity is marked for extension.

5 Move the cursor near the intersection of the extensions of the two entities.

An X indicates the intersection (3).




6 Click to accept the snap point.

***Прим.:*** You can snap to the extension of lines, polylines, arcs and elliptical arcs.

### 6.13.7 Using the Middle of 2 points snap mode

1 When prompted to specify a point, do one of the following:

- Activate the**Snap to the middle of 2 points**snap mode ( ).
- Type**mtp**or**m2p**in the Command line and press Enter.

You are prompted: First point of mid:

2 Specify the first point.

You are prompted: Second point of mid:

3 Specify the second point.

The point in the middle of the two points is snapped.

### 6.13.8 Using the Parallel entity snap mode

1 Make sureEntity Snap Trackingis active.

2 Launch an entity create command to create a linear entity (for example Line).

3 Specify the first point of the entity.

The line displays dynamically.

4 Press and hold the Shift key, then right click and choose**Snap to parallel**( ) in the context menu.

5 Hover the cursor over the entity you want to draw parallel to.

The**Parallel Entity Snap**marker ( ) displays on the selected entity.

Wait until a small cross (+) indicates the entity is recognized.

6 Move the cursor in order to place the dynamics of the line more or less parallel to the selected entity.

A tracking line displays from the start point of the line.

7 Do one of the following to specify the second point while the tracking line displays:

- Click a point.
- Enter the desired length and press Enter.
- Use a second entity snap tracking point, then move the cursor to the intersection of the parallel tracking line and the second entity snap tracking line.

## 6.14 3D entity snaps

In addition to the2D entity snaps, a series of 3D entity snap modes apply to points on 3D entities such as solids, surfaces, splines and meshes.

The current 3D entity snap modes are saved through the 3DOSMODE system variable as the sum of the bitcodes of the selected snap modes.

| Name | Icon | Marker | Description |
|---|---|---|---|
| Center |  |  | Snaps to the center of 3D faces. |
| Intersection |  |  | Snaps to the intersections of linear entities, edges, and polar or entity snap tracking lines with faces. |
| Knot |  |  | Snaps to knots (fit points) onsplines. |
| Midpoint |  |  | Snaps to the midpoint of a face edge. |
| Nearest |  |  | Snaps to a point on the face of a 3D entity that is nearest to the cursor. |
| None |  | | Disables all 3D snap modes. |
| Perpendicular |  |  | Snaps to a point perpendicular to a face on a 3D entity. |
| Vertex |  |  | Snaps to the closest vertex of a 3D entity and the control vertices ofsplines. |
| Point Cloud |  |  | Snaps to point cloud points. |

### 6.14.1 Setting the 3D Entity Snap modes

- Click the buttons on the**3D Entity Snaps**toolbar.

  The buttons of the currently active entity snap modes are pressed.
- When no command is running, clicking a tool button toggles the snap mode on/off.
- When executing a command, clicking a tool button sets the snap mode for the next point, temporarily disabling all other selected snap modes.
- Run the3DSONAPor-3DOSNAPthe commands.
- Shift + Right click, hover over**3d Entity Snap**in the context menu and click the 3D entity snap option on the flyout menu.

## 6.15 Polar tracking

Polar tracking (POLAR) allows you to draw at exact angles:

- At fixed intervals, starting from 3 o'clock (East).
- At specific additional angles.

A polar tracking path line displays from the origin point when you move the cursor close to one of the polar tracking angles. The tracking path lines display as long as theSnap Aperture box overlaps the polar tracking line.

In 3D views an additional tracking line displays parallel to the Z-axis of the current UCS.

Snap track lock: to lock a snap track path hit the Shift key. To unlock a track path, hit Shift again: it works as a toggle. Locked tracking paths display in thecolor defined by the SnapMarkerColoruser preference.

- The display of the tracking path line is controlled through the TRACKPATH system variable.
- The color of the tracking path line is controlled through the AutoTrackingVecColor user preference.
- Tracking paths parallel to the X-, Y- or Z-axis of the current coordinate system (WCS or UCS) display in the color of the axis they are parallel with (defined by COLORX , COLORY or COLORZ variables).
- Turning onPOLARautomatically disablesORTHO (Orthogonal Mode) and vice versa.
- The ANGBASE system variable defines the base angle of the tracking lines with respect to the current coordinate system.

### 6.15.1 Defining the tracking settings

1 Do one of the following:
   - Right click on thePOLARor the STRACK field in theStatus Bar, then right click and select**Settings**in the context menu.
   - Open the**Settings**dialog and expand the**Coordinate input**subcategory under**Drafting**in the**Drawing**settings class.

     Expand the**Snap Tracking**settings group.

2 Adjust the settings:
   - AUTOSNAP system variable: controls the behavior of Polar Tracking and whether polar tracking and/or entity snap tracking is active.
   - POLARANG system variable: sets the polar angle increment in degrees; default = 90.
   - POLARADDANG system variable: sets additional angles; separate angles by semicolons.
   - POLARMODE system variable: controls the behavior of polar and entity snap tracking.
   - Relative: the angles are measured with respect to the direction of the snapped entity; otherwise angles are measured according to the current coordinate system (WCS or UCS).
   - Use polar angle settings for entity snap tracking also.
   - Whether additional angles (POLARADDANG) are used or not.
   - Use the Shift key to acquire entity snapping points.
   - TRACKPATH system variable: controls whether and how the track paths display: full screen or partially.

### 6.15.2 Locking a tracking line

Press the Shift key while a snap track marker line displays to lock a tracking path, any (snap)point that is picked will be projected perpendicularly onto the tracking path. Press the Shift key again to unlock.

A locked tracking path changes to the color defined by theSnapMarkerColoruser preference.

### 6.15.3 Using temporary tracking points

When**Polar Tracking** is ON, temporary tracking points can be used to pick points at locations where no existing geometry is available to snap directly to, while you know the desired offset distance(s) from snap points on existing geometry. It is recommended to havedynamic dimensions(DYN) active as well.

1 Whenever a command prompts for a point, do one of the following:

- Click the middle mouse button (mouse wheel).
- Activate the**Temporary tracking points**entity snap mode ( ).
- Type**TK**in the Command line and press Enter.

  The command is suspended temporarily.

  You are prompted: Specify first temporary tracking point:

2 Enter a point, for example use an entity snap to specify a point.

You are prompted: Specify next temporary tracking point (press ENTER to end).

3 Do one of the following:

- Enter a point.
- Press Enter to accept the last temporary tracking.

  The command resumes.

## 6.16 Entity snap tracking

Entity Snap Tracking(STRACK) assists you in drawing entities with respect to other entities.

WhenEntity Snap Trackingis active, BricsCAD® displays temporary tracking lines from entity snap points.At least oneEntity Snapmode must be on if you want to useEntity Snap Tracking.

Click theSTRACKfield in theStatus barto enable/disableEntity Snap Tracking.

Entity Snap Trackinglines are parallel to the X- and Y-axis of the current UCS.

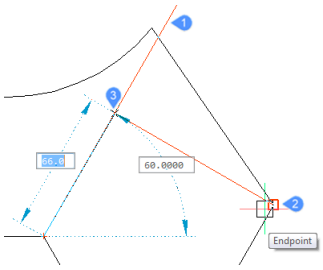
If bitcode 2 of the POLARMODE system variable is checked, the polar angles used inPolar Tracking(interval and additional angles) are also used inEntity Snap Tracking.

Alignment points are acquired when you hover over a point, using entity snaps. Acquired points display a small plus sign (+). To remove an acquired point, hover over the point again. The small plus sign (+) is then removed. If bitcode 8 of the POLARMODE is checked, press and hold the Shift key to acquire entity tracking points. Alignments relative to an acquired point are displayed as you move the cursor over their tracking paths. You can then define points either on the alignments or at the intersection of two alignments.

***Прим.:*** The ANGBASE system variable defines the base angle of the tracking lines with respect to the current coordinate system.

### 6.16.1 Locking a tracking line

Press the Shift key when a tracking line displays to lock the tracking line. Locked tracking lines display in the color defined by theSnapMarkerColoruser preference (1). Entity snaps (2) or specified points are projected perpendicularly onto a locked tracking line (3). Press the Shift key again to unlock a tracking line.




More options with locked tracking lines are:

- If the Intersection entity snap is active, it is possible to snap to the intersections of a locked tracking line and entities in the drawing.
- Acquire additional tracking lines and snap to the locations where these intersect with the locked tracking line.

### 6.16.2 Using Entity Snap Tracking

1 Launch a drawing command, for example LINE.

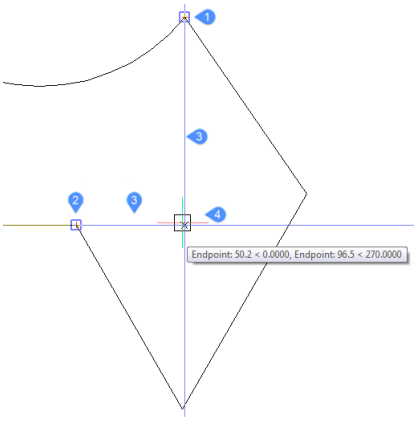
Prompts you: Start of line or [Follow] <Last point>:

2 Hover the cursor over the first tracking point (1).

A small plus sign (+) displays to indicate the tracking point is active.

3 Hover the cursor over the second tracking point (2).

A small plus sign (+) displays to indicate the tracking point is active.




4 Move the cursor close to position 4.

- Tracking lines display from both tracking points (3).
- A cross marker (x) indicates the intersection of the tracking lines.
- Snap markers display at the tracking points.

- An Entity Snap Tracking tooltip at the intersection of the tracking lines shows the angles and distances from the tracking points.

5 Click to accept theEntity Snap Trackingpoint.

## 6.17 User coordinate systems

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. TheELEVATION system variable specifies the distance from the XY-plane of the current coordinate system. IfELEVATION= 0 (default), entities are created in the XY-plane of the current coordinate system.

Every drawing has a fixed coordinate system called the World Coordinate System (WCS).

You can define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (UCS). You can create as many UCS as you want, saving or redefining them to help you construct entities in the three-dimensional space.

When you create a new drawing, the WCS is the current coordinate system by default, which is indicated by the letter W in the UCS icon. When you display a drawing in plan view, you see the coordinate system icon from the top, with the Z-axis directed straight toward you. When you display a three-dimensional drawing in a view other than plan view, the coordinate system icon changes to reflect the new viewpoint.

***Прим.:***

- You cannot delete or modify the WCS.
- When theUCSORTHOsystem variable is On, an orthographic UCS is restored automatically when the related orthographic view is restored by the-VIEWcommand or selected on theLook Fromcontrol.

  An orthographic UCS has its origin at the WCS origin or the UCS referred to by the UCSBASE system variable.

  Orthographic views are: top, bottom, left, right, front, back.
- When multiple viewports exist, theUCSVP(UCS viewports) system variable controls whether the UCS in a viewport is fixed (UCSVP = On) or changes to match the UCS in the current viewport (UCSVP = Off).
- When the UCSFOLLOW system variable is On, the plan view, zoomed to extents, is generated automatically whenever the coordinate system changes.
- ThePLANcommand restores the Plan view (Top view) of the current coordinate system (UCS or WCS).

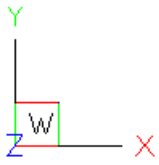
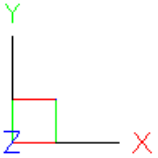
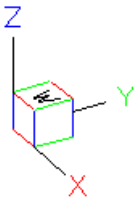
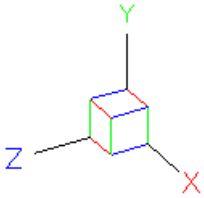
### 6.17.1 Controlling the display of the coordinate system icon

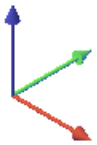
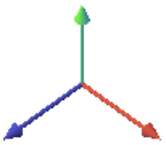
TheUCSICON and UCSICONPOS system variables control the display and location of the UCS icon:

- UCSICON:
  - **Show icon**: Controls whether the icon shows or not.
  - **At origin**: Controls the location of the UCS icon: if On, the icon indicates the origin of the current coordinate system (UCS or WCS). However, if the origin is not within the viewport borders, the UCS icon moves to the corner of the viewport, as defined by theUCSICONPOS

system variable. When the**At Origin**option is not checked, the icon always displays in the corner of the viewport defined byUCSICONPOS.

- UCSICONPOS : Defines the corner of the viewport for the non-origin location of the UCS icon.

| 2D Wireframe visual style | | | |
|---|---|---|---|
|  |  |  |  |
| WCS Top View | UCS Top View | WCS Isometric View | UCS Isometric View |

| Rendered Visual Styles | | | |
|---|---|---|---|
|  |  |  |  |

To help you keep your bearings in the current coordinate system, BricsCAD® displays a coordinate system icon (UCS icon). The visible portions of the axes are the positive directions.

The color of the three axes indicate the orientation of the three-dimensional space:

- X-axis: red
- Y-axis: green
- Z-axis: blue

The color of the axes is controlled by theCOLORX,COLORYandCOLORZsystem variables.

### 6.17.2 Defining a UCS

1 Type **ucs** at the command prompt.

You are prompted:

Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/Move/World] <World>:

2 Specify the origin point of the UCS.

You are prompted: Point of positive X axis <current point>:

3 Specify a point to define the positive X-axis.

The command window reads: Point in X-Y plane with positive Y value <current point>:

4 Specify a point to define the positive Y-axis.

The UCS is defined.

### 6.17.3 Defining and saving a User Coordinate System

1 Launch the EXPUCS command.

The Drawing Explorer - Coordinate Systems dialog box displays.

2 Click the New tool button on the Drawing Explorer dialog.

The BricsCAD Explore dialog closes.

You are prompted: Current/Entity/Origin/View/X/Y/Z/ZAxis;3point

3 Press Enter to accept the default option (define the UCS using 3 points).

You are prompted: New origin <current origin>:

4 Specify the origin point of the UCS.

You are prompted: Point of positive X axis <current point>:

5 Specify a point to define the positive X-axis.

The command window reads: Point in X-Y plane with positive Y value <current point>:

6 Specify a point to define the positive Y-axis.

The UCS is defined.

The Drawing Explorer dialog reopens.

7 Click in the UCS Name field of the newly defined UCS to replace the <NewUCS> default name.

8 Close the Drawing Explorer dialog.

### 6.17.4 Restoring a saved UCS

1 Launch the EXPUCS command.

The Drawing Explorer - Coordinate Systems dialog box displays.

2 Click the blank tile in front of the UCS name to make it current.

3 Close the Drawing Explorer dialog box.

### 6.17.5 Restoring the WCS

Type **ucs** in the Command line, then press Enter twice (= choosing the default option of the UCS command).

## 6.18 Dynamic UCS

With the Dynamic UCS feature enabled, the coordinate system is aligned automatically with the entity under the cursor.

TheUCSDETECTsystem variable controls the supported entity types:

- 1: faces of 3d Solids (the default)

- 2 :other entity types: point, line, polyline, 2D polyline, 3D polyline, ray, xline, arc, circle, ellipse, spline, text, mtext, solid, 3Dface, trace, block insert, viewport, mline, leader, mleader, hatch, helix, camera, light, section, shape, pdf underlay and image.

Right click the DUCS field in the status bar to select the supported entity types.

This feature is controlled by user preferences.

- RedHilite_DUCSLocked_Face_Colorcontrols the highlight color for the face to which the Dynamic UCS has been locked by pressing the Shift key.
- RedHilite_DUCSLocked_Face_Alphacontrols the highlight transparency for the face to which the Dynamic UCS has been locked by pressing the Shift key.

***Прим.:*** It is recommended to set UCSFOLLOW = Off when using the dynamic UCS feature.

### 6.18.1 Toggling the dynamic UCS

Do one of the following:

- Press the**F6**function key.
- Click the**DUCS**field in thestatus bar.

### 6.18.2 Using the Dynamic UCS

1 With the Dynamic UCS feature active, launch an entity creation command, for example Draw.

2 Move the cursor over an entity or 3D solid face.

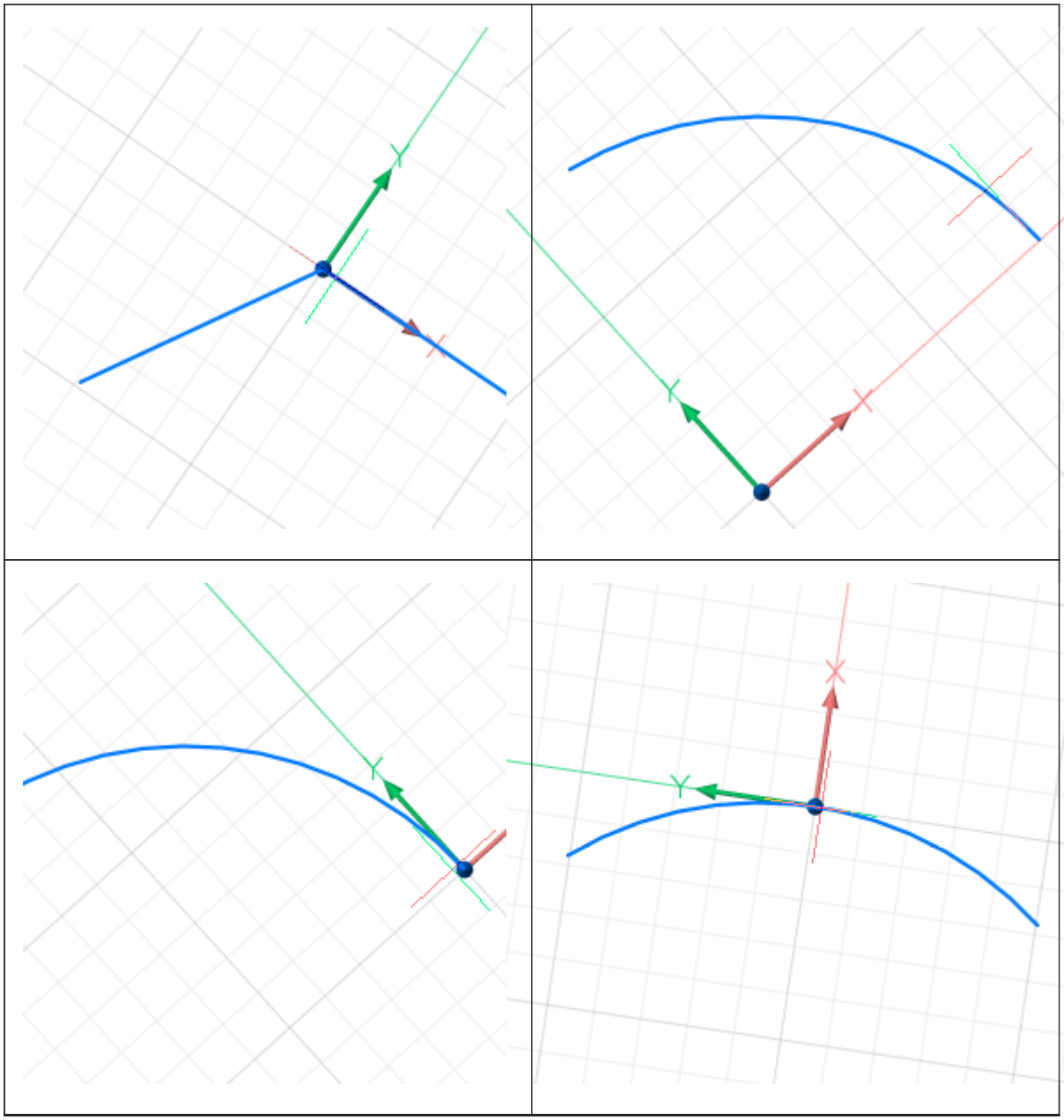
The entity or face highlights.

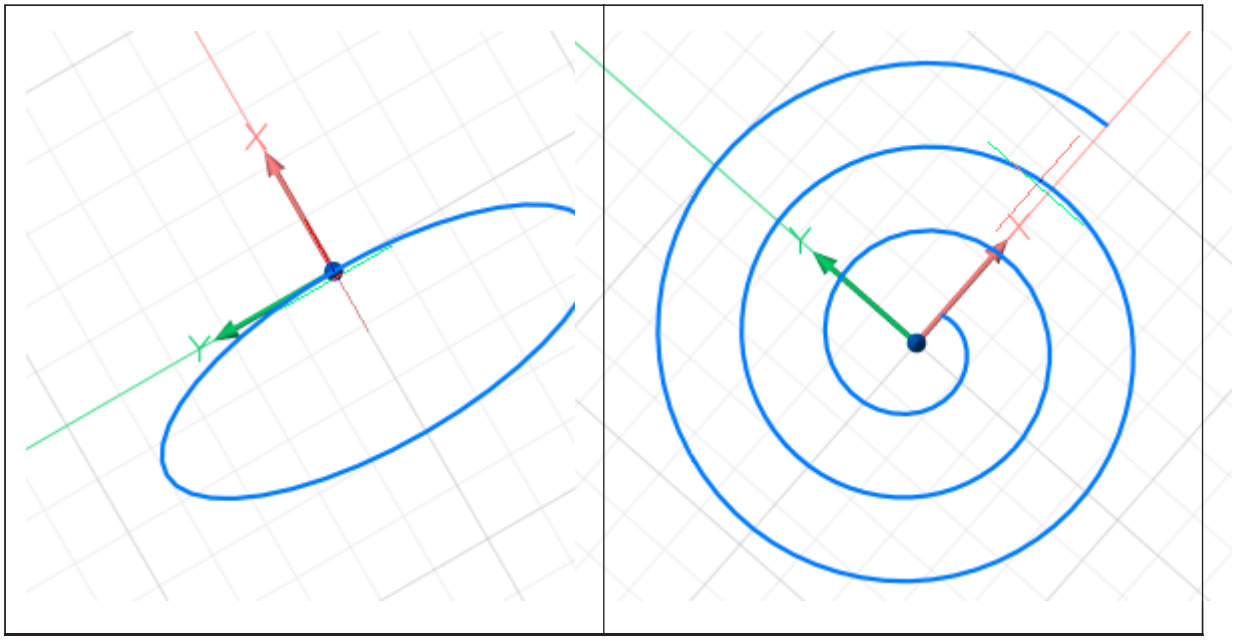
If GRIDMODE is On, the grid is aligned with the hovered entity or face.

3 IfUCSICON= 3 (On at Origin), the UCS icon displays.

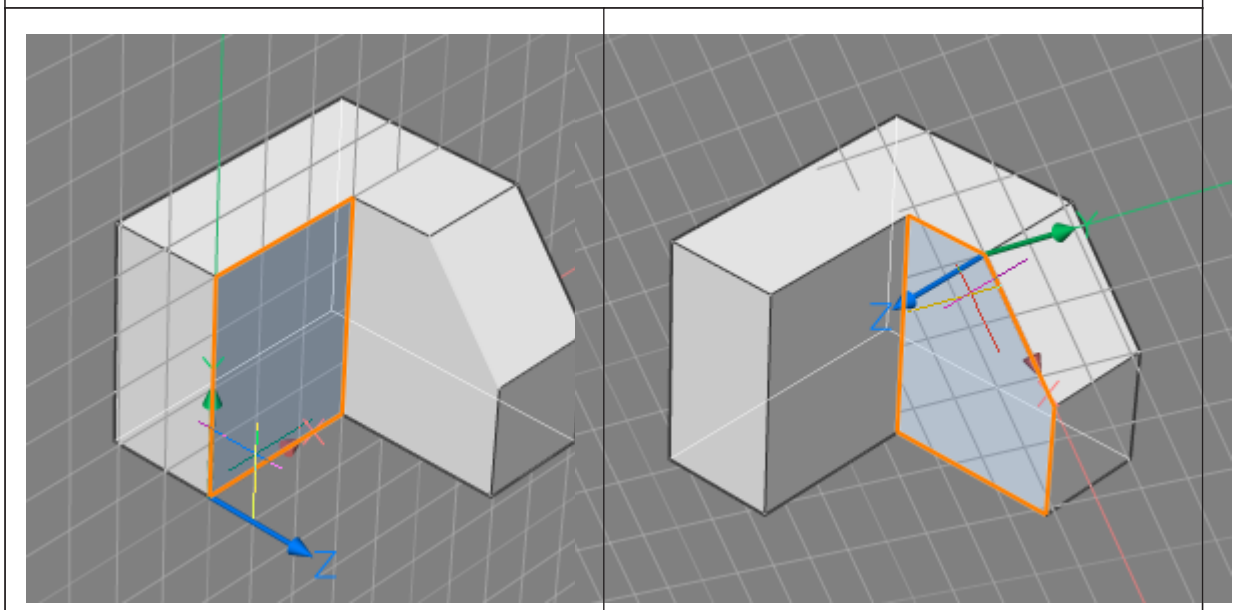
4

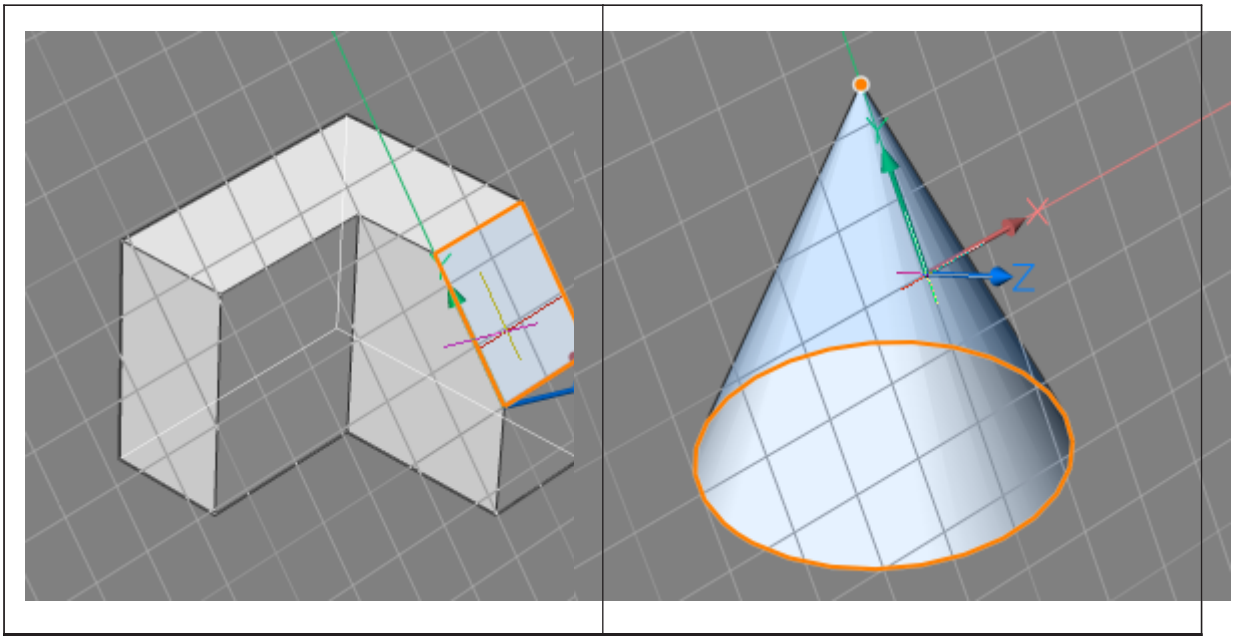
| Dynamic UCS on other entities |
|---|





Dynamic UCS on 3D solid faces






The dynamic UCS is oriented depending on how the cursor touches the hovered entity or face, thus providing users a way to manipulate the orientation of the DUCS. On 3D solid faces the origin of the DUCS is set to the start point of the solid edge and the X-axis is chosen co-directional to the tangent at the start point of the edge. On curved faces, the X-axis is tangent to the face, the Y-axis lies in the face and the Z-axis is perpendicular to the face.

5 (option) Tap the Shift key to lock the DUCS, which allows you to start drawing outside the hovered entity or face. Tap the Shift key again to unlock the DUCS.

6 Execute the entity creation command.

***Прим.:*** When bitcode 2 of the OSOPTIONS system variable is selected, entity snap ignores negative Z-values.

7 When the command concludes, the current coordinate system (WCS or UCS) is restored.

## 6.19 Measuring

The DIST command reports the distance and angle between two points.

The AREA command finds the area and perimeter (or length) of 2D entities.

The MASSPROP command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for **mass properties**).

The ID command reports the x,y,z coordinates of picked points.

The LIST command lists the properties of selected entities.

***Прим.:*** BricsCAD® reports the results in the Command line or in the Prompt History window when the Command line is closed. Depending on the number of prompt history lines in the Command line, it can be necessary to press the F2 function key to open the Prompt History window. Press the F2 function key again to close the Prompt History window.

### 6.19.1 Measuring distances

To measure a distance between 2 points:

1 Launch the DIST command.

Prompts you: Starting point for distance.

2 Identify the first point.

Prompts you: Enter second point or [Multiple points]:

3 Hover over the second point using an appropriate entity snap mode.

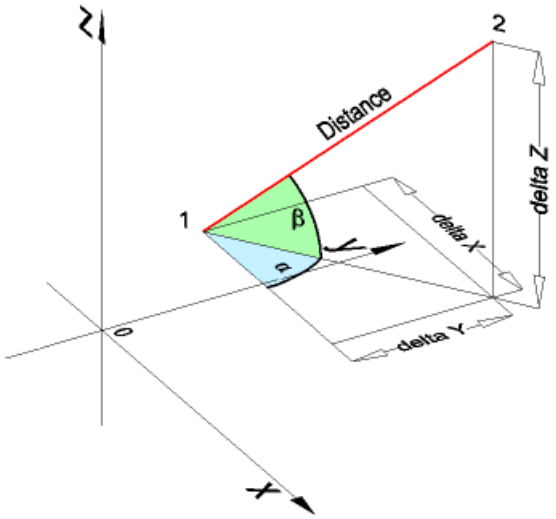
When Dynamic Input (DYN) is On, the distance displays in thedynamic dimensionfield.

4 Click to confirm the second point.

BricsCAD® prints the results in the Command line:

Distance, Angle in XY Plane (α), Angle from XY Plane (β)

Delta X, Delta Y, Delta Z




The values are in drawing units and with respect to the currentcoordinate system.

To measure a distance between multiple points:

1 Launch the DIST command.

Prompts you: Starting point for distance.

2 Identify the first point.

Prompts you: Enter second point or [Multiple points]:

3 Choose the Multiple points option.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

4 Identify the second point.

BricsCAD® displays the distance between the two points.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

5 Identify the next point.

BricsCAD® displays the total distance between the points.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

6 Keep adding points, then press Enter or right click to conclude.

### 6.19.2 Measuring lengths

To measure the length of single linear entity, click the entity then read the**Length**field in theProperties panel.

To combine the lengths of a series of linear entities:

1 Launch the AREA command.

Prompts you:

Select first point [find area of one Entity/Add areas together/Subtract areas]:

2 Choose the**Add areas together**option.

Prompts you: Draw first point to add to current area or [add Entities to area/Subtract from area]:

3 Choose the **ad Entities** option.

Prompts you: Select entities to add to area:

4 Select the first linear entity.

BricsCAD® reports in the Command line:

Area = xx, Length = xx

Total length = xx

Total area = xx

Prompts you: Select entities to add to area:

5 Select the second linear entity.

BricsCAD® reports in the Command line:

Area = xx, Length = xx

Total length = xx

Total area = xx

Prompts you: Select entities to add to area:

6 Keep selecting linear entities or press Enter or right click to stop.

***Прим.:*** You can combine the lengths of lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.

### 6.19.3 Measuring areas

To measure the area and length of single closed entity, click the entity then read the**Area**and**Length**fields in theProperties panel.

To measure areas and perimeters by specifying points in a drawing:

1 Launch the AREA command.

Prompts you:

Select first point [find area of one Entity/Add areas together/Subtract areas]:

2 Click the first point.

Prompts you: Next point.

3 Click the second point.

Prompts you: Next point.

A dashed outline displays dynamically.

4 Keep adding points, then press Enter or right click.

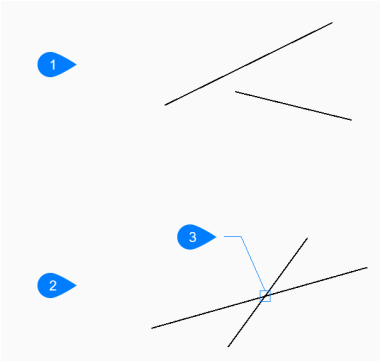
BricsCAD® reports in the command line:

Area = xx, Perimeter = xx

## 6.20 Entity snaps overview

### 6.20.1 Apparent intersection entity snap

The Apparent intersection entity snap enables the cursor to snap to the intersections of entities, even when they only appear to intersect in 3D space.




1 In one 3D viewpoint, the lines do not intersect.

2 In another 3D viewpoint, the lines appear intersect.

3 Apparent intersection.

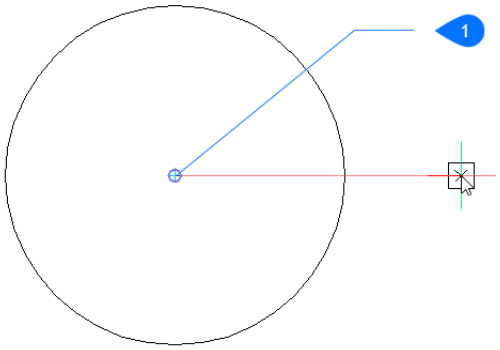
### 6.20.2 Center entity snap

The Center entity snap enables the cursor to snap to the center point of circular or elliptical entities.

Including:

- Circles
- Arcs
- Polyline arcs
- Ellipses
- Elliptical arcs
- Circular 3D faces

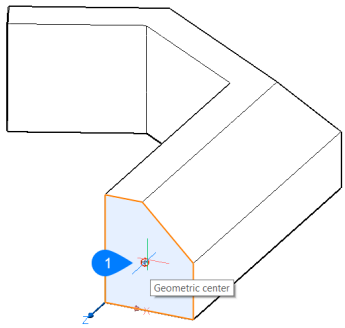
- Elliptical 3D faces




1 Center of circle.

### 6.20.3 Center of a Face 3D entity snap

The Center of a Face 3D entity snap enables the cursor to snap to the geometric center of a 3D face.



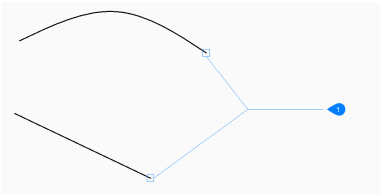

1 Geometric center entity snap.

### 6.20.4 Объектная привязка Конечная точка

Объектная привязка Конечная точка позволяет курсору привязываться к конечным точкам объектов.

Поддерживаются:

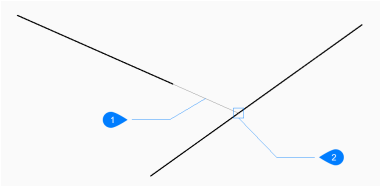
- Линии
- Дуги
- Сегменты полилиний
- Незамкнутые сплайны
- 3D ребра



1 Объектная привязка Конечная точка.

### 6.20.5 Extension entity snap

The Extension entity snap enables the cursor to snap to the intersection of one or two extended entities.



1 Extension

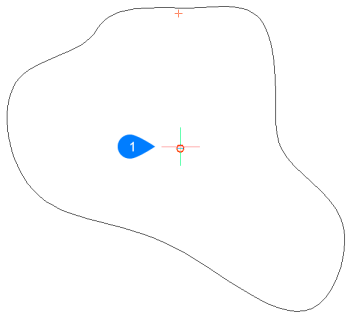
2 Extension entity snap.

### 6.20.6 Geometric center entity snap

The Geometric center entity snap enables the cursor to snap to the centroid of closed entities.

Including:

- Polylines
- Splines
- Regions
- Planar 3D faces



1 Geometric center entity snap.

### 6.20.7 Insertion entity snap

The Insertion entity snap enables the cursor to snap to the insertion points of entities.

Including:

- Text
- Mtext

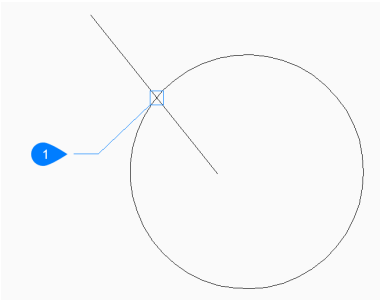
- Tables
- Attribute text
- Blocks
- External References
- PDF References
- Raster images




1 Insertion entity snap.

### 6.20.8 Объектная привязка Пересечение

Объектная привязка Пересечение позволяет курсору привязываться к физическому пересечению пересекающихся друг с другом объектов.



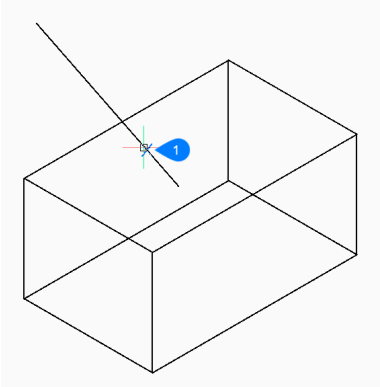

1 Объектная привязка Пересечение.

### 6.20.9 Объектная привязка 3D пересечение

Объектная привязка Пересечение позволяет курсору привязываться к физическому пересечению пересекающихся друг с другом объектов.

Поддерживаются:

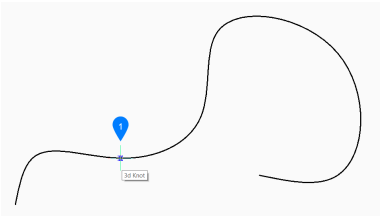
- Линии
- Сегменты полилиний
- 3D ребра
- Линии полярной трассировки
- Линии трассировки объектной привязки




1 Объектная привязка 3D пересечение.

### 6.20.10 Knot 3D entity snap

The Knot 3D entity snap enables the cursor to snap to knots on splines.



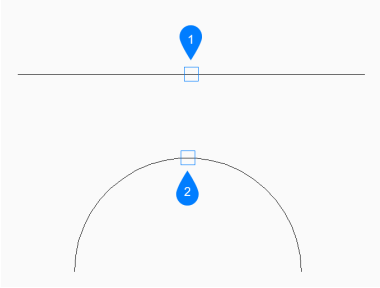

1 Knot 3D entity snap.

### 6.20.11 Midpoint entity snap

The Midpoint entity snap enables the cursor to snap to the midpoint of entities.

Including:

- Lines
- Polyline segments
- Arcs
- Elliptical Arcs
- Splines
- 3D edges

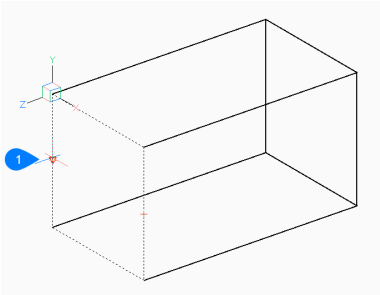



1 Midpoint (of line).

2 Midpoint (of arc).

### 6.20.12 Midpoint 3D entity snap

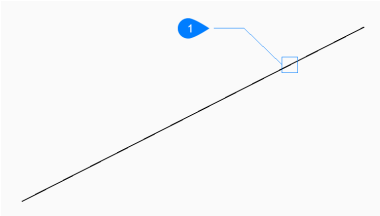
The Midpoint 3D entity snap enables the cursor to snap to the midpoint of 3D edges.




1 Midpoint 3D entity snap.

### 6.20.13 Nearest entity snap

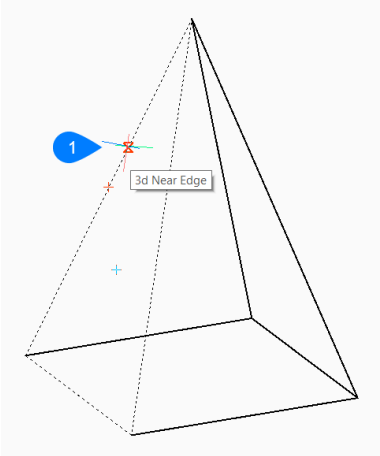
The Nearest entity snap enables the cursor to snap to 2D or 3D geometry nearest to the cursor.




1 Nearest entity snap.

### 6.20.14 Nearest 3D entity snap

The Nearest 3D entity snap enables the cursor to snap to a point on a 3D entity that is nearest to the cursor.



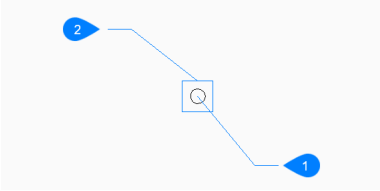

1 Nearest 3D entity snap.

### 6.20.15 Node entity snap

The Node entity snap enables the cursor to snap to point entities and the insertion points of entities.

Including:

- Text
- Mtext
- Attributes
- Tables

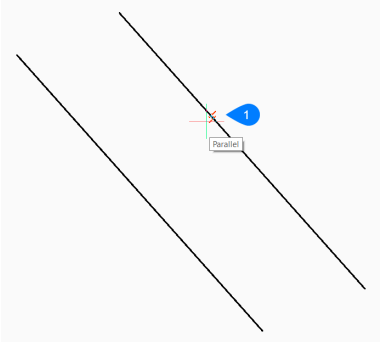



1 Point.

2 Node entity snap.

### 6.20.16 Parallel entity snap

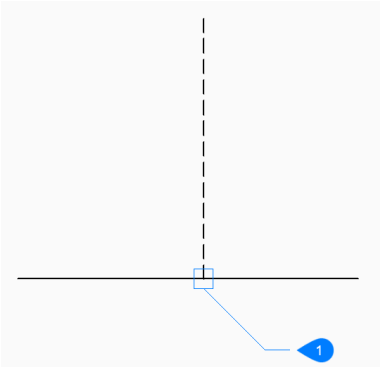
The Parallel entity snap enables the cursor to snap to a point parallel to an existing entity.




1 Parallel entity snap.

### 6.20.17 Perpendicular entity snap

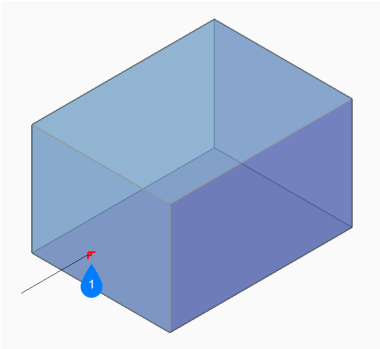
The Perpendicular entity snap enables the cursor to snap to a point perpendicular to an existing entity.




1 Perpendicular (to entity).

### 6.20.18 Perpendicular on a face 3D entity snap

The Perpendicular on a face 3D entity snap enables the cursor to snap to a point perpendicular to a 3D face.



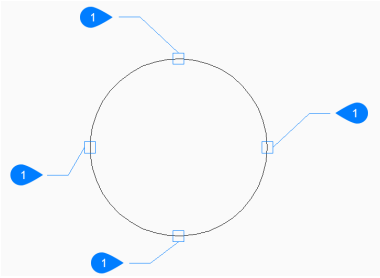

1 Perpendicular to a face 3D entity snap.

### 6.20.19 Quadrant entity snap

The Quadrant entity snap enables the cursor to snap to the quadrant points of circular or elliptical entities.

Including:

- Circles
- Arcs
- Polyline arcs
- Ellipses
- Elliptical arcs
- Circular 3D faces
- Elliptical 3D faces




1 Quadrant.

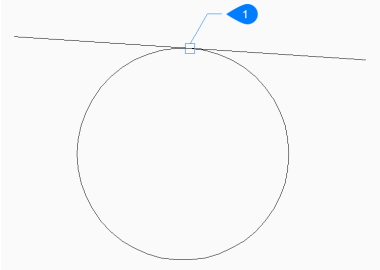
### 6.20.20 Tangent entity snap

The Tangent entity snap enables the cursor to snap to the tangent points of curved entities.

Including:

- Circles
- Arcs
- Polyline arcs

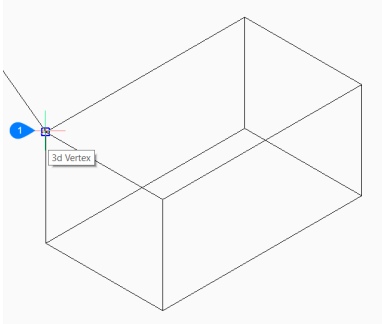
- Ellipses
- Elliptical arcs
- Splines
- Circular 3D faces
- Elliptical 3D faces




1 Point of tangency.

### 6.20.21 Vertex 3D entity snap

The Vertex 3D entity snap enables the cursor to snap to the closest vertex of a 3D entity.




1 Vertex 3D entity snap.

# 7. Просмотр чертежа

## 7.1 View commands overview

BricsCAD provides various tools and commands to control the display and view orientation of your drawing.

These tools can be found in the **View** menu, toolbar or ribbon tab.

| Icon | Tool Name | Command | Description |
|---|---|---|---|
|  | Redraw | REDRAW | Refreshes the screen display in the current viewport. |
|  | Redraw All | REDRAWALL | Refreshes the screen display in all open viewports. |
|  | Regenerate | REGEN | Recalculates the screen display in the current viewport. |
|  | Regenerate All | REGENALL | Recalculates the screen display in all open viewports. |
|  | Update Fields | UPDATEFIELD | Updates the selected fields. |
|  | Real Time Pan | RTPAN | Pans the view dynamically. |
|  | Pan | PAN | Pans the view. |
|  | Real Time Zoom | RTZOOM | Zooms in/out dynamically. |
|  | Zoom In | ZOOM + *I(*)* | Zooms in on the center of the window by a factor of 2. |
|  | Zoom Out | ZOOM + *O(*)* | Zooms out from the center of the window by a factor of 1/2. |
|  | Zoom Extents | ZOOM + *E(*)* | Displays all the entities in the drawing (referred to as the drawing extents). |
|  | Zoom Window | ZOOM + *W(*)* | You are prompted to pick two corners of a box on the existing view in order to enlarge that area to fill the display. |
|  | Zoom Previous | ZOOM + *P(*)* | Restores the displayed view prior to the current one. |

| Icon | Tool Name | Command | Description |
|---|---|---|---|
|  | Zoom All | ZOOM + *A(*)* | Displays the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two). |
|  | Zoom Left | ZOOM + *L(*)* | Pick the lower left corner of the next view, then specify the magnification factor or the view height. |
|  | Zoom Center | ZOOM + *C(*)* | Pick the center of the next view, then specify the magnification factor or the view height. |
|  | Zoom Right | ZOOM + *R(*)* | Pick the upper right corner of the next view, then specify the magnification factor or the view height. |
| | Look Around | RTLOOK | Look around in a 3D scene. |
| | Look Up and Down | RTUPDOWN | Move up / down and left / right in a 3D scene. |
| | Walk | RTWALK | Walk left / right and forward / backward in a 3D scene. |
|  | Real-Time Constrained Sphere | RTROT | Rotates the view dynamically in 3D.<br>Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the world Z-axis.<br>Moving the mouse vertically (parallel to the screen y-axis) rotates the screen x-axis. |
| | Real-Time Sphere Center | RTROTCTR | Rotates the view dynamically in 3D about a user-defined center point. Behaves like RTROT. |
|  | Real-Time Free Sphere | RTROTF | Rotates the view dynamically in 3D.<br>Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the screen Y-axis.<br>Moving the mouse vertically (parallel to the screen y-axis) rotates the view about the screen X-axis. |
|  | Real-Time X | RTROTX | Rotates the view dynamically about the screen X-axis. |
|  | Real-Time Y | RTROTY | Rotates the view dynamically about the screen Y-axis. |
|  | Real-Time Z | RTROTZ | Rotates the view dynamically about the screen Z-axis. |

| Icon | Tool Name | Command | Description |
|---|---|---|---|
| | Set View Point | DDVPOINT | Opens the **Preset Viewpoints** dialog. |
| | Plan View | PLAN | Restores the plan view with respect to the WCS, the current UCS or a saved UCS. |
| | Define View | DVIEW | Defines parallel and visual perspective views. |
| | Save/Restore View | VIEW | Saves and restores named views. |
| | Viewports | VPORTS | Manages viewports in model space. |
| | Paper Space Views | MVIEW | Creates viewports in paper space. |

(*) <command> <option>: type **zoom** followed by Enter, then type the command option letter, followed by Enter

## 7.2 Layer utilities

The layer utility commands help you to easily manipulate the **On/Off**, **Thaw/Freeze** and **Unlocked/Locked** layer properties by selecting entities. You can then save the result in a Layer State.

The layer utility commands sit on the **Layers** toolbar and on the **Home / Layers** ribbon panel in the **Drafting** and **Modeling** workspaces.

| Command | Icon | Description |
|---|---|---|
| LAYON | | Turns on all layers that have been turned off. |
| LAYOFF | | Turns off layers associated with selected entities. |
| LAYFRZ | | Freezes layers associated with selected entities. |
| LAYTHW | | Thaws all frozen layers. |
| LAYLCK | | Locks layers associated with selected entities. |
| LAYULK | | Unlocks layers associated with selected entities. |

| Command | Icon | Description |
|---|---|---|
| LAYISO |  | Isolates layers associated with selected entities. If no entity on the current layer is selected, the layer of the entity that is selected first temporarily becomes the current layer until the LAYUNISO command is executed. |
| LAYUNISO |  | Restores layers isolated by the LAYISO command. |

## 7.3 Hide and isolate entities

| Command | Icon | Description |
|---|---|---|
| HIDEOBJECTS |  | Hides the selected entities. |
| ISOLATEOBJECTS |  | Displays the selected entities only. |
| UNISOLATEOBJECTS |  | Brings back all previously hidden entities. |

The tools to launch these commands can be found on:

- the **Hide/Show Entities** and **Access 3D** toolbars.
- the **Select** Quad tab.
- The **Home / Layers** ribbon panel in the **Drafting** and **Modeling** workspaces.

Show, Hide and Isolate entities is also available in the right-click context menu in the **Structure** panel.

***Прим.:*** The OBJECTISOLATIONMODE system variable controls whether hiding and isolating entities is saved or not.

## 7.4 Redrawing and regenerating a drawing

The screen display of a drawing is really a simplified version of the drawing database. From time to time, it is necessary to synchronize the screen display and the drawing database.

The following commands are available:

- The REGEN command (short for "regenerate") regenerates the current viewport.
- The REGENALL command (short for "regenerate all") regenerates all viewports.
- The REGENAUTO command controls when BricsCAD automatically regenerates the drawing.
- The REDRAW command redraws the current viewport to clean it up.
- The REDRAWALL command redraws all viewports to clean them up.
- The REGENMODE system variable controls whether BricsCAD regenerates the drawing automatically. You can change the status of REGENMODE through the REGENAUTO command. When REGENMODE = ON, BricsCAD regenerates the display automatically, but in a few cases a forced regeneration of the drawing might still be necessary.

Take care not to confuse the REGEN command with the REDRAW command, which simply repaints the screen, without attempting to synchronize the screen display with the drawing database.

## 7.5 Panning

The PAN and RTPAN commands move the drawing in any direction: horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space. The only change is the portion of the drawing being displayed.

### 7.5.1 Using the pan command

1 Launch the PAN or RTPAN command.

1 The shape of the cursor changes to a hand.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.
- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

## 7.6 Zooming

The **ZOOM** command visually changes the size of the drawing within the current viewport.

The RT**ZOOM** command zooms the drawing in real-time.

You can change the magnification of your drawing at any time by zooming. The cursor changes to a magnifying glass when aN RTZOOM tool is active. Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail. Changing the magnification of the drawing affects only the way the drawing is displayed; it has no effect on the dimensions of the entities in your drawing.

### 7.6.1 Real-time zooming

1 Launch the RTZOOM command.

1 The shape of the cursor changes to a magnifying glass.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.

- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

### 7.6.2 Zooming to extents

Do one of the following:

- Double-click the middle mouse button (scroll wheel).
- Launch the ZOOM command, then type E and press Enter.

### 7.6.3 Dynamic zooming

Pans and zooms using a rectangular view box, representing the current viewport. The command first zooms out to display the entire drawing. You can then control the size of the view box dynamically and position the view box on the part of the drawing you want to display in the current viewport.

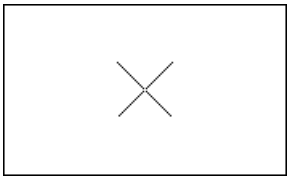
1 Launch the ZOOM command.

2 Choose the **Dynamic** option.

The drawing zooms out to extents.

A dashed rectangle indicates the area of the previous view.

A view box of the same size as the dashed rectangle is attached to the cursor.



3 (option) To resize the view box:

- Click, then move the cursor to resize.



- Click again to stop resizing.

4 Position the view box over the drawing area you want to enlarge, then right-click.

5 The drawing is zoomed in on the selected area.

## 7.7 View manipulation using the mouse and keyboard

In BricsCAD you can navigate through a 3D model using the mouse and keyboard shortcuts.

In perspective mode you can walk inside the 3D model. In isometric mode you can only zoom in very close, but not go inside.

| Mouse or Keyboard | Action | Result |
|---|---|---|
| Mouse wheel | scroll | Zoom in / out.<br>The incremental change in zoom with each mouse-wheel action is controlled through the **Zoom Factor** variable (ZOOMFACTOR system variable). Default = 60. Values between 3 and 100 are accepted.<br>When zooming in, the incremental step decreases gradually, allowing you to focus on a particular detail easily.<br>In a 3D model, the entity under the cursor has the zoom focus.<br>The ZOOMWHEEL system variable controls the zoom direction:<br>0 or OFF: scrolling forwards zooms in, backwards zooms out.<br>1 or ON: scrolling forwards zooms out, backwards zooms in. |
| Middle mouse button or wheel* | press and hold while moving the mouse | Real-time pan (RTPAN command) |
| Middle mouse button or wheel* | double-click | Zoom extents |
| Ctrl + Shift + Left mouse button | press and hold while moving the mouse | Real-time zoom in / out (RTZOOM command) |
| Ctrl + Shift + Right mouse button | press and hold while moving the mouse | Constrained real-time pan (RTPAN command). Release the Shift key for free panning. |
| Ctrl + Shift + Middle Button or wheel | press and hold while moving the mouse | Real-time sphere (RTROTF command) |
| Shift + Middle Button or wheel | press and hold while moving the mouse | Real-time constrained sphere (RTROT command) |
| Ctrl + Right Button | press and hold while moving the mouse | Real-time rotate about the screen Z-axis (RTROTZ command) |

| Mouse or Keyboard | Action | Result |
|---|---|---|
| Walk through navigation | | |
| Alt + Left Button | press and hold while moving the mouse | Walk left / right and forward / backward (RTWALK command) |
| Alt + Middle Button | press and hold while moving the mouse | Move up / down and left / right (RTUPDOWN command) |
| Ctrl + Middle Button | press and hold while moving the mouse | Look around (RTLOOK command) RTWALK command |
| Ctrl + Arrow Keys | press the arrow keys while holding the Ctrl key | Walk left / right and forward / backward () |
| Ctrl + Shift + Arrow Keys | press the arrow keys while holding the Alt + Ctrl keys | Look up / down and left / right (RTUPDOWN command) |
| Ctrl + Home | | Sets the view direction horizontal |
| Alt + Home | | Moves target point to center of scene |
| Alt + plus/minus | | Increase / decrease movement speed |
| Ctrl + plus/minus | | Increase / decrease rotation speed |

* On condition the **Middle Button Pan** (MBUTTONPAN) system variable is set **ON**.

## 7.8 View rotation

The following command are available for view rotation operations:

- The **RTROT** (Real-Time Constrained Sphere) command rotates 3D drawings in real-time.
- The **RTROTCTR** (Real-time Sphere Center) command freely rotates 3D drawings in real-time about a user-defined center point.
- The **RTROTF** (Real-time Free Sphere) command freely rotates 3D drawings in real-time.

- The **RTROTX, RTROTY** and **RTROTZ** commands rotates 3D drawings in real-time about the x-, y- or z-screen axis.
- The **DDVPOINT** command sets 3D viewpoints or plan view, through a dialog box (short for "dynamic dialog view point").
- The **PLAN** command restores the plan (top) view.
- The **VPOINT** command changes the 3D viewpoint.

***Прим.:***

- If the **CONTINUOUSMOTION system** variable is set, the view rotation continues until you press the Esc key to conclude the command.
- View rotation commands should not be used when working in 2D. Use the PLAN command or the LookFrom control to restore top view.

### 7.8.1 Rotating a view freely

1 Launch the RTROT, RTROTF, RTROTX, RTROTY or RTROTZ command.

1 The shape of the cursor changes according to the selected command.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.
- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

### 7.8.2 Using Preset Viewpoints

***Прим.:*** Switch off Perspective mode before starting the command.

1 Launch the DDVPOINT command.

1 The **Set Viewpoint** dialog box appears.

2 Click the **Angle Precision** button to set the display mode for the **Set Viewpoint** dialog box.

The options are: **45°**, **15°** or **5°**.

The **Horizontal** and **Vertical** select panes are divided accordingly.

3 Click an angle in the **Horizontal** and **Vertical** select panes.

The view orientation is updated.

4 (option) Click the **Plan** button to restore the plan (top) view.

5 (option) Repeatedly click the **Previous** button to restore previous viewpoints.

6 To conclude the command:
   - Click the **OK** button to keep the current view.
   - Click the **Cancel** button to restore the original view.

## 7.9 The LookFrom control

The **LookFrom** control allows you to select a number of preset views such as orthographic views and isometric views. By default, the **LookFrom** control is displayed in the top right corner of the graphic screen.

The NAVVCUBEDISPLAY system variable sets the display of the LookFrom control.

### 7.9.1 Context menu options

When you right-click the LookFrom control, a context menu appears. The options are:

- **UCS**: sets the current coordinate system.
  - **World**: restores the WCS (World Coordinate System)
  - **New**: runs the UCS command.
  - **UCS Manager...**: runs the EXPUCS command; displays the **Drawing Explorer / Coordinate Systems** dialog box.
- **View**:
  - **New**: runs the -VIEW command.
  - **View Manager...**: runs the VIEW command; displays the **Drawing Explorer / Views** dialog box.
- **Visual Style**: Allows you to set the visual style.
- **Isometric Mode**: Sets the isometric mode.
- **Twist Mode**: Sets the Twist mode.
- **WCS**: LookFrom uses the WCS as reference coordinate system.
- **UCS**: LookFrom uses the current coordinate system (WCS or UCS) as reference coordinate system.
- **Perspective**: Sets the perspective view mode.
- **Location**: Sets the location of the LookFrom control.
- **Dismiss**: Closes the LookFrom control.

  Edit the NAVVCUBEDISPLAY system variable to restore the LookFrom control.

### 7.9.2 LookFrom control settings

The behavior of the **LookFrom** control depends on a number of system variables and user preferences.

To access the LookFrom settings:

1 Right-click the LookFrom control and choose **LookFrom Settings...** in the context menu.

2 Run the LOOKFROM command and choose the **Settings** option.

### 7.9.3 Setting the LookFrom mode

The LookFrom control runs in one of two modes:

- Isometric mode for working in 3D models.
- Twist mode for 2D drafting.

To set the LookFrom mode:

1 Right-click the LookFrom control.

2 Select **Isometric Mode** or **Twist Mode** in the context menu.

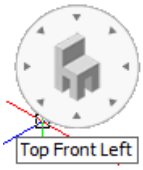
### 7.9.4 Using the LookFrom control in Isometric mode

1 Move the cursor to the **LookFrom** control.

The **LookFrom** control becomes active.

2 Hover the cursor over a view orientation.

3 The chair in the center of the LookFrom control rotates according to the view orientation.




4 (option) Hold down the Ctrl key to look bottom-up.

5 Click to confirm.

6 The view is rotated.

### 7.9.5 Using the LookFrom control in Twist mode

In **Twist** mode, the **LookFrom** control rotates the view around the screen Z-axis.

1 Move the cursor to the **LookFrom** control.

The **LookFrom** control becomes active.

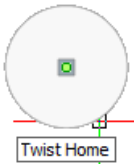
2 Move the cursor to the edge of the **LookFrom** control, on the left half for counterclockwise, on the right half for clockwise rotation. 15°, 30°,45°, 60°, 90°, 120°, 135° and 180° angles are available.




3 Click to rotate the view.

4 (option) Without moving the mouse, click again to rotate over the same angle again.

5 (option) Move the cursor to the center of the **LookFrom** control, then click to restore an unrotated view.



### 7.9.6 View transition options

The following system variables control the behavior of view transitions:

- VTENABLE: determines whether animated view transitions are enabled for zoom/pan and/or for view rotation operations.
- VTDURATION: determines the duration of view transition animations in milliseconds.
- VTFPS: defines the minimum frame rate required to allow animation. That is 7 by default, which means that the redraw time should take less than 143 (=1000/7) milliseconds. If the computer is not capable to redraw the view fast enough, no animation will be available.

## 7.10 Layer states

Layer states let you save and restore configurations of layer properties. You might want a layer to be displayed in blue sometimes and green at other times, or you need some layers to be off or frozen or locked when editing a specific part of a drawing. After spending a lot of time adjusting all layer settings you can use layer states to save your settings. You can save multiple layer states in a drawing. Layer states can be copied between drawings and you can save a layer state to an external file, which can then be imported in another drawing.

To easily create layer states, the Layer Utility commands might be useful.

The LAYERSTATE command opens the **Drawing Explorer / Layer States** dialog box.

### 7.10.1 Saving the current layer state

1 Click the **New** button on the **Drawing Explorer / Layer States** dialog box.

A new layer state is created.

2 Type a name for the layer state, replacing the **NewLayerState1** default name.

3 (option) When in **Detail View**, type a description in the **Description** field.

4 Select the **Layer Properties to restore**.

Click the tick box to select a layer property, click again to remove a property.

5 Select a **Restore option**.

6 When in paper space, tick the **Apply properties a viewport overrides** option.

***Прим.:*** In a new layer state, all layer properties are selected to be restored by default.

### 7.10.2 Editing layer states

1 (option) Click the **Overwrite** button to overwrite the layer state with the current layer settings.

2 (option) Rename the layer state.

3 (option) Edit the layer state description.

4 (option) Edit the **Layer Properties to restore**.

Click the tick box to select a layer property, click again to remove a property.

### 7.10.3 Restoring layer states

1 Do one of the following:
   - Click the **Restore** button on the **Drawing Explorer / Layer States** dialog box.
   - Right-click, then choose **Restore** in the context menu.

2 Click the **Regen** button, unless the **Regen Auto** button is pressed.

***Прим.:***

- The restore option of the layer state applies to layers which are added after the selected layer state was created. The options are:
   - Turn Off.
   - Freeze
   - Do nothing.
- Layer states can be restored using the **Layer State** control, which you can add to a ribbon panel or a toolbar.

### 7.10.4 Copying layer states between open drawings

1 Open both, the source and the target drawing.

2 Select the source drawing in the **Open Drawings** list on the **Drawing Explorer / Layer States** dialog box.

3 Select the layer state(s) you want to copy.

4 Press and hold the Ctrl key to select multiple layer states.

5 Hold down the left mouse button, then drag-and-drop the selection onto the target drawing in the **Open Drawings** list on the **Drawing Explorer / Layer States** dialog box.

The selected layer states are copied to the target drawing.

### 7.10.5 Exporting layer states

1 Select the layer state on the **Drawing Explorer / Layer States** dialog box.

2 Click the **Export** button.

The **Export Layer States** dialog box appears.

3 In the **File Name** field, type a name for the layer state file.

4 Select a destination folder.

5 Click the **Save** button.

### 7.10.6 Importing layer states

1 Click the **Import** button on the **Drawing Explorer / Layer States** dialog box.

The **Import Layer States** dialog box appears.

2 Select the layer state file to import.

3 Click the **Open** button.

The layer state is imported.

## 7.11 Named views

You can save the view in the current window as a named view. Saved views can be restored at any time.

A Background can be assigned to a named view, which will be used when the view is rendered.

### 7.11.1 Exploring views

1 Launch the VIEW command.

1 The **Drawing Explorer / Views** dialog box appears.

2 (option) Click the blank tile in front of the **View Name** to restore the view.

The current view is marked.

3 (option) Click the **New** button in the **Drawing Explorer** toolbar to save the current view (launches the -VIEW command).

4 (option) Click the **Delete** button in the **Drawing Explorer** toolbar to delete the selected view.

5 (option) Click the **View Name** , then right-click and choose **Rename** in the context menu.

Type a new name for the saved view.

6 (option) Check the **Perspective** property to make the view a visual perspective view.

If not checked the view is a parallel perspective view.

7 (option) Click the **Background** column of a view to choose a background. The background definition of the view is applied during rendering.

### 7.11.2 Saving a view

1 Do one of the following:

- Launch the -VIEW command.
- Click the **New** button in the **Drawing Explorer / Views** toolbar.

2 Choose the **Save** option.

-VIEW command workflow:

- **You are prompted**: Save view as:
- Type a name for the view in the Command line, then press Enter.

Drawing Explorer workflow:

- The **Drawing Explorer / Views** dialog box appears.
- Type a name to replace the NewView1 default name.

***Прим.:*** If you type a name in the Command line that is already used, the current view is saved, replacing the previously saved view. There is no warning when overwriting a saved view.

### 7.11.3 Restoring a view

In the **Drawing Explorer / Views** dialog box, click the **Current** field of the view.

### 7.11.4 Editing the view background

1 Select the view in the **Drawing Explorer - Views** dialog box.

2 Select the **Background** property.

3 Click the **Browse** button at the right hand side of the settings field.

The **Background** dialog box appears.

4 Click the **Type** list button, then select the background type.

The options are **None**, **Solid**, **Gradient** and **Image**.

5 Select a background type.

6 The settings for the selected type are displayed in the **Background** dialog box.

7 Specify the background settings.

8 Click the **OK** button on the **Background** dialog box.

9 (option) Click the **Current** field.

***Прим.:*** The GRADIENTMODE system variable sets the default for gradient backgrounds: 2 or 3 colors.

## 7.12 Visual styles

A visual style is a collection of settings that control the display of 3D solids and surfaces.

The VISUALSTYLES command opens **Drawing Explorer – Visual Styles**.

Here you can:

- Edit existing visual styles.
- Create new visual styles.
- Delete visual styles.
- Copy and paste visual styles between open drawings.

### 7.12.1 Setting the current visual style

The current visual style can be set in the **View / Visuals** style field in the **Properties** panel and in the right-click context menu of the **LookFrom** widget.

Click the desired visual style in the list.

The default options are:

- 2D Wireframe
- Conceptual
- Hidden
- Modeling
- Realistic
- Shaded
- Shaded with edges
- Shades of gray
- Sketchy
- Wireframe
- X-Ray

- BIM (BIM templates only)
- Mechanical (Mechanical templates only)

## 7.13 Model space and paper space

When you start a drawing session, your initial working area is called **Model Space**. Model Space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or a user coordinate system (UCS). You view and work in model space while using the **Model** tab.

In general, model space consists of a single view that fills the screen. If needed, you can create additional views, called viewports, which can show different views of your drawing or 3D model. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. Click in a viewport to make it the current viewport. You can print the current viewport only. The VPORTS command lets you manage your viewports in model space.

BricsCAD provides an additional work area, called **Paper Space**. Paper space represents a paper layout of your drawing. In this work area, you can create and arrange different views of your drawing similar to the way you arrange drawings on a sheet of paper. In paper space you can also add keynotes, annotations, borders, title blocks, and other print-related entities, which you don't want to see in model space.

Each drawing has at least one **Layout** in which you can have one or more **Layout Viewports**. Such layout viewports are to be considered as a view window on the entities in model space. Layout viewports can be placed anywhere in a paper space layout. Each viewport has its own scale and layer visibility. All viewports in the same layout can be printed simultaneously.

The MVIEW command lets you manage your viewports in paper space.

Although not necessary to print your drawing, paper space offers a lot of advantages:

- Create multiple layouts to print the same drawing with different print settings, such as pen widths, printer configuration files, lineweight settings, drawing scale, and more.
- Add print-related entities that are not essential to the model itself, such as keynotes, annotations, title blocks, etc.
- For a single layout, create multiple layout viewports to print multiple views of your drawing at different scales.

### 7.13.1 Switching between model space and paper space

- To switch between Model Space and Paper Space layouts, use the **Model** and **Layout** tabs at the bottom of the graphic window.
- When working in a layout, you can access model space through a paperspace viewport:
  - Double-click inside a viewport to access model space. Double-click outside a viewport to go back to paper space.
  - Click the **Paper/Model** field in the **Status** bar to toggle between the workspaces. This method has to be used when it is not possible to click outside the viewport.
  - Use the **MSPACE** and **PSPACE** commands. These commands are available in the right-click context menu of the **Paper/Model** field in the **Status** bar.

### 7.13.2 Using the layout tabs

- The name of the active tab is displayed in bold.
- Click the navigation buttons at the left hand side to switch.
- Drag layout tabs to change their order.
- Click the button between the navigation buttons and the **Model** tab to open the **Layout Manager** dialog box.
- Right-click a layout tab to display a context menu.
  - **New**: Create a new layout.
  - **From Template**: Imports a layout from another drawing.
  - **Rename**: Renames the layout.
  - **Copy**: Inserts a copy of the layout.
  - **Move Right / Move Left**: Moves the layout.
  - **Export...**: Exports the layout to a new drawing.

## 7.14 Model space viewports

The display in model space (Model Space, with tiled viewports) can be divided into multiple viewports, each of which can contain a different view of the current drawing. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. When no command is active, click in a viewport to make it the current viewport. You can print the current viewport only.

The coordinate system (WCS or UCS), grid display and snap can be set per viewport.

The VPORTS command creates and manages model space viewports.

### 7.14.1 Working with multiple viewports

- Click in a viewport to make it current.
- The border of the current viewport is highlighted.
- The cursor is shown in the current viewport only.
- You can start a command in one viewport, and finish the operation in a different viewport.

### 7.14.2 Joining adjacent viewports

You can join viewports which share an edge of equal length.

1 Launch the **VPORTS**command.

2 Select the **Join** command option.

**You are prompted**: Select inside dominant viewport <Current>:

3 Click in the dominant viewport or press Enter to choose the current viewport.

4 The border of the selected viewport is highlighted.

**You are prompted**: Select inside viewport to join.

5 Click the viewport to join again to confirm.

6 The two selected viewports are joined.

## 7.15 Paper space viewports

The **ALIGNSPACE** command adjusts viewport angle, zoom factor and pan position based on alignment points specified in model space and paper space.

In a layout, you can create multiple viewports, each of which displays a unique view of the entities created in model space. Each layout viewport functions as a window into your model space drawing. You can control the view, scale, and content of each layout viewport separately.

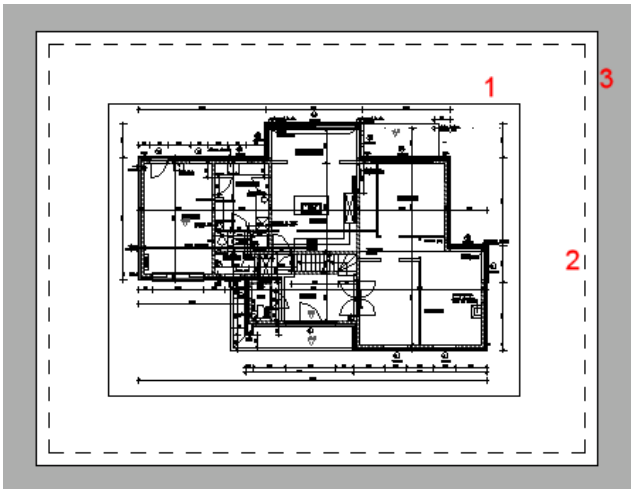
A layout viewport is created as a separate entity that you can copy, delete, move, scale, and stretch as you would any other drawing entity. You can snap to the viewport borders using entity snap. When you are working in model space in layout, click any layout viewport to make it the current viewport, and then add or modify model space entities in that viewport. Any changes you make in one layout viewport are immediately visible in the other viewports (if the other layout viewports are displaying that portion of the drawing). Zooming or panning in the current viewport affects only that viewport.

Each viewport has its own layer visibility settings. You can also turn off the display of the content of a viewport.

To preserve the scaling of a viewport, you can lock the display. It is no longer possible to zoom or pan in a locked viewport.

Apart from viewports, you can add print-related entities in a paper space layout that are not essential to the model itself, such as keynotes, annotations, title blocks, etc. Such entities are part of a specific paper space layout and do not appear in other layouts or in model space.

### 7.15.1 Displaying the paper sheet and the printable area




1 Viewport.

2 Printable area.

3 The display of the printable area is controlled through the DISPPAPERMARGINS system variable.

4 Paper sheet.

5 The display of the paper sheet is controlled through the DISPPAPERBKG system variable.

## 7.16 Creating paper space viewports

The MVIEW and VPORT commands create one or more viewports in layout tabs.

- A new viewport shows all entities on the layers which are currently visible in model space.
- It is recommended to use a dedicated layer for viewport entities. If you don't want the viewport outlines to be printed, set the **Do Not Plot** property of the viewport layer.

### 7.16.1 Creating an array of layout viewports

The MVSETUP command (short for "multiple viewports setup") creates multiple paper space viewports, and then aligns, rotates and scales them.

### 7.16.2 Defining the viewport properties

1 Select the viewport.

1 The viewport properties appear in the **Properties** panel.

2 Click the settings field of a property to modify.

| Name | Description |
|---|---|
| **Center** | XYZ coordinates of the center point of the viewport with respect to the bottom left corner of the printable area of the layout. |
| **Height (*)** | Height of the viewport in **drawing units**. |
| **Width (*)** | Width of the viewport in **drawing units**. |
| **On** | Controls the display of the content of the viewport. |
| **Clipped** | Indicates whether the viewport is clipped or not. |
| **Display locked** | Locks the scaling of the viewport content to preserve the scaling factor.<br>The VPLOCK field in the **Status** bar displays the Display Locked status of the selected viewport(s). Click the field to toggle the **Display Locked** property of the selection. |

| Name | Description |
|---|---|
| **Scale** | The SCALELISTEDIT command allows you to edit the available scales in the current drawing.<br>**Annotation scale**: Sets the annotation scale for the viewport. Editing the Annotation scale automatically sets the Standard scale.<br>If the ANNOALLVISIBLE system variable is OFF, annotative entities which do not support the annotation scale are not displayed.<br>**Standard scale**: By default, the Standard scale matches the Annotation scale, but you can adjust it manually. The **Standard scale** field reads '**Custom**' if the value in the **Custom scale** field is not available in the drawings scale list.<br>**Custom scale**: Displays the viewport scale factor. Allows you to set the viewport scale to a scale which is not available in the drawings scale list. |
| **UCS per viewport** | If **Yes**, lets you define a UCS for this viewport. |
| **Visual Style** | Sets the visual style for the viewport. This option only applies to 3D models. |
| **Shade Plot** | Allows you to adjust the plotted shading from the visual style. Default is '**As Displayed**'. |
| **Default Lighting** | Allows you to switch off the lights in the model and use the default lighting. |

## 7.17 Editing paper space viewports

### 7.17.1 Setting the layer properties in a paper space viewport

The layer visibility can be controlled per paper space viewport. When working in a paper space layout, a number of VP* (Viewport) properties are available, which affect the current viewport only and can be adjusted from the corresponding default properties:

- VP Freeze
- VP Color
- VP Linetype
- VP Lineweight
- VP Transparency
- VP Plot

1 Double-click inside the paper space viewport.

2 Launch the EXPLAYERS command.

3 The **Drawing Explorer / Layers** dialog box appears.

4 (option) Click the icon in the **VP Freeze** column to toggle the display of a layer.

5 (option) Click the icon in the **VP Color** column, then select a color in the **Select Color** dialog box.

6 (option) Click the **VP Linetype** column, then select a linetype in the drop-down list.

7 (option) Click the **VP Lineweight** column, then select a lineweight in the drop-down list.

8 (option) Click the **VP Transparency** column, then select a transparency value in the drop-down list.

9 (option) Click the **VP Plot** column, then select a plot style in the **Select Plot Style** dialog box. This option applies to STB (Style Tables) drawings only.

***Прим.:***

- When the **Indicate layers in use** option in the **View** menu of the **Drawing Explorer** dialog box is set, the icon in the **Current** column indicates whether VP property overrides exist.
- Use the **Viewports / Vpfreeze** option of the LAYOFF and LAYFRZ commands to freeze layers in a viewport by clicking entities.

### 7.17.2 Clipping a viewport

The **VPCLIP** command (short for "viewport clipping") clips viewports in layouts.

Only circles and closed polylines can be used to create a clipped viewport. Therefore, use the **Create polyline representations of an ellipse** option of the PELLIPS system variable when creating an elliptical viewport.

### 7.17.3 Removing the clipping boundary of a viewport

The **Delete** option of the VPCLIP command removes the clipping boundary.

### 7.17.4 Rotating the display in a paper space viewport

The display of a paper space viewport can be rotated only if the display of the viewport is not locked.

The **Align** > **Rotate** option of the MVSETUP command defines the angle of the content of a viewport relative to the X-axis of the world coordinate system (WCS).

If the VPROTATEASSOC system variable is ON, the ROTATE and MANIPULATE commands rotate the content of a viewport over a specified angle. The viewport is clipped by a rectangle of the dimensions of the original viewport. Use the **Delete** option of the VPCLIP command to remove the clipping.

In order to undo the rotation:

- Set the rotation angle = 0 (zero) in the **Align** > **Rotate** option of the MVSETUP command.
- Double-click inside the viewport, then launch the PLAN command and press Enter twice (= set plan view of the WCS).

### 7.17.5 Aligning viewports

1 Launch the MVSETUP command.

2 Choose **Align** in the prompt menu.

**You are prompted**: Enter an option [Angled/Horizontal alignment/Vertical alignment/Rotate view/Undo]:

3 (option) Choose the **Horizontal** alignment option.

**You are prompted**: Specify basepoint:

- 1. Pick a point in the reference viewport.
- 2. **You are prompted**: Specify point in viewport to be panned:
- 3. Pick a point in the viewport to be aligned.
- 4. The display in the viewport is moved vertically to align the two points horizontally.

4 (option) Choose the **Vertical alignment option**.

**You are prompted**: Specify basepoint:

- 1. Pick a point in the reference viewport.
- 2. **You are prompted**: Specify point in viewport to be panned:
- 3. Pick a point in the viewport to be aligned.
- 4. The display in the viewport is moved horizontally to align the two points vertically.

### 7.17.6 Aligning model space and paper space points

The ALIGNSPACE command adjusts viewport angle, zoom factor, and pan position based on alignment points specified in model space and paper space.

**Aligning 1 point**

1 Launch the ALIGNSPACE command.

1 **You are prompted**: First alignment point in Model space:

2 Click inside a layout viewport, then pick a point.

3 **You are prompted**: Second alignment point in Model space or <Return> for none:

4 Right-click or press Enter.

5 **You are prompted**: Alignment point in Paper space:

6 Pick a point in the layout.

7 **You are prompted**: Activate the viewport which will be used for alignment.

8 Click inside the viewport you want to align.

9 Right-click or press Enter to confirm.

10 The display in the selected viewport is moved to make the two points coincident.

**Aligning 2 points**

1 Launch the ALIGNSPACE command.

1 **You are prompted**: First alignment point in Model space:

2 Click inside a layout viewport, then pick a point.

3 **You are prompted**: Second alignment point in Model space or <Return> for none:

4 Pick a second point in the layout viewport.

5 **You are prompted**: First alignment point in Paper space:

6 Pick a point in the layout.

7 **You are prompted**: Second alignment point in Paper space:

8 Pick a second point in the layout.

9 **Prompts you:** Activate the viewport which will be used for alignment.

10 Click inside the viewport you want to align.

11 Right-click or press Enter to confirm.

12 The display in the selected viewport is moved, rotated and/or scaled to make the points two by two coincident.

## 7.18 Layouts

In BricsCAD, you can create up to 255 layouts in a single drawing. Each layout represents a sheet of paper. For each layout you can specify the paper size, print area, print scale, lineweight scale, pen mappings, and add viewports, dimensions, a title block, and other geometry specific to the layout. The entities you add to a layout in paper space do not appear in model space.

Each layout requires at least one layout viewport. If the CREATEVIEWPORTS system variable is ON, a viewport is added automatically in a new layout. If this variable is OFF, newly created viewports are empty. To add new layouts, you can either define them yourself or you can import layouts from another drawing: template file (.dwt), drawing file (.dwg) or drawing interchange file (.dxf).

Viewports can display all or part of the drawing's model space entities.

The CACHELAYOUT system variable controls whether the content of layouts is cached. The extra memory required for caching is made up for by improvements to the memory requirements for displaying entities, the net result being that the memory needed to display drawings has been reduced.

### 7.18.1 Creating a new layout

1 Launch the LAYOUT command, then select the **New** option.

2 **You are prompted**: Enter a layout name. <Layout1>:

3 Type a unique name for the layout, then press Enter.

4 The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign ($), hyphen (-), and underscore (_), or any combination.

5 Right-click or press Enter to accept the default name (e.g. Layout1).

6 The new layout tab is added.

***Прим.:*** Right-click a layout tab, then choose **New** in the context menu or click the plus (+) button after the last layout tab to add a new layout using a default name. See step 3 in the above procedure to rename the layout.

### 7.18.2 Copying layouts

1 Right-click the layout tab you want to copy.

1 A context menu appears.

2 Choose **Copy** in the context menu.

**You are prompted**: Enter new layout name:

3 Type a new unique name in the Command line, then press Enter.

4 A copy of the selected layout is added.

### 7.18.3 Arranging the layout tabs

Do one of the following:

- Drag the layout tab to the desired position.
- Right-click the layout tab, then choose a **Move** option in the context menu.

### 7.18.4 Importing layouts

1 Right-click the **Model** tab or one of the layout tabs.

A context menu appears.

2 Choose **From Template...** in the context menu.

The **Select Template From File** window opens.

3 Browse to the drawing that you want to import layouts from.

4 Click the **Open** button on the **Select Template From File** window.

The **Insert Layout(s)** window opens.

5 Select the layout(s) you want to import.

6 Hold down the Ctrl key to select multiple layouts.

7 Click the **OK** button.

The selected layouts are imported.

### 7.18.5 Renaming a layout

1 Double-click the layout tab you want to rename.

1 The **Rename Layout** dialog box opens.

2 Type a new unique name in the **Name** field.

3 Click the **OK** button.

### 7.18.6 Deleting a layout

1 Right-click the layout tab you want to delete.

1 A context menu appears.

2 Choose **Delete** in the context menu.

3 An **Alert** box appears.

4 Click the **OK** button to confirm.

### 7.18.7 Using the Layout Manager

1 Do one of the following:

- Launch the LAYOUTMANAGER command.
- Click the button between the navigation buttons and the **Model** tab.

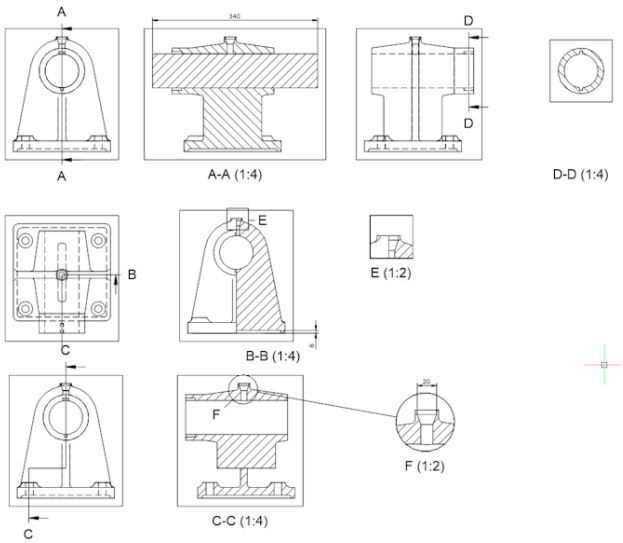
2 In the **Layout Manager** dialog box you can:

- Create layouts.
- Copy layouts.
- Delete layouts.
- Change the order of the layout tabs.
- Publish layouts.

## 7.19 Generated drawing views

The Generated Drawing Views functionality allows you to automatically generate associative orthographic and standard isometric views of a 3D solid model. All drawing views are placed in a paper space layout, they are not visible in model space.

Analytical hidden line removal (HLR) procedures are used to create the drawing views using standard 2D entities, mostly lines and arcs. Much like the result of the FLATSHOT command. Drawing views created in V14.1 will be automatically converted into HLR geometry by the VIEWUPDATE command.




The VIEWBASE command generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.

The VIEWSECTION command creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout.

The VIEWSECTIONSTYLE command allows you to modify the appearance of the section views created by the VIEWSECTION command.

The VIEWPROJ command generates additional projected views from an existing generated drawing view.

The VIEWDETAIL command creates a detail view of a portion of a standard generated drawing, at a larger scale.

The VIEWDETAILSTYLE command allows you to modify the appearance of the detail views created by the VIEWDETAIL command.

The VIEWUPDATE command updates a selection of drawing views obtained by VIEWBASE and VIEWSECTION when VIEWUPDATEAUTO = 0.

The VIEWEXPORT command exports the content of drawing views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing. This command can be used in Paper Space only.

The VIEWEDIT command allows you to change the scale and the hidden line visibility of drawing views.

Tools to generate drawing views of your 3D solid model are available on:

- The **Drawing Views** toolbar, which is a flyout of the **View** toolbar.
- The **Section Types** and **Detail Section Types** toolbars.
- The **View** | **Drawing Views** menu.
- The **Drawing Views** panel of the **Annotate Ribbon** tab (Mechanical Workspace).

### 7.19.1 Standard drawing views

Standard views include multiview orthographic and isometric projections.

In a technical drawing, a multiview orthographic projection is an illustration technique in which up to six images of an object are generated, with each projection plane parallel to one of the coordinate axes of the object.

Isometric projection is a method to represent three-dimensional objects in two dimensions in technical and engineering drawings. It is an axonometric projection in which the three coordinate axes appear equally foreshortened and the angles between any two of them are 120 degrees.

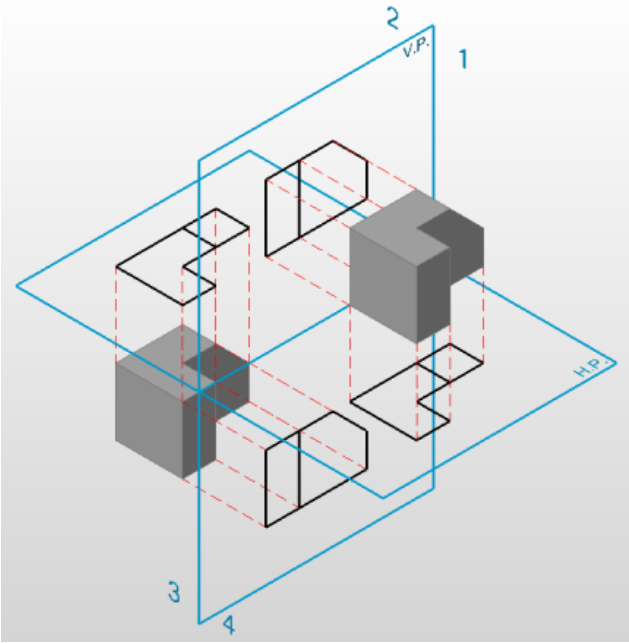
To generate the standard views of your 3D solid model, run the ViewBase command from Model Space.

You can choose between first angle (European) projection or third angle (American) projection.

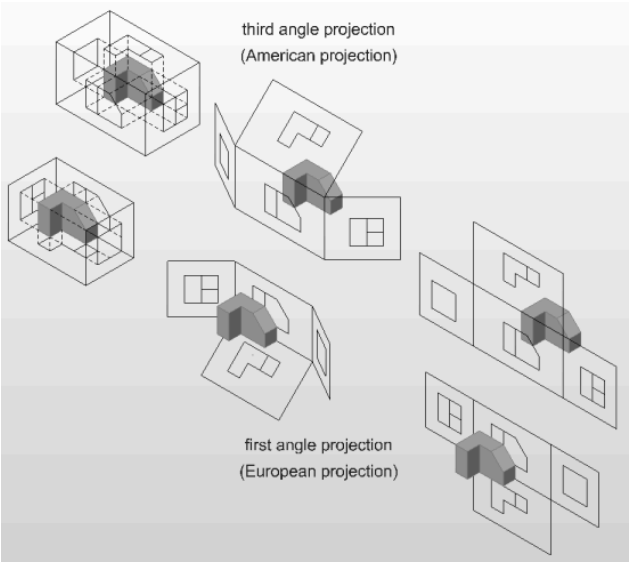
Orthographic projection (or orthogonal projection) is a means of representing a three-dimensional object in two dimensions. It is a parallel projection, where all the projection lines are orthogonal to the projection plane, resulting in every plane of the scene appearing in affine transformation on the viewing surface. It is further divided into multiview orthographic projections and axonometric projections.

Two projection planes, one horizontal (H.P.) and one vertical (V.P.) divide 3D space in four quarters.

- In the first angle projection system, the model is placed in the first quarter space (first angle).
- In the third angle projection system, the model is placed in the third quarter (third angle).




With multiview orthographic projections, up to six views of a 3D model are produced, with each projection plane parallel to one of the coordinate axes of the model. The views are positioned relative to each other, according to either of two schemes: first-angle or third-angle projection. In each, the appearances of views may be thought of as being projected onto the 6 faces of a box around the model. When unfolding the box, all views are organized in a 2D drawing.




The projection type is set by the **Projection Type** option of the VIEWBASE command or by editing the PROJECTIONTYPE system variable:

- 0 = First angle projection
- 1 = Third angle projection

The value of the PROJECTIONTYPE system variable is saved in the drawing.

Generated drawing view lines are created on default layers:

- BM_Isometric_Hidden;
- BM_Isometric_Visible;
- BM_Ortho_Hidden;
- BM_Ortho_Visible.

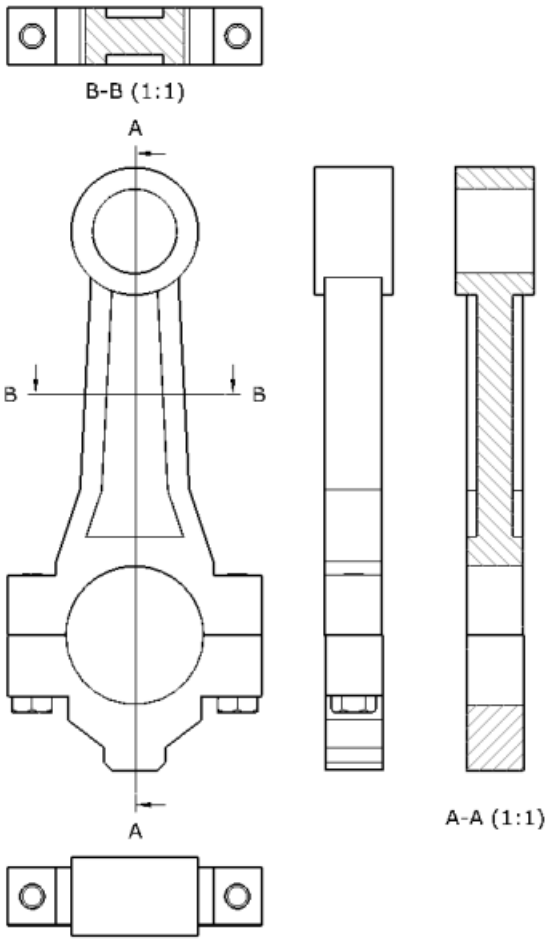
The display of the BM_Isometric_Hidden layer is turned off by default.

Additional projected views can be added using the VIEWPROJ command. A projected view inherits the scale, display settings and alignment from the parent view.

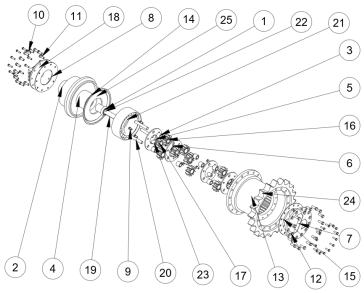
### 7.19.2 Generating section views

A section is the intersection of a 3D solid model with a section plane. In technical drawings, the internal parts of the 3D model are hatched in cross sections.

To generate section views of a 3D solid model, run the VIEWSECTION command from the paper space layout where the standard drawing views are located.



### 7.19.3 Special views




Bill of Materials Drive_Assembly

| No. | Component | Quantity |
|---|---|---|
| 1 | BBRG-03 | 2 |
| 2 | BBRG-03 | 1 |
| 3 | BBRG-60 | 9 |
| 4 | Body | 1 |
| 5 | Carrier_1 | 2 |
| 6 | Carrier_2 | 2 |
| 7 | Cover | 1 |
| 8 | Cover_Interal_Seal | 1 |
| 9 | DIN 933 | 12 |
| 10 | DIN 933 | 24 |
| 11 | DIN 933 | 16 |
| 12 | DIN 933 | 16 |
| 13 | Drum | 1 |
| 14 | External_Medium_Seal | 1 |
| 15 | External_Seal | 1 |
| 16 | Gear_Planet | 9 |
| 17 | Gear_Sun | 3 |
| 18 | Internal_Cover | 1 |
| 19 | Key | 1 |
| 20 | Pin | 3 |
| 21 | RLR BRG-32 | 1 |
| 22 | Shaft | 1 |
| 23 | Sleeve | 6 |
| 24 | Sprocket | 1 |
| 25 | Washer | 1 |

It is possible to generate drawing views for a selected exploded representation. Normally, an orthogonal projection is used for that; however, BricsCAD allows you to create any standard projection for exploded representations. Sections and detailed views are also supported.

To create a view for an exploded representation, run the VIEWBASE command with the **Special** views option, select the **Exploded view** option and then select the required exploded representation from the dialog box.

Generated views for exploded representation can be associative with the original model, so annotations and balloons can be placed. To do that, the views should be created or updated with GENERATEASSOCVIEWS = ON.

### 7.19.4 Updating drawing views

By default BricsCAD checks whether the source 3D solid model or the associated exploded representation was modified and automatically recalculates the drawing when:

- opening a paper space layout containing out-of-date views if the VIEWUPDATEAUTO system variable is ON.
- running the VIEWUPDATE command.
- running the BMUPDATE command.
- 3D geometry.

***Прим.:***

- Hidden line removal calculation might take some time for complex 3D models. In such cases, it is recommended to set VIEWUPDATEAUTO = OFF. The viewport border of out-of-date drawing views turns red to indicate an update is needed. When executing the ViewUpdate command, BricsCAD prompts you to either select the drawing views to be updated or update all drawing views.
- If GENERATEASSOCVIEWS = ON, associative dimensions will be updated according to the modified 3D geometry.

### 7.19.5 Exporting views to model space

The associativity mechanism of drawing views in a paper space layout allows for limited control by the user. Such views can be moved or scaled, but you cannot edit the geometry. In order to get the full control to the drawing views geometry, the ViewExport command allows you to move or copy drawing views to

model space or to a separate drawing. Exported drawing views lose their associativity with the 3D model and become standard blocks, which can be exploded, edited, erased, etc.

### 7.19.6 Customizing drawing views

The first time the VIEWBASE command is launched, four new layers are created:

- BM_Ortho_Hidden: hidden lines in drawing views;
- BM_Ortho_Visible: visible lines in drawing views;
- BM_Tangent_Hidden: hidden tangent lines in drawing views;
- BM_Tangent_Visible: visible tangent lines in drawing views.

Use the VIEWEDIT command to change the visibility of these layers for all views or for a selection of views only.

- Edit the properties of these layers, such as linetype, lineweight or color, to modify the display of the drawing views.
- Use ViewSectionStyle command to adjust different properties of section views such as style, color, and height of identifier text;
- start and end symbols, their colors, size and direction of arrows scale, patter, color, and angle of hatches;
- color, type, and weight of a line representing a cutting plane.

Use the VIEWDETAILSTYLE command to adjust different properties of details views such as:

- style, color, height of identified text, and the type of identifier
- symbol, and its color and size
- color, type, and weight of detail view boundary

# 8. 2D черчение

## 8.1 Создание объектов

Вы можете создавать в чертеже различные типы объектов. Это могут быть как простые объекты (отрезки, окружности, дуги, точки, лучи), так и сложные объекты (полилинии, сплайны, плоские фигуры). Кроме 2D объектов BricsCAD также позволяет создавать 3D поверхности и твердые тела.

Для полноценного оформления чертежа в BricsCAD включен дополнительный набор инструментов создания текстовых объектов, штриховок и размеров.

### 8.1.1 Основные операции при создании новых объектов

Новые объекты создаются с использованием текущих настроек создания объектов на текущем слое в плоскости XY текущей системы координат, в качестве которой по умолчанию используется Мировая система координат (МСК).

1 Установка текущего слоя (при необходимости).

2 Для определения параметров создания объектов вы можете использовать раздел **Общие** на **Панели свойств**.

3 Запуск команды.

4 Выполнение команды в соответствии с запросами программы.

## 8.2 Параметры создания объектов

Параметры создания объектов, используемые по умолчанию, определяются значениями системных переменных.

Для просмотра и редактирования этих переменных вы можете использовать:

- Диалоговое окно Параметры. Откройте раздел **Черчение** > **Создание объектов**. В верхней части списка параметров отображаются такие свойства объектов, как цвет, тип линий и слой. Ниже отображаются специфические свойства, сгруппированные по типам объектов.
- Разделе **Общие** на Панели свойств при отсутствии выбранных объектов.

В разделе **Общие** доступны следующие параметры:

- **Цвет**: Устанавливает текущий цвет объекта. По умолчанию = По слою.
- **Слой**: Устанавливает текущий слой.
- **Тип линий**: Устанавливает текущий тип линий. По умолчанию = По слою.
- **Масштаб типа линий**: Устанавливает текущий масштаб типа линий. По умолчанию = 1.
- **Толщина линии**: Устанавливает текущую толщину линии. По умолчанию = По слою.
- **Прозрачность**: Устанавливает прозрачность. По умолчанию = По слою.
- **Уровень**: Устанавливает высоту (Z) над плоскостью XY текущей системы координат (МСК или ПСК). По умолчанию = 0.

***Прим.:*** Большинство настроек создания объектов сохраняются в чертеже, поэтому они применяются только к текущему чертежу. Если вы хотите применить эти настройки для всех

новых чертежей, отредактируйте настройки по умолчанию в чертежах шаблонов, используемых вами при создании чертежей. Рекомендуется сохранять чертежи шаблонов в папке, указанной в значении системной переменной TEMPLATEPATH. Системная переменная BASEFILE определяет путь и имя файла шаблона, используемого по умолчанию.

## 8.3 Работа со слоями

Вы можете создавать слои и управлять ими с помощью раздела **Слои** диалогового окна Проводник по чертежам (команда EXPLAYERS) или панели Слои (команда LAYERSPANELOPEN).

В отличие от панели Слои, диалоговое окно Проводник по чертежам не может быть открыто в процессе выполнения других команд. Вы можете перемещать и изменять его размер по своему усмотрению или перемещать на второй монитор.

### 8.3.1 Текущий слой

Все новые объекты создаются на текущем слое. Чтобы нарисовать новый объект на другом слое, необходимо назначить этот слой текущим.

- Текущий слой определяется значением системной переменной CLAYER.
- Системная переменная CENTERLAYER* определяет слой, на котором размещаются осевые линии и маркеры центра, создаваемые командой DIMCENTER.
- Системная переменная DIMLAYER* определяет слой для размеров.
- Объекты, создаваемые командой INTERFEREE, размещаются на слое, который определяется значением системной переменной INTERFERELAYER.

* Значение по умолчанию: <Использовать текущий>

Чтобы установить текущий слой, выполните одно из следующих действий:

- При отсутствии выбранных объектов, щелкните на поле **Слой** на Панели свойств, нажмите кнопку со стрелкой и в открывшемся списке выберите новый текущий слой.
- На панели **Свойства объектов** нажмите кнопку со стрелкой рядом с полем **Слой** и в открывшемся списке выберите необходимый слой.
- На панели ленты **Слои** нажмите кнопку со стрелкой и в открывшемся списке выберите необходимый слой.
- Щелкните правой кнопкой мыши на поле **Слой** в строке состояния и в открывшемся контекстном меню выберите необходимый слой.

***Прим.:***

- Вы можете выключить отображение текущего слоя ( ). В результате все построенные на этом слое объекты будут невидимы до тех пор, пока вы снова не включите текущий слой ( ). При попытке выключить текущий слой отображается соответствующее предупреждение.
- Замораживание ( ) текущего слоя невозможно.

### 8.3.2 Состояния слоев

Команда LAYERSTATE.

Состояние слоев - это именованное содержимое, хранящееся в чертеже.

Управление состояниями слоев позволяет сохранять и восстанавливать конфигурации настроек слоев и их состояние. Возможно, вам будет необходимо иногда отображать слой синим цветом, а в остальных случаях - зеленым. Также может возникнуть необходимость отключить, заморозить или заблокировать некоторые слои при редактировании фрагмента чертежа. Чтобы экономить время на возврат к исходным настройкам слоев, вы можете использовать сохраненные состояния слоев. Вы можете сохранять множество состояний слоев в каждом чертеже. Состояния слоев могут быть скопированы с одного чертежа на другой и сохранены во внешний файл, который затем может быть импортирован в другой чертеж.

## 8.4 Entity linetype

You can use different linetypes to differentiate the purpose of one line from another. A linetype consists of a repeating pattern of dots, dashes, or blank spaces. Linetypes determine the appearance of entities both on the screen and when printed.

By default, every drawing has at least three linetypes: Continuous, Bylayer and ByBlock. Your drawing can also contain an unlimited number of named linetypes.

Linetype definitions are saved in the drawing. New linetypes are either imported from other drawings or loaded from a linetype file (*.lin). Which linetype source file is used by default is controlled by the MEASUREMENT system variable:

- MEASUREMENT = ON uses**Iso.lin**: Linetype definitions are expressed in mm.
- MEASUREMENT = OFF:**Default.lin**: Linetype definitions are expressed in inch.

New entities are drawn using the current linetype set by the CELTYPE system variable.

The options to set the Linetype are:

- ByLayer: Entities adopt the linetype of their layer. This allows you to change the linetype of all such entities by adjusting the linetype of the layer.
- ByBlock: Entities drawn as continuous lines until you include them into a block. The entities then inherit the linetype setting of the block entity when you insert the block into a drawing.
- Named linetype: Continuous or one of the linetypes saved in the drawing.

**Linetype scale**

Non-continuous linetypes consist of lines, gaps and dots put together in a variety of patterns. In complex linetypes also text or shapes can be included.

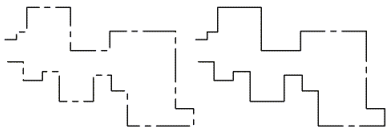
In the linetype definition the length of the lines and gaps are either defined in inch (default.lin) or mm (iso.lin). If the drawing units are different from inch or mm, linetypes need to be scaled. The linetype scaling is controlled by the LTSCALE system variable, which must be set with respect to the drawing units (DU):

| Drawing Unit | LTSCALE |
|---|---|
| Imperial (MEASUREMENT = OFF) | |
| Inch | 1 |
| Foot | 1/12 |
| Metric (MEASUREMENT = ON) | |
| mm | 1 |
| cm | 0.1 |
| m | 0.001 |

BricsCAD will always put a dash at the start and end of a line, polyline, arc or spline. The linetype pattern is also nicely centered, so that both ends of the entity look the same.

**Polyline Generation**

Linetypes are normally generated from vertex to vertex. Polylines of which the vertices are very close together might be rendered as a continuous line, if the linetype pattern does not fit between two subsequent vertices. ThePLINEGEN system variable addresses this problem: when set to 1 or ON, the linetype is drawn from one end of the polyline to the other end, instead of from vertex to vertex.



The same polyline drawn with PLINEGEN on (left) and off (right).

**Paper space linetype scale**

The PSLTSCALE system variable controls the linetype scaling in paperspace layouts.

If ON, linetype scaling is independent from the viewport scale.

In general, PSLTSCALE is best set ON. Notice that this variable can be set per layout.

**Model space linetype scale**

The MSLTSCALE system variable controls the linetype scaling in model space.

If ON, thecurrent annotation scale(CANNOSCALE system variable) is applied to linetypes in model space.

MSLTSCALE is best set OFF. It might be useful to switch it ON when printing from model space.

### 8.4.1 Setting the current linetype

Do one of the following:

- On theProperties panel, select**Linetype**.

Make sure that no entity is selected.

Choose a linetype in the drop down list.

- On the**Entity Properties**toolbar, click the down arrow next to the**Linetype**field, then choose a linetype in the drop down list.
- Right click on the**Linetype**field in theStatus Bar, then choose a linetype in the context menu.

### 8.4.2 Adding a new linetype

1 Do one of the following:

- Launch the LINETYPE command, then choose the **New** option.
- Select **Linetype** in the Properties panel, then click the down arrow and choose **Load...**

The Load linetypes dialog box displays.

2 (option) Click the **File...** button to load a different linetype source file.

3 Select one or more linetype(s) in the Load linetypes dialog box.

Hold down the Ctrl-key to add or remove linetypes.

4 Click the **OK** button

### 8.4.3 Setting the entity linetype scale

1 On theProperties panel, select**Linetype scale**.

Make sure no entity is selected.

2 Type a new value in the Linetype scale field.

The value of the Linetype Scale is saved through the CELTSCALE system variable. This variable is best set to 1, except for special linetypes, such as BATTING or GAS_LINE, which most likely need to be scaled.

***Прим.:*** The Entity Linetype Scale applies to new entities only.

## 8.5 Цвет объекта

Цвет объекта определяет его отображение на экране и при печати. Всем новым объектам при их создании присваивается текущий цвет, который соответствует значению системной переменной CECOLOR.

Вы можете выбрать между 255 **индексными цветами** или определить **реальный цвет**.

### 8.5.1 Индексные цвета

Индексный цвет представляет собой 8-битное описание цвета пикселя на экране дисплея. Доступно 256 различных цветов.

Каждый из индексных цветов имеет свой порядковый номер от 1 до 255. Семь из них кроме порядкового номера имеют имя: красный (1), желтый (2), зеленый (3), голубой (4), синий (5), розовый (6) и белый/черный (7). Индексный цвет 7 при черном цвете фона экрана отображается белым, а при белом цвете фона экрана - черным. При печати цвет 7 отображается черным.

Два дополнительных свойства цвета - По слою и По блоку. При применении этих свойств объект приобретает цвет того слоя или блока, на который выполняется вставка этого объекта. Цвет По слою имеет номер 256, а По блоку - 0. Для всех команд, использующих цвет, вы можете определять цвет как свойствами По слою и По блоку, так и номерами 256 и 0 соответственно.

Параметры установки цвета:

- По слою: Объекты имеют цвет того слоя, на котором они находятся. Это позволяет одновременно изменять цвет всех объектов, находящихся на одном слое.
- По блоку: Объекты имеют индексный цвет 7 (черный или белый, в зависимости от цвета фона экрана). При включении таких объектов в состав блока они приобретают цвет, назначенный этому блоку.
- Индексные цвета
- Реальные цвета

***Прим.:*** Цвет объектов при выводе чертежа на печать зависит также от **Таблицы цветов** (CTB) или **Таблицы стилей** (STB), используемых при печати. Цвет объекта при печати будет соответствовать цвету объекта на экране только в том случае, если для параметра **Цвет** в файле определения Таблицы цветов или Таблицы стилей включена опция **Использовать цвет объекта**.

### 8.5.2 Реальные цвета

Реальный цвет представляет собой 24-битное описание цвета пикселя на экране дисплея. Доступно 16 777 216 различных цветов.

Количество битов, используемых для описания оттенка цвета пикселя определяют глубину цвета. Реальный цвет также называют 24-битным цветом. Новые устройства отображения позволяют использовать 32-битный режим цвета. Дополнительный байт, называемый альфа-каналом, используется для хранения информации, необходимой для управления цветом или создания специальных эффектов.

Реальный цвет использует трехцветное **RGB**-описание цвета (**R**ed, **G**reen, **B**lue - Красный, Зеленый, Синий). Интенсивность каждого цвета изменяется в пределах от 0 до 255. RGB-описание для черного цвета имеет вид (0,0,0), для белого - (255,255,255).

### 8.5.3 Установка текущего цвета объекта

1 Выполните одно из следующих действий:

- Запустите команду COLOR.
- Нажмите кнопку **Цвет...** () на панели Параметры.
- На Панели свойств выберите поле **Цвет** и в выпадающем списке выберите **Выбрать цвет**.
- Дважды щелкните на поле **Цвет** в строке состояния.

Откроется диалоговое окно Выбор цвета.

2 Для выбора индексного цвета щелкните на закладке **Индексные цвета** в окне Выбор цвета и

выполните одно из следующих действий:

- Щелкните на одной из цветных ячеек.
- В поле **Номер цвета** введите или выберите номер необходимого цвета.

3 Для выбора реального цвета щелкните на закладке **Реальные цвета** и выполните одно из следующих действий:

- Выберите цвет щелчком на **Цветовой палитре**.
- Установите параметры цвета в полях **Параметры цвета**.

4 Нажмите кнопку **OK** для подтверждения.

## 8.6 Толщина линии

Толщина линий определяет толщину объектов при отображении их на экране и при печати.

Для толщины линий может быть установлено значение По слою, По блоку или определенное значение.

Толщина линий определяется в миллиметрах или дюймах в зависимости от значения системной переменной LWUNITS.

Вы не можете назначать толщину для плоских фигур, точек, ttf-шрифтов и растровых изображений.

Опции настройки толщины линии:

- ПО СЛОЮ: Толщина линии определяется значением свойства Толщина линии, установленным для текущего слоя. При изменении толщины линии, назначенной слою, изменяется толщина линий всех объектов этого слоя, для которых установлена толщина линии По слою.
- По блоку: Объекты, для которых установлена толщина линий По блоку, отображаются с толщиной линии По умолчанию до тех пор, пока объект не будет включен в блок. При вставке этого блока в чертеж объекту назначается толщина линий, установленная для блока.
- По умолчанию: Толщина линий определяется значением системной переменной LWDEFAULT. Изменение толщины линий по умолчанию влияет на все объекты чертежа, для которых установлено значение толщины линий По умолчанию.

### 8.6.1 Системные переменные

Системная переменная CELWEIGHT определяет толщину линии для новых объектов.

Системная переменная LWDEFAULT устанавливает значение толщины линии По умолчанию.

Системная переменная LWDISPLAY управляет отображением толщины линий. Для включения/выключения отображения толщины линий вы можете использовать поле ТОЛЩИНА строки состояния.

Системная переменная LWDISPLAYSCALE Управляет отображением масштаба толщины линии в пространстве модели. Допускаются значения от 0 до 1.

### 8.6.2 Толщина линий при печати

Для объектов, имеющих индексный цвет, толщина линий при печати с использованием **Таблицы цветов** (CTB) и **Таблицы стилей** (STB) будет соответствовать толщине линий объекта только при выборе в настройках стиля опции **Использовать толщину линий объекта**. В других случаях толщина линии определяется цветом объекта или стилем печати.

Объекты, имеющие реальный цвет при печати всегда используют толщину линий объекта.

## 8.7 Fill mode

TheFILLcommand toggles the display of filled and hatched areas and specifies whether wide polylines, traces, solids, hatches (including solid-fill) and wide polylines are filled in. It toggles the value of the FILLMODE system variable.

When FILLMODEis Off, all filled entities display and print as outlines and reduces the time it takes to display or print a drawing.

| | Polyline | Donut | Trace | Plane (Solid) |
|---|---|---|---|---|
| Fill On |  |  |  |  |
| Fill Off |  |  |  |  |

### 8.7.1 Setting Fill Mode

Do one of the following:

- Run the FILL command.
- Set the value of the FILLMODE system variable.

Run the REGEN or REGENALL command to regenerate the viewport(s).

## 8.8 Прозрачность

Свойство Прозрачность позволяет управлять прозрачностью объектов на чертеже. Вы можете устанавливать это свойство как для нескольких выбранных объектов, так и для всех объектов, размещенных на одном слое. Настройка прозрачности, которая используется по умолчанию при создании новых объектов, определяется значением системной переменной CETRANSPARENCY. Уровень прозрачности, используемый по умолчанию при создании штриховок и градиентных заливок, определяется значением системной переменной HPTRANSPARENCY.

Прозрачность может быть использована для улучшения восприятия ваших чертежей, например, путем уменьшения видимости малозначимых объектов или объектов, которые используются только для ссылки.

Опции настройки толщины линии:

- По слою: применяется настройки прозрачности, установленные для текущего слоя.
- По блоку: при вставке блока в чертеж объекты, созданные с использованием прозрачности По блоку, наследуют настройки прозрачности блока.
- Значение: Диапазон значений от 0 (без затемнения) до 90.

  Уровень прозрачности ограничен предельным значением 90 % для отличия прозрачных объектов от объектов, находящихся на замороженных или выключенных слоях.

### 8.8.1 Печать с прозрачностью

Прозрачность будет учитываться при печати только в том случае, если включена опция Печать с прозрачностью в диалоговых окнах Печать или Параметры страницы.

## 8.9 Editing entity properties

Entity properties can be edited either in the **Properties** panel, or by using the **Rollover Tips** feature.

### 8.9.1 Editing in the Properties panel

BricsCAD offers a **Properties** panel like the **Properties** palette in AutoCAD. You can access it with the PROPERTIES command (PR alias) as well as other methods such as double-clicking on an entity.

The properties displayed in the panel depend on the entity type(s) that you have selected. When multiple entities are selected on the shared properties display, the value of shared properties that vary between the selection set, display ***varies***.

As different from AutoCAD, the **Properties** panel in BricsCAD displays the **Handle** (read-only) of the selected entity, which can be useful to store and recall for programing purposes.

### 8.9.2 Editing with Rollover Tips

When Rollover Tips = ON (**RT** field in the **Status** bar), a selection of entity properties display at the cursor. Click a property to edit it.

When the Quad is on, click the **Quad** title bar to switch between Rollover Tips and the Quad cursor menu.

Right-click the **RT** field in the **Status** bar and choose **Customize…** in the context menu to open the **Customize / Properties** dialog box. Here you can define which properties display in the **Rollover Tips** for each entity type.

## 8.10 Drawing linear entities

Commands for drawing linear entities are:

| | |
|---|---|
| | Draws polylines with x,y,z coordinates. |

| | |
|---|---|
| | Draws arcs. |
| | Creates a polyline that forms a boundary around closed areas. |
| | Draws circles. |
| | Draws wide circles from polylines. |
| | Draws ellipses and elliptical arcs. |
| | Draws 2D spirals or a 3D helices. |
| | Draws line segments. |
| | Draws multilines. |
| | Draws connected lines and arcs. |
| | Draws regular polygons out of polylines. |
| | Draws semi-infinite lines. |
| | Draws rectangles and squares. |
| | Draws revision clouds. |
| | Draws sketched (freehand) lines. |
| | Draws open and closed splines in 2D and 3D. |
| | Draws wide line segments. |
| | Draws infinite lines. |

## 8.11 Отрезки

Отрезок определяется двумя точками: начальной и конечной. Используя команду LINE, вы можете создавать цепь соединенных между собой отрезков, каждый из которых при этом является отдельным объектом.



Инструмент Отрезок доступен:

- В меню Черчение
- На панели инструментов Черчение
- На панели ленты Черчение
- На закладке квадро-меню Черчение без выбора объектов.

### 8.11.1 Основные операции

1 Запустите команду LINE.

Запрос программы: Начальная точка или [Продлить] <Последняя точка>:

2 Укажите начальную точку.

Появится динамическое изображение создаваемого отрезка.

Запрос программы: Конечная точка или [Угол/Длина/Продлить/На шаг назад]:

3 Укажите конечную точку.

Будет создан отрезок. Появится динамическое изображение следующего отрезка.

Запрос программы: Конечная точка или [Угол/Длина/Продлить/На шаг назад]:

4 Укажите конечную точку.

Будет создан отрезок. Появится динамическое изображение следующего отрезка.

Запрос программы: Конечная точка или [Угол/Длина/Продлить/Замкнуть/На шаг назад]:

5 Выполните одно из следующих действий:

- Продолжайте указывать конечные точки для каждого нового сегмента линии.
- Выберите опцию **Замкнуть**, чтобы создать замкнутую полилинию.
- Нажмите клавишу Enter или щелкните правой кнопкой мыши.

### 8.11.2 Укажите угол и длину

Доступные опции:

- Определение параметров Угол и Длина.
- Использование динамических размеров (DYNDIVIS = 1 или 2).

  Используйте клавишу TAB для переключения между полями Угол и Расстояние.

### 8.11.3 Указание угла относительно другого линейного объекта

Щелкните правой кнопкой мыши на поле ДИН.ПСК в строке состояния, выберите опцию ДинПСК для других объектов, чтобы включить параметр **Включить для 2D объектов** системной переменной UCSDETECT.

1 Запустите команду LINE.

2 Подведите курсор мыши к базовому отрезку или линейному сегменту полилинии.

Временная ПСК установится по отрезку или сегменту полилинии.

Значок ПСК отображается в ближайшей средней или конечной точке объекта, находящегося

под курсором.

3 Нажмите клавишу Shift для фиксации временной ПСК.

4 Укажите начальную точку линии.

Теперь углы будут отсчитываться относительно выбранного отрезка или линейного сегмента полилинии.

***Прим.:*** После завершения выполнения команды LINE будет восстановлена МСК.

## 8.12 Лучи

Луч представляет собой линию бесконечной длины с началом в указанной точке. Т.к. лучи имеют бесконечную длину, они не учитываются при определении границ чертежа.

По умолчанию для построения луча необходимо указать начальную точку и его направление.

Лучи и Прямые линии иногда используют в качестве конструкционных линий при создании других объектов.



Инструмент Луч доступен:

- В меню Черчение
- На панели инструментов Черчение
- На панели ленты Черчение

### 8.12.1 Основные операции

1 Запустите команду RAY.

Запрос программы: Бесконечный луч: Начало луча или [Горизонталь/Вертикаль/Угол/Биссектриса/Параллель]:

2 Укажите начальную точку луча.

Появится динамическое изображение луча.

3 Укажите направление луча.

Появится динамическое изображение следующего луча.

4 При необходимости, укажите направления других лучей с началом в заданной точку.

5 Чтобы завершить построение лучей, щелкните правой кнопкой мыши.

## 8.13 Прямые

Прямая линия - это линия, проходящая через указанную точку под заданным углом и имеющая бесконечную длину в обоих направлениях. Т.к. прямые линии имеют бесконечную длину, они не учитываются при определении границ чертежа.

По умолчанию для построения луча необходимо указать начальную точку и его направление.

Лучи и Прямые линии иногда используют в качестве конструкционных линий при создании других объектов.



Инструмент Прямая доступен:

- В меню Черчение
- На панели инструментов Черчение
- На панели ленты Черчение

### 8.13.1 Основные операции

1 Запустите команду XLINE.

Запрос программы:

Укажите точку на прямой [Горизонталь/Вертикаль/Угол/Биссектриса/Параллель]:

2 Укажите начальную точку прямой.

Появится динамическое изображение прямой.

3 Укажите направление прямой.

Появится динамическое изображение следующей прямой.

4 При необходимости, укажите направления других прямых, проходящих через заданную точку.

5 Щелкните правой кнопкой мыши для завершения построения.

## 8.14 Полилинии

Полилиния представляет собой замкнутую или незамкнутую последовательность соединенных между собой прямолинейных или дуговых сегментов, которые рассматриваются как цельный объект. Каждый сегмент полилинии может иметь постоянную или переменную ширину.



Команда PLINE создает замкнутые и незамкнутые полилинии по указанным точкам.

Команды BOUNDARY и -BOUNDARY формируют замкнутую полилинию на основе области, определяемой другими объектами.

Команда JOIN создает полилинии из серии соединенных между собой отрезков, полилиний и дуг.

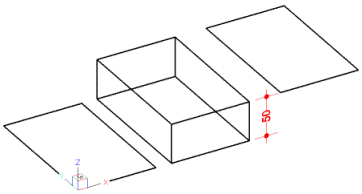
Инструмент **Полилиния** доступен:

- В меню Черчение
- На панели инструментов Черчение
- На панели ленты Черчение
- На закладке квадро-меню Черчение без выбора объектов.

Свойство **3D высота** определяет высоту полилинии. Если значение 3D высота = 0, визуальные отличия между полилиниями и серией отрезков или дуг отсутствуют.

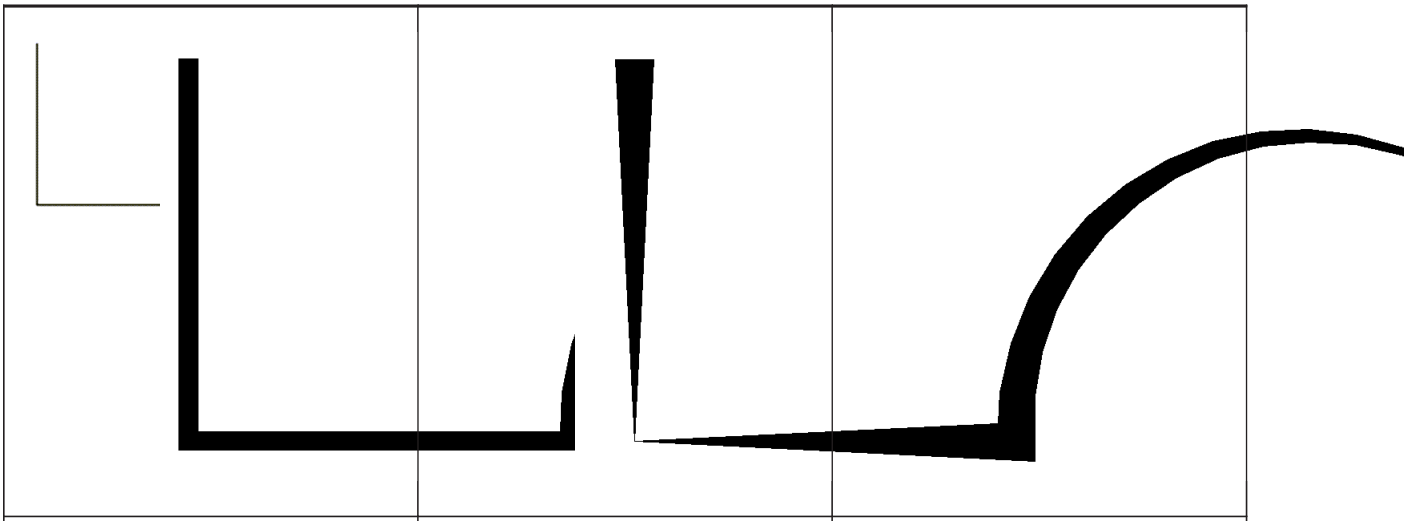
3D высота по умолчанию определяется значением системной переменной THICKNESS.

Свойство **Уровень** определяет расстояние между плоскостью полилинии и плоскостью XY WCS (Мировая система координат).




**THICKNESS**= 0 (слева),**THICKNESS**= 50 (по середине),**ELEVATION**= 50 (справа)

При редактировании полилинии Вы можете модифицировать полилинию в целом, или изменять отдельные ее сегменты, а также добавлять и удалять вершины.



| Полилиния с нулевой шириной | Полилиния с постоянной шириной | Полилиния с переменной шириной |
|---|---|---|

***Прим.:***

- Если значение системной переменной FILLMODE = 0, все объекты, имеющие заливку, в т.ч. полилинии и плоские фигуры, отображаются в виде контуров.
- Отображением узлов в конечных точках, вершинах и/или средних точках сегментов управляет системная переменная GRIPS
- Полилинии имеют направление, которое можно изменить с помощью опции **Обратить** команды PEDIT (Редактировать полилинию).

### 8.14.1 Основные операции

1 Запустите команду PLINE.

Запрос программы: Начало полилинии или [Продлить] <Последняя точка>:

2 Укажите точку.

Запрос программы: Укажите следующую точку или [Перейти к дугам/Расстояние/Полуширина/Ширина]:

3 Укажите точку.

Запрос программы: Укажите следующую точку или [Перейти к дугам/Расстояние/Продлить/Полуширина/Ширина/На шаг назад]:

4 Укажите точку.

Запрос программы:

Укажите следующую точку или [Перейти к дугам/Замкнуть/Расстояние/Продлить/Полуширина/Ширина/На шаг назад]:

5 Укажите другие точки.

6 Выполните одно из следующих действий:

- Выберите опцию **Замкнуть**, чтобы создать замкнутую полилинию.
- Выберите опцию **На шаг назад**, чтобы удалить предыдущую вершину.
- Для завершения команды щелкните правой кнопкой мыши или нажмите клавишу Enter.

### 8.14.2 Создание контурной полилинии

1 Запустите команду BOUNDARY.

Откроется диалоговое окно **Контур**.

2 Нажмите кнопку **Указать точки в пределах контуров**.

Диалоговое окно закроется.

3 Укажите точку внутри области, вокруг которой необходимо создать контурную полилинию.

Контурная полилиния будет выделена.

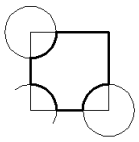
4 Укажите точки для создания других контурных полилиний.

5 Щелкните правой кнопкой мыши для подтверждения.

Диалоговое окно **Контур** снова откроется.

6 Нажмите кнопку **ОК**.

Будут созданы полилинии.



## 8.15 Мультилинии

Мультилинии состоят из двух или более параллельных линий в соответствии со стилем, определяемым с помощью команды MLSTYLE.

Инструмент Мультилиния доступен:

- в меню **Черчение**
- на панели инструментов **Черчение**

- на панели ленты **Черчение**

### 8.15.1 Основные операции

1 Запустите команду MLINE.

1 Текущие параметры: Выравнивание = Сверху, Масштаб = 20.00, Стиль = Standard

**Запрос программы**: Укажите начальную точку или [Выравнивание/Масштаб/Стиль]:

2 Укажите точку.

3 **Запрос программы**: Конечная точка или [Угол/Длина/На шаг назад]:

4 Укажите точку.

5 **Запрос программы**: Конечная точка или [Угол/Длина/Продлить/На шаг назад]:

6 Укажите точку.

7 **Запрос программы**: Конечная точка или [Угол/Длина/Продлить/Замкнуть/На шаг назад]:

8 Выполните одно из следующих действий:

- Укажите другие точки.
- Выберите опцию **Замкнуть** для создания замкнутой мультилинии.
- Щелкните правой кнопкой для завершения команды.

**Примечание**:

Мультилиниилинии создаются с использованием текущего стиля мультилиний, определяемого значением системной переменной CMLSTYLE.

Системная переменная CMLJUST управляет выравниванием мультилинии относительно указанных точек. Доступные опции:

- **Сверху**: По верхней линии мультилинии.
- **Нуль**: По линии нулевого смещения.
- **Снизу**: По нижней линии мультилинии.

Системная переменная CMLSCALE устанавливает масштаб мультилинии.

Свойства мультилинии могут быть изменены с помощью **Панели свойств**.

## 8.16 Drawing splines

A spline is an open or closed smooth curve defined by a set of points. You can use splines to draw curved lines, which cannot be drawn as a polyline.



The Spline tool is available on:

- The Draw menu.
- The Draw toolbar.
- The Draw ribbon panel.
- The Draw tab of the No Selection Quad.

### 8.16.1 General procedure

1 Launch the SPLINE command.

Prompts you: First point of spline:

2 Specify a point.

Prompts you: Second point:

3 Specify a point.

Prompts you: Set next point or [Close/Fit tolerance/Undo]:

4 (option) Repeat step 3 to define more points.

5 Right click to stop adding more points.

Prompts you: Select starting tangent point:

A line that is tangent to the spline displays dynamically from the start point of the spline.

6 Click to define the starting curve of the spline.

Prompts you: Select ending tangent point:

A line that is tangent to the spline displays dynamically from the end point of the spline.

7 Click to create the spline.

## 8.17 Эскизы

Эскиз представляет собой набор прямолинейных сегментов, созданных в виде отдельных объектов или полилинии.

Эскиз не сохраняется в чертеже до тех пор, пока Вы сами не "запишете" его.

Способ формирования эскиза из набора отрезков или полилинии определяется значением переменной SKPOLY (Создание эскиза).

Значение длины или приращения сегмента сохраняется в системной переменной SKETCHINC. Чем меньше приращение, тем точнее будет ваш эскиз и больше размер файла.



Инструмент Эскиз доступен:

- В меню Черчение
- На панели инструментов Черчение
- На панели ленты Черчение

### 8.17.1 Основные операции

1 Запустите команду SKETCH.

Запрос программы: Длина сегмента <текущее значение>:

2 Выполните одно из следующих действий:

- Нажмите клавишу Enter для подтверждения текущего значения.
- Введите значение расстояния в командной строке.

- Определите длину сегмента, указав две точки.

3 Щелкните кнопкой мыши, чтобы "опустить перо" и начать рисовать эскиз.

После указания длины сегмента символ курсора приобретает вид карандаша ()

Запрос программы:

Нажмите ENTER или щелкните мышью чтобы переключить [Удалить/Соединить/Сохранить и выйти/Выйти без сохранения/Сохранить]:

4 Щелкните для начала рисования эскиза: при перемещении курсора будет отображаться эскизная линия зеленого цвета.

5 Чтобы "поднять перо", щелкните кнопкой мыши. Чтобы снова "опустить перо", выполните повторный щелчок.

6 При необходимости, выберите опцию **Стереть**.

Символ курсора приобретет вид **ластика для стирания** ().

Переместите курсор к конечной точке линии эскиза для начала стирания.

Чтобы остановить стирание и продолжить рисование, щелкните кнопкой мыши.

7 Для сохранения эскиза выберите в контекстном меню пункт **Сохранить** или введите в командной строке R и нажмите клавишу Enter.

8 Для сохранения эскиза и завершения выполнения команды Эскиз выберите в контекстном меню пункт **Сохранить и выйти** или введите в командной строке X и нажмите клавишу Enter.

## 8.18 Окружности

По умолчанию для построения окружности необходимо указать ее центр и радиус.

Для создания окружности также могут использоваться другие методы, доступ к которым можно получить в подменю Окружность меню Черчение и на панели инструментов Окружности.

Команда AI_CIRCTAN строит окружности, касательные к трем объектам.



Инструмент Окружность доступен:

- В меню Черчение
- На панели инструментов Черчение
- На панели ленты Черчение
- На закладке квадро-меню Черчение без выбора объектов.

### 8.18.1 Основные операции

1 Запустите команду CIRCLE.

Запрос программы:

Укажите центр окружности или [2 точки/3 точки/Касательные и радиус/Преобразовать дугу в окружность/Несколько окружностей]:

2 Укажите точку.

Запрос программы: Укажите радиус или [Диаметр]:

3 Выполните одно из следующих действий:

- Нажмите клавишу Enter для подтверждения текущего значения радиуса.
- Введите в командной строке новое значение радиуса и нажмите клавишу Enter.
- Щелкните левой кнопкой мыши, чтобы определить радиус графически.

### 8.18.2 Построение окружности, касательной к трем объектам

1 Запустите команду AI_CIRCTAN.

Запрос программы: Укажите объект касания:

2 Укажите линейный объект.

В качестве линейных объектов могут быть использованы отрезки, дуги, окружности, сегменты полилиний, прямые и лучи.

Запрос программы: Укажите второй объект касания:

3 Укажите второй линейный объект.

Запрос программы: Укажите третий объект касания:

4 Укажите третий линейный объект.

Будет создана окружность, касательная к трем выбранным объектам.

### 8.18.3 Редактирование окружности

1 Выберите окружность.

Свойства окружности отображаются на Панели свойств.

2 Введите новое значение в полях**Center**,**Радиус**,**Диаметр**,**Длина окружности**или**Площадь**и нажмите клавишу Enter.

3 Нажмите клавишу Esc для завершения редактирования.

## 8.19 Drawing arcs

The default method for drawing circular arcs is to specify three points: the start point, a second point and the endpoint.

Other methods to draw arcs can be found in the Arc submenu of the Draw menu and in the Arcs toolbar.

The Arc tool is available on:

- The Draw menu.
- The Draw toolbar.
- The Draw ribbon panel.
- The Draw tab of the No Selection Quad.

### 8.19.1 General procedure

1 Launch the ARC command.

Prompts you: Specify the start point of the arc.

2 Specify a point.

Prompts you: Enter second point or [Angle/Center/Direction/End/Radius]:

3 Specify a point.

Prompts you: End point:

4 Specify a point.

### 8.19.2 Editing an arc

1 Select the arc.

The properties of the arc display in theProperties panel.

2 Type a new value in the**Center**,**Radius**,**Start angle**or**End angle**, then press Enter.

3 Press the Esc key to stop editing the arc.

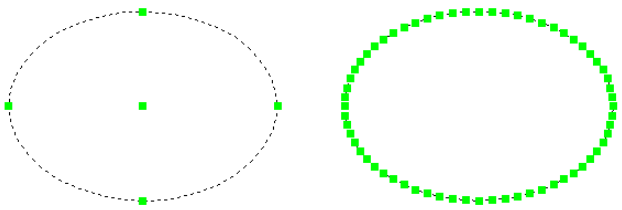
## 8.20 Эллипсы

По умолчанию для построения эллипса необходимо указать конечные точки одной оси эллипса и задать расстояние, равное половине длины второй оси. Конечные точки первой оси определяют расположение эллипса. Более длинная ось эллипса называется большой осью, а менее длинная - малой осью. Порядок указания осей не имеет значения. Программа определяет большую и малую оси только по их длине. Половина большой оси называется большим радиусом. Половина малой оси называется малым радиусом. Отношение малого радиуса к большому определяет величину эксцентриситета эллипса. Эллипс с эксцентриситетом равным 1 представляет собой окружность.

Для создания эллипса также могут использоваться другие методы, доступные в подменю Эллипс меню Черчение и на панели инструментов Эллипсы.

Способ создания эллипса также определяется значением системной переменной PELLIPSE (Полилиния - эллипс).

Отличие между "настоящим" эллипсом и эллипсом-полилинией становится заметным при выборе эллипса. "Настоящий" эллипс имеет пять захватов: в центре и конечных точках осей. Эллипс-полилиния представляет собой замкнутую полилинию из набора дуговых сегментов.



"Настоящий" эллипс (слева) и эллипс-полилиния (справа)

Инструмент Эллипс доступен:

- В меню Черчение
- На панели инструментов Черчение
- На панели ленты Черчение
- На закладке квадро-меню Черчение без выбора объектов.

### 8.20.1 Основной метод построения эллипса

1 Запустите команду ELLIPSE.

Запрос программы: Первая точка оси эллипса или [Дуга/Центр]:

2 Укажите точку:

Запрос программы: Вторая точка оси эллипса:

3 Укажите точку:

Запрос программы: Вторая ось эллипса или [Угол поворота вокруг оси]:

4 Укажите точку.

### 8.20.2 Основной метод построения эллиптической дуги

Эллиптическая дуга - это часть эллипса.

1 Запустите команду ELLIPSE.

Запрос программы: Первая точка оси эллипса или [Дуга/Центр]:

2 Выберите опцию Дуга.

Запрос программы:

3 Укажите точку:

Запрос программы: Вторая точка оси эллипса:

4 Укажите точку:

Запрос программы: Вторая ось эллипса или [Угол поворота вокруг оси]:

5 Укажите точку.

Запрос программы: Начальный угол дуги или [Параметр]:

6 Выполните одно из следующих действий:

- Введите значение угла в поле динамического ввода.
- Щелкните левой кнопкой мыши, чтобы определить дугу графически.

Запрос программы: [Параметр/Охваченный угол] <Конечный угол>:

7 Укажите точку на чертеже.

### 8.20.3 Редактирование эллипса или эллиптической дуги

1 Выберите эллипс или эллиптическую дугу.

Свойства эллипса отображаются на Панели свойств.

2 Введите новое значение в полях**Center**,**Радиус по главной оси**,**Радиус по вторичной**

**оси**,**Соотношение радиусов**,**Начальный угол**или**Конечный угол**и нажмите клавишу Enter.

3 Нажмите клавишу Esc для завершения редактирования.

***Прим.:*** При редактировании полей Начальный угол или Конечный угол эллипс превращается в эллиптическую дугу. Установка для обоих свойств значения 0 преобразует эллиптическую дугу в эллипс.

## 8.21 Прямоугольники

Прямоугольник представляет собой замкнутую четырехстороннюю полилинию.

Для построения прямоугольника необходимо указать расположение двух его противоположных углов. С помощью полей динамического ввода вы можете указать значения ширины и высоты прямоугольника.

Если при построении прямоугольника не была выбрана опция **Повернутый**, стороны прямоугольника всегда будут параллельны осям X и Y текущей системы координат (МСК или ПСК).



Инструмент Прямоугольник доступен:

- в меню **Черчение**
- на панели инструментов **Черчение**
- на панели ленты **Черчение**
- на закладке квадро-меню **Черчение** при **отсутствии выбранных объектов**

### 8.21.1 Основные операции

1 Запустите команду RECTANG.

1 **Запрос**:

Первый угол прямоугольника или [Фаска/Скруглить/Повернутый/Квадрат/3D уровень/3D высота/Ширина линии/Площадь/Размеры]:

2 Укажите точку.

3 Появится динамическое изображение прямоугольника.

**Запрос программы**: Противоположный угол прямоугольника:

4 Выполните одно из следующих действий:

- Укажите точку.
- Введите значения в полях динамического ввода.
- Для переключения между полями динамического ввода используйте клавишу TAB. Нажмите клавишу Enter для подтверждения.

***Прим.:***

***Т.к. прямоугольник представляет собой замкнутую полилинию, он имеет направление. Направление замкнутой полилинии считается положительным, если ее построение выполняется против часовой стрелки и отрицательным при построении по часовой стрелке. Прямоугольник имеет***

***положительное направление, если значения обеих координат (X и Y) второй точки выше или ниже значений координат первой точки (соответствует положительному или отрицательному направлению перемещения курсора вдоль обеих осей X и Y ). Прямоугольник имеет отрицательное направление, если значение одной из координат (X и Y) второй точки выше или ниже значения одной из координат первой точки (соответствует положительному или отрицательному направлению перемещения курсора вдоль оси X или Y ).***

***Для изменения направления полилинии используется опция Изменить направление команды PEDIT (Редактировать полилинию).***

## 8.22 Многоугольники

Многоугольники представляют собой замкнутые полилинии, содержащие не менее 3-х и не более 1,024 равных по длине сторон.

Вы можете создавать многоугольники с использованием следующих методов:

- **Центр - Вершина**: вначале указывается точка центра многоугольника, затем его вершина (метод указания радиуса описанной окружности).
- **Центр - Сторона**: вначале указывается точка центра многоугольника, затем точка середины его стороны (метод указания радиуса вписанной окружности).
- **Две вершины**: указывается длина стороны многоугольника.



Инструмент Многоугольник доступен:

- в меню **Черчение**
- на панели инструментов **Черчение**
- на панели ленты **Черчение**
- на закладке квадро-меню **Черчение** при **отсутствии выбранных объектов**

### 8.22.1 Основные операции

1 Запустите команду POLYGON.

1 **Запрос программы**: Укажите число сторон многоугольника или [Ширина линии/Несколько] <4>:

2 Выполните одно из следующих действий:

- Нажмите клавишу Enter или щелкните правой кнопкой мыши, чтобы принять текущее количество сторон.
- Введите значение и нажмите клавишу Enter.

**Запрос программы**: Укажите центр многоугольника или [По двум вершинам]:

3 Укажите точку.

4 **Запрос программы**: Укажите середину стороны многоугольника или [По вершине]:

5 Укажите точку.

***Прим.:***

- Опция, выбранная в шаге 3 (указание середины стороны или вершины) запоминается для многократного использования.
- Количество сторон определяется значением системной переменной POLYSIDES, но это значение между сеансами не сохраняется. При первом запуске команды POLYGON в текущем сеансе BricsCAD значение POLYSIDES всегда равно 4.

## 8.23 Drawing traces

Traces are 4-sided filled entities, created with two parallel sides. When you create a series of traces, they are L-connected to each other.



The Trace tool is available on

- the **Draw** menu.
- the **Draw** ribbon panel

The default trace width is saved through the TRACEWID system variable.

### 8.23.1 General procedure

1 Launch the TRACE command.

1 **Prompts you:** Width of trace <current width>:

2 Do one of the following:

- Right click to accept the current width.
- Type a value, then press Enter.
- Define a new current width graphically by clicking two points.

**Prompts you:** Start of trace:

3 Specify the start point of the trace.

4 **Prompts you:** Next point:

5 Specify the end point of the first trace.

6 A line indicates the direction of the first trace.

The direction of the second trace displays dynamically.

7 Do one of the following:

- Right click to create the trace.
- Specify the endpoint of the next trace.

8 Do one of the following:

- Repeat step 5 to keep adding traces.
- Right click to stop.

***Прим.:*** When the FILLMODE system variable is turned off, all filled entities, such as traces, display and print as outlines.

## 8.24 Кольца

Кольцо представляет собой окружность, созданную как замкнутая 2-сегментная широкая полилиния с заливкой.




По умолчанию для построения кольца необходимо задать внутренний и внешний диаметр, а затем указать точку центра кольца. Если внутренний диаметр равен 0, будет создана окружность с заливкой.



Инструмент Кольцо доступен:

- в меню **Черчение**
- на панели ленты **Черчение**

### 8.24.1 Основные операции

1 Запустите команду DONUT.

1 **Запрос программы**: Внутренний диаметр кольца [2 точки/3 точки/Касательные и диаметр] <текущий внутренний диаметр>:

2 Выполните одно из следующих действий:

- Щелкните правой кнопкой для подтверждения текущего значения внутреннего диаметра.
- Введите новое значение и нажмите клавишу Enter.
- Определите новое значение внутреннего диаметра графическим способом, указав две точки.

**Запрос программы**: Внешний диаметр кольца <текущий внешний диаметр>:

3 Выполните одно из следующих действий:

- Щелкните правой кнопкой для подтверждения текущего значения внешнего диаметра.
- Введите новое значение и нажмите клавишу Enter.
- Определите новое значение внешнего диаметра графическим способом, указав две точки.

**Запрос программы**: Центр кольца:

Укажите точку.

**Запрос программы**: Центр кольца:

4 Выполните одно из следующих действий:

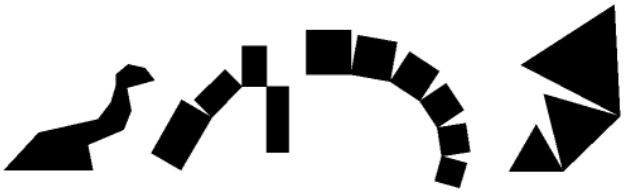
- Создайте другие кольца.
- Щелкните правой кнопкой мыши для завершения построения.

***Прим.:***

- Если Режим заливки (системная переменная FILLMODE) выключен, все объекты, имеющие заливку, в т.ч. полилинии и плоские фигуры, отображаются и выводятся на печать в виде контуров.
- Текущее значение внутреннего и внешнего диаметра сохраняется в значениях системных переменных DONUTID и DONUTOD. Эти значения не сохраняются между сессиями. При первом запуске команды DONUT в текущей сессии работы BricsCAD DONUTID = 0,5 и DONUTOD = 1.

## 8.25 Drawing solids

2D Solids are rectangular, square, triangular or quadrilateral areas filled with a solid color.



Left to right: quadrilateral, rectangular, square and triangular solids

The Solid tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel

### 8.25.1 General procedure

1 Launch the SOLID command.

1 **Prompts you**: Set first point of plane or [Rectangle/Square/Triangle]:

2 Specify a point.

3 **Prompts you**: Second point:

4 Specify a point.

5 **Prompts you**: Third point of plane:

6 Specify a point.

7 **Prompts you**: Fourth point:

8 Specify a point.

9 The solid is created.

**Prompts you**: Third point of plane:

10 Do one of the following:

- Keep specifying points to create more solids.
- Right click to stop.

***Прим.:*** When the FILLMODE system variable is turned off, all filled entities, such as solids, display and print as outlines.

## 8.26 Drawing revision clouds

A Revision Cloud is an open or closed polyline composed of arcs. Revisions clouds are used to for reviewing or redlining purposes to indicate the parts of the drawing that need to be adjusted or annotated.




Revision Clouds: 1. Normal 2. Calligraphy 3. Reversed calligraphy (based on a rectangle).

***Прим.:*** The **Reverse** option is available only in **Entity** mode or when creating an open revision cloud using the **Freehand** mode.

REVCLOUDCREATEMODE: sets the default mode.

REVCLOUDMINARCLENGTH: sets the minimum arc length.

REVCLOUDMAXARCLENGTH: sets the maximum arc length.

REVCLOUDARCSTYLE: sets the default style (normal or calligraphy).

REVCLOUDGRIPS: when On, grip editing a revision cloud also affects adjacent segments.



The Revision Cloud tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel
- the **Draw** tab of the **No Selection Quad**

### 8.26.1 General procedure

1 Launch the REVCLOUD command.

1 **Prompts you**:

Specify first point or [Arc length/Entity/Rectangular/Polygonal/Freehand/Style] <default mode>:

2 Specify a point to start creating a freehand mode revision cloud.

3 As you move the cursor, the revision cloud is being created.

4 Move the cursor to the start point to finish.

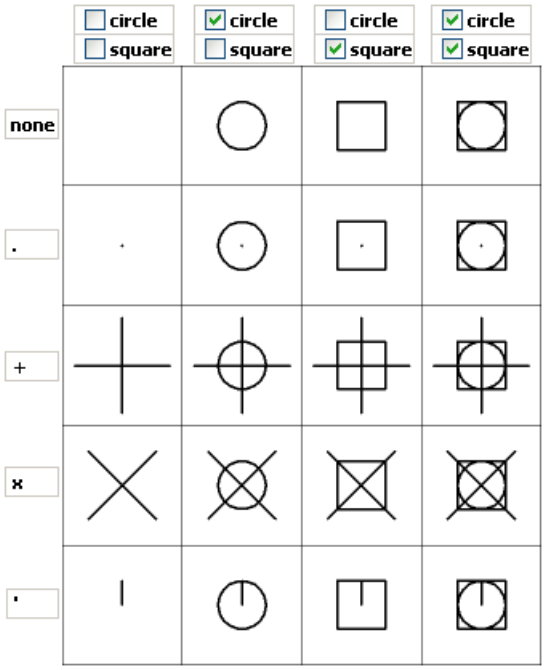
5 BricsCAD reports in the command line: Revision cloud finished.

***Прим.:***

- If you right click at the first prompt you start creating a revision cloud in the default mode (= current value of the REVCLOUDCREATEMODE system variable).
- If you right click before the cursor is at the start point, an open revision cloud is created.

## 8.27 Drawing points

Since points are dimensionless, a single pixel should be the (nearly) correct representation on the screen. But this is hardly visible, especially in a complex drawing. Therefore you can choose between a number of possible display styles, set by the PDMODE (Point Display Mode) system variable:




The size of the point representation style is controlled through the PDSIZE (Point Display Size) system variable. When PDSIZE=0 the point display size is 5% of the screen size.

***Прим.:*** If the PDSIZE=0 (zero), use the REGEN command to resize the display of the points to 5% of the screen size after zooming in or out.

The Point tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel

### 8.27.1 General procedure

1 Launch the POINT command.

1 **Prompts you**: Location of point or [point Settings.../Multiple points]:

2 Specify a point.

## 8.28 Drawing wipeouts

A wipeout is a polygonal area that hides the underlying entities using the current background color. Most commonly wipeouts are used to mask part of a drawing for clarity. For example, you may want to add text to a complicated part of a drawing. A wipeout could be used then to mask an area behind the text to increase the readability of the text.

The **Frame** option of the WIPEOUT command sets the value of the WIPEOUTFRAME system variable, which controls whether the wipe out outlines display or not, and whether they are printed or not.

***Прим.:*** Wipeouts hide existing entities only. Use the **Bring to front** option of the DRAWORDER command to hide all entities.



The Wipeout tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel
- the **Draw** tab of the **No Selection Quad**

### 8.28.1 General procedure

1 Launch the WIPEOUT command.

1 **Prompts you:** Specify start point or [Polyline/Frames]:

2 Specify a point.

3 **Prompts you**: Specify next point:

4 Specify a point.

5 **Prompts you**: Specify next point or [Undo]:

6 Specify a point.

7 **Prompts you**: Specify next point or [Undo/Close]:

8 Do one of the following:

- Keep specifying points.
- Right click or choose **Close** to stop.

### 8.28.2 To create a wipeout from a polyline

1 Launch the WIPEOUT command.

1 **Prompts you:** Specify start point or [Polyline/Frames]:

2 Choose the **Polyline** option.

**Prompts you**: Select a closed polyline [selection options (?)]:

3 Select a closed polyline, composed of line segments only.

4 **Prompts you**: Delete selected polyline? [Yes/No] <No>:

5 Do one of the following:
- Press Enter to keep the polyline.
- Choose **Yes** to delete the polyline.

## 8.29 Спирали

3D спираль - это кривая, поворачивающаяся вокруг оси по поверхности цилиндра (цилиндрическая спираль) или конуса (коническая спираль) с подъемом под постоянным углом вверх от основания.

2D спираль - это плоская кривая, которая огибает фиксированную центральную точку на постоянно увеличивающемся расстоянии от этой точки.

Команда HELIX может создавать 2D и 3D спирали.



Инструмент Спираль доступен:

- в меню **Черчение**
- на панели инструментов **Полилинии**
- на панели ленты **Черчение**

### 8.29.1 Основные процедуры при построении 2D спирали

1 Запустите команду HELIX.

1 Количество витков = 3 Закручивание = CCW

**Запрос программы:** Укажите центр основания:

2 Укажите точку.

3 Укажите радиус основания или [Диаметр] <1>:

4 Выполните одно из следующих действий:
- Укажите точку.
- Введите значение и нажмите клавишу Enter.

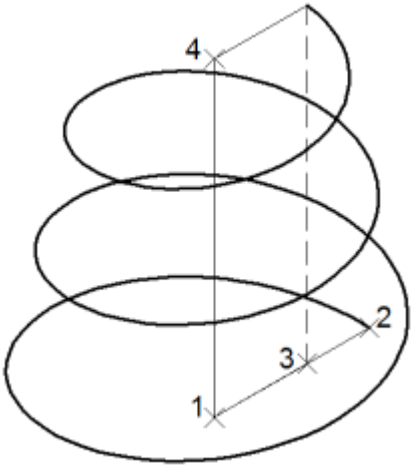
**Запрос программы:**Укажите верхний радиус или [Диаметр] <радиус основания>:

5 Выполните одно из следующих действий:
- Укажите точку.
- Введите значение и нажмите клавишу Enter.

**Запрос программы:**Укажите высоту спирали или [Конечная точка осей/Витки/Высота витка/Закручивание] <1>:

6 Введите 0 и нажмите клавишу Enter

### 8.29.2 Основные процедуры при построении 3D спирали




1 Запустите команду HELIX.

1 Количество витков = 3 Закручивание = CCW

**Запрос программы:** Укажите центр основания:

2 Укажите точку (1).

3 **Запрос программы:** Укажите радиус основания или [Диаметр] <1>:

4 Выполните одно из следующих действий (2):

- Укажите точку.
- Введите значение и нажмите клавишу Enter.

5 Выполните одно из следующих действий (3):

- Нажмите Enter, чтобы установить верхний радиус равным радиусу основания и создать цилиндрическую спираль.
- Укажите точку.
- Введите значение и нажмите клавишу Enter.

**Запрос программы:**Укажите высоту спирали или [Конечная точка осей/Витки/Высота витка/Закручивание] <1>:

6 Выполните одно из следующих действий (4):

- Укажите точку.
- Введите значение и нажмите клавишу Enter.

### 8.29.3 Редактирование спирали

1 Выберите спираль.

1 Свойства спирали отображаются в разделе **Геометрия** на **Панели свойств**.

2 Измените свойства:

- **Зависимость**: ограничивает высоту, высоту витка или количество витков. Неограниченные свойства настраиваются автоматически.

- **Высота**: общая высота спирали. высота = высота витка * количество витков.
- **Витки**: количество витков.
- **Высота витка**: высота одного витка.
- **Радиус основания**: начальный радиус.
- **Верхний радиус**: конечный радиус.
- **Закручивание**: направление CW (по часовой стрелке) или CCW (против часовой стрелки).

***Прим.:***

- Установите **Верхний радиус** = **Радиус основания** для создания цилиндрических спиралей.
- Установите нулевые значения параметров **Высота** или **Высота витка** для создания 2D спиралей.
- Команда EXPLODE преобразует спираль в сплайн.

## 8.30 3D полилинии

3D полилиния представляет собой набор линейных сегментов, расположенных в 3D пространстве.



Инструмент 3D Полилиния доступен:

- в меню **Черчение**
- на панели инструментов **Полилинии**
- на панели ленты **Черчение**

### 8.30.1 Основные операции

1 Запустите команду 3DPOLY.

1 **Запрос программы**: Начало полилинии или [Продлить] <Последняя точка>:

2 Укажите точку.

3 **Запрос программы**: Следующая точка:

4 Укажите точку.

5 **Запрос программы**: Следующая точка или [Замкнуть /На шаг назад]:

6 Выполните одно из следующих действий:

- Выберите опцию **Замкнуть**, чтобы соединить конечную и начальную точки и создать плоскую треугольную замкнутую полилинию.
- Укажите точку.

**Запрос программы**: Укажите следующую точку или [Замкнуть/На шаг назад]:

7 Выполните одно из следующих действий:

- Выберите опцию **Замкнуть**, чтобы соединить конечную и начальную точки.
- Укажите точку.

**Запрос программы**: Укажите следующую точку или [Замкнуть/На шаг назад]:

8 Продолжайте указывать точки или нажмите клавишу Enter для завершения.

### 8.30.2 Редактирование 3D полилинии

1 Выберите 3D полилинию.

1 Свойства выбранной полилинии отображаются на Панели свойств.

2 Выполните настройку свойств 3D полилинии в разделе **Дополнительно**:

- **Сглаживание**: выберите **Нет**, **Квадратичное** или **Кубическое**.

  При выборе значений **Квадратичное** и **Кубическое** 3D полилиния будет иметь вид сплайна.

- **Замкнутая**: При выборе значения **Нет** удаляется замыкающий сегмент полилинии.

## 8.31 Creating regions

The REGION command converts closed entities into a solid entity.

You can create regions from closed entities, such as polylines, polygons, circles, ellipses, closed splines and donuts.

Creating regions typically has no visible effect on a drawing using a wireframe visual style. However, if the original entity had a width or lineweight, that information is lost when you create the region.

In rendered visuals styles, region entities are displayed using a solid fill.



### 8.31.1 To create regions

1 Launch the REGION command.

1 **You are prompted**: Select entities or boundaries [selection options (?)]:

2 Select one or more entities.

3 Entities in set: x

**You are prompted**: Select entities or boundaries [selection options (?)]:

4 Select more entities or press Enter to stop selection.

5 BricsCAD reports: x region(s) created.

## 8.32 Creating hatches and gradient fills

The HATCH command fills an enclosed area in your drawing with a pattern.

The GRADIENT command creates solid fills in nine different patterns and one or two colors.

The area can either be a single entity such as a circle or a closed polyline or an area enclosed by a selection of entities.

To create the hatch you can click inside the closed perimeter of a boundary or you can select entities.

***Прим.:***

- Depending on the pattern, hatchings are memory intensive and might take a considerable amount of time to draw and display. To improve performance, add hatching as one of the last steps when you create a drawing, or insert hatches on a separate layer that you can freeze as you continue to work on your drawing. Or use the FILLMODE system variable to switch off the display of hatches, gradient fills and filled entities.
- The maximum number of dashes is controlled by the MAXHATCH settings variable. When the number of dashes is higher than the current MAXHATCH value, the hatch cannot be created. A warning box displays offering possible choices for subsequent actions. By default the maximum number of dashes is 100 000.



The Hatch and Gradient tools are available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel
- the **Draw** tab of the **No Selection Quad** (Hatch only)

### 8.32.1 General procedure

1 Launch the HATCH or GRADIENT command.

1 The **Hatch and Gradient** dialog displays.

2 Do one of the following:

3 (option) Click the **Hatch** tab to create a hatch, then define the hatch properties.

4 (option) Click the **Gradient** tab to create a gradient fill, then define the gradient fill properties.

5 Define the properties for the hatch or gradient fill.

6 Choose the boundary for the hatch or gradient fill.

7 Set the options.

8 Click the **OK** button to create the hatch or gradient fill.

Hatches and gradient fills are created on the layer defined by the HPLAYER system variable. Default = current layer.

### 8.32.2 To define hatch properties

1 Select the **Hatch** tab on the **Hatch and Gradient** dialog box.

2 (option) Click the **Inherit properties** button.

The **Hatch and Gradient** dialog closes temporarily.

**You are prompted**: Select hatch entity to copy properties from:

Select a hatch entity.

The **Hatch and Gradient** dialog reopens, displaying the properties of the selected hatch entity.

3 (option) Select the **Pattern Type**:
- **User Defined** - uses the pattern defined in this dialog box with the **Angle**, **Spacing**, and **Cross Hatch** options.
- **Predefined** - uses patterns defined in:
  - iso.pat (metric units -MEASUREMENT= ON)
  - default.pat (imperial units - MEASUREMENT = OFF).
- **Custom** - uses patterns defined in *.pat files (one pattern definition per file).

  BricsCAD searches for *.pat files in the folders that are defined in the SRCHPATH (Support File Search Path) system variable.

4 If the **Pattern Type** is **Predefined** or **Custom**:
- Double click the **Swatch** field,
- Select a **Pattern Name** in the **Hatch pattern** palette, then double click the pattern or click the **OK** Button.
- Type a scale in the **Scale** settings field or choose the scale from the drop down list.
- Type an angle in the **Angle** settings field or choose the angle from the drop down list.

5 If the **Pattern Type** is **User Defined**:
- Specify the **Pattern Spacing** in drawing units.
- Optionally set the **Cross Hatch** option.

6 Define the **Hatch Origin**.

The options are:
- **Use current origin** : The origin as defined by the HPORIGIN system variable is reused.
- **Specified origin** : Click the **Pick new origin** button, then pick a point in the drawing. The new origin is stored in the HPORIGIN system variable.

7 Set the **Color** for the hatch pattern.

8 Set the **Background color**.

### 8.32.3 To define gradient fill properties

1 Select the **Gradient** tab on the **Hatch and Gradient** dialog box.

2 (option) Click the **One color** radio button to create a 'color to white' or 'color to black' gradient fill.
- Click the colored tile to open the **Select color** dialog box, to pick a different base color.
- The color slider sets the transition from color to white or black.
- Press and hold the left mouse button to move the color slider.

  Move the slider to the far right to create a 'color to white' gradient fill.

  Move the slider to the far left to create a 'color to black' gradient fill.

3 (option) Click the **Two color** radio button to create 'two color' gradient fill.
- Click the colored tiles to open the **Select color** dialog to pick the base colors.
- Select one of the 9 gradient patterns.

- The options are: linear, cylindrical, inverted cylindrical, spherical, hemispherical, curved, inverted spherical, inverted hemispherical and inverted curved.

4 (option) **Centered**: If centered, the gradient fill is created symmetrical; if not centered, the gradient fill is built up from the left of the boundary.

5 **Angle**: Specifies the angle of the gradient fill, relative to the current UCS.

### 8.32.4 To define the boundary

1 Select a **Boundary retention** option.

2 Select an **Island** detection option.

This applies to enclosed areas inside hatch boundaries.

3 Set the **Boundary tolerance**. This value defines the maximum gap size in the boundary. When it is set to 0 (zero), BricsCAD sets the tolerance with respect to the current view size. Zoom out to increase the tolerance.

This value is saved through the HPGAPTOL system variable.

4 (option) Click the **Select** button to select a **Boundary set**.

The **Hatch and Gradient** dialog temporarily closes to let you select entities.

If a selection set was active when you launch the HATCH or GRADIENT command, this selection set will be used as the **Boundary Set**. In this case, the **Hatch and Gradient** dialog closes and immediately reopens when you click the **Select** button.

5 Do one of the following:

- Click the **Pick Points** button.

  The **Hatch and Gradient** dialog closes.

  **You are prompted**: Select a point to define a boundary or hatch area:

  • Specify a point in each of the areas you want to hatch.
  • **You are prompted**: Select a point to define a boundary or hatch area:
  • Right click or press Enter to stop adding areas.
  • The **Hatch and Gradient** dialog displays again.

- Click the **Select Boundary Entities** button.

  The **Hatch and Gradient** dialog closes.

  **You are prompted**: Select entities:

  • Select the entities you want to hatch.
  • Right click or press Enter to stop selecting entities.
  • The **Hatch and Gradient** dialog displays again.

- Click the **Remove Boundary Entities** button.

  The **Hatch and Gradient** dialog closes.

**You are prompted**: Select entities:

- Select the current boundary entities to be removed from the selection.
- Right click or press Enter to stop selecting entities.
- The **Hatch and Gradient** dialog displays again.

### 8.32.5 Set the options

- The **Annotative** option creates annotative hatches.
- Select the **Associative** option to associate the hatch with its boundary.
- Select the **Create Separate Hatches** option. When multiple boundaries are detected or multiple entities are selected, separate hatches are created, instead of a single hatch.
- Choose a **Draw order** option.

  This option is reset to **Send behind boundary** when the drawing is closed and reopened.
- **Layer**: choose an existing layer or **Use Current** (default).
- **Transparency**: sets the transparency; values between 0 and 90 are accepted.

## 8.33 Editing hatches and gradient fills

Hatches and gradient fills can be edited:

- In the **Properties** panel.
- Using the HATCHEDIT command, in a dialog box.
- Using tools in the Edit command group in the Quad when a hatch or gradient fill entity is selected.

### 8.33.1 To edit a hatch or gradient fill in the properties panel

1 Select the hatch or gradient fill in the drawing.

1 The properties of the selected entity display in the **Properties** panel.

2 Select the property you want to edit in the **Properties** bar.

The selected property field is activated.

3 Edit the selected property.

4 Press the Esc key to stop editing the hatch.

***Прим.:***

- When multiple hatches or gradient fills are selected properties which are different display as ***varies***. If you edit such property, the changes apply to entire selection.
- Linetype, Linetype scale and Lineweight properties apply to **User Defined** pattern types only.

### 8.33.2 To edit a hatch or gradient fill in a dialog box



The Edit Hatch tool is available on:

- the **Modify** menu
- the **Modify** toolbar

- the **Modify** ribbon panel

1 Do one of the following:
   - Launch the HATCHEDIT command.
   - Select a hatch or gradient entity.
   - Double click a hatch or gradient fill entity.

   The **Hatch Edit** dialog box displays.

2 Edit the properties and settings.

3 The changes are applied in real time.

4 Click the **OK** button to accept the changes.

### 8.33.3 To edit a hatch or gradient fill using the Quad

1 Pause the cursor on the hatch or gradient fill.

1 The Quad displays.

2 Move the cursor to the Quad.

3 (option) In the **Edit** tab, click the **Add Loop** button.

   **You are prompted**: specify internal point or [Select entities/Undo]:
   - Click inside a closed area,
   - or

     Type **S** , then press Enter to select entities.
   - Right click or press Enter to conclude.

4 (option) In the **Edit** tab, click the **Remove Loop** button.

   The boundaries of the selected hatch highlight.

   **You are prompted**: select entities:
   - Click the boundaries you want to remove.
   - Right click or press Enter to conclude.

5 (option) In the **Edit** tab, click the **Send to back** button.

   All hatches in the drawing are sent behind other entities.

6 (option) In the **Edit** tab, click the **Match** button.

   **You are prompted**: select entities to copy properties to or [Settings/selection options (?)]:
   - Select the hatch entities to copy the properties to.
   - Right click or press Enter to conclude.

7 (option) In the **Draw** tab, click the **Generate Boundary** button.

   The boundary of the hatch is generated.

# 9. 3D моделирование

## 9.1 Direct modeling overview

Direct modeling is much like sculpting on-screen: just like a sculptor starts with a lump of clay or a piece of stone or wood, you start with a base volume, which is then further detailed by adding, removing and deforming geometry. All operations are executed dynamically, and you decide whether to define dimensions intuitively or by specifying values. Unlike history-based modeling, the order in which features are created is not important. All operations are performed in interactive mode: the intermediate result of an operation shows dynamically, according to user input defined by mouse movement.

A key feature of direct editing capabilities provided by BricsCAD consists in preserving the design intent. 3D dimensional constraints (**Distance**, **Radius**, and **Angle**) imposed for faces and edges give you full parametric control over the solid model. Different configurations and scales of the model can be obtained simply by changing the parameter of a dimensional constraint. Geometrical constraints (**Coincident**, **Tangent**, **Parallel** and others) define how these changes influence the geometry of the model. However, there is no need to specify all geometrical constraints manually since most of them can be recognized automatically by setting **Design Intent Recognition** options.

Another feature of **Direct Editing** operations is that they can be easily applied with the Quad cursor menu. This tool provides a set of operations based on the currently selected or previewed entity or selection set. At the same time the Quad provides the other basic solid editing operations such as copy, union and subtract.

Direct modeling operations apply to all ACIS solid geometry, whether created in BricsCAD or imported. When the separately licensed BricsCAD Communicator is installed, the IMPORT command supports all major 3D model formats, both industry standard and proprietary formats.

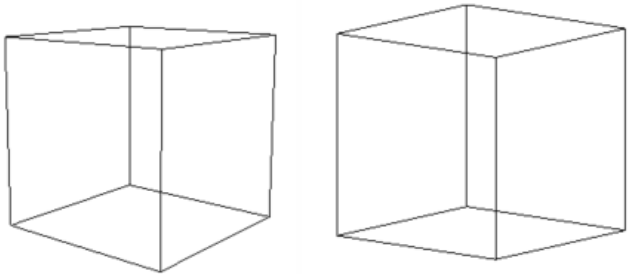
Depending on the value of the DELOBJ (Delete Entity) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities (see table below).

| Value | Result |
|---|---|
| 0 | Retain defining entities |
| 1 | Delete profile entities; retain paths and guide curves |
| 2 | Delete all defining entities |
| -1 | Prompt to delete profile entities; retain paths and guide curves |
| -2 | Prompt to delete all defining entities |

## 9.2 Navigate in 3D

As you navigate around and through your 3D model, the location of the model in space remains constant. It is your current view (viewpoint), of the model that is changing. You can see the current view properties in the **Properties** panel when nothing is selected. The **Camera** (X,Y,Z) represents your location as you view the model. Pretend you are viewing it through a camera lens. The **Target** (X,Y,Z) is where the camera lens is pointed. As you navigate around the model, the **Camera** and **Target** coordinates update accordingly. Those coordinates are read-only in the **Properties** panel. There are other **View** properties that you can edit. **Perspective** is the most important control to be aware of for navigation purposes.

You can set **Perspective** to **On** or **Off** for the current view. When set to **On**, the view displays the visual perspective mode (left). When set to **Off**, it displays the parallel perspective mode (right).



### 9.2.1 To control the Perspective property

Do one of the following:

- Select the **View / Perspective** property in the **Properties** panel.
- Right click the **Look From** widget, then choose **Perspective** in the context menu.

### 9.2.2 Mouse and keyboard navigation options

One of the easiest and most common methods for navigating around a 3D model is with the mouse and keyboard. Simply press the Shift key and middle mouse button as you move the mouse to orbit the model. The cursor changes to an orbit icon indicating you're in orbit mode.

There are many other mouse and keyboard navigation methods that you're probably familiar with from your 2D CAD experience. Most of those methods are also useful for 3D modeling.

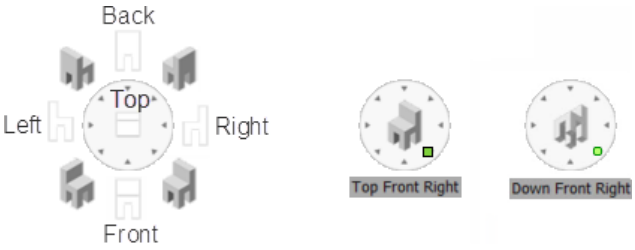
- Zoom in/out – Roll the mouse wheel. You can zoom all the way into a model and walk around inside it when in perspective mode. In isometric mode you can only zoom in very close, but not go inside.
- Zoom Extents – Double-click the mouse wheel.
- Pan – Press the mouse wheel or middle mouse button.

See the 'View Manipulation using the Mouse and Keyboard' article for a comprehensive list.

### 9.2.3 The LookFrom navigation tool

The **LookFrom** tool is displayed in the upper right corner of the **Drawing** area. Click on different places on the **LookFrom** tool to display the view from standard viewpoints. By default, it includes **Top**, **Left**, **Right**, **Front**, **Back**, **Top Back Left**, **Top Back Right**, **Top Front Left**, and **Top Front Right**. As you pass the cursor

over a viewpoint, the relevant icon appears at the center and a tooltip displays below. You can press the Ctrl key to access similar options for **Bottom** viewpoints.




A right-click menu offers access to additional controls and settings.

Choose **LookFrom Settings…** in the context menu to open the **Settings** dialog box and edit the **LookFrom** related system variables.

## 9.3 Direct modeling commands

BricsCAD direct modeling commands can create both solid and surface geometry. To distinguish between direct modeling commands and 'normal' commands, all direct modeling commands start with 'DM'.

Direct modeling commands are available in license levels Pro, BIM, Mechanical and Ultimate. Commands that create geometry are also available in BricsCAD Shape, along with basic modification commands such as DMPUSHPULL, DMCHAMFER and DMFILLET.

### 9.3.1 Creation commands:

DMEXTRUDE: creates 3D solids or surfaces by extruding open or closed 2D entities, faces of 3D solids, regions or closed boundaries. Optionally adds volume to solids or removes volume from solids.

DMREVOLVE: creates 3D solids or surfaces by revolution of open or closed 2D entities or regions about an axis.

DMTHICKEN: creates 3D solids by thickening (i.e. adding thickness to) surfaces, their faces, faces of 3D solids, regions and wire entities.

DMCOPYFACES: copies a set of 3D solid faces forming a feature (e.g. hole, rib or protrusion) to a new position on the same or another 3D solid.

DMSTITCH: converts a set of region and surface entities that bound a watertight area to a 3d solid.

### 9.3.2 Modification commands:

DMCHAMFER: creates equal distance and variable chamfers between adjacent faces sharing a sharp edge.

DMDEFORMCURVE: deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves.

DMDEFORMMOVE: deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges.

DMDEFORMPOINT: deforms a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction. The selected faces are deformed as smooth as possible.

DMFILLET: creates a smooth fillet between adjacent faces sharing a sharp edge.

DMPUSHPULL: adds or removes volume from a solid by moving/offsetting a face.

DMROTATE: rotates solids, faces, edges, or vertices of a solid around an axis.

DMTWIST: modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

### 9.3.3 Helper commands:

DMAUDIT: fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces).

DMAUDITALL: fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces) in drawings inserted as external references.

DMGROUP: creates group features, which are named persistent collections of faces and edges of 3D solids and surfaces.

DMSELECT: selects edges and faces of 3D solids or surfaces based on their geometric properties.

DMSELECTEDGES: selects the edges of faces and solids.

DMSIMPLIFY: simplifies the geometry and topology of 3d solid entities. The command removes unnecessary edges and vertices, merges seam edges, and replaces the geometry of faces and edges by analytic surfaces and curves, if possible within the user-specified tolerance.

The direct modeling commands are available in:

- The **Model** tab on the Quad cursor menu when the cursor is over an entity or subentity or when a selection set exists.
- The **Direct Modeling** Menu.
- The **Direct Modeling**, **Fillet**, **Chamfer** and **Deform** toolbars.
- The **Home / Direct Modeling** ribbon panel (**Modeling** workspace).

| Icon | Command | Applies to |
|---|---|---|
|  | | Any face of a 3D solid or surface. |
|  | | Cylindrical, spherical, conical, and toroidal faces of a 3D solid or surface.<br>Linear and circular edges of a 3D solid or surface.<br>Whole 3D solids, surfaces, and other entities. |
|  | | Planar, cylindrical, spherical, conical, and toroidal faces of a 3D solid or surface.<br>Whole 3D solids, surfaces, and other entities. |

| Icon | Command | Applies to |
|---|---|---|
|  | | 3D solids, surfaces, and their faces. |
|  | | Surfaces. |
|  | | 3D solids and surfaces. |
|  | | Planar faces of 3D solids and surfaces.<br>This tool launches the UCS command with the Face option selected.<br>***Прим.:*** When the Dynamic UCS feature is ON, the UCS aligns automatically to the solid face under the cursor and the UCS-icon changes accordingly. |
|  | Solid mode | Closed 2D entities and detected closed boundaries.<br>If the **Select detected boundaries** option of the SELECTIONMODES system variable is switched ON, boundaries enclosed by 2D linear entities in the XY-plane of the current UCS are detected automatically.<br>Planar faces of 3D solids and surfaces.<br>Closed 2D entities and detected closed boundaries on a planar face of a solid. |
|  | Solid mode | Closed 2D entities and detected closed boundaries.*<br>Planar faces of solids and surfaces.<br>Closed 2D entities and detected closed boundaries on the planar face of a solid. |
|  | Extract Faces | Surfaces and faces of 3D solids. |
|  | | A set of 3D solid faces. |
|  | | Faces of 3D solids and surfaces.<br>Surfaces.<br>Wire entities; lines, polylines, circles, ellipses, arcs, helices, and splines. |
|  | Constant radius mode | Sharp edges of a 3D solid or surface. |
|  | Variable radius mode | Sharp edges of a 3D solid or surface. |

| Icon | Command | Applies to |
|---|---|---|
|  | | Sharp edges of a 3D solid or surface. |
|  | Advanced Mode | Sharp edges of a 3D solid or surface. |
|  | | Faces of a solid or a surface.<br>Whole 3D solids and surfaces.<br>Free edges of a surface (a surface edge is called "free" if it has only one adjacent face)<br>Edges between coplanar faces. |
|  | Surface mode | 3D curves and edges of 3D solids and surfaces. |
|  | Surface mode | 3D curves and edges of 3D solids and surfaces. |
|  | | Faces of a 3D solid or a surface. |
|  | | Edges of a 3D solid or a surface. |
|  | | Edges of a 3D solid or a surface. |
|  | | 3D solids, surfaces or regions. |

## 9.4 Direct modeling operations

### 9.4.1 Controlling the coordinate system

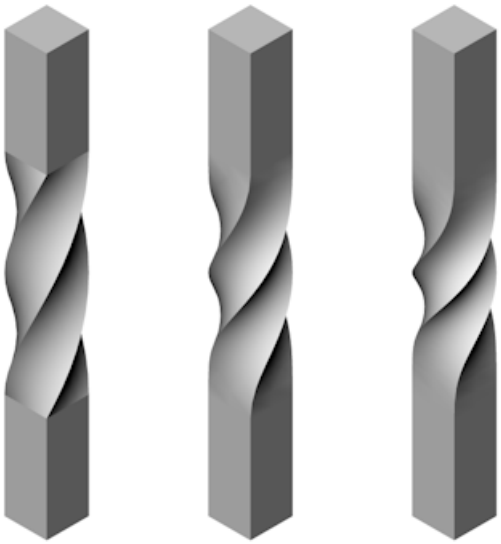
Since entities are created in the XY-plane of the current coordinate system (UCS), it is necessary to align the UCS with the 3D solid face. You have two options here:

- The **Align UCS** tool () launches the UCS command with the **Face** option selected.
- Click the **DUCS** field in the **Status Bar** to toggle the dynamic UCS feature ON and OFF. When ON, the UCS aligns automatically to the solid face under the cursor when executing a command and the UCS-icon changes accordingly. Press the Tab key to select an obscured face. Press the Shift key to temporarily lock the UCS, which allows you to work outside the hovered entity or face. Press the Shift key again to unlock it.

### 9.4.2 Selecting geometry

- Hit the Tab key to select obscured geometry.

- The SELECTIONMODES system variable controls which subentities highlight: edges, faces, and/or closed boundaries. Hold down the Ctrl key to overrule SELECTIONMODES, and select the subentity under the cursor.
- Window selection: to select the faces or edges of 3D solids and surfaces, press the CTRL key to cycle the options. The icon next to the cursor indicates what will be selected.
- The DMSELECT command allows you to select edges and faces based on their geometrical and topological properties:
  - Edges of a particular length.
  - Edges that form a concave/convex network.
  - Edge loops on the border of a set of connected faces.
  - Faces of a particular area.
  - Smoothly connected faces (G1 - tangent faces, or G2 - curvature continuity).



G0: No continuity (left), G1: Tangent Faces (middle), and G2: Curvature continuity (right)

  - Fillets of a specific radius.
  - Cylindrical protrusions of a specific radius.
  - Protrusion or depression features; pockets, extrusions, or ribs.

The value of a particular length, area, or radius can be taken from a selected edge/face.

### 9.4.3 Moving faces to control dimensions

A dynamic dimension displays with the DMPUSHPULL command.

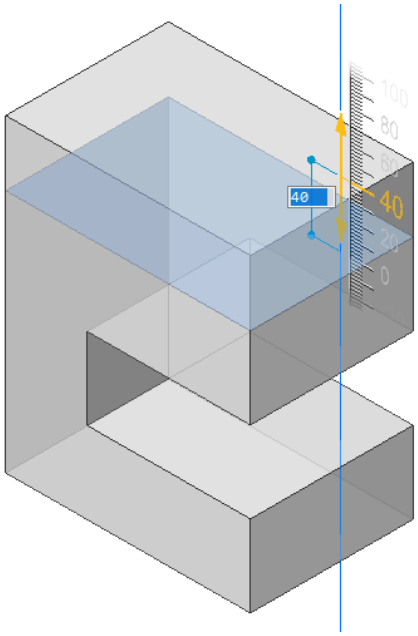
Make sure the **Quad** and **Dynamic Dimensions** are switched ON.

1 Hover the cursor over the face you want to move.

1 The face highlights.

2 Choose **Push/Pull** ( ) in the **Model** tab of the Quad.

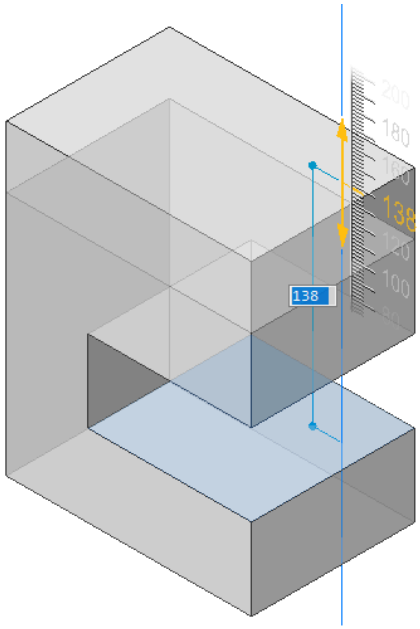
The selected face moves with the cursor and a dynamic dimension field displays the relative distance along with a ruler.




When the cursor is on the ruler, the face moves with fixed intervals. Zoom in or out to increase or decrease the interval.

When the cursor is outside the ruler, the face moves freely.

3 (Option) repeatedly hit the Tab key to see the distance with respect to an internal face, parallel to the selected face (left) or the bottom face (right).




Press and hold the Shift key and hit the Tab key to reverse the display order of reference faces.

4 Do one of the following.
   - Type a value in the dynamic dimension field, then hit Enter.
   - Using the ruler: click when the desired value displays.

### 9.4.4 Making holes

**Method 1**:

1 Create the subtractor solid,

2 Use the SUBTRACT command.

3 Depending on the value of the DELETETOOL system variable, the subtractor solid is either deleted or retained.

**Method 2**:

1 Create the outline of the opening on the 3D solid face.

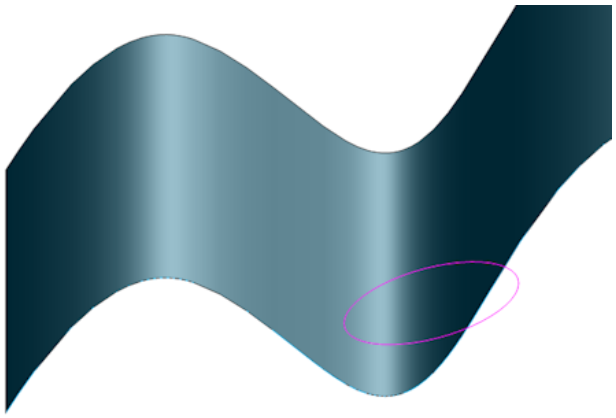
2 Place the cursor over the outline, then choose **Solid Extrude** ( ) (DMEXTRUDE command) in the **Model** tab of Quad.

3 Move the cursor inside the solid to make the opening.

4 Do one of the following:
   - Specify a point.
   - Type a value.

**To make a hole in a surface**

1 Create a 2D closed entity, e.g. an ellipse.

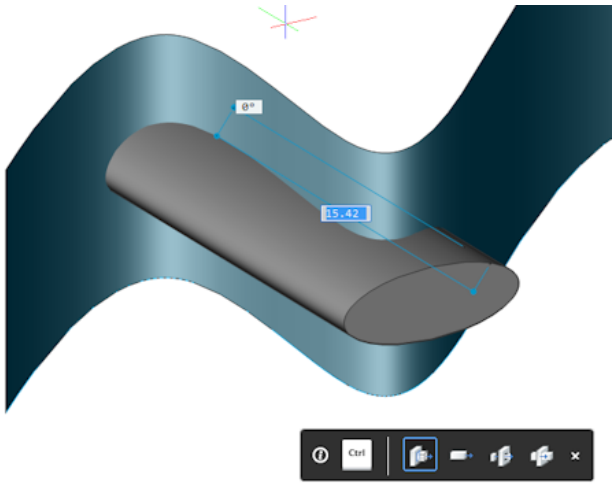


2 Place the cursor over the 2D closed entity, then choose **Solid Extrude** ( ) (DMEXTRUDE command) in the **Model** tab of Quad.

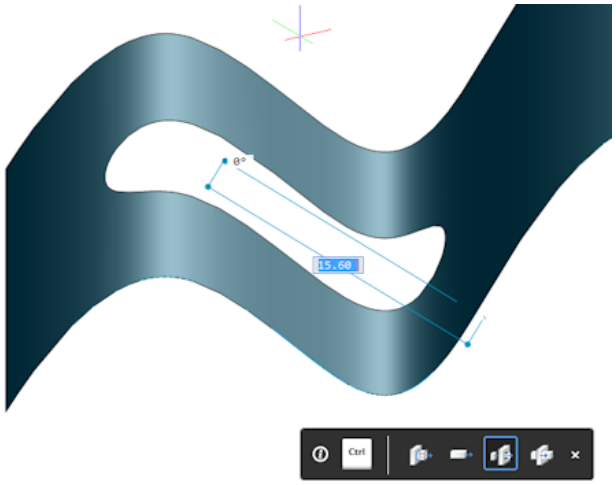
3 Extrude the 2D entity in the direction of the surface.

4 The default extrusion direction is perpendicular to the plane of the 2D entity.

The **Hotkey Assistant** (HKA) appears at the bottom of the screen.




5 Hit the CTRL key twice to select the **Subtract** option of the DMEXTRUDE command.




6 Click to create the opening.

### 9.4.5 Deleting features

The **Feature** option of the DMSELECT command allows to easily select all faces of a hole, depression or protrusion.

In the **Mechanical** workspace, the **Protrusion** and **Depression** tools are available in the **Select** panel of the **Solid** ribbon tab and the **Select** tab of the Quad.

The tools sit in the **Sheet Metal** toolbox. See **Available Tools** in the **Customize** dialog box.

**To delete a depression (hole or pocket) or a protrusion feature (extrusion or rib)**

1 Click the **Protrusion** ( ) or **Depression** ( ) tool button in the **Select** panel of the **Solid** ribbon tab or from the **Select** command group of the Quad.

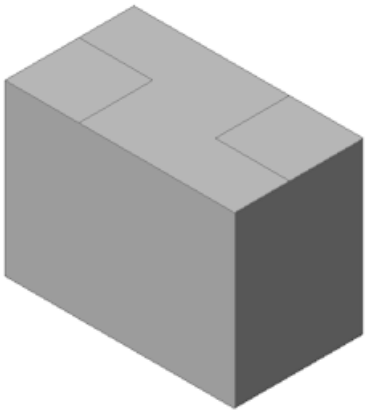
2 Select one or more faces of the feature.

3 All adjacent faces of the feature are detected.

4 Click the **Erase** ( ) tool button.

### 9.4.6 Removing edges between coplanar faces

When two or more faces are made coplanar, e.g. with the DMPUSHPULL command, it might be necessary to remove the shared edges using the DMSIMPLIFY command.

| | |
|---|---|
|  |  |
| Two depressed faces | Two depressed faces moved to the top face, to create three coplanar faces. |

1 +Launch the DMSIMPLIFY command.

1 **You are prompted**: Select entities to simplify or [Entire model/SEttings/selection options (?)] <Entire model>:

2 Select the solid, then press Enter.

3 The edges are removed.

### 9.4.7 Copying a Feature

The DMCOPYFACES command allows you to copy one or more features from a 3D solid (depressions - holes, grooves, and pockets - or protrusions - extrusions and ribs), to a new location on the same or a different 3D solid.

Make sure dynamic UCS is enabled.

1 Select all faces of the entire feature(s).

2 Click the **Copy Faces** ( ) tool:
- on the ribbon: **Solid** | **Edit** | **Copy Faces**.
- on the **Direct Modeling** toolbar.
- in the menu bar: **Model** | **Direct Modeling** | **Copy Faces**
- in the **Model** tab on the Quad

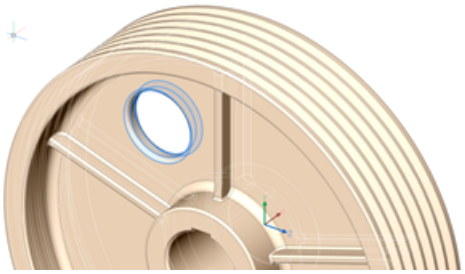
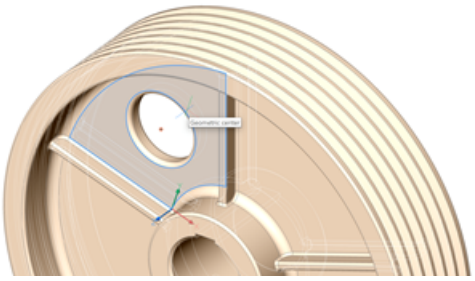
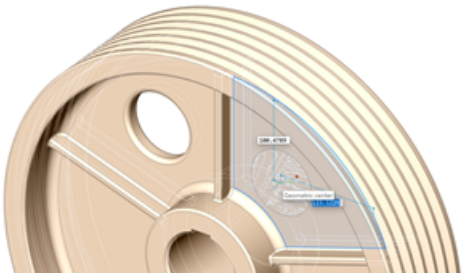
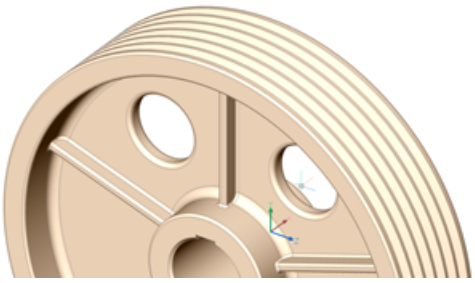
3 Specify the base point.

4 **You are prompted:** Select an insertion point or [Rotate/mUltiple/MOde]:

5 (Optional):
   - Use the **Rotate** option to rotate the faces around the Z axis of the UCS.
   - Use the **Multiple** option to insert the multiple copies of a feature.

6 Specify the insertion point.

7 The feature is copied from the base point to the insertion point with dynamic UCS.

| | |
|---|---|
|  |  |
| Select the faces of the entire feature | Specify base point |
|  |  |
| Specify the insertion point | The feature is copied |

### 9.4.8 Deformable modeling

Use the tools on the **Deform** toolbar, to deform a set of adjacent faces on a 3D solid or a surface, to create complex aesthetic shapes.

The faces are always deformed if:

- The external boundary (= edges, which are adjacent to only one of the selected faces) remains unchanged.
- The internal boundaries (= edges, which are adjacent to two of the selected faces) are deformed together with the faces.
- The initial continuity between deformed faces (G1 – tangent faces, or G2 – curvature continuity) is kept when deformed.

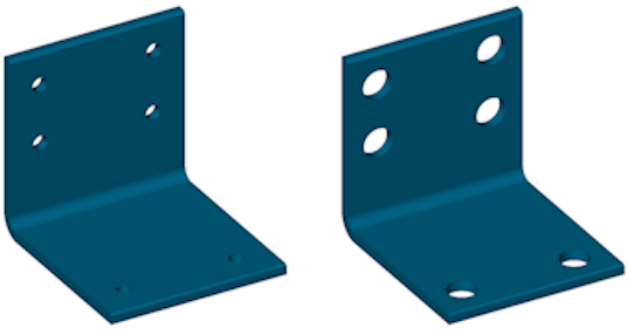
For a twist deformation, it is possible to specify the continuity between the deformed and the fixed portion of the entities.

Deformation can be defined in several ways:

- DMTWIST command: Twist the whole entity or a portion.
- DMDEFORMPOINT command: Move a point on a face in any direction in 3D space.
- DMDEFROMMOVE command: Move or rotate one or more edges.
- DMDEFORMCURVE command: Replace an edge with a given 3D curve.

## 9.5 Design intent recognition

Direct modeling operations allow you to change geometry of 3D solid models easily: when a single face of a solid is modified with a particular operation other faces are modified accordingly in order to keep design intent of the model and its topological and geometrical validity. For example, consider a model with a number of holes of the same radius. When the radius of a cylindrical face is modified with a push/pull operation or by changing the value of a radial constraint other faces that share the same diameter are recognized by BricsCAD and modified automatically to keep the radii synchronized.



Apart from the recognition of equal radii, BricsCAD automatically recognizes other geometrical relations between surfaces of a solid and preserves them during direct modeling operations.

To define the design intent recognition, click the buttons on the **Design Intent** toolbar. A pressed button indicates the feature is selected. An unpressed button indicates the feature is not active.

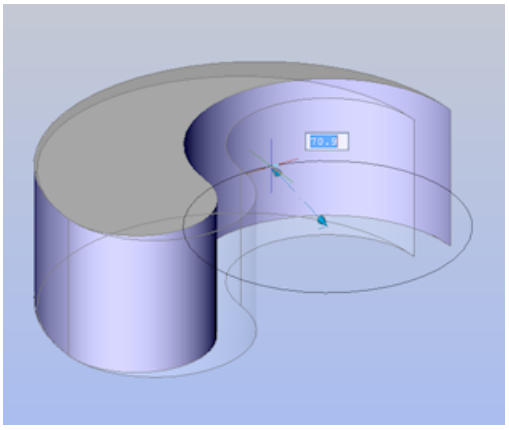
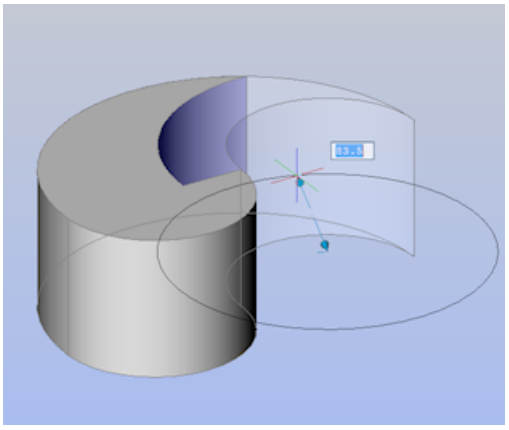
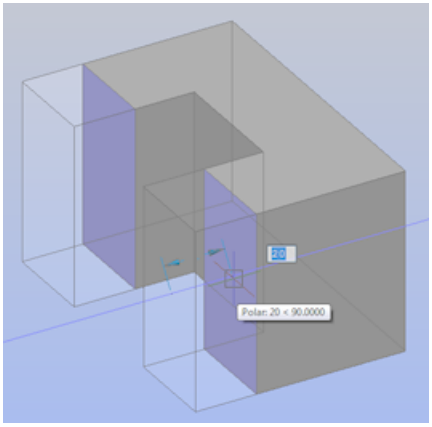
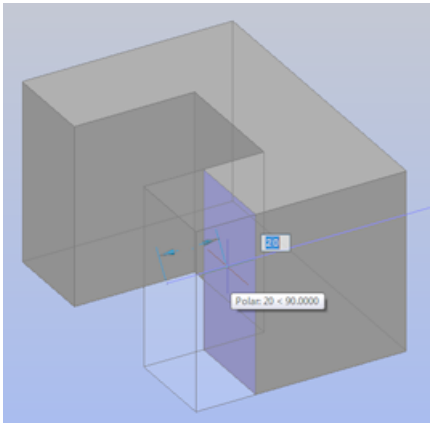
- tangent surfaces (planes, cylinders and cones) ()
- coincident planes ()
- parallel planes ()
- perpendicular planes ()
- cylinders perpendicular to planes ()
- coaxial surfaces (cylinders and cones) ()
- surfaces of equal radius (cylinders and spheres) ()
- vertices between 4 or more faces ()
- edges between coincident faces ()
- switch off/on all ()

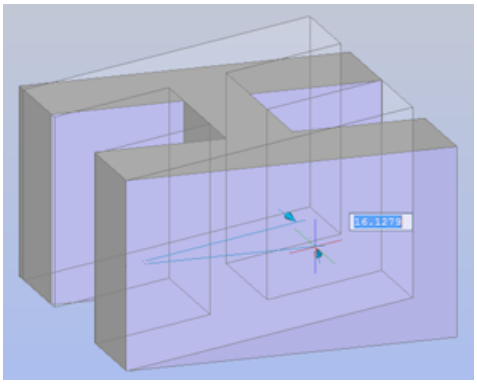
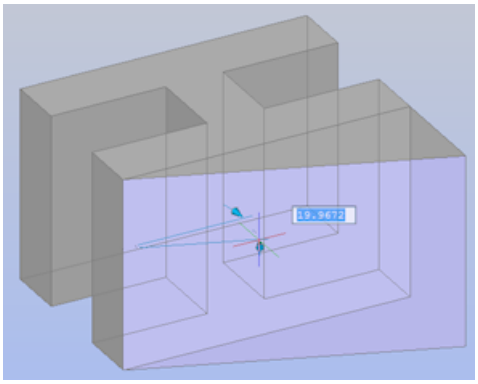
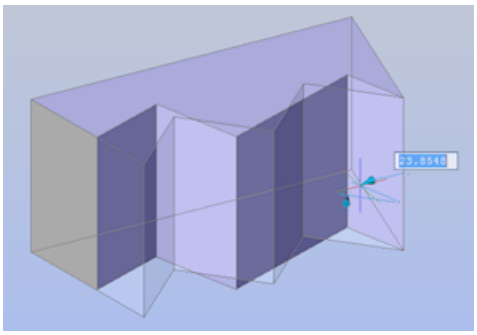
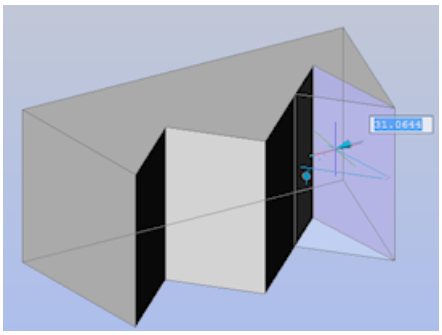
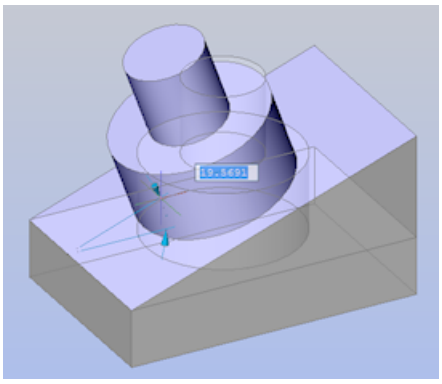
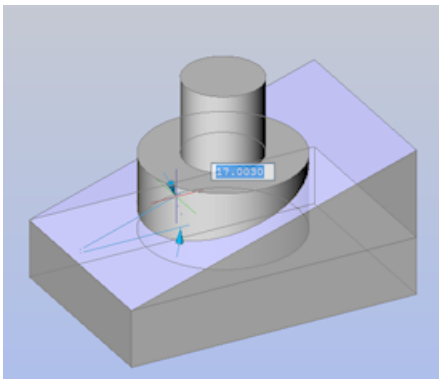
The design intent options can also be set by editing the DMRECOGNIZE system variable in the **Settings** dialog box.

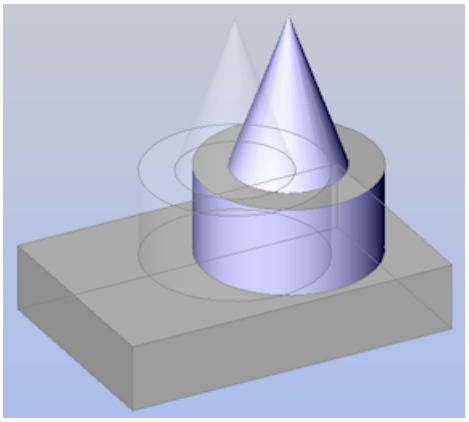
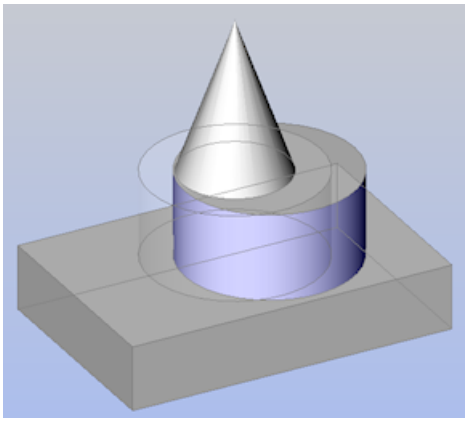
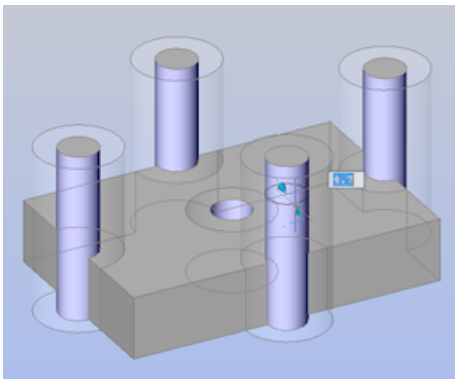
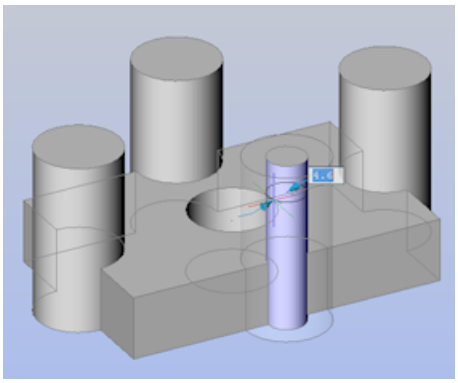
Click the **Options** tool button ( ) on the **3D Constraints** toolbar to set the **Automatic 3D geometry constraints recognition**. The state of tool buttons on the **Design Intent** toolbar (pressed or unpressed) is adjusted accordingly.

***Прим.:*** 3D Constraints which are explicitly defined by the user are taken into account when solid model is being modified with direct modeling commands. These constraints have a higher priority than automatically recognized ones and allow the user to better control the behavior of solid model modifications.

***The following examples illustrate the difference in the behavior whether automatic 3D geometry constraints are recognized or not.***

| | |
|---|---|
|  |  |
| tangent surfaces recognized ( ) | tangent surfaces not recognized ( ) |
|  |  |
| coincident plane recognized ( ) | coincident planes not recognized ( ) |

| | |
|---|---|
|  |  |
| parallel planes recognized () | parallel planes not recognized () |
|  |  |
| perpendicular planes recognized () | perpendicular planes not recognized () |
|  |  |
| perpendicular cylinders recognized () | perpendicular cylinders not recognized () |

| | |
|---|---|
|  |  |
| coaxial surfaces recognized ( ) | coaxial surfaces not recognized ( ) |
|  |  |
| equal radius recognized ( ) | equal radius not recognized ( ) |

## 9.6 Working with sections

The SECTION command creates section planes of 3D solids; the results are region entities.

The SECTIONPLANE command creates a section entity that creates sections of 3D solids, surfaces, and meshes.

The LIVESECTION command toggles the **Live Section** property of a section plane.

The SECTIONPLANESETTINGS command defines the properties of a section plane entity in the **Drawing Explorer - Section Planes** dialog box.

The SECTIONPLANETOBLOCK command saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

***Прим.:***

***When using a BIM license level:***

- The BIMSECTION command creates **BIM section** entities, which are Section entities with extended properties, such as type (plan, section, elevation) and initial scale of the 2D section result.

- The BIMIFY command can assign BIM properties to a Section entity.

### 9.6.1 Difference between Live Section and Clip Display

In both statuses all drawing content is sectioned and clipped-away parts are no longer displayed.

**Live Section**: entities in a drawing are temporarily modified, e.g. clipped solids are replaced by temporary solids with a different shape.

**Clip Display**: the display of entities is clipped. Entities are not replaced by temporary versions. Clip Display allows to continue using all modeling and editing operations on all drawing content. As opposed to the Live Section status, the Clip Display status can be set for multiple section entities simultaneously.

### 9.6.2 Align a section plane to a face

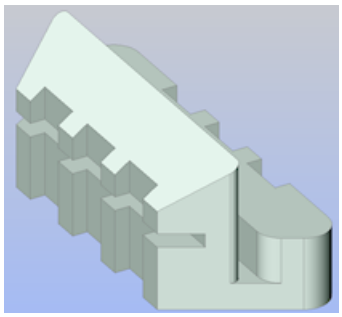
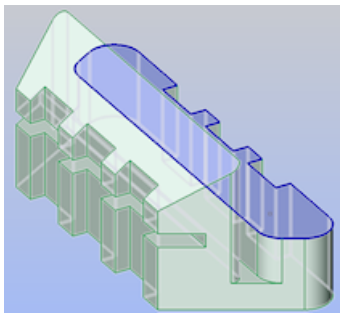
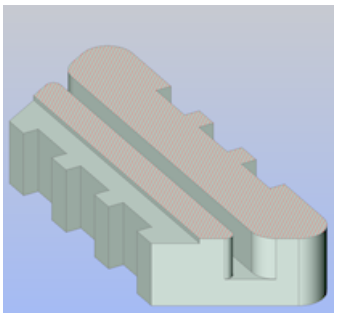
1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Select Face** option.

**You are prompted**: Select entity:

3 Select the face of a solid you want to align the section plane to.

| | | |
|---|---|---|
|  |  |  |
| Solid | Select face | Section plane aligned to face |

4 The section plane is aligned to the selected face.

5 The **Clip Display** property of the section entity is switched **On** automatically.

6 (option) Edit the properties of the section plane in the **Properties** panel.

### 9.6.3 To create a vertical section plane

***Прим.:*** The WCS must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

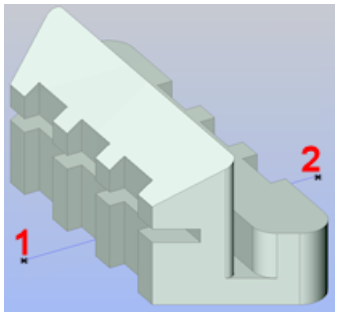
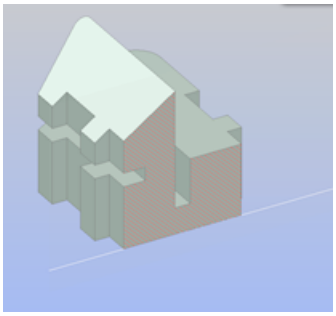
2 Specify two points.

3 The section plane is a vertical plane created through these points.

The part of the solid at the right hand side (with respect to the draw order of the points) of the section

plane is cut away.

The **Clip Display** property of the section entity is switched **On** automatically.

| | |
|---|---|
|   |  |
| Specify two points | Vertical section plane |

4 (option) Edit the properties of the section plane in the **Properties** panel.

### 9.6.4 To create a horizontal section plane

1 Choose **Front** in the **Look From** widget.

2 Align the UCS to the front view:

3 Use the **View** option of the UCS command.

***Прим.:*** If UCSORTH0 = ON and NAVVCUBEORIENT = 0, the UCS aligns with the view automatically.

4 Launch SECTIONPLANE command.

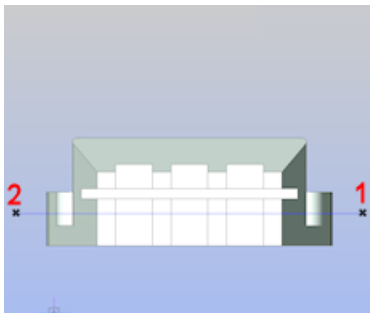
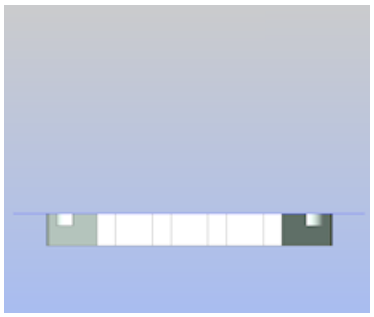
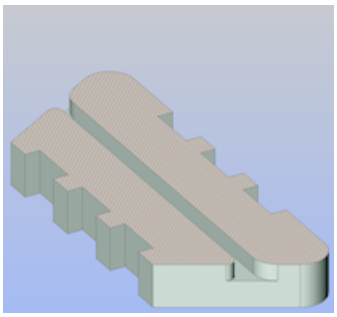
5 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

6 Specify two points.

7 The section plane is created through these points and perpendicular to the XY-plane of the current coordinate system.

8 The part of the solid at the right hand side (with respect to the draw order of the points) of the section plane is cut away.

9 The **Clip Display** property of the section entity is switched **On** automatically.

| | | |
|---|---|---|
|   |  |  |

| | | |
|---|---|---|
| Specify two points | The part of the solid at the right hand side of the section plane is cut away | Horizontal section plane |

10 (option) Edit the properties of the section plane.

### 9.6.5 To create a vertical jogged section plane

***Прим.:*** The WCS must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Draw** option.

**You are prompted**: Specify start point:

3 Specify the first point (1).

4 **You are prompted**: Specify next point:

5 Specify the second point (2):

6 **You are prompted**: Specify next point or ENTER to finish:

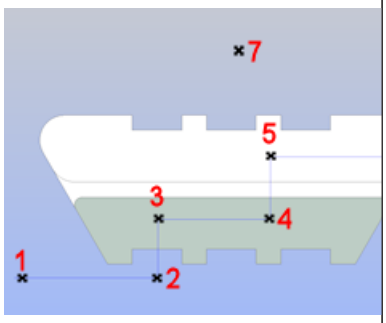
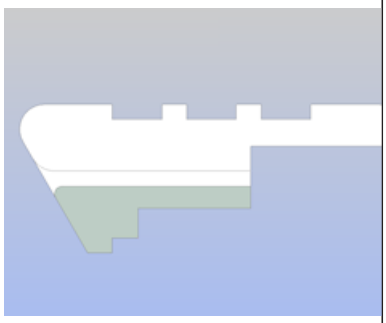
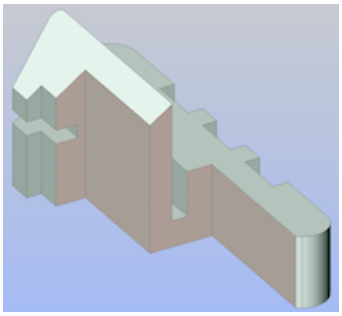
7 Specify the other points (3 - 6), then right click or press Enter to stop.

8 **You are prompted**: Specify the direction of the section entity:

9 Click a point (7) to specify the direction of the section entity.

10 The part of the solid opposite to the direction of the section entity is cut away.

The **Clip Display** property of the section entity is switched **On** automatically.

| | | |
|---|---|---|
|  |  |  |
| Specify points | The part of the solid opposite to the direction of the section entity is cut away | Vertical jogged section plane |

11 (option) Edit the properties of the section plane in the **Properties** panel.

### 9.6.6 To create a horizontal jogged section plane

1 Choose **Front** in the **Look From** widget.

2 Align the UCS to the front view:

3 Use the **View** option of the UCS command.

***Прим.:*** If UCSORTH0 = ON and NAVVCUBEORIENT = 0, the UCS aligns with the view automatically.

4 Launch SECTIONPLANE command.

5 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

6 Choose the **Draw** option.

**You are prompted**: Specify start point:

7 Specify the first point (1).

8 **You are prompted**: Specify next point:

9 Specify the second point (2).

10 **You are prompted**: Specify next point or ENTER to finish:

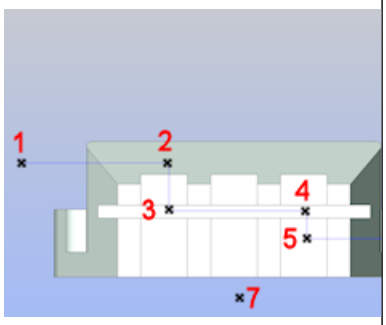
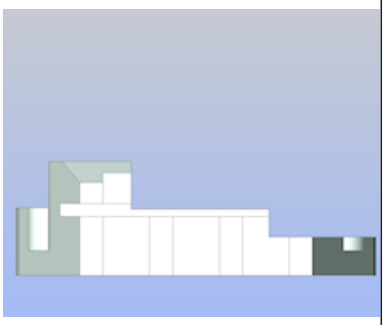
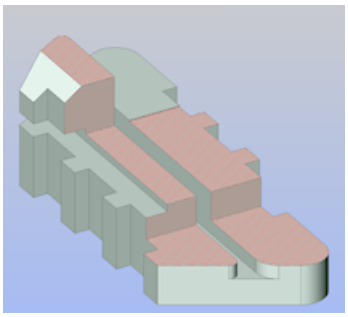
11 Specify the other points (3 - 6) , the right click or press Enter to stop.

12 **You are prompted**: Specify the direction of the section entity:

13 Click a point (7) to specify the direction of the section entity.

14 The part of the solid opposite to the direction of the section entity is cut away.

The **Clip Display** property of the section entity is switched **On** automatically.

| | | |
|---|---|---|
|  |  |  |
| Specify points | The part of the solid opposite to the direction of the section entity is cut away | Horizontal jogged section plane |

15 (option) Edit the properties of the section plane in the **Properties** panel.

### 9.6.7 To create an orthographic section plane

***Прим.:*** The WCS must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Orthographic** option.

**You are prompted**: Align section to [Front/bAck/Top/Bottom/Left/Right] <Top>:

3 Pick the orthographic section orientation of your choice in the prompt menu or type the corresponding option in the Command line.

4 Orthographic section planes are created through the center of the solid, parallel to the XY-plane (Top and Bottom), YZ-plane (Left and Right) or XZ-plane (Front and Back) of the current coordinate system.

The **Clip Display** property of the section entity is switched **On** automatically.

5 (option) Edit the properties of the section plane in the **Properties** panel.

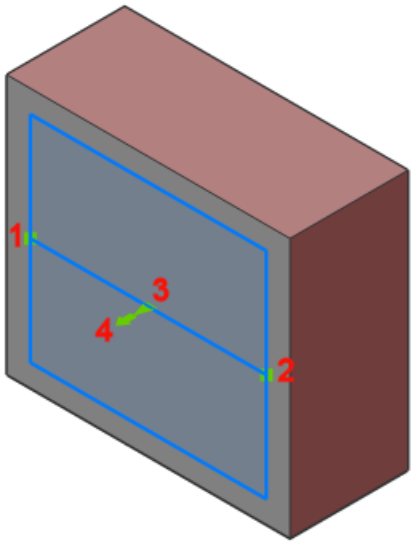
### 9.6.8 To modify the size and position of a section plane

1 Select the section plane entity in the drawing.

1 Depending on the current state of the section plane entity a number of grips display.

2 (option) Switch on the **Clip Display** property of the section plane in the **Properties** panel to see the result of the modifications dynamically.

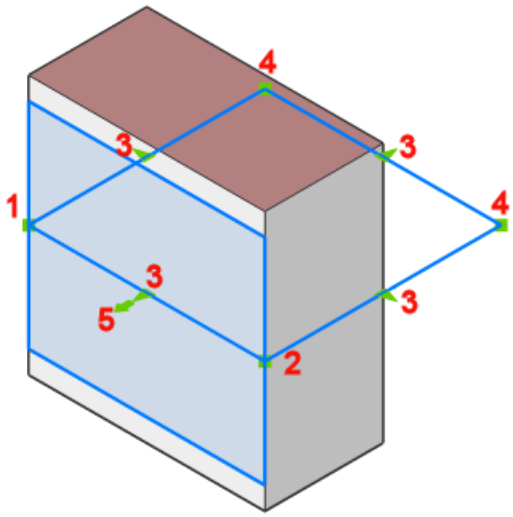
3 (option) Set the **State** property to **Plane**:




Four grips display:

- 1 ( ) = origin point: moves the position of the section plane in the XY-plane.
- 2 ( ) = through point: redefines the through point of the section plane.
  The section plane is rotated around the origin point (1).
- 3 ( ) = midpoint: moves the section plane parallel in a direction perpendicular to the section plane.
- 4 ( ): click to flip the view direction of the section plane.

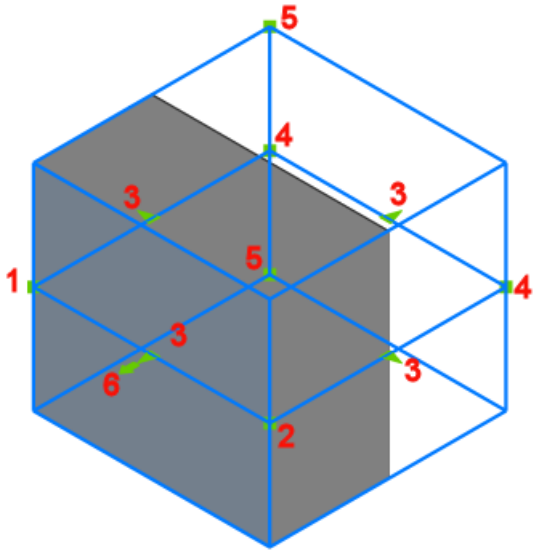
4 (option) Set the **State** property to **Boundary**:




Ten grips display.

- 1 ( ) = origin point: moves the position of the section plane in the XY-plane.
- 2 ( ) = through point: redefines the through point of the section plane.
  The section plane boundary is stretched and rotated around the origin point (1).
- 3 ( ): stretches the section plane boundary parallel.
- 4 ( ): modifies the shape of the section plane boundary.
- 5 ( )

5 (option) Set the **State** property to **Volume**:




Twelve grips display.

- 1 ( ) = origin point: moves the position of the section plane in the XY-plane.

- 2 () = through point: redefines the through point of the section plane.
  The section plane volume is stretched and rotated around the origin point (1).
- 3 (): stretches the section plane volume parallel.
- 4 (): modifies the shape of the section plane volume.
- 5 (): modifies the height of the section plane volume.
  **Polar Tracking** must be on to move grip 5.
- 6 (): click to flip the view direction of the section plane.

### 9.6.9 To save a section

The SECTIONPLANETOBLOCK command allows to:

- Insert a section in the drawing as a 2D section / elevation block or a 3D cutaway block.
- Replace a section block (e.g. when the section entity has been modified).
- Save a section to a new drawing.

## 9.7 Create a TIN Surface

### 9.7.1 Commands

TIN, TINASSIGNIMAGE, TINPROJECT

### 9.7.2 Overview

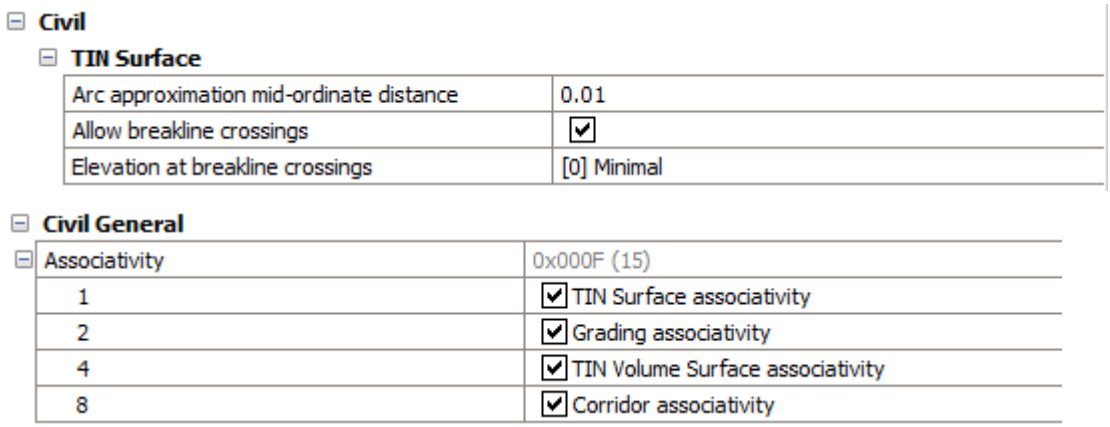
In BricsCAD, topographical surfaces are created using the TIN Surface tool (triangulated irregular network) or the TIN command.

A TIN surface uses two types of entry data: points and breaklines (polylines).

- Points contain X, Y coordinates and Z values. All points are used to make a connection with the two nearest neighbors to create triangles. The surface triangulation is based on the Delaunay algorithm, which ensures that no point is inside the circumcircle of any triangle.
- Breaklines represent linear infrastructure features like curbs, retaining walls, etc. These lines also define the edges of triangles. Breaklines can be created from linear entities such as line, polyline, arc or circle.
- Boundaries are the features to define bounded 3D surfaces. The surface boundaries can be created with closed polylines. Defining outer boundaries on a large 3D surface improves the performance of a TIN Surface plan.

Creating a TIN Surface is a process that needs to be followed to map a particular area in detail. On BIM models, these topographic surfaces are used to represent the site. The topographical surface can be generated based on drawing entities, data imported from a comma-delimited file format (CSV) or text file format (TXT), and points placed at different elevations, created from Civil 3D surface.

### 9.7.3 Setting the TIN Surface preferences




#### Arc approximation mid-ordinate distance

***Прим.:*** Mid-ordinate distance is the maximal distance between arc and chord (straight) segment which is used for arc approximation.

Defines how many points along the curve you want to add. When you have polylines with curves, this parameter allows you to tessellate the arcs in the polyline. The default value of this parameter is variable in BricsCAD predefined templates. For instance, the distance value is set to 0.01 in BIM-m, 1in BIM-cm, 10inBIM-mm, and 0.4in BIM imperial templates.

#### Allow breakline crossings

Defines if crossings breaklines are allowed. By default, it is set to **On**. If breakline crossings are allowed, intersections between breakline segments are calculated and added as points to the TIN Surface.

#### Elevation at breakline crossings

Defines how elevation at crossing breaklines should be determined. There are three options (e.g., minimal, maximal, and average) for defining breaklines elevation at the intersection.

#### Associativity

Enable/disable grading, surface and corridor associativities.

By default, the system variable is set to 15.

0: Neither TIN surface, grading nor corridor associativity is enabled.

1: Enables TIN surface associativity.

2: Enables grading associativity.

3: Both TIN surface and grading associativities are enabled.

4: Enables TIN volume surface associativity.

5: Both TIN surface and TIN volume surface associativities are enabled.

6: Both grading and TIN volume surface associativities are enabled.

7: TIN Surface, Grading and TIN volume surface associativities are enabled.

8: Enables corridor associativity.

9: Both TIN surface and corridor associativities are enabled.

10: Both grading and corridor associativities are enabled.

11: TIN surface, grading and corridor associativities are enabled.

12: Both TIN volume surface and corridor associativities are enabled.

13: TIN surface, TIN volume surface and corridor associativities are enabled.

14: Grading, TIN volume surface and corridor associativities are enabled.

15: All TIN surface, grading, TIN volume surface and corridor associativities are enabled.

### 9.7.4 Properties of a TIN Surface

| | |
|---|---|
| **General** | |
| Handle | 93 |
| Color | Green |
| Layer | 0 |
| Linetype | ByLayer |
| Linetype scale | 1 |
| Plot style | ByColor |
| Lineweight | ByLayer |
| Transparency | ByLayer |
| Hyperlink | |
| History | Current |
| **3D Visualization** | |
| Material | ByLayer |
| **Data** | |
| Name | Site |
| Description | School playground |
| **TIN Visibility** | |
| Boundary line | Off |
| Points | Off |
| Triangles | On |
| Contours | Off |
| **TIN Contours** | |
| Major contours interval | 10 m |
| Minor contours interval | 1 m |
| Major contours color | Red |
| Minor contours color | Green |
| **TIN Statistics** | |
| Number of points | 4264 |
| Number of triangles | 8230 |
| Minimum elevation | 4.4 m |
| Maximum elevation | 18.9192 m |
| 2D area | 10363.7386 m² |
| 3D area | 11990.9728 m² |

**TIN Visibility**

**Boundary line**
Change boundary line visibility.

**Points**
Change points visibility.

**Triangles**
Represents how the elevation points are connected to each other. It follows the Delaunay triangulation method where breaklines are not included in densification.

**Contours**
Represents the joining of points of equal elevation within a line.

**TIN Contours**

**Major contours interval**
Sets the major contours interval.

**Minor contours interval**
Sets the minor contours interval.

**Major contours color**
Defines the color of the major contour.

**Minor contours color**
Defines the color of the minor contour.

**TIN Statistics**

**The number of points**
Displays the number of points that define the TIN surface.

**The number of triangles**
Displays the number of non-overlapping triangles in the TIN surface.

**Minimum elevation**
Displays the minimum elevation value found in the TIN surface.

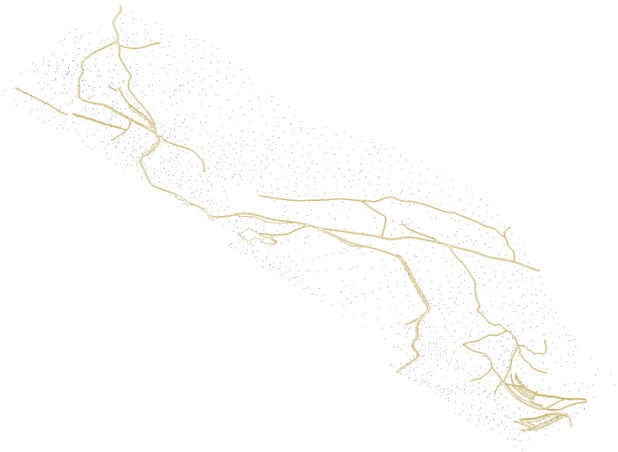
**Maximum elevation**
Displays the maximum elevation value found in the TIN surface.

**2D area**
Displays the 2D area calculated from the flat TIN surface.

**3D area**
Displays the 3D area that is calculated from the TIN surface.

### 9.7.5 Procedure: Creating a TIN Surface from a selection set of points and polylines

1 Open the drawing file that contains a set of points and polylines.

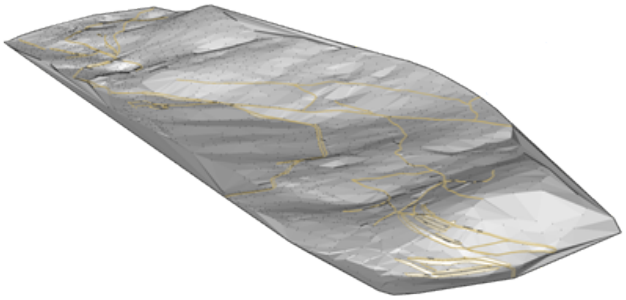


2 Launch the TIN command.

3 Select the entities in the drawing area to create a toposurface.

4 Press ENTER.

The toposurface is created from selected entities. Points are added to the surface as points, 3D polylines are added to the surface as breaklines.



### 9.7.6 Procedure: Creating a TIN Surface from an imported points file (.txt or .csv)

1 Launch the TIN command.

2 Select **Import from file** in the Command line.

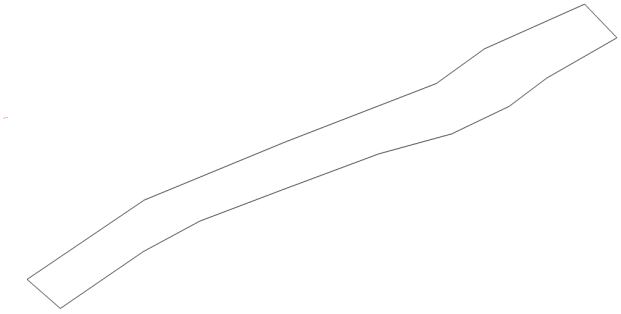
3 Choose the point data file you want to open or choose multiple files to import all at once.

4 Click Open to import the toposurface.

***Прим.:*** If the toposurface is not shown in the current viewport, use zoom extents to display it to the extents of the entities.

### 9.7.7 Procedure: Creating a TIN Surface of a selected area using Clip polygon

1 Open the drawing file that contains closed polyline.



2 Launch the TIN command.

3 Select the **cLip polygon** option in the Command line.

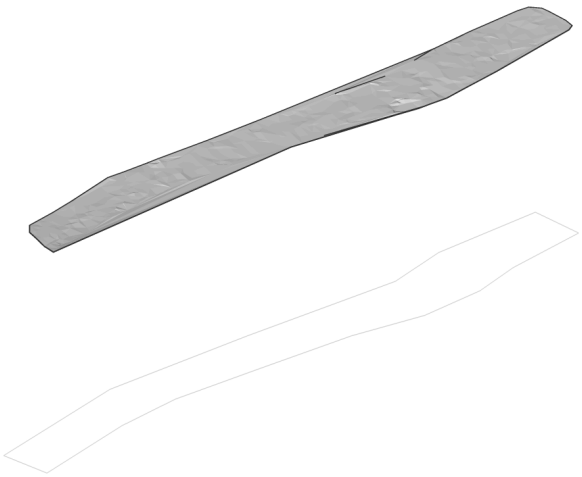
4 Select the closed polyline to use.

5 Specify if the crossed breaklines should be removed.

***Прим.:*** If the surface we are going to make contains breaklines, we define what should happen to the breaklines crossing the clipped polygon.

6 Select any of the options, **Import from file**, **Create from Faces**, **Create from civil 3D surface**.

***Прим.:*** Clip polygon works as a projection on the TIN surface. The TIN surface is created only in the area defined by the clip polygon.



### 9.7.8 Procedure: Creating a TIN Surface by placing points

1 Open a drawing.

2 Launch the TIN command.

3 Select the **place Points** option in the Command line.

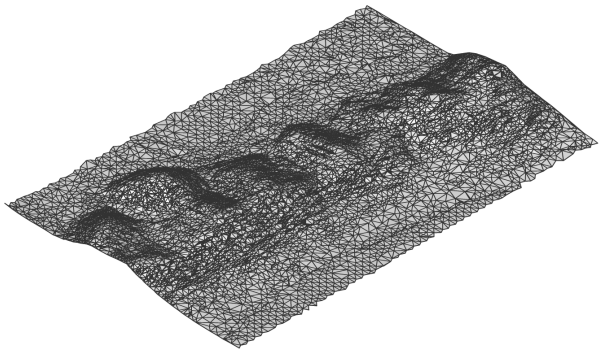
4 Pick a point where you want to start creating a TIN surface.

Specify the elevation value of each of those points, then press ENTER to accept.

***Прим.:*** At least three points are needed to create a TIN surface.

### 9.7.9 Procedure: Creating a TIN Surface from Faces

1 Open a drawing file that contains 3D faces.



2 Launch the TIN command.

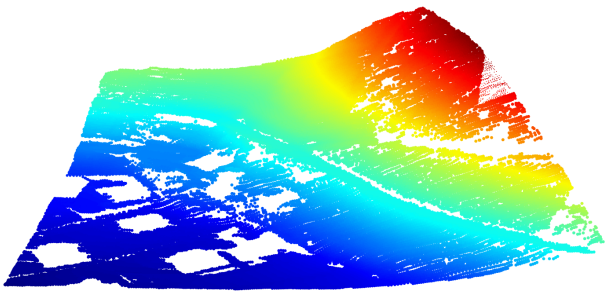
3 Select the **create from Faces** option in the Command line.

4 Select desired 3D faces.

5 Press Enter.

### 9.7.10 Procedure: Creating a TIN Surface from a Point Cloud

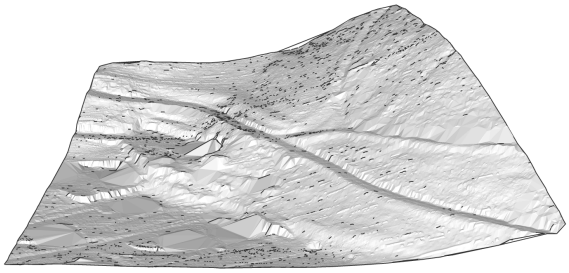
1 Open the drawing file that contains a point cloud.



2 Launch the TIN command.

3 Select the point cloud in the drawing area to create a toposurface.

4 Press ENTER. The toposurface is created from the point cloud.



### 9.7.11 Procedure: Creating a TIN Surface from a Civil 3D surface

1 Open a drawing with a Civil 3D surface.

2 Launch the TIN command.

3 Select **Create from civil3d surface**.

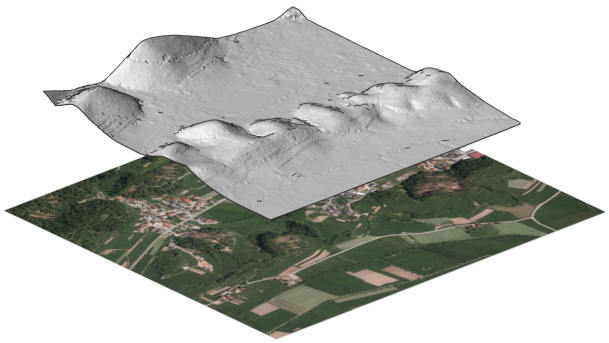
4 Select the Civil 3D surface. A TIN surface is created.

### 9.7.12 About TIN Assign Image

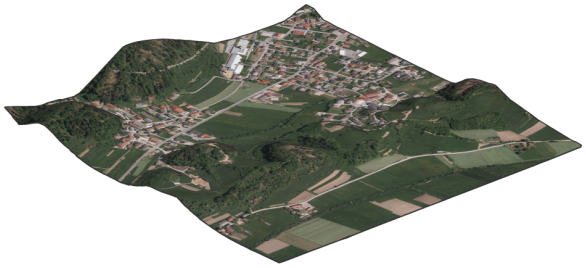
1 Open the drawing file that contains a TIN surface and an image.

2 Launch the TINASSIGNIMAGE command.

3 Select the TIN surface.



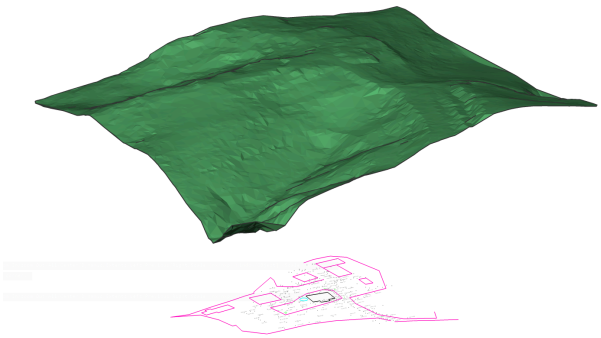
4 Select the raster image.



***Прим.:*** If the image is not shown on the TIN surface, change the Visual Style to Modeling or Realistic and set TIN surface triangles to ON.

### 9.7.13 About TIN Project

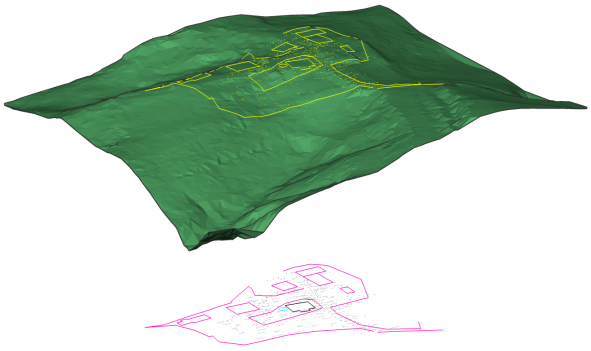
1 Open the drawing file that contains the TIN surface and the entities (points, blocks, text, lines, polylines, circles…).



2 Launch the TINPROJECT command.

3 Select the entities to project.

4 Select if the original entities are to be erased.



## 9.8 TIN Surface Edit

## 9.9 TIN Surface Direct Modeling

### 9.9.1 Commands

TINEXTRACT, TINMODIFY, TINMERGE

### 9.9.2 Overview

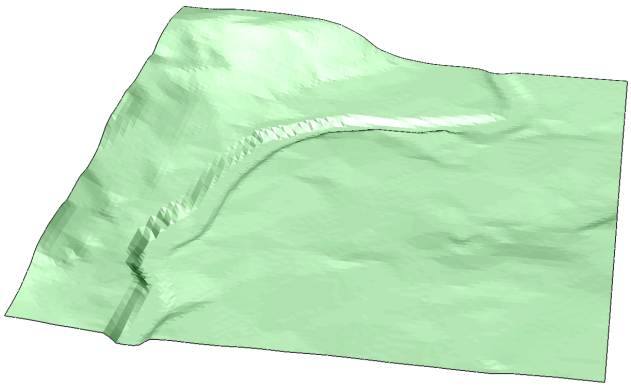
In BricsCAD, topographical surfaces created as a TIN Surface, can be adjusted using different tools.

### 9.9.3 About TINEXTRACT

The TINEXTRACT command is used to generate a solid, mesh, border, faces and points from a TIN surface. You can exchange these entities in other applications, to further analyze the surfaces. The solids or meshes can be created in different ways: as a vertical offset of the TIN Surface, as a model between two TIN Surfaces or as a model between a fixed elevation plane and the TIN Surface.

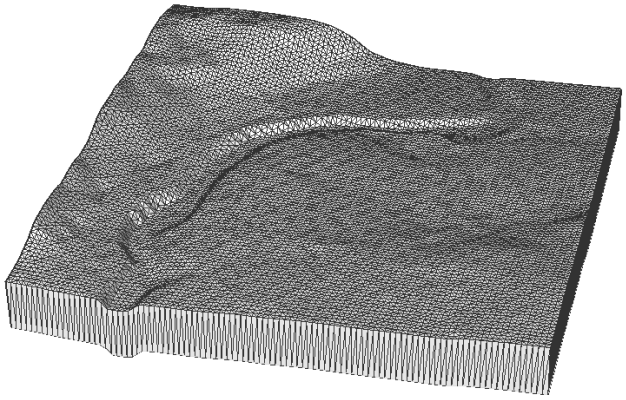
### 9.9.4 Procedure: Creating a solid from a TIN surface using option Elevation

1 Select a TIN Surface and type TINEXTRACT in the Command line.



2 Select the **Solid** option and press Enter.

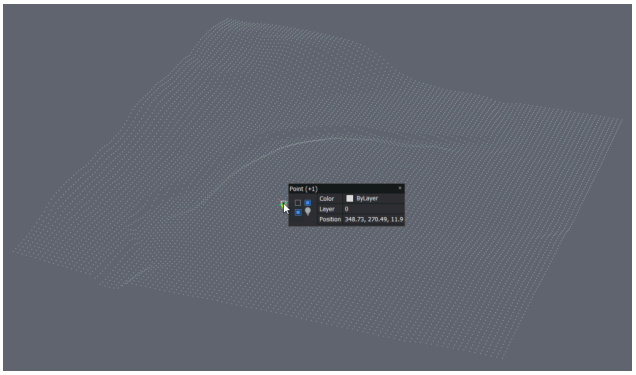
3 Choose the elevation height and press Enter.



### 9.9.5 Procedure: Creating Points from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

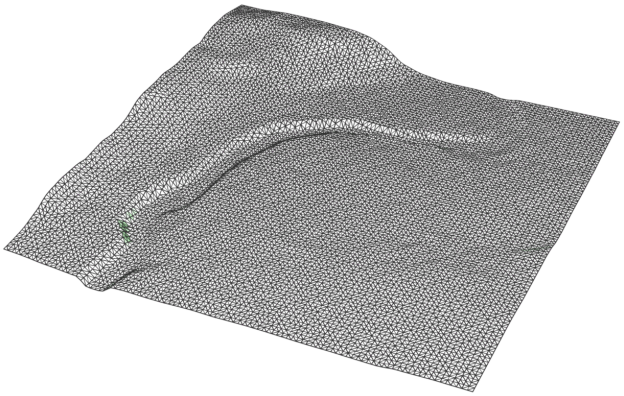
2 Select the **Points** option in the Command line.



### 9.9.6 Procedure: Creating Faces from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 Select the **Faces** option.

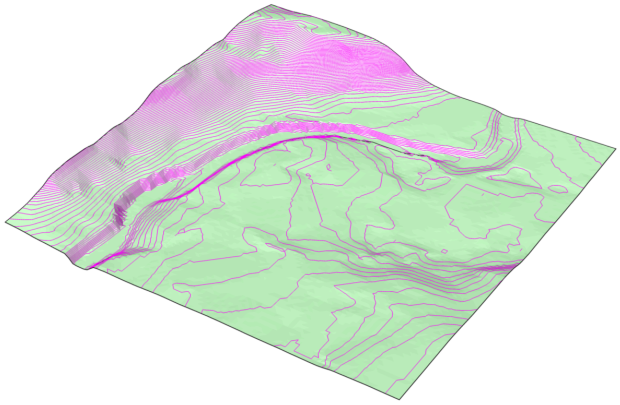


### 9.9.7 Procedure: Creating a Contours from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 Select the **Contours** option in the Command line.

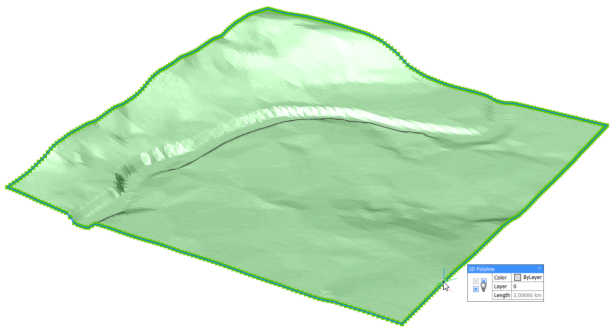
3 Type in **A** for All and press Enter.



### 9.9.8 Procedure: Creating a Border from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 Select the **Border** option in the Command line.

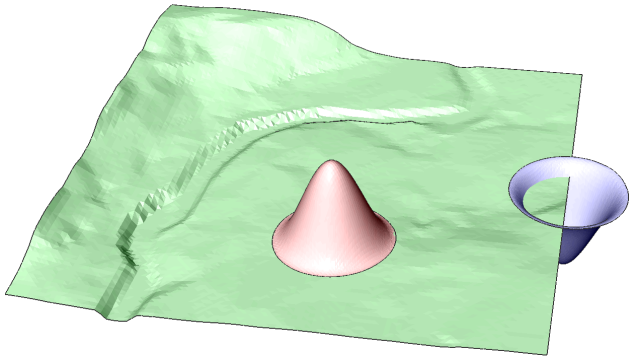


### 9.9.9 About TIN Merge

Different TIN Surfaces can be merged into one by using the TINMERGE command. A merge is executed between a base surface and one or more surfaces to merge with.

### 9.9.10 Merging TIN Surfaces

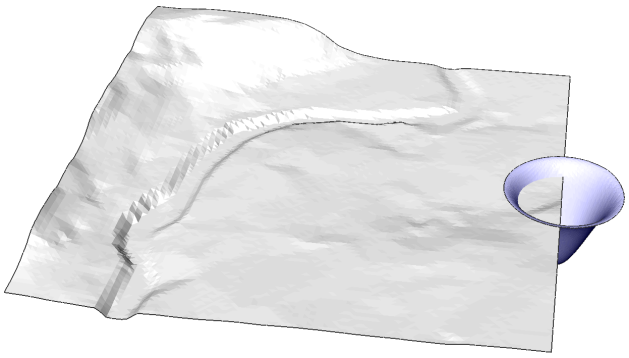
1 Type TINMERGE in the Command line.



2 Choose a base surface. In this example, we choose the green surface as the base surface.

3 Choose to keep or delete the original surfaces.

***Прим.:*** Selecting a different baseline, results in a different merge. It is important to remember that the surface to merge replaces the part of the base surface that it covers.



The first example shows the merge of the green surface as the base surface and the blue and red surface.

The second surface shows the merge of the blue surface as the base surface and the green surface.

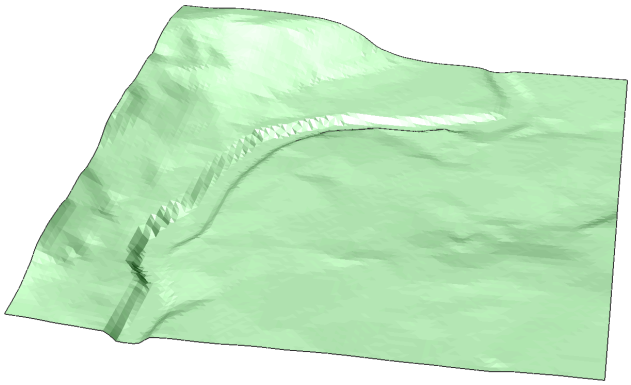
The third surface shows the merge of the red surface as the base surface and the green surface.

### 9.9.11 About TIN Modify

A TIN Surface can be modified by deforming or smoothening the surface. By deforming the surface, the TIN Surface is adjusted following the contours of an area defined by the user. By smoothening the surface, a selected area is leveled which is done by interpolation of this predefined area.

### 9.9.12 Procedure: Adding deformations on a TIN Surface using Radius

1 Select a TIN Surface and type TINMODIFY in the Command line.



2 Select the **Deform** option.

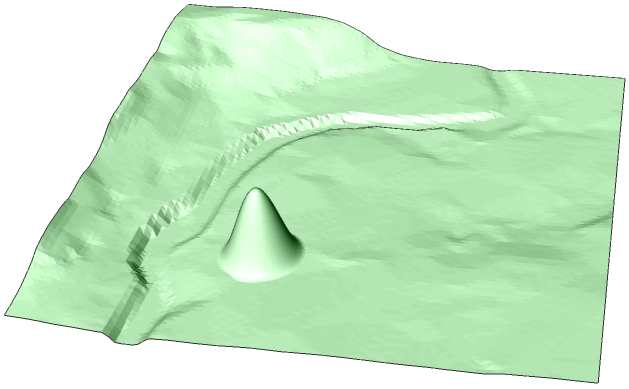
3 Choose a deformation radius.

4 Click on the area to apply the deformation.

5 Choose the elevation height.

6 Choose whether you want to add the adjustment as a new element or as a modification of the existing

TIN Surface.



***Прим.:*** By adding the modification as a new element, the structure of the original TIN Surface isn't modified but added as a new TIN surface.

7 Click on other areas to add new deformation or press ENTER to end the command.

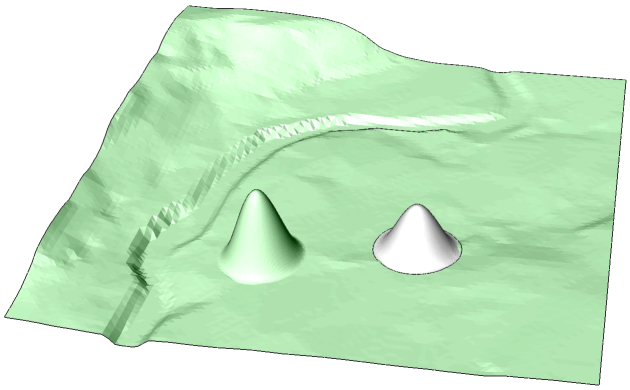
### 9.9.13 Procedure: Adding deformations on a TIN Surface using specify Contour

1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

3 Select the **specify Contour** option.

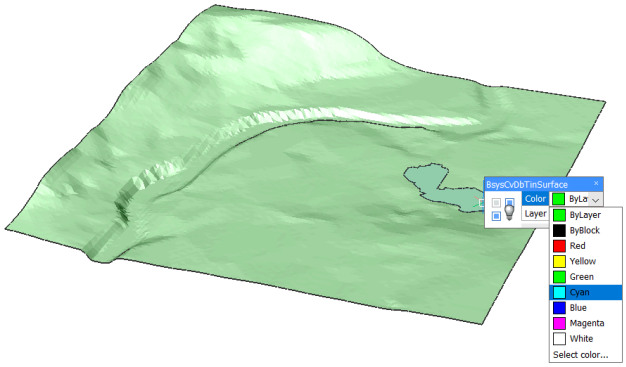
4 Pick a contour on the TIN Surface.



5 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a new element for the adjustment.

6 Click on other contours to add new deformation or press ENTER to end the command.

7 You have now created a new element on your TIN Surface. You can change its properties as you like. For example: change the color of the element.




### 9.9.14 Procedure: Adding deformations on a TIN Surface using select Entity

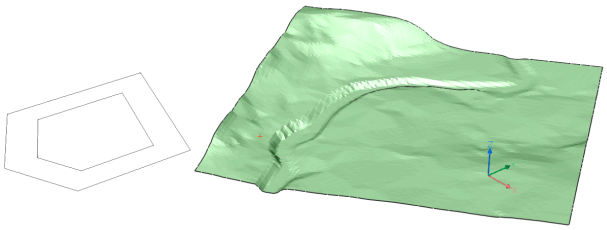
1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

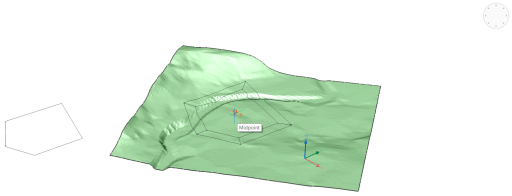
3 Type E and hit ENTER to choose **Select Entity**.

4 Select a closed entity (e.g. Polyline).

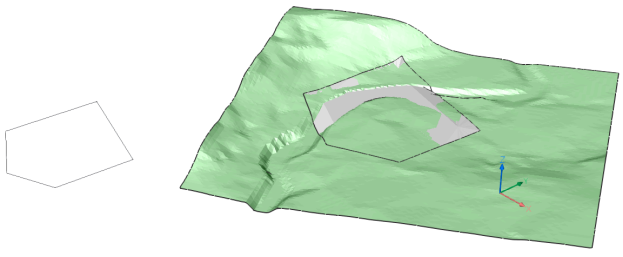
5 Choose an offset distance.



6 Click on the area to apply the deformation.



7 Choose the elevation height.



8 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a New element for your adjustment.

9 Click on other areas to add new deformation or press ENTER to end the command. You have now added a deformation using **select Entity**.

### 9.9.15 Procedure: Adding deformations on a TIN Surface using draw Polygon

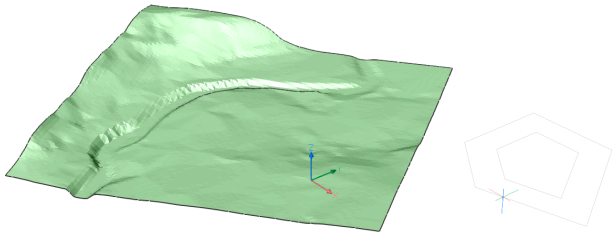
1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

3 Select **draw Polygon**.

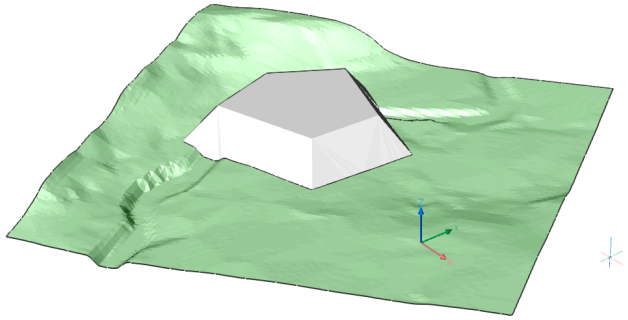
4 Draw a polygon, hit ENTER to close the polygon.

5 Choose an offset distance.



6 Select the area where to apply the deformation.

7 Choose the elevation height.

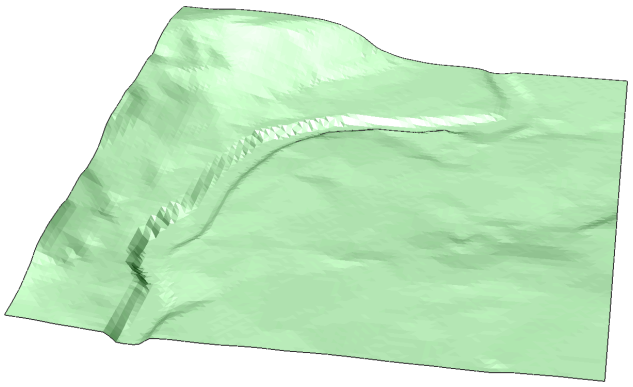


8 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a new element for your adjustment.

9 Click on other areas to add new deformation or press Enter to end the command. You have now added a deformation using draw polygon.

### 9.9.16 Procedure: Smoothening a TIN Surface

1 Select a TIN Surface and type TINMODIFY in the Command line.

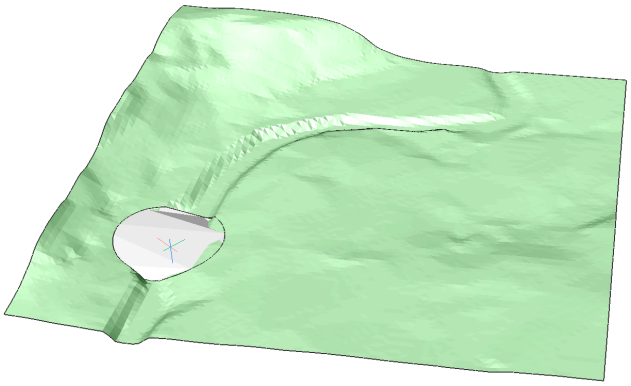


2 Select the **Smoothen** option.

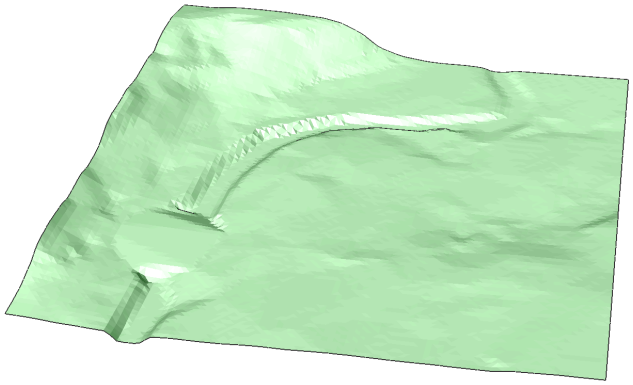
3 Choose a radius.

***Прим.:*** The procedure for **select Entity** or **draw Polygon** is analogue to **Adding deformations using select Entity** and **Adding deformations using draw Polygon** starting from step 3.

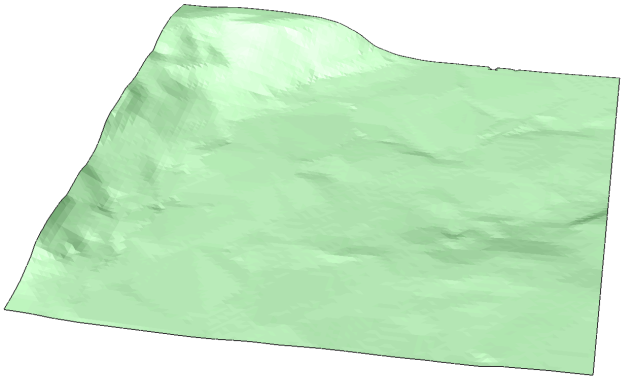
4 A preview of the intervention is shown.



5 Click to execute the command.



6 Click on other areas to smoothen or press ENTER to end the command.



## 9.10 Analyzing TIN Surfaces

### 9.10.1 Commands

TINVOLUME, TINWATERDROP

### 9.10.2 Overview

In BricsCAD, topographical surfaces created as a TIN Surface, can be analyzed using different tools.

### 9.10.3 About TIN Volume

A TIN volume surface provides an exact difference between the base and comparison surfaces. Therefore, the Z-value of any point in the volume surface is precisely the difference between the Z of the comparison surface at that point and the base surface at that point. A TIN Volume Surface is drawn as a custom entity with additional fill/cut volume properties.

You can use TIN Volume to calculate or analyze earthworks:

- Calculate fill/cut volumes inside bounding area.
- Draw zero contours which divide fill and cut areas.
- Visualize the mass distribution.

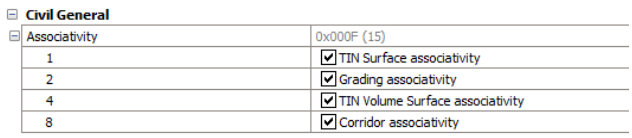
### 9.10.4 Procedure: Creating a TIN Volume

In the following procedures, a TIN Volume is created. Choose whether the volume is associative or not. If associativity is turned on: TIN Volume Surface is automatically updated if base or comparison surfaces are changed. To disable or enable the associativity, follow these steps:

1 Type **Settings** in the Command line and press Enter.

2 Go to **Civil** > **Civil General** > **Associativity**.

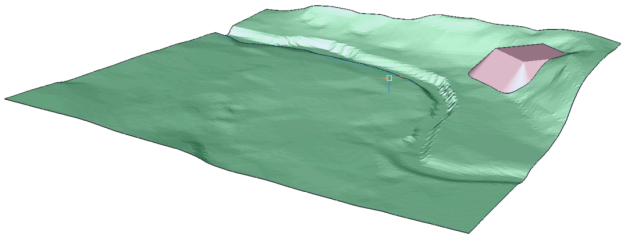
3 Check or uncheck **TIN Volume Surface Associativity**.



### 9.10.5 Creating a TIN Volume using a comparison TIN Surface

1 Launch the TINVOLUME command.

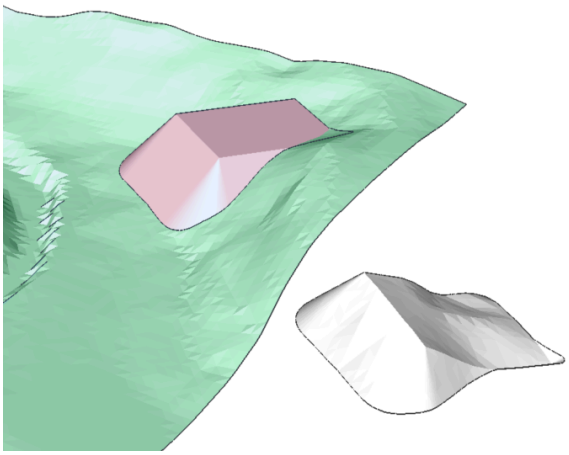
2 Choose a base surface. In this example, we choose the green TIN surface as the base surface.



3 Choose a comparison surface. In this example, we choose the red grading as the comparison surface.

4 Select a bounding area as the area to apply the operation orpress ENTER. The outer boundary of the surface will be used as a boundary.

5 The TIN Volume appears under the TIN Surface.



### 9.10.6 Creating a TIN Volume using Elevation

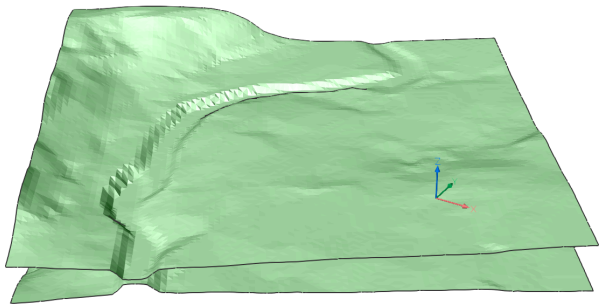
1 Launch the TINVOLUME command.

2 Choose a base surface.

3 Select **Elevation** in the Command line.

4 Choose the elevation height.

5 Select a bounding area as the area to apply the operation or press ENTER. The outer boundary of the surface will be used as a boundary.



### 9.10.7 About TIN Water Drop

The TINWATERDROP command creates water drop paths in real-time by moving the cursor over a TIN Surface. Water path is created as a 3D polyline in the current position if you click the mouse button.

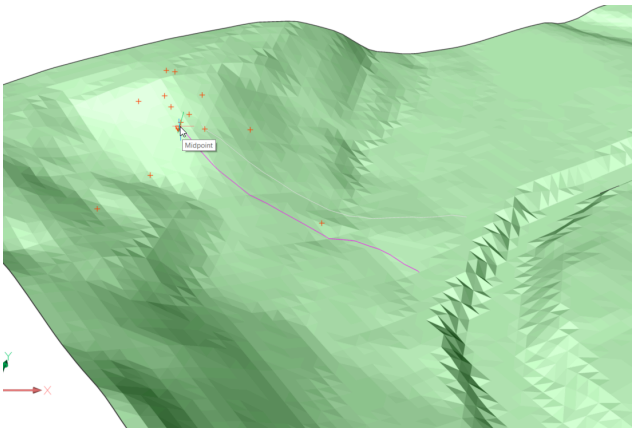
### 9.10.8 Procedure: Creating a water drop path

1 Launch the TINWATERDROP command.

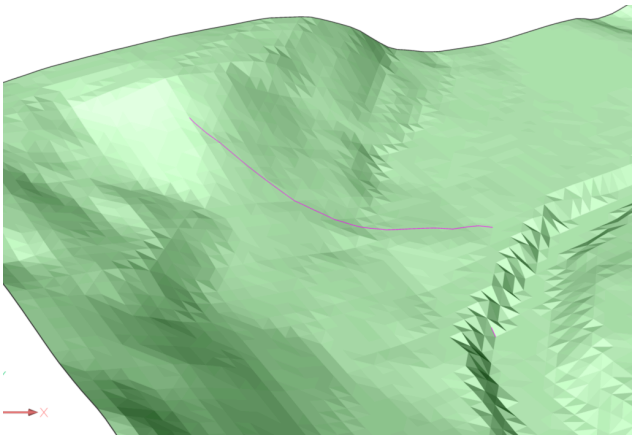
2 Select the TIN surface.

3 Move the cursor over the surface, a line representing the drop path is shown in real-time.

***Прим.:*** It would be better if the water drop path is colored so it could be seen clearer. To do so, you should change the current layer to another color, e.g. red.




4 Left click with the mouse to create a 3D polyline of the water drop path.



## 9.11 Working with Gradings

### 9.11.1 Commands

GRADING, GRADINGBALANCE

### 9.11.2 Overview

In BricsCAD, graded surfaces are created using the Create Grading tool.

Grading is a set of processes to maintain a base level for construction works or TIN surface design by defining a slope or adjusting the offset of a grading. For instance, creating parking areas, recreation reserves or road and railway designs.

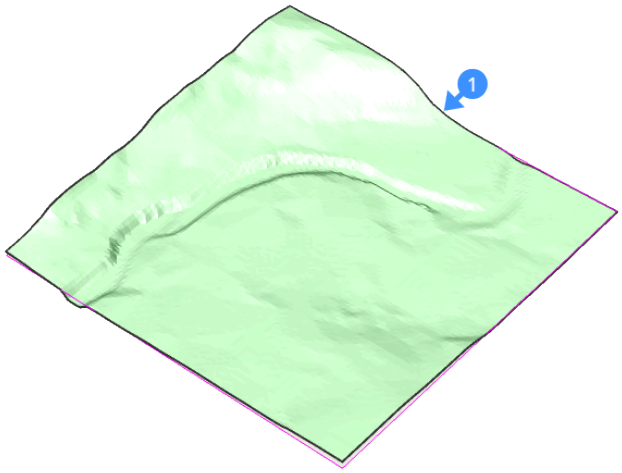
Two projection methods **Slope to surface** and **Offset to slope** are supported:

- **Slope to surface** needs an input entity (e.g., polyline, line, arc, circle), a surface and slope to grade the surface.
- **Offset to slope** needs an input entity, offset and slope.

Graded surfaces are not editable with the TINEDIT command.

### 9.11.3 Procedure: Grading a surface to create a pond

1 Open a drawing that has a topographical surface (1).

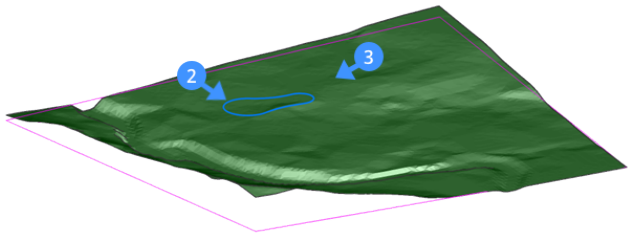



2 Launch the GRADING command.

3 Select the entity that defines the footprint area of your pond.

***Прим.:*** Before selecting the target surface you can change to **slOpe/offset** projection method which requires an input entity, offset and slope.

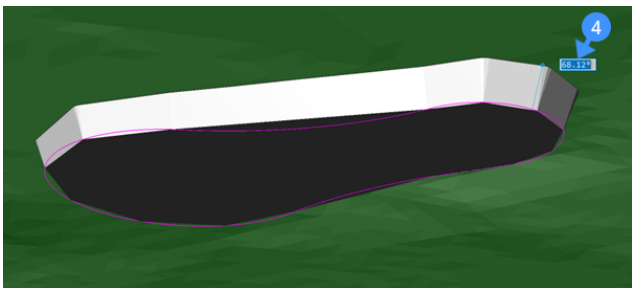
4 Select the target surface.




5 Define the slope of the grading.

***Прим.:*** Grading is attached to the input entity, and it is connected with the target surface. Moving the mouse in/out dynamically increases/decreases the slope of the grading.

***Прим.:*** If DYNMODE variable is set to 3, you can also input the slope angle directly from keyboard (4).




6 Press ENTER to accept the slope of the grading.

### 9.11.4 Displaying the properties of a graded surface

The properties are displayed in the Properties Panel after selecting the grading.

| Data | |
|---|---|
| Grading type | Slope to surface |
| Target surface | BB |
| Name | Grading |
| Description | |
| **Grading settings** | |
| Cut slope | 45 |
| Fill slope | 45 |
| Side | Right |
| Infill | On |
| Boundary | Off |
| **Grading statistics** | |
| Fill volume | 99.46 |
| Cut volume | 0.00 |
| Net volume | 99.46 |
| **TIN Visibility** | |
| Boundary line | Off |
| Points | Off |
| Triangles | On |
| Contours | Off |
| **TIN Contours** | |
| Major contours interval | 10 |
| Minor contours interval | 1 |
| Major contours color | Red |
| Minor contours color | Green |
| **TIN Statistics** | |
| Number of points | 389 |
| Number of triangles | 391 |
| Minimum elevation | 348.61 |
| Maximum elevation | 350 |
| 2D area | 182.79 |
| 3D area | 195.51 |

**Grading type**
Displays which projection method is used to grade the surface.

**Target surface**
Displays the handle number of the target surface

**Cut slope**
Displays the cut slope.

**Fill Slope**
Displays the cut slope.

**Side**

Controls on which side of the input entity the grading is drawn. Left or Right.

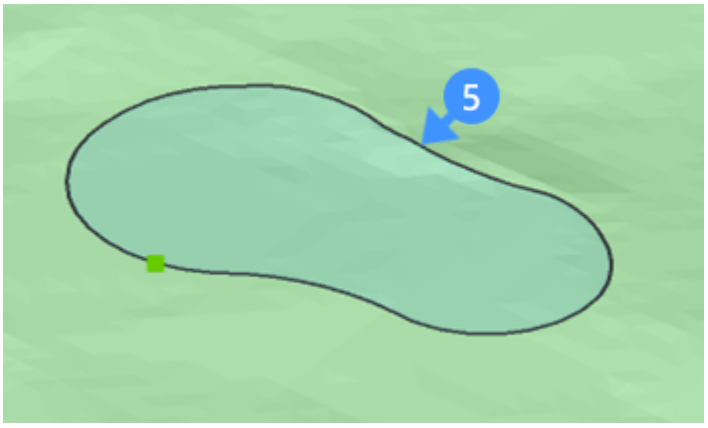
**Infill**

The internal are of entities is filled if this option is turned on. Valid for closed entities only (circle, closed polyline).

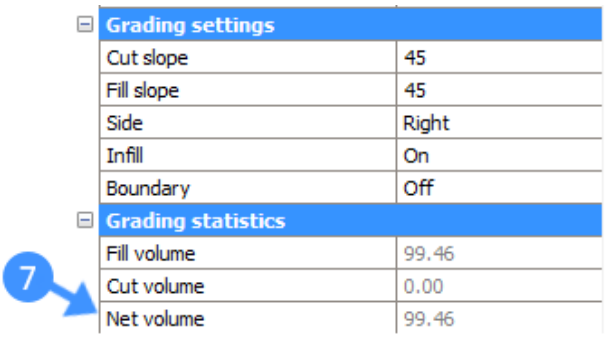
**Boundary**

Controls the boundary of the grading. When the grading boundary is on, that boundary is added as a **hide** boundary to the target surface.

The following images show when the grading boundary of the pond is off (5) and on (6). Based on the grading boundary, the net volume for the body of the pond is calculated and can be found in the properties panel (7).






| Grading settings | |
|---|---|
| Cut slope | 45 |
| Fill slope | 45 |
| Side | Right |
| Infill | On |
| Boundary | Off |
| **Grading statistics** | |
| Fill volume | 99.46 |
| Cut volume | 0.00 |
| Net volume | 99.46 |

**Fill volume**
Displays the fill volume for a body/shape limited with the target surface and graded surface.

**Cut volume**
Displays the cut volume for a body/shape limited with the target surface and graded surface.

**Net volume**
Displays the net volume for a body/shape

### 9.11.5 Procedure: Grading a surface to create a platform

1 Open a drawing that has a topographical surface (8).




2 Launch the GRADING command.

3 Select the entity that defines the footprint area of your platform (9).

***Прим.:*** Before selecting the target surface you can change to **slOpe/offset** projection method which requires an input entity, offset and slope.

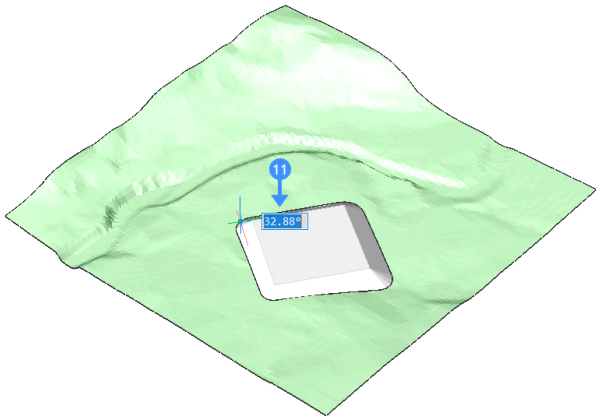
4 Select the target surface (10) to create a graded area.




5 Define the slope of the grading.

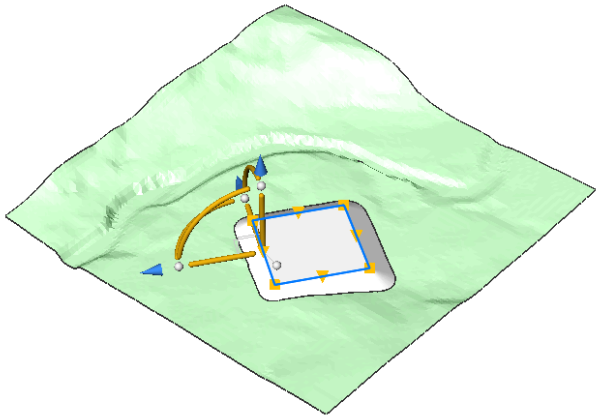
***Прим.:*** Grading is attached to the input entity, and it is connected with the target surface. Moving the mouse in/out dynamically increases/decreases the slope of the grading.

***Прим.:*** If DYNMODE variable is set to 3, you can also input the slope angle directly from keyboard (11).

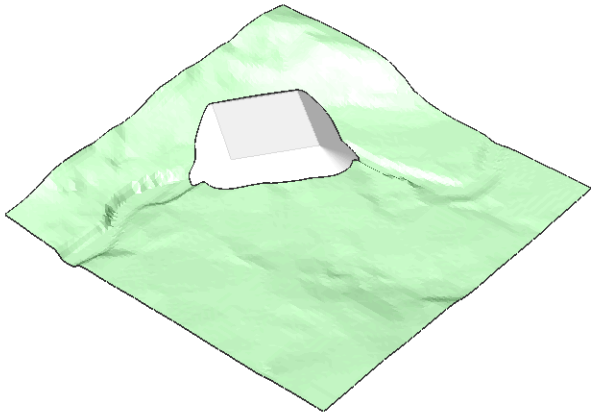



6 Press ENTER to accept the slope of the grading.

***Прим.:*** You can easily move the platform that you created. Do this by using the manipulator on the entity from which you created the platform. You can move both horizontally and vertically.



The grading will automatically be adapted to its surroundings.

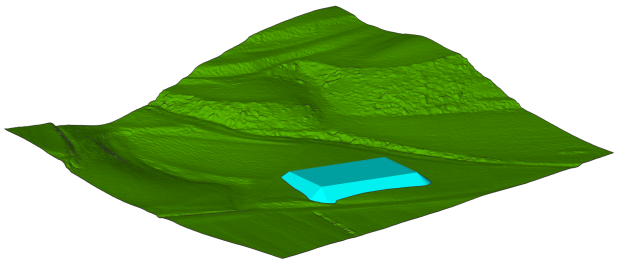


### 9.11.6 About Balance Grading Volumes

The GRADINGBALANCE command allows you to balance cut and fill volumes within the specified tolerance. This is achieved by raising/lowering the elevation of the grading input entity and the result is grading that has a net volume around zero, depending on the set tolerance.

### 9.11.7 Procedure: Adjusting the elevation of a grading to balance cut and fill volumes

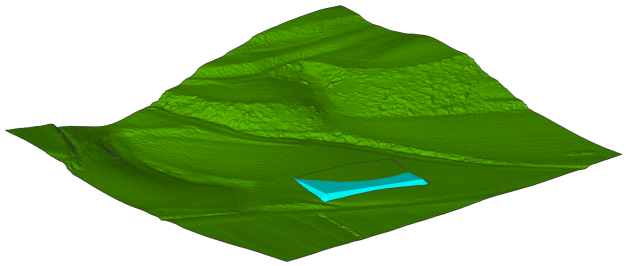
1 Open a drawing that contains a TIN surface and a grading representing a building pad.



2 Launch the GRADINGBALANCE command.

3 Select the grading.

4 Enter the tolerance in percent (1-100). 1 means that the difference between the cut and fill volume will be less than one percent and 100 means that the difference between the cut and fill volume will be less than 100 percent.



In the Properties panel you see that the cut and fill volumes are now balanced.

| Grading statistics | |
|---|---|
| Fill volume | 670.77 |
| Cut volume | 673.19 |
| Net volume | -2.42 |

***Прим.:*** Balance Grading Volumes command works only on gradings that has Boundary set to **OFF**.

| Grading settings | |
|---|---|
| Cut slope | 50 |
| Fill slope | 35 |
| Side | Left |
| Infill | On |
| Boundary | Off |

## 9.12 Create and edit alignments

### 9.12.1 Commands

ALIGNMENT, ALIGNMENTEDIT, ALIGNMENTVIEW

### 9.12.2 Overview

In BricsCAD, alignments are created using the Alignment tool. Alignment is the route of the linear objects, defined as a combination of lines and curves that are viewed as one object.

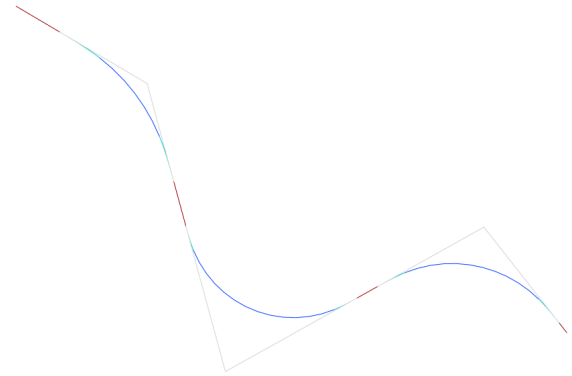
### 9.12.3 About ALIGNMENT

Alignment is a reference system used to position linear objects, like roads, railways, pipe networks, streets, retaining walls and bridges, in space. An alignment is represented with horizontal, vertical 2D curves, resulting in 3D alignment.

**Horizontal alignment**

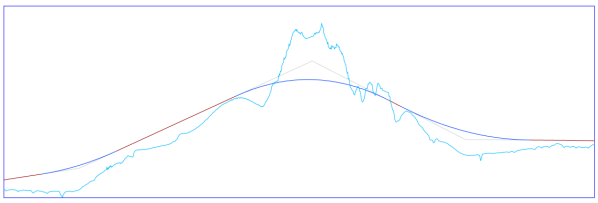
Horizontal alignment is a 2D curve that is needed to precisely position linear objects in a horizontal direction. It becomes defined by the Points of Intersection (PI). Horizontal alignment is the horizontal trace

on your TIN Surface which is used to generate the height profile and vertical alignment. A single horizontal alignment can have multiple vertical alignments. The horizontal alignment will appear under the surface.



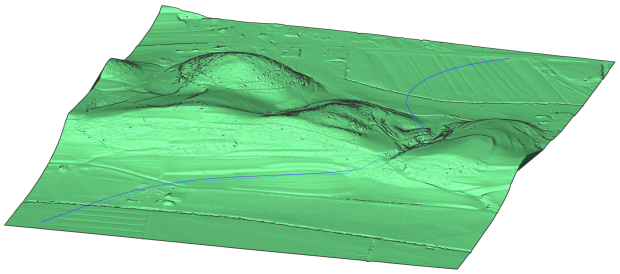
**Vertical alignment**

A vertical alignment is needed to precisely define a linear object in the vertical sense. The horizontal path defined by the horizontal alignment corresponds with a vertical profile of the TIN Surface. By defining Points of Vertical Intersection (PVI) on these vertical profiles of the target surfaces, the new 2D curve defines the vertical alignment.



**3D alignment**

A 3D Alignment is a 3D polygonal curve defined by the horizontal and vertical alignment.



### 9.12.4 Procedure: Creating an Alignment

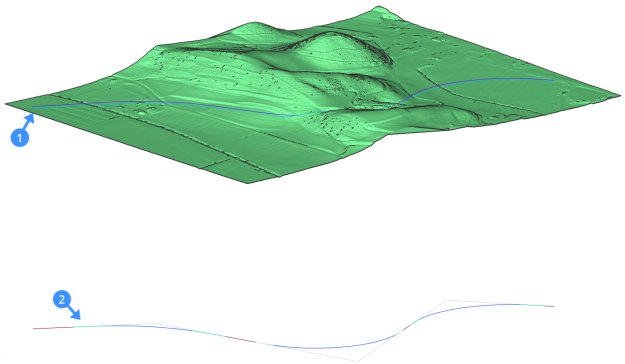
1 Open the drawing file that contains a TIN Surface.

2 Launch the ALIGNMENT command in the Command line.

3 Select **Horizontal** alignment in the Command line.

4 Select the TIN surface.

5 Pick the horizontal alignment PI points on your surface. Press ENTER when done. Your 3D alignments (1) and horizontal alignment (2) are now generated.



***Прим.:*** Three alignments are shown. The horizontal alignment on plane 0,0, light blue 3D alignment shows the projection of horizontal alignment on TIN Surface and the purple 3D alignment shows the design alignment that is automatically generated.

### 9.12.5 Procedure: Creating a Horizontal Alignment without 3D alignments

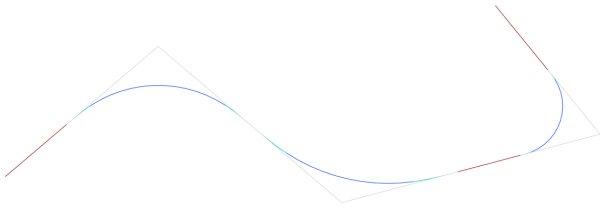
1 Open an empty drawing and launch the ALIGNMENT command in the Command line.

2 Select **Horizontal** alignment in the Command line.

3 Select the **Pick Pi point** option in the Command line.

4 Pick the horizontal alignment PI points. Press ENTER when done.

Your horizontal alignment is now generated.

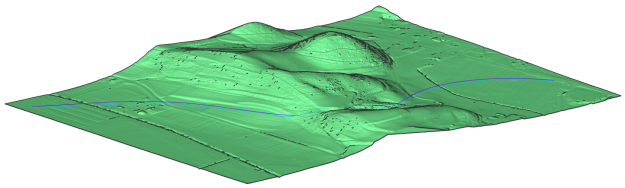


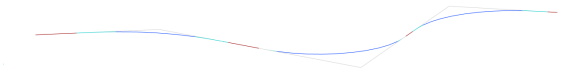
### 9.12.6 About Vertical Alignment View

The horizontal alignment creates a vertical section of the target surface. This 2D curve can be displayed in a separate viewport inside your model space using the command ALIGNMENTVIEW.

### 9.12.7 Procedure: Creating a Vertical Alignment View

1 Open the drawing file that contains a TIN Surface, Horizontal Alignment and 3D Alignments.



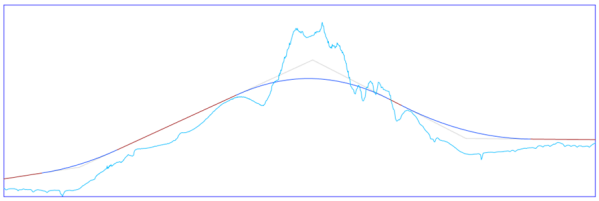


2 Launch the ALIGNMENTVIEW command in the Command line.

3 Select a horizontal or 3d alignment in the drawing.

4 Pick a point as origin for the vertical alignment view.

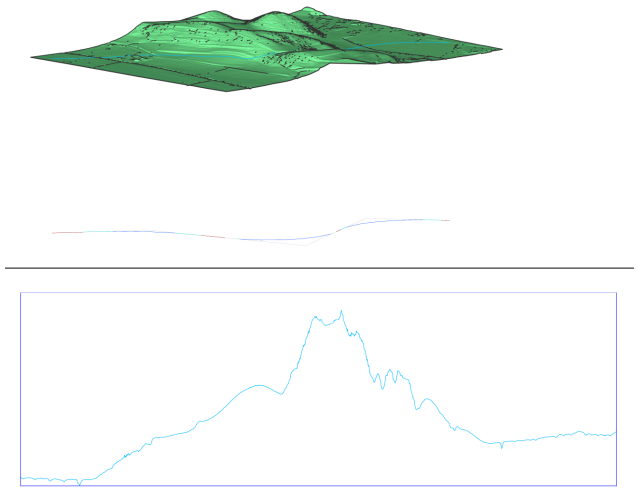
Your vertical alignment view is now generated.



***Прим.:*** Two vertical alignments are shown. Light blue represents terrain line from the TIN Surface, and another is design vertical alignment. The design vertical alignment has straight gradients and vertical curves and is created automatically based on the accuracy factor. You can change the accuracy factor in the Properties Panel from 1 to 100. When you choose 1 the accuracy will be the lowest.

### 9.12.8 Procedure: Creating a Vertical Alignment

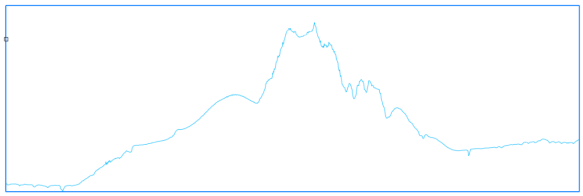
1 Open the drawing file that contains a TIN Surface, Horizontal Alignment and Vertical Alignment View.



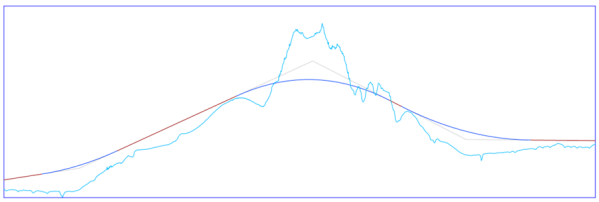
2 Launch the ALIGNMENT command in the Command line.

3 Select the **Vertical** option in the Command line.

4 Select vertical alignment view in the drawing.

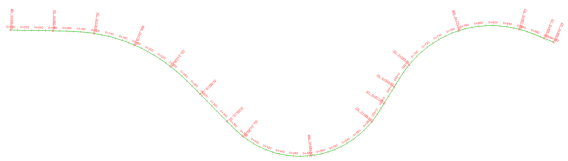


5 Pick the vertical alignment PVI points. Press ENTER when done. If you want to go one step back, use Undo. Automatic 3d alignment option generates design vertical alignment automatically. Your design vertical alignment is now generated.



### 9.12.9 Procedure: Creating an Alignment from Civil 3D drawing

1 Open the drawing file from Civil 3D that contains a Civil 3D Alignment.

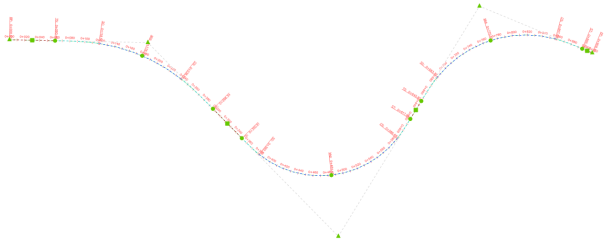


2 Launch the ALIGNMENT command in the Command line.

3 Select the **create from Civil 3d alignment**.

4 Select the civil alignment in the drawing and press ENTER.

Civil 3D alignment is converted to BricsCAD Alignment.



### 9.12.10 About ALIGNMENTEDIT

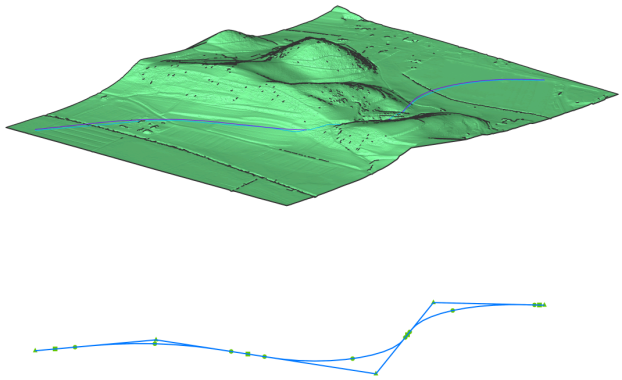
After creating an alignment, you can continue a horizontal or vertical alignment, add or delete PV/PVI points or change the TIN Surface using the command ALIGNMENTEDIT. The 3D alignment will be updated automatically.

### 9.12.11 Procedure: Continuing a horizontal alignment with ALIGNMENTEDIT

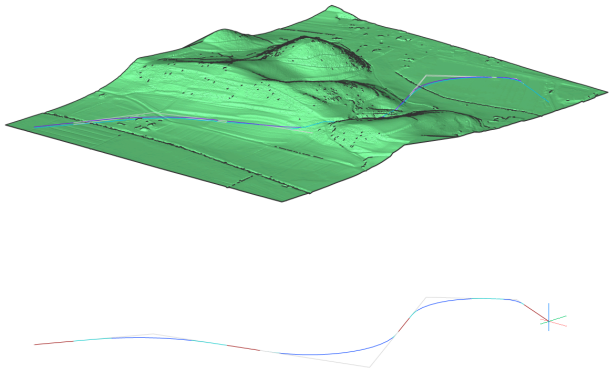
1 Draw a horizontal alignment following the steps from the Procedure: Creating an alignment.

2 Type ALIGNMENTEDIT in the Command line.

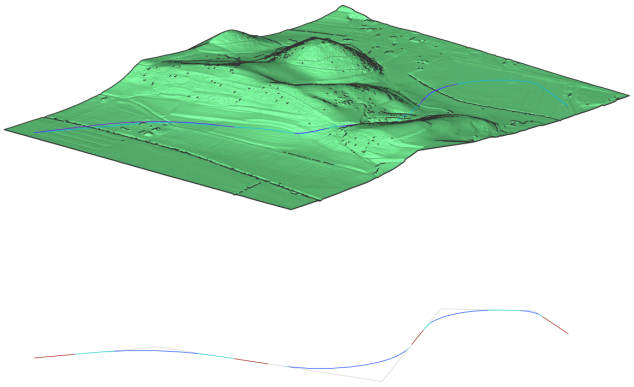
3 Select the horizontal alignment you want to edit.



4 Select points on your surface to continue your existing alignment.

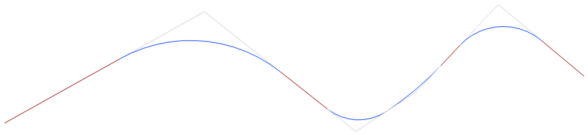


5 Press enter when finished. You have now continued your horizontal alignment. The 3D alignment will be updated automatically.



### 9.12.12 Procedure: Adding PI to an existing horizontal alignment

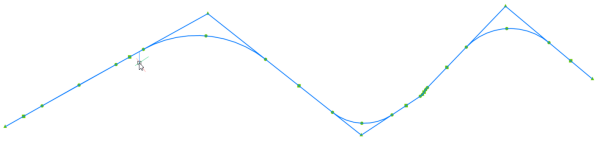
1 Type ALIGNMENTEDIT in the Command line.



2 Select the horizontal alignment you want to edit. You are prompted: Pick a point to continue alignment or [Add Pi/Remove Pi/change Tin surface].

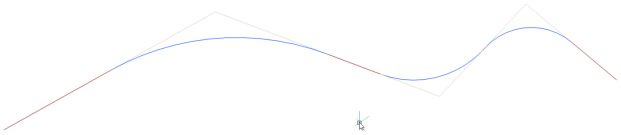
3 Select **Add Pi** in the Command line.

4 Pick point to add new PI.



### 9.12.13 Procedure: Removing PI to an existing horizontal alignment

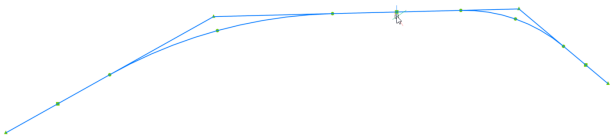
1 Type ALIGNMENTEDIT in the Command line.



2 Select the horizontal alignment you want to edit.

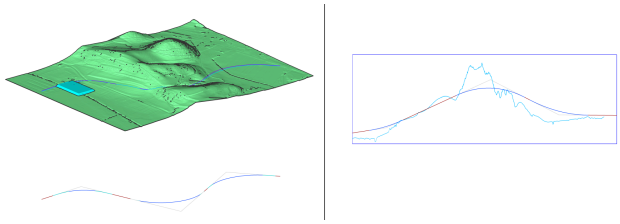
3 Select the **Remove Pi** option in the Command line.

4 Pick a PI point to remove.



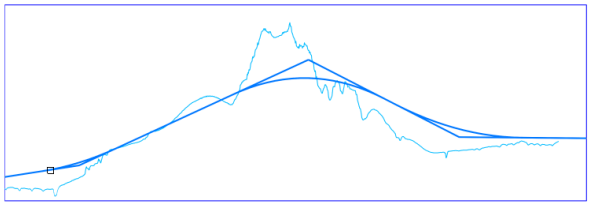
### 9.12.14 Procedure: Adding additional TIN Surface to Vertical alignment view

1 Open the drawing file that contains a TIN Surface, Alignments, Vertical Alignment View and grading.

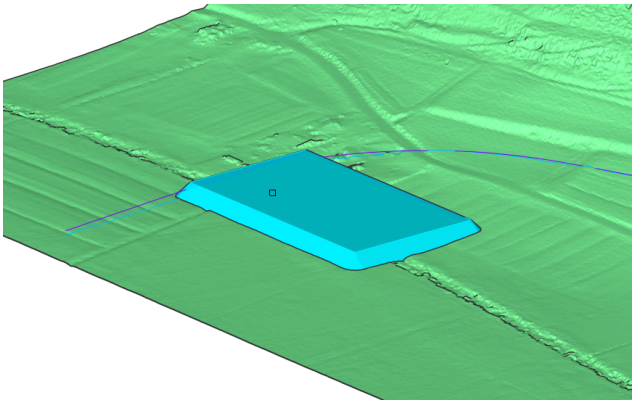


2 Launch the ALIGNMENTEDIT command in the Command line.

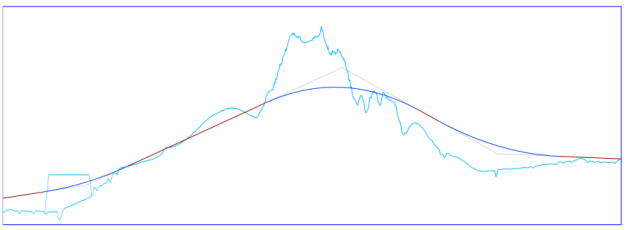
3 Select the **change Tin surface** option in the Command line.



4 Select the TIN surface to add.



In the vertical alignment view, you can see an additional light blue line representing the grading object.



## 9.13 Modeling a Corridor

### 9.13.1 Commands

CORRIDOR, CORRIDORTEMPLATE, CORRIDORTEMPLATEELEMENT, CORRIDOREDIT, CORRIDOREXTRACT.

### 9.13.2 Overview

In BricsCAD, Corridors that follow a certain 3D alignment are created using the Corridor tool.

### 9.13.3 About Corridor

A Corridor is a BricsCAD Civil drawing object used for modeling three-dimensional linear objects such as roads, railway, retaining walls, and bridges. It principally consists of a 3D alignment of the linear object and its typical cross-section – which is called Corridor Template in BricsCAD.

The CORRIDOR command enables you to create a Corridor, based on a Corridor Template, that follows a 3D alignment on a certain topographical surface.

### 9.13.4 About Corridor Template

In BricsCAD, a Corridor Template entity represents the framework to append a collection of template element objects. The command CORRIDORTEMPLATE enables you to create a corridor template entity and append one or more template elements to it.

### 9.13.5 About Corridor Template Element

In BricsCAD, a Corridor Template Element entity defines the geometry of a component used in a corridor section. The CORRIDORTEMPLATEELEMENT command enables you to create a corridor template element from closed polylines.

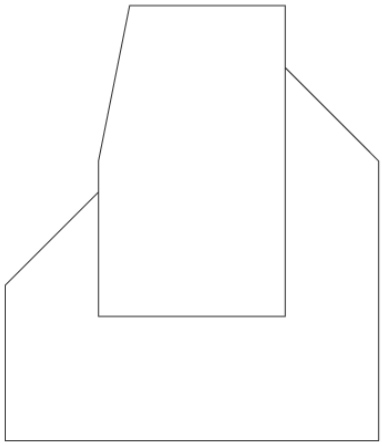
### 9.13.6 About Corridor Extract

The CORRIDOREXTRACT command is used to generate meshes, solids, polylines or outer boundaries from an existing Corridor.

### 9.13.7 Procedure: Creating a template element of kerb

1 Open an empty drawing.

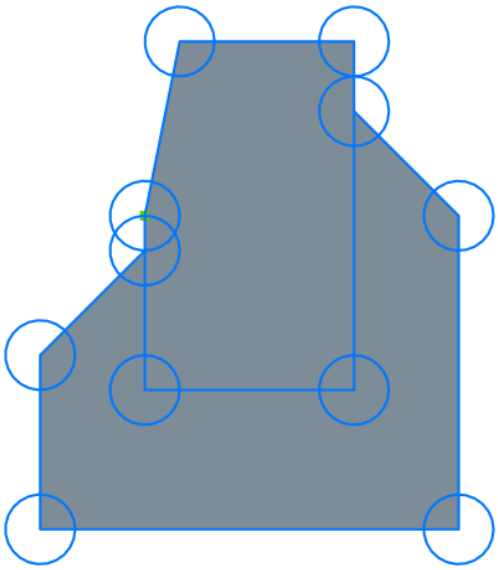
2 Draw closed polylines representing road kerb.



3 Launch the CORRIDORTEMPLATEELEMENT command in the Command line.

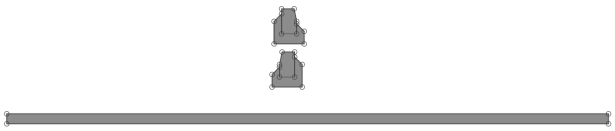
4 Select polylines representing kerb.

5 Pick one of the polyline vertices, the base point position can also be changed afterward.



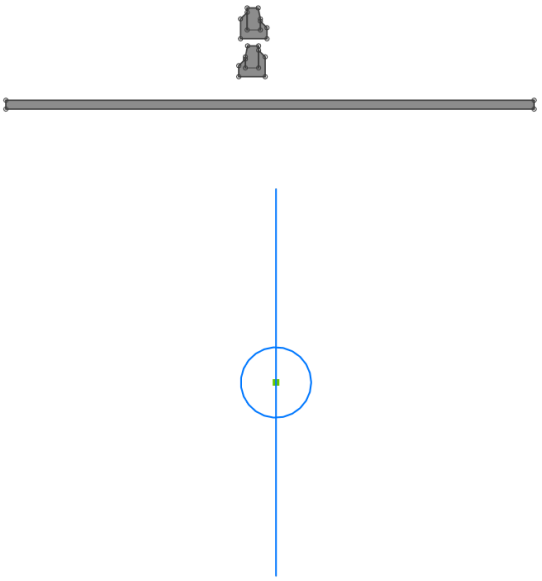
### 9.13.8 Procedure: Creating a template of road cross-section with asphalt layer and left and right kerb

1 Open the drawing file that contains a collection of template elements.



2 Launch the CORRIDORTEMPLATE command in the Command line.

3 Pick a point in the drawing where you want to create a corridor template.

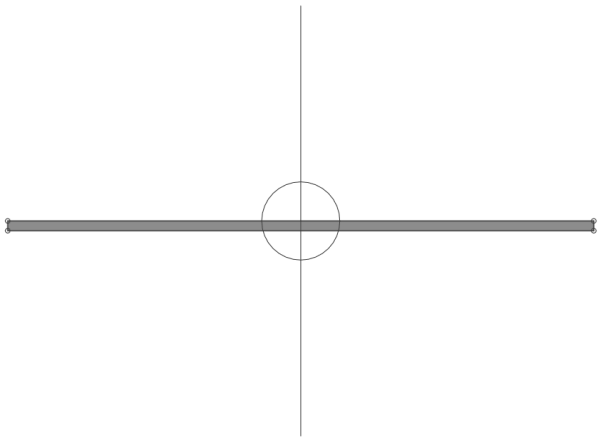


4 Launch the CORRIDORTEMPLATE command in the Command line.

5 Select the **Add template element** option in the Command line.

6 Select the template element representing the asphalt layer.

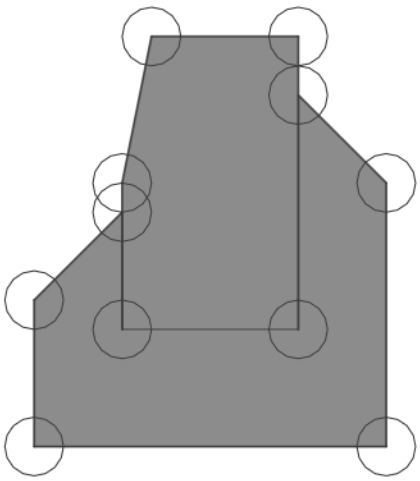
7 Select corridor template or template element point: (select the template)



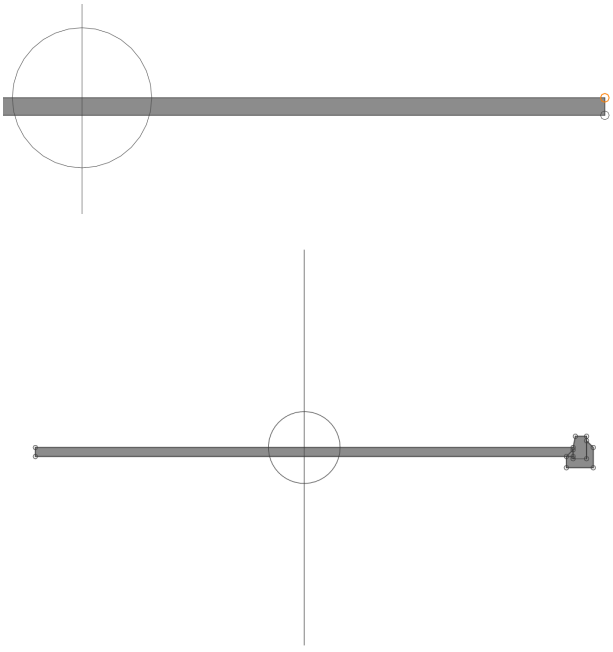
8 Launch the CORRIDORTEMPLATE command in the Command line.

9 Select the **Add template element** option in the Command line.

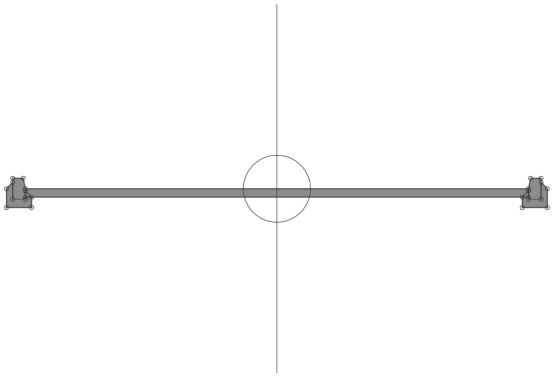
10 Select right top template element point of asphalt layer.



11 Select template element representing right kerb.

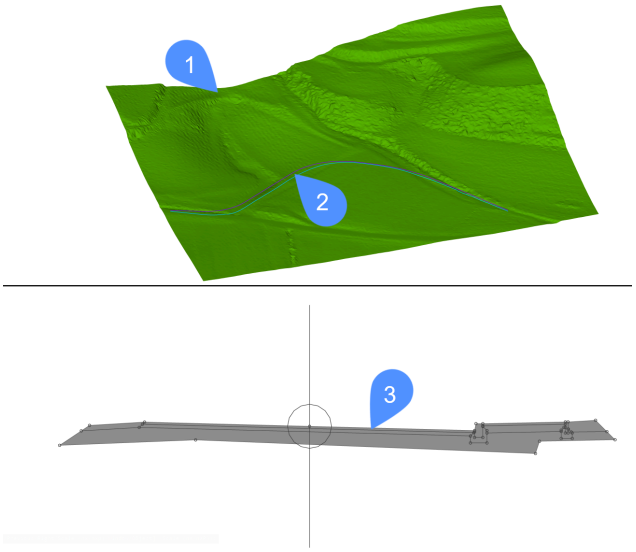


12 To add a kerb to the left side, repeat the steps described in points 9, 10, 11. By selecting the left kerb and left top point of asphalt layer this time.



### 9.13.9 Procedure: Creating a road corridor

1 Open the drawing file that contains the TIN Surface (1), 3D Alignment (2) and Corridor Template (3).



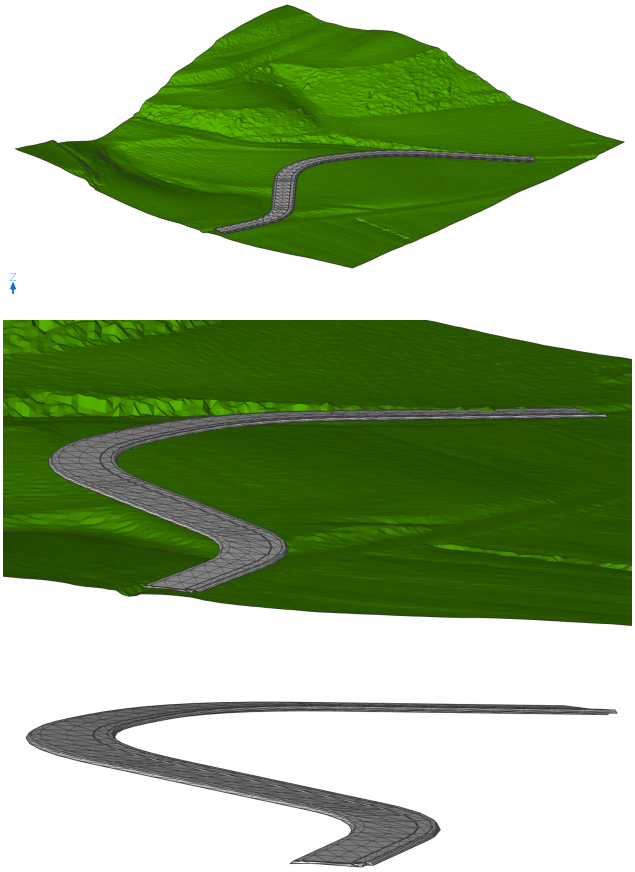

2 Launch the CORRIDOR command in the Command line.

3 Select 3D alignment (2) used as a corridor baseline.

4 Select corridor template (3).

5 Enter region start station. Press ENTER to start from beginning or type any station where you want your corridor to start.

6 Enter region end station. Press ENTER to create corridor to the end of selected alignment or type any station where you want your corridor to end.



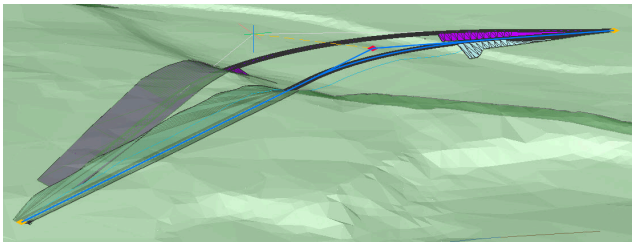
### 9.13.10 About Corridor Edit

The CORRIDOREDIT command adds or removes regions from an existing Corridor.

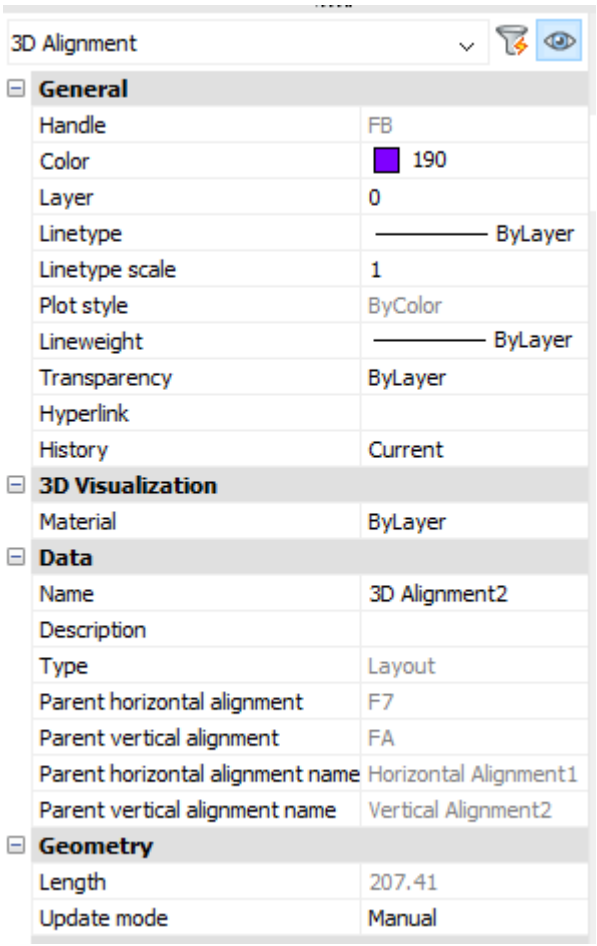
### 9.13.11 Procedure: Editing a corridor

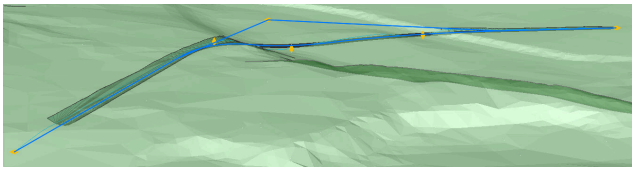
1 Open the drawing file that contains the corridor you would like to edit.

2 Select the 3D alignment that the corridor has as a reference and move the grip points to edit the corridor horizontally.

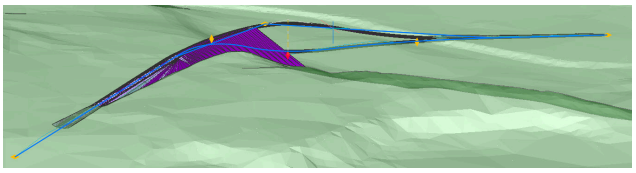


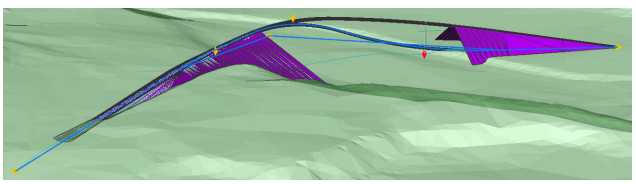
3 To edit the corridor vertically, select the 3D alignment that the corridor has as a reference and open the Properties Panel. Set the **Update mode** to Manual. Extra grip points will appear.



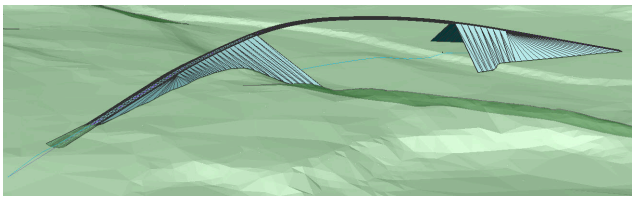



4 Move these grip points to edit the corridor vertically.





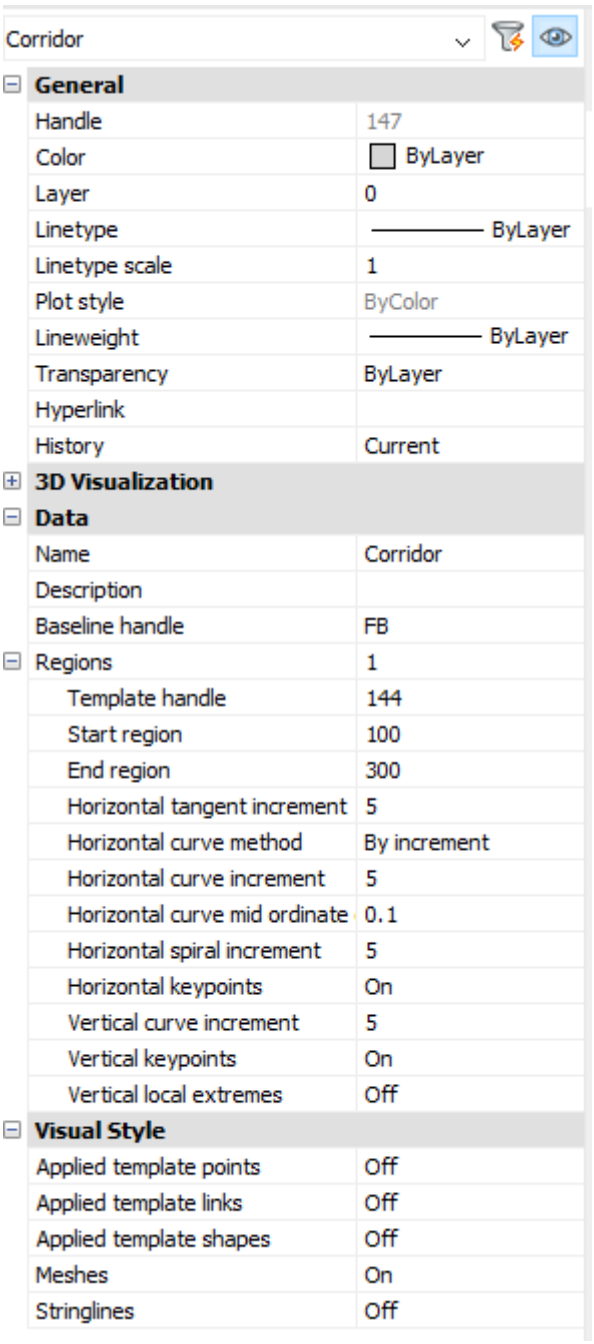
5 Press ENTER and you have finished your edited corridor.



### 9.13.12 Displaying the properties of a Corridor

1 Select the Corridor.

2 The properties of the Corridor are displayed in the Properties panel.



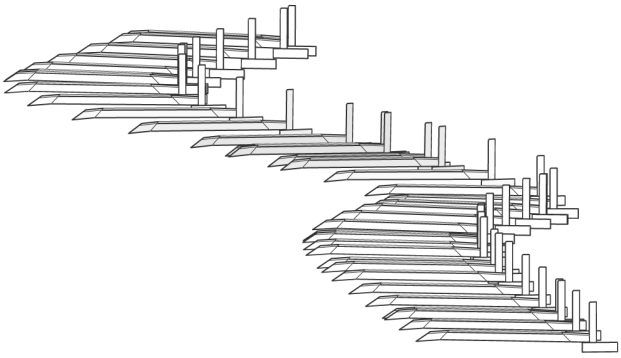

**Applied template points**

Represents the corridor with template element points.

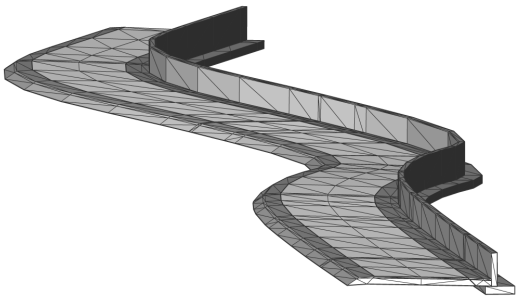
**Applied template links**
Represents the corridor with template element links.



**Applied template shapes**
Represents the corridor with template element shapes.



**Meshes**
Represents the corridor with meshes.



**Stringlines**

Represents the corridor with lines connecting the template element points along the corridor.

# 10. Параметрическое моделирование

## 10.1 Parametric modeling overview

BricsCAD provides a rich toolset to control 2D and 3D geometric models with parameters of different kinds. This includes:

- **Global parameters**: parameters of 2D/3D dimensional constraints, user-defined parameters, configuration parameters of design tables, etc.
- **Local parameters**: parameters of arrays and constrained entities inside arrays, parameters of inserts of parametric components, etc.

All parameters and their current values can be found in different sections in the **Mechanical Browser** panel. Parameter values can be linked together via expressions. When you change the value of a parameter, the model is updated automatically. This workflow allows you to add parametric behavior to any geometry and explore design variants easily.

***Прим.:*** In BricsCAD there are no parent-child dependencies between geometric elements: for example, if you change a sketch used to create an extruded 3d solid, the solid is not changed accordingly. However, you can create dependencies with parameters and expressions afterward.

### 10.1.1 Global and local parameters

Commands:

- -PARAMETERS command manages dimensional constraint parameters through the Command line.
- MECHANICALBROWSEROPEN command opens the **Mechanical Browser** panel.
- CLEANUNUSEDVARIABLES command purges parametric variables not used by constraint expressions and not linked to dimensions.

Parameters in BricsCAD can be either global or local.

Local parameters are attached to a particular entity, an associative array or a mechanical component.

Global parameters are not attached to a particular entity, they are subdivided into four groups:

- Parameters of 2D dimensional constraints
- Parameters of 3D dimensional constraints
- User-defined parameters
- Configuration parameters of design tables

### 10.1.2 Defining an expression for a parameter

You can define an expression for both parameter types, global and local. The simplest expression is a number or the name of a global parameter (local parameters cannot be referenced in expressions). A more complex expression can include standard operators and functions.

To define an expression for a parameter: in the **Mechanical Browser** panel, click on the parameter name and fill out the **Expression** field.

***Прим.:*** You can also define an expression for a parameter in the Command line using the -PARAMETERS command.

### 10.1.3 Defining dependent parameters

In expressions, you can use both: names of global parameters and numbers. This is known as a 'non-constant expression'. If a non-constant expression is set to a parameter, its icon changes from an open padlock () to a closed padlock () in the **Mechanical Browser** panel.

This means that the parameter becomes dependent on (an)other (defining) parameter(s) and its value will be automatically recalculated when the value(s) of the defining parameter(s) change(s).

You can create chains of parameters, where each element is dependent on the previous one. There is no limit to the length of such chain. However, be careful not to create a loop where the next element in a chain drives one of the previous elements. This might result in an infinite evaluation of expressions. BricsCAD automatically detects such problems and does not accept expressions that lead to a parametric loop.

### 10.1.4 Parameters of associative arrays

Associative arrays are created by the ARRAY, ARRAYRECT, ARRAYPOLAR and ARRATYPATH commands. The properties of associative arrays that can be controlled with an expression are the following.

- **Rectangular arrays**: Number of columns, Column spacing, Number of rows, Row spacing, Row elevation increment, Number of levels, Level spacing, and Axis angle.
- **Polar arrays**: Radius, Number of Items, Angle between items, Fill angle, Number of rows, Row spacing, Row elevation increment, Number of Levels, and Level spacing.
- **Path arrays**: Item spacing, Start offset, Rows, Row spacing, Row elevation increment, Levels and Level spacing.

To define an expression for any of these properties, select the corresponding array entity, either in the drawing area or in the **Mechanical Browser** panel and enter the expression in the corresponding properties field in the **Mechanical Browser** panel or in the **Properties** panel.

If an associative array contains entities constrained with 2D or 3D dimensional constraints, then parameters of these constraints can also be evaluated with expressions. These nested parameters are listed in the **Mechanical Browser** panel. If you select any of them, you can define an expression.

### 10.1.5 Parameters of mechanical components

The BMUPDATEMODE system variablecontrols whether external assembly components are always reloaded or only when modified.

If you insert a mechanical component or just a plain .dwg file into a drawing with BMINSERT command and the insert contains global exposed parameters, these parameters are converted to local parameters. They are associated with the corresponding component insert and are visible in the **Mechanical Browser** panel under the name of this insert. Click the parameter(s) to edit.

### 10.1.6 Parameter properties

All parameters such as; user-defined, 3D constraints, design tables, parameters of parametric components and arrays can be managed via **Mechanical Browser** panel.

- **Name**: Identifies the parameter. The parameter name can be used in expressions of other parameters in the same **component**. Name cannot begin with a number, it should be alphanumeric without spaces.
- **Expression**: Type a value or an expression.
- **Value**: Displays the current value of the parameter.
- **Geometry-driven**: If yes, disables the **Expression** property. The value of a parameter value is obtained from the model and can be changed with direct modeling operations.
- **Description**: Optional description of the parameter.
- **Exposed**: Defines whether the parameter is visible and can be modified when the component is inserted in an assembly. Select the field, then click the down-arrow button and choose an option. A new parameter gets the **Auto** option by default.
  - **Auto** = the parameter is exposed only if it does not depend on other parameters.
  - **ON** = the parameter is always exposed.
  - **OFF** = the parameter is never exposed.
- **Units**: Sets the dimension of the parameter: **Linear**, **2-dimensional** or **3-dimensional**. Select the field, then click the down-arrow button and choose an option.

### 10.1.7 3D constraints properties

BricsCAD provides a set of properties for 3D constraints which allow you to control mutual placement of the constrained geometrical entities (or constraint arguments) upon 3D constraints recalculation.

The properties may be assigned both to a constraint itself (**Directions** property) and/or to the constraints arguments, for each argument separately (**Placement** and **Use as** properties).

- **Type**: Indicates the constraint type: Path, Distance, Radius, Angle or Cone Angle.
- **Enabled**: Controls whether the constraint is On or Off.
- **Directions**: Specifies whether the directions of the constraint arguments are **Same** or **Opposite**. Changing from one to another flips constrained entities with respect to each other.

  Direction is used for lines, planes, circles, cylinders, cones and torii. It is not used for points and spheres.

  The direction of a plane is its normal vector. The direction of a circle is the normal vector of the circle's plane. The direction of a cylinder, cone, or torus is the direction of its axis. The direction of a planar face of a 3D solid is additionally adjusted to be external with respect to the 3D solid object.

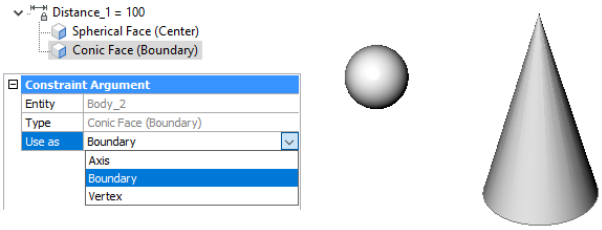
  Any option means that the directions may change upon direct modeling operations or upon a 3D constraint parameter change.

  The **Keep** option forces BricsCAD to keep the current directions. In other words, this option forces BricsCAD to analyze the mutual location of the arguments prior to 3D constraints solving and use either **Same** or **Opposite** (not both).

- **Placement**: Specifies how to place an argument with respect to another argument of the same constraint. For example, a point in a point-sphere distance may be placed either **Outside** or **Inside** of a sphere. Changing this property for a constraint argument moves the argument to the other side of another argument. The sides are defined for planes, spheres, cylinders, cones, and torii. And not defined for points, lines, and circles.

  **Any** means that both **Outside** and **Inside** placement is suitable for user's intentions.

  **Keep** option forces BricsCAD to keep the current placement of the argument. In other words, this option forces BricsCAD to analyze the mutual location of the arguments prior to 3D constraints solving and use either **Outside** or **Inside** (not both). The **Keep** option is used by default.
- **Use as**: This property is assigned to the arguments of the **Distance** and **Concentric** constraints and applies to circles, spheres, cylinders, cones and torii and defines which geometry is used in a 3D constraint to measure the distance or to coincide with the other entity.




  There are four options for the **Use as** property:

  - **Boundary**: the constraint is applied to the boundary of the argument;
  - **Axis**: applies to an argument that has an axis: circle, cylinder, cone, or torus;
  - **Center**: applies to an argument that has a central point: circle, sphere, or torus;
  - **Vertex**: applies to a cone vertex only.

### 10.1.8 Context menus

In the **Mechanical Browser** panel, the following context menus are available.

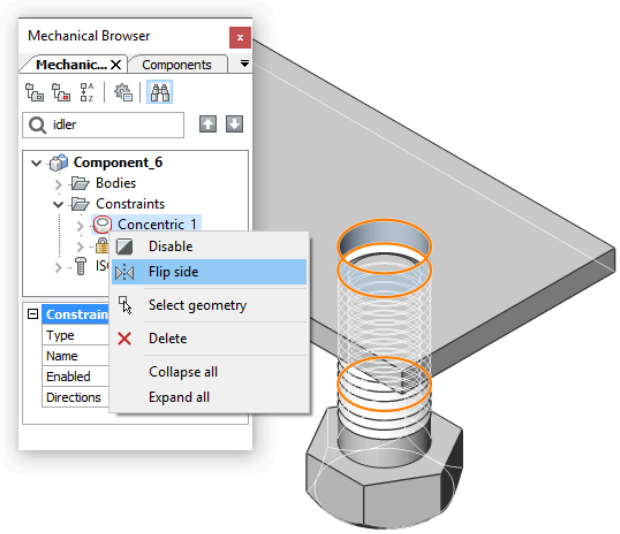
**Main Node**

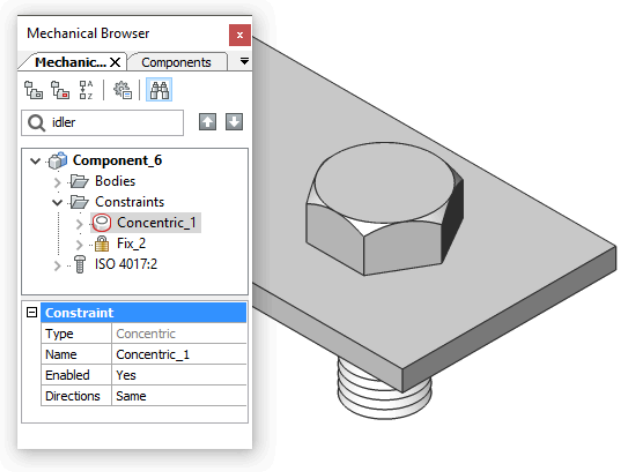
- **Add new parameter**: Creates a new parameter.
- **Collapse All**: Collapse all nodes.
- **Expand All**: Expand all nodes.

**Sub-nodes**

- **Delete**: deletes the selected parameter from file.
- **Create design table**: creates a design table to drive parametric block parameters.
- **Animate**: animates the parameter value within some range in the drawing.
- **Link to parameter**: links a subcomponent parameter to the main level parameter.
- **Enabled**: controls whether the constraint or parameter is evaluated or not.
- **Geometry driven**: if selected, disables the **Expression** property. The value of a parameter value is obtained from the model and can be changed with direct modeling operations.

- **Flip Side**: Allows you to change the relative position of an entity (vectors normal of the selected faces point in the same direction or in the opposite direction). This option can be applied to Parallel, Coincident and Concentric constraints only and on condition the constraint is applied to faces.




- **Select geometry**: Selects the geometry that is affected by the constraint, in the drawing.

### 10.1.9 The Parameters and Constraints panel

The **Parameters and Constraints** panel allows you to browse and manage global parameters of your model in tabular form. Every row in this table corresponds to one global parameter, while every column represents a particular property of a parameter - name, expression, value.

In the **Parameters and Constraints** panel you can:

- Create a new parameter
- Delete an existing parameter
- Click a parameter name field and change it
- Click a parameter expression field and change it

- Search a parameter by name
- Sort all parameters by name, expression, and value
- Hide all parameters except those relevant to the selected parameter
- Hide all parameters except those dependent on the selected parameter
- Animate a parameter

## 10.2 Operations and functions in expressions

### 10.2.1 Operators

| Operation | Character | Description |
|---|---|---|
| Addition | + | Produces the sum of numeric values. |
| Subtraction | - | Subtracts the two numeric values. |
| Multiplication | * | Multiplies the numeric values. |
| Division | / | Divides two numeric values. |
| Exponentiation | ^ | Calculates the exponential value of the given numbers. |
| Remainder operator | % | Gives the remainder after the division of one numeric value to another. e.g. 5%2 = 1, because 5/2 leaves remainder of 1. |

### 10.2.2 Expressions

Expressions are evaluated according to the standard mathematical rules of precedence:

1 Expressions within brackets; innermost sets first.

2 Standard operations order:
   - 1. exponent
   - 2. multiplication and division
   - 3. addition and subtraction

3 Operators of equal precedence from left to right.

The following functions can be used in expressions:

| Function | Syntax |
|---|---|
| Cosine | cos(expression) |

| Function | Syntax |
|---|---|
| Sine | sin(expression) |
| Tangent | tan(expression) |
| Arc cosine | acos(expression) |
| Arc sine | asin(expression) |
| Arc tangent | atan(expression) |
| Hyperbolic cosine | cosh(expression) |
| Hyperbolic sine | sinh(expression) |
| Hyperbolic tangent | tanh(expression) |
| Arc hyperbolic cosine | acosh(expression) |
| Arc hyperbolic sine | asinh(expression) |
| Arc hyperbolic tangent | atanh(expression) |
| Square root | sqrt(expression) |
| Signum function (-1,0,1) | sign(expression) |
| Round to nearest integer | round(expression) |
| Truncate decimal | trunc(expression) |
| Round down | floor(expression) |
| Round up | ceil(expression) |
| Absolute value | abs(expression) |
| Largest element in array | max(expression1;expression2) 1 |
| Smallest element in array | min(expression1;expression2) 1 |

| Function | Syntax |
|---|---|
| Degrees to radians | d2r(expression) |
| Radians to degrees | r2d(expression) |
| Logarithm, base e | ln(expression) |
| Logarithm, base | 10 log(expression) |
| Exponent, base e | exp(expression) |
| Exponent, base 10 | exp10(expression) |
| Power function | pow(expression1;expression2) 1 |
| Random decimal, 0-1 | Random |

(1) Use the list separator character as defined on your system: , (comma) or ; (semicolon).

***Прим.:***

- The constants **Pi=3.14...** and **e=2.72...** can be used in expressions. The constants names are not allowed to be used as a parameter or constraint name.
- The CLEANUNUSEDVARIABLES command purges parameters that are not used in constraint expressions or linked to dimensions.

## 10.3 3D constraints

### 10.3.1 Variational Direct Modeling

Use geometric constraints (aka variational direct modeling) to model parametric solids and surfaces. Define constraints between edges and faces to modify solid and surface models.

As soon as a constraint is added, BricsCAD automatically modifies the model and these constraints are updated with subsequent modifications to the model. The key advantage of variational direct modeling is that all constraints are taken into account simultaneously and the model behavior does not depend on the constraint creation order. It allows you to parametrize any feature of the 3D model without the worry of model creation history.

Dimensional constraints control the dimensions of the model. When such a constraint is introduced or the value of its parameter is changed, BricsCAD automatically updates the solid and surface geometry. For example, the dimensions of a box can be controlled by the parameters of three distance constraints applied to its opposite faces.

BricsCAD provides the following 3D constraints:

- Geometrical constraints: **Fix** (), **Coincident** (), **Concentric** (), **Parallel** (), **Perpendicular** (), **Tangent** (), and **Rigid Set** ().
- Dimensional 3D constraints: **Distance** (), **Radius** (), **Angle** (), **Cone Angle** (), **Path Constraint** ().

3D Constraints are available in the **3D Constraints** toolbar, the **Parametric/3D Constraints** menu, the **Home / Parametrize** ribbon panel and the **Constraints** tab on the Quad.

All 3D constraints are listed in the **Mechanical Browser** panel. Select a constraint to check and edit its properties.

The following (sub)entities and items are supported for 3D constraint creation:

- Vertices, edges and faces of 3D solids, surfaces, meshes and regions and surfaces of planar, cylindrical, spherical, toroidal and conical geometry.
- Lines, rays, xlines, circles and arcs.
- Segments and vertices of 3D polylines.
- Arbitrary curves for coincident and path constraints with a point as another argument.
- Planes, axes and origin points of the WCS or a block coordinate system.

3D constraints have a number of properties that allow you to control how BricsCAD treats them in various situations.

### 10.3.2 Geometry-driven constraints

Dimensional 3D constraints can also be **geometry-driven**. They update the model to accurately reflect the current state of the model. You can not specify their expression. Instead, their value is obtained from the model and can be changed with direct modeling operations and changes to other constraints.

Other parameters, including constraints, can depend on the values of geometry-driven constraints. In some circumstances a constraint dependent on geometry-driven constraints can make the model unresolvable. This should be avoided.

You can turn regular constraints into geometry-driven constraints and vice versa, in the **Mechanical Browser** panel and the **Parameters and Constraints** panel or using the DMCONSTRAINT3D command. After a regular constraint is turned into a geometry-driven one, its expression will be overwritten with the first modification of the model that affects this constraint.

### 10.3.3 3D constraints and direct modeling

3D constraints are taken into account when direct modeling operations are applied. If there is a 3D constraint that fixes the placement of faces or edges, this constraint will be preserved during Push/Pull, Move and Rotate operations. A 3D constraint can be temporarily disabled, or erased permanently in the **Mechanical Browser** panel.

Geometry-driven constraints update their values when a direct modeling operation is completed, which possibly leads to additional charges in the model.

If some faces or edges disappear during direct modeling or Boolean solid editing operations, 3D constraints applied to such entities are removed. However, if there is an exact correspondence between the initial faces and the faces obtained, 3D constraints are automatically applied to the new entities.

Depending on the value of the DMRECOGNIZE system variable, BricsCAD automatically recognizes geometrical relations between the surfaces of a solid and will preserve them during direct modeling operations. This feature is called 'design intent recognition'.

## 10.4 Suppression feature

The suppression feature temporarily suppresses a set of entities in your drawing. If the condition is equal to zero, all entities are displayed. Otherwise, entities are suppressed.

Suppressed entities are internally placed on a special frozen layer. This allows you to speed up orbit, zoom and pan operations, improves object selection performance and reduces regeneration time for complex drawings. Suppressed entities are also excluded from mass properties calculations (generated drawing views and bill of materials). Use a parametrically-driven expression for the condition parameter of a suppression feature, to create different design variants of your parametric components.

### 10.4.1 Creating a suppression feature

Execute the SUPPRESS command.

### 10.4.2 Managing suppression features

Select the suppression feature in the **Mechanical Browser** panel, then right-click to display a context menu.

The available options are:

- **Suppress**: Suppresses the entity set.
  - The **Suppressed** icon () indicates the suppressed state.
  - The option now reads **Unsuppress**.
- **Unsuppress**: Unsuppresses the entity set.
  - The **Unsuppressed** icon () indicates the unsuppressed state.
  - The option now reads **Suppress**.
- **Dissolve**: Deletes the suppression feature.
- **Edit Selection**: Allows you to add/remove entities from the selection.

  All members of the selection are highlighted.
  - Select entities to add.
  - Shift + Select (hold down the Shift key, then use a selection method on) entities to remove.
  - Right-click to stop editing.
- **Zoom to**: Zooms onto selected entities.

  To compose a selection set:
  - Click a suppression feature to select all its members.
  - Expand a suppression feature, then click to select an entity.

- Ctrl + Click to add/remove entities from the selection.

- **Select**: Selects all members of a suppression feature.

### 10.4.3 Setting a condition for a suppression feature

You can set an expressional condition for a suppression feature. Depending on the value of the current expression, the entities will be suppressed or unsuppressed automatically.

**In the Mechanical Browser panel:**

1 Select the suppression feature (use the "Search" field to search by name).

2 Enter an expression in **Condition** field in the properties grid.

**In the Parameters and Constraints panel:**

1 Select the parameter with the same name as the suppression feature (use the "Search" field to search by name).

2 Enter an expression in the **Expression** column.

## 10.5 Working with design tables

A design table allows you to evaluate several global parameters at once with the value of one configuration parameter. You can create design tables from scratch in BricsCAD or import them from a .csv file (a comma-separated values text file). You can create such files with a spreadsheet software package or a text editor.

### 10.5.1 Creating a design table from a .csv file

The **From file** option of the DESIGNTABLE command imports .csv files that contain sets of parameter values, or else creates them from scratch. Parameters are added to the **Mechanical Browser** panel and to parametric component properties.

The expected format of the .csv file is:

Configuration_Name,Parameter1,Parameter2,...,ParameterN

Configuration_1,Value1Parameter1,Value1Parameter2,...,Value1ParameterN

Configuration_2,Value2Parameter1,Value2Parameter2,...,Value2ParameterN

Configuration_3,Value3Parameter1,Value3Parameter2,...,Value3ParameterN

...

The first row lists the parameter names separated by a comma. The first name must be different from the name of any global parameter in your drawing. It is called the configuration name. No spaces are allowed in the configuration name. Other names in the first row can be identical to the names of a global parameter The next rows list possible values for these parameters as strings or numbers.

Example:

Size,Length,Width,Height

Small,300,200,100

Medium, 450,300,150

Large,600,400,200

When the .csv file is imported into BricsCAD using the DESIGNTABLE command, a special global parameter is created by the name of the first name in the first row of the .csv file ('Size' in the above example). This is called the 'configuration parameter'. It will drive the values of other parameters in the design table. These are called 'configuration-dependent parameters'.

***Прим.:*** BricsCAD does not maintain an association with the .csv file used to create a design table: if this file is modified, then the design table is not updated automatically. Use the -DESIGNTABLEEDIT command to replace a design table with a new or modified .csv file.

### 10.5.2 Direct lookup in a design table

To evaluate a configuration parameter:

- Select it in the **Mechanical Browser** panel and type the desired value in its **Expression** field.
- Select a value in the drop-down list of the **Value** field:

If you change the value of a configuration parameter, all configuration-dependent parameter values change automatically in the corresponding design table. The values of these parameters are taken from the row in the design table. The first element coincides with the current value of the configuration parameter.

### 10.5.3 Reverse lookup in a design table

You can evaluate parameters that are defined by a configuration parameter with a design table independently.

Select the dependent parameter and apply an expression to it.

- If the value of this expression matches one of the values in the corresponding design table, then the configuration parameter and all dependent parameters of the design table will automatically change to the values specified in the corresponding row in the table.
- If several configurations are possible for a particular value of a dependent parameter, BricsCAD selects the one closest to the current values of the other parameters.
- If you enter an expression and the result is not available in the design table, or if you enter expressions for two or more design table parameters and the values cannot be found in a single configuration (the values are not found in the same row in the design table), then the values of the configuration parameter are changed to 'Custom' and the values of the other dependent parameters are not changed. This case corresponds to a configuration that does not exist in the design table.

### 10.5.4 Adding a new configuration to a design table

You can add any 'Custom' configuration to a design table and assign it a name. To do so, right-click the configuration parameter in the **Mechanical Browser** and select **Save current configuration** in the context menu. The configuration will be added to the design tables as Configuration_N:

- **Configuration**: the name of the configuration parameter.
- **N**: an index number - one more than the current number of configurations in the design table.

If you want to save a configuration in the design table under a different name, launch the -DESIGNTABLEEDIT command.

### 10.5.5 Creating an empty design table

If you use the procedure described above, it is possible to start your design with an empty design table and then fill it with the desired values.

Use the **Empty** option of the DESIGNTABLE command.

You will be prompted to specify:

- The name of a new design table
- The name of the corresponding configuration parameter
- The names of global parameters that will be dependent on this design table.

### 10.5.6 Using a parametric component with design tables

You can use both configuration parameters and configuration-dependent parameters in assemblies. If you then insert such a component into your drawing, its parameters will include the exposed parameters of design tables. A drop-down list of available values will be available. Both direct and reverse lookup actions are supported for such parameters.

You must associate the design table with the parametric component drawing first.

1 Open the drawing.

2 On the **Mechanical Browser** panel, you will see the list of parameters and their values.

3 Launch the DESIGNTABLE command and press Enter to accept the **From file** option.

The **Select design table file** dialog box appears.

4 Select the .csv file, then double-click or click the **Open** button.

5 Check the **Value** property for each of the main parameters. You can select the default values to use when you insert the component.

6 Check the **Exposed** property for each of the main parameters. They should all be set to **Auto**.

7 Save the component drawing.

### 10.5.7 Other actions with design tables

The -DESIGNTABLEEDIT command allows you to:

- Delete a design table
- Replace a design table
- Link a new parameter
- Unlink an existing parameter
- Add new configurations, remove an existing one
- Export a design table as a .csv file.

# 11. Облака точек

## 11.1 Working with Point cloud

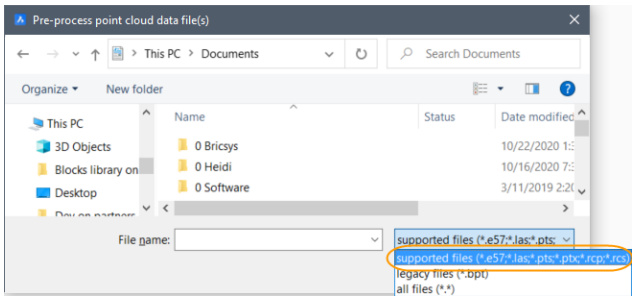
### 11.1.1 Commands

,,,,,,,, ,, ,,,,,,, ,, ,,

Point cloud functionality is significantly faster and more powerful in BricsCAD v21 and is even more powerful with a BIM license.

### 11.1.2 Pre-processing

The E57 file format is added to the supported files for pre-processing.

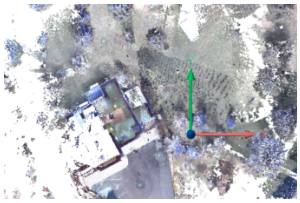


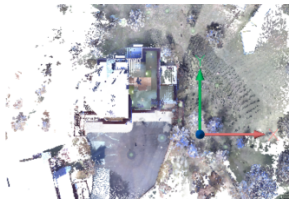

When you attach a point cloud that requires pre-processing, it converts 5 to 8 times faster in BricsCAD v21 than in v20.

The log file can be found in *C:\Users\USERNAME\AppData\Roaming\Bricsys\BricsCAD \V21x64\en_US\PointCloudCache\{Folder_for_processed_pointcloud}*, only if this path is set in POINTCLOUDCACHEFOLDER system variable. Inside this folder, the current state of processing can be observed.

### 11.1.3 Alignment

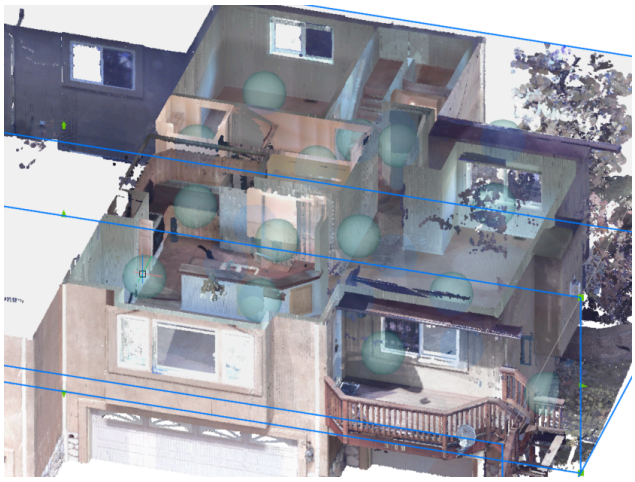
The new POINTCLOUDALIGN command automatically rotates a point cloud to optimally align it with the X and Y axis. To determine the best alignment, it can analyze the entire point cloud, or you can specify the most relevant area.





### 11.1.4 Bubble Viewer

The point cloud displays bubbles at all the scan locations. It’s at those locations where you will experience the most realistic visual representations.



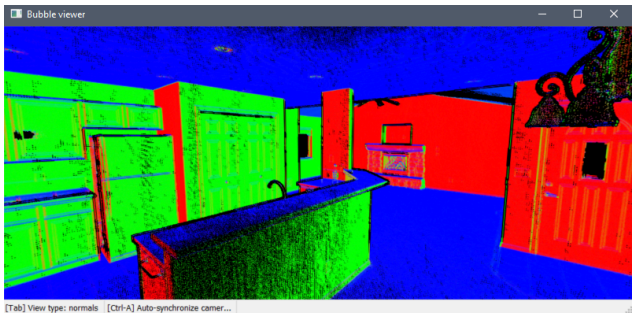
Double-click one of the bubbles in model space to open the Bubble viewer. You can press the middle mouse button and move the mouse to view the point cloud in any direction from that scan location. You can also zoom in and out using the mouse wheel.



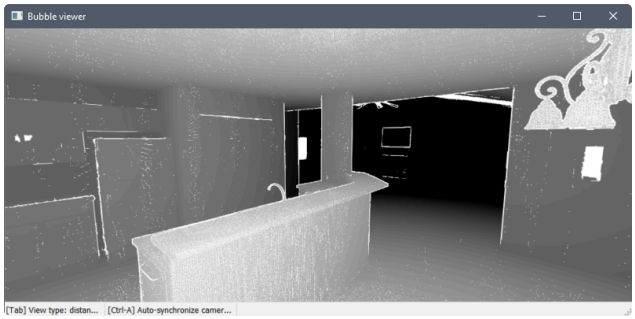

Press the Tab key to cycle between three different visual modes.

The first mode displays the points as their actual colors or in greyscale, depending how the data was scanned.

The second mode displays the points as red, green, or blue according to their normal vectors. The colors correspond to the UCS axes.




The third mode displays the points from light to dark as the distance from the scan location increases.




You can easily sync the drawing view to match the Bubble viewer by pressing Ctrl A.



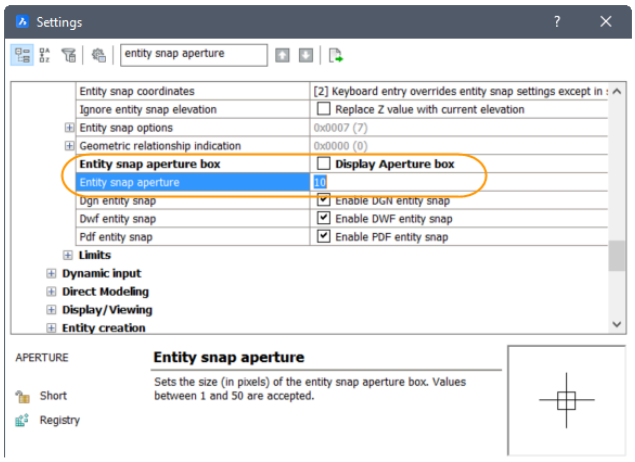
### 11.1.5 Entity snaps

The new Point Cloud Nearest Point entity snap significantly improves your ability to select relevant point cloud points. It uses an imaginary cylinder from the current viewpoint toward the cursor.



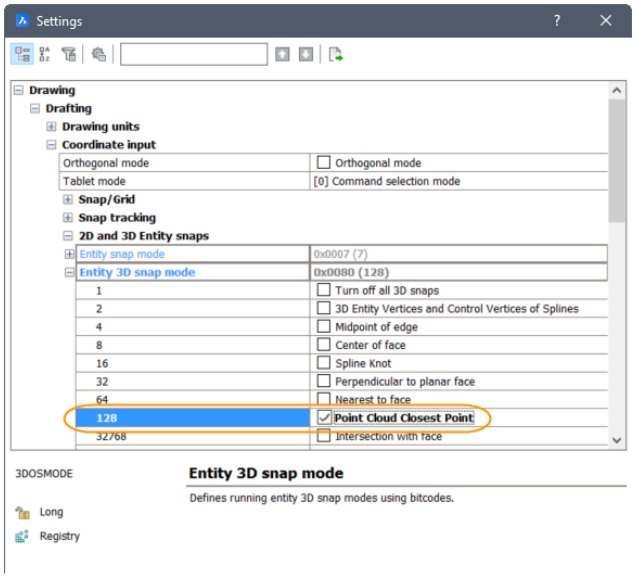


The radius of the imaginary cylinder is defined by the Entity snap aperture box setting.




Enable the new Point Cloud Nearest Point entity snap along with other 3D entity snaps in the Entity Snap menus, toolbar, and settings.

***Прим.:*** Make sure that only the bit 128 of 3DSMODE is set.



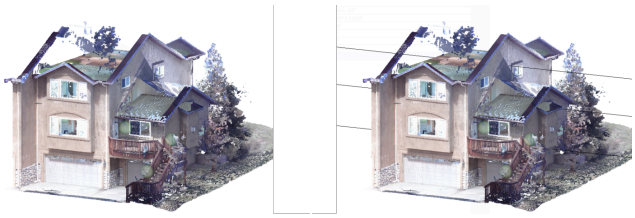
### 11.1.6 Export

The new POINTCLOUDEXPORT command allows you to export a cropped selection of a point cloud to a .pts file.

### 11.1.7 Floor Detection

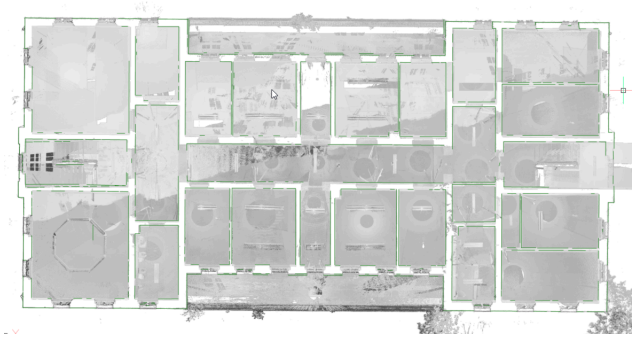
SectionPlanes work on point clouds as well, they can be used to show parts of point clouds. The difference between point cloud crops and SectionPlanes is that point cloud crop only clips the point cloud while section planes will clip all geometry in your drawing.

The new POINTCLOLUDDETECTFLOORS command generates volume sections for each floor found in a point cloud representing a building. The detection is based on regions of points with similar Z-coordinates. The generated volume sections can help in navigating point clouds of buildings.

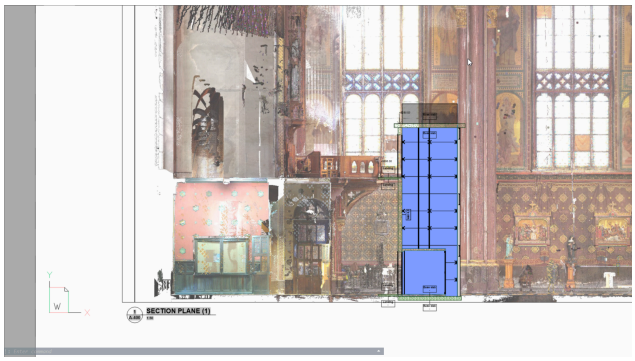


### 11.1.8 Point cloud projection

The newPOINTCLOUDPROJECTSECTIONcommand enables you to detect walls from the volume section of a point cloud based on a variety of wall detection options. You can create volume sections automatically for each floor in a building. You can use these sections to generate 2d lines to create a 2d floorplan or a vertical section. This is a background process and multiple sections can be processed in a queue. This way it is possible to run this command in full resolution on all sections.



At the same time, a raster image will be generated. In some cases, it is not necessary to recreate the existing building. Background images can give so much more context to the design documents. These can be used to verify the created 2d geometry but in high quality scans these images can also be used as graphical material. For example, as a background image for a BIM model in renovation projects where modern intervention are made in historical buildings.

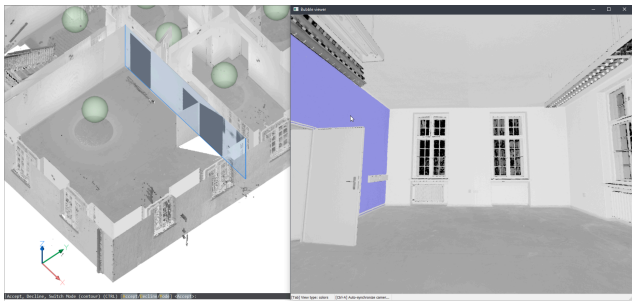


### 11.1.9 Planar Fit

The newPOINTCLOUDFITPLANARcommand enables you to create 3d geometry based on the point cloud. It will create a planar surface or solid after a selection of one point in a point cloud. The points that seem to be in a plane are never exactly in one plane, therefore a threshold value is set as a property of the point cloud entity. This also works in bubble view.

### 11.1.10 In bubble view

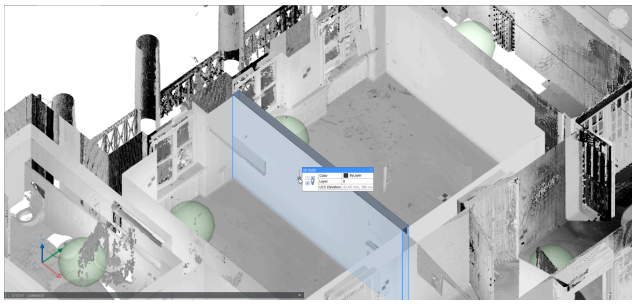
If the bubble viewer is open before launching the command, BricsCAD expects you to select in the bubble viewer. The cursor will give you a preview of the direction of the plane. When you click you get a preview in both bubble view and model view. You can toggle between 2 shape representations using the CTRL-key.



### 11.1.11 In model space

You can also use this command in the model space when the bubble viewer is not open. BricsCAD will ask you to select a point of the point cloud in model space. Depending on the size of the cropped point cloud, it takes more time, but it has 2 advantages by searching multiple scan positions:

- It can create larger surfaces where only parts are visible in each scan position
- It can detect wall and slab thickness since it can take the opposite surface into account.



# 12. Блоки, атрибуты, внешние ссылки и компоненты

## 12.1 Создание блоков

Команда BLOCK группирует существующие объекты в блок с помощью диалогового окна.

Команда -BLOCK создает блок из существующих объектов с помощью командной строки.

Команды BEDIT и BCLOSE используются для создания и редактирования блоков в режиме временного сеанса черчения.

Команда WBLOCK сохраняет блоки и другие элементы чертежа в виде отдельного файла чертежа.

Команда PASTEBLOCK вставляет объекты из **буфера обмена** в текущий чертеж в виде блока.

Команда SECTIONPLANETOBLOCK сохраняет результат применения выбранной секущей плоскости в виде 2D/3D блока.

### 12.1.1 Создание блока

1 При необходимости, выберите объекты для включения в блок.

2 Запустите команду BLOCK.

3 Откроется диалоговое окно **Описание блока**:

4 В поле **Имя** введите имя нового блока.

5 Чтобы указать **базовую точку** блока, выполните одно из следующих действий:

- Установите флажок в поле **Указать на экране**.

  После нажатия кнопки **ОК** программа выдаст запрос на указание базовой точки на чертеже.

- Нажмите кнопку **Указать точку** ( ).

  Окно **Описание блока** временно закроется для выбора точки на чертеже.

- Введите координаты базовой точки в полях X, Y и Z.

6 В разделе Объекты отображается количество выбранных в шаге 1 объектов.

7 Для выбора объектов, входящих в состав нового блока выполните одно из следующих действий:

- Выберите объекты перед запуском команды BLOCK.
- Установите флажок в поле **Указать на экране**.

  После нажатия кнопки **ОК** программа выдаст запрос на указание объектов на чертеже.

- Нажмите кнопку **Указать объекты** ( ).

  Окно **Описание блока** временно закроется для выбора объектов на чертеже.

8 Установите свойства блока:

- **Аннотативность**: при включении этой опции создается аннотативный блок.
- **Устанавливать ориентацию блоков по листу**: при включении этой опции сохраняется ориентация аннотативного блока при повороте содержимого видового окна.
- **Единый масштаб**: при включении этой опции масштабы вдоль осей X, Y и Z будут

одинаковыми.

- **Расчленяемый**: отключение этой опции защищает блок от расчленения.
- **Единицы измерения блока**: определяет единицы измерения, используемые при вставке блока (см. раздел Вставка блока).

9 Определите действия с выбранными объектами после создания блока:
- **Сохранить**: Выбранные объекты сохраняются в чертеже.
- **Преобразовать в блок**: Выбранные объекты замещаются созданным блоком.
- **Удалить**: Выбранные объекты удаляются.

10 Нажмите кнопку **ОК** для создания блока.

**Примечания:**

- Если выбранные для создания блока объекты были удалены (опция **Удалить** в п.6), вы можете восстановить их путем ввода в командной строке команды OOPS и нажатия клавиши Enter.
- Если при выполнении шага 3 вы введете имя существующего блока, программа предложит вам перезаписать существующее определение блока. Существующие экземпляры блока будут заменены новым определением блока.

### 12.1.2 Сохранение блока в отдельный файл

1 При необходимости, выберите объекты для включения в блок.

2 Запустите команду WBLOCK.

3 Откроется окно **Запись блока в файл**.

4 Нажмите кнопку обзора справа от поля **Имя и путь**.

Откроется окно **Сохранить блок**.

5 Выберите папку, введите имя в поле **Имя файла** и нажмите кнопку **Сохранить**.

Диалоговое окно **Сохранить блок** закроется.

6 **Единицы вставки**: определяет значение системной переменной INSUNITS для файла блока.

По умолчанию это значение переменной INSUNITS для исходного чертежа.

7 В разделе Объекты отображается количество выбранных в шаге 1 объектов.

8 Для выбора объектов, входящих в состав нового блока выполните одно из следующих действий:
- Выберите опцию **Блок** и укажите необходимый блок в выпадающем списке.
- Выберите опцию **Весь чертеж** для выбора всех объектов пространства модели текущего чертежа.
- Выберите опцию **Объекты** и нажмите кнопку **Указать объекты**.

  Окно **Запись блока в файл** временно закроется для выбора объектов на чертеже.

  Щелкните правой кнопкой мыши для завершения выбора объектов.

9 Нажмите кнопку **Указать точку** ( )для определения **Базовой точки** нового чертежа

или

Введите координаты базовой точки в полях X, Y и Z.

10 Определите действия с выбранными объектами после создания блока:
- **Сохранить**: Выбранные объекты сохраняются в чертеже.
- **Преобразовать в блок**: Выбранные объекты замещаются созданным блоком.
- **Удалить**: Выбранные объекты удаляются.

11 Нажмите кнопку **ОК** для создания файла чертежа.

***Прим.:*** Если в п.7 была выбрана опция **Удалить**, вы можете восстановить удаленные объекты блока с помощью команды OOPS.

## 12.2 Using Blockify



The Blockify tool enables you to automatically convert repetitive sets of entities to block definitions. Select a set of 2D entities or 3D solids and BricsCAD searches the drawing for identical sets of entities and automatically replaces them by block references.

For more information about this command, visit the BLOCKIFY Command Reference article.

### 12.2.1 Blockify from a selected input

1 Launch the BLOCKIFY command.

1 **You are prompted**: Select source entities or [match Equal solids/match existing Blocks/Settings/ selection options (?)] <match Equal solids>:

2 Select the entities you want to convert into a block and press Enter.

3 Entities selected: xx

**You are prompted**: Select source entities or [Settings/selection options (?)]:

4 Select more entities or press Enter or right click to stop selecting.

5 **You are prompted**: Select block insertion point or [use Default point] <use Default point>:

6 Identify the insertion point for the block.

7 **You are prompted**: Select search space or [Entire drawing/selection options (?)] <Entire drawing>:

8 Press Enter or right click to search the entire drawing.

9 **You are prompted**: Specify block name or <ENTER to use default name>:

10 Type a name for the block, then press Enter.

11 BricsCAD reports:

Duration of finding similar groups: x.xx sec

Number of block inserts: xx

***Прим.:*** Depending on the value of the BLOCKIFYMODE system variable, some prompts are suppressed and defaults are used for the insertion point, search space and/or block name.

### 12.2.2 Existing block definitions

One of the aims of the BLOCKIFY command is to automatically add and improve the structure of a model. If the model already contains block definitions, the existing block definitions are checked.

- **Use a selected input set of entities**: When one of the existing block definitions matches the BLOCKIFY input set, no definition will be created. All instances of the input will be replaced by block references to the existing block definition.
- **Match existing blocks**: The entire drawing, or selection set, is searched for matching geometry in existing block definitions. As such, repeated geometry corresponding to a block definition is replaced by block references to existing definition(s). This option allows to recreate block instances that were exploded accidentally.

### 12.2.3 Combining duplicate block definitions

Identical block definitions can be combined into a single one and optionally purge the duplicate using the OVERKILL command.

1 Launch the OVERKILL command.

1 **You are prompted**: Select entities or [Combine duplicate block definitions/selection options (?)]:

2 Choose the **Combine duplicate block definitions** option.

The **Combine duplicate block definitions** dialog box appears.

3 (option) Select the properties to ignore.

4 (option) Tick the **Purge duplicate definitions** option.

5 Click the **OK** button.

BricsCAD reports:

- Number of duplicate block definitions found
- Names of duplicate block definitions
- Purged block definitions

## 12.3 Редактирование блоков

Вы можете получить доступ к Редактору блоков с помощью команды BEDIT или двойным щелчком по блоку, предназначенному для редактирования. При использовании Редактора блоков BricsCAD отображает объекты блока во временном сеансе черчения. Он включает в себя все инструменты, необходимые для создания и редактирования определения блока. Чтобы выйти из сеанса редактирования, запустите команду BCLOSE и выберите опцию сохранения или отмены внесенных изменений. При выборе опции 'Сохранить' BricsCAD сохранит изменения в определении блока и закроет Редактор блоков.

Для редактирования блоков также может быть использована команда REFEDIT.

## 12.4 Вставка блоков

Команда INSERT позволяет выполнять вставку блоков с использованием диалогового окна.

Команда -INSERT вставляет блоки с помощью командной строки. BricsCAD выполняет поиск имени блока среди определений блоков текущего чертежа. Если блок не найден в текущем чертеже, будут использованы пути поиска, определенные значением системной переменной SRCHPATH. Если блок не будет найден и в этом случае, BricsCAD выведет сообщение 'Невозможно найти файл <имя блока>'.

