



materialise
innovators you can count on

Materialise Magics²⁸

User manual



Copyright Information

Materialise, the Materialise logo, Magics, Streamics and 3-matic are trademarks of Materialise NV in the EU, US and/or other countries.

Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries.

© 2024 Materialise NV. All rights reserved.

This work contains software owned by Siemens Product Lifecycle Management Software Inc.
©1986 – 2024



Table of contents

Part I: Introduction	15
Chapter 1. Installation	16
1.1. Recommended System Requirements	16
1. Hardware	16
2. Operating Systems	16
1.2. Installation	16
1.3. Silent installation	19
1. Magics silent package installation	20
2. Magics uninstallation command	21
3. MSI package installation	21
4. Floating License activation on local computer	22
Chapter 2. Getting started with Magics	23
2.1. Home Screen	23
1. Create new	23
2. Open files/Load project files	24
3. Options	24
2.2. Parts, Scenes & Projects	25
1. Parts	25
2. Scenes	25
3. Projects	25
2.3. Part & Platform Scenes	25
1. Part Scenes	25
2. Platform Scenes	25
3. Navigating between Part & Platform Scenes	26
4. Interaction between (Mesh) Part Scenes and Platform Scenes	26
5. Edit Mode from Platform Scenes	26
6. Scene management	26
2.4. Loading projects, scenes, and parts	27
2.5. Saving projects, scenes and parts	28
2.6. Getting Help	29
Chapter 3. Magics User Interface	30
3.1. Overview	30
1. Quick Access Toolbar	30
2. The Ribbon pages	31
3. Toolbars	31
4. Tool Pages	31
5. Scene	33
6. Bigger Work Space	35

3.2. Customizations	35
1. Customizing Ribbon & Toolbars	36
2. Customize Toolpages	43
3. Customize Shortcuts	46
Part II: Magics Features	51
Chapter 1. Quick Access Bar	52
1.1. Home	52
1.2. New Project	52
1.3. Load Project	52
1.4. Import Part	52
1.5. Open	52
1.6. New Mesh Part Scene	53
1.7. New BREP Part Scene	53
1.8. New Platform Scene	53
1.9. Save Selected Part(s) As	53
1.10. Save Scene As	53
1.11. Save Project	53
1.12. Undo	54
1.13. Redo	54
1.14. Settings	54
1.15. Quick Search	54
Chapter 2. File	55
2.1. Info	55
2.2. Home	55
2.3. New Project	56
2.4. Open	56
1. Load Project	57
2. Import Part	58
3. Import Part(s) from Materialise, MGX and 3mf file formats	60
4. Open	61
5. Open Streamics client	66
6. Batch Import	66
2.5. Save	71
1. Save Project	72
2. Save Project As	73
3. Save selected part(s) As	73
4. Save scene As	75
5. Save All in Directory	75
6. Save Selected Part(s) to Streamics	76

7. Save Platform to Streamics	76
2.6. Reporting	76
1. Generate Report	77
2. Create Report Template	77
3. Save Part(s) As 3D PDF	91
4. Print	91
2.7. Machines	95
1. Change Machine	95
2. Machine Properties	95
3. My Machines	95
2.8. Options	96
1. Settings	96
2. Customize UI	96
3. Licences	96
4. Export Magics Profile	96
5. Import Magics Profile	96
6. Exit	97
Chapter 3. Fix	98
3.1. Automatic Fixing	98
1. AutoFix	98
2. ShrinkWrap	98
3.2. Semi-Automatic Fixing	105
1. Normals Fix	105
2. Automatic Stitching	105
3. Holes Fix	105
4. Noise Shells	105
5. Unify	105
6. Shells to parts	106
7. Remove Small Parts	106
8. Filter Sharp Triangles	107
9. Remove Identical Triangles	107
10. Mark Overlapping Triangles	107
3.3. Manual Fixing	107
1. Invert Normals	107
2. Fill Hole Mode	107
3. Create Triangle	108
4. Translate Part Points	108
5. Move Part Points	108
3.4. Enhance	108

1. Triangle Reduction	108
2. Smoothing	109
3. Refine and Smooth Parts	110
4. Subdivide part	111
5. Remesh	111
Chapter 4. Edit	113
4.1. Insert	113
1. Create	113
2. Create from Image	114
4.2. Edit	116
1. Hollow Part	116
2. Polyline Cut	120
3. Section Cut	126
4. Perforator	135
5. Surface to Solid	138
6. Fillet	142
7. Chamfer	143
8. Extrude	144
9. Offset	146
10. Milling offset	150
11. Prop generation	150
12. Delete Face	152
13. Taper Face	153
14. Adapt Holes	155
15. Convert Scene to Mesh	160
4.3. Split & Merge	161
1. Shells to Parts	161
2. Boolean	161
3. Merge Parts	165
4.4. Label	165
1. Label	165
2. Mass Label	174
3. Label Tags	181
4.5. Fit 2 Ship	184
1. RapidFit	184
2. FormFit	206
Chapter 5. Lattice	207
5.1. Mesh Lattice	207
1. Honeycomb Lattice	207

2. Volume Mesh Lattice	211
3. Tetrahedron Lattice	218
5.2. Slice Based Lattice	222
1. Slice Based Volume Lattice	222
2. Slice Based Tetrahedron Lattice	225
5.3. Beam Lattice	226
1. Volume Beam Lattice	227
2. Filter Loose Beams	229
3. Edit Beam Lattice Properties	231
4. Convert Beam Lattice to Mesh	232
Chapter 6. Texture	234
6.1. Main	234
1. Select Texture	234
2. New Texture	234
3. Edit Texture	234
4. Update Textures	238
5. Copy Texture	238
6. Paste Texture	238
7. Delete Texture From Triangles	238
8. Delete Texture	238
9. Part to texture	238
6.2. Color	240
1. Paint Part	240
2. Color Surfaces	241
3. Split part by color	241
6.3. Visibility	241
1. Toggle Textures Visibility	241
2. Invert Textures Visibility	241
3. Triangle Colors	241
Chapter 7. Position	242
7.1. Transformation	242
1. Translate	242
2. Rotate	244
3. Pick and Place Parts	246
4. Rescale	246
5. Mirror	250
6. Alignment	252
7. Bottom/Top Plane	255
8. Minimize Bounding Box	257

9. Fit to Platform	257
7.2. Copies	258
1. Duplicate	258
2. Batch Duplicate	260
7.3. Automatic	262
1. Automatic Placement	262
2. Orientation Optimizer	271
3. Orientation Comparator	275
4. Shape Sorter	278
5. 3D nester	280
7.4. Group	280
1. Group	280
2. Ungroup	280
3. Remove From Group	280
4. Grouping visualization	280
7.5. Coordinate System	282
1. User Coordinate System	282
2. Import UCS File	285
7.6. Defaults	285
1. Translate to Default Z Position	285
2. Translate to Default Position	285
3. Original Position	285
4. Original in New Scene	285
5. Save Current Position	285
Chapter 8. Build Preparation	286
8.1. Scenes	286
8.2. Scenes: Virtual Copies	286
1. A Part and his Virtual Copies	286
2. Edit a Virtual Copy	286
3. Naming of virtual copies	287
4. Advised ways of working	288
8.3. Platform scene management	289
1. New Platform Scene	289
2. Add Part to Platform	290
3. Duplicate Platform	290
4. Rename Scene	291
5. Save Scene As	291
6. Close Scene	291
8.4. Machine	291

1. Export Platform	291
2. Machine Properties	293
3. Change Machine	315
4. My Machines	315
8.5. Positioning & Editing	319
1. Z-Compensation	319
2. Create base	320
8.6. Part Pages	321
8.7. Sinter	324
1. 3D Nester	324
2. 3D Nester - Check slices distribution	336
3. Subnester	338
4. Sinterbox	342
5. Toggle Nesting Density	345
Chapter 9. Support Generation	346
9.1. Introduction	346
9.2. Magics - Support Generation ribbon	346
1. Supported Area Preview	346
2. Generate Support	348
3. Generate Support of Selected	348
4. Manual Support	348
5. Re-trim supports on platform	349
6. Transfer Support	350
7. Export support	350
8. Unload support	351
9. Add No-Support Zones	351
10. Toggle No-Support Zones	353
11. Support Visibility	353
12. Baseplate Visibility	353
9.3. Automatic Support Generation	353
9.4. Machine Setup	354
1. Build Time Estimation	354
2. Cost Estimation	354
3. Support Generation Mode	355
4. Support generation parameters	356
9.5. Support Parameters	363
1. General	363
2. Block	373
3. Line	386

4. Line*	394
5. Point	395
6. Point*	398
7. Web	399
8. Contour	402
9. Gusset	406
10. Advanced Gusset	410
11. Combi	413
12. Volume	414
13. Tree	416
14. Tree*	421
15. Hybrid	422
16. Support on graphs	422
9.6. Modifying Surfaces	424
1. Modifying the surfaces	424
2. Part Info page	425
3. Surface Info page	425
4. Support List page	426
5. Support Toolbox	429
6. Support Types & Parameters	439
9.7. 2D editing of supports	439
1. 2D and 3D Editing of Supports	439
2. 2D Editing of Volume Supports	445
9.8. 3D editing of supports	448
1. Add Raft	448
2. Stabilization Wall	449
3. Create and edit tree support manually	450
9.9. Saving and Exporting Supports	451
1. Save Support	451
2. Export Support	451
9.10. Visualization of Support Structures	452
9.11. Support on 3D Texture and Slice Based Lattice	452
Chapter 10. Analyze & Report	455
10.1. Analyze Build	455
1. Out of bounds	455
2. Collision Detection	455
3. Interlocking Analysis	457
4. Wall Thickness Analysis	457
5. Detect Trapped Volumes	462

6. Build Risk Analysis	465
7. Check Slices Distribution	465
10.2. Estimate	467
1. Build Time Estimation	467
2. Cost Estimator	469
3. Material Cost Estimation	469
4. Volume Estimation	470
5. Toggle Nesting Density	471
10.3. Measure	471
1. Measurement Distance Point to Point	471
2. Measurement Thickness	471
3. Add Real Measurements	471
4. Measurements Quality	471
5. Part Comparison	471
10.4. Product and Manufacturing Information (PMI)	474
10.5. Report	476
1. Save Part(s) as 3Dpdf	476
2. Generate Report	476
3. Create Report Template	476
Chapter 11. Slicing	477
11.1. Introduction	477
11.2. The Slice ribbon	477
1. Slicer Properties	477
2. Preview Slicer	479
3. Machine Setup	480
Chapter 12. Materialise Software	487
12.1. Streamics	487
1. Open Streamics client	487
2. Save Selected Part(s) to Streamics	487
3. Save Platform to Streamics	487
4. Create Quote in Streamics	487
5. Create Order in Streamics	487
6. Reset revision(s)	487
12.2. CO-AM	487
1. Magics integration with CO-AM	487
2. CO-AM settings page in Magics	487
3. Connect with CO-AM	488
4. Log in with CO-AM account	488
5. Disconnect CO-AM account	490

6. Using Magics with multiple CO-AM accounts	490
7. Open CO-AM part(s) in Magics	490
8. Open CO-AM platform in Magics	491
9. Save parts and platforms to CO-AM	492
10. Supported Magics commands for CO-AM parts	495
11. Connection of Parts and Platforms with CO-AM	497
12. CO-AM Machine Properties	498
13. Supports on CO-AM platforms	499
14. Export-controlled parts	500
12.3. Machine Manager	501
1. Magics integration with Machine Manager introduction	501
2. Connect and Login to Machine Manager	501
3. Machine selection	502
4. Build preparation	503
5. Strategy assignment	503
Chapter 13. View	507
13.1. Views	507
1. ISO views	507
2. Smooth Shading	507
3. Fast Preview	507
13.2. Elements	508
1. Toggle Grid	508
2. Rulers	508
3. Coordinate System	508
4. No-build zones	508
5. Part Dimensions	508
6. Gravity Center	508
7. Combined Bounding Box	509
8. Tag ID	509
9. Tag Names	509
10. Tag Path	509
11. Selection Points	510
12. Toggle Graph Rendering	510
13.3. Export View	510
1. Export to Image	510
2. Copy Viewport	511
3. Print	511
Chapter 14. Options & Help	512
14.1. Settings	512

1. Mouse Input	512
2. General	515
3. Modules	522
4. File I/O	539
5. Analyze	554
14.2. Network	555
14.3. Customize UI	555
14.4. Magics Profile	556
1. Import settings of previous version	557
2. Import Magics Profile	558
3. Export Magics Profile	559
14.5. Licenses	560
1. Licenses	560
14.6. Help	561
1. Online Support	561
2. Manual	562
3. What's New	562
14.7. About	563
1. About Magics	563
14.8. Logging	563
1. Show log	563
2. Show history	564
Chapter 15. Toolbars	566
15.1. General toolbar	566
1. View tools	566
2. Marking tools	568
3. Errors visualization	575
15.2. Additional tools	576
1. Unload Part	576
2. Border marked Triangles	576
Chapter 16. The Toolpages	577
16.1. General Pages	578
1. Part List Page	578
2. Build Time Estimation page	583
16.2. Part Pages	585
1. Part Info page	585
2. Part Fixing Info page	586
16.3. Fixing Pages	590
1. Profiles	590

2. Autofix page	591
3. Near Bad Edges page	592
4. Hole page	592
5. Triangle page	598
6. Shell page	602
7. Point page	603
16.4. Measurements pages	604
1. Measurements on BREP parts	605
2. Distance page	605
3. Circle page	608
4. Angle page	609
5. Info page	610
6. Final Part page	611
7. Report page	613
8. Measurements on slices	614
16.5. Annotation Pages	614
1. Annotations page	614
2. Attachments page	616
3. Textures page	616
4. Beam Lattice page	618
16.6. Slices Pages	619
Part III: Extra Information	626
Chapter 1. Windows Configuration Info	627
1.1. Enabling long path support on Windows	627
1.2. Setting the default application to open specific file formats	628
1. Windows Settings	629
2. File properties page	630
Chapter 2. Contact Info	634

A large graphic consisting of two overlapping triangles. The top triangle is light blue and points downwards. The bottom triangle is a darker blue and points upwards. They meet at a horizontal line.

materialise
innovators you can count on

Part I: Introduction



Chapter 1. Installation

We recommend that you close all other applications before installing Magics. You must have administrative privileges to install the software.

Always take necessary IT backup precautions before installing (an update of) a software. We strongly recommend backing up all relevant data prior to initiating the installation process. For Magics specifically, we recommend exporting the Magics profile (.mpf), UI settings (.uimpf) and machine profiles (.bpprof, .mmcf) before upgrading. This not only safeguards your information against potential data loss or corruption during the installation but also provides a safety net in case you need to revert back to a previous version or installation.

1.1. Recommended System Requirements

1. Hardware

For each release of Magics, the minimum (recommended) hardware requirements can be found in the respective release notes. But depending on your needs and expectations, we can recommend [better hardware requirements](#) that will make Magics run even smoother.

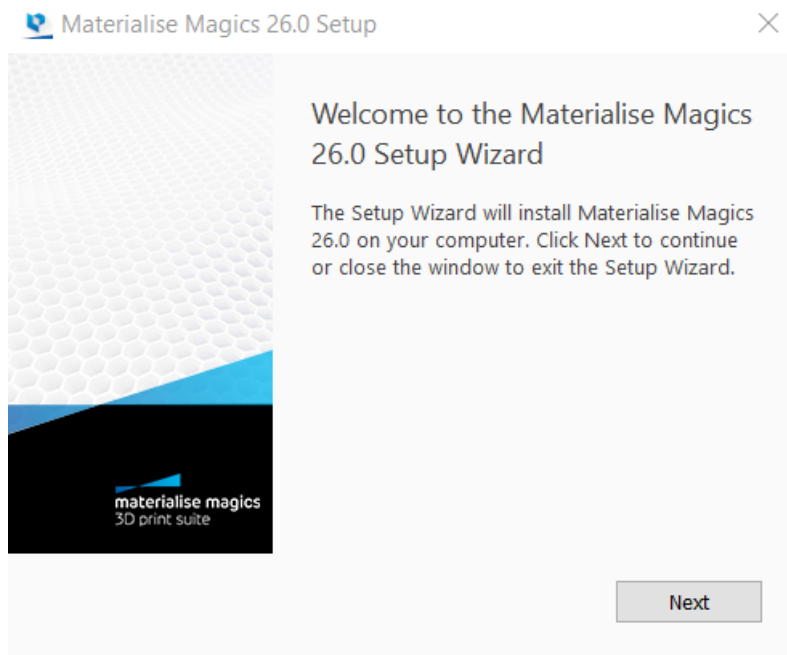
2. Operating Systems

It is important to run Magics on the correct Windows operating system. The required operating system is listed in the release notes of the respective Magics version. Please note that Materialise Software will discontinue supporting an operating system, or a specific version of an operating system, from the moment that the OS vendor discontinues support.

1.2. Installation

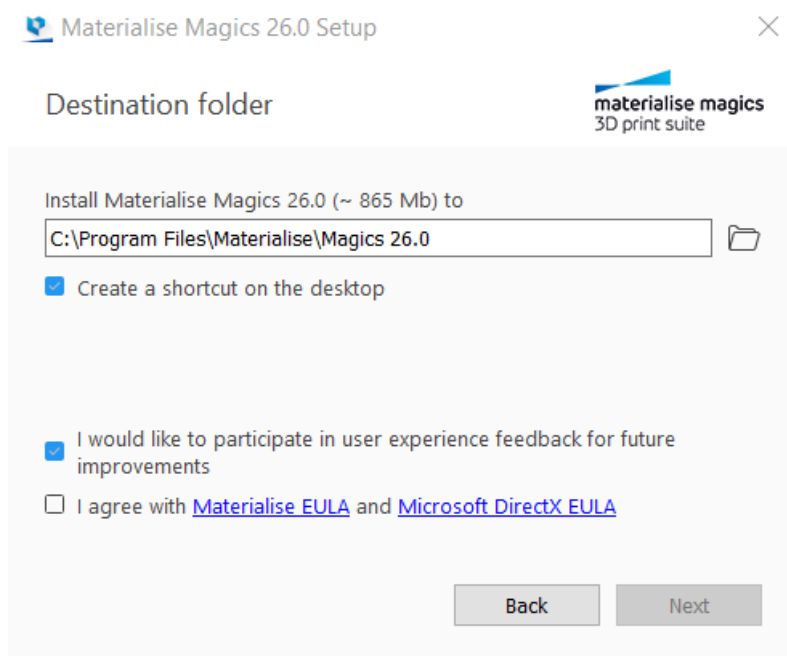
Step 1

Wait until the Windows installer is ready to start the installation.



If your operating system language is supported by Magics, the installer will default to that language. If it is not, then Magics will be installed in English.

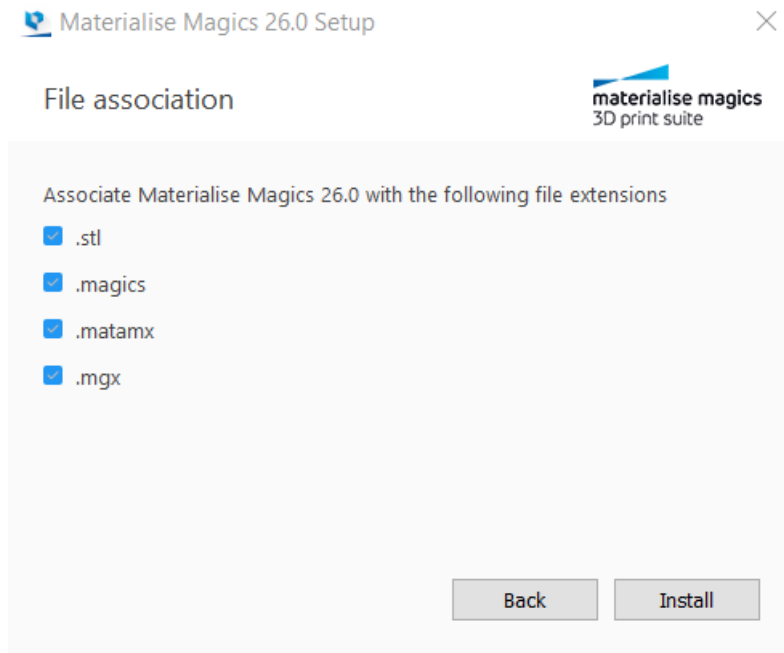
Step 2



Select the folder where you want to install Magics RP. Via the browse button you can specify a new directory, however we advise to use the default directory.

After reading the license agreement, select the “I agree with Materialise EULA and Microsoft DirectX EULA” checkbox and click on the **Next** button.

Step 3

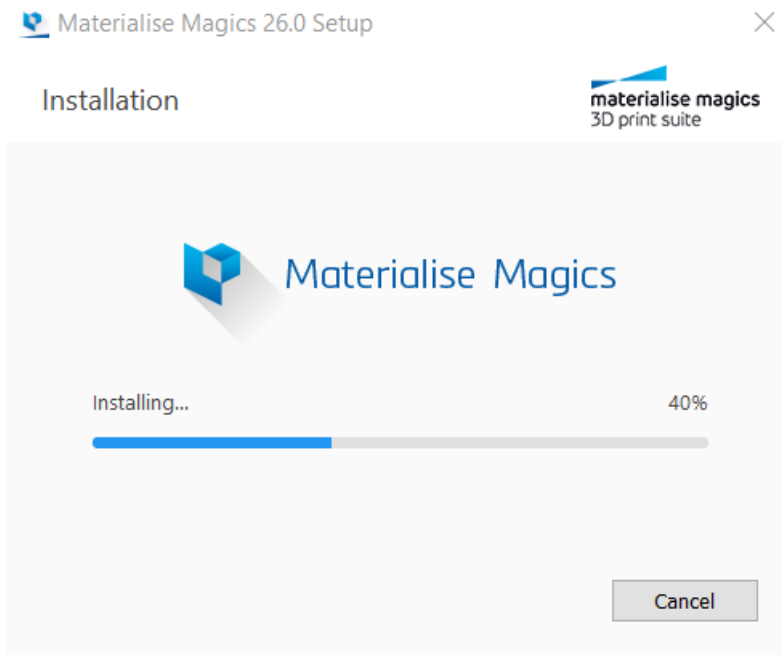


Select which file extensions you wish to open using Magics software and click on the ***Install*** button. It is advised to keep the standard settings.

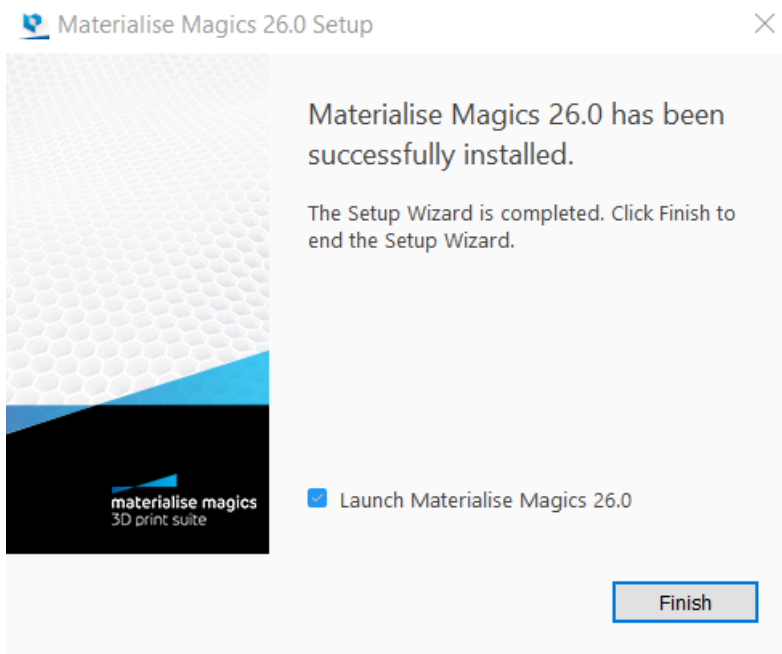


If you would like to manually set the default apps for any supported file format, see ***Setting the default application to open specific file formats, page 628***

Step 4



Step 5



The software is successfully installed. Choose if you want to launch Magics immediately by checking or unchecking "Launch Materialise Magics 26.0" checkbox and click on **Finish**.

1.3. Silent installation

Our standard installations work well for users who install our software themselves. Large companies with a centralized IT organization often would like to install software from a central location on a specific time. This is called "silent installation". This allows the IT organization to

install software quickly and easily over multiple computers without any inconvenience for the users.

It will allow the IT organization to write a simple program to do these installations using the commands described below. This can be done for nearly all our software, with the exception of some of our older products.

Below you can find the commands for the two installation packages which are used by Materialise as well as the command to connect Magics to your floating license server.

1. Magics silent package installation

To install silently Magics RP from version 26 or later you should use either of the following commands:

- when installing in the standard installation folder:
 - <installer.exe> install --accept-licenses --default-answer --confirm-command

```

Administrator: Command Prompt
Microsoft Windows [Version 10.0.19042.1706]
(c) Microsoft Corporation. All rights reserved.

C:\WINDOWS\system32>cd \Temp

C:\temp>Magics_setup_26.0.exe install --accept-licenses --default-answer --confirm-command
[0] Arguments: Magics_setup_26.0.exe, install, --accept-licenses, --default-answer, --confirm-command
[45] Operations sanity check succeeded.
[47] No target directory specified, using default value: "C:\Program Files\Materialise\Magics 26.0"
[1179] Preparing meta information download...
[1304] License "DirectX EULA" accepted by user.
[1305] License "EULA Chinese" accepted by user.
[1305] License "EULA English" accepted by user.
[1305] License "EULA French" accepted by user.
[1306] License "EULA German" accepted by user.
[1306] License "EULA Italian" accepted by user.
[1306] License "EULA Japanese" accepted by user.
[1306] License "EULA Spanish" accepted by user.
[1307] License "Third-Party License English" accepted by user.
[1312] Warning: QFont::setPixelSize: Pixel size <= 0 (-1) (text\qfont.cpp:950, void __thiscall QFont::setPixelSize(int))

[1314] Selected components without dependencies:
BuildProcessorSystemComponent
CommonProduct
CommunicationService
DotNetComponent
ExternalProjects
LocalLicenseServerComponent
MagicsAdditionalFiles
MatConvertComponent
  
```

- when installing in a specific installation folder:
 - <installer.exe> install --accept-licenses --default-answer --confirm-command --root <path-to-installation-folder-wrapped-with-quotes>

```

Administrator: Command Prompt
Microsoft Windows [Version 10.0.19042.1706]
(c) Microsoft Corporation. All rights reserved.

C:\WINDOWS\system32>cd \temp

C:\temp>Magics_setup_26.0.exe install --accept-licenses --default-answer --confirm-command --root "C:\My Program Files\Magics 26.0"
[0] Arguments: Magics_setup_26.0.exe, install, --accept-licenses, --default-answer, --confirm-command, --root, C:\My Program Files\Magics 26.0
[17] Operations sanity check succeeded.
[1099] Preparing meta information download...
[1238] License "DirectX EULA" accepted by user.
[1239] License "EULA Chinese" accepted by user.
[1239] License "EULA English" accepted by user.
[1240] License "EULA French" accepted by user.
[1240] License "EULA German" accepted by user.
[1240] License "EULA Italian" accepted by user.
[1240] License "EULA Japanese" accepted by user.
[1240] License "EULA Spanish" accepted by user.
[1240] License "Third-Party License English" accepted by user.
[1242] Warning: QFont::setPixelSize: Pixel size <= 0 (-1) (text\qfont.cpp:950, void __thiscall QFont::setPixelSize(int))
[1243] Selected components without dependencies:
BuildProcessorSystemComponent
CommonProduct
CommunicationService
DotNetComponent
ExternalProjects
LocalLicenseServerComponent
MagicsAdditionalFiles

```

Additional reference documentation can be found [here](#).

2. Magics uninstallation command

To uninstall Magics, use the following command:

```
C:\Program Files\Materialise\Magics 26.0>Uninstall.exe --default-answer --confirm-command purge
```



Make sure to specify the correct Magics version in the command.

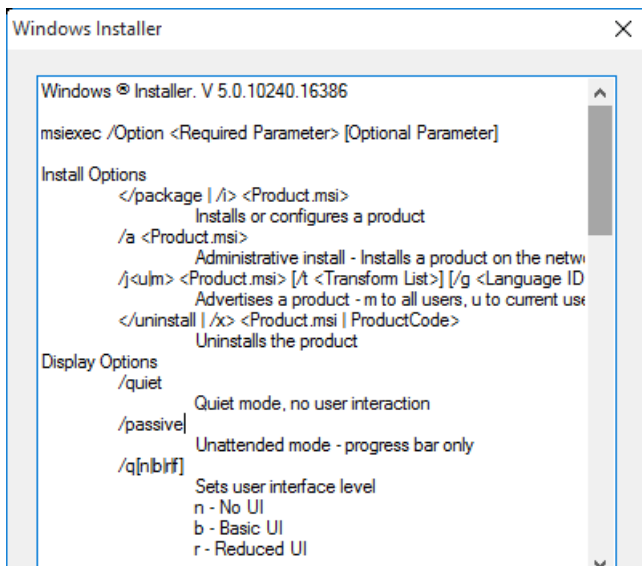
3. MSI package installation

e-Stage, 3-matic and a number of our other products have a Windows Installer package, in the form of an .msi file. For these packages, different commands will need to be used. msiexec.exe is the receiver of these commands, and will activate the .msi files. This means all commands are aligned with the standard msiexec commands. The most important for the installation are described below.

Commands

- Installation commands
 - /i - install
 - /uninstall – uninstall
- UI commands

- /passive - displays minimal UI with no prompts
 - /quiet – quiet mode, no user interaction
 - /qn – displays no UI
- Additional commands
- /norestart – suppresses any attempts to restart by the installer
 - /log <Logfile> – logs to a specific file
- Setup help
- msiserver.exe /? – setup help: will provide you more information on commands



Example

`msiserver.exe /quiet /norestart /i D:/e-Stage-7.0.4.157-x64.msi`

4. Floating License activation on local computer

To activate the floating license together with the installation of Magics, you can use the command below. This is done using the `Magics.exe` which will appear in installation folder after installation will be finished.

Command line:

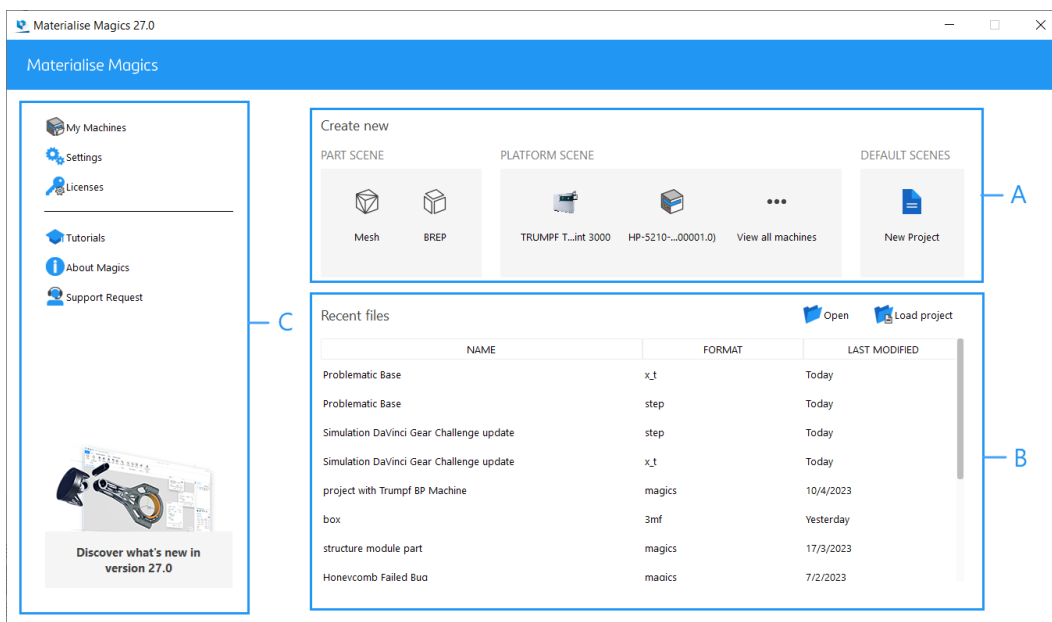
- `Magics.exe /flsregister {server_name}:{port}`

Chapter 2. Getting started with Magics

To easily get started with Magics, it's helpful to get familiar with some of its basic concepts. Note that in this chapter, some functionality will only be mentioned briefly to give you a high-level overview, more detailed descriptions can be found in **Part II: Magics Features, page 51**.

2.1. Home Screen

When opening Magics, the first dialog you will encounter is the Home Screen. From the Home Screen it is possible for you to perform a few tasks directly. This screen is shown when there are no active part or platform scenes in the current Magics session. There are a few sections on the Home Screen dialog that you should take note of :



- A. Create new
- B. Open files / Load project files
- C. Options



You can only minimize or close this screen, at the moment it is not possible to maximize this to full screen mode.

1. Create new

Part scene

You can create a new part scene from the home screen by selecting the appropriate create command that is available in the Part subsection. Selecting the Mesh or BREP command will respectively create part scenes of the specific type.

- See **New Mesh Part Scene, page 53** for more information about creating new mesh part scenes.

- See **New BREP Part Scene, page 53** for more information about creating new BREP part scenes.

Platform scene

Your recently used machines will be listed in the Platform subsection. Clicking on the desired machine will open a new platform scene with the target machine. If this section is empty, you would need to edit the "My Machines" list by opening the "My Machines" dialog from the Options section.

- See **New Platform Scene, page 289** for more information about creating new platform scene.

Default scenes

You can create a new untitled project with default scenes and parts based on a Magics project template file. This template file needs to be set in the Settings before this can be accessed from the home screen.

- See **New Project, page 56** for more information about the New Project command.

2. Open files/Load project files

In the open files section you will be able to find the command to "Open" to select for the target files to be opened through the Windows Explorer dialog. You can also observe and open the most recent files that were opened or loaded in Magics from the files list in this section.

- See **Open, page 61** for more information



Note: You can drag and drop one or more files into any section of the Home Screen to open the files quickly. If you drop exactly one Magics file, then the file will be loaded as a project file else if multiple files are dropped then they will all be opened automatically.

For loading magics project files the command "Load project" can be used. If magics project is loaded it's reflected on the top title bar accordingly.

- See **Load Project, page 57** for more information about loading your magics file as a project.



Note: Magics project files listed in the recent files will be loaded as a project automatically on click while all other file types will be opened automatically.

3. Options

In this section, you will be able to find the common commands needed to perform housekeeping for your Magics application. You will be able to find command to show the necessary settings and license dialogs in this section along with links to launch the What's New page of your Magics version for more detailed info.

2.2. Parts, Scenes & Projects

1. Parts

A part can be one among few different types of data such as Mesh (triangle-mesh), BREP (from CAD files) or Slices (slice stacks). A file may contain one or more parts of the same or different types. However the target output files that can contain this formats may vary between the different types. They may also come from a variety of different file formats, many of which are not native to Magics or other Materialise software. (For example: .stl, .step)

- See **Open, page 61** for more information on the file formats and the supported part types.

2. Scenes

Scenes are files which contain a collection of parts in a single scene that aids multi-part scene work flows. Opening a scene file will always result in one scene with multiple parts being loaded into the same scene.

3. Projects

Projects (*.magics) are files which can contain (multiple) part or platform scenes. These files are often native to Magics and can thus contain information specifically related to Materialise software, f.e. machine information, parametric support and measurements.

2.3. Part & Platform Scenes

To avoid cluttering your workspace with too much functionality that is not relevant for certain workflows, there are different types of scenes, which have different tools available.

1. Part Scenes

Within part scenes, you can focus on editing, fixing or simply viewing a single part or assembly. There are 2 types of Part Scenes: one for mesh parts and one for BREP parts. Depending on which type of file was opened (and whether it was converted to mesh or not), the right type of part scene will be shown. Both part scenes contain similar tools, but adjusted to work on either mesh or BREP parts.

If needed, it's possible to convert a BREP part scene to a mesh part scene by using the "Convert Scene to Mesh" command.

- See **Convert Scene to Mesh, page 160** for more information.

Use the "Import Part" command to import parts from additional files within the same part scene.

2. Platform Scenes

Platform scenes contain a machine and allow you to focus on build preparation. From this scene it is also possible to enter Support Generation mode.

Use the “Import Part” command to import parts from your files to the active platform scene. Currently it’s only possible to place mesh parts on a platform scene. BREP parts will be converted to mesh.

3. Navigating between Part & Platform Scenes

It’s possible to have both part & platform scenes open within one Magics instance, and to navigate between them. When on a mesh part scene, the “Add Part to Platform” function allows you to place a part on a platform. Since the platform scene currently does not support BREP parts, this function is not present on BREP part scenes.

On a part scene, it’s possible to open selected parts in respective part Scenes through the “Edit Part in Part Scene” command.

4. Interaction between (Mesh) Part Scenes and Platform Scenes

Parts on the part scene are the “master parts”, while parts on platform scenes are instances of these master parts. This means that when geometrical changes are made to master parts, these changes are also applied to the copies of this part on the platform scene(s).

When rotating or translating a part in the part Scene, these changes are not applied to parts on the platform Scene, and vice-versa (changing the position or orientation of parts on Platform Scenes does not affect the position or orientation of the master parts).



Note: if a single instance or multiple instances with the same orientation are opened in the part scene, the platform orientation of this instance(s) will be preserved upon the part scene creation. Any further position or orientation changes won't have any affect on the platform scene anymore.

When opening a part with multiple instances in the part Scene you can choose whether to edit only the selected instance or all instances in one go. For editing only selected instances, a new master part will be created to represent this geometrically different part and thus the link with other (unselected) instances will be lost. If you want to make a design change to all the instances of the part, do select 'All virtual copies' option in the dialog to proceed.

5. Edit Mode from Platform Scenes

‘Edit mode’ has been introduced at the platform level. This mode is designed to allow for limited editing and fixing of multiple parts or virtual copies within the platform context. Edit Mode is accessible via the command in the Home ribbon of Platform layout.

6. Scene management

- See **Platform scene management, page 289** for more information.

2.4. Loading projects, scenes, and parts

Loading project

If you want to load project file in Magics, you can use "Load project" command. Only single project file can be loaded at the same time that will replace the contents of the current session of the Magics with the contents of the project file.

- See **Load Project, page 57**

Importing part(s)

If you want to import part(s) to the active scene, you can use the "Import Part" command. Since scenes are type sensitive, this command will filter only the type of parts that can be added to the target scene automatically.

- See **Import Part, page 58**



Note: You can drag and drop files to the scene view port area to import parts quickly to the active scene.



Note: In Magics RP28, any CAD files loaded through the "Import Part" command on any platform scenes will trigger the MatConvert CAD to Mesh conversion automatically.

If you want to add mesh parts from an existing Mesh part scene quickly to the target platform, you can find the "Add Part to Platform" command from the Quick Access Toolbar or the Search menu to choose the target platform scene to add the part to.

- See **Add Part to Platform, page 290**

Open

The "Open" command would allow you to open one or more files of all supported file types quickly into the target Magics session. If you are unsure about the contents of the file you have received, opening the file is the best way to ensure all parts in the file can be accessed in the application.

- See **Open, page 61**

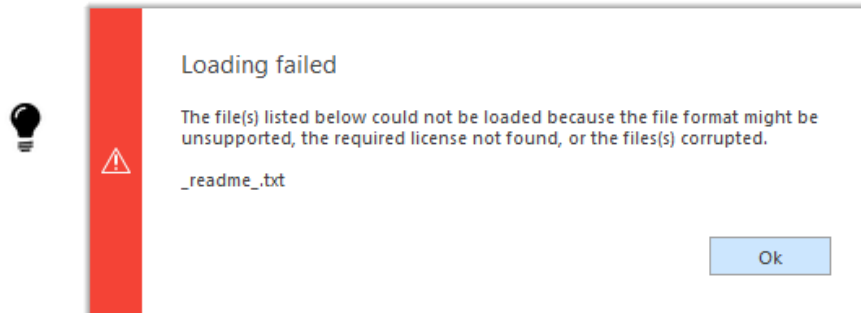


Note: You can drag and drop files to the Home Screen or the Scene Tab-Bars (while Scenes are active) to open the files quickly.

If you would like to avoid loading CAD files as BREP parts when opening the files, you can always disable this behavior in the settings.

- See **File I/O, page 539**

Note: When loading unsupported files, the following message will be shown to specify the files the problems encountered on.



2.5. Saving projects, scenes and parts

It's possible to save different types contents in to specific target files through Magics.

Projects

Lastly, if you want to save the full project with all the opened scenes to a single .magics file, you can use the "Save Project" or "Save Project As" commands.

- See **Save Project, page 72**

Closing a project with unsaved changes in any way (closing Magics, starting a new project, loading another project) will prompt a message to ask you whether you want to save the current project.

Scenes

You can save a target scene to an appropriate file type based on the contents of the scene using the "Save Scene As" command. For example, a platform scene can be saved to .magics, .matamx and .3mf file. Depending on the chosen format, the majority of the information of that scene (parts, machine, measurements...) will be preserved.

- See **Save scene As, page 75**

Closing a scene with unsaved changes will prompt a message to ask you whether you want to save the current scene.

Parts

Parts can be saved to many different formats using the "Save Selected Part(s) As" command, including formats that are not native to Magics or other Materialise software. Note that each file type supports different information, so some information might get lost when saving parts to non-native formats.

- See **Save selected part(s) As, page 73**



Unloading a part with unsaved changes will prompt a message to ask you whether you want to save the part.

- See ***Unload Part, page 576***

2.6. Getting Help

For Materialise software online help, please go to: <https://help.materialise.com>

For support request, see ***Online Support, page 561***

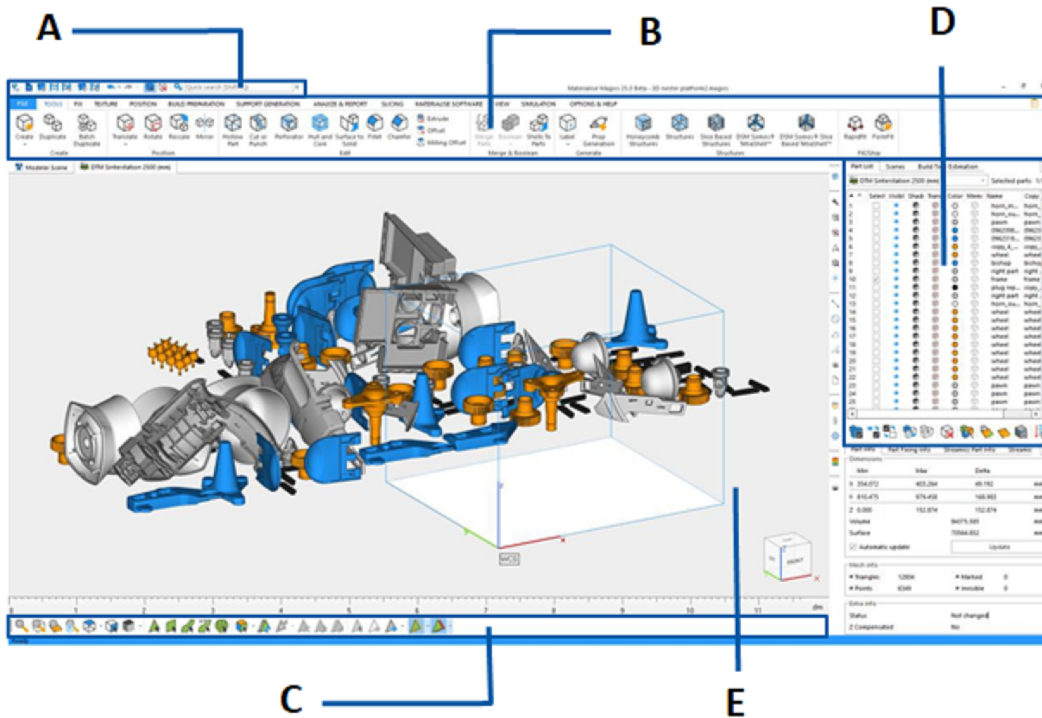
The manual of Magics offers you a clear and easy way to search for more information about a function.