Команда INSERTALIGNED вставляет блок с выравниванием по существующим объектам. Команда INSERTALIGNED аналогична команде INSERT, но расположение блока при вставке осуществляется динамически путем выравнивания его по существующим на чертеже отрезкам, полилиниям, дугам или окружностям. Блок выравнивается по объекту, выбранному в качестве привязки с помощью курсора. Если объекты привязки не выбраны, блок выравнивается по оси X МСК.

Команда MINSERT выполняет вставку блоков в виде прямоугольного массива, комбинируя команды -INSERT и ARRAY.

Вы можете выполнить вставку существующего определения блока или целого чертежа в виде блока. При вставке чертежа в текущем чертеже создается новое определение блока. При изменении исходного файла чертежа эти изменения не отображаются в текущем чертеже до тех пор, пока не будет выполнено переопределение блока путем повторной вставки измененного чертежа. Блок, вставленный в чертеж, представляет собой цельный объект.

**Примечание**:

- При вставке чертежа в качестве блока базовая точка вставляемого чертежа определяется значением переменной INSBASE и совпадает с точкой вставки блока. Значение переменной INSBASE устанавливается командой BASE.
- Если блок содержит атрибуты, в командной строке вам будет предложено ввести значение каждого атрибута.
- Если значение системной переменной DRAGOPEN равно 0, чертежи могут быть вставлены в виде блока путем перетягивания их файлов в текущий чертеж из окна **Проводника Windows**. Если значение переменной DRAGOPEN равно 1, то при перетягивании файла чертеж открывается.
- Системные переменные INSUNITS и INSUNITSSCALING управляют автоматическим масштабированием при вставке блоков или присоединении внешних ссылок, когда значение INSUNITS отличается в блоке или Xref и целевом чертеже.
- Если для переменной INSUNITS в исходном чертеже установлено значение 0 (единицы не указаны), вместо него будет использоваться значение переменной INSUNITSDEFSOURCE.
- Если для переменной INSUNITS в целевом чертеже установлено значение 0 (единицы не указаны), вместо него будет использоваться значение переменной INSUNITSDEFTARGET.
- Значения переменных INSUNITSDEFSOURCE и INSUNITSDEFTARGET сохраняются в реестре и применяются для всех чертежей, в которых не определено значение переменной INSUNITS.

- BricsCAD выполняет поиск указанного вами имени блока среди определений блоков текущего чертежа. Если блок не найден в текущем чертеже, будут использованы пути поиска, определенные значением системной переменной SRCHPATH. Если блок не будет найден и в этом случае, BricsCAD выведет сообщение 'Невозможно найти файл <имя блока>'.

В диалоговом окне **Вставка блока**:

- Значение в поле **Единицы** раздела **Единицы измерения** блока определяется значением переменной INSUNITS (Единицы чертежа), установленном в исходном чертеже или значением параметра **Единицы измерения блока**, установленном при определении блока (см. раздел Создание блока в чертеже).
- Значение в поле **Коэффициент** вычисляется как отношение значений переменной INSUNIT для внешней ссылки и исходного чертежа. Например, если в исходном и целевом чертежах значение INSUNITS=Миллиметры,значение в поле **Коэффициент** будет равно 1.0.

### 12.4.1 Использование команды INSERT

1 Запустите команду INSERT.

1 Откроется диалоговое окно **Вставка блока**.

2 Выполните одно из следующих действий:

- В раскрывающемся списке поля **Имя** выберите существующее определение блока.

  Имя выбранного блока отображается в поле **Имя**.
- Нажмите кнопку **Обзор** и выберите файл чертежа.

  В поле Путь отображается расположение выбранного чертежа.

3 При необходимости, включите опцию **Использовать базовые кривые для вставки**.

4 Выберите режим настройки параметров **Точка вставки**, **Масштаб** и **Угол поворота**.

Доступные опции:

- Настройка параметров вставки в диалоговом окне **Вставка блока**.
- **Указать на экране**: Программа будет выдавать запросы в процессе вставки.

5 При необходимости включите опцию **Расчленить** для расчленения блока после вставки.

6 Нажмите кнопку **ОК**.

Дальнейшие действия зависят от выбранной опции.

### 12.4.2 Вставка блока с помощью Проводника по чертежам

Вставка блока из текущего чертежа.

1 Запустите команду EXPBLOCKS.

1 Откроется диалоговое окно **Проводник по чертежам – Блоки**.

2 Выберите режим отображения блоков: **Подробно** или **Значки**.

3 Щелкните правой кнопкой мыши на выбранном блоке и в открывшемся контекстном меню выберите **Параметры…**.

Откроется диалоговое окно **Параметры Проводника по чертежам**.

Выполните настройку параметров вставки блока и нажмите кнопку **ОК**.

4 Дважды щелкните на значке блока или его номере (при использовании режима отображения **Подробно**).

Окно **Проводник по чертежам** временно закроется для выполнения вставки блока в чертеж.

5 Укажите точку вставки.

- В зависимости от выбранных ранее опций вы получите запросы на указание масштаба и/или угла поворота.
- Если была выбрана опция **Выровнять**, необходимо указать положение блока по отношению к выбранному линейному объекту.

  **Запрос программы**: Укажите управляющую точку для выполнения зеркального отображения блока:

Снова откроется диалоговое окно **Проводник по чертежам**.

Вставка блоков из другого чертежа.

1 Откройте диалоговое окно **Проводник по чертежам**.

2 В разделе **Чертежи** щелкните на закладке **Папки**.

3 Если на закладке **Локальные папки** нет доступных папок или, если вы хотите добавить папку, нажмите кнопку **Добавить локальную папку...**.

Откроется диалоговое окно **выбора папки**.

Укажите папку, которую необходимо добавить, и нажмите кнопку **Выбор папки**.

Выбранная папка и вложенные в нее папки будут добавлены в список **Локальные папки**.

4 Выберите чертеж в списке **Локальные папки**.

5 Раскройте структуру выбранного чертежа и выберите **Блоки**.

Все блоки выбранного чертежа отображаются в разделе **Блоки**.

6 Выполните действия предыдущей процедуры, начиная с п.2.

### 12.4.3 Вставка блока с выравниванием

1 Убедитесь, что включен необходимый режим объектной привязки.

2 Запустите команду INSERTALIGNED.

3 **Запрос программы**: Блок для вставки (~ чтобы открыть диалоговое окно файла) или [? для списка блоков чертежа] <текущий блок>:

4 Выполните одно из следующих действий:

- Нажмите клавишу Enter для подтверждения предложенного для вставки блока.
- Введите имя блока для вставки.
- Нажмите клавишу ? для просмотра списка доступных блоков.

**Запрос программы**: Точка вставки для блока или [Масштаб/Масштаб по X/Масштаб по Y/

Масштаб по Z/Массив блоков]:

5 При необходимости, выберите опцию **Несколько блоков** или укажите фиксированный масштаб.

6 Укажите точку.

7 При указании точки используйте объектную привязку.

**Запрос программы**: Укажите управляющую точку для выполнения зеркального отображения блока:

8 Переместите курсор для выбора необходимого расположения, затем щелкните кнопкой мыши, чтобы вставить блок в выбранную точку.

***Прим.:*** Выравнивание блока происходит следующим образом:

***Блок поворачивается так, что его ось Z становится параллельной оси Z текущей ПСК, а ось X - параллельна касательной к объекту в точке привязки. После указания точки привязки на объекте блок выравнивается по этому объекту. Затем, перемещая курсор вокруг точки вставки, вы можете выбрать вариант зеркального отображения блока. При этом точка вставки блока остается на объекте привязки.***

### 12.4.4 Вставка блоков в виде прямоугольного массива

1 Запустите команду MINSERT.

1 **Запрос программы**: Блок для вставки (~ чтобы открыть диалоговое окно файла) или [? для списка блоков чертежа] <текущий блок>:

2 Выполните одно из следующих действий:

- Нажмите клавишу Enter для подтверждения предложенного для вставки блока.
- Введите имя блока для вставки.
- Нажмите клавишу ? для просмотра списка доступных блоков.

**Запрос программы**: Точка вставки для блока или [Масштаб/Масштаб по X/Масштаб по Y/Масштаб по Z/Массив блоков]:

3 При необходимости, выберите опцию **Несколько блоков** или укажите фиксированный масштаб.

4 Укажите точку.

5 **Запрос программы**:Угол поворота блока <0°0'0">:

6 Нажмите клавишу Enter для подтверждения предложенного значения угла или укажите угол.

7 **Запрос программы**: Число строк массива <1>:

8 Введите число.

9 **Запрос программы**: Число столбцов массива <1>:

10 Введите число.

11 **Запрос программы**: Расстояние между строками или диагональ ячейки:

12 Выполните одно из следующих действий:

- Укажите значение.
- **Запрос программы**: Расстояние между столбцами:

  Укажите значение.

- Укажите две противоположные угловые точки прямоугольника, определяющего интервалы по горизонтали и вертикали.

Будет создан **Массив блоков**.

**Примечание**:

- При выборе опции **Несколько блоков** вы получите запрос на создание нескольких экземпляров массива блока.
- Расчленение объекта **Массив блоков** невозможно.
- Для редактирования свойств **Массива блоков** вы можете использовать **Панель свойств**.

## 12.5 Расчленение блоков

Если при определении блока была включена опция **Расчленяемый**, вы можете расчленять вставленные в чертеж блоки на входящие в их состав исходные объекты. Расчленение затрагивает только выбранные вставки блока. Исходное определение блока сохраняется в чертеже без изменений и вы можете вставлять в текущий чертеж другие копии этого блока. При расчленении блока, содержащего атрибуты, атрибуты теряются, но их исходные описания сохраняются в чертеже.

Системная переменная EXPLMODE управляет расчленением блоков, имеющих различный масштаб по осям X и Y.

### 12.5.1 Чтобы расчленить блок

1 Запустите команду EXPLODE.

1 **Запрос программы**: Укажите объекты для расчленения [Параметры выбора]:

2 Выберите один или несколько блоков.

3 Объектов в наборе: x

**Запрос программы**: Укажите объекты для расчленения [Параметры выбора]:

4 Выберите другие объекты или нажмите клавишу Enter, чтобы завершить выбор.

5 Выбранные блоки будут расчленены.

***Прим.:*** Расчленение разбивает сложные объекты на объекты с меньшей на одну ступень степенью сложности: блоки или полилинии, входившие в состав блока, после расчленения этого блока снова становятся блоками или полилиниями.

### 12.5.2 Настройка свойства Расчленяемый

Свойство **Расчленяемый** присваивается блоку при его создании с помощью команды BLOCK.

Для редактирования свойства выполните следующие действия:

1 Запустите команду EXPBLOCKS.

1 Откроется диалоговое окно **Проводник по чертежам – Блоки**.

2 Выберите режим отображения **Подробно**.

3 Установите/удалите флажок в столбце **Расчленяемый** для включения/выключения свойства

**Расчленяемый**.

## 12.6 Параметрические 2D блоки

Если вы работали ранее с AutoCAD®, то, возможно, уже знакомы с динамическими блоками. Возможно, вы также знакомы уже с параметрическими 2D зависимостями. Эти две различные функциональности могут иметь общую область применения.

Вместо применения динамических блоков в BricsCAD используется параметризация блоков. Вы можете создавать как 2D, так и 3D определения блоков, используя один и тот же набор инструментов и операций. При первом знакомстве с параметрическими зависимостями они могут показаться сложными. Но на самом деле их использование представляет собой достаточно интуитивный и логичный процесс и является очень мощным инструментом.

### 12.6.1 Основные процедуры при создании параметрических 2D блоков

Создание геометрии.

Создание геометрических зависимостей.

Создание размерных зависимостей и параметров.

Редактирование размерных зависимостей и параметров с помощью панелей **Параметры и зависимости** или **Обозреватель конструкции**.

Создание состояний видимости с помощью команды VISIBILITYSTATES для создания для отображения/скрытия геометрии.

Сохранение чертежа как библиотечного блока с помощью команды CREATELIBRARYBLOCK.

***Прим.:*** Команда PARAMETRIZE2D автоматически создает геометрические и размерные зависимости.

### 12.6.2 Вставка параметрических блоков

Вы можете выполнять вставку параметрических блоков в чертеж с помощью команд INSERT и BMINSERT или с использованием панели **Библиотека**.

### 12.6.3 Настройка значений параметров

Выберите параметрический блок и измените значения его параметров в разделе **Параметры** на **Панели свойств**.

## 12.7 Dynamic blocks

If you come from an AutoCAD® design environment, you may be familiar with Dynamic Blocks. You may also be familiar with 2D parametric constraints. They are two different functionalities that somewhat overlap.

If you insert AutoCAD dynamic blocks into your BricsCAD drawing, you can, in most cases, edit those block instances. You cannot, however, create or edit dynamic block definitions in BricsCAD. Instead of offering dynamic block functionality, BricsCAD focuses entirely on parametrics. It allows you to create both 2D

and 3D block definitions using the same set of tools and workflow. If you've never worked with parametric constraints, they may sound scary. But, they're quite intuitive and logical. And, incredibly powerful!

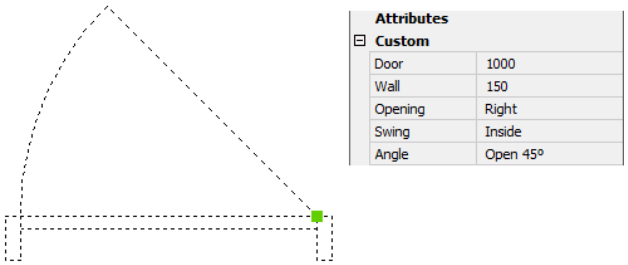
BricsCAD partly supports dynamic blocks, you can:

- insert dynamic blocks from your existing dynamic blocks libraries.
- edit the custom properties in the **Properties** panel.
- grip edit dynamic blocks.

### 12.7.1 Editing dynamic blocks

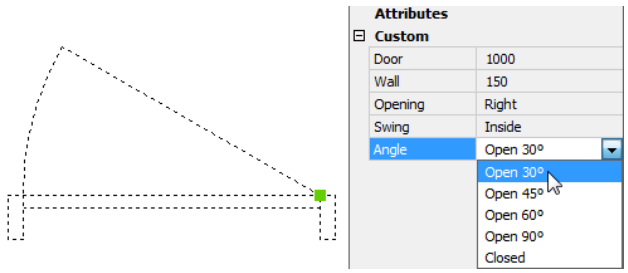
1 Select the dynamic block in the drawing.

1 The properties of the selected block are displayed in the **Properties** panel.




2 Under **Attributes** > **Custom**: edit the properties needed.

The dynamic block is updated automatically.




### 12.7.2 Grip editing dynamic blocks

1 Select the dynamic block in the drawing.

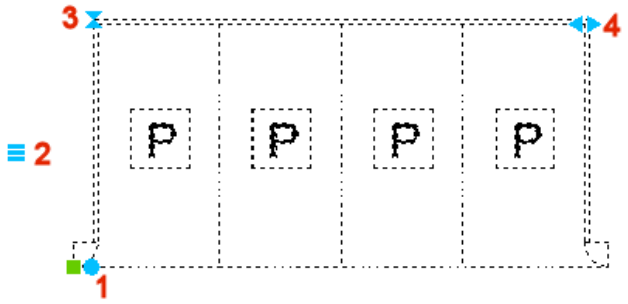
1 The following grip types are displayed:

- Point
- XY
- Linear
- Polar
- Rotation
- Flip
- Visibility

2 Drag a grip to edit the block.

3 In the example below:

- Rotate the block (1): drag the grip to rotate the block dynamically or type a value in the dynamic entry field.
- Show/hide entities (2): click to select an option.
- Flip (3): click to mirror the block
- Edit the length (4): select the grip to dynamically edit the length or type a value in the dynamic entry field.




## 12.8 Определение атрибутов

Команда ATTDEF определяет атрибуты блоков с использованием диалогового окна.

Команда -ATTDEF определяет атрибуты блоков с помощью командной строки.

### 12.8.1 Основные процедуры при создании атрибутов

- Определение параметров атрибута, включая его имя (тэг), приглашение и значение по умолчанию.
- Определение координат вставки атрибута.
- Определение свойств текста атрибута.
- Определение свойств атрибута.

### 12.8.2 Определение атрибута

1 Выполните одно из следующих действий:

- Нажмите кнопку > **Определить атрибуты...** ( ) на панели инструментов **Сервис**.
- Выберите инструмент **Определить атрибуты...** в меню **Сервис > Атрибуты**.
- Запустите команду ATTEDEF.

Откроется диалоговое окно **Определение атрибутов**.

2 Определите атрибут:

- В поле **Тег** введите имя атрибута.

  Тег используется для идентификации атрибутов и их извлечения. Для заполнения поля **Тег** могут использоваться любые комбинации символов. Использование пробелов не допускается. Строчные буквы при назначении тега автоматически преобразуются в заглавные.
- В поле **Приглашение** введите текст приглашения.

Запрос атрибута отображается при каждой вставке блока, содержащего атрибут. Если приглашение не указано, взамен него будет использован Тег атрибута.

- Выполните одно из следующих действий:
- В поле **По умолчанию** введите текстовое значение атрибута по умолчанию.

  Для атрибутов с переменным значением значение по умолчанию заменяется на фактическое значение при последующей вставке блока, содержащего атрибут.
- Нажмите кнопку **Вставить поле** () для присвоения значения поля атрибуту (см. 'Использование свойств подшивки в блоке основной надписи').
- Если выбрана опция **Многострочный**, поле **По умолчанию** будет затенено.
- Нажмите кнопку рядом с полем **По умолчанию**.

  Диалоговое окно **Определение атрибутов** временно закроется и Вам будет предложено создать объект в виде многострочного текста.

- Выполните одно из следующих действий:
- Укажите координаты X, Y и Z точки вставки атрибута.
- В разделе Координаты вставки нажмите кнопку Указать на чертеже ().

  **Запрос программы**: Укажите точку вставки.

  Диалоговое окно **Определение атрибутов** временно закроется.

- При необходимости, в разделе Флаги атрибута установите флаг **Скрытый**.

  Атрибуты с флагом **Скрытый**, не отображаются на экране и не выводятся на печать.

  Если для переменной ATTMODE (Режим отображения атрибутов) установлено значение **2**, будут отображаться все атрибуты, включая те, для которых включен режим **Скрытый**.
- При необходимости, в разделе Флаги атрибута установите флаг **Постоянный**.

  Атрибуты с флагом **Постоянный**, будут недоступны для редактирования.
- При необходимости, в разделе Флаги атрибута установите флаг **Проверяемый**.

  При вставке блока, содержащего атрибуты с флагом **Проверяемый**, выполняется проверка правильности значения атрибута.
- При необходимости, в разделе Флаги атрибута установите флаг **Определенный**.

  При вставке блока, содержащего атрибуты с флагом **Определенный**, программа не выдает запрос на ввод значения атрибута. Вы можете изменить это значение после вставки блока.
- При необходимости, в разделе Флаги атрибута установите флаг **Закрепленный**, чтобы зафиксировать положение атрибута в блоке.

  Незакрепленные атрибуты могут быть перемещены отдельно от блока с помощью узловых точек. Для незакрепленных многострочных атрибутов доступно изменение их размера.
- В разделе **Текст** определите параметры форматирования текста атрибута: **Стиль текста**,

**Выравнивание**, **Аннотативность**, **Высота** и **Угол поворота**.

3 Нажмите кнопку **OK**, чтобы создать атрибут на чертеже.

Диалоговое окно **Определение атрибута** закроется.

### 12.8.3 Использование свойств подшивки в основной надписи

1 Создайте подшивку.

2 Определите пользовательские свойства подшивки и листа.

3 Откройте лист из подшивки.

4 Создайте в пространстве листа исходные объекты блока основной надписи:

- Создайте необходимые рамки, рабочие и постоянные текстовые объекты.
- Создайте атрибуты с полями, которые содержат ссылки на свойства подшивки и листа.

См. описание команды SHEETSET для получения подробной информации о свойствах подшивок и наборов подшивок.

Пользовательские свойства подшивки:

| Пользовательские свойства подшивки | |
|---|---|
| Architect | Bricsys & Partners |
| ArchitectAddress | Bellevue 5/201 |
| ArchitectCity | 9050 Gent |
| ArchitectPhone | 09 244 01 91 |
| Contractor | Robert Stone ltd. |
| ContractorAddress | 123 Main Street |
| ContractorCity | New Town |
| Owner | Carpenter |
| OwnerAddress | 1 Church Road |
| OwnerCity | New Town |
| OwnerTitle | Mr. & Mrs. |
| SiteAddress | 48 Brick Lane |
| SiteCity | New Town |
| SiteReference | Section A no. 123 |

Свойства листа и пользовательские свойства листа:

| Лист | |
|---|---|
| Число | 08 |
| Наименование | Elevation South |
| Описание | South elevation drawing |
| Публикация | ☑ Include for Plot/Publish |
| Текущий лист | LAYOUT1 (C:\BricsCAD Trainir |
| Номер ревизии | 1 |
| Дата ревизии | 15/08/2013 |
| Назначение выпуска | Final Design |
| Категория | Elevation |
| **Пользовательские свойства листа** | |
| Date | 12/08/2013 |
| PaperSize | A2 |
| Printer | pdf |
| Remark | none |
| Scale | 1/50 |

5 Создайте блок из исходных объектов основной надписи.

6 Скопируйте блок в чертеж в одну из папок с вашими блоками.

7 Удалите исходные объекты основной надписи.

Блок основной надписи может быть вставлен в другие листы подшивки и в любую другую подшивку, в которой имеется такой же набор пользовательских свойств подшивки и листа. Все поля, которые содержат ссылки на свойства подшивки или листа, будут заполняться автоматически. Если значение для какого-либо поля не определено, оно будет отображаться в виде группы дефисов (----).

## 12.9 Работа с атрибутами

Атрибуты - это специфические объекты, которые могут быть сохранены как часть определения блока. Атрибут состоит из текстовых данных. Вы можете использовать атрибуты для отслеживания такой информации как учетный номер, цена и др. Атрибуты могут иметь постоянное или переменное значение. При вставке блока, содержащего атрибуты, программа автоматически добавляет в чертеж заданное значение постоянного атрибута или выдает запрос на ввод значения при использовании переменного атрибута.

После вставки блоков, содержащих атрибуты, вы можете извлечь данные атрибутов в отдельный файл и затем использовать эту информацию в электронной таблице или базе данных для создания перечня элементов и материалов. Вы можете также использовать атрибуты для отслеживания числа вставок определенного блока в чертеж.

Атрибуты могут быть видимыми или скрытыми. Скрытые атрибуты не отображаются и не выводятся на печать, но содержащаяся в них информация сохраняется в чертеже и при извлечении атрибутов записывается в файл.

Команда ATTDEF - определяет атрибуты блоков с использованием диалогового окна.

Команда -ATTDEF определяет атрибуты блоков с помощью командной строки.

Команда ATTDISP - включает/выключает отображение текста атрибута.

Команда ATTEDIT - редактирует значения атрибутов и свойства.

Команда -ATTEXT - извлекает данные из атрибутов в текстовый файл с использованием командной строки.

Команда ATTREDEF - переопределяет блок и обновляет связанные атрибуты.

Команда ATTSYNC - синхронизирует определения атрибутов во всех вставках указанного определения блока.

Команда BATTMAN - редактирует все аспекты атрибутов в определении блока, а затем, при необходимости, применяет изменения ко всем блокам с тем же именем на чертеже.

Команда EATTEDIT - редактирует значения и большинство свойств атрибутов.

### 12.9.1 Основные операции при работе с атрибутами

1 Определение атрибутов

2 Создание блока.

Вы можете присоединять атрибуты к блоку. Для этого необходимо указать атрибуты при выборе объектов, включаемых в состав определения блока. Если блок содержит атрибуты, то при каждой вставке этого блока в чертеж программа будет выдавать запрос на изменение значений атрибутов. Если блок содержит атрибуты, то при каждой вставке этого блока в чертеж программа будет выдавать запрос на изменение значений атрибутов.

1 Вставка блока в чертеж.

2 Редактирование атрибутов в блоке.

3 Извлечение данных атрибутов.

## 12.10 Редактирование определений атрибутов

Для редактирования свойств атрибутов вы можете использовать **Панель свойств**.

Команда ATTEDIT редактирует значения атрибутов и свойства.

Команда BATTMAN редактирует все аспекты атрибутов в определении блока, а затем, при необходимости, применяет изменения ко всем блокам с тем же именем на чертеже.

***Прим.:*** Атрибуты в определении блока могут быть изменены на **Панели свойств** при редактировании блока с использованием команды BEDIT.

## 12.11 Редактирование атрибутов блока

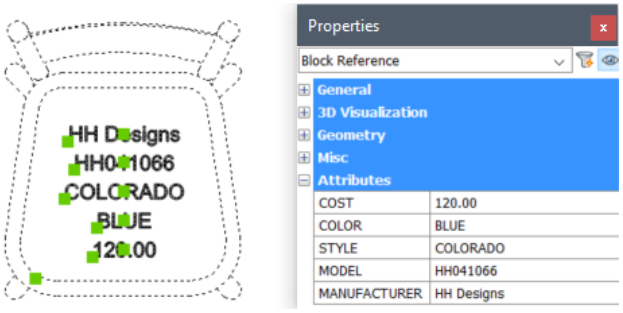
Для редактирования атрибутов блока вы можете использовать **Панель свойств**.

1 Выберите блок, содержащий атрибуты.

1 Текущие значения атрибутов выбранного блока отобразятся на **Панели свойств**.

2 Выберите атрибут, который необходимо изменить.

3 Поле со значением выбранного атрибута станет доступным для редактирования.




4 Введите новое значение и нажмите клавишу Enter или выберите другой атрибут.

## 12.12 Вставка блоков с атрибутами

При вставке блоков, содержащих атрибуты, в зависимости от значения системной переменной ATTDIA программа выдает запрос на определение атрибутов в командной строке или через диалоговое окно:

- Если для системной переменной ATTDIA установлено значение ON, при определении атрибутов используется диалоговое окно.

- Если для системной переменной ATTDIA установлено значение OFF, определение атрибутов выполняется по запросам командной строки.

### 12.12.1 Вставка блока с атрибутами

1 Выполните одно из следующих действий:

- Нажмите кнопку **Вставить блок** () на панели инструментов **Вставка** или панели ленты **Вставка / Блоки**.
- Выберите инструмент **Вставить блок...** в меню **Вставка**.
- Запустите команду INSERT.

Откроется диалоговое окно **Вставка блока**.

2 Выберите блок и нажмите кнопку **ОК**.

**Запрос программы**: Точка вставки для блока или [Масштаб/Масштаб X/Масштаб Y/Масштаб Z/Повернуть/Массив блоков]:

3 Укажите точку.

4 На чертеже будут размещены все объекты блока, за исключением атрибутов.

5 В зависимости от значения системной переменной ATTDIA:

- **Запрос программы**: Приглашение <Значение>:
- Откроется диалоговое окно **Редактор атрибутов**.

6 Выполните одно из следующих действий:

- В командной строке для каждого атрибута:
  - Нажмите клавишу Enter или щелкните правой кнопкой мыши для подтверждения текста по умолчанию.
  - Введите в командной строке новое значение атрибута, нажмите клавишу Enter и выполните определение следующего атрибута.
- В диалоговом окне **Редактор атрибутов**:
  - Выберите атрибут в списке.
  - Текущее значение отображается в поле **Значение**.
  - В поле **Значение** введите новое значение.
  - После завершения определения атрибутов нажмите кнопку **ОК**.

## 12.13 Извлечение данных из атрибутов

Команда ATTEXT копирует данные из атрибутов в текстовый файл с использованием диалогового окна.

Команда -ATTEXT копирует данные из атрибутов в текстовый файл с использованием командной строки.

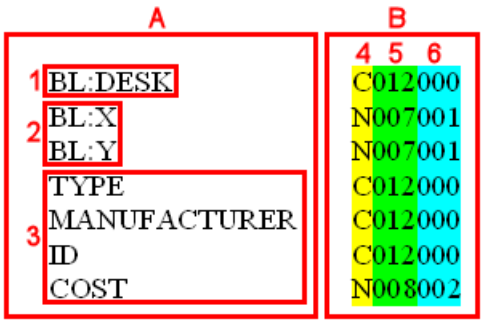
Вы можете выполнить импорт информации из этого текстового файла в базу данных или электронную таблицу.

Данные атрибутов могут быть сохранены в следующих форматах:

- **Comma Delimited Format** (CDF): Содержит одну строку для каждого блока. Поля атрибута разделяются запятыми. Текстовые поля заключаются в одиночные кавычки. При извлечении данных атрибута в формате CDF необходимо указать файл шаблона.
- **Space Delimited Format** (SDF): Содержит одну строку для каждого блока. Каждое поле атрибута имеет фиксированную длину. Разделители полей (в т.ч. и текстовых) не используются. При извлечении данных атрибута в формате SDF необходимо указать файл шаблона.
- **Drawing Exchange Format** (DXF): Создается разновидность стандартного DXF файла (*.dxx), содержащего всю информацию о блоках, включая точки вставки, углы поворота, и значения атрибутов. Файл шаблона не требуется.

Перед извлечением атрибута в файл CDF или SDF необходимо создать файл шаблона. Файл шаблона - это текстовый файл (ASCII), который определяет поля данных атрибута для записи в файл результатов извлечения. Каждая строка в файле шаблона определяет одно поле атрибута. BricsCAD распознает 15 различных полей, которые содержат такие элементы как имя блока; координаты x, y, и z точки вставки блока; слой, на который вставлен блок и другие. Вы можете включить в файл шаблона любое из этих полей. Файл шаблона должен содержать не менее одного имени атрибута.

Каждая строка файла шаблона должна начинаться с имени поля. Имя блока и значения координат точки вставки должны начинаться с **BL:**. Следующим символом должны быть буквы C (индикатор текстового поля)или N (индикатор числового поля). После этого символа следуют три цифры, определяющие длину поля (количество символов, включая запятые и десятичные точки). Последние три цифры указывают на число знаков после запятой (для числовых полей). Для текстовых полей последними тремя цифрами будут нули (000). Ниже приведен пример типового файла шаблона:




A. Имя поля

B. Формат поля

1. Имя блока (должно начинаться с **BL:** )
2. Координаты точки вставки (должны начинаться с **BL:** )
3. Тэги атрибутов
4. Тип данных: **C** для текстовых данных, **N** для числовых данных

5. Ширина поля (количество символов)

6. Количество десятичных знаков для числовых полей, **000** для текстовых полей.

### 12.13.1 Извлечение данных из атрибутов

1 Выполните одно из следующих действий:

- Нажмите кнопку **Извлечь атрибуты** на панели **Сервис**.
- Выберите пункт **Извлечь атрибуты** в меню **Сервис**.
- Запустите команду ATTEXT.

Откроется диалоговое окно **Извлечь атрибуты**.

2 Нажмите кнопку **Выбрать**.

Диалоговое окно **Извлечь атрибуты** временно закроется.

3 Выберите блоки, затем нажмите клавишу Enter или щелкните правой кнопкой мыши.

4 Диалоговое окно **Извлечь атрибуты** снова откроется.

5 Выберите формат текстового файла: **DXF** (drawing exchange format), **CDF** (comma delimited format) или **SDF** (space delimited format).

6 Выберите **Файл шаблона**.

7 В поле **Целевой файл** укажите имя файла для извлеченных данных.

8 Нажмите кнопку **Извлечь**.

В командной строке отобразится информация о количестве извлеченных записей.

## 12.14 Управление внешними ссылками

Команда EXPXREFS открывает раздел **Внешние ссылки** диалогового окна **Проводник по чертежам**.

Команда XREF открывает панель **Ссылки**.

Диалоговое окно **Проводник по чертежам > Внешние ссылки** позволяет:

- Подключать чертеж как внешнюю ссылку ( ).
- Отключать внешнюю ссылку ( ).
- Перезагружать внешнюю ссылку ( ).
- Выгружать внешнюю ссылку ( ).
- Связывать внешнюю ссылку ( ).
- Вставлять внешнюю ссылку ( ).
- Изменять тип подключения: **Attach (Присоединение)** или **Overlay (Наложение)**.
- Изменять **Путь** внешней ссылки.
- Изменять **Найденный путь** внешней ссылки.

### 12.14.1 Отключение внешней ссылки

1 Выберите внешние ссылки, которые необходимо отключить.

1 Нажмите и удерживайте клавишу CTRL, чтобы выбрать несколько внешних ссылок или отменить выбор внешней ссылки.

2 Нажмите кнопку **Отключить внешнюю ссылку** (✕) на панели инструментов раздела **Внешние ссылки**.

Выбранные внешние ссылки будут отключены. При этом запрос на подтверждение отключения не отображается.

### 12.14.2 Выгрузка внешней ссылки

Выполните одно из следующих действий:

- Щелкните на поле **Загрузить** для выгруженной внешней ссылки.
- Выберите внешние ссылки и нажмите кнопку **Выгрузить внешнюю ссылку** () на панели инструментов раздела **Внешние ссылки** или щелкните на поле **Загрузить** одной из выбранных внешних ссылок.

### 12.14.3 Перезагрузка внешней ссылки

Выполните одно из следующих действий:

- Щелкните на поле **Загрузить** для выгруженной внешней ссылки.
- Выберите внешние ссылки и нажмите кнопку **Перегрузить** () на панели инструментов раздела **Внешние ссылки** или щелкните на поле **Загрузить** одной из выбранных внешних ссылок.

## 12.15 Подключение внешней ссылки

Команда XATTACH подключает чертеж как внешнюю ссылку (xref). Подключенная ссылка отображается в текущем чертеже как вставка. Как и блоки, подключенные внешние ссылки могут быть вложенными. Подключенный чертеж отображается в виде последней сохраненной версии.

### 12.15.1 Чтобы подключить внешнюю ссылку

1 Выполните одно из следующих действий:

- Нажмите кнопку **Вставить внешнюю ссылку** () на панели инструментов **Вставка** или панели ленты **Вставка / Внешние ссылки**.
- Выберите пункт **Вставить внешнюю ссылку** в меню **Вставка**.
- Нажмите кнопку **Вставить внешнюю ссылку** () на панели инструментов раздела **Внешние ссылки** диалогового окна **Проводник по чертежам - Внешние ссылки**.
- Нажмите кнопку **Вставить DWG...** () на панели **Ссылки**.
- Запустите команду XATTCH.

Откроется диалоговое окно **выбора файла ссылки**.

2 Выберите файл ссылки и нажмите кнопку **Открыть**.

Откроется диалоговое окно **Подключить внешнюю ссылку**.

3 В поле **Путь** отобразится полный путь к выбранному чертежу.

При необходимости выберите **Тип выводимого пути**: **Полный**, **Относительный** или **Без пути**.

4 Выберите **тип внешней ссылки**:

- **Подключение**: любые внешние ссылки в чертеже, с которым устанавливается связь, также будут видны и становятся вложенными ссылками.
- **Наложение**: внешние ссылки в чертеже, с которым устанавливается связь, игнорируются и в текущем чертеже видны не будут.

5 Выберите способ указания **точки вставки** - на экране или в полях **X**, **Y**, **Z**.

6 Выберите способ указания **масштаба** - на экране или в полях **X**, **Y**, **Z**.

Включите опцию **Один масштаб по всем осям**, чтобы указать масштаб в поле **X**.

7 Позволяет указать **угол поворота** на экране или в поле **Угол**.

8 Нажмите кнопку **ОК** для подключения внешней ссылки.

В зависимости от выбранных в п. 5-7 опций вы получите запросы на указание точки вставки, масштаба и/или угла поворота.

**Примечание**:

- Системная переменная XDWGFADECTL (Управление затемнением внешних ссылок) устанавливает уровень затемнения для внешних ссылок.
- Базовая точка, используемая при подключении внешней ссылки, определяется значением переменной INSBASE (Базовая точка). Используя команду BASE Вы можете установить значение переменной INSBASE в чертеже внешней ссылки.
- Значение **Единиц измерения** в разделе **Единицы измерения блока** определяется значением переменной INSUNIT (Единицы измерения при вставке), установленном для внешней ссылки. Значение **Коэффициента** вычисляется как отношение значений переменной INSUNIT для внешней ссылки и исходного чертежа.

## 12.16 Working with external references

You can link entire drawings to the current drawing as external references. Unlike inserting a drawing as a block, in which you add all the entities from the separate drawing into the current drawing, external references attach a pointer to the external file. The entities in the external reference appear in the current drawing, but the entities themselves are not added to the drawing. Thus, attaching an external reference does not significantly increase the size of the current drawing file.

External references provide additional capabilities not available when you insert a drawing as a block. When you insert a drawing as a block, the entities are stored in the drawing. Any changes you make to the original drawing are not reflected in the drawing in which you inserted it. When you attach an external reference, however, any changes you make to the original drawing file are reflected in the drawings that reference it. These changes appear automatically each time you open the drawing containing the external

reference. If you know that the original drawing was modified, you can reload the external reference anytime you're working on the drawing.

External references are useful for assembling master drawings from component drawings. Use external references to coordinate your work with others in a group. External references help reduce drawing file size and ensure that you are always working with the most recent version of a drawing.

The XREF command (short for "external reference") attaches DWG files to the current drawing through the **Drawing Explorer**.

The -XREF command attaches DWG files to the current drawing through the Command line.

The XATTACH command (short for "external attach") attaches externally-referenced drawings through a dialog box.

The XCLIP command (short for "Xref clip") clips externally-referenced drawings, and adjusts front and rear clipping planes.

The XDWGFADECTL system variable controls the fading of external references. Values between 0 (no fading) and 90 are accepted. The fading factor affects the screen display only, not the printing.

The XNOTIFYTIME system variable defines the frequency to check for modified externally referenced resources: Xrefs, images and pdf attachments if XREFNOTIFY, IMAGENOTIFY and/or PDFNOTIFY is **On**.

The XREFNOTIFY system variable controls whether warnings for missing or modified Xrefs are issued.

## 12.17 Редактирование блоков и внешних ссылок

Команда REFEDIT позволяет редактировать содержимое внешних ссылок и блоков с использованием диалогового окна. Редактирование внешних ссылок и блоков выполняется непосредственно в том чертеже, в который они были вставлены (**in-place** редактирование). Команда REFEDIT работает как с фрагментом редактируемой внешней ссылки, так и со всем чертежом в целом и всеми объектами в блоке.

Команда -REFEDIT редактирует внешние ссылки и блоки с помощью командной строки.

Команда REFSET добавляет и удаляет объекты из редактируемой ссылки.

Команда REFCLOSE закрывает редактор ссылки.

Команда BEDIT редактирует объекты блоков во временном сеансе черчения, который открывается с помощью **Редактора блоков** и использует локальную систему координат блока.

Команда -BEDIT запрашивает имя блока в командной строке и открывает **Редактор блока**.

Системная переменная XFADECTL устанавливает уровень затемнения при отображении объектов, не используемых в текущей сессии редактирования внешней ссылки. Диапазон значений от 0 (без затемнения) до 90.

### 12.17.1 Редактирование внешних ссылок

1 Выполните одно из следующих действий:

- Дважды щелкните на ссылке.
- Нажмите кнопку **Редактор ссылок** () на панели **Редактор внешних ссылок**.
- Запустите команду REFEDIT.

**Запрос программы**: Укажите ссылку:

Щелкните на ссылке или блоке.

Откроется диалоговое окно **Изменение ссылки**.

Это диалоговое окно позволяет выбрать и визуально отобразить редактируемую ссылку.

- **Путь**: Расположение файла выбранной ссылки; если ссылка является блоком, путь не отображается.
- **Имя ссылки**: Имя текущей выбранной ссылки и всех вложенных в нее ссылок.
- **Предварительный просмотр**: отображает ссылку в том виде, как она была сохранена в чертеже. Необходимо помнить, что при сохранении изменений в ссылке обновление изображения предварительного просмотра не происходит.

2 (опция) Если в поле **Имя ссылки** отображаются несколько вложенных ссылок, выберите ссылку для изменения.

Нажмите кнопку Развернуть/Свернуть, чтобы показать/скрыть вложенные ссылки.

Одновременно может редактироваться только одна ссылка.

3 Выберите одну из опций раздела **Указать вложенные объекты...**:

- **Автоматически**: Все объекты и вложенные блоки выбранной ссылки доступны в текущей сессии редактирования ссылки.
- **Приглашение**: после закрытия диалогового окна **Изменение ссылки** в командной строке отображается запрос на выбор объектов и вложенных блоков внешней ссылки, предназначенных для редактирования. Только выбранные объекты будут временно извлечены и доступны для изменения в текущем чертеже.

Этот временный набор объектов может быть изменен и сохранен для обновления редактируемой ссылки. Чтобы отделить 'текущий чертеж' от 'редактируемого набора' BricsCAD отображает все объекты текущего чертежа с уменьшенной яркостью. Уровень яркости нередактируемых объектов при этом зависит от значения переменной XFADECTL, которое по умолчанию равно **50**.

4 Выберите необходимые опции в поле **Параметры**:

- **Использовать уникальные слои и имена**:
  - Если эта опция включена, все имена объектов внешней ссылки будут изменены добавлением префикса **$#$**.
  - Если эта опция отключена, имена слоев и других именованных объектов внешней ссылки сохраняются без изменений.
- **Отображать определения атрибутов для редактирования**: Управляет извлечением и отображением атрибутов блоков при редактировании.
- **Блокировать объекты, не входящие в рабочий набор**: при включении этой опции объекты редактируемой внешней ссылки, не включенные во временный рабочий набор, изменяться не будут. Опция аналогична использованию процедуры блокирования слоев.

5 Нажмите кнопку **ОК** для начала редактирования рабочего набора.

Откроется диалоговое окно **Изменение ссылки**.

6 При необходимости, нажмите кнопку **Добавить в редактор ссылок** (), чтобы добавить в рабочий набор новые объекты.

Выбранные объекты будут удалены из текущего чертежа и добавлены в редактируемую ссылку при сохранении рабочего набора.

7 При необходимости, нажмите кнопку **Удалить из редактора ссылок** () для удаления объектов из рабочего набора.

Выбранные объекты будут удалены из редактируемой ссылки при сохранении рабочего набора. Эти объекты добавляются на основной чертеж.

8 Выполните одно из следующих действий:

- Чтобы сохранить внесенные изменения в исходном файле ссылки или определении блока в текущем чертеже, нажмите кнопку **Сохранить и закрыть редактор ссылок** ().
- Чтобы завершить редактирование ссылки без сохранения внесенных изменений, нажмите кнопку **Закрыть редактор ссылок без сохранения** (). Исходный чертеж или определение блока останется без изменений.

## 12.18 Работа с группами

Команда GROUP создает и редактирует именованные группы объектов с использованием диалогового окна.

Команда -GROUP создает и редактирует именованные группы объектов с помощью командной строки.

Объединяя отдельные объекты в группу Вы можете управлять этим набором как цельным объектом. Группу можно рассматривать как именованный набор объектов. Как и для блока, вы можете выбрать весь набор объектов, указав только один объект, входящий в этот набор. Вы также можете редактировать отдельные объекты, входящие в группу, не разбивая ее и не изменяя другие элементы группы. При необходимости вы можете создавать вложенные группы. Один и тот же объект может входить в состав различных групп.

***Прим.:*** Убедитесь, что для переменной PICKSTYLE установлено значение 1 или 3, разрешающее выбор групп.

### 12.18.1 Создание группы

1 Выполните одно из следующих действий:

- Нажмите кнопку **Группа** () на панели **Сервис** или панели ленты **Главная / Группы**.
- Выберите инструмент **Группа** в меню **Сервис**.
- Запустите команду GROUP.

Откроется диалоговое окно **Группы**:

2 Выполните одно из следующих действий:
   - В поле **Имя** введите имя группы.
   - Включите опцию **Безымянная** для создания безымянной группы.
3 При необходимости, в поле **Описание** введите текст описания группы.
4 При необходимости включите/выключите опцию **Выбираемая**.
5 Нажмите кнопку **Выбрать объекты и создать группу**.

   Диалоговое окно **Группы** временно закроется для выбора объектов.
6 Выберите объекты, которые вы хотите включить в состав новой группы.
7 Щелкните правой кнопкой мыши для завершения выбора объектов.
8 Снова откроется диалоговое окно **Группы**.

   Будет добавлена созданная группа.
9 Нажмите кнопку **ОК**. Окно **Группы** закроется.

### 12.18.2 Редактирование группы

1 Выполните одно из следующих действий:
   - Нажмите кнопку **Группа** ( ) на панели инструментов **Сервис** или панели ленты **Главная / Группы**.
   - Выберите инструмент **Группа** в меню **Сервис**.
   - Запустите команду GROUP.

   Откроется диалоговое окно **Группы**.
2 Выберите группу, которую необходимо изменить.
3 При необходимости щелкните дважды на поле **Имя**, чтобы изменить имя группы.
4 При необходимости щелкните дважды на поле **Описание**, чтобы изменить описание группы.
5 При необходимости щелкните на поле **Выбираемая**, чтобы включить/выключить опцию **Выбираемая**.
6 При необходимости, нажмите кнопку **Добавить объекты в группу**, чтобы добавить в группу новые объекты.

   Диалоговое окно **Группы** временно закроется для выбора объектов.
7 При необходимости нажмите кнопку **Удалить объекты из группы**, чтобы удалить объекты из группы.

   Диалоговое окно **Группы** временно закроется для выбора объектов, удаляемых из группы.
8 Нажмите кнопку **ОК**. Окно **Группы** закроется.

### 12.18.3 Разгруппировка объектов

1 Выполните одно из следующих действий:
   - Нажмите кнопку **Группа** ( ) на панели инструментов **Сервис** или панели ленты **Главная / Группы**.

- Выберите инструмент **Группа** в меню **Сервис**.
- Запустите команду GROUP.

Откроется диалоговое окно **Группы**.

2 Выберите группу или группы, для удаления.

3 Нажмите кнопку **Разбить указанные группы**.

Выбранные группы будут разбиты и удалены из списка групп.

***Прим.:*** После того, как группа будет разбита, входившие в нее объекты сохраняются в чертеже, а группа удаляется.

### 12.18.4 Изменение порядка объектов группы

1 Выполните одно из следующих действий:

- Нажмите кнопку **Группа** () на панели инструментов **Сервис** или панели ленты **Главная / Группы**.
- Выберите инструмент **Группа** в меню **Сервис**.
- Запустите команду GROUP.

Откроется диалоговое окно **Группы**.

2 Нажмите кнопку **Упорядочить объекты**.

Откроется диалоговое окно **Упорядочить группу**:

3 При необходимости нажмите кнопку **Выделить** для просмотра текущего положения каждого объекта группы.

Откроется окно **Группы**.

4 При необходимости нажмите кнопку **Обратный порядок** для изменения порядка всех объектов группы.

5 Чтобы изменить порядок для одного из объектов группы, выполните следующие действия:

- В поле **Удалить с позиции** введите текущую позицию объекта, который необходимо переместить.
- В поле **Новый номер положения объекта** введите номер для нового положения объекта в группе.

6 При необходимости в поле **Число объектов** введите количество объектов, для которых необходимо изменить порядок.

7 Нажмите кнопку **Упорядочить**.

Откроется окно с сообщением, что группа была переупорядочена.

## 12.19 Работа с подосновами

PDF файл может быть присоединен к чертежу в виде подосновы. Так же, как файлы растровых изображений и внешних ссылок (Xrefs) подоснова в формате PDF не является частью чертежа. В чертеже сохраняется путь к файлу подосновы. Вы можете изменить этот путь, чтобы обеспечить

правильность ссылки на необходимый PDF файл. По умолчанию местом поиска файла подосновы является папка текущего чертежа. Если файл подосновы не был найден в папке чертежа, будет выполнен поиск по сохраненному пути. Если по сохраненному пути файл также не будет найден, BricsCAD выведет на чертеж сообщение 'Missing or invalid reference'.

Команда PDFADJUST позволяет выполнять настройку Затухания, Контрастности и Монохромности подосновы PDF с помощью командной строки.

Команды PDFATTACH и -PDFATTACH выполняют вставку PDF файла в текущий чертеж в качестве подосновы.

Если PDF подоснова содержит слои, вы можете управлять отображением этих слоев с помощью команды PDFLAYERS.

Команда PDFCLIP выполняет обрезку PDF подосновы. Вы можете выбрать многоугольный или прямоугольный контур обрезки. Каждый экземпляр одной и той же PDF подосновы может иметь различный контур обрезки.

Если для системной переменной PDFOSNAP установлено значение **On**, вы можете выполнять привязку к геометрии подосновы PDF.

Системная переменная PDFNOTIFY управляет выводом сообщений об отсутствии или изменении файлов подоснов PDF.

Системная переменная XNOTIFYTIME определяет частоту проверки наличия изменений внешних ресурсов: внешних ссылок, изображений и pdf файлов, если для системных переменных XREFNOTIFY, IMAGENOTIFY и/или PDFNOTIFY установлено значение **ON**.

Если значение системной переменной FRAME = 3, системная переменная PDFFRAME управляет видимостью фреймов PDF. Доступные опции:

- 0: Не отображать контуры PDF.
- 1: Отображать и печатать контуры PDF.
- 2: Отображать, но не печатать контуры PDF.

### 12.19.1 Управление кешированием подосновы PDF

При отображении прикрепленных подоснов PDF применяется постоянный кеш изображения с переменным разрешением, который позволяет значительно повысить скорость выполнения операций масштабирования и панорамирования. Максимальное разрешение для кеширования - 5000 x 5000 пикселей, поэтому при большом приближении изображение будет пикселизироваться. Поэтому имеется возможность перехода в гибридный режим, при котором регенерация изображения подосновы выполняется в реальном времени. Генерация кеша изображения может занять несколько секунд и после этого всегда будет обеспечиваться высокая производительность.

Для управления кешированием подоснов PDF используется системная переменная PDFCACHE:

- 0 = не применять кеширование PDF, всегда использовать генерацию изображения подосновы PDF в реальном времени. (Этот режим использовался по умолчанию в предыдущих версиях BricsCAD).

- 1 = применять кеширование PDF с переходом на генерацию изображения в реальном времени при разрешении, превышающем разрешение кеширования.
- 2 = всегда применять кеширование PDF (новое значение по умолчанию).

### 12.19.2 Подключение подосновы PDF

1 Выполните одно из следующих действий:

- Нажмите кнопку **Вставить PDF** () на панели инструментов **Вставка** или панели ленты **Вставка / PDF**.
- Нажмите кнопку **Создать** () на панели инструментов раздела **Подосновы PDF** окна **Проводник по чертежам - Подосновы PDF**.
- Введите в командной строке **pdfattach** и нажмите клавишу Enter.

Откроется диалоговое окно **Выбор файла подосновы**.

2 Выберите файл, дважды щелкните на нем или нажмите кнопку **Открыть** в окне **Выбор файла подосновы**.

Откроется диалоговое окно **Подключение подосновы PDF**.

3 При подключении многостраничного PDF файла нажмите на кнопку со стрелкой у правого края поля **Имя** и выберите необходимую для загрузки страницу PDF файла.

4 Выберите одну из опций поля **Путь**:

- **Полный путь**: При загрузке чертежа поиск файлов подосновы выполняется по полному пути к файлу подосновы. Если файл подосновы не будет найден по указанному пути, будет выполнен поиск в папке текущего чертежа. Если по сохраненному пути файл также не будет найден, в точке вставки PDF подосновы отобразится сообщение 'Missing or invalid reference'.
- **Относительный путь**: при загрузке чертежа поиск файлов подосновы выполняется по относительному пути (относительно папки текущего чертежа). Если файл подосновы не будет найден по указанному пути, будет выполнен поиск в папке текущего чертежа. Если по сохраненному пути файл также не будет найден, в точке вставки PDF подосновы отобразится сообщение 'Missing or invalid reference'.
- **Без путей**: при загрузке чертежа поиск файла подосновы выполняется только в папке текущего чертежа. Если по сохраненному пути файл также не будет найден, в точке вставки PDF подосновы отобразится сообщение 'Missing or invalid reference'.

5 Определите способ выбора **Точки вставки**, **Масштаба** и **Угла поворота**. Для этого включите/выключите опцию Указать на экране.

6 Нажмите кнопку **OK**.

Диалоговое окно **Вставка PDF** закроется.

7 В соответствии с выбранной в предыдущем пункте опцией определите **Точку вставки**, **Масштаб** и/или **Угол поворота**.

### 12.19.3 Вставка подосновы PDF

Если Вам необходимо выполнить повторную вставку PDF файла, который уже был подключен к чертежу, или другой страницы того же файла, выполните следующие действия:

1 Запустите команду EXPPDFS.

1 Откроется диалоговое окно **Проводник по чертежам - Подосновы PDF**.

2 Выберите подоснову PDF.

3 Выполните одно из следующих действий:

- Щелкните правой кнопкой на выбранной PDF подоснове и в открывшемся контекстном меню выберите **Вставить**.
- Выберите пункт **Вставить** в меню **Правка** окна **Проводник по чертежам - Подосновы PDF**.

Окно **Проводник по чертежам - Подосновы PDF** закроется.

4 Выполните действия п.3-7 предыдущей процедуры.

### 12.19.4 Управление отображением подосновы PDF

1 Щелкните на рамке подосновы PDF.

1 Рамка подосновы PDF будет выделена.

Свойства подосновы отображаются на **Панели свойств**.

2 В разделе **Дополнительно** на **Панели свойств** щелкните строку **Отображать подоснову** и выберите в выпадающем списке опцию **Да** или **Нет**.

### 12.19.5 Настройка отображения слоев подосновы PDF

1 Запустите команду PDFLAYERS.

1 **Запрос программы**: Укажите подоснову PDF:

2 Щелкните на рамке подосновы PDF.

3 Рамка подосновы PDF будет выделена.

Откроется диалоговое окно **Слои подосновы**.

4 Значок в колонке **Включить** указывает на текущее состояние слоя - **Включен** () или **Выключен** ().

5 Щелкните на значке в колонке **Включить** для изменения текущего режима отображения слоя.

6 При необходимости, щелкните на поле **Поиск слоя** и введите имя слоя, который необходимо найти.

Для ограничения количества слоев в списке используйте подстановочные символы (? или *).

7 При необходимости, нажмите кнопку со стрелкой у правого края поля **Имя ссылки** для выбора другой подосновы PDF.

8 Нажмите кнопку **ОК** для завершения.

### 12.19.6 Обрезка подосновы PDF

1 Запустите команду PDFCLIP.

1 **Запрос программы**: Укажите подоснову PDF:

2 Щелкните на рамке подосновы PDF.

3 Рамка подосновы PDF будет выделена.

**Запрос программы**: Укажите параметры обрезки PDF [Включить/Выключить/Удалить/Создать] <Создать>:

4 Нажмите клавишу Enter для выбора опции **New** как опции по умолчанию.

**Запрос программы**: Укажите используемый тип контура обрезки [Многоугольный/Прямоугольный] <Прямоугольный>:

5 Выполните одно из следующих действий:

- Нажмите клавишу Enter, чтобы выбрать опцию **Прямоугольный**, предлагаемую в запросе командной строки по умолчанию для определения прямоугольного контура обрезки.
- Введите в командной строке P и нажмите клавишу Enter для выбора многоугольного контура обрезки.

6 Определите контур обрезки.

7 Точки, определяющие контур, должны располагаться внутри рамки подосновы PDF подосновы. При указании точки за пределами подосновы она будет автоматически перемещена на рамку подосновы.

### 12.19.7 Отображение отсеченной части подосновы PDF

1 Щелкните на рамке подосновы PDF.

1 Рамка подосновы PDF будет выделена.

Свойства подосновы отображаются на **Панели свойств**.

2 В разделе **Дополнительно** на **Панели свойств** щелкните строку **Отсекание** и выберите в выпадающем списке опцию **Да** или **Нет**.

### 12.19.8 Удаление контура обрезки

1 Запустите команду PDFCLIP.

1 **Запрос программы**: Укажите подоснову PDF:

2 Щелкните на рамке подосновы PDF.

3 Рамка подосновы PDF будет выделена.

**Запрос программы**: Укажите параметры обрезки PDF [Включить/Выключить/Удалить/Создать] <Создать>:

4 Введите в командной строке D и нажмите клавишу Enter.

5 Контур обрезки будет удален и подоснова PDF отобразится в исходном виде без обрезки.

### 12.19.9 Настройка параметров отображения подосновы PDF

Команда PDFADJUST позволяет выполнять настройку **Затухания**, **Контрастности** и **Монохромности** подосновы PDF с помощью командной строки.

Вы также может выбрать подоснову PDF и выполнить настройку параметров ее отображения в разделе **Настройки подосновы** на **Панели свойств**.

## 12.20 Components and component inserts

A component is a named group of entities, that can be inserted in a drawing with the BmInsert command. Components can be parametric: e.g. windows and doors, in BricsCAD BIM and standard mechanical components in BricsCAD Mechanical.

Any .dwg file can be used as a component. The same component can be inserted once or several times into one or more other components. When the component is modified, all inserts of the component will reflect these changes automatically. Components cannot self-reference.

A component insert is a named entity. The default name of a component insert is composed of the name of the component and the serial number of the insert. Internally a component insert is represented either as a reference to an anonymous block or as an external reference.

Properties of inserted components, including their parameters, can be edited in the **Mechanical Browser** panel and **Properties** panel.

### 12.20.1 External and local components

Components can be either local or external. A local component is stored in the .dwg file with the assembly as an anonymous block. An external component is defined in a separate .dwg file and represented by an Xref or as a local copy stored as an anonymous block. When the corresponding .dwg file is modified all inserts of external components can be updated using the BMUPDATE command or the **Mechanical Browser** context menu. If you use external components in your assembly, always remember to transfer all related files when you want to share your model.

Use the ETRANSMIT command to create a package of a drawing file and all its dependencies, such as external references, images, font files, plot configuration files, plot style tables and font map files.

You can always convert a local component to an external one and vice versa.

To edit a local component, it first needs to be converted to an external component using the BMEXTERNALIZE command. Alternatively, you can use the BMOPENCOPY command to open a copy of a component, edit and save it. Then use the BMREPLACE command to replace the original with the copy.

You can choose the default insertion type for your component: **Local** or **External**. To change the type, select the root node in the **Mechanical Browser** and select **Local** or **External** from **Insert as properties**.

Depending on the value of BMAUTOUPDATE, inserts of external components are updated automatically on opening the assembly document or manually with the BMUPDATE command. You can change this value in the **Settings** dialog box. The options are:

- Update only when the BMUPDATE command is used.
- Update automatically upon opening the file.

### 12.20.2 Converting components

The BMEXTERNALIZE command converts local components to external components.

The BMLOCALIZE command converts external components to local components.

Do one of the following:

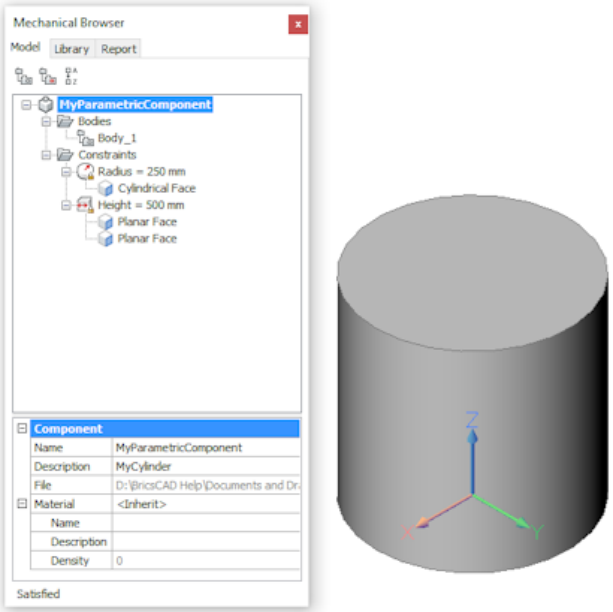
- Click **Switch Component to External** ( ) or **Switch Component to Local** on the **Assembly** ribbon tab. The tools are also available from the **Assembly** toolbar and the **Assembly** menu.
- Right-click on the component insert in the **Mechanical Browser** and select **Switch to External** or **Switch to Local** in the context menu.
- Hover your mouse cursor over a component insert in a drawing window and select **Switch to External** ( ) or **Switch to Local** in the **Assembly Modeling** command group in the Quad.

When you convert an external component to a local one, the .dwg file with the component definition is not deleted, but future changes will not affect your assembly.

You can always distinguish between inserts of local components ( ) and inserts of external components ( ) by their icons in the **Mechanical Browser**.

### 12.20.3 Parametric Components

You can make parametric changes of inserted components at the assembly level. A component is considered parametric if it contains at least one 2D or 3D dimensional constraint or an associative array with an expression.



***Прим.:***

- BricsCAD automatically maintains associativity between the definition of a parametric component and its inserts. So, you can open the parametric component for editing, modify its geometry and save it. To see the result, execute the BmUpdate command to update the mechanical structure of your assembly.
- If you need to go back to the default value of a component's parameter, just edit the **Expression** field and delete its content.

You can control if a particular component parameter will be available for editing in a drawing that contains the inserts of these components (at the assembly level). To do that, select a parameter in the **Mechanical Browser** and change the value of the **Exposed** field in the properties grid of the parameter.

The options are:

- **Off**: Hides the parameters at the assembly.
- **On**: Makes the parameter visible at the assembly level.
- **Auto**: Hides all parameters with values controlled by expressions and exposes all other parameters.

You can associate units with a parameter. If you assign a value, directly or via an expression, to a component parameter at the assembly level, it automatically converts from drawing units of the assembly drawing to the units of the parameter. Values assigned to unitless parameters are not converted.

By default, all parameters are unitless except parameters of dimensional constraints and parameters, which are used as expressions for dimensional constraints. BricsCAD automatically assigns drawing units to such parameters.

### 12.20.4 To edit the parameters of a component insert

When a parametric component is inserted in a model, you can edit the parameters either in the **Mechanical Browser** panel or in the **Properties** panel.

The component insert immediately updates according to the new value of a parameter. Please note that this does not modify the definition of the parametric component, which is a separate drawing file. As a result, multiple inserts of the same parametric component can exist in the same model, with different values for each of their parameters:

**To edit a parameter in the Mechanical Browser**

1 Expand the **Parameters** node of the component.

2 Select the parameter.

3 In the **Parameter** section, type a new value in the **Expression** field.

**To edit a parameter in the Properties bar**

1 Select the component in the drawing.

2 Expand the **Parameters** settings group.

3 Type a new value for one of the available parameters.

***Прим.:***

- In the **Properties** panel, the current value is displayed. If this value is controlled through an expression, the expression will be overwritten by the newly entered value.
- If multiple parametric components are selected, the shared parameters can be edited simultaneously. The settings fields of parameters that currently have different values reads: *varies*.

### 12.20.5 To replace components

The BMREPLACE command replaces component inserts. Only top-level inserts can be replaced.

***Прим.:*** Parameter values that are shared between the replacement component and the replaced component are copied automatically.

### 12.20.6 Components, blocks or external references?

Components are based on blocks and external references (Xrefs) but the behavior of components is different. An external component is linked to a file, much like an Xref, but an external component can be represented by either an Xref (BMINSERT acts like XATTACH) or by a local anonymous block (BMINSERT acts like INSERT).

Components can be either parts or assemblies:

- Part: composed of entities only; no inserts of subcomponents.
- Assembly: composed of entities and subcomponent inserts. Even when the subcomponents are removed, the assembly status is kept.

When an external component is a part and its parameters are not modified, the component is inserted as an Xref. This is listed in the **External References** section of the **Drawing Explorer** dialog. Xref layers have a 'file_name' prefix.

When an external component is an assembly, it is inserted as a local anonymous block. All of its layers are merged with the layers of the main document and no prefix is added to the layer name.

### 12.20.7 Converting blocks and external references

The BMMECH command converts the current drawing into a mechanical component. If the drawing already is a mechanical component BMMECH does nothing.

Block references are converted to inserts of local components and external references are converted to inserts of external components.

The BMUNMECH command converts the current mechanical component into a plain drawing. Mechanical components are converted back to blocks and external references. The command applies to drawings that are a mechanical component only.

### 12.20.8 Component-based features

A feature is a physical constituent of a part with engineering significance. Examples of features are holes, ribs, slots, pockets, forms. If applied to a 3D solid, a feature removes and/or adds geometry. To create your own features, use the BricsCAD component extension technology.

An extended component has BC_SUBTRACT and/or BC_UNITE layers. When you use the BMINSERT command to insert such a component on the face of a 3D solid, the target solid is modified as follows:

- All 3D solids on the BC_SUBTRACT layer of the component are subtracted from the target 3D solid.
- All 3D solids on the BC_UNITE layer of the component are united with the target 3D solid.

All faces created after Boolean operations form a component-based feature. They are updated when you move the component insert or when you modify any parameters. When the insert is deleted, the target solid is healed. Adjust the target 3D solids with the BMLINK command.

Some examples of parametric component-based features are included with BricsCAD. You can use them as samples to create your own features.

### 12.20.9 Assign physical materials to components

You can assign a physical material to a mechanical component. If a mechanical component does not have an assigned material, it will inherit the material from its parental component. If the parental component does not have an assigned material, the material of the nearest component, will be used. As a result, the same material can be assigned to all components that have a common parent.

1 Select the root component in the **Mechanical Browser**.

2 Select the **Material** property and press the **Browse** button (...).

3 In the **Physical Materials** dialog box, select a material from either the **In Project** or the **In Library** list.

Materials that are chosen from the **In Library** are copied to the **In Project** database automatically.

4 Click the **OK** button to assign the selected material to the component.

**To remove the material from the component**:

1 Select the root component in the **Mechanical Browser**.

2 Select the **Material** property and press the **Clear** button (X).

**Inserting a component with a physical material**:

When you insert a mechanical component in a model, all its materials are copied to the material library of the model. If a material with the same name already exists in the target model, this material will be used instead. As a result, the properties of the material in the target model are used. This replaces the properties of the material from the inserted component.

***Прим.:*** Materials that are copied to the document material library will not be erased in **Undo** operations.

# 13. Управление изображениями

## 13.1 Managing images

The EXPIMAGES command displays the Drawing Explorer – Images dialog box.

The IMAGE command opens the Attachments panel.

The IMAGEADJUST command adjusts the properties of images through the Properties panel.

The IMAGEATTACH command attaches raster images to the drawing.

The IMAGECLIP command clips images.

The IMAGEFRAME command toggles the display of the frame around images; sets the value of the IMAGEFRAME system variable.

The IMAGEQUALITY command toggles the display quality of images between draft and high.

The IMAGENOTIFY system variable controls whether warnings for missing or modified image files are issued.

The XREFNOTIFYTIME system variable defines the frequency to check for modified externally referenced resources: Xrefs, images and pdf attachments if XREFNOTIFY, IMAGENOTIFY and/or PDFNOTIFY is ON.

You can manage images in the **Drawing Explorer - Images** dialog box and in the **Attachments** panel.

Here you can:

- Attach an image () or ().
- Load / Unload inserted images.
- Detach images ().
- Edit the path to the image.
- See a preview of the attached images (**Drawing Explorer** dialog box only).

To open the **Drawing Explorer - Images** dialog box, do one of the following:

- Choose **Drawing Explorer** > **Images** in the **Tools** menu.
- Select **Images** in the **Open Drawings** sub-window of the **Drawing Explorer** dialog box.
- Launch the EXPIMAGES command.

To open the **Attachments** panel, do one of the following:

- Click the **Image Management** tool button () in the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGE command.

### 13.1.1 Attaching an image

1 Do one of the following:

- Click the **New** tool button () in the **Drawing Explorer - Images** dialog box.
- Click the **Attach Raster Image** tool button () in the **Insert** toolbar or the **Insert / Images** ribbon panel.

- Launch the IMAGEATTACH command.

The **Select Image File** dialog box appears.

2 Select the image file, then double-click or click the **Open** button.

The **Attach Raster Image** dialog box appears.

3 (option) Check the **Use Geocoding Information** option, then click the **Browse** button to open an appropriate world file to be used with the selected raster image.

If the **Use Geocoding Information** option is checked, the **Insertion Point**, **Scale** and **Rotation** settings fields are not available.

4 Specify an **insertion point** or select the **Specify on-screen** check box.

5 Specify a **scale** or select the **Specify on-screen** check box.

6 Specify a **rotation** angle or select the **Specify on-screen** check box.

7 Click the **OK** button.

8 If none of the **Specify on-screen** options are selected, the image is inserted at the desired insertion point, scale and rotation angle;

else you are prompted to specify the insertion point, scale and/or rotation angle on-screen.

### 13.1.2 Loading and unloading images

When an image is attached and then inserted in the drawing, you can unload the image to temporarily hide it. Unloaded images are still inserted in the drawing, but they are no longer displayed. If the **Imageframe** property is ON, the frame around the image is still displayed.

**To load / unload a single image**

Click the **Load** column of the image in the **Attachments** panel or the **Drawing Explorer – Images** dialog box.

**To load / unload multiple images**

1 Hold down the Ctrl key, then select the images in the **Attachments** panel or the **Drawing Explorer – Images** dialog box. Click a selected image to de-select.

2 Click the **Load** column of a selected image.

### 13.1.3 Detaching images

Since an image is first attached, then inserted in the drawing, you can delete an inserted image, in the drawing without detaching it.

To detach an image:

1 Select the image(s) in the **Attachments** panel or the **Drawing Explorer – Images** dialog box.

Hold down the Ctrl key to select multiple images; click a selected image to de-select.

2 Do one of the following:

- Click the Delete or Detach ( ✕ ) tool button.
- Right-click, then choose **Delete** or **Detach** in the context menu.

## 13.2 Image utilities

The IMAGEADJUST command adjusts the properties of images through the Properties panel.

The IMAGECLIP command clips images.

The IMAGEFRAME command toggles the display of the frame around images; sets the value of the IMAGEFRAME system variable.

The TRANSPARENCY command toggles the transparency of images that have a transparent alpha channel. The alpha channel is a special channel that handles transparency. When an image has an alpha channel on it, you can adjust the image's opacity levels and make bits translucent or totally see-through.

***Прим.:*** When the image frames do not appear (IMAGEFRAME = 0), you must use a window selection method to select an image.

### 13.2.1 Clipping an image

1 Do one of the following:

- Click the **Clip Image** tool button () on the **Image** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGECLIP command.

**You are prompted**: Select image:

2 Select the image.

3 **You are prompted**: Enter image clipping option [ON/OFf/New boundary] <New boundary>:

4 Press Enter to accept the **New Boundary** default option.

5 **You are prompted**: [Select polyline/Polygonal/Rectangular] <Rectangular>:

Press Enter to accept the **Rectangular** default option.

6 **You are prompted**: Specify first corner.

7 Click a point to define the first corner of the clipping rectangle.

8 The clipping rectangle is dynamically displayed.

**You are prompted**: Specify opposite corner:

9 Click a point to define the clipping rectangle.

10 The image is clipped.

### 13.2.2 Modifying the Image Frame setting

**Using the Image Frame tool (IMAGEFRAME command)**:

1 Do one of the following:

- Click the **Image Frame** tool () on the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGEFRAME command.

**You are prompted**: Enter image frame setting <current value>:

2 Do one of the following:

- Type 0 to hide image frames.
- Type 1 to display and plot image frames.

- Type 2 to display but not print image frames.

**Editing the IMAGEFRAME system variable**:

1 Open the **Settings** dialog box.

2 Go to **Drawing / Underlays**.

3 Set the FRAME system variable = 3: Use individual settings.

4 Select the IMAGEFRAME system variable.

5 The options are:

- 0: Hide image frames.
- 1: Display and plot image frames.
- 2: Display but do not plot image frames.

### 13.2.3 Setting the image transparency

1 Do one of the following:

- Click the **Image Transparency** tool button ( ) on the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the TRANSPARENCY command.

**You are prompted**: Select images.

2 Select the images you want to set the **Transparency** property of.

**You are prompted**: Enter transparency mode [ON/OFF]<current mode>:

3 Do one of the following:

- Press Enter to accept the current mode.
- Choose **On** to switch the **Transparency** property of the selected images on.
- Choose **Off** to switch the **Transparency** property of the selected images off.



Transparency of the camera image: OFF (left) and ON (right)

### 13.2.4 Setting the display properties of an image

1 Select an image.

2 The properties of the image appear in the **Properties** panel.

3 Go to the **Misc** properties category.

4 The following properties can be set **Yes** or **No**:

- **Show Image**: shows or hides the image.
- **Show clipped**: sets the clipping boundary on/off.
- **Clip inverted**: inverts the clipping boundary.
- **Background transparency**: sets the transparency on/off.

# 14. Аннотация

## 14.1 Dimensions

### 14.1.1 Обзор инструментов для создания размеров

Инструменты для создания размеров доступны:

- На панели инструментов **Размеры**
- В меню **Размеры**
- На панели ленты **Аннотации / Размеры**
- На закладке квадро-меню **Черчение** при **отсутствии выбранных объектов**.

| Значок | Инструмент | Команда |
|---|---|---|
|  | Линейный | DIMLINEAR |
|  | Совмещенный | DIMALIGNED |
|  | Arc | DIMARC |
|  | Радиус | DIMRADIUS |
|  | Diameter | DIMDIAMETER |
| | | DIMDISASSOCIATE |
| | | DIMREASSOCIATE |
|  | Угловой | DIMANGULAR |

| Значок | Инструмент | Команда |
|---|---|---|
|  | Baseline | DIMBASELINE |
|  | Continue | DIMCONTINUE |
|  | Координатный | DIMORDINATE |
|  | Повернутый | DIMLINEAR |
|  | Сделать наклонным | DIMEDIT + O |
|  | Мультивыноска | MLEADER |
|  | Редактировать мультивыноску | MLEADEREDIT |
|  | Добавить стрелку | AIMLEADEREDITADD |
|  | Удалить стрелку | AIMLEADEREDITREMOVE |
|  | Допуск | TOLERANCE |
|  | Маркер центра | DIMCENTER |

| Значок | Инструмент | Команда |
|---|---|---|
|  | Редактировать размерный текст | DIMEDIT + E |
|  | Повернуть размерный текст | DIMEDIT + R |
|  | Переместить размерный текст | DIMTEDIT |
|  | Восстановить размерный текст | DIMEDIT |
|  | Применить стиль | -DIMSTYLE + A |
|  | Сохранить стиль | -DIMSTYLE + S |
|  | Восстановить стиль | -DIMSTYLE |
|  | Статус размерных переменных | -DIMSTYLE + ST |
|  | Выноска | DIMLEADER |
|  | Быстрая выноска | QLEADER |

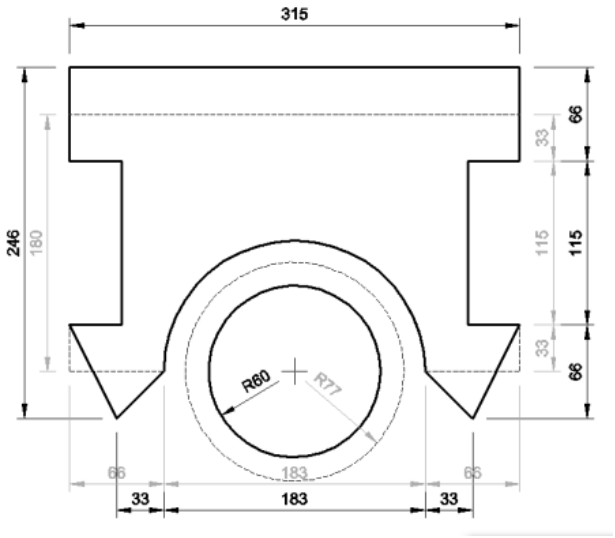
### 14.1.2 Ассоциативные размеры

Ассоциативные размеры обновляются автоматически при перемещении или изменении размеров соответствующего объекта.

BricsCAD создает ассоциативные размеры, если значение системной переменной DIMASSOC равно 2 (по умолчанию).

Свойство ассоциативности также применяется к размерам, которые размещаются в пространстве листа и связаны с объектами пространства модели.

При необходимости, вы можете использовать команду DIMREGEN для обновления всех ассоциативных размеров.




При создании ассоциативных размеров вы можете использовать следующие типы объектной привязки: **конечная точка**, **середина**, **центр**, **перпендикуляр**,**квадрант**, **точка вставки**, **точка**, **пересечение**, а также **мнимое пересечение**.

Команда DIMDISASSOCIATE удаляет ассоциативность для выбранных размеров.

Команда DIMREASSOCIATE восстанавливает или устанавливает ассоциативность размеров с объектами или точками на объектах.

Системная переменная GENERATEASSOCVIEWS позволяет создавать ассоциативные размеры для видов и чертежей, создаваемых с помощью команд VIEWBASE, VIEWSECTION и BIMSECTIONUPDATE.

### 14.1.3 Параметры размеров

Настройки размеров сохраняются в размерном стиле. Размеры создаются с использованием текущего размерного стиля.

Команда DIMSTYLE используется для создания и редактирования размерных стилей с помощью диалогового окна **Проводник по чертежам**.

Команда -DIMSTYLE используется для создания и редактирования размерных стилей с помощью командной строки.

Размерный стиль включает 6 групп настроек:

- **Линии и стрелки**
- **Text**
- **Fit**
- **Основные единицы**
- **Альтернативные единицы**
- **Допуски**

### 14.1.3.1 Отображение текущего состояния размерных переменных

1 Выполните одно из следующих действий:

- Нажмите кнопку **Статус размерных переменных** () на панели **Размеры**.
- Выберите пункт **Статус размерных переменных** в меню **Размеры**.

Список всех размерных переменных отобразится в командной строке.

2 Нажмите клавишу **F2** для открытия окна **История команд BricsCAD**.

3 Нажмите клавишу Enter для продолжения просмотра списка.

4 Нажмите клавишу **F2** для закрытия окна **История команд BricsCAD**.

## 14.1.4 Working with dimension styles

The DIMSTYLE command opens the **Drawing Explorer – Dimension Styles** dialog box.

Here you can:

- Create a new dimension style
- Create dimension child styles
- Edit a dimension style
- Define dimension style overrides
- Rename a dimension style
- Set a dimension style current
- Copy dimension styles between drawings
- Compare dimension styles
- Delete a dimension style

### 14.1.4.1 To set a dimension style current

First method:

1 Right click the dimension style field in the **Status bar**.

A context menu appears.

2 Select the dimension style in the context menu.

Second method:

1 Launch the DIMSTYLE command.

1 The current dimension style is marked in the **Current** column.

2 Click the current column of the dimension style.

#### 14.1.4.2 To create a new dimension style

1 Launch the DIMSTYLE command.

2 Select an existing dimension style, which will be used as a template for the new style.

3 Do one of the following:
- Click the **New** button.
- Place the cursor on an existing dimension style, then right click and choose **New** in the context menu.

4 Select the **Name** field and type a name for the new dimension style.

5 Define the settings for the new dimension style.

***Прим.:*** Use the **Save to new style** option of the context menu to create a new dimension style as a copy of the selected dimension style.

#### 14.1.4.3 Creating a dimension child style

For each dimension style, you can create a child style for each dimension type: **Linear**, **Angular**, **Radius**, **Diameter**, **Ordinate** and **Leader**.

If the current dimension style has a child style for a dimension type, the child style will be used automatically when such dimension type is created.

To create a dimension child style, do the following:

1 Select the parent dimension style, then right click and choose a **New child style** in the context menu.

A context menu appears.




2 Select a dimension entity type in the context menu.

3 The name of the child style is: <parent style>:<dimension entity type>

4 Select the child style and define the properties that need to be different from the parent style for the selected dimension entity type.

#### 14.1.4.4 To edit a dimension style

1 Launch the DIMSTYLE command.

2 Select the dimension style.

3 Adjust the dimension settings.

4 Close the **Drawing Explorer** dialog box.

#### 14.1.4.5 To define dimension style overrides

1 Launch the DIMSTYLE command.

2 Select the dimension style.

3 The first line of the dimension style list reads: <overrides on dimstylename>

4 Define the overrides.

5 Close the **Drawing Explorer** dialog box.

***Прим.:***

- It is not recommended to create dimensions with overrides.
- Use the dimension override settings to test changes, without modifying the current dimension style definition, then use the **Save overrides to current style** or **Save to new style** option to save the overrides.
- Setting another style current will discard the existing dimension style overrides.

#### 14.1.4.6 To rename a dimension style

1 Launch the DIMSTYLE command.

2 Select the dimension style.

3 Right click, then choose **Rename** in the context menu.

4 Type a new name in the **Name** field, then press enter.

#### 14.1.4.7 To copy dimension styles between drawings

1 Open both the source and the target drawing.

2 Launch the DIMSTYLE command.

3 Open drawings are listed in the **Open drawings** list on the **Drawing Explorer** dialog box.

4 Select **Dimension styles** of the source drawing in the **Open drawings** list.

5 Select the dimension style(s) to be copied.

6 Hold down the Ctrl key to select multiple dimension styles.

7 Place the cursor over one of the selected dimension styles, then press and hold the left mouse button and drag the selection onto the target drawing, then release the left mouse button.

#### 14.1.4.8 To compare dimension styles

1 Launch the DIMSTYLE command.

2 Select the dimension styles.

3 Click the first dimension style, then hold down the Ctrl key to select the next dimension style(s).

The properties of the selected styles display. The settings that are different, are highlighted.

4 (option) Hold down the Ctrl key to add or remove a dimension style from the comparison table.

### 14.1.5 Using the dimension style tools

The dimension style tools on the **Dimensions** toolbar and the **Dimensions** ribbon panel allow to:

- Apply the current dimension style to a selection of dimension entities.
- Create a new dimension style.
- Restore a dimension style.
- List the current status of the dimension settings.

#### 14.1.5.1 To apply the current dimension style

1 Do one of the following:

- Click the **Apply Style** tool ( ).
- Choose **Apply Style** in the **Dimensions** menu.
- Launch the -DIMSTYLE command, then type **a** or **apply** and press Enter.

2 **You are prompted**: Select dimensions to apply the current style.

3 Select the dimensions, then press Enter.

***Прим.:*** Since the dimension style settings only apply to dimension entities, you can select all entities in step 2 to apply the current dimension style to all dimensions in the drawing.

#### 14.1.5.2 To save a dimension style

1 Do one of the following:

- Click the **Save Style** tool button ( ) on the **Dimensions** toolbar.
- Choose **Save Style** in the **Dimensions** menu.
- Launch the -DIMSTYLE command, then type **S** or **save** and press Enter.

**You are prompted**: Save style: ? to list styles/Name for new dimension style:

2 Type a name for the new dimension style, then press Enter.

3 If the name already exists, the Command line prompts you to redefine it or not.

#### 14.1.5.3 To restore a dimension style

1 Do one of the following:

- Click the **Restore Style** tool button ( ) on the **Dimensions** toolbar.
- Choose **Restore Style** in the **Dimensions** menu.
- Launch the -DIMSTYLE command, then type **r** or **restore** and press Enter.

**You are prompted**: ? to list styles/Enter to select dimension/<Dimension style>:

2 Do one of the following:

- Type a dimension style name and press Enter.
- Press Enter, then click a dimension entity in the drawing.
- The dimension style of the selected entity is the current dimension style now.
- Type ? to see a list of the dimension styles in the current drawing.

#### 14.1.5.4 To list the settings in the current dimension style

Do one of the following:

- Click the **Dimension Variable Status** tool button () on the Dimensions toolbar.
- Choose **Dimension Variable Status** in the **Dimensions** menu.
- Launch the -DIMSTYLE command, then type **st** or **status** and press Enter.

The status of the settings in the current dimension style are listed in the **Prompt History** window. Press the F2 function key to close the **Prompt History** window.

### 14.1.6 Creating linear dimensions

Linear dimensions annotate linear distances or lengths and can be oriented either horizontally, vertically or aligned parallel to an existing entity or to the selected origin points.

The following commands create linear dimension entities:

- DIMLINEAR: Creates horizontal and vertical linear dimensions with respect to the current coordinate system (World or UCS).
- DIMALIGNED: Creates dimensions parallel to (aligned with) lines and polyline segments. The command also works with polyline arc segments, arcs, and circles, but dimensions their diameters (see DIMDIAMETER command).
- DIMBASELINE: Places multiple linear and angular dimensions, all starting at the same base point of an existing dimension. This command works only when at least one other dimension already exists in the drawing.
- DIMCONTINUE: Places a chain of dimensions by continuing linear or angular dimensions from the endpoint of the previous dimension or another existing dimension.
- DIMARC: Places dimensions that measure the lengths of arcs and polyline arc segments.

#### 14.1.6.1 To create a horizontal or vertical linear dimension

1 Launch the DIMLINEAR command.

1 **You are prompted**: Origin of first extension line <Select entity>:

2 Do one of the following:

- Specify the origin of the first extension line, then specify the origin of the second extension line.
- Press Enter, then select the line or linear segment of a polyline you want to dimension.

The dimension line is dynamically displayed.

Move the cursor in to create a horizontal dimension, or horizontally to create a vertical dimension.

3 Specify a point to position the dimension line.

#### 14.1.6.2 To create an aligned linear dimension

1 Launch the DIMALIGNED command.

1 **You are prompted**: Origin of first extension line <Select entity>:

2 Do one of the following:
   - Specify the origin of the first extension line, then specify the origin of the second extension line.
   - Press Enter, then select the line or linear segment of a polyline you want to dimension.

   The dimension line is dynamically displayed.

3 Specify a point to position the dimension line.

#### 14.1.6.3 To create a rotated linear dimension

1 Do one of the following:
   - Click the **Rotated** tool button on the **Dimensions** toolbar or ribbon panel.
   - Choose **Rotated** in the **Dimensions** menu.

   **You are prompted**: Origin of first extension line <Select entity>:Do one of the following:

2 Do one of the following:
   - Specify the origin of the first extension line, then specify the origin of the second extension line.
   - Press Enter, then select the line or linear segment of a polyline you want to dimension.

   **You are prompted**: Angle of dimension line <0>:

3 Do one of the following:
   - Type the rotation angle in the Command line, then press Enter.
   - Enter the rotation angle by specifying two points.

   The dimension line is dynamically displayed.

4 Specify a point to position the dimension line.

#### 14.1.6.4 To create baseline dimensions

1 Launch the DIMBASELINE command.

1 **You are prompted**: Baseline: Origin of next extension line or [Select starting dimension/Undo] <Select starting dimension>:

2 Do one of the following:
   - Go to step 2 to accept the baseline dimension which is dynamically displayed, starting from the most recently added linear or angular dimension.
   - Press Enter, then select the starting dimension entity.

3 Specify a point to create the first baseline dimension.

4 Keep specifying points or press Enter to stop.

***Прим.:***

- When selecting the starting dimension make sure to click the first extension line or the first half of the dimension line in order to use the first origin point of the starting dimension as the origin point for the baseline dimension.
- If you select an angular dimension, stacked angular dimensions are created.
- The distance between the stacked dimension lines is controlled by the DIMDLI system variable (Dim baseline spacing property in the dimension style).

#### 14.1.6.5 To create continued dimensions

1 Launch the DIMCONTINUE command.

1 **You are prompted**: Continue: Origin of next extension line or [Select starting dimension/Undo] <Select starting dimension>:

2 Do one of the following:

- Go to step 2 to accept the continued dimension which is dynamically displayed, starting from the most recently added linear or angular dimension.
- Press Enter, then select the starting dimension entity.

3 Specify a point to create the first continued dimension.

4 Keep specifying points or press Enter to stop.

***Прим.:***

- When selecting the starting dimension, make sure to click the second extension line or the second half of the dimension line in order to use the second origin point of the starting dimension as the origin point for the continued dimension. Otherwise, the new dimension line will partly overlap the starting dimension.
- If you select an angular dimension, continued angular dimensions are created.

#### 14.1.6.6 To create an arc length dimension

1 Launch the DIMARC command:

1 **You are prompted**: Select arc or polyline arc segment:

2 Select the arc or polyline arc segment.

3 The arc length dimension is dynamically displayed.

4 Specify a point to position the dimension line.

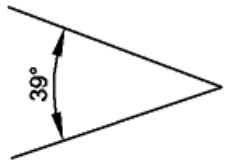
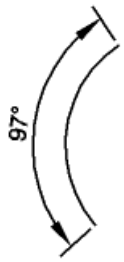
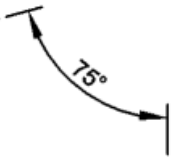
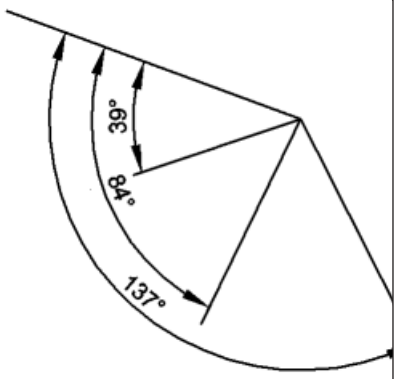
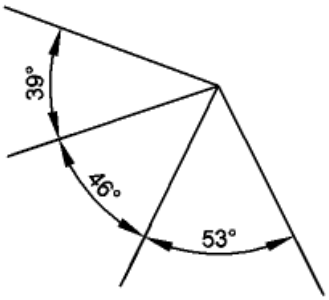
### 14.1.7 Создание угловых размеров

Для создания угловых размеров используется команда DIMANGULAR.

Угловой размер указывает угол, измеренный между двумя отрезками, двумя прямолинейными сегментами полилинии или внутренний угол дуги окружности. Для измерения угла Вы можете также указать вершину этого угла и две точки, определяющие стороны угла.

Используя существующие угловые размеры Вы можете создавать базовые размеры или размерную цепь.

- Базовый угловой размер создает размер от первой выносной линии предыдущего углового размера.
- При создании цепи угловых размеров создается размер от второй выносной линии предыдущего углового размера.

| | | |
|---|---|---|
|  |  |  |
| угловой размер между двумя отрезками | угловой размер дуги | угловой размер, заданный вершиной и двумя точками |
|  |  | |
| базовый угловой размер | цепь угловых размеров | |

### 14.1.7.1 Угловой размер между двумя линиями

1 Запустите команду DIMANGULAR.

1 **Запрос программы**: Укажите линию, дугу или окружность [Параметры выбора] <ENTER для выбора угла>:

2 Выберите отрезок или линейный сегмент полилинии.

3 **Запрос программы**: Вторая линия углового размера [Параметры выбора]:

4 Выберите второй отрезок или линейный сегмент полилинии.

5 Угловой размер будет отображаться динамически.

**Запрос программы**: Положение размерной дуги [Угол/Текст]:

6 Укажите точку для размещения размера.

7 Переместите курсор, чтобы измерить либо внутренний угол между линиями, либо его дополнение.

### 14.1.7.2 Угловой размер для дуги

1 Запустите команду DIMANGULAR.

1 **Запрос программы**: Укажите линию, дугу или окружность [Параметры выбора] <ENTER для

выбора угла>:

2 Выберите дугу.

3 Угловой размер будет отображаться динамически.

**Запрос программы**: Положение размерной дуги [Угол/Текст]:

4 Укажите точку для размещения размера.

#### 14.1.7.3 Угловой размер по вершине и двум точкам

1 Запустите команду DIMANGULAR.

1 **Запрос программы**: Укажите линию, дугу или окружность [Параметры выбора] <ENTER для выбора угла>:

2 Нажмите клавишу Enter.

3 **Запрос программы**: Вершина угла:

4 Укажите точку.

5 **Запрос программы**: Первая сторона угла:

6 Укажите точку.

7 **Запрос программы**: Вторая сторона угла:

8 Укажите точку.

9 Угловой размер будет отображаться динамически.

**Запрос программы**: Положение размерной дуги [Угол/Текст]:

10 Укажите точку для размещения размера.

11 Переместите курсор, чтобы измерить внутренний или внешний угол.

### 14.1.8 Создание диаметральных и радиальных размеров

Команды DIMCENTER и DIMDIAMETER создают диаметральные и радиальные размеры для аннотирования радиусов и диаметров окружностей и дуг окружностей.

#### 14.1.8.1 Создание диаметрального размера

1 Запустите команду DIMDIAMETER.

1 **Запрос программы**: Укажите окружность или дугу:

2 Укажите окружность или дугу.

3 Диаметральный размер отобразится в динамическом режиме.

4 Укажите точку размещения размерного объекта.

#### 14.1.8.2 Создание радиального размера

1 Запустите команду DIMRADIUS.

1 **Запрос программы**: Укажите окружность или дугу:

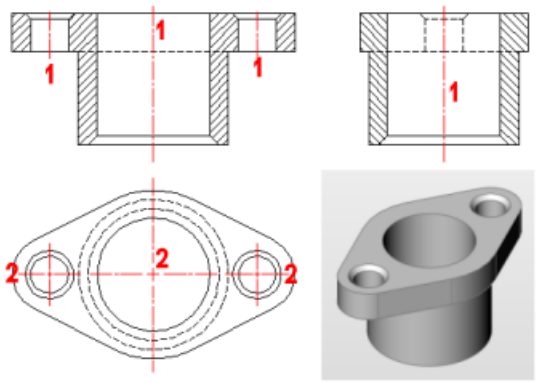
2 Укажите окружность или дугу.

3 Радиальный размер отобразится в динамическом режиме.

4 Укажите точку размещения размерного объекта.

### 14.1.9 Centerlines and center marks

Centerlines and center marks are drawing reference entities to indicate axes of symmetry and centers of circular openings. Centerlines and center marks are associative entities: when the referenced lines, circles or circular arcs are modified they are adjusted automatically.




1 Centerline

2 Center mark

The CENTERLINE command creates a centerline entity associated with two selected lines.

The CENTERMARK command creates a center mark entity associated with a selected circle or arc.

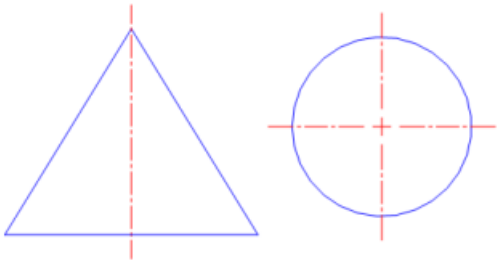
The CENTERREASSOCIATE command recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc.

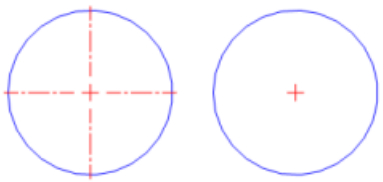
The CENTERDISASSOCIATE command breaks the association of centerline and a center mark entities.

The CENTERRESET command resets centerline and center mark entities.

#### 14.1.9.1 System variables for center entities

The properties and geometry of centerline and center mark entities are controlled through system variables.

| Name | Title | Description |
|---|---|---|
| CENTERCROSSGAP | Center mark cross gap | Defines the gap between the center mark and its centerlines.<br><br>Type a value to specify the absolute size of the gap in drawing units.<br>Type a value followed by x to set the size of the gap relative to the diameter of the circle or circular arc.<br>The default value 0.05x creates a gap of 5/100 of the diameter.<br>. (dot) creates no gap. |
| CENTERCROSSSIZE | Center mark cross size | Defines the size of the associative center mark.<br>Type a value to specify the absolute size of the center mark lines in drawing units.<br>Type a value followed by x to set the size of the center mark relative to the diameter of the circle or circular arc.<br>The default value 0.1x creates center marks 1/10th of the diameter.<br>Type a . (dot) to create center marks with zero length lines. |
| CENTEREXE | Centerline extensions length | Defines the length of the centerline extensions. The value is expressed in drawing units.<br> |
| CENTERLAYER | Default layer for center mark or centerline | Specifies the default layer for new centerlines and center marks.<br>Type a name. If such layer not yet exists, the layer will be created when a centerline or centemark is created.<br>Type a . (dot) to create centerlines and center marks on the current layer. |

| Name | Title | Description |
|---|---|---|
| CENTERLTYPEFILE | Linetype file for center mark or centerline | Specifies the linetype file used to create centerlines and center marks.<br>Default in imperial unit drawings: **default.lin**<br>Default in metric unit drawings: **iso.lin** |
| CENTERLTYPE | Linetype for center mark or centerline | Specifies the linetype for newly created centerlines and center marks.<br>Type a new linetype name or press Enter to accept the current name.<br>Type a . (dot) for continuous lines. |
| CENTERLTSCALE | Linetype scale for center mark or centerline | Specifies the linetype scale for newly created centerlines and center marks. |
| CENTERMARKEXE | Automatic extension for center mark or centerline | Determines whether centerlines extend (left) or not (right) for new center marks.<br> |

### 14.1.9.2 Editing center entities

The properties of center entities can be edited in the **Properties** panel.

## 14.1.10 Creating ordinate dimensions

An ordinate dimension annotates the perpendicular distance from an origin or base point (the origin of the current coordinate system (WCS or UCS). Ordinate dimensions consist of an x- or y-coordinate and a leader. An x-ordinate dimension measures distances along the x-axis, while a y-ordinate dimension measures distances along the y-axis.

As you select ordinate points, the program automatically detects whether the point is x- or y-ordinate based depending on the direction you drag the second point. You can also specify whether the ordinate represents an x- or y-ordinate. Ordinate dimension text is always aligned with the ordinate leader lines, regardless of the text orientation specified by the current dimension style.

### 14.1.10.1 To create an ordinate dimension

1 Set the origin of the coordinate system at the reference point for the ordinate dimensions.

2 Launch the DIMORDINATE command.

3 **You are prompted**: Select point for ordinate dimension:

4 Specify a point.

5 The ordinate dimension is dynamically displayed.

**You are prompted**: Leader endpoint [Xdatum/Ydatum/Text/Angle]:

6 Click to position the ordinate dimension.

### 14.1.11 Editing dimensions

You can edit dimension entities:

- Using grips.
- Edit the dimension text.
- Rotate the dimension text.
- Reposition the dimension text.
- Restore the text position.
- Make the extension lines oblique.

#### 14.1.11.1 Using grips to edit a linear dimension

1 Select the dimension entity.

1 5 grips display:

2 origin point grips (1)

2 dimension line grips (2)

1 dimension text grip (3)




2 (option) Drag the origin point grips to adjust the dimension origin points.

3 The dimension text is adjusted automatically.

4 (option) Drag dimension line grips to move the dimension line.

5 (option) Drag the text grip to move the dimension text.

***Прим.:***

- Depending on the **Text movement** setting (DIMTMOVE system variable), the dimension line will move along with the text or the text will move independently. A leader will be drawn if the **Text movement** setting is set to **Move text, add leader**.
- If two adjacent dimensions (continued dimensions) are selected both, you can move their coinciding grips simultaneously.

### 14.1.11.2 Editing the dimension text

Dimension text can be edited:

- in the MText editor:
  - using the DDEDIT command
  - double clicking the dimension entity
- in the **Text override** field in the **Properties** panel.
- using the **Edit Text** option of the DIMEDIT command.

***Прим.:***

***The following tags can be used to format the dimension text:***

- **\L** (backslash upper case L) starts underlined text, **\l** (backslash lower case L) to stop underlined text
- **<>** to add the default dimension text (= measured distance)
- **[]** to add the alternate units, when alternate units are currently switched off in the dimension style.
- **%%d** to add the degree symbol (°)
- **%%c** to add the diameter symbol (Ø)
- **%%p** to add the plus/minus symbol (±)

e.g. if you type **%%c\L<> cm** in the **Text override** field, dimension text displays as:

Ø24,43cm

### 14.1.11.3 To rotate the dimension text

1 Do one of the following:
   - Click the **Rotate Dimension Text** tool () in the **Dimensions** toolbar or ribbon panel.
   - Choose **Rotate Dimension Text** in the **Dimensions** menu.
   - Launch the DIMEDIT command then choose the **Rotate text** option.

   **You are prompted**: Angle of dimension text:

2 Type a value, then press Enter.

3 **You are prompted**: Select entities to rotate text [selection options (?)]:

4 Select the dimensions and press enter.

### 14.1.11.4 To reposition the dimension text

1 Do one of the following:
   - Click the **Reposition Dimension Text** tool () in the **Dimensions** toolbar or the ribbon panel.
   - Choose **Reposition Dimension Text** in the **Dimensions** menu.
   - Launch the DIMTEDIT command.

   **You are prompted**: Select dimensions to reposition text:

2 Select the dimension.

3 The dimension text moves dynamically.

4 Click to reposition the dimension text.

#### 14.1.11.5 To restore the text position

1 Do one of the following:

- Click the **Restore Text Position** tool ()
- Choose **Restore Text Position** in the **Dimensions** menu.
- Launch the DIMEDIT command and choose the **Restore text** option.

**You are prompted**: Select dimensions to restore text to default position [selection options (?)]:

2 Select the dimensions, then press Enter.

#### 14.1.11.6 To make the extension lines oblique

1 Do one of the following:

- Click the **Make Oblique** tool () in the **Dimensions** toolbar or ribbon panel.
- Choose **Make Oblique** in the **Dimensions** menu.
- Launch the DIMEDIT command and choose the **Oblique lines** option.

**You are prompted**: Select linear dimensions to make oblique [selection options (?)]:

2 Select the dimension entities, then press Enter.

3 **You are prompted**: Enter obliquing angle:

4 Type a value, then press Enter.

***Прим.:*** To restore non-rotated extension lines, execute the above procedure and type 0 (zero) at the 'Enter obliquing angle' prompt.

### 14.1.12 Создание выносок с комментариями

Выноска состоит из стрелки, отрезка, полилинии или сплайна и комментария.

Основные действия при создании выноски с комментарием:

- Определение положения стрелки выноски.
- Создание сегментов выноски.
- Ввод текста комментария.

#### 14.1.12.1 Создание выноски с комментарием

1 Выполните одно из следующих действий:

- Нажмите кнопку **Выноска** () на панели инструментов **Размеры** или закладке ленты Аннотации.
- Выберите пункт **Выноска** в меню **Размеры**.
- Запустите команду LEADER или DIMLEADER.

**Запрос программы**: Начало выноски:

2 Определение положения стрелки выноски.

3 **Запрос программы**: Следующая точка.

4 Укажите конечную точку первого сегмента выноски.

5 **Запрос программы**: В точку [Формат/На шаг назад/Надпись] <Надпись>:

6 Добавьте необходимое количество сегментов и нажмите клавишу Enter или щелкните правой кнопкой мыши для завершения создания сегментов.

7 **Запрос программы**: Первая строка надписи/<Параметры>:

8 Введите первую строку текста надписи и нажмите клавишу Enter.

9 **Запрос программы**: Следующая строка надписи:

10 Выполните одно из следующих действий:

- Нажмите клавишу Enter или щелкните правой кнопкой мыши для завершения создания выноски и комментария.
- Добавьте необходимое количество строк комментария, затем нажмите клавишу Enter или щелкните правой кнопкой мыши для создания выноски с комментарием.

#### 14.1.12.2 Создание выноски без комментария

1 Выполните одно из следующих действий:

- Нажмите кнопку **Выноска** () на панели инструментов **Размеры** или закладке ленты Аннотации.
- Выберите пункт **Выноска** в меню **Размеры**.
- Запустите команду LEADER или DIMLEADER.

**Запрос программы**: Начало выноски:

2 Определение положения стрелки выноски.

3 **Запрос программы**: Следующая точка.

4 Укажите конечную точку первого сегмента выноски.

5 **Запрос программы**: В точку [Формат/На шаг назад/Надпись] <Надпись>:

6 Добавьте необходимое количество сегментов и нажмите клавишу Enter или щелкните правой кнопкой мыши для завершения создания сегментов.

7 **Запрос программы**: Первая строка надписи/<Параметры>:

8 Нажмите клавишу Enter.

9 **Запрос программы**: Параметры размерного текста: [Блок/Копировать/Нет/Допуск/Многострочный текст] <Многострочный текст>:

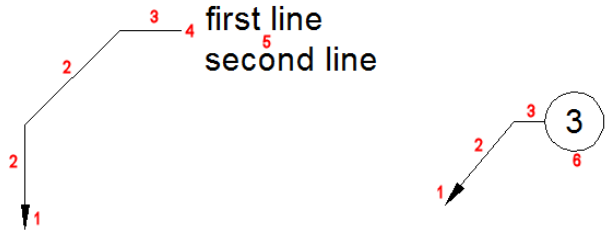
10 Выберите опцию **Нет**.

### 14.1.13 Creating multileaders

A multileader consists of:

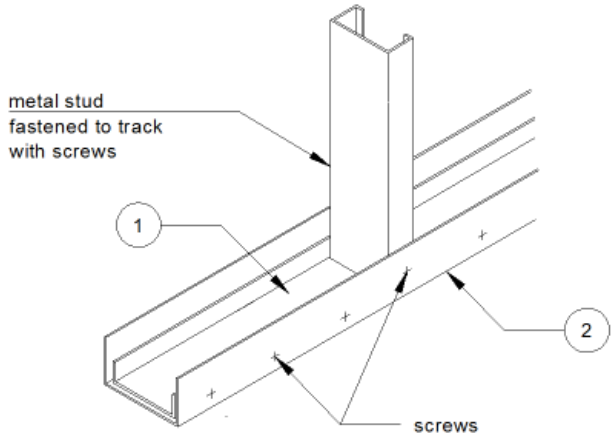
- an arrowhead (1)
- one or more leader lines (2), which can be straight or curved
- optionally a horizontal landing (3) and landing gap (4)

- content: either a multiline text (5) or a block (6)




Multileaders are created using the current multileader style. Each drawing contains a **Default** leader style, which cannot be renamed or deleted; the **Default** leader style can be customized though. The MLEADER command allows to define overrides on the current multileader style.

Multileaders can be created arrowhead first, landing first or content first.




### 14.1.13.1 To create a multileader

1 (option) Launch the MLEADERSTYLE command, then set the current multileader style.

2 Do one of the following:

- Click the **Multileader** tool button ( ) on the **Dimensions** toolbar or ribbon panel.
- Choose **Multileader** in the **Dimensions** menu.
- Launch the MLEADER command.

**You are prompted**: Specify leader arrowhead location or [leader Landing first/Content first/Options] <Options>:

3 (option) Select a placement option:

- arrowhead first
- landing first
- content first.

The most recently used placement option is repeated.

4 The default sequence is: arrowhead - landing - content.

5 (option) Choose **Options** if you want to define overrides on the current multileader style.

**You are prompted**: Enter an option [Leader type/leader lAnding/Content type/Maxpoints/First angle/Second angle/eXit options] <eXit options>:

Specify the option(s), then choose **Exit** to proceed.

6 Specify a point to define the leader arrowhead location.

7 **You are prompted**: Specify leader landing location:

8 Specify a point to define the leader landing location.

9 Depending on the content type:
- **MText**: The MText command is launched.
- **Block**: A block is inserted.
- **None**: The command is concluded.

### 14.1.13.2 Adding vertices to a leader line

1 If necessary press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the leader line you want to add a vertex to, then hold down the Ctrl key.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu and expand the **Modify** tab.

5 Click the **Add Vertex** tool ( ).

A new vertex is added dynamically.

6 Specify the position of the vertex.

### 14.1.13.3 Removing vertices from a leader line

1 If necessary press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the leader line you want to add a vertex to, then hold down the Ctrl key.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu expand the **Modify** tab.

5 Click the **Remove Vertex** tool ( ).

The vertex is removed.

### 14.1.13.4 Adding leader lines to a multileader

1 If necessary press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the multileader entity.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu and expand the **Modify** tab.

5 Click the **Add Leader** tool ( ).

A new leader line is dynamically displayed.

6 Specify the origin of the new leader line*.

7 Another new leader line is dynamically displayed.

8 Do one of the following:
- Specify the origin of the new leader line.
- Right click or press Enter to stop adding leader lines.

(*) When the Maximum leader points property of the multileader style is higher than 2, right click to stop adding vertices for the new leader.

### 14.1.13.5 Removing leader lines from a multileader

1 If necessary, press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the leader line you want to remove.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu and expand the **Modify** tab.

5 Click the **Remove Leader** tool ( ).

The leader line is removed.

## 14.1.14 Добавление геометрических допусков

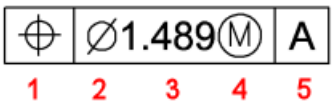
Геометрические допуски указывают предельные отклонения формы и расположения поверхностей от значений, определенных чертежом. Геометрический допуск определяет ширину или диаметр зоны допуска, в пределах которой поверхность или ось отверстия (или цилиндра) располагаются так, что будет обеспечена функциональность и взаимозаменяемость изделия.

Информации о геометрических допусках в BricsCAD отображается в рамке допуска, которая представляет собой разделенный на ячейки прямоугольник.

Каждая рамка допуска содержит не менее двух ячеек. Первая ячейка содержит символ геометрического допуска, относящийся к одному из типов допуска (расположение, ориентация, форма). Например, допуск формы может определять плоскостность или прямолинейность поверхности.

Вторая ячейка рамки допуска содержит значение предельного отклонения. При использовании символа диаметра в обозначении допуска этот символ располагается перед значением допуска. После значения допуска располагается символ зависимого допуска. Зависимый допуск применяется к изделиям с переменным размером.

После значения допуска могут располагаться идентификаторы баз - буквенные символы первой, второй и третьей баз с соответствующим символом зависимого допуска для каждой из баз. Базой называется теоретически точный элемент детали (точка, ось или плоскость), относительно которой производятся измерения и контроль размеров.




1 Символ геометрического допуска

2 Символ диаметра

3 Значение допуска

4 Символ зависимого допуска

5 Идентификатор базы

Когда для одной геометрии применяется два допуска, вы можете вставить обозначение составного допуска, состоящее из значения первого допуска и значения второго допуска. Дополнительно к позиционным допускам для их уточнения можно задавать выступающие поля допуска. Они могут, например, определять допуск перпендикулярности для детали, вставляемой или вворачиваемой в отверстие другой.

#### 14.1.14.1 Создание рамки геометрического допуска

1 Выполните одно из следующих действий:

- Нажмите кнопку **Допуски** (⌖) на панели **Размеры**.
- Выберите пункт **Допуски** в меню **Размеры**.
- Введите в командной строке **tolerance** и нажмите клавишу Enter.

Откроется диалоговое окно Геометрический допуск.

2 В окне **Геометрический допуск** укажите необходимые составные элементы рамки допуска.

3 Нажмите кнопку **ОК**.

Диалоговое окно **Геометрический допуск** закроется.

4 Укажите точку вставки рамки допуска на чертеже.

#### 14.1.14.2 Редактирование рамки геометрического допуска

1 Дважды щелкните на рамке геометрического допуска.

1 Откроется диалоговое окно **Геометрический допуск**.

2 Измените параметры.

3 Нажмите кнопку **ОК**.

Диалоговое окно **Геометрический допуск** закроется.

Содержимое рамки геометрического допуска будет обновлено.

## 14.2 Texts

### 14.2.1 Text system variables

System variables can be set using the **Settings** dialog box.

| Variable Name | Variable Title | Description |
|---|---|---|
| FONTALT | Alternate font | Defines the font which will be used if a text font cannot be found. |

| Variable Name | Variable Title | Description |
|---|---|---|
| HIDETEXT | Hide text on hide | Specifies whether the HIDE command processes text entities. |
| MIRRTEXT | Mirror text | Determines whether text is mirrored by the MIRROR command or not. |
| MTEXTED | Multiline text editor | Sets the primary and secondary text editors to use for multiline text entities. |
| MTEXTFIXED | Multiline text fixed | Controls whether BricsCAD zooms, rotates and/or pans the view to fit the Mtext being edited. |
| QTEXTMODE | Quick text mode | Controls how text entities are displayed. If ON, displays text as a box. |
| TEXTANGLE | Text angle | Stores the angle of the last added text entity. |
| TEXTFILL | Text fill | Determines whether True Type fonts show as either filled or outlined.<br> |
| TEXTQLTY | Text quality | Defines the smoothness of True Type fonts for plotting and rendering. |
| TEXTSIZE | Text size | Sets the default height for new text entities.<br>TEXTSIZE has no effect if the current text style has a fixed height. |
| TEXTSTYLE | Text style | Stores the name of the current text style. |
| TSPACEFAC | Text space factor | Specifies the line spacing distance of multiline text measured as factor of the text height.<br>Values between 0.25 and 4 are accepted. |
| TSPACETYPE | Text space type | Specifies the type of line spacing used for multiline text. At least: adjusts line spacing based on the tallest character(s) in a line. Exactly: used the specified line spacing, regardless of in individual character sizes. |
| TSTACKALIGN | Text stack align | Determines the vertical alignment of stacked text: bottom, center or top. |

| Variable Name | Variable Title | Description |
|---|---|---|
| TSTACKSIZE | Text stack size | Specifies the percentage a which the stacked text fraction height is relative to the current height of the selected text. Values between 25 and 125 are accepted. |

### 14.2.2 Работа со стилями текста

При вставке текста в чертеж используется текущий стиль текста, который определяет шрифт, размер, угол наклона, ориентацию и другие свойства создаваемого текстового объекта.

Каждый чертеж по умолчанию содержит стиль текста **Standard**, который первоначально использует шрифт **Arial**. Вы не можете удалить или переименовать стиль **Standard**, но вы можете изменить его с учетом своих предпочтений.

При изменении **шрифта** или **ориентации** для существующего стиля текста, все текстовые объекты, созданные с использованием этого стиля, будут автоматически обновлены в соответствии с внесенными изменениями. Изменение любого другого атрибута текста не окажет никакого действия на существующие текстовые объекты.

Вы можете создавать неограниченное количество дополнительных стилей текста.

Аннотативные стили текста создают аннотативные объекты. В аннотативном стиле текста параметр **Высота** определяет высоту текста в пространстве листа; высота текста в пространстве модели зависит от текущего значения масштаба аннотации, определяемого значениями системных переменных CANNOSCALE и CANNOSCALEVALUE.

#### 14.2.2.1 Окно Проводник по чертежам - Стили текста

Выполните одно из следующих действий:

- В меню **Сервис** > **Проводник по чертежам** выберите **Стили текста...**.
- Щелкните поле **Текущий текстовый стиль** в **строке состояния**.
- Запустите команду СТИЛЬ.

#### 14.2.2.2 Создание стиля текста

1 Откройте диалоговое окно **Проводник по чертежам - Стили текста**.

2 Выполните одно из следующих действий:

   - Нажмите кнопку **Создать** () в на панели инструментов раздела **Стили текста** диалогового окна **Проводник по чертежам**.
   - Щелкните правой кнопкой мыши на существующем стиле текста и в открывшемся контекстном меню выберите пункт **Создать**.

   Будет создан **новый стиль**, наследующий свойства текущего стиля.

3 В поле **Имя стиля текста**, введите новое имя стиля текста заменив имя **Стиль**, предложенное по умолчанию.

4 Определите свойства стиля текста.

5 Закройте диалоговое окно **Проводник по чертежам - Стили текста**.

#### 14.2.2.3 Редактирование стиля текста

1 Откройте диалоговое окно **Проводник по чертежам - Стили текста**.

2 Выберите стиль текста.

3 Определите свойства стиля текста.

4 Закройте диалоговое окно **Проводник по чертежам - Стили текста**.

#### 14.2.2.4 Установка текущего стиля текста

Выполните одно из следующих действий:

- Щелкните правой кнопкой мыши на поле **Текущий текстовый стиль** в **строке состояния** и выберите стиль текста.
- Откройте диалоговое окно **Проводник по чертежам - Стили текста**.

  Текущий стиль текста выделен маркером в столбце **Текущий**.

  Выполните одно из следующих действий:

  - Щелкните на поле **Текущий** рядом с выбранным стилем теста.
  - Щелкните правой кнопкой мыши на выбранном стиле текста и в открывшемся контекстном меню выберите **Установить текущим**.

#### 14.2.2.5 Удаление стиля текста

1 Откройте диалоговое окно **Проводник по чертежам - Стили текста**.

2 Выберите стиль текста.

3 Выполните одно из следующих действий:

- Нажмите кнопку **Удалить** ().
- Щелкните правой кнопкой мыши на выбранном стиле текста и в открывшемся контекстном меню выберите **Удалить**.

Если выбранный стиль текста в чертеже не используется, то он будет удален. Если стиль используется, появится сообщение.

4 Выполните одно из следующих действий:

- Нажмите кнопку **Удалить** для удаления стиля текста и всех использующих этот стиль объектов.
- Нажмите кнопку **Изменить...** для удаления стиля текста и назначения другого стиля текста для всех объектов, использующих удаляемый стиль.
- Нажмите кнопку **Отмена** для отмены процедуры удаления стиля.

***Прим.:*** Удаление текущего стиля текста невозможно.

### 14.2.3 Создание текста

Команда TEXT создает однострочные текстовые объекты.

Команда MTEXT создает форматированный многострочный текст в пределах указанных границ.

#### 14.2.3.1 Создание однострочного текста

1 Выполните одно из следующих действий:
   - Нажмите кнопку **Текст** (A) на панели инструментов **Черчение** или панели ленты **Аннотации / Текст**.
   - Выберите инструмент **Текст** в меню **Черчение**.
   - Запустите команду TEXT.

2 **Запрос программы**: Начальная точка текста или [Выбрать стиль/Подогнать с подбором высоты/Подогнать с подбором сжатия/Выровнять по центру строки/Выровнять по центру текста/Выровнять по правому краю/Дополнительно...]:

3 Укажите точку.

4 **Запрос программы**: Высота текста <текущее значение>:

5 Выполните одно из следующих действий:
   - Нажмите клавишу Enter или щелкните правой кнопкой мыши для подтверждения текущего значения высоты.
   - Введите значение.
   - Укажите точку, чтобы задать высоту текста графическим способом.

   **Запрос программы**: Угол поворота текста <текущее значение>:

6 Выполните одно из следующих действий:
   - Нажмите клавишу Enter или щелкните правой кнопкой мыши для подтверждения текущего значения угла.
   - Введите значение.
   - Укажите точку, чтобы задать угол поворота текста графическим способом.

7 Введите текст и нажмите клавишу Enter.

8 Будет создан новый текстовый объект.

9 Выполните одно из следующих действий:
   - Введите текст.
   - Нажмите клавишу Enter для завершения.

10 При необходимости, создайте другие текстовые объекты. Для завершения нажмите клавишу Enter.

***Прим.:*** Если на чертеже имеется текст, созданный ранее в текущей сессии работы BricsCAD, то при повторе команды TEXT последний созданный текстовый объект выделяется. Если Вы хотите разместить новый текст непосредственно под этим текстовым объектом и использовать такую же высоту и угол поворота текста, нажмите клавишу Enter.

**Параметры выравнивания однострочного текста**:




### 14.2.3.2 Создание многострочного текста

1 Выполните одно из следующих действий:

- Нажмите кнопку **Многострочный текст** (A) на панели инструментов **Черчение** или панели ленты **Аннотации / Текст**.
- Выберите инструмент **Многострочный текст** в меню **Черчение**.
- Запустите команду MTEXT.

**Запрос программы**: Многострочный текст: Первый угол текстового блока:

2 Укажите точку вставки многострочного текста.

3 **Запрос программы**: Укажите противоположный угол текстового блока [Выравнивание/ Угол поворота/Стиль текста/Высота текста/Направление/Ширина/Межстрочный интервал/ Столбцы]:

4 Выполните одно из следующих действий:

- Укажите противоположный угол текстового блока для определения максимальной ширины многострочного текста.
- Если включен режим **Динамические размеры**, высота и ширина текстового поля отображается динамически. Введите значения высоты и ширины в соответствующих полях динамических размеров. Нажмите клавишу TAB для переключения между полями и клавишу Enter для подтверждения введенного значения.
- Ведите в командной строке @ и нажмите клавишу Enter для использования неограниченной ширины текста.

Над текстовым блоком откроется панель **Форматирование текста**.

5 Введите текст.

6 Максимальная длина строки определяется шириной текстового блока, определенного в п.3.

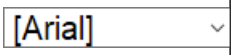
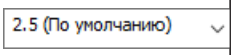
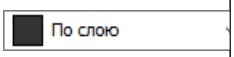
Для начала нового абзаца нажмите клавишу Enter.

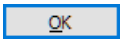
7 Для завершения команды и закрытия панели **Форматирование текста** выполните одно из

следующих действий:

- Нажмите кнопку **OK**
- Нажмите клавиши Ctrl+Enter.
- Щелкните за пределами многострочного текста.

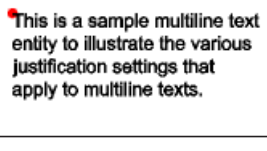
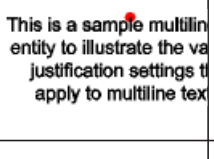
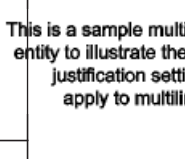
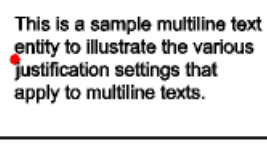
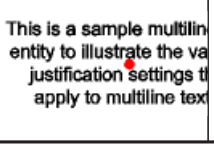
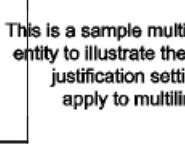
**Инструменты и настройки панели Форматирование текста**

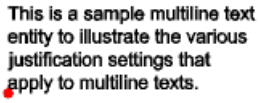
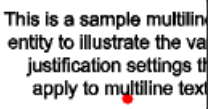
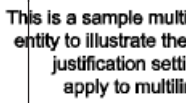
| Кнопка или параметр | Имя | Description |
|---|---|---|
|  | Стиль текста | Устанавливает стиль текста |
|  | Шрифт | Устанавливает шрифт текста |
|  | Высота текста | Устанавливает высоту текста |
|  | Полужирный | Создает полужирный текст |
|  | Курсив | Создает курсивный текст |
|  | Подчеркнутый | Создает подчеркнутый текст |
|  | Надчеркнутый | Создает надчеркнутый текст |
|  | Верхний регистр | Переводит выбранный текст в верхний регистр |
|  | Нижний регистр | Переводит выбранный текст в нижний регистр |
|  | Аннотативность | Включает/выключает свойство Аннотативность |
|  | Междустрочный интервал | Устанавливает интервал между строками |
|  | Столбцы | Выравнивает текст в столбцах |
|  | Цвет | Устанавливает цвет текста |
|  | Наклон | Устанавливает угол наклона символов текста |
|  | Сжатие | Устанавливает коэффициент сжатия символов текста |

| Кнопка или параметр | Имя | Description |
|---|---|---|
|  | Интервал символов | Устанавливает интервал между символами текста |
|  | На шаг назад | Отменяет последнее действие |
|  | Вернуть | Возвращает последнее отмененное действие |
|  | Дробный текст | Создает текст в виде дроби |
|  | Специальный символ | Выполняет вставку специального символа |
|  | Поле | Выполняет вставку поля с переменной |
|  | Выравнивание | Позволяет выбрать способ выравнивания текста |
|  | Фоновая маска | Добавляет цветной фон |
|  | Линейка | Включает/выключает отображение шкалы линейки |
|  | OK | Закрывает панель **Форматирование текста** и применяет изменения |
|  | Отмена | Закрывает панель **Форматирование текста** и сохраняет многострочный текст без изменений |

**Выравнивание многострочного текста**

Красная точка соответствует точке вставки многострочного текста.

| | | |
|---|---|---|
|  |  |  |
| Влево-Вверх | По центру-Вверх | Верхний правый |
|  |  |  |

| Влево-По середине | По центру-По середине | Вправо-По середине |
|---|---|---|
|  |  |  |
| Влево-Вниз | По центру-Вниз | Нижний правый |

### 14.2.3.3 Вставка специальных символов

Вы можете использовать управляющие коды и юникод для вставки специальных символов в текстовые объекты.

Для вставки управляющих кодов при вводе текста введите два символа процента (%%) и управляющий код. Одиночный символ процента обрабатывается программой как обычный текстовый символ.

**Специальные текстовые символы**

| Управляющий код | Строка Unicode | Результат |
|---|---|---|
| %%d | \U+00B0 | Символ градуса (°) |
| %%p | \U+00B1 | Символ плюс-минус (±) |
| %%c | \U+2205 | Символ диаметра (Ø) |
| %%k | \U+0336 | Начало зачеркивания текста. Для окончания повторите этот код. |
| %%u | \U+005F | Начало подчеркивания текста. Для окончания повторите этот код. |
| %%o | \U+203E | Начало надчеркивания текста. Для окончания повторите этот код. |

**Символы и строки Юникода**

| Имя | Symbol (*) | Строка Unicode |
|---|---|---|
| Приблизительно равно | ≈ | \U+2248 |

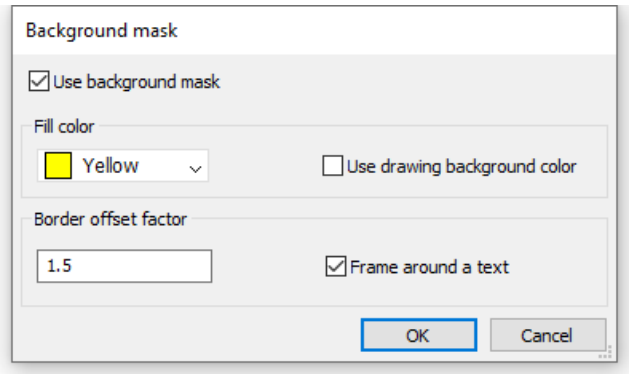
| Имя | Symbol (*) | Строка Unicode |
|---|---|---|
| Угол | ∠ | \U+2220 |
| Осевая линия | ℄ | \U+2104 |
| Дельта | Δ | \U+0394 |
| Электрическая фаза | ϕ | \U+0278 |
| Обозначение | ≡ | \U+2261 |
| Не равно | ≠ | \U+2260 |
| Омега | Ω | \U+03A9 |
| Квадратичная | ² | \U+00B2 |
| Возведение в куб | ³ | \U+00B3 |

(*) при поддержке используемым шрифтом.

### 14.2.3.4 Добавление фоновой маски

1 При создании или редактировании многострочного текста нажмите кнопку **Фоновая маска** ( ) на панели **Форматирование текста**.

Откроется диалоговое окно **Фоновая маска**:




2 Check the **Use background mask** option.

3 Выполните одно из следующих действий:

- Выберите цвет в выпадающем списке поля **Цвет заливки**.

- Выберите строку **Выбрать...** в выпадающем списке поля **Цвет заливки** и выберите цвет на закладке Индексные цвета или Реальные цвета диалогового окна **Выбор цвета**.
- Включите опцию **Использовать цвет фона чертежа**.

4 Определите **Коэффициент смещения контура**, который устанавливает размер полей контура фона вокруг текста. Значение коэффициента смещения контура зависит от размера текста и устанавливается в пределах 1-5. При коэффициенте, равном 1, фоновая маска совпадает с габаритами многострочного текста.

5 При необходимости, включите опцию **Рамка вокруг текста**.

6 Нажмите кнопку **OK**, чтобы создать фоновую маску.

### 14.2.4 Редактирование текста

Как и для других объектов чертежа, для текстовых объектов могут использоваться такие операции редактирования как удаление, перемещение, копирование, поворот и масштабирование.

Для редактирования свойств текста вы можете использовать **Панель свойств**.

Системная переменная TEXTED управляет режимом редактирования однострочного текста:

- 0 = редактирование по месту
- 1 = редактирование в диалоговом окне
- 2 = редактирование по месту с повторным вводом

Системная переменная MTEXTED позволяет использовать для редактирования многострочного текста внешний текстовый редактор.

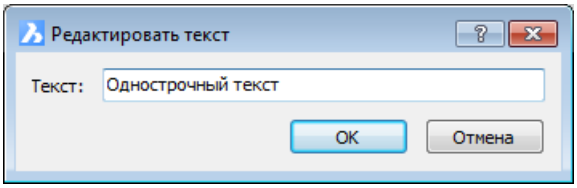
Системная переменная MTEXTFIXED управляет возможностью масштабирования, панорамирования и вращения вида для более удобного отображения многострочного текста при его редактировании.

#### 14.2.4.1 Редактирование однострочного текста

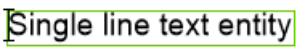
1 Щелкните дважды на текстовом объекте.

2 В зависимости от значения системной переменной TEXTED:

3 Откроется диалоговое окно **Редактирование текста**




или текст будет доступным для редактирования по месту.




4 Измените текст и нажмите клавишу Enter.

5 Выполните одно из следующих действий:

- Выберите другой текстовый объект для редактирования.

- Нажмите клавишу Enter для завершения редактирование текстовых объектов.

#### 14.2.4.2 Редактирование многострочного текста

1 Дважды щелкните на многострочном тексте.

2 Отобразится панель инструментов **Форматирование текста**.

3 Выполните редактирование текста и нажмите кнопку **ОК** или щелкните за пределами текстового блока.

4 Выполните одно из следующих действий:
- Выберите другой многострочный текст для редактирования
- Нажмите клавишу Enter для завершения редактирование многострочных текстов.

#### 14.2.4.3 Редактирование набора многострочных текстов

1 Выберите многострочные тексты.

2 На **Панели свойств** выберите поле **Содержимое**.

3 Нажмите кнопку **Обзор** (...) у правого края поля **Содержимое**.

4 Панель инструментов **Форматирование текста** откроется для первого из выбранных многострочных текстов.

5 Выполните редактирование многострочного текста.

6 Нажмите кнопку **ОК** или щелкните за пределами текстового блока.

Теперь вы можете приступить к редактированию следующего многострочного текста, включенного в набор для редактирования.

7 Повторите действия по п.5 и 6 для изменения всех выбранных многострочных текстов.

#### 14.2.4.4 Использование внешнего редактора многострочного текста

1 Запустите команду SETTINGS, чтобы открыть диалоговое окно **Параметры**.

2 Переместить назад **Чертеж** > **Черчение** > **Объект** > **Создание** > **Тексты** > **Многострочный текст**выберите системную переменную MTEXTED.

3 Введите в командной строке имя файла и полный путь к внешнему текстовому редактору, затем нажмите клавишу Enter.

4 например, C:\Program Files\Windows NT\Accessories\Wordpad.EXE

***Прим.:*** Чтобы вернуться к использованию встроенного текстового редактора, удалите содержимое в поле значения системной переменной MTEXED.

### 14.2.5 Exploding text

The TXTEXP command explodes Text and Mtext entities into 2D polylines.

#### 14.2.5.1 To explode text

1 Select the text(s)

2 Do one of the following:
   - Choose **Explode** in the **Edit** tab of the **Quad**.
   - Launch the TXTEXP command.

***Прим.:***

- The solid fill of certain TTF fonts is removed when exploding such texts, leaving the text outlines only.
- Use the JOIN command to create closed polylines from the exploded text outlines.
- The **Explode** tool in the **Edit** tab of the **Quad** launches the TXTEXP command. The EXPLODE command cannot explode text entities.

### 14.2.6 Поиск и замена текста

Команда FIND позволяет задать параметры поиска текста, найти этот текст на чертеже и выполнить его замену. При необходимости Вы можете приблизить текстовые объекты, содержащие искомый текст, для его просмотра.

#### 14.2.6.1 Настройка параметров поиска и замены текста

1 Запустите команду FIND.

1 Откроется диалоговое окно **Поиск и замена текста**.

2 Нажмите кнопку **Параметры**.

3 Выберите необходимые опции в окне **Параметры поиска и замены текста**.

4 Нажмите кнопку **ОК**.

#### 14.2.6.2 Поиск текста

1 Запустите команду FIND.

1 Откроется диалоговое окно **Поиск и замена текста**.

2 При необходимости, настройте параметры **Поиска и замены текста**.

3 При необходимости, нажмите кнопку **Указать объекты** ( ) для создания набора объектов.

Если объекты не выбраны, поиск выполняется по всему чертежу.

   - Окно **Поиск и замена текста** временно закроется для выбора объектов.
   - Щелкните правой кнопкой мыши для завершения выбора объектов.

В поле **Искать в** опция **Весь чертеж** будет заменена опцией **Выделенные объекты**.

4 В поле **Искать** введите заменяемый текст.

5 Нажмите кнопку **Найти**.

6 В поле **Результаты поиска** отобразится первый текстовый объект, содержащий искомый текст.

7 При необходимости нажмите кнопку **Показать**.

Найденный текстовый объект отобразится в центре экрана.

8 Выполните одно из следующих действий:

9 Повторите действия п. 5 и 6 для поиска других текстовых объектов, содержащих искомый

текст.

10 Повторите действия п. 2-6 для поиска другого текста.

11 Нажмите клавишу Esc или кнопку **Закрыть** для завершения поиска.

#### 14.2.6.3 Замена текста

1 Запустите команду FIND.

1 Откроется диалоговое окно **Поиск и замена текста**.

2 При необходимости, настройте параметры **Поиска и замены текста**.

3 При необходимости, нажмите кнопку **Указать объекты** ( ) для создания набора объектов.

Если объекты не выбраны, поиск выполняется по всему чертежу.

- Окно **Поиск и замена текста** временно закроется для выбора объектов.
- Щелкните правой кнопкой мыши для завершения выбора объектов.

В поле **Искать в** опция **Весь чертеж** будет заменена опцией **Выделенные объекты**.

4 В поле **Искать** введите заменяемый текст.

5 В поле **Заменить на** введите новый текст.

Отобразится список всех текстовых объектов, содержащих строку поиска.

6 Выполните одно из следующих действий:

- Щелкните на элементе списка, чтобы выделить соответствующий текстовый объект на чертеже. Для замены выделенного текста нажмите кнопку **Заменить**.
- Нажмите кнопку **Заменить все**.

### 14.2.7 Проверка правописания

Функция проверки орфографии осуществляет контроль правильности написания текста в текущем чертеже. Вы можете выполнить проверку как одного или нескольких выбранных объектов, так и всего чертежа в целом.

В процессе проверки команда SPELL сравнивает слова на чертеже или из выбранного набора текстовых объектов со словами в текущих основном и пользовательских словарях. Если слово не обнаружено ни в одном из этих словарей, вы можете выбрать правильное написание этого слова из предложенных вариантов или добавить новое слово в текущий пользовательский словарь. Пользовательский словарь используется для хранения специальных терминов.

#### 14.2.7.1 Настройка словаря

Словари, используемые командой SPELL, определяются значениями системных переменных DCTMAIN (для основного словаря) и DCTCUST (для пользовательского словаря).

***Прим.:*** Вместе с BricsCAD поставляется только английский словарь, что связано с лицензией на право использования словарей: английская версия словаря распространяется свободно; все остальные языковые версии словарей являются платными. Но вы можете выполнить загрузку других словарей и также использовать их бесплатно (см. Добавление основного словаря).

### 14.2.7.2 Диалоговое окно Проверка орфографии

Выполните одно из следующих действий:

- Выберите пункт **Проверка орфографии** в меню **Сервис**.
- Нажмите кнопку **Проверка орфографии** на панели ленты **Аннотации / Текст**.
- Запустите команду SPELL.

### 14.2.7.3 Выполнение проверки орфографии

1 Откройте диалоговое окно **Проверка орфографии**.

2 При необходимости нажмите кнопку **Выбрать** ( ) для создания набора объектов.

Если объекты не выбраны, поиск выполняется по всему чертежу.

- Диалоговое окно **Проверка орфографии** временно закроется для выбора объектов.
- Щелкните правой кнопкой мыши для завершения выбора объектов.

В поле **Область проверки** опция **Весь чертеж** будет заменена опцией **Текущий набор**.

3 При необходимости измените текущие словари (см. ниже).

4 Нажмите кнопку **Начало**.

Первое найденное слово, содержащее возможную ошибку, отобразится в поле **Текущее слово**.

5 Содержимое текстового объекта, в котором было найдено **текущее слово**, отобразится в поле **Контекст**.

6 Выполните одно из следующих действий:

- Нажмите кнопку **Пропуск** для сохранения **Текущего слова** без изменений.
- Нажмите кнопку **Пропуск-Все** для сохранения без изменений всех объектов, содержащих **Текущее слово**.
- Используйте слово в поле **Рекомендации** или выберите слово в списке **Рекомендации**, затем нажмите кнопку **Изменить**, чтобы заменить слово в поле **Текущее слово** словом из поля **Рекомендации**.
- Используйте слово в поле **Рекомендации** или выберите слово в списке **Рекомендации**, затем нажмите кнопку **Замена-Все**, чтобы заменить слово в поле **Текущее слово** словом из поля **Рекомендации**.
- Нажмите кнопку **Добавить** для добавления **Текущего слова** в текущий пользовательский словарь.
- Нажмите кнопку **Поиск** для поиска слов, подобных предложенному в поле **Рекомендации**.

7 Повторяйте предыдущие действия до тех пор, пока не появится сообщение **Проверка орфографии закончена**.

### 14.2.7.4 Изменение словарей

1 Откройте диалоговое окно **Проверка орфографии**.

2 Нажмите кнопку **Редактировать словарь**.

Откроется диалоговое окно **Редактирование словаря**.

3 Для изменения основного словаря нажмите кнопку раскрытия списка в разделе **Основной словарь**.

4 Выберите новый основной словарь.

5 При необходимости нажмите кнопку **Обзор...** для выбора нового **Пользовательского словаря**.

6 Чтобы добавить слово в текущий пользовательский словарь, введите в поле **Содержимое пользовательского словаря** новое слово и нажмите кнопку **Добавить**.

7 Нажмите кнопку **Применить и закрыть**.

**Примечание**:

- Пользовательские словари рекомендуется сохранять в папке **Support** пользовательского каталога **Roamable**.
- Если Вы хотите использовать данные пользовательского словаря MS Word, при выполнении шага 5 укажите путь к этому словарю. По умолчанию в качестве пользовательского словаря в MS Word используется файл CUSTOM.DiC, размещенный в папке *C:\<имя пользователя>\AppData\Roaming\Microsoft\Proof\CUSTOM.DIC*.

### 14.2.7.5 Добавление нового основного словаря

1 Откройте диалоговое окно **Проверка орфографии**.

2 Нажмите кнопку **Редактировать словарь**.

3 Нажмите кнопку **Загрузить...**.

В интернет - браузере откроется страница загрузки словарей https://extensions.openoffice.org/dictionaries.

4 Выберите необходимый язык и щелкните на соответствующей ему ссылке, чтобы открыть страницу этого языка.

5 Нажмите кнопку **Download extension** для начала загрузки ***.oxt** файла.

6 Измените расширение файла **oxt** на **zip**.

7 Извлеките из загруженного архива файлы ***.aff** и ***.dic** и переместите их в каталог **C:\Program Files\Bricsys\BricsCAD V...\Support**.

После выполнения этой операции словарь станет доступным в списке **Основной словарь** диалогового окна **Редактирование словаря**.

### 14.2.7.6 Добавление нового пользовательского словаря

1 Откройте текстовый редактор, например **Notepad**.

2 Введите слова, которые Вы хотите использовать в пользовательском словаре.

3 Каждое слово должно находиться в отдельной строке.

4 Сохраните созданный файл в папке **Support** пользовательского каталога **Roamable**. Расширение файла **.cus** .

### 14.2.8 Working with fields

A field is a text entity which contains instructions to display the status or property of another entity or setting in the drawing. When a field is updated it displays the latest value of the data source it refers to. Fields can be used to insert drawing properties, custom properties or object properties in multiline texts or tables or as an attribute in a block. Fields in attributes are especially useful in title blocks: using drawing, sheet set, subset and sheet properties title blocks can be filled out automatically.

See the DWGPROPS command to learn more about default and custom drawing properties.

See the SHEETSET command to learn more about default and custom sheet set, subset and sheet properties.

If no value is available, a field displays hyphens (----).

If the FIELDDISPLAY system variable is ON, fields are displayed on a light gray background which does not print.

The FIELDEVAL system variable controls when fields are updated.

#### 14.2.8.1 General procedure to create fields

1 Do one of the following:

- Click the **Insert Field** tool button () on the **Insert** toolbar and the **Annotate / Text** ribbon panel.
- Choose **Insert Field...** in the **Insert** menu.
- Launch the FIELD command.
- Click the **Field** tool button () on the **Text Formatting** toolbar when creating or editing Mtext or a table cell content.

The **Field** dialog box appears.

2 Under **Field Names**, expand a group, then choose a field name.

The settings for the selected field name appear.

3 Adjust the settings, then click the **OK** button.

The **Field** dialog box closes.

The field is created in the Mtext or table cell being edited.

or

**You are prompted**: Specify start point or [Height/Justify]:

Do the following:

- (option) Type **H** or choose **Height** in the context menu to define the text height.
- (option) Type **J** or choose **Justify** in the context menu to set the text justification.
- Specify the insertion point of the field in the drawing.

An Mtext entity containing the field is created.

#### 14.2.8.2 To display an entity property in a field

1 Launch the FIELD command.

1 The **Field** dialog box appears.

2 Expand **Objects** in the **Field names** list and select **Object**.

3 Click the **Select Entity** button () on the **Field** dialog box.

The **Field** dialog box closes temporarily.

**You are prompted**: Select Entity:

4 Click the entity you want to display a property of.

5 The **Field** dialog box appears again.

6 Select the entity property in the **Property** list.

7 Set the display layout:

- **Format**: choose a unit format.
- **Precision**: select a precision in the list button
- **Convert**: this option allows to recalculate the field value.
  - Choose a formula in the list; e.g. f*#, where f is the factor you type in the **Convert** field and # is the field value.
  - Type a factor in the **Convert** field; e.g. 0.0001 to express the area of a polyline in square meters in a cm drawing.

8 Click the **OK** button.

9 Specify a point to place the field in the drawing.

#### 14.2.8.3 To edit a field

1 Edit the table cell or Mtext entity that contains the field.

2 Double click the field.

3 The **Field** dialog box appears.

4 Adjust the properties of the field.

5 Click the **OK** button to apply the changes.

#### 14.2.8.4 To update a field

1 Do one of the following:

- Click the **Update Field** tool button () on the **Redraw / Regen** toolbar.
- Choose **Update Fields...** in the **Tools** menu.
- Launch the UPDATEDFIELD command.

**You are prompted**: Select field(s) to update:

2 Select the field(s) you want to update, then right click or press Enter.

3 The selected fields are updated.

***Прим.:*** If bitcode 16 of the FIELDEVAL system variable is set, executing the REGEN command updates all fields in the drawing.

#### 14.2.8.5 To convert a field to text

1 Edit the table cell or Mtext entity that contains the field.

2 Select the field.

3 Right click, then choose **Convert Field to Text** in the context menu.

### 14.2.9 Работа с таблицами

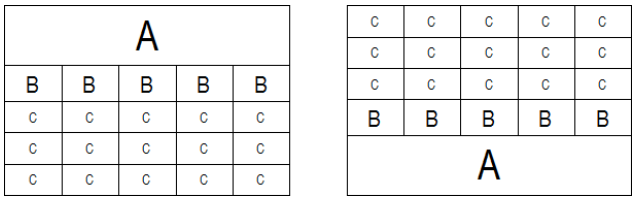
Таблица представляет собой объект, состоящий из строк и столбцов по типу таблиц Microsoft Excel. Вы можете создать пустую таблицу или выполнить импорт данных из CSV (Comma Separated Values) файла. Отображение текста и линий сетки таблицы определяется параметрами стиля таблицы. Стили таблицы сохраняются в файле чертежа.

По умолчанию таблица состоит из:

- ячейки **Наименование** (A)
- строки ячеек **Заголовок** (B)
- одна или несколько строк ячеек **Данные** (C).

Таблица может иметь одно из двух направлений:

- **Вниз** (строка заголовков сверху)
- **Вверх** (строка заголовков внизу).




#### 14.2.9.1 Создание пустой таблицы

1 Выполните одно из следующих действий:

- Нажмите кнопку **Таблица...** ( ) на панели **Черчение** или панели ленты **Главная / Аннотации**.
- Выберите инструмент **Таблицы...** в меню **Черчение**.
- Запустите команду TABLE.

Откроется диалоговое окно **Вставка таблицы**.

2 Выберите Стиль таблицы в списке **Стиль таблицы**.

3 При необходимости, задайте стиль ячейки для первых, вторых и других строк.

4 По умолчанию для первой строки используется стиль **Наименование**, для второй строки - стиль **Заголовок** и для других строк - стиль **Данные**.

5 В разделе **Параметры таблицы** выберите опцию **Создать пустую таблицу**.

6 В разделе **Поведение при вставке** выполните одно из следующих действий:
   - Выберите опцию **Указать точку вставки** и определите количество столбцов, ширину столбца, количество строк с данными и высоту строки.
   - Выберите опцию **Указать окно** и выполните одно из следующих действий:
   - Определите количество столбцов и строк с данными.
   - Определите ширину столбца и высоту строки.
   - Определите ширину столбца и количество строк с данными.
   - Определите количество столбцов и высоту строк.
7 Нажмите кнопку **ОК**.

   В зависимости от вашего выбора в шаге 5 **вам будет предложено** указать точку вставки или окно.

   Будет создана таблицы и откроется панель **Форматирование текста** (см. Создание текста).
8 Выполните одно из следующих действий:
   - Начните добавлять текст в таблицу.
   - Щелкните за пределами таблицы для закрытия панели инструментов **Форматирование текста**.

### 14.2.9.2 Создание таблицы по исходным данным

При импорте данных из CSV( Comma Separated Values) файла убедитесь, что символ разделителя элементов списка в вашей системе такой же, как и в CSV файле (см. Определение символа разделителя элементов списка).

1 Выполните одно из следующих действий:
   - Нажмите кнопку **Таблица...** () на панели **Черчение** или панели ленты **Главная / Аннотации**.
   - Выберите инструмент **Таблицы...** в меню **Черчение**.
   - Запустите команду TABLE.

   Откроется диалоговое окно **Вставка таблицы**.
2 В разделе **Параметры таблицы** выберите опцию **Загрузить файл**.
3 Нажмите кнопку **Обзор** ().
4 Откроется диалоговое окно **выбора файла**.

   Выберите CSV файл и нажмите кнопку **Открыть**.
5 Нажмите кнопку **Вставить** в диалоговом окне **Вставка таблицы**.
6 Укажите на чертеже точку для вставки таблицы.
7 Будет создана таблицы и откроется панель **Форматирование текста** (см. Создание текста).
8 Выполните одно из следующих действий:
   - Выполните редактирование текста таблицы.
   - Щелкните за пределами таблицы для закрытия панели инструментов **Форматирование**

**текста**.

9 При необходимости выберите таблицу и переместите ее узловые точки для изменения размеров таблицы.

### 14.2.9.3 Создание таблицы из таблицы Excel

1 Откройте таблицу в **Microsoft Office Excel**.

2 Выберите ячейки, которые Вы хотите использовать для создания таблицы.

3 Чтобы скопировать выбранные ячейки в буфер обмена, выполните одно из следующих действий:

- Нажмите сочетание клавиш **Ctrl + C**.
- Щелкните правой кнопкой и в контекстном меню выберите **Копировать**.

4 В BricsCAD выберите пункт **Специальная вставка** в меню **Правка** или запустите команду PASTESPEC.

5 Откроется диалоговое окно **Специальная вставка**.

6 Выберите **Электронная таблица XML** в списке **возможных вариантов** вставки.

7 Щелкните в чертеже для вставки таблицы.

### 14.2.9.4 Редактирование таблицы с помощью узлов

Для выбранной таблицы отображаются несколько узловых точек:

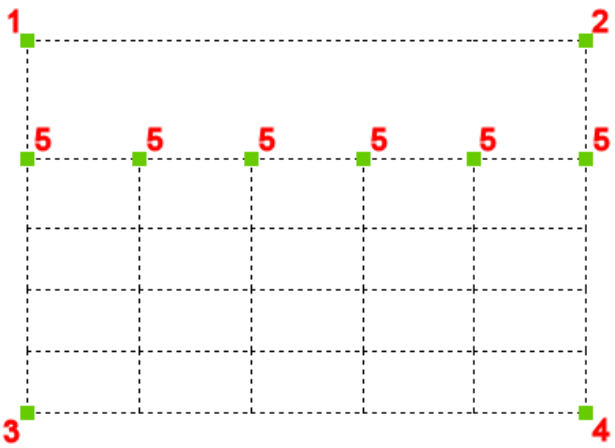
1 Перемещение таблицы.

2 Равномерное изменение ширины.

3 Равномерное изменение высоты.

4 Равномерное изменение высоты и ширины.

5 Изменение ширины столбца




### 14.2.9.5 Редактирование таблицы с использованием инструментов панели Таблицы

1 Чтобы открыть панель инструментов **Таблица**, выполните одно из следующих действий:

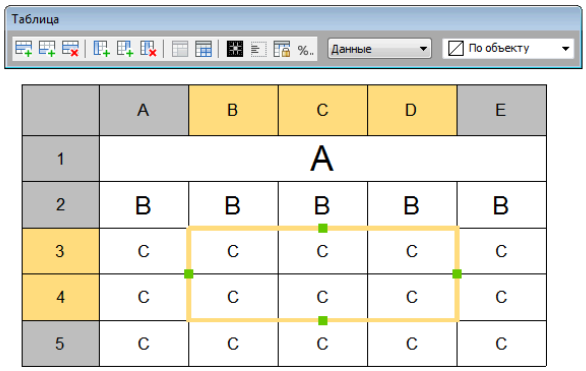
- Щелкните на ячейке, которую необходимо изменить.

- Нажмите левую кнопку мыши и, удерживая ее нажатой, определите прямоугольник для выбора нескольких ячеек.

2 Отобразятся буквы обозначения столбцов и номера строк.

3 Выбранные столбцы и строки будут выделены.

Выбранные ячейки будут ограничены прямоугольником с четырьмя управляющими точками.




4 При необходимости, перетяните узловые точки для изменения размеров.

5 При необходимости, выберите инструмент на панели **Таблица**.

6 При необходимости, щелкните правой кнопкой мыши для отображения контекстного меню:

- **Удалить содержимое ячейки**: удаляет текст в выбранных ячейках.
- **Очистить переназначения стиля ячейки**: сбрасывает все свойства выбранных ячеек к первоначальным настройкам **стиля ячейки**, определенным в **Стиле таблицы**.
- **Очистить переназначения стиля таблицы**: сбрасывает свойства всех ячеек таблицы к первоначальным настройкам **стиля ячейки**, определенным в **Стиле таблицы**.

7 При необходимости, повторите шаги 3 и 4 для продолжения редактирования таблицы.

8 При необходимости, щелкните дважды на ячейке для изменения текста ячейки.

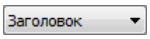
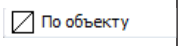
9 Выполните одно из следующих действий:

- Выберите другую ячейку.
- Нажмите левую кнопку мыши и, удерживая ее нажатой, переместите курсор для выбора нескольких ячеек.

10 Щелкните за пределами таблицы для закрытия панели инструментов **Таблица**.

**Инструменты для работы с таблицами**

| Значок | Инструмент | Description |
|---|---|---|
|  | Вставить строку выше | Вставляет пустую строку над выбранной областью таблицы. |

| Значок | Инструмент | Description |
|---|---|---|
|  | Вставить строку ниже | Вставляет пустую строку под выбранной областью таблицы. |
|  | Удалить строку | Удаляет выбранные строки. |
|  | Вставить столбец слева | Вставляет пустой столбец слева от выбранной области таблицы. |
|  | Вставить столбец справа | Вставляет пустой столбец справа от выбранной области таблицы. |
|  | Удалить столбец | Удаляет выбранные столбцы. |
|  | Объединить | Объединяет выбранные ячейки. |
|  | Разбить ячейки | Разбивает объединенные ранее ячейки. |
|  | Установить стиль обрамления | Устанавливает стиль обрамления выбранной области таблицы (пока не реализовано). |
|  | Выравнивание текста | Устанавливает режим выравнивания текста в выбранной области таблицы. |
|  | Блокировка ячейки | Включает/выключает блокировку выбранной области таблицы. |
|  | Установить формат ячейки | Устанавливает формат содержимого для выбранных ячеек (пока не реализовано). |
|   | Стиль ячейки | Устанавливает стиль для выбранных ячеек: **Наименование**, **Заголовок** или **Данные**. |
|   | Фон ячейки | Устанавливает цвет фона для выбранных ячеек. |

### 14.2.9.6 Редактирование таблицы с помощью Панели свойств

1 Щелкните на одной из линий сетки таблицы.

1 Свойства таблицы отобразятся на **Панели свойств**.

2 При необходимости, примените другой **Стиль таблицы**.

3 При необходимости, измените направление таблицы.

4 При необходимости, измените ширину и/или высоту таблицы.

5 При необходимости измените вертикальные или горизонтальные поля ячейки.

6 Нажмите клавишу Esc для завершения редактирования.

#### 14.2.9.7 Редактирование ячейки таблицы с помощью Панели свойств

1 Щелкните на ячейке таблицы.

1 Свойства ячейки отобразятся на **Панели свойств**.

2 При необходимости, измените ширину и/или высоту ячейки.

3 Ширина столбца и/или высота строки, содержащие выбранную ячейку таблицы, изменятся в соответствии с новыми значениями.

4 При необходимости, измените угол поворота текста.

5 Доступные значения: 0°, 90°, 180° и 270°.

6 При необходимости, измените текст содержимого ячейки.

7 При необходимости, измените способ выравнивания текста в ячейке.

8 Нажмите клавишу Esc для завершения редактирования.

#### 14.2.9.8 Вычисление формул

В настоящее время поддержка формул реализована частично. Обработка сложных формул пока не поддерживается , но в большинстве случаев функциональность таблиц Excel при выполнении их импорта в виде таблиц сохраняется. Редактирование и создание формул с помощью диалогового окна пока недоступны, но имеется возможность редактирования/создания формул по месту. Например, если вы введете в ячейке выражение =SUM(A2:A4), оно будет распознано программой как формула и в этой ячейке будет отображено значение суммы значений ячеек A2, A3 и A4.

#### 14.2.9.9 Редактирование содержимого таблицы

1 Выполните одно из следующих действий:

- Щелкните дважды на ячейке, которую необходимо изменить.
- Запустите команду TABLEDIT.
- **Запрос программы:** Укажите ячейку для редактирования.

  Щелкните на ячейке, которую необходимо изменить.

Над выбранной ячейкой откроется панель инструментов **Форматирование текста** (см. Создание многострочного текста).

2 Отредактируйте ячейку.

3 При необходимости, используйте сочетание клавиш Alt + Enter для создания второй строки в той же ячейке.

4 Чтобы изменить следующую ячейку:

- Нажмите клавишу TAB для перехода к следующей ячейке в этой же строке.
- Нажмите сочетание клавиш Shift + TAB для перехода к предыдущей ячейке в этой же строке.

- Нажмите клавишу Enter для перехода к следующей ячейке в этой же колонке.
- Нажмите сочетание клавиш Shift + Enter для перехода к предыдущей ячейке в этой же колонке.
- Нажмите одну из клавиш управления курсором.

5 Для завершения редактирования щелкните за пределами таблицы.

#### 14.2.9.10 Экспорт таблицы

При экспорте данных в файл CSV( Comma Separated Values) в качестве символа разделителя в CSV файле используется символ разделителя элементов списка, установленный в Вашей системе (см. Определение символа разделителя).

1 Запустите команду TABLEEXPORT.

1 **Запрос программы:** Укажите таблицу для экспорта:

2 Щелкните на таблице, которую необходимо экспортировать.

3 Откроется диалоговое окно **Экспорт данных**.

4 Выберите папку для сохранения данных таблицы.

5 В поле **Имя файла** введите имя файла.

6 Нажмите кнопку **Сохранить**.

Данные выбранной таблицы будут сохранены в файл CSV( Comma Separated Values).

#### 14.2.9.11 Установка символа разделителя списка

1 Откройте диалоговое окно **Параметры** в Windows.

2 Откройте категорию настроек **Регион**.

3 Выберите **Дополнительные параметры даты, времени, региона**.

4 Выберите **Изменение форматов даты, времени и чисел**.

5 Нажмите кнопку **Дополнительные параметры**.

6 Задайте символ **Разделителя элементов списка**.

7 Нажмите кнопку **ОК** для подтверждения.

## 14.3 Annotative Entities

### 14.3.1 Аннотативные объекты

Аннотации - это объекты чертежа, которые не относятся к модели или к объектам геометрии, например, такие как размеры, текст, символы и другие. Общей проблемой для аннотаций является их зависимость от масштаба. Некоторые аннотации должны масштабироваться в соответствии с масштабом печати или отображаться только в определенных масштабах.

Аннотативные стили и объекты изменяют свои параметры отображения (размер, расположение и ориентация) в зависимости от текущего коэффициента масштабирования и видового окна.

Свойство аннотативности может быть присвоено следующим объектам:

- текст и многострочный текст
- штриховки
- блоки и атрибуты
- размеры и допуски
- выноски и мультивыноски

Свойство аннотативности может быть присвоено следующим стилям:

- стили текста (команда STYLE)
- размерные стили (команда DIMSTYLE)
- стили мультивыноски (команда MLEADERSTYLE)

**Примечание**:

- Объектам, созданным с применением аннотативного стиля, автоматически присваивается текущий масштаб аннотации.
- При применении аннотативного стиля к объекту, который не является аннотативным, автоматическое присвоение свойства аннотативности не происходит.

#### 14.3.1.1 Команды

ANNORESET: Для каждого выбранного аннотативного объекта все альтернативные масштабируемые представления возвращаются к местоположению текущего масштабируемого представления объекта, синхронизируя таким образом все масштабируемые представления.

ANNOUPDATE: Обновляет все выбранные аннотативные объекты в соответствии с определением текущего стиля.

OBJECTSCALE: Добавляет или удаляет поддерживаемые масштабы аннотативных объектов с помощью диалогового окна.

-OBJECTSCALE: Добавляет или удаляет поддерживаемые масштабы аннотативных объектов с помощью командной строки.

SCALELISTEDIT: Позволяет редактировать список масштабов чертежа, используемого в разделе **Масштаб печати** диалоговых окон **Печать** и **Параметры страницы** и свойство **Стандартный масштаб** видового окна пространства бумаги. Определяет набор доступных масштабов для аннотативных объектов.

-SCALELISTEDIT: Позволяет редактировать список масштабов чертежа с помощью командной строки.

#### 14.3.1.2 Системные переменные

ANNOALLVISIBLE: Включает/выключает отображение аннотативных объектов, которые не поддерживают текущий масштаб аннотации.

ANNOTATIVEDWG: Определяет поведение чертежа как аннотативного блока при его вставке в другой чертеж.

CANNOSCALE: Устанавливает имя текущего масштаба аннотации для текущего пространства. Могут быть использованы только масштабы, существующие в текущем списке масштабов. Значение параметра CANNOSCALE сохраняется в пространстве модели и в каждом видовом окне различных компоновок листов. При создании аннотативного объекта автоматически применяется текущий масштаб аннотации.

CANNOSCALEVALUE: Отображает значение текущего масштаба аннотации. Например, масштабу 1:2 соответствует значение 0.5.

HPANNOTATIVE: Управляет присвоением свойства аннотативности вновь создаваемым объектам штриховки.

MSLTSCALE: Если для этой системной переменной установлено значение ON, типы линий в пространстве модели масштабируются в соответствии с текущим масштабом аннотации. При изменении значения MSLTSCALE необходимо выполнить обновление чертежа. Данная переменная эквивалентна переменной PSLTSCALE (Масштаб типа линий пространства листа) для пространства листа.

MSOLESCALE: Управляет размером OLE-объектов, содержащих текст, при их вставке в пространство модели. Системная переменная MSOLESCALE оказывает влияние только при вставке объекта. Изменение этого параметра не влияет на существующие OLE объекты.

SAVEFIDELITY: Управляет включением/выключением сохранения чертежа с визуальной точностью для открытия в программах, не поддерживающих аннотативные объекты.

SELECTIONANNODISPLAY: Управляет включением/выключением отображения всех масштабных представлений аннотативного объекта при выборе этого объекта.

#### 14.3.1.3 Установка текущего масштаба аннотаций

Чтобы установить текущий масштаб аннотации, выполните одно из следующих действий:

- Щелкните на поле **Масштаб аннотаций** в разделе **Дополнительно** окна **Панели свойств** и нажмите кнопку со стрелкой для выбора необходимого значения масштаба в выпадающем списке.
- Щелкните правой кнопкой на поле **Масштаб аннотаций** в **строке состояния** и в открывшемся списке выберите необходимое значение масштаба. Если поле **Масштаб аннотаций** не отображается, нажмите кнопку со стрелкой у правого края **строки состояния** и в открывшемся списке выберите пункт **Масштаб аннотаций**.

***Прим.:*** Вы можете использовать только те масштабы, которые входят в **Список масштабов** чертежа. Для редактирования **Списка масштабов** используйте команду SCALELISTEDIT.

#### 14.3.1.4 Присвоение объекту свойства аннотативности

Аннотативным объектам, которые были созданы с использованием неаннотативного стиля, может быть присвоено свойство аннотативности, для чего необходимо выполнить следующие действия:

1 Выберите аннотативный объект.

2 На **Панели свойств** выберите свойство **Аннотативность**.

3 Выберите опцию **Да**.

Текущий масштаб аннотации будет применен к выбранному объекту.

### 14.3.2 Annotation scales

Annotation scales are used to calculate the scaling factor for annotative entities. It defines the height of annotative text entities or the overall scaling of annotative blocks. In general, the annotation scale allows you to create annotative entities in model space at the correct paper space size automatically.

- Texts, MTexts, Dimensions, MLeaders: The current annotation scale is applied upon creation if the current style is annotative.
- Hatches and Gradient Fills: The current annotation scale is applied if the HPANNOTATIVE system variable is ON.
- Blocks: The current annotation scale defines the scaling of annotative blocks upon insertion.

Annotative entities can support multiple annotation scales. When the SELECTIONPREVIEW system variable is active or when using the **Quad** cursor menu, an icon is displayed next to the highlighted entity, indicating whether a single annotation scale () applies to an annotative entity or multiple ().

#### 14.3.2.1 To add or remove annotation scales

The following procedure allows to:

- Check which annotation scales currently apply.
- Add annotation scales.
- Remove annotations scales.

1 Select the annotative entity or entities.

2 In the **Properties** panel, select the **Annotative Scale** field.

3 Click the **Browse** button () of the **Annotative Scale** field.

The **Annotation Object Scale** dialog box appears.

4 (option) If multiple entities were selected, choose whether to list all scales for the selection, or only the common scales.

5 (option) Do one of the following:

- Select one or more scales in the list, then click the **Delete** button. The selected scale(s) are removed from the list.
- Click the **Add...** button, then select one or more scales in the list and click the **OK** button.

The selected scale(s) are added to the list.

6 Click the **OK** button to close the **Annotation Object Scale** dialog box.

The changes are applied to the selection set.

***Прим.:*** Only scales that exist in the **Scale** list of the drawing are available. Use the SCALELISTEDIT command to edit the **Scale** list.

### 14.3.3 Аннотативные текстовые объекты

Текстовые объекты создаются с использованием текущего стиля текста:

- Если стиль текста является аннотативным, то созданному с использованием этого стиля текстовому объекту автоматически присваивается текущий масштаб аннотаций.
- Если стиль текста не является аннотативным, то созданному с использованием этого стиля текстовому объекту необходимо вручную присвоить свойство аннотативности.

Аннотативные текстовые объекты имеют два размерных параметра:

- **Высота текста в пространстве модели**: определяет высоту текста на экране в пространстве модели относительно текущего масштаба аннотации.
- **Высота текста в пространстве листа**: определяет высоту текста на бумаге при печати с использованием поддерживаемого масштаба аннотации.

#### 14.3.3.1 Назначение объекту свойства аннотативности

1 Выберите текстовый объект.

1 Свойства выбранного текстового объекта отображаются на **Панели свойств**.

2 На **Панели свойств** выберите свойство **Аннотативность**.

3 В списке значений выбранного свойства выберите **Да**.

Будет назначен текущий масштаб аннотации.

Текст не масштабируется: текущая высота текста соответствует значению **Высота текста в пространстве модели**.

#### 14.3.3.2 Как удалить свойство аннотативности

1 Выберите текстовый объект.

1 Свойства выбранного текстового объекта отображаются на **Панели свойств**.

2 На **Панели свойств** выберите свойство **Аннотативность**.

3 В списке значений выбранного свойства выберите **Нет**.

Если выбранный объект поддерживает несколько масштабов аннотации, для этого объекта будет установлено исходное значение масштаба.

### 14.3.4 Аннотативные штриховки и градиентные заливки

Системная переменная HPANNOTATIVE управляет присвоением свойства аннотативности новым штриховкам и градиентным заливкам.

По умолчанию для переменной HPANNOTATIVE установлено значение OFF. Значение этой системной переменной не сохраняется и при закрытии BricsCAD восстанавливается ее значение по умолчанию.

#### 14.3.4.1 Настройка системной переменной HPANNOTATIVE

Выполните одно из следующих действий:

- Включите параметр Аннотативность шаблона штриховки в диалоговом окне **Параметры** в разделе **Черчение** > **Создание объектов** > **Штриховки**.
- Включите параметр **Аннотативность** в диалоговом окне **Штриховка и градиентная заливка**.
- Выберите параметр **Аннотативность** в командной строке после запуска команды -HATCH.

#### 14.3.4.2 Включение аннотативности штриховки и градиентной заливки

1 Выберите штриховку или градиентную заливку.

1 Свойства выбранного объекта отображаются на **Панели свойств**.

2 Выберите свойство **Аннотативность**.

3 В списке значений выбранного свойства выберите **Да**.

Текущий масштаб аннотации будет применен к выбранной штриховке или градиентной заливке с соответствующим изменением масштаба штриховки.

#### 14.3.4.3 Как удалить свойство аннотативности

1 Выберите штриховку или градиентную заливку.

1 Свойства выбранного объекта отображаются на **Панели свойств**.

2 Выберите свойство **Аннотативность**.

3 В списке значений выбранного свойства выберите **Нет**.

### 14.3.5 Annotative blocks

The annotative property of a block is either set when creating the block definition using the BLOCK command, or next, in the **Drawing Explorer / Blocks** dialog.

Annotative blocks can have an optional property to always match the orientation of a layout, independent from the rotation of the paper space viewport content.

#### 14.3.5.1 To make a block definition annotative

1 Launch the EXPBLOCKS command.

1 The **Drawing Explorer / Blocks** dialog box appears.

2 If necessary, choose **Detail View**.

Annotative block definitions have the **Annotative** icon () in the **Annotative** column.

3 Click in the **Annotative** field of a block definition to toggle the annotative property on/off.

#### 14.3.5.2 To set the Match Orientation to Layout property

1 Launch the EXPBLOCKS command.

1 The **Drawing Explorer / Blocks** dialog box is displayed.

2 If necessary, choose **Detail View**.

3 Tick the **Match Orientation to Layout** checkbox of a block definition to set the property.

### 14.3.6 Аннотативные размеры

Размерные объекты создаются с использованием текущего размерного стиля:

- Если размерный стиль является аннотативным, вновь созданному размеру автоматически присваивается свойство аннотативности и назначается текущий масштаб аннотаций.
- Если размерный стиль не является аннотативным, свойство аннотативности для размеров присваивается вручную.

**Примечание**:

- При использовании аннотативного размерного стиля параметр **Масштаб стиля** (DIMSCALE) не доступен для редактирования и в аннотативных размерах не учитывается.
- Для аннотативных размерных стилей рекомендуется устанавливать значение параметра **Масштаб стиля** равным 1.

#### 14.3.6.1 Присвоение размеру свойства аннотативности

1 Выберите размер.

1 Свойства выбранного текстового объекта отображаются на **Панели свойств**.

2 Выберите свойство **Аннотативность**.

3 В списке опций выберите опцию **Нет**.

Масштаб размера изменится в соответствии с текущим значением системной переменной DIMSCALE.

### 14.3.7 Аннотативные выноски

Мультивыноски создаются с использованием текущего стиля мультивыноски:

- Если стиль мультивыноски является аннотативным, выноске автоматически присваивается свойство аннотативности и текущий масштаб аннотации.
- Если стиль мультивыноски не является аннотативным, необходимо вручную сделать объект Мультивыноска аннотативным.

#### 14.3.7.1 Как сделать объект Мультивыноска аннотативным

1 Выберите объект **Мультивыноска**.

Свойства выбранной **мультивыноски** отобразятся на **Панели свойств**.

2 Выберите свойство **Аннотативность**.

3 В списке значений выбранного свойства выберите **Да**.

Будет назначен текущий масштаб аннотации.

Масштаб **мультивыноски** изменится в соответствии с текущим масштабом аннотаций.

#### 14.3.7.2 Как удалить свойство аннотативности

1 Выберите объект **Мультивыноска**.

Свойства выбранной **мультивыноски** отобразятся на **Панели свойств**.

2 Выберите свойство **Аннотативность**.

3 В списке значений выбранного свойства выберите **Нет**.

# 15. Настройка

## 15.1 Startup options

When starting up BricsCAD, it is possible to add arguments or switches. For example:

- The Command line "BricsCAD.exe drawing1.dwg drawing2.dwg drawing3.dwg" starts BricsCAD and loads the specified drawings.
- The Command line "bricscad.exe /P MyProfile" starts BricsCAD using the MyProfile user profile.

"/P MyProfile" is called a Command line switch. Both a slash or a hyphen are accepted to precede the switch, e.g. "-P MyProfile" will works. The space is optional, e.g. "/PMyProfile" would also work, and the switch character is case insensitive, e.g. "-p MyProfile" is also fine.

| Switch | Description |
|---|---|
| /P [userProfileName] | Starting without the /P option, BricsCAD uses the current profile (= the most recently used profile) which is not necessarily the Default profile.<br>If the user profile specified with the /P option doesn't exist at startup, then a new empty user profile will be created with that name.<br>If the /P argument is an .arg file the arg file is used to initialize the profile settings on the first run (See Starting BricsCAD with a specific profile). |
| /B [scriptFileName.scr] | Runs a script (*.scr) or lisp (*.lsp) at startup.<br>Opens all the file types that can be dragged and dropped on the application main frame window.<br>If you pass filenames without a parameter, they are opened as if they had been passed with /B. Therefore, bricscad.exe mylisp.lsp is exactly the same as bricscad.exe /B mylisp.lsp, and bricscad.exe my.dwg is identical to bricscad.exe /B my.dwg. |
| /L | Suppresses the startup splash screen. |
| /LD [application.arx/ brx/drx/lsp] | Loads an application on startup and on every new or opened drawing depending on the file extension. If the file extension is .lsp/.des/.mnl, it is treated as a lisp file and loaded in every drawing. Other file types are loaded only once at startup. |
| /Safemode | Prevents all third party plug-in code, enablers and user lisp files from loading in the current session. Starting in a clean environment can prevent potential crash causes.<br>See Launching BricsCAD in safe mode. |
| /S [path] | Redefines the SRCHPATH system variable. e.g. BricsCAD.exe /S C: \BricsCAD\ |
| /T [TemplateFileName] | Creates a new drawing based on the template drawing specified. |

| Switch | Description |
|---|---|
| regserver<br>unregserver | To register/unregister BricsCAD COM. Please note these arguments are to be used without a preceding "/". |

***Прим.:***

- Switch arguments containing spaces should be enclosed by double quotes, e.g. /P "My user profile".
- Switches can be combined, for example **/P MyProfile /L /S C:\BricsCAD** starts BricsCAD using the **MyProfile** user profile without the splash screen and sets the SRCHPATH system variable to **C:\BricsCAD** .
- Switches /B and /LD enable all sorts of files to be loaded, some maybe not expected. For example, .cui, .dsd, .txt, .pdf types are also processed.
- If GETSTARTED = ON when you launch BricsCAD, the **BricsCAD Launcher** dialog box appears, which lets you choose the workspace.

Besides these switches, BricsCAD supports file names as Command line arguments.

Supported file types are:

- DWG - to load a drawing
- DXF - to load a drawing
- SCR - to load a script
- LSP - to load a LISP program
- DLL - to load an ADS/SDS or DRX/ARX program
- SLD - to load a slide file
- MNU - to load a menu file
- CUI - to load a menu file

## 15.2 Пользовательские файлы BricsCAD

В BricsCAD пользовательские файлы хранятся в определенных папках:

- **PlotConfig**: файлы конфигурации плоттера (*.pc3)
- **Plotstyles**: файлы таблиц стилей печати (*.stb) и файлы таблиц цветозависимых стилей печати (*.ctb)
- **Support**:
  - файлы шаблонов штриховки (*.pat)
  - файлы стилей линий (* .lin)
  - файлы псевдонимов (* .pgp)
  - файлы единиц (* .unt)
  - файлы настройки пользовательского интерфейса (* .cui)
  - Файлы библиотеки BIM (* .bsyslib), см. команду BIMPROJECTINFO.

- Файлы конфигурации структуры (*.cst)|*.cst, см. команду STRUCTUREPANEL.
- Файлы конфигурации панели Библиотека (*.ccf), см. команду LIBRARYPANELOPEN.

- **Templates**: шаблоны чертежей (* .dwt)

| Платформа | Папки | Вложенные папки |
|---|---|---|
| Windows | C:\Users\<ИмяПользователя>\AppData\Local \Bricsys\BricsCAD\<Версия>\ru_RU | \Templates |
| Windows | C:\Users\<ИмяПользователя>\AppData \Roaming \Bricsys\BricsCAD\<Версия>\ru_RU | \PlotConfig<br>\PlotStyles<br>\Поддержка |
| Mac | /Users/<ИмяПользователя>/Library/ Preferences/Bricsys/BricsCAD/<Версия>/ru_RU | /PlotConfig<br>/PlotStyles<br>/Поддержка<br>/Templates |
| Linux | /home/<ИмяПользователя>/Bricsys/ BricsCAD/<Версия>/ru_RU | /PlotConfig<br>/PlotStyles<br>/Поддержка<br>/Templates |

### 15.2.1 Диспетчер пользовательских файлов BricsCAD

При первом запуске BricsCAD после выполнения его обновления, содержимое папки **UserDataCache**, расположенной в папке установки BricsCAD, сравнивается с содержимым **папок с пользовательскими файлами**. При наличии в этом каталоге измененных пользователем файлов или если один или более соответствующих файлов в каталоге **UserDataCache** являются более новыми, чем устанавливаемые, **Диспетчер пользовательских файлов BricsCAD** отобразит запрос на сохранения ваших файлов или их замену файлами новой версии.

Выполните одно из следующих действий:

1 Выберите необходимый файл в списке файлов, нажмите кнопку **Да** для перезаписи или **Нет** для сохранения выбранного файла.
   - Повторите процедуру для каждого файла из списка.
   - Нажмите кнопку **Да для всех** для перезаписи всех файлов.
   - Нажмите кнопку **Остановить копирование** для сохранения всех файлов.

2 Если Вы не обновите один или более файлов, появится окно с предупреждением и списком файлов, сохраняемых без обновления. Программа предложит напоминать об этом при следующем запуске BricsCAD.

3 Выполните одно из следующих действий:

- Нажмите кнопку **Да**, если вы хотите обновить один из этих файлов позже.

  В этом случае эта процедура будет перезапущена при следующем запуске BricsCAD.

- Нажмите кнопку **Нет**, если вы не хотите получать напоминания.

## 15.3 Loading and unloading CUI files

When you install BricsCAD, the default CUI files are copied to the **Support** folder (see BricsCAD User Files).

When you launch BricsCAD the first time, the **default.cui** file is loaded automatically.

There are two procedures to load a CUI file:

- Load main CUI file: replaces the currently loaded main CUI file.
- Load partial CUI file: partially replaces the content to the currently loaded main CUI file.

### 15.3.1 Loading a main CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 In the **File** menu, choose **Load main CUI file ...** .

The **Choose a customization file** dialog box displays the content of the **Support** folder.

3 Select a CUI file, then double-click or click the **Open** button.

The selected CUI file is loaded.

4 Close the **Customize** dialog.

### 15.3.2 Loading a partial CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 In the **File** menu, choose **Load partial CUI file ...** .

The **Choose a customization file** dialog box displays the content of the **Support** folder.

3 Select a CUI file, then double-click or click the **Open** button.

The content of the selected CUI file is appended: definitions which are different from the ones in the main CUI are used.

4 Close the **Customize** dialog.

***Прим.:*** If a CUIX file is selected in step 2 it will be converted to a CUI file on-the-fly. When a CUIX file contains bitmaps, they are copied into a .RESZ resource file, which are supported as an alternative to resource DLL (zip file containing tool bitmaps; works on all platforms). The following image formats are supported: BMP, JPG, PNG, ICO, GIF, TIFF, XBM, XPM.

### 15.3.3 Unloading a partial CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 Click the **Menus** tab.

3 Right-click the partial CUI name and choose **Unload <PartialCuiName>** in the context menu.

4 Close the **Customize** dialog.

### 15.3.4 Managing customization groups

1 Launch the MENULOAD command.

1 The **Customization groups** dialog opens.

2 (Option) To load a partial CUI file, click the browse button (...) to select a partial CUI file, then click the **Load** button.

3 (Option) To unload a partial CUI file, select the partial CUI file in the **Loaded Customization Groups** list, then click the **Unload** button.

4 Click the **Close** button to stop.

## 15.4 Creating a partial CUI file

1 Launch the CUSTOMIZE command.

2 Do one of the following:

- In the **File** menu, choose **Create new partial CUI file ...**
- Place the cursor on a Menu group, then right-click and choose **Create new partial CUI file ...** in the context menu.

The **Create a Customization File** dialog opens.

By default the new partial CUI file will be created in the **Support** folder.

3 (Option) Select a different folder to save the CUI file in.

4 Type a name for the new CUI file in the **File name** field.

5 Click the **Save** button.

The newly created **Partial CUI file** is loaded and a **Menu Group** of the same name is created in the **Menus**, **Toolbars**, **Ribbon**, **Keyboards**, **Mouse**, **Tablet** and **Quad** tab and in the **Available Tools** pane of the different tabs.

## 15.5 Managing toolboxes

A toolbox contains a set of tools, which can be added to menus, toolbars, ribbon panels, Quad tabs and keyboard or mouse shortcuts. Toolboxes can be created in the main CUI or in a partial CUI.

### 15.5.1 Creating a toolbox in a menu group

1 In the **Customize** dialog box, select a menu group in the **Available Tools** panel:

- BricsCAD (= default main CUI)
- <PartialCuiName> (= partial CUI)

2 Right-click and choose **Append toolbox** in the context menu.

The **Add Toolbox** dialog box appears.

3 Type a name for the new toolbox in the **Name** field, then click the **OK** button.

The toolbox is added in the **Available Tools** tree.

### 15.5.2 Creating a tool in a toolbox

1 In the **Customize** dialog box, select a toolbox in the **Available Tools** panel.

2 Right-click and choose **Append tool** in the context menu.

The **Add tool** dialog box appears.

3 **Title**: type a name for the new tool.

4 **Help**: type a help string.

5 **Command**: type a command or a command macro*.

6 **Image**: Click the **Browse** button (…).

The **Tool Image** dialog box appears.

7 Do one of the following:

- Select **Builtin**, then choose one of the built-in icons.
- Select **Bitmap file**, first choose to select one image file or a small and a large image file, then click the **Browse** button (…) to select the image file(s).

8 Click the **OK** button to create the tool.

* Syntax to create command macros:

- Start the macro with **^c^c**, the equivalent of pressing the Esc key twice, which first ends a running command if necessary.
- Type an underscore (_) in front of a command name if you want to use the macro in a non-English version of BRICSCAD.
- A period (.) in front of a command name ensures that the macro still works if this command is temporarily undefined.
- A minus sign (-) in front of a command launches the Command line version of a command, thus preventing the command dialog window to open.
- A semicolon (;) is the equivalent of pressing the Enter key, e.g. to accept a default prompt option.
- A backslash (\) temporarily pauses the macro to allow user input, e.g. specifying a point.

In the following macro: ^c^c_-layer;m;InteriorWalls;;mline

- **^c^c** concludes a running command, if any.
- **-layer;** launches the -LAYER command.
- **m;** selects the **Make new current layer** option.
- **InteriorWalls** makes the InteriorWalls layer current and creates it first if necessary.
- **;;mline** concludes the -LAYER command and launches the MLINE command.

## 15.6 Creating toolbars

1 In the **Customize** dialog box, click the **Toolbars** tab.

2 Do one of the following:
   - Select a menu group, then right-click and choose **Append toolbar** in the context menu.

     The new toolbar will be added below the last existing toolbar.
   - Select an existing toolbar in a menu group, then right-click and choose **Insert toolbar** in the context menu.

     The new toolbar will be added above the selected toolbar.

   The **Add toolbar** dialog box appears.
3 Type a name in the **Title** field.
4 Click the **OK** button.

   The toolbar is created.
5 Define the properties of the toolbar:
   - **Title**: The toolbar name must include alphanumeric characters with no punctuation other than a dash (–) or an underscore (_). The **Title** along with the **Alias** enables the toolbar to be referenced programmatically.
   - **Position**: Defines the display position of the toolbar when the CUI file is loaded.

     The options are: **Floating**, **Top**, **Left**, **Bottom** or **Right**. The latter four refer to the window edge where the toolbar will be docked.
   - **Display**: Defines whether the toolbar appears or not when the CUI file is loaded.
   - **Rows**: Specifies the number of rows when its position is floating. Toolbars with controls cannot have multiple rows.
   - **Xval**: When the position is Floating, specifies the X coordinate in pixels, measured from the left edge of the screen to the left side of the toolbar.
   - **Yval**: When the position is Floating, specifies the Y coordinate in pixels, measured from the top edge of the screen to the top of the toolbar.
6 Add items to the toolbar.

### 15.6.1 To add a flyout to a toolbar

1 In the **Customize** dialog box, click the **Toolbars** tab.
2 Do one of the following:
   - Select a toolbar, then right-click and choose **Append Flyout** in the context menu, to add the flyout at the end of the toolbar.
   - Expand the toolbar, right-click a tool then choose **Insert flyout** in the context menu, to add the flyout above the selected tool.
3 The **Add flyout** dialog box appears.
4 Type a name in the **Title** field of the **Add flyout** dialog.
5 Click the **OK** button.

   A new empty toolbar is created and inserted as a flyout in the selected toolbar.

6 Define the flyout properties of the toolbar.
   - **Title**: The toolbar name must include alphanumeric characters with no punctuation other than a dash (–) or an underscore (_). The **Title** along with the **Alias** enables the toolbar to be referenced programmatically.
   - **Target alias**: Identifies the referenced toolbar.
   - **Own Icon**:
   - **No**: The icon of the first tool or the icon of the most recently used tool is displayed in the parent toolbar.
   - **Yes**: The selected icon is displayed in the parent toolbar.
7 Define the properties of the toolbar when the flyout is teared off and inserted as a floating toolbar.
8 Add items to the toolbar.
9 Items added to the flyout are automatically added to the toolbar and vice versa.

## 15.7 Creating menus

Main menus sit in the menu bar at the top of the BricsCAD application window. Submenus show as a flyout of a main menu.

The MENUBAR system variable controls the display of menus.

### 15.7.1 Creating a main menu

1 In the **Customize** dialog, click the **Menus** tab.
2 Do one of the following:
   - Click the **Main Menus** subgroup of a menu group, then right-click and choose **Append main menu** in the context menu.

     The new menu will be added below the last existing menu.
   - Select an existing menu in the **Main Menus** subgroup, then right-click and choose **Insert main menu** in the context menu.

     The new menu will be added above the selected menu.

   The **Add Main Menu** dialog box appears.
3 Type a name in the **Title**.
4 Click the **OK** button.

   The main menu is created.
5 Add items to the menu.

***Прим.:*** The new main menu is not visible in the menu bar until at least one menu item is added to it.

### 15.7.2 Adding a submenu to a main menu

1 In the **Customize** dialog, click the **Menus** tab.

   If collapsed, expand the menu group.

   If collapsed, expand the **Main menus** group of the menu group.

2 Do one of the following:
   - Select a main menu, then right-click and choose **Append submenu** in the context menu.
     The new submenu will be added below the last existing submenu.
   - Select an existing submenu, then right-click and choose **Insert submenu** in the context menu.
     The new submenu will be added above the selected submenu.

   The **Add submenu** dialog box appears.
3 Type a name in the **Title** field.
4 Click the **OK** button.
   The submenu is created.
5 Add items to the submenu.

### 15.7.3 Adding a submenu to a submenu

1 In the **Customize** dialog, click the **Menus** tab.
2 Expand the submenu.
3 Select an item in the submenu, then right-click and choose **Insert submenu** in the context menu.
   The **Add submenu** dialog box appears.
4 4. Type a name in the **Title** field.
5 Click the **OK** button.
   The submenu is created.
6 Add items to the submenu.

## 15.8 Creating context menus

The availability of context aka shortcut menus is controlled through the SHORTCUTMENU system variable. There are three context menu modes:

- **Default mode**: Displays the **Default** context menu when no command is active and nothing is selected.
- **Edit mode**: Displays the **Edit** context menu when a selection set exists. This mode is context sensitive. If a specific shortcut menu for the selected entity or entities exists, its content is added to the **Edit** context menu. For example, the content of the **Multiline Text Object** context menu is available in the **Edit** context menu when an MText entity is selected.
- **Command mode**: Displays the **Command** context menu when a command is running.

Shortcut menus appear when you right-click. However:

- When bitcode 16 of the SHORTCUTMENU system variable is set, a right-click repeats the previous command and you need to hold down the right mouse button longer (= long right-click) to display shortcut menus.
- When bitcode 4 of the QUADDISPLAY system variable is set (= Show Quad on right-click), shortcut menus only show when bitcode 16 of the SHORTCUTMENU system variable is set.

Special shortcut menus are:

- **Grips**: Appears on a right-click when using grips to edit an entity.
- **Entity Snap**:
  - MBUTTONPAN = OFF: appears when you click the mouse wheel (middle mouse button).
  - MBUTTONPAN = ON: displays on Shift + Right-Click

### 15.8.1 Creating shortcut menus

1 In the **Customize** dialog, click the **Menus** tab.

2 Do one of the following:
  - Click the **Context Menus** subgroup of a menu group, then right-click and choose **Append context menu** in the context menu.

    The new context menu will be added below the last existing context menu.
  - Select an existing context menu in the **Context Menus** subgroup, then right-click and choose **Insert context menu** in the context menu.

    The new context menu will be added above the selected context menu.

  The **Add Context Menu** dialog box appears.

3 Type a name in the **Menu title** field.

4 (option) To create a regular context menu:
  - Choose **As regular context menu**.
  - Click the **Add Context…** button.

    The **Add context alias** dialog box appears.
  - Select a context alias. The options are: CMCOMMAND, CMDEFAULT, CMEDIT, GRIPS and SNAP.
  - This controls when the context menu is available. See above.
  - (option) add another context alias.
  - Click the **OK** button to create the context menu.

5 (option) To create a context sensitive context menu:
  - Choose **As context menu on specific entities**.
  - Click the **Add Entity Types…** button.

    The **Add Entity Alias** dialog appears.
  - Tick the **One entity** and/or **Multiple entities** boxes.
  - Select one or more entities in the list, then click the **OK** button.
  - (option) Click the **Add Custom…** button add a custom alias.
  - Click the **OK** button to create the context menu.

6 Add tools to the context menu.

## 15.9 Creating keyboard shortcuts

1 In the **Customize** dialog box, click the **Keyboard** tab.

2 Do one of the following:
   - Select a menu group, then right-click and choose **Append shortcut** in the context menu.

     The new shortcut will be added below the last shortcut.
   - Select an existing shortcut in a menu group, then right-click and choose **Insert shortcut** in the context menu.

     The new shortcut will be added above the selected shortcut.

   The **Add keyboard shortcut** dialog box appears.
3 Do one of the following:
   - Select **Add an existing tool**, then select an existing tool.
   - Select **Add a new tool**, then create a new tool.

     See the **Create a tool in a toolbox** procedure in **Managing Toolboxes**.
4 In the **Properties** grid of the new shortcut, select the **Key** field, then press the key combination you want to assign to the shortcut.

## 15.10 Customizing the Quad

To open the **Quad** tab of the **Customize** dialog box, do one of the following:

- Right-click the **QUAD** field in the **Status** bar and choose **Customize Quad items...** in the context menu.
- Run the CUSTOMIZE command, then select the **quad** tab.

### 15.10.1 Creating a quad tab

1 Do one of the following:
   - Right-click a menu group and choose **Append quad tab** in the context menu.

     The quad tab will be added below the last existing quad tab in the menu group.
   - Right-click an existing quad tab and choose **Insert quad tab** in the context menu.

     The quad tab will be added above the selected quad tab.

   The **Add Quad Tab** dialog box appears.
2 Type a name in the **Label** field.

   The label appears in the Quad cursor menu.
3 Type a name in the **Title** field.

   The tab title must be unique in the menu group.
4 Click the **OK** button.
5 Add tools to the tab.

### 15.10.2 Copying a quad tab

You can insert a copy of a quad tab in the same menu group or to another menu group.

- To insert a copy of a quad tab, select the tab, then right-click and choose **Insert Copy** in the context menu.

The **Add Quad Tab** dialog box appears.

Specify a label and title (see above), then click the **OK** button.

- To insert a copy of quad tab in another menu group: drag-and-drop the quad tab onto the target menu group.

### 15.10.3 Setting the Entity Filter property of a tool

The Quad cursor menu is context sensitive. The available tools depend on the entity under the cursor or the current selection set. When no selection exists, the 'No Selection Quad' appears on a right-click if bitcode 4 of the QUADDISPLAY system variable is selected.

1 Select the tool.

1 The **Quad Button** properties are displayed in the bottom section of the **Customize** dialog box.

2 Select the **Entity Filter** property.

The default setting is NO_SELECTION.

3 Click the **Browse** button (…) at the right hand side of the field.

The **Edit Quad Button Filter** dialog box appears.

4 (option) Click the **Add Filter…** button.

The **Add quad button filter** dialog box appears.

Click the down arrow at the right hand side of the field and choose a filter:

- NO_SELECTION: available when nothing is selected.
- OBJECT_ANY: available when a single entity is selected.
- OBJECTS_ANY: available when multiple entities are selected.

Click the **OK** button.

5 (option) Repeat the previous step to add a filter or remove a filter.

6 (option) Click the **Add Entity Type(s)…** button.

The **Add Entity Alias** dialog box appears.

- Tick the **One entity** and/or **Multiple entities** boxes.
- Tick the box the entity type(s) the tool applies to.

Click the **OK** button.

***Прим.:*** It is not possible to modify the Entity Filter of a built-in Quad Button.

## 15.11 Adding tools to user interface controls

Tools can be drag-and-dropped from the toolboxes in the **Available Tools** section of the **Customize** dialog box onto menus, context menus, toolbars, rows in a ribbon panel, keyboard shortcuts and Quad tabs.

1 Select the target in the **Customize** dialog box.

2 In the **Available Tools** section, expand the source toolbox.

3 Select the tool and drag-and-drop it on the target item.

4 (option) Expand **Controls** in the **Available Tools** section, then drag-and-drop a control onto the target item.

5 (option) Drag-and-drop tools to reposition them or to move them to a different target item.

## 15.12 Настройка ленты

Лента состоит из набора закладок, которые, в свою очередь, состоят из набора панелей. Панели ленты включают кнопки инструментов, элементы управления и вложенные панели.

Закладки ленты определяют набор панелей, отображаемых на ленте.

Для отображения закладки на ленте необходимо добавить эту закладку в рабочее пространство.

### 15.12.1 Создание закладки ленты

1 В диалоговом окне **Настройка** щелкните на закладке **Лента**.

2 Выполните одно из следующих действий:

- Раскройте группу меню, щелкните правой кнопкой мыши на узле **Закладки ленты** и в открывшемся контекстном меню выберите **Добавить закладку на ленту**.

  Новая закладка ленты будет добавлена под последней существующей закладкой.

- Щелкните правой кнопкой мыши на имени существующей закладки ленты и в открывшемся контекстном меню выберите **Вставить закладку ленты**.

  Новая закладка ленты будет добавлена над выбранной закладкой.

Откроется диалоговое окно **Добавить закладку на ленту**.

3 Определите свойства закладки ленты:

- **Надпись**: Имя закладки, отображаемое на ленте.
- **Наименование**: Полное имя закладки.
- **ID**: Уникальное имя, которое используется для идентификации закладки ленты в группе меню. Этот идентификатор используется для программной ссылки на закладку ленты.

4 Нажмите кнопку **OK** для создания закладки ленты.

### 15.12.2 Создание панели ленты

1 В диалоговом окне **Настройка** щелкните на закладке **Лента**.

2 Выполните одно из следующих действий:

- Раскройте группу меню, щелкните правой кнопкой мыши на узле **Панели ленты** и в открывшемся контекстном меню выберите **Добавить панель на ленту**.

  Новая панель ленты будет добавлена под последней существующей панелью.

- Щелкните правой кнопкой мыши на имени существующей панели ленты и в открывшемся контекстном меню выберите **Вставить панель ленты**.

  Новая панель ленты будет добавлена над выбранной панелью.

Откроется диалоговое окно **Добавить панель на ленту**.

3 Определите свойства панели ленты:

- **Надпись**: Имя панели, отображаемое на ленте.

- **Наименование**: Полное имя панели.
- **ID**: Уникальное имя, которое используется для идентификации панели ленты в группе меню. Этот идентификатор используется для программной ссылки на панель ленты.

4 Нажмите кнопку **OK** для создания панели ленты.

### 15.12.3 Добавление панели ленты на закладку

1 В диалоговом окне **Настройка** щелкните на закладке **Лента**.

2 Раскройте узел **Закладки ленты**.

3 Выполните одно из следующих действий:

- Щелкните правой кнопкой мыши на закладке ленты и в открывшемся контекстном меню выберите пункт **Добавить панель**.

  Новая панель ленты будет добавлена под последней существующей панелью.
- Раскройте узел закладки ленты, щелкните правой кнопкой мыши на существующей панели ленты и в открывшемся контекстном меню выберите пункт **Вставить панель**.

  Новая панель ленты будет добавлена над выбранной панелью ленты.

Откроется диалоговое окно **выбора панели ленты**.

В этом диалоговом окне перечислены все панели ленты, имеющиеся в группе меню.

4 Выберите в списке необходимую панель, дважды щелкните на ней или нажмите кнопку **OK**.

Выбранная панель будет добавлена на закладку ленты.

### 15.12.4 Добавление строки на панель

1 В диалоговом окне **Настройка** щелкните на закладке **Лента**.

2 Раскройте узел **Панели ленты**.

3 Выберите панель и выполните одно из следующих действий:

- Щелкните правой кнопкой мыши и выберите в контекстном меню пункт **Добавить строку**.

  Строка будет добавлена под последней существующей строкой выбранной панели.
- Щелкните правой кнопкой мыши на существующей строке и в открывшемся контекстном меню выберите **Вставить строку ленты**.

  Строка будет добавлена над выбранной строкой.

### 15.12.5 Добавление элементов в строку

При щелчке правой кнопкой мыши на строке в контекстном меню доступны следующие опции:

- **Добавить панель строк**: Добавляет вложенную панель, которая по умолчанию состоит из трех строк. Вы можете добавить в эти строки один или несколько элементов или добавить/удалить выбранные строки.
- **Добавить кнопку команды**: См. **Добавление инструментов в пользовательский интерфейс**

- **Добавить раскрывающуюся кнопку**: Добавляет раскрывающуюся кнопку, к которой вы можете добавить несколько элементов. На ленте будет отображаться только первый или последний из использованных элементов.
- **Добавить переключатель**: Добавляет кнопку для переключения параметра или опции, например, объектная привязка или параметры выбора объектов. Кнопка-переключатель отображает текущее состояние параметра или опции.
- **Добавить разделитель**: Добавляет пробел между строками.

### 15.12.6 Определение свойств элемента

Выберите элемент и определите его свойства в таблице в нижней части диалогового окна **Настройка**.

### 15.12.7 Изменение положения элементов

Чтобы изменить положение любого элемента на панели или в строке, выберите элемент, затем нажмите и удерживайте левую кнопку мыши и переместите элемент на новую позицию. Отпустите кнопку мыши, чтобы разместить элемент в выбранной позиции.

### 15.12.8 Удаление объектов

Щелкните правой кнопкой мыши на выбранном элементе и в открывшемся контекстном меню выберите **Удалить**.

## 15.13 Tool palettes

On a tool palette you can add blocks, hatches and commands in a tabbed window.

Drag a tool from the palette to the graphic screen to launch the related tool.

### 15.13.1 Opening the Tool Palettes panel

Do one of the following:

- Hover over a toolpanel (Windows) or an empty toolpanel area (Windows and Linux), then right-click and choose **Palettes** > **Tool Palettes** in the context menu.
- Launch the TOOLPALETTES command.

### 15.13.2 Setting the view options

1. Right-click an empty space of the icon area on the **Tool Palettes** panel and choose **View Options...** in the context menu.

   The **View Options** dialog appears.
2. Drag the **Image Size** slider to adjust the image size.
3. Set the viewstyle: **Icon with text**, **Icon only** or **List view**.
4. Choose either **Current Palette** or **All Palettes** from the **Apply to** list button.
5. Click the **OK** button to confirm.

### 15.13.3 Customizing tool palettes

***Прим.:*** The default folder for tool palettes is defined by the TOOLPALETTEPATH system variable.

### 15.13.4 Opening the Customize dialog box

1 Move the cursor to the icon area of the **Tool Palettes** panel, then right-click.

A context menu appears.

2 Choose **Customize Palettes** in the context menu.

The **Customize** dialog box appears.

### 15.13.5 Importing a tool palette

1 In the **Palettes** pane, right-click and choose **Import** in the context menu.

The **Import Palette** dialog box appears.

2 Select a folder, then select a palette file (*.atc, *.btc or *.xtp).

3 Click the **Open** button or double-click to import the palette.

The palette is added in the **Palettes** list.

4 Click the **Close** button to close the **Customize** dialog box.

### 15.13.6 Creating a palette group

1 Place the cursor on the **Palette Groups** root folder name.

2 Right-click and choose **New Group** in the context menu.

A new group folder is created.

3 Type a name for the new group.

### 15.13.7 Creating a palette subgroup

1 Place the cursor on an existing group name.

2 Right-click and choose **New Palette** in the context menu.

A new group folder is added as a subgroup of the selected group.

3 Type a name for the new group.

### 15.13.8 Adding a palette to a group

1 Place the cursor on a palette.

2 Select a palette, then drag the palette onto a palette group name.

### 15.13.9 Removing a palette from a group

1 Select the palette in the group.

2 Right-click and choose **Remove** in the context menu.

### 15.13.10 Setting a palette group current

1 Select the palette group in the palette groups tree.

2 Right-click and choose **Set Current** in the context menu.

### 15.13.11 Deleting a palette group

1 Select the palette group in the palette groups tree.

2 Right-click and choose **Delete** in the context menu.

### 15.13.12 Renaming a tool palette

1 Right-click tool palette tab and choose **Rename Palette** in the context menu.

2 Type a new name for the palette in the **Name** field.

3 Press Enter to confirm.

### 15.13.13 Deleting a tool palette

1 Right-click tool palette tab and choose **Delete Palette** in the context menu.

2 Click the **OK** button on the **Confirm Palette Deletion** box.

### 15.13.14 Managing tools

1 Right-click the tool.

1 A context menu appears.

2 Choose an option:

- **Cut**: Removes the item from the palette. Open another panel, then right-click and choose **Paste** in the context menu.
- **Copy**: Copies the tool. Open another panel, then right-click and choose **Paste** in the context menu.
- **Delete**: Deletes the tool. A dialog box appears, asking you to confirm the deletion.
- **Rename**: Type a new name for the tool.
- **Update tool image**: Updates the image if the image source file has been modified.
- **Specify Image...**: Allows you to replace the default image.

  Choose a new image in the **Select Image File** dialog box, then double-click or click the **Open** button.
- **Remove Image...**: Restores the default image.
- **Properties**: When you click an item on a tool palette, a command is launched. The properties of the tool palette item control the execution of the command.

  The available properties on the **Tool Properties** dialog box depend on the tool type: Command, Hatch, Block or Component.

  Edit the tool properties, then click the **OK** button.

### 15.13.15 Creating a new tool palette

1 Move the cursor to the icon area of the **Tool Palettes** panel, then right-click.

A context menu appears.

2 Choose **New Palette** in the context menu.

A new tab is added.

3 Type a name for the new palette in the **Name** field.

### 15.13.16 Adding items to a tool palette

The following items can be added to a tool palette:

- Hatch patterns
- Blocks
- Components
- Commands

### 15.13.17 Adding hatch patterns

1 Select the palette you want to add a hatch pattern to in the **Tool Palettes** panel.
2 Open the **Hatch and Gradient** dialog box.
3 Select either **Predefined** or **Custom** from the **Type** list button.
4 Double-click the **Swatch** field on the **Hatch and Gradient** dialog box.

The **Hatch Pattern Palette** appears.

5 Select the hatch pattern you want to add, then right-click and choose **Add to Current Tool Palette**.

The hatch pattern is added on the selected tool palette.

6 (option) Repeat steps 4 and 5 to add more hatch patterns.
7 (option) Drag-and-drop patterns to change their order on the palette.

### 15.13.18 Adding blocks

Blocks can be added from:

- the current drawing.
- any drawing.
- all drawings in a selected folder.

### 15.13.19 Adding blocks from the current drawing

1 Open the **Drawing Explorer** dialog box.
2 Select **Blocks**.
3 Select the block(s) you want to add. Hold down the Ctrl key to select multiple blocks.
4 Right-click and choose **Add to Current Tool Palette** in the context menu.

### 15.13.20 Adding blocks from any drawing

1 Open the **Drawing Explorer** dialog box.
2 Select **Folders** in the left hand pane.
3 If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.
4 Select the drawing in the **Local Folders** list.

5 Click the **Expand** icon (+) of the containing drawing and click the **Blocks** icon.

All blocks in the selected drawing are displayed in the **Details** pane.

6 Select the block(s) you want to add. Hold down the Ctrl key to select multiple blocks.

7 Right-click and choose **Add to Current Tool Palette** in the context menu.

### 15.13.21 Adding a drawing as a block

1 Open the **Drawing Explorer** dialog box.

2 Select **Folders** in the left hand pane.

3 If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.

4 Select the drawing in **Local Folders** list.

5 Right-click and choose one of the following in the context menu.

- **Add to Current Tool Palette as block**.

  Blocks are inserted using the INSERT command.

- **Add to Current Tool Palette as component**.

  Components are inserted using the BMINSERT command.

### 15.13.22 Adding all blocks from multiple drawings

This procedure collects all blocks in each of the drawings of the selected folder and adds them to a palette of the drawing name. New palettes are created if necessary.

1 Open the **Drawing Explorer** dialog box.

2 Select **Folders** in the left hand pane.

3 If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.

4 Select the folder, then right-click and choose **Add blocks to Tool Palettes** in the context menu.

### 15.13.23 Adding commands

1 Open the **Customize** dialog box.

2 In the **Available Tools** pane, on the **Customize** dialog, expand a menu group (e.g. BricsCAD).

All toolboxes in the selected menu group are listed.

3 Expand a toolbox.

4 All commands in the selected toolbox are listed.

5 Right-click a tool and choose **Add to Current Tool Palette** in the context menu.

## 15.14 Command aliases

Command aliases are alternative names for BricsCAD commands. You can launch the LINE command by entering L at the Command line. There is no need to type the complete name. After typing the command alias the command is executed. Some commands have more than one alias: e.g. REC, RECT and

RECTANGLE are all aliases for the RECTANG command, whereas the last alias is even longer than the command name. In general, command aliases can increase your speed in entering the commands using the keyboard.

Command aliases are saved in a *.pgp file (P roG ram P arameters file). By default the **default.pgp** file in the **Support** folder of the Roamable Root folder is used.

If a command can be executed transparently (= while another command is running), the alias can also be used transparently. To execute a command or an alias transparently, precede the command name or alias with an apostrophe (').

***Прим.:***

- Unlike commands, aliases are only local and cannot be used as global commands preceded by a underscore (_).
- Command aliases cannot be used in scripts (*.scr).

### 15.14.1 Editing the alias file in a text editor

In order to modify existing aliases or to create new ones, the **default.pgp** file can be edited using an ASCII text editor (e.g. Notepad).

The first section of the **default.pgp** file defines the **Shell** commands. The second section of the **default.pgp** file defines command aliases.

The following syntax is used to define a command alias:

**alias, *command**

**alias** = the character string to be entered at the command prompt. The command alias must be followed by a comma (,).

**command** = the BricsCAD command. The command name must be preceded by an asterisk (*).

Comment lines must be preceded by a semicolon (;). Comment lines can be used to add additional information, e.g. when the file was last edited and by whom.

Some commands, such as HATCH or BOUNDARY, can be preceded by a hyphen (-) to access the Command line version of the command instead of the dialog box version. Command aliases can be created for these commands also, e.g.:

**vi, *-view**

Execute the REINIT command to reload the alias file when it was externally edited.

### 15.14.2 Using the Customize dialog to edit the alias file

1 Click the **Command Aliases** tab on the **Customize** dialog.

2 (option) To create a new alias,
   - Do one of the following:
   - Click the **Add** button.
   - Right-click an alias and choose **Add Alias** in the context menu.
   - Right-click a command and choose **Add Alias** in the context menu to create an alias for the selected command.

The **Add Alias** dialog box appears.

- Type the command alias in the **Alias** field.
- Click the down arrow at the right hand side of the **Command** field, then select a command. You can start typing the command name in the **Command** field to easily find the command in the list.
- Click the **OK** button.

3 (option) To edit an existing alias,

- Select the alias in the list.
- Click the **Edit** button, or right-click the alias and choose **Edit Alias** in the context menu.

  The **Edit Alias** dialog box appears.
- Type a new alias in the **Alias** field.

  If the alias already exists, you are prompted to overwrite the existing alias.
- Click the **OK** button.

4 (option) To delete an alias,

- Select the alias in the list.
- Click the **Delete** button, or right-click and choose **Delete** in the context menu.

  A **Confirm** dialog box appears.
- Click the **OK** button on the **Confirm** dialog box to delete the alias.

***Прим.:*** If you use these procedures to edit the .pgp file, it is not necessary to execute the REINIT command.

## 15.15 User profiles

In BricsCAD, you can customize the settings that control your drawing environment, which can then be saved in a **user profile**. You can create multiple user profiles, e.g. a profile for each of your drawing types or workflows. User profiles can also be used when multiple users share the same computer, and each user wants to create their own drawing environment.

User profiles can be exported to a file, which can then be imported on a different computer. This allows you to copy your settings to a new computer or to install the same settings on multiple computers.

If you use third party applications on top of BricsCAD, you might want to create a user profile to use with each of these applications.

The **User Profile Manager** is a standalone tool that lists the existing profiles and provides all tools needed to **Load**, **Create**, **Set Current**, **Copy**, **Rename**, **Delete**, **Export** and **Import** user profiles.

### 15.15.1 Content of a user profile

User profiles save many settings that control the drawing environment. Once you start BricsCAD using a user profile, it automatically tracks and stores changes that you make to your drawing environment, such as the following.

- Main Cui file.

- Current Workspace. BricsCAD has six standard workspaces: Drafting, Drafting (toolbars), Modeling, Modeling (toolbars), Mechanical and BIM.
- Configuration settings: all system variables which are saved in the Registry and user preferences.
- Default plot settings.
- Settings in dialog windows, for example, search options in the **Settings** dialog.
- Project settings.
- Recent paths.
- Status bar settings.
- Properties panel settings.
- Toolbar and menu configuration.
- Worksets.

### 15.15.2 The User Profile Manager

To open the **User Profile Manager** dialog box:

- When BricsCAD is running: launch the PROFILEMANAGER command.
- Execute **profilemanager_app.exe** in the BricsCAD program folder (e.g. *C:\Program Files\Bricsys \BricsCAD Vx*).

In the **User Profile Manager** dialog:

- The **Current Profile** is marked.
- **Create**: Creates a new profile, using default settings.
- **Set Current**: Loads the selected profile.
- **Copy**: Creates a copy of the selected profile in the **Copy profile** dialog box.
- **Delete**: Deletes the selected profile.
- **Export...**: Exports the selected profile to a .arg file.
- **Import...**: Imports a **.arg** file as a new profile.
- **Start**: Starts BricsCAD with the selected user profile. This option is available only when the dialog box is opened using **profilemanager_app.exe** in the BricsCAD installation folder.
- **OK**: Closes the dialog box.

### 15.15.3 Starting BricsCAD with a specific user profile

When you restart BricsCAD, the user profile that was current when you closed the previous session is used.

To create a desktop shortcut to launch BricsCAD using a specific user profile, do one of the following:

- Add the **/P option** to the **Target** settings of the BricsCAD shortcut. To open the shortcut settings dialog: right-click the shortcut icon on the desktop and choose **Properties** in the context menu.

  A typical BricsCAD shortcut could be: **"C:\Program Files\Bricsys\BricsCAD Vx\bricscad.exe" /P UserProfileName**
- Execute **profilemanager_app.exe**, then either double-click a user profile or select a user profile and click the **Start** button.

***Прим.:***

- The **/P** option is case insensitive.
- A space is needed between the file location and /P option.
- When BricsCAD is already running, a second instance of BricsCAD is launched, at least on condition that the SINGLETON variable is OFF. This variable controls whether multiple BricsCAD sessions can run simultaneously. See the Settings dialog page for more information about variables and settings.
- If the user profile in the argument of the **/P** option does not exist, the user profile is created automatically using default hard-coded values for all settings.
- If the user profile name contains spaces, use double quotes: e.g. /P "background white".

## 15.16 Workspaces

The WORKSPACE command sets the current workspace and lets you create, modify and save workspaces.

The WSSAVE command saves the current workspace under a different name, thus creating a new workspace.

The WSSETTINGS command opens the **Customize** dialog box at the Workspaces tab.

Workspaces define settings and preferences regarding a specific working environment.

The following workspaces are available:

- Drafting
- Drafting (toolbars)
- Modeling
- Modeling (toolbars)
- Mechanical
- BIM

The name of the workspace is stored in the WSCURRENT system variable of the current User Profile.

If the GETSTARTED system variable is ON, you can select the workspace on the **BricsCAD Launcher** dialog box on startup.

The availability of a startup option depends your license level.

| Startup Option | Workspace | License Level |
|---|---|---|
| 2D Drafting | Drafting | Lite or higher |
| 3D Modeling | Modeling | Pro or higher |
| Mechanical design | Mechanical | Mechanical or Ultimate |
| BIM | BIM | BIM or Ultimate |

### 15.16.1 Managing workspaces

Workspaces are defined in the **Workspace** tab page on the **Customize** dialog box.

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Expand a workspace.

3 A workspace definition is composed of:

- **Menus**: defines which menus are available.
- **Toolbars**: defines which toolbars are displayed by default.
- **Panels**: controls the visibility of dockable panels.

  Right-click a palette, then choose an option in the context menu:

  • **Show**: the panel is visible.

  • **Hide**: the panel is not visible.

  • **Don't change**: keep the current state.
- **Ribbon**: defines the available ribbon tabs.
- **Quad Tabs**: defines the order and availability of the Quad tabs.
- **On Switch**: defines the system variables and user preferences to be set, and the commands to be executed, when switching to the workspace.

### 15.16.2 Setting the current workspace

Do one of the following:

- Right-click the **Current Workspace** field in the **Status** bar, then select a workspace in the context menu.
- Choose a workspace in the **Workspaces** toolbar.
- Type **WSCURRENT** at the command prompt.

  **You are prompted**: New value for WSCURRENT <"2d DRAFTING">:

  Type the name of the new current workspace, then press Enter.

***Прим.:***

- If the WSAUTOSAVE system variable is ON, upon switching to another workspace, the toolbar positions of the current workspace are saved in the CUI file. The default value for WSAUTOSAVE is ON.
- It is not recommended to switch workspaces if the name of the user profile refers to a workspace.

### 15.16.3 Creating a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Do one of the following:

- Select Workspaces, then right-click and choose **Append workspace** in the context menu.

  The new workspace will be added below the last existing workspace.
- Select an existing workspace, then right-click and choose **Insert workspace** in the context menu.

  The new workspace will be added above the selected workspace.

The **Add workspace** dialog box appears.

3 Type a name in the **Name** field then press Enter or click the **OK** button.

A newly created workspace contains the following items:

- A standard menu set
- A standard toolbar set
- All palettes set to **Don't change**
- An empty ribbon
- A default set of Quad tabs
- An empty **On Switch** sequence

Default properties. See the **Settings** dialog box under **Program Options / Workspace**.

### 15.16.4 Editing a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Expand the workspace.

3 (option) Edit the menu set:

- Change the order: Drag a menu.
- Delete a menu: select the menu, then right-click and choose **Remove** in the context menu.
- Insert a menu:
  - Right-click a menu in the list and choose **Insert menu** in the context menu.

    The **File** menu is inserted by default.
  - In the **Workspace Menu** grid, select the **ID** field and select the desired menu in the list.

4 (option) Edit the toolbar set:

- Change the order: Drag a toolbar.
- Delete a toolbar: select the toolbar, then right-click and choose **Remove** in the context menu.
- Insert a toolbar:
  - Right-click a toolbar in the list, and choose **Insert toolbar** in the context menu.

    The **Standard** toolbar is inserted by default.
  - In the **Workspace Toolbar** grid, select the **ID** field and select the desired toolbar in the list.

5 Edit the properties of the panels:

- Set the desired 'on switch' behavior: **Show**, **Hide** or **Don't change**.
- Select a panel, then edit its properties in the property grid.

6 Edit the composition of the ribbon:

- To add a tab: right-click an existing tab, then choose **Insert tab** in the context menu and select the tab in the **Select Ribbon Tab** dialog.
- To remove a tab: right-click the tab and choose **Remove** in the context menu.
- To change the position of a tab: drag the tab.

7 Edit the Quad settings:
   - To change the order of the Quad command groups: press and hold the left mouse button, then drag a command group to its new position.
   - Right-click a Quad command group, then choose **Switch off** in the context menu to remove the command group from the Quad cursor menu.
   - Right-click a currently switched off Quad command group, then choose **Switch on** in the context menu to add the command group to the Quad cursor menu.

8 Edit the **On Switch** procedure:

   To add a command to the **On Switch** sequence: select the command in the available commands pane and drag it to the **On Switch** node.

### 15.16.5 Deleting a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Right-click the workspace, then choose **Delete** in the context menu.

   A **Confirm** dialog box appears.

3 Press the **Yes** button to delete the selected workspace.

***Прим.:*** Do not delete default workspaces, as they are used in the **BricsCAD Launcher** dialog box. When one of the default workspaces is deleted, use the **Importing a Workspace** procedure below to restore the missing workspace from Default.cui in the .../UserDataCache\Support\en_US subfolder of the BricsCAD installation folder.

### 15.16.6 Importing a workspace

The option in the **File** menu on the **Customize** dialog box can import workspaces from another CUI file into the current main CUI.

1 In the **Customize** dialog box, choose **Import workspaces...** in the **File** menu.

   The **Choose a Customization File** dialog appears.

2 Browse to the folder where the CUI file, you want to import the workspace from, is saved.

3 By default, BricsCAD saves CUI files in the **Support** folder of the Roamable Root folder. See BricsCAD User Files.

4 Select the CUI file.

5 A dialog box appears, listing all workspaces in the selected CUI file.

6 Select the workspace(s) to import in the list, then click the **OK** button.

   Press and hold the Ctrl key to select multiple workspaces.

## 15.17 Express tools

### 15.17.1 About Express tools

Express tools is a collection of productivity tools that can be added to BricsCAD. You might already know these tools from AutoCAD.

The tools are available for you to provide additional options or possibilities but they are unsupported. BricsCAD is not responsible for the successful operation of these tools.

If you want to know more about how these tools work, go to CADwiesel.

### 15.17.2 Adding Express tools to bricsCAD

1 Go to the Bricsys application page.

2 If Express Tools is not present on the application page, enter **Express tools** in the search field.

3 Click the Express Tools tile.

The Express Tools detail page displays.

4 Do the following:

- Select the BricsCAD version.
- Tick the **I agree to the extended privacy statement** box.
- Click the Download button.

You will receive a zip file.

5 Unzip this file to a location of your choice.

6 Without BricsCAD running, double click on the installation file and install the Express Tools.

- Select a language.
- Specify the installation folder or accept the default folder.
- Click the **Install** button.

The Express tools are installed.

The **Express.cui** file is added in the Support folder of the roamable root folder. By default this is *C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\<version>\en_US\Support*

7 The next time you launch BricsCAD, the Express tools are available in:

- The Express menu.
- Four Express tools toolbars.
- The Express Tools tab on the ribbon.

### 15.17.3 Unloading the Express tools

1 Launch the CUIUNLOAD command.

The Customization Groups dialog box displays.

2 Select EXPRESS.

3 Click the Unload button.

### 15.17.4 Reloading the Express tools

Type **expressmenu** in the Command line, then press Enter.

The **Express.cui** file is reloaded as a partial cui and the Express tools menu, toolbars and ribbon tab are available again.

## 15.18 Lisp

How to create LISP routines for drawing setup.

### 15.18.1 BricsCAD Lisp

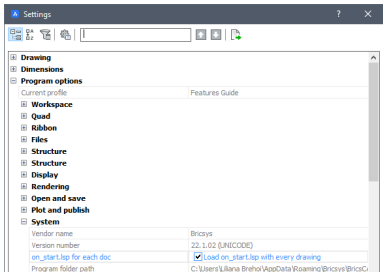
#### Migrating from AutoCAD

When you start migrating to BricsCAD, you'll find that there are virtually no differences in BricsCAD LISP versus OtherLISP. Your code loads and runs, and the functionality is identical. The primary differences when "porting" your apps will be a few minor setup steps, Command line structure, and possibly file locations. To start, let's review the setup and launch of your existing programs in BricsCAD.

If you are auto-launching your programs using **Acad.lsp** or **Acaddoc.lsp**, you will simply need to rename them as **on_start.lsp** or **on_doc_load.lsp** respectively.

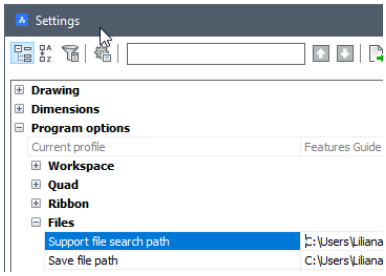
#### BricsCAD Settings Panel

To enable auto-launching your program set the ACADLSPASDOC system variable to 1 or simply toggle the option in *Settings / Program options / System*:




It is also recommended that you spend a little time familiarizing yourself with the Options/Setting menu in BricsCAD, as there are settings and options specific to BricsCAD environment.

The next step in the settings dialog would be to add your support folders to the Files search path, which is found within Settings / Program Options / Files:




It is recommended to use the support paths added as in the manner above and/or located one level below the current project folder. It is a good practice to use a "findfile" function to validate the file exists before proceeding with your commands, as in the method below:

```
(if (findfile "custom-code.lsp") (load "custom-code"))
```

This is one of the methods you may use to demand-load your functions within the **on_start.lsp** file:

```
(defun c:my-function ()                 ; this defines the command
(if (findfile "custom-code.lsp")        ; check if the file exists
(load "custom-code")                    ; if True, launch
(alert "Custom-code not in Support Path!")  ; not True, so alert)
)                                       ; end if statement
)                                       ; end function
```

The command defun (DEfine FUNction) is the same name as the function inside of your **on_start.lsp** file and that function redefines the calling function when it's launched.

## 15.18.2 Drawing Setup Lisp Routines for BricsCAD

### Code sample

The below simple drawing setup routine could be adapted to your needs with minimal changes.

```
(defun C:MYSETUP ( / )                  ;define the command line function "mysetup"
  (setvar "cmdecho" 0)                  ;set display of prompts and input to OFF
  (command "_.audit" "_y")              ;audit current drawing
  (command "_-purge" "_all" "*" "_n")   ;purge all unused styles, etc
  (command "_-layer" "_m" "1" "_c" "1" "" "")  ; create layer "1" color "1"
  (command "_-layer" "_m" "2" "_c" "2" "" "")
  (command "_-layer" "_m" "3" "_c" "3" "" "")
  (command "_-layer" "_m" "4" "_c" "4" "" "")
  (command "_viewres" "_y" "20000")     ;set view resolution to maximum
  (setvar "facetres" 10)                ;set facet resolution to max
  (setvar "filedia" 1)                  ;turn file dialogs on
  (setvar "attdia" 1)                   ;turn attribute dialogs on
  (setvar "attreq" 1)                   ;enable attribute editing on placement
  (setvar "cecolor" "bylayer")          ;set current color bylayer
  (setvar "clayer" "0")                 ;set current layer to "0"
  (setvar "maxsort" 5000)               ;set maximum layer sort to 5,000
  (command "_zoom" "_e" "_zoom" "0.95x")  ;zoom extents, then out a little
  (setvar "cmdecho" 1)                  ;set display of prompts and input to ON
  (prino)
  )
```

Each individual line in this drawing setup routine could have been typed in at the command line in BricsCAD. By wrapping them into a Lisp routine, the **defun** (*define function*) makes the drawing setup routine “callable” from a menu or by typing its name on the BricsCAD command line.

If you don’t include the **defun**, the program will run once, when you load it by its file name. Using the **defun** and loading this routine within your **lsp** will make the function available at any time by typing its function name: **mysetup**.

The commands with the preceding dashes (e.g. **-layer** and **-purge**) are used to prevent the command’s dialog boxes from launching.

***Прим.:*** Always use English command and options names to avoid fails on localized (non-English) BricsCAD versions.

A good practice when opening files from others is to execute an **audit** followed by a **purge** to remove any unused items.

**Purge** command could be run multiple times to remove nested items. See the below example:

```
(repeat 3 (command "-purge" "all" "*" "n"))
```

***Прим.:*** **Purge** is done before adding the new, empty layers, as they would be removed if we did it after.

The **command** calls for setting layers could be combined into a single command statement, but it’s easier to read when each line stands alone.

The **setvar** calls for setting a system variable to a specified value, e.g. style settings for dimensions and text, unit settings, etc.

The **maxsort** setting was introduced for two reasons. First, **Maxsort** controls the maximum number of dialog box entries that will be sorted. The second reason is that the “default value” for **maxsort** is 1,000. With today’s computers and often found larger project files, that number can be too low. If you open drawings where the layer names won’t sort into alphabetical order, the default **maxsort** value could be the reason!

### A more versatile Layer Maker for your drawing setup routine

The following example could be added to your version of mysetup.lsp. By creating a simple text file “Layers.txt” with an editor like Notepad, you can add a list of layer names to be created when the routine is run. This is an alternative to the hard-coded process shown above. By using the findfile function, BricsCAD will first search the current directory, then sequentially through your defined support folders to find the Layers.txt file. You can place your default layer setup text file in the support path and place alternative versions of it in individual project folders.

```
(if (not (findfile "layers.txt"))   ;"not" = looking for no result from findfile
(alert "Can't find layers.txt!")    ;No file? Yell at the user and exit!
(progn                              ;Otherwise, do the stuff in the progn group
(setq layerfile (open (findfile "layers.txt") "r")) ;open layers.txt to "read"
(while (setq layername (read-line layerfile))       ;while reading each line, store the text to layername
(command "-layer" "m" layername "")                 ;make layer with the layername
)                                   ;end the while loop
(close layerfile)                   ;close the layers.txt file
)                                   ;end the progn
)                                   ;end the if
```

# 16. Редактирование объектов

## 16.1 Modifying commands overview

| Icon | Command | Description |
|---|---|---|
|  | 3DARRAY | Creates multiple copies of entities in three dimensions. Entities are arrayed in a three-dimensional rectangular (rows, columns and levels) pattern or a two-dimensional polar (circular) pattern in three-dimensional space. |
|  | ALIGN | Aligns selected entities with other entities in three-dimensional space. |
|  | ARRAY | Creates 2D and 3D arrays of entities in rectangular or polar patterns, or along a path. |
| | ARRAYCLASSIC | Creates multiple copies of entities in one of two symmetrical patterns: rectangular (rows and columns) or polar (circular). |
|  | ARRAYPATH | Associatively distributes entity copies evenly along a path into multiple rows, and levels. |
|  | ARRAYPOLAR | Associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels. |
|  | ARRAYRECT | Associatively distributes copies of entities into any number of rows, columns, and levels. |
|  | BREAK | Splits an entity into two entities. |
|  | CHAMFER | Creates chamfers at intersections of linear entities, defined by two lengths, or a length and an angle. |
|  | CHANGE | Changes the properties of entities, through the Command line. This command has largely been superseded by the **Properties** panel. |
|  | COPY | Draws duplicates of the selected entities. |
|  | COPYGUIDED | After selecting the entities, their relation to their reference curves will automatically guide you to inserting the copy in similar locations. |
|  | DIVIDE | Places markers (points or blocks) along a selected entity. The markers evenly divide the entity into the specified number of equal parts. |

| Icon | Command | Description |
|---|---|---|
|  | EXPLODE | Reduces complex entities to simpler ones, such as blocks into constituent parts, or polylines into lines and arcs. |
|  | EXTEND | Lengthens lines, arcs, polylines or rays to meet another entity. |
|  | FILLET | Fillets intersections; joins intersecting lines with an arc of any suitable radius. |
|  | FLATTEN | Projects 2D and 3D entities onto the XY-plane of the current view. |
|  | HATCHEDIT | Edits hatches and gradient fills in a dialog box. |
|  | JOIN | Joins lines, lightweight and old-style 2D polylines, 3D polylines, arcs, elliptical arcs, polyarcs, splines, and helices at their common endpoints. |
|  | LENGTHEN | Changes the length of a line, polyline, freehand entity or arc. |
|  | MANIPULATE | Launches the Manipulator widget; rotates, moves, mirrors and scales entities. Keep the Ctrl key pressed while starting an action to manipulate a copy of the selected entities. |
|  | MEASURE | Divides a selected entity into segments by placing markers (points or blocks) at specified intervals along its length or circumference. |
|  | MIRROR | Moves or copies the reflected image of selected entities about a line. |
|  | MIRROR3D | Moves or copies the reflected image of entities about a plane. |
|  | MOVE | Moves entities to another location in the same drawing. |
|  | MOVEGUIDED | After selecting the entities, their relation to their reference curves will automatically guide you to moving the entities in a similar location. |
|  | OFFSET | Creates a parallel or offset copy of linear entities. |
| | OVERKILL | Deletes duplicate entities and overlapping lines, arcs, and polylines, and unifies partly overlapping or contiguous ones; optionally moves duplicates to a dedicated layer.<br>Combines duplicate blocks; optionally purges duplicate block definitions. |

| Icon | Command | Description |
|---|---|---|
|  | PEDIT | Edits a two-dimensional or three-dimensional polyline, or a polygon mesh. |
|  | PROPERTIES | Opens the BricsCAD **Properties** panel. |
|  | ROTATE | Rotates entities around a specified point. |
|  | ROTATE3D | Rotates entities around an axis. |
|  | SCALE | Changes the scale of existing entities, either enlarging them or reducing them proportionately in X, Y, and Z directions. |
| | SIMPLIFY | Reduces the number of vertices of polylines and hatch boundaries without changing their general shape. Simplified entities are easier to manipulate and can significantly reduce file size. |
|  | SPLINEDIT | Edits a two-dimensional or three-dimensional spline. |
|  | STRETCH | Moves a portion of a drawing while retaining connections to other parts of the drawing. |
|  | TRIM | Erases the portions of selected entities that cross a specified boundary. |

## 16.2 Entity modification settings

Entity modification settings are found in the **Settings** dialog box under **Drawing** > **Drafting** > **Entity modification**.

## 16.3 Match properties

The MATCHPROP command (short for "match properties") assigns the properties of one entity to other entities.



### 16.3.1 Copying properties between entities

1 Launch the MATCHPROP command.

2 **You are prompted**: Select entity to copy properties from:

3 Select the source entity.

4 **You are prompted**: Active Settings: 'Color' 'Layer' 'Linetype' 'Linetype Scale' 'Lineweight' 'Thickness' 'Plot Style' 'Dimension' 'Polyline' 'Material' 'Text' 'Shadow Display' 'Hatch' 'Multileader' Select entities to copy properties to or [Settings/selection options (?)]:

5 (option) Choose the **Settings** option, then press Enter.

Adjust the settings, then click the **OK** button.

6 Select the target entities.

7 **You are prompted**: Select entities to copy properties to or [Settings/selection options (?)]:

8 Keep selecting entities or right-click to stop.

### 16.3.2 Copying properties between entities with BIM data

Whether the MATCHPROP command copies BIM data depends on the value of the BIMMATCHPROP system variable.

- **BimMatchProp = 0**: Match standard properties

  Only standard properties will be copied, according to the currently selected options (see above).

- **BimMatchProp = 1**: Match standard and BIM properties

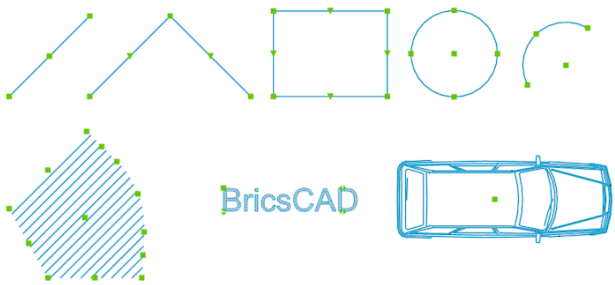
  BIM data, including IFC properties and user defined properties are copied, except:

  - BIM Type*
  - Composition/Profile
  - GUID
  - Quantities

* When using MATCHPROP between a BIM classified entity and an entity without any BIM data, the BIM type and the corresponding BIM properties will be copied along to the unclassified entity, except Composition/Profile, GUID and Quantities.

## 16.4 Grips editing

Grips are displayed as small colored squares at key-points on selected entities.




Whether grips are displayed or not and the appearance of grips are controlled through a series of grips settings in the **Settings** dialog box under **Drawing** > **Drafting** > **Display/Viewing** > **Grips**.

In complex drawings, using too many grips may clutter the display. The GRIPOBJECTLIMIT system variable allows you to suppress the display of grips if the number of selected entities exceeds a certain limit. If set to zero, all grips are displayed.

Grips can be dragged using the mouse in order to stretch, move, mirror, rotate or scale entities.

When the GEOMRELATIONS system variable is set, the geometric relation (perpendicular or tangent) between the entity being edited and other lines, polylines, arcs or circles is indicated (see Geometric Relations).

To select a grip, move the mouse over it, then click. Selected grips are displayed in a different color, which is defined by the HOTGRIP system variable. A selected grip is called a hot grip. By default, hot grips are displayed in red.

The **midpoint** grip on lines and infinite lines, the **startpoint** grip on rays, the **center** grip on circles, arcs, ellipses and elliptical arcs, the grip on points and the **insertion point** grip of texts, mtexts, blocks and Xrefs can be used to quickly move such entities.

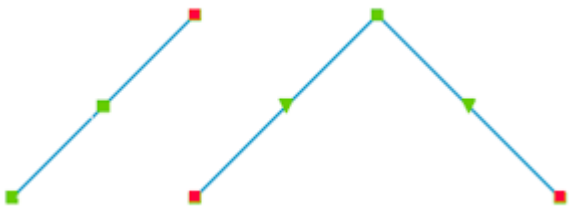
***Прим.:*** When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.

### 16.4.1 Selecting multiple grips

1 Select all entities you want to edit.

2 Press and hold the Shift key, then click the first grip. The selected grip is displayed in red.

3 Keep the Shift key pressed, then click more grips. All selected grips are displayed in red.



4 Release the Shift key, then click the hot grip you want to use as the base point. The hot grips move simultaneously.

### 16.4.2 Grip Edit Command Options

When you start dragging a selected grip, BricsCAD displays a prompt in the command line:

Enter new point [Base point/Copy/Undo/eXit]:

You can choose an option using the keyboard, or in the **Prompt** menu.

| Keyboard | Prompt Menu | Description |
|---|---|---|
| B | Base Point | You are prompted to specify a new base point, other than the selected grip. |
| C | Copy | Copies the selected entities and keeps the originals. By default, multiple copies can be created. Press Enter to stop making copies. |
| U | Undo | If the **Copy** option is chosen, deletes the latest copy. |

| Keyboard | Prompt Menu | Description |
|---|---|---|
| R | Reference | (Applies to Grip Rotate only) Allows you to specify a reference angle. Type the angle in the Command line or specify two points. |
| X | Exit | Aborts the grip edit command. |

### 16.4.3 The grips editing context menu

The grips editing commands can be chosen by repeatedly hitting the space bar or the Enter key, or in a right-click context menu.

The command options are: stretch, move, mirror, rotate, scale.

The stretch command option is selected by default.

### 16.4.4 Stretching entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point.

4 All selected grips move simultaneously.

5 **You are prompted**: Enter new point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

**You are prompted**: Enter base point.

Specify a point.

8 Do one of the following to define the displacement point:

- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

### 16.4.5 Moving entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the move.

4 The **Grip Stretch** command is launched.

5 Do one of the following:

- Right-click, then select **Move** in the context menu.
- Hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Move** is active.

**You are prompted**: Enter move point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Base point** option.

**You are prompted**: Enter base point.

Specify a point.

7 Do one of the following to define the displacement point:
- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

### 16.4.6 Copying entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the move.

4 The **Grip Stretch** command is launched.

5 Do one of the following:
- Right-click, then select **Move** in the context menu.
- Hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Move** is active.

**You are prompted**: Enter move point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Base point** option.

**You are prompted**: Enter base point.

Specify a point.

7 Do one of the following:
- Choose **Copy** in the context menu.
- Type **C**, then press **Enter**.

8 Do one of the following to define the displacement point:
- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

9 Repeat the previous step to create more copies or press the Space bar or Enter to stop.

### 16.4.7 Mirroring entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the mirroring.

4 The **Grip Stretch** command is launched.

5 Do one of the following:
- Right-click, then select **Mirror** in the context menu.
- Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Mirror** is active.

**You are prompted**: Enter second point: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

**You are prompted**: Enter base point.

Specify a point.

8 Specify a second point to define the mirror axis.

9 The selection is mirrored.

### 16.4.8 Rotating entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the rotation.

4 The **Grip Stretch** command is launched.

5 Do one of the following:

- Right-click, then select **Rotate** in the context menu.
- Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Rotate** is active.

**You are prompted**: Enter rotation angle: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

**You are prompted**: Enter base point.

Specify a point.

8 Do one of the following to rotate the selection:

- Specify a point to define the rotation angle.
- Type the rotation angle in the Command line, then press Enter.

### 16.4.9 Scaling entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the scaling.

4 The **Grip Stretch** command is launched.

5 Do one of the following:

- Right-click, then select **Scale** in the context menu.
- Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Scale** is active.

**You are prompted**: Enter scale factor: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

**You are prompted**: Enter base point.

8 Specify a point.

9 Type the scale factor in the Command line, then press Enter to scale the selection.

## 16.5 Selection methods

Before you start to use modification commands, you need to know how to compose a selection set. If the PICKFIRST variable is ON, you can build the selection set before starting the command, else you are prompted to select the entities after you launched the modification command.

Using QUICK SELECT, you can specify filters, such as entity type, color, linetype or any other entity property, to compose a selection set.

The SELECTIONPREVIEW and PREVIEWEFFECT system variables facilitate the selection procedure.

The SELECTIONPREVIEWFILTER system variable allows you to exclude entities from being selected: entities on locked layers, entities in Xrefs, tables, MTexts, hatches, entities in group.

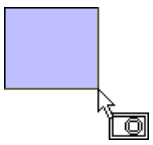
### 16.5.1 Composing a selection set before (pre-pick)

To compose a selection set before launching a modification command do one or more of the following:

- Click entities.
- Selected entity highlight and grips are shown.
- Area selection:
  - Click to define the first corner of a selection area.
  - Move the mouse to the left to define a **Crossing area**:

  - All entities which overlap or are completely inside the rectangle, are added to the selection set.
  - By default, the selection area is displayed in green with a dashed outline.
  - Click to define the first corner of a selection area.
  - Move the mouse to the right to define a **Window area**:

  - All entities which are completely inside, are added to the selection set.
  - By default the selection area is displayed in blue with a continuous outline.
- When working with 3D solids and surfaces, pressing the Ctrl key during window selection allows you to select faces or edges instead of the entire solid or surface. The icon next to the cursor indicates the selection mode:
  - Solids (default)

- Faces
- Edges

Depending on the value of the TIPS system variable, the Hotkey Assistant widget indicates the current selection mode:




- Press an hold the **Shift** key, then use one of the above methods to **remove** entities from the selection set.

***Прим.:***

- The Selection Area system variables control the display of the area selection effects.
- When the PICKADD system variable is **Off**, you cannot add entities to a selection set.
- Press and hold the left mouse button to start window selection when the entity under the cursor highlights. This method is especially helpful for window selection in rendered 3D models.

### 16.5.2 Composing a selection set afterwards (post-pick)

If you launch a modification command when no selection set is active, **you are prompted** to select entities first. BricsCAD provides a range of selection methods to let you compose your selection set easily.

The various selection methods are:

- **Add to set**: Selected entities are added to the selection set.
- **Subtract from set**: Selected entities are removed from the selection set.
- **Picking**: Place the pickbox over a part of the object and click.
- **Select all entities**: Select all entities in the entire drawing, including entities which are on hidden layers. Entities on frozen layers are not selected.
- **Previous selection**: Re-use the previous selection set.
- **Last entity in drawing**: Select the most recent entity.
- **Window**: All entities which are inside, overlap or are completely outside a window are selected. You can choose between a **rectangle**, **polygon** or a **circle**.
- **Fence**: All entities that cross a multi-segment line are selected.
- **Point**: Click a point to select all closed entities which enclose this point.
- **Select by Properties**: Displays the **Select by Properties** context menu to select entities by **Color**, **Layer**, **Linetype**, **Name**, **Thickness**, **Type**, **Value**, **Width** or **Handle**.
- **Undo**: Undoes the last selection action.
- **Location**: Displays the **Select Location** context menu.

### 16.5.3 Selecting overlapping entities

When entities lie on top of each or are obscured by other entities, it is difficult or sometimes impossible to select such entities..

Cycling through the overlapping entities, highlighting each entity one by one, helps you to select such entities.

Make sure SELECTIONPREVIEW is ON and/or the Quad cursor menu is active.

1 Place the cursor over the overlapping entities.

1 One of the entities highlights.

2 Repeatedly hit the Tab key.

3 The overlapping entities highlight one by one.

4 Click when the desired entity highlights.

5 Repeat the previous steps to select more entities.

***Прим.:***

- Do not move the mouse while hitting the Tab key, because this will restart the highlight order.
- Hold down the Shift key to cycle the overlapping entities in reverse order.

### 16.5.4 Selecting subentities

By default the entire entity is selected. This method allows you to select subentities of compound entities, such as polylines, 3D polylines, regions, 3D solids, etc..

1 Place the cursor over the subentity.

2 When the entire entity highlights, hold down the Ctrl key.

3 Click when the subentity is highlighted. The properties of the selected subentity show in the **Properties** panel.

4 (option) Repeat the previous steps to select more subentities.

The following subentities can be selected: polyline segments, vertices and segments of 3D polylines, vertices, edges and faces of regions, surfaces, meshes and 3D solids.

## 16.6 Quick select

Using **Quick Select**, you can specify filters, such as entity type, color, linetype or any other entity property, to compose a selection set.

After composing the filter criteria, you can choose to add the matching entities to a new selection set, add the entities to the current selection set or remove them from the current selection.

**Quick Select** applies to the entire drawing or to a previously created selection.

### 16.6.1 To create a selection set

1 Click the **Quick Select** switch button () on the **Properties** panel.

The **Properties** panel switches to **Quick Select** mode.

If a selection set exists then the **Apply to current selection set** option is checked.

The **Selection list** button indicates the number of entities in the drawing, or the number of selected entities.

2 (option) Click the **Selection list** button, to see a list of all entity types in the current drawing or current selection set.

A list of all entities in the drawing and the number of entities of each type is displayed.

3 Select an entity type in the list, e.g. **Line**.

All Line properties show in the **Properties** panel.

4 Select a property you want to use as a filter, e.g. **Color**.

5 Specify a color.

6 By default all entities of this color will be selected.

7 Optionally, specify a different selection operator.

Click the = sign, then select an operator in the list.



8 Do one of the following:

- Click the **Add to new selection set** button ( ).

  The entities that match the selected criteria are selected.

- Click the **Add to the current selection set** button ( ).

  The entities that match the selected criteria are added to the current selection.

- Click the **Remove from current selection** set button ( ).

  The entities that match the selected criteria are removed from the current selection.

9 (option) Repeat the above procedure to further refine the selection set.

## 16.7 Draw order

When new entities overlap existing entities, they display and print on top of the previously drawn entities. The DRAWORDER command can change the order in which overlapping entities are displayed and printed. You can move entities to the front or the back, or above or under another entity.

1 Launch the DRAWORDER command.

2 **You are prompted**: Select entities to change the draw order:

3 Select the entities, then right-click or press enter.

4 **You are prompted**: Change draw order: Above/Under/Clear/Front/<Back>:

5 Choose an option.

6 Depending on your choice:

- If you have chosen **Above** or **Under**, the Command line reads: Select reference entity:

  Identify the reference entity. The display of the selection set is changed with respect to the

reference entity.

- If you have chosen **Front** or **Back**, the display of the selection set is changed accordingly.

### 16.7.1 Using Draw Order tools

Draw Order tools are available in the **Tools** menu and the **Draw Order** toolbar.

| Icon | Tool Name | Description |
|---|---|---|
|  | Bring To Front | Places the selection set on top. |
|  | Bring Above | Places the selection set above the reference entity. (*) |
|  | Send Under | Place the selection set under the reference entity. (*) |
|  | Send to Back | Moves the selection set to the back. |
|  | Hatch to Back | Sends all hatch entities to the back (HATCHTOBACK command). |
|  | Text and Dimensions to Front | Brings all texts and/or dimension entities to front.<br>Options: **Text / Dimensions / Both** can be chosen either in the Command line or in the prompt menu. The default option is **Both** (TEXTTOFRONT command). |

## 16.8 The manipulator

The manipulator is a versatile grip tool to swiftly move, rotate, scale and mirror selected entities. Holding down the Ctrl key when starting the action keeps the original entities, thus creating copies. Upon selecting an entity, the manipulator is displayed at a user-controlled location and orientation. The location and orientation are controlled in a similar way to a dynamic UCS. When multiple entities are selected simultaneously, the manipulator is displayed at their geometric center. When a selection is built up step by step, the initial location of the manipulator is preserved. The manipulator can be moved to any desired location or orientation using its handles and the context menu options.

Depending on the operation type and if dynamic dimensions are active, a dynamic entry field is available to set the distance, angle or scale factor.

### 16.8.1 Adaptive snap step

The snap step size automatically adapts to the current screen zoom factor if SNAPTYPE= 2. The ADAPTIVEGRIDSTEPSIZE system variable controls the smallest available step size. The adaptive step size is used to dynamically adjust the step size in the Manipulator Ruler, which enables you to quickly modify entities using precise values, without having to key in numbers.

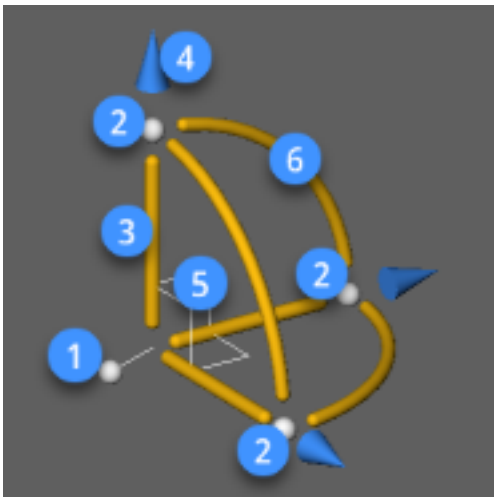
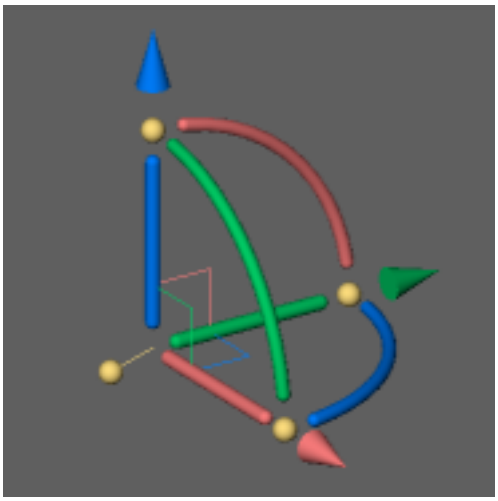
### 16.8.2 The manipulator settings

The manipulator settings can be found in the **Settings** dialog box under **Program Options** > **Manipulator**.

When the Manipulator is displayed, select **Manipulator Settings** in the right-click context menu.

### 16.8.3 The manipulator layout

Check the **Classic Color** option in the manipulator context menu for the classic layout. Uncheck the option for the default layout.

| | |
|---|---|
|  |  |
| Default | Classic |

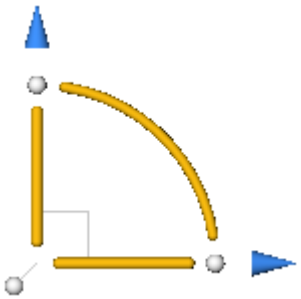
4 handles

- **1**: manipulator handle
- **2**: axis handle

12 command trigger zones

- **3**: axis
- **4**: arrowhead
- **5**: plane swatch
- **6**: rotation arc

When the current view is parallel to one of the main axes of the coordinate system, as in a plan view, the manipulator is simplified to a 2D version.



### 16.8.4 The location and orientation of the manipulator

The initial location and orientation of the 3D manipulator tripod depends on:

- The entity type.
- The point you click on the entity.

| Entity Type | Selection Point | Manipulator - Anchor Point | Manipulator - Orientation |
|---|---|---|---|
| Line | Near endpoint | At endpoint | X-axis through the other endpoint |
| Line | Near midpoint | At midpoint | X-axis through the nearest endpoint |
| Polyline Line Segment | Near vertex | At vertex | X-axis through the other vertex of the selected line segment |
| Polyline Line Segment | Near segment midpoint | At the segment midpoint | X-axis through the nearest vertex |
| Polyline Arc Segment | On the arc segment | At the arc center | X-axis through the nearest vertex |
| Arc | Random | At the arc center | X-axis through the nearest endpoint |
| Circle | Random | At the circle center | X-axis parallel with the X axis of the current coordinate system |
| Spline | Random | At the nearest endpoint | Through the other endpoint |
| Region | On a line segment | At the nearest endpoint of the line segment | X-axis through the other endpoint |

| Entity Type | Selection Point | Manipulator - Anchor Point | Manipulator - Orientation |
|---|---|---|---|
| Region | On an arc segment | At the center of the arc segment | X-axis through the nearest endpoint of the arc segment |
| Text, MText, Block, Xref | Random | At the origin of the entity | X-axis at the rotation angle of the entity |
| 3D Solid<br>3D Solid Face | On a face of the solid | At the nearest vertex | X-axis along the nearest edge<br>XY-plane in the selected face |
| Multiple Entities | Random | At the geometric center of the selection set** | Parallel with the axes of the current coordinate system |

* See Selection Modes to learn how to select a 3D solid face or the entire solid.

** The position of the manipulator does not change when more entities are added to the selection set.

### 16.8.5 Displaying the manipulator

The value of the MANIPULATOR system variable controls when the MANIPULATOR appears automatically:

- 0: Never
- 1: When entities are selected.
- 2: When the left mouse button is hold down longer when selecting an entity.

In order to display the Manipulator when multiple entities are selected:

- Select the entities one by one or use window selection, then long-click to select the last entity.
- Set the value of the MANIPULATOR system variable to 1.
- Compose the selection set, then launch the MANIPULATE command.

### 16.8.6 Moving the manipulator

Do one of the following:

- Long-click anywhere on the manipulator and move it to a new location.
- MANIPULATORHANDLE = 0: Click the manipulator handle (1) and move it to a new location*.
- The anchor handle stands out to avoid overlap with entity grips.
- MANIPULATORHANDLE = 1: Long-click the manipulator handle (1) and move it to a new location*.
- Place the cursor over the manipulator, then right-click and choose **Move** in the context menu and move it to a new location*.

* Specify the new location using entity snaps.

### 16.8.7 Rotating the manipulator

Select one of its axis handles (2) to rotate the manipulator. Pick a point or type a value in the dynamic entry field to specify the rotation angle.

Press the Shift key to rotate the Manipulator 90 degrees about its normal axis (Z-axis).

### 16.8.8 Reorienting the manipulator

1 Place the cursor over the manipulator, then right-click and choose **Reorient Manipulator** in the context menu.

**You are prompted**: Specify origin of manipulator:

2 Pick a point.

3 The anchor point of the manipulator is placed at the specified point.

**You are prompted**: Point on X-axis or <Accept>:

4 Pick a point.

5 The manipulator rotates about its X-axis.

**You are prompted**: Point on the XY-plane with positive Y value or <Accept>:

6 Pick a point to define new position of the manipulator.

### 16.8.9 Aligning the manipulator with a coordinate system

Move the cursor over the manipulator, then right-click and select an option in the context menu:

- **Align with WCS**: the manipulator axes are forced to become parallel with the corresponding axes of the World Coordinate System.
- **Align with UCS**: if a UCS is currently the active coordinate system, the manipulator axes are forced to become parallel with the corresponding axes of the UCS.

### 16.8.10 Aligning the manipulator with the face of a solid

Dynamic UCS (DUCS) must be ON for this operation to succeed.

1 Place the cursor over the manipulator then right-click and choose **Move** in the context menu.

2 Hover over the face of a solid.

3 The XY-plane of the manipulator aligns with the solid face under the cursor.

4 Do one of the following:
   - Snap to a point on the selected face or on one of its edges.
   - Hit the Shift key to lock the XY-plane to the selected face, then snap to a point outside the face.

## 16.9 Moving entities

The MOVE command moves entities in the drawing.

The CUTCLIP command copies entities to the Clipboard, and then erases them from the drawing.

The PASTECLIP command (short for "paste clipboard") pastes entities from the Clipboard into the current drawing.

The PASTEORIG command pastes entities from the Clipboard into another drawing, using the coordinates of the source drawing.

The PASTEBLOCK command pastes entities from the Clipboard as a block into the current drawing.

Use CUTCLIP, then either PASTECLIP, PASTEORIG or PASTEBLOCK to move entities to another drawing.

The **Copy** option of the Grips Editing commands allows you to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

***Прим.:*** When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.

### 16.9.1 Moving entities in a drawing



The default method is to create a selection set and then specify a starting point (base point) and an endpoint (displacement point) to define the relocation of the entities. You can also move the entities using a direction vector.

1 Launch the MOVE command.

1 **You are prompted**: Select entities to move [selection options (?)]:

2 Select the entities, then right-click or press Enter.

3 The selection set is now attached to the cursor.

**You are prompted**: Enter base point [Displacement] <Displacement>:

4 Specify a point.

5 **You are prompted**: Enter second point or <Use base point as displacement>:

6 Specify a point.

7 The selection set is moved.

### 16.9.2 Moving entities between drawings

1 In the source drawing, select the entities you want to move.

2 Do one of the following:

- Right-click, then choose **Cut** in the context menu.

  Hold down the right-mouse button longer when the Quad is active.
- Choose **Cut** in the **Edit** menu.
- Press **Ctrl + X**.

The entities are deleted in the source drawing and copied to the Clipboard.

3 Switch to the target drawing.

4 Do one of the following:

- Choose **Paste** in the **Edit** menu.
- Press **Ctrl + V**.

The bottom left corner of the bounding rectangle of the selection is attached to the cursor.

5 Do one of the following:
   - Specify a point.
   - Press to paste the selection set using the coordinates of the source drawing.

***Прим.:*** Launch the UNDO command in the source drawing bring back the moved entities.

### 16.9.3 Moving a selection as a block

1 Select the entities you want to move.

2 In the source drawing, select the entities you want to move.

3 Do one of the following:
   - Right-click, then choose **Cut** in the context menu.

     Hold down the right mouse button longer when the Quad is active.
   - Choose **Cut** in the **Edit** menu.
   - Press **Ctrl + X**.

   The entities are deleted in the source drawing and copied to the Clipboard.

4 The entities are deleted in the drawing and copied to the Clipboard.

5 (option) Switch to the target drawing.

6 Choose **Paste as a Block** in the **Edit** menu or press **Ctrl + Shift + V**.

   The bottom left corner of the bounding rectangle of the selection is attached to the crosshairs in the target drawing.

7 Specify the insertion point.

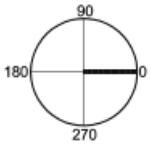
8 The selection is pasted as a block, using a default name.

9 (option) Launch the EXPBLOCKS command and rename the block in the **Drawing Explorer / Blocks** dialog box.

## 16.10 Rotating entities

The ROTATE command rotates entities at a specified point along a specified rotation angle or by an angle referenced to a base angle. The default method rotates the entities using a relative rotation angle from their current orientation.

By default, angles start at 3 o'clock (East) and increase in a counter-clockwise direction. If you want to rotate in a clockwise direction, you can enter a negative value.




***Прим.:***

- The **Angular Base** is controlled by the ANGBASE system variable, which in turn refers to the current UCS.

- The **Angle Direction** is controlled by the ANGDIR system variable, which sets the positive angle direction from angle 0 with respect to the current UCS.



### 16.10.1 General procedure

1 Launch the ROTATE command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

**You are prompted**: Select entities to rotate [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Rotation point:

6 Specify a point.

7 **You are prompted**: Rotation angle or [Base angle/Copy] <0>:

8 Do one of the following:

- Specify a point.
- Type a value and press Enter.

The selection set is rotated.

### 16.10.2 Rotating a selection set relative to a base angle

1 Launch the ROTATE command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

**You are prompted**: Select entities to rotate [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Rotation point:

6 Specify a point.

7 **You are prompted**: Rotation angle or [Base angle/Copy] <0>:

8 Choose the **Base angle** option.

**You are prompted**: Base angle <0>:

9 Do one of the following:

- Type a value and press Enter.
- Click two points to specify the base angle.

10 **You are prompted**: New angle <30.84>:

11 Do one of the following:

- Type a value and press Enter.

- Click a point.

The selection is rotated.

### 16.10.3 Rotating in 3D



1 Launch the ROTATE3D command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

**You are prompted**: Select entities to rotate [selection options (?)]:

4 Select more entities, or press Enter to stop selecting.

5 **You are prompted**: Select axis by [Entity/Last/View/Xaxis/Yaxis/Zaxis/2points] <2points>:

6 Press Enter to accept the 2points method to select the rotation axis.

7 **You are prompted**: Specify the first rotation axis point:

8 Specify a point.

9 **You are prompted**: Specify the second rotation axis point:

10 Specify a point.

11 **You are prompted**: Rotation angle or [Base angle/Copy]:45

12 Type a value and press Enter.

13 The selection is rotated.

## 16.11 Mirroring entities

The MIRROR command creates a mirrored copy of a selection set. The selection is mirrored along a mirror line, which you define by specifying two points. You can choose to either delete or retain the original entities.

The MIRROR3D command creates a mirrored copy of selected entities in three-dimensional space. In this case the selection is mirrored along a mirror plane, which can be defined by either specifying three points, selecting an existing two-dimensional planar entity, aligning the plane parallel to the XY, YZ, or XZ plane of the current UCS, or aligning the plane with the current view. You can choose to either delete or retain the original entities.

***Прим.:*** Whether text is mirrored or not by the MIRROR command is controlled by the MIRRTEXT system variable.

### 16.11.1 Mirroring entities along a line



1 Launch the MIRROR command.

1 **You are prompted**: Select entities to mirror [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

**You are prompted**: Select entities to mirror [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Start of mirror line:

6 Specify a point.

7 The selection set rotates dynamically.

**You are prompted**: End of mirror line:

8 Specify a point.

9 The selection is mirrored.

**You are prompted**: Delete the original entities? [Yes-delete entities/No-keep entities] <No-keep entities>:

10 Decide whether or not to keep the original entities.

### 16.11.2 Mirroring entities along a plane



1 Launch the MIRROR3D command.

1 **You are prompted**: Select entities to mirror [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

**You are prompted**: Select entities to mirror [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Define mirror plane by [Entity/Last/View/Z axis/X-Y plane/Y-Z plane/Z-X plane/3 points] <3 points>:

6 Press Enter to accept the 3 points method to define the mirror plane.

7 **You are prompted**: First point on plane:

8 Specify a point.

9 **You are prompted**: Second point on plane:

10 Specify a point.

11 **You are prompted**: Third point on plane:

12 Specify a point.

13 **You are prompted**: Delete the original entities? [Yes-delete entities/No-keep entities] <No-keep entities>:

14 Decide whether or not to keep the original entities.

## 16.12 Aligning entities

The ALIGN command lets you reposition a selection set with respect to an existing entity in the drawing. The selection set is moved and rotated in a single action. If necessary, you can also scale the selection to fit in its new location.

The command can be used both in a 2D and 3D environment.

### 16.12.1 Aligning in 2D

1 Launch the ALIGN command.

2 **You are prompted**: Select entities:

3 Select the entities, then right-click or press Enter.

4 **You are prompted**: Specify first source point:

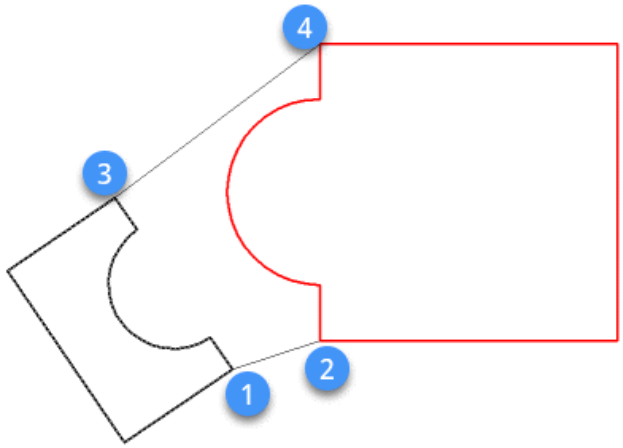
5 Snap to the first source point in the selection set (1).

6 **You are prompted**: Specify first destination point:

7 Snap to the first destination point on the reference entity (2).

8 A witness line is drawn between the source point and the target point.

**You are prompted**: Specify the second source point:




9 Snap to the second source point in the selection set (3).

10 **You are prompted**: Specify second destination point:

11 Snap to the second destination point on the reference entity (4).

12 A witness line is drawn between the source point and the target point.

**You are prompted**: Specify the third source point:

13 Right-click to use the ALIGN command in 2D.

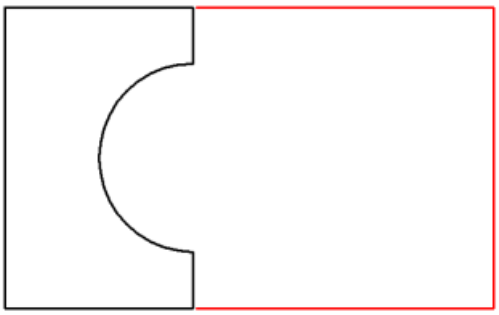
14 **You are prompted**: Scale objects based on alignment points [Yes/No] <No>:

15 Do one of the following:

- Press Enter if you don't want to scale the selection set.

- Type Y and press Enter to scale the selection set.
- Scaling adjusts the distance between the source points to the distance between the target points.

16



Result of the Align procedure with scaled selection set.

### 16.12.2 Aligning in 3D

1 Launch the ALIGN command.

1 **You are prompted**: Select entities:

2 Select the entities, then right-click or press Enter.

3 **You are prompted**: Specify first source point:

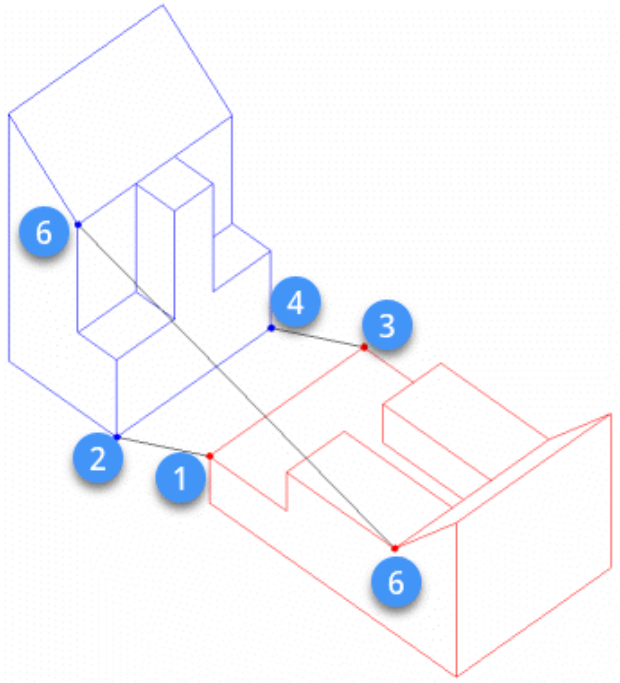
4 Snap to the first source point in the selection set (1).

5 **You are prompted**: Specify first destination point:

6 Snap to the first destination point on the reference entity (2).

7 A witness line is drawn between the source point and the target point.

**You are prompted**: Specify the second source point:




8 Snap to the second source point in the selection set (3).

9 **You are prompted**: Specify second destination point:

10 Snap to the second destination point on the reference entity (4).

11 A witness line is drawn between the source point and the target point.

**You are prompted**: Specify the third source point:

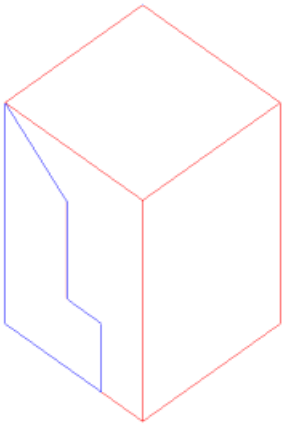
12 Snap to the third source point in the selection set (5).

13 **You are prompted**: Specify third destination point:

14 Snap to the third destination point on the reference entity (6).

15 A witness line is drawn between the source point and the target point.

The entity is aligned.



## 16.13 Changing entities

The CHANGE command lets you change the insertion point and orientation of texts and block inserts.

The **Properties** option of the command can also change the layer, line type, line type scale, line weight, etc. but the BricsCAD Properties panel is much more versatile.



### 16.13.1 Changing text entities

1 Launch the CHANGE command.

1 **You are prompted**: Select entities to change [selection options (?)]:

2 Select the text entities, then right-click or press Enter.

3 **You are prompted**: Change point [special Entities/Properties...]:

4 Specify a new insertion point or press Enter to keep the original.

5 **You are prompted**: New text style Standard:

6 Type the name of a new text style or press Enter to keep the current style.

7 The text entity scales dynamically.

**You are prompted**: New height <current height>:

8 Do one of the following:

- Click to define the new text height.
- Type a value to specify the new text height.
- Press Enter to keep the current text height.

The text entity rotates dynamically.

**You are prompted**: New rotation angle <0>:

9 Do one of the following:

- Specify a point
- Type a value and press Enter.
- Press Enter to keep the current rotation angle.

**You are prompted**: New text <current text>:

10 Type a new text or press Enter to keep the current text.

11 When multiple text entities are selected, the procedure restarts for each entity.

### 16.13.2 Relocating blocks

1 Launch the CHANGE command.

1 **You are prompted**: Select entities to change [selection options (?)]:

2 Select the blocks, then right-click or press Enter.

3 **You are prompted**: Change point [special Entities/Properties...]:

4 Specify a point.

5 The block rotates dynamically.

**You are prompted**: New rotation angle <current angle>:

6 Do one of the following:
- Specify a point.
- Type a value and press Enter.
- Press Enter to keep the current rotation angle.

7 When multiple blocks are selected, the procedure restarts for each entity.

## 16.14 Copying entities

The COPY command duplicates entities within the current drawing using. The default method is to create a selection set and then specify a starting point (base point) and an endpoint (second point) for the copy. You can also specify the displacement as a direction vector (X,Y,Z).

Depending on the COPYMODE system variable, you can create multiple copies of the selection set or just one.

The COPYCLIP command copies entities to the clipboard, which you can then paste in the current drawing or in another drawing using the PASTECLIP or the PASTEBLOCK commands.

***Прим.:*** When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.



### 16.14.1 General procedure

1 Launch the COPY command.

1 **You are prompted**: Select entities to copy [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

4 **You are prompted**: Select entities to copy [selection options (?)]:

Current copy mode: Multiple

**You are prompted**: Enter base point [Displacement/mOde] <Displacement>:

5 Specify a point.

6 The copied selection moves dynamically.

**You are prompted**: Enter second point or [Array] <Use base point as displacement>:

7 Specify a point.

8 A first copy of the selection is created.

A second copy moves dynamically.

**You are prompted**: Enter second point [Undo/Array/Repeat/Exit]:

9 Do one of the following:
- Keep specifying points to create multiple copies.
- Press Enter to stop.

### 16.14.2 Using the Repeat option

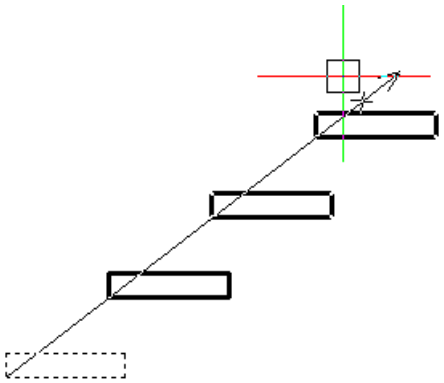
1 Execute steps 1-5 of the general procedure (see above).

1 **You are prompted**: Enter second point [Undo/Array/Repeat/Exit]:

2 Choose the **Repeat** option.

Multiple copies at regular intervals are created when you move the cursor.

**You are prompted**: Select end point (<ENTER> to key in number of repeats):



3 Do one of the following:

- Click when the desired number of copies is displayed.
- Press Enter.
- **You are prompted**: Enter number of repeats <1>:

  Type the desired number of copies, then press Enter.

### 16.14.3 Using the Array option

1 Execute steps 1-4 of the general procedure (see above).

1 **You are prompted**: Enter second point or [Array] <Use base point as displacement>:

2 Choose the **Array** option.

**You are prompted**: Enter number of items to array:

3 Specify the number of items.

4 The array is dynamically displayed.

**You are prompted**: Enter second point or [Fit]:

5 Do one of the following:

- Specify a point to define the distance between each item in the array.
- Choose the **Fit** option.

  **You are prompted**: Enter second point or [Array]:

  Specify a point to define the distance between the first and the last item in the array.

6 Do one of the following:

- Keep creating copies.
- Press Enter to stop.

### 16.14.4 Copying a selection as a block

1 Select the entities you want to copy as a block.

2 Launch the COPYBASE command.

3 **You are prompted**: Select base point:

4 Specify a point.

5 xx selected

6 (option) Switch to another drawing.

7 Launch the PASTEBLOCK command.

8 **You are prompted**: Select insertion point:

9 Specify a point.

10 The selection is pasted as a block.

11 (option) Launch the EXPBLOCKS command and rename the block in the **Drawing Explorer / Blocks** dialog box.

### 16.14.5 Copying entities between drawings

Entities can be copied between drawings using the following methods:

- COPYCLIP and PASTECLIP*: to copy the selection using a default base point.
- COPYCLIP and PASTEORIG: to copy the selection using the coordinates of the source drawing.
- COPYBASE and PASTECLIP*: to copy the selection using a user-defined base point.
- COPYBASE and PASTEBLOCK: to copy the selection as a block in the target drawing.

* Press Enter at the 'Specify insertion point' prompt to paste the selection using the coordinates of the source drawing (= equivalent of the PASTEORIG command).

## 16.15 Making parallel copies

The OFFSET command creates a copy of linear entities and aligns them parallel to the original entities at a specified distance. You can make parallel copies of arcs, circles, ellipses, elliptical arcs, lines, two-dimensional polylines, rays and infinite lines.

Depending on the selected options set by the SELECTIONMODES system variable, the OFFSET command accepts:

- an edge of a 3D solid if the **Select Edges** option is set.
- the outline of a 3D solid face if the **Select Faces** option is set.
- enclosed boundaries if the **Select Detected Boundaries** option is set.

***Прим.:*** The OFFSETGAPTYPE settings variable controls how potential gaps between segments are treated when closed polylines are offset.

### 16.15.1 Making a parallel copy at a specified distance

1 Launch the OFFSET command.

1 **You are prompted**: Enter offset distance or [Through point/Erase/Layer] <Through point>:

2 Do one of the following:

- Press Enter to accept the current distance.
- Type a new distance in the Command line.
- Define a new distance by specifying two points.

**You are prompted**: Select entity/subentity or [Exit/selection options (?)] <Exit>:

3 Select a linear entity.

4 A preview of the parallel copy is displayed depending on the current cursor position with respect to the selected entity.

**You are prompted**: Select side for parallel copy or [Both sides/Multiple]:

5 Click a point.

6 The parallel copy is created.

If OFFSETERASE = ON, the original entity is deleted.

7 **You are prompted**: Select entity/subentity or [Undo/Exit/selection options (?)] <Exit>:

8 Do one of the following:

- Select another entity.
- Press Enter to stop.

### 16.15.2 Making a parallel copy through a point

1 Launch the OFFSET command.

1 **You are prompted**: Enter offset distance or [Through point/Erase/Layer] <Through point>:

2 Do one of the following:

- Press Enter to accept the default **Through point** option (= when last used).
- Choose the **Through point** option.

**You are prompted**: Select entity/subentity or [Exit/selection options (?)] <Exit>:

3 Select a linear entity.

4 The parallel copy is dynamically displayed.

5 Click a point to create the parallel copy.

6 If OFFSETERASE = ON, the original entity is deleted.

**You are prompted**: Select entity/subentity or [Undo/Exit/selection options (?)] <Exit>:

7 Do one of the following:

- Select another entity.
- Press Enter to stop.

## 16.16 Arraying entities

The **ARRAY** commands copy a selection set in a rectangular or polar (circular) pattern or along a linear entity. For a rectangular array, you must specify the number of rows and columns and also the spacing between subsequent rows and columns. To create a polar array, you will be prompted to specify the center point of the array, the rotation step and the number of items in the array or the angle to fill. You can choose to rotate the selection set around the center point or to keep its original orientation.

The ARRAYCLASSIC command creates a static polar or rectangular array of entities through a dialog box.

The -ARRAY command creates a static polar or rectangular array of entities.

The 3DARRAY command constructs static 3D rectangular arrays and rotated polar arrays.

The ARRAY command distributes entity copies in an evenly spaced associative rectangular array (ARRAYRECT command), a polar array (ARRAYPOLAR command) or a path array (ARRAYPATH command) array.

The ARRAYEDIT command allows you to edit associative arrays and their source entities.

### 16.16.1 Creating a static rectangular array

1 Launch the ARRAYCLASSIC command.

1 The **Array** dialog box appears.

2 Under **Array Type**, choose **Rectangular**.

3 Click the **Select Entities** button.

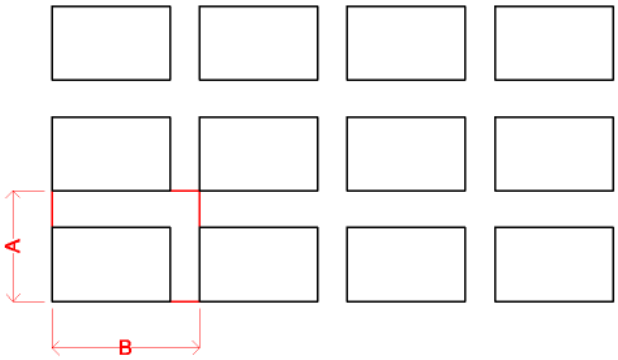
The **Array** dialog box closes temporarily to let you select entities.

4 Select the entities, then right-click or press Enter.

5 The **Array** dialog box reopens.

6 Fill out the **Settings** fields or press the **Pick Points** buttons to define the settings graphically.

The **Array** dialog box closes temporarily to let you pick a point in the drawing.




Row offset (A) and column offset (B) can be defined by the spacing rectangle (red).

7 Click the **OK** button to create the array.

***Прим.:*** Positive values in step 5 are measured along the positive X- and Y-axis of the current UCS. Negative values are measured in the opposite direction.

### 16.16.2 Creating a static polar array

1 Launch the ARRAYCLASSIC command.

1 The **Array** dialog box appears.

2 Under **Array Type**, choose **Polar**.

3 Click the **Select Entities** button.

The **Array** dialog box closes temporarily to let you select entities.

4 Select the entities, then right-click or press Enter.

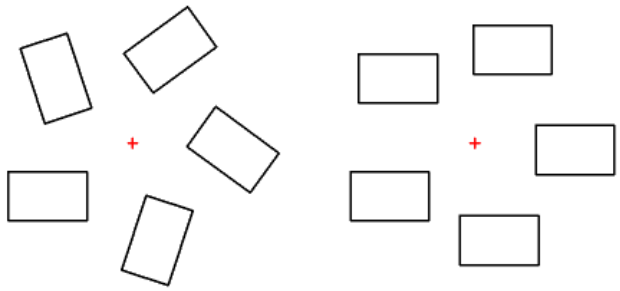
5 The **Array** dialog box reopens.

6 Choose a **Calculate** option: **Angle between**, **Angle to fill** or **Number of items**.

7 Fill out the **Settings** fields or press the **Pick Points** buttons to define the settings graphically.

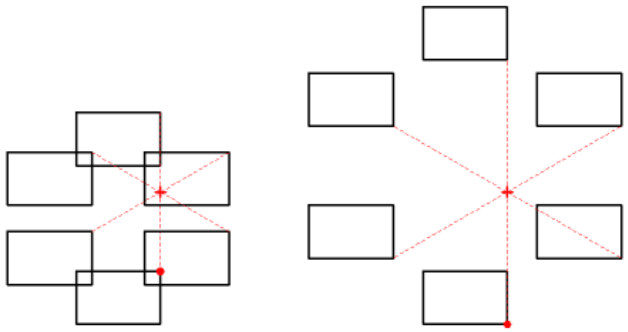
The **Array** dialog box closes temporarily to let you pick a point in the drawing.

8 (option) Check the **Rotate items as copied** option.



Entities rotated (left) or not (right) about the center point of the array.

9 (option) Define an **Entity base point**.



Polar array with base point (dot) and center point (cross).

10 Click the **OK** button to create the array.

### 16.16.3 Creating a static 3D rectangular array

1 Launch the 3DARRAY command.

1 **You are prompted**: Select entities to array [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

**You are prompted**: Select entities to array [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Type of array [Polar/Rectangular] <Rectangular>:

6 Choose the **Rectangular** option.

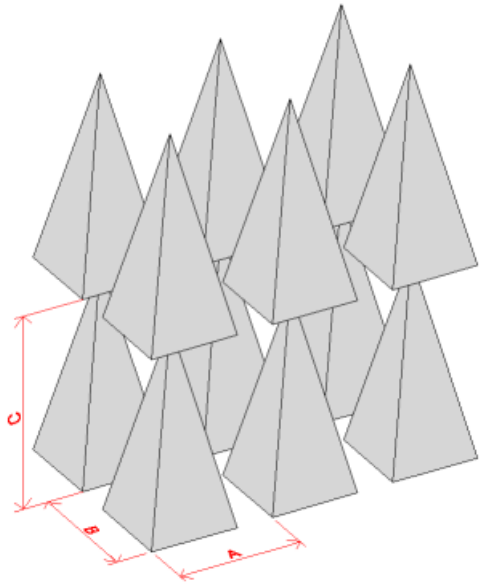
**You are prompted**: Number of rows in the array <1>:

7 Specify the number or rows.

8 **You are prompted**: Number of columns <1>:

9 Specify the number of columns.

10 **You are prompted**: Number of levels <1>:




Rectangular 3D Array:

row spacing A, column spacing B, level spacing C

11 Specify the number of levels.

12 **You are prompted**: Vertical distance between rows:

13 Type a value or click two points.

14 **You are prompted**: Horizontal distance between columns:

15 Type a value or click two points.

16 **You are prompted**: Depth between levels:

Type a value or click two points.

The array is created.

***Прим.:*** Positive values in steps 7, 8 and 9 are measured along the positive X-, Y- and Z-axis of the current UCS. Negative values are measured in the opposite direction.

### 16.16.4 Creating a static 3D polar array

1 Launch the 3DARRAY command.

1 **You are prompted**: Select entities to array [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

**You are prompted**: Select entities to array [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Type of array [Polar/Rectangular] <Rectangular>:

6 Choose the **Polar** option.

**You are prompted**: Number of items to array or [enter to specify Angle between items] <enter to specify Angle between items>:

7 Specify the number of items.

8 **You are prompted**: Angle to fill (+ for ccw, - for cw <360>):

9 Type a value or press Enter to accept the default.

10 **You are prompted**: Rotate entities around the array? [Yes/No] <Yes>:

11 Choose **Yes** or **No**, or press Enter to accept the default.

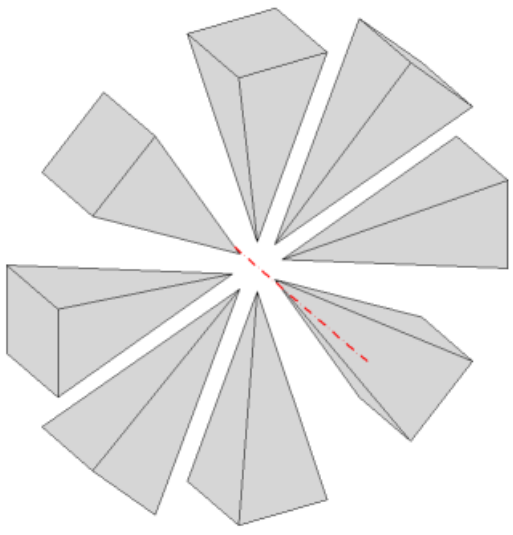
**You are prompted**: Center point of array:

12 Specify a point.

13 **You are prompted**: Specify second point along central axis of array:

14 Specify a point.

15 The array is created.



Polar 3D array around a horizontal axis (red).

## 16.17 Extending entities

The EXTEND command extends entities to a boundary, which is defined by one or more other entities.

If the EDGEMODE system variable is **On**, entities extend to an implied edge of the boundary entities.

If the boundary entity is not in the same plane as the entity you want to extend, the PROJMODE system variable lets you choose how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view plane
- True 3D mode (No projection).

When extending entities, first select the boundary edges, and then specify the entities to extend, selecting them either one by one or using the fence selection method.

The following entities can be extended: arcs, lines, two-dimensional polylines, rays.

Boundary entities can be: arcs, circles, ellipses, lines, splines, polylines, rays, infinite lines, layout viewports.



### 16.17.1 Extending entities

1 Launch the EXTEND command.

1 **You are prompted**: Select boundary entities for extend [selection options (?)] <ENTER to select all>:

2 Select a boundary entity.

3 Entities in set: 1

**You are prompted**: Select boundary entities for extend [selection options (?)] <ENTER to select all>:

4 Select more boundary entities or press Enter to stop selecting.

5 **You are prompted**: Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/ Projection/eRase]:

6 Select an entity:

7 The entity extends to the boundary entity.

**You are prompted**: Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/ Projection/eRase/Undo]:

8 Select more entities or press Enter to stop.

***Прим.:***

- Entities which are selected when you launch the EXTEND command will be used as boundary entities.
- Hold down the Shift key when you select an entity to trim the line by the boundary entity.
- Use the Fence or Crossing selection method to select a series of entities simultaneously.
- If you select an entity near the end that cannot make an extension to one of the boundary entities, the EXTEND command is aborted.

## 16.18 Trimming entities

The TRIM command clips or trims entities by cutting entities.

If the EDGEMODE system variable is **On**, entities extend to an implied edge of the boundary entities.

If the boundary entity is not in the same plane as the entity you want to extend, the PROJMODE system variable lets you choose how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view pane
- True 3D mode (No projection).

When trimming entities, you first select the cutting edges, and then specify the entities to trim, selecting them either one by one or using the fence selection method.

The following entities can be trimmed: lines, two- and three dimensional polylines, arcs, circles, ellipses, elliptical arcs, splines, rays and infinite lines.

Cutting entities can be: lines, splines, polylines, arcs, circles, elliptical arcs, ellipses, rays, infinite lines, layout viewports.

Entities in a block can be selected as cutting entities.



### 16.18.1 Trimming entities

1 Launch the TRIM command.

1 **You are prompted**: Select cutting entities for trim [selection options (?)] <ENTER to select all>:

2 Select a cutting entity.

3 Entities in set: 1

**You are prompted**: Select cutting entities for trim [selection options (?)] <ENTER to select all>:

4 Select more cutting entities or press Enter to stop selecting.

5 **You are prompted**: Select entity to trim or shift-select to extend or [Fence/Crossing/Edge mode/ Projection/eRase]:

6 Select an entity.

7 The entity is trimmed by the cutting entity.

**You are prompted**: Select entity to trim or shift-select to extend or [Fence/Crossing/Edge mode/ Projection/eRase/Undo]:

8 Select more entities or press Enter to stop.

***Прим.:***

- Entities which are selected when you launch the TRIM command will be used as cutting entities.
- Hold down the Shift key when you select an entity to extend the line to the cutting entity.
- Use the Fence or Crossing selection method to select a series of entities simultaneously.

## 16.19 Lengthening entities

The LENGTHEN command changes the length of lines, open polylines and arcs dynamically, incrementally or relatively. You can also modify the included angle of arcs.

### 16.19.1 Changing the length dynamically

1 Launch the LENGTHEN command.

1 **You are prompted**: Edit length: Select entity to list length or [DYnamic/Increment/Percent/Total]:

2 Choose the **Dynamic** option.

**You are prompted**: Select entity to change or [edit Mode]:

3 Click the entity near the endpoint you want to lengthen.

4 The entity is lengthened (shortened) dynamically.

**You are prompted**: New end point for line:

5 Specify a point to define the new length.

6 **You are prompted**: Select entity to change or [edit Mode/Undo]:

7 Do one of the following:

- Edit another entity.
- Press Enter to stop.

***Прим.:*** The **Geometry** properties in the **Properties** panel allow you to edit the total length of lines and arcs and the start and end angle of arcs.

## 16.20 Selection and grip settings

When you select entities in a drawing, the selection method and the display of selected entities, e.g. whether grips are displayed or not, is controlled by a number of settings.

### 16.20.1 Selection settings

Selection settings sit in the **Selection** settings group of the **Program Options** settings category in the **Settings** dialog box.

| Name | System Variable | Description |
|---|---|---|
| Highlight | HIGHLIGHT | Determines whether or not entities highlight on screen when selected. |
| Image highlight | IMAGEHLT | Determines whether the entire raster image highlights or only the raster image frame when selected. |

| Name | System Variable | Description |
|---|---|---|
| Pick add | PICKADD | Controls whether subsequent selections replace the current selection set or add to it.<br>**ON**: The selection set is extended if you select additional entities. Press and hold the Shift key to remove entities from the selection set.<br>**OFF**: You cannot add entities to a selection set. The newly selected entity or entities replace the existing selection set. However, if you press and hold the **Shift** key, you can add entities. If you select an entity that was already selected while pressing the **Shift** key, this entity is removed from the selection set. |
| Pick automatic | PICKAUTO | Controls automatic window selection (inside or crossing) when selecting entities.<br>The value is stored as a bitcode using the sum of the selected options.<br>negative: Disables window selection, saving current value.<br>1 = draws a selection window if the mouse is over a blank area of the drawing when clicked.<br>2 = draws a selection window if the mouse is over an entity when dragged. |
| Pick box | PICKBOX | Defines the size of the small square at the end of the selection cursor (). If you select an entity by clicking the **Pick Box** must touch or overlap the entity.<br>The default size of the **Pick Box** is 3. |
| Pick drag | PICKDRAG | Controls the method of drawing a selection window.<br>**ON**: Allows you to define a selection window by dragging: press and hold the left mouse button to define the first corner of the rectangle, then move the mouse to define the size of the selection window and release the mouse button to define the opposite corner.<br>**OFF**: Define the selection window by clicking two opposite corners. |
| Pick first | PICKFIRST | Controls whether you select objects before or after you issue a modification command.<br>**ON**: Allows you to first compose a selection set, then launch a modification command.<br>**OFF**: You must first start the command, then compose the select entities. |

| Name | System Variable | Description |
|---|---|---|
| Pick style | PICKSTYLE | Controls the selection of groups and associative hatches.<br>0 = No group selection, nor associative hatch selection.<br>1 = Group selection: if a member of a selectable group is selected, all members of the group are selected.<br>2 = Associative hatch selection: the hatch and its boundary are selected, no matter what is picked: the hatch or the boundary (not supported yet).<br>3 = Group selection and Associative hatch selection. |
| Match options for SELECTSIMILAR | SELECTSIMILARMODE | Controls which properties must match for an entity of the same type to be selected with the SELECTSIMILAR command. |
| Selection area | SELECTIONAREA | Controls the display of selection area effects. |
| Selection area opacity | SELECTIONAREAOPACITY | Controls the transparency of the selection area: the higher the value, the more opaque the selection area. |
| Crossing area color | CROSSINGAREACOLOR | Specifies the color for crossing selection areas. |
| Window area color | WINDOWAREACOLOR | Specifies the color for window selection areas. |
| Selection preview display | SELECTIONPREVIEW | Controls when entities highlight when the cursor hovers over them:<br>0 = never;<br>1 = when no commands are active;<br>2 = when a command prompts for entity selection;<br>3 = both, when no commands are active and when a command prompts for entity selection.<br>When the QUAD cursor menu is active, SELECTIONPREVIEW is neglected, unless a command prompts for entity selection. |
| Selection preview effect | PREVIEWEFFECT | Controls the appearance of highlighted entities: 0=dashed lines; 1=thickened lines; 2=dashed and thickened lines (not yet supported). |

| Name | System Variable | Description |
|---|---|---|
| Selection preview filter | PREVIEWFILTER | Controls which entity types are excluded from selection previewing.<br>The options are:<br>Entities on locked layers<br>Entities in Xrefs<br>Tables<br>MText entities<br>Hatches<br>Groups |
| GL Selection Highlight Style | HIGHLIGHTEFFECT | Specifies the highlight method:<br>0 = dashed lines<br>1 = use different color<br>2 = use thickened line<br>3 = use different color and thickened line. |
| Selection Highlight color | HIGHLIGHTCOLOR | Specifies the highlight color to be used when HIGHLIGHTEFFECT is 1 or 3. |
| Selection Modes | SELECTIONMODES | Defines which subentities should highlight in selection preview.<br>The variable is stored as a bitcode using the sum of the values of all selected options.<br>Click the buttons on the **Selection Modes** toolbar to set the value of SELECTIONMODES. A pressed button indicates the option is selected.<br>1 = Enable detection of 3D solid edges ()<br>2 = Enable detection of 3D solid faces ()*<br>4 = Enable boundary detection ().<br>Controls whether closed boundaries in XY-plane of the current coordinate system or on the face of 3D solids highlight when no commands are active.<br>The BOUNDARYCOLOR system variable specifies the color for highlighting detected boundaries.<br>* Hold down the Ctrl key to invert the status of the **Select Faces** option: if ON it is possible to select 3D solids, if OFF it is possible to select solid faces.<br>Repeatedly pressing the Tab key, while keeping the cursor at the same position, highlights all included subentity types one by one.<br>See also Window Selection. |
| Constraint Bar Display | CONSTRAINTBARDISPLAY | Controls the visibility of a geometric constraint at its creation or selection. |

| Name | System Variable | Description |
|---|---|---|
| Dynamic Constraint Mode | DYNCONSTRAINTMODE | Controls the visibility of hidden dimensional constraints when the constrained entity is selected. When the cursor is over an entity with a dimensional constraint, the blue constraint glyph is displayed. |
| Preview selection delay | PREVIEWDELAY | Specifies how many milliseconds to wait before highlighting the (sub)entities under the cursor. |

### 16.20.2 Grips settings

If the Grips system variable is different from zero, grips are displayed on selected entities.

Grips settings sit under **Display/Viewing** in the **Drafting** settings group of the **Drawing** settings category in the **Settings** dialog.

Grips can be used to manipulate entities (see Grips Editing).

| Name | System Variable | Description |
|---|---|---|
| Grips | GRIPS | The value of GRIPS can be 0, 1 or 2.<br>0 : No grips.<br>1 : Display grips. Polyline endpoint and midpoint grips.<br>2 : Display grips. No midpoint grips on polyline segments. |
| Grips in blocks | GRIPBLOCK | Toggles the display of grips on entities in blocks on/off. |
| Grip color | GRIPCOLOR | Sets the color of unselected grips. Default value is green (index color no. 72). |
| Selected grip color | GRIPHOT | Sets the color of selected grips, so called 'hot grips'.<br>Default value is red (index color no. 240). |
| Hover grip color | GRIPHOVER | Sets the color of an unselected grip when the cursor pauses over it.<br>Default value is blue (index color no. 150). |
| Grip size | GRIPSIZE | Sets the size of the grip box, expressed in pixels.<br>Default value = 4. |
| Grip object limit | GRIPOBJLIMIT | Suppresses the display of grips if the number of selected objects exceeds the value of GRIPOBJLIMIT.<br>If set to 0 (zero), all grips are displayed. |

| Name | System Variable | Description |
|---|---|---|
| Grip tips | GRIPTIPS | (Not supported yet) Determines whether grip tips are displayed when the cursor hovers over grips on custom entities or dynamic blocks that support grip tips. |
| Grips attraction | ENABLEATTRACTION | Toggles the magnetism of grips on/off. |
| Grips attraction distance | ATTRACTIONDISTANCE | Sets the grips attraction distance: if the cursor is within this distance, the cursor is attracted by the grip and jumps to it. |

## 16.21 Stretching entities

The size and shape of entities can be changed by stretching them. You select an area in your drawing using either a rectangular window or a polygon, then you specify a base point and a displacement point. All points and nodes inside the selected area will be moved over the specified distance. As a result, entities that cross the window or polygon boundary are stretched; those completely within the window or polygon are simply moved.



### 16.21.1 Stretching entities

1 Launch the STRETCH command.

1 **You are prompted**: Select entities to stretch by [Crossing window/Crossing Polygon/Remove/Add/ selection options (?)]:

2 Use a crossing window (or crossing polygon) to specify the points that need to be moved.

3 Specify the bottom right corner of the crossing window.

A green rectangle is dynamically displayed.

**You are prompted**: Opposite Corner:

4 Specify the top left corner of the crossing window.

5 Entities in set: xx.

**You are prompted**: Select entities to stretch by [Crossing window/Crossing Polygon/Remove/Add/ selection options (?)]:

6 Do one of the following:

- Use a crossing window (or crossing polygon) to specify the points that need to be added or removed from the selection.
- Right-click or press Enter to confirm the current selection.

**You are prompted**: Base point of displacement:

7 Specify a point.

8 The selected points move with the cursor.

**You are prompted**: Second point of displacement:

9 Specify a point to confirm the stretch distance.

## 16.22 Scaling entities

The SCALE command resizes a selection set in relation to a base point. You can specify the scale factor by selecting a base point and a length or by typing an explicit scale factor in the Command line. The scale factor can also be defined with respect to a base scale factor. E.g. when the base scale factor is 2 and the new scale is 3, the new size is 3/2 of the original. The base scale and the new scale can also be defined graphically in the drawing.

### 16.22.1 Scaling a selection set

1 Launch the SCALE command.

1 **You are prompted**: Select entities to scale [selection options (?)]:

2 Select entities.

3 Entities in set: xx

**You are prompted**: Select entities to scale [selection options (?)]:

4 Select more entities or press Enter to continue.

5 **You are prompted**: Base point:

6 Specify a point.

7 The base point is not moved in the scaling.

**You are prompted**: Scale factor or [Reference/Copy] <current scale*>:

8 Type a scale factor or press Enter to accept the previously used value.

* The previously used scale factor in a drawing is remembered.

### 16.22.2 Scaling a selection set using a reference scale

1 Repeat steps 1 through 4 of the procedure above.

1 **You are prompted**: Scale factor or [Reference/Copy] <current scale*>:

2 Choose the **Reference** option.

**You are prompted**: Reference scale <1>:

3 Click the base point again (see step 3 in the previous procedure) (1).

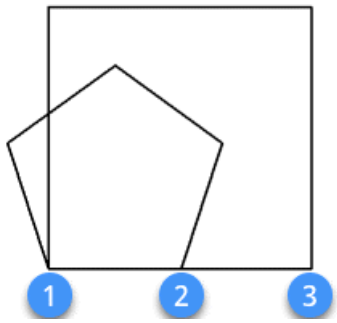
4 **You are prompted**: Second point:

5 Specify a point. (2)

6 The distance between the base point and the second point will be used as the reference scale.

The selection scales dynamically.

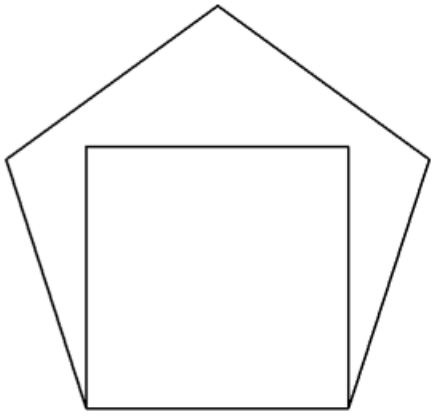
**You are prompted**: New scale factor or [Points] <2>:




7 Do one of the following:

- Type a value.
- The distance between the reference points is set to the specified value.

  The size of the selection set is scaled accordingly.
- Specify a point to position the second reference point (see step 4) (3).



## 16.23 Breaking entities

The BREAK command removes a portion of an entity, thus breaking it into two parts.

You can break arcs, circles, ellipses, lines, polylines, rays and infinite lines. Breaking a circle converts it to an arc. A ray is broken into a ray and a line, an infinite line is broken into two rays.

When breaking entities, you must specify two points for the break. By default, the point you use to select the entity becomes the first break point. However, you can use the **First** option to select a break point different from the one that selects the entity.

### 16.23.1 To break an entity

1 Launch the BREAK command.

1 **You are prompted**: Select entity to break [selection options (?)]:

2 Select the entity.

3 **You are prompted**: Select second breakpoint or [First break point/same as first point (@)]:

4 Select the **First** break point option.

**You are prompted**: First break point:

5 Specify a point.

6 **You are prompted**: Second break point:

7 Specify a point.

8 The part of the entity between the two break points is removed.

## 16.24 Joining entities

The JOIN command joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.

When you join colinear lines, the farthest endpoints remain at their existing locations. BricsCAD draws a new line between these points.

The resulting entity type depends on the types of the input entities and their coplanarity.



### 16.24.1 General procedure

1 Launch the JOIN command.

1 **You are prompted**: Select source entity or multiple entities to join at once [selection options (?)]:

2 Select source entities, then right-click to stop selecting.

3 BricsCAD reports the result:

4 37 entities joined into 10 polylines, 4 entities discarded from the operation

### 16.24.2 Joining two arcs

Coplanar arcs, sharing midpoint and radius (= lie on the same circle), are joined counterclockwise, therefore the result depends on the selecting order.

1 Launch the JOIN command.

1 **You are prompted**: Select source entity or multiple entities to join at once [selection options (?)]:

2 Select the first arc.

3 **You are prompted**: Select entities to join [selection options (?)]:

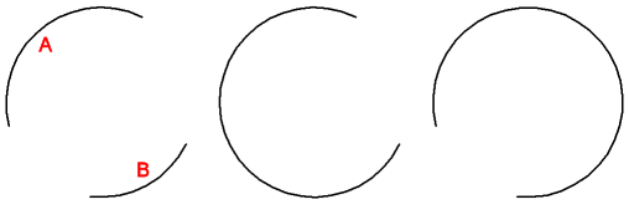
4 Select the second arc.

5 Entities in set: 1

6 **You are prompted**: Select entities to join [selection options (?)]:

7 Right-click.

8 BricsCAD reports: 2 arcs joined into 1 arc.




start situation (left) and result when selecting A first (middle) or B first (right).

## 16.25 Chamfering entities

The CHAMFER command connects two non-parallel entities by extending or trimming them and then joining them with a line to create a beveled edge.

In BricsCAD you can choose between two chamfer methods:

- **distance-distance**: specify how far to trim the entities back from their intersection
- **distance-angle**: specify the length of the chamfer and the angle it forms along the first entity.

The current chamfer method, distances and angle are saved through a series of system variables:

- CHAMMODE: sets the chamfer method
- CHAMFERA: defines the first distance
- CHAMFERB: defines the second distance
- CHAMFERC: defines the chamfer length
- CHAMFERD: defines the chamfer angle
- TRIMMODE: defines whether or not to keep the original length of the chamfered entities or to trim or lengthen them to the endpoints of the chamfer line.

You can set the values in the **Settings** dialog box under **Drawing** > **Drafting** > **Entity Modification** > **Chamfer/Fillet**. The **Settings...** option of the CHAMFER command allows you to edit the variables in the CHAMFER command procedure.

The following entities can be chamfered: lines, polylines, rays and infinite lines. When chamfering a polyline, you can create a chamfer between two polyline segments or you can chamfer all vertices of the polyline. When chamfering a polyline, the chamfer line is joined with the polyline segments if TRIMMODE=ON.



### 16.25.1 Chamfering using the distance-distance method

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings...** option, edit the Chamfer variables in the **Settings** dialog box.

3 Select the first entity.

4 **You are prompted**: Directional/Select second entity (select with pressed SHIFT to make corner)

[Directional]:

5 Select the second entity.

6 The chamfer is created.

### 16.25.2 Chamfering using the length-angle method

1 Launch the CHAMFER command.

1 Chamfer (length=20, angle=45) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings...** option, edit the Chamfer variables in the **Settings** dialog box.

3 Select the first entity.

4 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

5 Select the second entity.

6 The chamfer is created.

### 16.25.3 Chamfering all vertices of a polyline

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings...** option, edit the Chamfer variables in the **Settings** dialog box.

3 Choose the **Polyline** option.

**You are prompted**: Select 2D polyline to chamfer:

4 Select the polyline.

5 The polyline is chamfered.

***Прим.:*** When the chamfer method is **distance-angle**, the direction of the polyline defines which is the first entity of a vertex. Use the **Reverse direction** option of the PEDIT command if necessary to get the desired result.

### 16.25.4 Removing a chamfer

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 Select the first entity.

3 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

4 Hold down the Shift key, then select the second entity.

5 The chamfer is removed.

***Прим.:*** You can use this method to create a corner between two lines or an end segment of a polyline. When the two entities intersect, the part you click is kept.

## 16.26 Filetting entities

The FILLET command connects two entities with an arc of a specified radius to create a rounded edge.

You can fillet pairs of line segments, straight polyline segments, arcs, circles, rays, and infinite lines. You can also fillet parallel lines, rays, and infinite lines.

When filleting a polyline, you can fillet multiple segments between two selected segments or you can fillet the entire polyline.

The TRIMMODE system variable defines whether or not to keep the original length of the filleted entities or to trim or lengthen them to the endpoints of the fillet arc.



### 16.26.1 Filleting two entities or polyline segments

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Choose the **Radius** option.

**You are prompted**: Fillet radius <current radius>:

3 Type a value.

4 **You are prompted**: Fillet (radius=new radius): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

5 Select the first entity or polyline segment.

6 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

7 Select the second entity or polyline segment.

8 The fillet is created.

### 16.26.2 Filleting all vertices of a polyline

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Choose the **Radius** option.

**You are prompted**: Fillet radius <current radius>:

3 Type a value.

4 **You are prompted**: Fillet (radius=new radius): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

5 Choose the **Polyline** option.

**You are prompted**: Select 2D polyline to fillet [selection options (?)]:

6 Select the polyline.

7 The polyline is filleted.

### 16.26.3 Filleting two parallel lines

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Select the first line.

3 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

4 Select the second line.

5 The fillet is created.

## 16.27 Editing polylines

Use the PEDIT command to:

- Convert an entity into a polyline
- Fit a curve to a polyline
- Taper a polyline uniformly along its length
- Reverse the direction of a polyline

Edit a polyline in the **Properties** panel to:

- Open or close a polyline
- Apply a uniform width to a polyline
- Change the width of a polyline segment
- Set the linetype mode (PLINEGEN system variable)

Use the edit polyline segments tools in the Quad to:

- Insert a vertex
- Add vertices at the start or end
- Modify the bulge factor of a segment
- Stretch a vertex or a segment
- Delete a segment

Use grips editing to:

- Move a vertex
- Move a segment

### 16.27.1 Launching the PEDIT command



Do one of the following:

- Double-click a polyline.
- Click the Edit Polyline tool in the **Modify** menu.
- Click the Edit Polyline tool on the **Modify** toolbar.

### 16.27.2 Converting an entity into a polyline

1 Launch the PEDIT command.

2 Select the entity.

**You are prompted**: The selected entity is not a polyline. Turn it into one? [Yes-turn into polyline/No-leave as is] <Yes-turn into polyline>:

3 Press Enter or right-click.

4 **You are prompted**: Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/Spline/Taper/Width/Undo] <eXit>:

5 Choose an option or right-click to stop.

### 16.27.3 Fitting a curve to a polyline

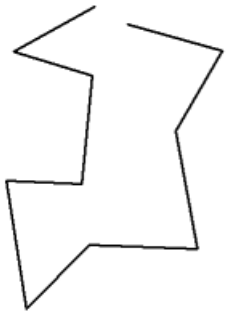
1 Launch the PEDIT command.

2 Select the polyline.

3 Do one of the following:

- Choose the **Fit** option.
- Choose the **Spline** option.

4 Press Enter or right-click to conclude the PEDIT command.

| | | |
|---|---|---|
|  |  |  |
| Original polyline | After applying Fit | After applying Spline |

***Прим.:***

- Use the **Decurve** option to restore the original polyline.
- The SPLINESEGS (Spline Segments) system variable controls the number of segments (lines or arcs) to be generated for spline-fit polylines.

### 16.27.4 Tapering a polyline uniformly along its length

1 Launch the PEDIT command.

2 Select the polyline.

3 Choose the **Taper** option.

**You are prompted**: Enter new starting polyline width <current width>:

4 Type a value.

5 **You are prompted**: Enter new ending polyline width <current width>:

6 Type a value.

7 The polyline is tapered.

8 Press Enter or right-click to conclude the PEDIT command.

### 16.27.5 Reversing the direction of a polyline

1 Launch the PEDIT command.

2 Select the polyline.

3 Choose the **Reverse direction** option.

The direction is reversed.

4 Press Enter or right-click to conclude the PEDIT command.

### 16.27.6 Opening or closing a polyline

1 Select the polyline.

2 Select the **Closed** property under **Misc** in the **Properties** panel.

3 Choose **No** or **Yes**.

Or

1 Double-click the polyline.

2 Type C to close, or O to open.

### 16.27.7 Applying a uniform width to a polyline

1 Select the polyline.

2 Select the **Global width** property under **Geometry** in the **Properties** panel.

3 Type the desired width.

### 16.27.8 Changing the width of a polyline segment

1 Select the polyline.

2 Select the **Vertex** property under **Geometry** in the **Properties** panel.

An X mark is displayed at the start vertex of the polyline.

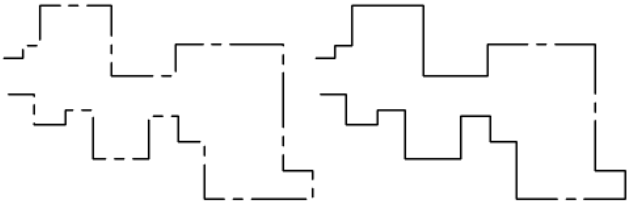
3 Click the arrow keys at the right hand side of the vertex property field to set the X marker at the start vertex of the segment you want to change the width of.

4 Select the **Start width** property and type the desired width.

5 Select the **End width** property and type the desired width.

### 16.27.9 Setting the linetype mode

The Linetype generation mode (PLINEGEN system variable) controls how a dashed linetype is applied to a multi-segment polyline. When disabled, dashed linetypes are applied per segment: start/end with a dash at each vertex.



PLINEGEN On (left) and Off (right)

1 Select the Polyline.

2 Select the **Linetype generation** property under **Misc** in the **Properties** panel.

3 Select **disabled** or **enabled**.

### 16.27.10 Using the edit polyline segments tools in the Quad

- Add vertex: adds a vertex to the segment under the cursor.
- Add vertex at end: adds as a new vertex at the end of the polyline.

  This option is only available if the cursor is on one of the endpoints of a polyline.
- Adjust bulge: modifies the bulge factor of the segment under the cursor.
- Delete segment: deletes the segment under the cursor.
- Stretch segment: moves the segment under the cursor.
- Stretch vertex: moves the vertex closest to the cursor.

1 Place the cursor over the segment you want to edit.

1 The entire polyline is highlighted.

2 Hold down the Ctrl key.

3 Only the segment under the cursor is highlighted.

The editing tools are available in the Quad.

4 Select a tool, then follow the instructions in the Command line.

## 16.28 Exploding entities

The EXPLODE command converts complex entities, such as blocks, polylines, solids or dimensions into their component parts.

Exploding a polyline or dimension reduces it to a collection of individual line and arc entities that you can then modify individually.

Blocks are converted to the individual entities, possibly including other, nested blocks that composed the original entity.

MText entities explode into single line texts.

In general, exploding entities will have no visible effect in the drawing, except for:

- If the original polyline had a width, the width information is lost when you explode it. The resulting lines and arcs follow the centerline of the original polyline.
- If you explode a block containing attributes, the attributes are lost, but the original attribute definitions remain.
- Colors and linetypes assigned BYBLOCK may appear different after exploding an entity, because they will adopt the default color and linetype.

The TXTEXP command explodes text entities into polylines. You can then use the JOIN command to create a single polyline for each letter. For some letters, such as a, e or p, a second polyline is needed for the 'hole').

### 16.28.1 Exploding entities

Do one of the following:

- Launch the EXPLODE command.
- Select entities, then press Enter.
- Select entities, then choose **Explode** in the **Modify** command group in the Quad.

## 16.29 Measuring entities

The MEASURE command places markers - points or blocks - at a specified interval along the length or circumference of an entity. The markers are placed starting at the closest endpoint to where you select the entity.

You can measure lines, polylines, arcs, circles, ellipses, elliptical arcs and splines. You cannot measure rays and infinite lines.

### 16.29.1 Measuring an entity using points

1 Launch the MEASURE command.

1 **You are prompted**: Select entity to measure:

2 Select an entity.

3 **You are prompted**: Segment length [insert Blocks]:

4 Do one of the following:

- Type a value.
- Specify two points.

The points are placed along the entity at the specified interval.

***Прим.:*** The visibility of the points depends on the current value of the PDMODE system variable.

### 16.29.2 Placing blocks a specified interval along an entity

1 Launch the MEASURE command.

1 **You are prompted**: Select entity to measure:

2 Select an entity.

3 **You are prompted**: Segment length [insert Blocks]:

4 Choose the **Blocks** option.

**You are prompted**: Name of block to insert [? to list]:

5 (option) Press the ? key to see a list of available blocks in the drawing.

6 Type the name of a block.

7 **You are prompted**: Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-align blocks>:

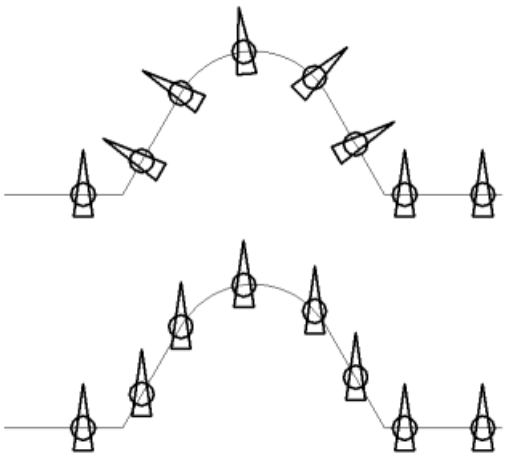
8 Choose whether or not to align the blocks.

9 **You are prompted**: Segment length:

10 Do one of the following:

- Type a value.
- Specify two points.

The blocks are placed along the entity at the specified interval.



Blocks aligned (top) or not aligned (bottom)

## 16.30 Dividing entities

The DIVIDE command places markers - points or blocks - along a selected entity. The markers divide the entity into a specified number of equal parts.

You can divide lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.

### 16.30.1 Dividing an entity using points

1 Launch the DIVIDE command.

1 **You are prompted**: Select entity to divide:

2 Select an entity.

3 **You are prompted**: Number of segments [insert Blocks]:

4 Type a value, then press Enter.

5 The entity is divided by points.

***Прим.:*** The visibility of the points depends on the current value of the PDMODE system variable.

### 16.30.2 Dividing an entity using blocks

1 Launch the DIVIDE command.

1 **You are prompted**: Select entity to divide:

2 Select an entity.

3 **You are prompted**: Number of segments [insert Blocks]:

4 Choose the **Blocks** option.

**You are prompted**: Name of block to insert [? to list]:

5 (option) Press the ? key to see a list of available blocks in the drawing.

6 Type the name of a block.

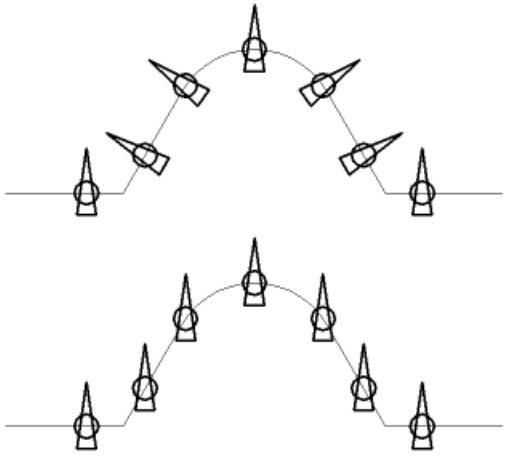
7 **You are prompted**: Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-align blocks>:

8 Choose whether or not to align the blocks.

9 **You are prompted**: Number of segments:

10 Type a value, then press Enter.

11 The entity is divided by blocks.



Blocks aligned (top) or not aligned (bottom)

# 17. Печать

## 17.1 General procedure for printing

The PLOT and PRINT commands plot drawings to printers and to files, through the **Print** dialog box.

The -PLOT command plots drawings to printers and to files, through prompts at the command bar; this command is meant for scripts and routines.

The QPRINT command prints the drawing with the default plot configuration, without displaying the **Print** dialog box.

The PAGESETUP command creates and edits page setups for plotting drawings in the **Drawing Explorer** dialog box.

A standard print procedure consists of the following steps:

- Choose either a **Layout** tab or the **Model** tab.
- Open the **Print** dialog.
- Select a **Page Setup name**. (1)

The following steps are optional and only necessary if you want to use overrides on the selected page setup definition.

- Choose a **Printer or Plotter configuration**. (2)
- Choose a **Plot style table**. (3)
- Choose a **Paper size**. (4)

  The image reflects the position and orientation of the drawing on the selected paper size.
- Choose the paper size unit. (5)
- Choose the **Drawing orientation**.(6)
- Define the **Plot offset**. (7)
- Specify the **Plot area** (the area to plot). (8)
- Define the **Plot scale**. (9)
- Check the **Plot options**. (10)
- When printing model space, choose a **Shade plot** mode. (11)

  This option is disabled when printing a paper space layout.
- Check/uncheck the **Print to file** option. (12)
- Specify the **Number of copies**. (13)
- Click the **Preview** button to see a preview of the plot output.
- When in a layout, click the **Apply** button to update the layout.
- Click the **Print** button to start printing.

***Прим.:*** All print settings and options are saved in the **Model** and each of the **Layouts**.

***Прим.:***

***On Mac or Linux, printing will always print to PDF. So a PDF document will be generated that then still needs to be sent to the printer for physical printing.***

***On MAC and Linux there is a relevant PAGESETUP command.***

### 17.1.1 Opening the Print dialog box

Do one of the following:

- Launch the PRINT or PLOT command.
- Press **Ctrl + P**.
- Choose **Print...** in the **File** menu.
- Click the **Print** tool in the **Output / Plot** ribbon panel (Drafting workspace).
- Select the **Model** tab or a **Layout** tab, then right-click and choose **Print** in the context menu.

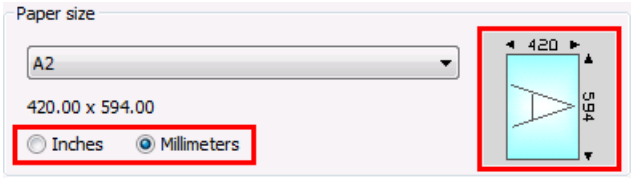
### 17.1.2 Choosing the paper size

Press the down arrow, then select the paper size in the **Paper Size** list.

The available sizes depend on the selected printer.

### 17.1.3 Choosing the paper unit

Select either **Inches** or **Millimeters**.




The paper size image indicates the size, position and orientation of the plot area on the selected paper size.

***Прим.:*** The **Scale** and **Plot offset** setting units are changed according to the chosen **Paper unit**.

### 17.1.4 Choosing the paper orientation

Select either **Portrait** or **Landscape**.

- **Portrait**: The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.
- **Landscape**: The drawing or layout x-axis is aligned with the longest edge of the selected paper size.

The paper size image changes accordingly.

When in a layout, click the **Apply** button to update the display.

### 17.1.5 Defining the plot offset

Select either **Center on page** or type the plot offset in the **X:** and **Y:** fields.

The offset is calculated from the bottom left corner of the paper.

When in a layout, click the **Apply** button to update the display.

To specify the plot area:

| Option | Result |
|---|---|
| View | Prints the portion of the drawing that is displayed in the current viewport or a saved view. |
| Extents | Prints all entities in the drawing. |
| Limits | Prints to the limits defined in the drawing.<br>This option is only available when printing in Model space. |
| Layout | Prints the current layout.<br>This option is only available when printing in Paper space. |
| Window | Prints the portion of the drawing contained in a user defined window, maintaining the aspect ratio of the windowed area to the drawing.<br>Click the **Select print area** button (), then specify the two corners of the print area in the drawing.<br>or<br>Type the X- and Y-coordinates of the corners of the print area in the **X:** and **Y:** fields. |

### 17.1.6 Defining the plot scale

***Прим.:*** The SCALELISTEDIT command allows you to edit the list of available scales in the **Print** dialog and the **Standard Scale** property of a paper space viewport.

**... when printing a paper space layout:**

If you print in paper space, the final scale of the plot output is the product of the viewport scale and the plot scale.

In order to keep control over the scale of the plot output:

- Specify the width and height of the viewports in drawing unit.
- Set the scale of each viewport to the scale you want in the plot output.
- The **plot scale** reflects the relationship between paper unit and drawing unit. (see table below)

| Paper unit | Drawing unit | Plot scale | Printed mm or inches = | Drawing unit |
|---|---|---|---|---|
| mm | mm | 1:1 | 1 | 1 |
| mm | cm | 10:1 | 10 | 1 |

| Paper unit | Drawing unit | Plot scale | Printed mm or inches = | Drawing unit |
|---|---|---|---|---|
| mm | m | 1000:1 | 1000 | 1 |
| inches | inches | 1:1 | 1 | 1 |
| inches | feet | 12:1 | 12 | 1 |

**... when printing model space:**

If you print in model space, the scale of the plot output equals the plot scale, on the condition that your drawing unit matches the paper unit. Otherwise you need to multiply the plot scale with respect to the relationship between the paper unit and the drawing unit. E.g. if you want to plot at a 1/50 scale in a drawing of which 1 DU = 1 cm, the **plot scale** = 1/5, if 1 DU = 1 m the **plot scale** = 20/1 or 1/0.05

### 17.1.7 Setting the plot options

Click to toggle the **plot options** On / Off.

| Option | Result |
|---|---|
| Plot object lineweights | If **Plot with plot styles** is off, object lineweights are plotted. |
| Plot with plot styles | Settings of the assigned STB or CTB file are used.<br>***Прим.:*** This applies to viewports which have the 2D Wireframe or Wireframe visual style only. |
| Plot transparencies | Plots entities using the value of the **Transparency** property. |
| Plot paperspace last | Entities in Model space are plotted first, paperspace entities are plotted last. |
| Hide paperspace entities | Removes hidden lines from 3D entities in paper space.<br>This option is disabled when printing model space. |
| Plot stamp | Adds a header and / or footer.<br>Click the **Edit plot stamp button** () to define the plot stamp. |
| Save changes to layout | All changes made in the **Print** dialog are saved in the layout.<br>This option toggles the **Save changes to layout** variable. |

### 17.1.8 Setting the shade plot mode

By default the Shade Plot mode is set to **As displayed**.

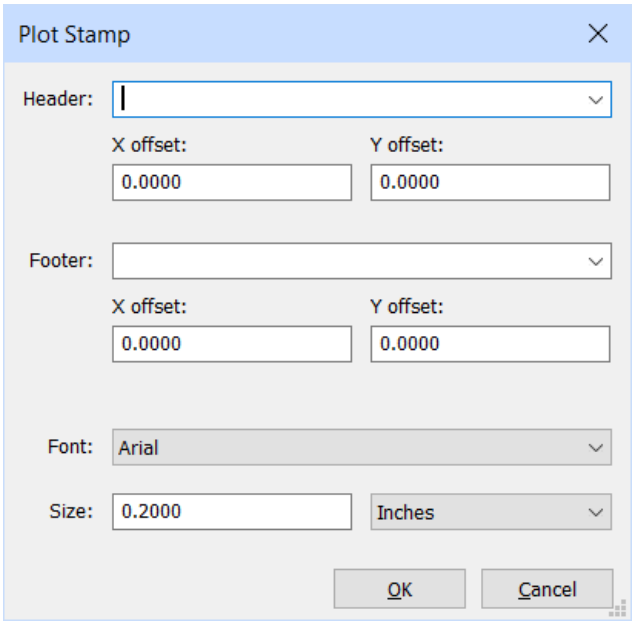
Click the list button, then select an option to plot using a different visual style.

***Прим.:*** This option applies to model space printing only. The Shade Plot mode of a paper space viewport is defined in the Shade Plot property of the viewport.

### 17.1.9 Defining the plot stamp

1 Click the **Edit plot stamp** button.

The **Plot stamp** dialog box appears.




2 In the **Header** field, do one of the following:

- Type the header text.
- Click the down arrow button, then choose a variable in the drop down list.
- Type one or more variable names (see table below).
- Leave the field empty.

3 Define the X and Y offset for the header.

4 In the **Footer** field, do one of the following:

- Type the footer text.
- Click the down arrow button, then choose a variable in the drop down list.
- Type one or more variable names (see table below).
- Leave the field empty.

5 Define the X and Y offset for the footer.

6 Select a font in the **Font** list button.

7 Enter a value in the **Font Size** field.

8 Set the unit type to **Millimeters** or **Inches**.

9 Click the **OK** button to save the plot stamp.

| Variable Name | Description |
|---|---|
| $Name | Adds the name of the current user, as defined in the **User Name** field of the **License Information**. |
| $Company | Adds company name of the current user, as defined in the **Company** field of the **License Information**. |
| $Date | Adds the current date. |
| $DateTime | Adds the current date and time. |
| $DwgName | Adds the file name and path of the drawing. |

***Прим.:***

- The header and footer text consists of three parts (left, center and right), separated by commas.
- By default, the header and footer text is centered.
- The header and footer text can be outlined left center and/or right by separating the text with commas.
- e.g. **$Name,$Company,$Date** places your name left, company name centered and the current date at the right.
- Type a space before one of the commas to leave one of the parts empty, thus outlining the header or footer left or right.
- e.g. **$Company, ,** ($Company,<space>,<space>) outlines the company name left. , ,$Company (<space>,<space>,$Company) outlines the company name right.
- The plot stamp settings are saved to a series of user preferences in the current user profile, which can be edited in the **Settings** dialog box under **Program Options** > **Plot and Publish** > **Plot Stamp**.

### 17.1.10 Specifying the number of copies

Do one of the following:

- Type the desired number in the **Number of copies** field.
- Click the up-arrow and the down-arrow buttons to specify the number of copies.

### 17.1.11 Previewing the plot output

Click the **Preview** button on the **Print** or **Page Setup** dialog box.

Choose **Print Preview...** in the **File** menu.

Type **preview** in the Command line, then press Enter.

## 17.2 Plotter configuration

Configurations for Windows system printers are stored in plotter configuration files (.PC3 files). Unlike in AutoCAD®, PC3 plotter configuration files in BricsCAD™ apply to Windows system printers only.

In a plotter configuration file, you override one or more settings of the system printers installed on your computer. You can configure BricsCAD for many plotting/printing devices and keep multiple configurations for each single device.

Each plotter configuration file contains information such as:

- The device driver and model
- The output port to which the device is connected
- Various device-specific settings (not supported on the Linux platform).

PC3 files are saved in the **Plotconfig** subfolder of your Roamable root folder.

After creating a PC3 file, it is available in the list of plotter configuration names in the **Printer / Plotter configuration** list on the **Print** or **Page Setup** dialog boxes.

### 17.2.1 Configuring the built-in PDF printer

1 Open the **PRINT** dialog box.

2 Click the **Printer/Plotter configuration** list button and choose **Print As PDF.pc3** in the list.

3 Click the **Edit Plotter Configuration** button.

The **Plotter Configuration Editor** dialog is displayed with the **Print As PDF.pc3** configuration file loaded.

4 Click the **Settings** tab on the **Plotter Configuration Editor** dialog.

5 (Option) Edit the **Graphics** settings.

6 (Option) Execute the **Scale Calibration**.

7 Click the **Custom Properties** button.

The **Print As PDF - Custom Properties** dialog is displayed.

8 (Option) Adjust the **Output quality** and **Layers support** settings.

9 Click the **Manage Custom sizes ...** button.

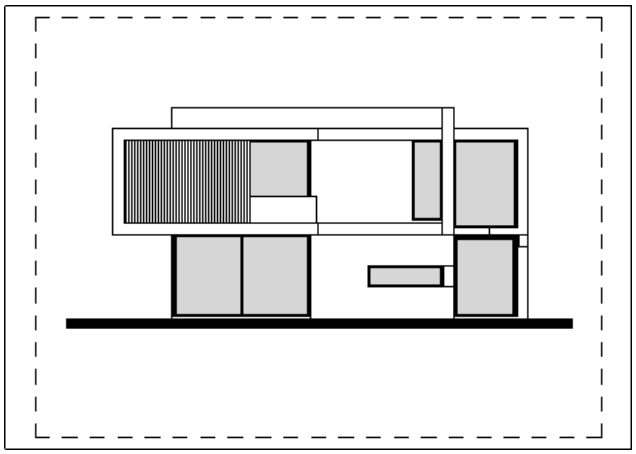
10 To create a custom paper size, do the following:

- Click the **Add** button.

  A new paper size is added.
- Double-click the newly created paper size.
- Type a name for the new paper size, then click the **OK** button.
- Define **Width** and **Height** for the new paper size.
- Define the **Paper Margins**.

Paper margins define the printable area of paper sheet.

If the DISPPAPERMARGINS system variable is ON, dashed lines indicate the paper margins on the paper sheet in a layout.



The paper sheet background is displayed if the DISPPAPERBKG system variable is ON.

- Click the **OK** button to create the custom paper size.

11 Click the **OK** button on each of the previously opened dialogs.

The custom paper size(s) can now be selected in the **Paper Size** list button.

### 17.2.2 Creating a plotter configuration

1 Launch the PLOTTERMANAGER command.

1 A **File** dialog box is displayed with the folder, defined by the PLOTCFGPATH system variable.

2 Double-click **Create a Plotter Configuration** shortcut.

The **Plotter Configuration Editor** dialog box is displayed.

3 In the **Printer name** list, select the printer you want to create a plotter configuration (PC3) for.

4 Click the **Settings** tab on the **Plotter Configuration Editor** dialog.

5 Click the **Custom Properties...** button.

The **<Selected Printer> Properties** dialog opens.

The layout of this dialog is different for each printer.

Custom Properties are not supported on the Linux platform.

6 On the **Properties** dialog of the selected printer, adjust the printer settings, then click the **OK** button to confirm.

The **<Selected Printer> Properties** dialog closes.

7 Click the **OK** button on the **Plotter Configuration Editor** dialog.

The **Save Plotter Configuration As** file dialog appears.

8 (option) Edit the PC3 file name.

9 The default name is **<Selected Printer>.pc3**.

10 Click the **OK** button on the **Changes to a Printer Configuration File** dialog.

The PC3 file is created.

***Прим.:*** You can create multiple plotter configurations for the same printer if you save them under different names.

### 17.2.3 Editing a plotter configuration

1 Launch the PRINT command.

1 The **Print** dialog box appears.

2 Select a plotter configuration (PC3) in the **Printer / Plotter configuration** list.

3 Click the **Edit Plotter Configuration** button.

The **Plotter configuration editor** dialog opens.

4 Proceed in the **Creating a plotter configuration** procedure above starting from step 4.

***Прим.:*** If you select a system printer in step 2, a new plotter configuration is created for the selected printer. When you click the **Edit Plotter Configuration** button in step 3, the **Properties** dialog of the selected printer opens first. After clicking the **OK** button on the **Properties** dialog, the **Plotter configuration editor** dialog opens. You can then complete the **Creating a plotter configuration** procedure.

### 17.2.4 Assigning a plotter configuration

1 Choose **Model** or the **Layout** to which you want to assign a specific plotter configuration file.

2 Do one of the following:

- Choose **Page Setup** in the **File** menu.
- Type **pagesetup** in the Command line, then press Enter.

3 Choose one of the following from the **Printer / Plotter configuration** list:

- A system printer
- A configuration file (PC3 file)
- None (uses Default)

4 Click the **OK** button.

***Прим.:***

- When BricsCAD cannot find the plotter configuration file assigned to the drawing, it changes the configuration file assigned to your drawing to **None**.
- The **None** printer device has its own set of paper sizes that can be stored in the drawing, very much the same as for any 'real' printer. If a layout is opened and the stored printer is absent, the printer switches to **None** while the paper size is maintained. The missing printer and the previous paper size are indicated upon opening the **Print**dialog. The previous paper size can be saved in the drawing.
- PREVIOUS PAPER SIZE: drawings created by other parties are often set up for a printer that is not available in your office. In such case, BricsCAD resets the printer device to **None**, which results in using the default printer. Instead of resetting the paper size to a default, BricsCAD shows the Previous Paper size, in order to inform you about the intended paper size and allow you to choose a similar size on any of the available printers.

## 17.3 Plot style tables

The **Plot Style Table Editor** displays the plot styles contained in the specified plot style table.

### 17.3.1 Opening the Plot Style Table editor

Do one of the following:

- Click the **Edit Plot Style** button on the **Print** or **Pagesetup** dialog box.
- Choose **Plot Style Manager...** in the **File** menu, then double-click a **CTB** or **STB** file.
- In a **Windows Explorer** window, double-click a **CTB** or **STB** file.

The Plot Style Table Editor includes the following tabs:

- The **General** tab
- The **Form View** tab
- The **Lineweights** tab
- The **General** tab

The **General** tab lists general information about the plot style table.

| Name | Description |
|---|---|
| Plot style table file name | Displays the name of the plot style table file you are editing. |
| Description | Displays a description you want to include about the plot style table. |
| File info | Displays information about path, number of plot styles and the version number of the Plot Style Table Editor.<br>***Прим.:*** A color-dependent plot style table always contains 255 plot styles. |
| Apply global scale factor to non-ISO linetypes | Specifies whether or not you want to scale all non-ISO linetypes in the plot styles of entities controlled by the plot style table. |
| Scale factor | Specifies the factor by which to scale non-ISO linetypes and fill patterns. |

The **Form View** tab lists the plot styles contained in the plot style table and their settings. Plot styles are style overrides for your drawings that occur during plotting.

The first plot style in a **Named Plot Style** table is **NORMAL** and represents an entity's default properties (no plot style applied). You cannot modify or delete the **NORMAL** style.

| Property | Description |
|---|---|
| **Plot styles** | Displays the names of plot styles contained in the selected plot style table.<br>• Always contains 255 styles (1 for each color).<br>• They are tied to entity color.<br>• You cannot add nor delete a plot style.<br>• You cannot rename a plot style.<br>Named plot style table (STB):<br>• Contains one or more plot styles.<br>• You can add or remove plot styles, except for the **Normal** style, which cannot be deleted.<br>• Plot styles can be renamed, except for the **Normal** style, which cannot be renamed. |
| **Description** | Description for the selected plot style. |
| **Color** | Specifies the plotted color for an entity.<br>The default setting for plot style color is **Use Object Color**.<br>If you assign a plot style color, the color overrides the entity's color in the plot output. |
| **Dither** | Depending on the capabilities of your plotter, dithering approximates the colors with dot patterns. When this option is not active, the colors are mapped to the nearest color, resulting in a smaller range of colors when plotting.<br>Dithering is available only when you select the object's color or assign a plot style color.<br>By default, this option is active. Setting it to Off has no effect in BricsCAD. |
| **Grayscale** | Depending on the capabilities of your plotter, converts the colors to grayscale in the plot output. |
| **Pen #** | Specifies a pen to use when plotting entities that use this plot style.<br>You can select a pen number from 1 to 32. |

| Property | Description |
| --- | --- |
| **Virtual pen #** | Specifies a virtual pen number between 1 and 255.<br>Choose **Automatic** to specify that BricsCAD should make the virtual pen assignment from the BricsCAD Color Index.<br>The virtual pen setting in a plot style is used only by non-pen plotters and only if they are configured for virtual pens (select **255 Virtual Pens** under **Color Depth** under **Vector Graphics** on the **Settings** tab in the **Plotter Configuration Editor**). |
| **Screening** | Specifies the color intensity of the plot on the paper.<br>The valid range is 0 through 100.<br>If you select 100 the drawing will plotted with its full color intensity. In order for screening to work, the **Dithering** option must be active. |
| **Linetype** | Displays a list with an example and a description of each linetype.<br>The default setting for plot style linetype is **Use Object Linetype**.<br>The selected linetype overrides the entity's linetype in the plot output. |
| **Adaptive** | Adjusts the scale of the linetype to complete the linetype pattern.<br>You can activate this property if a complete linetype pattern is more important than a correct linetype scaling. |
| **Lineweight** | Displays a sample of the lineweight as well as its numeric value.<br>You can modify an existing lineweight if the one you need is not available.<br>The default setting for plot style lineweight is **Use Object Lineweight**.<br>The selected lineweight overrides the entity's lineweight in the plot output. |

| Property | Description |
|---|---|
| **Line end style** | Displays a list of several end styles for the lines.<br>By default, the option **Use Object End Style** is selected.<br>If you select a line end style, the object's line end style will be overridden in the plot output. |
| **Line join style** | Displays a list of several join styles for the lines.<br>By default, the option **Use Object Join Style** is selected.<br>If you select a line join style, the object's line join style will be overridden in the plot output. |
| **Fill style** | Displays a list of several fill styles.<br>By default, the option Use **Object Fill Style** is selected.<br>If you select a fill style, the object's fill style will be overridden in the plot output. |

***Прим.:*** Entities which are created in **true color** are always plotted using their own object properties (color, lineweight, ...).

The **Lineweights** tab lists the available lineweights.

Lineweights with a tick are used in the **Lineweight** field on the **Form View** tab.

To edit a lineweight, select the lineweight in the list, then click the **Edit** button.

### 17.3.2 Assigning a plot style table

In BricsCAD, you can assign plot style tables to model space and to each of the layouts in a drawing. By assigning different plot style tables to each layout in your drawing, you can control how entities in the layout are plotted. The plot style table affects both model space and paper space entities. To plot the drawing without applying plot style properties, choose **None (use Default)** from the list of plot style tables.

In STB-type drawings, each entity in the drawing either is assigned a plot style explicitly or **By Layer**.

Plot style tables are used only if the **Plot with plot styles** plot option on the **Print** dialog box is ticked. When you turn off plot style tables, entities print according to their own properties.

**Assigning a plot style table**

1 Open the **Print** or **Page Setup** dialog box.

2 Select a **Plot style table** from the list box in the **Plot Style Table (pen assignments)** section.

3 Click the **OK** button.

## 17.4 The plot styles manager

The **Plot Styles Manager** is a window that lists plot style table files (CTB and STB files).

The Plot Styles Manager allows you to:

- Create Color-dependent Plot Style (CTB) and Named Plot Style files (STB) using the **Create a Plot Style Table** wizard.
- Edit existing Plot Style Table files.

### 17.4.1 Launching the plot styles manager

Do one of the following:

- Choose **Plotstyle manager...** in the **File** menu.
- Launch the STYLESMANAGER command.

A **File** dialog box is displayed, showing the content of the PlotStyles folder, which is defined by the PLOTSTYLEFOLDER system variable.

### 17.4.2 Creating a new plot style table

1 Launch the Plot Styles Manager.

2 In the PlotStyles folder, double-click the **Create a Plot Style Table** shortcut. The **Add Plot Style Table Editor** dialog box opens.

3 Choose **Create new from scratch**.

4 Do one of the following:

- Select either **CTB** or **STB** from the **Table type** selection button.
  - **CTB** (Color-dependent Plot Style Table) - bases plots on entity colors. Plotting parameters, such as pen thickness and linetype, are matched to the color of entities. This system is older and less flexible than named plot styles, but is the default in all new drawings. Creates a CTB file.
  - **STB** (Named Plot Style Table) - bases plots on plot styles. Plotting parameters can be given to every entity and every layer. This system is new and more flexible, but may be incompatible with other software that reads DWG files. Creates an STB file.
- Choose **Create from existing table**, then choose a file in the drop down list box or click the browse button to select an existing CTB or STB file.

5 Type a name in the **Table name** field, then click the **OK** button.

The **Plot Style Table Editor** dialog box is displayed.

### 17.4.3 Creating a new plot style table in the Print dialog box

1 Launch the PRINT command.

1 The **Print** dialog box appears.

2 Under **Plot style table**, click the **Create new plot style**.

The **Add Plot Style Table** dialog box appears.

3 Type a name in the **Table name** field, then click the **OK** button.

If the **Launch Editor** option is chosen, the **Plot Style Table Editor** dialog box appears.

If the **Use this plotstyle for the current drawing** option is chosen, the new plot style table is used in the current layout.

***Прим.:*** The above procedure creates a CTB file in CTB-type drawing and a STB file in a STB-type drawing.

### 17.4.4 Editing a plot style table

1 Launch the Plot Styles Manager.

2 In the PlotStyles folder, double-click the plot style table you want to edit.

3 The **Plot Style Table Editor** dialog appears.

### 17.4.5 Editing a plot style table in the Print dialog box

1 Open the **Print** dialog box.

2 Select a plot style table from the **Plot Style Table** list button.

3 Click the **Edit plot style** button.

## 17.5 Batch plot

The PRINT command prints a single layout at a time. The PUBLISH command allows to print a sheet list (= a list of model space or paper space layouts) to a printer. A sheet list can be saved to a DSD (Drawing Set Description) file.

The SHEETSET command manages sheet sets and allows you to print, publish or etransmit a sheet set as a whole or a selection of sheets. A sheet set is saved to a DST file (Sheet Set Data) file.

# 18. Визуализация

## 18.1 Using TwinMotion

***Прим.:*** This feature is only available for BricsCAD on Windows.

Twinmotion offers real-time, immersive 3D architectural visualization using the power of Unreal Engine from Epic Games. All your BricsCAD model data are exported using the Unreal Engine file type (*.udatasmith) in the EXPORT command.

To use Twinmotion with BricsCAD V21 you need to:

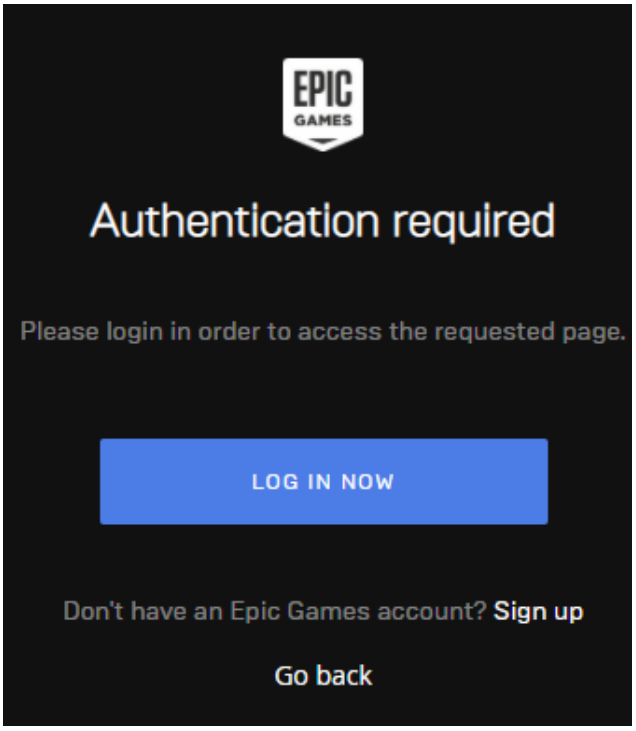
- Install the **Epics Games Launcher**.
- Download and install **Twinmotion**.
- Export your model to a ***.udatasmith** file.

### 18.1.1 Installing Twinmotion

From the Twinmotion Help pages:

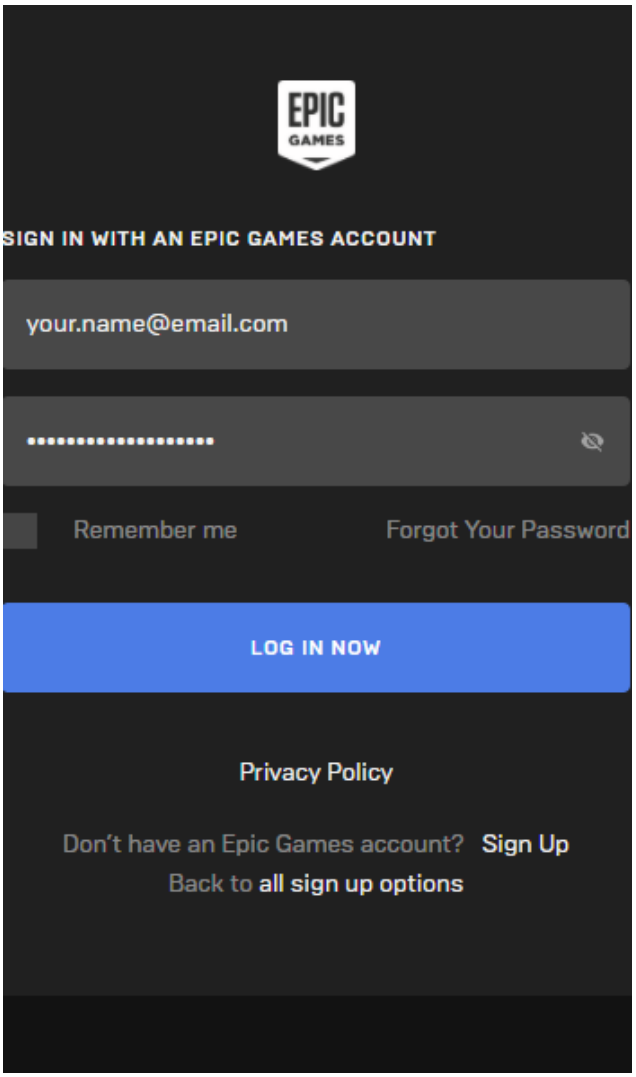
1 Go to https://www.unrealengine.com/en-US/twinmotion.

2 Click on **Get Started Now**.




If you do not have an Epic Games account yet, click **Sign up**, then fill in all required information and click **Create Account**.

3 Click **LOG IN NOW**.




4 Once you are logged in, click on **Download the Epic Games Launcher**.

5 Once the Epic Games Launcher is downloaded, make sure to click on the **Unreal Engine** tab on the left side.

You will now see the **Twinmotion** tab in the top right.

6 Click on **Install** to install Twinmotion.

7 Carefully choose your file location, it is strongly recommended to use the default location.

### 18.1.2 Exporting your model

1 Open your model in BricsCAD BIM.

2 Launch the DATASMITHEXPORT command.

3 The **Export drawing to** dialog box appears.

The **Save as type** is Unreal Engine (*.udatasmith).

4 Type a name or accept the default name, then click the **Save** button.

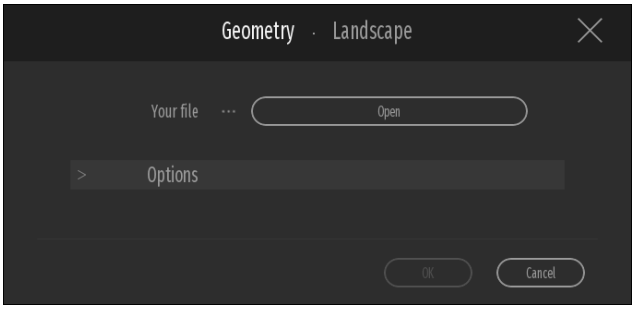
The exported model can now be imported in Twinmotion.

### 18.1.3 Importing your model in Twinmotion

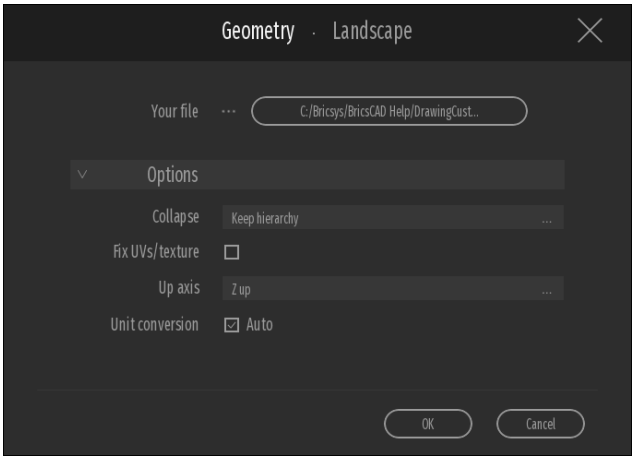
1 Launch Twinmotion.

2 Do one of the following:
- Click the **Import Files** button.
- Press **Ctrl + I**.

3 Make sure **Geometry** is selected, then click the **Open** button.




On the **Import** dialog, select a **Unreal Engine** (*.udatasmith) model file.




4 Click **Options**, and set the **Collapse** option to **Keep Hierarchy**.

This structures the model the same way as the layers used in BricsCAD.

5 Click the **OK** button to import the model.

## 18.2 Rendering overview

Rendering is the process of generating an image from a 3D model. The model contains the geometry, viewpoint, texture, lighting and shading information. The result is a digital image which can be saved as a raster image file: bmp, jpeg or png.

Rendering in BricsCAD is based on the Redway3d rendering engine. By default the rendering engine will try to leverage the hardware on the graphics card. If experiencing problems, please switch off the

RenderUsingHardware system variable. It is recommended to install the most recent drivers for your graphics card.

The RENDERPRESETS command allows you to create and edit render presets, and to set the current render preset via dialog box.

## 18.3 Rendering preferences

The RENDERPRESETS command allows you to create and edit render presets, and to set the current render preset through a dialog box.

### 18.3.1 Setting the render preferences

1 Open the **Settings** dialog box.

2 Under **Program Options**, expand the **Rendering** settings category.

**Current material**:

Specifies the material for new entities. By default, the **Global** material is used.

**Lighting units**:

- 0: No lighting units are used. Enables generic lighting.
- 1: Enables American lighting units: Foot-Candles
- 2: Enables International lighting units: Lux

Web lights can be created only if the LIGHTINGUNITS system variable is set to 1 or 2.

**Default lighting**:

A distant light that follows the view direction. This setting can be different per viewport.

- 0: Default lighting is used if no lights are switched ON.
- 1: Default lighting is used instead of the light sources defined in the drawing.

**Linear contrast**:

Specifies the ambient light intensity. Values between -10 and 10 are accepted. A value of -10 results in maximum ambient light. A value of 10 results in no ambient light.

**Linear brightness**:

Specifies a scaling factor for the light intensity. Values between -10 and 10 are accepted. The default value is 0 (no scaling). Negative values decrease the light intensity, positive values increase the light intensity. This setting can be different per viewport.

**Render using hardware**:

Choose **User software only (slower)** if hardware rendering fails. It may be necessary to restart BricsCAD after changing the setting.

**Real world scale**:

Controls the rendering of materials with units set to real-world scale.

**Sky status**:

Specifies whether sky illumination is computed at render time. (Not implemented yet)

**Texture map path**:

Defines the search path for texture map images. In the BricsCAD program folder exist three subfolders under **Textures**, each containing a number of texture files of the same name. Images in folder 1 have a size of 256 x 256 pixels, folder 2 contains images of 512 x 512 pixels, images in folder 3 have a size of 1024 x 1024 pixels. If the **Diffuse map** setting of a material uses the image name only (not path), you can control the quality of a rendered image by setting the TextureMapPath user preference to folder 1, 2 or 3.

**Tile mode light synch**:

Controls the synchronization of lighting in all model space viewports. (Internal use only)

## 18.4 Rendering tools

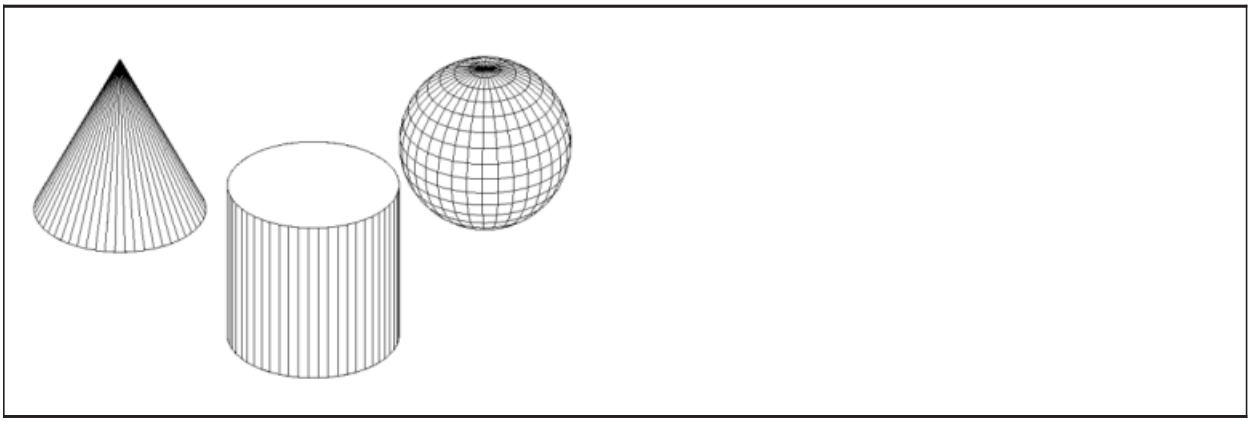
Rendering tools are available in:

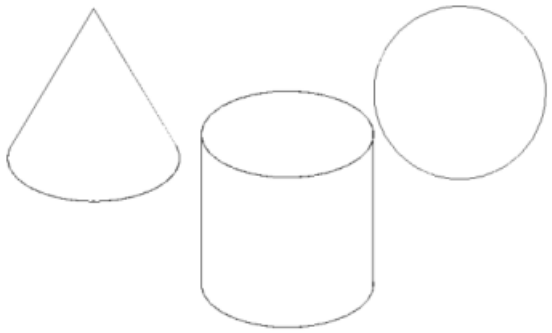
- The **Rendering** toolbar.
- In the **View** > **Rendering** menu.
- The **View** and **Visualize** ribbon tabs.
- In the **General** tab of the **No Selection** Quad.

| Icon | Tool Name | Command | Description |
|---|---|---|---|
|  | Render | RENDER | Photorealistic rendering using materials and lights |
|  | Hide | HIDE | Hidden line view (*).<br>If the previous shademode option was **2Dwireframe**, wireframe representation is restored if you zoom or pan.<br>If the previous shademode option was one other than **2Dwireframe**, the hidden line representation is kept until a different shade mode is chosen. |
|  | Shade | SHADE | Shades the entities between the polygon faces. Curved surfaces are segmented. |
|  | 2D Wireframe | SHADEMODE + 2 | Wireframe representation |
|  | 3D Wireframe | SHADEMODE + 3 | Wireframe representation |
|  | Hidden | SHADEMODE + H | Hidden line view (*) |
|  | Flat | SHADEMODE + F | Shades the entities between the polygon faces. Curved surfaces are segmented |

| Icon | Tool Name | Command | Description |
|---|---|---|---|
| | Gouraud | SHADEMODE + G | Shades the entities between the polygon faces. Curved surfaces are smoother and more realistic |
| | Flat + Edges | SHADEMODE + L | Combines flat shading and wireframe lines and curves |
| | Gouraud + Edges | SHADEMODE + O | Combines gouraud shading and wireframe lines and curves |
| | Materials | MATERIALS | Opens the **Drawing Explorer - Materials** |
| | Material Map | MATERIALMAP | Allows you to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes. |
| | Light List | LIGHTLIST | Opens the Drawing Explorer - Lights |
| | Sun Properties | SUNPROPERTIES | Opens the Drawing Explorer - Lights |
| | Geographic Location | GEOGRAPHICLOCATION | Opens the **Geographic Location** dialog box. |
| | Animation Path | ANIPATH | Records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file. |
| | Animation Editor | ANIMATIONEDITOROPEN | Opens the **Animation Editor** panel. |

(*) Set the DISPSILH system variable On to display silhouette lines instead of a polygon mesh for curved surfaces.



| Hidden line of a cone, a cylinder and a sphere with DISPSILH Off (top) and On (bottom). |
| --- |
|  |

## 18.5 Render materials

The MATERIALS command creates materials and edits their properties through the Drawing Explorer.

The MATBROWSEROPEN command opens the **Render materials** panel, which shows an overview of available high-resolution render materials.

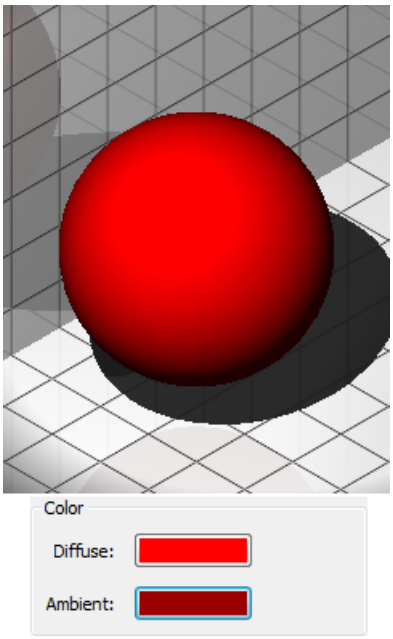
The MATBROWSERCLOSE command closes the **Render materials** panel.

The MATERIALMAP command allows you to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes.

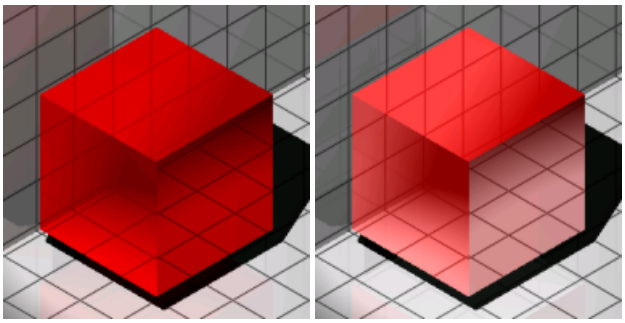
Material definitions are saved in the drawing. Each drawing contains the **Global** material, which you cannot to delete or rename. You can edit the properties of the **Global** material though.

### 18.5.1 Materials glossary

- **Diffuse color**: the color that the object reveals under pure direct, white light. It is perceived as the color of the object itself.
- **Ambient color**: the color of the object in the shadow, under ambient light rather than direct light.

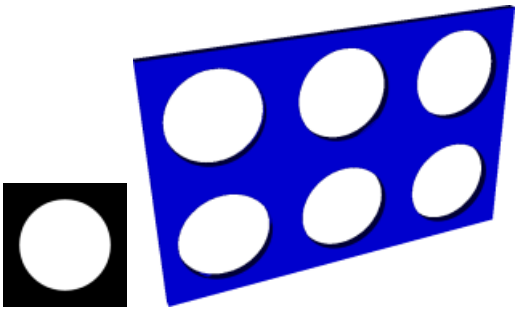


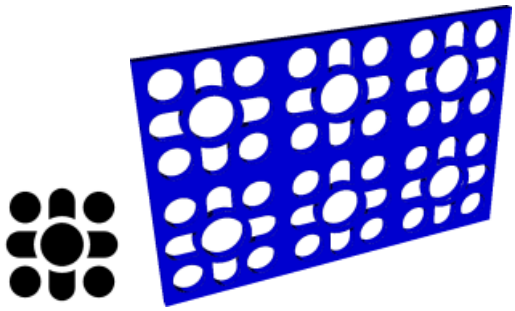

- **Highlights**: highly reflective highlights in the color of the material.



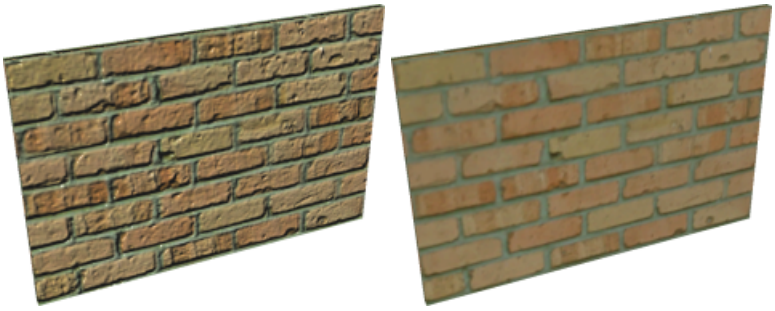
Metallic Highlights (left) and Non-metallic Highlights (right)

- **Shading**: how the color and brightness of a surface varies with lighting.
- **Texture-Mapping**: adds detail to a surface that is not included in the 3D model itself.
- **Diffuse map**: adds a structure or pattern to a surface to simulate a material.
- **Transparency map**: sets the transparency of a texture from 0% (black) to 100% (white) and anything between (grayscales). (Black=Transparency map)





- **Bump map**: simulates a certain roughness or bumpiness on surfaces. (left = with bump map)



- **Reflectivity**: the property of a surface to reflect light.
- **Transparency**: sharp transmission of light through solid objects.
- **Translucency**: scattered transmission of light through solid objects.
- **Refraction**: bending of light when passing through transparent objects.
- **Self-illumination**: it appears as if a surface is emitting its own light. No light is cast on other objects.

### 18.5.2 Defining render materials

See the MATERIALS command.

### 18.5.3 Assigning render materials

Materials can be assigned to a layer and to entities or subentities. If the **Material** property of an entity is **BYLAYER**, the material assigned to the layer of the entity will be used.

By default, new entities get the **BYLAYER** material setting.

### 18.5.4 Adding a render material to the drawing

1 Select the material in the **Render Materials** panel.

2 Right-click and choose the **Add to drawing** option.

### 18.5.5 Assigning a render material to entities

When the current visual style is BIM, Modeling, Realistic or X-Ray, entities and subentities are rendered using their material.

**Method 1: Using the Properties panel**

1 Select the entities or sub-entities (e.g. faces or 3D solids).

2 Under **3D Visualization** > **Material**, click the down arrow and select the material in the **Material** list.

***Прим.:*** Materials in the drawing are available only.

**Method 2: Using the Render Materials panel**

1 Select the material, which launches the MATERIALASSIGN command.

1 The cursor turns into a paintbrush.

**You are prompted**: Select object to assign material to:

2 Click the entities or subentities to assign the material.

3 (option) Select a different material, then click more entities or subentities.

4 (option) Hold down the Alt key.

5 The cursor turns into a dropper.

**You are prompted**: Select material from object:

Click an entity or subentity to copy the material from, then click entities or subentities to assign the material to.