Chapter 3. Magics User Interface

3.1. Overview

You create and manipulate your parts and projects using various elements, such as scene, toolpages, toolbars, and windows.

The following image identifies the main elements of the user interface:

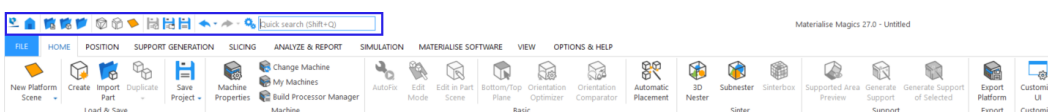


- A: Quick access bar
- B: Ribbon pages
- C: Toolbar
- D: Toolpages
- E: Scene

1. Quick Access Toolbar

The Quick Access Toolbar gives you instant access to a number of often-used options based on the current active scene. The usual commands that are available on this toolbar are commands that helps users to add or remove parts or scenes into the current Magics session along with commands to save parts quickly.

The quick search allows you to find and access any option within Magics without having to search through all the menus.



- See **Quick Access Bar, page 52**

2. The Ribbon pages

Within the Ribbon pages, you can find nearly all the possible options of Magics according to different types of scenes. All options are sorted and grouped in different ribbons in which you can find all the relevant options. To improve the ease of use in Magics we have also created tabbed toolpages. These toolpages contain the most used functions of the Main Menu and show them as small icons.



3. Toolbars

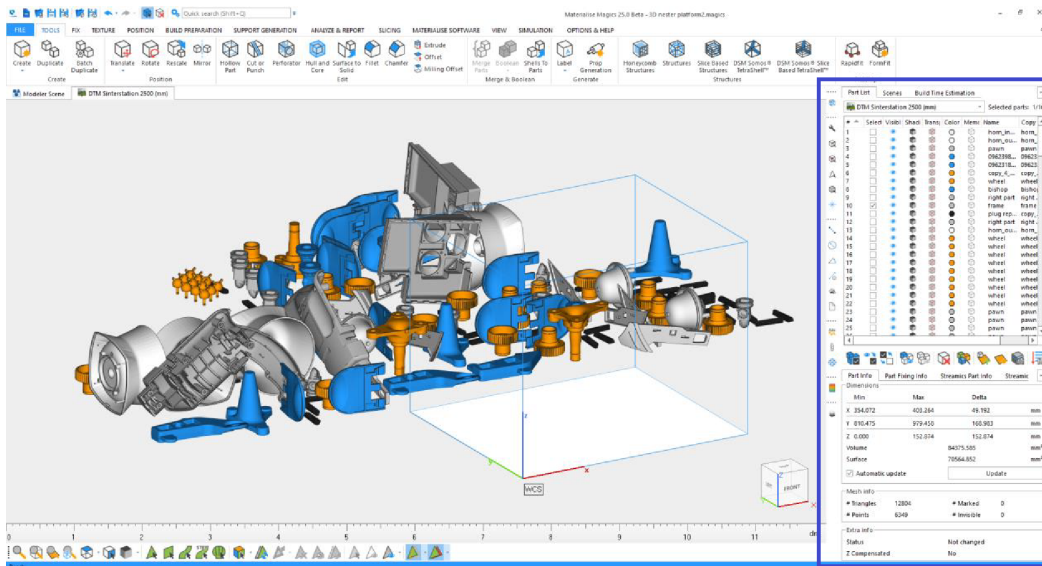
There is one default toolbar available: the general toolbar. This toolbar includes view and marking tools.

The view tools allow you to manipulate the view (e.g. zoom options or standard views) and the visualization of the parts on the scene (e.g. shading modes).

The marking functions allow you to mark a STL file or parts thereof. The marking tool is used in combination with other functions like fixing or editing tools. Due the different marking tools you can mark very fast and accurate, you can choose to mark triangles, planes, surfaces and shells, but you can also mark several triangles by drawing a window, a polyline or polygon.

A user may create custom toolbars.

4. Tool Pages



General Pages

These pages give you an overview on the parts present on the scene and on the build time estimation library. The Part List page provides you a list of all parts on the active scene; in this list you can also hide parts, change the part name (double click on Part name) or show any additional information.

- See **General Pages, page 578**

Part Pages

The part pages show you all the needed information on the selected part on the scene; the Part Info page contains general information about part dimensions, volume, mesh information, etc...; the Part Fixing Info page can be used to analyze possible errors of the selected part and get suggestions on how to fix it.

- See **Part Pages, page 585**

Fixing Pages

You can use the fixing pages to manually fix the different errors of an STL file: near bad edges, holes, overlapping and intersecting triangles, shells, etc...

To fix a corrupted STL file, the trial and error method is used in many cases. Sometimes, the fixing can damage the part, for example when stitching is done with a too large tolerance. That is why it is advised to save regularly the result of a set of fixing operations. This way, the user does not need to start all over when severe damage was done to the part he was fixing. In case you only want to go back on or two steps of fixing, the undo-function can be used.

- See **Fixing Pages, page 590**

Measurement Pages

Magics offers extended measurement functions. It is possible to measure distances or angles between two features like points, lines, circle centers, middle lines of a cylinder, sphere centers, etc... You also can retrieve coordinate information of a feature and create a report based on a template.

- See **Measurements pages, page 604**

Annotation Pages

In the Annotations and Attachments pages you can create scenes wherein you can draw, add comments and load attachments. This can be very helpful in the communication between customer, sales person and production.

The Textures page allows you to add textures on a part. You can load any image format, choose dimensions, direction and position and print the design on the selected triangles of a part.

- See **Annotation Pages, page 614**

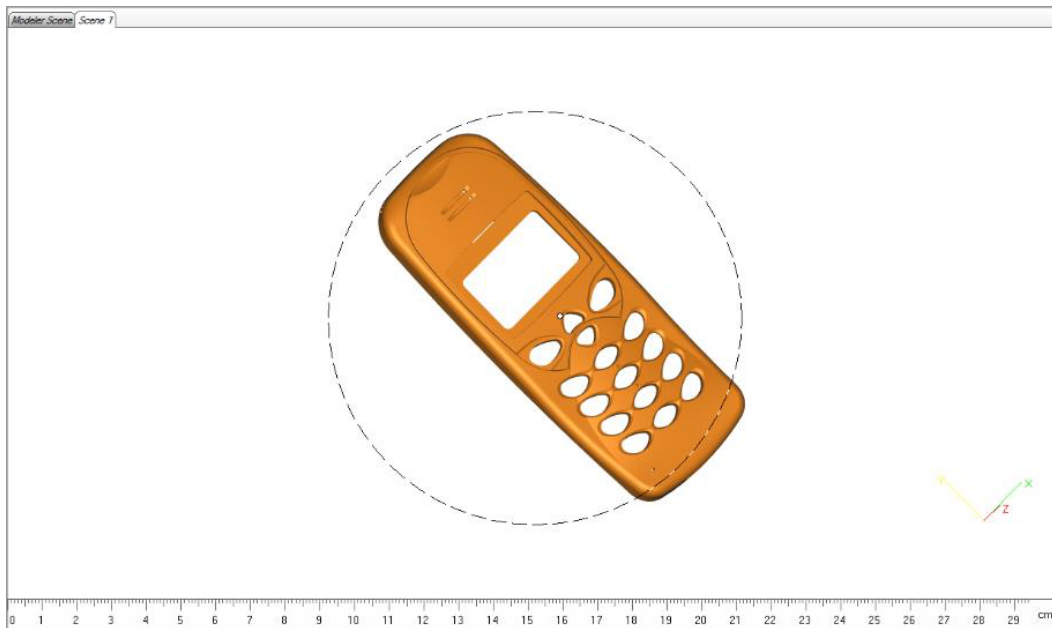
Slices Page

This page offers you the tools to visualize the slices preview of a part loaded on a scene or to analyze a slice file loaded in Magics. Scroll through the slices and visualize the different vector types.

- See **Slices Pages, page 619**

5. Scene

Rotate View



Rotate view mode can be activated by calling the command of the same name to activate the to rotate your scene view freely. When completing the rotation, press Esc key to exit the rotate view mode.

When the rotate mode is active a circle in the center of your workspace is visualized to show which behavior to expect. When the cursor is inside the circle, it will have a quadruple arrow shape (3D movement). When it is outside the circle, it will have a circular arrow shape (2D movement).



Rotate view mode can also be activated temporarily while holding down right mouse button (RMB) on the scene and dragging on the scene.



Multi-plane rotate, Quadruple arrow with ball

The movement of the cursor is translated in a rotation around the three axes in the screen (3D movement). Holding down the RM) while hovering within the rotation circle will temporarily activate the multi-plane rotate view mouse. Holding down and dragging the mouse on the scene, will trigger the scene view rotation.



Note: Holding Alt key while click and dragging with the RMB will allow you to change the view respectively along the vertical or horizontal screen axes.

Screen plane rotate, Circular arrow

The movement of the cursor is translated in a rotation around the axis that is perpendicular to the screen plane (An axes that is parallel to the camera view). Holding down the RMB while hovering outside of the rotation circle will temporarily activate the screen plane rotate view mouse



Panning

The scene view can also be translated along the screen plane freely to pan the scene view quickly. Panning view mode can be activated by calling the command. "Panning" to pan your view freely. When completing the panning, press the Esc key to exit the panning view mode.



Panning view mode can also be activated temporarily by holding down the middle mouse button (MMB) and dragging on the scene to pan.



Pan view, Quadruple arrow

This cursor is shown when the panning view mode is active.



Zooming

The scene view can also be zoomed such that models are enlarged to the your views. It is advised to enable the Rulers as well to ensure zoom factors relative to the actual size of the model can be observed easily. The zoom view mode can be activated by clicking the command of the same name. Once activated, you can draw rectangle box on the scene to zoom in one the rectangle box area.

Alternatively, using the scroll wheel to zoom in and out is preferable the main way to perform zooming of the view.



Part Selection

This is the default mouse mode in a Magics scene. When this mouse mode is active, users will be able to select parts freely by clicking on the parts directly on the scene. Alternatively if the selection points are visible, you can also click on the selection point to select or deselect parts. You can activate this mouse mode manually by calling the Select command.

Select part, Mouse icon with a dot

A left mouse button (LMB) click on parts will select the specific part quickly while deselecting all other parts in the scene. You can also drag select parts based on the selection points to add multiple parts to selection quickly.



Note: If you only want to add or remove some specific parts in the selection, then hold down the CTRL key while clicking on a part with LMB, to toggle the selection state of the specific part while preserving the selection state of all other parts as it is.

To clear selection quickly, you can click on an empty space in the scene to clear all selection. If there is only one part in the current scene, then clearing selection is only possible from the Part list or using the specific command Invert selection.



Note: Slice parts can only be selected from the scene by clicking on the selection points. As such always enable selection points when working with slice parts.

6. Bigger Work Space

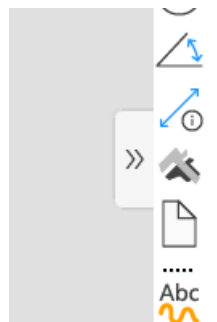
To have a bigger work space, the ribbon pages and the tool pages can be collapsed.

- Collapse ribbon pages

Double click the ribbon group title will collapse or expand the whole ribbon pages. When they are collapsed, clicking on the title will make it appear.

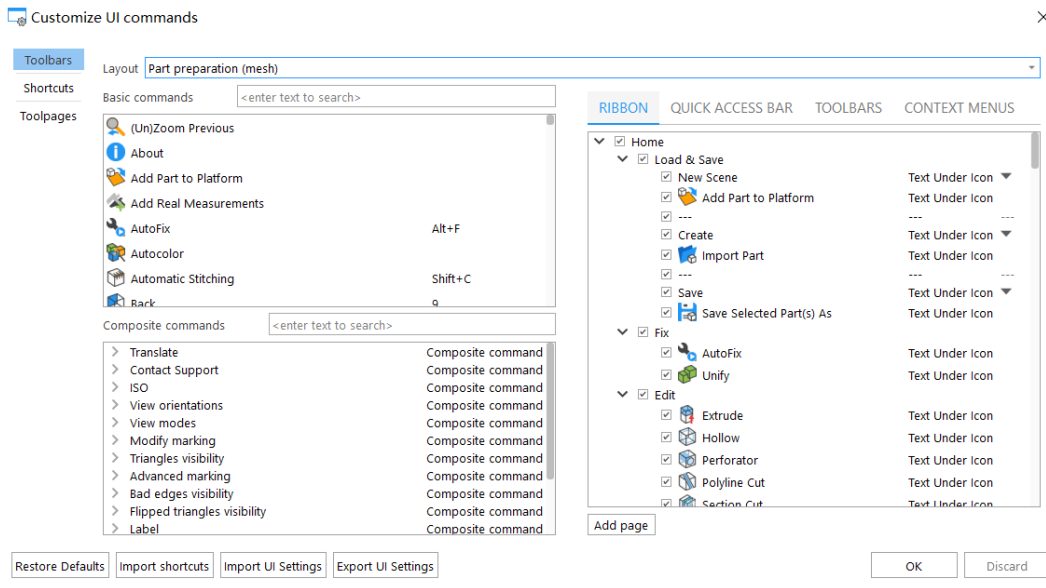
- Collapse tool pages

A button is there to collapse and expand the tool pages.



3.2. Customizations

It is possible to customize the different Ribbon pages, the Quick Access Bar, Toolbars and Context menus.




A ribbon page consists of a number of icons that allow easy access to a particular dialog window. By holding the mouse pointer on an icon, the Tooltip of the function will appear.

1. Customizing Ribbon & Toolbars

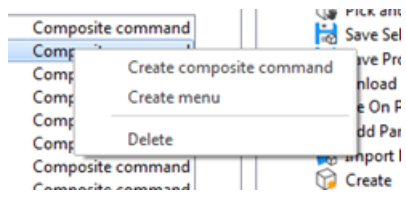
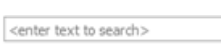
Magics gives the user total freedom to customize the ribbon pages.

- Default ribbon pages
- Custom ribbon pages

The user cannot delete, rename or edit the Build Processor and Plug-ins ribbon pages. The other default ribbon pages can be renamed, edited and deleted. The custom ribbon pages are created, named and defined by the user. These custom ribbon pages can be deleted.

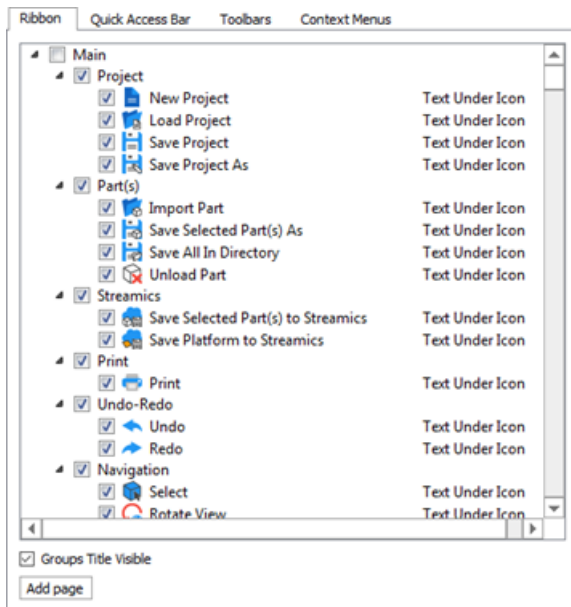
 Remark: All the contents of the "Toolbars & Shortcuts" menus are layout sensitive. As such please ensure the correct layout is set before making any changes.

Commands	Two lists:	
	Basic commands	A list of all the basic commands in Magics. If present, the keyboard shortcut is shown next to the command. Scroll through the list, or use the search box to find a specific command.
	Composite commands	The second list contains all the composite commands. A composite command contains a

	<p>number of grouped functions.</p> <p>Scroll through the list, or use the search box to find a specific command.</p> <p>Make your own Composite commands or menu by clicking the right mouse button:</p> 	
<p>Search box</p> 	Search box for the Basic Commands and Composite Commands lists.	
Rename	To rename, doubleclick on the name you wish to change.	
Delete	To delete a custom ribbon...	
Restore Defaults	Restores the defaults settings for the entire Customize UI window.	
Drag & Drop	The selected function in the Function list of the commands will be added to the Function List of the selected ribbon or toolbar. The selected function in the Function list of the commands remains in this list, so it can be visualized multiple times	
Layout	Depending on the active licences, different layouts are listed.	
	Platform preparation	Target the platform scene.
	Part preparation (mesh)	Targets the mesh part scene.
	Part preparation (b-rep)	Targets theBREP part scene.
	Support generation	Only available when SG/SG+ licenses are available. Targets the SG Mode.
	RapidFit	Available with Fit2Ship. Targets the RapidFit Mode.
	Concept laser	Available with Concept Laser Slicer for the Concept Laser Mode.
Ribbon & toolbars	<p>Four lists:</p> <ul style="list-style-type: none"> - Ribbon - Quick Access Bar - Toolbars - Context Menus 	
	Ribbon	The Ribbon list contains all existing ribbon pages. The visible ribbons and functions have a check.
	Quick Access Bar	The Quick Access bar contains all the functions available via the Quick Access Bar. The visible

	functions have a check.
Toolbars	<p>The Toolbars list contains all Toolbars and their available functions. The visible toolbars and functions have a check.</p> <p>There is one default toolbar: Marking.</p> <p>Other toolbars can be custom made and added by the user.</p>
Context Menus	The Context Menus list shows all available context menus and their available functions.

Ribbon



Checkbox <input checked="" type="checkbox"/>	Checkmark indicates whether an item will be visible or not.
	Adds a separator line before/after functions in the Function List of the selected toolbar.
Visualization	<p>At the right side, the visualization in the ribbon or toolbar is shown.</p> <p>The following visualizations are available:</p> <ul style="list-style-type: none"> - Icon only - Text only - Text under icon - Text next to icon
Groups title visible	When checked, the groups title in the ribbons is visible
Add page	The Add page command adds a custom page, to make a custom ribbon.

– The Default Ribbon pages



The default ribbon pages are standard in Magics. Each groups a number of functions that logically fit together. Some icons may appear on multiple ribbon pages, if they fit in more than one cluster of functions.

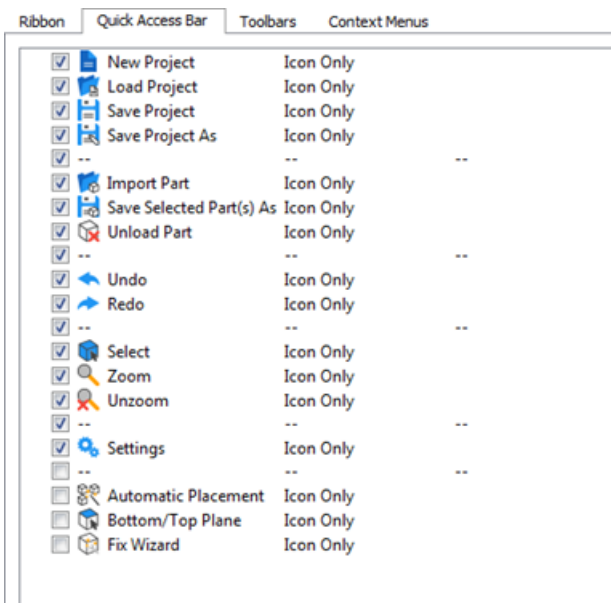
The default ribbon pages are:

- File
- Tools
- Fix
- Texture
- Position
- Build Preparation
- Support Generation (available with SG/SG+ or Tree Supports module)
- Analyze & Report
- Slicing (available with Slice module)
- Streamics (available with Streamics)
- View
- Options & Help

– For more information about these ribbon pages, see **Overview, page 30**.

– For detailed information about every ribbon pages, see **The Ribbon pages, page 31**

Quick Access Bar



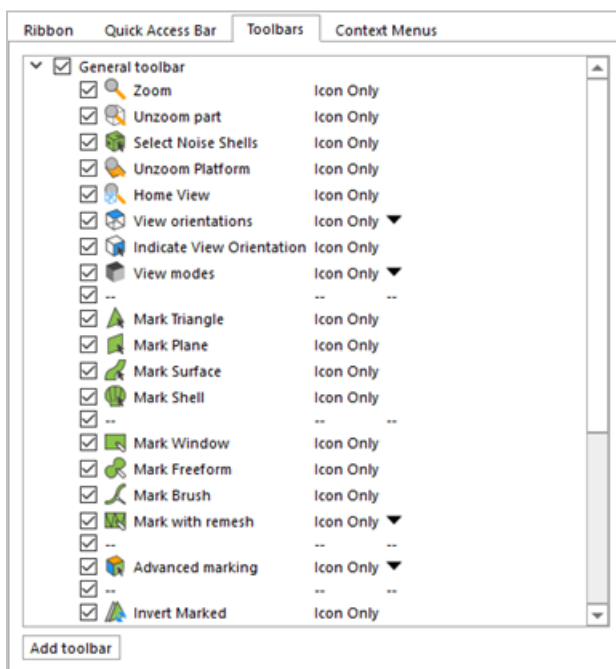
Checkbox <input checked="" type="checkbox"/>	Checkmark indicates whether an item will be visible or not.
Visualization	At the right side, the visualization in the ribbon or toolbar is shown. The following visualizations are available: <ul style="list-style-type: none"> – Icon only – Text only – Text next to icon
Add separator	The Add separator command adds a separator to the quick access bar.



Note: You can also drag and drop commands to the Quick Access Toolbar (QAT) box to add commands that are not added by default to customize your own QAT.

Toolbars

Toolbar menus can be created by the customer according to his needs. The general toolbar is by default present.

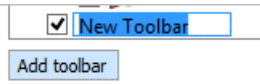


Checkbox <input checked="" type="checkbox"/>	Checkmark indicates whether an item will be visible or not.
Visualization	At the right side, the visualization in the ribbon or toolbar is shown. The following visualizations are available: <ul style="list-style-type: none"> – Icon only – Text only – Text under icon

	<ul style="list-style-type: none"> – Text next to icon
Add toolbar	The Add toolbar command adds a custom toolbar to the toolbar's menu..
Insert toolbar before	Visible when a toolbar is selected. Insert a toolbar before the selected one.
Insert toolbar after	Visible when a toolbar is selected. Insert a toolbar after the selected one.
Delete toolbar	Visible when a toolbar is selected. Delete the selected toolbar.
Add separator	Visible when a toolbar or command is selected. Add a separator into the toolbar.

- Creating a custom toolbar

To create a custom toolbar, proceed as follows:

1.	Click Add toolbar to add a new toolbar.
2.	<p>Enter a name for the new toolbar.</p> 
3.	<p>Add commands by selecting a basic or composite command and dragging it to the new toolbar.</p> <p>By adding a separator, you can make different groups within your toolbar.</p>
4.	<p>Click OK to confirm the creation.</p> <p>The toolbar is now visible in the toolbars at the left side of the work area. The visibility of the toolbar can be altered by dragging:</p> <ul style="list-style-type: none"> – fixed height of the toolbar – Location of the toolbar: below or next to other toolbars.

Context Menus

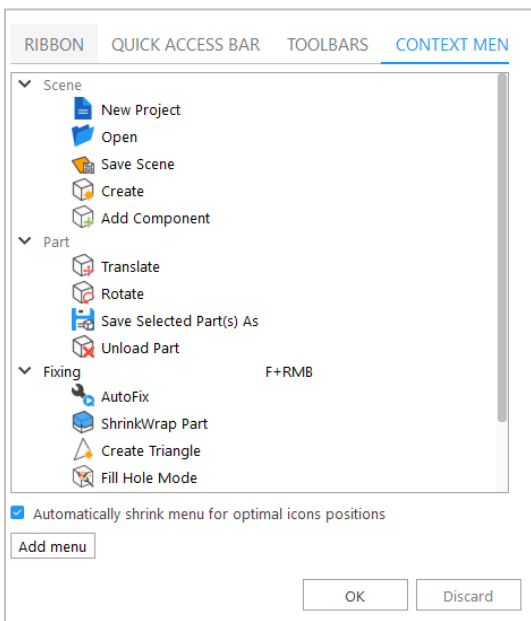
The context menu in the workspace is a quick access point to frequently used function. The usage of the context menu can speed up the general workflow. Depending from where the menu is called, the functions can be different.



Magics gives the user total freedom to customize the menus. Two types of menus are present:

- Default menus
- Custom menus

The user cannot delete nor rename the default menus, the freedom concerning these menus is limited to the removal or adding of buttons in the respective menu. The custom menus are created, named and defined by the user. These custom menus can be deleted.



<p>Default menus</p>	<p>The Menu list contains some predefined menus depending on the target layout and the location of the mouse cursor when summoning the context menu.</p> <ul style="list-style-type: none"> - Scene - Part
<p>Automatically shrink</p>	<p>The menu will be sized according to the chosen functions.</p>

menu for optimal icons position	
Add menu	Initiates the creation of a custom menu, the user has to define a name.

- Default menus

By default there are atleast 2 default menus available.

The predefined menus can be accessed by clicking on the right mouse button along with a shortcut key (if defined).

The context menu shown is sensitive to:

- The target scene
- Location of mouse cursor when the context menu is summoned.



Note: Please check the Customize UI dialog with the desired target layout to identify the default commands available on the context menus.

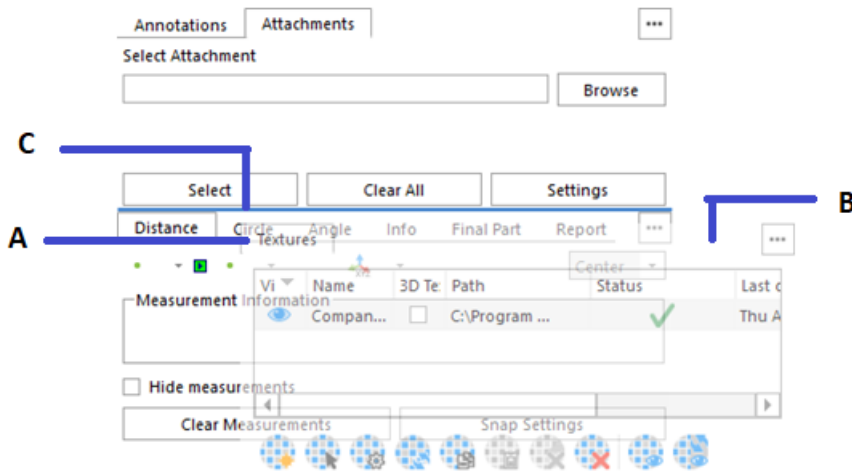
2. Customize Toolpages

You can create a custom workspace by moving and visualizing toolpages, based on your specific needs. This customization can be achieved directly from the workspace or from the Customize UI window.

Move and group toolpages

As you move toolpages, you will see highlighted drop zones, which are areas where you can drop the panel into – e.g. user can move a toolpage up or down in a dock by dragging it to the narrow drop zone above or below another toolpage. The position of the mouse (not the position of the toolpage), activates the drop zone.

- To move a single toolpage, drag it by its tab
- To move a group, drag the tab bar (the solid space next to the last tab)



A: tab – B: tab bar – C: highlighted drop zone

A toolpage can be dragged to an area where there is no drop zone: the toolpage (or toolpages group) floats freely on the workspace – e.g. user can move a toolpage to a second screen connected to the laptop.

You can manipulate a toolpages group:

- To move a toolpage into a group, drag the toolpage tab to the highlighted drop zone in the group (the group tab bar)
- To rearrange toolpages in a group, drag a toolpage tab horizontally to a new location in the group
- To remove a toolpage from a group, drag the toolpage by its tab outside the group

Dock area

A *dock* is a collection of toolpages or toolpages groups displayed together on the extreme left or right side of the workspace. The dock area occupies the entire height of the scene side, and the toolpages are always visible.

You can dock and undock toolpages by moving them into and out of a dock.

- To dock a single toolpage, drag it by its tab into the dock, at the top, bottom or in between other toolpages
- To dock a toolpages group, drag it by its tab bar (the solid space next to the last tab) into the dock, at the top, bottom or in between other toolpages
- To remove a toolpage or a toolpages group, drag it out of the dock by its tab or tab bar. You can drag it back into the dock on another position, into the toolbar or make it free-floating

If you remove all toolpages from a dock, the dock disappears. A dock can be created again by moving a toolpage or a toolpages group to the left or right side of the workspace until a drop zone appears.

Toolpages toolbar

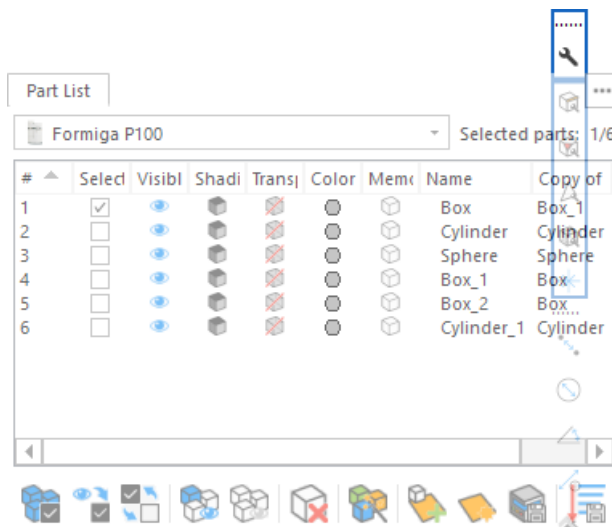
A *toolpages toolbar* is a collection of toolpages or toolpages groups displayed together in between the dock area and the scene, on the left or right side of the scene.

Each toolpages group is identified by a section in the toolbar. At the top of the section there is a dotted line. Each section contains icons, which are linked to each toolpage present in the group. You can click on the icon to show or hide the toolpage.

- To move a toolpage on a toolbar, drag its icon
- To move a toolpages group on a toolbar, drag the section dotted line

You can move toolpages to the toolbar by moving them into and out of a toolbar:

- To add a toolpage to the toolbar, drag it by its tab into the toolbar, at the top, bottom or in between other toolpages
- To add a toolpages group to the toolbar, drag it by its tab bar into the toolbar, at the top, bottom or in between other toolpages
- To add a toolpage or a toolpages group to an existing group, drag it at the top, bottom of a section or in between icons by its tab or tab bar.



- To remove a toolpage or a toolpages group, drag it out of the toolbar by its icon or section dotted line. You can drag it back into the toolbar on another position, into the dock or make it free-floating

If you remove all toolpages from a toolbar, the toolbar disappears. The toolbar can be created again by moving a toolpage or a toolpages group to the left or right side of the workspace until a drop zone appears.

Free-floating toolpages

When user drags a toolpage or a toolpage group out of its dock or toolbar but not into a drop zone, it floats freely. The floating toolpage allows user to position it anywhere in the workspace.

You can stack floating toolpages or toolpages groups so that they move as a unit when you drag the topmost bar.

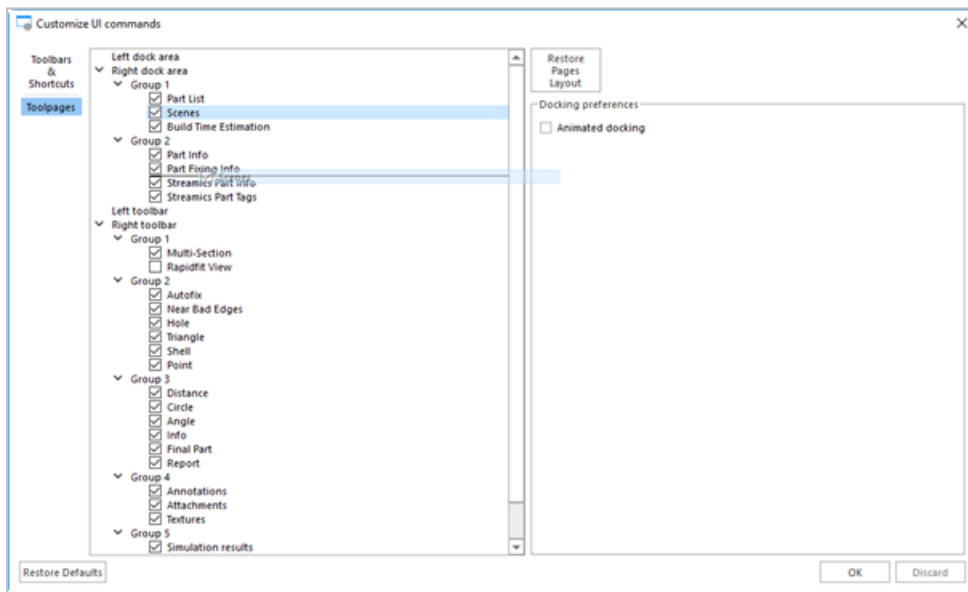
- To stack floating toolpages, drag a toolpage to the top or bottom of the stack or in between other toolpages
- To change the stacking order, drag a toolpage or a toolpages group up or down by its tab or tab bar
- To remove a toolpage or toolpages group from the stack, so that it floats by itself, drag it out by its tab or tab bar

Customize UI window

The following customizations can be achieved from the Customize UI window:

- Control visualization of a toolpages by selecting the checkbox next to it
- Add a toolpage to an existing group by dragging the toolpage item to a group item or in between toolpages items.
- Create a new group by dragging the toolpage item to an area item (dock area, toolbar, floating pages)

The customization is applied automatically to the workspace, and the tree list always represents the current location of toolpages.

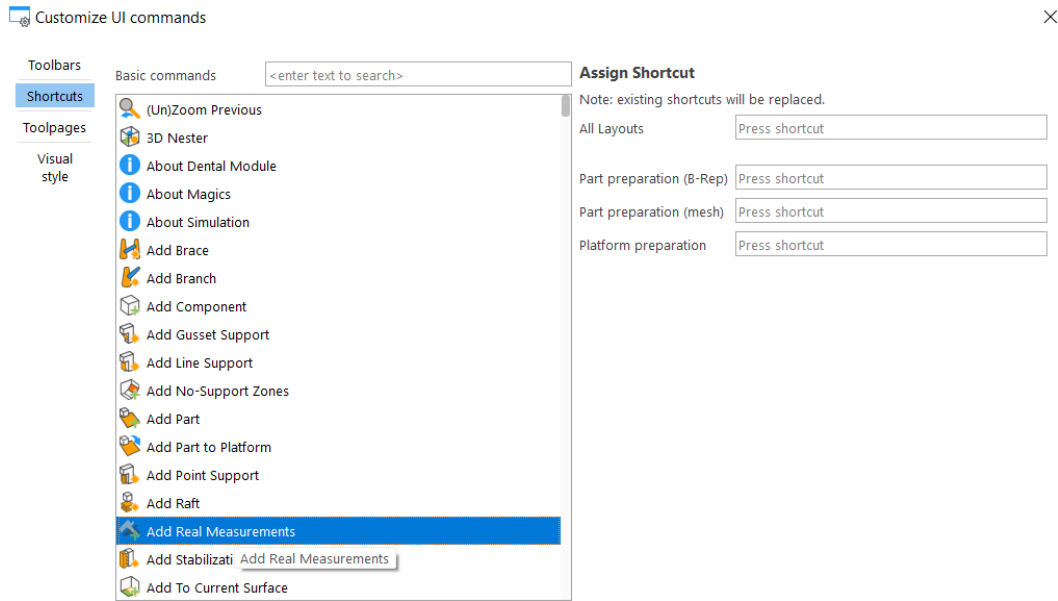


Checkbox	Checkmark indicates whether an item will be visible or not.
Restore Defaults	Restores the defaults settings for the entire Customize UI window.
Restore Pages Layout	Restores the default settings for the position and visibility of the pages.

3. Customize Shortcuts

Magics gives the user the freedom to assign shortcuts to frequently used functions. The user can define shortcuts for functions that by default do not have a shortcut, or already predefined shortcuts can be changed to the needs of the user.

Changing shortcuts for different layouts is available in the Shortcuts page.



Default shortcuts overview

– General

Manual	F1
Settings	F12
Undo	Ctrl+Z
Redo	Ctrl+Y
Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Select	F2
Group	Ctrl+G
Ungroup	Shift+G
Customize UI	Alt+C
Print	Ctrl+P
Switch Units	Ctrl+Alt+1

– File

Home	Ctrl+N
Open	Ctrl+O



Load project	Ctrl+L
Save Scene As	Ctrl+Shift+S
Save Selected Part(s) As	Ctrl+S
Rename part(s)	Shift+R
Unload Part	Ctrl+U

– View

Back	9
Front	7
Left	4
Right	6
Top	8
Bottom	2
Home View	Q
Zoom	Alt+Z
Zoom In	Ctrl++
Zoom Out	Ctrl+-
Unzoom part	Alt+U
Tag ID	F9
Tag Names	F10
Tag Path	F11

– Align

Translate	T
Translate to Default Z Position	Home
Original Position	Ctrl+Shift+P
Rotate	R
Pick and Place Parts	F3
Mirror	Ctrl+M
Automatic Placement	Ctrl+A
3D Nester	Ctrl+3

– Mark

Mark Triangle	F5
Mark Plane	F6
Mark Shell	F7
Mark Window with Remesh	Alt+Shift+R
Shrink Selection	Down
Expand Selection	Up
Invert Marked	O
Unmark All	F8
Hide Marked	Ctrl+Shift+H
Make All Visible	Ctrl+H
Invert Triangle Visibility	Ctrl+I
Delete Marked Triangles	Del
Copy Marked	Alt+Shift+D
Separate Marked	Alt+Shift+X

– Fix

AutoFix	Alt+F
ShrinkWrap Part	W
Normals Fix	Shift+N
Automatic Stitching	Shift+C
Stitch (manual)	Shift+E
Holes Fix	Shift+H
Fill Hole Mode	Ctrl+Shift+B
Shells	Shift+S
Noise Shells	Shift+I
Triangles	Shift+T
Filter Sharp Triangles	Shift+F
Detect Intersecting	Ctrl+Shift+Q
Detect Overlapping Triangles	Ctrl+Shift+O
Overlaps	Shift+O
Create Triangle	Ctrl+Shift+E
Delete Triangle	Shift+D
Create Bridge	Ctrl+Shift+Z
Triangle Reduction	Ctrl+T



– Edit

Boolean	Ctrl+B
Cut or Punch	C
Duplicate	Ctrl+D
Extrude	Ctrl+E
Rescale	Ctrl+R
Edit Mode	E
Exit Edit Mode	E

– Analyze

Wall Thickness Analysis	Shift+W
Measurement Thickness	Ctrl+Shift+C
Measurement Distance Point to Point	Ctrl+Shift+X

– Slice

Slice Preview	Alt+P
Slice Selected	Alt+I

– Support Generation

Generate Support	S
Generate Support of Selected	Shift+S
Manual Support	Alt+S

The logo graphic consists of two overlapping triangles. The top triangle is a light blue color and points downwards. The bottom triangle is a darker blue color and points upwards. They meet at a horizontal line that is slightly above the text.

materialise
innovators you can count on

Part II: Magics Features

Chapter 1. Quick Access Bar

Most commonly used command to load, save and undo redo projects will be found in this section. The commands placed in this section can be customized to add or remove commands from the Customize UI dialog.

- See **Quick Access Bar, page 39** for more information on the customization of Quick Access Toolbar.

The general commands that can be found in this section are:

1.1. Home



Close all the open scenes and return to home screen. (CTRL+N)

- See **Home, page 55**

1.2. New Project



Start a new untitled project with default scenes and parts from a Magics project template file.

- See **New Project, page 56** for more information.

1.3. Load Project



Load an existing project. (CTRL+L)

- See **Load Project, page 57**

1.4. Import Part



Import existing 3D model(s) to the current active scene.

- See **Import Part, page 58**

1.5. Open



Open existing file(s) in new scene(s) (CTRL+ O)

- See ***Open***, page 56

1.6. New Mesh Part Scene



This command will create a new mesh part scene which then you can load mesh parts in for part preparation.

1.7. New BREP Part Scene



This command will create a new BREP part scene which then you can load brep parts in for part preparation.

1.8. New Platform Scene



This command will show the New Platform dialog that allows user to specify the target machine for a new platform scene.

- See ***New Platform Scene***, page 289

1.9. Save Selected Part(s) As



Save the selected part(s) and select the desired name, file format and location. (CTRL+S)

- See ***Save selected part(s) As***, page 73 for more information.

1.10. Save Scene As



Save the current active scene and select the desired name, file format and location.

- See ***Save scene As***, page 75 for more information.

1.11. Save Project



Save the current project. (CTRL+SHIFT+S)

1.12. Undo



With this command you can undo the previous action. All actions that change the STL file will be noted in a list, the Log Window (Menubar/View/Log Window). In case of a computer-crash when Magics is open, you will be able to recover the work you did before (auto-recovery). (CTRL+Z)

The undo and auto-recovery functions are default ON. If you would like to change this, go to Settings (Settings > General > Undo and Recovery)

1.13. Redo



The actions that were undone by the undo operations can be redone by redo. (CTRL+Y)

1.14. Settings



Change Magics' settings

- See **Settings, page 512**

1.15. Quick Search



Search a functionality within Magics. Start entering the name of a functionality, and all relevant functionalities are instantly shown. Click on the desired functionality to directly activate. (SHIFT+Q)

Chapter 2. File

File menu provides a set of commands allowing you to open or load .magics files as well as other supported formats, save them, going back to the home screen when the workflow is finished. Also reporting functionality, machine information and access to the Magics options is possible from there.

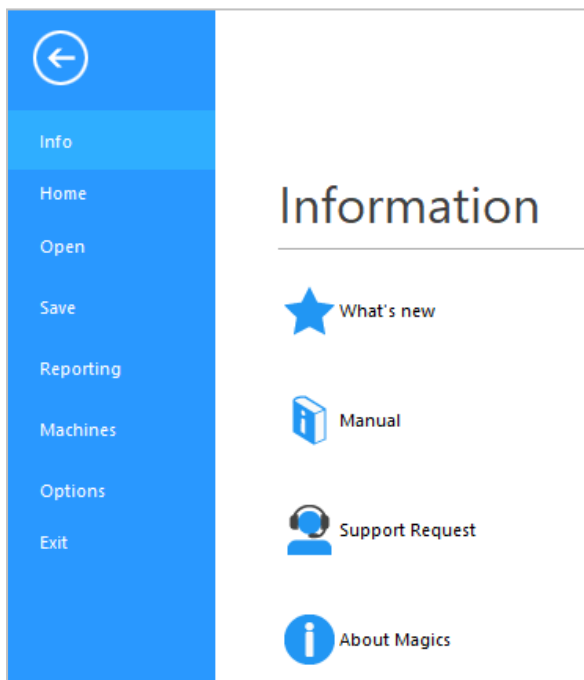
The File screen:

2.1. Info

In this section you can find more information on the Magics software:

- What's new: the new features in your Magics version, compared to previous versions
- Manual: Reference manual on the Magics software
- Support Request: contact info to request support from Materialise or its affiliates.
- About Magics: info about your currently installed Magics version and its active licenses.

The features of this section are equally available via the Options & Help ribbon.



2.2. Home



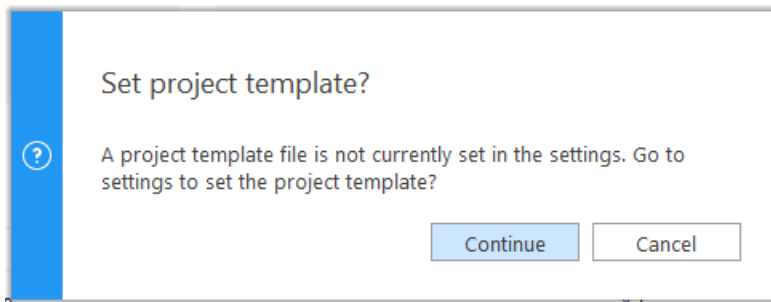
This command removes all parts and scenes in the current Magics session and shows the Home Screen. If the project is not saved, then the user is prompted to save the project (See **Save Project, page 72**) before all the contents of the session are cleared.

2.3. New Project



The New Project command will close the current loaded project and start a new untitled project with default scenes and parts based on a Magics project template file. If an existing project is already loaded, you will be prompted to save the project and proceed to start the new untitled project.

By default creating a new project from a template file is disabled with a clean installation. This can be enabled by modifying the Project Template settings in the settings dialog. When you call the command without enabling this setting, a message dialog is shown and you can navigate to the settings page automatically from the dialog.



Clicking continue in this dialog will open the Project Template preference page in the Setting dialog automatically so that you can make the necessary changes to enable the new project command.

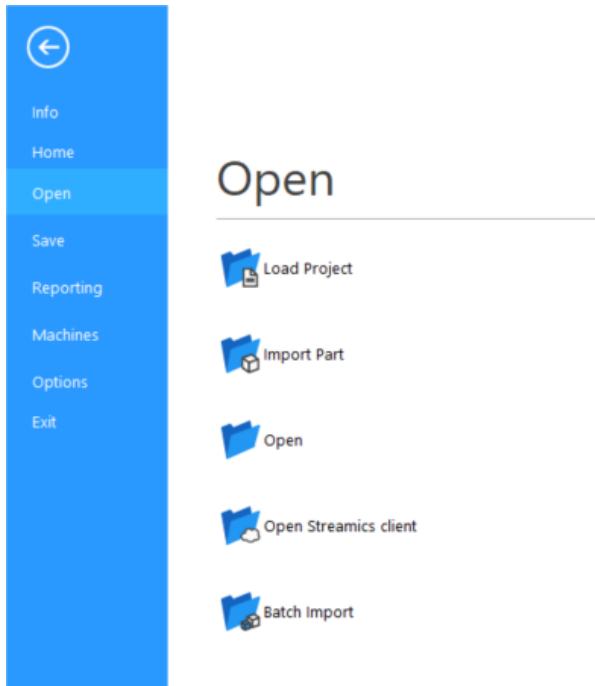
- See [Project Template in Settings](#) for more information on settings page for project template.

A project template file should be a valid Magics project file and all information that can be saved to a project file can be loaded back automatically through the new project command.

- See **Save Project, page 72** for more on information that can be saved to a Magics file.

2.4. Open

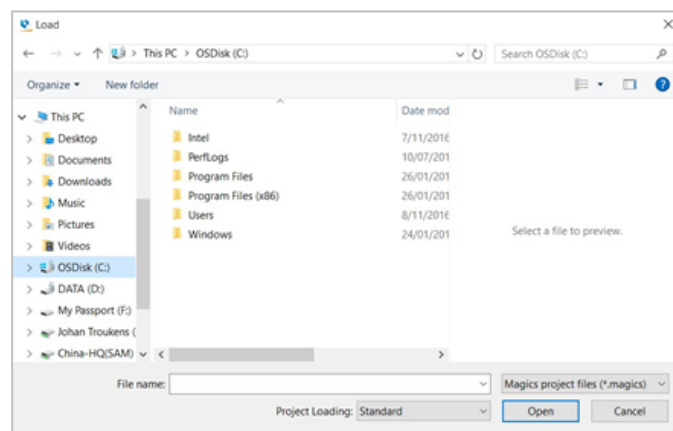
Via this section you can load a project or import a part into an open project. You can also open a Streamics client.



1. Load Project



The Load Project command starts the standard dialog box to load files.



Only *.magics format files can be loaded as project files in Magics.

The memory state of the loaded project can be defined, the following states can be chosen:

Standard	This is the standard memory state of a STL file. Magics knows the placement of the triangles and the mutual dependencies of the triangles. The user is able to perform actions on STL level (E.g. deleting triangles)
Compact	The STL resides in the memory as read-only, therefore it uses far less memory than the Standard memory state. Magics does not know the placement of the triangles nor the mutual dependencies of the triangles. The user is not able to

	perform actions on STL level.
On Disk	The STL is saved on disk and unloaded from the memory. The STL will stay in the project but the user can't perform any actions on it.
As Saved	The Project will be loaded as previously saved.

The default memory state for the loaded project is defined in the Settings -> File I/O -> Import -> STL.



A Magics file can be loaded as a project through drag and drop to the home screen, the application executable or application shortcut. The memory state defined in the settings menu will be used automatically when dragging and dropping.



If the latest version of Magics is set as the default app for .magics files then Magics file can be loaded as a project also by double clicking on the file in the Windows file explorer. See **Setting the default application to open specific file formats, page 628** for more information.

All the saved contents can be loaded back when loading a project file along with the original path of the file to ensure the file can be updated quickly with new or changed information.

- See **Save Project, page 72**

2. Import Part



The "Import Part" command would allow you to import parts from all supported files into the current active scene. This command will evaluate the file formats and the contents of the file to ensure only supported file formats and part types can be imported to the respective active scenes.

- When importing parts from specific file formats to a platform scene, virtual copies in the file can be preserved as is, see **Import Part(s) from Materialise, MGX and 3mf file formats, page 60**
- When importing parts from CAD files into a BREP Part scene, the BREP Import part dialog will be shown parameters to customize, see **Opening CAD files as BREP Part (s), page 64** for more information on the dialog parameter

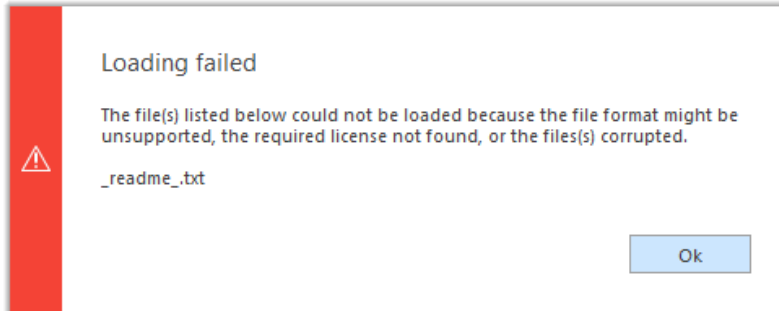


Note: You can drag and drop files to the scene view port area to import parts quickly to the active scene.



Note: In Magics RP28, any CAD files loaded through the "Import Part" command on any platform scenes will trigger the MatConvert CAD to Mesh conversion automatically.

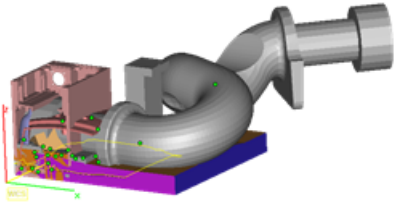

Note: When dragging and dropping unsupported files, the following message will be shown to the user through any file loading flow.

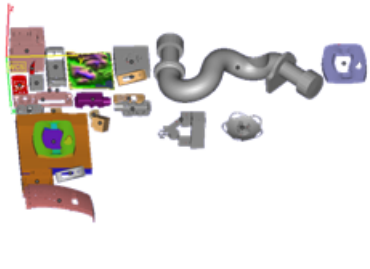



Part Placement and Orientation

When importing parts into the current active scene, user can specify the position of the part relative the active coordinate system. This options can be set in the Import Part dialog or the Settings page for File IO.

Only the position of the part will be changes and not the orientation based on this part placement values.

As Is	The original part position is maintained as it is.
Default Position	<p>The part is placed at the default position. This default position is defined in the Machine Properties and represents the minimal X, Y and Z of a part. (Default position in Part Scene: Xmin = 0mm; Ymin=0mm; Zmin = 0mm)</p> 
Aside Of Others	<p>Parts are loaded one after the other while the original Y-position is maintained. If a line is full, a new line is started.</p> 
Automatic Placement	<p>The part(s) will be added using automatic placement. The parts that already are loaded will not be moved. This can be done later by selecting all the parts and using Automatic Placement of the Tools menu. Changing the settings for automatic placement can be done in the Nesting settings (see Parts placement page, page 296).</p>





Note: Automatic Placement method is only available when importing parts into the platform scene.

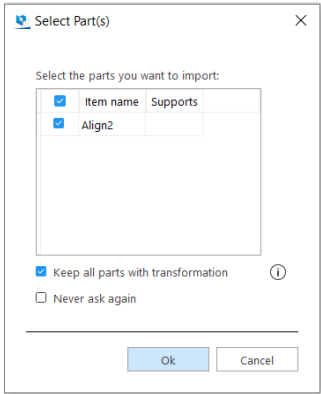
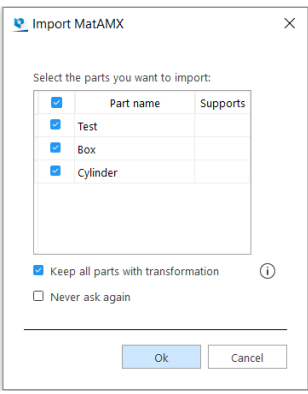
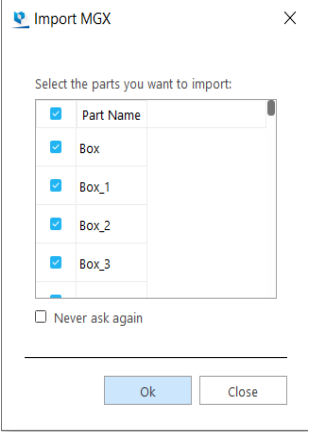
When the imported part position or size exceeds the expected range of position and size value, then either the Rescale large parts or Move part into the workspace dialog is shown.

- See **Rescale large parts, page 63** for more information on parts larger than the expected size.
- See **Move part into workspace dialog, page 64** for more information on parts placed outside of the expected position range.


3. Import Part(s) from Materialise, MGX and 3mf file formats

When loading Materialise native file formats or 3mf file formats through the "Import Part" command, a special dialog is shown to allow users to select the specific parts to be loaded in the active scene.

You can specify the specific parts that you want to load into scene along with related supports or copies that may exist in the file.

Importing from .magics or .mxp files	Importing from .matamx and .3mf files	Importing from MGX files
		

Select Part(s)	This list contains all the unique parts that are available in the target File.		
	<table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 30%; padding: 5px;">Item name</td> <td style="padding: 5px;">The unique part name in the file. The user is able to select the parts that he wants to load.</td> </tr> </table>	Item name	The unique part name in the file. The user is able to select the parts that he wants to load.
Item name	The unique part name in the file. The user is able to select the parts that he wants to load.		

	Supports	This column displays whether there is support for the part. The user is able to import the support for every single part.
Keep all parts with transformation		<p>If True, then all the design and platform copies of the selected unique part(s) that may exist in the file will be loaded with its position preserved relative to the active coordinate system. Default is True.</p> <div style="border: 1px solid black; padding: 5px;">  Note: Virtual copy relationship will only be preserved as is when adding to a platform scene. </div>
Never ask again		This dialog is never shown again in the future when loading similar files. All parts will be loaded automatically with all the last set loading parameters remembered.

File formats that triggers this dialog are:

- i. Magics project file (.magics)
- ii. Materialise AM Exchange File - MatAMX (.matamx)
- iii. Materialise 3-matic project file (.mxx) (without support for Virtual copies)
- iv. 3D Manufacturing Format - 3mf (.3mf)
- v. MGX files



Note: for MGX files protected with a password, the pop up dialog is given where the password must be entered first. Once succeeded, the content of the file is imported to the active scene.

4. Open



The "Open" command would allow you to open one or more files of all supported file types quickly into the target Magics session. The open command will evaluate the contents of the files and open the appropriate scene(s) that are available in the target files. The open command will always result in one or more scenes created in the target Magics session.



Note: Opening .magics, .matamx or .3mf is no longer considered as targeting a project document for the target Magics session, instead only the contents of the files are appended to the session. The existing content in the target Magics session will stay unmodified by the open operation. To ensure that your current full Magics session is targeting a Magics project file, please save the project once upon loading or load magics files via 'Load project' command.



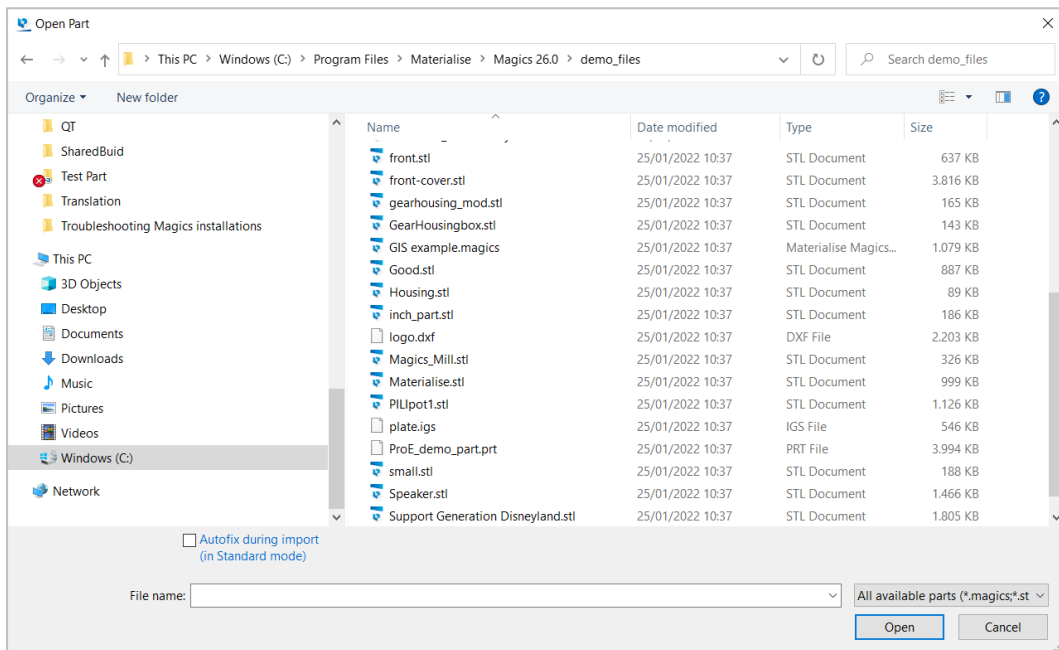
Note: You can drag and drop files to the Home Screen or the Scene Tab-Bars (while Scenes are active) to open files quickly.



If a single Magics file is dropped to the Home Screen, the file will be loaded as a project automatically.



Note: If you open a slice file that contains no platform information, you will need to select a new platform in Magics in which the slice stacks will be opened. If you want to open slice stacks in the active platform, do load the file via "Import Part" command on an active platform scene or simply drag and drop on the platform directly.



For supported types of files, see **MatConvert, page 525**



Remark: All supported formats for open can be loaded through a drag and drop action as well.

Part Memory Modes

The memory state of the loaded Mesh part can be defined, the following states can be chosen:

Standard	This is the standard memory state of a STL file. Magics knows the placement of the triangles and the mutual dependencies of the triangles. The user is able to perform actions on STL level (E.g. deleting triangles).
Compact	The STL resides in the memory as read-only, therefore it uses far less memory than the Standard memory state. Magics does not know the placement of the triangles nor the mutual dependencies of the triangles. The user is not able to

	perform actions on STL level.
On Disk	The STL is saved on disk and unloaded from the memory. The STL will stay in the project but the user can't perform any actions on it.

The default memory state for the loaded part is defined in the Settings -> File I/O -> Import -> STL.

– See **Settings, page 512**



Remark: BREP models can only be loaded in Standard memory state.

Rescale large parts

When opening or importing a mesh part into Magics that is too large for the application scene and operation, a rescale large part dialog will be shown to let the user choose an appropriate rescale factor for the large part.

Rescale Large Part

Part from "Box very very big from 3-matic.stl" is oversized.
Dimensions of the part exceed the dimensions of the workspace.
Part needs to be rescaled by a factor to ensure the part will fit in the workplace.

Rescale factor by percentage: 99.99 %

Appromixated bounding box sizes

	Original	mm	Rescaled	mm	
X	400000	mm	400000	mm	✗
Y	61584	mm	61584	mm	✓
Z	61584	mm	61584	mm	✓

Update part name with rescale naming template
 Apply to all

Ok Close

Rescale factor by percentage	Rescale factor as a percentage value. Changing the rescale factor will update the check or cross mark labels automatically to indicate the if the rescale factor is acceptable.
Original bounding box	Approximation of the current size of the part's bounding box
Rescaled bounding box	Approximated of the rescaled part's bounding box. The cross or check marks indicates if the rescaled bounding box is within the acceptable range of part size. If cross mark is shown in any one of the sizes, then import will blocked automatically.
Update part name with rescale naming template	The used rescale factor will be appended to the part name upon import as per the defined rescale naming

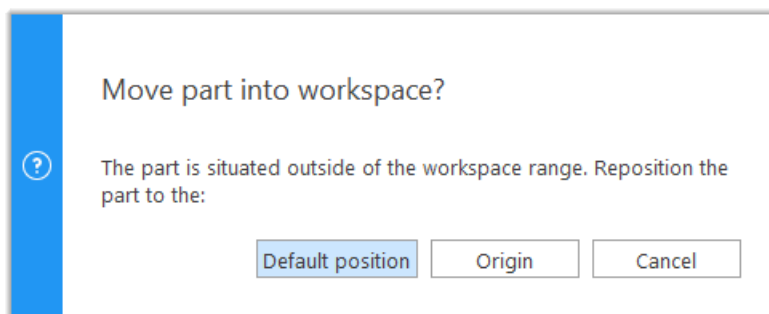
	template.
Apply to all	The same rescale factor will be used for other upcoming oversized parts
Ok	The part is rescaled and loaded into the application
Close	The part is not rescaled and loaded into the application.

Part Placement and Orientation

When opening files the original position and the orientation of the parts are preserved as is in the final loaded part as it is.

Move part into workspace dialog

An exception to the preservation of the original position of the part is when the part is placed very far away from the World coordinate system such that the part points exceed the expected range of part position within the application. When such parts are loaded, the move part into workspace message dialog is shown.

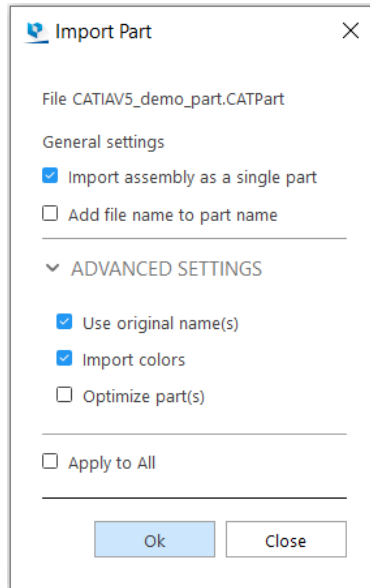


User can specify to which position should the part be moved into. If the user selected the Default Position option, the part's minimum bounding box will be moved the default position of the scene.

Alternatively if the user specified the Origin option, the part center will be moved to the origin of the scene. Cancel will abort the open process and not load the part into the application.

Opening CAD files as BREP Part(s)

Opening a BREP file triggers import parameter dialog to pop up.



<p>Import assembly as a single part</p>	<p>If checked, each assembly in the file will be preserved as a unique part in Magics, with its whole assembly structure preserved.</p>
<p>Use original part name(s)</p>	<p>If checked, all the assembly and its components names available in the file will be used to name the parts and its components in Magics. If unchecked, Magics will assign the appropriate names to the entities. The top most level will be part, intermediate levels subpart and lowest level (bodies) as component.</p>
<p>Add file name to part name</p>	<p>If checked, file name is automatically added as a prefix to part name, separated by a dash "-" (e.g filename_partname). Only applied to the top most level entity. <i>Note: If the "Use original part name (s)" is unchecked while this parameter is checked then the part name will just be the file name.</i></p>
<p>Import colors</p>	<p>If checked and color definitions are present in the file, then the face colors are preserved as in the file, otherwise the part color is used to render all the faces.</p>
<p>Optimize part(s)</p>	<p>If checked on [default] the imported BREP part(s) will be optimized for better geometry: Example of optimization:</p> <ul style="list-style-type: none"> • Cylindrical/Conical faces are identified and made into a single face whenever possible • Redundant smoothly connected

	<p>faces are always merged into a single face whenever possible</p> <p>Note: on complex parts optimization can be time-consuming. If it takes too long to import your BREP part in Magics, it's advised to turn it off</p>
Apply to All	<p>Selected parameters will be applied to all parts that are loaded. If unchecked, an import dialog will be shown for each file consecutively.</p>



Note: When opening or importing parts, the parts name in the application will be made unique by appending an index to the part name if an existing part already exists with the same name.

5. Open Streamics client



Opens the associated Streamics Clients. The client can be associated in the settings.
(Settings – Modules – Streamics Client)

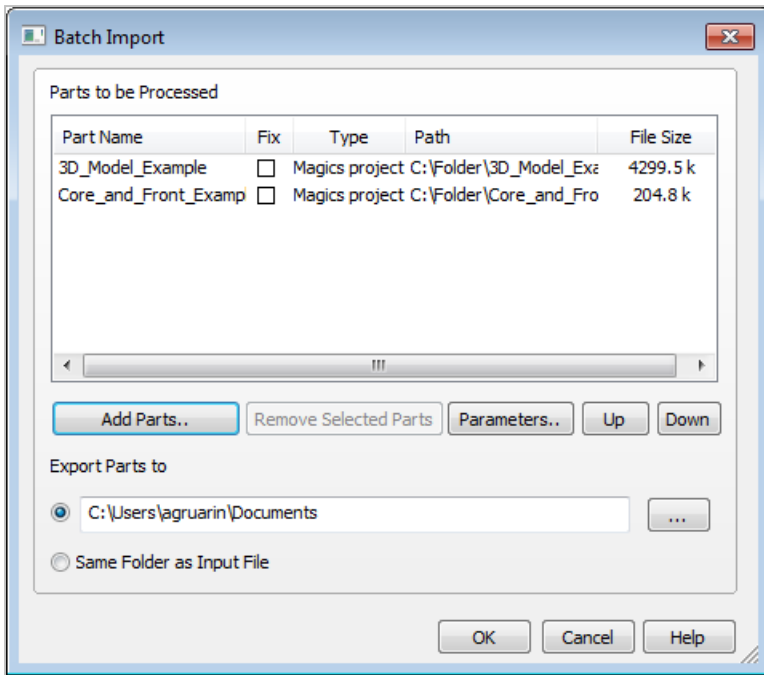
6. Batch Import



This command loads automatically CAD, STL, MGX and *.MAGICS files on the current platform from a selected location. All CAD files that are loaded are translated into STL-files. If no machine is selected, the part can be processed also. To load several parts at the same time, the CTRL or the Shift button are used. In this list, you can select parts by highlighting a row. The same type of files can be loaded as with the import part.

This command can save you a lot of time. For example: You need to load a large amount of files. You can specify which files need to be loaded, the files can be loaded overnight, and the next morning all files can be imported in Magics.

This command is only available with an additional license.



Parts to be Processed	The list contains all parts that you added to the queue. Magics will convert these files from top to bottom	
	Part name	The name of the part.
	Fix	When the checkbox is checked, the part will be fixed after importing, using the parameters of the parameters Dialog box. The best is to use this function only for parts you know. The automatic fixing can change the geometry of a part.
	Type	The type of CAD-file.
	Path	The location of the CAD file.
	File Size	Size of the selected part in KB.
Add parts	With this button, you can add a CAD-part to the list. A dialog box opens, where you can select the parts you want to add to the import-list.	
Remove selected parts	Use this button to remove the selected files from the list.	
Parameters	Opens the parameters-Dialogue box	
Up	Move a part upwards in the list so it will be converted earlier.	
Down	Move a part downwards in the list so it will be converted later.	
Export parts to		
	Specify a folder to export the file to. Use Parameters to set its default value	

Same folder as Input File	as	Export the files to the same folder as the Cad-file
---------------------------	----	---

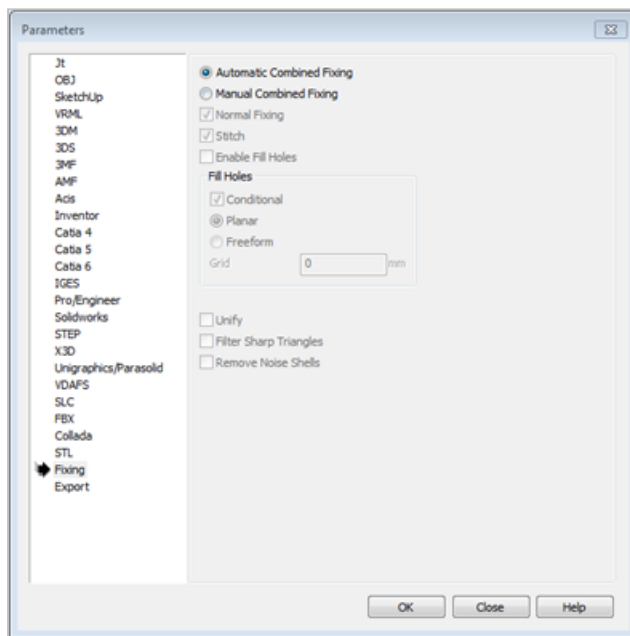
Parameters

1. Import parameters

Here you can specify the parameters for the files you want to load. You can apply these parameters for all the files in the list.

- See **Open, page 61** for the file types.

2. Fixing parameters



Here you can select in what way the file is fixed after it's imported.

There are two fixing methods available: automatic or manual combined fixing.

- Automatic Combined Fixing

If selected, Magics will do a predefined list of fixing actions. Some actions are conditional, this means that the action will only be performed when Magics is sure that the result will be ok.

As mentioned above, be careful with this option. Only use it on parts you know.



– Manual Combined Fixing

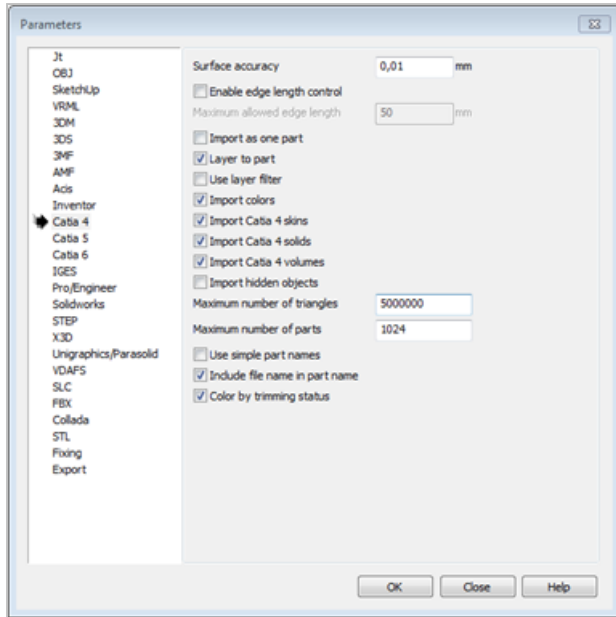
Instead of the fully automatic fixing algorithm, it might be safer to use a manual selection of fixing functionality.

The user can decide which fixing functions he wants to apply.

Normal Fixing	A normal fix is applied.
Stitch	A stitching is applied with the estimated stitching tolerance
Fill Holes	All contours that seem to be holes are filled automatically. (Type of hole fixing see combined fixing)
Unify	This will remove all internal geometry and intersecting triangles. This operation will only be done if the geometry allows it.
Filter Sharp Triangles	Sharp triangles will be removed to improve surface quality.
Remove Noise Shells	The automatic removal of detected noise shells, these noise shells make no geometrical sense: they are not connected to the bigger part, nor have a volume.

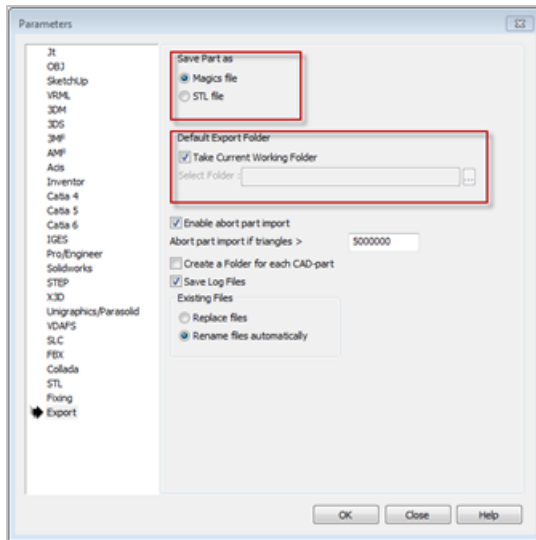
3. Export - parameters

Here you can set how the converted CAD-files are saved to disk.



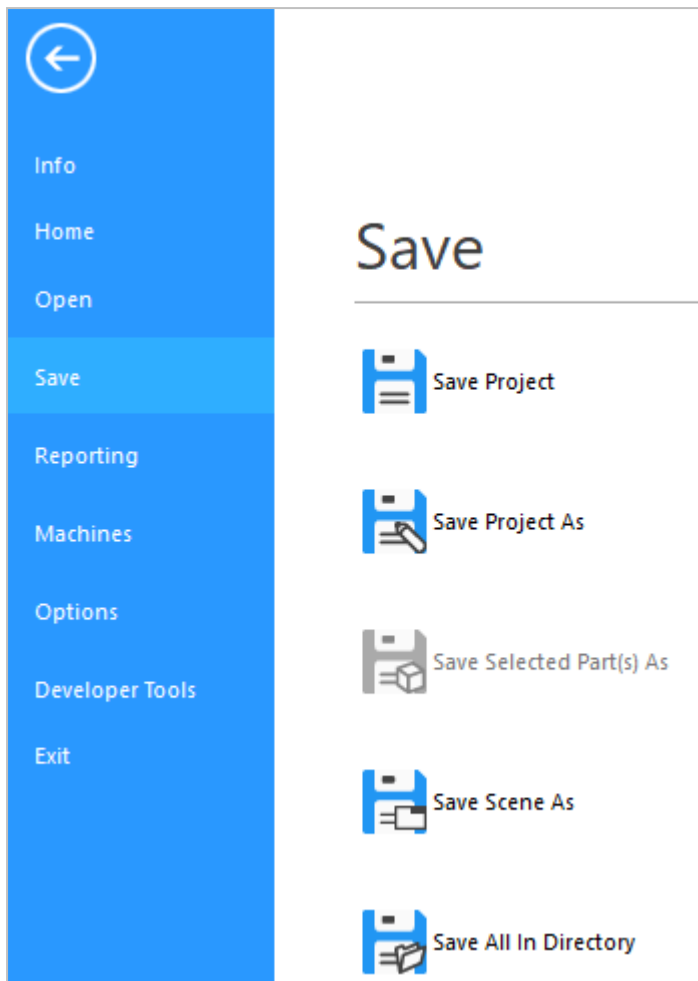
4. Default Export Folder

Default Export folder	
	<p>Take current working folder</p> <p>Export the files to the current working folder.</p> <hr/> <p>Specify a folder to export the file to. Each time, it will take this folder.</p>
Abort part import after ...	<p>Set a limit of triangles. When the limit is reached, Magics will stop the import and moves to the next part in the queue. This option can be set when you need to load a very large file, to avoid that all the memory is being used.</p>
Create a folder for each CAD-part	<p>For each imported part, a folder is created. The name of the folder is the name of the CAD-part.</p>
Save log files	<p>For each part, a log file is saved in the same folder as the Magics file (or STL) is saved.</p> <p>This log file contains:</p> <ul style="list-style-type: none"> - the CAD File - To which version it is converted. - When it is converted - Result of the batch import
Existing Files	
Replace files	<p>Files that already exist in the chosen folder will be overwritten.</p>
Rename files automatically	<p>Files that already exist in the chosen folder will be renamed automatically.</p>



2.5. Save

Via this section you can save your opened project, select part(s) or scene. You can save selected part(s) or the platform to Streamics.



1. Save Project



When you have prepared a project in Magics, you can save it to disk in order to load it again later on.

Magics Project Files

The Magics Project File is a dedicated file format for Magics. The Project File can contain information concerning:

- Mesh Parts and Composed Mesh Parts
- Slice Parts
- BREP Parts and Assemblies
- Support
- RapidFit
- Measurements and annotations
- Machine information
- Annotation and Platform Scenes

The user is able to save a whole project instead of saving all parts and supports separately. Another advantage of the Magics Project File is that the information is stored together with the preprocessed data, so the loading of a project file is much faster than loading the individual parts. The Magics Project File format is also a compressed format, dedicated for the RP industry.

For support, Magics will automatically extend the project name with '_sup' if this option is checked in the Settings. So the reader is noticed that a file contains support data.

– See **Settings, page 512**

2. Save Project As



Save Project As allows you to save the project with another name, it contains the same functionality as Save Project.

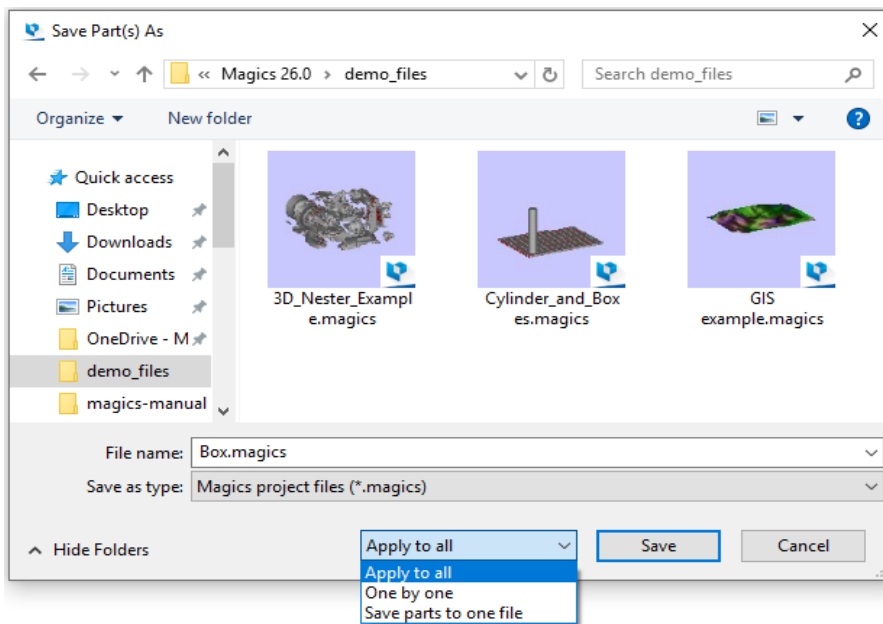
3. Save selected part(s) As



This command allows you to save all the selected parts quickly. The destination of the saved parts can be changed when using this option. Each part can be saved either in a separate files or to the same file with the option "Save parts to one file".

Note: be aware that not all formats support the option to save parts to one file.

The supported target files to save to may differ per type of the selected part.



The file can be saved to the following file types:

X - Available | O - Not Available

Name	Mesh	BREP	Slice	Multi-parts saving in single file	Extension
Magics Project files	X	X	X	X	*.magics
STL files	X	O	O	O	*.stl

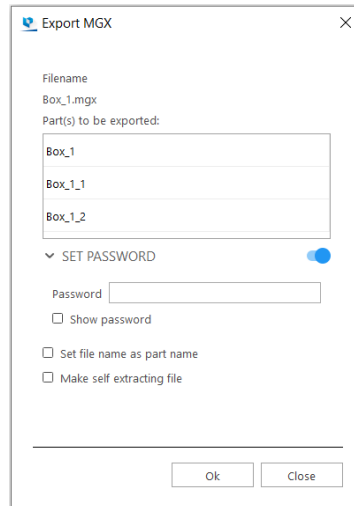
Name	Mesh	BREP	Slice	Multi-parts saving in single file	Extension
STL (ASCII) files	X	O	O	O	*.stl
STL (COLOR) files	X	O	O	O	*.stl
Materialise AM exchange files	X	O	X	X	*.MatAMX
Connect part files	X	O	O	O	*.mpart
Materialise Kernel files	X	O	O	O	*.mdck
AMF files	X	O	O	X	*.amf
MGX files	X	O	O	X	*.mgx
Materialise part files	X	O	O	O	*.matpart
PLY files	X	O	O	O	*.ply, *.zcp
DXF files	X	O	O	O	*.dxf
ZPR files	X	O	O	O	*.zpr
VRML files	X	O	O	O	*.wrl, *.vrml
3D PDF files	X	O	O	X	*.pdf
Iges Wireframe files	O	O	O	O	*.igs
X3D files	X	O	O	O	*.x3d
OBJ files	X	O	O	O	*.obj
3MF files	X	O	X	X	*.3mf
STEP files	O	X	O	O	*.step



Remark: The Save Selected Part(s) As dialog will only show the appropriate file formats that the selected part can be saved to.

Save selected part(s) to MGX

When you save selected parts to MGX format the 'Export to MGX' dialog is given



File name	It allows you to observe the name of the exported file
Part(s) to be exported	It visualizes all parts that will be exported to MGX format
Set password	It allows you to protect the MGX file with a password
Show password	If On, the password will be shown on the screen while typing. If Off, it will be hidden instead
Set file name as part name	The part(s) to be exported to the MGX file will be renamed to the filename with the suffix "_x" for each subsequent part
Make self extracting file	The resulting MGX file upon export will be a self extracting Windows executable file.

4. Save scene As

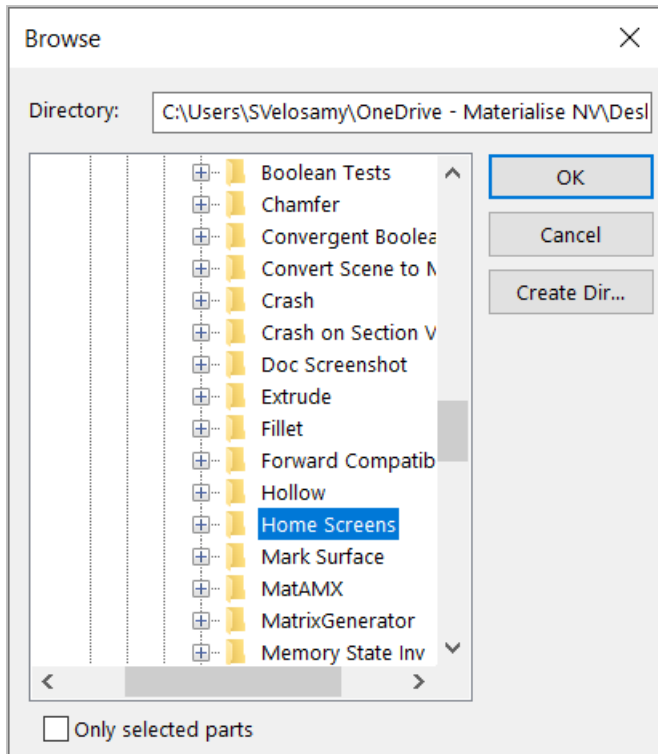


Save the active scene in to a target file.

5. Save All in Directory



This command allows all the selected mesh parts on the current active scene (Part or Platform) to be saved to a .stl file quickly at a target output directory. You can use an existing directory, or create a new one.



6. Save Selected Part(s) to Streamics



Save selected part(s) to Streamics.

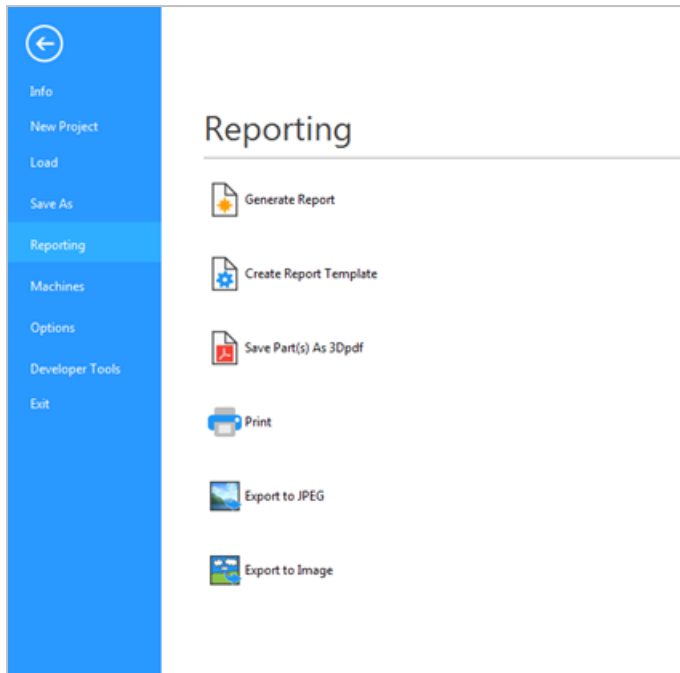
7. Save Platform to Streamics



Save the platform to Streamics.

2.6. Reporting

In the Reporting section, you can generate a report, create a report template, save part(s) as 3Dpdf, print or export a view.



1. Generate Report



Magics can use Microsoft WORD and EXCEL templates to generate reports. In these templates some specific tags are used. When generating the report, these tags will be replaced by the value that they are representing. Of course, you have the freedom to create the templates yourself.

Choose the template Magics should process. When processing the template, Magics will replace the tags by the values they are representing.

2. Create Report Template



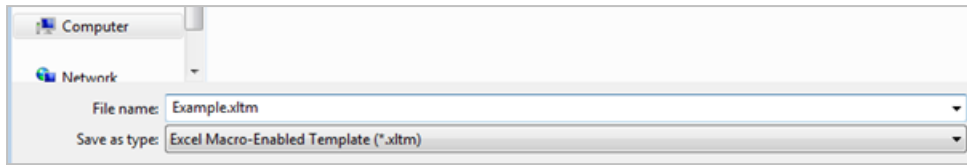
Magics will start the desired template in Microsoft Word or Microsoft Excel for further editing. 'Start here' is an empty template with all the tags in the menu. For Office 2007 these tags can be found under the Add-Ins menu, older version of Office can find the tags under the Insert menu.

Save the template you made in the directory C:\Program Files\Common Files\Materialise\Templates.

The Templates

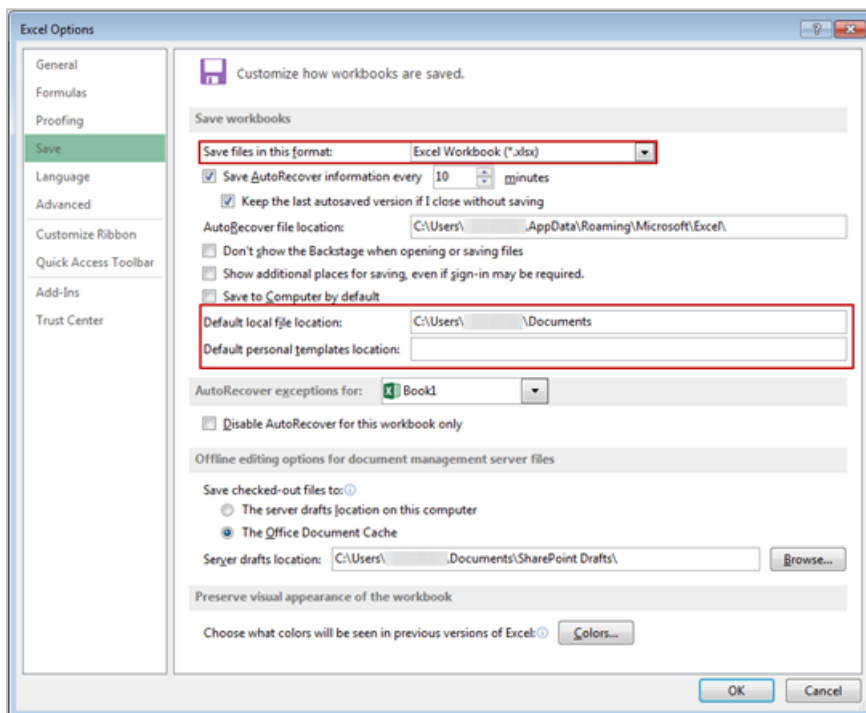
1. Saving a Report Template

When saving your own report template, the file type has to be changed to "Excel Macro-Enabled Template (*.xltm)".