***Прим.:*** The Hotkey Assistant appears at the bottom of the screen, indicating the current selection mode: 3D solids or 3D solid faces. Hold down the Ctrl key to invert the selection mode.

To assign a material to a selection set:

1 Select the entities.

2 Search the material in the **Render Materials** panel.

3 Click the material.

4 The material is assigned to all entities or subentities in the selection.

### 18.5.6 Removing a render material

Assigning the Global material, removes a rendering material from an entity or subentity.

1 Select the entities and/or subentities.

2 Select the **3D Visualization / Material** field in the **Properties** panel and select the **Global** material.

***Прим.:*** Clicking the **Erase Material** tile in the **Render Materials** panel sets the Material property to BYLAYER.

### 18.5.7 Assigning a material to a layer

1 Select the **Material** field of the layer in the **Layers** panel or the **Drawing Explorer – Layers** dialog box.

2 Click the down arrow and select the material in the list.

3 Materials in the drawing are available only.

### 18.5.8 Adding the Redway material library

Materials in the folder linked under **RENDERMATERIALSPATH** appear in the **Render** materials panel. To add the **Redway Material Library**, follow this procedure:

1 Download the complete Redway Material Library here.

2 Unzip the downloaded file and copy the path of the folder. The path should end with ...

\all_redway_materials.

3 In the **Render Materials** panel, click the hamburger menu ( ).

4 Select **Manage libraries** in the menu.

The **Settings** dialog box is displayed with the RENDERMATERIALPATH system variable selected.

5 Click the Browse button ( ).

The **Path List** dialog box appears.

6 Click the **Add Path** button.

An empty path field is added.

7 Do one of the following:

- Paste the path you have copied in the empty path field.
- Click the Browse button ( ), then select the folder in the **Choose a folder** dialog box.

The downloaded materials appear in the **Render Materials** panel.

### 18.5.9 Adding your own materials

1 Download the textures you want to use.

***Прим.:*** you can download a photo of your texture, but in order to get the most realistic view it is best to download a zip file that gives you different representations of the texture:




2 Launch the MATERIALS command.

3 The **Drawing Explorer – Materials** dialog box appears.

4 Click the **New** button.

5 Give your new material a name.

6 Check on the **Diffuse Map** and the **Bump Map**.

7 Edit the path of the diffuse map ( ) to the colored image of your texture.

8 Edit the path of the bump map ( ) to the normal image of your texture.

This will give your render material a more 3D and realistic look.

9 Edit the scale of your image until you get a realistic image in the preview.

10 The scale of the texture represents the number of drawing units covered by the full width of your image.

11 Right-click on your new material and choose **Add material(s) library** in the context menu.

12 Close the **Drawing Explorer** dialog box.

13 Open the **Render Materials** panel and click on the folder **User Defined**.

You find your new material in this folder of the library.

This folder is automatically generated in the following path: C:\ProgramData\Bricsys\RenderMaterials\UserMaterials\User defined.

## 18.6 Lights

Lighting is the key to make the viewer believe they are looking at a realistic scene.

### 18.6.1 Lighting glossary

- **Ambient light**: light from an unspecified source.
- **Back light**: light that adds depth and dimension by creating highlights.
- **Distant light**: light that is cast evenly throughout a scene with shadows all in the same direction.
- **Fall-off**: the attenuation or decrease in brightness the further from the light source.
- **Fill light**: the light that brightens dark areas and softens shadows from the main light.
- **Point light**: light that shines in all directions from a central spot.
- **Spot light**: focusable light that is aimed in one area.
- **Photometric web**: a 3D representation of the light intensity distribution of a light source.
- **Web light**: approximates real-world light distribution using a 3D representation of the light intensity. Web lights can be created only if the LIGHTINGUNITS system variable is set to 1 (American lighting units) or 2 (International lighting units).
- **Attenuation**: the fall-off in brightness the further from the light source.

### 18.6.2 System variables

LIGHTINGUNITS (Lighting units):

- 0: No lighting units are used. Enables generic lighting.
- 1: Enables American lighting units: Foot-Candles.
- 2: Enables International lighting units: Lux.

DEFAULTLIGHTING (Default lighting): A distant light that follows the view direction. This setting can be different per viewport.

- 0: Default lighting is used if no lights are switched ON.
- 1: Default lighting is used instead of the light sources defined in the drawing.

LINEARCONTRAST (Linear contrast): Specifies the ambient light intensity. Values between -10 and 10 are accepted. A value of -10 results in maximum ambient light. A value of 10 results in no ambient light.

LINEARBRIGHTNESS (Linear brightness): Specifies a scaling factor for the light intensity. Values between -10 and 10 are accepted. The default value is 0 (no scaling). Negative values decrease the light intensity, positive values increase the light intensity. This setting can be different per viewport.

LIGHTGLYPHDISPLAY (Light glyph display): Controls the display of light glyphs.

FBXEXPORTLIGHTS (Fbx Export Lights): Specifies whether to export lights from model space.

### 18.6.3 General properties

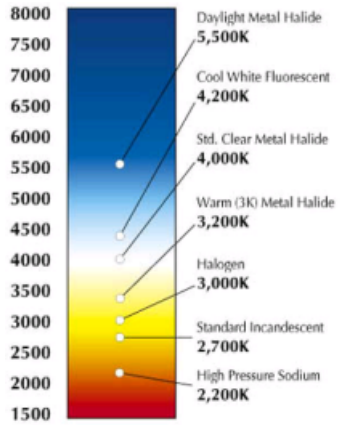
The general properties are common to all lights:

- **Name**: a user defined name for the light.
- **Type**: defines the type of light: spot, point, web or distant.
- **On/Off**: specifies whether the light is turned on or off.
- **Shadows**: specifies whether the light casts shadows or not. Turning shadows off to increases performance.
- **Intensity factor**: multiplies the **Lamp Intensity** property, the result is the **Resulting Intensity**.
- **Filter color**: defines the color of the light.
- **Plot glyph**: if on, the light glyph is plotted.
- **Glyph display**: controls the display of the light glyph.

### 18.6.4 Photometric light properties

LIGHTINGUNITS = 1 (American units) or 2 (International units).

- **Lamp intensity**: Defines the brightness of the light. The lamp intensity is expressed in candela (cd) which is the SI base unit of luminous intensity: the power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function).
- **Resulting intensity**: the product of the lamp intensity and and intensity factor.
- **Lamp color**: defines the inherent color of the light or color temperature in Kelvin or standards.

| | |
|---|---|
|  | D65White<br>Fluorescent<br>Cool White<br>White Fluorescent<br>Daylight Fluorescent<br>Incandescent<br>Xenon<br>Halogen<br>Quartz<br>Metal Halide<br>Mercury<br>Phosphor Mercury<br>High Pressure Sodium<br>Low Pressure Sodium |
| color temperatures | lamp color list |

- **Resulting color**: this is the final color of the light: a combination of the lamp color and the filter color.
- **Photometric web**: If the light type is **Web**, **Photometric Web** and **Web offsets** are available (not implemented yet).

### 18.6.5 Geometry properties

- **Position**: controls the location of the light.
- **Target**: defines the target point for point lights, spotlights and weblights.
- **Targeted**: switches the **target** property on/off.

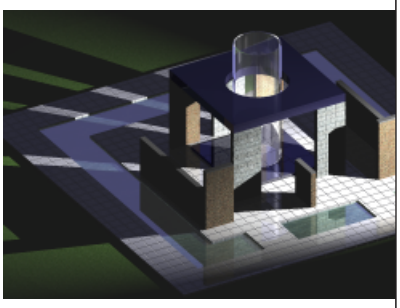
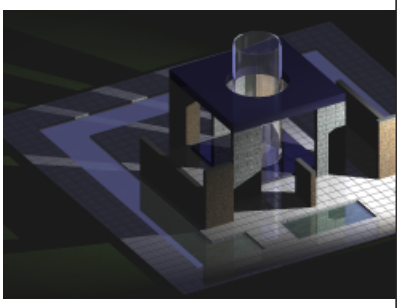
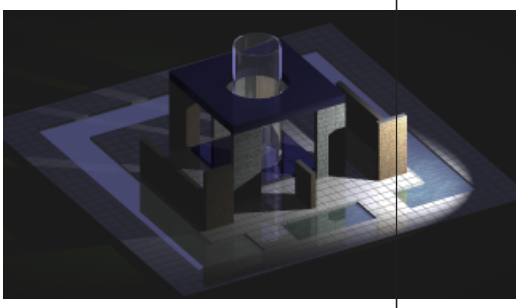
### 18.6.6 Attenuation properties

In the real world, an object appears darker if it is farther away from the light source. The attenuation properties define how the light diminishes over distance. Attenuation does not apply to photometric lights.

An alternative method to control the distance a light shines is the use of limits, which define the points from where a light starts to shine and where it stops. Using limits decreases the time needed to calculate the illumination of a scene.

Attenuation properties apply to spot lights and point lights.

- **Type**: the options are **none**, **inverse linear** and **inverse square**
  - **None**: no attenuation. The distance to the light source has no influence.
  - **Inverse Linear** : the attenuation is the inverse of the linear distance from the light: at a distance of 2 units from the light source, light is half as strong; at a distance of 4 units, light is one quarter as strong.
  - **Inverse Square**: the attenuation is the inverse of the square of the distance from the light: at a distance of 2 units, light is one quarter as strong; at a distance of 4 units, light is one sixteenth as strong.
- **Use Limits**: if this property is set to Yes, the **Start limit offset** and **End limit offset** fields become active.
  - **Start limit offset**: defines the point where the light starts to shine, measured from the center of the light.
  - **End limit offset**: defines the point where the light stops to shine, measured from the center of the light.

| | | |
|---|---|---|
|  |  |  |
| No attentuation | Inverse linear attentuation | Inverse square attentuation |

***Прим.:***

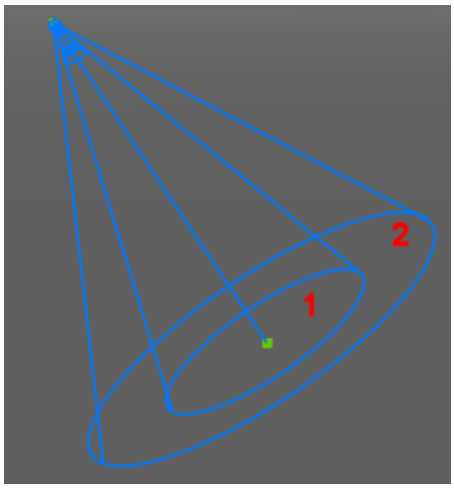
- The **Attenuation** properties apply to generic lights only (LIGHTINGUNITS = 0).
- Inverse square attenuation is applied automatically for photometric lights ( LIGHTINGUNITS = 1 or 2).

### 18.6.7 Rendered shadow details

- Type: Sets the shadow type.
- Sharp (raytraced): generates sharp edged shadows.
- Soft (shadow map): generates realistic shadows with soft edges.
- Soft (sampled): Not supported yet
- Map size: specifies the amount of memory for shadow calculation. Click the field, then select a value in the drop-down list (64/128/256/512/1024/2048/4096).
- Softness: sets the softness of the shadow map. Enter a value between 1 and 10.

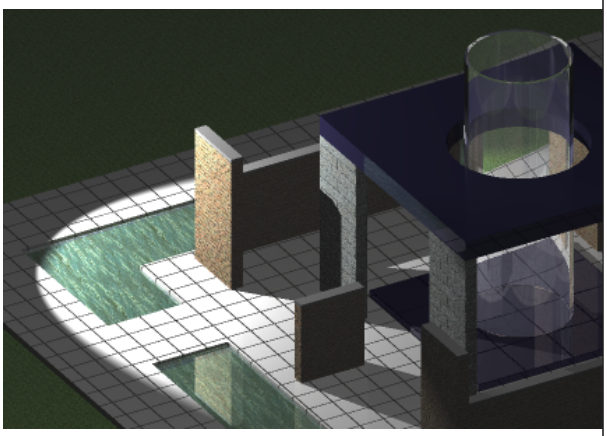
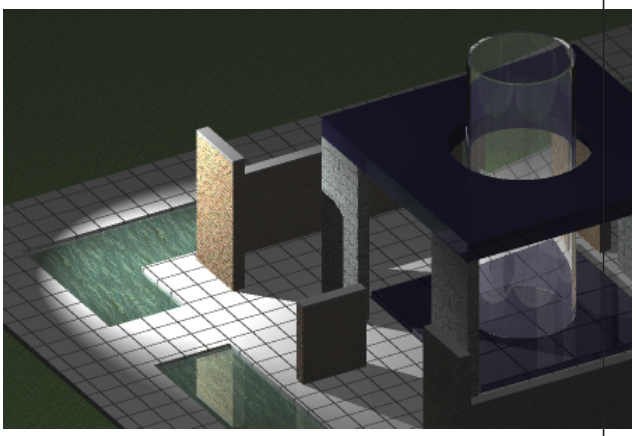
### 18.6.8 Spotlight hotspot and falloff properties

When a surface is illuminated by a spotlight, there is an area of maximum illumination (hotspot = 1) that is surrounded by an area of lesser intensity (falloff = 2).




**Hotspot angle**: angle of the central light cone (defines the hotspot)

**Falloff angle**: angle of the full light cone

| | |
|---|---|
|  |  |
| Hotspot angle = 55, Falloff angle = 60 | Hotspot angle = 30, Falloff angle = 60 |

The difference between the hotspot angle and the falloff angle defines the area of lesser light intensity.

If the falloff angle and the hotspot angle are almost equal the edge of light cone is rather sharp (left).

The greater the difference between both angles, the softer the edge of the light cone (right).

### 18.6.9 Managing Lights

The LIGHTLIST command opens the **Drawing Explorer – Lights** dialog box.

Here you can:

- Create new lights.
- Delete lights.
- Select lights in the drawing.
- Switch lights On/Off.

### 18.6.10 Defining a light

1 Do one of the following:
   - Launch the LIGHT command.
   - Click the New () button on the **Drawing Explorer – Lights** dialog box.

     **You are prompted**: Enter light type [Point/Spot/Web/Distant] <Web>:

     Specify the light type.
   - Click a toolbutton on the **Lights** toolbar or the **View** or **Visualize** ribbon tab.
   - : spot light
   - : point light
   - : distant light
   - : web light

   **You are prompted**: Specify source position <0,0,0>:

2 Specify a point in the drawing.

3 **You are prompted**: Specify target position <0,0,-10>:

4 Specify a point in the drawing.

5 **You are prompted**: Enter an option to change: Name/Intensity factor/Status/Photometry/Hotspot/Falloff/Shadow/Attenuation/FilterColor/<eXit>:

6 Specify the properties of the light.

7 Choose **Exit** to conclude the command and create the light.

   A light glyph is displayed in the model, if LIGHTGLYPHDISPLAY = On.

***Прим.:*** It is not possible to define web lights if **LIGHTINGUNITS** =0 (No lighting units).

### 18.6.11 Editing lights

Lights can be edited in the **Properties** panel.

1 Do one of the following to select the light in the drawing:
   - Click the glyph of the light(s).
   - Select the light(s) in the **Drawing Explorer – Lights**, then right-click and choose **Select lights** in the drawing in the context menu.

   The properties of the selected light(s) appear in the Properties panel:

2 Edit the properties.

3 Press the Esc key to stop.

### 18.6.12 Defining the sun properties

1 Launch the SUNPROPERTIES command.

1 The **Drawing Explorer - Lights** dialog appears, showing the sun properties in the **Editor** pane.

2 Edit the properties.

***Прим.:*** When LIGHTINGUNITS = 1 or 2, the **Color** property of the sun cannot be edited.

### 18.6.13 Defining the geographic location

1 Launch the GEOGRAPHICLOCATION command.

1 The **Geographic Location** dialog box appears.

2 (option) Check the **Store geographic location information in drawing** option.

3 (option) Check the **Use Map Grid coordinate system** option.

4 Do one of the following:
   - Type the latitude and longitude of the location in the **Latitude** and **Longitude** fields and select the appropriate hemisphere for each setting.
   - Click the **Choose location...** button, then select a city and timezone and click the **OK** button.

5 (option) If you did not select a time zone in the previous step, choose a time zone in the **Time Zones** list.

6 (option) Specify the **Coordinates of the corresponding location in the drawing**.

7 Define the direction of the north.

8 Click the **OK** button.

## 18.7 Creating a rendered image

The rendered image appears in the current viewport in a separate window or can be exported to a file.

The background of a rendered view can be controlled by the Background property of the current named view.

The ANTIALIASRENDER system variable controls the amount of anti-aliasing (edge smoothing) applied during the rendering generation. The default value is 1, maximum value is 5.

***Прим.:***

- High anti-alias values incur a high calculation cost.
- The values of the lighting system variables influence the rendering result.

### 18.7.1 Rendering a viewport

1 Launch the RENDER command ( ).

The **Render** dialog box appears.

2 Click the **Viewport** option.

3 Click the **OK** button.

The view is rendered.

### 18.7.2 Rendering in a window

1 Launch the RENDER command ( ).

The **Render** dialog box appears.

2 Click the **Window** option.

3 Click the **OK** button.

The view is rendered in a popup window.

### 18.7.3 Saving a rendering to a file

1 Launch the RENDER command ( ).

The **Render** dialog box appears.

2 Click the **File** option.

3 Click the **Browse** button ( ), at the right hand side of the **File** field. The **Render output file** dialog box appears.

4 Select a folder and type a name for the output file in the **File name** field.

5 Click the **Save as type** button to select the file type.

The options are: **bmp**, **jpg** and **png**.

6 Click the **OK** button.

The **Render Output File** dialog closes.

7 Do one of the following:

- Type the desired resolution for the output size in the **Width** and **Height** fields.
- Check the **Same as viewport** size option to set the outsize to the size of the current viewport.

8 Click the **OK** button to create the file.

# 19. Обратная связь

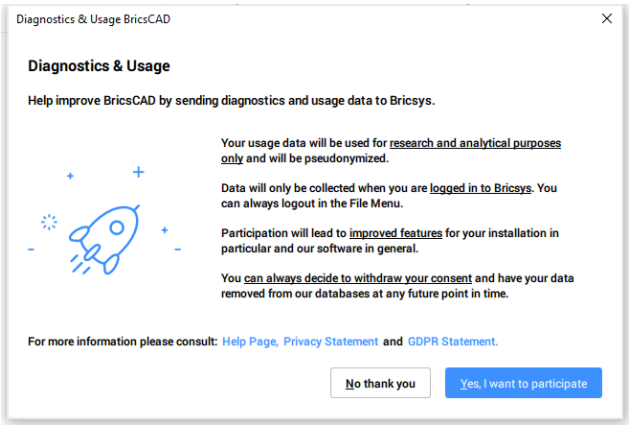
## 19.1 Diagnostics and usage data collection

The Diagnostics and Usage Data Collection process allows users to help improve BricsCAD by sending pseudonymized diagnostics and usage data to Bricsys. You can decide at any time to withdraw your consent and have your data removed from our databases.

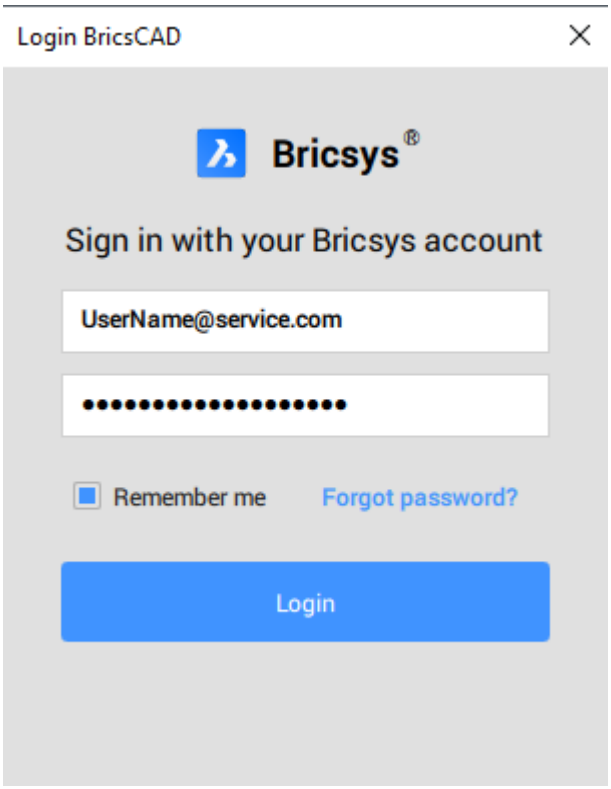
When transferring data:

- A unique, anonymous ID is used.
- The following data and information are removed:
- All personal project information .
- All texts and leaders.
- All information that is stored in the Settings dialog.

When you launch BricsCAD the second time, a dialog box displays asking you to participate.




When you agree, you are prompted to log in with your Bricsys account.




If you tick the**Remember me** checkbox, you will be signed in automatically each time you launch BricsCAD. The status of your agreement is saved through theDATACOLLECTIONsystem variable as indicated in the table below.

| Value | Status | Result |
|---|---|---|
| -2 | Not Asked | When you close BricsCAD the value is set to [-1] Not Set. |
| -1 | Not Set | When you restart BricsCAD the**Diagnostics & Usage**dialog displays, asking you to participate. |
| 0 | Off | The Diagnostics and Usage Data Collection process is switched Off. |
| 1 | On | The Diagnostics and Usage Data Collection process is switched On.<br>Data will be transferred when you are logged in only.<br>To temporarily stop transferring data, choose**Logout...**in the**File**menu or launch the-LOGOUTcommand.<br>To restart transferring data, choose**Login...**in the**File**menu or launch the-LOGINcommand. |

### 19.1.1 Stopping the process

1 In the Settings dialog, under User Preferences, setDATACOLLECTION to[0] Off.

2 Close and restart BricsCAD.

### 19.1.2 Restarting the process

1 In the Settings dialog box, under User Preferences, setDATACOLLECTIONto [-1] Not Set.

2 Close and restart BricsCAD.

3 Choose**Yes, I want to participate**, then log in with your Bricsys account.

### 19.1.3 Removing your data from the database

To withdraw your consent and have your data removed from our databases send in a support request and select the **Data Collection** category under BricsCAD.

# 20. BricsCAD BIM

## 20.1 Что нового

### 20.1.1 Overview

BricsCAD V21 offers a plethora of new and improved tools to help maximize your productivity.

This section gives an overview of all the new features and improvements in **BricsCAD V21 BIM**:

Modeling techniques 970

Building data 977

Project collaboration 986

Design documentation 987

Point clouds 996

For information about the new features and improvements in **BricsCAD V21 Core and Civil tools**, go to the What's new in BricsCAD V21 section. It contains the following articles:

Overview 31

User interface 32

Productivity 35

Drafting 49

Modeling 51

Civil tools 60

Performance and compatibility 65

For information about the new features and improvements in **BricsCAD V21 Mechanical tools**, go to the What's new in BricsCAD V21 Mechanical section. It contains the following articles:

Обзор 1540

Диспетчер спецификаций 1541

Библиотеки стандартных изделий 1544

Параметры и зависимости 1545

Стандартные детали 1545

Листовой металл 1545

#### 20.1.1.1 Release notes

Click here for an overview of the release notes.

#### 20.1.1.2 Diagnostics and Usage Data Collection

Many of the improvements were inspired by requests and suggestions from you, BricsCAD users. At Bricsys, we continue to expand our methods of gathering user feedback with new data collection tools. When you launch BricsCAD for the second time, a dialog box displays, inviting you to participate in the

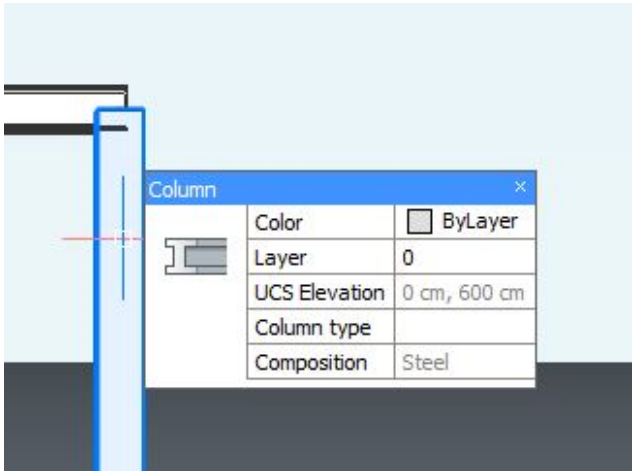
Diagnostics and Usage Data Collection program. The data collection process enables you to share anonymous diagnostic and usage data with Bricsys to help improve BricsCAD. You can withdraw your consent and have your data removed from our databases at any time.

### 20.1.2 Modeling techniques

#### 20.1.2.1 BIM Profiles

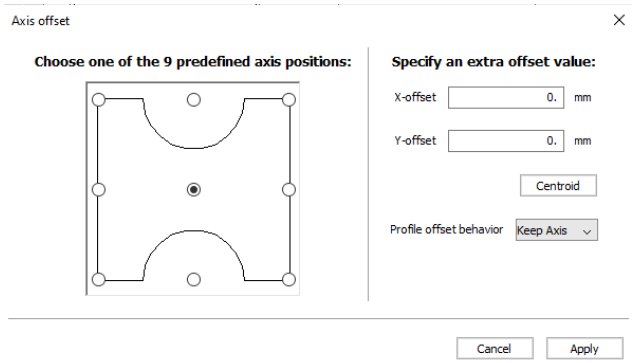
**BIMAPPLYPROFILE / BIMLINEARSOLID**

The composition material of a profile is now attached to the created solid in the BIMAPPLYPROFILE command and BIMLINEARSOLID commands.




**BIMPROFILES**

When changing the Offset property of a CustomShape profile via the Profiles dialog, you can choose whether to keep the position of the solid or its axis, for linear solids using said profile.




**BIMPROFILESTANDARDS**

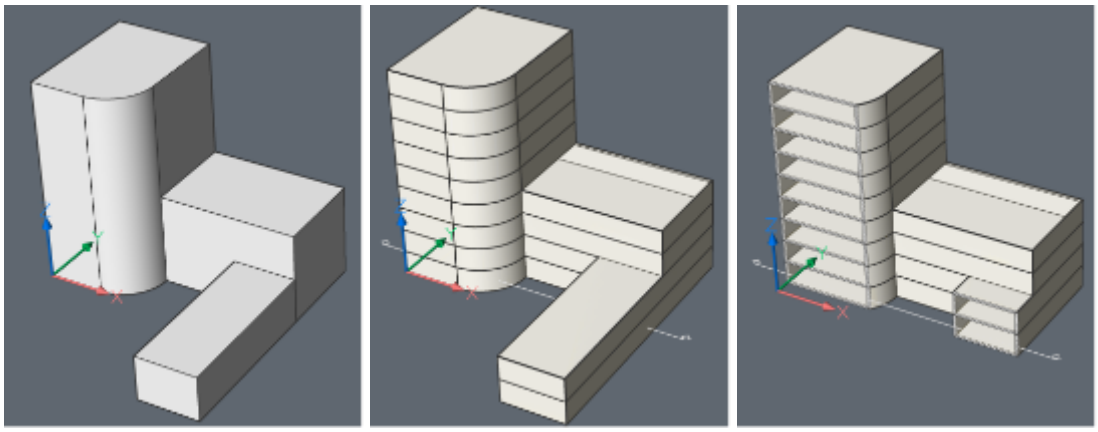
The BIMPROFILESTANDARDS setting is no longer case sensitive. In case there are no profiles matching the value of the BIMPROFILESTANDARDS setting, a warning message displays.

### 20.1.2.2 Mass Modeling tools

The first prototype of the new BIMQUICKBUILDING command has been implemented. The QuickBuilding tool allows you to generate a BIM building just from one massing solid. It automatically generates different levels, walls, spatial locations, etc. It can be very useful in early design stages, volume studies and conceptual design. The tool will assign BIM properties to each part of the new building.

Spatial Locations can now be used as input for BIMQUICKBUILDING. This workflow allows you to process different buildings simultaneously.

The command also works on solids without edges, for example, a sphere or a torus.



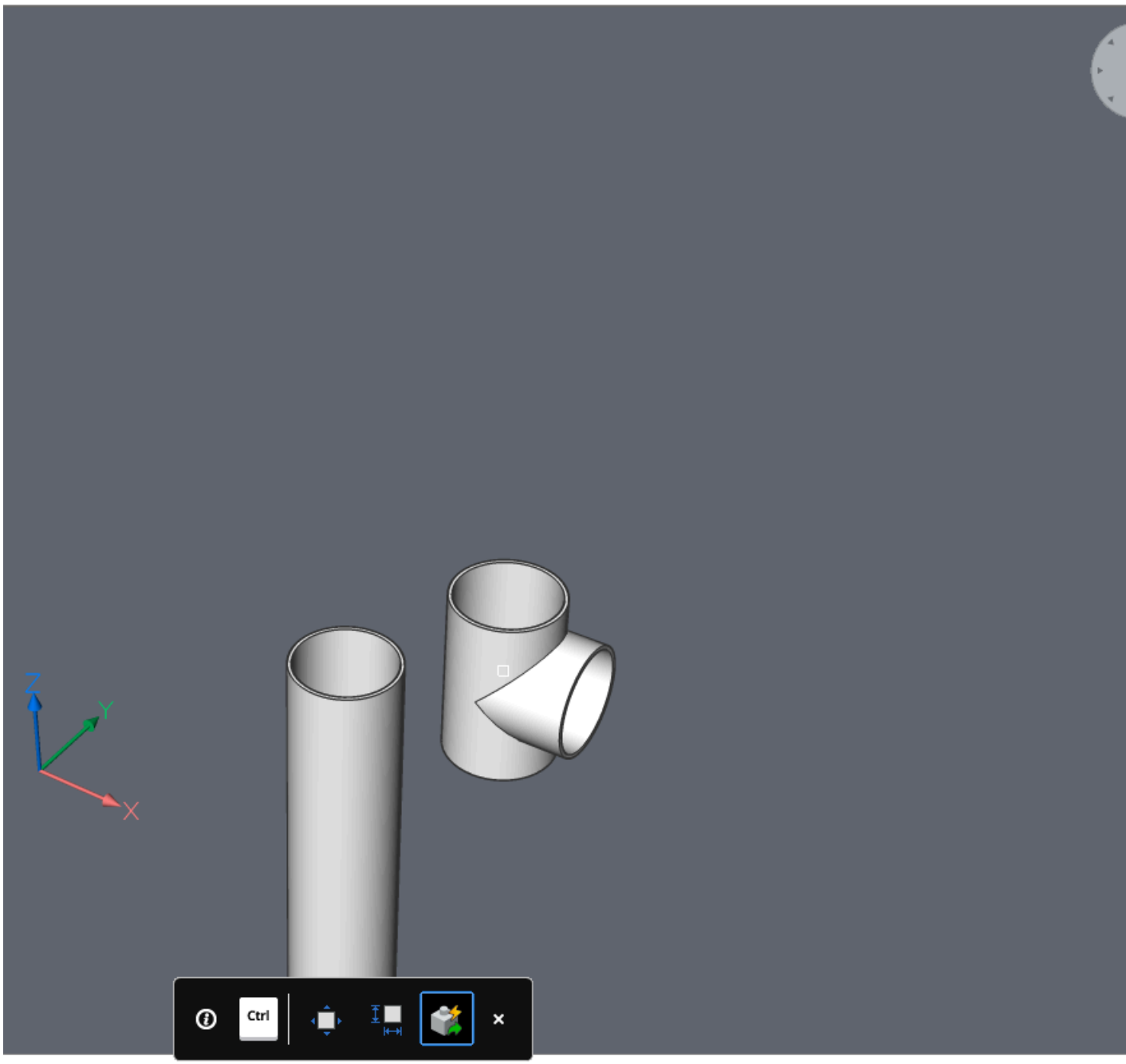
### 20.1.2.3 Insert BIM Component

The BMINSERT command now includes a SMART insert option, which automatically adjusts parts to match current and relevant parts in the model file when inserted.

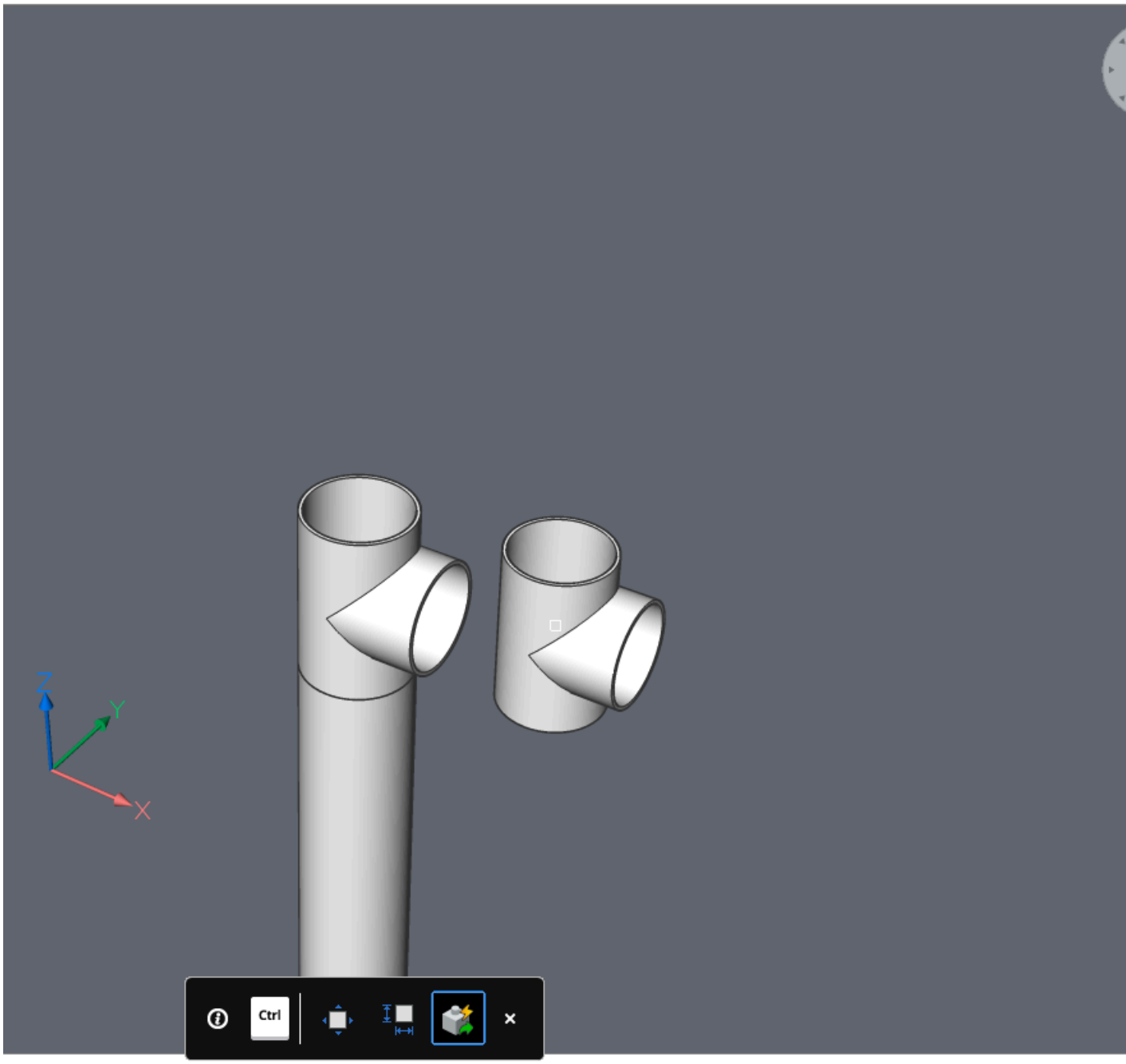
You can select SMART insert by clicking the option in the Command line, or toggle CTRL to the rightmost icon when prompted in the HotKey Assistant.

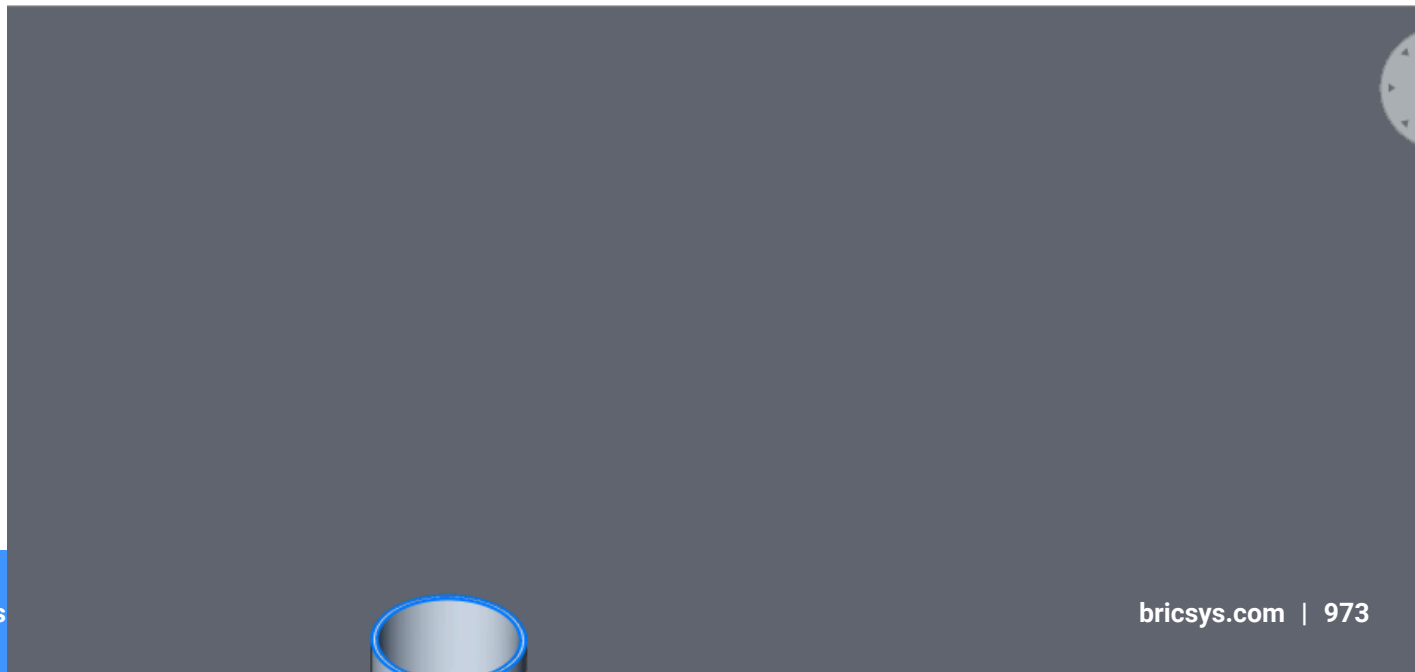



By detecting and creating 3D constraints between the relevant connection points, SMART insert resizes and attaches your desired component onto the current component/s with assembly constraints and common parameters to form a bigger part of a parametric assembly. This is particularly useful in modeling parametric MEP piping systems.


Z
Y
X
Ctrl

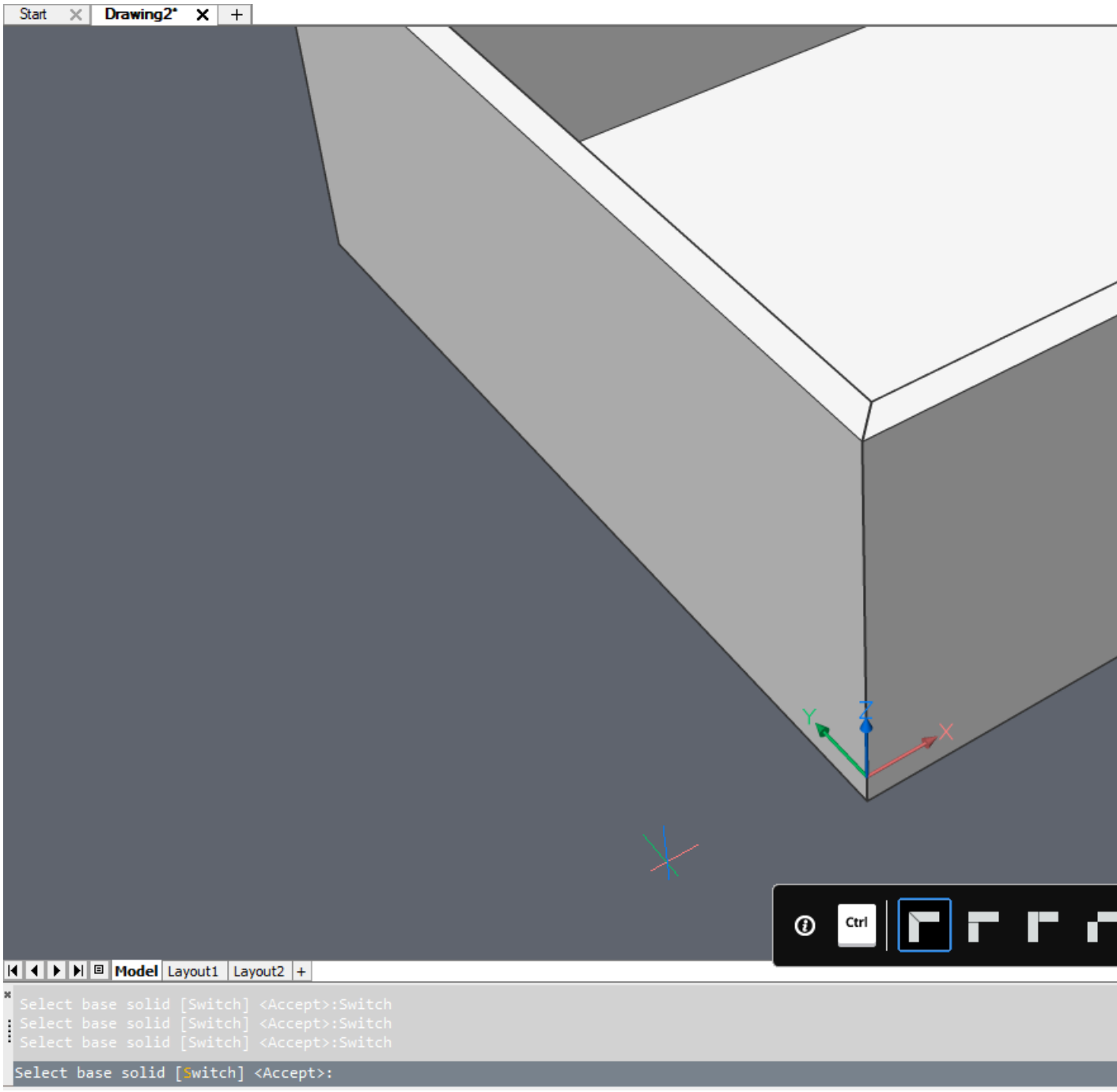


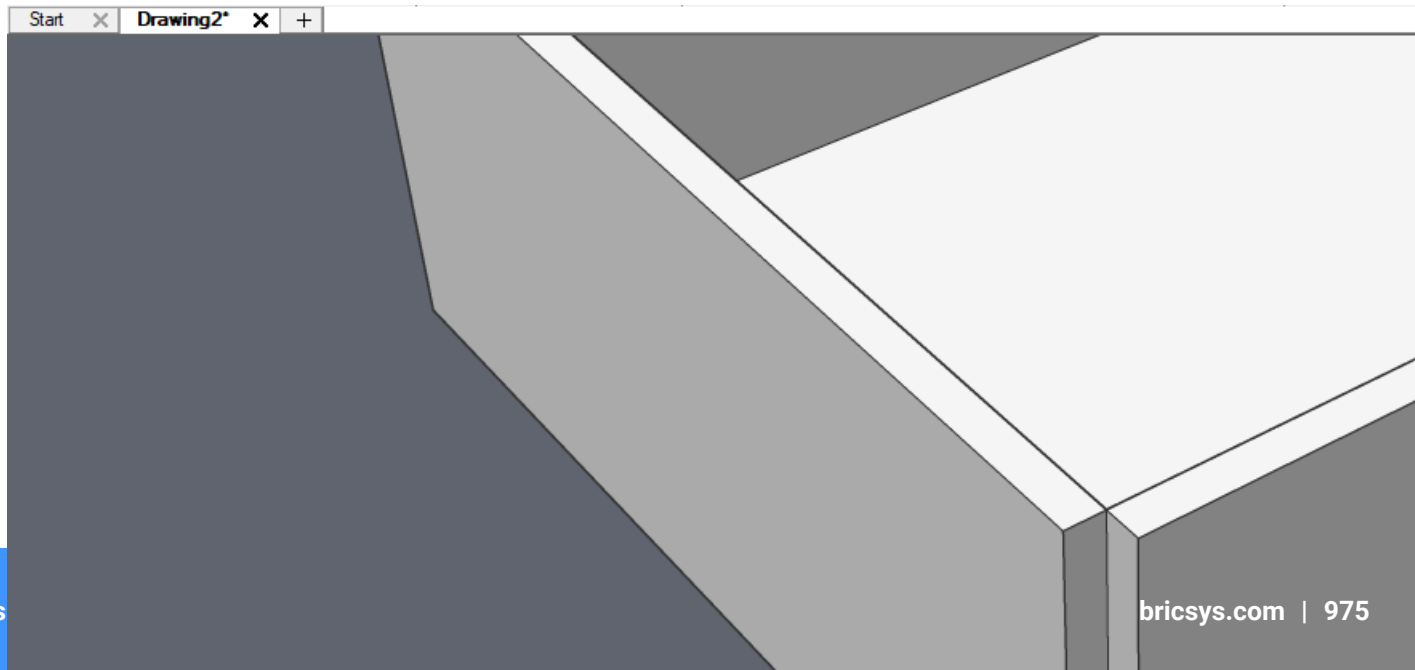

Z
Y
X
Ctrl

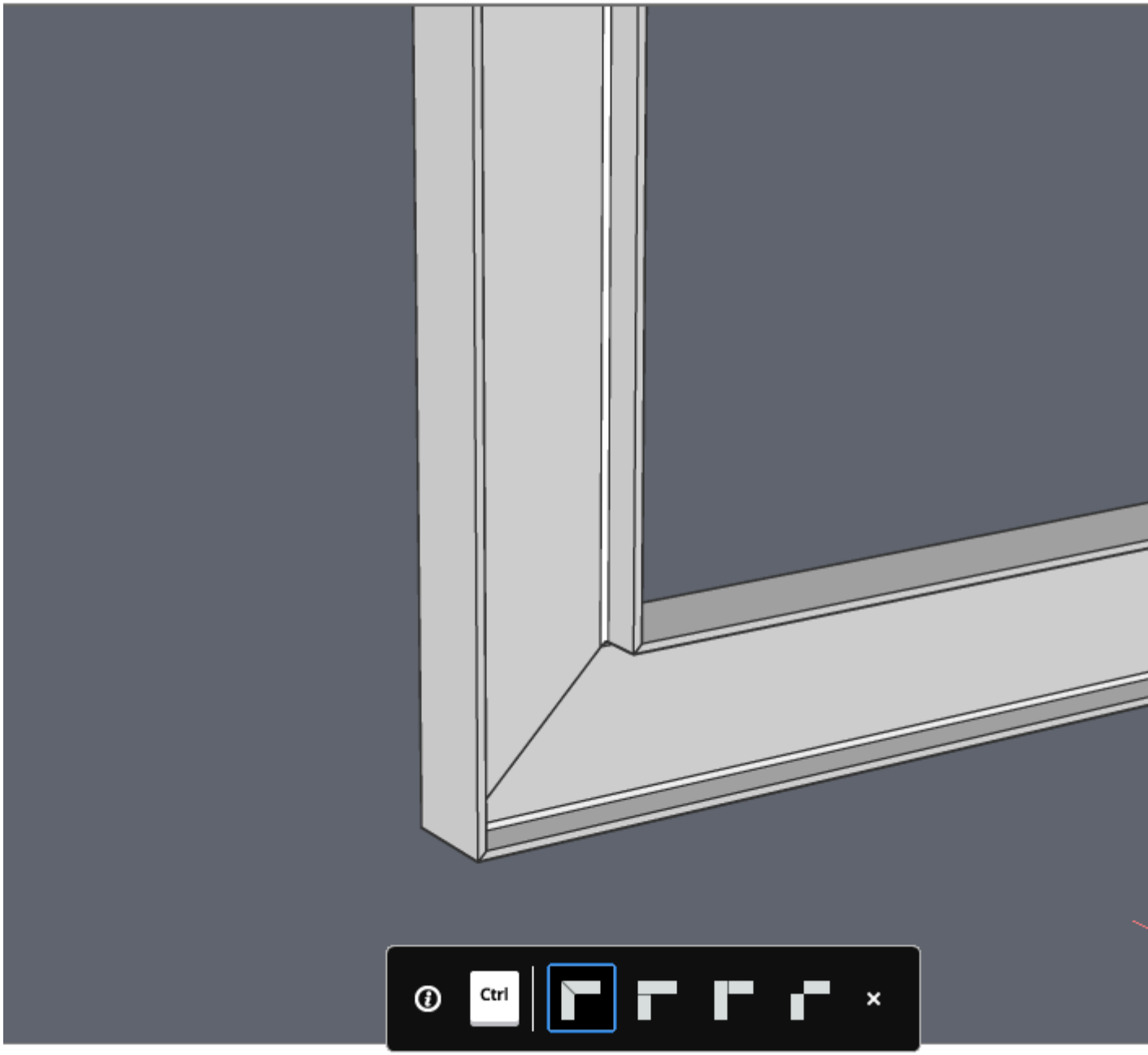


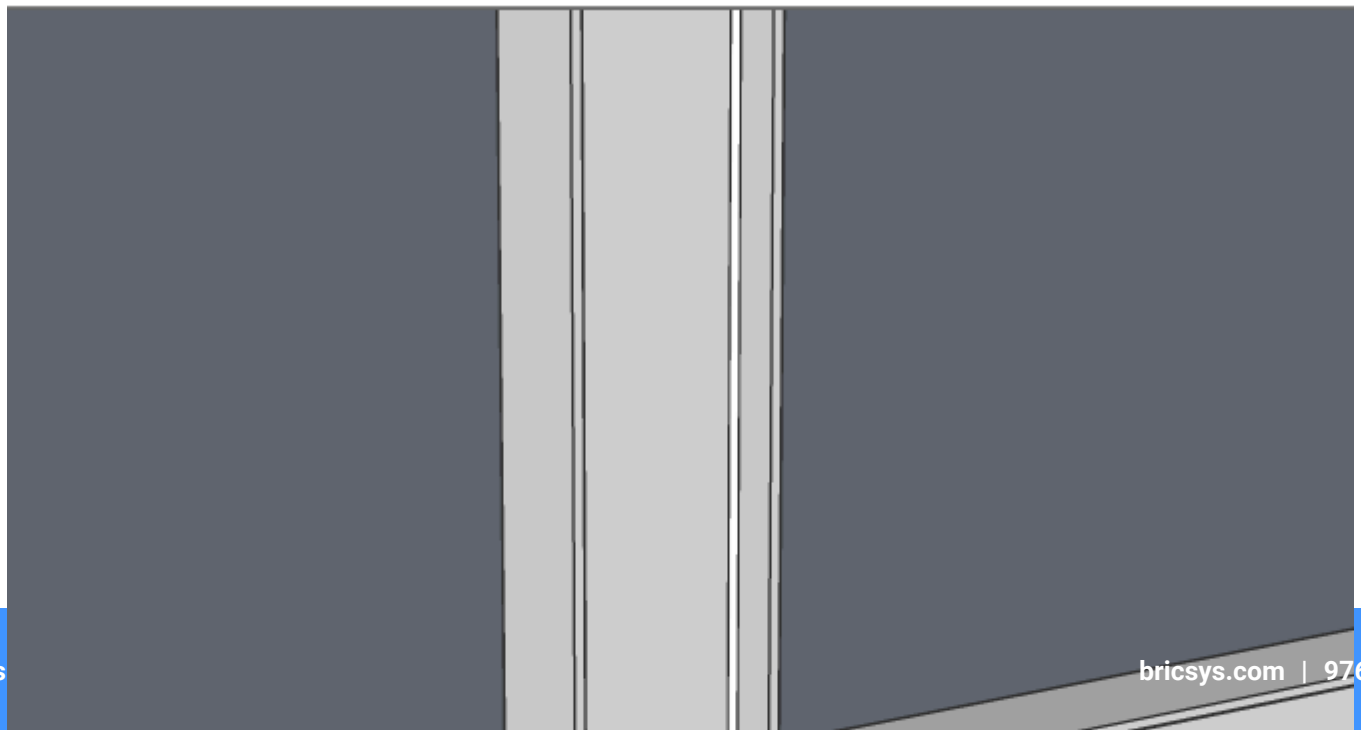
#### 20.1.2.4 Disconnect

The LCONNECT and BIMSTRUCTURALCONNECT commands now include a fourth option: DISCONNECTED. You can cycle through the options with the CTRL button in the HotKey Assistant.


Start
Drawing2*
Y
Z
X
Ctrl
Model
Layout1
Layout2
Select base solid [Switch] <Accept>:Switch
Select base solid [Switch] <Accept>:Switch
Select base solid [Switch] <Accept>:Switch
Select base solid [Switch] <Accept>:


Start
Drawing2*


Ctrl

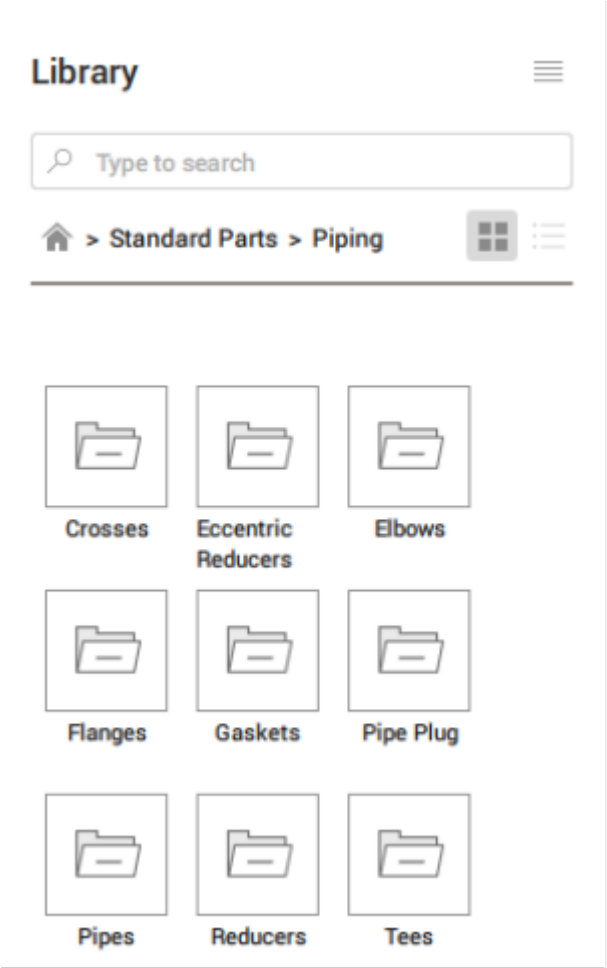


#### 20.1.2.5 Quickdraw

The Quickdraw cursor now supports snapping to (double) lines inside XREFs. Walls now inherit the spatial location of the slab on which they are created.

#### 20.1.2.6 Standards Parts

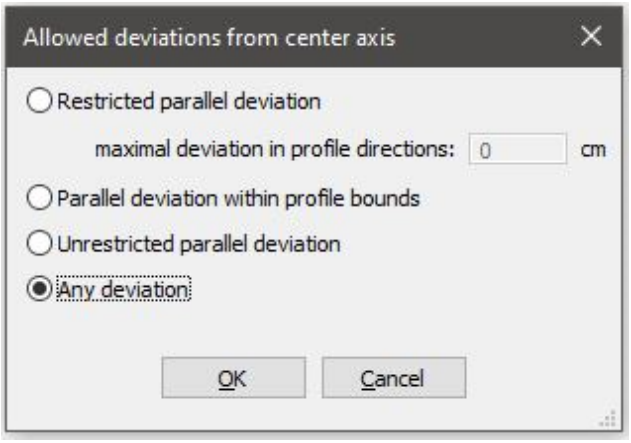
Piping parts were added to the standard parts category of the Library panel.



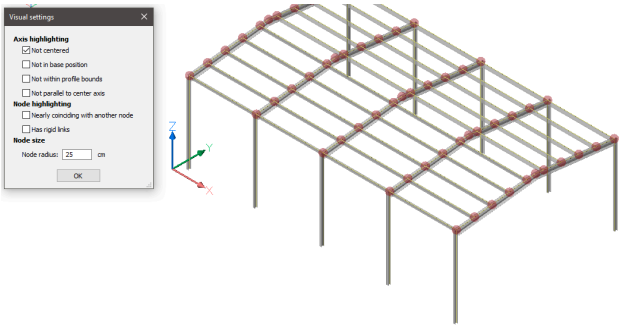

### 20.1.3 Building data

#### 20.1.3.1 Analytical Model

This command allows you to generate an analytical model from a full 3D model of a structure. The command starts with an automatically generated proposal, but allows you to further adapt the model to your desire.




The automatic proposal will try to minimize rigid links (eccentricities) as much as possible by shifting, extending and even rotating some axis lines. You can control the amount of such changes by the deviation settings.



After editing the model, you can export it to an IFC.

### 20.1.3.2 BIM API

Configurable quantities are now available in the BIM API, as well as the BIM API for Spaces.

### 20.1.3.3 BIM Compositions

In V21, the render material display and LOD setting have been separated. A new button has been added in the Home/View ribbon panel, next to the LOD button to toggle the rendering composition materials.

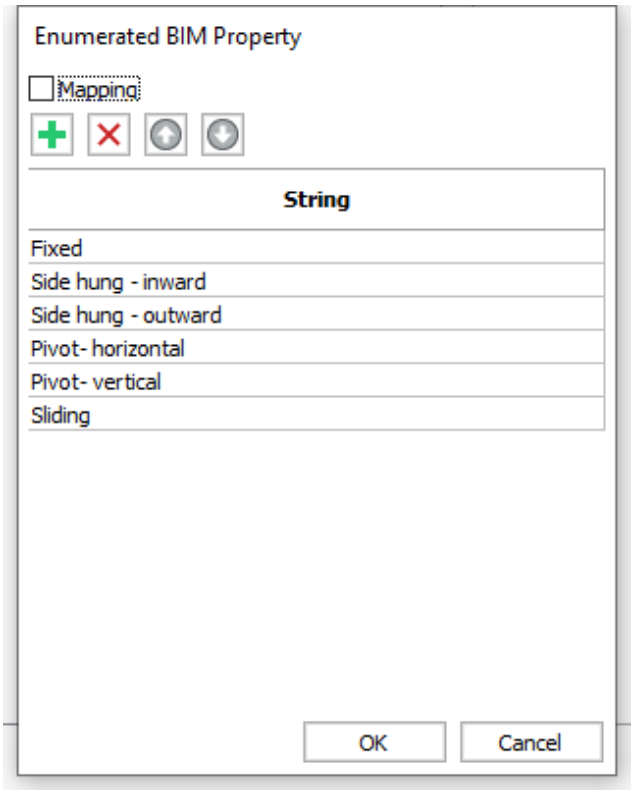


LOD settings control whether ply solids are shown or not. If ply solids are not visible and render composition materials is on, only major faces of the multi ply solids will get render materials.

### 20.1.3.4 Custom BIM Properties

The Per Instance option of a custom property set allows you to enable/disable user-defined properties, on a per-entity basis.

The Enum option allows you to define a fixed number of possible values for custom BIM properties.



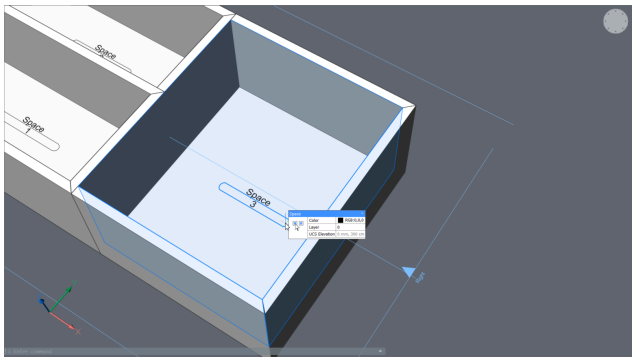

### 20.1.3.5 Spatial Elements

**BimRoom**

The BIMROOM command has been replaced by the BIMSPACE command. You can still open BIMROOMs created in V20, but it is not possible to create new BIMROOMs or update the geometry of existing BIMROOMs in BricsCAD BIM V21. Use the Space option of the BIMCLASSIFY command to convert a BIM room to a BIM space. Room properties and interior elevations, if any, are transferred to the corresponding Space.

**BimSpace**

You can create BIM Spaces entities from enclosed boundaries. When the wall elements enclose a space in your model, you can use the BIMSPACE tool to define a BIM Space. The space footprint preview now displays when executing the BIMSPACE command. These spaces are represented as regular solids so they can be easily manipulated. The Edit option of the BIMSPACE command allows you to add or remove the bounding elements of a space.




**BimUpdateSpace**

Recalculates a BIM space which is out-of-date, for example after adding or removing bounding entities. In the Properties panel, the Update Method property is added with the options Automatic and Manual.

The new Representation property allows toggling between Solid or Footprint representation of a space.

The Space tag has been added. The Space tag is a block that contains a hatch and attributes. The default attributes of the Space tag include a space name and a space number. You can change the values of each attribute using the Properties panel.

The new Status property indicates whether a space is up-to-date or out-of-date.

| BIM | |
|---|---|
| Type | Space |
| Spatial Element Type | Space |
| Representation | Footprint |
| Update method | Manual |
| Status | Up to date |
| Name | Office |
| Description | Standard office |
| Building | Building |
| Story | Floor 0 |
| Composition | |
| GUID | 1zcNYneH56hRdMV23U7Iw$ |
| Number | 2 |
| Interior Elevations | Off |

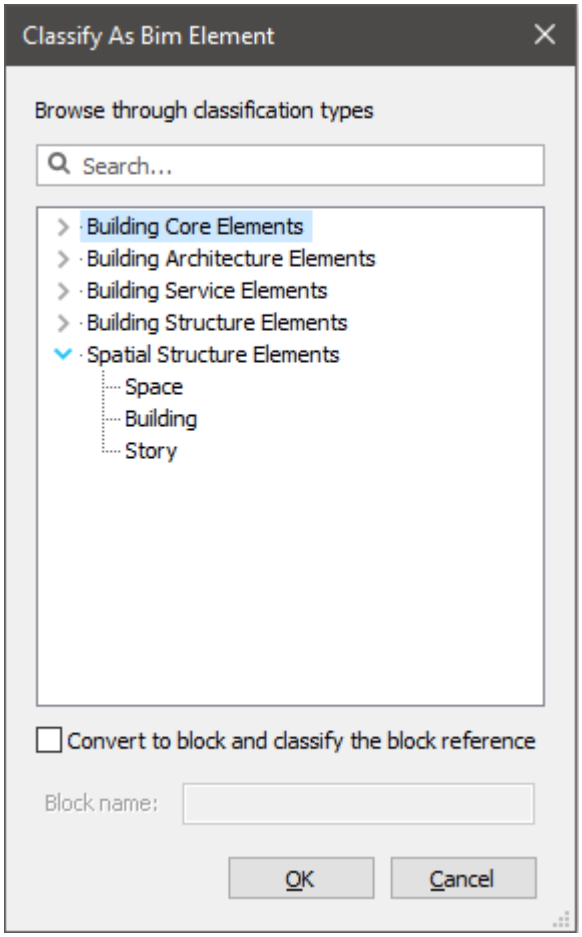
**Space icons**

BIM Space tool icons for Edit Space, Update Space and Define Interior Elevations are available in the Quad, the Ribbon and the BIM Spaces toolbar.



**Spatial Structure Elements**

When using the BIMCLASSIFY command, a new category of classification has been added to the dialog box. The name of the new category is Spatial Structure Elements. Under this category you will find Space, Building and Story.

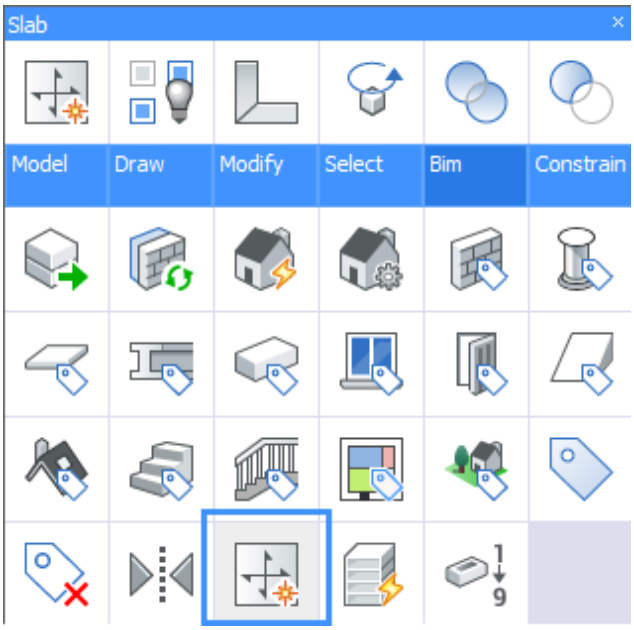



**DefaultSpaceHeight**

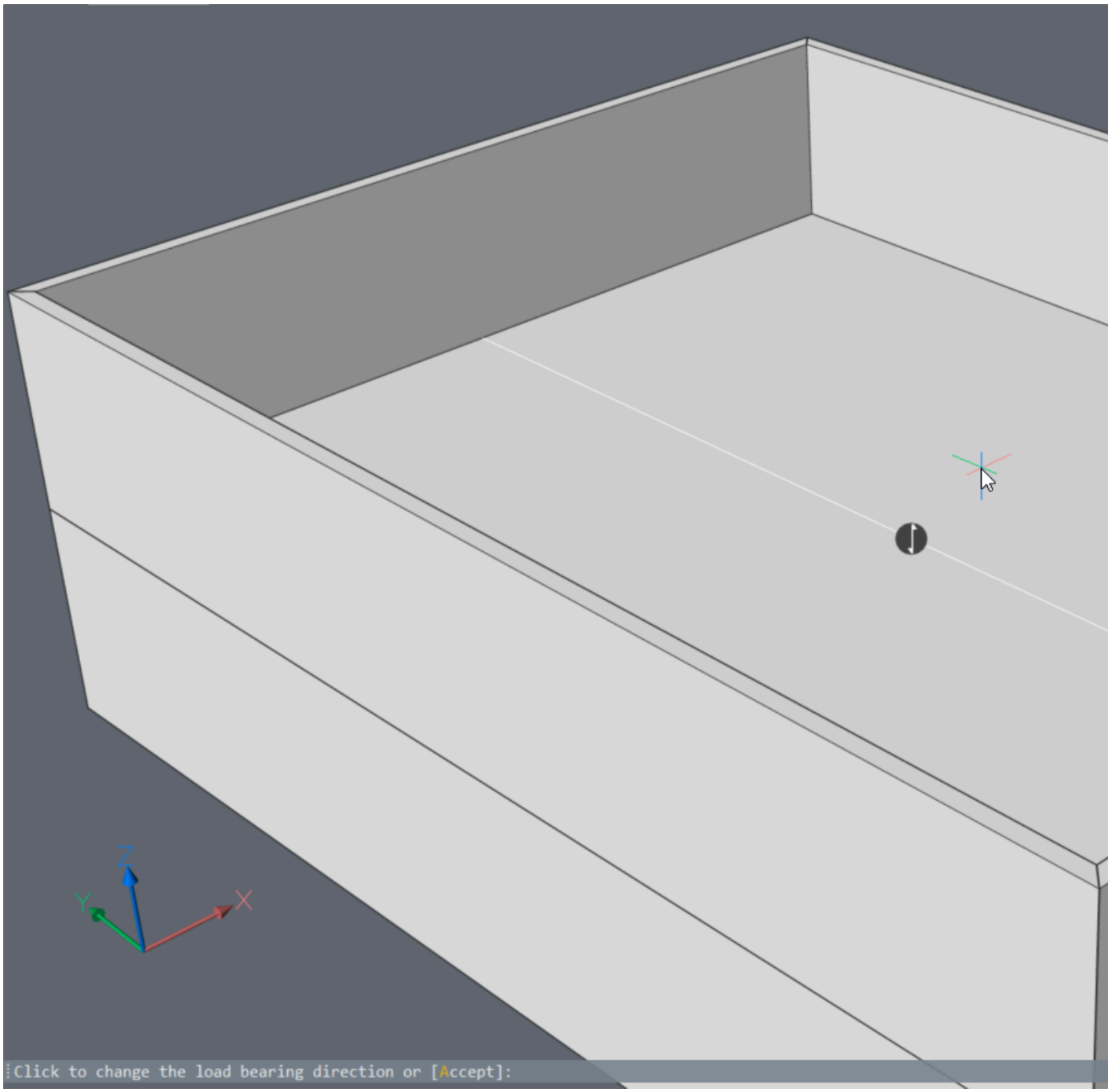
The ROOMHEIGHT system variable is replaced by the DefaultSpaceHeight user preference, the setting applies to Spaces.

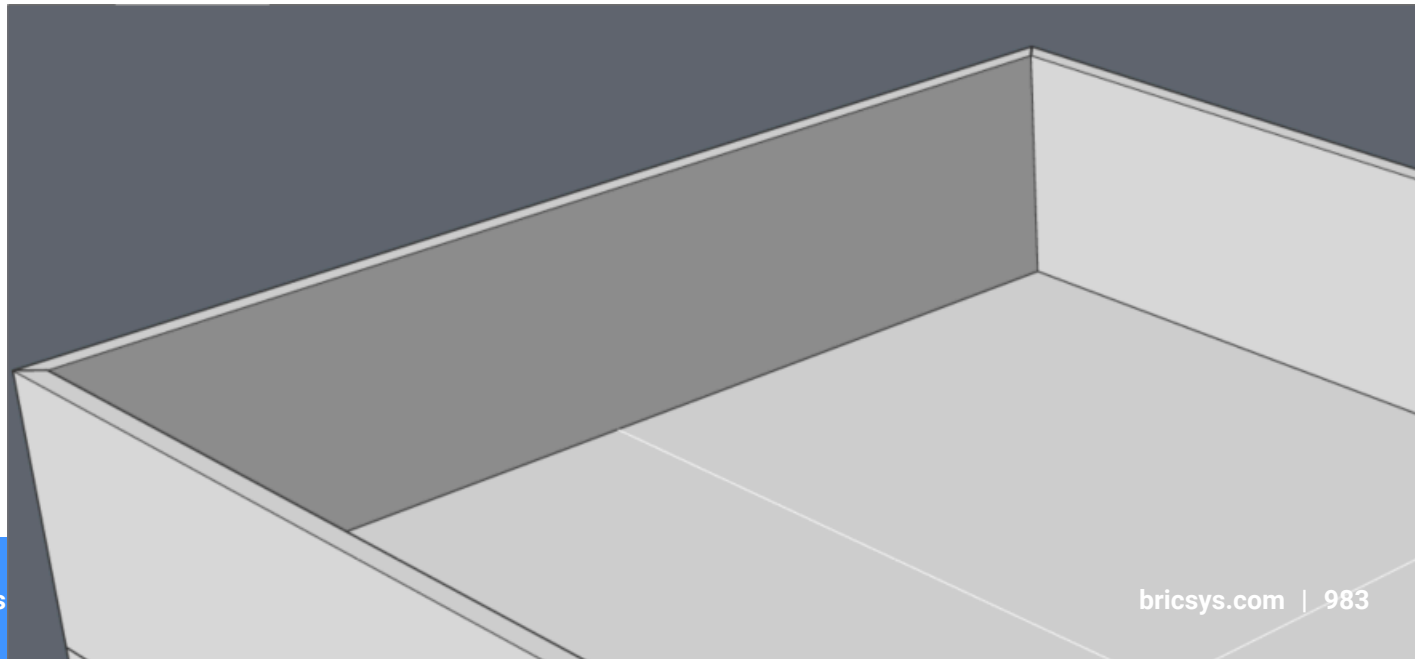
### 20.1.3.6 BIMSETLOADBEARINGDIRECTION

With the new tool button that launches the BIMSETLOADBEARINGDIRECTION command in V21, it is easy to set your loadbearing direction of your slab through the Quad.




As a result, you will get a black dot on your slab. Once you click on it, you can change the load-bearing direction of the slab in one of the two main directions of the slab or in both.


Z
Y
X
Click to change the load bearing direction or [Accept]:



### 20.1.3.7 Python scripts in BricsCAD BIM

***Прим.:*** This is a V21 Beta feature.

The new BIMPYTHON command enables you to query and manage data from a model within BricsCAD BIM, with a .py python script.

The python scripts can be as simple as obtaining quantities and associated properties of objects and BIM entities, or as elaborate as implementing a series of calculations based on the model parameters available.

Executing BIMPYTHON opens a Windows dialog box where you can navigate and run your python script in .py format.

The new BIMACTIVATEPYTHON command allows you to activate and deactivate the virtual python environment in BricsCAD BIM, by toggling the values 0 and 1 for **OFF** and **ON** respectively. You need to ensure it is turned **ON** before using BIMPYTHON.

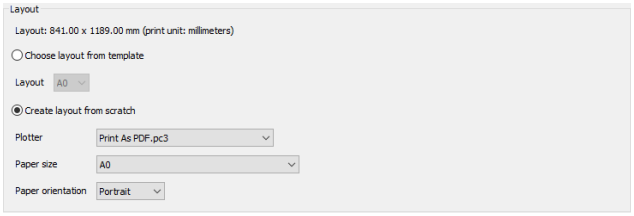
The Python Programming Language, together with its standard libraries, is embedded in BricsCAD BIM so there is no need for you to install them separately, unless you have custom packages and libraries which you wish to use in your scripts.

### 20.1.3.8 Project Browser

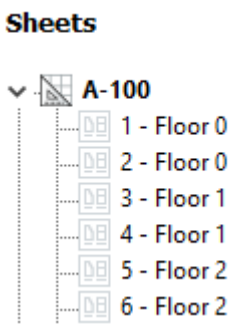
BIM project data are now stored in an external project **.bsyslib**, instead of the previously used **.projectroot**. When opening a pre-V21 BIM project, you can convert the **.projectroot** to a V21 project **.bsyslib**. To simplify the opening of a BIM project at a new location, paths to **.bsyslib** files are now stored with a relative path.

**Sheets**

It is now possible to create a layout from scratch in the Sheetset setup dialog box.



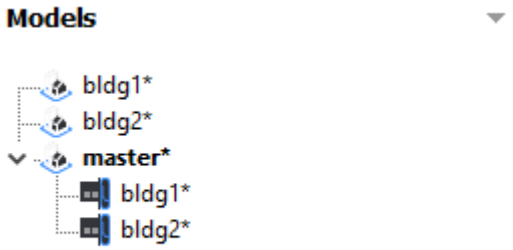

The Project Browser displays the Sheet Number (if it exists). The Sheet Number is assigned by the **Create Sheets** procedure. Sheet View Numbers are assigned starting from 1 for all section results created by the BIMSECTIONUPDATE command.




**Models**

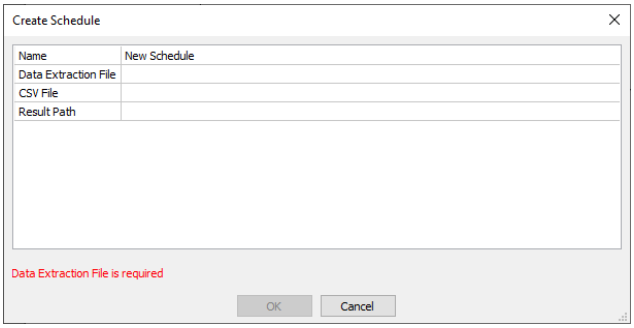
Master drawings and external references are listed in a tree-structured manner in the Models tab of the Project Browser:

A balloon is available to fix a project libraries' connection to a drawing sheet.

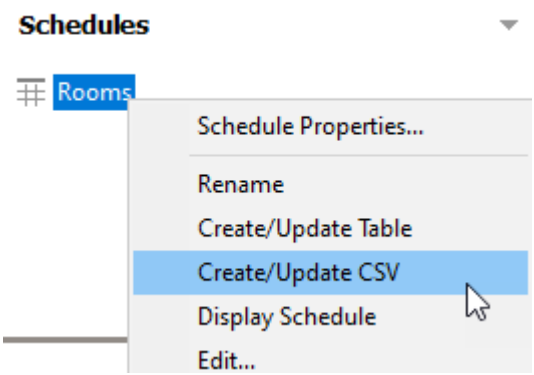



**Schedules**

The **Add Schedule** wizard, available in a context menu, allows you to quickly add a BIM schedule without the creation of a new .dxd file. The use of relative paths is now supported for Schedules.




The **Create/Update Table** and **Create/Update CSV** options are available in a context menu for creating or updating a schedule or a CSV output file.




### 20.1.4 Project collaboration

#### 20.1.4.1 IFC IMPORT/EXPORT

A mechanism has been implemented to map IFC/BIM entities to BIM/IFC entities, to control which entities are imported/exported. BricsCAD BIM also added IFC Export of Annotation entities.

You can do this mapping by using mapping files. The IfcImportMappingMath and IfcExportMappingPath user preferences set the paths to the necessary mapping files. These files are simple text files containing some rules.

This mapping can be used to export BIM entities as IFC entities of another type, or to skip certain types altogether. An example of such an export file can be:

- BimDbColumn IfcMember
- BimDbBeam IfcMember
- BimDbFurnishingElement skip

In this example, columns and beams in the BricsCAD BIM model will be exported as Members in the IFC file, and furnishing elements will be skipped in the export process.

A similar process can be used when importing IFC files. An example of such an import file can be:

- IfcWall BimDbSlab
- IfcSlab BimDbWall

* skip

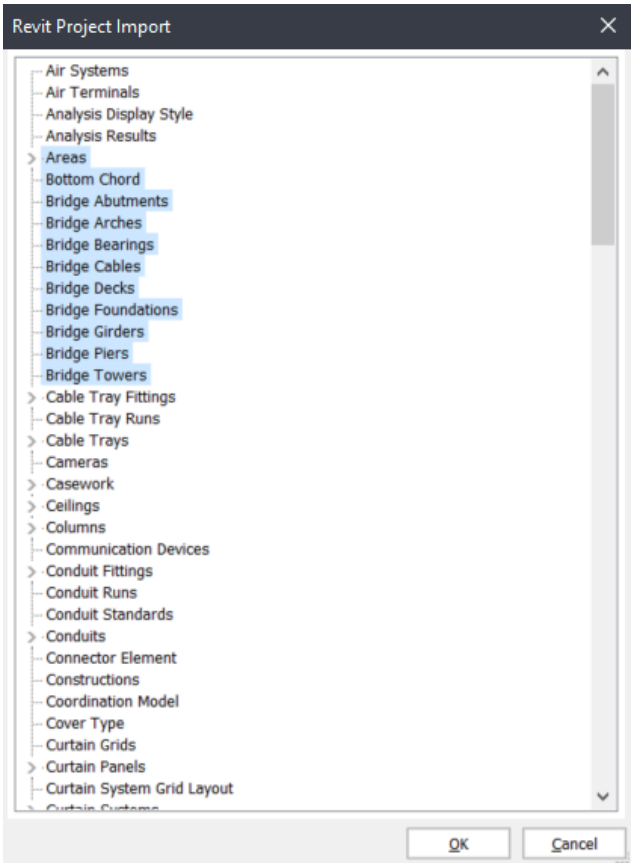
In this case, walls will be imported as slabs and slabs will be imported as walls. No other entities will be imported, as the asterisk means that only listed entity types will be taken into account, all unlisted entity types will be skipped.

The IfcCreateUniqueGuid user preference has been added to enable/disable the creation of unique GUIDs for nested elements inside classified blocks and Xrefs.

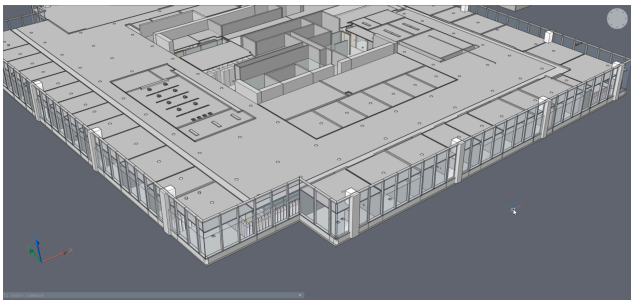
The flow connection point is exported to IFC as IfcBuildingElementProxy. Only 2D geometry or a block reference with 2D geometry can be classified as a flow connection point.

### 20.1.4.2 Importing RVT files

When importing Revit Objects using IMPORT, you can choose for Import all the solids or only import as an Overlay. You can filter the import based on Revit Views, Categories, Levels or WorkSets.




This is an example for the option Categories, where you can map specific categories for your RVT import. Colors and transparencies are now imported when importing a RVT file.



## 20.1.5 Design documentation

### 20.1.5.1 Sections

**Activate SectionView**

This new command allows you to look at the model from a section plane's perspective, without generating the section. This view allows you to manipulate the 3D model from a 2D point of view. To exit the active view, just orbit around the model.

**Open BimSections**

When switching to the model from a sheet view using the BIMSECTIONOPEN command, the model is rotated and zoomed in to the same zoom level as the sheet view.

Available properties and property values display in the filter parameters form.

The BIMSECTIONOPEN command now allows you to transition between the sheet viewport and drawing model seamlessly at the same zoom level, without the need to re-orientate yourself and the model to where you have been working earlier on the sheet.

When you switch between a sheet viewport to the drawing model using the updated BIMSECTIONOPEN command, the drawing model displays and orientates itself to the same zoom level as in the sheet viewport, regardless of the scale of the Standard Scale or Annotation Scale of the drawing or viewport.

**Update BimSections**

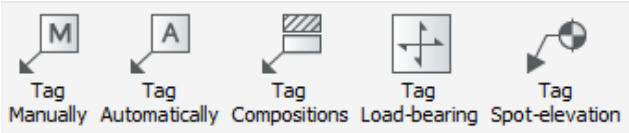
The associative data for 3D solids, needed for section generation and tagging of the section results by the BIMTAG command, are generated automatically in BricsCAD BIM V21 as you model, unless the new GENERATEASSOCATTRS (Generate Associative Attributes) system variable is set to Off.

**Interior Elevations**

Since Rooms in V20 and older are now deprecated and replaced by Spaces, the Interior option of the BIMSECTION command now supports Spaces.

### 20.1.5.2 Tags

The BIM Tags tool now includes options of tagging load-bearing directions and spot elevations on viewports on Paper Space. Both tag types work on plan and section drawings. Spot elevation tags also work on elevation drawings.




Automatically generated tags can have an offset, which is defined through the Offset attribute in TagTypeToStyle.xml found in the **SUPPORTFOLDER** > **Bim** > **Sections** folder. If the offset is not specified, the center of the tag block will be used.

Similarly, automatic and manually placed tags can be rotated if the autoRotation attribute is set to **true** in TagTypeToStyle.xml.

**Tag Load-bearing**

Load-bearing tags is only applicable if the load-bearing direction of the slab has already been set. Refer to the BIMSETLOADBEARINGDIRECTION command for more information.

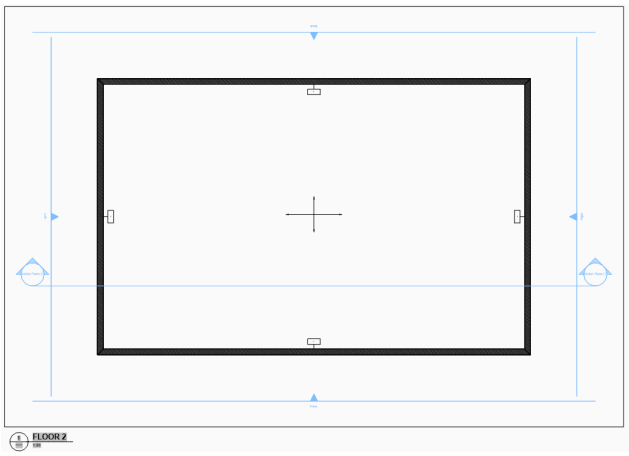
**Tag Spot-elevation**

The spot elevation level tag displays the actual elevation of a selected point of a BIM object in a section view. The height is measured relative to level 0.

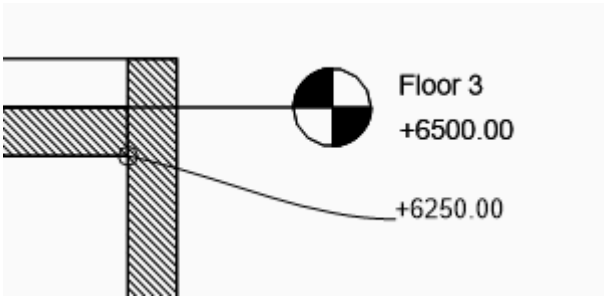
**Tags in general**

On Paper Space, click **Tag Load-bearing** from the ribbon and pick point on a sectioned BIM element as prompted. You will be prompted to pick a point on your desired sectioned BIM element or to change the current mleader style settings, like how you set BIMTAG in V20.

For example, you can have a two-way load-bearing span graphic tag on structural drawing plans.



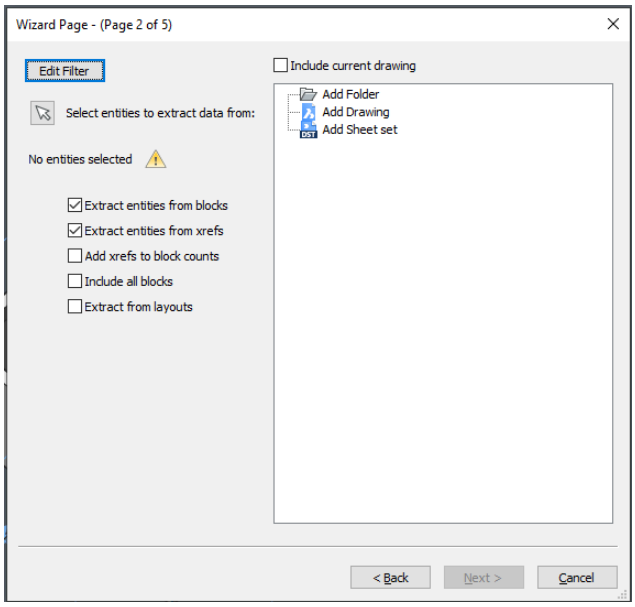

Or you can have spot elevations displaying elevation levels relative to the WCS.



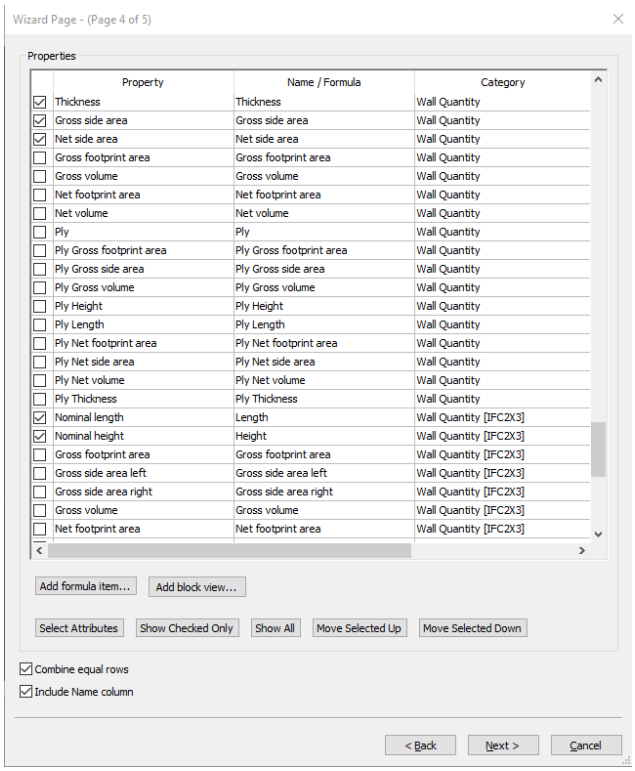

Like all other BIM tags, they are customizable multileader blocks. Therefore, for example, Spot Elevation tags can be further adjusted to suit your graphic and information preferences.

### 20.1.5.3 Data Extraction

The DATAEXTRACTION command now allows you to include all blocks and also extract from layouts in the new filter rule settings. You can now extract composition and physical material data.



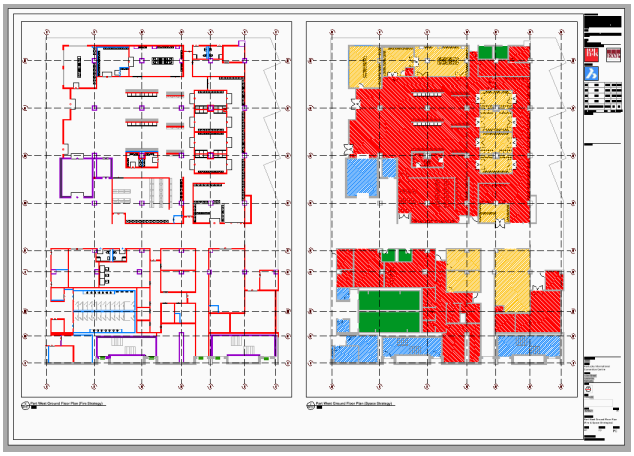

You can now extract and schedule quantity data based on IFC2x3 Element Types.




### 20.1.5.4 Drawing Customizations

Drawing Customization is a new tool that is available and only accessible from the panel. All the controls of Drawing Customizations are represented as graphic icons within the User Interface panel, such as the **+** symbol in a blue circle to create a new template. You can duplicate, rename or delete your customization templates.

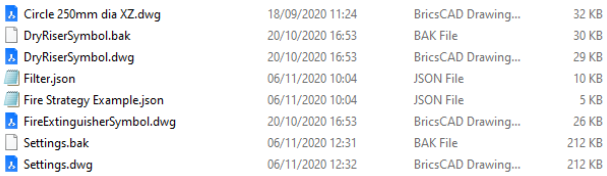
It allows you to create view templates to control and customize the visibility settings of your drawings on layouts. The Drawing Customizations tool in BricsCAD is operated by 3 principles – Entity Customizations, Styles and Filter Rules.



Drawing Customization templates are stored within the Customization folder located in the SUPPORTFOLDER, under **Bim folder** > **Sections subfolder**.

Each customization template is saved as an individual folder which contains in its basic form Settings.dwg, a Filter.json and a New customization.json files. The folder will also contain any external symbol sources which you have created new using the Create New buttons.

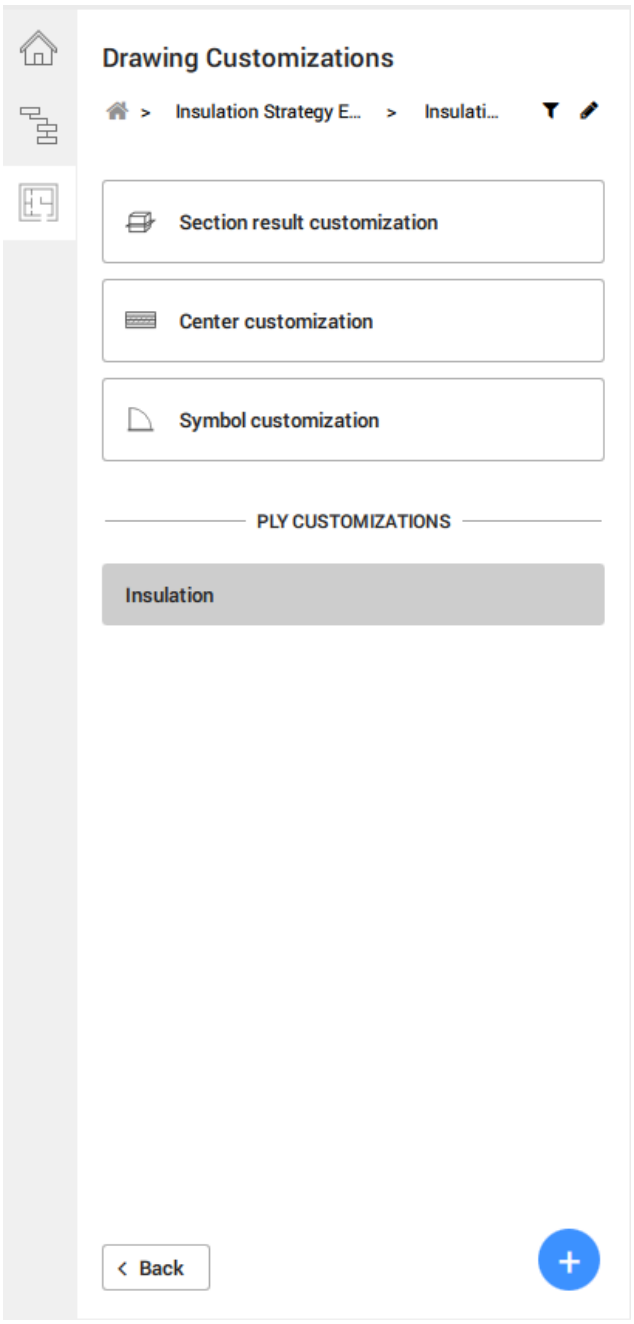
Settings.dwg contains the values created in the Style tab in the form of layers and if you have created any custom properties that you wish to use as a Filter parameter, you have to ensure they are also reflected in the Settings.dwg.




**Entity Customizations**

Entity Customizations refer to the visibility settings of how you would like to have your elements in your model drawing to look like.

One level deeper within one of those entity customization tabs brings you to its settings tab, where you can choose how the element should look like as a section result, or how a center line type representation should overlap, or how an external symbol representation should replace the original asset. You can further modify and customize a specific ply layer of your desired entity composition, but only if appropriate.




The section result customization displays a series of controls with drop-down values from the Styles tab for more specific control of the section result graphics. You can override the entities' hatches via the Appearance Override with preset or custom Physical Materials available in your project.

For the Symbol Customization settings, you can find an input dialog to specify your desired external symbol drawing in .dwg format. There are additional buttons to allow you to edit the selected symbol drawing or to create a new .dwg symbol.

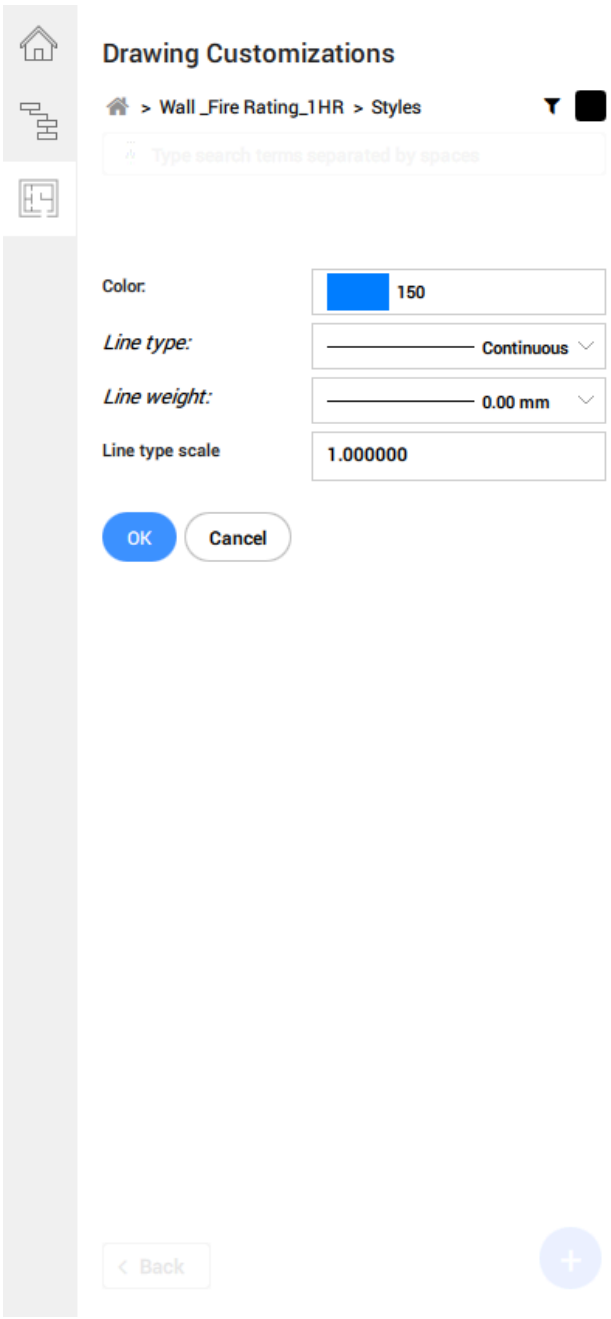
You can further apply your saved Styles to specific layers within your external symbol source, thereby allowing you to retain a general symbol drawing across several customization templates with different output results.

**Styles**

This is where your desired style options like Lineweight, line colors, etc, are stored and will appear as drop-down options in the Entity Customizations and Center Customizations settings as shown in the previous section above.

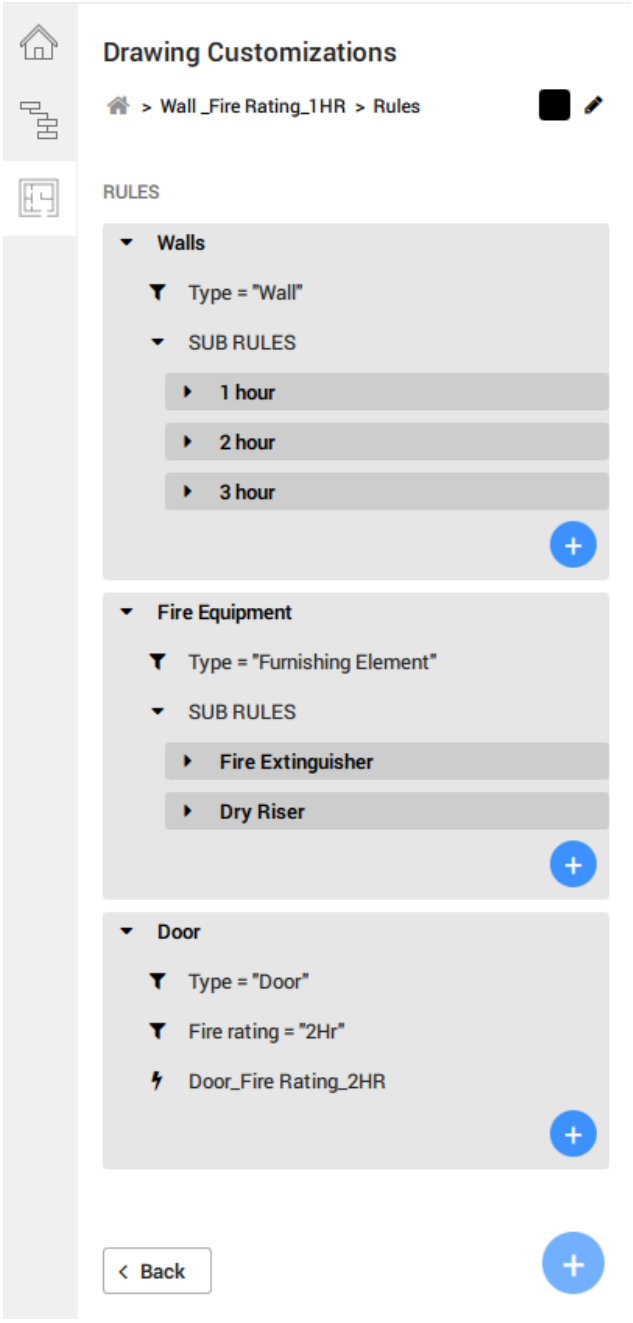
The operation of the Styles tabs is like how Entity Customization tabs can be controlled.

To edit any of the styles available, click the relevant tab to enter the styles settings, where you can specify your desired values for the style's color, linetype, lineweight and linetype scale.

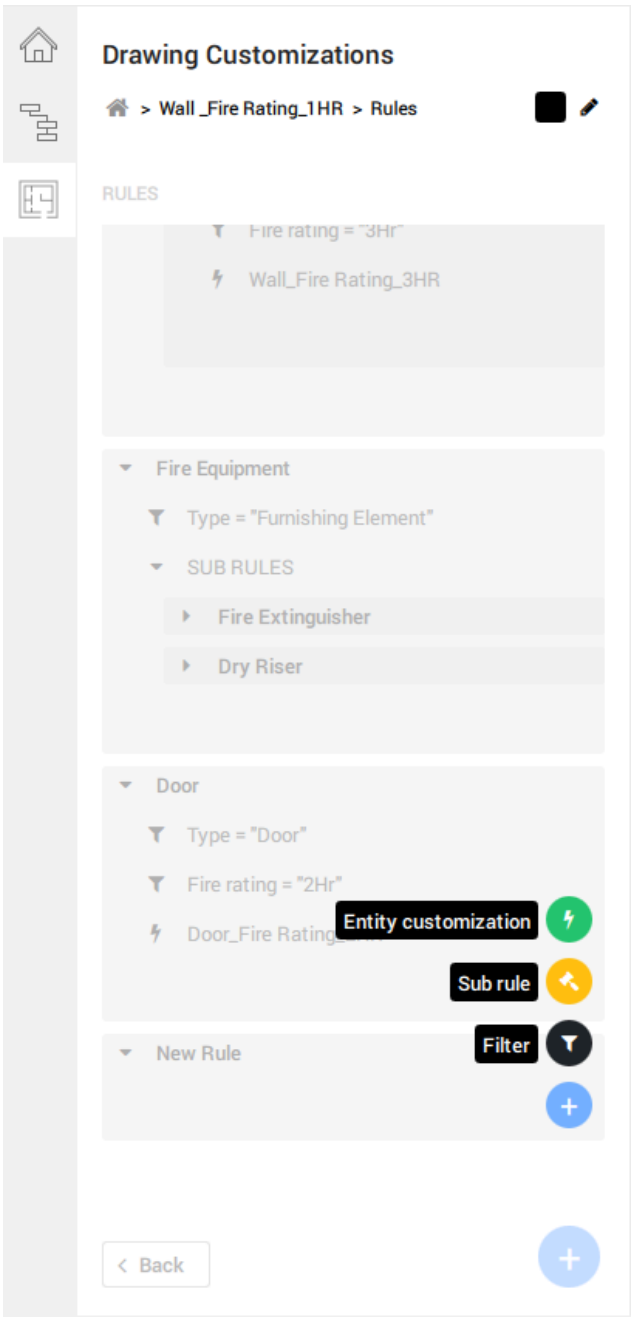



**Filter Rules**

To be able to apply the right customization styles to specific entities, you have to set the appropriate filter rules to target those entities correctly in the Filter Rules tab.




Filter rules can be further classed into sub-rules, which falls as a sub-set of the parent rule, ie. Fire Equipment as a Filter Rule, whereas Fire Extinguisher is classed as a sub-rule. The blue **+** symbol also appears respectively and accordingly at each rule level, where at the end of each rule, regardless of it being a subset or otherwise, has to end with an Entity Customization.




Adding and editing a filter parameter will lead you to a prompt, where the options of filter property is determined by the BIMPROPERTIES imported in the namespace settings.

To end and legitimize a filter rule, the entity customization value must be added and filled in. The options available in the drop-down selection reflects the existing customizations tab available in the Entity Customization page.

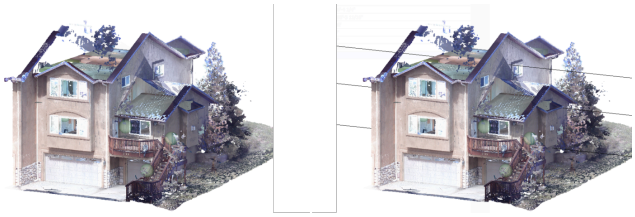
### 20.1.6 Point clouds

Point cloud functionality is even more powerful with a BIM license.

### 20.1.6.1 Floor Detection

Sectionplanes work on point clouds as well, they can be used to show parts of point clouds. The difference between point cloud crops and sectionplanes is that point cloud crop only clips the point cloud while section planes will clip all geometry in your drawing.

The new POINTCLOUDDETECTFLOORS command generates volume sections for each floor found in a point cloud representing a building. The detection is based on regions of points with similar Z-coordinates. The generated volume sections can help in navigating point clouds of buildings.

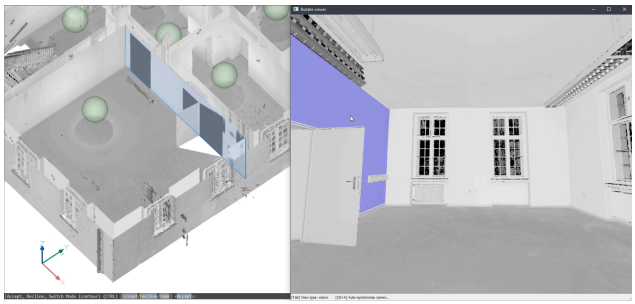


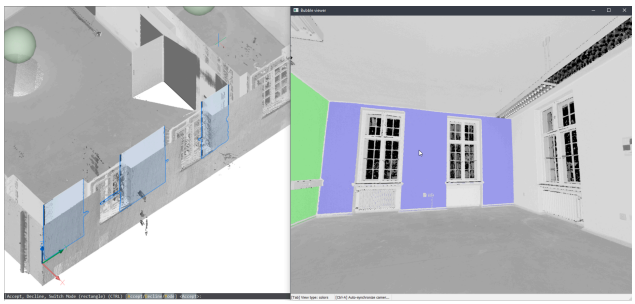
### 20.1.6.2 Planar Fit

The new POINTCLOUDFITPLANAR command enables you to create 3d geometry based on the point cloud. It will create a planar surface or solid after a selection of one point in a point cloud. The points that seem to be in a plane are never exactly in one plane, therefore a threshold value is set as a property of the point cloud entity. This also works in bubble view.

**In bubble view**

If the bubble viewer is open before launching the command, BricsCAD expects you to select in the bubble viewer. The cursor will give you a preview of the direction of the plane. When you click you get a preview in both bubble view and model view. You can toggle between 2 shape representations using the CTRL key.

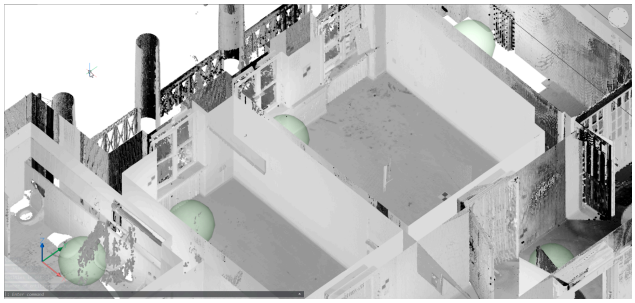


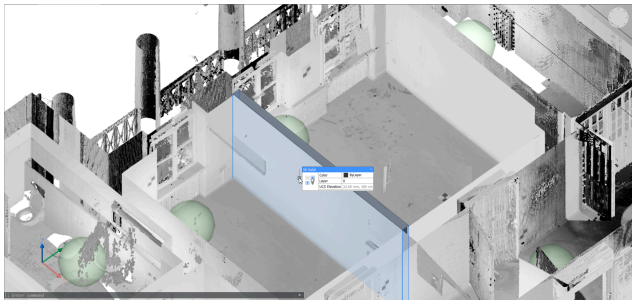


**In model space**

You can also use this command in the model space when the bubble viewer is not open. BricsCAD will ask you to select a point of the point cloud in model space. Depending on the size of the cropped point cloud, it takes more time but it has 2 advantages by searching multiple scan positions:

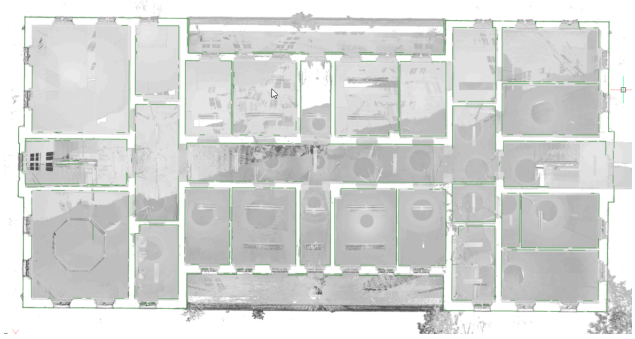
- It can create larger surfaces where only parts are visible in each scan position
- It can detect wall and slab thickness since it can take the opposite surface into account.





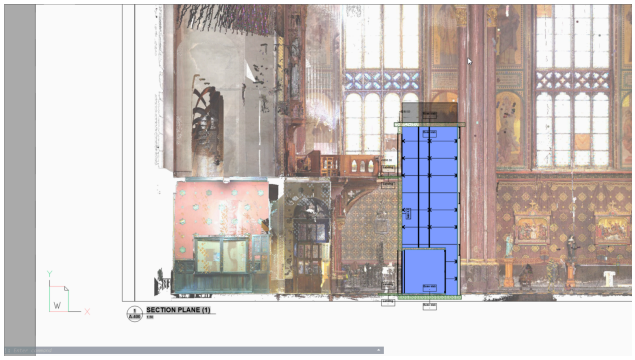
### 20.1.6.3 Point cloud projection

The new POINTCLOUDPROJECTSECTION command enables you to detect walls from the volume section of a point cloud based on a variety of wall detection options. You can create volume sections automatically for each floor in a building. You can use these sections to generate 2d lines to create a 2d floorplan or a vertical section. This is a background process and multiple sections can be processed in a queue. This way it is possible to run this command in full resolution on all sections.



At the same time, a raster image will be generated. In some cases it is not necessary to recreate the existing building. Background images can give so much more context to the design documents. These can be used to verify the created 2d geometry but in high quality scans these images can also be used

as graphical material. For example, as a background image for a BIM model in renovation projects where modern intervention are made in historical buildings.




## 20.2 Начало работы

### 20.2.1 Creating a BIM model

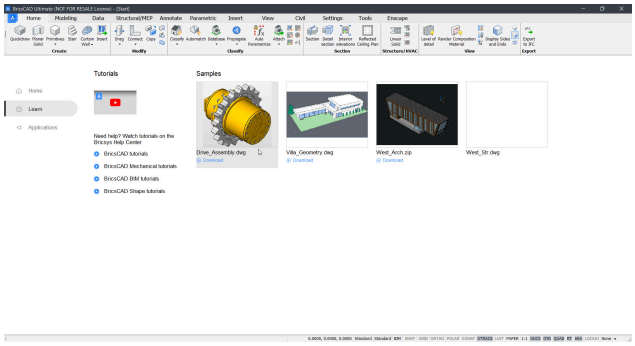
**Do the following:**

1 Start BricsCAD and choose the BIM workspace on the **BricsCAD Launcher**. In this workspace, all BIM tools and resources are available in the Ribbon, the Quad and various dockable panels.

2 On the Start tab, select a **BIM** template.

3 Click the **New Drawing** button to create the model.

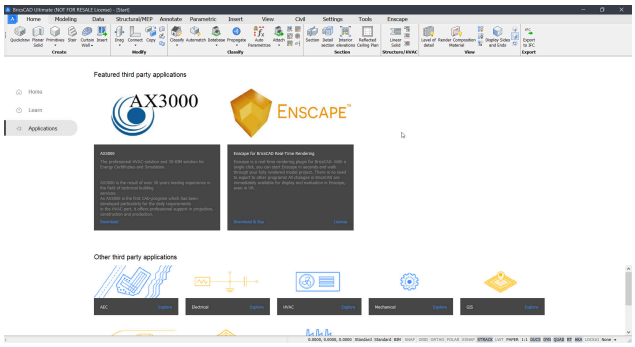
#### 20.2.1.1 Learn




Here you will find links to tutorials for Shape, Core, BIM and Mechanical. You can also find some sample files that you can download and try out yourself.

#### 20.2.1.2 Applications

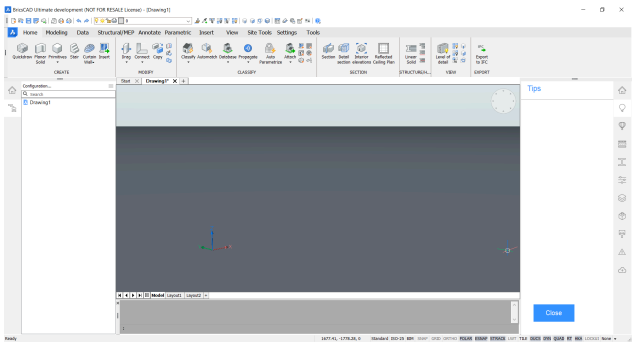
Here you can find some of the third party applications that are available for BricsCAD. You can find more applications on our Applications Store website.



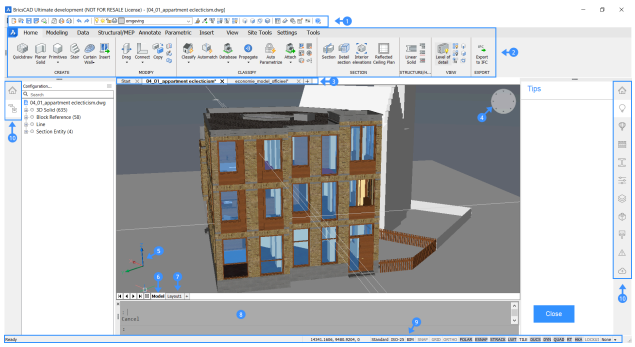

## 20.2.2 Exploring the interface

### 20.2.2.1 User interface

The BricsCAD BIM interface includes all tools, commands, and settings required to create and edit building elements. Start by creating building elements, opening an existing drawing or importing a building model.



### 20.2.2.2 Interface elements

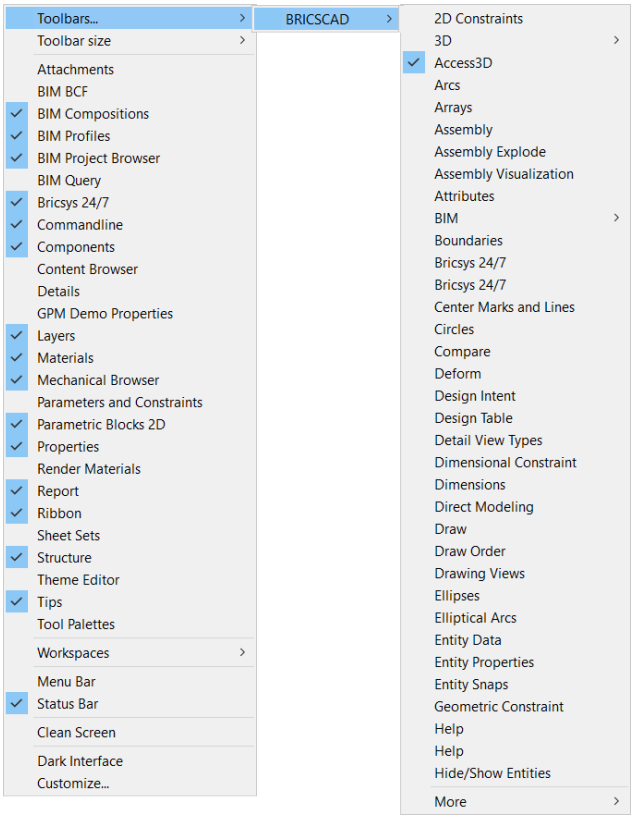


**1. Toolbar**

In BricsCAD, toolbars are arranged in collections of similar functions within a bar of tools. Each function is displayed with an icon and is categorized by topic. Toolbars are available for all command categories. For example, a BIM toolbar is one of the tool categories available in BricsCAD BIM. This toolbar includes the most used BIM tools for modeling. It is possible to customize existing toolbars or create your own custom toolbars.

By default, the Access3D Toolbar is shown on top of the screen.

***Прим.:*** To open more toolbars you can right-click, choose toolbars and check the topics you want.

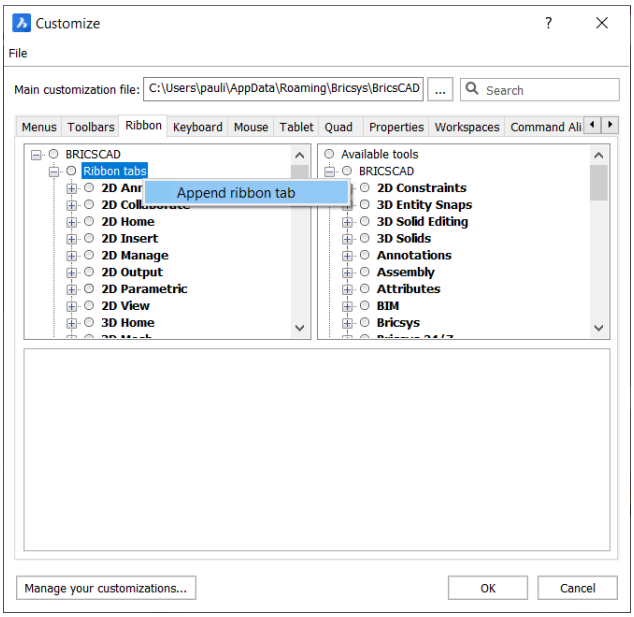
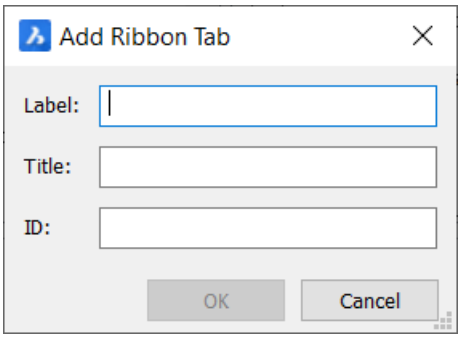


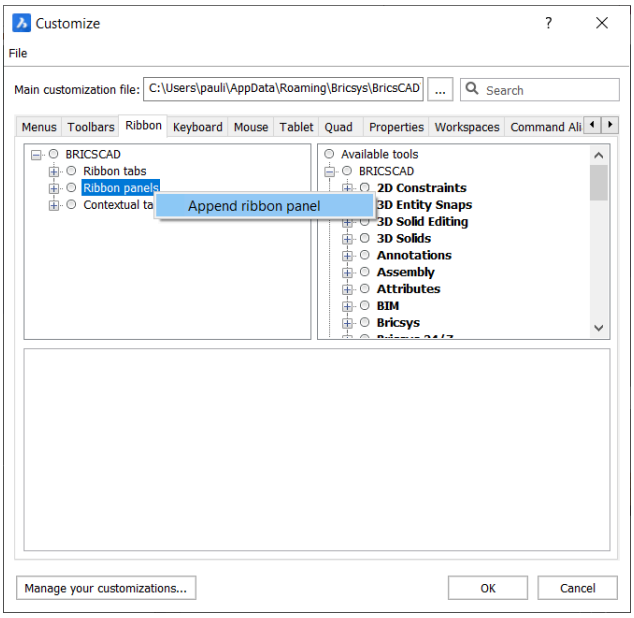
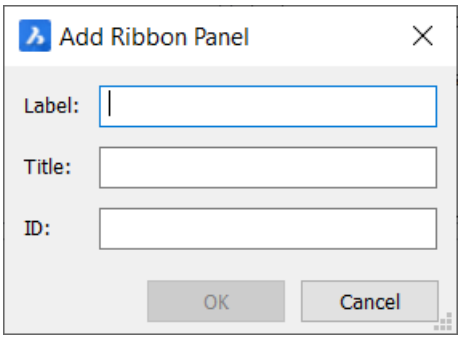

### 2. BIM Ribbon

A ribbon organizes tools in a series of panels (11) which are grouped in different tabs (12). Each tab contains panels and each panel contains a group of buttons and flyouts.

***Прим.:*** The content of the ribbon depends on the current workspace (see Editing In Workspace).

***Прим.:*** Use the RIBBON and RIBBONCLOSE commands to show or close the ribbon.

| | |
|---|---|
| Creating a ribbon tab | Open the **Customize** Dialog box and click on the **Ribbon** tab.<br>Right-click the **Ribbon tabs** group and choose **Append ribbon tab** in the context menu.<br><br>Fill out the fields in the **Add Ribbon Tab** dialog box:<br><br>**Label**: Caption on the ribbon tab.<br>**Caption**: Used in the Select Ribbon Tab dialog when selecting ribbon tabs for a workspace.<br>**ID**: A unique name to identify the ribbon tab. |
| Editing a ribbon tab | Open the **Customize** Dialog box and click on the **Ribbon** tab.<br>Expand the **Ribbon tabs** group and you can edit each tab. |

| | |
|---|---|
| Creating a ribbon panel | Open the **Customize** dialog box and click on the **Ribbon panels**.<br>Right-click the **Ribbon panels** group and choose **Append ribbon panel** in the context menu.<br><br><br>Fill out the fields in the **Add Ribbon Panel** dialog box:<br><br><br>**Label**: Caption of the panel in the ribbon.<br>**Title**: Used in the Select Ribbon Panel dialog when selecting the panels for a ribbon tab.<br>**ID**: A unique name to identify the ribbon panel. |
| Editing a ribbon panel | Open the **Customize** dialog box and click on the **Ribbon** panel.<br>Expand the **Ribbon panels** group and you can edit each panel. |

### 3. Working with Document Tabs

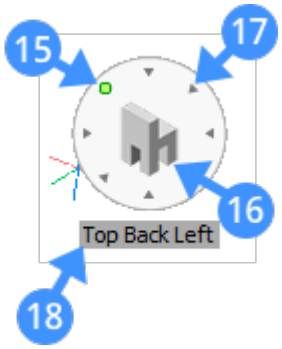
Document tabs allow you to:

- Easily switch between all open drawings.
- Drag and drop the tabs to change their order.
- Click the **X** icon (13) at the right-hand side of the tab to close it.

- Right-click on a document tab to display a context menu.
- Click the **+** icon (14) at the end of the document tab to open a new drawing.

**4. Look from Widget**

The Look From Widget appears in the upper right corner of the drawing area. When you hover the cursor over the widget, a small green rectangle appears (15), as does the preview of a chair (16). Clicking the cursor over one of the triangle shapes (17) shows what the 3D view will look like (18). The default location of the widget is the top right corner of the drawing area.




**5. UCS Icon**

UCS stands for a User-defined Coordinate System. The UCS icon shows the nature of the current coordinate system.




The display of the icon is controlled through the UCSICON system variable and the position of the UCS icon is defined by the UCSICONPOSITION system variable.

**6. Model Tab**

When you start a drawing session, your initial working area is called **Model Space**. **Model Space** is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or User Coordinate System (UCS). You view and work in model space while using the **Model** tab.

**7. Layout Tab**

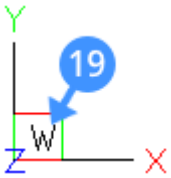
Each drawing has at least one layout and each layout is composed of one or more viewports. Each viewport can show a different part of the drawing at a different scale.

Paper space is a work environment that provides the model space view at a given scale, depending on the size of the paper.

In each layout, you can add the entities needed to complete a printed copy of the drawing; title blocks, legends, frames, etc. These entities are only visible in the layout when you add. They are not visible in the other layouts or in the model space.

**UCS in Layout Mode**

In paper space of layout modes, a W (19) letter appears in the UCS icon, which means the WCS is active.

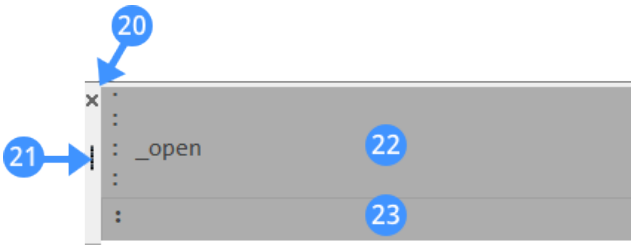



***Прим.:*** WCS stands for World Coordinate System.

**8. Command line**

The Command line consists of two fields.

- In the lower field, you can type the commands and BricsCAD shows prompt, options and other information regarding the execution of commands here. If the Command line is closed, this information shows in the Status Bar.
- The command history displays in the upper field of the Command line.

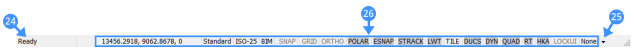



| | |
|---|---|
| Close (20) | Click to close the Command line. |
| Grip (21) | Press and hold the left mouse button to drag the command bar. |
| Command history (22) | Shows the previously typed commands. |
| Command edit (23) | Type the commands and command options here. |

**9. Status Bar**

The Status Bar sits along the bottom edge of the BricsCAD application window. It contains a lot of information about the settings in the current drawing. The status bar consists of 16 fields. All of these fields are optional, except for the Status field (24). If you click the small down arrow button at the right end of the status bar (25) a list of all field displays. Click a field to toggle its display.

Right-click a field to display an options menu for this field.



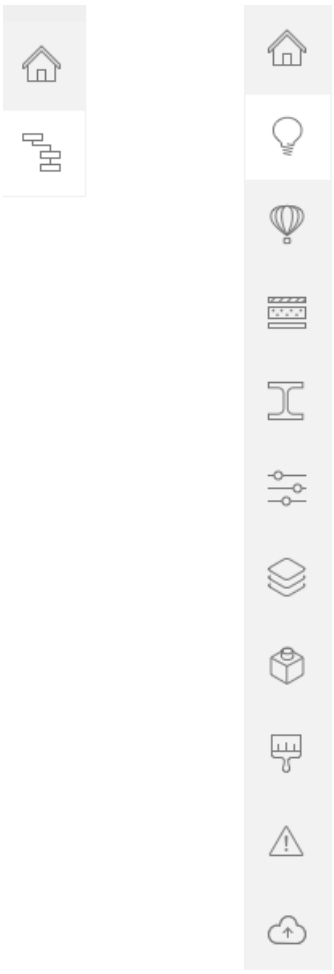

| | |
|---|---|
| Status field (24) | Displays the status of the software. |

| | |
|---|---|
| List button (25) | Click the arrow to display a list of available Status Bar fields. |
| Fields (26) | Click a field to toggle it ON/OFF. Right-click to display a context menu. |

**10. Tool Panel**

In BricsCAD BIM, the Tool Panel is displayed on both sides of the workspace. The Tool panel consists of BIM panels. The BIM panels are dockable and thus they are movable. The panels can be moved to another docked panel using the drag and drop method.

By default the Tool Panel looks like the image below: on the left-hand side it displays the **Structure Browser** and the **Project Browser** and on the right-hand side it displays **Tips**, **Components**, **BIM Compositions**, **BIM Profiles**, **Properties**, **Layers**, **Mechanical Browser**, **Materials**, **Report** and **Bricsys 24/7**.



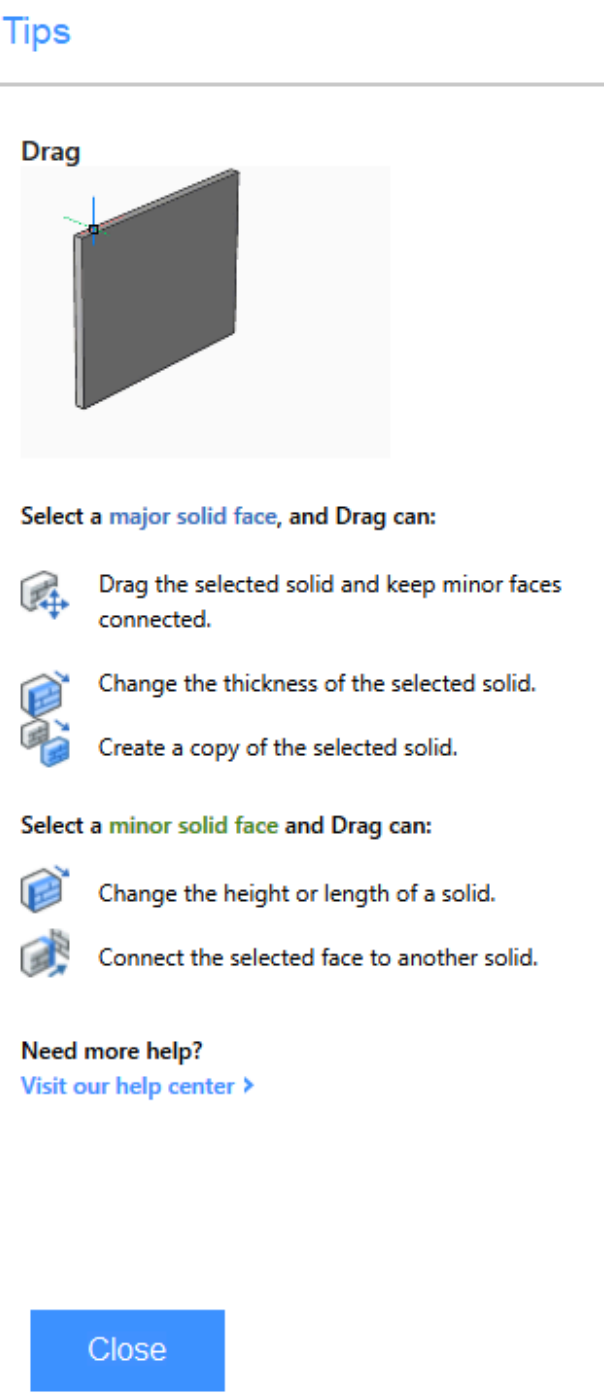
***Прим.:*** You can always add panels in the Tool Panel by right-clicking and checking those you want.

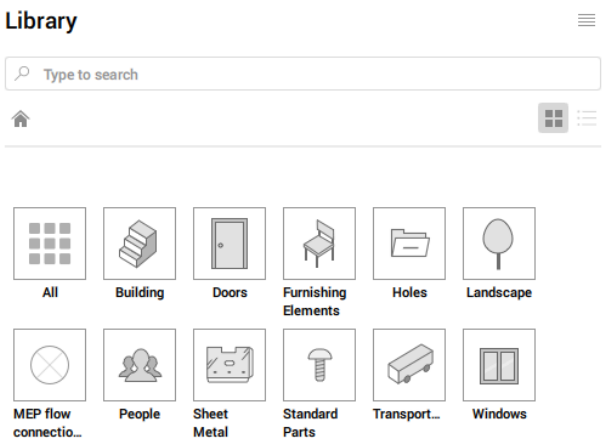
We will now have a look at the functions of the different dockable Panels from the Tool Panel:

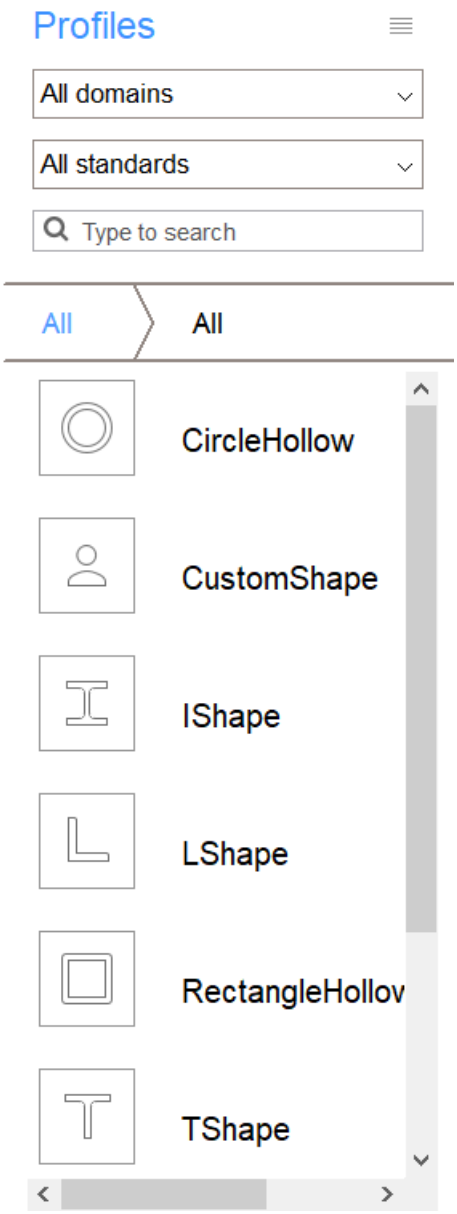
| | |
|---|---|
| **Project Browser** | The BIM Project Browser allows you to easily navigate through the entire content of a project: models, sections, drawings, and schedules.<br>See The **BIM Project Browser** article on how to use this panel. |
| **Structure Browser** | The **Structure Browser** is an interactive tree that displays the entities in the current model. Using the configurable structure tree on the **Structure Browser** the BIM model can be organized in a way that you want to view the elements.<br>Once you configured the tree, you can easily save this structured tree as a .cst file. By default, the .cst files are stored in the **Support Folder**.<br>See The **Structure Browser** article on how to use this panel. |

**Tips**

**Tips** panel gives general information about the tools and commands. The tips can change depending on the active command. The tips can only be seen when the panel is open.
The following illustration shows the **Tips** panel when the DRAG command is active:

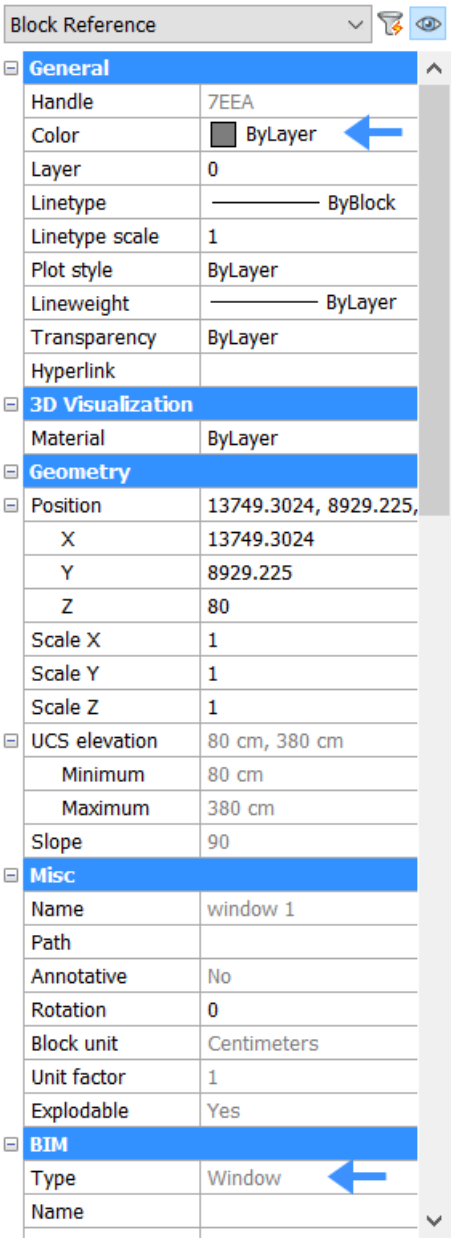


| | |
|---|---|
| **Library** | The **Library** panel gives a list of components that you have by default. You can simply click and drag these components into your drawing.<br><br>At the bottom of the panel, you have the button . When you have made for example a drawing of a window, you can click on this button and add your own drawing to the component library. This way your window can be used as a component in all of your other drawings. |
| **BIM Compositions** | In the **BIM Compositions** panel, you find a library of predefined composition that you can assign to various solids of the BIM model by dragging them on.<br>Assigning compositions to the elements in your model is a core step to increase the BIM model accuracy. Compositions can contain either multiple materials or a single ply.<br>See The **BIM Compositions** article on how to use this panel in more detail |

| | |
|---|---|
| **BIM Profiles** | The **BIM Profiles** panel shows a library of predefined profiles that you can assign to linear solids by dragging them on.<br><br>See The **BIM Profiles** article to see how to work with this panel in more detail. |

**Properties**

The **Properties** panel shows all the properties of a selected element. You always have some general info first, the other different tabs within this panel depend on the type of element that is selected.

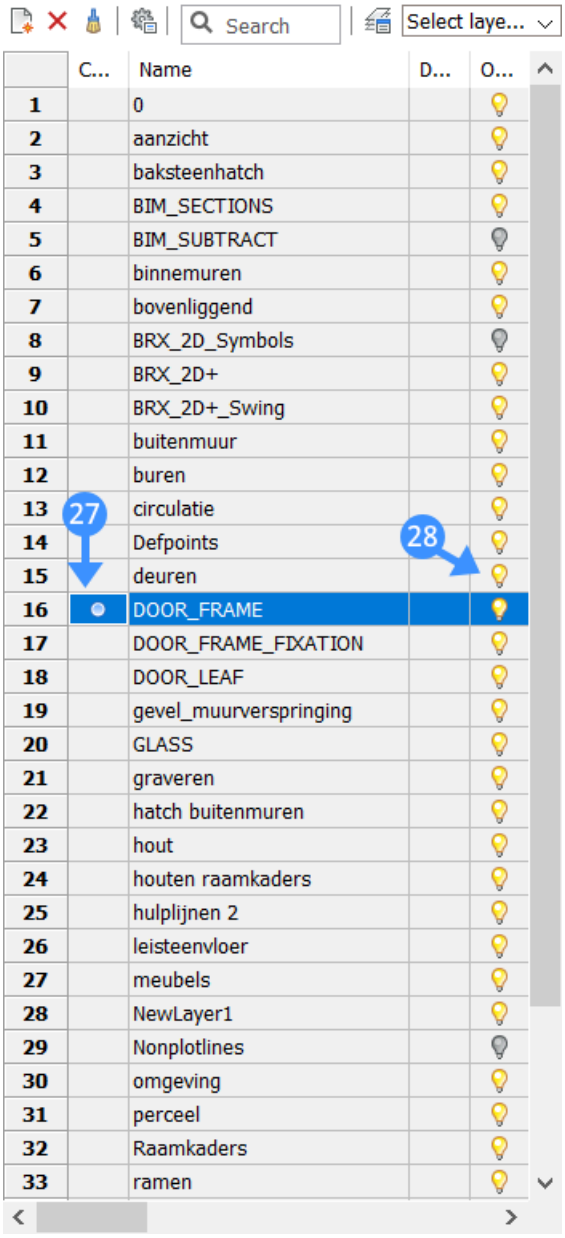



For example, here we can see that the selected item is a window and is colored by layer. You can adjust the properties as you like.

See the **Properties** article to see how to work with this panel in more detail.

**Layers**

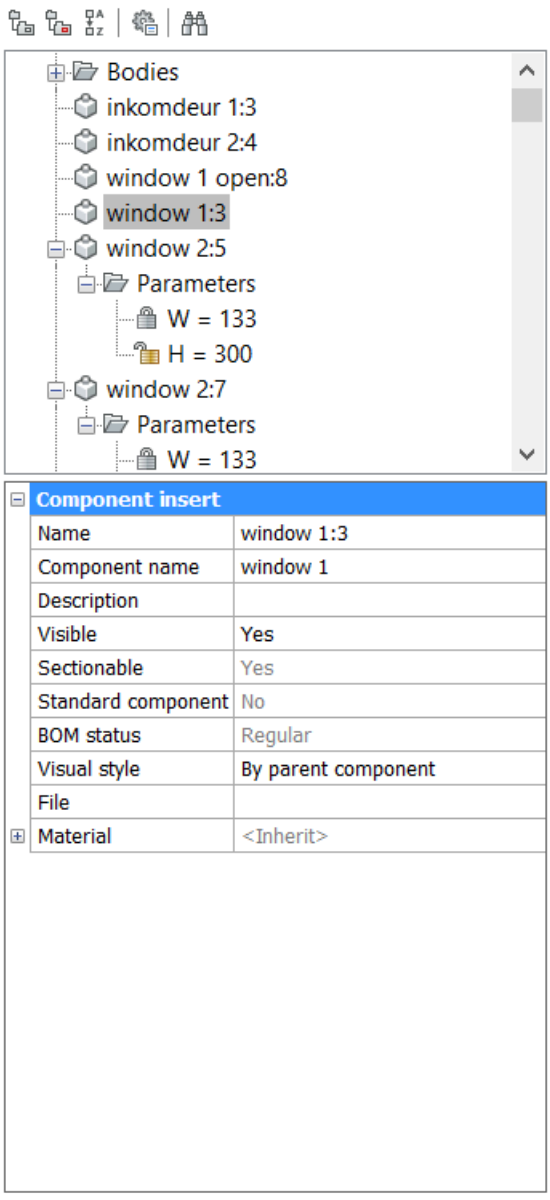
The **Layers** panel shows all the layers of your drawing.
It shows you the current layer (27), you can change it by clicking left of another layer. It also shows if a layer is ON or OFF by the state its lightbulb icon (28). Other layer properties such as color, line type,... can be found and adjusted by scrolling to the right.




In this example the current layer is **DOOR_FRAME** and it is turned ON.

**Mechanical Browser**

The **Mechanical Browser** panel displays the hierarchy of components, features, arrays, parameters, and constraints for the current drawing.
They are displayed in a tree-like form: you can expand and collapse every node to see its child nodes (for example, nested components).

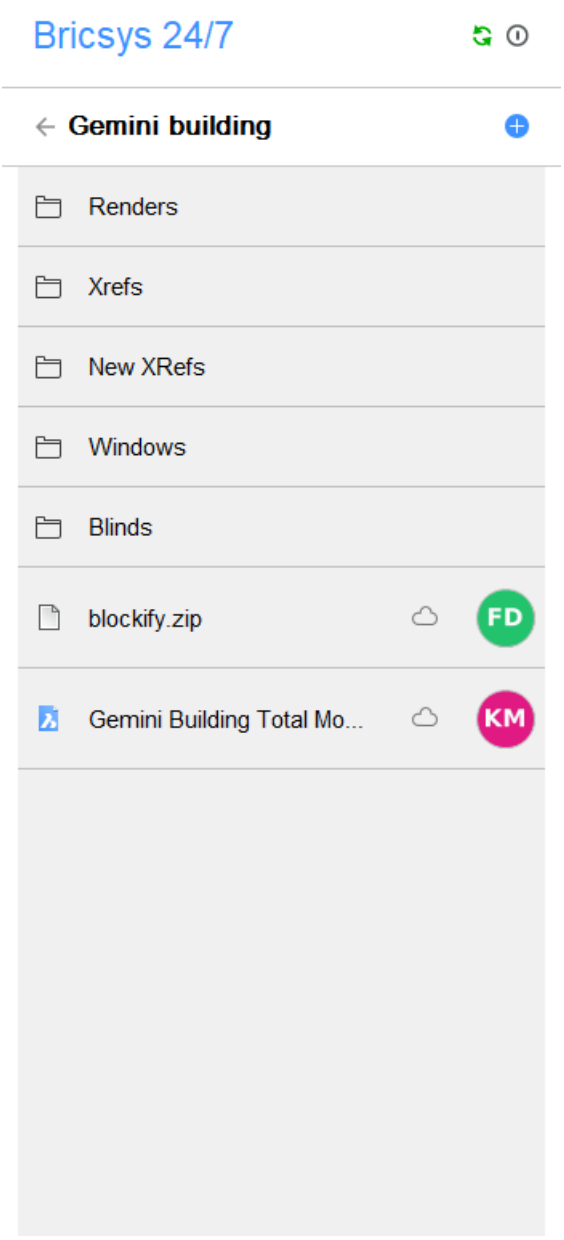



The **Mechanical Browser** allows you to edit properties of a particular node, displayed at the bottom of the panel and calls different tools depending on what type is available in the context menu by right click on a particular node. For example, in the image above you could add a description for the selected window.

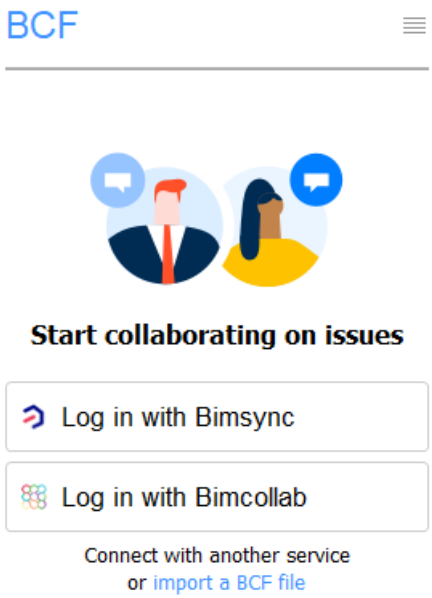
| | |
|---|---|
| **Render materials** | The **Render materials** Panel contains a library of predefined materials that you can assign to solids by dragging them on.<br>You can also see a list of all the materials that you are currently using in your drawing.<br>See the **Materials** article to see how to edit and create materials. |
| **Report** | Some commands use the **Report** panel to report the command output; e.g. all SHEETMETAL (Sm*) commands, DMAUDIT and DMSTITCH. |

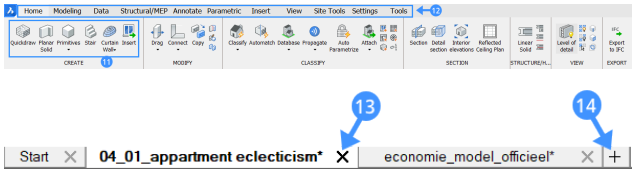
**Bricsys 24/7**

Login to your Bricsys 24/7 account here to collaborate on your project in the cloud. BricsCAD 24/7 makes project management, a collaboration between project teams, cloud computing security, and task automation possible.
When logged on, you can see an overview of your in the cloud Projects, Folders and Documents as shown in the image below:




See the **BricsCAD 24/7** panel article to see in detail how this panel works.

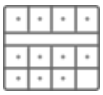
| | |
|---|---|
| **BIM BCF** | Here you can import BCF (Bim Collaboration Format) files and connect to a BCF server. The BCF topics are listed in this panel.<br><br>The top part of the new **BCF** panel shows all the topics and can look similar to the **Library** panel.<br>The lower part of the new **BCF** panel shows the currently active topic, with a bigger snapshot and the most common metadata: comment, author, status, etc.<br>Double-clicking on an issue activates the topic. |
| **Details** | In the **Details** panel you can:<br>• View saved details.<br>• Organize saved details.<br>• Propagate details.<br>• Open details and adjust them in the model space. |




### 20.2.3 Quad cursor

#### 20.2.3.1 About

The **Quad cursor** menu, or **Quad** for short, is an alternative to the Command line or toolbars, offering a rich set of tools while requiring fewer clicks, without cluttering the screen with loads of grip-glyphs. The rollover tips are a limited set of properties displayed when selecting or hovering over an entity.



To toggle the **Quad** on/off, do one of the following:

- Click the **Quad** field in the Status bar.
- Press the F12 function key.
- Edit the QUADDISPLAY system variable in the **Settings** dialog.
- Type QUADDISPLAY in the Command line, then choose an option.

To toggle the **Rollover Tips** on/off, do one of the following:

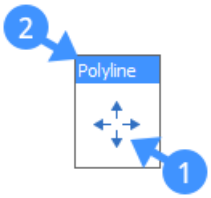
- Click the **RT** field in the Status bar.
- Type ROLLOVERTIPS in the Command line then set a value.

### 20.2.3.2 Quad layout

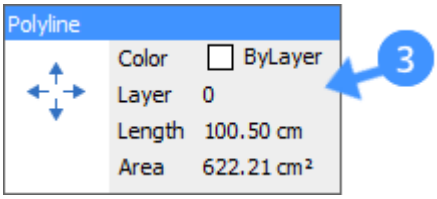
The **Quad** has a floating feature set that adjusts its content, depending on what you are or are not highlighting, and what you may have selected in the current workspace.

- Based on the highlighted entity type, the **Quad** displays the most recently used tool (1) with the entity name (2).

  ***Прим.:*** When multiple entities are under the cursor, hit the TAB key to cycle through the different entities.

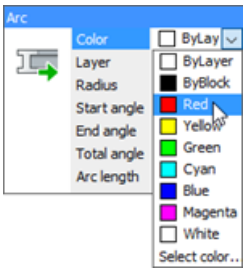



- When the **Rollover Tips** setting is on, a limited set of properties (3) of the entity under the cursor display in the **Quad**.




  The property values in the **Rollover Tips** can be edited.

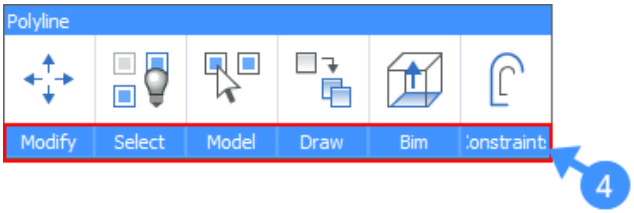
  The following figure shows the color of an arc being changed using the **Rollover Tips**.




***Прим.:*** To display the **Rollover Tips**, simply click the title bar of the **Quad** after expanding it.

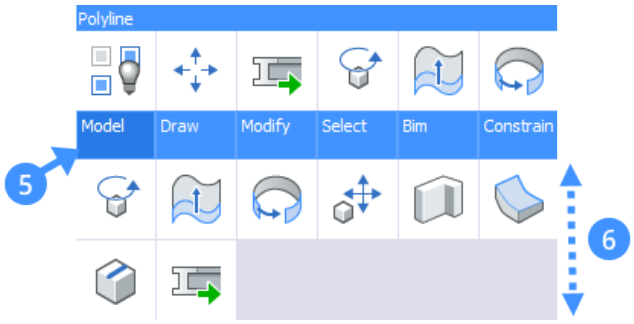
- When the cursor is moved on the **Quad**, a number of tool tabs display at the bottom (4).

  ***Прим.:*** The number of tool tabs is controlled by the **Quadwidth** user preference (default = 6).

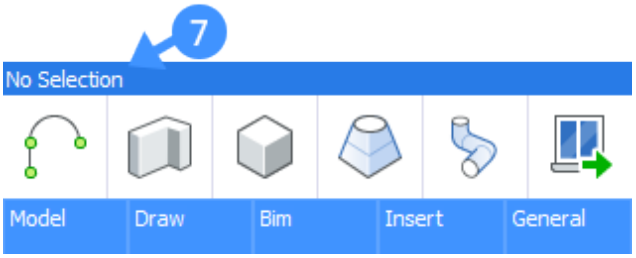



- When you move the cursor over the tool tabs title bar (5), the tool groups expand (6) under the cursor, which provides you with more tool options.

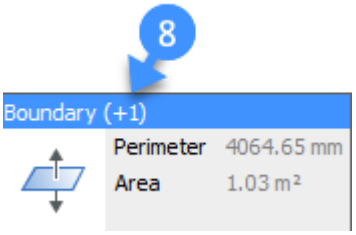
  ***Прим.:*** The number of available tool groups and the number of tools within a tool group depend on the entity type under the cursor.




- The **Quad** can also be accessed in the no selection If the cursor is on an entity, hold down the right mouse button longer to display the **No Selection** Quad (7). Or when there is no selected/highlighted entity in the drawing, right-click to display the **No Selection** Quad.



- When an entity is selected and another one is highlighted, next to the entity type (title bar on the top) a plus one (8) appears in the parentheses. The count, regardless of the type, increases each time you select an entity. To reveal the selected the entity types which are hidden in the parentheses, expand the quad by moving your cursor on it.

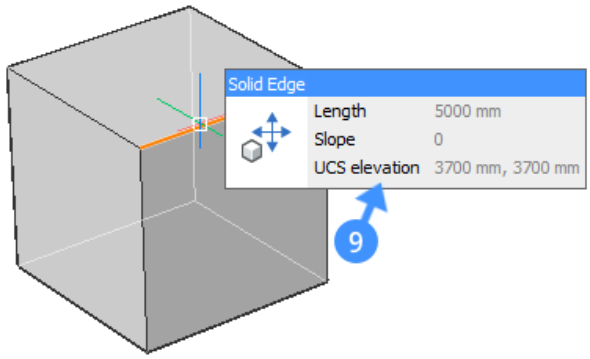



***Прим.:*** The **Quad** also adapts to the type of entities that have been selected. If you select two surfaces, you will be provided with different tool options in the Quad, then if you selected a solid and a surface.

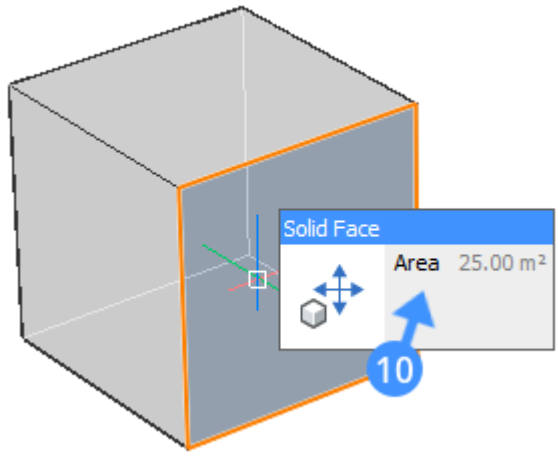
### 20.2.3.3 About: selecting entities

The **Quad** adapts to the type of entities that you have selected. If you select a surface, you will be provided with different tool options in the Quad, then selecting a solid or edge.

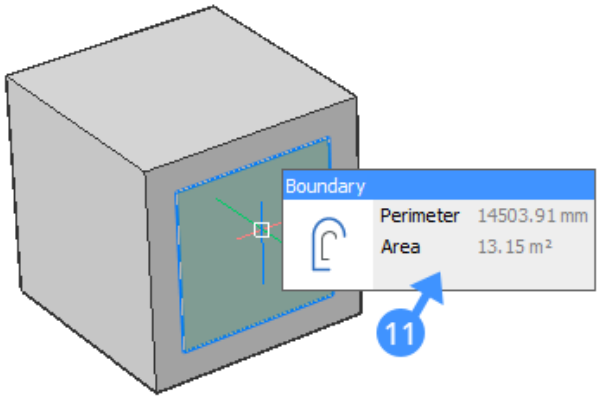
- Selecting an **edge**, the **Quad** appears showing the edge length, slope, and elevation (9).



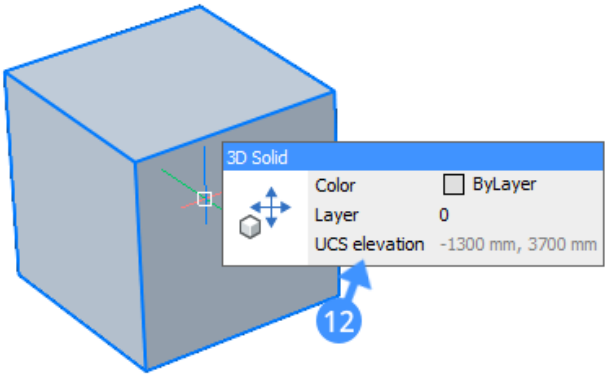

- Selecting a **solid face**, the surface area is shown in the **Quad** (10).




- Highlighting a **boundary**, the **Quad** displays the perimeter and area of this boundary (11).




- Highlighting a **solid**, the **Quad** displays the color, layer, and elevation of the solid (12).




### 20.2.3.4 Procedure: using the quad

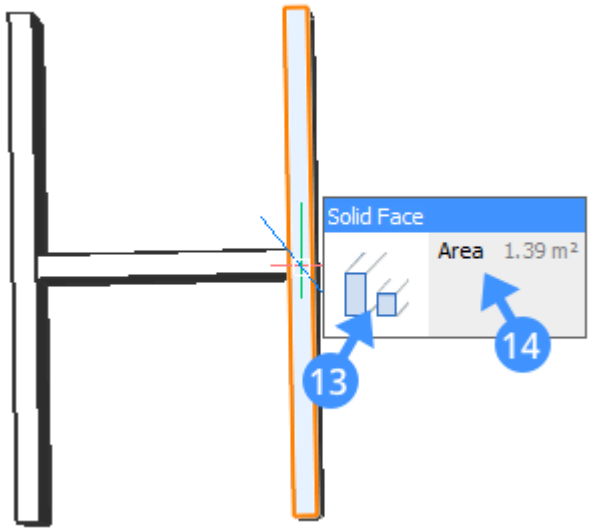
The following are general steps to use the **Quad** cursor.

***Прим.:*** The **Quad** and **Rollover tips** are both enabled.

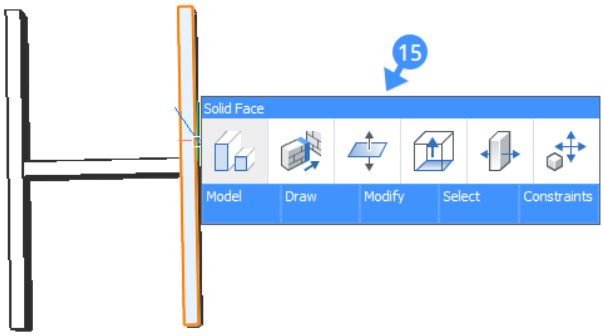
1 Hover the cursor over the entity you want to manipulate.

1 The entity under the cursor highlights.

The **Quad** displays a single icon (13), which is the most recently used tool with this entity type, and a limited set of properties (14) of the highlighted entity.




2 Right-click to launch the recently used tool. Or move the cursor to the single icon to expand the **Quad** (15), then select one of the available tools to manipulate the entity.




3 Move the cursor over the blue field to further expand the **Quad**.

All Quad tool groups for the current workspace display. When the first tool group is expanded, click an icon to use a tool or move the cursor to a different tool group.

### 20.2.4 Viewing entities

#### 20.2.4.1 Commands

HIDEOBJECTS, ISOLATEOBJECTS, UNISOLATEOBJECTS

### 20.2.4.2 About

If you work with many entities in a drawing, everything that you have in the drawing is visible in the view. When you want to temporarily make only a few entities or a particular entity visible and manage them in a view, you can use the **Hide** or **Isolate** tools.

### 20.2.4.3 Hiding entities

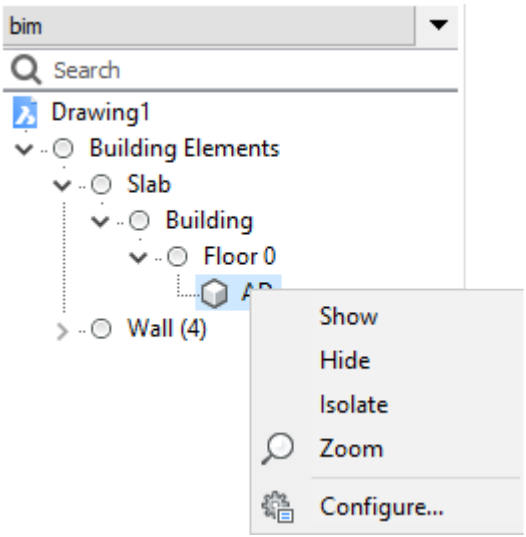
The **Hide** tool temporarily hides the selected entities in the view.



For more information about this command, visit the Command Reference article HIDEOBJECTS.

(**Optional**) **Structure Browser** can also be used to hide and isolate selected entities as well as revealing all hidden entities in the model space. This is an easy way to quickly hide or isolate a certain category of objects or entities with a certain property value etc.

To do so, right-click on the entity in the **Structure Browser** and choose one of the options in the context menu (see below image).



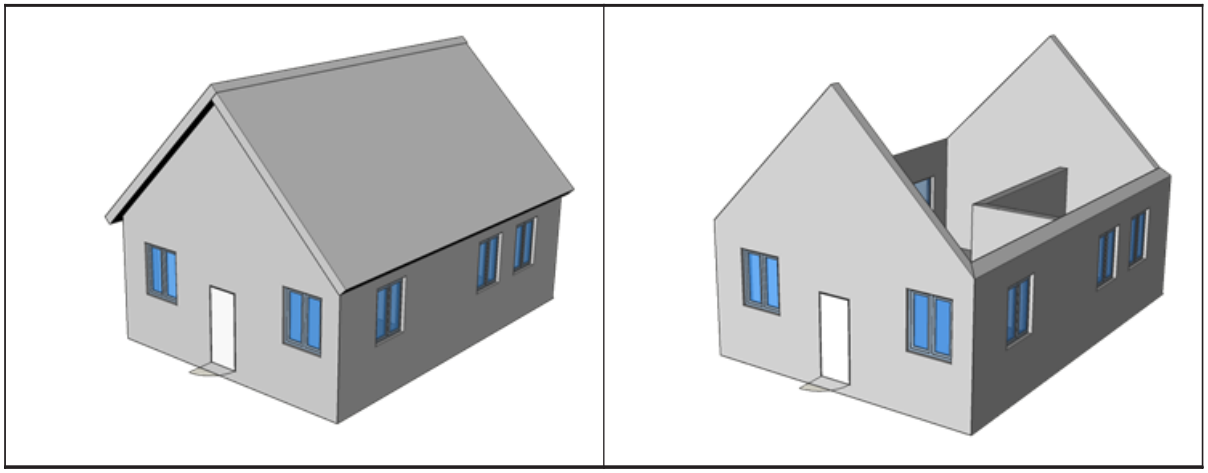

### 20.2.4.4 Procedure: hiding entities

1 Select one or more entities in the drawing area.

2 Select **Hide Entities** from the Quad.

All entities that have been selected are hidden in the view.

The following illustration shows an entire model on the left. In the second image, the roof was temporarily hidden.



### 20.2.4.5 Isolating entities

The **Isolate** tool temporarily hides all entities except those that have been selected.



For more information about this command, visit the Command Reference article ISOLATEOBJECTS.

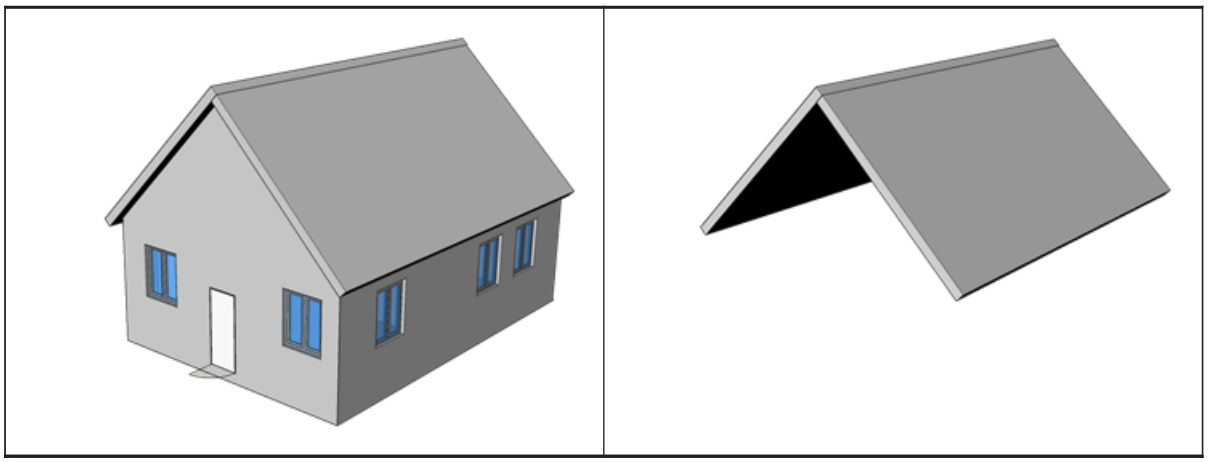
### 20.2.4.6 Procedure: isolating entities

1 Select one or more entities in the drawing area.

2 Select **Isolate Entities** from the Quad.

All entities that have been selected before are isolated in the view.

The following illustration shows the same building model from the below. In the second image, the roof was temporarily isolated.



### 20.2.4.7 Showing entities

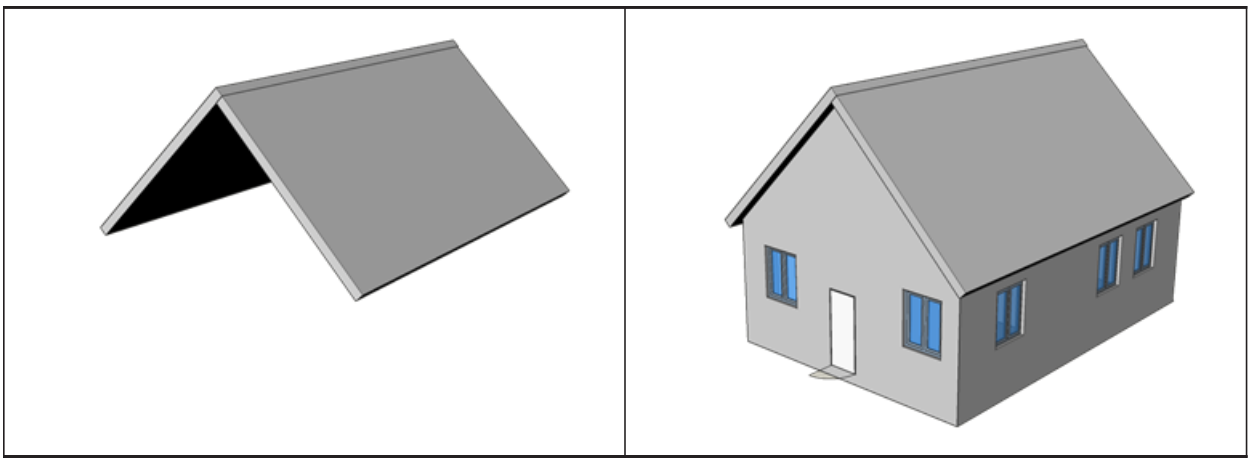
The **Show Entities** tool brings back all hidden entities in the view.



For more information about this command, visit the Command Reference article UNISOLATEOBJECTS.

If you have any hidden entities in your drawing, you can reveal them in the view using the show entities tool.

The following illustration shows the isolated roof elements from the above example. The second image shows the result after using the **Show Entities** tool.



### 20.2.5 Selecting entities

#### 20.2.5.1 About

In BricsCAD, when you hover over an object, it will be highlighted. After highlighting an entity, you left click to select the highlighted entity, then it will be selected.

**Highlighting**: you only can highlight one entity at a time.

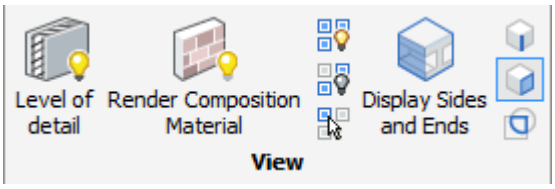
**Selecting**: you can select as many entities as you want.

Most of the modeling process can be done by highlighting the entity and using the Quad. For example, when you highlight a surface, the Quad will open, pick the command you want, without having to select the surface first.

#### 20.2.5.2 Selection modes

The selection modes allow you to control which sub-entities (faces, edges, and boundaries) should highlight in selection preview, and can be selected.

Ribbon: **Home tab** > **View panel** > **Selection modes**




***Прим.:*** To display the **Ribbon**, right-click on a toolbar, then choose **Ribbon** in the context menu.

**Settings**: **Program options** > **Selection** > **Selection Preview** > **Selection Modes**

| Selection modes | 0x0007 (7) |
|---|---|
| 1 | Select edges |
| 2 | Select faces |
| 4 | Select detected boundaries |

**Keyboard**: SELECTIONMODES

In the following steps, each selection mode option is given with its figure and explanation.

- **Enable detection of 3D solid edges** ( ): controls whether 3D solid edges are highlighted by selection preview and can be selected. If you enable this option, you can select the edges of 3D solids by clicking it once.
- **Enable detection of 3D solid faces** ( ): controls whether 3D solid faces are highlighted by selection preview and can be selected. If you enable this option, you can select the faces of the 3D solids by clicking it once.
- **Enable boundary detection** ( ): controls whether closed boundaries in XY-plane of the current coordinate system or on the face of 3D solids are detected. If you enable this option, you can select closed boundaries from the faces of 3D solids by clicking it once.
- **Enable selection of Sides and Ends** ( ): controls the bimSides and BimEnds visibility of all linear elements.

***Прим.:*** BIM Sides/Ends are only available on classified linear elements with the attached library profile.

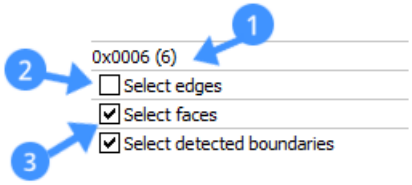
### 20.2.5.3 Changing selection modes setting

A sub-entity is a face, edge or boundary of a 3D solid. To control which sub-entities you select in your model do one of the following:

***Прим.:*** The default value of the SELECTIONMODES system variable is 6, which means the detection of 3D solid edge is disabled, while the detection of 3D solid faces and boundaries are enabled.

- Click the buttons on the **Settings** panel on the **Ribbon** and find **Selection Modes**.

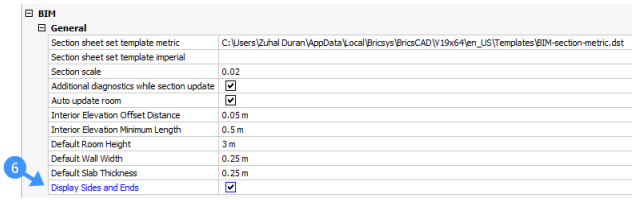
  The following illustration shows the sum of the values (1) of the SELECTIONMODES system variable when the **Select edges** option is disabled (2) after clicking the checkbox once, the **Select faces** and **Selected detected boundaries** are enabled (3).




- Type SELECTIONMODES in the Command line and change the current parameter.
- Use the CTRL key on the keyboard to temporarily toggle between the different selection modes. These key functions enable you to select any type of entity.

- To display the BimSides/Ends for linear elements you can use DISPLAYSIDESANDENDS system variable in the **Settings** dialog box or set the DISPLAYSIDESANDENDS system variable to 1 in Command line.

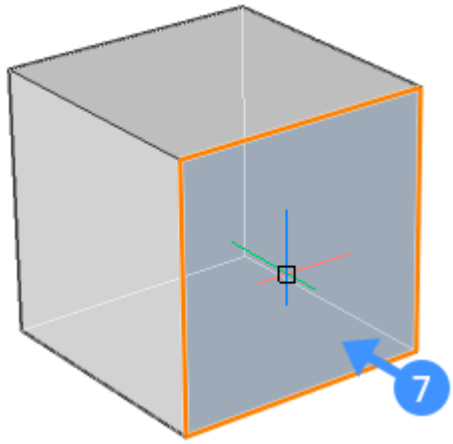
  The following illustration shows the **BIM** > **General** system variable when the DISPLAYSIDESANDENDS is enabled (6) after clicking the checkbox once.



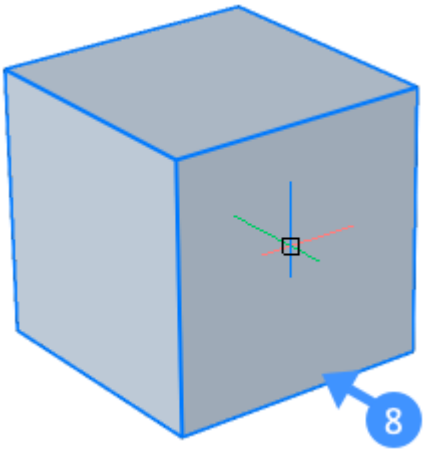

### 20.2.5.4 Selecting sub-entities

When **Select edges** is off, **Select faces** is on, **Boundary Detection** is on (default), do one of the following:

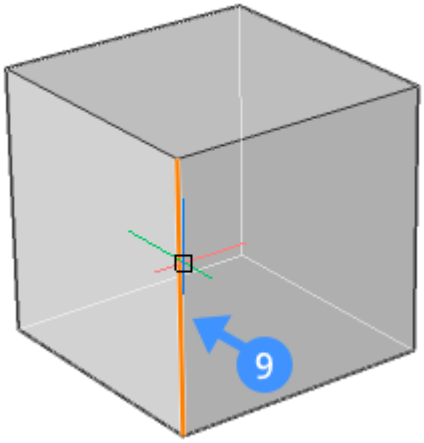
- To highlight a solid face, hover over the face with your mouse cursor. The highlighted face displays in orange (7). To select the face, left-click it when the face is highlighted. To select multiple faces of the solid, left-click them one by one.



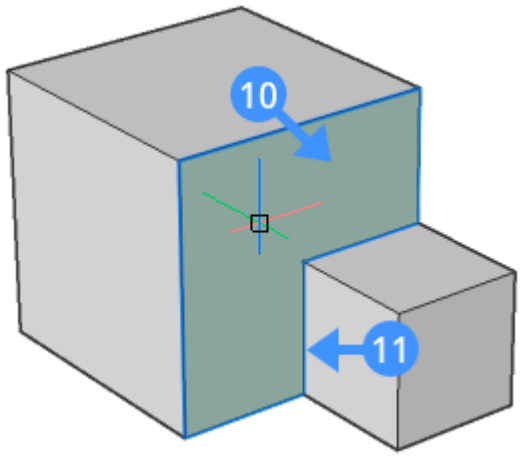

- To highlight a solid, hover over one of its faces while holding down the CTRL-key. The solid displays in blue (8). To select the solid, left-click it while the solid is highlighted. To select multiple solids left-click them one by one.




- To highlight an edge, hover over the edge of the solid while holding down the CTRL-key. The edge displays in orange (9). To select the edge, left-click it while the edge is highlighted. To select multiple edges left-click them one by one.



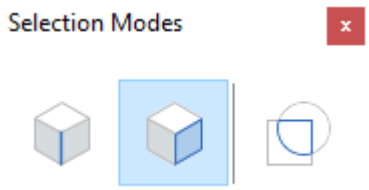

- To highlight a boundary, hover over the detected boundary on the face of the 3D solid.
- When the detected boundary is highlighted, it displays with a color. By default, inside of the boundary displays in green (10) and the outer edge of the boundary displays in blue (11).




***Прим.:*** This setting can be changed by typing BOUNDARYCOLOR on the Command line (default value = 95).

### 20.2.5.5 Procedure: Modify the thickness of a solid

1 Make sure the **Select faces** option of the SELECTIONMODES system variable is selected.




2 Move the cursor over the face of a solid

3 If necessary, press Tab to highlight an obscured face.

4 When the face highlights, choose **Push/Pull** in the Model command group of the Quad.

5 The selected face of the solid is dragged

6 The dynamic dimension field displays the relative displacement of the face (= with respect to its original position).

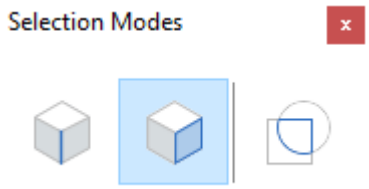
7 (**Optionally**) Press Tab to change the face selected as the reference face for the dynamic dimension.

***Прим.:*** When a composition with locked thicknesses (non-variable thickness) is applied to a solid and if the thickness of the solid is not equal to the total amount of the composition thickness, the solid displays with red hatches. Execute the BIMUPDATETHICKNESS command to re-apply the composition to such solid.

### 20.2.5.6 Procedure: Modify the height of a wall

**Modify the dimensions of a single wall**

1 Make sure the **Select faces:** option of the SELECTIONMODES system variable is selected.




2 Move the cursor over the top face of the wall.

3 If necessary, press Tab to highlight an obscured face.

4 When the face highlights, choose **Push/Pull** in the Model command group of the Quad.

The selected face moves dynamically with the cursor.

The dynamic dimension fields display the incremental height.

Do one of the following:

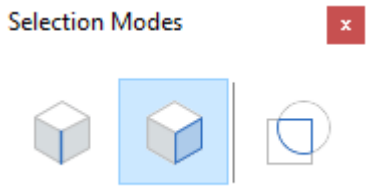
5 Type a value in the dynamic dimension field and press Enter.

6 Press Tab to use another reference face, e.g. the bottom face of the solid, then type a value in the dynamic dimension field.

**Modify the height of multiple walls**

Use the SELECTALIGNEDFACES command (**Quad** > **Select**: ) it is easy to select the top face of all solids that lie in the same horizontal plane.

1 Make sure the **Select faces** option of the SELECTIONMODES system variable is selected.




2 Move the cursor over the top face of one of the walls.

3 If necessary, press Tab to highlight an obscured face.

4 When the face highlights, choose SELECTALIGNEDFACES in the **Solid/Face Selection** command group in the Quad.

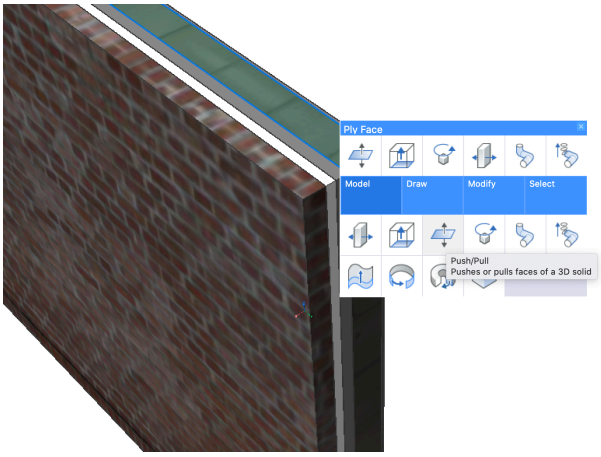
All faces lying in the plane of the selected face are selected.

5 Now follow the same steps as for a single wall.

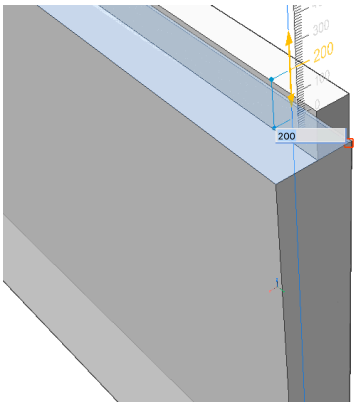
**Modify the height ply-by-ply**

When the **Display Composition** property of a solid is set (LEVELOFDETAIL system variable =2 and, optional, RENDERCOMPOSITIONMATERIAL system variable =1) and the **Select faces** option of the SELECTIONMODES system variable is selected, the height or length of the composition plies can be modified separately.

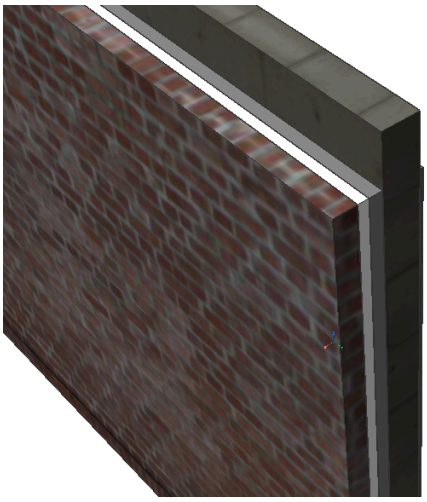
1 Move the cursor over the ply you want to edit and choose **Push/Pull** in the **Model** command group of the Quad when the ply highlights.




2 Type value in the dynamic distance field.




3 Press Enter to accept the distance.



### 20.2.5.7 Selecting obscured sub-entities

In your drawing, some entities or sub-entities (faces, edges, and boundaries) might be hidden behind other entities. The Tab key enables you to cycle through sub-entities.

The following steps show how to highlight a hidden face that lies beneath the cursor.

When **Select edges** is off, **Select faces** is on, **Boundary Detection** is on (default).

1 Place your cursor on one of the solid faces.

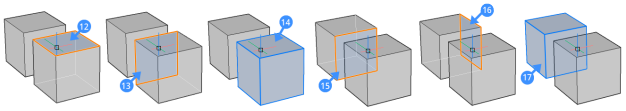
1 In the example below, the top face of the front solid is highlighted (12).

2 Press Tab key to cycle through the hidden entity.

3 Now, the back face of the front solid is highlighted (13).

4 Repeatedly pressing the Tab key, while keeping the cursor at the same position, allows you to highlight the entire solid in the front (14) and the front face (15) and right face (16) of the rear solid. Finally, the entire rear solid is highlighted (17).

5 When the entity or sub-entity that you want to select is highlighted, left-click it to select.




### 20.2.5.8 Selecting multiple entities using the selection boxes

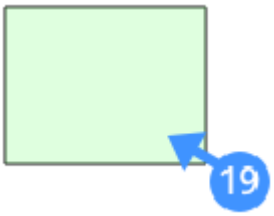
The selection boxes allow you to select one or more than one entity at a time. To use a selection box, left-click on an empty space in the drawing area, and move the cursor until the selection box covers the entities that you want to select.

There are two types of selection boxes in BricsCAD;

**Blue selection box** (18) appears when you create a box from left to right. The blue selection box will only select entities that are entirely covered by this selection box.




**Green selection box** (19) appears when you create a box from right to left. The green selection box will select all elements that are entirely covered by the selection box and all the elements that intersect with one of the edges of the box.

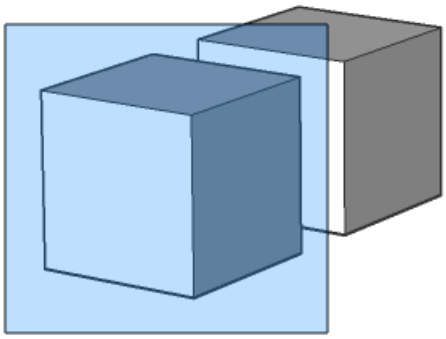


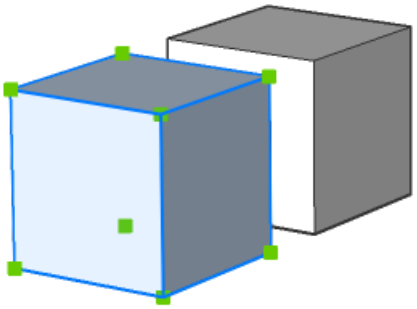

The following steps show how to select multiple entities using the blue selection box:

***Прим.:*** By default, the selection box only selects entire entities (e.g. solids, lines, polylines, blocks, …) but not sub-entities (e.g. solid faces, solid edges).

1 Click and move the mouse to the right to create a blue box around the entities you want to select. When the entity is completely inside the window, it will be added to the selection set.

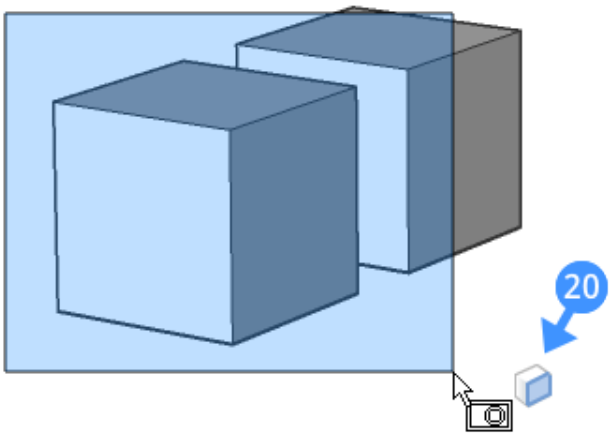
1 The example below shows the blue selection box usage when the selection mode is set to **Solids** (default).

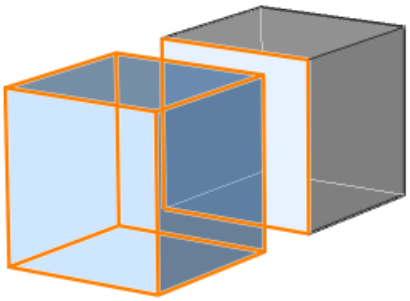




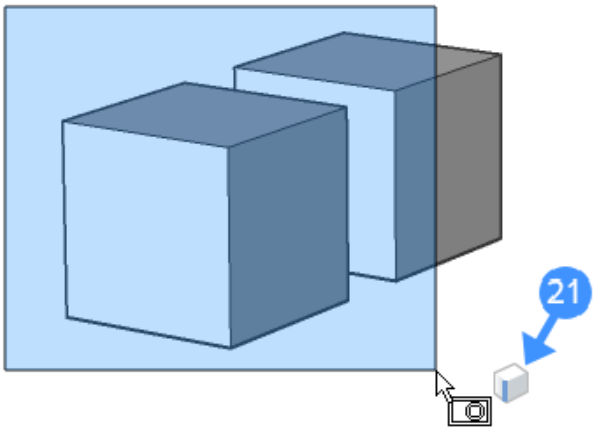
The blue selection box covers the left cube completely and thus only this cube is selected in the model.

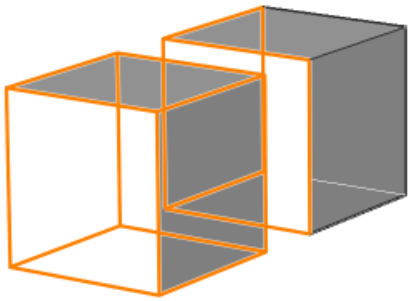
2 To select faces of 3D solids and surfaces, press CTRL key once during window selection. The icon (20) next to the cursor indicates the selection mode. The faces that are completely inside the window will be selected when the selection mode is set to **Faces**.



3 To select edges of 3D solids and surfaces, press the CTRL-key twice during window selection. The 'Edges' icon (21) appears next to the cursor. The edges that are completely inside the window will be selected when the selection mode is set to **Edges**.

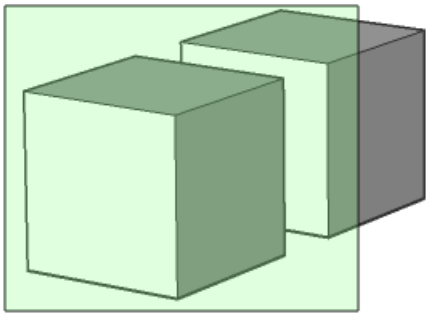


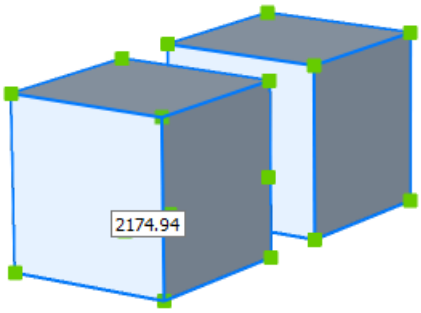



The following steps show how to select multiple entities using the green selection box:

4 Click and move the mouse to the left (crossing window) to create a green box around the entities you want to select. When the entities overlap the window or are completely inside the window, they will be added to the selection set.

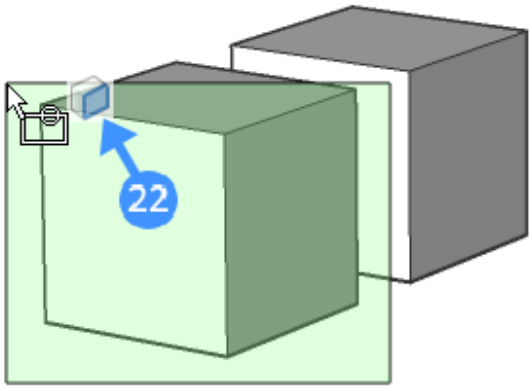
5 The example below shows the green selection box usage when the selection mode is set to default.

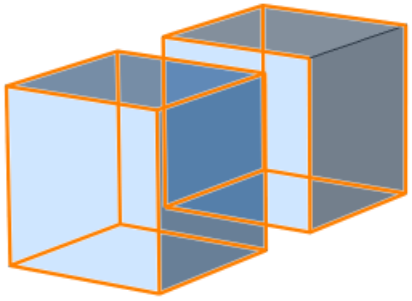




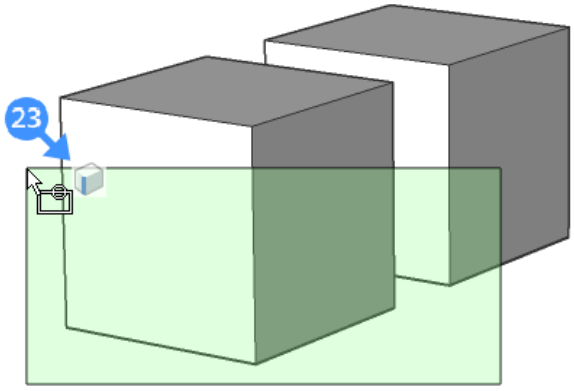

The green selection box selects both cubes in the model as the selection box overlaps with the two cubes.

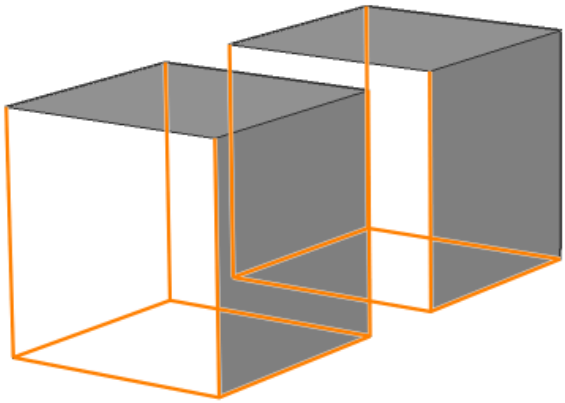
6 To select faces of 3D solids and surfaces, making use of the green selection box, press the CTRL key once during window selection. The ‘Faces’ icon appears next to the cursor (22). Any faces that are completely or partially inside the crossing window will be selected.



7 To select edges of 3D solids and surfaces, making use of the green selection box, press the CTRL key twice during window selection. The ‘Edges’ icon appears next to the cursor (23). Any edges that are completely or partially inside the crossing window will be selected.



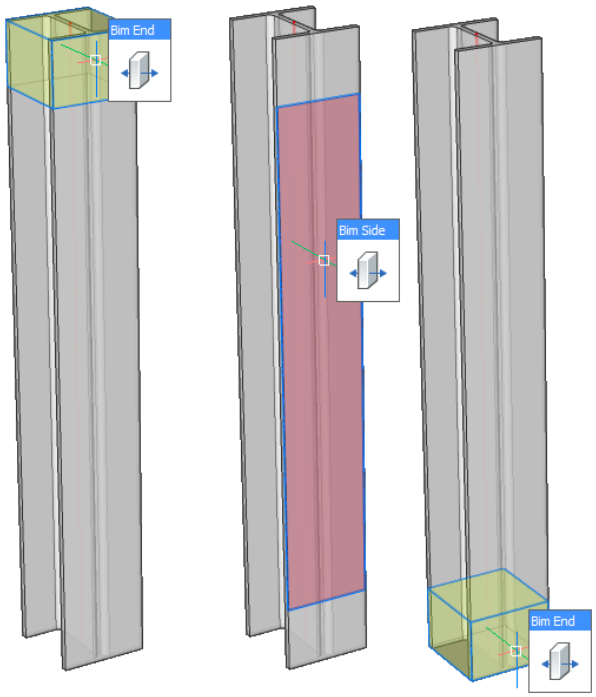



### 20.2.5.9 Selecting/Highlighting BIM sides and ends on profiles

1 Click on **Enable selection of Sides and Ends** icon in the BIM Toolbar ( see below image).



2 Hover over the linear solid in the model space when the cursor is on the top or bottom side of linear solid, **Bim End** face displays in yellow whereas the side face of linear solid displays in red.




### 20.2.6 Level of detail

The RENDERCOMPOSITIONMATERIAL and LEVELOFDETAIL system variables control the display of materials and multiply compositions.

#### 20.2.6.1 Level of detail



Quad: Bim

Ribbon: **Home** | **View**

Toolbar: **BIM**

Click the Icon to toggle the setting.

| Option | Description |
|---|---|
|  [0] Low | Multiply compositions do not display. |
|  [2] High | The individual plies of the multiply compositions are visible and editable. |

#### 20.2.6.2 Render composition materials



Quad: Bim

Ribbon: **Home** | **View**

Toolbar: **BIM**

Click the Icon to toggle the setting.

| Option | Description |
|---|---|
|  [0] Off | Materials are not rendered. |
|  [1] On | Materials are rendered. When Level Of Detail = 0 (Low), major faces of solids, that have a multiply composition attached, are rendered in the material of the underlying ply. Minor faces are rendered in the material of the exterior ply. |

## 20.3 Методы моделирования

### 20.3.1 BIM linear solid

#### 20.3.1.1 Commands

BIMLINEARSOLID, BIMAPPLYPROFILE, BIMPROFILES, BIMADDECCENTRICITY, BIMRECALCULATE

#### 20.3.1.2 About BIM linear solid

The Linear Solid tool allows you to create structural members or flow segments. The tool can be made use of creating a variety of structural member profiles or MEP elements.



For more information about this command, visit the Command Reference article BIMLINEARSOLID.

For more information about **MEP modeling**, visit the help article **MEP Modeling** and **MEP Flow Connection Points**.

For more information about **Structural Steel**, visit the help article **Structural Steel**.

Before starting the process, make sure that Quad is active. Next, follow these steps:

1 Hover over the **Model** tab in the Quad and select the Linear Solid tool.

***Прим.:*** You can also launch the **BimLinearSolid** command by entering it in the command line.

You are prompted: Set start point or [Follow/Rectangular/Circular/Library/select in Model/adJust

profile] <Last point>:

2 Select a start point.

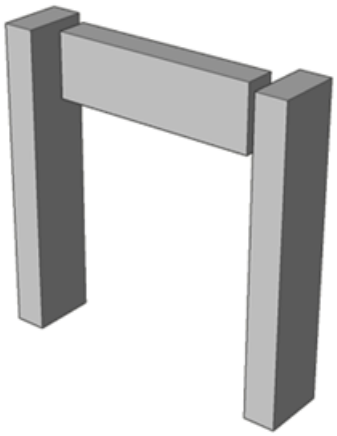
3 You are prompted: Set next point or [Angle/Length/Undo/Quarter turn/Rotate/choose other Profile/adJust profile]:

4 Move the mouse cursor to draw the beam or column.

5 Click to specify the endpoint or enter a value for the distance using the dynamic dimension field.

6 Press Enter key to accept it.

7 The following illustration shows a beam and two columns with the default rectangular profile.

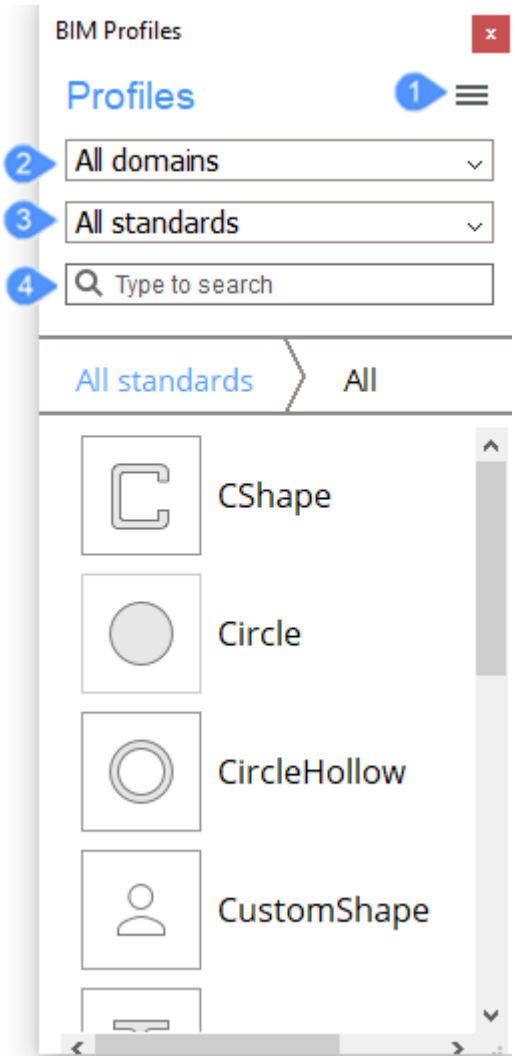


### 20.3.1.3 About BIM profiles

You can access the **BIM profiles** panel by clicking the I-beam on the right-hand side of your screen. If the **BIM Profiles** toolbar is not displayed on the right-hand side of your screen, right-click on the **Tool** panel at the right-hand side of your screen, choose **Panels** in the context menu, then check **BIM Profiles**.

***Прим.:*** You can also find the **BIM profiles** panel by typing BIMPROFILES in the Command line.

The **BIM Profiles** panel will be displayed:




The drop-down box at the top (2) acts as a filter. There are different domains on which you can filter: **Generic**, **Structural Concrete**, **Structural Steel**, **HVAC**, **Piping** and **Electrical**.

***Прим.:*** If you don't have any profiles in a certain domain, you will not be able to filter on it.

The second drop-down boxes (3) allow you to filter the profiles by the country standards.

The search box (4) allows you to filter by name.

The menu (1) on the right-hand side allows you to access the **BIM Profiles** dialog box. Clicking the menu icon displays a context menu with a couple of options.

Select **Open profile dialog** on the menu to open the **Profiles** dialog box.

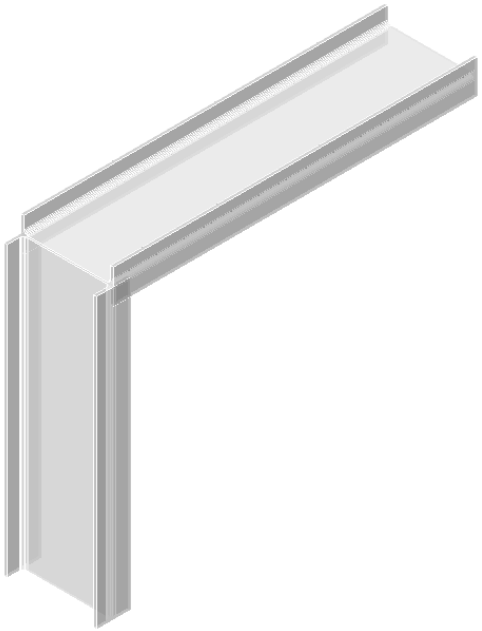
### 20.3.1.4 How to use the BIM Profiles panel

1 Drag and drop the desired profile from the **BIM Profiles** panel to the model space.

***Прим.:*** This command works in the same way as the LINE command.

2 To define the starting point and the following point(s) click in the model space or use the dynamic dimension fields.

3 Press Enter to end the command.



***Прим.:*** BricsCAD doesn't automatically generate the connections between the different segments because the chosen profiles are structural steel.

### 20.3.1.5 About BIMAPPLYPROFILE

The **Apply Profile** tool allows you to change the profile of linear entities and linear solids. Linear solids have a constant cross-section over a linear extrusion path. The tool recognizes clippings and openings. Using the **Apply Profile** tool, you can create columns, railings, beams, pipes, ducts and more.



For more information about this command, visit the Command Reference article BIMAPPLYPROFILE.

**How to apply profiles to an existing line or solid: with the BIMAPPLYPROFILE command**

1 Select the line or solid you want to apply a profile to and launch the BIMAPPLYPROFILE command.

2 Press Enter and the **Profiles** dialog box, appears.

3 Choose a profile and press the **Select** button in the bottom-right corner.

4 Press Enter to end the command. The profile is applied to the line or solid in your model.

**How to apply profiles to an existing line or solid: with the BIM Profiles panel**

1 Open the **BIM Profiles** panel.

***Прим.:*** You can access the **BIM Profiles** panel by clicking the I-beam icon on the right-hand side of your screen.

2 Select all the entities you want to apply a profile to.

3 In the **BIM Profiles** panel, search for the profile you want apply to the selected entities.

4 Drag the chosen profile onto one of the selected entities.

5 The profile is applied to the line or solids in your model.

***Прим.:*** By applying a profile to a solid, the solid will be replaced by the chosen profile. In case of a line, the line will not be replaced, but a profile was added with his axis the same as the selected line.

6 Since the line is no longer needed, you can delete the line.

### 20.3.1.6 About BIMADDECCENTRICITY

BIMADDECCENTRICITY command allows you to control the relative position of the axis of a linear solid.



For more information about this command, visit the Command Reference article BIMADDECCENTRICITY.

1 Select an existing linear solid or profile in the drawing area.

2 Launch the BIMADDECCENTRICITY command by entering it in the Command line or select the command in the Quad (under the model tab) or in the Ribbon (in the **Structural/MEP** tab).

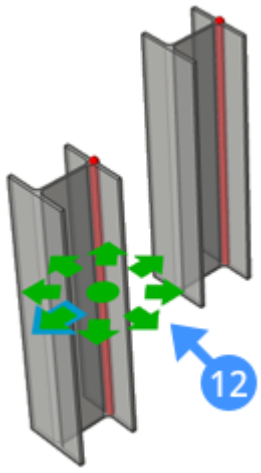
The eccentricity widget appears.

3 To set an eccentricity, click one of the nine locations from the widget, then right-click to confirm.

4 Optionally, type a location in the Command line.

5 The options of the locations are: **TR**: Top Right, **TM**: Top Middle, **TL**: Top Left, **MR**: Middle Right, **C**: Center (resets the axes of the selection set to the default location), **ML**: Middle Left, **BR**: Bottom Right, **BM**: Bottom Middle, **BL**: Bottom left .

6 Press Enter to accept changes. The following illustration shows the Eccentricity widget (12) on the linear solid profile.




### 20.3.1.7 About BIM recalculate axis

The Recalculate Axis tool recalculates the axis of linear building elements and repositions it back to the centerline of a linear element.



For more information about this command, visit the Command Reference article BIMRECALCULATEAXIS.

**Recalculate the axis of linear building elements**

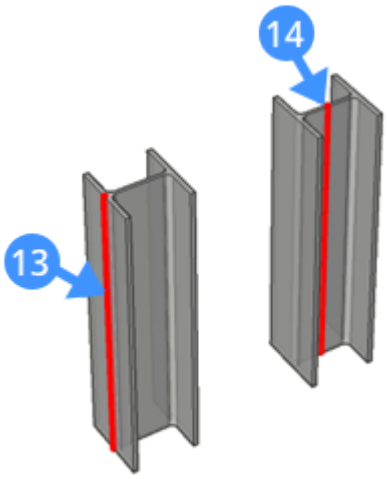
1 Select any linear building element in your model.

2 Launch the BIMRECALCULATEAXIS command.

3 Press Enter.

4 The axis of the element is moved to the default position.

5 The following illustration shows the result of using BIMRECALCULATEAXIS on the linear solid. The axis (13) is moved to the centerline (14) of the linear element.




### 20.3.1.8 Helpful settings when modeling Linear Solids

**Display Axes**

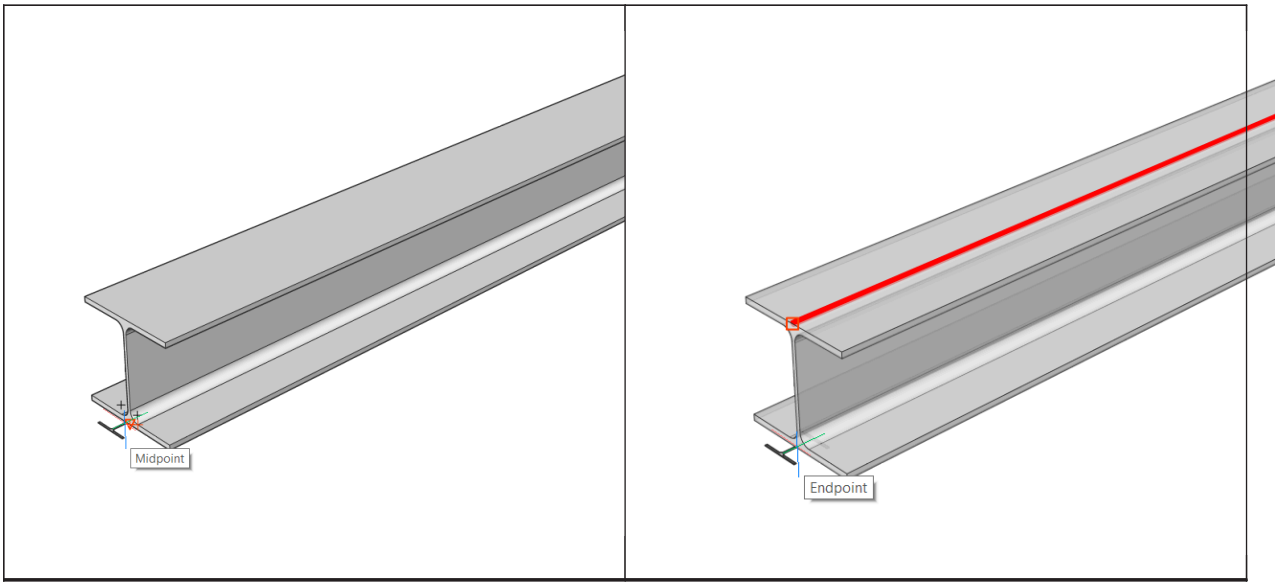
The DISPLAYAXES variable system can hide or view the Axes of linear solids in order to edit the structure more easily.



If DISPLAYAXES is toggled on, the axes of the linear solids are visible as a thick red line. In this case, you will only be able to snap the endpoint of the axis as a starting point to add more linear solids or profiles to your already existing structure.

When DISPLAYAXES is toggled off, no axis is visible in your drawing area and you're able to snap all specific points you have ticked in the context menu of ESNAP.

| DISPLAYAXES OFF | DISPLAYAXES ON |
|---|---|




**BIMOSMODE**

The BIMOSMODE variable system gives you the possibility to choose the level of detail you are snapping to.

1 Enter BIMOSMODE in the Command line or in Settings.

1 You are prompted: New current value for BIMOSMODE (0 to 3) <2>:

2 Choose a new value (type a number from 0 to 3).

| Value | Snapping detail |
|---|---|
| 0 | Turns on all snapping. |
| 1 | Turns off all snapping to solids except for the axis of the solid. |
| 2 | Turns off all snapping to BIM Grid axes except for axis intersections. |
| 3 | Turns off all snapping to BIM Grid axes except for axis intersections.<br>Turns off all snapping to solids except for the axis of the solid. |

3 Press Enter

### 20.3.2 BIM window create

#### 20.3.2.1 Commands

BIMWINDOWCREATE

### 20.3.2.2 About

In BricsCAD, the **Window Creator** tool allows you to turn a different set of design ideas into a window. You can use the tool on windows of all shapes and sizes, which enables you to utilize a significant amount of window styles in your BIM project.

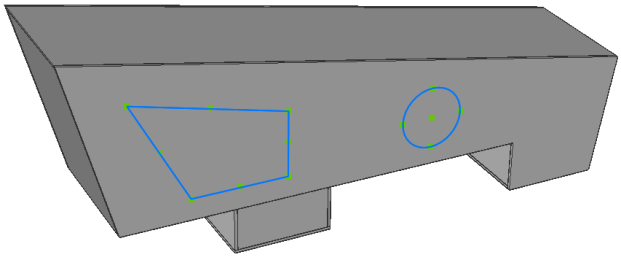
When you have a closed 2D entity or boundary on the face of a solid (wall), the window creator tool enables you to use these closed entities to make your window. Using the window creator tool, you can select any window styles in the dialog box and change the properties of this created window in a BIM model.



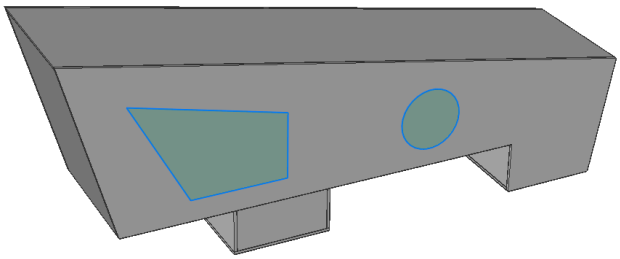
### 20.3.2.3 Creating a parametric window using boundaries

1 Draw a polygon on the face of a solid (wall).

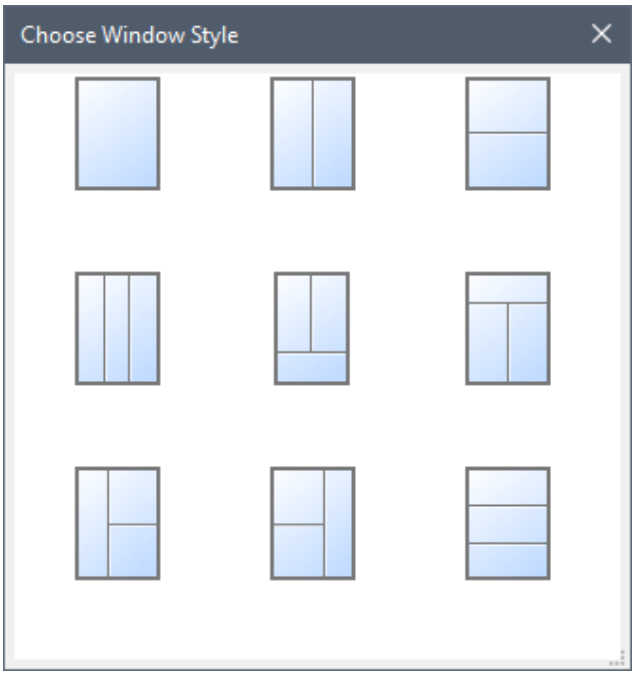
1 Based on this base profile, the window opening will be created.



2 Make sure **Boundary Detection** (select detected boundaries) is enabled into SELECTIONMODES system variable. Hover over the inside of the polygon and select the **Create Window** tool from the Quad.

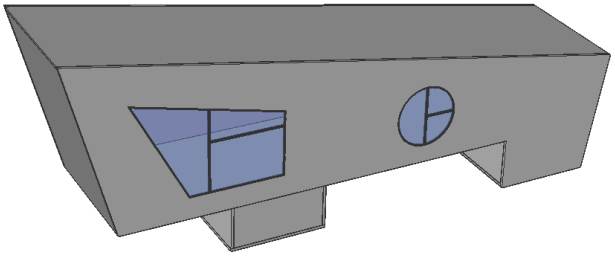


A **Choose Window Style** dialog box displays:

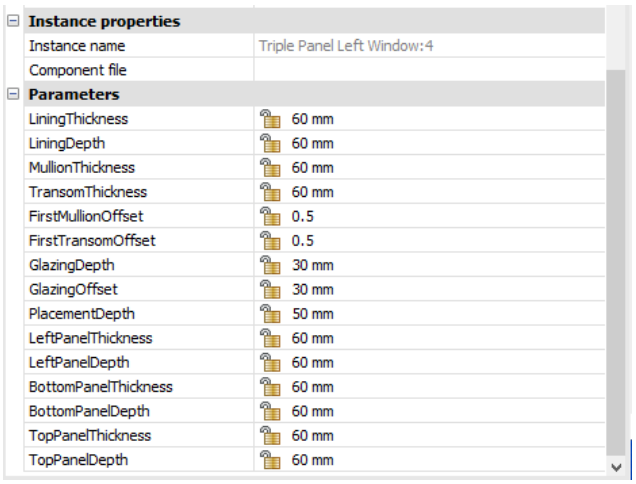



3 Select a window style in the dialog box.

4 The window is created on the wall with the selected style. The following illustration shows the triple panel left window style which was selected to create the windows on the face of the wall.



5 (**Optional**) You can refine the window design by changing the default values of the parameters. To do this, select the window and change the desired parameters in the **Properties** panel.



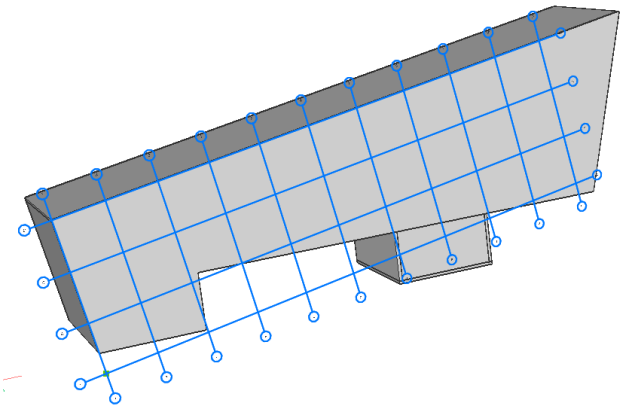
To change one of the default parameters of this window, type a new value in the property field.

Press Enter to accept the new value of the parameter. The window should be updated automatically.

### 20.3.2.4 Create windows based on a grid

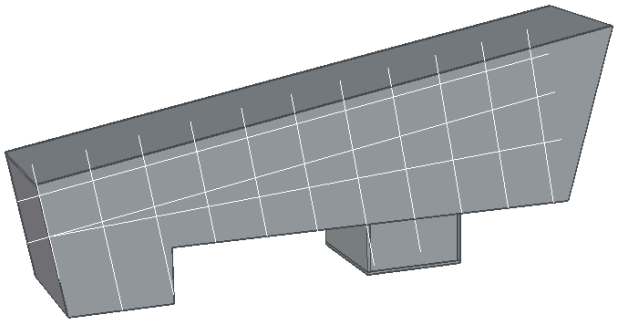
1 Use BIMGRID command to create a Grid on the face of the wall. Choose the basic dimensions in the Command line. Changing the **U-line** and **V-line** panel options allows you to specify the desired size and number of panels on the surface.

***Прим.:*** To draw on the face, make sure you first highlight the face and then click. Otherwise, the Grid will be created in the XY-plane.

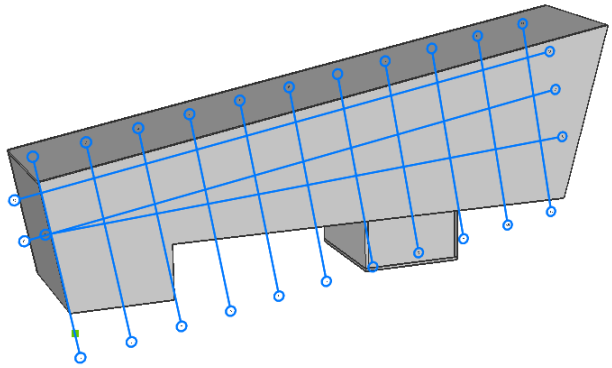


2 Use BEDIT or REFEDIT to edit the Grid.

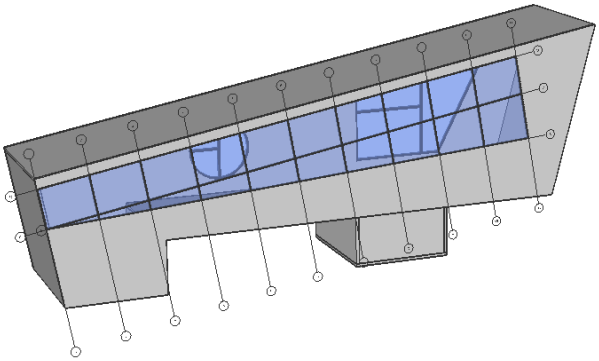
3 Use the Manipulator to change the position of the gridlines on the wall.



4 Select the Grid and select **Create Window** in the Quad.



5 The window is generated based on the gridlines.



***Прим.:*** You can edit the parameters in the **Properties** panel. Type a new value in the property field and press Enter. The window will automatically update itself.

| Parameters | |
|---|---|
| FrameDepth | 60 mm |
| GlazingDepth | 30 mm |
| GlazingOffset | 30 mm |
| PlacementDepth | 50 mm |

### 20.3.2.5 Creating window openings

**BIM Opening Create** creates an opening without window geometry. It uses the same command as BIMWINDOWCREATE. The command allows to create an opening at the location of the selected base profile. The base profile can be a boundary or a closed polyline.



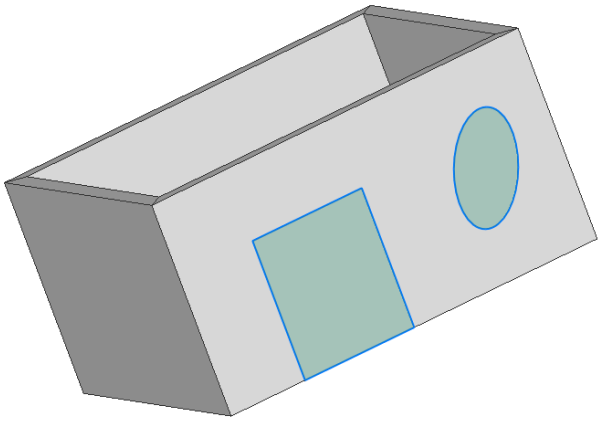
For more information about this command, visit the Command Reference article BIMWINDOWCREATE.

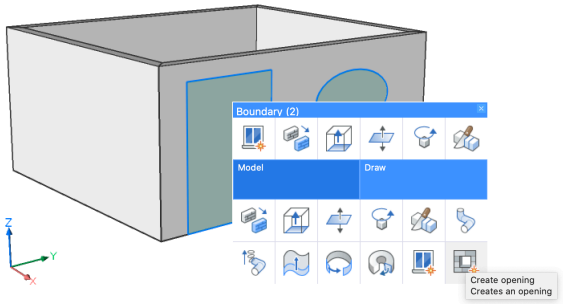
1 Draw one or more closed polylines on the face of a wall.

2 While boundary detection is on, hover over the polygon.

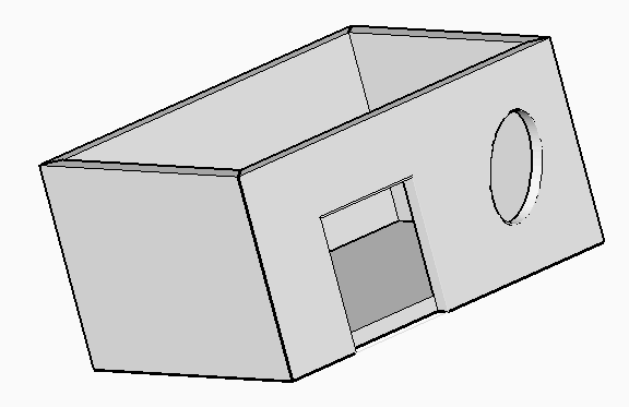
3 Or select the closed polyline.

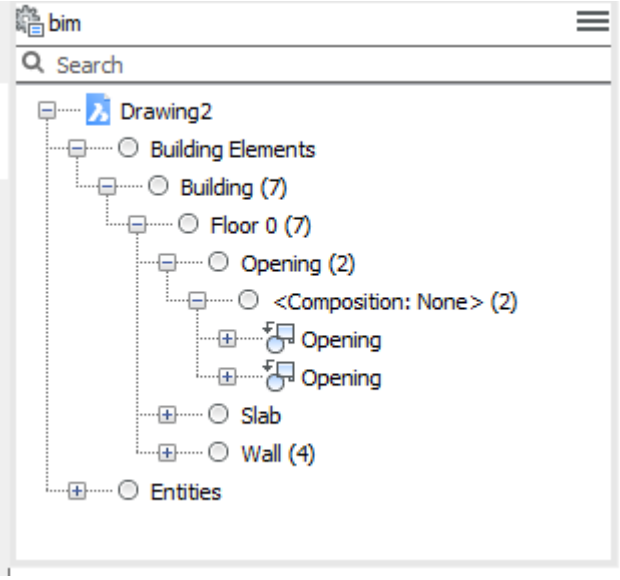
4 Select the **Create Opening** tool from the Quad.



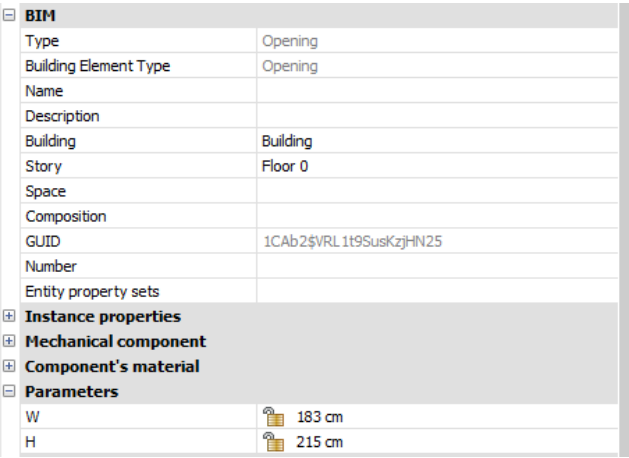



5 The opening is created based on the closed contour. In the **Structure Browser**, the opening is added as a block reference. The block reference is classified as an opening.






If you select the opening from the structure browser, you will see a set of properties related to this opening element in the **Properties** panel.

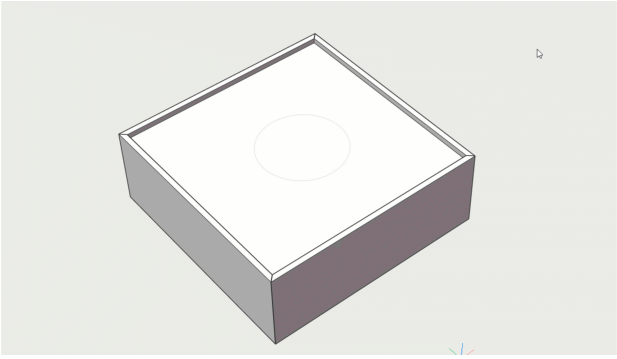



***Прим.:*** If your closed contour has a rectangle shape, the opening element will include Width and Height parameters.

***Прим.:*** Opening elements contain subtractor solids. To display the subtractor solid, open the **Layers** panel and select the **BIM_Subtract** layer.

***Прим.:*** The associative tags can be generated using the BIMTAG command for the opening element in a sectioned drawing.

The following animated gif demonstrates how to create an opening based on a closed contour.



### 20.3.3 The library panel

#### 20.3.3.1 Commands

LIBRARYPANELOPEN, LIBRARYPANELCLOSE, CREATELIBRARYBLOCK, -CREATELIBRARYBLOCK

#### 20.3.3.2 About

The **Library** panel is a quick way to keep your BIM blocks at your fingertips or access predefined ones. You can add any CAD model to the **Library**. You can also fully customize the name and category to make it easier to find. You can then insert a library block in your drawing by dragging and dropping. Keep your library clean by deleting the items you no longer need.

#### 20.3.3.3 Content of the Library panel

You can control the content of the **Library** panel in the **Option** menu.

***Прим.:*** When the COMPONENTSPATH system variable is empty (all paths deleted), the default **C:\ProgramData\Bricsys\Components** path is added automatically when you close and reopen the **Settings** dialog box.

#### 20.3.3.4 Creating a library block

1 Open or create the drawing which contains the geometry for the new block.

2 Do one of the following:

- Click the **Home** icon ( ).

  This allows you to select the destination category; or you can create a new category.
- Click the **Category** thumbnail.

  The new item will be created in the selected category or in a subcategory of the selected category.
- Launch the CREATELIBRARYBLOCK command.
- The **Add block to library** dialog box displays.
- Select the entities or choose the **Entire drawing** option.

- Type a name for the block.
- (**Option**) Type a new category name or choose a category from the list.
- (**Option**) Click **Show Options** and set the block creation options.
- Click the **Create** button.

### 20.3.3.5 Managing library blocks

1 In the **Grid view** display, hover over the block icon.

2 Click the **Menu** icon (...).




3 (**Option**) Click the **File Open** icon (1) to open the block file.

Edit the file, then save the changes.

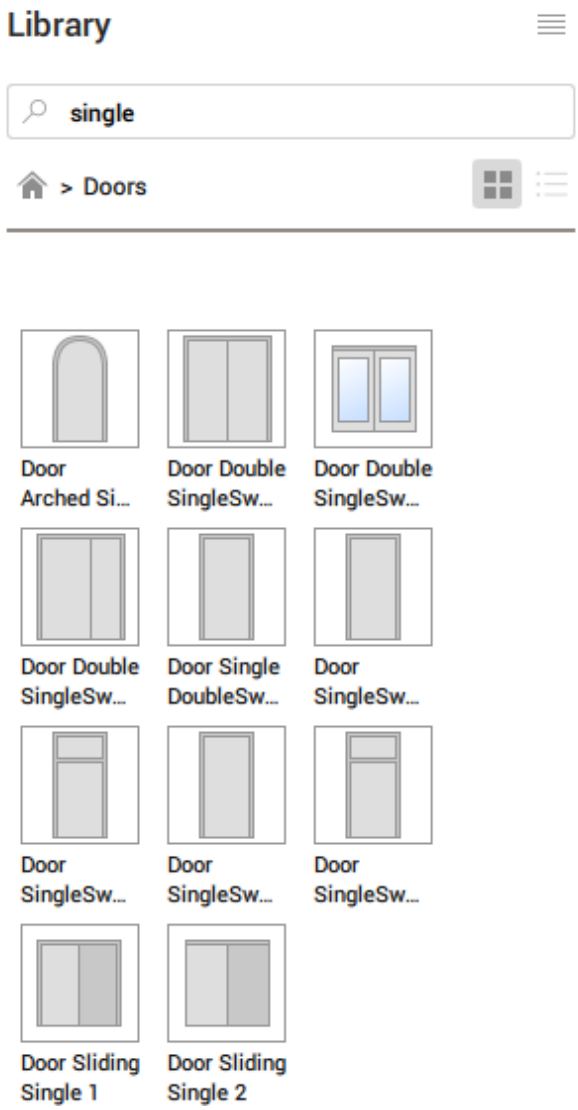
4 (**Option**) Click the **Delete** icon (2) to delete the block file.

Click **Yes** to confirm.

### 20.3.3.6 Searching the block libraries

Type a search string in the search field.

Only matching library blocks in the current category display.

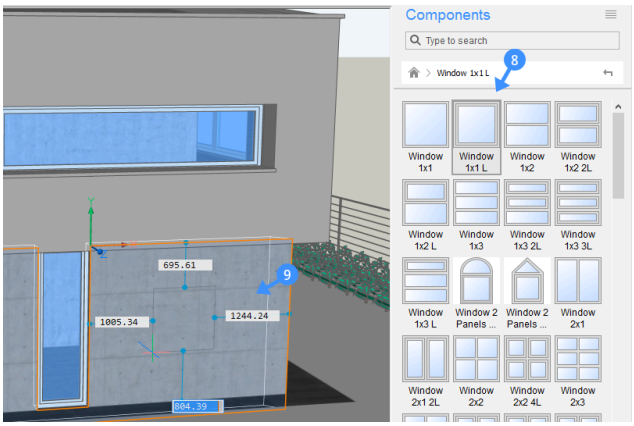



### 20.3.3.7 Inserting a BIM window into your 3D model

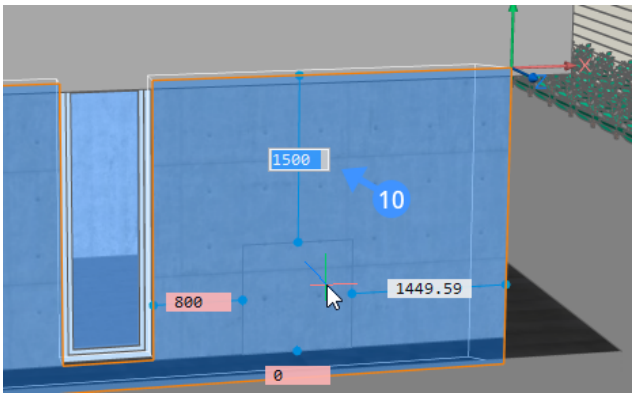
1 Click the **Windows** category tile in the **Library** panel.

2 Click a window (8).

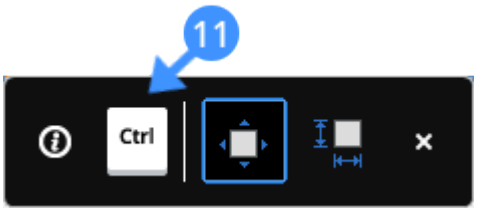
3 Drag and drop the window onto the wall (9).



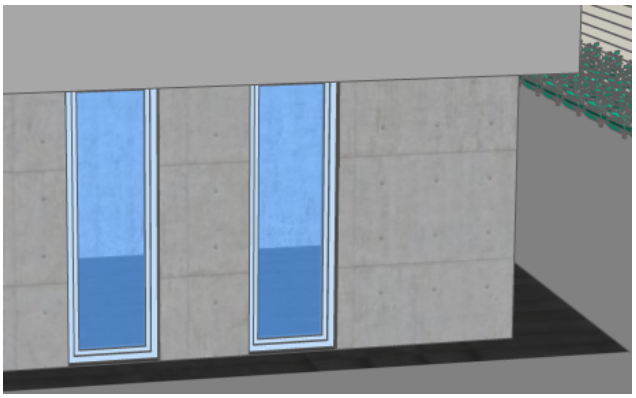

4 If Dynamic dimensions (**DYN**) is switched on, set the window location by entering the dimensions into the DYN fields (9). Press the Tab key to cycle through the DYN fields (10). A DYN field turns red to indicate the distance is locked.



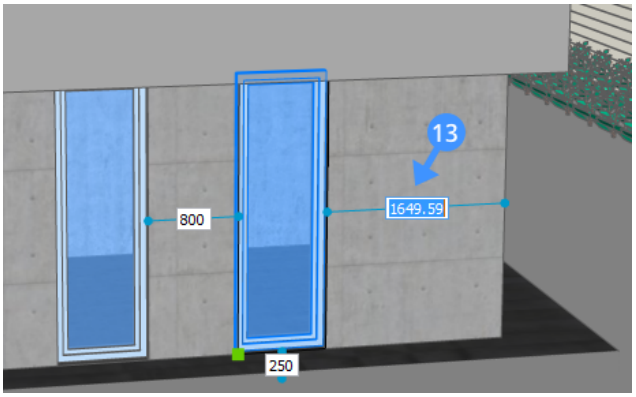

5 To edit the dimensions of the window press Ctrl (11) once and then use the Tab-key to cycle through the DYN fields (12).




6 Press Enter. The window is created. This same technique can be used for most of the blocks in the **Library**.



7 (**Optional**) Once you have a window you can adjust the position by selecting the window and then clicking on the dimension lines and editing the dimensions (13).




8 (**Optional**) To adjust the dimensions of the window after the window has been created, go to the **Properties** panel. Scroll down to the **Parameters** section and change the parameters as required. All windows and doors in the Library are parametric.

| Parameters | |
|---|---|
| Name | Window_1x1_L:9 |
| File | |
| Physical material | |
| W | 800 mm |
| H | 250 cm |
| Rebate | 0 mm |
| PlacementDepth | 90 mm |
| FrameThickness | 60 mm |
| FrameDepth | 50 mm |
| LeafThickness | 45 mm |
| LeafDepth | 60 mm |
| PaneOffset | 20 mm |
| LeafOffset | 20 mm |

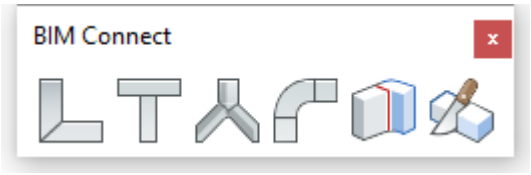
### 20.3.4 Connecting solids

#### 20.3.4.1 Commands

LCONNECT, TCONNECT, BIMSTRUCTURALCONNECT

#### 20.3.4.2 About

Walls, profiles, plies, etc. are all created as native 3D solids. So, any of the 3D Solids Editing commands can be used to modify them. Use **BIM Connect** commands to connect two solids in different ways.




The LCONNECT command creates an L-connection (bisector or parallel) between two solids.

The TCONNECT command creates a T-connection between two solids.

The BIMSTRUCTURALCONNECT command connects structural profile solids such as beams and columns.

The BIMFLOWCONNECT command creates a connection between flow segments, such as pipes or HVAC ducts.

The procedures below use the Quad cursor menu.

#### 20.3.4.3 Procedure: creating L-connections

1 Make sure SELECTIONMODES system variable = 0.

2 Type **LConnect** in the Command line or select it via the Quad.

3 Select two solids or faces to connect.

4 The two selected solids are connected: the default connection type is L-bisector.

5 If HOTKEYASSISTANT system variable is ON (HKA is switched on), the Hotkey Assistant widget displays.



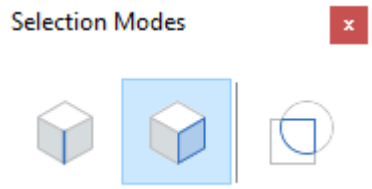

Use the CTRL key to cycle between the connection types **or**
You are prompted: Select base solid [Switch] <Accept>. Type **A**to use the selected connection, type **S** to switch to the desired connection type.
The 3D model updates dynamically when cycling the L-connection types.

***Прим.:*** L-connections can also be created between non-vertical solids such as roof slabs.

### 20.3.4.4 Procedure: Creating T-connections

The following procedure allows connecting minor faces of a solid to the major faces of other solids, regardless of what the orientation of the faces is. As a result, you can use TCONNECT command to T-connect solids.

1 Make sure the **Select Faces** option of the SELECTIONMODES system variable is selected.

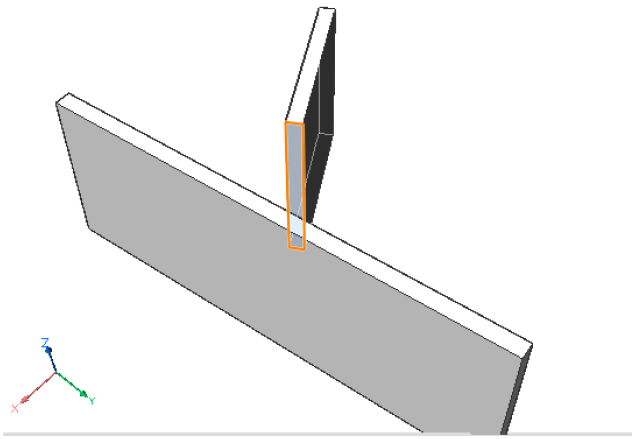



2 Type TCONNECT in the Command line or select it via the Quad.

3 Move the cursor over the minor face of the first solid at the end that needs to be connected.

4 Press Tab to highlight obscured faces.

Click when the face highlights.



5 (**Optional**) Repeat the previous step to select more faces, then press Enter.

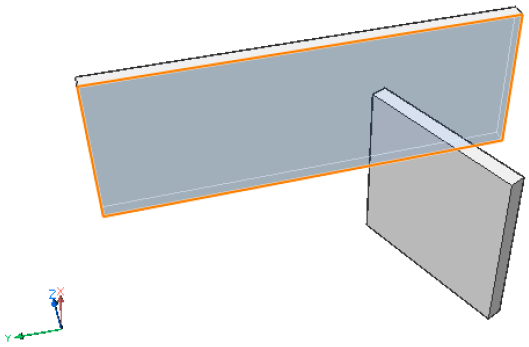
***Совет:*** The SELECTALIGNEDFACES command in the **Select** command group in the Quad selects all faces in the 3D model that are in the same plane (coplanar) with a selected face.

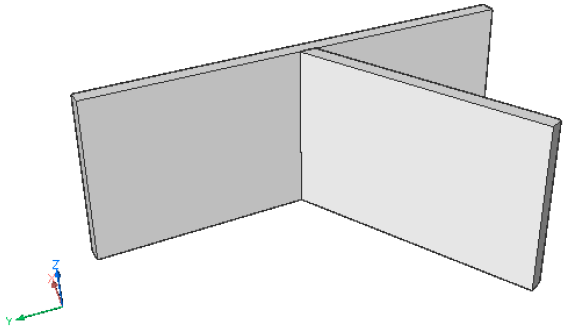
6 You are prompted: Select entities to connect to or [Connect to nearest/Disconnect/selection options (?)] <Connect to nearest>:

7 Do one of the following:

- Press Enter to connect to the nearest face(s).
- Select one or more faces.
- If necessary, press Tab to highlight an obscured face; click when the face is highlighted.

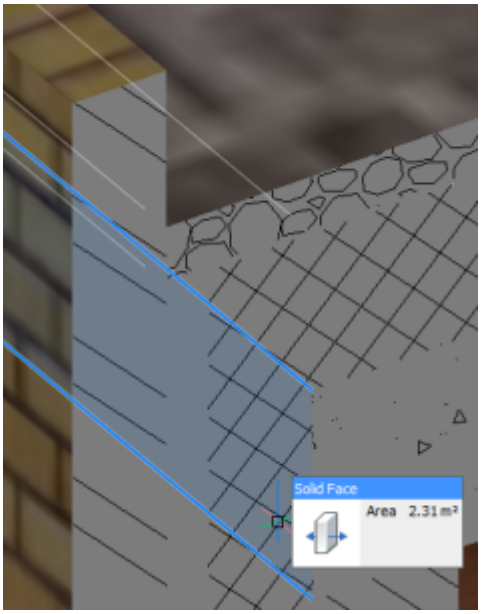
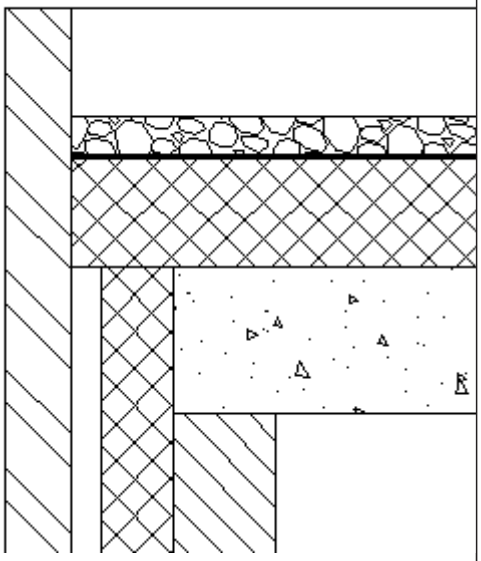
  Right click to stop selecting faces.





***Совет:*** Hover over the minor face of the solid and choose **Connect with Nearest in the Model** command group of the Quad.

**Creating T-connections ply-by-ply**

| | |
|---|---|
|  |  |
| 3DModel | Calculated Section |

To control the connection of composition plies between two building elements, you can **Push/Pull** a ply in a clipped BimSection (if Show Composition is ON). For more in-depth information, visit the BIM Ply Editing article.

To create a T-connection ply-by-ply in a section, do the following.

1 Place the cursor over the face of the ply you want to connect. If necessary, repeatedly press Tab to select an obscured face.

2 When the ply face highlights, choose **Push/Pull** in the Quad.

3 Specify to which point the selected ply should reach by moving the cursor or typing in the distance.

4 Right-click or press Enter to confirm.

5 The ply of the first solid is connected to the ply of the second solid. The connected ply is subtracted from the second solid.

6 Optionally, use the BIMPROPAGATEPLANAR command to apply the connection to similar locations in the model.

### 20.3.4.5 Procedure: Setting the distance between parallel solids

1 Select the solid of which the position must be fixed.

2 Select the solid which will be moved when defining the new distance.

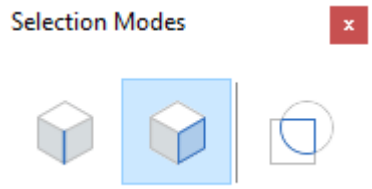
3 The nearest distance between the solids displays.

4 Type a value in the distance field and press Enter.

### 20.3.4.6 Procedure: Creating a copy of a solid

**Creating a parallel copy of a solid**

1 Make sure the **Select Faces** option of the SELECTIONMODES system variable is selected.




2 Move the cursor over a face of the solid to be copied.

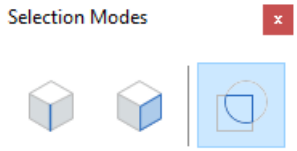
3 When the face highlights, choose **BimCopy** in the **Model** command group of the Quad.

A copy of the solid displays dynamically. The ruler indicates the current distance.

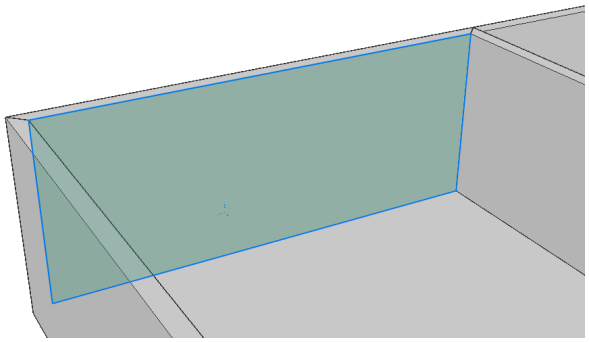
4 Type a value in the dynamic entry field and press Enter.

**Creating a partial copy of a solid**

1 Make sure the **Enable boundary detection** option of the SELECTIONMODES system variable is selected.



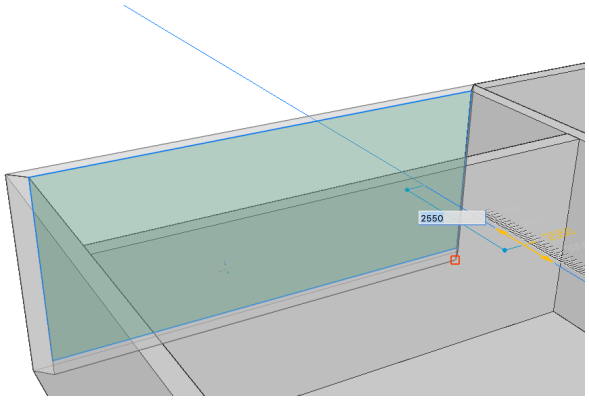

2 Highlight the part of the boundary for the part of the solid you want to copy.



3 When the face highlights, choose **BimCopy** in the **BIM** command group of the Quad.

A partial copy of the solid displays dynamically. The ruler indicates the current distance.

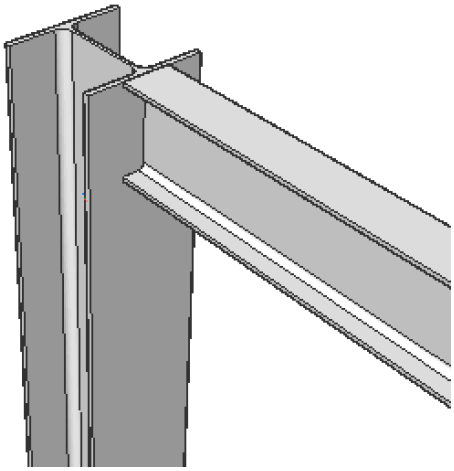
4 Type a value in the dynamic entry field and press Enter.




### 20.3.4.7 Procedure: disconnecting two connected solids

In some cases, you need to disconnect 2 solids of a node to do some further direct modeling.

You select two solids and it breaks the connection by separating the two solids physically and removes all end clippings at the side of the connection. As such, these solids can direct modeled upon again.



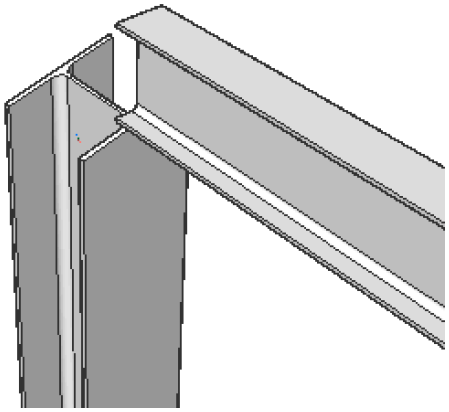
1 Type LCONNECT in the Command line.

2 Select the two solids.

3 The connection is automatically made, press CTRL until the disconnect icon is highlighted.




4 Press Enter.



***Прим.:*** You can use Propagate to disconnect similar nodes.

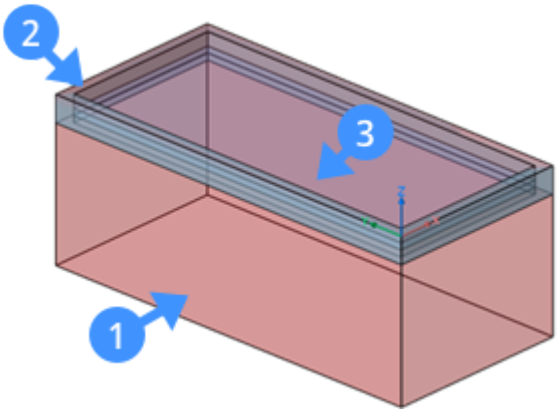
### 20.3.5 Creating a custom window

#### 20.3.5.1 Creating a custom window

In BricsCAD, you can create any geometry or component from scratch. Once you have your geometry in 3D you can insert and use it in your BIM model. You can make changes or add more details to your window at any time.

This process is explained in 4 parts to make the steps of creating a 3D window geometry straightforward.

- Create the file
- Create the subtractor solid (1)
- Create the window frame (2)
- Create the (Glass) panes (3)

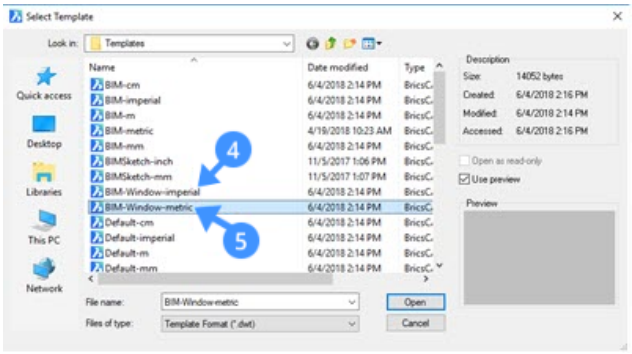



**Step 1: Creating the file**

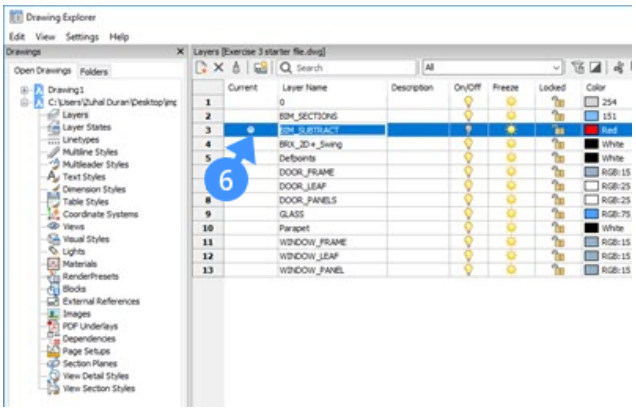
1 Click **New** in the **File** menu.

2 Select one of the pre-defined templates to create a simple window specifying the unit system of your drawing.

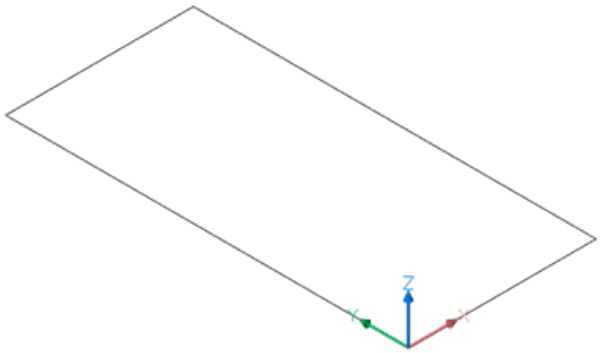
***Прим.:*** The pre-defined templates contain the layers needed to create the geometry of the window or door. These .dwt file format templates are **BIM-Window-imperial** (4) and **BIM-Window-metric** (5) which assure the required layers and properties are displayed in your drawing.




3 Set the **BIM_Subtract** layer current (6) either using the drawing explorer or layer field in the **Properties** panel or **Layers** panel on the right-hand side.




4 Create a closed polyline (e.g. a rectangle) that represents the form of the opening for the insert.




5 Save the file in any location. The library of predefined windows and doors is located in the support folder.

6 By default this is: C:\Users<user_name>\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\Support

\Bim\Components\Windows This path can also easily be found by typing SUPPORTFOLDER in the Command line.

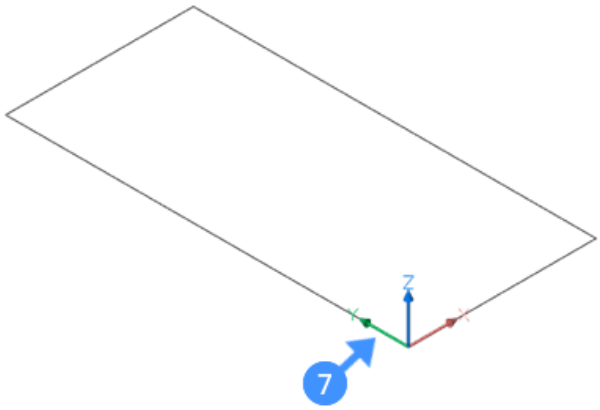
**Step 2: Creating the subtractor solid**

The subtractor solid will be used to create an opening.

***Прим.:*** **BIM Subtract** layer must be a current layer.

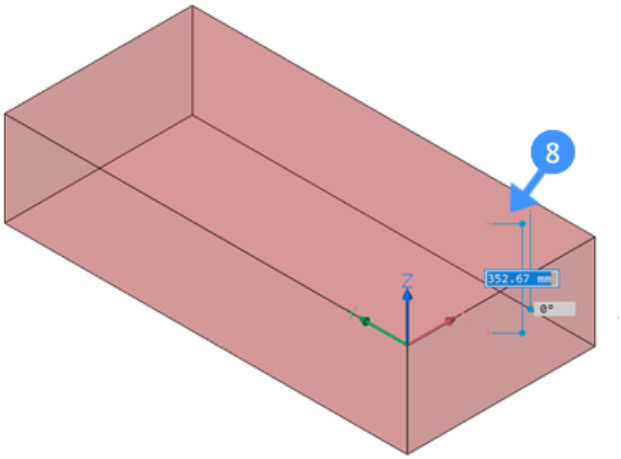
1 Use the outline of the solid which you have previously created ( e.g. a rectangle).

1 The outline lies in the first quadrant of the XY plane of the World Coordinate System (WCS).




The lower left corner of the rectangle lies at the origin (0,0,0) of the WCS (7).

2 Select the **Extrude** tool and move the cursor downward (8) to extrude your solid outline in the negative Z direction.




3 Do one of the following to finish the subtractor solid creation.

- Click a point.
- Type a value in the dynamic dimension fields, then press Enter.

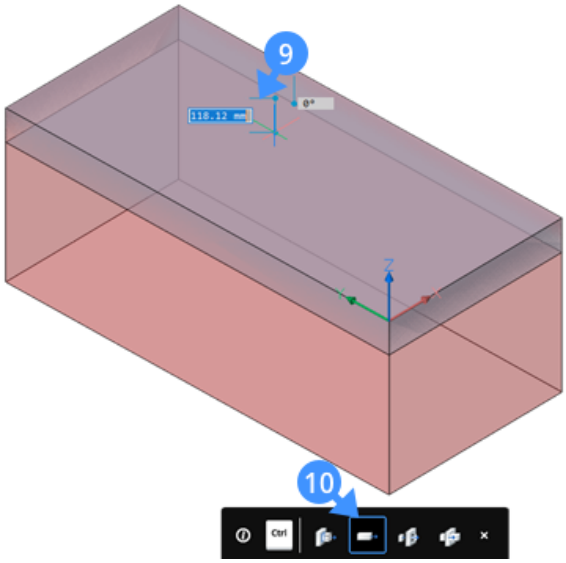
**Step 3: Creating the fixed frame**

The outline of the fixed frame coincides or is parallel (in case of a rebate) with the outline of the subtractor solid.

1 Set the **Window_Frame** layer current.

2 Highlight the top face of the subtractor solid.

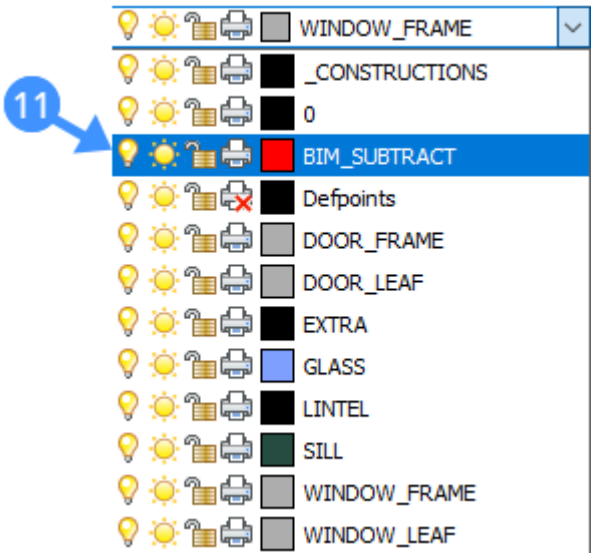
3 Next, extrude the top face of the subtractor solid in the negative Z direction (9) using the **Extrude** tool 'Create' (10) option by pressing the CTRL key once.



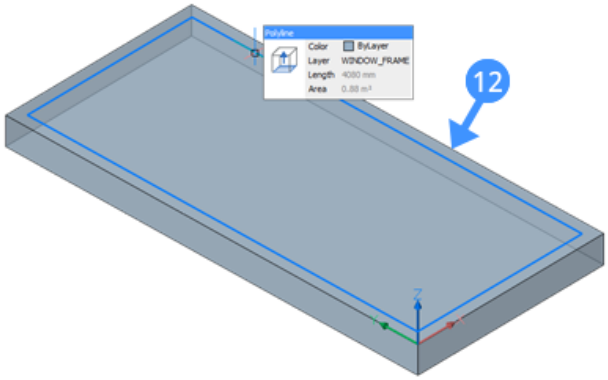

4 Type a value in the dynamic dimension field to specify the height of the fixed frame.

***Прим.:*** The height of the extrusion will be the thickness of the fixed frame.

5 Turn off **BIM_SUBTRACT** layer by clicking the light bulb (11) next to the layer name.

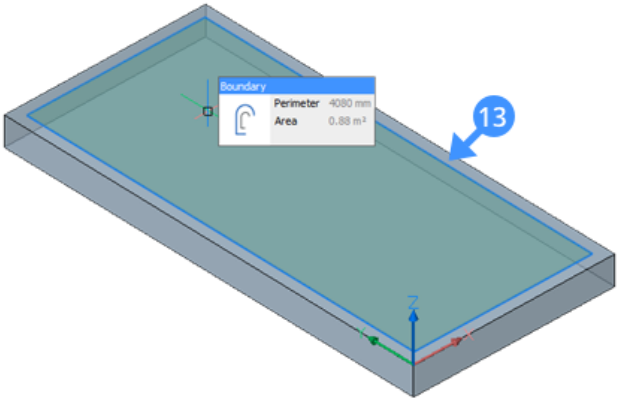



6 Use the **Offset** tool to create a parallel offset of the outline of the fixed frame (12).

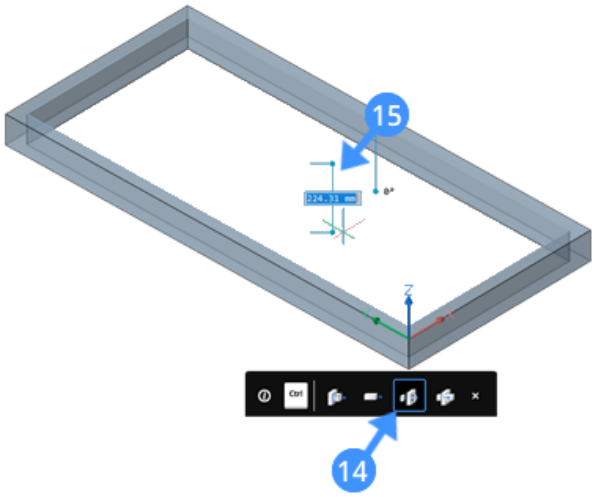



***Прим.:*** The distance of the offset equals the width of the frame.

7 Hover over the inside boundary of the parallel offset (13) and select **Extrude** tool from the Quad.




8 When the extrude mode is on the **Subtract** option (14) move your cursor downward (15) to create an opening in the fixed frame.




9 Do one of the following to finish the creation of the opening in the fixed frame.

- Click a point.

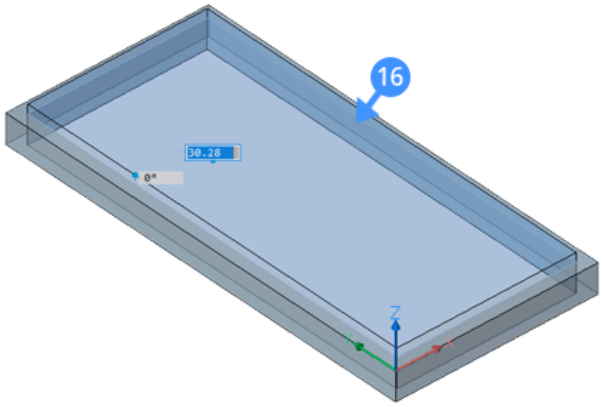
- Type a value in the dynamic dimension fields, then press Enter.

**Step 4: Creating the glass panes**

1 Set the **Glass** layer current.

2 Use the DMEXTRUDE command to extrude the inside boundary of the window frame.

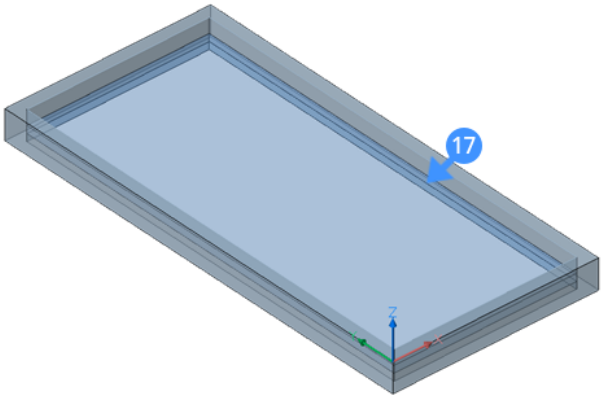
***Прим.:*** The height of the extrusion equals the thickness of the glass pane. The following illustration shows the inside boundary of the window frame after using DMExtrude to create a window glass pane.




Note that to create a glass pane, the create option should be selected from the Tooltip of **DMExtrude**.

3 Move the glass pane solid down in the negative Z-direction if the outside face of the fixed frame does not coincide with the outside face of the solid (wall) the insert will be placed in.

4 The following illustrations show the glass pane solid after moving it (17) down.




5 Optionally, before saving the drawing, classify the BIM as a window element.

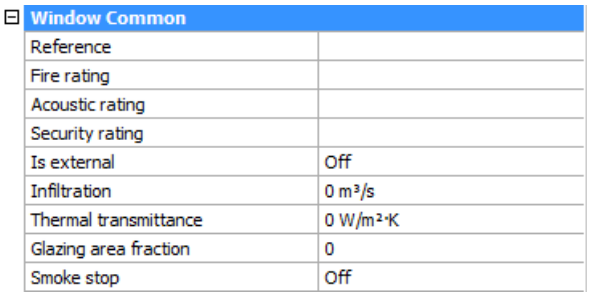
6 To do so, follow these steps:

- Type BIMCLASSIFY in the Command line.
- Press 'I' for window and press enter to accept it.
- Next, press 'D' for Drawing and press the Enter key until the command is executed.

The drawing is classified as a window. When you insert this block in your drawing, it will automatically

be classified as a window.

7 Optionally, when no selection is active, you see a set of BIM properties on the right-hand side of your screen. Here you can assign information about the thermal transmittance coefficient of the window materials, fire rating according to the national fire safety classification, and so on. The following illustration shows the BIM properties of the window in the properties panel.



| Window Common | |
|---|---|
| Reference | |
| Fire rating | |
| Acoustic rating | |
| Security rating | |
| Is external | Off |
| Infiltration | 0 m³/s |
| Thermal transmittance | 0 W/m²·K |
| Glazing area fraction | 0 |
| Smoke stop | Off |

8 Save the drawing.

**Adding 2D entities**

Optionally, windows and doors drawings can contain 2D symbols. These 2D symbols will either replace the actual **3D Solid** section of the window or door or be added to the **3D Solid** section, depending on their layer:

- Symbols on a layer with prefix BRX_2D_will be used to replace the geometry resulting from the actual section.
- Symbols on a layer with prefix BRX_2D+_will be added to the section result. Only those 2D entities in a plane parallel to the section plane are.

### 20.3.6 Creating roofs

#### 20.3.6.1 The roof creation workflow consists of the following steps:

1 Creating the roof footprint.

2 Extrude the footprint and create a hip roof (DMEXTRUDE command).

3 Adjust the slope of the roof faces if necessary (**Manipulator** widget).

4 Defining the thickness of the roof solid (**Body/Shell** option of the SOLIDEDIT command).

5 Create additional roof components.

#### 20.3.6.2 Creating the roof footprint

You can create the roof footprint:

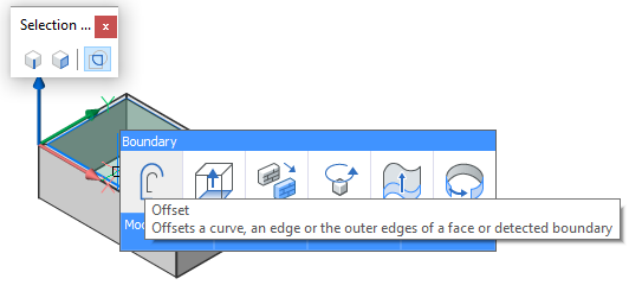
- From the exterior walls boundary.
- Creating a closed polyline by clicking points or other 2D procedures.

**To create the roof outline from the exterior walls boundary**

1 Isolate the exterior walls. Isolating entities is easily done using the **Structure** Panel. See also the ISOLATEOBJECTS command.

2 Set the coordinate system to the top face of the exterior walls:
   - Type UCS and press Enter.
   - You are prompted: Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/Move/World] <World>:
   - Snap to a top corner of an exterior wall, then press Enter.

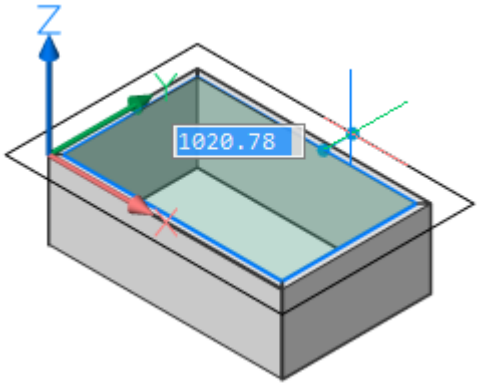
3 Set the **Enable boundary detection** ON in the **Selection Modes** toolbar or include the **Select detected boundaries** into SELECTIONMODES system variable.




4 Move the cursor to avoid area inside the exterior walls.

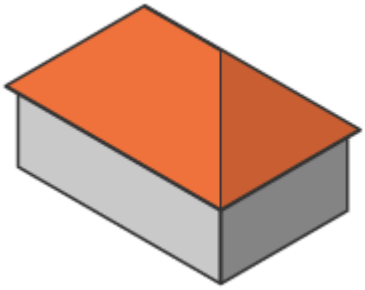
5 The inside boundary highlights.

6 Choose **Offset** in the **Draw** tab of the Quad.



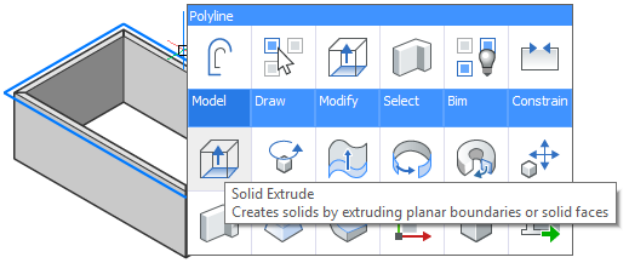

7 Do one of the following:
   - Type a value in the dynamic distance field.
   - Click to accept the value.

### 20.3.6.3 Creating a hip roof

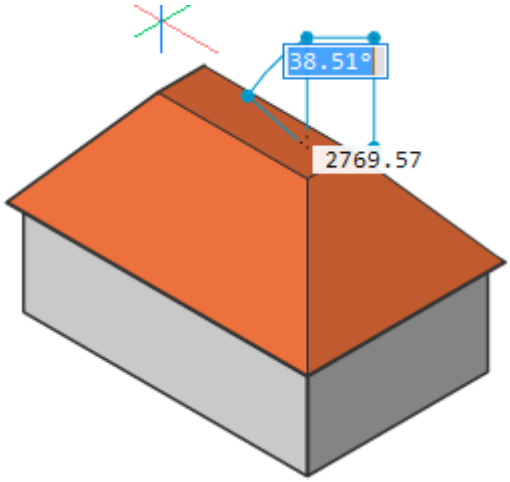


1 Create the roof footprint.

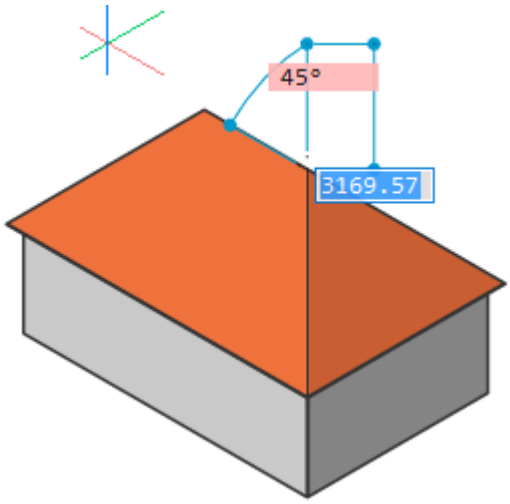
2 Move the cursor over the roof outline and choose **Extrude** in the **Model** tab of the Quad.




3 Move the cursor vertically, then hit the Tab key.




4 Type a value in the **Taper Angle** field and hit the Tab key.




5 Do one of the following:

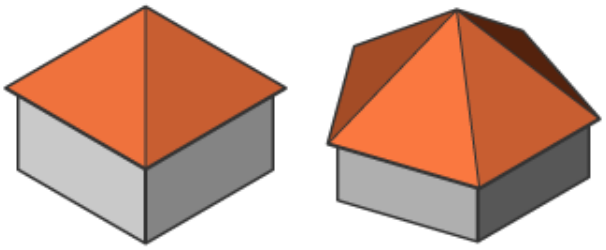
- Hit the Tab key and type another value in the **Taper Angle** field.

  The **Taper Angle field** is the complement of the inclination angle of the roof.

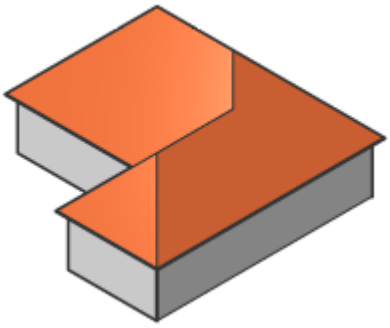
- Press Enter to create the roof.

6 Depending on the shape of the roof outline, the result is:

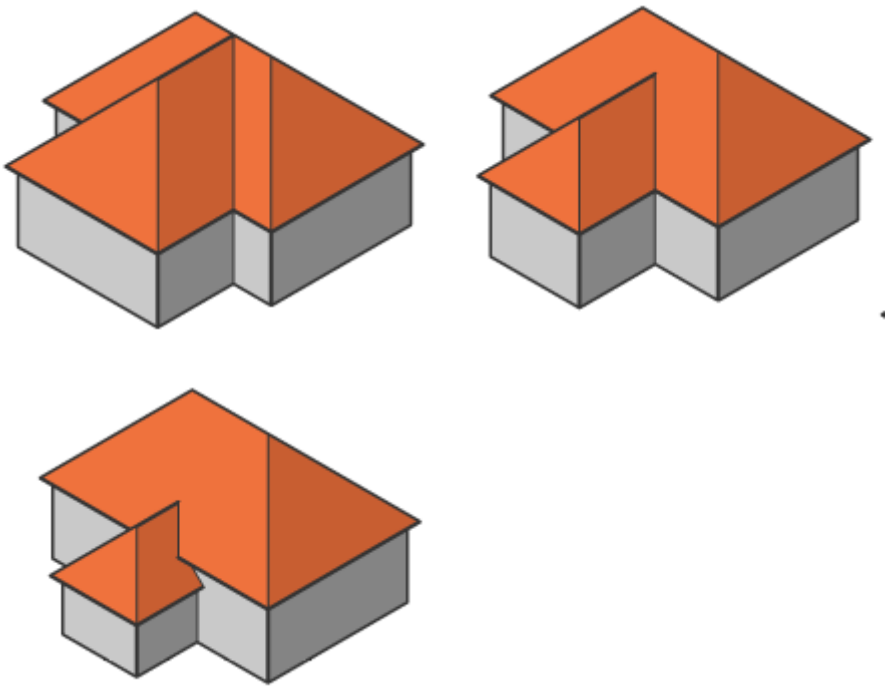
- Pyramid roof:



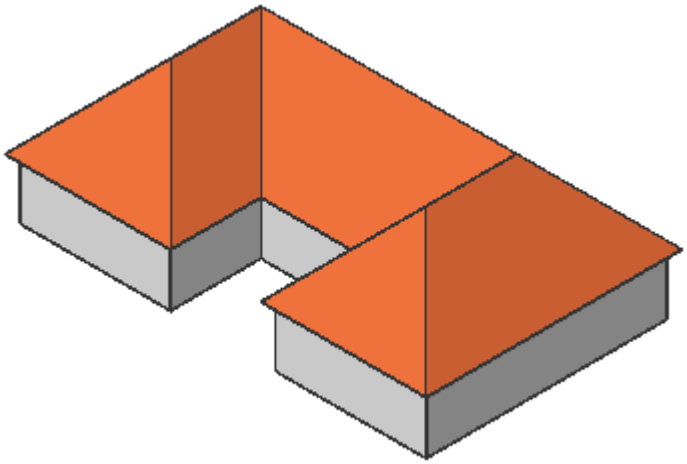
- L-shaped hip roof:

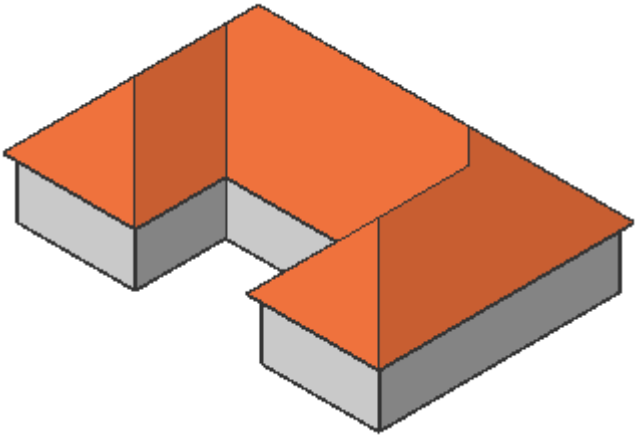


- T-shaped hip roof:

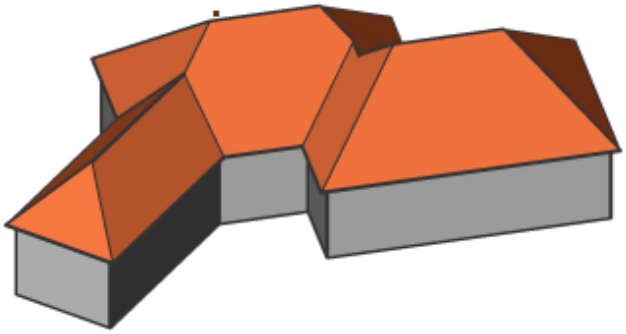


- U-shaped hip roof:





- Complex hip roof:



7 To define the thickness of the roof solid we will use the **Body/Shell** option of the SOLIDEDIT command.

Do one of the following:

a Choose **Shell** in the **Solid Edit | 3D Solid Editing** ribbon panel.
Or Choose **Shell** in the **3D Solid Editing** toolbar.

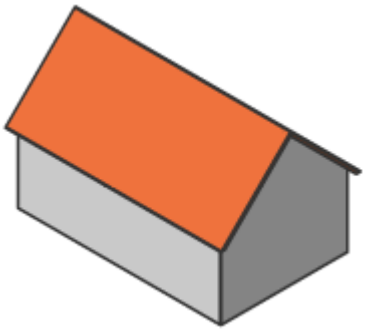
b You are prompted: Select 3D solid: Select the roof solid.

c You are prompted: Select face to remove or [Add/Undo/ALL]: Select the bottom face of the roof

solid.

d You are prompted: Enter the shell offset distance: Type a value to define the thickness of the roof solid.

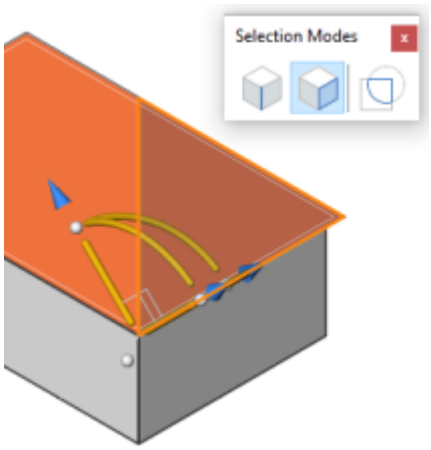
### 20.3.6.4 Creating a gable roof



1 Create a hip roof.

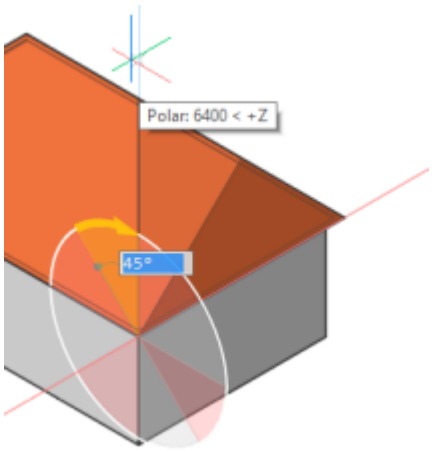
2 Make both end faces vertical:

- Move over the end face then hold down the left mouse button about half a second; the Manipulator widget displays (set the value of the MANIPULATOR system variable = 2).



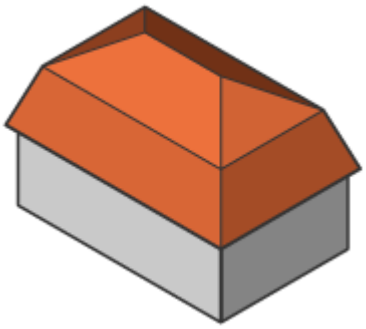

- Place the cursor over the vertical rotation arc. When the arc highlights, move the cursor.
- The selected face rotates dynamically.

  Click when the face is in a vertical position.




3 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

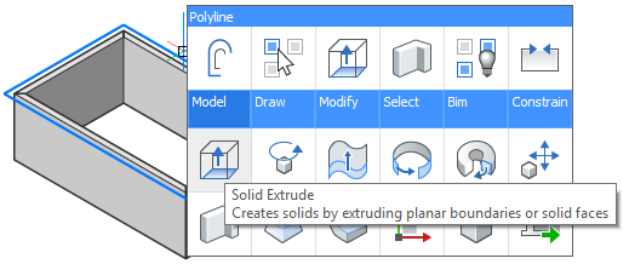
### 20.3.6.5 Creating a mansard roof



A mansard roof is composed of two solids, which we will create as two separate extrusions.

1 Create a roof footprint.

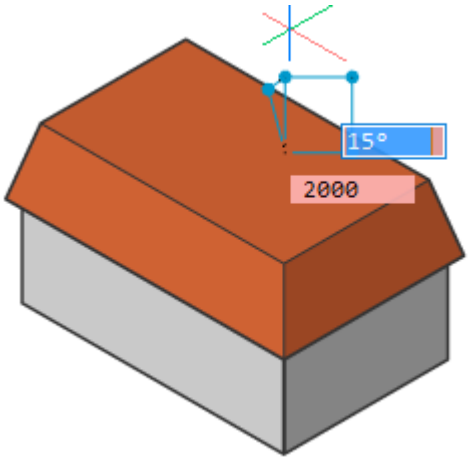
2 Move the cursor over the roof outline and choose to **Extrude** in the **Model** tab of the Quad.




3 Move the cursor vertically, then hit the Tab key.

4 Type a value in the **Taper Angle** field then hit the Tab key.

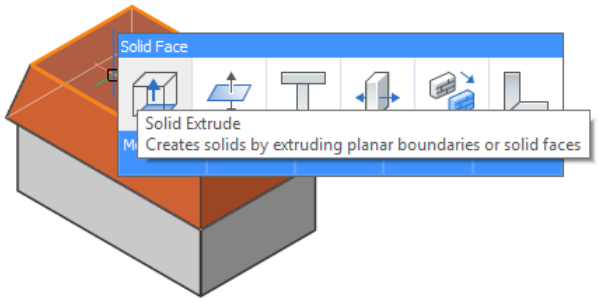
The **Taper Angle** field is the complement of the inclination angle of the roof.




5 Type a value in the **Extrusion Height** field, then do one of the following:
   - Hit the Tab key and type another value in the **Taper Angle** field.
   - Press Enter to create the first solid.

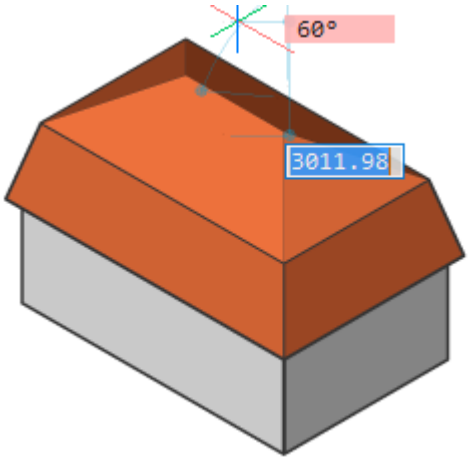
6 Optionally hit the Tab key to go back and forth between the entry fields and press Enter when you are satisfied with the result.

7 Move the cursor of the top face of the first roof solid and choose **Extrude** in the Quad.



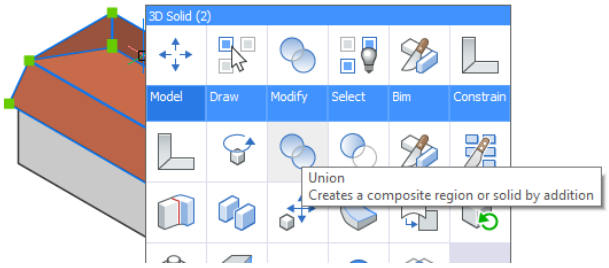

8 Move the cursor vertically, then hit the Tab key.

9 Type a value in the **Taper Angle** field then hit the Tab key.



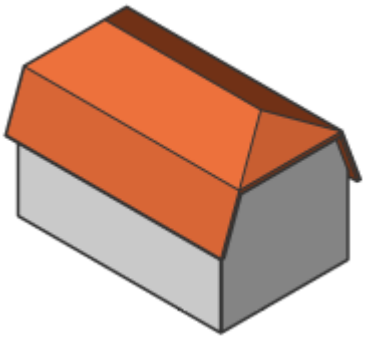
10 Do one of the following:
- Hit the Tab key and type another value in the **Taper Angle** field.
- Press Enter to create the first solid.

11 Select both solids and choose **Union** in the **Model** tab in the Quad.




12 Repeat step 7 of the Create a hip roof procedure.

### 20.3.6.6 Creating a jerkinhead roof



A jerkinhead roof is composed of two solids, which we will create as two separate extrusions.

1 Execute steps 1 to 6 of Creating a mansard roof procedure, to create the first solid.

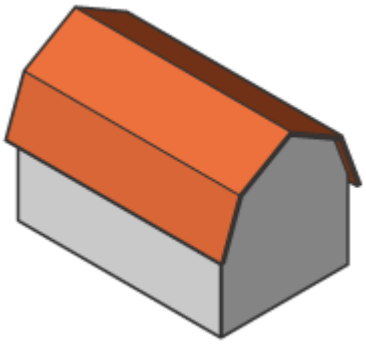
2 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

3 Execute steps 7 to 11 of the Creating a mansard roof procedure.

4 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

### 20.3.6.7 Creating a gambrel roof

A gambrel roof is composed of two solids, which we will create as two separate extrusions.



1 Execute steps 1 to 6 of Creating a mansard roof procedure, to create the first solid.

2 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

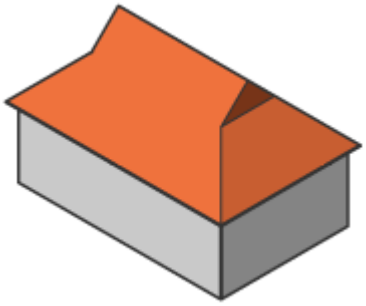
3 Execute steps 7 to 10 of Creating a mansard roof procedure to create the second solid.

4 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

5 Execute step 11 of Creating a mansard roof procedure.

6 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

### 20.3.6.8 Creating a dutch gable roof

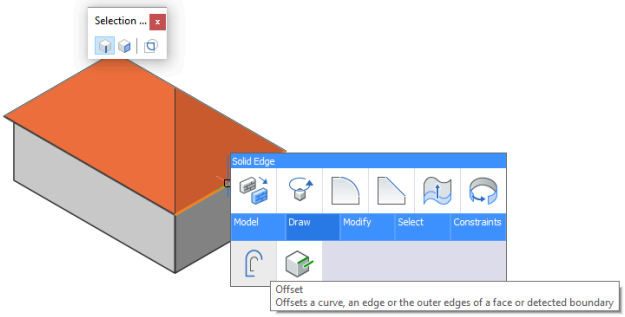


A Dutch gable roof is based on a hip roof.

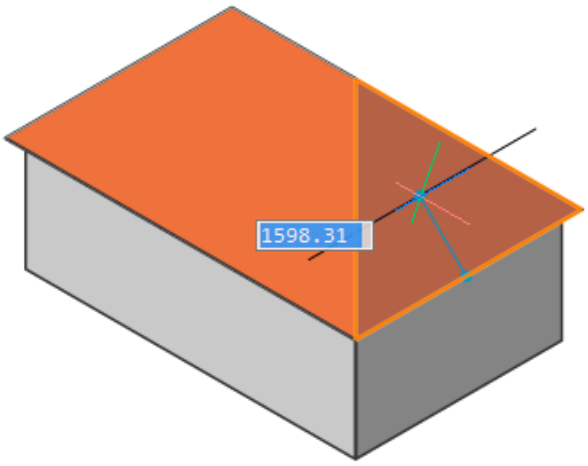
1 Create a hip roof.

2 Set the **Select Edges** option of the SELECTIONMODES system variable ON.

3 Move the cursor over the bottom edge of the roof face, then choose **Offset** in the **Draw** tab in the Quad.




4 Move the cursor and type a value in the dynamic distance field or click a point.




5 Do one of the following:
   - Choose **Imprint** in the **3D Solid Editing** toolbar.
   - Choose **Imprint** in the **Solid Edit | Solid Editing** ribbon panel.

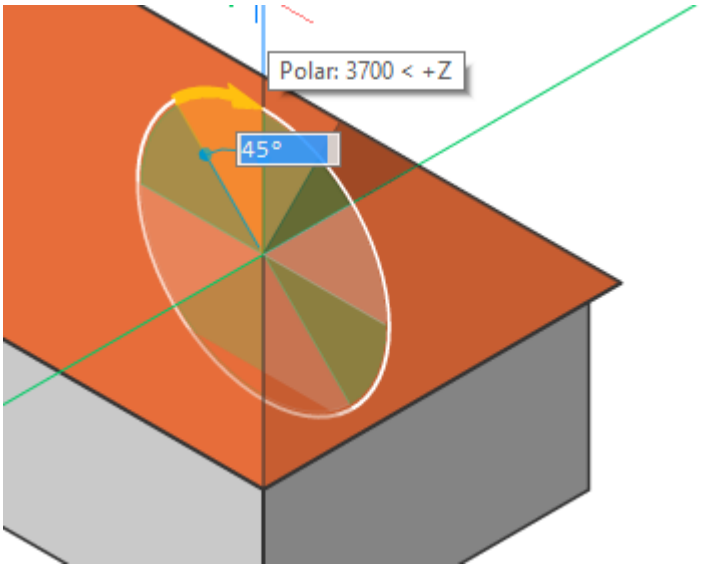
6 You are prompted: Select 3D solid, surface or region: Select the roof solid.

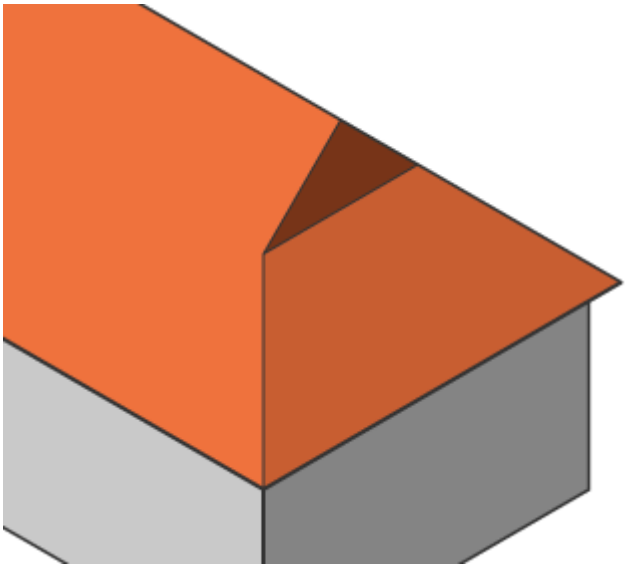
7 You are prompted: Select an entity to imprint: Select the offset roof edge.

8 You are prompted: Delete the source object[Yes/No] <No>: Choose Yes.

9 A triangular face is created.

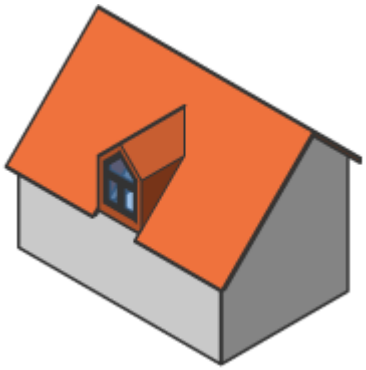
10 Execute step 2 of Creating a gable roof procedure to set the triangular face vertical.



11 Execute step 7 of the Create a hip roof procedure to set the thickness of the roof solid.

### 20.3.6.9 Creating dormers



The procedure to create a dormer consists of the following steps:

- Create a box solid at the bottom edge of the roof plane and connect the box to the roof.
- Define the position and dimensions of the dormer.
- Create a dormer roof.

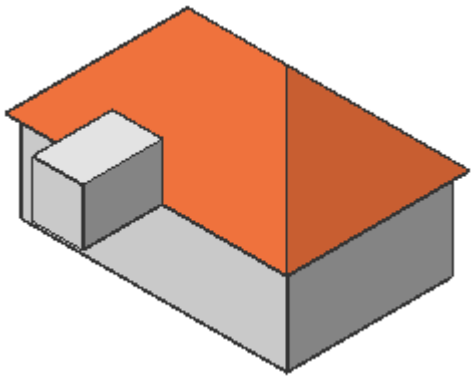
1 Right-click, then choose **Box** in the **Model** tab of the Quad.

You are prompted: Set corner of box or [Center] <0.00,0.00,0.00>:

2 Click a point at the bottom edge of the roof plan.

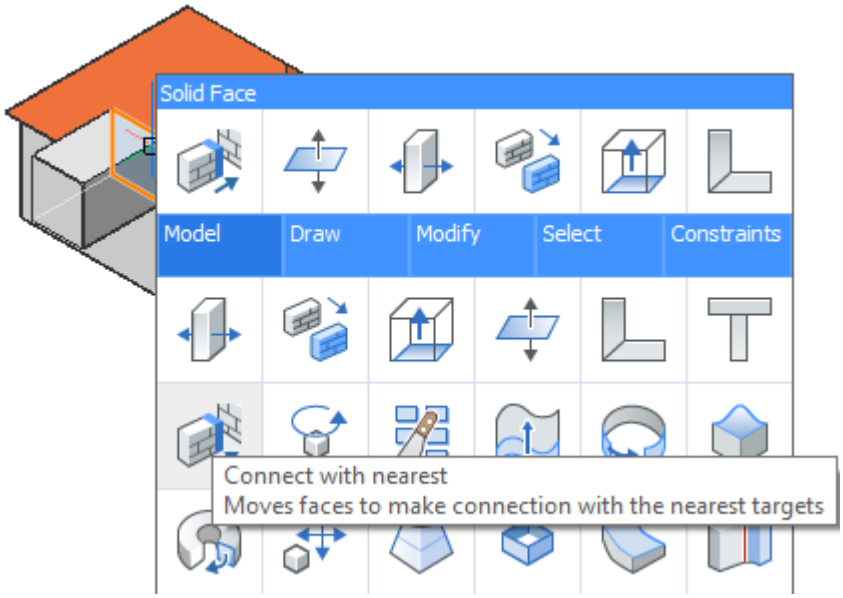
3 You are prompted: Set opposite corner or [Cube/Length of side]:

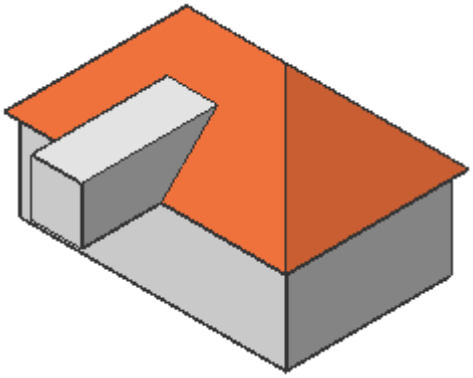
4 Click a point, then define the height of the cube: either click a point or type value.



5 Select the back face of the cube: move the cursor over the cube at the location of the back face, then hit the Tab key until the face highlights (Selection Modes: Select Faces = ON).

6 Choose Connect with nearest in the Model tab in the Quad.



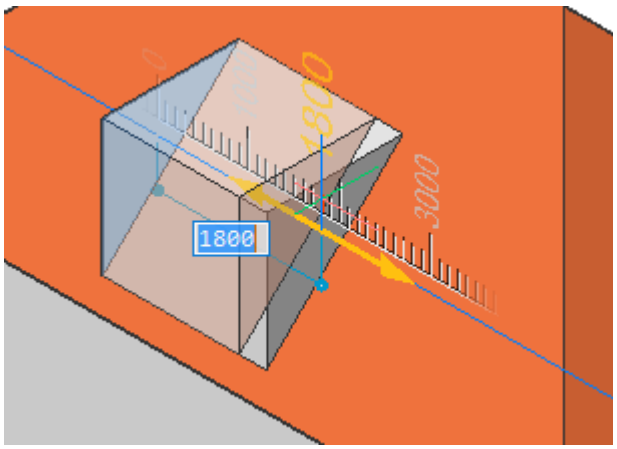



7 Select the front face of the box, choose **Push/Pull** in the **Model** tab in the Quad and move the face to the roof plane.

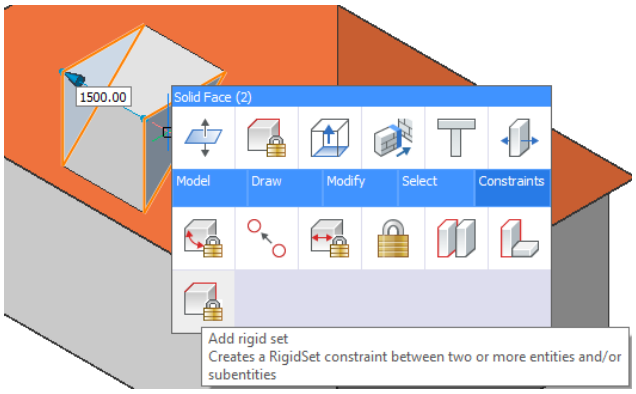
8 Use PUSHPULL command on the front, top and side face of the dormer solid to define the dimensions

and the position of the dormer on the roof:

- Select a side face, then hit the Tab key to define the width.




- Select both side faces, then choose to **Add rigid set** in the **Constraints** tab in the Quad to fix the width.



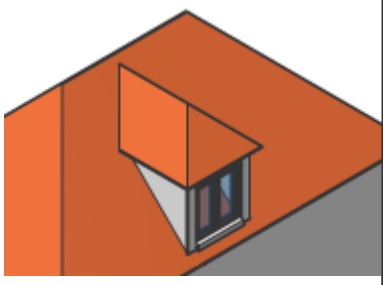
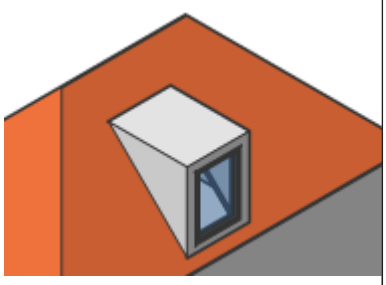
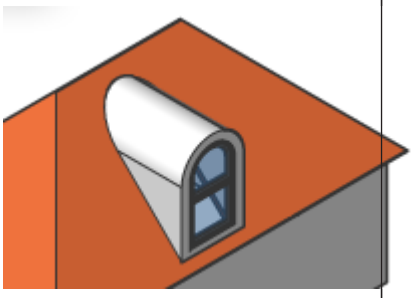
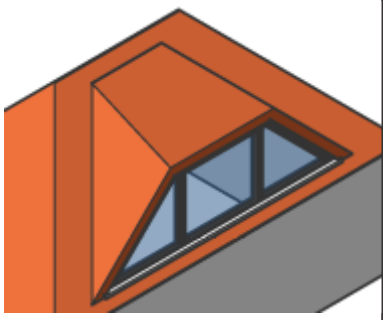
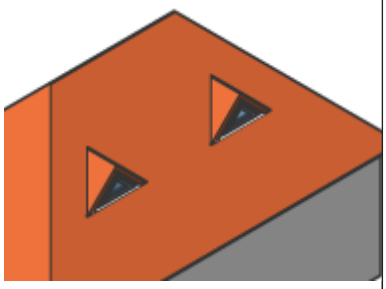
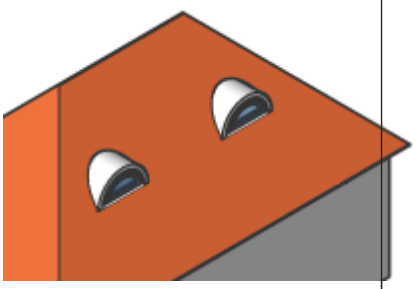

- Drag one of the side faces to position the dormer. Do one of the following:
  - Type a distance in the dynamic entry field.
  - Click to accept the value.
  - Move the cursor over a face that is parallel to the face being dragged and hit the Tab key. The distance field now shows the distance to this face.
- Drag the top face to define the height of the dormer.
- Drag the front face to set the distance from the roof eave.

Move the cursor over the exterior face of the wall beneath the dormer, then hit the Tab key. The distance field now shows the distance to this face.

9 Modify the shape of the dormer:

- Add a hip roof.
- Add a hip roof, then rotate the front face vertical to create a gable roof.
- Add a vault.
- Rotate top and side faces.

- Starting from a gable roof move the front face of the dormer backward until the front face is triangular.

| | | |
|---|---|---|
|  |  |  |
| Hip | Flat | Vault |
|  |  |  |
| Special | Triangle | Eyebrow |

### 20.3.7 Creating slabs

#### 20.3.7.1 Commands

DMEXTRUDE, POLYSOLID

***Прим.:***

***The DMEXTRUDE command allows to extrude a closed 2D entity such as polylines, circles, ellipses and extrude a detected boundary, enclosed by walls and/or 2D entities.***

***The POLYSOLID command allows to create pitched roofs.***

#### 20.3.7.2 Creating floor slabs between walls

Make sure that:

- The **Enable Boundary Detection** option of the SELECTIONMODES system variable is ON.
- The bottom faces of the surrounding walls are in the XY-plane of the current coordinate system.
- If necessary, move the origin of the coordinate system.
- Dynamic dimensions are ON.

1 Move the cursor inside the surrounding walls.

2 When the boundary highlights, choose **Extrude** in the **Model** command group in the Quad.

The slab is extruded dynamically.

3 Do one of the following:

- Type the thickness of the slab in the dynamic dimension field.
- Pick a point.

### 20.3.7.3 Creating a pitched roof slab

Make sure that:

- **Dynamic UCS** is ON.
- **Dynamic dimensions** (DYN) is ON; it is recommended to have the **Tracking dynamic dimensions** option of the DYNMODE system variable set.
- **Entity Snap Tracking** (STRACK) is ON.
- **Endpoint entity snap** is ON.

1 Launch the POLYSOLID command.

1 You are prompted: Start point or [Height/Width/Justification/Entity/Separate solids/Dynamic] <Entity>:

2 Move the cursor over the outside face of a wall which is perpendicular to the roof slab you want to create.

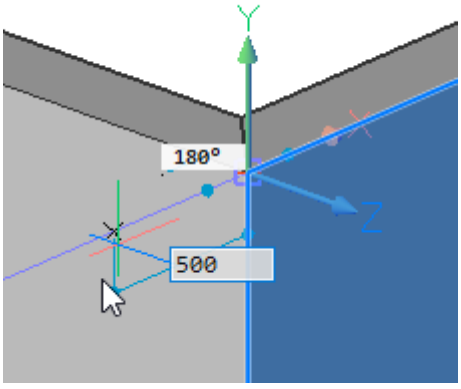
3 When the face highlights, hit the Shift key to lock the **dynamic UCS**.

4 Do one of the following:

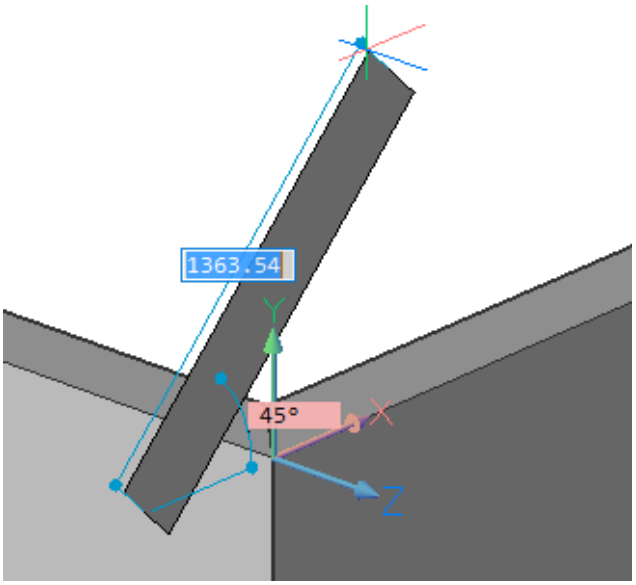
5 To create the roof at the top edge of the wall, snap to the exterior top corner of the wall.

To create a roof overhang (eave):

- Move the cursor over the exterior top corner of the wall to require a snap tracking point.
- When a small red cross indicates the tracking point is acquired, move the cursor to the outside and type the overhang distance in the dynamic entry field.



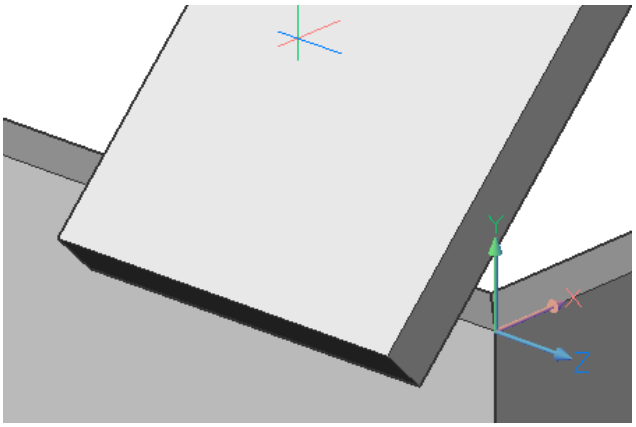

6 Hit the TAB key to activate the Angle field, then type the pitch angle and hit the TAB key again to jump to the Distance field.




7 Specify the height of the roof slab, then press Enter twice: the first Enter creates the footprint of the roof slab, the second Enter interrupts the creation of a second solid.

8 The roof slab is extruded dynamically.

9 Specify a point to define the length of the extrusion.




10 Repeat the previous steps to create more roof slabs.

11 Use CONNECTWITHNEAREST to connect the walls to the roof slabs.

### 20.3.8 Creating walls

#### 20.3.8.1 Commands

POLYSOLID, LCONNECT, TCONNECT, BIMQUICKDRAW

### 20.3.8.2 About the Polysolid tool

The **Polysolid** tool allows you to create wall solids by selecting a start point, an endpoint and entering a height. Note that, in BricsCAD BIM, you always start by creating the geometry first and adding materials or compositions later. This way there is no need to worry about details when modeling a preliminary design.

The use of Dynamic Dimensions and Polar Tracking is recommended. Dynamic Dimensions can be used to specify the length of the wall and Polar Tracking controls the direction.

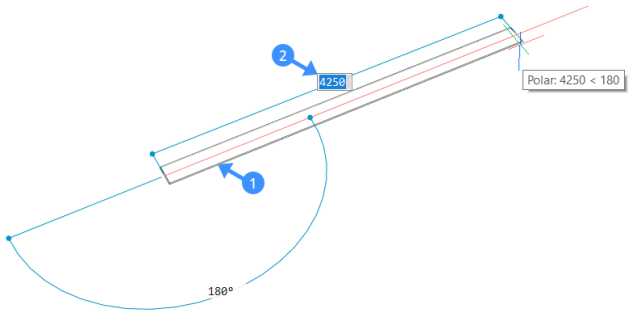
For more information about this command, visit the Command Reference article POLYSOLID.

### 20.3.8.3 Procedure: Create a simple wall

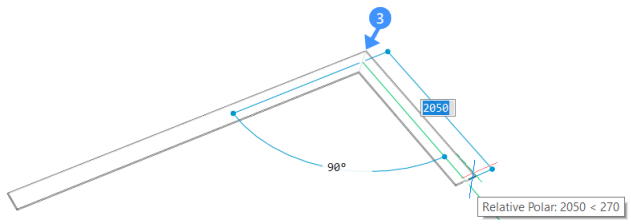
1 Launch the **Polysolid** tool.

2 Select a start point, the footprint (1) of the wall is displayed automatically.

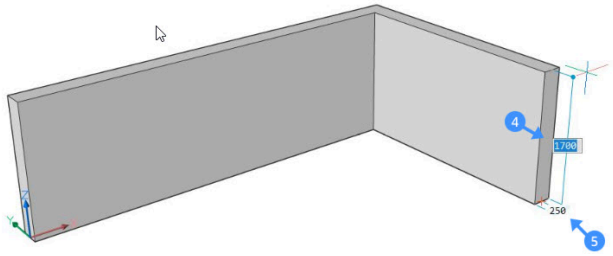
3 Move the cursor in the desired direction, the current length value is displayed in the length dynamic entry field (2).




4 Select an endpoint or enter a value in the dynamic input field and press Enter. The footprint (3) of the adjacent wall is displayed. Select a new endpoint or enter values in the dynamic input field to create adjoining wall segments.




5 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field (4). Here, you can change the default height of the wall. To change the width of the wall, use the width field (5). You can switch between the two dynamic input fields by pressing Tab.




6 To change the justification of the wall hit CTRL while the 'Polysolid justification Hotkey Assistant widget' appears at the bottom of the screen.




***Прим.:*** The width and height of the previous wall will be the new default values of the next Polysolid.

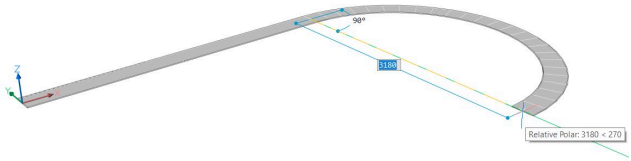
### 20.3.8.4 Procedure: creating a curved wall

1 Launch the **Polysolid** tool.

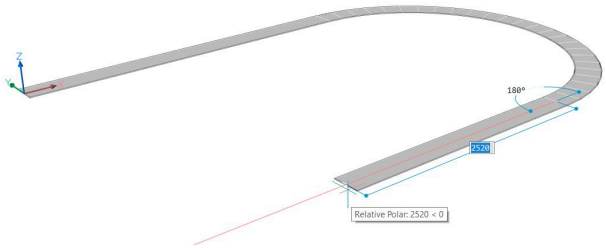
2 Select a start point and move the cursor to set the length or click to set the endpoint.

3 To create a curved wall, type A and press Enter to choose **Draw arcs** or select **Draw arcs** in the prompt menu.

4 Now move the cursor in the desired direction to curve the wall. Enter a value in the dynamic input field to set the degree and length of the arc. Press Enter.

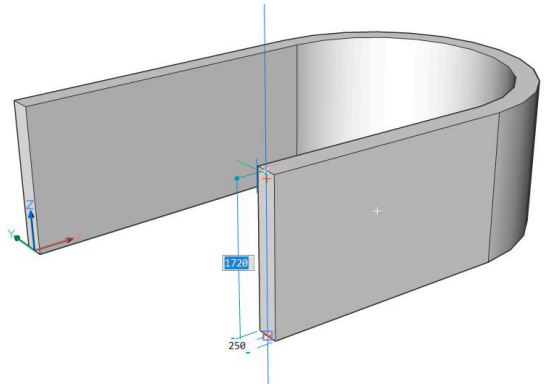



5 You can continue creating curved, adjoining wall segments by clicking or by entering values in the dynamic input fields. To go back to drawing straight wall segments, type L and press Enter or select **Draw lines** in the options dialog.




6 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field. Type a value to change the default height of the wall. To change the width of the wall, use the width

field. You can switch between the two dynamic input fields by pressing Tab.




### 20.3.8.5 Procedure: using an existing 2D plan to create walls

Using an existing 2D plan view enables you to create a wall element in 3D.

1 Click the **Polysolid** tool from the Quad and type E on the Command line or select **Entity** in the prompts menu. Using the **Entity** option in **Polysolid** enables you to create 3D geometry from a 2D layout.

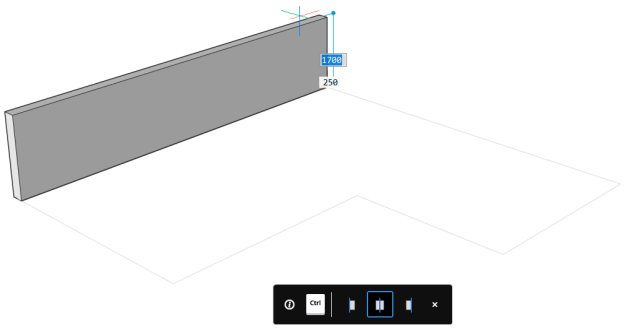
2 Choose a 2D linear entity which will be the Polysolid base.

3 Lines, open and closed polylines, arcs, circles, ellipses, elliptical arcs, and splines are accepted as a polysolid base.

4 Move the solid up or down and left-click to set the height of the wall, enter a value in the dynamic input field or right-click to accept the default value.

5 The current width of the wall appears in the Width Field. Press Tab switch between the Height and the Width field.

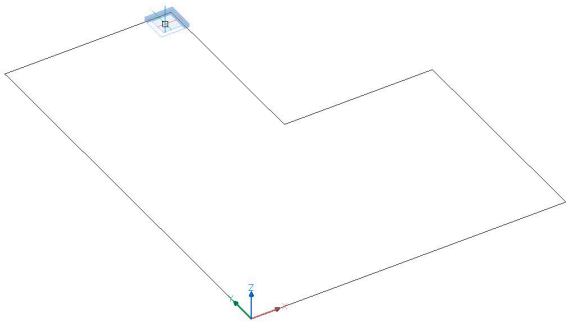
6 The Hotkey Assistant appears and displays the possible justification options. Press CTRL to cycle through the wall justification options.



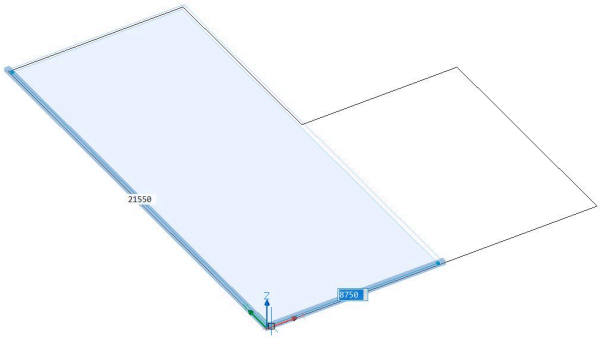

It is also possible to use an existing 2D plan to create walls using the **BimQuickDraw** tool:

1 Open the **Quickdraw** tool from the Quad or type Quickdraw in the Command line.

2 The cursor automatically snaps to the lines of your 2D drawing. Place your cursor where you want to place your first corner and left-click.



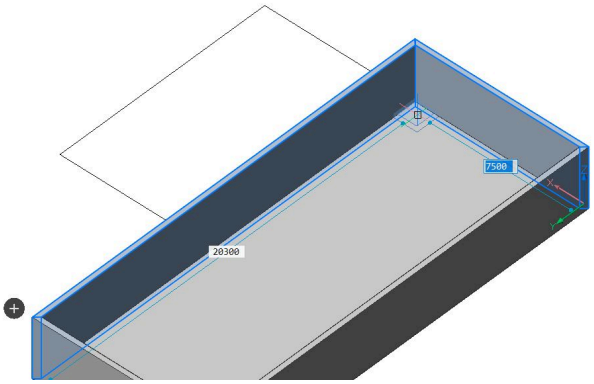
3 Hover your mouse to the opposite corner and left click again or type the values of the lengths of the sides in the respective fields and press Enter.



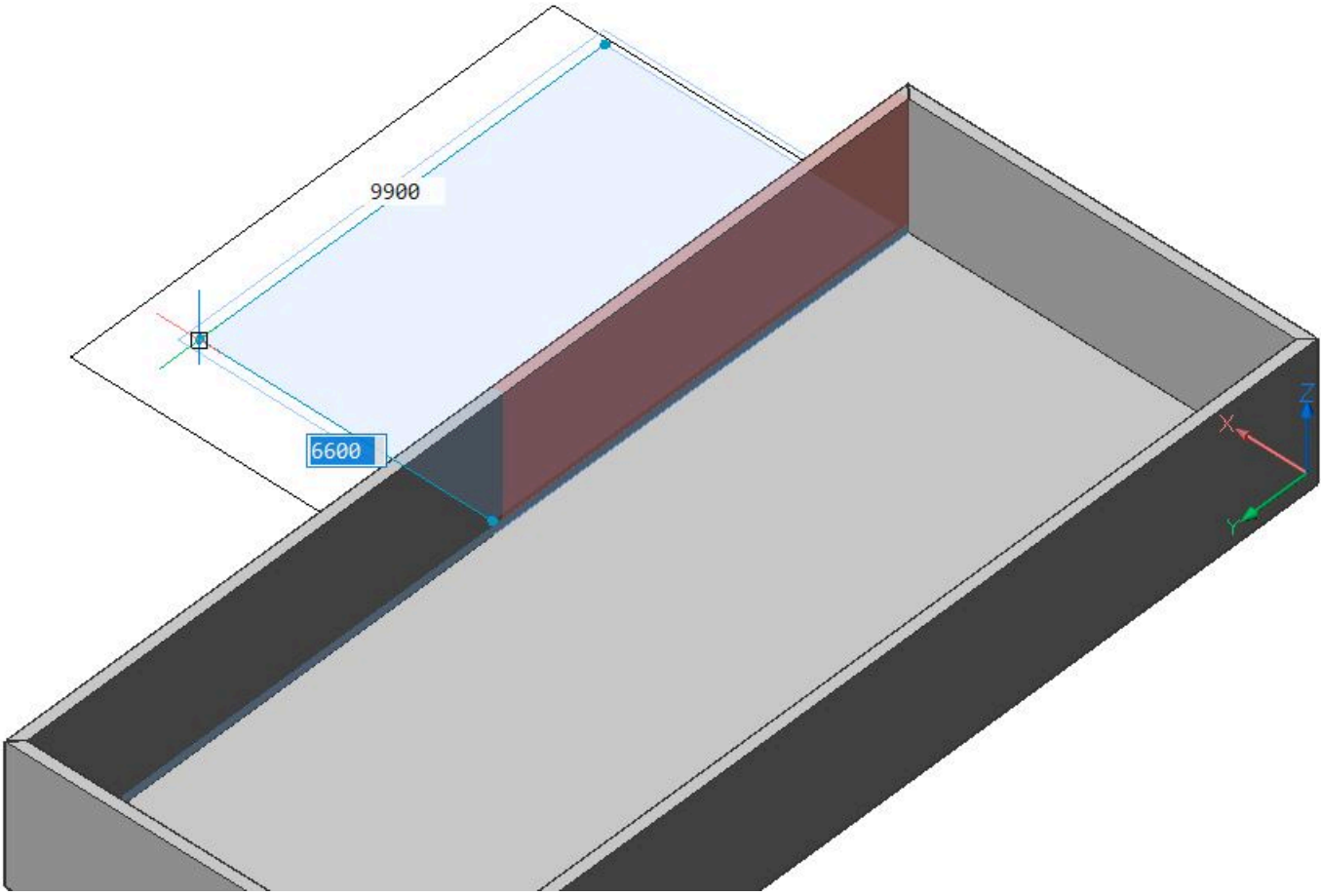

4 The **Quickdraw** tool always uses the default height and width settings.

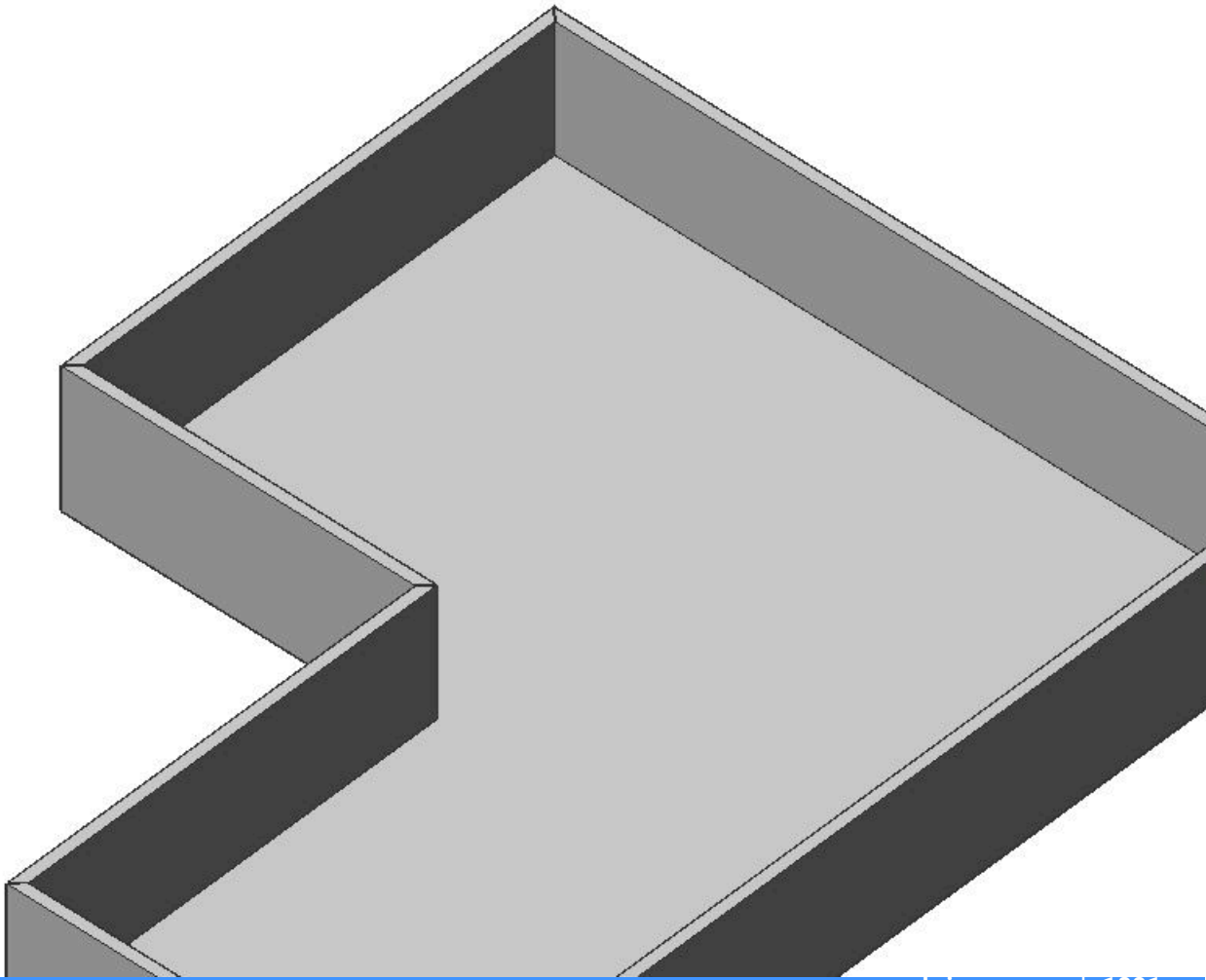
To change these values type S and press Enter to open the Quickdraw Settings. Change the values in this menu.

5 To finish the L-shaped form, you can draw a new rectangular on the second part of the figure. The common wall of the two rectangles will automatically be removed if you draw a second rectangular starting from the interior side of the first rectangular as indicated below:




6 While drawing the second rectangular, you can see the wall that will be deleted is displayed in red.


9900
6600
Z
X
Y



### 20.3.8.6 About the BimConnect tool

The **BimConnect** tool allows you to connect one wall to another element.

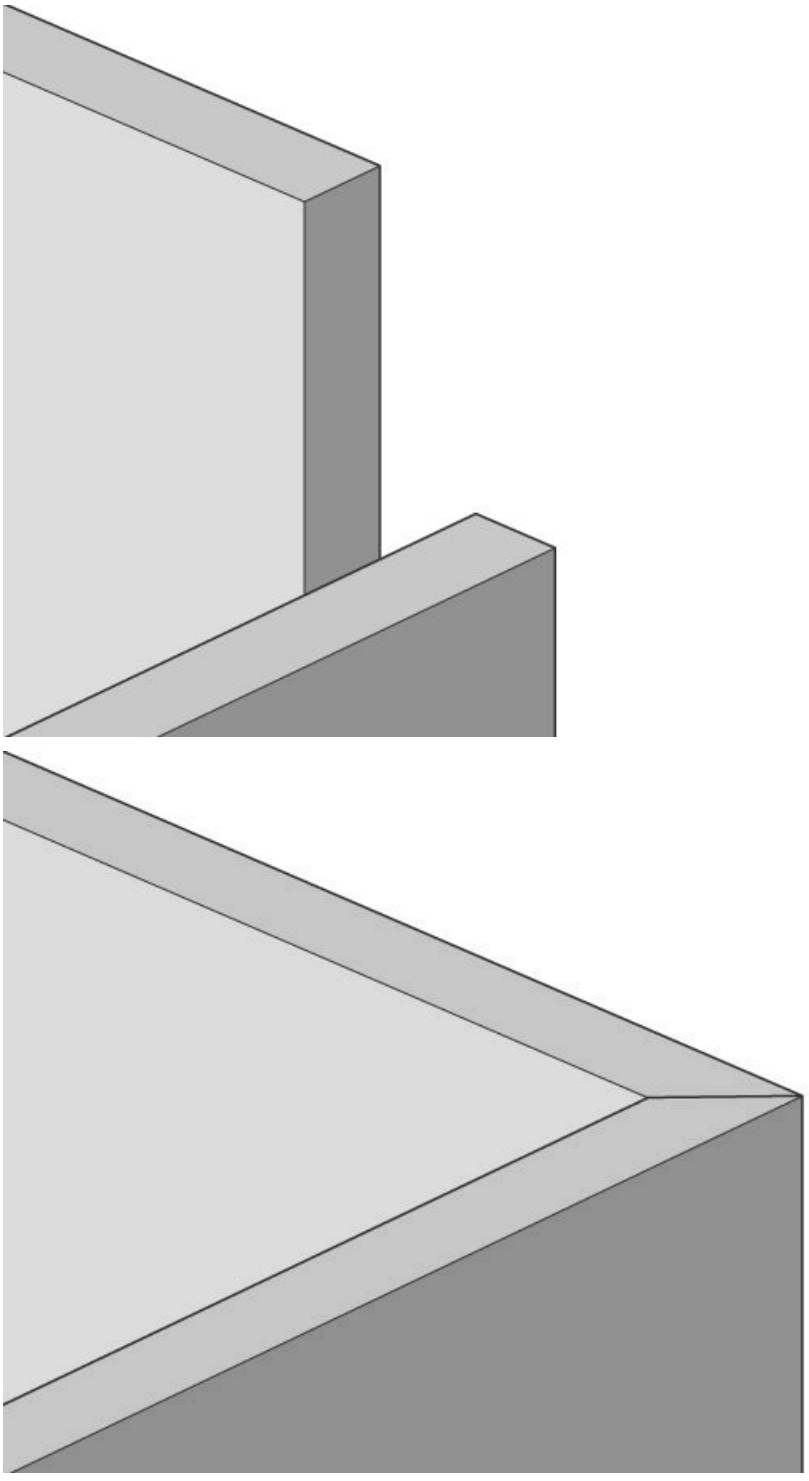


For more information about this command, visit the Command Reference article LCONNECT.

### 20.3.8.7 Procedure: connecting two walls

1 Select two wall elements and apply the **LConnect** tool.

When two walls intersect, a mitered connection will be created by default.



2 Change the layout of the wall connections using the ‘LConnect HotKeyAssistant widget’.

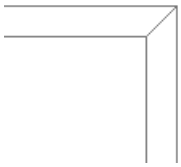
3 The selected connection type is indicated by a blue frame.




The connection can be (from left to right) a bisector L-connection , L parallel type 1 , L parallel type 2 or a disconnection.
Switch between the options by pressing CTRL.

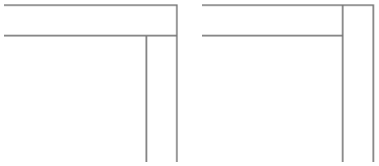
- **Bisector L-connection (miter)**

  When the **bisector L-connection** is selected, the walls are connected at an angle of 45°.



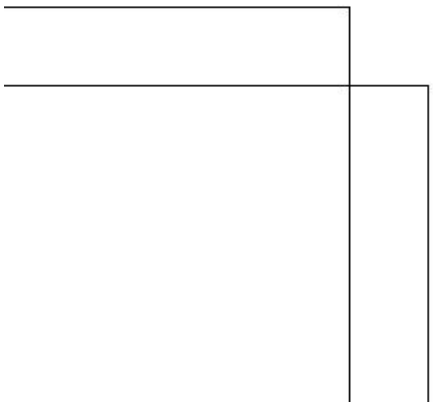
- **Parallel L-connection 1 Parallel L-connection 2**

  When the L parallel type 1 or type 2 connections are selected, the walls are connected at an angle of 90°.



- **Disconnection**

  When the disconnected option is selected, the existing connection between the walls is removed.



Press Enter to accept the current connection type or press CTRL to cycle through the possible connection options.

### 20.3.8.8 Connecting walls to other elements

The **Connect with Nearest** tool allows you to connect the minor face of a wall to the major face of another element, regardless of the orientation of the faces. This tool can be used to connect the top face of a wall to a roof, the bottom face to a slab or the side face of a slab to a wall.

To connect the face to the nearest object, select the face of the object that you would like to connect with the nearest object and choose the Connect With Nearest tool in the Quad.

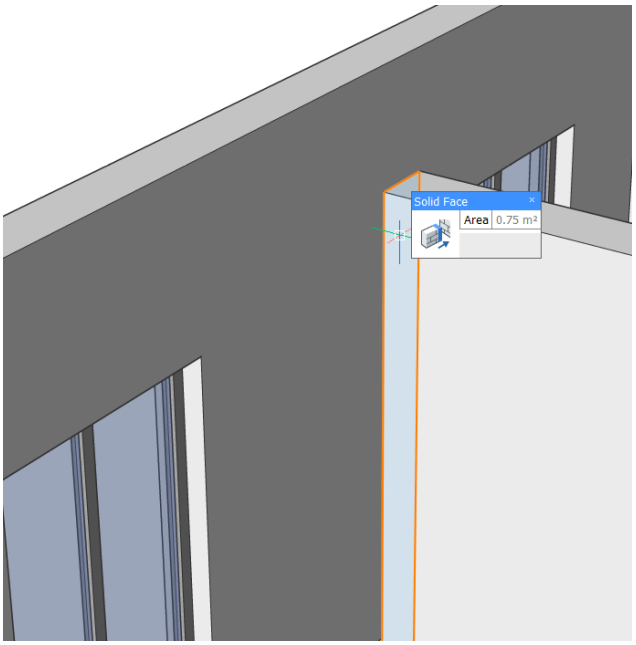
***Прим.:*** Before using the **Connect with Nearest** tool, make sure the **Select Faces** option is enabled in the selection mode settings.



For more information about this command, visit the Command Reference article TCONNECT.

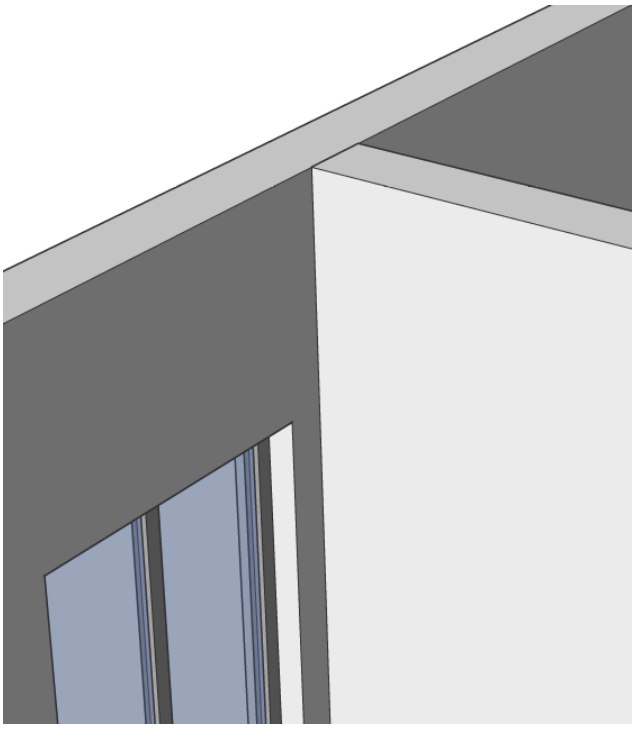
### 20.3.8.9 Procedure: creating T- connections between two walls

1 In the drawing area, highlight the face you want to connect to another wall.




2 Use the **Connect with Nearest** tool from the Quad.

The **Connect with Nearest** tool automatically makes a T-connection between the two walls by extruding the face of the first wall to the face of the other wall.



### 20.3.8.10 About the select aligned faces tool

The **Connect with Nearest** tool can also be used to connect multiple faces of a wall to the nearest face of another element. To do this use the **Select aligned faces** tool. This tool enables you to highlight all faces that are coplanar with the current selection.

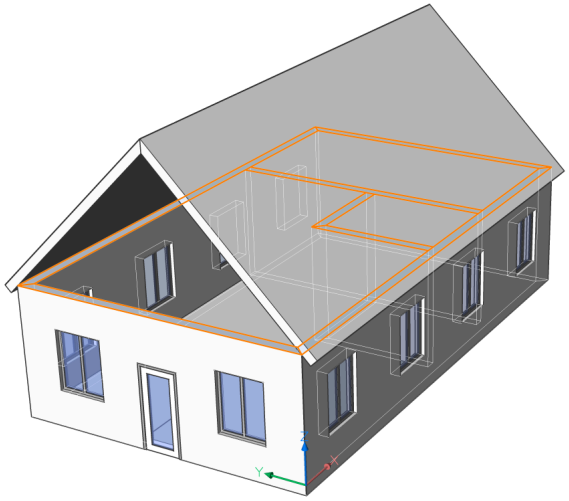


For more information about this command, visit the Command Reference article SELECTALIGNEDFACES.

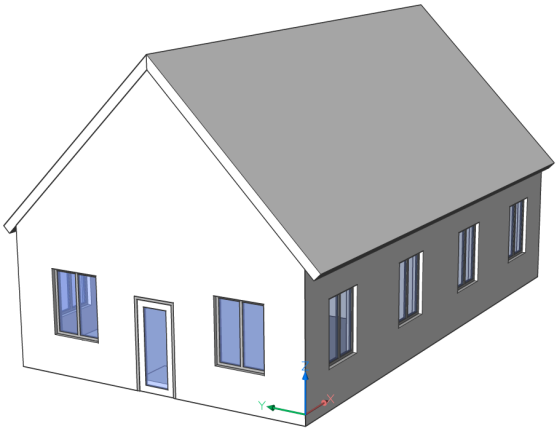
### 20.3.8.11 Procedure: connecting multiple faces of the wall to the roof slab

1 To select all faces aligned to the current selection, highlight the top face of one of the walls and click

the **Select aligned faces** tool in the **Select** tab of the Quad.



2 Launch the **Connect with Nearest** tool. The top faces of the wall are attached to the roof.



### 20.3.9 Curtain wall

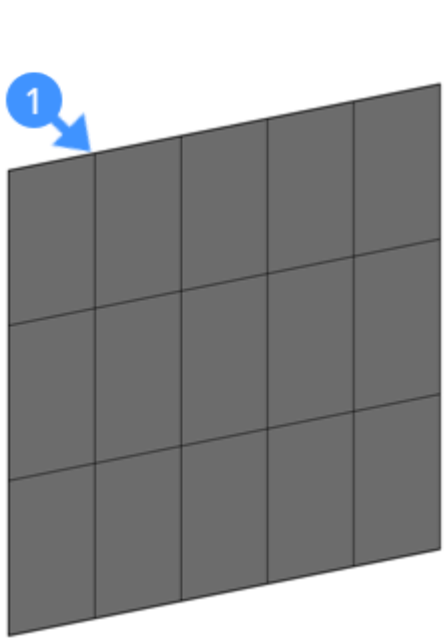
#### 20.3.9.1 Commands

BIMCURTAINWALL

#### 20.3.9.2 About

In BricsCAD, curtain walls are created from the face of a 3D solid or surface as a block by using the **Curtain Wall** tool.

- Curtain walls can be created from any planar and curved surface.
- Window Frames are placed on a BIM Grid when creating a Curtain Wall.
- Changing the U-line and V-line panel options allows you to specify the desired size and number of panels on the planar or curved surface.



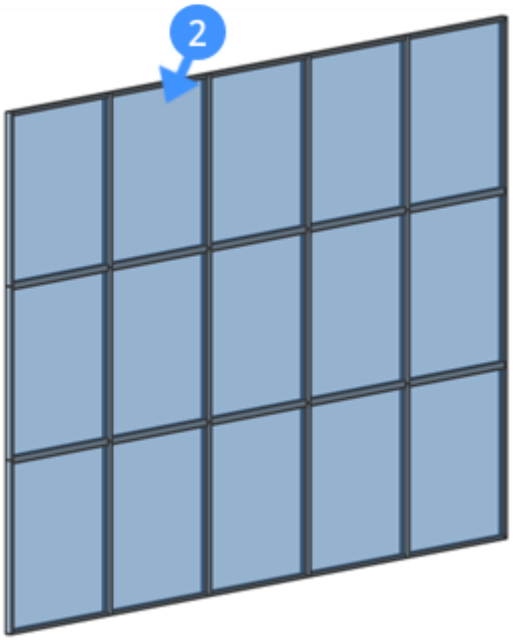




Figure 1, Curtainwall grid (1).

Figure 2, Curtainwall with glazing (2).

The curtain wall system consists of panels of glass. This is a panel infill for a curtain wall.

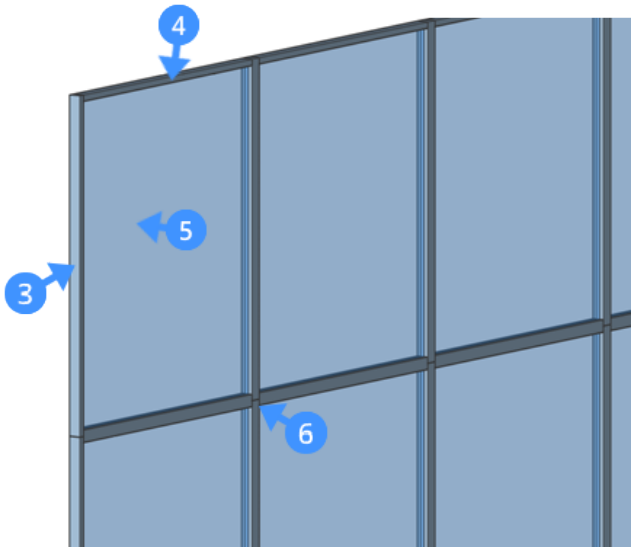
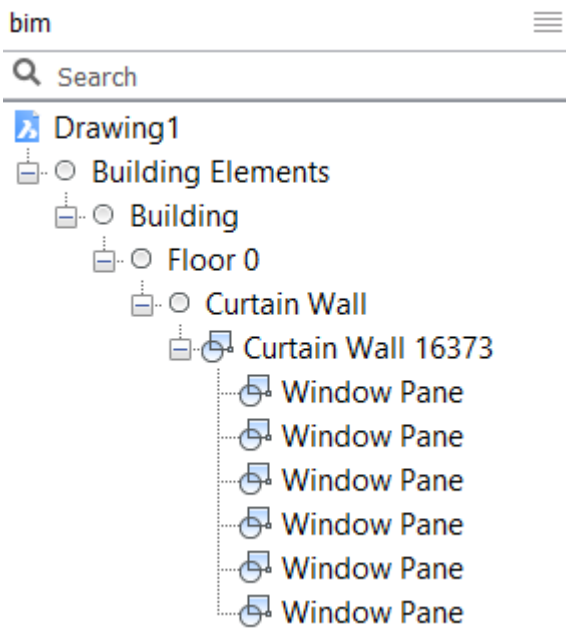



Figure 3, A curtain wall. The frame that coincides with the grid vertical (U) panels (3), the frame that coincides with the grid horizontal (V) panels (4), and glass in-fill (5) are shown. By default, the straight connection (6) is made between the beams.

Curtain walls are automatically classified as a **Curtain wall** in the **Structure browser**.

If you explode the Curtain wall, the sub-entities are classified as windows and as frames. The frames are linear solids. This allows you to calculate the length of a curtain wall.




### 20.3.9.3 Procedure: creating a curtain wall

1 Launch the BIMCURTAINWALL command in the Command line.

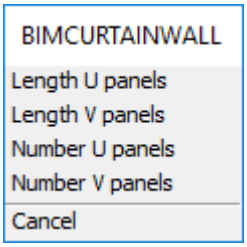
1 You are prompted: Select a face:

2 Select a face in the drawing area to create a curtain wall.

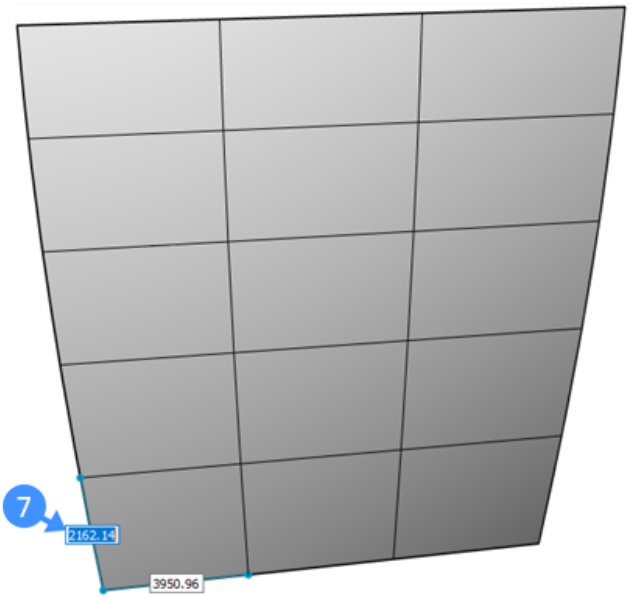
3 The curtain wall grid is automatically generated on this face.

You are prompted: Create grid or [Lenght U panels/Length V panels/Number U panels/Number V panels]:

4 Change the length of the U or V panels and number of U or V panels using the prompt menu and the Command line.




Or, if Dynamic dimension (DYN) is on, the length of the panels can be changed using the DYN fields (7).




5 Hit the TAB key to switch between dynamic dimension boxes.

6 Or Press Enter to accept the current value.

***Прим.:*** The maximum deviation from planarity is displayed in the command line.

**Optional**: If the value of the max deviation from planarity is different to "0", the prompt menu (8) will appear and the Command line prompts a message (9).




The prompt menu and the Command line allow you to planarize the grid cells.

Click "Yes" to planarize the grid cells. By default, the target deviation from planarity is set to "0.00001".

7 Press Enter to accept the default value.

8 You are prompted: Change parameters or accept [Accept/Width/Depth/Glass thickness/Connections type] <Accept>:

The prompt menu:




(**Optional**) To change the frame width, select the **Width** option in the prompt menu or type "W" in the Command line.

Default frame width = 60.000000

(**Optional**) To change the frame depth, select the **Depth** option in the prompt menu or type "D" in the Command line.

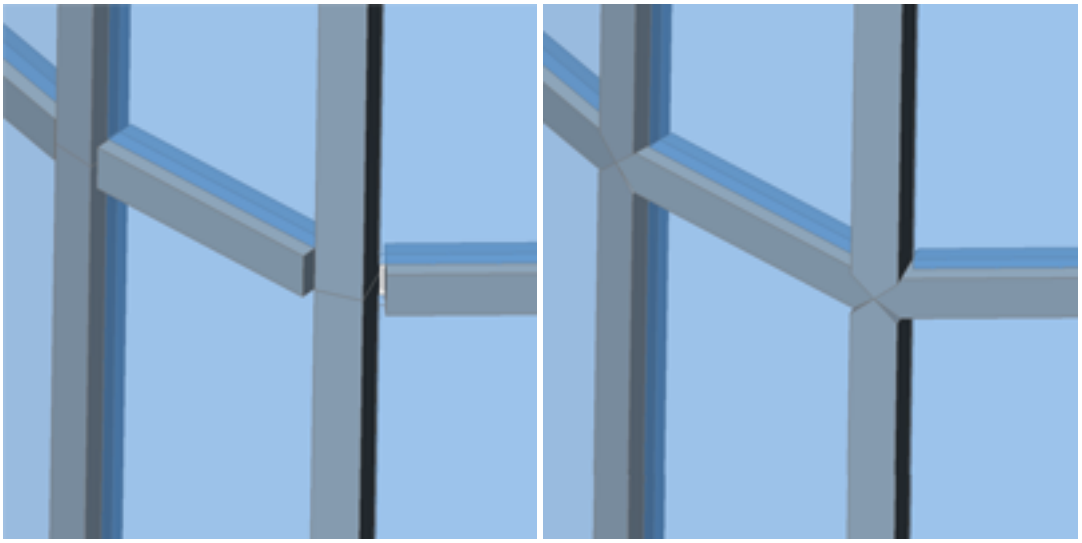
Default frame depth = 60.000000

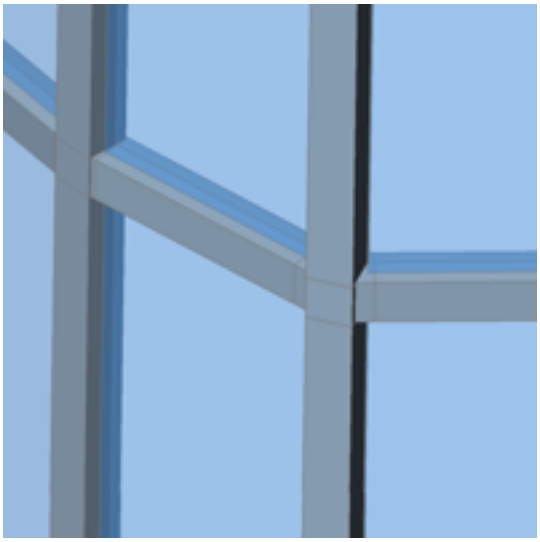
(**Optional**) To change the glass thickness, select the **Glass thickness** option in the prompt menu or type "G" in the Command line.

Default glass thickness = 20.000000

(**Optional**) To choose a connection type between, select the **Connections type** option in the prompt menu or type "C" in the Command line. For smooth connections, select the **Smooth connection** option or type "S" in the Command line. For nodes connection, select the **Nodes connection** option or type "N" in the Command line.

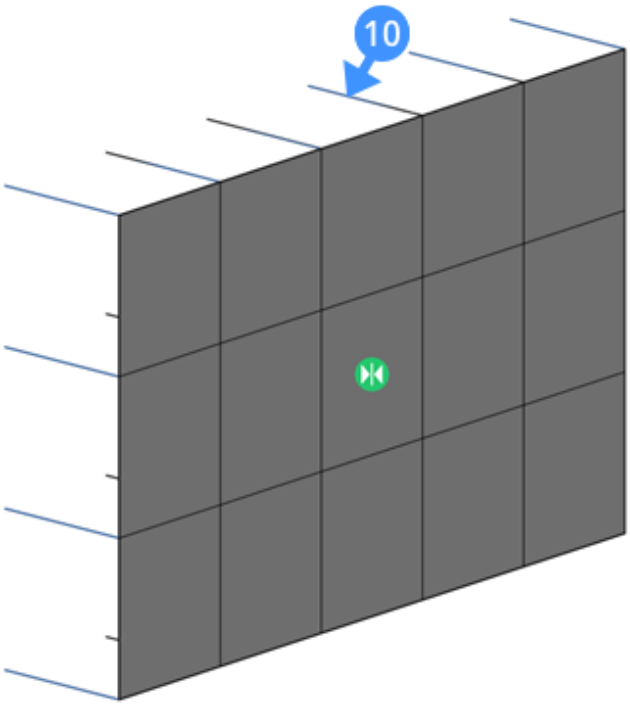
The default connection is straight.





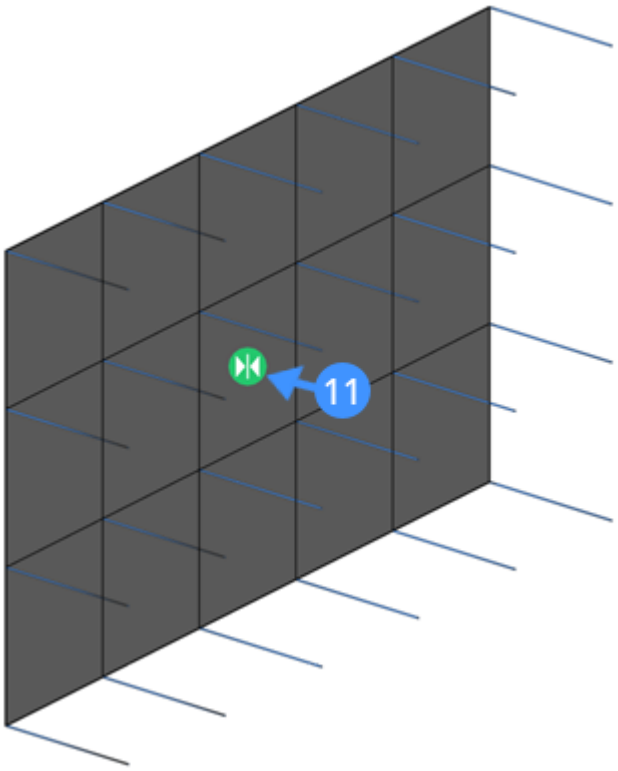
There are three types of connection. From left to right: Straight - Smooth - Nodes.

9 Blue lines (10) appear on the face to show the side a curtain wall will be created.




***Прим.:*** Step 6 and step 7 are skipped when the face is selected from a solid. In this case the inside normal is chosen by default.

10 Use the widget (11) to flip the curtain wall.

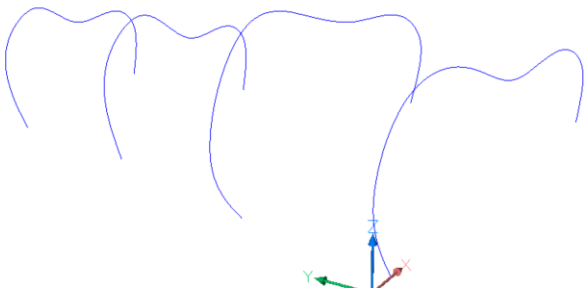



11 Press Enter to accept.

***Прим.:*** The surface or solid used to create the curtain wall will be deleted by default. If you don't want this, set the **DELOBJ** value to '0' in **Settings**.

### 20.3.9.4 Procedure: Creating a curtain wall from a free form surface

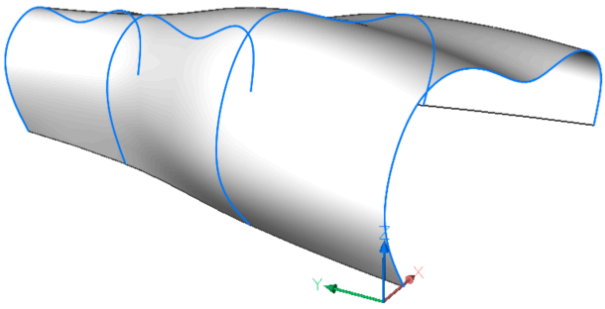
1 Start with 4 splines.



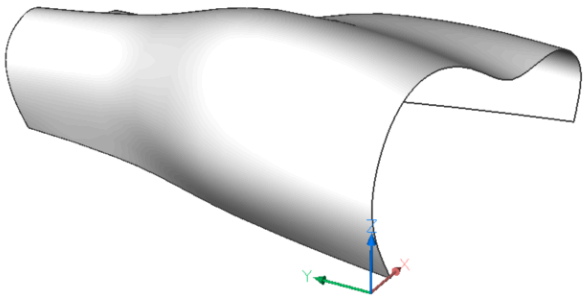
2 Launch the LOFT command in the Command line.

3 You are prompted: Select cross-sections in lofting order [MOde]:

Select the 4 splines in lofting order.



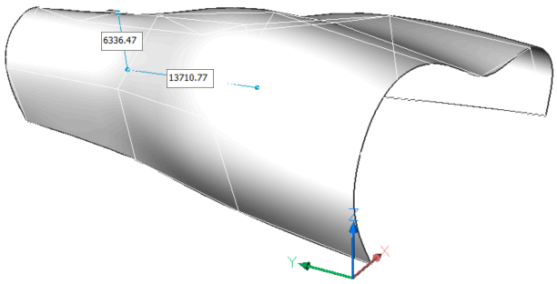
4 Press Enter twice to end the command.



5 Launch the BIMCURTAINWALL command in the Command line.

6 You are prompted: Select a face

Select the lofted surface.




7 A curtain wall grid is automatically generated from this surface.

8 You are prompted: Create grid or [Lenght U panels/ /Length V panels/Number U panels/Number V panels].

Change the number of the U and V panels in the prompt menu or the Command line. In this example: U = 20 and V = 8.

9 You are prompted: Planarize [Yes/No] <Yes>

10 Click "Yes" to planarize the grid cells.

11 You are prompted: Enter target deviation <0.00001>:

12 By default, the target deviation from planarity is set to "0.00001".

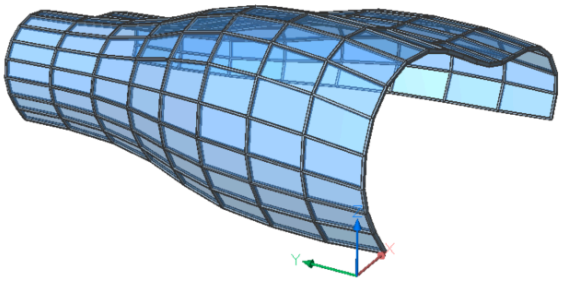
Press Enter to accept it.

13 You are prompted: Change parameters or accept [Accept/Width/Depth/Glass thickness/Connections type] <Accept>:

14 Change the parameters to:

- Width: 150
- Depth: 300
- Glass thickness: 20
- Connection type: Smooth

15 Press Enter to accept. A curtain wall is created.



### 20.3.10 DmExtrude

#### 20.3.10.1 Commands

DMEXTRUDE

#### 20.3.10.2 About

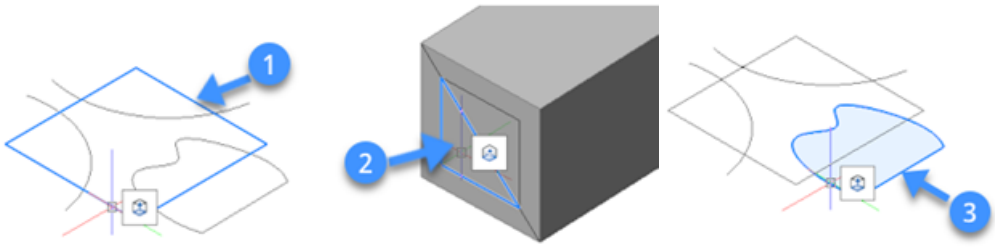
The **Extrude** tool allows you to create 3D solid(s) by extruding closed 2D entities, faces of 3D solid(s), regions or closed boundaries. The **Extrude** tool enables you to cut the solid model partially by removing the volume from the main solid and it can also create new solid(s) by adding volume to the main solid model.



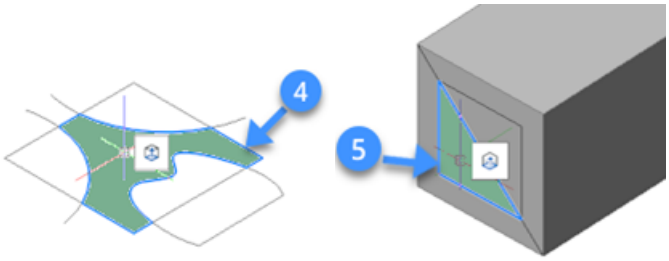
For more information about this command, visit the Command Reference article DMEXTRUDE.

The following illustration shows the selected entities to be extruded.

Closed entities (1) (2) are shown. A region object (3) is shown in the figure.



The following illustration shows a closed boundary in the XY-plane of the current coordinate system (4) and the closed boundary on the face of the solid (5) is highlighted while hovering over it.




The closed boundaries are recognized automatically in BricsCAD, when the **Enable Boundary Detection** option on the **Selection modes** toolbar is active.

***Прим.:*** As a closed entity only closed polylines, circles, and ellipses are accepted.

### 20.3.10.3 Procedure: using DMEXTRUDE in solid mode

The following are general steps to add and remove volume from the solid(s) using the **Extrude** tool.

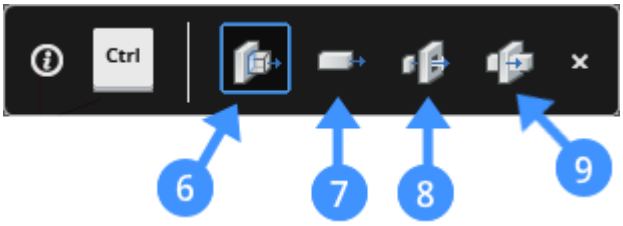
1 Hover the cursor over an entity or sub-entity in the drawing area.

2 Select the **Extrude** tool in the **Model** tab from the Quad.

You are prompted: Specify height of extrusion or set [Auto(subtract or create)/Create/Subtract/Unite/ Both sides/Taper angle/Direction/Limit] <Auto>.

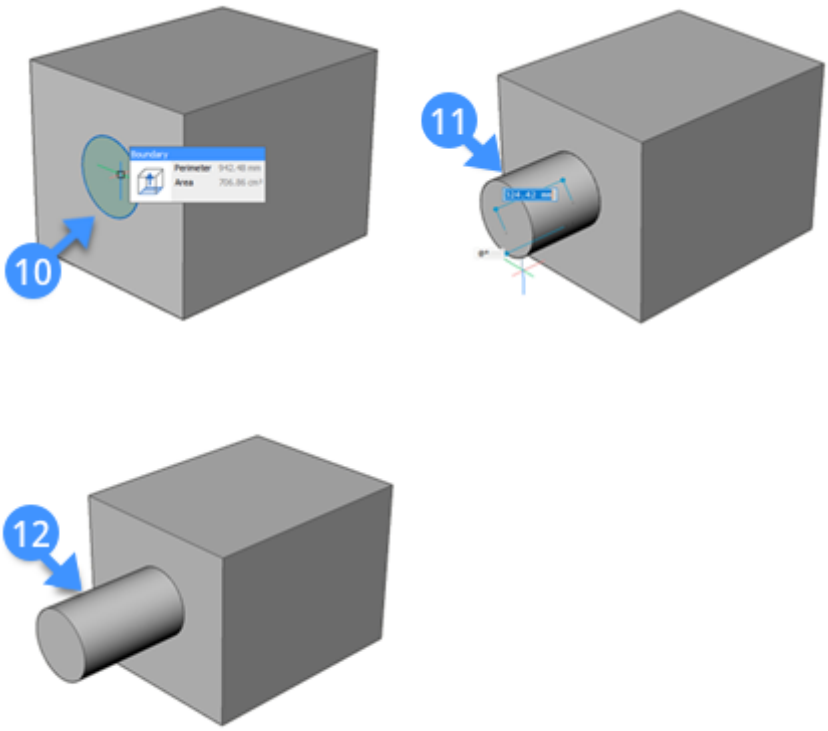
3 The “DmExtrude Hotkey Assistant” appears at the bottom of the screen with a set of extrusion options.

***Прим.:*** The default mode is indicated by a blue frame.

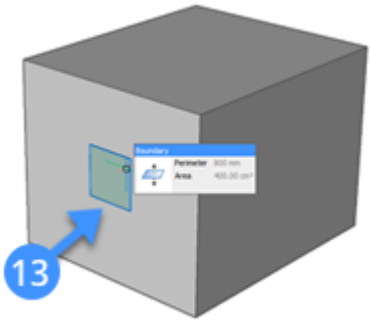


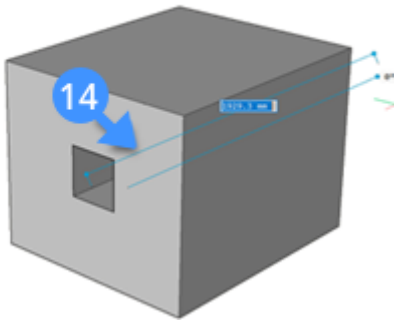

- **Auto (6)**: The result of the auto option depends on the extrusion direction of the 3D solid and the value of the DMEXTRUDEMODE system variable value. Note: By default, the value of the DMEXTRUDEMODE is set to “3”. The following illustration shows the extruded entity from the main solid while the default value is set to “3”. When you highlight an entity (10) and extrude it

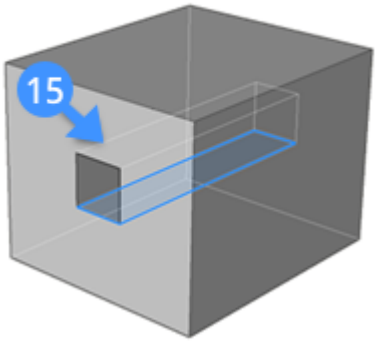
to the outward (11), a new volume is added (12) to the main solid model.



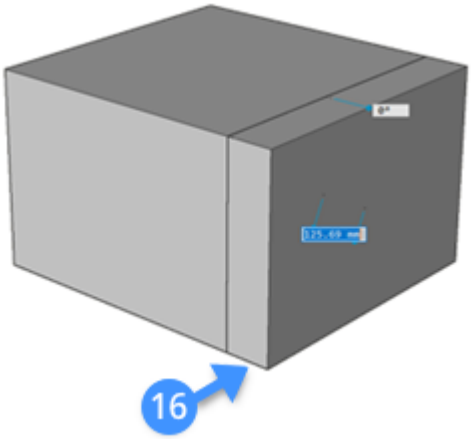

The following illustration shows the subtracted entity from the main solid. When you highlight an entity (13) and extrude it to the inward (14), a volume is subtracted (15) from the main solid.