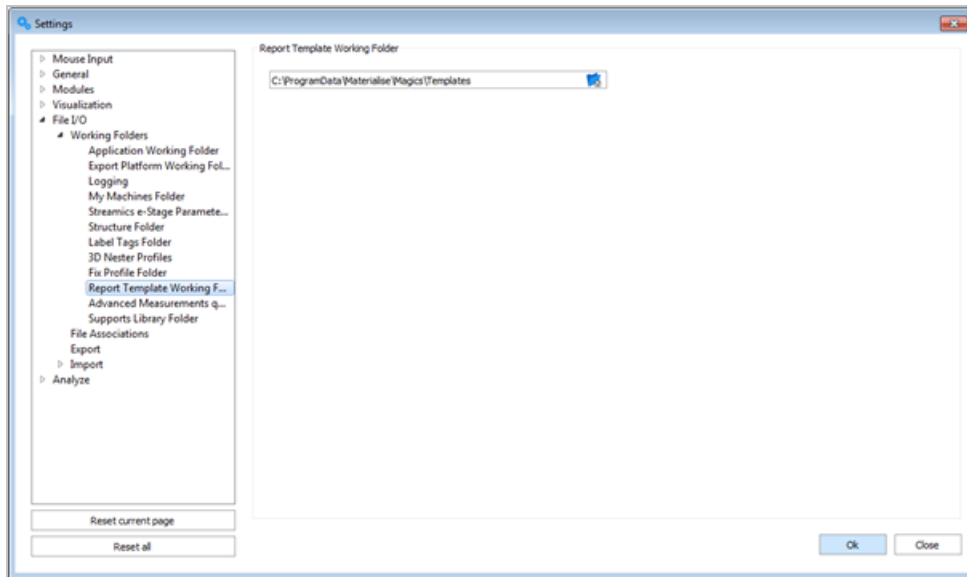


Remark: By default, selecting this file format will change the current folder to “C:\Users\[username]\Documents\Custom Office Templates”. This is standard Excel-behavior and cannot be changed by Magics. Because of this, it is easier to select the “Excel Macro-Enabled Template” first, and only then navigate to the desired save location.

If you often create templates, changing the default save format and the default templates location might speed up your workflow. This can be done in Excel by going to File -> Options -> Save.



Alternatively, it is also possible to change the default Report Template Working Folder in Magics by going to Settings -> File I/O -> Working Folders -> Report Template Working Folder. Using “C:\Users\[username]\Documents\Custom Office Templates” here can also result in a smoother workflow.



2. Tags

A tag is code representing a certain value; Magics will scan the document, recognize the tags and replace them with the wanted value. You have 2 kinds of tags.

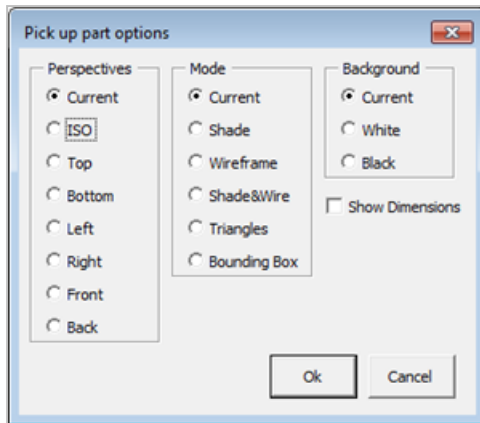
- Text

Text tags are tags, which will be replaced by text. They begin with %% and ends with %%. In between, the property is placed (for example: %%SurfaceArea%%). These tags can be inserted using the insert Materialise field's menu or just by typing the right syntax.

- Pictures

You can also insert pictures (screenshots). They are inserted with a menu. The macro will place a dummy JPEG where the screenshot shall come. When Magics is using the template to generate the document, he will replace this dummy JPEG by the wanted view. A screenshot can only be inserted using the insert Materialise fields menu.

In Office 2000 – 2003 you can find the Materialise fields under the Insert menu. In Office 2007 they can be found under the Add-Ins menu.



Perspectives	In this column, you indicate which view you want. You will recognize the standard views (ISO, top, bottom, etc...). The Current will be the view that is shown in Magics when generating the document.
Mode	You can just take the current active mode or you can make sure that a certain mode (shaded, shaded and wireframe, etc.) will always be used.
Background	You can use the current background color but sometimes it can be better that the background color is forced to be white or black.
Unzoom	Unzoom will always do an unzoom before taking the screenshot.
Show platform	The platform can either be shown or hidden in print screens.

A JPEG will be inserted; the entered properties just entered are linked to the JPEG. You can now resize, move, align, etc. the JPEG. When generating the view, Magics replaces this dummy image by the selected view with exactly the same position and size. The properties of the JPEG can be reviewed by using the 'show all shape names' dialog. In Office 2000 – 2003 you can find the Materialise fields under the Insert menu. In Office 2007 they can be found under the Add-Ins menu.

3. List of tags

– General tags

These tags are representing general information.



User Name
Current Time
Current Time Ext
Current Date
Current Date Ext
Current TimeDate
Current TimeDate Ext
Unit
Platform fields >
Part fields (for table) >
Measurements (for table) >
RapidFit fields >
Objet fields >
Machine properties >
Remove Menu

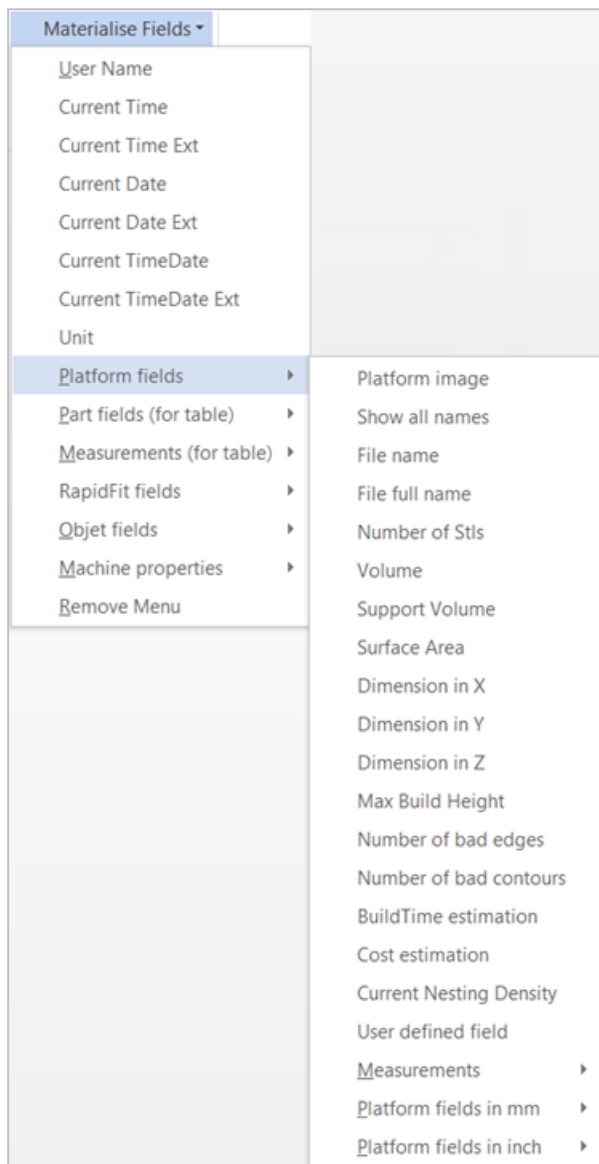
Name	Description												
%%UserName%%	The user name that is currently logged on												
%%CurrentTime%%	<p>The current time</p> <p>Extension:</p> <p>%%CurrentTime:%H:%M:%S%%</p> <table border="1"> <tr> <td>%H</td> <td>Hours</td> </tr> <tr> <td>%M</td> <td>Minutes</td> </tr> <tr> <td>%S</td> <td>Seconds</td> </tr> </table>	%H	Hours	%M	Minutes	%S	Seconds						
%H	Hours												
%M	Minutes												
%S	Seconds												
%%CurrentDate%%	<p>The current date</p> <p>Extension:</p> <p>%%CurrentDate:%A, %d %B, %Y%%</p> <table border="1"> <tr> <td>%A</td> <td>Name of the day</td> </tr> <tr> <td>%d</td> <td>Day (number)</td> </tr> <tr> <td>%B</td> <td>Month (name of the month)</td> </tr> <tr> <td>%Y</td> <td>Year</td> </tr> </table>	%A	Name of the day	%d	Day (number)	%B	Month (name of the month)	%Y	Year				
%A	Name of the day												
%d	Day (number)												
%B	Month (name of the month)												
%Y	Year												
%%CurrentDateTime%%	<p>The current date and time</p> <p>Extension:</p> <p>%%CurrentDateTime: %H:%M:%S, %A, %B %d, %Y%%</p> <table border="1"> <tr> <td>%H</td> <td>Hours</td> </tr> <tr> <td>%M</td> <td>Minutes</td> </tr> <tr> <td>%S</td> <td>Seconds</td> </tr> <tr> <td>%A</td> <td>Name of the day</td> </tr> <tr> <td>%B</td> <td>Month (name of the month)</td> </tr> <tr> <td>%d</td> <td>Day (number)</td> </tr> </table>	%H	Hours	%M	Minutes	%S	Seconds	%A	Name of the day	%B	Month (name of the month)	%d	Day (number)
%H	Hours												
%M	Minutes												
%S	Seconds												
%A	Name of the day												
%B	Month (name of the month)												
%d	Day (number)												



Name	Description
	%Y Year
%%Unit%%	The current unit size used in Magics (mm or inches)

– Group tags

These tags are representing the properties of groups of STL-files.



Tag	Description
Common view	

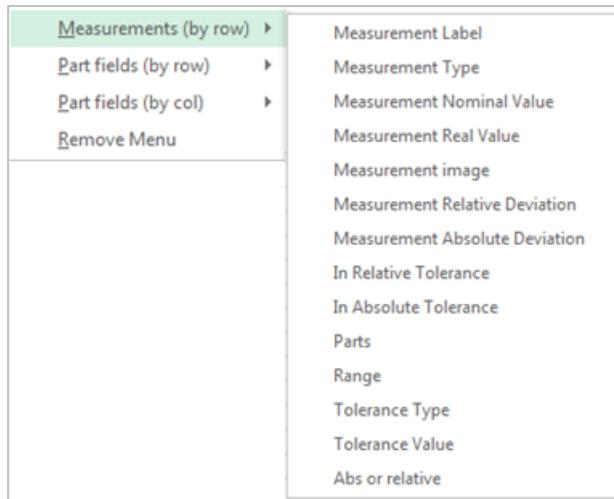


Tag	Description
Show all names	
%%FileName%%	The name of the platform file (only useable in the RP-version)
%%FileFullName%%	The name of the platform file + path (only useable in the RP-version)
%%NumOfStl%%	The number of STL-files in the active scene at the moment the document is generated
%%Volume%%	The total volume of all the parts in the active scene using the current unit size
%%VolumeMM%%	The total volume of all the parts in the active scene in cubic mm
%%VolumeInch%%	The total volume of all the parts in the active scene in cubic inches
%%SupportVolume%%	The total volume of the support of all parts in the active scene using the current unit size
%%VolumeMM%%	The total volume of the supports of all the parts in the active scene in cubic mm
%%VolumeInch%%	The total volume of the supports of all the parts in the active scene in cubic inches
%%SurfaceArea%%	The total surface of all the parts in the active scene using the current unit size
%%SurfaceAreaMM%%	The total surface of all the parts in the active scene in square mm
%%SurfaceAreaInch%%	The total surface of all the parts in the active scene in square inches
%%DimX%%	The X-dimension of the bounding box around all loaded STL-parts in the active scene in the current unit size
%%DimY%%	The Y-dimension of the bounding box around all loaded STL-parts in the active scene in the current unit size
%%DimZ%%	The Z-dimension of the bounding box around all loaded STL-parts in

Tag	Description
	the active scene in the current unit size
%%MaxZ%%	Maximum height of the build
%%DimXmm%%	The X-dimension of the bounding box around all loaded STL-parts in the active scene in mm
%%DimYmm%%	The Y-dimension of the bounding box around all loaded STL-parts in the active scene in mm
%%DimZmm%%	The Z-dimension of the bounding box around all loaded STL-parts in the active scene in mm
%%DimXInch%%	The X-dimension of the bounding box around all loaded STL-parts in the active scene in inches
%%DimYInch%%	The Y-dimension of the bounding box around all loaded STL-parts in the active scene in inches
%%DimZInch%%	The Z-dimension of the bounding box around all loaded STL-parts in the active scene in inches
%%NumOfBadEdges%%	Total amount of bad edges of the loaded parts in the active scene
%%NumOfBadContours%%	Total amount of bad contours of the loaded parts in the active scene
%%NumOfShells%%	The total number of shells in the active scene
%%Machinename%%	The name of the selected machine in the active scene
%%Materialname%%	The material name of the selected machine in the active scene
%%Comments%%	The comments concerning that machine in the active scene
STL View	Insert a JPEG
%%ScanTimeEstimation%%	The estimated scan time in the active scene
%%RecoatTimeEstimation%%	The estimated recoat time in the active scene
%%BuildTimeEstimation%%	Estimation of build time of the loaded parts (only used in the RP-

Tag	Description
	version) in the active scene
%%CostEstimation%%	Estimation of cost of the loaded parts (only used in the RP-version) in the active scene
%%CostEstimationBuildtime%%	The cost fragment depending on the build time
%%CostEstimationFixed%%	The cost fragment depending on the fixed time
%%CostEstimationVolume%%	The cost fragment depending on the volume
%%CostEstimationSupportVolume%%	The cost fragment depending on the support volume
%%CostEstimationSurface%%	The cost fragment depending on the surface
%%CostEstimationDeltaX%%	The cost fragment depending on the delta X value
%%CostEstimationDeltaY%%	The cost fragment depending on the delta Y value
%%CostEstimationDeltaZ%%	The cost fragment depending on the delta Z value
%%CostEstimationNumberOfSTL%%	The cost fragment depending on the number of STLs
%%CostEstimationBoundingBoxVol%%	The cost fragment depending on the bounding box volume
%%UserDef:"Remark"%%	While generating the document, a dialog box will pop up for user input

– Measurement fields



These tags serve to make measuring reports. The Labels for the measuring report are made with the “Report” button in the measuring functionality. Use these tags in a label or in a loop.

Tag	Description
%%Measlabel%%	The label that is given to measurement by Magics by pressing the “report” button.
%%MeasNominalValue%%	The value of the Measurement
%%MeasType%%	The type of measurement: a distance, radius or angle.

– STL fields

<u>U</u> ser Name	
C <u>u</u> rrent Time	
C <u>u</u> rrent Time Ext	
C <u>u</u> rrent Date	
C <u>u</u> rrent Date Ext	
C <u>u</u> rrent TimeDate	
C <u>u</u> rrent TimeDate Ext	
U <u>n</u> it	
<u>P</u> latform fields ▶	
<u>M</u> easurements (by row) ▶	
<u>P</u> art fields (by row) ▶	Part formula
<u>P</u> art fields (by col) ▶	Part image
<u>R</u> emove Menu	Part name
	Part path name
	Part index
	Part Guid
	Number of points
	Number of triangles
	Volume
	Surface Area
	Dimension in X
	Dimension in Y
	Dimension in Z
	Min X
	Min Y
	Min Z
	Center X
	Center Y
	Center Z
	Number of bad edges
	Number of bad contours
	Number of shells
	User defined field
	Estimated scan time
	Estimated part scan time
	Estimated support scan time
	Number of virtual copies
	Part fields (Mm) ▶
	Part fields (Inch) ▶

These tags will be replaced with the information of one single part. In cases where multiple parts are loaded, this information will be repeated once for each part. To achieve this, the tags must be placed into a table or a loop. Magics will repeat this information in the table for each part. See below how to make a loop.

Tag	Description
%%StlName%%	The separate name of the STL-part
%%StlFullName%%	The separate name of the STL-part + path
%%StlGuid%%	The GUID identifies the STL-part
%%StlIndex%%	An index number will appear in this column
%%NumOfCopies%%	The number of virtual copier of an STL part in the active scene
%%StlNumOfPoints%%	The amount of points of the STL-part
%%StlNumOfTriangles%%	The amount of triangles of the STL-

Tag	Description
	part
%%StlVolume%%	The volume of the part using the current unit size
%%StlVolumeMM%%	The volume of the part in cubic mm
%%StlVolumeInch%%	The volume of the part in cubic inches
%%StlSurfaceArea%%	The surface of the part using the current unit size
%%StlSurfaceAreaMM%%	The surface of the part using square mm
%%StlSurfaceAreaInch%%	The surface of the part using square inches
%%StlDimX%%	The X-dimension of the bounding box in the current unit size in the active scene
%%StlDimY%%	The Y-dimension of the bounding box in the current unit size in the active scene
%%StlDimZ%%	The Z-dimension of the bounding box in the current unit size in the active scene
%%StlDimXmm%%	The X-dimension of the bounding box in mm in the active scene
%%StlDimYmm%%	The Y-dimension of the bounding box in mm in the active scene
%%StlDimZmm%%	The Z-dimension of the bounding box in mm in the active scene
%%StlDimXInch%%	The X-dimension of the bounding box in inches in the active scene
%%StlDimYInch%%	The Y-dimension of the bounding box in inches in the active scene
%%StlDimZInch%%	The Z-dimension of the bounding box in inches in the active scene
%%StlMinPosX%%	The minimal X value of the bounding of the parts in the active scene in the current unit size
%%StlMinPosY%%	The minimal Y value of the bounding of the parts in the active scene in the current unit size
%%StlMinPosZ%%	The minimal Z value of the bounding

Tag	Description
	of the parts in the active scene in the current unit size
%%StlMinPosXmm%%	The minimal X value of the bounding of the parts in the active scene in mm
%%StlMinPosYmm%%	The minimal Y value of the bounding of the parts in the active scene in mm
%%StlMinPosZmm%%	The minimal Z value of the bounding of the parts in the active scene in mm
%%StlMinPosXInch%%	The minimal X value of the bounding of the parts in the active scene in inches
%%StlMinPosYInch%%	The minimal Y value of the bounding of the parts in the active scene in inches
%%StlMinPosZInch%%	The minimal Z value of the bounding of the parts in the active scene in inches
%%StlCenterPosX%%	The X value of the center point of the parts in the active scene in the current unit size
%%StlCenterPosY%%	The Y value of the center point of the parts in the active scene in the current unit size
%%StlCenterPosZ%%	The Z value of the center point of the parts in the active scene in the current unit size
%%StlCenterPosXmm%%	The X value of the center point of the parts in the active scene in mm
%%StlCenterPosYmm%%	The Y value of the center point of the parts in the active scene in mm
%%StlCenterPosZmm%%	The Z value of the center point of the parts in the active scene in mm
%%StlCenterPosXInch%%	The X value of the center point of the parts in the active scene in inches
%%StlCenterPosYInch%%	The Y value of the center point of the parts in the active scene in inches
%%StlCenterPosZInch%%	The Z value of the center point of the parts in the active scene in inches
%%StlNumOfBadEdges%%	The amount of bad edges of the part
%%StlNumOfBadContours%%	The amount of bad contours of the

Tag	Description
	part
%%StlNumOfShells%%	The amount of shells
%%StlSupportScanTimeEstimation%%	The estimated scan time of the supports of the part
%%StlPartScanTimeEstimation%%	The estimated scantime of the part
%%StlScanTimeEstimation%%	The total scantime of the part
%%StlUserDef:"Remark"%%	While generating the document, a dialog box will pop up for user input for each STL-file



Remark: for excel templates the 'STL view' tag is available for row and columns.

– Machine properties fields

Machine properties	Machine Information
Remove Menu	Support Parameters

Tag	Description
%%MachinePlatZComp%%	The platform settings and Z-compensation settings from the machine properties
%%MachineSupport%%	The support parameter settings from the machine

– Object fields

Specific tags for documenting Object set-ups.

– RapidFit fields

Specific tags for documenting RapidFit set-ups are documented in the RapidFit section of the manual.

– Loops

Instead of using tables for STL-fields, Measurements and EDM's, you can use loops. All the text and tags between the loop-tags %%For Each: MEASUREMENT%% and %%Next:MEASUREMENT%% will be repeated for each (in this example) measurement. This allows E.g. a complete page per part.

3. Save Part(s) As 3D PDF

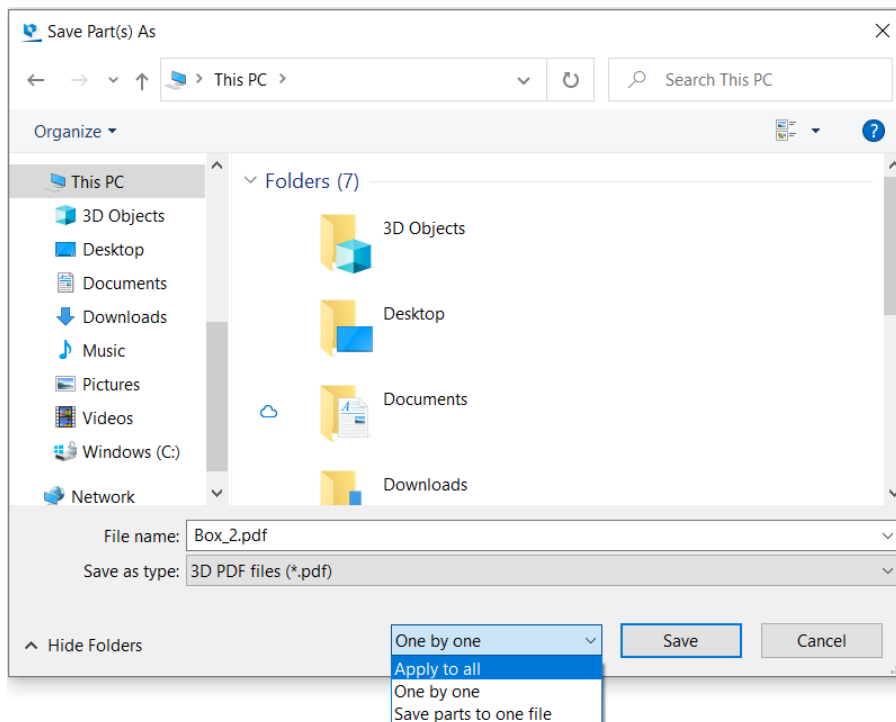


By saving parts as 3D PDF, you create a PDF-file which can be viewed in 3D with a default Adobe Acrobat Reader. This enables you to send or view the design in 3D without specific software needed.

The user can save the parts by clicking on 'Save Part(s) as 3Dpdf' in the report ribbon. Alternatively the user can click 'Save Selected Parts As' in the toolbar and then select the '3D PDF (*.pdf)' file type.

If the drop down menu is set to 'Apply to all', all parts will be saved individually under their respective names. If it is set to 'One by one', the user can change the file name of every part, with all the parts still being saved individually. Setting it to 'Save parts to one file' will save all the parts in one file, which can be viewed by the Adobe Acrobat Reader as an assembly or a collection of parts.

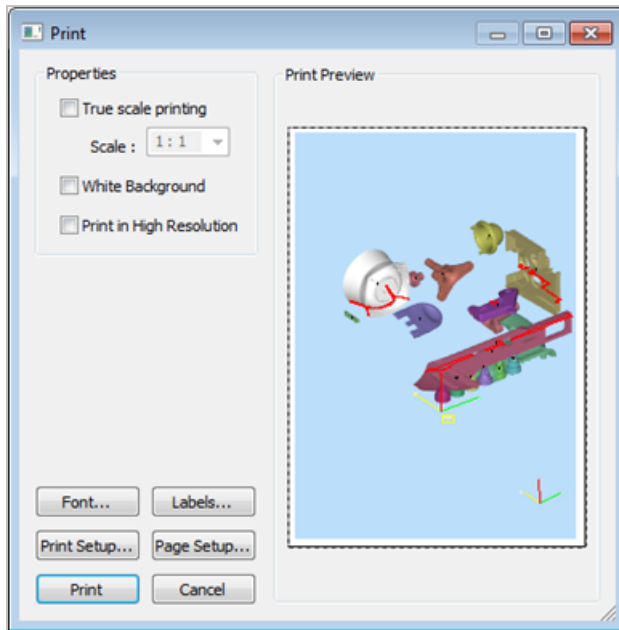
If there's support connected to the part, it will also be visible in the reader. When viewing the part, it's possible to hide the support by going to the Model Tree tab and unchecking the box in front of 'Support [filename]' or right clicking it and clicking 'Hide'. The same can be done with the part, to only show the support.




4. Print

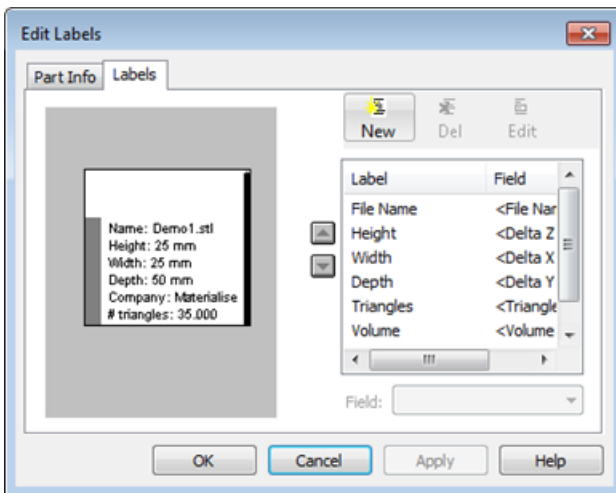
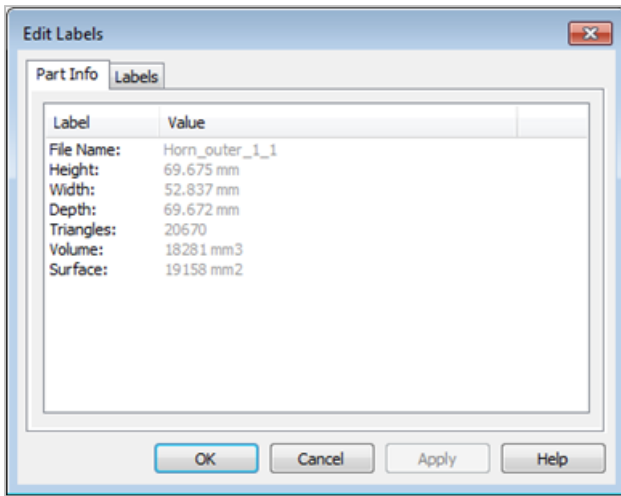


This command starts the Magics print wizard that leads you to the Page Setup and the standard Windows Print dialog box.



Print Preview	The Print Preview is visible in the Print-action. The right half of the print dialog box is the Print preview.
True Scale Printing	<p>This feature allows you to print in scale. This means that you can measure on the printed page.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p> Remark: Be aware of the fact that perspective views never allow measuring.</p> </div>
Font...	This allows you to define the font of the labels.
Labels...	Specify the information about the part that should be printed. For instance: File name, Height, Width, Depth, Number of triangles....
Print Setup...	The user can choose another printer, paper size and paper orientation using the standard printer setup interface.
Page Setup...	With Page Setup, one can define a print style. It is a standard Windows dialog box.

Edit Labels




Part Info

This can be used to see information about the part, but it is also the information that will appear on paper when a part is printed (only one part may be visible to show this information). This info is selected via the Label Sheet.

Labels

New	A New Label and Field are added at the bottom of the list. The user can immediately change the label and the field type.
Del	To delete a label and a field: Select the label, and press on the Delete button.
Edit	The user can edit a label by selecting one, and pressing the Edit button.
Rearrange Label	The order of the labels can be changed with the arrow buttons. Select a label, and use the arrow buttons to move it one place up, or one place down the list.

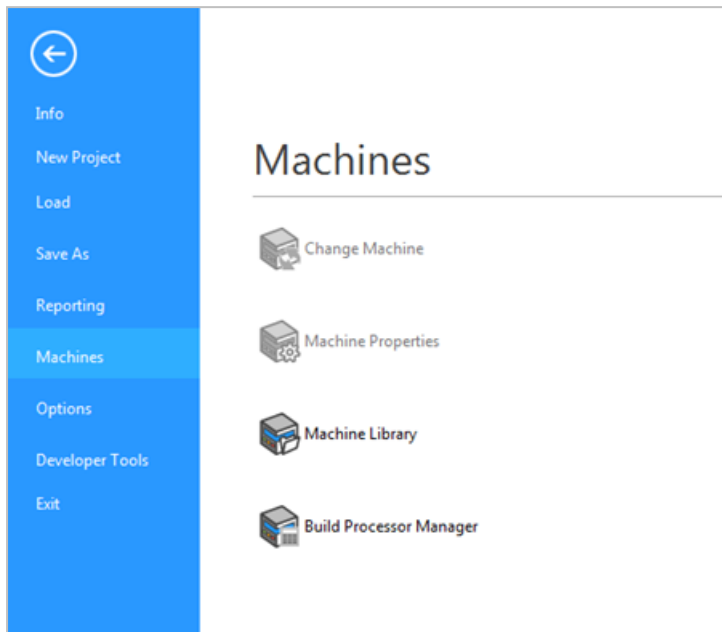
	
Field Types	Several Field Types are predefined. See the following table.
Changing Field Types	Select a Label, and select another field type.

– Field types

Label	Comment
Delta X	Width of the STL part
Delta Y	Depth of the STL part
Delta Z	Height of the STL part
Dimensions	Start and end coordinates in all three dimensions
File Name	Name of the STL file
File Path	File Path of the STL file
Triangles	Number of triangles in the file
Volume	Volume of the STL part
Surface	Surface of the STL-part

With the 'User Input' field, the user can input any string before printing (e.g. reference number, client name...). It is always possible to change to another field type.

2.7. Machines



1. Change Machine

- See ***Change Machine, page 315***

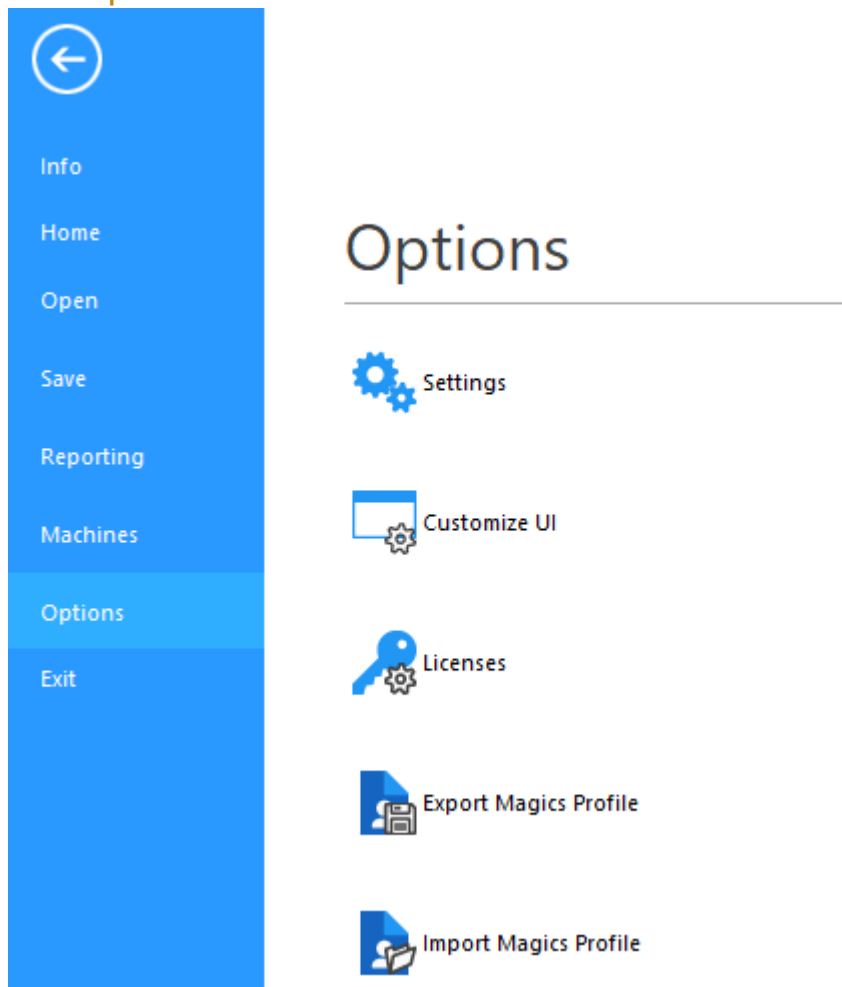
2. Machine Properties

- See ***Machine Properties, page 293***

3. My Machines

- See ***My Machines, page 315***

2.8. Options



1. Settings

- See **Settings**, page 512

2. Customize UI

- See **Customizations**, page 35

3. Licences

- For details, please go to: <https://help.materialise.com/>

4. Export Magics Profile

- See **Export Magics Profile**, page 559

5. Import Magics Profile

- See **Import Magics Profile**, page 558



6. Exit

Closes Magics.

Chapter 3. Fix

3.1. Automatic Fixing

1. AutoFix



Clicking on Auto Fix will perform the fixing operation automatically.

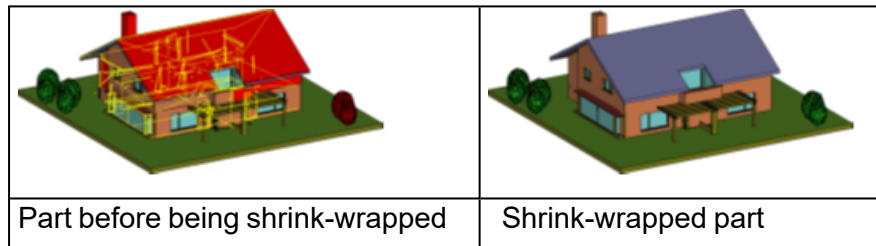
2. ShrinkWrap



Principle

The principle is that a new surface is wrapped around the existing geometry so it takes over its shape and then the original geometry is removed. You can compare it with the real life shrink-wrapping, where plastic is wrapped around a part and shrunk to fit.

The main aim for the shrinkwrap is to make a part buildable, accuracy is the second priority.



When to use ShrinkWrap?

If the STL-file contains following kind of errors, consider using the ShrinkWrap to fix the part.

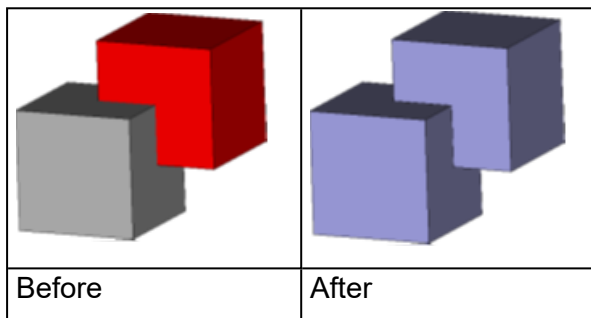
- *Complex inverted surfaces*: Inverted surfaces are so complex that automatic or manual flipping of the triangles is almost infeasible.
- Shells
- High shell count: Due to the way of drawing, the amount of shells can be very high. Then it's getting difficult for Magics to fix handle this amount of shells. For example, each beam of a windowframe is a separate shell and the building has a few hundred windows.
- Many of those shells are touching and/or coming together at the same spot, resulting in many triangle sides on exact the same spot. This confuses the traditional fixing algorithms because he cannot determine which triangle is connected to which triangle.
- Internal shells: Some models contain a lot of internal shells that are not necessary for building the part. Due to the nature of the part, it's difficult to isolate these shells for an efficient removal.
- Complicated bad edges: The bad edges are so complex that stitching, hole filling or the creation of triangles does not help.
- Gaps

- The elements of the part are not properly connected to each other. There's sometimes a small gap between the parts.
- Single surfaces
- Some sides are represented by single surfaces having no thickness. This is ok for visualization but not for building on a RP machine.

How does ShrinkWrap fix these problems?

1. Flipped triangles

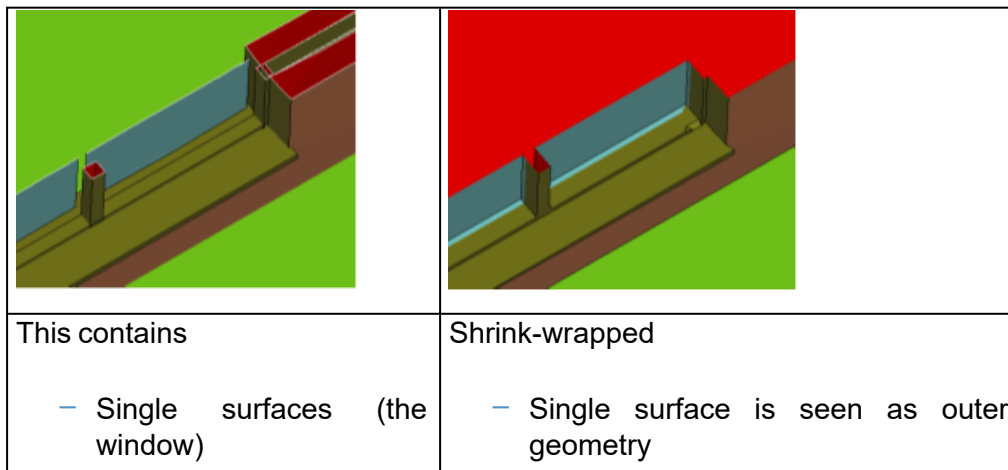
The ShrinkWrap algorithm will lay the surface on triangles, regardless their orientation.



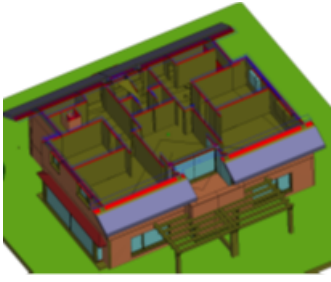
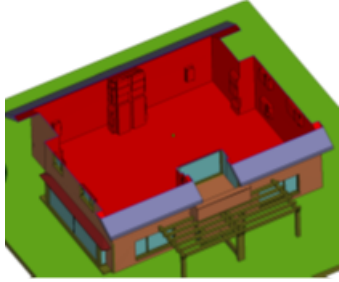
2. Outer geometry

The Shrink-wrapped surface will

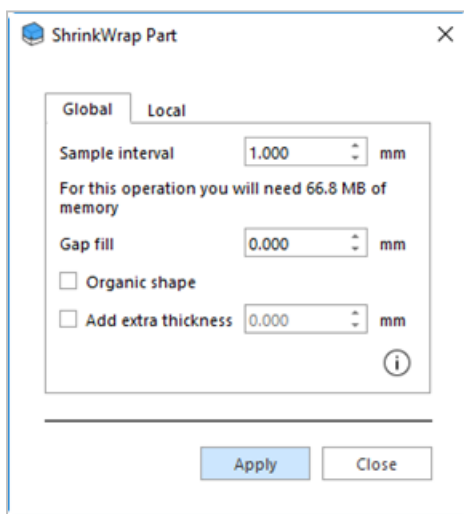
- Only take the outer geometry in account
- Joining all the shells (and so handling overlapping & double surfaces)
- Ignoring inner geometry, noise shells and their errors (complex bad edges)
- Treat single surfaces (surface with no thickness) as outer geometry
- Bridge small gaps
- Bad edges very close to each other
- Different shells not joined properly



<ul style="list-style-type: none"> – Overlapping shells – Gaps between beam & Window 	Multiple	<ul style="list-style-type: none"> – Only the outer geometry of the shells is kept – Small gaps are bridged
--	----------	---

	
Many internal features inside the house.	All internal features are removed. Only the outside is kept.

Dialog Box and its parameters



1. Sample Interval

The smaller, the more precise the Shrink-wrapped surface will follow the original geometry.

Please consider:

- The smaller the sample interval, the higher the memory usage. Because we're dealing with an object in 3D object, a division of the sample interval by 2 will increase the memory use by 8 (each dimension gets 2 times more samples → $2*2*2=8$).



- The smaller the sample interval, the more details will be preserved. Details that fall between 2 sample points (E.g. small walls, ribs, decorations, etc...) typically fall away. (see below “Wall thinner than...”)
- The smaller the detail, the more sensitive the Shrink-wrapped surface is for gaps and the surface might “leak” to the inside via a gap. To compensate this, use Gap Fill.

Sometimes sharp edges are not followed perfectly. This can happen into areas with a high complexity.

2. Relation between Sample Interval & Gaps

The surface will fill small gaps and holes.

But if you use a smaller sample rate, the accuracy of the wrapped surface will increase. It might increase so much that the gaps will be considered as geometry and these defects will be shrink-wrapped. The result is that the surface might “leak” to the inside so that all internal geometry will be shrink-wrapped too and there’s also the risk that walls might disappear because they are too thin. If this is the case, use “gap fill” to compensate.

Original Geometry	ShrinkWrap accuracy low The Gap is bridged	ShrinkWrap High Accuracy So accurate, the gap is seen as geometry and the surface “leaks” to the inside.

Walls thinner than ... might disappear

Depending on the sample interval, geometry thinner than the given thickness might disappear. This might be a problem with walls & pipes from architectural parts.

Keep in mind that only geometry will disappear that is shrink-wrapped at both sides. Thin walls on the outside of a house, where the inside is not wrapped will not disappear.

How to deal with this:

- Use “Add thickness” to avoid removal of this geometry (see below)
- Try to detect the thin areas (use double surface detector)
- Try using offset to make them thicker

Try to separate these thin areas from the main part & process these parts separately.



Sample Data to process

This gives you an indication of the size of the 3D Sample data.

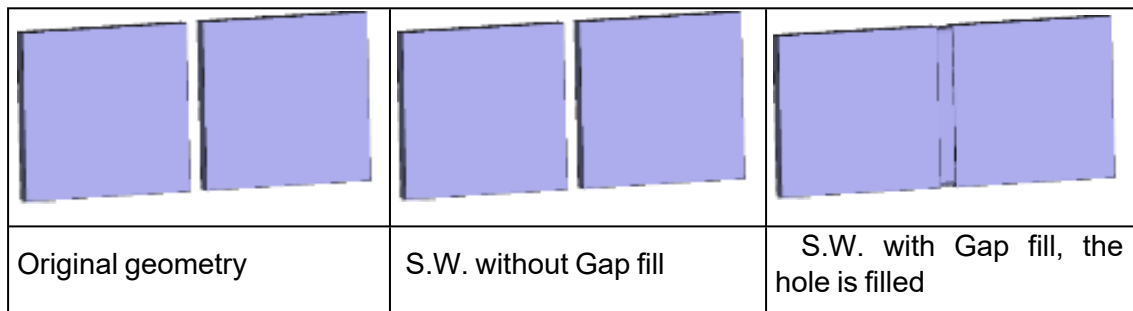
Please keep in account that this is not the final memory usage.