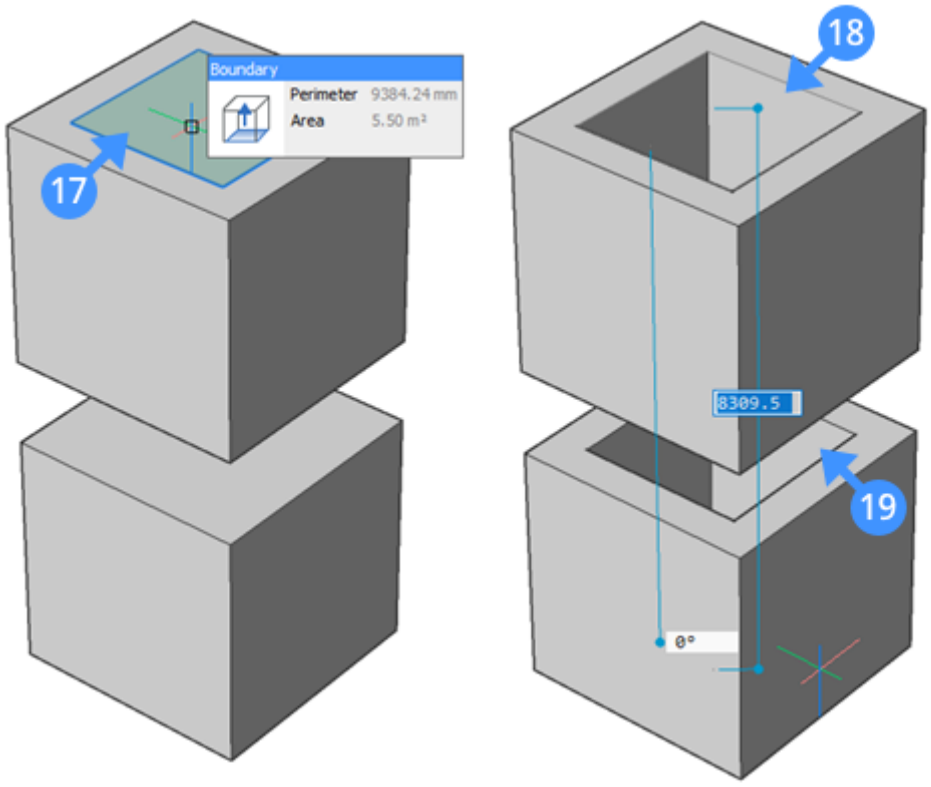




- **Create (7)**: Regardless of the extrusion direction, a new volume is created (16) from the existing solid.



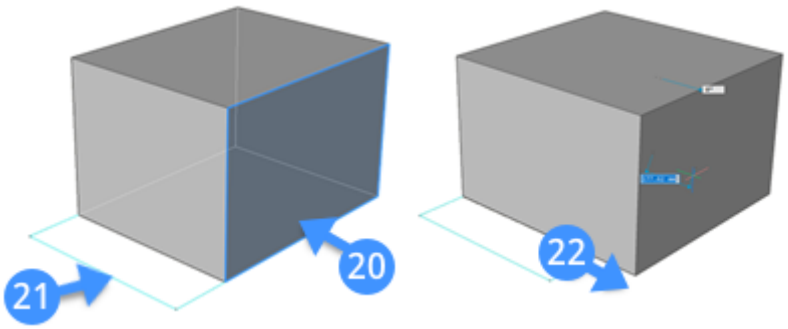

- **Subtract (8):** The 3D solid is subtracted from each interfering existing solid.

  The following illustration shows the subtracted solids (18,19) using the highlighted boundary (17).

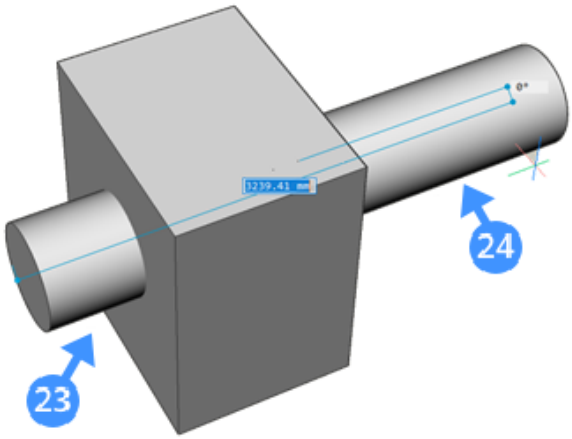



- **Unite (9):** The new volume is unified with each interfering existing solid.

  The following illustration shows the before and after when using the extrusion tool unite option. The solid surface is highlighted (20) and the distance between the left and right face is shown with the aligned line (21). The new volume is unified (22) with the first solid.




- **Both sides**: New symmetric volumes are created (23), (24) in both directions from the main solid.




To create symmetric volumes from the existing solid, type “B” on the the Command line which corresponds to Both sides option of the **Extrusion** tool.

- Select one of the available options to operate the extrusion function on your solid.
- Next, move the cursor in the desired direction to specify the extrusion direction. The selected entity or sub-entity is extruded dynamically.
- Define the height of extrusion by specifying a point in the drawing, entering a value in the dynamic dimension field or choose the limit option.
- Press Enter or right-click to accept the extrusion from your existing solid.

### 20.3.10.4 DMEXTRUDE in Surface Mode

The **extrude** tool allows you to create 3D surface(s) by extruding 2D entities such as lines or polylines.



For more information about this command, visit the Command Reference article DMEXTRUDE.

### 20.3.10.5 Procedure: using the extrude tool in surface mode

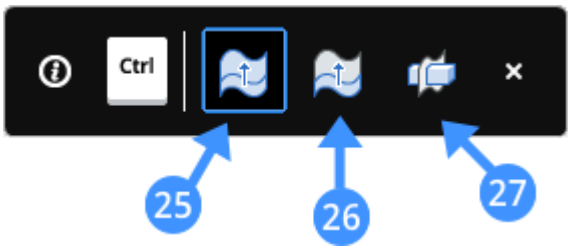
The following are general steps to create 3D surface(s) using the **Extrude** tool in the Surface mode.

1 Move the cursor over an entity or sub-entity in the drawing area.

2 Select the **Surface Extrude** tool in the **Model** tab from the Quad.

You are prompted: Specify height of extrusion or set [Auto(subtract or create)/Create/Subtract/Unite/Both sides/Taper angle/Direction/Limit] <Auto>.

3 The “DM Extrude Surface Hotkey Assistant” appears at the bottom of the screen with a set of surface extrusion options.



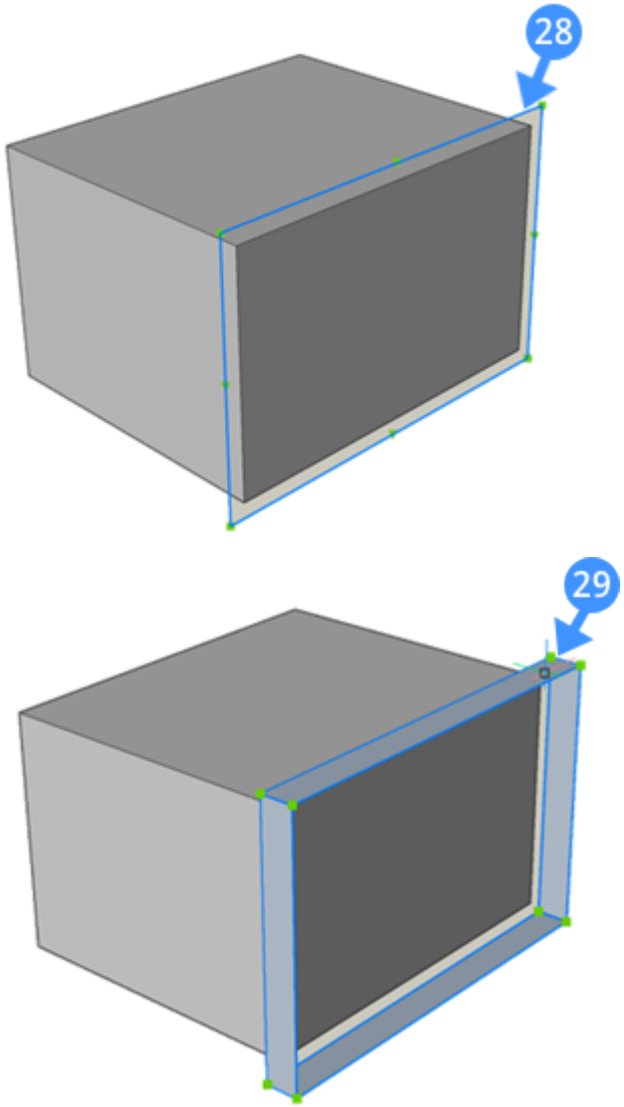

***Прим.:*** The default mode is indicated by a blue frame.

- **Auto (25):**Select either a surface or the face of a solid element. When the surface's edge is used as a contour for extrusion, the created surface is stitched with the base surface. When the surface's face is used for extrusion, the surface results in not stitched with the base surface.
- **Create (26):**Create new surfaces.
- **Slice (27):**Cut through solids with the extruded surface. This option is much like the SLICE command.

4 Select one of the options and move the mouse to give the extrusion direction, entering a value in the dynamic dimension field.

5 Press Enter or right-click to accept the extrusion for the surface.

6 The following illustrations show the created surface (29) by extruding the selected 2D entity (28) in the drawing.




### 20.3.11 DmPushPull

#### 20.3.11.1 Commands

DmPushPull

#### 20.3.11.2 About

The **Push/pull** tool allows you to add and remove volume from solids by moving highlighted faces.

***Прим.:*** **Push/pull** tool applies to planar, cylindrical, spherical, conical and toroidal faces of a solid.



For more information about this command, visit the Command Reference article DMPUSHPULL.

### 20.3.11.3 Procedure: using Push/Pull tool to add or remove volume to the solid

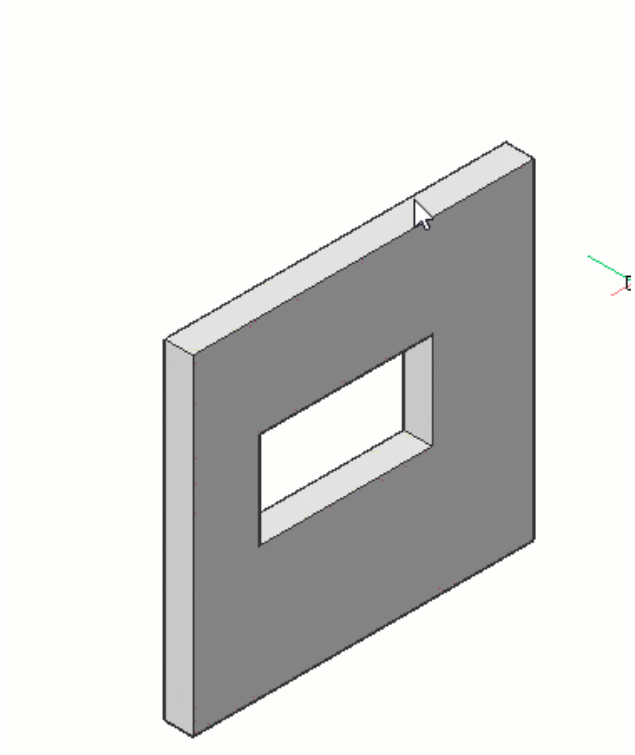
The following are general steps to add or remove volume from a solid(s) using the **Push/Pull** tool.

1 Highlight any face(s) of the solid(s) in the drawing.

2 Select the **Push/pull** tool in the **Model** tab from the Quad.

3 Move the cursor to pull the highlighted face.

4 The following movie shows the highlighted face behavior. The selected face moves dynamically, adding or removing volume.



5 Specify a point in the drawing or enter a value in the dynamic dimension field, then right click to accept changes.

## 20.3.12 Editing grids

### 20.3.12.1 Commands

BIMGRID, BEDIT, NUMBER, PROPERTIES

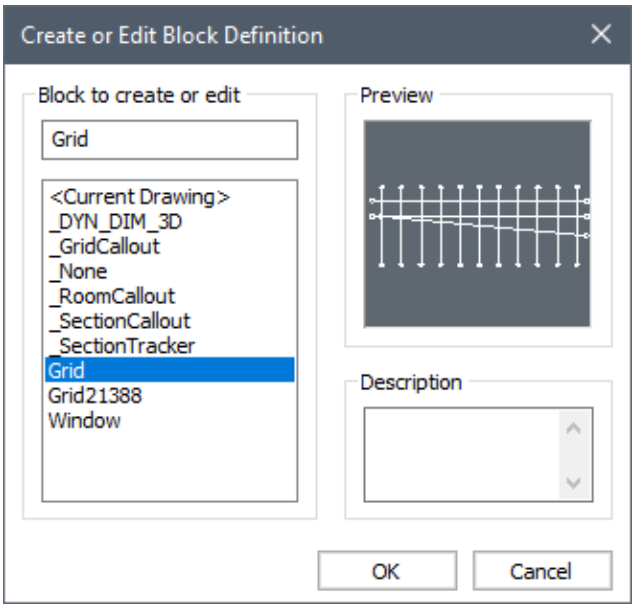
### 20.3.12.2 About

BIM grids are created as a block. The BEDIT (Block Edit) command allows to edit a BIM grids.

### 20.3.12.3 Editing a rectangular grid

1 Double click the grid.

1 The **Create or Edit Block Definition** dialog box displays.




2 Select the grid, then double click or click the OK button.

3 A **Block Edit** session is started.

4 Edit the grid axes.

5 You can move, rotate, copy, stretch, grip-edit or delete grid axes.

6 Renumber grid axes or edit axis labels if necessary (see the procedures below).

7 Click the **Bclose and Save** icon ( ) on the **Block Edit** toolbar.

### 20.3.12.4 Renumbering the grid labels

Renumbering grid labels is needed:

- If you want a different numbering style.
- When grid axes are added (copied) or deleted.

Do the following:

1 Double click the grid to start a **Block Edit** session.

2 Select all vertical axes.

3 Start the NUMBER command and choose the appropriate number style.

4 Make sure to overwrite the existing numbers.

5 Repeat the previous steps for the horizontal axes.

6 Click the **Bclose and Save** icon ( ) on the **Block Edit** toolbar.

### 20.3.12.5 Editing a grid label

1 Double click the grid to start a **Block Edit** session.

2 Double click an axis.

3 The properties of the axis entity (line or arc) display in the **Properties** panel.

4 Select the **BIM/Axis number** property.

5 Type a value or name.

6 Click the **Bclose and Save** icon () on the **Block Edit** toolbar.

## 20.3.13 Inserting and editing windows and doors

### 20.3.13.1 Commands

BIMINSERT

### 20.3.13.2 About

In BricsCAD, the **Insert BIM Component** tool allows you to insert windows, doors and other components, such as furniture, sanitary equipment, plants, etc. in any face of a 3D solid and is typically used to insert windows and doors.



For more information about this command, visit the Command Reference article BIMINSERT.

### 20.3.13.3 Controlling the position of the window or door

To control the position of the window in the face of the solid (wall), you can use the following drawing aids:

- **Dynamic Dimensions**: During placement of a window or door on a 3D Solid face, dynamic dimensions are created, starting from the middle of each edge of the bounding rectangle of the opening to the nearest parallel edge on the face of the 3D Solid. Make sure **Dynamic Dimensions** are active by checking the DYN field in the status bar and **Dynamic UCS** is active by checking the DUCS field in the status bar.

  The **Dynamic UCS** aligns the window to the face of the solid. The edge of the face by which the cursor enters the face defines the orientation of the X-axis. Hit the Shift key to temporarily lock the UCS, which allows to use reference points outside the face of the solid. Hit the shift key again to unlock.
- **Temporary Tracking points**: Click the mouse wheel or press the **Temporary Tracking Points** tool button () on the **Entity Snaps** toolbar to start specifying temporary points.
- **OSNAPZ**: If this system variable is ON, the Z-value of any entity snapping point is replaced by the current value of the ELEVATION system variable. If ELEVATION=0, entity snapping points are forced to lie in the face of the solid. OSNAPZ is toggled by the Ignore entity snap elevation tool button () on the **Entity Snaps** toolbar. The state of the tool button (pressed or not) indicates the current value of **OSNAPZ**.

- **Entity snap to negative Z-values**: Press the **Entity snap to negative-z** tool button ( ) on the **Entity Snaps** toolbar to enable entity snaps on points that lie behind the selected face. If **OSNAPZ** is ON, the point is projected onto the selected face.

### 20.3.13.4 Procedure: inserting windows or doors

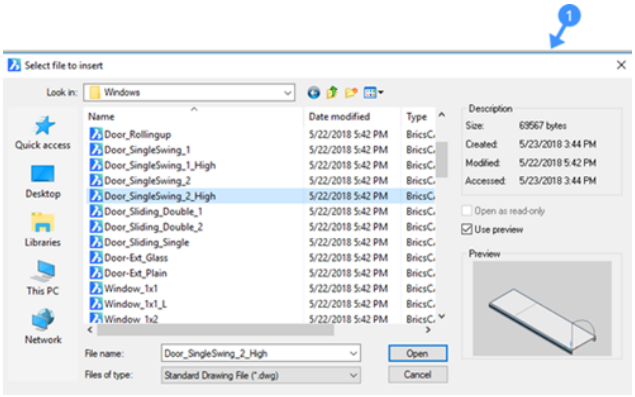
***Прим.:*** Make sure **Dynamic UCS** (DUCS) and **Dynamic Dimensions** (DYN) are active.

**Using the BIMINSERT command**

1 Select **Insert BIM Component**from the **Model** tab of the No Selection Quad.

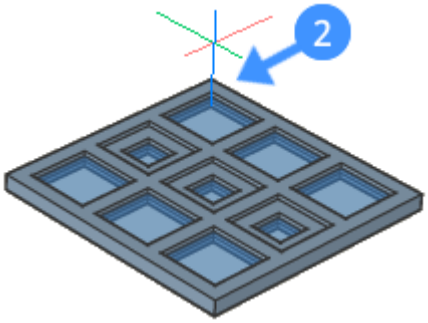
The file explorer opens, displaying the content of the most recently used folder to insert a BIM component. The default folders for BIM components sit in the following subfolder of the BricsCAD installation folder: ...\UserDataCache\Support\en_US\Bim\Components.

2 Select the **Windows** (1) or **Doors** folder.




3 Select a window or door you want to insert in your drawing, then double click or click the **Open** button.

The selected insert is attached to the cursor (2).



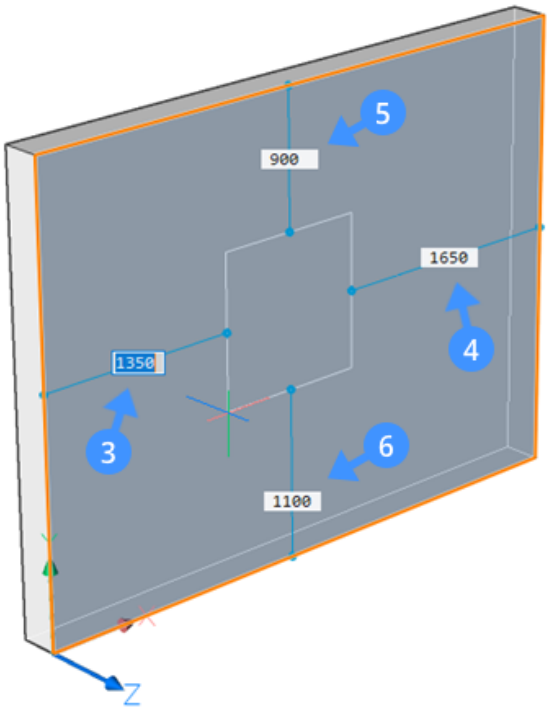

The parameters of the insert display in the **Properties** panel.

Prompts you: Select insertion point or [Edit inserted entity/Rotate component/set Base point/Name/insertion Type/Flip/mUltiple/Change target 3d solids] <0, 0, 0>:

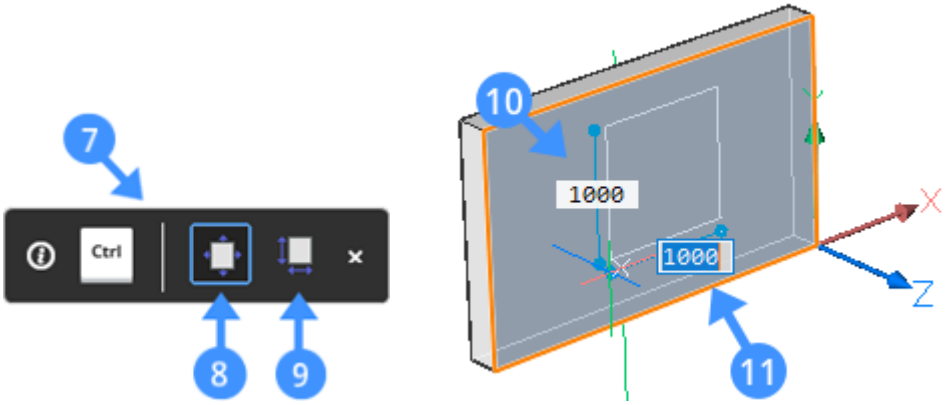
4 Optionally, edit the parameters in the **Properties** panel or choose an option (see Command Options of the BIMINSERT command).

5 When you move over a face of the solid, the insert aligns with the face under the cursor. The distances

from the start (3), end (4), top (5) and bottom (6) face of the wall to the placement rectangle of the insert display dynamically.




If the HKA (Hot Key Assistant)* is ON, the HKA Widget (7) displays at the bottom of the screen to show you the available options. Use 'pick position' (8) option to adjust the insertion point of the wall or door. Use the 'change parameters' option (9) by hitting the CTRL key to specify the width and height of the insert. The Width (10) and Height (11) field will indicate the current dimensions of the insert.



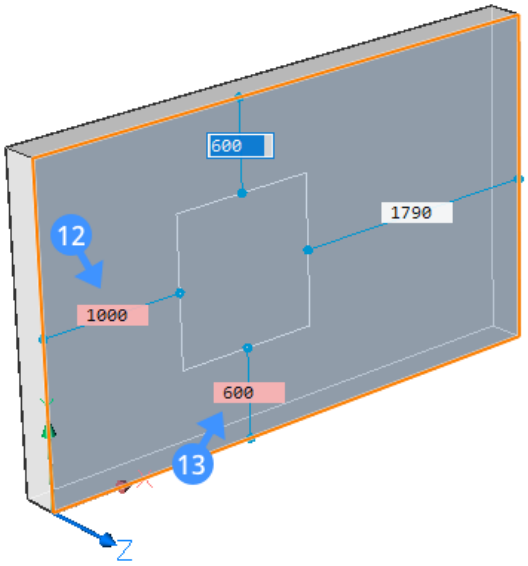

* Click the HKA field in the Status bar to toggle the display of the Hot Key Assistant.

6 Do one of the following to place the selected window or door using the dynamic dimension fields:

- Hit the TAB key to select one of the dynamic dimension fields, then type a value and hit the TAB key to lock the dimension.
- The locked dimension turns red (12).
- Hit the TAB key to select one of the other dynamic dimension fields, then type a value and hit

the TAB key to lock the dimension.

- The locked dimension turns red (13).




**Using the components panel**

See: Insert a window from the Components Panel into your drawing.

### 20.3.13.5 Procedure: editing the parameters of an insert

If parameters are defined in the window or door source drawing, these parameters can be used to edit each detail of a window/door separately.

1 Select any door or window in the drawing. The current values of the various parameters display in the **Parameters** section of the **Properties** panel. In the following illustration, window parameters are shown in the **Properties** panel.

| Parameters | |
|---|---|
| Name | Window_3x2:1 |
| File | |
| W | 1000 mm |
| H | 1000 mm |
| Rebate | 0 mm |
| PlacementDepth | 90 mm |
| FrameThickness | 60 mm |
| FrameDepth | 60 mm |
| PaneDepth | 20 mm |
| PaneOffset | 20 mm |
| MullionThickness | 60 mm |
| TransomThicknes | 60 mm |

2 Select a parameter you want to change, then type a value in the settings field.

### 20.3.13.6 Moving an insert

In BricsCAD, there multiple methods to move an insert in the face of a solid:

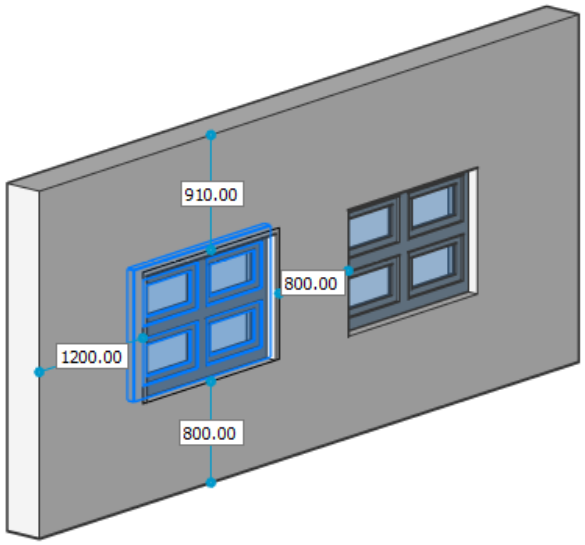
- Using Distance fields.

- Using **Push/Pull**
- Using the Manipulator

**Procedure: using distance fields**

1 Select the insert in the model.

1 Current distances from the insert to the nearest parallel faces of the parent solid display:

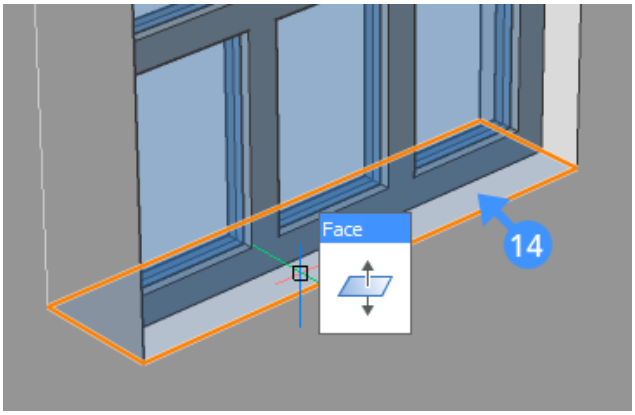


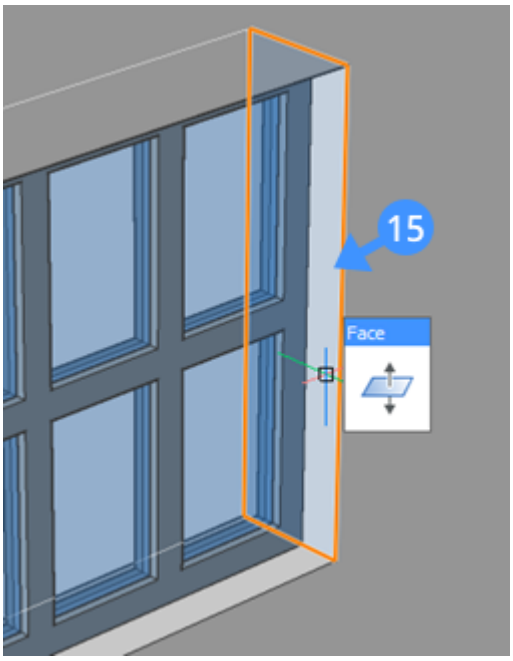

2 Double click a distance field, then type a value and press Enter.

**Procedure: using Push/Pull to move an insert**

1 Hover the cursor over the top or the bottom inner face of the opening (14) made by the insert to move the insert vertically. Or to move the insert horizontally, hover the cursor over the left or the right inner

face of the opening (15) made by the insert.




2 When the face highlights, select **DmPushPull** from the **Model** tab of the Quad.

The insert will move vertically if the top or bottom inner face is highlighted.

The insert will move horizontally if the left or right inner face is highlighted.

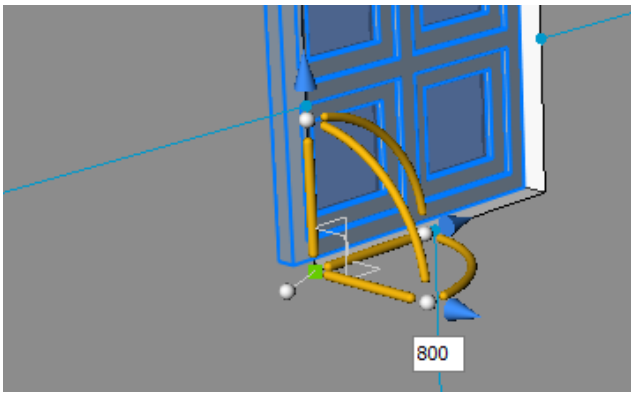
3 Do one of the following to complete the moving insert process.

- Specify a point.
- Type a distance in the dynamic dimension field.
- Use **Adaptive Grid Snap** of the Ruler.

**Procedure: using the Manipulator**

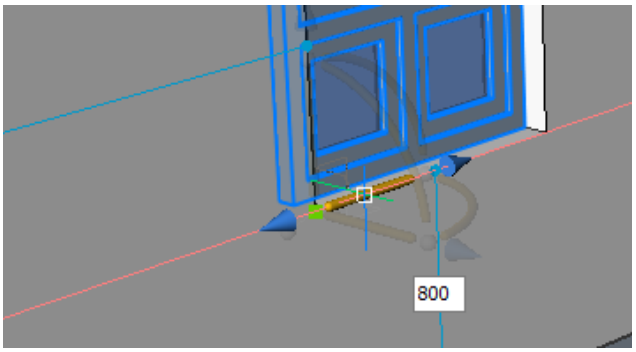
Select the insert by holding the left mouse button down a bit longer. Set the MANIPULATOR system variable to 2.

The Manipulator displays.




1 Hover over the vertical or horizontal axis trigger zone of the Manipulator.

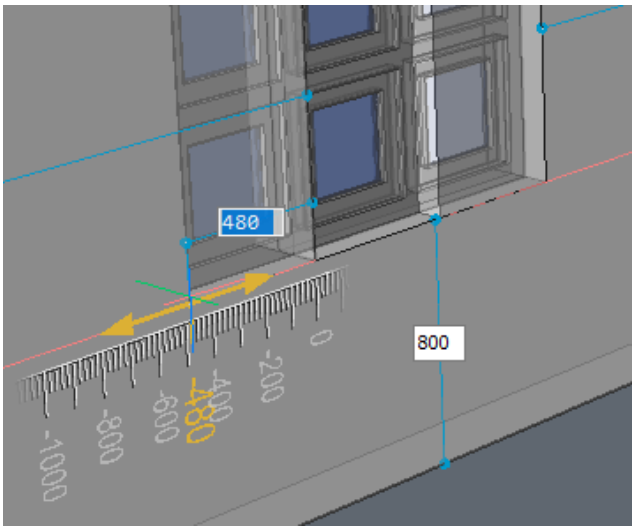
1 The axis highlights and the rest of the Manipulator fades.




2 Click to start the movement operation.

3 The window moves dynamically or if you hold down the Ctrl key creates a copy of the selected insert.

The Ruler displays.




4 Do one of the following:

- Type a distance in the dynamic dimension field.
- Use **Adaptive Grid Snap** of the Ruler.

### 20.3.13.7 Copying Inserts

**Inserting a copy of an insert**

1 Hover over the insert. When the insert highlights, choose **Insert BIM Component** in the **Model** command group in the Quad.

2 You are prompted: Select insertion point or [Edit inserted entity/Rotate component/set Base point/ Name/insertion Type/Flip/mUltiple/Change target 3d solids] <0, 0, 0>:

3 Optionally choose the **Multiple** option to insert multiple copies.

4 Proceed with the Inserting Windows and Doors procedure from step 2 on.

**Copying an insert in a solid**

The **Copy** tool can be used to place copies of an insert in the same solid. Using the repeat option on the **Copy** tool you can create a series of copies at the same interval.

***Прим.:*** The movement vector for the copy should be parallel to the face of the wall. If the orientation of the wall, in which the source window or door is placed, is not parallel to either the X- or Y axis of the current UCS, use the **Face** option of the UCS command to align the UCS.

### 20.3.13.8 Procedure: copying an insert on the wall face

***Прим.:*** Make sure that **Dynamic UCS** (DUCS) and **Dynamic Dimensions** (DYN) are active.

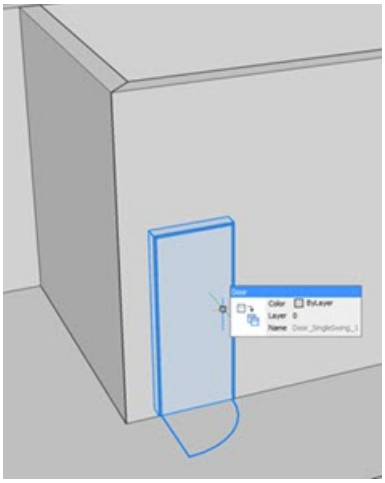
1 Select an insert using the **BIM Insert** tool.

You are prompted: Select insertion point:

2 Place the insert on the wall face.

3 The insert should align with the wall face.

4 Next, hover over the insert to create a copy of it. When the insert highlights, choose **Copy** in the **Modify** command group in the Quad.



You are prompted: Enter base point [Displacement/ mode] <Displacement>:

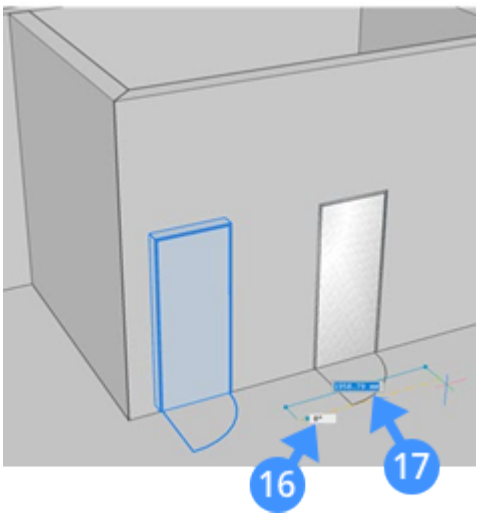
5 Pick a point in the drawing area.

6 You are prompted: Enter second point <Use base point as displacement>.

7 Type a displacement distance from the coordinates of the base point.

***Прим.:*** Displacement specifies the distance to position the copy.

You can use the dynamic dimension fields (16,17) to specify the displacement.




You are prompted: Enter second point [Undo/Repeat/Exit].

8 Optionally, use the **Repeat** option to create multiple copies of the insert keeping the displacement distance of each copy same.

9 Press Enter to accept the copying.

### 20.3.13.9 Replacing an insert

Use the BMREPLACE command to replace an insert.

## 20.3.14 Mass modeling with quickbuilding

### 20.3.14.1 Commands

BIMQUICKBUILDING, BIMSPATIALLOCATIONS

### 20.3.14.2 About

The **QuickBuilding** tool allows you to generate a BIM building just from one or more massing solids. BricsCAD creates a separate drawing and in this drawing walls, slabs and roofs, as well as spatial elements such as floors and spaces will be automatically created

It's useful in early design stages, volume studies and conceptual designs.

The tool will assign BIM properties and spatial information to each part of the new building.

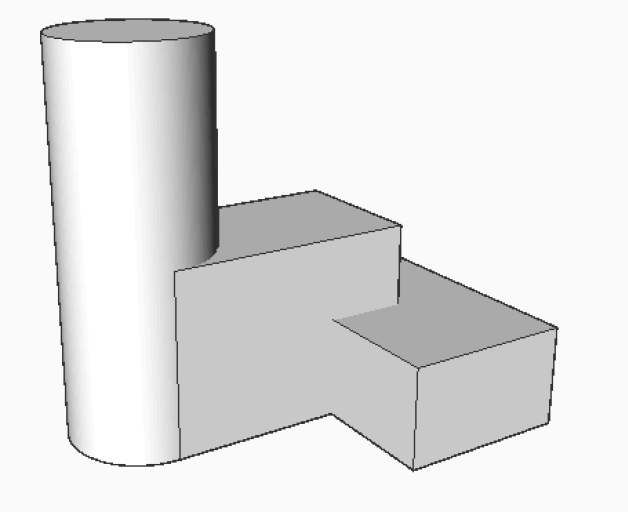


For more information about this command, please visit the Command Reference article BIMQUICKBUILDING.

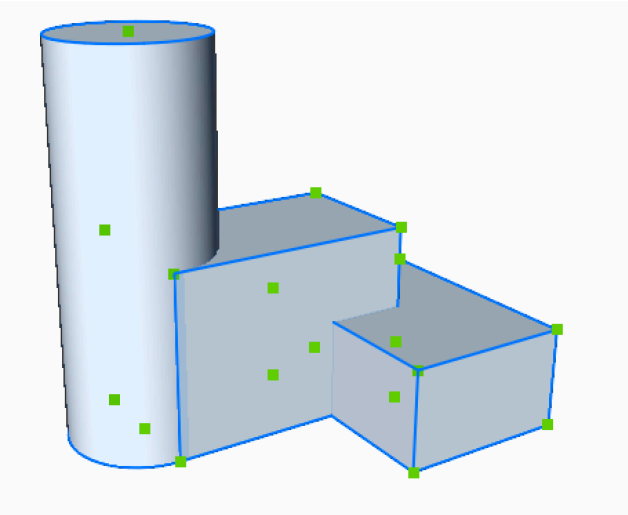
### 20.3.14.3 Procedure: top-down workflow

1 Draw one or more solids in your model space.

***Прим.:*** The solids don't need to be unioned, the command will do it automatically.



2 Type the command BIMQUICKBUILDING in the Command line or select from the Quad, select the solids and press Enter.

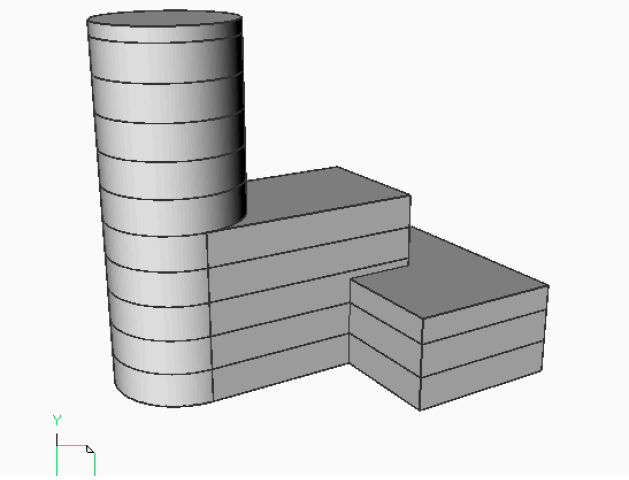


3 You are prompted: Do the selected solids represent [Buildings/Spaces]: Type B to continue with the top-down workflow and press Enter.

4 You are prompted: Enter desired story height (mm) or [BuildingRequirementFile/Spatialallocations]:

5 Enter the desired story height.

Press Enter and the building will be generated.



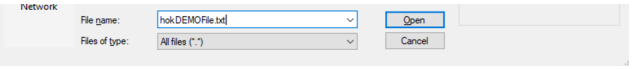
***Прим.:*** To generate different floor heights, type (2x4000,*3000,1x5000) which stands for two stories of 4000mm, the rest 3000mm and the top floor has a story height of 5000mm.

**Or** import the story height values from a .txt file.

Enter BIMQUICKBUILDING in the Command line,

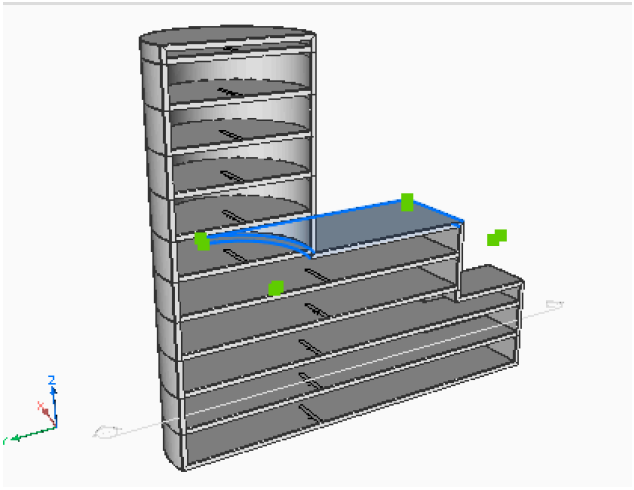
Type BUILDINGREQUIREMENTFILE and press Enter.

Then search for the text file in the file folder and click open.




Now the properties of the text file are applied to the building.
*Example of a text file with parameters for your building:*
*Space:Name='Bricsys Tower',Area=15000*
*Space:Name='Level 0 - Reception',Area=6%,'Floor-to-Floor Height'=5000*
*Space:Name='Level 1-10',Area=5%,'Floor-to-Floor Height'=4000,Count=10*
*Space:Name='Level 11-17',Area=4%,'Floor-to-Floor Height'=3000,Count=7*
*Space:Name='Level 18-19',Area=4%,'Floor-to-Floor Height'=4000,Count=2*
*Space:Name='Level 20-21',Area=1.5%,'Floor-to-Floor Height'=4000,Count=2*
The area and area ratios that are specified in the file are requirements. If the result of **Quickbuilding** does not satisfy the required areas this will be printed in the Command line, see also the section about Command Line Output later in this article.

6 If you make a section, you can see the newly formed slabs and walls which each have individual BIM properties.

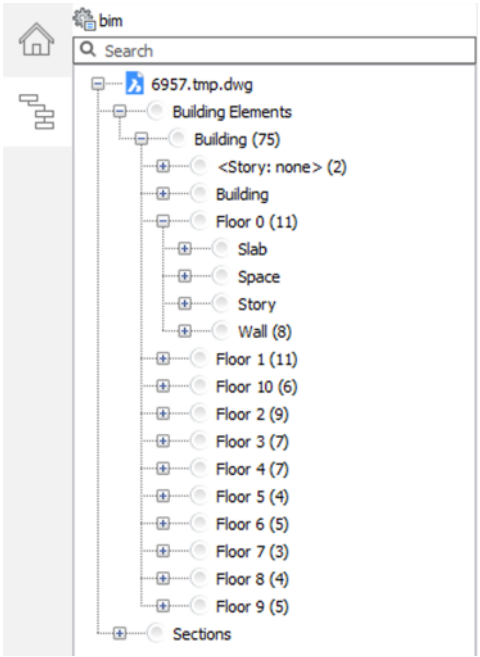


Select the roof and navigate to the **Properties** Panel - **BIM** tab. You will see that this entity has **Roof type** property, as well as some other properties such as spatial locations, quantities or a **GUID**.

| BIM | |
|---|---|
| Type | Roof |
| Building Element | Roof |
| Name | |
| Description | |
| Building | Building |
| Story | Floor 5 |
| Composition | |
| GUID | 15iQzJUHr8rxP2hWT$pWj7 |
| Roof type | Not defined |
| Space bounding | On |
| Number | |
| Entity property s | |

***Прим.:*** All the elements are categorized, in the **Structure Browser** you will see a list of building elements, sorted by floors and building element.

The different buildings elements will be placed in different layers according to their building type (External wall/slab, Flat/pitched roof, Internal wall/slab).

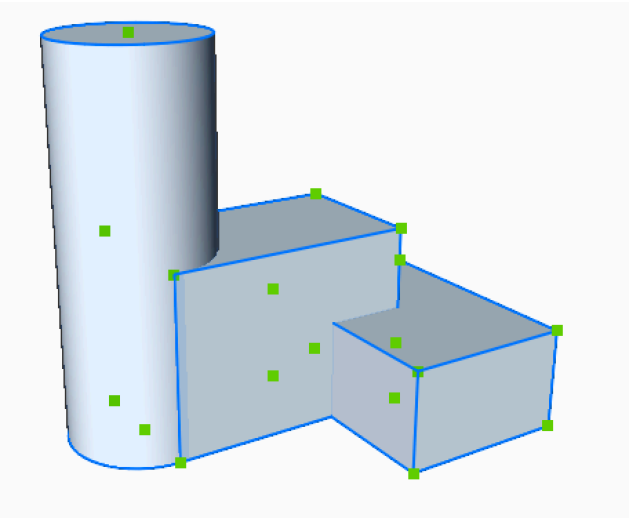


### 20.3.14.4 Procedure: bottom-up workflow

The bottom-up workflow creates a building out of solids that represent spaces. This approach allows you to first shape the spaces, then use **QuickBuilding** to create the shell and the new spaces.

1 Type QuickBuilding in the Command line.

2 Select the solids and press Enter.



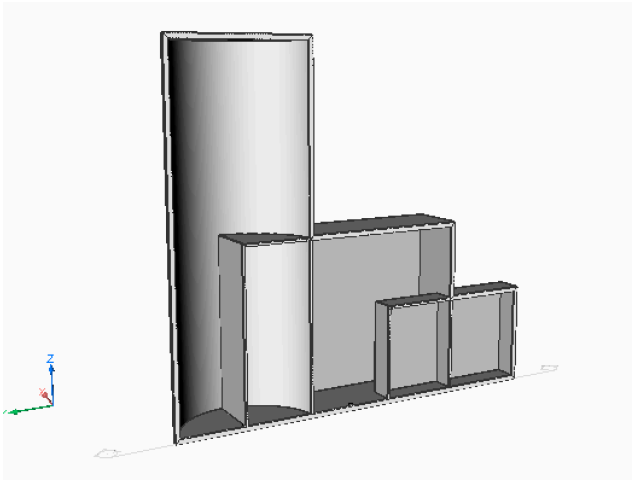
3 You are prompted: Do the selected solids represent [Buildings/Spaces]:

4 Type Spaces to create a building from the selected massing solids.

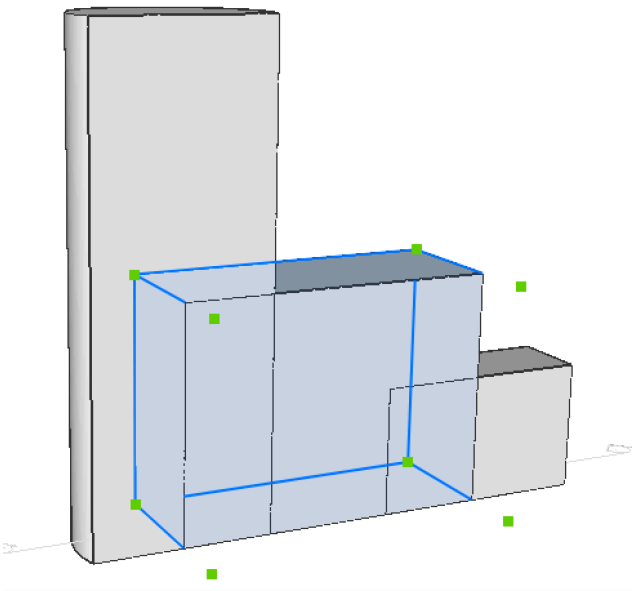
***Прим.:*** BricsCAD takes these defined solids and converts them to BIM Spaces with walls and slabs.

5 Press Enter to create the building.

6 (Optional) make a section to see the newly created spaces.



***Прим.:*** The original solids are kept, **Quickbuilding** just creates a Building from it with the information that these input solids represent spaces, so it knows that inner walls and slabs should be created based on those input spaces.

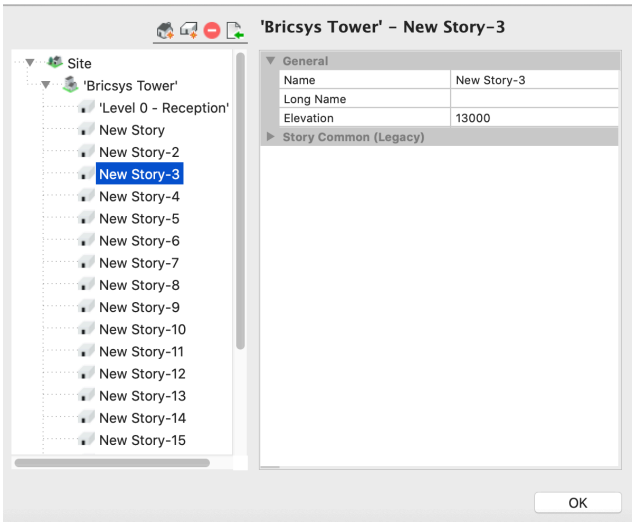


### 20.3.14.5 Procedure: SPATIALLOCATIONS as input

It is also possible to use a Spatial Structure as input for **Quickbuilding**. So, you can import a Spatial Hierarchy and use that data to subdivide the building.

1 Import or Edit the SPATIALLOCATIONS.

1 For more information about this command, please visit the help center article SPATIALLOCATIONS.




2 Make sure that the solid(s) you wish to use for **Quickbuilding** are:
- classified as a Building (under **Spatial Structure Elements** in **BimClassify**), and
- are assigned the correct 'building' in its BIM properties.

3 Type QUICKBUILDING in the Command line.

4 Select the building and press Enter.

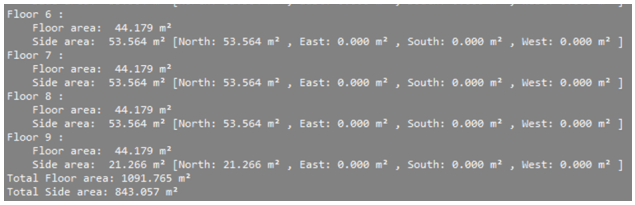
5 You are prompted: Enter desired story height (mm) or [BuildingRequirementFile/Spatiallocations]:

6 Type SPATIALLOCATIONS to use the Spatial Structure as input.

***Прим.:*** If you add more than one building in the Spatial Locations manager and then assign the building to the solids, **QuickBuilding** will use the different story heights on the different buildings.

### 20.3.14.6 Command line output

After you run the BIMQUICKBUILDING command, BricsCAD will create an overview with information on some basic building information. For instance, the total floor area of the building or other information for each floor.




***Прим.:*** If you used a Building Requirement File, and some requirements are not passed (e.g. not enough total area or floor area), you can see the problems in the output.

```
Total Floor area: 330.990 m² Requirement NOT passed, area should be >= 15000.000 m² m²
Total Side area: 1001.571 m²
Floor to Side ratio: 33.047 %
```

## 20.3.15 MEP modeling

### 20.3.15.1 Commands

BIMAPPLYPROFILE, BIMLINEARSOLID, BIMFLOWCONNECT, DRAG, BIMSTRETCH

### 20.3.15.2 About BIMFLOWCONNECT

It's also possible to connect different segments using the BIMFLOWCONNECT command.

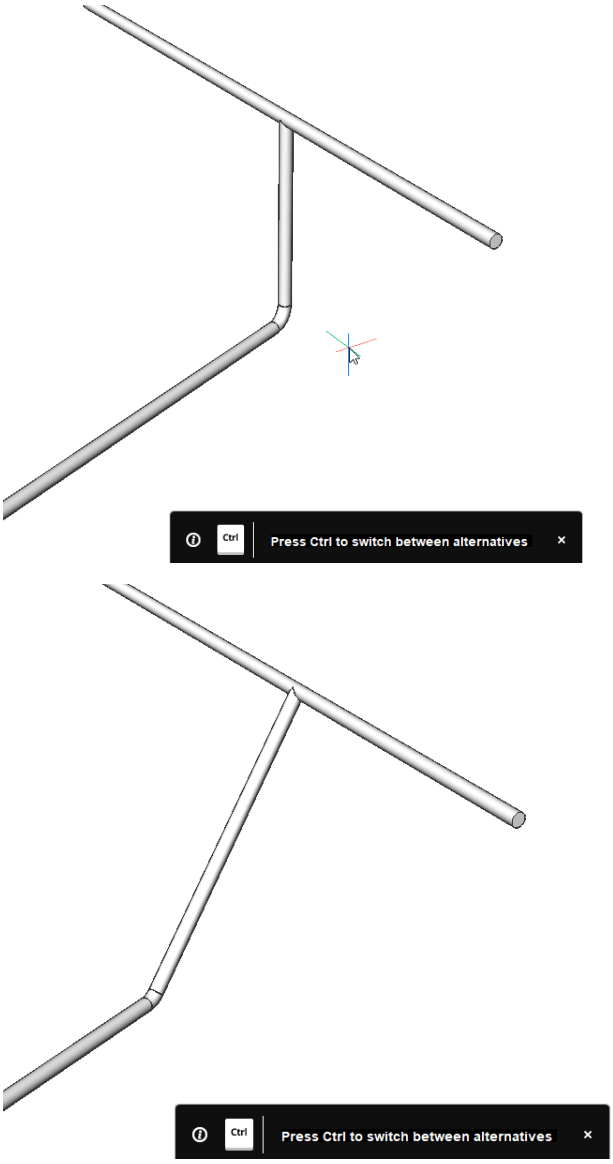


For more information about this command, visit the Command Reference article BIMFLOWCONNECT.

**Connecting different segments to each other**

Select the segments you want to connect and launch the BIMFLOWCONNECT command.

When the Hot Key Assistant is switched on (**HKA** on the status bar), the following widget appears on the screen press the CTRL-key to cycle through the different options.




End the command by pressing Enter, the segments are now connected.

### 20.3.15.3 Procedure: altering the position of segments

Before starting with the procedures: enable the selection of sides and ends:



Click the icon on the **Selection Modes** toolbar or set the DISPLAYSIDESANDENDS system variable ON.

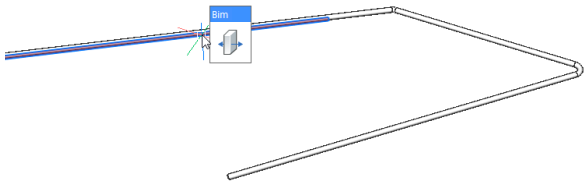
**DRAG**

The DRAG command makes it possible to alter a segment of a drawn profile without losing its connection to the other segments.



For more information about this command, visit the Command Reference article DRAG.

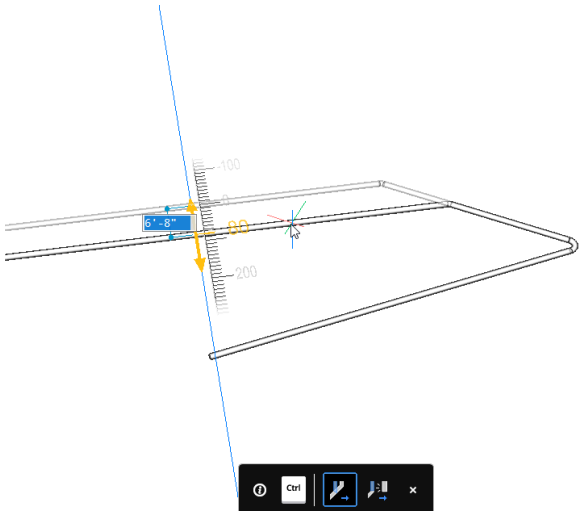
1 Hover over a segment, select a face perpendicular to the section of the profile.



2 Launch the DRAG command.

3 Choose a new position for the segment or use the dynamic dimension field to enter a value for the displacement.

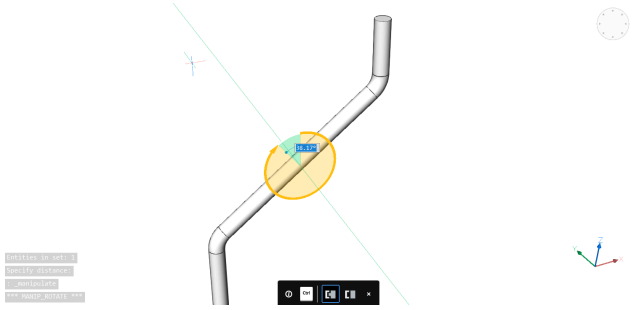
***Прим.:*** The software automatically regenerates the connection between the segments.



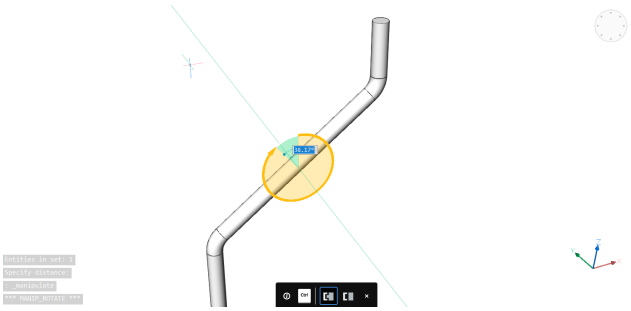
**Manipulator**

1 Hover over a segment, select a face perpendicular to the section of the profile.

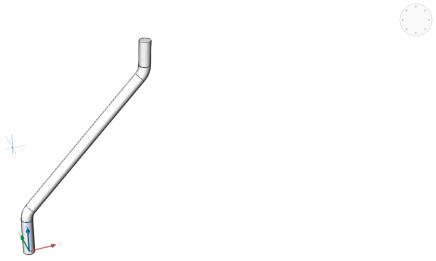
2 Select **Manipulate**.



3 Choose a new position for the segment or use the dynamic dimension field to enter a value for the rotation.

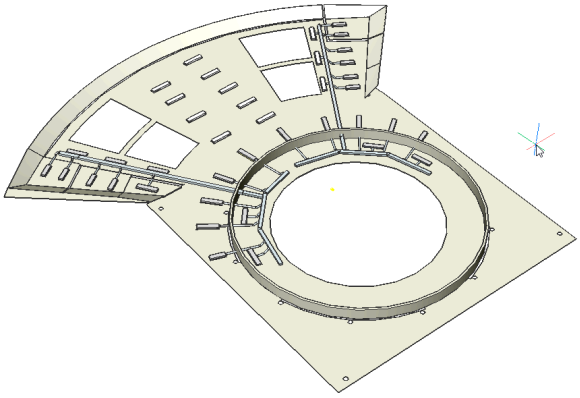


4 Click on the new position or enter the rotation angle and hit enter.



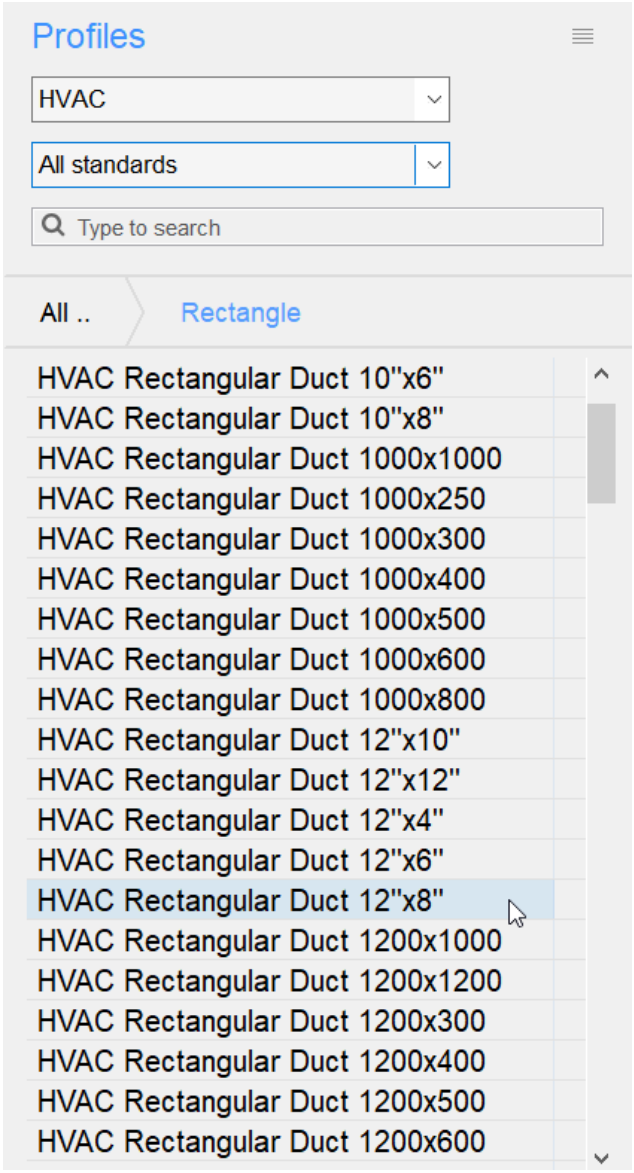
### 20.3.15.4 Procedure: using the MEP modeling tools to draw and connect profiles

1 Open the drawing that contains the MEP model. You can download this **MEP.zip**-file containing the drawings at the end of this article.



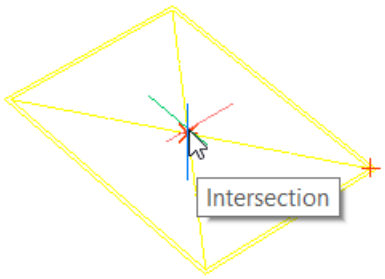
2 Open the BIM **Profiles** panel on the right-hand side of your screen (see Dockable Panels).

Filter on **HVAC** and select the Rectangular 12"x 8" profile, drag it to the model space.




3 Define the starting point of the profile.

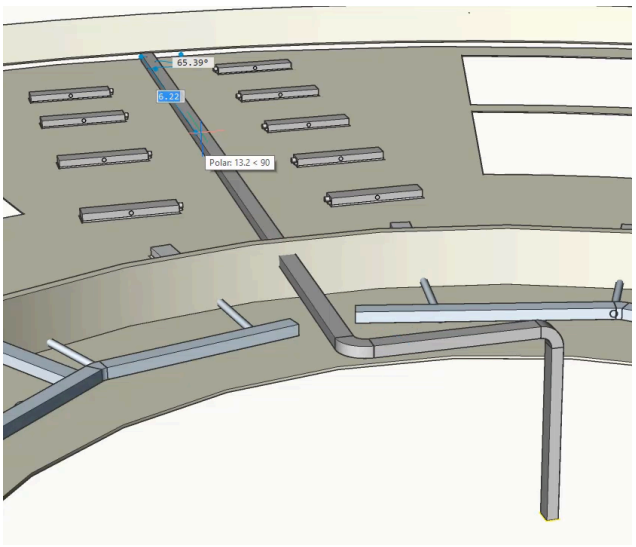
4 Snap to the geometric center of the yellow rectangle.




5 Turn **Ortho** on and define the next points of the profiles by using the dynamic dimension field.

Quarter turns the profile by entering Q in the Command line.

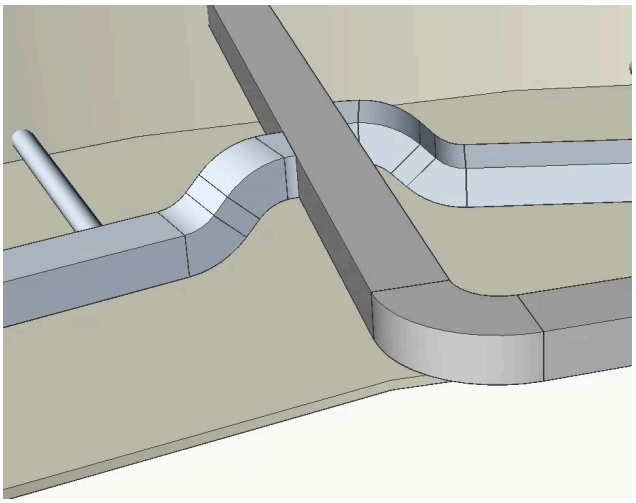
Go up 3.5, go 3.5 to the left and go 21 horizontally.




***Прим.:*** If you had chosen to draw with a non-HVAC profile, you will not get the automatic classification as **Flow Segment**. You can manually classify the profiles through the BIMCLASSIFY command as **Building Service Elements** - **Flow Segments**.

6 Connect the other rectangular ducts on the side to the one we have just created.

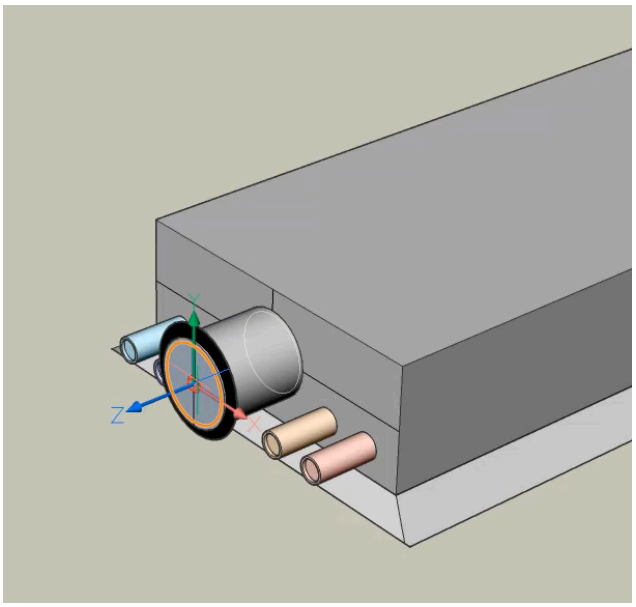
7 Select the ducts together with the main duct and launch the BIMFLOWCONNECT command. Your connection should look like the image below.



8 Create a duct from the flow terminal.

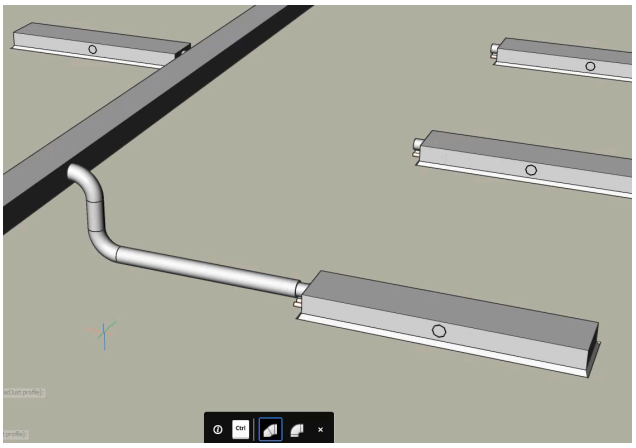
9 Launch the BIMLINEARSOLID command and select the flow connection point.

This connection point contains profile information, so the software automatically creates the predefined circular duct. Give the duct a length of 1.



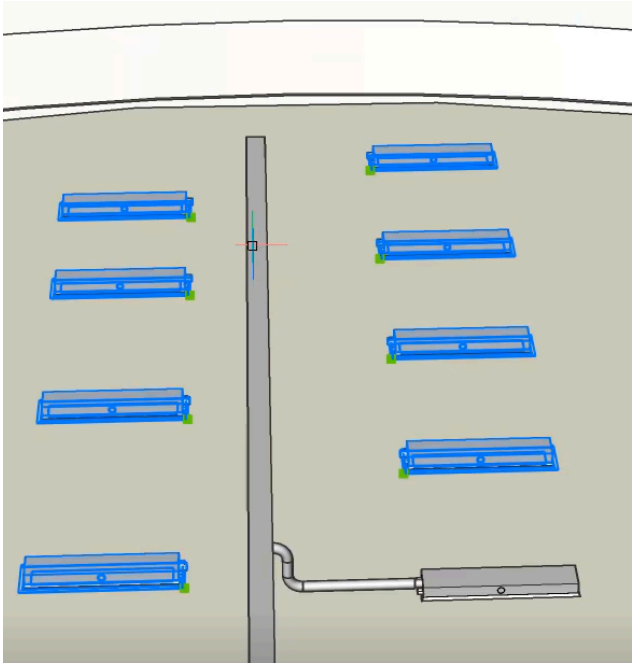
10 Connect the ducts from the flow terminal to the rectangular main duct.

11 Select the main duct and the duct from the flow terminal and launch the BIMFLOWCONNECT command.



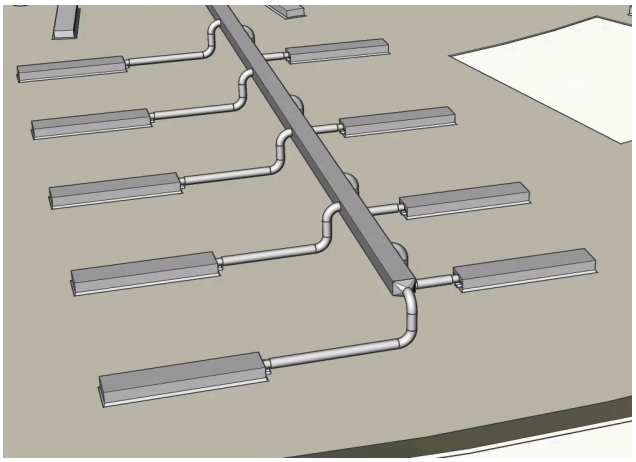
12 Connect all the other flow terminals to the rectangular duct at once.

13 Select the other flow terminals and the rectangular duct and launch the BIMFLOWCONNECT command.



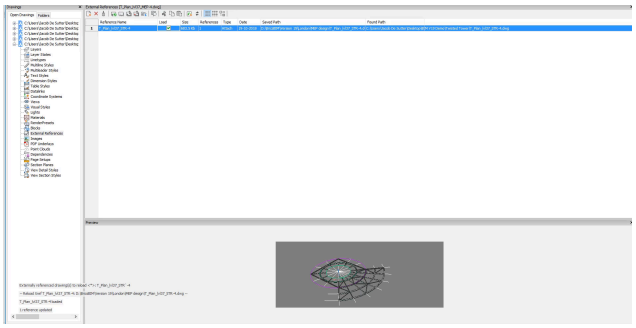
14 BricsCAD uses the information from the flow connection point to determine the type of duct used to connect with the main duct.

***Прим.:*** The flow terminal on the back is connected to the back of the rectangular duct.



### 20.3.15.5 Procedure: Using the MEP Modeling tools to alter profiles

1 Type XRef in the Command line and check the load box next to the structural model to load the structural drawing.



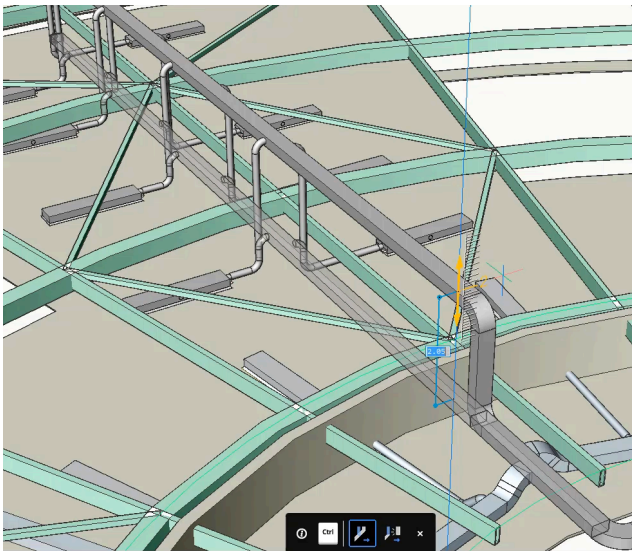
2 Use SLICE command and **Multislice** option to divide the rectangular duct.

Enter 3 in the dynamic dimensions field and press Enter to end the command.

3 Alter the positioning of the rectangular duct, so it doesn't interfere with the structural model.

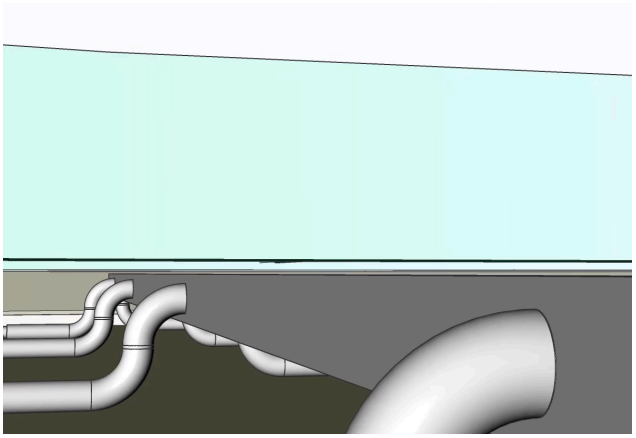
4 Select the upper face of the duct and launch the DRAG command.

***Прим.:*** BricsCAD preserves the connections with the circular ducts. You can change this by pressing the CTRL-key if you want to disconnect the ducts.



Move the duct down by 0.23.

5 The structural model no longer interferes with the MEP Model, while the connections between the ducts of the flow terminals and the main rectangular duct were preserved.



### 20.3.15.6 Procedure: using Bimstretch to modify flow segments

1 Open a New drawing.

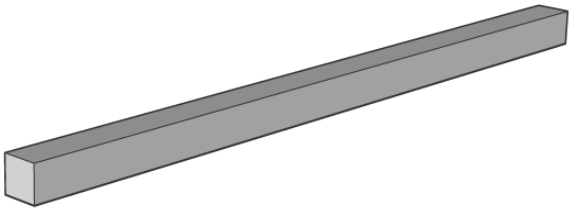
2 Open the BIM **Profiles** panel by clicking on it on the right-hand side of your screen.

***Прим.:*** If the BIM Profile panel icon isn't visible yet, right-click on the ribbon and check **Profiles** in the context menu under.

3 Filter on the **HVAC** standard. These profiles will be automatically classified as flow segments once they are dragged into the model space.

4 Search for the profile **HVAC Rectangular Duct 10"x10"** in the **Profiles** panel and drag the profile in the model space.

5 Select a starting point and draw a profile. Press Enter to stop drawing with this profile. Your model should look like this:

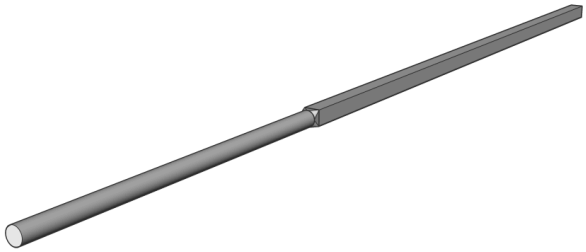


6 Search for the profile **HVAC Round Duct 8"** in the **Profiles** Panel and drag the profile in the model space.

7 Click on the end of the Rectangular profile, after hovering on its profile face.

***Прим.:*** To easily snap on the end of the rectangular profile, turn on the **Display Axes** setting or set the DISPLAYAXES system variable to 1.

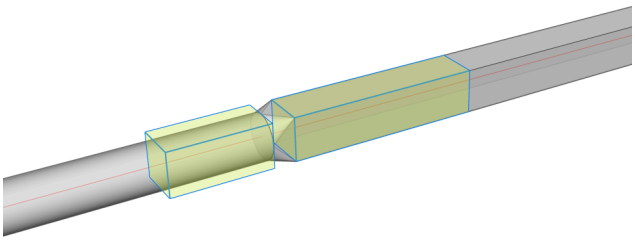
8 Make sure to draw the circular profile along the same axis as the rectangular one. After accepting the last point of the circular profile, it will be connected to the rectangular profile through an automatically generated reducer.



***Прим.:*** If you had chosen to draw with a non-HVAC profile, you will not get the automatic classification as Flow Segment and the connection will not be made immediately. You can still manually classify the profiles through the BIMCLASSIFY command and then use BIMFLOWCONNECT to connect the two through a reducer.

***Прим.:*** Make sure the **Display Sides and Ends** and **Display Axes** is enabled.

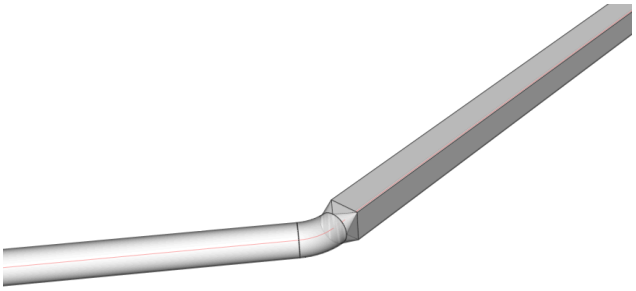
9 Select the connecting ends of both segments.



10 Launch the **BimStretch** tool.

11 Hover your cursor to one of the sides and click on the desired location. BricsCAD automatically stretches the 2 Flow Segments in their new position and adds an extra curved Flow Fitting in the

middle.



Attach **MEP example file.zip**
2 MB Download

### 20.3.16 MEP flow connection points

#### 20.3.16.1 Commands

BIMLINEARSOLID, BIMFLOWCONNECT

#### 20.3.16.2 About

When creating flow elements like Flow Terminals and Flow Segments, you can add a 2D symbol and classify it as a **Flow Connection Point**. A flow connection point contains BIM data where you can predefine the profile you want to use with this flow component. When you select the flow connection point and use the commands BIMLINEARSOLID or BIMFLOWCONNECT, BricsCAD will automatically use a BIM Profile based on the given information.

See the related links for more information about Linear Solids.

See the related links for more information about MEP modeling.

#### 20.3.16.3 Procedure: creating a flow connection point

In this procedure, you will learn how to create your own HVAC components with flow connection points. As an example, we will create an air heat pump unit.

**Creating a flow connection point in the Library panel**

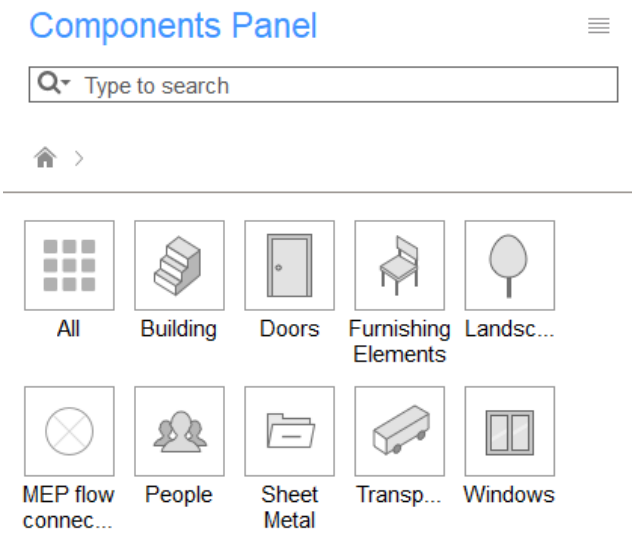
1 Draw a box with the desired dimensions (80x40x80 cm) representing the heat pump.

2 Select the box and launch BIMCLASSIFY with the <Other> option or hover over the box and select **Classify Manually** in the **BIM** tab of the Quad.

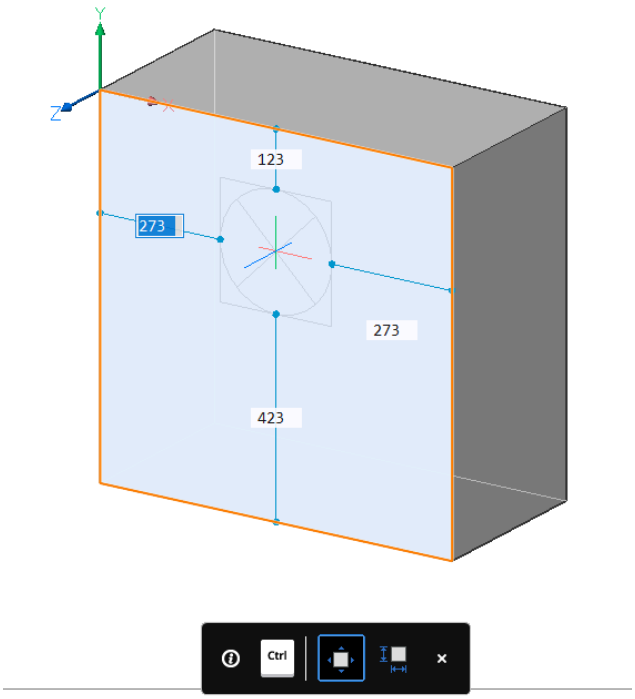
3 Classify the box as **Building Service Elements** > **Flow Terminal** and click **OK**.

4 Open the **Library** panel.

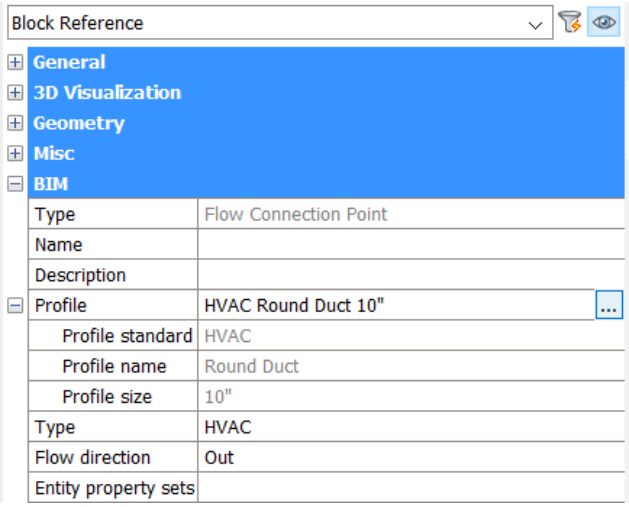
5 Go to the **MEP Flow Connection Points**.




6 Place HVAC Outlets Round on the heat pump box.




7 Select the Outlet and click the **BIM**/**Profile** property in the **Properties** panel.




8 Click the browse (…) button.

The **Profiles** dialog box displays.

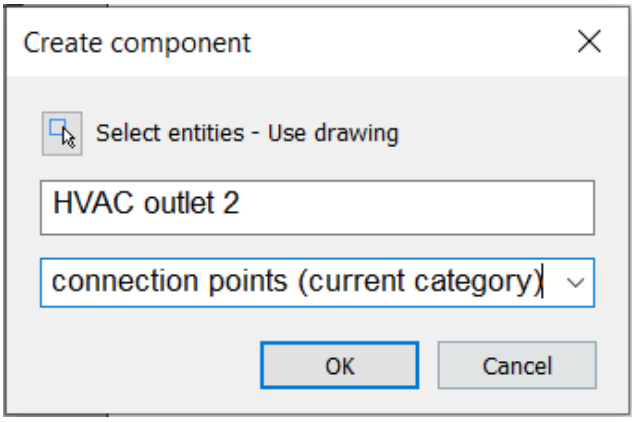
9 Set **Type** and **Standard** = **HVAC**.

10 Choose the desired profile and click the **Select** button.

11 Go back to the **Library** panel and click **Create Component**.

12 Choose a name and category and click the **OK** button.

See the **Components Panel** article for more information about adding a component to the library.




The drawing is now saved as a component and can be used in other drawings.

**Creating a flow connection point from scratch**

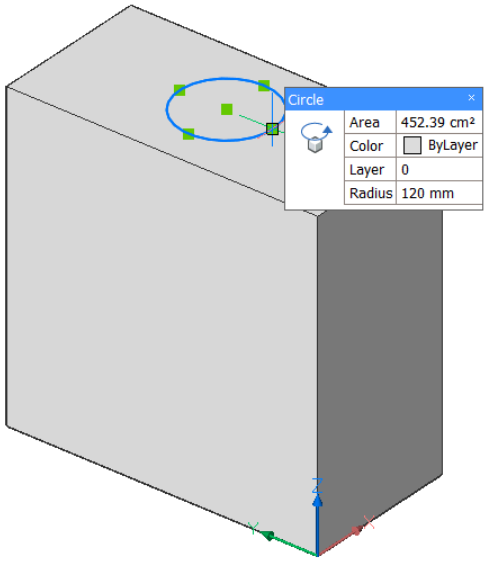
1 Draw a box with the desired dimensions (80x40x80 cm) representing the heat pump.

2 Select the box and do one of the following:

- Launch BIMCLASSIFY with the **Other** option.
- Hover over the box and select **Classify Manually** in the **BIM** tab of the Quad.

3 Classify the box as **Building Service Elements** > **Flow Terminal** and click **OK**.

4 Draw a circle on the top face of the box.

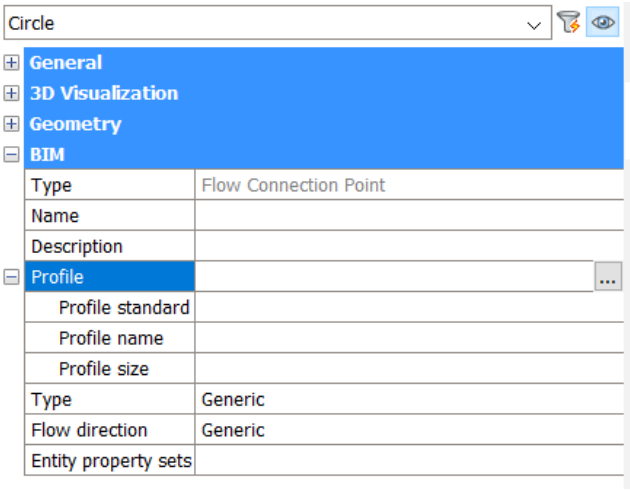



5 Select the circle and launch BIMCLASSIFY with the **Other** option.

6 Classify the circle as **Building Service Elements** > **Flow Terminal** and click **OK**.

7 Select the circle.

8 In the **Properties** panel, select the **BIM**/**Profile** property and click the Browse (…) button.




The **Profiles** dialog displays.

9 Select the desired profile and click **Select**.

10 On the **Library** panel and click **Create Component**.

11 Choose a name and category and click **OK**.

The drawing is now saved as a component and can be used in other drawings.

### 20.3.16.4 Procedure: Creating an HVAC System

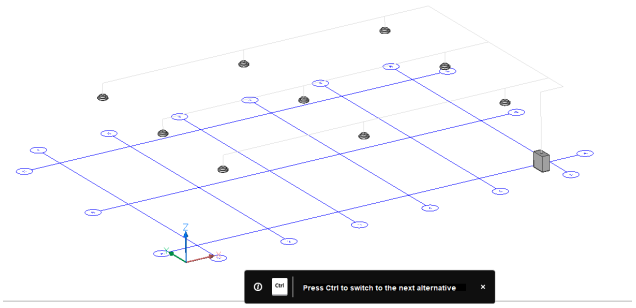
In this procedure, we will use flow connection points to create a ventilation system.

1 Let's start from a file with a series of flow terminals and a ventilation unit.

2 Select the flow terminals and the ventilation unit.

3 Select **FlowConnect** in the Quad or in the **Structural**/**MEP** ribbon tab.

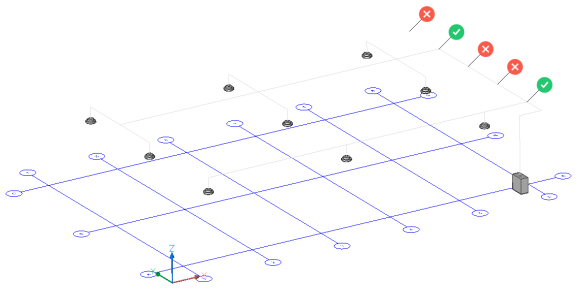
4 Press the CTRL-key to switch between the alternative setups and press Enter.




5 Click the **Tune** option or enter T in the Command line to modify the chosen setup.

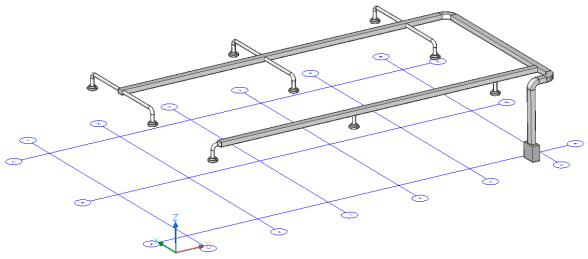
6 Select **Topology** in the toolbar or enter T and press Enter.

7 Select the check icons to modify the layout of the system.



8 Press Enter 3 times to exit the command.

9 ***Прим.:*** BricsCAD automatically uses the profiles that were assigned to the ventilation units. This information is stored in the **Flow Connection Points**.



## 20.3.17 Offset

### 20.3.17.1 Commands

OFFSET

### 20.3.17.2 About

The **Offset** tool allows you to create copies of linear entities and align them parallel to the original entities at a specified distance.



Depending on the selected options in the **Selection Modes** toolbar, the **Offset** tool allows to create a parallel copy of:

- An edge of a 3D solid if the **Select Edges** option ( ) is enabled or on or set the SELECTIONMODES system variable to 1.
- The edges of a 3D solid face if the **Select Faces** option ( ) is enabled on or set the SELECTIONMODES system variable to 2.
- Detected boundaries if the **Select Detected Boundaries** option ( ) is enabled on or set the SELECTIONMODES system variable to 4.

### 20.3.17.3 Procedure: make a parallel copy of a 3D solid edge

Before starting the process, make sure that Quad is active and the **Select Edges** option in the **Selection Modes** toolbar is on. Next, follow these steps:

1 Move the cursor over the edge of the solid to highlight the edge.

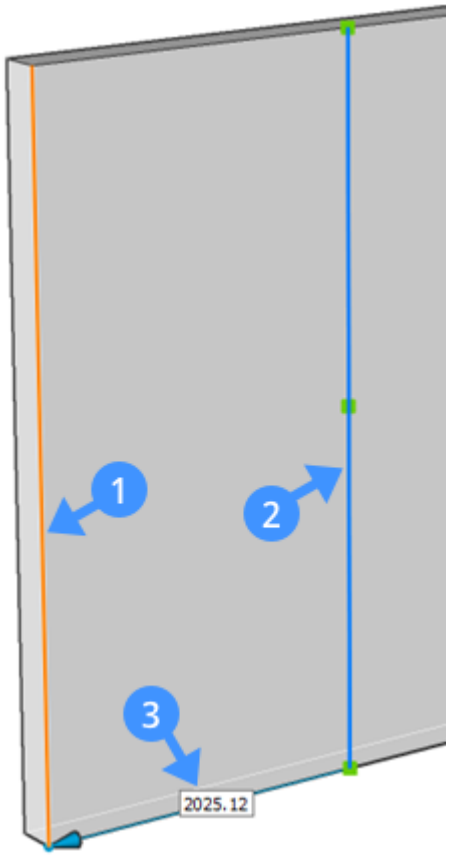
2 Choose the **Offset** tool from the Quad.

3 Specify the offset distance by dragging the edge of the solid face.

***Прим.:*** The distance of the offset displays in the dynamic dimension field. The parallel copy of the edge is created dynamically.

4 While dragging the edge to set a precise offset distance, type a distance in the dynamic entry field and press Enter or left-click to accept the offset.

5 The following illustration shows the offset edge on the 3D solid. The edge highlights in orange is the original edge (1) of the 3D solid, whereas the blue highlighted line with grips (2) is the offset copy of the original one. The line (3) between the original edge and copied edge gives the offset distance.




### 20.3.17.4 Procedure: Make a parallel copy of a 3D solid face

Before starting the process, make sure that Quad is active and the **Select Faces** option in the **Selection Modes** toolbar is on. Next, follow these steps:

1 Move the cursor over the face of the solid to highlight the solid face.

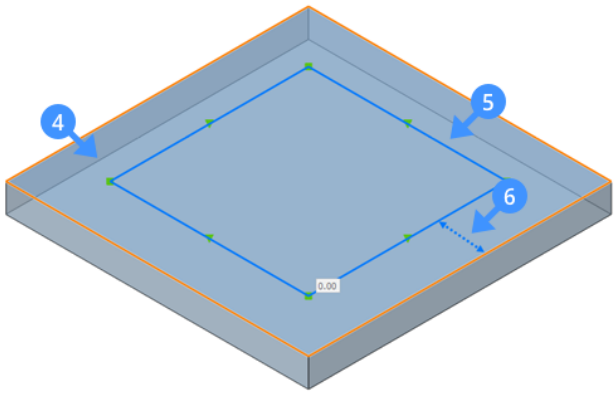
2 Choose the **Offset** tool from the Quad.

3 Specify the offset distance by moving the cursor outside or inside of the solid.

4 The parallel copy of the face is created dynamically.

5 Enter a distance in the dynamic entry field and press Enter, or left-click to finish the offset.

6 The following illustration shows the offset of the 3D solid face. The highlighted face is the original 3D solid face (4), whereas the created polyline (5) is the offset from the original one. The line (6) between the original 3D solid face and copied polyline gives the offset distance.




### 20.3.17.5 Procedure: make a parallel copy of a detected boundary

Before starting the process, make sure that Quad is active and the **Select Detected Boundaries** option in the **Selection Modes** toolbar is on. Next, follow these steps:

1 Move the cursor inside the boundary to highlight the detected boundary.

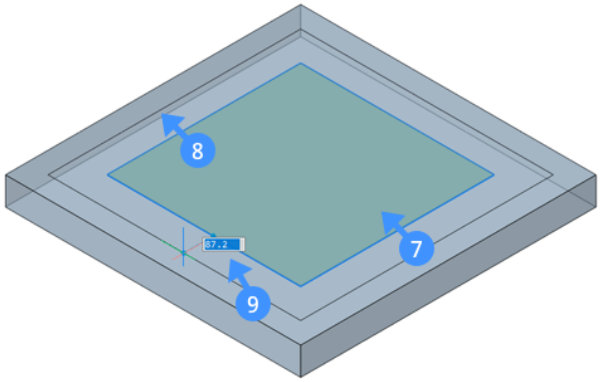
2 Choose the **Offset** tool from the Quad.

3 Move the cursor to define the offset distance and left-click.

4 The parallel copy of a detected boundary is created dynamically.

5 Enter a distance in the dynamic entry field to set a precise offset distance and press Enter or left-click to accept the offset.

6 The following illustration shows the offset boundary on the face of a 3D solid. The highlighted face is the detected boundary (7), whereas the created polyline (8) is the copy from the original one. The line (9) between the detected boundary and copied polyline gives the offset distance.




## 20.3.18 Splitting solids

### 20.3.18.1 Commands

BIMSPLIT

### 20.3.18.2 About

The BIMSPLIT command either splits segmented solids into separated solids automatically or allows to split a solid using a selection of cutting faces.

### 20.3.18.3 To split a solid automatically

Make sure that the value of the SELECTIONPREVIEW system variable is 2 or 3.

1 a Do one of the following:
- Click the BimSplit tool button ( ) on the **Model** tab of the ribbon.
- Choose **Split** in the BIM menu.
- Type BIMSPLIT in the Command line, then press Enter.

You are prompted: Select solid for auto-splitting or cutting faces for manual splitting:

2 Place the cursor over a face of the solid.

3 The face highlights.

4 Without moving the mouse, hit the TAB key until the complete solid highlights, then click.

5 You are prompted:

Entities in set: 1

Select solid for auto-splitting or cutting faces for manual splitting:

6 Right click.

7 The selected solid is split.

The number of parts is reported in the command window.

#### Using the Quad

Make sure the **Select Faces** option ( ) of the SELECTIONMODES system variable is switched off.

Place the cursor over a face of the solid.

When the solid highlights, choose Split ( ) in the BIM command group of the Quad.

The solid is split.

The number of parts is reported in the command window.

### 20.3.18.4 To split a solid using cutting faces

Make sure that the value of SELECTIONPREVIEW system variable is 2 or 3.

1 Do one of the following:
- Click the **BimSplit** tool button on **Model** tab of the ribbon.
- Choose **Split** in the **BIM** menu.
- Type BIMSPLIT in the Command line, then press Enter.

You are prompted: Select solid for auto splitting or cutting faces for manual splitting:

2 Place the cursor over a face of the solid.

3 Hit the TAB key to highlight an obscured face.

When the face highlights, click to select the face.

You are prompted:

Entities in set: 1

Select solid for auto splitting or cutting faces for manual splitting:

4 (Optional) Repeat step 4 to select more faces.

5 Right-click to stop selecting faces.

6 The solid is split using the selected faces as cutting planes.

The number of parts is reported in the command window.

**Using the Quad**

Make sure the **Select Faces** option of the SELECTIONMODES system variable is set.

1 Place the cursor over a face you want to use as a cutting plane.

1 When the face highlights, do one of the following:

2 Go to step 3.

3 Go to step 2 to select more faces.

4 (Optional) Place the cursor over a face you want to use as a cutting plane.

5 When the face highlights, do one of the following:

6 Go to step 3.

7 Repeat this step to select more faces.

8 Choose **Split** in the **BIM** command group of the Quad.

The solid is split using the selected faces as cutting planes.

The number of parts is reported in the command window.

### 20.3.19 Stair tool

#### 20.3.19.1 Commands

BIMSTAIR

#### 20.3.19.2 About

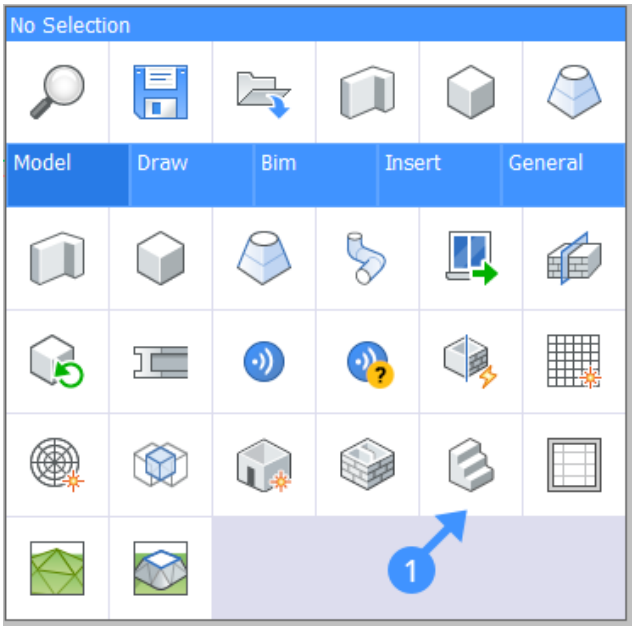
The **Stair** tool is a powerful tool that allows you to add a custom flight of stairs for your project in just two clicks. You can change the default stair settings as required. The **Stair** tool will automatically try to fit these requirements. It is possible to create single-flight straight, double-flight L-shaped, and double or triple-flight U-shaped stairs.



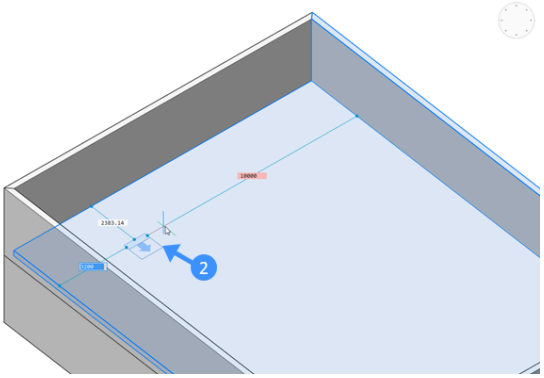
For more information about this command, visit the Command Reference article BIMSTAIR.

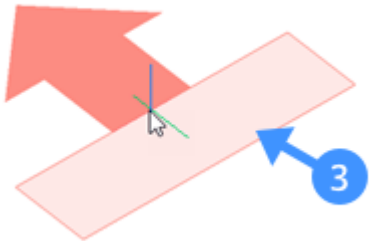
### 20.3.19.3 Create a straight flight of stairs

1 Select the **Stair** tool from the Quad (1).




2 Hover over a slab. The Stair Cursor will appear (2) and the slab on which the stair cursor hovers, is highlighted. This indicates the position of the first step. If the position of the cursor is invalid, the cursor will turn red (3).



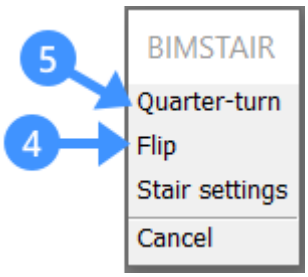



3 To flip the direction of the first step, click the **Flip** button (4) or type F.

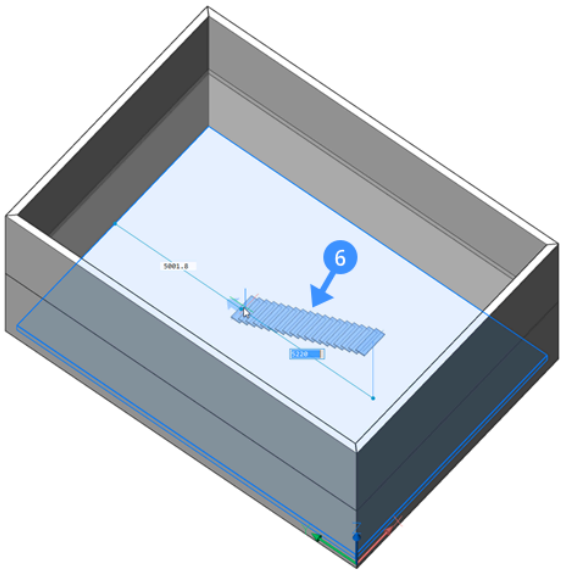
To turn the direction of the first step 90°, click the **Quarter-turn** button (5) or type Q.

To cycle through the different stair variants, the CTRL-key can be used when the Hotkey Assistant is active.

***Прим.:*** The cursor will automatically flip or quarter-turn when it is positioned close to a wall or the edge of a slab. This auto-quarter-turning/flipping behavior is only active when the relevant Stair Setting is turned on.

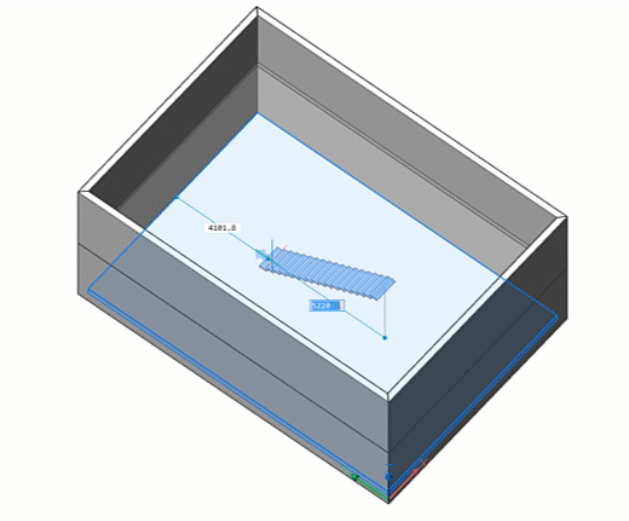



4 Once the cursor is in position, press Left-click. If the upper slab is blocking the view, the slab will become transparent.

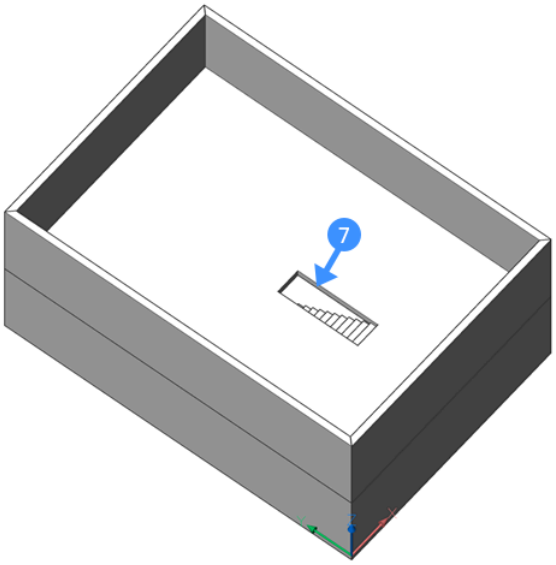



5 To create a straight flight of stairs, align the mouse perpendicular to the first step.

***Прим.:*** A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (6). The **Stair** tool will automatically snap to the configurations that correspond to "ideal step length" setting, as can be seen in the animation below. When the optimal configuration is satisfied, this preview will display in a darker shade of blue. The other settings are taken into account as well of course, but not to this specific snapping feature (where the stair preview turns darker blue).



6 Once you have the cursor in the correct position, left-click. A flight of stairs will be created. A hole is automatically generated in the upper slab, corresponding to the headroom setting (7).

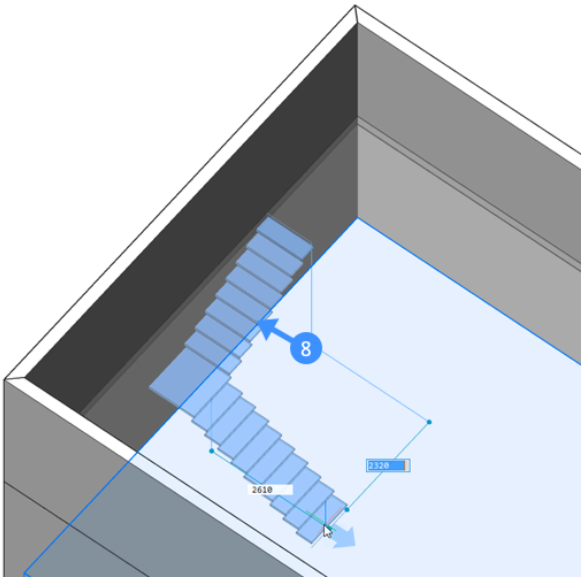



### 20.3.19.4 Create an L-shaped staircase

1 Select the **Stair** tool from the Quad.

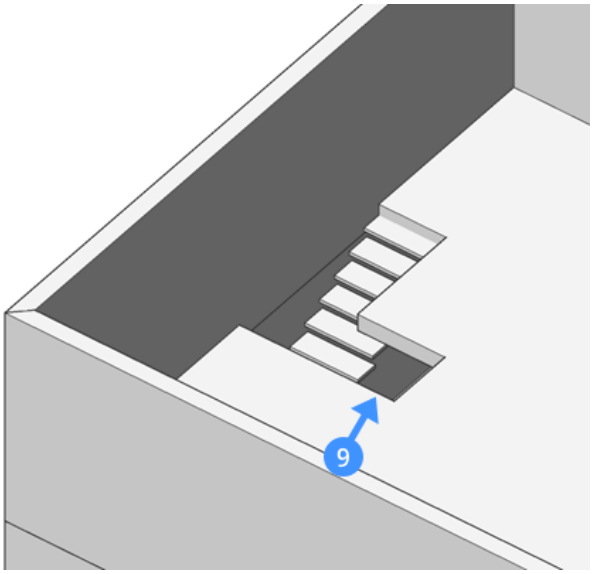
2 Hover over a slab. The Stair Cursor will appear. Left-click once the stair cursor is in the correct position.

3 Move the cursor away from the perpendicular to create an L-shape flight of stairs (8).




***Прим.:*** A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (8). The **Stair** tool will automatically try to find optimal configurations of the stair that correspond with the dimensions given in the settings dialog. It will snap to the optimal configuration and display this preview in a slightly darker color.

4 When you are satisfied with the preview, left-click. A flight of stairs will be created and a hole will be automatically generated (9). This time, the hole may be L-shaped, depending on the length of the upper flight and the "Headroom" stair setting. If the headroom is so large it cannot be covered with only the topmost stair flight, it will span also a part of the second stair flight and thus be L-shaped.




### 20.3.19.5 Create a U-shaped staircase

1 Select the **Stair** tool from the Quad.

2 Hover over a slab. The Stair Cursor will appear. Left-click when the stair cursor is in the correct

position.

3 Move the cursor away. You can type Q and Enter so that the direction of the second cursor is rotated 180° with respect to the first cursor.

**Or** use the CTRL-key when the Hot Key Assistant (**HKA** field in the **Status** bar) is ON.

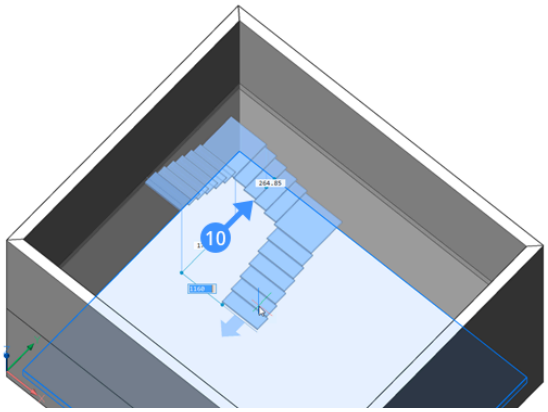
1 Select the **Stair** tool from the Quad.

2 Hover over a slab. The Stair Cursor will appear. Left-click when the stair cursor is in the correct position.

3 Move the cursor away from the perpendicular to get an L-shaped stair.

4 Press the CTRL-key for switching between an L-shaped stair and a U-shaped stair.

5 **Or**align the last step with the first fixed step. When entering a strip parallel to the first step, the last step will automatically quarter turn.



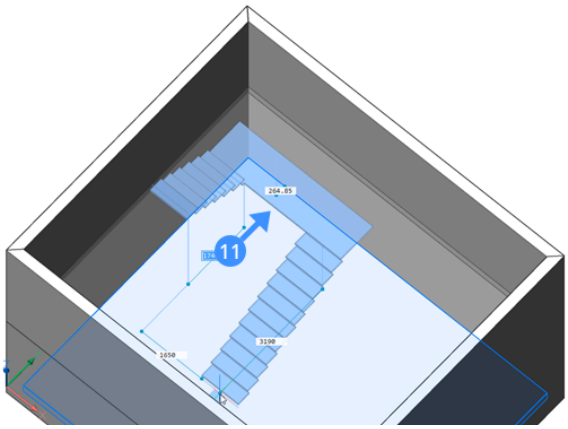

***Прим.:*** A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (10). The **Stair** tool will automatically snap to the configurations that correspond to the current settings. When the optimal configuration is satisfied, this preview will display in a darker shade of blue.

6 There are two possible types of U-shaped stairs you can switch between:

- The middle section includes steps (10).
- The middle section is a landing in between the two opposite stair flights (11).

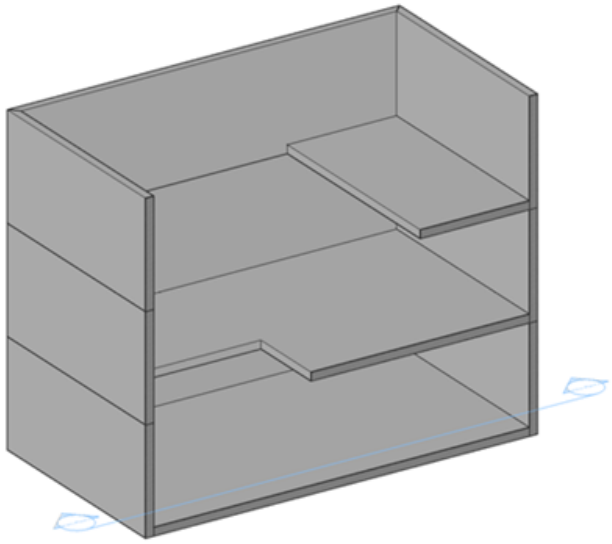
It is possible to switch between these two modes by typing U and hitting Enter.

**Or**press CTRL to cycle between different stair variants.




### 20.3.19.6 Create a flight of stairs in an existing opening

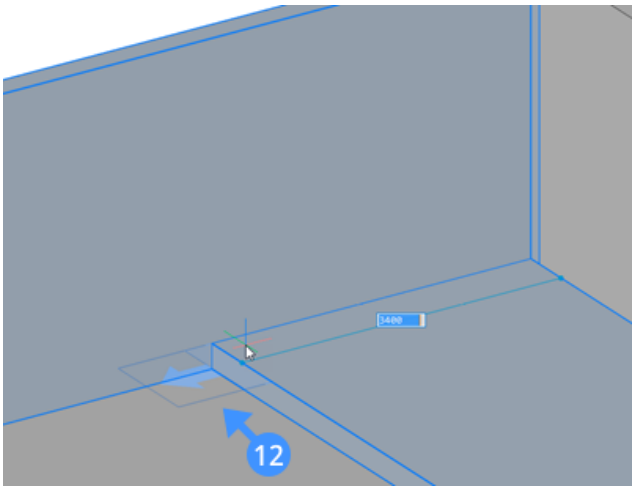
Sometimes it is necessary to create a flight of stairs starting from the edge of a slab, where an opening already exists.



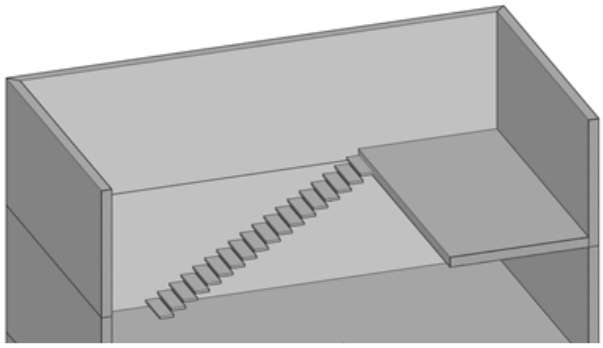
In the example above, different floors need to be connected with flights of stairs. However, an opening does not need to be created.

1 Select the **Stair** tool from the Quad.

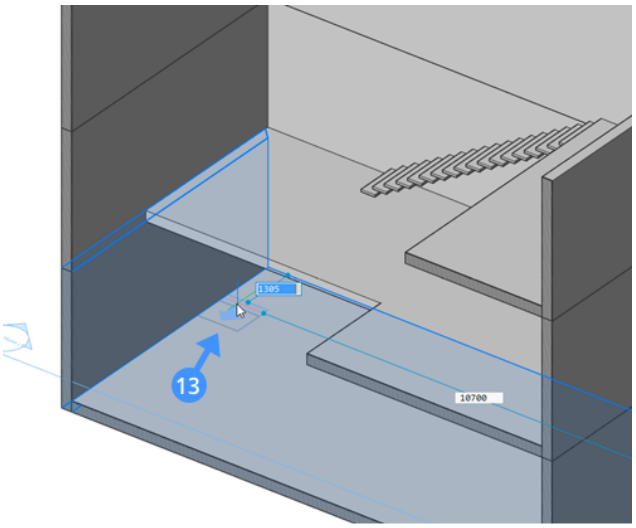
2 Hover the Stair Cursor near the edge of the slab (12). The cursor automatically snaps to the outer side of the edge.



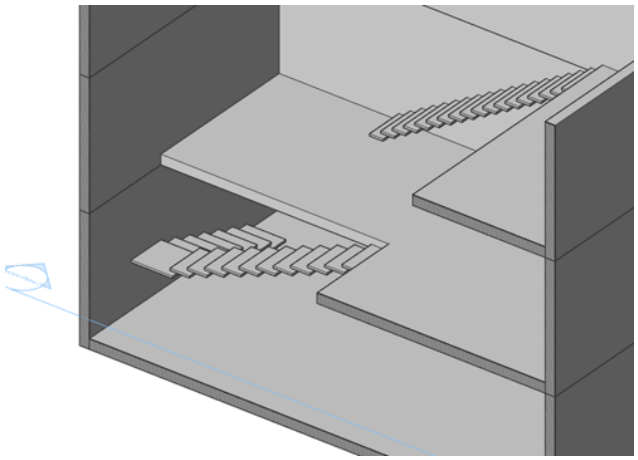

3 You can now create a flight of stairs as described above.



4 You can also make a flight of stairs by starting at the lowest point. This time select a starting point on the first floor (13).




5 Now hover over the edge of the slab on the second floor. The Stair Cursor automatically snaps to the edge of this slab. Left-click to create the flight of stairs.



### 20.3.19.7 Change the Stair Tool Settings

You can change the stair settings by clicking the **Stair settings** button in the **Stair** tool menu after the **Stair** tool command is launched or simply open the **Settings** dialog and search 'Stair'.

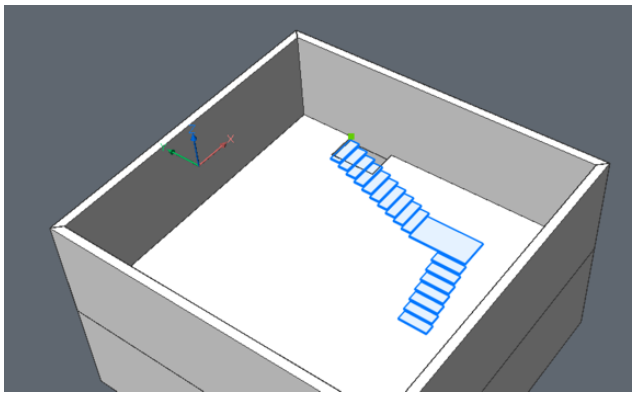
**Edit a flight of stairs**

You can adjust multiple parameters of a flight of stairs created with the **Stair** tool in the **Properties** panel. These parameters include:

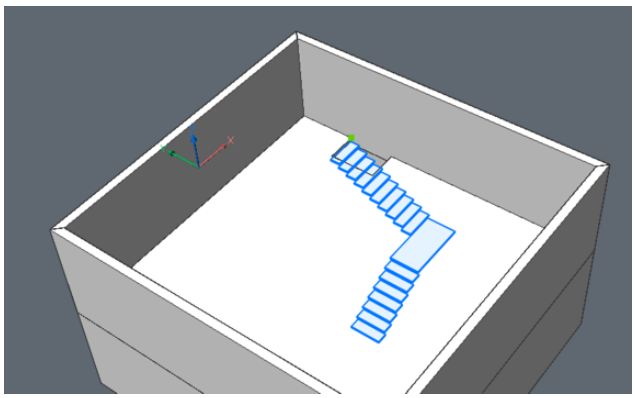
| Parameters | |
|---|---|
| Height_Staircase | 325 cm |
| Stair_Width | 1000 mm |
| Ideal_Riser_Height | 170 mm |
| Step_Thickness | 50 mm |
| Nosing | 50 mm |
| Length_StairFlight_0 | 145 cm |
| Length_StairFlight_1 | 174 cm |
| Length_StairFlight_2 | 145 cm |

| | |
|---|---|
| **Height_Staircase** | The overall height of the flight of stairs. The height is fixed at the top and will adjust from the base. |
| **Stair_Width** | The width of the flight of stairs. When a flight of stairs is selected a green dot appears on the upper slab. This dot indicates the insertion point of the flight of stairs. When this point is in the middle of the flight of stairs, the flight of stairs will widen from the center. When it is located at the side, for example when a stair is attached to a wall, the stair will only widen the side away from the dot. |

| | |
|---|---|
| **Ideal_Riser_Height** | The ideal riser height. When creating a flight of stairs, the real riser height will be calculated based on the ideal riser height and the height of the steps, so that the risers are equally divided as close to the ideal riser height as possible. |
| **Thickness_Step** | The step thickness. The upper face of the steps is fixed so that the thickness increases downwards. |
| **Nosing** | The horizontal distance of overlap between subsequent treads. |
| **Length_StairFlight_0** | The length of the first flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs. |
| **Length_StairFlight_1** | The length of an optional, second flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs. This option is only available for L-stairs and U-stairs. |
| **Length_StairFlight_2** | The length of an optional third flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs. This option is only available for U-stairs. |
| **Landing_Extension_Up** | The length of the landing going down the first flight of stairs. The first flight is fixed so the second flight adjusts to the given parameter value. |
| **Landing_Extension_Down** | The length of the landing going down the second flight of stairs. The first flight is fixed so the second flight adjusts to the given value. |



Setting: **Landing_Extension _Up**.



Setting: **Landing_Extension_Down**.

You can re-position a flight of stairs created with the **Stair** tool using the MOVE command.

For complex edits to a flight of stairs created with the **Stair** tool, use the BEDIT command to edit the block definition. When in the BEdit-mode navigate to the **Mechanical Browser** and here you can see and edit all the constraints that apply to a flight of stairs.

### 20.3.20 Structural steel

#### 20.3.20.1 Commands

BIMAPPLYPROFILE, BIMLINEARSOLID, BIMSTRUCTURALCONNECT, BIMADDECCENTRICITY, BIMRECALCULATEAXIS, DRAG, BIMSTRETCH

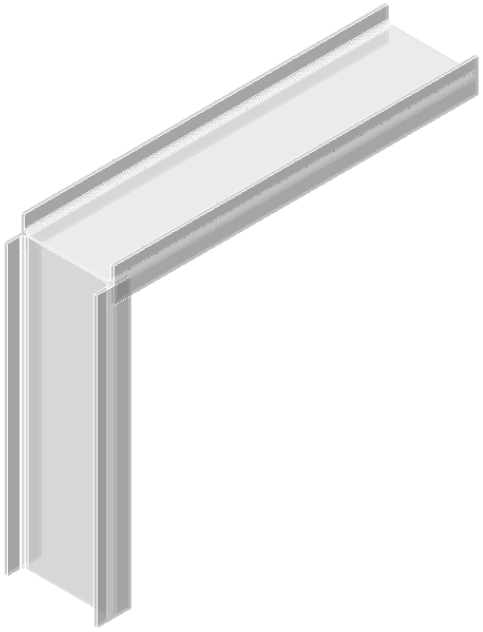
#### 20.3.20.2 How to use the BIM Profiles panel

1 Drag and drop the desired profile from the BIM **Profiles** panel to the model space.

***Прим.:*** This command works in the same way as the LINE command.

2 To define the starting point and the following point(s) click in the model space or use the dynamic dimension fields.

3 Press Enter to end the command.



***Прим.:*** BricsCAD doesn't automatically generate the connections between the different segments because the chosen profiles are structural steel.

### 20.3.20.3 About BIMSTRUCTURALCONNECT

The BIMSTRUCTURALCONNECT command automatically generates connections for profiles.

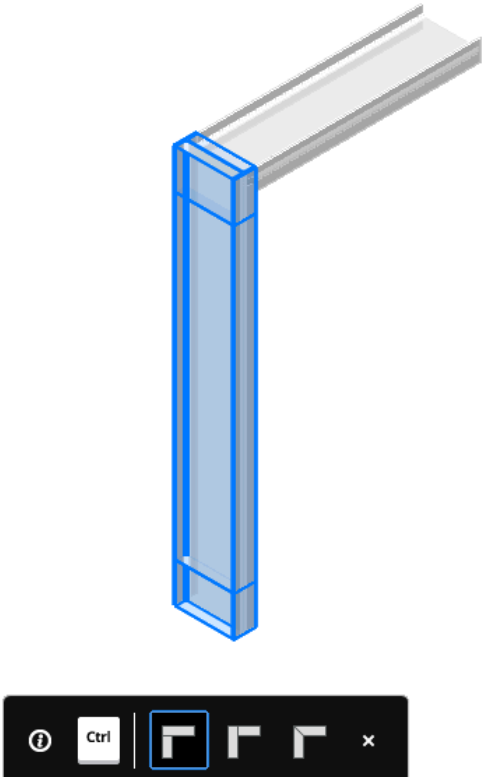
For more information about this command, visit the Command Reference article BIMSTRUCTURALCONNECT.

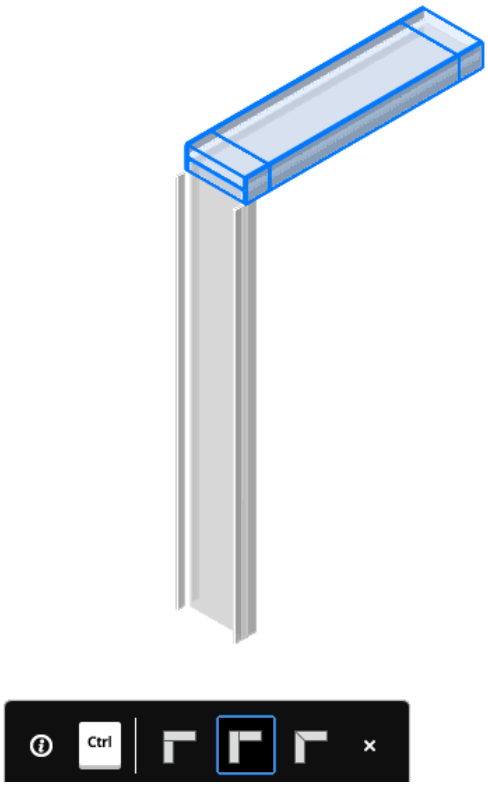
**Making a structural connection to linear solids**

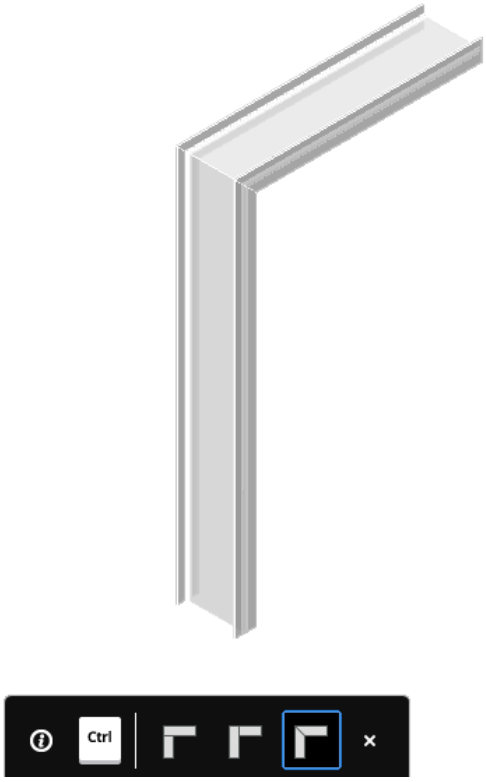
1 Select the profiles you want to connect with.

2 Launch the BIMSTRUCTURALCONNECT command. by typing it in the Command line or select **StructuralConnect** in the **Model** tab in the Quad.

3 When the Hot Key Assistant is on (**HKA** on the **status** bar), a widget (as displayed below) appears on the screen. Tap the CTRL-key to cycle through the different options.


Ctrl


Ctrl




4 Press Enter to accept the selected option and end the command.

### 20.3.20.4 About DRAG

The DRAG command makes it possible to alter a segment of a drawn profile without losing the connection to other segments.



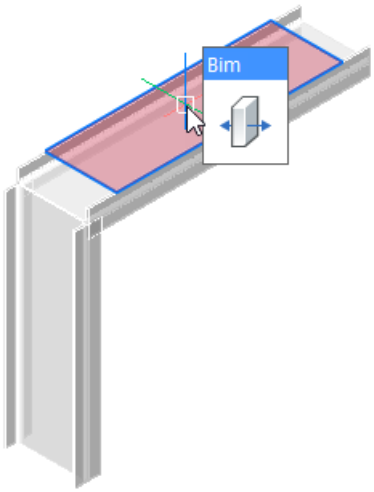
For more information about this command, visit the Command Reference article DRAG.

**Altering the positioning of segments**

***Прим.:*** This only works if the connections were created with the BIMSTRUCTURALCONNECT command.

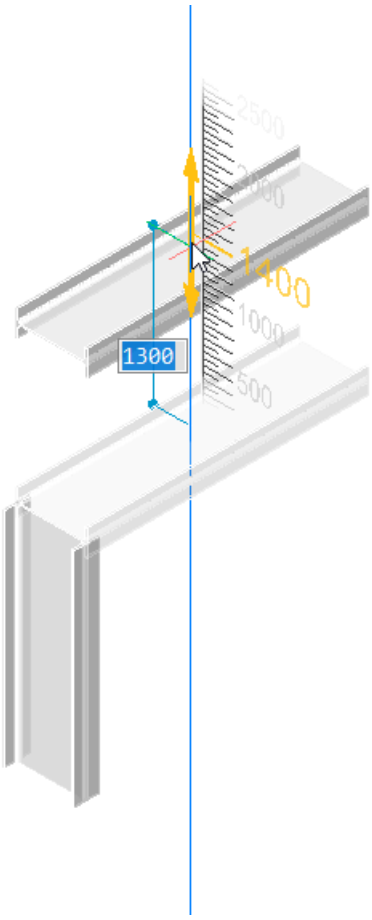
1 Set the value of DISPLAYSIDESANDENDS system variable to 1.

2 Select a face perpendicular to the section of the profile.




3 Launch the DRAG command.

4 Click the mouse to set a new position for the segment. Or use the dynamic dimension field to enter a value for the displacement.




### 20.3.20.5 Procedure: use modeling tools to create a structural steel model

1 Draw a BimGrid as a base.

2 Launch the BIMGRID command.

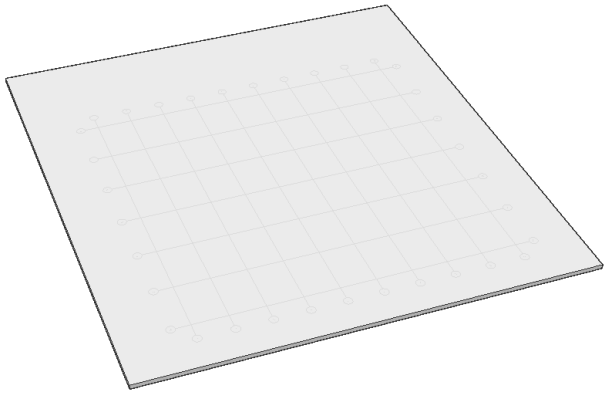
3 Click a starting point and enter values into the dynamic dimension fields.



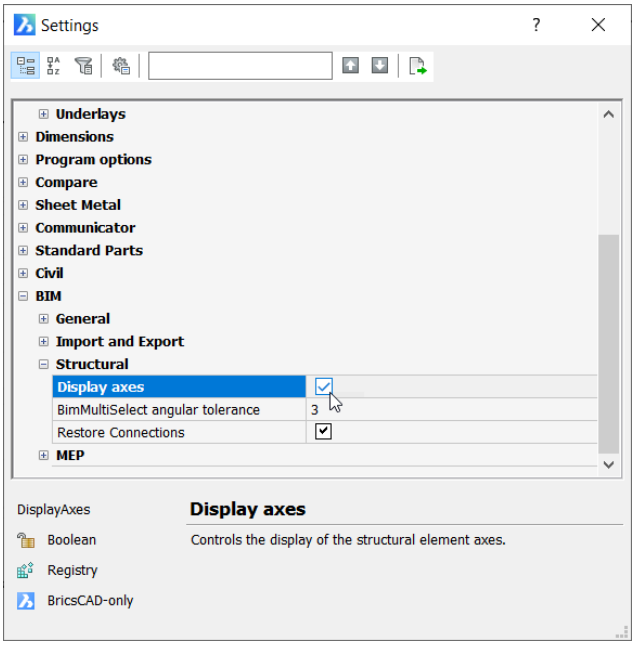
4 Draw a base slab underneath the grid.

5 Create a slab using the BOX command.

6 Ensure that the grid is entirely on top of the slab and that the axes of your profiles are displayed.

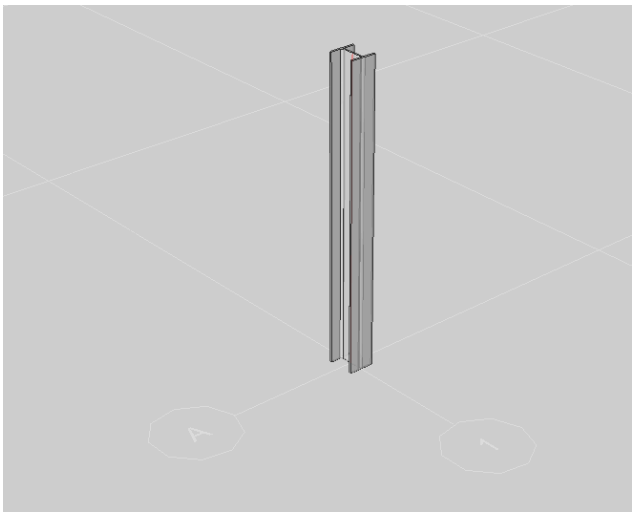


7 Go to the **Settings** dialog and under the **BIM** > **Structural** section. Ensure that **Display Axes** is checked.




8 Open the **Profiles** panel and drag a structural steel profile into your model. Start drawing at the intersection point of two grid axes and draw vertically upward.

***Прим.:*** To snap specific points, make sure you have toggled on the different types of points you want to snap in the context menu when right-clicking on ESNAP.



9 Launch the BIMPROPAGATE command.

10 Select the slab as base solid and press Enter.

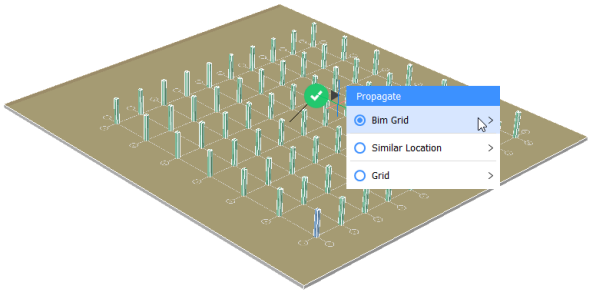
11 Select the column as your detail element and press Enter. BricsCAD will zoom into the 3D detail. Press C to copy the column as a solid, not as a block.

***Прим.:*** The default setting for BIMPROPAGATE is to copy the object as a block.

12 Hit Enter.

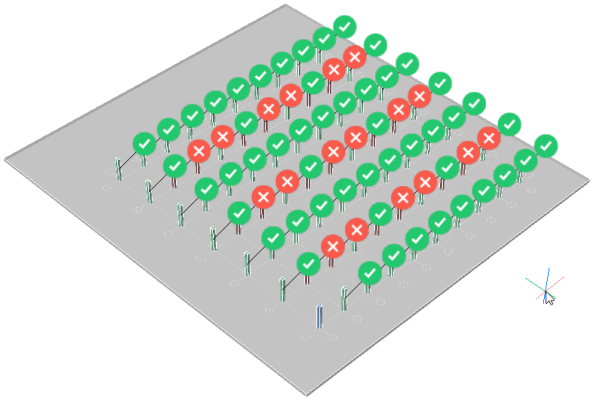
***Прим.:*** BricsCAD automatically copies the beam to every grid intersection. This is because you chose the floor slab with a BimGrid as your base solid.

13 Hover over the green checkmark and click **Bim Grid** in the dialog box.




14 Click **Explode**. You can now manually toggle individual suggestions to create the pattern of your choice.

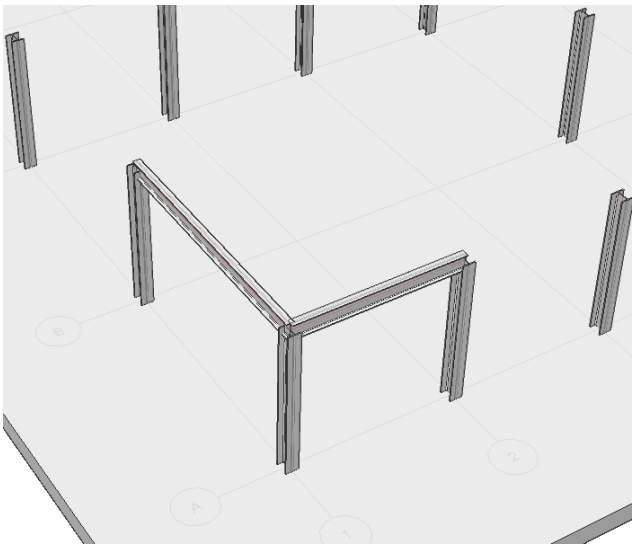
15 Press Enter to accept and end the command.



**Create beams between the columns**

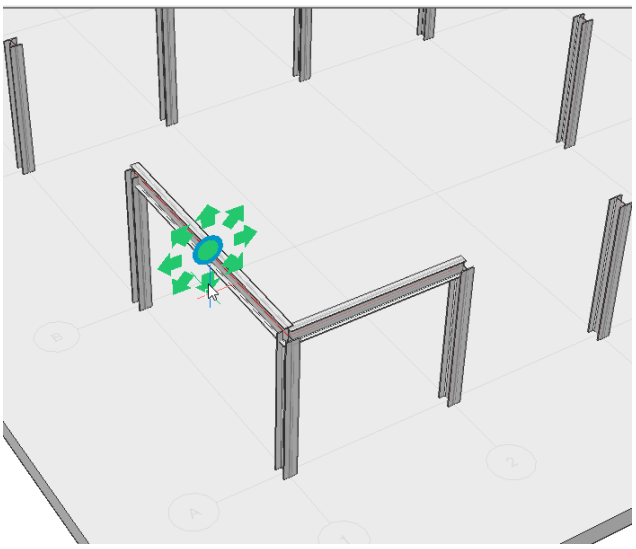
1 Open the **Profiles** panel and drag a structural steel profile into your model. To set the start and endpoints, click the top of the column and then the next. Do this in both directions.

***Прим.:*** If you want to turn your profiles to make the flanges are facing upwards, press Q (from Quarter turn). The profile will turn 90 degrees around its axis.



2 Drag and drop a beam profile from the **Profiles** panel onto this beam. Give the second beam a large profile, as it has a longer span.

3 Select a beam and launch the BIMADDECCENTRICITY command to alter the relative position of the axes of the beams.

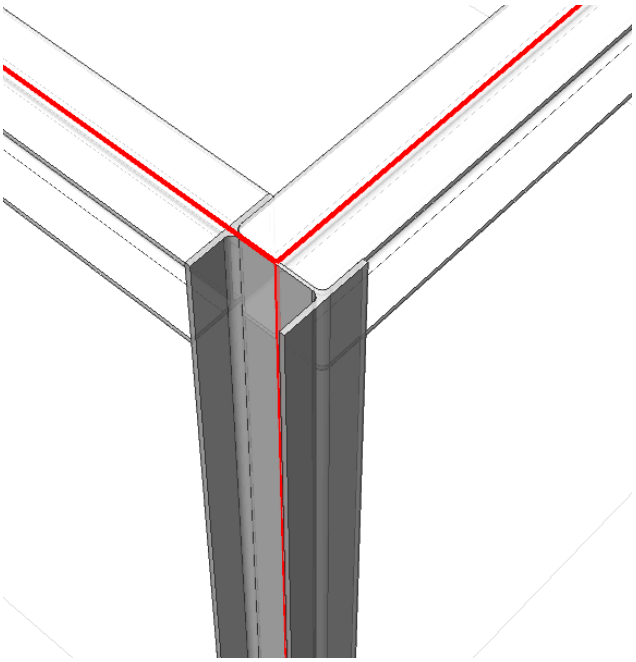


4 Click the down arrow. The beam moves down in relation to its axis.

5 Press Enter to end the command.

6 Repeat steps 3-5 for the other beam.

7 Select the column and the two beams then launch the BIMSTRUCTURALCONNECT command to make a simple connection. Press Enter to accept the default option.



8 Repeat step 7 for the other 2 junctions.

9 Select the two columns connected to the beam in the Y-direction and launch the BIMPROPAGATE command.

10 Select the beam in the Y-direction as the detail solid.

11 Press C to copy it as a solid, not as a block.

12 Press Enter.

13 Choose which suggestions to accept. Press Enter to accept and finish the command.

***Прим.:*** Suggestions are made at the bottom of every column because the situation is symmetrical. You can toggle the suggestions off.

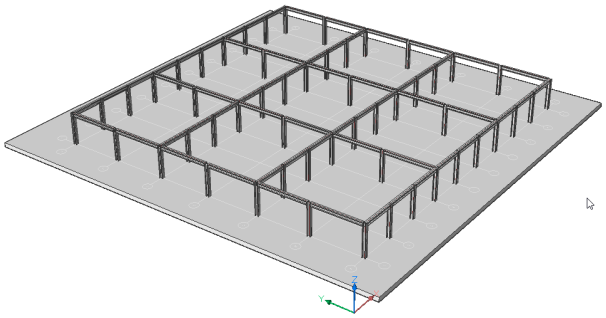
14 Repeat steps 9 - 13 to create beams on the X-direction.

***Прим.:*** You can use box selections to select all the lower beams at once.

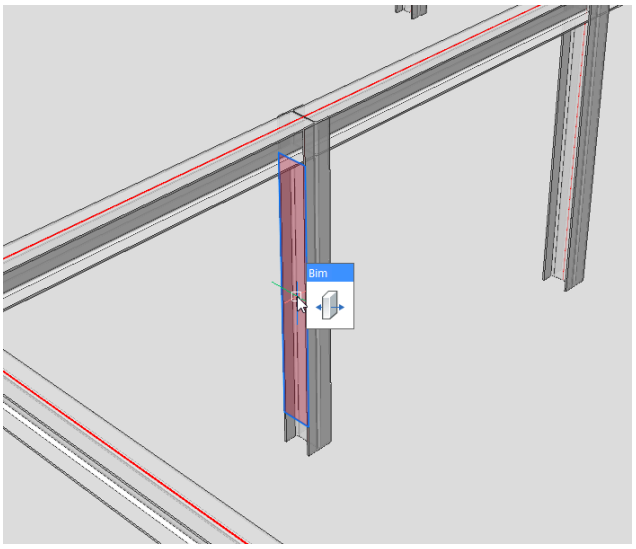
**Move a Beam**

1 Set the value of DISPLAYSIDESANDENDS system variable to 1.

2 Highlight the side face of the desired column and launch the DRAG command.



3 Move the column to the desired position.



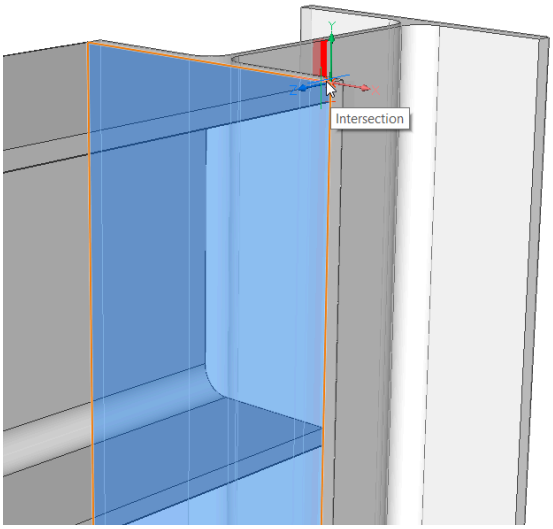

***Прим.:*** The connections with the adjoining beams are maintained.

### 20.3.20.6 Procedure: use the modeling tools to create a more detailed connection

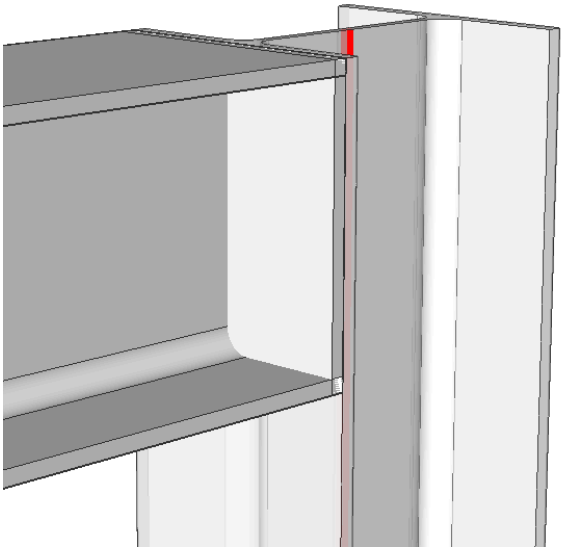
1 Set the value of DISPLAYSIDESANDENDS system variable to 0.

2 Launch the BOX command and hover over the side face of the column.

3 Press Shift once so that the face is highlighted in blue.

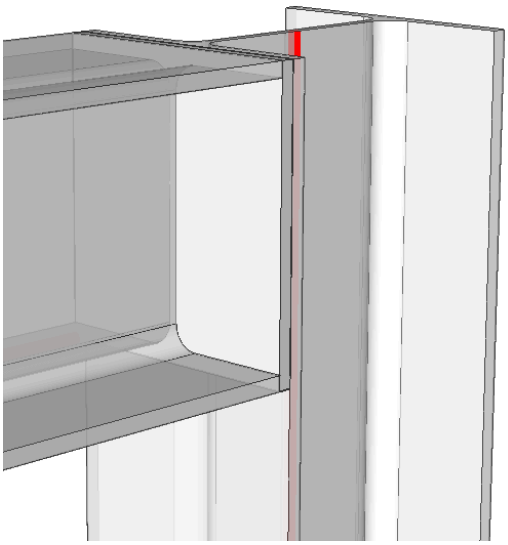



4 Now draw a plate on the flange of the column.



5 Select the beam and launch the SUBTRACT command.

6 Select the endplate and press Enter.

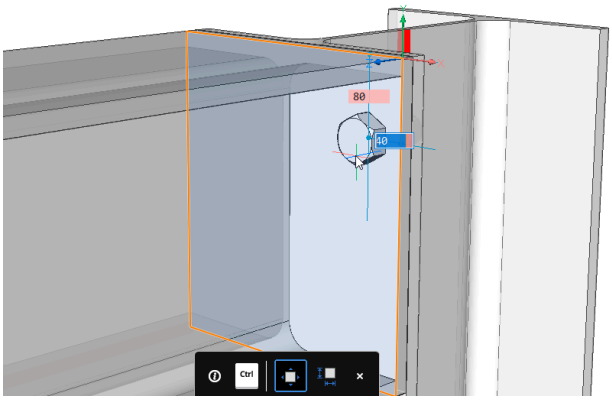


7 Select the face and launch the BIMIFY command from the Quad.

***Прим.:*** You can also Classify the plate manually by Quad selecting **Classify Manually** in the **BIM** tab.

8 Open the **Library** panel and drag a bolt onto the plate.

***Прим.:*** You can position it precisely by entering values in the dynamic dimension fields.

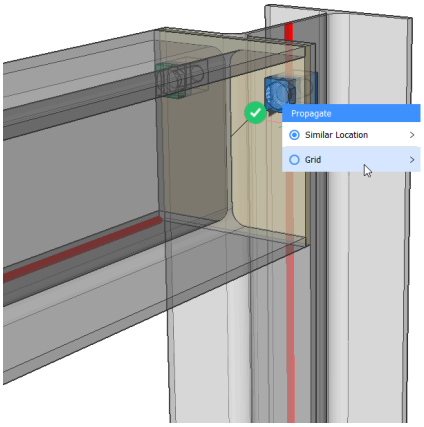



9 Select the plate and launch the BIMPROPAGATE command.

10 Select the bolt as the detail and press Enter.

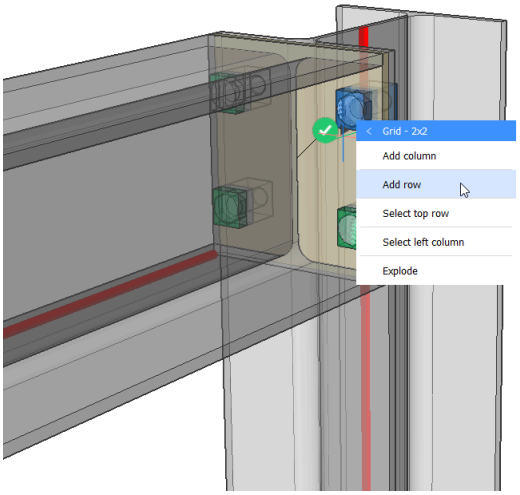
11 Press Enter again and click on the blue question mark. It will turn into a green checkmark.

12 Hover over the checkmark and select **Grid**.

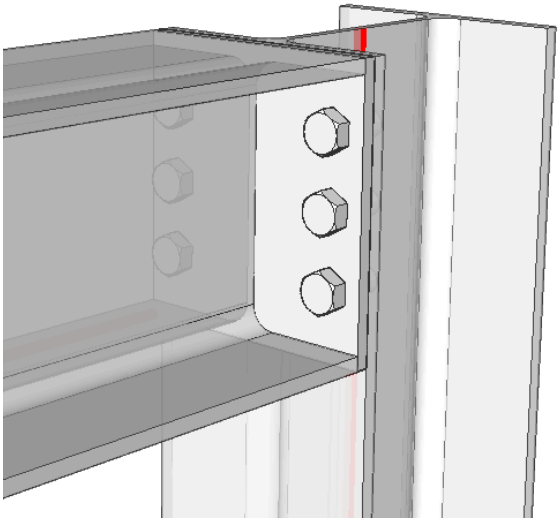



13 Add one row.

14 Press Enter to end the command.




The detailed connection has now been created.



### 20.3.20.7 Procedure: using BIMSTRETCH to modify structural steel

1 Open a New drawing.

2 Open the **Profiles** panel by clicking on it on the right-hand side of your screen.

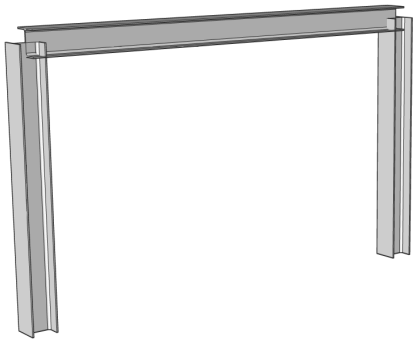
***Прим.:*** If the **Profiles** panel icon isn't visible yet, right-click on the ribbon and check **BIM Profiles** in the context menu under **Panels**.

3 Drag the profile you want to draw a structure with within the model space.

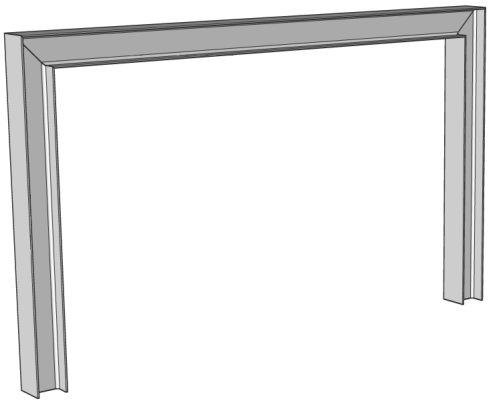
4 Draw a column with a height of 3000 mm. Don't exit the command and draw a connecting beam with

a length of 5000 mm horizontally. Now, add another column at the end of the beam. The height of this column is also 3000 mm.

***Прим.:*** Make sure you turned the profiles in the right way.



5 Select the 3 beams and select **Connect Structural** in the ribbon or select the command from the Quad. You will be able to choose which connection you prefer. Choose the L-connection option and press enter. Your entities are now connected.



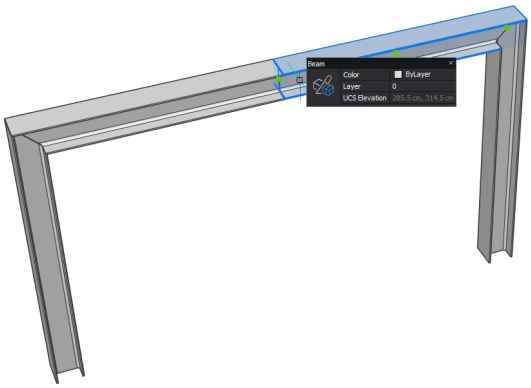
6 In the ribbon, make sure you enable the **Display sides and ends** and **Prioritize selection of Faces**. You can find them in the **Structural**/**MEP** – **View** tab or set the value of DISPLAYSIDESANDENDS system variable to 1 and SELECTIONMODES system variable to 2.

7 Check if the profiles are correctly classified as **Beam** and **Columns**.

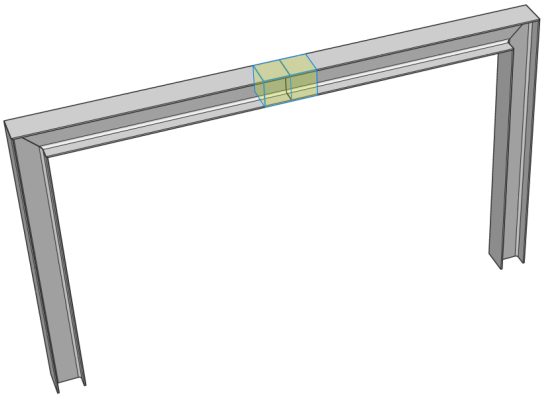
8 Launch the SLICE command and slice the beam in the middle.

9 You will be prompted: Slice along axis? Press Enter to accept the default option (Yes).

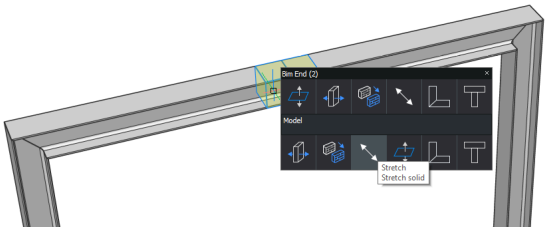
10 Select the midpoint of the beam and press Enter. Your beam is now sliced into two beams.




11 Select the 2 ends of the beams in the middle of the structure. They will be highlighted in yellow.

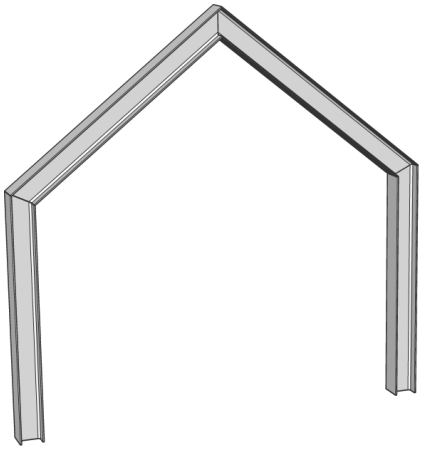


12 Quad select the BIMSTRETCH command in the **Model** tab.



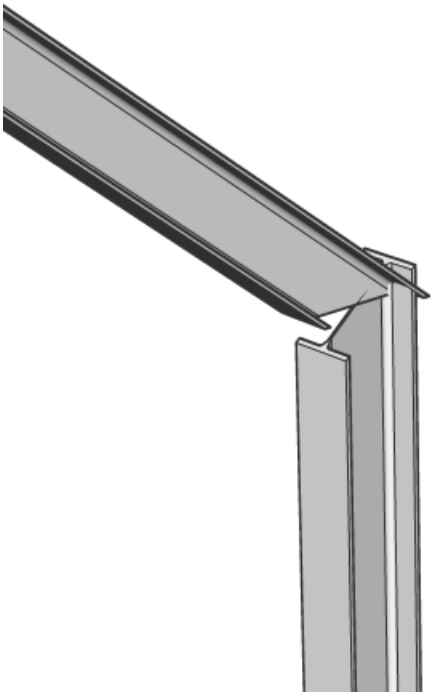

13 Hover orthogonally upwards, enter a height of 2000 mm and press Enter to accept.

14 The result should look like this.



As you can see, the connections between the profiles are automatically adapted to the new angle between the entities.

***Прим.:***If you would only have selected one of the two side-ends in the middle of the beam, the connections on the side that was not selected won't be included in the stretch and will look like this:



### 20.3.21 Using BIM quickdraw

#### 20.3.21.1 Commands

BIMQUICKDRAW

### 20.3.21.2 About

The **Quickdraw** tool allows you to get started with conceptual modeling easily. It also automatically adds BIM data. You can use the **Quickdraw** tool to create a room or a whole building. Once in use, **Quickdraw** will remain active until you explicitly exit the command or until another command is executed.

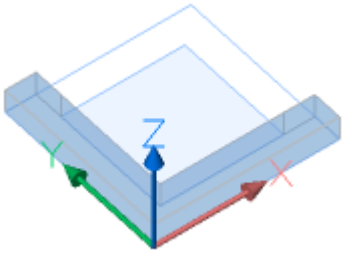


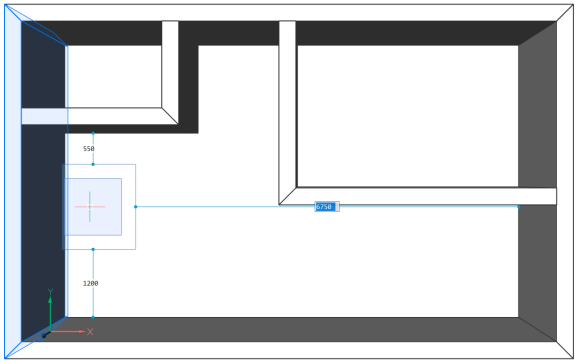
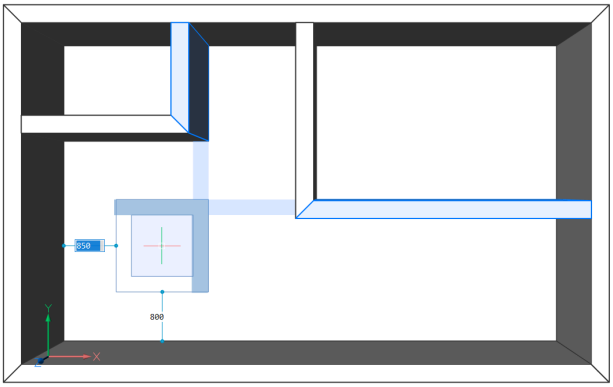
For more information about this command, visit the Command Reference article BIMQUICKDRAW.

### 20.3.21.3 The magnetic snapping feature

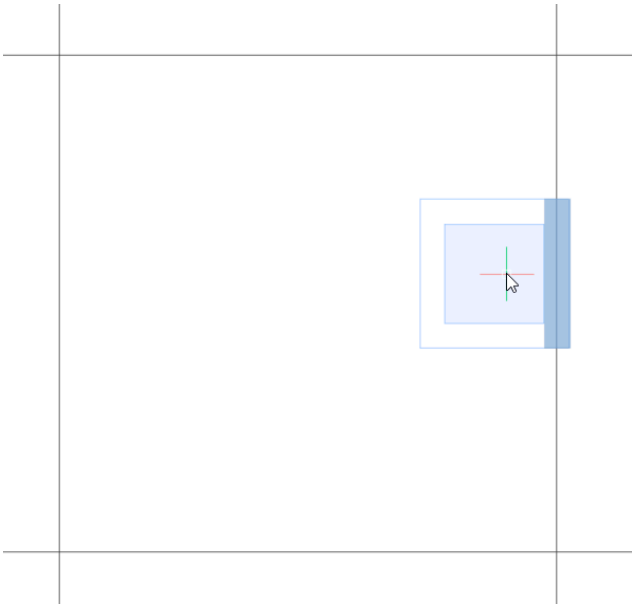
With the magnetic snapping feature, you can easily align the QuickDraw cursor. When the command is active, move the cursor close to the object you want to align to. The QuickDraw cursor automatically snaps to the object.

You have the following options:

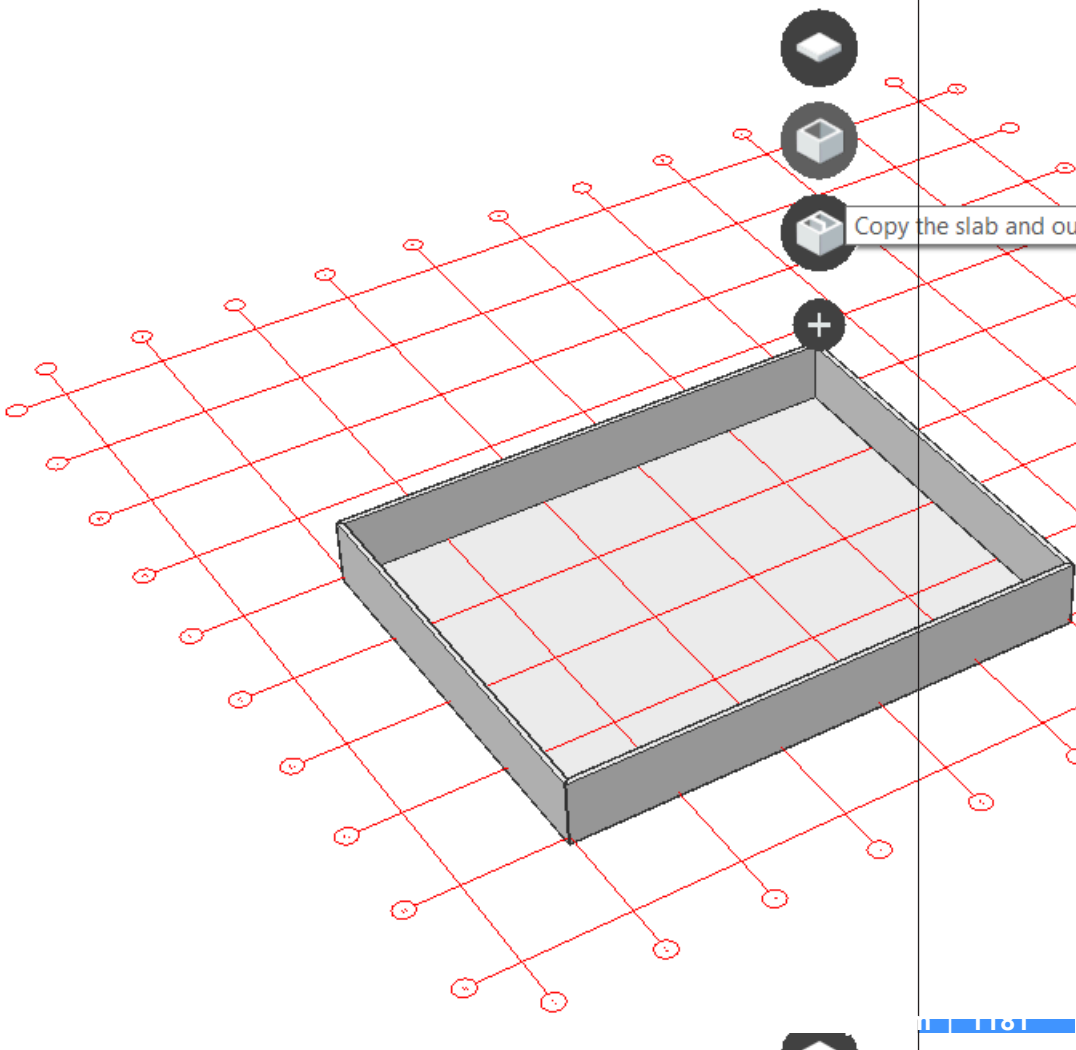
| | |
|---|---|
| 1. Align to the origin |  |

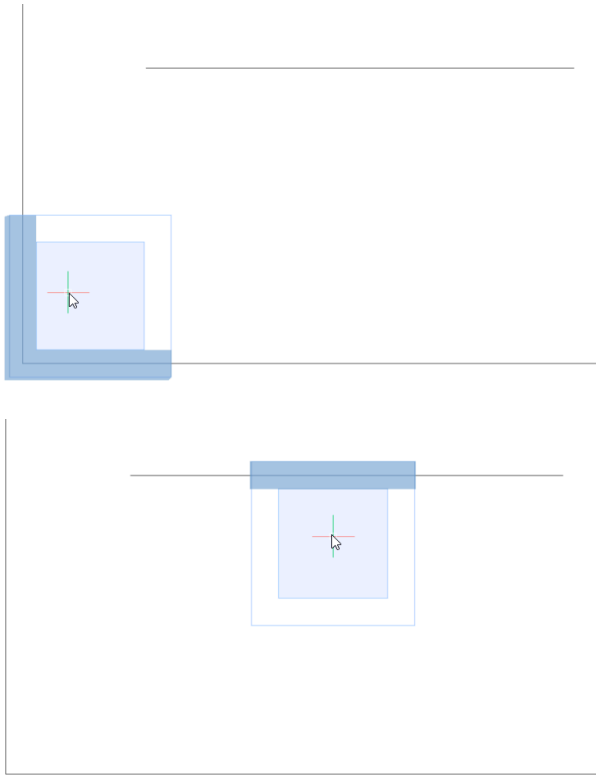
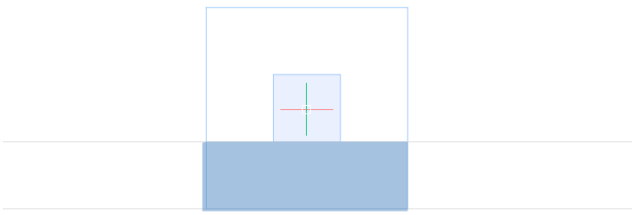
| | |
|---|---|
| 2. Align to existing walls | To the wall itself<br><br>To the extents of the wall<br> |

| | |
|---|---|
| 3. Align to lines | To grid lines |



***Прим.:*** If you use **Quickdraw** to copy a story, the gridlines will only be copied if the grid is assigned to the same spatial location of the ground floor slab and walls. If this is not the case, the gridlines that lie in the boundary of the ground slab will be projected on the slab of the top floor. This is only a visual help. The gridlines on the slab of the top floor will disappear when you exit the QUICKDRAW command.



| | |
|---|---|
| | To X- and Y- aligned lines and line segments<br> |
| | To parallel lines<br> |

### 20.3.21.4 Setting the Quickdraw system variables

You can use the **Quickdraw Settings** to set the Quickdraw system variables:

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** from the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the

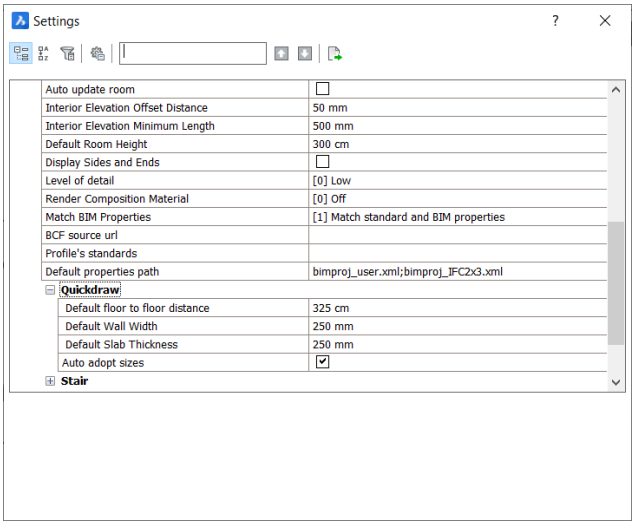
Command line.

2 You are prompted: Select first point or [quickdraw Settings]:

3 Enter S for the **Quickdraw Settings**.

4 Specify a new value to the system variables of **Quickdraw** to change their default values (e.g. **Default Wall Width**, **Default Slab Thickness**, and Default floor to floor distance).

When the **auto adapt sizes** option is turned on, the dimensions of the walls and slab of the new room will be the same as the walls and slab you snap to.




### 20.3.21.5 Changing system variables while running Quickdraw

It is possible to edit the dimensions of the walls you will draw while the QUICKDRAW command is launched. Change these settings in the **Quickdraw Settings** dialog box, select the settings in the prompts menu or enter the specific letters in the Command line and press Enter to change the dimensions.

- H: Change the dimension of the floor height.
- W: Change the dimension of the wall width.
- T: Change the dimension of the slab thickness.
- A: Adapt the dimensions of the room you are drawing in/against automatically.

***Прим.:*** Make sure that you enable the hidden options in the prompt menu. Set the value for PROMPTMENUFLAGS system variable to 1.

### 20.3.21.6 Procedure: creating an L-shaped house

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** from the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the Command line.

You are prompted: Select the first point or [quickdraw Settings]:

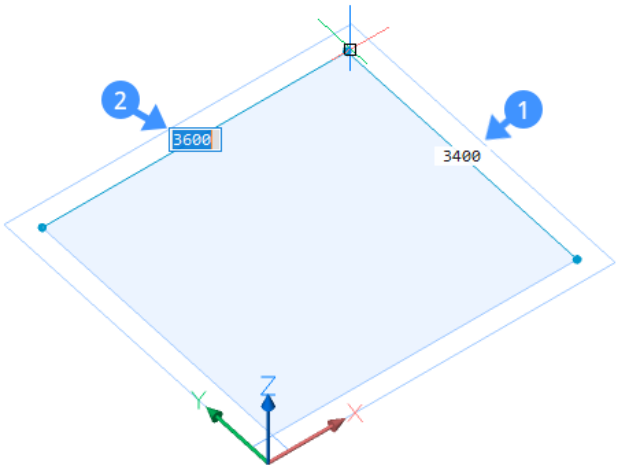
***Прим.:*** Use the **Quickdraw Settings** to change the wall width, wall height and slab thickness of the room before you click and then specify the first corner of a building.

2 Select any point in the drawing area to set the first corner.

***Прим.:*** To start from the origin, use the magnetic snapping feature of **Quickdraw**.

3 Drag the mouse cursor to define the footprint of the building.

***Прим.:*** If the **Dynamic dimension** (DYN) option is on, the current values of the room width (1) and length (2) are displayed in a dynamic dimension field.

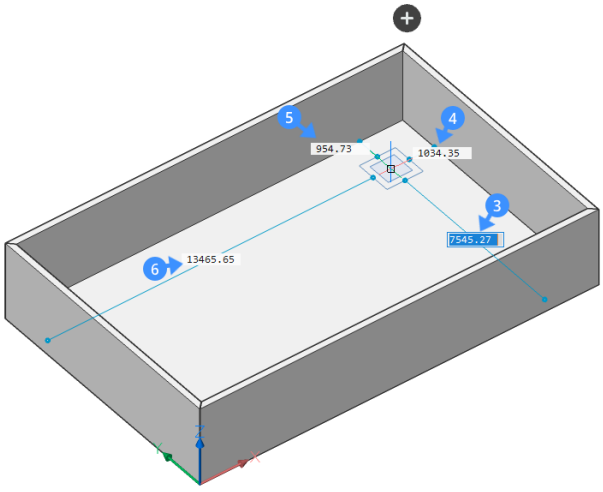



Specify the values of the room width and length in the Dynamic dimension field. Press the Tab key to switch between width and length and press Enter or just click freely. Press Enter or Escape to leave the QUICKDRAW command or you can continue to create more rooms.

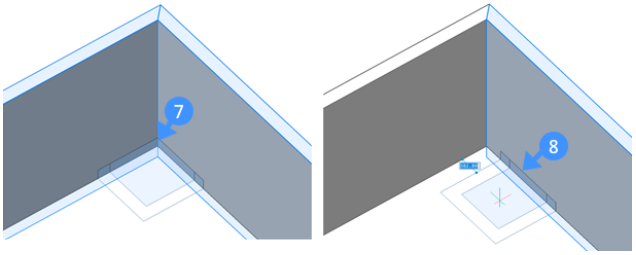
4 Place the Quickdraw cursor inside the existing building to create more rooms.

5 When you place the Quickdraw cursor inside the existing building, the dynamic dimension fields display the distances from the Quickdraw cursor to the adjacent walls (3,4,5,6).

When the "auto adapt sizes" option is turned on, the dimensions of the walls of the new room will be the same as the wall you snap to.




If the cursor aligns with one or two existing walls in the model, the layout of the cursor will adapt and depict the alignment.



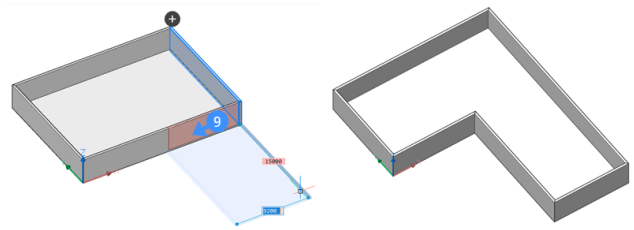

To actually draw new walls, choose the distances from the already existing walls to specify your first point. Move your cursor or type the values in the dynamic dimension fields to do this. Draw walls in the same way as explained in steps 2 and 3.

6 To remove a section of a wall from the existing room, first, align the Quickdraw cursor with an existing wall at the inner side of the building. Then click and drag the cursor outside of the existing building.

7 The outer wall will turn red (9) to indicate the part of the wall that will be removed.

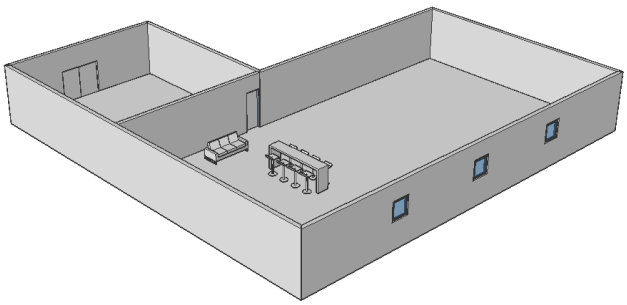
***Прим.:*** You can also align your cursor with an existing wall at the outer side of the building to remove a part of the slab. Then click and drag the cursor inside the existing building.

8 Select a second point. The wall highlighted in red will automatically be deleted and an L-shaped room will be created.




### 20.3.21.7 Procedure: creating a story and a roof

Start with a similar configuration:



Go through the following steps to create two extra stories, a floating part and a flat roof.

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab or type BIMQUICKDRAW in the Command line.

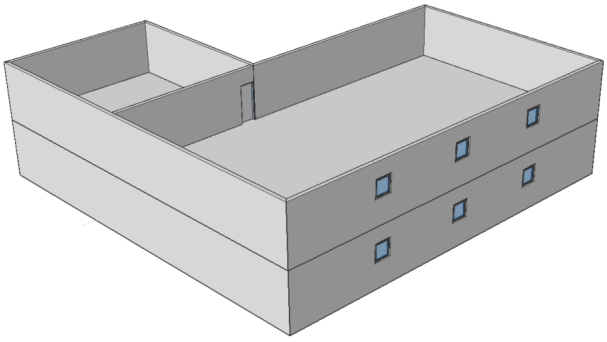
2 Click on the icon (⊕) to see the options:

- Fully copy the top floor. All features will be copied (windows, doors, interior walls, and

furniture, excluding doors on the outer walls of the ground floor).

- Copy the slab and the outer walls of the top floor.
- Finish the building with a flat roof.

3 For the first story, choose the first option (Full copy).

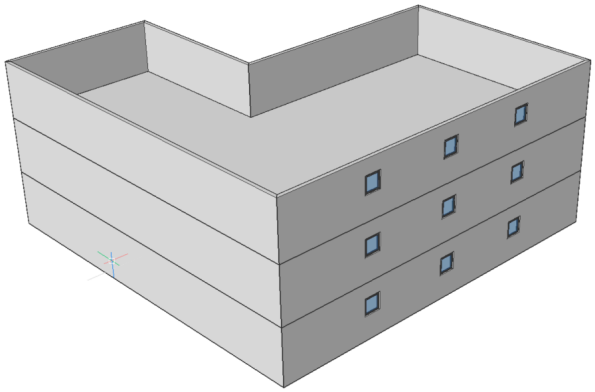


***Прим.:*** The interior wall and interior door and the windows in the exterior wall are copied. The door in the exterior wall on the ground floor is not copied.

***Прим.:*** The furniture is not copied in this case, because it is not assigned to the ground floor (see the **Properties** panel). If you want to copy the furniture, use BIMIFY to assign the furniture to a spatial location. The new furniture will be automatically assigned to the new spatial location.

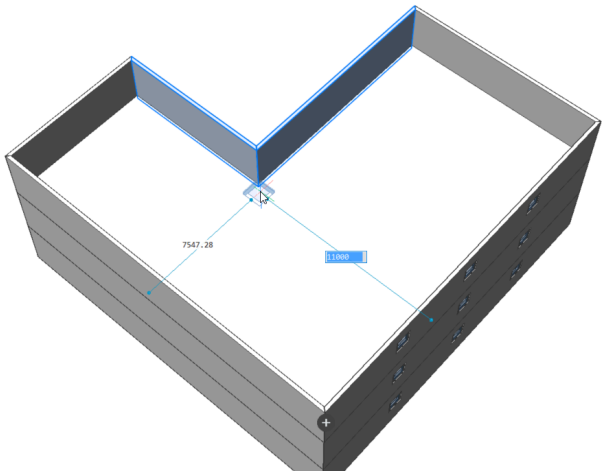
***Прим.:*** Walls inherit the spatial location of the slab on which they are created.

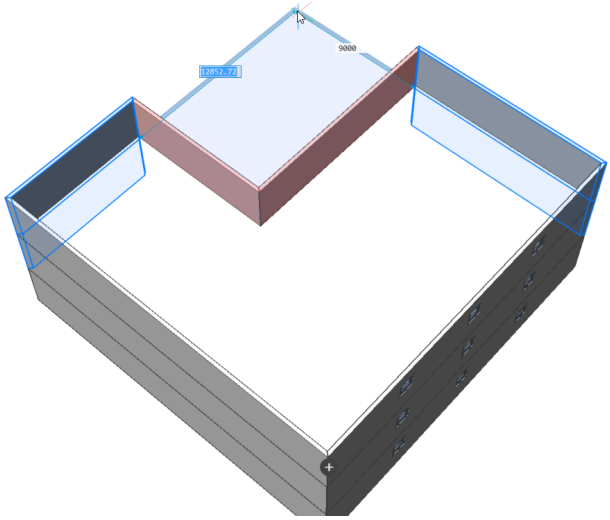
4 Repeat step 2. Choose the second option (Copy the slab and the outer walls) to create the top story.



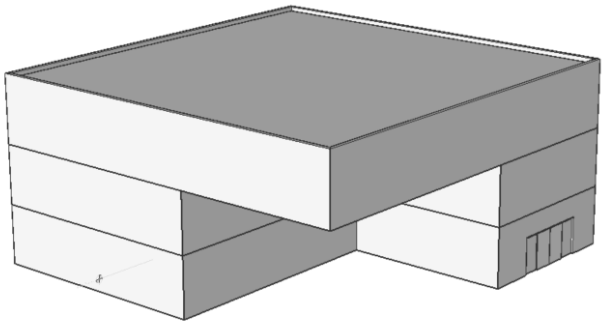
5 To make the overhang you should remove parts of the outer walls of the second floor. First, snap the Quickdraw cursor in the corner of the two outer walls, inside the building. Then click and drag the cursor outside.

6 The outer walls will turn red to indicate that the walls will be removed.



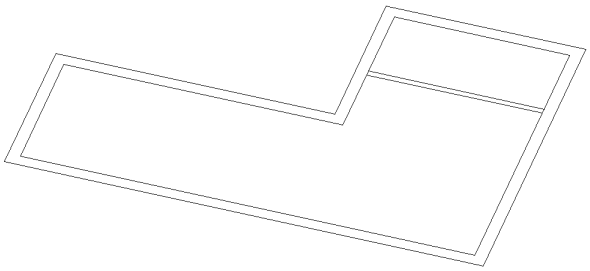




7 Use the same method as in step 2, selecting option 3 (Finish the building with a flat roof).



### 20.3.21.8 Procedure: using Quickdraw starting from a 2D layout

1 Open a 2D floorplan of a rectangular or L-shaped building.

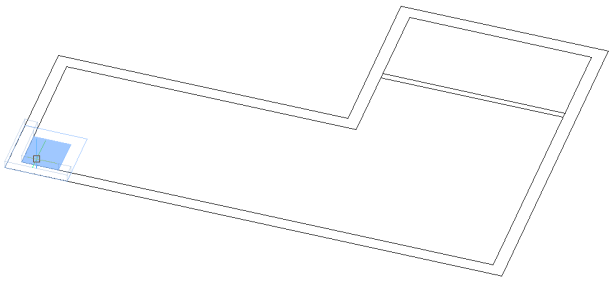


***Прим.:*** It is also possible to draw a simple 2D floor plan with the (POLY)LINE command to start from. In this case, the default wall thickness of the BIMQUICKDRAW command will be used.

***Прим.:*** The Quickdraw cursor now supports snapping to (double) lines inside XREFs.

2 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the Command line.

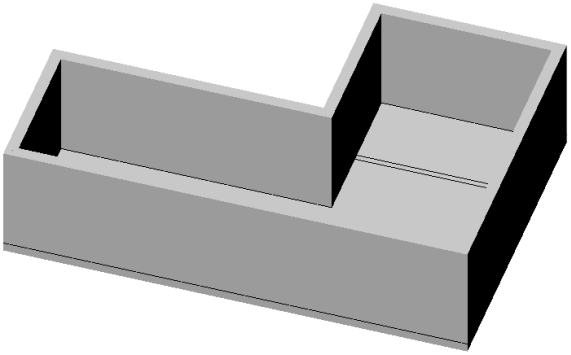
3 Hover over one of the outer walls with the cursor.



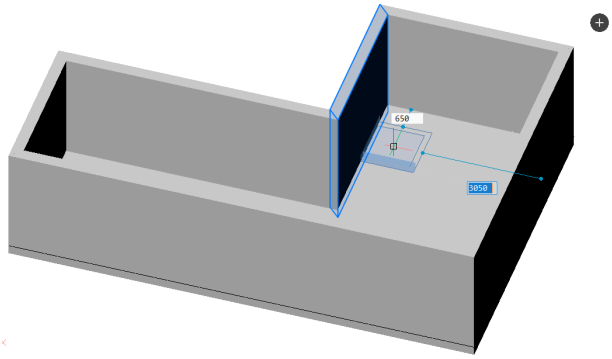
***Прим.:*** The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

***Прим.:*** If the lines in the (imported) 2D floorplan are not optimal (e.g. not exactly X- or Y-aligned), the QuickDraw-cursor will not snap to it (as it is perfectly X-Y-aligned). In that case, use the Optimize command to make sure all 2D lines exactly X- or Y-aligned.

4 Draw the outer walls, using the magnetic snapping feature to snap to the lines of the 2D drawing.

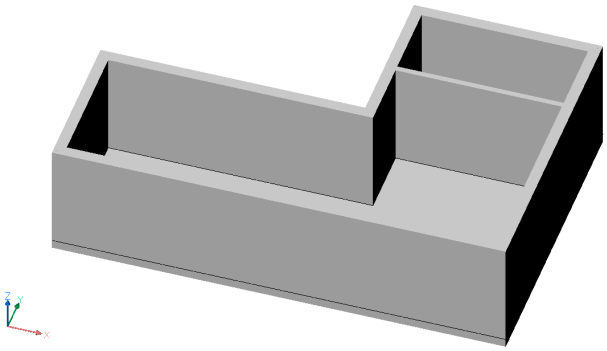


5 Hover over one of the inner walls with the cursor, using the magnetic snapping feature to snap to the lines of the 2D drawing.




***Прим.:*** The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

6 The 2D plan has been transformed into a 3D model.



***Прим.:*** you can use the same method as in step 2 and 6 of the previous procedure to add floors or finish the building with a flat roof.

## 20.3.22 Using drag

### 20.3.22.1 Commands

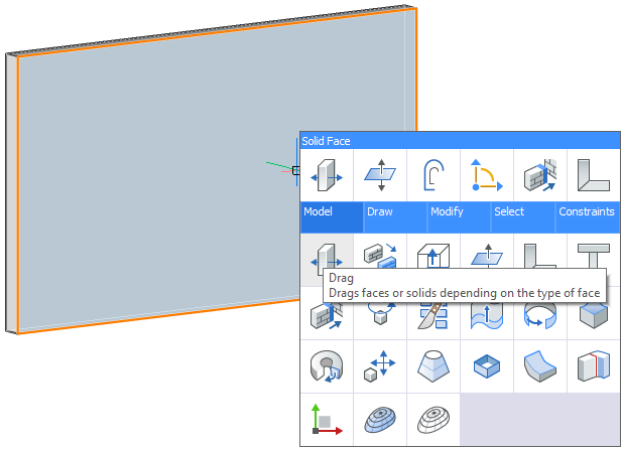
DRAG

### 20.3.22.2 About

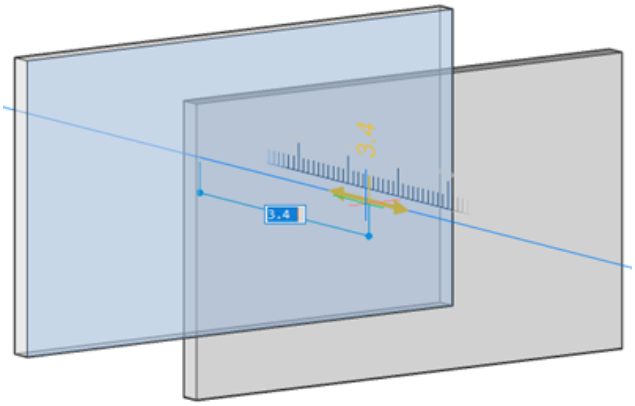
The **Drag** tool allows you to move walls, slabs or linear elements on the highlighted face of a solid. For instance, the **Drag** tool can be used to move linear solids by highlighting/selecting the end or side face of a linear solid. Depending on the modeling operation, the connection between solids can be either preserved while dragging one or more solids at a time or solid (s) can be moved independently by disjoining the connectivity to other solid(s).

### 20.3.22.3 Procedure: using drag on a 3D solid

1 Highlight/select any faces of the solid and select **Drag** from the **Quad** > **Model** tab.

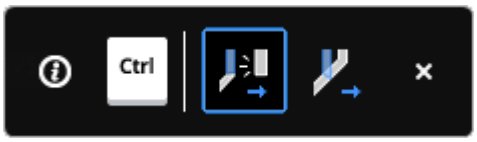



2 Move the cursor to adjust the position of the moving solid in the dynamic dimension field, then click or enter a precise value in the dynamic dimension field and press Enter.




***Прим.:*** A ruler appears to visually represent the distance over which you will move the entity.

3 The Hotkey Assistant appears at the bottom of the screen showing the available options.

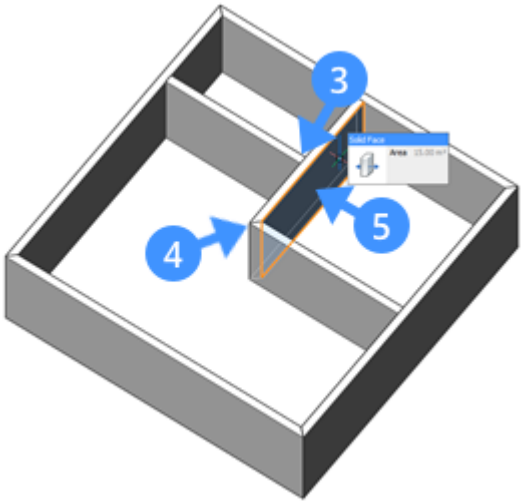


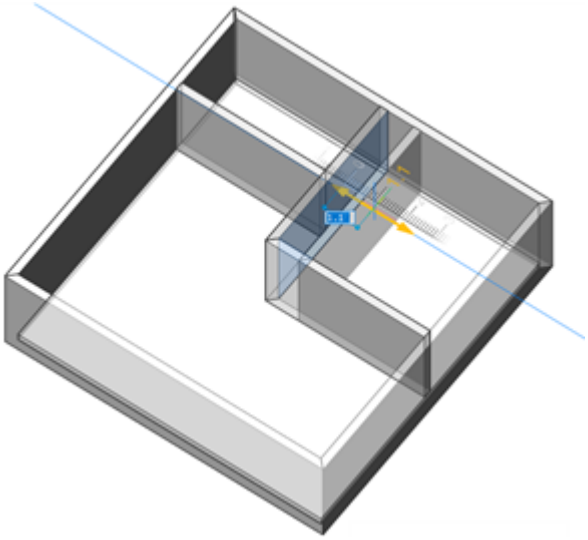

***Прим.:*** The current mode is indicated by a blue frame.

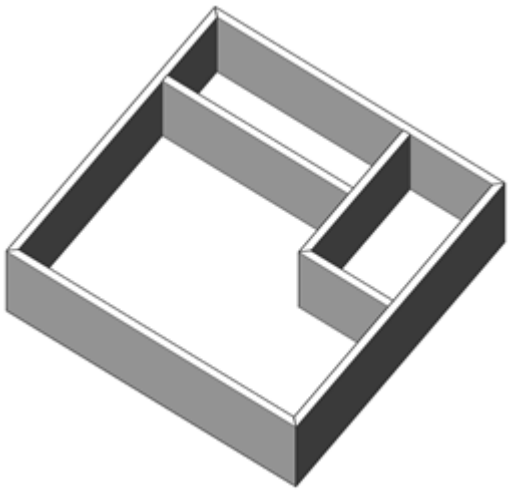
Hit the CTRL-key to toggle the connectivity mode. Alternatively, type D+Enter to disable connectivity or type E+Enter to enable connectivity.

- Select the **Enable connectivity mode** option, to move the solid while preserving the connections between solids.

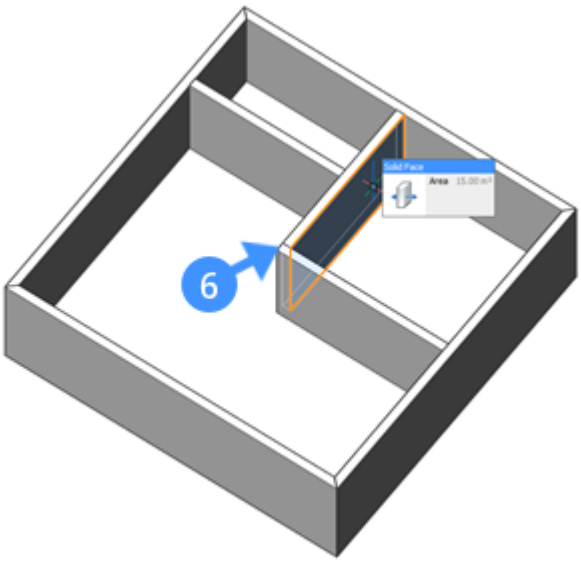
In the following illustration, the solid behavior is shown before and after applying the DRAG in **Enable connectivity mode**. The connectivity between walls, i.e. T- (3) and mitered (4) connections are preserved after dragging an inner wall surface (5).

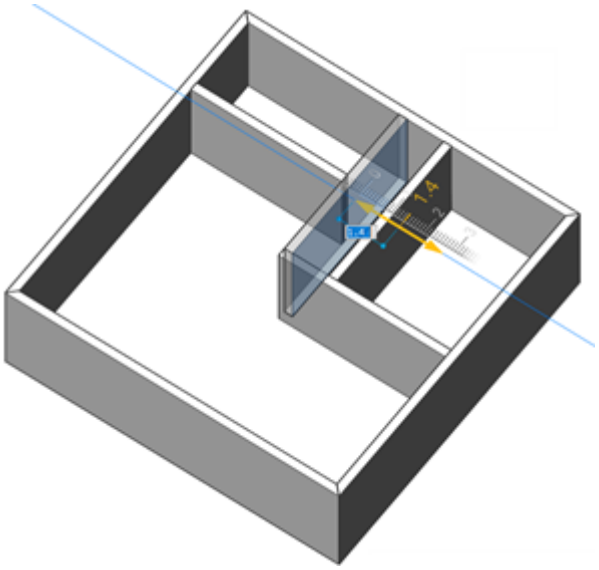


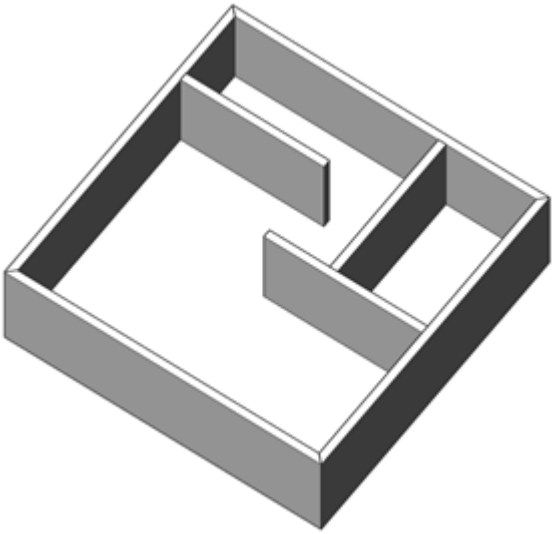





- Select the Disable connectivity mode (default) option to move the solid while breaking the connections between solids.
- In the following illustration, the solid behavior is shown before and after applying the Drag in "Disable connectivity mode". The connectivity between walls, i.e. T- (3) and L- (6) connections are disjointed after dragging an inner wall surface (5).







### 20.3.23 Using parametrize

#### 20.3.23.1 Commands

PARAMETRIZE, PARAMETERSPANELOPEN

#### 20.3.23.2 About

Parametrizing is a process that automatically adds constraints and parameters to a set of 3D solids, without any manual work. For each unique value, a specific parameter is created. The constraints and parameters are represented by their name and expression in the **Mechanical Browser**. Each parameter can be animated to show what the parameters correspond to in the model. The parametrized geometry becomes controllable in terms of shape and size. When one of the parameters is changed, the geometric model will be updated automatically.

### 20.3.23.3 Procedure: parametrizing a component automatically

1 Open the geometric model that represents your component.

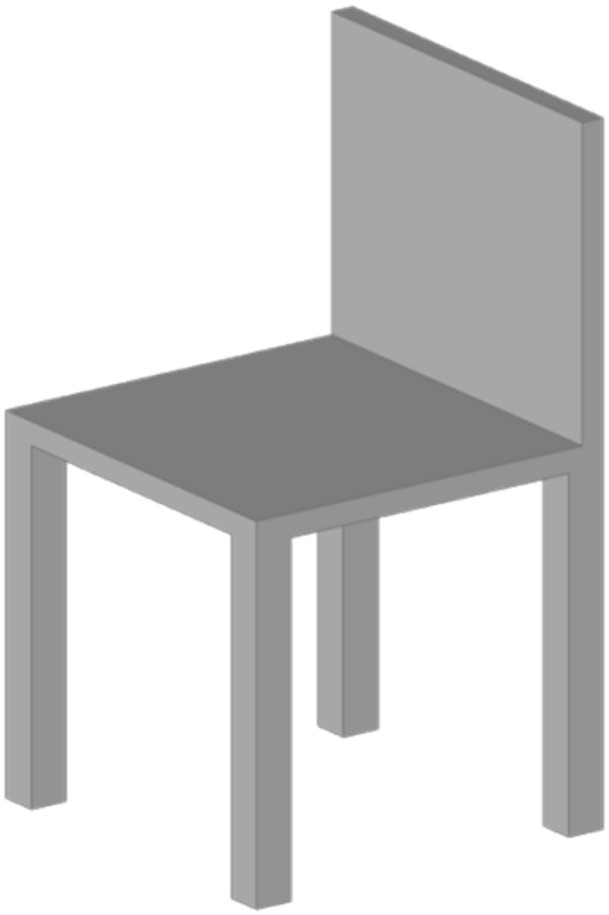
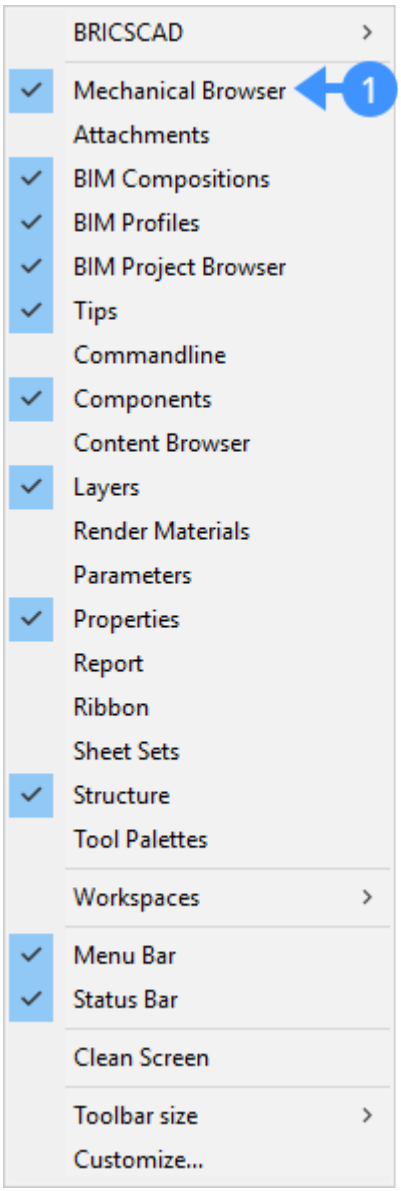


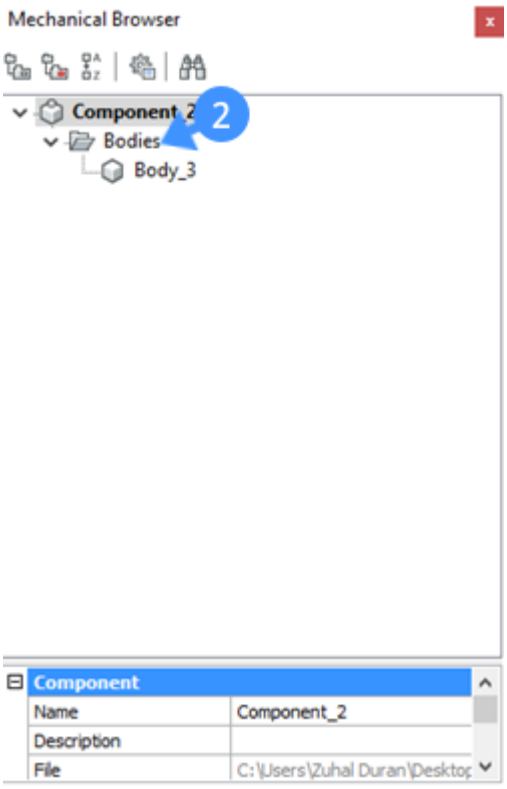
Figure 1, a geometric model of the sample chair.

2 Open the **Mechanical Browser** by clicking its icon in the **Toolbar** panel or type the MECHANICALBROWSEROPEN in the Command line.

***Прим.:*** If the Browser or Parameters icons are not displayed in the **Toolbar** panel, place the cursor on a toolbar, then right-click to open **Panels** and choose **Mechanical Browser** (1) in the context menu.




If the model does not contain any parameters and constraint it will display only the 3D solids under the **Bodies** (2) section. If the model contains any constraint or parameter, the parameterized tool asks whether you want to reuse the existing ones.




3 Select the component geometry you want to parametrize.

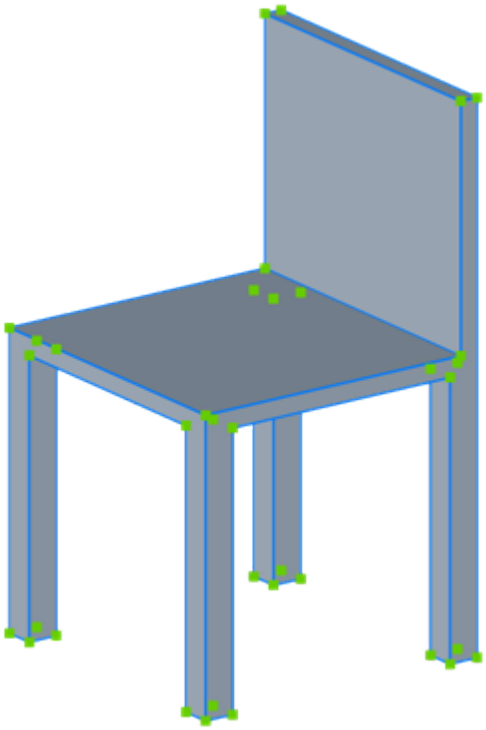


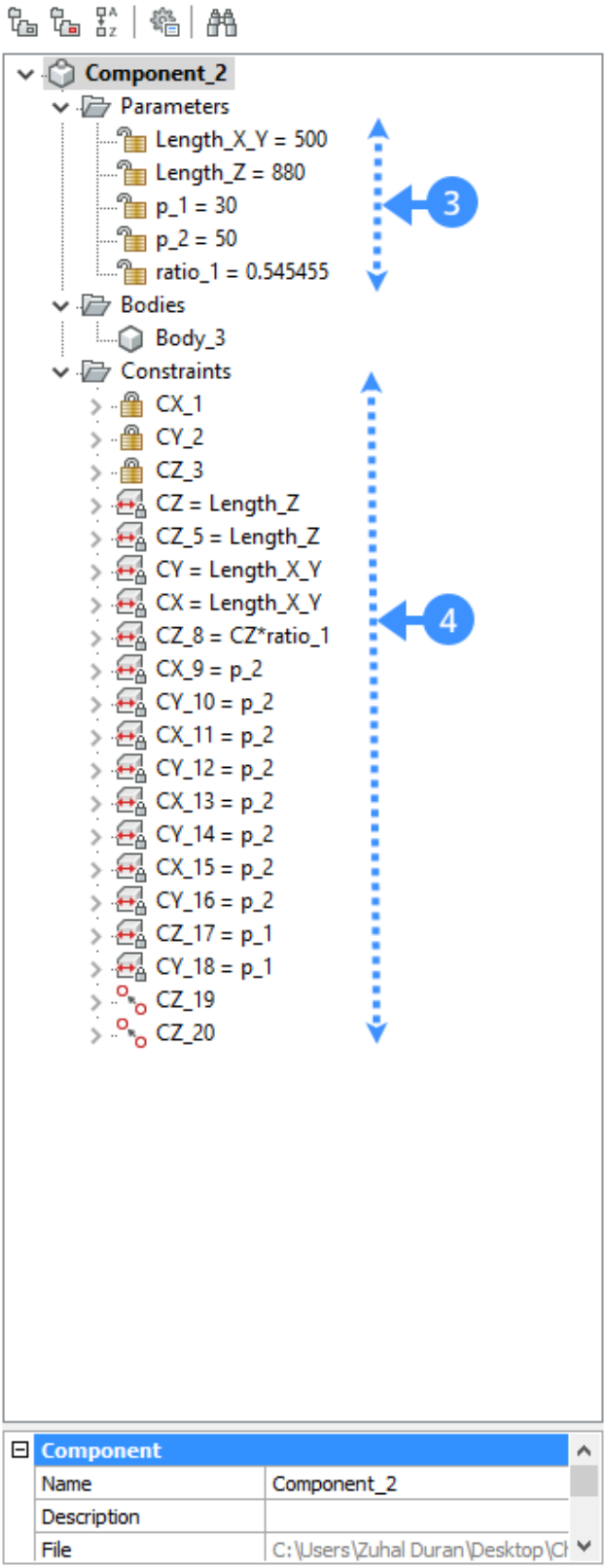
Figure 2, selected object.

4 Launch the **Parametrize** tool in the **Constraints** tab from the Quad.

Or type PARAMETRIZE in the Command line.

A set of parameters and constraints are defined automatically.

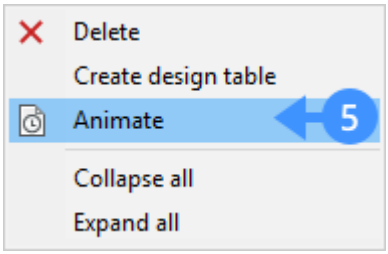
These parameters and constraints are shown in the **Mechanical Browser** (3 and 4).


Component_2
Parameters
Length_X_Y = 500
Length_Z = 880
p_1 = 30
p_2 = 50
ratio_1 = 0.545455
3
Bodies
Body_3
Constraints
CX_1
CY_2
CZ_3
CZ = Length_Z
CZ_5 = Length_Z
CY = Length_X_Y
CX = Length_X_Y
CZ_8 = CZ*ratio_1
4
CX_9 = p_2
CY_10 = p_2
CX_11 = p_2
CY_12 = p_2
CX_13 = p_2
CY_14 = p_2
CX_15 = p_2
CY_16 = p_2
CZ_17 = p_1
CY_18 = p_1
CZ_19
CZ_20
Component
Name
Component_2
Description
File
C:\Users\Zuhal Duran\Desktop\Cl

***Прим.:*** **Parametrize** creates Distance, Radius, Rigid Sets, Fix, Coincidence, Concentric constraints depending on the geometric model.

***Прим.:*** Perpendicularity and tangency can be set (if they are applicable) by changing the DMRECOGNIZE system variable in the **Settings** dialog box.

5 To check easily which parameter corresponds to what, right-click one of the parameters, and choose **Animate** (5) in the context menu.



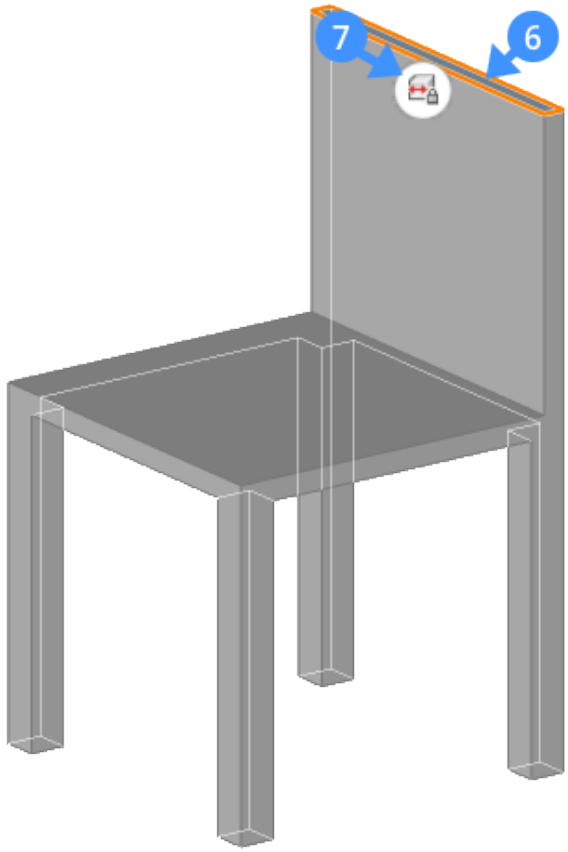

***Прим.:*** In the attached .dwg file after parametrizing the chair geometry, parameters and constraints will be assigned and shown in the **Mechanical Browser**. Each parameter corresponds to one measurement in the chair geometry. For instance, **Length_X_Y** controls the width and depth, **Length_Z** controls the overall height, **p_1** controls the seat thickness **p_2** controls the leg thickness **ratio_1** controls the ratio between seat height and overall height.

### 20.3.23.4 Procedure: changing the parameters and constraints in the parameters manager

***Прим.:*** After the auto parametrizing, it's possible to change what you do not find clear about the parameters and constraints using the parameters manager.

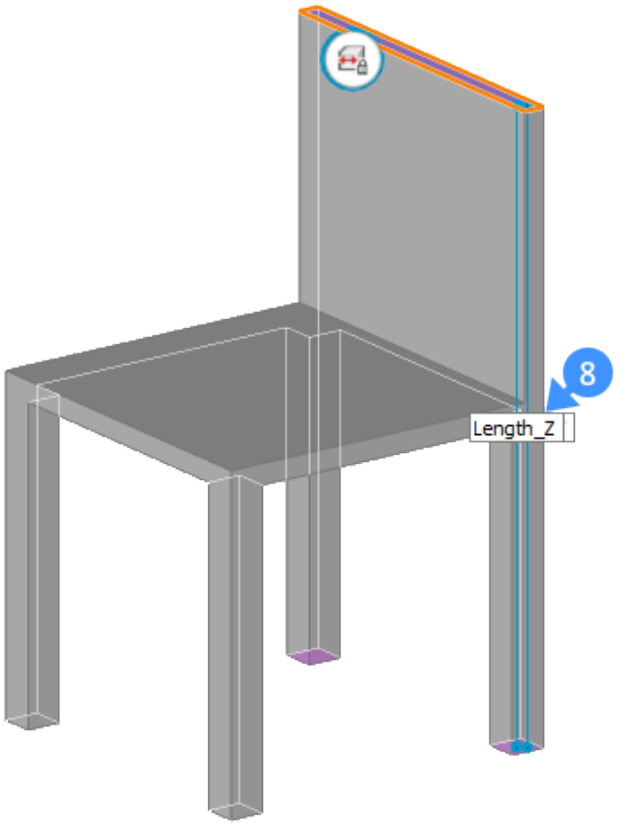
1 Select one of the faces (6) of your component. The constraint type (e.g. distance, fix) regarding the

face is displayed.

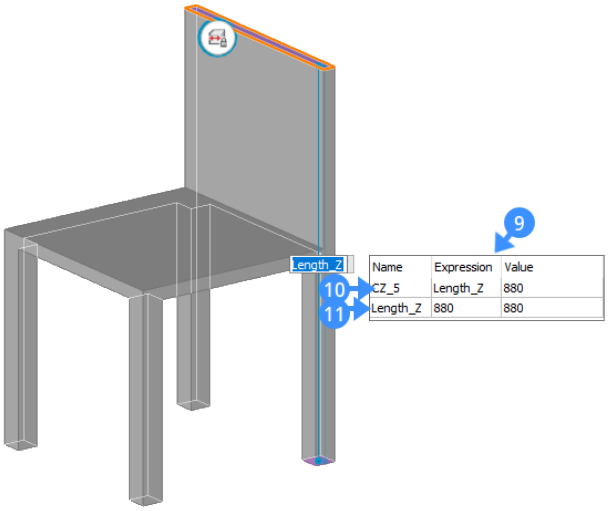



2 Click the distance constraint (7) icon on the element. The editable dimension(s) is shown inside the box.

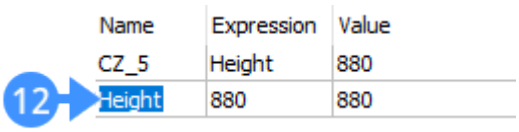
3 Click the editable dimension box (8).




A parameter manager displays (9). The parameter manager shows the expressions of the distance constraint (10) and the parameter (11) depending on the selected face.



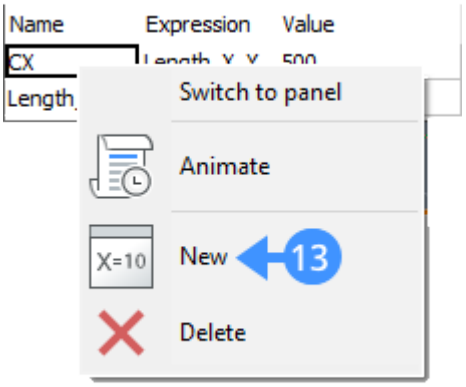

4 Change the parameter name by clicking its field (12) in the parameters manager.



5 (**Optional**) A new parameter can be defined using the **New** option in the parameter manager's context menu.

6 To do so, right-click the parameters manager and select **New** (13).

7 A new parameter is added to the list (14).




| Name | Expression | Value |
|---|---|---|
| CX | Length_X_Y | 500 |
| Length_X_Y | 500 | 500 |
| v22 | 1 | 1 |

14

8 Specify a **Name** (15) for the newly created parameter and enter an expression using the **Expression** field (16).

| Name | Expression | Value |
|---|---|---|
| CX | Length_X_Y | 500 |
| Length_X_Y | 500 | 500 |
| Width | 500 | 500 |

15 16

9 To link the newly created parameter with an existing constraint, change the expression field of the constraint with the new parameter name (17).

| Name | Expressi | Value |
|---|---|---|
| CX | Width | 500 |
| Length_X_Y | 500 | 500 |
| Width | 500 | 500 |

17

***Прим.:*** The above procedure can also be done using the **Mechanical Browser**.

### 20.3.24 Working with parameters and constraints

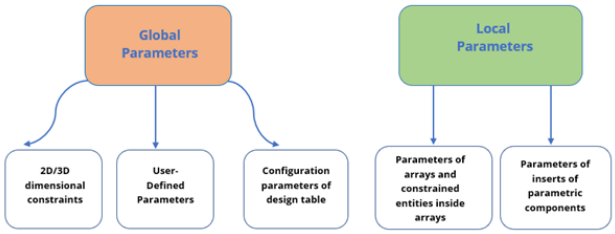
#### 20.3.24.1 Parametric modeling

In BricsCAD, parametric modeling is done through the use of geometrical and dimensional constraints. A rich toolset controls the 2D and 3D geometric model with a set of parameters. Each parameter will be

displayed with a value in the **Mechanical Browser**. The value from the different parameters can be linked together via expressions. When one of the parameter values is changed, the geometric model is updated automatically. The toolset allows BricsCAD users to add parametric behavior to any geometry and easily explore design intent.

***Прим.:*** In BricsCAD, there are no parent-child dependencies between geometric elements. For instance, if you change a sketch used to create an extruded 3D solid, the solid is not changed accordingly. Any kind of dependencies can be created using the parameters and expressions.

There are two types of parameters: Local parameters are attached to a particular entity. Global parameters are not attached to a particular entity.




### 20.3.24.2 Working with constraints

If you create some components (e.g. windows, doors,...) in your BIM model, you can parametrize them by applying 3D constraints. Defining constraints allows you to control the shape and size of the elements. Together with constraints, parameters determine the positions of entities through an expression.

There are two types of 3D constraints in BricsCAD: one specifies the size of the entities, the other locate their positions.

### 20.3.24.3 Geometrical constraints

Geometric constraints allow you to control the position of 3D entities with respect to each other.




Toolbar: **Parametric** > **3D constraints**

Quad: **Constraints**

The following table shows the 3D geometrical constraints.

| | |
|---|---|
| **Fix** | Keeps solids, edges or faces of solids in-place in the drawing. |
| **Coincident** | Applies a coincident constraint between two edges, two faces or an edge and a face of two different solids. |
| **Concentric** | Keeps two cylindrical, spherical or conical surfaces centered. |

| | |
|---|---|
| **Parallel** | Keeps the two faces of a solid or of different solids parallel. |
| **Perpendicular** | Keeps the two faces of a solid or of different solids perpendicular. |
| **Tangent** | Keeps a face and a curved surface of different solids tangent. |
| **Rigid set** | Makes a set of entities or sub-entities a rigid body. |

### 20.3.24.4 Dimensional constraints

Dimensional constraints allow you to control the sizes of 3D entities in drawings and the distances between them.




Toolbar: **Parametric** > **3D constraints**

Quad: **Constraints**

Ribbon: **Parametric Tab** > **3D constraints**

The following table shows the dimensional constraints.

| | |
|---|---|
| Distance | Creates a distance between two sub-entities. |
| Radius | Creates a radius to cylindrical surfaces or circular edges. |
| Angle | Creates an angle between two faces of a solid or of different solids. |

### 20.3.24.5 Setting an expression to a parameter

In BricsCAD, you can set an expression to any parameter, be it global or local. As an example, you can create a simple expression that only contains a number or a name of a global parameter and apply it to the related geometry in the drawing. You can also use more complex formulas that include the use of standard operators and functions.

| | |
|---|---|
| NOTES | • Local parameters cannot be referenced by names in expressions.<br>• The constants Pi=3.14... and e=2.72... can be used in expressions. The constants names are not allowed to be used as parameter name and name of 3D constraint. |

The following table shows the operators that can be used in expressions.

| | |
|---|---|
| Addition (+) | Produces the sum of numeric values. |
| Subtraction or Negative (-) | Subtracts the two numeric values. |
| Multiplication (*) | Multiplies the numeric values. |
| Division (/) | Divides two numeric values. |
| Exponentiation (^) | Calculates the exponential value of the given numbers. |
| Modulo or Remainder operator (%) | Gives the remainder after the division of one numeric value to another<br>e.g. The expression "5 % 2" would evaluate to 1, because 5 divided by 2 leaves a quotient of 2 and a remainder of 1. |

The following table shows the functions and the syntax which can be used in expressions.

| | |
|---|---|
| Cosine | cos(expression) |
| Sine | sin(expression) |
| Tangent | tan(expression) |
| Arc cosine | acos(expression) |
| Arc sine | asin(expression) |
| Arc tangent | atan(expression) |
| Hyperbolic cosine | cosh(expression) |
| Hyperbolic sine | sinh(expression) |
| Hyperbolic tangent | tanh(expression) |
| Arc hyperbolic cosine | acosh(expression) |
| Arc hyperbolic sine | asinh(expression) |
| Arc hyperbolic tangent | atanh(expression) |

| | |
|---|---|
| Square root | sqrt(expression) |
| Signum function (-1,0,1) | sign(expression) |
| Round to nearest integer | round(expression) |
| Truncate decimal | trunc(expression) |
| Round down | floor(expression) |
| Round up | ceil(expression) |
| Absolute value | abs(expression) |
| Largest element in array | max(expression1;expression2) * |
| Smallest element in array | min(expression1;expression2) * |
| Degrees to radians | d2r(expression) |
| Radians to degrees | r2d(expression) |
| Logarithm, base e | ln(expression) |
| Logarithm, base | 10 log(expression) |
| Exponent, base e | exp(expression) |
| Exponent, base 10 | exp10(expression) |
| Power function | pow |
| Random decimal, 0-1 | Random(expression1;expression2) * |

*Use the list separator character as defined on your system: , (comma) or ; (semicolon).

### 20.3.24.6 Parametric blocks

Parametric Blocks refer to blocks that have parameters that determine the size and shape of the 3D geometry. Once the parametric block is created, it can be inserted as a block reference in the drawing. The parameters of your block can be changed after you insert it in your model. Thus, no need to modify the parametric block file individually. The parametric blocks also enable you to use the same block in different sizes and in different shapes in your model.

### 20.3.24.7 Procedure: parametrizing a custom component

The window that was created in the previous lesson is used to demonstrate how the parametric design works. This window consists of three solids: subtractor, fixed frame, and glass pane.

**Step 1: Before starting to add constraints:**

1 Open the **Mechanical Browser** with the MECHANICALBROWSEROPEN command, to control and manage the values of 3D constraints and parameters.

***Прим.:*** The **Mechanical Browser** allows to navigate through all the constraints and parameters in the drawing and to edit dimensional constraints.

2 Make sure that the **Enable selection of 3D solid faces** () is toggled on in the Selection Modes To easily follow the steps, make sure Boundary Detection is toggled off.

**Step 2: Applying fix constraints**

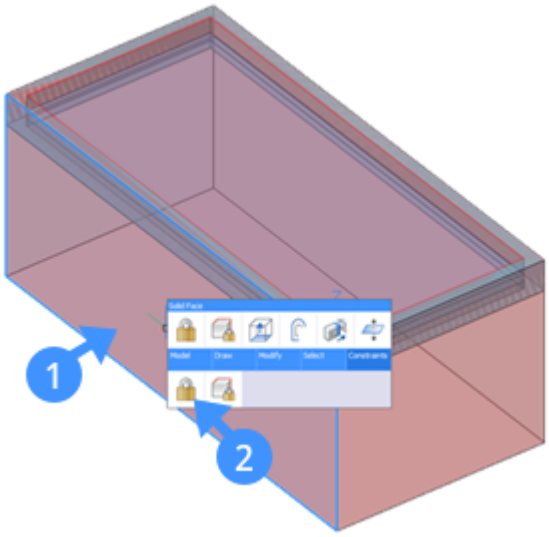
Command: DMFIX3D

1 Turn on the **BIM_Subtract** layer.

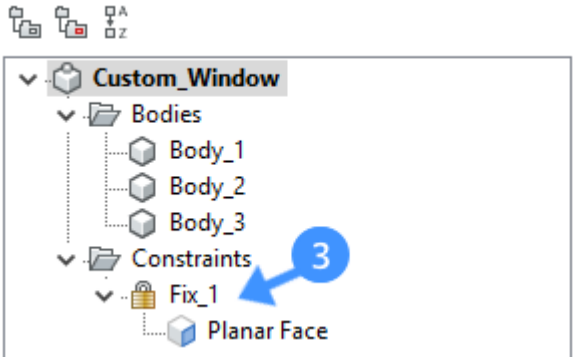
2 Select the entities (1) you want to add a fixation to.

3 (**Optional**) Press the TAB key to select obscured geometry.

4 Choose **Add Fixation** (2) in the 3D constraints command group in the Quad.




The fixation appears in **Mechanical Browser** as **Fix_1**.




**Step 3: Applying rigid constraints**

Command: DMRIGIDSET3D

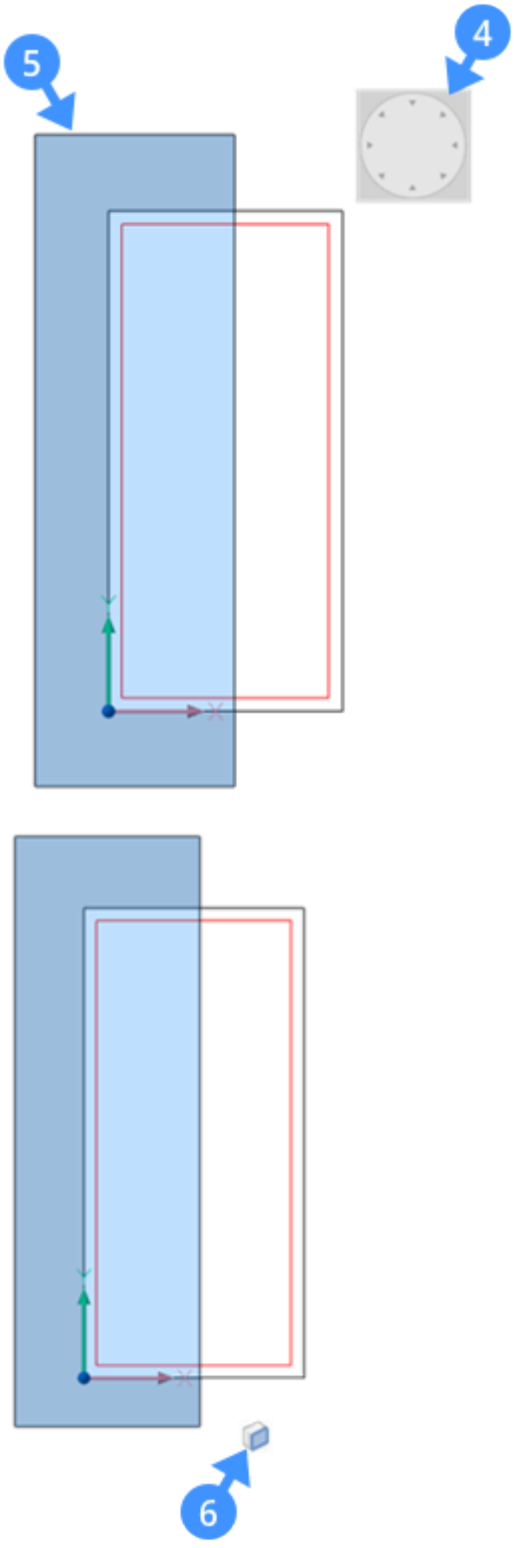
1 Select the entities to apply the **Rigid Set** constraints.

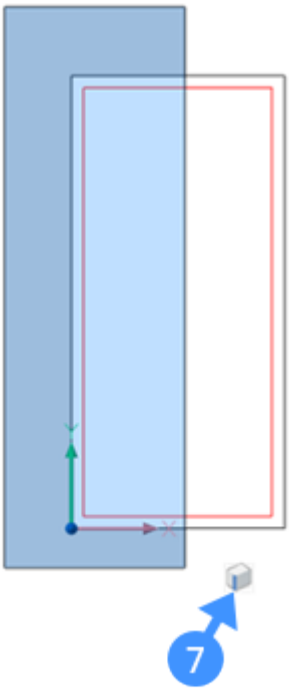
The entities remain in the same position with respect to each other when they are in a Rigid Set.

To select more than one sub-entity of the same type at a time, take the following steps:

- Use **Look From Widget** (4), by default it is at the top right of your screen, to change the view from 3D to TOP.
- Select the entities using the selection box (5). While drawing a selection box, press CTRL key to change the type of sub-entity selection. Every time you press CTRL key the type of the sub-entity selection will change. The type of the sub-entity can be seen under your mouse cursor, for instance, faces (6) or edges (7).

***Прим.:*** By default, the selection boxes see and select solids.


5
4
6




2 Choose **Add Rigid Set** ( ) in the 3D constraints command group in the Quad.

3 Repeat the same process for the other sides: top, bottom, and right side.

**Step 4: Applying distance constraints**

Command: DMDISTANCE3D

Dimensions of the window and the thickness of the glass pane.

1 Turn off the **BIM_Subtract** layer.

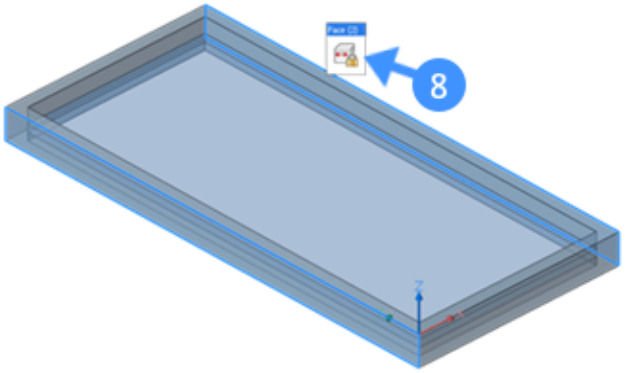
2 Select the first face.

3 The face highlights.

4 Hover over the second face.

5 Hit the TAB key to highlight an obscured face.

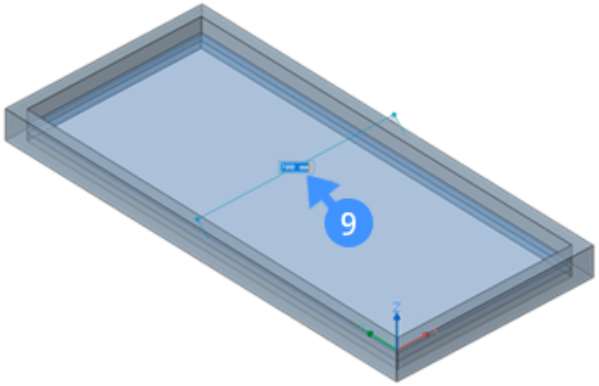
6 When the second face highlights, choose **Add Distance** (8) in the 3D Constraints command group in the Quad.




You are prompted: Specify distance value or [Geometry-driven] <xxx.xx>:

7 Do one of the following:

- Press Enter key or right-click to accept the current value.
- The current value displays in the dynamic dimension field (9) when the **Dynamic Dimensions** (DYN) is active.




- Type a value in the Command line.

8 The distance value between the two surfaces appears in the **Mechanical Browser**.

***Прим.:*** To remove any constraint in your model, select the constraint in the **Mechanical Browser**, e.g. fix, rigid set..., and press the Delete key or right-click and select the **Delete** option.

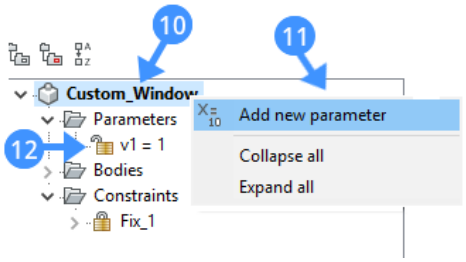
**Step 5: Applying parameters**

We will create a parameter that controls the width of the window.

1 Right-click the file name (10) in the **Mechanical Browser**.

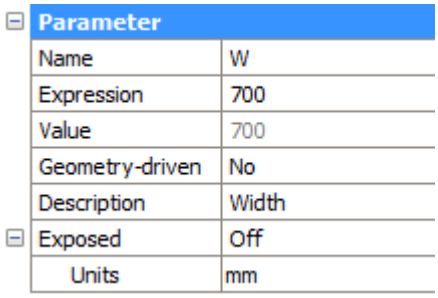
2 Select **Add New Parameter** (11) in the context menu.

The parameter settings grid is created and the parameter is added in **Mechanical Browser** as **v1 = 1** (12).




3 Select the parameter, then edit its properties in the settings grid.

4 The following illustration shows the properties of a **Width** parameter of the custom window. The table below demonstrates each property.




| | |
|---|---|
| **Name** | Type a parameter name. |
| **Expression** | Type a value or formula. |
| **Value** | Shows the current value of the parameter. |
| **Geometry-driven** | If yes, makes the parameter geometry-driven. |
| **Description** | Define an optional description of the parameter. |
| **Exposed** | Controls whether the parameter is available in the properties panel when the component is inserted into a model. |
| **Units** | Specifies whether the parameter is linear, areal or volumetric. |

5 To link the parameter you just created with the dimensional constraint, select the distance constraint in the **Mechanical Browser** and use the parameters to formulate an expression in the **Expression** field in the settings grid of the constraint.

| Constraint | |
|---|---|
| Type | Distance |
| Name | Distance_1 |
| Expression | W |
| Value | 700 mm |
| Enabled | Yes |
| Exposed | Off |
| Directions | Same |

6 **(Optional)** Change the expression value of the parameter and press Enter. The assigned distance constraint is changed accordingly.

*In your drawing, you can design as many constraints as possible. Note that the software will keep you away from applying more constraints than are necessary.

## 20.4 Данные здания

### 20.4.1 Analytical model

#### 20.4.1.1 Commands

BIMANALYTICALMODEL

#### 20.4.1.2 About the BIM analytical model

An Analytical Model is in essence a simplified wireframe model that only contains the load-carrying building elements. The linear building elements like columns, beams and members are represented by lines. The planar building elements like walls and slabs are not yet supported in the analytical model.

In BricsCAD, the BIMANALYTICALMODEL command allows you to generate an analytical model from a full 3D model of a structure. The command starts with an automatically generated proposal but allows you to further adapt the model to your desire. The automatic proposal will try to minimize rigid links (eccentricities) as much as possible by shifting, extending and even rotating some axis lines. The number of such changes can be controlled by the deviation settings.

You can modify the generated proposal afterward by using the options when you select an element (nodes or axes). To propagate the effect of a local change further you can use recalculate. This will launch a global solution that again tries to minimize rigid links but will respect the changes you made.

After editing the model, you can export it to an IFC file and a CIS/2 file. This analytical model can then be used for further structural analysis after import in a dedicated analysis software package.

It's important to emphasize that the input model has to be fully classified since the BIMANALYTICALMODEL command assumes that all the load-carrying elements are indicated in the building element properties.

In the procedures, the analytical model is explained with an example model. You can find this model in the appendix.

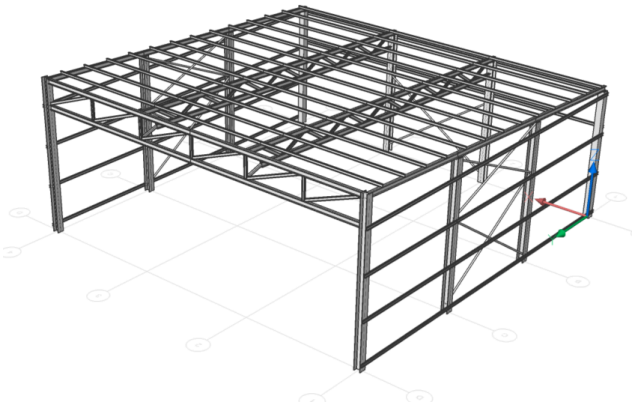
### 20.4.1.3 Procedure: creating an analytical model with strict deviation settings

1 First, you have to classify your BIM model. This is necessary for using the BIMANALYTICALMODEL command.

1 Use the BIMIFY command on your model or classify elements manually.

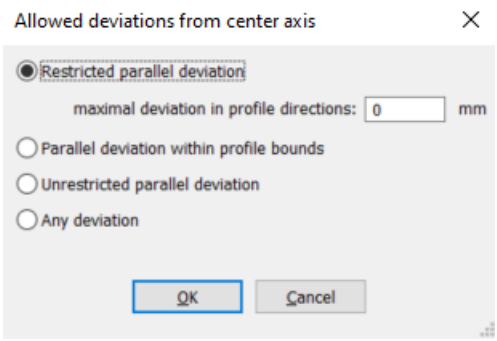
**You are prompted:** *run Bimify on entire model or [Advance] <model>*

2 Press Enter if you want to classify everything in your model or type **A** if you want a selection of elements.



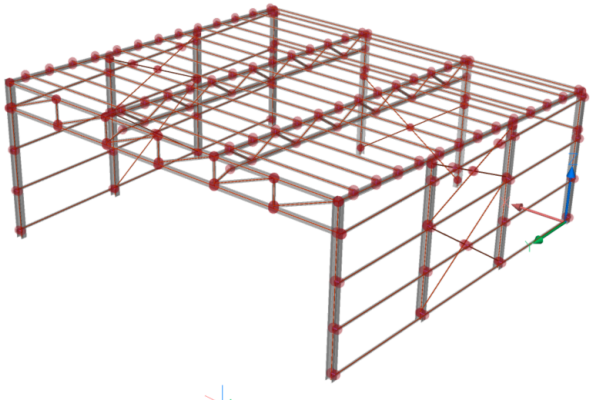
3 Launch the BIMANALYTICALMODEL command in the Command line.

4 A pop-up window opens.

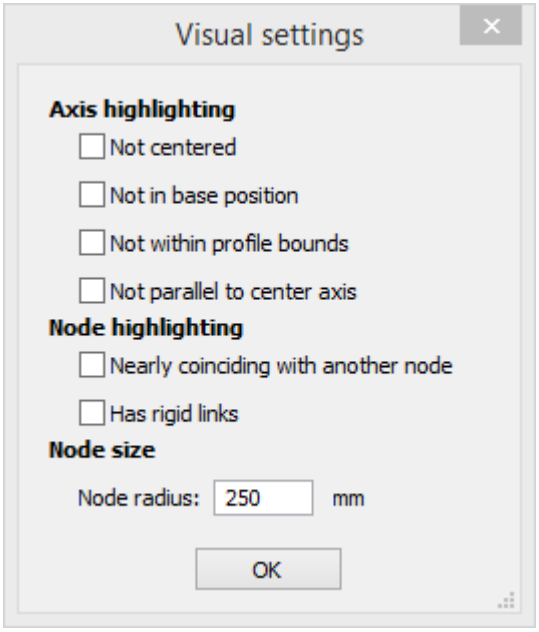



This window allows you to set the level of freedom used when generating an analytical model. When hovering over one of the options, a tooltip shows you what type of restrictions are imposed for that option. In this case, choose the **Restricted parallel deviation** with a maximal deviation in profile directions of 0 mm. No axis is allowed to deviate from its central position in the profile.

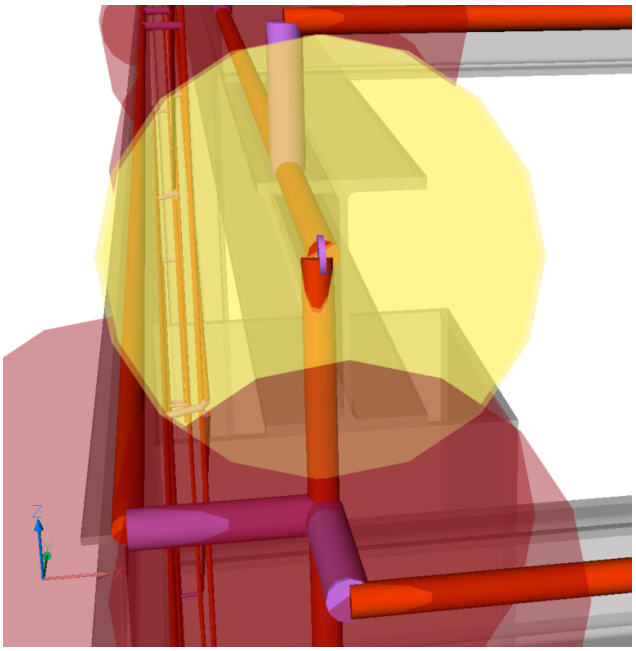
5 An interactive preview of the analytical model is constructed based on the chosen global deviation setting.



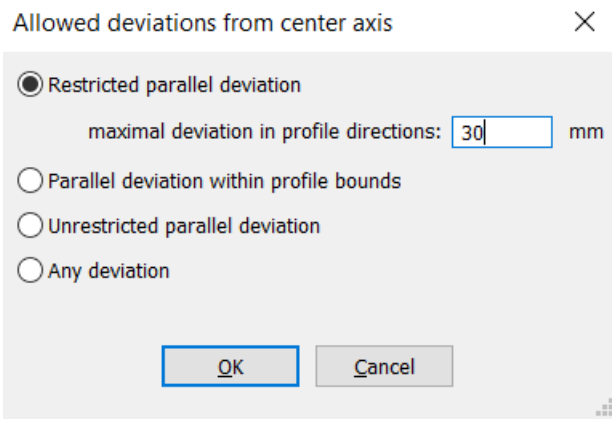
Also, the visual settings window appears. In this window, you can check and uncheck different features of axes and nodes to highlight (in yellow) the axes and nodes which correspond to those features. This allows you to explore the current properties of the axis and nodes of the analytical model. Based on what you find you can determine if any adjustments need to be made. You can also change the size of the disks representing the nodes. This is useful in combination with the node highlighting, which can be used for example to find nearly coinciding nodes.




6 Select the checkbox **Nearly coinciding with another node** in the **visual settings** window. You will see that the center axes at these nodes do not align perfectly. As a result, small rigid links (eccentricities) had to be introduced. This is because of the previous setting in step 4. These small rigid links are displayed in purple.

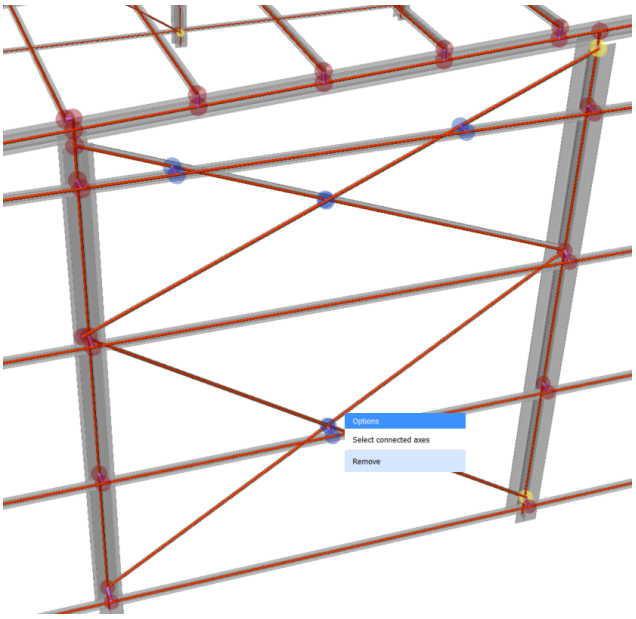


To avoid this, restart the command with a slightly more flexible setting. In the **Allowed deviations from center** window, choose again for **restricted parallel deviation** but this time with a maximal deviation in profile directions of 30 mm.

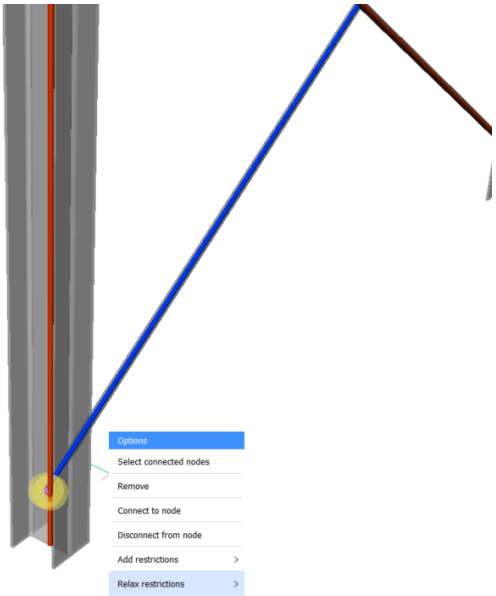



7 For the detection of nodes in the preview of the analytical model, the proximity of structural elements is used. Because of this, it is possible that there arise nodes in the model where tie rods cross. You can easily remove these nodes by clicking on the node and select **Remove**.

***Прим.:*** You can select several nodes at once using box selection.

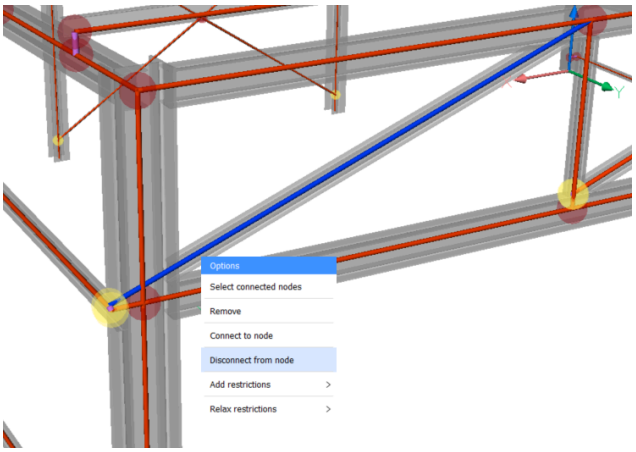



8 If you take a look at the remaining coinciding nodes, you see that the tie rods are not aligned perfectly with the center lines of the columns and small rigid links are formed. You can remove these rigid links by relaxing the position of the axes of the tie rods to a non-parallel position. You can do this by clicking on the tie rod, click **Relax restriction** and click **Non-parallel**. The extra freedom will be used to remove the rigid links by moving the axes of the tie rods a bit.




9 The only coinciding nodes that are left are at the front truss of the hangar structure. If you take a look at the side of the front truss, you see that a tilted member is connected to the horizontal beam rather than the column.

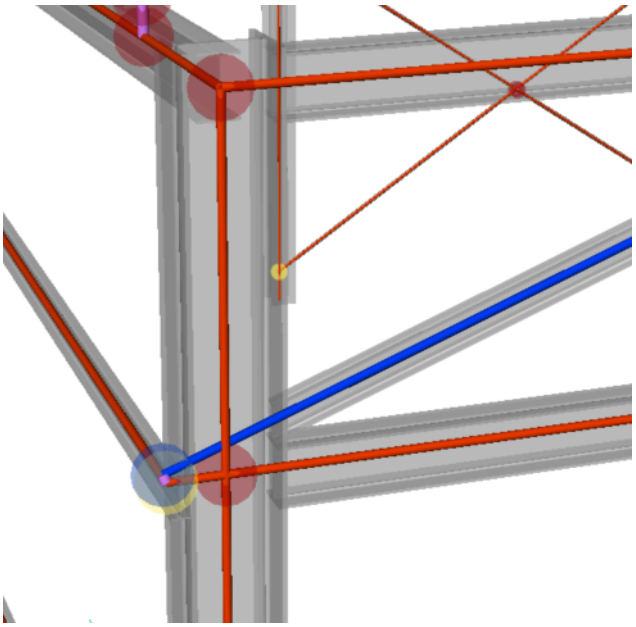
10 You can disconnect the tilted member by clicking on it and choose **Disconnect from node**.



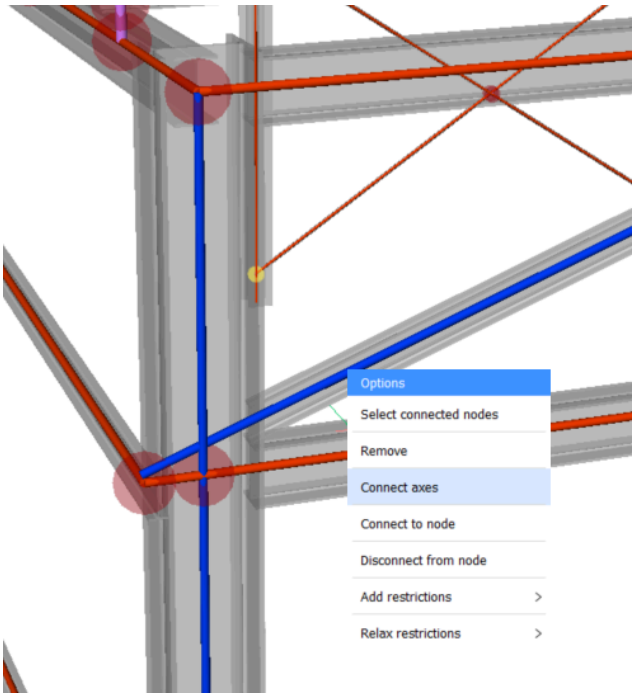

**You are prompted:***Select node disconnect from axes [Ok]*

Select the node you want to disconnect from.

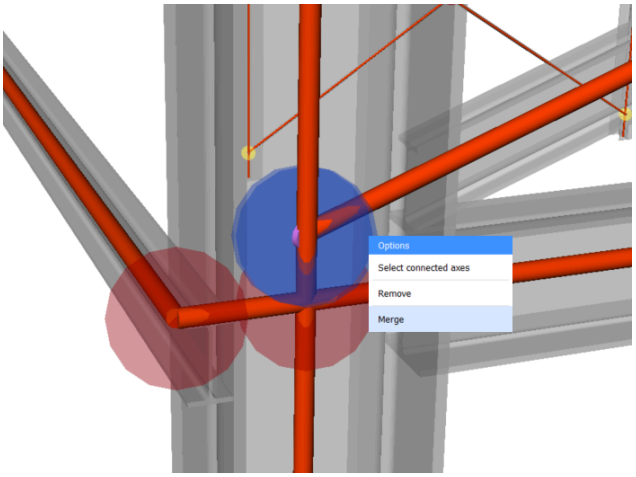
Type **O** to choose Ok.



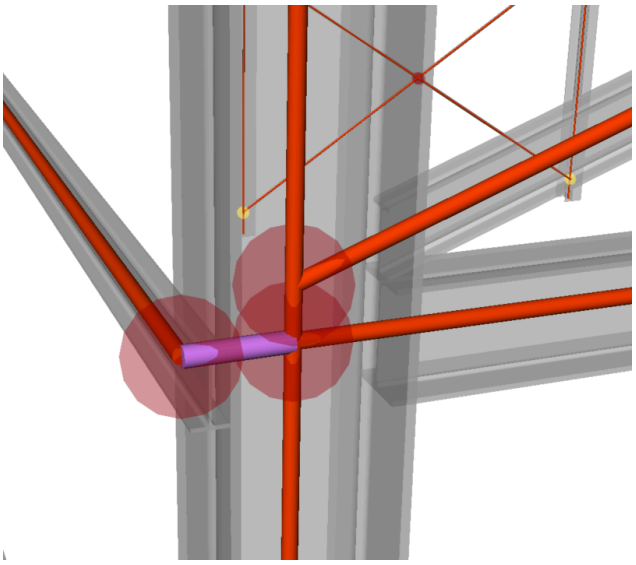
Now select the two axes that you want to connect and click **Connect axes**.




With the global deviation setting, you chose this can only be done by introducing a small rigid link again. You can remove this by merging the two nodes. Select both nodes and click **Merge**.

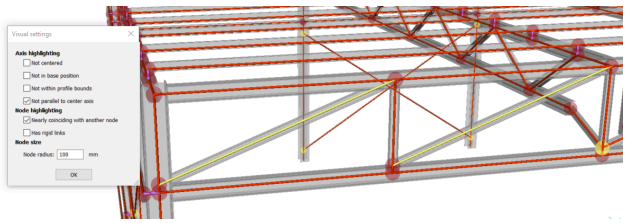



11 You see that the side member also is attached to the front beam instead of the column. Repeat step 9 for the side member.



12 You can remove the last coinciding nodes in the front truss by merging the nodes. This will shift the axes of the tilted members and introduces rigid links on the other end.

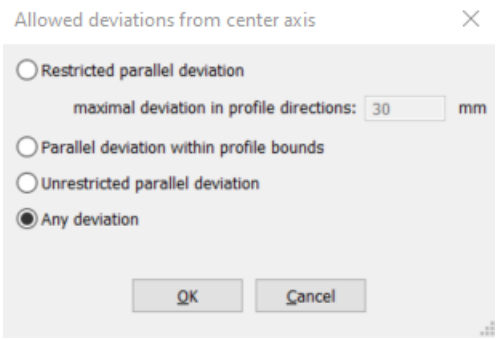
***Прим.:*** If you take a look at the axis **Not parallel to center axis** by using the **Visual settings** window, you will see that the tilted member in the front truss has been allowed to be non-parallel to the center axis. So, you could also remove the small rigid links by relaxing the tilted member to a non-parallel position like as we did for the tie rods.



### 20.4.1.4 Procedure: creating an analytical model with the least restrictive deviation settings

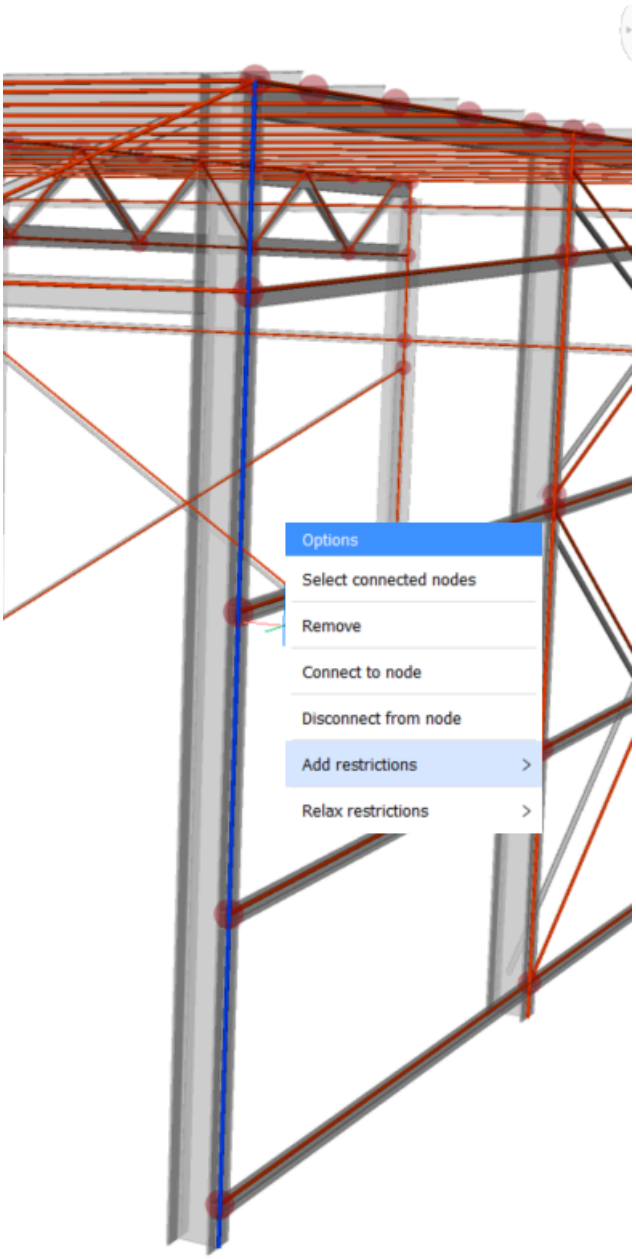
1 Repeat the steps from the previous procedure until step 4.

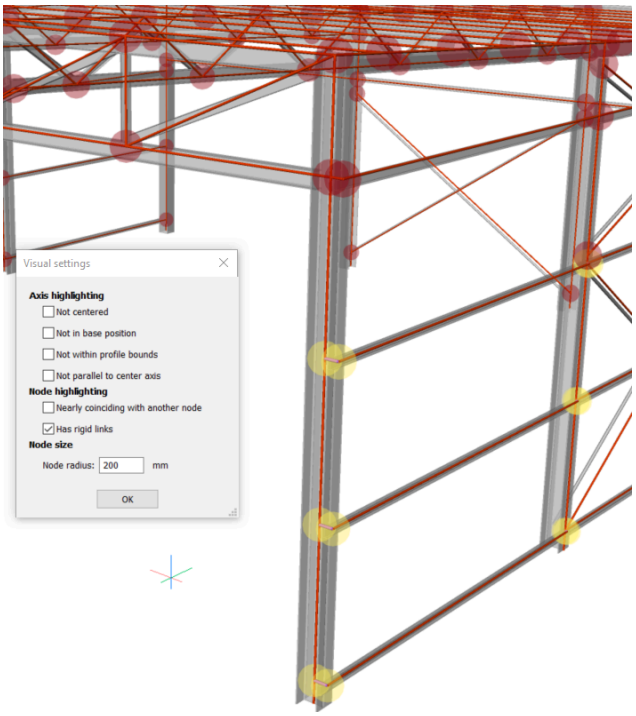
2 Choose **Any deviation** in the **Allowed deviations from center axis**. This allows all kinds of freedom to avoid any rigid links. In the preview of the analytical model, you will see that some axes have been tilted. For example, the small members in the trusses are not centered and the column axis is moved towards the profile bounds to find connections with smaller side members there.




3 Remove the unwanted tie rod nodes as in step 7 of the procedure with a strict deviation setting.

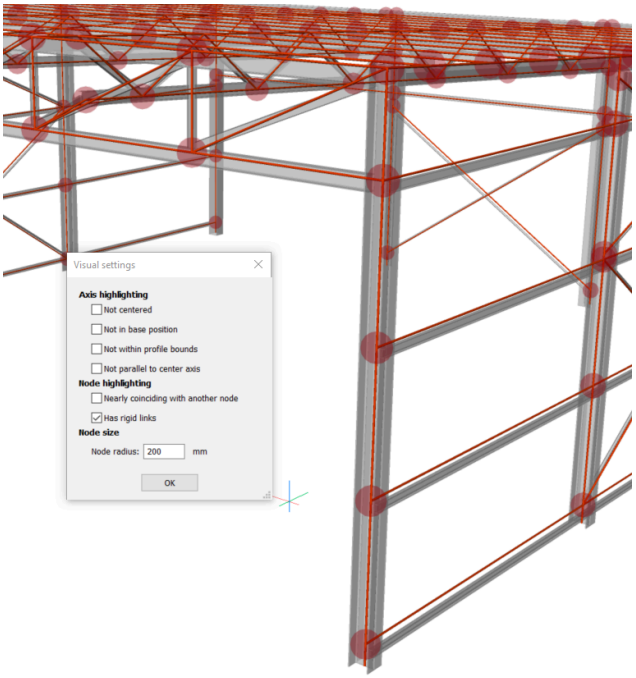
4 In the interactive preview of the analytical model, the column axes were moved towards the profile bounds. In this step, you will center the axis of the columns. You can do this by clicking on the axis of the column, click **Add restrictions** and click **To center**. This introduces rigid links very locally.


Options
Select connected nodes
Remove
Connect to node
Disconnect from node
Add restrictions
Relax restrictions




5 In the visual settings window, select the checkbox **Has rigid links**.

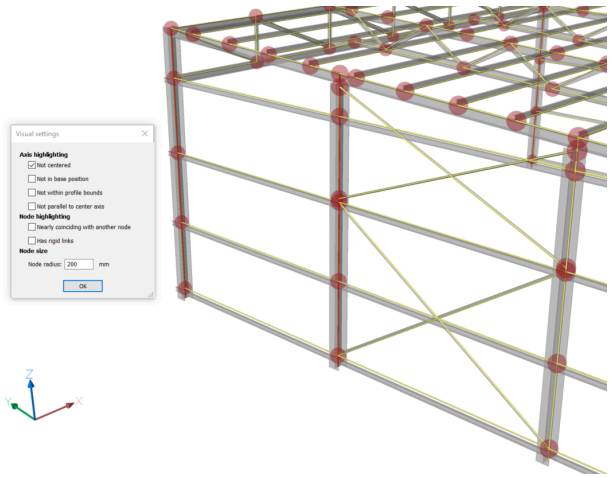
Type **R** in to choose **Recalculate**.




6 In the visual settings window, select the checkbox **Not centered**.

You will see that one of the columns is still not centered because the connection at its top includes not only the larger beams but also one of the smaller ones. You change this by imposing a center

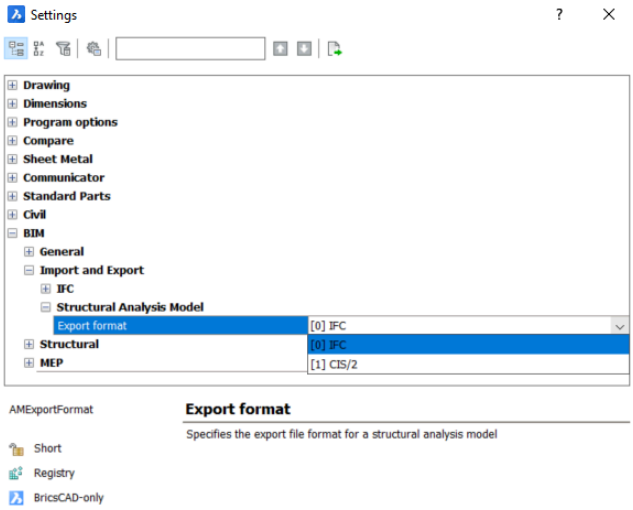
restriction on this column as well.



### 20.4.1.5 Procedure: exporting an analytical model

1 Before starting the command, open the **Settings** dialog.

2 In the Settings dialog, click the option **Structural Analysis Model** and choose the export format of the analytical model. You can choose between **IFC** and **CIS/2**.




3 Launch the BIMANALYTICALMODEL command.

4 Change the analytical model to your desire.

5 When you have finished the model, type **E** to choose Export and exit.

6 The document window appears where you can name and save the file in the desired folder.

***Прим.:*** There are some remarks about the exported result.

- The **node concept** used by BIMANALYTICALMODEL does not completely coincide with **CIS/2** and **IFC** node concept:

  In the BIMANALYTICALMODEL, only connections between linear elements are modeled with a node. For example, columns will not have a floor node if there is no connection to another

element there. During export, missing end nodes are added to be compatible with CIS/2 and IFC export.

In the BIMANALYTICALMODEL, a node always lies exactly on every connected axis. Eccentricities are modeled with explicit visual rigid links (purple bars in preview). During export, each cluster of BricsCAD nodes that are interconnected with rigid links is translated into a single connection node with eccentricities in **CIS/2** and **IFC**.

- The **axis concept** used by BIMANALYTICALMODEL does not always coincide with the **CIS/2** and **IFC** axis concept:

  In the BIMANALYTICALMODEL, there will be one single axis for each physical element. An axis can be attached to more or less than two nodes. During export, an axis that has multiple nodes along its length gets split up into several analysis members, each with one end and one start node, for the **CIS/2** and **IFC** export.
- The concept of **Base Position** in BricsCAD is similar to **cardinal points**. Base positions in BricsCAD form a subset of the cardinal points specified in the **CIS/2** standard. For now, only cardinal positions 1 to 9 are supported.
- In the analysis model, as calculated by BricsCAD, an axis can shift away from the center of the physical entity, essentially to any point within certain bounds. This means that the final position of the analysis axis does not necessarily coincide with a cardinal point or base position of the profile. During export, the section profile is linked to the analysis axis using the closest base position of the profile to the final analysis axis, using the cardinal point concept in **CIS/2** and **IFC**. This means that if you visualize the structure in a **CIS/2** or **IFC** viewer that shows physical members as extrusions of the profile section along the analysis axes, you will see that the structure shown there can slightly deviate from the original structure as modeled or imported into BRICSCAD. The structure as described by the **exported analysis model** is therefore an **approximation of the actual structure**.

model.dwg

### 20.4.2 AutoComplete BIM data

#### 20.4.2.1 Commands

BIMAUTOMATCH

#### 20.4.2.2 About the AutoComplete BIM data

Instead of assigning compositions to every element manually, BIMAUTOMATCH detects the difference between interior and exterior walls and their respective composition and automatically assigns it to other walls. It can also make distinctions between different floors, different sides of the building, walls with a balcony, etc.



***Прим.:***

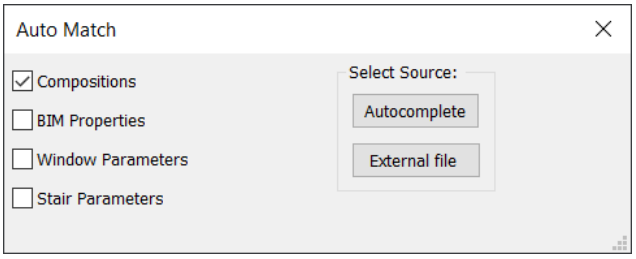
***Before you use BIMAUTOMATCH make sure that you have Bimified ( ) everything. AutoMatch works best with a fully Bimified model.***

### 20.4.2.3 AutoComplete BIM data in one file

1 Do one of the following:

- Select **Automatch** from the **Classify** tab in the Ribbon.
- Select **Model** > **Automatch** in the Quad.
- Type BIMAUTOMATCH and press Enter.

**You are prompted:** Compositions/BIM Properties/Window Parameters/Stair Parameters:



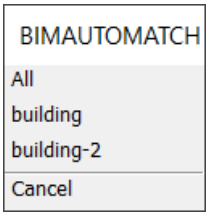

2 Select one of the following:

- **Compositions**: Copy compositions.
- **BIM Properties**: Copy BIM Properties.
- **Window Parameters**: Copy window parameters.
- **Stair Parameters**: Copy Stair Parameters.

3 Under **Select Source**, choose **Autocomplete**.

4 f the file contains multiple buildings, BIMAUTOMATCH asks you to define the source building.

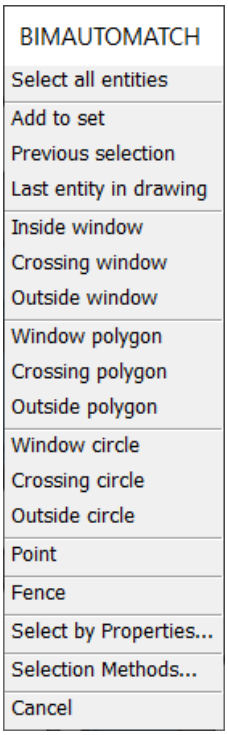
5 **You are prompted:**




Select the building from the menu.

6 If BIMAUTOMATCH finds a logical distinguishing feature between objects it will automatically use this. When it can not, BIMAUTOMATCH asks you to define the source composition or window.

7 **You are prompted:**

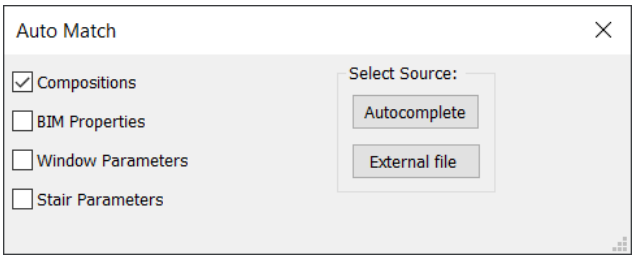



### 20.4.2.4 AutoComplete BIM data with an external file

1 Do one of the following:

- Select **Automatch** from the **Classify** tab in the Ribbon.
- Select **Model** > **Automatch** in the Quad.
- Type BIMAUTOMATCH and press Enter.

**You are prompted:** Compositions/BIM Properties/Window Parameters/Stair Parameters:




2 Select one of the following:

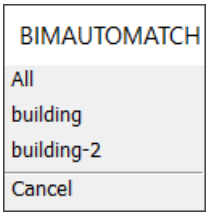
- **Compositions**: Copy compositions.
- **BIM Properties**: Copy BIM Properties.
- **Window Parameters**: Copy parameters of windows.
- **Stair Parameters**: Copy properties of stairs.

3 Under **Select Source**, choose **External file**.

4 Select source file and click **Open**.

5 If the file contains multiple buildings, BIMAUTOMATCH asks you to define the source building.

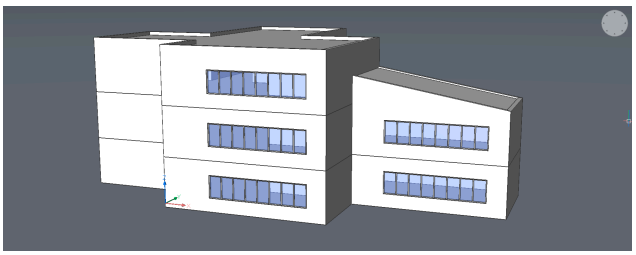
6 **You are prompted:**




7 Select the building from the menu.

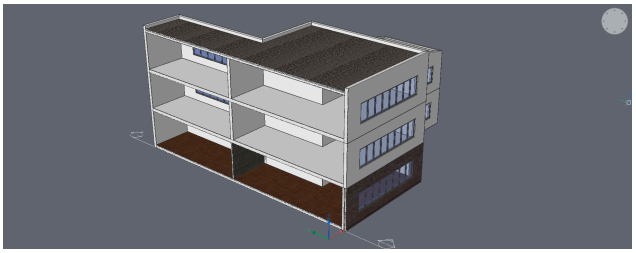
### 20.4.2.5 Procedure: using autocomplete in one file

1 Open up a file to start from. Here we use a basic rectangular building that looks like this:



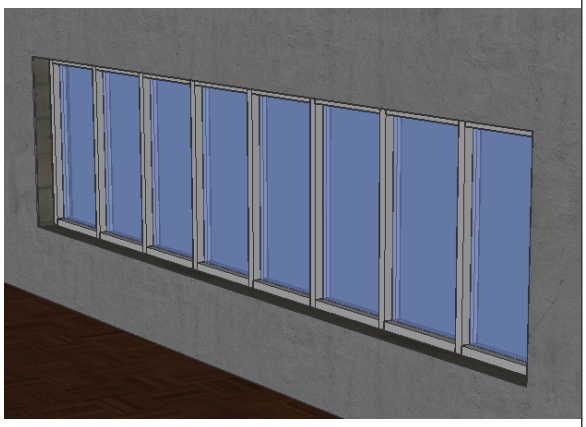
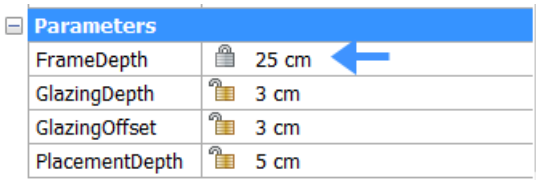
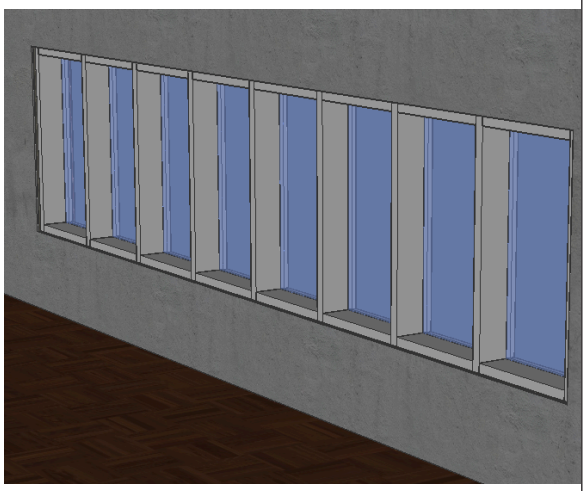
***Прим.:*** No BIM data or compositions should be added yet.

2 Add the compositions that you want. Don't do this for all your elements, but only for one element of each type. In our example we give a composition to 1 outer wall, 1 inner wall, 1 floor slab and the roof:



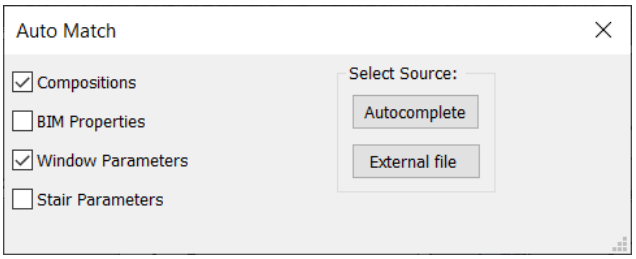
***Прим.:*** It could be that you have different types of outer walls, for example the wall facing north that needs more insulation. In this case, you give one of these northern walls a different composition.

3 In this example, we also want to adjust the parameters of our windows. Select only 1 of the windows and change the parameter you want in the properties panel.

| | |
|---|---|
|  | The window before changes. |
|  | Changing the **FrameDepth** in the property panel of the window. |
|  | The window after the changes. |

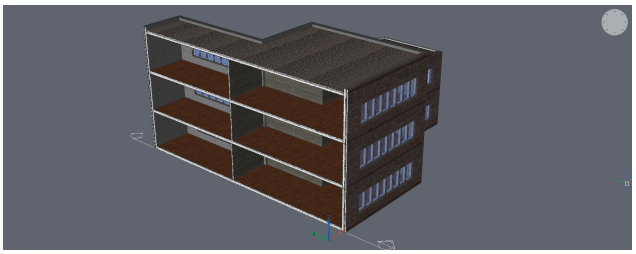
4 Select **Automatch** from the **Classify** tab in the Ribbon **or** select **Model** > **Automatch** in the Quad **or** type BIMAUTOMATCH and press Enter.

The following box appears:




We want the compositions and the window parameters that we chose to be completed in our entire drawing, so we check the boxes from **Compositions** and **Window Parameters**.

5 Select **Autocomplete**. The compositions and window parameters within your file are automatically copied. The drawing looks like this:

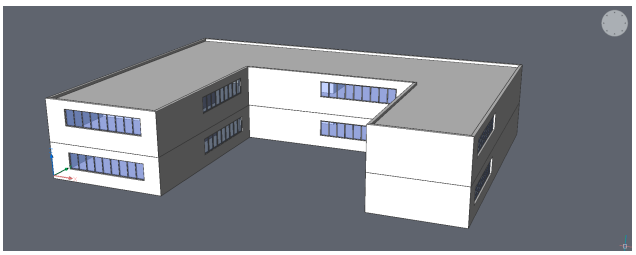


By using BIMAUTOMATCH, we very quickly have all our compositions and window parameters as we want them, without having to worry about forgetting anything.

***Прим.:*** You can see that different types of outer walls are no problem. All the walls facing north have been given the correct adjusted composition.

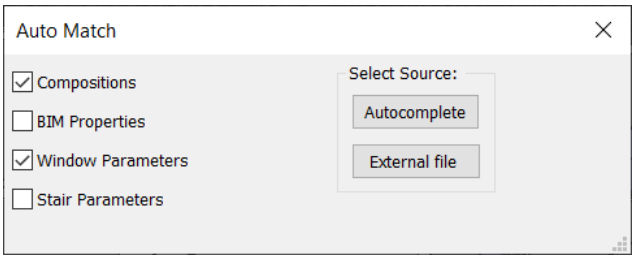
### 20.4.2.6 Procedure: using autocomplete with an external file

1 Open up a project that is similar in style to another project that you already finished. In this case, we take another rectangular building, similar to the one we looked at before:



2 Select **Automatch** from the **Classify** tab in the Ribbon **or** select **Model** > **Automatch** in the Quad **or** type BIMAUTOMATCH and press Enter.

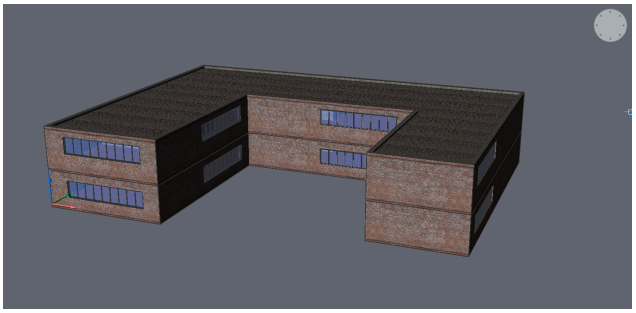
The following window appears:

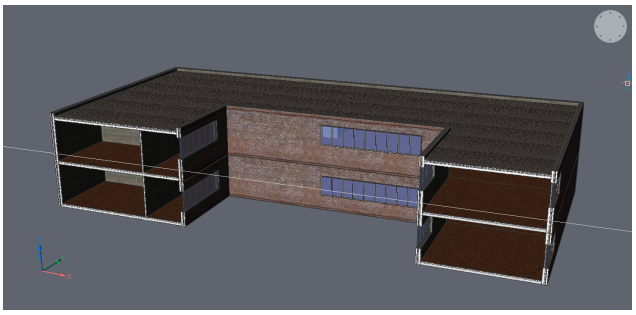


We want the compositions and the window parameters that we chose to be completed in our entire drawing, so we check the boxes from **Compositions** and **Window Parameters**.

3 Select **External file**.

4 Select the file that we made in the previous procedure. The compositions and window parameters from the external file will be copied to your new file. The drawing looks like this:





***Прим.:*** It can be handy to make a library of building styles with their typical compositions, in order to use BIMAUTOMATCH with external files in a very efficient way.

### 20.4.3 Attaching compositions

#### 20.4.3.1 Commands

BIMATTACHCOMPOSITION, BLCOMPOSITIONS

#### 20.4.3.2 About BIM attach composition

The BIMATTACHCOMPOSITION command allows you to easily apply a composition to a selection of solids. The assigned composition can be flipped or redefined to a different reference face. Optionally, you can align solids based on a reference face or the center of a composition.



See BIMATTACHCOMPOSITION for more information about attaching a composition.
See BIM Compositions for more information about managing compositions.

#### 20.4.3.3 Attaching a composition

To attach a composition, you can:

- Use the BIMATTACHCOMPOSITION command.
- Drag a composition from the **BIM Compositions** panel on the solid(s).

**Using the BIMATTACHCOMPOSITION command**

1 Launch the BIMATTACHCOMPOSITION command.

1 **You are prompted:** Enter the composition name or [Dialog/Entity] <Dialog>.

2 Do one of the following:

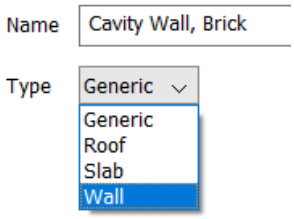
- Type a composition name in the Command line and press Enter.

  ***Прим.:*** Composition names are case sensitive.

- Press Enter to display the **Compositions** dialog and click on a Composition in the **In Project** or **In Library** list to open the menu of the composition.

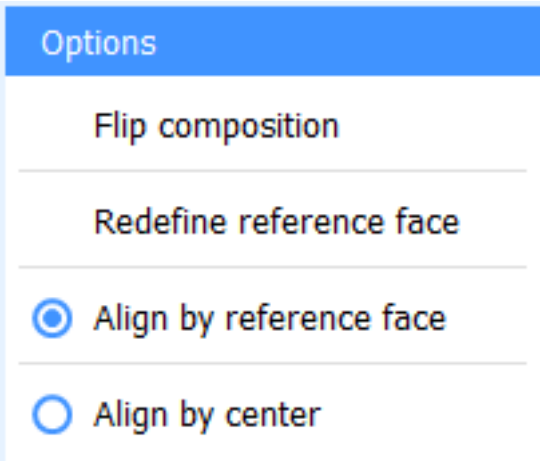
  If you want to select the composition without opening the menu, double-click it.

3 Choose a Composition **Type** from the drop-down menu and press **Select** to accept.




4 Select entities to attach the composition to and press Enter.

5 Choose the reference face. If you hover over the widget, a submenu displays showing different options. For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.




6 Press Enter to accept the reference face.

7 BricsCAD responds: The composition has been assigned to <xxx> element(s).

***Прим.:*** Using Bimify building elements are automatically classified. This makes it easy to select the right entities.

**Dragging a composition on a solid**

1 Select the composition in the BIM Compositions panel ( ). See Dockable panels.

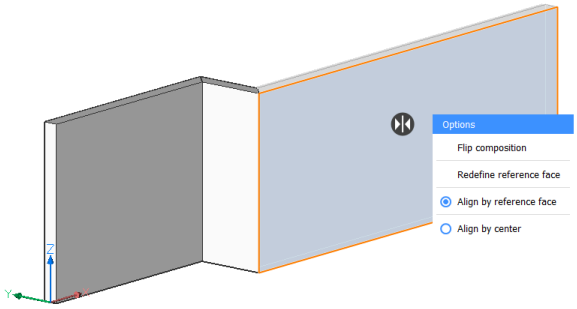
2 Hold down the left mouse button and drag the composition on the solid.

3 The solid face under the cursor highlights, indicating that this face is the reference face to attach the composition.

4 Release the left mouse button to attach the composition.

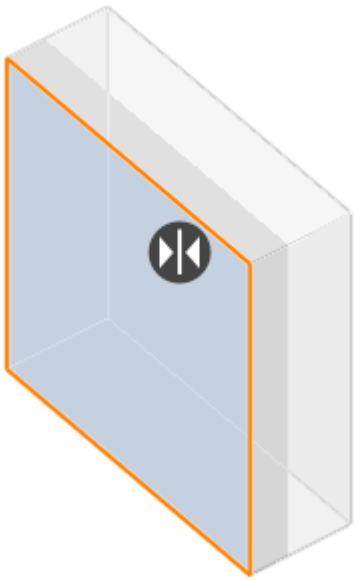
5 A widget appears that allows you to choose the reference face.

6 The reference face is fixed, while the opposite face moves if the thickness of the composition does not fit the current thickness of the solid.




For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

7 A darker volume indicates the volume and position of the solid after attaching the composition.



8 Press Enter to accept the current reference face and attach the composition.

***Прим.:*** When a single ply composition with a variable thickness is selected, it is not needed to select a reference face. The composition is attached to the selected solid.

**Dragging a composition on a selection of solids**

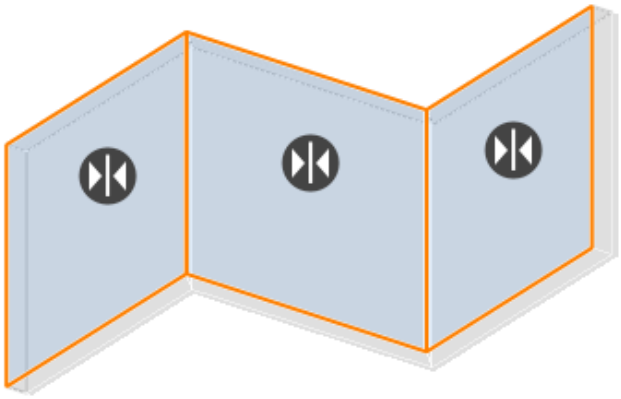
1 Select the solids you want to apply the composition to.

2 Select the composition in the **BIM Compositions** panel.

3 Hold down the left mouse button and drag the composition on any of the selected solids.

4 Release the left mouse button to attach the composition.

5 A widget appears which allows you to choose the reference face for every segment separately.



For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

6 Press Enter to accept the current reference face and attach the composition.

***Прим.:***

- BIM compositions are copied to the Project database upon attachment.
- Attaching a **Wall** composition to a solid, automatically classifies the solid as a wall.
- Attaching a **Slab** or **Roof** composition to a solid, automatically classifies the solid as a slab.

#### 20.4.3.4 Copy compositions from other solids

1 Launch the BIMATTACHCOMPOSITION command.

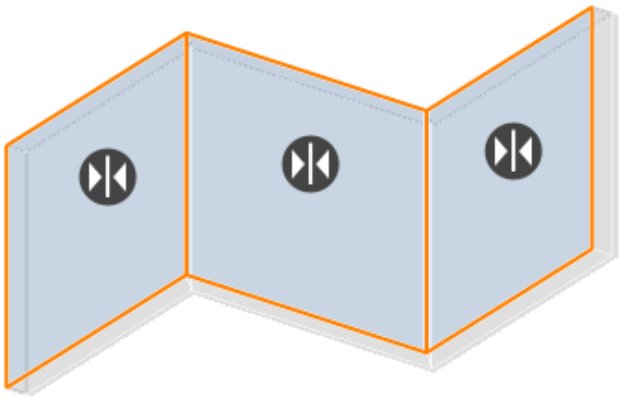
2 Select the **Entity** option.

Prompts you: **Select other entity to get composition:**

3 Select the solid to copy the composition from.

4 Select the solids(s) to attach the copied composition to.

5 A widget appears which allows you to choose the reference face for every segment separately.



For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

6 Press Enter to accept the current reference face and attach the composition.

#### 20.4.3.5 Removing compositions from solids

The **Unclassify** option of the BIMCLASSIFY command removes a composition from a solid.

1 Select the solid(s).

2 Choose **Unclassify** from the **BIM** command group in the Quad.

### 20.4.4 BIM classify

#### 20.4.4.1 Commands

BIMCLASSIFY

#### 20.4.4.2 About

If you have a conceptual model without any classification, you can turn these elements into classified BIM elements using BIMCLASSIFY.

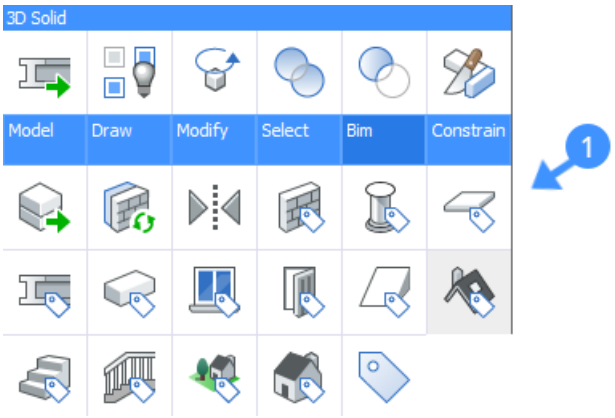


For more information about this command, visit the Command Reference article BIMCLASSIFY.

#### 20.4.4.3 Classifying entities

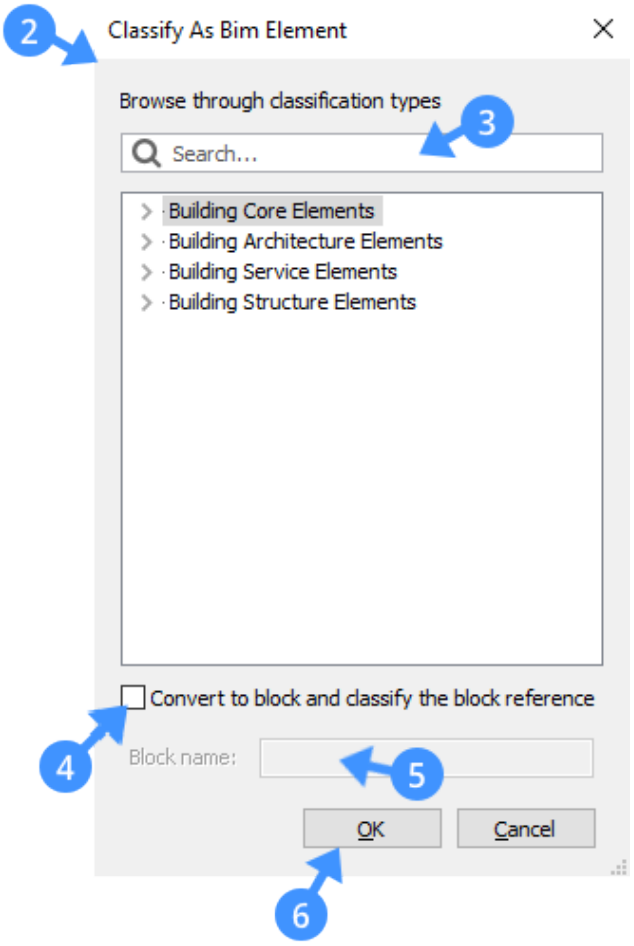
1 Select an entity in the drawing to classify.

2 Choose a classification type (1) in the **BIM** command group in the Quad.




The BIM data is assigned to the selected entity.

3 **(Optional)** Choose the **classify manually** option from the Quad to access the BIM Element classification types. This displays the **Classify as BIM Element** dialog box (2):




***Прим.:*** The classification types are grouped into the following categories:

- **Building Core Elements**
- **Building Architecture Elements**
- **Building Service Elements**
- **Building Structure Elements**
- **Spatial Structure Elements**

**Do one of the following:**

- Enter a specific name of the BIM Element type in the **Search** field (3) to filter the available element types.
- Select a Building Element type from the list.
- (**Optional**) Check the **Convert to block** option (4) and enter a name in the **Block name** field (5).

This option will convert the selected entity to a block and classify the block reference to the defined Building Element type.

- Click the **OK** button (6) to accept it.

4 **(Optional)** Use the Auto ( ) option from the Quad to classify the selected entities automatically.

This automatic classification is done based on the geometrical features of the object. Always check whether it was correctly classified.

5 To remove the assigned BIM data from the entity, select the entity in the drawing and choose the Unclassify () option from the Quad.

The BIM data is removed from the selected entity.

#### 20.4.4.4 Properties of the classified entities

When you classify an entity, a number of properties with regard to the building element type is displayed in the **Properties** panel.

The following illustrations show the properties that are available for windows and doors:

| Door Window Glazing Type | |
|---|---|
| Glass layers | 0 |
| Glass thickness 1 | 0 mm |
| Glass thickness 2 | 0 mm |
| Glass thickness 3 | 0 mm |
| Fill gas | |
| Glass color | |
| Is tempered | Off |
| Is laminated | Off |
| Is coated | Off |
| Is wired | Off |
| Translucency | 0 |
| Reflectivity | 0 |
| Beam radiation transmittance | 0 |
| Solar heat gain transmittance | 0 |
| Thermal transmittance summer | 0 W/m²·K |
| Thermal transmittance winter | 0 W/m²·K |

| BIM | |
|---|---|
| Type | Window |
| Building Element Type | Window |
| Name | |
| Description | |
| Building | |
| Story | |
| Space | |
| Composition | |
| GUID | 0shXs2TdfFB9RulemExNT4 |
| Space bounding | Off |
| Construction type | Not defined |
| Operation type | Not defined |
| Window inset | |
| Overall height | 0 mm |
| Overall width | 0 mm |
| Sill height | 0 mm |
| Head height | 0 mm |
| Assembly code | |
| Assembly subtype | |
| Number | |
| Entity property sets | |

| Window Common | |
|---|---|
| Reference | |
| Fire rating | |
| Acoustic rating | |
| Security rating | |
| Is external | Off |
| Infiltration | 0 m³/s |
| Thermal transmittance | 0 W/m²·K |
| Glazing area fraction | 0 |
| Smoke stop | Off |

| Door Window Shading Type | |
|---|---|
| External shading coefficient | 0 |
| Internal shading coefficient | 0 |
| Inset shading coefficient | 0 |

### 20.4.5 BIM compositions

#### 20.4.5.1 Commands

BLCOMPOSITIONS, BIMCLASSIFY, BLMATERIALS, BIMATTACHCOMPOSITION

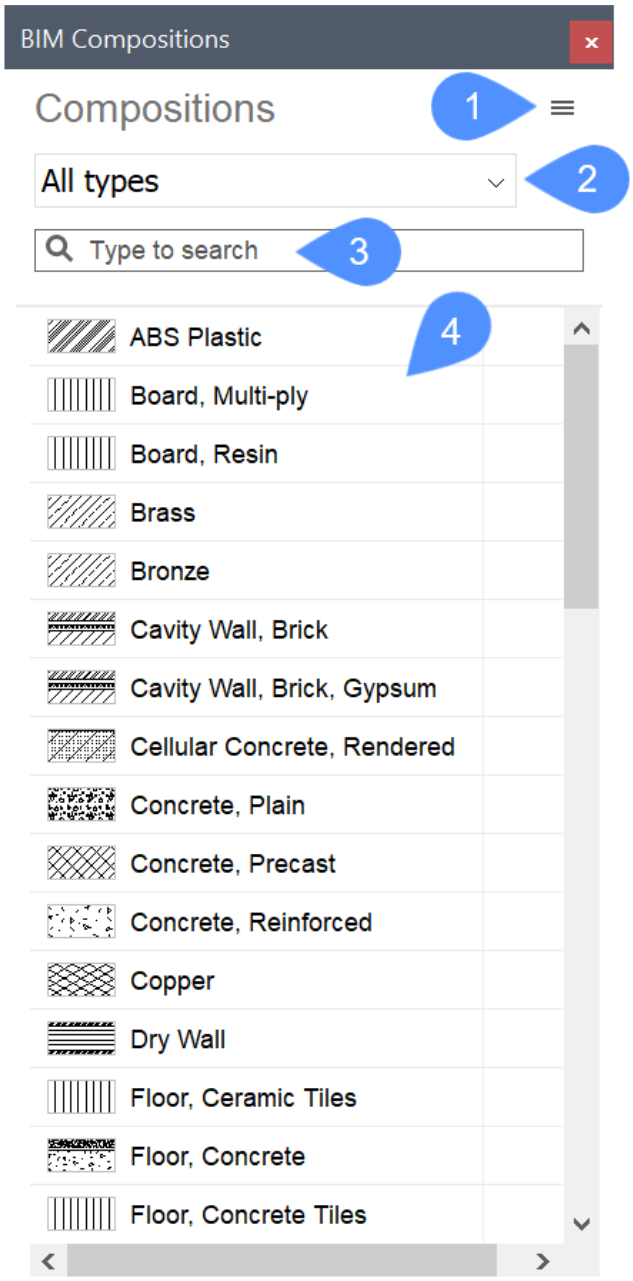
#### 20.4.5.2 About

BIM compositions:

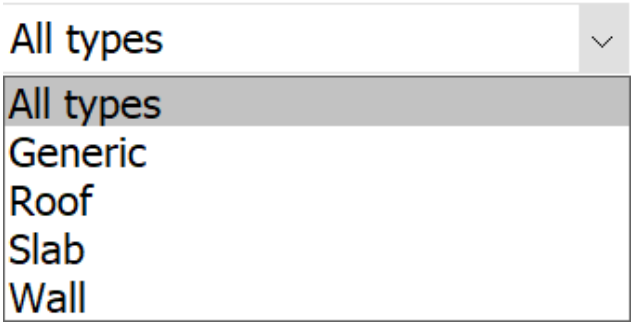
- Are stored in the library database.
- Are either multiply or single-ply.
- Ply materials are managed in the **Physical Materials** editor ( ).
- Can be assigned to any solid in a BIM model.
- Are listed in the **BIM Compositions** panel ( ).
- Are managed in the **Compositions** editor ( ).

For more information about Attaching Compositions, see the procedure article Attaching Compositions.

#### 20.4.5.3 Using the BIM compositions panel



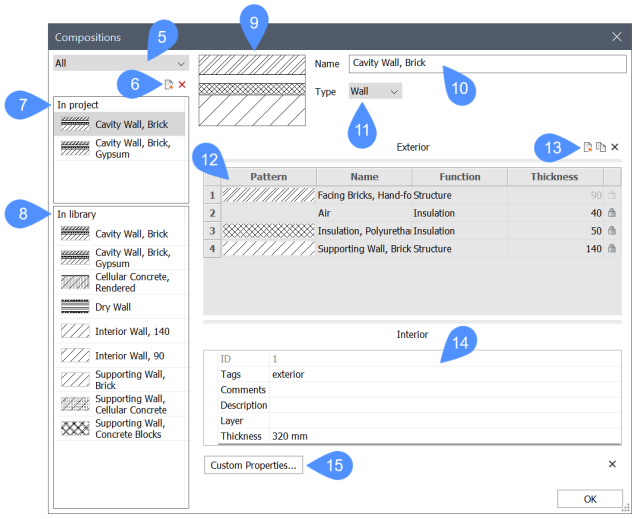

| | |
|---|---|
| **(1) Menu** | Open material dialog...<br>Open compositions dialog...<br>Open project dialog...<br>Show only compositions in project... |

| | |
|---|---|
| **(2) Composition types list** |  |
| **(3) Search field** | Filters the list of the compositions. Only compositions that contain the current search term are displayed. |
| **(4) Available compositions** | All compositions of the selected type in both the **In library** and the **In project** database display, unless the **Show only compositions in project...** menu option is selected. |

### 20.4.5.4 Using the BIM compositions editor

The BIM **Compositions** editor is a dialog box that contains options to create, modify and delete compositions. Using the **Compositions** editor, you can also sort the compositions by category and view the properties of the composition.

The BLCOMPOSITIONS command opens the BIM **Compositions** Editor:




| | |
|---|---|
| **(5) Composition category selection** | Filters by category: **Wall**, **Slab**, **Roof**, or **Generic**. |
| **(6) New composition** | Creates a new composition. |

| | |
|---|---|
| **(7) In project** | Displays the compositions of the selected category (**wall**, **slab**…) in the Project database. |
| **(8) In library** | Displays the compositions of the selected category (**wall**, **slab**…) in the Library database. |
| **(9) Preview pane** | Displays the currently selected composition. |
| **(10) Name field** | Displays the selected composition name. You can change the default name. |
| **(11) Category drop-down box** | Displays a category for the new composition. |
| **(12) Structure grid** | Displays the material and thickness of the composition plies. The ply on top (exterior) is applied to the reference of the solid. |
| **(13) Add ply** | Opens the physical materials editor which allows you to choose a material from the library or project database. Here you can also create a new material.<br>Inserts a copy of the selected ply. |
| **(14) Properties grid** | Shows the properties of the selected composition. |
| **(15) Custom properties...** | Opens the **custom properties** dialog. |

### 20.4.5.5 Creating a new composition

You can create a new composition in either the **Project** or the **Library** database.

1 To open the BIM **Compositions** editor, do one of the following:
- Select the **Open the compositions dialog...** in the menu on the **Compositions** panel.
- Type BLCOMPOSITIONS in the Command line.

2 Click the **New** icon ( ) on the left of the compositions toolbar.

A new (empty) composition is added to the selected database.

3 To add and edit plies, go to **Edit a composition**.

### 20.4.5.6 Editing a composition

**Step 1: Changing name and type**

1 Select the new composition.

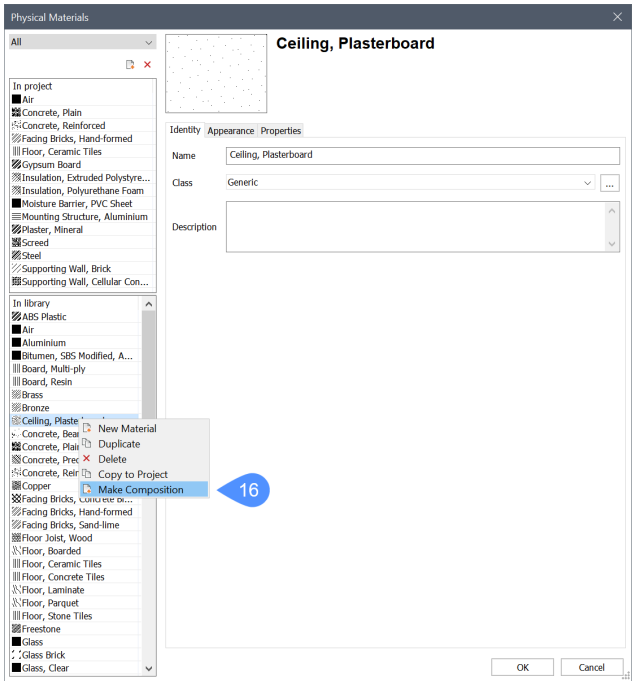
2 Type a new name in the **Name** field for the composition.

3 Use the Type drop-down box to specify the composition type: **Wall**, **Slab**, **Roof** or **Generic**.

Composition categories are used to filter the list of the compositions and the **Compositions** dialog.

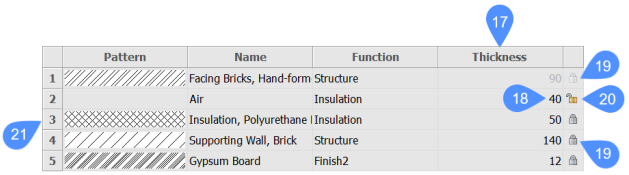
### Step 2: Changing the plies

- **Adding, editing and deleting plies**
  a Click the Add Ply button ( ) on the right of the compositions editor.
  The **Physical Materials** dialog displays:




  b Select a material either in the Project **or** the Library database and double-click **or** press the **OK** button to assign the material to ply in the composition.
  c Double-click on the **Pattern** or **Name** of the ply to edit the material. See the **Physical Materials** article for more information.
  d To choose another material, pick a material in the Library **or** Project list and press **OK**.
  e To delete a ply, select the ply and click on the **Delete** tool button.
  f **(Optional)** Right-click the material and choose **Make Composition** (16) in the context menu to create a single-ply composition.

- **Modifying the thickness and changing the ply order**
  a Click the **Thickness** field (17) to modify the thickness of a ply in the Structure grid.
  You can only modify the ply thickness if the **Allow Custom Thickness** property of the material is set to **Yes**.




  b Enter a value in the field (18) and press Enter to accept it. The overall thickness of the composition is updated accordingly.
  c The plies of a composition are ordered from Exterior (top) to Interior (bottom). Drag the ply

number (21) on the desired position to change the ply order.

***Прим.:*** When a composition has been modified, the solids that have this composition attached are updated automatically. Use the BIMUPDATETHICKNESS command apply the changes in the model.

- **Locking/unlocking the composition thickness**

  The overall thickness of a composition is equal to the sum of the thicknesses of the composing plies.

  The thickness of a ply can be either locked or unlocked (see image above).

  - Locked thickness (19): the thickness of the ply is fixed.
  - Unlocked thickness (20): the thickness of the ply is variable.

  ***Прим.:*** Only one ply in a composition can have an unlocked thickness. As a result, the total thickness of a composition can be:

  - **Fixed:** all plies have locked thicknesses.
  - **Minimal:** the composition contains at least two plies and one ply has an unlocked thickness. The minimum thickness is equal to the sum of the thicknesses of the locked plies.
  - **Free:** a single ply composition has an unlocked thickness.

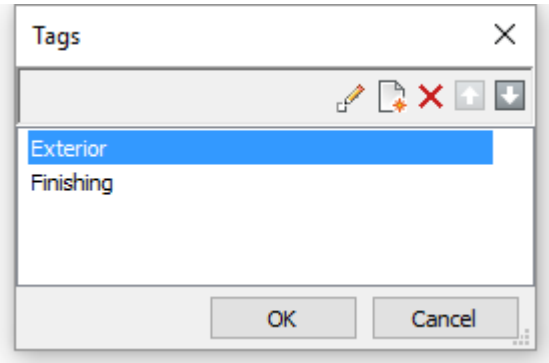
  The thickness of a ply can only be unlocked if the **Allow Custom Thickness** property of the material is set to **Yes**.

- **Adding Tags**

  a Select the **Tags** field.

  b Type the tags, separated by commas. Tags are case sensitive.

  c Click the **Browse** button ( ) next to the tags field in the **Compositions** dialog box to open the **Tags** dialog.




  d Click the **New Item** button ( ) to add a new tag at the bottom of the list.

  e Select a tag in the list and click a tool button:

  - 
  - 
  - 
  - 

  f Click the **OK** button to save the changes.

**Step 3: Creating new materials**

You can create new materials from scratch or copy them from an existing material. See **Physical Materials** for more details.

### 20.4.5.7 Displaying compositions

Whether the various plies of a composition show in the 3D model is controlled through the LEVELOFDETAIL user preference. The RENDERCOMPOSITIONMATERIAL user preference controls whether the ply material is rendered or not.

Click the icons in the **Home/View** panel in the ribbon or the **BIM** toolbar to toggle the status of the composition display:

- Level of Detail: 
- Render Composition Material: 

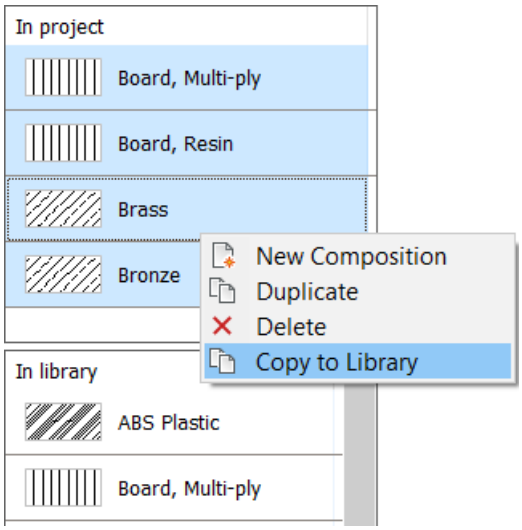
When LEVELOFDETAIL=0, the major faces of a solid are rendered using the material of the outside plies. The minor faces (for example top, bottom, start and end face of a wall) are rendered using the material of the exterior ply.

### 20.4.5.8 Copying compositions

You can use the following procedure in both the **BIM Compositions** panel and the **Compositions** editor:

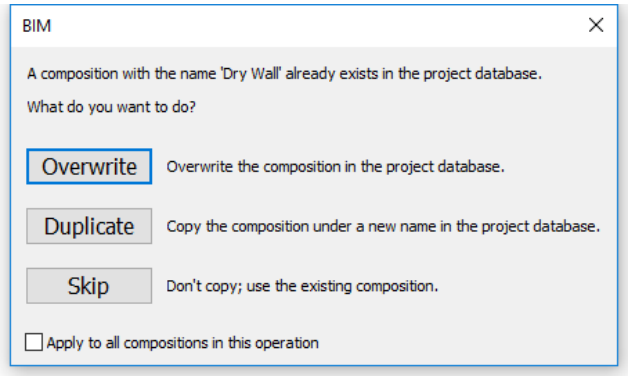
1 Select the composition(s) in the source database. Press and hold the Ctrl key to select multiple compositions.

2 Do one of the following:
   - Drag the selection to the other database.
   - Right-click and choose **Copy to Library** or **Copy to Project** in the context menu.




If the value of the COPYOVERWRITE user preference is 0 (zero), a dialog box displays if the composition already exists in the target library. You can choose to overwrite, duplicate or skip existing compositions.

**(Optional)** Check the **Apply to all compositions in this operation** option.




### 20.4.5.9 Deleting compositions

1 Select the compositions in the **In Project** or **In Library** list on the **BIM Compositions** panel or the **Compositions** editor.

2 Do one of the following:

- Right-click and choose **Delete** in the context menu.
- Click the **Delete** button (**Compositions** editor only).

***Прим.:*** Compositions that are used in the project cannot be deleted from the project database.

## 20.4.6 BIM data structure

### 20.4.6.1 Commands

BIMSPATIALLOCATIONS, BIMATTACHCOMPOSITION, BLCOMPOSITIONS, BIMPROFILES, BLMATERIALS, BIMPROPERTIES, BIMPROJECTINFO

### 20.4.6.2 Project database and library database

A BIM Project consists of dwg entities, classified as **Building Elements**, to which Compositions can be attached. All Compositions being used in a project, are stored in the Project database. The Project database can be stored inside a dwg file, or in a separate file with extension **.bsyslib:**

- When the Project database is embedded in a dwg file, that dwg file contains the complete BIM Project.
- When the Project database is stored as an external file, the BIM Project can consist of multiple dwg files, all using the same Project database.

To reuse compositions and building materials across multiple BIM Projects, a Library database can be specified, in addition to the Project database. The Library database is a **.bsyslib**file, usually stored in a location which is common to several projects. When Compositions and Building Materials from the Library database are used in a BIM Project, they are automatically imported in the Project database. To maintain and extend BIM databases, users can drag Building Materials and Compositions from one database to the other, provided that they have write access to the destination database.

| | |
|---|---|
| **Note** | Sample databases are installed in the **Support** subfolder in the **Roamable** root folder. When a BricsCAD update is installed, the **BricsCAD User File Manager** will prompt you whether to overwrite the sample databases or not. Therefore, it is recommended to rename your library databases and/or save them at a different location. The location of the library databases is defined in the **Library DB** tab of the **BIM Project Info** dialog. |

### 20.4.6.3 Materials and compositions

A BIM database consists of Building Materials and Compositions. A Building Material contains information about one specific material: its appearance, cost, manufacturer, etc. A Composition contains information about the structure of a building element, by defining Composition layers. Each Composition layer refers to a Building Material and has a certain thickness.

**Flexibility**

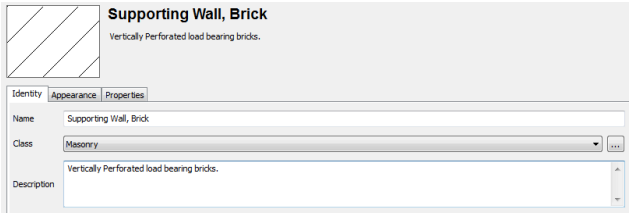
The same flexibility you have in modeling is available in attaching building information. Although Compositions are grouped into **Wall**, **Slab**, **Roof** and **Generic** Compositions, any type of Composition can be attached to any dwg entity. When attached to a 3D Solid, with specific geometrical properties, the thickness of the Composition is used to define the thickness of walls and slabs, and the cross section will show the wall or slab structure by applying the hatch patterns from the various layers on the section geometry. When a 3D Solid is assigned a Composition with a fixed thickness or a minimal thickness, and this thickness can not be applied to the geometry, the section appears in red to indicate the incompatibility between the selected Composition and the geometry of the solid.

**BIM Compositions panel**

All BIM database features are accessible from the dockable **BIM Libraries** panel.

**Materials**

A Material definition describes the appearance and properties of a Building Material.




**Composition layers**

A Composition describes the structure of a building element by an ordered set of composition layers. Each composition layer has the following fields:

- Pattern and Material **Name**: The Building Material for this layer.
- Function: One of (**None**, **Structure**, **Substrate**, **Insulation**, **Finish1**, **Finish2**, **Membrane**). Currently, this field serves for information purposes only.
- Thickness: Defines the thickness of the layer.

### 20.4.7 BIM ply editing

#### 20.4.7.1 Commands

LEVELOFDETAIL, RENDERCOMPOSITIONMATERIAL, DMPUSHPULL

#### 20.4.7.2 About

The **BIM Ply Editing** function allows you to manipulate all the composition plies in your model. In BricsCAD, each drawing has settings stored inside to control the composition plies, called LOD (LEVELOFDETAIL), and the render materials, called RENDERCOMPOSITIONMATERIAL. These two settings can be turned on/off separately.

#### 20.4.7.3 Composition plies and render materials

Composition plies display only when LEVELOFDETAIL = 2 (high).

Building elements do not need to be selected, they will be automatically adjusted. When LEVELOFDETAIL = 0 (low) the material definition of the solid is used.

To toggle the value of the LEVELOFDETAIL user preference, click the icon in the ribbon under the **Home** tab or in the quad under the **BIM** tab:



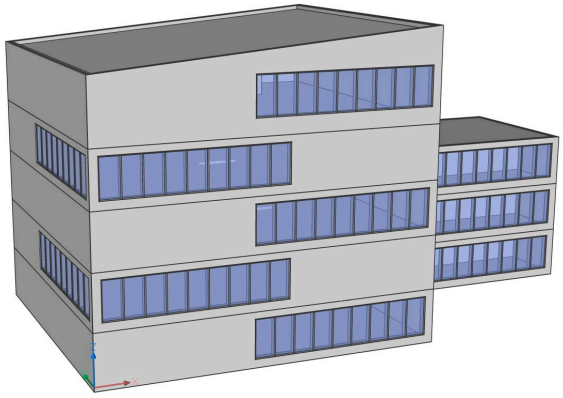
For more information about this setting visit the article LEVELOFDETAIL.

Render materials display only when RENDERCOMPOSITIONMATERIAL = 1 (on). The render material of each ply is defined in the 3D Render Material setting of the material definition.

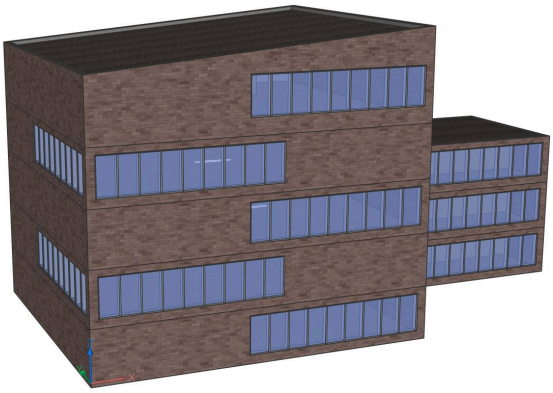
To toggle the value of the RENDERCOMPOSITIONMATERIAL user preference, click the icon in the ribbon under the **Home** tab or in the quad under the **BIM** tab:



For more information about this command, visit the article RENDERCOMPOSITIONMATERIAL.



*RENDERCOMPOSITIONMATERIAL = 0 (off)*



*RENDERCOMPOSITIONMATERIAL = 1 (on)*

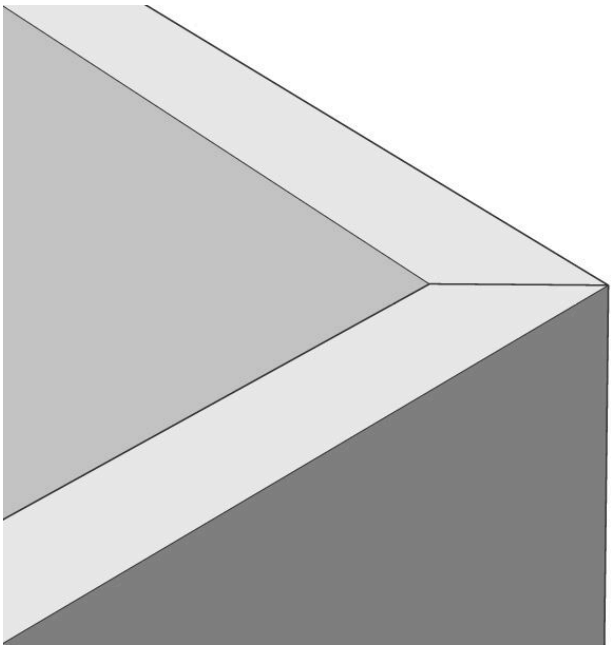
***Прим.:*** Only when Visual Styles have their **Face Settings / Material Display settings** set to **Materials and Textures** (e.g. **BIM**, **Modeling**, and **Realistic**), a visual representation of materials is displayed. Composition plies are displayed in the model as well as render materials.

***Прим.:*** If the render materials of the selected composition are not downloaded or the composition is incorrect, the entity will be displayed in red.

There are different combinations possible with LOD and RENDERCOMPOSITIONMATERIAL turned on or off.

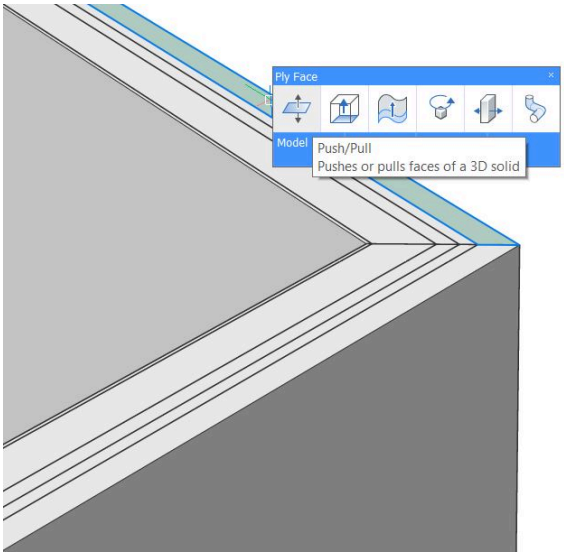
- **LOD =0, RENDERCOMPOSITIONMATERIAL = 0**

  The material definition of the solid is used and no materials are displayed.



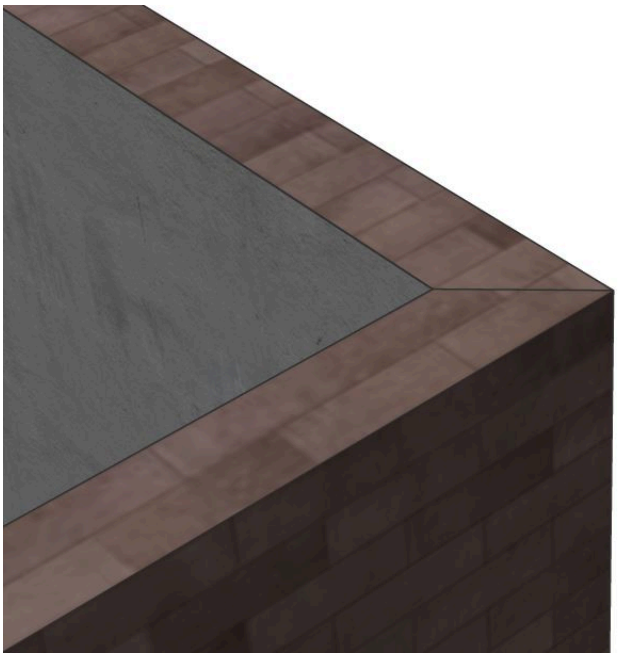
- **LOD =2, RENDERCOMPOSITIONMATERIAL = 0**

  The solid is subdivided into plies, no materials are displayed. When **face detection** is turned on, you can select the faces of different plies to **push/pull** for example.




- **LOD =0, RENDERCOMPOSITIONMATERIAL = 1**

  The material definition of a solid is used. The materials of the exterior and interior face are displayed and the material of the exterior layer will be wrapped around the object.



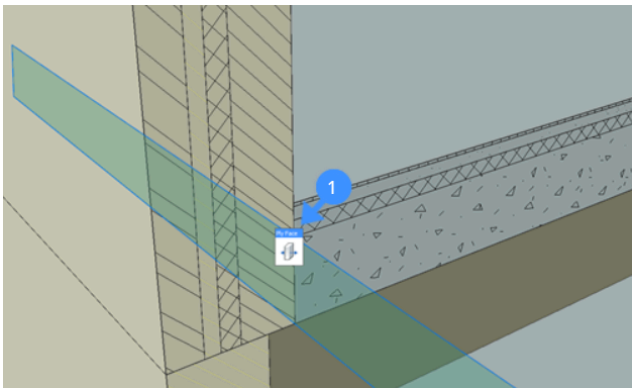
- **LOD =2, RENDERCOMPOSITIONMATERIAL = 1**

  The solid is subdivided into plies. Each ply will be displayed with its own material. When face detection is turned on, you can select the faces of different plies to push/pull for example.

### 20.4.7.4 Manipulating the composition plies

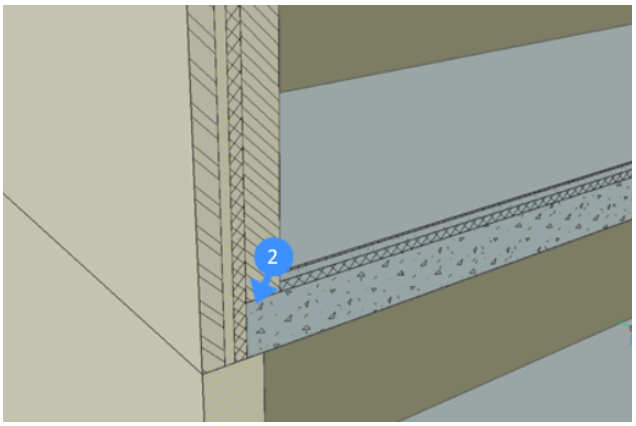
1 Make sure the value of **LEVELOFDETAIL = 2 (High)**.

2 Place your cursor near the composition ply you want to edit and press the Tab key until the **Ply face** (1) highlights.




***Прим.:*** The highlighted ply face is displayed in the detected boundary color (default = 95). Face detection must be enabled to do this.

3 Select **Push/Pull** from the Quad and drag the ply to a new position (2).




***Прим.:*** Make sure DMPUSHPULLSUBTRACT = 1. This ensures that plies being push/pulled are subtracted from any of the other 3D solids they intersect.

***Прим.:*** Make sure ESNAP is turned on to snap to the edge of a ply.

### 20.4.7.5 Procedure: creating a rooftop detail

It is also possible to create more complex details with extra solids added apart from the existing compositions. As an example, a rooftop detail will be created, starting from a rooftop edge.

1 tart with this roof-wall connection.

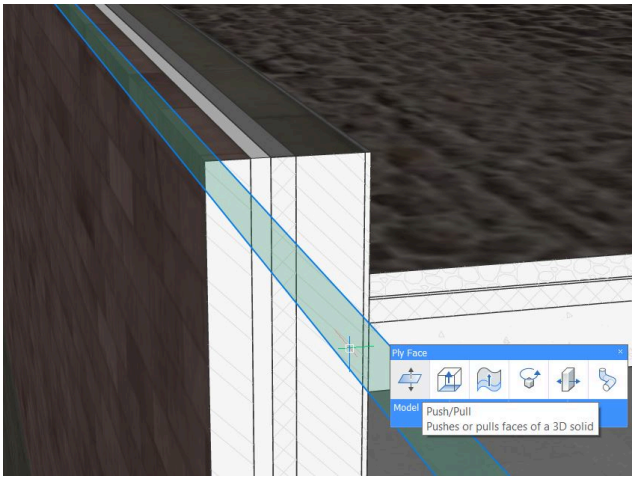


2 Click the **Face detection** icon in the ribbon to enable face detection.

3 Turn on LOD to be able to select ply faces. Click the icon in the ribbon **or** type LEVELOFDETAIL in the Command line, type **2** and press Enter.

4 Hover over the edge of the concrete ply of the floor slab and press Tab to select the face of the ply.

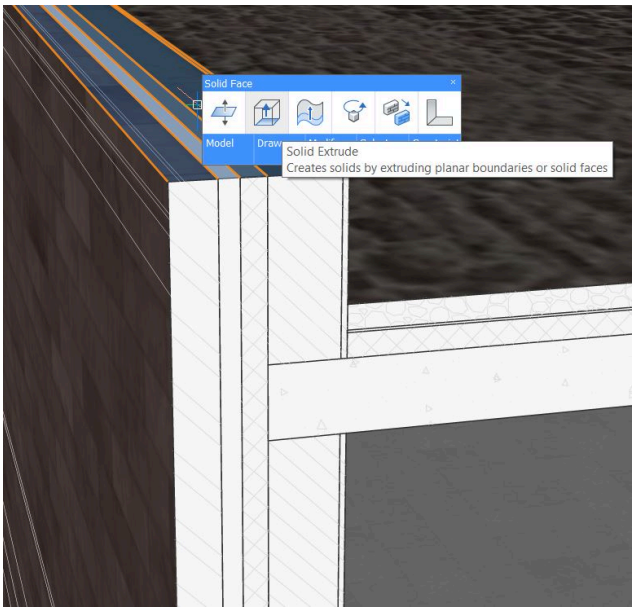
5 Choose **Push/Pull** in the **model** tab of the Quad.



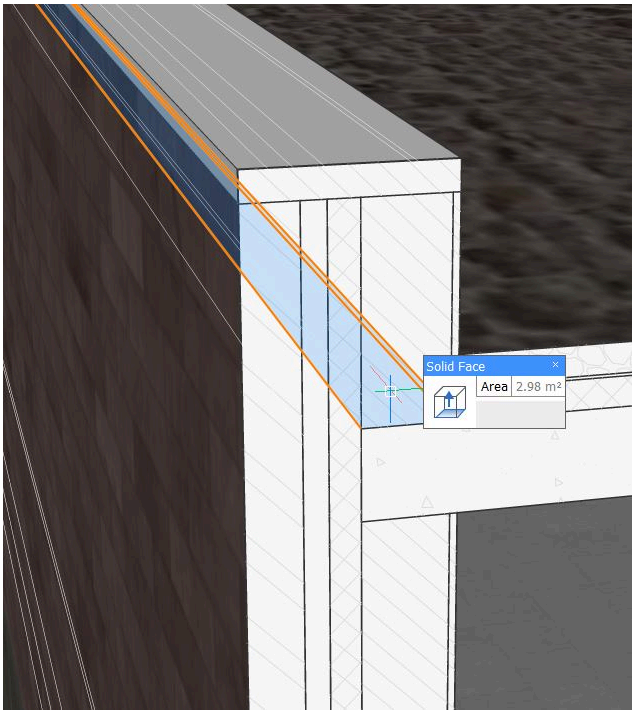

6 Push/Pull the face to the next ply of the wall.



7 To create a wall cap, select the top surface of the wall by pressing Tab and extrude the surface up for 50mm. The created cap isn't classified yet.




8 Delete the part where the insulating building block will come. Hover over the surface of the supporting wall and the gypsum board and press tab to select the surface. Then Extrude till the ply of the wall cap.





9 **Extrude** the same surface again, now a new solid will be created.



### 20.4.8 BIM project info

#### 20.4.8.1 Commands

BIMPROJECTINFO

#### 20.4.8.2 About

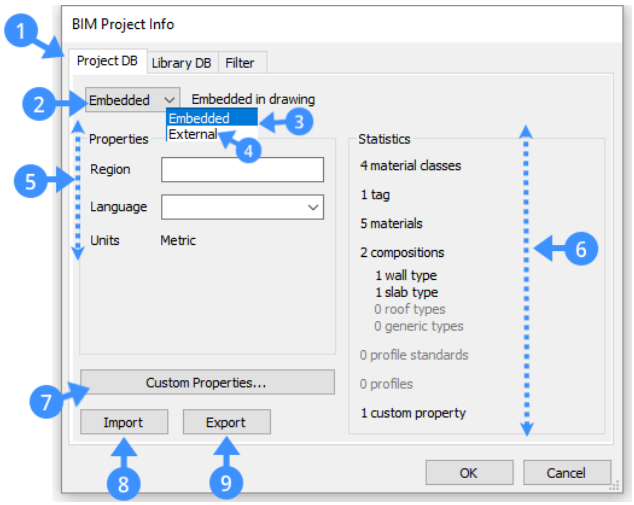
The **BIM Project Info** allows you to control the location, properties, and content of the Project and Library databases. When you launch the BIMPROJECTINFO command in BricsCAD, the BIM project info displays in a dialog box that includes certain components and tabs. Project and library databases of BIM models can be changed using this dialog box.

For more information about this command, visit the Command Reference article BIMPROJECTINFO.

#### 20.4.8.3 Procedure: setting the project database

1 Open the **BIM Project Info** dialog box by typing BIMPROJECTINFO in the Command line.

The following illustration shows the dialog box when the **Project DB** tab is current.




| | |
|---|---|
| **Project database (1)** | Shows the BIM project info. |
| **Project database drop-down list (2)** | Choose between **Embedded** or **External**. |
| **Embedded (3)** | Saves the project library in the drawing file. |
| **External (4)** | Saves the project library in a BIM database file (.bsyslib). |
| **Properties (5)** | Shows the properties of the BIM project e.g. region, language, and units. |
| **Statistics (6)** | Shows the location, properties, and the content of the current project library of the BIM model. |
| **Custom properties (7)** | Adds custom properties to materials and compositions. |
| **Import (8)** | Imports the selected .xml and .csv library files. |
| **Export (9)** | Saves .xml library files in the defined location. |

2 To change the project database from **Embedded** to **External** or vice versa, select the desired option in the Project Database drop-down list.

***Прим.:*** The external database keeps the database independent from the current drawing. This is especially useful if you want to share the library between multiple drawings.

- When switching from **Embedded** to **External** a **File** dialog displays.

  Do one of the following:

- Type a name in the **File Name** field to export the content of the project library to a new .bsyslib file, then click the **Open** button.

- Select an existing .bsyslib file, then click the **Open** button.

  The current content of the project library is merged with the selected .bsyslib file.

  - When switching from **External** to **Embedded** the content of the external library is copied to the embedded project library.

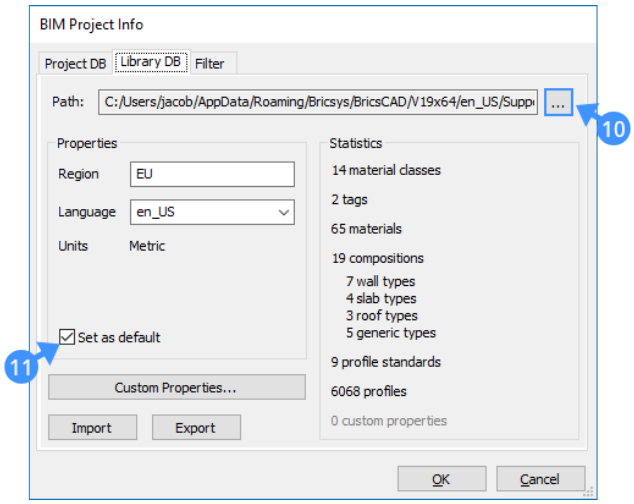
3 To specify the Project Database properties, such as Region and Language, fill out the **Region** field and choose one of the language options in the drop-down list.

4 Optionally, use the custom properties button to add material and composition custom properties in the Project Database.

5 Press the **OK** button to accept it.

### 20.4.8.4 Procedure: setting the library database

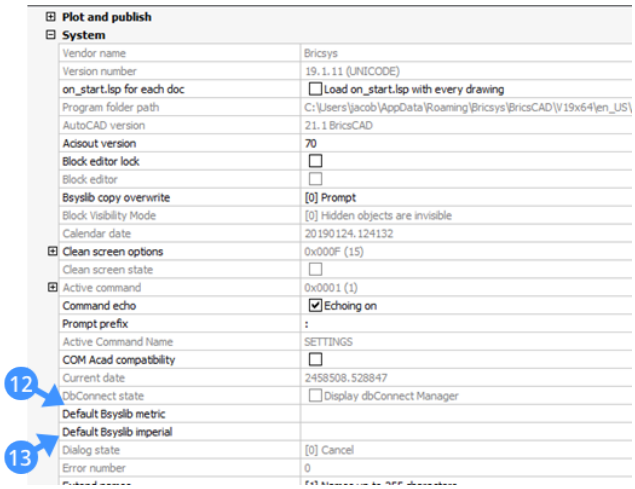
1 Click the **Library DB** tab page in the **BIM Project Info** dialog box. The following illustration shows the Library Database section in the dialog box.




2 To select a BIM database file (.bsyslib), click the Browse (10) button of the **Path** field, then choose a database file in the file dialog.

3 Optionally, check the **Set as default** (11) option to set the selected database as default library database for next projects.

***Прим.:*** You can edit the default library settings (12, 13) in the **Settings** dialog under **Program options** > **System**.



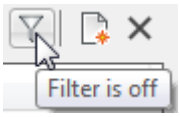

### 20.4.8.5 The project filter

To each Building Material or Composition, any number of tags can be attached. Any string can be entered as a tag.

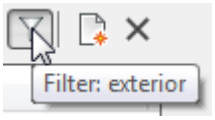
Each of the tags can be used as a filter on the database. There is a button at the top of each list ( ), to switch the filter on or off. If the filter is on, only Compositions and Building Materials that match at least one of the filter tags will be listed.

The **Filter** button ( ) has three states:

- If the project filter is empty, the button is not active (**BIM Compositions** panel) or is not available (**Materials** and **Compositions** editors).
- The project filter is defined, but not active.




- The project filter is defined and active.

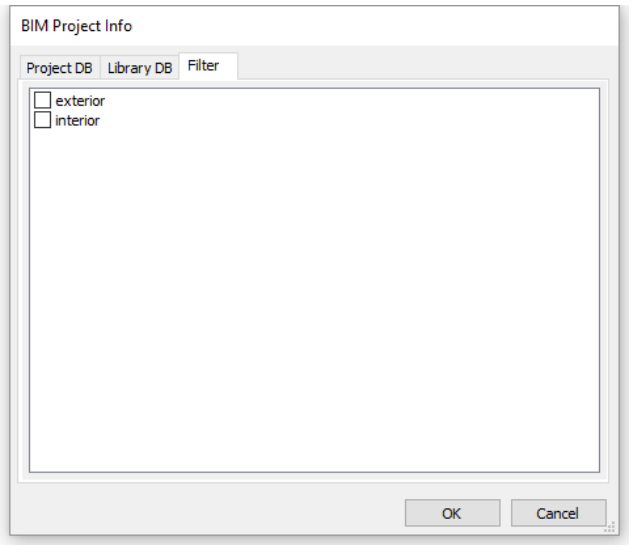



A tooltip shows the current project filter.

### 20.4.8.6 To set the project filter

1 Select the **Filter** tab page on the **BIM Project Info** dialog box.

2 Check the tags you want to use in the filter.




### 20.4.9 Using Python scripts (beta)

#### 20.4.9.1 Commands

BIMPYTHON

#### 20.4.9.2 About BIMPYTHON

The new BIMPYTHON command enables you to query and manage data from a model within BricsCAD BIM, with a suitable .py Python script.

The Python scripts can be as simple as obtaining quantities and associated properties of objects and BIM entities, or as elaborate as implementing a series of calculations based on the model parameters available.

BricsCAD does not ship with a Python Shell, so the scripts would need to be prepared in a text or code editor application.

The Python Programming Language, together with its standard libraries, is embedded within BricsCAD BIM so there is no need for you to install them separately unless you have custom packages and libraries which you wish to use in your scripts.

Before using BIMPYTHON, you need to first turn on the virtual Python environment within BricsCAD.

#### 20.4.9.3 Procedure: activating the Python environment in BricsCAD

1 Open a new or a BricsCAD file where you would like to run a Python script.

2 Type BIMACTIVATEPYTHON in the Command line and press Enter.

3 **You will be prompted:** New current value for BIMACTIVATEPYTHON [1 for ON/0 for OFf] <1 for ON>:

4 By default, the value should display **<1 for ON>**. That means the Python environment is already activated.

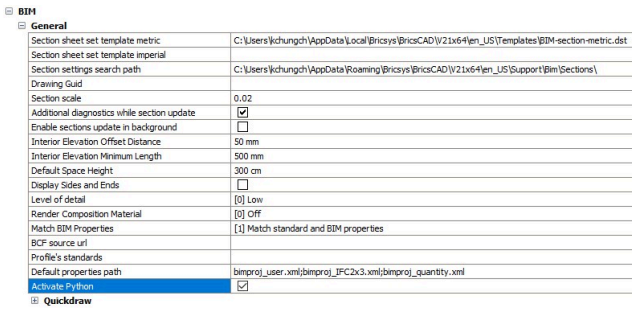
However, If the value shows **<0 for OFF>**, type **1**in the Command line and press Enter to confirm.

**Or**

1 Open a new or a BricsCAD file where you would like to run a Python script.

2 Access the **Settings** dialog box.

3 Tick the box where it says **Activate Python** under the **General** subcategory within **BIM**.




4 Close the dialog box to confirm your updated settings.

### 20.4.9.4 Procedure: setting up a Python script with the BIMPYTHON module

1 Before gaining access to the API, import or 'call' any desired modules you wish to have in the script.

1 If they are not part of the Python Standard library, make sure they have been installed separately beforehand.

You can "import" a standard module, such as math and "import as" an external one, for example Pyplot, which is a collection of functions in the Matplotlib package:

```
import math
import matplotlib.pyplot as plt
```

2 With your desired libraries "called" in, the next step is to gain access to the API in order to query the elements within the BricsCAD model. This is the range of bim objects and serves as an entry point to the model.

3 Import the briqpy.bim_model:

```
from briqpy import bim_model
```

4 Now, you can query the model with these example statements.

```
# Display info about the lengths of walls in the model
lengths = [wall.prop('Length') for wall in bim_model.filter(Type='Wall')]
print(f'wall lengths. max: {max(lengths)}, avg: {sum(lengths)/len(lengths)}')
```

```
# Create a selection and print the objects
bim_model.filter(Type='Wall', IsExternal=True).select()
for wall in bim_model.filter(Type='Wall', IsExternal=True,
 Length=max(lengths)):
    print(wall)
```

The mappers of the briqpy.ObjectRange are chainable. This is an example to get the parts of the roof that are close to a wall:

```
# Get all parts of roof within 40cm range of all walls
roof_parts = bim_model.filter(Type='Roof').parts()
```

```
roof_parts_close_to_wall = bim_model.filter(Type='Wall').within_distance(40, 'cm',
 search_range=roof_parts)
```

Whereas this is an example of a function statement to filter:

```
# Filter roof parts longer than 50 project units
def is_long(obj):
return obj.prop('Length') > 50
roof_parts.filter(is_long)
```

You can also export and show data in various formats:

```
# create a dictionary list
wall_info = [
    {   'Handle': wall.prop('Handle'),
        'Length': wall.prop('Length'),
        'Height': wall.prop('Height')
    } for wall in bim_model.filter(Type='Wall')]
```

```
# export to .json
import json
file = open('path/to/file.json', 'w+')
file.write(json.dumps(wall_info, indent=4))
file.close()
```

```
# plotting a histogram
import matplotlib.pyplot as plt
import pandas as pd
df = pd.DataFrame(wall_info)
df.hist();
plt.show()
```

```
# export to .csv
df.to_csv(r'path/to/file.csv', index = False, header=True)
```

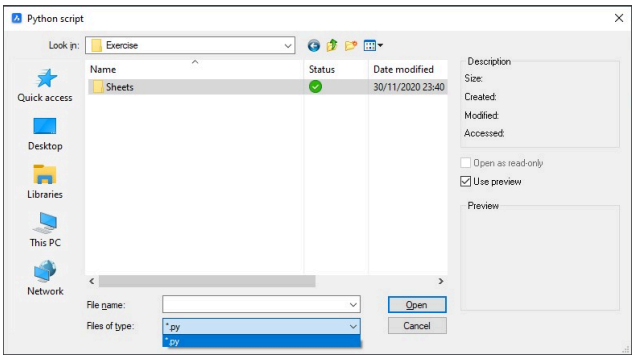
For more information on the BricsCAD API and its various classes, please visit the API chapter below.

### 20.4.9.5 Procedure: executing the Python script

1 Open a new or a BricsCAD file where you would like to run a Python script.

2 Type BIMPYTHON in the command-line and press Enter.

3 A dialog box is displayed where you can only select a Python script file (*.py). Select it and click **Open** to execute the file.




4 Unless you have specified the data to be exported or displayed in an external program outside BricsCAD, BricsCAD reports the output in the Command line panel.

### 20.4.9.6 API dictionary

**Classes and syntaxes**

**class briqpy.Object**- defines BricsCAD BIM Objects

```
prop(prop_name)
```

- Returns the property value of the Object with the given name.

```
distance_to(  other_obj, units='mm', distance_mode='exact' )
```

- Calculate the distance between two Objects.
- Options for argument *units*: any units value form insunits ('Centimeters', 'Feet', 'Parsecs' etc.) and these abbreviations: 'mm', 'cm, 'm', 'km', 'ft'.
- Options for argument *distance_mode*: 'bbox_center', 'bbox', exact'.

```
parts()
```

- Return the ObjectRange containing the sub-elements of this object.

```
parent()
```

- The opposite of **parts**(), returns the parent object of this sub-element.

```
within_distance(  distance, unit='mm', distance_mode='exact',
 search_range=bim_model )
```

- Return the ObjectRange of objects that are within the distance of argument *distance*.
- Options for argument *units*: see distance_to.
- Options for argument *distance_mode*: see distance_to.

```
select()
```

- Add Object to selection

```
deselect()
```

- Remove Object from selection.

```
__eq__() and __hash__()
```

- Makes Object interoperable with e.g. python set or dictionary.

**class briqpy.ObjectRange** - defines collection of BricsCAD BIM Objects

```
filter( function )
```

- Filtering this range with a function parameter.

```
filter( **conditions )
```

- Filtering this range with conditions given as keyword arguments.

```
parts()
```

- Return the ObjectRange of all the parts of all the elements in this range.

```
parents()
```

- Return the ObjectRange of all parent objects of elements in this range.

```
within_distance(  distance, unit='mm', distance_mode='exact',
 search_range=bim_model )
```

- Returns the ObjectRange of objects that are within distance to any object in this range.

```
select()
```

- Add Objects to selection.

```
deselect()
```

- Remove Objects from selection.

```
__len__ ()
```

- Return the number of Objects in this range.

### 20.4.10 Classification codes

#### 20.4.10.1 About

Assigning classification codes organizes the BIM and increases the level of detail. For instance, they allow to organize library materials and project reports, estimate project costs and define product specifications. Currently, **GUBIMClassCA**, **GUBIMClassES**, **Master Format**, **NLSfb**, **OmniClass**, **UniClass2015**, **Uniformat**, and **VMSW** are available classification systems in BricsCAD.



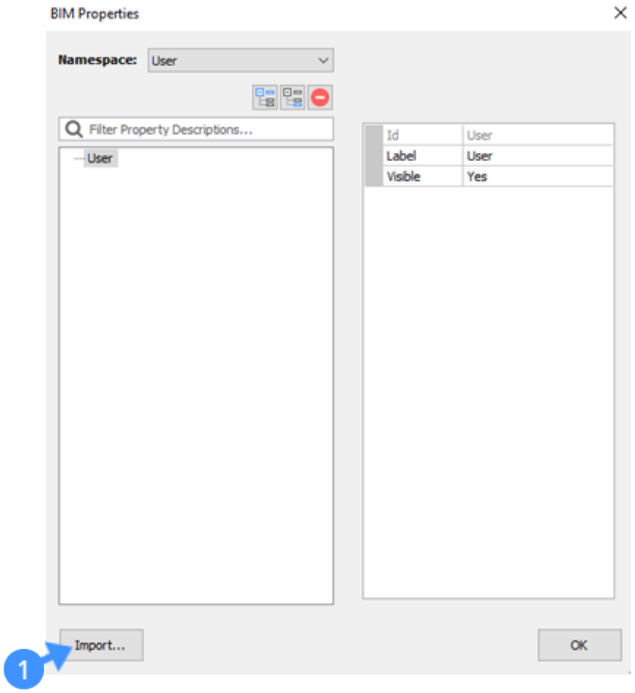
For more information about this command, visit the Command Reference article BIMPROPERTIES.

#### 20.4.10.2 Assigning classification codes to the BIM

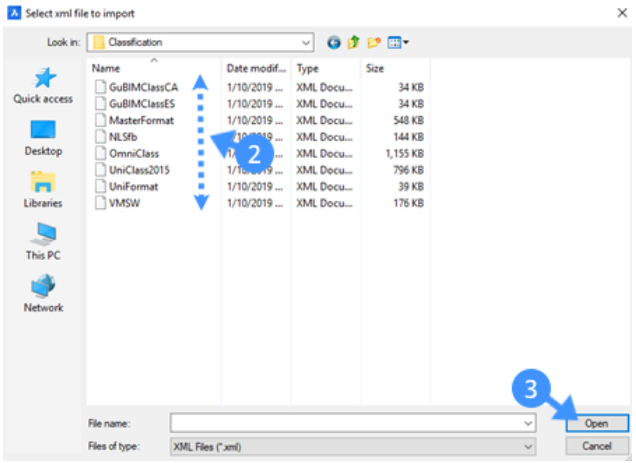
1 Open the BIM Properties dialog box from the Quad (**BIM** > **Model**).

***Прим.:*** Quad should be in **No Selection** state:

A dialog box appears:




2 Click the **Import** (1) button at the bottom-left corner of the dialog box. A file dialog displays:




3 Choose the classification system on the list (2), and click **Open** (3) to import it into the BIM.

***Прим.:*** If the file dialog does not show the **Classification** folder, you should manually enter this path in the address bar: C:\Users\<User>AppData\Roaming\Bricsys\BricsCAD V19x64\en_US\Sup Support \Bim\Classification.

**Available classification systems:**

| | |
|---|---|
| **GUBIMClassCA** | Classification system for the BIM Users group of Catalonia. |
| **GUBIMClassES** | Unified classification system of building elements designed for the AEC industry in Spain. |

| | |
|---|---|
| **Master Format** | Standards for commercial and institutional building projects in the US and Canada. |
| **NLSfb** | Standards for the AEC industry in the Netherlands. |
| **OmniClass** | Classification system for the construction industry. |
| **UniClass2015** | Standards for all sectors of the UK construction industry. |
| **Uniformat** | Standards for classifying building specifications, cost estimating, and cost analysis in the US and Canada. |
| **VMSW** | Standards for the AEC industry in Belgium. |

4 The classification system is added to the **namespace** (4), the selected classification system is added as a property set (5), and the classification code (6) is attached to this property set.

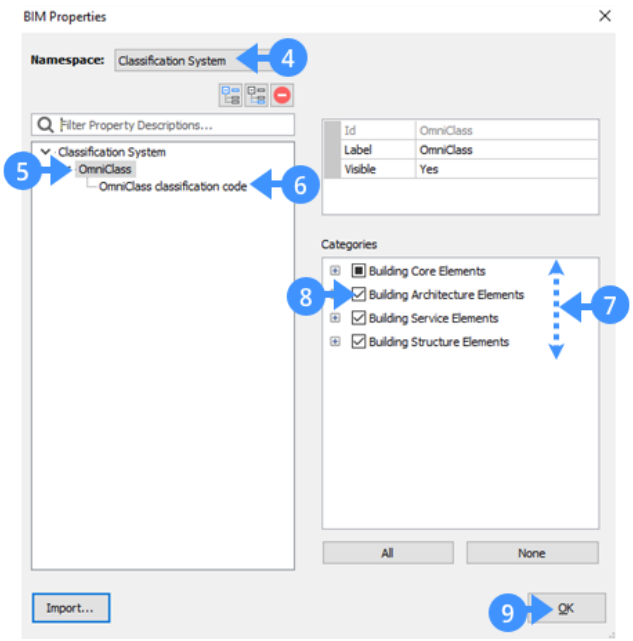
By default, the classification system is applied to all sorts of building elements (7).

**Optional:** Deselect the categories of building elements to which the classification system is applied.

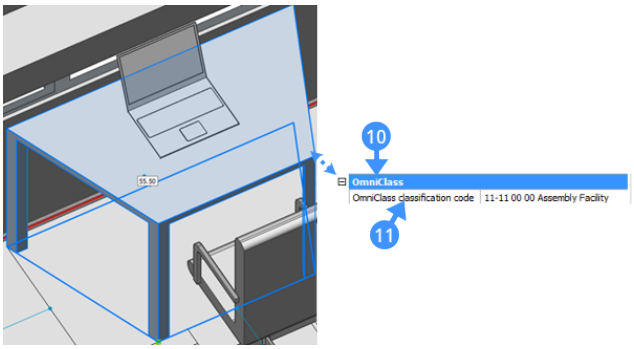
To do so, check the box next to the category name (8) (all elements in this category will be excluded), or expand the list and check the box next to the element type only.

Click the **OK** button (9) to finalize the procedure.

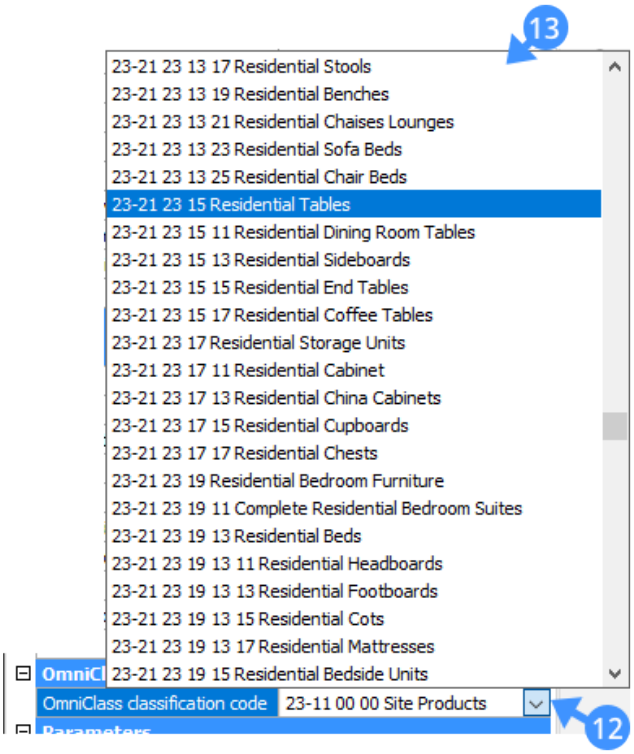
**Or**import another classification system following the previous steps.



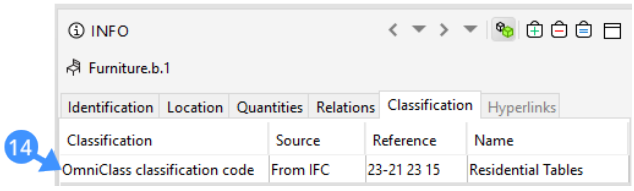

5 Select the element to which you want to assign a classification code, and open the properties panel. The classification system (10) and code (11) should be available in the properties panel.




6 Click the arrow icon on the right-hand side (12), and select the classification code for the selected element in the context menu (13).




***Прим.:*** Exporting the BIM to an IFC file contains the classification system for each element with assigned classification codes. These properties are exported with proper labels and descriptions. The following image shows the OmniClass Classification (14) in Solibri after exporting and importing BIM with the **OmniClass Classification code** from Bricsys to Solibri.



## 20.4.11 Creating and editing BIM spaces

### 20.4.11.1 Commands

BIMSPACE, BIMUPDATESPACE, BIMIFY

### 20.4.11.2 About

You can create BIM Spaces from enclosed spaces. When the wall elements enclose a space in your model, you can use the **BimSpace** tool to define a BIM Space. These spaces are solids so they can be easily manipulated. The 3D space geometries added to the current model are associated with space tags. The space tag is a block that contains a hatch and attributes. The default attributes of the Space tag include a space name and a space number. The values of each attribute can be changed using the properties panel.





For more information about this command, visit the Command Reference article BIMSPACE.

### 20.4.11.3 Defining a BIM space

1 Select the **Space** tool under **Home** > **Classify** > **Space or** type BIMSPACE in the Command line.

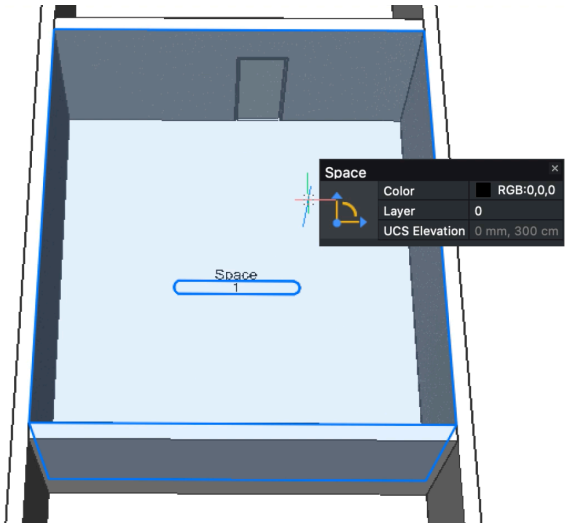
**You are prompted:***Pick a point inside the space or [Edit];*

2 To define a BIM space, pick a point inside the space area in your BIM model.

3 Move the cursor inside the space area to pick a point, the **Dynamic UCS** enables you to define the bottom plane of the space. A green checkmark will appear next to the cursor to indicate a possible location. A red cross will appear if the plane isn't available.

4 In this plane, an area enclosed by wall elements or any space bounding elements is found and a space tag is placed in the center of the space.

5 **Space Bounding elements** can be Walls, curtain walls, slabs, columns, but also hatches and polylines can be counted as space bounding elements.



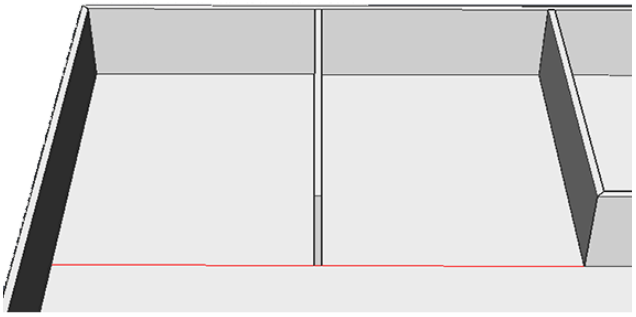

6 Edit the properties of the BIM Space in the properties panel.

7 Change the representation from solid to footprint for a less notable visualization.

### 20.4.11.4 Procedure: creating BIM spaces from virtual boundaries

There are cases in which a virtual boundary is needed. For example, separate parking spaces, large auditorium spaces for energy analytics, etc.

1 Divide the Space with linear entities. This can be hatches or lines.

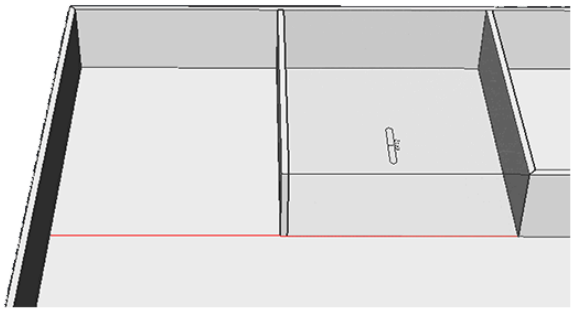


2 Select the dividing elements (the red line).

3 Classify them as a space bounding element: **Home** > **Classify** > **Wall**.

**Or**in the Quad under **BIM**.

4 Type BIMSPACE in the Command line and pick a point inside the space.



***Прим.:*** If the line doesn't divide the space, go to the **Properties** panel and then to **BIM**. Check if **Space bounding** is turned on.

#### 20.4.11.5 Creating BIM spaces automatically

BIMIFY and BIMQUICKBUILDING create BIM spaces and automatically assign information to them. For example, the story and building of the space (see BIMSPATIALLOCATIONS). The commands will search for space bounding elements and create spaces wherever possible.

These spaces act like manually made spaces and can be edited according to your own needs.

### 20.4.12 Custom properties

#### 20.4.12.1 Commands

BIMPROPERTIES

#### 20.4.12.2 About

BricsCAD provides a way to assign your own properties to desired building elements in a single command. This is called assigning custom properties. The approach is first to create a custom property set, then assigning it to the building elements to which the property set is to be applied. The method is carried out by the BIMPROPERTIES command, which allows you to create, edit, and delete the custom property set in a dialog box. This dialog box also allows you to add properties to the currently selected property set. Important to note is that these custom properties are also included in IFC export.

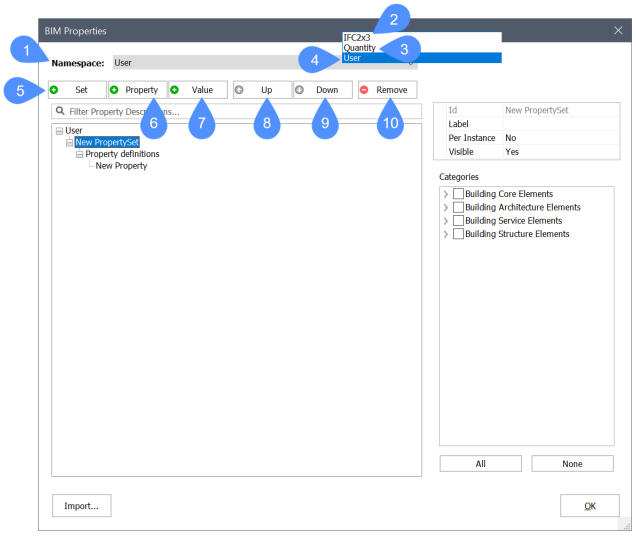
For more information about this command, visit the Command Reference article BIMPROPERTIES.

To learn more about creating user-defined quantity definitions, visit the Procedure article for Customizing BIM Quantities.

### 20.4.12.3 Using the BIM properties dialog box

1 Open the **BIM Properties** dialog box by typing BIMPROPERTIES in the Command line. The dialog box displays for specifying and editing properties of BIM projects. The following illustration shows the

dialog box.




| | |
|---|---|
| **(1) Namespace** | Groups property sets in namespaces. |
| **(2) IFC2X3 namespace** | Shows the IFC2x3 properties. |
| **(3) Quantity namespace** | Shows some hardcoded quantities<br>***Прим.:*** These have been deprecated and replaced by the Quantities in the **IFC2x3** namespace. |
| **(4) User namespace** | Shows the custom properties. |
| **(5) Add property set button** | Creates a new property set. |
| **(6) Add property button** | Creates a new property definition. |
| **(7) Add value button** | Creates a new value definition. |
| **(8) Move up button** | Moves the selected property or value up the list. |
| **(9) Move down button** | Moves the selected property or value down the list. |
| **(10) Remove button** | Removes the selected property set, property definition or value definition. |

2 To create a custom property set, select the **User** option in the namespace drop-list, then click the Add

Property Set. A new property set is added to the **User** space. To edit the property set do the following:

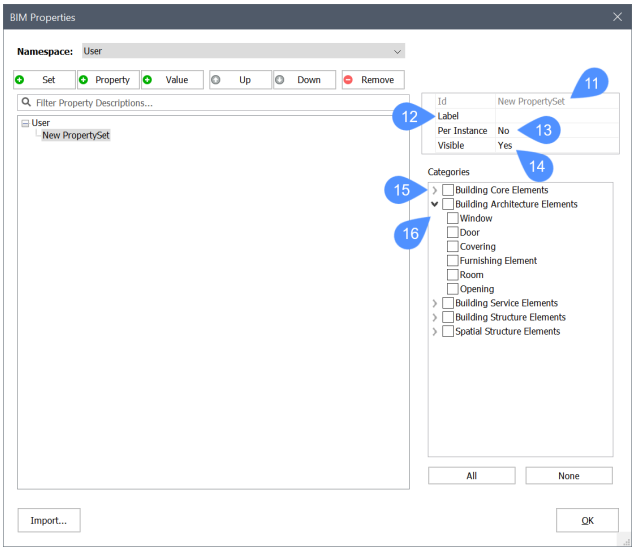
- Type a name in the **Id** (11) field.
- Type a name in the **Label** (12) field.

The **Label** name appears in the properties tree. If the **Label** name is not defined, the **Id** name displays. The **Id** must be unique.

Optionally, you can choose to have the custom property be only applied to specific instances (13), and choose the visibility (14).

- Select the building element categories (15) the property set applies to.

When you expand one of the categories, a different type of building elements (16) displays, check the box next to its name to associate with the property set. The following illustration shows the **BIM Properties** dialog box after inserting a new property set.

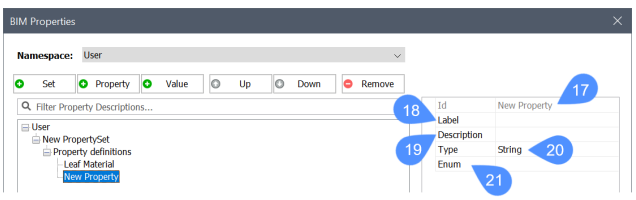



3 To add a property to the currently selected property set, first, select a property set, then click the Add Property button to add a new property.

4 To edit the property do the following:

- Type a name in the **Id** (17).
- Type a name in the **Label** (18).
- Optionally, type a description in the **Description** (19).
- Select the property type. The options are **Boolean**, **Integer**, **Real**, and **String** (20).
- Optionally, define enumerated values (21).

The following illustration shows the **BIM Properties** dialog box after inserting a new property.



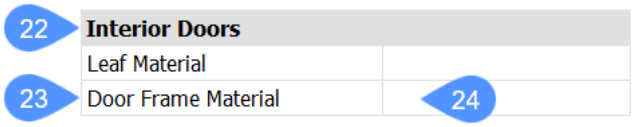
5 To remove the property set/property, select a property set/property in the tree, then click the **Remove** button to remove.

6 Press the **OK** button to accept it.

Now if you select a building element, which is associated with the property set, you will see its newly assigned properties (22) are placed in a new category (23) in the properties panel.

To assign a value to your custom property, select it in the properties panel, then type a new value in its field (24).

The following illustration shows the custom properties in the properties panel.




### 20.4.13 Customizing BIM quantities

#### 20.4.13.1 Commands

BIMPROPERTIES

#### 20.4.13.2 About

Entities have a set of properties which can be viewed in the **Properties** Panel. If an entity has been assigned a certain **BIM classification** or **Type** (e.g. **Wall** or **Slab**), additional properties will be displayed such as IFC Common properties and Base quantities. Which properties are displayed in the panel is managed in the **BIM Properties** dialog, which is opened by the BIMPROPERTIES command.

#### 20.4.13.3 Namespaces

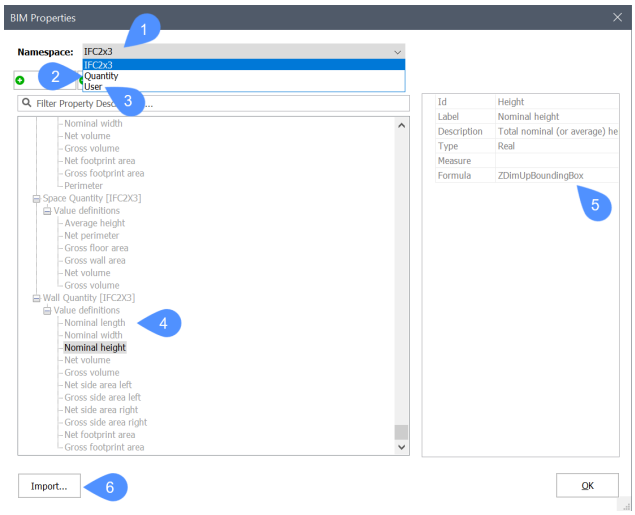
In the BIM Properties dialog, by default three namespaces are available: **IFC2x3** (1), **Quantity** (2) and **User** (3). The **IFC2x3** namespace contains properties that are defined by the IFC2x3 schema. This contains both common properties (e.g. Wall Common properties, such as **Fire Rating** and **Acoustic Rating**), and Quantities (e.g. **Wall Quantity** > **Nominal lenght** (4)). These Quantities are based on so-called Core quantities (5) of the solid. See Core quantities for more information.

The **Quantity** namespace contains some hard-coded quantities of BIM objects also based on the Core quantities, much like in the **IFC2x3** namespace.

> ***Прим.:*** This **Quantity** namespace is deprecated. It has been replaced by the Quantities in the **IFC2x3** namespace, however it is still available in the **BIM Properties** dialog for backward compatibility with versions V20 and older.

The **User** namespace exists for the user to create their own property definitions. See also the Procedure article Custom Properties for more information on custom properties.

It is also possible to import namespaces, e.g. the **IFC4** namespace. This can be done by clicking the **Import** button (6) on the bottom left.




***Прим.:*** using this method the namespace will only be imported for this particular project. If you want namespaces to be imported over all your projects, open the **Settings** dialog and under the **BIMGeneralDefault properties** path you can add a new path. By default, it should take you to *C:\Program Files\Bricsys\BricsCAD Vxx en_US\Support*, where you can find for instance the **bimproj_IFC4.xml** containing the **IFC4** namespace.

### 20.4.13.4 Core quantities

Core quantities are quantities of solids that are calculated based on the geometric features of that solid, and that can be used in user-defined quantities. These core quantities are also the basis for the quantities defined in the **IFC2x3** and **IFC4** properties as described in the previous section.