Pay attention because the algorithm is also needing memory to create the triangle surface around the part (the ShrinkWrap surface) and (depending of the complexity of the geometry) this can also need a lot of memory.

Advanced Parameters

1. Gap fill

If you notice that there are still gaps that need to be filled, you can increase the gap filling rate with this function.



2. Organic shape

If you have a part with a shape without essential edges, you can skip the step of restoring these by checking "Organic shape". This will also save you time.

3. Add extra thickness

If the part contains features that are too thin, checking this checkbox will add extra thickness.

Please take in account that:

- The thickness will be added to the whole part
- The thickness has a minimal value, depending of the ShrinkWrap parameters.

You can use Add extra thickness

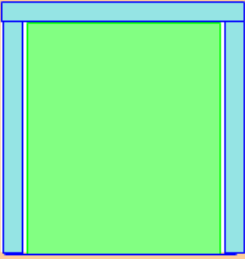
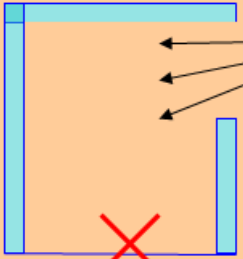
- To avoid that geometry disappears because it's too thin
- To add thickness to the part to make it build able.

Working with the ShrinkWrap

- Please take in account the high processing time & memory use of this algorithm. Some parts are still faster & more accurate to fix with the Fixing Wizard.
- The ShrinkWrap algorithm should be considered as a step in the fixing process. You need to prepare the model. You can check if the model is “closed” so that the outside is not leaking to the inside. If this happens, the risk that geometry will disappear because the walls are too thin is very high. You can close the model using hole filling, creating triangles or by extruding sides of walls.

Below is an example what might happen if you don't prepare a part properly.

- Red: outside
- Green: Inside

	
<p>Part is closed properly. The result is that the “outside’ stays at the outside and the result will be a solid block.</p>	<p>A gap is not closed The “outside leaks inside”. This results also in the thin wall that will disappear.</p>

- You can run some quick test to see if the part is shrink-wrapped correctly by using the add thickness option.

When using add Thickness, some steps are skipped what gives you faster feedback so you can have an idea of the end result or an indication where it goes wrong.

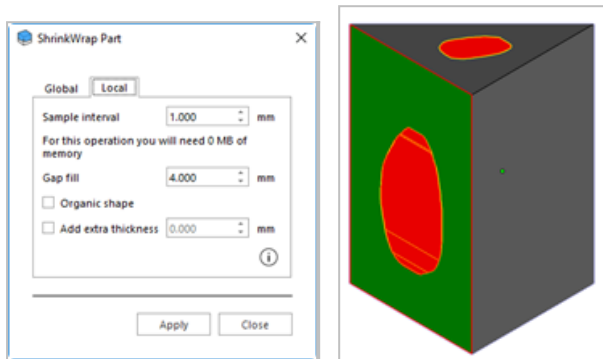
Local ShrinkWrap

The user can also perform a Local ShrinkWrap, which means that the ShrinkWrap operation will be applied only on an indicated area. The Local Shrinkwrap is located in the ShrinkWrap Part dialog box, under the ‘Local’ tab.

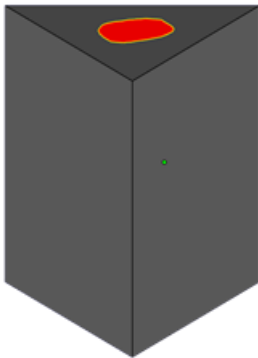
In order to mark the surface and, implicitly, the triangles on which the Local ShrinkWrap will be applied, use the already available marking tools from the marking toolbar.



After marking the area, open the ShrinkWrap window, switch the tab to “Local” and indicate what parameters should be used. For further details on the parameters, please read again through sections Dialog Box and its parameters and its parameters & Advanced Parameters.



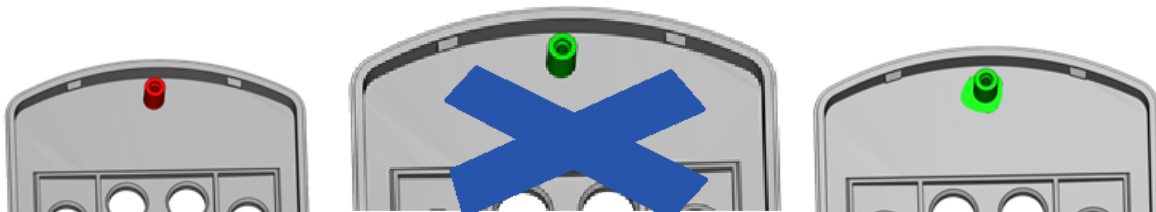
The Local ShrinkWrap will be applied on the surface that you have selected and will create an organic tie between this surface and the rest of the part.



The Local ShrinkWrap operation might encounter issues if:

- The selection is a shell that is made out of inverted triangles

Solution: When selecting, use a marking tool that allows you to select also non-inverted triangles around the shell made out of inverted triangles





3.2. Semi-Automatic Fixing

1. Normals Fix



This option allows you to automatically fix inverted normals on the selected part(s).

- See ***Inverted Normals***, page 587.

2. Automatic Stitching



This option allows to automatically stitch near edges on the selected part(s). Magics will do an estimation of the needed tolerance and stitch iteratively using this tolerance.

- See ***Bad edges***, page 588.

3. Holes Fix



This option allows to automatically fix holes on the selected part(s). When pressing the button, all the detected planar holes will be filled. A planar hole will not be filled with triangles if Magics detects that these new triangles will intersect with other (existing) triangles.

After the hole filling, check if the holes are filled correctly. It can be that the geometry was misunderstood.

- See ***Planar hole***, page 588.

4. Noise Shells



This option allows to automatically remove noise shells from the selected part(s). We prefer to play it on the safe side. Occasionally the algorithm does not remove some noise shells.

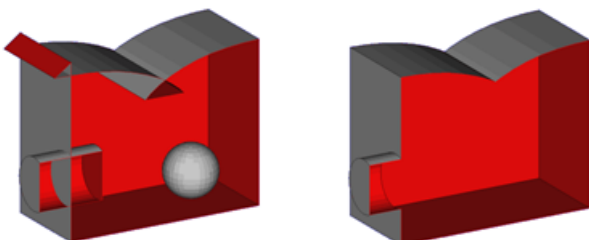
- See ***Noise shells***, page 590.

5. Unify



When applying the unify function to a part, only the outer triangles will be preserved and all the inner triangles will be thrown away.

Example:



You see there are 4 changes:

- Sphere: The triangles of a sphere are all internal triangles, so they are removed.
- Cylinder: The cylinder intersects the cube. As you see, only the triangles that are outside remain. The internal surfaces are removed after a re-triangulation was done.
- Overlap: the internal part of the intersection is removed.
- Intersection at the outside: Surfaces, which are inverted, are removed.

For parts with a lot of internal intersections, unify will help you to make clean parts.

6. Shells to parts



Converts shells to parts.

- See **Shells, page 589**.

7. Remove Small Parts



When doing a Shells to Parts (See **Shells, page 589**), the result may be a lot of small parts, which are just noise and which are not needed anymore. You can specify the criterion to filter the parts quickly. All of the filtered part will be unloaded from the current scene if it fulfills at least one of the set criteria.



Note: If you would like remove shells from large parts quickly, using the **Shell page, page 602** to sort and remove the shells will be much more efficient than using the Remove Small Parts operations along with the Shells to Parts operation.

Remove Small Parts
✕

Remove selected parts with

- Volume less than mm³
- Surface area less than mm²
- # Triangles less than

Parts are always filtered based on the current selected parts. If you would like to apply the operation on all parts in the scene, please select all parts.

Filtered parts will be automatically unloaded from the current scene.

Volume less than

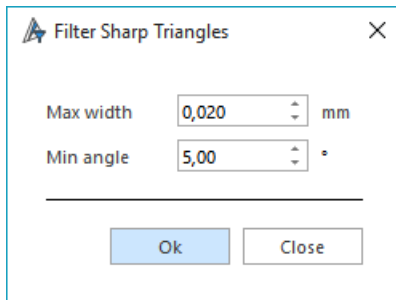
If set to true, any selected part that has volume smaller than specified will be filtered.

Surface area less than	If set to true, any selected part that has surface area smaller than specified will be filtered.
# Triangles less than	If set to true, any selected part that number of triangles less than specified will be filtered.

8. Filter Sharp Triangles



When the part has long thin triangles, this filter will remove them.



- The parameters used for this command are synchronized with the ones in Triangle tool page: See **Triangle page, page 598**.

9. Remove Identical Triangles



This tool gives you the opportunity to remove identical triangles quickly. Two parameters define when two triangles are considered identical.

- The parameters used for this command can be found in Triangle tool page: See Triangle page, page 1.

10. Mark Overlapping Triangles



This tool allows you to detect overlapping triangles (double surfaces). The definition of an overlapping triangle is made with the help of some parameters.

- The parameters used for this command can be found in Triangle tool page: See Triangle page, page 1.

3.3. Manual Fixing

1. Invert Normals



Click to activate. You can now manually invert all the marked triangles.

2. Fill Hole Mode



Click to activate. You can now manually select and fill holes.



- See **Single or Multi-contour hole**, page 594.

3. Create Triangle



Click to activate. You can now manually create a triangle or bridge.

- See **Create triangle**, page 596.

4. Translate Part Points



Click to activate. You can now manually select and translate a part point. Select multiple points by holding CTRL while selecting, or by drawing a window.

- See **Translate**, page 242

5. Move Part Points



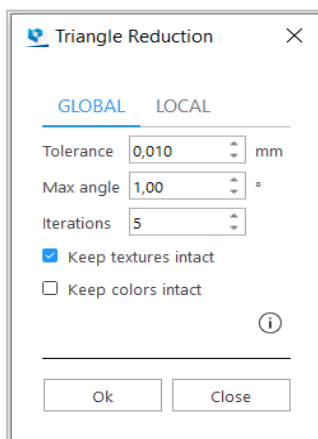
Click to activate. You can now manually select a part point and replace it. This point will snap to the neighboring edge or points.

3.4. Enhance

1. Triangle Reduction



Magics allows you to reduce the number of triangles in an STL file. This makes it easier to manipulate the file. You can re-triangulate the whole part, or only a selected portion. A Local Triangle Reduction will reduce only the marked triangles, but some neighboring triangles can be changed too.



Tolerance	If 2 triangles are replaced by one triangle, there may be a little deviation in position. The tolerance indicates the maximum deviation allowed between the original surface and the new one.
-----------	---

Max angle	<p>The Max angle value defines two limits:</p> <ul style="list-style-type: none"> – When two triangles have an angle value bigger than Max angle, they may not be reduced. The edge between them may not be eliminated; otherwise too much geometrical information would be lost. When the program meets such an edge, the reduction will keep the edge but reduce the number of points on it. – When there is no critical edge, this Max angle value determines the maximum angle that can be created during the reduction. This means where there is an edge present, there will remain one. Where there is no edge, no edge will be added.
Iterations	Magics can perform the operation in different iterations to improve the reduction of triangles. It is better to increase the number of iterations than performing the triangle reduction twice (to maintain the smallest detail).
Keep textures intact	Textures on the part stay intact. Some triangles may not be filtered out due to texture that is kept.
Keep colors intact	Colors on the part stay intact. Some triangles may not be filtered out due to color that is kept.



It is advised not to use the reducer on very noisy objects. In this case it is better to perform a smoothing first.

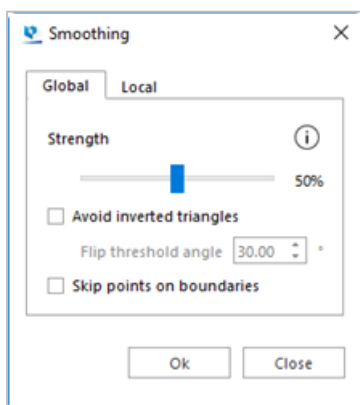



If the tolerance and angle values are too big, essential part information may get lost.

2. Smoothing



Smooth the selected part or marked area.

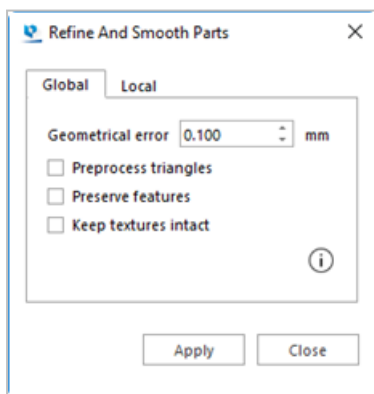


Global / Local	Global will smooth the whole part. Local will smooth only the selected triangles.
Strength	<p>We will explain this factor with the figure below. The figure shows eight triangles with one common point in the center.</p>  <p>The algorithm changes the position of this center point according to the positions of the eight other points. The importance of the other points can be raised or lowered by adjusting the Ratio-parameter. If this Ratio is low (0.01) the new position is mainly dependent on the old position of the point. If this ratio is 1, the dependency is spread over the points. The new position is still 50% dependent on the old position. With high values for the ratio, the new position is mainly determined by the position of the other points of the triangles. In this last case it is obvious that we talk about smoothing. This algorithm is exercised on every point of the part.</p>
Avoid Inverted Triangles	To avoid the creation of flipped triangles, the movement of the points will be aborted when the angle between the normal of the neighboring triangles is bigger as the given angle.
Skip points on boundaries	When the point is on the part boundaries edges (with Global option) or on the marked area boundaries edges (with Local option), the point won't be moved.

3. Refine and Smooth Parts



This feature has as purpose to give gaunty parts a more smooth surface while making sure that the original shape is maintained as much as possible.



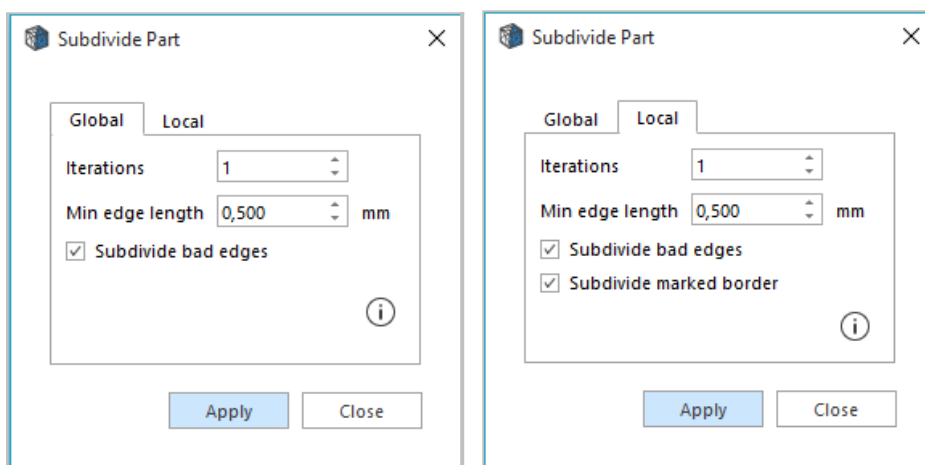
Global / Local	Global will smooth the whole part. Local will smooth only the marked triangles.
Geometrical error	Geometrical indicates the maximum deviation allowed between the original surface and the new one.
Preprocess	Before the refine and smooth algorithm is performed, a filtering of sharp

triangles	triangles is done.
Preserve features	Sharp features will be preserved. The refine and smooth algorithm will only smooth the surfaces that are already pretty smooth. This makes sure that surfaces that don't need smoothing will keep their geometry.
Keep textures intact	Textures on the part stay intact. Some triangles may not be smoothed due to texture that is kept.

4. Subdivide part



With the Subdivide Part option, you can add extra triangles to the selected part or area without changing its appearance.

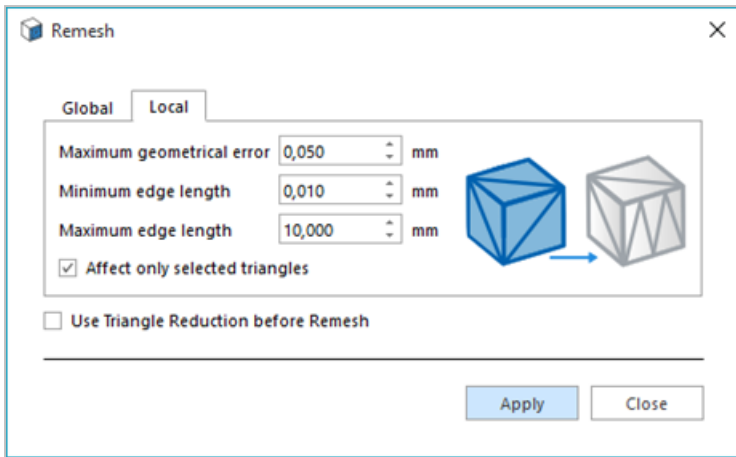


Global / Local	Global will subdivide the whole part. Local will subdivide only the marked triangles.
Number of iterations	Number of iterations indicates the number of iterations that will be executed. The more iterations, the more additional triangles the will be added. The maximum number of iterations is 1,000
Max edge size	Max. Edge Size indicates the maximum edge size that will be subdivided.
Subdivide Bad Edges	Bad edges will be subdivided.
Subdivide Marked Border	The border of the selected triangles will be subdivided.

5. Remesh



Remesh feature gives users a option to generate new mesh topology in frames of existing part. It can be applied to the whole part or marked region.



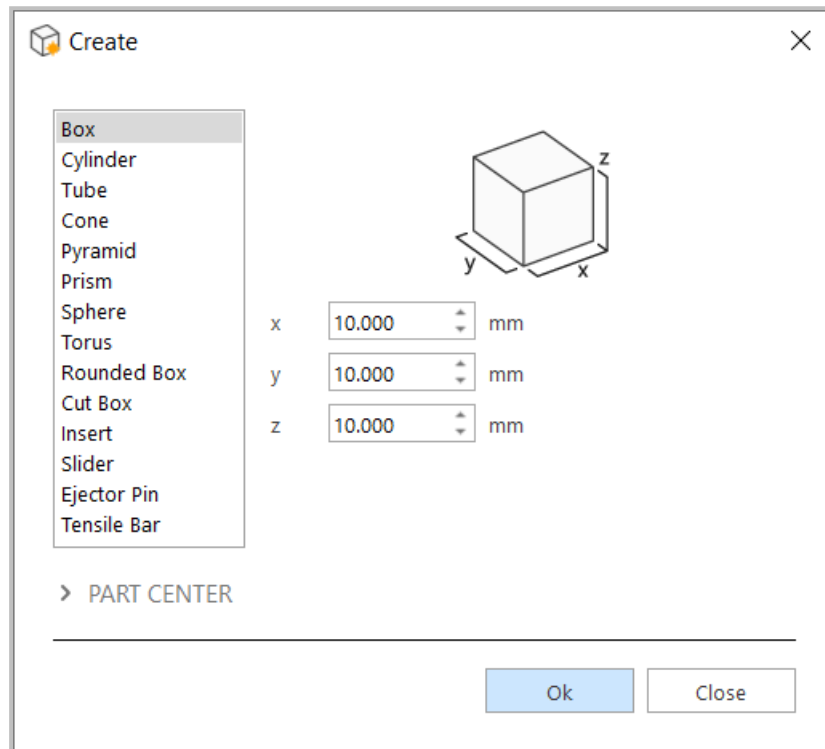
Chapter 4. Edit

4.1. Insert

1. Create



With this command, basic shapes can be created (Mesh or BREP Part).



A window with two panes respectively will be opened. The first pane contains a list with the volumes the user can choose. The second pane contains the object definition for the specified volume. A third pane, accessed by clicking “PART CENTER” enables to enter position coordinates.

The Create command is accessible from mesh and BREP part scenes where it creates primitives in the respective types. From the respective part scenes, only the specific primitives relevant for the scenes are shown in the Create dialog.

Note: BREP Part creation is only supported on Magics RP version

Object Definition

The object definition-sheet contains all the parameters to define the object that the user would like to create. The parameters correspond with certain dimensions that are illustrated on the drawings. The tolerance parameter influences the number of triangles that will be created. The first time, the default values are displayed. When you change these values and execute the operation, the used values will be remembered.



Note: Only a limited set of objects are included for creation as a BREP part.

Part Center

▼ PART CENTER			
	X	Y	Z
Coordinate	<input type="text" value="0.000"/>	<input type="text" value="0.000"/>	<input type="text" value="0.000"/> mm

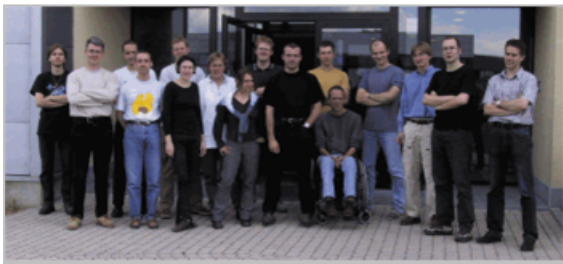
Here you can enter the coordinates of the center of the created part.

2. Create from Image

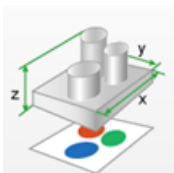
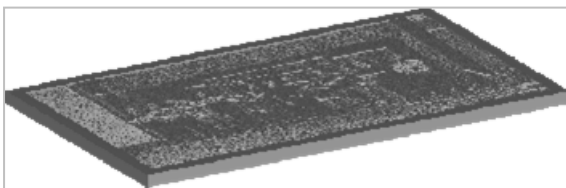


This function will create a new mesh part of a kind of 3D-picture from a given image.

Example:



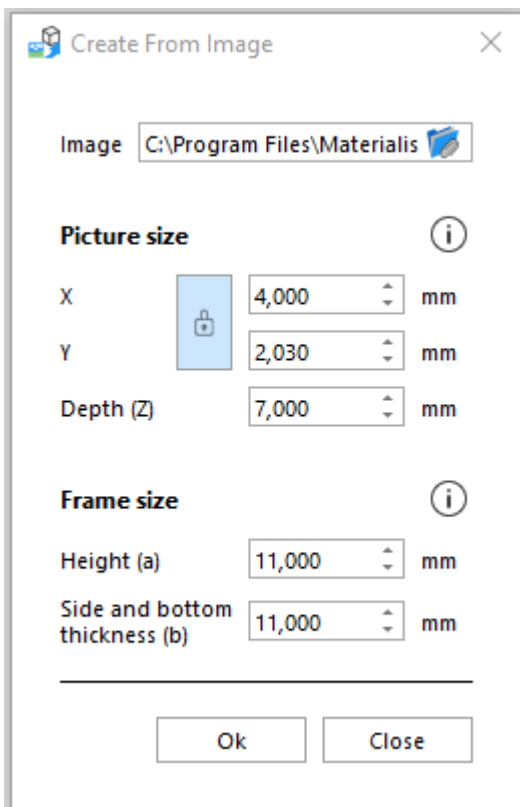
Above you see an image and below you see the created mesh part.



Nothing special, the only thing you can notice if you really look close is that the dark parts on the image are thick and the light areas are thin. But if you keep it in front of a light source with the relief pointing to you, you will see this.



Because of the relief, the picture will be visible. This part was made with a ThermoJet (a kind of 3D-wax printer from 3D systems) but of course other techniques will also give nice results. As long as the material lets light through but is not too clear, the result should be fine.



File Name	Source image from which you would like to make an STL-file. An image with a lot of contrast is advised.
Picture size: X & Y	The size of the picture inside the frame.
Keep aspect ratio	The aspect ratio expresses the proportion between the X and Y direction.
Depth	The relief is based on the gray values of the pixels of the bitmap. If a pixel is white, the thickness of the part will be minimal at that place, so a maximum of light will get through. If a pixel is black, the thickness is maximal and so a maximum of light will be blocked and so it will seem darker. Depending of the material, this value must be adapted. Some small experiments can be useful.

Frame height & Frame & bottom thickness	The size of the borders around the picture.
---	---

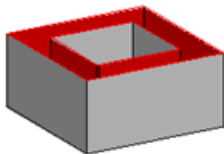
4.2. Edit

1. Hollow Part

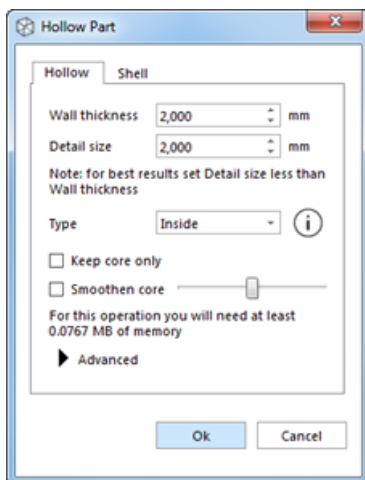


You can use this command to easily remove material from a part; final printed parts will become lighter, saving you material and build time. The result of the hollow operation is one STL file that present a cavity with a constant wall thickness. This cavity is build from triangles whose size is determined by the parameter detail size. There is the option to keep the original part closed with all the faces (Hollow tab), or choose to remove selected faces (Shell tab).

Note: Hollow operation is also available for BREP Part(s)













Hollow for Mesh Part(s)

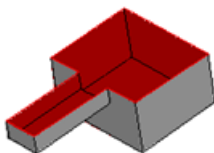


The result of the hollow operation is one STL file with two shells: the original shell and a new one that gives the part a constant thickness. You can select several parts and make them hollow in one operation.

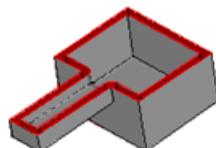
Wall Thickness	This value displays the distance over which the triangles of the original shell get an offset in order to generate a hollow part.
Detail Size	This value displays the level of detail that will remain in the new shell. Standard, this value should be the same as the smallest detail of the part. The smaller this value, the more triangles will be included in the new shell

	<p>and the more detail can thus be incorporated.</p> <div style="border: 1px solid blue; padding: 5px;"> <p> Remark: If the value is chosen too high, it is possible that the internal wall intersects with the external wall.</p> </div>						
Type	<p>Here you determine if you want to create a new shell at the interior or at the exterior of the existing shell, or create a self-supporting structure.</p> <table border="1"> <tr> <td>Keep Core Only</td> <td> <p>If you only wish to retain the newly created shell, you can check this box.</p> <div style="border: 1px solid blue; padding: 5px;"> <p> Note: available for inside and outside values of Type parameter.</p> </div> </td> </tr> <tr> <td>Smoothen Core</td> <td> <p>By checking this option a smoothing will be performed on the created core. (See Smoothing, page 109)</p> <div style="border: 1px solid blue; padding: 5px;"> <p> Note: available for inside and outside values of Type parameter.</p> </div> </td> </tr> <tr> <td>Surface angle</td> <td> <p>This value defines the self-supporting angle used to generate the internal cavity, making sure that no support is needed to build the part successfully.</p> <div style="border: 1px solid blue; padding: 5px;"> <p> Note: available for self-support value of Type parameter.</p> </div> </td> </tr> </table>	Keep Core Only	<p>If you only wish to retain the newly created shell, you can check this box.</p> <div style="border: 1px solid blue; padding: 5px;"> <p> Note: available for inside and outside values of Type parameter.</p> </div>	Smoothen Core	<p>By checking this option a smoothing will be performed on the created core. (See Smoothing, page 109)</p> <div style="border: 1px solid blue; padding: 5px;"> <p> Note: available for inside and outside values of Type parameter.</p> </div>	Surface angle	<p>This value defines the self-supporting angle used to generate the internal cavity, making sure that no support is needed to build the part successfully.</p> <div style="border: 1px solid blue; padding: 5px;"> <p> Note: available for self-support value of Type parameter.</p> </div>
	Keep Core Only	<p>If you only wish to retain the newly created shell, you can check this box.</p> <div style="border: 1px solid blue; padding: 5px;"> <p> Note: available for inside and outside values of Type parameter.</p> </div>					
	Smoothen Core	<p>By checking this option a smoothing will be performed on the created core. (See Smoothing, page 109)</p> <div style="border: 1px solid blue; padding: 5px;"> <p> Note: available for inside and outside values of Type parameter.</p> </div>					
Surface angle	<p>This value defines the self-supporting angle used to generate the internal cavity, making sure that no support is needed to build the part successfully.</p> <div style="border: 1px solid blue; padding: 5px;"> <p> Note: available for self-support value of Type parameter.</p> </div>						
Memory requirements	<p>While you set the parameters, Magics makes an estimation of the quantity of free RAM that will be needed during the calculation and of the number of triangles that will be created. You'll need to enter new values in the Wall thickness and Detail size fields to see a new estimation of the amount of RAM and new triangles. The amount of triangles can later on be reduced with the Triangle Reduction function (see Advanced, page 119). The memory requirements depend greatly on the value set for Detail size.</p>						

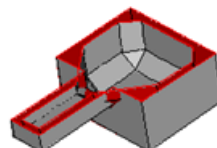
Original



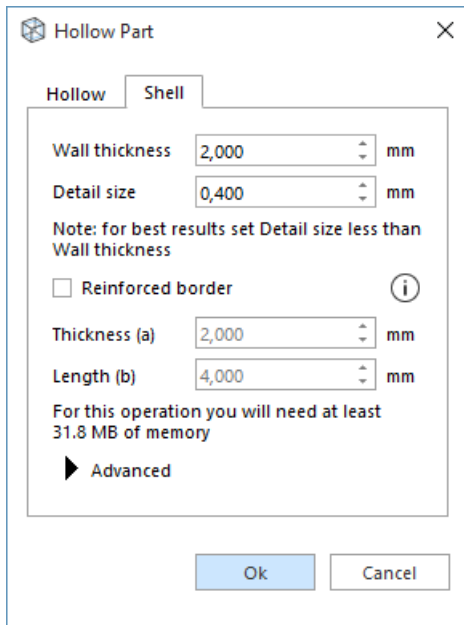
Hollow: inside direction





Detail size value too big



Shell

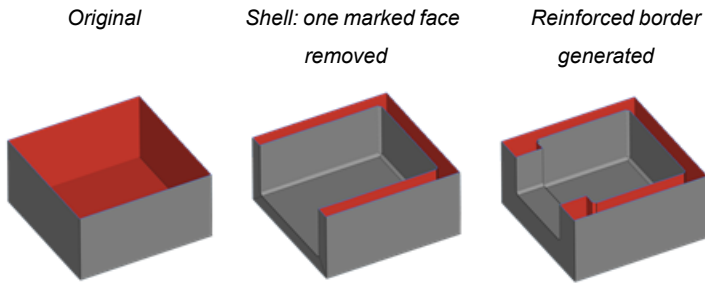


Mark the surfaces on the part that you would like to remove; the remaining part will be hollowed out. The result of the shell operation is normally one STL with one shell, but, depending on the marked surfaces and the part geometry, multiple shells can be created too.

Wall Thickness	This value displays the distance over which the triangles of the original shell get an offset in order to generate a hollow part.	
Detail Size	This value displays the level of detail that will remain in the new shell. Standard, this value should be the same as the smallest detail of the part. The smaller this value, the more triangles will be included in the new shell and the more detail can thus be incorporated.	
	 Remark: If the value is chosen too high, it is possible that the internal wall intersects with the external wall.	
Reinforced border	You can generate an extra border on the open side of the part, to increase strength around the open area or to add stability to the part when the open side lies on the part base.	
	 Remark: the reinforced border will follow the original shape of the marked surface that created the open side.	
	Thickness	You can assign a specific thickness to the border which can be different compared to the wall thickness.
	Length	This value determines the distance between the inner cavity and the end of the border.
Memory requirements	While you set the parameters, Magics makes an estimation of the quantity of free RAM that will be needed during the calculation and of the number of	



). The memory requirements depend greatly on the value set for Detail size.



Advanced

▼ Advanced

Reduce triangles of core

Tolerance mm

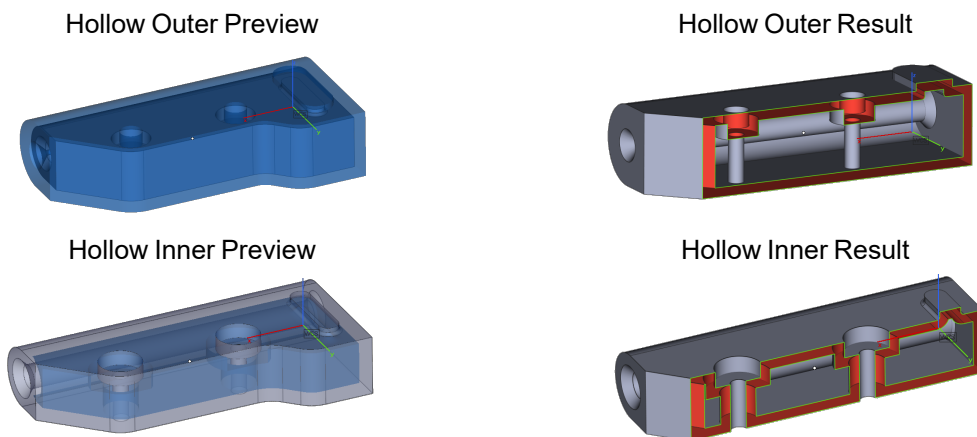
Angle °

Iterations

Reduce triangles of Core	Because the hollow function creates a lot of triangles, you have the possibility to reduce these at once.						
	<table border="1"> <tr><td>Tolerance</td><td></td></tr> <tr><td>Angle</td><td></td></tr> <tr><td>Number of Iterations</td><td></td></tr> </table>	Tolerance		Angle		Number of Iterations	
Tolerance							
Angle							
Number of Iterations							

Hollow for BREP Part(s)

Hollow tool is intended to create an inner or an outer shell on BREP Part(s). It comes with instant preview allowing you to assess result before applying the operation



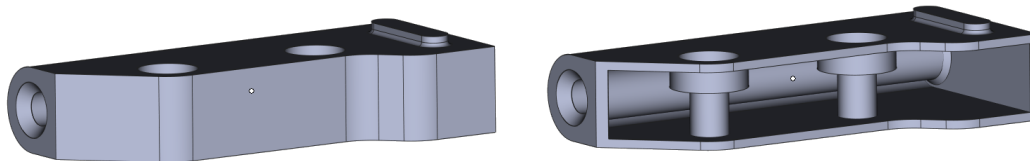
Wall Thickness	This value displays the distance over which the faces of the original BREP Part get an offset in order to generate a hollow part.	
Shell type	Inner Outer	Shell type defines direction in which BREP Part(s) will be hollowed out

Re-hollow for BREP parts

You can also apply Hollow on already hollowed BREP part(s) with the new wall thickness parameter. In this case all the shells of the BREP part(s) except the outer most shell (Shell within the Infinite void region) will be deleted automatically and the outer shell will be re-hollowed with the specified wall thickness automatically.

Shell for BREP part

Shell methods allows to create an open hollow with the specified wall thickness always to the inside of your part.



Indicated pierce face (s)	When this button is focused, you will be able to mark any of the faces on the selected parts as pierce faces for shell operation. You can enable/disable the marking mouse mode by clicking on this button.
Wall Thickness	This value displays the distance over which the faces of the original BREP part get offsetted to the inside

2. Polyline Cut

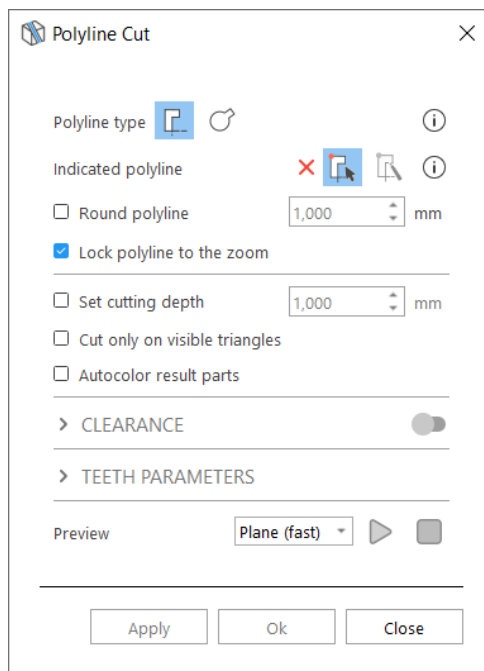
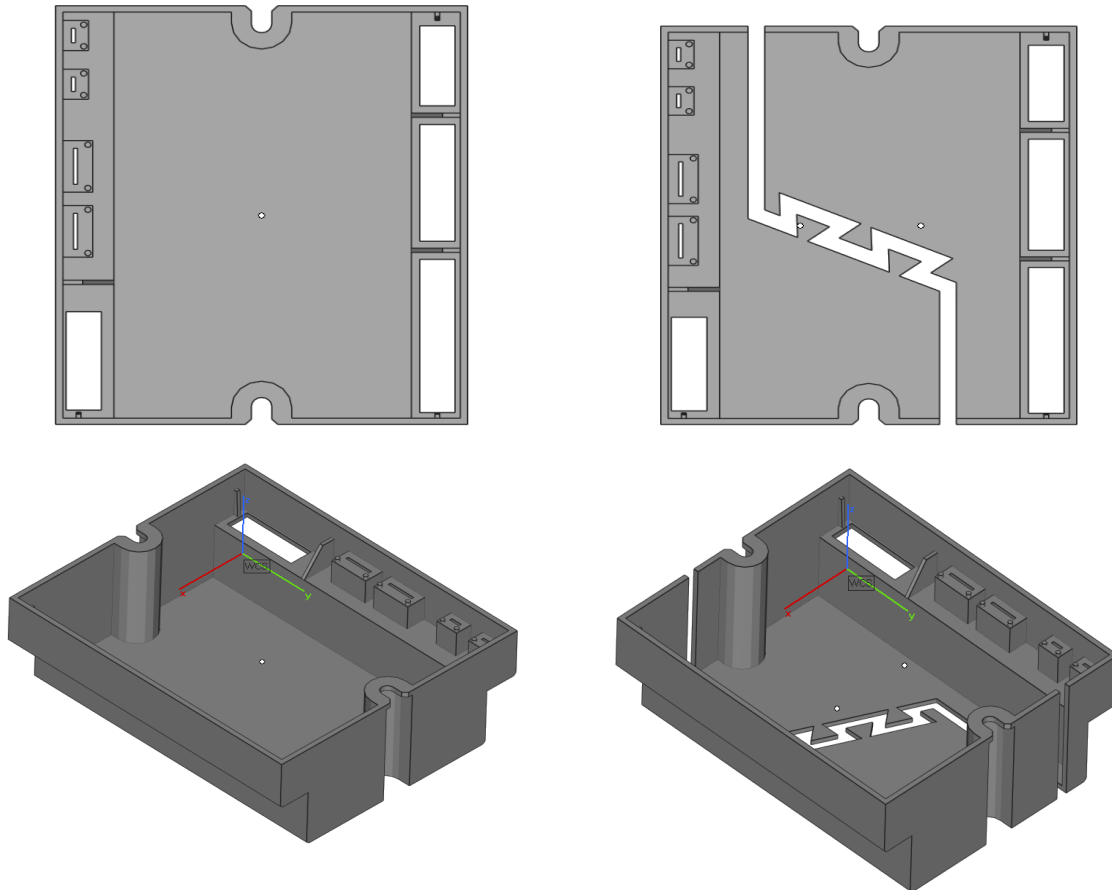


With this command you can cut parts and punch holes through your parts. First a cutting line or polygon shall be defined, which will be used to drive the cut all selected parts along the perpendicular axis to the screen view.


This command is useful to split parts to smaller components quickly where precision and speed is more important than dimensional accuracy.


Part before cut

Part after cut from top view





Polyline type(s)


Polyline type	<p>One of two methods can be selected to specify the type of Polyline.</p> <p> Polygon type: Allows users to specify a few points that</p>
---------------	---

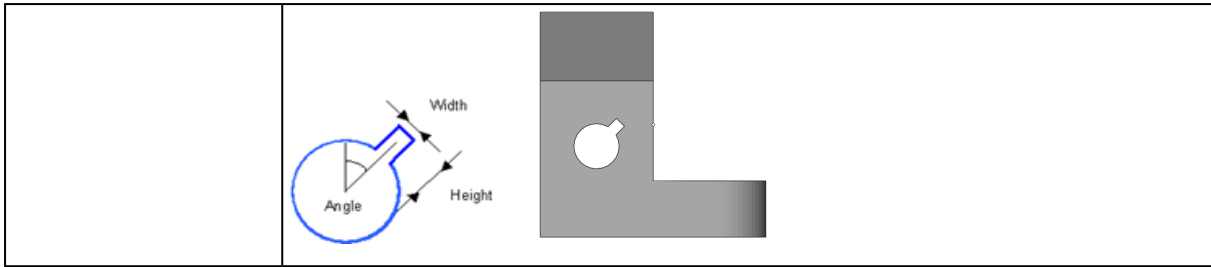
	<p>forms a closed polygon.</p> <p> Circle type: Allows user to pick a point with a smooth circle polyline as the cutting draw.</p>
--	---

Polyline parameters for Polygon type

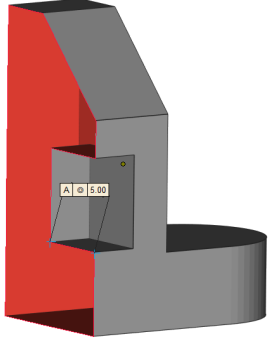

Indicated polyline	<p> Create polyline button activates the mouse mode to specify the polyline points. The button will be checked automatically on launch of the dialog, hence you can specify the point as soon as the dialog is launched.</p> <p>When a valid polyline is set (at least 2 points) then a checkmark will be shown to indicate a valid input has been set.</p> <p> Edit polyline button activates the mouse mode to move or delete existing polyline points. The button will be checked automatically once the polyline is created.</p>
Round polyline	If set to True then the polyline will be rounded on the sharp corners of the polyline polygons based on the rounding radius.
Lock polyline to the zoom	The polyline is locked to your camera of the viewport while panning and zooming is used. Useful to achieve much more precise creation of polyline for cutting.

Polyline parameters for Circle type

Indicated circle	<p> Create circle button activates the mouse mode to specify the circle on the part. The button will be checked automatically on launch of the dialog, hence you can specify the point as soon as the dialog is launched.</p> <p>When a valid circle is set then a checkmark will be shown to indicate a valid input has been set.</p>
Radius	The parameter determines the radius of the circle, and thus the cylinder you will cut out of the loaded part
Tolerance	The tolerance value determines the deviation between the circle drawn by the program (a polygon) and a real circle. It is the distance along the radius perpendicular on and through the middle of one of the edges of the polygon between the cross point of the radius with the polygon and the cross point of the radius with the circle. The higher the tolerance, the bigger the deviation.
Notch on circle	You can choose to add a small notch. The notch is defined by the parameters Angle, Width and Height

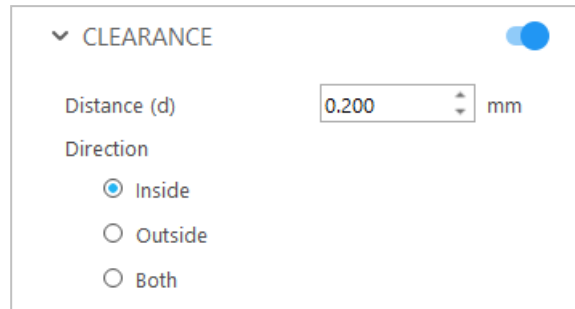


Cut parameters

<p>Set cutting depth</p>	<p>When set to true, user can specify the depth of the cutting along the view direction. If set to False, then the cutting is always done through the selected parts.</p>  <p>cutting depth= 5 mm</p> <div style="border: 1px solid blue; padding: 5px; margin-top: 10px;"> <p> Note: If polyline is created on the curved surface the cutting depth will be calculated starting from the closest to the screen point of this surface</p> </div>
<p>Cut only on visible triangles</p>	<p>Set some section planes to shown with some clipping to cut only the visible triangle. This can be useful to isolate only a specific region of the part to be affected by the cut operation.</p>
<p>Autocolor results parts</p>	<p>All the resultant cut parts will be colored to different unique color respectively.</p>

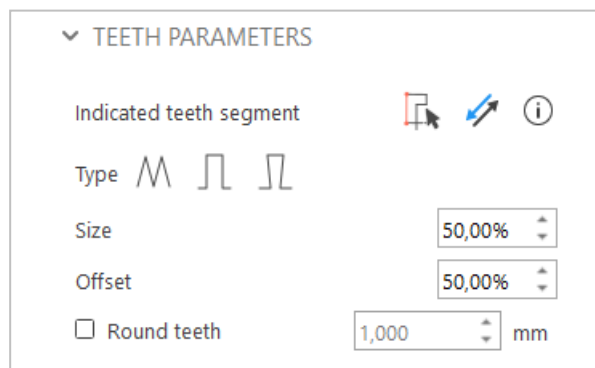



Clearance parameters



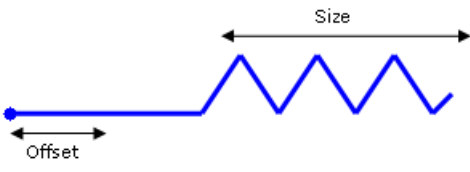

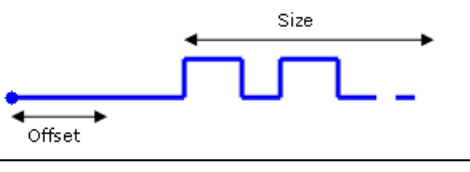

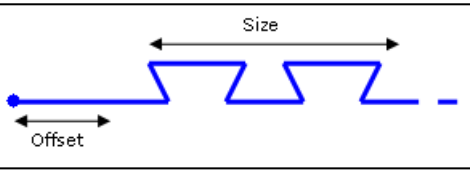


Clearance	If set to True, then a clearance will be created in the specified direction relative to the indicated polyline.	
Distance	The clearance distance.	
Direction	Inside	
	Outside	
	Both	

Teeth parameters



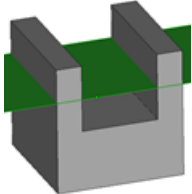


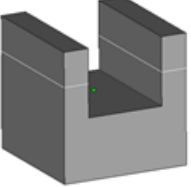
Indicated polyline segment	 Button to activate the mouse to select a polyline segment to be set as teeth segment. As soon as a segment is selected, the segment will be assigned as a teeth segment. This button will only be active when a valid polyline has been set in the Polyline
----------------------------	---

	parameters.  Button to flip the teeth segment direction.	
Type	Sets the type of the selected teeth segment.	
	 Triangle	
	 Square	
	 Jigsaw	
Size	Scale of the teeth segment relative to the rest of the polyline segment. Changing the size will allow you to scale the teeth to the appropriate size for the geometry and zoom.	
Offset	The offset of the teeth segment from the start point of the segment. Changing the offset should allow some degree of control on the position of the teeth feature on the part.	
Round teeth	If set to True then the teeth segment will be rounded based on the rounding radius set. Rounded teeth can be manufacture more easily.	

Preview

The cut preview gives you a first indication how the actual cut will be after applying. In this stage it is still possible to move your part according to the defined cutting line

Preview		<input type="text" value="Plane (fast)"/>  
Play	Start the preview of the indicated polyline	
Stop	Stop the preview of the indicated polyline	
Preview type	Two different types of preview are available:	
	Plane (fast)	The polyline preview cut is represented by a plane. 

	Projection	<p>The polyline preview cut is represented by a projection on the part where the actual cut will take place.</p> 
--	------------	--

Polyline cut mouse modes

Basic Mouse Steps	Activation Method
Create Point	Indicate Polyline Mouse Mode
Move Point	Editing Polyline MM
Delete Point	Editing Polyline MM + Delete key
Close Polyline	Indicate Polyline MM + LMB double click
Clear All Polyline	Indicate Polyline MM + Esc key
Switch Mouse Modes	Right Click on Both Mouse Mode
Indicate Polyline Mouse Mode States	
Create Point	Indicate Polyline MM
Create Point with Angle Constraints	Indicate Polyline MM + Shift
Create Point with Snap to Object	Indicate Polyline MM + Ctrl
Close Polyline	Indicate Polyline MM + LMB double click
Clear All polylines	Indicate Polyline MM + ESC key
Delete Last Created Point	Indicated Polyline MM + Delete key
Combination of Move Points	
Select Point	Editing Polyline MM + LMB Click on a Point
Move Selected Point(s)	Editing Polyline MM + LMB Click & Drag on a Point
Move Selected Point(s) with Angle Constraints	Editing Polyline MM + Shift + LMB Click & Drag on a Point
Move Selected Points Snap to Object	Editing Polyline MM + Ctrl + LMB Click & Drag on a Point
Delete Point	Editing Polyline MM + Delete key + LMB on a Point or with a Selected Point/ Double click on Point
Select Multiple Points	Editing Polyline MM + Rectangular drag over Point (s) / Editing Polyline MM + CTRL + LMB click on Point
Delete multiple points	Editing Polyline MM + Delete key with Multiple selected

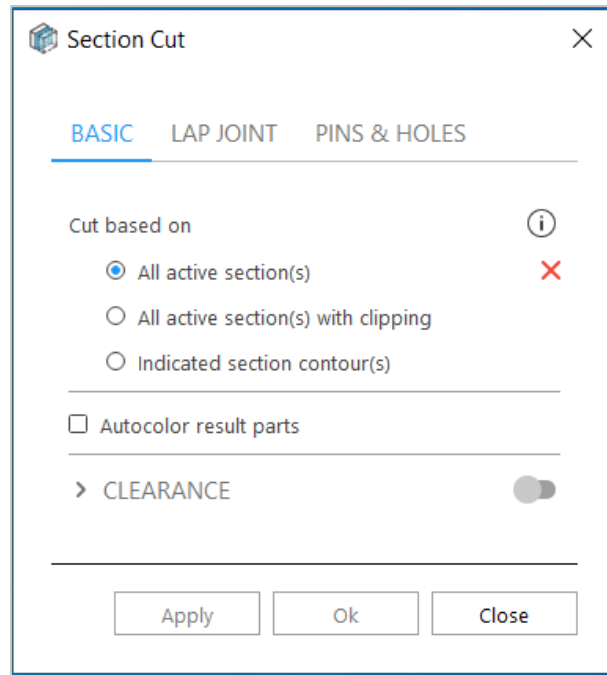
3. Section Cut



This command contains a few different cutting methods that require some active section contour as input. This is usually useful for creating assembly with connector that are along a predefined plane.

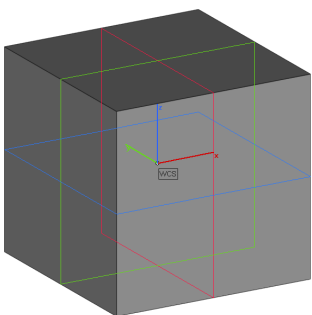
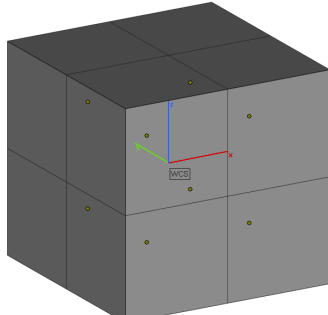
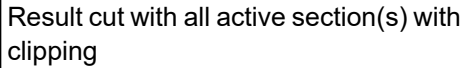
Basic Section Cut

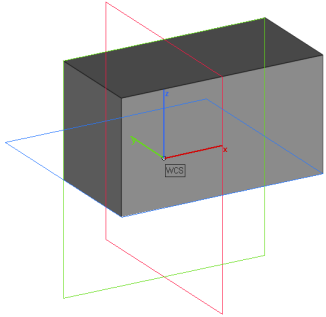
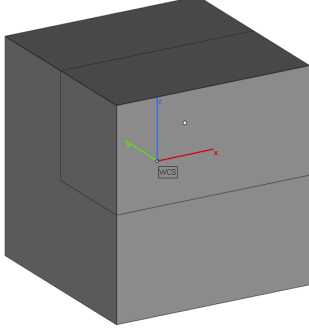

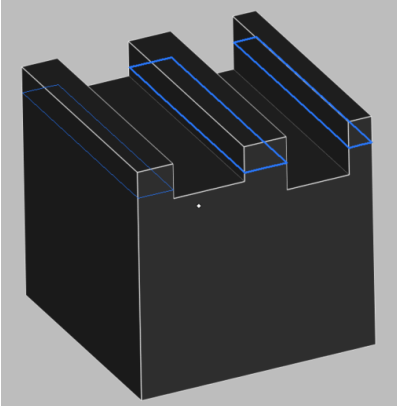
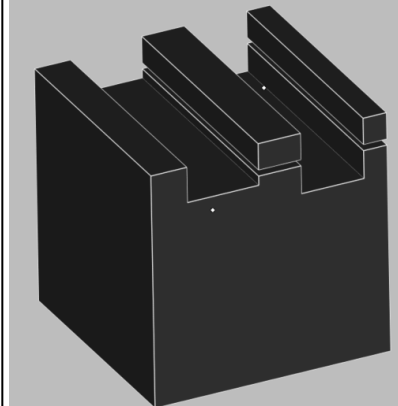

With this method you can do a quick through cut of parts based on the current active section plane(s).




Note: Basic section cut is available for Mesh and BREP part(s).

Cut parameters

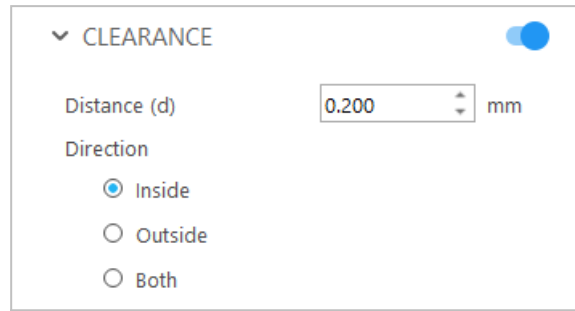
Cut based on	
All active sections	<p>When this option is set, the part will be cut by every section plane that is intersecting the selected part(s) iteratively.</p> <p>Before cut with all active section (s)</p> 
	<p>After cut with all active section(s)</p> 
All active section (s) with clipping	<p>When this option is set, only the visible parts based on the active sections with clipping will be cut. The clipped region of the parts will be preserved as a single part always.</p> <p>Before cut with all active section (s) with clipping</p>
	<p>Result cut with all active section(s) with clipping</p> 

		
<p>Indicated section contour(s)</p>	<p>When this option is set, one or more section contours from the same active section plane can be specified to localize the cut only to the specific region.</p> <p>✓  Button to activate indicate section contour mouse.</p>	
	<p>Before cut with indicated section contour(s)</p>	<p>After cut with indicated section contour(s)</p>
		
<p> Note: Currently cut based on indicated section contour(s) is not available for BREP parts.</p>		
<p>Autocolor result parts</p>	<p>All the resultant cut parts will be colored to different unique color respectively.</p>	

 When an appropriate section plane is not active or an appropriate section contour is not set then the operation cannot be applied.



Clearance parameters

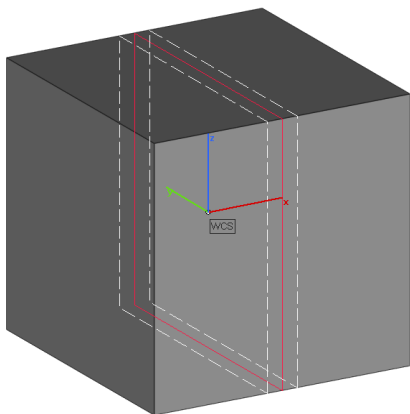


Clearance	If set to True, then a clearance will be created in the specified direction relative to the indicated section.	
Distance	The clearance distance.	
Direction	Inside	
	Outside	
	Both	

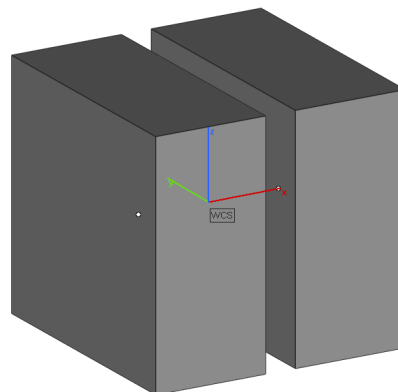
Note: Currently Clearance option is not available when cutting based on indicated section contour(s).

Dotted lines are rendered on the parts with intersecting section planes to indicate the clearance position on the final cut result.

Both sides clearance preview



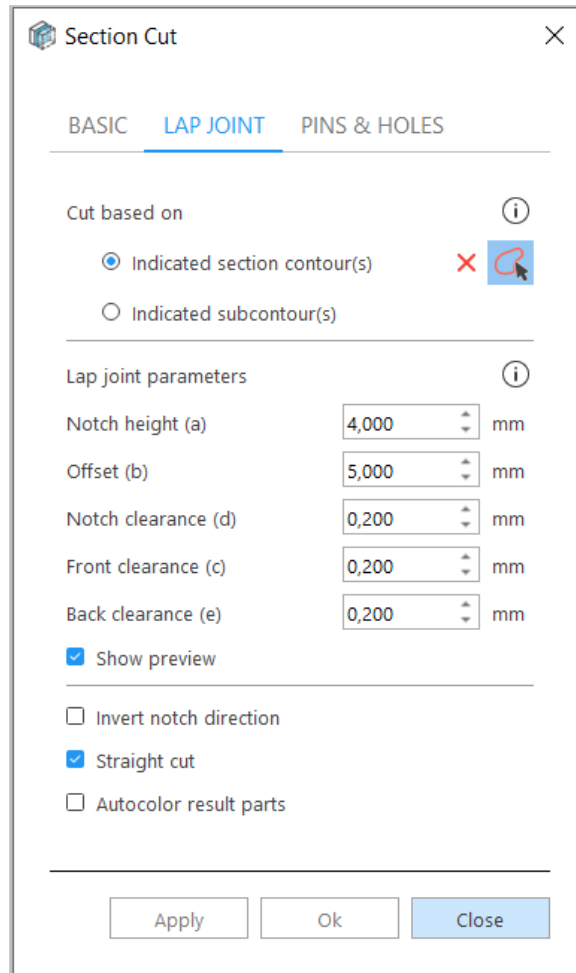
Both sides clearance result







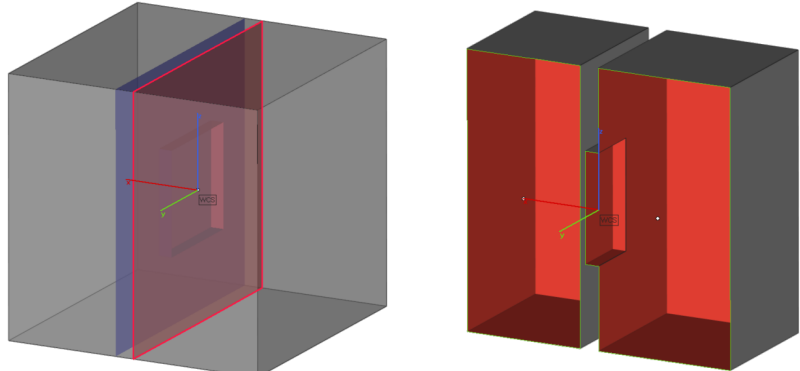

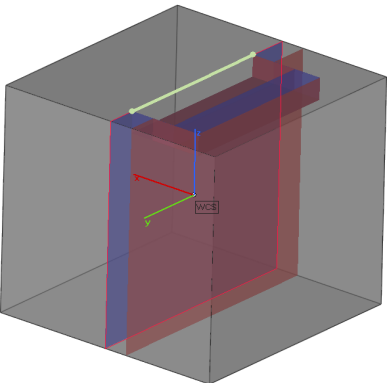
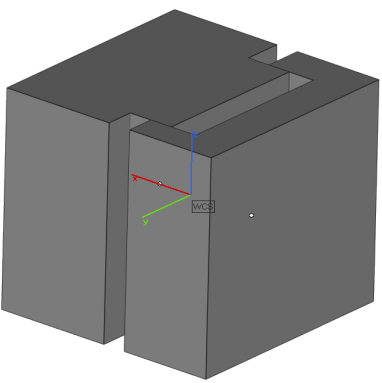
Lap Joint Cut

With this method, you can create cut parts into components with lap joint connector interface to produce assemblies quickly based on a pre-defined section contour.






Note: Lap Joint cut is only available for Mesh part(s).

Cut based on	<p>Indicated section contour (s)</p> <p>When the cut based on indicated section contour(s) is set, one or more section contours from the same active section plane can be specified to create lap joints only on the specified contours after cutting.</p> <p>  When a valid section contour is set then a checkmark will be shown to indicate that accordingly</p> <p style="text-align: center;">Lap Joint preview Lap Joint result (with clipped section)</p>
--------------	---

	
	<p>Indicated subcontour(s)</p> <p>✓  The cut can be performed between a part of a contour limited by two customizable points(subcontour). Multiple subcontours can be selected but they all should belong the same contour of the same part.</p> <p>Once the subcontour is selected, a checkmark will indicate that accordingly</p> <div style="display: flex; justify-content: space-around;"> <div data-bbox="534 907 922 1339"> <p>Lap Joint cut (preview)</p>  </div> <div data-bbox="986 907 1369 1339"> <p>Lap Joint cut (result)</p>  </div> </div>

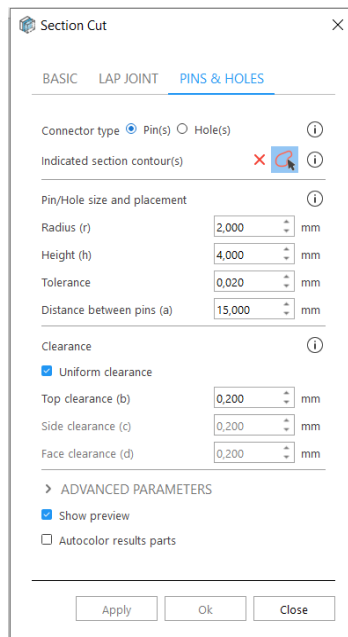
Lap Joint cut parameters

<p>Notch height</p>	<p>This parameter defines the z-coordinate of the incision (height between the top and bottom incision plane.)</p>
<p>Offset</p>	<p>This is the distance between the wall of the part and the step in the cutting surface.</p> <div style="border: 1px solid blue; padding: 10px; margin-top: 10px;"> <p> Remark: The offset cannot be taken too big. It is also important that the part is uniform in the Z direction over the length of the step. Otherwise deformations will occur.</p> </div>
<p>Notch clearance</p>	<p>You can decide to make a little gap along the cut. This way the two parts can easily slide into each other if you have to</p>

	assemble them.
Front clearance	Clearance added in the direction perpendicular to the section.
Back clearance	Gap left at the back side of the cut
Show preview	The preview gives you a first indication how the actual cut will be after applying. In this stage it is still possible to move your part according to the defined cutting line
Invert notch direction(Default OFF)	Notch will be created in opposite direction 
Straight cut(Default ON)	
Autocolor result parts	All the resultant cut parts will be colored to different unique color respectively.

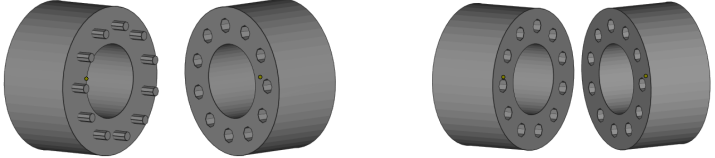

Pins & Holes

With this method, pins and holes can be created along the cutting section contours to simplify part assembling process after printing

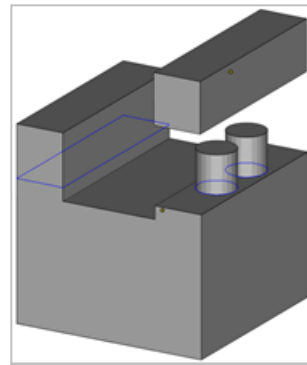
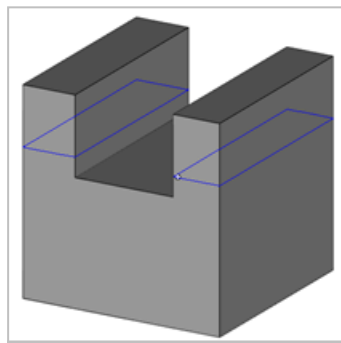


Connector type	Specify which type of connections has to be created.
----------------	--




	<p>Pin(s): Pins on one side and holes on the other side Hole(s): Holes at 2 sides of the performed cut</p> 
Indicated section contour(s)	<p>✓  When a valid section contour is set then a checkmark will be shown to indicate that accordingly</p>

If you have defined an connector Pins/ Holes section, and you only would like to perform the cut on a specific contour of the section, you can do so by using the 'indicate contour' function.



In the figure above a Z section is made. The contour on the right is selected, so only the right 'leg' will be separated from the main part using an advanced cutting.

Pin/Hole size and placement

Radius (r)	Here you determine the radius of the connector pin/hole.
Height (h)	Here you determine the height of the connector pin/hole.
Tolerance	The tolerance parameter influences the number of triangles that will be created.
Distance between pins	<p>Indicates the distance between centers of pins</p>  <p style="text-align: center;">distance = 2 mm distance = 4 mm</p>

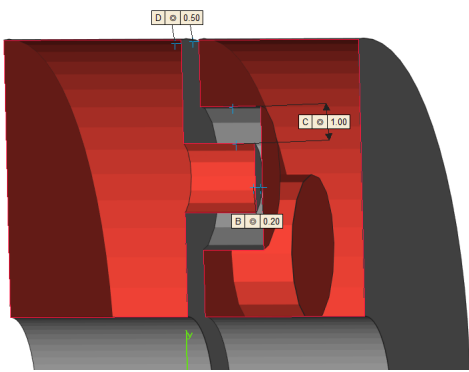
Clearance parameters

Uniform clearance

Top clearance (b) mm

Side clearance (c) mm

Face clearance (d) mm

Uniform clearance	If set to True ,cut results have the same clearance for all the different clearance regions
Top clearance	Clearance add the end of the pin
Side clearance	Clearance that should be applied to the sides of the pins
Face clearance	You can decide to make a little gap along the cut. This way the two parts can easily slide into each other if you have to assemble them.
	 <p>Top clearance = 0.2 mm Side clearance = 1 mm Face clearance = 0.5 mm</p>

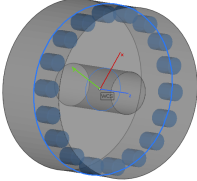
Advanced parameters

Set pin distance from border (e) mm

Remove pins that are close to each other

Remove pins that don't fit the cut surface

Set pin distance from border	The parameter defines how far from the border pins should be placed.
Remove pins that are close to each other	With this option the pins which are too close to each other will not be created to avoid undesired collisions
Remove pins that don't fit the cut surface	If pins go beyond the cut surface they will be removed

Show preview	<p>The preview gives you a first indication how the actual cut will be after applying. In this stage it is still possible to move your part according to the defined cutting line</p> 
Autocolor result parts	All the resultant cut parts will be colored to different unique color respectively.

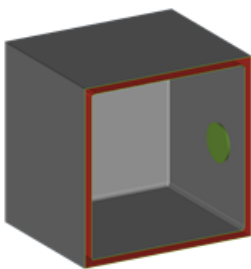
4. Perforator



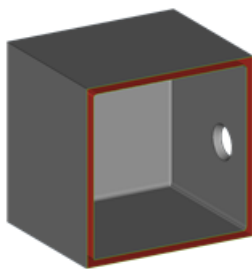
With this command you can make perforations through your parts. The perforation is defined by a flattened cone that will be subtracted from the part, thus creating a perforation. This is especially useful if you work with hollow parts(See Hollow).

Firstly click on a point of the part to create a preview of the perforation. This preview is shown in BLUE color. When a preview is selected, this will be colored in GREEN.

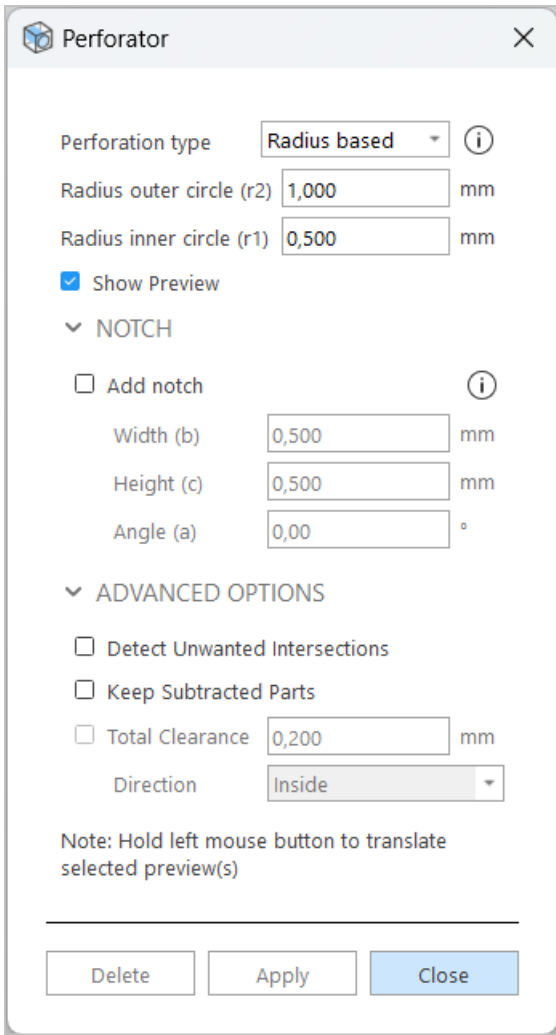
- Edit the parameters. Hold CTRL key to select multiple previews and edit them at the same time.
- Translate selected preview to the desired position by clicking and holding left mouse button.
- Apply preview(s) to the STL by clicking Apply.
- Delete selected preview(s) by pressing Delete key or Delete button.



Selected perforation preview



Perforation applied to the STL



Perforation type	The perforation can be defined by either two radii (Radius based type), or one radius and an angle (Angle based type).	
	Radius outer circle (r2)	Here you determine the radius of the outer circle of the perforation.
	Radius inner circle (r1)	Here you can determine the radius of the inner circle of the perforation.
	Angle (a)	Here you can determine the angle of the perforation
Show preview	A preview of the perforation is shown. This gives you immediately an indication of how the perforation will be executed.	
Notch	Add notch	You can choose to add a notch to the perforation. The notch is defined by the parameters Angle, Width and Height. The notch is particularly useful in combination with “Keep Subtracted Parts” option, for part alignment after printing the parts.
	Width	
	Height	
	Angle	



Advanced options

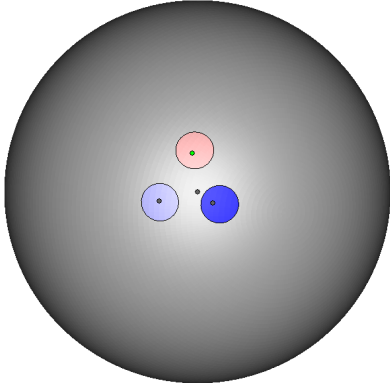
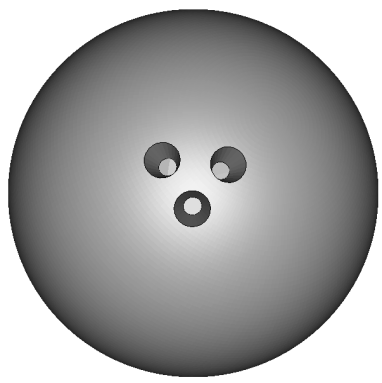
▼ ADVANCED OPTIONS

Detect Unwanted Intersections

Keep Subtracted Parts

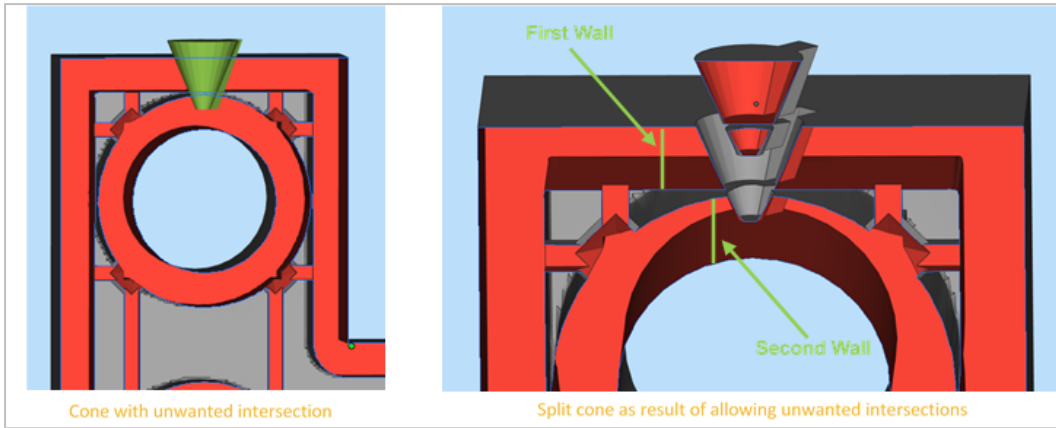
Total Clearance mm

Direction

Detect Unwanted Intersections	Collisions will be detected when the added cone is intersecting with more than one wall. (See Remark: Unwanted Intersections) Default OFF	
Keep Subtracted Parts	When checked, the subtracted part(s) are kept. If not checked subtracted part(s) are unloaded automatically.	
		
	With subtractions	Without subtractions
Total Clearance	If you would like to have a little gap between the two parts that result from the perforation, you can add a clearance. This either to the inside, outside or on both sides.	
	Inside	Subtract clearance from cone
	Outside	Subtract clearance from wall
	Both sides	Subtract clearance from both: half of the clearance is taken from one side and half of clearance is taken from the other side

Remark: Unwanted Intersections

When you want to make a perforation, Magics might detect that an unwanted intersections will take place. Normally a perforation should go through only one wall (crossing just one red area). If the **detect unwanted intersections** option is selected, the warning will be given when the perforation goes through more than one wall of the same part. When you allow unwanted intersections, your subtracted part will be split into two pieces.



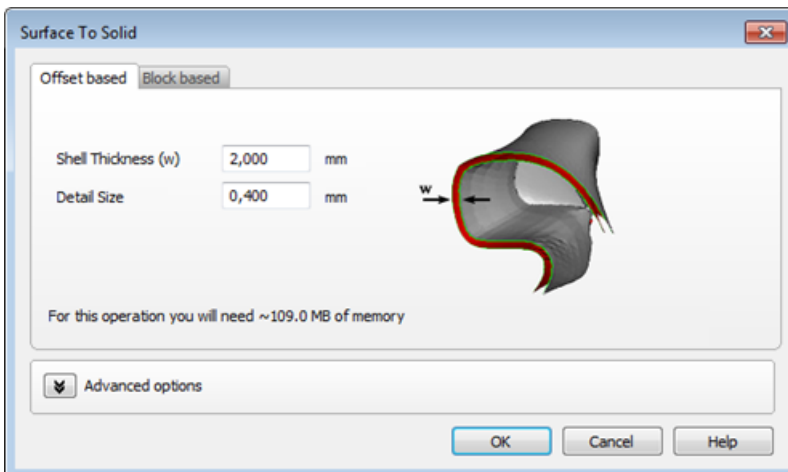
Ok	The cone will be constructed with the intersection.
Cancel	The flattened cone is not constructed.

5. Surface to Solid



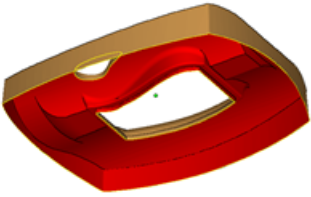
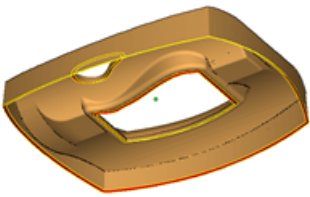
This function creates a solid from a surface. In some applications (e.g. GIS) only a surface is generated, but that cannot be built on an RP machine. Therefore you have to make a solid part.

Offset based



Shell Thickness	This value displays the distance over which the triangles of the original shell get an offset in order to generate the required shell
-----------------	---



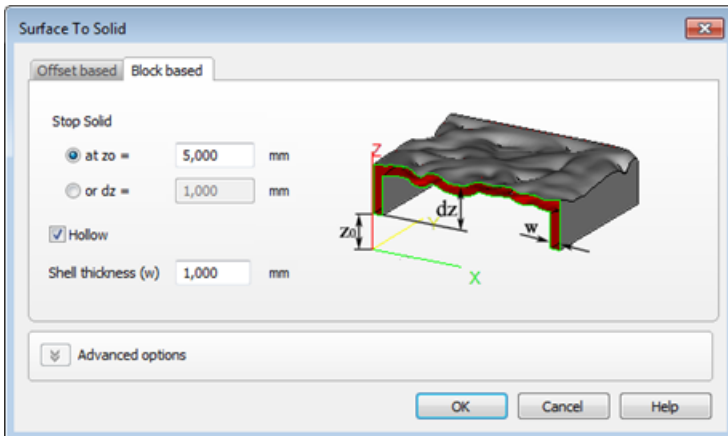
Detail Size	This value displays the level of detail that will remain in the new shell.
	
Shell without thickness	Offset based thickness added to surface

1. Advanced

Remove noise shells		All noise shells (no geometrical sense) are removed from the part
Apply filter sharp triangles on original surface		Filter out long thin triangles on the original surface.
	Max width filter	Triangles thinner than this distance will be removed, depending on your choice.
	Max angle	The thin triangle will only be removed when the angle it makes with its neighbors is bigger than the given angle. This is easy to filter only thin triangles of folds and leave thin triangles of curves untouched.
Close hole (s) automatically		When checked, the gap in between the original surface and newly created surface is automatically closed
	Straight edge	The gap in between the two surfaces will be closed by using a straight edge.
	Sloped edge	The gap in between the two surfaces will be closed by using a sloped edge, as the internally created surface is slightly smaller.
		When checked, Triangle Reduction is applied to the newly created surface.
Apply Triangle Reduction for new surface	Smallest detail	If 2 triangles are replaced by one triangle, there may be a little deviation in position. The tolerance indicates the maximum deviation allowed between the original surface and the new one.
	Max Angle	The Max Angle value defines two limits:

		<ul style="list-style-type: none"> – When two triangles have an angle value bigger than Max Angle, they may not be reduced. The edge between them may not be eliminated; otherwise too much geometrical information would be lost. When the program meets such an edge, the reduction will keep the edge but reduce the number of points on it. – When there is no critical edge, this Max Angle value determines the maximum angle that can be created during the reduction. This means where there is an edge present, there will remain one. Where there is no edge, no edge will be added.
	Number of Iterations	Magics can perform the operation in different iterations to improve the reduction of triangles. It is better to increase the number of iterations than performing the triangle reduction twice (to maintain the smallest detail).

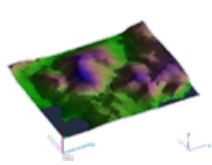
Block based



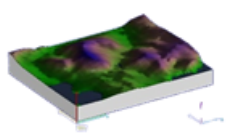
Stop Solid at	The surface you want to make a solid from needs to be positioned above the XY plane.	
	z0	Extrude the solid till this Z value
	dz	Extrude the solid over this Z distance
Hollow	You can hollow the solid to save material.	
	Shell Thickness (w)	The wall thickness of the hollowed solid.



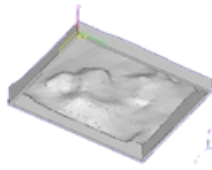
Original Surface



Created Solid



Hollowed Solid



6. Fillet

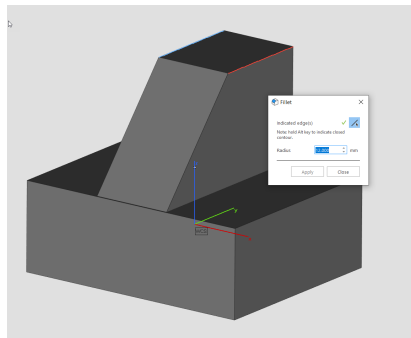


This command allows you to round interior and exterior sharp edges with a constant radius. Before applying the fillet, select single edge(s) of a part on the scene, or hold Alt key to select one closed contour. You can also select multiple edges and/or contours and run the fillet operation at the same time.

Note: Fillet operation is also available for BREP Part(s).

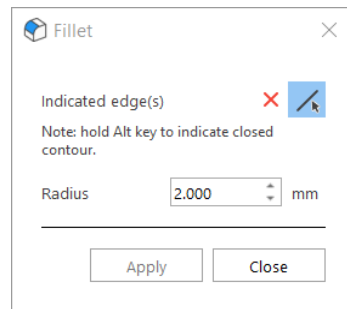
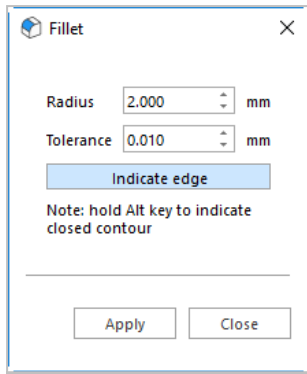
Selected edges		Fillet Result	
Preview on Mesh operation	Preview on BREP operation	Mesh Result	BREP Result

Note: When the operation is unable to perform the rounding successfully, all previews will be hidden. For the operation on BREP Part, at least 1 failing edge will be highlighted in red to indicate a need to modify the selection or the set parameters. You may also attempt to perform the fillet operation one after another in such cases to improve the possibility of operation success.



Fillet Dialog for Mesh Part(s)

Fillet Dialog for BREP Part(s)



Radius	This value defines the radius of the resulting circular edge. Fillet is performed with a constant radius over the entire edge.
Tolerance *	This value determines the maximum deviation allowed between the circle drawn by the software (a polygon) and a parametric circle; the higher the tolerance, the bigger the deviation. The tolerance has also influence on the number of triangles that will be created.

**Tolerance parameter is unavailable for the Fillet operation for BREP Part(s)*

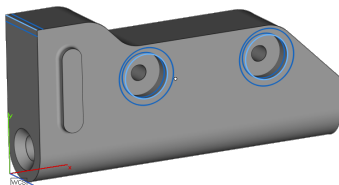
7. Chamfer



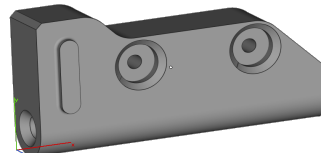
This command allows you to create a slanted edge between two surfaces of a part. Before applying the Chamfer, select single edge(s) of a part on the scene, or hold Alt key to select one closed contour.

Note: Chamfer operation is also available for BREP Part(s).

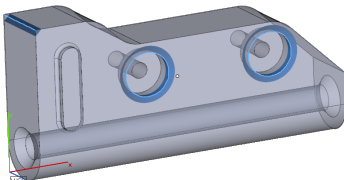
Chamfer Preview for Mesh Part(s)



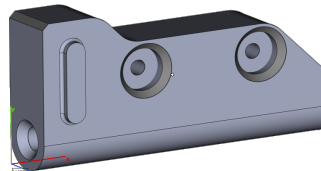
Chamfer Result for Mesh Part(s)



Chamfer Preview for BREP Part(s)

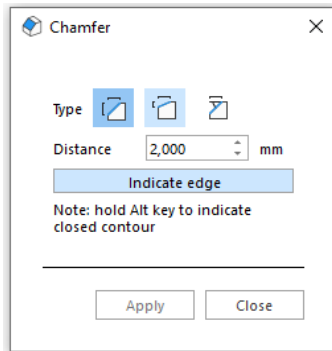


Chamfer Result for BREP Part(s)

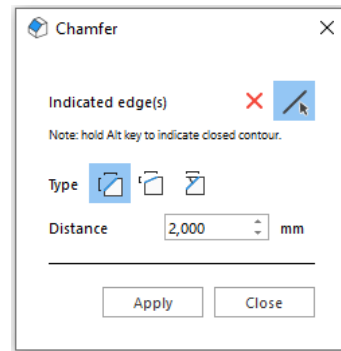


Note: in case the Chamfer operation can't be applied on specific edge (s) of BREP Part (s), the failing edge will be highlighted in red color. In this case, you should adjust the edge or change parameters accordingly

Chamfer Dialog for Mesh Part(s)



Chamfer Dialog for BREP Part(s)



	Equal distance	Distance	This value defines the equal distances calculated starting from the selected edges in both directions.
	Two distances	Distance 1	This value defines the first distance calculated starting from the selected edges.
		Distance 2	This value defines the second distance calculated starting from the selected edges.
		Reverse direction	When selected, Distance 1 and Distance 2 will be calculated as opposite.
	Distance and angle	Distance	This value defines the distance calculated starting from the selected edges in one direction.
		Angle	This value defines the angle created by the part surface where the distance is calculated and the surface of the chamfer.
		Reverse direction	When selected, the distance and angle will be calculated on the opposite side of the edge.

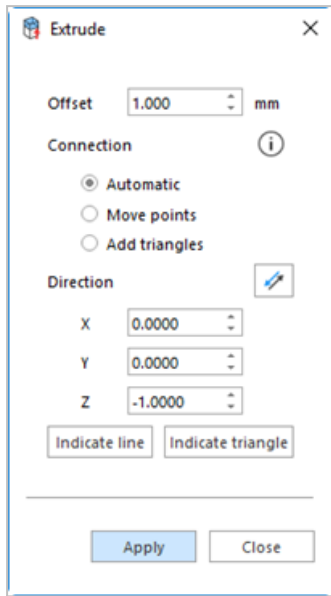
8. Extrude



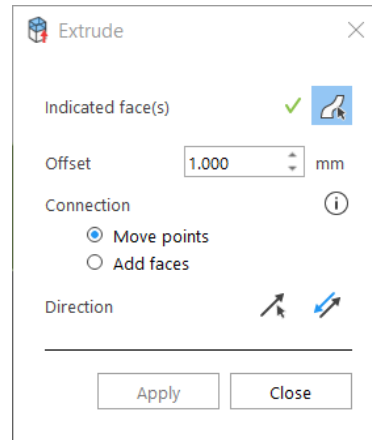
This command allows you to extrude surface(s) in a certain direction. Before the extrusion, you first have to mark the triangles or faces you'd like to extrude. All marked surfaces are moved in the same direction, over a defined distance. A surface can be made up one or more triangles on a mesh part.



Note: Extrude operation is also available on BREP Part(s). On a BREP Part, a surface can be made up of one or more faces.