For example, under **Wall Quantities** in the **IFC2x3** namespace, the **Nominal Height** of a wall is defined as the **ZDimUpBoundingBox** of the solid, which is a Core quantity. A full list of Core quantities can be found below.

| Core Quantity | Description | Measure | Type |
|---|---|---|---|
| **NumberOfPlies** | Number of plies in the attached composition. | | **Integer** |
| **XDimUpBoundingBox** | X dimension of the bounding box found with global Z axis rotation freedom. | **Length** | **Real** |
| **YDimUpBoundingBox** | Y dimension of the bounding box found with global Z axis rotation freedom. | **Length** | **Real** |
| **ZDimUpBoundingBox** | Z dimension of the bounding box found with global Z axis rotation freedom. | **Length** | **Real** |

| Core Quantity | Description | Measure | Type |
|---|---|---|---|
| **XDimFreeBoundingBox** | X dimension of the bounding box found with 3D rotation freedom. | **Length** | **Real** |
| **YDimFreeBoundingBox** | Y dimension of the bounding box found with 3D rotation freedom. | **Length** | **Real** |
| **ZDimFreeBoundingBox** | Z dimension of the bounding box found with 3D rotation freedom. | **Length** | **Real** |
| **DistanceBetweenMajorSurfaces** | Distance between major surfaces. | **Length** | **Real** |
| **FirstMajorSurfacePerimeter** | Perimeter of the first (largest) major surface. | **Length** | **Real** |
| **SecondMajorSurfacePerimeter** | Perimeter of the second major surface. | **Length** | **Real** |
| **LinearAxisLength** | Length of the linear element's axis. | **Length** | **Real** |
| **VariablePlyThickness** | Thickness of a variable ply. | **Length** | **Real** |
| **FirstMajorSurfaceNetArea** | Area of the first (largest) major surface after subtracting openings. | **Area** | **Real** |
| **FirstMajorSurfaceGrossArea** | Area of the first (largest) major surface before subtracting openings. | **Area** | **Real** |
| **SecondMajorSurfaceNetArea** | Area of the second major surface after subtracting openings. | **Area** | **Real** |
| **SecondMajorSurfaceGrossArea** | Area of the second major surface before subtracting openings. | **Area** | **Real** |
| **IsMajorSurfacesParallel** | Indicator if the major surfaces are parallel. | | **Boolean** |
| **TotalSurfaceNetArea** | Area of the solid's surfaces after subtracting openings. | **Area** | **Real** |
| **TotalSurfaceGrossArea** | Area of the solid's surfaces before subtracting openings. | **Area** | **Real** |

| Core Quantity | Description | Measure | Type |
|---|---|---|---|
| **FootprintNetArea** | Area of the bottommost solid's surfaces after subtracting openings. | **Area** | **Real** |
| **FootprintGrossArea** | Area of the bottommost solid's surfaces before subtracting openings. | **Area** | **Real** |
| **ProjectedNetArea** | Area of the solid's surfaces projected to global XY plane after subtracting openings. | **Area** | **Real** |
| **ProjectedGrossArea** | Area of the solid's surfaces projected to global XY plane before subtracting openings. | **Area** | **Real** |
| **CrossSectionNetArea** | Area of the cross section for linear elements after subtracting openings. | **Area** | **Real** |
| **OuterSurfaceNetArea** | Area of the outer surface for linear elements after subtracting openings. | **Area** | **Real** |
| **NetVolume** | Volume of the solid after subtracting openings. | **Volume** | **Real** |
| **GrossVolume** | Volume of the solid before subtracting openings. | **Volume** | **Real** |

### 20.4.13.5 User-defined quantities

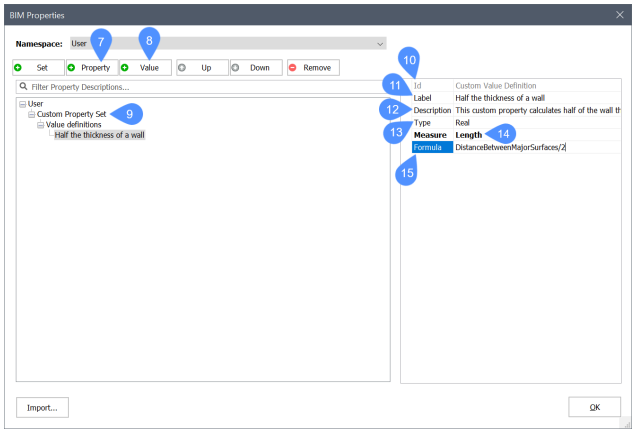
In the **User** namespace it is possible to create custom properties inside a custom property set using the **Property** (7) button, see Procedure article Custom Properties.

It is also possible to create user-defined quantities, using the **Value** (8) button. In the image below, a user-defined quantity was created in a **custom property set** (9). This user-defined quantity (or Value Definition) has the following attributes:

- **Id** (10): a unique identifier.
- **Label** (11): the name of quantity which will show up in the properties panel. This can be the same as the ID, however **the ID must be unique**.
- **Description** (12): a description of the quantity.
- **Type** (13): the type of the quantity. Typically, a quantity will be defined as a **Real** value.
- **Measure** (14): the measure of the quantity, i.e. is it a **length** value, an **area**, **volume** or **mass**?

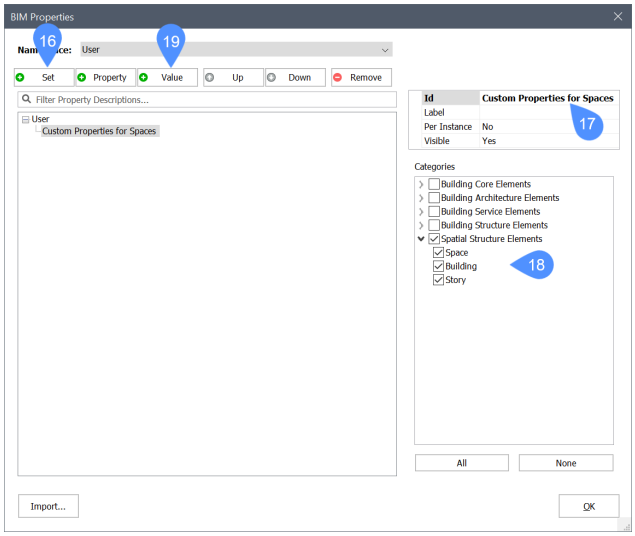
- **Formula** (15): the actual expression of the quantity. Here Core quantities can be used, as well as mathematical expressions as defined in the Procedure article Working with Parameters and Constraints.

In this example, a user-defined quantity is defined that displays half the thickness of a wall, which is calculated as **DistanceBetweenMajorSurfaces/2**.




### 20.4.13.6 Procedure: creating a user-defined quantity

In this part you will learn how to make user-configurable quantities. By way of example, we will define a quantity of Spatial objects such as a room. The quantity will be the **Compactness** which can be calculated as the volume of the space divided by the sum of the areas on all sides of the space.




1 Open the **BIM Properties** dialog by typing BIMPROPERTIES in the Command line.

2 Set the **User** namespace as current namespace.

3 Create a new Property Set by clicking the **Set** button (16), and give it a name (17).

4 Assign the Property Set to one or more categories (18). In this example we will assign it to spatial elements.

5 Create a new Value Definition by clicking the **Value** button (19).

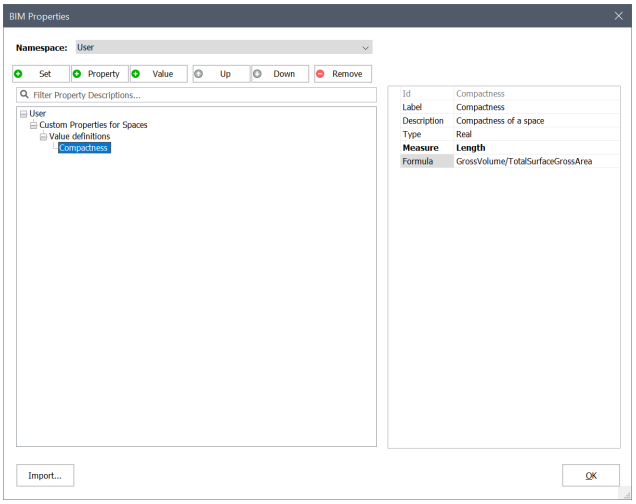
6 Give this Value Definition an **Id** and a **Label**, e.g. **Compactness**.

7 **(Optional)** Give this Value Definition a description.

8 Assign the correct **Type** to this Value Definition. Because our Value Definition will be based on a calculation of two **Real** values, this will also be a **Real** value.

9 Assign a **Measure** to the Value Definition. Because the result of the calculation will be a volume divided by an area, we will end up with a **Length** measure.

10 Fill in an expression in the **Formula** field. The volume of a space is a Core quantity with the name **GrossVolume**. The sum of areas of the space on all sides is also a Core quantity with the name **TotalSurfaceGrossArea**. Hence, we can define the **Compactness** = **GrossVolume/ TotalSurfaceGrossArea**.




11 Click **OK** to close the **BIM Properties** dialog.

12 Create a volume that represents a space, either by creating surrounding elements like walls and a slab, and by defining a space inside using the BIMSPACE or BIMIFY commands, or by creating a solid and using BIMCLASSIFY to classify it as a **Space**.

13 Select the space. In the **BIM Properties** panel you should now see your user-defined property under a separate property set.

### 20.4.14 Load bearing direction for slabs

#### 20.4.14.1 Commands

BIMSETLOADBEARINGDIRECTION

#### 20.4.14.2 About BIMSETLOADBEARINGDIRECTION



In BricsCAD BIM, the BIMSETLOADBEARINGDIRECTION command allows you to define 0, 1, or 2 load bearing directions for a slab.

After defining the load bearing direction in the 3D model, you can indicate the load bearing direction in the floorplan using the **Manual** option of the BIMTAG command.

### 20.4.14.3 Specifying the load bearing direction(s)

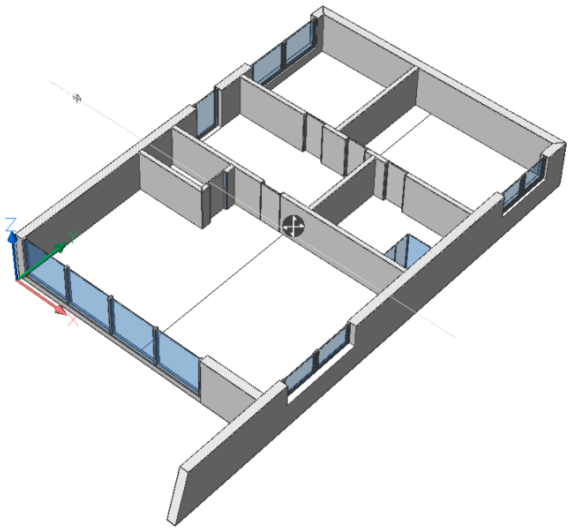
Only solids which are classified as a **BIM Slab** can be processed by the BIMSETLOADBEARINGDIRECTION command. Use the BIMIFY or BIMCLASSIFY command to classify a solid as a **BIM Slab**.

1 Launch the BIMSETLOADBEARINGDIRECTION command.

1 **You are prompted:** Select a slab

2 Click the slab. It may help to set the **Clip Display** property of a plan section to select the slab.

A widget displays at the center of the slab.



3 Repeatedly click the widget. Press Enter or right-click to confirm the loadbearing direction(s).

### 20.4.14.4 Adding a load bearing symbol in plan section

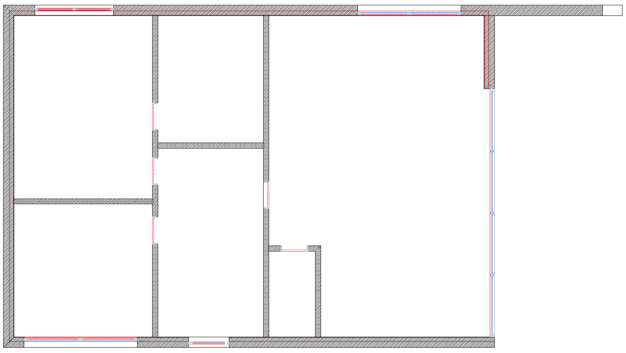
1 When you insert the load bearing tag in a plan section, you need to select a line, which is generated by the slab. Since slabs usually overlap with walls, you need to set the **Hidden Lines** > **Show** property of the **BIM Section** entity to **Yes**.

**Hidden Lines**

| | |
|---|---|
| Show | Yes |
| Color | Red |
| Layer | 0 |
| Linetype | Continuous |
| Linetype scale | 1 |
| Plot style | ByColor |
| Lineweight | ByLayer |

Next, execute the BIMSECTIONUPDATE command of the plan section.

2 Open the **Plan** section.



3 Launch the BIMTAG command.

4 Choose the **Manual** option.

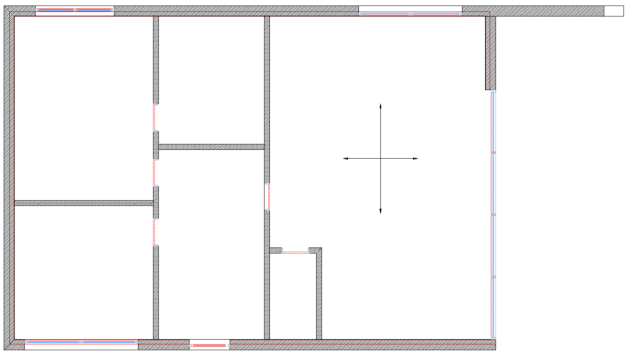
5 Choose the **Change Tag Type** option.

6 Choose the **Load-Bearing Direction** tag type.

7 Click on a point on the edge of the slab.

8 The tag displays, attached to the cursor.

9 Click to place the tag.



## 20.4.15 Physical materials

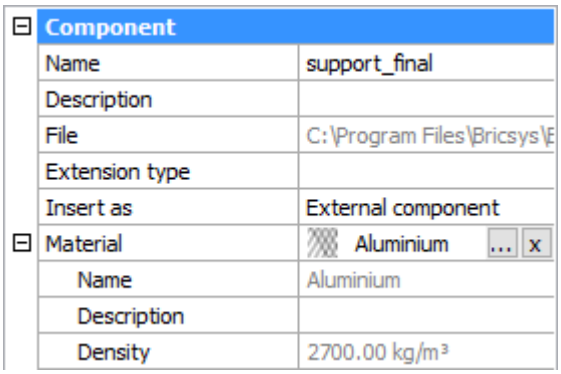
### 20.4.15.1 Commands

BLMATERIALS

### 20.4.15.2 Opening the physical materials editor

Do one of the following:

- Launch the BLMATERIALS command.
- Click the **Browse** button (…) of the **Material** property of a component in the **Mechanical Browser**.




### 20.4.15.3 Creating physical materials

You can create physical materials in either the Project or the Library database. You can create new materials from scratch or copy them from an existing material.

**Creating a physical material from scratch**

Select a material in either the **In Project** or the **In Library** material list.

Do one of the following:

1 Click the **New Material** button ( ) on the **Physical Materials** dialog.

2 Right-click and choose **New Material** in the context menu.

A material named **New** is added to the selected database. Now you can edit the new material.

**Adding a physical material as a copy of an existing material**

Select a material in either the **In Project** or the **In Library** material list. Do the following:

1 Right-click on the material you want to copy and choose **Duplicate** in the context menu.

2 A material named **<existing_material>-2** is added in selected database.

3 Adjust the copy of the material.

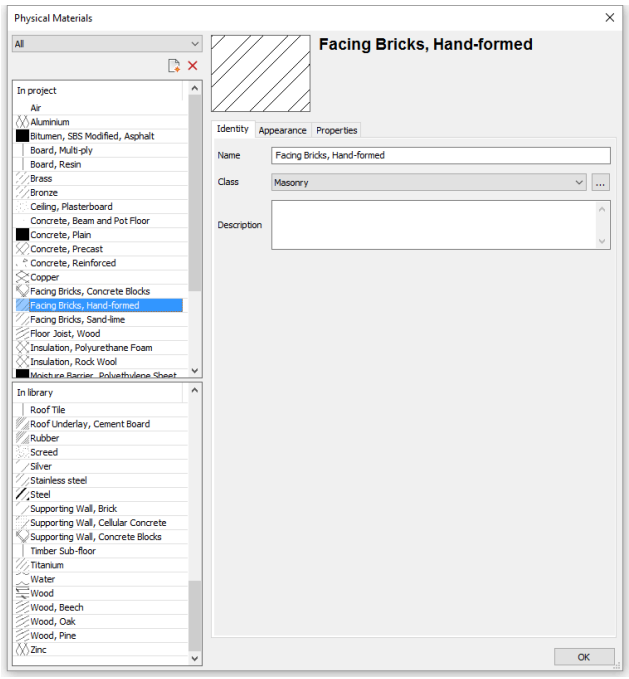
### 20.4.15.4 Editing a physical material

The various settings of a physical material definition are divided into three groups under three tabs on the **Physical Materials** dialog:

- **Identity**: **Name**, **Class** and **Description** of the material.
- **Appearance**: The BIM Materials layers the composition is composed of.
- **Properties**: Properties of building material such as **Cost**, **Manufacturer**, **Thickness**...

Select the material you want to edit in either the **In Project** or in the **In Library** list on the **Physical Materials** dialog.

**Identity tab page**

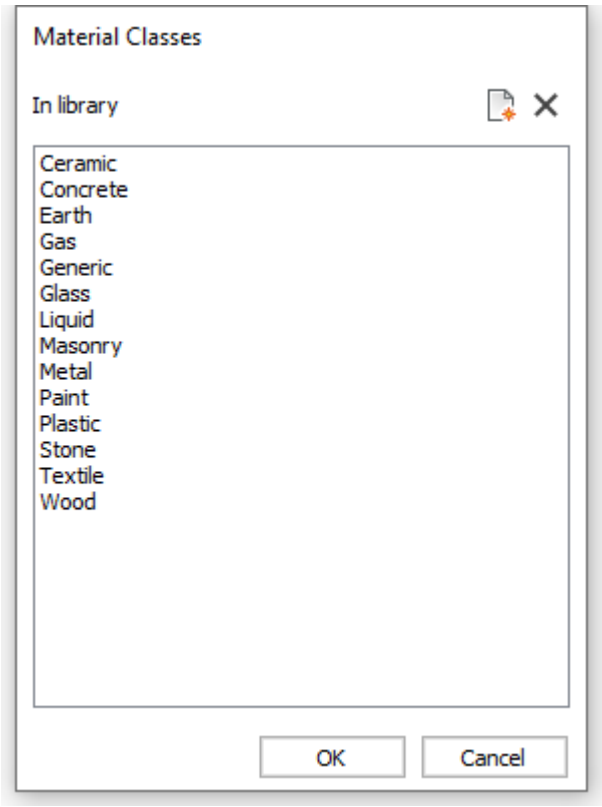



1 Type a **Name** for the material.

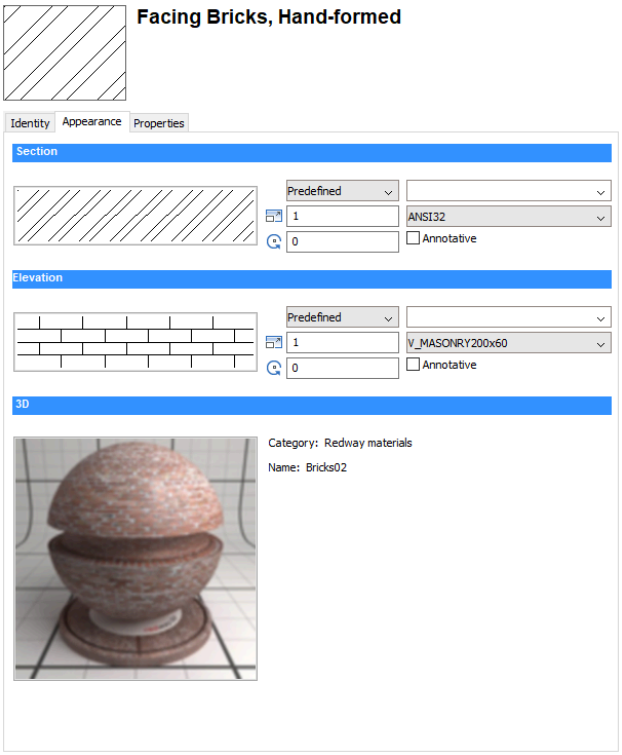
2 Select a **Class** from the drop-down list*.

3 **(Optional)** Type a **Description**.

* Click the **Browse** button (...) to open the **Material Classes** dialog where you can create new material classes.




**Appearance tab page**




- **Section settings**

The **Section** settings define the hatching of the section boundary lines.

a Click the **Hatch Type** button and select the type of the hatch: **None**, **Solid**, **Linear**, **Predefined** or **Custom**.

b Do one of the following to select a predefined or custom hatch pattern:

- Click the **Hatch** preview and select the pattern in the the **Hatch Pattern** dialog box.
- Click the **Hatch Pattern Name button** and select the pattern in the list.

c Define the **Scale**.

The final pattern scale in a section is the product of the **Scale** setting of the physical material and the value of the **Scale** property of the **BIM** section. As a result, the printed scale of the hatch pattern is independent of the section plane scale.

d Set the **Rotation** angle.

e In the **Layer** field, either type a name or click the arrow button and select a layer in the list.

f Optionally, check the **Annotative** option.

- **Elevation settings**

  The **Elevation** settings define the surface hatching of the section background.

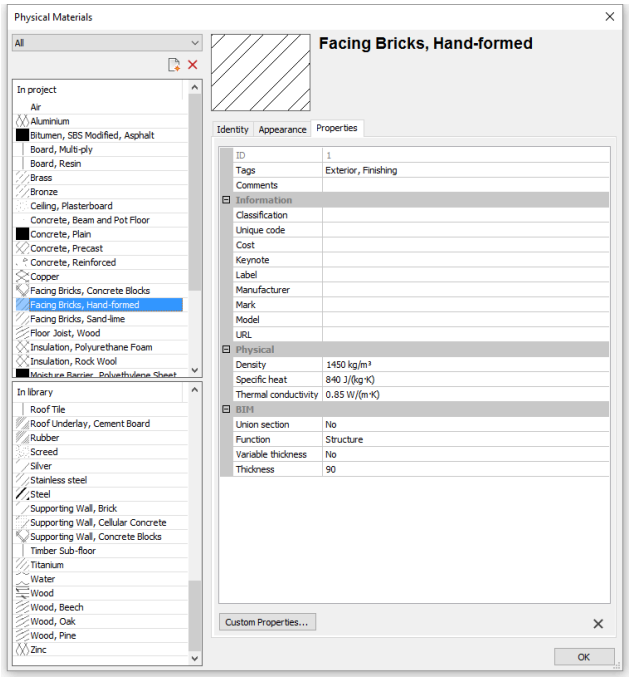
  See **Section** settings above, except for the **Scale** setting. The elevation hatch is independent of the **Scale** property of the **BIM** section.

- **3D settings**

  The **3D** settings define the rendering material.

  Double-click the preview image and select a material in the **Select Render Material** dialog box.

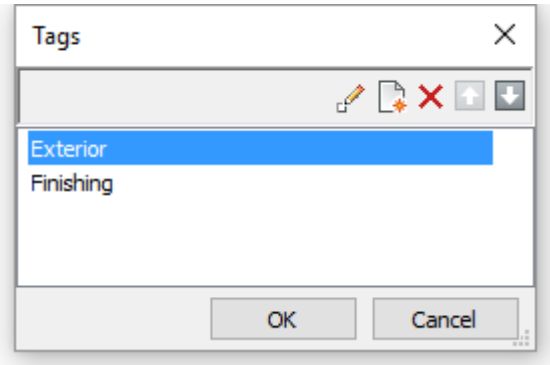
### Properties tab page




1 Edit the **Tags** field: tags are separated by commas.

Tags are case sensitive.

Click the **Browse** button (...) to open the **Tags** dialog.




2 Click the **New Item** button () to add a new tag at the bottom of the list.

3 Select a tag in the list and click a tool button:

- : Edit the selected tag.
- : Delete the tag.
- : Move the tag up.
- : Move the tag down.

4 **(Optional)** Expand the **Information** properties and fill out the necessary fields.

5 **(Optional)** Expand the **Physical** properties and specify the physical properties of the material.

6 Expand the **BIM** properties.

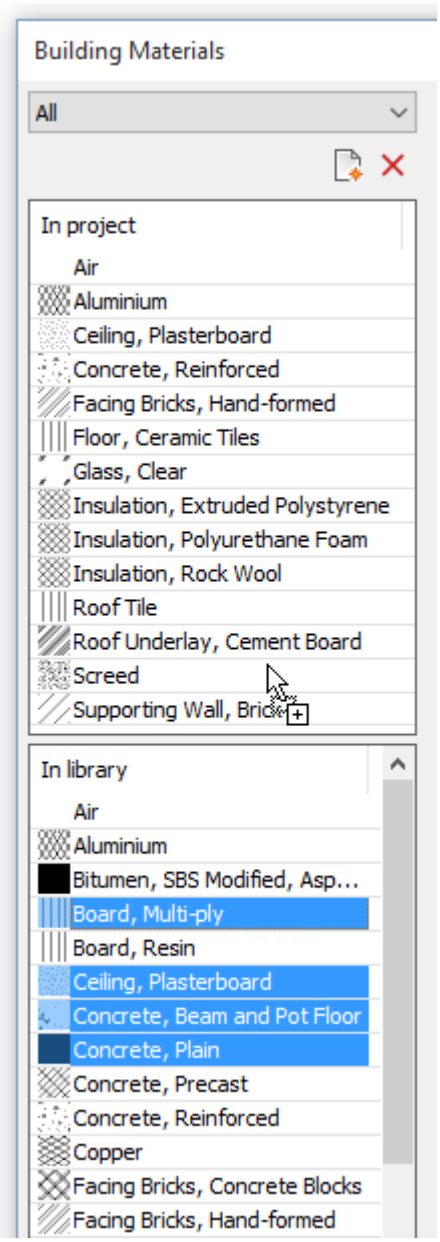
- **Union Section**: Select **Yes** to remove connection lines in sections between different solids of this material. Select **No** to keep the connection lines.
- **Union Elevation**: Select **Yes** to remove connection lines in elevations between different solids of this material. Select **No** to keep the connection lines.
- Select a **Function** from the drop-down list.
- **Variable Thickness**: select **Yes** to allow a variable thickness. Select **No** to not allow a variable thickness.
- Type a default thickness in the **Thickness** field.

7 Click **OK**.

### 20.4.15.5 Copying materials between the project and library databases

1 Select the material(s) in the source database.

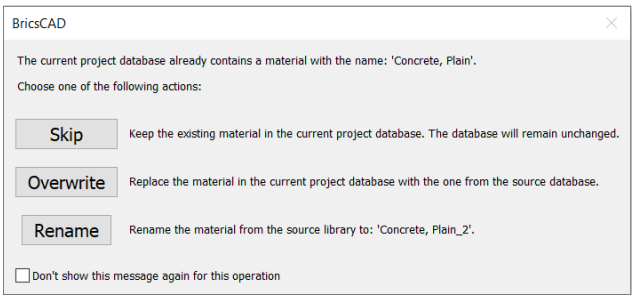
1 Press and hold the Ctrl-key to select multiple materials.




2 Drag the materials to the target database.

3 The selection set is copied.

If one or more materials already exist in the target database, an alert box displays:




Click the appropriate button to either overwrite, duplicate or skip the existing material. If the **Apply to all materials in this operation** option is selected, the option applies to all existing materials. Duplicated materials have an index number added to their name.

### 20.4.15.6 Deleting physical materials

1 On the **Physical Materials** dialog box, select the material(s) in the Project or Library database.

2 Click **Delete**.

***Прим.:*** You cannot delete materials that are used in a composition or a mechanical component.

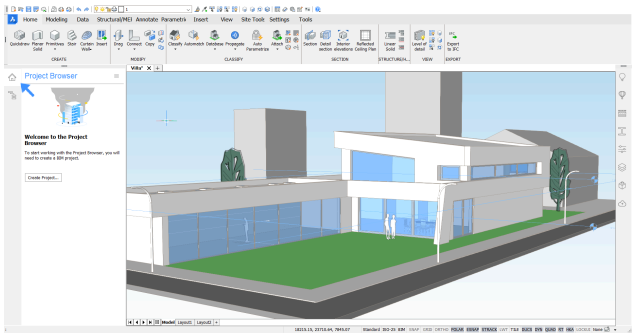
## 20.4.16 Project browser

### 20.4.16.1 Commands

DATAEXTRACTION, BIMSECTIONUPDATE, BIMSECTIONOPEN

### 20.4.16.2 The project browser panel

You can find the **Project Browser** panel on the left-hand side of the screen.



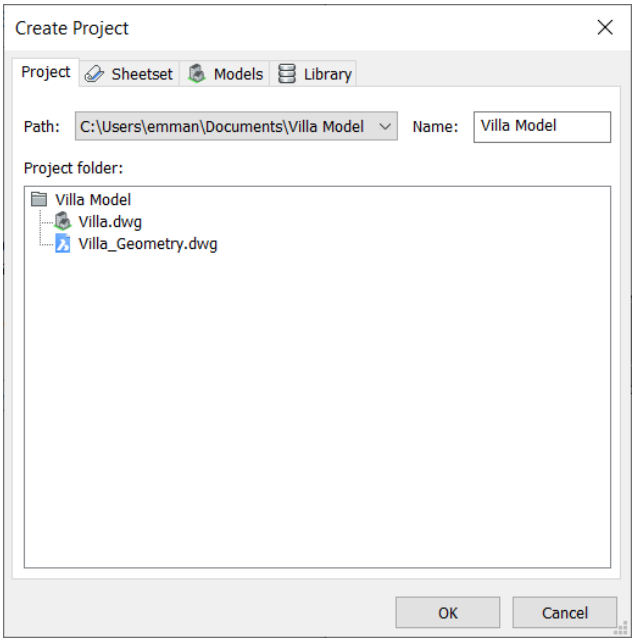

***Прим.:*** If the **Project Browser** doesn't display in the panel, right-click in the panel and select **Panels** > **BIM Project Browser**

### 20.4.16.3 Create a BIM project

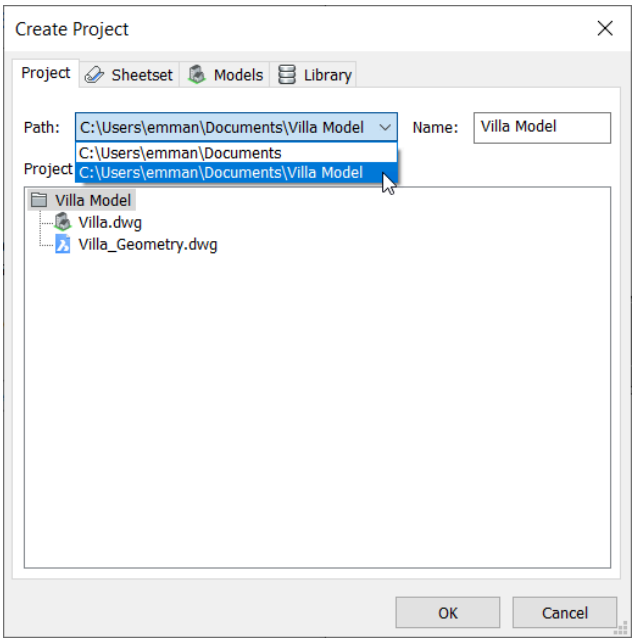
To create a BIM project, you must select one or more model drawings.

### Project

1 Click the **Create Project...** button in the **Project Browser**. A dialog box appears.



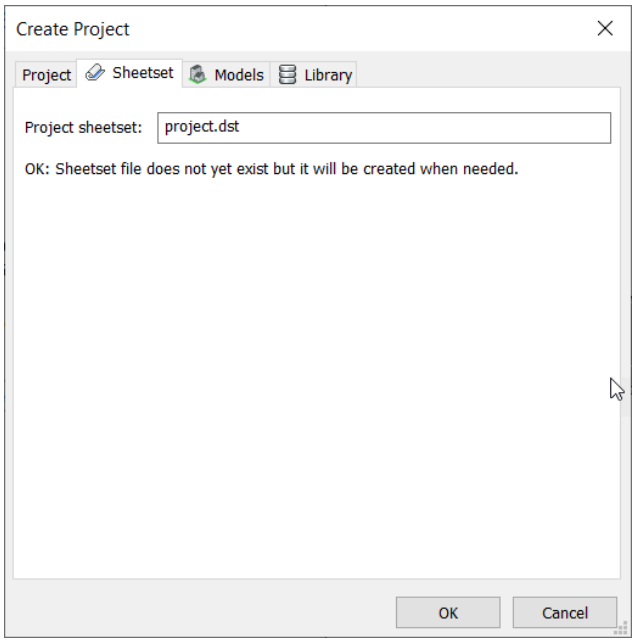

2 **(Optional)** Click the **Path** list button and select a parent folder.




3 **(Optional)** Type a name in the **Name** field (the default name is the name of the parental folder).

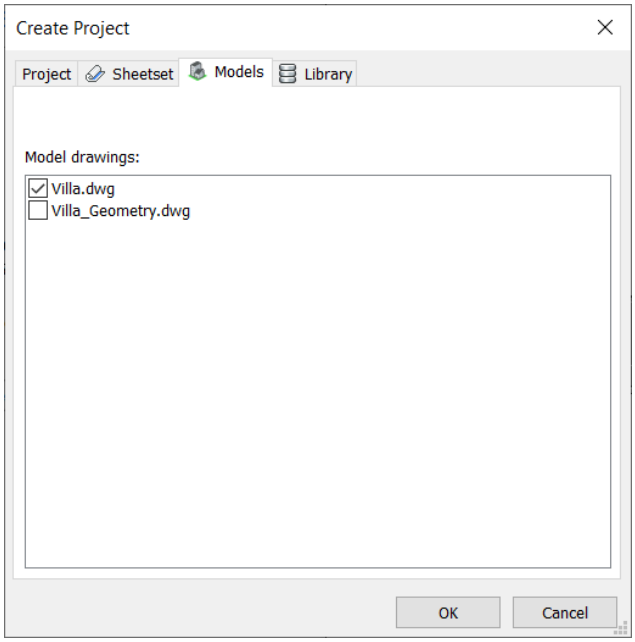
### Sheetset

Click the **Sheetset** tab. Enter a name for the **Project sheetset** (the default name is project.dst).




### Models

Click the **Models** tab. Here, the files in the selected parent folder (and subfolders) are displayed. Check all the models you want to incorporate in your drawing.



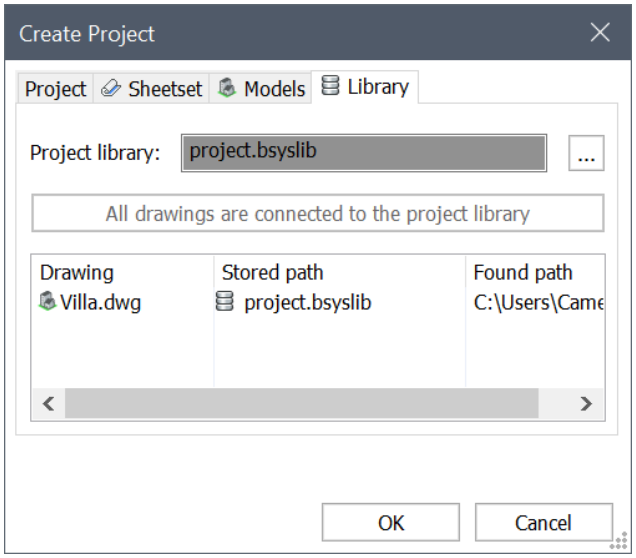

### Library

The **Project Library** can either be **external** or **embedded**. Sharing the **project library** means that you can use the same library over different drawings, this is called an external library. The shared library is stored in an external BIM Database File with the .bsyslib extension. An embedded library stores the **Project Library** in the drawing file.

- To make an external library:

***Прим.:*** By default, the library is set to **Embedded**.

a Go to the **Library** tab. Click the **Browse** button (...) of the **Project Library** option. Enter a name for the **Project library** (the default name is project.bsyslib).




b Click **OK** to display the **Sheetset Setup** dialog box.

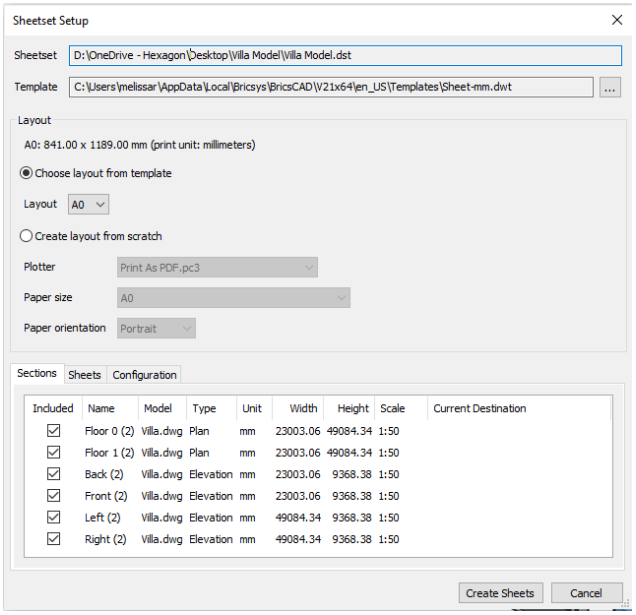
### 20.4.16.4 Sheetset setup

**Sheetset**: Shows the path of the sheetset file.

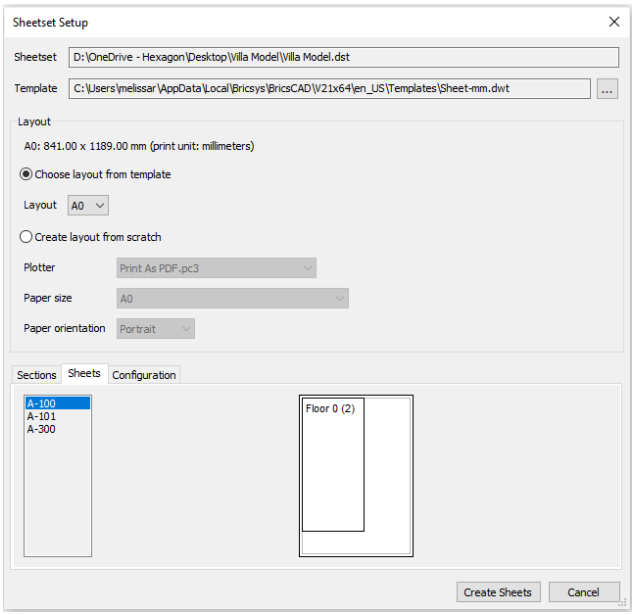
**Template**: Shows the path of the current template. Click **Browse** (...) to change the template.

**Layout**: Selects which layout (different paper sizes) the section results will be generated.

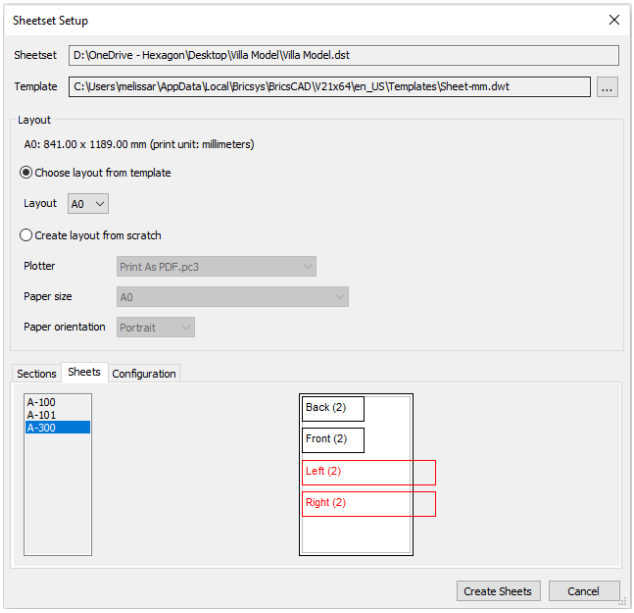
1 The **Sections** tab lists all sections. For each section with an empty **Current Destination** field, a viewport and sheet view are generated.



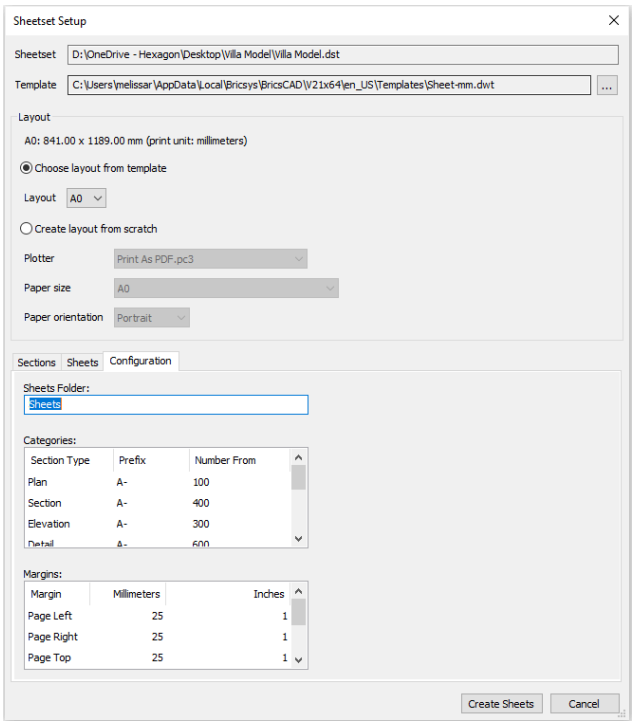
2 Click **Sheets** to preview each sheet (within the selected layout).




***Прим.:*** When the viewport is larger than the layout, it displays in red.

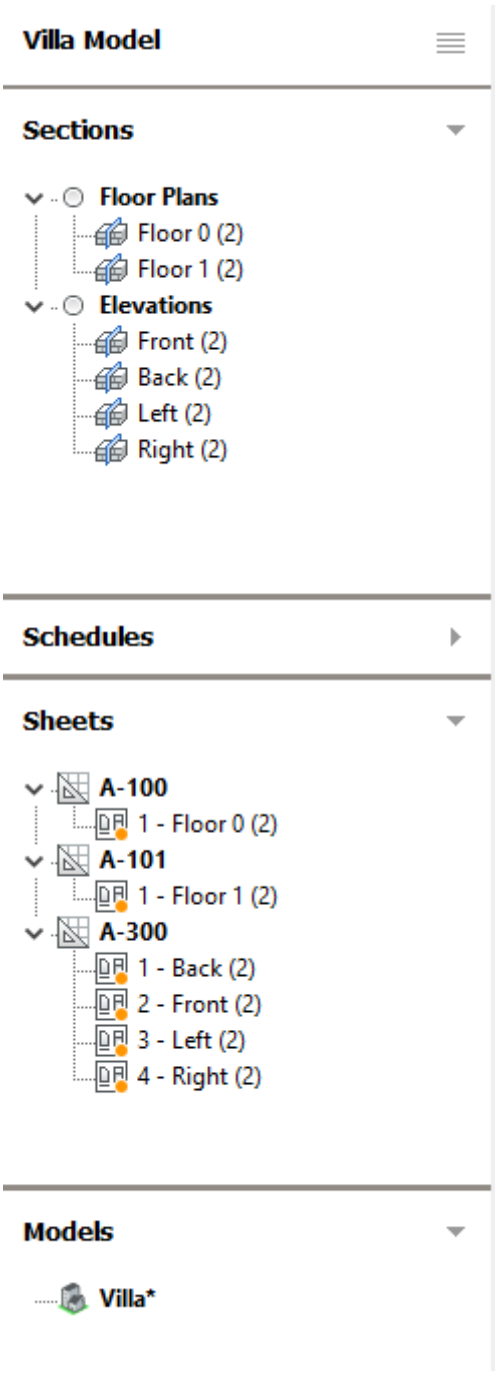



3 Click the **Configuration** tab. Slowly double-click (or click and hit Space/Enter) to change the properties. Under **Sheets Folder** you can change the name of the folder where the sheets will be stored.



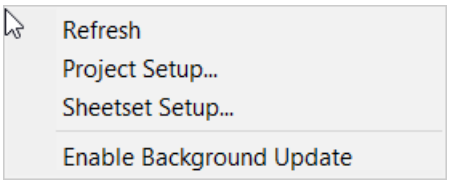

4 Click **Create Sheets**. An overview of the project displays and the sheets are generated.

***Прим.:*** The drawings with an orange fill circle icon are out of date. Right-click on them and select **Update** to match changes in the models included in the project.




### 20.4.16.5 Editing a project

Click on the **Menu** button (). A context menu displays:




- **Refresh**: Refreshes the Project Browser to match changes in the model.
- **Project Setup...**: Opens the Project Setup dialog box.
- **Sheetset Setup...**: Opens the Sheetset Setup dialog box.
- **Enable Background Update**: Calculates section results automatically (without interrupting user workflow). Sections are assigned a color; Queued: blue, Outdated: red and In progress: green.

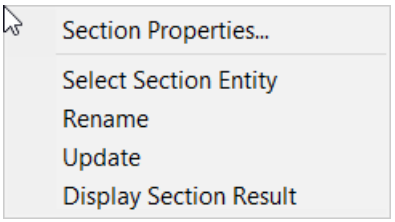
### 20.4.16.6 Creating schedules

See the Schedules article for more information about Schedules.

### 20.4.16.7 Working in the project browser

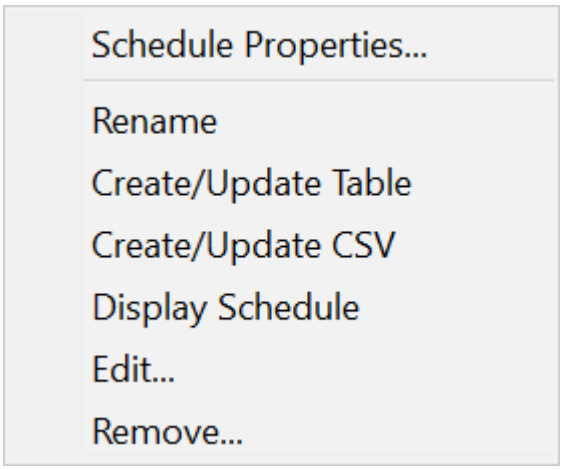
Right-click an item in the BIM **Project Browser** to display a context menu.

**Sections**



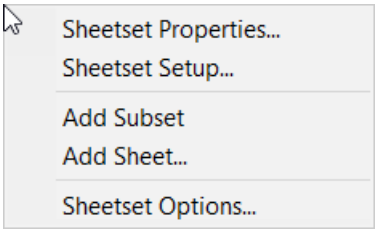

- **Section Properties...**: Opens the **Section Properties** dialog box where you can see and edit the properties of a section, such as the name and the result path.
- **Select Section Entity**: Selects the section entity in the project model drawing.
- **Rename**: Renames the section entity.
- **Update**: Recalculates the section result.
- **Display Section Result**: Displays the section result in the corresponding sheet and zooms in on its viewport. If not already open, the drawing is opened.

**Schedules**



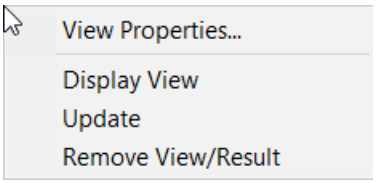

- **Schedule Properties...**: Opens the **Schedule Properties** dialog box that allows you to modify the properties of a schedule.
- **Rename**: Renames the schedule.
- **Create/Update Table**: Creates a sheet with the schedule or updates the sheet.
- **Create/Update CSV**: Creates a CSV file with the schedule information and saves it in the folder where the model file is located or updates the CSV file if it already exists.
- **Display Schedule**: Displays the schedule in the resulting drawing. If not already open, the drawing is opened.
- **Edit**: Opens the **Wizard Page** dialog box that allows you to edit the schedule.
- **Remove...**: Displays the **Remove** dialog box that allows you to choose between **Schedule Entity** and **Sheet View/Result** and removes the schedule.

**Sheets**




- **Sheetset Setup...**: Opens the **Sheetset Setup** dialog box that allows you to create a layout from scratch.
- **Add Subset**: Adds a subset in the **Sheets** tab.
- **Add Sheet...**: Adds a new sheet in a subset in the **Sheets** tab.
- **Sheetset Options...**: Opens the **Sheet sets** tab in the **Settings** dialog box that allows you to modify the sheet sets options.

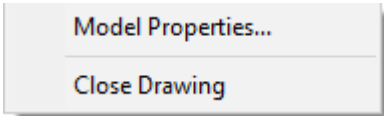
**Sheet views**



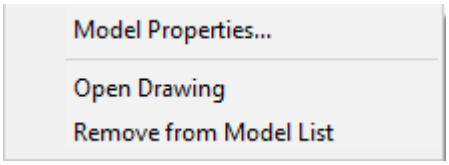

- **View Properties...**: Opens the **Sheet View Properties** dialog box that allows you to modify the properties of a sheet view.
- **Display View**: Displays the corresponding result and zooms in on its viewport.
- **Update**: Updates the sheet view. It executes the BIMSECTIONUPDATE command for the related **BIM** section.
- **Remove View/Result**: Removes the view and result from the corresponding sheet.

**Model drawings**

Context menu of a currently open model:




Context menu of a currently closed model:




- **Model Properties...**: Opens the **Model Properties** dialog box that allows you to see more information about the model.
- **Close Drawing**: For a currently open model, it closes the model.
- **Open Drawing**: For a currently closed model, it opens the model.
- **Remove from Model List**: Removes the drawing from the project model drawings.

### 20.4.17 Spatial locations

#### 20.4.17.1 Commands

BIMATTACHSPATIALLOCATION, BIMSPATIALLOCATIONS

#### 20.4.17.2 About

Spatial locations allow you to specify building and its stories in a project. In BricsCAD, a BIM model file can contain one site and multiple buildings and a building can contain multiple stories. When the entities are classified as a building element in the project, these classified entities have a **Building** and a **Story** property. Every building element in the BIM model resides in a particular story within a particular building.



For more information about this command, visit the Command Reference article BIMSPATIALLOCATIONS.

### 20.4.17.3 Procedure: assigning a building and story to a building element

1 Launch the BIMSPATIALLOCATIONS command.

2 Click the **New Building** icon ( ) to add a new building to the model.

3 Fill out the properties grid to specify a new name or value of the existing property.

4 Click the **New Story** icon ( ) to add a new story in the selected building.

5 Fill out the properties grid to specify a new name or value of the existing property.

6 **(Optional)** Click the **Delete** icon ( ) to remove the selected building or story.

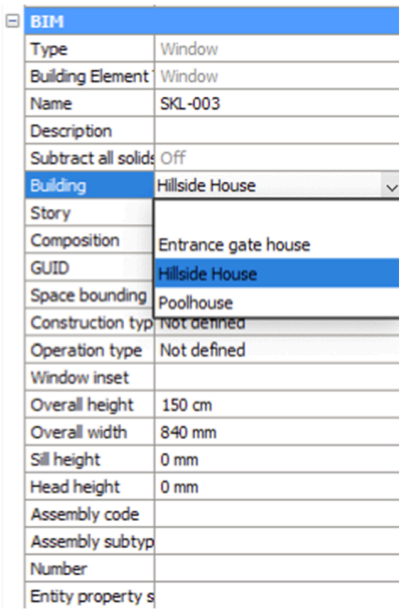
7 **(Optional)** Click **Import Spatial Locations** button ( ) to import a spatial location from a .txt file.

8 Select building element(s) in the drawing area to associate them with the appropriate spatial locations.

9 The properties of the selected building elements display in the **Properties** panel.

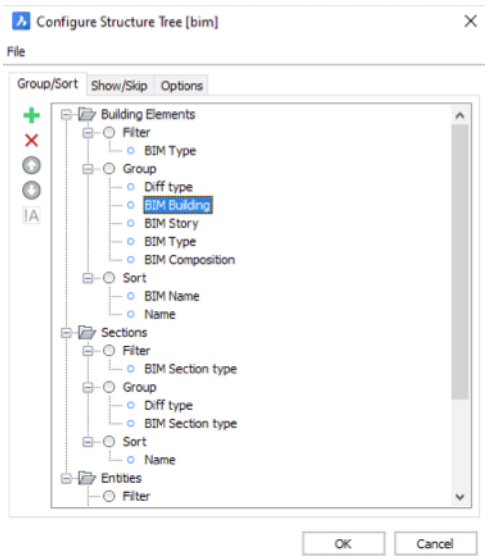
10 Do the following to assign the building and story to these building elements:

- Expand the **BIM** section in the **Properties** Panel.
- Click the **Building** field, then use the arrow to select a building.
- Click the **Story** field, then use the arrow to select a story.




### 20.4.17.4 Viewing the spatial locations in the structure browser

The **Structure browser** allows to view all of the entities in a drawing. By default, the structure of the building elements is organized in a particular form. Building elements are grouped by Building, Story, BIM type, and by composition respectively. This organization can be fully configured. Right click in the **structure browser** and click **Configure**, a new window will open. See the STRUCTUREPANEL command for more info.



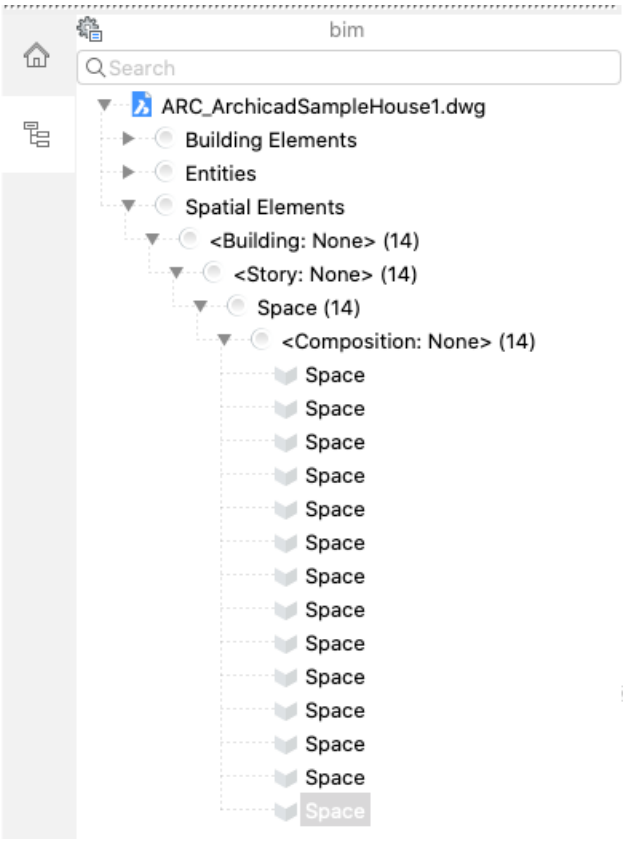

If there is no **BIM Building** or **BIM Story** in the **structure browser**, you can add them manually. See Structure Panel for more info.

### 20.4.17.5 About BIMATTACHSPATIALLOCATION



Assigning new Spaces can be done using the BIMSPACE command. Spaces are not always assigned to the according buildings because they don't have a spatial location yet. Use the BIMATTACHSPATIALLOCATION command to assign a spatial location to spaces.

For more information about this command, visit the Command Reference article BIMATTACHSPATIALLOCATION.



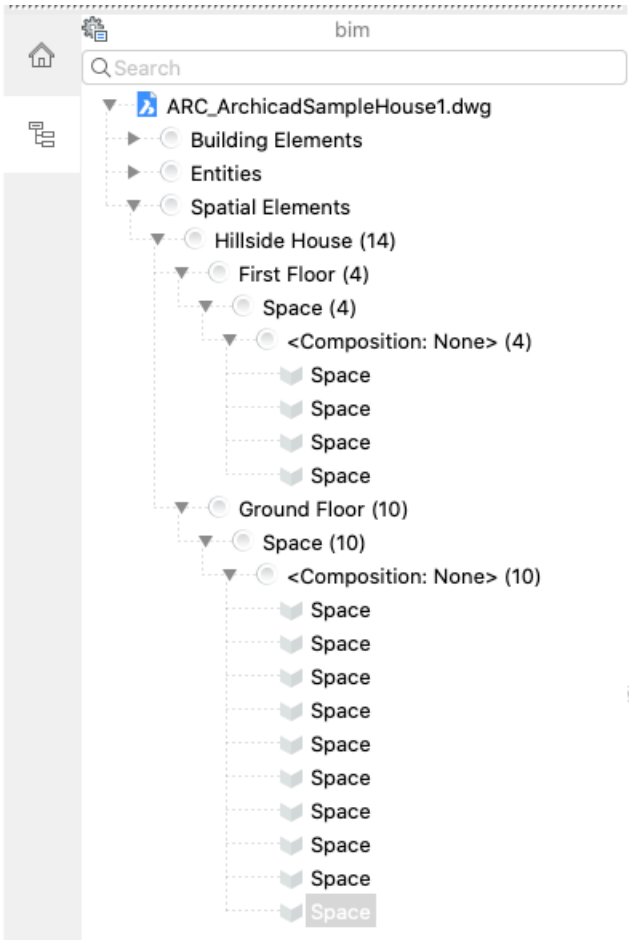

### 20.4.17.6 Procedure: attaching spatial locations

1 Launch the BIMATTACHSPATIALLOCATIONS command.

2 **You are prompted:***Enter location number or [Auto attach locations/Unattach current location]:* Press **A** to auto attach the spaces to their according building and floor level.

3 Select the spaces to update and press Enter.

4 **(Optional)** Press U to unattach the spatial location.




### 20.4.18 Structure browser

#### 20.4.18.1 Commands

STRUCTUREPANEL

#### 20.4.18.2 About

The **structure browser** is an interactive tree that displays the entities in the current model. Using the configurable structure tree on the **Structure Browser** the BIM model can be organized in a way that you want to view the elements. Once you configured the tree, you can easily save this structured tree as a .cst file. By default, the .cst files are stored in the **Support Folder**.



Place the cursor over a toolbar or ribbon panel, then right-click and choose **Structure** in the context menu.

**Or**

Type STRUCTUREPANEL in the Command line, then press Enter.

### 20.4.18.3 Procedure: using the structure browser

1 Open the **structure browser**.

By default, the **structure browser** displays on the left-hand side on your screen. With a **search** field at the top of the panel (1).

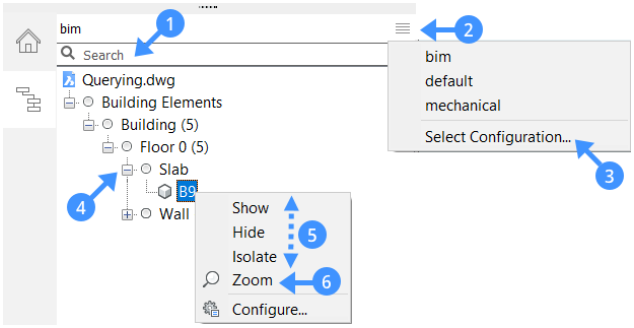
2 Click the horizontal stripes button (2) to load a configuration file in the list. Click **Select Configuration...** to select a .cst file, which specifies a configuration (3). The current configuration displays on the left. Based on the selected file the BIM model will be organized in a tree-structure.

3 Click the expand/collapse icon (4) of a node to expand or collapse the node. Right-click on a group name to select **Expand All** or **Collapse All** and expand or collapse all the subnodes at once.

You can **Show**, **Hide** or **Isolate** the entities, selected in the **structure browser**, in your model space, by right-clicking on the selected names (5).

This context menu also provides you to **Zoom** (6) into a specific entity in the model.

The following illustration shows the **Structure Browser**.




### 20.4.18.4 Procedure: querying in the structure browser

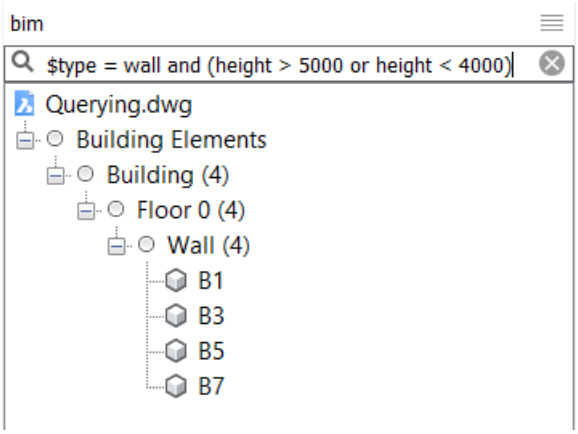
1 Typing something in the **Search** field at the top of the **structure browser** will issue a simple Text search: e.g. if you type 'wall' in the search field and press **Enter**, the **structure browser** will be filtered to everything with 'wall' in its name, including walls and curtain walls.

2 Text searches can be useful for a quick search, but more powerful queries are possible using expressions.

3 An expression starts with the **$** E.g. if you enter *$type = wall* and press **Enter,** the browser will be filtered on entities that have the BIM type = wall. Thus, using this expression, the curtain wall doesn't show up in the browser anymore.

4 Expressions can be more complex, such as filtering on multiple properties. E.g., you can enter *$type = wall and height > 5000* to filter on all walls with a height over 5000 mm.

5 Logical expressions can be nested using brackets: *$type = wall and (height > 5000 or height < 4000)*.




### 20.4.18.5 Procedure: managing the tree-structure

The tree structure of the **Structure browser** represents the hierarchical set of rules in a graphical form. Each rule has a filter property, a grouping property, and a sorting property. Organizing the properties in the **Structure Browser** enhances the performance of your project while querying the entities among others. When you have a tree for your BIM model you can establish a hierarchy across its building element types, buildings, stories, etc. You can use the hierarchy to manage the properties at various levels of the data display.

Do the following to configure the tree-structure:

1 Right-click on the name of your drawing in the **Structure Browser**, then choose **Configure** in the context menu.

The **Configure Structure Tree** (7) dialog box displays.

2 Select any existing rule in the structure tree, then click the **Add icon** (8) or right-click and choose **Add Rule** in the displayed context menu. This menu can also be used to delete and rename the existing rule.

The new rule is added below the previously selected rule. Type a new name for the newly created rule by clicking its name field (9).

3 Select the **Group** (10) node of the rule, then use the Add icon or right-click and choose **Add Grouping Property** (11) in the context menu.

The **Select Property** dialog box displays. Select a property on the list that you want to use to group the properties in your **Rule** setting. Then, click the **OK** button. The new property is added to the **Group** node.

If you use the **Filter** node to add a property, the selected property will be added in this node.

4 To move an item in the tree, select the item, then click the **Move Up** (12) or **Move Down** (13).

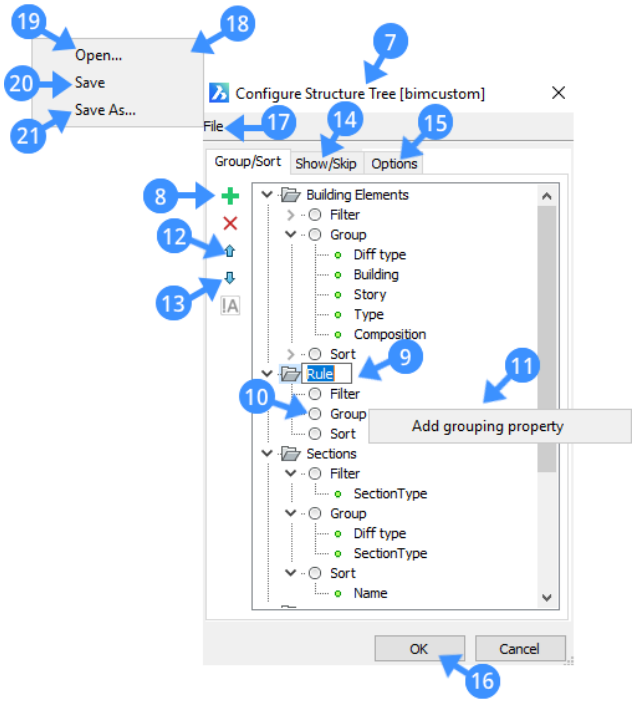
5 Select the **Show/Skip** (14) tab to sort the entity types or exclude them on the **Structure Browser**.

6 Select the **Options** (15) tab to set the entities action in the model using the structure tree. You can here specify whether or not selecting an entity in the **Structure Browser** also selects the entity in the model space.

7 Do one of the following to save the configured tree:

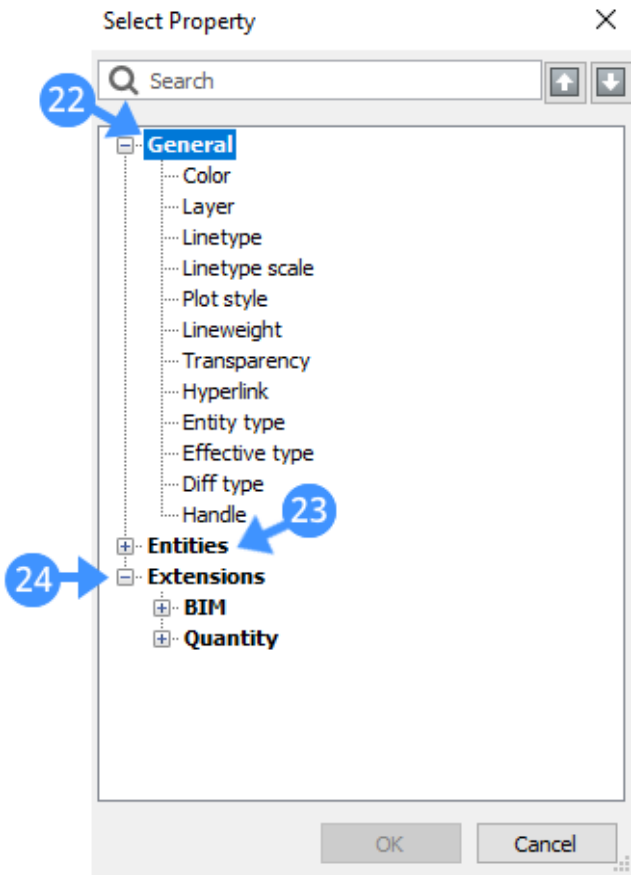
- Click the **OK** (16) button.
- Click the **File** (17) menu in the dialog box and choose one of the available options on the list (18).

The following illustration shows the **Configure structure tree** dialog box.




| | |
|---|---|
| **Open (19)** | Opens a file dialog to select another .cst configuration file. |
| **Save (20)** | Saves the current configuration and keep the file dialog open. |
| **Save as (21)** | Saves the current configuration under a different name and keep the file dialog open. |

The following illustration shows the **Select Property** dialog box.




| | |
|---|---|
| **General (22)** | Contains general properties. |
| **Entities (23)** | Contains various types of entity properties. |
| **Extensions (24)** | Contains various types of properties for **BIM** and **Quantity** extensions. |

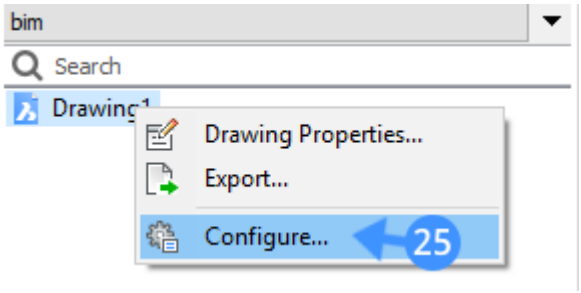
### 20.4.18.6 Procedure: managing the tree-structure with custom properties

In BricsCAD, the user-defined/custom properties are available to add in the **Select Property** dialog using the GUI. The **Select Property** dialog provides a list of dynamic properties for the drawing based on the current workspace. Note that custom properties, through the **Select Property** dialog box, can be used to group the tree structure.

Do the following to group the tree-structure using the existing custom properties:

1 Right-click on the name of your drawing in the **Structure Browser**, then choose **Configure** (25) in the

context menu.




The **Configure Structure Tree** dialog box displays.

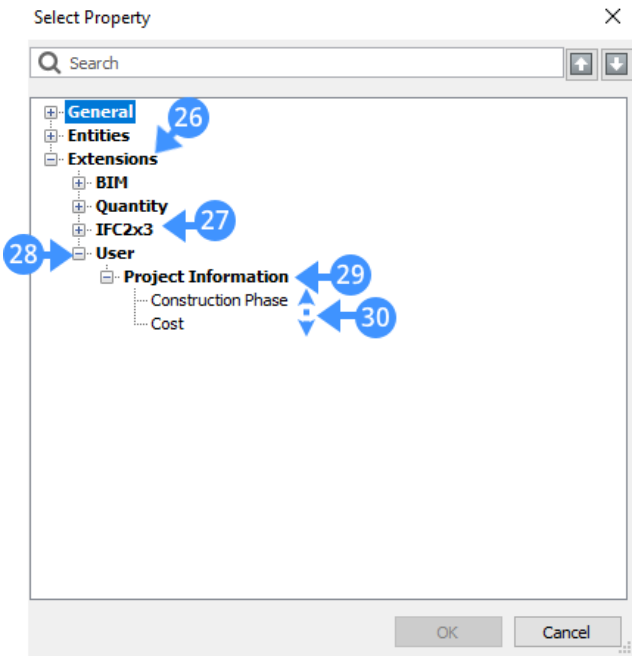
2 Select the **Group** node of the rule, and click the **Add rule or property**(+) icon.

**Or** right-click on the Group node and choose **Add Grouping Property** in the context menu. The **Select Property** dialog box displays (see below image for the **Select Property** dialog). If you have custom properties, these properties will be shown under **Extensions** (26) and grouped by its namespace.

3 Click the expand/collapse icon of a node to expand and collapse the node.

4 The following image shows the **IFC2X3** (27) and **User** (28) namespaces in the **Select Property** dialog.

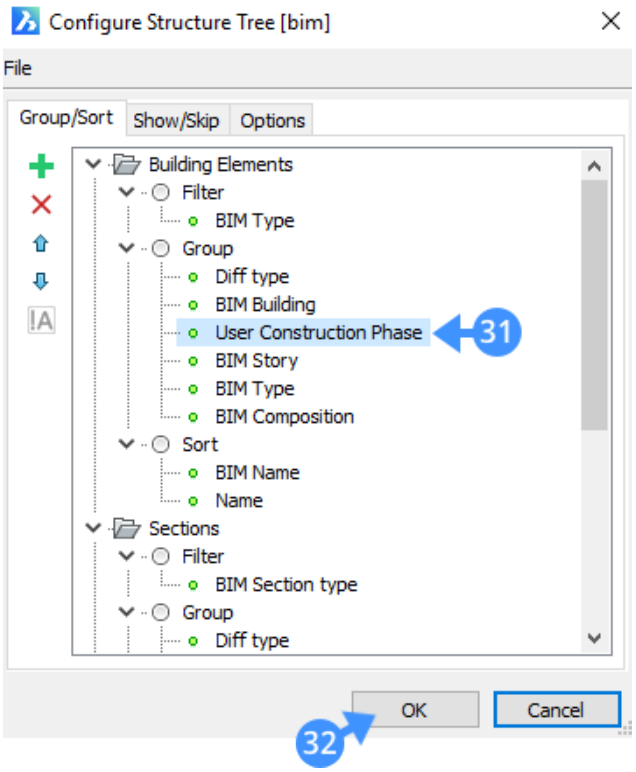
For example, the **User** namespace is expanded to display the property set (29) and properties (30) that are in this node.




5 Select the property to use for grouping in the tree structure, and press **OK**.

The property is added (31) to the **Group** node.

6 Press **OK** (32) to use the property for grouping entities in the **Structure Browser**.




## 20.5 Средства проектирования

### 20.5.1 Using bimify

#### 20.5.1.1 Commands

BIMIFY

#### 20.5.1.2 About BIMIFY

BIMIFY analyzes a model and will automatically classify spatial locations, spaces, buildings, and stories. It also detects and classifies the external and internal walls.

BIMIFY can automatically detect whether a model is an Architectural, Structural or MEP model. BricsCAD uses this information during the autoclassification process. Define this information before you launch BIMIFY, to improve the accuracy of the automatic classification.

The solids are auto-classified as columns, beams or members. If their profiles match a definition in the profile library, the profile name will be automatically added to the entity's properties as meta-data. If no matching profile definition is found, a new one is created in the library.

BIMIFY can be used in a model that has both classified and unclassified entities. The classified entities are also linked to the appropriate spatial locations.

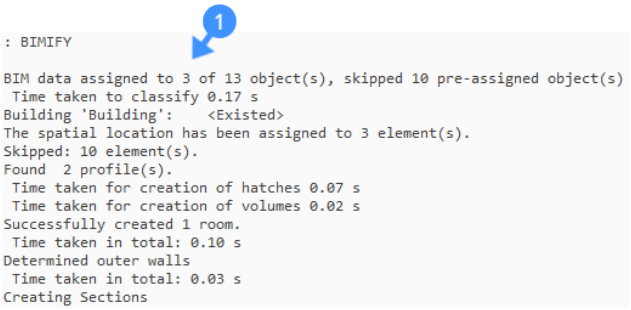


For more information about this command, visit the Command Reference article BIMIFY.

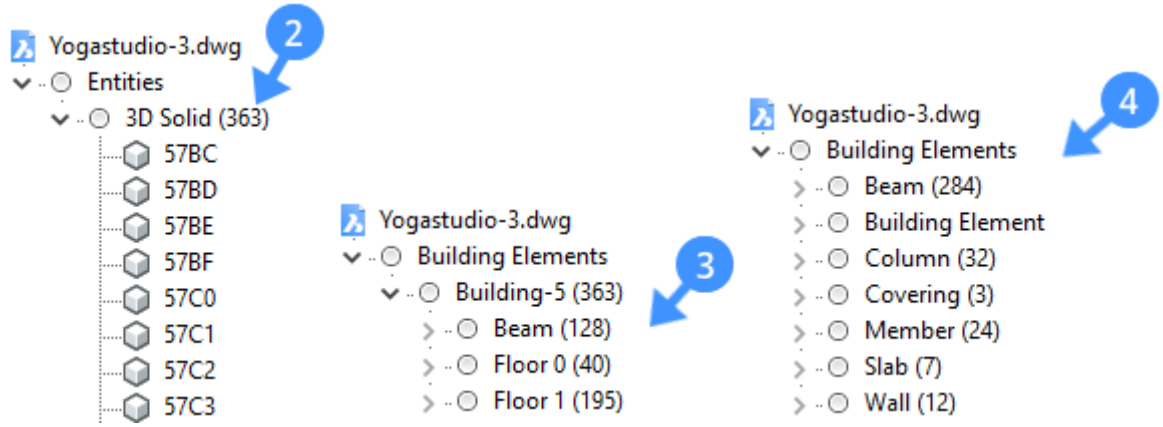
### 20.5.1.3 Procedure: bimify your entire drawing

Choose BIMIFY from the Quad or type BIMIFY in the Command line and choose **Entire model**.

When the BIMIFY process is completed, an overview of the result (1) is provided in the **Prompt History** panel.



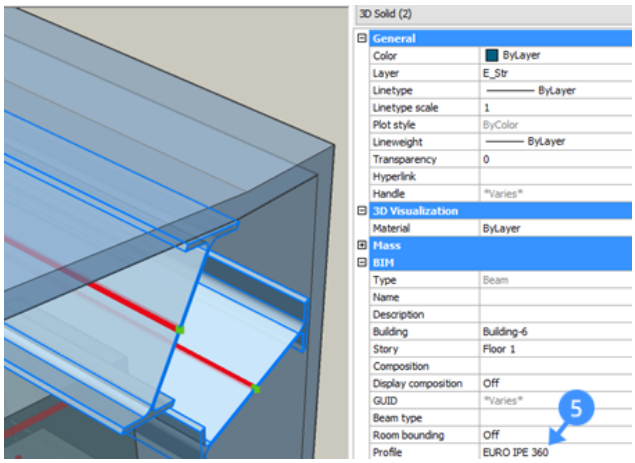

Bimify creates new spatial locations if necessary.




*Left to right:*

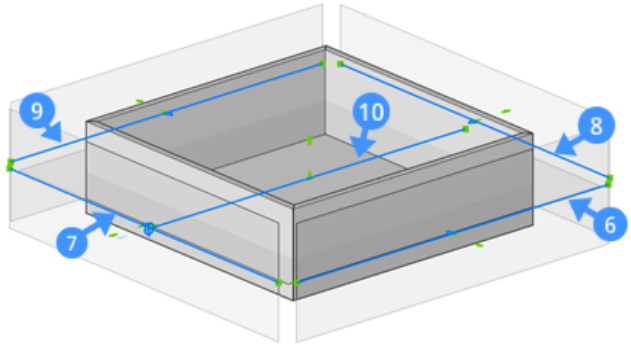
- ***Before Bimify:*** *The structure tree displayed only 3D solids (2)*
- ***After Bimify:*** *The entities are sorted into their corresponding building, story (3) and building element type (4).*

***Прим.:*** The **structure browser** can be customized to control the content of the tree.



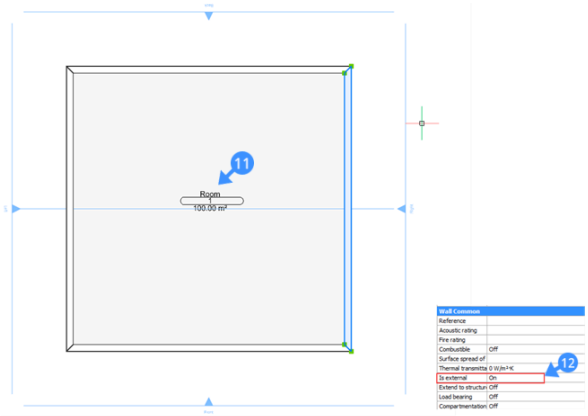
*The profile data of a beam in the properties panel (5). BIMIFY recognized the profile of these model elements as structural members and detected the axis location of the profile.*

The elevation views and floor plan sections are created last.



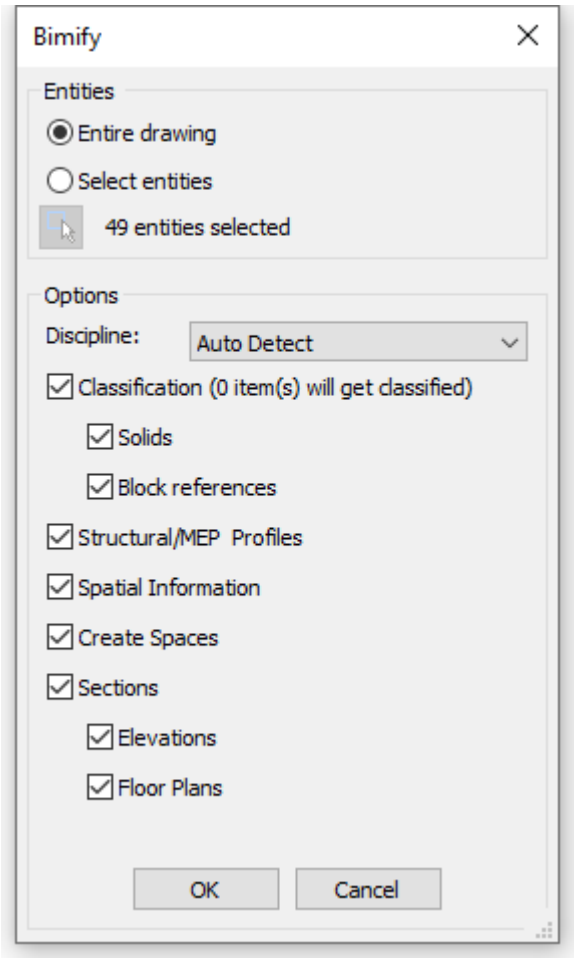

*This building contains front (6), Left (7), Right (8) and Back (9) views as well as a floor plan (10).*

For this simple building, one room and 4 external walls have been detected.




*The BIM room (11) and **Is External** (12) property under the **Wall Common** properties.*

If you choose to use the **Advanced** option of BIMIFY, a dialog box will appear:



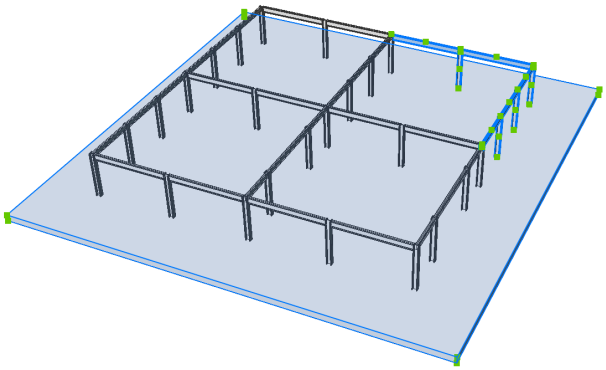

| | |
|---|---|
| **Discipline** | Auto Detect<br>Architectural<br>Structural<br>MEP<br>Mixed model<br>Defines the discipline of the drawing manually. This information helps improve the accuracy of Bimify. |
| **Classification** | Classifies **Solids** and **Block References** automatically. |
| **Structural/MEP profiles** | Assigns columns, beams, members or flow segments profiles if available in the BricsCAD profile library.<br>This option is available when **Classification** is selected only. |
| **Spatial Information** | Assigns spatial locations such as buildings and floors, detects spaces and identifies internal and external walls.<br>This option is available when **Classification** is selected only. |

| | |
|---|---|
| **Sections** | Creates an elevation view and a plan section per floor.<br>This option is available when **Spatial Information** is selected only. |

***Прим.:*** To disable the autoclassification option for solids or blocks simply uncheck the related checkbox. By default, all checkboxes are ticked. However, BricsCAD remembers your previous settings. Ticking the Spatial Information checkbox automatically includes the Classification option. This information is needed to detect the other elements.

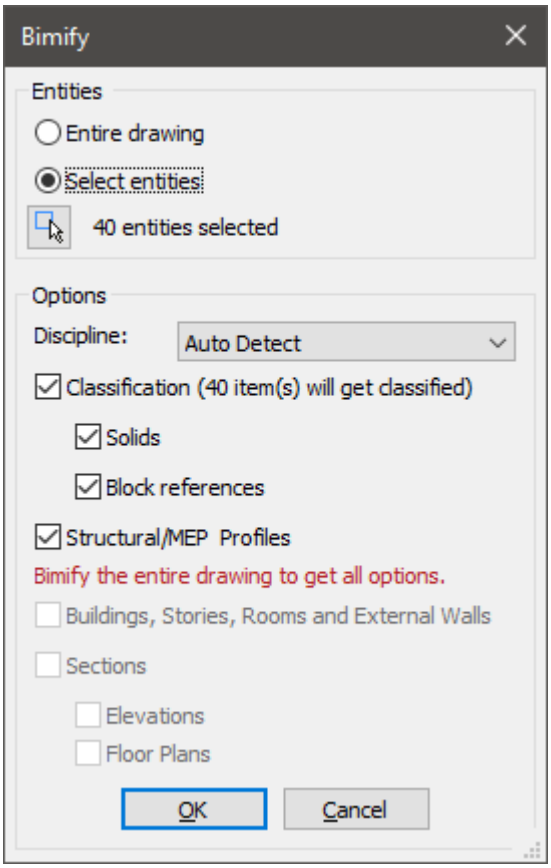
### 20.5.1.4 Procedure: bimify a selection of your BIM Model

1 Select the part of your drawing you want to Bimify.



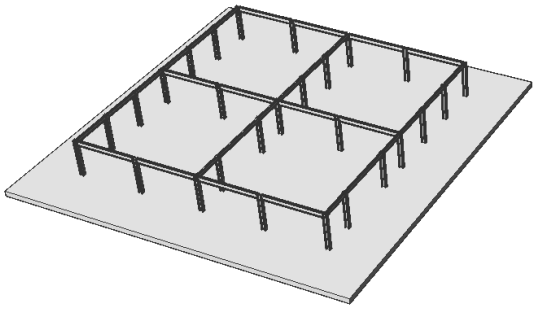
2 Type BIMIFY in the Command line and type in **A** for Advanced.

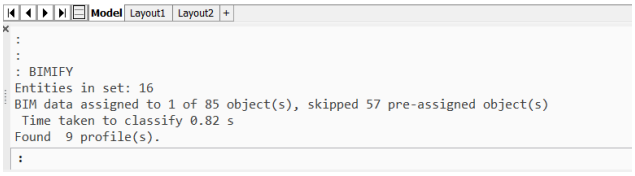
A dialog box displays:



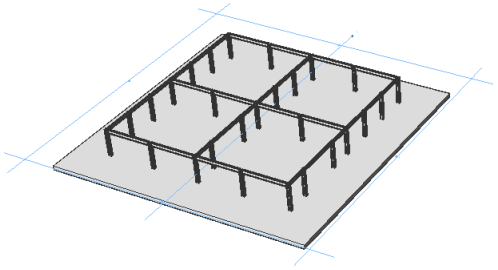

***Прим.:*** If you only select a part of your model, only solids, blocks and profiles can be classified. BricsCAD can only detect entities like stories, spaces, and sections when the whole drawing is selected.

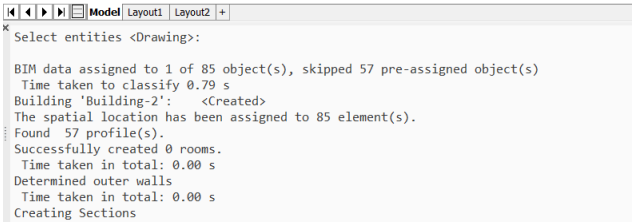
3 When the Bimify process is completed, an overview of the result is provided in the **Prompt History** panel.






*BIMIFY classified the selected part of the model. No sections for floor plans or elevations are created.*






*This is the result if BIMIFY was launched on the whole drawing and all the boxes in the dialog box were checked. The entire model is classified additionally there are sections for the floor plan and the 4 elevations of the model.*

### 20.5.2 Using propagate

#### 20.5.2.1 Commands

BIMPROPAGATE

### 20.5.2.2 The five flavors

Before getting into the BIMPROPAGATE workflow it is important to note that there are five variants of BIMPROPAGATE, each working on a limited set of situations. These five variants (or flavors) were designed to guide new users through the process of BIMPROPAGATE. The tool has so many different uses that it can seem overwhelming to a new user; these variants help keep the tool more clear and concise, as they are designed for more specific use cases. However, they are still distillates from the original BIMPROPAGATE command, so every problem you can solve with a particular flavor you can **also** solve using the standard BIMPROPAGATE command. A more advanced user will even be able to solve problems that can not be solved by any of the particular flavors.

The BIMPROPAGATE variants are:

- BIMPROPAGATEPLANAR: Propagation of connections between planar elements such as walls, slabs and roofs.
- BIMPROPAGATELINEAR: Propagation of connections between linear elements such as beams, columns, pipes and ducts, and maximum one planar element.
- BIMPROPAGATEPATTERN: Propagation of a single element or group of elements on a flat surface to multiple locations and grids; can be used to propagate light fixtures, light switches, windows, air diffusers, columns on a grid.
- BIMPROPAGATEEDGES: Propagation along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc.
- BIMPROPAGATECORNER: Propagates details connected to multiple planar reference solids coming together in one node.

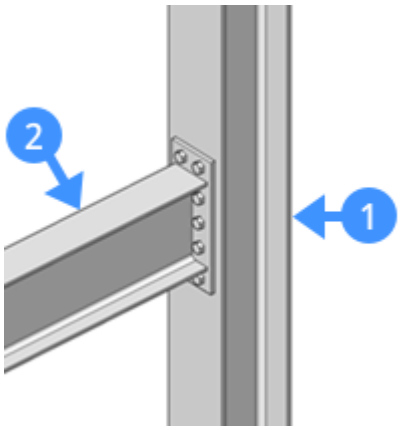
The general workflow of BIMPROPAGATE consists of two major steps: defining the detail volume, and choosing where to apply this detail.

### 20.5.2.3 Defining the detail volume

**You are prompted:**

Select reference solids. These will not be copied, but used to match other locations. [selection options (?)]:

**The first step** of every propagation is specifying the **reference solids**. These are the solids that define the situation where this particular detail should be applied. For instance, the image below shows a column-beam connection using an end plate and several bolts. This detail was modeled for this particular beam connecting with this particular column, and now we want to propagate this detail for every situation where a column and beam of these types come together. Thus, the reference solids in this case will be the column (1) and the beam (2). Later in the process, BIMPROPAGATE scans the model for situations where similar solids occur, and tries to map the detail onto these locations.




***Прим.:*** This is an example of a situation that can be solved by BIMPROPAGATELINEAR.

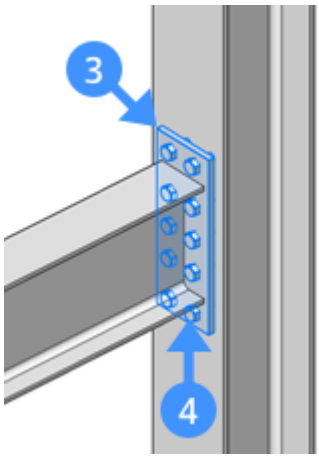
Press **Enter** to stop selecting reference solids.

**The second step** is specifying the detail objects or sub entities. These are the objects that you wish to propagate or copy throughout the model. They can be solids, block references or even faces (e.g. a hole in the reference solid can be propagated by selecting its faces).

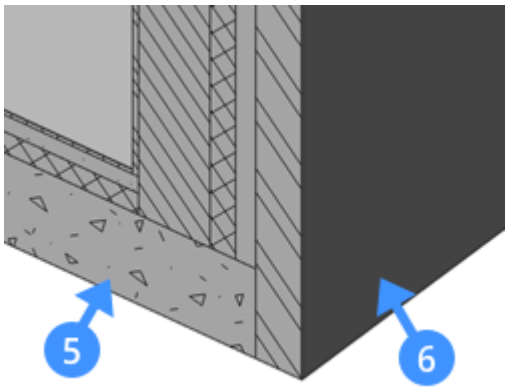
**You are prompted:**

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied. [selection options (?)]:

In the example at hand the detail objects will be the end plate (3) and the bolts (4).




***Прим.:*** In some cases detail objects are not even required. Let's say we want to propagate the connection between the floor slab and the wall shown in the image below. Our reference solids are then the floor slab (5) and the wall (6), and no detail objects are required. In this case you can skip this step by pressing Enter or right click.



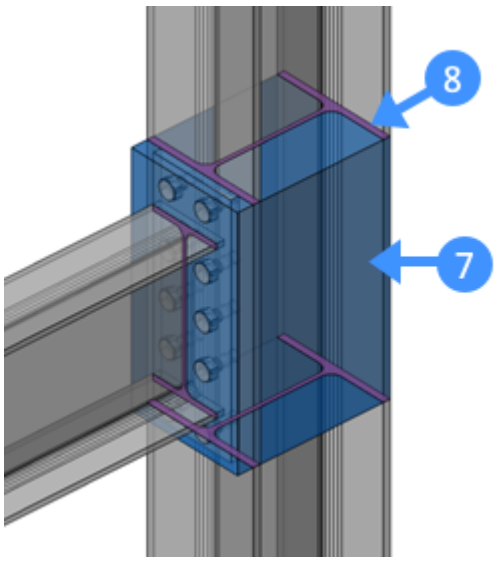

If we were to model extra solids in this connection (e.g. an insulation block or a skirting board), these extra solids would then be detail objects.

***Прим.:*** This is an example of a situation that can be solved by BIMPROPAGATEPLANAR.

Once the **reference solids** and the **detail objects** are specified, a detail volume is defined automatically. The visual style is temporarily set to X-ray, and the view is automatically zoomed in onto the detail volume. Everything that is inside this volume and is either a detail object or part of a reference solid will be copied to the other locations when executing BIMPROPAGATE.

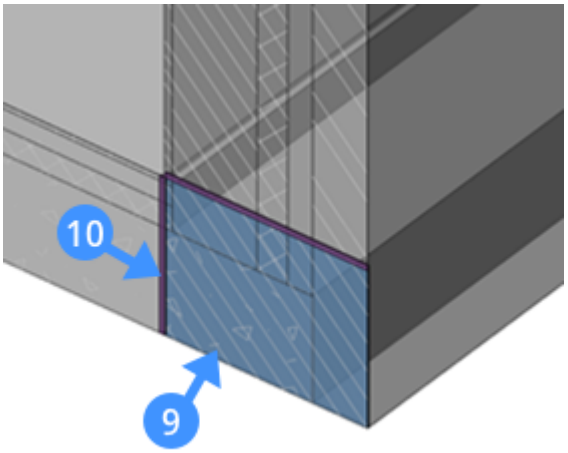
A detail volume can be either a **3D volume** or a **2D slice through a linear detail**.

In our example of the column-beam connection, the detail volume will be as shown in the image below. This is an example of a 3D volume. The connection of the reference solids to the detail volume is highlighted in purple (8).




**Something interesting to note:** In this particular detail, there are holes in the column where the bolts are bolted into the column. These holes are not necessarily detail objects, but since they are part of a reference solid and they are inside the detail volume, they will be copied as well. Thus, in this respect, BIMPROPAGATE does more than a 'dumb' copy of objects.

The detail volume of our wall-slab connection will look different. As shown in the image below, the detail (9) will be planar, and displayed in a section view. This is because the detail can be propagated as an extrusion of this 2D detail slice. Here as well, the locations where the reference solids connect to the detail volume are highlighted in purple (10).




***Прим.:*** The detail of BIMPROPAGATEPLANAR and BIMPROPAGATEEDGES will always be a 2D slice through a linear detail.

**You are prompted:**

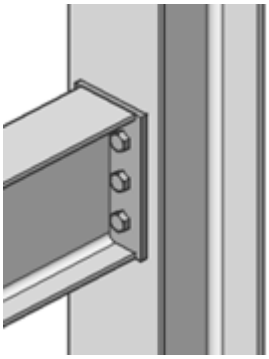
Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

**You are prompted:** Propagate detail as an extrusion of this 2D section? [Yes/No/Inflate first] <Yes>:

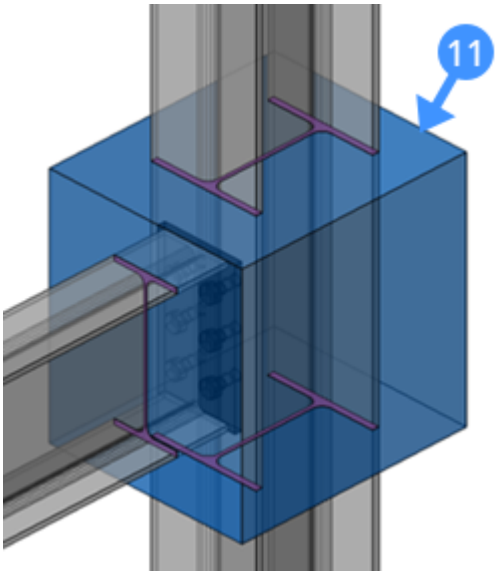
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Usually, the detail volume as it is proposed by BIMPropagate will be sufficient for what the user is trying to achieve. However, it is possible to inflate the detail to include a larger volume. This feature has two particular use cases:

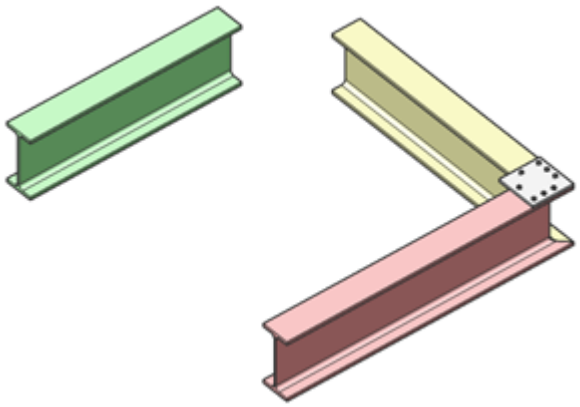
- If you want to replace an already existing detail with a new detail that is **smaller** than the original one. For example, we want to replace the previously proposed column-beam connection by the following:

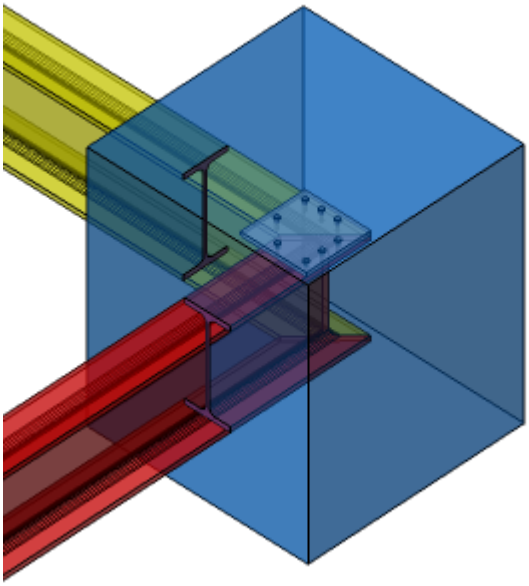
  In this case, we will have to inflate the detail volume (11) so that it is larger than the original detail:

- You might want to include a larger part of the reference solid, for example to connect with target solids that are further apart. In the image below, we want to propagate the connection between the **red** and the **yellow**beam, so that the **yellow** and the **green** beam have a similar connection.



To achieve this, we need to inflate the detail volume so that it overlaps the end faces of the red and the yellow beam.



Once the detail volume includes everything you want to propagate, you can go over to the second step: choosing where to apply the detail. For 3D detail volumes there is one last choice to make: Propagate the detail as a **Block** or as a **Copy**. If you choose to propagate the detail as a block, then it will group all detail objects into a new **Block Definition** and copy around references to this **Block Definition**.

#### 20.5.2.4 Choosing where to apply the detail

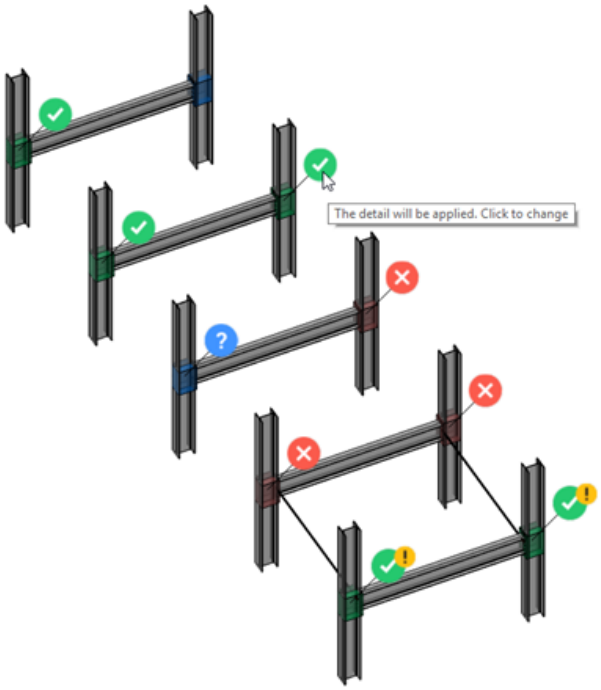
**You are prompted:**

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:

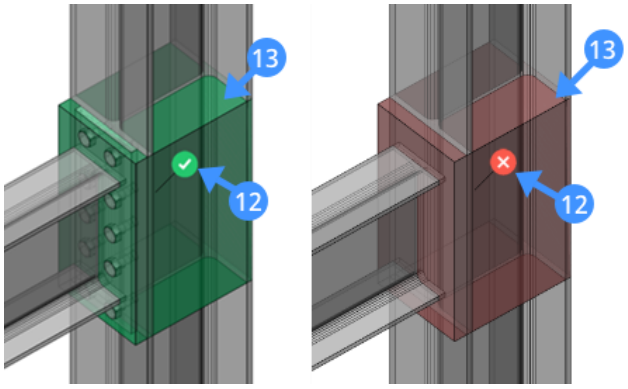
In this step the view zooms out showing all the possible suggestions on screen. Each suggestion is a location where this detail could be applied, and is indicated by one of the following icons (or widgets):

| | |
|---|---|
|  | The detail will be applied. |
|  | The detail will not be applied; it was manually toggled off by the user. |
|  | The detail will not be applied, because of one of several possible reasons. Hovering over the icon will provide more information. |
|  | The detail will be applied, but interferences were detected. |
|  | The detail will be applied, but there are several alternatives. Hovering over the icon will allow you to cycle between alternatives. |

Hovering over a widget reveals some more information. Clicking a widget changes its status: you can toggle suggestions individually by clicking their respective widgets. Pressing Enter will then apply all the widgets as they are shown at that point: suggestions with a green checkmark will be applied, others will not be applied.



A suggestion also shows a preview of what the detail will look like. In the images below, you can both see the widgets (12) and the previews (13).




Widgets can also be **selected** using a window selection. A selected widget is highlighted by a blue border. Once one or more widgets are selected, pressing Enter will no longer apply **all** the suggestions, but **only those that are currently selected:**

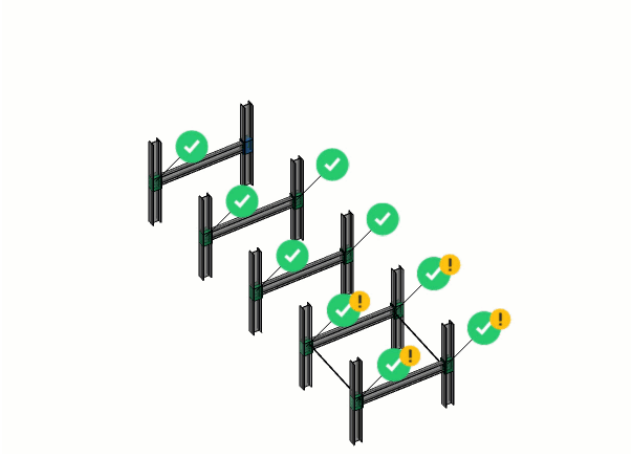
**You are prompted:**

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:

This procedure can be used to gradually solve all the suggestions.

In the video below, the following steps are taken:

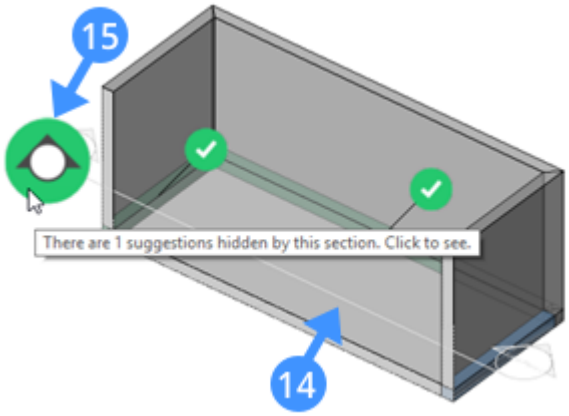
- Four widgets are selected and toggled off
- Enter is pressed: the four selected suggestions were all toggled off, so they are not applied, and the four suggestion are removed from the view
- Four more suggestions are selected
- Enter is pressed: the four selected suggestions are applied, and removed from the view
- There is one suggestion left; Enter is pressed to accept this suggestion, and the BIMPROPAGATE command is terminated.



It is important to emphasize that BIMPROPAGATE only processes entities that are currently visible. As a result, hidden entities, or entities on a hidden or frozen layer are not included when BIMPROPAGATE is scanning the model for locations where the detail can be applied. This can be a useful when executing BIMPROPAGATE in a large model; e.g. when you want to propagate a steel connection, only turn on those layers that contain steel members. This will improve the propagation performance.

It is possible to activate or move existing section planes during BIMPROPAGATE.

- A section plane (14) can be activated by double-clicking it.
- Left-click it once to move it.
- When a section plane is activated, a widget (15) might appear next to it indicating that there is one or more suggestion currently hidden because of the active section. Click the widget to deactivate this section and show the hidden suggestions.



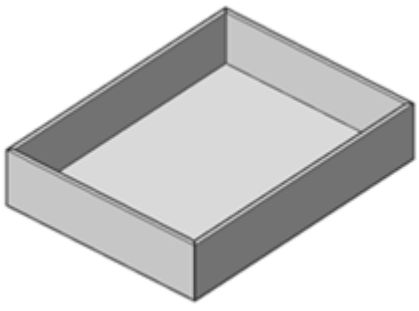

### 20.5.3 Propagate planar

#### 20.5.3.1 Commands

BIMPROPAGATEPLANAR

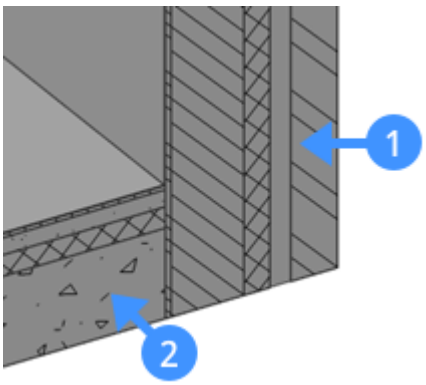
#### 20.5.3.2 Procedure: propagating a wall-slab connection

1 Consider the following situation of four walls and one slab, created with **QuickDraw**.

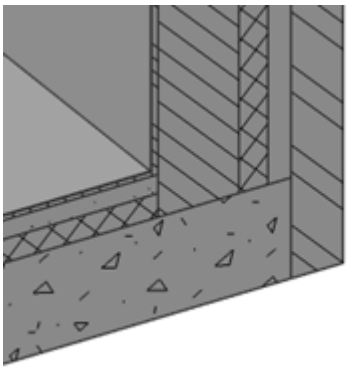


2 The four walls were given a composition, as well as the floor slab. Without any further editing or modeling, creating a vertical section through the building will show the connection one of the walls (1)

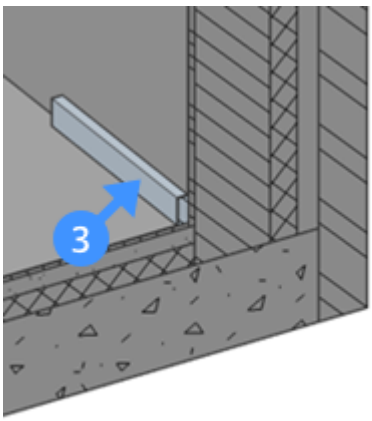
and the slab (2) to be something similar to the following image.




3 While this section is active, we can **Push/Pull** the different ply faces to obtain the desired connection between the wall and the slab:



4 We can detail this connection even further, for example by locally adding a skirting board (3). This was done by simply drawing a box and classifying this solid as a **Building Element**.




5 Now that we have this detailed connection, we want it to be applied to every location where a wall and slab of this type come together. Let's deactivate the section and launch BIMPROPAGATEPLANAR.

6 **You are prompted:**

Select planar reference solids that form the connection. The connection detail will be copied to similar solids. [selection options (?)]:

Select the wall and slab, and press Enter.

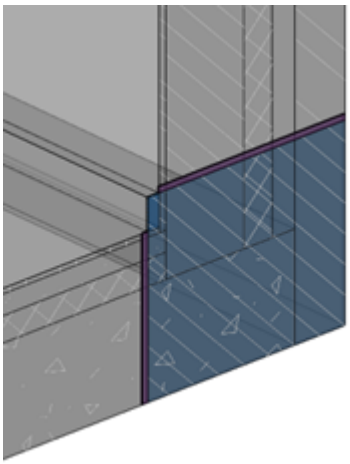
7 **You are prompted:**

Optional. Select detail objects to be copied as part of the connection. [selection options (?)]:

Select the skirting board solid, and press Enter.

8 **You are prompted:**The detail section varies too much along the extrusion direction.

In this particular case, BIMPROPAGATEPLANAR cannot find a solution: because the skirting board is not modeled along the entire length of the wall, BIMPROPAGATEPLANAR finds contradicting detail volumes. This problem can be solved easily by activating a section where the skirting board is present (e.g. the section we used earlier to create this detail) and trying again. If we now follow the same steps as before, BIMPROPAGATE finds a detail volume as shown in the image below.



9 **You are prompted:**

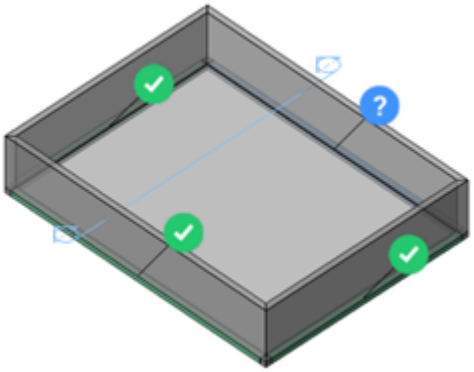
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command. Press **Enter** to accept.

10 **BimPropagatePlanar** now scans the entire drawing for similar locations. When this is done, all sections are deactivated and the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed:

**You are prompted:**

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:



A green checkmark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information.

In this case, it says **Location of original detail. Click to replace original detail by adapted extrusion**. Because we modeled the skirting board only locally, we should replace the original detail by an adapted extrusion so that it covers the entire length of the wall. Clicking the widget will turn it into a green checkmark.

11 Press **Enter** to complete the command.

Now, all four walls should have this particular connection with the floor slab, including the skirting board.

### 20.5.4 Propagate linear

#### 20.5.4.1 Commands

BIMPROPAGATELINEAR

#### 20.5.4.2 About

BIMPROPAGATELINEAR is one of the five Propagate flavors. It can be used to copy detailed connections between two or more linear solids. Common examples are column-beam connections or column-slab connections.

For a more in-depth explanation of the general principles of Propagate, read the Using Propagate article.

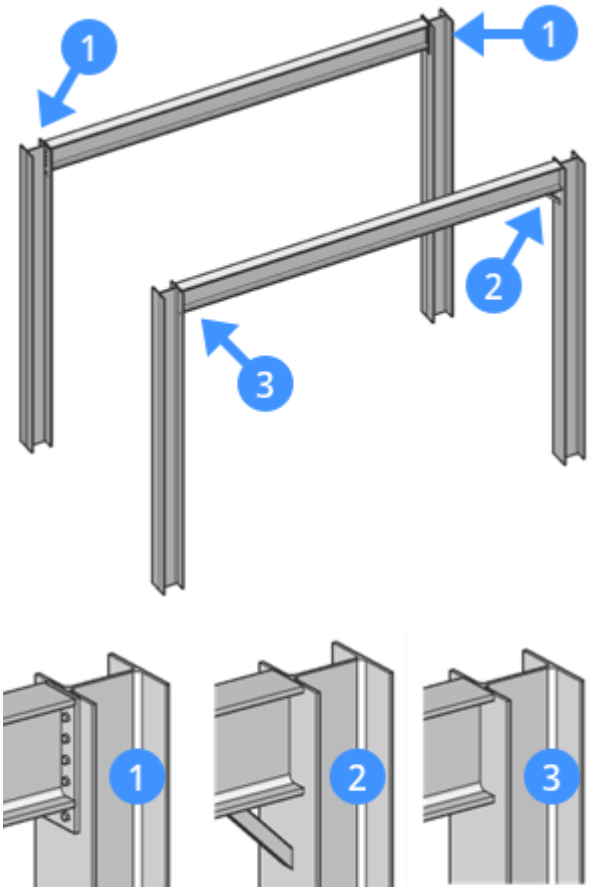


For more information about this command, visit the Command Reference article BIMPROPAGATELINEAR.

#### 20.5.4.3 Procedure: propagating a column-beam connection

1 Consider the following situation of four columns and two beams. Two of the connections have been designed with a plate and bolts through the column flanges (1). One connection has been designed with a diagonal member for additional rotational stiffness (2). The last connection has not been

designed yet (3).




2 We can now use Propagate Linear to copy one of the details to the other locations. In this example we will copy connection 2 to all four locations.

3 We will launch BIMPROPAGATELINEAR.

4 **You are prompted:**

Select the linear or planar reference solids that form the connection. The connection detail will be copied to similar solids. [selection options (?)]:

At least one should be linear.

Select the column and the beam that are connected by the diagonal member, and press **Enter.**

5 **You are prompted:**

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied as part of the connection. [selection options (?)]:

Select the diagonal member and press **Enter**. A detail volume is created that completely encompasses the diagonal member and parts of the column and beam.

6 **You are prompted:**

Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain

Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

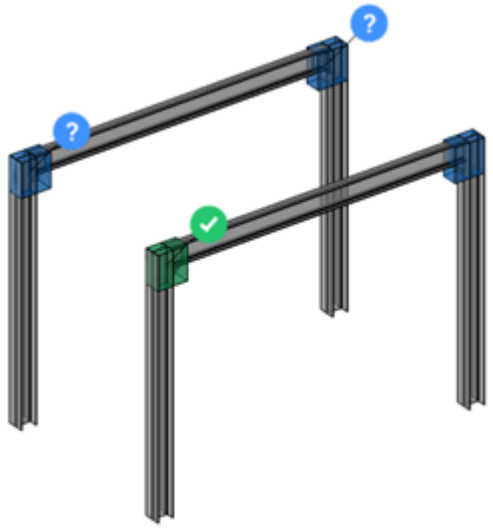
You can choose to propagate this detail as a block to keep the file smaller and more manageable, or as a copy. Inflating the detail is not necessary in this case. For more information on inflating, read the Using Propagate article. Choosing **No** will exit the command.

Press **Enter** to accept.

7 BIMPROPAGATELINEAR now scans the entire drawing for similar locations. When this is done, the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed:

8 **You are prompted:**

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:



A green check mark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information. In this case it says **Existing details detected. Suggestion will not be applied. Click to change**. We want to replace the bolted connection by this diagonal member, so we click both the blue widgets so they turn into green check marks.

9 If we now press **Enter**, all connections should be replaced by the diagonal member.

### 20.5.5 Propagate pattern

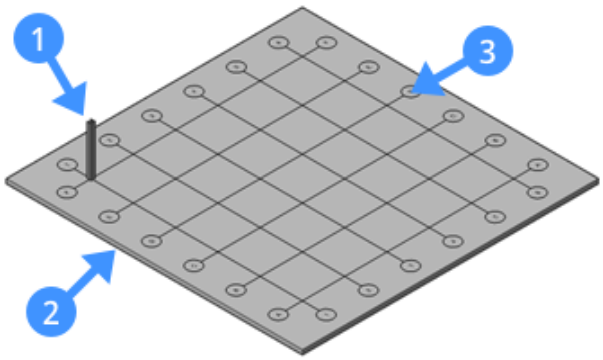
#### 20.5.5.1 Commands

BIMPROPAGATEPATTERN

#### 20.5.5.2 Procedure: propagating columns on top of a slab

1 Consider the following situation: a column (1) is placed on top of a slab (2). There is also a grid drawn

on top of this slab (3).




2 We want to place this column on specific axis intersections. We could manually copy the column but, Propagate allows us to do this faster.

3 Let's launch BIMPROPAGATEPATTERN.

4 **You are prompted:**

Select planar reference solid to which the detail is related. The detail will be copied to similar solids. [selection options (?)]:

Select the slab, and press **Enter.**

5 **You are prompted:**

Select detail objects (solids, block references, faces, edges,...) to be copied. [selection options (?)]:

Select the column, and press **Enter.**

6 A 3D detail volume is created.

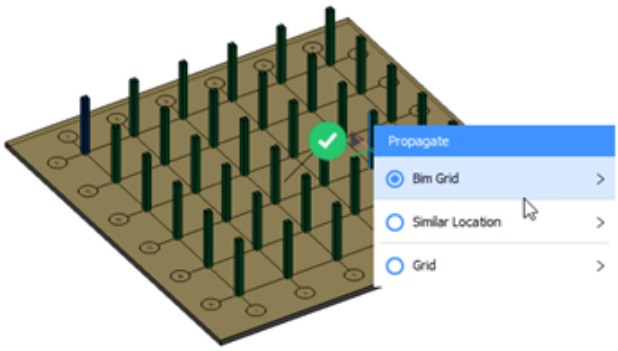
7 **You are prompted:**

Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command.

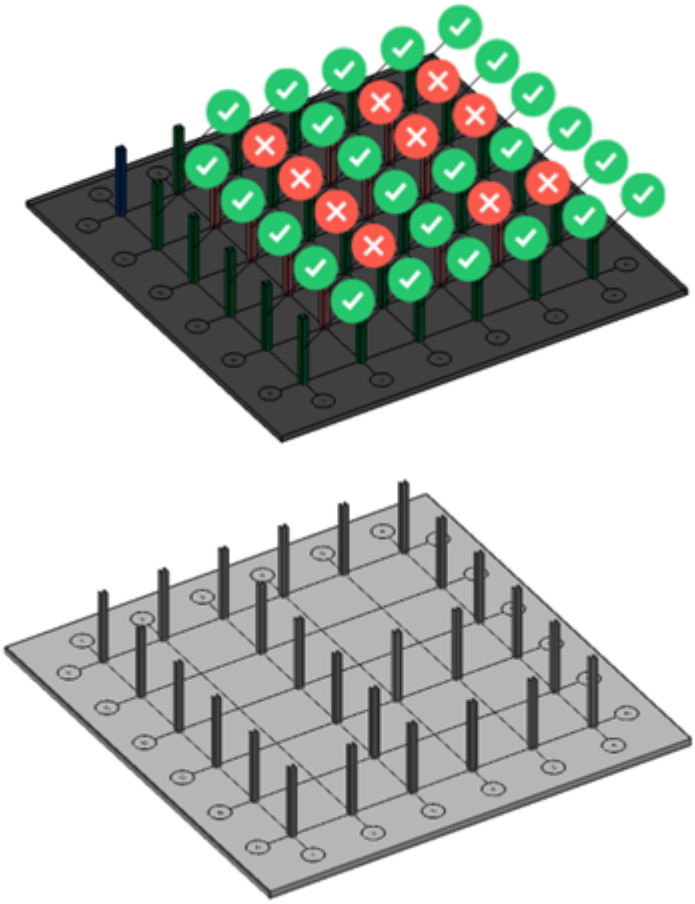
Press **Enter** to accept.

8 A widget appears. Hovering over this widget reveals a flyout menu with several options:




- **Bim Grid**: this is the grid that was drawn on top of the slab. Choosing this option and pressing Enter will copy this column to every axis intersection of this grid.
- **Similar Location**: this will copy the column to similar locations on this slab; in this case the four corners of the slab.
- **Grid**: this allows you to create your own grid of columns on top of this slab. You can choose to add or remove rows and columns. These rows and columns will always be evenly spaced.

9 We'll choose the BIM **Grid** option. Clicking this will expand the menu further; we now have the option to **Explode**. This will create a separate widget for each instance of this column, giving you more control over which suggestions you wish to apply (in this case which axis intersection you wish to place a column). We can now manually turn off suggestions:



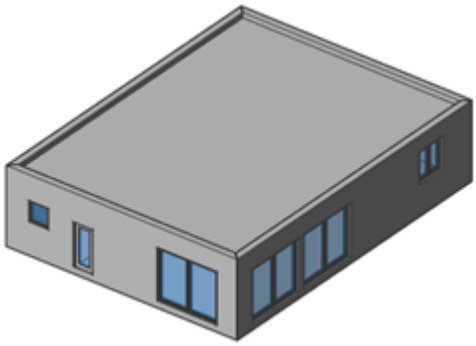
### 20.5.6 Propagate edges

#### 20.5.6.1 Commands

BIMPROPAGATEEDGES

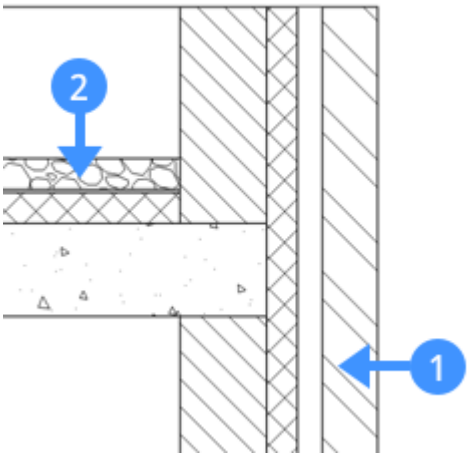
#### 20.5.6.2 Procedure: propagating a wall cap

1 Consider the following situation of a one-story building with a flat roof.

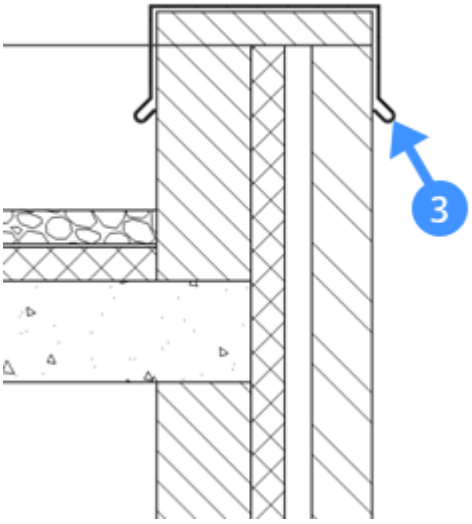


2 Let's create a vertical section plane to cut through our building and take a look at where the wall (1)

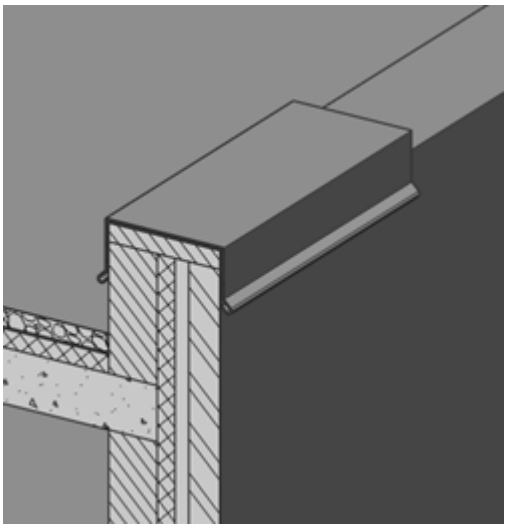
and the roof slab (2) come together.




3 We want to create a capping (3) on top of this wall to get the following result:




We can start by creating this locally on top of one wall, either by using direct modeling tools or inserting a block from somewhere else and exploding it. In our section it might look like this:



4 ow we can use Propagate to copy this detail over to all applicable locations. Let's deactivate the section and launch BIMPROPAGATEEDGES.

5 **You are prompted:**

Select planar reference solid to which the edge detail is related. The detail will be copied to edges of similar solids. [selection options (?)]:

Select the wall on top of which the capping was modeled, and press **Enter.**

6 **You are prompted:**

Select detail objects to be copied. These should be solids or faces that are parallel to an edge of the reference solid. [selection options (?)]:

Select the objects that make up the capping, and press **Enter.**

7 **You are prompted:**

Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command. Press **Enter** to accept.

8 **You are prompted:**

Propagated to edges with similar orientation only? [Yes/No] <Yes>:

Choosing **Yes** will only apply this detail to edges with a similar orientation, in this case wall edges that point upwards. Press **Enter** to accept.

9 **You are prompted:**

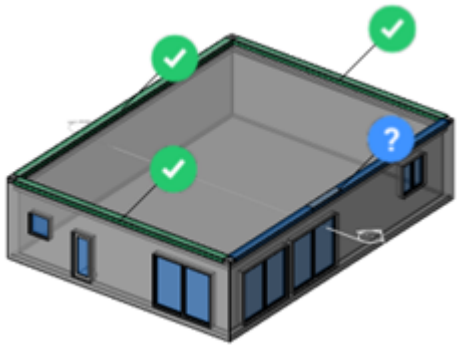
Choose location of detail relative to edge: [Outside reference solid/Inside reference solid] <Outside reference solid>:

This option controls whether the detail will be placed inside or outside (on top of) the base solids. When placed inside, the detail is subtracted from the base solids.

10 BIMPROPAGATEEDGES now scans the entire drawing for similar locations. When this is done, the

view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed:

11 **You are prompted:**Choose next action [Apply all/Cancel] <Apply all>:



A green check mark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information. In this case it says **Location of original detail. Click to replace original detail by adapted extrusion**. Because we modeled the wall cap only locally, we should replace the original detail by an adapted extrusion so that it covers the entire length of the wall. Clicking the widget will turn it into a green check mark.

12 Press **Enter** to complete the command.

Now, all four walls have this wall cap and the corner connections are created automatically.

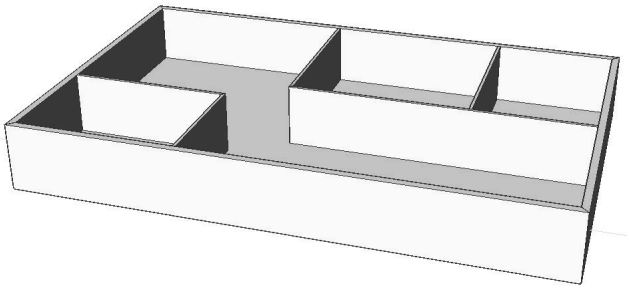
### 20.5.7 Propagate corner

#### 20.5.7.1 Commands

BIMPROPAGATECORNER

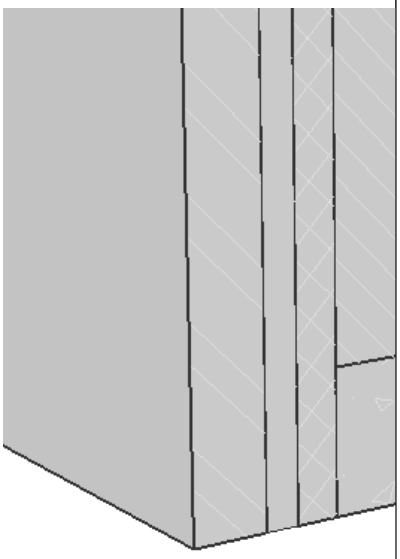
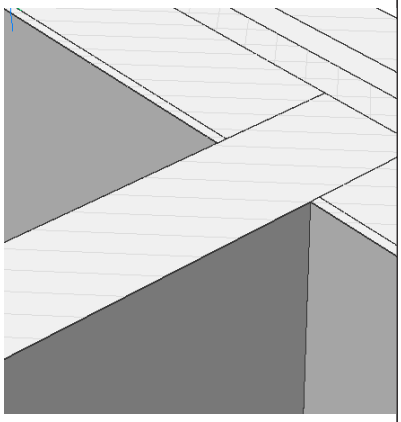
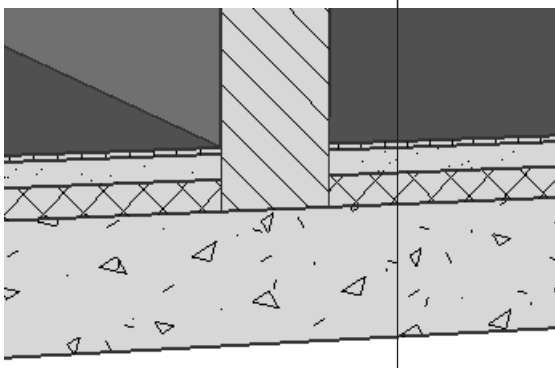
#### 20.5.7.2 Setting the foundations

In this example the 4 exterior walls, 5 interior walls, and floor slab have already been assigned a composition.

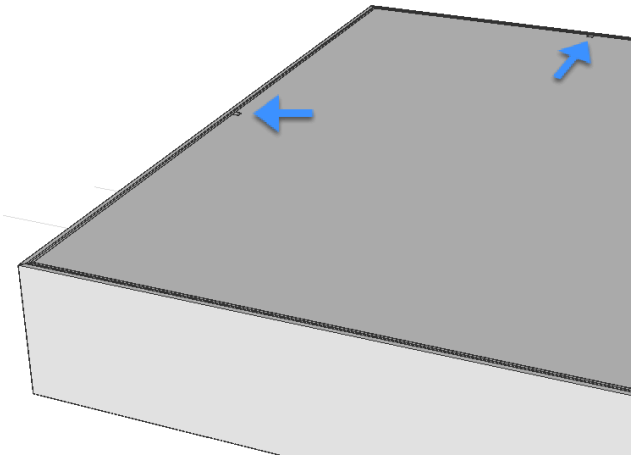


First, you must connect the 3 planar types of connections separately. For a detailed explanation of how to do this, read the article Propagate planar.

You should now have the following connections:

| | | |
|---|---|---|
|  |  |  |
| Exterior wall - Floor slab | Exterior wall - Interior wall | Interior wall - Floor slab |

The nodes where the exterior wall, interior wall, and floor slab come together can't be correctly propagated using BIMPROPAGATEPLANAR. Instead, use BIMPROPAGATECORNER. BIMPROPAGATECORNER can propagate details connected to three planar base solids (i.e. corners).



### 20.5.7.3 Propagate corner

1 Launch BIMPROPAGATECORNER.

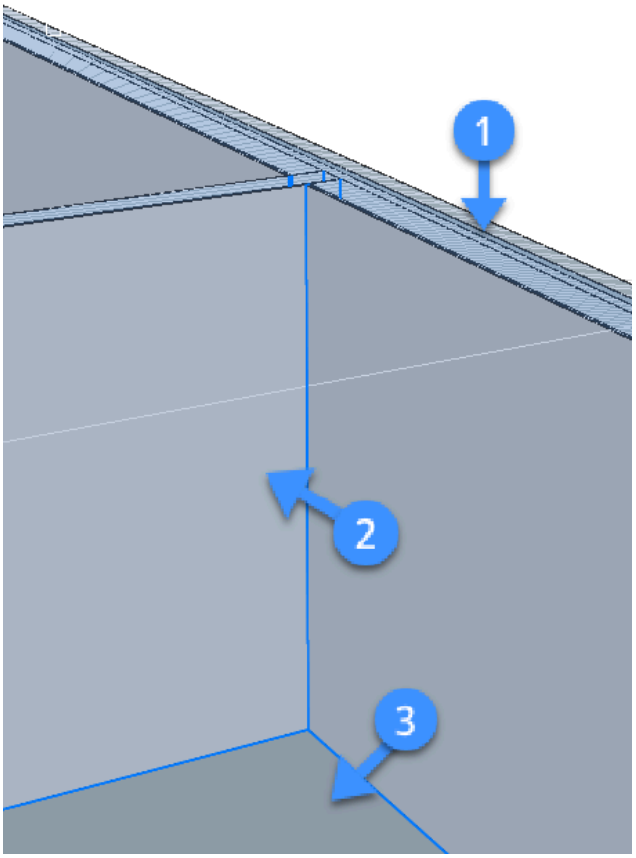
1 **You are prompted:**

Select at least three planar reference solids that form a 3D corner. The detail will be copied to similar

corners. [selection options (?)]:

2 Select the exterior wall (1), the interior wall (2), and the floor slab (3).

3 Press **Enter** to accept.




**You are prompted:**

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied as part of the corner detail. [selection options (?)]:

If you have extra solids in your detail, i.e. an insulation block, you can select this as an extra detail object and it will be copied to the other nodes when using BIMPROPAGATECORNER.

4 Press **Enter** when ready.

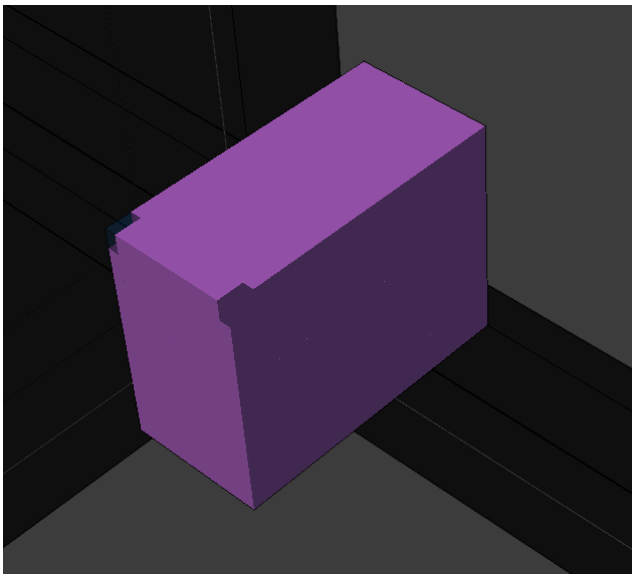
***Прим.:*** If you don't have any extra details, just press **Enter** to continue.

5 The volume that will be propagated is encased in a purple box.

6 **You are prompted:**

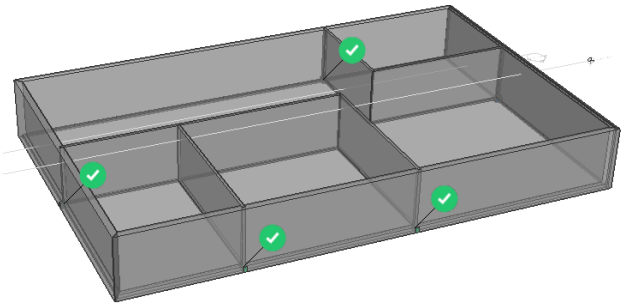
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

7 Press **Enter** to accept the detail.



***Прим.:*** BIMPROPAGATECORNER shows a 3D volume instead of a 2D section.

8 BIMPROPAGATECORNER now scans the entire drawing for similar locations. When this is done, the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed. A green checkmark means that this suggestion will be applied when pressing **Enter**.



**You are prompted:**Choose next action [Apply all/Cancel] <Apply all>:

9 Press **Enter** to complete the command.

The nodes where the exterior wall, interior wall, and floor slab come together should now be propagated correctly.

### 20.5.8 Parametrize details

#### 20.5.8.1 Commands

BIMPARAMETRIZEDETAIL

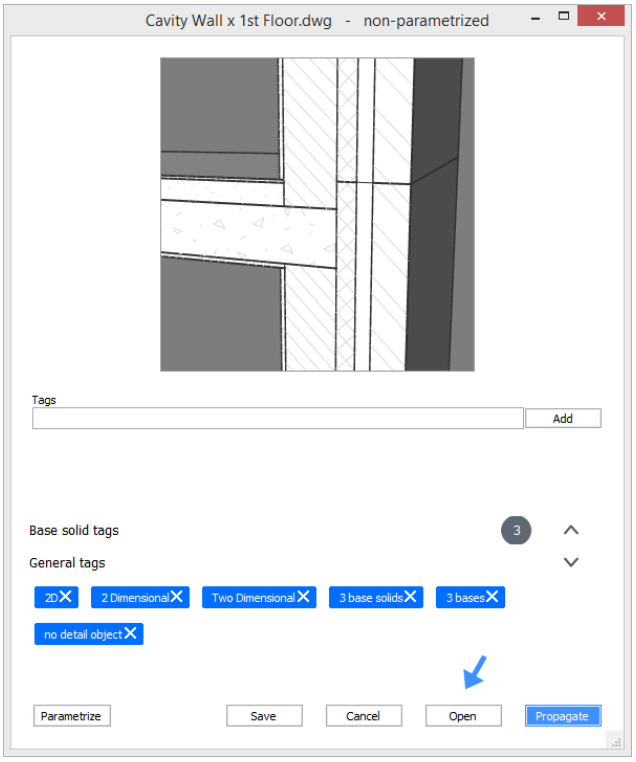
### 20.5.8.2 About BIMPARAMETRIZEDETAIL

Use the BIMPARAMETRIZEDETAIL command to automatically generate parameters for a saved detail. Standard parameters such as angles, offsets in different directions and tolerance parameters are created. These parameters mean you can propagate details to target situations similar to the example below.

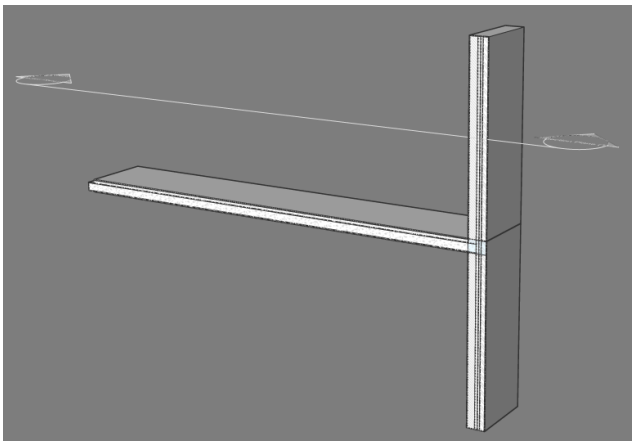
The automatically generated parameters form a framework that can control the position and shape of the reference solids. You can extend this framework adding custom constraints. You can evaluate the effect of these changes by animating the parameters from the **Mechanical Browser**. You can add custom constraints to link detail objects to the reference solids and to each other. You can also use the framework parameters in the expression of those constraints.

### 20.5.8.3 Parametrize and edit a wall-slab connection detail

1 To open the .dwg file of the detail you want to parametrize click on the detail in the **Details** panel. The **Detail** dialog box displays:




2 Click **Parametrize**, or click **Open** and run BIMPARAMETRIZEDETAIL. The detail .dwg file will open and the detail will be parametrized.



3 Open the **Mechanical Browser**.

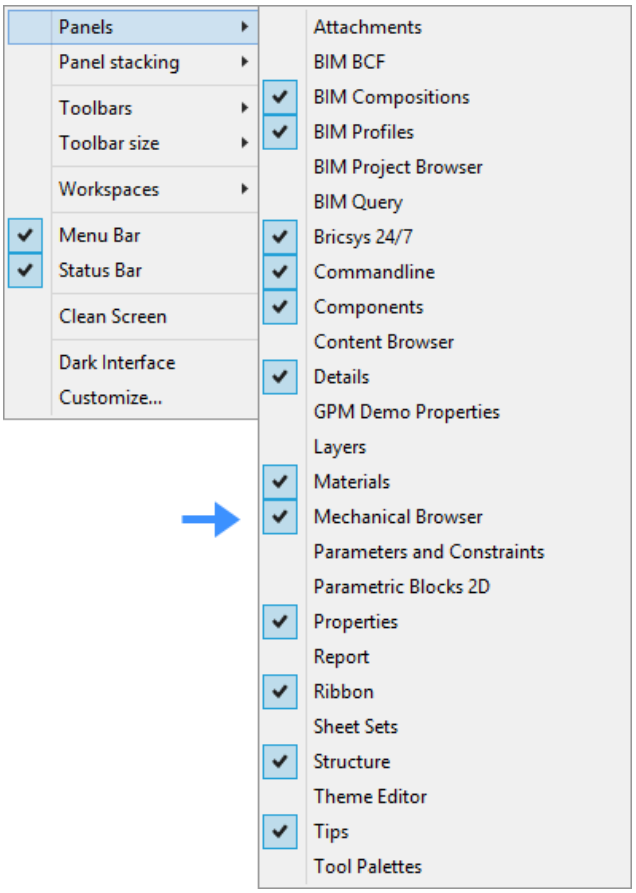
4 Click the **Mechanical Browser** button on the right of the screen.



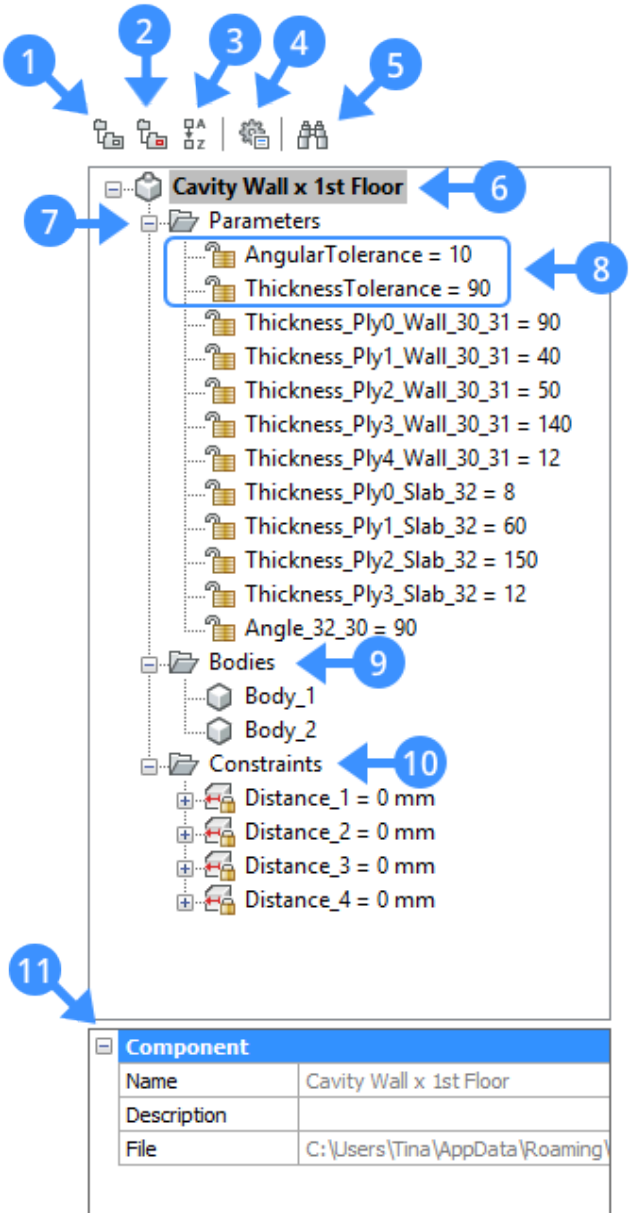
5 If the **Mechanical Browser** button isn't visible yet, move the cursor to the toolbar or ribbon panel and right-click. A context menu displays.

Expand the **Panels** option.

Select **Mechanical Browser** in the context menu.




6 Once you have clicked on the **Mechanical Browser** button, the **Mechanical Browser** displays:



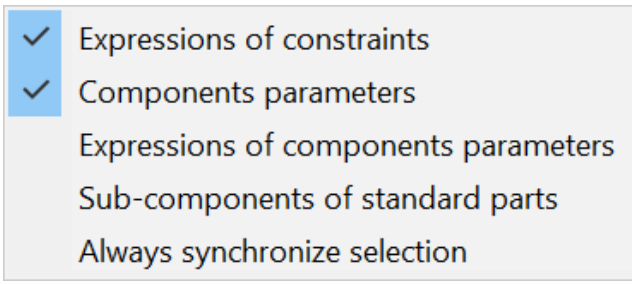

***Прим.:*** You can also use the MECHANICALBROWSEROPEN command.

(1) **Group by entity**: Group the parameters and constrains by entity.

(2) **Group by type**: Group the parameters and constrains by type.

(3) **Sort**: Sort alphabetically.

(4) **Settings**: An options menu will display:

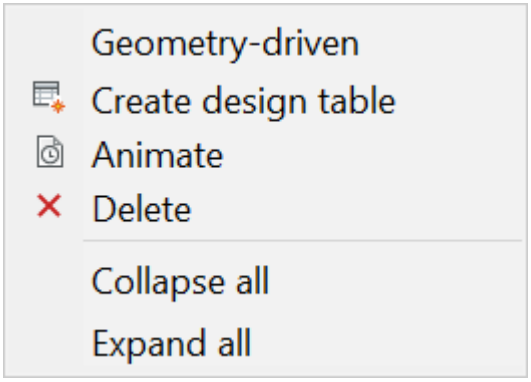



- **Expressions of constraints**:Controls whether the numeric value or the assigned parameter name displays.
- **Components parameters**:Expressions at sub-components control the visual representation of sub-component parameters: numeric value or assigned parameter name.
- **Expressions of components parameters**:Sub-component parameters control the visibility of sub-component parameters.
- **Sub-components of standard parts**:Parameters at properties add a parameters section for the selected instance to the **Mechanical Browser** properties.
- **Always synchronize selection**:When checked,theselectednodes in the browser mirror the selection in the document.

(5) **Show search** (Ctrl+F): Show or hides the search bar.

(6) The name of the parametrized detail.

(7) **Parameters**: Right-click on a parameter to open a context menu:




- **Geometry-driven**:If checked, it makes the parametergeometry-driven.
- **Create design table**: Creates a design table to drive parametric block parameters.
- **Animate**:Animates models by means of parameters.
- **Delete**: Deletes the selected parameter.
- **Collapse all**: Collapse the whole **Mechanical Browser**.
- **Expand all**: Expand the whole **Mechanical Browser**.

(8) **Tolerance Parameters**: The maximum and minimum variation of an angle, thickness or dimension from the original parameter value.

(9) **Bodies**: Display the bodies in the model.

(10) **Constraints**: The constraints you can delete, modify or add.

(11) **Component**: The properties of the detail file.

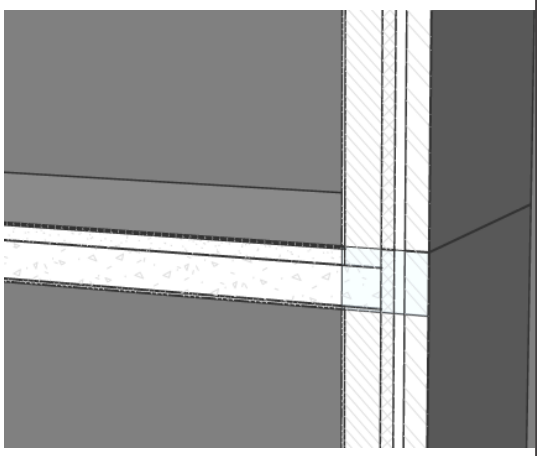
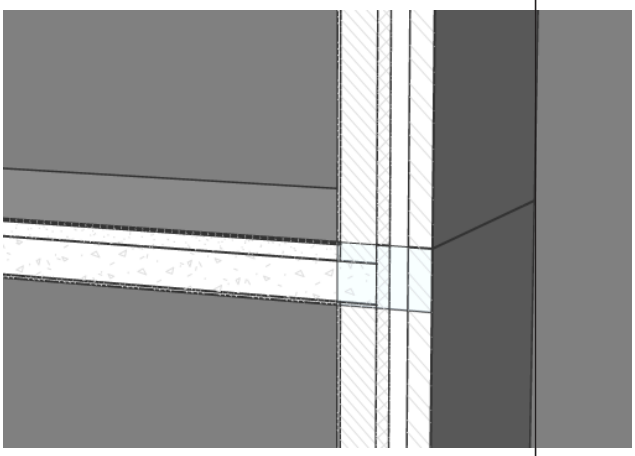
7 ***Прим.:*** Some default constraints are generated that link the detail volume and the detail objects to the reference solids. You can delete and replace them with other constraints.

8 Edit the parameters.

9 The parameters that were generated have specific values to ensure the detail looks exactly as it was created. If you change these values, the detail will change.

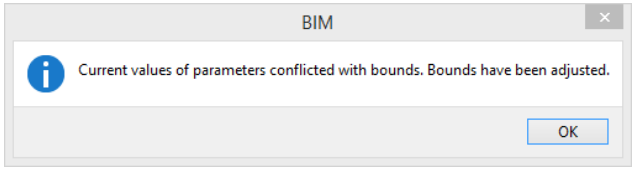
**For example**:

- Change the **Thickness_Ply1_Wall_30_31** parameter from 40 to 70 (in the **Expression** field). The new detail is visibly different in the model space:

| Before | After |
| --- | --- |
|  |  |

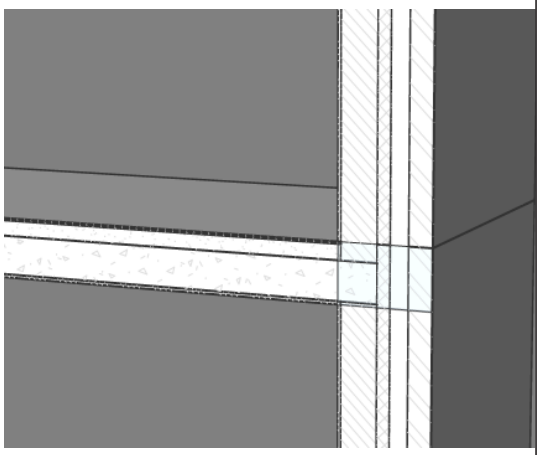
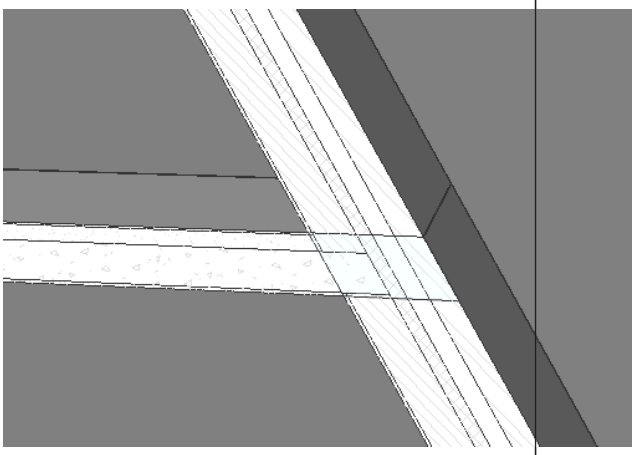
- Change the **Angle_32_30** parameter from 90 to 120. The new angle between the wall and the slab is different in the model space.

  A warning pops up:



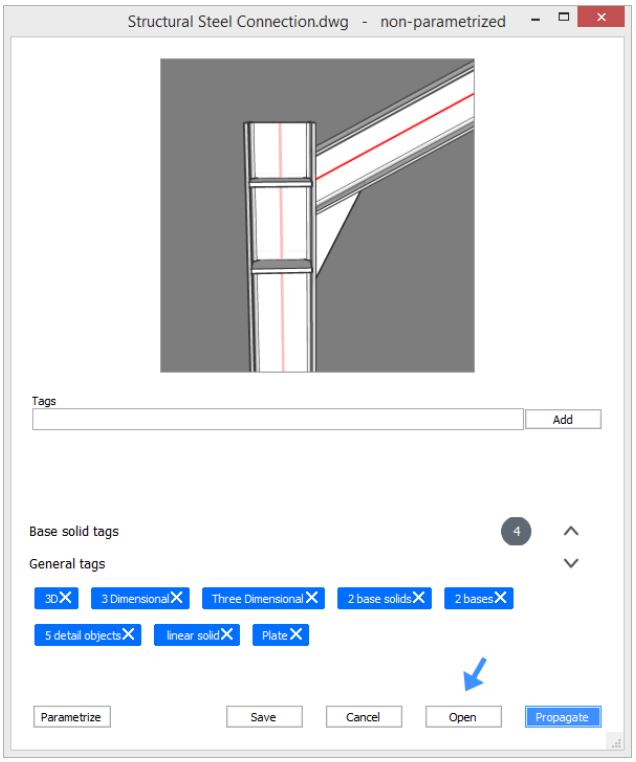

  This is because the **AngularTolerance** parameter was set to 10. The new **AngularTolerance** parameter automatically changes to 30 because you changed the **Angle_30_31** parameter by 30 degrees.

- Click **OK**

| Before | After |
| --- | --- |
|  |  |

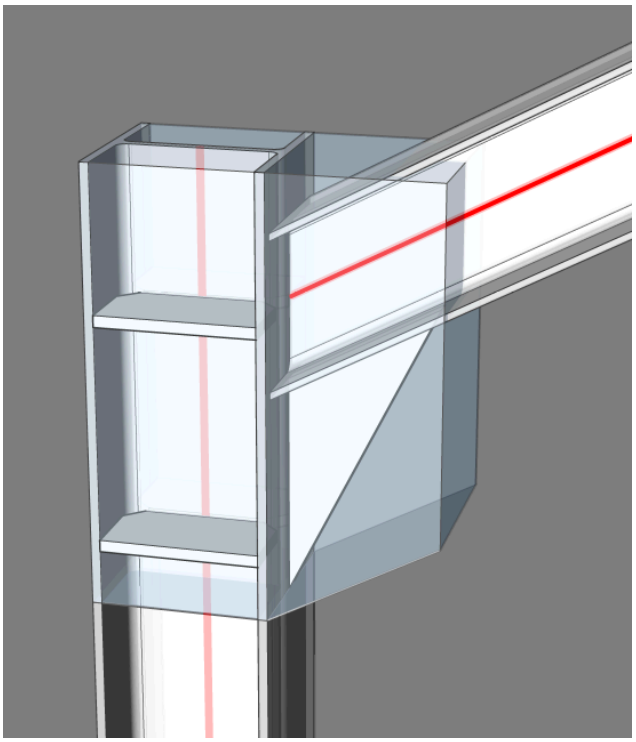
### 20.5.8.4 Parametrize and edit a structural steel connection

1 To open the .dwg file of the detail you want to parametrize, click on the detail in the **Details** panel. The **Detail** dialog box displays:

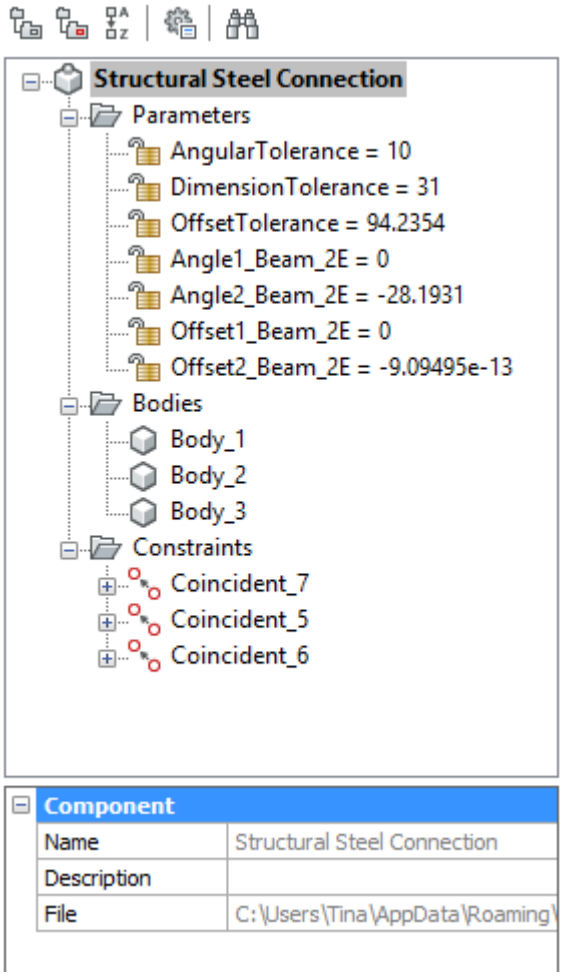



2 Click **Parametrize**, or click **Open** and run BIMPARAMETRIZEDETAIL.

The detail .dwg file will open and the detail will be parametrized.



3 Open the **Mechanical Browser**:




***Прим.:*** For connections between linear reference solids, only the framework parameters are automatically generated. You must manually add constraints to specify the behavior of detail objects in relation to the reference solids. In this example, three coincidence constraints have been manually added: two to make sure that the connection planes of the triangular steel plate maintain coincident with the flanges of the steel column and girder, and one to make sure that the end section of the girder maintains coincident with the column flange. You can add coincidence constraints before or after the parametrize tool. If you add the coincidence constraints after you can see how the reference solids change with new parameters. This makes it clear which constraints have to be added to make all detail objects move correctly.

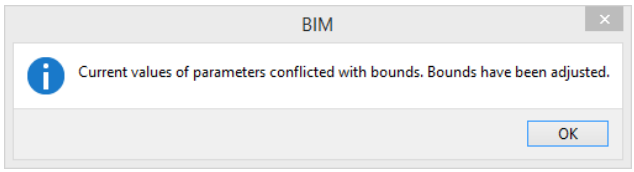
4 Edit parameters.

5 The parameters that were generated have specific values that make sure the detail looks exactly as it was created. If you change these values, the detail will change.

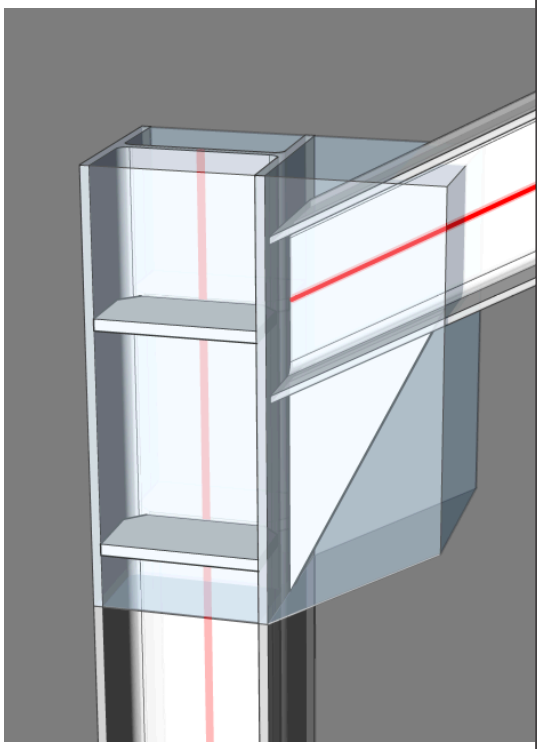
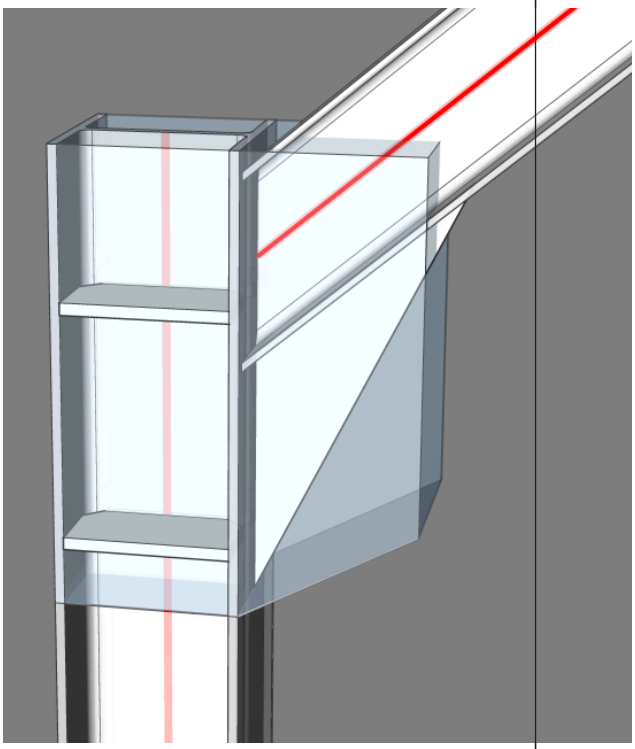
**For example**:

- Change the **Angle2_Beam_2E** from -28.1931013282 to -40. The column and the girder change in the model space.

  A warning pops up:

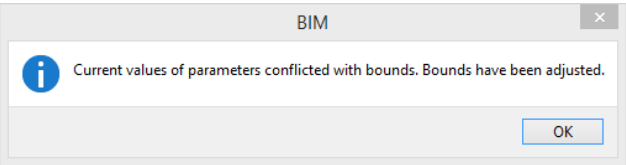



This is because the **AngularTolerance** parameter was set to 10. The **AngularTolerance** parameter automatically changed to 11.8069 because you have changed the **Angle2_Beam_2E** by 11.8069 degrees. Click **OK**.

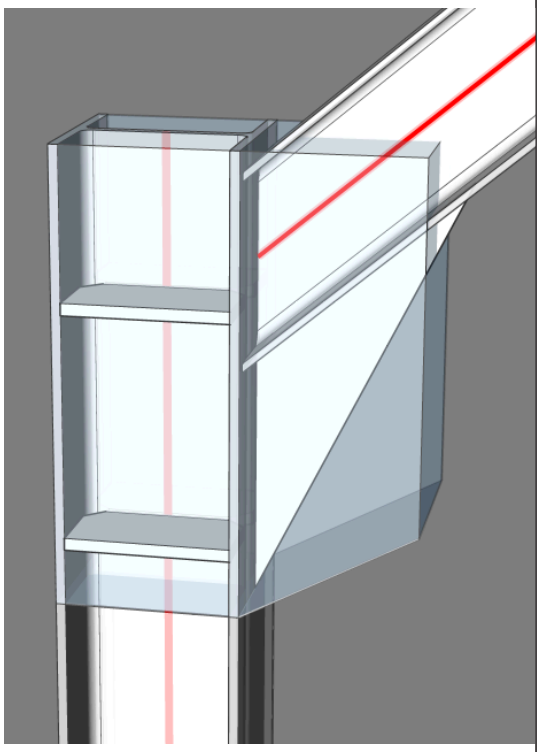
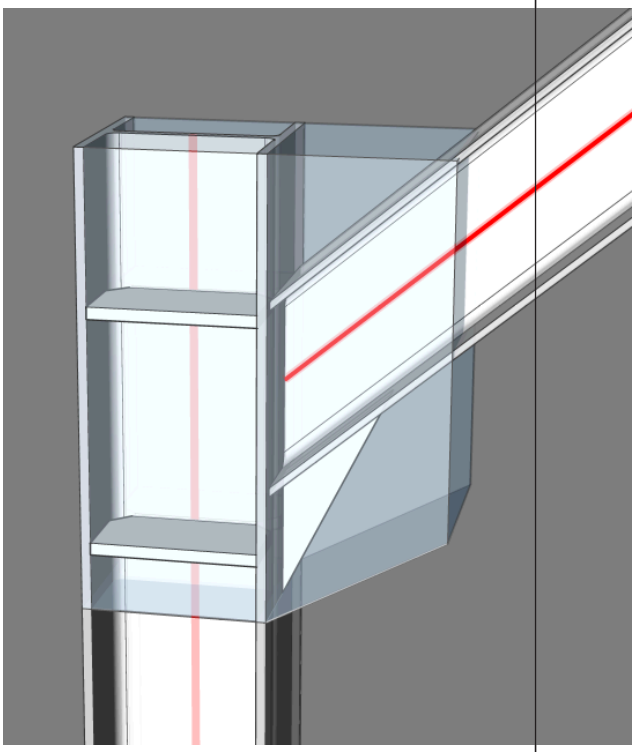
| Before | After |
| --- | --- |
|  |  |

- Change the **Offset1_Beam_2E** 0 to 200. You see the girder moves 200 mm downwards.

  A warning pops up:




This is because the **OffsetTolerance** parameter was set to 31. The **OffsetTolerance** parameter automatically changed to 200 because you changed the **Offset1_Beam_2E** parameter by 200 mm. Click **OK**.

| Before | After |
| --- | --- |
|  |  |

### 20.5.8.5 Propagate a detail in a project

There are different methods to propagate a detail:

- With the **Details** Panel: propagate a detail from the project on all similar connections in the project.
- With the BIMPROPAGATEFROMFILE command: propagate a detail from a file.

Fewer suggestions are given by the BIMPROPAGATE command for a detail that is not parametrized. This is because a detail that is not parametrized has fewer possible locations.

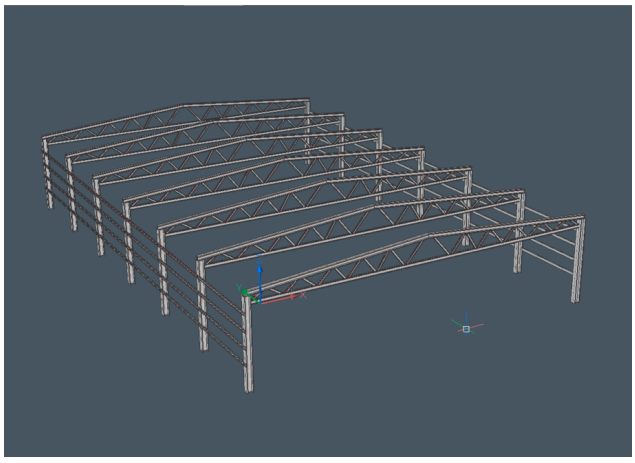
For example: if a non-parametrized detail of a wall-slab connection is propagated, BricsCAD BIM won't be able to propagate this detail to a wall-slab connection when the composition is different or a particular ply in the wall differs 10 mm from the corresponding ply in the wall of the original detail. To solve this problem parametrize the detail.

The BIMPROPAGATE command can adapt a detail with parameters to situations that differ from the original (eg: another corner, a smaller profile, etc.). You can decide how much the connections can differ from the original with the tolerance parameters.

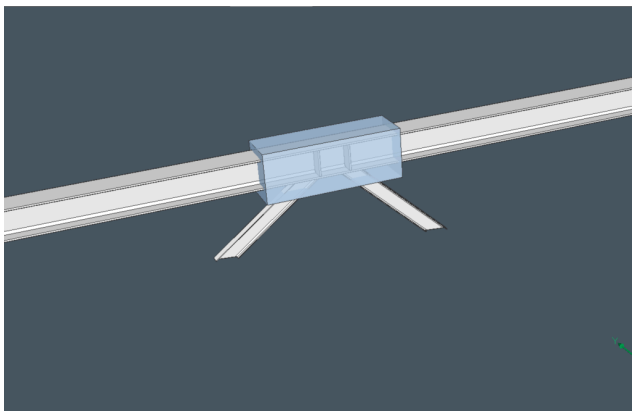
Different tolerance parameters:

- **Angular**: Sets the tolerance on angle parameters of the detail in degrees. Angles in the target situation cannot deviate more than this tolerance value from the original angles in the detail.
- **Dimension**: Sets the tolerance for the profile dimensions of linear reference solids in the detail. Profile widths and heights in the target situation cannot deviate more than this tolerance from the original widths and heights in the detail.
- **Offset**: Sets the tolerance for offset parameters of the detail. Offsets in the target situation cannot deviate more than this tolerance from the original offsets in the detail.
- **Thickness**: Sets the tolerance for the total thickness of planar reference solids in the detail. Thicknesses in the target situation cannot deviate more than this tolerance from the original thicknesses in the detail.

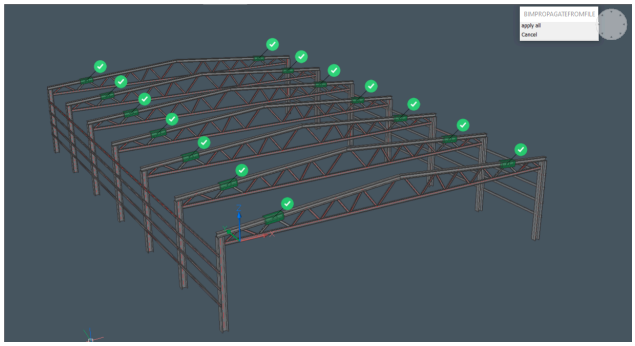
In the following project, there are a lot of similar connections present. However, the angles of the connections vary and the lower beam in the truss and has a different profile size than the upper beam.



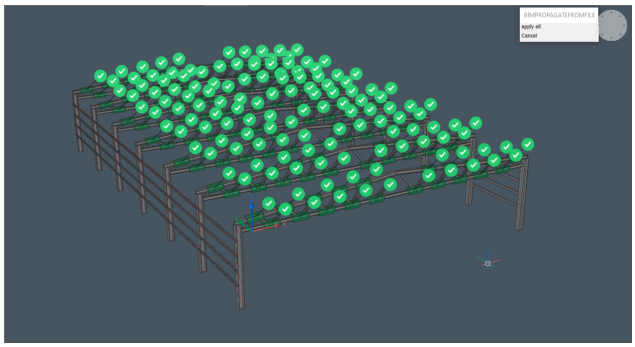
Propagate a detail in the project:

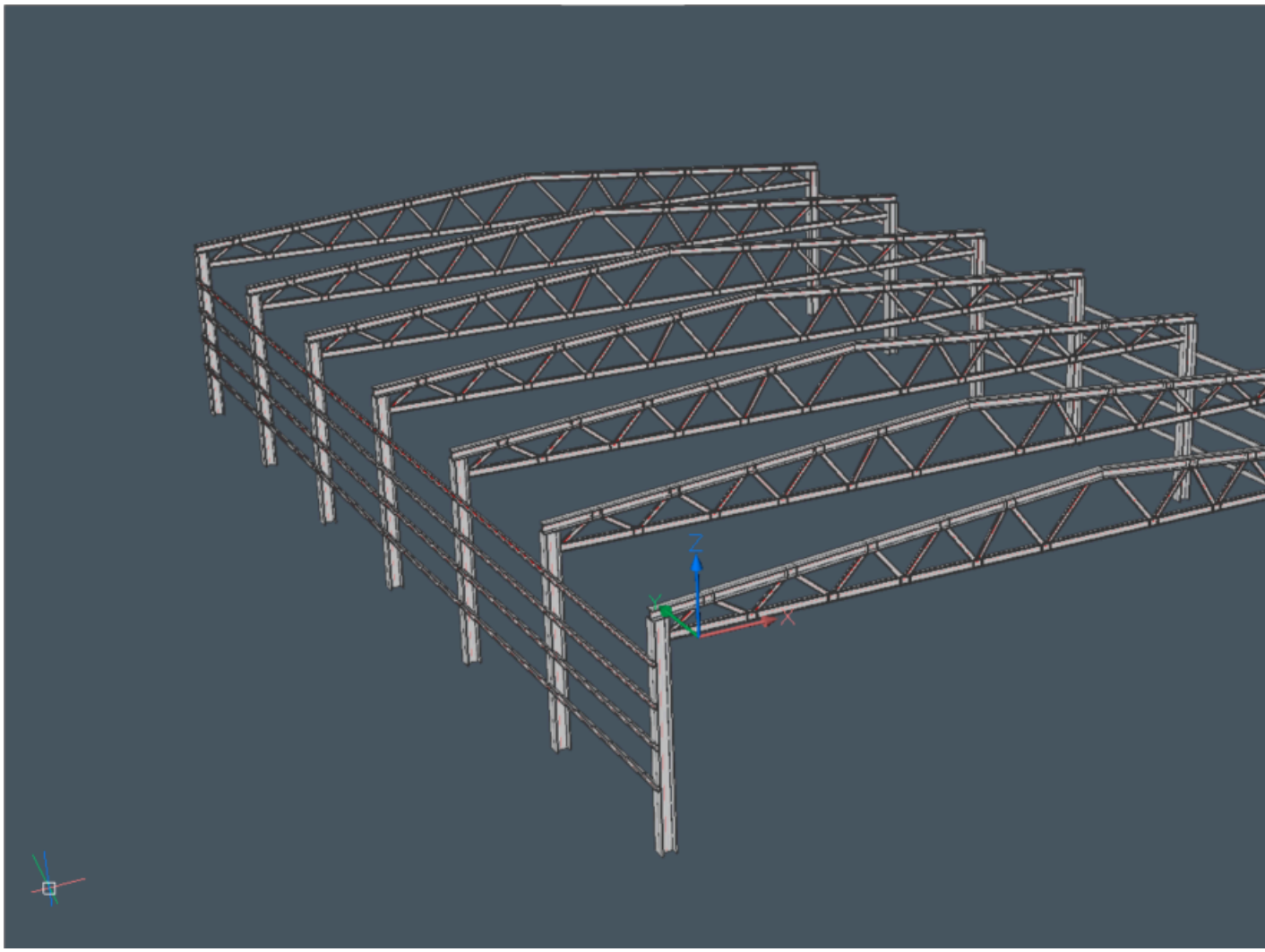


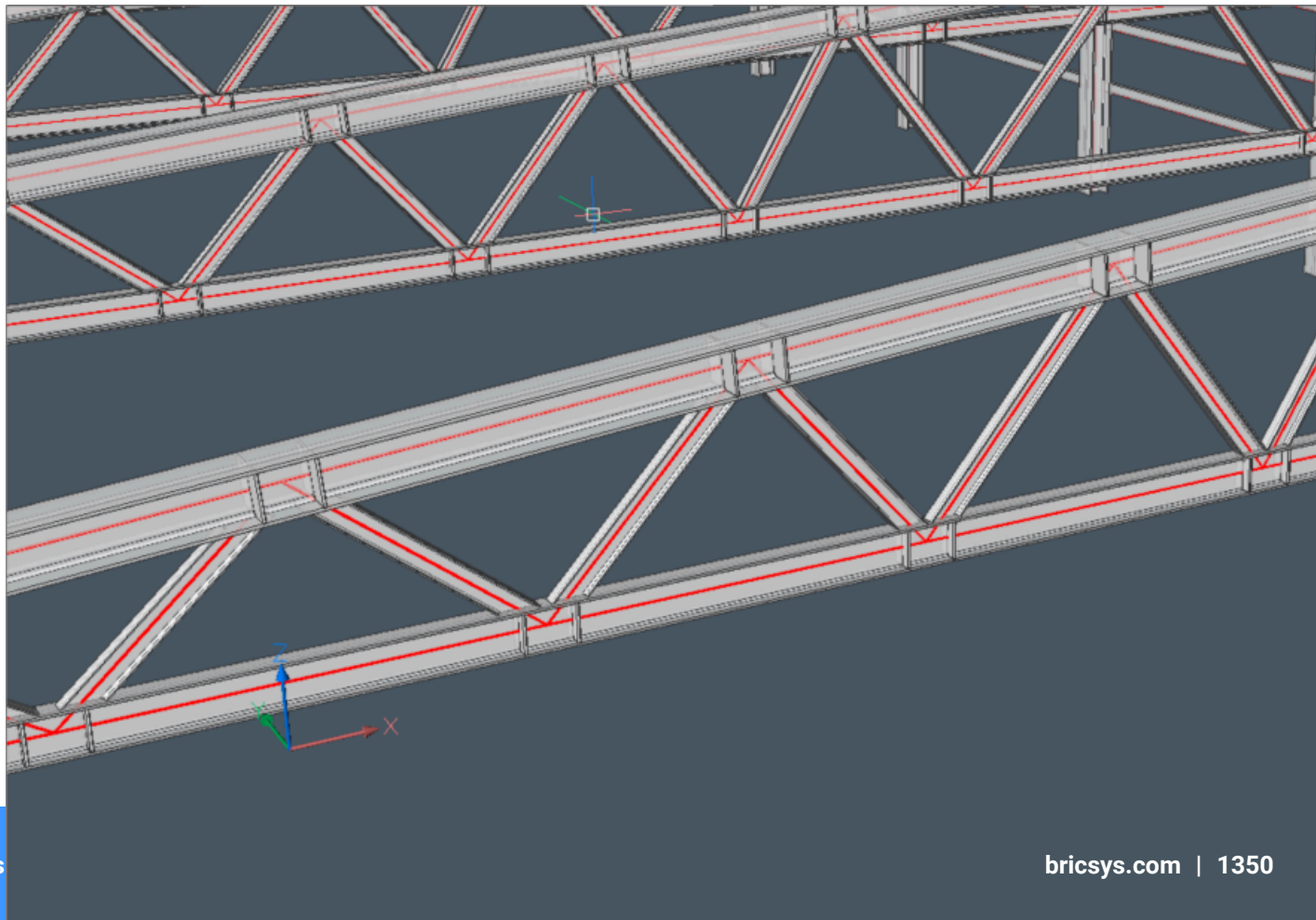
The **Mechanical Browser** displays the parameters. The **AngularTolerance** parameter is set to 0 degrees, the detail will only propagate to connections with exactly the same angle.



Change the **AngularTolerance** parameter to 45 degrees. The detail will propagate to all the similar connections. This is because none of the connections exceed the tolerance value.







### 20.5.9 Details panel

#### 20.5.9.1 Commands

DETAILSPANELOPEN, DETAILSPANELCLOSE

#### 20.5.9.2 Opening the details panel

Execute the DETAILSPANELOPEN command.

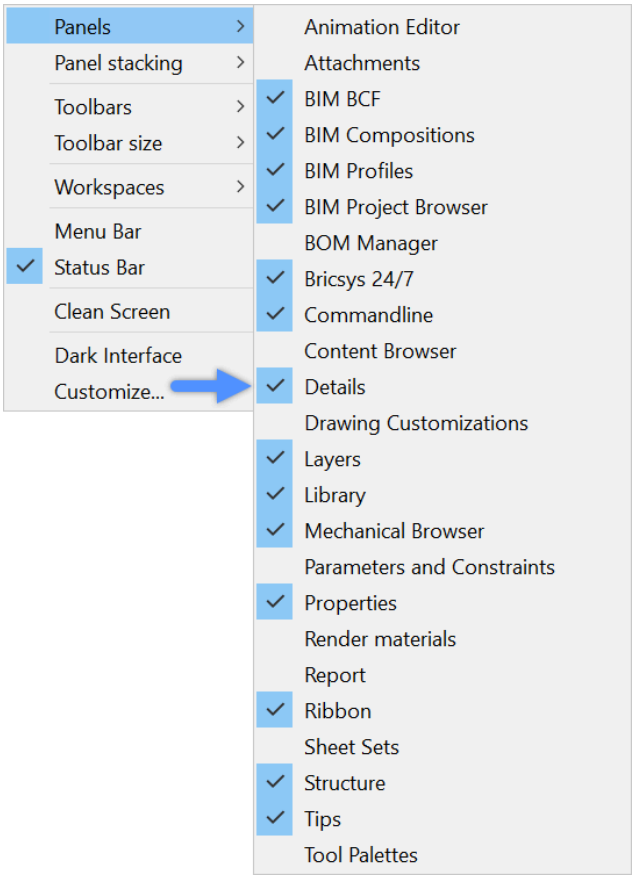
or

1 Place the cursor on a toolbar or ribbon panel and right-click.

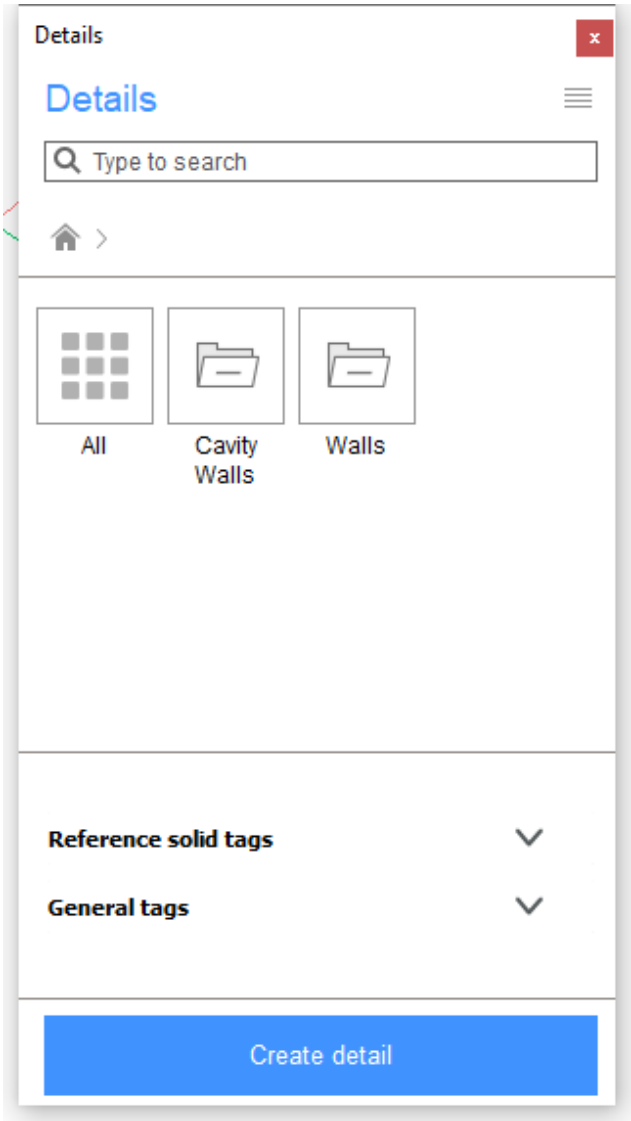
1 A context menu displays.

2 Expand the **Panels** option.

3 Select **Details** from the context menu.




The Details panel displays and the **Details** button ( ) is added to the collapsible panelset at the right-hand side of the BricsCAD application window.



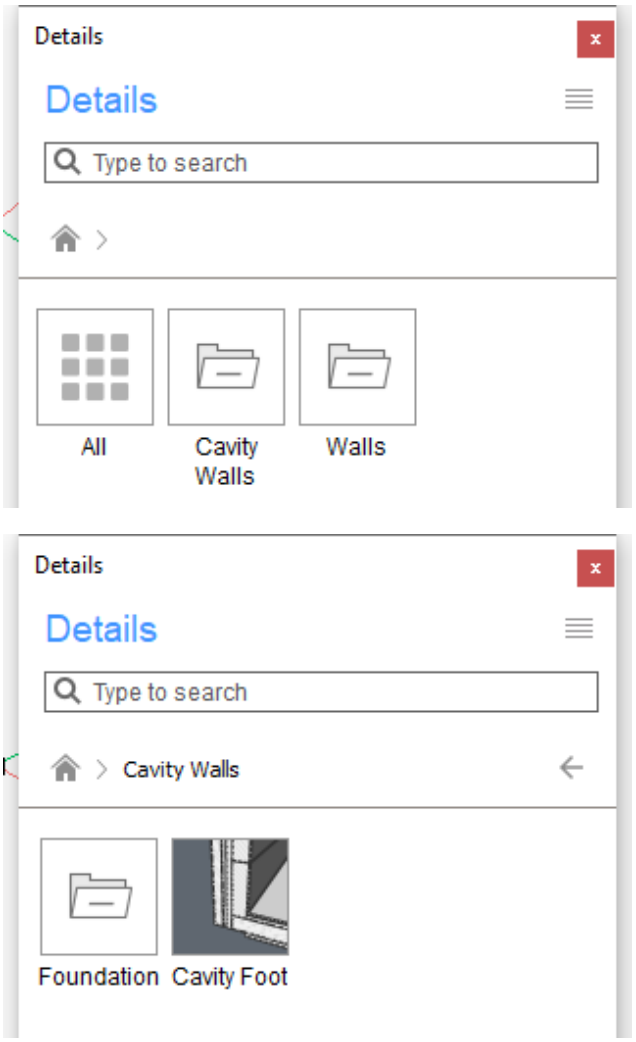

Click on the **Details** button ( ) to expand/collapse the **Details** panel.

### 20.5.9.3 Saving details to the details panel

After you created a new detail, it's possible to save it to the **Details** Panel. This way you can use the same detail in other files as well. Use the command BIMCREATEDETAIL.

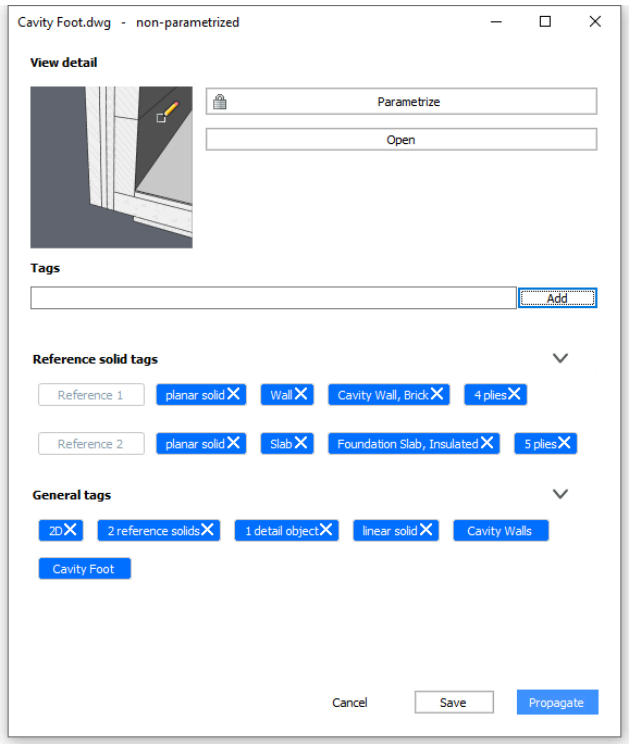
### 20.5.9.4 View a detail

1 Click on the category of the detail you want to view. The details assigned to that category display.




2 Click on the detail you want to view.

3 The **Detail** dialog box displays:




- Edit the thumbnail image:
  - Click the **Pencil** icon (✎).

    The detail dwg opens.

    The CREATETHUMBNAIL command is launched.
  - Use view commands (zoom, pan, rotate) to adjust the view.
  - Choose **OK** or press Enter to save the thumbnail.
- **Parametrize**: Opens the .dwg detail file and executes the BIMPARAMETRIZEDETAIL command.
- **Open**: Opens the .dwg file of the detail.
- **Tags**: Add general tags.
- **Reference solid tags**: Click the down arrow to display the tags. You cannot add reference solid tags; you can only delete the existing tags.
- **General tags**: Click the down arrow to display the tags.
- **Save**: Saves the changes and closes the **Detail** dialog box.
- **Propagate**: Propagates the detail. See the BIMPROPAGATE command.
- **Cancel**: Closes the detail dialog box without saving.

Tags can be added to each detail. These are used to search for a specific detail in the library. There are two different tag categories:

- **Reference solid tags**: Tags that identify an individual reference solid from the detail.
- **General tags**: Tags that generally identify the detail.

During the BIMCREATEDETAIL command, tags are automatically generated for the detail. You can delete unrequired tags and add new tags. New tags can only be **General tags**.

You can add tags during the BIMCREATEDETAIL command or at any moment after the detail has already been created.

To add tags during the BIMCREATEDETAIL refer to the BIMCREATEDETAIL article. To add tags at another moment, see below.

**Managing tags**

1 In the **Details** panel, click on the detail you want to adjust.

The **Detail** dialog box displays.

2 Expand the **Reference solid tags** category.

You can only delete these tags, you cannot add new tags.

3 Click in the **Tags** field and type the name for the new tag, then click the **Add** button to add **General tags**.

The new tag is added to the **General tags** category in an orange box.

Repeat this procedure to add more tags.

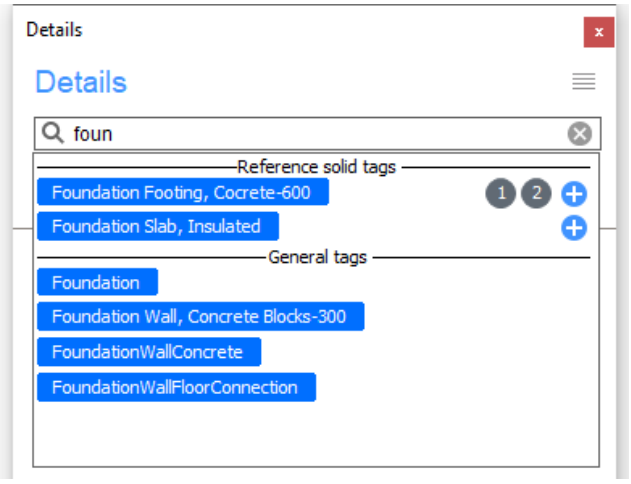
4 Click the **Save** button to save the changes and close the **Detail** dialog box.

**Searching details with tags**

In the **Details** panel you can search the library for specific details using tags. You can add one or more tags at once to your search query to select all matching details.

1 Click the search bar in the **Details** panel and type in a search string. The matching tags display:

**Reference solid tags** and the **General tags**.



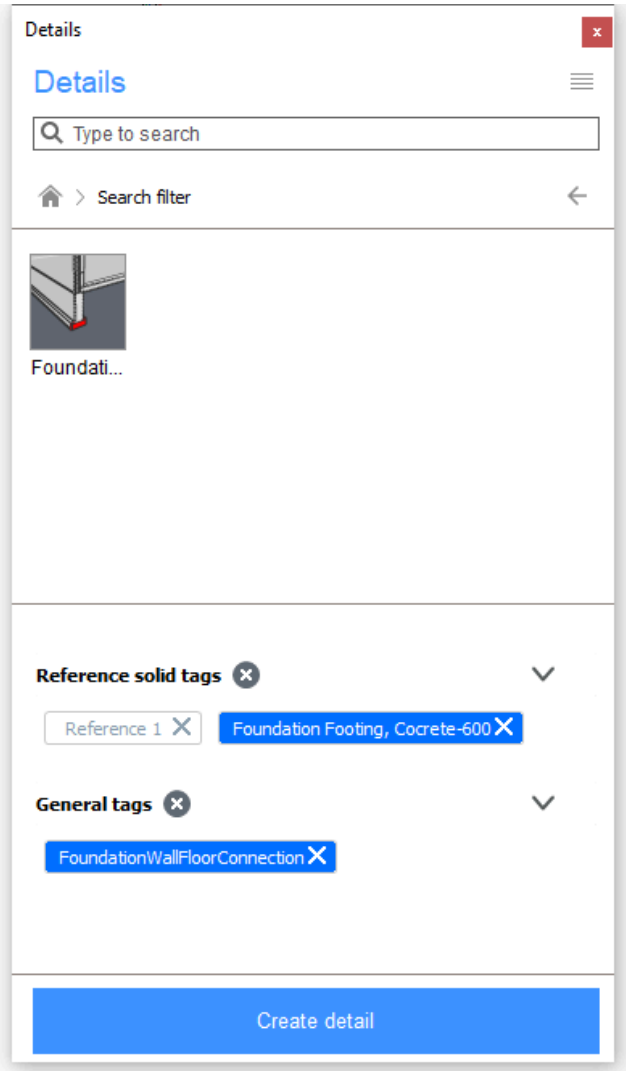

2 Do the following:

- **General tags**: Click the desired tag.
- **Reference solid tags**:
  - To add the tag to a reference solid that is not yet in your search query: click the plus sign next

to the required tag.

- Add the tag to a reference solid that is already in your search query: click the number of the required reference solids next to the tag.
- Once these tags have been added to your search query, they will appear at the bottom of the **Details** panel under the right tag category.

The filtered search results display.




### 20.5.9.5 Editing a detail

1 Select the detail you want to edit.

2 Click the **Open** button.

The .dwg file of the detail opens in a separate drawing tab.

3 Make the necessary adjustments to the detail.

4 Save the detail .dwg file.

5 The detail is updated in the **Details** panel.

6 The existing details won't be updated automatically. To replace the existing details by the new one, follow the steps below to propagate a detail.

### 20.5.9.6 To propagate a detail

You can propagate a detail to all of the similar connections in your project.

Do one of the following:

Drag and drop the detail from the library into the drawing.

or:

1 Select the detail in the library.

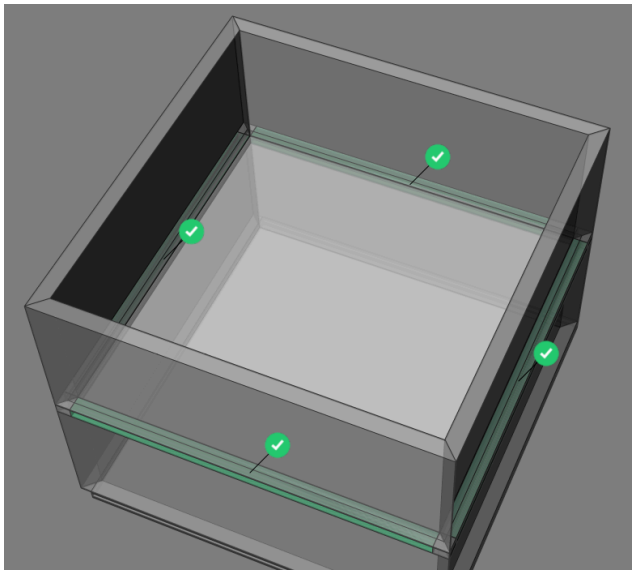
1 The **Detail** dialog box dispays.

2 Click the **Propagate** button.

To apply all suggestions, hit Enter.

To reject a suggestion click on the checkmark. A red cross icon replaces the checkmark. Click the icon again to accept the suggestion.

Press Enter to accept.



You can also propagate a detail in your project using the BIMPROPAGATEFROMFILE command. This command allows you to propagate detail files that are not in the library.

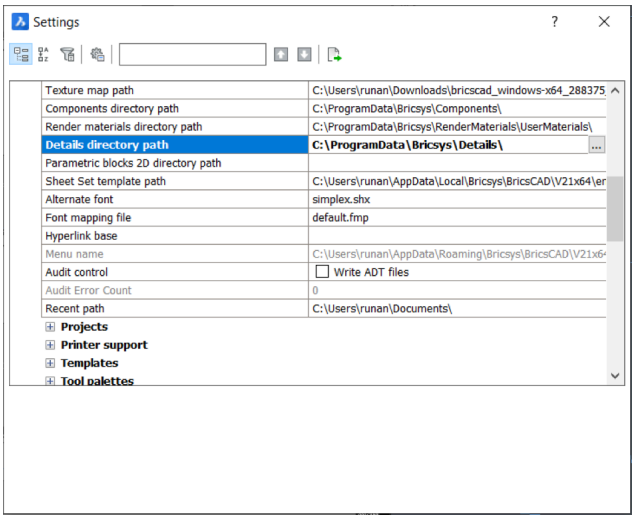
### 20.5.9.7 Sharing details

You can save details as a dwg-file and share the details folder/library. Save the detail drawings at the correct location which will then display in the details panel.

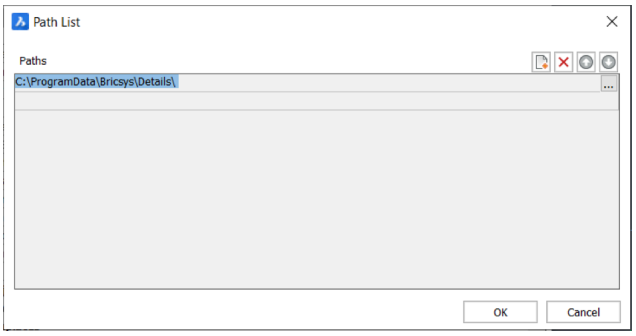
1 If you want to make a new detail to share, you can create a new detail first, as is explained above;

1 The following steps apply both to new details as well as existing details.

2 All the details you have created are automatically saved in the folder, and its subfolders, which is set through the DETAILSPATH system variable. By default, this is **C:\ProgramData\Bricsys\Details**.




Then click on the three dots next to the existing name in the **Path List** to change the location.

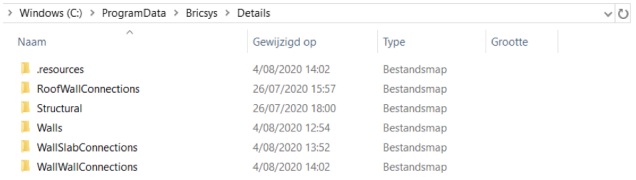



You can also create additional paths.

Click the **New** icon (), then click on the three dots of the new row to add a path for the **Details** library.

3 If you want to share a folder, go to the file explorer on your computer and follow the path to the details folder. If you haven't created any details yet, this folder only contains the folder '.resources'.

4 If you have created some details, you can find the same folders here as in your **Details** panel.



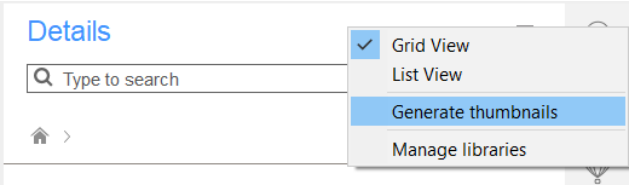

5 Copy the folder that you want to share and turn it into a zip file. This zip file can be shared with others.

6 If you receive a zip file you should unpack it and copy the folders containing the details to the folder *C:\ProgramData\Bricsys\Details*.

***Прим.:*** Make sure the details are saved in a folder that you place in the **Details** folder. If you just place the dwg files in your **Details** folder, they will not appear in your **Details** panel.

7 To load the new details in your library panel, go to the **Details** panel, open the menu and click **Generate thumbnails**.

The new details folders will appear in your **Details** panel.




### 20.5.10 Rhino/Grasshopper integration

#### 20.5.10.1 Commands

RHINO, GRASSHOPPER, TOGRASSHOPPER

#### 20.5.10.2 About

Rhinoceros3D is a 3D modeler developed by Robert McNeel & Associates of Seattle, Washington USA. Rhino is used to create, edit, analyze, document, render, animate and translate NURBS (curves, surfaces, solids, point clouds and polygon meshes).

Grasshopper is a visual programming language environment that runs within the Rhinoceros 3D computer-aided design application. It is tightly integrated with Rhino's 3D modeling tools.

#### 20.5.10.3 Installing Rhino/Grasshopper connection

1 Download Rhino 7 from https://www.rhino3d.com/download.

***Прим.:*** Since the releases of BricsCAD V21 and Rhino 7, you can use the Rhino/Grasshopper Connection within BricsCAD with trial license keys of both BricsCAD BIM V21 and Rhino 7. Keys for older versions of Rhino will not work for this integration.

2 Download the Grasshopper-BricsCAD Connection from the BricsCAD Application Store: https://www.bricsys.com/applications/a/?rhino-grasshopper-connection-for-bricscad-a1353-al2360. The installer copies all required files to the 'Program Files\Bricsys\Grasshopper-BricsCAD Connection' folder.

***Прим.:*** The Grasshopper-BricsCAD Connection currently available from the Application Store will only work within BricsCAD BIM V21, not within older versions.

3 Restart BricsCAD V21.

4 To launch Grasshopper or Rhino, open and save a new drawing. Then use the buttons in the new tab that is displayed in the ribbon or launch with the Quad.

**To launch a Grasshopper file from BricsCAD**

1 Open an empty BricsCAD file.

2 Save it.

3 Open Grasshopper with the button in the new **Grasshopper** tab inside BricsCAD.

4 You can close the Rhino pop-up window. Rhino will continue to work in the background.

5 In the **File** tab of Grasshopper, choose **New document** to open a new and empty Grasshopper script.

6 Choose **Open document...** to open an existing Grasshopper script. You can use any script you made with Grasshopper in the past and open it in BricsCAD, as long as the input geometry from another program has been internalized. (To internalize data right-click the input geometry component and hit **Internalise data**.)

7 The geometry you created with the script should now pop-up in your BricsCAD model space, displayed in red. Make sure you don't close the **Grasshopper** window, otherwise the preview geometry will disappear, as the Grasshopper document is closed.

8 If your Grasshopper script is open and you still don't see the preview geometry, it probably means your Grasshopper file is linked to the wrong BricsCAD document. To link it to the correct one, see 'To link a Grasshopper file with a BricsCAD file'.

**To link a Grasshopper file with a BricsCAD file**

By default, the BricsCAD file that the Grasshopper document will link to is the one that was active when you launched Grasshopper from BricsCAD. If you want to link it to another BricsCAD file, you need to:

1 Make the target BricsCAD drawing the current drawing (i.e. the one you see in the model space window of your current session).

2 Link the Grasshopper script to this open drawing. You do that by clicking the Link icon  in the bar just above the Grasshopper canvas that contains the script that you are trying to link.

**To launch a sample file**

The sample files are provided with the Grasshopper-BricsCAD Connection. They can show you what is possible with the Connection. They are stored in the 'Program Files\Bricsys\Grasshopper-BricsCAD Connection' folder.

1 Open up a sample .dwg file by opening a .dwg from the 'Program Files\Bricsys\Grasshopper-BricsCAD Connection' folder.

2 You will get a warning that the file is read-only. This happens because the document is located in a secured folder. Click **Yes** to open as read-only.

**Optional**: If you don't want this warning, or if you want to edit the file, copy-paste the sample files into one of your own folders and open them from there instead.

3 From inside Grasshopper, choose **File** > **Open Document**. Go to the 'Program Files\Bricsys \Grasshopper-BricsCAD Connection' folder again.

4 Choose the .ghx-file with the same name as the .dwg file.

5 The sample file now opens and you should see red preview geometry in your BricsCAD model space.

6 Make sure you don't close the **Grasshopper** window, otherwise the preview geometry will disappear, as the Grasshopper document is closed. If your Grasshopper script is open and you still don't see the preview geometry, it probably means your Grasshopper file is linked to the wrong BricsCAD document. To link it to the correct one, see 'To link a Grasshopper file with a BricsCAD file'.

7 You can play around with the input sliders in the Grasshopper script.

8 If you are happy with your model and would like to convert the preview geometry into real BricsCAD geometry, you can 'bake' the geometry into BricsCAD. You do that by selecting all the **Bake Building Element** components (Shift-clicking them all) and pressing the **Bake** button .

9 A pop-up window appears, just click **OK** to accept.

10 The geometry is now converted into real BricsCAD geometry, even with BIM data attached to it.

11 You won't be able to save your changes to your files, as they were read-only. So, discard your changes or save the files as new ones in a different location.

### 20.5.10.4 Customizing the Rhino/Grasshopper connection

The code for the Rhino/Grasshopper Connection is 100% open-source and is available at https://github.com/Bricsys/rhino.inside-bricscad. This allows you to fully customize your Connector by coding to your heart's content (assuming, of course, that you know how to code).

### 20.5.10.5 Uninstall the Rhino/Grasshopper connection

1 Launch **Add or Remove Programs** in Windows.

2 Search for **Grasshopper-BricsCAD connection**.

3 Hit **Uninstall**.

### 20.5.10.6 How to draw a box in Rhino and bake into BricsCAD via Grasshopper

1 Open a new BricsCAD file.

2 Save the file.

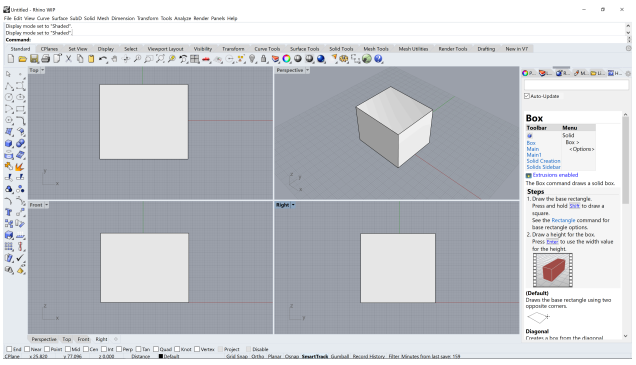
3 In BricsCAD, click on **Grasshopper** > **Rhino**.

4 Draw a Rhino box by typing BOX in the Rhino Command line. (This works in the same way as the BricsCAD BOX command).

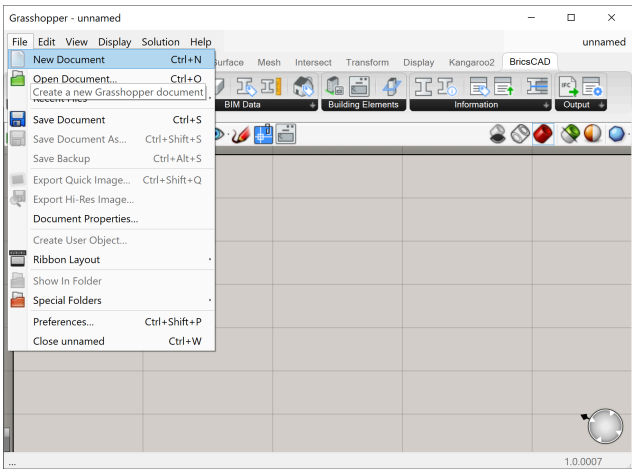
5 Click for the first corner of the base.

6 Click for the other corner of the base.

7 Click to define the height of the box.



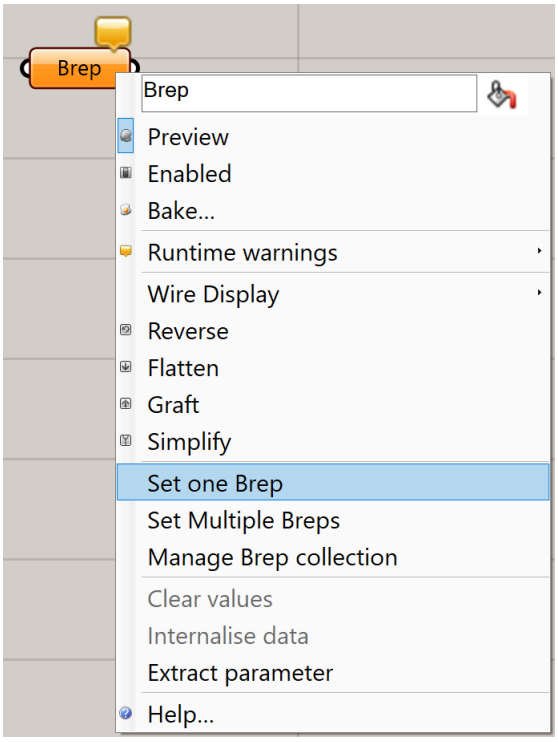

8 To open a new Grasshopper script, inside the Grasshopper environment, click on **GrasshopperGrasshopper** then click on **File** > **New Document**.




9 Save the file.

10 Drag and drop the **Params** > **Geometry** > **Brep component** onto the Grasshopper canvas.

11 Right-click on the component and select **Set one Brep**.



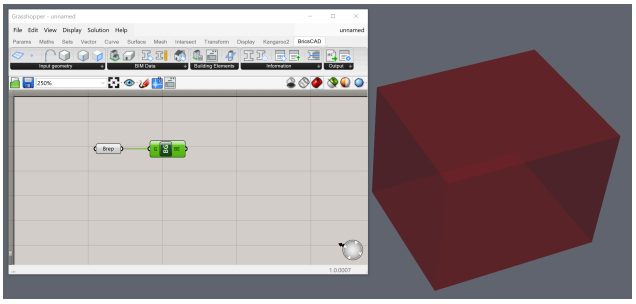

12 Go to the Rhino canvas and select your cube.

13 The Brep component will change color from orange to grey. If you close the Rhino window and return to the **Grasshopper** window, a preview of the cube will display in the BricsCAD model space.

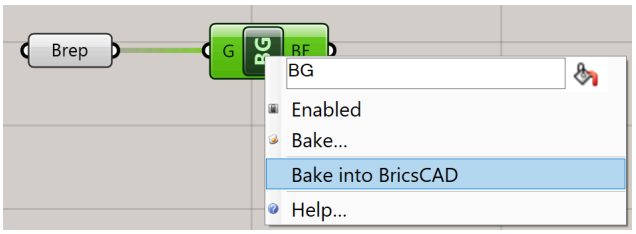
14 Drag and drop the **BricsCAD** > **Building Element** > **Bake Geometry component** onto the Grasshopper canvas.

15 To link the two components, click and drag from the Brep component's right dot to the BricsCAD Bake

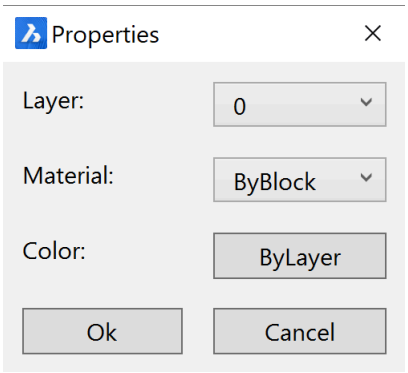
component's left dot.



16 Right-click onto the **Bake Geometry component** and choose **Bake into BricsCAD**.




The following dialog displays. Here you can choose the destination **Layer**, **Material** and **Color**:

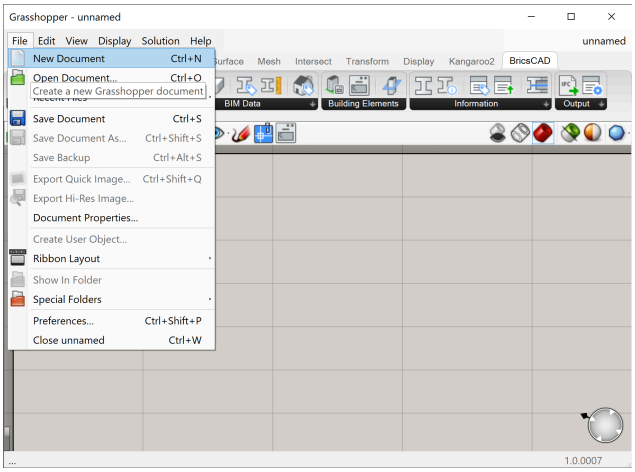



17 Click **OK**. The box is baked as a solid into BricsCAD.

### 20.5.10.7 How to draw a box in Grasshopper from a BricsCAD rectangle

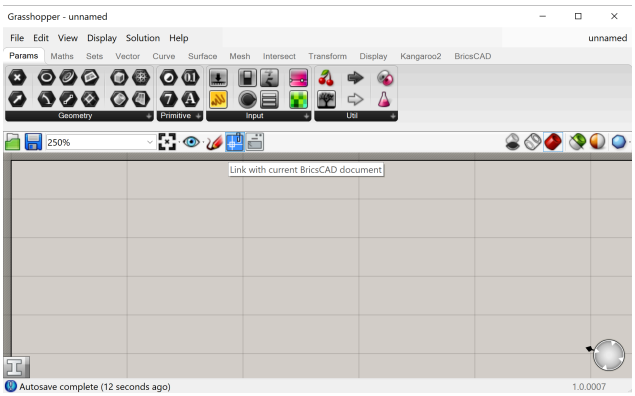
1 To open a new Grasshopper script, click on **Grasshopper** > **Grasshopper** and clicking on **File** > **New**

**Document** inside the Grasshopper environment. Save the file.




2 Open a new BricsCAD file in a **meter template** and save it.

3 To link the new script to the open .dwg file hit the **Link** button in the **Canvas** Toolbar.




4 Go to the BricsCAD model space and draw a rectangle.

***Прим.:*** Use a polyline to create irregular geometry.

5 Drag and drop the **BricsCAD** > **Input Geometry** > **Curve component** onto the Grasshopper canvas.

6 Right-click onto the **Curve component** and choose **Set One BricsCAD Curve**.

7 Go to the BricsCAD model space and select the rectangle.

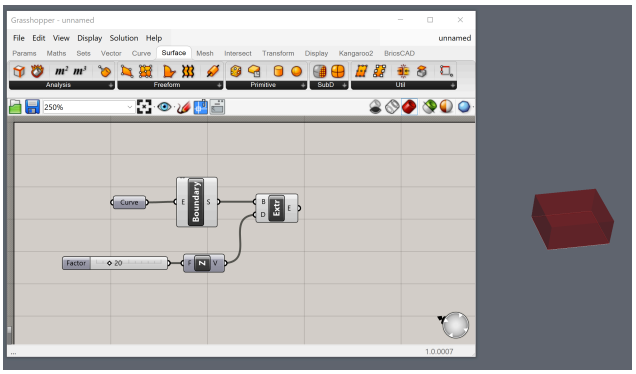
***Прим.:*** Save time with the Quad: select the curve in BricsCAD and then select **ToGrasshopper** under the **Grasshopper** tab of the Quad. A Grasshopper component appears in the upper left corner of your Grasshopper canvas.

8 Link the Curve component to a **Surface** > **Freeform** > **Boundary Surfaces component**.

9 Link this to a **Surface** > **Freeform** > **Extrude component**.

10 In the Direction input of the Extrude component link a **Vector** > **Vector** > **Unit Z component** with **Params** > **Input** > **Number Slider component**. Set this to the height of your box, 17 for example.

***Прим.:*** You can edit the range and preciseness of the slider by right-clicking the component and selecting **Edit...**. A dialog box pops up where you can set the digits and min and max value of the slider.



11 You now have drawn a box in Grasshopper, using BricsCAD geometry as an input.

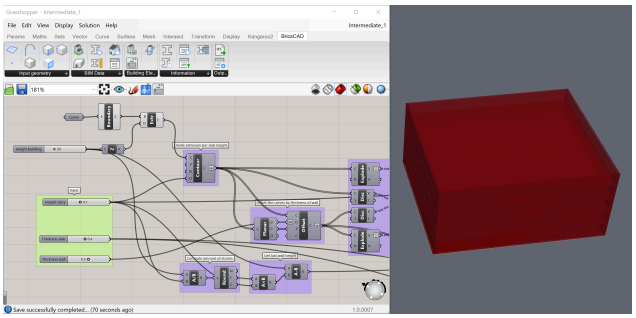
### 20.5.10.8 How to split up a box into different floors using Grasshopper

1 Add the 'Start_1.ghx'-code to your previous Grasshopper script.
   - You do that by downloading the zip file at the bottom of this page and extracting its content.
   - Then open the 'Start_1.ghx' file by going to **File** > **Open Document...** in your **Grasshopper** window.
   - You can now select all of the Grasshopper code by hitting **Ctrl+A** on the keyboard and copy it using **Ctrl+C**.
   - Then switch back to the document you were working on in the previous procedure by going to the upper right corner of the **Grasshopper** window and clicking on the title of the current document (Start_1).
   - You then get a drop-down of all the active Grasshopper scripts, so open the one you saved in 'How to draw a box in Grasshopper from a BricsCAD rectangle'.
   - Now click on the canvas and hit **Ctrl+V** to paste the 'Start_1.ghx'-code.

2 Right-click to rename the number slider from the previous procedure to **Height building**.

3 Link this number slider to input A from the Division component in the **Compute amount of stories** group and to input A from the Subtraction component in the **Get last wall height** group.

4 Link your extrusion to the Shape input of the Contour component.



5 Do some cleaning up of the file:
   - Drag the green **Input** group to the front of your canvas.
   - Add the **Height building** slider to the group by selecting the component and then right-clicking on the group and choosing **Add to group**.
   - Put off the preview for your Boundary and Extrusion components: select the components (Shift-click them) and right-click on an empty space of your canvas and choose **Preview Off**.

6 **(Optional)** For intermediate files that already have these changes, use 'Intermediate_1.ghx' and 'Intermediate_1.dwg'. (Located in the zip file at the bottom of this page.)

7 The script you just made allows you to:
   - Divide the extrusion per slab height, using the Contour component.
   - **Offset** the curves by the thickness of the walls.
   - Make the bottom surfaces of the walls (the surfaces are split into lower floor walls and roof walls).
   - Compute the last wall height.
   - **Extrude** the surfaces into roof, slabs, and walls.
   - Adapt the height of the building, the height of the stories, and the thickness of the slabs and walls, by using the sliders in the green **Input** group.

8 Add 3 **BricsCAD** > **Building Element** > **Bake Building Element components** to the end of your Grasshopper script.

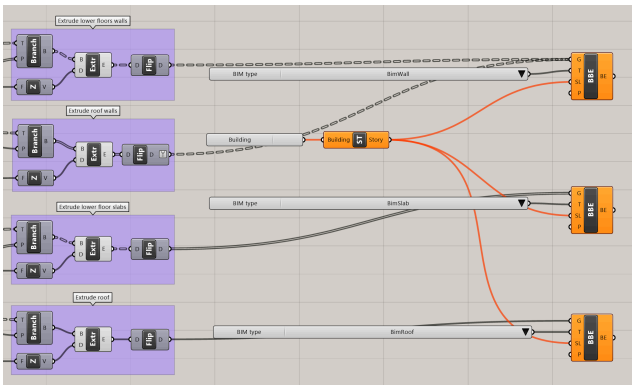
9 Attach the extruded elements (the Flip components) to the Geometry input nodes of the **Bake Building Element** components:
   - Connect the extruded lower floor walls and extruded roof walls to one **Bake Building Element** component. (Hold Shift while connecting multiple nodes to one input node).
   - Connect the extruded lower floor slabs to another **Bake Building Element** component.
   - Connect the extruded roof to the remaining **Bake Building Element** component.

10 Add the following components to your canvas from the **BricsCAD** > **BIM Data** category: 1 **Buildings**, 1 **Stories**, and 3 **BIM Types components**.

11 Attach Buildings to the Building input of the Stories component and attach this to the Spatial Location input of all the **Bake Building Element** components.

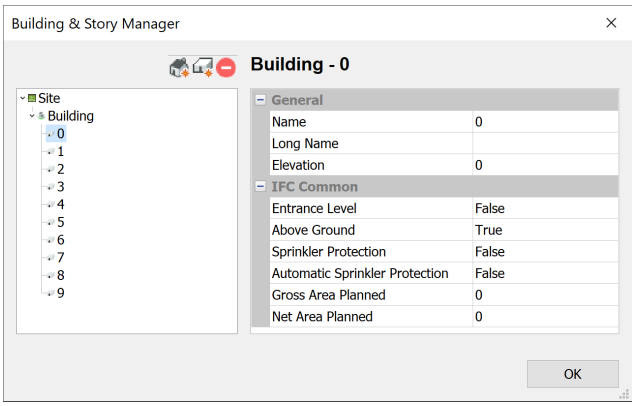
12 Attach BIM Types (set respectively to **BimWall**, **BimSlab**, and **BimRoof** in the selection menu) to the Element Type input of the **Bake Building Element** components of the extruded walls, the slabs, and the roof respectively.

13 The following warning will display: **Input parameter Building failed to collect data**. This is because, when no buildings are defined in the BricsCAD drawing, Grasshopper fails to collect input.

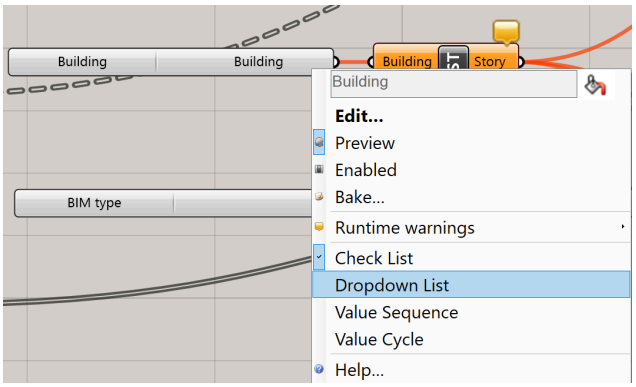


14 To make spatial locations, for the component to detect, go to BricsCAD and click on **Home** > **Classify** > **Spatial Locations**.

15 Add a building and a few stories (more stories than you want to divide your building into).




16 Go to Grasshopper and right-click the Buildings component and put it onto **Drop-down List** instead of **Check List**.

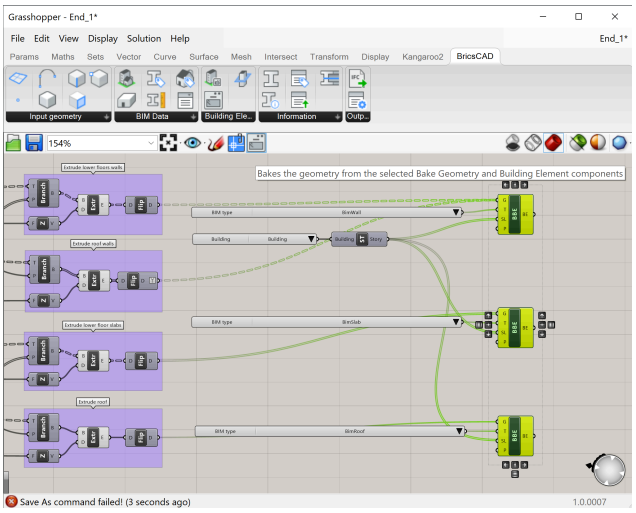



17 Select the newly made building in the drop-down list.

18 Shift-select all the **Bake Building Element** components and choose the **Bake** option from the **Canvas** Toolbar. Click **OK** to accept the **Layer**, **Material**, and **Color**.

19 The drawing will now have different floors.

***Прим.:*** Check the properties to see that the geometry is correctly classified as **Wall**, **Slab** or **Roof**. These elements will also have a spatial location assigned to them.

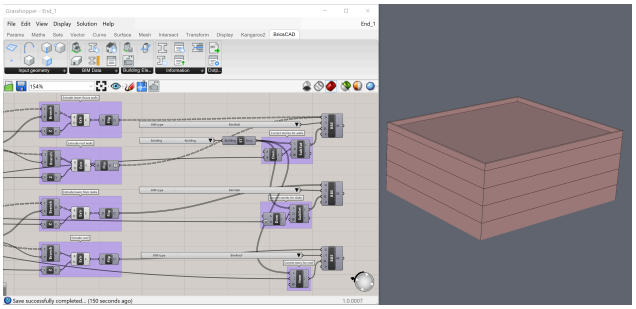



20 The last input geometry will be baked onto all remaining stories as there are now more stories than geometry. To correct this, select a correct subset of the stories to assign. Bake the geometry in BricsCAD and the stories will be correct. No 'duplicate' geometry will be baked.

21 You can see how it's done by opening the 'End_1.dwg' and then opening the Grasshopper file 'End_1.ghx' that is linked to the 'End_1.dwg'.

22 You have now successfully created a building.

23 If you want to see the result of the baked building, check the file 'End_1_Baked.dwg' or bake it yourself.



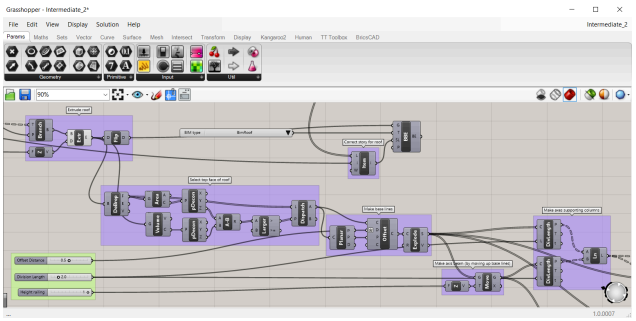
### 20.5.10.9 How to add a railing to the top of the building

We will continue with the building made in the previous procedure. So open the 'End_1.dwg' and then open the Grasshopper file 'End_1.ghx' that is linked to the 'End_1.dwg'.

1 Add the 'Start_2.ghx'-code to the Grasshopper canvas.
   - You do that by downloading the zip file at the bottom of this page and extracting its content.
   - Then open the 'Start_2.ghx' file by going to **File** > **Open Document...** in your **Grasshopper** window.
   - You can now select all of the Grasshopper code by hitting **Ctrl+A** on the keyboard and copy it using **Ctrl+C**.

- Then switch back to the document you were working on in the previous procedure by going to the upper right corner of the **Grasshopper** window and clicking on the title of the current document (Start_2).
- You then get a drop-down of all the active Grasshopper scripts, so open 'End_1.ghx'.
- Now click on the canvas and hit **Ctrl+V** to paste the 'Start_2.ghx'-code.

2 Link the **Extrude Roof** group's end component to the Brep input of the Deconstruct Brep component of the **Select top face of roof** group.



3 The script does the following:
- The **Select top face of roof** group will select the top face of the roof.
- The remaining script will take that face and use it to make the baselines for the railing.
- It will then make the axes of the supporting columns and beams.
- It will also calculate half of the height of the beam profile.
- There is still a profile required, however, to complete this step and the following one. We will, therefore, create a profile from step 6 onward.
- With that profile and the axes, the beams and supporting columns are made as extrusions.

4 The intermediate result is found in the 'Intermediate_2.ghx' and 'Intermediate_2.dwg' files. (Located in the zip file at the bottom of this page.)

5 **(Optional)** Skip the **Select top face of roof** group and replace it with a **BricsCAD** > **Input Geometry** > **Face component**. Set it to the top face of the roof slab in your BricsCAD baked building. (Do this by right-clicking on the component, click on **Set one BricsCAD face** and select the face in the BricsCAD drawing). Connect this Face component with a **Params** > **Geometry** > **Surface component** and connect this surface to the correct inputs.

6 Go to **BricsCAD** > **BIM Data**.

Drag the **Profile Names** and the **Profile Sizes** components onto the canvas.

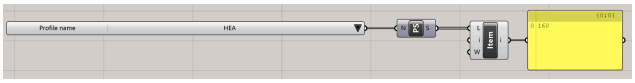
7 Select **HEA** from the drop-down list in **Profile Names**. This drop-down list displays all the names of the profiles in the **Profile Library** of BricsCAD.

8 Link the component to the input **ProfileName** of the **Profile Sizes** component.

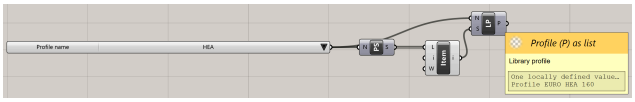


9 Link the **Profile Sizes** component to the input node **List** in a **Sets** > **List** > **List Item component**, where i is set to '4' (right-click on the **i** input and **Set integer** to **4**). This **List Item** lets you pick one size from the list with all available profile sizes for the **Profile Name** linked to it.

***Прим.:*** If you link a **Params** > **Input** > **Panel component** to the Profile Sizes output, you will see that the index 4 stands for size 160.

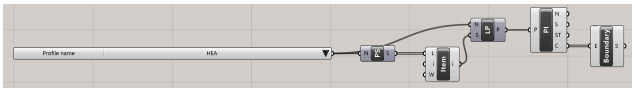


Link the **Profile Names** and **List Item** components to the **BricsCAD** > **Information** > **Library Profile component**, to get an HEA 160 profile as an output.



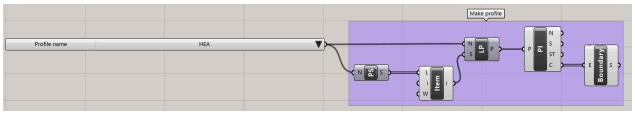
10 To get the curves that make up this profile, attach a **BricsCAD** > **Information** > **Profile Info component** to the **Library Profile**. One of the outputs created is the **Profile Curves** as a tree.

11 Attach a **Surface** > **Freeform** > **Boundary Surfaces component** to the **Profile Curves** output node. This is to later extrude the surface as a solid, instead of the lines as a surface.

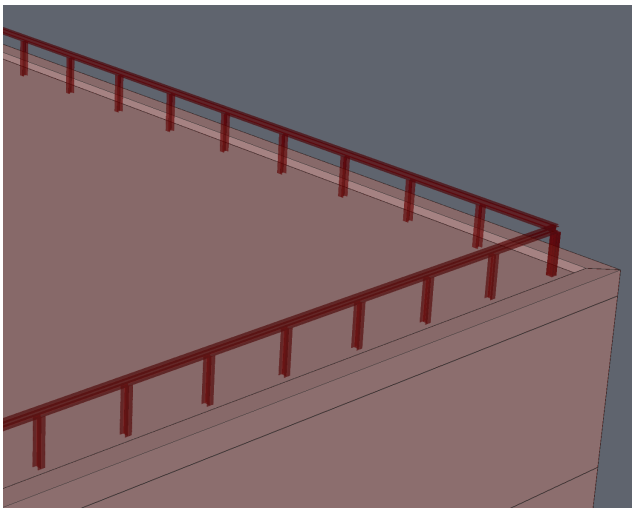


12 To group together the components just created, select them (with a selection box) and right-click on **Group**.

13 **(Optional)** To rename the group, right-click on the purple rectangle. Rename the group in the upper text box, e.g. to **Make profile**.



14 The Boundary Surfaces component should now be linked to the Content input of the Bounding Box component of the **Calculate half of height of beam** group. The Boundary Surfaces component should also be linked to the two Profile inputs of the Extrude components in the groups **Make supporting columns** and **Make beams**. The extrusions should now show you a railing with HEA 160 beams and columns.



15 To bake those beams and columns into BricsCAD attach two **BricsCAD** > **Building Elements** > **Bake Building Element components** to the Grasshopper canvas.

16 Do the following:
- Set the **Geometry** input to the extrusions.
- Set the **Element Type** input to a **BricsCAD** > **BIM DataBricsCAD > BIM Data > BIM Types component**, which is set to **BimColumn** for the column extrusions and to **BimBeam** for the beam extrusions.
- Set the **Spatial Location** input to the same as the roof.
- Set the **Profiles** input to the **Library Profiles** component from the **Make profile** group.

17 You now successfully created a railing on top of your building. (The full script to do this is in the 'End_2.ghx' file at the bottom of this page, which is linked to the 'End_2.dwg'.)

18 Bake into BricsCAD. The profiles are now set to the one in the Grasshopper script. You can still modify the connections of the beams in the corners by using L-Connect in BricsCAD. (To see the finished building with the railing on top of it, check out the 'End_2_baked.dwg' file at the bottom of this page.)

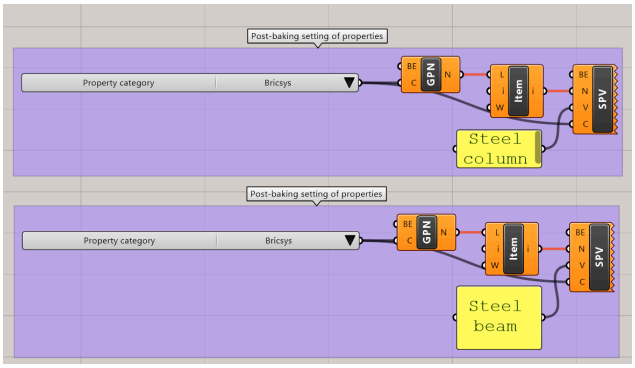
### 20.5.10.1 Procedure: add information to a baked model 0

We will continue with the building made in the previous procedure. So, open the 'End_2.dwg' and then open the Grasshopper file 'End_2.ghx' that is linked to the 'End_2.dwg'.

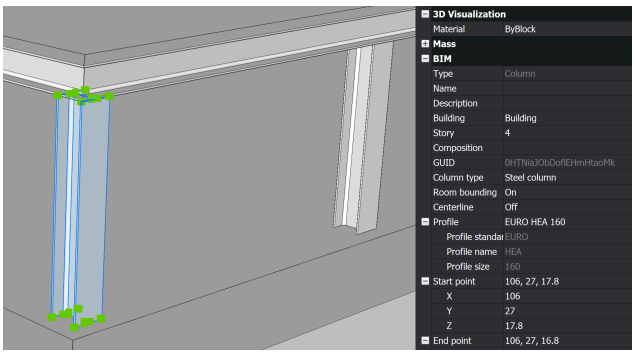
1 Add the 'Start_3.ghx' code to the canvas.
- You do that by downloading the zip file at the bottom of this page and extracting its content.
- Then open the 'Start_3.ghx' file by going to **File** > **Open Document...** in your **Grasshopper** window.
- You can now select all of the Grasshopper code by hitting **Ctrl+A** on the keyboard and copy it using **Ctrl+C**.
- Then switch back to the document you were working on in the previous procedure by going to the upper right corner of the **Grasshopper** window and clicking on the title of the current

document (Start_3).

- You then get a drop-down of all the active Grasshopper scripts, so open 'End_2.ghx'.
- Now click on the canvas and hit **Ctrl+V** to paste the 'Start_3.ghx'-code.




2 This code has some post-baking settings of classified and baked beams and columns:
   - The script uses the **Get Properties Names component** to display the BIM properties in the Property category **Bricsys**.
   - In those properties, it selects item 1 for the columns and item 0 for the beams, **ColumnType** and **BeamType** respectively.
   - To conclude, it sets the properties to the value specified in the **Panel components**, using the **Set Property Value components**.
3 To make the script work, bake the two **Bake Building Element components** that make up the railing.
4 Now link the Building Element output of the **Bake Building Element components** to the respective Building Element input of the **Get Properties Names components**.
5 Also, link the Building Element output of the **Bake Building Element components** to the respective Building Element input of the **Set Property Value components**.
6 If you now select a column in BricsCAD and open its Properties, the **ColumnType** will be set to **Steel column**. When you change the value of the **Panel component**, the Properties in BricsCAD change in real-time.
7 The resulting script: 'End_3.ghx' can be found in the zip file, located at the bottom of this page. It works with the 'End_3.dwg' file in which nothing is yet baked. Bake all the Building Elements, to get the post-baking settings.



### 20.5.10.1 The BricsCAD toolset in Grasshopper

1

| Icon | Name | Description |
|---|---|---|
|  | Plane | Represents a plane in BricsCAD. |
|  | Point | Represents a point in BricsCAD. |
|  | Curve | Represents a BricsCAD curve. |
|  | Entity | Represents a BricsCAD entity. |
|  | Edge | Represents a BricsCAD edge. |
|  | Face | Represents a BricsCAD face. |
|  | Vertex | Represents a BricsCAD vertex. |
|  | Buildings | Provides a name picker for all the buildings present in **Spatial Locations** in BricsCAD. |
|  | Stories | Returns all the stories attached to the input building. |
|  | Profile Names | Provides a name picker for all the profiles present in **Profiles** in BricsCAD. |
|  | Profile Sizes | Returns all the sizes attached to the input profile. |

| Icon | Name | Description |
|---|---|---|
|  | BIM Types | Provides a type picker for all the **BIM Types** available in BricsCAD. |
|  | Property Categories | Provides a category picker for all the property categories available in BricsCAD. |
|  | Bake Geometry | Bake the Grasshopper geometry into the current BricsCAD drawing, while disregarding the BIM data attached to it. The output of Bake Geometry is a reference to the baked building element without BIM data. |
|  | Bake Building Element | Bake the Grasshopper geometry into the current BricsCAD drawing, while adding BIM data to it. The output of Bake Building Element is a reference to the baked building element with BIM data. |
|  | Elements on Spatial Location | By default, returns all the building elements present in BricsCAD. When using input parameters, returns the building elements filtered by element type and/or spatial location. |
|  | Library Profile | Returns a profile from the library, according to the given name and size. |
|  | Profile Info | Returns the information (name, size, standard and curves) of the specified profile. |
|  | Property Names | Returns the property names, attached to a building element, in the specified property category. |

| Icon | Name | Description |
|---|---|---|
|  | Property Value | Returns the property value, attached to a building element, for the specified property name and category. |
|  | Linear Solid Info | Returns information (axis, extrusion path and profile curves) about a linear solid present in the BricsCAD drawing. |
|  | Set Property | Sets the property value of the building element according to the specified name, category and value. |
|   | IFC Export | Exports the specified building elements to IFC. |
|  | Link | Links the Grasshopper script to the open BricsCAD .dwg file. |
|  | Bake | Bakes the geometry from the selected Bake Geometry and Bake Building Element components. |

Starter files.zip

## 20.6 Совместная работа над проектом

### 20.6.1 BIM collaboration format (BCF) panel

#### 20.6.1.1 About BCF

The BIM Collaboration Format (BCF) is an open file format used to exchange textual comments, images, and more on top of an IFC model. This allows better communication between different parties in the BIM process.

BCF makes it possible to link information such as a comment, a screenshot, a list of involved objects, and a camera viewpoint to an issue. An issue can be anything ranging from a small change in the design to clashes between different disciplines in the model. These issues can then either be saved as .bcf files or they can be managed on the cloud, using any sort of BCF cloud management system.

The advantage of BCF is that it allows communication over different software packages (e.g. a BCF issue created in Solibri can be opened in BricsCAD BIM), and that it enables easy navigation through a model based on issues.

#### 20.6.1.2 BCF files

Issues can be saved in standalone files. This is generally a more cumbersome workflow than working with a cloud system but it can be useful in case you are working offline. A .bcf file can then be imported in any BCF manager (e.g. BricsCAD BIM) to read the issues and see where in the model it applies.

Note that a .bcf file cannot be created in BricsCAD BIM. For this, you will need access to another service e.g. Solibri Model Checker or BIMcollab ZOOM. This software allows you to create a .bcf file based on one or more .ifc files. You can then create issues containing:

- Comments
- Screenshots or other images
- A list of involved objects, based on GUID (Global Unique IDs) to distinguish them unambiguously
- A camera position

#### 20.6.1.3 Using the cloud

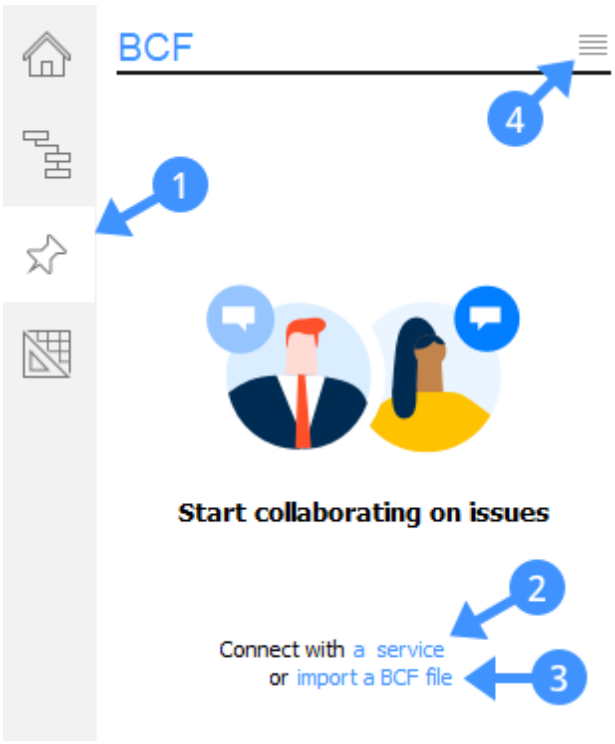
It is generally easier to use cloud services to manage BCF issues. Currently, BricsCAD BIM supports connection to three such services: **BIMcollab**, **BimSync**and **BIMtrack**. If you wish to connect to these services from within BricsCAD BIM you will need an account of these services first. These allow you to create issues and manage them in real-time from anywhere.

Note that it is not possible to **create** issues within BricsCAD BIM. The BCF Panel allows you to connect to these services from within BricsCAD BIM, log into your account and add comments, screenshots or change statuses of existing issues.

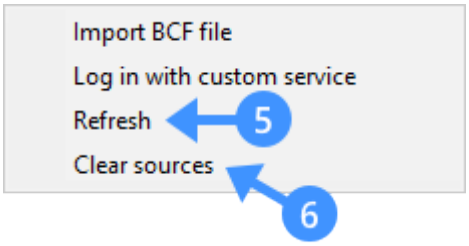
#### 20.6.1.4 The BCF panel

The **BCF** panel can be opened by clicking the board pin icon on the left of your screen (1). If the icon is not there, right-click an empty space in the ribbon and under **Panels**, enable the BIM **BCF** panel.

Using the panel, you can either **Connect with a service** (2) or **import a BCF file** (3), as stated above. The hamburger menu at the top (4) gives you the same options, as well as refreshing the panel and clearing the sources.



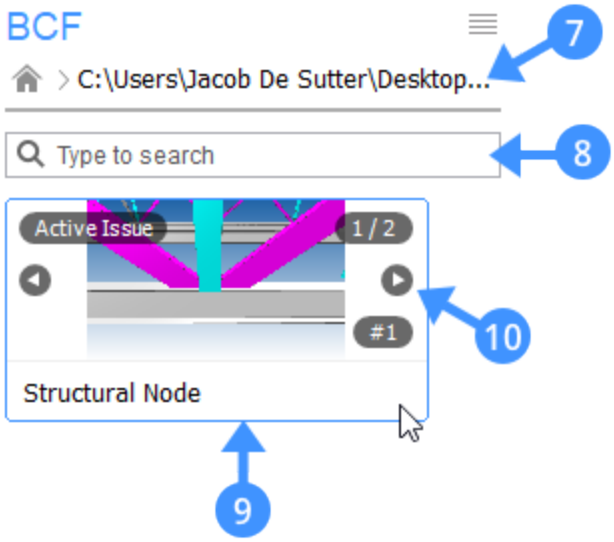

**Refreshing the panel** (5) will make sure you see the latest updates in case you are working on the cloud. **Clearing the sources** (6) will disconnect you with the cloud service in case you are online or close the .bcf file in case you are working on an imported .bcf file.



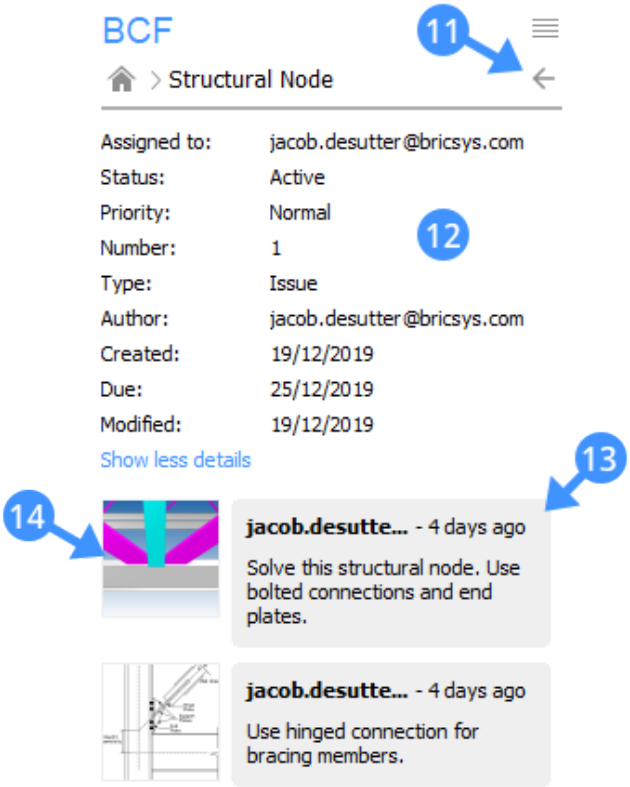

### 20.6.1.5 Procedure: working with the BCF panel

The following example will be shown with the files attached at the end of this article. If you want to follow along, open **West_str.dwg** and import **Structural_node.bcf**.

When importing a .bcf file you will see the path of the currently opened file (7), a search bar that can be used to filter on issues (8) and a list of issues. In the case of this imported .bcf file, there is only one issue (9) with two associated views (10).




Click on the issue to open it. When opening an issue, you will see a button to return to the issue overview (11), some more detailed information about the issue such as status, creation date and so on (12) and different comments on this issue (13). If this comment has an associated camera position, then clicking the thumbnail image (14) will take the camera in the current drawing to this position. This is useful if you want to know where exactly the issue is and you don't want to manually search through the model.

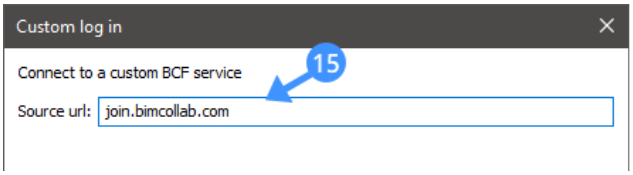


When using a cloud service, you are presented with some more options, such as setting statuses and adding comments and screenshots.

To connect to a cloud service, click the **Connect with a service** button (2). You are prompted to enter a **Source URL** (15). The correct Source URL's for the respective services can be found in the table below.

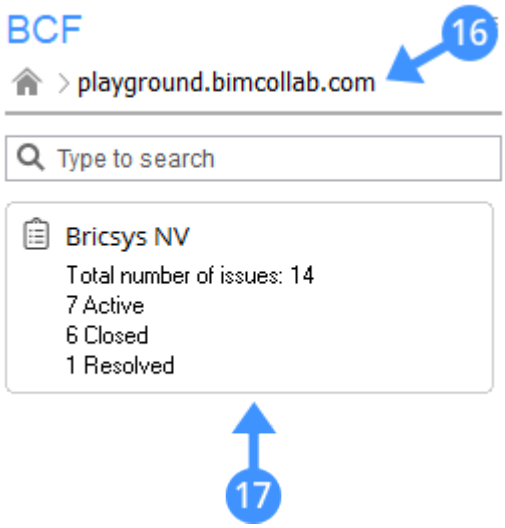
| Cloud service | Source URL |
|---|---|
| BIMcollab | *<space name>*.bimcollab.com * |
| BimSync | bcf.bimsync.com |
| BIMtrack | bcfrestapi.bimtrackapp.co |

* Note that the Source URL for BIMcollab includes the name of the space you bought. If you are using a free account, use *join.bimcollab.com* instead.

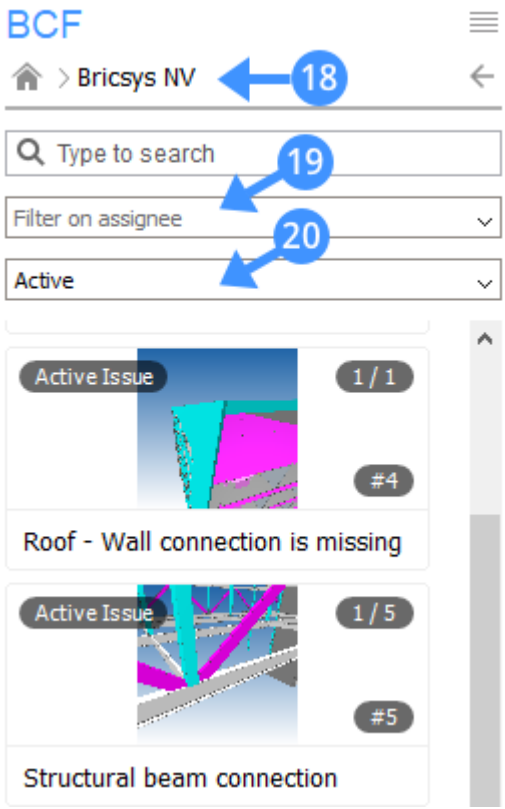



The following examples will be shown on an internal cloud space so it will not be possible to follow along with these steps exactly. It is possible however to create a free account on, for example, BIMcollab. Clicking the **Log In** button should take you to a login page of the chosen service.

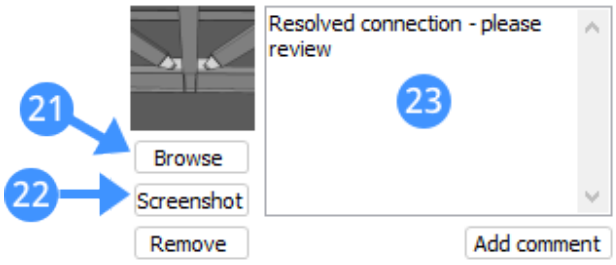
After logging in, the source URL should be shown (16), along with a list of projects that you are involved in (17). In this example, only one project is shown with 14 issues. Left-clicking the project will open it and show a list of the issues, similar to what we saw when importing the .bcf file.



At the top of the panel, the current project is displayed (18). Note that now it is also possible to filter the issues on assignee (19) and status (20). This makes it easy to quickly find the issues that are relevant to you.




Left-clicking an issue will open it and you will see similar options as before: detailed information about the issue (12) and comments on this issue (13). However, note that at the bottom it is also possible to add bitmap images (21) or a screenshot from model space (22). You can add some textual comments as well (23).




Structural_node.bcf

West_Str.dwg

### 20.6.2 Bricsys 24/7 panel

#### 20.6.2.1 About Bricsys 24/7

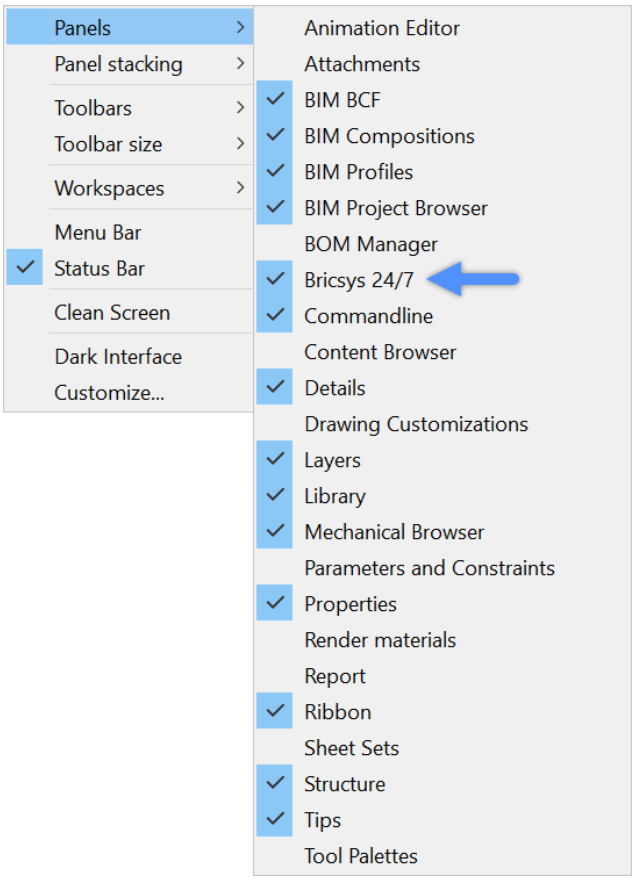
Bricsys 24/7 helps team members to find and share their drawings. Bricsys 24/7 makes project management, a collaboration between project teams, cloud computing security, and task automation possible. A big advantage of this cloud-based service is that BricsCAD is directly linked to it by the Bricsys 24/7 Panel.

#### 20.6.2.2 Use the Bricsys 24/7 panel in BricsCAD BIM

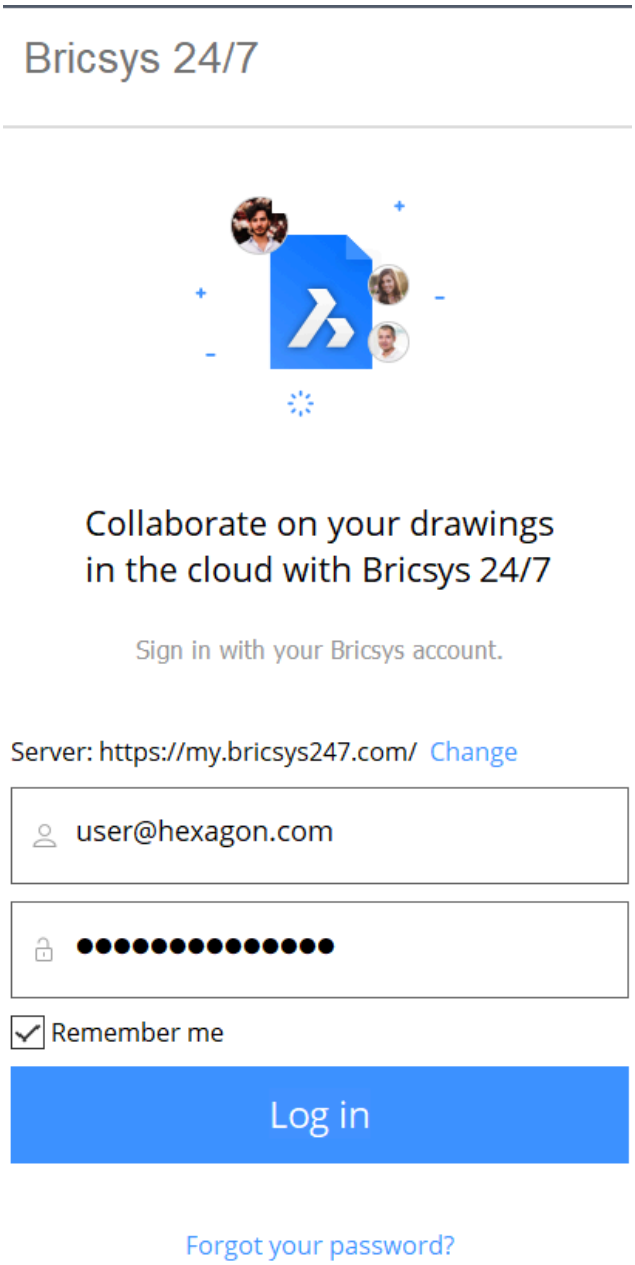
1 Open the **Bricsys 24/7** panel by clicking on the Bricsys 24/7 icon in the **Tool** panel on the right-hand side of the workspace.



***Прим.:*** If you do not find this icon in the **Tool** panel, right-click in the **Tool** panel and check **Bricsys 24/7** in the drop-down menu.



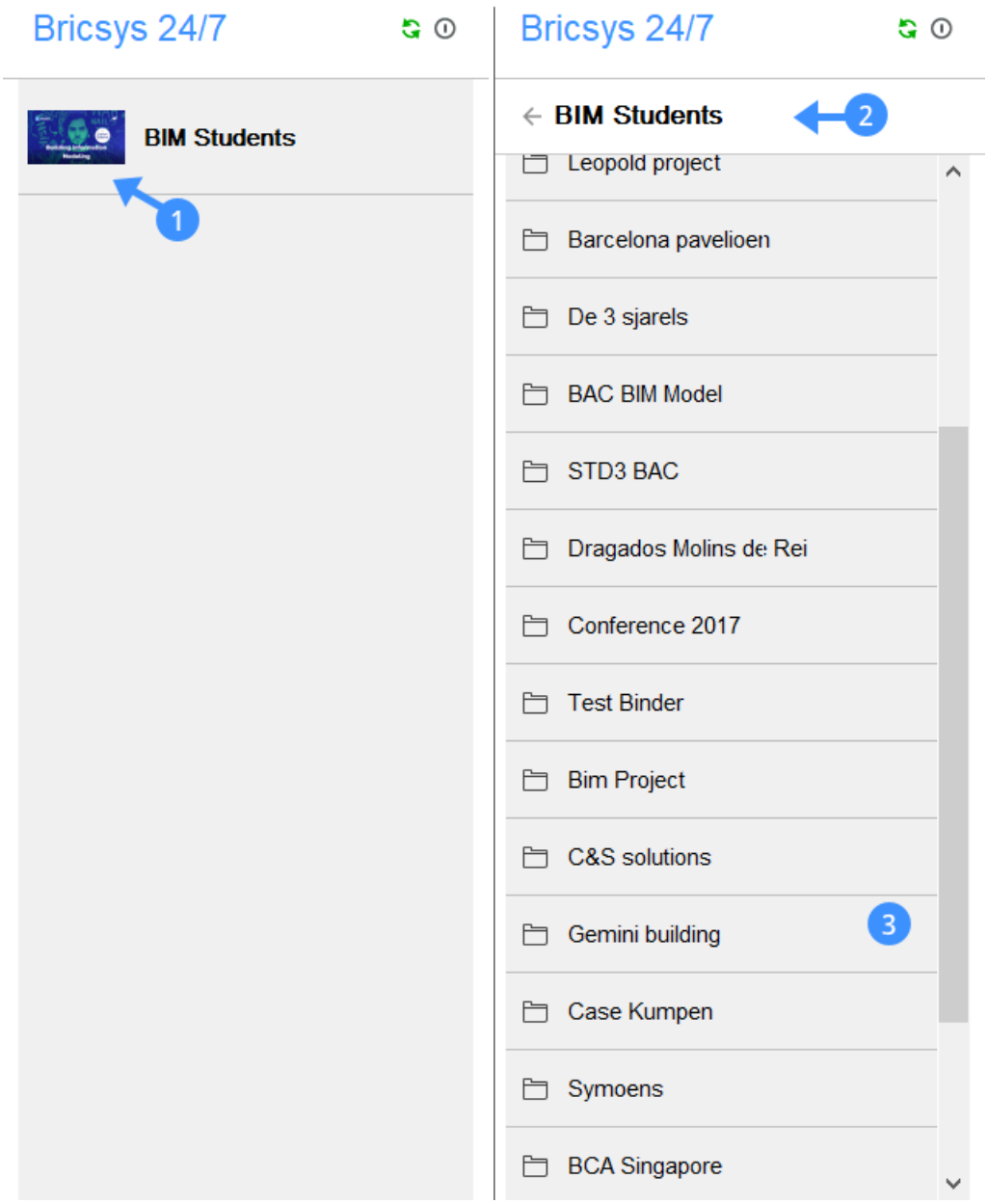

If you are logged out of your 24/7 account a screen will appear:

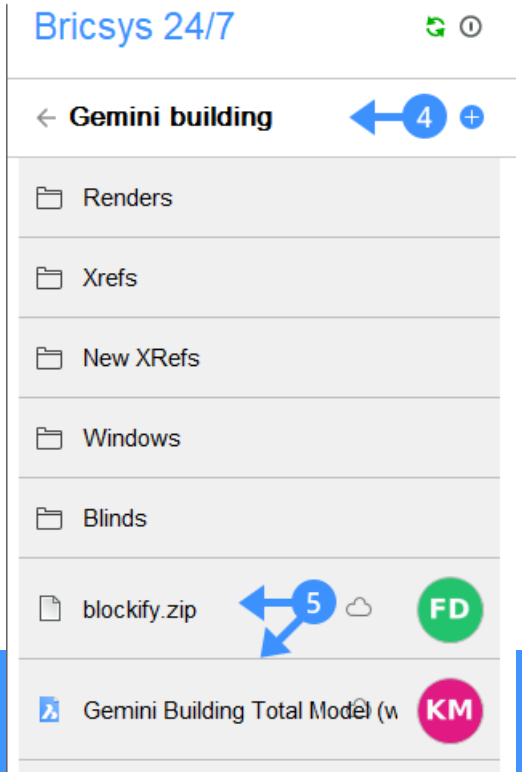



2 Enter your email address and password and press the **Log in** button.

3 (Option) Tick the **Remember Me** checkbox to store username and password.

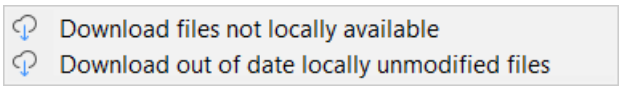
4 Once logged on you will see an overview of your 'in the cloud' Projects, Folders, and Documents.


Bricsys 24/7
BIM Students
1
Bricsys 24/7
BIM Students
2
Leopold project
Barcelona pavelioen
De 3 sjarels
BAC BIM Model
STD3 BAC
Dragados Molins de Rei
Conference 2017
Test Binder
Bim Project
C&S solutions
Gemini building
3
Case Kumpen
Symoens
BCA Singapore


Bricsys 24/7
Gemini building
4
Renders
Xrefs
New XRefs
Windows
Blinds
blockify.zip
5
FD
Gemini Building Total Model (w
KM

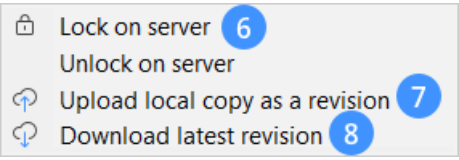
| | | |
|---|---|---|
| **(1)** | **project** | The first thing you see on the 24/7 Panel when logged on, are all the projects you are assigned to. |
| **(2)** | | At the top of the panel you see your project. You can go back to the overview of projects by clicking on it. |
| **(3)** | **folder** | When you open a project, a list of folders is shown. |
| **(4)** | | At the top of the panel you see the folder you're in. You can go back to the overview of by clicking it. |
| **(5)** | **documents** | In a folder you find documents. On the right-hand icons are displayed:<br>The profile picture or the initials from the author of the document.<br>The document is in the cloud.<br>Someone is currently working on the document.<br>The document is currently uploading.<br>The document is locked by someone, another person cannot change it. |

Right-click on a folder to get a menu with 2 possible actions:




You can download a BIM model from your Bricsys 24/7 project to a local folder.

5 Right-click on a document to get a menu with possible document actions:




| | |
|---|---|
| **(6)** | Lock the document so no changes can be made. |
| **(7)** | Upload your drawing as a revision of the original drawing. |
| **(8)** | Download the latest revision. |

### 20.6.3 IFC import and export in BricsCAD

#### 20.6.3.1 Commands

XREF, XATTACH, XOPEN, XCLIP

#### 20.6.3.2 About

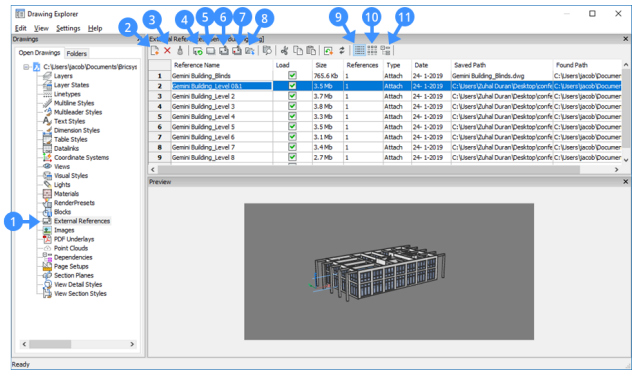
Using external references or Xrefs allows you to insert any model in the current BIM model. Instead of copying the model, only a reference to the source model is saved in the master model. You can attach multiple models to a single master model without increasing the size of the current model significantly. Further advantages of using the Xrefs are that the source models can be edited simultaneously by different members of the design team, and the master model can always be updated to show the latest version of each of the source models.

***Прим.:*** You can distinguish between XRef entities and entities in the current drawing using the XDWGFADECTL system variable. This system variable allows to fade the display of XRefs. You can give the values between 0 (no fading) and 90 to change the default value of the fading.

#### 20.6.3.3 Procedure: managing XRefs in the current drawing

1 Open the **Drawing Explorer** dialog box and click the **External References** section.

The XRef preview pane appears, allowing the user to attach a model or manage the existing XRefs on the list. The following illustration shows the XRef preview pane.

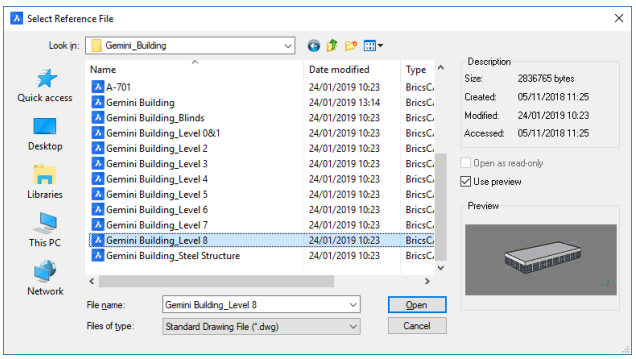


| | |
|---|---|
| **External References(1)** | Shows the External References section. |
| **Attach XRef (2)** | Attaches a model as an external reference. |
| **Detach XRef (3)** | Removes the selected XRef completely from the current model. |
| **Reload XRef (4)** | Reloads the selected XRef. This is useful when the source models have been changed, and you want to see the updated versions in the master model. |
| **Unload XRef (5)** | Unloads the selected XRef. This hides the XRef from the view. |
| **Bind XRef (6)** | Merges the source models permanently to the current drawing. Merging the XRef to the current drawing makes the XRef part of the drawing. References to the XRef disappear from the Drawing Explorer. |
| **Insert XRef (7)** | Converts XRefs into blocks. |
| **Open XRef (8)** | Opens the referenced drawing. |
| **Detail view (9)** | Displays the details of XRefs. |
| **Icon view (10)** | Displays icons of the XRefs. |
| **Tree view (11)** | Displays the relationship between XRefs in a tree-like view. |

2 To attach externally referenced models, click the **Attach XRef** tool button on the **Drawing Explorer** dialog box.

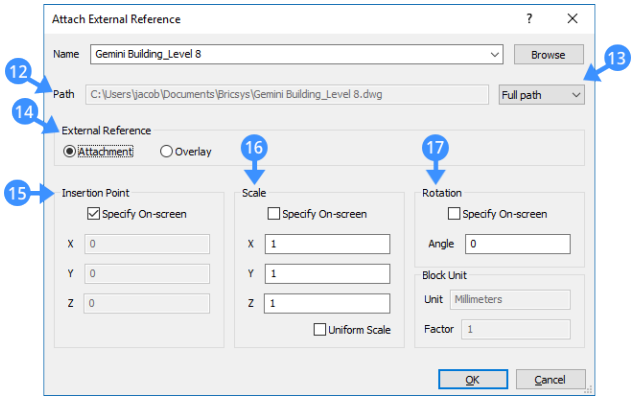
The **Select Reference File** dialog box displays.

The following illustration shows the **Select Reference File** dialog box. From this dialog box, select the DWG file you wish to attach in your main model, and then click **Open**.




Next, the **Attach External Reference** dialog box displays.

The following illustration shows the **Attach External Reference** dialog box. The table below presents each option of the **Attach External Reference** dialog box. From this dialog box, select the options you want to apply, and then click **OK**.




| | |
|---|---|
| **Path field (12)** | Displays the full path of the selected drawing. |
| **Path type (13)** | Includes **Full path**, **Relative path**, and **No path** options.<br>**Full path**: When you move your drawing containing an XRef to another location, BricsCAD will still search for the XRef in the previous location and it will not find this XRef.<br>**Relative path**: When you move your drawing containing an XRef to another location, the new file path will be created in the new location. The file folder will set up correctly using the same folder structure as the first one. |

| | |
|---|---|
| **External Reference type (14)** | Includes **Attachment** and **Overlay** reference types.<br>**Attachment**: Attaches all XRefs, including nested XRefs. For example, you have the first level of XRef “X” and this contains nested XRefs “Y” and “Z”. When the attachment option is selected the first level of XRef and its nested XRefs Y and Z will be attached to the drawing.<br>**Overlay**: Attaches only the first level of xref (X), nested xrefs (Y, Z) will be not loaded. This can also be used to prevent circular references. |
| **Insertion Point (15)** | Allows the user to specify the Insertion point on the screen or in the X-, Y-, Z- fields. |
| **Scale (16)** | Allows the user to specify the Scale on the screen or in the X-, Y-, Z- fields. Check the **Uniform Scale** option to specify the scale in the X- field. |
| **Rotation (17)** | Allows the user to specify the Rotation angle on the screen or in the **Angle** field. |

Note that the **BIM XReference** option from BIM Classify allows you to classify the model as an external reference. The identification of information will not explicitly be represented in the current model or in the project database. Such information appears after attaching the model to another project. To do so, follow the above step.

The following illustration shows the BIM category in the **properties** panel after attaching the model that has been classified as an XRef.




3 To detach an existing XRef, select the XRef you wish to detach in your main model, and then click the **Detach XRef** tool button on the **Drawing Explorer** dialog box.

The selected XRefs are detached.

***Прим.:*** Press and hold the CTRL key to select multiple XRefs or to unselect a selected XRef.

4 Do one of the following to unload and reload an existing XRef.

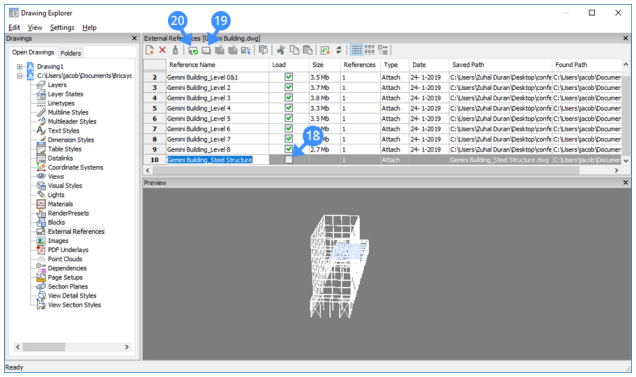
- Click the **Load** (18) column of a loaded XRef.

  This XRef is unloaded in your main model.

  To reload the XRef check the load box of this XRef.

- Select the XRef, then click the Unload XRef (19) tool button on the Details toolbar.
- Select the unloaded XRef, then click Reload XRef (20) tool button.

  The following illustration shows the **Load** column (18), Unload XRef (19) and Reload XRef (20) tool buttons on the **Drawing Explorer**.




5 To modify the attachment type, click the **Type** column of an existing XRef and switch from **Attach** to **Overlay** or vice versa. The same process can be followed for modifying the **Saved path** of an XRef and **Found path** of an overlay.

6 Click **X** to close the dialog box. Notice that the changes that have been made appear after you close the **Drawing Explorer**.

### 20.6.4 Import and underlay Autodesk® Revit® files in BricsCAD

#### 20.6.4.1 Commands

IMPORT

#### 20.6.4.2 Disclaimer

Unless otherwise indicated, the word "Revit" used in this article refers to the Revit® software and its peripherals by Autodesk®.

#### 20.6.4.3 About Revit import

The Revit import function allows you to import .RVT files into BricsCAD as 3D solids or as a geometry underlay for referencing purposes. However, any project data and model metadata, i.e. Constraints, Wall Type structure assembly, materials etc., from the Revit model are not imported.

You can also choose to import selected geometries by 3D Views, by Family Categories, by Levels or by View Worksets, depending on how the Revit project file has been set out.

If you import the RVT file as a solid, only the geometry will be imported with no BIM metadata attached to it. The solids can later be classified and edited with direct modelling tools in BricsCAD.

If you import the RVT file as an underlay, almost like how an XREF is used, the BricsCAD model performs faster and smoother generally. This process also takes a shorter time than importing as a solid. However, the underlay model becomes an OdRvtEntity, like a non-editable, singular block. This method is particularly

useful if you are looking to import the Revit model only for reference and coordination purposes, i.e. importing a Revit structural model to look for clashes with your architectural / MEP model within BricsCAD.

BricsCAD V21 currently supports the import of **Revit 2015-2020** files only. If you would like to import a model created in a later version of Revit, please consider importing the model into BricsCAD as an IFC file. For more information, please visit the IFC Import and Export in BricsCAD article.

### 20.6.4.4 Procedure: importing a Revit file as a solid in BricsCAD

1 Open a new BricsCAD file.

2 Type **Import** in the command-line and press Enter **or** click the BricsCAD icon in the ribbon and select **Import** to launch the Import command.

3 The **Import File** dialog box pops up. Select Revit Project (*.rvt) in **Files of type**, select the file you want to import from your folder and click **Open**.

4 **You are prompted:** Choose import mode as [Import/Underlay]:

Type **I** and Press Enter to select Import.

5 **You are prompted:** Choose elements [**A**ll/by **V**iews/by **C**ategories/by **L**evels/by **W**ork**S**ets]:

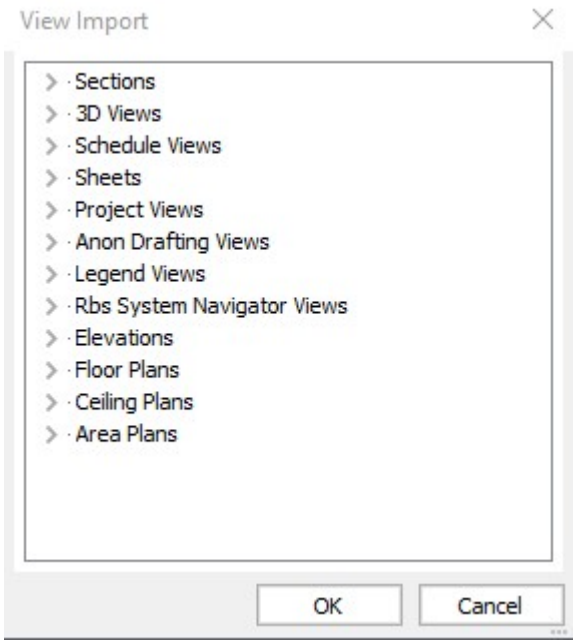
There are different filters to import a file. Select the filter you want to use by clicking into your desired option in the command-line or by hitting their respective letters in bold.

- **All**

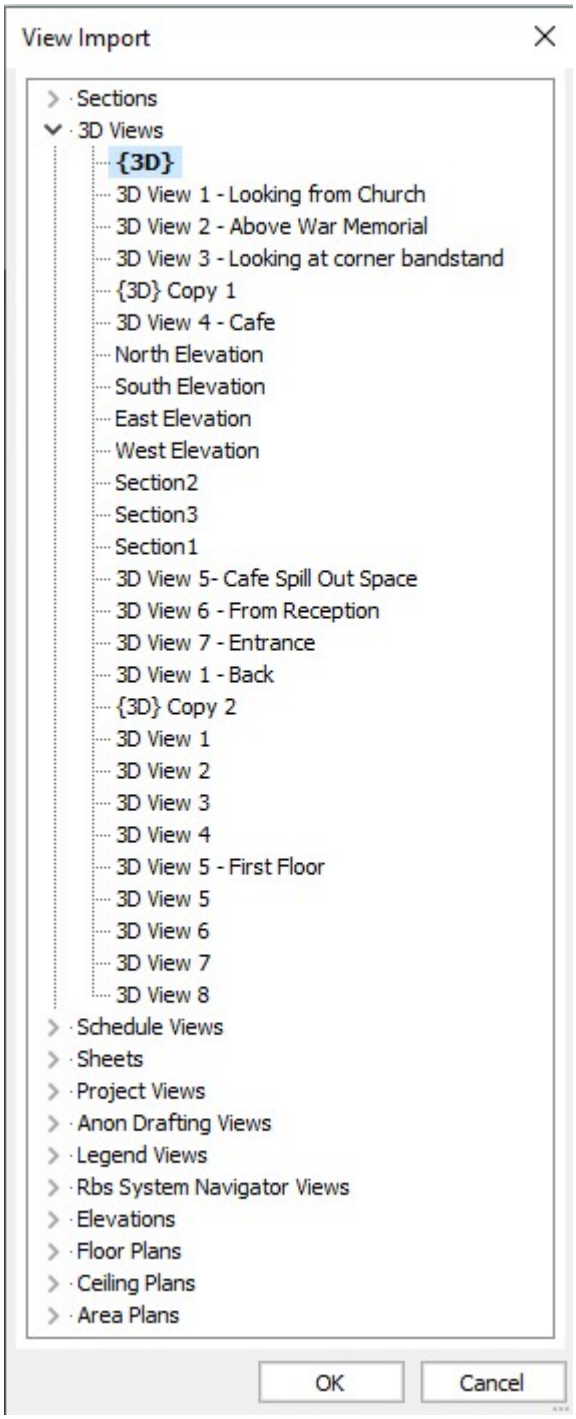
  All elements present within the RVT file will be imported.

- **By Views**

  Select a view from the **View Import** dialog box and the elements will be imported based on the view's Visibility/Graphics Overrides settings in Revit. The views are structured based on their View Types.



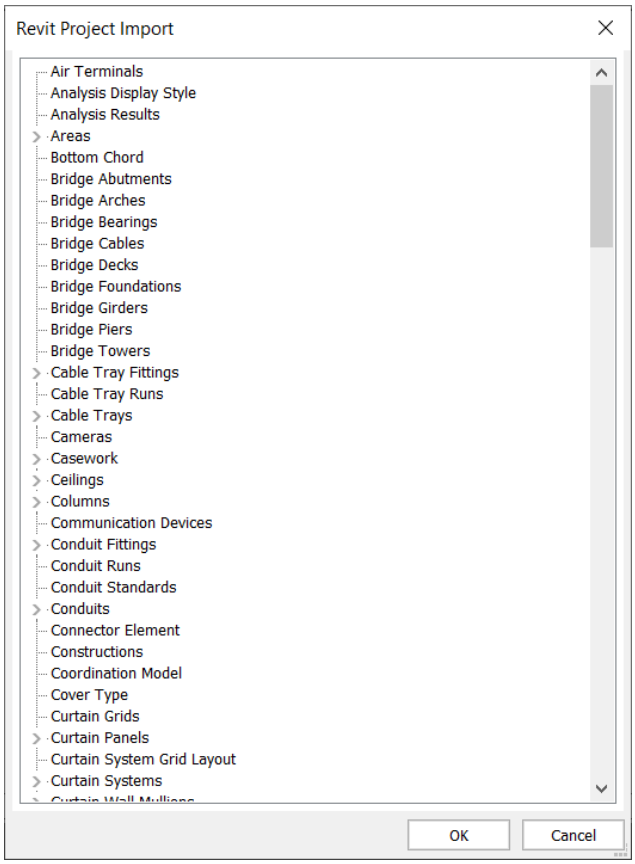
You can expand any relevant or applicable View Type category for the specific views. It could look like this, if the **3D Views** category is expanded:




***Прим.:*** Importing elements via a 2D view, ie. Floor Plans etc., only brings elements that are made visible in the Visibility/Graphic Overrides in the Revit model settings for that particular view. It does not necessarily bring elements hosted onto its corresponding level, if they are made hidden on the plan view.

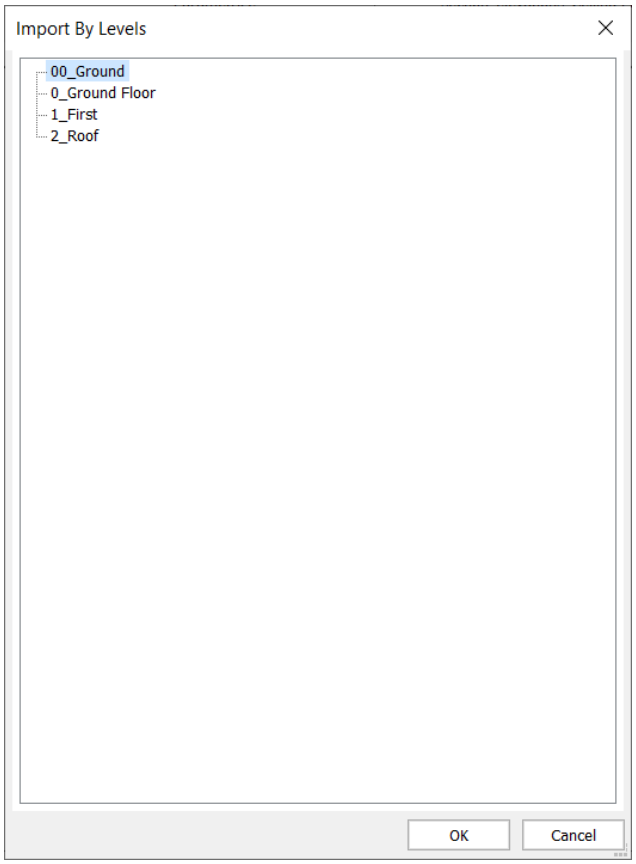
- **By Categories**

  You can select one or multiple built-in Categories and Element Types from the **Revit Project Import** dialog box. Elements of those chosen categories and element types will be imported.




- **By Levels**

  You can select one or multiple levels from the **Import By Levels** dialog box. Elements hosted to the chosen levels will be imported.

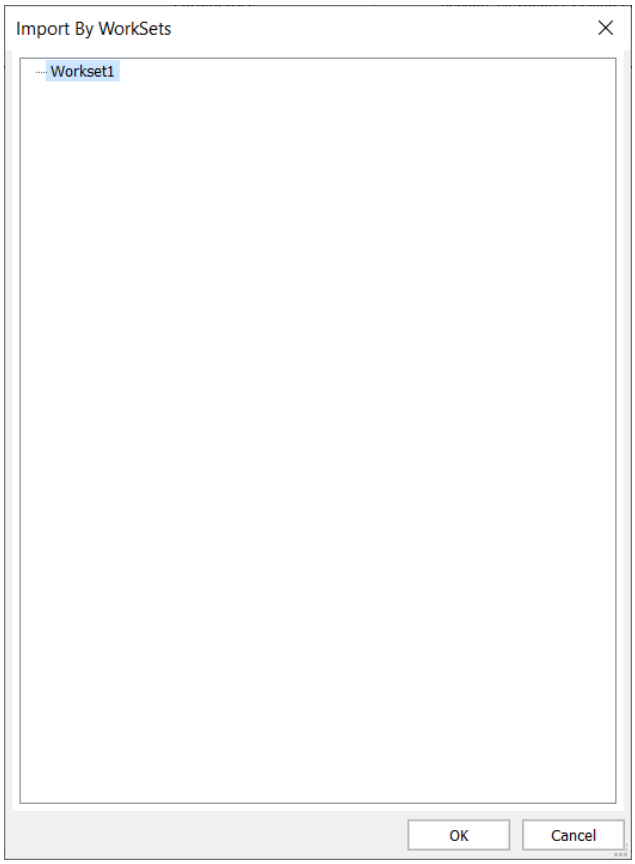



- **By WorkSets**

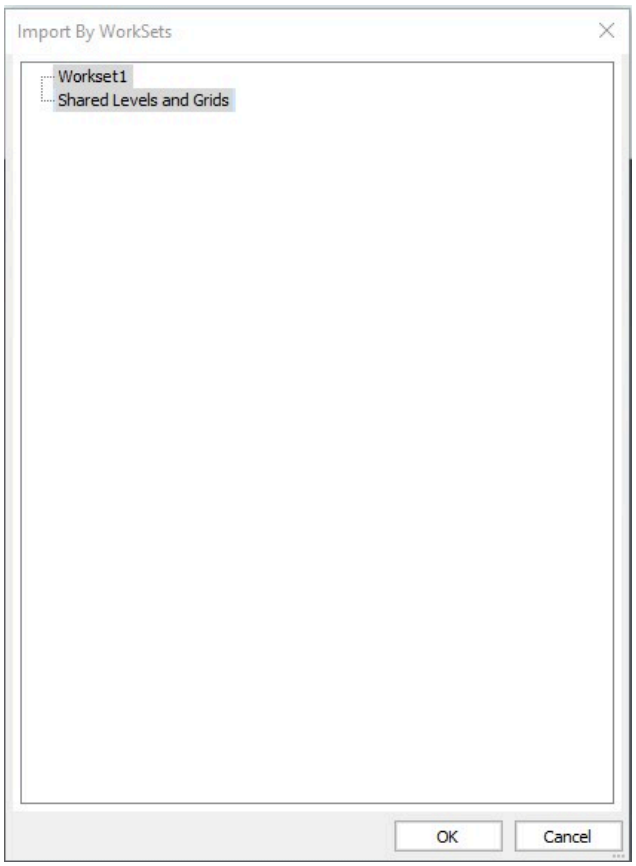
  You can select one or multiple worksets from the **Import By Worksets** dialog box.

  This method of importing works best on a workshared Revit model. It is recommended you import a Central Model to ensure you are importing the latest information in the worksets. Detaching the Central Model is optional.

  If you import a non-workshared Revit model, only **Workset1** is available.




If you import a workshared Revit model, either Central or Local, the User-Created worksets within the model will be available in the dialog. The elements which belong to this workset will be imported.




***Прим.:*** The **Shared Levels** and Grids and **Workset1** worksets are created in Revit by default when Worksharing is enabled and can be managed by the Revit user.

6 If you have chosen to import by view, you do not have to specify the level of detail. The Detail Level of the selected view set in Revit will be used.

7 For the rest of the other options,

8 **you are prompted:** Import Elements with Detail Level [Fine/Medium/Coarse]:

Select the level of detail you want. Type **F** in the command-line for fine, **M** for medium and **C** for coarse and press Enter.

***Прим.:*** When selecting the level of detail, make sure you are aware of how the incoming geometries have been set out in terms of Detail Levels within the Family Elements Visibility Settings in Revit.

9 Select the View, Categories, Levels or WorkSets you want to import and click **OK**. The RVT file will be imported as 3D solid geometries.

10 Now, you can classify the solid manually or automatically and/or further edit it with direct modeling tools in BricsCAD.

***Прим.:*** Some of Revit's system families, such as walls and floors etc., are imported into BricsCAD as 3D solids, where you can further manipulate their geometries directly. This also applies to Model-In-Place components and Element Types, such as Fascias, that can be defined geometrically by a Profile.

However, other system families designed with in-built constraints, i.e. stairs and curtain walls etc. are imported into BricsCAD as separate Block References. For eg., Curtain Panecl and Curtain Wall Mullion Types, which make up a Curtain Wall Type in Revit, will exist as separate Block References. This also applies to all other loadable families, like furniture, placed within the Revit model. For more information, please visit the article on Working with Blocks.

#### 20.6.4.5 Procedure: importing a Revit file as an underlay in BricsCAD

1 Open a new BricsCAD file.

2 Type **Import** in the command-line and press Enter **or** click the BricsCAD icon in the ribbon and select **Import** to launch the Import command.

3 The **Import File** dialog box pops-up. Select Revit Project (*.rvt) in **Files of type**, select the file you want to import from your folder and click **Open**.

4 **You are prompted:** Choose import mode as [Import/Underlay]:

Type **U** and Press Enter to select **Underlay**.

5 Follow Steps 5-7 of the 'Procedure: importing a Revit file as a solid in BricsCAD' above.

6 **You are prompted**: Deviation value:

By default, the deviation value is 0.001. Press Enter to accept **or** type a new value and press Enter.

***Прим.:*** The deviation value defines the accuracy of the imported model. A lower value will result in a finer geometry, but it will take longer time to load.

7 The RVT file is now imported as a non-editable OdRvtEntity.

#### 20.6.4.6 Procedure: importing a Revit family in BricsCAD

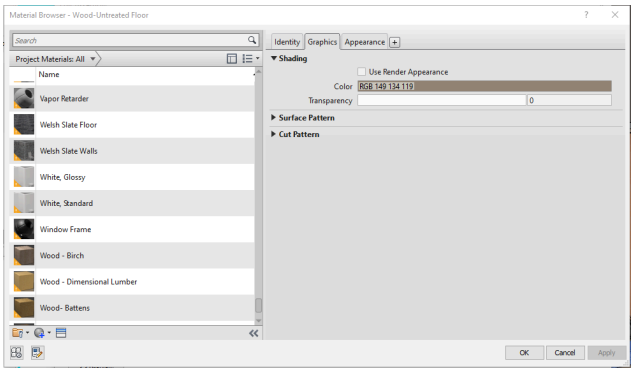
Please refer to the article **Importing Revit RFA Families in BricsCAD** for more information.

#### 20.6.4.7 Texture and graphics of the imported geometries

Although project information and element metadata in a Revit model cannot be brought into BricsCAD (just yet), there are some aspects of the imported 3D solids, i.e. Material Color, Transparency, which can be brought over and therefore correspond to the original graphics set in the Revit.

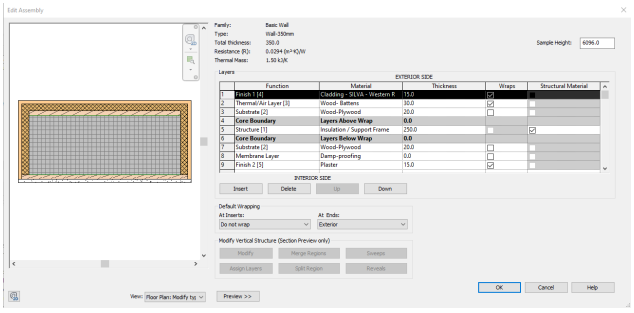
To retain the consistency of the graphical information during the import process, ensure your RGB color and transparency settings set in Revit's Material Browser are set to your desired values.

In the example below, the values taken will be **RGB 149 134 119** and **0** (%) for the colors and transparency values respectively.




If the imported element is a Revit System Family with the ability to define its structure assembly, i.e. Wall Type, and is made up of various layers of materials or functions, BricsCAD applies the color template values from the outermost materials (on both the Exterior and Interior sides) accordingly on to the corresponding geometry's faces.

Let's take an example from the illustration below, the values will be derived from the materials assigned on Layers 1 and 9, which are **Cladding...** and **Plaster** respectively.




Otherwise, BricsCAD applies the color value, and the transparency settings if applicable, to the 3D solid replacing the respective parts of the Revit Family. This also applies to those Loadable Families and Components, having loaded into the imported Revit project.

## 20.6.5 Importing Autodesk® Revit® families in BricsCAD

### 20.6.5.1 Commands

IMPORT, BMINSERT

### 20.6.5.2 Disclaimer

Unless otherwise indicated, the word "Revit" used in this article refers to the Revit® software and its peripherals by Autodesk®.

### 20.6.5.3 Important notification before getting started

RFA import is a work-in-progress. See the BMINSERT command.

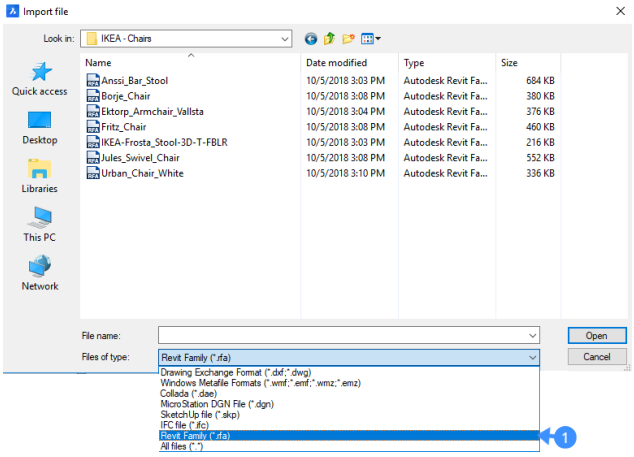
BricsCAD V21 currently supports the import of **Revit 2015-2020** files only. If you would like to import a model created in a later version of Revit, please consider importing the model into BricsCAD as an IFC file. For more information, please visit the **IFC import and export in BricsCAD** topic.

### 20.6.5.4 Importing RFA family files

1 Launch the Import command.

1 The **Import file** dialog box displays.

2 Select **Revit Family (*.rfa file)** (1) in the **Files of type** list.




3 Select the .rfa file you want to import.

4 Click **Open** to import the Revit Family.

When the RFA family file is imported the geometry from this file is included in BricsCAD as separate 3D solids.

### 20.6.5.5 Procedure: inserting RFA family files

***Прим.:***

- When DUCS = ON (Dynamic UCS) the RFA entity aligns with the 3D solid face under the cursor.
- When DYN = ON (Dynamic Dimensions) the distances from the nearest parallel edge on the face of the 3D solid display.

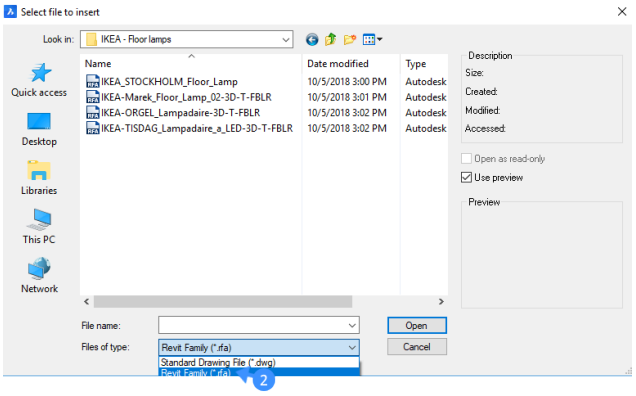
**Using BmInsert**

1 Launch the BMINSERT command.

1 A **File** dialog box appears.

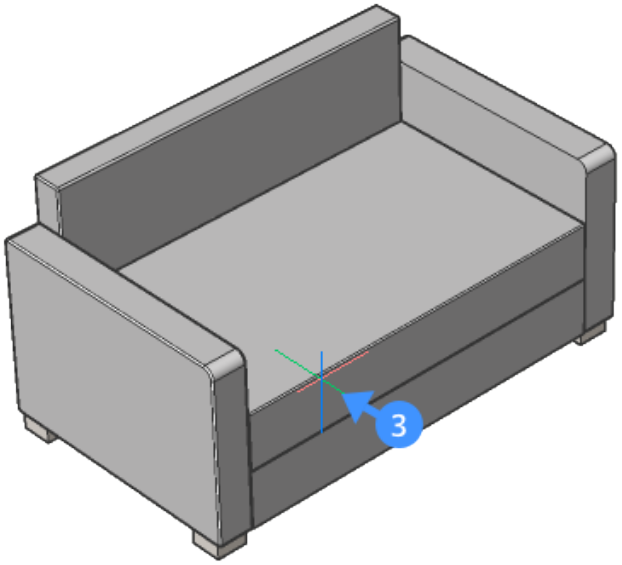
2 Browse to the folder where your .rfa family files are saved.

3 Select **Revit Family (*.rfa file)** (2) in the **Files of type** list.

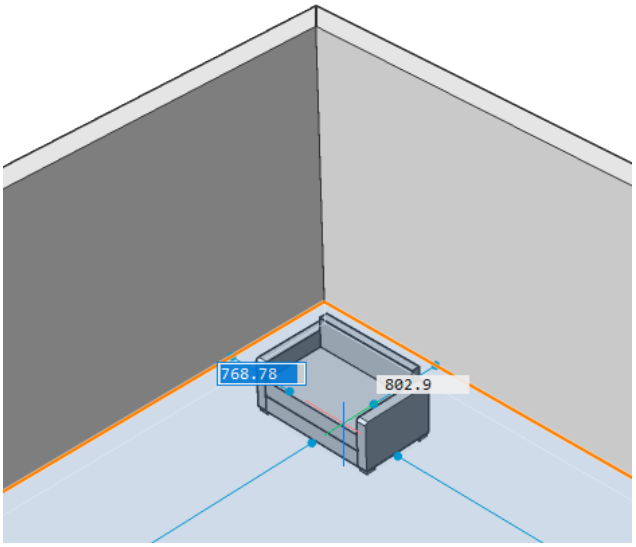



4 Select the .rfa file you want to import.

5 The selected file is attached to the cursor (3).




The following illustration shows inserting a Revit family sofa when both DYN and DUCS are on.



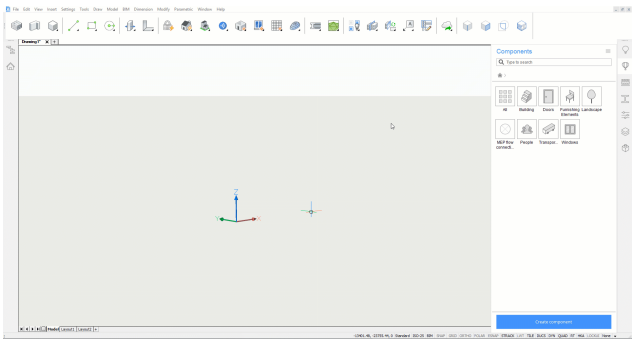

6 Do the following to place the selected RFA family file using the dynamic dimension fields:

- Hit the Tab key to select one of the dynamic dimension fields, then type a value and hit the Tab key to lock the dimension. The locked dimension will turn red.
- Repeat the previous step to lock another dynamic dimension field.
- Optionally hit the Tab key to adjust the dynamic dimensions.
- Press Enter to accept the current position.

**Using the Library panel**

1 To insert a RFA family folder in the **Library** panel, add the RFA root folder path to the Library directory path in the **Settings** dialog box. The content of the root folder and all subfolders will be available in the **Library** panel.

The following animated gif shows how to add the RFA root folder path to the components directory path.



The RFA family files are ready to insert in the model space.

If necessary, choose to **Generate Thumbnails** in the hamburger menu on the **Library** panel.

2 Drag and drop the RFA file from the **Library** panel into your model space.

## 20.6.6 Interference checking

### 20.6.6.1 Commands

INTERFERE

### 20.6.6.2 About interference checking

The **Interference Checking** tool is used during the modeling phase to find out conflicts in the model by comparing 3D solid models of building elements. Interference occurs when two 3D entities overlap in 3D space. In BricsCAD BIM, the Interfere tool helps you to find intersections between the entities that you have selected. You can also do the interference checking for all elements in your model at a time. Optionally, the tool creates interference solids after running the command. The newly created solids are placed on their own layer, as specified by the **InterfereLayer** variable. Running an interference check on existing models allows you to prevent the conflicts in your entire model.



For more information about this command, visit the Command Reference article INTERFERE.

### 20.6.6.3 Procedure: performing an interference check

1 Select the desired 3D solids to be checked for interference, then launch the INTERFERE command.

***Прим.:*** Multiple solids can be selected using the selection box or in the Structure Browser, depending on whether the selected entities appear when selected in the tree.

***The first set of entities is defined.***

***The Command line displays how many entities in total are set for interference checking e.g. Entities in set: 20.***

***Note that the Nested Selection option allows you to select 3D solid models that are inside blocks and external references (XRef) and once they are selected you can compare them against other entities in the selection set.***

2 Select the 3D solids in the model to add them to the second set of entities.

3 The entities in the first set are compared with the entities in the second set.

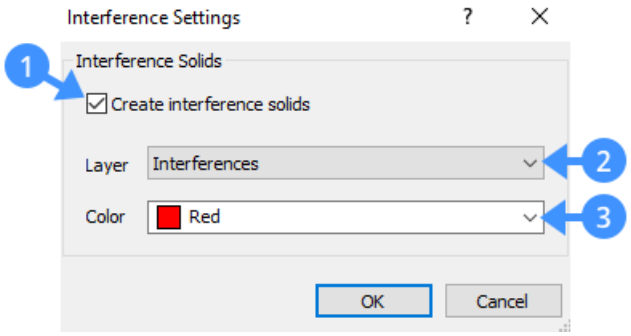
***Прим.:*** If you press the **Enter** key without selecting any solids for defining the second set of entities, the entities in the first set will be compared to each other.

4 Press **Enter** to run the interference check.

5 The interference volume is created for each collision and it is shown in a solid model.

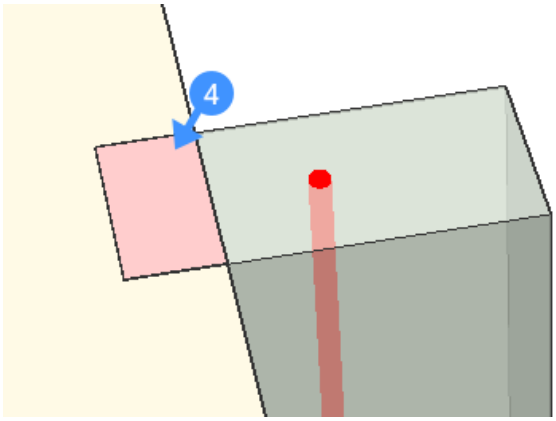
***Прим.:*** The solids of the interference volume are displayed in red since they are placed on layer **Interferences**. The default settings on the Interference Checking tool makes it easy to find the intersecting solids in the model. This setting can be changed using the prompt menu or the Command line. To open it, click the **Settings** option in the prompt menu or type "**S**" while the interfere command is still active. The **Interference Settings** dialog box displays. The dialog box allows you to configure interference settings, which are stored in variables.

***The following illustration shows the Interference Settings dialog box. The table below presents each component of the dialog box.***



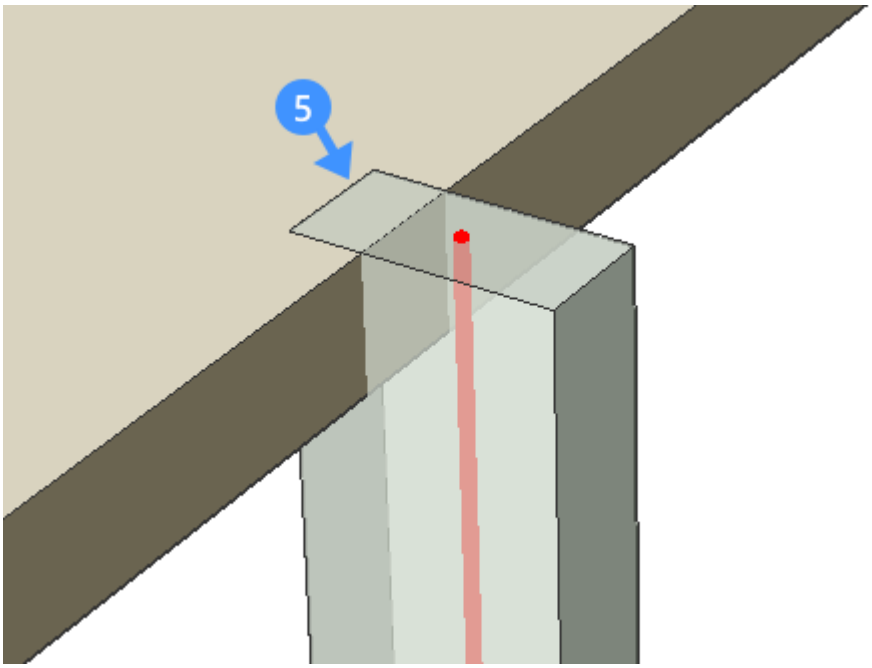

| | |
|---|---|
| **Create interference solids box (1)** | Determines whether a solid or region of the interference is created.<br>Check to create a solid of the interference volume or region of the interference area. |
| **Layer (2)** | Shows the current layer for interferences. |
| **Color (3)** | Shows the current layer color. |

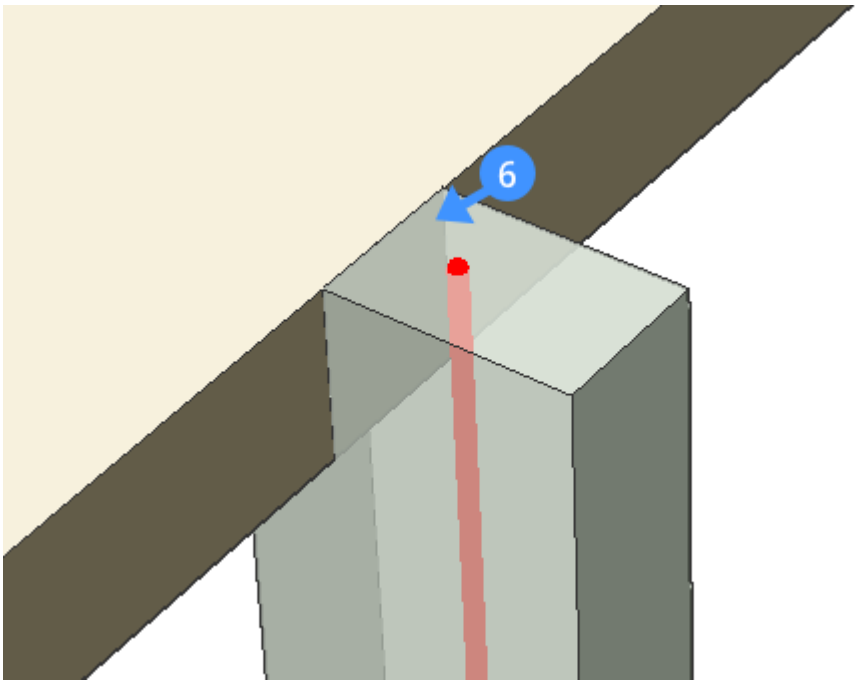
The following illustration shows the detected collision (4) between the floor slab and column.




6 To remove the undesired intersections, turn off the **Interferences** layer, and use one of the 3D solid modifying tools e.g. **BIM Drag** or **Subtract** on the interfering solid. This process will let you make the necessary corrections in your model.

In the following illustration, the created solid of the interference volume is shown (5). In the second image, the intersection is fixed by subtracting the slab geometry from the column.




7 Run the INTERFERE command a second time to check whether all interferences are fixed.

8 If there are no collisions left, the Command line reports that "**Solids do not interfere**".

### 20.6.7 Using Xrefs

#### 20.6.7.1 Commands

XREF, XATTACH, XOPEN, XCLIP

### 20.6.7.2 About

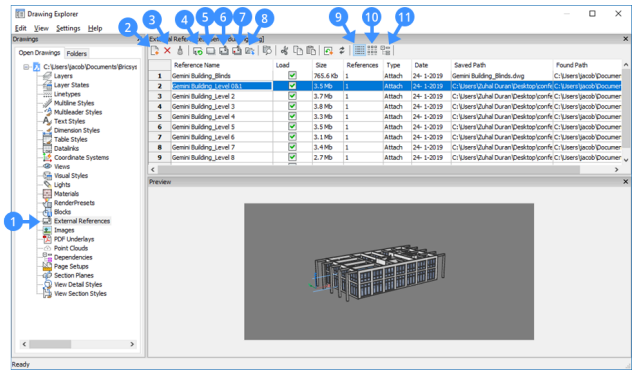
Using external references or Xrefs allows you to insert any model in the current BIM model. Instead of copying the model, only a reference to the source model is saved in the master model. You can attach multiple models to a single master model without increasing the size of the current model significantly. Further advantages of using the Xrefs are that the source models can be edited simultaneously by different members of the design team, and the master model can always be updated to show the latest version of each of the source models.

***Прим.:*** You can distinguish between XRef entities and entities in the current drawing using the XDWGFADECTL system variable. This system variable allows to fade the display of XRefs. You can give the values between 0 (no fading) and 90 to change the default value of the fading.

### 20.6.7.3 Procedure: managing XRefs in the current drawing

1 Open the **Drawing Explorer** dialog box and click the **External References** section.

The XRef preview pane appears, allowing the user to attach a model or manage the existing XRefs on the list. The following illustration shows the XRef preview pane.



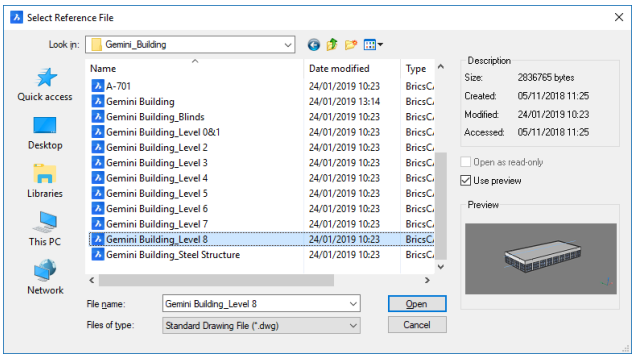

| | |
|---|---|
| **External References(1)** | Shows the External References section. |
| **Attach XRef (2)** | Attaches a model as an external reference. |
| **Detach XRef (3)** | Removes the selected XRef completely from the current model. |
| **Reload XRef (4)** | Reloads the selected XRef. This is useful when the source models have been changed, and you want to see the updated versions in the master model. |
| **Unload XRef (5)** | Unloads the selected XRef. This hides the XRef from the view. |

| | |
|---|---|
| **Bind XRef (6)** | Merges the source models permanently to the current drawing. Merging the XRef to the current drawing makes the XRef part of the drawing. References to the XRef disappear from the Drawing Explorer. |
| **Insert XRef (7)** | Converts XRefs into blocks. |
| **Open XRef (8)** | Opens the referenced drawing. |
| **Detail view (9)** | Displays the details of XRefs. |
| **Icon view (10)** | Displays icons of the XRefs. |
| **Tree view (11)** | Displays the relationship between XRefs in a tree-like view. |

2 To attach externally referenced models, click the **Attach XRef** tool button on the **Drawing Explorer** dialog box.

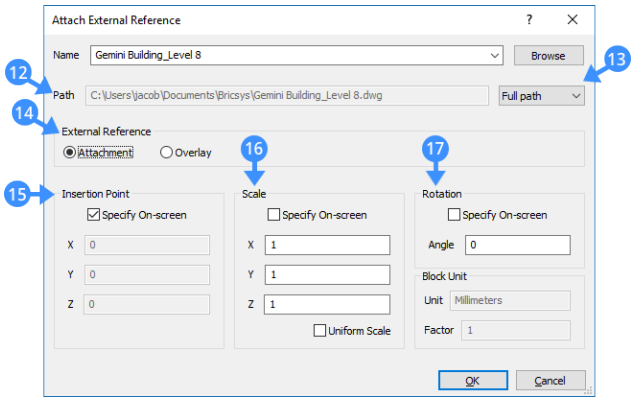
The **Select Reference File** dialog box displays.

The following illustration shows the **Select Reference File** dialog box. From this dialog box, select the DWG file you wish to attach in your main model, and then click **Open**.




Next, the **Attach External Reference** dialog box displays.

The following illustration shows the **Attach External Reference** dialog box. The table below presents each option of the **Attach External Reference** dialog box. From this dialog box, select the options you want to apply, and then click **OK**.




| | |
|---|---|
| **Path field (12)** | Displays the full path of the selected drawing. |
| **Path type (13)** | Includes **Full path**, **Relative path**, and **No path** options.<br>**Full path**: When you move your drawing containing an XRef to another location, BricsCAD will still search for the XRef in the previous location and it will not find this XRef.<br>**Relative path**: When you move your drawing containing an XRef to another location, the new file path will be created in the new location. The file folder will set up correctly using the same folder structure as the first one. |
| **External Reference type (14)** | Includes **Attachment** and **Overlay** reference types.<br>**Attachment**: Attaches all XRefs, including nested XRefs. For example, you have the first level of XRef “X” and this contains nested XRefs “Y” and “Z”. When the attachment option is selected the first level of XRef and its nested XRefs Y and Z will be attached to the drawing.<br>**Overlay**: Attaches only the first level of xref (X), nested xrefs (Y, Z) will be not loaded. This can also be used to prevent circular references. |
| **Insertion Point (15)** | Allows the user to specify the Insertion point on the screen or in the X-, Y-, Z- fields. |
| **Scale (16)** | Allows the user to specify the Scale on the screen or in the X-, Y-, Z- fields. Check the **Uniform Scale** option to specify the scale in the X- field. |

| | |
|---|---|
| **Rotation (17)** | Allows the user to specify the Rotation angle on the screen or in the **Angle** field. |

Note that the **BIM XReference** option from BIM Classify allows you to classify the model as an external reference. The identification of information will not explicitly be represented in the current model or in the project database. Such information appears after attaching the model to another project. To do so, follow the above step.

The following illustration shows the BIM category in the **properties** panel after attaching the model that has been classified as an XRef.




3 To detach an existing XRef, select the XRef you wish to detach in your main model, and then click the **Detach XRef** tool button on the **Drawing Explorer** dialog box.

The selected XRefs are detached.

***Прим.:*** Press and hold the CTRL key to select multiple XRefs or to unselect a selected XRef.

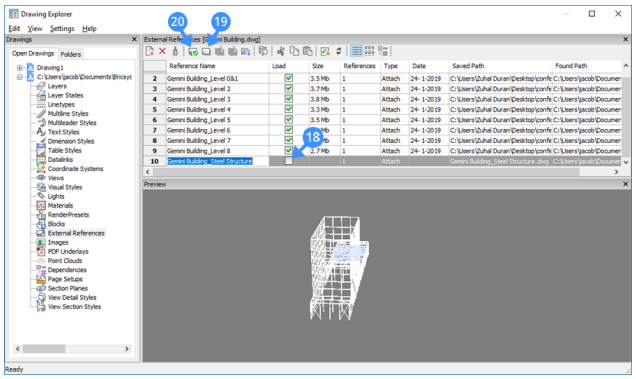
4 Do one of the following to unload and reload an existing XRef.

- Click the **Load** (18) column of a loaded XRef.

  This XRef is unloaded in your main model.

  To reload the XRef check the load box of this XRef.
- Select the XRef, then click the Unload XRef (19) tool button on the Details toolbar.
- Select the unloaded XRef, then click Reload XRef (20) tool button.

  The following illustration shows the **Load** column (18), Unload XRef (19) and Reload XRef (20) tool buttons on the **Drawing Explorer**.




5 To modify the attachment type, click the **Type** column of an existing XRef and switch from **Attach** to **Overlay** or vice versa. The same process can be followed for modifying the **Saved path** of an XRef and **Found path** of an overlay.

6 Click **X** to close the dialog box. Notice that the changes that have been made appear after you close the **Drawing Explorer**.

## 20.7 Проектная документация

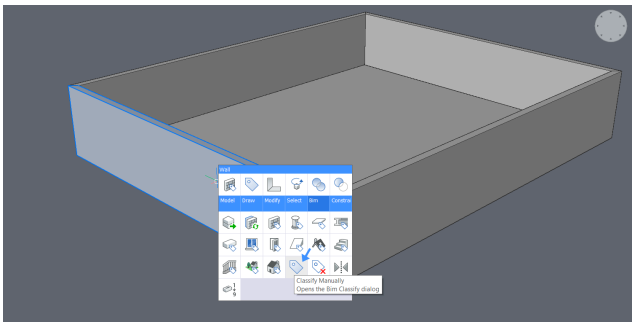
### 20.7.1 Annotations in 3D

#### 20.7.1.1 Commands

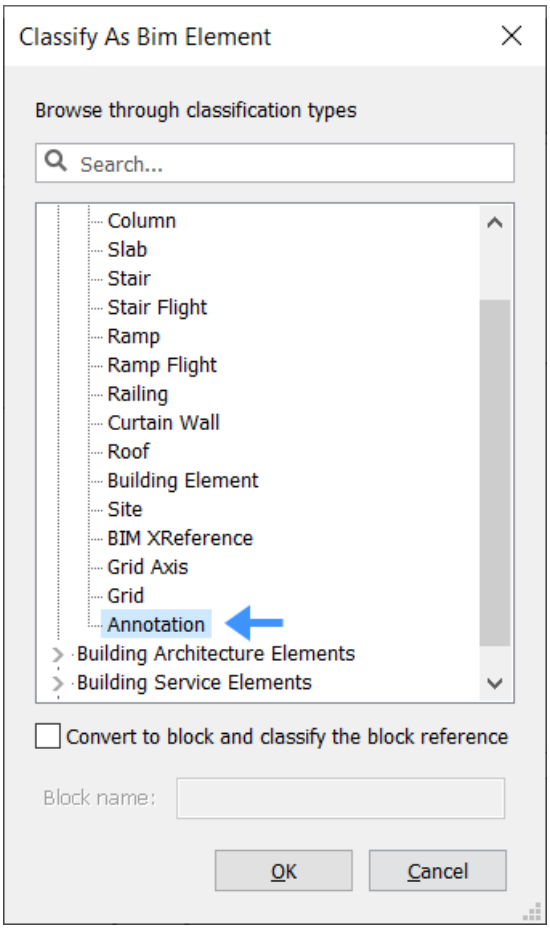
BIMTAG, BIMCLASSIFY

#### 20.7.1.2 BIM classify as an annotation

1 Select a solid and select **Classify Manually** in the Quad or classify as other after launch the BIMCLASSIFY in the Command line:

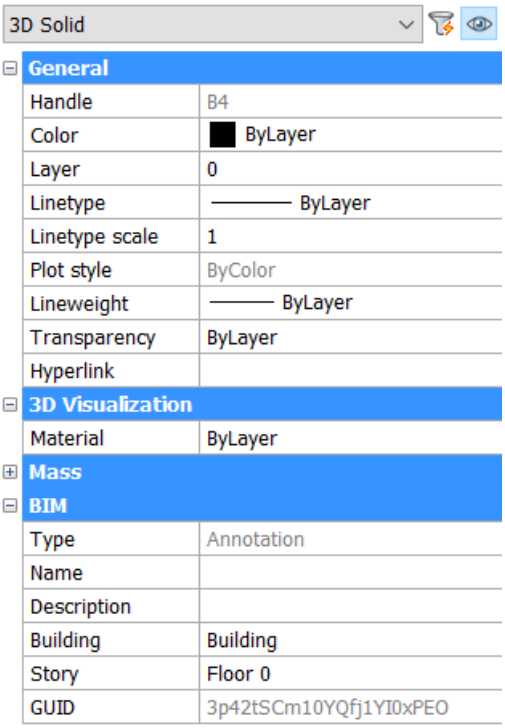


2 Choose to classify as **Annotation** in the folder of **Building Core Elements** and click **OK**:




3 When you open the Properties panel of the selected solid you can change the **Name**, **Description**, **Building** and **Story**.

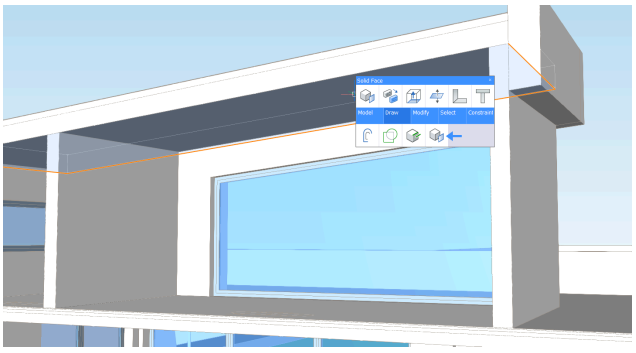
***Прим.:*** The property GUID cannot be changed because it has a unique, automatically generated string.




### 20.7.1.3 Show added information of a finishing material

In this procedure, you will add design information to a concrete roof structure.

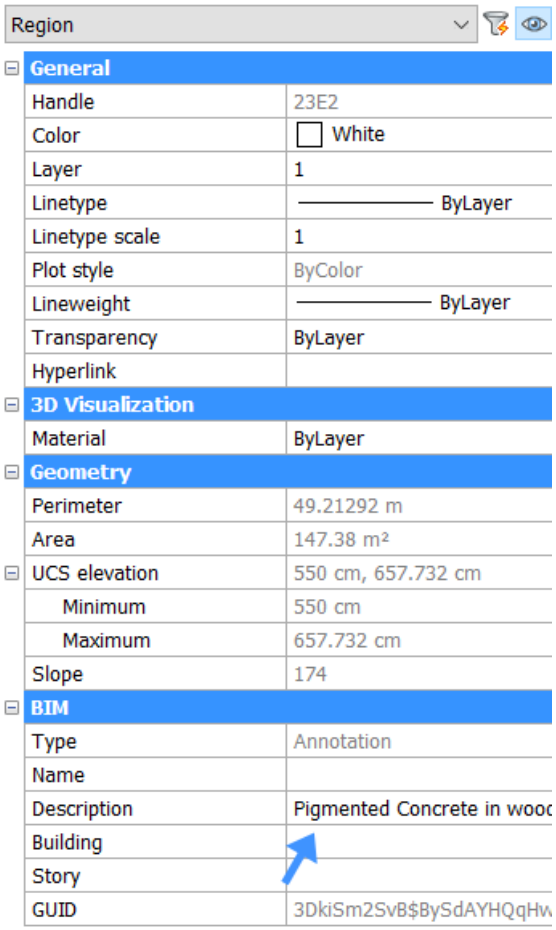
1 Use the EXTRACTFACES command under **Draw** in the quad, to extract the face of the **Roof** entity. A region is now created.



2 Classify the region as an **Annotation** as seen above.

3 Enter the desired text in the **Description** field (found in the Properties panel):

***Прим.:*** By default, the BIMTag references the **Description** property.



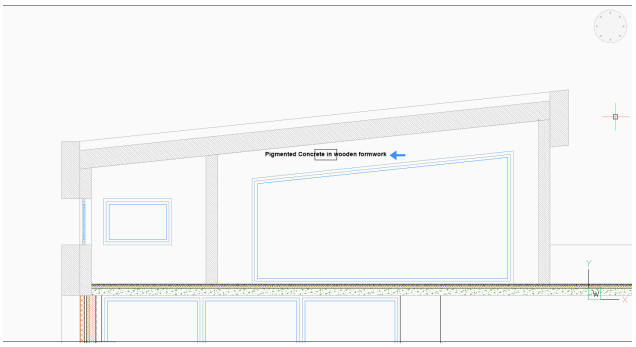

4 Make a section through the region (with BIMSECTION command) and select **Generate Section**( ) from the Quad:



5 Open the generated section drawing.

6 Type BIMTAG in the Command line OR select **Tag Automatically** in the **Annotate** tab of the Ribbon.

The text that you added is now in the 2D drawing:



### 20.7.1.4 Show the surface area of an extracted surface

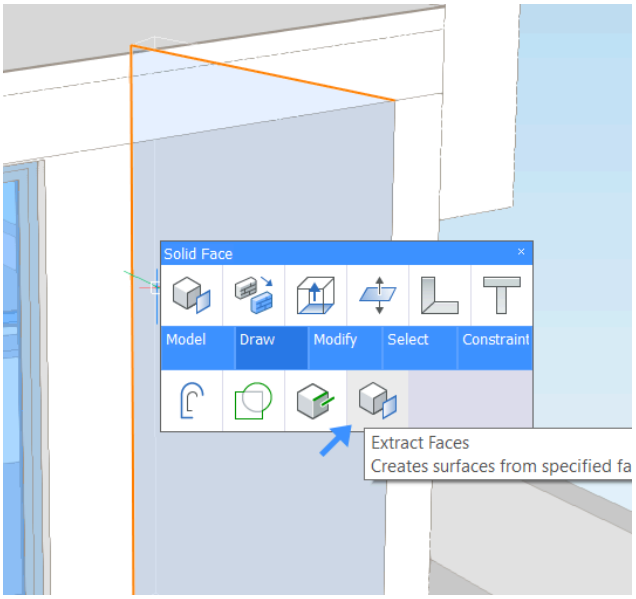
This procedure will teach you how to display other properties of an entity classified as an **Annotation** object.

***Прим.:*** The annotation tag pulls information from the **Description** property by default.

For more information about tagging, see the article BIM Tags.

1 Select the face you want to extract the information from.

2 Select **Extract Faces** from the Quad. A region is now created.




3 Classify the region as an **Annotation** as above.

4 Make a section through the solid (with BIMSECTION command) and select **Generate Section** ( ) from the Quad.

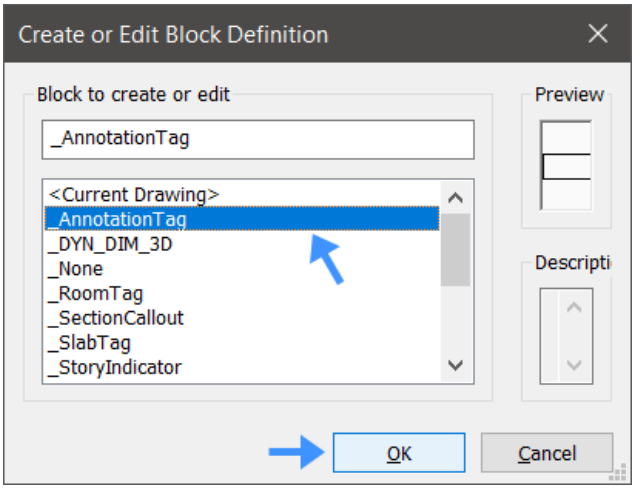
5 Open the generated section drawing.

6 Type BIMTAG in the Command line OR select **Tag Automatically** in the **Annotate** tab of the Ribbon.

***Прим.:*** The tag appears empty because it refers to the **Description** property by default.

7 Launch the BEDIT command.

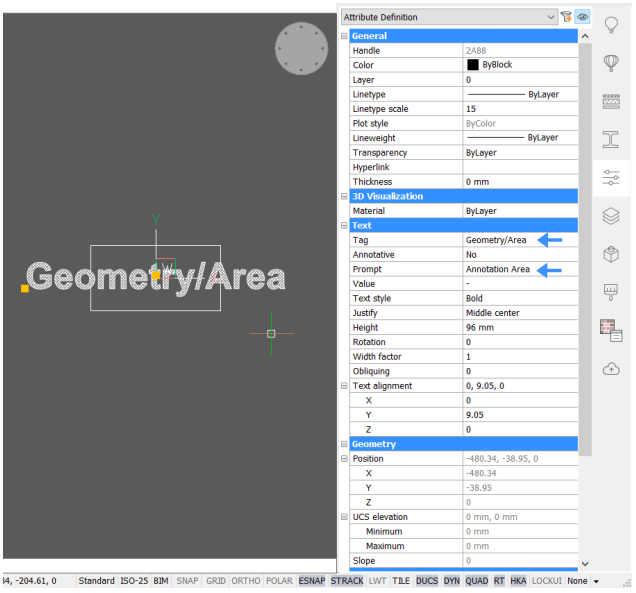
8 Select **_AnnotationTag** and click **OK**.




9 Select the attribute definition, in the properties panel.

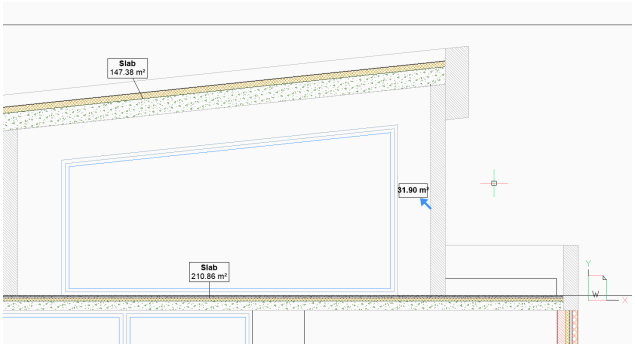
10 Edit the **Tag** property, change it to **Geometry/Area**.

11 **(Optional)** You can also change the **Prompt** property.




12 Enter BCLOSE in the Command line and click **Save**.

13 Select the viewport and pick **Generate Section** ( ) from the Quad. The area of the region is now displayed in the tag.




## 20.7.2 BIM sections

### 20.7.2.1 Commands

BIMSECTION

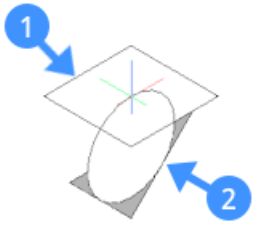
### 20.7.2.2 Defining a section

In BricsCAD BIM, the **Define Section** tool allows you to create BIM section entities. The BIM section allows you to section a BIM model by a plane and thus view the interior details.



For more information about this command, visit the Command Reference article BIMSECTION.

When defining a BIM section, the section tracker is attached to the cursor.




The section plane (1) lies in the XY-plane of the current coordinate system (WCS or UCS). The view direction (2) is in the negative direction of the current coordinate system.

***Прим.:***

- If Dynamic UCS (DUCS) is ON, the section plane aligns to the face of the 3D solid under the cursor. Hit the TAB key to select an obscured face.
- You can press the Shift key to lock a highlighted plane, allowing you to specify the definition point for the BIM section entity outside the face of the solid.
- The section line is parallel to the X-axis of the coordinate system (WCS, UCS or DUCS).

**General procedure to create a BIM section**

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

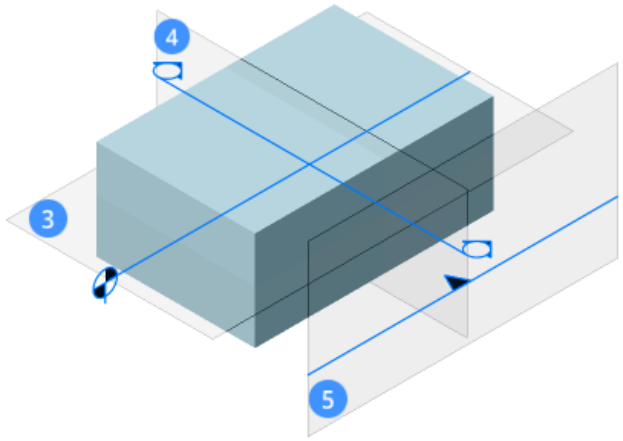
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

2 Do one of the following:
   - To create a plan section, click a point anywhere outside the model.
   - Hover the cursor over the face of a 3D solid, parallel to the section plane you want to create.
   - The section tracker aligns to the face under the cursor, on the condition that DUCS is active. Left-click.
   - **(Optional)** Hit the Shift key to lock the highlighted plane. This allows you to start from a point outside the selected 3D solid face. Next, left-click.
3 The initial section plane displays dynamically and the 3D model is clipped accordingly.
4 You are prompted: Specify distance:

   Do one of the following:
   - Type a value in the dynamic dimension field to offset the section from the initial position.
   - Left-click when the section plane is at the location you want it to be.

   The BIM Section entity is defined.

   The Section Type depends on the direction of the section plane (horizontal or vertical) and whether it intersects with the model.



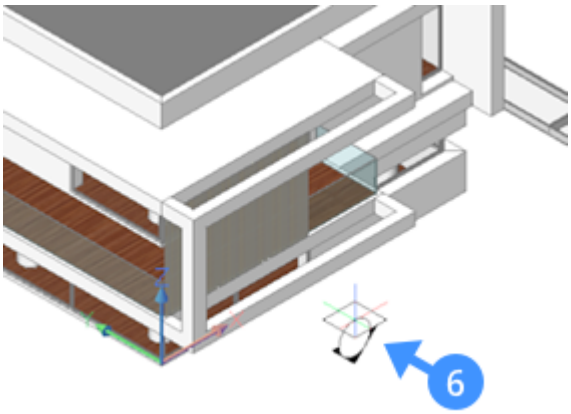

| | |
|---|---|
| Plan (3) | Displays horizontal section plane. |
| Section (4) | Displays a vertical section that cuts through the model. |
| Elevation (5) | Displays the exterior elevation view. |

**Creating a plan section**

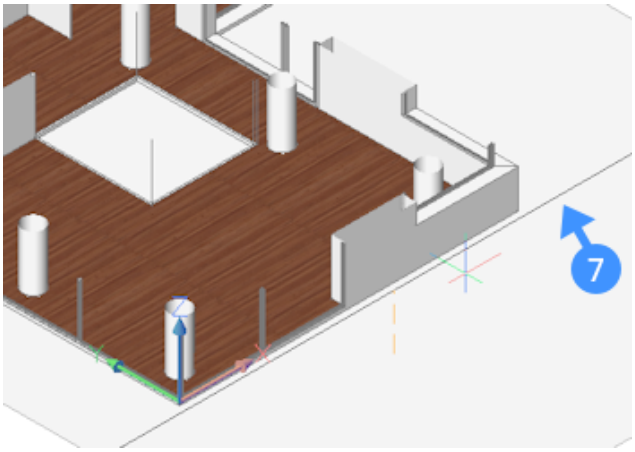
***Прим.:*** A plan section is best defined in the WCS (World Coordinate System).

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

   You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:
2 Place the section tracker near the 3D model and click (6).




The initial section plane lies in the XY-plane (Z = 0) of the current coordinate system (e.g. the WCS). The section plane displays dynamically and the 3D model is clipped accordingly (7).




You are prompted: Specify distance:

3 If Dynamic Dimension is ON, the current distance displays in a dynamic dimension field.

4 Do one of the following:
- Move the cursor and click when the section plane is at the desired height.
- Use an Entity Snap to specify a point.
- Type a value to define the offset from the initial position of the section plane.

The plan section is defined.

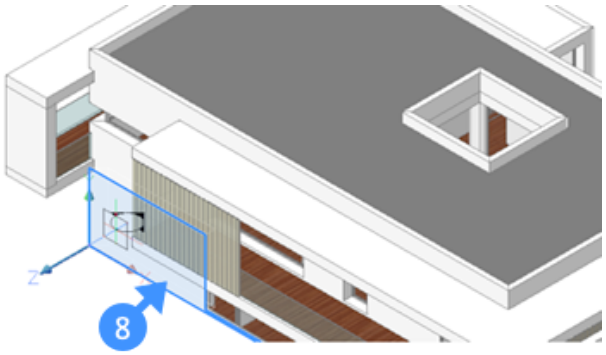
**Creating a cross section**

Make sure Dynamic UCS (DUCS) is enabled.

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

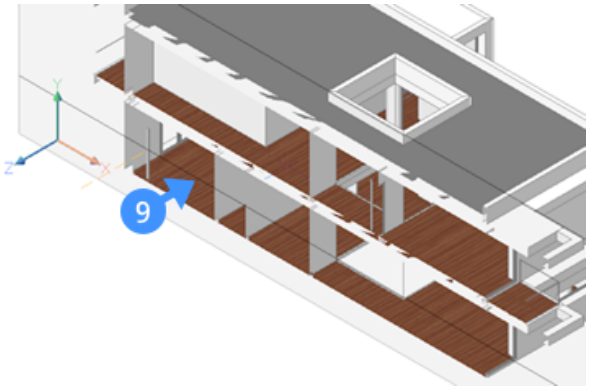
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

2 Place the section tracker over a 3D solid face (8) parallel to the cross-section to be defined.




3 Click when the face highlights and move the section tracker inside the 3D model.

4 The section plane displays dynamically and the 3D model is clipped accordingly (9).




You are prompted: Specify distance:

If Dynamic Dimension is ON, the current distance displays in a dynamic dimension field.

5 Click when the section plane is at the desired location or specify a distance.

6 The cross-section is defined.

7 **(Optional)** Click the section entity and hold down the left mouse key a little longer to display the **Manipulator**. Drag an axis of the manipulator to move the section.

8 **(Optional)** Click the Arrow grip to flip the view direction.

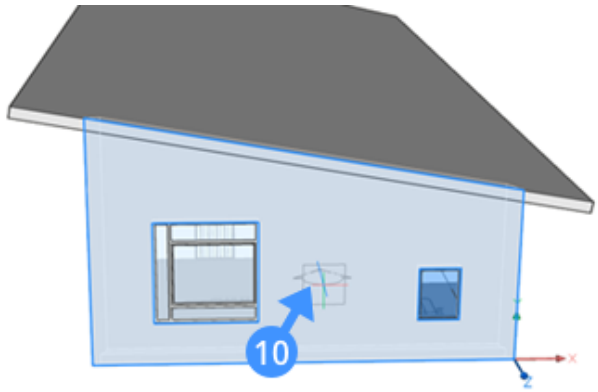
**Creating an elevation**

Make sure Dynamic UCS (DUCS) is enabled.

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

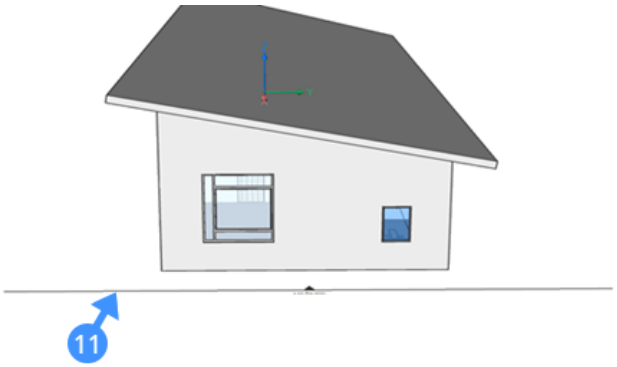
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

2 Place the section tracker over a 3D solid face (10) which is parallel to the elevation to be defined and click when face highlights.




3 You are prompted: Specify distance:

4 Move the section tracker outside the 3D model and click to define the elevation (11).




Vertical sections outside the model (no geometry is sectioned) get the Elevation section type automatically.

### 20.7.2.3 Editing the default properties of a BIM section

The default properties of a BIM section entity are copied from the section entities in _SectionSettings.dwg in the **BIM/Sections** subfolder of the **Support** folder.

1 Select the BIM section in the drawing or the **Structure panel**.

The section properties display in the Properties panel.

2 Define the properties of the BIM section entity.

| General | |
|---|---|
| Handle | 14F2 |
| Color | ByLayer |
| Layer | 0 |
| Linetype | ByLayer |
| Linetype scale | 1 |
| Plot style | ByColor |
| Lineweight | ByLayer |
| Transparency | ByLayer |
| Hyperlink | |
| History | Current |
| **3D Visualization** | |
| Material | ByLayer |
| **Section Entity** | |
| Name | Floor 0 |
| State | Plane |
| Live section | No |
| Clip display | No |
| Use lids | Yes |
| Plane transparency | 70 |
| Plane color | 9 |
| Destination file | Plans |
| **Geometry** | |
| Elevation | 120 cm |
| Top plane | 15.86 m |
| Bottom plane | 15.86 m |
| Vertex | 1 |
| Position | -4122.5, 7000, 1200 |
| X | -4122.5 |
| Y | 7000 |
| Z | 1200 |
| UCS Elevation | 899.57 mm, 139.957 cm |
| Minimum | 899.57 mm |
| Maximum | 139.957 cm |
| **BIM** | |
| Name | |
| Description | |
| Section type | Plan |
| Callout position | Start |
| Initial scale | 1:50 |
| Layer state | |
| Apply layer state | Off |
| Project section | On |
| Generate tags | On |
| Entity property sets | |

**Section Entity**:

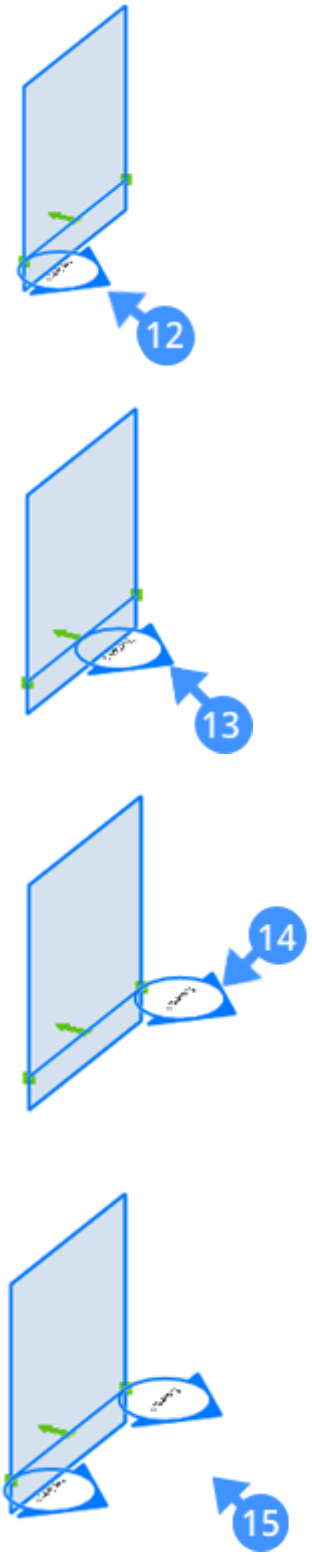
| | |
|---|---|
| **Name** | Type a name. |
| **State** | Click the field and select a state. The selected state determines how the section result will be clipped.<br>The options are: **Plane** (default), **Boundary** and **Volume**. |
| **Live section** | Sets the live section property of the section plane. Only one section entity can have the **Live section** property **ON**. |

**Geometry**:

| | |
|---|---|
| **Elevation** | Defines the Z-value of the start point of the section plane. |
| **Top plane** | Defines the top plane of the section volume. |
| **Bottom plane** | Defines the bottom plane of the section volume. |
| **Vertex** | Defines to coordinates of the control points of the section plane. The number of vertices depends on the current section plane state.<br>Plane: 2 vertices<br>Boundary and Volume: 4 vertices.<br>Select the field, then click the arrow buttons to select a vertex. An X indicates the selected vertex in the model. |

**BIM**:

| | |
|---|---|
| **Name** | Type a BIM name. |
| **Description** | Type a description. |
| **Section type** | Sets the BIM section type. |

| | |
|---|---|
| **Callout position** | Defines the position of the callout(s).<br>The callout block and default position depend on the section type.<br>The options are: **Start** (12), **Mid** (13), **End** (14) and **StartEnd** (15).<br><br><br>Click the green arrow ( ) to flip the view direction of the section. |

| | |
|---|---|
| **Initial scale** | Sets the initial scale of the papers pace viewport in the generated drawing. |
| **Layer state** | Sets the layer state to be used to calculate the section. |
| **Apply layer state** | Determines whether the layer state will be used to calculate the section or not.<br>Applying a layer state allows to control which layers are ON/OFF when calculating the section result. |
| **Project section** | Controls the display of the section callouts and section line in section drawings which are perpendicular to this section. |
| **Generate tags** | Controls whether BIM tags are generated automatically. |
| **Entity property sets** | Controls whether 'per instance' entity properties can be used to generate tags.<br>Click the **Browse** (…) button, then select the property sets in the **Enable per-instance property set** dialog box. |

### 20.7.2.4 How to toggle the clipped display property of a BIM section entity

The **Clip display** property of the **BIM** section temporarily clips 3D models. The **Clip display** can be used in all modeling and editing operations on all drawing content.

The following illustration shows the 3D model of a villa with a plan section (16) for the ground floor plan.

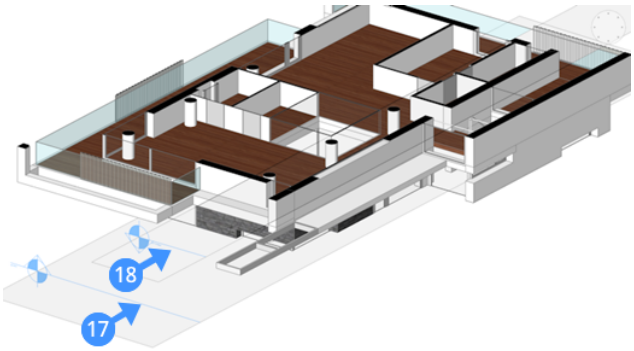



To clip the 3D model, do one of the following:

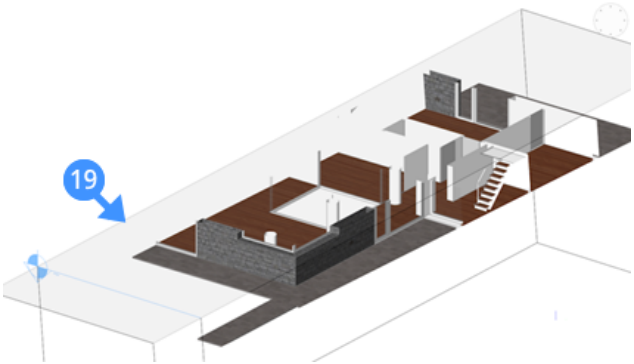
- Double click the section entity.
- Select **Clip display** from the Quad.
- Select the section plane and set the **Clip display** property to **Yes** in the properties panel.
- When **RT** (Rollover Tips) is selected in the **Status** Bar, place the cursor over a section entity to set the **Clip display** property.

***Прим.:*** **Clip display** can be on for multiple section entities simultaneously.

In the following illustration, **Clip display** is turned ON for two BIM section entities (17,18). These section entities are in opposite view directions: section plane 18 is a plan section, section plane 17 is a reflected ceiling section.




Option: Set the **State** property for the current section to **Volume** in the properties panel.



A clipped section shown in the **Volume** state (19).

#### 20.7.2.5 Converting a section entity to a BIM section

The BIMIFY command converts a section entity to a BIM section.

1 Launch the BIMIFY command.

1 You are prompted: Run Bimify on entire model or [Advanced] <Model>:

2 Choose the **Advanced** option.

The **Bimify** dialog box appears.

3 Choose **Select entities**, then click the **Select** button.

The **Bimify** dialog box closes.

4 Select the section entity, then right click.

5 The **Bimify** dialog box reopens.

6 Click the **OK** button.

### 20.7.3 BIM tags

#### 20.7.3.1 Commands

BIMTAG

#### 20.7.3.2 Source files

The following source files are used to create BimTags in a generated BIM section drawing:

- **_SectionTag.dwg**: contains the tag blocks and multileader styles.
- **_TagTypeToStyle.xml**: links a BIM entity type to a multileader style.

Both files sit in the folder:

C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\Support\Bim\Sections

***Прим.:*** when assigning tags, BricsCAD needs to access the 3D model. For this reason it may take several seconds to place the first tag, if the 3D model is not currently opened, especially in large models. If you want to avoid this waiting time, consider having the 3D model open while placing tags.

#### 20.7.3.3 Assign BIM tags manually

The BIMTAG Manual command allows you to select the BIM element that you wish to create a BIM tag for.

For more information about this command, visit the Command Reference article BIMTAG.



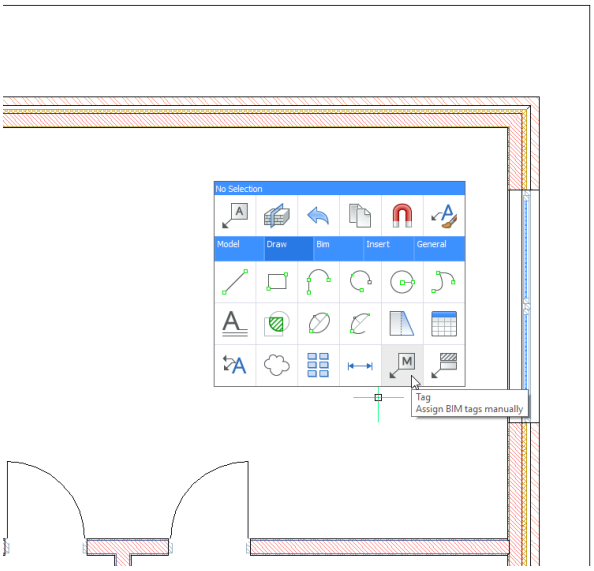
**Manually create a BIM tag**

1 Open the section drawing for which you wish to create BIM tags.

***Прим.:*** For more information about generating section drawings, please read the BIM article **Generating Drawings** first.

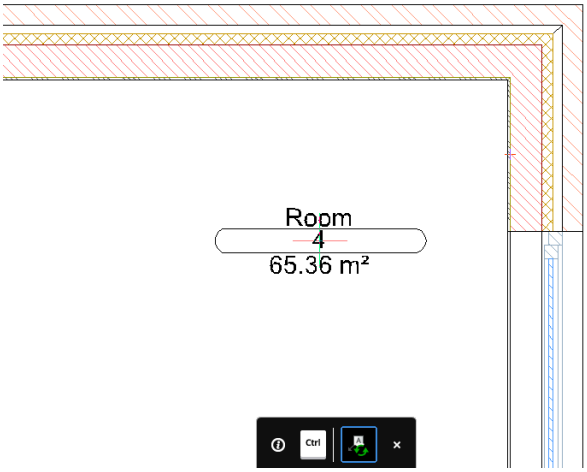
2 With no selection, select **Tag Assign BIM tags** manually from the Quad or enter BIMTAG followed by M

into the Command line to choose the manual option.




You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

3 Select a point on a line of a BIM element to generate a tag for it. The tag will appear. If the point selected is shared between multiple BIM entities, you can use the CTRL key to cycle through the different possible tags.



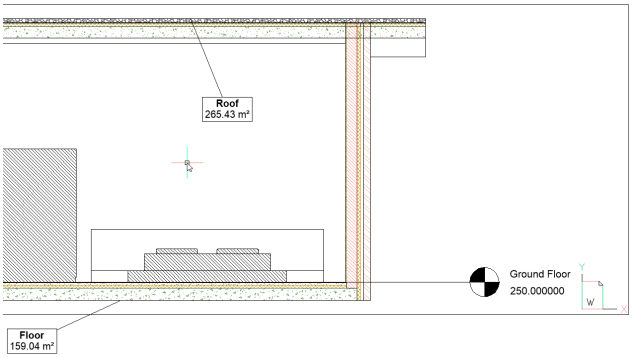

You are prompted: Place the tag [Next element]:

4 Move your cursor in order to adjust the position of the tag and left-click to place it.

5 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

6 Repeat steps 3 and 4 for all of the entities you wish to be tagged. If you do not require more tags, press the ESC or ENTER key to complete the command. If you want all elements to be tagged, you can use the BimTag Select all entities command which we will explore next.

7 The same steps above can be used to tag a floor slab or a roof.




### 20.7.3.4 Assign BIM tags automatically

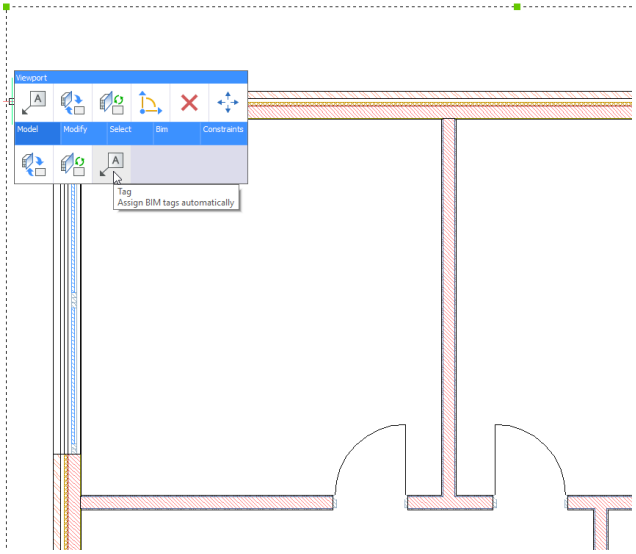
If you want all BIM elements in a section drawing to be tagged it is easiest to use the BIMTAG Select all entities command. This command automatically assigns BIM tags to all BIM entities inside a viewport. If the model is altered or new elements are added, the BIM tags can be updated and new tags added.

For more information about this command, visit the Command Reference article BIMTAG.

**Automatically create BIM tags**

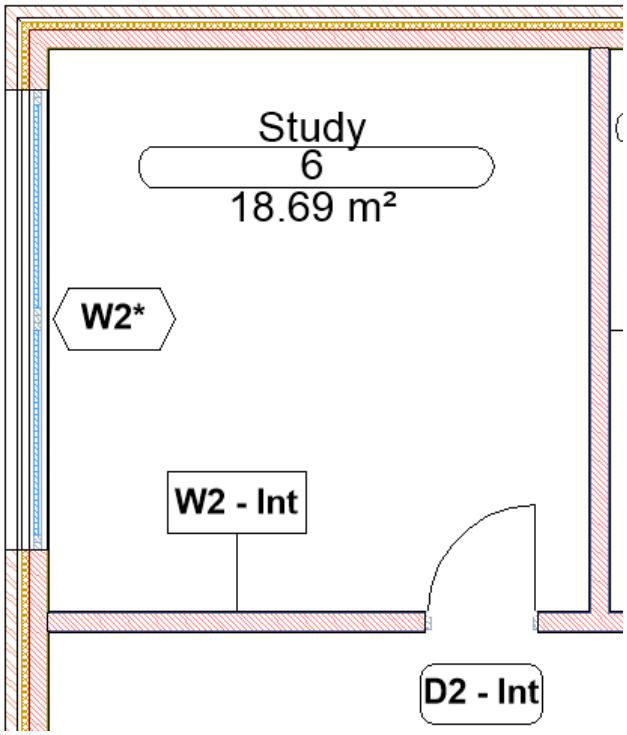
1 Open the section drawing on which you wish to create BIM tags.

2 Select a viewport containing the section which you will add BIM tags to.




3 Select **Tag Assign BIM tags automatically** from the Quad or enter BIMTAG into the Command line.

The BIM tags have now automatically been generated for each of the BIM entities in the viewport.



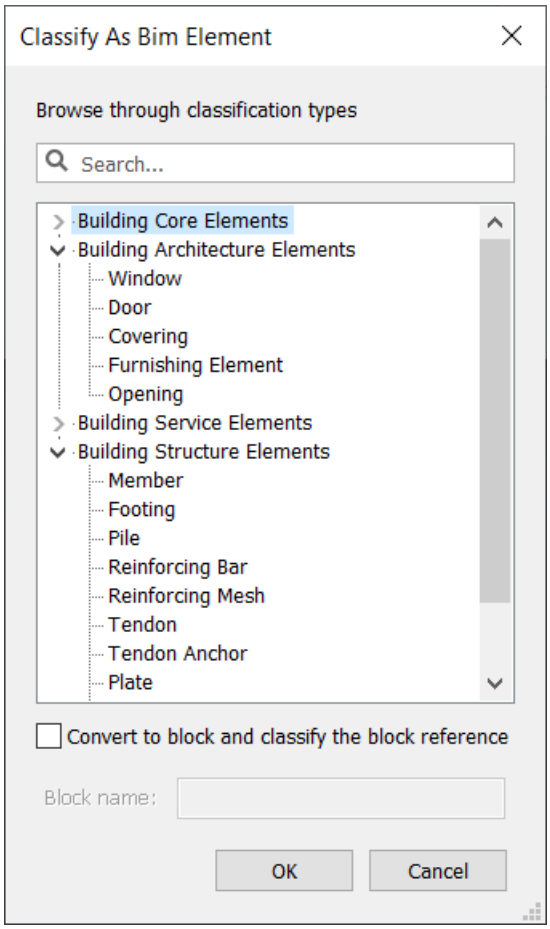

You are prompted: Select BIM section viewport [Manual]:

4 If you wish to automatically create BIM tags in another viewport on the sheet, select the viewport and these will be generated. If you do not require any further BIM tags, press Esc to complete the command.

### 20.7.3.5 Create the source data for a BIM entity type

By default the source data for the following BIM entity types already exist; **Annotation**, **Beam**, **Column**, **Curtain Wall**, **Door**, **Plies**, **Railing**, **Ramp**, **Roof**, **Room**, **Slab**, **Stair**, **Wall**, and **Window**.

Available BIM entity types are listed in the **Classify As Bim Element** dialog box. To open this dialog box, choose the **Other command** option of the BIMCLASSIFY command.




**Create the tag block and multileader style**

To create the source data for a Bim Member entity type:

1 Type SUPPORTFOLDER in the Command line and press Enter.

2 Once the folder has opened, navigate to the **BIM** > **Sections** folder and open the _SectionTag.dwg file.

3 Launch the EXPBLOCKS command.

4 The **Drawing Explorer / Blocks** dialog box will display.

5 Select the **_AnnotationTag** block (or any other existing block that you want to use as a template).

6 Right-click and choose **Copy** from the context menu.

7 Right-click and choose **Paste** from the context menu.

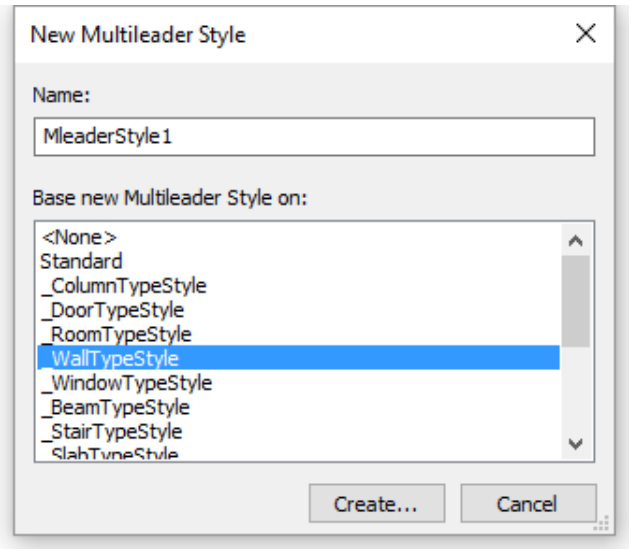
8 Click **Copy**, but keep both on the **Copy/Paste** dialog box.

9 Rename the copied block to **_MemberTag**.

10 Select **Multileader Styles** in the **Drawing Explorer** dialog box.

11 Click the New ( ) tool button.

The **New Multileader Style** dialog box will display:




12 In the Name field type _MemberTypeStyle.

13 Select a source style in the **Base new Multileader Style on** list.

14 Click the **Create...** button.

15 **(Optional)** Edit the **Leader** properties.

16 Click the **Content** tab.

17 Under **Block options**, click the **Source Block list** button and select **_MemberTag** from the list.

18 Close the **Drawing Explorer** dialog box.

19 Save _SectionTag.dwg.

**Link the entity type to its multileader style**

1 Type SUPPORTFOLDER in the Command line, then go to the **BIM** > **Section** folder and open _TagTypeToStyle.xml.

Use an ASCII text editor, e.g. Notepad.

2 Add the following right before </TagTypeToStyle>:

```
<tag type="BIM_MEMBER">
<style
name="_MemberTypeStyle"
autoPlacement="true"
autoRotation="true"
offset="1"
/>
</tag>
```

3 Save the file.

4 Note that this style has three options:

- **autoPlacement = "true"** will make sure tags are automatically generated when using the automatic BIMTAG option. If this option is set to false, then this tag type will not be generated automatically (you can still generate these tags manually).
- **autoRotation = "true"** will rotate the tag along the main axis of the tagged entity. If set to false,

the tag will be placed horizontally.

- **offset** will offset the tags perpendicular to the main axis of the tagged entity.

### 20.7.3.6 Customize tag blocks

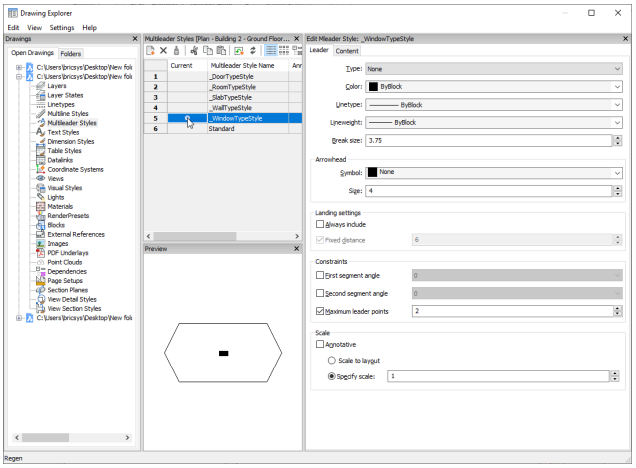
The content of a Bim Tag block consists of:

- Layout entities Linework, fixed texts, etc.
- Attributes Bim entity properties.

***Прим.:*** When you execute the BIMTAG command, BricsCAD checks whether the source data (tag blocks and their multileader styles) already exist in the Bim section drawing. If not, the source data is copied from the _SectionTag.dwg file. Otherwise, the tag block definitions and multileader styles in the section drawing are used. Therefore, editing tag blocks in the _SectionTag.dwg file only affects the creation of new Bim tags the first time BIMTAG is executed in a section drawing. If you edit tag blocks in a section drawing it affects Bim tags in that drawing only.

### 20.7.3.7 Customize tag blocks for the current drawing only

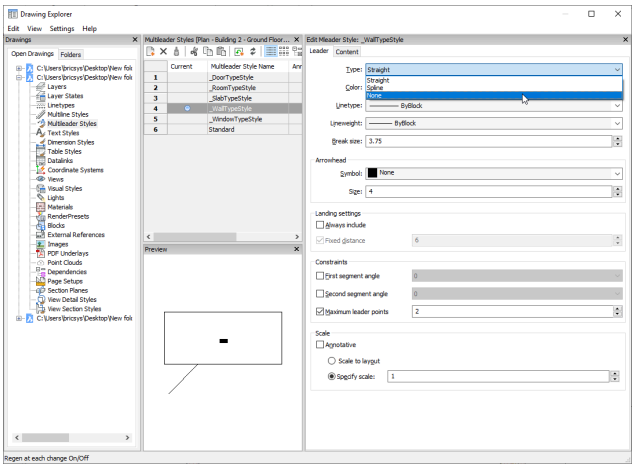
1 In the section drawing for which you wish to customize the BIM Tags, type MLEADERSTYLE in the Command line. The drawing explorer will open in the **Multileader Styles** tab.



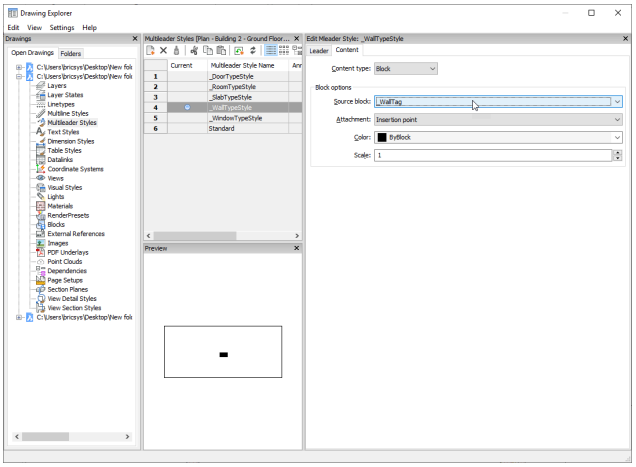

Each BIM tag is a multileader. The multileader style depends on the type of BIM element being tagged. All the multileader styles in the drawing are listed in the drawing explorer.

2 Select **_WallTypeStyle** from the list to see a preview of what the tag will look like. You can change the tag style from here.

3 Change the **Leader Type** from **Straight** to **None**. Now there will be no line connecting the tag to the wall element.



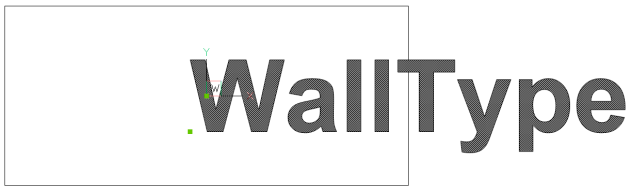
4 In the **Content** tab, the **Source block** is called **_WallTag**. Remember this and now close the drawing explorer.