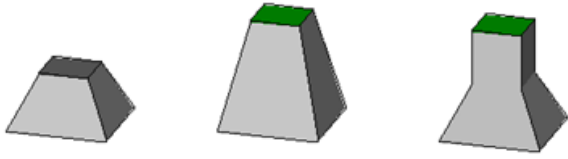


Extrude Dialog for Mesh Part(s)



Extrude Dialog for BREP Part(s)



Offset	The user must specify the extrude offset. Each surface will undergo an offset in the defined direction over a distance defined by this value. The area of the extruded marked surfaces will be unchanged after the operation	
 Indicated Face Button	When this button is focused, user will be able to mark any of the faces on the selected parts for the Extrude operation. User can enable/disable the marking mouse mode by clicking on this button. Note: Currently this is only available for Extrude operation for BREP part(s). <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Remark: Once at least 1 face is marked on the selected part(s), the cross mark will turn to a check mark to indicate that at least some face is marked. </div>	
Connection	Move Points	The triangles or faces adjacent to the selected surfaces will be redrawn. They are extended like shown in the figure below to preserve the current topology as much as possible. The common points are moved. The gradient of the adjacent triangles or faces may be changed.
	Add Triangles/Add Faces	The triangles or faces adjacent to the marked surfaces that are extruded will be unchanged. The common points of the selected and adjacent geometries remain on their position. The gap between the offset geometry and existing geometry is filled with new triangles or faces. This is shown in the figure below.
	Automatic	The program will make the choice between Move Points or Add Triangles.

		<i>Note: This option is not currently available on the Extrude for BREP Part(s).</i>
	<p><i>Original part</i> <i>Move points</i> <i>Add triangles</i></p> 	
Direction	<p>Select the direction that the extrusion will follow. The direction is defined by a vector: fill in the direction vector in the X, Y and Z fields</p> <p><i>Note: For the Extrude for BREP Part(s), it is not possible to set the direction vector manually. You'd have to pick a direction from an edge or planar face.</i></p>	
	 Reverse direction	You can click the Reverse direction button to quickly change the direction of the vector to the opposite side.
	Indicate line**	You can click the Indicate line button and afterwards click on a line. The extrude offset will be in the direction of the line.
	Indicate triangle**	You can click the Indicate triangle button and afterwards click on a triangle. The extrude offset will be in the direction of the triangle normal.
 Indicate direction*	You can click the Indicate direction button and afterwards click on a straight edge or a planar face. The extrude offset for will be along the edge direction or the face normal.	

* Indicate direction button is only available on Extrude for BREP Part(s).

** Indicate line and triangle button is only available on Extrude for Mesh Part(s).

9. Offset



The Offset function for STL Part(s) allows offsetting the whole part or some selected triangles. The triangles are moved along a direction proper to the normal of that triangle over a defined distance, which is the same for all triangles.

The Offset function for BREP Part(s) is similar and allows offsetting the whole body or local face(s) along the same direction.

Offset Operation

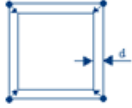
The offset will offset the points of the triangles/faces over the offset distance in a direction defined by the mean of the normal of the triangles/faces adjacent to the point. When looking at a 2D-representation the offset looks like this:





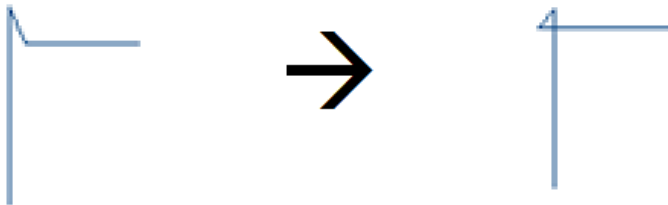
The nodes represent the corners of the triangles/faces; the lines between the nodes represent the triangles/faces; the arrows represent the direction and distance (the vector) of the offset. The offset works on open shells as shown in the example.

In case of a closed surface, we will get the following:



The nodes represent the corners of the triangles/faces; The lines between the nodes represent the triangles/faces; The small arrows represent the direction and distance (the vector) of the offset; 'd' represents the distance one would measure. This will be smaller than the offset distance mentioned in the dialog box.

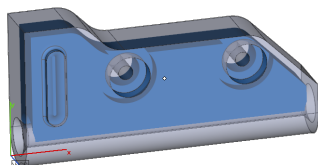
If the STL-file you are working on has some irregularities like in the figure below, the result may not be as expected, but as shown in the right figure.



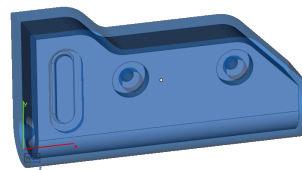
So when you have STL noise, the offset could amplify the noise at bad portions of your part. Therefore we would recommend not using this feature for offsetting over large distances, it is better to use hollow (*Menu/ Tools/ Hollow*) with the option 'keep core only' checked.

The Offset operation for BREP Part(s) is enhanced with real time preview allowing you to assess the output before applying the operation. If preview is not generated, the operation can't be applied and thus some of the parameters must be adjusted to make it a success.

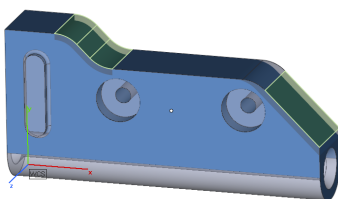
Global Offset Preview Inside



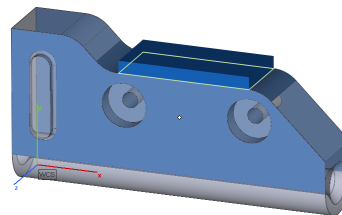
Global Offset Preview Outside



Local Offset Preview Inside

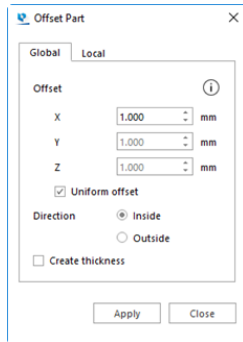


Local Offset Preview Outside

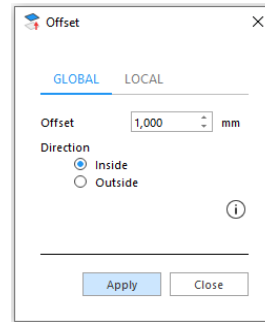


Global

Global Offset Dialog for Mesh Part(s)



Global Offset Dialog for BREP Part(s)



Offset	In this field, you enter the offset value. Each triangle/face will be moved along the direction of its normal over a distance set by the offset value.
Uniform*	Select uniform if you wish that the offset is uniformly applied in X, Y and Z. If you switch it off, the dialog box will change so you can put different offset values in the X, Y and Z direction fields.
Direction	Choose whether you want the offset to be inward or outward.
Create thickness*	If you selected global you can choose to create a thickness. In this case, the original shell will remain and a thickness is created.

* These parameters are unavailable for Offset operation for BREP Part(s)

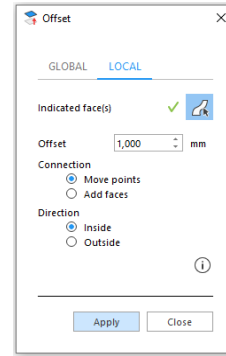
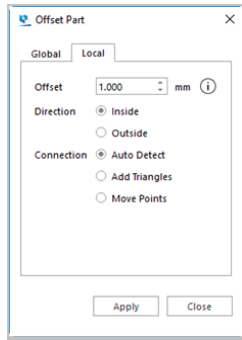
Local





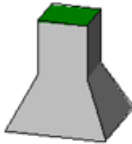
For STL Part(s) you can select (mark) triangles on the part that you would like to offset. These triangles will move. The algorithm is made so that no holes are introduced. There are two possibilities to fill the gap that rises when some triangles have undergone an offset.

For BREP Part(s) you can select local faces after launching the Offset dialog. Once selection is done, the green check icon appears in the Offset dialog allowing you to proceed with the operation

Local Office Dialog for Mesh Part(s)

Local Offset Dialog for BREP Part(s)



 Indicate Face Button	<p>When this button is focused, you will be able to mark any of the faces on the selected parts for the Offset operation. You can enable/disable the marking mouse mode by clicking on this button.</p> <p><i>Note: Currently this is only available for the Offset operation for BREP part(s).</i></p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p> Remark: Once at least 1 face is marked on the selected part(s), the cross mark will turn to a check mark to indicate that at least some face is marked.</p> </div>	
Offset	In this field, you enter the offset value. Each triangle will be moved along the direction of its normal over a distance set by the offset value.	
Direction	Choose whether you want the offset to be inward or outward.	
Connection	Move Points	The triangles adjacent to the selected triangles will be changed. They are stretched like shown in the figure below. The common points are moved. The area of the triangles selected remains the same after the offset. The slope of the adjacent triangles will change a bit.
	Add Faces/Triangles	The common points of the selected and adjacent triangles/faces remain on their position. The gap between these triangles/faces and the triangle/faces that have undergone an offset is filled with new triangles/faces. This is shown in the figure below.
	Auto Detect*	The program will make the choice between Move Points or Add Triangles.
	<div style="display: flex; justify-content: space-around; text-align: center;"> <div data-bbox="384 1581 603 1624">Original part</div> <div data-bbox="608 1581 826 1624">Move Points</div> <div data-bbox="831 1581 1394 1624">Add Triangles/Faces</div> </div> <div style="display: flex; justify-content: space-around; align-items: center; margin-top: 10px;">    </div>	

* This parameter is unavailable for Local Offset for BREP Part(s)



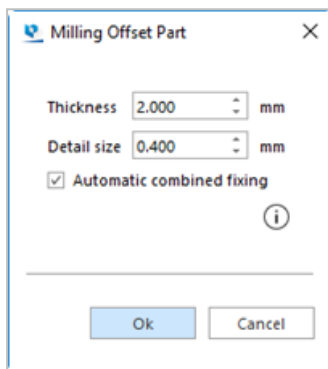
Remark: The local offset differs from the *Extrude* operation. In an extrude operation all triangles/faces are moved along the same direction. In the local offset operation, the direction of offset depends on the triangle/face and its neighbors.

10. Milling offset



The milling offset will add a specified thickness for the marked triangle(s), the edges of the marked area are rounded based on the specified detail size.

By adding this thickness there will be no distortion of the original surface contour.

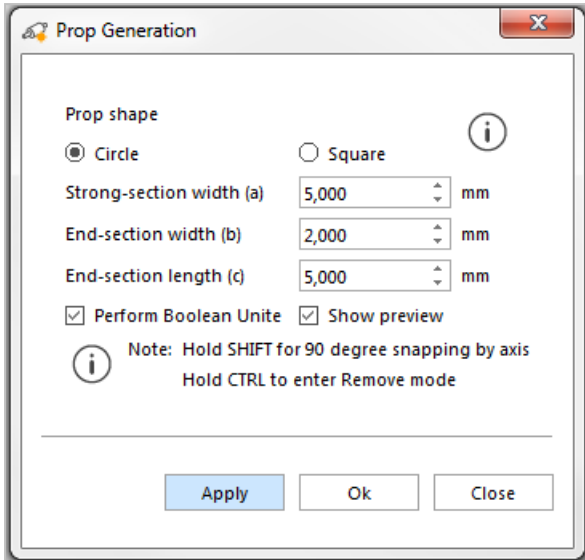


Thickness	The amount of material to be added.
Detail size	This value displays the level of detail that will remain. As a rule, this value should be the same as the smallest detail of the part. The smaller this value, the more triangles will be included and the more detail can thus be incorporated.
Automatic combined fixing	Run automatic combined fixing after the milling offset operation to fix small errors.

11. Prop generation



To avoid distortion on your part during building you can create 'props' to make sure that the shape of the part is kept.



Prop Shape	Circle/ Square	
	The shape of prop of this cross section can be changed between a circle or a square.	
Strong- section width (a)	The width of the circle/ square connector	
End- section width (b)	The width of the connection (in between the strong section and the part)	
End- section length (c)	The length of the connection between the strong section and the part.	

Perform Boolean Unite	The part and prop(s) are united into one file and all surfaces are trimmed to make one shell of both parts.	
Show Preview	While adding the props a preview is created of how the prop will be placed on the part. 	

Hold SHIFT for 90° snapping	After indicating the prop's starting point, holding SHIFT will make sure the generated props are straight (aligned to the world coordinate system, not locally)
-----------------------------	---

Hold CTRL to enter Remove mode	Hold CTRL and click unapplied props to remove them.
--------------------------------	---

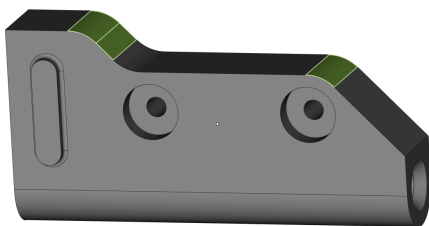


Remark: After creating the prop, it will be merged together with the part.

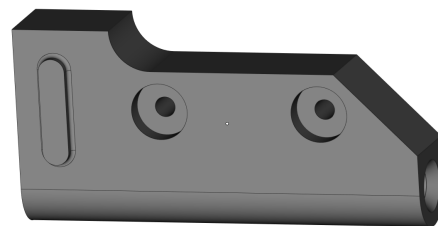
12. Delete Face



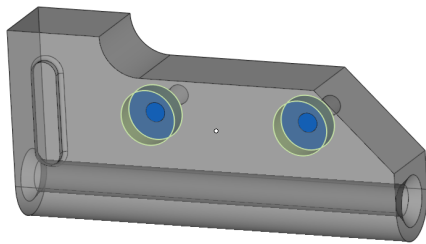
This powerful command allows you to delete selected BREP face(s) and heal it with one of two preferred healing methods.



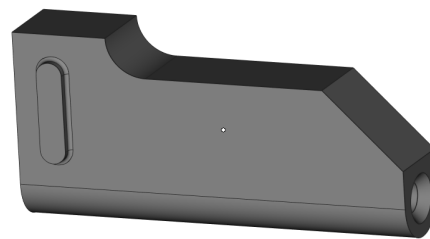
Original BREP with fillets



Result with Grow method

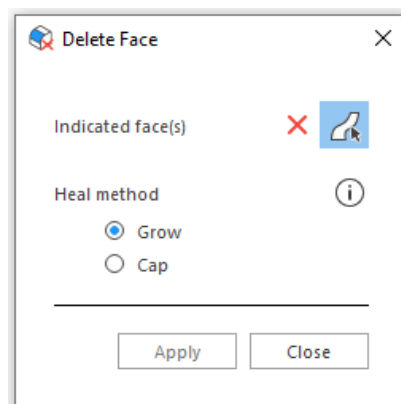


Original BREP with holes



Result with Cap method

Note: in case the Delete Face operation can't be applied on specific face(s) of BREP Part(s), the BREP body will be highlighted in red color. In this case, you should adjust the face or change parameters accordingly



Grow heal method

With the Grow healing method, the adjacent faces will be grown to close the void adaptively based on the existing geometries.



	This method is recommended to remove fillets, chamfers and holes quickly.
Cap heal method	With the Cap option, a new trimmed planar surface will be used to close the void without changing any of the adjacent faces. This method is recommended when the faces shall be deleted without any changes to the existing faces.

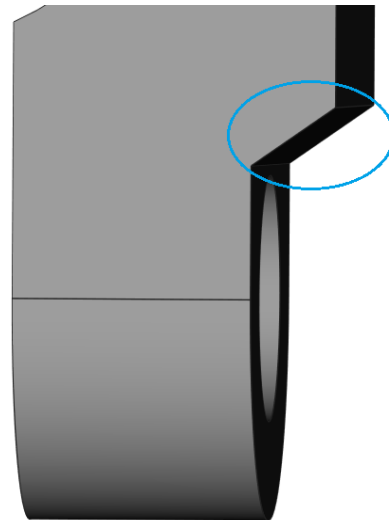
13. Taper Face



'Taper face' tool allows you to taper a single or multiple faces of a BREP model at a specific angle with respect to a reference plane.

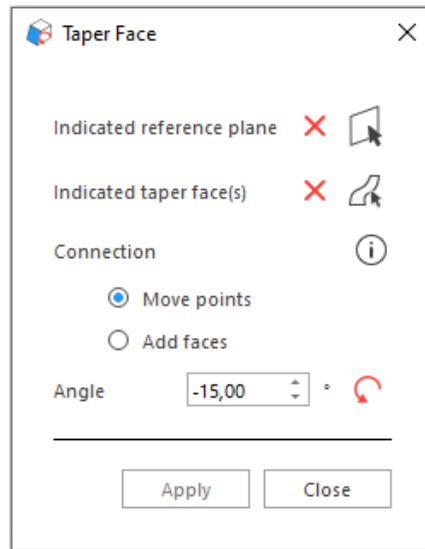


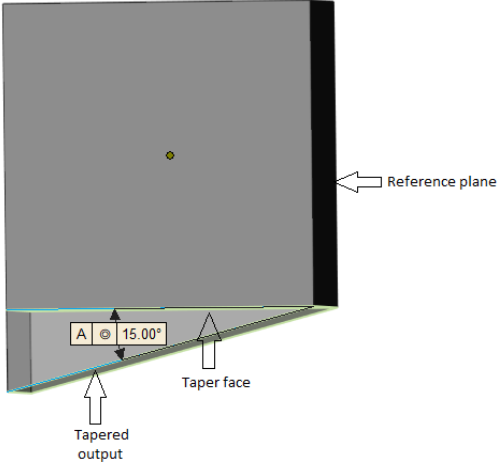
Original BREP

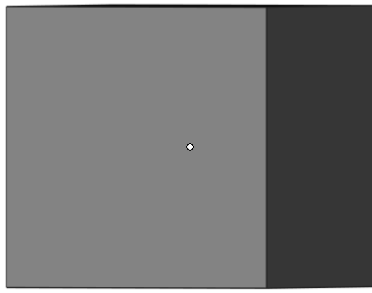


Tapered output

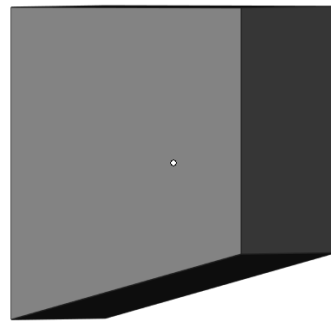
Note: in case the Taper Face operation can't be applied on specific face(s) of BREP Part(s), the BREP body will be highlighted in red color. In this case, you should adjust the face or change parameters accordingly



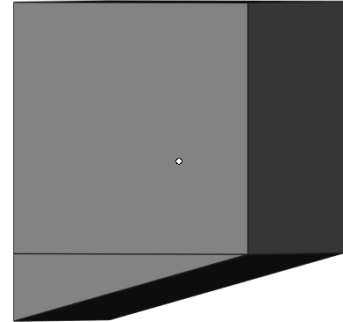
Indicated reference plane	The face to be used as reference for the specified taper face and angle.
Indicated taper face	The face(s) to be tapered. When this button is focused, user will be able to mark any of the faces on the selected parts. User can enable/disable the marking mouse mode by clicking on this button.
Move points	The faces adjacent to the selected ones will be redrawn. They are extended to preserve the current topology as much as possible. The common points are moved. The gradient of the adjacent faces may be changed (see figure below)
Add faces	The faces adjacent to the marked ones will be unchanged. The common points of the selected and adjacent geometries remain on their position. The gap between the tapered geometry and existing geometry is filled with new faces (see figure below)
Angle	<p>It defines the angle for the tapered output face(s). Range (-180 to 180)</p> 



Original BREP



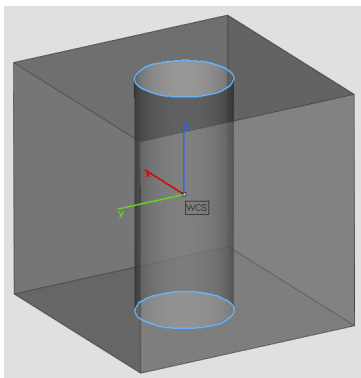
Connection 'Move points'



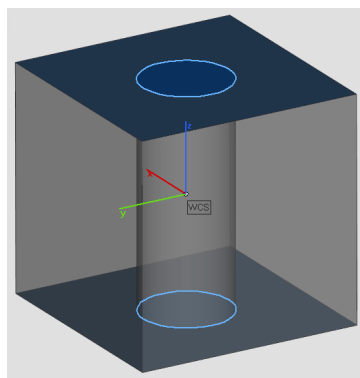
Connection 'Add faces'

14. Adapt Holes

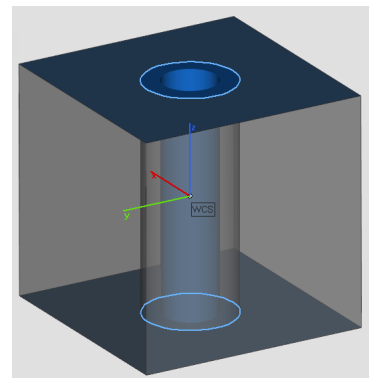
Detect holes on a BREP body quickly based on the hole properties and quickly delete or resize the holes. While most holes can be detected, only constant radius holes can be modified easily by deleting or resizing the hole feature itself.



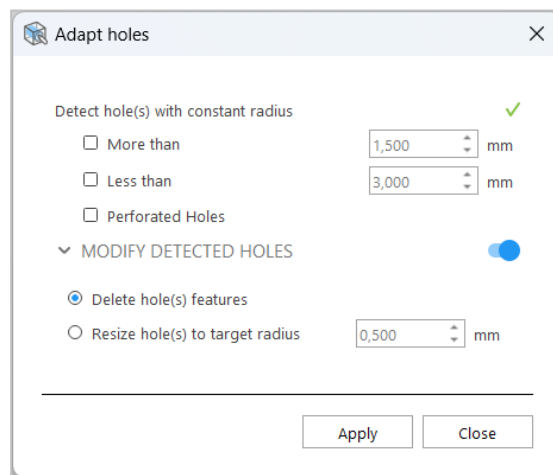
Hole detection



Hole deletion



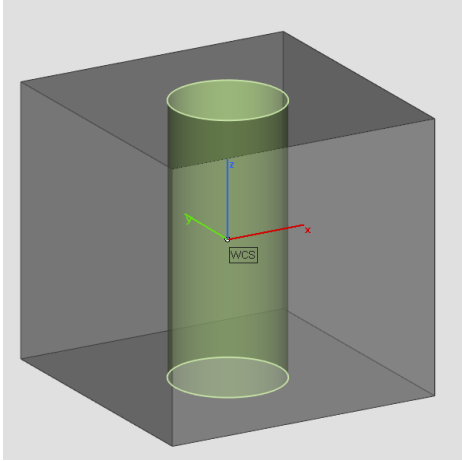
Hole resize



Detect hole(s) with constant radius	This shows the status of the detected hole(s). If a checkmark is shown then hole(s) that are within the filtering options are detected.
-------------------------------------	---

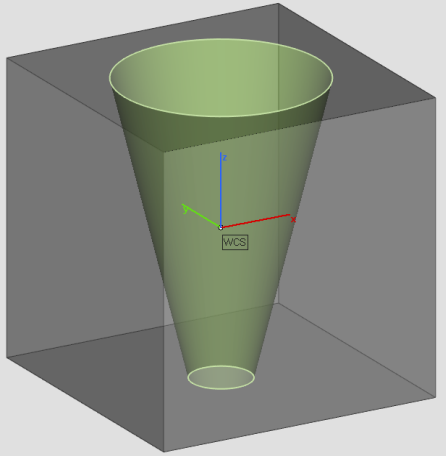
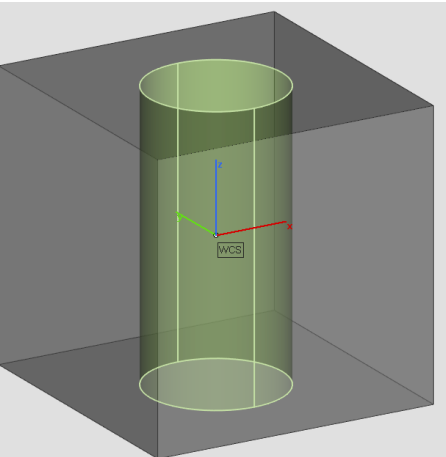
Hole filtering options	More than	Detect hole(s) larger than the specified radius.
	Less than	Detect hole(s) smaller than the specified radius.
	Perforated holes	Detect hole(s) which has other perforated internal features.
Modify detected holes	When modification is set to On, the detected holes will be modified based on the specified modification options else the detected hole(s) face(s) will be marked only.	
Modification options	Delete hole(s) features	When the delete option is selected, all the detected holes will be attempted to be removed with the gaps filled by growing the neighboring face(s).
	Resize hole(s) to target radius	When the resize option is selected, all the detected hole(s) will be resized to the specified target radius.

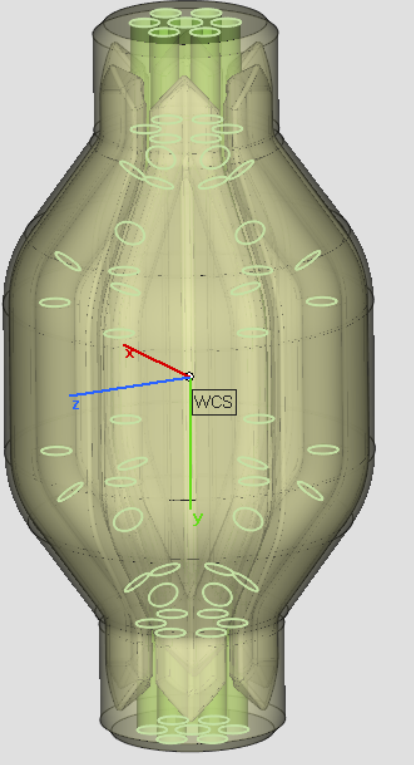
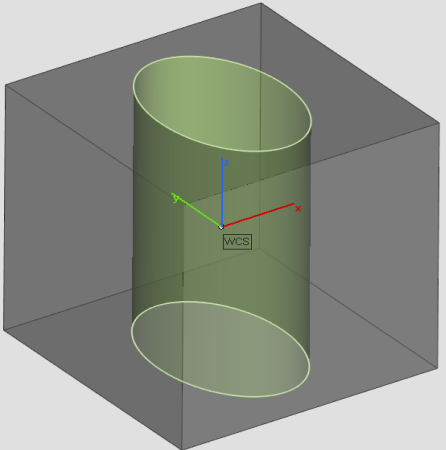
Hole types

Hole classifications	Description	Example
<i>Holes that can be detected currently</i>		
Constant radius	Hole where the radius of the hole is uniform across the whole axis of the hole feature.	<p>Example: Cylindrical hole</p> 
Multi-feature axial hole	<p>Hole feature(s) where there are multiple radius holes with an additional intermediate face(s)</p> <p>Note: There can be holes with multiple radius in this features and only the minimum radius are currently accounted for.</p>	<p>Example: Counterbore or countersunk hole.</p>

<p>Perforated</p>	<p>Hole feature(s) where there are other additional holes through the cylindrical face itself.</p>	<p>Example: Cylindrical hole with a perforation through it</p>
<p>Non-drillable blind holes</p>	<p>Blind hole where the bottom face is not perpendicular to the axis of the hole.</p>	
<p><i>Holes that cannot be detected currently</i></p>		
<p>Variable radius</p>	<p>Hole where the radius of the hole is variable across the whole axis of the hole feature.</p>	<p>Example: Conical hole</p>



		
Multiple-faced	Holes where the central axis of the hole is made up of more than 1 spun- face.	<p>Example: Hole that is made up of 2 different half-circular faces.</p> 
Variable hole axis	Holes with the direction of the central axis changing along the length of the hole.	Example: Circular cooling channels

		
<p>Elliptical</p>	<p>Holes that are are not circular.</p>	

Hole filtering options

The hole filtering radius options can be used with different combinations to detect different types of holes quickly. The table below shows the different conditions of the "More than" and "Less than" values that can be specified to detect holes of different parameters.

Expected filtered hole(s)	More condition than	Less than condition	More than condition > Less than value
Get all cylindrical hole features in the body.	False	False	Not applicable
Get all cylindrical hole features below a certain radius value.	False	True with the specified radius	Not applicable.

		value.	
Get all cylindrical hole features more than a certain radius value	True with the specified radius value.	False	Not applicable
Get all cylindrical hole features that are within a maximum and minimum radius range.	True with minimum radius value.	True with maximum radius value	More than value shall be smaller than the less than value.
Get all cylindrical hole features that are outside the maximum and minimum radius range.	True with maximum radius value.	True with minimum radius value.	More than value shall be larger than the less than value.



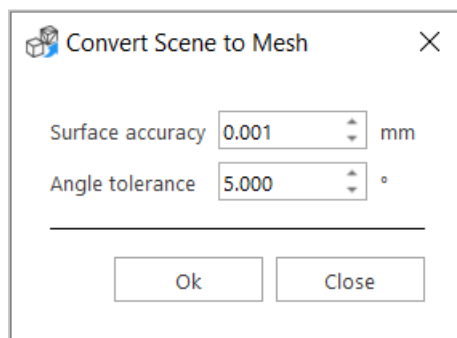
Perforated holes filter can be switched on independent of the radius specifications.

15. Convert Scene to Mesh



With this command, you would be able to quickly convert BREP parts in the current active BREP part scene into mesh parts. All the converted mesh parts will be added to a new mesh part scene. This command is only activated and can be performed in a BREP part scene with at least 1 BREP part. Topological information such as faces and edges that exist on the original BREP part is lost upon the conversion to mesh and the new edges are generated based on the mesh edge calculation. Assembly parts will be converted to mesh in a single merged part.

The default parameters are optimized for best quality for models designed for practical manufacturing models with the average bounding box scales of 10 mm to 500 mm.

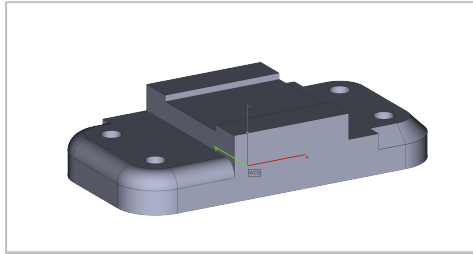


Surface accuracy	Specifies maximum deviation of the mesh representation from its boundary representation (BREP) model. Bigger values produces less triangles, but rougher surfaces while smaller values produce smoother surfaces but more triangles. This unit is in the target system unit always.
Angle tolerance	Specifies the maximum angle deviation allowed between the triangles of the mesh representation on a curved feature. With a smaller angle tolerance much more smoother representations of curved features can be generated on meshes. This parameter is only advised to be changed, when the output mesh representation has long thin triangles or intersecting triangles in its output. This unit is always in degrees of angle.

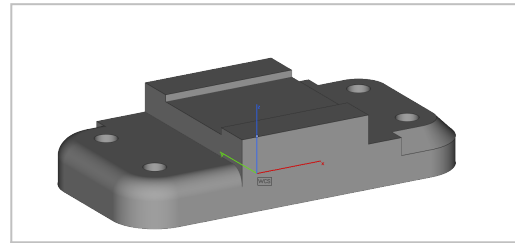


If your part is a larger part, larger than 500 mm, you are advised to consider using a larger surface accuracy value for much more scalable mesh representation.

BREP part before conversion



Mesh part after conversion



4.3. Split & Merge

1. Shells to Parts



If a part consists of several shells, it can be divided so that every shell becomes a distinct part. These parts will be named "shell_#_of" where # is a number. The parts will be sorted in order of amount of triangles. Shell_1 has more triangles than shell_2. This is the inverse operation of Merge Parts.

2. Boolean

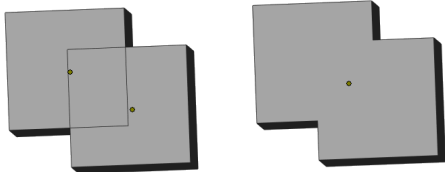
Boolean enables you to combine different design shapes together. There are five types of Boolean: Unite, Intersect, Subtract, Clip and Boolean undercut.

Operation	Select type of Boolean
Unite	Merge the parts into one part and trim all the surfaces to make a single shell out of them. <i>Note: Available for BREP and mesh parts</i>
Intersect	Create intersection of selected geometries. <i>Note: Available for BREP and mesh parts</i>
Subtract	Subtract the selected body(es) from the main body. <i>Note: Available for BREP and mesh parts</i>
Clip	Combine Intersect and Subtract operations in a single step. Create pins or fitted parts quickly. <i>Note: Available for mesh parts only</i>
Undercut	Use it in complex situations where the end product is built in separate parts that need to be assembled afterwards. Detect undercuts along an indicated

	direction and decide to add material from one part to another to have the easiest assembly after the parts have been produced <i>Note: Available for mesh parts only</i>
Indicate part(s)	Select at least 2 parts to perform boolean
Main bodies (Green)	Overview of main bodies (parts) for boolean
Bodies to subtract (Red)	Overview of bodies (parts) to be subtracted from the main bodies <i>Note: move parts from one list to another with drag & drop</i>
Body to undercut (Green)	Overview of main bodies (parts) for boolean undercut
Body to fit (Red)	Overview of bodies to fit for boolean undercut
Mode	Define accuracy of boolean result <i>Note: Available for mesh parts only. Not available for Boolean undercut</i>
Fast	Default option suitable for majority of cases.
Accurate	Recommended for specific cases when high accuracy is required
Clearance	Create a gap between resulted parts by offsetting them towards inner side by clearance value
Add to	Indicate to which part (red or green) the material effected by the undercut should be added
Moving direction	Indicate in which direction the undercut detection needs to be performed
Unload original parts	Unload original parts after boolean is performed
Set result part name	Set custom name for Boolean result

Boolean logic

Unite

Mesh and Mesh = Mesh only	
------------------------------	--

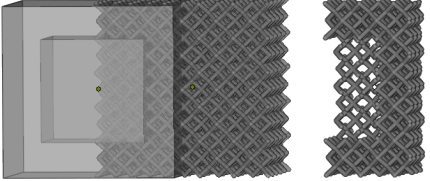
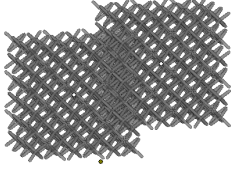


<p>Mesh and Hollow = Mesh only</p>	
<p>Mesh and Lattice = Mesh only</p>	
<p>Hollow and Lattice = Hollow and Lattice</p>	
<p>Lattice and Lattice = Lattice</p>	


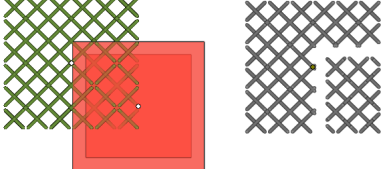
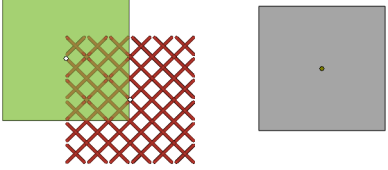
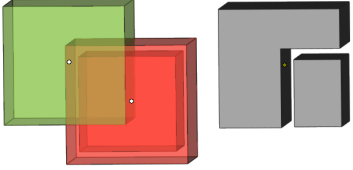
Intersect

<p>Mesh and Mesh = Mesh</p>	
<p>Mesh and Hollow = Hollow</p>	
<p>Mesh and Lattice = Lattice</p>	

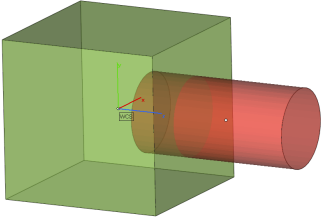
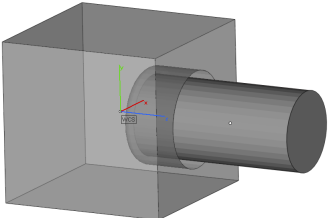


<p>Hollow and Lattice = Hollow</p>	
<p>Lattice and Lattice = Empty</p>	

Subtract

<p>Lattice - Mesh = Empty</p>	
<p>Lattice 1 - Lattice 2 = Lattice 1</p>	
<p>Lattice - Hollow = Lattice</p>	
<p>Mesh - Lattice = Mesh</p>	
<p>Mesh - Hollow = Mesh</p>	

Undercut

	
---	--



3. Merge Parts

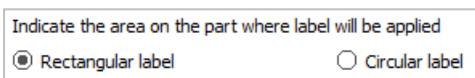


Shells can be merged to one STL part. This function is used to save all loaded parts as one STL. This is the inverse operation of Convert Shells to Parts. Please note that merge will not remove intersecting volumes, like the Boolean operation will do.

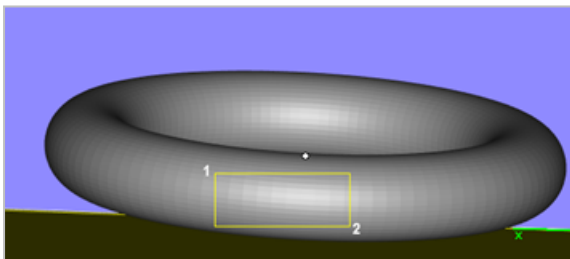
4.4. Label

1. Label

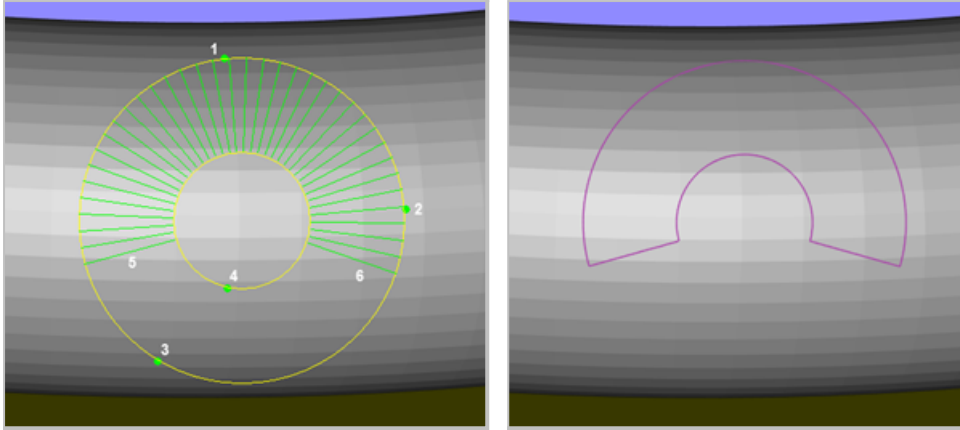
This feature allows you to put text or a figure on a part. First you need to indicate the area where you want the label to be applied. There are 2 options for the label. Or you chose to have a rectangular label or you chose for the circular label.



The rectangular label area needs to be defined by clicking on the part and drawing a rectangle. In the picture point 1 is the starting point of the rectangle, point 2 is the end point. This rectangular area can then be used as label area. Make sure the label area completely fits on the part. A warning message will be shown when the label doesn't fit on the part.



The circular label area needs to be defined by defining 3 points to define the main circular shape. The fourth point will define the size of the label area. The fifth and sixth click will determine the start and end point of the label area.



Label content Auto part name
 Remember text

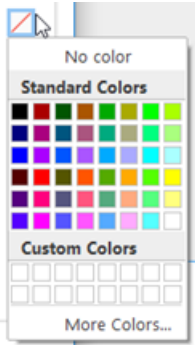





Times New Roman **B** / U / ~~ABC~~

12.0 pt 4,234 mm

Paragraph [Left] [Right] [Align] [Justify]

Label height 1,000 mm [Icon] [Icon]

Label content	Specify in this box the text which should be used as label content.	
	Auto part name	Add the part name of the selected part automatically to the label content.
	Remember text	The last entered value will be remembered the next time the labeling functionality is used.
	Part name	Pushing this button will automatically add the current part name to the label content.

Font options	Font	Specify the font type for the label.
	Bold	Select if the font should be Bold.
	Italic	Select if the font should be Italic.
	Underline	Select if the font should be Underlined.
	Color	Specify the color for the label. 
Font size	Specify the dimension of the label text. The dimension can be specified in mm or pt. If the dimension is specified in mm, then the size in pt will be automatically deduced. If the dimension is specified in pt, then the size in mm will be automatically filled in. *	
Paragraph options	Paragraph 	Will work only if Fit text to label boundaries is switched off. Align the text in the label area to Left/Center/Right or Top/Middle/Bottom.
	Flip 	The label content will be flipped before being applied.
	Mirror 	The label content will be mirrored before being applied.
	Fit text to label boundaries 	Will use the maximal possible size of the text in the label planning area. Overrides any specification on Font size (Font size is greyed out if Fit text to label boundaries is checked).
Label height	Specify the height of the label (in or outside).	
	Raised / Engraved 	Specify if the label needs to be raised (outside) or engraved (inside).

* How is the size determined?

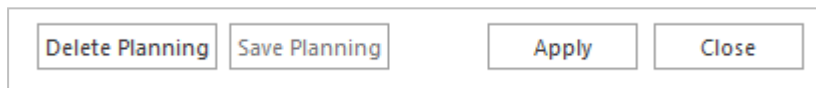
The Size of the text is dependent on the selected Font. The Size of the text is calculated as the distance between the tallest letter and the lowest letter of the selected Font, regardless of the letters used in your label. This makes it easy to be constant in the sizes of the labels created across your applications. It also provides the same understanding and application of Font Size between Magics and other software programs that you are using.

– Advanced options

▼ ADVANCED

- Label through
- Automatically save planning
- Apply as separate STL

Label through	Use this option to label multiple parts at once with the same label. Make sure the parts are aligned perpendicular to the view
Automatically save planning	The label planning area on the part is automatically saved on the part when applied. If the box is unchecked, then the label area will be saved only when the button Save planning (at the bottom of the window) is pressed.
Apply as separate STL	The label will be generated as a separate part.



Delete planning	Use this button to delete previously defined label planning areas.
Save planning	Pressing this button will keep the label planned, but not yet generate it as STL. It can still be edited in a later stage.
Apply	Pressing this button will generate the label and modify the STL.
Close	Press this button to close the dialog.

** In order to process the Label Planning by a Build Processor, an extra license is required and you will have to reach out to your local Materialise contact for it. If the Label Planning is Applied as STL before saving the project and sending it to the Build Processor, no extra license is required.*

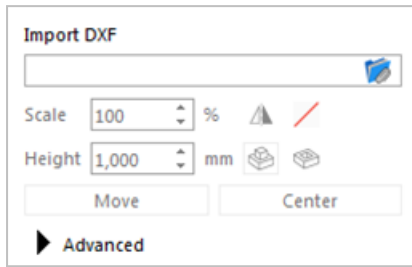
Save label to STL



A part with label planning can be saved as STL, without having to apply the Label to STL in the Label window. The user needs to go to File -> Save as and select STL as the file format. The user will be asked if he/she wants to apply the label planning to the part as an STL. If selected yes, the label will be saved as STL to the part and the part will be saved in a STL file.

The part with Label planning inside Magics will remain unchanged. So the user can continue editing the label planning or save the part with label planning as a Magics project for later editing.

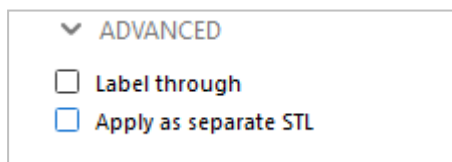


Drawing



Import DXF	Browse to the wanted DXF file by clicking on the folder icon.	
Scale	Resize the label to the desired size.	
Move	Pushing this button allows you to move the 2D label over your screen in order to position it in the optimal position.	
Center	Reposition the label text in the center of the screen.	
Color	Select the color to be applied on the label.	
Height	Specify the height or depth of the label.	
	Raised / Engraved  	Choose whether the label should go inside the part or be placed on top of the part

– Advanced Options



Apply as separate STL	The text will be applied as separate part.
-----------------------	--



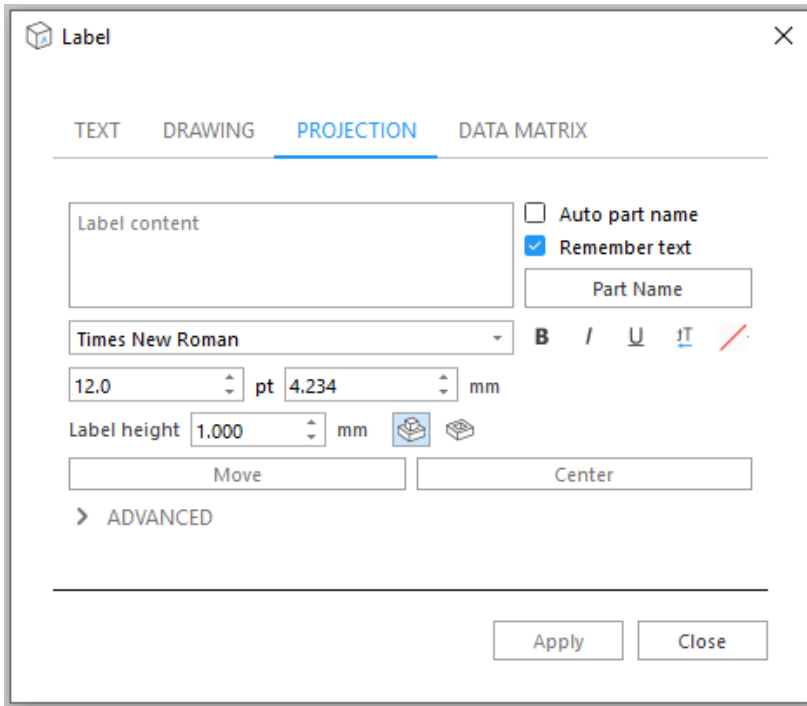
Remark: Pieces of the part can be hidden by using a section view, this allows you to apply labels in the most flexible way.