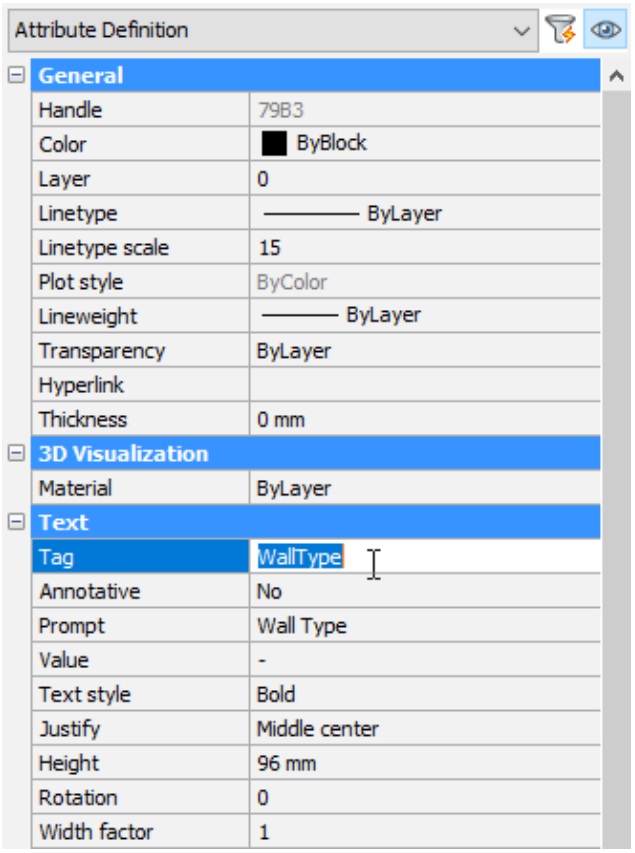


5 In the Command line type BEDIT and select **_WallTag** from the list. Click **OK**.

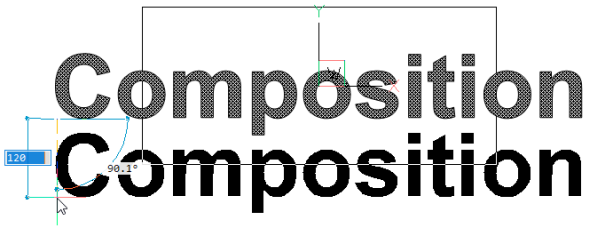
6 Select the **WallType** text and **Tag** value of the properties panel.

7 Replace **WallType** with **Composition** and press ENTER (see the next procedure for the syntax of other property types). Now the attribute text will display **Composition**.




8 Hover over this attribute and select **Copy** from the Quad. Select a base point to paste the copied attribute below the current one.



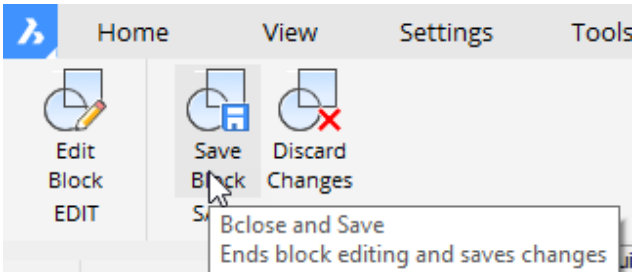

9 Repeat steps 5 - 7 to change the new attribute **Tag** value (**Quantity/Thickness**).

10 Delete the rectangle.

# Composition Quantity/Thickness

11 Select the **Save Block** icon from the **Block Editor** tab of the ribbon.

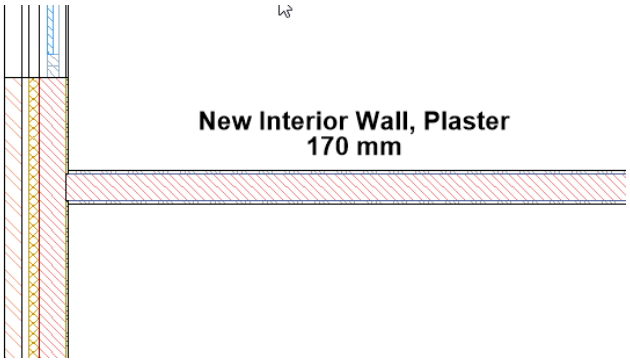
Your section viewport will now display.




12 Highlight the viewport and select **Tag Assign BIM tags automatically** from the Quad.

You are prompted: Tags already have been assigned to the section. Choose an update option [Update existing/update existing and add New/Regenerate all] <Regenerate all>:

13 Click hit the ENTER key to accept the default option of **Regenerate all**. The wall tags will be regenerated with the new style.




### 20.7.3.8 Edit the source data to customize BIM tags for all section drawings

1 With any drawing open, type SUPPORTFOLDER in the Command line followed. Hit Enter.

2 Once open navigate to the **BIM** > **Sections** and open the _SectionTag.dwg file.

3 Launch the BEDIT command.

4 Select the tag block you want to edit, then click the **OK** button.

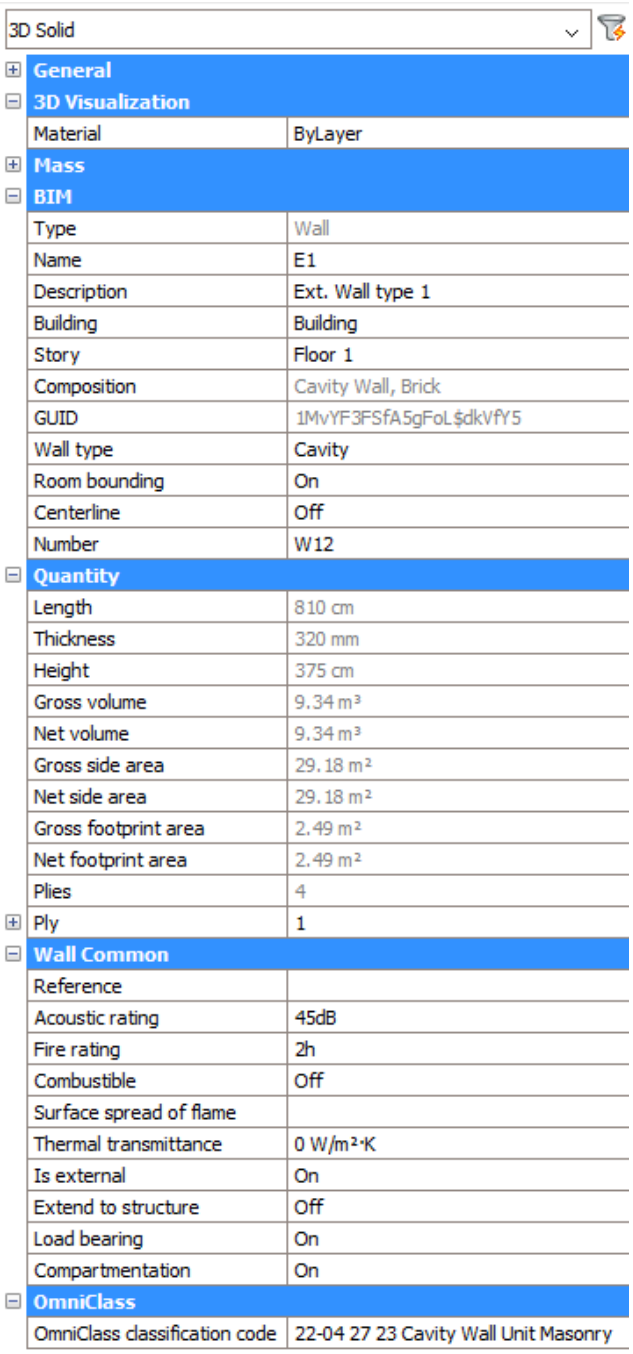
5 Add attributes using the following syntax, in the **Tag** field, in the properties panel:

| Property type | Syntax | Examples |
|---|---|---|
| **BIM** | <property name> | name<br>room bounding |
| **Parameters** | parameters/<parametername> | parameters/w<br>parameters/framethickness |
| **General** | general/<property name> | general/layer<br>general/linetype scale |
| **Mass** | mass/<property name><br>mass/<property name>:<child> | mass/surface area<br>mass/ucs elevation:minimum |
| **Geometry** | geometry/<property name><br>geometry/<property name>:<child> | geometry/position<br>geometry/ucs elevation:maximum |
| **Quantity** | quantity/<property name> | quantity/length<br>quantity/net footprint area |
| **IFC** | <ifc property set>/<ifc property> | wall common/load bearing<br>door window glazing type/ glass layers |
| **User Defined** | <property set label>/<property label> | exterior walls/price |
| **Classification system** | <classification system label>/ <classification code label> | omniclass/omniclass classification code |

***Прим.:*** To know which properties are available for a specific BIM entity type, select such entity in the model, then open the Properties panel.

In the image below the properties of a BIM **Wall** entity are listed.

Click the + button to expand a property type node.



| 3D Solid | |
|---|---|
| **General** | |
| **3D Visualization** | |
| Material | ByLayer |
| **Mass** | |
| **BIM** | |
| Type | Wall |
| Name | E1 |
| Description | Ext. Wall type 1 |
| Building | Building |
| Story | Floor 1 |
| Composition | Cavity Wall, Brick |
| GUID | 1MvYF3FSfA5gFoL$dkVfY5 |
| Wall type | Cavity |
| Room bounding | On |
| Centerline | Off |
| Number | W12 |
| **Quantity** | |
| Length | 810 cm |
| Thickness | 320 mm |
| Height | 375 cm |
| Gross volume | 9.34 m³ |
| Net volume | 9.34 m³ |
| Gross side area | 29.18 m² |
| Net side area | 29.18 m² |
| Gross footprint area | 2.49 m² |
| Net footprint area | 2.49 m² |
| Plies | 4 |
| Ply | 1 |
| **Wall Common** | |
| Reference | |
| Acoustic rating | 45dB |
| Fire rating | 2h |
| Combustible | Off |
| Surface spread of flame | |
| Thermal transmittance | 0 W/m²·K |
| Is external | On |
| Extend to structure | Off |
| Load bearing | On |
| Compartmentation | On |
| **OmniClass** | |
| OmniClass classification code | 22-04 27 23 Cavity Wall Unit Masonry |

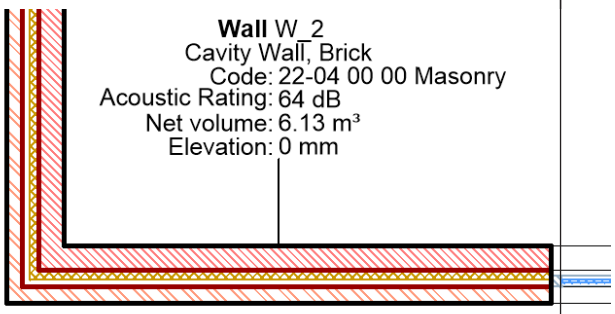
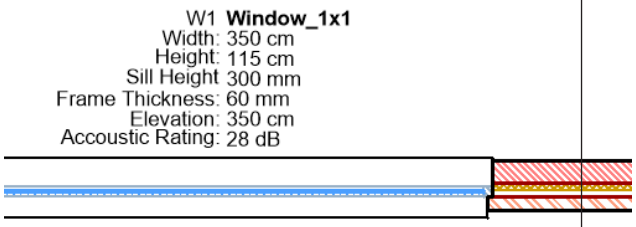
6 For each new attribute do the following:

- Copy an existing attribute or create a new attribute (ATTDEF command).
- Select the new attribute.

  ***Прим.:*** Do not double click the attribute. The **Edit Attribute Definition** dialog box does not accept spaces in the **Tag** field.
- In the Properties panel, type the property reference in the **Tag** field (see the syntax in the above table).
- **(Optional)** Define the Text properties (**Text Style**, **Justify**, **Height**, …) in the Properties panel.

- **(Optional)** Add linework and fixed text.

7 On the **Block Edit** toolbar: click the **Save** icon to conclude the BEDIT command.

| Definitions in the tag block | Resulting tag |
|---|---|
| **type** number<br>composition<br>Code: Omniclass/omniclass classification c<br>Acoustic Rating: wall common/acoustic rating<br>Net volume: Quantity/net volume<br>Elevation: mass/ucs elevation:minimum |  |
| _WallTag block | Wall tag in a plan section |
| number **name**<br>Width: parameters/W<br>Height: parameters/H<br>Sill Height sillheight<br>Frame Thickness: parameters/framethickness<br>Elevation: geometry/ucs elevation:minim<br>Accoustic Rating: window common/acoustic rati |  |
| WindowTag block | Window tag in a plan section |

### 20.7.3.9 Assign composition tags

Composition tags allow you to tag a wall or a slab element in order to specify its composition. These BIM tags are a list of the ply materials which make up the element.

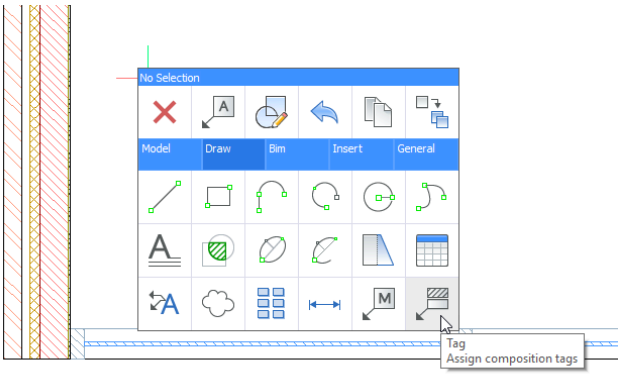
For more information about this command, visit the Command Reference article BIMTAG.



**Create composition tags**

1 Open the section drawing for which you wish to create composition tags.

2 Select **Tag Assign composition tags** from the Quad.

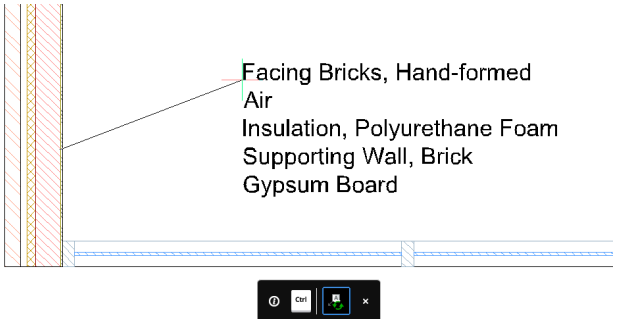



Or type BIMTAG in the Command line, then do one of the following:

- Type M+Enter for the manual option; then T + Enter for **Change Tag** type, followed by O+Enter for the composition option.
- Click the options in the Command line.
- Click the options in the **Prompt** Menu.

3 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

4 Select a point on a BIM element to generate a tag for it. The tag will appear. If the point selected is shared between multiple BIM entities, you can use the CTRL key to cycle through the different possible composition tags.




You are prompted: Place the tag [Next element]:

5 To adjust the position of the tag, move your cursor and left click to place it.

6 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

7 Repeat steps 3 and 4 for all of the entities you wish to be tagged. If you do not require any more tags, press the Esc or Enter to complete the command.

### 20.7.4 Create drawing types with drawing customization

#### 20.7.4.1 About drawing customizing

BricsCAD BIM uses drawing templates to customize your generated drawings. For a general understanding of how these templates are created and work, please visit the Templates for generated drawings article.

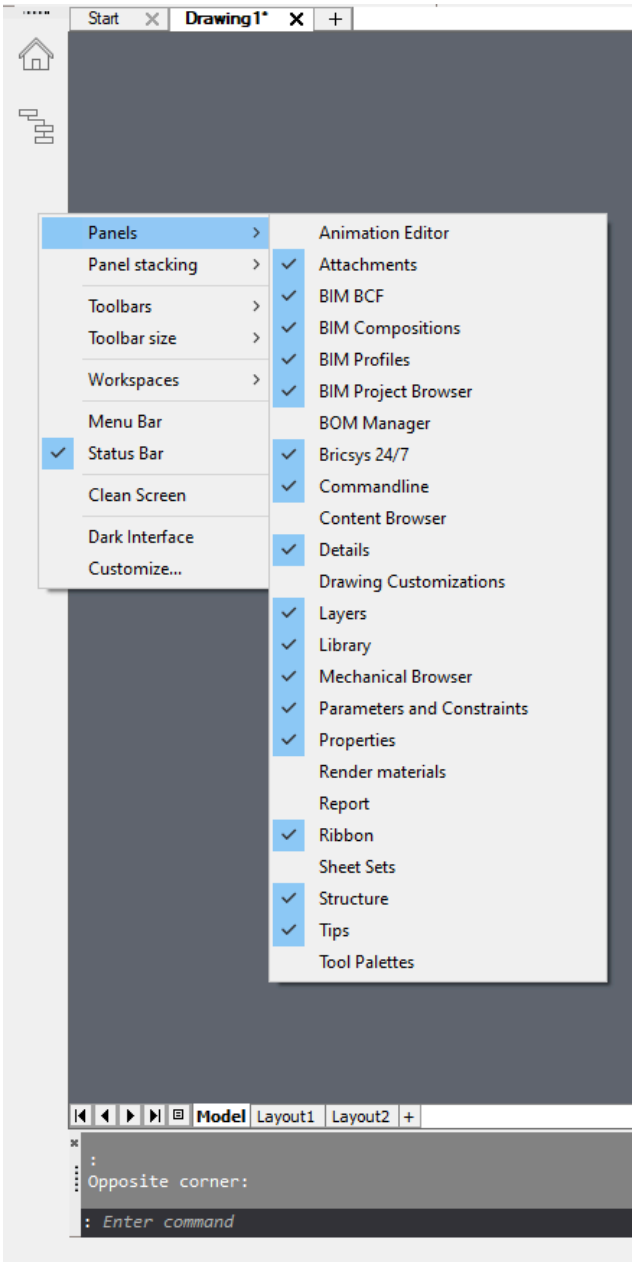
In V21, **Drawing Customization** allows you to create bespoke templates for drawing types based on a set filter parameter rule against an entity type's value or property within the 3D BIM model. This is especially helpful for customizing graphics of model entities independent of their assigned layers or layer states, therefore transferable and usable on other projects with similar entity properties.

The **Drawing Customizations** tool is operated by 3 principles – **Entity Customizations, Filter Rules & Styles** – and is only accessible from the panel.

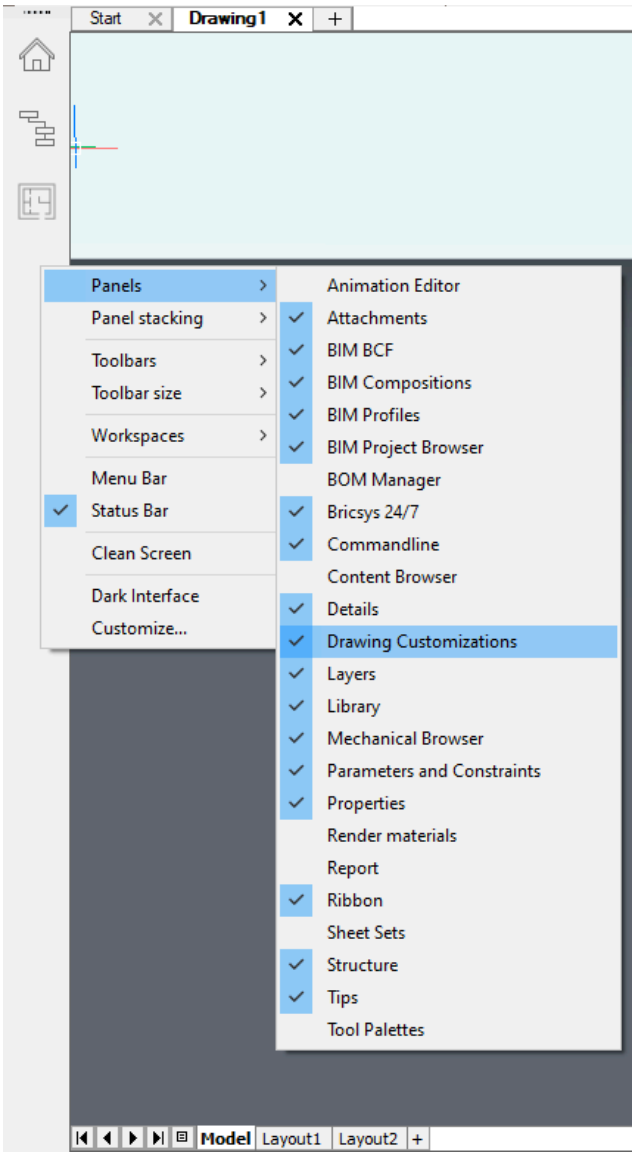
***Прим.:*** Version Control for **Drawing Customization** will be implemented from V21.1.06. As features and parameters are constantly improved and added in future V21 updates, using older versions of BricsCAD to open **Drawing Customization** templates may affect their behavior and performance. Therefore, a versioning system would help to prevent customizations from being accidentally opened or modified by BricsCAD versions that are too old to handle them properly.

### 20.7.4.2 Drawing customizations panel

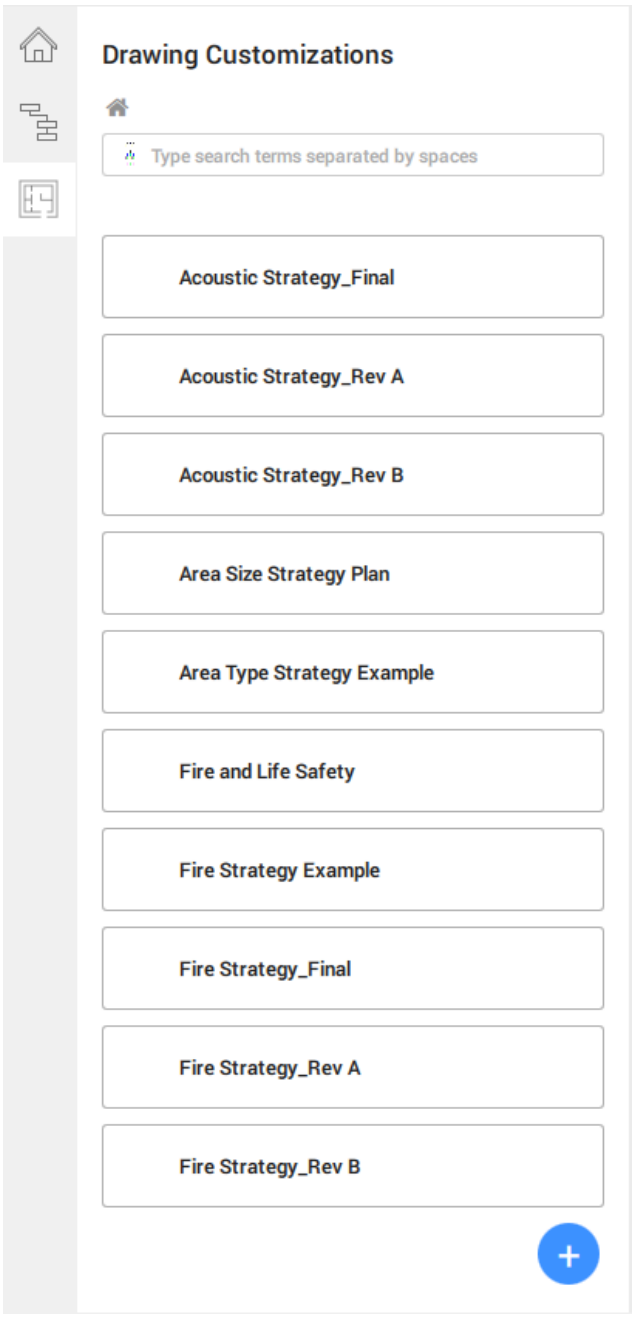
1 Right-click the panel and hover to **Panels**.




2 Select **Drawing Customization** to dock the tool icon in the panel.




3 Open the panel, it allows you to create view templates to control and customize the visibility settings of your drawings on layouts.



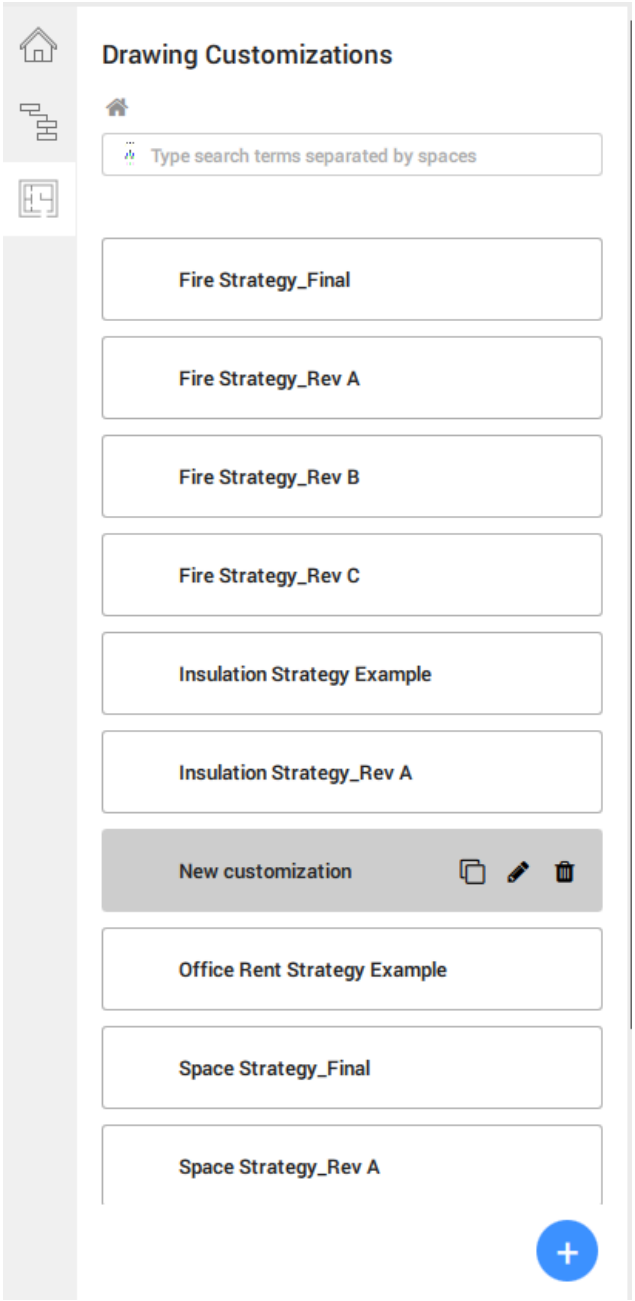

4 Click to create a new template. By default, the template is named **New Customization** with the text highlighted.




***Прим.:*** All the controls of **Drawing Customizations** are represented as graphic icons within the **User Interface** panel, such as to create a new control item. They are repeated throughout the tool in

different settings, for example **Entity Customizations**, **Ply Customizations**..., but their functions and purposes remain consistent.

5 Rename the template whilst the text is being highlighted. Once done, press Enter to save and apply the name.

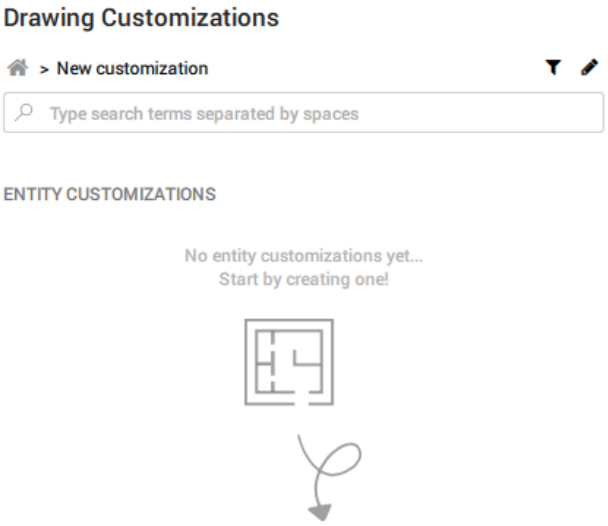



***Прим.:*** You can duplicate, rename or delete your customization templates.

6 Click the newly created **Drawing Customization template** tab to begin defining its customization settings.

The first step into the template brings you to **Entity Customizations**, where you will create the

necessary entity customizations template and will be first described later in this article.

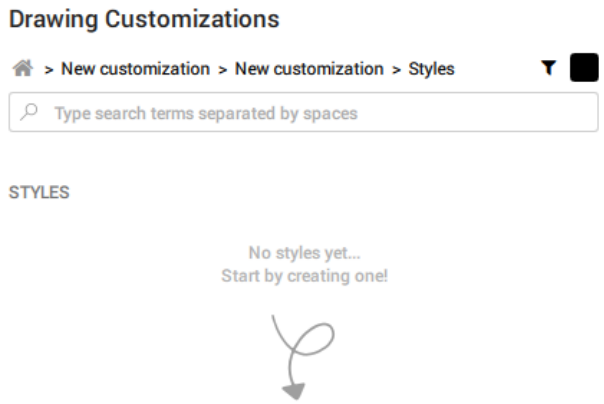



You will also notice a funnel icon and a pencil icon on the top right part of the panel. They are the **Filter Rules** and **Styles** tabs respectively.
The 3 principles – **Entity Customizations, Filter Rules & Styles** – which make up the **Drawing Customization** tool as described earlier in the article, are now accessible on this main page.

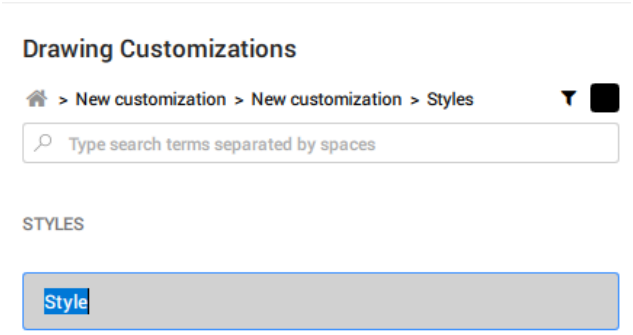
### 20.7.4.3 Drawing customizations styles

1 Click the pencil icon to access the **Styles** tab.

This is where your desired style options, such as line weight, line colors..., are stored and will appear as drop-down options in the **Entity Customizations** and **Center Customizations** settings as shown in the previous section above.



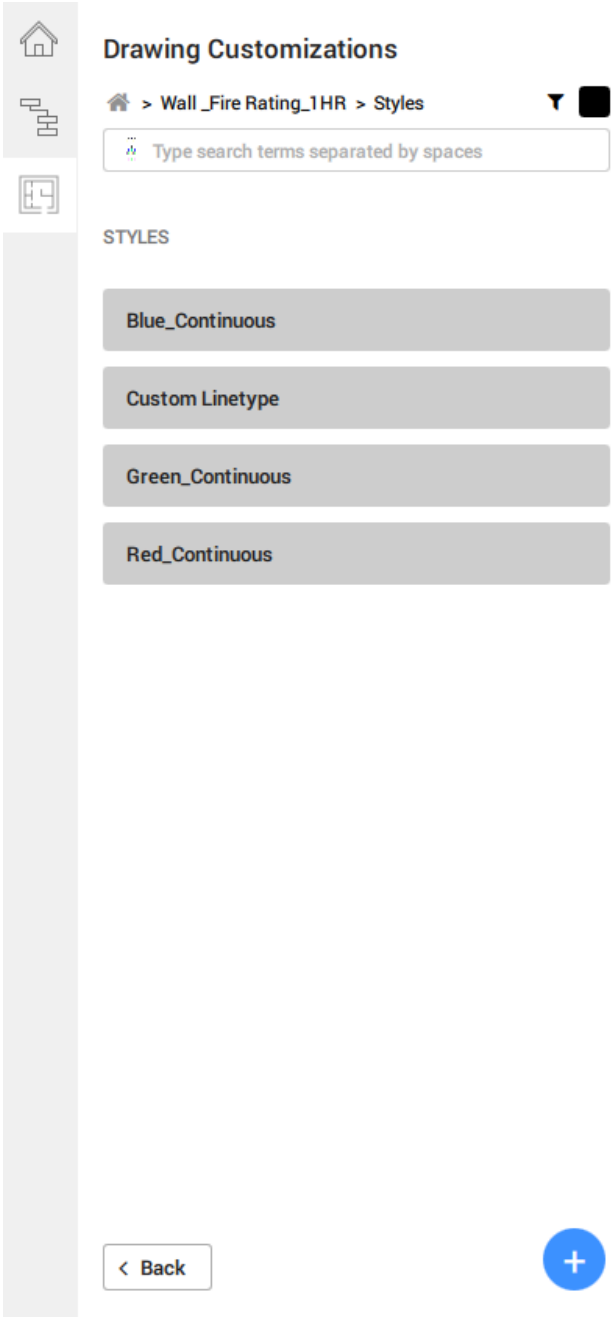

2 To create a new style, click + at the bottom right of the panel and a new **Style** template tab will appear.




3 By default, the tab is named **Style**. Rename it whilst the text is being highlighted as shown. After you are done, hit Enter to save and apply.

4 Create a list of desired styles that you wish to use in this **Drawing Customization** template. Each tab contains its individual styles settings, such as the style's color, line type, line weight and line type scale.

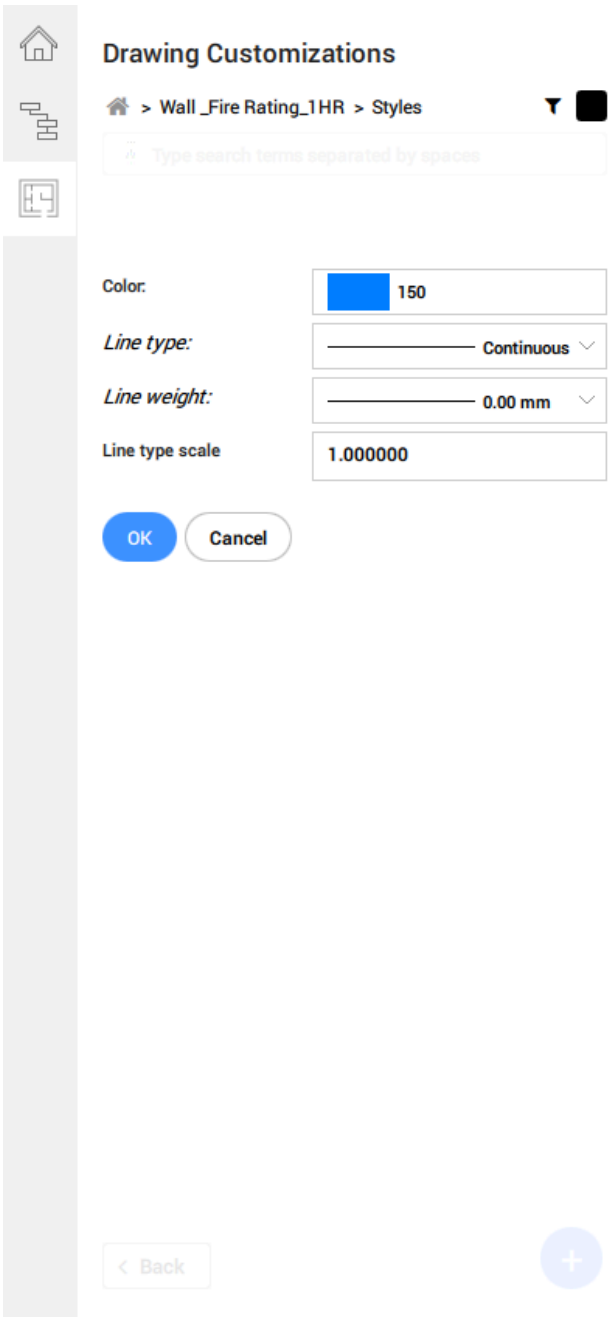
An example of the style pages a list of templates:




5 To edit any of the styles template available, click on its relevant tab to access its settings.

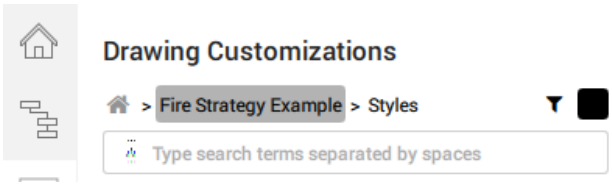
***Прим.:*** Similar to the **Drawing Customization template** tabs, hover over the styles template tabs to reveal the function icons to duplicate, rename or delete.

Style settings include the options to define **Color**, **Line Type**, **Line Weight** and **Line type scale** in their respective drop-down dialogs. Once done, click **OK** to save and apply the changes.




6 To leave the **Styles** page, click **Back** at the bottom left side of the panel to return to your previous page. From there, you can navigate to modify the **Entity Customizations** or **Rules** templates.

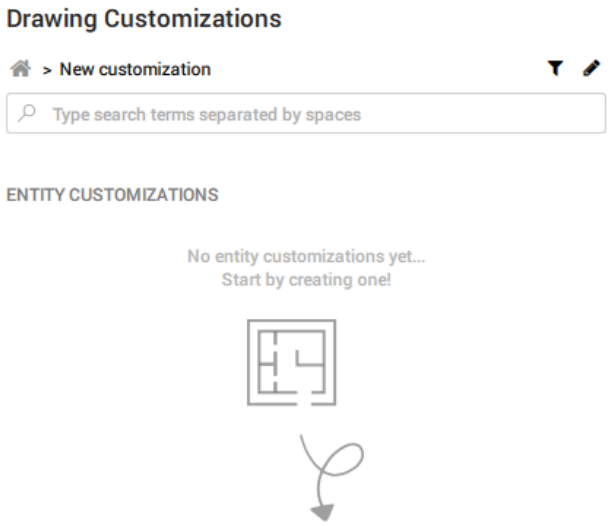
**Or** click on your desired branch in the tree structure map above the search bar to navigate to your desired page.



***Прим.:*** The tree structure map is helpful to keep yourself orientated within the **Drawing Customization** tool. Similarly, the search bar allows you to narrow to a specific template if you have a long list of values to navigate through.

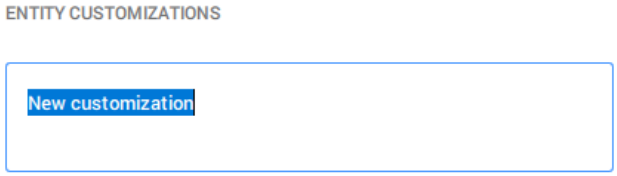
#### 20.7.4.4 Entity customizations

**Entity Customizations** refer to the visibility settings of how you would like to have your elements in your model drawing to look like.

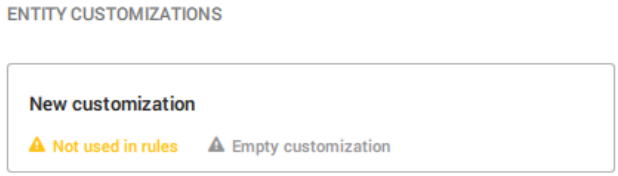



1 Click ⊕.

2 To create a new **Entity customization** template. By default, the template is named **New Customization** with the text highlighted.



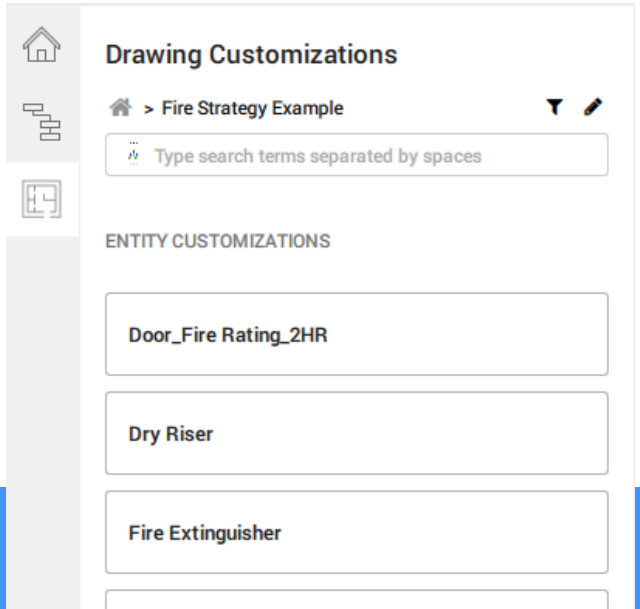

3 Rename it whilst the text is being highlighted as shown. After you are done, hit Enter to save and apply.




As the **Entity customization** was newly created and not fully set up, you will find the relevant notices at the bottom of the **Entity customization** tab. This is a helpful mechanism to remind you if any customization settings have not been defined yet, saving you the need to check the settings manually.

4 Create a list of templates for the entities you wish to customize.

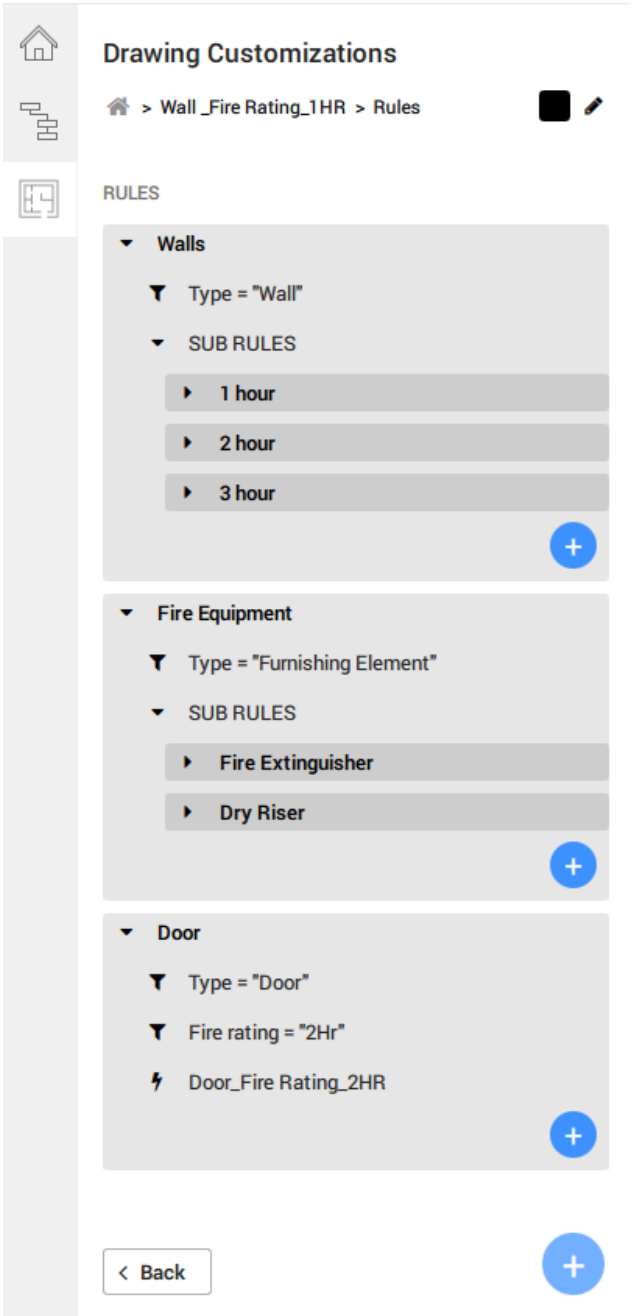
5 A list of **Entity Customizations** templates could look like this:



### 20.7.4.5 Filter rules

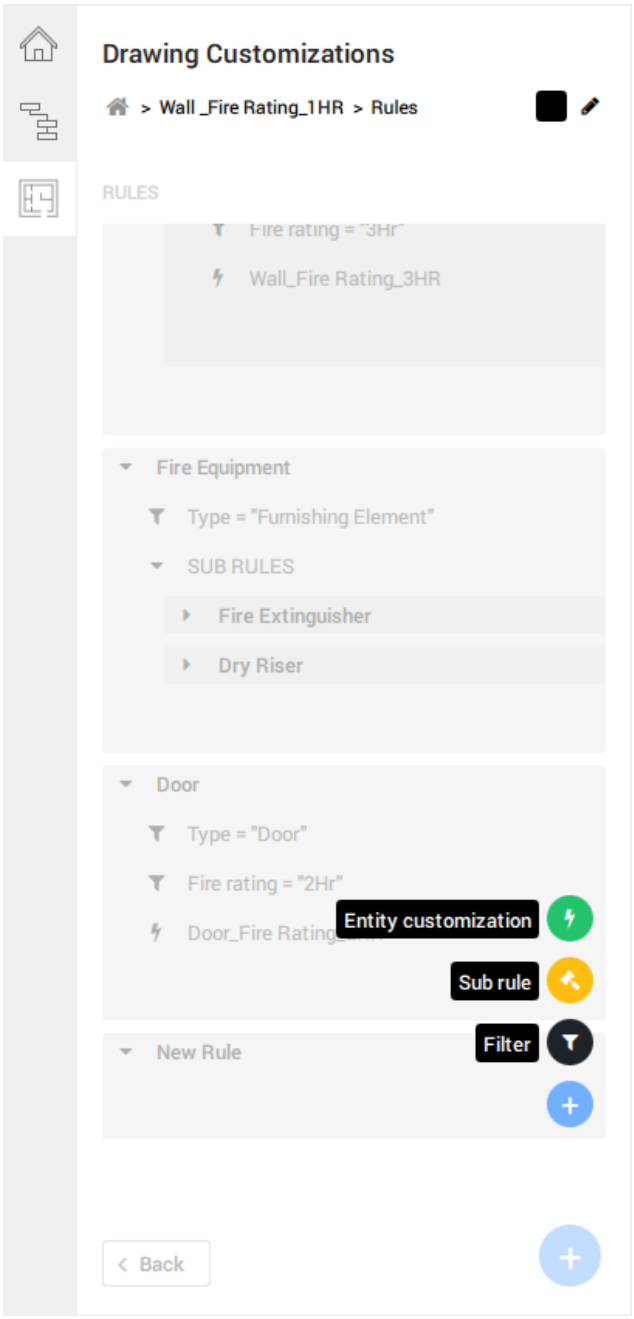
To be able to apply the right customization styles to specific entities, you have to set the appropriate filter rules to target those entities correctly in the **Filter Rules** tab located as a funnel icon on the top right-hand part of the panel next to the **Styles** icon.

***Прим.:*** The **Nested Property Search** function has been introduced from V21.1.06 onwards, allowing you to specify how nested objects within an assembly are filtered by and displayed.

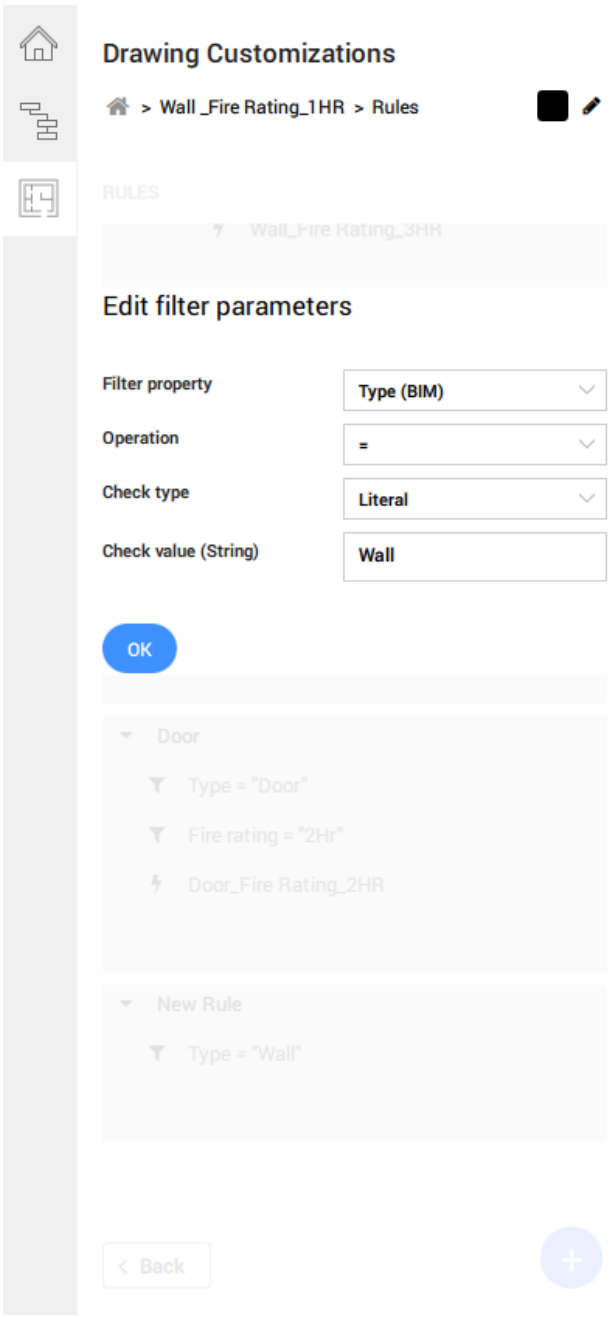



1 **Filter rules** can be further classed into **sub rules**, which falls as a sub-set of the parent rule, e.g. **Fire Equipment** as a **Filter Rule**, whereas **Fire Extinguisher** is classed as a sub rule.

also appears respectively and accordingly at each rule level. At the end of each rule, regardless of it being a subset or otherwise, it has to end with an **Entity Customization**.

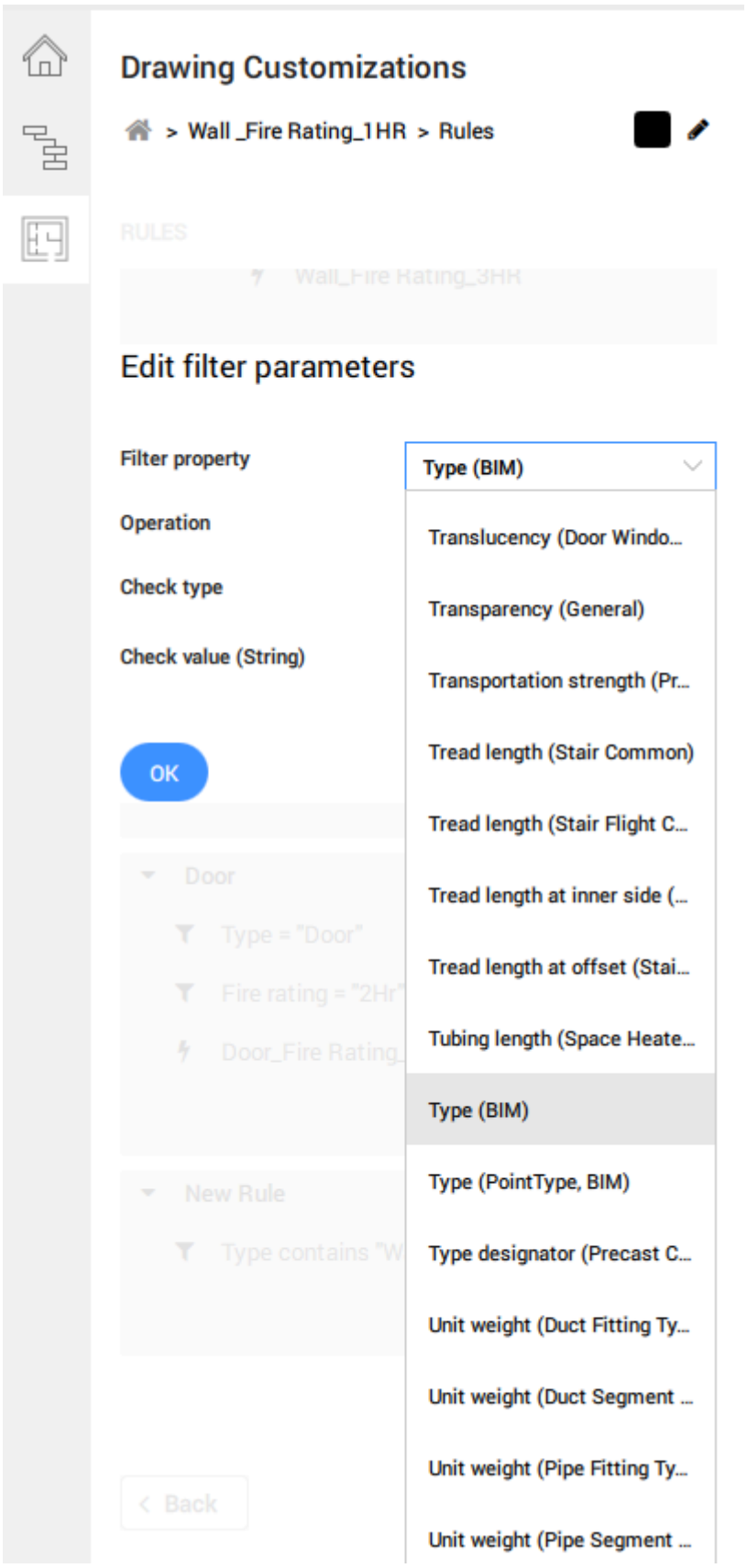



2 Adding and editing a filter parameter will lead you to a prompt, where the options of filter property are determined by the BIMPROPERTIES imported in the **namespace** settings.



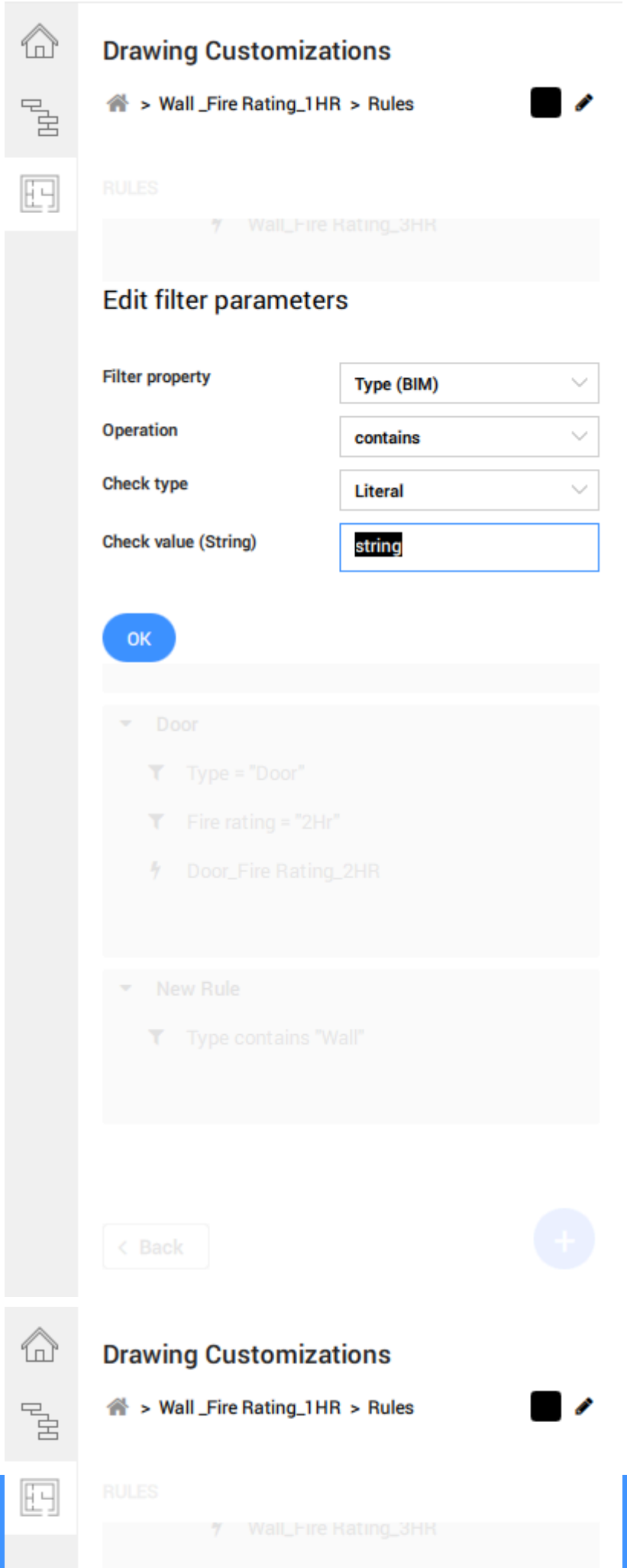

3 The filter **Operation** values are also available as a drop-down list.

***Прим.:*** You can also start typing a certain keyword to search a specific property that you want to filter on.

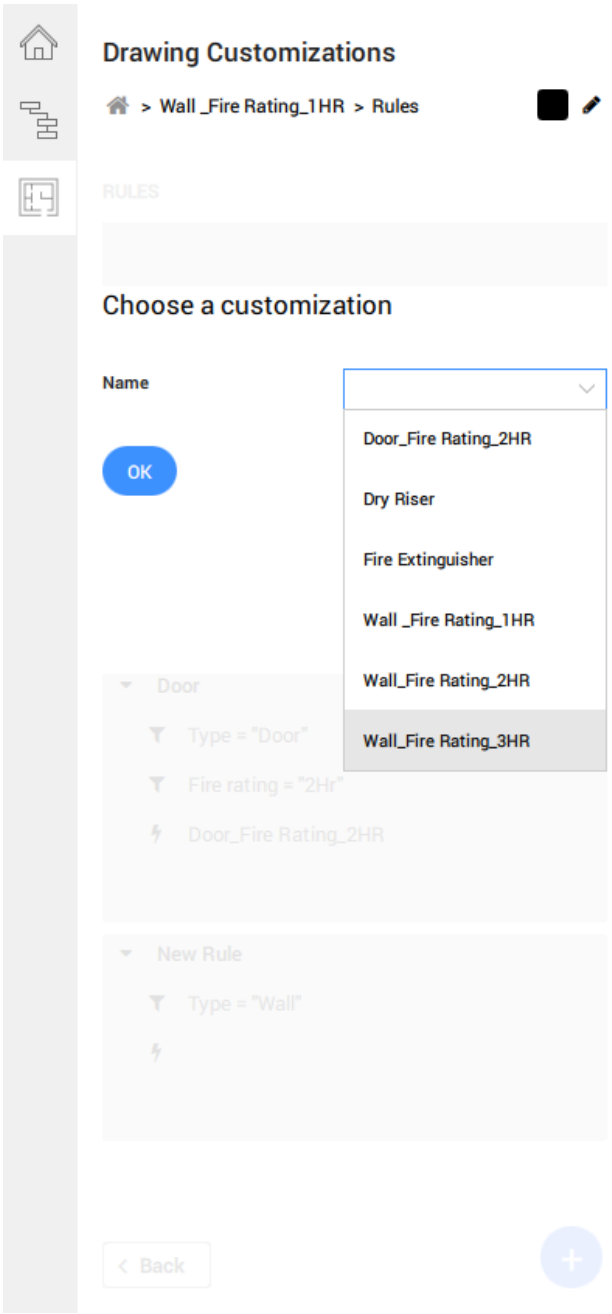



Drawing Customizations
> Wall _Fire Rating_1HR > Rules
RULES

4 You can specify a string manually in your check value if your **Check** Type is a **Literal**. If you have selected a **Property** value for **Check Type**, your **Check Property** value will refer to the same values found in the **Filter Property** drop-down.


Drawing Customizations
> Wall _Fire Rating_1HR > Rules
RULES
Edit filter parameters
Filter property
Type (BIM)
Operation
contains
Check type
Literal
Check value (String)
string
OK
Door
Type = "Door"
Fire rating = "2Hr"
Door_Fire Rating_2HR
New Rule
Type contains "Wall"
Back
Drawing Customizations
> Wall _Fire Rating_1HR > Rules
RULES

5 To end and legitimize a **filter rule**, the **Entity Customization** value must be added and filled in. The options available in the drop-down selection reflects the existing customizations tab available in the **Entity Customization**page.

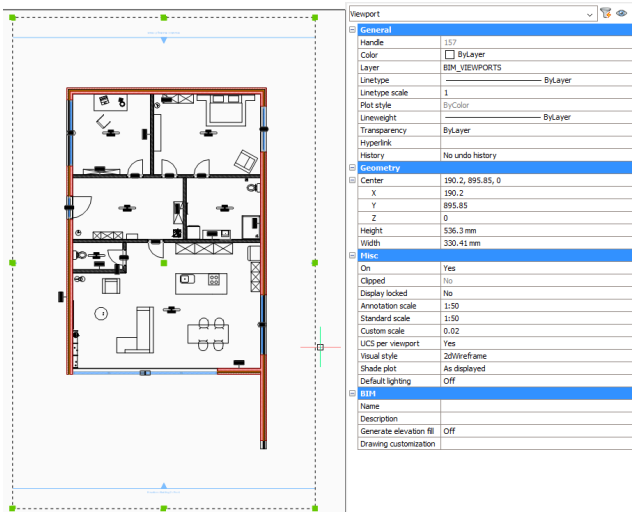



### 20.7.4.6 Applying templates to viewports on sheets

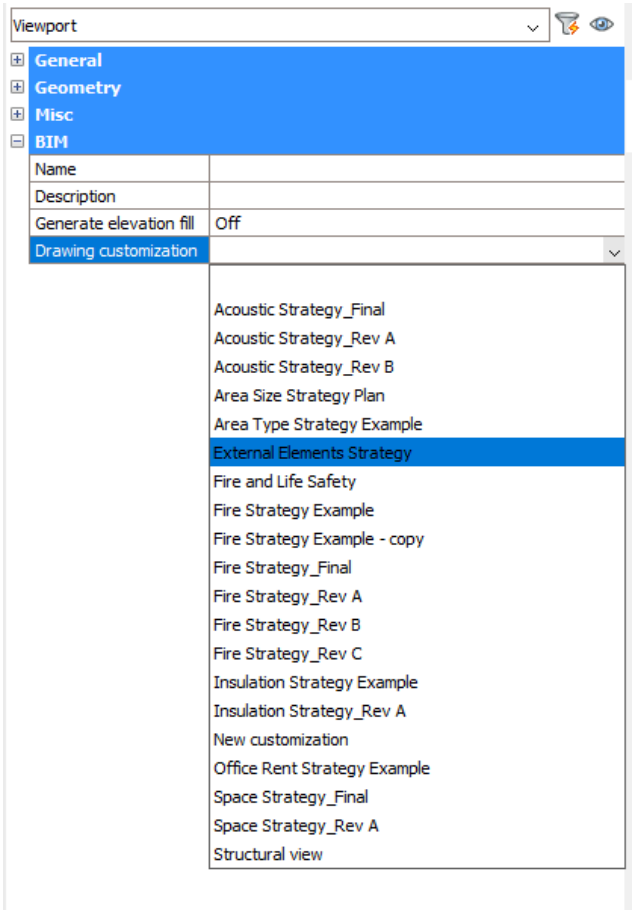
With your **Entity Customizations**, **Filter Rules** and **Styles** all set, your **Drawing Customization** template is now ready.

1 To apply the template, navigate to your sheet drawing with your desired viewport.

2 Open the Properties panel and select the viewport to view its properties.

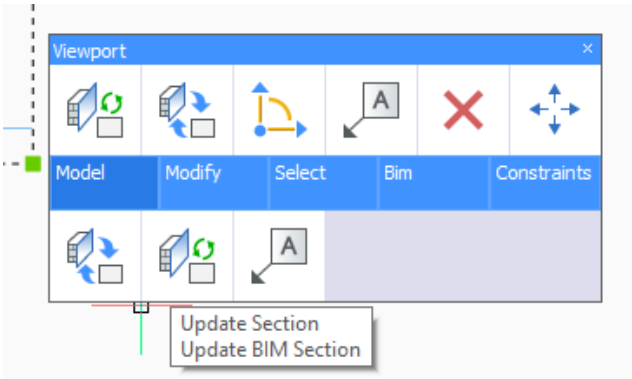



3 Navigate to the **Drawing Customization** property and click on the drop-down selection to the right.




4 Select your desired **Drawing Customization** template.

5 Once done, hover over your selected viewport to access the Quad. Click **Update Section** in the Quad under **Model** tab.




6 You should see a refreshed **Viewport** with the graphical changes as per your **Drawing Customization** settings.

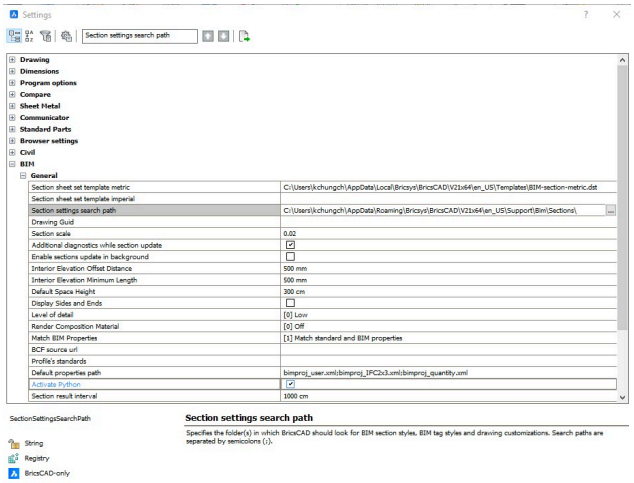
### 20.7.4.7 Accessing customization templates

By default settings, these templates are stored in the **Customizations** folder within your **Support** folder in the following path, under the **Bim** folder and then **Sections** subfolder:

*C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\V21x64\en_US\Support\Bim\Sections\Customizations*

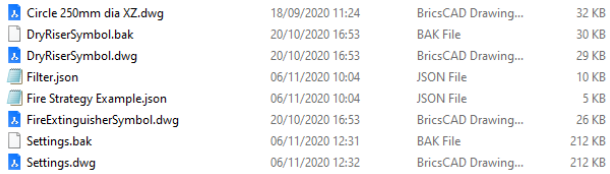
The **AppData** folder is hidden by default, so you may have to unhide it first. This path can also be accessed by entering SUPPORTFOLDER in the Command line.

You can specify a different path if your Customization files are stored elsewhere in an accessible folder. Simply redefine the path in the **Section settings search path** in the **Settings** dialog box or typing SRCHPATH in the command-line:




New customizations created with the panel will be stored in a **Customizations** subfolder in the specified path. Each customization template is saved as an individual folder which contains in its basic form Settings.dwg, a Filter.json and a New customization.json files. The folder will also contain any external symbol sources which you have created new using the **Create New** buttons.

Settings.dwg contains the values created in the **Style** tab in the form of layers and if you have created any custom properties that you wish to use as a **Filter** parameter, you have to ensure they are also reflected in the Settings.dwg.




***Прим.:*** As these **Customization** folders are stored locally in the machine you have created it with, you need to ensure the folders and their contents within are copied and pasted into the same location on other machines, should you wish to share it with other users on other machines. This is the current, temporary procedure to do so, as a dedicated function to streamline the sharing process within the **Drawing Customization** tool will be released shortly in an update.

## 20.7.5 Define detailed section

### 20.7.5.1 Command

BIMSECTION

### 20.7.5.2 Define a detailed section from scratch

1 Select **Detail Section** from the ribbon or use the Command line by entering BIMSECTION followed by **D** to choose the detail option.

**You are prompted**: Select the first corner of the section box or [Based on an existing section/Scale]:

2 Select the first corner of the area you wish to be contained within the detailed section.

3 **You are prompted**: Select the opposite corner of section box:

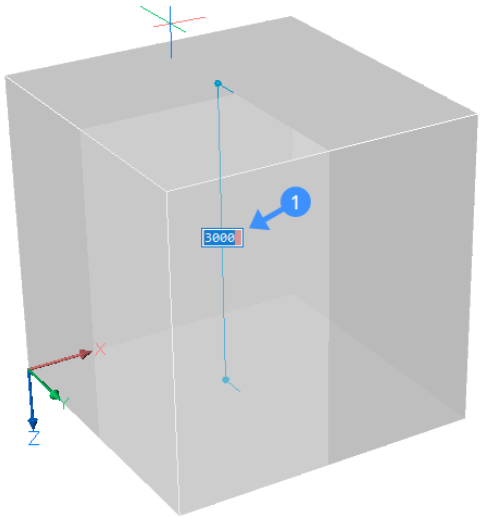
4 Select the opposite corner snap to a point or enter the distances. Use the TAB key to switch between dynamic dimension fields.

5 **You are prompted**: Select the height of volume box <4000>:

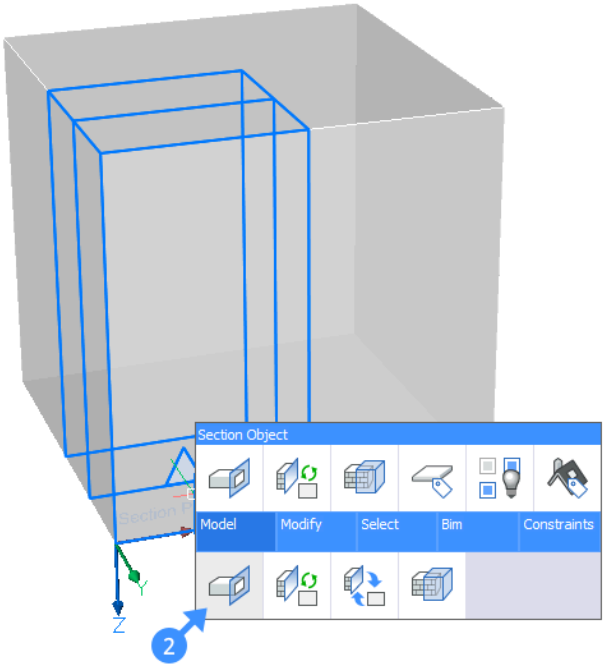
In this case, the default height is 4000mm.

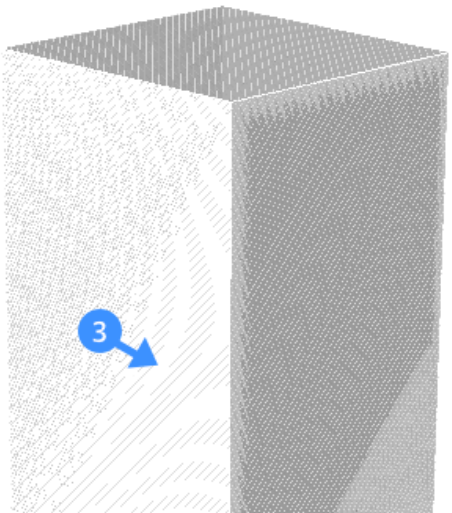
6 To complete the volume, do one of the following:

- Press Enter to accept the default height.
- Specify a point.
- Type a value in the dynamic dimension field (1).




To view the detailed section which has just been defined, hover over the section and select **Clip display** (2) from the Quad **or** double-click the selected section. Now only the detailed section will be visible (3).

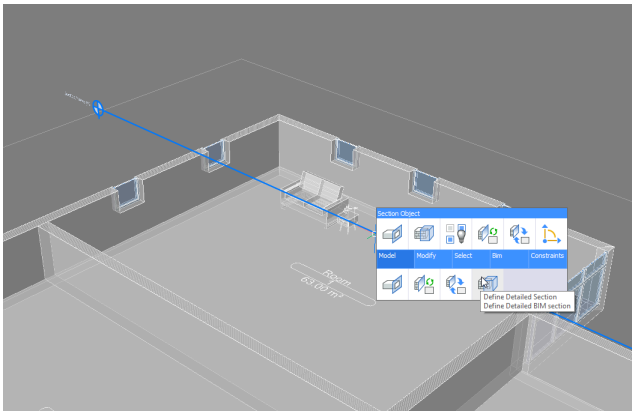


### 20.7.5.3 Define a detailed section based on an existing section

Here you will be shown how to position the base of your detailed section on an already existing section.

1 Highlight the existing section that you want the detailed section base to be positioned on and select **Detail Section** from the Quad.

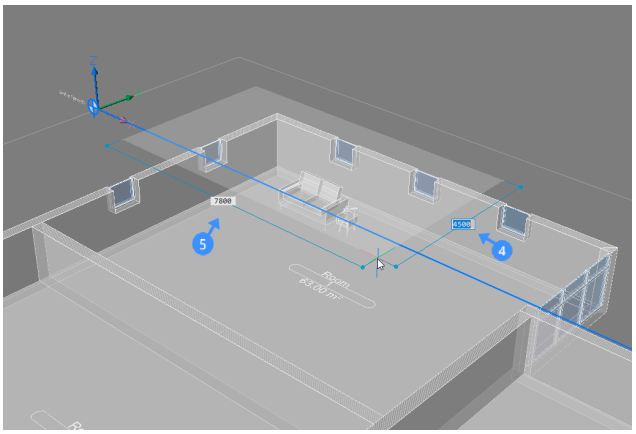


**You are prompted**: Select first corner of volume box base rectangle:

2 Draw a rectangle around the desired area by typing the values in the dynamic dimension fields.

3 Hit the TAB key to go back and forth between the dynamic dimension fields (4, 5). Your selection will be aligned with the plane of the existing section.

4 Press Enter to confirm the values. The **volume section** box displays dynamically.



**You are prompted**: Select the height of the volume section box.

5 Press Enter to accept the default height.

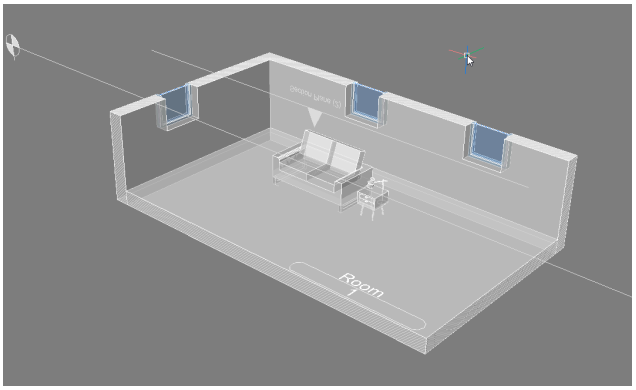
6 Specify a point.

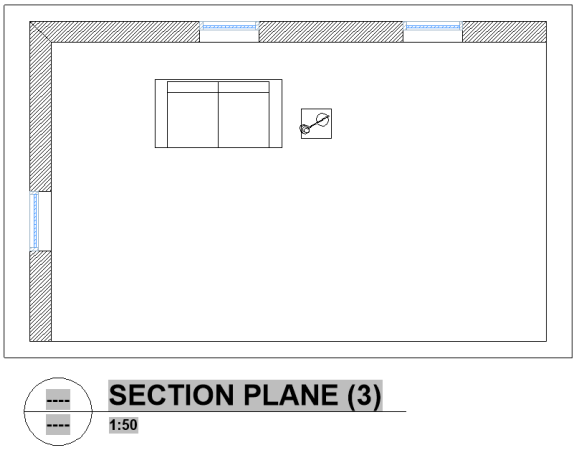
7 Type a value in the dynamic dimension field.

8 The detailed section has now been defined and can be found in the structure browser. Hover over the section and select **Clip display** from the Quad **or** double-click the selected section to view it.

9 To generate the detailed section on a sheet, select the detailed section and select **Generate Section** from the Quad.

10 You can open the section results by clicking **Open Model** when the section plane(s) are selected. The sheet is saved in the same folder containing the model drawing.




If you need more help with how to generate drawings, visit the **Generating Drawings** page.
If you want to annotate your drawings, see how to add tags and linear dimensions.

### 20.7.6 Define reflected ceiling plan

#### 20.7.6.1 Command

BIMSECTION

#### 20.7.6.2 About

Reflected ceiling plans show a mirror image of the ceiling. These are useful to show the layout of lights and other installations in the ceiling. To produce the reflected ceiling plan a section is created. This section displays the part of the building above the section boundary.



For more information about this command, visit the Command Reference article BIMSECTION.

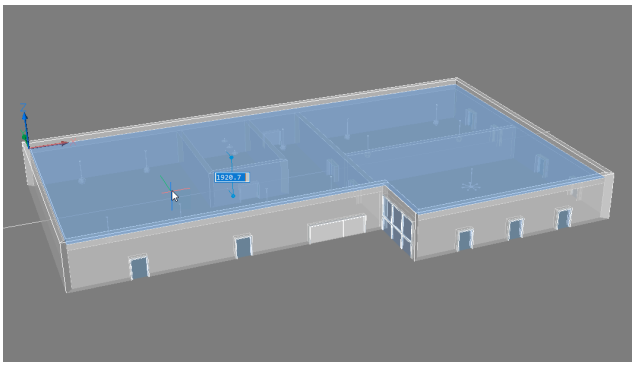
#### 20.7.6.3 Create a reflected ceiling plan

1 Select **Reflected Ceiling Plan** from the ribbon **or** enter BIMSECTION into the Command line followed by

**R** to choose the **Reflected ceiling** option.

**You are prompted**: Select a point to place section:

2 Select the ceiling or a face with a plane parallel to that of the ceiling.

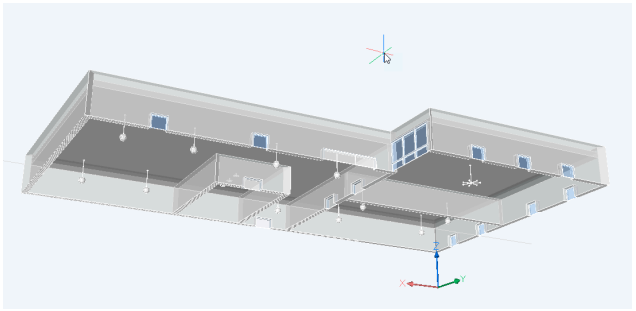


**You are prompted**: Specify distance:

3 Do one of the following:

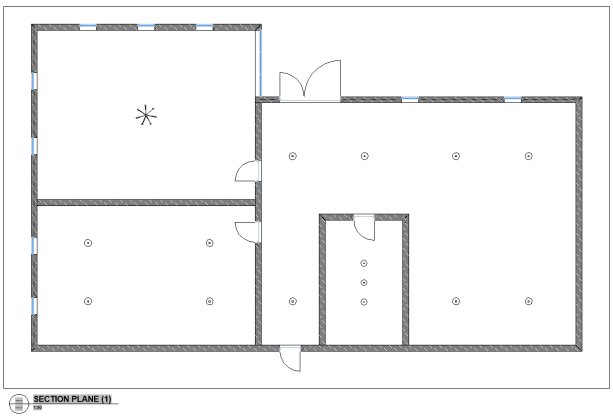
- Type a value in the dynamic dimension field to offset the section from the initial position.
- Left-click when the section plane is at the location you want.

4 The reflected ceiling section has now been defined. To view hover over the section and select **Clip display** from the Quad.



5 To generate the reflected ceiling plan on a sheet, hover over the section in the model and select **Generate Section** from the Quad.

6 To open the section result drawing click **Open Model**. The sheet is saved to the same folder that contains the model drawing.




If you need more help with how to generate drawings, visit the Generating Drawings page.
If you want to annotate your drawings, see how to add tags and linear dimensions.

### 20.7.7 Detailing with BimPatch

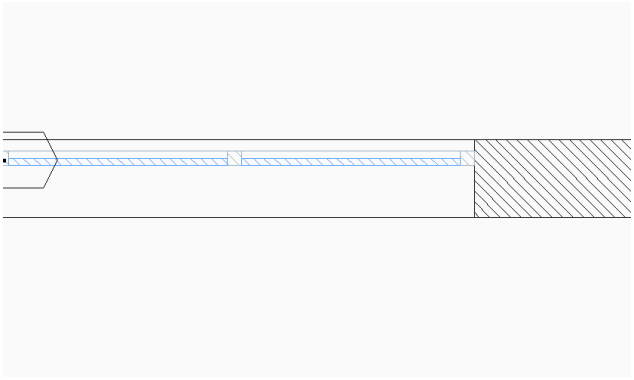
***Прим.:*** BIMPATCH cuts out part of geometry and replaces it by a block reference. In this cutting process, entity data linked to the 3D model are lost. Therefore, starting from V21.2, this command is no longer available.

#### 20.7.7.1 Command

BIMPATCH

#### 20.7.7.2 About

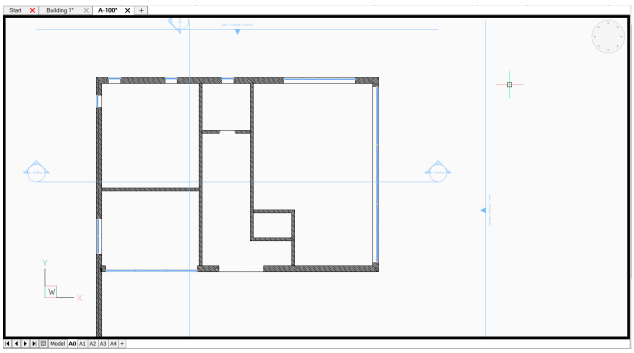
In this procedure, we will add fixtures and a vapor barrier to our window-wall connection. We will detail this window-wall connection.



#### 20.7.7.3 Procedure: detailing a window connection

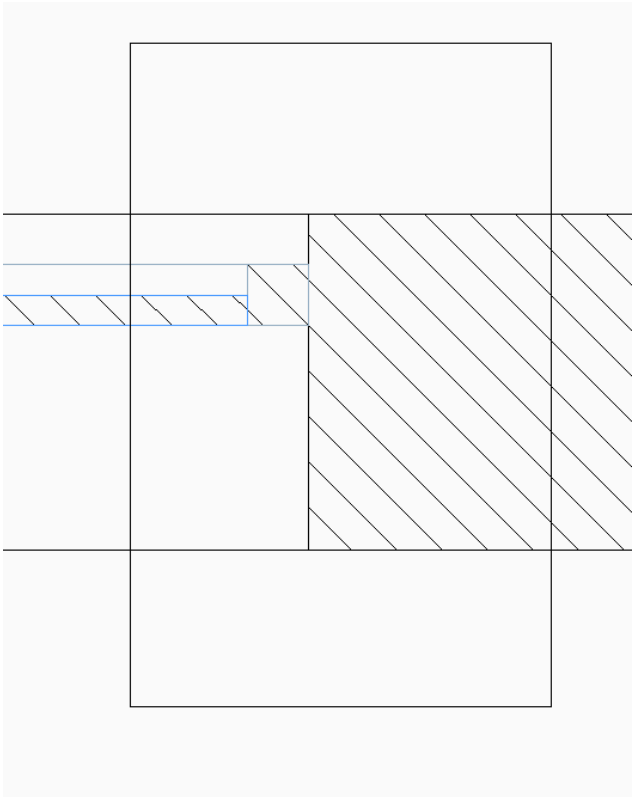
1 Open the section.

2 Go to the model space by double-clicking in the viewport **or**toggle the viewport mode in the **Status** Bar.

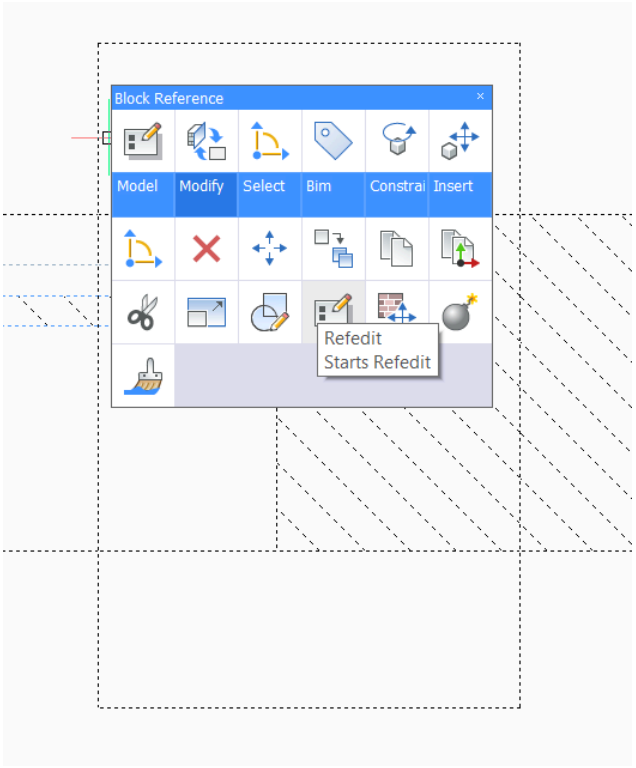


3 Launch the BIMPATCH command in the model space.

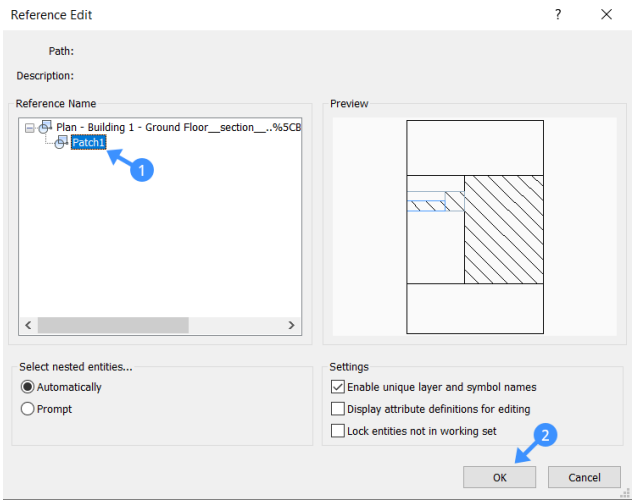
4 Draw the rectangle over the detail and hit enter.



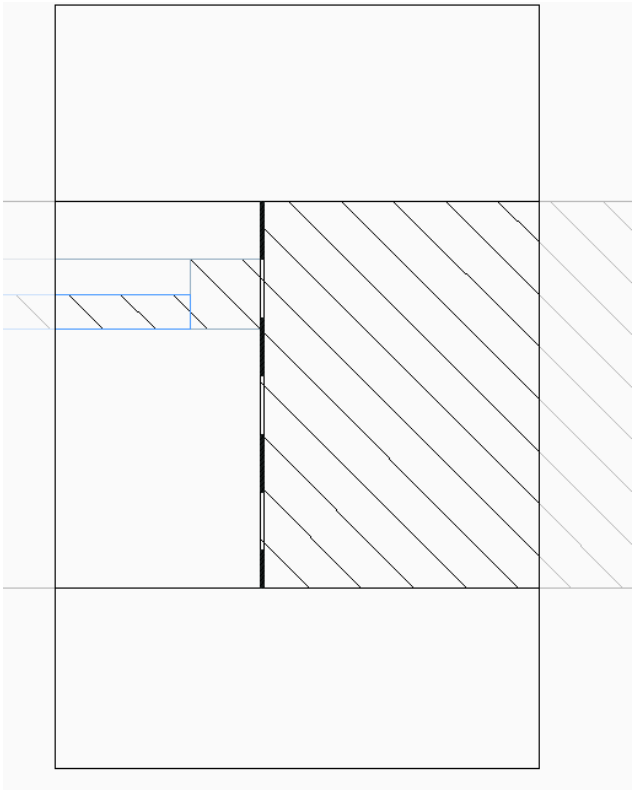
5 Hover over the section object and launch **Refedit** from the quad.




6 A menu appears, select the patch (1) and click **OK** (2).

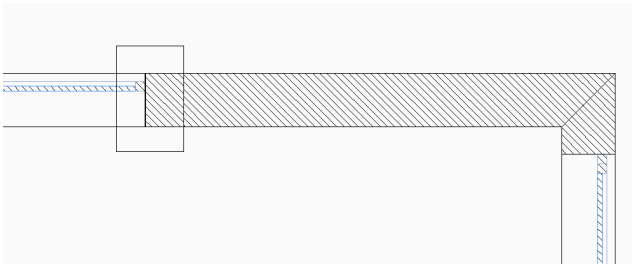



7 Draw the required detail for the window-wall joint. We will add a vapor barrier.

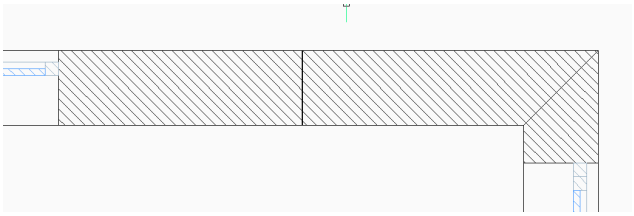


8 Save the changes made in the **RefEdit** session with the **Save** option of the REFCLOSE command.

9 Go back to the paper space. The detailing of the window connection is done. If we now move the window to the left in the 3D model, you will see that the detail you made a patch of moves along.



***Прим.:*** If you would have added the vapor barrier in the model space of the section without making a patch of it, the vapor barrier wouldn't be seen as a part of the detail. If you would move your window, the vapor barrier would stay in the same position as you drew it before.



### 20.7.8 Editing data extraction definition (.dxd) files

DXD (Data Extraction Definition ) files are plain text files. You can edit DXD files created by the DATAEXTRACTION command using a text editor such as MS Notepad and use them in the Creating Schedules procedure in the **Project Browser**.

Lines starting with semicolon (;) are comments and ignored on parsing. Empty lines are ignored as well.

The file can contain 4 sections, each section name is written between square brackets, section names are English case-insensitive (ECI) strings. The sections are:

- **Settings**
- **Entity Types**
- **Properties**
- **Filter**

#### 20.7.8.1 The [Settings] section

The section contains values of the following parameters which control the behavior:

- **ExtractFromBlocks** - whether to take entities from blocks into account, considering all nesting levels.
- **ExtractFromXrefs** - whether to take entities from external reference blocks into account.
- **CountXrefs** - whether xrefs are taken into account at all.
- **CombineEqualRows** - whether to group equal rows in the output table; if yes, then the **Count** column is added, which indicates how many equal data rows are included.
- **IncludeNameColumn** - whether the output includes the **Name** column. The **Name** column indicates the source entity type: 3D Solid, 3D Solid Ply, ...

Parameter names are ECI strings. The format is:

ParameterName=value,

The value is either 1 or 0. No spaces should be written around the equal (=) character.

#### 20.7.8.2 The [Entity Types] section

This section contains list of entity types which will be processed. The format of each line can be one of the following types:

**EntityType** | **DisplayEntityType**

- **EntityType** is the ECI entity type name, such as **Circle** or **AttributeDefinition**. This is the mandatory part of the line.
- **DisplayEntityType** is optional and can be any character string including spaces in any language. If present, this string appears in the **DATAEXTRACTION** dialog and in the output table.

**Block** | **BlockName** | **DisplayBlockName**

- **Block** is the predefined ECI word, and **BlockName** is case-insensitive name of the block to be processed, including **xRefs** and **MInsert**(s).
- **DisplayBlockName** is optional and has the same meaning and format as **DisplayEntityType**.

**Block** | *

This is a variant of the previous format meaning that all blocks will be taken into account; if present, then all other lines regarding blocks in this section are ignored. You can use the Filter section to filter the content to be processed.

*

All entity types and blocks will be taken into account, all other lines in this section are ignored then. You can use the **Filter** section to filter the content to be processed.

### 20.7.8.3 The [Properties] section

This section contains a list of all properties to be included in the data extraction output. The format of each line can be one of the following types: **Entity Property**, **Attribute**, **Drawing**, **Formula** or **Block View**.

**EntityPropertyName | DisplayEntityPropertyName | ValueOutputFormat**

- **EntityPropertyName** is the ECI name of entity property, e.g. **Lineweight**, **Color**, **Center Y** and so on. Note, all names do not contain spaces except those which represent coordinates of 2D or 3D points or vectors. In the last case, coordinate letters X, Y and Z should be separated by single space from the general property name.

  **EntityPropertyName** is the mandatory part of the line, the remaining part is optional.
- **DisplayEntityPropertyName** is user-defined and can be in any string including spaces in any language. By default, it is equal to **EntityPropertyName**.
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

**Attribute | AttributeTagName | DisplayAttributeTagName | ValueOutputFormat**

- **Attribute** is the predefined ECI word.
- **AttributeTagName** is the ECI attribute tag name as it is stored in the drawing.
- **DisplayAttributeTagName** and has the same meaning as for **EntityPropertyName** (see above).
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

**Drawing | DrawingPropertyName | DisplayPropertyName | ValueOutputFormat**

- **Drawing** is a predefined ECI word.
- **DrawingPropertyName** is an ECI name, which can be one of the following:
  - **Comments**
  - **DrawingRevisionNumber**
  - **FileAccessed**
  - **FileCreated**
  - **FileLastSavedBy**
  - **FileLocation**
  - **FileModified**
  - **FileName**
  - **FileSize**

  - **HyperlinkBase**
  - **Keywords**
  - **Subject**
  - **Title**
  - **TotalEditingTime**

- **DisplayPropertyName** has the same meaning for **EntityPropertyName** (see above).
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

**Formula** | **EntityPropertyName** | **DisplayEntityPropertyName=FormulaExpression** | **ValueOutputFormat**

- **Formula** is a predefined ECI word.
- **EntityPropertyName** is a unique name for this formula item.
- **DisplayEntityPropertyName** is an optional output name of the property, if it is omitted, then **EntityPropertyName** is used as the table column title.
- = is a mandatory sign, which must exist even if **DisplayEntityPropertyName** is omitted.
- **FormulaExpression** is the expression in format as in \AcExpr formula fields, which can contain constants such as PI or functions such as SIN (see Using Expressions). An operand in an expression can be a **DisplayPropertyName** enclosed in arrow brackets (<>).

  For instance: =2 * PI * <Radius> calculates the circumference of a circle.
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

**BlockView** | **BlockViewPropertyName** | **DisplayBlockViewPropertyName** | **BlockViewOptions**

- **BlockView** is a predefined ECI word
- **BlockViewPropertyName** is a unique name for this block view item
- **DisplayEntityPropertyName** is the output name of the property; cannot be omitted
- **BlockViewOptions** is a string in the following format:

  ViewType;DwgPath;SectionName

  - **ViewType** is one of the following: **Front**, **Top**, **Left** or **Right**; cannot be ommited.
  - **DwgPath** is path to dwg file, may be relative with respect to the dxd file; can be ommited.
  - **SectionName** is the name of the section to extract the section settings from for the block view construction,;can be ommited.

If **DwgPath** or **SectionName** are ommited or not found, default settings will be applied.

An example:

BlockView|BlockViewProperty1|Elevation symbol|Front;../Sections/_SectionSettings.dwg;_BlockViewSettingsSection

*

If present all entity properties are taken into account; all other lines in this section, except Drawing and Formula lines, are ignored.

### 20.7.8.4 The [Filter] section

**Syntax description:**

Relational operators:

| Operator | Meaning |
|---|---|
| == | Equal to |
| != | Not equal to |
| < | Less than |
| > | Greater than |
| <= | Less than or equal to |
| >= | Greater than or equal to |

- Relational expressions are enclosed in '()' brackets.
- Logical operators AND and OR are case-insensitive and enclosed in '<>' brackets.
- Logical expressions use a prefix notation.
- No spaces are allowed between '<' and a logical operator ('<OR', '<AND').
- Tokens can be separated by zero or any number of spaces, except logical expression begin,
- example:

  <AND ( Layer == "MyLayer" )(Thickness<=3) >
- Expressions can be split into any number of lines; tokens are not broken.
- Use the **EntityPropertyName** to refer to properties (not the **DisplayEntityPropertyName**).
- Lines can be commented with ';'.
- String values are enclosed in double quotation marks "" .

<OR (Type == Window)(Type == Door)>

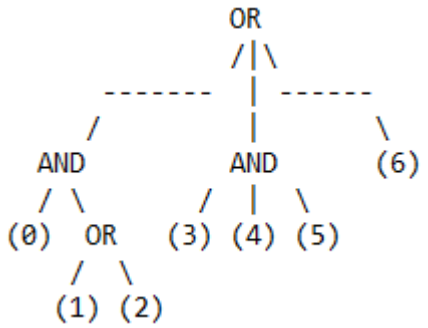
The result of the filter in the above example is that only BIM entities classified as windows or doors are included in the data extraction. See the BIMCLASSIFY command to learn more about BIM entity types.

The filter expression is either a single relational one or several ones grouped by logical operations.

Example:

<OR <AND (0) <OR (1) (2)>> <AND (3) (4) (5)> (6)>

where (0),...,(6) are relational expressions such as (Radius >= 3).




### 20.7.9 Editing generated drawings

When a BIM Section is generated, a block containing the drawing generation result is added to the model space of the target drawing, as well as a paper space viewport looking onto this block. The block and the viewport initially are placed so they do not overlap with existing geometry contained in the drawing. You can move this block or viewport to a new location, which will be preserved on subsequent updates.

In the default template drawing which is used to create the generated drawings **Viewlabel** and **NumberBubble** blocks exist. You can place them associated to the selected sheet view. The **Title** block in this drawing contains attributes that will be filled in automatically when the associated **Sheet Set** properties are defined.

A sheet list table can be placed on any sheet of the sheet set. In the current implementation, the table content is not associative yet.

#### 20.7.9.1 Inserting the title block

1 Open the **Drawing Explorer - Blocks** dialog.

2 Double click the **Title** block.

The **Drawing Explorer**dialog closes and the block is attached to the cursor.

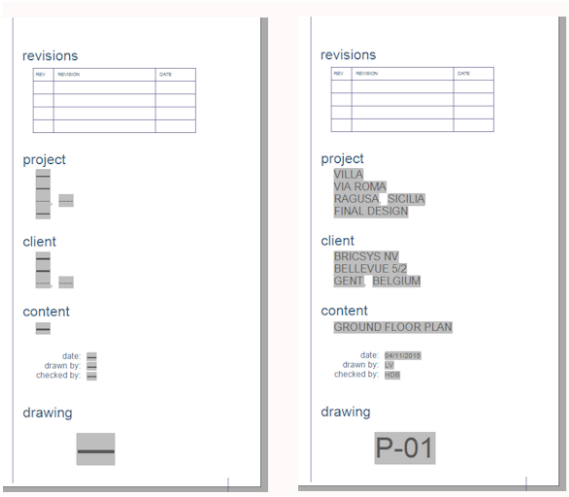
In a metric units project the size of the **Title** block fits the A1 paper size (594 x 841 mm)

In an imperial units project the size of the **Title** block fits the Arch D paper size (24 x 36 inches)

3 Click at the lower left corner of the paper sheet in the layout.

4 Open the **Sheet Sets** panel (SHEETSET command) and define the **Project** and **Custom Properties** available in the context menu.

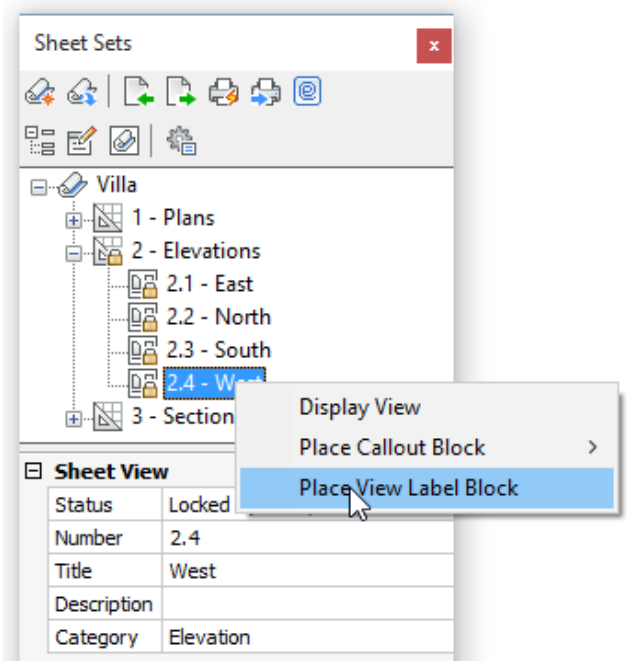
5 Select the **Title** block, then click the **Update Fields...** tool button ( ) in the **Redraw/Regen** toolbar.




### 20.7.9.2 Inserting a view label for a sheet view

1 Select the sheet view in the **Sheet Sets** panel.

2 Right-click the sheet view for which you want to insert the view label block.




3 Choose **Place View Label Block** in the context menu.

**You are prompted**: _VIEWLABEL

Units: Millimeters Conversion: 1

Insertion point for block:

4 Specify the insertion point.

5 **You are prompted**: Scale factor for block <1.0>:

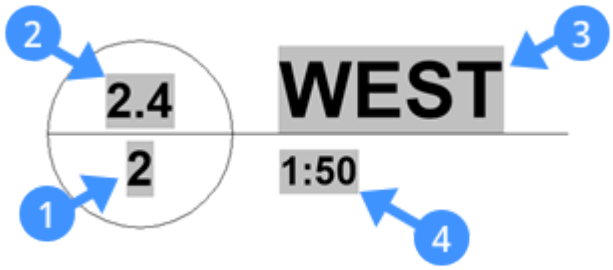
6 Specify the scale factor or right-click to accept the current scale.

7 **You are prompted**: Rotation angle for block <0>:

8 Specify the rotation angle or right-click to accept the current angle.

9 The View Label block is inserted.

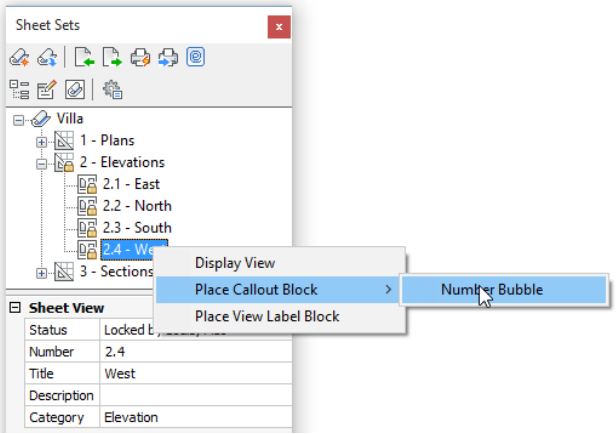
The Sheet Number (1), Sheet View Number (2), Sheet View Title (3) and Viewport Scale (4) of the selected sheet view are filled in automatically.




### 20.7.9.3 Inserting a number bubble block for a sheet view

1 Select the sheet view In the **Sheet Sets** panel.

2 Right-click the sheet view for which you want to insert the **Number Bubble** block.




3 Choose **Place Callout Block** > **Number Bubble** in the context menu.

**You are prompted**: _CALLOUT

Units: Millimeters Conversion: 1

Insertion point for block:

4 Scale factor for block <1.0>:

5 Rotation angle for block <0>:

6 Specify the insertion point.

7 **You are prompted**: Scale factor for block <1.0>:

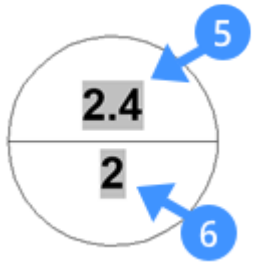
8 Specify the scale factor or right-click to accept the current scale.

9 **You are prompted**: Rotation angle for block <0>:

10 Specify the rotation angle or right-click to accept the current angle.

11 The **Number Bubble** block is inserted.

The Sheet Number (5) and Sheet View Number (6) of the selected sheet view are filled in automatically.




### 20.7.9.4 Inserting a sheet list table

A **Sheet List** table can be inserted on any sheet of the sheetset. In the current implementation, the table content is not yet associative and therefore will not be updated automatically when sheets are added, removed or renamed.

1 Right-click the sheetset name in the **Sheet Sets** panel and choose **Insert Sheet List Table** in the context menu.

**You are prompted**: Select insertion point.

2 Specify a point in the drawing.

## 20.7.10 Elevation hatch

### 20.7.10.1 About elevation hatch

**Generate elevation fill** is a property of a section that can display the texture or solid hatch of a composition when a 2D elevation is generated. Hatches can also be generated for other types of sections when activated in the properties panel.

The hatch is generated for every visible boundary on the 2D elevation including boundaries that are not perpendicular to the section. Hatches are generated without perspective corrections. The union of elevation lines can be changed independently from the 'union section' settings in the **Physical Materials** dialog.

### 20.7.10.2 Make elevations

***Прим.:*** **Elevation** here refers to a *Section Plane* in a 3D model that has its **Type** property set to **Elevation**. This can then be used to generate 2D drawings that represent the elevations of your model.

1 Create BIM elevations by using **Bimify**.

2 Select the BIM elevation and choose **Generate BIM Section** in the **Model** command tab in the Quad.

3 Select the BIM elevation and choose **Open BIM Section Model** in the **Model** command tab in the Quad.

The section result displays in the layout viewport of the destination drawing.

4 Select the paper space viewport.

5 The viewport properties display in the Properties panel.

6 Under BIM, set the **Generate Elevation Fill** property to **On**.

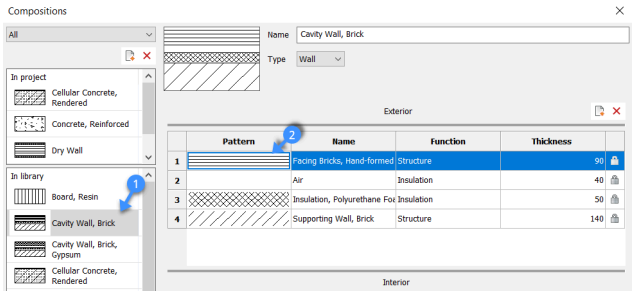
7 Select the viewport and choose **Update BIM Section** in the **Model** command tab in the Quad.

8 **(Optional)** Copy the viewport, then set the **Generate Elevation Fill** property to **Off** to create two versions of the section result, one with and one without elevation hatch.

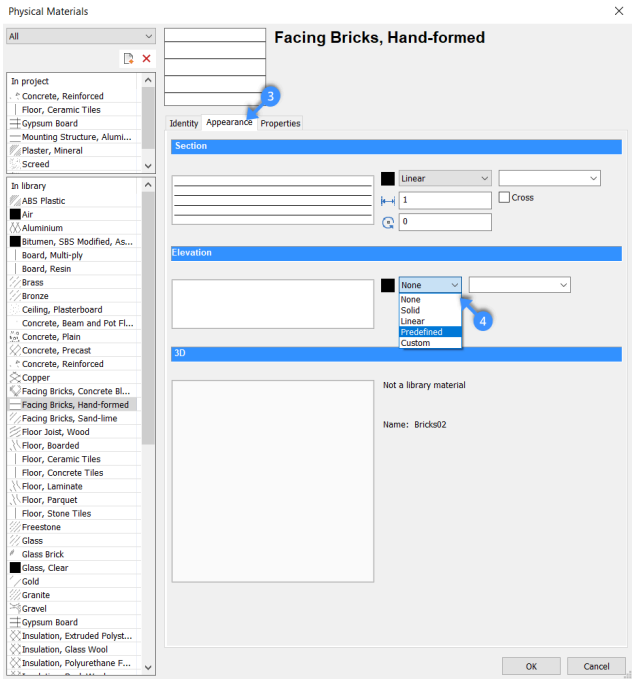
### 20.7.10.3 Configure the hatch

1 Go to the **Physical Materials** dialog by typing BLMATERIALS or BLCOMPOSITIONS (1) into the Command line.

2 Click on the composition and double click on the pattern of the composition to open the **Physical Materials** dialog (2).

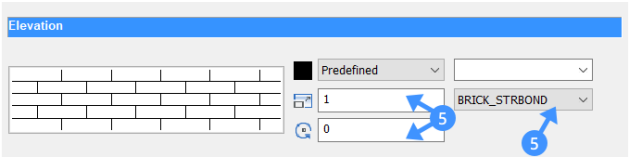



3 To configure the texture or solid fill, go to the **Appearance** tab page (3). The elevation hatch is set by default to **None**. Click the elevation hatch type button and select the type of hatch: **None**, **Solid**, **Linear**, **Predefined** or **Custom** (4).



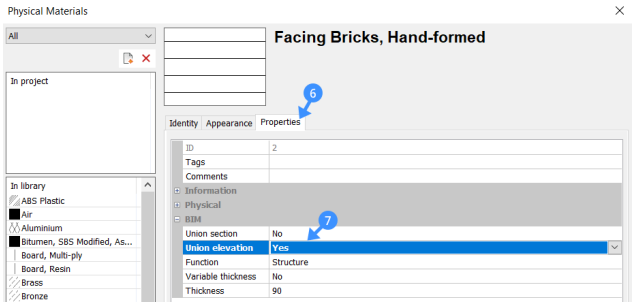

4 Configure the hatch. Define the scale, angle, color and/or layer of the pattern. (5)

***Прим.:*** Type a name in the layer field or click the arrow button and select a layer in the list.

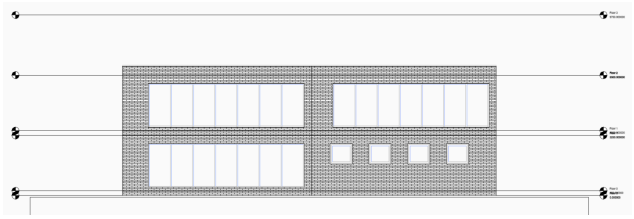



5 To remove connection lines in the elevation between different solids of the material go to the **Properties** tab (6) and set **Union elevation** to **Yes** (7).

***Прим.:*** **Union elevation** works independently from the **Union section** setting.




6 After regeneration, the hatch will appear on the elevation.



### 20.7.11 Enscape integration

#### 20.7.11.1 Commands

STARTENSCAPE, CLOSENESCAPE, ENSCAPEEXPORTSETTINGS, ENSCAPEIMPORTSETTINGS, ENSCAPEBATCHSCREENSHOT, ENSCAPELOADVIDEOPATH, ENSCAPEPANORAMA, ENSCAPESAVEVIDEOPATH, ENSCAPESCREENSHOT, ENSCAPESETTINGS, ENSCAPESTARTVIDEO, ENSCAPESTOPVIDEO, ENSCAPETOGGLELIVEUPDATE, ENSCAPECREATESTANDALONE

#### 20.7.11.2 About

Enscape is a real-time rendering plugin for BricsCAD. With a single click, you can start Enscape in seconds and walk through your fully rendered 3D model. There is no need to export to other programs. All changes in BricsCAD are immediately available for display and evaluation in Enscape.

You can quickly explore different materials and design options to present to clients. With the option to create output renders, videos, and panoramas of your project, you can send the output files to your clients or colleagues, which allows for quick demonstrations. You can edit the changes in BricsCAD, and Enscape will immediately show the changes you have made to the project, even in VR.



#### 20.7.11.3 Enscape system requirements

The minimum requirements to run Enscape:

- Windows 7 64 Bit or higher.
- Intel i5 CPU or higher.
- NVIDIA or AMD GPU with 2GB VRAM.
- Supports OpenGL 4.3.
- NVIDIA GeForce GTX 660/Quadro K2000 and newer.
- AMD Radeon R9 260/FirePro W5100 and newer.
- 4 GB RAM (8 GB recommended).

- 2 GB Video-RAM (4 GB recommended).

For more information about the Enscape system requirements, visit https://enscape3d.com/knowledgebase/system-requirements/.

For more information about specific Enscape features, you can visit their knowledge-base on their website.

### 20.7.11.4 How to install/uninstall Enscape

1 Download the Enscape-BricsCAD Connection from the Application Store.

2 The installer copies the necessary files to the 'Program Files\Bricsys\Enscape'-folder.

3 Restart BricsCAD.

4 You now have a new tab in the ribbon. Use this to launch Enscape.

***Прим.:*** The first time you will start the Enscape-plugin, you will be asked to provide a license key. If you don't have a license key yet, you can start by choosing the trial option.

**How to uninstall Enscape**

Uninstall the connector from **Add or Remove Programs** in Windows, by searching Enscape and clicking **Uninstall**.

### 20.7.11.5 Navigate your model in Enscap

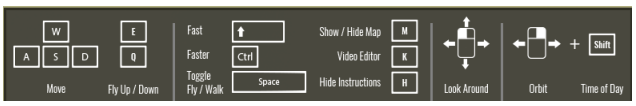
1 Click on **Start Enscape** in the ribbon **or** type in STARTENSCAPE in the Command line.

Changes made in BricsCAD (e.g. deleted furniture, new material, a new section, etc.) can be seen instantly in Enscape.

2 Follow the instructions to navigate through your model.

3 You can switch between **Walk** mode and **Fly** mode by pressing the Space bar.

Type H to hide the instructions.




### 20.7.11.6 Settings in Enscape

Open the **Enscape Settings** dialog by clicking **Settings** in the ribbon or type in ENSCAPESETTINGS in the Command line. When you change the settings, they will be automatically saved for the user, not in the drawing. There is also an option to create and import presets for the Enscape settings.

You can find more information about saving and importing presets in the following link: https://enscape3d.com/community/blog/knowledgebase/settings-presets/.

The **Enscape Settings** dialog consists of 7 tabs:

- **General**: general **Rendering Style** options, **Depth of Field** and **Exposure** settings.
- **Image**: Contrast and Color settings.
- **Atmosphere**: Horizon/background options, **Fog**, **Clouds** and **Sky Orbs**.
- **Input**: Mouse navigation settings.
- **Advanced**: Spectator options.

- **Capture**: **Image**, **Video** and **Panorama** options.
- **Customization**: Interface options.

### 20.7.11.7 Render materials in BricsCAD

In BricsCAD BIM, there are two methods to add render materials to your model.

- Directly apply to render materials (without compositions) to the solids/surfaces in your model, using the **Render Materials** panel.
- Apply render materials via compositions, using the **Compositions** panel.

***Прим.:*** When render materials are attached to an entity, both applied directly with the **Render Materials** panel and compositions attached, the composition materials will be shown (when the LEVELOFDETAIL system variable = 2).

***Thus, if an entity already has a composition and you want to change the render material, don't try to overrule it with the Render Materials panel, but change the composition plies instead.***

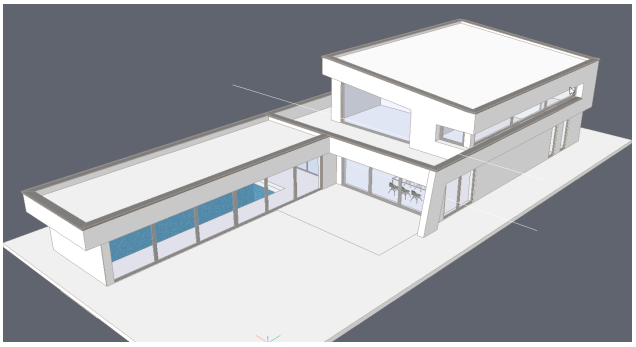
**Render materials with the Render Materials panel**

- To assign a material to a single entity
  a Select the material in the **Render Materials** panel.
  b Click on the entity or press and hold the left mouse button, then drag the material onto the entity.
- To assign a material to multiple entities
  a Select the entities.
  b Search the material in the **Render Materials** panel.
  c Click on the material.

  For more information about render materials, see the article Rendering materials.

**Render materials with Compositions**

1 Set the LEVELOFDETAIL system variable to 2. The materials of the composition plies will display.



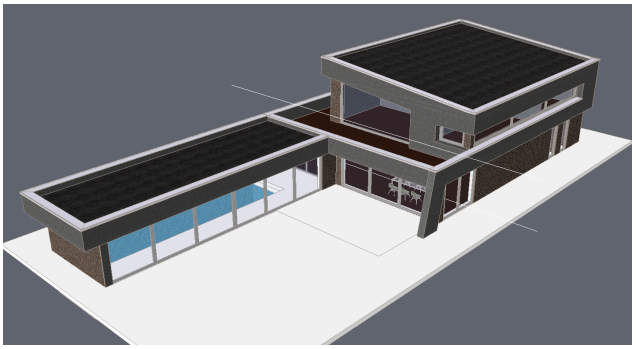


*Figure 1: Level of detail = 0,Figure 2: Level of detail = 2*

2 Open the **BIM Compositions** panel.

For more information about compositions, see the article BIM compositions.

3 Double-click on the composition **Cavity Wall, Brick**. The **Compositions** dialog box will display.

4 Double-click the material **Facing Bricks, Hand-formed** to edit the ply material. The **Physical Materials** window displays.

5 Click on the material image under **Appearance** > **3D** to change the render material for the ply.

**Editing render materials in BricsCAD**

1 To edit all the render materials used in your drawing, type in MATERIALS in the Command line.

### 20.7.11.8 Keywords in Enscape

**Grass**

Any time the word ‘Grass’ is used in a material name, grass will be rendered in Enscape. You can use diffuse colors to choose options of green, but also texture maps can be used as well.

1 Go to the **Materials** dialog box.

2 Click on **Add New Material**.

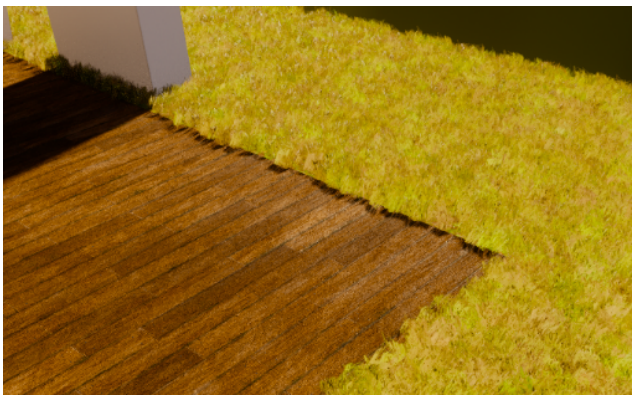
3 Give the new material a name that includes the name 'Grass', for example, ‘Grass_ground’.

4 Close the **Materials** dialog.

5 Select the solid that you want to render in grass.

6 Open the **Properties Panel**.

7 Change the property **3D Visualization** > **Material** from **ByLayer** to **Grass_ground**.



**Water**

Any time the word 'Water' is used in a material name, water will be rendered in Enscape. You can also create your own water material by including the word 'Water' in the material name and give it a diffuse color in a shade of blue.

1 Select the solid that you want to render in water.

2 Open the **Render Materials** panel and choose the folder named **Water**.

3 Drag and drop material **Water Blue Two** into the drawing.



### 20.7.11.9 Saved views in Enscape

1 Navigate in the BricsCAD drawing until you get a view that you want to save.

***Прим.:*** Make sure your viewing style is in **Perspective** to have a more accurate saved view in Enscape. You can check this in **Properties Panel** > **View** > **Perspective** > **ON**. And close the **Properties Panel** afterward.

2 Click on **Create View** in the Enscape ribbon or type -VIEW in the Command line.

**You are prompted**: View [? to list/Delete/Orthographic/Restore/Save/Window]:

3 Type **S** and press Enter to save the view.

**You are prompted**: Save view as:

4 Name the view and press Enter to confirm.

5 You can delete or adjust existing views by clicking **Views** in the Enscape ribbon **or** type in VIEW in the Command line.

6 In the Enscape window: on the right-hand side of the window, you can find all the saved views in the drawing.

7 Click on the view you want to see to switch between views.




### 20.7.11.1 Render images with Enscape 0

1 Navigate through the Enscape window.

2 **Or** you can click on a saved view on the right side of the Enscape window. The saved view will display.

3 Go back to BricsCAD and select **Render Image** in the Enscape ribbon to render the current view or type in ENSCAPESCREENSHOT in the Command line.

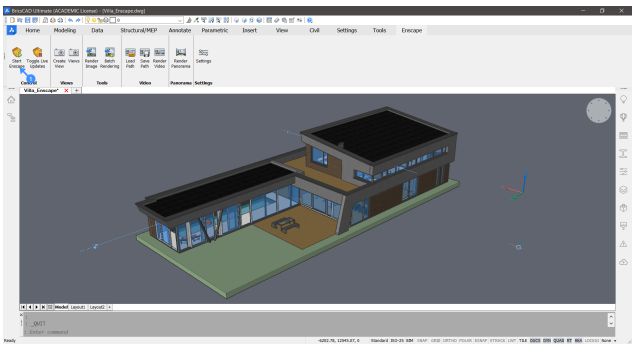
4 Select **Batch Rendering** in the ribbon, or type in ENSCAPEBATCHSCREENSHOT in the Command line, to render all the saved views to a folder.

5 Specify a destination file to save the rendered image and click **Save**.

The rendered image is now created.

### 20.7.11.1 Render a video with Enscape 1

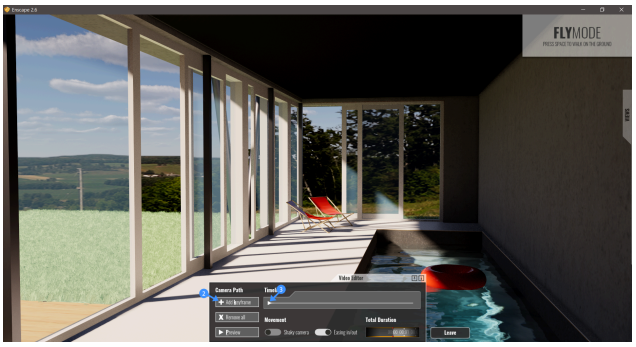
1 Click on **Start Enscape** in the ribbon or type in STARTENSCAPE in the Command line.



2 In the Enscape window: type K to open the **Video Editor**.

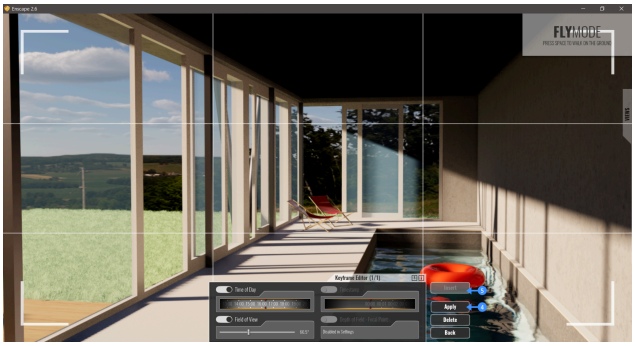
3 Click **Add Keyframe** (2) to add the first keyframe.

4 Click on the triangle to open the keyframe settings (3).



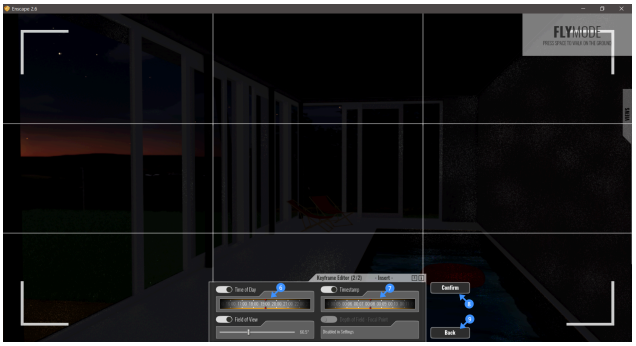
5 Change the parameters as required.

6 Click **Apply** (4).



7 Click **Insert** (5) to add a second keyframe.

8 Change the **Time of Day** to 7 pm (6) and the **Timestamp** to 8 seconds (7).



9 Click **Confirm** (8).

10 Click **Back** (9).

***Прим.:*** Before you click **Leave** to exit the **Video Editor**, your video path will not be automatically saved. So, before you exit your **Video Editor**, you should save your video path using the command ENSCAPESAVEVIDEOPATH.

11 Go back to BricsCAD and click on **Render Video** in the Enscape ribbon. Specify a destination file and

click **Save**. The video is now rendered and saved.



#### 20.7.11.1 Export a standalone .exe
#### 2

The Executable export function provides the same experience and quality that Enscape does, but does not require Enscape and BricsCAD to run.

1 Click **Export Standalone** in the Enscape ribbon or type in ENSCAPECREATESTANDALONE in the Command line.

2 A new dialog will pop up asking you where to save your standalone file. Choose your destination file, name your standalone and click Save.

### 20.7.12 Generating Drawings

#### 20.7.12.1 Commands

BIMSECTION, BIMSECTIONUPDATE, BIMSECTIONOPEN

#### 20.7.12.2 About BIMSECTIONUPDATE

BIMSECTIONUPDATE generates section drawings from BIM Section entities. The command can be used either by selecting a BIM Section entity in the 3D model or by selecting an existing section in a generated drawing. The result is placed in a new drawing, saved to the same folder as the model drawing unless a different folder is specified in the **Destination File** property of the BIM Section entity.



For more information about this command, visit the Command Reference article BIMSECTIONUPDATE.

#### 20.7.12.3 Procedure: generating BIM sections

BIM Sections can be generated in 2 ways:

- In the model space.
- Using the BIM **Project Browser**.

**Model space**

1 Select one or more BIM section entities.

2 **(Optional)** Edit the section properties (e.g. **Scale**, **Name**, **Destination File**).

3 Choose **Generate Section** in the **Model** command tab of the Quad.

BricsCAD reports the number of generated sections and their name(s) in the Command line e.g.:

8 section(s) were generated:

D:\Bricsys Reports\V16\Villa\Elevations.dwg

D:\Bricsys Reports\V16\Villa\Plans.dwg

D:\Bricsys Reports\V16\Villa\Sections.dwg

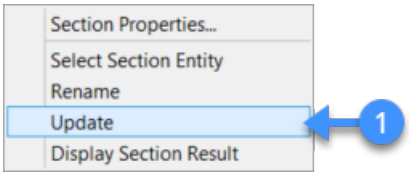
In the example above 2 plans, 4 elevations, and 2 cross-sections were generated and exported to 3 drawings.

**Project browser**

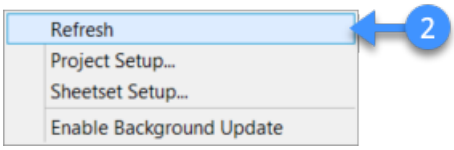
1 In the **Sections** segment of the **Project Browser**, right-click the section.

2 **(Optional)** Choose **Select Section Entity** to change its properties in the properties panel.

3 Choose **Update** (1).




4 Choose **Refresh** in the project browser menu (2).




A new section is generated in the **Sheets** segment of the **Project Browser**.

| | |
|---|---|
| **Notes** | • BIM_SECTIONS_PREVIOUS layer: the result of the previous section generation is stored on the dedicated BIM_SECTIONS_PREVIOUS layer. This is frozen by default.<br>• BIM_VIEWPORTS layer: generated viewports are placed on the dedicated BIM_VIEWPORTS layer if desired.<br>• The **SheetSet manager** automatically and silently adjusts the file paths after the project folder has been relocated if the same folder is used for all files. |

### 20.7.12.4 Open BIM sections

BIMSECTIONOPEN navigates between the model drawing containing a BIM Section entity and the corresponding generated drawing. The command works bi-directionally.



For more information about this command, visit the Command Reference article BIMSECTIONOPEN.

BIM Sections can be opened in different ways:

- From the 3D Model.

- Using the BIM **Project Browser**.

**From the 3D model**

To open a 2D document from the 3D model:

1 Select one or more BIM sections.

2 Choose **Open Model** in the **Model** command group of the Quad.

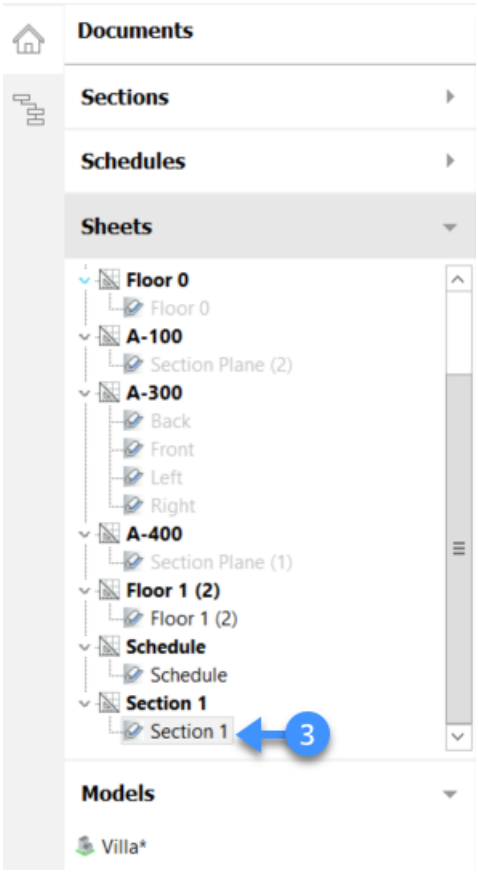
3 The 2D section drawing(s) open(s).

**Project browser**

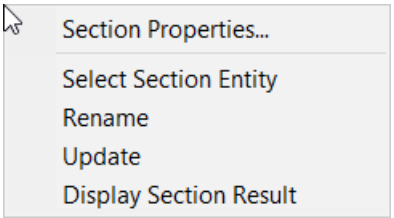
To open the 2D section drawings from the **Project Browser**, do one of the following:

- Double click the sheet view in the **Sheets** segment of the **Project Browser** (3).

  The 2D section drawing opens with the selected sheet view.




- In the **Sections** segment of the **Project Browser**, right-click the section and choose **Display Section Result** in the context menu.

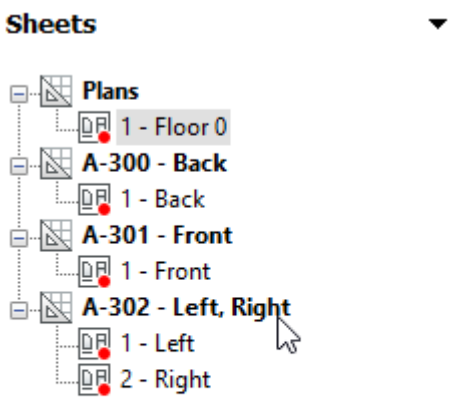


### 20.7.12.5 Update BIM sections

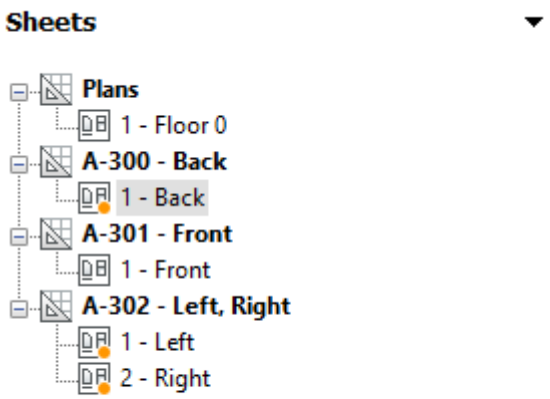
BIM Sections can be updated in different ways:

- In the 3D model.
- In the generated 2D documentation file.
- In the BIM **Project Browser**.

When the 2D documentation is no longer in sync with the 3D model, the corresponding sheet views display a red mark in the **Sheets** segment of the **Project Browser**.




Type BIMIFY command, then the corresponding sheet views display an orange mark in the **Sheets** segment of the **Project Browser**.



### 20.7.12.6 Update BIM sections in your 3D model

To update BIM Sections via your 3D model using the same process as used to generate views.

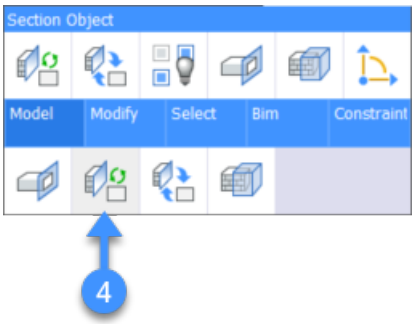
**Update BIM sections in the 2D documentation file**

A red frame around the section result indicates that the BIM Section generated block is no longer in sync with the 3D model and needs to be updated. This can be done in **Model** space or in the **Layout**.

**Update a BIM section in model space**

1 Place the cursor over the section block to update.

2 Choose **Update Section** in the **Model** command group in the Quad (4).

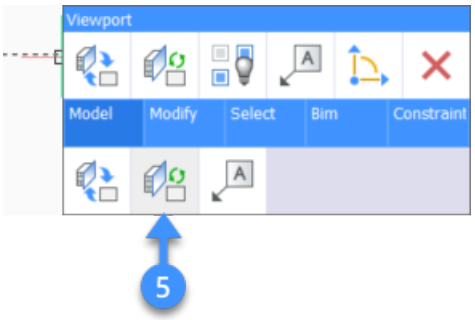



**Update a BIM section in a layout**

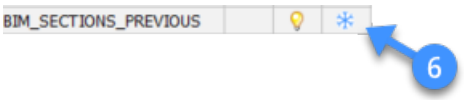
1 Click the **Layout** tab.

2 Place the cursor over the viewport of the BIM Section to update.

3 Choose **Update Section** in the **Model** command group of the Quad (5).



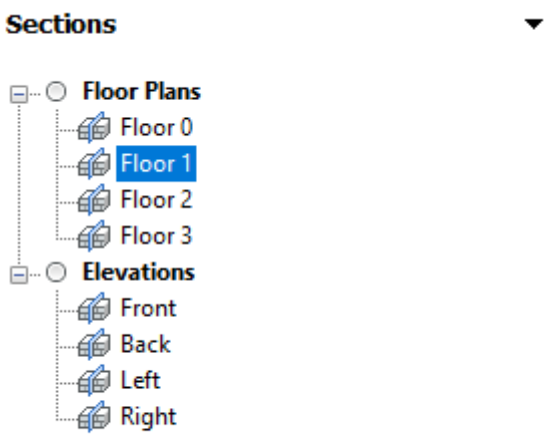

The result of the previous section generation is stored on the dedicated BIM_SECTIONS_PREVIOUS layer. Thaw the layer (6) to check the modifications.




The content of the BIM_SECTIONS_PREVIOUS layer displays in magenta.

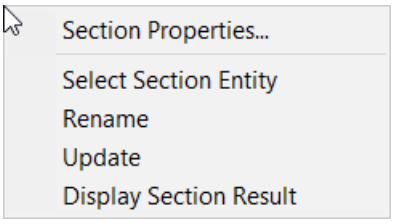


**Update BIM sections in the project browser**




1 Right-click on a section in the **Project Browser**.

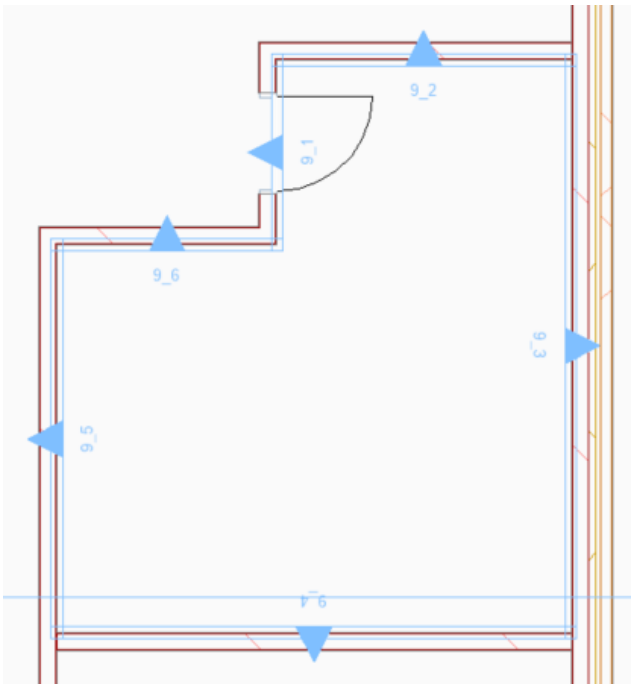
2 Select **Update** in the context menu.




### 20.7.12.7 Section indicators

A generated plan and reflected ceiling plan provide section indicators for interior elevations and detail sections if they intersect with the section plane.

The section indicators are stored on the dedicated BIM_SECTIONS layer. To change the visibility of indicators, turn the layer ON or OFF.




## 20.7.13 Working with Point Clouds in BIM

### 20.7.13.1 Commands

POINTCLOUDDETECTFLOORS, POINTCLOUDFITPLANAR, POINTCLOUDPROJECTSECTION

### 20.7.13.2 About POINTCLOUDDETECTFLOORS

The new POINTCLOUDDETECTFLOORS command generates volume sections for each floor found in a point cloud representing a building. The detection is based on regions of points with similar Z-coordinates. The generated volume sections can help in navigating point clouds of buildings.

Section planes work on point clouds as well, they can be used to show parts of point clouds. The difference between point cloud crops and section planes is that point cloud crop only clips the point cloud while section planes will clip all geometry in your drawing.

If the point cloud building has multiple floors, there will be multiple section volumes.



For more information about this command, visit the Command Reference article POINTCLOUDDETECTFLOORS.

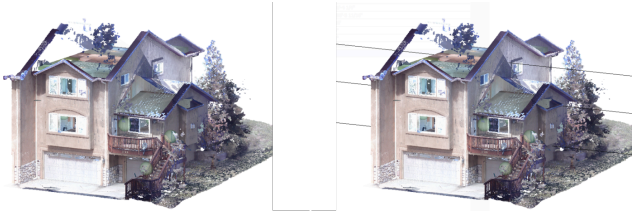
**Detecting floors from Pointclouds**

1 Type POINTCLOUDDETECTFLOORS in the Command line.

2 **You are prompted**: Operate on full point cloud or a selection? [Full/Selection]

- **Full**: The command will search the entire point cloud for floors and generates a section for each of them.
- **Selection**: The command only runs on the selected area.

3 Press Enter.



***Прим.:*** This volume selection is a detail section of the selected building. You can manipulate the section box if you want to include more or fewer points from the point cloud using the green grips.

### 20.7.13.3 About POINTCLOUDFITPLANAR

POINTCLOUDFITPLANAR is used to semi-automatically make 3D geometry based on a point cloud scan. It will create a planar surface or solid after a selection of one point in a point cloud. The points that seem to be in a plane are never exactly in one plane, therefore a threshold value is set as a property of the point cloud entity. This also works in bubble view.

Select planes in the point cloud to generate 3D solids based on the given geometries. These 3D solids can be further used in the process of creating a full BIM model.



For more information about this command, visit the Command Reference article POINTCLOUDFITPLANAR.

**Creating 3D solids from the bubble view**

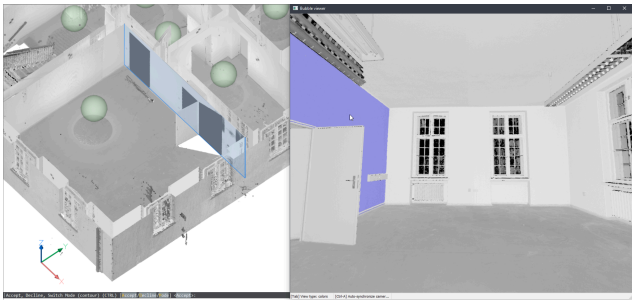
If the bubble viewer is open before launching the command, BricsCAD expects you to select in the bubble viewer. The cursor will give you a preview of the direction of the plane. When you click you get a preview in both bubble view and model view. You can toggle between 2 shape representations using the CTRL-key.

1 Double-click a green point cloud bubble to open the **Bubble viewer**.

2 Type POINTCLOUDFITPLANAR in the Command line.

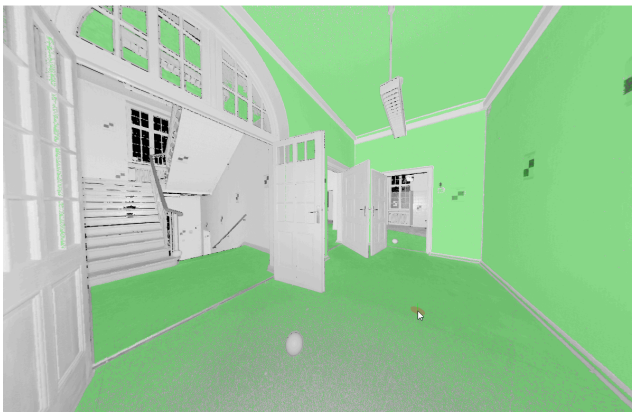
3 Select the desired wall.

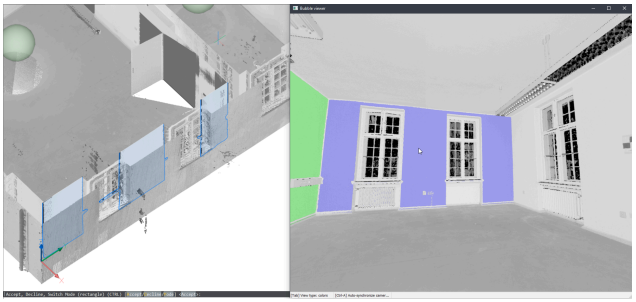




4 Press A to accept the selection.

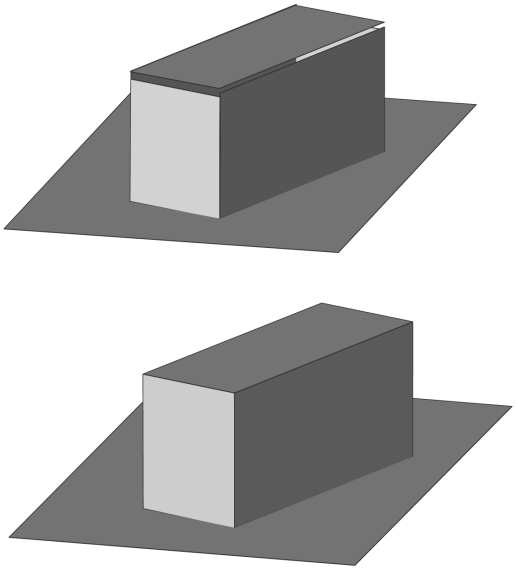
5 **(Optional)** Select more walls.





6 Close the **Bubble viewer**.

The surfaces are created and you can edit them to closed spaces with LCONNECT or other connecting tools.



**Create 3D entities from the model space**

You can also use this command in the model space when the bubble viewer is not open. BricsCAD will ask you to select a point of the point cloud in model space. Depending on the size of the cropped point cloud, it takes more time but it has 2 advantages by searching multiple scan positions:

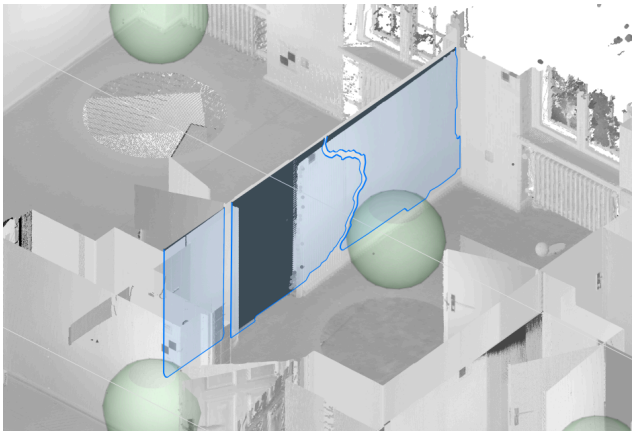
- It can create larger surfaces where only parts are visible in each scan position.
- It can detect wall and slab thickness since it can take the opposite surface into account.

1 Type POINTCLOUDFITPLANAR in the Command line.

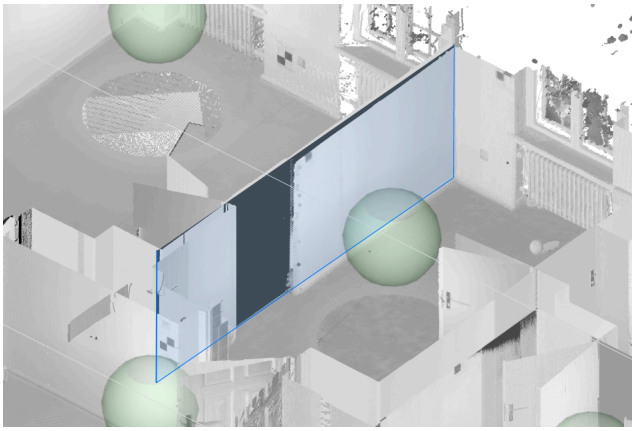
2 Select a wall where you want to create a plane or solid.

3 You are prompted: Accept, Decline, Switch Mode (contour) (CTRL) [Accept/Decline/Mode]:
   - Press A to create the wall.
   - Press D to decline.
   - Press M or CTRL repeatedly to switch between different modes.

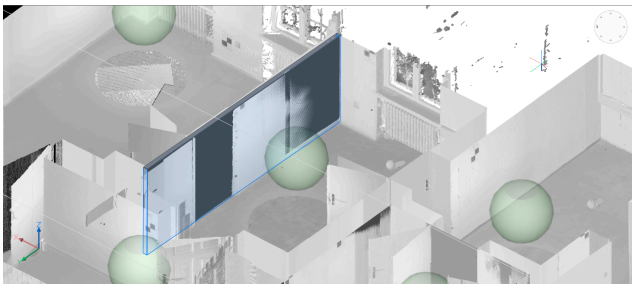
*Contour mode will create the best fitting plane:*



*Rectangular mode will create a rectangular plane around region of planar points:*



*Solid mode will create a solid from two parallel point cloud planes*:



***Прим.:*** If the point cloud exists out of multiple rooms, you can use the solid mode to automatically create solid walls.

4 Press A to create the wall.

These solids can be the base for a more complex BIM Model. Use modeling commands to complete the building and add BIM data with **Bimify**, **Automatch** and **Propagate** to quickly transform your point cloud into a complete BIM Building.

It may be helpful to turn **on** the **Point Cloud Closest Point Esnap** setting. This snap setting significantly improves your ability to select relevant point cloud points.

### 20.7.13.4 About POINTCLOUDPROJECTSECTION

POINTCLOUDPROJECTSECTION automatically generates a raster image with optional contour lines out of a defined section box. These lines can be easily manipulated into a detailed 2D drawing.

The command enables you to detect walls from the volume section of a point cloud based on a variety of wall detection options. You can create volume sections automatically for each floor in a building with POINTCLOUDDETECTFLOORS. You can use these sections to generate 2D lines to create a 2D floorplan or a vertical section. This is a background process and multiple sections can be processed in a queue. This way it is possible to run this command in full resolution on all sections.



For more information about this command, visit the Command Reference article POINTCLOUDPROJECTSECTION.

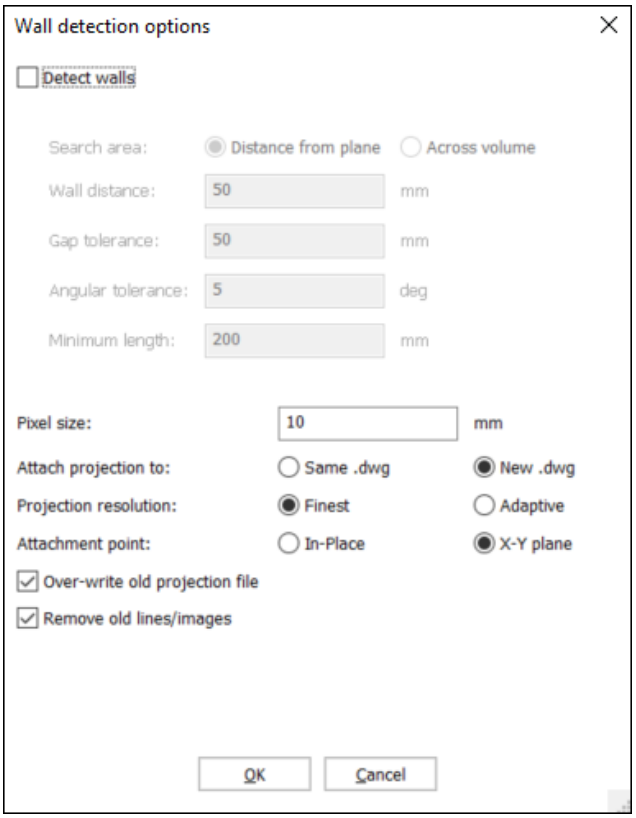
### 20.7.13.5 Procedure: creating a raster image from a point cloud

1 Type POINTCLOUDPROJECTSECTION in the Command line.

2 Select a **Section** volume and press Enter.

3 The **Wall detection options** dialog appears.

Edit the options to customize your projected section. Turn **Detect Walls** off.




4 Click **OK** to generate the raster image.



Point Cloud scan by Realistic Visual styles.

### 20.7.13.6 Procedure: detecting walls from a point cloud and generate in 2D

**Making horizontal sections from point clouds**

1 Type POINTCLOUDPROJECTSECTION in the Command line.

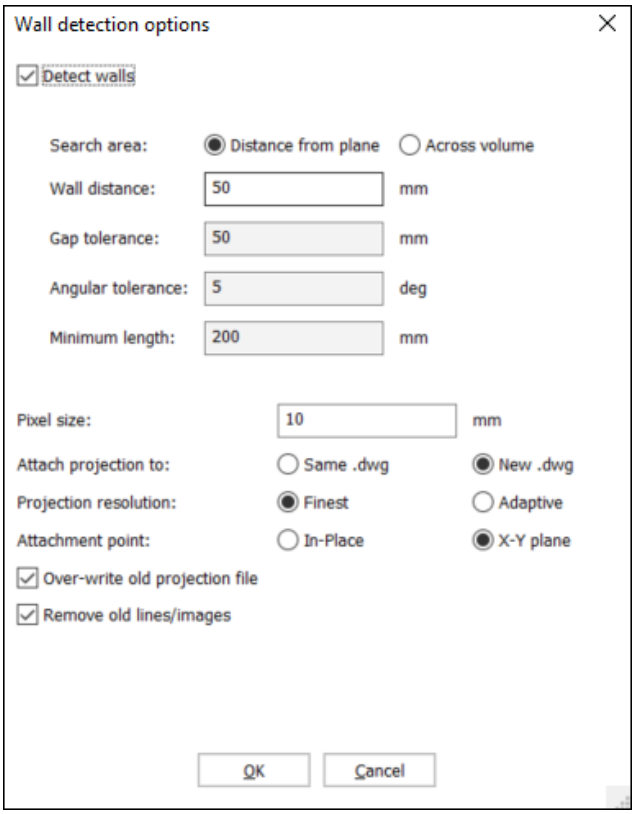
2 Select a section volume and press Enter.

3 The **Wall detection options** dialog appears.

Edit the options to customize your projected section.

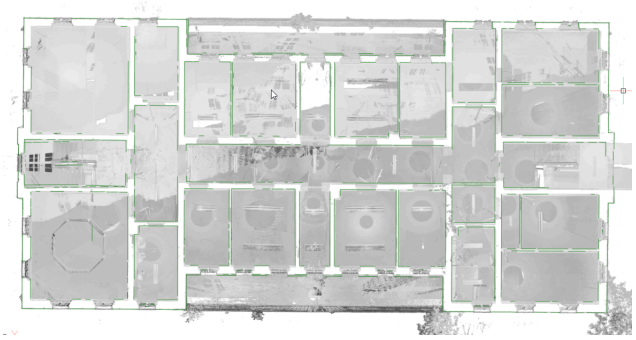
***Прим.:*** See the command reference page for more information about the customization options;

***Прим.:*** Turn on **Detect Walls** to generate the green lines as seen in the image below.




4 Press Enter to create the projection.

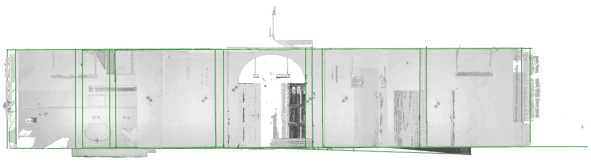
***Прим.:*** If there are some missing lines, you can draw them yourself by tracing the background image.



**Making vertical sections from point clouds**

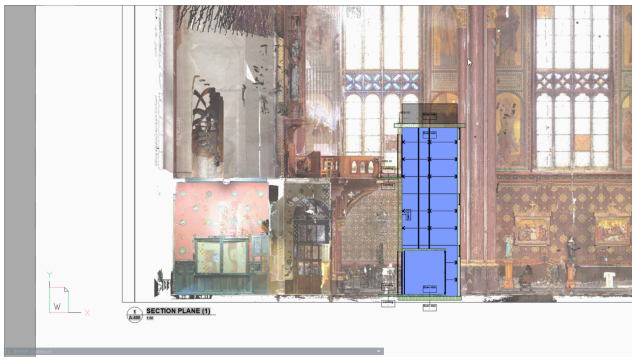
Make a vertical section box on the place you want the vertical projection.

Make sure that the blue round arrow aims to the side you want your projection.



### 20.7.13.7 Generating BIM sections with point clouds

In some cases it is not necessary to recreate the existing building. Background images can give so much more context to the design documents. These can be used to verify the created 2D geometry, in high quality scans these images can also be used as graphical material. For example, as a background image for a BIM model in renovation projects where modern interventions are made in historical buildings.



Point Cloud scan by Realistic Visual styles.

## 20.7.14 Interior elevations

### 20.7.14.1 Command

BIMSECTION

### 20.7.14.2 About

Interior elevations show wall elevations and floor plans for individual Spaces. They provide the details and dimensions of building elements. This can be particularly useful for interior designers or to make a layout of power sockets and light switches. In BricsCAD they can be automatically defined by a BIM Space.

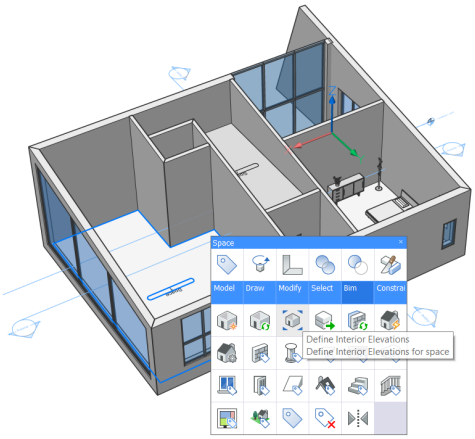


For more information about this command, please read BIMSECTION.

### 20.7.14.3 Create interior elevations from a BIM space

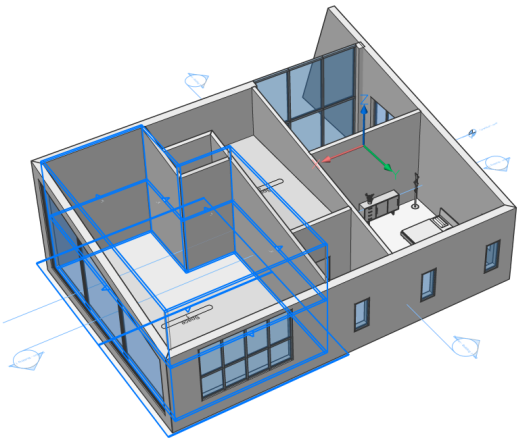
1 Hover the cursor over a Space Tag and select **Define Interior Elevations** from the **BIM** Tab in the Quad
**Or** type BIMSECTION in the Command line, followed by i and Enter to choose the interior option.

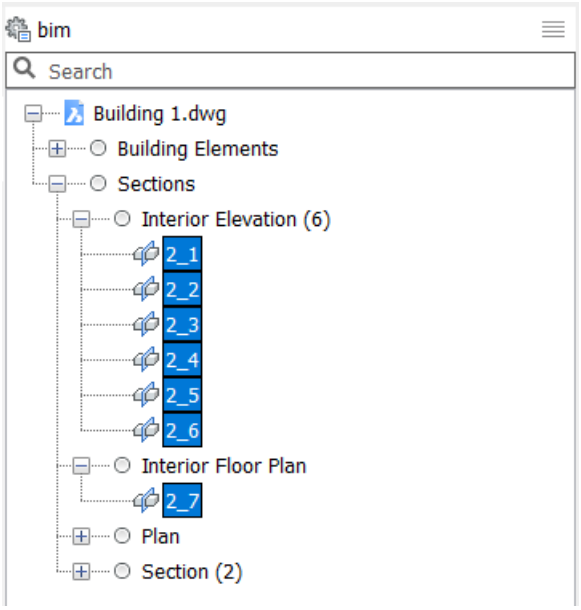
**You are prompted**: Select BIM Spaces to generate Interior Elevations or [Attach section]:




2 (Skip if you used the Quad) Select the BIM Space for which you want to create the interior elevations and hit Enter.

3 The interior elevations for each wall and the interior floor plan are now defined. They can be found in the **Structure Browser**.

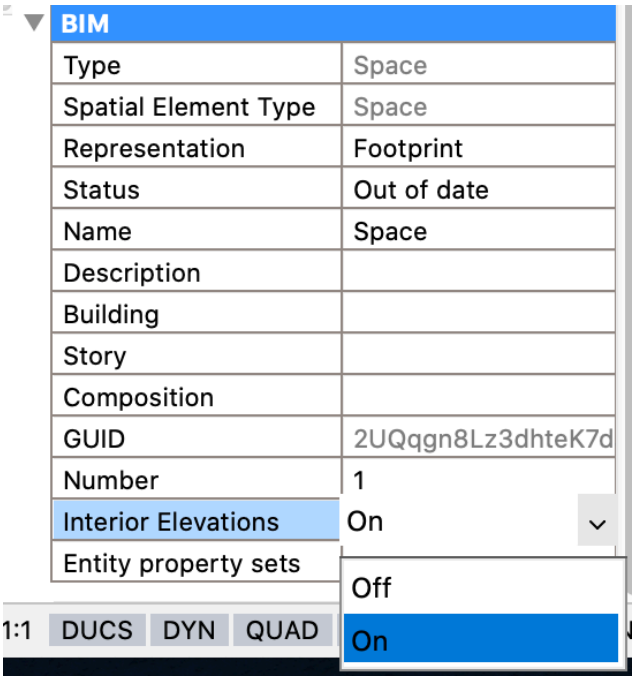





*Six interior elevations and one interior floor plan have been created and will appear in the structure browser.*

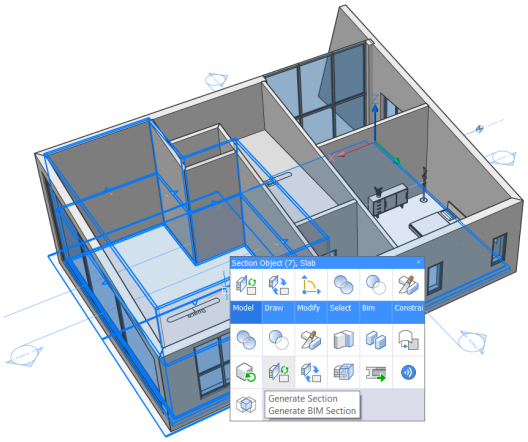
***Прим.:*** If there already are interior sections for a Space, running BIMSECTION again on that Space will have no effect. To regenerate the interior elevations, you must delete the existing interior elevations in that Space and run BIMSECTION again.

4 To show or hide the interior elevations you can use the HIDE/ISOLATE/UNISOLATE objects commands **or** you can go to the properties of the Space and switch the interior elevations on and off in the **BIM** tab.



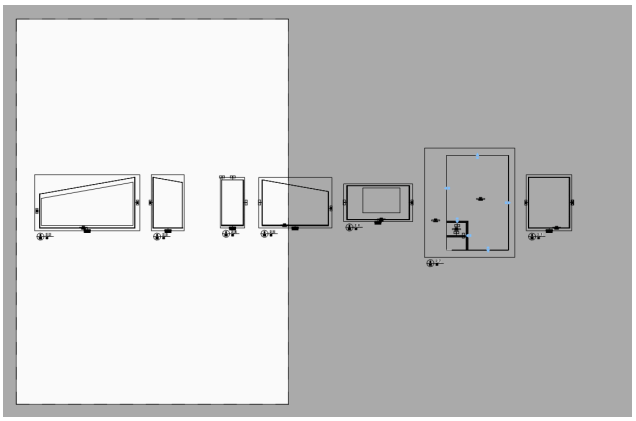

5 Select all elevations, as illustrated above and select **Generate Section** from the Quad to generate the

interior elevations on a sheet.




6 To open the section results click **Open Model**whilst the section planes are selected.

The sheet is automatically saved in the same folder as the model drawing. The following elevations have been produced and can be edited for the selected Space in the model above:



***Прим.:*** The section drawings are stored in the file **InteriorElevations** by default. If you generate sections of a second Space, the sections will be added to the same file as the previous ones.

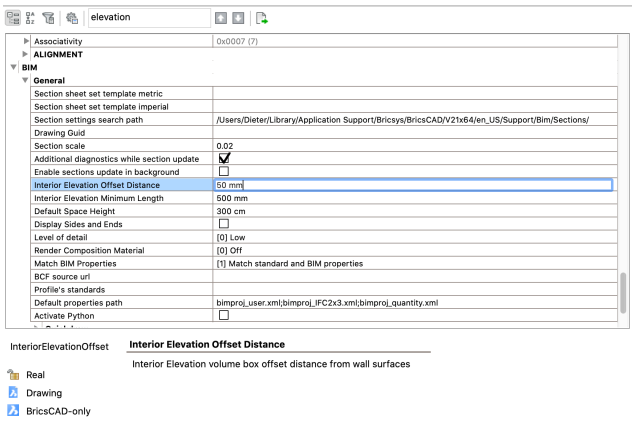
If you need more information on how to generate drawings, visit the Generating Drawings page.
If you want to annotate your drawings, see **Annotate** > **Creating linear dimensions**.

### 20.7.14.4 Edit the size of the interior elevations

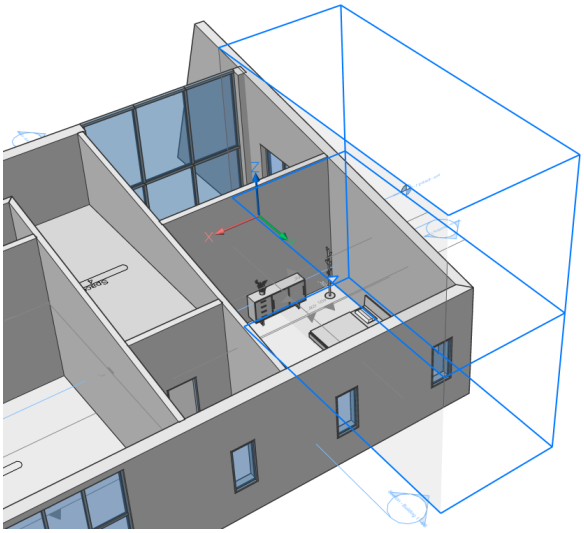
It's possible to edit the size of these sections to include furniture for example. If you create interior elevations for a BIM Space, the offset value is 50mm by default. To include furniture in a section, you have to change this value.

1 To change this setting type INTERIORELEVATIONOFFSET in the Command line and hit Enter. Type a new value and press Enter **or** click on two points in your drawing and the distance between them will be selected.

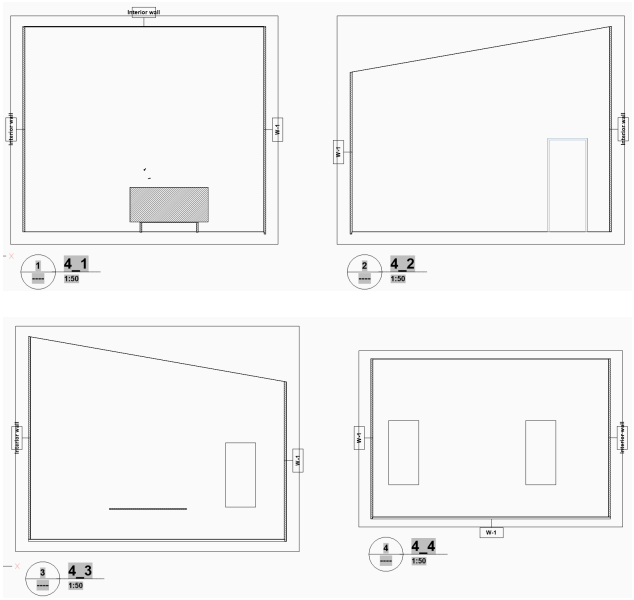
You can also change the offset by typing Settings in the Command line, search for elevation and adapt the **Interior Elevation Offset Distance** value.



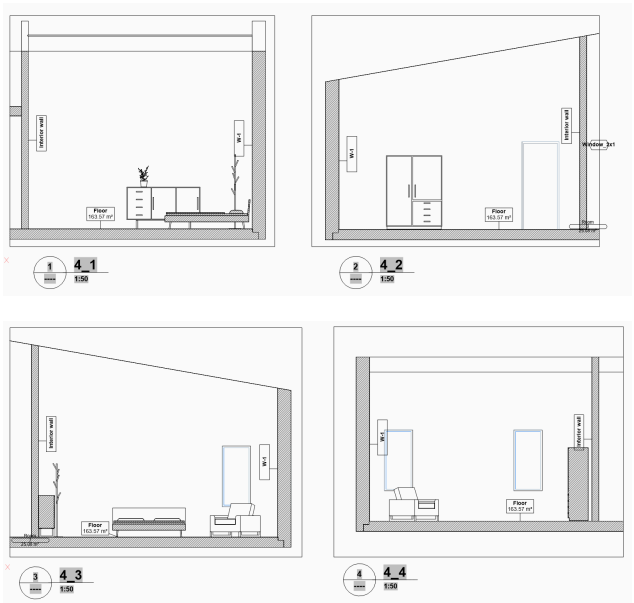

2 If you make new Interior Elevations, the offset distance will be the chosen value. This is also visible when you select the interior elevations in your drawing:



3 This results in more detailed section sheets with the furniture included.




*Without specified offset distance (50 mm)*




*With specified offset distance (2100 mm)*

## 20.7.15 Schedules

### 20.7.15.1 Open the BIM project browser panel

1 Move the cursor over a toolbar, ribbon item or a dockable panel.

2 Right-click.

3 Choose **Panels** > **BIM Project Browser** in the context menu.

See the article Project browser, for more information about the BIM **Project Browser**.

### 20.7.15.2 How to create schedules

Schedules are exports, entity properties, block attributes and drawing information to tables on sheets and tables.

**Step 1: create a project**

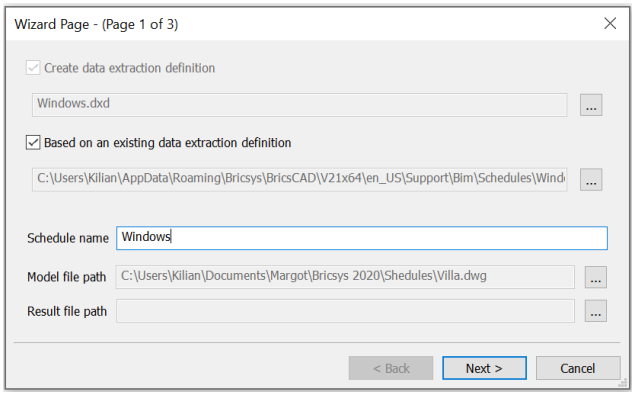
For more information about creating a project, see the article Project browser.

When the project is created, right-click in the **Schedules** section. Click **Add Schedule (Wizard)...** to open the wizard.

**Step 2: the schedule wizard**

A wizard of 3 pages displays, this will help you to create a schedule.

- Page 1




**(1) Create data extraction definition** creates a new .dxd (data extraction definition) file:

a Turn the option on.
b Click the **Browse** button (...).
c In the file dialog box, select a folder.
d Enter a name in the **File** name field.
e Click the **Select Folder** button.

***Прим.:*** By default, the .dxd file will be placed in the project folder. To change this path click the **Browse** button.

**(2) Based on an existing data extraction definition** uses an existing .dxd file as the template for the new one:

a Turn the option on.
b Click the **Browse** button (...).

***Прим.:*** The .dxd files are stored in ..\bim\schedules serve as templates.

c In the file dialog box, select a .dxd file.
d Click the **Open** button.

***Прим.:*** By default .dxd files are saved in the ...\Bim\Schedules subfolder of the **Support** folder. You can use newly created custom .dxd files too. Also, you can create a .dxd files by yourself

using a text editor. Please scroll down for the procedure. "How to create and edit your own Data Extraction file" in this article.

You can either choose a new name for your schedule or keep the default name.

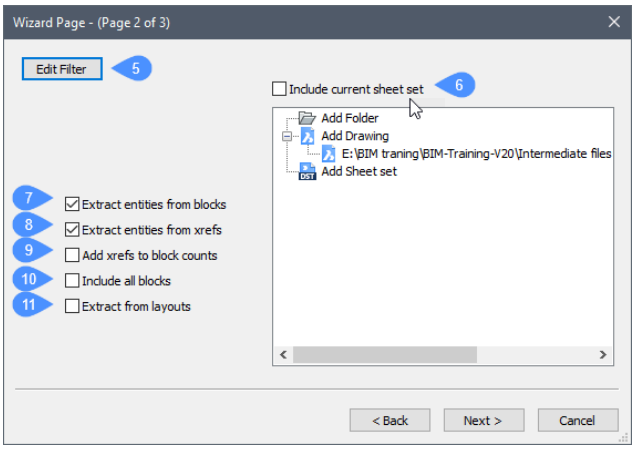
**(3) Model file path**: The model path links to the model. This is filled in automatically if the wizard is started in the model file. If the wizard is started in another file from the project, link the model file:

a Click the **Browse** button (...).
b In the file dialog box, select the model file.
c Click the **Open** button.

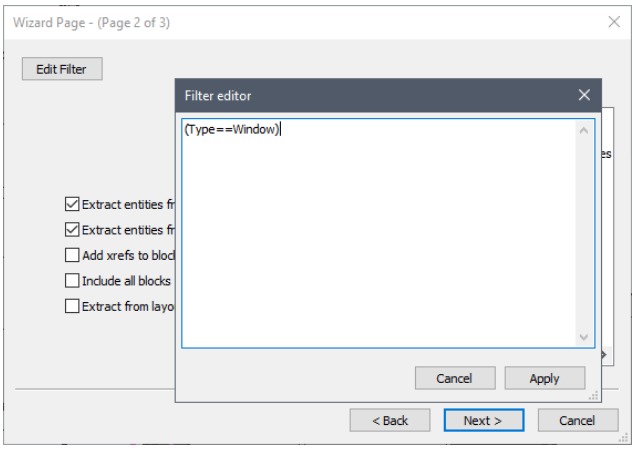
**(4) Result file path**: The **Result file path** determines where the schedule is created. When left blank, the Schedule is created as a new sheet:

a In the file dialog box, select the sheet file.
b Click the **Open** button.
Click **Next** for the next page.

- Page 2




**(5) Edit Filter**: The filter selects which BIM entity types are included in the Schedule. Use an existing data extraction definition to automatically define the filter. See the Editing Data Extraction Definition (.dxd) files to learn more about the filter syntax.



**(6) Include current sheet set**: Adds the current sheet set for data extraction.

**(7) Extract entities from blocks**: Adds entities in blocks which will also be processed in the schedule.

**(8) Extract entities from xrefs**: Adds all entities in xrefs which will also be processed in the schedule.

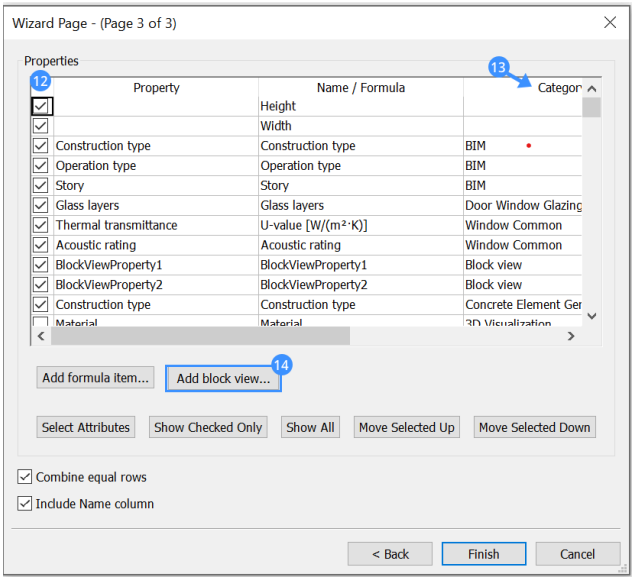
**(9) Add xrefs to block counts**: Counts selected xrefs as blocks.

**(10) Include all blocks**: Includes all blocks into the schedule.

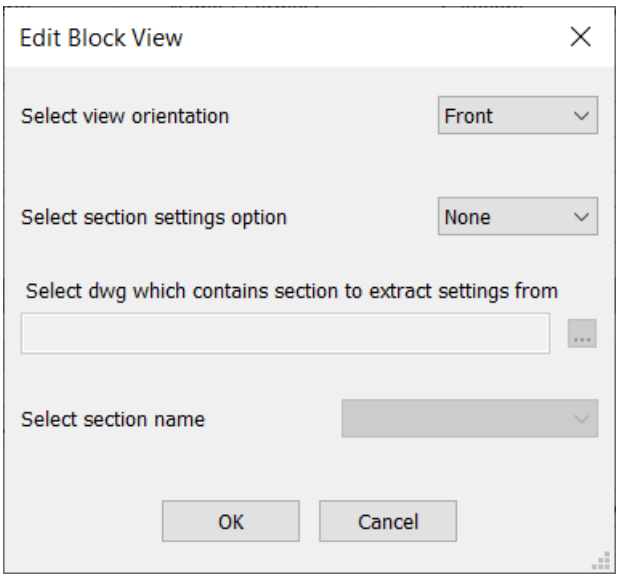
**(11) Extract from layouts**: When toggled on, all entities layouts will also be processed in the schedule.

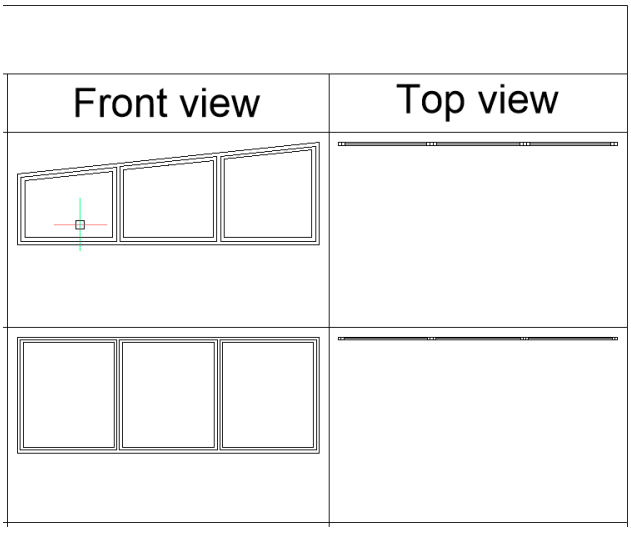
***Прим.:*** The Wizard automatically selects all entities in the drawing.

- Page 3



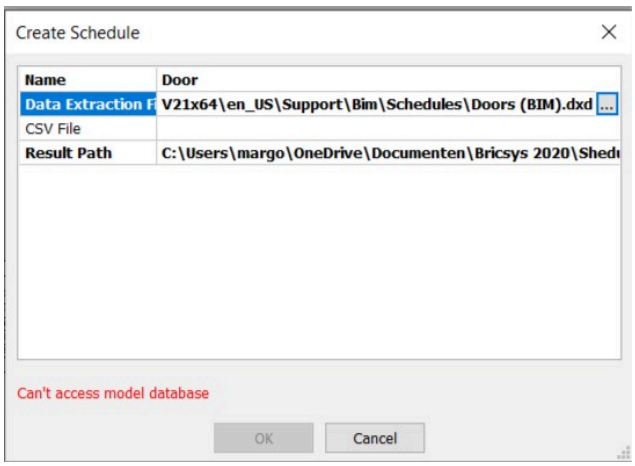

**(12)** Select the properties to be included in the schedule: click the checkbox to check/uncheck a property: selected (☑) or not selected (☐).
(**13**) Click on the **Category** column to change how the property is shown in the schedule.
(**14**) To include plan and elevation symbols, click on the **Add block view...** button.



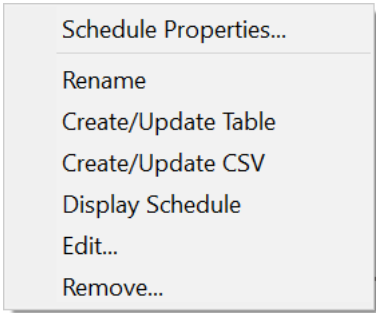

To sort the list, click the title in the column heading. Click again to sort in reverse order.
Click **Finish** to close the wizard (or click **Back** to switch back to page 2).

When the project is created, right-click in the **Schedules** section. Click **Add Schedule (From dxd)...** to open the wizard.




### 20.7.15.3 Update and display schedules

1 Right-click on the new schedule (created in the BIM **Project Browser**). A context menu appears:




| | |
|---|---|
| **Schedule Properties...** | **Name**: Renames the schedule.<br>**Model**: Displays the link to the model file.<br>**DWG Handle**: Displays the DWG Handle.<br>**Data Extraction File**: Displays the link to the Data Extraction Definition.<br>**CSV File**: Displays the link to the .csv file.<br>**Result Path**: Displays the resulting path. |
| **Rename** | Renames the schedule. |
| **Create/Update Table** | Creates a sheet with the schedule or updates the sheet. |
| **Create/update CSV** | Creates a CSV file with the schedule information and saves it in the folder where de model file is located or updates the CSV file if it already exists. |
| **Display Schedule** | Displays the schedule in the resulting drawing. (If not already open, the drawing is opened. |

| | |
|---|---|
| **Edit...** | Opens the **Wizard Page** dialog box where you can edit your schedule. More information about this setup can be found in the procedure in this article. |
| **Remove...** | Removes the schedule. |

2 Click on **Update**.

3 Once the schedule is updated, click on **Display Schedule**. The schedule is shown in the desired sheet.
**Or** double-click on the schedule's name to display the schedule.

### 20.7.15.4 Procedure: create a window schedule

In this example, you will learn how to make a window schedule from scratch. You can create a door schedule in the same way.

**Step 1: open schedule wizard**

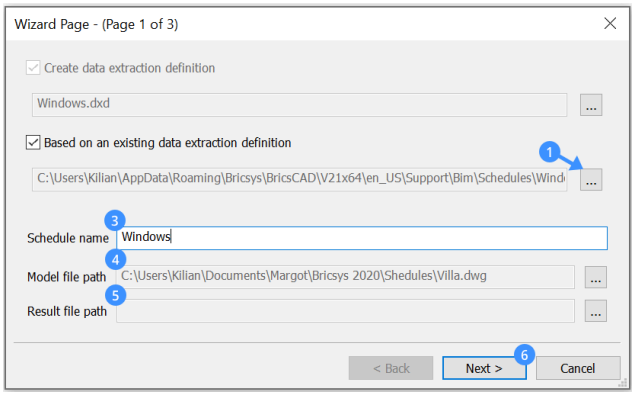
1 Open the BIM **Project Browser**.

2 Right-click on **Schedules**. A context menu appears.

3 Click on **Add Schedule (wizard)...** to open the wizard.

**Step 2: the schedule wizard**

- Page 1




For this example, use an existing data extraction definition.

a Click on the **Browse** (1) button. A folder pops up. This contains the predetermined data extraction definitions.

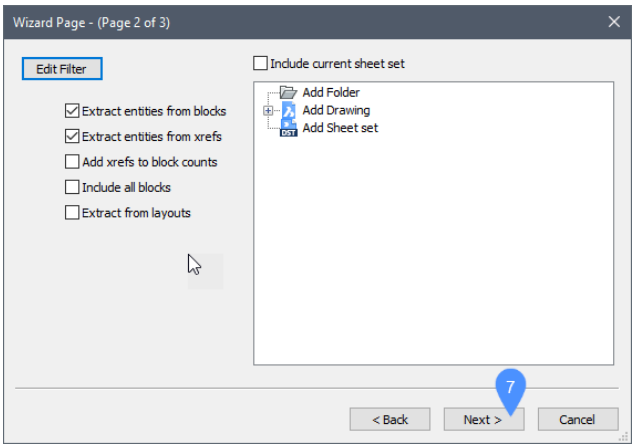
b In the folder, select *'Windows.dxd'*.

c Change the **Schedule name** (3) to, in this case, **Windows**.

d The **Model file path** (4) is correct because it was taken from the model file. To create the schedule on a separate sheet keep the **Result file path** (5) empty.

e Click **Next** (6).

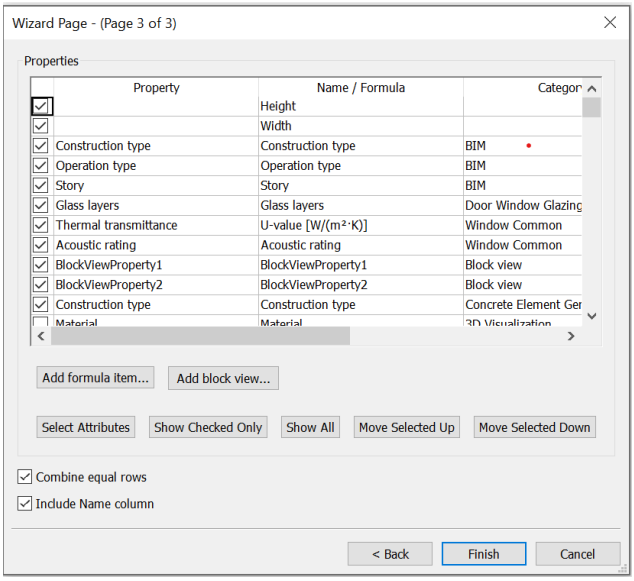
- Page 2




a The wizard automatically selects all the entities in the model.

b Click **Next** (7).

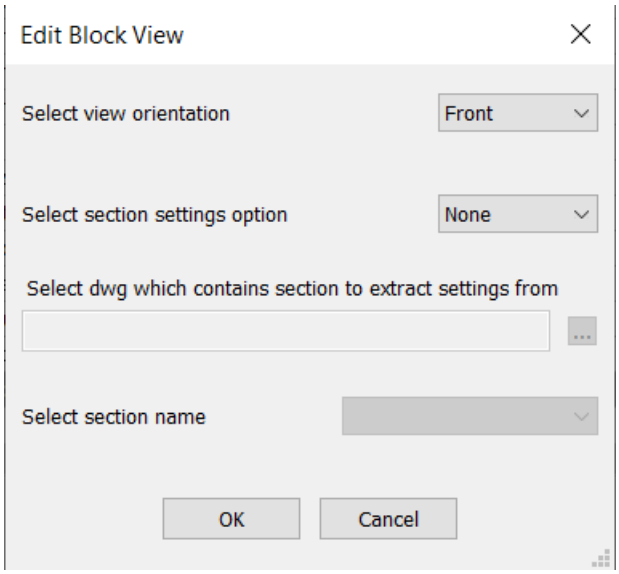
- Page 3




a Because you use an existing .dxd, the properties are already pre-selected. Views such as **Front** and **Top** are also pre-set.

b Click on the **Block view** category, the **Edit Block View** window appears.

c Choose the view.




d Click **Finish**.

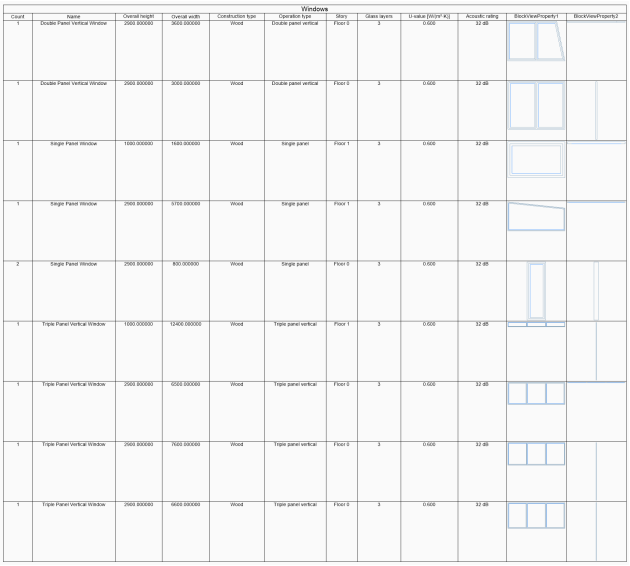
**Step 3: open the schedule**

1 Right-click on the new schedule.

2 In the context menu select **Create/Update Table**.

- Double click on the schedule.
- Right-click on the schedule and select **Display Schedule**.
- Go to **Sheets** and double-click on the schedule sheet. Open the schedule.

3 The result is as follows:



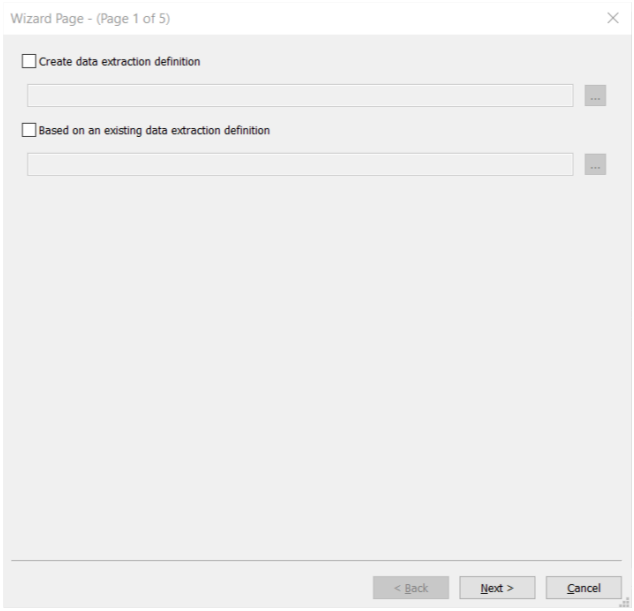
### 20.7.15.5 Procedure: how to create and edit your own data extraction file: furnishing elements

You can also create a Data Extraction File by yourself and use it to make new schedules. The DATAEXTRACTION command will, just like the Schedules wizard, make a new .dxd file with your settings, but it will also make a .csv file with your data in it.

1 Open your model file.

2 In the **Structure Browser**, select all Furnishing Elements.

3 Type in DATAEXTRACTION in the Command line. A wizard of 5 pages appears:

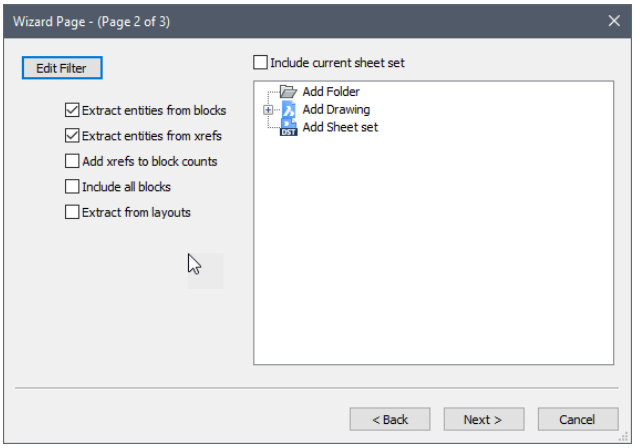



4 Check the **Create data extraction definition** box and choose a file. Name it “FurnishingElements.dxd” and save it in your project folder.

***Прим.:*** You can base you dxd file on an existing Data Extraction Definition file by checking **Based on an existing data extraction definition**. And choose the file you want to base on.

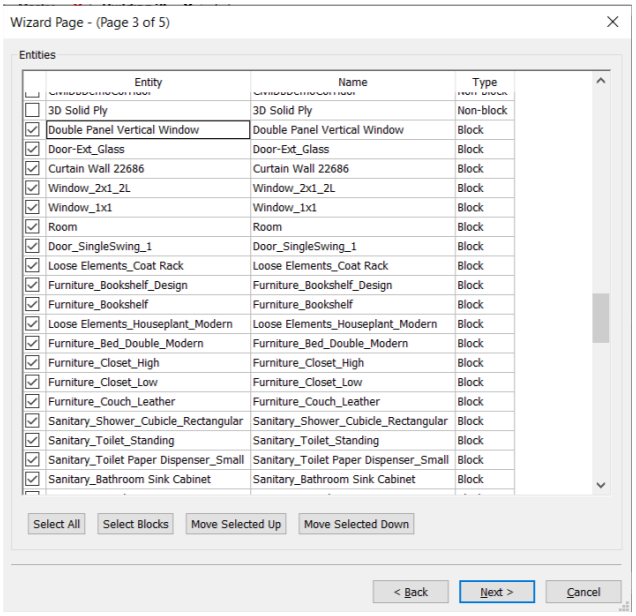
5 Click **Next**.

6 Select all entities you want to include in your schedule.



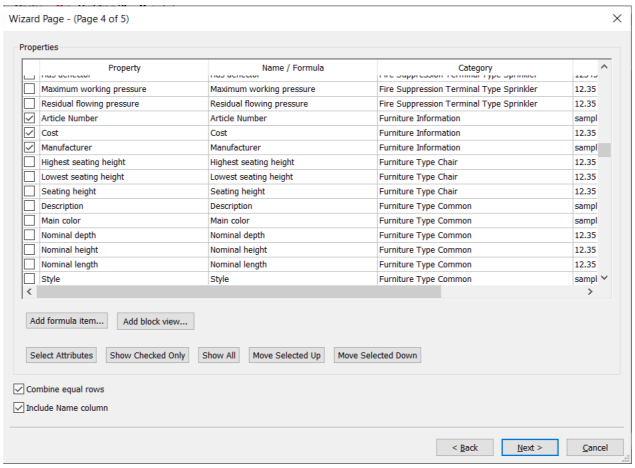

7 Click **Next**.

8 Select all entities you want to have a data extraction of.

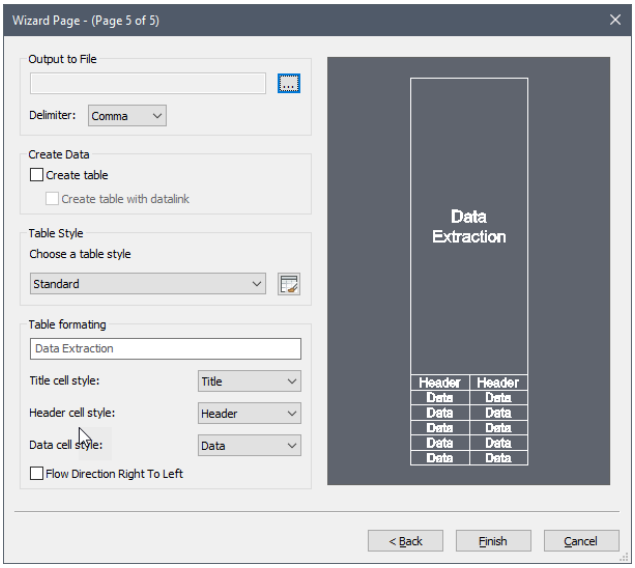



9 Click **Next**.

10 Check the all properties you want in the **Furniture Information**.




11 Click **Next**. The next page appears:




12 Click on the three dots next to the **Output to File**. Give in a new name for your .csv file and click Save. You can choose between different options for the delimiter, but it's best to use a semicolon.

13 You can create a Data table as well. When checking this, you will have the option to place the table any where inside your drawing either model space or paper space.

14 Click **Finish**.

15 This should have created 2 files: a .csv file containing the results of the data extraction (basically the same as a schedule, but in an external file) and a .dxd file. This .dxd file can now be re-used for creating a schedule. (See procedures above).

16 Open the .dxd file in a text editor. We can see 5 sections: **Settings**, **Entity Types**, **Properties**, **TableFormatOptions** and **SELECTION SET**.

17 Leave the **Settings** as they are.

18 The **Entity Types** are chosen because of the selection set you had active when using data extraction.

However, instead of explicitly choosing which types (and handles, see later) you want to extract data from, you can replace the explicit entity types by an asterisk (*), to define that we want to extract information from all entity types.

19 Leave the **Properties** as they are.

20 Leave the **TableFormatOptions** as they are.

21 You can delete the entire **SELECTION SET** section, if you don't want to explicitly define which entities you'll be extracting data from.

22 You can add a filter by adding a new section: The section title is called [Filter] The section content is (Type == Furnishing Element) (in the .dxd files that are given as example in the support folder, a Filter on Door and Window type is used).

23 Thus, the contents of the .dxd file should be as follows:

24 *[Settings]*

*ExtractFromBlocks=0*

*ExtractFromXrefs=1*

*CountXrefs=0*

*CombineEqualRows=1*

*IncludeNameColumn=1*

*FooterStyle=0*

*FlowDirectionRtoL=0*

*CVSdelimiter=1*

*IncludeLayouts=0*

*[Entity Types]*

*

*[Properties]*

*Furniture Information:Article Number|Article Number|*

*Furniture Information:Cost|Cost|%1u2%pr2*

*Furniture Information:Manufacturer|Manufacturer*

*[TableFormatOptions]*

*TableStyle=Standard*

*TitleCellStyle-TITLE*

*HeaderCellStyle-HEADER*

*DataCellStyle-DATA*

*[Filter]*

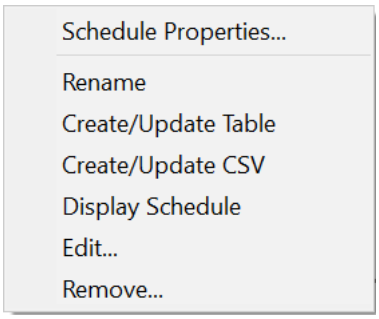
*(Type==Furnishing Element)*

25 Save this .dxd file in your project folder.

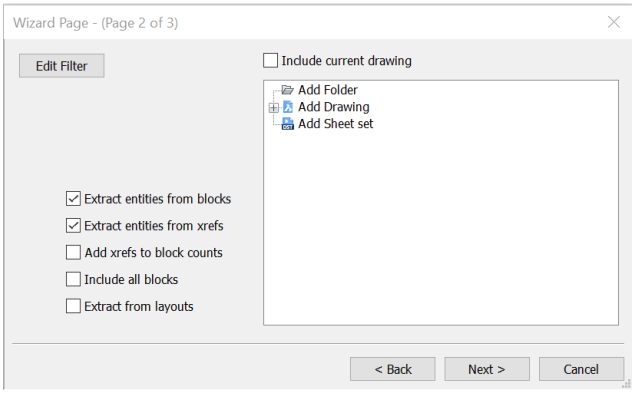
### 20.7.15.6 Procedure: how to edit a schedule

1 Go to the **Schedules** tab in the **Project Browser**.

2 Right-click the selected schedule. A context menu appears:




3 Click **Edit...** A Wizard of 2 pages appears:




4 Check or uncheck the options you want.

5 Click **Next**.

6 Check all properties you want to be in the schedule or uncheck all properties you want to leave out.

7 Click **Finish**.

8 Right-click on the schedule again and choose **Create/Update Table**.

9 Repeat this action but choose **Create/Update CSV**.

10 The table and CSV file are now updated. You can update and open your sheet with the schedule again and see the changes.

## 20.7.16 Spot elevation level

### 20.7.16.1 Command

BIMTAG

### 20.7.16.2 About

The Spot Elevation tag displays the absolute elevation of a selected point of a BIM object displayed in a section view, based on the **World Coordinate System (WCS)**.

As the Spot Elevation tag is an associative tag type, it can only be used in the **paper space** of a drawing sheet.

For more information on how to set up Sheets and create Projects, please visit the **Project Browser** article.

### 20.7.16.3 Procedure: how to place a spot elevation tag on a section sheet

1 Open a section sheet of the model. Right-click on the section in the **Project Browser** and select **Display View**.

2 Type BIMTAG in the Command line **or** click on **tag** in the **Draw** menu of the Quad.

3 **You are prompted**: [Manual/selection options (?)]

Type **M**t o choose **Manual** and press Enter.

4 **You are prompted**: [Auto/Current mleader style/change Tag type]

Type **T** to change the **Tag** type and press Enter.

5 **You are prompted**: [Tag/cOmposition/Load-bearing directions/Spot-elevation]

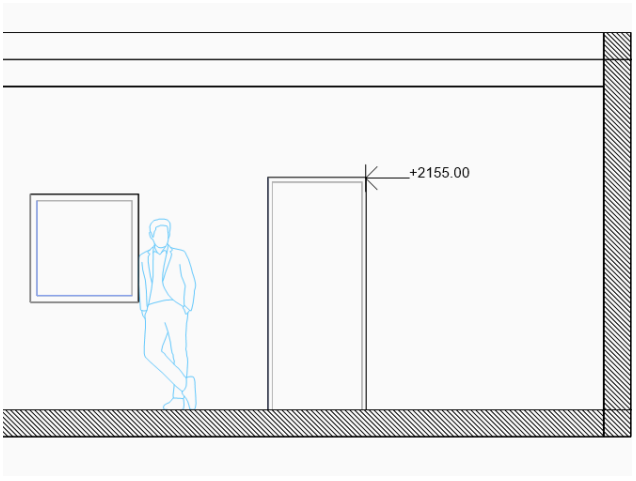
Type **S**to choose **Spot-elevation** and press Enter.

6 Click on a point on the edge of a BIM object to display the elevation height. The displayed value is the elevation level measured relative to level 0 of the model.

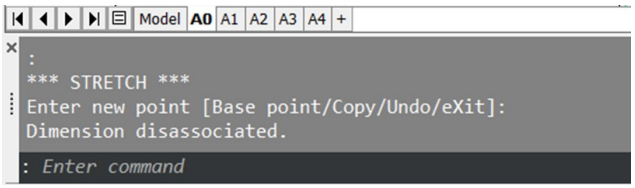
***Прим.:*** You can select any point of the edge of a BIM object. Turn off ESNAP if you don't only want to select snapping points.

7 When there are several tags possible on the location where you place the tag, hit CTRL to scroll through the different options while the Hotkey Assistant (HKA) appears at the bottom of the screen.

8 Move your cursor in order to adjust the position of the tag and left-click to place the tag.

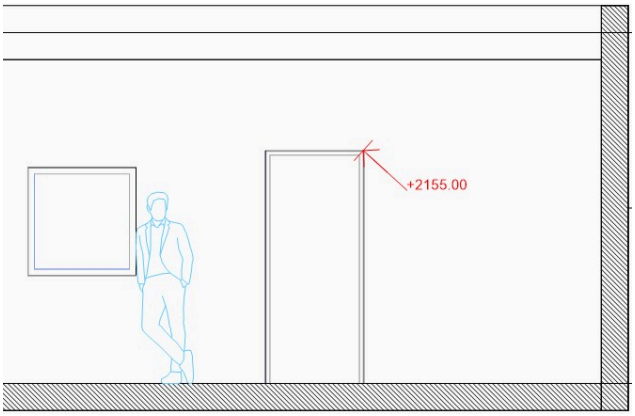



***Прим.:*** The spot elevation level is a non-dynamic value. When you move the tag, the value does not always change along. The moment you move a tag that will not be updated, you can see the tag value becomes dissociated in the Command line.

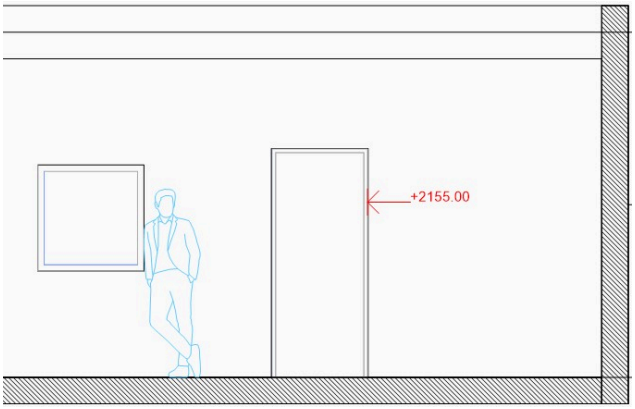



9 Update the section to make sure the right values are displayed. Select the section, open the Quad and click on **Update Section** in the **Model** tab.

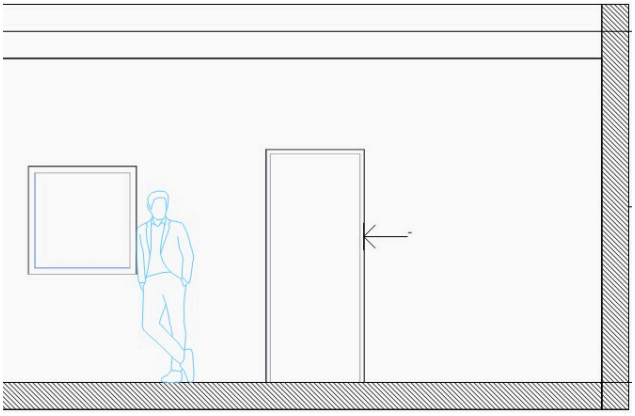
***Прим.:***If the tag has been displaced, it may behave irrationally or haphazardly by either turning red or omitting its value, as illustrated below:




***Tag turned red after displacing the arrowhead of the leader and updating the section.***




***Tag turned red after displacing the entire tag and updating the section.***



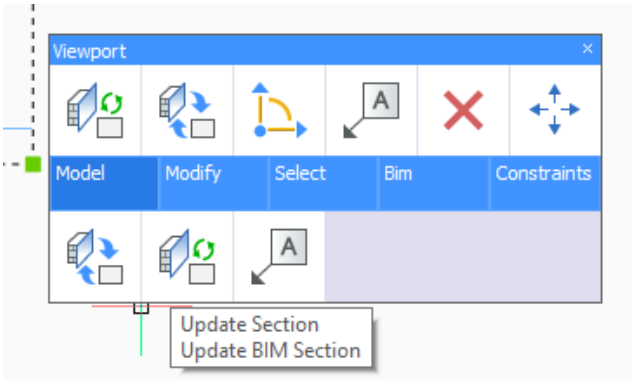
*Tag value disappeared after displacing the tag and updating the section.*

### 20.7.16.4 Procedure: how to place a spot elevation tag on an elevation sheet

1 Open an elevation sheet of the model. Right-click on the elevation sheet in the **Project Browser** and select **Display View**.

2 Type BIMTAG in the Command line,

3 **or** click on **Tag** in the **Model** menu of the Quad.




4 **You will be prompted**: [Manual/selection options (?)]

Type **M** to choose **Manual** and press Enter.

5 **Later, you will be prompted**: [Auto/Current mleader style/change Tag type]

Type **T** to change the **Tag** type and press Enter.

6 **Finally, you will be prompted**: [Tag/cOmposition/Load-bearing directions/Spot-elevation]

Type **S** to choose **Spot-elevation** and press Enter.

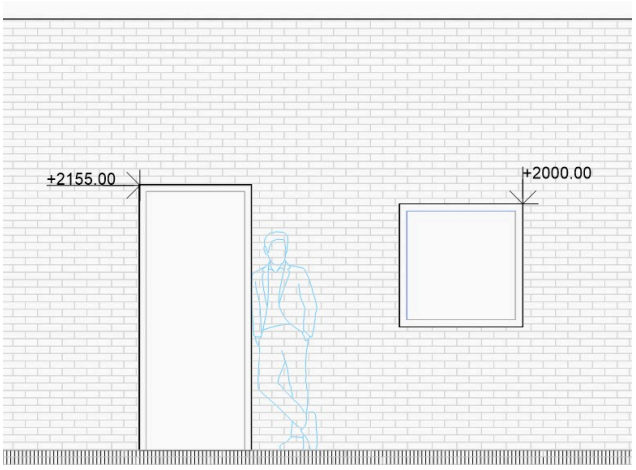
7 Click on a point on the edge of a BIM object to display the elevation height. The displayed value is the absolute elevation level measured relative to Floor 0 of the model.

***Прим.:*** You can select any point of the edge of a BIM object. Turn off ESNAP if you don't only want to select snapping points.

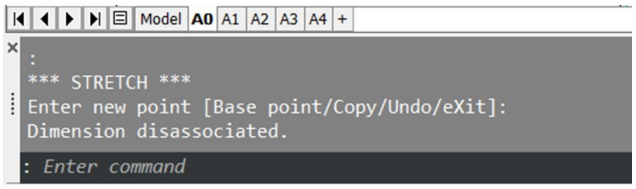
8 When there are several tags possible on the location where you place the tag, hit CTRL to scroll

through the different options while the HotKey Assistant (HKA) appears at the bottom of the screen.

9 Move your cursor in order to adjust the position of the tag and left-click to place the tag.

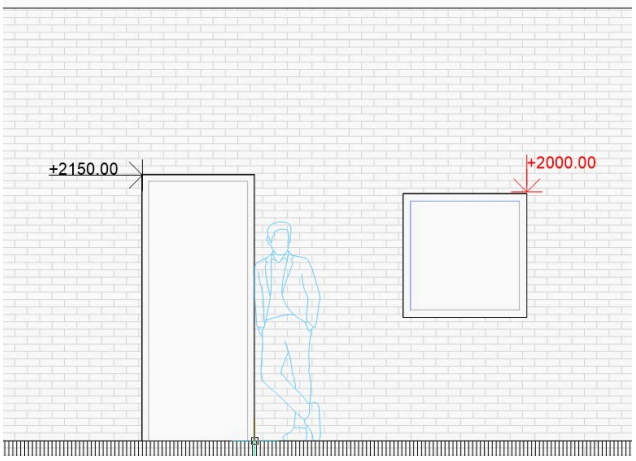



***Прим.:*** The spot elevation level is a non-dynamic value. When you move the tag, the value does not always change along. The moment you move a tag that will not be updated, you can see the tag value is dissociated in the Command line.

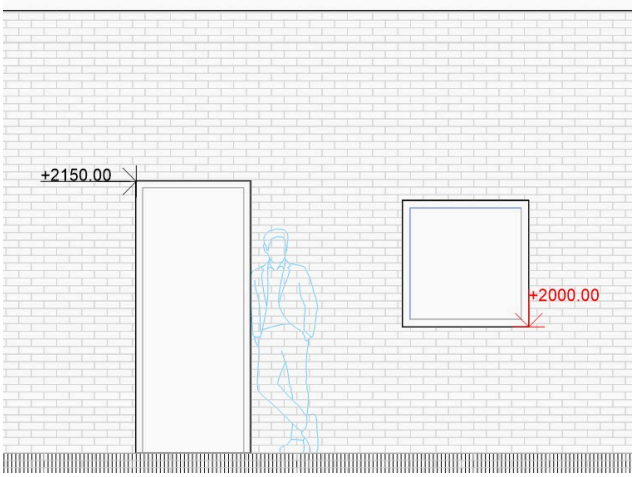



10 Update the section to make sure the right values are displayed. Select the section, open the Quad and click on **Update Section** in the **Model** tab.

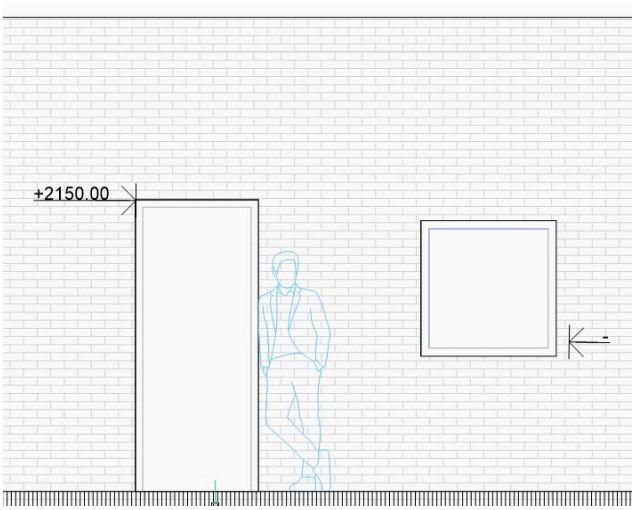
The tag should not behave irrationally or haphazardly by either turning red or omitting its value when it's displaced.




*Tag turned red after displacing the arrowhead of the leader and updating the section.*




*Tag turned red after displacing the entire tag and updating the section.*




*Tag value disappeared after displacing the tag and updating the section.*

### 20.7.16.5 Procedure: placing spot elevation level tags on floor plans or reflected ceiling plans

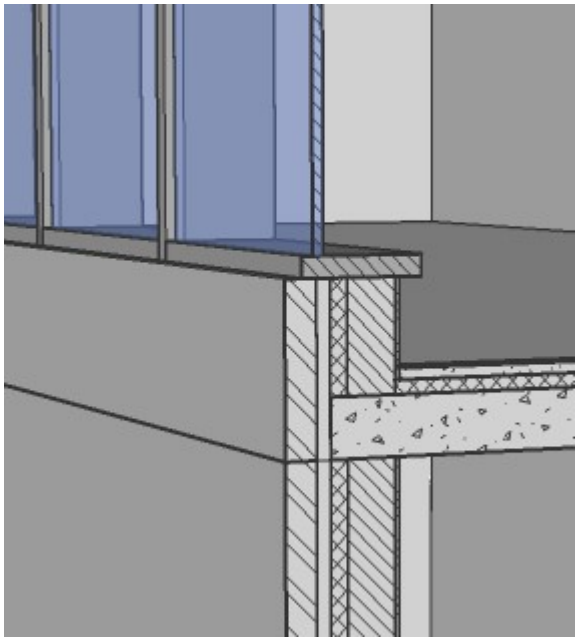
1 Open a floor plan or a reflected ceiling plan from the **Project Browser**.

2 Go through step 2-7 of 'How to place spot elevation level tag in a section sheet'.

3 In a floor or ceiling plan, there are 2 different tags possible.

- **With MLeader**

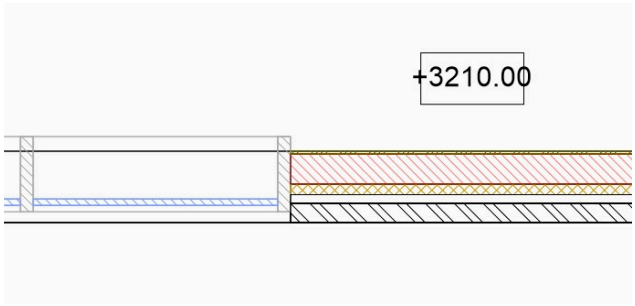
  This annotation tag type, which illustrates with a leader line, is also normally used in the section or elevation sheets. You can use this tag on entities that do not reach or aligned to the floor/ceiling level, such as a window sill in the picture below.

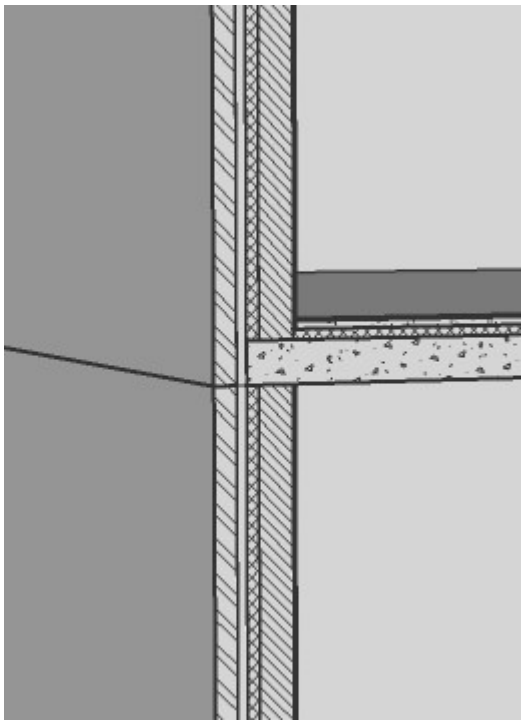


*Spot elevation level tag on window sill level which is not on floor level.*

- **Without MLeader**

  This tag has a different layout to the tags found in the elevation or section sheets. This tag appears when you place a spot elevation tag on entities that reaches the floor/ceiling plane, thereby displaying the elevation level of the floor plane.



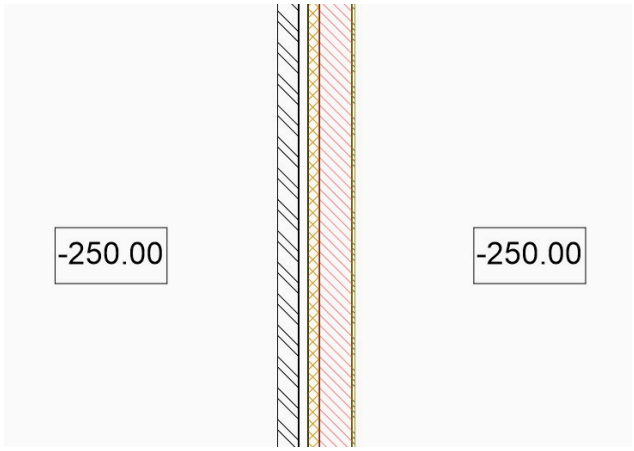


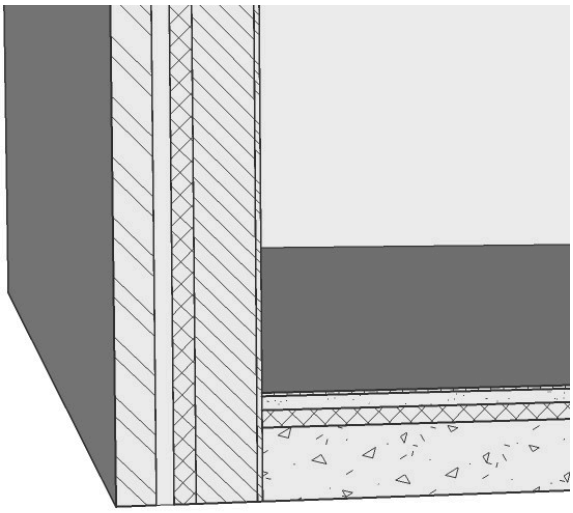
*Spot elevation level tag on wall that reaches till the floor slab.*

4 When placing a tag on a wall in plan view, you may encounter different results for different plies of the wall. This is due to how elevation levels are measured In BricsCAD. When placing a spot elevation tag on an entity in plan view, it refers to the elevation value of the lower level of the entity's ply, if applicable. Similarly, when you place a tag on a ceiling plan, the upper level of the ply is measured. Since plies of a wall may differ in their lengths based on how they are designed at detail junctions, the spot elevation tag value may differ according to the ply you have selected.

- **Construction detail with unedited plies**

  The interior and exterior plies have the same length. All the spot elevation tags you place on this wall in plan view will therefore display the same value.

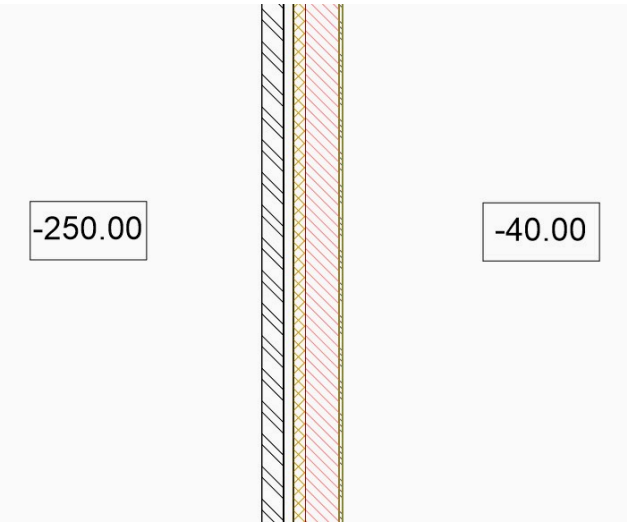


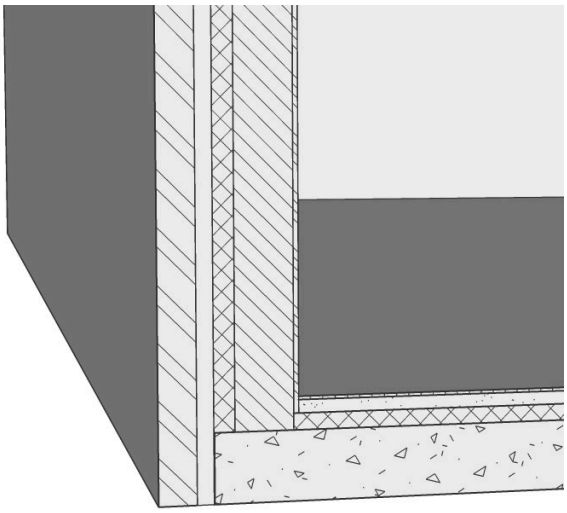


***Прим.:*** The value is -250 instead of 0 because Floor 0 is defined as the level of the upper side of the ground floor in this model.

- **Edited construction detail**

  If you have adjusted how the wall would connect to a floor slab based how it would be constructed, the plies of the wall will therefore have different lengths. Depending on which ply you select to place the spot elevation level tag, a different value will be displayed.

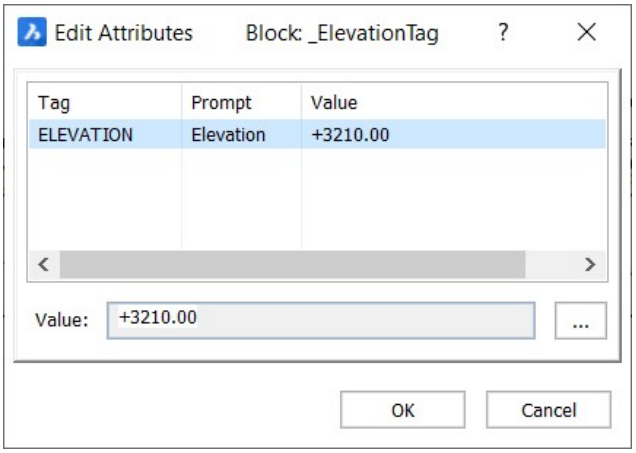




*Elevation level of interior ply is higher because ply does not reach till the base level of the floor.*

### 20.7.16.6 Procedure: adjusting the layout of the tag

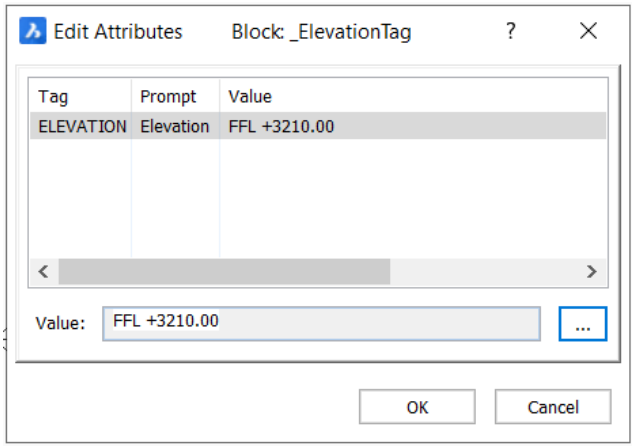
1 Double-click on the value of the spot elevation tag to adjust the layout. A dialog box will pop-up where you can edit the attributes.

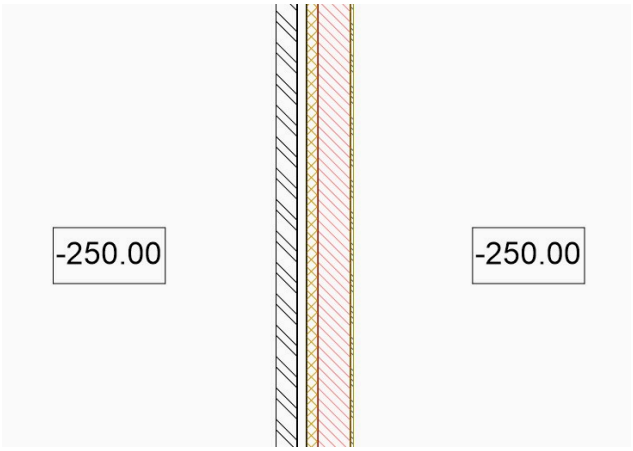



2 Click on the three dots to open the **Text Formatting** dialog box.




3 Here you can edit the attributes of the displayed value of the spot elevation level tag. You can put prefixes and/or suffixes to the values, e.g. FFL, SSL, to better illustrate the levels you are referring to.



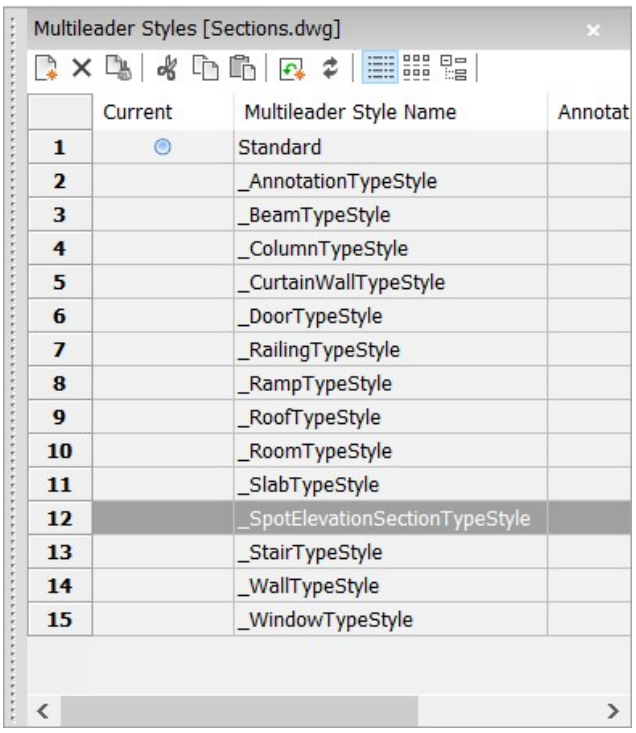

4 If the prefixes and/or suffixes are commonly used, you can create or edit the tag blocks so you do not have to repeat Step 3. For more info, visit the command article BEDIT.

5 For a detailed explanation about attributes, visit the command article MText.

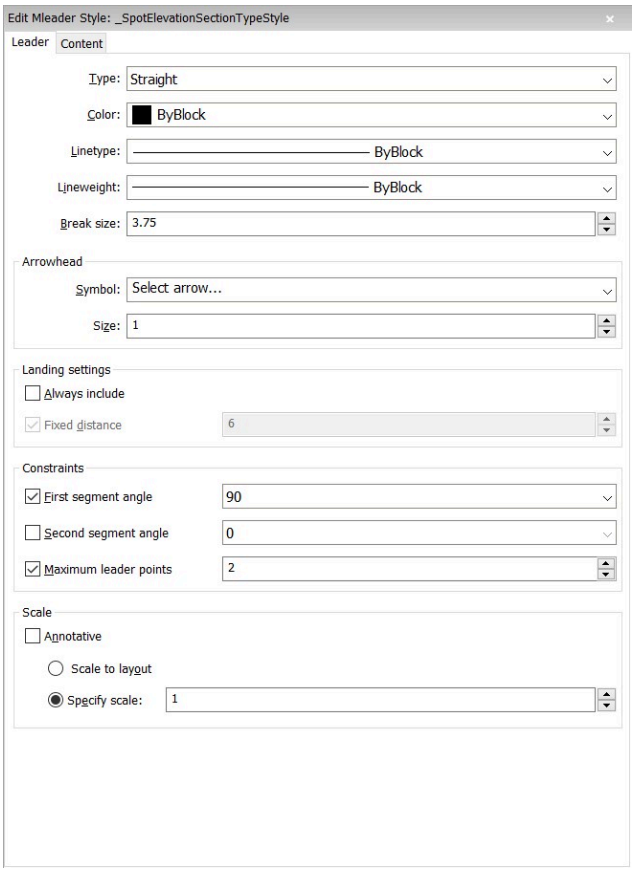
### 20.7.16.7 Procedure: adjusting the layout of the leader

1 Type MLEADERSTYLE in the Command line and press Enter. The **Drawing Explorer** dialog box will pop-up where you can adjust the layout of the leader.

2 Click on **SpotElevationSectionTypeStyle** in the **Multileader Styles** dialog box.

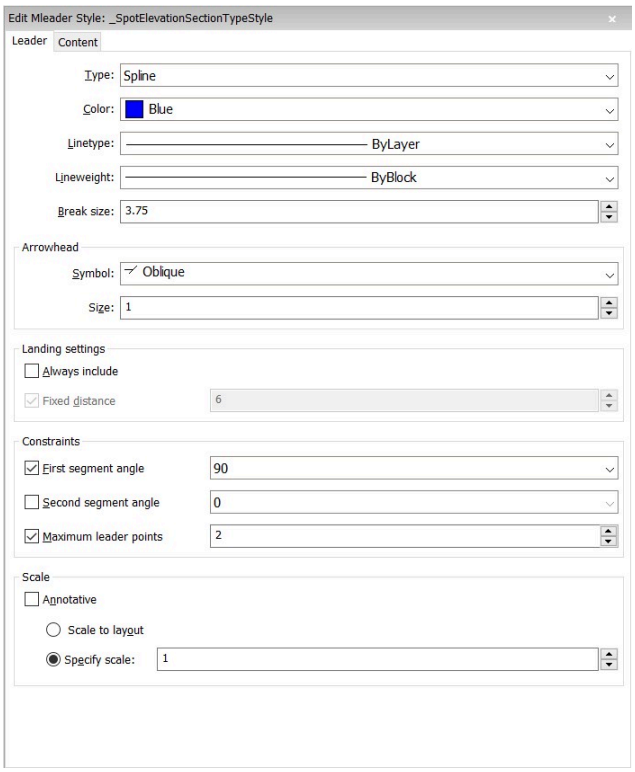



3 You can edit the settings of the leader in the **Edit MLeader Style** dialog box. For more info about the MLeader style settings, visit the article MLeaderStyle.

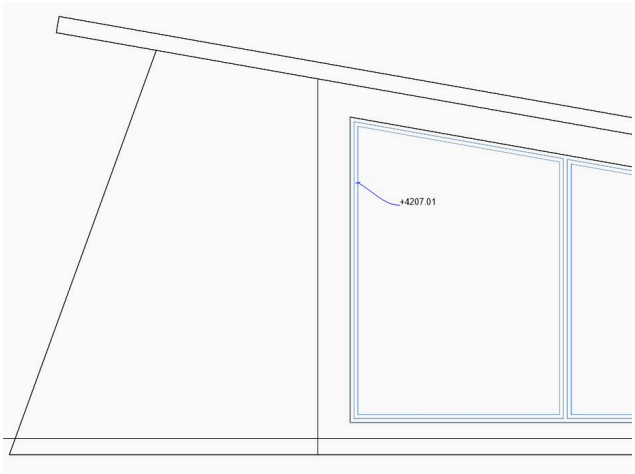



4 After adjusting the settings, click on regen, close the **Drawing Explorer** and update the section to load the new settings.

***Прим.:*** If the new settings don't load, type BIMTAG in the Command line, select the viewport, click **R** to select **Regenerate All** and press Enter to load the new settings.




***New Settings***




***Resulting BimTag***

## 20.7.17 Templates for generated drawings

### 20.7.17.1 About

BricsCAD BIM has a series of templates to customize your generated drawings. These templates are stored in your **Support** folder in the following path:

C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\Support\Bim\Sections.

The **Appdata** folder is a hidden folder by default, you may have to unhide folders first. This path can also be accessed by the SUPPORTFOLDER command.

To understand how it works, we must know how BricsCAD creates layers, hatches, tags, and call-out blocks. Then, for each template, we will discuss how to customize them.

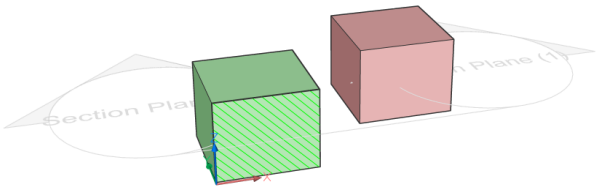
### 20.7.17.2 Procedure: understanding the created layers

1 Open a millimeter template **BIM-mm.dwt** and save the drawing.

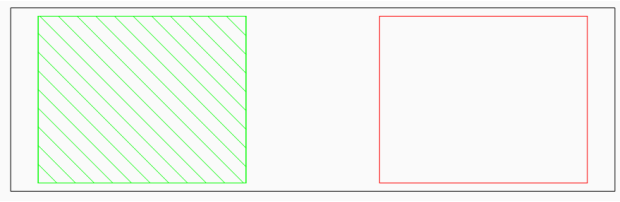
2 Draw two boxes of 500x500x400, one being a bit behind the other.

3 Put the first box on the layer **Layer1** (with layer color green) and the second box on the layer **Layer2** (with layer color red).

4 Create a BIM section, aligned with the front face of the first, green box and cutting through it in the middle, but not cutting through the second, red box.



5 Generate and open the section drawing.



6 Take a look at the **Layers** panel: three groups of layers have been created: **Background**, **Boundary**, and **Fill**.

| | Current | Name | D.. | On/Off | Freeze | Locked | Color |
|---|---|---|---|---|---|---|---|
| 1 | | 0 | | | | | White |
| 2 | | Background_Layer1 | | | | | Green |
| 3 | | Background_Layer2 | | | | | Red |
| 4 | | BIM_SECTIONS_PREVIOUS | | | | | Magenta |
| 5 | | BIM_VIEWPORTS | | | | | White |
| 6 | | Boundary_Layer1 | | | | | Green |
| 7 | | Defpoints | | | | | White |
| 8 | | Fill_Layer1 | | | | | Green |
| 9 | | Title Block Frame | | | | | 175 |
| 10 | | Title Block Labels | | | | | 157 |
| 11 | | Title Block Text | | | | | 251 |

7 The generated layers for the green solid are:

- **Boundary_Layer1**: the boundary of the section through the box.
- **Fill_Layer1**: the hatching of the section.
- **Background_Layer1**: shows subentities (e.g., vertices of solids) that are not sectioned.

8 The layers all inherit the color of the **Layer1** layer.

9 Since the red solid is not sectioned, only the **Background_Layer2** layer is created. It inherits the color of the **Layer2** layer.

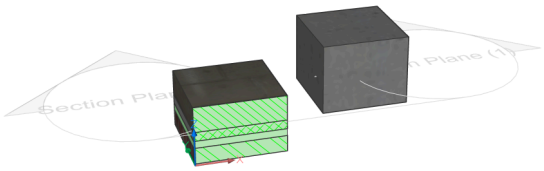
10 Drag the **Cavity Wall, Brick** composition from the **Compositions** panel onto the green box.

***Прим.:*** Set the LEVELOFDETAIL system variable to 1 or 2, to display the ply materials.

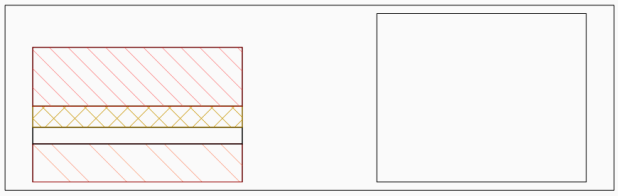
11 Double-click the **Cavity Wall, Brick** composition in the **Compositions** panel.

This composition consists of 4 plies, each with its own material. A physical material is associated with a hatch pattern, used for floor plans and sections. Another pattern is specified for elevations.

12 Assign the **Concrete, Reinforced** composition to the red box.



13 Update and open the section.



14 Go to your layers and execute the PURGE command to remove the unused layers.

| | Current | Name | D.. | On/Off | Freeze | Locked | Color |
|---|---|---|---|---|---|---|---|
| 1 | | 0 | | | | | White |
| 2 | | Background_Air | | | | | White |
| 3 | | Background_Concrete_ Reinforced | | | | | White |
| 4 | | Background_Facing Bricks_ Hand-form | | | | | White |
| 5 | | Background_Insulation_ Polyurethane | | | | | White |
| 6 | | Background_Layer1 | | | | | Green |
| 7 | | Background_Layer2 | | | | | Red |
| 8 | | Background_Supporting Wall_ Brick | | | | | White |
| 9 | | BIM_SECTIONS_PREVIOUS | | | | | Magenta |
| 10 | | BIM_VIEWPORTS | | | | | White |
| 11 | | Boundary_Air | | | | | White |
| 12 | | Boundary_Cavity Wall_ Brick | | | | | White |
| 13 | | Boundary_Facing Bricks_ Hand-formed | | | | | 14 |
| 14 | | Boundary_Insulation_ Polyurethane Fo | | | | | 42 |
| 15 | | Boundary_Layer1 | | | | | Green |
| 16 | | Boundary_Supporting Wall_ Brick | | | | | 14 |
| 17 | | Fill_Facing Bricks_ Hand-formed | | | | | 21 |
| 18 | | Fill_Insulation_ Polyurethane Foam | | | | | 42 |
| 19 | | Fill_Layer1 | | | | | Green |
| 20 | | Fill_Supporting Wall_ Brick | | | | | 11 |
| 21 | | Title Block Frame | | | | | 175 |
| 22 | | Title Block Labels | | | | | 157 |
| 23 | | Title Block Text | | | | | 251 |

You see the three categories (**Background**, **Boundary**, and **Fill**) again. Now the second part of the layer name is the name of the physical material used in the solid, instead of the layer name of the box:

- **Background_Facing Bricks_Hand-formed**, **Background_Insulation_Polyurethane Foam**, and **Background_Supporting Wall_Brick**.
- **Boundary_Facing Bricks_Hand-formed**, **Boundary_Insulation_Polyurethane Foam**, and **Boundary_Supporting Wall_Brick**.
- **Fill_Facing Bricks_Hand-formed**, **Fill_Insulation_Polyurethane Foam**, and **Fill_Supporting Wall_Brick**.

15 Set the **Generate elevation fill** property of the section **ON** to add an elevation hatch.

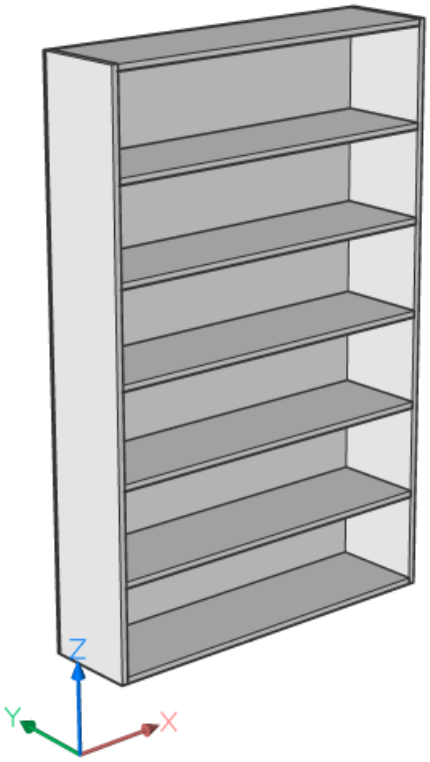
16 Update the section.

17 The outline of the second box is now hatched and the layer **Elevation_Concrete_Reinforced** is added.

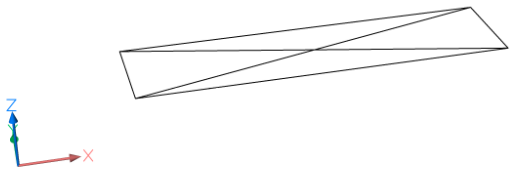
### 20.7.17.3 Procedure: understanding the BRX_2D layers

1 Open a millimeter template **BIM-mm.dwt** and save the drawing.

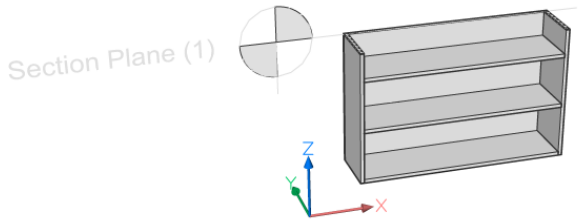
2 Drag the component **Furniture Bookshelf** into the drawing. You can find the component in the **Furnishing Elements** category in the **Library** panel (LIBRARYPANELOPEN command).



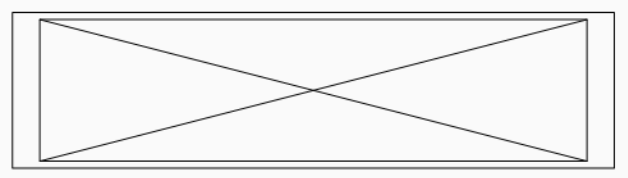

3 If you now show and isolate the **BRX_2D_Symbols** layer, you will see a symbol drawn at the insertion point of the bookshelf.




4 Now show all layers again (LAYUNISO command) and make a section through the bookshelf.




5 Generate and open the Plans drawing.



6 You will see (if the **Boundary_BRX_2D_Symbols** layer is on) that only the symbol is drawn onto the plan, and not the real section. It happens because this component had a 2D symbol on the BRX_2D layer attached to it.

7 Now return to your 3D model and Open a copy of the bookshelf.

8 Hide the bookshelf itself and make sure the **BRX_2D_Symbols** layer is on.

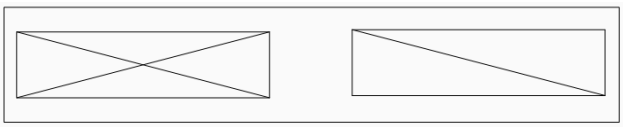
9 Delete one of the lines (of the cross).

10 **(Optional)** Draw extra 2D lines on the **BRX_2D_Symbols** layer.

11 Save the drawing.

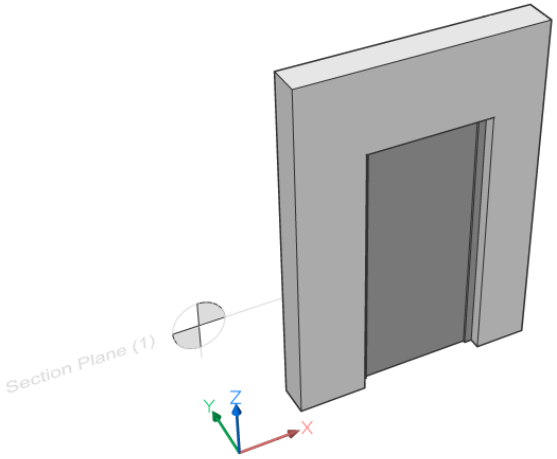
12 Go back to the original 3D model and type in BMINSERT in the Command line and go to the file path where you just saved the new bookshelf. Click **OK** and choose a point to insert the bookshelf.

13 Regenerate the section and open it. Widen the viewport if necessary, by dragging the existing viewport.



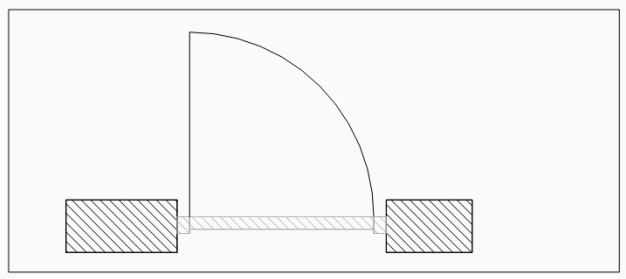
14 You will now see that the second bookshelf has the adapted symbol displayed.

15 Draw a wall in your 3D model and insert the door **Door Ext Plain** into this wall.




16 Repeat steps 3 to 5 but now use the **BRX_2D+_Swing** layer instead of the **BRX_2D_Symbols** layer.

17 The generated plan displays the symbol of the door swing as well as the door section itself. The **BRX_2D+** layer of a component makes sure you can show the real section together with a symbol.
   - Symbols on a layer with prefix **BRX_2D_** will be used to replace the geometry resulting from the actual section.
   - Symbols on a layer with prefix **BRX_2D+_** will be added to the section result. Only those 2D entities in a plane parallel to the section plane are.



### 20.7.17.4 Procedure: customizing the colors and line weights of hatch patterns and lines

1 Firstly, in the above procedure 'Understanding the created layers', you saw that for objects without a composition, the colors and line weights of the **Fill_**, **Background_**, and **Boundary_** layers are copied from the layer properties in the 3D model.

2 However, for objects with a composition, the default colors are set by a template found by typing in SUPPORTFOLDER command and browsing to *Bim/Sections/_SectionSettings.dwg*.

3 Open this drawing.

4 Go to the **Layers** panel, you will find a list of all the possible layers that can be created for every material present in the material library. They all have a color and line weight. Changing the color or line weight and saving the file will change the default color/line weight for this layer in your future generated drawings. You can fully customize how your drawing will look for every type of line (**Fill**,

**Background**, **Boundary**, or **Elevation**) per material.

| | | | | | | |
|---|---|---|---|---|---|---|
| 195 | | Fill_Ceiling_ Plasterboard | | | | 253 |
| 196 | | Fill_Concrete_ Beam and Pot Floor | | | | 85 |
| 197 | | Fill_Concrete_ Plain | | | | 85 |
| 198 | | Fill_Concrete_ Precast | | | | 85 |
| 199 | | Fill_Concrete_ Reinforced | | | | 85 |
| 200 | | Fill_Copper | | | | 151 |
| 201 | | Fill_Facing Bricks_ Concrete Blocks | | | | 21 |
| 202 | | Fill_Facing Bricks_ Hand-formed | | | | 21 |
| 203 | | Fill_Facing Bricks_ Sand-lime | | | | 21 |
| 204 | | Fill_Floor Joist_ Wood | | | | 33 |
| 205 | | Fill_Floor_ Boarded | | | | 9 |
| 206 | | Fill_Floor_ Ceramic Tiles | | | | 251 |
| 207 | | Fill_Floor_ Concrete Tiles | | | | 251 |
| 208 | | Fill_Floor_ Laminate | | | | 251 |
| 209 | | Fill_Floor_ Parquet | | | | 251 |
| 210 | | Fill_Floor_ Stone Tiles | | | | 43 |
| 211 | | Fill_Freestone | | | | 43 |
| 212 | | Fill_Glass | | | | 141 |
| 213 | | Fill_Glass Brick | | | | 141 |
| 214 | | Fill_Gold | | | | 151 |
| 215 | | Fill_Granite | | | | 43 |
| 216 | | Fill_Gravel | | | | 251 |
| 217 | | Fill_Gypsum Board | | | | 251 |
| 218 | | Fill_Insulation_ Extruded Polystyrene | | | | 42 |
| 219 | | Fill_Insulation_ Glass Wool | | | | 42 |
| 220 | | Fill_Insulation_ Polyurethane Foam | | | | 42 |
| 221 | | Fill_Insulation_ Rock Wool | | | | 42 |
| 222 | | Fill_Iron | | | | 251 |
| 223 | | Fill_Lead | | | | 151 |
| 224 | | Fill_Marble | | | | 43 |

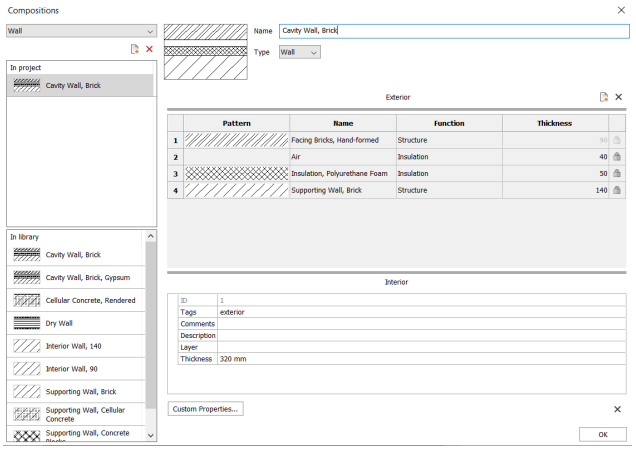
## 20.7.17.5 Procedure: customizing the hatch patterns

1 Open the drawing made in the above procedure 'Understanding the created layers'.

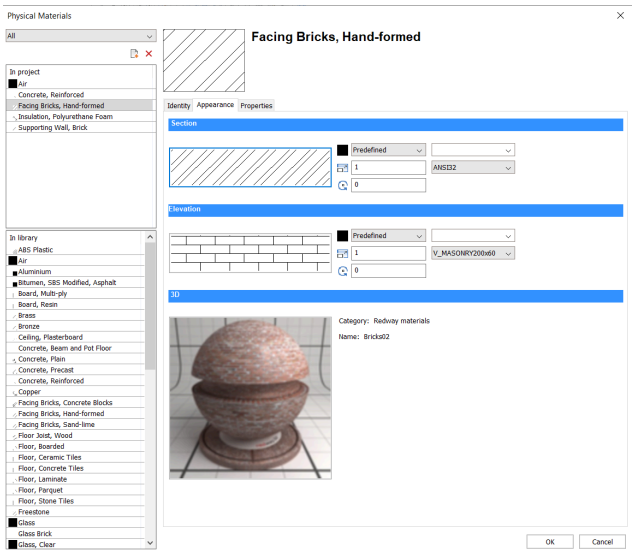
2 Open the **Compositions** panel.

3 Double-click the composition **Cavity Wall, Brick**.

4 In the procedure 'Understanding the created layers said that you can change the hatch patterns of the materials here. How you need to do this will follow in the next steps.



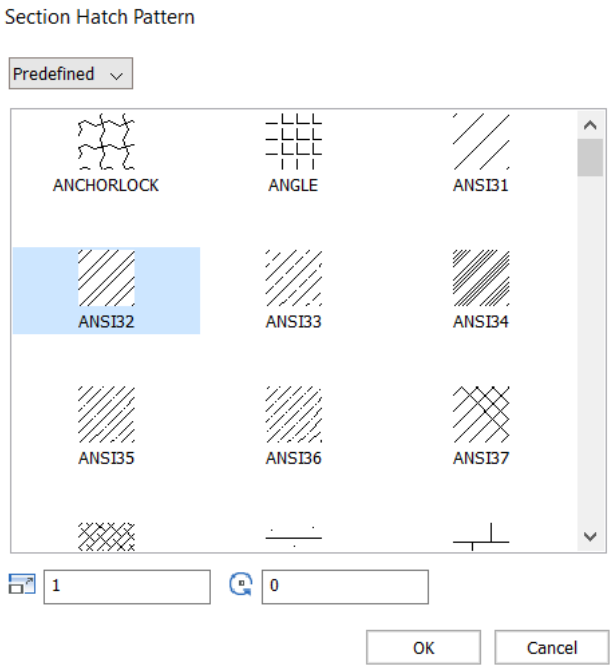

5 Double-click on the row of **Facing Bricks, Hand-formed**. The **Physical Materials** dialog will pop-up on the material **Facing Bricks, Hand-formed**. You could have entered this dialog box by typing in the command BLMATERIALS, after which you could have selected the **Facing Bricks, Hand-formed** material.




6 Go to the tab **Appearance**.

7 Under the title **Section**, double-click on the hatch pattern.

8 You can now choose your new standard section hatch representation for this particular material. In every composition with this material, the same hatch representation will be used.



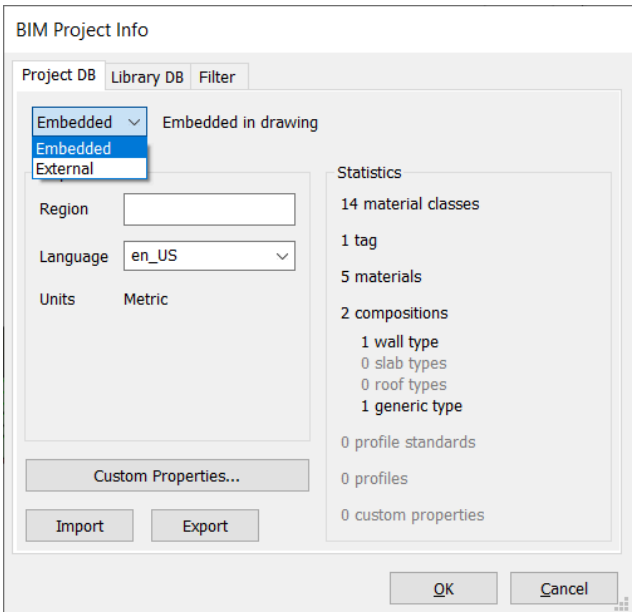

9 If you want to change the appearance of an elevation hatch, you can do so by double-clicking the hatch pattern under the **Elevation** title. The outer most visible material of a composition will be used to determine the elevation hatch.

10 Note that the representation of these hatches will not be copied into new drawings as the **Physical Materials** dialog is not a template, but rather a way to define the representation in a specific drawing.

If you would like to store this information, it is recommended to create a library of compositions. The best way to do so is to make a drawing where you attach the compositions you would like to store to little cubes. Make sure each composition has the hatch appearance you want by going through the process described above.

11 Once this is finished, type in BIMPROJECTINFO in the Command line. You will enter the **BIM Project Info** dialog box.

12 In the **Project DB** tab, switch the value for the database from **Embedded** to **External**. You will be asked to save the database on your computer and give it a name, for example, 'CompositionsTemplate.bsyslib'.




13 If you now want to use this database (Project DB) in a new drawing, switch the value for the database from **Embedded** to **External**. This time select the .bsyslib file you created.

### 20.7.17.6 Procedure: customizing tags per entity type for all section drawings

For this procedure, please visit the following procedures from the BIM Tags page:

- Create the source data for a BIM entity type.
- Customize tag blocks.
- Edit the source data to customize BIM Tags for all section drawings.

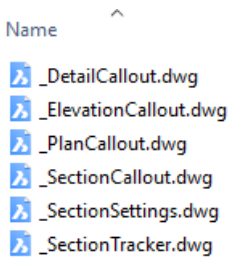
To change the tags in the current drawing, visit the procedure 'Customize tag blocks for the current drawing only'.

### 20.7.17.7 Procedure: customizing call-out blocks

1 Source files of the various call-out blocks and the section tracker exist in the **..\Support\Bim\Sections** subfolder of the **Roamable Root** folder.

By default this is: **C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\Vxx\en_US\Support**

**\Bim\Sections.** You can easily access this file by typing in SUPPORTFOLDER command and browsing to **Bim** > **Sections**.




2 You can edit these files to customize the layout of the call-outs and the section tracker blocks.

3 When a drawing is generated, the needed call-out blocks are created from the call-out blocks source file(s). When a BIM section is updated, the existing call-out blocks in the target drawing are used (type BEDIT to choose one to edit).

### 20.7.17.8 Procedure: customizing the sheet set template

1 The sheet set templates are stored in the **C:\Users\<user_name>\AppData\Local\Bricsys\BricsCAD \Vxx\en_US\Templates** folder.

2 If you want to edit an existing template, you have two options. Either you edit it directly, or you make a copy.
   - To edit it directly, you can go to **Open** under the BricsCAD icon, choose **.dwt** in the **Files of type** drop-down list and choose the file you want to edit. You can then make the modifications and save the file.
   - To make a copy, double-click on the .dwt file you want to copy. A new .dwg with this template will open. Make your modifications and **save your file as .dwt**. Save it in the templates folder to easily use the copied template in a new drawing.

3 If you want to start from scratch, using a wizard, follow the procedure 'To create a drawing using a wizard'. After the creation and modification of the dwg, save it as a .dwt file in your sheet set templates folder.

4 If you have saved the template .dwt to the sheet set templates folder, you will see it appear when you use the **New Wizard** to open a new drawing. You will be able to select it as the template for the creation of generated drawings in the sheet set set-up of the **Project Browser**.

5 What can you store into a template?

6 You can make a title block for your template, with sheet set properties, by following the procedure 'Using sheet set properties in a title block'. Do this for every layout in your template. You can also store the following things by using them in your .dwt file:
   - Layers
   - Colors
   - Line types
   - Hatch patterns

- Blocks
- Text styles
- Dimension styles
- Page set-ups

# 21. BricsCAD Mechanical

## 21.1 Что нового

### 21.1.1 Обзор

BricsCAD Mechanical V21 предлагает весь набор новых возможностей, доступных в BricsCAD V21, и включает ряд изменений, направленных на усовершенствование процесса машиностроительного проектирования.

В этом разделе представлен обзор всех новых возможностей **BricsCAD V21 Mechanical**:

Диспетчер спецификаций 1541

Библиотеки стандартных изделий 1544

Параметры и зависимости 1545

Стандартные детали 1545

Листовой металл 1545

Информация о новых возможностях **BricsCAD V21** доступна в разделе Что нового в BricsCAD V21. Он содержит следующие статьи:

Overview 31

User interface 32

Productivity 35

Drafting 49

Modeling 51

Civil tools 60

Performance and compatibility 65

Информация о новых возможностях **BricsCAD V21 BIM** доступна в разделе Что нового в BricsCAD BIM V21. Он содержит следующие статьи:

Overview 969

Modeling techniques 970

Building data 977

Project collaboration 986

Design documentation 987

Point clouds 996

#### 21.1.1.1 Последние изменения

Нажмите здесь для просмотра перечня последних изменений.

### 21.1.1.2 Диагностика и сбор данных об использовании

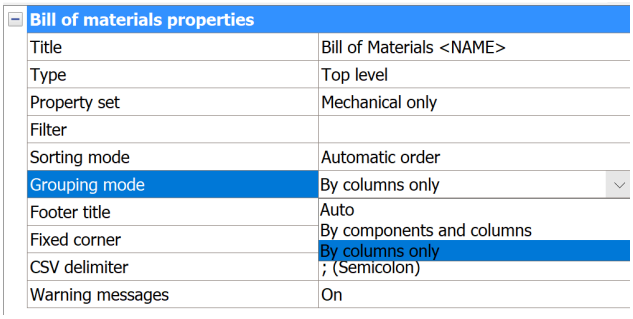
Многие новые инструменты и возможности были добавлены по запросам пользователей BricsCAD. Bricsys постоянно расширяет методы обратной связи с пользователями и предлагает новые инструменты для сбора данных об использовании своих продуктов. При втором запуске BricsCAD отображается диалоговое окно с приглашением принять участие в Программе диагностики и сбора данных об использовании. Процесс сбора данных происходит анонимно и позволяет вам делиться диагностическими данными и данными об использовании BricsCAD с Bricsys, что способствует дальнейшему улучшению нашего продукта. Вы всегда можете отозвать свое согласие и удалить предоставленную вами информацию из наших баз данных.

## 21.1.2 Диспетчер спецификаций

Процесс создания и редактирования спецификаций стал еще проще, чем ранее.

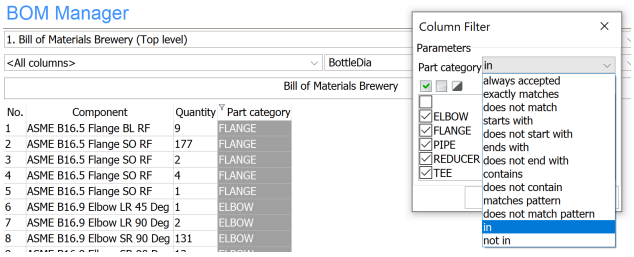
### 21.1.2.1 Группировка

Добавлено свойство Режим группировки, которое позволяет быстро сгруппировать содержимое таблицы спецификации в сжатые строки по общим свойствам существующих столбцов.

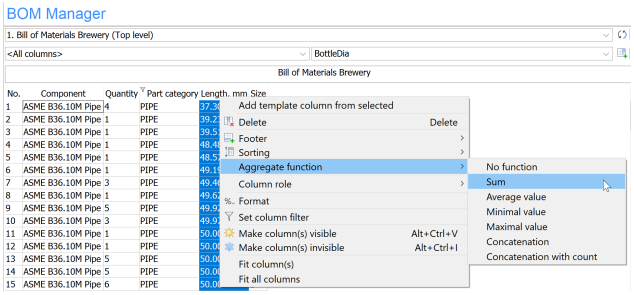



### 21.1.2.2 Столбцы

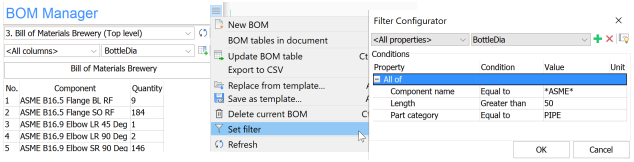
Добавлена возможность установки одного из нескольких предопределенных фильтров столбцов для любого столбца в спецификации.



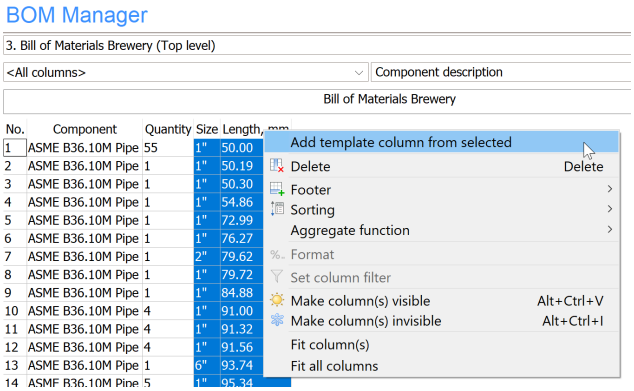

Агрегированные столбцы теперь поддерживаются в таблицах спецификаций верхнего и нижнего уровней, созданных с использованием команды BMBOM. Они позволяют пользователям вычислять сумму, минимальное, максимальное или среднее значение данного свойства для текущей группы значений.



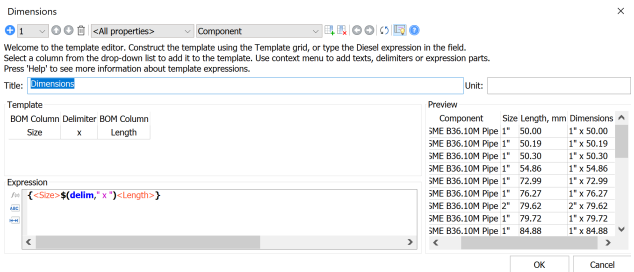

Опция в основном меню Диспетчера спецификаций открывает новое диалоговое окно Конфигуратор фильтра. В этом окне вы можете просмотреть содержимое таблицы спецификации. Для свойств строки в фильтре спецификации поддерживаются выражения со специальными символами.



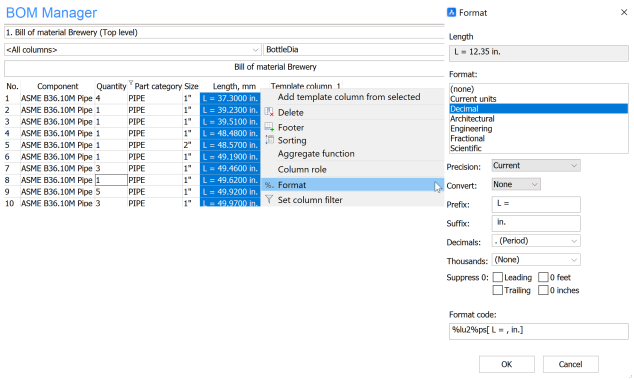

Столбцы шаблона - это новый тип расчетных столбцов, который позволяет пользователям объединять значения нескольких свойств в один столбец спецификации в виде текста, используя выражения DIESEL.



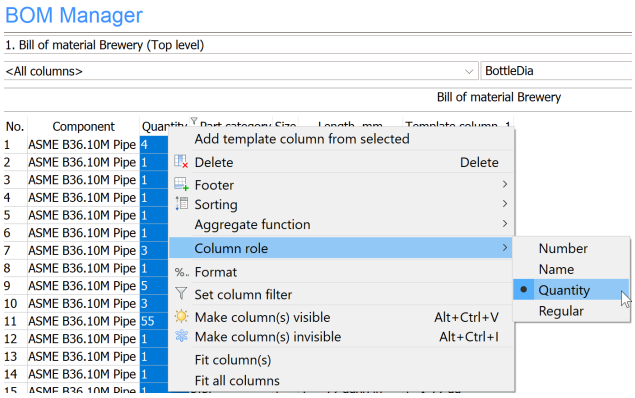

Вы можете настроить выражения столбцов формул или шаблонов, используя диалоговое окно редактора выражений.



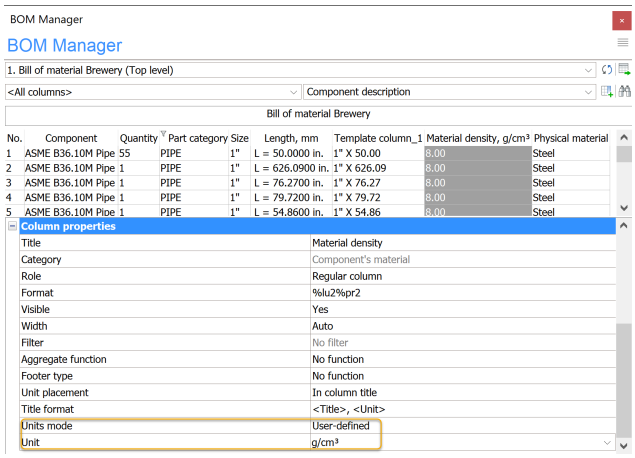

Диалоговое окно Формат позволяет формировать выражения для данных таблиц спецификаций с использованием префиксов, суффиксов и разделителей.



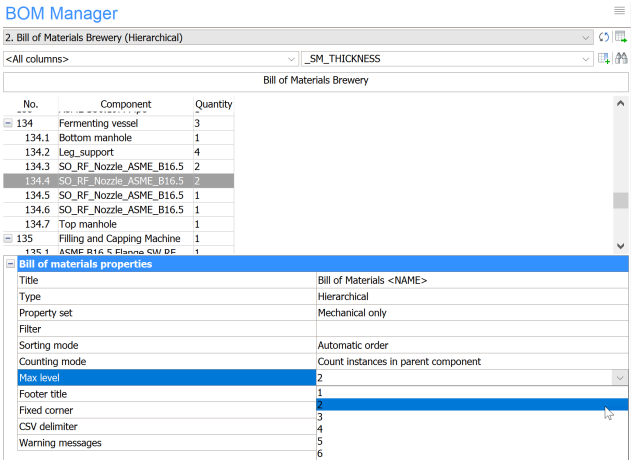

Новый параметр Назначение столбца позволяет пользователям указывать, какой столбец должен быть источником данных для видов чертежа.




Добавлена возможность выбора единицы измерения плотности материала в диалоговом окне Физические материалы и в таблицах спецификации. Поддерживаемые единицы: кг/м³, кг/дм³, г/см³, фунт/фут³ и фунт/дюйм³.

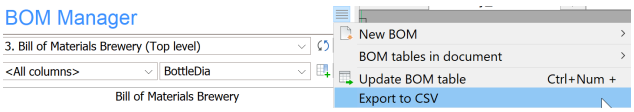



Добавлена возможность ограничения уровня отчета иерархической спецификации.




### 21.1.2.3 Экспорт

Реализована возможность экспорта таблицы спецификаций в .csv файл.

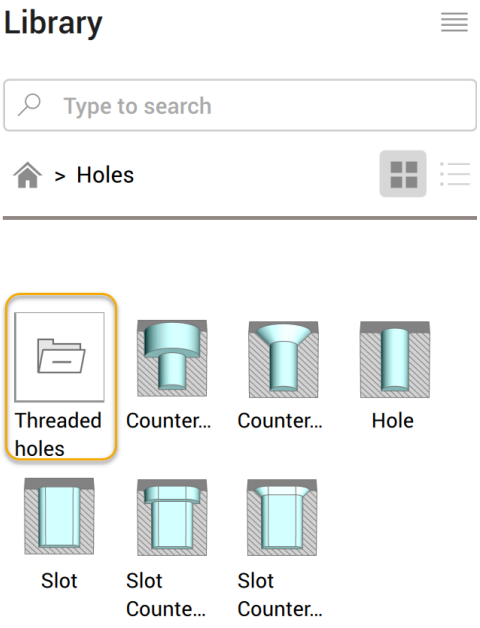



## 21.1.3 Библиотеки стандартных изделий

В BricsCAD V21 значительно расширена библиотека стандартных изделий и элементов.

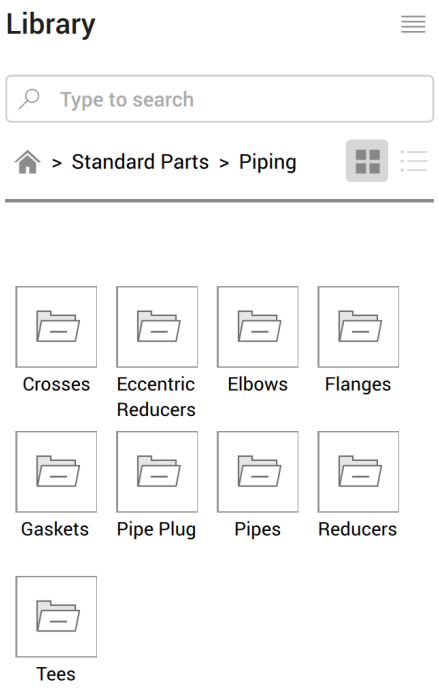
### 21.1.3.1 Отверстия

Добавлена библиотека отверстий с резьбой.



#### 21.1.3.2 Трубы

Добавлена библиотека элементов трубопроводов.




### 21.1.4 Параметры и зависимости

#### 21.1.4.1 Трубопроводы

Новая команда BMCONNECT автоматически устанавливает зависимости для выбранных стандартных деталей трубопроводов в соответствии с их взаимным расположением.

### 21.1.5 Стандартные детали

Теперь после изменения параметров стандартной детали зависимости восстанавливаются автоматически.

### 21.1.6 Листовой металл

BricsCAD Mechanical V21 предлагает усовершенствованный набор гибких и высокопроизводительных инструментов для разработки изделий из листового металла.

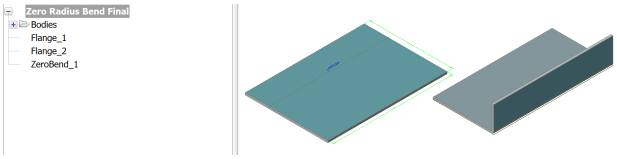
Например, для каждого твердого тела может быть назначена определенная таблица гибов, а в одном блоке могут существовать элементы различной толщины.

#### 21.1.6.1 Фаски

Новая функция Фаска теперь поддерживается командами SMCONVERT, SMREPAIR и SMUNFOLD.

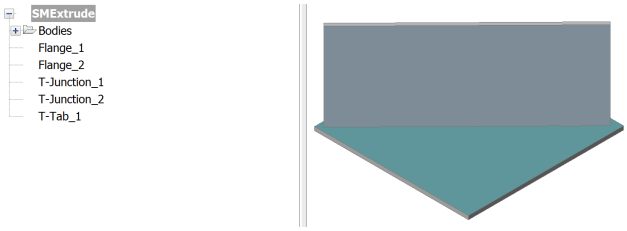
### 21.1.6.2 Гибы с нулевым радиусом

Новая функция ZeroBend позволяет создавать гибы элементов листового металла с нулевым внутренним радиусом.



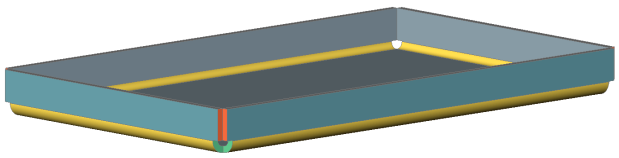

### 21.1.6.3 T-образный стык

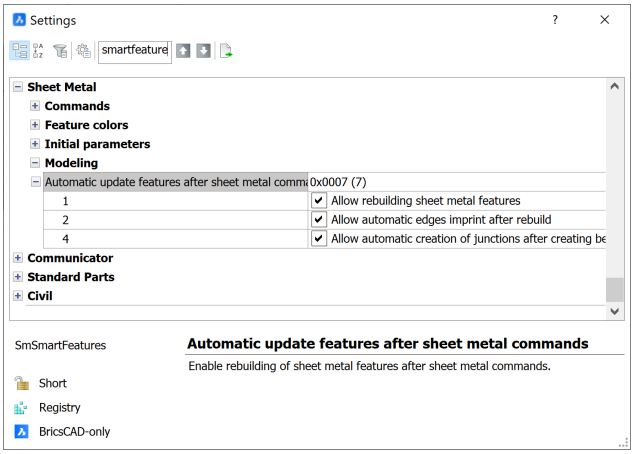
Команда SMEXTRUDE позволяет выдавить односегментную полилинию, лежащую на стенке, чтобы создать новую стенку, ортогональную первой и сформировать T-образный стык. Созданный T-образный стык может быть автоматически преобразован в T-образное соединение шип-паз.




### 21.1.6.4 Интеллектуальные возможности моделирования

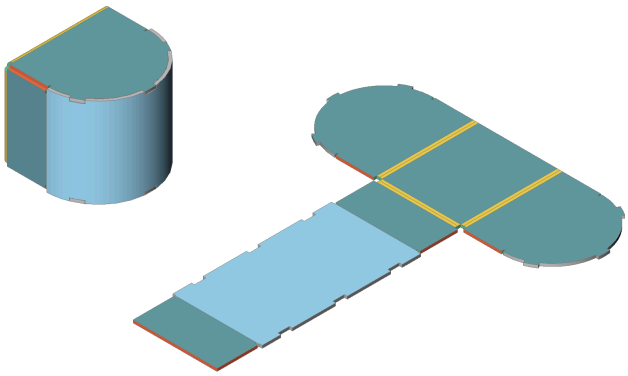
Обновленная команда SMSMARTFEATURES позволяет автоматически создавать стыки при использовании инструмента Гиб.



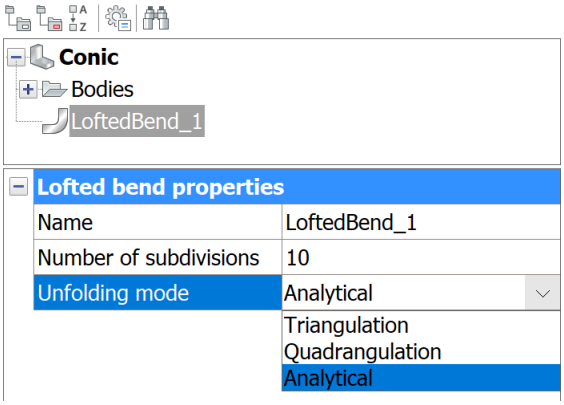


### 21.1.6.5 Соединение Шип-паз

Для команды SMTABCREATE добавлена возможность создания криволинейного шип-паза на изогнутой кромке.

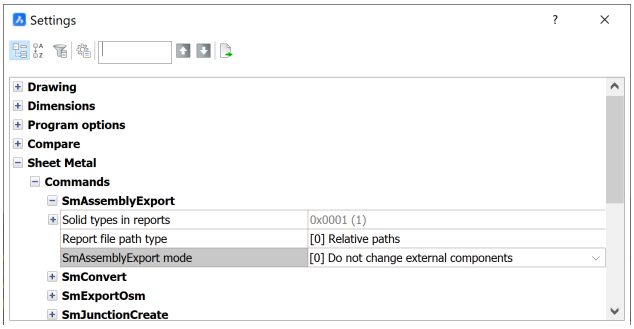


Команда SMUNFOLD обеспечивает улучшенную обработку конических лофт-гибов. Теперь можно получить их развертку аналитическим методом без дополнительного разделения на несколько гибов и стенок.




### 21.1.6.6 Экспорт сборок

Системная переменная SMASSEMBLYEXPORTMODE позволяет сохранять распознанные элементы листового металла и простого листового металла при использовании команды SMASSEMBLYEXPORT.



## 21.2 Начало работы

### 21.2.1 Рабочее пространство Mechanical

#### 21.2.1.1 Начало работы с машиностроительным проектом

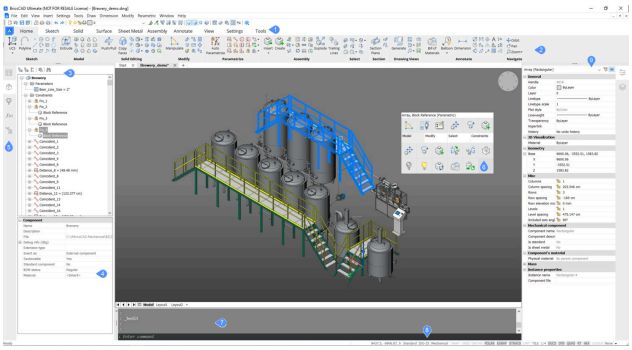
Перед началом работы с машиностроительным проектом:

1 Запустите BricsCAD.

Откроется диалоговое окно *Запуск BricsCAD*.

2 Нажмите кнопку **Пуск** в разделе **Mechanical**.

#### 21.2.1.2 Основное окно BricsCAD Mechanical



1 **Закладки ленты**: нажмите, чтобы перейти на соответствующую вкладку ленты.

2 **Лента**: каждая вкладка ленты содержит несколько панелей, которые логически группируют подобные инструменты.

3 **Обозреватель конструкции**: отображает древовидную структуру модели и позволяет выделить конкретный элемент модели с возможностью редактирования его свойств.

4 **Свойства выбранного элемента**: некоторые из свойств доступны только для чтения.

5 **Закрепляемые панели инструментов**: Обозреватель конструкции и Библиотека.

6 **Квадро-меню курсора**: позволяет выбрать необходимый инструмент при подведении курсора к объекту.

7 **Командная строка**: используется для ввода команд.

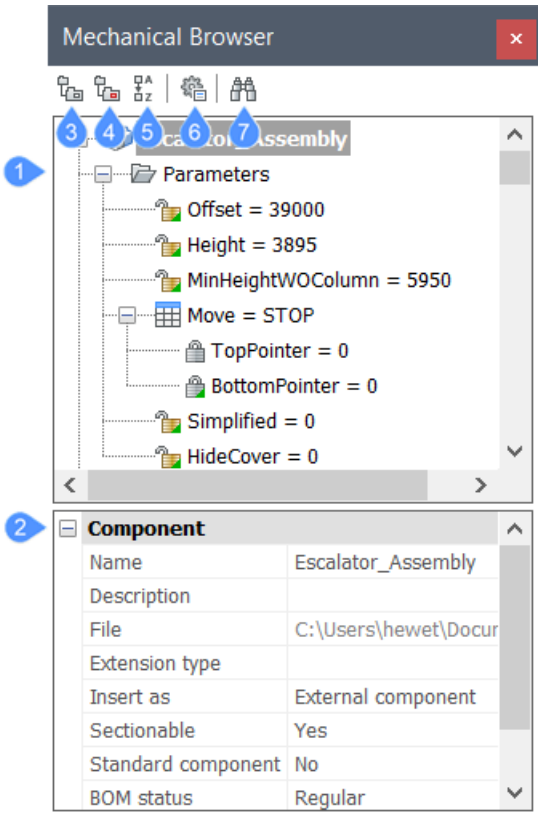
8 **Строка состояния**: отображает текущее состояние параметров черчения и позволяет изменять их значения.

9 **Панель свойств**: используется для просмотра и редактирования немеханических свойств выбранных объектов; таких как цвет или слой.

**Рисунок 4:** Навигация в BricsCAD Mechanical

### 21.2.2 Mechanical browser panel

The Mechanical browser panel offers a central location to view and modify parametric properties of mechanical parts and assemblies.




1 Parameter list

2 Properties

3 Group by entity

4 Group by type

5 Sort

6 Settings

7 Show search

### 21.2.2.1 Parameter list

Displays a list of all the parameters and constraints in the drawing. They are organized in expandable categories which may include user parameters,2Ddimensional constraints, 3D dimensional constraints, 3D geometric constraints,block parameters and arrayparameters.

A context menu displays when right-click the main component name:

**Update**
Updates the hierarchy of mechanical components for the current drawing in case referenced drawing files of sub-components have been modified.

**Visual style > All by Viewport**
Applies the current visual style to all components in the assembly.

**Insert standard hardware**
Inserts a standard hardware part as a mechanical component in the current drawing.

**Switch all to local**
Converts all external components in the model to local components.

**Switch all to external…**
Switches all internal components to external components.

**Select all**
Selects all components with the same definition.

**Select same**
Selects all components with the same name and the same parameter values.

**Highlight all**
Highlights all components with the same definition.

**Highlight same**
Highlight all components with the same name and the same parameter values.

**Create exploded view**
Creates a block with an exploded representation of the current.

**Dissolve**
The selected feature is removed from the part, but it will keep its geometry. However, design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

**Delete**
This is the analogue of the SMDELETE command. In this case the feature is removed from the browser and geometry is changed depending upon the type of the feature.

**Disable**
The feature will remain in the Mechanical Browser, but the part will behave like the feature was dissolved. It is beneficial in contrast with dissolve, that feature still updates on geometry changes and you can turn it on when you want, avoiding recreating it or recognizing by SMCONVERT.

Right-clickmenusoffer additional toolsfor components:

**Open**
Opens the referenced drawing.

**Open a copy**
Opens a copy of a component insert as a new drawing.

**Update**
Reloads all referenced components from external files and updates BOM tables.

**Replace…**
Replaces a component insert.

***Прим.:*** Replacing a local insert turns it into an external insert.

**Set material to component**
Opens the Physical Materials dialog box, which allows you to assign a physical material to the component.

**BOM status**
Controls the appearance of the component in BOM tables.

Right-clickmenusoffer additional toolsfor parameters:

**Geometry-driven**
If checked, it makes the parameter geometry-driven.

**Create design table**
Creates a design table to drive parametric block parameters.

**Animate**
Animates models by means of parameters.

**Link to parameter**
Links subcomponent parameter to the main level parameter.

#### 21.2.2.2 Properties

Specifies the properties of the selected item.

#### 21.2.2.3 Group by entity

Groups 3D constraints by entity. Expand an entity to view its associated constraints.

#### 21.2.2.4 Group by type

Groups 3D constraints by type. Expand a constraint to view the affected entities.

#### 21.2.2.5 Sort Alphabetically

Lists mechanical components and 3D constraints alphabetically. Otherwise, they are listed in the order they are added to the assembly.

#### 21.2.2.6 Settings

**Expressions of constraints**
Controls whether the numeric value or the assigned parameter name displays.

**Components parameters**
Expressions at sub-components control the visual representation of sub-component parameters: numeric value or assigned parameter name.

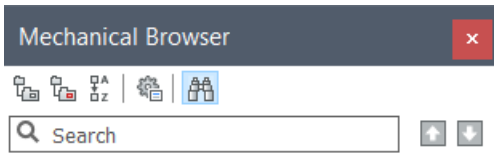
**Expressions of components parameters**
Sub-component parameters control the visibility of sub-component parameters.

**Sub-components of standard parts**
Parameters at properties add a parameters section for the selected instance to the Mechanical browser properties

#### 21.2.2.7 Show search

Toggles the search field and associated tools on and off. You can also press CTRL+F to toggle Search.

**Search field**
Searches the parameter list for the character string you enter. The arrow tools to the right of the search field enable you find the next or previous occurrence of the text string.




### 21.2.3 Сравнение 3D моделей

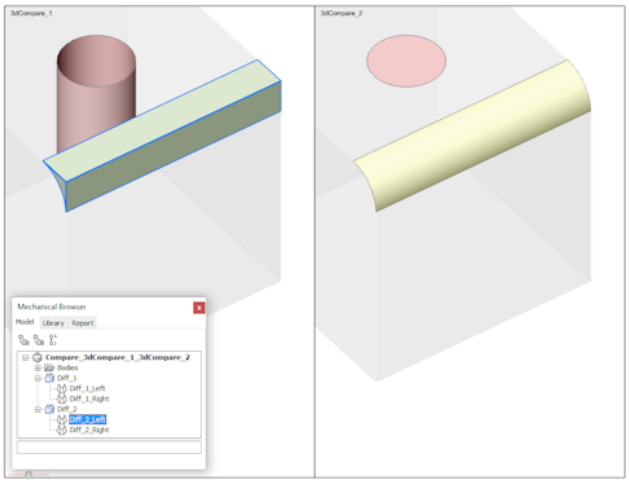
Команда: *3DCOMPARE*.

Вы можете выполнить сравнение 3D моделей, сохраненных в двух .dwg файлах, включая те, которые были получены путем импорта в BricsCAD файлов других форматов. Инструмент эффективно обрабатывает большие модели и формирует отчет об обнаруженных отличиях на панели **Обозреватель конструкции**. Поддерживаются различные режимы визуализации.

#### 21.2.3.1 Определения

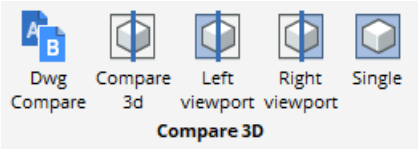
3D модель представляет собой .dwg файл с одним или несколькими 3D телами или поверхностями. При сравнении двух 3D моделей BricsCAD сначала пытается автоматически выровнять модели в 3D пространстве. Затем выполняется сравнение геометрии всех граней 3D тел и поверхностей с учетом заданного допуска. Если каждой грани в первом файле соответствует аналогичная грань во втором файле и наоборот, такие модели считаются одинаковыми. Если какая-либо грань в первом файле не имеет аналогичной грани во втором файле или совпадает только часть этой грани, такие модели считаются различными. Таким образом, отличие - это грань или часть грани 3D тела или поверхности, для которых не найдены соответствующие грани или части граней в другом файле. Если два и более отличия модели имеют смежную геометрию, они группируются вместе.

Для каждой группы отличий в **Обозревателе конструкции** создается отдельный узел. Если Вы раскроете такой узел, то сможете увидеть входящие в него отдельные отличия. При выборе отличия в **Обозревателе конструкции** выделяется соответствующий этому отличию элемент на чертеже.



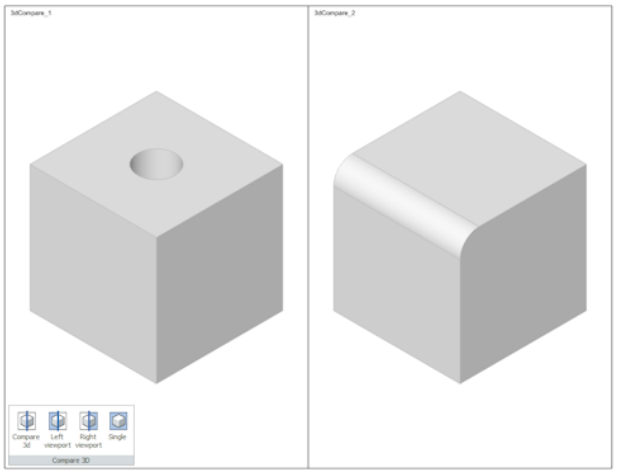
### 21.2.3.2 Панели инструментов

Для запуска процедуры сравнения 3D моделей Вы можете использовать панель **Сравнение модели** на закладке ленты **Сервис**.




### 21.2.3.3 Визуализация

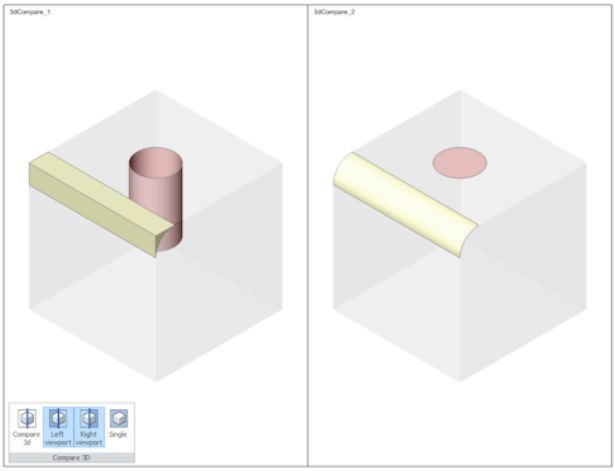
Команда *3DCOMPARE* создает новый .dwg файл с именем **Сравнение_<Файл_1>_<Файл_2>.dwg**. Сравниваемые модели подключаются к этому чертежу как внешние ссылки. Специальный лист с именем **Сравнение** содержит два видовых окна, по одному для каждой из двух моделей. Обнаруженные при сравнении отличия выделяются разными цветами.



Режим отображения моделей при сравнении определяется с помощью системной переменной *3DCOMPAREMODE*. Если значение переменной *3DCOMPAREMODE* не равно 0, модель отображается полупрозрачной, а обнаруженные отличия выделяются разными цветами. Для каждого отличия используется свой цвет. Используя контекстное меню Обозревателя конструкции, вы можете приблизить каждое отличие для более удобного просмотра.

Для отображения отличий в моделях выполните одно из следующих действий:

1 Нажмите кнопку **Левое окно** ( ) и/или **Правое окно** ( ).



2 Дважды щелкните в одном из видовых окон.

3 Используйте мышь/клавиатуру для изменения отображения модели.

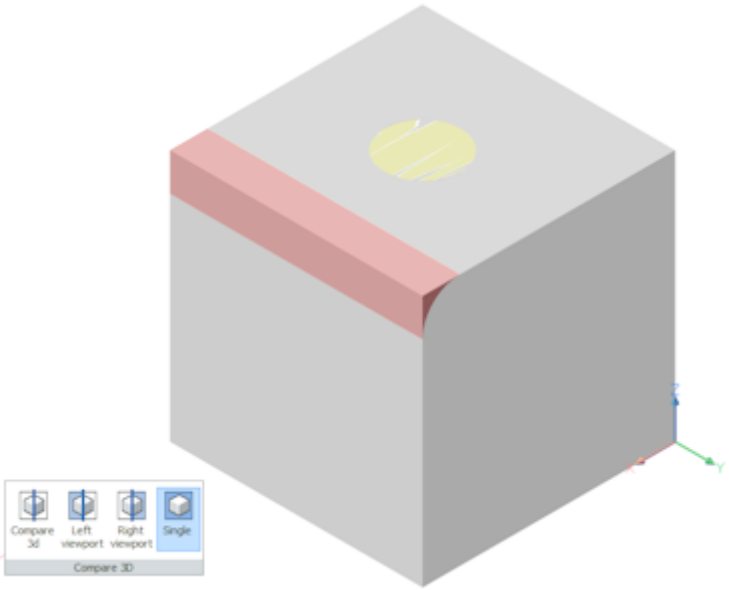
Пример: Приблизить/Отдалить или поворот.

Отображение модели во втором окне будет синхронизироваться автоматически. Это позволит выполнить подробное сравнение элементов обеих моделей.

Альтернативный вариант:

Нажмите кнопку **Единое окно** ( ).

Найденные отличия отобразятся в пространстве модели.

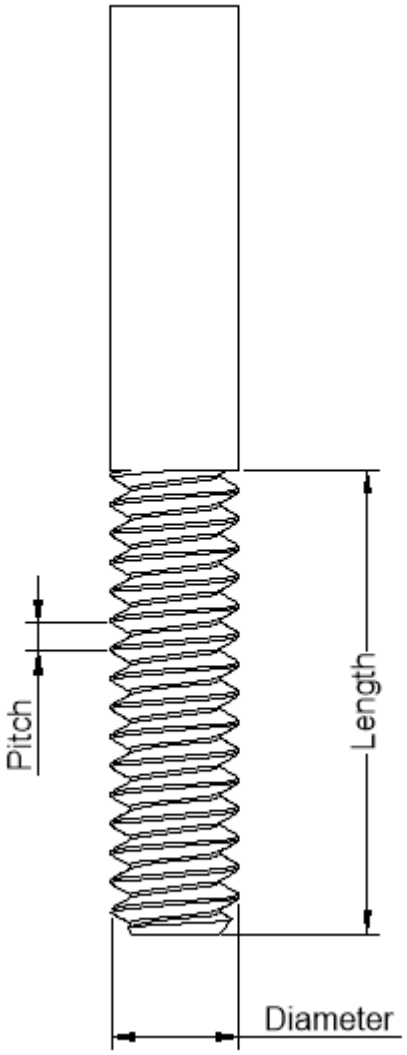


## 21.3 Моделирование деталей

### 21.3.1 Элемент Резьба

Для создания элемента Резьба используйте команду DMTHREAD.

На следующем рисунке представлена структура элемента Резьба:




Элемент Резьба имеет следующие параметры:

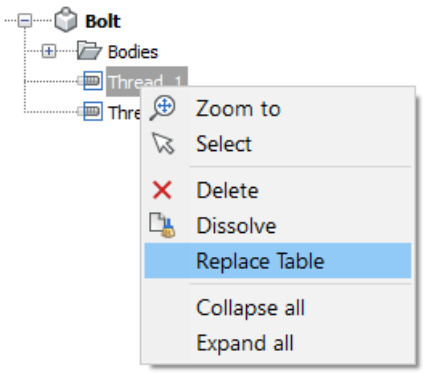
- **Стандарт резьбы**

  Это набор стандартных резьб. Этот список может быть расширен пользовательскими резьбами. При изменении стандарта резьбы геометрия элемента изменяется автоматически.
- **Шаг** резьбы.
- **Длина** резьбы.
- **Диаметр** резьбы.

- **Тип** резьбы: наружная или внутренняя. Если для параметра Тип установлено значение Автоматически, тип будет определен автоматически.

#### 21.3.1.1 Добавление конфигурации резьбы

1 Щелкните правой кнопкой на элементе в **Обозревателе конструкции**.




2 Выберите **Заменить таблицу**.

3 Выберите .csv файл со следующей конфигурацией:

```
Size, Diameter, Pitch Name1, Diameter_value1, Pitch_value1 Name2,
 Diameter_value2, Pitch_value2 Name3, Diameter_value3, Pitch_value3
```

### 21.3.2 Методы проектирования деталей

BricsCAD Mechanical позволяет создавать и редактировать 3D модели.

Способы создания 3D модели:

- Создание твердотельного 3D примитива - параллелепипед, цилиндр и т. д.
- Создание 2D профиля и его выдавливание или вращение для создания 3D тела.
- Создание 3D поверхности с последующим приданием ей толщины или соединение нескольких поверхностей для создания замкнутого ("водонепроницаемого") 3D тела.
- Импорт существующей 3D геометрии (твердые тела и поверхности) из других САПР.

Способы редактирования 3D моделей:

- Выдавливание/вытягивание, перемещение и поворот граней.
- Создание фасок и скруглений на ребрах модели.
- Создание отверстий.
- Добавление выступов и впадин, применение логических операций, использование инструментов прямого и деформационного моделирования.

**Нет зависимости между исходными и дочерними объектами = без использования истории создания объектов**

BricsCAD Mechanical не создает родительско-дочерних зависимостей между геометрическими примитивами и элементами. Все изменения, внесенные в деталь, являются локальными; они не вызывают регенерацию всей геометрии. Отсутствие зависимости от истории создания объектов,

позволяет пользователю сосредоточиться на процессе проектирования и не думать о том, как вносить изменения в будущем.

**3D зависимости = Параметризация**

BricsCAD Mechanical предоставляет мощный набор инструментов для параметризации детали путем добавления 3D зависимостей к ее граням и ребрам и связывания их с глобальными параметрами с использованием выражений и таблицы параметров.

### 21.3.3 Sketching

The Sketching toolset allows you to draw 2D profiles, which can be extruded or revolved to create 3D solids and surfaces, or to add/subtract volume to/from existing entities. The sketching tools are available on the **Sketch** tab of the **Mechanical** workspace ribbon:

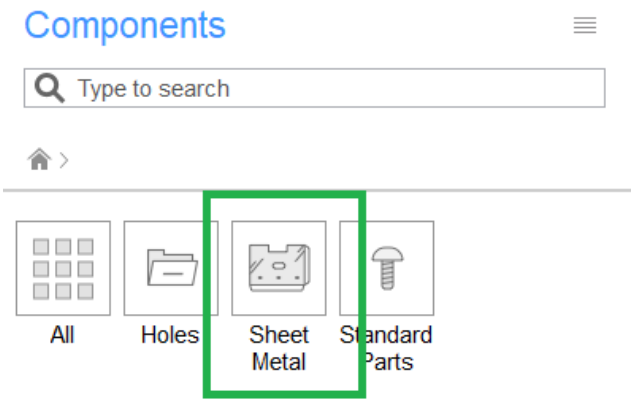



**Рисунок 5:** Mechanical workspace ribbon

Sketches are created in the XY plane of the current User Coordinate System (UCS). Since the dynamic UCS is enabled by default, you can create 2D entities on the 3D solid face under the cursor. Use the Align UCS tool to position the UCS according to your needs.

The drawing tools include familiar 2D primitives, such as a rectangle/polygon, a circle/arc/ellipse, a polyline, and a spline.

Use the Move, Copy, 2D Rotate, 2D Mirror and Array tools to manipulate the sketches.

Use commands *OFFSET*, *STRETCH*, *TRIM*, *EXTEND*, *FILLET*/*CHAMFER*, and *BREAK* to edit the sketches.

Use 2D constraints to make the sketches parametric.

***Прим.:*** The value of the *DELOBJ* (Delete Entity) system variable controls whether sketching entities are deleted when used by *DMEXTRUDE* or *DMREVOLVE* to create 3D solids or surfaces.

### 21.3.4 Solid Modeling

BricsCAD Mechanical provides a rich toolset for solid modeling, including traditional 3D modeling tools like Extrude or Subtract and intuitive Direct Editing tools like; Push/Pull, Copy Faces and powerful 3D Constraints, which allow you to adjust the shape and size of your 3D part.

3D solids tools are available on the **Solid** tab of the **Mechanical** workspace ribbon:



**Рисунок 6:** Solid

**Таблица 1: Solid Modeling**

| Tool/toolset | Description |
|---|---|
| Extrude | Create 3D solids from sketches. |
| Revolve | |
| Loft | |
| Sweep | |
| Thicken | |
| Primitive toolset:<br>Box, Cylinder, Cone, Pyramid, Wedge, Sphere, and Torus | Create 3D solids directly, without underlying sketches. |

**Таблица 1: Solid Modeling**

| Tool/toolset | Description |
|---|---|
| Push/Pull | Edit 3D solids (applied to faces and/or edges). |
| Move | |
| Rotate | |
| Copy Faces | |
| Fillet | |
| Chamfer | |
| Shell | Edit 3D solids. |
| Slice | |
| Boolean toolset:<br>Union, Subtract, Intersect, and Separate | |
| Scale | |
| Mirror | |
| Array | |
| Erase | Can be applied to the whole 3D solid, and to a set of its faces forming a feature (for example a fillet or a hole). |
| Audit | Find and fix errors in imported geometry. |
| Simplify | Automatically recognize canonical faces (planar, cylindrical, conical, spherical, toroidal) from spline faces of your 3D solid. This is often needed before applying 3D constraints. |
| Stitch | Stitch touching surfaces and regions, that form a watertight boundary, into a single 3D solid. |

**Таблица 1: Solid Modeling**

| Tool/toolset | Description |
|---|---|
| Deform toolset:<br>Twist, Move Point, Move Edge, and Transform Curve | Deform your 3D solid uniformly. |
| Smart Selection | Select faces and edges of your 3D solids based on particular geometric and topological criteria, such as length, area and smoothness. |
| Parametrize | Add 3D constraints to faces and edges of 3D solids, to align and resize them. Make your 3D part parametric. |
| Design Intent | Automatically apply geometric 3D constraints, such as parallel, perpendicular, coaxial, and tangent. |

***Прим.:*** BricsCAD is a history-free modeler; it does not create parent-child dependencies between features, nor regenerates geometry on parameter change. You can embed full-blown parametric control into any 3D part (including imported ones) using 3D constraints.

### 21.3.5 Поверхностное моделирование

BricsCAD Mechanical предоставляет мощный набор инструментов для поверхностного моделирования и преобразования 3D тел в поверхности и обратно.

Инструменты для моделирования поверхностей доступны на закладке ленты **Поверхность** рабочего пространства **Mechanical**:

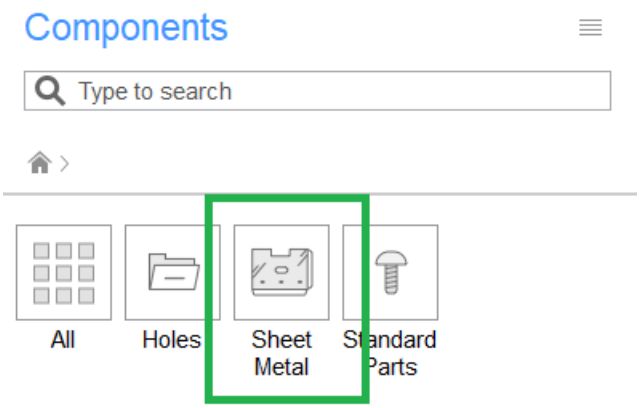



**Рисунок 7:** Поверхность

Инструменты моделирования поверхностей во многом похожи на инструменты редактирования объектов и создания 3D зависимостей, применяемые при твердотельном моделировании.

Единственное отличие состоит в том, что при моделировании поверхностей инструменты Выдавливание/Вращение/Лофт/Сдвиг могут создавать поверхности из замкнутых 2D профилей.

Инструмент Область создает область из замкнутого 2D профиля. Области могут быть сшиты вместе для создания большей поверхности.

Другие методы создания поверхности состоят в том, чтобы выделить одну или несколько смежных граней 3D тела/поверхности и разбить 3D тело на набор поверхностей и областей.

### 21.3.6 Моделирование произвольных форм

#### 21.3.6.1 Обзор

Редактируйте Ваши твердотельные и поверхностные модели с помощью инструментов деформационного моделирования BricsCAD Mechanical для придания им необходимой формы. Изменяйте геометрию твердых тел и поверхностей, перемещая, вращая и скручивая их ребра независимо от истории создания объектов.

Отсутствие зависимости от истории создания объектов, позволяет пользователю сосредоточиться на процессе проектирования и не думать о том, как вносить изменения в будущем.

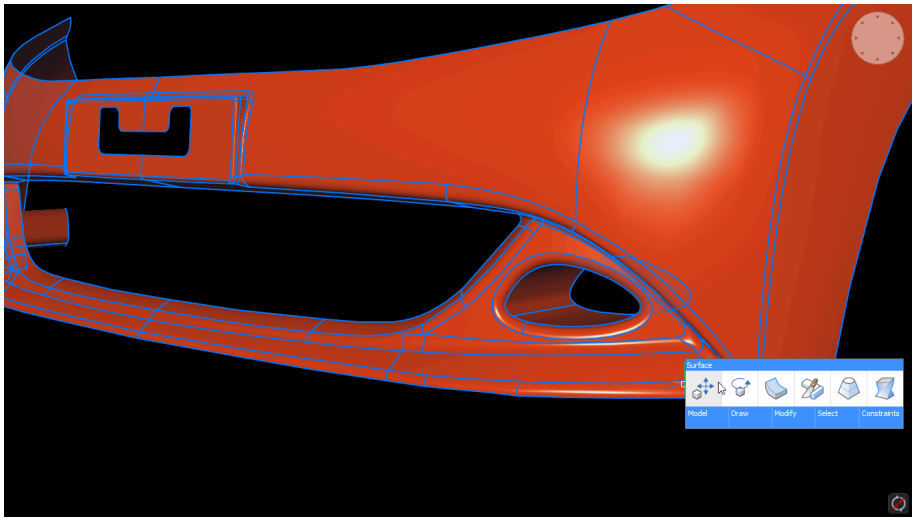



**Рисунок 8:** Моделирование произвольных форм

#### 21.3.6.2 Инструменты деформационного моделирования

Вы можете использовать следующие инструменты деформации:



**Рисунок 9:** Деформация

**Таблица 2: Инструменты деформационного моделирования**

| Инструмент | Команда | Description |
|---|---|---|
| Переместить ребро | *DMDEFORMMOVE* | Деформирует грани перемещением или вращением их ребер. |
| Переместить точку | *DMDEFORMPOINT* | Деформирует выбранный набор граней перемещением точки. |
| Деформировать ребро | *DMDEFORMCURVE* | Деформирует грани изменением геометрии их ребер. |
| Скрутить | *DMTWIST* | Деформирует 3D объект, скручивая его часть вокруг заданной оси. |

***Совет:*** Видеоурок: Поверхностное и деформационное моделирование

## 21.4 Моделирование сборок

### 21.4.1 Восходящее и нисходящее проектирование сборок

#### 21.4.1.1 Обзор

Моделирование сборки состоит в использовании механических компонентов для группировки геометрических объектов в иерархии, отображающие структуру разрабатываемого изделия. Моделирование сборок обычно используется в системах компьютерного механического проектирования MCAD (Mechanical Computer-Aided Design), значительно упрощая разработку сложных изделий, состоящих из тысяч и миллионов геометрических объектов.

В этой статье используются следующие термины:

- Сборка - это компонент, который содержит вставки других компонентов
- Подсборка - это отдельная сборка, которая вставляется в другую сборку

- Деталь - это компонент, который не содержит подкомпонентов.

Сборка со вставленными в нее компонентами (которые в свою очередь могут быть сборками, состоящими из других вставок компонентов) образуют иерархию, которая называется механической структурой 3D модели. Эта структура отображается в Обозревателе конструкции.

Существует два основных метода разработки механических изделий: **нисходящее проектирование** и **восходящее проектирование**. Так же имеется возможность добавления механической структуры к уже существующей геометрии (созданной в BricsCAD или импортированной из другого источника).

Команда *BMNEW* создает новый чертеж для нового механического компонента.

Команда *BMMECH* используется для преобразования существующих блоков и внешних ссылок в компоненты. Команда *BMUNMECH* выполняет обратное действие.

#### 21.4.1.2 Восходящее проектирование

При восходящем проектировании разработка изделия начинается с компонентов нижнего уровня. Эти компоненты группируются в компоненты более высокого уровня. Процесс продолжается до тех пор, пока не будет разработан самый верхний уровень, представляющий собой готовый продукт.

Основной командой восходящего проектирования является команда *BMINSERT*, которая используется для вставки существующего компонента в текущий чертеж.

#### 21.4.1.3 Нисходящее проектирование

При нисходящем проектировании разработка изделия начинается с определения (но не деталировки) компонентов верхнего уровня. Затем выполняется детализация каждого компонента путем определения составляющих его компонентов более низкого уровня. Процесс продолжается до тех пор, пока не будут определены все компоненты самого низкого уровня.

Команда *BMFORM* позволяет формировать новые подкомпоненты для существующего компонента.

#### 21.4.1.4 Добавление механической структуры к существующей геометрии

Вы можете добавлять иерархию компонентов в любой существующий чертеж, созданный в BricsCAD или импортированный из другой САПР.

Команда *BMFORM* преобразует набор выбранных геометрических объектов в подкомпонент текущего чертежа.

#### 21.4.1.5 Изменение механической структуры

В процессе проектирования у пользователя может возникнуть необходимость смещения компонента вверх или вниз в иерархии структуры.

Команда *BMFORM* формирует для текущего чертежа новый подкомпонент , который будет содержать набор выбранных компонентов.

Команда *BMDISSOLVE* удаляет компонент и перемещает все составляющие его подкомпоненты на тот уровень, на который эти компоненты были вставлены.

#### 21.4.1.6 Использование стандартных компонентов

Примеры стандартных компонентов:

- Винты
- Гайки
- Шайбы
- Подшипники
- Отверстия

BricsCAD предоставляет большую библиотеку стандартных изделий, включающую более 30,000 элементов. Это позволяет пользователям значительно ускорить свою работу.

Для вставки стандартных компонентов используется панель Компоненты.

#### 21.4.1.7 Использование зависимостей в сборке

Зависимости в сборке представляют собой обычные 3D зависимости, связывающие между собой компоненты сборки подобно тому, как связываются между собой грани и ребра твердых тел.. 3D зависимости могут быть применены как к компонентам верхнего уровня, так и к их подкомпонентам.

#### 21.4.1.8 Видимость компонентов

Компонент в сложной сборке может оказаться невидимым из-за окружающих его других компонентов.

Например, поршень невидим внутри блока цилиндров. Чтобы сделать такой компонент видимым, пользователю может потребоваться скрыть другие окружающие его компоненты.

Вы можете скрыть любой компонент, вставленный в текущий чертеж или любой его подкомпонент на любом уровне.

Вы также можете сделать компонент прозрачным и/или визуализировать его в виде каркасной модели.

Команда *BMHIDE* скрывает подкомпонент.

Команда *BMSHOW* отображает скрытый ранее подкомпонент.

### 21.4.2 Команды и панели инструментов

#### 21.4.2.1 Команды

**Таблица 3: Команды для моделирования сборок**

| Значок | Команда | Description |
|---|---|---|
|  | *BMNEW* | Создает новый механический компонент в виде нового чертежа. |
|  | *BMMECH* | Выполняет преобразование текущего чертежа в механический компонент. |
|  | *BMINSERT* | Выполняет вставку существующего механического компонента в текущий чертеж. |
|  | *BMINSERT* | Вставляет существующий механический компонент в текущий чертеж с автоматическим изменением размера новой детали и соединением ее с родительской деталью. |
|  | *BMHARDWARE* | Открывает панель Компоненты со стандартными механическими компонентами. |
|  | *BMFORM* | Формирует новый механический компонент и вставляет его в текущий чертеж. |
|  | *BMCONNECT* | Соединяет две стандартные детали путем создания 3D зависимостей между их элементами соединения. |
|  | *BMREPLACE* | Выполняет замену вставки компонента. |
|  | *BMLOCALIZE* | Выполняет преобразование внешних компонентов в локальные. |
|  | *BMEXTERNALIZE* | Выполняет преобразование локальных компонентов во внешние. |
|  | *BMOPEN* | Открывает исходный чертеж внешнего механического компонента. |
|  | *BMOPENCOPY* | Открывает копию вставки компонента в новом чертеже. |
|  | *BMDISSOLVE* | Расчленяет механический компонент, вставленный в текущий чертеж. |

**Таблица 3: Команды для моделирования сборок**

| Значок | Команда | Description |
|---|---|---|
|  | *BMLINK* | Изменяет целевые 3D тела компонентных элементов. |
|  | *BMUNLINK* | Удаляет связь между вставкой компонента (например, окно) и 3D телом (например, стена) с сохранением проема. |
|  | *BMHIDE* | Скрывает механический компонент или подкомпонент, вставленный в текущий чертеж. |
|  | *BMSHOW* | Отображает скрытые ранее механические компоненты или подкомпоненты текущего чертежа. |
|  | *BMVSTYLE* | Применяет визуальный стиль к выбранному механическому компоненту. |
|  | *BMBALLOON* | Выполняет вставку ассоциативного маркера позиции для компонентов сборки в пространстве модели и их генерированных видов. |
|  | *BMBALLOON* | Позволяет автоматически размещать ассоциативные маркеры на всех компонентах определенного вида чертежа в пространстве листа. |
|  | *BMBOM* | Выполняет вставку таблицы спецификации в текущий чертеж. |
|  | *BMMASSPROP* | Вычисляет массовые характеристики текущей модели на основе данных о плотности, назначенной ее отдельным компонентам. |
|  | *BMUPDATE* | Выполняет перезагрузку всех компонентов со ссылками на внешние файлы и обновляет таблицы спецификаций. |
|  | *BMUPDATE* | Выполняет перезагрузку всех выбранных компонентов со ссылками на внешние файлы и обновляет таблицы спецификаций. |

**Таблица 3: Команды для моделирования сборок**

| Значок | Команда | Description |
|---|---|---|
|  | *BMEXPLODE* | Создает покомпонентное отображение сборки путем перемещения каждого компонента для обеспечения его видимости, сохранения в виде блока и последующей вставки в пространство модели. |
|  | *BMEXPLODEMOVE* | Перемещает выбранные детали для формирования покомпонентного вида в заданном направлении с учетом возможных физических коллизий между компонентами (аналогично параметру Линейный команды *BMEXPLODE*). |
|  | *BMTRAILINGLINES* | Создает направляющие линии для покомпонентного вида. |
|  | *MECHANICALBROWSEROPEN* | Отображает Обозреватель конструкции. |
|  | *PARAMETERSPANELOPEN* | Отображает Диспетчер параметров. |
|  | *BMDEPENDENCIES* | Формирует в командной строке список всех файлов, содержащих определения компонентов, вставленных в сборку. |
|  | *BMRECOVER* | Восстанавливает механическую структуру. |
|  | *BMUNMECH* | Выполняет преобразование текущего механического компонента в простой чертеж. |
| | *BMXCONVERT* | Выполняет для текущего чертежа преобразование твердых тел X-Hardware в механические компоненты. |

### 21.4.2.2 Инструменты

Для работы с механическими компонентами вы можете использовать следующие инструменты:

1 На панелях инструментов **3D Сборка**:

- **3D Сборка**

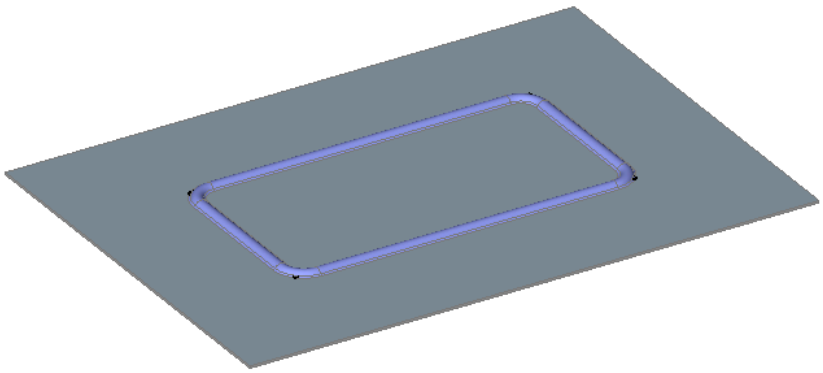


**Рисунок 10:** Сборка

- **3D Сборка: Вставить компонент**



**Рисунок 11:** 3D Сборка: Вставить компонент

- **3D Сборка: Покомпонентный вид**

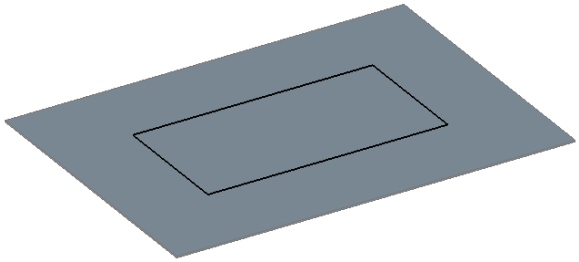


**Рисунок 12:** Покомпонентный вид

- **3D Сборка: Визуализация компонента**

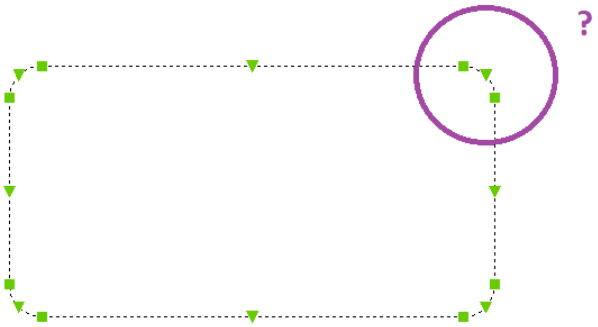



**Рисунок 13:** Визуализация сборки

2 В меню **Сборка** (только для рабочего пространства Mechanical):

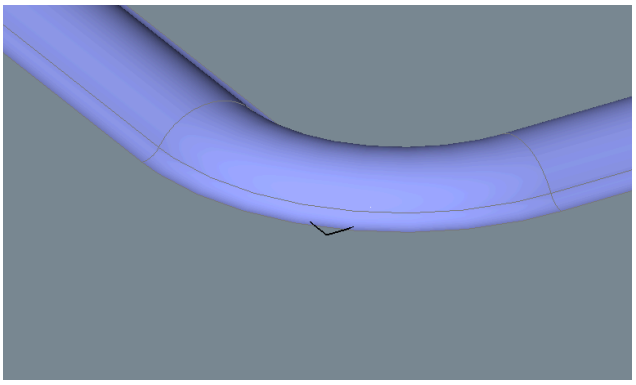


**Рисунок 14:** Меню Сборка

3 На закладке **Сборка** ленты рабочего пространства Mechanical:

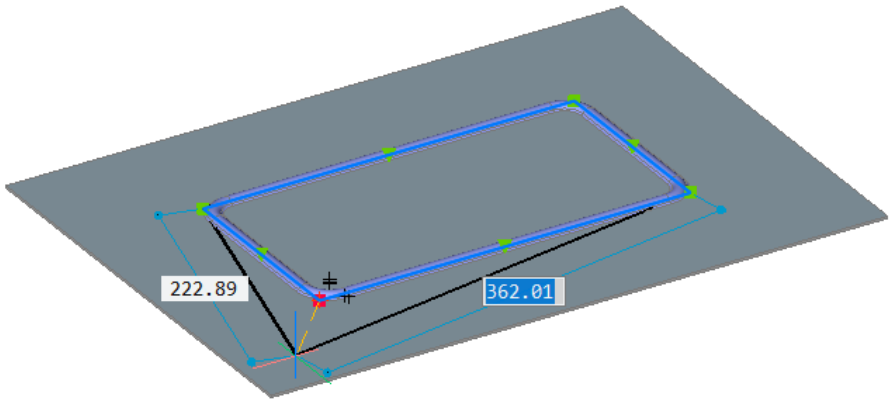



**Рисунок 15:** Сборка

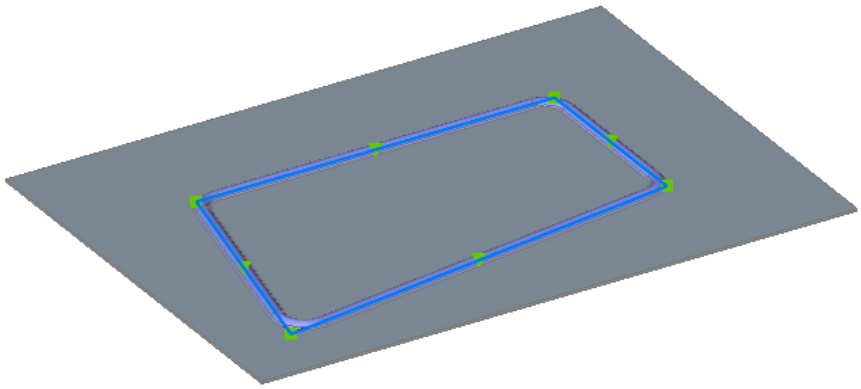


**Рисунок 16:** Инструменты

### 21.4.3 Свойства компонентов

Свойства компонентов и контекстные меню доступны на панели **Обозреватель конструкции**.

Контекстное меню компонента открывается щелчком правой кнопки мыши на выбранном компоненте, а в окне Обозревателя конструкции автоматически отображаются свойства выбранного компонента.



**Рисунок 17:** Обозреватель конструкции

Компоненты делятся на следующие типы: **Основные**, **Вставки** и **Подкомпоненты**.

**Таблица 4: Компоненты**

| Тип компонента | Визуальный стиль Визуализация | Визуальный стиль Каркас | Hidden |
|---|---|---|---|
| Компонент без подкомпонентов | * | * | * |
| Компонент с подкомпонентами | * | * | * |

**Таблица 4: Компоненты**

| Тип компонента | Визуальный стиль Визуализация | Визуальный стиль Каркас | Hidden |
|---|---|---|---|
| Компонент Стандартное изделие |  |  |  |

* Внешние компоненты имеют метку в виде зеленой стрелки:  или .

**Таблица 5: Свойства компонентов**

| Контекстные меню | |
|---|---|
| Контекстное меню основного компонента | Открывается щелчком правой кнопкой мыши на имени основного компонента:<br><br>**Рисунок 18:** Контекстное меню основного компонента<br>• **Обновить**: обновляет иерархию механических компонентов для текущего чертежа в случае, если ссылки на файлы чертежей подкомпонентов были изменены.<br>• **Визуальный стиль**: **Все по видовому окну**: применяет текущий **визуальный стиль** ко всем компонентам в сборке (см. команду *SHADEMODE*).<br>• **Вставить стандартное изделие**: вставляет стандартную деталь как механический компонент в текущий чертеж.<br>• **Преобразовать все компоненты в локальные**: выполняет преобразование всех компонентов в локальные (см. команду *BMLOCALIZE*).<br>• **Преобразовать все компоненты во внешние**: выполняет преобразование всех компонентов во внешние (см. команду *BMEXTERNALIZE*).<br>• **Добавить новый параметр**: создает новый параметр в сборке.<br>• **Выбрать все**: выбирает все компоненты с одинаковым определением.<br>• **Выбрать одинаковые**: выбирает все компоненты с одинаковыми именами и одинаковыми значениями параметров.<br>• **Выделить все**: выделяет все компоненты с одинаковым определением.<br>• **Выделить одинаковые** : выделяет все компоненты с одинаковыми именами и одинаковыми значениями параметров.<br>• **Создать покомпонентный вид**: создает блок с покомпонентным отображением текущей сборки.<br>• **Свернуть все**: сворачивает основной компонент и все его компоненты и подкомпоненты.<br>• **Раскрыть все**: Раскрывает основной компонент и все его компоненты и подкомпоненты. |

**Таблица 5: Свойства компонентов**

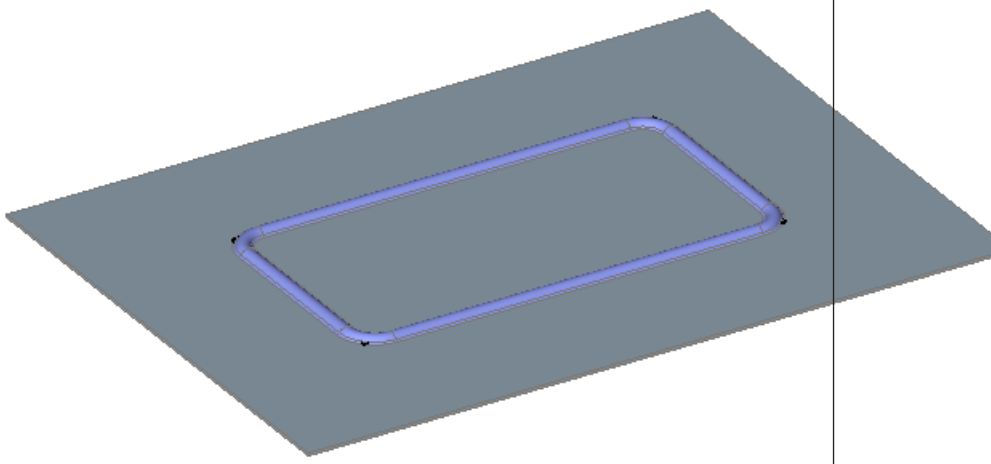
| Контекстные меню | |
|---|---|
| Контекстное меню компонента | Открывается щелчком правой кнопки мыши на имени компонента.<br>Контекстное меню внешних компонентов (слева), контекстное меню локальных компонентов (справа):<br><br>• **Открыть**: открывает чертеж ссылки (см. команду *BMOPEN*)<br>• **Открыть копию**: открывает копию компонента в виде нового чертежа (см. команду *BMOPENCOPY*).<br>• **Обновление**: перезагружает все компоненты из внешних ссылок и обновляет таблицы спецификации (см. команду *BMUPDATE*)<br>• **Заменить**: заменяет вставку компонента (см. команду *BMREPLACE*). При замене экземпляра локального компонента выполняется его преобразование в экземпляр внешнего компонента.<br>• **Заменить все экземпляры**: Заменяет все экземпляры, относящиеся к одному источнику (см. команду *BMREPLACE*). При замене экземпляров локальных компонентов выполняется их преобразование в экземпляры внешних компонентов.<br>• **Преобразовать в локальный**: выполняет преобразование внешних компонентов в локальные (см. команду *BMLOCALIZE*).<br>• **Преобразовать во внешний**: выполняет преобразование компонентов во внешние (см. команду *BMEXTERNALIZE*).<br>• **Назначить материал компоненту**: назначает физический материал для локального компонента.<br>• **Удалить материал компонента**: удаляет определение физического материала из локального компонента.<br>• **Статус в спецификации**: параметр для управления внешним видом компонента в таблицах спецификаций .<br>• **Расформировать**: Расформировывает компонент, вставленный в текущий чертеж (см. команду *BMDISSOLVE*).<br>• **Отменить связь**: удаляет связь между вставкой компонента (например, окно) и 3D телом (например, стена) с сохранением проема.<br>• **Скрыть** ( ) / **Показать** ( ): скрывает или отображает выбранный компонент.<br>• **Скрыть все экземпляры на разрезе**: Устанавливает для свойства **Рассекаемый** всех одинаковых экземпляров значение *Нет*. Определяет, применяется ли команда *VIEWSECTION* к данному экземпляру.<br>• **Визуальный стиль**: Отображает меню **Визуальный стиль**. Доступны все визуальные стили, сохраненные в текущем чертеже. |

**Таблица 5: Свойства компонентов**

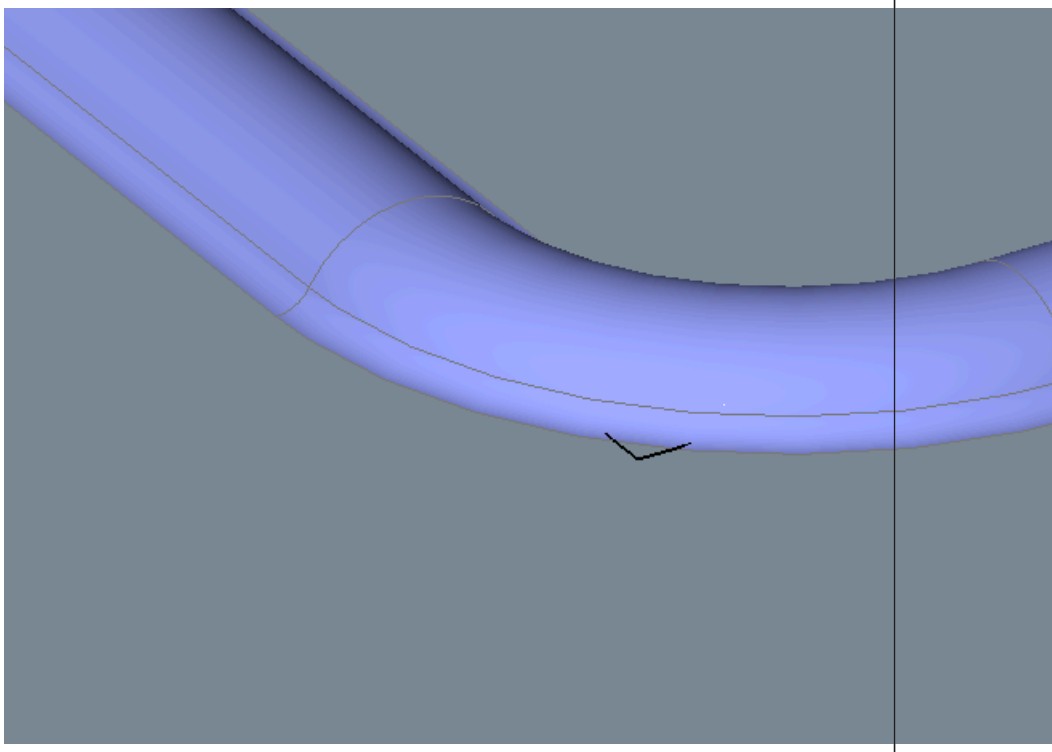
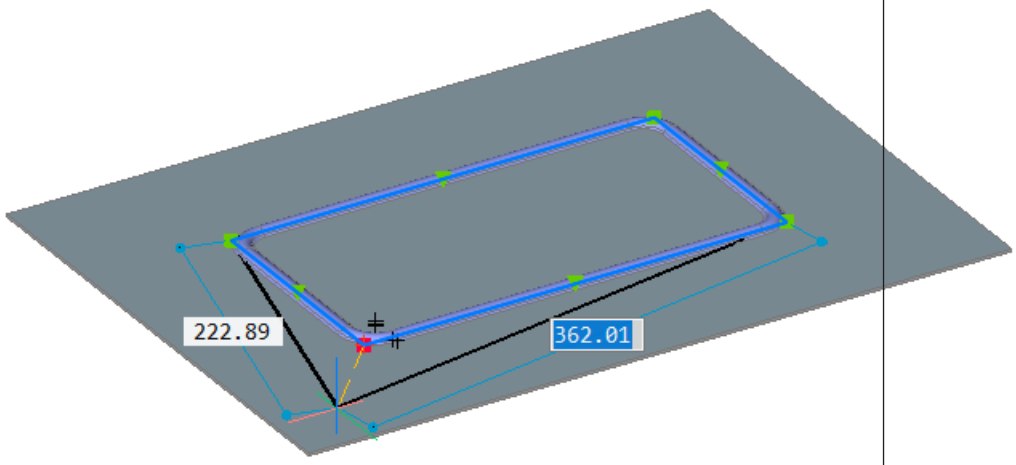
| Контекстные меню | |
|---|---|
| Контекстное меню подкомпонента | Контекстное меню внешних подкомпонентов (слева), контекстное меню локальных подкомпонентов (справа):<br><br><br>• **Открыть**: открывает чертеж ссылки (см. команду *BMOPEN*)<br>• **Открыть копию**: открывает копию компонента в виде нового чертежа (см. команду *BMOPENCOPY*).<br>• **Скрыть** ( ) / **Показать** ( ): скрывает или отображает выбранный подкомпонент.<br>• **Скрыть все экземпляры на разрезе**: Устанавливает для свойства **Рассекаемый** всех одинаковых экземпляров значение *Нет*. Определяет, применяется ли команда *VIEWSECTION* к данному экземпляру.<br>• **Визуальный стиль**: см. Визуальный стиль компонента.<br>• **Показать в модели**: Выполняет масштабирование по габаритам выбранного набора.<br>• **Выбрать все**: выбирает все компоненты с одинаковым определением.<br>• **Выбрать одинаковые**: выбирает все компоненты с одинаковыми именами и одинаковыми значениями параметров. |

**Таблица 5: Свойства компонентов**

| Контекстные меню | |
|---|---|
| Контекстное меню параметра компонента | Открывается щелчком правой кнопки мыши на параметре<br>• **Связать с параметром**: устанавливает связь между одним или несколькими параметрами компонента и параметрами сборки.<br>• **Свернуть все**: сворачивает основной компонент и все его компоненты и подкомпоненты.<br>• **Раскрыть все**: Раскрывает основной компонент и все его компоненты и подкомпоненты. |
| **Свойства** | |

**Таблица 5: Свойства компонентов**

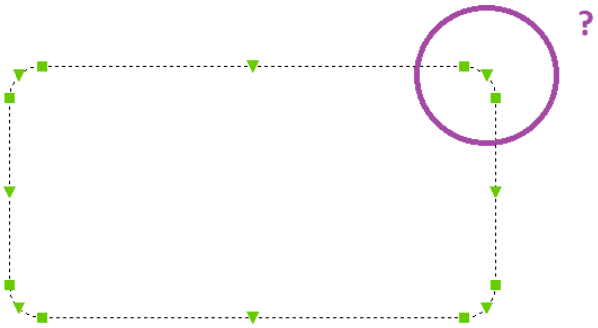
| Контекстные меню | |
|---|---|
| Свойства основного компонента | Открываются щелчком на имени основного компонента:<br><br>**Рисунок 19:** Свойства основного компонента<br>• **Имя**: введите имя основного компонента. По умолчанию имя основного компонента совпадает с именем файла.<br>• **Описание**: дополнительное описание основного компонента.<br>• **Файл**: отображает путь и имя чертежа.<br>• **Тип элемента**: позволяет идентифицировать компонент как BIM компонент или элемент листового металла.<br>• **Вставить как**: выберите **Локальный компонент** или **Внешний компонент**.<br>• **Рассекаемый**: Определяет, влияет ли на компонент команда *VIEWSECTION*.<br>• **Стандартный компонент**: определяет, является ли этот компонент стандартным или нет.<br>• **Статус в спецификации**: определяет режим отображения компонента в таблице спецификации.<br>• **Материал**: материал основного компонента.<br>**Имя**, **Плотность** и **Описание** определяются в диалоговом окне Физические материалы.<br>Выберите раздел **Материал** и выполните одно из следующих действий:<br>- Нажмите кнопку **Обзор** (...), чтобы открыть диалоговое окно Физические материалы и выбрать материал в списке проекта или библиотеки.<br>Плотность выбранного материала используется командой *BMMASSPROP*.<br>- Нажмите кнопку **Удалить** (x).<br>В поле **Материал** теперь указано *<Наследовать>*. Когда компонент вставляется в сборку, определение **материала** копируется из основного компонента сборки. |

**Таблица 5: Свойства компонентов**

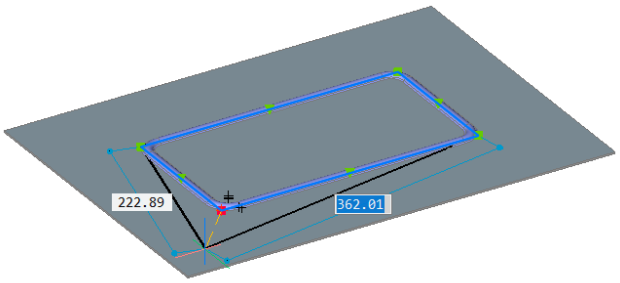
| Контекстные меню | |
|---|---|
| Свойства экземпляров компонента | Щелкние на имени компонента:<br><br><br>**Рисунок 20:** Свойства экземпляров компонента<br>• **Имя**: введите имя компонента. По умолчанию имя компонента совпадает с именем исходного файла.<br>• **Имя компонента**: имя компонента по исходному файлу.<br>• **Описание**: описание компонента по исходному файлу.<br>• **Видимость**: позволяет отображать или скрывать компонент в сборке.<br>Выполните одно из следующих действий:<br>- Щелкните на поле **Видимость**, затем нажмите кнопку со стрелкой и в выпадающем списке выберите *Да* или *Нет*.<br>- Дважды щелкните на поле **Видимость**, чтобы включить/выключить видимость компонента.<br>• **Рассекаемый**: Определяет, влияет ли на компонент команда *VIEWSECTION*.<br>• **Стандартный компонент**: определяет, является ли этот компонент стандартным или нет.<br>• **Статус в спецификации**: определяет режим отображения компонента в таблице спецификации.<br>• **Визуальный стиль**: позволяет управлять визуальным стилем для компонента. По умолчанию наследуется визуальный стиль видового окна.<br>Щелкните на поле **Визуальный стиль**, затем нажмите кнопку со стрелкой и в выпадающем списке выберите необходимый визуальный стиль.<br>• **Файл**: отображает имя файла и путь к чертежу ссылки.<br>• **Материал**: отображает материал компонента.<br>Если для материала установлено значение *<Наследовать>*, копируется материал основного компонента<br>.<br>Если материал явно определен в исходном файле, то используется этот материал. |

**Таблица 5: Свойства компонентов**

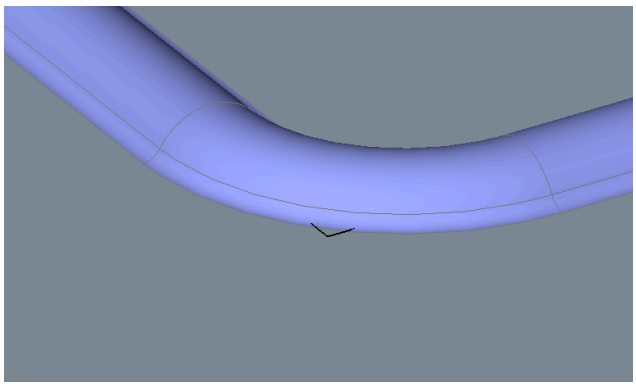
| Контекстные меню | |
|---|---|
| Свойства подкомпонента | Открываются щелчком правой кнопки мыши на имени подкомпонента<br><br>**Рисунок 21:** Свойства подкомпонента<br>• **Имя компонента**: имя компонента по исходному файлу родительского компонента.<br>• **Имя компонента**: имя компонента по исходному файлу.<br>• **Описание**: описание компонента по исходному файлу.<br>• **Видимость**: позволяет отображать или скрывать компонент в сборке.<br>Выполните одно из следующих действий:<br>- Щелкните на поле **Видимость**, затем нажмите кнопку со стрелкой и в выпадающем списке выберите *Да* или *Нет*.<br>- Дважды щелкните на поле **Видимость**, чтобы включить/выключить видимость компонента.<br>• **Рассекаемый**: Определяет, влияет ли на компонент команда *VIEWSECTION*.<br>• **Стандартный компонент**: определяет, является ли этот компонент стандартным или нет.<br>• **Статус в спецификации**: определяет режим отображения компонента в таблице спецификации.<br>• **Визуальный стиль**: управляет визуальным стилем для компонента. По умолчанию наследуется визуальный стиль видового окна.<br>Щелкните на поле **Визуальный стиль**, затем нажмите кнопку со стрелкой и в выпадающем списке выберите необходимый визуальный стиль.<br>• **Файл**: отображает имя файла и путь к чертежу ссылки.<br>• **Материал**: отображает материал подкомпонента.<br>Если для материала установлено значение *<Наследовать>*, копируется материал основного компонента<br>.<br>Если материал явно определен в исходном файле, то используется этот материал. |

**Таблица 5: Свойства компонентов**

| Контекстные меню | |
|---|---|
| **Методы выбора** | |
| Выбор компонентов в модели | Выбранные компоненты выделяются в Обозревателе конструкции. |
| Выбор компонента в Обозревателе конструкции | Выбранный компонент выделяется на чертеже. Для выбора выделенного компонента нажмите клавишу Space. |

### 21.4.4 Массовые характеристики сборки

#### 21.4.4.1 Обзор

Используемая команда *BMMASSPROP*.

Сборка может состоять из нескольких деталей, выполненных из различных материалов. Для корректного вычисления массы, центра тяжести и моментов инерции таких сборок BricsCAD поддерживает раздельное определение плотности для компонентов.

Для определения материала механического компонента используется свойство Материал в **Обозревателе конструкции**.

По умолчанию компонент наследует плотность сборки, в которую он был вставлен.

#### 21.4.4.2 Установка единиц измерения массы

1 Откройте диалоговое окно *Параметры*.

2 Раскройте опции настройки параметра **Единицы измерения для свойств** в разделе **Документ / Вычерчивание / Чертежные единицы**.

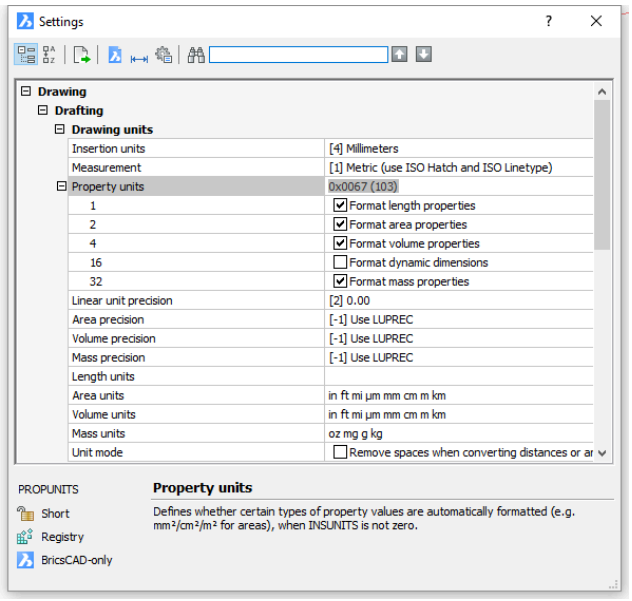



**Рисунок 22:** Параметры: Единицы измерения для свойств

3 Выберите опцию **Форматирование масс** для включения форматирования значений массы в соответствии с заданными единицами измерения.

4 Выберите поле **Единицы измерения массы**.

5 Нажмите кнопку **Обзор** ....

6 Укажите необходимые единицы массы в диалоговом окне *MASSUNITS*.

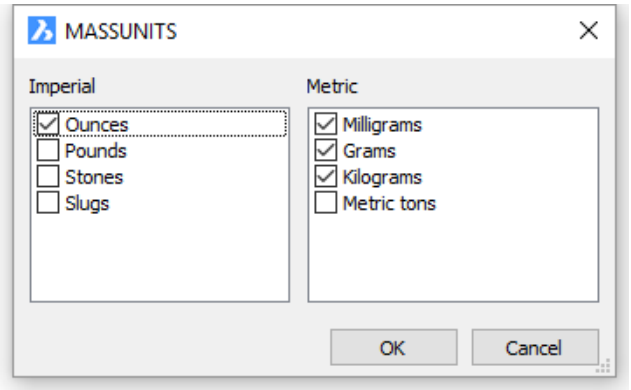



**Рисунок 23:** MASSUNITS

***Прим.:*** Параметр **Единицы массы** используется только для значений массы. Другие массовые характеристики, такие как плотность или моменты инерции, используют формат единиц СИ для метрической системы и английские единицы для английской системы, независимо от значения параметра **Единицы массы**.

7 В поле **Точность массы** выберите необходимое значение точности.

8 Закройте диалоговое окно *Параметры*.

#### 21.4.4.3 Вычисление массовых характеристик

Используемая команда *BMMASSPROP*.

Вы можете выполнить вычисление массовых характеристик как для всей модели, так и для набора 3D тел и/или подкомпонентов. Отчет, содержащий значения массы, объема, моментов инерции, центра тяжести, центробежных моментов инерции, радиуса инерции выводится в командной строке.

При необходимости Вы можете выровнять ПСК по осям инерции. В результате, центр тяжести компонента будет совпадать с точкой начала ПСК, что может быть полезным при изучении поведения изделия под действием сил тяжести.

### 21.4.5 Standard Mechanical Components

#### 21.4.5.1 Overview

Use command *COMPONENTSPANELOPEN*.

BricsCAD comes with a library of more than 1,000 standard mechanical parts available in more than 170,000 different sizes. They are available in the Library panel, and can be drag-and-dropped from the library into your assembly. You can adjust the parameters in the Properties panel, which temporarily lists parameters of the component being inserted. After insertion, parameters can be edited in Mechanical Browser and the Properties panel.

In the Mechanical workspace, the Library panel contains 3 categories:

- Standard Parts
- Holes
- Sheet Metal

#### 21.4.5.2 Standard Parts (Hardware)

The Standard Parts category contains parametric standard hardware grouped into 4 subcategories:

- Fasteners, such as bolts, nuts, and washers.
- Machine Parts, such as bearings, and sprockets.
- Piping, such as pipes, elbows, and flanges.
- Structural Shapes, such as beams, channels, and bars.

The content of each subcategory is structured by type, subtype, and standards.

Example: Standard Parts / Fasteners / Bolts / Hex Head Bolts / ISO / ISO 4017.

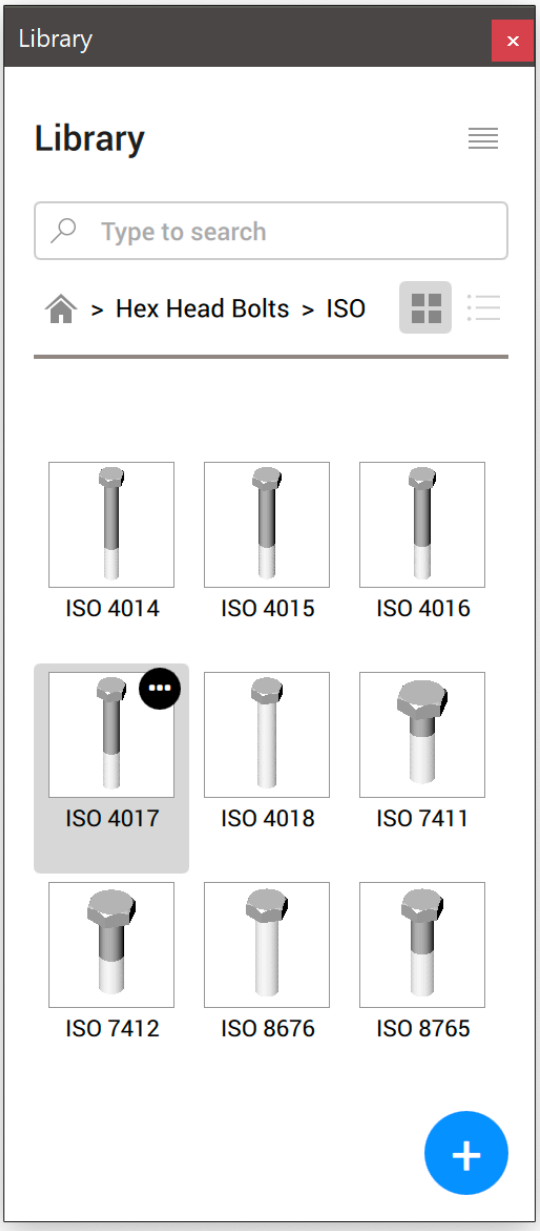



**Рисунок 24:** Library

***Прим.:*** There are 2 specific settings you can configure for Standard Parts in the *Settings* dialog box:

- **Thread representation (THREADDISPLAY)** is used to represent (or not to represent) a 3D thread for bolts and nuts. This setting allows to decrease the memory consumption and increase the performance for large assemblies
- **Maximum number of sprocket teeth (GEARTEETHNUMBER)** is used to limit the number of teeth for sprocket wheels (available under **Standard Parts / SPROCKET)**. This setting allows to decrease memory consumption and increase the performance.

By default, these settings are set at *Off*, in order to optimize the performance of the model. When changed, the settings apply to all new documents.

**Standard representation of thread in drawing views**

Drawing views of your assemblies, generated with the *VIEWBASE* command, contain thread annotations for standard parts as thin lines for side views, and as a 270-degree thin arc for top/bottom views.




**Рисунок 25:** Example

### 21.4.5.3 Holes

The Holes category contains a parametric library of standard holes, such as slots, counterbores, and countersinks.

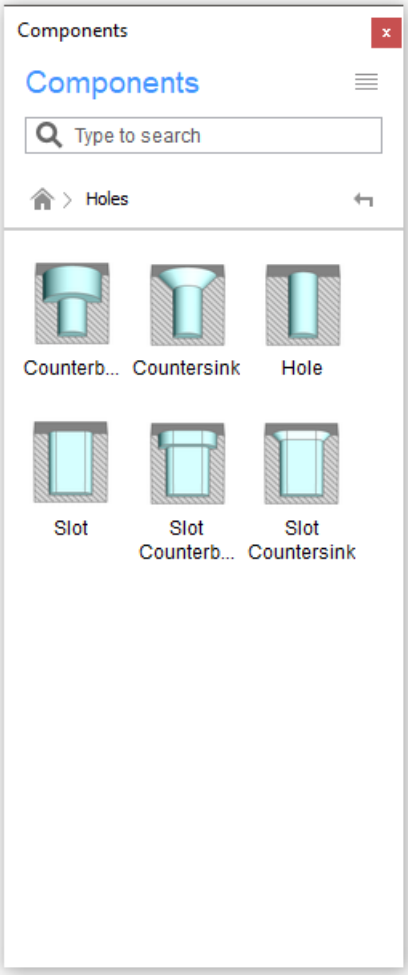



**Рисунок 26:** Holes

There is also a subcategory of threaded holes, represented in drawing views.

### 21.4.5.4 Sheet Metal Form Features

The Sheet Metal category contains a parametric library of sheet metal form features, such as bridges and embosses.

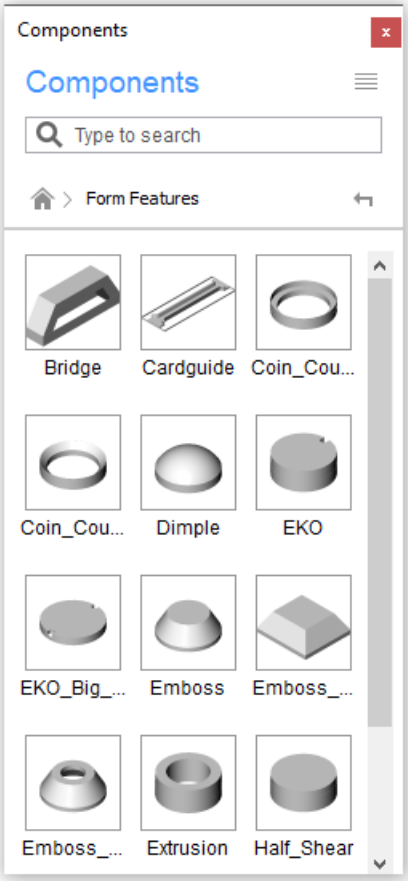



**Рисунок 27:** Form Features

### 21.4.5.5 Place a standard component into the current drawing

1 Choose a category.

2 Choose a standard.

3 Select a type.

4 Click on the desired standard component.

5 Edit the parameters in the properties panel.

6 Click in the drawing area to insert the selected component.

### 21.4.5.6 Create your own standard component

Use command *BMCREATECOMPONENT* to create your own standard parts and add them to Components panel.

### 21.4.5.7 Control the display of threads

1 Open the *Settings* dialog box.

2 Find the *Thread representation* variable (**THREADDISPLAY**).

3 Enable the setting.

4 If applicable, update the document to make threads appear on existing parts:
   - Temporarily set the **Assembly components updating mode (BMUPDATEMODE)** to **[1] Update all components**.
   - Use command *BMUPDATE* to update the document.
   - If applicable, return the **Assembly components updating mode (BMUPDATEMODE)** to the previous value.

#### 21.4.5.8 Control the number of sprocket teeth

1 Open the *Settings* dialog box.

2 Find the *Maximum number of sprocket teeth* variable (**GEARTEETHNUMBER**).

3 Set the value to be sufficient for used sprockets (*500* is recommended).

   ***Прим.:*** Larger values slow down the update of standard sprockets. If the value is smaller than the required number, only that number of sprocket teeth displays.

4 If applicable, update the document to make sprocket teeth appear on existing parts:
   - Temporarily set the **Assembly components updating mode (BMUPDATEMODE)** to **[1] Update all components**.
   - Use command *BMUPDATE* to update the document.
   - If applicable, return the **Assembly components updating mode (BMUPDATEMODE)** to the previous value.

### 21.4.6 Piping in Assembly Design

#### 21.4.6.1 Overview

Use commands: *BMINSERT*, *BMCONNECT*, and *BMBOM*.

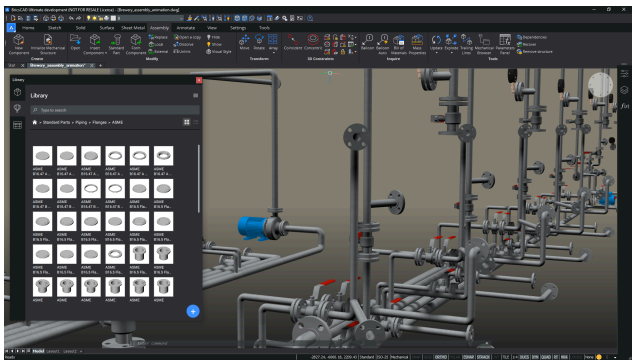


**Рисунок 28:** Piping in Assembly Design

Seamless and welded pipes, elbows, tees, crosses, reducers and flanges are widely used in machinery, automotive, and aviation industries. They are also found in oil-and-gas, chemical industry, and water supply. BricsCAD has a library of parametric piping parts of different standards, such as ASME, DIN, and ISO. These standard parts are parametric. You can easily adjust their parameters, link them to different global parameters, and use 3D constraints to properly connect the parts. BricsCAD allows you

to streamline these routine operations using the **Smart mode** of command *BMINSERT* and a dedicated command *BMCONNECT*.

### 21.4.6.2 Insert piping parts in your assembly

Drag and drop any piping standard part from the **Library** panel into your assembly.

By default, all parts from the Piping category are inserted in Smart mode, which means that if you hover over a previously inserted piping part, BricsCAD:

- Copies parameter expressions of that part to the corresponding parameters of the part being inserted.
- Adds a set of 3D constraints to connect the parts.

***Прим.:*** Press the **Ctrl** key to switch between Smart mode and Regular mode, in which 3D constraints with other parts are not generated.

### 21.4.6.3 Create 3D constraints between piping standard parts manually

Piping standard parts have a set of wireframe entities (points and lines) on the STD_PARTS_PIPING_CONNECTION layer, which is switched off by default.

**Constraining a fitting**

To constrain a fitting (elbow, reducer, flange, ...) with another fitting or a pipe, create 2 Coincident 3D constraints between:

- 2 points of the fittings, which should coincide
- 2 lines of the fittings, which should be collinear.

**Constraining a pipe**

Constraining pipes is more complex because a pipe can have any length. If a pipe is already constrained with a fitting at one side using 2 Coincident 3D constraints, then constraining it from the other side must be different.

You must create:

- A Coincident 3D constraint between the line, which represents the axis of the pipe, and the corresponding line of the other fitting
- A geometry-driven Distance 3D constraint between the planar face of the pipe on the opposite (already constrained) side and the corresponding point of the fitting, which is being connected.

You must then type the name of the distance constraint in the expression field for the Length parameter of the pipe. Because geometry-driven constraints allow controlling of a particular dimension, the pipe will always have the correct length, and such a parametrization will not limit any further editing of your assembly using direct manipulation of additional non-geometry-driven dimensional constraints.

### 21.4.6.4 Automatically connect piping parts

The set of constraints is created automatically using the Smart mode of command *BMINSERT* or the dedicated command *BMCONNECT* to connect 2 standard parts.

#### 21.4.6.5 Create a Bill of Materials for a piping assembly

In a BOM you may want all pipes of the same type and size (PartNumber), but with different lengths, listed as the same part

Use the Aggregate function of the BOM Manager to combine all such parts in a single row with their total length.

### 21.4.7 Массивы компонентов

#### 21.4.7.1 Обзор

Используемые команды: *ARRAY*, *ARRAYRECT*, *ARRAYPOLAR*, *ARRAYPATH* and *BMINSERT*.

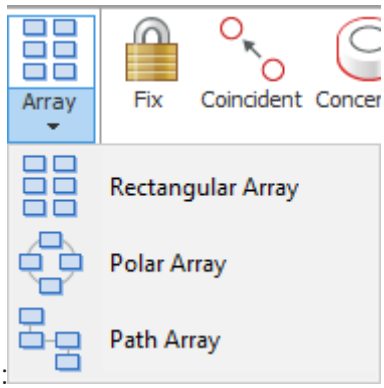
BricsCAD позволяет выполнять многократную вставку копий одного и того же компонента с их равномерным распределением вдоль линии, окружности или произвольной кривой. Также поддерживается создание массивов более высокого порядка (например, прямоугольных или цилиндрических). Это достигается с помощью технологии ассоциативных массивов: прямоугольных, полярных и вдоль траектории.

Ассоциативные массивы - это постоянные объекты, которые доступны для ручного и автоматического редактирования в любой момент после их создания. Например, вы можете изменить количество элементов массива или расстояние между ними. Все массивы Вашей модели отображаются в соответствующем списке Обозревателя конструкции.

Для создания массива может быть использован любой компонент, включая детали, узлы, стандартные изделия и отверстия.

#### 21.4.7.2 Создание массива компонентов

Используйте один из инструментов создания массива, доступных на панели закладки ленты

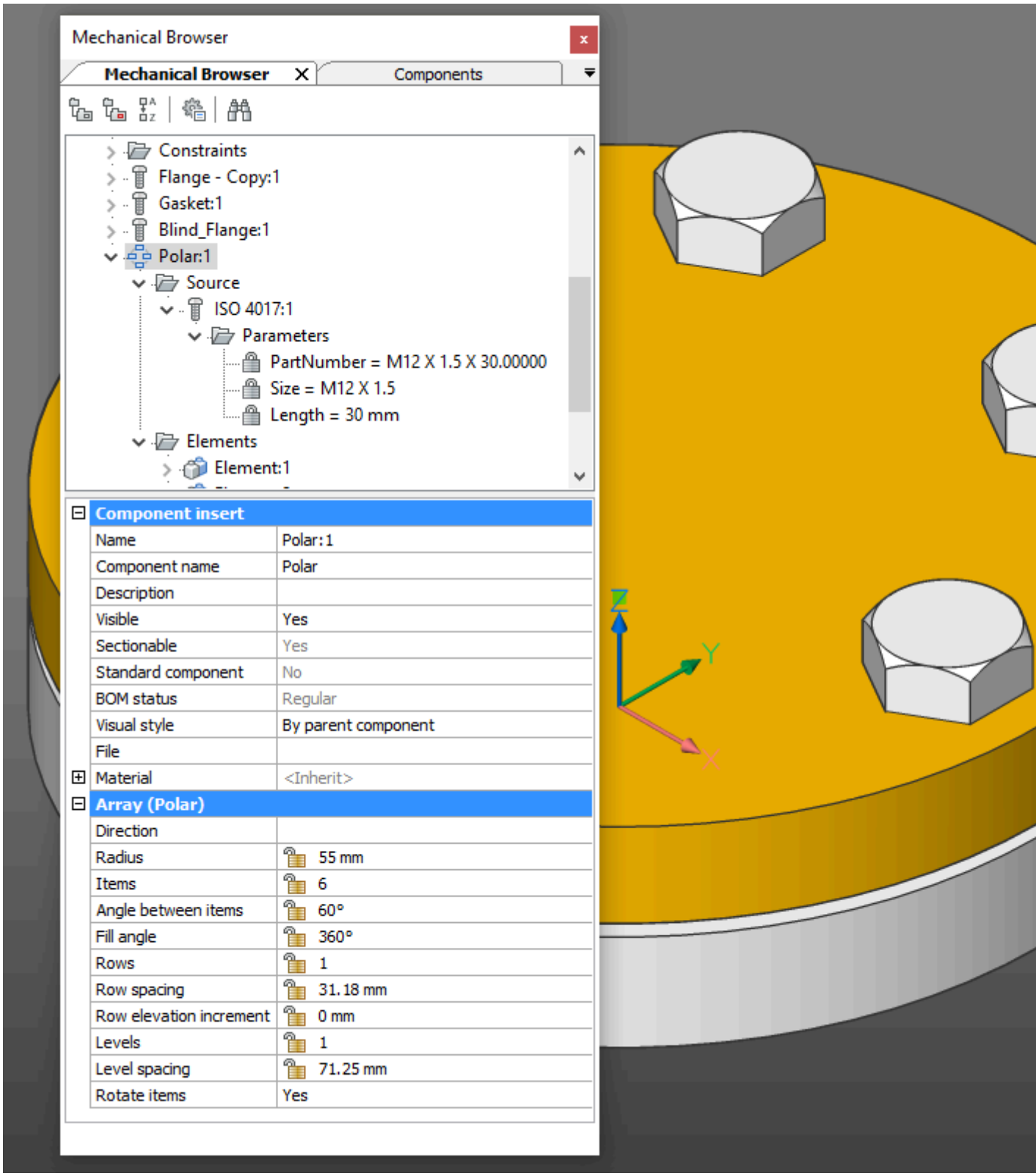



**Сборка** при использовании профиля Mechanical:

- **Прямоугольный массив**
- **Полярный массив**
- **Массив вдоль пути**.

Другой способ создания массива компонентов - использование опции Несколько/Массив команды Вставить компонент.

### 21.4.7.3 Массивы компонентов в Обозревателе конструкции



В Обозревателе конструкции массивы компонентов отображаются как отдельные узлы. Их имя состоит из типа массива (Rectangular, Polar или Path) и его индекса (например, Rectangular:1, Polar:2, Path:3). Эти узлы имеют те же свойства, что и обычные ассоциативные массивы. Редактирование свойств массива позволяет изменить количество элементов в массиве, расстояние и угол между ними, а также другие параметры массива. Вы также можете назначить постоянное выражение для любого свойства. Оно будет использоваться для автоматического пересчета значения свойства при изменении определяющих его параметров.

Каждый узел массива имеет два дочерних узла: Источник и Элементы.

- Исходный объект - узел, содержащий исходный объект и его параметры.
- Узел Элементы включает список элементов массива и позволяет скрывать или изменять визуальный стиль отображения выбранного элемента массива.

#### 21.4.7.4 Массивы компонентов и спецификации

Массивы компонентов прозрачны для структуры продукта. Компоненты внутри массивов считаются вставленными непосредственно в сборку. Компоненты, вставленные в массивы верхнего уровня, включаются в список спецификации как компоненты верхнего уровня.

Столбец **Количество** в спецификации отображает данные об общем количестве экземпляров одного и того же компонента, включая компоненты, входящие в состав массивов.

#### 21.4.7.5 Массивы параметрических компонентов, стандартных изделий и отверстий

Если параметрический компонент вставлен как массив, его параметры также доступны для редактирования и связаны с общими параметрами. Технически параметры связаны с исходным элементом массива. Редактирование параметров исходного элемента массива приводит к соответствующему изменению всех остальных элементов массива.

Стандартные изделия и отверстия также являются параметрическими компонентами, и вы можете редактировать их параметры внутри массивов. Вы можете легко создать детали с несколькими отверстиями, изменить число этих отверстий, их положение и размер, а также вставить в эти отверстия параметрические болты. Эта технология позволяет создавать различные конфигурации сборок с использованием параметрических фланцев.

### 21.4.8 Свойства массива

#### 21.4.8.1 Обзор

Массив компонентов представляет собой ассоциативный массив, состоящий из параметрических компонентов. Массивы компонентов имеют специальное представление в **Обозревателе конструкции**.

Массив компонентов имеет два основных подузла:

- Исходный объект: узел, содержащий исходный объект и его параметры. Исходный объект может представлять собой компонент или подсборку со своим собственным набором свойств.
- Элементы: узел, который содержит все экземпляры исходного объекта.

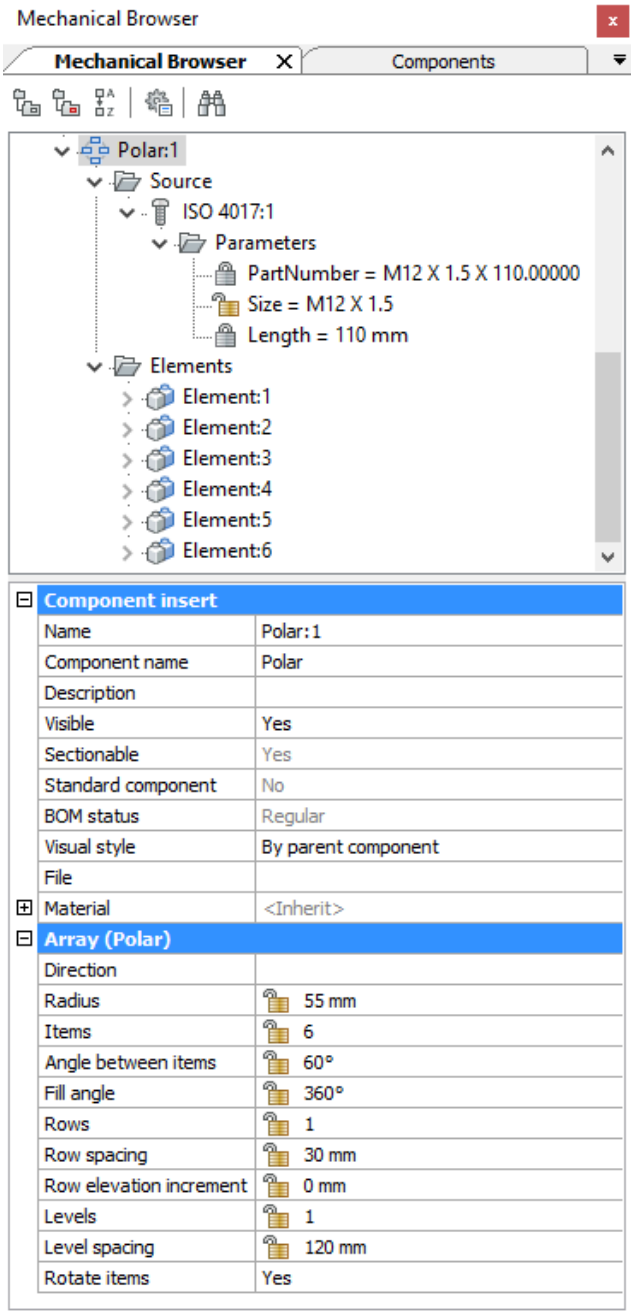
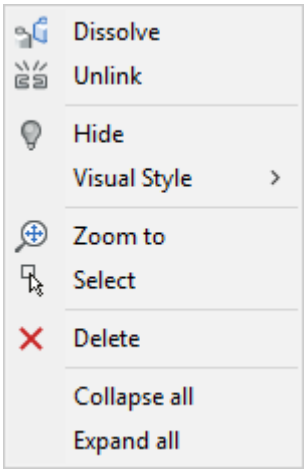
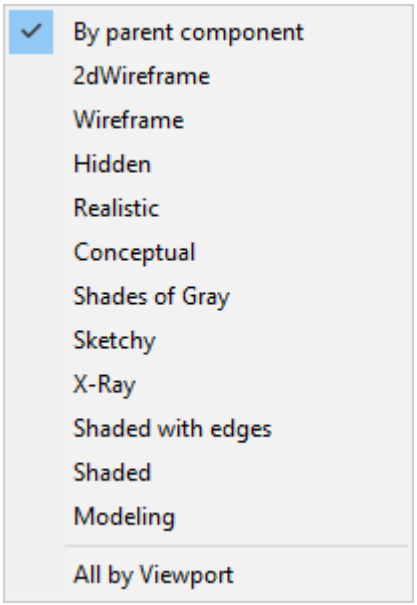



**Рисунок 29:** Обозреватель конструкции

При щелчке правой кнопкой мыши на узле массива отображается контекстное меню для этого массива, а при выборе массива его свойства автоматически отображаются в Обозревателе конструкции.

**Таблица 6: Контекстное меню**

| Контекстное меню | Description |
|---|---|
| Контекстное меню массива компонентов | Щелчок правой кнопкой на массиве открывает контекстное меню, которое содержит следующие опции:<br><br><br>• **Расформировать**: Расформировывает массив, созданный в текущем чертеже (см. команду *BMDISSOLVE*).<br>• **Отменить связь**: Удаляет связь массива с 3D телом путем удаления всей информации, имеющей отношение к сборке.<br>• **Скрыть** / **Показать** : Скрывает или отображает выбранный массив.<br>• **Скрыть все экземпляры на разрезе**: Устанавливает для свойства *Рассекаемый* всех одинаковых экземпляров значение *Нет*. Определяет, применяется ли команда *VIEWSECTION* к данному экземпляру.<br>• **Визуальный стиль**: Отображает меню **Визуальный стиль**. Доступны визуальные стили, сохраненные в текущем чертеже.<br><br><br>• **По видовому окну**: Отображение выбранного массива в соответствии с визуальным стилем, установленным для текущего видового окна (см. команду *SHADEMODE*).<br>• **Все по видовому окну**: Отображение все массивов в соответствии с визуальным стилем, установленным для текущего видового окна (см. команда *SHADEMODE*).<br>• **Показать в модели**: Выполняет масштабирование по габаритам выбранного набора.<br>• **Выбрать**: Выбор массива.<br>• **Удалить**: Удаляет выбранный массив из сборки.<br>• **Свернуть все**: сворачивает основной компонент и все его компоненты и подкомпоненты |

#### 21.4.8.2 Связанные команды

**Таблица 7: Связанные команды**

| Команда | Description |
|---|---|
| *3DARRAY* | Создает 3D массивы. |
| *ARRAY* | Создает полярный или прямоугольный массив объектов с использованием командной строки. |
| *ARRAYCLASSIC* | Создает статический полярный или прямоугольный массив объектов с использованием диалогового окна. |
| *ARRAYEDIT* | Позволяет редактировать объекты ассоциативного массива и их исходные объекты. |
| *ARRAYPATH* | Ассоциативно распределяет копии объектов равномерно по пути в несколько строк и уровней. |
| *ARRAYPOLAR* | Создает полярный ассоциативный массив копий объекта, равномерно распределенных по окружности вокруг точки центра или оси массива с использованием нескольких строк и уровней. |
| *ARRAYRECT* | Ассоциативно распределяет копии объектов по любому количеству строк, столбцов и уровней. |
| *MINSERT* | Выполняет вставку массивов блоков. |

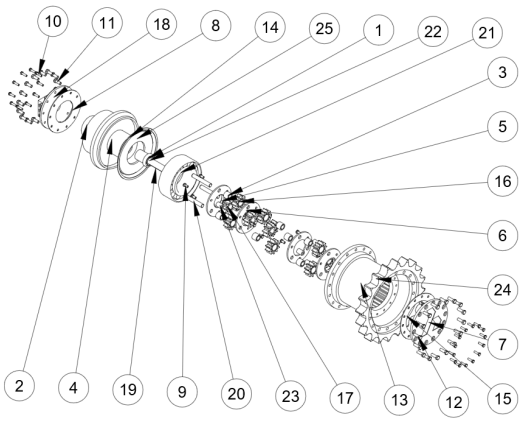
### 21.4.9 Покомпонентные виды

#### 21.4.9.1 Обзор

Используемые команды: *BMEXPLODE*, *BMEXPLODEMOVE*, *BMTRAILINGLINES* и *VIEWBASE*.

Функция покомпонентного отображения позволяет отображать сборку в виде ассоциативного набора ее компонентов без изменения самой сборки. Покомпонентные отображения сборок сохраняются в виде блоков, которые могут быть отредактированы и использованы для вставки в чертеж и генерации видов.

Покомпонентный вид является ассоциативным. Каждая деталь при покомпонентном отображении связана с соответствующей ей другой деталью сборки. Вы можете обновлять покомпонентный вид в соответствии с текущим состоянием сборки, а также добавлять маркеры позиций на сгенерированных видах с помощью команды *BMBALLOON*.




| Bill of Materials Drive_Assembly | | |
|---|---|---|
| No. | Component | Quantity |
| 1 | BBRG-03 | 2 |
| 2 | BBRG-03 | 1 |
| 3 | BBRG-60 | 9 |
| 4 | Body | 1 |
| 5 | Carrier_1 | 2 |
| 6 | Carrier_2 | 2 |
| 7 | Cover | 1 |
| 8 | Cover_Interal_Seal | 1 |
| 9 | DIN 933 | 12 |
| 10 | DIN 933 | 24 |
| 11 | DIN 933 | 16 |
| 12 | DIN 933 | 16 |
| 13 | Drum | 1 |
| 14 | External_Medium_Seal | 1 |
| 15 | External_Seal | 1 |
| 16 | Gear_Planet | 9 |
| 17 | Gear_Sun | 3 |
| 18 | Internal_Cover | 1 |
| 19 | Key | 1 |
| 20 | Pin | 3 |
| 21 | RLR BRG-32 | 1 |
| 22 | Shaft | 1 |
| 23 | Sleeve | 6 |
| 24 | Sprocket | 1 |
| 25 | Washer | 1 |

**Рисунок 30:** Покомпонентное отображение сборки

### 21.4.9.2 Уровни покомпонентного отображения

Покомпонентные виды содержат ссылки на компоненты.

Вы можете выбрать один из двух типов покомпонентных видов: крупноузловой или детальный.

- **Крупноузловой**

  покомпонентный вид содержит ссылки на компоненты верхнего уровня. Компоненты верхнего уровня - это компоненты, которые вставлены непосредственно в основную сборку. Они могут состоять из нескольких деталей и/или подсборок, но при этом сами не являются деталями подсборок. Крупноузловой покомпонентный вид позволяет выполнять просмотр основной композиции сборки.

- **Детальный**

  покомпонентный вид содержит ссылки только на компоненты нижнего уровня. Компоненты нижнего уровня - это детали, которые не включают в себя другие детали или подсборки. Детальный покомпонентный вид позволяет выполнять просмотр всех деталей сборки по отдельности.

***Прим.:*** Если выбран **Детальный** покомпонентный вид и некоторые компоненты среднего уровня содержат не только вставки компонентов, но и другие объекты, эти объекты не будут показаны в покомпонентном отображении, так как для них невозможно определить корректную ссылку. Такие объекты будут преобразованы в соответствующие компоненты нижнего уровня.

### 21.4.9.3 Алгоритмы

В BricsCAD используется три алгоритма для автоматической генерации покомпонентных видов и ручной режим.

**Таблица 8: Алгоритмы**

| Алгоритм | Description |
|---|---|
| Табличный по типам | Создает покомпонентное отображение в виде таблицы, в которой каждая строка объединяет детали одного типа. |
| Линейный | Определяет последовательность разборки компонентов в заданном направлении и упорядочивает компоненты.<br>***Прим.:*** Этот алгоритм учитывает возможные физические коллизии между компонентами. Компоненты могут быть перемещены алгоритмом только в том случае, если нет других компонентов (еще не перемещенных), которые блокируют их перемещение. |
| Авто | Определяет последовательность разборки компонентов в соответствии с иерархией сборки.<br>***Прим.:*** При этом учитываются все возможные физические столкновения. Направление перемещения каждой детали или узла определяется автоматически. |
| Вручную | Создает точную копию сборки для ее редактирования. В этом режиме вы можете создать пользовательский вариант покомпонентного отображения сборки без изменения исходной сборки. |

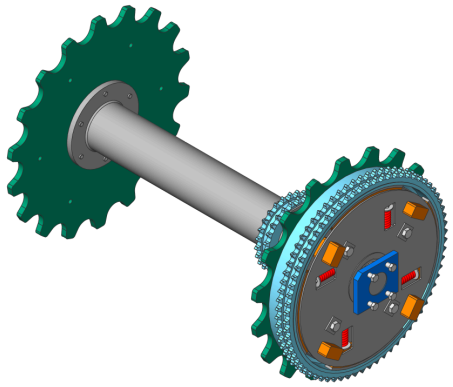
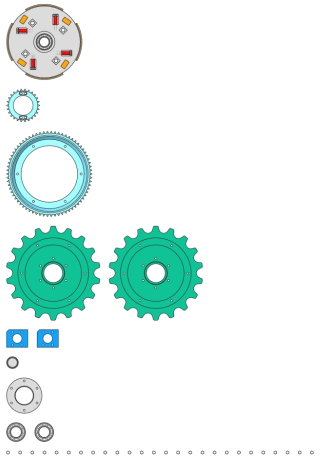
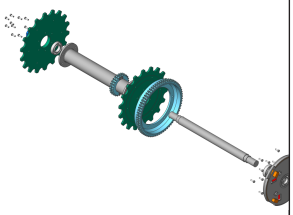
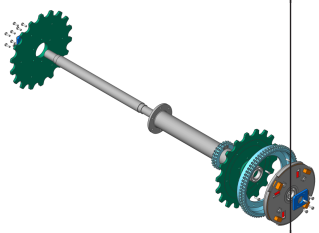
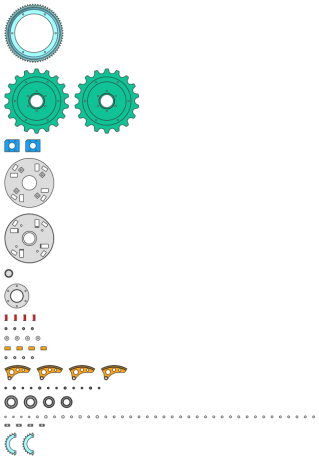
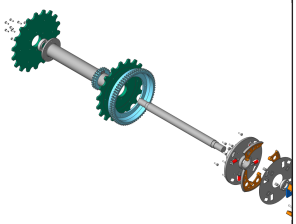
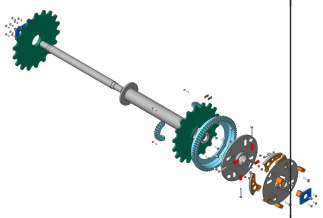


**Рисунок 31:** Пример: Главный привод в сборе

**Таблица 9: Пример: Главный привод в сборе**

| | Табличный по типам | Линейный | Авто |
|---|---|---|---|
| Крупноузловая |  |  |  |
| Детальная |  |  |  |

### 21.4.9.4 Шаги и анимации

Покомпонентные виды могут содержать один или несколько шагов, которые описывают конкретную последовательность сборки или разборки. Каждый шаг соответствует набору компонентов, которые необходимо переместить на этом конкретном этапе, чтобы собрать или разобрать основной узел. Все алгоритмы покомпонентного отображения создают необходимые шаги. Вы можете применять к шагам операции удаления, объединения, разделения или перемещения и при необходимости добавлять дополнительные шаги.

Каждому шагу присваивается уникальное имя, которое может использоваться для описания этого шага.

Вы можете применить анимацию для конкретного шага, а также для всей последовательности шагов, как в прямом, так и в обратном направлении (для анимации разборки и сборки). Чтобы

анимация работала правильно, покомпонентное отображение должно включать начальный шаг, определяющий начальное состояние сборки. Используйте свойство **Автоматически скрыть** покомпонентного отображения, чтобы автоматически скрывать компоненты, которые не важны для определенного шага во время анимации.

### 21.4.9.5 Редактирование покомпонентных видов

Каждый покомпонентный вид сохраняется в виде блока. Вы можете редактировать его с помощью Редактора блоков.

***Важное замечание:*** Не используйте команду *REFEDIT* для редактирования покомпонентных видов. Эта команда несовместима с шагами покомпонентного отображения.

Чтобы открыть покомпонентный вид для редактирования:

1 Щелкните правой кнопкой на покомпонентном виде в Обозревателе конструкций.

2 В открывшемся контекстном меню выберите **Изменить**.

Редактор блоков откроется автоматически.

***Совет:*** Покомпонентный вид также можно открыть для редактирования при завершении команды *BMEXPLODE*.

Когда покомпонентный вид открыт для редактирования, вы можете изменить состояние текущего шага. По умолчанию это последний шаг алгоритма. Но вы можете выбрать один из двух вариантов:

- Используйте свойство **Текущий шаг** покомпонентного вида в Обозревателе конструкции.
- Щелкните правой кнопкой мыши на необходимом шаге в Обозревателе конструкции и выберите в контекстном меню **Применить**.

***Прим.:*** Если вы измените текущий шаг, вы отмените все несохраненные изменения. Перед изменением текущего шага или выполнением любой другой операции убедитесь, что все изменения сохранены.

Для перемещения компонентов внутри покомпонентных отображений в требуемые позиции используйте команды *DMMOVE*, *MOVE*, *DMROTATE* и *BMEXPLODEMOVE*. Команда *BMEXPLODEMOVE* автоматически добавит все необходимые шаги после текущего шага. Для других операций сохраните текущий шаг после завершения редактирования. Сохраняются только отличия между текущим и предыдущим шагами.

Для изменения покомпонентного отображения вы также можете применять операции к шагам.

Например, если вы удалите шаг, все компоненты, перемещенные на этом шаге, будут возвращены на свои позиции, на которых они находились до этого шага.

Сохраните текущий шаг после редактирования, в противном случае изменения будут отменены.

### 21.4.9.6 Направляющие линии

Направляющие линии помогают указать взаимосвязь между деталями. Это 3D полилиния, которая определяет траекторию перемещения детали в процессе разборки.

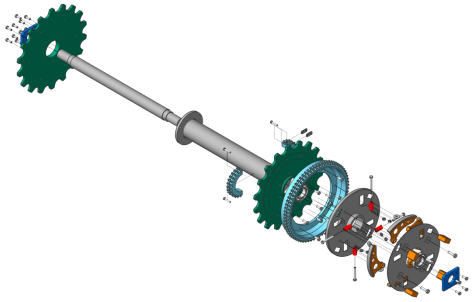


**Рисунок 32:** Пример направляющих линий

Для добавления направляющих линий в открытое для редактирования покомпонентное отображение используется команда *BMTRAILINGLINES*. Этот инструмент создает все направляющие линии. Для этого используется информация, полученная из шагов покомпонентного вида.

Вы можете генерировать направляющие линии как для всех деталей, так и для отдельных узлов сборки. Для узлов сборки направляющие линии будут построены только для деталей этого узла. Будут добавлены только необходимые направляющие линии. Некоторые детали могут вообще не иметь направляющих. Вы также можете вручную выбрать две детали и добавить линию между ними.

При расчете направляющей линии алгоритм автоматически определяет местоположение детали и место, откуда деталь была удалена, чтобы вычислить ее траекторию с учетом перемещения обеих деталей.

Каждая направляющая линия отображается в Обозревателе конструкции. В Обозревателе конструкции вы можете выделять, выбирать, переименовывать и удалять направляющие линии, а также приближать детали, связанные с выбранной линией.

Для редактирования направляющих линий используются стандартные инструменты для работы с полилиниями.

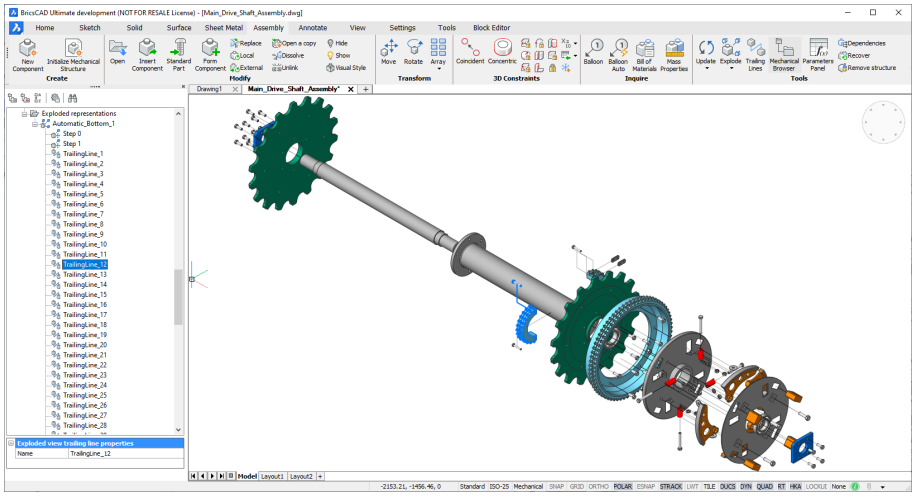


**Рисунок 33:** Свойства направляющей линии покомпонентного вида

### 21.4.9.7 Создание покомпонентного вида

Команда *BMEXPLODE* позволяет создавать покомпонентное отображение текущей сборки.

***Прим.:*** Для работы этой команды необходима сборка, содержащая хотя бы одну вставленную в нее деталь.

1 Для запуска команды *BMEXPLODE* выполните одно из следующих действий:

- Нажмите кнопку **Покомпонентный вид** на панели **Сервис** закладки ленты **Сборка**.
- Нажмите кнопку **Покомпонентный вид** на панели инструментов **Сборка**.
- Выберите пункт **Покомпонентный вид** в меню **Сборка**.
- Введите в командной строке **bmexplode** и нажмите клавишу Enter.

Запрос программы: Укажите алгоритм размещения элементов [Табличный по типам/Линейный/Автоматически/Вручную/Параметры] <Вручную>:

2 При необходимости определите *Уровень* и *Имя* покомпонентного вида, выбрав для этого опцию **Параметры**.

3 Выберите алгоритм.

Запрос программы: Определите поведение покомпонентного вида [Изменить/Создать чертежные виды/Завершить] <Завершить>:

4 Выполните одно из следующих действий:

- Для редактирования покомпонентного вида выберите **Изменить**.
- Для генерации чертежа покомпонентного вида выберите **Создать чертежные виды**.
- Для завершения команды выберите **Завершить**.

### 21.4.9.8 Генерация чертежных видов на основе покомпонентного отображения

1 Запустите команду *VIEWBASE*.

Запрос программы: Укажите объекты или [Вся модель/Шаблоны/Специальные виды/Параметры выбора] <Вся модель>:

2 Выберите опцию **Специальные виды**.

Запрос программы: Указать вид [Покомпонентный вид/Развертка/Назад] <Назад>:

3 Выберите опцию **Покомпонентный вид**.

Откроется диалоговое окно выбора покомпонентного вида.

4 Выберите покомпонентный вид.

Запрос программы Введите имя нового или существующего листа для установки текущим <Лист1>:

5 При необходимости, введите *Имя листа* для покомпонентного вида.

6 Укажите расположение сгенерированных видов чертежа в пространстве листа.

***Прим.:*** Для использования маркеров позиций (см. команду *BMBALLOON*) необходимо для системной переменной *GENERATEASSOCVIEWS* установить значение *ON*.

### 21.4.9.9 Обновление покомпонентного вида

Для синхронизации покомпонентных видов с текущим состоянием сборки выполните команду *BMUPDATE*.

При необходимости выполните настройку покомпонентного вида. Такая настройка может потребоваться при добавлении и/или удалении некоторых деталей.

***Прим.:*** Автоматическое обновление расположения деталей для покомпонентного отображения сборки, созданного в **ручном** или **линейном** режиме, недоступно.

### 21.4.9.10 Управление покомпонентными видами

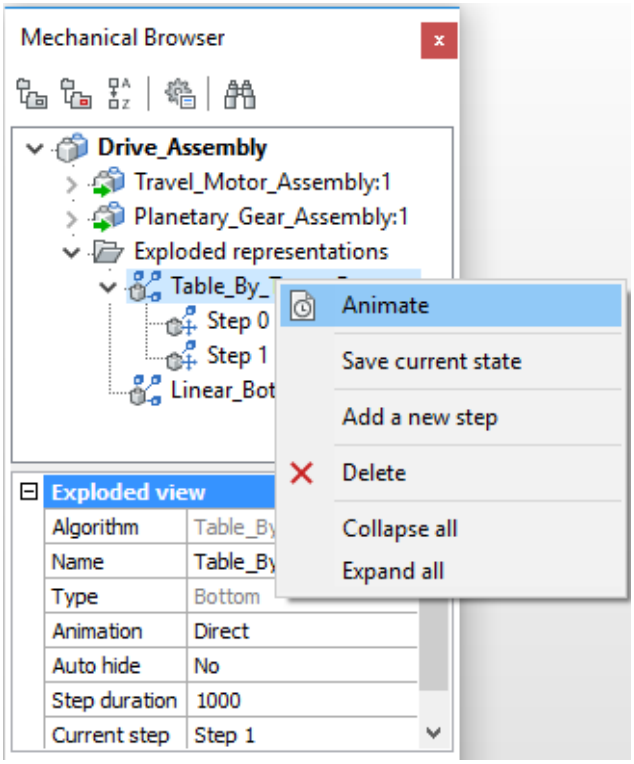



**Рисунок 34:** Управление покомпонентными видами

Откройте Обозреватель конструкции для просмотра всех покомпонентных видов текущего документа. Все покомпонентные виды будут перечислены в разделе **Покомпонентные виды**. Щелчок правой кнопкой мыши на выбранном покомпонентном виде открывает контекстное меню с перечнем доступных операций.

**Свойства покомпонентных видов**

Каждый покомпонентный вид имеет набор свойств, некоторые из которых могут быть изменены пользователем.

**Таблица 10: Свойства покомпонентных видов**

| Свойство | Description |
|---|---|
| Алгоритм | Наименование алгоритма, использованного для создания этого покомпонентного вида. |
| Имя | Имя покомпонентного вида. Оно должно быть уникальным для всех покомпонентных видов и блоков модели. |
| Type | Тип покомпонентного вида: *Крупноузловой* или *Детальный*. |
| Анимация | Направление анимации. Выберите одно из двух значений: *Прямое* (от начала до конца) или *Обратное* (от конца до начала). |
| Автоматически скрыть | Если выбрано значение *Да*, все второстепенные детали для определенного шага будут скрыты во время анимации. |
| Длительность шага | Продолжительность каждого шага анимации по умолчанию в миллисекундах. Значение по умолчанию - 1000, что соответствует длительности 1 с. |
| Текущий шаг | Шаг, соответствующий текущему состоянию покомпонентного вида. |

**Операции с покомпонентными видами**

Перечень доступных операций зависит от того, открыт ли покомпонентный вид для редактирования.

Если покомпонентный вид **не открыт для редактирования**, будут доступны следующие операции:

**Таблица 11: Операции с покомпонентными видами**

| Операция | Description |
|---|---|
| Изменить | Открывает покомпонентный вид для редактирования.<br>***Совет:*** Для редактирования покомпонентных видов вы также можете использовать команду *BEDIT*. |
| Удалить | Удаляет из документа покомпонентный вид и связанный с ним блок. |

Если покомпонентный вид **открыт для редактирования**, будут доступны следующие операции:

**Таблица 12: Операции с покомпонентными видами**

| Операция | Description |
|---|---|
| Анимировать | Анимирует всю последовательность шагов. |
| Сохранить текущее состояние | Сохраняет позиции деталей на текущем шаге. |
| Добавить новый шаг | Добавляет новый шаг после последнего.<br>***Прим.:*** Новый шаг будет автоматически установлен как *текущий*. Все несохраненные изменения покомпонентного вида будут потеряны. |
| Удалить | Удаляет из документа покомпонентный вид и связанный с ним блок. |

Все шаги этого покомпонентного вида будут перечислены в Обозревателе конструкции.

### 21.4.9.11 Управление шагами покомпонентного вида

Откройте Обозреватель конструкции и выберите покомпонентный вид для его редактирования. Все шаги будут перечислены под соответствующим узлом структуры сборки.

**Свойства шагов**

**Таблица 13: Свойства шагов**

| Свойство | Description |
|---|---|
| Имя | Наименование шага. Должно быть уникальным для текущего покомпонентного вида. |
| Длительность, мс | Длительность шага. По умолчанию равна длительности шага, установленной для покомпонентного вида. |
| Тип длительности | Указывает, следует ли брать длительность по покомпонентному виду или устанавливать ее отдельно для данного конкретного шага. |
| Значение длительности (только для типа Абсолютная длительность) | Длительность этого шага в миллисекундах. |

**Операции с шагами**

**Таблица 14: Операции с шагами**

| Операция | Description |
|---|---|
| Применить | Устанавливает шаг как текущий и обновляет расположение деталей в покомпонентном виде в соответствии с этим шагом. |
| Анимировать | Анимирует выбранный шаг. |
| Установить по предыдущему | Устанавливает позиции всех деталей такими же, как и на предыдущем шаге. |
| Объединить с предыдущим | Объединяет текущий шаг с предыдущим. Предыдущий шаг удаляется. |
| Разделить | Разделяет шаг на последовательность шагов, каждый из которых соответствует одной детали. |
| Переместить шаг вверх | Перемещает шаг на одну позицию вверх. |
| Переместить шаг вниз | Перемещает шаг на одну позицию вниз. |
| Удалить | Удаляет шаг и связанные с ним перемещения деталей. После этого происходит обновление позиций деталей в покомпонентном виде. |

### 21.4.9.12 Редактирование покомпонентных видов

1 Выберите покомпонентный вид в Обозревателе конструкций.

2 Щелкните правой кнопкой мыши на покомпонентном виде и в открывшемся контекстном меню выберите **Изменить**.

***Прим.:*** В качестве альтернативы вы можете запустить команду *BEDIT* и выбрать блок покомпонентного вида.

3 Выполните необходимые операции редактирования.

4 Сохраните текущий шаг.

5 Для сохранения изменений используйте команду *BCLOSE* с опцией *Сохранить*.

***Прим.:*** Избегайте использования опции *Отменить изменения* при выполнении команды *BCLOSE*, так как это может привести к рассогласованию состояния блока и шагов покомпонентного вида. Вместо этого используйте опцию *Undo (На шаг назад)*.

### 21.4.9.13 Добавление направляющих линий

1 Открывает покомпонентный вид для редактирования.

2 Запустите команду *BMTRAILINGLINES*.

3 Укажите детали для создания направляющих линий или выберите всю модель.

4 Выберите, какие точки деталей следует использовать для создания траектории: начало координат или центр.

5 Выполните редактирование направляющих линий (если доступно).

6 Сохраните изменения.

## 21.4.10 Диспетчер спецификаций

### 21.4.10.1 Обзор

Используемые команды: *BMBOMPANELOPEN*, *BMBOMPANELCLOSE* и *BMBOM*.

Диспетчер спецификаций позволяет просматривать, создавать и редактировать спецификации (BOM) в текущем чертеже. Вы можете использовать Диспетчер спецификаций для интерактивной настройки как новых, так и существующих спецификаций. Вы можете сохранить конфигурацию спецификации в виде файла шаблона и повторно использовать его в других чертежах. Вы также можете добавить спецификацию, которая не связана с таблицей на чертеже.

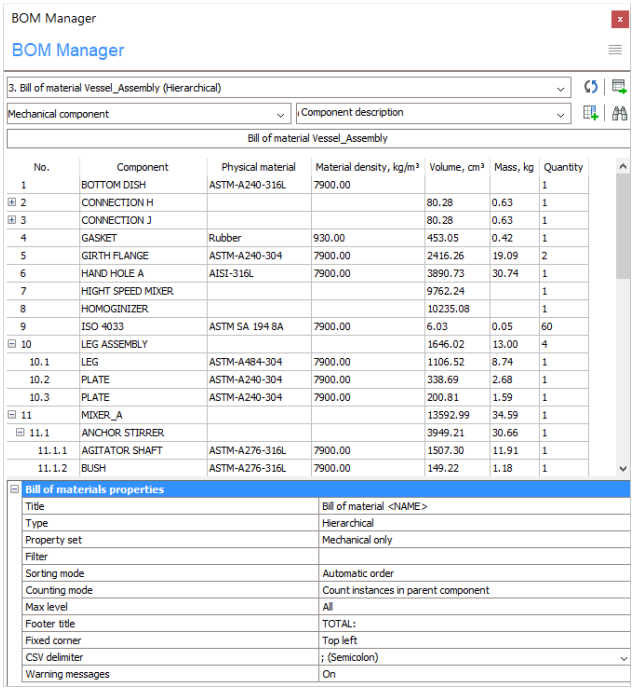



**Рисунок 35:** Диспетчер спецификаций

С помощью Диспетчера спецификаций вы можете:

- Выполнять просмотр существующих спецификаций.
- Создавать новые спецификации, используя один из доступных стандартных типов (Крупноузловая, Детальная, Иерархическая) или существующий шаблон спецификации.

- Редактировать спецификации:
  - Добавлять или удалять столбцы (включая столбцы с формулами).
  - Перемещать выбранные столбцы на новые позиции.
  - Переименовывать столбцы.
  - Устанавливать режим сортировки в спецификации.
  - Настраивать конфигурацию нижнего колонтитула для каждого столбца.
  - Управлять отображением единиц измерения для каждого столбца.
  - Устанавливать ширину каждого столбца.
  - Управлять видимостью каждого столбца.
  - Использовать фильтры компонентов с выражениями.
  - Изменять строки заголовка и нижнего колонтитула.
- Выполнять поиск компонентов в таблице.
- Выделять и выбирать экземпляры компонентов.
- Сохранять конфигурацию спецификации в виде файла шаблона.
- Загружать таблицу спецификации из файла шаблона.
- Вставлять таблицу спецификации в чертеж.
- Удалять таблицы спецификаций.

Для создания и редактирования спецификаций требуется лицензия Mechanical или Ultimate. Но Диспетчер спецификаций доступен и при использовании лицензии Pro.

### 21.4.10.2 Открытие Диспетчера спецификаций

Выполните одно из следующих действий:

- Щелкните правой кнопкой на панели инструментов и в открывшемся контекстном меню выберите **Панели > Диспетчер спецификаций**.
- Запустите команду *BMBOMPANELOPEN*.

### 21.4.10.3 Инструменты

Большинство операций в Диспетчере спецификаций доступны в главном меню. Для выбранных столбцов и строк доступны контекстные меню. В разделе свойств Диспетчера спецификации вы можете изменить свойства выбранных столбцов и спецификации в целом. Чтобы работать с определенным столбцом, выберите его в сетке спецификации. При необходимости, вы можете выбрать несколько столбцов одновременно.

**Основное меню**

Нажмите кнопку **Меню** (☰) в правом верхнем углу панели Диспетчер спецификации.

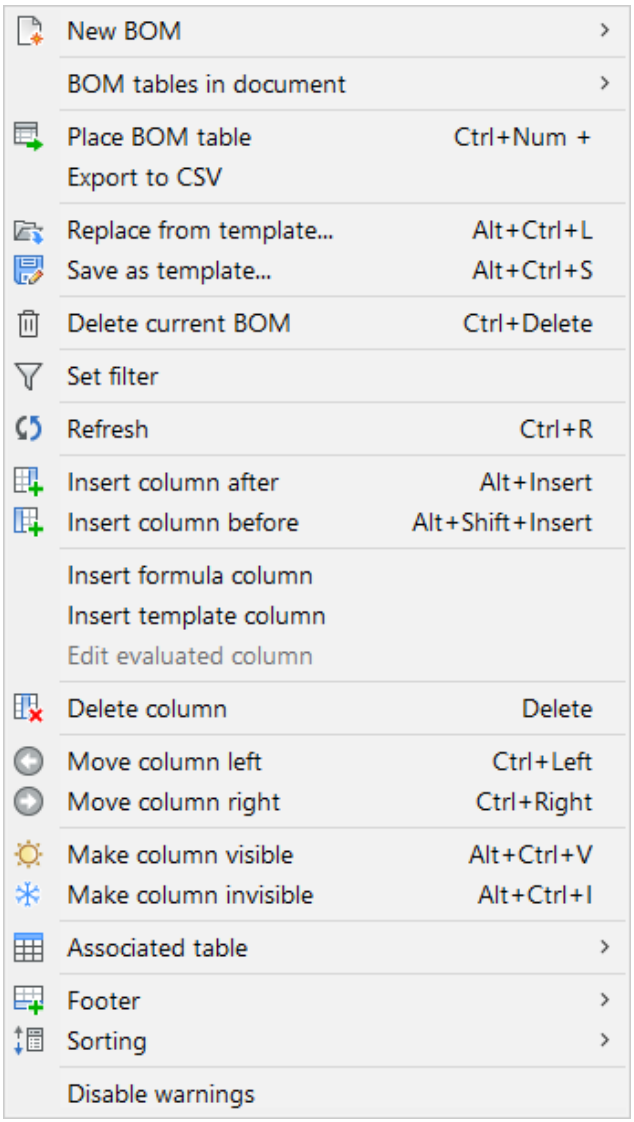



**Рисунок 36:** Основное меню

**Таблица 15: Основное меню**

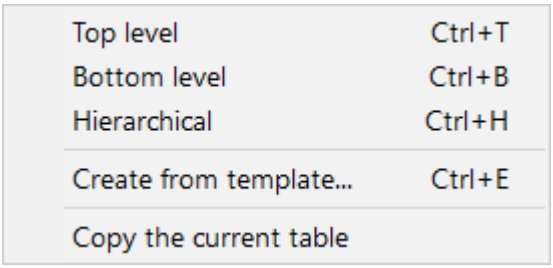
| Значок | Инструмент | Description |
|---|---|---|
|  | **Создать спецификацию** | Открывает подменю:<br><br>**Рисунок 37:** Подменю<br>• **Крупноузловая**: создает новую крупноузловую спецификацию.<br>• **Детальная**: создает новую детальную спецификацию.<br>• **Иерархическая**: создает новую иерархическую спецификацию.<br>• **Создать из шаблона**: открывает диалоговое окно для выбора файла шаблона конфигурации спецификации.<br>• **Копировать текущую таблицу**: создает новую спецификацию на основе текущей таблицы. |

**Таблица 15: Основное меню**

| Значок | Инструмент | Description |
|---|---|---|
| | **Спецификации в документе** | Открывает подменю:<br><br>**Рисунок 38:** Подменю<br>• **Перейти к первой таблице**: переход к первой таблице спецификации в документе.<br>• **Перейти к предыдущей таблице**: переход к предыдущей таблице спецификации в документе.<br>• **Перейти к следующей таблице**: переход к следующей таблице спецификации в документе.<br>• **Перейти к последней таблице**: переход к последней таблице спецификации в документе.<br>• **<Имя спецификации>**: переключает на соответствующую таблицу спецификации в документе. |
|  | **Вставить/обновить таблицу спецификации** | Запускает интерактивное размещение таблицы в чертеже. Если таблица уже размещена на чертеже, она будет обновлена.<br>***Прим.:*** Все операции редактирования автоматически обновляют связанную таблицу на чертеже. |

**Таблица 15: Основное меню**

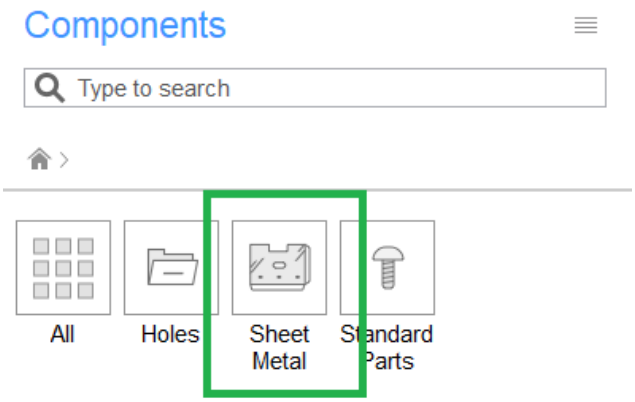
| Значок | Инструмент | Description |
|---|---|---|
| | **Экспорт в CSV** | Сохраняет содержимое текущей таблицы в .csv файл.<br>Открывает диалоговое окно для выбора местоположения .csv файла:<br><br>**Рисунок 39:** Выберите файл для экспорта таблицы спецификации<br>***Прим.:*** Для настройки разделителя значений CSV используйте свойства таблицы. |
| | **Заменить из шаблона** | Заменяет текущую конфигурацию таблицы спецификации выбранным шаблоном.<br>Открывает диалоговое окно выбора файла шаблона. |
| | **Сохранить как шаблон** | Сохраняет конфигурацию текущей таблицы спецификации как шаблон.<br>Открывает диалоговое окно выбора места расположения нового файла шаблона. |
| | **Удалить текущую спецификацию** | Удаляет текущую таблицу спецификации из документа. |

**Таблица 15: Основное меню**

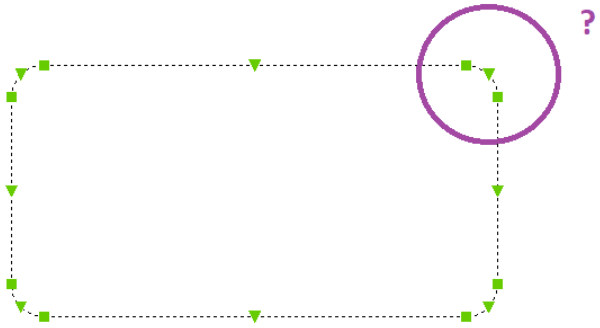
| Значок | Инструмент | Description |
|---|---|---|
| | **Установить фильтр** | Открывает диалоговое окно настройки основного фильтра:<br><br>**Рисунок 40:** Конфигуратор фильтра |
| | **Обновить** | Обновляет текущую спецификацию и пересчитывает все таблицы спецификаций на чертеже. |
| | **Вставить столбец после** | Если выбран столбец: вставляет столбец из **Списка столбцов** после выбранного столбца таблицы.<br>Если столбец не выбран: вставляет столбец после последнего столбца таблицы. |
| | **Вставить столбец перед** | Если выбран столбец: вставляет столбец из **Списка столбцов** перед выбранным столбцом таблицы.<br>Если столбец не выбран: вставляет столбец как первый столбец таблицы. |

**Таблица 15: Основное меню**

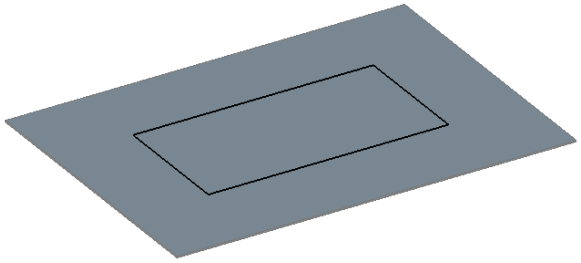
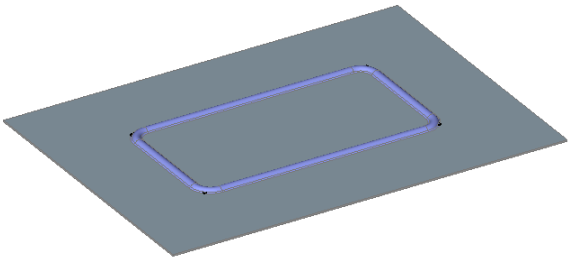
| Значок | Инструмент | Description |
|---|---|---|
| | **Вставить столбец формулы** | Открывает диалоговое окно для настройки столбца формулы:<br><br>**Рисунок 41:** Столбец формул<br>Нажмите кнопку **OK**, чтобы добавить новый столбец формулы после выбранного. Если столбец не выбран, новый столбец формулы добавляется после последнего столбца в таблице. |
| | **Вставить столбец шаблона** | Открывает диалоговое окно для настройки столбца шаблона:<br><br>**Рисунок 42:** Столбец шаблона<br>Нажмите **OK**, чтобы добавить новый столбец шаблона после выбранного. Если столбец не выбран, новый столбец шаблона добавляется после последнего столбца в таблице. |
| | **Изменить расчетный столбец** | Открывает диалоговое окно для редактирования выбранной формулы или столбца шаблона. |

**Таблица 15: Основное меню**

| Значок | Инструмент | Description |
|---|---|---|
|  | **Удалить столбец** | Удаляет выбранный столбец из таблицы.<br>Вы также можете удалить столбец, выбрав его в **сетке спецификации** и нажав клавишу **Удалить**. |
|  | **Переместить столбец влево** | Перемещает выбранные столбцы на один столбец влево. |
|  | **Переместить столбец вправо** | Перемещает выбранные столбцы на один столбец вправо. |
|  | **Сделать столбец видимым** | Показывает выбранный столбец в связанной таблице на чертеже. |
|  | **Сделать столбец невидимым** | Удаляет выбранный столбец из связанной таблицы. Невидимые столбцы остаются видимыми в Диспетчере спецификаций, но выделяются диагональной штриховкой фона. |

**Таблица 15: Основное меню**

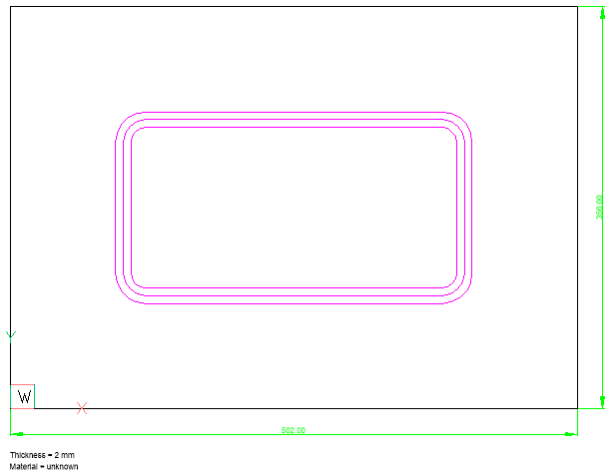
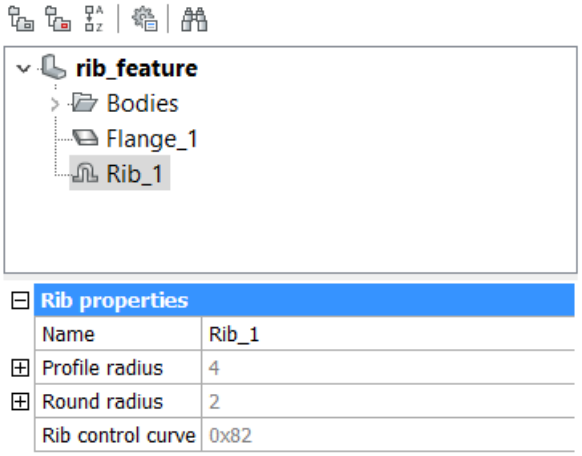
| Значок | Инструмент | Description |
|---|---|---|
| | **Связанная таблица** | Открывает подменю:<br><br>**Рисунок 43:** Подменю<br>• **Показать в модели**: отображает таблицу в модели. При необходимости, установите лист таблицы текущим.<br>• **Выбрать**: выбирает таблицу. При необходимости, установите лист таблицы текущим.<br>• **Удалить**: удаляет таблицу из документа.<br>• **Отключить**: отключает таблицу. После отключения таблицы в документе она не будет обновляться при изменении таблицы спецификации. |
| | **Нижний колонтитул** | Открывает подменю:<br>**Рисунок 44:** Подменю<br>Для каждого из выбранных столбцов:<br>• **Без функции**: удаляет нижний колонтитул.<br>• **Сумма**: отображает в нижнем колонтитуле сумму.<br>• **Среднее значение**: отображает в нижнем колонтитуле среднее значение.<br>• **Минимальное значение**: отображает в нижнем колонтитуле минимальное значение.<br>• **Максимальное значение**: отображает в нижнем колонтитуле максимальное значение.<br>Дополнительные сведения см. в разделе Нижний колонтитул Спецификация 1641. |

**Таблица 15: Основное меню**

| Значок | Инструмент | Description |
|---|---|---|
|  | **Сортировка** | Открывает подменю:<br><br><br>**Рисунок 45:** Подменю<br>• **Автоматически**: сортировка в таблице выполняется автоматически.<br>• **Исходный порядок**: сортировка в соответствии с исходным порядком экземпляров компонентов в документе.<br>• **По возрастанию**: сортировка выбранных столбцов в порядке возрастания.<br>• **По убыванию**: сортировка выбранных столбцов в порядке убывания.<br>Дополнительные сведения см. в разделе Сортировка Спецификация 1641. |
| | **Включить/выключить уведомления** | Управляет отображением предупреждающих сообщений во время создания или обновления таблиц спецификации. |

**Панели инструментов**

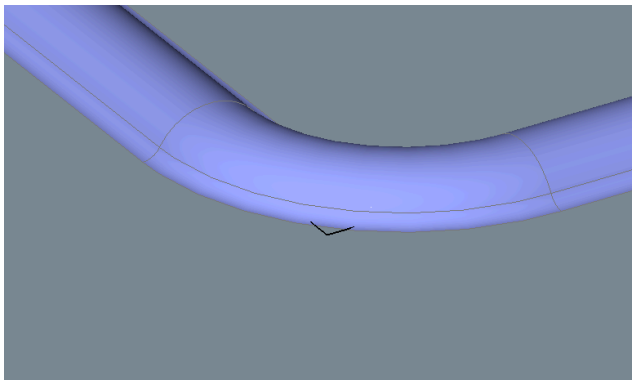


**Рисунок 46:** Диспетчер спецификаций

**Таблица 16: Диспетчер спецификаций**

| Значок | Инструмент | Description |
|---|---|---|
| | **Основное меню** | Открывает основное меню. |
| | **Список доступных спецификаций** | Позволяет выбрать текущую таблицу спецификации. Информация о выборе текущей таблицы сохраняется в чертеже. При закрытии и повторном открытии чертежа в Диспетчере спецификаций отображается выбранная ранее спецификация. Выбранная таблица обновляется до текущего состояния чертежа. |
| | **Обновить** | Обновляет как текущую спецификацию, так и список всех таблиц спецификаций на чертеже. |
| | **Вставить/ обновить таблицу спецификации** | Запускает интерактивное размещение таблицы на чертеже.<br>Обновляет таблицу, если она уже существует.<br>***Прим.:*** Все операции редактирования автоматически обновляют связанную таблицу на чертеже. |
| | **Список категорий свойств** | Позволяет выбрать категорию свойств. |

**Таблица 16: Диспетчер спецификаций**

| Значок | Инструмент | Description |
|---|---|---|
|  | **Список столбцов** | Позволяет выбрать столбец для добавления в текущую спецификацию.<br>Вы также можете использовать этот элемент при вводе формул.<br>Чтобы добавить формулу, введите выражение, начинающееся с "=", например "=<Mass>* <Quantity>". Формула может ссылаться на другие столбцы. Используйте их отображаемые имена, заключенные в <>. |
|  | **Вставить столбец после** | Если столбец выбран: вставляет столбец из **списка столбцов** после выбранного столбца в таблице.<br>Если столбцы не выбраны: новый столбец добавляется после последнего столбца в таблице. |
|  | **Поиск** | Включает/выключает поле поиска.<br>Для выбора предыдущего совпадения вы также можете использовать сочетание клавиш **Ctrl + F3**. |
| | **Поле поиска** | Ввод строки для поиска; поддерживаются специальные символы. |
| | **Найти предыдущий** | Выбирает предыдущее совпадение со строкой для поиска в таблице спецификации.<br>Для выбора предыдущего совпадения вы также можете использовать сочетание клавиш **Ctrl + F3**. |
| | **Найти следующее** | Выбирает следующее совпадение со строкой поиска в таблице спецификации.<br>Для выбора следующего совпадения вы также можете использовать клавишу **F3**. |
|  | **Выбрать все найденные элементы** | Выбирает все совпадения со строкой поиска в таблице спецификации. |

**Контекстное меню столбца**

Контекстное меню столбца открывается щелчком правой кнопки мыши на заголовке этого столбце. Столбец добавляется к текущему набору выбранных столбцов.

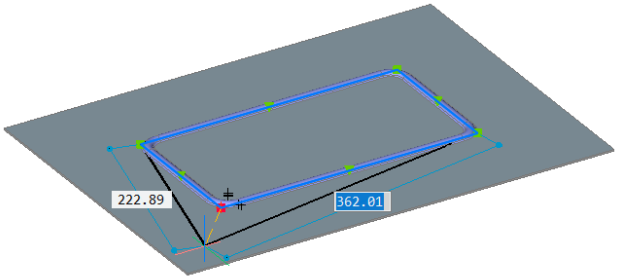



**Рисунок 47:** Контекстное меню столбца

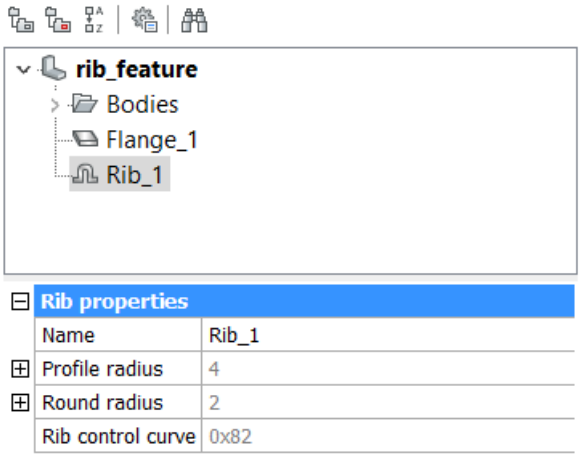
**Таблица 17: Контекстное меню столбца**

| Значок | Инструмент | Description |
|---|---|---|
| | **Добавить столбец шаблона из выбранного** | Открывает диалоговое окно настройки нового столбца шаблона. Выбранные столбцы будут автоматически добавлены в выражение шаблона. |
|  | **Удалить столбец** | Удаляет выбранные столбцы из таблицы.<br>Вы также можете выбрать столбец в **сетке спецификации** и нажать клавишу **Delete**. |
|  | **Нижний колонтитул** | Открывает подменю для выбранных столбцов:<br>• **Без функции**: удаляет нижний колонтитул.<br>• **Сумма**: отображает в нижнем колонтитуле сумму.<br>• **Среднее значение**: отображает в нижнем колонтитуле среднее значение.<br>• **Минимальное значение**: отображает в нижнем колонтитуле минимальное значение.<br>• **Максимальное значение**: отображает в нижнем колонтитуле максимальное значение. |
|  | **Сортировка** | Открывает подменю:<br>• **Автоматически**: сортировка в таблице выполняется автоматически**.**<br>• **Исходный порядок**: сортировка в соответствии с исходным порядком экземпляров компонентов в документе.<br>• **По возрастанию**: сортировка выбранных столбцов в порядке возрастания.<br>• **По убыванию**: сортировка выбранных столбцов в порядке убывания. |

**Таблица 17: Контекстное меню столбца**

| Значок | Инструмент | Description |
|---|---|---|
| | **Агрегатная функция** | Открывает подменю для выбранных столбцов:<br>• **Нет функции**: удаляет агрегатную функцию.<br>• **Сумма**: отображает сумму всех значений в группе.<br>• **Среднее значение**: отображает среднее значение всех значений в группе.<br>• **Минимальное значение**: отображает минимальное значение в группе.<br>• **Максимальное значение**: отображает максимальное значение в группе.<br>• **Объединение**: объединенное отображение всех значений в группе в порядке возрастания с разделением указанным разделителем (по умолчанию ';').<br>• **Объединение со счетчиком**: объединенное отображение всех значений в группе с соответствующим им количеством в порядке возрастания. Пары значений и количества разделяются указанным разделителем (по умолчанию ';'). |
| | **Назначение столбца** | Открывает подменю для выбранных столбцов:<br>• **Нумерация**: столбец будет использоваться в качестве источника нумерации при выполнении команды *BMBALLOON*.<br>• **Имя**: столбец будет использоваться в качестве источника имен при выполнении команды *BMBALLOON*.<br>• **Количество**: столбец будет использоваться в качестве источника данных о количестве при выполнении команды *BMBALLOON*.<br>• **Обычный**: столбец не имеет специального назначения. |
| %.. | **Формат** | Позволяет установить формат для выбранных столбцов.<br>Открывает диалоговое окно:<br><br>**Рисунок 48:** Формат |

**Таблица 17: Контекстное меню столбца**

| Значок | Инструмент | Description |
|---|---|---|
| | **Установить фильтр столбца** | Открывает диалоговое окно для настройки фильтра столбца:<br><br>**Рисунок 49:** Фильтр столбца |
| | **Сделать столбец видимым** | Отображает выбранные столбцы в связанной таблице на чертеже. |
| | **Сделать столбец невидимым** | Удаляет выбранный столбец из связанной таблицы. Невидимые столбцы остаются видимыми в Диспетчере спецификаций, но выделяются диагональной штриховкой фона. |
| | **Подогнать столбцы** | Подгоняет ширину выбранных столбцов по их содержимому.<br>Ширина столбца автоматически устанавливается по содержимому, если не была изменена вручную. |
| | **Подогнать все столбцы** | Подгоняет ширину всех столбцов по их содержимому.<br>Ширина столбца автоматически устанавливается по содержимому, если не была изменена вручную. |

### Контекстное меню строки

Контекстное меню строки открывается щелчком на ячейке.

**Рисунок 50:** Контекстное меню

**Таблица 18: Контекстное меню строки**

| Значок | Инструмент | Description |
|---|---|---|
|  | **Показать в модели** | Масштабирует и отображает выбранные детали в модели. |
|  | **Выбрать** | Выбирает детали. Заменяет существующий выбор. |
|  | **Добавить в набор** | Добавляет детали в набор выбранных деталей. |
|  | **Удалить из набора** | Удаляет детали из набора выбранных деталей. |
| | **Удалить выделение** | Удаляет выделение. |
| | **Очистить выбор** | Очищает набор выбранных деталей. |
| | **Свернуть** | Сворачивает группу подсборки в сетке спецификации. |
| | **Раскрыть** | Раскрывает группу подсборки в сетке спецификации. |
| | **Раскрыть этот уровень** | Раскрывает все группы подсборок на текущем уровне дерева сборки. |
| | **Свернуть этот уровень** | Сворачивает все группы подсборок на текущем уровне дерева сборки. |
| | **Раскрыть все** | Раскрывает все группы подсборки в текущей спецификации. |
| | **Свернуть все** | Сворачивает все группы подсборки в текущей спецификации. |
| | **Ограничить этим уровнем** | Устанавливает текущий уровень как максимальный уровень для иерархических таблиц спецификации. |
| | **Показать все уровни** | Удаляет ограничение максимального уровня для иерархических таблиц спецификации. |

**Заголовок спецификации**

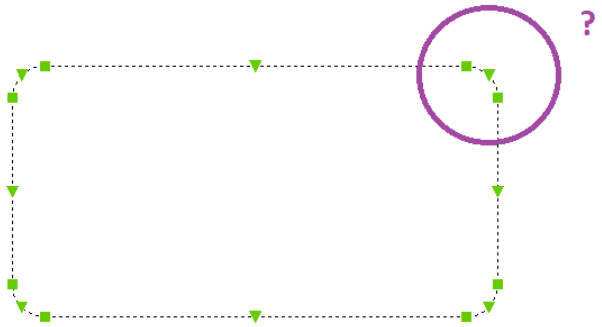



**Рисунок 51:** Заголовок спецификации

Отображает заголовок текущей спецификации так, как он отображается на чертеже. Щелкните левой кнопкой мыши по этому элементу управления, чтобы отредактировать чертеж. В этом режиме вместо **заголовка спецификации** отображается **шаблон заголовка**. **Шаблон заголовка** - это строка, которая может содержать заполнитель *<ИМЯ>* заменяемый именем сборки.

**Сетка**

| No. | Component ▲₂ | Physical material | Material density | Is standard | Quantity ▼₁ |
|---|---|---|---|---|---|
| 1 | ISO 4033 | | | Yes | 60 |
| 2 | MM-THD | | | Yes | 30 |
| 3 | LEG | | | No | 4 |
| 4 | FLANGE | | | No | 2 |
| 5 | CONNECTION H | ASTM-A276-316L | 7900.00 | No | 1 |
| 6 | CONNECTION J | ASTM-A276-316L | 7900.00 | No | 1 |
| 7 | GASKET | | | No | 1 |
| 8 | HAND HOLE | | | No | 1 |
| 9 | HIGHT SPEED MIXER | | | No | 1 |
| 10 | HOMOGINIZER | | | No | 1 |
| 11 | MIXER_A | | | No | 1 |
| 12 | NOZZLE B | | | No | 1 |
| 13 | NOZZLE D | | | No | 1 |
| 14 | NOZZLE F | AISI-316L | 7900.00 | No | 1 |
| 15 | NOZZLE L | | | No | 1 |

**Рисунок 52:** Сетка

В сетке отображается содержимое выбранной таблицы спецификации. Каждая строка связана с соответствующими экземплярами. При выборе строк соответствующие им детали выделяются на чертеже. С помощью контекстного меню строки вы можете выбрать соответствующие детали, показать их в модели, добавить или удалить их из текущего набора. Чтобы показать деталь в модели, дважды щелкните на любой ячейке соответствующей этой детали строки.

Перемещение столбца

Чтобы переместить столбец в таблице спецификации, перетяните его на новое место.

Видимые и невидимые столбцы



**Рисунок 53:** Видимые и невидимые столбцы

Вы можете сделать любой столбец в таблице спецификации невидимым, чтобы он не отображался в итоговой таблице. Невидимые столбцы остаются видимыми в Диспетчере спецификаций, но выделяются диагональной штриховкой фона. Вы можете изменить видимость столбцов в главном меню, контекстном меню столбца или изменив его свойство **Видимость** в **сетке свойств**.

Настройка ширины

Вы можете настроить ширину любого столбца в сетке спецификации, который не зависит от ширины в связанной таблице. Если ширина столбца меньше, чем необходимо для отображения текста ячейки, высота соответствующей строки увеличивается в соответствии с размером текста.

Ширина столбца, установленная вручную, остается неизменной независимо от содержимого таблицы. Ширина столбца также сохраняется в конфигурации спецификации. При закрытии и повторном открытии чертежа все настроенные значения ширины столбцов остаются неизменными.

Для автоматической подгонки ширины столбца по содержимому выполните одно из следующих действий:

- Выберите столбцы, щелкните правой кнопкой мыши и в открывшемся контекстном меню выберите **Подогнать столбцы**.
- Дважды щелкните на разделителе заголовка столбца.

Сортировка спецификации

Для управления сортировкой содержимого используется сетка. Дважды щелкните на заголовке столбца, чтобы изменить его режим сортировки, переключаясь между сортировкой по возрастанию, убыванию и исходным порядком. Если к столбцу таблицы была применена сортировка, к его заголовку добавляется символ сортировки (▲ для возрастания и ? для убывания). Двойной щелчок на заголовке изменяет порядок сортировки.

Для сортировки могут быть выбраны несколько столбцов. Символы сортировки для таких столбцов имеют дополнительный числовой индекс. Например, если вы хотите отсортировать по количеству в порядке убывания, а затем по имени компонента в порядке возрастания, столбец Количество будет иметь символ сортировки ??, а столбец Компонент - ▲?.

Раскрытие/сворачивание групп

**Рисунок 54:** Раскрытие/сворачивание групп

В иерархических спецификациях каждая подсборка образует группу, которая может быть свернута или раскрыта в сетке спецификации.

Чтобы свернуть или раскрыть группу, выполните одно из следующих действий:

- Щелкните на значке узла в первом столбце сетки спецификации.
- Щелкните на строке правой кнопкой мыши и в открывшемся контекстном меню выберите **Свернуть** или **Раскрыть**.

Используя контекстное меню, вы также можете свернуть или раскрыть все группы или все группы, соответствующие одному уровню сборки.

***Прим.:*** Статус группы узлов сборки (свернутая или развернутая) в документе не сохраняется. При закрытии и повторном открытии таблицы спецификации все группы узлов сборки по умолчанию раскрываются.

**Свойства**

**Рисунок 55:** Свойства спецификации

Сетка свойств расположена в нижней части панели Диспетчер спецификаций. Чтобы настроить ее размер, используйте разделитель, расположенный между сеткой спецификации и сеткой свойств.

Сетка свойств содержит свойства текущей таблицы; заголовок, тип, фильтр, режим сортировки и заголовок нижнего колонтитула. Если выбраны какие-либо столбцы, сетка также отобразит их общие свойства. Вы можете изменить любое общее свойство сразу для всех выбранных столбцов. Свойства столбца позволяют изменить его заголовок, видимость и ширину.

Для числовых значений вы можете установить тип нижнего колонтитула, положение и режим единиц измерения (если в столбце есть единицы, которые можно изменить).

**Положение единиц** определяет, где будут отображаться единицы измерения: в заголовке столбца, в той же ячейке, что и само значение, или в дополнительном столбце.

**Режим единиц** определяет, как должны быть получены отображаемые единицы; лучшая единица для всех значений, лучшая единица для отображения определенного значения в ячейке или единица, указанная пользователем. Вы можете выбрать **Режим единиц** в свойствах столбцов Расстояние, Площадь, Объем или Масса.

### 21.4.10.4 Фильтр

В текущую спецификацию можно добавить настраиваемый фильтр. Чтобы применить фильтр, выполните одно из следующих действий:

- Используйте свойство **Фильтр** в разделе **Свойства спецификации сетки свойств**.
- Используйте команду **Установить фильтр**, доступную в **основном меню**.

Откроется диалоговое окно *Конфигуратор выражений*:

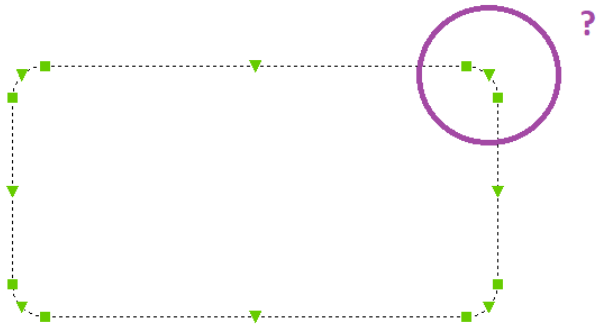



**Рисунок 56:** Конфигуратор фильтра

Каждая строка в этом диалоговом окне представляет собой элементарное условие. Вы также можете добавлять группы, чтобы сгруппировать несколько элементарных условий и/или групп. Группы позволяют создавать сложные фильтры.

**Добавление нового элементарного условия**

Выполните следующие действия:

1 При необходимости, выберите категорию, чтобы ограничить список свойств этой категорией.

2 Выберите необходимое свойство из списка свойств. При выборе свойства условие для этого свойства добавляется автоматически.

3 При необходимости, задайте необходимый тип условия:
- Не равно
- Меньше, чем
- Больше, чем
- Не меньше, чем
- Не больше, чем

4 Установите целевое значение. Для текстовых значений целевое значение можно выбрать из списка всех возможных значений. Для числовых значений вы можете указать единицу целевого значения. Значение будет автоматически преобразовано в единицу измерения документа.

**Удаление одного или нескольких условий**

Выполните следующие действия:

1 Выберите условия для удаления. Для выбора нескольких условий нажмите клавишу Ctrl. Для выбора условий в указанном диапазоне используйте клавишу Shift.

2 Нажмите кнопку **Удалить** (✕)

или щелкните правой кнопкой мыши на строке, которую необходимо удалить, и в открывшемся контекстном меню выберите **Удалить условие**

.

**Добавление новой группы условий**

Выполните следующие действия:

1 Нажмите кнопку **Добавить** (✚).

2 Выберите **Добавить группу «Все из»**, чтобы добавить новую группу «Все из» или **Добавить группу «Любой из»**, чтобы добавить новую группу «Любой из».

3 Добавьте необходимые условия.

**Группировка выбранных условий**

Выполните следующие действия:

1 Выберите условия для удаления. Вы можете использовать клавишу Ctrl, чтобы выбрать несколько условий, и клавишу Shift, чтобы выбрать условия в указанном диапазоне.

2 Щелкните правой кнопкой мыши на строке, которую необходимо сгруппировать.

3 Выберите **Создать группу «Все из»**, чтобы сформировать новую группу «Все из» или **Создать группу «Любой из»**, чтобы сформировать новую группу «Любой из».

Чтобы применить фильтр столбца, нажмите кнопку **ОК**.

### 21.4.10.5 Фильтры столбцов

Фильтры столбцов позволяют вам установить условия, при которых строки должны быть включены в таблицу. Каждый фильтр столбца задает условие, которому должны соответствовать значения в столбцах для их включения. Например, вы можете настроить фильтр столбца, чтобы он включал только детали, количество которых превышает определенное число, или детали, имя которой содержит текст **Сопло**. В каждом столбце может быть только один фильтр. Но вы можете добавить фильтры столбцов к любому количеству столбцов. Столбцы с активными фильтрами отмечены значком фильтра ().

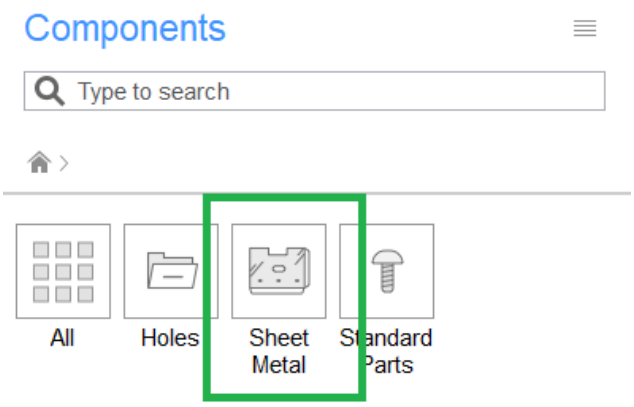



**Рисунок 57:** Пример столбцов с активными фильтрами

Чтобы добавить фильтр столбца, щелкните правой кнопкой мыши на заголовке столбца и в открывшемся контекстном меню выберите команду **Установить фильтр столбца**.

Откроется диалоговое *Фильтр столбца*.

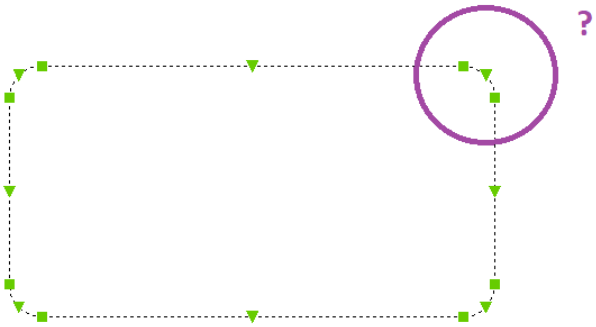



**Рисунок 58:** Фильтр столбца

Выберите необходимый тип фильтра. Затем вам необходимо будет установить целевые значения для фильтра в зависимости от типа фильтра и содержимого столбца.

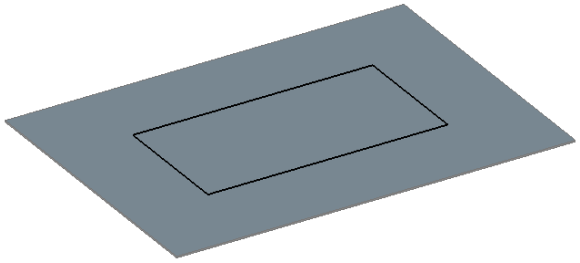


**Рисунок 59:** Целевые значения для фильтра

Для столбцов с числовым содержимым вы можете установить единицу измерения целевого значения. Целевое значение будет автоматически преобразовано в единицу измерения документа.

Для текстовых фильтров вы можете выбрать между опциями **Не учитывать регистр** и **Учитывать регистр**. Фильтры **В** и **Не в** отображают список всех возможных значений для этого столбца (в отношении уже примененных фильтров). Выберите значения, которые вы хотите включить или исключить соответственно. Вы можете выбрать все значения, снять отметку выбора со всех значений или инвертировать текущий выбор, используя соответствующие кнопки на панели инструментов или команды из контекстного меню.

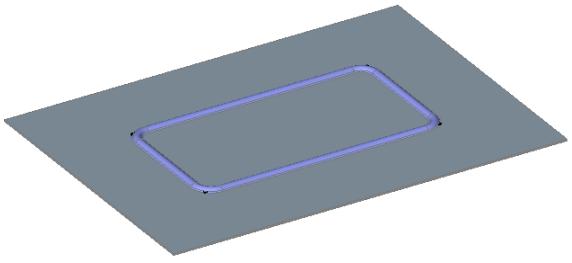


**Рисунок 60:** Фильтр столбца

Чтобы применить фильтр столбца, нажмите кнопку **ОК**.

Чтобы удалить фильтр столбца, щелкните правой кнопкой мыши на заголовке столбца и в открывшемся контекстном меню выберите команду **Удалить фильтр столбца**.

Дополнительные сведения см. раздел Использование фильтра в Спецификации.

### 21.4.10.6 Сортировка

Таблицы спецификации поддерживают несколько режимов сортировки (см. раздел Сортировка в Спецификации).

Для переключения между этими режимами используйте свойство **Режим сортировки** в разделе **Свойства спецификации** в **сетке свойств**. Используйте раскрывающийся список **Режим сортировки** или дважды щелкните на заголовке столбца в **сетке спецификации** для переключения между режимами сортировки По возрастанию, По убыванию или Исходный порядок для соответствующего столбца.

### 21.4.10.7 Единицы измерения столбца

Вы можете изменить **положение** и **режим** единиц измерения для большинства числовых свойств с единицами измерения. Выберите необходимые столбцы в **сетке** и используйте раздел **Свойства столбца сетки свойств**, чтобы изменить **положение** и **режим** единиц измерения.

Дополнительные сведения см. раздел Единицы столбцов в Спецификации.

См. также: Изменение единиц измерения и режимов размещения единиц измерения столбца.

### 21.4.10.8 Расчетные столбцы

Спецификации поддерживают расчетные столбцы. Доступны два типа расчетных столбцов: столбцы формул и столбцы шаблона. Подробнеее о полях формул и синтаксисе выражений см. раздел Расчетные столбцы в Спецификации.

См. также: Добавление расчетного столбца 1636.

**Столбцы формул**

Чтобы добавить столбец формул, выполните следующие действия:

1 Выберите категорию **Расчетные столбцы** в **списке категорий**.

2 Выберите **Столбец формул** в **списке столбцов**.

3 Используйте инструменты **Вставить столбец после** или **Вставить столбец перед**.

В качестве альтернативы вы можете использовать опцию **Вставить столбец формулы**, которая доступна в **основном меню**.

Откроется диалоговое окно *Конфигуратор выражений*:

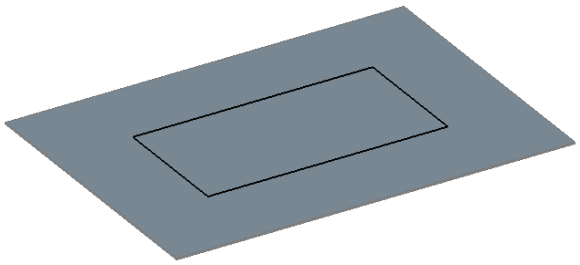


**Рисунок 61:** Конфигуратор выражений

В этом диалоговом окне вы можете задать заголовок, единицы измерения и выражение для столбца формулы и выполнить предварительный просмотр результатов применения формулы. Поле выражения поддерживает автозаполнение.

Чтобы добавить ссылку на столбец или свойство к выражению в позиции курсора, выберите необходимое свойство из списка свойств. Вы также можете выбрать инструмент **Добавить ссылку на столбец** (ABC) и указать свойство в меню или выбрать необходимую функцию с помощью инструмента **Добавить функцию** ( ).

Чтобы добавить столбец формулы в таблицу спецификации, нажмите кнопку **ОК**.

**Столбцы с шаблонами**

Чтобы добавить столбец шаблона, выполните следующие действия:

1 Выбрать *<Расчетные столбцы>* в **списке категорий**.

2 Выбрать *<Столбец шаблона>* в **списке столбцов**.

3 Используйте инструменты **Вставить столбец после** или **Вставить столбец перед**.

В качестве альтернативы вы можете использовать опцию **Вставить столбец шаблона**, которая доступна в **основном меню**.

Откроется диалоговое окно *Конфигуратор шаблона*.

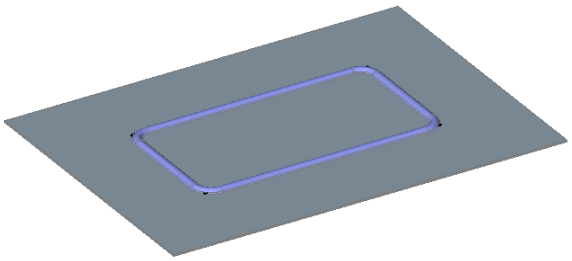


**Рисунок 62:** Конфигуратор шаблонов

В этом диалоговом окне вы можете задать заголовок, выражение и, при необходимости, единицу измерения для столбца вашего шаблона и выполнить предварительный просмотр результатов применения шаблона. Поле выражения поддерживает автозаполнение.

Столбец шаблона может содержать несколько выражений. Эти выражения будут вычисляться от первого до последнего, и будет использовано первое выражение, дающее непустой результат. **Конфигуратор шаблонов** позволяет добавлять, удалять и переключаться между выражениями шаблона с помощью панели инструментов. Вы также можете изменять порядок выражений шаблона.

***Прим.:*** В поле выражения одновременно отображаются все выражения, причем каждое выражение заключено в фигурные скобки {}.

Чтобы добавить выражение, нажмите кнопку .

Чтобы удалить текущее выражение, нажмите кнопку .

***Прим.:*** При удалении последнего выражения оно заменяется пустым выражением.

Чтобы изменить текущее выражение, выберите номер этого выражения из выпадающего списка.

Используя кнопки и , вы можете изменять порядок выражений.

Выбранное выражение можно настроить с помощью раздела **Шаблон**. Каждый столбец в этом разделе представляет собой элемент выражения шаблона, который вы можете изменить.

Типы элементов шаблона:

- **Ссылка на столбец**: представляет собой ссылку на столбец спецификации или свойство. Чтобы добавить ссылку на столбец, выберите необходимый столбец из списка свойств. Он будет автоматически добавлен после последнего выбранного столбца или, если столбец не выбран, после последнего столбца. При необходимости перед **ссылкой на столбец** будет автоматически добавлен **разделитель**.
- **Разделитель**: представляет собой разделитель между двумя элементами. Этот разделитель будет добавлен при выводе результата только в том случае, если с обеих сторон имеется текст. Например, если у вас есть ссылки на столбцы PartNumber и Length с разделителем «X» между ними, «X» будет добавлен к окончательному тексту только в том случае, если и PartNumber, и Length не являются пустыми. В большинстве случаев разделители добавляются автоматически. Но вы можете добавить разделители, нажав кнопку и выбрав в раскрывающемся списке опцию **Добавить разделитель**. Кроме того, вы можете добавить разделитель с помощью контекстного меню.
- **Текст**: представляет собой пользовательский текст. Вы можете добавить пользовательский текст, нажав кнопку и выбрав в раскрывающемся списке опцию **Добавить текст**. Кроме того, вы можете добавить пользовательский текст с помощью контекстного меню.
- **Расчетный текст**: представляет собой элемент, который должен быть добавлен к окончательному выражению. Расчетный текст может содержать функции или другие специальные символы, которые изменяют результат. Вы можете добавить расчетный текст, нажав кнопку и выбрав в раскрывающемся списке опцию **Добавить выражение**. Кроме того, вы можете добавить расчетный текст с помощью контекстного меню.

В любой момент вы можете изменить содержимое любого элемента в шаблоне. Вы также можете удалить элементы, выбрав их и нажав кнопку или выбрав опцию **Удалить столбец** в контекстном меню. Чтобы переместить элемент, выберите столбец в шаблоне и перетяните его в другое место. Любые изменения шаблона будут отражаться в поле выражения. Точно так же любые изменения в поле выражения будут отражаться в области шаблона.

Чтобы добавить столбец шаблона в таблицу спецификации, нажмите кнопку **ОК**.

### 21.4.10.9 Строка нижнего колонтитула

Таблица спецификации может содержать строку нижнего колонтитула. Чтобы добавить строку нижнего колонтитула, укажите тип нижнего колонтитула для одного или нескольких столбцов. По умолчанию используется заголовок строки нижнего колонтитула *ВСЕГО:*. Измените заголовок нижнего колонтитула с помощью свойства **Заголовок нижнего колонтитула** в разделе **Свойства спецификации** в **сетке свойств**.

Дополнительные сведения о строке нижнего колонтитула в спецификации см. раздел Нижний колонтитул в Спецификации.

См. также: Добавление нижнего колонтитула.

### 21.4.10.1 Агрегатные столбцы 0

В таблицах крупноузловых и детальных спецификаций почти любому столбцу может быть назначена агрегатная функция. Агрегатные функции группируют значения нескольких строк спецификации (которые отличаются только этими значениями) в одно значение. Например, агрегатные функции могут использоваться для вычисления общей длины труб с учетом их размера и толщины или для объединения всех размеров болтов и гаек, используемых в сборке, в одной ячейке.

Дополнительные сведения см. раздел Столбцы в Спецификации.

Чтобы задать агрегатную функцию, выполните одно из следующих действий:

- Щелкните правой кнопкой мыши на заголовке столбца и выберите необходимую агрегатную функцию в подменю **Агрегатная функция**.
- Выберите столбец и измените агрегатную функцию в соответствующем свойстве столбца.

Агрегатные столбцы отмечены символом ∑.

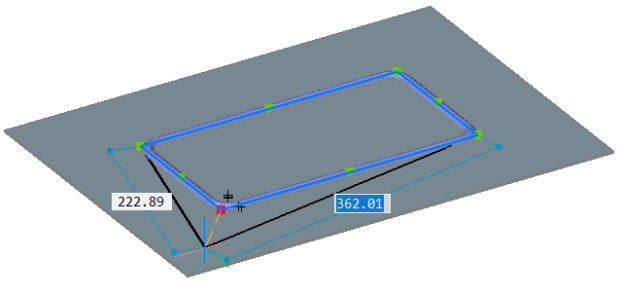



**Рисунок 63:** Пример агрегатных столбцов

### 21.4.10.1 Шаблоны

1

Вы можете сохранить конфигурации спецификации как шаблоны. Используйте Диспетчер спецификаций, чтобы создать новую таблицу из шаблона и заменить текущую конфигурацию таблицы таблицей из шаблона. Вы можете сохранить конфигурацию как шаблон в любой момент. Вы можете изменить загруженную конфигурацию шаблона с помощью обычных инструментов и сохранить ее как другой шаблон.

Дополнительные сведения о шаблонах спецификации см. раздел Шаблоны в Спецификации.

### 21.4.10.1 Поиск

2



**Рисунок 64:** Поиск

Диспетчер спецификаций включает поле поиска, которое облегчает поиск определенного значения в таблице. Поле поиска позволяет использовать специальные символы. По умолчанию BricsCAD выполняет поиск по всей таблице. Чтобы ограничить область поиска, выберите необходимые столбцы. Если значение найдено, курсор перемещается в эту ячейку. Если столбцы не выбраны, выбирается строка. **Сетка** автоматически прокручивается для отображения необходимой ячейки. Чтобы найти другое значение, используйте кнопку или . Вы также можете использовать для этого клавишу **F3** или **Ctrl + F3**. Чтобы выбрать все найденные результаты, нажмите кнопку .

По умолчанию выполняется поиск текста, введенного в поле поиска. Но вы также можете использовать **специальные символы**.

Поддерживаются следующие специальные символы:

**Таблица 19: Специальные символы**

| Имя | Символ | Description |
|---|---|---|
| Обратная кавычка | ` | Игнорирует следующий символ (воспринимает его буквально). |
| Запятая | , | Разделяет шаблоны |

**Таблица 19: Специальные символы**

| Имя | Символ | Description |
|---|---|---|
| Астериск | * | Соответствует любой последовательности символов, включая нулевую последовательность в любом месте. |
| Знак вопроса | ? | Соответствует одиночному символу. |
| Точка | . | Соответствует одиночному не буквенно-цифровому символу. |
| "Собака" | @ | Соответствует одиночному буквенному символу. |
| Решетка | # | Соответствует одиночному цифровому символу. |
| Тильда | ~ | Если первый символ в шаблоне, соответствует чему-либо, кроме шаблона. |
| Скобки | [...] | Соответствует одиночному закрытому символу. |
| Скобки с тильдой | [~...] | Соответствует одиночному незакрытому символу. |
| Дефис | - | В скобках соответствует одиночному символу в указанном диапазоне. |

Чтобы открыть поле поиска, нажмите кнопку **Поиск** на панели инструментов или используйте сочетание клавиш **Ctrl + F**.

### 21.4.10.1 Связанная таблица 3

Каждая таблица спецификации в диспетчере может иметь связанную (ассоциированную) таблицу, которая обновляется при изменении конфигурации спецификации. Таблица спецификации может иметь только одну связанную таблицу. Но вы можете создать таблицы спецификаций, которые не имеют связанной таблицы в документе.

Чтобы создать ассоциированную таблицу, используйте инструмент **Вставить/Обновить таблицу**. Если таблица спецификации была создана с помощью команды *BMBOM*, ассоциированная таблица создается автоматически.

Вы можете найти ассоциированную таблицу в документе, выбрав в основном меню Диспетчера спецификаций **Ассоциированная таблица > Показать в модели** или **Ассоциированная таблица > Выбрать.**При необходимости, выполняется переключение на лист ассоциированной таблицы.

Вы можете удалить ассоциированную таблицу, выбрав в основном меню Диспетчера спецификации **Ассоциированная таблица > Удалить.**Вы также можете отключить

ассоциированную таблицу от таблицы спецификации, выбрав в основном меню **Ассоциированная таблица > Отключить.**После отключения ассоциированная таблица теряет связь с таблицей спецификации и при ее изменении не обновляется.

### 21.4.10.1 Обновление спецификации 4

Начиная с BricsCAD V20.2, Диспетчер спецификаций автоматически обновляет содержимое таблицы при изменении или обновлении сборки. Для больших сборок это может быть неудобно. Диспетчер спецификаций автоматически обновляет таблицу, когда он активен.

Чтобы избежать автоматического обновления, закройте Диспетчер спецификаций, чтобы сделать панель Диспетчер спецификаций неактивной.

Для ручного обновления содержимого Диспетчера спецификаций используйте инструмент **Обновить**.

***Прим.:*** При изменении или модификации сборки ассоциированная таблица автоматически не обновляется.

Для обновления ассоциированной таблицы используйте инструмент **Вставить/обновить таблицу**.

### 21.4.10.1 Создание новой спецификации 5

1 Используйте подменю **Создать спецификацию** в **основном меню**.

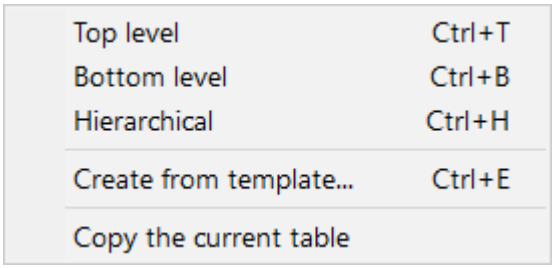



**Рисунок 65:** Новое подменю спецификации

2 Укажите тип спецификации или выберите опцию **Из шаблона** для выбора файла шаблона спецификации.

### 21.4.10.1 Выбор текущей спецификации 6

Выберите существующую спецификацию в **списке доступных спецификаций**.

***Прим.:*** Таблица автоматически обновится до текущего состояния сборки и связанной таблицы на чертеже (если она существует).

### 21.4.10.1 Изменение типа текущей спецификации 7

Измените свойство **Тип** в разделе **Свойства спецификации** в **сетке свойств**.

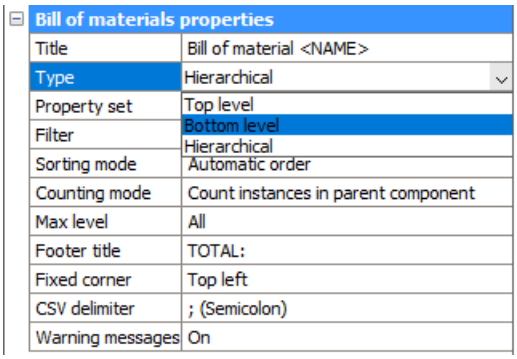



**Рисунок 66:** Свойства спецификации

### 21.4.10.1 Изменение набора свойств текущей спецификации 8

Измените свойство **Набор свойств** в разделе **Свойства спецификации** в **сетке свойств**.

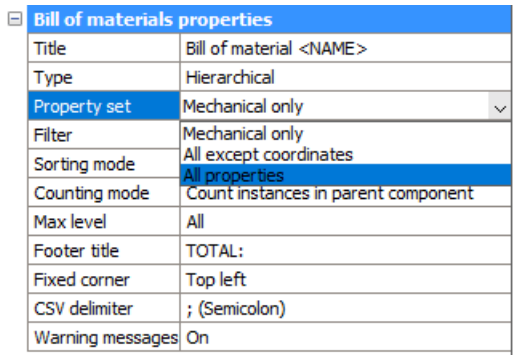



**Рисунок 67:** Все свойства

### 21.4.10.1 Изменение режима подсчета иерархической спецификации 9

Измените свойство **Режим подсчета** в разделе **Свойства спецификации** в **сетке свойств**.

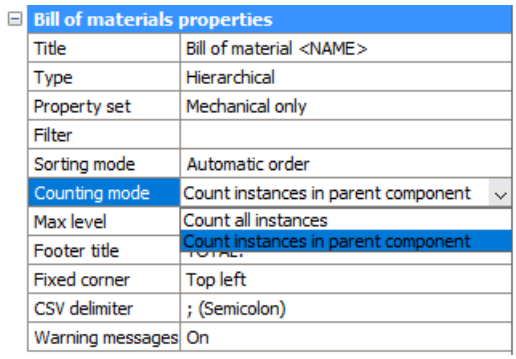



**Рисунок 68:** Свойства спецификации

### 21.4.10.2 Изменение режима группировки крупноузловой и детальной спецификации 0

Измените свойство **Режим группировки** в разделе **Свойства спецификации** в **сетке свойств**.

**Рисунок 69:** Свойства спецификации

### 21.4.10.2 Настройка пользовательского фильтра для текущей спецификации 1

1 Выберите команду **Установить фильтр** в **основном меню**.

Откроется диалоговое окно *Конфигуратор фильтра*.

2 Настройте фильтр с помощью диалогового окна.

Дополнительные сведения см. Фильтр 1623 .

3 Нажмите кнопку **OK**, чтобы применить фильтр.

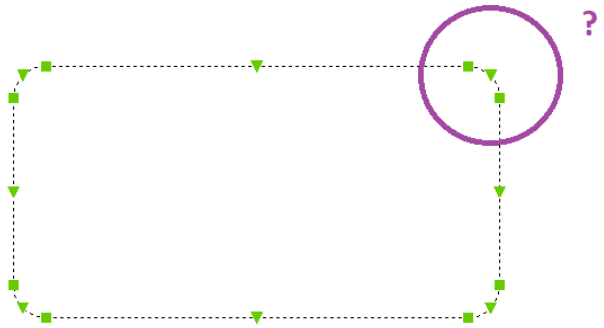



**Рисунок 70:** Конфигуратор фильтра

### 21.4.10.2 Добавление столбцов в текущую спецификацию 2

1 При необходимости, используйте **список категорий** для выбора категории добавляемых свойств.

**Рисунок 71:** Все столбцы

2 Выберите добавляемое свойство из списка доступных столбцов. Этот список содержит все столбцы выбранной категории, которые еще не были добавлены в спецификацию. Если вы начнете вводить имя поля, BricsCAD автоматически предложит вам список возможных имен. Для выбора имени щелкните на нем в предложенном списке.

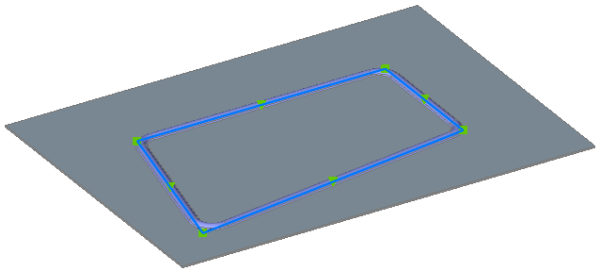


**Рисунок 72:** Описание компонента

3 При необходимости, выберите столбец спецификации, рядом с которым необходимо разместить новый столбец.

4 Нажмите кнопку **Вставить столбец после** (), чтобы разместить новый столбец после выбранного, или кнопку **Вставить столбец перед** (), чтобы разместить новый столбец перед выбранным.

Если столбец не выбран, новый столбец помещается соответственно в последнюю или первую позицию.

### 21.4.10.2 Добавление расчетного столбца 3

1 Выберите категорию **<Расчетные столбцы>** в **списке категорий**.

2 Выберите **<Столбец формулы>**, чтобы создать столбец формулы или **<Столбец шаблона>**, чтобы создать столбец шаблона.

3 Нажмите кнопку **Вставить столбец после** (), чтобы разместить новый столбец после выбранного, или кнопку **Вставить столбец перед** (), чтобы разместить новый столбец перед выбранным.

Если столбец не выбран, новый столбец будет добавлен как последний или первый столбец таблицы.

Откроется диалоговое окно настройки расчетного столбца.

4 Измените **заголовок** столбца.

5 При необходимости, измените**Единицы измерения**.

6 Задайте выражение с помощью инструментов диалогового окна.

7 Нажмите кнопку **ОК**, чтобы добавить столбец в таблицу спецификации.

### 21.4.10.2 Добавление столбца шаблона из выбранных столбцов 4

1 Выберите столбцы, которые необходимо объединить в столбец шаблона. Используйте клавишу Ctrl, чтобы выбрать несколько столбцов, или клавишу Shift, чтобы выбрать столбцы в указанном диапазоне.

2 Щелкните правой кнопкой мыши на заголовке одного из выбранных столбцов.

3 В открывшемся контекстном меню выберите **Добавить столбец шаблона из выбранного**.

Откроется диалоговое окно настройки расчетного столбца.

4 Измените **заголовок** столбца.

5 При необходимости, измените**Единицы измерения**.

6 Задайте выражение с помощью инструментов диалогового окна.

7 Нажмите кнопку **ОК**, чтобы добавить столбец в таблицу спецификации.

8 При необходимости, удалите или скройте столбцы, которые больше не нужны.

### 21.4.10.2 Изменение расчетного столбца
**5**

1 Щелкните правой кнопкой мыши на заголовке столбца в **сетке спецификации**.

2 В открывшемся контекстном меню выберите **Изменить выражение столбца**.

Откроется диалоговое окно настройки расчетного столбца.

3 При необходимости, измените заголовок столбца, единицы измерения и выражение.

4 Нажмите кнопку **ОК**, чтобы применить изменения.

### 21.4.10.2 Удаление столбца из текущей спецификации
**6**

1 Выберите столбец в сетке.

2 Нажмите кнопку **Удалить столбец** ()

или нажмите клавишу **Delete**.

### 21.4.10.2 Изменение заголовка спецификации
**7**

Используйте поле **Заголовок спецификации** для редактирования заголовка

или используйте свойство **Заголовок** в разделе **Свойства спецификации** в **сетке свойств**.

Вы можете использовать заполнитель *<ИМЯ>* для ссылки на имя сборки.

### 21.4.10.2 Изменение заголовка строки нижнего колонтитула
**8**

Используйте свойство **Заголовок нижнего колонтитула** в разделе **Свойства спецификации материалов** в **сетке свойств**.

### 21.4.10.2 Добавление префикса или суффикса нумерации
**9**

1 Выберите столбец **Нумерация**.

2 Выберите свойство **Префикс номера** в разделе **Свойства столбца**.

| Column properties | |
|---|---|
| Title | No. |
| Category | BOM Standard |
| Role | Number |
| Number prefix | A |
| Number suffix | |
| Number type | Full |
| Delimiter | . |
| Format | |
| Visible | Yes |
| Width | Auto |
| Filter | No filter |

**Рисунок 73:** Свойства столбца

3 При необходимости, введите новую строку префикса.

4 Выберите свойство **Суффикс номера** в разделе **Свойства столбца**.

5 При необходимости введите новую строку суффикса.

### 21.4.10.3 Изменение разделителя между номерами в иерархической спецификации 0

1 Выберите столбец **Нумерация**.

2 Выберите свойство **Разделитель** в разделе **Свойства столбца**.

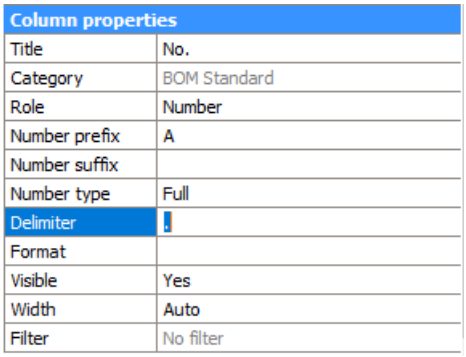



**Рисунок 74:** Свойства столбца

3 Введите новую строку-разделитель.

### 21.4.10.3 Изменение типа нумерации в иерархической спецификации 1

1 Выберите столбец **Нумерация**.

2 Выберите свойство **Тип нумерации** в разделе **Свойства столбца**.

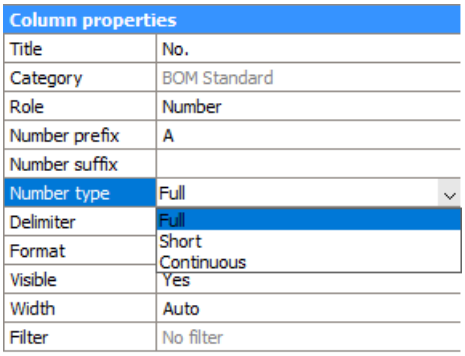



**Рисунок 75:** Свойства столбца

3 Выберите необходимый тип нумерации.

### 21.4.10.3 Изменение заголовка столбца 2

1 Выберите столбец в **сетке спецификации**.

2 Выберите свойство **Заголовок** в разделе **Свойства столбца**.

**Рисунок 76:** Свойства столбца

3 Измените значение свойства **Заголовок** на новый заголовок столбца.

### 21.4.10.3 Изменение формата столбца 3

1 Выберите столбцы.

2 Щелкните правой кнопкой мыши на заголовке выбранного столбца.

3 В открывшемся контекстном меню выберите **Формат**.

Откроется диалоговое окно *Формат*.

**Рисунок 77:** Формат

4 Задайте необходимый формат.

5 Нажмите кнопку **ОК**, чтобы применить изменения.

### 21.4.10.3 Изменение единиц измерения и режимов размещения единиц измерения 4 столбца

1 Выберите столбцы.

2 При необходимости, измените свойство **Положение единиц** в разделе **Свойства столбца**.

3 При необходимости, измените свойство **Режим единиц** в разделе **Свойства столбца**.

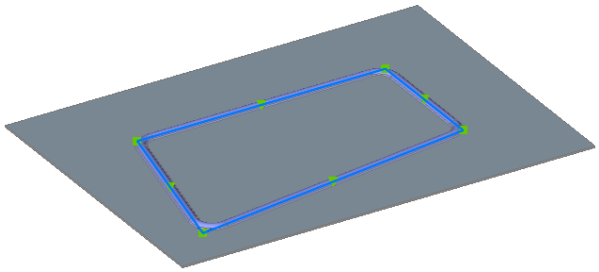


**Рисунок 78:** Свойства столбца

4 При необходимости, измените **Формат заголовка**.

- Используйте *<Заголовок>* для добавления заголовка столбца.
- Используйте *<Единица>* для добавления единицы столбца.
- Используйте *\n* для добавления разрыва строки.

### 21.4.10.3 Добавление нижнего колонтитула к столбцу 5

1 Выберите столбцы.

2 Выберите тип значения нижнего колонтитула. Для этого используйте подменю **Нижний колонтитул** в **основном меню**

или щелкните правой кнопкой мыши на заголовке столбца и выберите подменю **Нижний колонтитул** в открывшемся контекстном меню.

Доступны следующие типы значений:

- **Без функции** (столбец будет удален из нижнего колонтитула. Если нет столбца с другим типом нижнего колонтитула, строка нижнего колонтитула будет полностью удалена)
- **Сумма**
- **Среднее значение**
- **Минимальное значение**
- **Максимальное значение**

Альтернатива:

1 Выберите столбцы.

2 В **сетке свойств** выберите значение для свойства **Тип нижнего колонтитула**.

**Рисунок 79:** Свойства столбца

***Прим.:*** Для переключения между значениями **Сумма** и **Без функции** нажмите кнопку **Нижний колонтитул** ().

### 21.4.10.3 Сохранение текущей конфигурации как шаблона 6

1 Выберите опцию **Сохранить как шаблон** () в основном меню ().

Откроется диалоговое окно для выбора имени и расположения файла шаблона:



**Рисунок 80:** Выберите файл для сохранения шаблона таблицы спецификации

2 Укажите имя и расположение файла шаблона.

3 Нажмите **Сохранить** .

### 21.4.10.3 Экспорт текущей таблицы в CSV 7

1 При необходимости, выполните настройку свойства **Разделитель CSV** для таблицы.

**Рисунок 81:** Разделитель CSV

2 Выберите в основном меню команду **Экспорт в CSV** ().

Откроется диалоговое окно для выбора имени и расположения .csv файла.

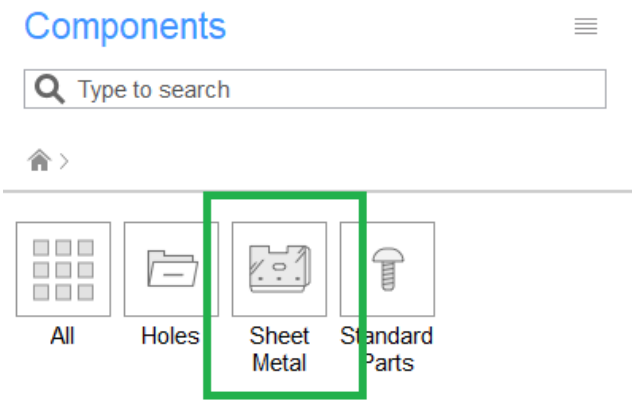



**Рисунок 82:** Выберите файл для экспорта таблицы спецификации

3 Укажите имя и расположение .csv файла.

4 Нажмите **Сохранить** .

### 21.4.11 Спецификация

#### 21.4.11.1 Обзор

Используемые команды: *BMBOM*, *BMBOMPANELOPEN* и *BMBOMPANELCLOSE*.

Спецификация представляет собой перечень уникальных механических компонентов, используемых в сборке. В BricsCAD этот перечень выполняется в виде таблицы (см. раздел Таблицы) и может быть размещен в любой точке чертежа. Для каждого элемента указывается номер его позиции (начиная с 1) и количество.

Для просмотра и настройки спецификаций используется панель Диспетчер спецификаций.

#### 21.4.11.2 Типы спецификаций

В BricsCAD доступны следующие три типа спецификаций:

- **Крупноузловая**:

  Содержит только компоненты верхнего уровня. Компоненты верхнего уровня - это компоненты, которые вставлены непосредственно в основную сборку. Этот тип спецификации включает информацию только об основных компонентах без подробных сведений об их подкомпонентах.
- **Детальная**:

  Содержит только компоненты (детали) нижнего уровня. Компоненты нижнего уровня - это детали, которые не включают в себя другие детали или подсборки. Этот тип спецификации включает все детали основной сборки, независимо от того, к какому компоненту принадлежат эти детали.
- **Иерархическая**:

Содержит все компоненты и их подкомпоненты в виде иерархии. Серийные номера подкомпонентов представляют собой последовательность чисел, разделенных точками (например, "1.2.3"). Этот тип спецификации отображает иерархическую структуру сборки.

***Прим.:*** Свойство **Статус в спецификации** определяет, какие детали включены в спецификацию. Уровень компонента в таблице спецификации устанавливается в соответствии со **Статусом в спецификации** всех связанных компонентов. Например, компонент со статусом Конечный всегда рассматривается как компонент нижнего уровня, даже если он содержит другие детали или узлы.

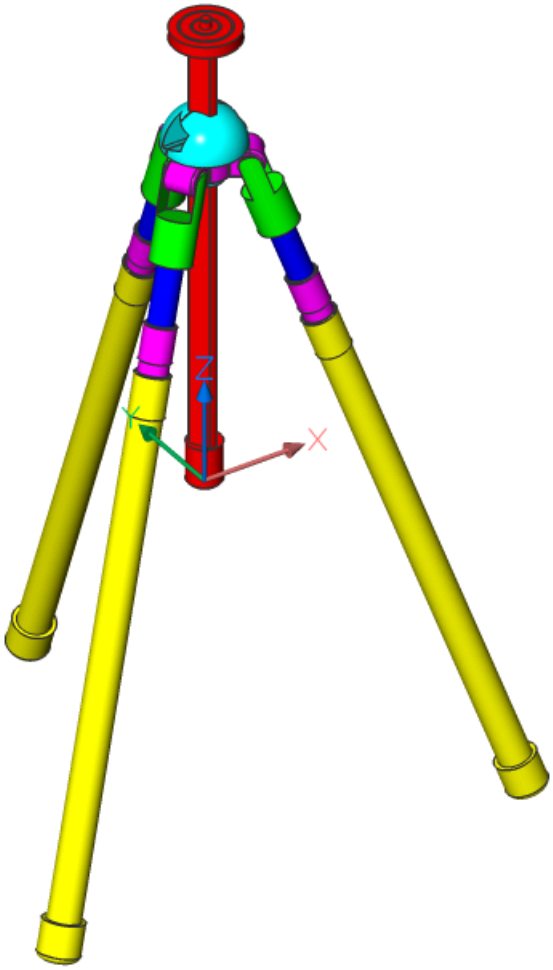



**Рисунок 83:** Пример сборки: Штатив

**Таблица 20: Типы спецификаций**

| Type | Спецификация |
|---|---|
| Крупноузловая | Bill of Materials tripod_main_assy<br>No. \| Component \| Quantity<br>1 \| tripod_axis_assy \| 1<br>2 \| tripod_foot_assy \| 3<br>**Рисунок 84:** Крупноузловая |
| Детальная | Bill of Materials tripod_main_assy<br>No. \| Component \| Quantity<br>1 \| lower_tube_prt \| 3<br>2 \| middle_tube_prt \| 3<br>3 \| tripod_axis_prt \| 1<br>4 \| tripod_bond_prt \| 3<br>5 \| tripod_crosspiece_prt \| 1<br>6 \| tripod_sphere_prt \| 1<br>7 \| upper_tube_prt \| 3<br>**Рисунок 85:** Детальная |
| Иерархическая | Bill of Materials tripod_main_assy<br>No. \| Component \| Quantity<br>1 \| tripod_axis_assy \| 1<br>1.1 \| tripod_axis_prt \| 1<br>1.2 \| tripod_crosspiece_prt \| 1<br>1.3 \| tripod_sphere_prt \| 1<br>2 \| tripod_foot_assy \| 3<br>2.1 \| bond_assy \| 3<br>2.1.1 \| tripod_bond_prt \| 3<br>2.2 \| tubes_assy \| 3<br>2.2.1 \| lower_tube_prt \| 3<br>2.2.2 \| middle_tube_prt \| 3<br>2.2.3 \| upper_tube_prt \| 3<br>**Рисунок 86:** Иерархическая |

### 21.4.11.3 Режим подсчета и максимальный уровень

Для иерархических спецификаций поддерживаются два режима подсчета:

- **Подсчет всех экземпляров**:

  Общее количество узлов и деталей во всей сборке.

- **Подсчет экземпляров в родительском компоненте**:

  Общее количество узлов и деталей в родительском компоненте. Родительский компонент считается корневым.

Например, основной узел штатива содержит 3 узла опор штатива. Каждый экземпляр узла опоры штатива содержит 1 узел крепления и 1 узел трубки.

Если для **режима подсчета** установлено значение **Подсчет всех экземпляров**, результат будет следующим:

**Таблица 21: Сборка штатива со спецификацией материалов (подсчет всех экземпляров)**

| # | Компонент | Количество |
|---|---|---|
| 1 | Сборка штатива | 1 |
| 1.1 | Сборка Опора штатива | 3 |
| 1.1.1 | Сборка Крепление | 3 |
| 1.1.2 | Сборка Трубка | 3 |

Если для **режима подсчета** установлено значение **Подсчет экземпляров в родительском компоненте**, результат будет следующим:

**Таблица 22: Спецификация сборки штатива (число экземпляров каждого компонента)**

| # | Компонент | Количество |
|---|---|---|
| 1 | Сборка штатива | 1 |
| 1.1 | Сборка Опора штатива | 3 |
| 1.1.1 | Сборка Крепление | 1 |
| 1.1.2 | Сборка Трубка | 1 |

По умолчанию для **режима подсчета** установлено значение - **Подсчет экземпляров в родительском компоненте.** Вы можете изменить режим подсчета в Диспетчере спецификаций или с помощью команды *BMBOM*.

***Прим.:*** До BricsCAD V20.2 все иерархические таблицы спецификаций по умолчанию использовали режим **Подсчет всех экземпляров**. При обновлении таких таблиц в BricsCAD V20.2 и более новых версиях в качестве режима по умолчанию устанавливается **Подсчет экземпляров в родительском компоненте**. Это может привести к изменению количества некоторых деталей.

При необходимости, вы можете вручную установить необходимый режим подсчета в Диспетчере спецификаций.

По умолчанию в иерархической спецификации отображаются все уровни модели. Вы можете ограничить иерархическую спецификацию определенным уровнем с помощью свойства **Максимальный уровень**. Когда установлен максимальный уровень, будут включены детали, входящие в сборку только до этого уровня. Например, если **Максимальный уровень** = 2, будут перечислены только компоненты и подкомпоненты верхнего уровня, непосредственно входящие в компоненты верхнего уровня.

### 21.4.11.4 Столбцы спецификации

По умолчанию таблица спецификации содержит только три столбца: номер позиции компонента, его наименование и количество экземпляров данного компонента в сборке.

Опция Настроить команды *BMBOM* позволяет добавлять дополнительные столбцы: Описание, Плотность, Объем, Масса, Материал, Толщина (для листового металла) и Параметры (для параметрических компонентов).

В таблицу спецификации можно добавить любое встроенное свойство компонентов, экземпляров или блоков. Также поддерживаются настраиваемые столбцы для вычислений с использованием технологии DataExtraction.

| Bill of Materials tripod_main_assy | | | | | |
|---|---|---|---|---|---|
| No. | Component | Quantity | Density | Volume | Mass |
| 1 | tripod_axis_assy | 1 | 8000.00 kg/m³ | 330.77 cm³ | 2.65 kg |
| 1.1 | tripod_axis_prt | 1 | 8000.00 kg/m³ | 194.42 cm³ | 1.56 kg |
| 1.2 | tripod_crosspiece_prt | 1 | 8000.00 kg/m³ | 60.06 cm³ | 0.48 kg |
| 1.3 | tripod_sphere_prt | 1 | 8000.00 kg/m³ | 76.29 cm³ | 0.61 kg |
| 2 | tripod_foot_assy | 3 | 8000.00 kg/m³ | 166.24 cm³ | 1.33 kg |
| 2.1 | bond_assy | 3 | 8000.00 kg/m³ | 21.32 cm³ | 0.17 kg |
| 2.1.1 | tripod_bond_prt | 3 | 8000.00 kg/m³ | 21.32 cm³ | 0.17 kg |
| 2.2 | tubes_assy | 3 | 8000.00 kg/m³ | 144.92 cm³ | 1.16 kg |
| 2.2.1 | lower_tube_prt | 3 | 8000.00 kg/m³ | 62.85 cm³ | 0.50 kg |
| 2.2.2 | middle_tube_prt | 3 | 8000.00 kg/m³ | 34.83 cm³ | 0.28 kg |
| 2.2.3 | upper_tube_prt | 3 | 8000.00 kg/m³ | 47.24 cm³ | 0.38 kg |

**Рисунок 87:** Спецификация с данными о плотности, объеме и массе

#### Режимы группировки

В таблицах спецификации перечислены только уникальные компоненты; несколько вставок одних и тех же компонентов группируются в соответствии с режимом группировки.

Начиная с BricsCAD V21 поддерживаются следующие режимы группировки:

- **Авто** (режим по умолчанию).

  Несколько вставок одних и тех же компонентов группируются вместе по их параметрам и добавленным свойствам. Многократные вставки одного параметрического компонента группируются в отдельные строки спецификации в зависимости от значения их параметров, независимо от того, добавлены ли эти параметры в спецификацию как столбцы.

| Bill of Materials Bookshelf | | | | | |
|---|---|---|---|---|---|
| No. | Component | Quantity | L | T | W |
| 1 | Frame Board | 2 | 360.0 | 20.0 | 200.0 |
| 2 | Frame Board | 2 | 700.0 | 20.0 | 200.0 |
| 3 | Shelf Board | 2 | 266.7 | 2.5 | 200.0 |
| 4 | Shelf Board | 2 | 466.7 | 2.5 | 200.0 |

**Рисунок 88:** Спецификация Книжная полка

- **По компонентам и столбцам**

  В этом режиме несколько вставок одних и тех же компонентов группируются вместе только по добавленным свойствам. Это означает, что несколько вставок одного и того же параметрического компонента могут быть сгруппированы в одной строке, даже если их параметры различны, в зависимости от того, какие столбцы добавляются в спецификацию. Но экземпляры разных компонентов всегда группируются в разные строки. Этот режим полезен, если некоторые параметры соответствуют разным состояниям компонентов, но не приводят к тому, что эти компоненты являются разными деталями.

- **Только по столбцам**

  В этом режиме вставки компонентов группируются по их свойствам, добавленным в спецификацию, независимо от того, являются ли они вставками одного и того же компонента. Этот режим полезен для получения сводных таблиц, таких как список всех материалов или всех размеров труб.

**Видимые и невидимые столбцы**

Любой столбец в таблице спецификации может быть сделан невидимым. Невидимые столбцы не появятся в таблице документа. Но вы по-прежнему можете использовать эти столбцы в формулах. Невидимые столбцы также будут отображаться в Диспетчере спецификаций.

**Агрегатные столбцы**

Начиная с BricsCAD V21, можно установить агрегатную функцию почти для каждого столбца в таблицах крупноузловых и детальных спецификаций. Агрегатные функции группируют значения нескольких строк спецификации (которые отличаются только этими значениями) в одно значение. Поддерживаются следующие агрегатные функции:

- **Сумма**: Суммирует сгруппированные значения.
- **Среднее**: Находит среднее значение для группы значений.
- **Минимум**: Находит минимальное значение для группы значений.
- **Максимум**: Находит максимальное значение для группы значений.
- **Объединить**: Объединяет все уникальные значения в группе в порядке возрастания.
- **Объединить с подсчетом**: Объединяет все уникальные значения с указанием их количества в группе.

Рассмотрим, например, следующую спецификацию:

**Таблица 23: Таблица спецификации**

| Компонент | Ширина | Длина | Высота | Количество |
|---|---|---|---|---|
| Балка | 16 | 250 | 10 | 1 |
| Балка | 16 | 200 | 10 | 2 |
| Балка | 16 | 220 | 20 | 1 |
| Балка | 20 | 215 | 20 | 3 |
| Балка | 20 | 225 | 20 | 1 |

Применим агрегатную функцию к столбцу **Длина**. Результаты объединены в следующей таблице:

**Таблица 24: Таблица спецификации**

| Компонент | Ширина | Высота | Длина (сумма) | Длина (средняя) | Длина (минимальная) | Длина (максимальная) | Длина (объединить) | Длина (объединить с подсчетом) |
|---|---|---|---|---|---|---|---|---|
| Балка | 16 | 10 | 650 | 216.6667 | 200 | 250 | 200; 250 | 2x200; 250 |
| Балка | 16 | 20 | 220 | 220 | 220 | 220 | 220 | 220 |
| Балка | 20 | 20 | 870 | 217.5 | 215 | 225 | 215; 225 | 3x215; 225 |

Для столбцов **Объединить с подсчетом** вы можете:

- Настроить расположение числа количества (впереди или позади значения).
- Добавить разделитель между значениями.
- Добавить разделитель между значениями и их количеством.
- Определить постфикс и/или префикс для количества.

Если в расчетном столбце используется агрегированный столбец, по умолчанию будут использоваться окончательные значения. К расчетному столбцу можно применить агрегатную функцию. Результаты для такого расчетного столбца будут сначала рассчитываться для всех исходных значений (до агрегирования), а затем полученные значения будут агрегированы.

**Столбец нумерации и назначение столбца**

Столбец нумерации - это специальный столбец для нумерации строк. В нем автоматически присваиваются номера всем строкам в таблице спецификации, без пропусков и независимо от других таблиц. Нумерация строк начинается с 1.

Для столбца нумерации могут быть изменены следующие настройки:

- **Префикс**: Строка, которая будет добавлена перед любым числом. Например, если вам необходимо, чтобы все числовые значения начинались с буквы "A".
- **Суффикс**: Строка, которая будет добавлена после числа. Например, вы можете добавить "-1" ко всем числовым значениям.

Для иерархической спецификации доступны следующие настройки:

- **Тип нумерации**: Определяет формат нумерации:
  - **Полная**: Номера будут представлены в формате "1.2.3".
  - **Сокращенная**: Отображается нумерация только текущего уровня. Его предполагается использовать вместе со столбцом Уровень, который отображает уровень текущей детали в сборке.
  - **Непрерывная** все компоненты будут нумероваться непрерывно, начиная с 1, независимо от их уровня.
- **Разделитель**: Указывает символ для разделения чисел разных уровней, если для типа нумерации установлено значение *Полная*..

Как правило, столбец нумерации используется в качестве источника для аннотирования чертежа командой *BMBALLOON*. Но любой столбец в таблице также можно использовать в качестве источника для аннотирования, изменив его роль. Вы можете выбрать одно из следующих назначений столбца:

- **Нумерация**: В столбце указаны номера позиций для аннотаций.
- **Имя**: Столбец содержит наименования деталей для аннотаций.
- **Количество**: Столбец содержит количество деталей для аннотации.
- **Обычный**: столбец не имеет специального назначения.

Только один столбец может иметь роль **Номер**, **Имя** или **Количество**. Если ни один из столбцов не имеет одного из этих назначений, автоматическое аннотирование невозможно.

**Ширина столбца**



**Рисунок 89:** Свойства столбца

По умолчанию ширина каждого столбца устанавливается автоматически, поэтому его содержимое размещается без дополнительных разрывов строк. При необходимости, вы можете установить фиксированную ширину столбца.

Чтобы задать ширину столбца, выполните одно из следующих действий:

- Выберите столбец в Диспетчере спецификаций и введите для него значение свойства **Ширина** в единицах листа.
- Измените ширину столбца в связанной таблице. Начиная с BricsCAD V20.2, таблицы спецификации автоматически сохраняют ширину столбцов, которая была установлена вручную.

Чтобы восстановить автоматический режим определения ширины, выберите столбец в Диспетчере спецификаций и выберите опцию **Авто** в раскрывающемся списке свойства **Ширина**.

**Формат столбца**

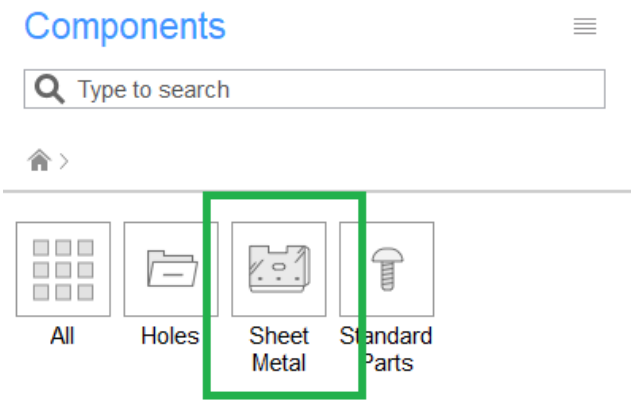



**Рисунок 90:** Формат

Вы можете установить формат значений столбца:

1 Выберите необходимый столбец в Диспетчере спецификаций.

2 Выберите в контекстном меню опцию **Формат**.

3 Установите необходимый формат в диалоговом окне **Формат**.

***Прим.:*** Доступные настройки зависят от типа данных выбранного столбца. Вы можете выбрать несколько столбцов одного типа и установить для них одинаковый формат значений.

**Набор свойств**

Так как в таблицах спецификаций используется технология DataExtraction, вы можете перечислять в таблицах спецификаций свойства механических компонентов и свойства объектов, например, ссылки на блоки.

***Прим.:*** Но эти свойства не всегда являются необходимыми. В больших сборках они могут отрицательно повлиять на производительность. Это связано с тем, что BricsCAD кэширует свойства во время создания таблицы спецификации.

Наборы свойств представляют собой встроенные наборы свойств. Вы можете выбрать наиболее оптимальный набор свойств, соответствующий вашим задачам. В настоящее время поддерживаются следующие наборы свойств:

- **Только механические** (по умолчанию): Будут перечислены только свойства механических компонентов.
- **Все кроме координат**: Выводит список всех свойств компонентов, кроме координат. Координаты обычно используются только для экземпляров компонентов.
- **Все свойства**: Перечисляет все возможные свойства компонентов и экземпляров компонентов.

Вы можете изменить набор свойств для таблицы спецификации в Диспетчере спецификаций или использовать команду *BMBOM* при добавлении новой таблицы.

### 21.4.11.5 Фильтр таблицы спецификации

Вы можете использовать фильтр для деталей, включенных в состав спецификации. Начиная с BricsCAD V21, используются два типа фильтров: **основной фильтр** и **фильтр по столбцам**.

**Основной фильтр**

Главный фильтр работает так же, как команда *DATAEXTRACTION*. Сначала все детали в сборке на требуемом уровне (верхний, нижний или иерархический) сопоставляются с выражением фильтра. В спецификацию будут включены только те детали, которые прошли фильтр.

Выражение фильтра может не относиться к столбцам спецификации, поскольку фильтрация выполняется до создания спецификации. Например, можно создать фильтр для всех стандартных деталей, так как **стандартность** является обычным свойством компонента. Но невозможно настроить основной фильтр для фильтрации всех деталей с количеством больше 1. Для этого необходимо использовать фильтры столбцов.

Конфигуратор фильтра

Самый простой способ настроить основной фильтр - использовать **Конфигуратор фильтров** в Диспетчере спецификаций.

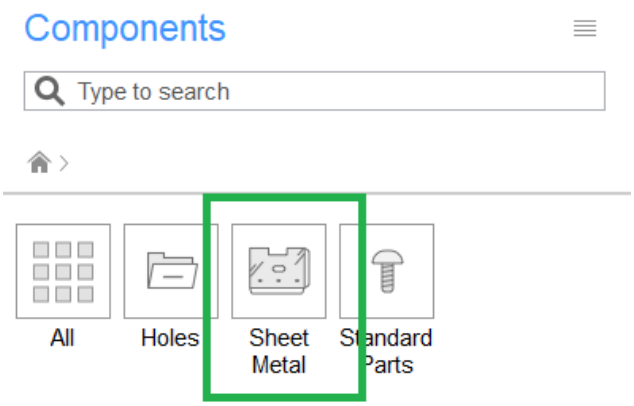



**Рисунок 91:** Конфигуратор фильтра

Конфигуратор фильтров позволяет задать набор условий. Каждое условие определяет соотношение между выбранным свойством и целевым значением.

Возможные отношения:

- Равно
- Не равно
- Меньше, чем
- Больше, чем
- Меньше или равно
- Больше или равно

Условия могут быть добавлены в группу, которая определяет, как следует обрабатывать эти состояния. Есть 2 типа групп:

- Группа **Все из**: "true", если все условия в этой группе "true". Соответствует оператору "and".
- Группа **Любая из**: "true", если хотя бы одно условие в этой группе "true". Соответствует оператору "or".

Группа может содержать произвольное количество условий. Она также может содержать вложенные группы, что делает возможным создание сложных фильтров.

Чтобы добавить условие:

- Выберите свойство из списка свойств.
- Выберите требуемый оператор сравнения и укажите целевое значение.

  Для текстовых значений целевое значение можно выбрать из списка всех возможных значений. Для числовых значений можно указать их целевую единицу, чтобы автоматически преобразовать значение относительно единицы измерения, используемой в документе по умолчанию.

Для текстовых значений операторы сравнения "Равно" и "Не равно" поддерживают специальные символы.

Конфигуратор фильтров автоматически преобразует набор условий в выражение фильтра, которое можно сделать видимым, нажав кнопку :

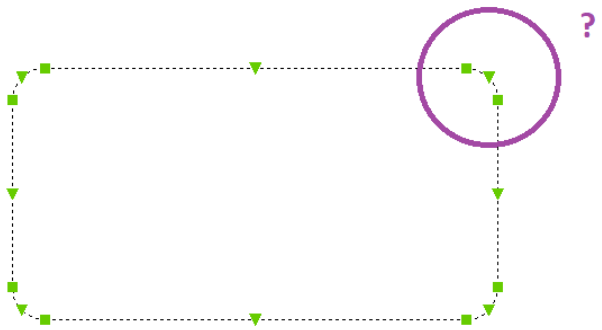



**Рисунок 92:** Выражение

Это выражение полностью редактируемое и имеет двустороннюю ассоциацию с набором условий.

Описание синтаксиса:

**Таблица 25: Операторы сравнения:**

| Оператор | Значение |
|---|---|
| == | равно |
| != | не равно |
| < | меньше |
| > | больше |
| <= | меньше или равно |
| >= | больше или равно |

- Наименования свойств необходимо заключать в кавычки.
- Операторы отношения можно комбинировать с помощью операторов "and" и "or". Оператор "and" приведет к "true", если обе стороны оператора "true". Оператор "or" приведет к "false", если либо левая, либо правая сторона оператора - "true".
- Логические операторы "and" и "or" чувствительны к регистру.
- Используйте "(" и ")", чтобы объединить два оператора в один.
- Строковые значения заключаются в двойные кавычки "".

- Операторы сравнения для строковых значений чувствительны к регистру.
- Операторы "==" и "! =" поддерживают шаблоны со специальными символами.

Например, чтобы получить спецификацию всех стандартных деталей, используйте следующее выражение:

"Is standard"=="Yes"

Чтобы получить спецификацию всех стандартных деталей ISO, используйте следующее выражение:

"Is standard"=="Yes" and "Part standard"=="ISO"

Чтобы получить спецификацию всех стандартных деталей ISO или DIN, используйте следующее выражение:

"Is standard"=="Yes" and ("Part standard"=="ISO" or "Part standard"=="DIN")

Специальные символы

Специальные символы позволяют сопоставить значения свойств с определенным шаблоном. Например, если вы хотите найти все компоненты, содержащие в своем названии слово «сопло», можно использовать следующее выражение: "Component name"=="*сопло*".

Специальные символы также можно использовать в качестве целевых значений в Конфигураторе фильтров.

Специальные символы поддерживаются только реляционными операторами "Равно" и "Не равно".

***Прим.:*** Обратите внимание, что специальные символы в основном фильтре чувствительны к регистру. Для фильтров столбцов вы можете указать, требуется ли вам соответствие с учетом регистра или без учета регистра.

Поддерживаются следующие специальные символы:

**Таблица 26: Специальные символы**

| Имя | Символ | Description |
|---|---|---|
| Обратная кавычка | ` | Игнорирует следующий символ (воспринимает его буквально). |
| Запятая | , | Разделяет шаблоны. |
| Астериск | * | Соответствует любой последовательности символов, включая нулевую последовательность в любом месте. |
| Знак вопроса | ? | Соответствует одиночному символу. |
| Точка | . | Соответствует одиночному не буквенно-цифровому символу. |

**Таблица 26: Специальные символы**

| Имя | Символ | Description |
|---|---|---|
| "Собака" | @ | Соответствует одиночному буквенному символу. |
| Хэштег | # | Соответствует одиночному цифровому символу. |
| Тильда | ~ | Если первый символ в шаблоне, соответствует чему-либо, кроме шаблона. |
| Скобки | [...] | Соответствует одиночному закрытому символу. |
| Скобки с тильдой | [~...] | Соответствует одиночному незакрытому символу. |
| Дефис | - | В скобках соответствует одиночному символу в указанном диапазоне. |

**Фильтр столбца**

Фильтры столбцов были введены в BricsCAD V21. Они позволяют быстро определить условия отображения строк в зависимости от содержимого столбца (по аналогии с фильтрами, применяемыми в некоторых редакторах электронных таблиц). В отличие от основного фильтра, они применяются после создания таблицы спецификации. Это позволяет использовать фильтры столбцов для столбцов с агрегированным содержимым, например количеством.

В каждом столбце может быть только один фильтр. Но вы можете добавить фильтры столбцов к любому количеству столбцов. В спецификации отображаются только строки, прошедшие через все фильтры столбцов.

Вы можете настроить фильтры столбцов с помощью диалогового окна **Фильтр столбцов** в Диспетчере спецификаций:

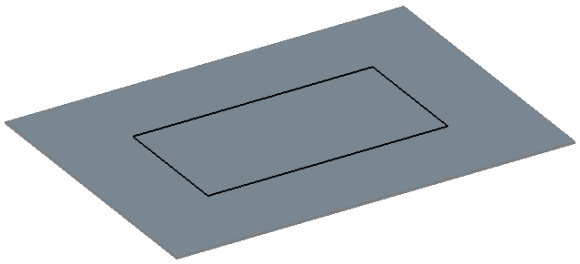


**Рисунок 93:** Быстрый фильтр

В этом диалоговом окне вы можете выбрать тип фильтра из списка и настроить его соответствующим образом. Список доступных фильтров зависит от типа содержимого столбца, как показано в таблице ниже:

**Таблица 27: Фильтры**

| Тип фильтра | Тип столбца | Description |
|---|---|---|
| всегда принимается | любой | принимаются все значения столбца |
| равно | числовое значение | значение должно быть равно указанному значению |
| не равно | числовое значение | значение не должно быть равно указанному значению |
| меньше | числовое значение | значение должно быть меньше указанного значения |
| больше | числовое значение | значение должно быть больше указанного значения |
| меньше или равно | числовое значение | значение должно быть меньше или равно указанному значению |
| больше или равно | числовое значение | значение должно быть больше или равно указанному значению |
| между | числовое значение | значение должно быть в заданном диапазоне, включая оба граничных значения |
| не в диапазоне | числовое значение | значение должно быть в заданном диапазоне, включая оба граничных значения |
| точно соответствует | текстовое значение | значение должно точно соответствовать указанному значению |
| не соответствует | текстовое значение | значение должно отличаться от указанного значения |
| начинается с | текстовое значение | значение должно начинаться с указанного текста |

**Таблица 27: Фильтры**

| Тип фильтра | Тип столбца | Description |
|---|---|---|
| не начинается с | текстовое значение | значение не должно начинаться с указанного текста |
| заканчивается на | текстовое значение | значение должно заканчиваться указанным текстом |
| не заканчивается на | текстовое значение | значение не должно заканчиваться указанным текстом |
| содержит | текстовое значение | значение должно содержать указанный текст |
| не содержит | текстовое значение | значение не должно содержать указанный текст |
| соответствует образцу | текстовое значение | значение должно соответствовать заданному шаблону |
| не соответствует шаблону | текстовое значение | значение не должно совпадать с заданным шаблоном |
| in | текстовое и числовое значение | значение должно быть в заданном наборе значений |
| not in | текстовое и числовое значение | значение должно быть в заданном наборе значений |

Для всех текстовых фильтров вы также можете указать, должны ли они быть чувствительными к регистру или нет.

### 21.4.11.6 Сортировка в таблице спецификации

В таблицах спецификации доступны три режима сортировки:

- **Автоматически**:

  Содержимое таблицы сортируется следующим образом: имя компонента, имя физического материала, толщина (если применимо), параметры (в алфавитном порядке) и количество.
- **Исходный порядок**:

  Содержимое таблицы не сортируется. Вместо этого все компоненты расположены в порядке их добавления в документ. В большинстве случаев детали, добавленные ранее, будут предшествовать деталям, добавленным позже.
- **Пользовательский порядок**:

Содержимое таблицы сортируется по выбранным столбцам. Если столбцы не указаны, используется исходный порядок.

Начиная с BricsCAD V21, таблицы спецификации используют так называемую "естественную сортировку" текста. Это означает, что если текст содержит сочетание букв и цифр, он будет отсортирован по этим буквам и числам. Например, строки «N1», «M2», «N11», «N2», «AA1» будут отсортированы как «M2», «N1», «N2», «N11», «AA1». Следовательно, можно назначать компонентам пользовательские номера с различными префиксами с помощью настраиваемых параметров, а затем сортировать таблицы спецификации по этим номерам в правильном порядке.

### 21.4.11.7 Расчетные столбцы

Расчетные столбцы - это специальные столбцы, значения которых зависят от других столбцов и определяются с использованием пользовательских выражений. Начиная с BricsCAD V21, поддерживаются два типа расчетных столбцов: **столбцы с формулами** и **столбцы с шаблонами**.

**Столбцы формул**

Спецификация поддерживает поля с формулами. Их синтаксис аналогичен синтаксису, используемому в команде *DATAEXTRACTION*. **FormulaExpression** - это выражение в формате полей формулы **\AcExpr**. Он может содержать константы, такие как PI, или функции, такие как SIN (см. Использование выражений ). Операндом в выражении может быть **ЗаголовокСтолбца**, заключенный в скобки со стрелками (<>).

Например, формула *=<Mass>/<Volume>* вычисляет среднюю плотность компонента.

Если строка не содержит значения, вместо него будет использоваться значение по умолчанию. Вы можете указать значение по умолчанию для операнда, используя следующий синтаксис: *<"ColumnTitle",DefaultValue>* где **ColumnTitle** - это заголовок столбца, а **DefaultValue** - соответствующее значение по умолчанию. **ColumnTitle** должен быть заключен в двойные кавычки ("). Если значение по умолчанию не указано, будет использоваться пустое значение, что в большинстве случаев приводит к пустому конечному результату.

Столбец формулы может содержать несколько выражений. Каждое выражение должно быть заключено в квадратные скобки суммы ({}). Эти выражения будут вычисляться от первого до последнего, и будет использовано первое выражение, дающее непустой результат.

Начиная с BricsCAD V21, формула может ссылаться на другие столбцы формулы, при условии, что они не образуют цикл. В иерархической спецификации формула также может ссылаться на значения родительской строки с добавлением *Parent:* перед именем столбца. Она также может относиться к свойствам, которые явно не добавлены в таблицу, но все остальные столбцы формулы, указанные в формуле, должны существовать в таблице. Вы по-прежнему можете отображать такие столбцы как невидимые в выражениях формул таблицы спецификации.

Для свойств расстояния, площади, объема и массы перед их использованием в выражении формул значения будут преобразованы в **пользовательские** единицы. Если **пользовательские** единицы не указаны, будут использованы единицы документа. Для массовых характеристик

в документах с метрической системой будет использоваться килограмм, а в документах с английской системой - фунт.

**Столбцы с шаблонами**

Столбцы с шаблонами позволяют объединить несколько свойств, а также пользовательский текст в один столбец. Для определения столбцов с шаблонами используется специальный синтаксис, основанный на выражениях Diesel. Выражения для столбцов с шаблонами должны начинаться со знака **$**. Операндом в выражении может быть **ЗаголовокСтолбца**, заключенный в скобки со стрелками (<>). Также возможно использование специальных функций:

**Таблица 28: Специальные функции**

| Функция | Description |
|---|---|
| $(+, arg1, arg2[, ... , arg9]) | Добавляет к первому аргументу последующие документы. |
| $(-, arg1, arg2[, ... , arg9]) | Вычитает из первого аргумента последующие аргументы. |
| $(*, arg1 , arg2[, ... , arg9]) | Умножает первый аргумент на последующие аргументы. |
| $(/, arg1, arg2[, ..., arg9]) | Делит первый аргумент на последующие аргументы. |
| $(=, arg1, arg2) | Возвращает 1, если arg1 равен arg2; в противном случае возвращается 0. Оба аргумента arg1 и arg2 должны быть числами. |
| $(<, arg1, arg2) | Возвращает 1, если arg1 меньше arg2; в противном случае возвращается 0. Оба аргумента arg1 и arg2 должны быть числами. |
| $(>, arg1, arg2) | Возвращает 1, если arg1 больше arg2; в противном случае возвращается 0. Оба аргумента arg1 и arg2 должны быть числами. |
| $(!=, arg1, arg2) | Возвращает 1, если arg1 не равен arg2; в противном случае возвращается 0. Оба аргумента arg1 и arg2 должны быть числами. |
| $(<=, arg1, arg2) | Возвращает 1, если arg1 меньше или равен arg2; в противном случае возвращается 0. Оба аргумента arg1 и arg2 должны быть числами. |

**Таблица 28: Специальные функции**

| Функция | Description |
|---|---|
| $(>=, arg1, arg2) | Возвращает 1, если arg1 больше или равен arg2; в противном случае возвращается 0. Оба аргумента arg1 и arg2 должны быть числами. |
| $(and, arg1, arg2[, ... , arg9]) | Применяет к аргументам побитовую операцию И. |
| $(angtos, угол[, формат, точность]) | Преобразует угол в строку в соответствии с заданными форматом (0 - градусы, 1 - г/м/с, 2 - грады, 3 - радианы) и точностью. |
| $(delim, строка_разделителя) | Вставляет умный разделитель. Строка разделителя будет добавлена к выходным данным только в том случае, если выходные данные до и после этого разделителя не пусты. |
| $(edtime, time, формат) | Форматирует дату в соответствии с заданным форматом. |
| $(eq, arg1, arg2) | Возвращает 1, если arg1 равен arg2; в противном случае возвращается 0. Оба аргумента arg1 и arg2 должны быть текстовыми. |
| $(eval, выражение) | Вычисляет заданное выражение как выражение Diesel. |
| $(fix, значение) | Отсекает действительное значение до целого. |
| $(getvar, переменная) | Возвращает значение указанной системной переменной. |
| $(if, arg, then[, else]) | Возвращает 'then', если 'arg' не равен 0; в противном случае возвращает 'else' или пустую строку, если 'else' не указано. |
| $(index, позиция, строка_разделенная_запятыми) | Вернуть подстроку в заданной позиции в заданной строке с разделением запятой. |
| $(nth, позиция, arg1[, arg2, ... , arg9]) | Возвращает аргумент в заданной позиции. |
| $(or, arg1, arg2[, ..., arg9]) | Применяет к аргументам побитовую операцию ИЛИ. |
| $ (rtos, значение[, режим, точность]) | Форматирует заданное значение с учетом требуемого режима и точности. |

**Таблица 28: Специальные функции**

| Функция | Description |
|---|---|
| $ (strlen, строка) | Возвращает длину заданной строки. |
| $ (substr, строка, начало[, длина]) | Возвращает подстроку строки от указанного символа и заданной длины. Нумерация символов начинается с 1. |
| $(upper, строка) | Преобразует строку в верхний регистр. |
| $(xor, arg1, arg2[, ... , arg9]) | Применяет побитовую исключающую операцию ИЛИ (XOR) ко всем аргументам. |

Столбец шаблона может содержать несколько выражений. Каждое выражение должно быть заключено в квадратные скобки суммы ({}). Эти выражения будут вычисляться от первого до последнего, и будет использовано первое выражение, дающее непустой результат.

Начиная с BricsCAD V21, столбец шаблона может ссылаться на другие расчетные столбцы, при условии, что они не образуют цикл. В иерархической спецификации столбец шаблона также может ссылаться на значения родительской строки с добавлением *Parent:* перед именем столбца. Он также может ссылаться на свойства, которые явно не добавлены в таблицу, но все остальные расчетные столбцы, указанные в выражении, должны существовать в таблице. Вы по-прежнему можете использовать в выражениях шаблона столбцы, отображаемые как невидимые в таблице спецификации.

Для добавления разрыва строки используйте '\n'.

Например, в следующем выражении комбинируются свойства PartNumber, Width, Height и Length:

$ {<PartNumber> $ (разделитель, "X") <Width> $ (разделитель, "X") <Height> $ (разделитель, "X") <Lenght>}

### Конфигуратор выражений

Вы можете использовать диалоговое окно *Конфигуратор выражений* для настройки расчетного столбца. Это диалоговое окно доступно в Диспетчере спецификаций.



**Рисунок 94:** Редактор шаблонов

В этом диалоговом окне вы можете задать заголовок и единицы измерения столбца, отредактировать выражение с помощью редактора с подсветкой синтаксиса и автозавершением, а также выполнить просмотр таблицы спецификаций с расчетным столбцом. Раздел **Шаблон** позволяет легко вставлять, перемещать и удалять свойства, разделители и пользовательский текст столбца шаблона.

### 21.4.11.8 Единицы измерения столбца

Вы можете настроить режим **размещения единиц измерения** для столбцов. Доступны следующие опции:

- **Не показывать единицы**

  В таблице спецификации не будут отображаться единицы измерения этого столбца
  .

- **В заголовке столбца**

  Если все значения в таблице используют одну и ту же единицу измерения или могут быть преобразованы в общую единицу, все значения преобразуются в эту общую единицу. Наименование единицы измерения будет добавлено в заголовок столбца. Вы можете настроить положение заголовка и единицы измерения, используя свойство **Формат заголовка**. Используйте *<Заголовок>* для обозначения заголовка и *<Единица>* для обозначения единицы измерения. Например, формат заголовка по умолчанию - "*<Заголовок>*, **<Unit>**." Для добавления разрыва строки используйте '\n'.

  Обычно единицы измерения выбираются в следующем порядке:

  a Единицы, установленные пользователем.

  b Если свойство представляет собой расстояние, площадь, объем или массу, а режим единиц измерения - **Лучшее для всех значений**, используется представление, которое является наилучшим для большинства значений.

  c Если все значения имеют одну и ту же единицу измерения, будет использоваться это значение.

- **В ячейке**

  К значению будет добавлено наименование единицы измерения. Значение и единица измерения отображаются в одной ячейке.

- **В дополнительном столбце**

  Наименование единицы измерения будет размещено в отдельном столбце.

Вы можете установить **режим единиц измерения** для значений расстояния, площади, объема и массы, чтобы определить единицы, используемые для отображения определенного значения. Например, если масса может принимать значения от 1 грамма до нескольких тонн, выберите режим **Лучшее для каждого значения**, чтобы отобразить каждую массу с преобразованием в наиболее подходящую единицу массы. Чтобы отобразить все массы в килограммах, установите килограммы в качестве **пользовательской** единицы измерения для столбца. По умолчанию используется режим **Лучшее для всех значений**.

Параметры **режима единиц измерения**:

- **Не показывать единицы**

  Единицы не отображаются.
- **Лучшее для всех значений**

  Отображается единица с наилучшим представлением для большинства значений. Поддерживаются только размеры, площадь, объем и массовые характеристики.
- **Пользовательские**

  Вы выбираете единицы измерения для отображения. Кроме размеров, площади, объема и массовых характеристик вы также можете выбрать необходимую единицу измерения для таких свойств материала, как плотность, теплопроводность и удельная теплоемкость.
- **Лучшее для каждого значения**

  Используется единица, наиболее подходящая для каждого значения. Режим **В заголовке столбца** свойства **Положение единиц** автоматически переключается в режим **В дополнительном столбце**. Поддерживаются только размеры, площадь, объем и массовые характеристики.

***Прим.:*** Если для столбца задана пользовательская единица измерения и этот столбец используется в поле формулы, значения, преобразованные в эту единицу, будут подставлены в формулу. В противном случае будут использоваться единицы документа.

### 21.4.11.9 Нижний колонтитул

В таблицах спецификаций поддерживаются нижние колонтитулы. Чтобы определить, какие столбцы будут формировать нижний колонтитул, установите **Тип нижнего колонтитула** для выбранных столбцов:

- **Без функции**

  Столбец не будет включен в строку нижнего колонтитула.
- **Сумма**

  в нижнем колонтитуле отображается сумма всех значений для каждого отдельного экземпляра компонента.

  ***Прим.:*** Это не эквивалент суммы всех значений, отображаемых в таблице, поскольку учитывается количество каждого компонента. В полях формул используется сумма значений для Quantity=1.
- **Среднее**

  в нижнем колонтитуле отображается среднее значение из всех значений для каждого отдельного компонента. В полях формул будет использоваться сумма значений для Quantity=1.
- **Минимальное**

  В нижнем колонтитуле отображается наименьшее из всех значений для каждого отдельного компонента. В полях формул будет использоваться сумма значений для Quantity=1.
- **Максимальное**

В нижнем колонтитуле отображается наибольшее из всех значений для каждого отдельного компонента. В полях формул будет использоваться сумма значений для Quantity=1.

Если все столбцы имеют тип нижнего колонтитула **Без функции**, строка нижнего колонтитула не отображается.

### 21.4.11.1 Шаблоны

**0**

Вы можете сохранить конфигурацию спецификации в качестве шаблона и использовать ее для создания новых таблиц спецификации. Файл шаблона содержит все необходимые данные. Он включает в себя тип таблицы, заголовок, фильтр и набор столбцов со всеми связанными свойствами. Вы можете использовать <NAME> в заголовке в качестве фактического имени сборки. Например, если имя вашей сборки - 'Моя сборка', а заголовок спецификации указан как Спецификация для <NAME>, заголовком спецификации будет 'Спецификация для моей сборки', поскольку 'Моя сборка' заменяет <NAME>.

По умолчанию файлы шаблонов спецификации имеют расширение .BOM.

Файл шаблона спецификации является расширением файла определения извлечения данных (.dxd). Но вы не можете использовать .dxd файл в качестве шаблона спецификации, так как в этом файле отсутствуют некоторые данные, относящиеся к спецификациям. Вы также не можете использовать файл .BOM в качестве файла определения шаблона извлечения данных.

Для редактирования файлов шаблонов вы можете использовать любой текстовый редактор. Но Bricsys рекомендует для создания и редактирования файлов шаблонов использовать команду *BMBOM* или Диспетчер спецификаций.

### 21.4.11.1 Статус в спецификации

**1**

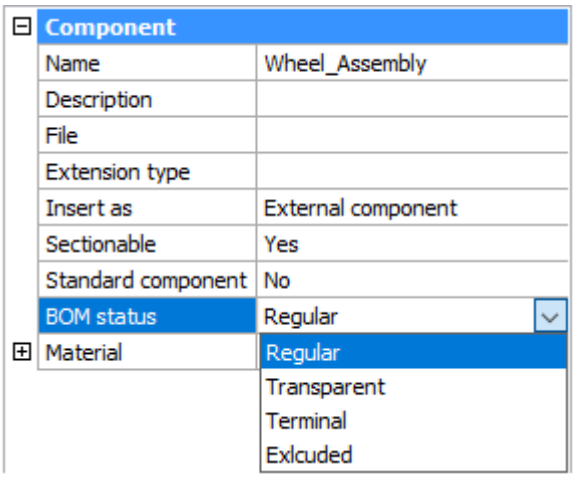



**Рисунок 95:** Компонент

Каждый компонент имеет свойство **Статус в спецификации**. Вы можете использовать это для управления отображением компонента в таблицах **спецификаций**. Это свойство может иметь одно из следующих значений:

- **Обычный**

  Компонент отображается в спецификации в той же позиции, что и в структуре сборки, используется значение по умолчанию выбранного режима спецификации.
- **Прозрачный**

  Компонент представляет собой прозрачный контейнер для своих подкомпонентов. Компонент не отображается в спецификации, но подкомпоненты могут отображаться в спецификации в зависимости от их статуса в спецификации, положения в структуре сборки и режима таблицы спецификаций.
- **Конечный**

  Компонент рассматривается как компонент без подкомпонентов. Это полезно для сложных деталей или стандартных компонентов.
- **Исключенный**

  компонент и его подкомпоненты не отображаются в Спецификации материалов.

**Установка статуса в спецификации внешнего или основного компонента**

1 Откройте файл, содержащий компонент.

2 Выберите основной компонент в Обозревателе конструкций.

3 Установите значение для свойства **Статус в спецификации**.

**Установка статуса в спецификации локального компонента**

1 Выберите компонент в Обозревателе конструкции.

2 Щелкните правой кнопкой мыши на выбранном узле.

3 В открывшемся контекстном меню в подменю **Статус в спецификации** выберите необходимый статус в спецификации.



**Рисунок 96:** Подменю Статус в спецификации

***Прим.:*** Использование контекстного меню для установки статуса в спецификации для локальных компонентов, которые вставляются во внешний компонент, невозможно. Для этого необходимо открыть компонент, содержащий локальный компонент.

#### 21.4.11.1 Фиксированные углы

2

Начиная с BricsCAD V21, вы можете фиксировать положение любого угла таблицы спецификации. Фиксированный угол таблицы всегда будет сохранять свое исходное положение во время обновления спецификации. При ручном перемещении таблицы будет сохранено новое положение ее фиксированного угла. Фиксированные углы позволяют выравнивать таблицы спецификации по другим объектам.

Вы можете выбрать фиксируемый угол таблицы при ее размещении с помощью команды *BMBOM* или Диспетчера спецификации; для циклического перемещения по углам используйте клавишу Tab. Вы можете изменить фиксированный угол таблицы в Диспетчере спецификаций.

### 21.4.12 Параметрическая 3D анимация

#### 21.4.12.1 Обзор

Используемые команды: *ANIMATIONEDITOROPEN* и *ANIMATIONEDITORCLOSE*.

Вы можете анимировать несколько параметров и запрограммировать перемещение камеры с помощью панели редактора анимации. После того, как анимация запрограммирована, вы можете воспроизвести ее в BricsCAD или визуализировать как 2D видео.

У каждого параметра анимации есть своя «временная шкала». Вы можете добавить к параметру любое количество анимационных последовательностей при условии, что эти последовательности не перекрываются.

Также есть специальная шкала перемещений камеры. Вы можете добавить любое количество последовательностей камер. Вы можете установить как точку фокусировки, так и точку камеры с помощью полилинии или точки. Эта процедура аналогична выполнению команды *ANIPATH*.

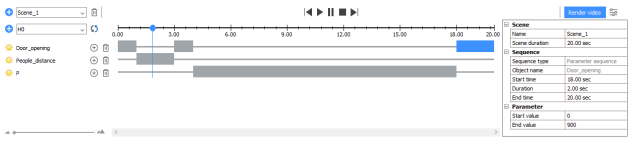


**Рисунок 97:** Временная шкала

#### 21.4.12.2 Сцены, временные шкалы и последовательности анимации

**Сцены**

Сцена состоит из одной или нескольких **временных шкал** и каждая из этих временных шкал имеет одну или несколько **анимационных последовательностей**. При первом запуске редактор автоматически создает сцену. Сцена устанавливает продолжительность анимации. Сцена и продолжительность определяют **расписание анимации**, используемое для воспроизведения и визуализации. Редактор анимации позволяет визуализировать только текущую сцену. Но в одном документе может быть много сцен, и вы можете переключаться между ними.

**Последовательности анимации**

Последовательность анимации - это основа анимации. Она представляет собой одиночную непрерывную анимацию объекта. Последовательность анимации имеет связанные между собой параметры Время начала и Продолжительность. Она также включает специфические значения параметров анимированного объекта, такие как начальное и конечное значения для параметров или пути камеры и точки цели для последовательностей камер:

- **Параметр**

  Последовательность анимации представляет собой непрерывное изменение параметра от начального до конечного значения. Например, для зависимости Расстояние между двумя гранями анимация перемещает одну грань на максимальное и минимальное значение расстояния.
- **Камера**

  Последовательность анимации соответствует непрерывному движению камеры по траектории. Вы можете управлять перемещением камеры с помощью временной шкалы камеры.

**Временная шкала**

Временная шкала - это визуальное представление последовательности анимации для объекта. На временной шкале показано, как и когда будут анимированы параметры. У каждого параметра анимации есть своя временная шкала. Перекрытие последовательностей анимации не допускается. Временную шкалу можно временно отключить, чтобы исключить ее из анимации. Все временные шкалы независимы. Это означает, что вы можете анимировать несколько параметров одновременно и увидеть комбинированный эффект от применения этих анимаций.

### 21.4.12.3 Открытие Редактора анимации

Выполните одно из следующих действий:

- Щелкните правой кнопкой на панели инструментов и в открывшемся контекстном меню выберите **Панели > Редактор анимации**.
- Нажмите кнопку **Редактор анимации** ( )на закладке ленты **Вид**.
- Выберите инструмент **Редактор анимации** в меню **Вид**.
- Введите в командной строке команду *ANIMATIONEDITOROPEN*.

### 21.4.12.4 Интерфейс Редактора анимации

Редактор анимации представляет собой панель. Он включает три области:

- **Панель инструментов**: содержит инструменты для выбора сцены, настройки анимации и визуализации видео.
- **Область временных шкал**: используется для добавления и редактирования временных шкал и последовательностей анимации.
- **Конфигуратор**: используется для настройки параметров текущей сцены и выбранной последовательности анимации.

### Панель инструментов



**Рисунок 98:** Панель инструментов

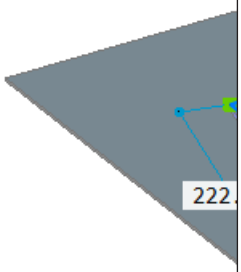
Панель инструментов содержит инструменты для:

- Добавления и удаления сцен
- Управления просмотром анимации
- Установки длительности текущей сцены
- Добавления временной шкалы камеры
- Запуска визуализации.

**Таблица 29: Панель инструментов**

| Инструмент | Имя | Description |
|---|---|---|
| | **Добавить сцену** | Добавляет пустую сцену с длительностью по умолчанию |
| | **Выбрать сцену** | Выбор текущей сцены |
| | **Удалить сцену** | Удаляет текущую сцену из документа |
| | **К предыдущей последовательности** | Переход к началу предыдущей последовательности анимации |
| | **Воспроизвести** | Запускает анимацию |
| | **Пауза** | Приостанавливает анимацию |

**Таблица 29: Панель инструментов**

| Инструмент | Имя | Description |
|---|---|---|
|  | **Остановить** | Останавливает анимацию и возвращает курсор в начальную позицию |
|  | **К следующей последовательности** | Переход к началу следующей последовательности анимации |
|  | **Визуализация** | Открывает диалоговое окно запуска визуализации 2D видео |
|  | **Показать/скрыть панель эффектов** | Управляет отображением панели эффектов |

**Область временных шкал**

**Область временных шкал** используется для добавления и удаления временных шкал, добавления и удаления последовательностей анимации и отображения расписания анимации. Щелкните на последовательности анимации, чтобы выбрать ее, или перетащите начало или конец соответствующей полосы, чтобы настроить продолжительность. Выбранная последовательность анимации будет выделена синим цветом, остальные последовательности - серым. Каждую временную шкалу можно временно отключить, чтобы исключить ее из анимации.

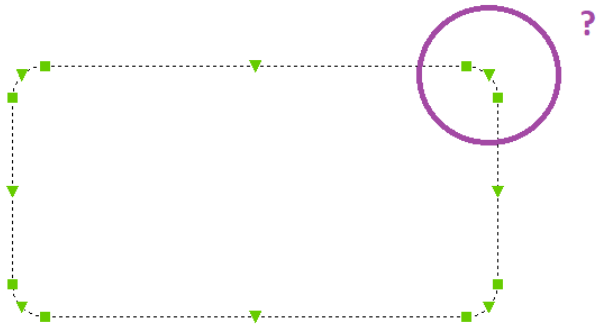


**Рисунок 99:** Область временных шкал

В **области временной шкалы** отображается курсор, который указывает на текущую позицию в анимации. При воспроизведении анимации курсор указывает на текущую позицию воспроизведения. При необходимости, вы можете вручную переместить курсор в другую позицию. При перемещении курсора все параметры изменяются в соответствии с текущим кадром. Также обновляется положение камеры.

Чтобы обеспечить соответствие состояния модели текущей позиции анимации, нажмите кнопку .

***Прим.:*** Вы можете выполнить это вручную в любой момент, но это не окажет влияния на анимацию.

### Масштаб времени

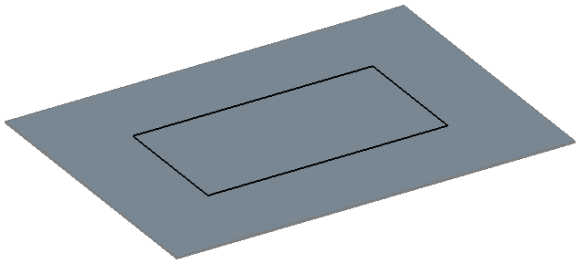
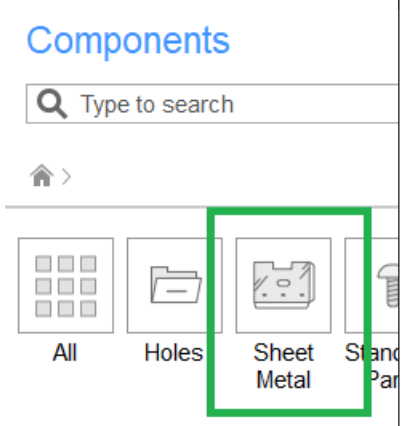
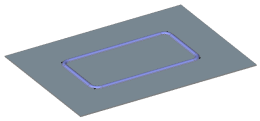


**Рисунок 100:** Масштаб времени

**Таблица 30: Масштаб времени**

| Инструмент | Имя | Description |
|---|---|---|
| | **Добавить временную шкалу** | Добавляет выбранный параметр во временную шкалу |

**Таблица 30: Масштаб времени**

| Инструмент | Имя | Description |
|---|---|---|
|  | **Селектор объекта** | Выбирает объект для добавления на временную шкалу |
|  | **Обновить модель** | Обновляет состояние модели в соответствии с текущей позицией курсора |
|  | **Масштаб времени** | Отображает метки времени на временной шкале |
|  | **Курсор** | Отображает текущую позицию воспроизведения |

### Временная шкала

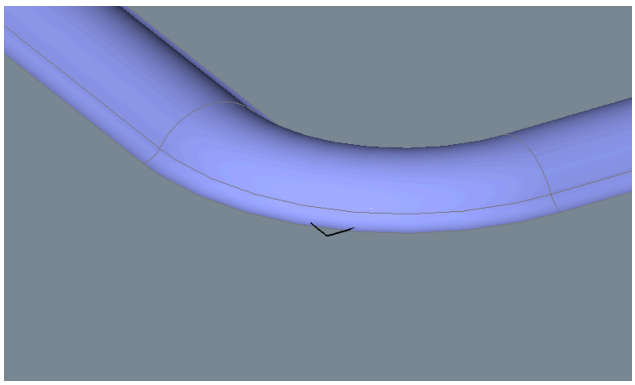


**Рисунок 101:** Временная шкала

**Таблица 31: Временная шкала**

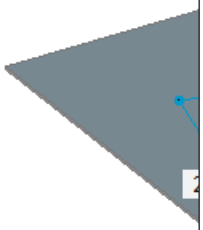
| Инструмент | Имя | Description |
|---|---|---|
|  | **Временная шкала включена** | Указывает, что временная шкала включена. Щелчок на этом элементе отключает временную шкалу. |
|  | **Временная шкала отключена** | Указывает, что шкала времени отключена. Щелчок на этом элементе включает временную шкалу. |
|  | **Объект временной шкалы** | Отображает имя связанного со шкалой объекта. |
|  | **Добавить последовательность** | Добавляет новую последовательность анимации в точке положения курсора |
| | **Удалить временную шкалу** | Удаляет временную шкалу |
|  | **Последовательность анимации** | Отображает длительность и позицию последовательности анимации. Для выбора последовательности щелкните левой кнопкой. |

**Таблица 31: Временная шкала**

| Инструмент | Имя | Description |
|---|---|---|
| | **Выбранная последовательность анимации** | Отображает длительность и позицию выбранной последовательности анимации. Просмотр и изменение свойств доступно в **Конфигураторе**.<br>Переместите мышь в конец последовательности. Курсор изменит свой вид на двустороннюю стрелку. Затем нажмите левую кнопку мыши и переместите курсор, чтобы изменить время начала, время окончания и продолжительность последовательности.<br>Щелкните левой кнопкой мыши в середине последовательности и перетащите его в новое положение на шкале времени, не изменяя длительность последовательности. |

### Конфигуратор

**Конфигуратор** предоставляет доступ ко всем свойствам сцены анимации и выбранной последовательности. Все свойства группируются по объекту.

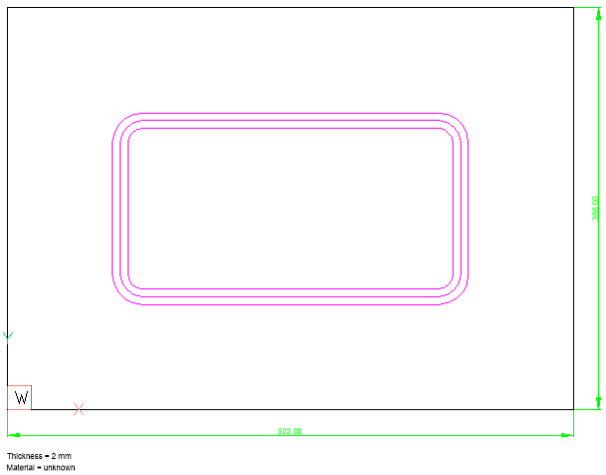



**Рисунок 102:** Конфигуратор

- **Сцена**: свойства текущей сцены.
  - **Имя**: наименование сцены.
  - **Длительность**: продолжительность сцены в секундах.
- **Последовательность**: свойства выбранной последовательности.
  - **Тип последовательности** (только для чтения): тип последовательности.
  - **Имя объекта** (только для чтения): имя объекта, контролируемого этой последовательностью.

- **Время начала**: время начала последовательности в секундах. Изменение значения влияет на длительность, время окончания остается неизменным.
- **Длительность**: продолжительность последовательности в секундах. Изменение значения влияет на время окончания. Время начала остается неизменным.
- **Время окончания**: время окончания последовательности в секундах. Изменение значения влияет на длительность, время начала остается неизменным.

- **Параметр** (для параметрических последовательностей)
  - **Начальное значение**: начальное значение параметра.
  - **Конечное значение**: конечное значение параметра.
- **Камера** (для последовательности камеры)
  - **Тип пути камеры**: тип объекта пути камеры. Может принимать одно из следующих значений:
    - **Точка**: камера будет зафиксирована в указанной точке.
    - **Путь**: камера будет двигаться по заданному пути.
  - **Точка камеры**: если для параметра **Тип пути камеры** задано значение **Точка**, отображаются следующие элементы управления:

**Таблица 32: Точка камеры**

| | |
|---|---|
| | Список ранее использованных точек. Вы можете выбрать путь камеры из раскрывающегося списка. |
|  | Нажав на эту кнопку, вы можете указать точку в модели. После выбора пути отображается диалоговое окно:<br><br>**Рисунок 103:** Имя пути<br>• **Имя**: введите имя точки.<br>• **ОК**: нажмите, чтобы подтвердить выбор точки.<br>• **Отмена**: нажмите, чтобы отменить выбор точки. |
| | Нажмите, чтобы удалить точку из списка. |

  - **Путь камеры**: если для параметра **Тип пути камеры** задано значение **Путь**, отображаются следующие элементы управления:

**Таблица 33: Путь камеры**

| | |
|---|---|
| | Список ранее использованных путей. Вы можете выбрать точку камеры из раскрывающегося списка. |
|  | Нажав на эту кнопку, вы можете указать путь в модели. После выбора пути отображается диалоговое окно:<br><br>**Рисунок 104:** Имя пути<br><br>• **Имя**: введите имя пути<br>• **ОК**: нажмите, чтобы подтвердить выбор точки.<br>• **Отмена**: нажмите, чтобы отменить выбор точки. |
| | Нажмите, чтобы удалить путь из списка. |

- **Тип пути цели**: тип пути цели. Цель определяет точку, в которую камера будет направлена в данный момент. Может принимать одно из следующих значений:
  - **Точка**: цель будет зафиксирована в указанной точке.
  - **Путь**: цель будет двигаться по заданному пути.
- **Точка цели**: если для параметра **Тип пути цели** задано значение **Точка**, отображаются следующие элементы управления:

**Таблица 34: Точка цели**

| | |
|---|---|
| | Список ранее использованных точек. Вы можете выбрать путь камеры из раскрывающегося списка. |
|  | Нажав на эту кнопку, вы можете указать точку в модели. После выбора пути отображается диалоговое окно:<br><br>**Рисунок 105:** Имя пути<br><br>• **Имя**: введите имя точки.<br>• **ОК**: нажмите, чтобы подтвердить выбор точки.<br>• **Отмена**: нажмите, чтобы отменить выбор точки. |
| | Нажмите, чтобы удалить точку из списка. |

- **Путь цели**: если для параметра **Тип пути цели** задано значение **Путь**, отображаются следующие элементы управления:

**Таблица 35: Путь цели**

| | |
|---|---|
| | Список ранее использованных путей. Вы можете выбрать точку камеры из раскрывающегося списка. |
|  | Нажав на эту кнопку, вы можете указать путь в модели. После выбора пути отображается диалоговое окно:<br><br>**Рисунок 106:** Имя пути<br>• **Имя**: введите имя пути<br>• **ОК**: нажмите, чтобы подтвердить выбор точки.<br>• **Отмена**: нажмите, чтобы отменить выбор точки. |
| | Нажмите, чтобы удалить путь из списка. |

***Прим.:*** По аналогии с *ANIPATH*, все выбранные точки и пути сохраняются в чертеже.

### 21.4.12.5 Воспроизведение анимации

Используйте элементы управления воспроизведением анимации для предварительного просмотра анимации.

- Чтобы **начать воспроизведение** анимации, нажмите кнопку **Воспроизвести**.

  Воспроизведение начнется с текущей позиции курсора.
- Чтобы **приостановить** воспроизведение анимации, нажмите кнопку **Пауза**.
- Чтобы **остановить** воспроизведение анимации и переместить курсор в начало, нажмите кнопку **Стоп**.

Вы можете перейти к следующей или предыдущей последовательности анимации (в хронологическом порядке) с помощью кнопок **Переход к следующей последовательности** и **Переход к предыдущей последовательности**.

***Прим.:***

- Во время воспроизведения параметры сцены и положение камеры изменяются непосредственно на чертеже. Для сложных моделей пересчет модели может занять значительное время. В целях экономии времени вы можете использовать курсор для просмотра ключевых моментов анимации.

- Во время воспроизведения камера будет перемещаться в соответствии с установленной последовательностью камер. Если последовательности камер отсутствуют, будет использоваться текущий вид. Конечное видео будет таким же, как в текущем видовом окне.
- По окончании воспроизведения модель автоматически вернется в состояние до начала анимации. В некоторых случаях это может привести к тому, что модель будет находиться в состоянии, отличном от того, на которое указывает положение курсора. Чтобы обновить модель в соответствии с текущей позицией курсора, используйте кнопку .

### 21.4.12.6 Визуализация

1 Нажмите кнопку **Визуализация** для визуализации анимации как 2D видео.

Откроется диалоговое окно *Визуализация*:

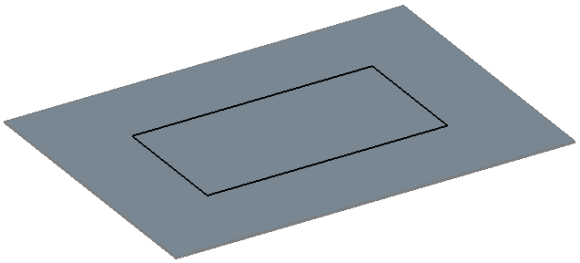


**Рисунок 107:** Визуализация

2 Выполните настройку параметров визуализации:
- **Частота кадров (fps)**: Частота кадров.
- **Разрешение**: Разрешение видео.
- **Формат файла**: Формат видео (поддерживаются форматы .avi, .mpg и .wmv ).
- **Время начала**: Время начала визуализации.
- **Время конца**: Время окончания визуализации.

3 Нажмите кнопку **Визуализация** для запуска процесса визуализации.

Откроется диалоговое окно Сохранить ролик как:

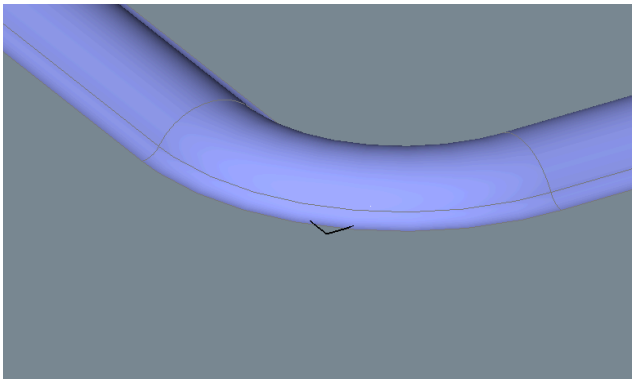


**Рисунок 108:** Сохранить ролик как

4 Укажите расположение и имя выходного файла.

5 Нажмите кнопку **Сохранить** для начала визуализации.

В окне предварительного просмотра будет отображается прогресс выполнения операции.

### 21.4.12.7 Добавление новой сцены

1 Нажмите кнопку **Добавить сцену**.

Откроется диалоговое окно Добавить сцену:

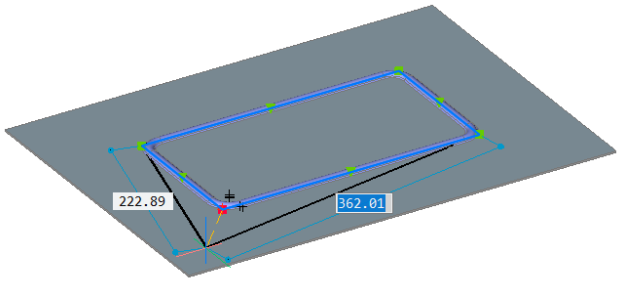



**Рисунок 109:** Добавить сцену

2 Укажите имя сцены.

3 Укажите длительность сцены в секундах.

4 Нажмите кнопку **Добавить сцену**.

### 21.4.12.8 Удаление сцены

1 Выберите сцену с помощью **селектора сцены**.

2 Нажмите кнопку **Удалить сцену**.

### 21.4.12.9 Добавление анимации параметра

1 Добавьте необходимый параметр на временную шкалу:

- Выберите параметр с помощью **селектора параметра**.
- Используя кнопку **Добавить временную шкалу** добавьте новую временную шкалу.

2 Установите курсор в начальную позицию на **временной шкале**.

3 Для добавления новой последовательности анимации нажмите кнопку **Добавить последовательность**

или дважды щелкните на временной шкале в том месте, где необходимо добавить новую последовательность.

4 Установите длительность последовательности путем перетягивания ее границ

или используйте **Конфигуратор**, чтобы установить время начала, время окончания или длительность в секундах.

5 Используйте **Конфигуратор**, чтобы установить начальные и конечные значения для параметра.

### 21.4.12.1 Добавление перемещения камеры

**0**

1 При необходимости добавьте временную шкалу камеры, выбрав **Камера** в **селекторе объектов**.

2 Установите курсор в начальную позицию на **временной шкале**.

3 Для добавления новой последовательности камеры нажмите кнопку **Добавить последовательность**

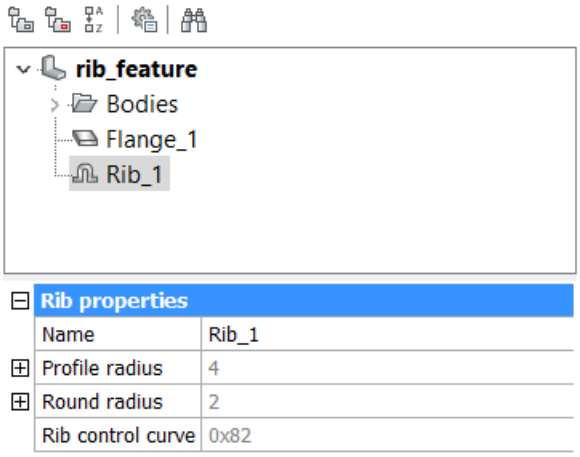
или дважды щелкните на временной шкале в том месте, где необходимо добавить новую последовательность.

4 Установите длительность последовательности путем перетягивания ее границ

или используйте **Конфигуратор**, чтобы установить время начала, время окончания или длительность в секундах.

5 Используя **Конфигуратор**, задайте путь камеры:

- Выберите **Тип пути камеры**:
- Точка, чтобы зафиксировать положение камеры
- **Путь**, чтобы камера перемещалась вдоль заданного пути.
- Используйте свойство Точка камеры для выбора точки или **Путь камеры**, чтобы выбрать путь.
- Выберите свойство для редактирования, затем используйте кнопку




, чтобы указать путь или точку на модели

или выберите одну из предыдущих точек или путей из раскрывающегося списка.

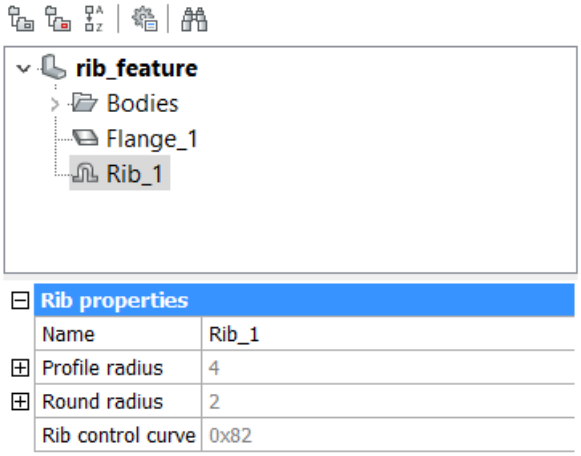
6 Используя **Конфигуратор**, задайте путь цели. **Цель** это точка, на которой фокусируется камера.

Для выбора **Типа пути цели** выполните одно из следующих действий:

- Укажите, что цель должна быть фиксированной.
- **Путь**, чтобы камера перемещалась вдоль заданного пути.

Используйте свойство **Точка цели** для выбора точки или **Путь цели** для выбора пути:

Выберите свойство для редактирования, нажмите




для выбора точки или пути в модели, или выберите одну из предыдущих точек или путей из выпадающего списка.

### 21.4.12.1 Удаление последовательности

**1**

1 Щелкните правой кнопкой мыши на последовательности.

2 В открывшемся контекстном меню выберите **Удалить**.

Альтернативный вариант:

1 Щелкните левой кнопкой мыши на последовательности.

2 Нажмите клавишу **Удалить**.

### 21.4.12.1 Включение/отключение временной шкалы

**2**

1 Проверьте состояние значка временной шкалы. Если значок имеет вид , временная шкала отключена. Если значок имеет вид , временная шкала включена.

2 Для включения/отключения временной шкалы щелкните на этом значке.

Последовательности с отключенной временной шкалой в анимацию не включаются.

### 21.4.12.1 Анимация покомпонентного вида

**3**

Так как в режимах Редакторование блоков и Редактирование ссылок редактор анимации не поддерживается, анимация покомпонентных видов, доступна только в пространстве модели.

1 Вставьте покомпонентный вид в пространство модели.

2 При необходимости, включите/выключите покомпонентный вид и модель, если вы хотите показать или скрыть их во время анимации.

3 Выберите объект покомпонентного вида в **Селекторе объектов**.

4 Нажмите кнопку , чтобы добавить временную шкалу.

5 Для добавления новой последовательности покомпонентного вида нажмите кнопку **Добавить последовательность**

или дважды щелкните на временной шкале в том месте, где необходимо добавить новую последовательность.

6 Установите длительность последовательности путем перетягивания ее границ

или используйте **Конфигуратор**, чтобы установить время начала, время окончания или длительность в секундах.

7 При необходимости, используйте **Конфигуратор**, чтобы установить начальный и конечный шаги покомпонентного вида или включить опцию автоматического скрытия.

#### 21.4.12.1 Создание визуализации 4

См. раздел Визуализация.

## 21.5 Листовой металл

### 21.5.1 Методы проектирования из листового металла

Модуль Sheet Metal позволяет создавать детали из листового металла и автоматически генерировать их развертки, включая информацию, необходимую для изготовления детали.

С помощью BricsCAD вы можете легко и быстро создавать сложные детали из листового металла, поскольку процесс проектирования отличается от процесса производства. Вы работаете не с плоским листом, который необходимо разрезать и сгибать. а моделируете свою деталь напрямую, создавая твердые тела с помощью инструментов прямого моделирования.

**Видео: Знакомство с BricsCAD Sheet Metal**

**Видео: Разработка детали в BricsCAD Sheet Metal с "нуля"**

Для создания деталей из листового металла в BricsCAD вы можете использовать 4 различных способа (или их комбинацию):

1 Создание базового листа из замкнутого плоского профиля с последующим добавлением к нему стенок:
   - Вытягивание стенок по ребрам базового листа с формированием гибов.
   - Создание частичных стенок с автоматическим формированием технологических вырезов для сгиба.
   - Одновременное вытягивание нескольких стенок на смежных ребрах листа с автоматическим формированием угловых технологических вырезов и стыков между ними.
   - Изгиб части листа для создания новой стенки.

2 Создание комбинации стенок и лофт-гибов, соединяющих два 3D профиля, с использованием

инструмента

Лофт-гиб.

3 Создание комбинации из стенок и гибов из плоского профиля с использованием инструмента

Выдавить полилинию.

4 Создание оболочки из 3D тела с использованием

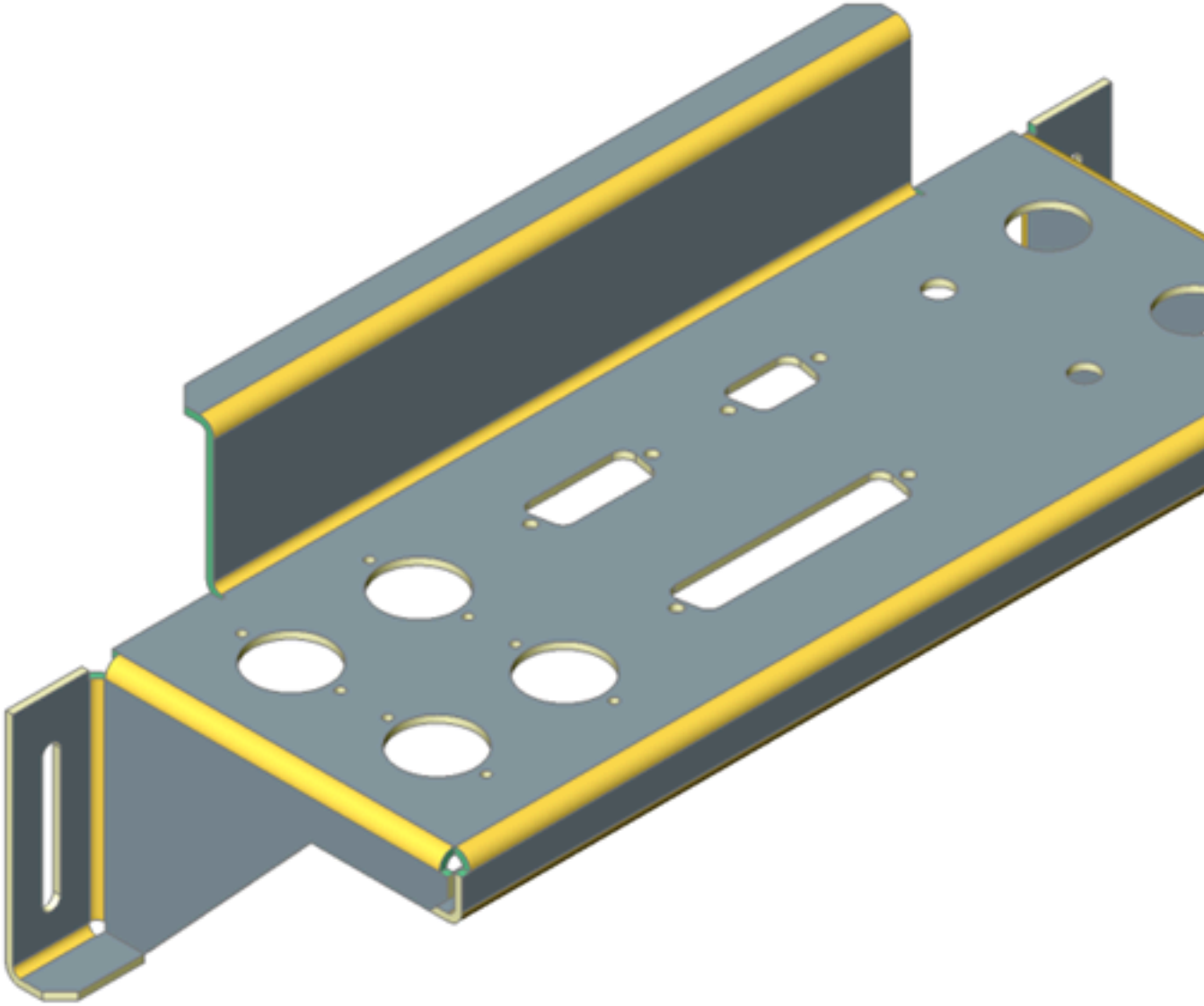
опции Оболочка *Оболочка* команды *SOLIDEDIT* и преобразованием тела в деталь из листового металла с заданной толщиной и созданием гибов и стыков вместо острых кромок с помощью команды *SMCONVERT*.

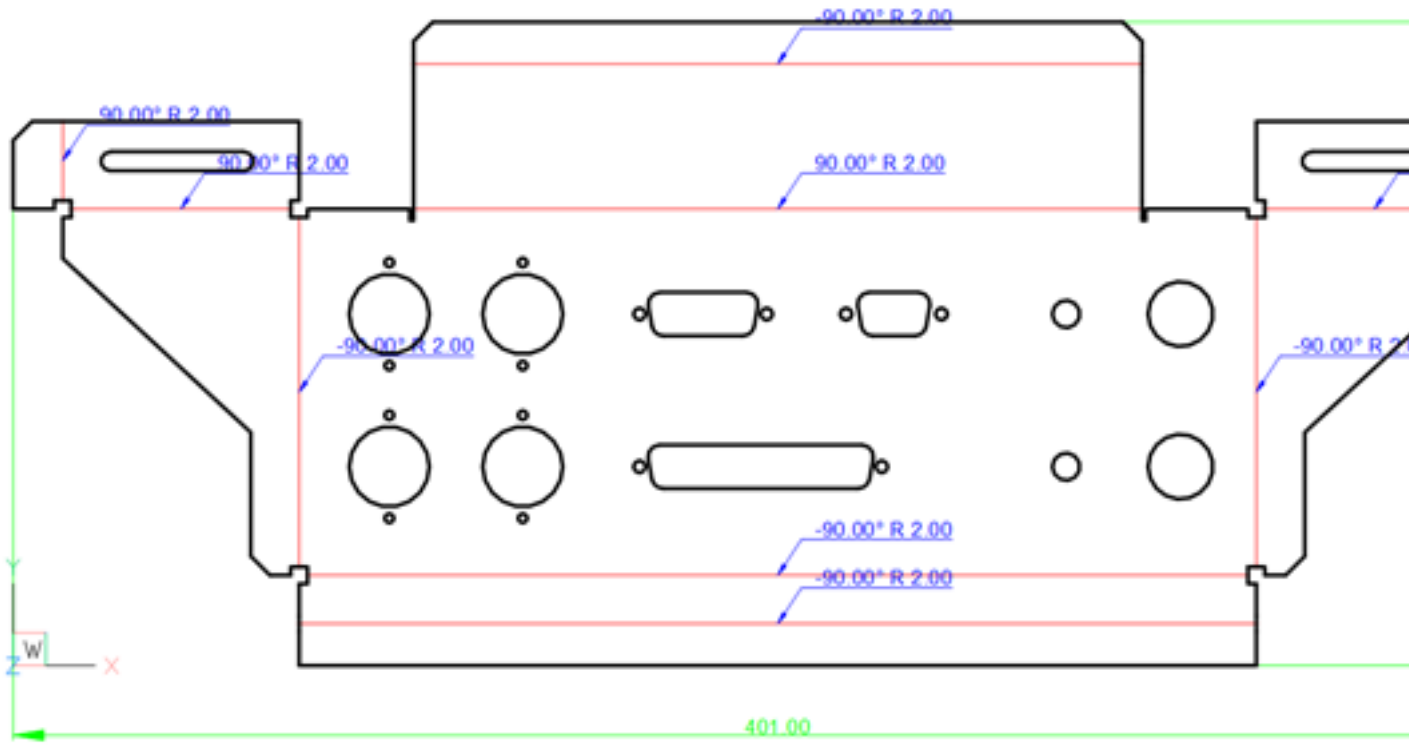
- Выполнение отверстий в детали путем вычерчивания замкнутых контуров на гранях детали и последующего выдавливания этих контуров через стенки.
- Корректировка толщины материала и радиуса гиба с использованием предустановленных параметров.
- Корректировка конструкции детали с применением операций прямого моделирования и 3D зависимостей.
- Автоматическая генерация развертки детали и ее передача в CAM систему путем экспорта соответствующих 2D чертежей с информацией о сгибах в .dwg и .dxf файлом.

Вы также можете выполнять следующие дополнительные операции:

- Выполнение отверстий в детали путем вычерчивания замкнутых контуров на гранях детали и последующего выдавливания этих контуров через стенки.
- Вставка элементов форм из библиотеки. Библиотека параметрических элементов форм устанавливается вместе с BricsCAD Sheet Metal. Вы также можете создавать свои собственные элементы форм.
- Создание ребер жесткости на основе 2D кривой, лежащей на стенке, с помощью команды *SMRIBCREATE*.
- Корректировка толщины материала и радиуса гиба с использованием предустановленных параметров.
- Корректировка конструкции детали с применением операций прямого моделирования и 3D зависимостей.

По окончании разработки детали Вы можете выполнить ее автоматическую развертку и отправить полученный результат в производство путем экспорта соответствующего 2D чертежа.






**Видео: Пакетная обработка в BricsCAD Sheet Metal**

### 21.5.2 Команды и панели инструментов

**Таблица 36: Команды и панели инструментов**

| Значок панели инструментов Листовой металл | Значок выпадающей панели инструментов | Команда | Description |
|---|---|---|---|
| | | *SMFLANGEBASE* | Создает заготовку - базовый лист металла из плоского замкнутого профиля. |
| | | *SMEXTRUDE* | Создает заготовку детали из листового металла выдавливанием полилинии в направлении нормали плоскости кривой. Толщина назначается с учетом внешних размеров полилинии. |

**Таблица 36: Команды и панели инструментов**

| Значок панели инструментов Листовой металл | Значок выпадающей панели инструментов | Команда | Description |
|---|---|---|---|
|  | | *SMLOFT* | Создает заготовку детали из листового металла с лофт-гибами и стенками из двух некомпланарных кривых. |
|  | | *SMCONVERT* | Автоматически распознает стенки, гибы и лофт-гибы в 3D теле. |
|  |  | *SMFLANGEEDGE* | Создает одну или несколько стенок детали листового металла путем вытягивания одного или нескольких торцов существующей стенки. |
| |  | *SMFLANGECONTOUR* | Создает стенку из замкнутого контура и присоединяет ее к существующей модели. |
| |  | *SMFLIP* | Меняет местами грани выбранной стенки. Опционально, позволяет сдвигать стенку по толщине детали из листового металла. |
| |  | *SMFLANGEROTATE* | Выполняет поворот стенки. |
|  |  | *SMHEMCREATE* | Создает одну или несколько замкнутых отбортовок путем вытягивания одного или нескольких торцов существующей стенки. |
| |  | *SMHEMCREATE* | Создает одну или несколько открытых отбортовок путем вытягивания одного или нескольких торцов существующей стенки. |
| |  | *SMHEMCREATE* | Создает одну или несколько каплевидных отбортовок путем вытягивания одного или нескольких торцов существующей стенки. |
| |  | *SMHEMCREATE* | Создает одну или несколько закругленных отбортовок путем вытягивания одного или нескольких торцов существующей стенки. |
|  |  | *BMINSERT* | Выполняет вставку элемента формы. |

**Таблица 36: Команды и панели инструментов**

| Значок панели инструментов Листовой металл | Значок выпадающей панели инструментов | Команда | Description |
|---|---|---|---|
| |  | *SMFORM* | Создает элемент формы на выбранных гранях. |
| |  | *SMFORM* | Разбивает элемент формы на стенки и гибы (если возможно). |
|  | | *SMRIBCREATE* | Создает ассоциативный элемент формы в виде ребра жесткости на грани стенки из плоской 2D кривой. |
|  | | *SMREPLACE* | Заменяет элементы форм в деталях из листового металла (включая распознанные при импорте) на элементы форм из встроенной или пользовательской библиотеки. |
|  |  | *SMBENDCREATE* | Преобразует острые ребра между гранями стенок и стыки кромок стенок в гибы. |
| |  | *SMBENDSWITCH* | Преобразует гибы в лофт-гибы. |
| |  | *SMFLANGEBEND* | Изгибает существующую стенку вдоль указанной линии с учетом значения K-фактора для заданного радиуса гиба. |
|  |  | *SMJUNCTIONCREATE* | Преобразует кромки стенок в стыки. |
| |  | *SMJUNCTIONSWITCH* | Заменяет симметричный стыковой элемент стыком с наложением граней. |
| |  | *SMFLANGECONNECT* | Замыкает зазоры между двумя произвольно ориентированными стенками. |
|  |  | *SMTABCREATE* | Преобразует существующий стык в одиночный шип-паз. |
| |  | *SMTABCREATE* | Преобразует существующий стык в массив шип-пазов. |

**Таблица 36: Команды и панели инструментов**

| Значок панели инструментов Листовой металл | Значок выпадающей панели инструментов | Команда | Description |
|---|---|---|---|
|  |  | *SMRELIEFCREATE* | Формирует угловые вырезы и подрезы гиба. Угловые вырезы создаются в углах, в которых сходятся три и более смежные стенки. Подрезы гиба создаются в начале и конце участка изгиба стенки. |
| |  | *SMRELIEFSWITCH* | Преобразует угловой вырез в круговой угловой вырез. |
| |  | *SMRELIEFSWITCH* | Преобразует угловой вырез в прямоугольный угловой вырез. |
| |  | *SMRELIEFSWITCH* | Преобразует угловой вырез в V-образный угловой вырез. |
| |  | *SMRELIEFSWITCH* | Преобразует подрез гиба в плавный подрез гиба. |
| |  | *SMRELIEFSWITCH* | Преобразует подрез гиба в скругленный подрез гиба. |
| |  | *SMRELIEFSWITCH* | Преобразует подрез гиба в прямой подрез гиба. |
|  |  | *SMSPLIT* | Разделяет стенку вдоль линии, построенной на ее грани. |
| |  | *SMSPLIT* | Разделяет стенку вдоль линии, проходящей через угловую точку. |
| |  | *SMSPLIT* | Разделяет стенку вдоль линии, проходящей через угловую точку, и автоматически применяет это разделение в аналогичных точках. |
|  | | *SMIMPRINT* | Разделяет торцевые грани детали из листового металла на смежные стенки и гибы. |

**Таблица 36: Команды и панели инструментов**

| Значок панели инструментов Листовой металл | Значок выпадающей панели инструментов | Команда | Description |
|---|---|---|---|
|  | | *SMDISSOLVE* | Удаляет все свойства листового металла для выбранных граней или объектов. |
|  | | *SMDELETE* | Удаляет гибы и соединения с восстановлением острой кромки между двумя стенками; удаляет стенку вместе со всеми примыкающими к ней гибами. Примыкающие стенки удлиняются в соответствии с конфигурацией соединения. Удаляет скосы кромок путем восстановления исходной геометрии. |
|  |  | *SMREPAIR* | Исправляет проблемы деталей из листового металла. Исправляет проблемы деталей с лофт-элементами: объединяет смежные сгибы и обеспечивает их соединение со стенками по касательной. |
| |  | *SMREPAIR* | Исправляет проблемы деталей из листового металла. Применяется только к выбранным стенкам и гибам. |
|  |  | *SMSELECT* Подобные элементы форм | Выбирает все одинаковые элементы форм в модели детали из листового металла. |
| |  | *SMSELECT* Острые ребра | Выбирает все острые ребра детали из листового металла. |
| |  | *SMSELECT* Неортогональные торцевые ребра | Выбирает неортогональные торцевые ребра детали из листового металла. |
| |  | *SMSELECT* Ребра грани | Выбирает ребра плоской грани детали из листового металла. |
| |  | *SMSELECT* Торцевая грань | Выбирает торцевую грань детали из листового металла. |

**Таблица 36: Команды и панели инструментов**

| Значок панели инструментов Листовой металл | Значок выпадающей панели инструментов | Команда | Description |
|---|---|---|---|
|  | | *SMPARAMETRIZE* | Создает набор 3D зависимостей для выбранной детали из листового металла. |
|  |  | *SMUNFOLD* | Создает развертку детали из листового металла. |
| |  | *SMUNFOLD* Ассоциативная | Открывает дополнительное окно с разверткой модели детали из листового металла. |
| |  | *SMEXPORTOSM* | Выполняет экспорт твердотельной детали из листового металла в файл формата .osm (Open Sheet Metal). |
| |  | *SMEXPORT2D* | Выполняет экспорт развертки детали из листового металла в виде 2D профиля в .dxf или .dwg файл. |
| |  | *SMASSEMBLYEXPORT* | Выполняет пакетный экспорт компонентов сборки для получения их развертки и создания .dxf файла для каждого 3D тела, распознанного как деталь из листового металла. |
|  | | *FEATURECOLORS* | Переключает значение системной переменной *FEATURECOLORS*. |

Инструменты модуля Sheet Metal доступны:

- В **квадро-меню** курсора (в зависимости от элемента, находящегося под курсором):

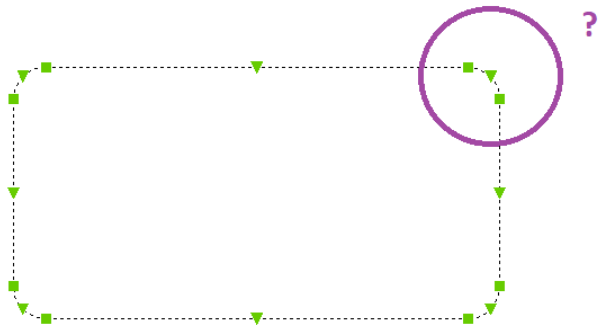
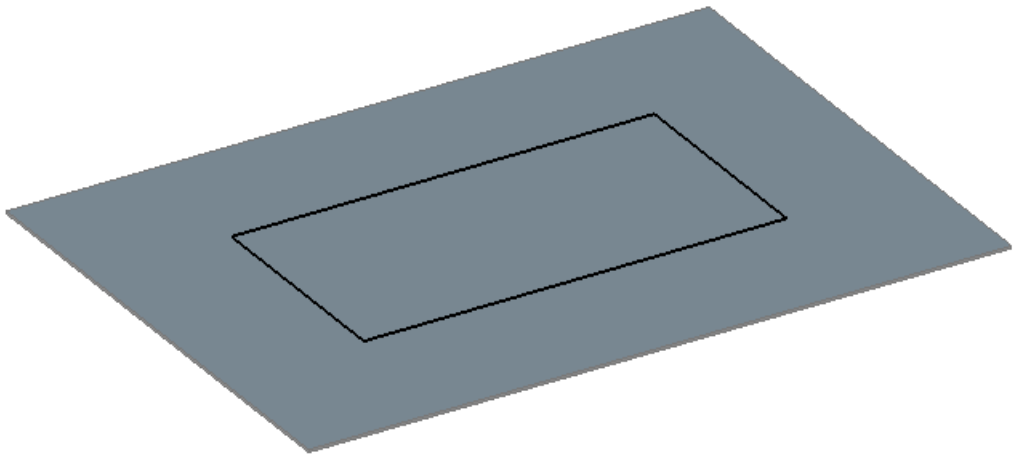
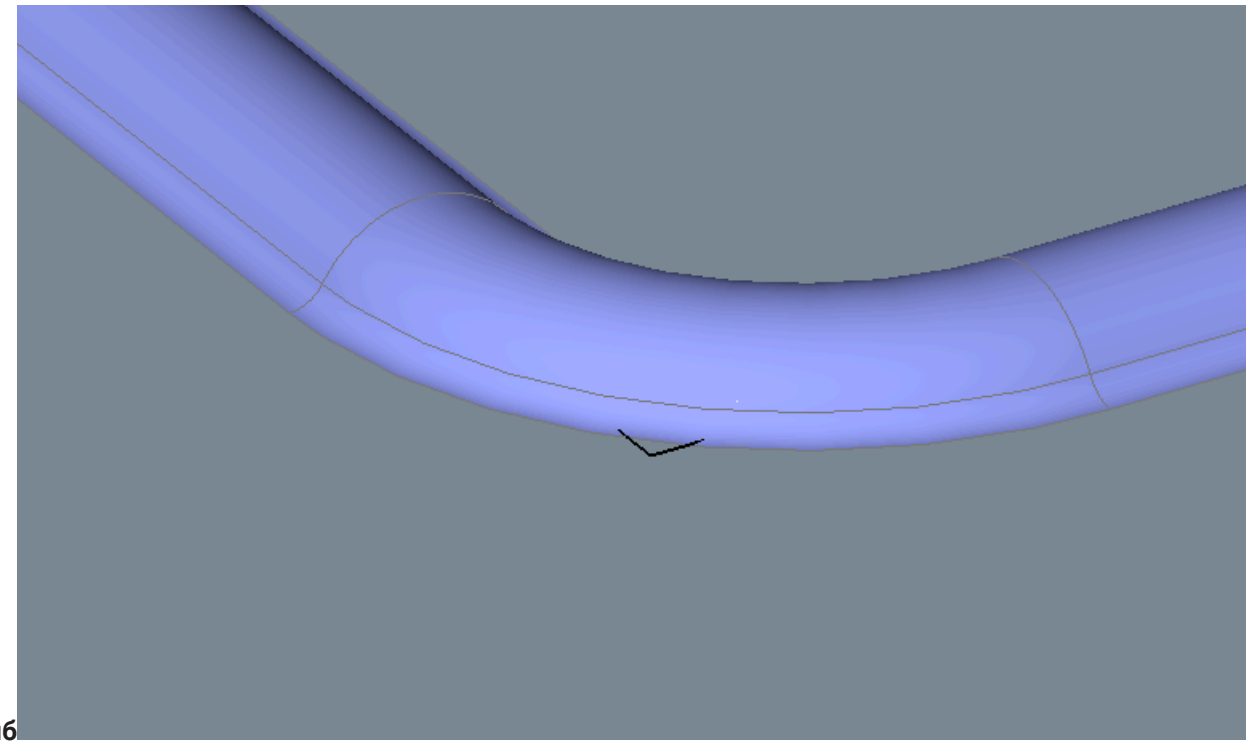



**Рисунок 110:** Квадро-меню курсора

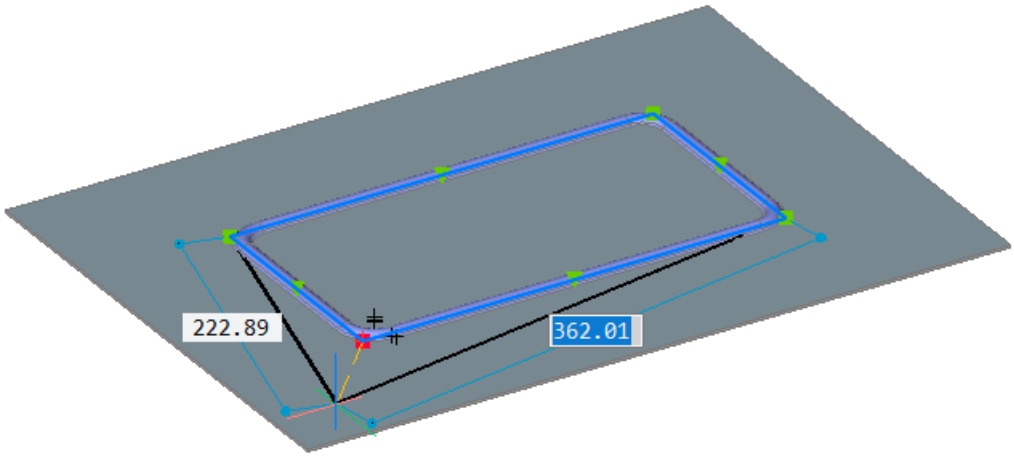
- На панелях инструментов модуля **Sheet Metal**:
  - **Листовой**



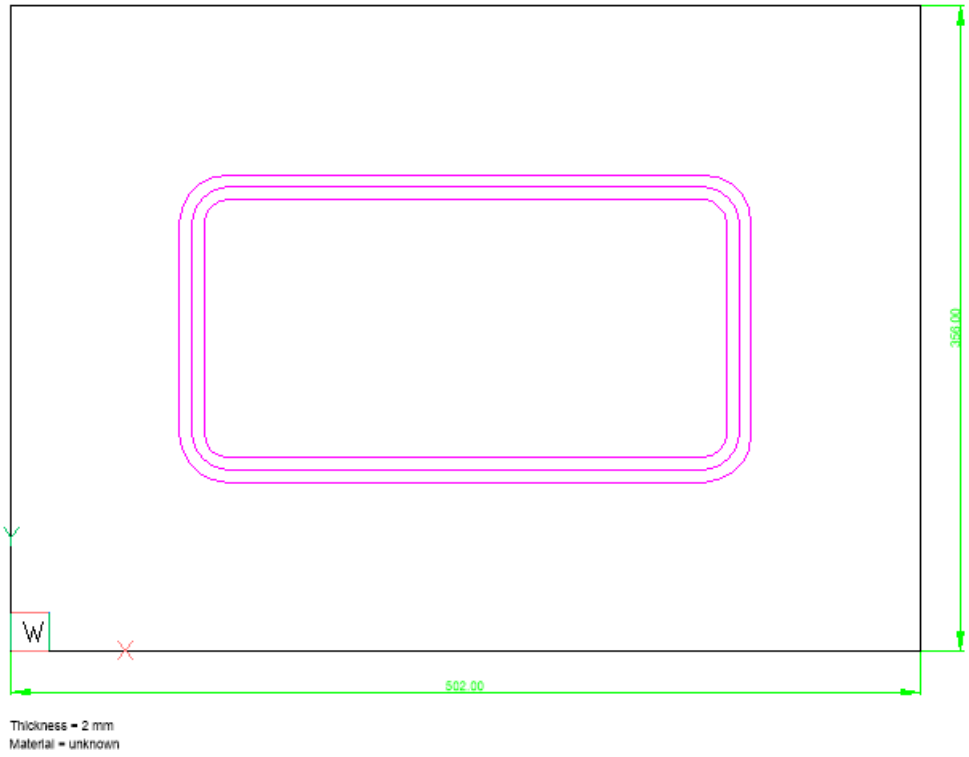
**металл**



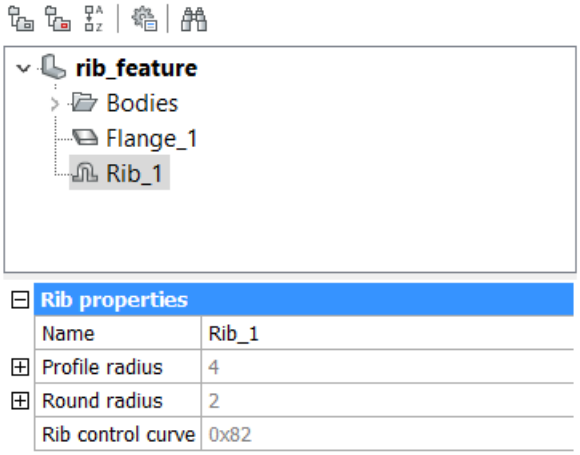
- **Гиб**
- **Лист**
- **Форма**




- **Отбортовка**




- **Стык**
- **Подрез**
- **Восстановить деталь**
- **Рассечь**
- **Шип-паз**




- **Развертка**

- В меню **Листовой металл** (только для рабочего пространства **Mechanical**):

**Рисунок 111:** Меню Листовой металл

- На закладке ленты **Листовой металл** рабочего пространства




**Mechanical**.

См. также Прямое моделирование инструмент.

### 21.5.3 Элементы деталей из листового металла

#### 21.5.3.1 Обзор

В BricsCAD элементы 3D модели детали из листового металла представляют собой интеллектуальные наборы граней. Для каждого элемента устанавливаются определенные пространственные и параметрические связи между его гранями и несколькими смежными гранями. Эти элементы позволяют использовать функцию распознавания намерений проектирования. Элементы создаются автоматически в зависимости от применяемой вами геометрической операции. На приведенном ниже рисунке показаны примеры элементов детали из листового металла, используемые в BricsCAD:

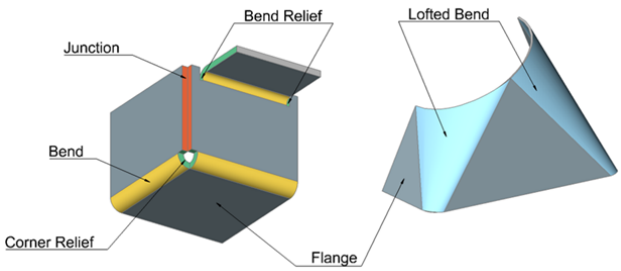



**Рисунок 112:** Пример 1



**Рисунок 113:** Пример 2

### 21.5.3.2 Стенка (Лист)

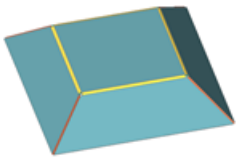
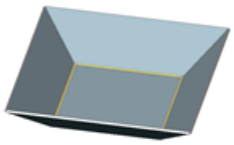
Главной особенностью любой детали из листового металла является наличие стенки, которая состоит из двух параллельных плоских граней, расположенных друг напротив друга (с некоторым объемным пространством между ними) на расстоянии, которое значительно меньше линейного размера граней. Это расстояние равно толщине материала. При изменении модели толщина поддерживается автоматически. Другие грани, которые примыкают к граням стенки и не относятся к гибам, называются торцевыми гранями. Они всегда перпендикулярны граням стенки.

Для создания стенок используются команды: *SMFLANGEBASE*, *SMEXTRUDE*, *SMLOFT*, *SMFLANGEEDGE*, *SMFLANGEBEND* и *SMCONVERT*.

Если для системной переменной *FEATURECOLORS* установлено значение *ON*, цвет базовой грани стенки отличается от цвета противоположной грани этой же стенки. Как правило, изменение параметров детали из листового металла, например, толщины листа, не приводит к перемещению базовой грани. Команда *SMFLIP* меняет местами базовую и противоположную ей грани выбранной стенки и позволяет, при необходимости, сдвигать стенку по толщине детали из листового металла.

На приведенном ниже рисунке показано цветовое выделение граней стенок одной и той же детали.

**Таблица 37: Лист**

| | |
|---|---|
|  |  |
| Базовые грани | Другие грани |

### 21.5.3.3 Гиб

Гиб соединяет две стенки. Он состоит из двух соосных цилиндрических поверхностей, касательных к примыкающим к ним плоским поверхностям стенок.

Для создания гибов используются команды: *SMFLANGEBASE*, *SMEXTRUDE*, *SMFLANGEEDGE*, *SMFLANGEBEND* и *SMCONVERT*.

### 21.5.3.4 Гиб с нулевым радиусом

Это конструктивная концепция гиба, геометрия которого имеет цилиндрическую грань с внешней стороны гиба с радиусом, равным толщине. Внутренняя грань гиба представляет собой ребро. Такая геометрия формирует гиб с нулевым радиусом, который также используется при проектировании. Команда *SMUNFOLD* поддерживает гибы такого типа и генерирует размеры, как если бы эти гибы были обычными гибами с радиусом, равным толщине.

### 21.5.3.5 Некорректный гиб

Некорректные гибы распознаются командой *SMCONVERT*. Как и обычные гибы, некорректные гибы соединяют стенки, но имеют ошибки в геометрии, например, несоответствие осей гиба, некасательное соединение гиба со смежной стенкой. Эти ошибки не позволяют использовать такие гибы как обычные. Перед получением развертки детали все некорректные гибы должны быть исправлены с помощью команды *SMREPAIR* или удалены с помощью команды *SMDELETE* и созданы заново с помощью команды *SMBENDCREATE*.

### 21.5.3.6 Лофт-гиб

Элемент лофт-гиб состоит из двух линейчатых поверхностей, расположенных друг напротив друга (со смещением, равным толщине материала), между которыми имеется некоторый сплошной объем. Линейчатая поверхность может быть описана как набор точек, перемещаемых по прямой линии. Примерами являются цилиндры, конусы, спирали. Лофт-гибы могут быть созданы путем придания толщины линейчатым поверхностям. Лофт-гиб может соединяться с одной или двумя стенками или быть автономным, без стенок. Также допускается комбинированное соединение, состоящее из обычного гиба и лофт-гиба.

Различия между гибами и лофт-гибами:

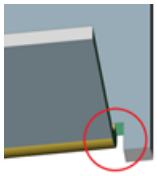
**Таблица 38: Отличия**

| | Гибы | Лофт-гибы |
|---|---|---|
| Поверхность | Цилиндрическая | Цилиндрическая, коническая или сплайн |
| Развертка | Точная | Развертка лофт-гибов является приблизительной и контролируется регулируемым количеством сегментов гиба. Чем больше сегментов гиба, тем точнее будет реальная деталь и тем лучше она будет соответствовать первоначальному проекту.<br>Цилиндрические и конические лофт-гибы имеют опцию создания точной аналитической развертки. |

Для создания элемента Лофт-гиб используются команды *SMLOFT* и *SMCONVERT*.

### 21.5.3.7 Подрез гиба

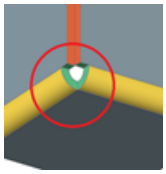
Технологический подрез в виде группы граней формируется между двумя стенками разной ширины, соединенных элементом гиба. Ширина подреза определяет расстояние между двумя противоположными гранями выреза.



Для создания элемента Подрез гиба используются команды *SMFLANGEEDGE*, *SMBENDCREATE* и *SMRELIEFCREATE*.

### 21.5.3.8 Угловой вырез

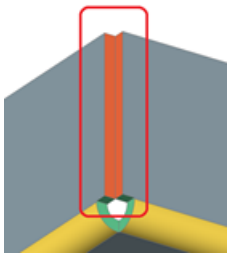
Угловой вырез формируется группой граней выреза в углу, где сходятся три или более стенки. Этот вырез необходим для предотвращения нежелательной деформации материала при изгибе детали из листового металла. Свойства элемента Угловой вырез определяют форму и размер этого выреза.



Для создания угловых вырезов используются команды *SMFLANGEEDGE*, *SMBENDCREATE* and *SMRELIEFCREATE*.

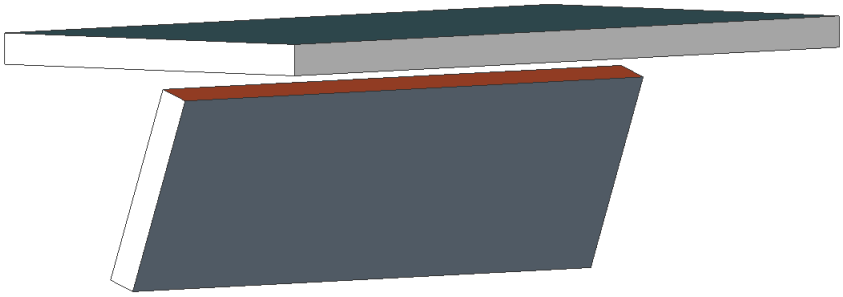
### 21.5.3.9 Стыки и Т-Стыки

Стык образуется торцевыми гранями двух смежных стенок, не соединенных между собой гибом. Стык может быть симметричным или с наложением граней. Свойства элемента Стык определяют тип стыка и минимальное расстояние между стенками, которое называется зазором стыка.



Т-Стык формируется между торцевой гранью одной стенки и гранью другой стенки. Обычно Т-Стык используется для последующего создания Т-образного соединения Шип-паз.

На приведенном ниже рисунке для наглядности показан большой зазор.

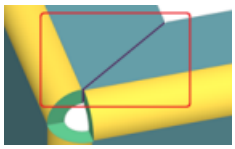


Для создания элемента Стык используются команды *SMFLANGEEDGE*, *SMJUNCTIONCREATE* и *SMFLANGECONNECT*.

Для создания элемента Т-Стык используются команды *SMJUNCTIONCREATE* и *SMFLANGECONNECT*.

### 21.5.3.10 Элемент Скос

Элемент Скос используется для стыковки компланарных стенок, созданных с помощью команды *SMFLANGEEDGE* и *SMSPLIT* . Свойства элемента Скос определяют зазор между торцевыми гранями стыкуемых стенок.



### 21.5.3.11 Элементы формы и жесткости

Элементы формы и жесткости представляют собой набор различных форм, получаемых при выполнении операции штамповки и прокатки.

Элементы форм имеют два представления: в 3D и в виде 2D символа для развертки. При импорте геометрии 2D символ формируется как проекция 3D граней элемента форм. При вставке/замене элемента форм из библиотеки в качестве символа для развертки используется символ из библиотеки.

Элемент жесткости представляет собой рельефный оттиск на листе, выполненный роликом, перемещающимся по заданной траектории.

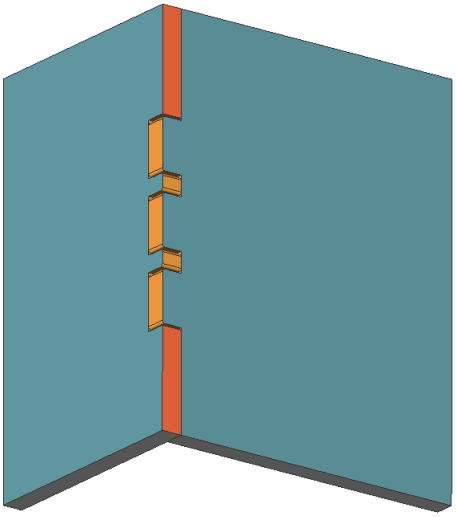
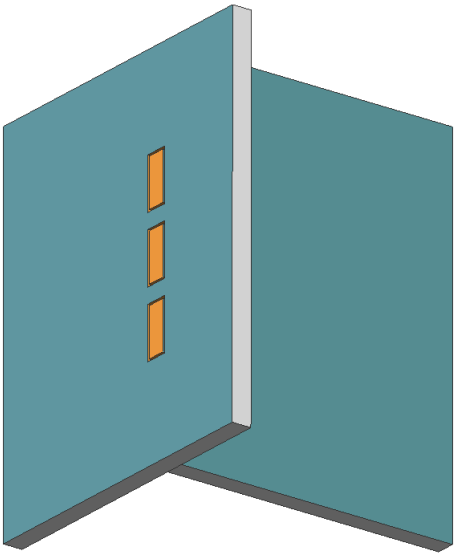
Для создания элементов формы и жесткости используются команды *BMINSERT*, *SMRIBCREATE* и *SMCONVERT*.

### 21.5.3.12 Шип-паз и Т-образный шип-паз

Оба элемента используются для соединения двух листов и ограничивают их взаимное смещение.

Для создания элементов Шип-паз и Т-образный шип-паз используется команда *SMTABCREATE*.

**Таблица 39: Шип-паз и Т-образный шип-паз**

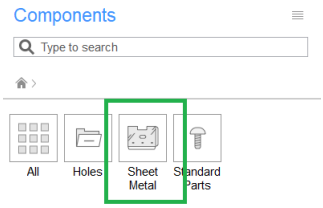
| | |
|---|---|
| <br>**Рисунок 114:** Пример шип-паза | <br>**Рисунок 115:** Пример Т-образного шип-паза |
| Элемент Шип-паз | Элемент Т-образный шип-паз |



### 21.5.3.13 Фаска

Элемент Фаска представляет собой скос на торцевой грани детали из листового металла. Фаска формируется наклонными и ортогональными гранями. Элемент Фаска может принадлежать только одной стенке и не может примыкать к гибу.

Для распознавания или удаления элементов Фаска может быть использована команда *SMCONVERT*. Эти элементы поддерживаются командой *SMUNFOLD* .




### 21.5.4 Цветовое выделение элементов

Для облегчения распознавания элементов геометрии детали из листового металла каждому из этих элементов назначается определенный цвет. Этот цвет используется только при работе с моделью и не отображается на генерируемых документах, видах пространства листа и т.д.

Цвет элементов определяется значением системной переменной *FEATURECOLORS* и набора системных переменных, каждая из которых управляет цветом отдельного элемента. Используемые по умолчанию цвета были тщательно отобраны для обеспечения их наилучшей комбинации, но при необходимости вы можете изменить их в группе настроек **Листовой металл** диалогового окна *Параметры*.

**Рисунок 116:** Листовой металл

Чтобы включить/выключить отображение цветов элементов:

- Нажмите кнопку **Цвета элементов** () на панели инструментов **Листовой металл**.
- Нажмите кнопку **Цвета элементов** () на панели ленты **Листовой металл**.
- Выберите пункт **Цвета элементов** в меню **Листовой металл**.

**Видео: Цветовое выделение элементов в BricsCAD Sheet Metal**

### 21.5.5 Обозреватель конструкции для листового металла

#### 21.5.5.1 Обзор

Панель Обозреватель конструкции открывается автоматически при переключении на рабочее пространство **Mechanical**.

Чтобы открыть панель Обозреватель конструкции вручную, выполните следующие действия:

1 Щелкните правой кнопкой мыши на панели инструментов.

Откроется контекстное меню:

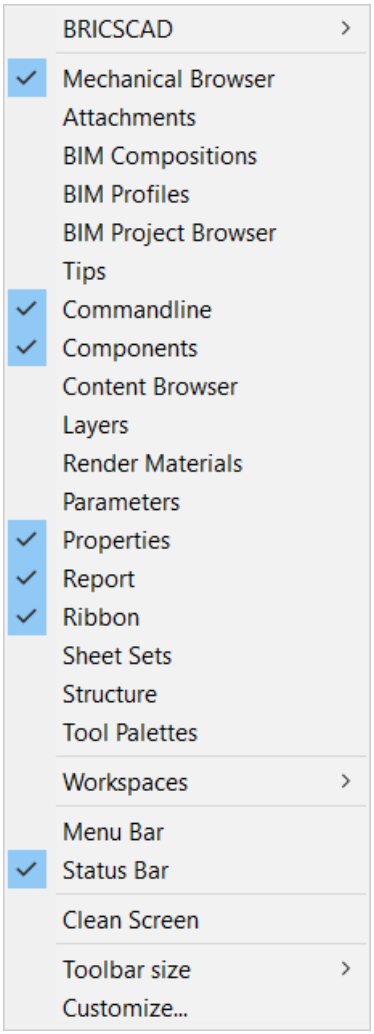



**Рисунок 117:** Контекстное меню

2 Выберите **Обозреватель конструкции**.

### 21.5.5.2 Контекст листового металла

Если вы работаете с деталью из листового металла и создали какой-либо элемент изделия из листового металла, свойства корневого узла в **Обозревателе конструкции** будут содержать раздел «Листовой металл» (также называемый Контекстом листового металла, связанным с документом).

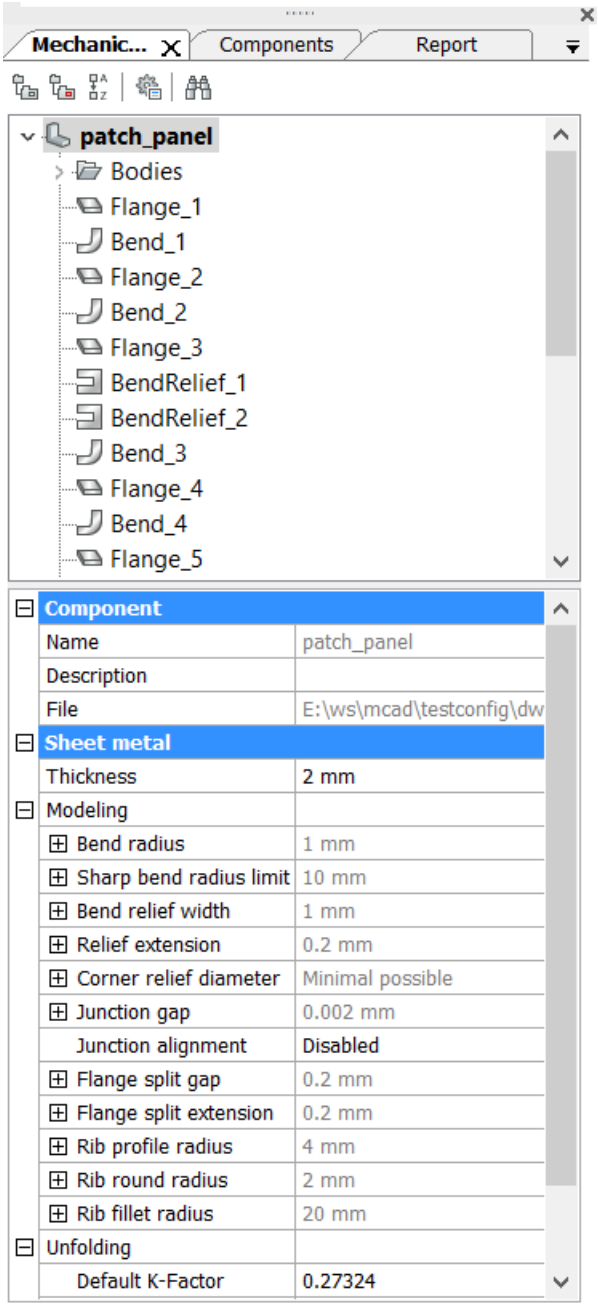



**Рисунок 118:** Обозреватель конструкции

Свойства листового металла представлены в виде сгруппированных полей.

Поле **Толщина** управляет толщиной листа, которая должна быть одинаковой для всех элементов детали из листового металла в документе.

Группа 3D модель играет две роли:

- Набор исходных параметров определяет настройки по умолчанию, используемые при создании новой модели с использованием команд *SMFLANGEBASE*, *SMEXTRUDE*, *SMFLANGEEDGE*, *SMFLANGEBEND*, *SMLOFT* и *SMCONVERT*. Исходные параметры сохраняются в чертеже, что позволяет создавать .dwt шаблоны для различных типов моделей из листового металла.
- Новые элементы, созданные с помощью указанных команд, обычно получают свои свойства из контекста листового металла. Например, для свойства **Тип радиуса** элемента Гиб по умолчанию устанавливается значение Использовать глобальное значение. При изменении **радиуса гиба** в контексте листового металла будет соответственно изменен и радиус всех элементов Гиб. Таким образом, еще одно применение свойств контекста листового металла состоит в том, что они позволяют глобально изменять свойства детали из листового металла.

В группе свойств **Развертка** в контексте листового металла перечислены некоторые свойства для команд *SMUNFOLD*, *SMEXPORT2D* и *SMASSEMBLYEXPORT*.

Контекст листового металла формируется при создании в документе первой детали из листового металла. При его инициализации используются настройки для листового металла, указанные в группе настроек Параметры по умолчанию диалогового окна Параметры. Типовые процессы создания детали из листового металла перечислены в разделе Методы проектирования изделий из листового металла. Для удаления контекста листового металла используется команда *SMDISSOLVE*.

***Прим.:*** Если вы просто выделите все детали из листового металла в документе и примените к ним команду Erase, контекст листового металла будет сохранен, несмотря на то, что в документе не будет твердых тел и элементов из листового металла.

### 21.5.5.3 Работа с элементами

Все элементы детали из листового металла отображаются в списке **Обозревателя конструкции**. При выборе элемента в **Обозревателе конструкции** грани выбранного элемента выделяются в области модели:

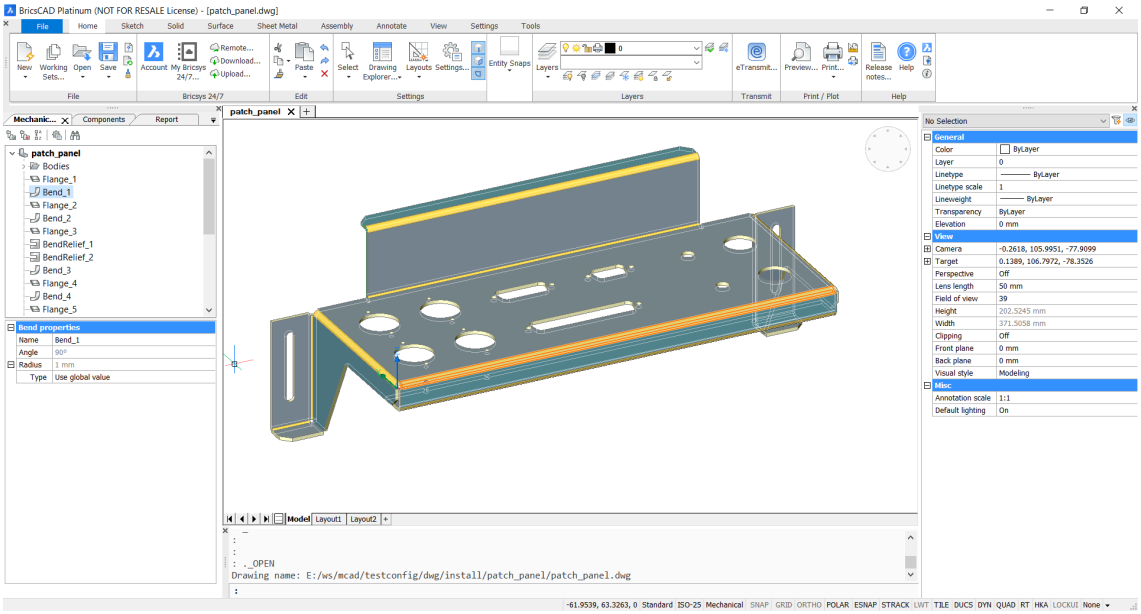



**Рисунок 119:** Обозреватель конструкции

***Прим.:*** Теперь вместо группы свойств **Листовой металл** отображается группа **Свойства гиба**. Таким образом, выбор элемента в Обозревателе конструкции позволяет:

- Найти и выделить геометрию выбранного элемента.
- Выполнить просмотр свойств выбранного элемента.

### 21.5.5.4 Изменение толщины

Чтобы изменить толщину листовой детали, выполните следующие действия:

1 Выберите основной узел детали в **Обозревателе конструкции**.

2 Введите необходимое значение в поле **Толщина**.

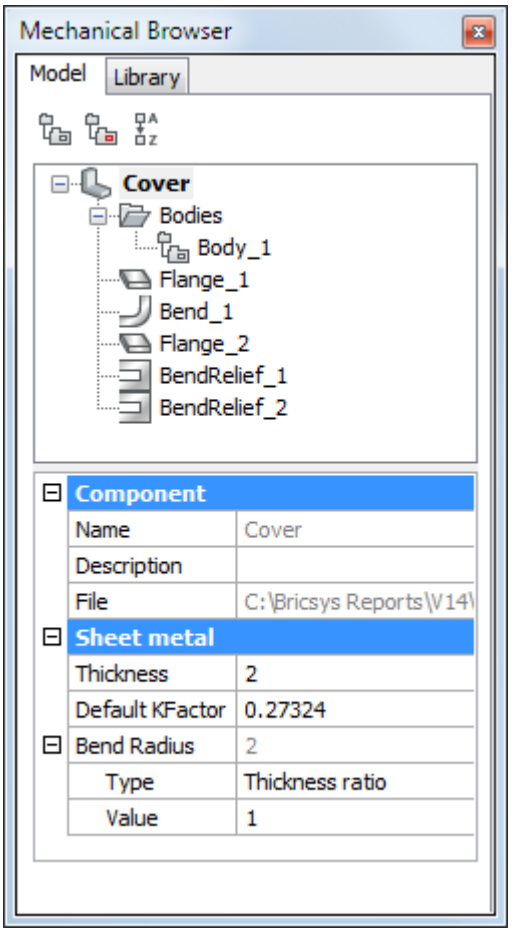
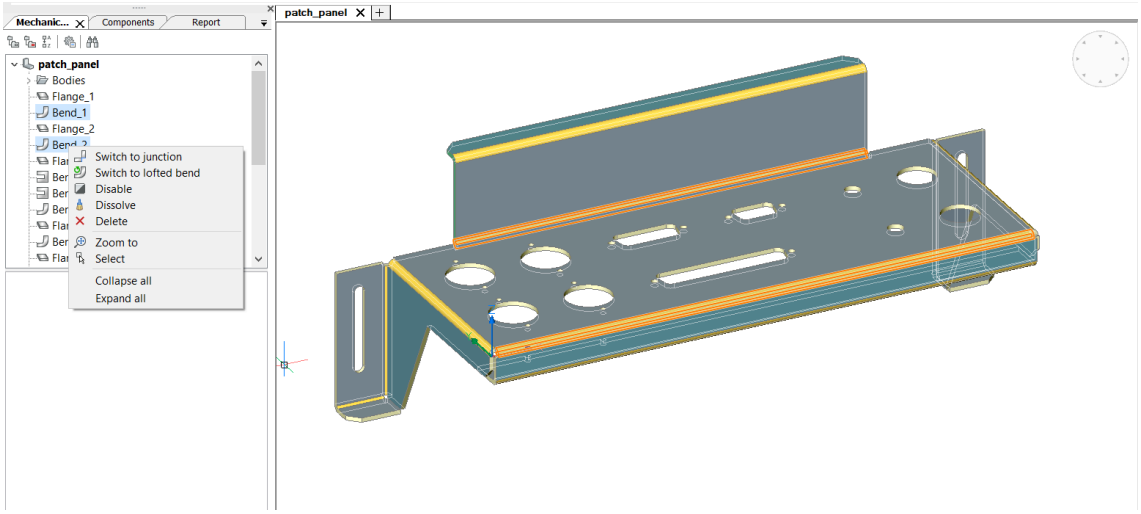



**Рисунок 120:** Толщина

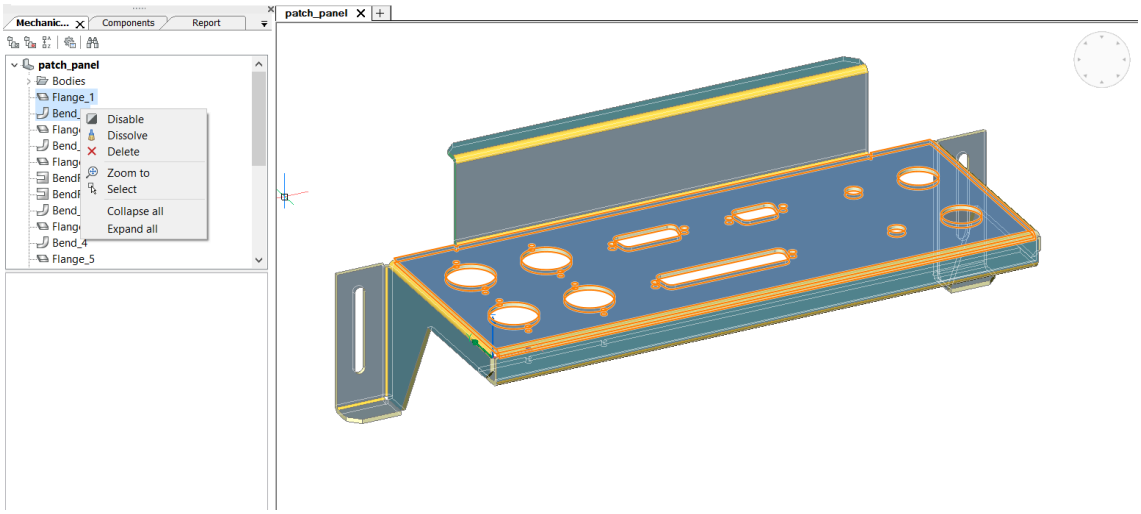
### 21.5.5.5 Операции с элементами

**Обозреватель конструкции** обеспечивает быстрый доступ к операциям над выбранным элементом детали из листового металла. Набор доступных операций зависит от типа выбранного элемента.

При выборе двух гибов вы можете применить к ним операции, доступные для элемента Гиб:




При выборе листа и гиба будут доступны операции, применимые к обоим выбранным элементам:




### 21.5.5.6 Операции Расформировать, Отключить и Удалить

При щелчке правой кнопкой мыши на элементе в **Обозревателе конструкции** и выборе в контекстном меню опции **Расформировать** выбранный элемент исключается из состава детали, но его геометрия сохраняется. При этом пространственные и параметрические зависимости, связанные с гранями этого элемента удаляются.

Если вам необходимо временно "забыть" о выбранном элементе, вы можете выбрать опцию **Отключить**. Элемент останется в **Обозревателе конструкции**, но деталь будет вести себя так, как если бы этот элемент был расформирован. Это удобно тем, что в отличие от расформирования элемента, в этом случае выполняется обновление при изменении геометрии и сохраняется возможность последующего включения элемента без необходимости его повторного создания или распознавания с помощью команды *SMCONVERT*.

Для некоторых элементов деталей из листового металла доступна опция **Удалить**, которая также отображается в контекстном меню элемента в **Обозревателе конструкции** и является аналогом команды *SMDELETE* команды. В этом случае элемент удаляется из браузера, а геометрия изменяется в зависимости от типа выбранного элемента.

### 21.5.6 Свойства деталей из листового металла и их элементов

#### 21.5.6.1 Обзор

Элементы деталей из листового металла и их контекстные меню доступны в **Обозревателе конструкции**. См. Обозреватель конструкции для листового металла для понимания основных принципов. В этой статье описываются свойства и контекстное меню для каждого типа элементов, отображаемых в **Обозревателе конструкции**.

При работе с деталями из листового металла щелчок на корневом узле **Обозревателя конструкции** отображает свойства всего чертежа. Иногда эти свойства называют глобальным контекстом листового металла, связанным с документом: этот контекст по умолчанию используется для всех деталей из листового металла.

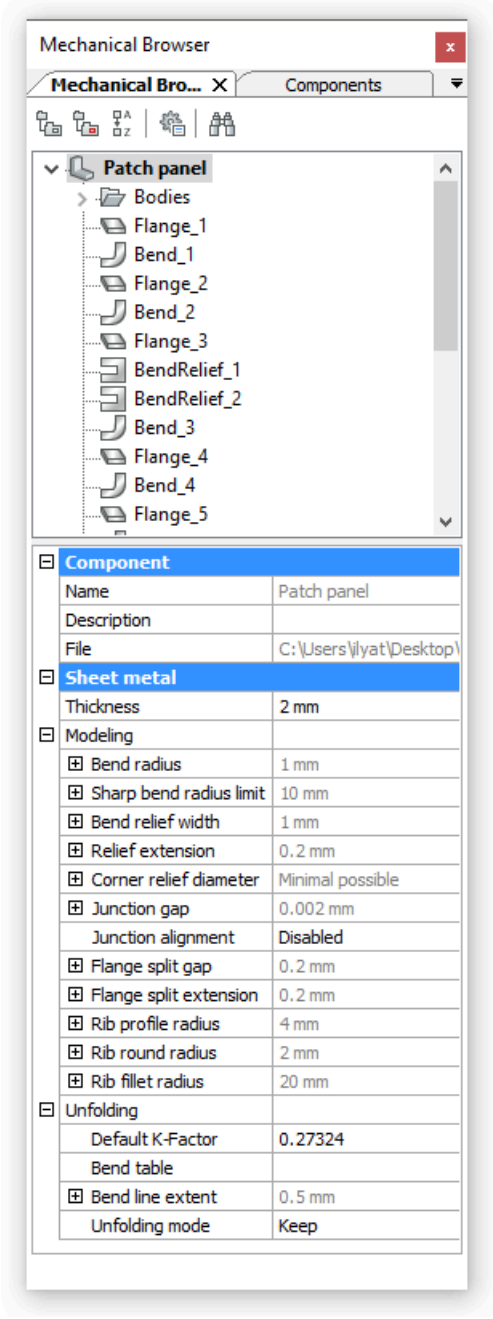



**Рисунок 121:** Обозреватель конструкции

Начиная с BricsCAD V21, появилась возможность назначения локального контекста листового металла для данного твердого тела, что позволяет определять детали из листового металла с различными конструктивными параметрами и, что наиболее важно, изменять толщину детали. До BricsCAD V21 все детали в пространстве модели должны были иметь одинаковую толщину, что заставляло пользователя распределять детали с разной толщиной по разным блокам или компонентам. Также эта новая функция облегчает использование сценариев с несколькими импортированными телами, использующими команды *SMCONVERT* и *SMASSEMBLYEXPORT*. Еще

одно преимущество заключается в том, что локальный контекст листового металла может определять свою собственную Таблицу гибов.

### 21.5.6.2 Переключение контекста на локальный

Выполните следующие действия:

1 Выберите тело в **Обозревателе конструкции**.

2 В разделе Листовой металл переключите **Тип контекста** с **Глобальный контекст** на **Контекст для твердого тела**.

***Прим.:*** Команда *SMCONVERT* в случае нескольких исходных объектов с разной толщиной автоматически переключает некоторые контексты на локальные.

Глобальные свойства листового металла размещены в нижней части **Обозревателя конструкции**.

**Таблица 40: Глобальные свойства**

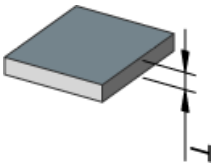
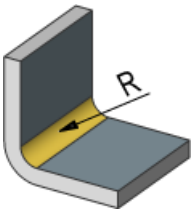
| Глобальные свойства | Description |
|---|---|
| **Толщина** | Глобальная толщина листового металла.<br> |
| **Радиус гиба** | Радиус гиба по умолчанию; может быть изменен для каждого гиба отдельно.<br>• **Тип**: радиус может быть определен с помощью коэффициента толщины или как абсолютное значение.<br>• **Значение**: коэффициент или абсолютное значение.<br> |
| **Предельное значение радиуса гиба** | Максимальный радиус гиба, при котором гиб считается обычным. Используется в командах *SMCONVERT* и *SMRELIEFCREATE* для различия обычных и лофт-гибов.<br>• **Тип**: радиус может быть определен с помощью коэффициента толщины или как абсолютное значение.<br>• **Значение**: коэффициент или абсолютное значение. |

**Таблица 40: Глобальные свойства**

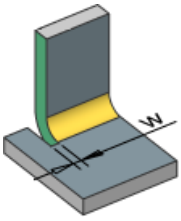
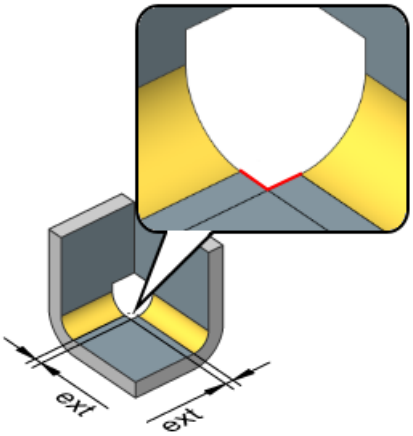
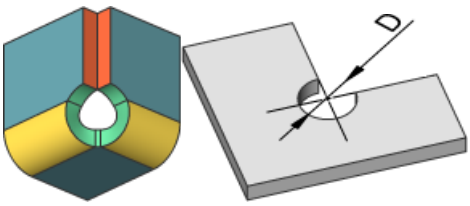
| Глобальные свойства | Description |
|---|---|
| **Ширина подреза гиба** | Ширина технологического подреза для одиночного гиба.<br>• **Тип**: радиус может быть определен с помощью коэффициента толщины или как абсолютное значение.<br>• **Значение**: коэффициент или абсолютное значение.<br> |
| **Заступ подреза** | Значение заступа подреза гиба или углового выреза.<br>• **Тип**: радиус может быть определен с помощью коэффициента толщины или как абсолютное значение.<br>• **Значение**: коэффициент или абсолютное значение.<br> |
| **Диаметр углового выреза** | Диаметр по умолчанию для кругового углового выреза.<br>**Тип**: минимально возможное или абсолютное значение.<br> |

**Таблица 40: Глобальные свойства**

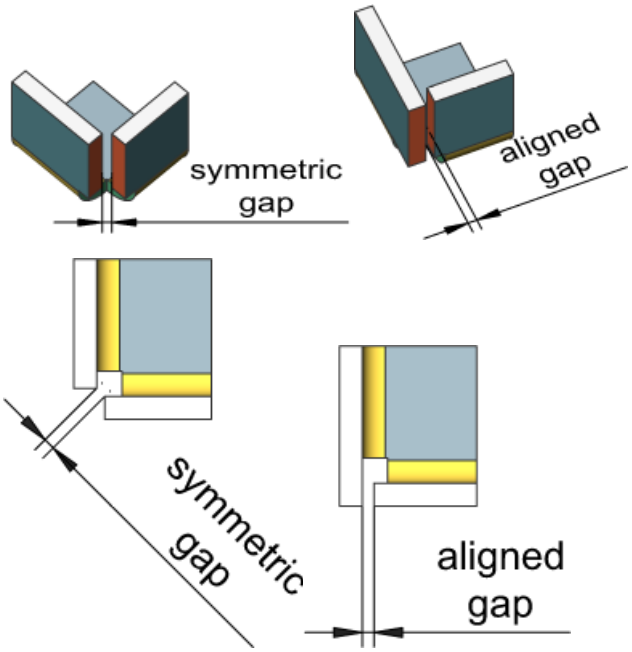
| Глобальные свойства | Description |
|---|---|
| **Зазор стыка** | Зазор между двумя смежными стенками.<br>• **Тип**: радиус может быть определен с помощью коэффициента толщины или как абсолютное значение.<br>• **Значение**: коэффициент или абсолютное значение.<br> |

**Таблица 40: Глобальные свойства**

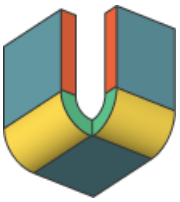
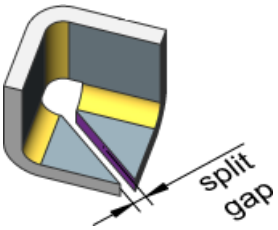
| Глобальные свойства | Description |
|---|---|
| **Выравнивание стыка** | Управляет выравниванием граней стыка с соответствующими гранями угловых вырезов.<br>• **Тип**: радиус может быть определен с помощью коэффициента толщины или как абсолютное значение.<br>• **Значение**: коэффициент или абсолютное значение.<br>• **Выравнивание стыка**: включено или отключено<br><br><br>**Рисунок 123:** Отключено<br>**Рисунок 122:** Включено |
| **Зазор при разделении листа** | Зазор между торцевыми гранями листа или смежными гибами при разделении, см. *SMSPLIT*.<br>• **Тип**: радиус может быть определен с помощью коэффициента толщины или как абсолютное значение.<br>• **Значение**: коэффициент или абсолютное значение.<br> |

**Таблица 40: Глобальные свойства**

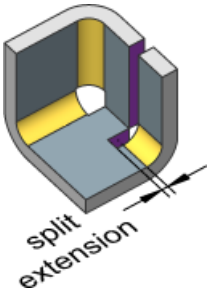
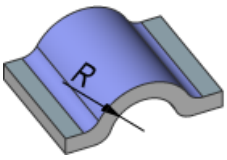
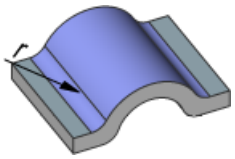
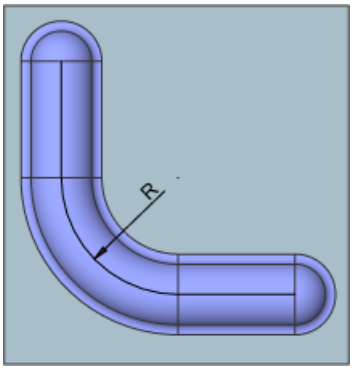
| Глобальные свойства | Description |
|---|---|
| **Заступ при разделении листа** | Управляет глубиной реза смежного листа при разделении листа.<br>• **Тип**: радиус может быть определен с помощью коэффициента толщины или как абсолютное значение.<br>• **Значение**: коэффициент или абсолютное значение.<br> |
| **Радиус профиля ребра** | Радиус профиля инструмента, который создает элемент жесткости.<br>• **Тип**: радиус может быть определен с помощью коэффициента толщины или как абсолютное значение.<br>• **Значение**: коэффициент или абсолютное значение.<br> |
| **Радиус скругления элементе жесткости** | Радиус скругления между листом и элементом жесткости.<br>• **Тип**: радиус может быть определен с помощью коэффициента толщины или как абсолютное значение.<br>• **Значение**: коэффициент или абсолютное значение.<br> |

**Таблица 40: Глобальные свойства**

| Глобальные свойства | Description |
|---|---|
| **Радиус закругления** | Радиус закругления контрольной кривой элемента жесткости. См. Работа с элементами форм и жесткости.<br>• **Тип**: может быть определен с помощью коэффициента радиуса профиля или как абсолютное значение.<br>• **Значение**: числовое значение.<br> |
| **Развертка** | Параметры создания развертки. |
| **K-фактор по умолчанию** | K-фактор для радиуса изгиба, равного толщине листа. См. K-фактор. |
| **Таблица гибов** | Отображение диалогового окна выбора файла таблицы гибов в формате CSV. |
| **Заступ линии гиба** | Контролирует, насколько далеко линия сгиба выходит за границы развертки (допускается нулевое или отрицательное значение).<br>• **Тип**: радиус может быть определен с помощью коэффициента толщины или как абсолютное значение.<br>• **Значение**: коэффициент или абсолютное значение. |
| **Режим развертки** | Управляет отображением элементов формы в развертке.<br>• **Сохранить**: сохраняет 3D представление элементов формы.<br>• **Удалить**: полностью удаляет элементы формы из развернутого представления.<br>• **Проекция**: проецирует все ребра элементов формы как 2D объекты.<br>• **Контур**: проецирует только контур элементов формы как 2D объекты.<br>• **Символ**: размещает на развертке 2D символы элементов формы. |

При выборе элемента в **Обозревателе конструкции** отображаются его свойства.

Щелчок правой кнопкой мыши на выбранном элементе открывает контекстное меню этого элемента.

**Таблица 41: Контекстное меню**

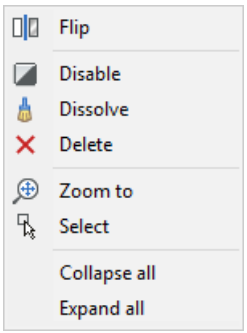
| Контекстное меню | Description |
|---|---|
| | Общие инструменты<br>• **Отключить**: временно отключает элемент и не учитывает связанные с ним зависимости.<br>• **Расформировать**: удаляет элемент с сохранением его геометрии. Информация о намерениях проектировщика, связанная с геометрией расформированного элемента, будет удалена.<br>• **Удалить**: см. команду *SMDELETE*. Удаляет элемент из Обозревателя конструкции и соответствующую ему геометрию из модели.<br>• **Показать в модели**: увеличивает модель по габаритам выбранного элемента.<br>• **Свернуть все**: сворачивает основной компонент и все его компоненты и подкомпоненты.<br>• **Раскрыть все**: Раскрывает основной компонент и все его компоненты и подкомпоненты. |
| Элемент Лист | Щелчок правой кнопкой мыши на элементе Лист открывает контекстное меню:<br><br>**Инверсия**: переставляет стороны выбранного листа так, чтобы опорная грань находилась на другой геометрической стороне листа. Эта операция перемещает лист на его толщину таким образом, чтобы положение опорной грани оставалось без изменений. |

**Таблица 41: Контекстное меню**

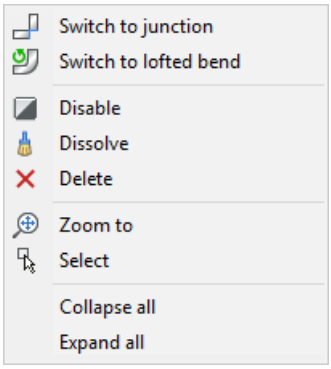
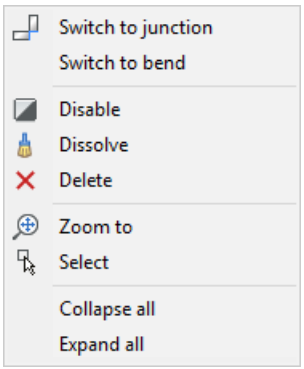
| Контекстное меню | Description |
|---|---|
| Элемент Гиб | Щелчок правой кнопкой мыши на элементе Гиб открывает контекстное меню: <br>• **Преобразовать в стык**: выполняет преобразование в элемент Стык с зазором между смежными листами.<br>• **Преобразовать в лофт-гиб**: выполняет преобразование в элемент Лофт-гиб. |
| Элемент Лофт-гиб | Щелчок правой кнопкой мыши на элементе Лофт-гиб открывает контекстное меню: <br>• **Преобразовать в стык**: выполняет преобразование в элемент Стык с зазором между смежными листами.<br>• **Преобразовать в гиб**: выполняет преобразование в элемент Гиб. |

**Таблица 41: Контекстное меню**

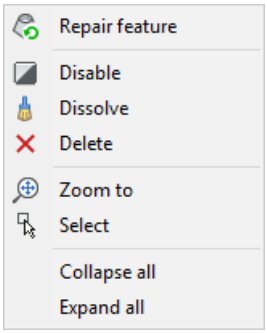
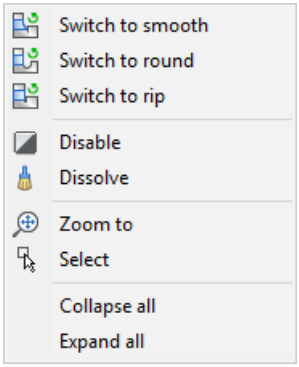
| Контекстное меню | Description |
|---|---|
| Элемент Некорректный гиб | Щелчок правой кнопкой мыши на элементе Некорректный гиб открывает контекстное меню: <br><br>**Исправить элемент**: исправляет некорректный элемент гиба, создавая элемент гиба с правильной геометрией, если это возможно. |
| Элемент Подрез гиба | Щелчок правой кнопкой мыши на элементе Подрез гиба открывает контекстное меню:<br><br>**Преобразовать в плавный, скругленный или прямой подрез**: переключает тип элемента Подрез гиба. |

**Таблица 41: Контекстное меню**

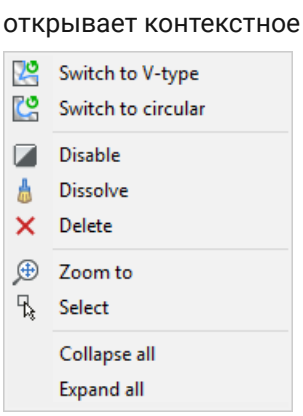
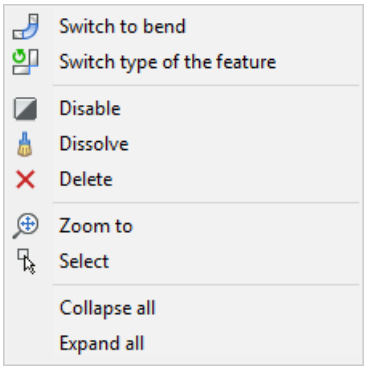
| Контекстное меню | Description |
|---|---|
| Элемент Угловой вырез | Щелчок правой кнопкой мыши на элементе Угловой вырез открывает контекстное меню:  **Преобразовать в V-образный или круговой вырез**: переключает тип элемента Угловой вырез. |
| Элемент Стык | Щелчок правой кнопкой мыши на элементе Стык открывает контекстное меню:  • **Преобразовать в гиб**: выполняет преобразование в элемент Гиб.<br>• **Переключить тип элемента**: переключает тип стыка между симметричным стыком и стыком с выравниванием по одному из листов. |

**Таблица 41: Контекстное меню**

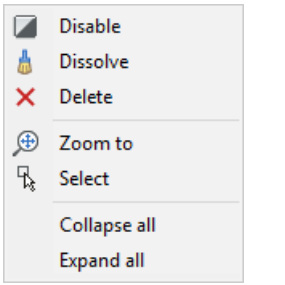
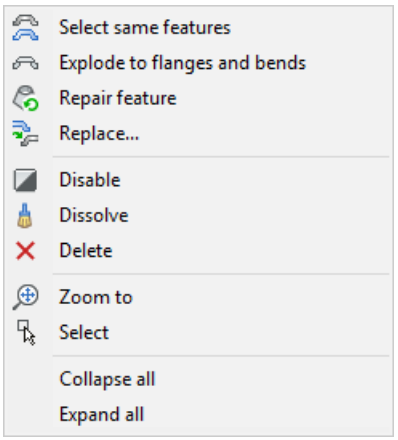
| Контекстное меню | Description |
|---|---|
| Элемент Скос | Щелчок правой кнопкой мыши на элементе Скос открывает контекстное меню:  |
| Элемент формы | Щелчок правой кнопкой мыши на элементе формы открывает контекстное меню: <br>• **Выбрать подобные элементы**: выбирает в модели все элементы формы такого же типа.<br>• **Расчленить на листы и гибы**: расчленяет элемент формы на набор листов и гибов, если это возможно.<br>• **Исправить элемент**: пересчитывает точку вставки элемента формы.<br>• **Заменить...**: отображает диалог выбора элемента формы для замены. |

**Таблица 41: Контекстное меню**

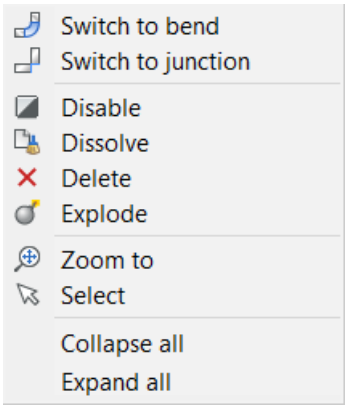
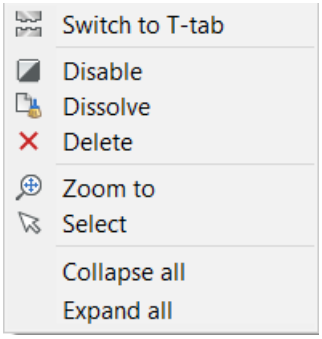
| Контекстное меню | Description |
|---|---|
| Элемент Шип-паз | Щелчок правой кнопкой мыши на элементе Шип-паз открывает контекстное меню: <br><br>• **Преобразовать в гиб**: выполняет преобразование элемента Шип-паз в элемент Гиб<br>• **Преобразовать в стык**: выполняет преобразование элемента Шип-паз в элемент Стык |
| Элемент Т-Стык | Щелчок правой кнопкой мыши на элементе Т-Стык открывает контекстное меню: <br><br>**Преобразовать в Т-Шип**: выполняет преобразование элемента Т-Стык в элемент Т-Шип. |

**Таблица 41: Контекстное меню**

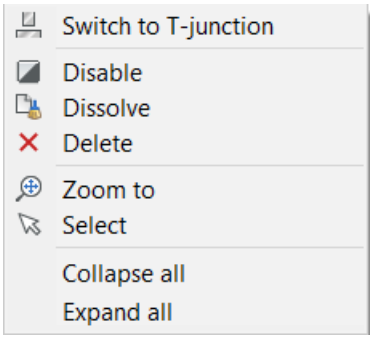
| Контекстное меню | Description |
|---|---|
| Элемент T-образный шип-паз | Щелчок правой кнопкой мыши на элементе T-Шип открывает контекстное меню:  **Преобразовать в T-Стык**: выполняет преобразование элемента T-Шип в элемент T-Стык. |

**Таблица 42: Свойства элемента**

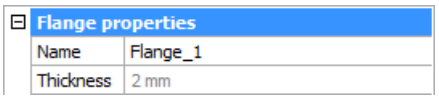
| Свойства элемента | Description |
|---|---|
| | Общее свойство:<br>**Имя**: имя выбранного элемента листового металла. При необходимости, введите имя элемента. По умолчанию в качестве имени объекта используется наименование типа элемента с его порядковым номером. Лист_1, Лист_2,... |
| **Элемент Лист** | Свойства отображаемые при выборе листа:<br> **Толщина**: текущая толщина. Значение доступно только для чтения и соответствует свойству Толщина компонента из листового металла. |

**Таблица 42: Свойства элемента**

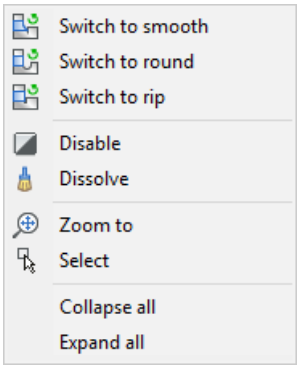
| Свойства элемента | Description |
|---|---|
| **Элемент Гиб** | Свойства, отображаемые при выборе гиба:<br><br><br>• **Угол**: текущий угол гиба; может быть изменен с помощью команды *SMFLANGEROTATE*.<br>• **Радиус**: текущий радиус гиба.<br>• **Тип значения радиуса**:<br>- Коэффициент толщины: коэффициент отношения к толщине<br>- Абсолютное значение: глобальное значение (основные свойства компонентов из листового металла). |
| **Подрез гиба** | Щелчок правой кнопкой мыши на элементе Подрез гиба открывает контекстное меню:<br><br><br>• **Тип подреза**: текущий тип подреза - прямоугольный, плавный, скругленный или прямой.<br>• **Ширина**: ширина подреза гиба; глобальное, абсолютное или коэффициент отношения к толщине. |

**Таблица 42: Свойства элемента**

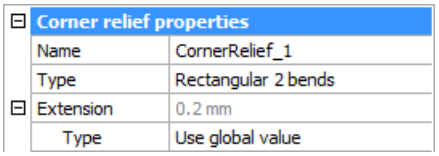
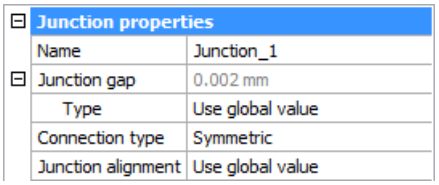
| Свойства элемента | Description |
|---|---|
| **Угловой вырез** | Щелчок правой кнопкой мыши на элементе Угловой вырез открывает контекстное меню:<br><br>Corner relief properties<br>Name: CornerRelief_1<br>Type: Rectangular 2 bends<br>Extension: 0.2 mm<br>Type: Use global value<br>• **Тип углового выреза**: текущий тип углового выреза - 2 прямоугольных гиба, прямоугольный, v-образный и круговой.<br>• **Ширина**: ширина подреза гиба; глобальное, абсолютное или коэффициент отношения к толщине. |
| **Элемент Стык** | Щелчок правой кнопкой мыши на элементе Стык открывает контекстное меню:<br><br>Junction properties<br>Name: Junction_1<br>Junction gap: 0.002 mm<br>Type: Use global value<br>Connection type: Symmetric<br>Junction alignment: Use global value<br>• **Зазор стыка**: зазор между двумя смежными стенками.<br>• **Тип зазора стыка**: может быть глобальным, абсолютным или определенным с помощью коэффициента толщины.<br>• **Тип соединения**: управляет взаимным расположением смежных листов - симметрия или выравнивание по одному из листов.<br>• **Выравнивание стыков**: управляет положением граней стыка.<br>- глобальный<br>- включено (выравнивание по соответствующим граням угловых подрезов)<br>- выключено. |

**Таблица 42: Свойства элемента**

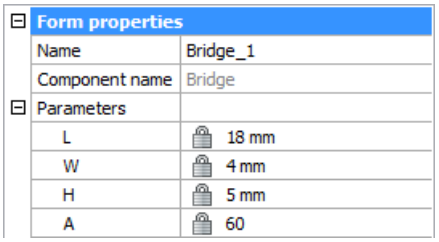
| Свойства элемента | Description |
|---|---|
| **Элемент Скос** | Щелчок правой кнопкой мыши на элементе Скос открывает контекстное меню:<br><br><br>• **Зазор скоса**: зазор между торцевыми гранями скоса.<br>• **Тип**: зазор может быть определен с помощью коэффициента толщины или как абсолютное значение. |
| **Элемент формы** | Щелчок правой кнопкой мыши на элементе формы открывает контекстное меню:<br><br><br>• **Имя компонента**: имя определения компонента элемента формы.<br>• **Параметры**: раздел содержит параметры параметрического элемента формы листового металла. |

**Таблица 42: Свойства элемента**

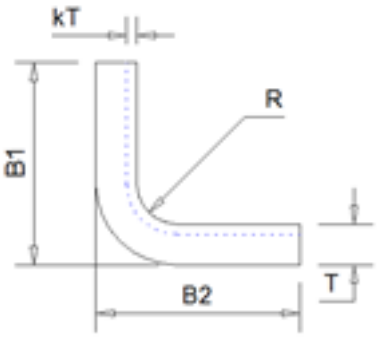
| Свойства элемента | Description |
|---|---|
| **Элемент Фаска** | Щелчок правой кнопкой мыши на элементе Фаска открывает контекстное меню:<br><br>• Тип фаски: определяет тип фаски в зависимости от геометрии. Возможные значения: односторонняя, Y-, X- и K-образная. Набор параметров фаски зависит от ее типа.<br>• Торцевая фаска: параметры торцевой фаски.<br>• Ортогональная фаска: параметры ортогональной фаски. |

### 21.5.7 K-фактор

#### 21.5.7.1 Обзор

При гибе листового металла происходит деформация материала, которая приводит к разнице длины листа в развернутом и неразвернутом состоянии. BricsCAD автоматически учитывает эту деформацию при создании развертки детали из листового металла. Для ее расчета используются два подхода: упрощенный (с учетом K-фактора) и расширенный (с использованием таблицы гибов).

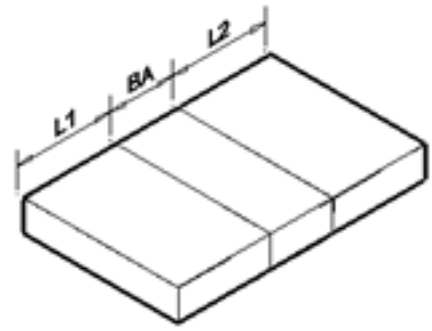
Свойства деформации материала при изгибе основаны на предположении о существовании внутреннего недеформируемого (нейтрального) слоя листового металла. Из всех внутренних слоев выбирается один, который одинаково удален от одной стороны изгиба. Поверхность является локальной для каждого изгиба и для упрощения можно применить ее ко всей детали, как это показано на рисунке. Мы предполагаем, что эта поверхность при изгибе листа не удлиняется.




T: толщина детали из листового металла

R: внутренний радиус гиба

K-фактор (k) определяет соотношение положения нейтрального слоя и толщины материала. В результате, нейтральный слой располагается на расстоянии kT от внутренней поверхности изгиба. Радиус изгиба для нейтрального слоя равен R + kT. K-фактор является простым геометрическим расчетом положения нейтрального слоя. Он не учитывает формирующие изгиб усилия и другие факторы, которые могут привести к ошибке. Значение K-фактора зависит от многих параметров, включая свойства материала, тип операции гиба, применяемое оборудование и др. и обычно находится в диапазоне 0.3 - 0.5. По умолчанию при радиусе изгиба, равном толщине материала (T) значение K-фактора равно 0.27324.




В развернутом состоянии детали из листового металла ее стенки (L1 и L2) не растянуты. Изгибаемая часть обозначена как BA (Bend Allowance): припуск на изгиб. Значение BA равно длине развертки нейтральной поверхности:

BA = УголИзгиба * (R + kT)

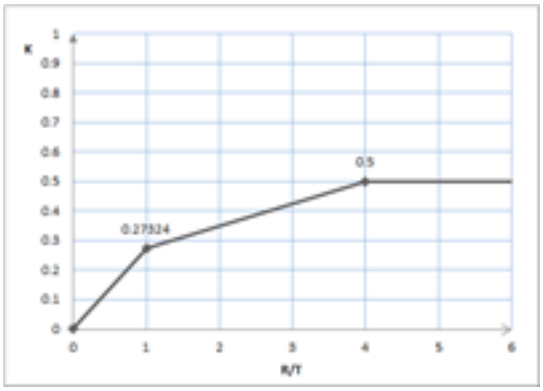
На практике тяжело измерить значение K-фактора или величину припуска на изгиб. Приведенная ниже формула позволяет вычислить величину уменьшения сгиба BD (Bend Deduction):

BD = (B1 + B2) - Длина развертки = (B1 + B2) - (L1 + L2 + BA)

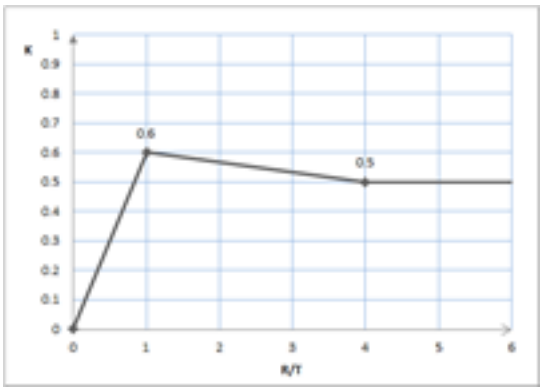
В простых случаях значение K-фактора может быть изменено в окне Обозревателя конструкции. Для обеспечения максимальной точности гиба должна предоставляться таблица гиба, содержащая значения величины уменьшения сгиба. Для каждой детали из листового металла вы можете указать значение K-фактора или использовать его значение предложенное по умолчанию.

### 21.5.7.2 Изменение K-фактора детали из листового металла

Для изменения значения K-фактора детали из листового металла на панели Обозреватель конструкции выберите корневой узел этой детали и в поле **K-фактор** введите необходимое значение. Значение должно находиться в пределах [0 1], т.к. нейтральная поверхность располагается внутри детали из листового металла. В BricsCAD значение K-фактора определяется для радиуса изгиба, равного толщине металла (R/T = 1). При вычислении значений **K-фактора** для радиусов изгиба, превышающих толщину материала, BricsCAD использует специальный механизм интерполяции, проверенный в промышленных приложениях для листового металла. Приведенный ниже график демонстрирует принципы интерполяции: для R/T < 1 применяется линейная интерполяция значений от 0 до 0.27324, для 1< R/T < 4 применяется линейная интерполяция значений от 0.27324 до 0.5 и для 4 < R/T значение K-фактор равно постоянному значению 0.5. Физически это означает что при значении радиуса изгиба, превышающем толщину листа более чем в 4 раза, удлинение материала не учитывается.




Значение K-фактора может быть изменено в обозревателе для соотношения R/T. Например, при установке значения 0.6 применение закона интерполяции приведет к следующему результату:

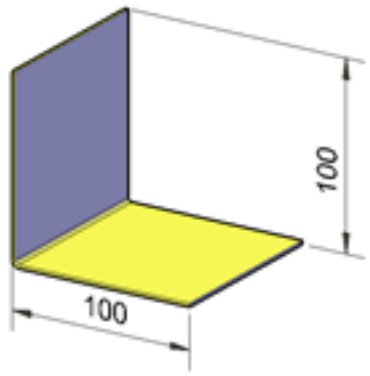



Изменение K-фактора в Обозревателе конструкции приводит к следующей эффективной кривой. Отличительной особенностью кривой является то, что изменение R в диапазоне (0, T], L1 + L2 + BA даст то же самое значение. Это объясняется тем, что радиусы меньше толщины, к которым они

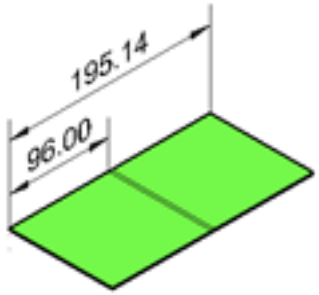
относятся, предназначены для удобства проектирования, а не для моделирования физического процесса, поскольку во время гибки радиус инструмента обычно больше или равен толщине.

Рассмотрим пример с изменением K-фактора с помощью настройки контрольной кривой (радиус изгиба остается постоянным).

T = 2, R = 2, B1 = 100, B2 = 100



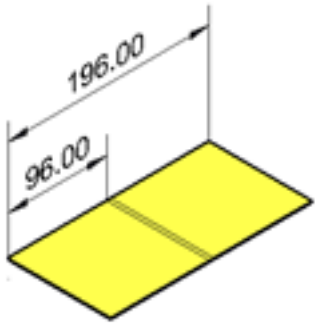

При установке значения K=0 получаем значения L1=L2=96, длина развертки = 96, и значение BA будет равно 3.14.



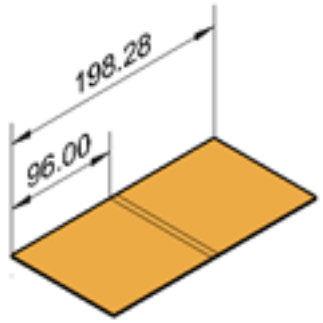

Это значение точно соответствует формуле: прямой угол в радианах равен приблизительно 1.57 (PI/2). При значении K-фактора, равном по умолчанию 0.27324, получаем длину развертки = 196.

Или: L1=L2=96 (стенки не деформируются), BA = 4 что следует из следующих вычислений (196 - 2*96) или из формулы расчета уменьшения сгиба:

BA = УголИзгиба * (R + kT) = 1.57 * (2 + 2*0.27324) = 4



Увеличение значения K-фактора приводит к результату Длина развертки = 198.28. Это максимальное значение, которое может быть достигнуто при совпадении нейтральной поверхности с внешней стороной изгиба.




Уменьшение сгиба (BD) для k = 0.27324:

BD = (B1 + B2) - Длина развертки = (100 + 100) - 196 = 4

### 21.5.8 Таблица гибов

#### 21.5.8.1 Обзор

Таблица сгибов предоставляет более точную информацию о свойствах деформации материала, чем K-фактор. При изгибе плоского листа из определенного материала вы можете выполнить замер его длины до и после выполнения операции изгиба. Вы можете повторить эту процедуру для разной толщины листа при различных значениях углов и радиусов изгиба и сохранить полученные результаты в таблице гибов. Эти измерения могут быть использованы для точного вычисления разверток всех деталей, изготавливаемых из этого материала.

Вы можете добавить в таблицу гибов любое количество результатов для различных углов гиба. Но в большинстве случаев достаточно иметь измерения для угла 90 градусов. BricsCAD автоматически вычисляет длину для других углов изгиба, используя проверенный механизм интерполяции.

Таблицы гибов сохраняются в файлах с расширением .csv (Comma Separated Values). Вы можете создавать такие таблицы в любом текстовом редакторе ASCII, например, Блокнот. Но более надежный и простой способ - создание электронной таблицы, например, в Microsoft Excel или LibreOffice с последующим ее экспортом в .csv файл.

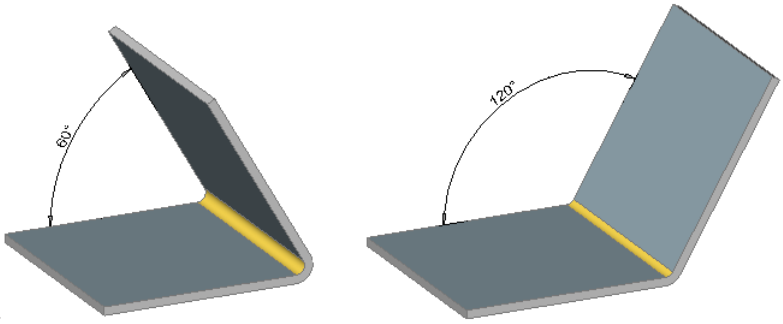
Таблицы сгибов в BricsCAD имеют следующую структуру:

| | A | B | C | D | E | F | G |
|---|---|---|---|---|---|---|---|
| 1 | BricsCAD | | | | | | |
| 2 | Version | 1 | | | | | |
| 3 | AngleType | Internal | | | | | |
| 4 | LengthType | BendDeductionTangent | | | | | |
| 5 | | | | | | | |
| 6 | Thickness | 1 | | | | | |
| 7 | AngleValues | | 45 | 60 | 90 | 120 | 135 |
| 8 | Radius | 1 | 1 | 1.333 | 2 | 0.976 | 0.657 |
| 9 | Radius | 2 | 0.466 | 1.081 | 2.31 | 1.004 | 0.641 |
| 10 | Radius | 3 | -0.069 | 0.828 | 2.621 | 1.033 | 0.624 |
| 11 | | | | | | | |
| 12 | Thickness | 1 | | | | | |
| 13 | DieWidth | 10 | | | | | |
| 14 | AngleValues | | 45 | 60 | 90 | 120 | 135 |
| 15 | Radius | 1 | 1.225 | 1.533 | 2.15 | 1.076 | 0.732 |
| 16 | Radius | 2 | 0.616 | 1.214 | 2.41 | 1.071 | 0.691 |
| 17 | Radius | 3 | 0.006 | 0.895 | 2.671 | 1.066 | 0.649 |

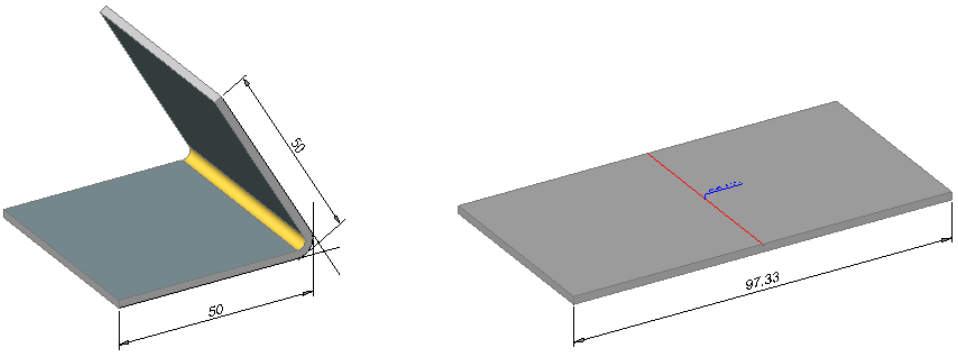
**Рисунок 124:** Пример таблицы гибов

При создании таблицы гибов используйте следующие правила:

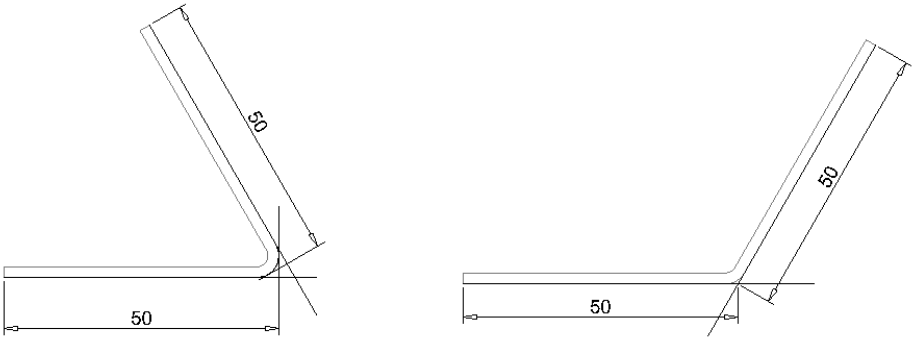
- Первая ячейка должна содержать текст "BricsCAD".
- **Version (Версия)**: текущая поддерживаемая версия - 1.
- **AngleType (Тип значения угла)**: в настоящее время поддерживаются только внутренние углы:




- **LengthType (Тип значения длины)**: метод получения значений в ячейках таблицы, соответствующих определенному значению угла и радиуса изгиба. В настоящее время поддерживается только метод BendDeductionTangent, при котором уменьшение гиба определяется по точкам касания. Уменьшение гиба (Bend deduction - BD) - это разница между суммарной длиной двух стенок, измеренной до точки касания в 3D модели и длиной того же фрагмента в развернутом состоянии:




Методика расчета уменьшения гиба для острых и тупых углов.




- Один заголовок может объединять несколько отдельных таблиц. Для каждой из этих таблиц первый ключ **Thickness (Толщина)** содержит толщину детали из листового металла. Второй ключ, **DieWidth (Ширина инструмента)**, не является обязательным. Пара ключей **Thickness**/**DieWidth** должны быть уникальными. Если ключ **DieWidth** не определен, будет доступно только одно уникальное значение ключа **Thickness**, но могут быть добавлены несколько значений толщины. При использовании таблицы гибов для управления процессом развертки значения ключей **Thickness** и **DieWidth**, определенные для детали из листового металла, должны присутствовать в применяемой таблице гибов. Если таблица не найдена, будет применено значение K-фактора, установленное по умолчанию.
- Сразу после того, как в файле с расширением .csv будет найдена таблица сгибов, он будет использован для создания развертки детали из листового металла. Рассмотрим несколько случаев для приведенного выше примера таблицы сгибов:

  T = Thickness (Толщина), R = Radius (Внутренний радиус изгиба), A = Angle (Угол изгиба)
  - T = 1, R = 2, A = 45. В результате развертки величина уменьшения гиба составит BD = 0.466, так как значения R и A имеются в таблице гибов для указанной толщины.
  - T = 1, R = 2.5, A = 45. В таблице имеется значение A = 45, но отсутствует значение R = 2.5: при определении BD будет применена интерполяция между R=2 и R=3.
  - T = 1, R = 2.5, A = 75. В таблице отсутствуют указанные значения R и A: при расчете будут использованы ячейки со следующими значениями: (R=2, A=60), (R=3, A=60), (R=2, A=90) и (R=3, A=90).

  При вычислении будет применен алгоритм top-of-the-edge. Например, линейная интерполяция близких значений BD приводит к результатам, не соответствующим действительным результатам изгиба. Также необходимо учитывать, что не все значения BD имеют смысл для заданных значений T, R и A: нейтральная поверхность может случайно выйти за пределы толщины листового металла или полученное значение K-фактора (определяемого по значению BD) может оказаться за пределами допустимого диапазона значений [0, 1]. В этом случае будет применено значение BD для значения K-фактора, используемого по умолчанию.
- Если для производства деталей из листового металла используется различное оборудование, Вы можете описать его в таблице гибов, используя параметр **DieWidth**. Пример таблицы гибов доступен в папке **[BricsCAD Installation folder]\Samples\Mechanical\bend_tables**.

***Прим.:*** Необходимо учесть, что права доступа в эту папку могут быть ограничены. В этом случае необходимо скопировать таблицы гиба в другой каталог, к которому вы имеете полный доступ.

### 21.5.8.2 Установка таблицы гибов для детали из листового металла

1 Выберите основной узел детали в Обозревателе конструкции.

2 Выберите поле **Таблица гибов**. 


3 Нажмите кнопку **Обзор** ().

4 Выберите *.csv файл.

5 Дважды щелкните на выбранном или нажмите кнопку **Открыть**.

### 21.5.8.3 Обновление таблицы гибов

1 Выберите основной узел детали в Обозревателе конструкции.

2 Выберите поле **Таблица гибов**.

3 Нажмите кнопку **Обновить** ().

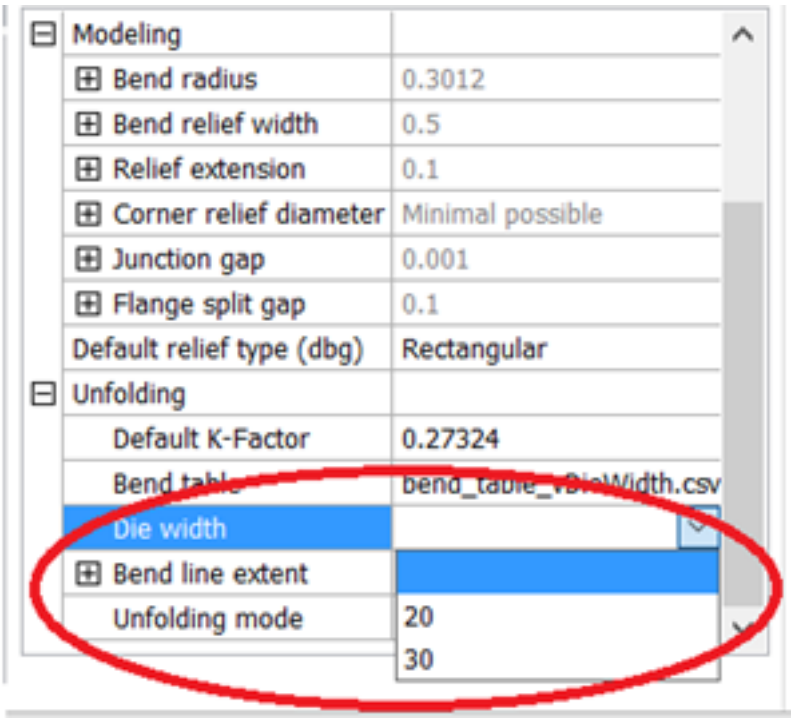
Если документу не назначена таблица гибов или отсутствует второй ключ **DieWidth**, назначение параметра **DieWidth** для одиночного гиба или для всех гибов невозможно. Для его назначения выполните следующие действия:

| | A | B | C | D |
|---|---|---|---|---|
| 1 | BricsCAD | | | |
| 2 | Version | 1 | | |
| 3 | AngleType | Internal | | |
| 4 | LengthType | BendDeductionTangent | | |
| 5 | | | | |
| 6 | Thickness | 1 | | |
| 7 | AngleValues | | 90 | |
| 8 | Radius | 1 | 2.1 | |
| 9 | | | | |
| 10 | Thickness | 1 | | |
| 11 | DieWidth | 20 | | |
| 12 | AngleValues | | 90 | |
| 13 | Radius | 1 | 1.7 | |
| 14 | | | | |
| 15 | Thickness | 1 | | |
| 16 | DieWidth | 30 | | |
| 17 | AngleValues | | 90 | |
| 18 | Radius | 1 | 1.8 | |

1 Выберите в .csv файле значения второго ключа **DieWidth** и толщины, соответствующие текущей детали. В приведенном примере толщина детали и радиус гиба приняты равными 1 мм. Ширина инструмента изменяется в зависимости от инструмента: первый раздел таблицы - без указания инструмента, второй раздел таблицы для **DieWidth** = 20, и последний раздел - для **DieWidth** = 30.

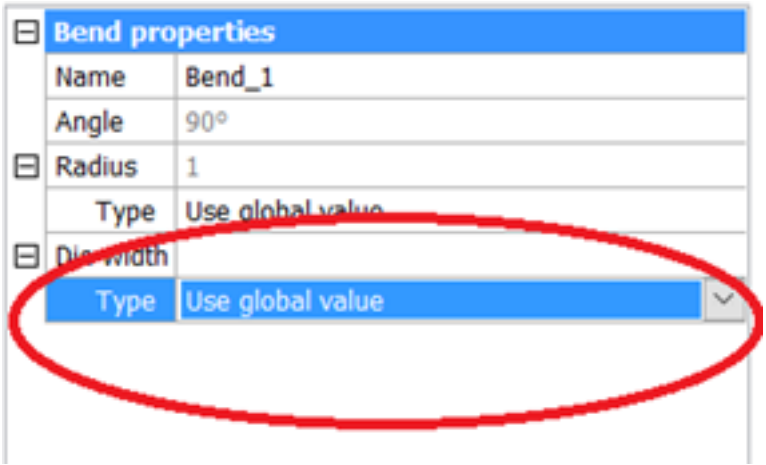
2 Назначьте таблицу гибов детали из листового металла.

3 Убедитесь, что новые поля отображаются в Обозревателе

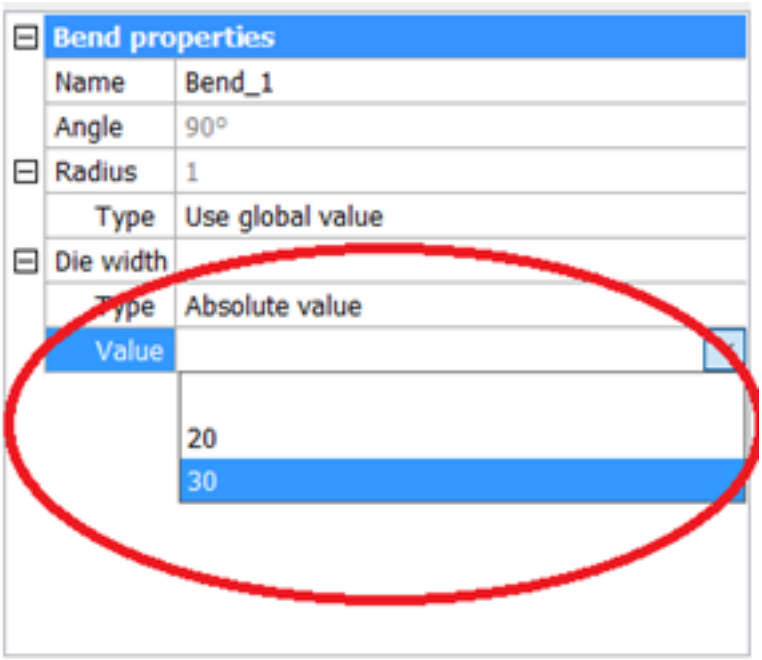



конструкции:

В этой общей настройке Вы можете выбрать любое подходящее значение **DieWidth** для использования в качестве значения по умолчанию для определенной толщины детали. Это значение будет использоваться по умолчанию для всех гибов детали при их выборе в Обозревателе конструкции:




4 Кроме того, для отдельных гибов можно назначить определенный инструмент, выбранный из




выпадающего списка:

5 В приведенном примере для гиба Гиб_1 будет применяться третий раздел таблицы и радиус 1.8. Для других гибов, кроме тех, для которых установлен ключ **DieWidth**, будет использоваться значение по умолчанию из первого раздела таблицы - 2.1 для 90 градусов и радиуса гиба, равного 1.

***Прим.:*** После назначения гибам ключа **DieWidth** и сохранения документа, значения сохраняются в гибах. Но при следующем открытии чертежа для назначения **DieWidth** другому гибу необходимо снова загрузить таблицу гибов, нажав кнопку с зеленой стрелкой (см. Обновление таблицы гибов). Значения таблицы гибов будут доступны после перезагрузки.

***Прим.:*** Примечание: Начиная с BricsCAD V21, вы можете изменять таблицы гибов для различных тел в пространстве модели, определяя локальный контекст листового металла для каждого тела.

### 21.5.9 Параметры листового металла

#### 21.5.9.1 Обзор

Раздел Листовой металл в диалоговом окне *Параметры* управляет различными свойствами листового металла в BricsCAD.. Свойства разделены на четыре группы: **Команды**, **Цвета элементов**, **Параметры по умолчанию** и **3D модель**.



**Рисунок 125:** Параметры: Листовой металл

***Прим.:*** Большинство настроек листового металла сохраняются в чертеже. Для достижения наилучших результатов всегда начинайте с одного из .dwt файлов шаблонов для листового металла или создавайте свои собственные шаблоны.

### 21.5.9.2 Команды

Раздел **Команды** содержит параметры, управляющие поведением различных команд для проектирования изделий из листового металла.

***Прим.:*** До выхода BricsCAD V19 эти элементы управления были доступны в качестве опций отдельных команд. Для упрощения интерфейса они были перенесены в диалоговое окно Параметры.

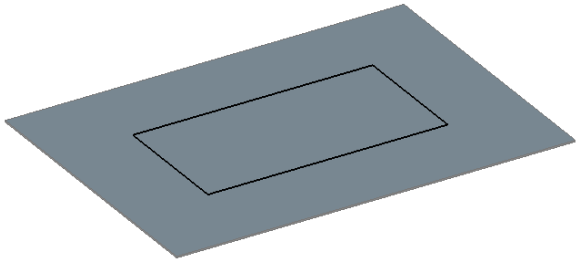


**Рисунок 126:** Параметры: Команды

Слои для разверток определяют внешний вид разверток деталей из листового металла.

### 21.5.9.3 Цвета элементов

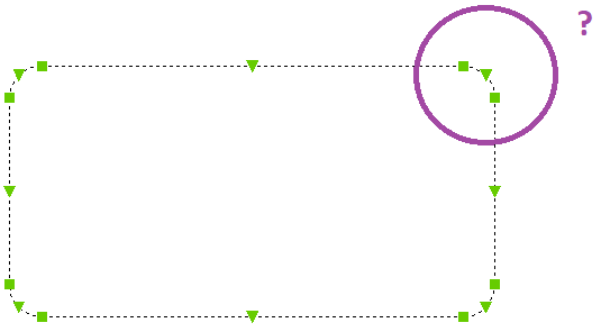



**Рисунок 127:** Параметры: Цвета элементов

**Настройки цветов** элементов позволяют визуально различать элементы из листового металла по цвету. Вы можете настроить цвет для каждого типа элемента.

### 21.5.9.4 Параметры по умолчанию

Эти параметры отвечают за инициализацию **контекста листового металла**, связанного с чертежом.

**Моделирование**

Параметры моделирования управляют поведением элементов из листового металла. Настройки сгруппированы по типам элементов.

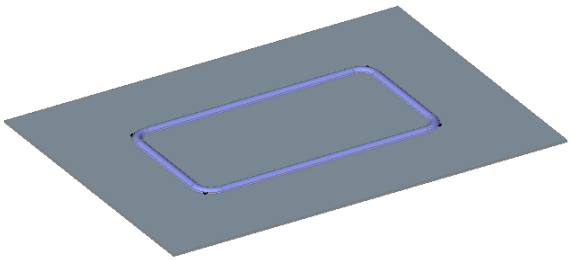


**Рисунок 128:** Параметры: 3D модель

**Развертка**

Параметры развертки управляют значением **K-фактора**, **режимом развертки элементов формы**, а также **типом и значением заступа линии гиба**.

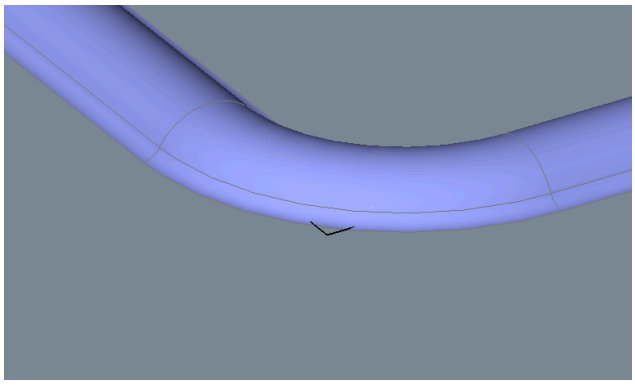


**Рисунок 129:** Параметры: Развертка

### 21.5.9.5 Моделирование

Раздел Моделирование включает пользовательский параметр *SmSmartFeatures*, который управляет автоматическим обновлением геометрии после модификации модели изделия из листового металла.

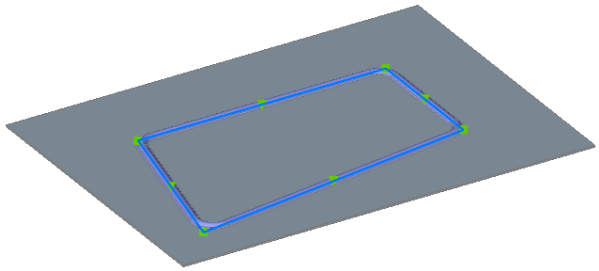


**Рисунок 130:** Параметры: 3D модель

**Видео: Параметризация деталей из листового металла**

## 21.5.10 Creating Flanges

### 21.5.10.1 Overview

A flange is the main feature of most sheet metal parts. In BricsCAD you can create flanges using many different techniques:

- Create a base flange from a closed 2D profile, which defines its boundary.
- Create one or several edge flanges by pulling edges of existing flanges.
- Create several flanges at once by extruding a 2D polyline (every linear edge of the polyline becomes a flange).

***Совет:*** There are also other techniques you can use to create flanges. These are described in:

- Flanges can be created as elements of lofted sheet metal parts.
- Flanges can be recognized in existing geometry.

In contrast to history-based MCAD, BricsCAD applies the same set of tools to modify any flange - independently.

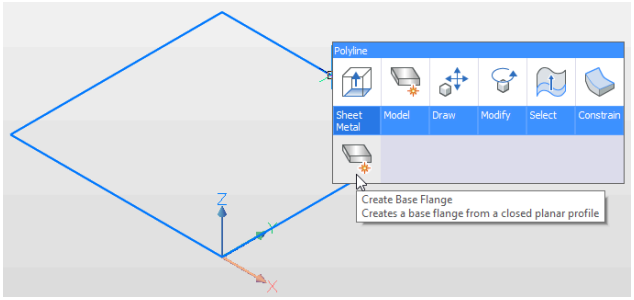
### 21.5.10.2 Create a Base Flange

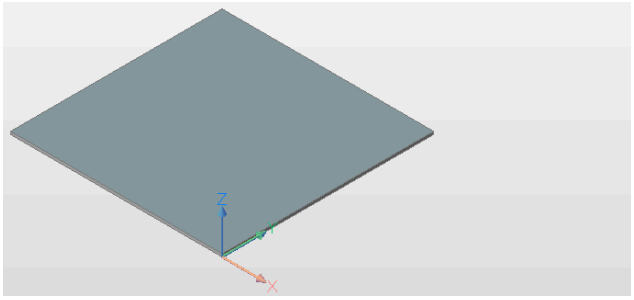
In general, you create a base flange to start your sheet metal part design. To create a base flange, select a closed planar profile and run the *SMFLANGEBASE* command.

Do one of the following:

- Click the **Create Base Flange** tool button ( ) on the **Sheet Metal** toolbar.
- Choose **Create Base Flange** in the **Sheet Metal** menu.
- Choose **Create Base Flange** in the **Sheet Metal** group of the **Quad** cursor menu:

Create a Base Flange




To create a base flange the *SMFLANGEBASE* command extrudes a profile.

The height of the base flange is the default **Thickness** property of the sheet metal part. To change the thickness of your sheet metal part, type the appropriate value in the **Thickness** field of the Mechanical Browser.

### 21.5.10.3 Create Edge Flanges

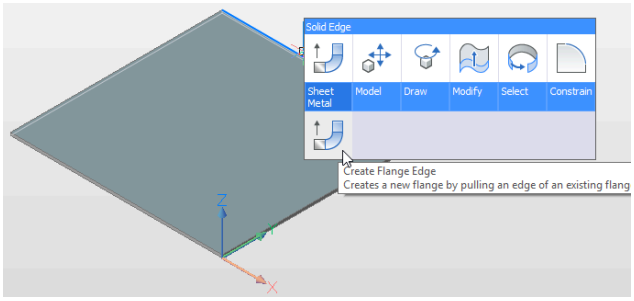
You can add an additional flange to your sheet metal:

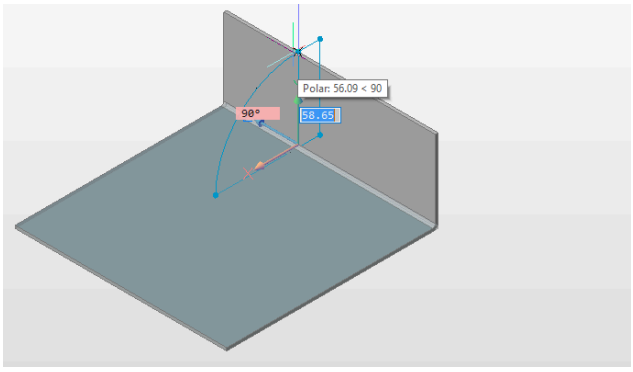
1 Select a linear edge.

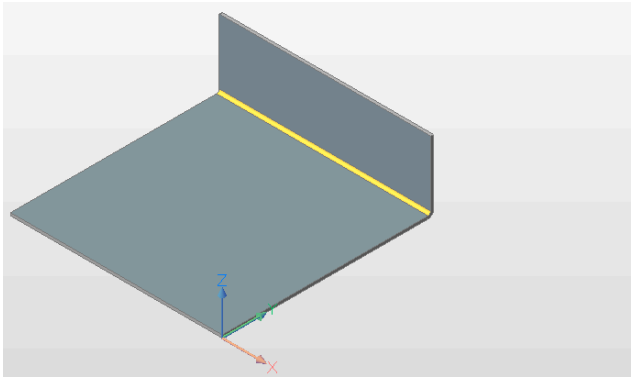
2 Select the *SMFLANGEEDGE* command in the **Sheet Metal** section of the **Quad** cursor menu.

3 Move your mouse pointer to define the desired length of the edge flange and the angle between the 2 flanges. You can also use the dynamic dimensions fields to enter the required values.

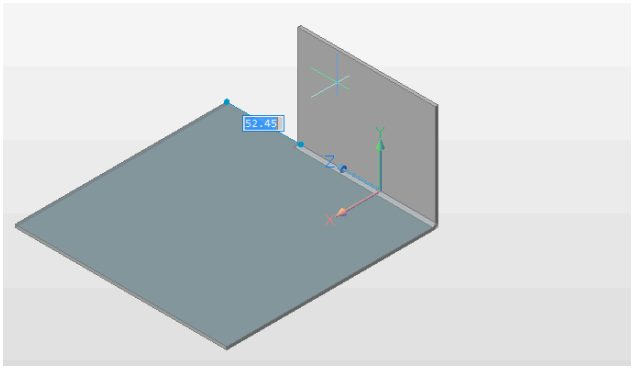
Create Flange Edge

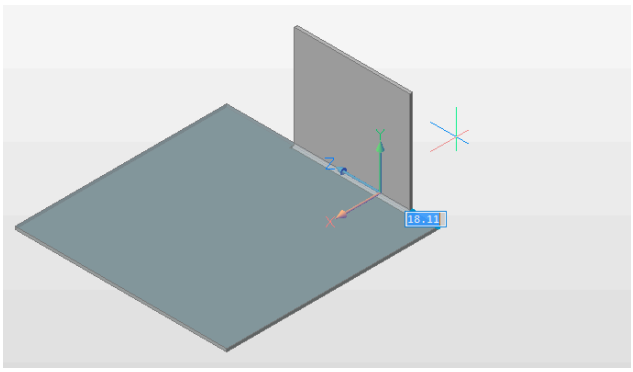


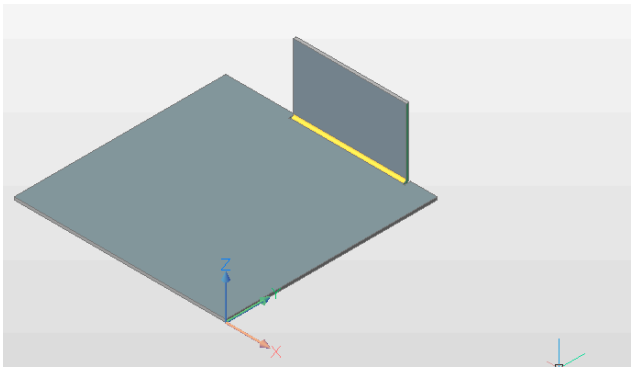



To create an edge flange where the width is different to the width of the edge, select the **Width** option of the *SMFLANGEEDGE* command and define the offset distance from one or both sides of the edge.



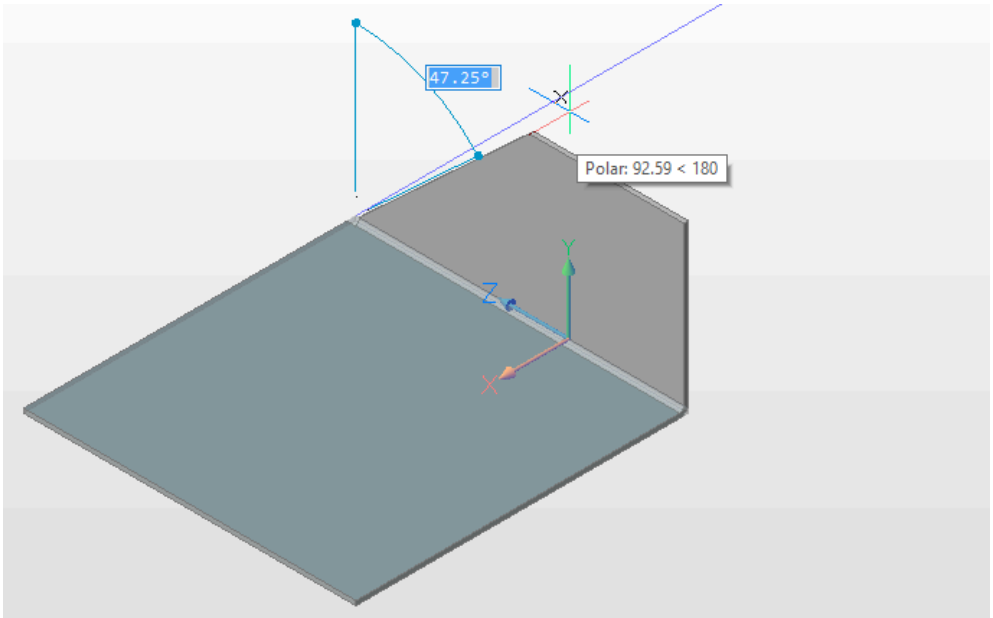


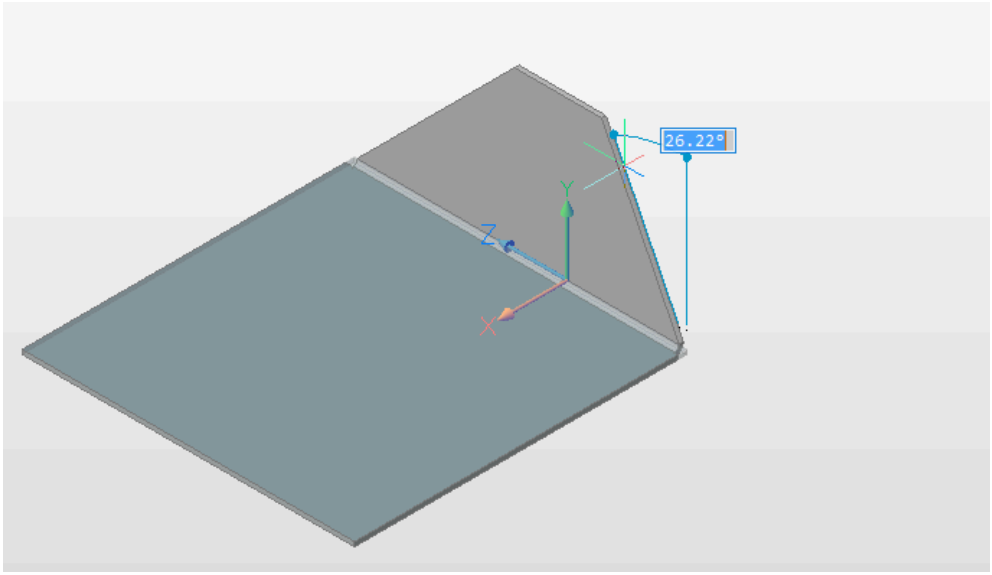



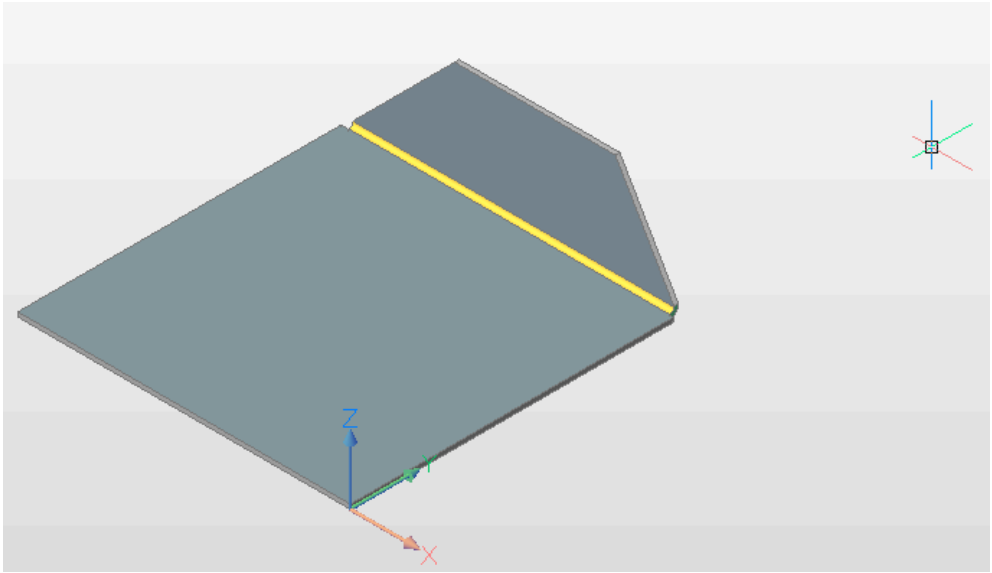
***Прим.:*** When you create an edge flange of a different width to the edge, bend reliefs are created automatically.

### 21.5.10.4 Create Edge Flange with Taper Angle(s)

You can create trapezoidal edge flanges with the **Taper angle** option of the *SMFLANGEEDGE* command:

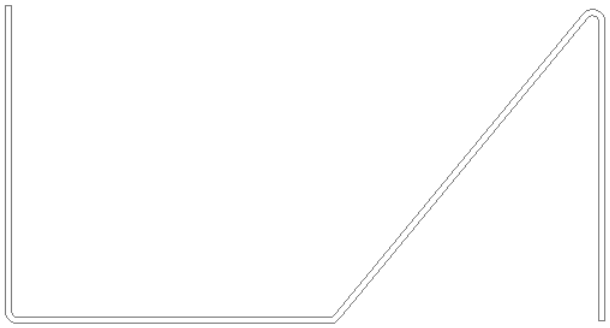





### 21.5.10.5 Create Multiple Flanges by Extruding a Polyline

There is a simple but important class of sheet metal parts which are seen as a thickened polyline if we view on the part from the "front" side:

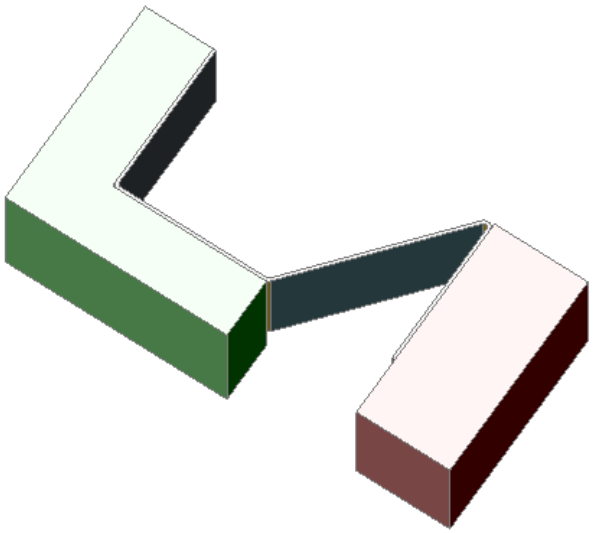


To create this part you can:

- use sequential calls of the *SMFLANGEEDGE* command, or
- draw a polyline and use the *DMEXTRUDE* command, followed by the *DMTHICKEN*, *SMCONVERT*, and *SMBENDCREATE* commands.

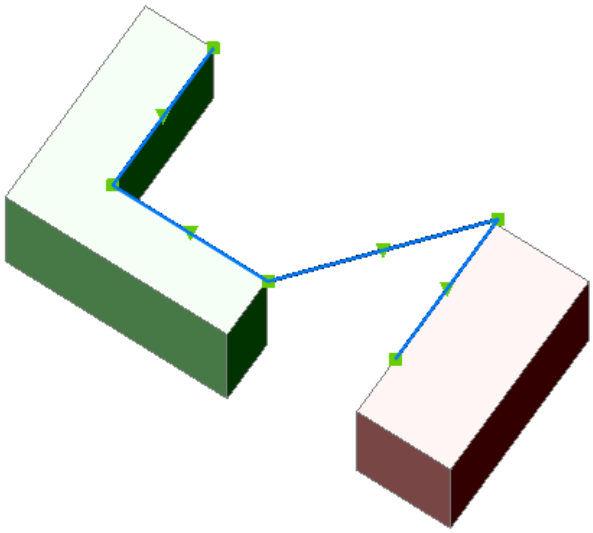
However, we recommend the following procedure:

BricsCAD allows you to draw a part in a complex assembly with other parts so that it does not interfere with other entities in the assembly.



You can control the thickening direction of every line segment individually with *SMEXTRUDE*.

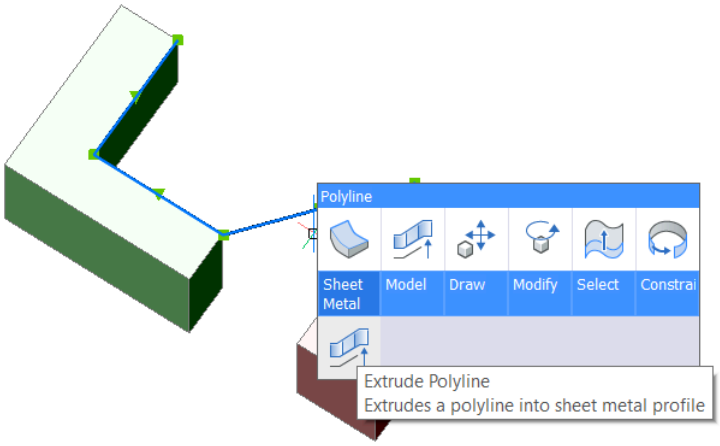
1 Draw a polyline that represents a sketch of the future sheet metal



part:

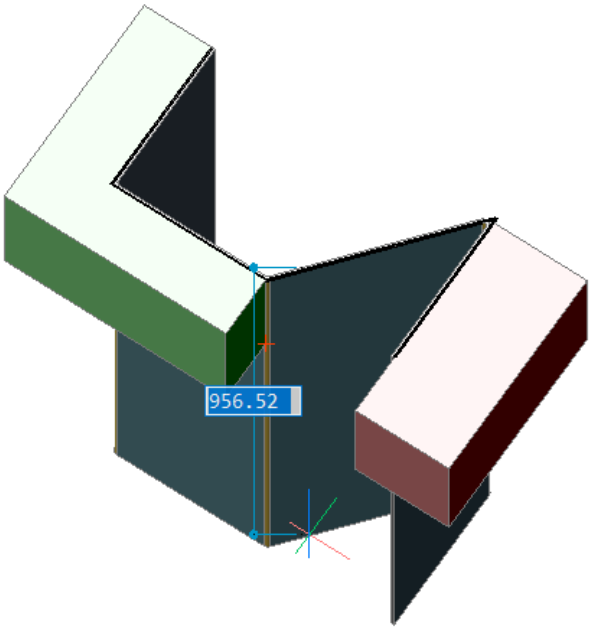
Do one of the following:

- Click the **Extrude Polyline** tool button ( ) on the **Sheet Metal** toolbar.
- Choose **Extrude Polyline** in the **Sheet Metal** menu.
- Choose **Extrude Polyline** in the **Sheet Metal** group of the **Quad** cursor



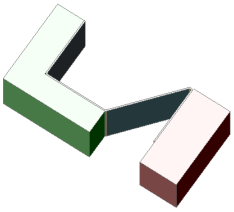
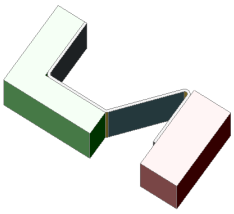
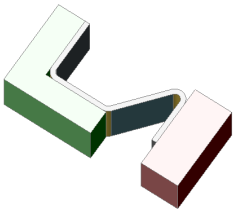

menu:

2 Hover your cursor to select **extrusion height** or enter the value and press **Enter** to




accept:

The body is created and the extensions are respected. The sheet metal part does not exceed the dimensions of the original polyline. The behavior is the same, even with different thicknesses:

| Thickness 1 mm | Thickness 2 mm | Thickness 4 mm |
|---|---|---|
|  |  |  |

***Прим.:*** The policy of preserving external dimensions also works for sheet metal parts created by other workflows: sequential flange building with *SMFLANGEEDGE*, part bends with *SMFLANGEBEND* and sheet metal design recognition with *SMCONVERT*. If you want to change the extrusion direction for a flange, apply the *SMFLIP* command to it.

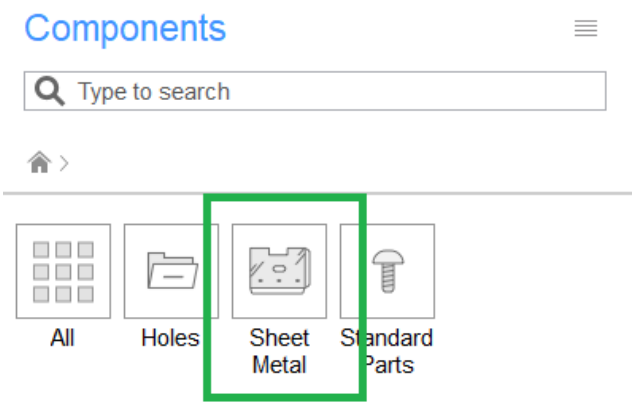
### 21.5.10.6 Extrude a Contour to Create a Flange

The *SMFLANGECONTOUR* command allows you to extrude a given contour to a new flange and attach it to the model. It can be thought of as a sequence of the *SMFLANGEBASE*, *SMFLANGECONNECT*, and *SMBENDCREATE* commands. In reality, it intelligently chooses the thickness faces on selected flanges and connects them with the thickness faces on a new flange.

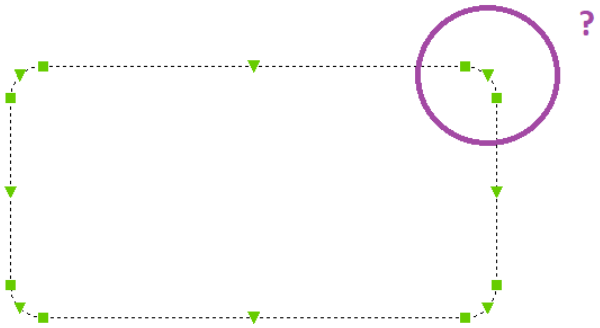
Do one of the following:

- Click the **Create Flange from closed Contour** tool button ( ) on the **Sheet Metal** toolbar.

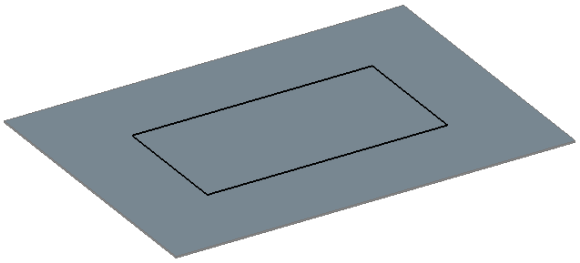
- Choose **Create Flange from Contour** in the **Sheet Metal** menu.
- Choose **Create Flange from Contour** in the **Sheet Metal** group of the **Quad** cursor menu:




Once all the required flanges are selected, click on the contour:

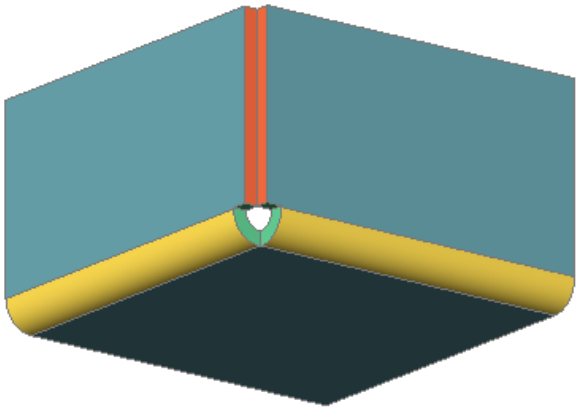



The command creates a new flange and merges it with the selected flanges on the part:



### 21.5.10.7 Bends, Corner Reliefs and Junctions

When you pull an edge flange with the *SMFLANGEEDGE* command, a bend feature between the edge flange and the base flange is automatically created. A corner relief feature is automatically created in a corner where 3 flanges meet together. This operation also creates a junction between 2 flanges, which are not connected with a bend.

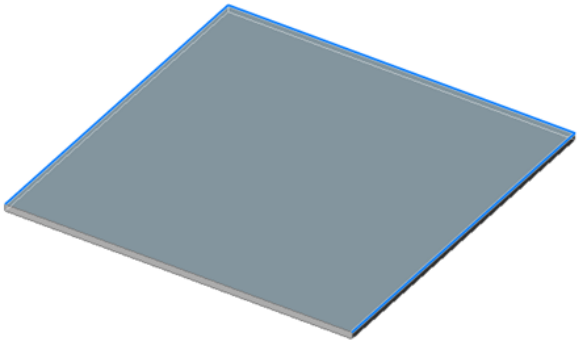


### 21.5.10.8 Create Multiple Flanges

You can create several flanges simultaneously.

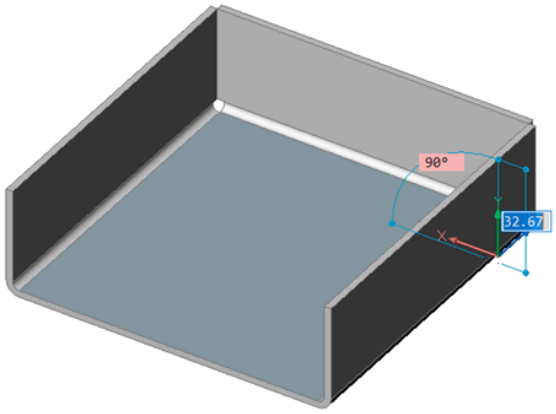
Do one of the following:

- Select multiple edges, then choose **Create Flange Edge** ( ) in the **Sheet Metal** command group of the **Quad** cursor menu.
- Launch the *SMFLANGEEDGE* command and select multiple edges.

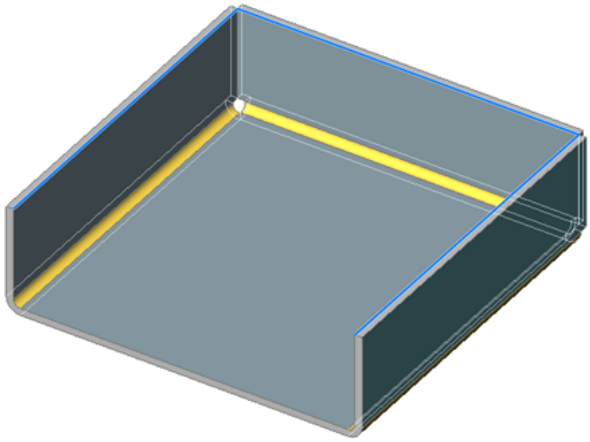


- Create the edges dynamically.

  Define the angle and the flange height using the dynamic entry fields. Press the **TAB** key to switch between the **Angle** and the **Distance** field.

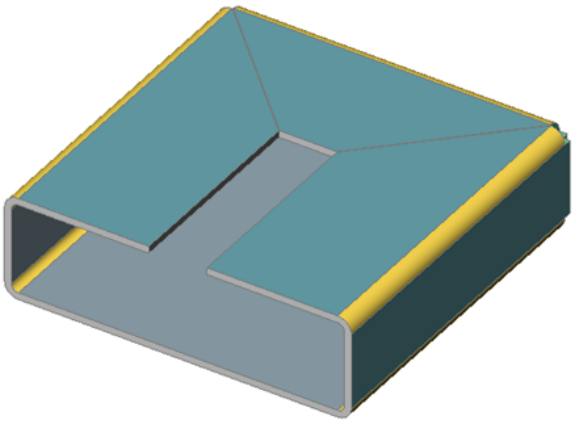



- Select the inside edges of the newly created flanges:

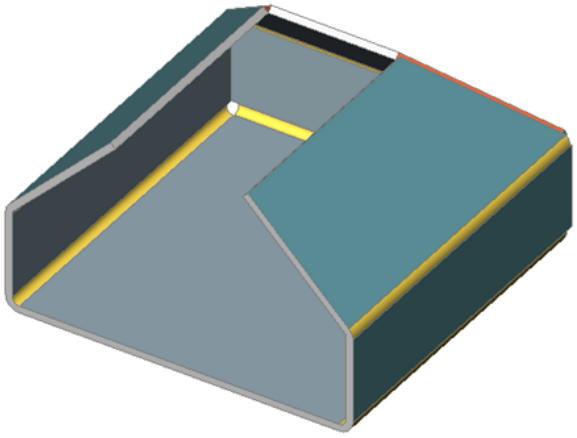


- Create the flanges.

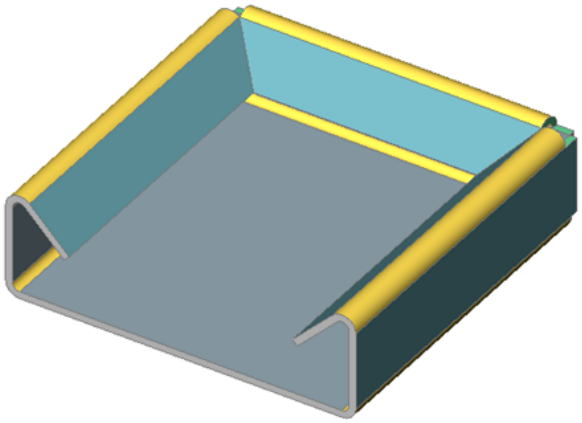
  The command automatically calculates the intersection of the flanges and creates a junction or miter features. At 90 degrees miters are created:



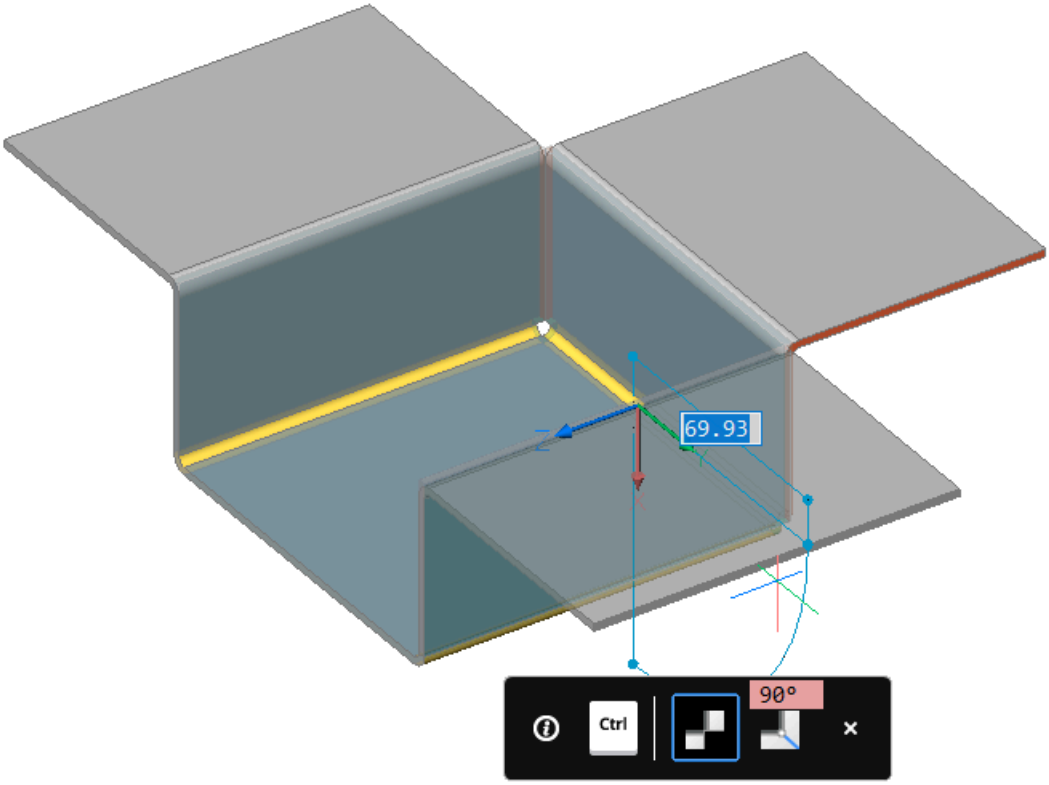
  For other angles junctions are created:

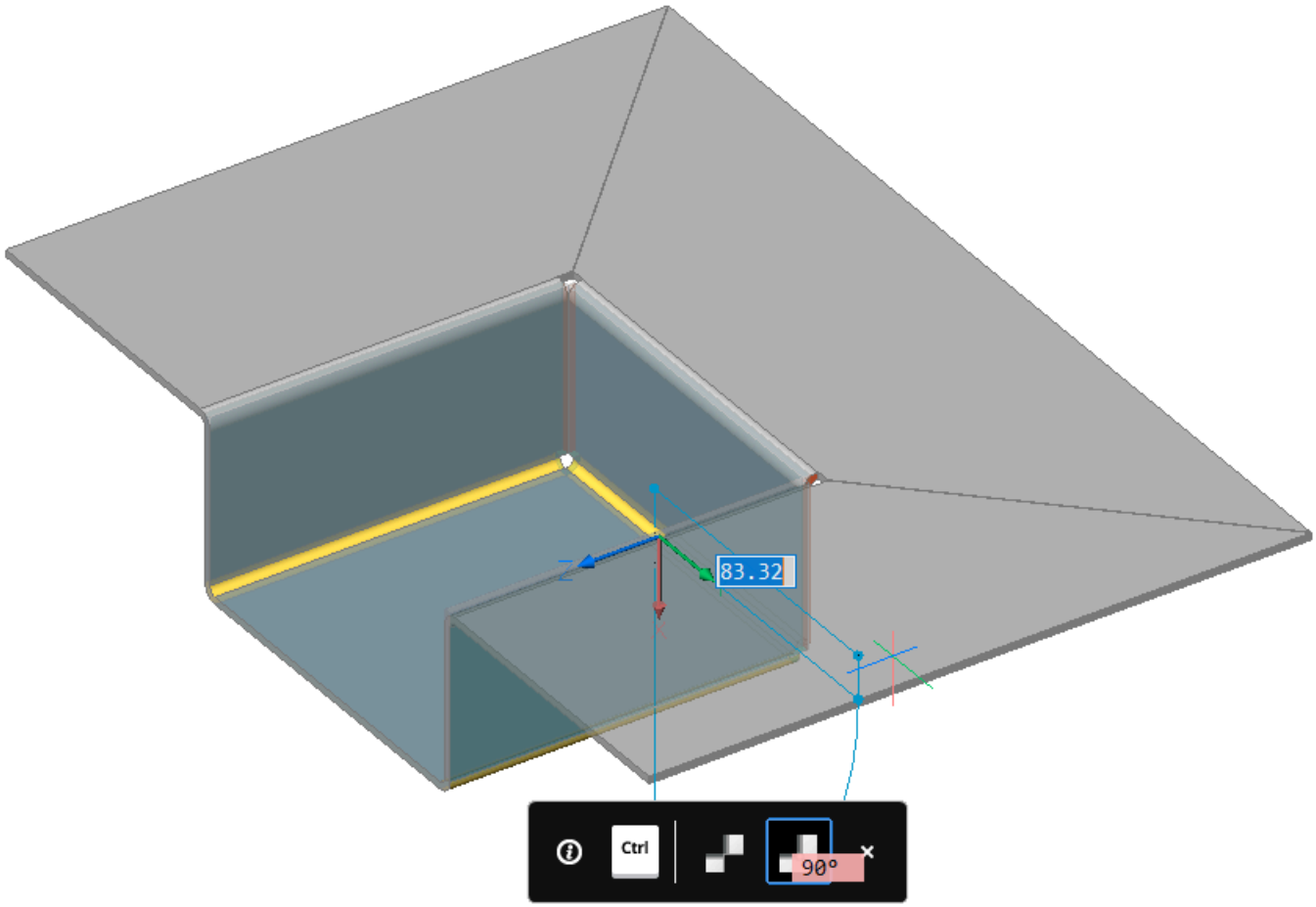


In all cases material conflicts are avoided:

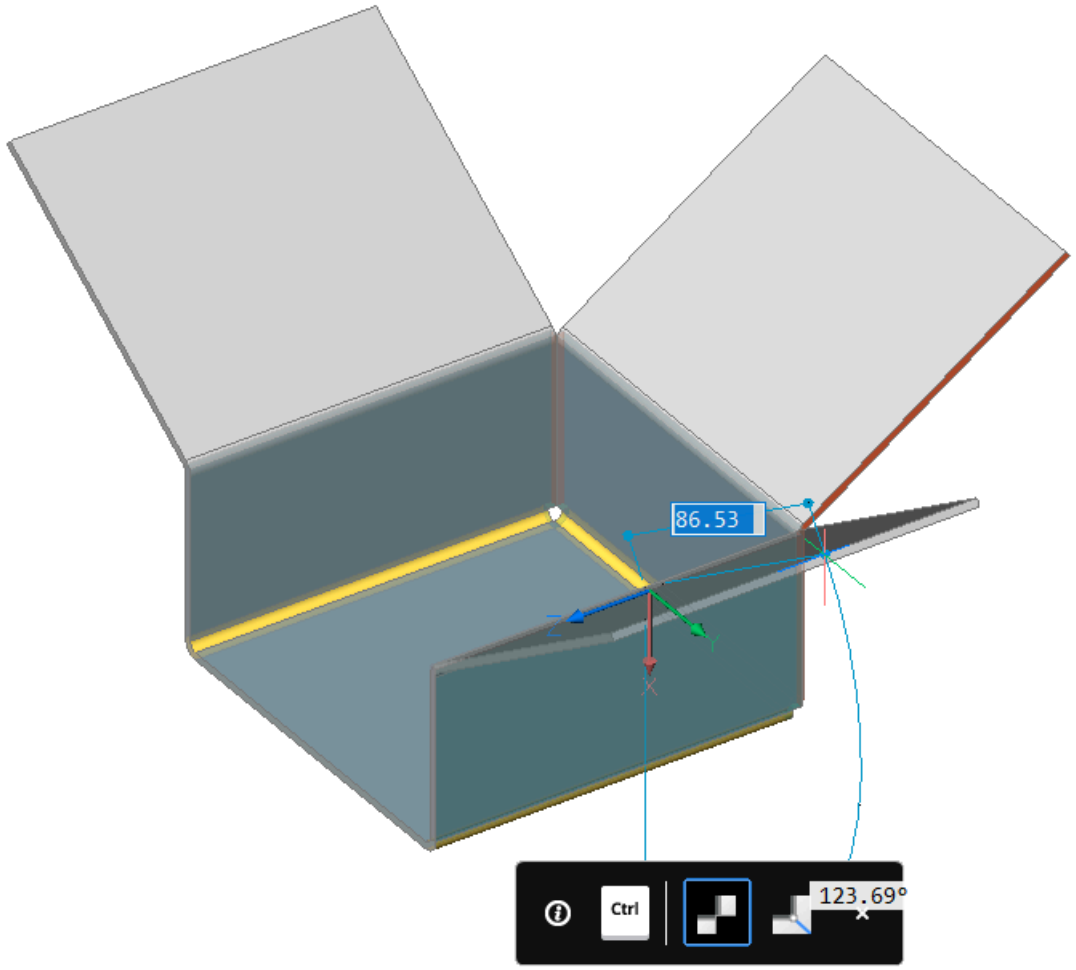


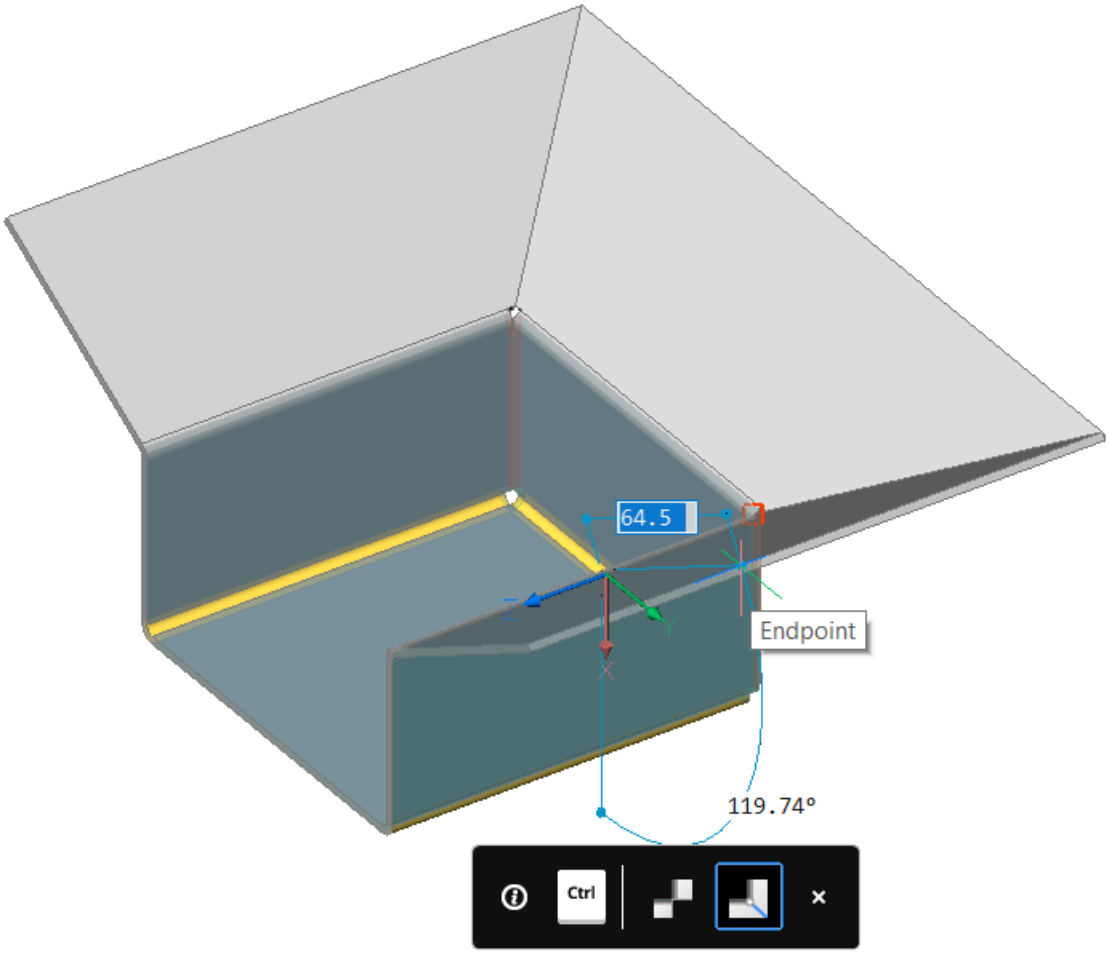
If the flanges are created externally, use the **Toggle** option (as well as **widget control**) to control the connection type for miters:


69.93
Ctrl
90°


83.32
Ctrl
90°

and junctions:


86.53
Ctrl
123.69°


64.5
Endpoint
119.74°
Ctrl

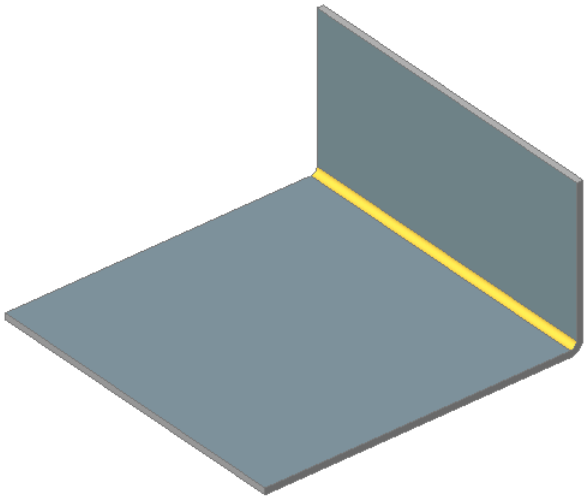
### 21.5.11 Редактирование стенок из листового металла

BricsCAD предоставляет набор инструментов для редактирования стенок деталей из листового металла. Эти инструменты могут быть применены к любой детали независимо от истории ее проектирования. В этой статье мы рассмотрим применение инструментов твердотельного моделирования и специальных инструментов, предназначенных для проектирования изделий из листового металла. Почти все инструменты твердотельного моделирования могут быть применены к деталям из листового металла с сохранением возможности интеллектуальной обработки элементов листового металла.

#### 21.5.11.1 Изменение размера и формы стенки

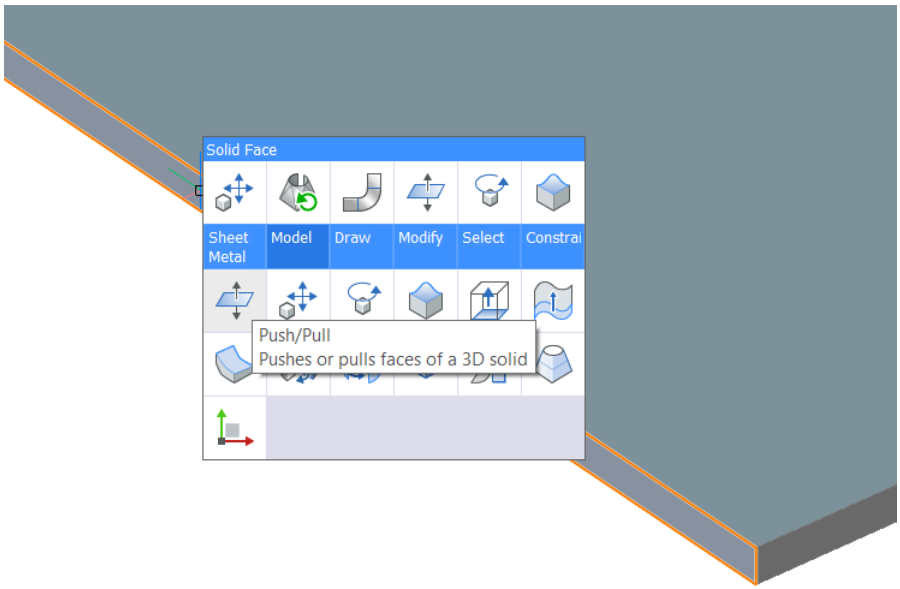
В рассматриваемом примере мы применим к простой модели несколько операций твердотельного моделирования, чтобы изменить форму ее стенок.

***Прим.:*** Мы работаем только с торцевыми гранями



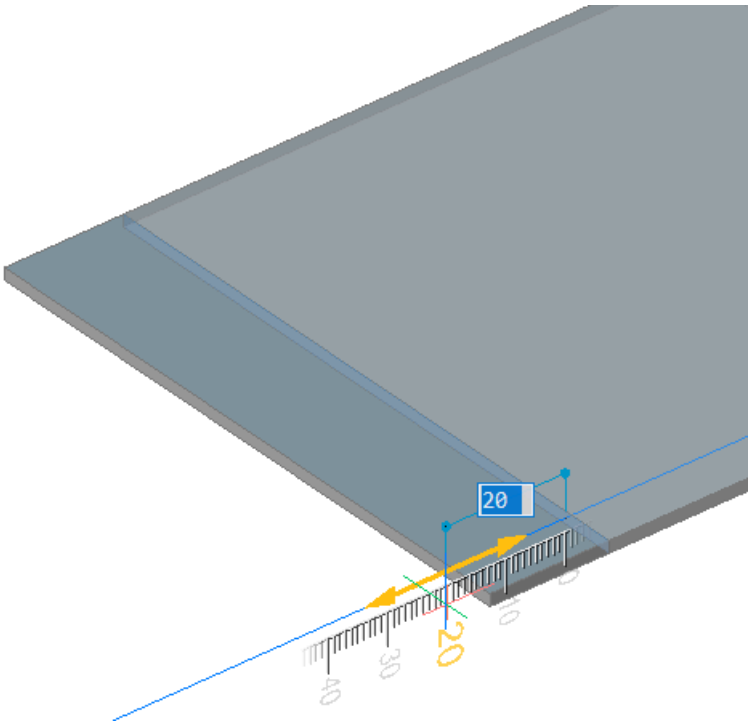
#### 21.5.11.2 Удлинение стенки

1 Установите курсор над торцевой гранью стенки и нажмите кнопку **Выдавить/Вытянуть** на закладке **квадро-меню**




**Моделирование**.

2 Перетяните грань и щелкните правой кнопкой

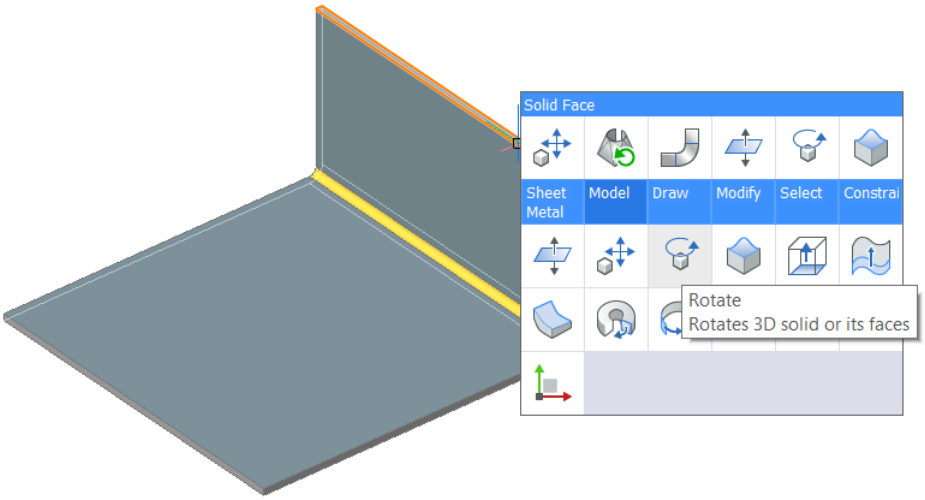



мыши.

Стенка детали будет удлинена на указанное расстояние.

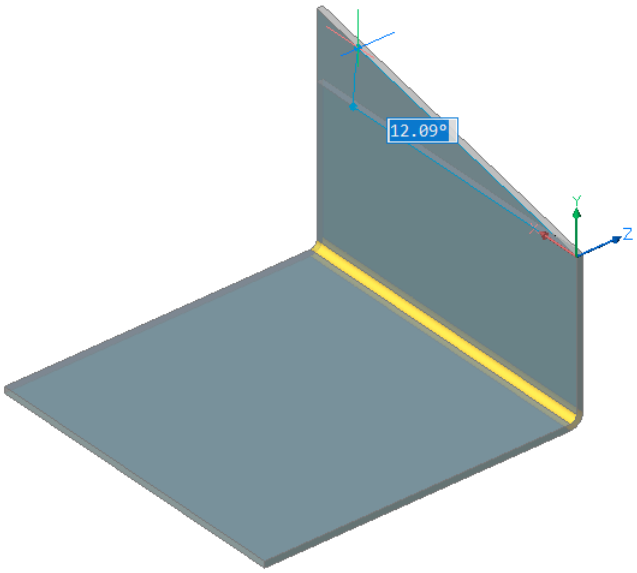
### 21.5.11.3 Поворот торцевой грани

1 Установите курсор над торцевой гранью стенки и нажмите кнопку **Повернуть** на закладке **квадро-меню**




**Моделирование**.

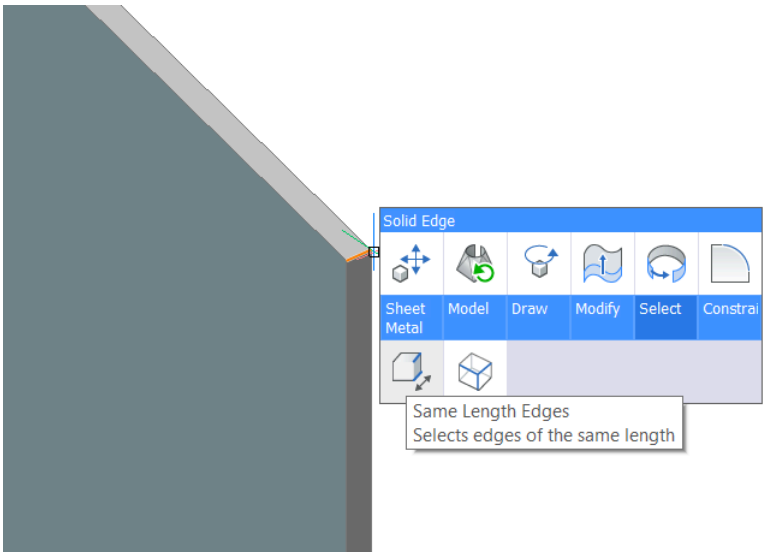
2 Укажите ось вращения, базовую точку и угол (см. описание команды



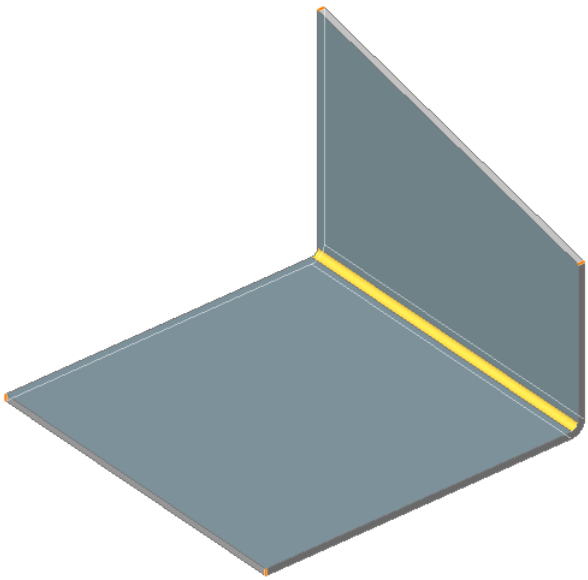

DMROTATE):

3 Выполним скругление острых ребер.

Установите курсор над ребром торцевой грани стенки и нажмите кнопку **Ребра равной длины** () на закладке **квадро-меню Выделение**.



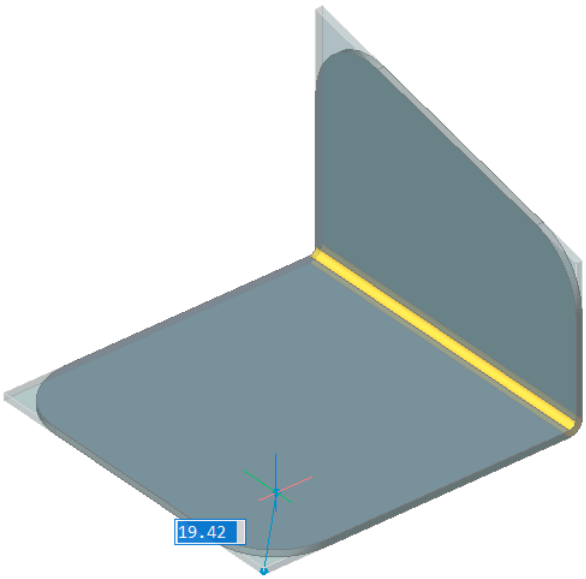

4 Нажмите клавишу **Enter**. Выбраны ребра с той же

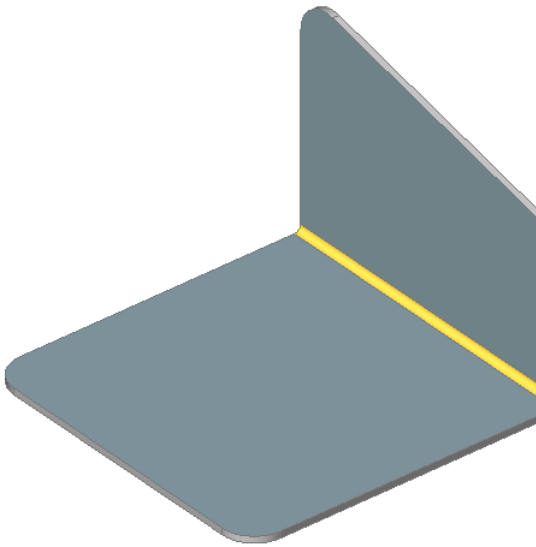


длиной.

5 Выберите инструмент **Скругление** на закладке **квадро-меню Моделирование**.

6 Укажите радиус

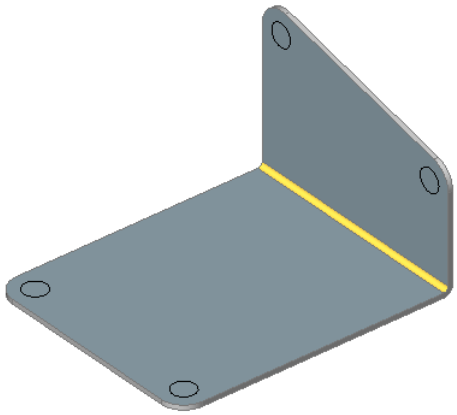


скругления:

### 21.5.11.4 Создание отверстия в стенке

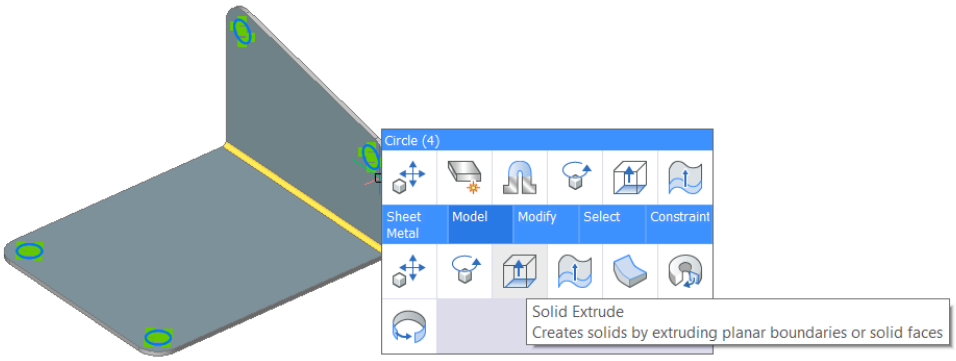
1 Постройте 4 окружности, как показано на приведенном ниже рисунке. При определении центральных точек окружностей используйте привязку к



центру.

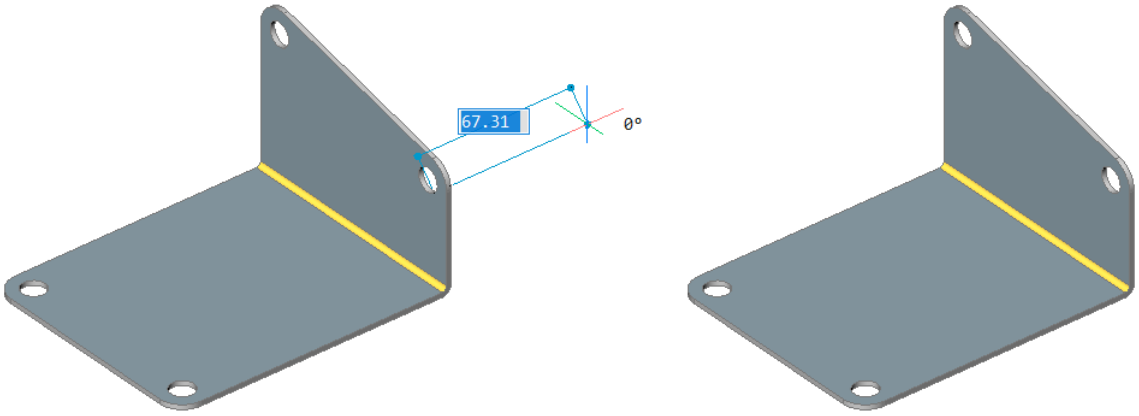
2 Выберите окружности.

3 Выберите инструмент **Выдавить** на закладке **квадро-меню**




**Моделирование**.

4 Переместите курсор для создания

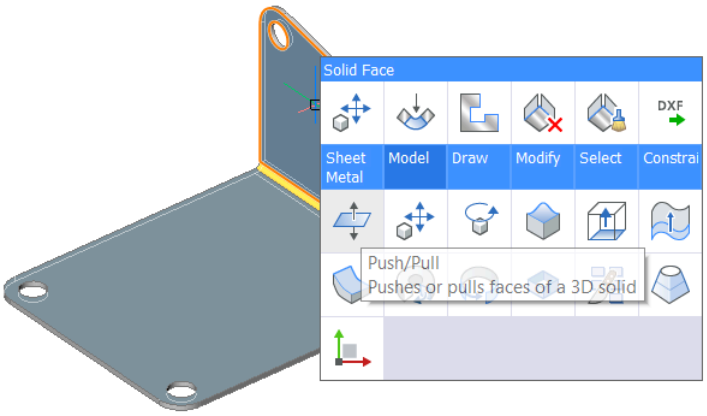



отверстия.

### 21.5.11.5 Перемещение стенки

1 Выполните одно из следующих действий:

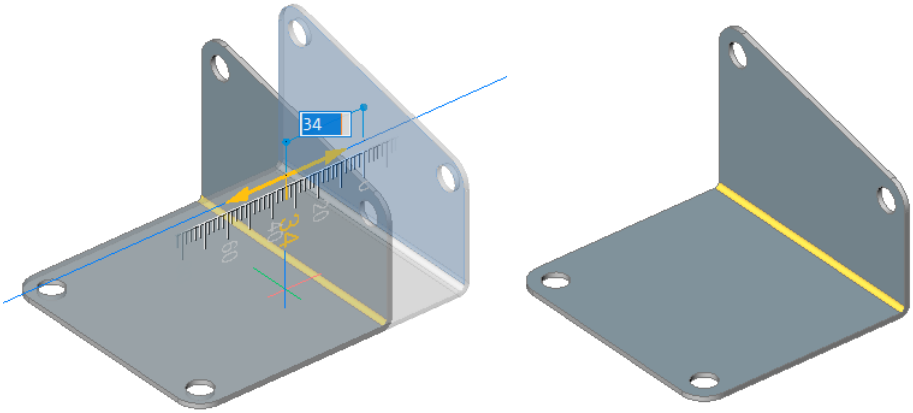
- Нажмите кнопку **Выдавить/Вытянуть** () на панели инструментов **Прямое моделирование**.
- Выберите **Прямое моделирование > Выдавить/Вытянуть** в меню **Моделирование**.
- Выберите инструмент **Выдавить/Вытянуть** () на панели **Редактирование** закладки ленты **Листовой металл**.
- Примените к грани команду *DMPUSHPULL*.
- Установите курсор над гранью перемещаемой стенки и нажмите кнопку **Выдавить/Вытянуть** на закладке **квадро-меню**




**Моделирование**.

2 Укажите новое расположение стенки.

***Прим.:*** Обратите внимание, что для **элементов листового металла** толщина листа сохраняется.




### 21.5.11.6 Поворот стенки

Вы можете выполнить поворот стенки с помощью команды *DMROTATE* (), но обычно удобнее использовать команду *SMFLANGEROTATE* (), которая автоматически выбирает ось поворота.

1 Запустите команду *SMFLANGEROTATE* ().

Запрос программы: Укажите поверхность листа для поворота:

Будут выделены грани стенок, находящиеся под курсором.

2 Щелкните на выбранной грани листа.

Положение стенки будет меняться динамически при изменении положения курсора.

При повороте будет отображаться текущее значение угла поворота относительно базовой поверхности.

3 Выполните одно из следующих действий:

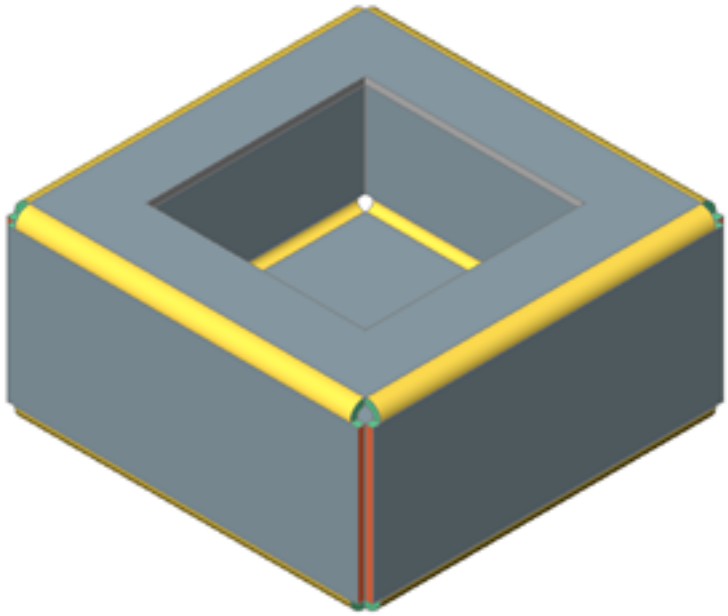
- Укажите точку.
- Введите значение угла в поле динамического ввода.
- Нажмите клавишу **TAB** и укажите абсолютное значение угла поворота в поле

динамического размера.

***Прим.:*** При выполнении команды *DMROTATE* элементы соединения поворачиваемой стенки с другими стенками удаляются автоматически.

### 21.5.11.7 Разделение стенки

В некоторых случаях необходимо выполнять разделение листа металла для минимизации расхода материала. Например, если в листе имеется большой проем как показано на приведенном ниже рисунке:



1 Запустите команду *SMSPLIT* ( ).

Запрос программы: Выберите грань стенки или лофт-гиба:

2 Выберите грань стенки.

Запрос программы:

Укажите отрезки или ребра для разделения листа или [Умное разделение/Создать новую линию/Параметры выбора] <Создать новую линию>:

3 Если необходимая линия на грани листа отсутствует, щелкните правой кнопкой мыши.

Запрос программы: Начальная точка:

4 Укажите начальную точку линии.

Запрос программы: Конечная точка линии:

5 Укажите конечную точку линии:

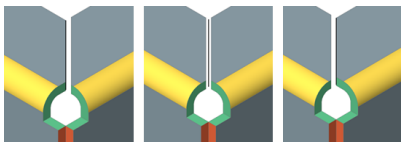
Запрос программы: Разделить по Центру/Слева/Справа/<Принять модель>:

6 Для определения положения реза относительно линии выполните одно из следующих действий:

- Нажимая клавишу **CTRL** выберите необходимую опцию **По центру**/**Слева**/**Справа**.

Если для переменной **TIPS** установлено значение ON, выбранная опция будет отображаться на виджете Ассистент.

- Выберите в контекстном меню опцию **Центр**, **Слева** или **Справа**.
- Введите опцию в командной строке и нажмите клавишу **Enter**.



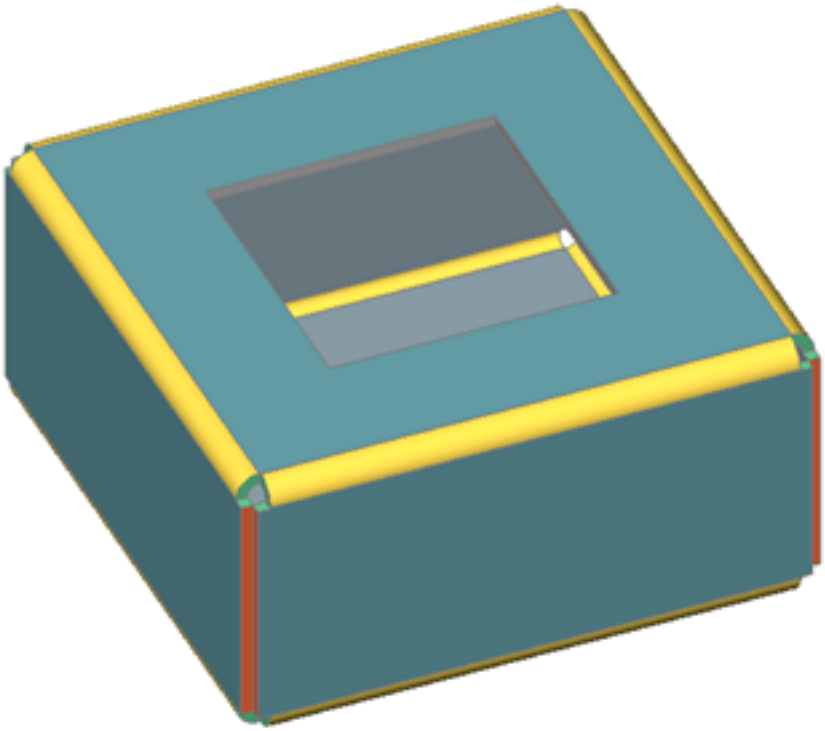
Стенка будет разрезана по линии: слева, по центру или справа.

***Прим.:*** При выполнении разделения создается элемент Скос. Вы можете восстановить начальную стенку, удалив объект Скос из списка объектов в **Обозревателе конструкции**.

### 21.5.11.8 Интеллектуальные инструменты для разделения стенки

Приведенные ниже примеры демонстрируют другие возможности команды *SMSPLIT*:

- "Умное" разделение листа путем указания одной угловой точки.
- Разделение гиба и преобразование его элементов, примыкающих к углу, в



стык.

1 Выполните одно из следующих действий:

- Нажмите кнопку **Умное разделение** ( ) на закладке ленты **Листовой металл**.
- Выберите инструмент **Умное разделение** в меню **Листовой металл > Разделить**.
- Введите в командной строке *SMSPLIT*.

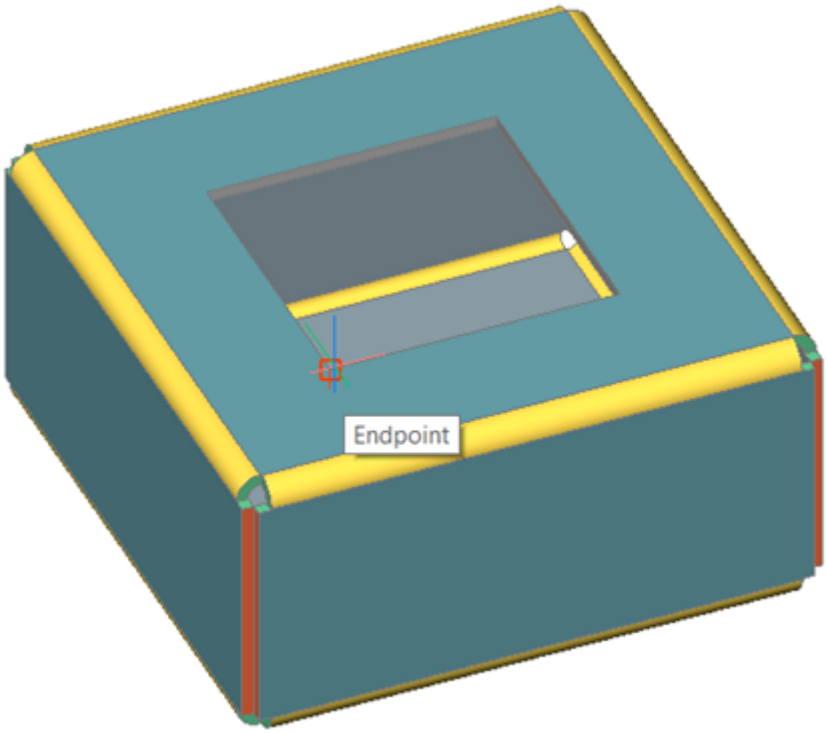
Запрос программы: Выберите грань стенки или лофт-гиба:

2 Выберите грань стенки.

Запрос программы:

Укажите отрезки или ребра для разделения листа или [Умное разделение/Создать новую линию/Параметры выбора] <Создать новую линию>:

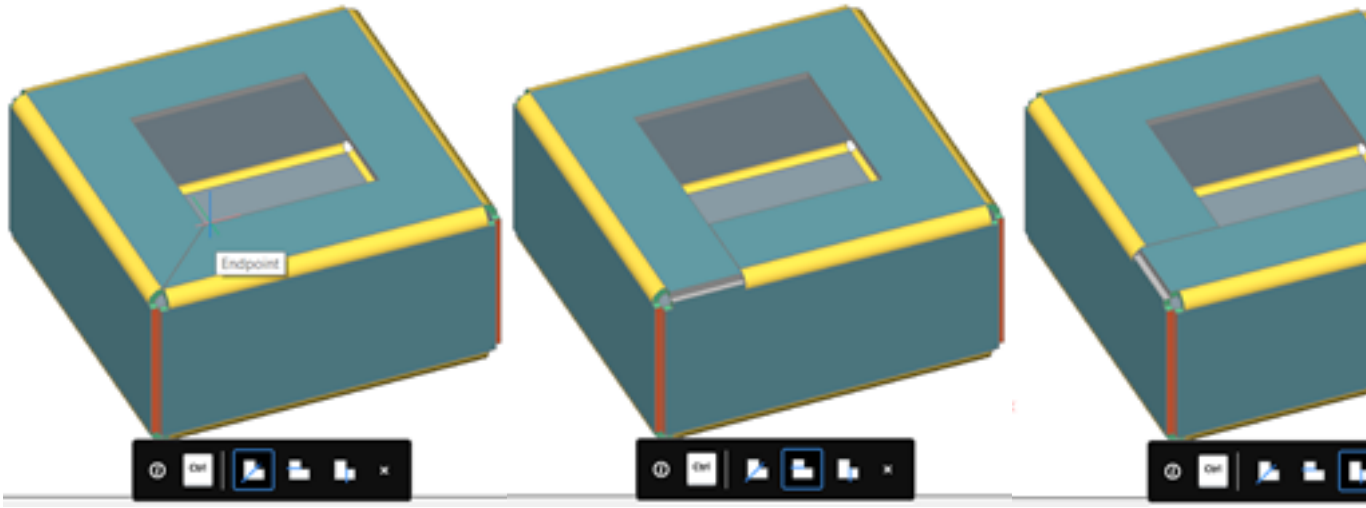
3 Выберите опцию **Умное разделение** и укажите угловую




точку:

Запрос программы: Разделить [Центр/Слева/Справа/Принять] <Принять>:

Доступные опции:



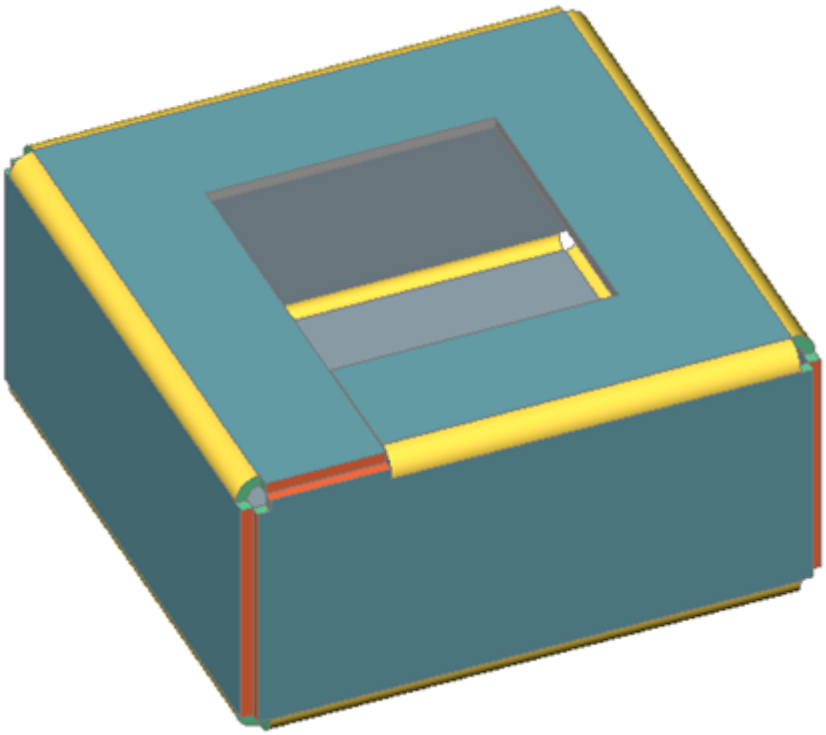

***Прим.:*** Преобразование части гиба в стык не связано с применением **Умного разделения**.

4 Для выбора опции выполните одно из следующих действий:

- Нажмите клавишу **CTRL** несколько раз.

  Если для переменной **TIPS** установлено значение ON, выбранная опция будет отображаться на виджете Ассистент.
- Выберите в контекстном меню опцию **Центр**, **Слева** или **Справа**.
- Введите опцию в командной строке и нажмите клавишу **Enter**.

  Стенка будет разрезана по линии, а гиб частично преобразован в стык:



При выборе опции **По центру** и включенном режиме **Smart Features**, угловые вырезы, прилегающие к разделению, будут преобразованы в подрезы гиба.

### 21.5.11.9 Сгиб стенки

Команда *SMFLANGEBEND* сгибает одну или несколько стенок по заданной линии. Вы можете сначала нарисовать линию, а затем запустить команду или использовать опцию **Создать линию гиба**.

**Одновременный изгиб нескольких стенок**

1 Запустите команду *SMFLANGEBEND*.

  Запрос программы: Укажите грань листа:

2 Выберите стенку, которую необходимо согнуть.

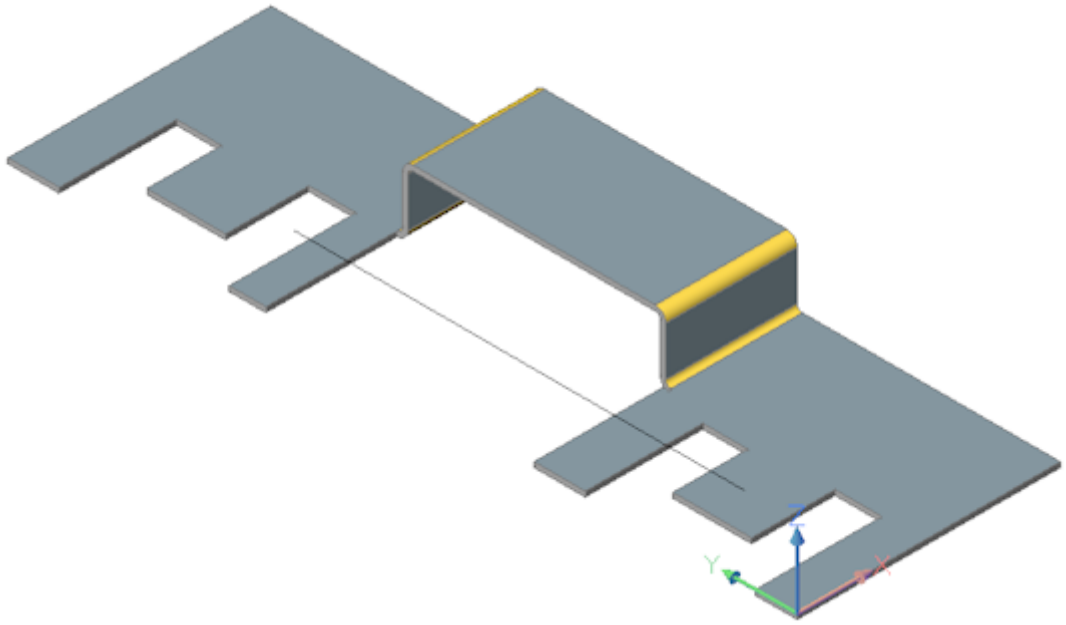
  Запрос программы: Укажите линию или ребро для выполнения гиба или начертите [Создать линию гиба/Параметры выбора] <Создать линию гиба>:

3 Выполните одно из следующих действий:

- Укажите линию, проходящую по граням сгибаемых стенок.
- Выберите опцию **Создать линию гиба**.

  Программа выдаст запросы на указание двух точек, определяющих новую линию гиба.

4 Запрос программы: Укажите точку на стороне листа, которая будет



перемещаться:

Укажите точку с той стороны от линии, которую вы хотите переместить.

Выбранная грань будет сгибаться в динамическом режиме.

Запрос программы: Введите угол гиба или укажите [Радиус/Переключить сторону]:

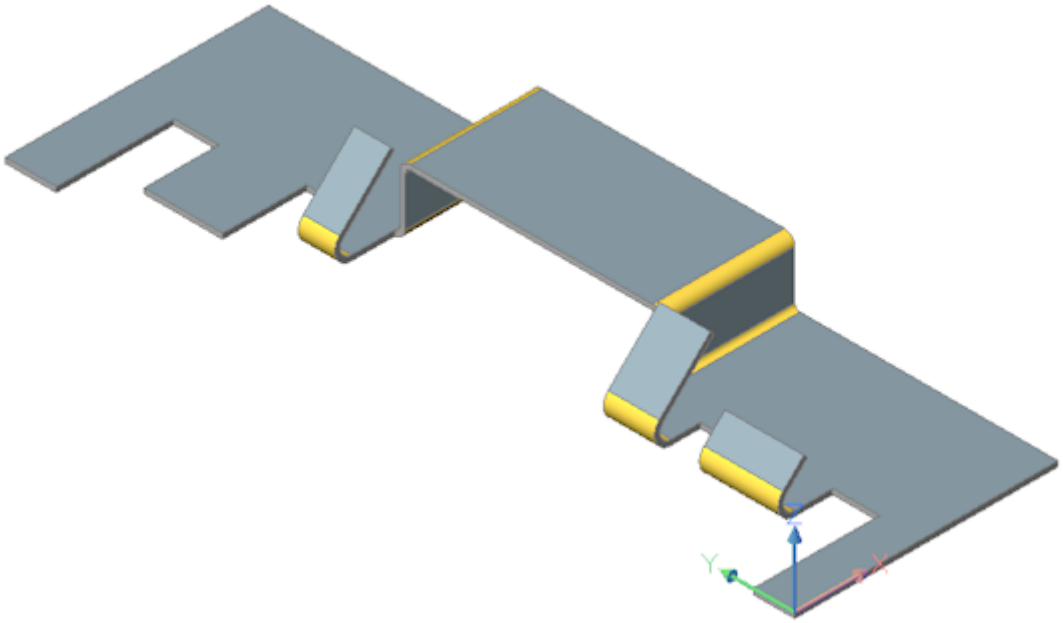
5 При необходимости, введите значение угла гиба, используя радиус гиба по умолчанию.

6 При необходимости, выберите опцию **Радиус**.

- Запрос программы: Укажите радиус гиба [Назад] <Назад>:

  Введите значение, заменяющее значение радиуса изгиба, предложенное по умолчанию, и нажмите клавишу **Enter**.
- Запрос программы: Введите угол гиба или укажите [Радиус/Переключить сторону].

  Введите значение угла гиба.



### 21.5.11.1 Соединение стенок

**0**

Команда *SMFLANGECONNECT* замыкает зазоры между двумя произвольно ориентированными стенками.

1 Запустите команду *SMFLANGECONNECT*.

Запрос программы: Укажите плоские торцевые грани двух листов:

Будут выделены грани стенок, находящиеся под курсором.

2 Выберите торцевую грань первого листа.

Запрос программы:

Объектов/подобъектов в наборе: 1

Укажите плоские торцевые грани двух листов.

3 Выберите торцевую грань второго листа.

Выбранные торцевые грани будут соединены.

***Прим.:*** После соединения двух стенок элемент Стык может быть преобразован в элемент Гиб.

## 21.5.12 Working with Bends and Junctions

Two adjacent flanges of sheet metal parts are connected by a bend or a junction. As different from many other mechanical CAD systems, in BricsCAD, bends and junctions are separate features. A bend can be switched to a junction and vice versa, which allows you to fix or optimize the unfolded representation of a sheet metal part easily.

### 21.5.12.1 Converting Hard Edges to Bends or Junctions

If adjacent flanges of a sheet metal part are not connected by a bend or junction, the part cannot be unfolded or exported to a CAM system. Therefore such hard edges must be converted to bends or junctions.

**Converting hard edges to bends**

1 Do one of the following:

- Hover over a hard edge and click the **Create Bend** () tool in the **Quad** menu
- Click the **Create Bend** () tool button in the **Sheet Metal** ribbon tab
- Choose **Create Bend** in the **Sheet Metal** menu
- Type *SMBENDCREATE* or the Command line.

  You are prompted:

  Select hard edges or junctions or flanges or 3D solids or [Entire model] <Entire model>:

2 Select a flange face or a hard edge.

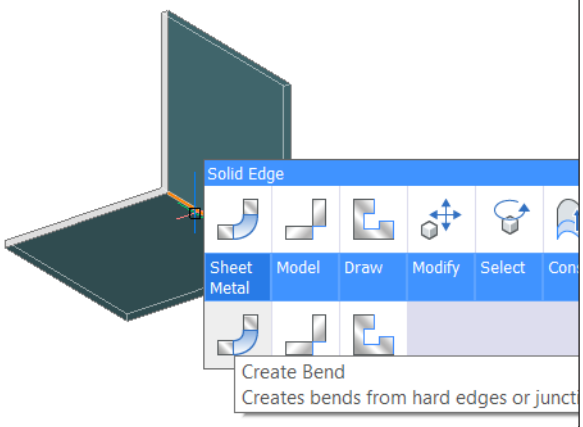
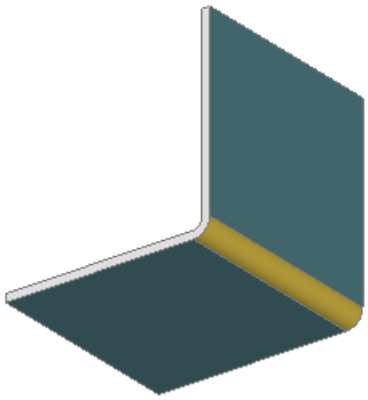
You are prompted:

Entities in set: 1

Select flange faces or hard edges:

3 Do one of the following:

- Select more faces or hard edges.
- Right-click or press **Enter** to create the bends or junctions on the selected items.

| | |
|---|---|
|  |  |
| Selected a hard edge | A bend is created |

**Converting Hard Edges to Junctions**

1 Do one of the following:

- Hover over a hard edge and click the **Create Junction** () tool in the **Quad** menu.
- Click the **Create Junction** () tool buttons in the **Sheet Metal** ribbon tab.

- Choose **Create Junction** in the **Sheet Metal** menu.
- Type *SMJUNCTIONCREATE* in the Command line.

  You are prompted: Select hard edges or bends or flanges or 3D solids [Entire model] <Entire model>:

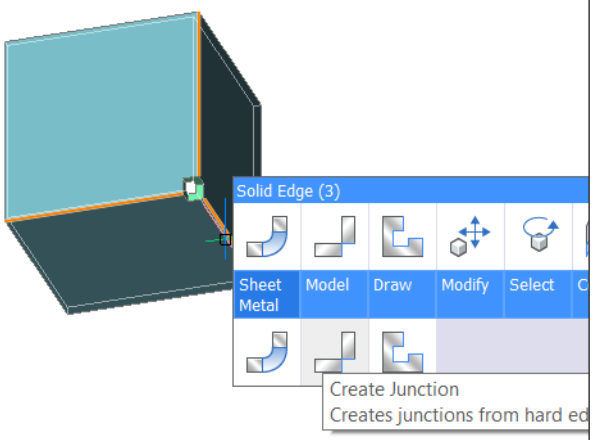
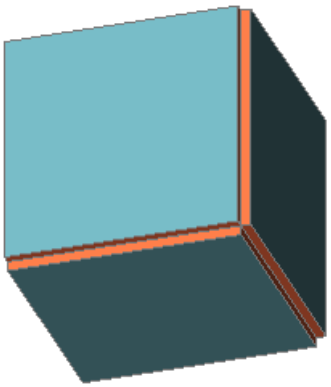
2 Select a flange face or a hard edge.

  You are prompted:

  Entities in set: 1

  Select flange faces or hard edges:

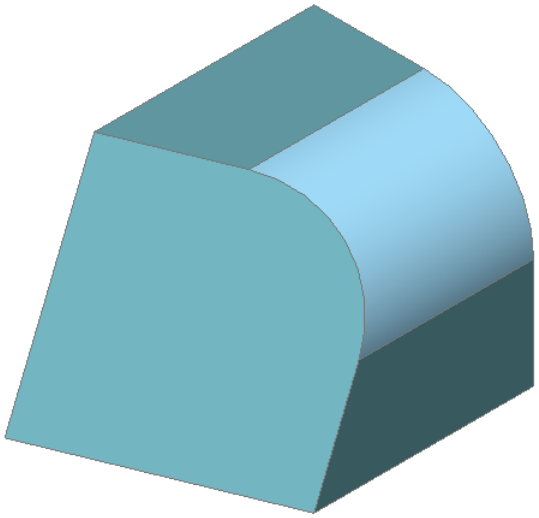
3 Do one of the following:
- Select more faces or hard edges.
- Right-click or press **Enter** to create the bends or junctions on the selected items.

| | |
|---|---|
|  |  |
| Selected hard edges.<br>Notice that a corner relief must be created first. | Junctions created.<br>Notice that unnecessary material cuts have been removed due to **Smart Features**. |

### 21.5.12.2 Creating Curved Junctions

Let us have a look at a more complex scenario where a junction has to be cut over a non-linear edge. To make the part unfoldable, we must:

- Cut the bend relief for one of the linear hard edges.
- Cut the junctions for the remaining linear and curved hard edges.



- Create a bend on the remaining hard edge.

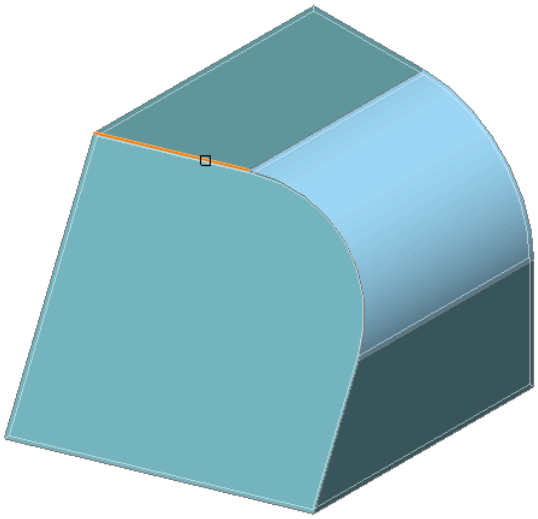
1 Create bend reliefs.

Do one of the following:

- Click the **Create Relief** tool button ( ) on the **Sheet Metal** ribbon tab.
- Choose **Create Relief** in the **Sheet Metal** menu.
- Type *SMRELIEFCREATE* in the Command line.

You are prompted:

Select a hard edge, bend face, flange face or 3D solid [Entire model]:



2 Select a hard edge.

You are prompted:

Select another hard edge or bend face for the corner relief:

3 Press **Enter** or right-click.

You are prompted:

Enter relief size by bend radius ratio or [force Bend reliefs/Auto] <Auto>:

4 Press **Enter** or right-click.

2 bend reliefs are created for the edge:



We created 2 bend reliefs because a hard edge side with a "free" vertex needs a relief in order to make orthogonal thickness face.

We created a bend relief on the side adjacent to the lofted bend because this feature is different from a regular bend: creating a corner relief there would be too big.

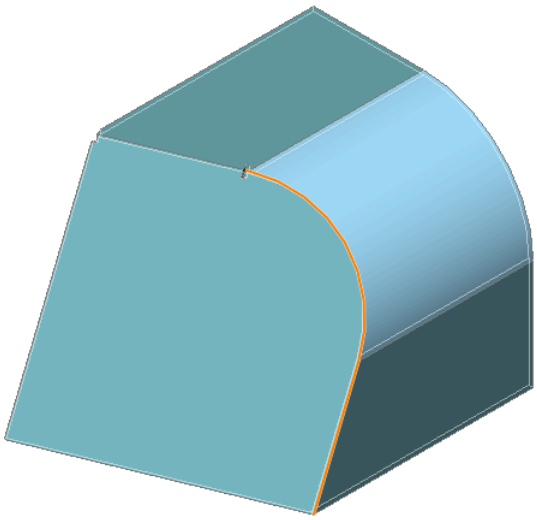
5 Create 2 junctions.

Do one of the following:

- Click the **Create Junction** ( ) tool buttons in the **Sheet Metal** ribbon tab;
- Choose **Create Junction** in the **Sheet Metal** menu;
- Type *SMJUNCTIONCREATE* in the command bar.

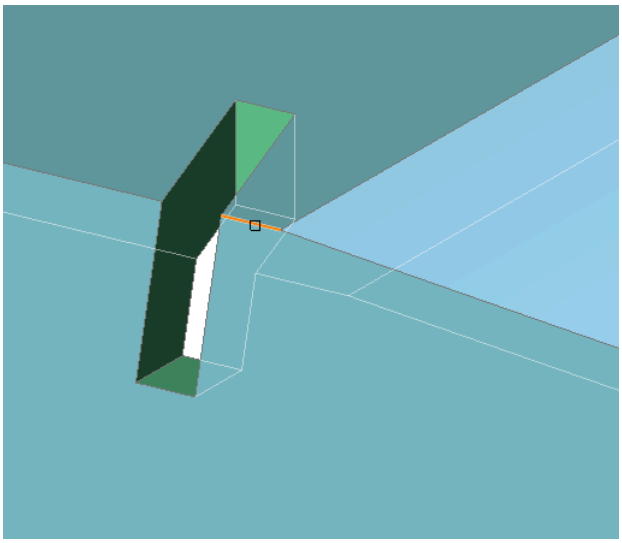
  You are prompted:

  Select hard edges or bends or flanges or 3D solids [Entire model] <Entire model>:



6 Select the following 3 hard edges:

Consider the small hard edge:



The cut is created. Parts which corresponded to linear edge get regular junction features.

7 Create a bend on the hard edge. Because it is the only hard edge left on the model, apply the command to the entire model.

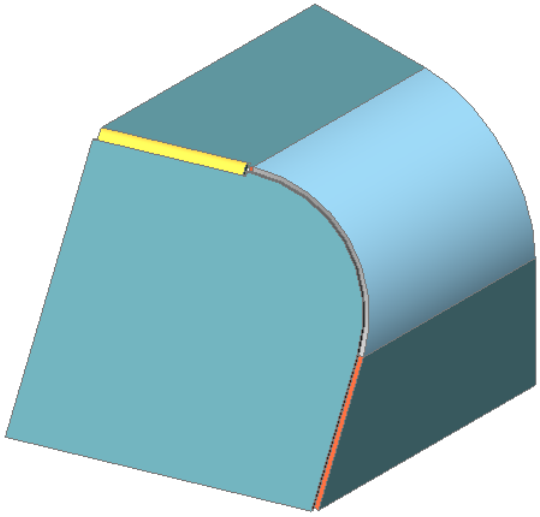
- Click the **Create Bend** ( ) tool buttons in the **Sheet Metal** ribbon tab.
- Choose **Create Bend** in the **Sheet Metal** menu.
- Type *SMBENDCREATE* or the Command line.

  You are prompted:

  Select hard edges or junctions or flanges or 3D solids or [Entire model] <Entire model>:

8 Press **Enter**.

The bend is created.



9 Unfold the part.

- Click the **Unfold Body** ( ) tool buttons in the **Sheet Metal** ribbon tab.
- Choose **Unfold Body** in the **Sheet Metal** menu.

- Type *SMUNFOLD* or the Command line.

  You are prompted:

  Select a flange or lofted bend face to start unfolding [Associative]:

10 Select some flange face.

   You are prompted:

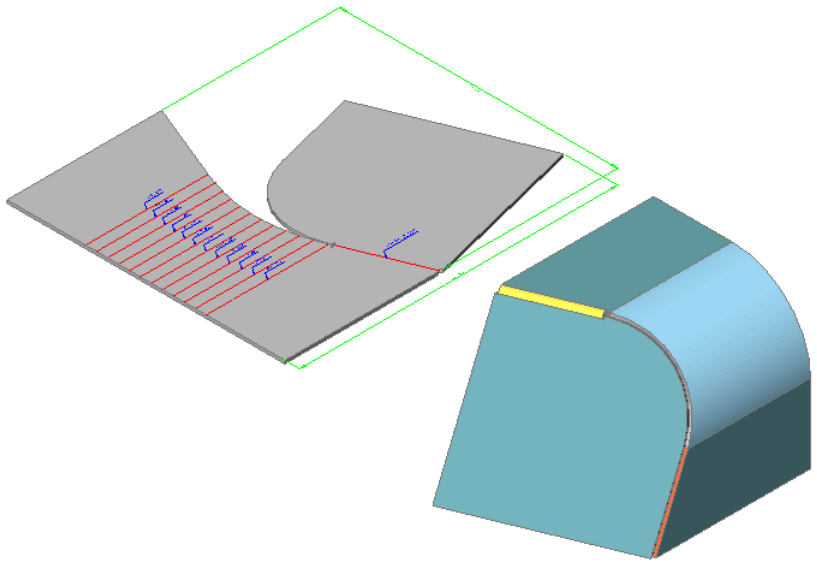
   Select position of the unfolded body:

11 Place the unfolded part.

   You are prompted:

   Validate the unfolded body and select an option [save 2D geometry/save 3D geometry/Optimize bend annotations/Keep] <Keep>:

12 Press **Enter** to keep the



result.

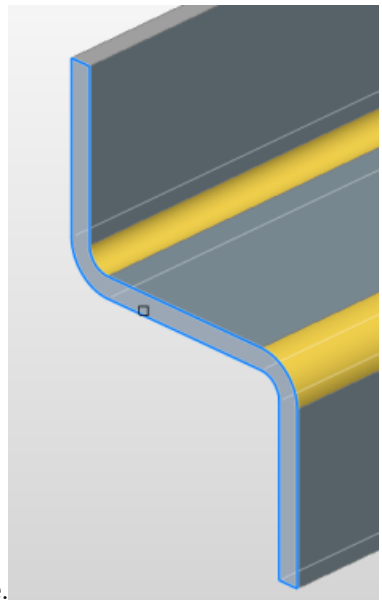
### 21.5.12.3 To Split a Thickness Face

To extend a flange which shares the same thickness face with other flanges, the thickness face must be split.

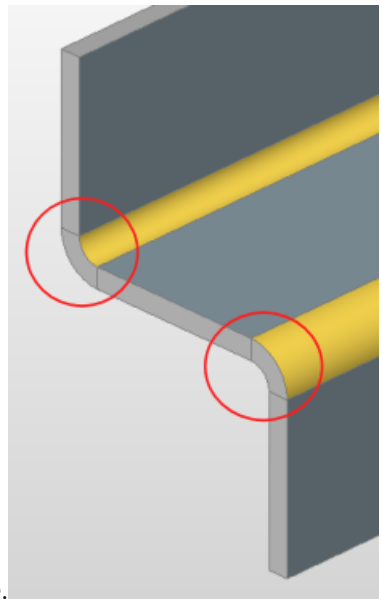
1 Launch the *SMIMPRINT* command ( ).

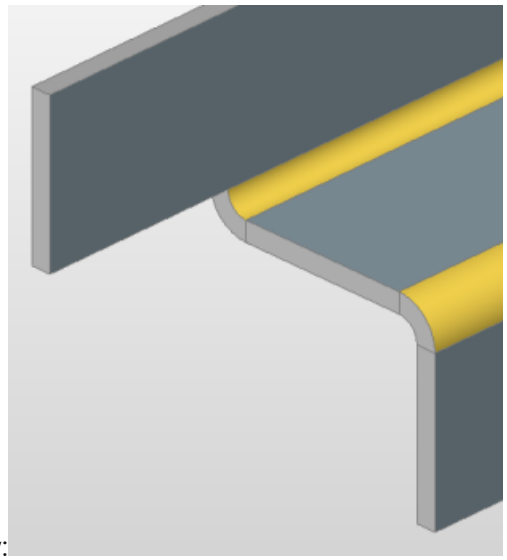
  Prompts you: Select a thickness face:

2 Select the shared thickness face.



3 Right-click or press **Enter** to split the face.





4 The flanges can now be extended separately:

5 Execute the *DMSIMPLIFY* command to remove the unneeded splits, thus unifying the coplanar thickness faces.

### 21.5.12.4 Changing the Bend Radius

**To change the overall bend radius:**

1 Select the root node in the **Mechanical Browser**.

2 Find the **Bend radius** group in the bottom part of the **Mechanical Browser**.

3 Do one of the following:

- Set **Type** to **Absolute Value**, then type the radius value in the **Value** field.

  The **Bend Radius** field changes accordingly.

- Set **Type** to **Thickness ratio**, then type a value in the **Value** field.

  The **Bend Radius** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

**To change the bend radius for a particular bend:**

1 Select the bend node in the **Mechanical Browser**. The selected bend highlights in the model. The Bend Properties display in the bottom part of the **Mechanical Browser**.

2 Do one of the following:

- Set the **Type** to **Global value**.

  The current global bend radius is applied to the selected bend.

- Set the **Type** to **Absolute value**, then type the radius value in the **Value** field.

  The **Bend Radius** field changes accordingly.

- Set the **Type** to **Thickness Ratio** and type a value in the value field.

  The **Bend Radius** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

### 21.5.12.5 Changing the Junction Gap

**To change the overall junction gap:**

1 Select the root node in the **Mechanical Browser**.

2 Locate the **junction gap** group in the bottom part of the **Mechanical Browser**.

3 Do one of the following:

- Set **Type** to **Absolute Value**, then type the gap value in the **Value** field.

  The **junction gap** field changes accordingly.

- Set **Type** to **Thickness ratio**, then type a value in the **Value** field.

  The **junction gap** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

**To change the junction gap for a particular bend:**

1 Select the bend node in the **Mechanical Browser**. The selected bend highlights in the model. The Bend Properties display in the bottom part of the **Mechanical Browser**.

2 Do one of the following:

- Set the **Type** to **Global value**.

  The current global junction gap is applied to the selected bend.

- Set the **Type** to **Absolute value**, then type the gap value in the **Value** field.

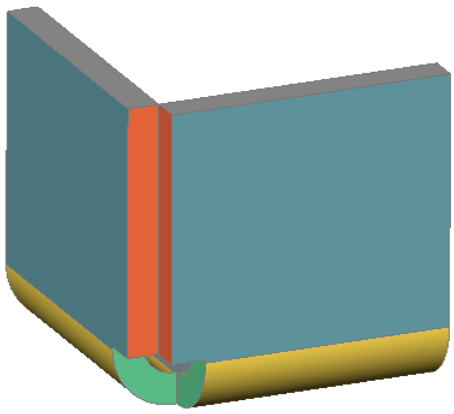
  The **junction gap** field changes accordingly.

- Set the **Type** to **Thickness Ratio** and type a value in the **Value** field.

  The **junction gap** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

### 21.5.12.6 Changing the Junction Type

By default, BricsCAD creates symmetric junctions, as shown in the image below:

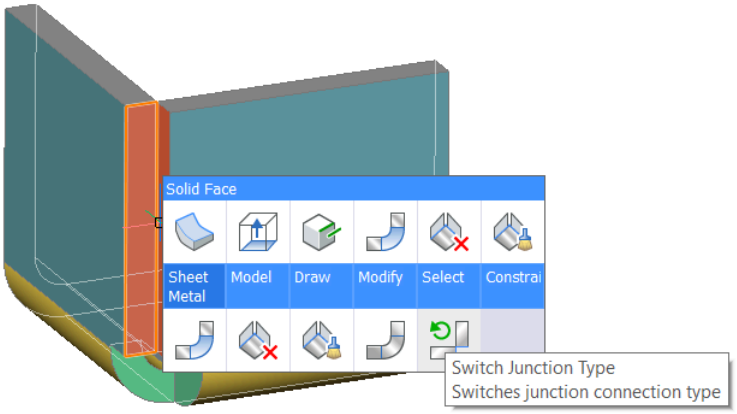


The *SMJUNCTIONSWITCH* command allows changing a symmetrical junction feature to one with overlapping faces.

**To change the junction type using the Quad menu:**

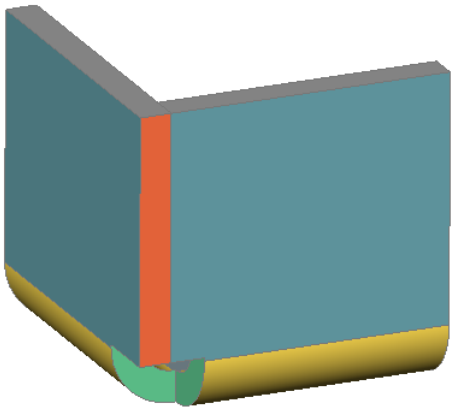
1 Hover the cursor over one the thickness faces of this junction.

2 Select **Switch Junction Type** in the **Sheet Metal** command group in the **Quad**




menu.

The selected thickness face is aligned with the flange face.



**To change the junction type in the Mechanical Browser:**

Method 1:

1 Select the corresponding junction in the tree.

The junction properties display in the bottom part of the **Mechanical Browser**.

2 Select the **Connection type** property.

3 Choose the desired junction type.

The options are: **Symmetric**, **Aligned to Flange_X** or **Aligned to Flange_Y**.

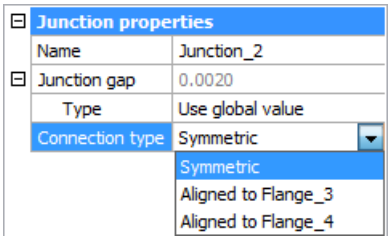


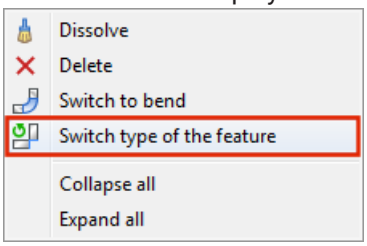

**Рисунок 131:** Junction properties

The junction is updated accordingly.

Method 2:

1 Select the corresponding junction in the tree.

A context menu displays:




2 Right-click and choose **Switch type of feature** in the context menu.

***Прим.:*** Both symmetric and aligned junctions maintain the same gap value between connected flanges.

### 21.5.13 Working with Hems

A hem feature is produced when the edge of a sheet metal flange is folded back on itself. *SMHEMCREATE* simplifies the creation of sheet metal hems. In BricsCAD hems have their own set of properties depending on the type, "flange" and "bend" type of the geometry. Hem features are recognized by the *SMCONVERT* command and have a special symbol when unfolded with *SMUNFOLD*.

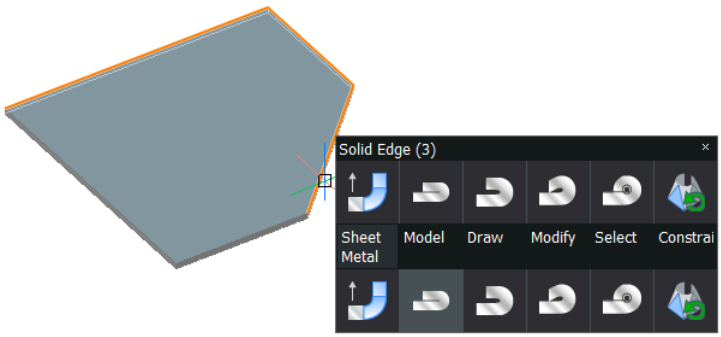
#### 21.5.13.1 Create a Hem

1 Do one of the following:

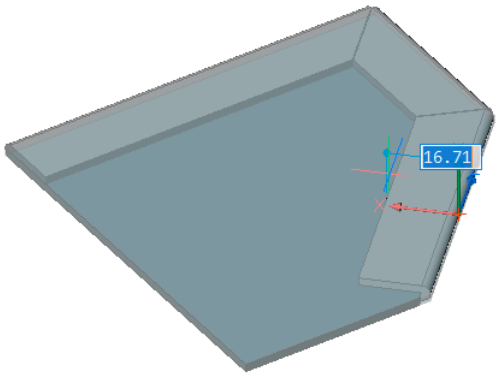
- Select several linear edges on the flange.
- Hover the cursor over a linear edge of the flange.

2 Do one of the following:

- Select **Create a Closed Hem** () tool in the **Quad** menu.
- Click the **Create a Closed Hem** () tool buttons in the **Sheet Metal** ribbon tab.
- Choose **Create a Closed Hem** in the **Sheet Metal** menu.
- Type *SMHEMCREATE* in the Command

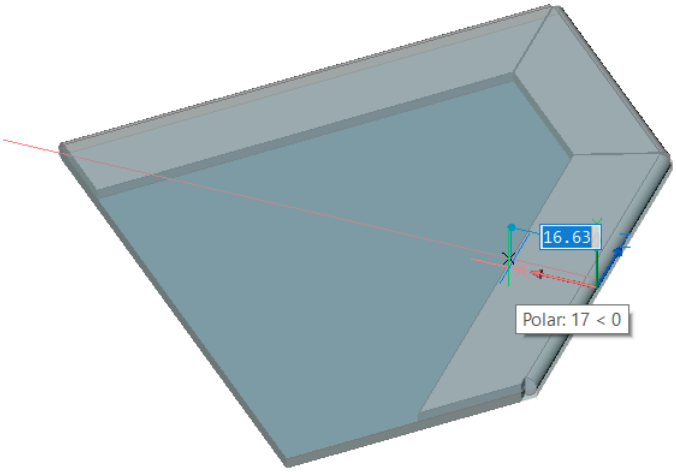



line.



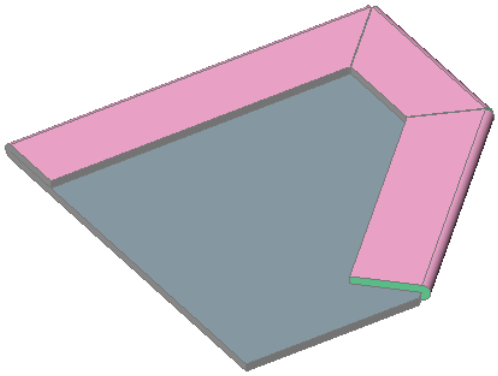

3 Hover the cursor to adjust the hem length:

4 Use the **Ctrl** widget or **Toggle tapering** option to flip the adjustment of a hem to the




edge:



5 When finished, press **Enter**:

Hems and auxiliary features (reliefs and miters) are created.

### 21.5.13.2 Modify Hem Parameters

To modify Hem parameters select the feature and use the **Mechanical Browser**. Different parameters are available, depending on the type of the hem.

**Таблица 43: Hem parameters**

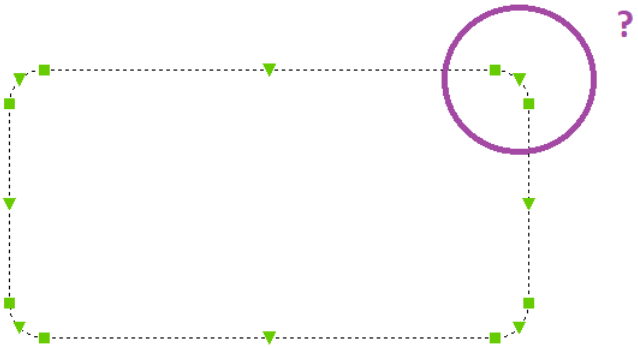
| Type of hem | Available parameters |
|---|---|
| Closed | Radius<br>Length |
| Open | Radius<br>Length |
| Teardrop | Radius - change the angle of the hem<br>Length - change the angle of the hem |
| Round | Radius - change the "planar" part of the hem |

In the case of any parametric change, the hem type is preserved and the geometry is moved inside, to maintain external dimensions.

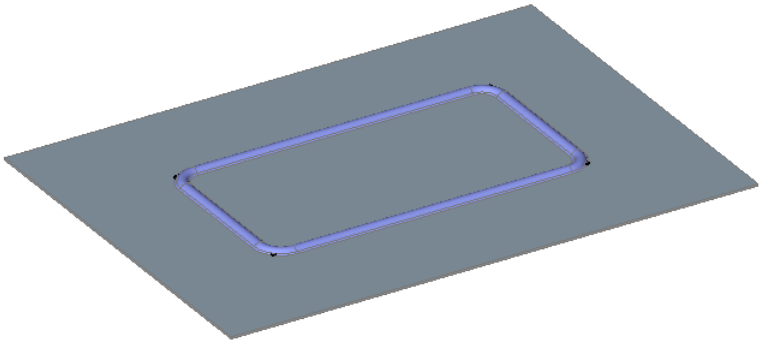
### 21.5.13.3 Erase a Hem

When a hem feature is erased, BricsCAD intelligently restores the geometry's dimensions and reliefs.

The flange part below has external dimensions equal to 100.

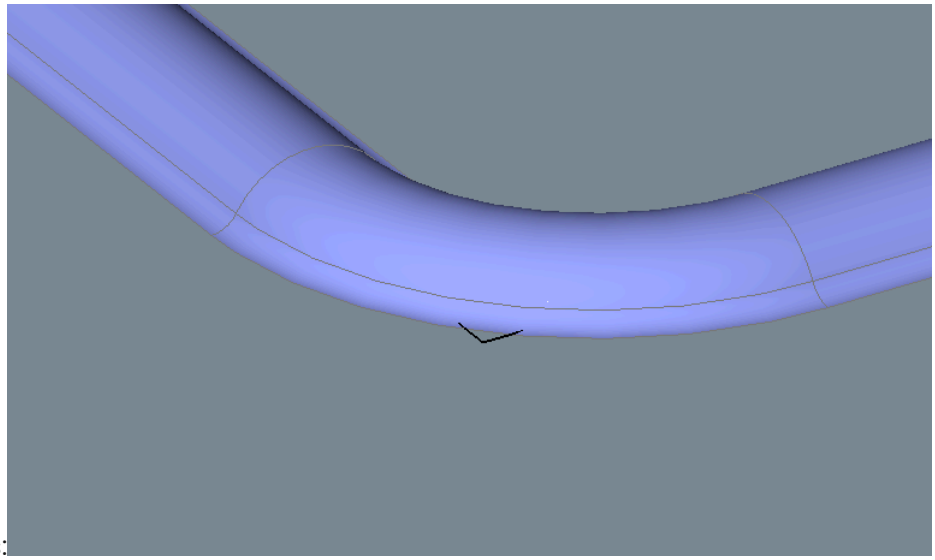



1 Use the Width option of the *SMHEMCREATE* command to create a partial hem on a single



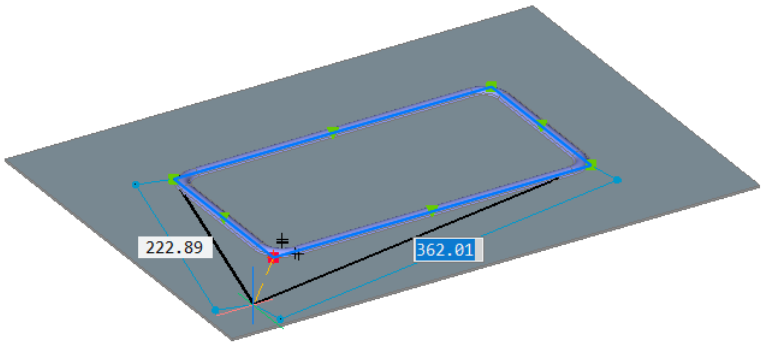
edge:

2 Create hems on 2 other

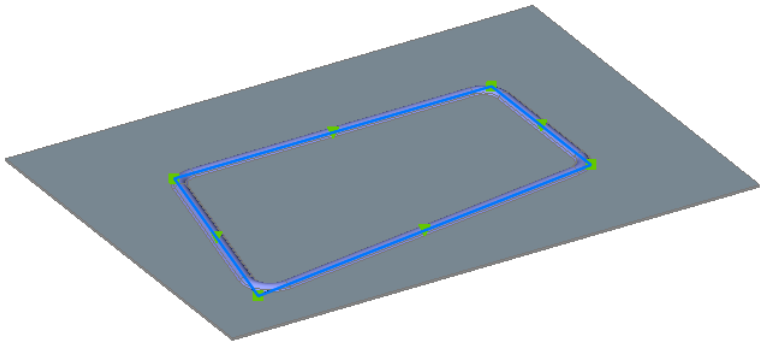


edges:

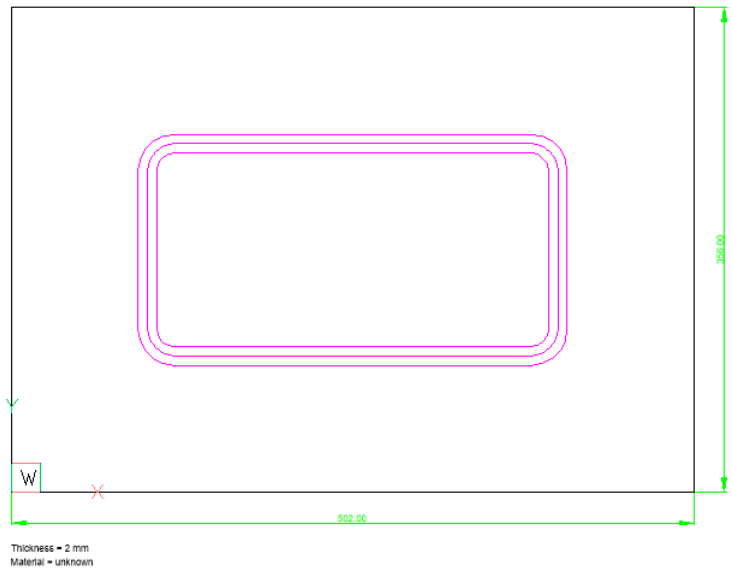
These operations lead to the appearance of complex relief geometry:




***Прим.:*** The external dimensions of the part are preserved.

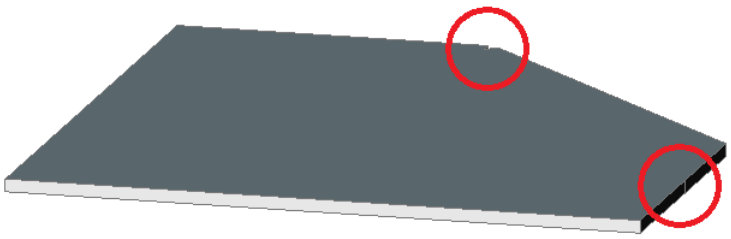


3 Select the hem features and delete

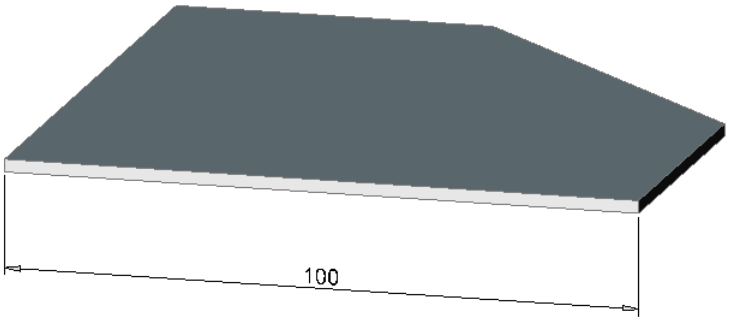



them.

Some imperfections are left on the corner relief feature and the imprinted edge:



4 Delete the corner relief feature and use *DMSIMPLIFY* to get the original part with original




dimensions.

### 21.5.13.4 Related commands

The *MECHANICALBROWSEROPEN* command displays the Mechanical Browser.

The *SMDELETE* command restores a hard edge between 2 flanges to remove a bend or a junction - removes a flange and all adjacent bends. The adjacent flanges are extended up to a junction configuration and the flange is deleted. Restores geometry removed by the miter.

The *SMHEMCREATE* command creates one or more hems to a sheet metal part by pulling one or more edges of an existing flange.

## 21.5.14 Работа с шип-пазами

Элемент Шип-паз используется для создания простых и надежных соединений между деталями из листового металла.

Шип-пазы создаются из стыков.

### 21.5.14.1 Создание одиночного шип-паза

Для создания одиночного шип-паза по умолчанию используется команда *SMTABCREATE*. Этот элемент имеет только один паз.

1 Выполните одно из следующих действий:

- Подведите курсор к грани стыка и нажмите кнопку **Создать одиночный шип-паз** ( ) на закладке квадро-меню **Листовой металл**.
- Нажмите кнопку **Создать одиночный шип-паз** ( ) на закладке ленты **Листовой металл**.
- Выберите инструмент **Создать одиночный шип-паз** в меню **Листовой металл**.
- Введите в командной строке команду *SMTABCREATE*.

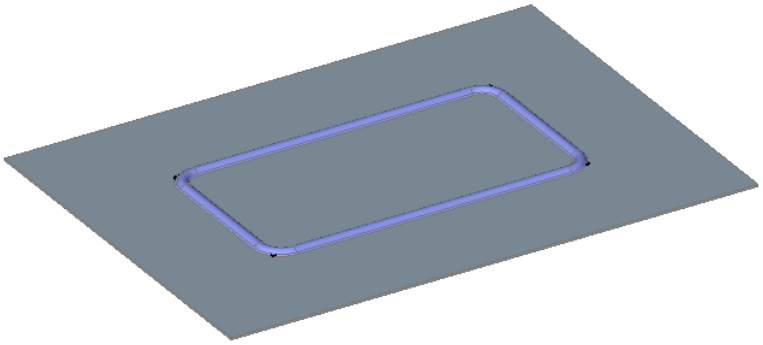
  Запрос программы: Выберите стык для создания одиночного шип-паза или [Массив шип-пазов/Параметры выбора]:

2 Выберите грань стыка.

  Запрос программы: Введите длину шипа или [Точка центра]:

3 Укажите длину шипа.

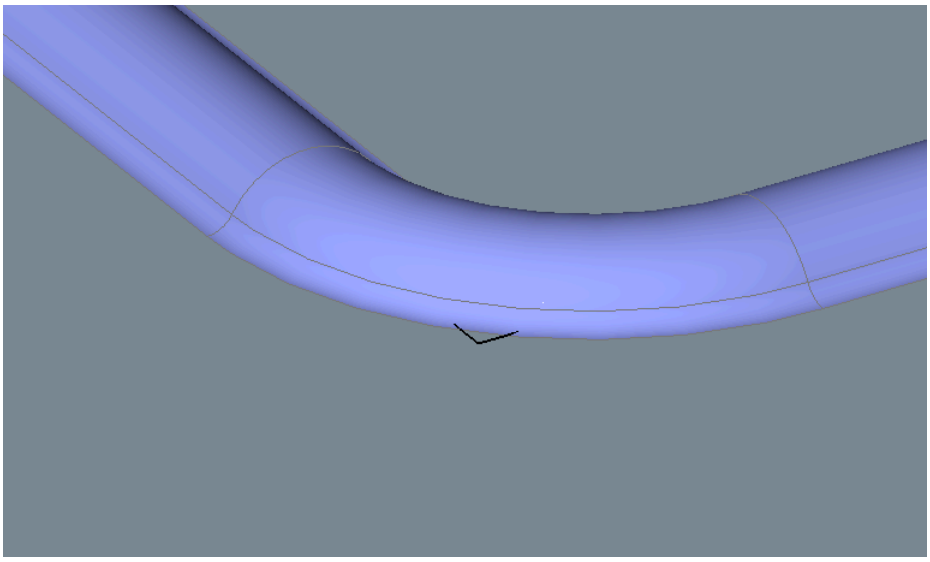
Шип размещается симметрично относительно точки центра. Введите значение длины шипа в поле динамического ввода.



4 Выберите опцию **Точка центра** для определения положения шипа.

Запрос программы: Выберите точку центра или [Длина]:

При указании точки центра шипа отображаются два динамических размера, которые определяют положение этой точки относительно концов стыка:



5 Выберите опцию **Длина**, чтобы указать длину, или нажмите клавишу **Enter** для завершения команды.

### 21.5.14.2 Создание массива шип-пазов

Массив шип-пазов, в отличие от одиночного шип-паза, имеет несколько пазов.

1 Чтобы создать массив шип-пазов, выполните одно из следующих действий:

- Подведите курсор к грани стыка и нажмите кнопку **Создать массив шип-пазов** ( ) на закладке квадро-меню **Листовой металл**.
- Нажмите кнопку **Создать массив шип-пазов** ( ) на закладке ленты **Листовой**

**металл**.

- Выберите инструмент **Создать массив шип-пазов** в меню **Листовой металл**.
- Введите в командной строке команду *SMTABCREATE*.

  Запрос программы: Выберите стык для создания одиночного шип-паза или [Массив шип-пазов/Параметры выбора]:

2 Выберите опцию **Массив**.

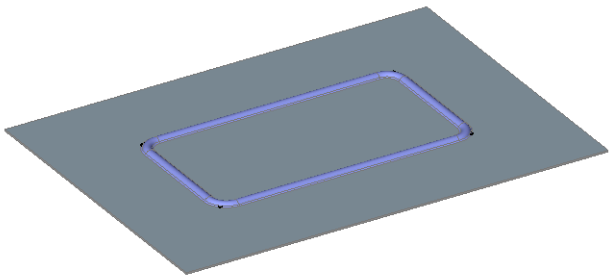
Запрос программы: Выберите стык или криволинейное острое ребро для создания массива шип-пазов или [Инвертированный криволинейный шип-паз/Одиночный шип/Параметры выбора]:

3 Выберите грань стыка.

Запрос программы: Подтвердите текущую конфигурацию шипа или [Перевернуть/Число пазов/Расстояние/Длина/Радиус скругления/Расстояние фаски/Принять] <Принять>:

4 Вы можете использовать отображаемые в запросе опции команды для изменения внешнего вида шип-паза. (см. пример ниже).

- **Число пазов = 3** - обратите внимание на несимметричность пазов. При выборе опции **Перевернуть** 3 шип-паза будут расположены на противоположной стороне стыка.
- **Длина = 8** - длина паза.
- **Расстояние = 40** - расстояние между центрами

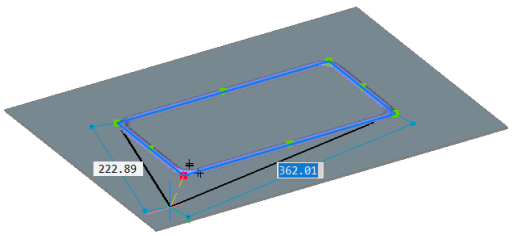
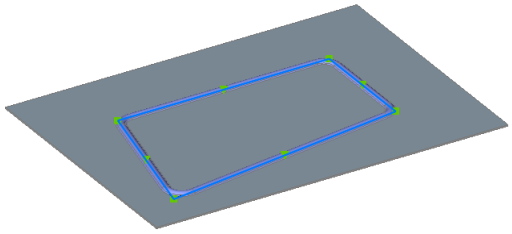


пазов.

5 Измените значения параметров **Радиус скругления** и **Расстояние фаски**, чтобы определить скругления и фаски для ребер.

***Совет:*** Если вы задали скругления или фаски по ошибке, используйте нулевые значения, чтобы отключить эти параметры.

6 После того, как вы установили все параметры, выберите **Принять** или нажмите клавишу **Enter**, чтобы создать массив шип-пазов и завершить команду.

| Скругления | Фаски |
| --- | --- |
|  |  |

### 21.5.14.3 Создание криволинейного массива шип-пазов

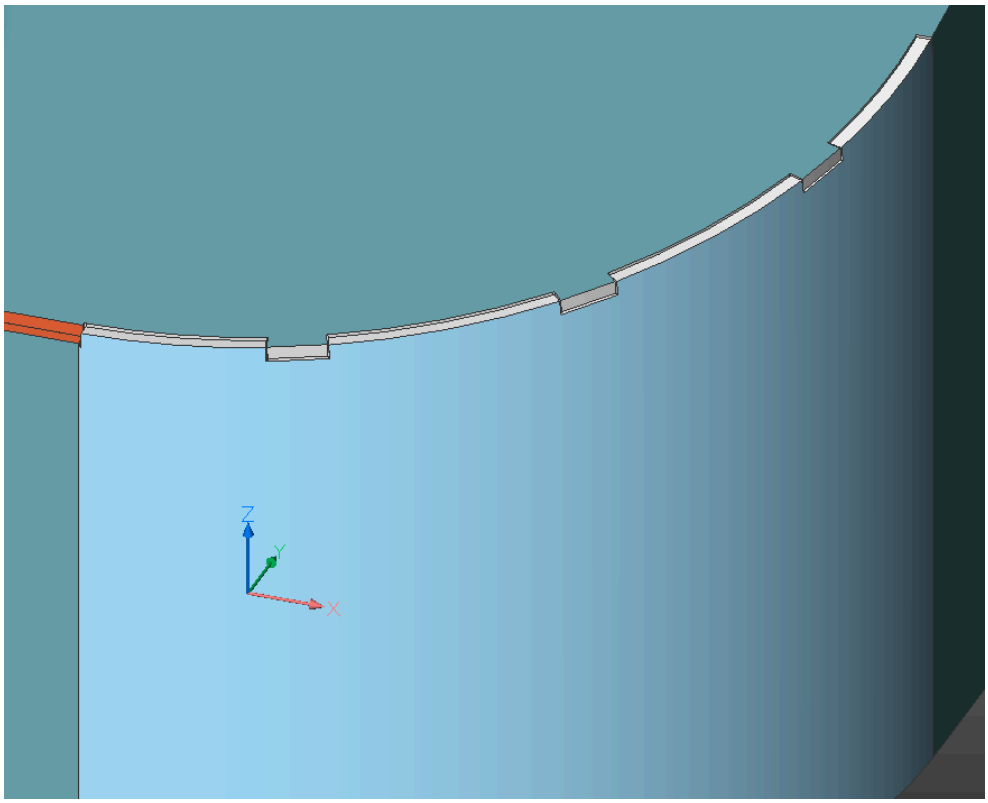
В отличие от прямолинейного массива шип-пазов, в этом случае шип-пазы размещаются вдоль заданной кривой.

1 Чтобы создать криволинейный массив шип-пазов:

- Подведите курсор к криволинейному острому ребру детали из листового металла и нажмите кнопку **Создать массив шип-пазов** ( ) на закладке квадро-меню **Листовой металл**.
- Нажмите кнопку **Создать массив шип-пазов** ( ) на закладке ленты **Листовой металл**.
- Выберите инструмент **Создать массив шип-пазов** в меню **Листовой металл**.
- Введите в командной строке команду *SMTABCREATE*.

  Запрос программы: Выберите стык или криволинейное острое ребро для создания массива шип-пазов или [Инвертированный криволинейный шип-паз/Одиночный шип/Параметры выбора]:

2 Выберите криволинейное острое ребро.

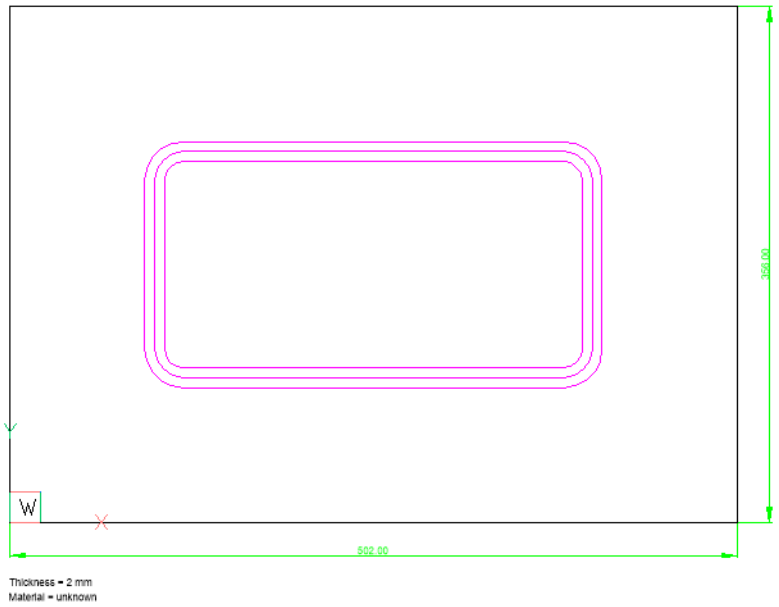



### 21.5.14.4 Редактирование шип-пазов

Одиночные шип-пазы и массивы шип-пазов имеют набор редактируемых свойств, которые отображаются в **Обозревателе конструкции**.

Для выбора элемента Шип-паз щелкните на его имени.

Одиночные шип-пазы имеют следующие свойства: **Длина**, **Зазор**, **Радиус скругления** и **Расстояние фаски**. Свойство **Зазор** определяет величину зазора между шипом и гранями паза.




Массивы шип-пазов имеют следующие свойства: **Число пазов**, **Расстояние**, **Длина**, **Зазор**, **Радиус скругления** и **Расстояние фаски**.

***Прим.:*** Редактирование криволинейных массивов шип-пазов недоступно.

### 21.5.14.5 Удаление шип-пазов

Для удаления шип-пазов (кроме криволинейных) используется команда *SMDELETE*. После удаления шип-паза на его месте создается стык.

### 21.5.14.6 Расчленение шип-паза

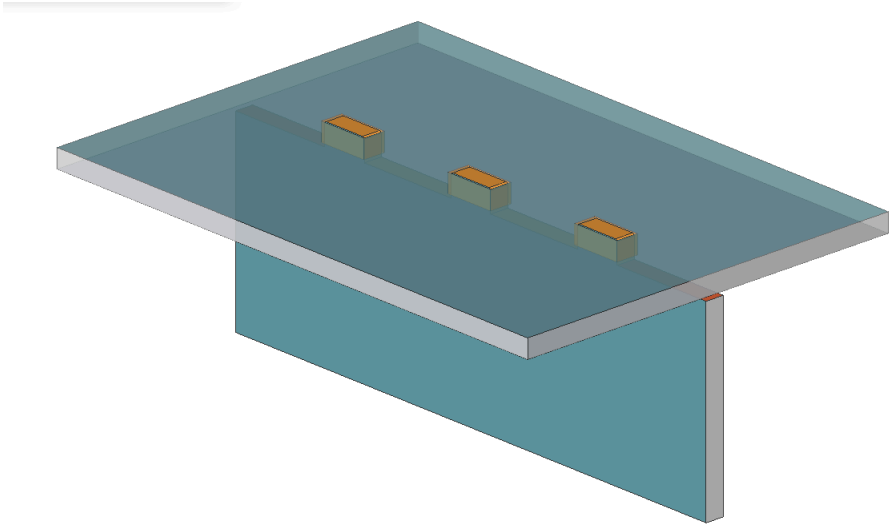
Для расчленения соединения с шип-пазами используется команда *SMEXPLODE*. При расчленении шип-паза его геометрия сохраняется, а грани распределяются между элементами стыка.

| Исходный элемент Шип-паз | Результат удаления | Результат расчленения |
|---|---|---|
| | | |

## 21.5.15 Working with T-tabs and T-junctions

### 21.5.15.1 Overview

You can use T-tab features to create a connection with robust positioning between 2 sheet metal flanges. Unlike normal tabs, T-tabs form a T-like configuration, when one flange is attached to the other.



T-tabs can be created from T-junctions by the *SmTabCreate* command. T-junction features are often used as an intermediate stage for making T-tabs.

### 21.5.15.2 Creating a T-junction

A T-junction can be created on existing separate flanges in the following way:

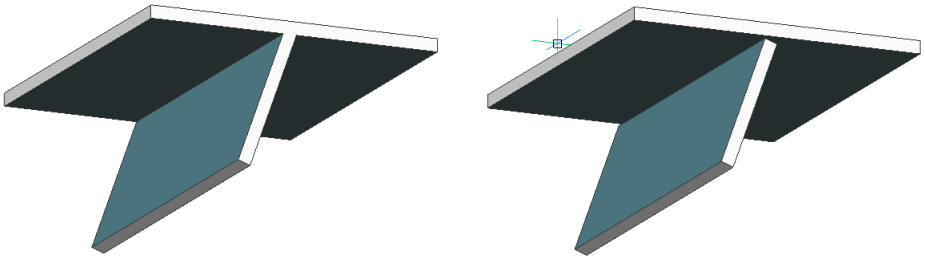
1 Select one thickness face on the flange.

2 Do one of the following:

- Click the **Connect Flanges** ( ) tool button in the **Sheet Metal** | **Junction** ribbon tab.
- Choose **Connect Flanges** in the **Sheet Metal** | **Junction** menu.
- Type *SMFLANGECONNECT* in the Command line.

3 Press **Enter**.

Alternatively, you can create a T-junction feature in place of a T-like hard edge between 2 flanges (image at the left):



1 Hover over a hard edge between the flanges and click the **Create Junction** ( ) tool in the **Quad** menu, or do one of the following:

- Click the **Create Junction** ( ) tool button in the **Sheet Metal** ribbon tab.
- Choose **Create Junction** from the **Sheet Metal** | **Junction** menu.

- Type *SMJUNCTIONCREATE* in the Command line.

2 Select a hard edge.

### 21.5.15.3 Creating a Single T-tab

The *SMTABCREATE* command creates a single T-tab by default. This feature has only a single slot.

To create a single T-tab:

1 Hover over a T-junction face and click the **Create single Tab** ( ) tool in the **Quad** menu, or do one of the following:

- Click the **Create single Tab** ( ) tool button in the **Sheet Metal** ribbon tab.
- Choose **Create single Tab** from the **Sheet Metal** | **Tab** menu
- Type *SMTABCREATE* in the Command line.

2 Select a T-junction face.

### 21.5.15.4 Creating an Array T-tab

Different from Single T-tabs, array T-tab features have multiple slots.

To create an array T-tab:

1 Hover over a T-junction face and click the **Create array Tab** ( ) tool in the **Quad** menu, or do one of the following:

- Click the **Create array Tab** ( ) tool button in the **Sheet Metal** ribbon tab.
- Choose **Create array Tab** from the **Sheet Metal** | **Tab** menu.
- Type *SMTABCREATE* in the Command line.

  You are prompted: Select junction to create single tab or [Array]: select **Array** option.

2 Select a T-junction face.

### 21.5.15.5 Modifying T-junction and T-tab Parameters

A T-junction has only the **Junction gap** parameter.

Array T-tabs and Single T-tabs have the following properties: **Distance**, **Length**, **Clearance**, **Fillet radius**, and **Chamfer distance**. Array T-Tabs also have the **Slot number** parameter.

You can set the value of all parameters in the **Mechanical Browser** panel.

### 21.5.15.6 Deleting a T-junction or T-tab

When a T-tab feature is deleted, BricsCAD restores a T-junction feature with the corresponding geometry. Erasing a T-Junction joins the flanges and creates a T-like hard edge between them.

### 21.5.15.7 Related commands

The *MECHANICALBROWSEROPEN* command displays the Mechanical Browser.

The *SMDELETE* command erases a feature.

## 21.5.16 Working with Bevels

### 21.5.16.1 Overview

Bevel features represent a chamfer operation made on Sheet Metal parts. On the level of BricsCAD V21 each bevel should be exactly on a single flange. There are no specific operations to create a bevel, so the *DMCHAMFER* command must be used on an edge of a flange, and then use *DMROTATE* to adjust the bevel angle. The *SMCONVERT* command recognizes the bevels and is processed by *SMUNFOLD*. Bevel features are also supported by the *SMDELETE* and *SMREPAIR* commands.

This procedure applies to 2 scenarios:

- Working with importing geometry, including batch export by *SMASSEMBLYEXPORT*.
- Modifying parts.

We distinguish between the following bevel types:

**Таблица 44: Bevel types**

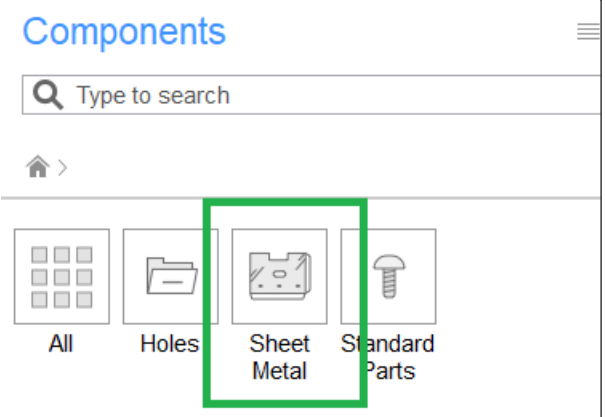
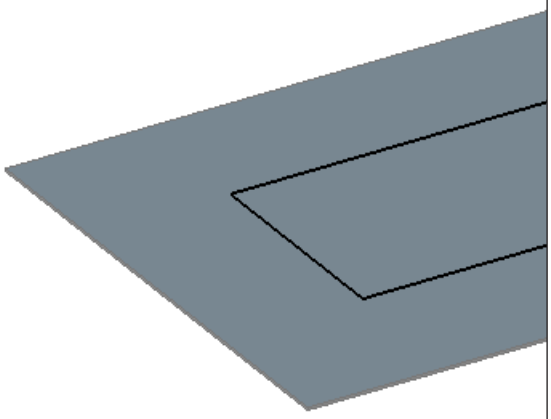
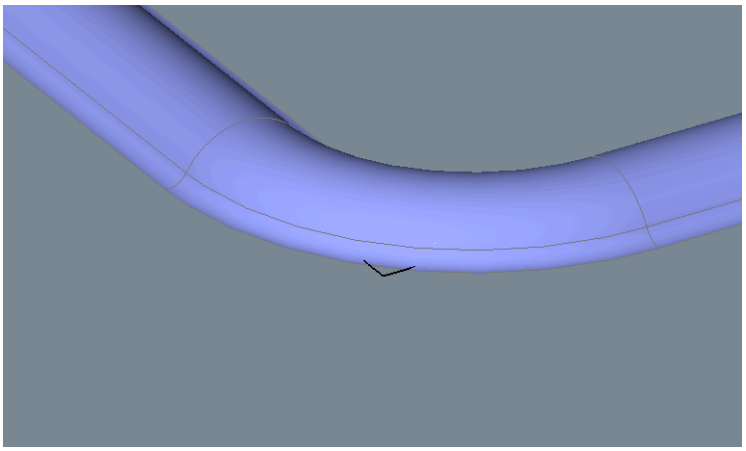
| | |
|---|---|
|  | One-side bevel |
|  | Land bevel |

**Таблица 44: Bevel types**

| | |
|---|---|
|  | K-bevel |
|  | X-bevel |

In the **Mechanical Browser**, the properties of each chamfer of a bevel feature are available separately. These properties are:



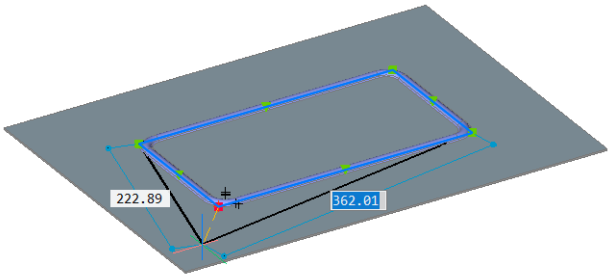
**Height (H)**: the depth of the chamfer operation, much like the *DMCHAMFER* command does.

**Angle (A)**: the slope of the chamfered face.

***Прим.:*** In the Mechanical Browser properties, Top or Bottom are not used to describe bevel features.

### 21.5.16.2 Recognizing bevels

Let us start with a sheet metal part with bevel geometry (chamfered flange edges).

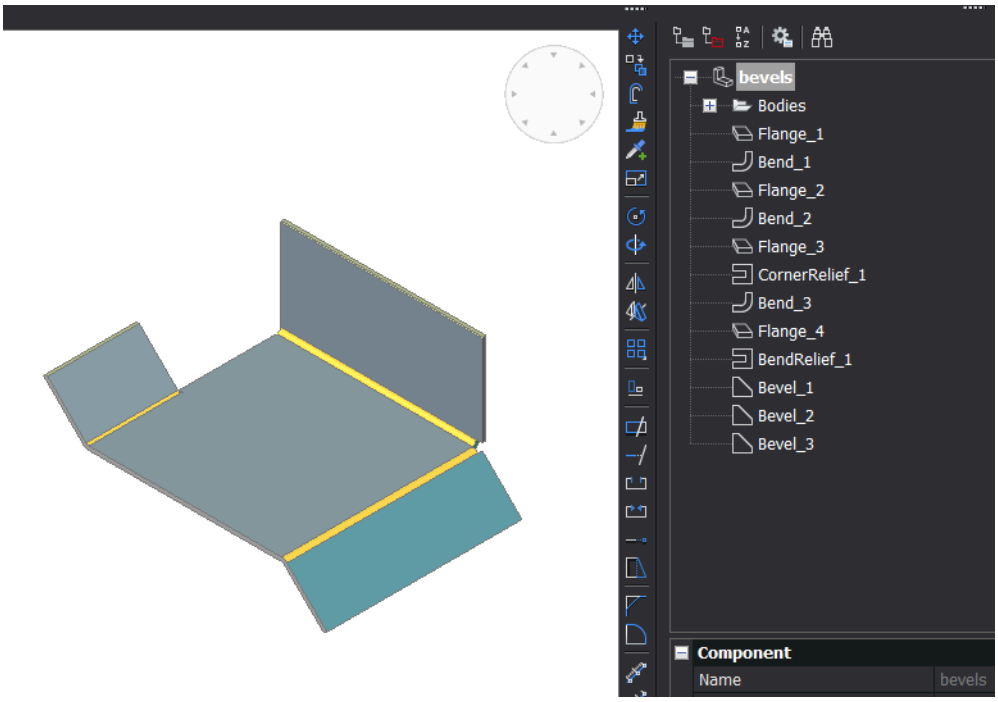



1 Click the **Convert to Sheet Metal** ( ) tool button on the **Sheet Metal / Create** Ribbon panel.

You are prompted: Select 3D solids or seed faces or [Entire model/selection options (?)] <Entire model>:

2 Press **Enter** to convert the 3D solid to a sheet metal part.

The bevel features are recognized and appear in **Mechanical Browser** and show on the part:




### 21.5.16.3 Delete and Repair

The following round trip does not change the overall dimensions of the part: Create the bevel geometry and feature using *DMCHAMFER*/*SMCONVERT*, then delete the bevel using *SMDELETE* and restart the procedure.

When using *SMDELETE* to delete a bevel the volume is **always added** to the part because the dimensions of the part can not be reduced.

Support in the *SMREPAIR* command is also important, since this command is a part of the Import and Rework workflow.

***Прим.:*** If you apply the *SMREPAIR* command to a part with chamfers but without bevel features, chamfers will be lost and dimensions will change, depending on the selected side of the part.

### 21.5.16.4 Unfold

Bevel features are unfolded according to the **Unfolding Mode** setting in the **Mechanical Browser**.

The options are:

**Keep**: Does nothing with bevel geometry, resulting in a 3D solid with chamfers. These chamfered faces will likely produce artifacts in the 2D .DXF.

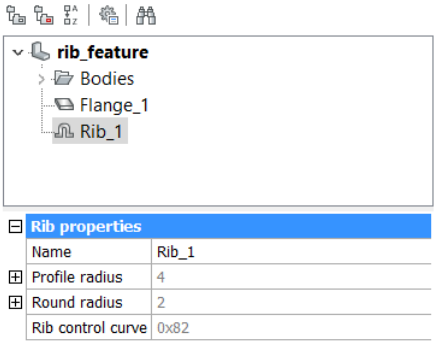
**Remove**: Same result as applying *SMDELETE* on all the bevel features: all the chops are filled.

**Annotate**: Each bevel feature is removed and the edge is annotated. This mode is recommended.

This is the result for the example part:

- Each bevel edge has been annotated with a pink text on the **Bevel Annotations** layer.

- UP and DOWN are with respect to the selected side for unfolding. When choosing the opposite side, all the UP annotations become DOWN, and vice versa.
- For **One-side** and **Land bevels** a single chamfer operation is needed only: a single UP or Down annotation is created.
- For **K-** and **X- bevels** 2 chamfer operations are needed: UP and DOWN annotations are created.




### 21.5.17 Creating and Splitting Lofted Bends

BricsCAD allows you to create or recognize lofted sheet metal parts with curvilinear geometry, which can be manufactured using multiple bending.

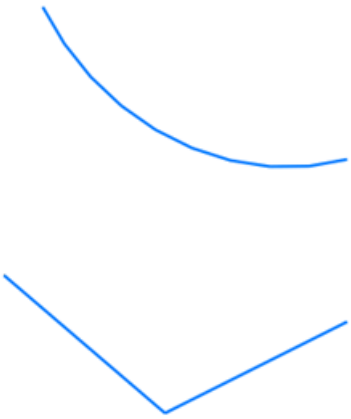
#### 21.5.17.1 General Procedure to Create a Sheet Metal Part with a Lofted Bend

1 Launch the *SMLOFT* command.

2 Do one of the following:

- Click the **Create Lofted Bend** tool button ( ) on the **Sheet Metal** toolbar.
- Click the **Create Lofted Bend** tool button ( ) on the **Sheet Metal** ribbon bar.
- Choose **Create Sheet Metal Loft** in the **Sheet Metal** menu.

You are prompted: Select two cross sections:



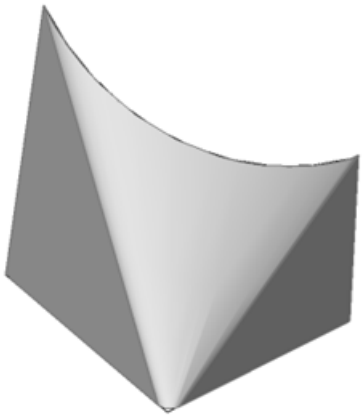
3 Select 2 profile entities.

You are prompted: Set fillets Radius/Thickness/Single bend/<Flanges with bends>:

4 Press **Enter** to accept the default settings.

A preview of the sheet metal part is created.

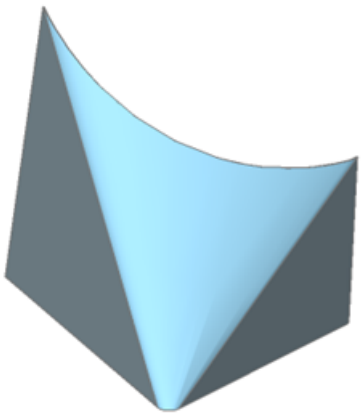
You are prompted: Set fillets Radius/Thickness/Single bend/Flanges with bends or make thicken Both sides/Inside/<Outside>:






5 Press the **CTRL** key to choose the position of the sheet metal part thickness with respect to the source profiles: **Left**, **Center** or **Right**.

If the **HKA** field in the Status Bar is ON, the Hotkey Assistant widget displays.



6 Press **Enter** to create the sheet metal part.

If **FEATURECOLORS** is ON, you can distinguish between the 2 flanges ( ) and the lofted bend ( ).

The properties of the lofted bend display in the **Mechanical Browser**:

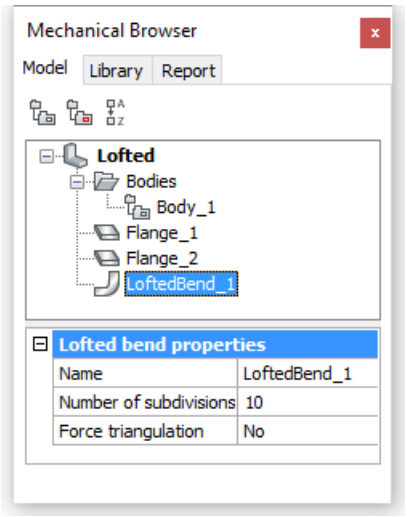



**Рисунок 132:** Lofted bend properties

Optionally select the lofted bend feature to edit its properties:

- **Number of subdivisions**: the number of flat regions in the unfolded representation of a lofted bend feature (the greater the number, the more bend lines will be added to the unfolded view for the lofted bend).
- **Force triangulation**: most of lofted bends can be unfolded using an intelligent algorithm of quad tessellation. However, sometimes it might produce non-optimal results. In such cases set this property to *Yes* to switch to a more straight-forward algorithm, which creates triangle-like quads.

### 21.5.17.2 To Split a Lofted Bend

Lofted bends can split along a line on the lofted surface at an arbitrary point:

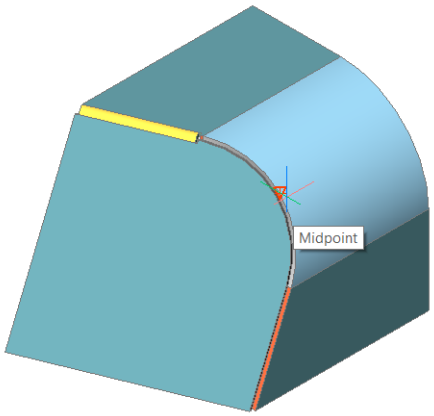
1 Do one of the following:
   - Click the **Split** tool () on the **Sheet Metal** ribbon tab.
   - Choose **Split** the **Sheet Metal** menu.
   - Type **smsplit** in the Command line.

   You are prompted: Select flange or lofted bend face:

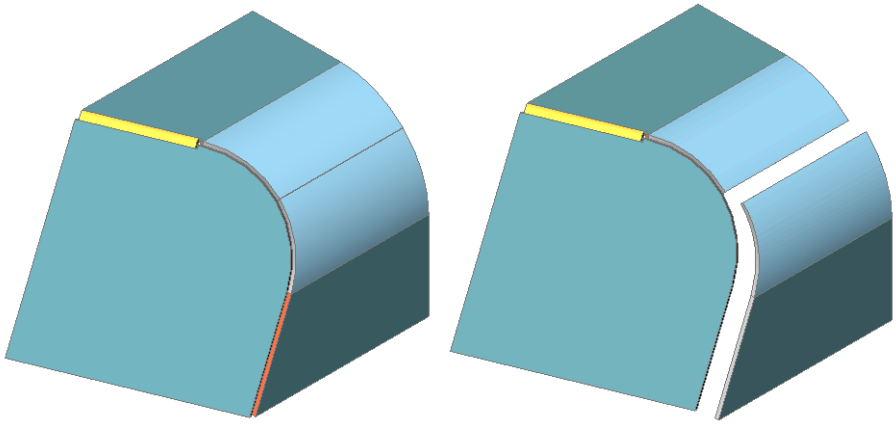
2 Select a lofted bend face.

   You are prompted: Select point on lofted bend face:

3 Select a point on the lofted bend surface.




The lofted bend is split. If the connecting junction is dissolved, you can freely move the separated part.



### 21.5.18 Bend Relief and Corner Relief Feature Operations

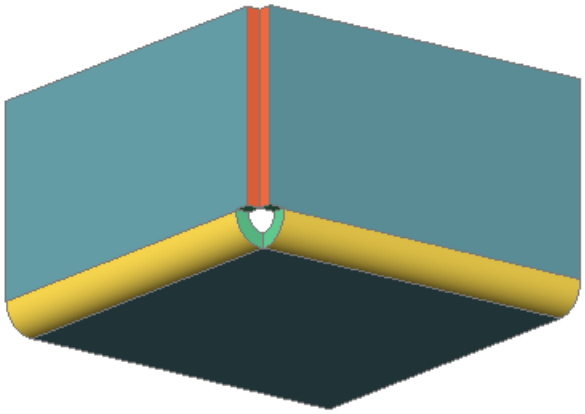
#### 21.5.18.1 Overview

Use the *SMFLANGEEDGE*, *SMRELIEFCREATE*, *SMJUNCTIONSWITCH*, and *SMRELIEFSWITCH* commands.

Reliefs are needed in sheet metal parts to avoid material tear when bent on press breaks. Basically you must not care about reliefs, because BricsCAD automatically creates all needed reliefs when you create a bend (implicitly or explicitly). However, if you rework poor sheet metal parts, you may find it useful to create corner and bend reliefs with the dedicated tools.

#### 21.5.18.2 Corner Reliefs and Junctions

When you create a flange using the *SMFLANGEEDGE* command, adjacent to a bend edge, a corner relief is created automatically in the corner where the 3 flanges meet. In addition, a junction is created between the 2 flanges, which are not connected with a bend.



### 21.5.18.3 Creating Missing Reliefs

1 Do one of the following:

- Click the **Create Relief** tool button ( ) on the **Sheet Metal** ribbon tab.
- Choose **Create Relief** in the **Sheet Metal** menu.
- Type *SMRELIEFCREATE* in the Command line.

  You are prompted: Select a hard edge, bend face, flange face or 3D solid [Entire model]:

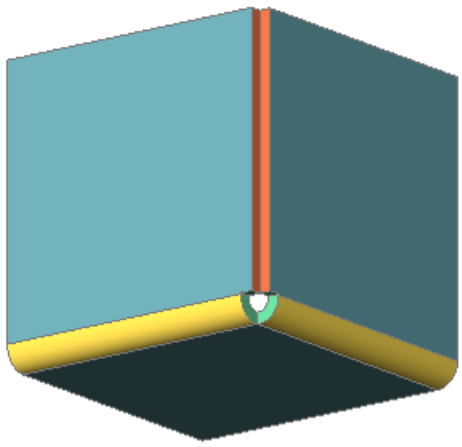
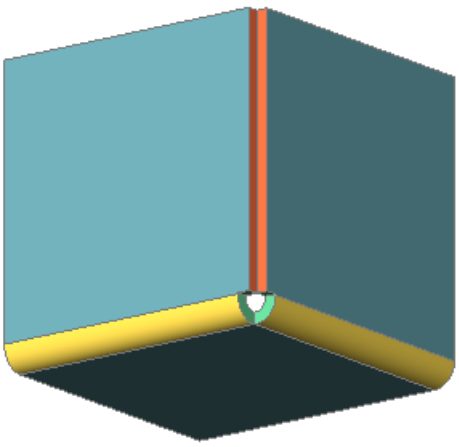
2 Select an item.

  Enter relief size by bend radius ratio or [force Bend reliefs/Auto] <Auto>:

3 Do one of the following:

- Press **Enter** or right-click to accept the default bend radius ratio.
- Type a value at the command prompt, then press **Enter**.

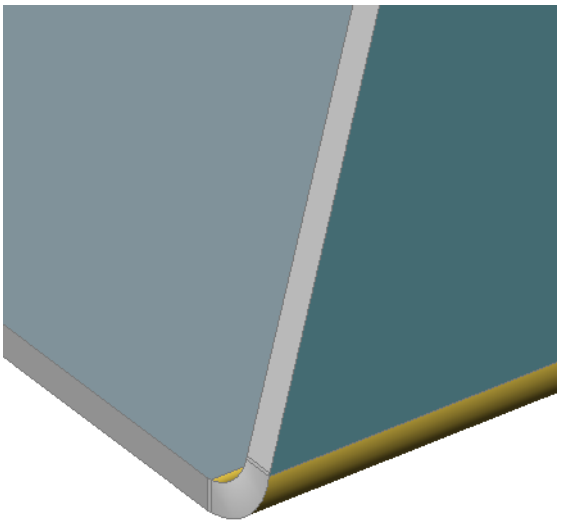
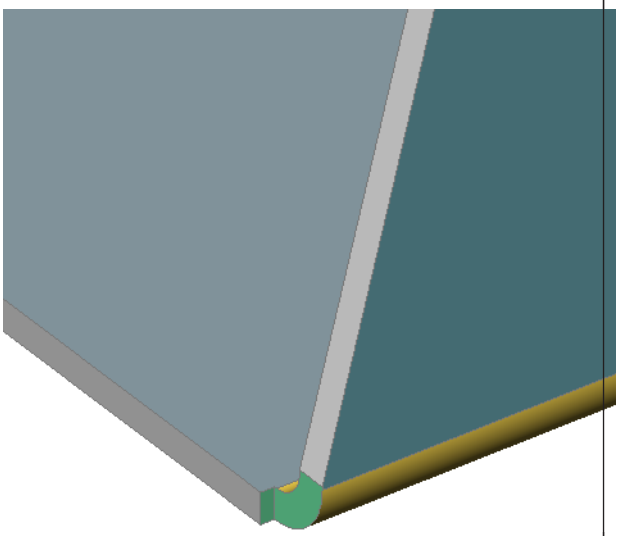
  The relief is created.

| | |
|---|---|
|  |  |
| Selected bend faces | Created reliefs |

***Прим.:***

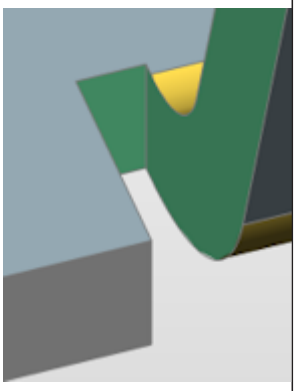
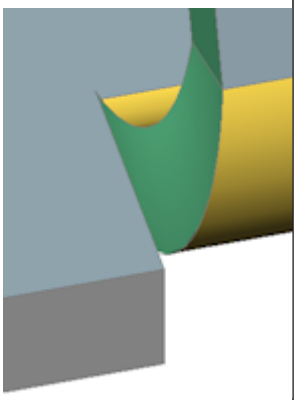
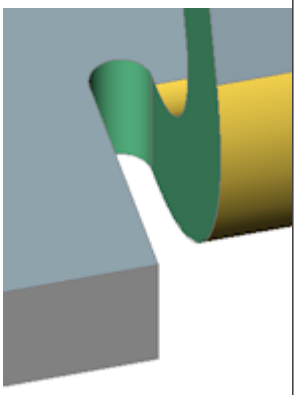
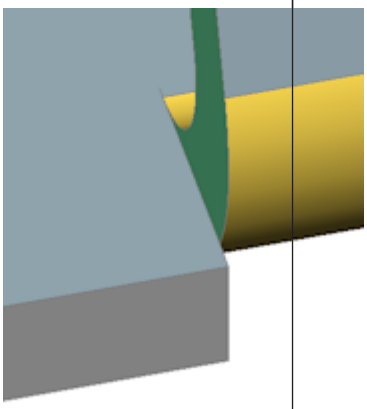
- The selected items can be a 3D solid with sheet metal features, a face or a hard edge of the solid.
- To create a relief locally, you can select 1 or 2 hard edges, 1 or 2 bends or a bend and a hard edge.
- If a 3D solid is selected, the command recognizes the places where corner or bend reliefs are needed and creates them automatically.
- If 2 hard edges or 2 bends or a bend and a hard edge at a corner are selected, the command tries to create a corner relief.

Option **force Bend reliefs** forces creation of the bend relief even if there is no need to create a bend relief. As shown in the image below, there is no need to create a bend relief, but the geometry is different from rectangular cut.

| | |
|---|---|
|  |  |
| A bend relief is not needed because of a smooth transition of the geometry. | A rectangular bend relief is created only if the option **force Bend reliefs** is enabled. |

### 21.5.18.4 Switching the Bend Relief Type

The bend relief type can be switched between 4 types:

| | | | |
|---|---|---|---|
|  |  |  |  |
| Rectangular | Smooth | Round | Rip |

***Прим.:***

- A **Rip** relief has a very small width of 0.001 * thickness. If you need to remove the relief completely, change its width to 0. This operation removes both the bend relief geometry and the bend relief feature.
- **Smooth**, **Round** and **Rip** bend relief types can be applied only if the current bend relief type is **Rectangular**.

**Using the ribbon, menu or toolbar**

1 To choose the desired relief type, do one of the following:
   - Click the **Relief** drop-down button in **Sheet Metal / Modify** panel on the ribbon and select **Switch to Rectangular**, **Switch to Smooth**, **Switch to Round** or **Switch to Rip**.
   - Click a tool button on the **SM Relief** toolbar: **Switch to Rectangular Relief** (), **Switch to Smooth Relief** (), **Switch to Round Relief** () or **Switch to Rip Relief** ().
   - Choose the relief type in the **Sheet Metal / Relief** menu.

     You are prompted: Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/relief EXtension/Entire model] <Entire model>:

2 Do one of the following:
   - Press **Enter** to apply the selected relief type to the entire model.
   - Click a neighboring flange or bend or a face of the bend relief.

     The bend relief is converted.

3 If the command has been called from the Ribbon, the command repeats the main prompt until you press the **ESC** key to stop.

**Using the Quad**

1 Hover over a neighboring flange or bend or a face of the bend relief.

  If the **Select Faces** option of the **SELECTIONMODES** system variable is selected, the flange or face highlights; otherwise hold the **CTRL** key down when hovering until the face highlights.

2 Select the desired relief type in the **Sheet Metal** tab in the **Quad** menu.

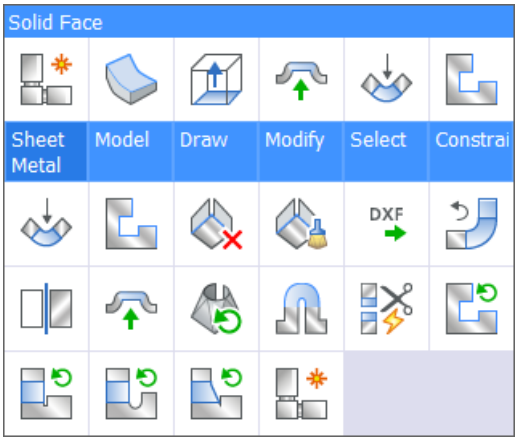
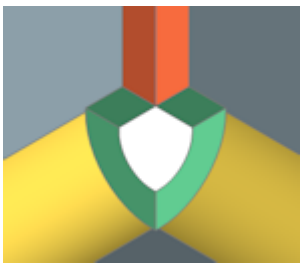
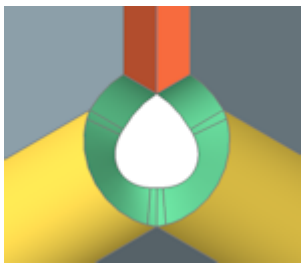
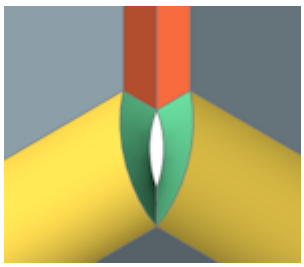



**Рисунок 133:** Sheet Metal

### 21.5.18.5 Switching the Corner Relief Type

The corner relief type can be switched between 3 types:

| | | |
|---|---|---|
|  |  |  |
| Rectangular | Circular | V-type |

**Using the ribbon, menu or toolbar:**

1 To choose the desired relief type, do one of the following:
- Click the **Relief** drop-down button in **Sheet Metal / Modify** panel on the ribbon and select a relief type.
- Click a tool button on the **SM Relief** toolbar.
- Choose the relief type in the **Sheet Metal / Relief** menu.

  You are prompted: Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/relief EXtension/Entire model] <Entire model>:

2 Do one of the following:
- Press **Enter** to apply the selected relief type to the entire model.
- Click a neighboring flange or bend or a face of the corner relief.

  The corner relief is converted.

3 If the command has been called from the Ribbon, the command repeats the main prompt until you

press the **ESC** key to stop.

**Using the Quad:**

1 Make sure the **Select Faces** option of the **SELECTIONMODES** system variable is selected.

2 Hover over a neighboring flange or bend or a face of the corner relief.

3 Select the desired relief type in the **Sheet Metal** command group in the **Quad** menu.

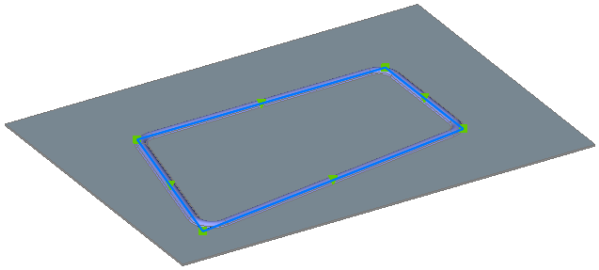


**Рисунок 134:** Sheet Metal

### 21.5.19 Работа с элементами формы и ребрами жесткости

#### 21.5.19.1 Обзор

Элементы формы и ребра жесткости создаются с помощью формообразующего инструмента. Ребро жесткости представляет собой рельефный оттиск на листе, выполненный роликом, перемещающимся по заданной траектории. Мы будем использовать понятие 'форма' для обозначения как элементов формы, так и ребер жесткости. Элементы формы могут быть распознаны в импортированной геометрии или вставлены из встроенных или пользовательских библиотек. Элементы формы могут быть отредактированы напрямую или параметрически. Элементы формы и их параметры отображаются в **Обозревателе конструкции**. Для отображения и редактирования параметров элементов форм также используется **Панель свойств**.

При работе с элементами форм и ребрами жесткости применяются следующие команды:

| Значок | Команда | Description |
|---|---|---|
|  | *SMFORM* | Преобразование выбранного набора граней в элемент формы. Файл чертежа с пользовательским элементом формы можно сохранить и использовать для последующей вставки этого элемента с помощью команды *BMINSERT*. |
|  | *SMCONVERT* | Распознает элементы формы и жесткости в импортированной геометрии. |
|  | *SMREPLACE* | Заменяет элементы форм в деталях из листового металла (включая распознанные при импорте) на элементы форм из встроенной или пользовательской библиотеки. |

| Значок | Команда | Description |
|---|---|---|
|  | *SMRIBCREATE* | Создает ассоциативные элементы жесткости на деталях из листового металла с использованием 2D профилей. При изменении базового профиля ассоциированный с ним элемент формы обновляется автоматически. |
|  | *BMINSERT* | Выполняет вставку существующих механических компонентов или элементов форм из листового металла в текущий чертеж. |
|  | *SMSELECT* | Выделяет элементы форм, подобные данной. Команда работает в двух режимах:<br>• **Подобные элементы**: выбор элементов с одинаковым набором граней, но, возможно, разных размеров.<br>• **Одинаковые элементы**: выбор идентичных элементов. |

Библиотека параметрических элементов форм доступна на закладке Элементы форм панели **Инструментальные палитры**. Библиотека включает наиболее часто используемые типы элементов форм. Для вставки элемента формы на деталь из листового металла перетяните выбранный элемент из панелей **Инструментальные палитры** или **Компоненты** на стенку детали.

Системная переменная *SMFORMFEATUREUNFOLDMODE* управляет отображением элементов форм в 2D и 3D представлениях развертки модели.

### 21.5.19.2 Вставка элемента формы

Элементы формы добавляются в детали из листового металла путем их вставки на существующие стенки. Элементы форм могут быть вставлены с помощью команды *BMINSERT* или путем перетягивания необходимого элемента с закладки **Элементы форм** панели **Инструментальные палитры**.

1 Убедитесь, что включены режимы **Динамические размеры** (ДИН.ВВОД) и **Динамическая ПСК** (ДИН.ПСК).

2 Выполните одно из следующих действий:

- Нажмите кнопку **Элемент формы** () на панели инструментов **Листовой металл**.
- Нажмите кнопку **Элемент формы** () на закладке ленты **Листовой металл**.
- Выберите инструмент **Вставить элемент формы** в меню **Листовой металл**.
- Введите в командной строке команду *BMINSERT* и нажмите клавишу **Enter**.

  Откроется диалоговое окно *Укажите файл для вставки*, в котором отображается содержимое папки **FormFeatures**, расположенной в каталоге [Папка установки BricsCAD]\Support\DesignLibrary\SheetMetal\FormFeatures:

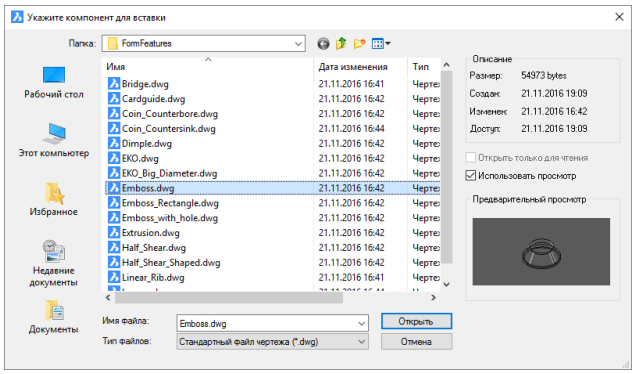



**Рисунок 135:** Укажите файл для вставки

Выберите элемент формы и дважды щелкните на нем или нажмите кнопку **Открыть**.

- Выберите элемент формы в категории Листовой металл > Элементы формы на панели **Библиотека**:

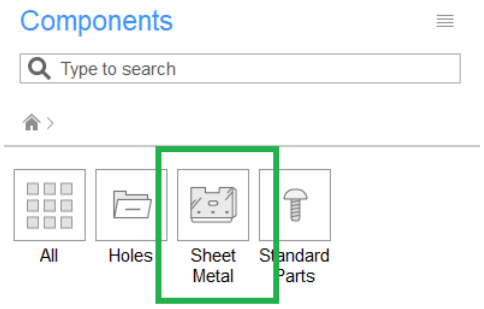



**Рисунок 136:** Компоненты

- Выберите элемент формы на закладке **Элементы форм** панели **Инструментальные палитры**:

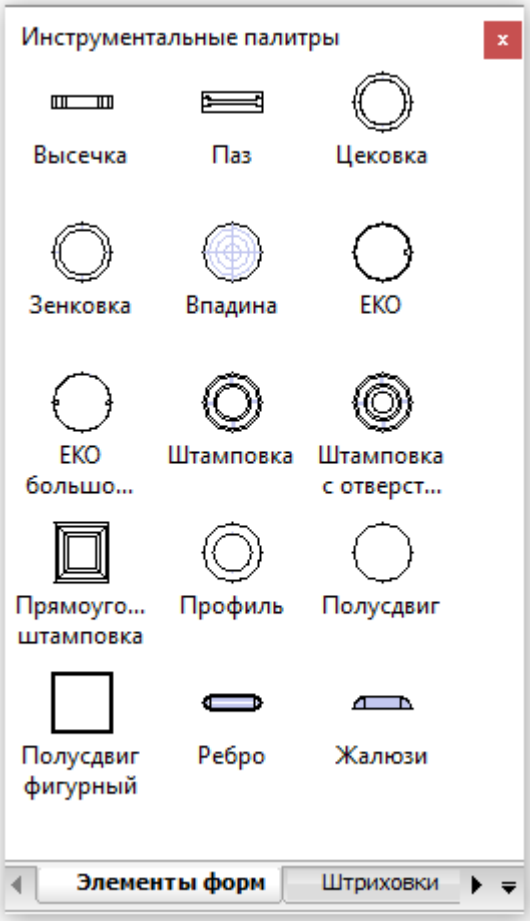



**Рисунок 137:** Инструментальные палитры

3 Подведите курсор к грани листа, на который вы хотите вставить элемент.

Динамическая ПСК будет установлена по выбранной грани.

Ориентация ПСК зависит от ориентации ребра, через которое курсор входит в грань.

Динамические размеры указывают текущие значения расстояний от начала координат ПСК до точки вставки элемента формы.

Запрос программы:

Укажите точку вставки или [Изменить объект/Повернуть компонент/Установить базовую точку/Имя/Тип вставки/Инверсия/Массив/Изменить целевые 3D объекты]<0, 0, 0>:

4 При необходимости, выберите одну из опций (см. описание команды *BMINSERT*):

- **Изменить объект**: Позволяет изменить параметры выбранного для вставки объекта.
- **Повернуть компонент**: Позволяет указать угол поворота компонента.
- **Установить базовую точку**: Позволяет переопределить базовую точку вставки компонента.
- **Имя**: Позволяет определить имя вставляемого компонента.

- **Тип вставки**: Позволяет определить тип вставляемого компонента - Локальный или Внешний.
- **Инверсия**: Позволяет выполнить вставку элемента формы на противоположной грани стенки.
- **Массив**: Позволяет выполнить несколько вставок одного компонента или создать массив.

5 Для вставки элемента формы выполните одно из следующих действий:
- Укажите точку.
- Введите значения расстояний в полях динамического ввода. Для переключения между полями динамического ввода используйте клавишу **TAB**.

6 Повторите действия по п.5 для вставки следующей копии или нажмите клавишу **Enter** для завершения.

Для отображения элементов форм на панели **Библиотека** необходимо создать или использовать профиль **Mechanical** в диалоговом окне **Начало работы**.

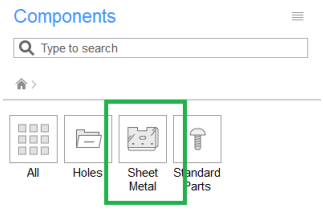



**Рисунок 138:** Компоненты

### 21.5.19.3 Редактирование параметров элемента формы из библиотеки

Элементы форм из библиотеки являются параметрическими компонентами. Их размерами управляют несколько параметров. Параметры для управления длиной, шириной, высотой и радиусом элемента формы обозначаются соответственно L, W, H и R. Толщина элемента формы автоматически устанавливается равной толщине детали из листового металла.

Параметры одиночного элемента формы могут быть изменены с помощью **Панели свойств** или **Обозревателя конструкции**. Одновременное редактирование параметров нескольких элементов форм возможно только при использовании **Панели свойств**.

**Методы выбора элементов форм**

- Выберите опцию **Выделять грани** в настройках системной переменной *SELECTIONMODES* и укажите элементы модели.
- Выберите элементы в **Обозревателе конструкций**, нажмите клавишу **Space** или щелкните правой кнопкой мыши и в открывшемся контекстном меню выберите пункт **Выбрать**. Для выбора нескольких элементов форм нажмите и удерживайте клавишу **CTRL**.

- Деталь из листового металла может иметь множество различных элементов формы, которые могут усложнять такие операции, как удаление или замена, особенно если требуется обработка набора элементов, соответствующих определенным условиям. Команда **SmSelect** позволяет упростить выбор элементов форм путем выбора подобных или идентичных элементов.

**Изменение параметров с помощью Панели свойств**

1 Выберите элементы форм.

2 В разделе **Параметры** на **Панели свойств** выберите необходимый параметр.

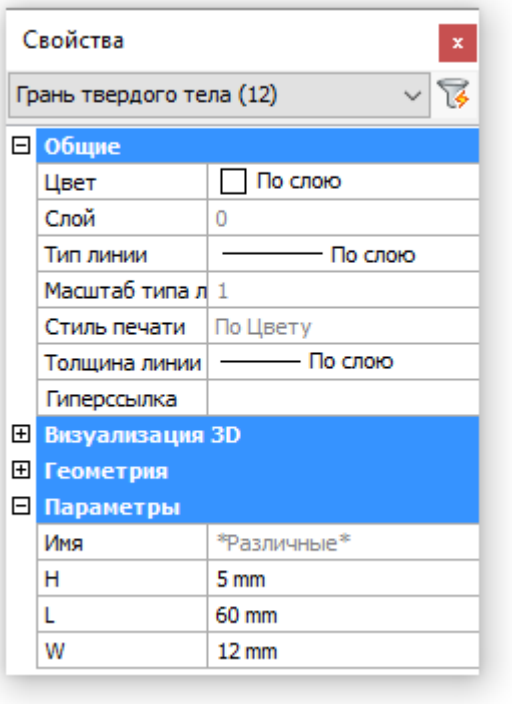



**Рисунок 139:** Свойства

**Редактирование параметров с помощью Обозревателя конструкции**

1 Выберите элемент **формы** в структуре компонента.

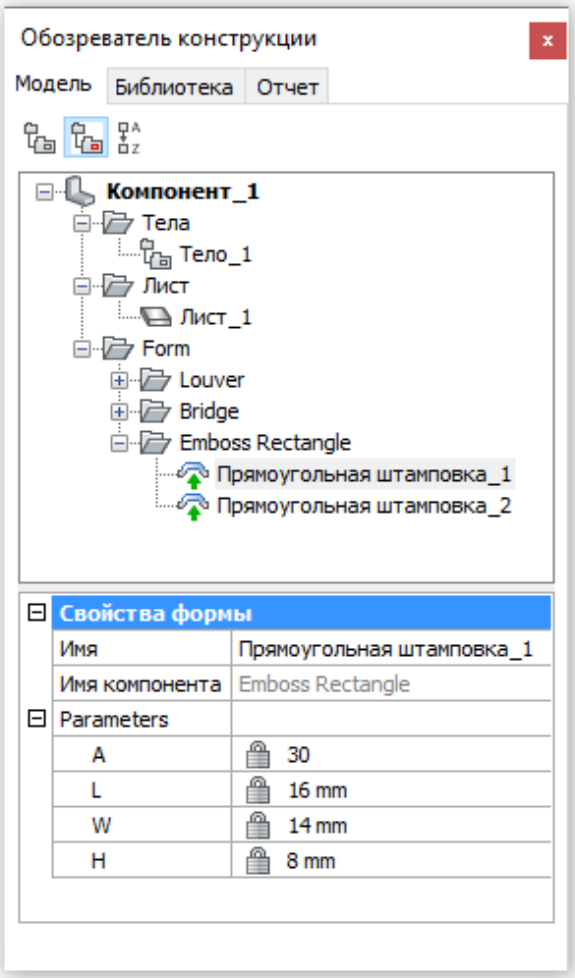


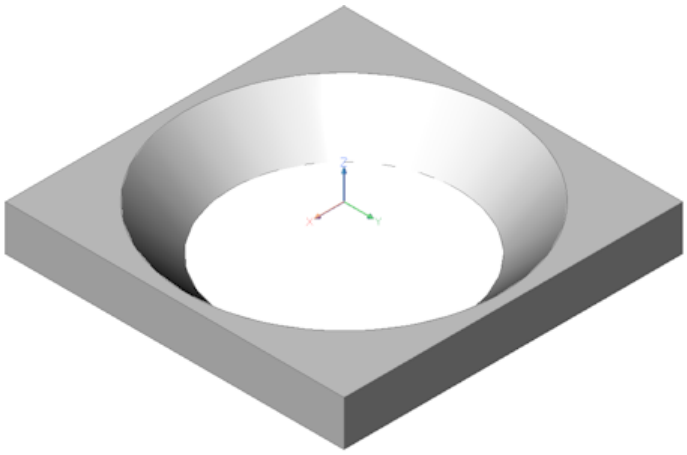

**Рисунок 140:** Обозреватель конструкции

2 Выберите параметр в нижней панели **Обозревателя конструкции** и введите новое значение.

### 21.5.19.4 Создание пользовательского элемента формы

1 Откройте новый чертеж на основе шаблона **Mechanical-metric.dwt**.

2 Создайте 3D геометрию элемента



формы.

3 Запустите команду *SMCONVERT* ( ).

4 Запустите команду *BMMECH* ( ).

Запрос программы: Имя компонента:

5 Введите имя элемента формы.

Запрос программы: Преобразовать все блоки и внешние ссылки в компоненты конструкции? [Да/Нет]<Да>

6 Выберите опцию **Нет**.

7 В поле **Тип элемента** в **Обозревателе конструкции** выберите **Элементы форм листового металла**.

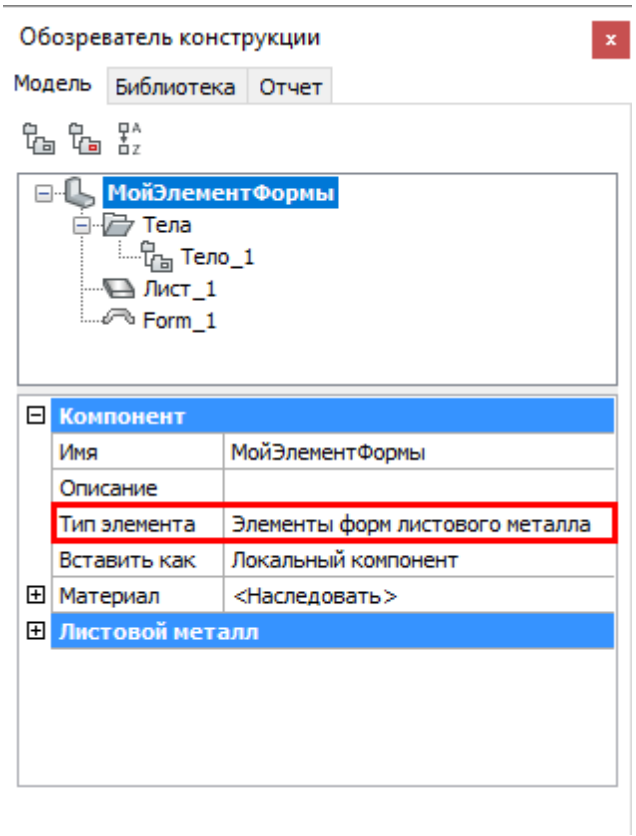



**Рисунок 141:** Обозреватель конструкции

8 В разделе **Тела** выберите элементы Лист_1 и Форма_1, щелкните правой кнопкой мыши и в открывшемся контекстном меню выберите **Отключить**.

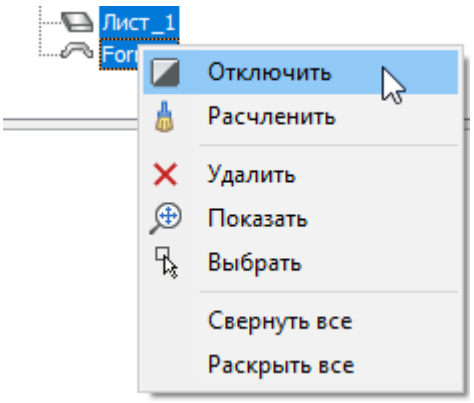
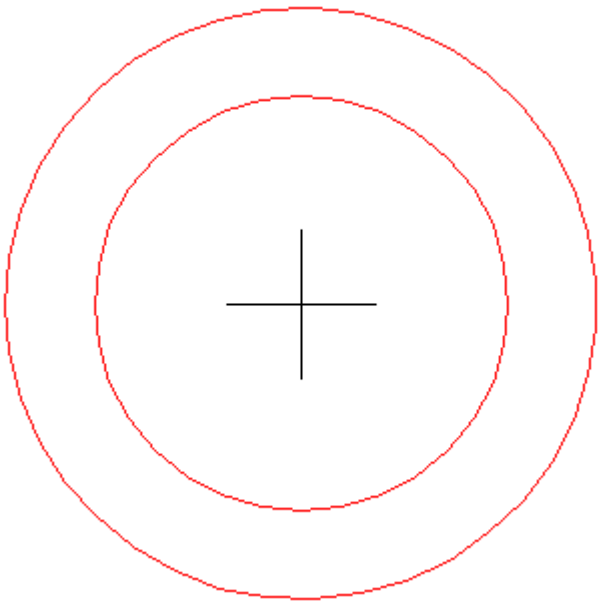



**Рисунок 142:** Отключить

9 Создайте 2 новых слоя: UNFOLD_SYMBOL_UP и UNFOLD_SYMBOL_DOWN.

10 **Выключите** слои 0 и UNFOLD_SYMBOL_DOWN и назначьте слой UNFOLD_SYMBOL_UP **текущим**.

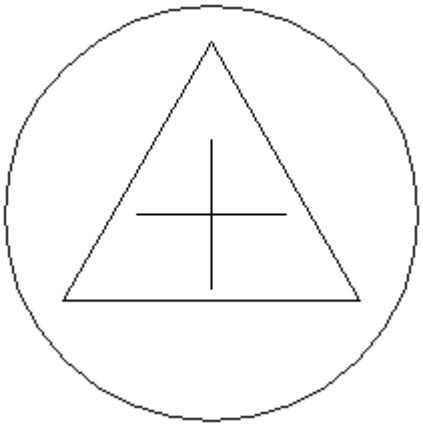
11 Создайте символ для развертки, который будет вставлен на грань листа по месту вставки



элемента формы.

12 **Выключите** слой UNFOLD_SYMBOL_UP и **включите** слой UNFOLD_SYMBOL_DOWN. Назначьте слой UNFOLD_SYMBOL_ DOWN **текущим**.

13 Создайте символ для развертки, который будет вставлен на грань по месту вставки элемента



формы.

14 **Включите** слой 0 и **выключите** слои UNFOLD_SYMBOL_UP и UNFOLD_SYMBOL_DOWN. Назначьте слой 0 **текущим**.

15 Сохраните чертеж.

### 21.5.19.5 Создание пользовательского параметрического элемента формы

Добавим параметры для созданной ранее формы.

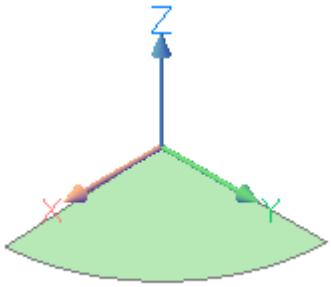
В этом случае необходимо создать два параметра:

- **D** - диаметр большего ребра конического отверстия.
- **A** - угол наклона грани конического отверстия.

В листовом металле используется параметр **_SM_THICKNESS**, который управляет толщиной листа детали.

1 Сначала создайте вспомогательное тело на новом слое CONSTRUCTION.

Сделайте этот слой текущим и отключите все остальные слои.




2 Создайте область в плоскости XY МСК на слое CONSTRUCTION.

3 Примените к этой области 3D зависимость *Фиксация* ( ).

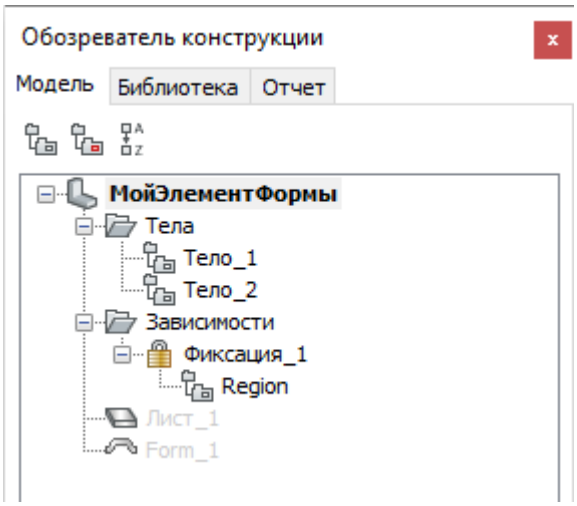
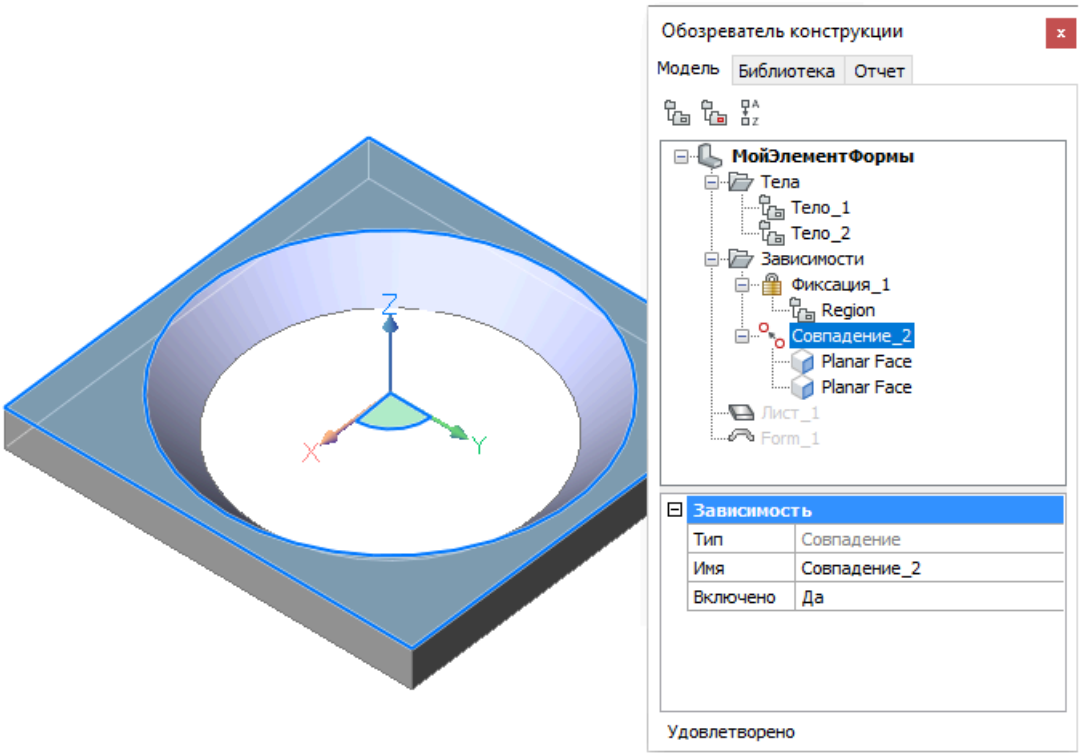



**Рисунок 143:** Обозреватель конструкции

4 **Включите** слой 0.

5 Примените 3D зависимость *Совпадение* ( ) для грани области и верхней грани



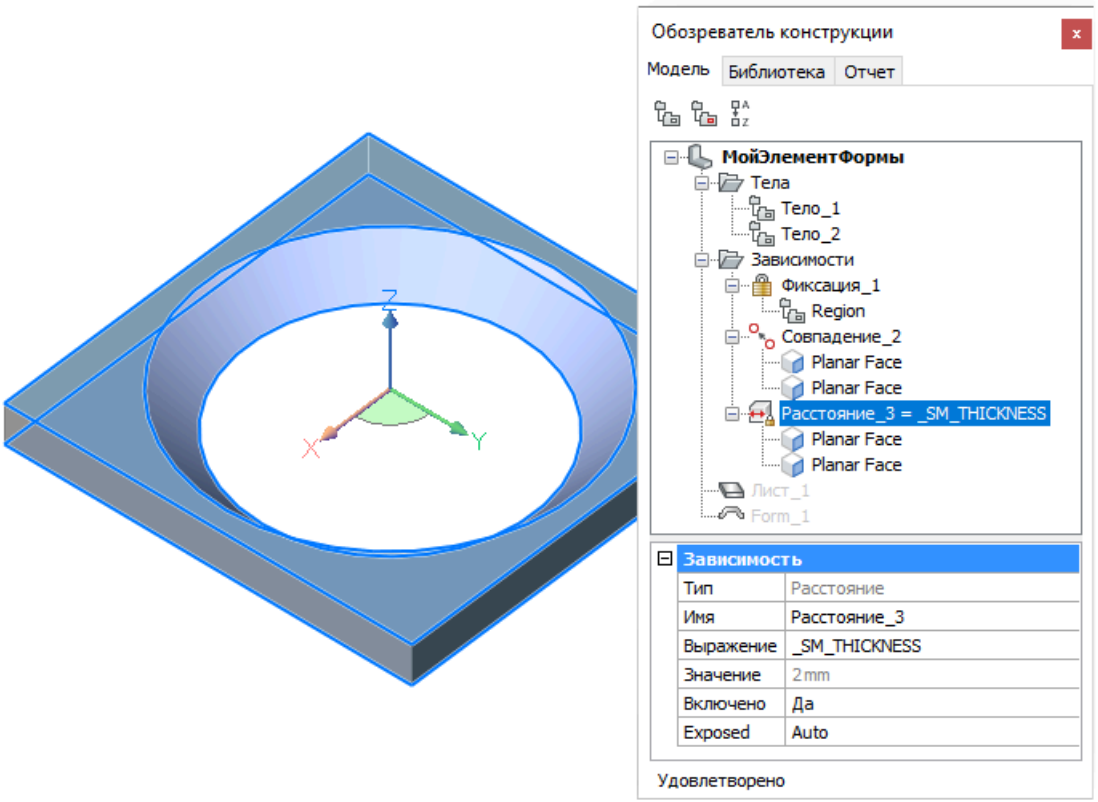

стенки.

6 Примените 3D зависимость *Расстояние* () для верхней и нижней граней стенки.

Сначала выберите верхнюю грань, а затем, нажав клавишу **TAB**, выберите нижнюю грань.

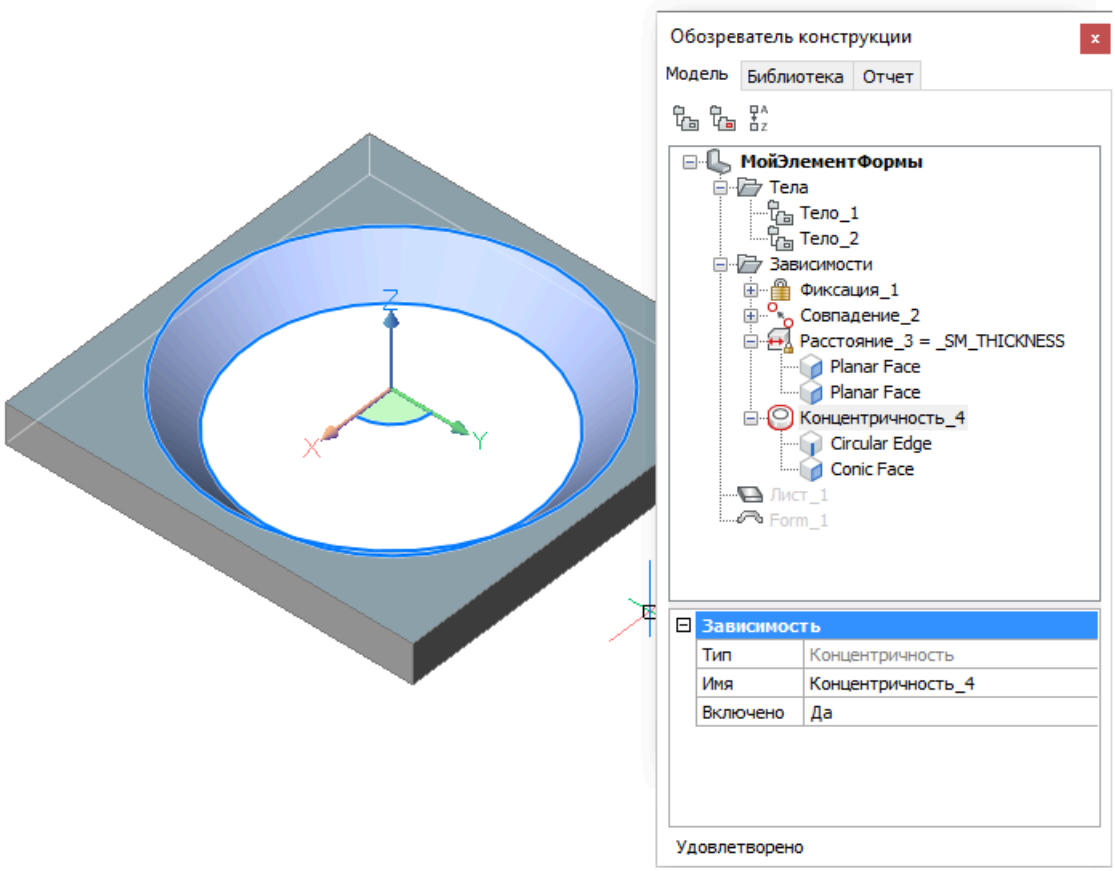
Подтвердите значение динамического размера по умолчанию.

7 Для зависимости Расстояние_3 в поле **Выражение** введите




_SM_THICKNESS.

8 Так как наш элемент формы конический, мы можем изменить форму листа с параллелепипеда на

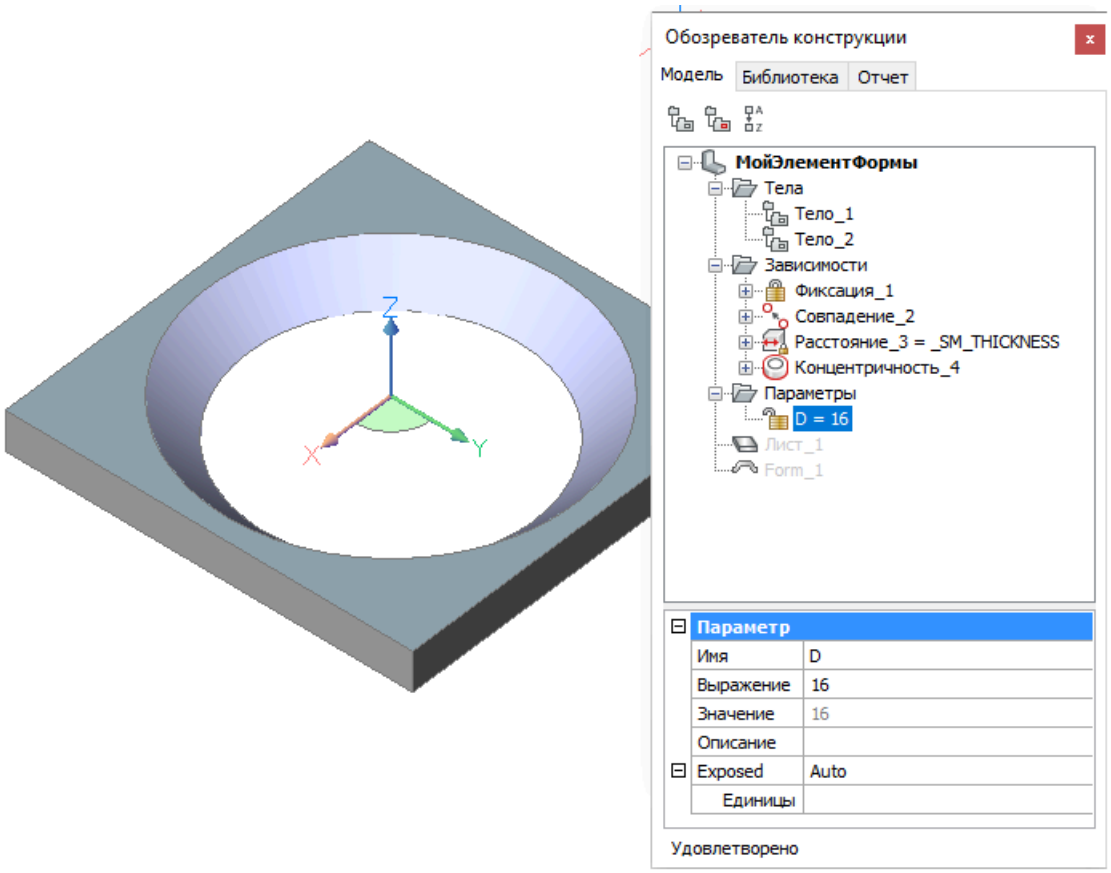



цилиндрическую.

9 В **Обозревателе конструкции** щелкните правой кнопкой на имени элемента и в открывшемся контекстном меню выберите пункт **Добавить новый параметр**.

Введите для нового параметра имя **D**.
В поле **Единицы** выберите мм.

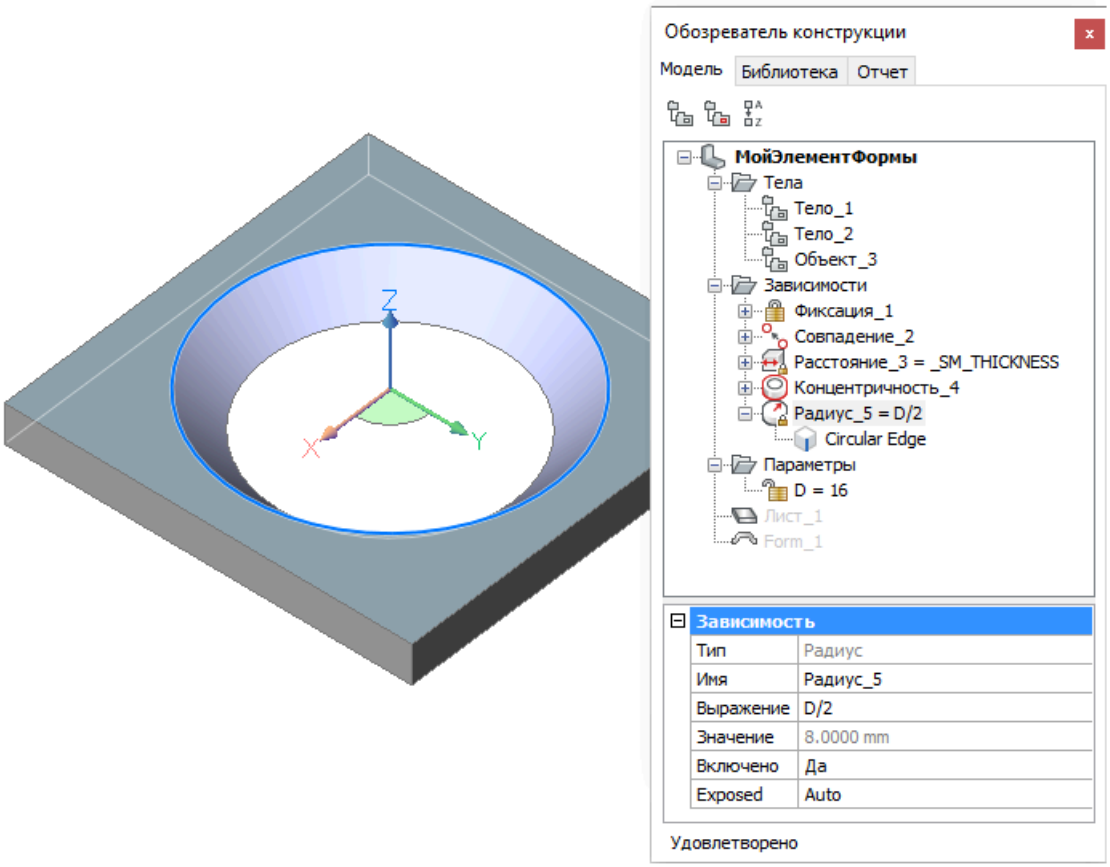
В поле **Выражение** введите 16.




10 Примените 3D зависимость *Радиус* ( ) к верхнему ребру конической грани.

Примите значение динамического размера, предложенное по умолчанию.

11 Для зависимости Радиус_5 в поле **Выражение** введите




D/2.

12 В **Обозревателе конструкции** щелкните правой кнопкой на имени элемента и в открывшемся контекстном меню выберите пункт **Добавить новый параметр**.

Переименуйте новый параметр в **A**.
Оставьте поле **Единицы** пустым.

В поле **Выражение** введите 45.

13 Для создания зависимости **Угол конусности** выполните одно из следующих действий:

- Запустите команду *DMANGLE3D* (), выберите опцию **Угол конусности** и укажите коническую грань.
- Установите курсор над конической гранью и выберите зависимость **Угол конусности** () на закладке **Зависимости** в **квадро-меню**.

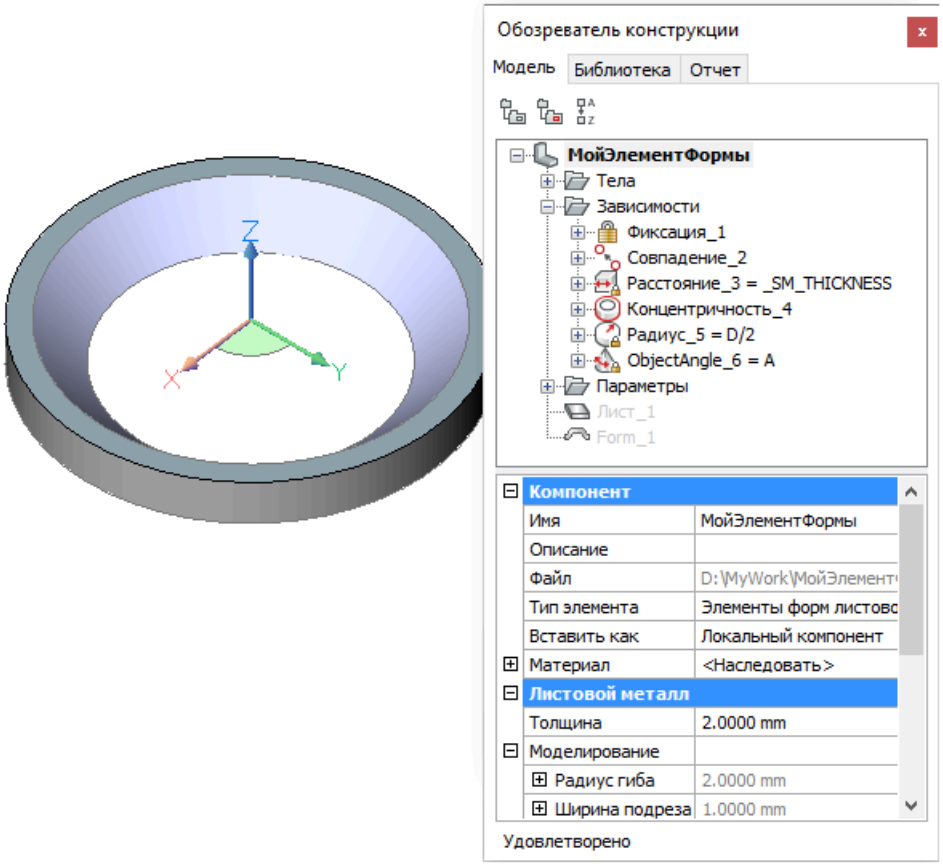
Подтвердите значение по умолчанию.

14 В поле **Выражение** для зависимости ObjectAngle_6 введите A.

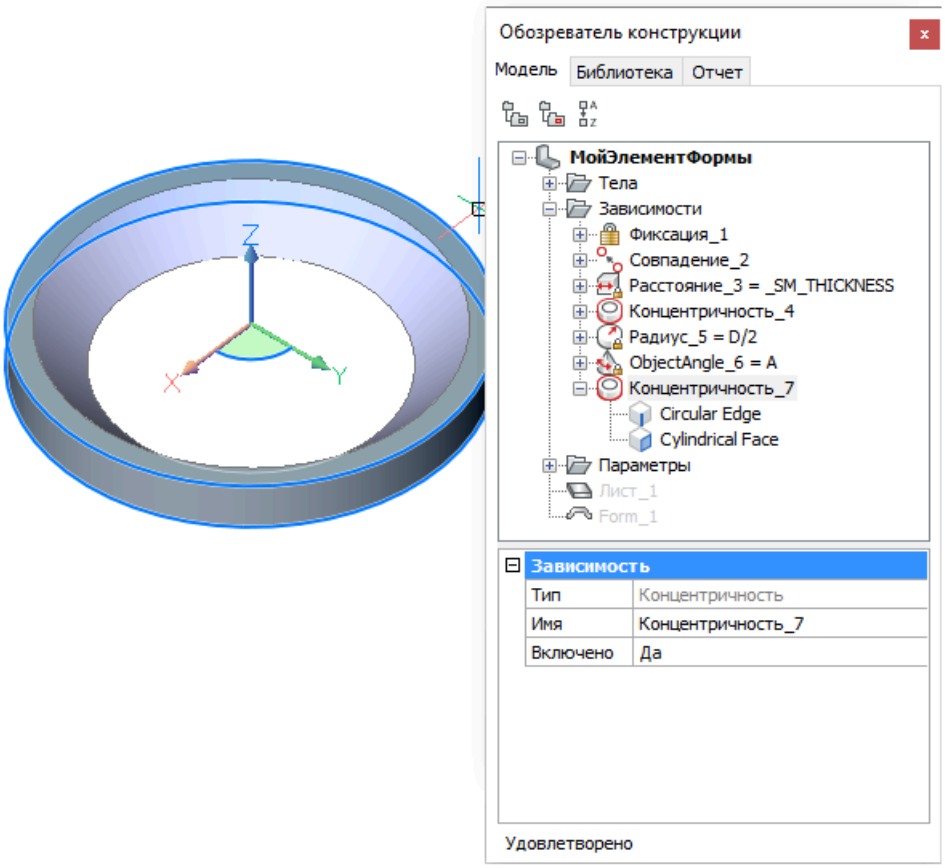
15 Так как наш элемент формы конический, мы можем преобразовать форму листа из параллелепипеда в цилиндрическую:

- Постройте на верхней грани листа окружность радиусом 9 мм.
- Включите режим **Поиск контуров** на панели **Параметры** закладки ленты **Главная**.
- Установите курсор над областью между окружностью и ребром листа стенки. Нажмите

кнопку **Выдавливание тела** ( ) в **квадро-меню** и переместите курсор вниз.




16 Примените 3D зависимость *Концентричность* ( ) для круглого ребра области и цилиндрической

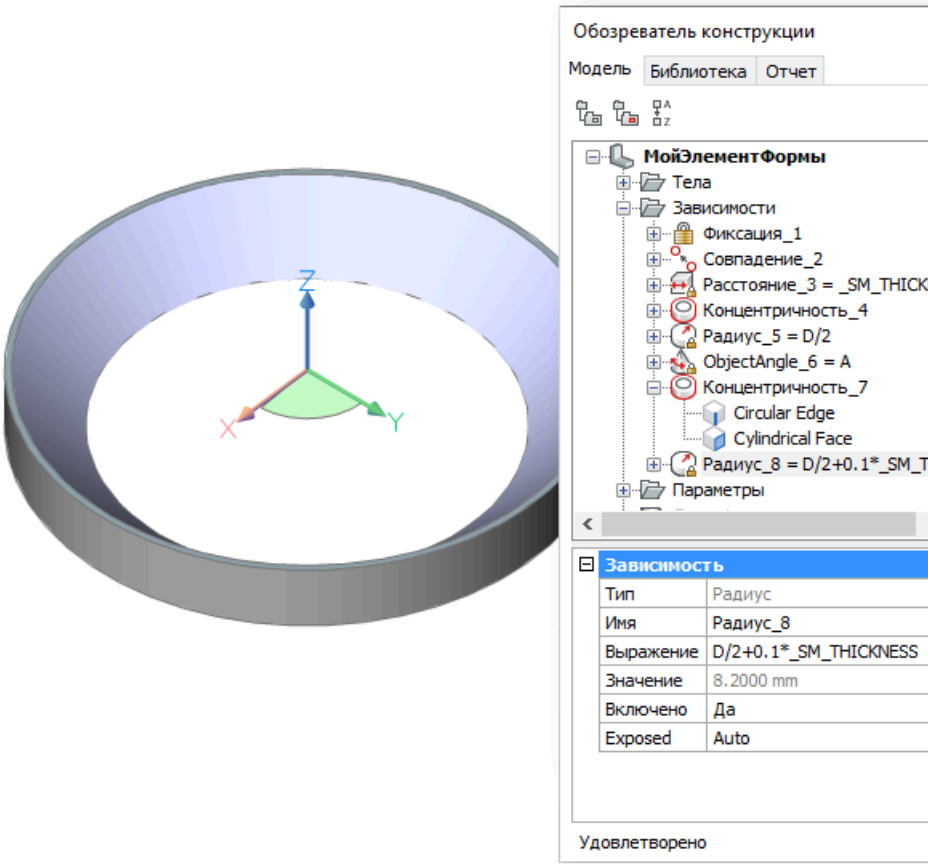



грани.

17 Примените 3D зависимость *Радиус* ( ) к цилиндрической грани.

Примите значение динамического размера, предложенное по умолчанию.

18 Для зависимости Радиус_8 в поле **Выражение** введите




D/2+0.1*_SM_THICKNESS.

19 В поле **Открытый** для обоих параметров A и D установите значение **On**.

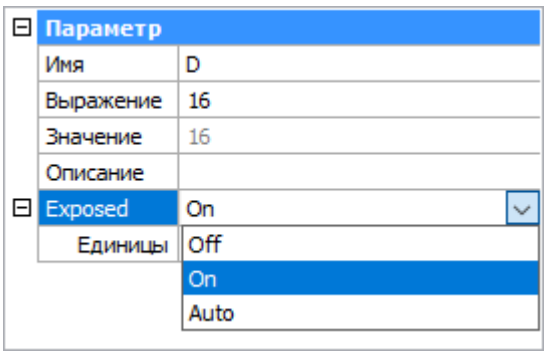



**Рисунок 144:** Параметр

Теперь создадим 2D зависимости для символов развертки.

20 Назначьте слой UNFOLD_SYMBOL_UP **текущим** и **выключите** остальные слои.

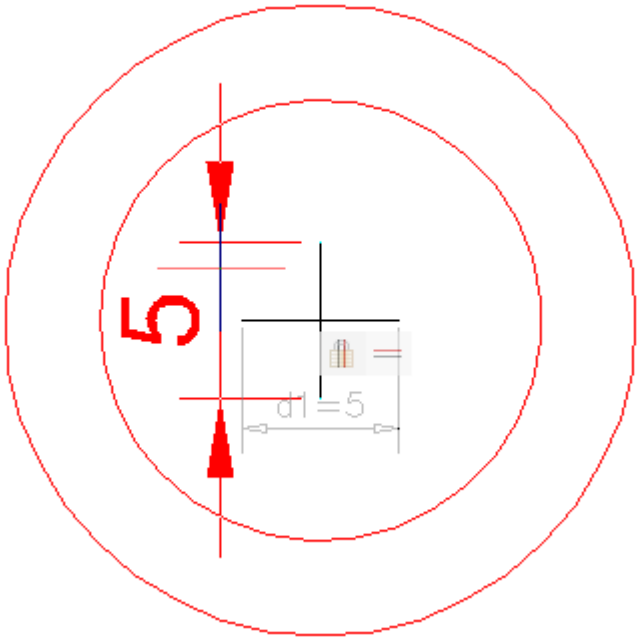
21 Примените 2D зависимость *Фиксация* ( ) к обеим линиям перекрестия центра окружности.

22 Примените 2D зависимость *Горизонталь* ( ) к горизонтальной линии перекрестия центра.

23 Примените 2D зависимость *Вертикаль* ( ) к вертикальной линии перекрестия центра.

24 Примените 2D зависимость *Концентричность* ( ) для двух окружностей.

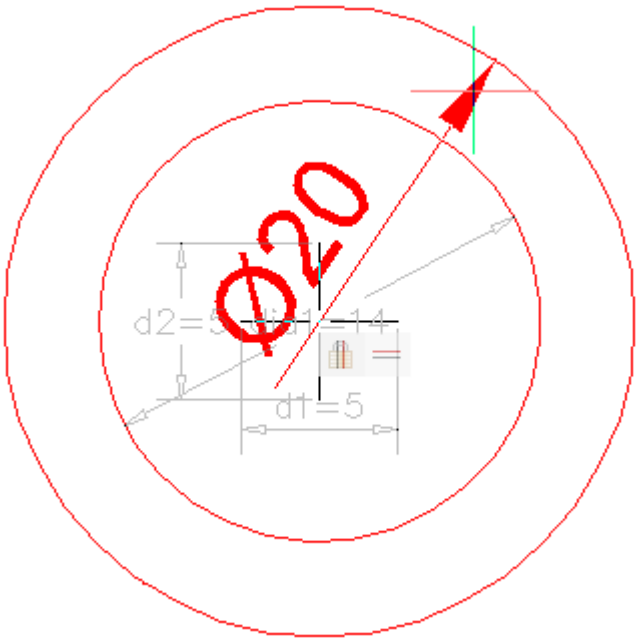
25 Примените 2D зависимость *Линейный размер* () между конечными точками линий перекрестия центра. Подтвердите значение по




умолчанию.

26 Примените 2D зависимость *Диаметр* () для обеих окружностей.

Подтвердите значение по умолчанию для dia1 (= внутренняя окружность) и dia2 (= внешняя окружность).




27 Для 2D зависимостей в поле **Выражение** введите:

- d1 = 1

- d2 = d1
- dia 1 = D - 2*_SM_THICKNESS/tan(A)
- dia 2 = D

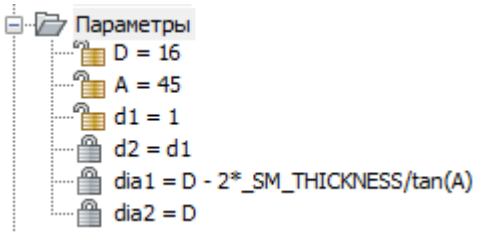
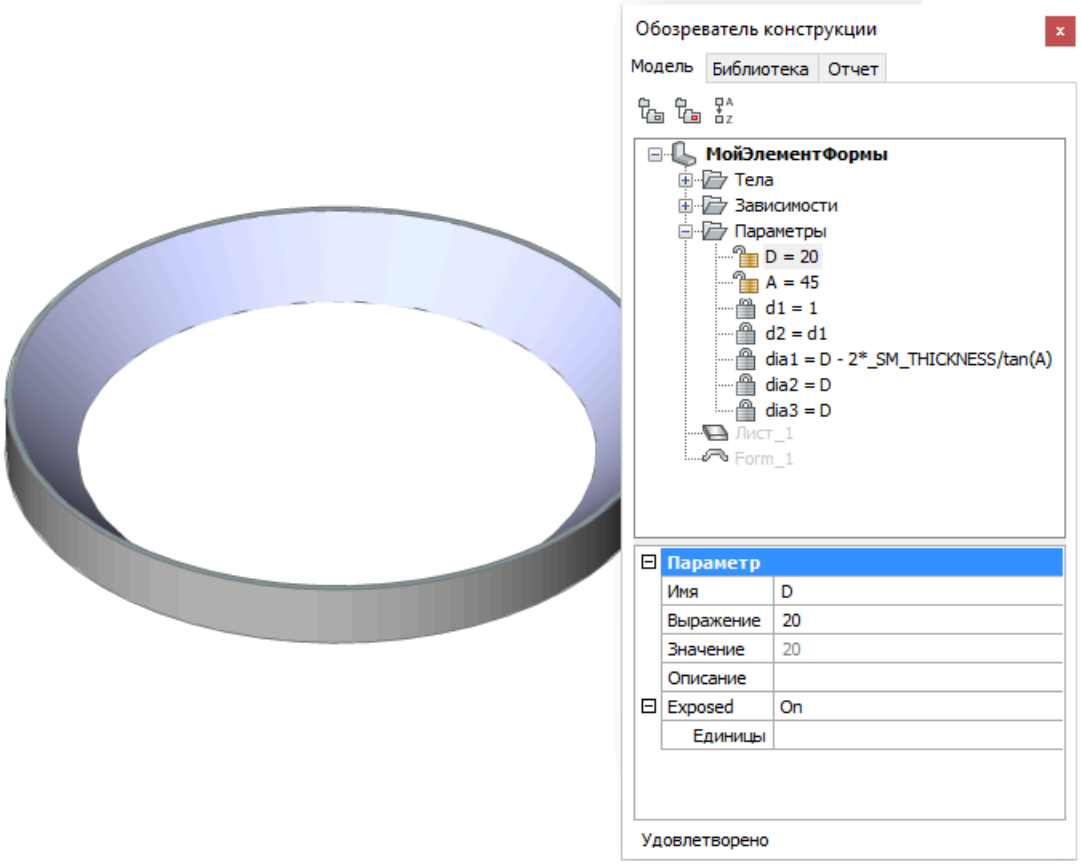



**Рисунок 145:** Параметры

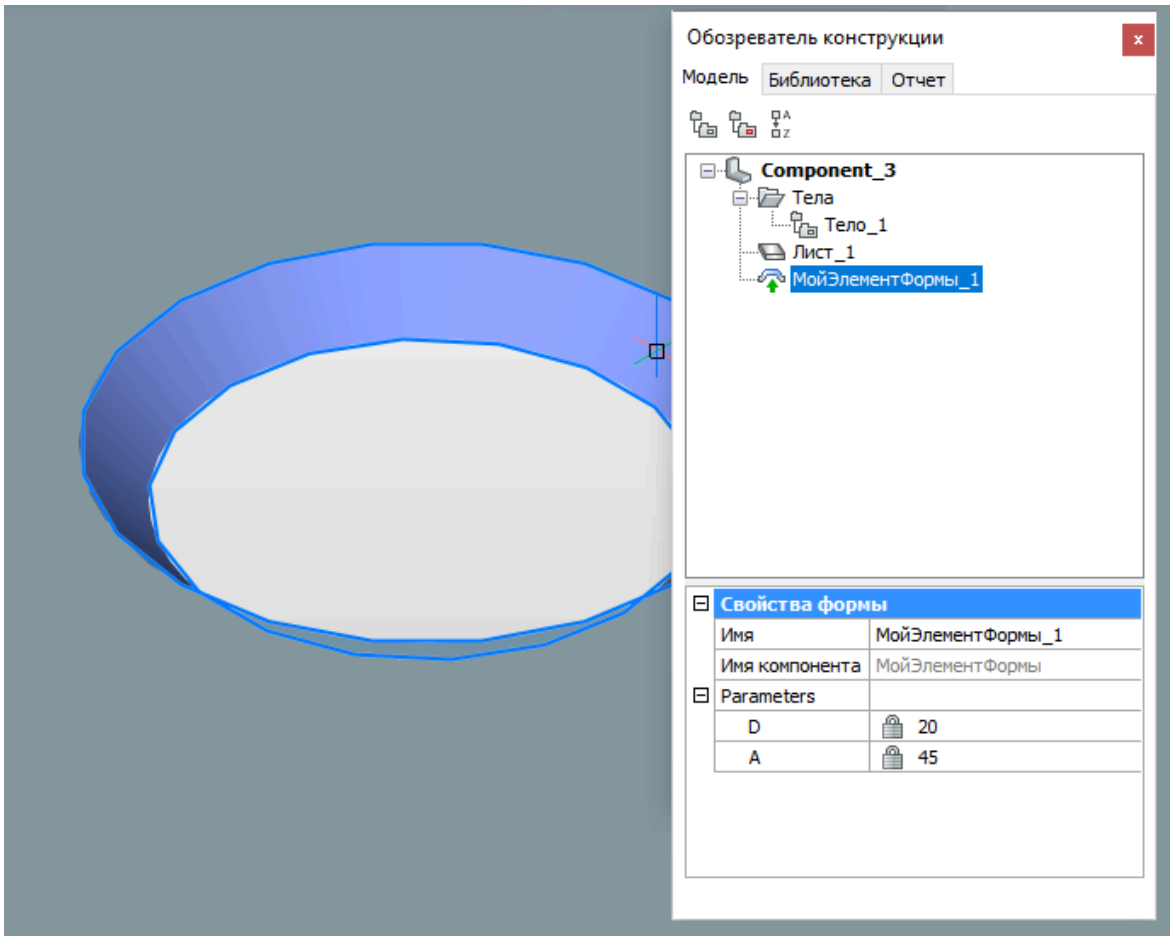
28 **Выключите** отображение всех слоев, кроме слоя 0 и сохраните




файл.

Теперь элемент формы может быть использован для создания конических параметрических отверстий в стенке детали из листового металла.

Для этого примените инструмент **Вставить элемент формы** (), который запускает команду *BMINSERT*.




См. также: Добавление элемента формы на инструментальную палитру

### 21.5.19.6 Создание ребра жесткости

Для создания ребра жесткости на основе 2D кривой, расположенной на грани стенки, используется команда *SMRIBCREATE*.

1 Выполните одно из следующих действий:

- Нажмите кнопку **Создать ребро** () на панели **Листовой металл**.
- Нажмите кнопку **Создать ребро** () на панели **Изменение** закладки ленты **Листовой металл**.
- Введите в командной строке команду *SMRIBCREATE* и нажмите кнопку **Enter**.

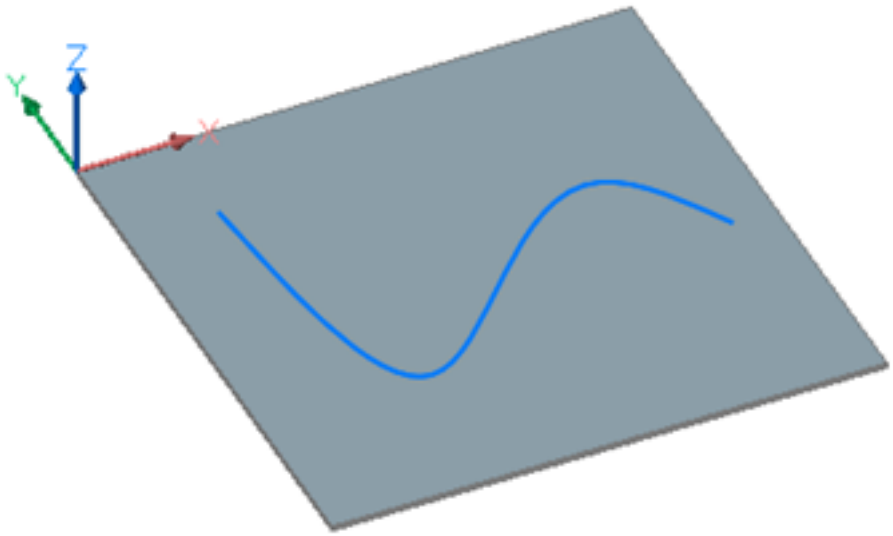
  Запрос программы: Укажите грань листа:

2 Укажите грань листа.

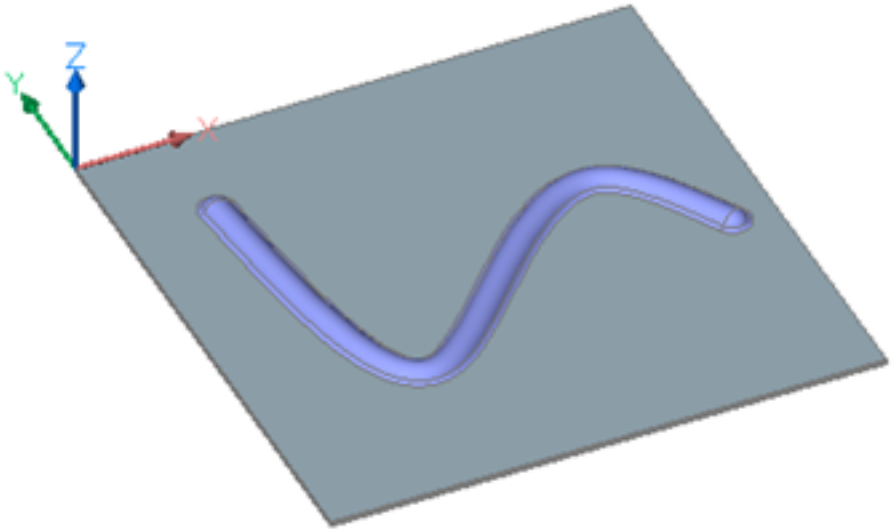
  Запрос программы: Укажите кривую или [Радиус профиля/Радиус скругления]:

3 Укажите кривую на выбранной грани листа.

Вы можете выбрать отрезок, полилинию, окружность, дугу, эллипс, эллиптическую дугу или сплайн. 2D кривые могут быть замкнутыми и незамкнутыми, но не должны пересекать себя или другие элементы формы.




После выбора кривой будет создано ребро жесткости:




### 21.5.19.7 Изменение ребра жесткости

При редактировании ребра жесткости доступны следующие операции:

- Изменение базовой кривой с помощью узловых точек.
- Изменение значений **Радиуса профиля** и **Радиуса скругления** в **Обозревателе конструкции**.

**Изменение управляющей кривой**

1 Выберите кривую.

Так как кривая скрыта ребром, нажмите клавишу **TAB** для выбора скрытых объектов или выберите кривую на панели **Структура**.

2 Измените положение узловых точек кривой.

Форма ребра жесткости изменится в соответствии с новым расположением узловых точек управляющей кривой.

3 Укажите грань листа. Запрос программы:

Укажите управляющую кривую [Радиус профиля/Радиус скругления/Параметры выбора]:

4 Укажите кривую на выбранной стороне листа.

**Редактирование свойств в Обозревателе конструкции**

1 Выберите ребро жесткости в **Обозревателе конструкции**.

Выбранный элемент будет выделен в модели.

2 Выберите свойство в разделе **Свойства ребра**.

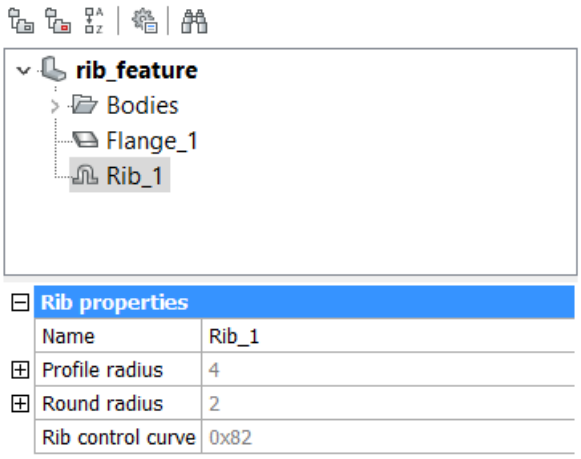



**Рисунок 146:** Свойства ребра

3 Введите необходимое значение для выбранного свойства.

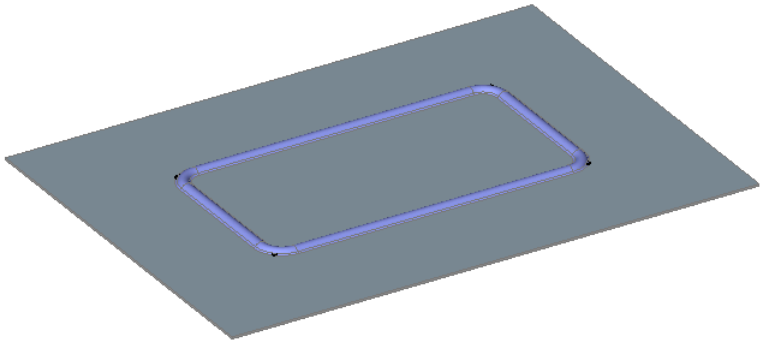
Выбранное ребро будет обновлено.

### 21.5.19.8 Отличие между управляющими кривыми и кривыми траектории

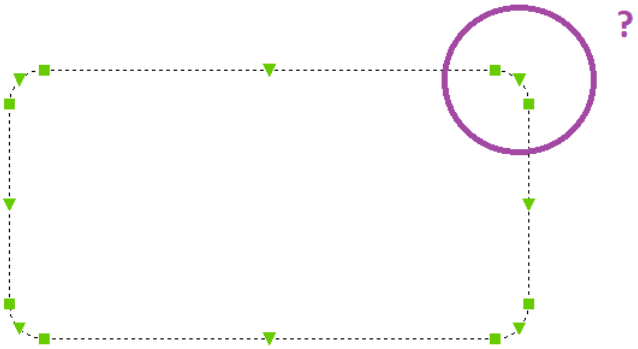
При создании прямоугольного ребра жесткости мы сначала рисуем прямоугольник, который затем используется для создания ребра с помощью команды *SMRIBCREATE*. При редактировании управляющей кривой с помощью узлов ребро обновляется в соответствии с изменением редактируемой кривой.



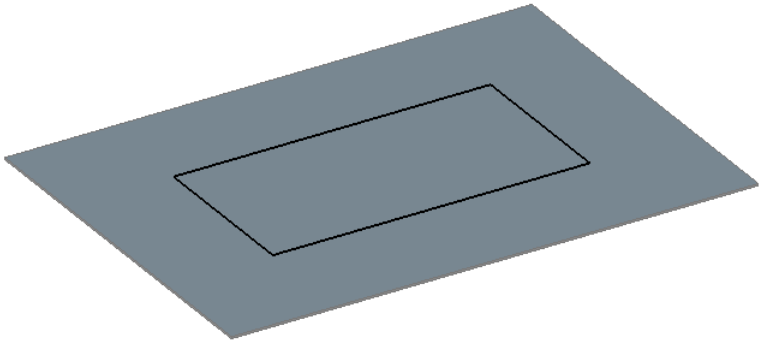
На производстве прямоугольная управляющая кривая заменяется скругленной кривой траектории в соответствии с физическими ограничениями прокатного инструмента:



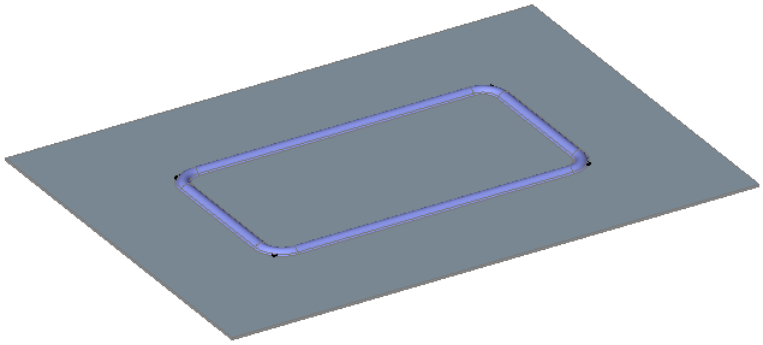
Кривой траектории, какой бы правильной она ни была, трудно управлять, например, когда необходимо переместить угол.



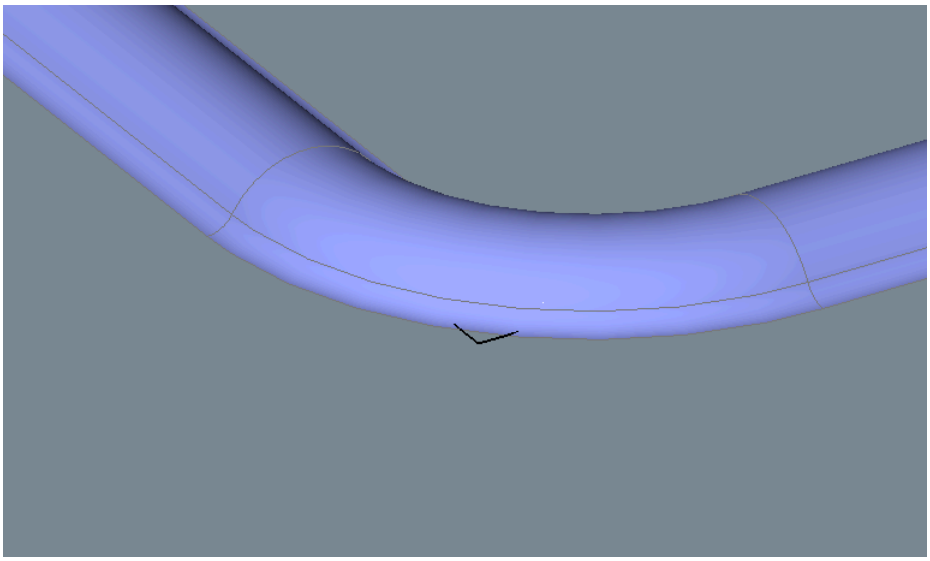

На этапе проектирования мы используем **управляющую кривую**, работать с которой намного проще:



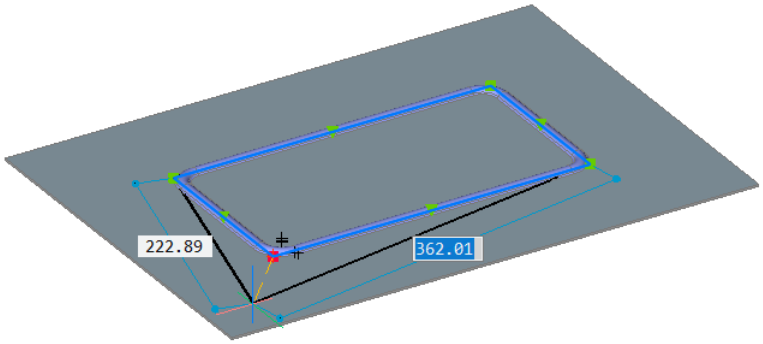
При создании ребра жесткости команда *SMRIBCREATE* добавляет скругления кривой в соответствии со значением параметра **Радиус закругления**, устанавливаемого в **Обозревателе конструкции**. Этот параметр не устанавливается отдельно для каждого элемента, так как зависит от физических параметров прокатного ролика инструмента, используемого при изготовлении детали:



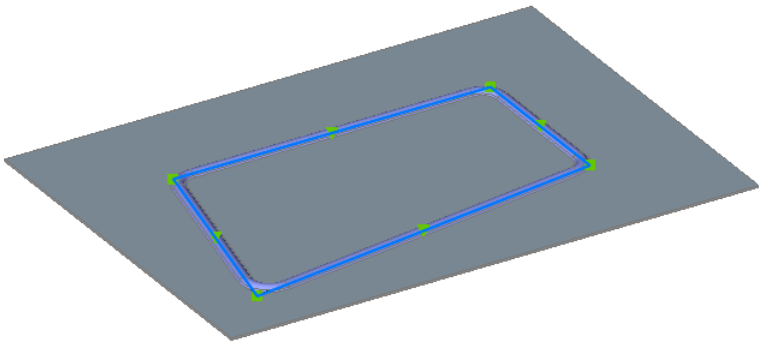
Масштабирование показывает разницу между 3D геометрией ребра и управляющей кривой:



При редактировании кривой с помощью узловых точек...

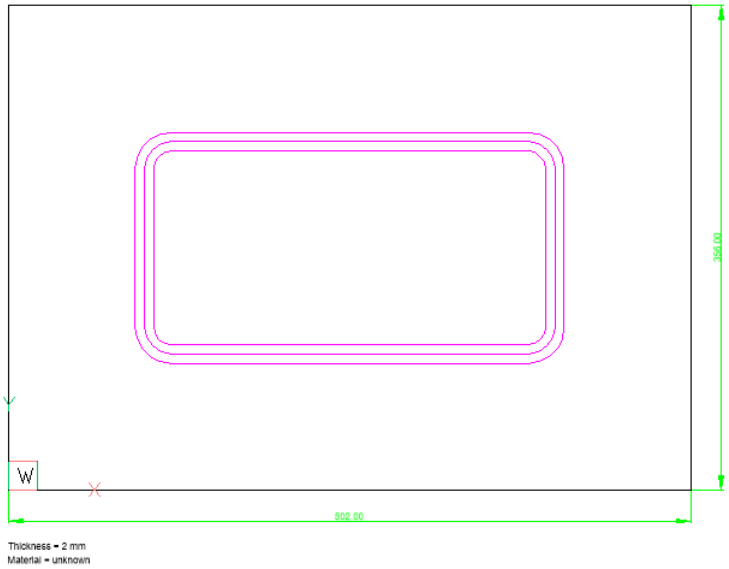



...форма ребра изменяется в соответствии с изменениями управляющей кривой:



Это то, что мы называем ассоциативностью между 2D кривой и 3D ребром жесткости. Распознавание ребер жесткости при выполнении команды *SMCONVERT* управляется пользовательской настройкой *SmConvertRecognizeRibControlCurves*.

На развертке, полученной с помощью команды *SMUNFOLD* видно, что кривые соответствуют 3D проекту и имеют скругления. Средняя кривая называется **кривой траектории**:

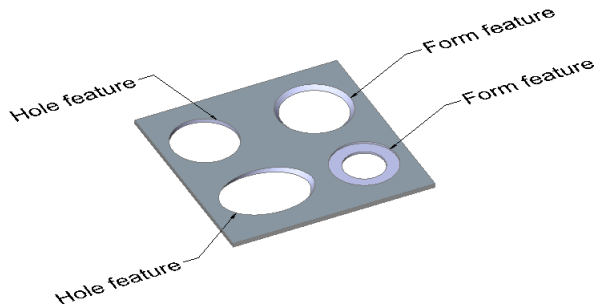



## 21.5.20 Работа с отверстиями

### 21.5.20.1 Обзор

Элемент Отверстие является упрощенной версией элемента формы. На приведенном ниже рисунке представлены различные элементы, которые состоят только из торцевой грани и распознаются командой *SMCONVERT* следующим образом:

- Ортогональные отверстия распознаются как отверстия.
- Простые неортогональные отверстия распознаются как отверстия.
- Ортогональные отверстия с фаской распознаются как элементы формы.
- Отверстия с двумя торцевыми гранями распознаются как элементы форм.




Отверстия распознаются командой *SMCONVERT*, если для пользовательской настройки *SMCONVERTRECOGNIZEHOLES* установлено значение ON.

Для отверстий доступны следующие базовые операции:

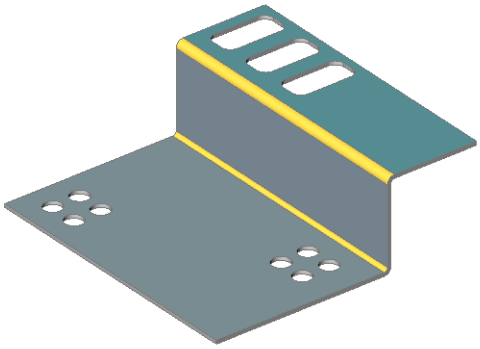
- *Удалить*, *Расформировать*, *Выбрать* и Отключить, которые могут быть выполнены с помощью соответствующих команд или опций, доступных в **Обозревателе конструкции** для листового металла.
- Выбор **Режима развертки элемента формы** при выполнении команды *SMUNFOLD*, включая режим **Символ**.
- Связывание вашей собственной 2D геометрии с разверткой.
- Исправление с использованием команды *SMREPAIR*. Применение этой команды обеспечивает ортогональность плоских граней отверстий.

### 21.5.20.2 Распознавание массивов отверстий

Команда *SMPARAMETRIZE* распознает прямоугольные массивы отверстий на стенках деталей из листового металла в соответствии со следующими правилами:

- Отверстия принадлежат одной стенке
- Отверстия расположены по прямоугольной сетке
- Массив отверстий не имеет пропусков (отсутствующих элементов)
- Отверстия имеют одинаковую ориентацию
- Минимальный размер отверстий 1x3 или 2x2.

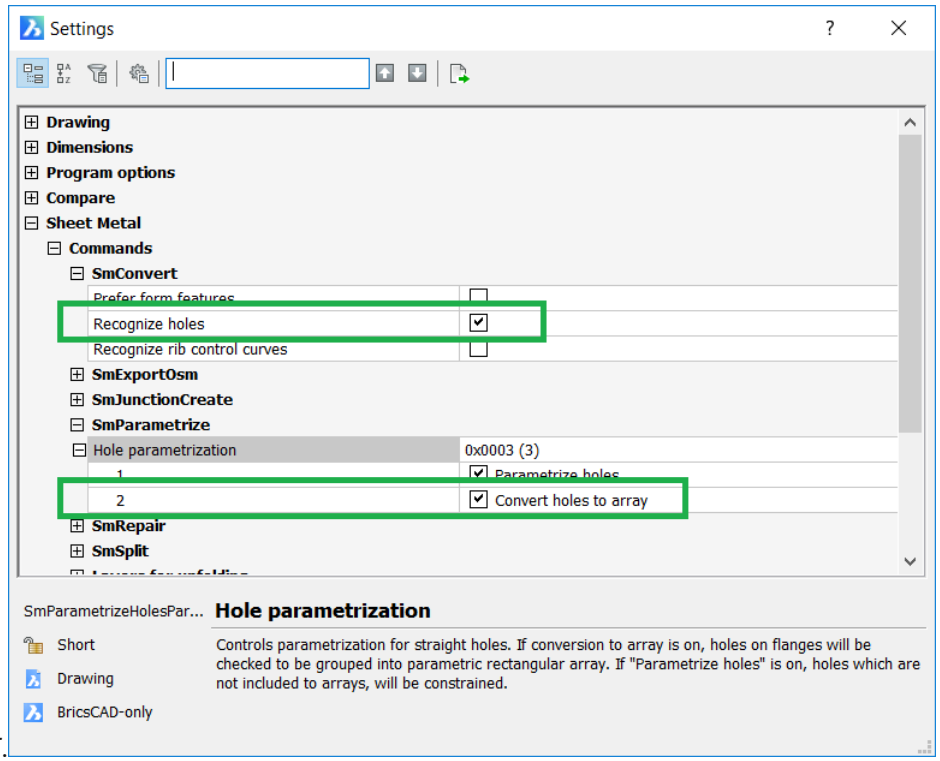
Рассмотрим пример детали со стенками и гибами.



1 В диалоговом окне *Параметры* Откройте раздел **Листовой металл/Команды**.

2 Включите опцию **Распознавать отверстия** для команды *SMCONVERT*.

3 Включите опцию **Объединять отверстия в массивы** и **Параметризовать отверстия**для команды




*SMPARAMETRIZE*.

4 Запустите команду *SMCONVERT*.

5 Выполните одно из следующих действий:

- Нажмите кнопку **Преобразовать в лист** ()на панели инструментов **Листовой металл**.
- Нажмите кнопку **Преобразовать в лист** ()на закладке ленты **Листовой металл**.
- Выберите инструмент **Преобразовать в листовой металл** в меню **Листовой металл**.

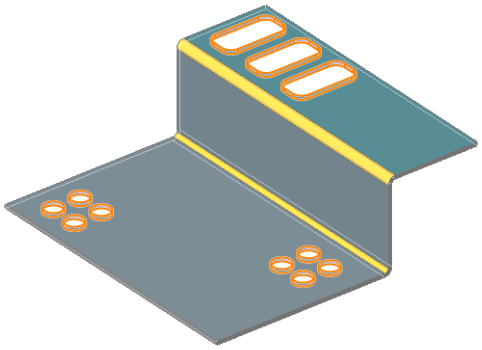
  Запрос программы: Выберите 3D тела или исходные грани или [Вся модель/Параметры выбора] <Вся модель>:

6 Выберите 3D тело для преобразования.

Запрос программы: Укажите 3D тела или [Вся модель/Параметры выбора] <Вся модель>:

7 Нажмите клавишу **Enter**, чтобы преобразовать выбранное 3D тело в деталь из листового металла.

На детали будут распознаны одиннадцать отверстий.



8 Запустите команду *SMPARAMETRIZE*.

9 Выполните одно из следующих действий:

- Нажмите кнопку **Параметризовать** () на панели инструментов **Листовой металл**.
- Нажмите кнопку **Параметризовать** () на закладке ленты **Листовой металл**.
- Выберите инструмент **Параметризовать** в меню **Листовой металл**.

  Запрос программы: Укажите тело для параметризации или [Вся модель/Параметры выбора] <Вся модель>:

10 Нажмите клавишу **Enter** для обработки детали из листового металла.

Отчеты команд должны выглядеть так, как показано на приведенном ниже рисунке:

Создано 4 зависимости расстояния

Создано 3 зависимости фиксации

Создано 0 зависимостей совпадения

Создано 0 зависимостей касания

Создано 0 зависимостей жестких связок

Создан прямоугольный массив 2x2 на основе элемента Hole_3.

Создан прямоугольный массив 2x2 на основе функции Hole_7.

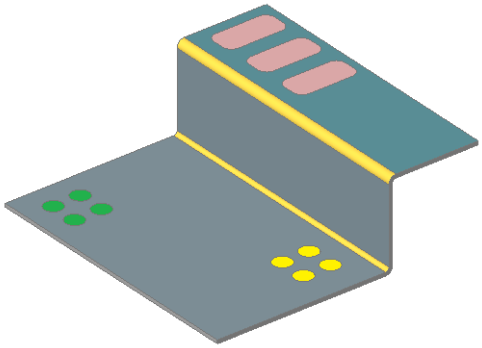
Создан прямоугольный массив 3x1 на основе функции Hole_9.

Итого: создано 7 зависимостей и 3 массива

***Прим.:*** Наименования элементов и распределение зависимостей могут отличаться в зависимости от версии BricsCAD.

Чтобы исследовать массивы, вы можете временно разморозить слой **BC_SUBTRACT**.

На приведенном ниже рисунке массивы были окрашены вручную.

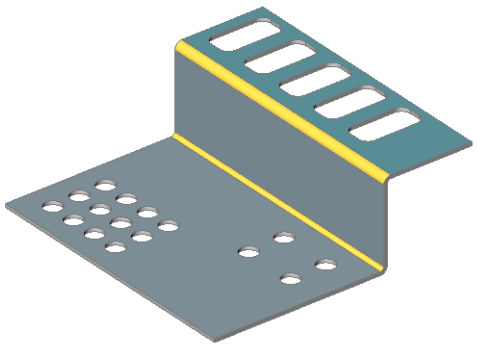


***Прим.:*** Обратите внимание, что желтый и зеленый массивы не объединены. В противном случае в массиве не было бы отверстий.

11 Измените **Свойства массива** в **Обозревателе конструкции**.

В приведенном ниже примере были внесены следующие изменения:

- Зеленый массив: число строк и столбцов массива изменено с 2x2 на 3x4.
- Желтый массив: интервалы по X и Y.
- Красный массив: число строк и столбцов изменено с 3x1 на 5x1.



### 21.5.21 Converting to Sheet Metal and Repairing

Use the *SMCONVERT* and *SMREPAIR* commands.

#### 21.5.21.1 To convert 3D geometry to a valid Sheet Metal part

A sheet metal part can be created from any 3D solid geometry, whether created in BricsCAD or imported from another CAD system. However, the geometry might not be a valid sheet metal model: missing bends and reliefs or thickness faces which are not orthogonal to the flange faces. BricsCAD Sheet Metal contains a dedicated toolset to convert such geometry to a valid sheet metal part and add all missing sheet metal features.

1 Do one of the following:

- Click the **Convert to Sheet Metal**tool button ( ) on the **Sheet Metal** toolbar.
- Click the **Convert to Sheet Metal** tool button ( ) on the **Sheet Metal** ribbon bar.
- Choose **Convert to Sheet Metal** in the **Sheet Metal** menu.

  You are prompted: Select 3D solids or [Entire model] <Entire model>:

2 Select the 3D solid to convert.

  You are prompted: Select 3D solids or [Entire model] <Entire model>:

3 Press **Enter** to convert the 3D solid to a sheet metal part.

The properties of the sheet metal part display in the **Mechanical Browser for Sheet Metal**.

***Прим.:*** Converted bodies can be unfolded or exported to .DXF or .OSM file formats for processing by a CAM system.

In some cases it might be necessary to fix converted bodies before unfolding/exporting:

- Create missing reliefs: *SMRELIEFCREATE* command.
- Convert hard edges to bends or junctions: *SMBENDCREATE* and *SMJUNCTIONCREATE* commands.
- Split flanges and lofted bends: *SMSPLIT* command.
- Repair sheet metal bodies: *SMREPAIR* command.

#### 21.5.21.2 To repair a Sheet Metal part

A sheet metal part may have inaccuracies in geometry and topology or sheet metal design issues. This is often the case for imported parts which are designed in geometrical kernels different from ACIS.

Examples of such problems are:

- Thickness faces, which are not perpendicular to the flange/bend faces.
- Lofted bends are not tangential to the adjacent flanges/bends.
- Wrong bends.
- General ACIS issues.

Such problems can prevent editing operations from being executed successfully.

The recommended workflow to fix such problems is:

- Run the *DMAUDIT* command to fix general modeling issues.
- Run the *SMREPAIR* command.

### 21.5.21.3 To repair a Sheet Metal body

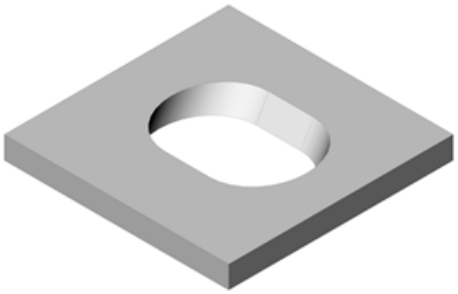
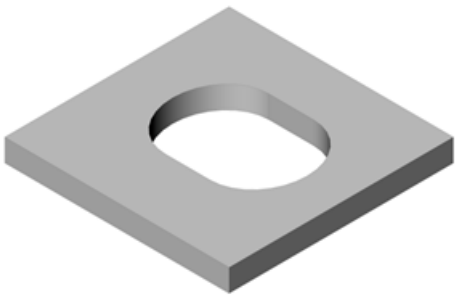
Sheet metal bodies with thickness faces, which are not perpendicular to the flange/bend faces, can not be correctly manufactured using traditional tools.

1 Do one of the following:

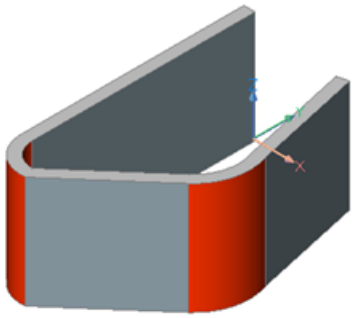
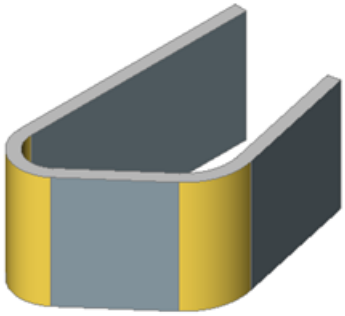
- Click the **Repair Sheet Metal** tool ( ) on the Heal panel of the Sheet Metal ribbon tab;
- Click the **Repair Sheet Metal** tool ( ) on the **Sheet Metal** toolbar;
- Choose **Repair Sheet Metal** in the **Sheet Metal** menu;
- Type *SMREPAIR* in the Command line.

  You are prompted: Select flange face or lofted bend face to repair or [switch on Local repairing/ Scan for lost details]:

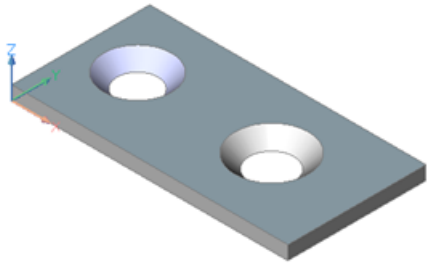
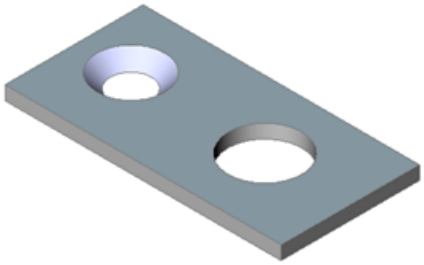
2 Select the face to be repaired.

| | |
|---|---|
|  |  |
| The same model before ... | ... and after applying a repair operation: |

When the model has wrong bends *SMREPAIR* transforms them to regular bends:

| | |
|---|---|
|  |  |
| Wrong bends, which have a non-tangential connection with the flanges. | Correct bends after repairing. |

Form features are translated to the new repaired model:

| | |
|---|---|
|  |  |
| Model before repairing. | The form feature has not been changed, while a hole, which is not a form feature, is orthogonalized. |

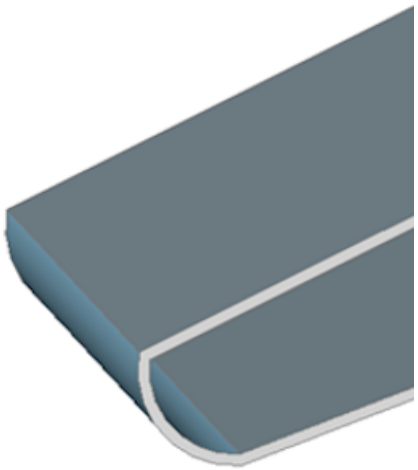
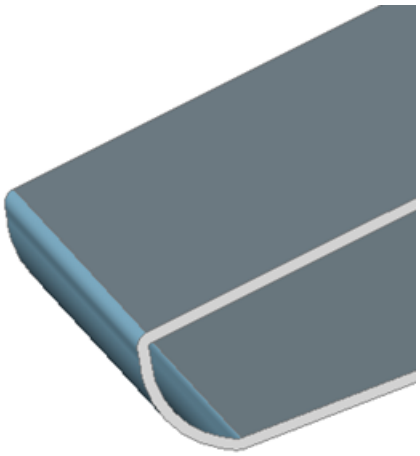
### 21.5.21.4 To repair a Sheet Metal body with lofted bends

1 Do one of the following:

- Click the **Repair Sheet Metal** tool ( ) on the **Heal** panel of the **Sheet Metal** ribbon tab.
- Click the **Repair Sheet Metal** tool ( ) on the **Sheet Metal** toolbar.
- Choose **Repair Sheet Metal** in the **Sheet Metal** menu.
- Type *SMREPAIR* in the Command line.

  You are prompted: Select flange face or lofted bend face to repair or [switch on Local repairing/ Scan for lost details]:

2 Select a face of a flange or a lofted bend. The selected face provides the surface to be thickened.

| Before | After |
| --- | --- |
|  |  |
| The lofted bend is not tangent to the flange. Unfolding is impossible. | Notice the difference in the connection of the blue lofted bend with the top flange, which is smooth now. Unfolding is possible. |

Option: If the **SmRepairLoftedBendMerge** user preference is ON, adjacent lofted bends with a tangential connection are merged to a single lofted bend.

### 21.5.22 Unfolding Sheet Metal Parts and Exporting to CAM

#### 21.5.22.1 Overview

Use the *SMUNFOLD* command.

Sheet metal parts can be designed in many different ways. To manufacture the sheet metal part, a flat pattern with bend annotations is needed, which is then used in a CAM system to program CNC machines, such as laser cutters and press brakes. This flattening operation is called unfolding.

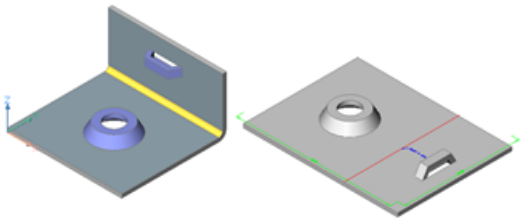
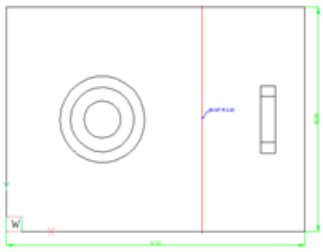
#### 21.5.22.2 Unfolding a sheet metal body

The *SMUNFOLD* command unfolds sheet metal parts. BricsCAD automatically creates a solid body corresponding to the flat metal sheet needed to manufacture the sheet metal part using bending techniques. This sheet is placed on the XY-plane and is oriented along the coordinate axes similarly to the orientation of the initial body in 3D space. To change the orientation of the unfolding in Z axis, run the *SMUNFOLD* command again and select the opposite face of the initial body.

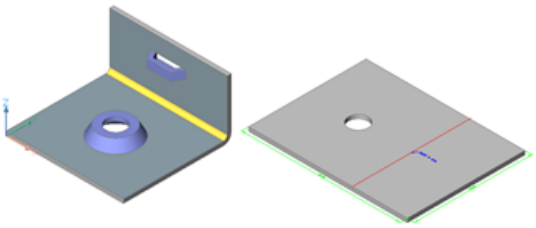
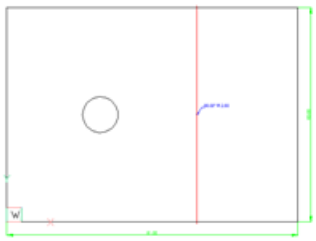
Form features are unfolded according to the **Unfolding Mode** setting in the **Mechanical Browser**.

The options are:

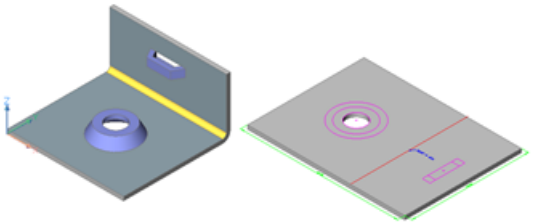
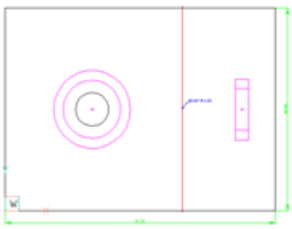
- **Keep** (0): allows to keep form features geometry on the unfolded representation.

| 3D model and unfolded body | 2D |
|---|---|
|  |  |

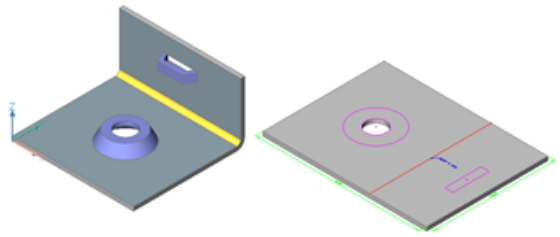
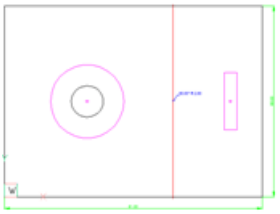
- **Remove** (1): removes the form feature geometry from the unfolded representation.

| 3D model and unfolded body | 2D |
|---|---|
|  |  |

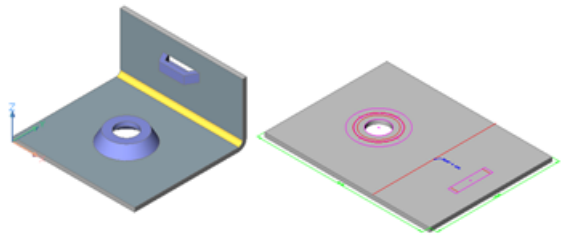
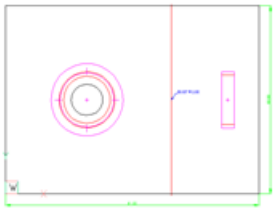
- **Project** (2): generates a form feature symbol by projecting the form feature edges onto the flange plane.

| 3D model and unfolded body | 2D |
|---|---|
|  |  |

- **Contour** (3): generates a form feature symbol by projecting the external contour and holes of the form feature onto the flange plane.

| 3D model and unfolded body | 2D |
| --- | --- |
|  |  |

- **Symbol** (4): creates the unfold symbol on the **UNFOLD_SYMBOL_UP** or **UNFOLD_SYMBOL_DOWN** layer depending on the direction of the form feature.

| 3D model and unfolded body | 2D |
| --- | --- |
|  |  |

The unfold mode for the current sheet metal part can be set in **Mechanical Browser**.

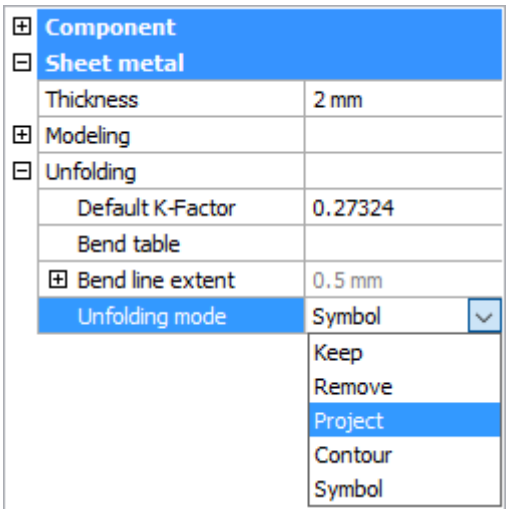



**Рисунок 147:** Unfolding mode

Lofted bends lying on cylindrical and conical surfaces can be unfolded analytically: the dimensions and shape of the unfolded lofted bend will be as if its middle surface was mathematically unrolled on the

plane. This shape can also be achieved by a highly increased number of samples for the lofted bend. You must set the **Unfolding mode** individually for each lofted bend feature. If analytical unfolding is allowed, the option will be present in a drop down list:

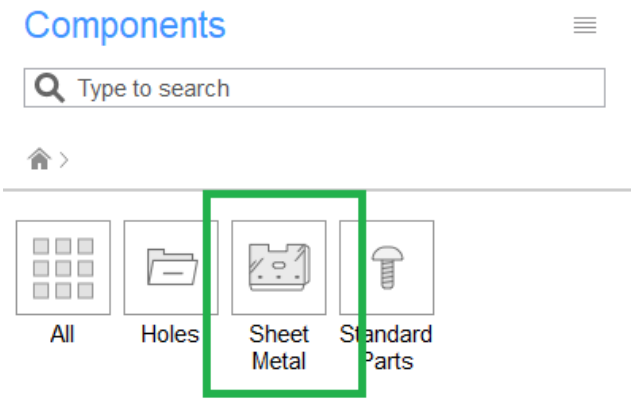



**Рисунок 148:** Analytical

### 21.5.22.3 To unfold a sheet metal body

1 Launch the *SMUNFOLD* command.

You are prompted: Select a flange or lofted bend face to start unfolding [Associative]:

2 Select a face on sheet metal part.

You are prompted: Select position of the unfolded body:

3 Specify a point.

You are prompted: Validate the unfolded body and select an option [save 2D geometry/save 3D geometry/Optimize bend annotations/Keep] <Keep>

4 Do one of the following:

- Save the unfolded 3D solid in a separate drawing file.
- Convert the unfolded 3D solid to a 2D drawing file (*.DWG or *.DXF).
- Rearrange the bend annotations.
- Keep the block with the unfolded part in the sheet metal part model.

The *SMUNFOLD* command considers the deformation of the sheet metal material during bending. When a flat sheet of metal is bent into a 3D part with a bending tool (for example a press brake), the material is plastically deformed, which means it is compressed at the inside of the bend and stretched at the outside. As a result, the length of the part measured along its surface is different in the flat and bend states. BricsCAD can automatically compute the proper unfolded length of your part based on the material deformation properties. These properties can be defined by setting the value of **K-Factor** parameter or by attaching a Bend Table.

### 21.5.22.4 Managing layer properties of the 2D geometry

The 2D geometry generated by the *SMUNFOLD*, *SMEXPORT2D*, and *SMASSEMBLYEXPORT* commands has different colors for contours, bend lines up, bend lines down, overall dimensions, form features up, form features down, bend annotations and attributes.

You can set the colors differently in the **Sheet Metal/Commands/Layers for unfolding** settings group on the *Settings* dialog box. Each entity type is placed on a dedicated layer of which the color is controlled through a series of user preferences:

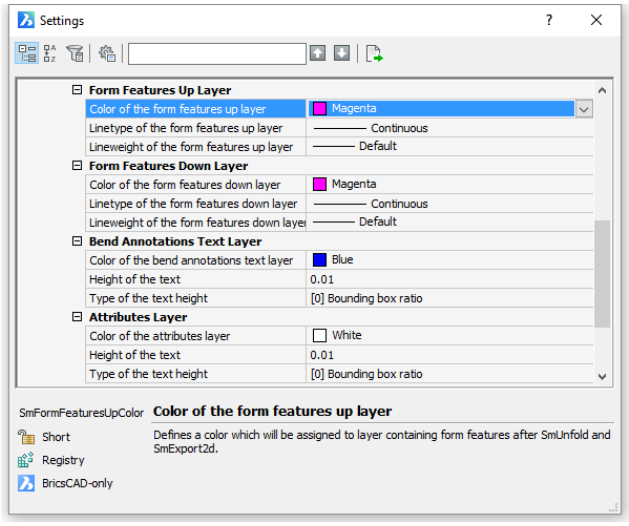



**Рисунок 149:** Settings

If the 2D geometry of a form feature was taken from a library component, its properties are controlled by the properties of this component at the moment of insertion. There are 2 other modes of **Unfolding Mode** which lead to generation of 2D geometry for form features: **Project** or **Contour**. In these modes 2D geometry is generated automatically from 3D geometry and its appearance is controlled by **Form Features Up Layer** and **Form Features Down Layer** settings groups.

### 21.5.22.5 To export a sheet metal body

Sheet metal parts created or edited in BricsCAD can be processed by different CAM systems (such as JETCAM or CADMAN-B) to generate NC code for cutting and bending machines.

The user preference controls the **save 2D geometry** option of the *SMUNFOLD* command to customize the .DXF file to be compatible with the list of popular CAM systems.

The *SMEXPORT2D* command exports the unfolded representation of a sheet metal body as a 2D profile in *.DXF or *.DWG file format.

The *SMEXPORTOSM* command exports a sheet metal solid to the *.OSM (Open Sheet Metal) file format (native for the CADMAN-B CAM system).

### 21.5.22.6 To export an assembly with sheet metal parts to .DXF

The *SMASSEMBLYEXPORT* command searches the assembly tree for sheet metal design and exports it to a *.DXF file. As a result, a set of *.DXF files, sorted by thickness in different subfolders is created

in the output folder. The command generates a HTML report which allows to control the output. Most of the sheet metal parts will be processed automatically. Poor sheet metal parts will be reported by the command to be reworked manually.

### 21.5.23 Ассоциативные развертки 3D тел

#### 21.5.23.1 Обзор

BricsCAD обеспечивает двунаправленную ассоциативность между 3D моделью детали из листового металла и ее разверткой. В этом режиме оба представления детали существуют в отдельных документах: исходная модель детали сохраняется в основном чертеже, а ее развертка размещается во временном документе. Ассоциативность чертежа детали и ее развертки является временной и существует до тех пор, пока открыты оба документа. Чтобы работать в режиме ассоциативности в следующем сеансе BricsCAD или после закрытия документа, необходимо повторить процедуру создания развертки.

#### 21.5.23.2 Что означает ассоциативность?

Ассоциативная развертка позволяет вам делать резы на плоской геометрии и передавать изменения в 3D модель. Это помогает упростить выполнение двух задач:

- Создание сложных, нестандартных подрезов для гиба и угловых вырезов.
- Обеспечение отсутствия самопересечений при создании развертки.

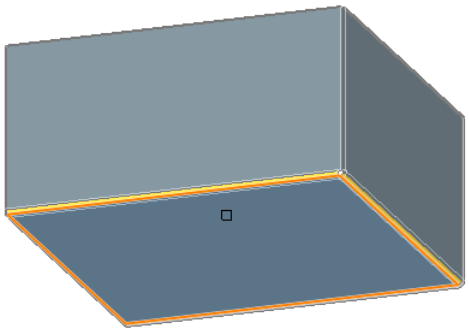
Эти задачи легче выполнять в режиме развертки, когда деталь из листового металла представлена как набор отдельных плоских твердых тел (по одному для каждого элемента). Для удаления твердотельных листов и гибов на развертках используйте команду *SMDELETE*.

Ассоциативность является двунаправленной - изменения могут передаваться из развертки в 3D модель и наоборот. Изменения, сделанные в одном из этих представлений, передаются в документ с другим представлением после его активации..

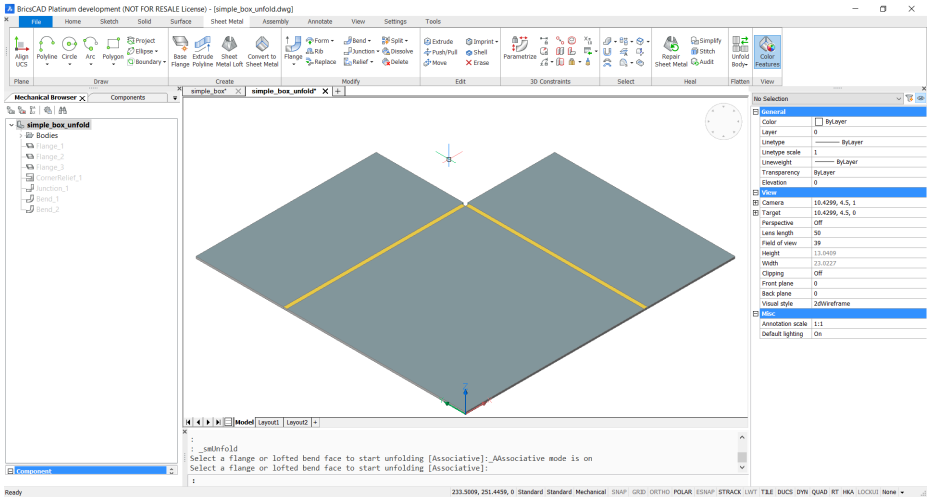
#### 21.5.23.3 Переключение в режим ассоциативности

Режим ассоциативности включается с помощью команды *SMUNFOLD*. Убедитесь, что нет выбранных граней.

На закладке ленты **Листовой металл** нажмите кнопку **Ассоциативная развертка** () и выберите грань **листа**.



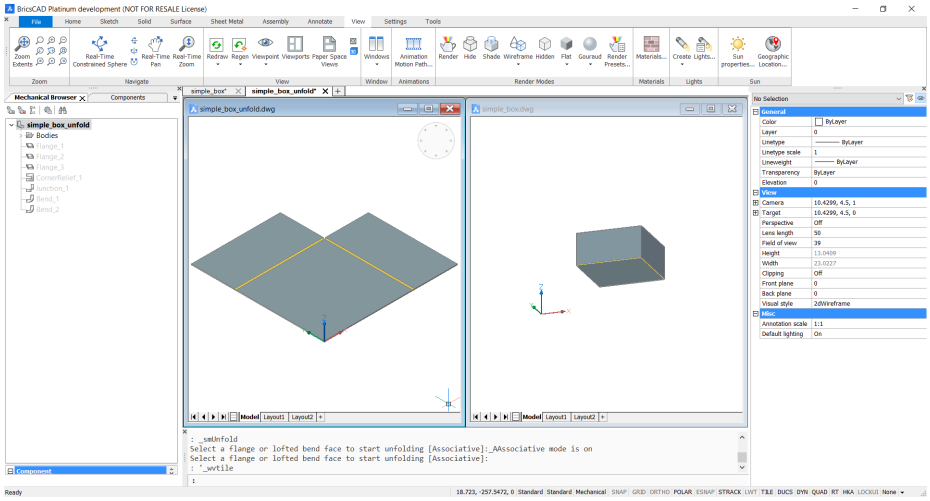
Приложение выполнит расчет развертки и переключит ее в ассоциативный режим. Условие успешности данной операции такое же, как и при обычном раскладывании. Продолжайте работу в режиме ассоциативности только в том случае, если во время операции не появилось никаких ошибок или предупреждений. Будет создан новый временный документ, содержащий развертку детали.




***Прим.:*** Нет необходимости сохранять этот документ. Он используется для передачи результатов операции редактирования из развертки в 3D модель.

Для просмотра 3D модели и разверток используйте **вертикальное разделение** рабочего окна BricsCAD.

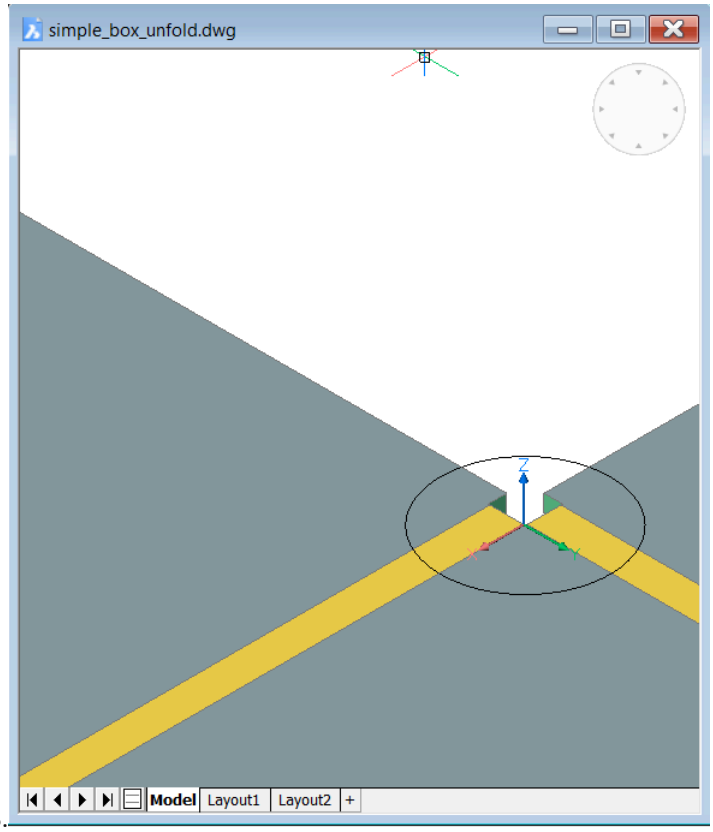
Нажмите кнопку **Разделить вертикально** на панели инструментов или на ленте.



***Прим.:*** В **Обозревателе конструкции** при выборе **геометрии развертки** отображается список отключенных элементов.

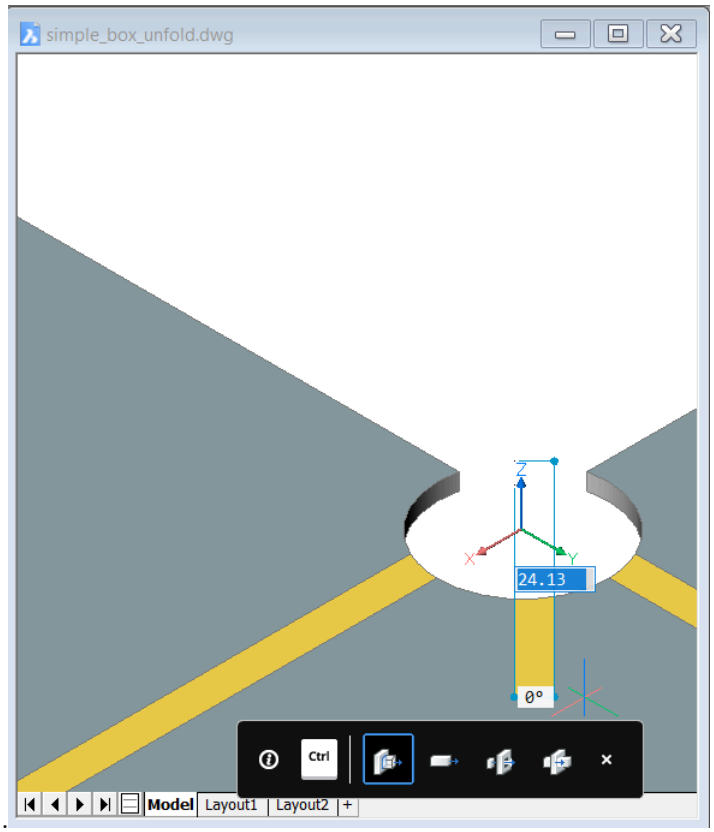
### 21.5.23.4 Редактирование геометрии

1 Работая с разверткой, начертите в области углового выреза



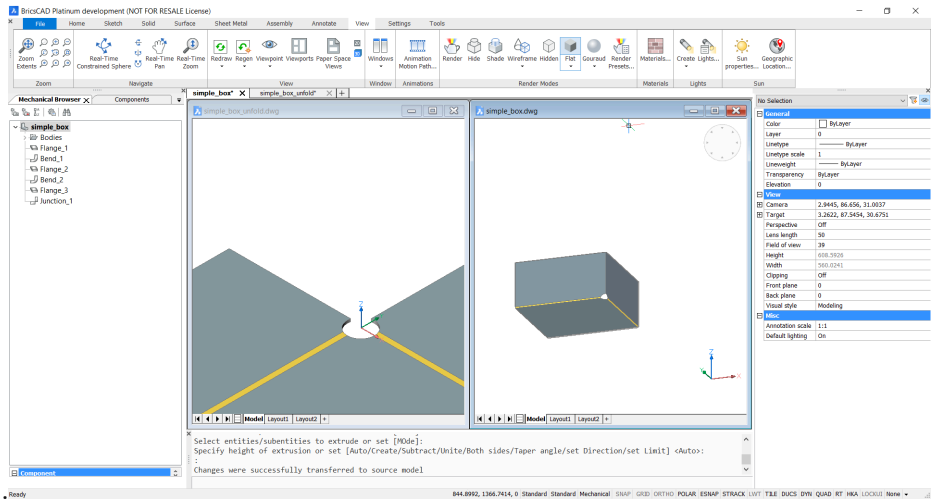

окружность.

2 Примените команду *DMEXTRUDE* к окружности, чтобы вырезать часть листа по контуру этой




окружности.

3 Снова выберите исходный документ, чтобы сделать его

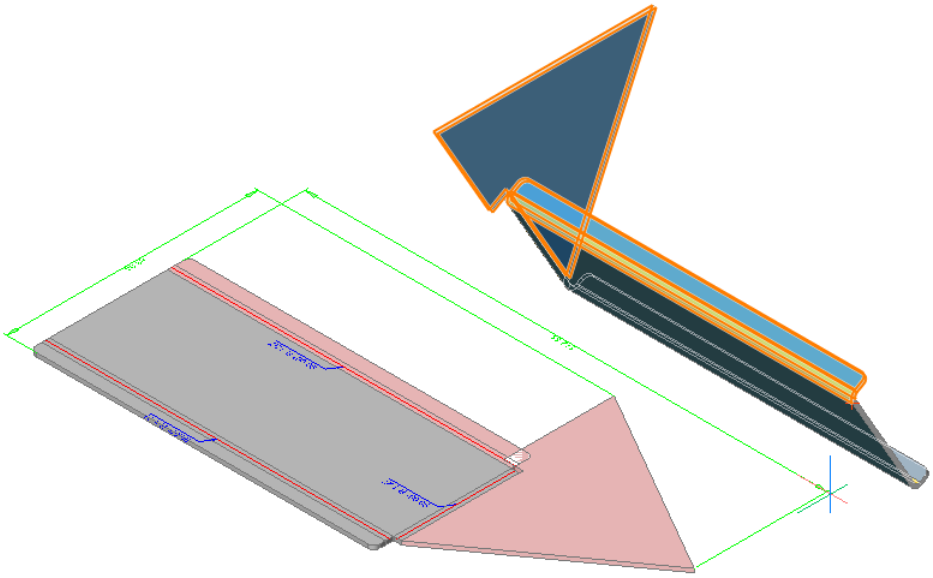


активным.

4 Модель будет обновлена с учетом изменения формы углового выреза. В командной строке отобразится сообщение: Изменения были успешно перенесены в исходную модель.

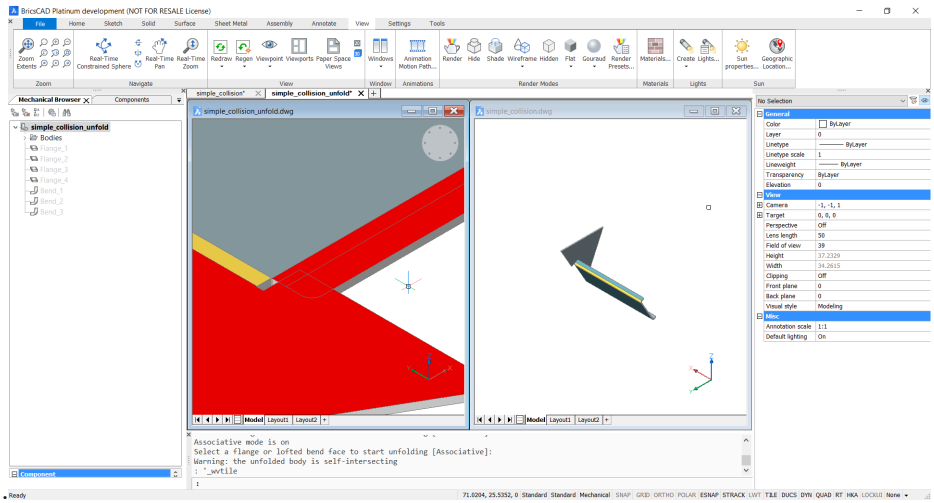
### 21.5.23.5 Предотвращение самопересечений

Другим полезным применением ассоциативной развертки является исправление возможных коллизий.

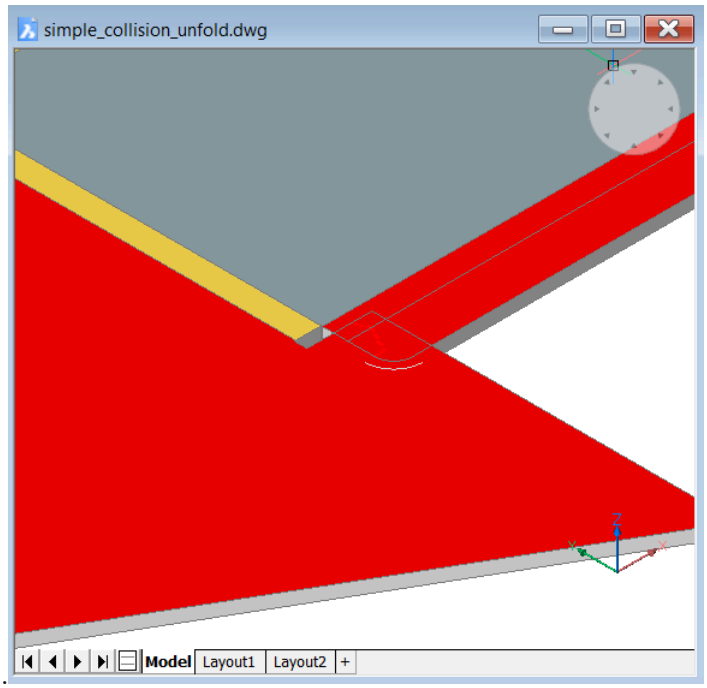


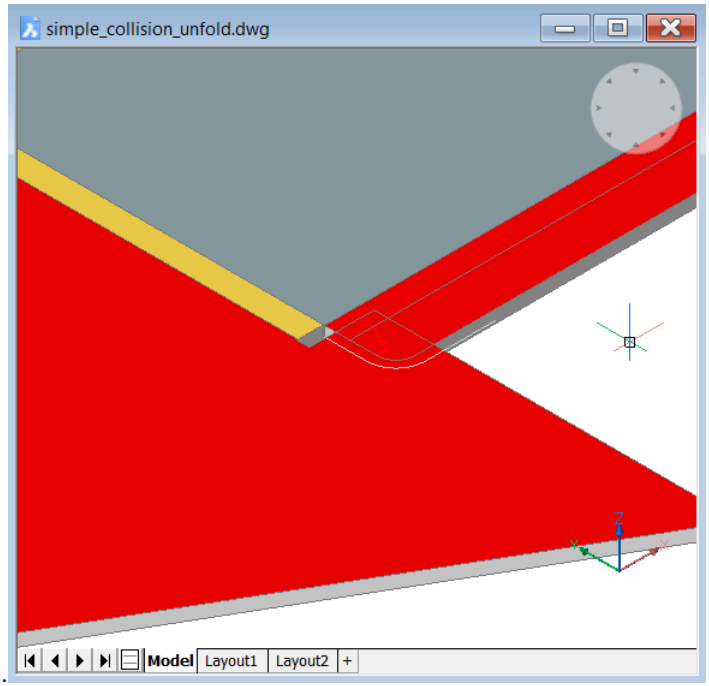
В этом примере мы не знаем, как необходимо изменить 3D модель, чтобы избежать коллизий при получении ее развертки. Выполним обработку модели с помощью инструмента Ассоциативная развертка.

1 На закладке ленты **Листовой металл** нажмите кнопку **Ассоциативная развертка** (), выберите грань **листа**, создайте развертку и нажмите кнопку **Разделить вертикально** () на панели инструментов или на ленте, чтобы расположить открытые документы вертикально и приблизить область коллизии.

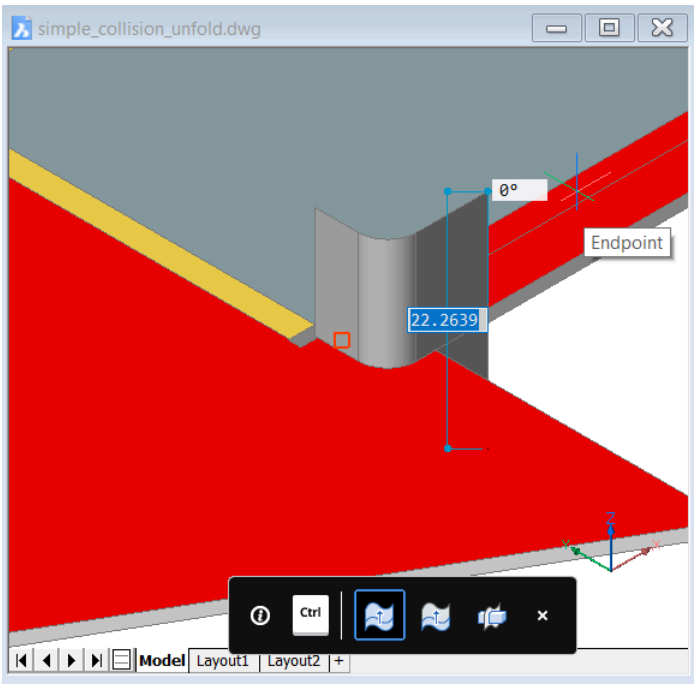


2 Выберите скругленную кромку и с помощью инструмента Подобие сместите ее на

1. 

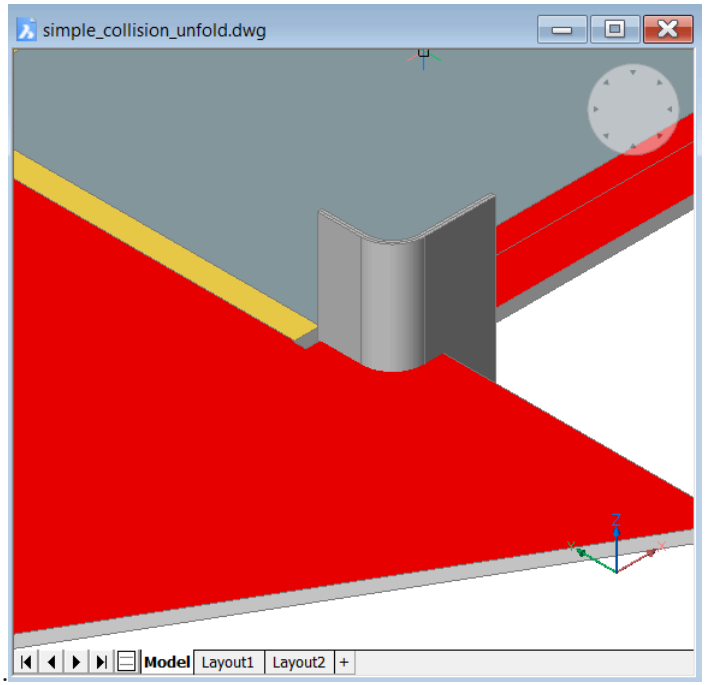

3 Добавьте к скруглению прямые линии и соедините их. 


4 Примените команду *DMEXTRUDE* и выдавите полилинию в обе




стороны.

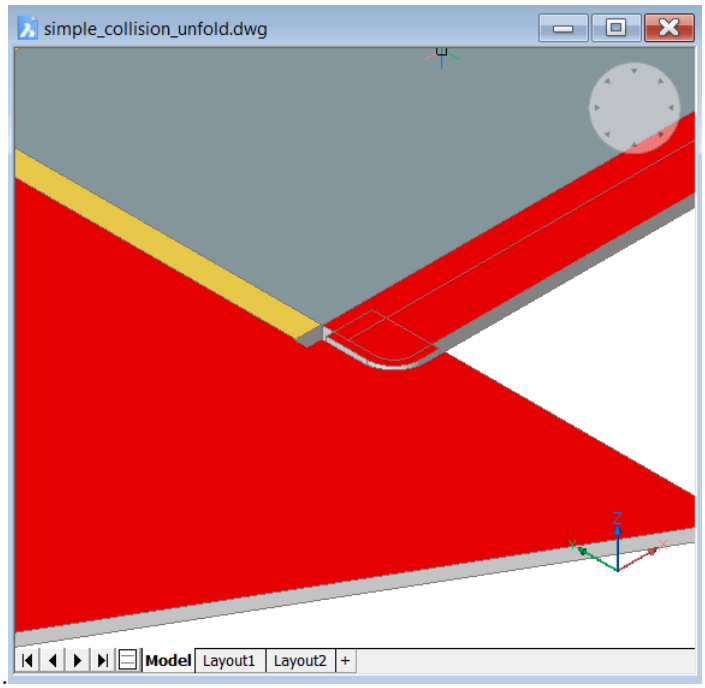
5 Примените к полученной поверхности команду *DMTHICKEN* для придания ей



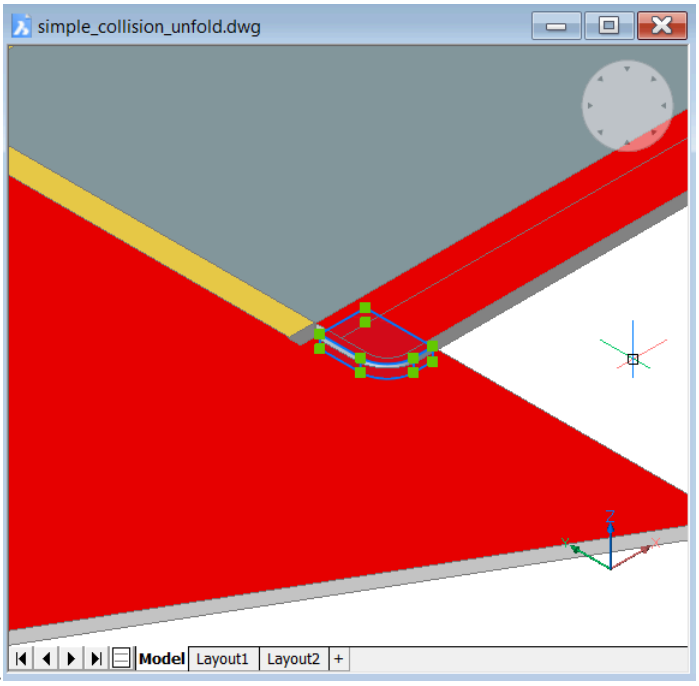

толщины.

6 Выполните *вычитание* полученного тела из

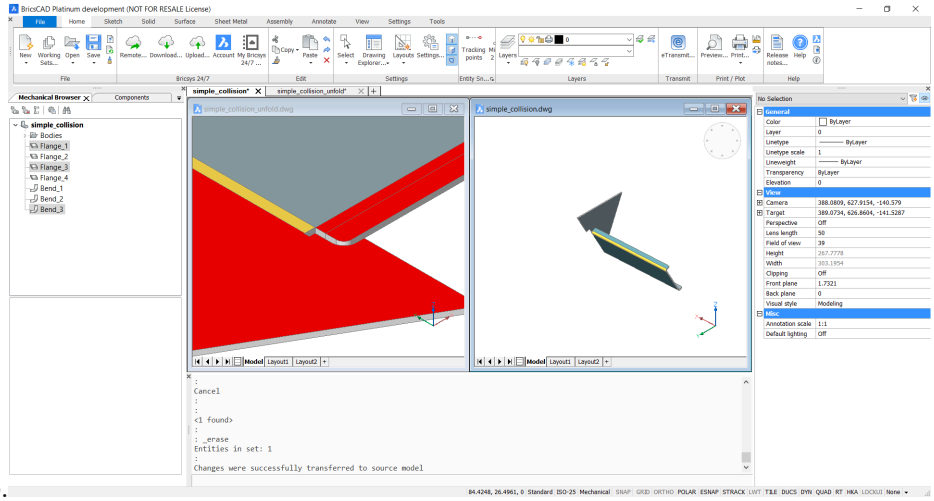
развертки.




7 *Удалите* вспомогательные элементы.




8 Вернитесь к документу с 3D

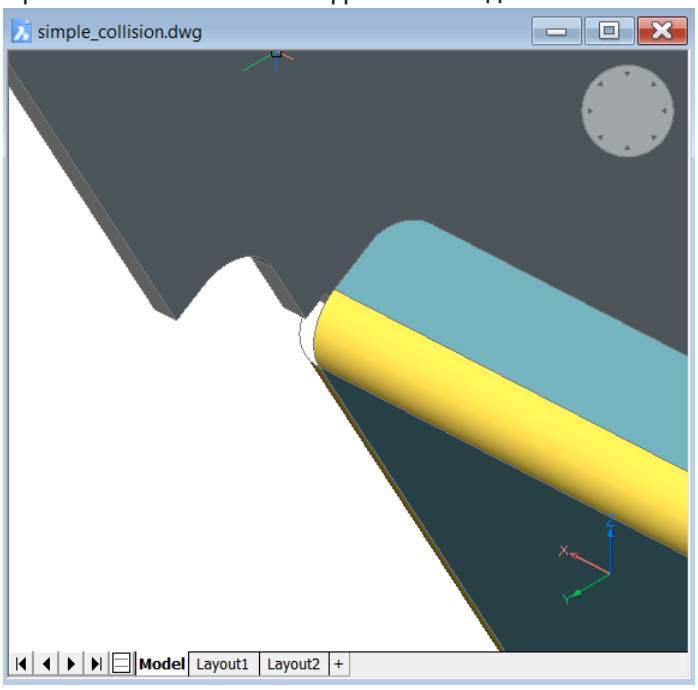



моделью.

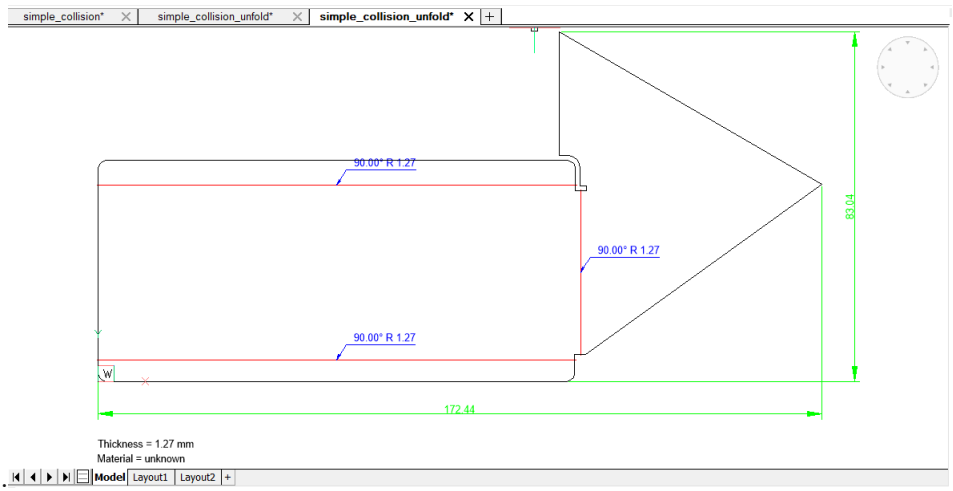
9 Убедитесь, что изменения были переданы в модель.

В командной строке должно отобразиться сообщение: Изменения были успешно перенесены в исходную модель.

Приблизьте измененный фрагмент модели.




10 Для проверки развертки выполните ее экспорт в .dxf




файл.

### 21.5.24 Smart Features

#### 21.5.24.1 Overview

The Smart Features technology manages the local intelligent update of the geometry after executing Sheet Metal commands. The behavior is controlled through the **SmSmartFeatures** user preference.



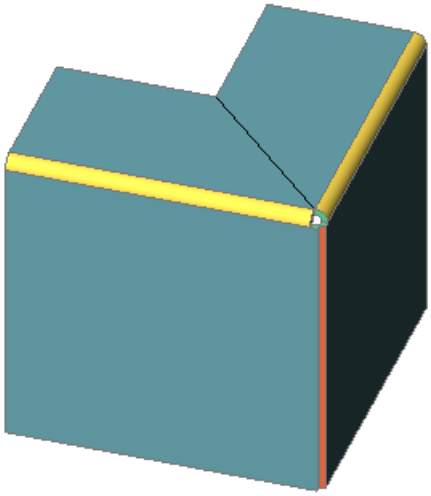
3 options are available:

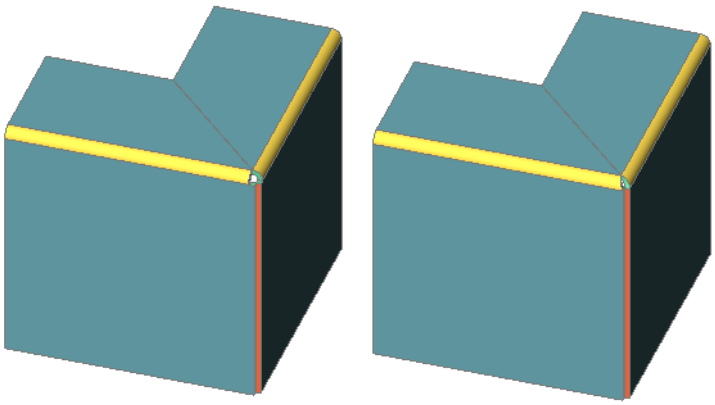
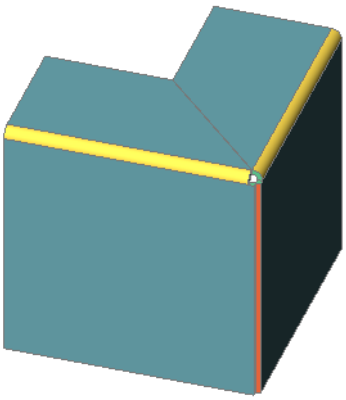
- Allow rebuilding sheet metal features:
  - Corner relief is transformed into 2 bend reliefs after executing *SMSPLIT* on a flange.
  - Unnecessary reliefs are removed.
- Allow automatic edges imprint after rebuild: automatic imprinting edges for thickness faces.
- Allow automatic creation of junctions after creating bends.

***Прим.:*** Always keep Smart Features options ON, unless inconsistent behavior occurs.

### 21.5.24.2 Transformation of a corner relief into bend reliefs

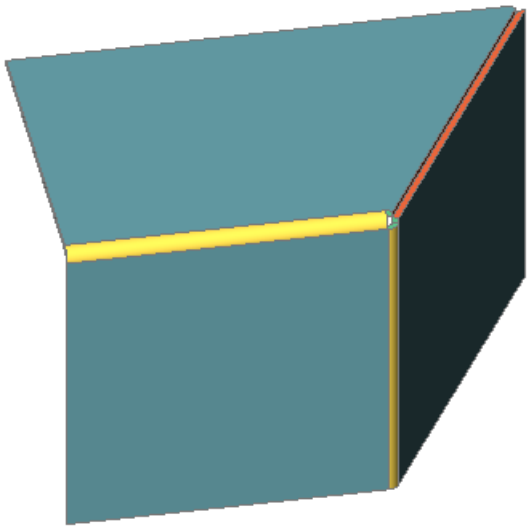
When splitting a top flange by a line into 2 separate parts, the corner relief is transformed into 2 bend reliefs.



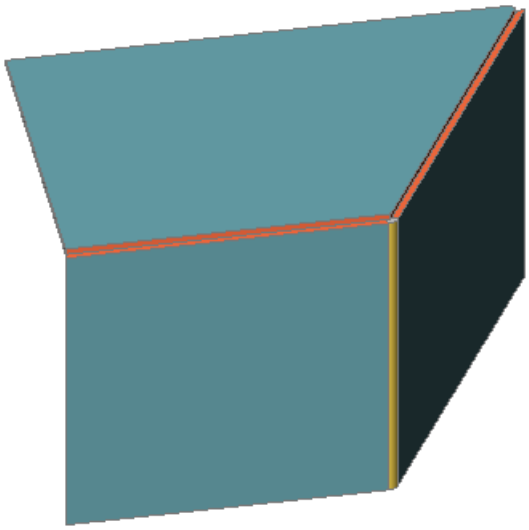
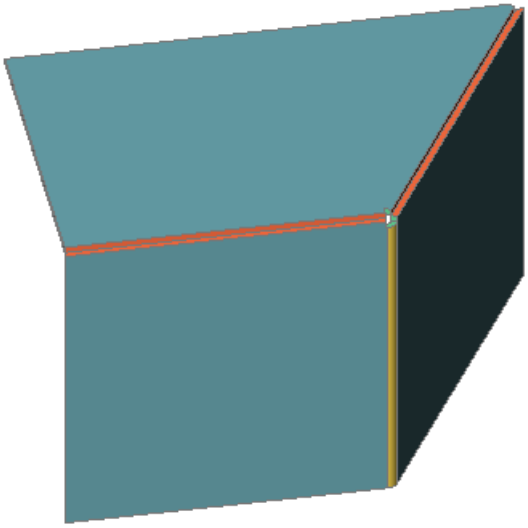
| | |
|---|---|
|  |  |
| Smart features create bend reliefs automatically, which can then be switched to smooth. | Without smart features, the corner relief is kept and cannot be switched. |

### 21.5.24.3 Removing unnecessary cuts

In the model below, a corner relief cut exists, which is needed to prevent the material from being stretched.



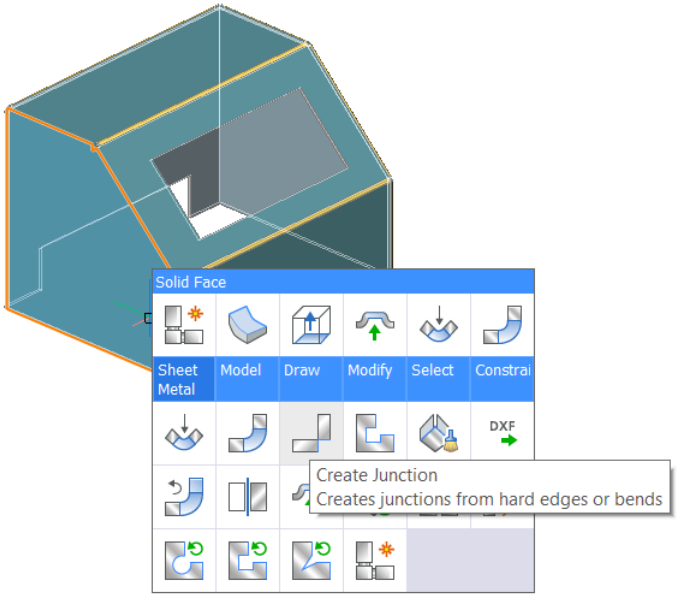
When the top bend is converted into a junction, we get to separate parts. The smart features automatically remove the now unnecessary relief.

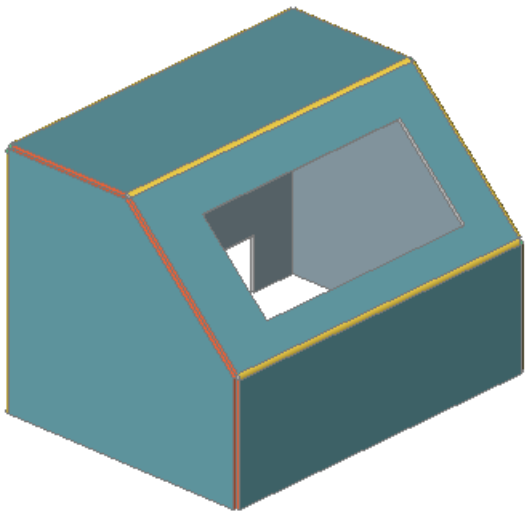
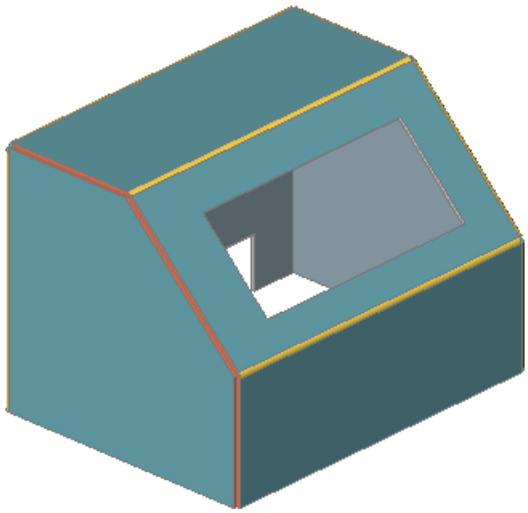
| | |
|---|---|
|  |  |
| Smart features automatically fill unnecessary cuts, thus simplifying the geometry. | Without smart features, the corner relief still exists. |

### 21.5.24.4 Automatic imprinting of edges

Imprinted edges on thickness faces separate them into areas which can be reworked automatically. That means localized junctions can be converted into bends. Without automatically imprinted edges the *SMIMPRINT* command is needed here.

In the example below, we will separate a flange using junctions. We turn **Rebuilding sheet metal features** ON, and then use **Automatic edges imprint after rebuild**.

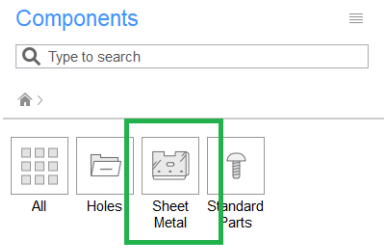



| | |
|---|---|
|  |  |
| Allow automatic edges imprint after rebuild = ON.<br>Junction faces are split correctly. | Allow automatic edges imprint after rebuild = OFF.<br>Junction features overlap each other. |

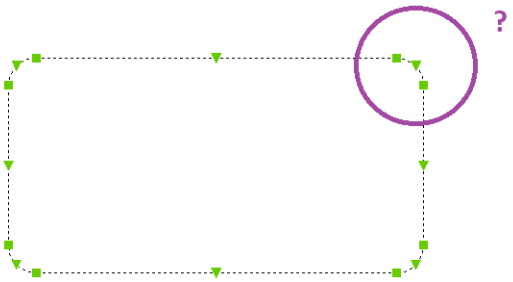
#### 21.5.24.5 Automatic creation of junctions

In certain situations, when hard edges meet at the corner and we create a bend on all but one, which has to become a junction in order to make the part unfoldable.

Assume we create bends on the top face of a sheet metal box:




Then the remaining hard edges adjacent to the face must become junctions. When **SmSmartFeatures** = ON, this is performed automatically:




To be precise, the area of automatic junction creation is located to the common vertex of hard edges (or corner relief when bends are created). The option does not optimize the layout of the unfolded part.

## 21.6 Генерация чертежей

### 21.6.1 Drawing Views

#### 21.6.1.1 Overview

BricsCAD Mechanical provides a toolset to generate drawing views from 3D models of parts and assemblies. Drawings views are created in the paper space of the same DWG file. The 3D model is created in model space. Drawing views can be annotated using dimensions, leaders, text and tables. Views and annotations are updated when the associative 3D model is changed.

The tools to generate, update and annotate drawing views are located on the **Annotate** tab of the **Mechanical workspace**:



**Рисунок 150:** Annotate

**Таблица 45: Annotate**

| Drawing view | Description |
|---|---|
| **Base Views** | Generate associative orthographic and standard isometric views of a 3D solid model in a paper space layout. |
| **Projected Views** | Generate additional projected views from an existing drawing view. |
| **Section View** | Create a cross-section view based on a standard drawing view generated by command *VIEWBASE* in a paper space layout. |
| **Detail View** | Create a detail view of a portion of a standard generated drawing at a larger scale. |
| **Edit View** | Change the scale and the hidden line visibility of drawing views. |
| **Update Views** | Updates a selection of drawing views obtained by *VIEWBASE* and *VIEWSECTION* when *VIEWUPDATEAUTO = 0*. |
| **Section Style** | Predefined and user-defined styles. |
| **Detail Style** | |
| **Export Views** | Export the content of drawing views, obtained by command *VIEWBASE* and *VIEWSECTION*, to the Model Space of the drawing, or to a new drawing. |

### 21.6.1.2 Draft-quality drawing views

By default, BricsCAD generates precise (high-quality) drawing views for any 3D model. The edges of such views (including silhouette edges) are represented by 2D entities; lines, circles, arcs and splines. The generation of precise views for a large number of 3D entities can take a significant amount of time. This is because every silhouette edge must be approximated with an analytical 2D curve.

You can reduce the time needed to generate drawing views by changing the *DRAWINGVIEWQUALITY* setting to *0*. This enables draft-quality drawing views. Such drawing views are internally represented as

3D solids, which makes it impossible to put annotations on their silhouette edges. However, they look very similar to a precise (high-quality) drawing view and you can use them to quickly create layouts.

A draft-quality drawing view can be switched to a high-quality drawing view using its *Geometry* property (change it from *Draft* to *2D*).

### 21.6.2 Аннотации чертежа

Процедуры создания и редактирования аннотаций описаны в разделе Обзор инструментов создания размеров.

Размеры и выноски в чертежных видах обновляются при изменении исходной 3D модели.

На чертежах сборок вы можете использовать аннотации в виде маркеров позиций. Каждый маркер соответствует строке в таблице спецификации, которая генерируется на основе сборке.

Таблица спецификации может быть размещена в пространстве листа вместе с чертежным видом сборки.

Покомпонентные виды сборок могут быть использованы для генерации чертежных видов и аннотированы с помощью маркеров позиций.

# 22. Communicator для BricsCAD

## 22.1 BricsCAD Communicator

BricsCAD Communicator is an optional plug-in for Pro and higher license levels of BricsCAD that imports and exports 3D data between major CAD formats. Its high-quality conversions facilitate the re-use of 3D CAD data to improve productivity and reduce project costs.

### 22.1.1 Supported export file formats

Windows

- **3D PDF**: .pdf
- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4:** .model
- **CATIA v5:** .CATPart, .CATProduct
- **IGES:** .iges, .igs
- **STEP:** .step, .stp

  The following export protocols can be specified; AP203, AP214, AP242
- **VDA-FS:**.vd
- **XCGM:** .xcgm

Linux

- **3D PDF**: .pdf
- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4:** .model
- **IGES:** .iges, .igs
- **STEP:** .step, .stp

  The following export protocols can be specified; AP203, AP214, AP242
- **VDA-FS:**.vda

Mac

- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4:** .model
- **IGES:** .iges, .igs
- **STEP:** .step, .stp

  The following export protocols can be specified; AP203, AP214, AP242
- **VDA-FS:**.vda

***Прим.:*** When the EXPORTPRODUCTSTRUCTURE system variable is set, you can export a BricsCAD assembly with mechanical components as a file with product structure.

### 22.1.2 Supported import file formats

Windows

- **ACIS:** .asab, .asat, .sab, .sat
- **Autodesk Inventor:** .iam, .ipt
- **CATIA v4:**.model
- **CATIA v5/v6:**.CATPart, .CATProduct, .CGR
- **Creo Elements / Pro Engineer:** .asm, .prt
- **IGES:** .iges, .igs
- **NX:** .prt
- **Parasolid:**.xmt_bin, .xmt_txt, .x_b, .x_t
- **Siemens:** .jt
- **Solid Edge:** .asm, .par, .psm
- **SolidWorks:** .sldasm, .sldprt
- **STEP:** .step, .stp, .stpz
- **VDA-FS:** .vda
- **XCGM:** .xcgm

Linux

- **ACIS:**.asab, .asat, .sab, .sat
- **CATIA v4:**.model
- **Creo Elements / Pro Engineer:** .asm, .prt
- **IGES:** .iges, .igs
- **Parasolid:**.xmt_bin, .xmt_txt, .x_b, .x_t
- **Siemens:** .jt
- **Solid Edge:** .asm, .par, .psm
- **SolidWorks:** .sldasm, .sldprt
- **STEP:** .step, .stp, .stpz
- **VDA-FS:** .vda
- **XCGM:** .xcgm

Mac

- **ACIS:**.asab, .asat, .sab, .sat
- **CATIA v4:**.model
- **Creo Elements / Pro Engineer:** .asm, .prt
- **IGES:** .iges, .igs
- **Parasolid:**.xmt_bin, .xmt_txt, .x_b, .x_t
- **STEP:** .step, .stp, .stpz
- **VDA-FS:** .vda

***Прим.:***

***Depending on the Communicator settings:***

- You can import an assembly file, as a plain geometry (a nonhierarchical set of entities) or map its product structure into native blocks or mechanical components.
- PMI (Product and Manufacturing Information) is supported for the following file formats (Communicator license is required): ACIS, CATIA V5, NX, Pro/E / Creo, SolidWorks, XCGM. PMI annotations are imported as graphical elements grouped in blocks.
- If the assembly contains hidden parts, you can decide how to represent them in BricsCAD: 'import and hide', 'import and set visible' or just 'do not import'. This feature is currently supported on CATIA v5 only.
- You can set import options to automatically repair, stitch and simplify imported geometry.
- The IGES format has its own options to stitch and simplify. If you import a 3D model from SolidWorks, it makes sense to set an option to map Y axis into Z (because in SolidWorks the vertical axis is Y).
- For SolidWorks and other native CAD formats, it is possible to specify an alternate search path. This will be used to look for parts absent in the imported assembly folder.

## 22.2 BricsCAD Communicator settings

The **Communicator** section in the **Settings** dialog box allows you to customize the behavior of the related commands:

- setting environment options.
- control the import/export of data.
- toggling post-processing operations.

Settings are applicable when the EXPORT and IMPORT commands engage one of Communicator file formats.

### 22.2.1 Import settings

### 22.2.2 Product structure

**Name**: ImportProductStructure

**Description**: Determines how the imported entities will be structured and whether it will form a soup of entities, a block hierarchy or an assembly structure.

**Values**:

- **[0] None**: Creates a flat structure without blocks in the modelspace of the target database, regardless of whether the imported data have an assembly structure or not.
- **[1] As blocks**: The imported data have an assembly structure, which will be converted into a hierarchy of plain blocks; so the structure will be preserved, while the assembly metadata are lost. If the imported data is composed of entities only, they are placed in the modelspace of the target database.

- **[2] As mechanical components**: In this mode, data will be translated into BricsCAD assembly data, the structure and its properties (physical materials – BLMATERIALS command). If the imported file has no assembly data, a mechanical component will be created in the BricsCAD target document root anyway.

### 22.2.3 Translate colors

**Name**: ImportColors

**Description**: Determines how the colors of the imported entities will be assigned: either as true colors or as matching the current color palette.

**Values**:

- **[0] To RGB**: All entity colors will be converted to RGB, regardless of the current palette.
- **[1] To RGB if no matching palette index**: If the entity color is found in the palette, then the entity gets an index color. Otherwise, it is given a true color.
- **[2] To nearest palette index**: For any true color of the imported entity, the nearest match is searched in the palette and this index color is assigned to the entity.

### 22.2.4 Product and manufacturing information

**Name**: ImportPMI

**Description**: Determines whether product manufacturing information is imported. Currently such information is imported as exploded data (lines, text, ...) instead of compound entities (e.g. annotations).

**Values**: **on**/**off** checkbox.

### 22.2.5 Perform stitching

**Name**: ImportStitch

**Description**: In some cases, imported geometry represents solid geometry as a set of separate surfaces. In order to be able to work with solid operations on the imported geometry, the DMSTITCH command has to be applied. If this setting = ON, the DMSTITCH command is executed automatically when the geometry is imported.

***Прим.:***

- Stitch operations are time-consuming when importing large files.
- Check the IMPORTIGESSTITCH setting, which can set an override for the IGES file format.

**Values**: **on**/**off** checkbox.

### 22.2.6 Repair model on import

**Name**: ImportRepair

**Description**: Applies operations, similar to the DMAUDIT command, in order to improve the quality of the imported geometry. Geometry modeled in CAD systems which use a kernel different from ACIS, often needs to be healed because of possible flaws.

**Values**: **on**/**off** checkbox.

### 22.2.7 Perform simplification

**Name**: ImportSimplify

**Description**: Applies operations similar to the DMSIMPLIFY command:

- Convert imported splines into canonic surfaces.
- Simplify topology (remove imprinted edges) if possible.

***Прим.:*** Also check the ImportIgesSimplify setting, which can set an override for the IGES file format.

**Values**: **on**/**off** checkbox.

### 22.2.8 Hidden parts

**Name**: ImportHiddenParts

**Description**: Defines how hidden (invisible) entities in the imported geometry are processed.

**Values**:

- **[0] Import and hide**: All entities are imported; invisible entities are hidden. Note that currently there are no user tools to make these hidden entities visible again.
- **[1] Import and set visible**: All entities are imported and visible with regardless of the visibility in the source file.
- **[2] Do not import**: Hidden entities in the source file are not imported.

### 22.2.9 Creo options

**Alternate search paths**

**Name**: ImportCreoAlternateSearchPaths

**Description**: This setting is taken into account when Creo files are imported. The value is a set of semi-colon (;) delimited, fully qualified paths used to search for missing assembly references during product structure import operations.

**Values**: string; if empty, the search is performed in the folder of the imported file only.

### 22.2.10 IGES options

1 **Perform stitching**

**Name**: ImportIgesStitch

**Description**: In some cases, imported geometry represents solid geometry as a set of separate surfaces. In order to be able to work with solid operations on the imported geometry, the DMSTITCH command has to be applied. If this setting = ON, the DMSTITCH command is executed automatically when the geometry is imported. This setting determines whether a stitch operation has to be applied to imported entities when the format of the source file is IGES.

***Прим.:***

- Stitch operations are time-consuming when importing large files.
- For file formats other than IGES, the ImportStitch setting applies.

**Values**: **on**/**off** checkbox.

2 **Perform simplification**

**Description**: Determines whether a DMSIMPLIFY operation has to be applied to the imported geometry when the source file format is IGES

***Прим.:*** For other formats, see the ImportSimplify setting.

**Values**: **on/off** checkbox.

3 **Inventor options**

**Alternate search paths**

**Name**: ImportInventorAlternateSearchPaths

**Description**: Setting is taken into account when Inventor files are imported. The value is a set of semi-colon (;) delimited, fully qualified paths used to search for missing assembly references during product structure import operations.

**Values**: string; if empty, the search is performed in the folder of the imported file only.

4 **NX options**

**Alternate search paths**

**Name**: ImportNxAlternateSearchPaths

**Description**: Setting is taken into account when NX files are imported. The value is a set of semi-colon (;) delimited, fully qualified paths used to search for missing assembly references during product structure import operations.

**Values**: string; if empty, the search is performed in the folder of the imported file only.

5 **SolidEdge options**

**Alternate search paths**

**Name**: ImportSolidEdgeAlternateSearchPaths

**Description**: Setting is taken into account when SolidEdge files are imported. The value is a set of semi-colon (;) delimited, fully qualified paths used to search for missing assembly references during product structure import operations.

**Values**: string; if empty, search is performed in the folder of the imported file only.

6 **SolidWorks options**

- **Map SolidWorks Y to BricsCAD Z axis**

  **Name**: ImportSolidWorksRotateYZ

  **Description**: Corrects the difference between the SolidWorks and BricsCAD coordinate systems. If unchecked, imported geometry in BricsCAD is rotated by 90 degrees compared to its position in SolidWorks.

  **Values**: **on/off** checkbox.

- **Alternate search paths**

  **Name**: ImportSolidWorksAlternateSearchPaths

  **Description**: Setting is taken into account when SolidWork files are imported. The value is

a set of semi-colon (;) delimited, fully qualified paths used to search for missing assembly references during product structure import operations.

**Values**: string; if empty, search is performed in the folder of the imported file only.

7 **STEP options**

**Map Y to BricsCAD Z axis**

**Name**: ImportStepRotateYZ

**Description**: Corrects the difference between coordinate systems in different coordinate systems, i.e. SolidWorks and BricsCAD. This can be helpful to prevent undesired rotation when the origin CAD of STEP file is known, i.e. if it is SolidWorks.

**Values**: **on**/**off** checkbox.

### 22.2.11 Export settings

### 22.2.12 Product structure

**Name**: ExportProductStructure

**Description**: Determines whether the assembly structure will be preserved in the target document.

**Values**:

- **[0] No product structure**: Exports a flat structure without components in the target document, whether the BricsCAD document has a product structure or not.
- **[1] Export product structure**: Exports the BricsCAD product structure data (if existing) to the target document.

### 22.2.13 Hidden parts

**Name**: ExportHiddenParts

**Description**: Determines if invisible entities are exported or not. Entities can be invisible because of:

- the result of the HIDEOBJECTS command.
- sitting on a hidden layer.
- owned by an invisible component.

**Values**:

- **[0] Export and hide if possible**: Exports hidden entities. If the target format supports hidden entities, hidden entities in the source document will be hidden also in target document.
- **[1] Do not export**: Invisible entities are skipped.

### 22.2.14 STEP options

**STEP export format version**

**Name**: ExportStepFormatVersion

**Description**: Allows you to select the available specifications of the STEP format.

**Values**:

- **[0] AP203**: Selects AP203 protocol.
- **[1] AP214**: Selects AP214 protocol.
- **[2] AP242**: Selects AP242 protocol.

# 23. BricsCAD Shape

## 23.1 Introduction to BricsCAD Shape

### 23.1.1 What is BricsCAD Shape

BricsCAD® Shape is the new conceptual modeling tool from Bricsys. It's made to help architects and engineers streamline their 3D schematic design work. Shape is the fastest way to get designs out of your head and in front of your clients. It's CAD-accurate, using a powerful direct modeling engine.

You can use it as a live presentation tool, too. Potentially best of all, Shape offers a clean, simple user interface that you can learn in 30 minutes or less.

### 23.1.2 BricsCAD Shape lets you create without limits

Shape's smart modeling tools make it easy to capture in 3D. The multifunctional "Shape" tool can be used to create walls, slabs, structural elements and more. Direct modeling features let you drag, connect, push/pull and extrude as you like. Wall connections are automatically mitered for you.

BricsCAD Shape also offers everything you need to present your designs. It comes with a library of materials, textures and pre-built 3D components. You can use the included collection of doors and windows, or create your own custom versions easily. Make your model look the way you want using the Visual Styles panel in Shape.

### 23.1.3 How is BricsCAD Shape different from other tools?

First, BricsCAD Shape is CAD-accurate from the start. Under the simple UI is the same fast and proven parametric, 3D direct modeling engine that's in BricsCAD Platinum. Shape creates solid models, stored in industry-standard DWG – just like BricsCAD® BIM. Unlike competing products, Shape's solid models are fully accurate – never approximated. Every element that you create in Shape can be modified deeply, anytime, on the fly. Your concept models open directly in BricsCAD BIM. You'll never lose time starting over again. Make the smart move – directly from concept to BIM with BricsCAD Shape with no loss of detail.

### 23.1.4 BricsCAD Shape accelerates your BIM workflow

The Bricsys BIM workflow starts in 3D with Shape, and stays in 3D – speeding 2D construction documentation with the world's best drafting tools. No breaks or transitions in your workflow, all in DWG, with a friendly, familiar user interface that you'll appreciate. It won't take long for you to be productive in BricsCAD Shape or BricsCAD BIM.

### 23.1.5 Stop sketching. Start shaping!

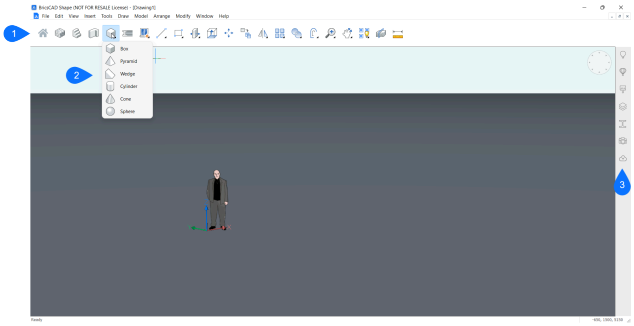
Get your free copy of BricsCAD Shape! BricsCAD Shape delivers the tools you need, and none that you don't. Even the learning materials are clean and simple – check out "10 Minutes to BricsCAD Shape" on YouTube, Bricsys channel.

## 23.2 BricsCAD Shape user interface

The toolbar at the top of your screen (1) gives you fast access to the most used tools. Left-click on any icon in the toolbar to use the tool.

Many top-level tools offer sub-menus (2). Hover over the icon, left click and hold. The sub-menu will fly out to show these additional tools.

The icons on the right side of your screen (3) give you access to a set of collapsible panels. The top-most icon opens the **Tips panel**. You'll find more details on how to use the tools in Shape here.




***Прим.:*** Many of the modeling tools that are used in BricsCAD and BricsCAD BIM are also available in Shape. You can get an overview of all available commands in Shape by typing in COMMANDS in the Command line.

## 23.3 Basic concepts of modeling in BricsCAD Shape

### 23.3.1 Selection methods

**Highlighting and selecting objects**

Entities in BricsCAD Shape are solids, which are made up of edges and faces.

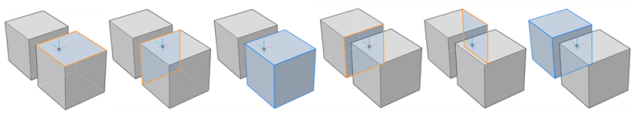
When hovering over an entity, a part of the entity is highlighted. To select the highlighted part of the entity, left-click the edge or face.

***Прим.:*** To select or highlight the whole entity, hold down the **CRTL** key and left-click.

For some operations, it's not necessary to select entities. When a surface or edge is highlighted, the **Quad** opens and several commands can be accessed.

***Прим.:*** You can only highlight one entity at a time, while it is possible to select multiple entities at once by left-clicking them one-by-one, or using a window selection method.

Sometimes, the entity you want to select is hidden behind other entities. Place the cursor over the obscured entity, then repeatedly press the **TAB** key. All (sub)entities under the cursor highlight one by one. Click to select the currently highlighted (sub)entity.



To clear a selection of objects, press the **ESC** button.

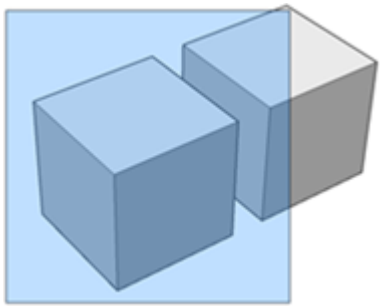
### Selection windows

To select one or multiple elements at once, a selection window is used. Left click on an empty space in the model area. When you move the cursor a selection window displays.

***Прим.:*** The selection window selects entire solids by default. Press the **Ctrl** key to select faces, press the **Ctrl** key again to select edges and again to select solids. The current selection method is indicated in the **Hot Key Assistant** (HKA) at the bottom of the screen.

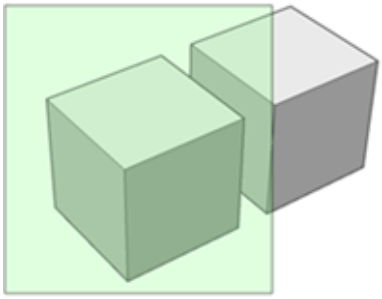
#### *Blue selection window*

Moving the cursor from left to right, a blue selection window displays: all entities that are completely inside the rectangle are selected. This selection method is called 'Window Inside'.



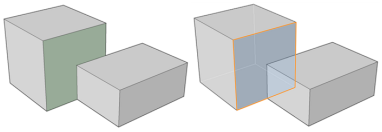
#### *Green selection window*

Moving the cursor from right to left, a green selection window displays: all entities that are completely inside or overlapping the selection rectangle are selected. This selection method is called 'Window Overlap'.



### Boundaries vs Faces

Boundaries can be defined by a set of 2D entities in the XY field of the current coordinate system or on a 3D solid face or by partially overlapping faces of 3D solids. BricsCAD Shape automatically detects and highlights closed boundaries, as you hover over different model elements. When a boundary is detected, it will be highlighted. To select it, just left-click inside the highlighted boundary. Boundaries can be used to create new solids or making holes or offsets and recesses.



***Прим.:*** To select the entire solid face instead of a boundary, press the **TAB** key.

### 23.3.2 Quad

The Quad cursor menu, or Quad for short, is an alternative to the Command line, toolbars or ribbon, offering a rich set of tools while requiring fewer clicks, without cluttering the screen with loads of grip-glyphs. It includes rollover tips, which display a limited set of properties when selecting or hovering over an entity.

The Quad displays when:

- You hover over an entity.
- A selection set exists.
- You right click on an empty space on the screen. This will open the No Selection Quad.

The initial content of the Quad when hovering over an entity is:

- Entity type.
- The icon of the most recently used command on this entity type.
- Relevant properties.

When you move the cursor to the Quad, it expands. On top you can find the most relevant or most recently used commands for the entity. Below you see a series of command tabs:

- Move the cursor over a tab to expand it.
- Place the cursor over an item to see a tool tip.
- Click an icon to launch a command.
- Click the Quad title bar to collapse, click again to expand.

### 23.3.3 Manipulator

The manipulator allows you to move, rotate scale and mirror selected entities and optionally create copies.

To display the Manipulator do one of the following:

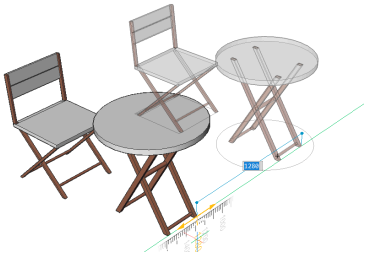
- Select the entity or subentity and choose **Manipulator** in the **Modify** tab on the **Quad**.
- Hold down the left mouse button a little bit longer when selecting.

***Прим.:*** The Manipulator aligns along the edge that you enter the entity by. Its origin sits at the closest endpoint of that edge.

**Select an axis**

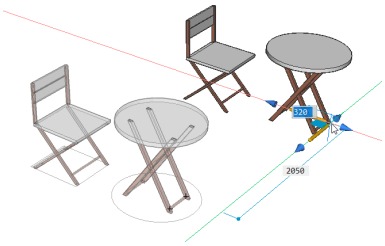
By selecting an axis of the manipulator you are able to move the entity. Specify the value of displacement by entering a value or picking a point. The step size of the ruler depends on the zoom factor. Zoom in to decrease the step size, zoom out to increase.

***Прим.:*** To copy the entity, hold down the **CTRL** key before clicking the axis.



**Select a plane**

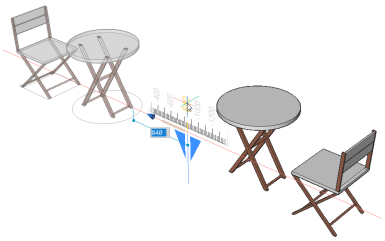
By selecting a plane of the manipulator you are able to move the entity in the selected plane after specifying the value of displacement by entering a value or picking a point.

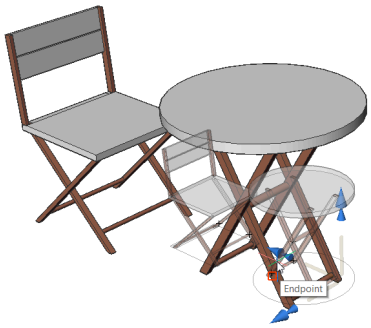


**Select an arrowhead**

By selecting an arrowhead of the manipulator you are able to scale or rotate the selected entity. By right clicking when hovering over the entity, a contexts menu with other options is displayed.There the function of an arrowhead can be changed between 3D mirror and 3D scale.

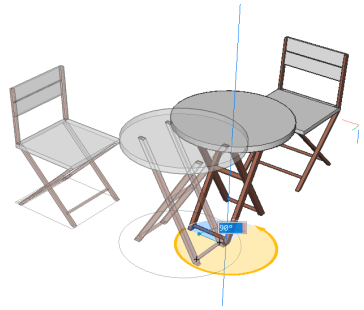
Select one of the arrowheads and specify the mirror axis or scale.






**Select a rotation arc**

By selecting an arc of the manipulator you are able to rotate the entity after entering the rotation angle value or specifying by clicking on a point in the drawing.



**Select the anchor handle**

By selecting the anchor of the manipulator you are able to relocate the manipulator.

## 23.4 Navigation in BricsCAD Shape

### 23.4.1 Panning

You can move the drawing in any direction: horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space.

There are several ways to pan:

- Hold down the middle mouse button.
- Press and hold the **CRTL+SHIFT** and right mouse button.
- Type the *PAN* in the Command line.
- Type the *RTPAN* in the Command line.
- Click the **Real-Time Pan** icon in the toolbar.

To abort the *PAN* command, do one of the following:

- Right click and choose **Exit** in the context menu.
- Press **Enter** key, **space bar** key or **Esc** key on the keyboard.

### 23.4.2 Rotating a view

You can rotate a view freely, in a constrained manner.

To rotate in real-time:

- Hold down the **SHIFT** and middle mouse button.
- Type *RTROT* in the Command line.
- Click the **Real-Time Constrained Sphere** icon in the toolbar.

To abort the *RTROT* command, do one of the following:

- Right click and choose **Exit** in the context menu.
- Press **Enter** key, **space bar** key or **Esc** key on the keyboard.

### 23.4.3 Zoom in/out

You can change the magnification of your drawing at any time by zooming.

Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail.

Changing the magnification of the drawing affects only the way the drawing is displayed. It has no effect on the dimensions of the entities in your drawing.

To zoom in/out:

- Scroll the mouse wheel.
- Press and hold the **CRTL+SHIFT** and left mouse button to zoom in or out.
- Type **ZOOM** in the Command line.
- Click the **Zoom In** or **Zoom Out** icons in the toolbar.

***Прим.:*** **Zoom Extents** displays the drawing to the extents of entities

### 23.4.4 Look From control

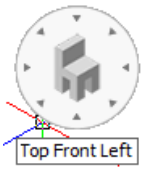
The **Look From** control allows you to select a number of preset views such as orthographic views and isometric views. By default the **Look From** control displays in the top right corner of the graphic screen.

1 Move the cursor to the **Look From** control.

The **Look From**control becomes active.

2 Position the cursor on the **Look From** control to choose a view orientation.

A preview image and a tooltip indicate the currently selected view orientation.



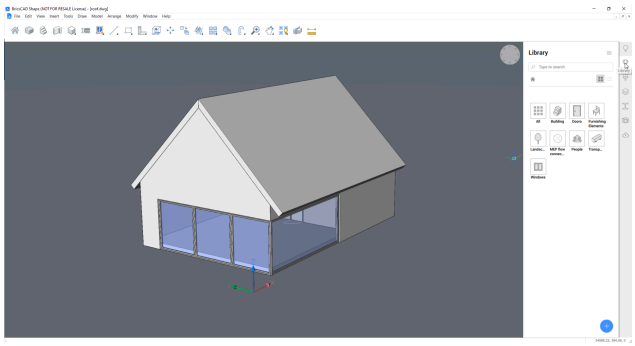

3 Click to confirm.

4 The view is updated.

## 23.5 Components in BricsCAD Shape

BricsCAD Shape includes a collection of components in the **Library** panel.

Open the library by clicking the airballoon icon on the right hand edge of the screen. Several categories are available.



Click the menu button at the top right corner of the panel to select or hide the available libraries:

- **Bricsys BIM library**: content of *C:\Program Files\Bricsys\BricsCAD Vxx en_US\UserDataCache\Support \en_US\Bim\Components*.
- **Bricsys 2D library**: content of *C:\Program Files\Bricsys\BricsCAD Vxx en_US\UserDataCache\Support \en_US\Blocks2D*.
- **User library**: content of *C:\ProgramData\Bricsys\Components*

***Прим.:*** Choose **Manage Libraries** in the menu to add folders to the COMPONENTSPATH system variable in the **Settings** dialog box.

### 23.5.1 To insert a component

1 Click the component in the library.

2 Move the cursor into the model.

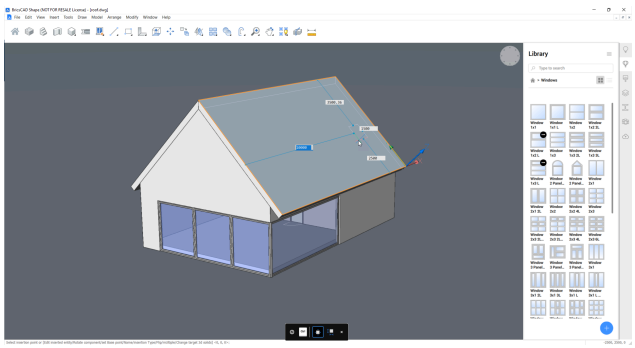
***Прим.:*** The selected component is attached to the cursor. The X and Y distances from the origin of the current coordinate system display dynamically.

3 Do one of the following:

- Click to place the component.
- Type a value in the X and Y distance fields.

***Прим.:*** Press the **TAB** key to jump to the other field. Press Enter to place the component.

### 23.5.2 To insert a window in the face of a solid



1 Click the window in the library.

2 Move the cursor into the model over the face of 3D solid.

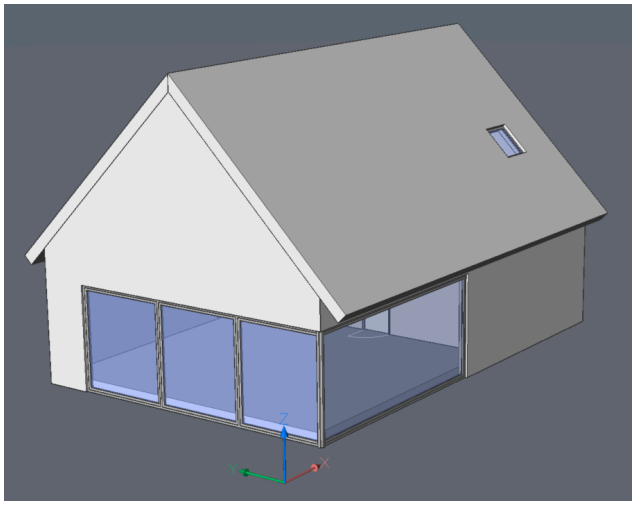
***Прим.:*** The window aligns to the face under the cursor. Distance fields from the sides of the face display dynamically.

3 Do one of the following:

- Click to insert the window.
- Type a value in distance fields.

  ***Прим.:*** Press the **TAB** key to jump to the next field. Press Enter to place the component.

  The window automatically makes an opening in the solid.



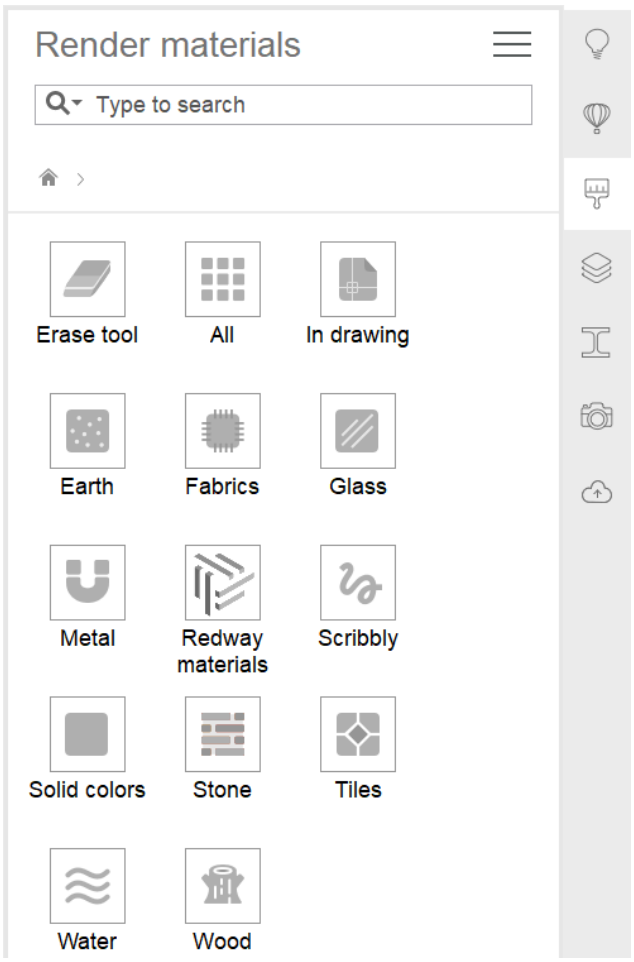
***Прим.:*** When you delete a window, the opening is removed.

## 23.6 Materials in BricsCAD Shape

BricsCAD Shape includes a collection of materials in the **Render materials** panel.

Render materials are stored in: *C:\ProgramData\Bricsys\RenderMaterials\UserMaterials*.

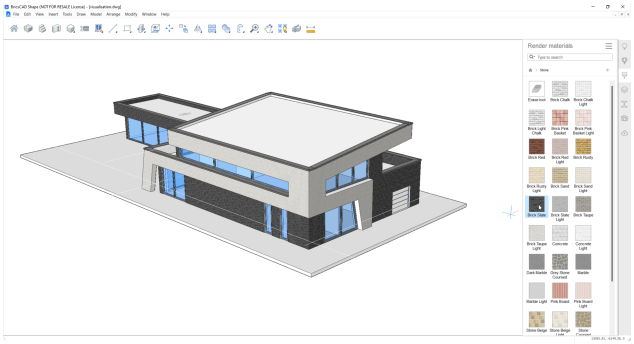
Open the **Render materials** panel by clicking the **paintbrush** icon on the right-hand edge of the screen. Several categories are available. To view all materials that are used in the drawing, select the **"In drawing"** category.




To assign a material to an entity in your model:

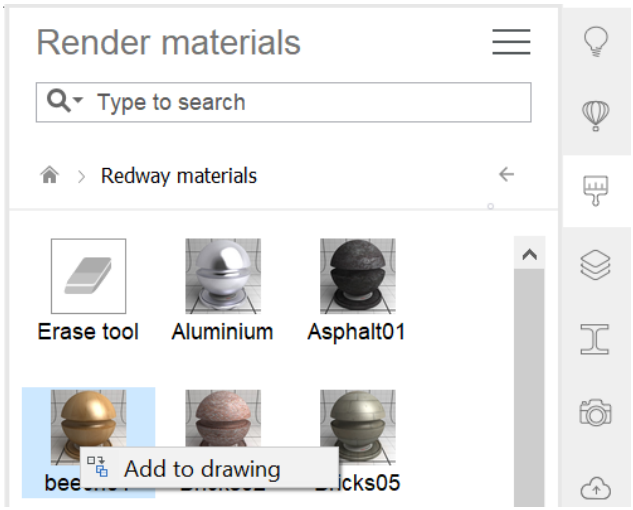
- Select the material in the **Render materials** panel.
- Select the entity you want to assign the material.

***Прим.:*** You can apply a material to multiple entities at once by preselecting them.



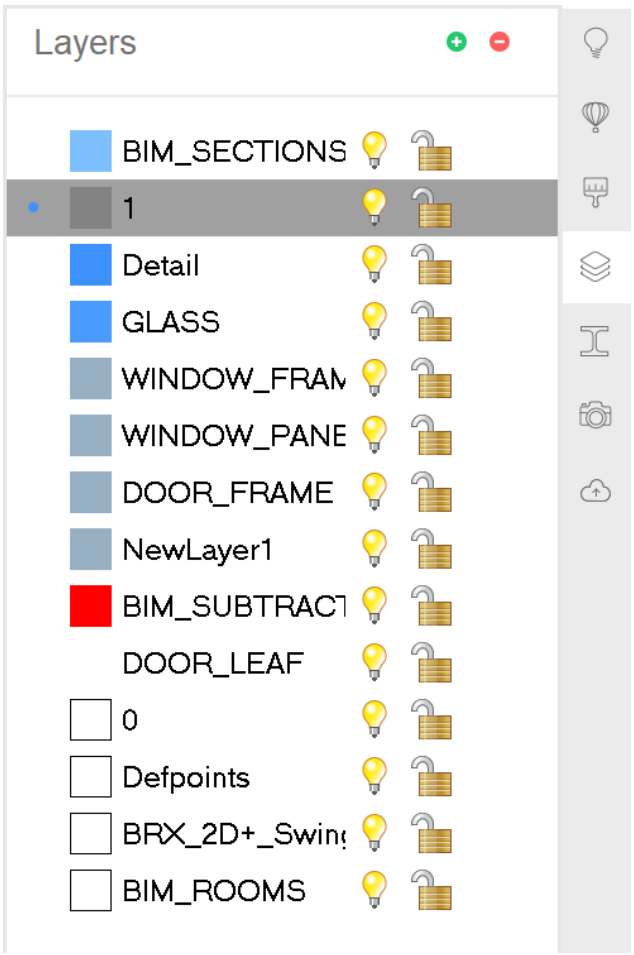
***Прим.:*** To remove a material from an entity, use the erase tool and click any entity to remove the material from.

It you want to add materials to the drawing to use later, right-click the material icon and select the "**Add to drawing**" option.




## 23.7 Layers in BricsCAD Shape

Layers can be used to group entities to organize your 3D model. The **Layer** panel can be accessed through the stacked sheets icon on the right side of the screen.




***Прим.:*** The current layer is indicated with the small blue dot. To change the current layer, click in the same column next to the layer you want to set as current layer.

### 23.7.1 Turn on/off

To turn on and of specific layers, toggle the lightbulb of the layer on and off. This way elements can be hidden or isolated. This can be of huge help with navigation.

- **On**: Displays entities that are on that layer.
- **Off**: Hides entities that are on that layer.

### 23.7.2 Lock/unlock

To lock or unlock layers, toggle the lock icon of the layer.

- **Unlocked**: Allows entities on that layer to be edited.
- **Locked**: Prevents entities on that layer from being edited.

### 23.7.3 Create/delete

To create a new layer, click the **green plus** sign. A new layer with a generic name is created. To delete a layer, make sure the layer to delete is selected and click the **red minus** sign.

### 23.7.4 Edit

Properties such as name and color of layers can be edited:

- To rename a layer, double-click the layer name or select **Rename** from the right-click menu.
- To change the color, select one or more layers and then click the current color for one of the selected layers. The **Select Color** dialog box is displayed. The color you select is applied to all selected layers.

## 23.8 Working with steel profiles

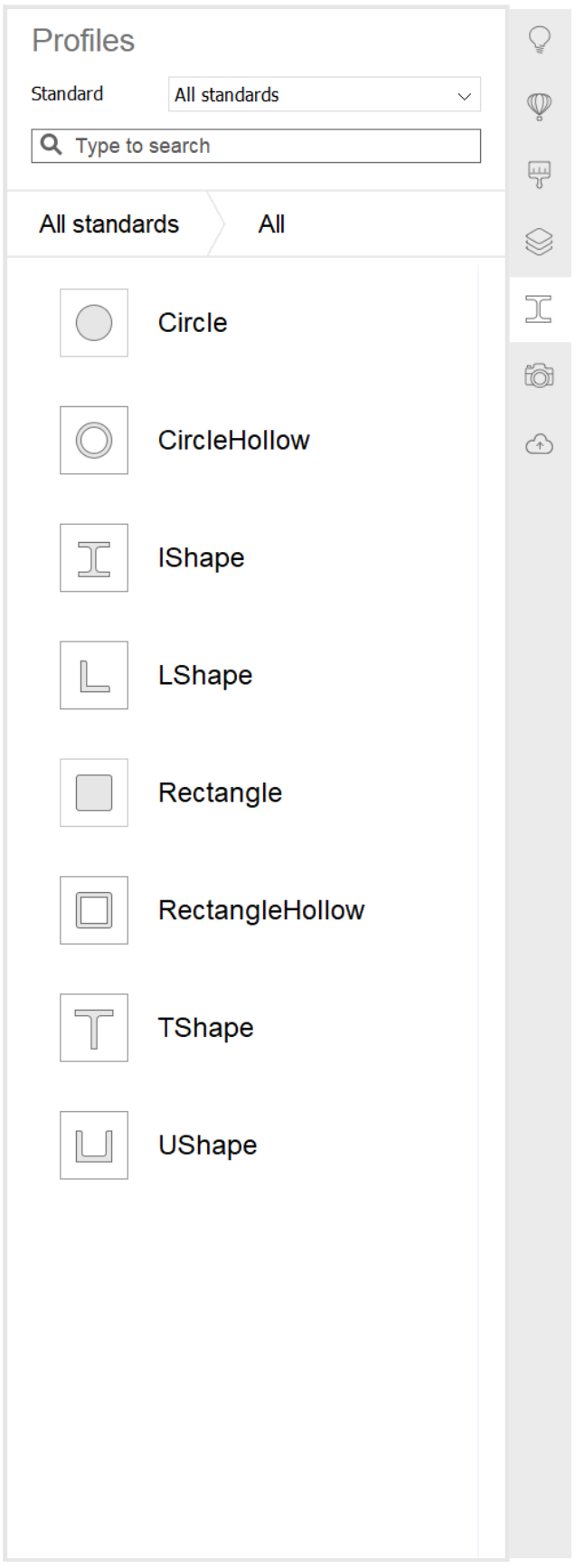
BricsCAD Shape offers standard structural steel profiles which enable you to easily create structural elements. Multiple standards are available, including American, British and European standards.

Steel profiles are created as linear solids.

### 23.8.1 Creating a steel profile

1 Click the **Profile** icon the vertical toolbar, on the right-hand edge of the screen.

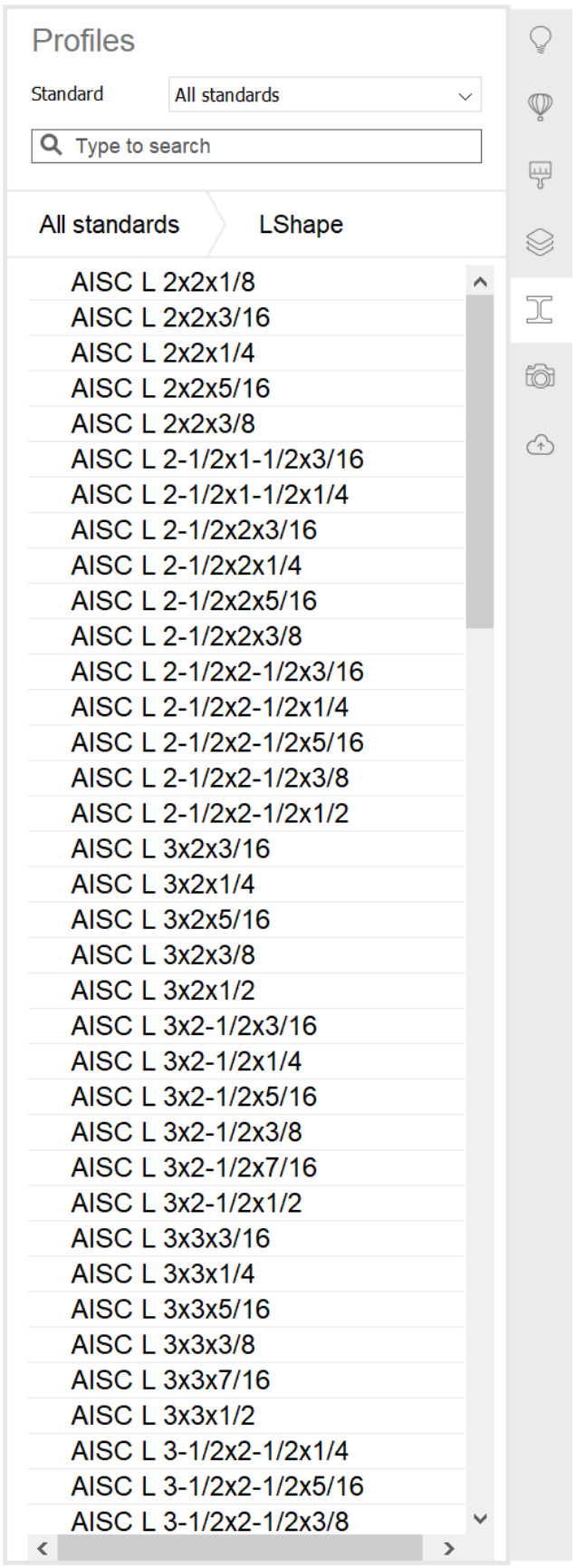
The **Profiles** panel displays.


Profiles
Standard
All standards
Type to search
All standards
All
Circle
CircleHollow
IShape
LShape
Rectangle
RectangleHollow
TShape
UShape

2 Optionally select a standard.

3 Select a profile shape.

4 A list of all profiles of the selected shape displays.


Profiles
Standard
All standards
Type to search
All standards
LShape
AISC L 2x2x1/8
AISC L 2x2x3/16
AISC L 2x2x1/4
AISC L 2x2x5/16
AISC L 2x2x3/8
AISC L 2-1/2x1-1/2x3/16
AISC L 2-1/2x1-1/2x1/4
AISC L 2-1/2x2x3/16
AISC L 2-1/2x2x1/4
AISC L 2-1/2x2x5/16
AISC L 2-1/2x2x3/8
AISC L 2-1/2x2-1/2x3/16
AISC L 2-1/2x2-1/2x1/4
AISC L 2-1/2x2-1/2x5/16
AISC L 2-1/2x2-1/2x3/8
AISC L 2-1/2x2-1/2x1/2
AISC L 3x2x3/16
AISC L 3x2x1/4
AISC L 3x2x5/16
AISC L 3x2x3/8
AISC L 3x2x1/2
AISC L 3x2-1/2x3/16
AISC L 3x2-1/2x1/4
AISC L 3x2-1/2x5/16
AISC L 3x2-1/2x3/8
AISC L 3x2-1/2x7/16
AISC L 3x2-1/2x1/2
AISC L 3x3x3/16
AISC L 3x3x1/4
AISC L 3x3x5/16
AISC L 3x3x3/8
AISC L 3x3x7/16
AISC L 3x3x1/2
AISC L 3-1/2x2-1/2x1/4
AISC L 3-1/2x2-1/2x5/16
AISC L 3-1/2x2-1/2x3/8

5 Click a profile in the list, then move the cursor to the model.

6 The outline of the selected profile is attached to the cursor.

7 Specify a point.

8 A linear solid displays dynamically.

A length and angle field display the current length an angle of the profile.

When the current direction is close to the X, Y or Z direction of the current coordinate system a colored tracking line displays. Press the **Shift** key to lock the tracking line. Press again to unlock.

9 Do one of the following to create the profile:

- Specify a point.
- Type a value in the length field, then press **Enter** or press the **TAB** key to activate the angle field.

The next profile displays dynamically.

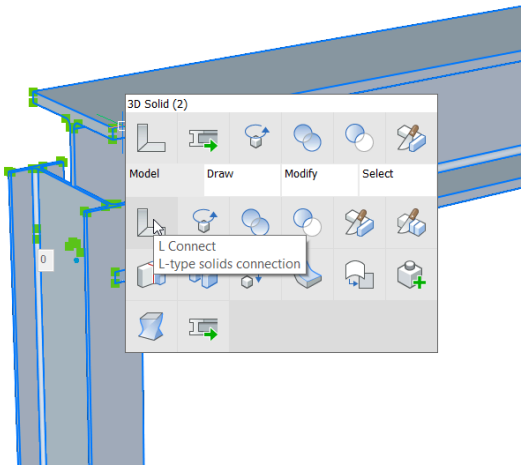
10 Repeat the previous steps to create more profiles or right click to stop.

### 23.8.2 Connecting Profiles

The L-Connect tool creates various types of L-connections between two selected profiles.

1 Select both profiles.

2 Click the L-Connect tool in the Model tab on the Quad.




- A bisector L-connection is created.
- The **Hotkey Assistant** (HKA) displays at the bottom of the screen.

3 Press the **Ctrl** key to modify the connection type.

The current connection type highlights in the **HKA**.

4 Right click to accept the connection type.

### 23.8.3 Assigning a different profile

1 Place cursor over the new profile in the **Profiles** panel. Do not click.

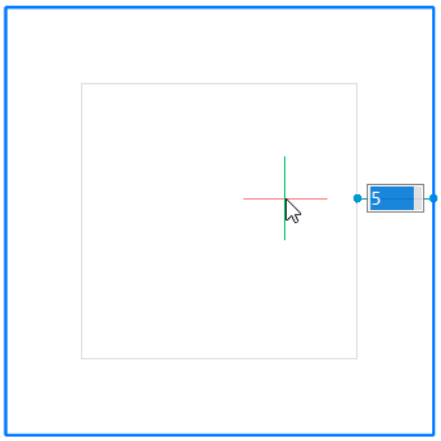
2 Hold down the left mouse button, then drag the profile onto the linear solid.

The selected profile is assigned to the linear solid.

### 23.8.4 Assigning a custom profile

1 Draw a custom profile shape using the tools in the **Draw** tab on the **Quad**:
- A closed polyline.
- A square or rectangle using the **Rectangle** tool.
- A circle.
- An ellipse.

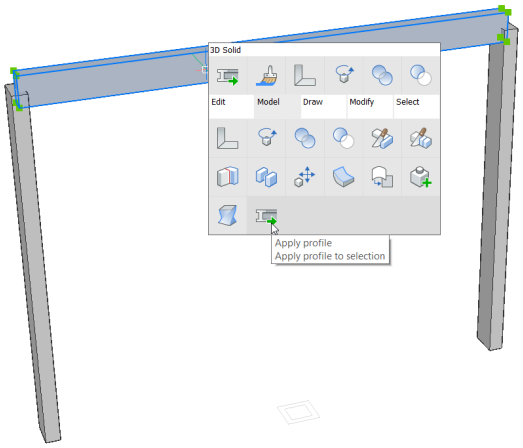
2 Optionally, use the **Offset** tool to create a hollow profile: select the outline, then specify the offset.




3 Hold down the **Ctrl** key, then select one or more linear solids. The linear solid(s) highlight(s).

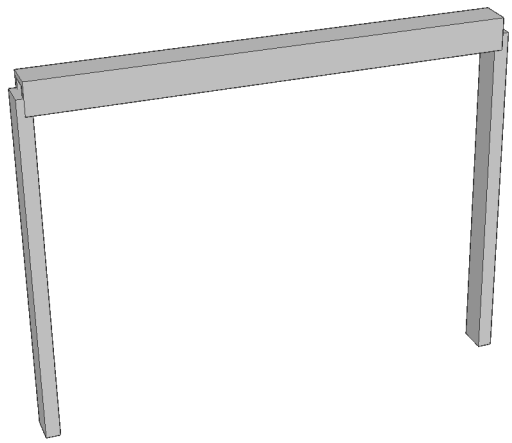
4 Click the **Apply profile** tool in the **Model** tab on the **Quad**.

Prompts you: Select profile [selection options (?)]:



5 Do the following:

- Select the custom profile.
- Use the window overlap selection method to select the profile outline and the offset when applying a hollow profile.
- A preview of the result displays.
- Prompts you: Apply profile? [Ok/Quarter turn/Rotate/Delete clippings] <Ok>:



6 Do one of the following:

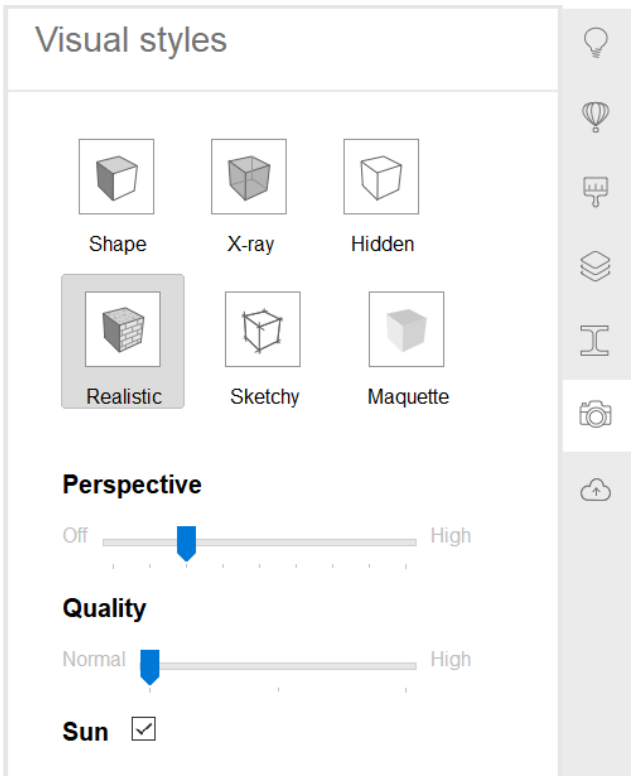
- Right click to apply the profile.
- Choose an option.
- Press the Esc key to quit.

## 23.9 Visualization in BricsCAD Shape

### 23.9.1 Visual styles panel

Visual styles control the appearance of your model.

The **Visual Styles** panel can be accessed by clicking the camera icon at the right side of the screen.




The look of the entire model will be changed when selecting another style.

The available visual styles are:

- Shape
- X-ray
- Hidden
- Realistic
- Sketchy
- Maquette

**Perspective**

You can change the perspective view by moving the **Perspective** slider.

***Прим.:*** Moving the perspective slider all the way to the left will turn the perspective off.

**Quality**

To control the on-screen rendering quality of the model, move the **Quality** slider.

**Sun**

To make the shadows of the sun appear, toggle the **Sun** checkbox.

### 23.9.2 Hide/Isolate/Show entities

**Hide**

When model elements block your view, you can hide them easily.

To hide one or more elements:

- Select the entities you want to hide.
- Click the **Hide Entities** icon in the **Quad** or in the toolbar.

**Isolate**

The opposite of hiding an element is isolating it, which will hide all the non-highlighted elements in your model.

To isolate entities:

- Select the entities you want to isolate.
- Click the **Isolate Entities** icon in the **Quad** or in the toolbar.

**Show**

Hiding and Isolating are temporary functions that you can use to simplify your model while working on it.

When you want to see all the elements in your model:

- Select the **Show Entities** icon in the “No Selection” Quad or in the toolbar.

### 23.9.3 Sections

You can use Sections to cut through your model dynamically.

To create a section:

- In the “No Selection” Quad, select the **Define Section** tool, found under the **Model** tab.
- Select any solid face in your model to create the section, parallel to the selected solid face.

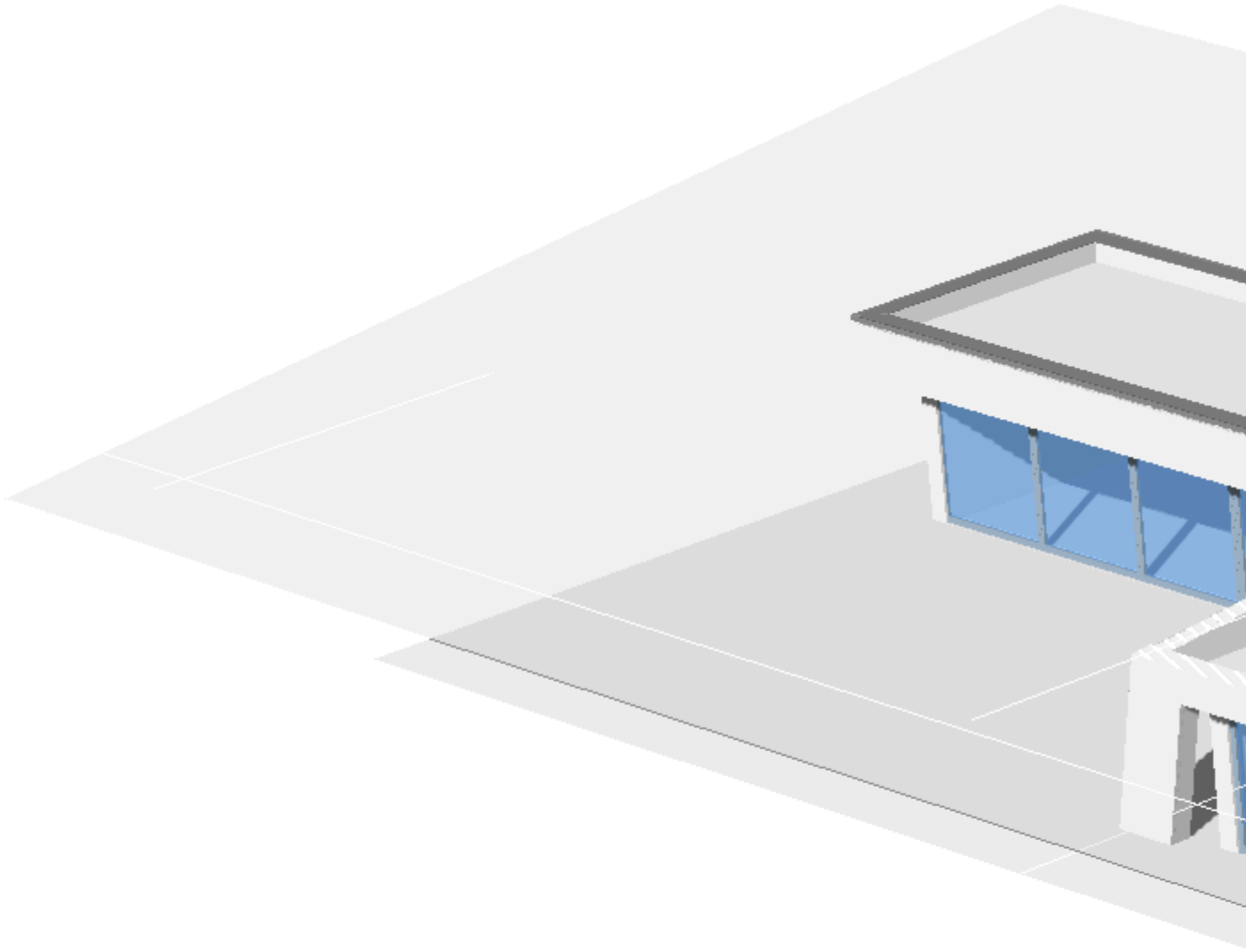
***Прим.:*** If you don’t choose a surface to attach the Section, a horizontal Section plane will be created.

You can toggle a section on and off by:

- double-clicking its handle
- highlighting the section plane and using **Clip Display** tool from the **Quad**.

A Section element has three grip edit points. Selecting the Section’s handle turns the grips on for editing:

- You can move the start and end grips to change the orientation and width of the Section.
- You can move the midpoint grip to reposition the section plane.
- If you click the arrow displayed near the midpoint grip, you will flip the clipping direction of the Section.



## 23.10 Solids in BricsCAD Shape

### 23.10.1 Drawing a box

1 Start creating a box by accessing the *BOX* command through the command line or by clicking the **Box** icon in the Quad or toolbar.

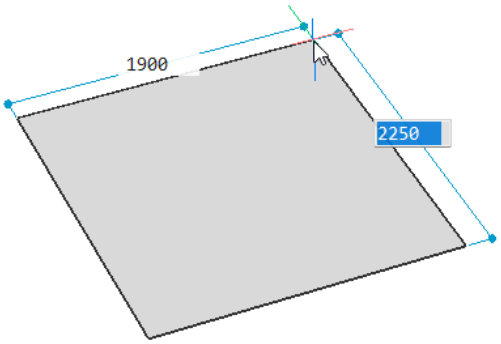
2 Specify a corner for the base of the box.

3 Specify the opposite corner for the base of the box to apply both the length and width.

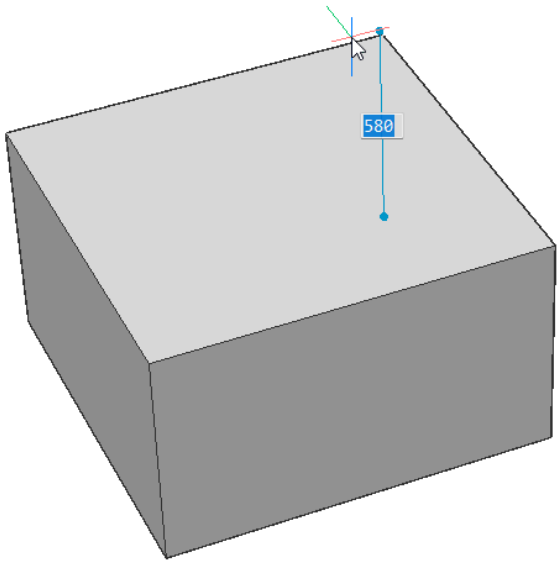
4 You can do this dynamically, by hovering the cursor at the opposite corner, or by entering the desired values in the dynamic input fields.

The box is created parallel to the x and y axes.

***Прим.:*** The blue dynamic field is the active one.




5 Specify the height of the box.



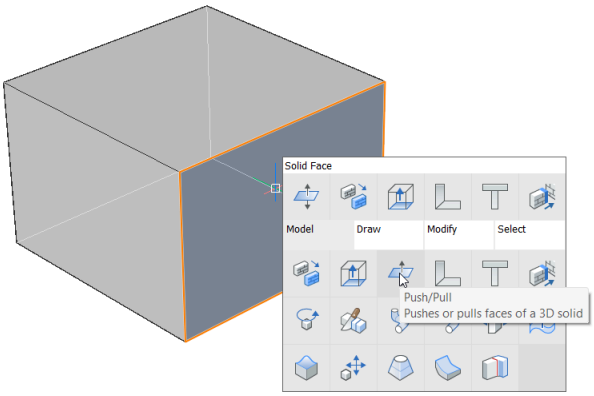

### 23.10.2 Manipulate a solid face

To manipulate a solid face:

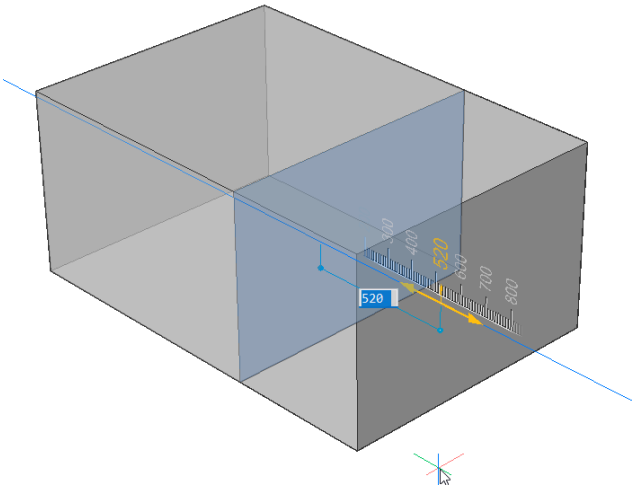
1 Hover the cursor over a face until highlighted.

2 Hover over the appearing icon to open the **Quad**.

3 Under the **Model** tab, the *DMPUSHPULL* command can be accessed.




4 Specify the displacement value or move with your cursor to define the end position of the face.




***Прим.:*** When selecting multiple faces and accessing the command, both faces will be pushed or pulled over same distances.

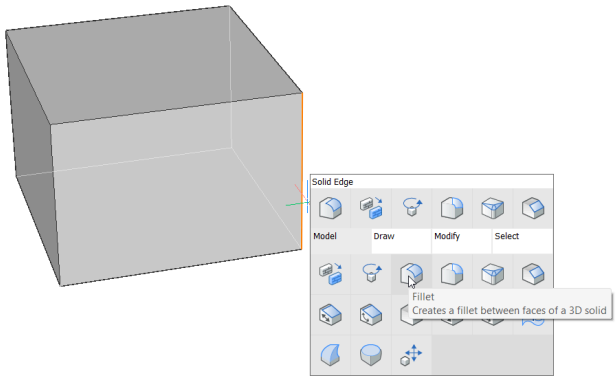
### 23.10.3 Manipulating a solid edge

To manipulate a solid edge:

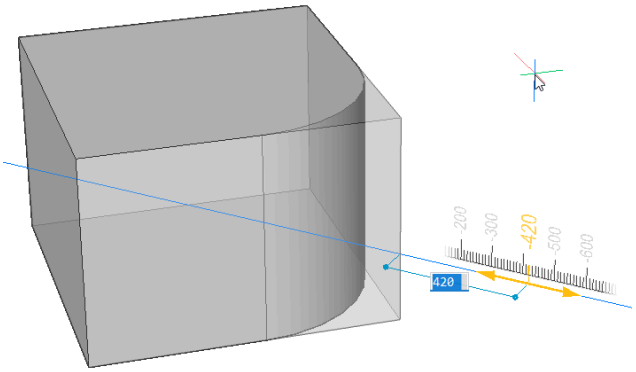
1 Hover your cursor over an edge while holding the **CRTL** key until highlighted.

2 Hover over the appearing icon to open the **Quad**.

3 Under the **Model** tab, the *FILLET* command can be accessed.

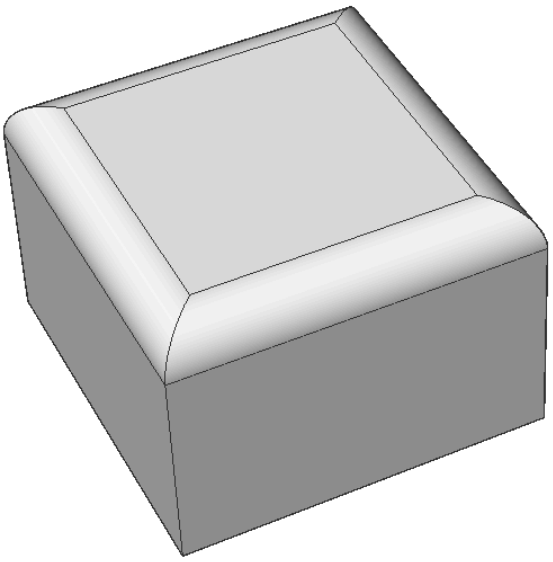



4 Specify the displacement value or move with your cursor to define the end position of the face.




***Прим.:*** When selecting multiple edges and accessing the command, all selected edges will have a fillet of the same radius.

***Прим.:*** When using a selection window to select edges, make sure you press the **CRTL** key twice so the edge selection icon is selected at the bottom of your screen.



### 23.10.4 Extrude solids

Selected boundaries can be extruded to create solids or surfaces. For example, a floor slab can be extruded between existing walls.

The extrude command can be accessed by the Quad or in the toolbar.

***Прим.:*** The extrude command can also subtract geometry also.

## 23.11 Creating walls in BricsCAD Shape

There are two ways to create a simple wall:

- Using the *POLYSOLID* command.
- Using the *BIMQUICKDRAW* command.

***Прим.:*** For a detailed explanation about *BIMQUICKDRAW*, read the article Quickdraw in BricsCAD Shape article.

### 23.11.1 About the Polysolid tool

The *POLYSOLID* command allows you to create wall solids by selecting a start point, an endpoint and entering a height.

### 23.11.2 Procedure: Create a simple wall

1 Launch the *Polysolid* command:

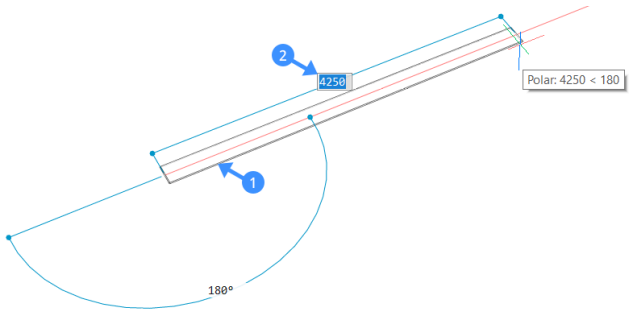
- Click the **Polysolid** icon

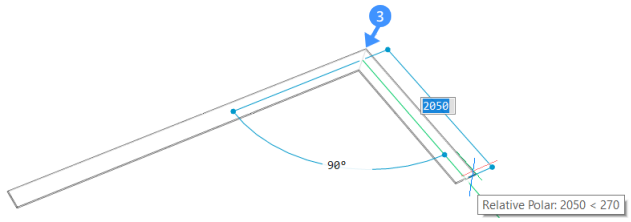
  in the toolbar
- Type *POLYSOLID* in the command line.

2 Select a start point, the footprint (1) of the wall is displayed automatically.

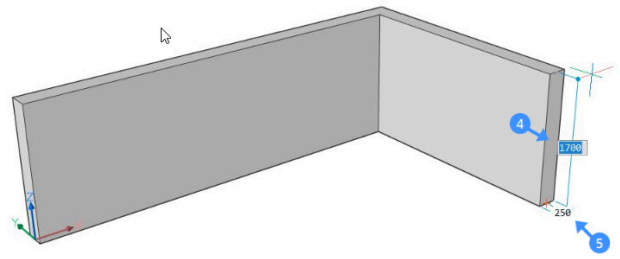
3 Move the cursor in the desired direction, the current length value is displayed in the length dynamic entry field (2).




4 Select an endpoint or enter a value in the dynamic input field and press **Enter** key. The footprint (3) of the adjacent wall is displayed. Select a new endpoint or enter values in the dynamic input field to create adjoining wall segments.




5 Press **Enter** once more or right-click. The height of the wall is displayed dynamically in the height field (4). Here, you can change the default height of the wall. To change the width of the wall, use the width field (5). You can switch between the two dynamic input fields by pressing **Tab** key.




6 To change the justification of the wall hit **CTRL** while the 'Polysolid justification Hotkey Assistant widget' appears at the bottom of the screen.




***Прим.:*** The width and height of the previous wall will be the new default values of the next Polysolid.

### 23.11.3 Procedure: Using an existing 2D plan to create walls

1 Launch the *POLYSOLID* command.

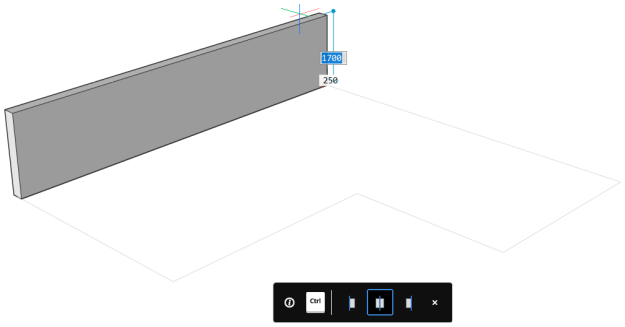
2 Type E in the command line or select **Entity** in the prompts menu.

3 Choose a 2D linear entity which will be the Polysolid base. Lines, open and closed polylines, arcs, circles, ellipses, elliptical arcs, and splines are accepted as a polysolid base.

4 Move the solid up or down and left-click to set the height of the wall, enter a value in the dynamic input field or right-click to accept the default value.

5 The current width of the wall appears in the **Width Field**. Press **Tab** switch between the Height and the Width field.

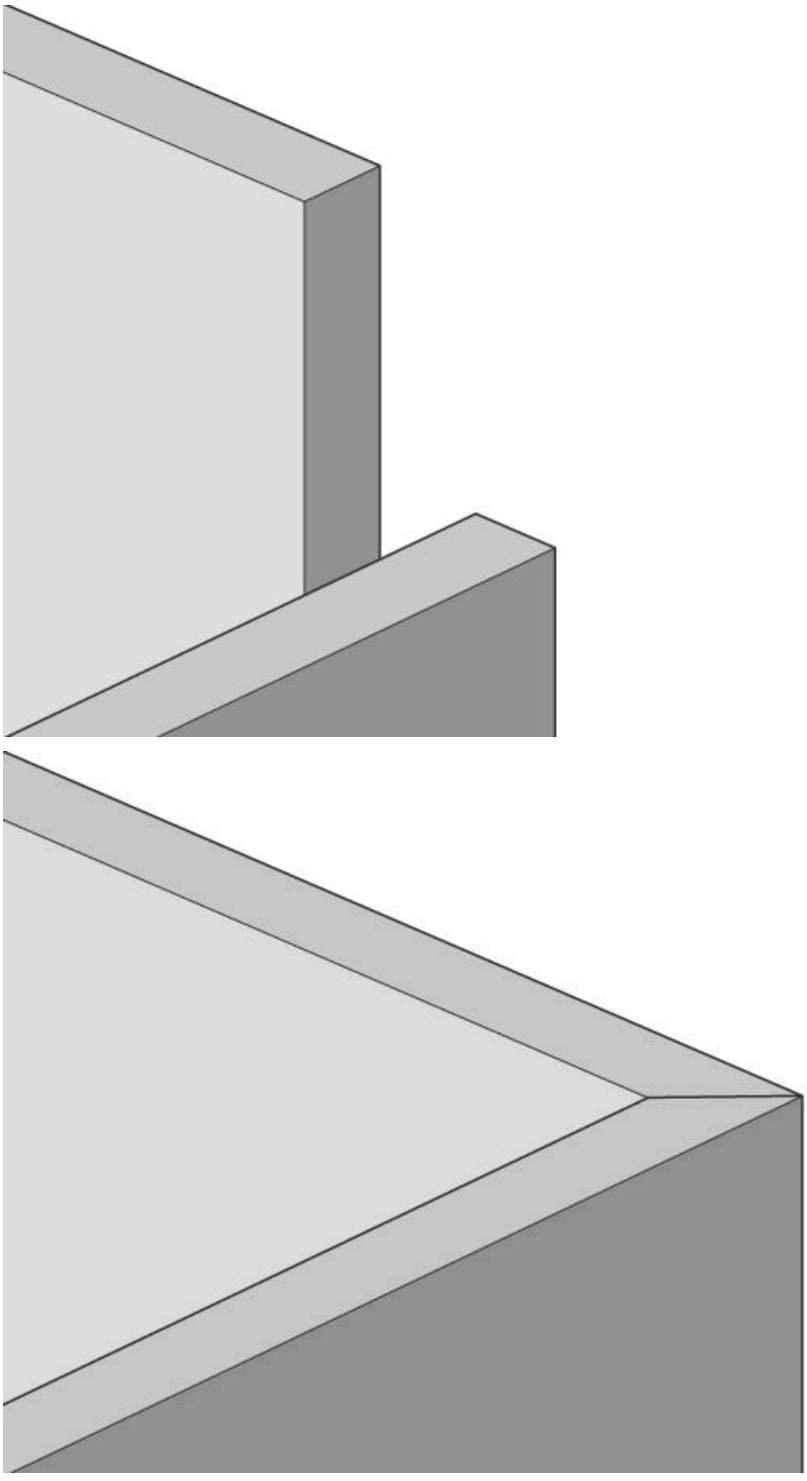
6 The **Hotkey Assistant** appears and displays the possible justification options. Press **CTRL** to cycle through the wall justification options.




### 23.11.4 Procedure: Creating L-connections between two walls

1 Launch the **LConnect** tool, by typing *LConnect* in the Command line and press Enter.

Select the two wall elements you want to connect.



***Прим.:*** When two walls intersect, a mitered connection will be created by default.

2 Change the layout of the wall connections using the ‘LConnect HotKeyAssistant widget’.

3 The selected connection type is indicated by a blue frame.




The connection can be (from left to right) a bisector L-connection , L parallel type 1 , L parallel type 2 or a disconnection.

Press **Enter** to accept the current connection type or press **CTRL** to cycle through the possible connection options.

### 23.11.5 Procedure: Creating T-connections between two elements

1 Launch the *TConnect* command.

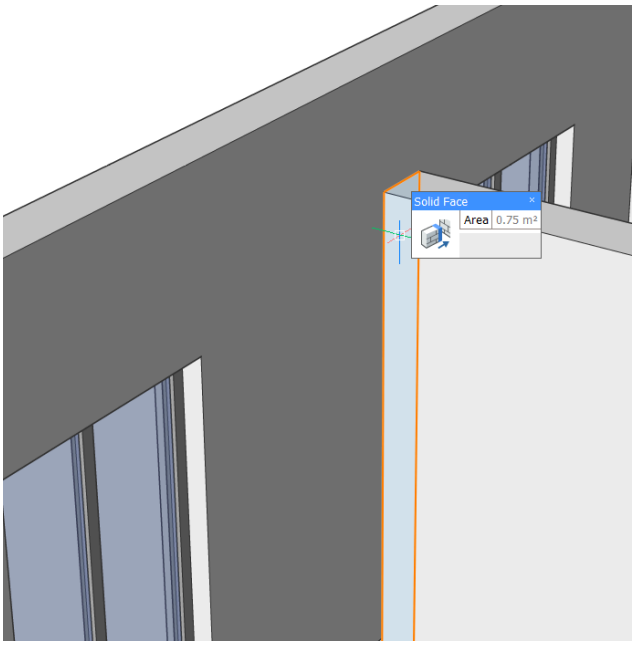
The tool allows you to connect the minor face of a wall to the major face of another element or wall. This tool can be used to connect the top face of a wall to a roof, the bottom face to a slab or side face of a slab to a wall.

2 Select the faces of the objects that you would like to connect and press **Enter**.

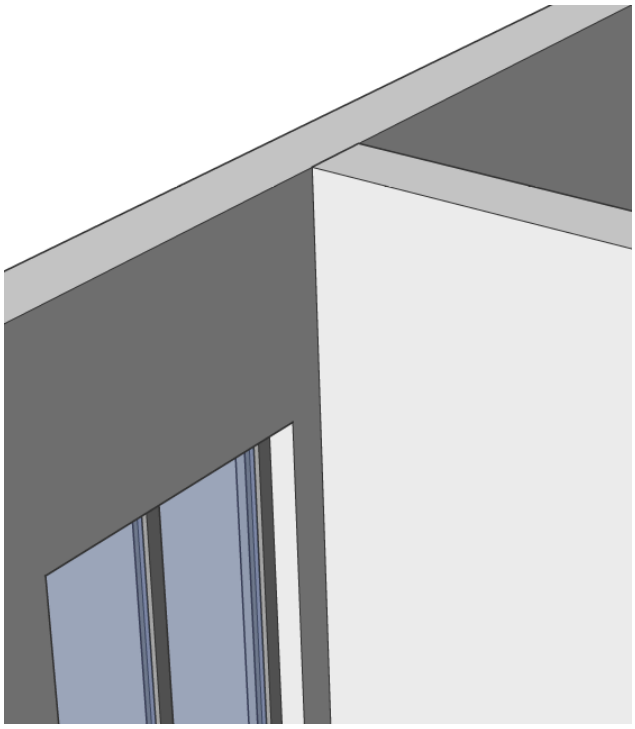
3 Select **Connect to nearest** in the prompt menu.

You can also use the **Quad** menu to create a T-connection between two walls or other elements:

- In the drawing area, highlight the face you want to connect to another wall.
- Use the **Connect with Nearest** tool from the **Quad**.




The Connect with Nearest tool automatically makes a T-connection between the two walls by extruding the face of the first wall to the face of the other wall.



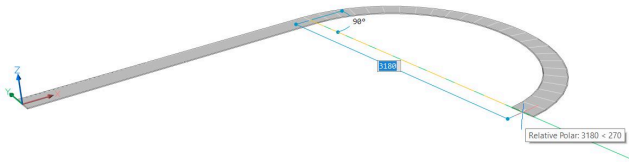
### 23.11.6 Procedure: Creating a curved wall

1 Launch the *POLYSOLID* command.

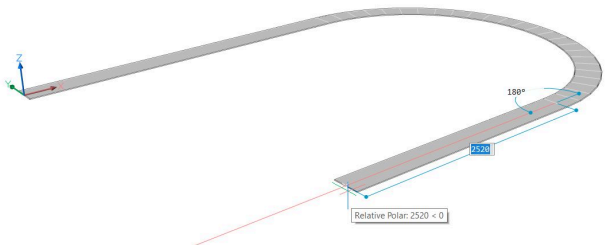
2 Select a start point and move the cursor to set the length or click to set the endpoint.

3 To create a curved wall, type A and press Enter to choose **Draw arcs** or select **Draw arcs** in the prompt menu.

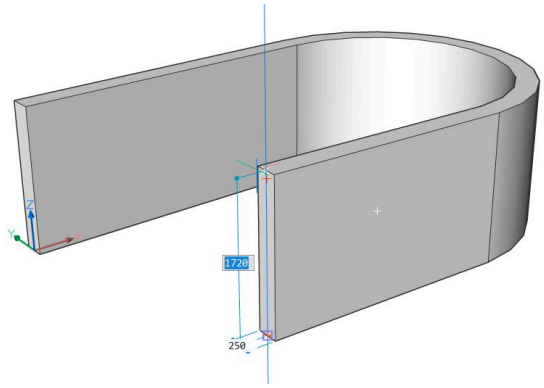
4 Now move the cursor in the desired direction to curve the wall. Enter a value in the dynamic input field to set the degree and length of the arc. Press **Enter**.




5 You can continue creating curved, adjoining wall segments by clicking or by entering values in the dynamic input fields. To go back to drawing straight wall segments, type L and press Enter or select **Draw lines** in the options dialog.

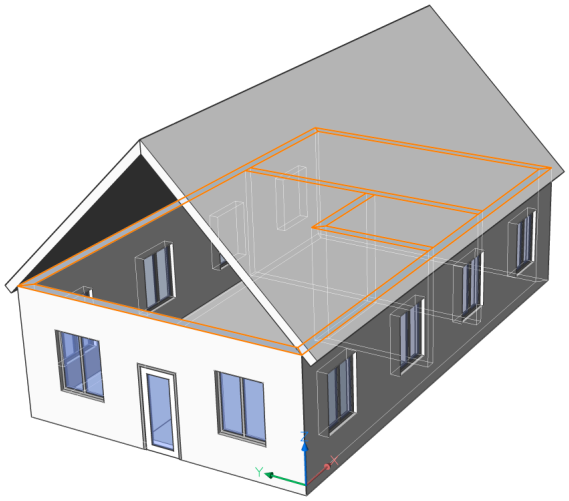


6 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field. Type a value to change the default height of the wall. To change the width of the wall, use the width field. You can switch between the two dynamic input fields by pressing Tab.

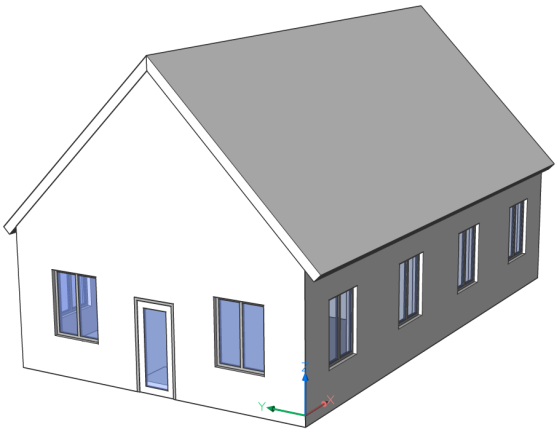



### 23.11.7 Procedure: Connecting multiple faces of the wall to the roof slab

1 To select all faces aligned to the current selection, highlight the top face of one of the walls and click the **Select aligned faces** tool in the **Select** tab of the Quad.



2 Launch the **Connect with Nearest** tool. The top faces of the wall are attached to the roof.



## 23.12 Quickdraw in BricsCAD Shape

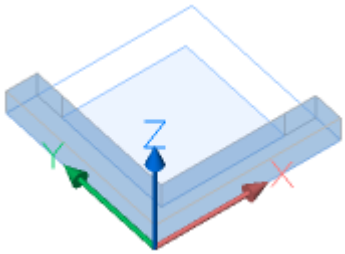
### 23.12.1 About the Quickdraw tool

The **Quickdraw** tool allows you to get started with conceptual modeling easily. You can use the **Quickdraw** tool to create a room or a whole building. Once in use, Quickdraw will remain active until you explicitly exit the command or until another command is executed.

### 23.12.2 The magnetic snapping feature

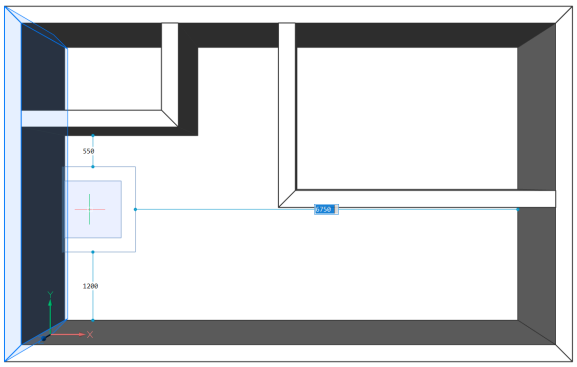
With the magnetic snapping feature, you can easily align the QuickDraw cursor. When the command is active, move the cursor close to the object you want to align to. The QuickDraw cursor automatically snaps to the object. You have the following options:

1 Align to the origin

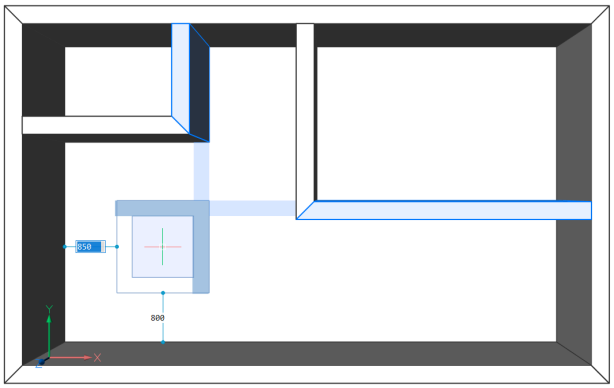


2 Align to existing walls

- To the wall itself:



- The extends of the wall:



### 23.12.3 Procedure: Creating an L-shaped house

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** from the **Model** tab, click the **Quickdraw** icon in the ribbon or type BIMQUICKDRAW in the Command line.

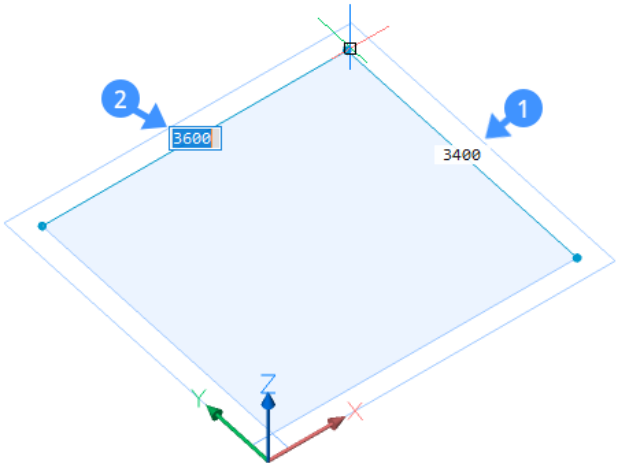
You are prompted: Select first point or [floorHeight/wallWidth/slabThickness/autoAdoptsizes]:

***Прим.:*** Select one of the options in the command bar to change the wall width, floorHeight and slab thickness of the roombeforeyou click and then specify the first corner of a building.

***Прим.:*** When the **autoAdoptsizes** option is turned on, the dimensions of the walls and slab of the new room will be the same as the walls and slab you snap to. If you don't want this to happen, you can turn this option off in the command line.

2 Select any point in the drawing area to set the first corner.

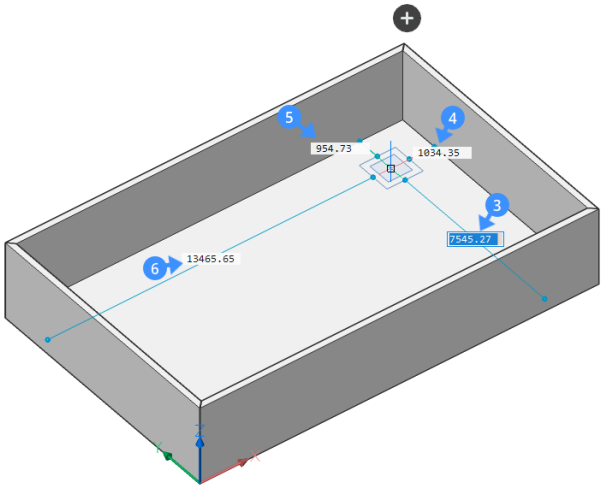
3 Drag the mouse cursor to define the footprint of the building.



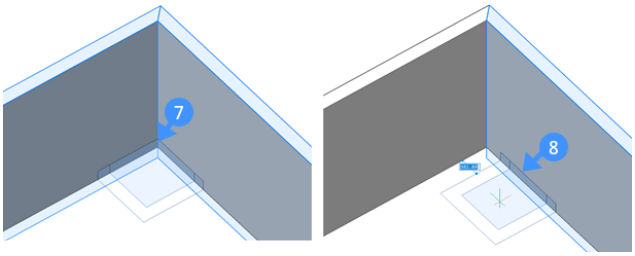

Specify the values of the room width and length in the Dynamic dimension field. Press the **Tab** key to switch between width and length and press **Enter** or just click freely. Press Enter or Escape to leave the QUICKDRAW command or you can continue to create more rooms.

4 Place the Quickdraw cursor inside the existing building to create more rooms.

5 When you place the Quickdraw cursor inside the existing building, the dynamic dimension fields display the distances from the Quickdraw cursor to the adjacent walls (3,4,5,6).



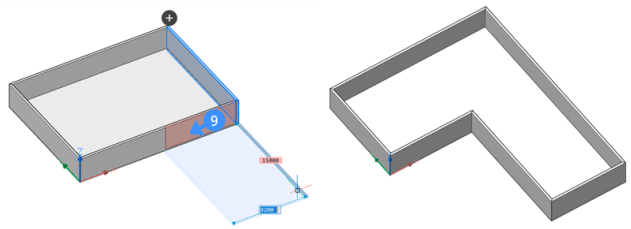

If the cursor aligns with one or two existing walls in the model, the layout of the cursor will adapt and depict the alignment.



To actually draw new walls, choose the distances from the already existing walls to specify your first point. Move your cursor or type the values in the dynamic dimension fields to do this. Draw walls in the same way as explained in steps 2 and 3.

6 To remove a section of a wall from the existing room, first, align the Quickdraw cursor with an existing wall at the inner side of the building. Then click and drag the cursor outside of the existing building.

7 The outer wall will turn red (9) to indicate the part of the wall that will be removed.

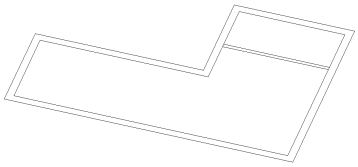


***Прим.:*** You can also align your cursor with an existing wall at the outer side of the building to remove a part of the slab. Then click and drag the cursor inside the existing building.

8 Select a second point. The wall highlighted in red will automatically be deleted and an L-shaped room will be created.

### 23.12.4 Procedure: using Quickdraw starting from a 2D layout

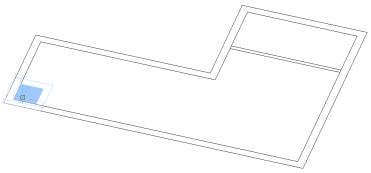
1 Open a 2D floorplan of a rectangular or L-shaped building.



***Прим.:*** It is also possible to draw a simple 2D floor plan with the POLYLINE command to start from. In this case, the default wall thickness of the BIMQUICKDRAW command will be used.

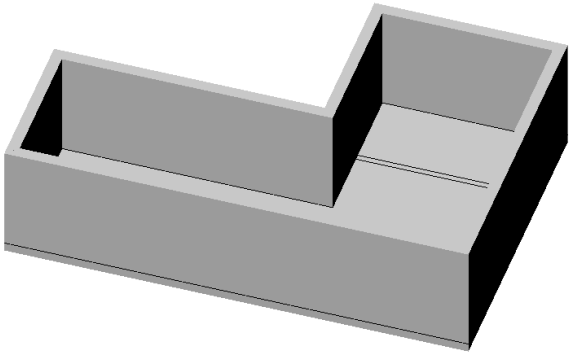
2 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the Command line.

3 Hover over one of the outer walls with the cursor.

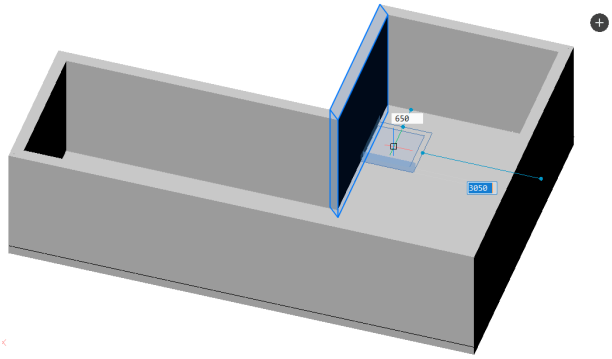


***Прим.:*** The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

4 Draw the outer walls, using the magnetic snapping feature to snap to the lines of the 2D drawing.

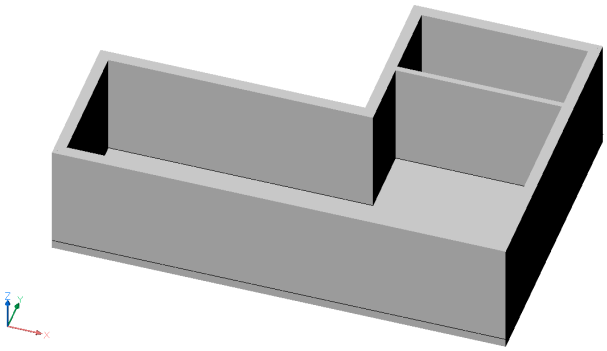


5 Hover over one of the inner walls with the cursor, using the magnetic snapping feature to snap to the lines of the 2D drawing.




***Прим.:*** The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

6 The 2D plan has been transformed into a 3D model.



## 23.13 Creating windows and doors in BricsCAD Shape

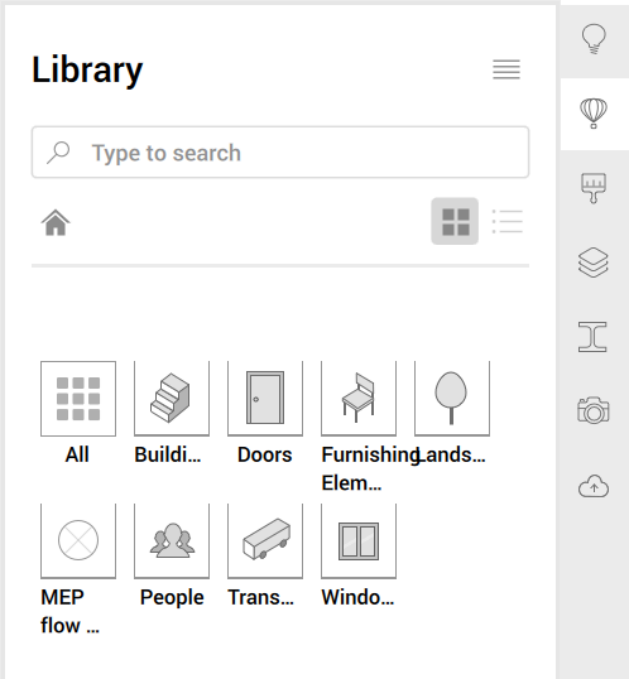
There are two ways to create windows or doors in BricsCAD Shape.

- Pick a door or window from the **Library** panel.
- Create your own custom window.

### 23.13.1 Procedure: Pick windows or doors from the library

1 Open the **Library** panel by clicking the airballoon icon on the right hand edge of the screen.

2 Click the **Doors** section in the **Library** panel.




3 Select a door from the panel.

4 Move the cursor into the model.

5 Click to place the component.

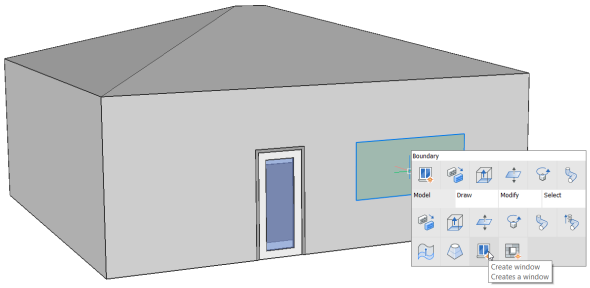
6 Press the **CTRL** key to toggle between door positioning and door size. Enter the desired values for the height and width of the door by pressing the **Tab** key. When you're done sizing and placing the door, press the **Enter** key.

***Прим.:*** This is the same for a window. Change Doors to Window section. Drag a window into the wall. You can change the size and location of a window just as you did with the door.

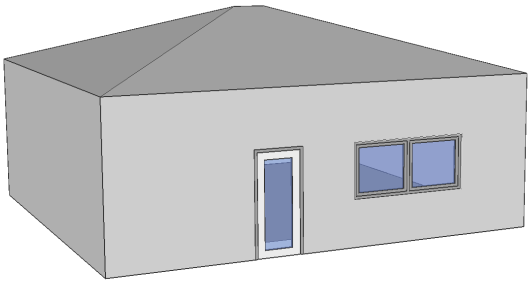
### 23.13.2 Procedure: Create a custom window

1 Use the either the **Polyline** tool, the **Polygon** tool or the **Rectangle** tool to draw the outline of the window on the wall face.

2 Move the cursor inside the window outline and, when the boundary is highlighted, choose **Create window** from the Quad.




3 The available window styles are shown in a dialog box. Move your cursor over the window design of your choice, and left-click it. The window is automatically created.



You can also create a custom window from a wall boundary:

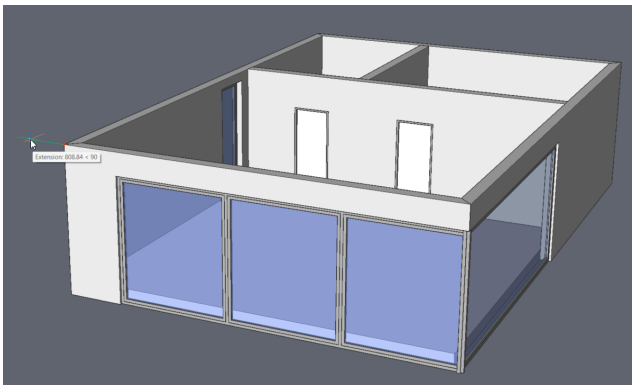
1 Select face of the outer wall and use the **Offset** tool from the Quad.

2 Move the cursor to re-size the new boundary, or key in a value and press Enter. Then, highlight the offset boundary and select **Create window** from the Quad.

## 23.14 Creating roofs in BricsCAD Shape

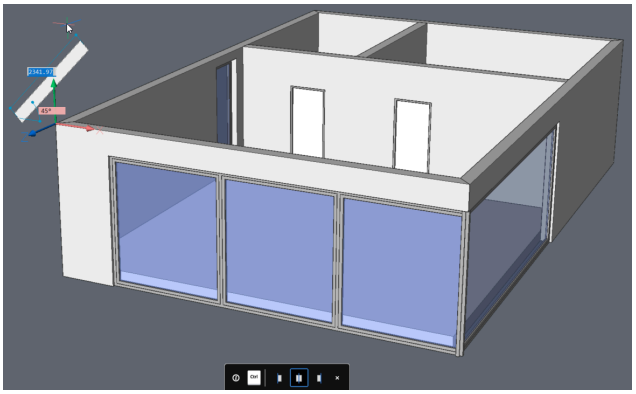
Launch the *POLYSOLID* command trough the toolbar or the "No selection" Quad.

1 To align the drawing lane to the wall surface, press the **SHIFT** button while hovering over the wall surface. The alignment of the slab can be changed by pressing the **CRTL** button.

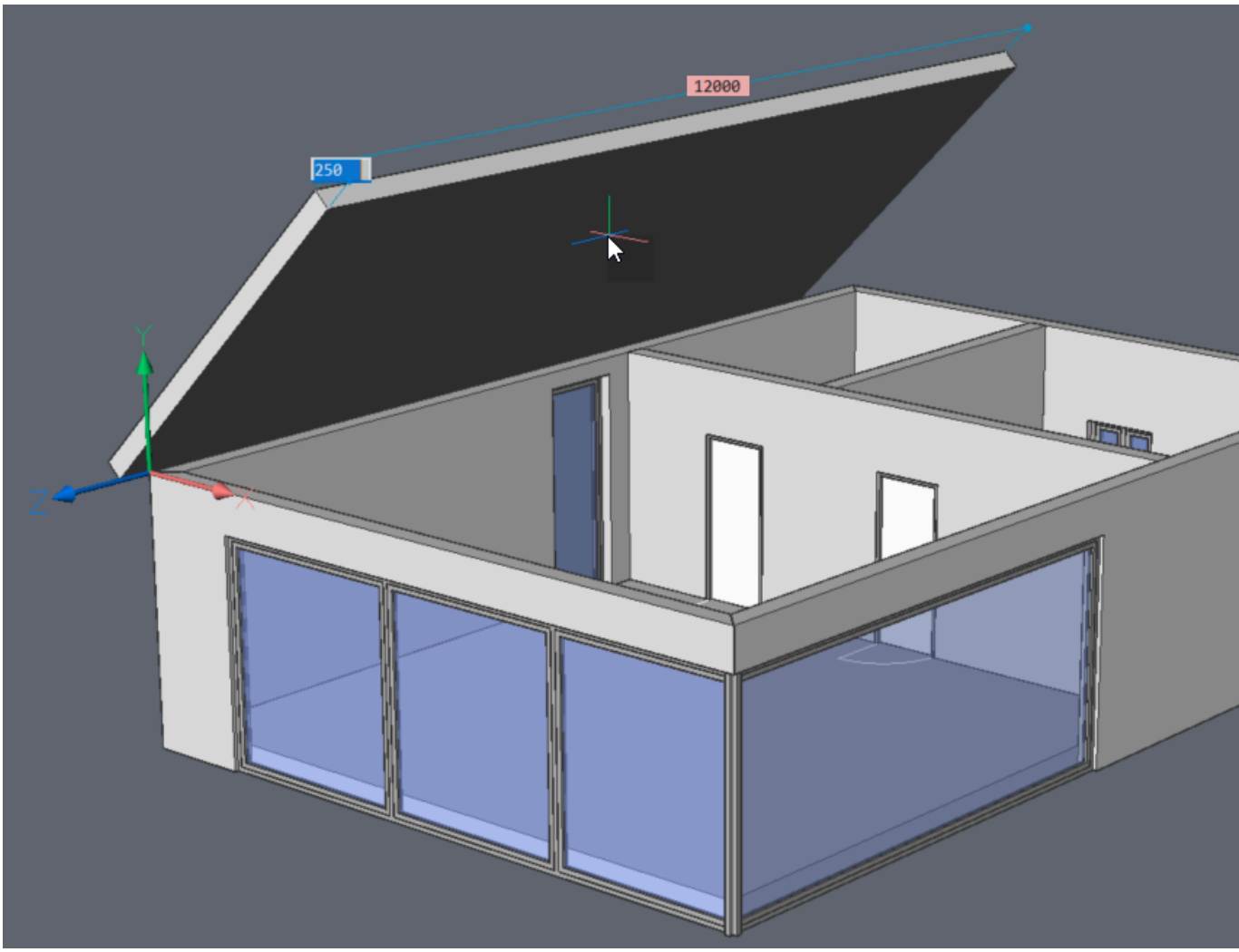


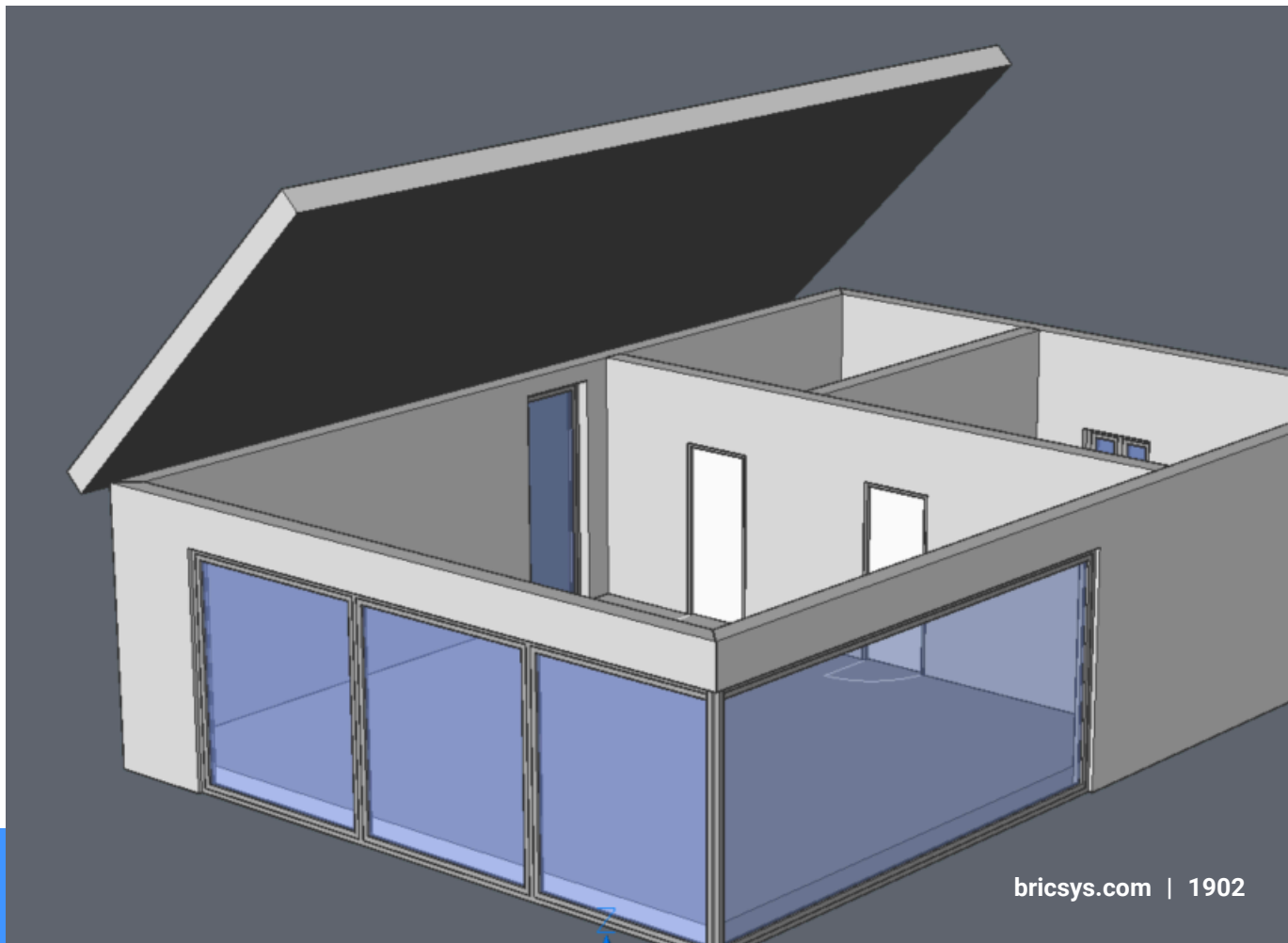
2 Pick an origin point to start drawing.

3 To lock the inclination, press TAB until the dynamic dimension of the inclination turns blue. Enter a value and press Enter.

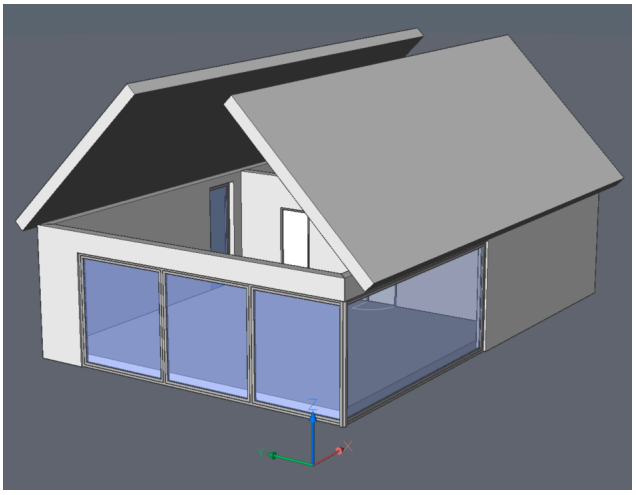


4 The profile thickness can be changes by pressing the TAB key and entering the desired value.

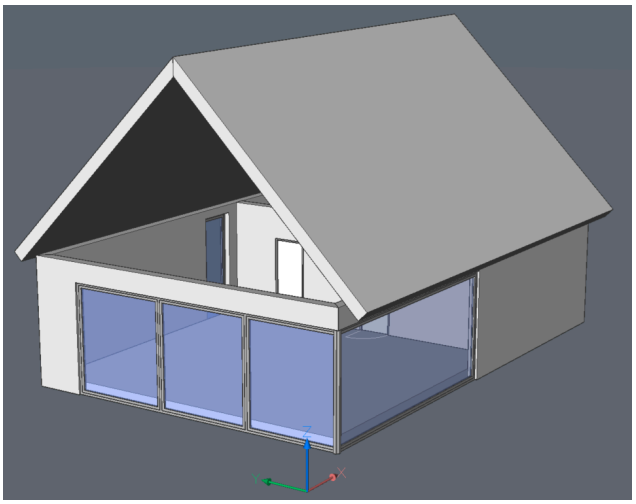

12000
250
Y
Z
X



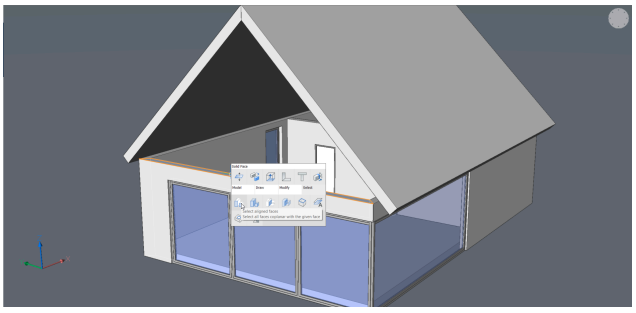
5 Repeat the same process on the other side of the building. Another way to have identical slabs on both sides is by 3D mirroring the first slab.



6 Access the **L connect** tool to connect both slabs.



7 To connect the walls to the roof, select one of the top surfaces of a wall. Trough the Quad, the **Select aligned faces** option can be accessed in the **Select** tab.



8 Once the top surfaces are selected, the **Connect with nearest** command can be accessed trough the Quad in the **Model** tab. Walls are extended up to the roof.