Projection


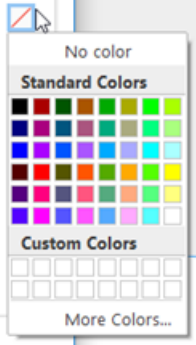


This feature allows you to put text on a part, without having to make a planning first. The label text is first shown on the screen and then perpendicularly projected on the part when applying



the label.



Label content	Specify in this box the text which should be used as label content.	
	Auto part name	Add the part name of the selected part automatically to the label content.
	Remember text	The last entered value will be remembered the next time the labeling functionality is used.
	Part name	Pushing this button will automatically add the current part name to the label content.

Font options	Font	Specify the font type for the label.
	Bold	Select if the font should be Bold.
	Italic	Select if the font should be Italic.
	Underline	Select if the font should be Underlined.
	Mirror 	The label content will be mirrored before being applied.
	Color	Specify the color for the label. 
Font size	Specify the dimension of the label text. The dimension can be specified in mm or pt. If the dimension is specified in mm, then the size in pt will be automatically deduced. If the dimension is specified in pt, then the size in mm will be automatically filled in. *	
Label height	Specify the height of the label (in or outside).	
	Raised / Engraved  	Specify if the label needs to be raised (outside) or engraved (inside).
Move	Pushing this button allows you to move the 2D label over your screen in order to position it in the optimal position.	
Center	Reposition the label text in the center of the screen.	

* How is the size determined?

The Size of the text is dependent on the selected Font. The Size of the text is calculated as the distance between the tallest letter and the lowest letter of the selected Font, regardless of the letters used in your label. This makes it easy to be constant in the sizes of the labels created across your applications. It also provides the same understanding and application of Font Size between Magics and other software programs that you are using.

– Advanced Options

▼ ADVANCED

Label through

Apply as separate STL

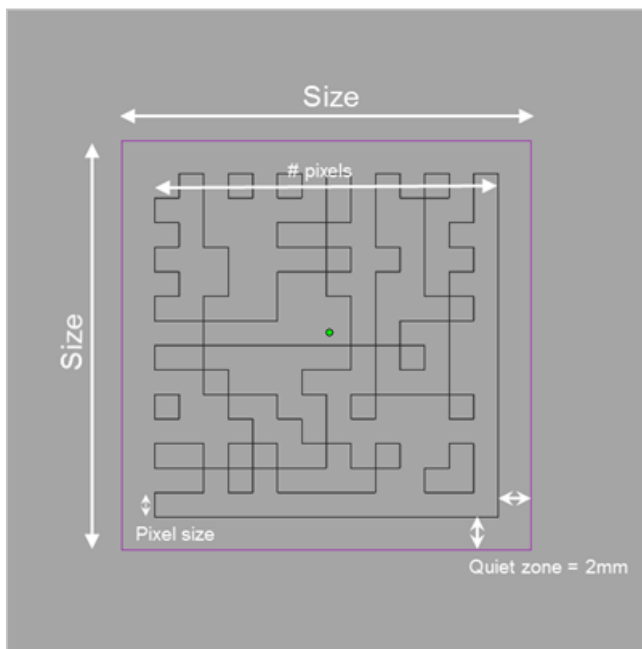
Label through	Use this option to label multiple parts at once with the same label. Make sure the parts are aligned perpendicular to the view
Apply as separate STL	The label will be generated as a separate part.

Data matrix label

This feature allows you to convert text into a Data matrix and apply it to the part as STL. The user needs in the first place to select the area on the part where the label planning will be applied. Once a size is selected from the label window, a preview of the label planning area will be shown together with the mouse cursor. The user needs to position the cursor with the area on the part and click on the part to apply the label planning area.

Once applied, the label planning area can be selected and moved around the part, similar to label planning areas created with other label features.

The following parameters help define the data matrix within the label planning area:



Label

TEXT DRAWING PROJECTION **DATA MATRIX**

Size mm

pixels 0.0555 mm

Label content

Max numeric characters: 172
Max alphanumeric characters: 127

Pixel height (a) mm

Height applied to

Black pixels
 White pixels

Offset label area (i)

Depth (b) mm

ADVANCED

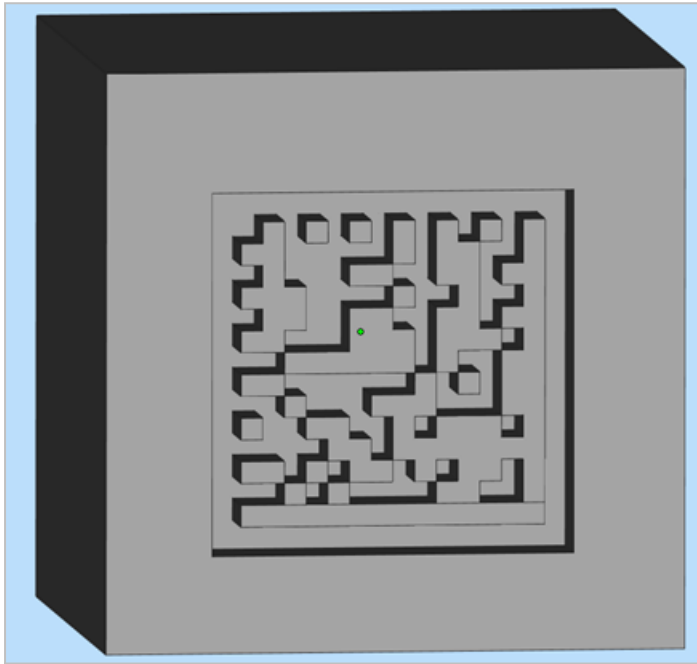
Automatically save planning

Delete Planning Apply Close

Size	This parameter defines the size of the label planning area and, implicitly, of the data matrix. The default shape of the data matrix is a square, therefore all sides equal the value set for the parameter Size (mm).	
(Quiet zone)	The quiet zone is not specified in the UI. The value of the quiet zone is predefined and equals 2mm.	
# pixels	Defines the number of pixels that will appear on each side of the data matrix. The bigger the number of pixels, the larger amount of characters that can be encoded.	
Pixel size	$(\text{Size} - \text{Quiet zone}) / \text{Number of pixels}$	
Label content	The text that needs to be encoded	
Max numeric characters	The maximum number of numeric characters that can be encoded.	
Max alphanumeric characters	The maximum number of alphanumeric characters that can be encoded.	
Pixel height (a)	The height assigned to the pixels which will be read by the scanning device.	
Height applied to	Black pixels	The black pixels will be elevated.
	White pixels	The white pixels will be elevated.
Offset label area	Add an engraved margin around the label area.	
Automatically save planning	The label planning area on the part is automatically saved on the part when applied. If the box is unchecked, then the label area will be saved only when the button Save planning (at the bottom of the window) is pressed.	

Delete planning	Removes a label planning area
Apply	Applies the label planning as STL to the part
Close	Closes the window without making any changes. If the window is closed without applying the label as STL, then the label planning is automatically removed before the window is closed.

Result:



2. Mass Label

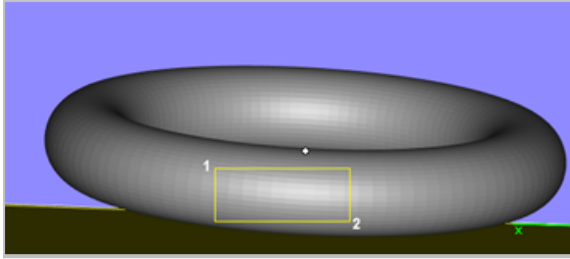


This feature allows you to apply a custom text label on a series of parts in a single batch. First the user needs to select the part which shall be used for the mass labeling operation. This part will be duplicated and a predefined label can be applied on the newly created part. When the master part is selected the user needs to define the label area. There are two different options for the label area: a rectangular label or a circular label.

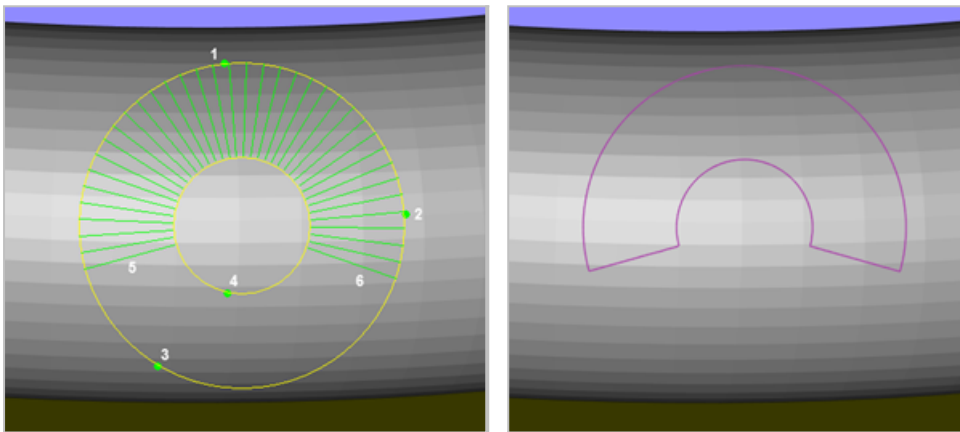
Indicate the area on the part where label will be applied

Rectangular label
 Circular label

The rectangular label area needs to be defined by clicking on the part and drawing a rectangle. In the picture point 1 is the starting point of the rectangle, point 2 is the end point. This rectangular area can then be used as label area. Make sure the label area completely fits on the part. A warning message will be shown when the label doesn't fit on the part.



The circular label area needs to be defined by defining 3 points to define the main circular shape. The fourth point will define the size of the label area. The fifth and sixth click will determine the start and end point of the label area.

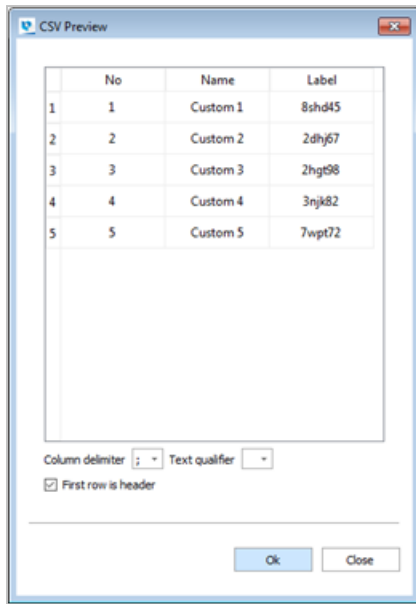


CSV labeling

The label content can be retrieved from a predefined CSV document from which the location needs to be specified inside the CSV tab.



When a CSV file is selected the location will be visible in the CSV tab. Also a preview of the content is shown. In this window you can define if the first row will be used as header or not. You can use the first row to easily distinguish between different columns like for example final part name and label content. In our example it looks like this:

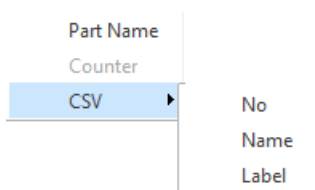


In the example below we use a file with 2 columns. The first column contains the desired part name after the mass labeling operation and the second column contains the label content for the specific parts.

Once a proper CSV document is selected the label content and part name content edit boxes become active. You can type any text in these text boxes or use the Add content button. Behind this button you can choose the part name or the content from the CSV file. As you can see in this example there are 2 CSV options: Name and Label.

Label content

Part name content



After selecting the name and label for the desired result the editboxes look like this

%%ML.CSV.Name%%

%%ML.CSV.Label%%



Also a preview is shown in the label planning area.



Counter labeling

In addition to the CSV labeling option you can add a counter to a label which you can specify manually in the Label content editbox.

CSV	Counter	
Number of copies	2	▲ ▼
Start	1	▲ ▼
Step	1	▲ ▼

The counter allows you to define the number of copies that will be created and also allows you to create a unique label content by use of a counter.

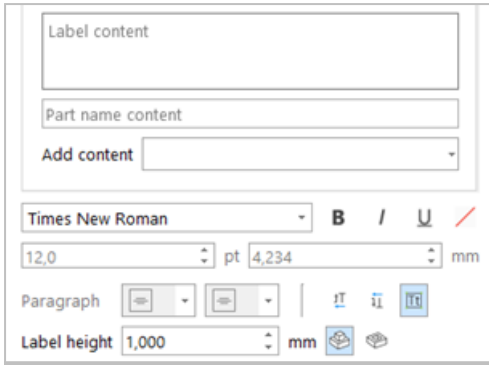
In the Add content dropdown menu you can select the counter option which can be added in the Label content and Part name content editboxes.

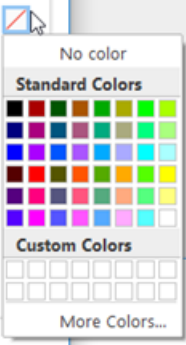




Add content ▼
Part Name
Counter
CSV ▶


In this case a possible option would be:

Materialise %%ML.Counter%%
Torus_%%ML.Counter%%

Finally some additional parameters can be defined for the label.

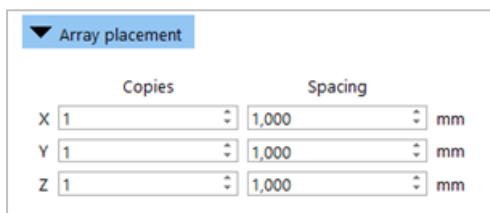


Add content	Use this dropdown button to load counter content in the label area and in the Part name area.	
Font options	Font	Specify the font type for the label.
	Bold	Select if the font should be Bold.
	Italic	Select if the font should be Italic.
	Underline	Select if the font should be Underlined.
Font options	Color	Specify the color for the label. 
	Font size	Specify the dimension of the label text. The dimension can be specified in mm or pt. If the dimension is specified in mm, then the size in pt will be automatically deduced. If the dimension is specified in pt, then the size in mm will be automatically filled in. *
Paragraph options	Paragraph 	Will work only if Fit text to label boundaries is switched off. Align the text in the label area to Left/Center/Right or Top/Middle/Bottom.
	Flip 	The label content will be flipped before being applied.
	Mirror 	The label content will be mirrored before being applied.
	Fit text to label boundaries 	Will use the maximal possible size of the text in the label planning area. Overrides any specification on Font size (Font size is greyed out if Fit text to label boundaries is checked).

Label height	Specify the height of the label (in or outside).	
	Raised / Engraved 	Specify if the label needs to be raised (outside) or engraved (inside).

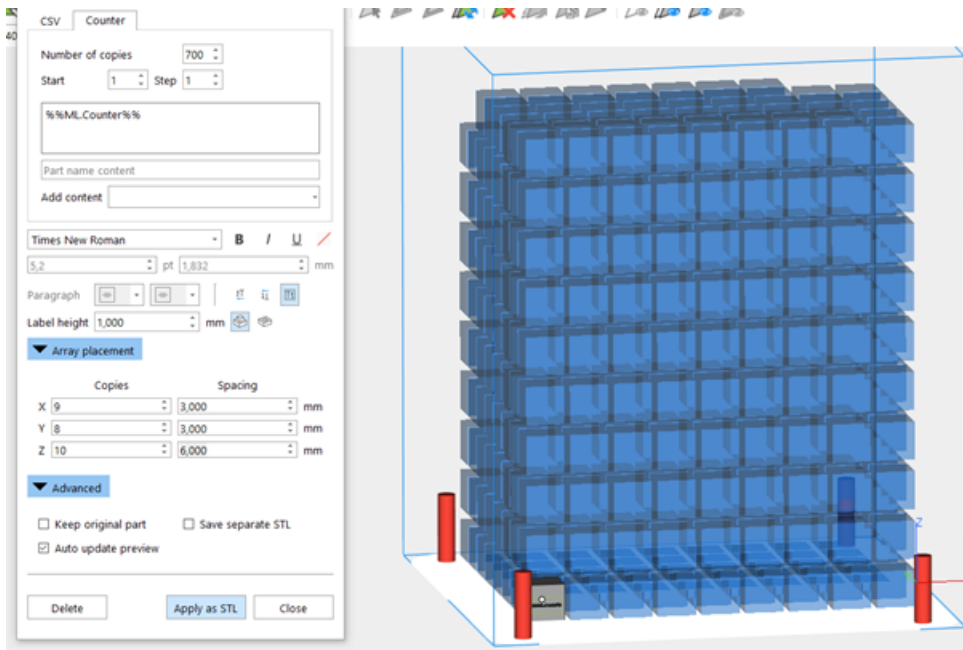
* How is the size determined?

The Size of the text is dependent on the selected Font. The Size of the text is calculated as the distance between the tallest letter and the lowest letter of the selected Font, regardless of the letters used in your label. This makes it easy to be constant in the sizes of the labels created across your applications. It also provides the same understanding and application of Font Size between Magics and other software programs that you are using.



The parts can be positioned on the platform by using the Array placement. Both negative and positive values can be used for Spacing.

A preview of the arrangement is shown automatically. To switch this off, in Advanced settings, the checkbox Auto update preview needs to be unchecked.



The advanced options:

▼ **Advanced**

Keep original part Save separate STL

Auto update preview

Keep original part	The master part used for the mass labeling will be kept after the operation.
Automatically update preview	The content of the label content textbox will be automatically updated in the label planning area.
As separate STL	The label will be generated as a separate part.

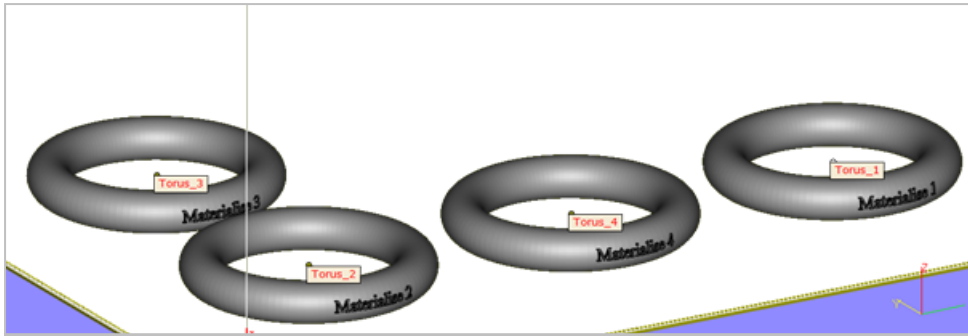
Delete
Apply as STL
Close

Delete	Pressing this button will allow you to go in the delete mode to actually delete previously created label planning areas.
Apply as STL	Pressing this button will generate the mass labels according to the settings specified in the mass labeling dialog.
Close	This button will close the dialog without performing the actual mass labeling operation.

In our example for the CSV labeling the result will be as follows after the label is applied. 5 parts with all different labels and different part names.



In the case of the counter labeling example, the result would be like this: 5 parts with all a unique name and label defined by the counter.

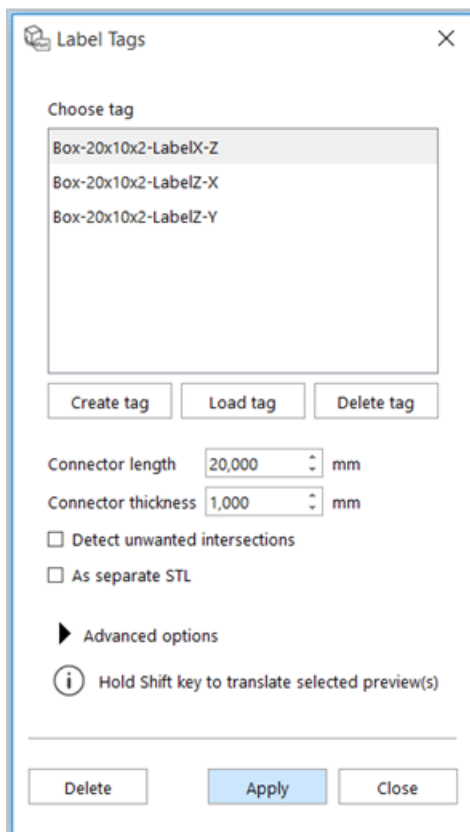



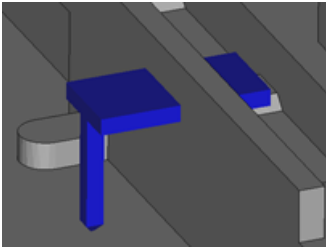
3. Label Tags

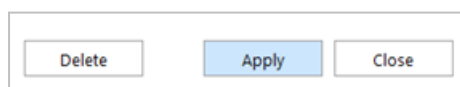


This feature allows you to connect a label tag at a part. You need to identify on the part where the label tag must be placed; just click on desired location on the part, and a preview of the label tag will be shown in BLUE color. To select a preview, just click on it, and it will become GREEN. Since the tag is not yet generated, you are still able to:

- Edit the parameters. Hold CTRL key to select multiple previews and edit them at the same time.
- Translate selected preview(s) by holding Shift key.
- Delete selected preview(s) by pressing Delete key.



Choose tag	This shows a library with unit tags which can be used to create label tags.
Create tag	Create a new tag out of part that is selected on the scene. This part can contain label planning information; this will be saved together with the part for fast editing once adding this tag to a part. The tag will be automatically saved as a *.matPart file in the Label Tags library.
Load tag	Import an existing *.matPart file and add it to the Label Tags library.
Delete tag	Remove the selected tag from the Label Tags library.
Connector Length (a) and Connector thickness (b)	
Detect Unwanted Intersections	The possibility exists that an unwanted intersection is created. Magics will detect this and give a warning.  <i>Label tag preview in blue</i>
As separate STL	The generated tag will become a separate part on the Part list.



Delete	Remove the selected preview(s).
Apply	Convert the preview(s) to STL and boolean it to the part.
Close	Close the dialog.



Advanced Options

▼ Advanced options

Break off point

Tip diameter mm

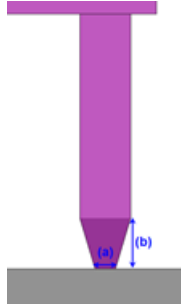
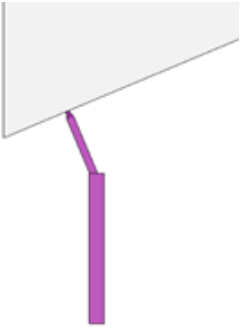
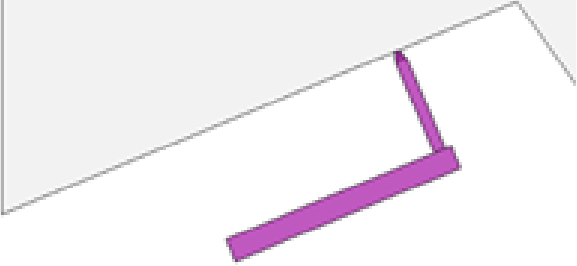
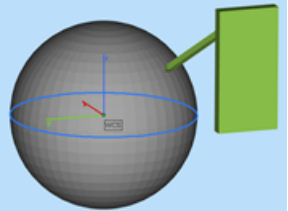
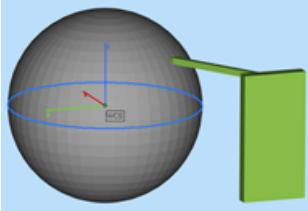
Offset mm

Rotate structure

Lock rotation Z

Lock rotation 90°

Horizontal connector

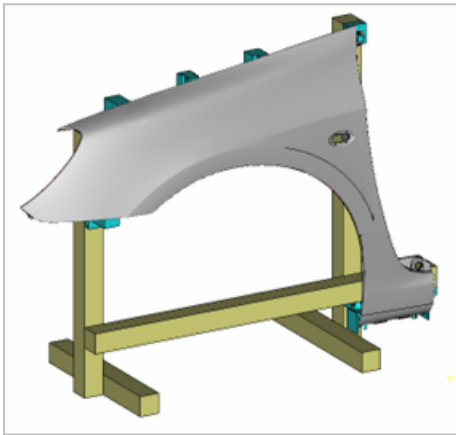
Break off point	Tip Diameter (a)	
	Offset (b)	
Rotate structure	<p>The label tag will be rotated perpendicular to the selected surface.</p> <div style="display: flex; justify-content: space-around;"> <div style="text-align: center;">  <p>Not rotated</p> </div> <div style="text-align: center;">  <p>Rotated</p> </div> </div>	
	Lock Rotation Z	The label tag is always and only rotated into the Z direction.
	Lock Rotation 90°	The label tag is always and only rotated with steps of 90°.
Horizontal connector	<p>OFF = the connector is parallel to the normal</p> 	<p>ON = the connector is parallel to the XY plane</p> 

4.5. Fit 2 Ship

1. RapidFit



The automated solution for manufacturing operations and quality control on complex components. Quickly design cost-efficient fixtures to secure complex, large or unstable parts! Firmly positioned, these parts can then easily be checked, measured, machined, transported, glued or assembled. Speed and quality control are two of the biggest concerns in Rapid Prototyping. You can achieve both when creating and using Magics RapidFit fixtures. It's the ideal solution for the design and setup of a support system for your parts.



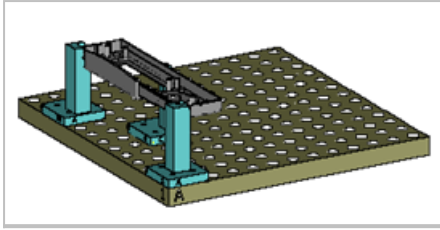
Introduction

Magics RapidFit software automatically designs fixtures based on your customized settings. It takes only a few steps:

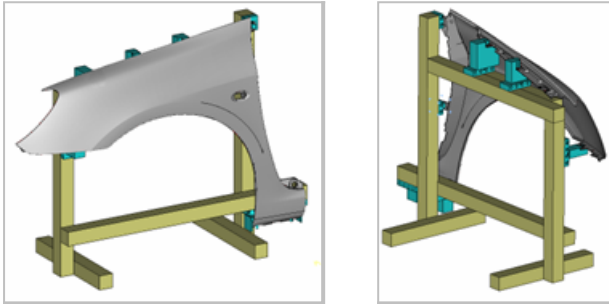
- Create the supporting system with base plates and/or beams
- Indicate on the part the contact points where the fixtures are needed
- Define the type of fixture (e.g. shape and orientation)
- Let Magics create the fixture automatically

The fixtures have a unique and well-defined fit: they will only fit to the part on the exact spots you've chosen. The fixtures are automatically labeled with the assembly position and the name of the parts. This doesn't only allow for setting up the system in no time, but also for easy identification, storage and reuse if required. Due to a very simple assembly method, the system is ready for use in a few minutes. The fixtures can be fixed quickly on a RapidFit reference plate or on beams of a fixture system. You can combine Magics RapidFit fixtures with modular fixture systems as the software can design fixtures that are compatible with any type of beam.

Once created, you can easily produce the fixtures with any RP technique. As RP technologies are functional and cost-effective, these are interesting techniques for fixture manufacturing. The fixtures can be built simultaneously with the part, which further increases your gain in time. The fixtures fit either on beams or on standard grids (baseplates).



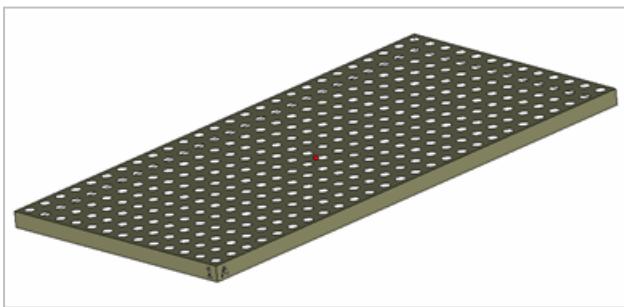
The following picture shows the placement of fixtures on a base plate.



We thus can distinguish 4 typical features in the RapidFit module: the part itself, the fixtures (supporting pillars for the part), the beams (supporting pillars for the fixtures) and/or the base plates (standard grids).

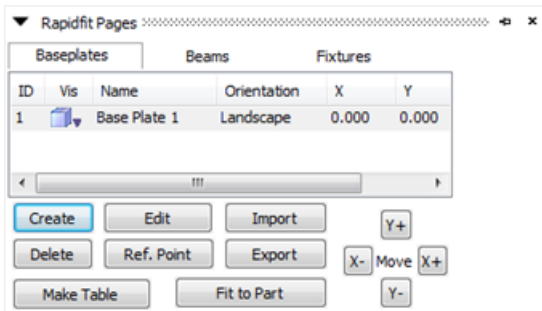
Base Plate

The base plate is a standard metal plate with a grid of threaded holes that can be bought or made. An example of a base plate is:



As you can see the sides of the plate are marked with '1' and 'A'. This is the coordinate system for the holes in the plate. It works just like the grid of a spreadsheet. The hole in the front is 'A1'. Each hole has a unique reference.

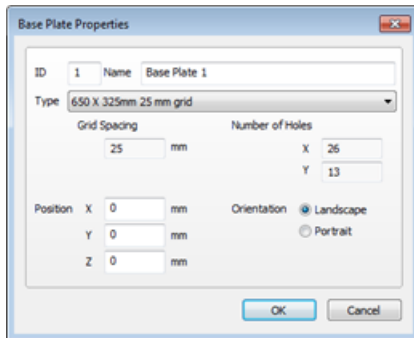
- The Base Plate Toolpage



List of Plates		
	ID	This is the ID of the base plate
	Vis(ibility)	Hide the selected base plate.
		Wireframe shows the edges of the selected base plate.
		The selected base plate is showed in a combination of the shade-mode and the wireframe-mode.
	Name	Each Base Plate has a name that can be changed. If you use several base plates for one setup, they can be distinguished by their name. The default name is 'Base Plate' + ID
	Orientation	The orientation can be compared with Landscape and Portrait of a page setup in other programs
	X	X-position of the plate
	Y	Y-position of the plate
Z	Z-position of the plate	
Create	Creates a new item in the Base Plate list and leads you to the Base Plate Parameters dialog box	
Edit	Leads you to Base Plate Parameters dialog box	
Delete	Deletes the marked base plate from the list	
Ref. Point	Leads you to a small dialog box to change the X, Y and Z column of the list	
Import	This will import a bpd – file that contains a baseplate configuration. When you import an mrf-file, only the baseplates will be imported.	
Export	This will save the current baseplate setup as a bpd – file	
	Moves the Base Plate over 'gridsize' in the positive X-direction.	
	Moves the Base Plate over 'gridsize' in the negative X-direction.	
	Moves the Base Plate over 'gridsize' in the positive Y-direction.	
	Moves the Base Plate over 'gridsize' in the negative Y-direction.	

Make Table	You need to select the type of baseplate you want to use in the dialog box base plate table.
Fit to Part	This wizard will make a baseplate setup which is adapted to the size of the particular part. You have to choose the type of baseplate.

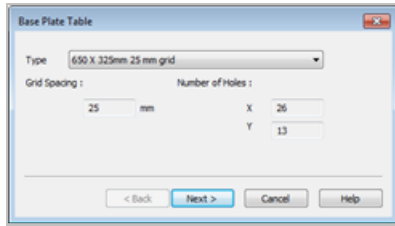
1. Base Plate Properties



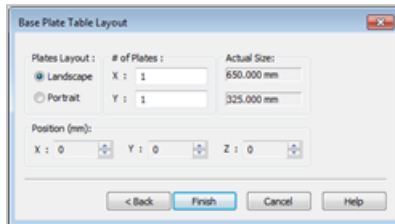
ID	This is a unique number that represents the Base Plate. You cannot change it.
Name	The name is a string that has a default value 'Base Plate'+ ID. This name can be changed and you'll use this name for reference to the plate.
Type List	You choose a Plate Type out of the list, which is filtered in baseplates linked to the chosen grid size. The Base Plates are standardized. The type mentions the size of the plate in X and Y and also the Grid size.
Grid Spacing	The grid Parameter is a property of a Plate Type and can thus not be changed. It is the distance between the centers of the threaded holes. The distance between the side edge and the first row of holes is half the grid Parameter. Therefore the size of the plate is a multiple of this grid parameter.
Number of Holes	The Number of holes in the plate is the size (here 325) divided by the grid spacing (here 25).
Position	This is the position of the hole with lowest X and Y coordinate.
Orientation	The orientation can be compared with Landscape and Portrait of a page setup in other programs.

2. Make Table

First you need to select the type of baseplate you want to use in the dialog box base plate table.

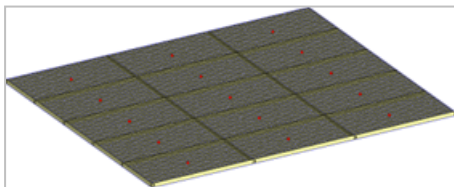


Then, the following dialog box pops up:



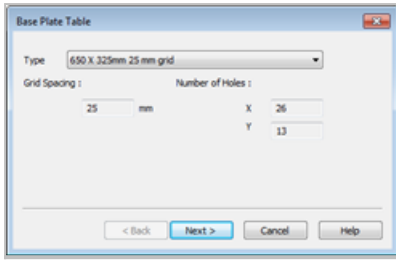
Plates Layout	The user can choose whether to place the baseplates horizontal or vertical.
# of Plates	Enter here how much baseplates you want in the X and Y direction
Actual Size	Magics indicates the actual size of the chosen base plates
Position	Over here you can adapt the position of the setup (X, Y and Z position)

When you press finish, Magics will create this setup. The baseplates are horizontal, 3 rows in the X-direction and 5 rows in the Y-direction.

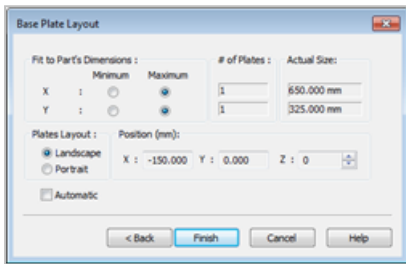


3. Fit to Part

This wizard will make a baseplate setup which is adapted to the size of the particular part. After having chosen the type of baseplate:

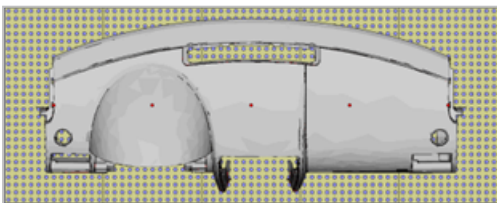


The following base plate layout dialog box pops up:



Fit to Part's Dimensions	Minimum	The setup will be a bit smaller as the part.
	Maximum	The setup will be bigger as the part.
# of Plates	Magics will show how much plates will be used in the X and Y-direction.	
Plates Layout	The user can choose whether to place the baseplates horizontal or vertical.	
Position	Over here you can adapt the position of the setup.	
Automatic	When automatic is selected, the plate layout will be chosen automatically so that the amount of baseplates is minimized.	

As you can see in the following picture, Magics has automatically generated 5 baseplates to cover the whole part.



4. Pick & Place Base Plates

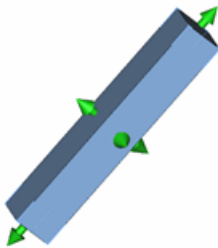


You can also move the baseplates by dragging the baseplates. Select the baseplates you want to move by clicking on the tags or by drawing a rectangle around the tags. Move the cursor to the center of the baseplate indicated by the red dot, hold the left mouse button and move the baseplates in the XY plane.

Beam

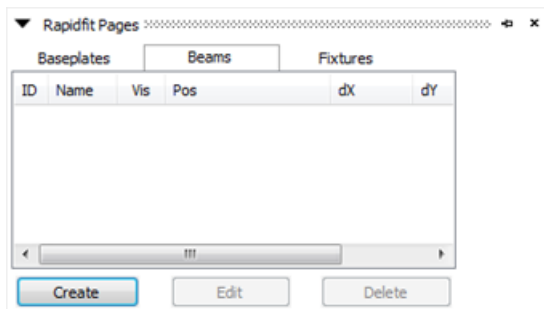
A beam is a post that can replace the baseplates or be added to the setup with baseplates. Fixtures can be attached to these beams. The advantage of the beams is the possibility of placement in the X, Y and Z direction.

A beam looks like:









The picture clearly shows the arrows on each side of the beam. With those arrows, both the position as the size of the beam can be changed. This will be discussed in more detail at Pick & Place & Edit.

– The Beams Toolpage

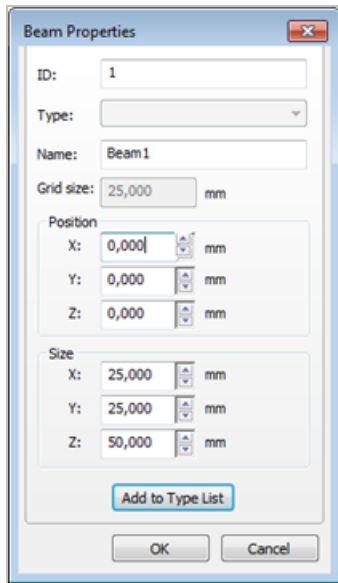


List of Beams

	ID	The ID of the beam.
	Name	Each beam has a name that can be changed. If you use several beams, they can be distinguished by their name. The default name is Beam + ID
	Vis(ibility)	 Hide the selected beam.
		 The beam is displayed with shades according to the direction of the triangles.
		 Wireframe shows the edges of the selected beam.
		 The selected beam is showed in a combination of the shade-mode and the wireframe-mode.
		 The triangles of the beam will be displayed.
		 The bounding box of the beam is showed.
	Pos	The (x, y, z) position where the beam will appear after creation.
	dX	The length of the beam in the X direction.
	dY	The length of the beam in the Y direction
	dZ	The length of the beam in the Z direction
Create	Click on the button create in order to create a new beam. The dialog box Beam Properties appears.	
Edit	If you highlight a row in the Beams Toolpage and you click on edit, the Beam Properties dialog box appears in order to edit the selected beam.	
Delete	To delete a selected (highlighted) beam.	

1. Beam Properties

There are 2 ways to change the properties of a beam. After creation you can either use the following dialog box, or you can use the 'Pick and Place beams mode' (arrows attached to the beam).



ID	The ID of the beam.
Type List	One can also chose predefined types of beams (particular size) from the type list.
Name	The name of the beam
Grid Size	The Grid Size of the Base Plate. The Grid Size indicates the step of the movement on the base plate.
Position	The (x, y, z) position where the beam will appear.
Size	The size of the beam.
Add to Type List	If the user wants to work with a self-defined type of beam, a particular size (of a beam) can be saved by clicking Add to Type List and giving a name to the newly created type.

2. Pick & Place & Edit

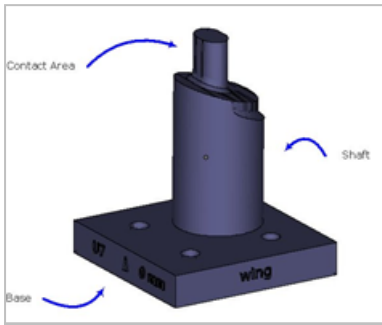


The position and size of the can be changed by using the Pick & Place & Edit mode. First enter the mode, then click on the beam (CTRL+click in order to select more than one beam). There are 3 possible actions:

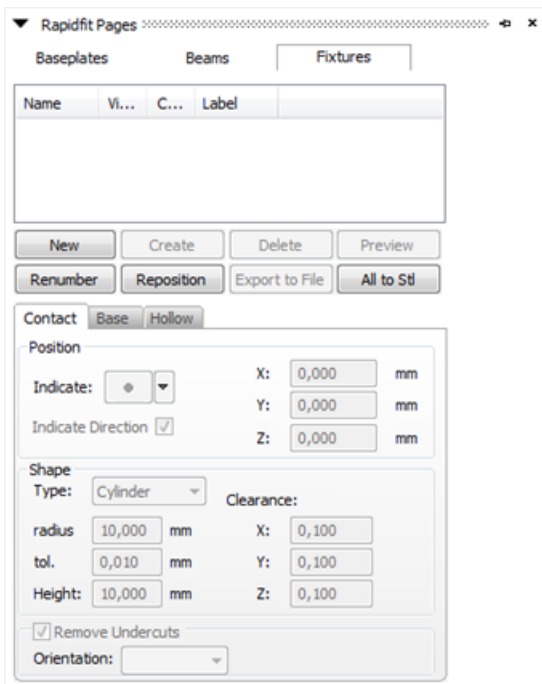
Drag		After selecting the beam, go to the arrow indicating the desired direction and move the beam in the desired direction. You can pull or push the beam.
Resize		By clicking CTRL+left click mouse on arrow and moving in the desired direction, the beam will be resized.
Rotate		By clicking SHIFT+left click mouse on arrow, the beam will rotate 90° around the arrow.



Fixture





The fixture is the pillar that is created to support and hold the part. A fixture looks like:



– The Fixtures Toolpage

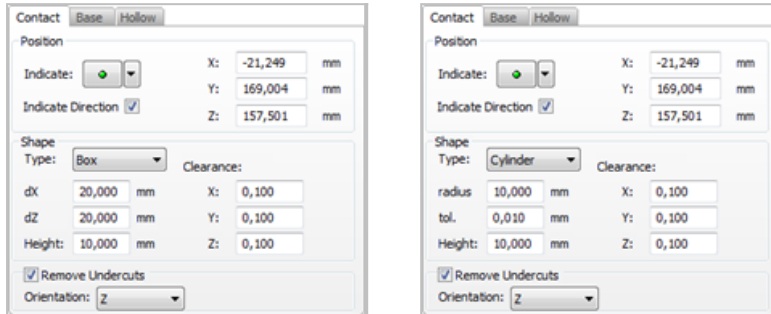


List of Fixtures			
	Name	This is the name of the fixture. It helps the user to see which fixture is manipulated.	
	Visible		Hide the selected fixture.
		The fixture is displayed with shades according to the direction of the triangles.	

		Wireframe shows the edges of the selected fixture.
		The selected fixture is showed in a combination of the shade- mode and the wireframe-mode.
		The triangles of the fixture will be displayed.
		The bounding box of the fixture is showed.
	Color	The circle shows the color of the fixture. Clicking on this circle and selecting a different color can change the color.
Label	This is the label that will appear on the base of the fixture. By default, this is the part name but it can be changed to the name of an assembly, a project or whatever.	
New	To create a new fixture.	
Delete	To delete the selected fixture.	
Create	Magics will create the selected beam so you can study it in its final form.	
Preview	This button refreshes the preview of the fixtures on the screen.	
Renumber	This button renumbers your fixtures should you have a list, in which fixtures have been created and deleted.	
Reposition	Reposition recalculates the position of the fixture. Suppose that the base of the fixture has been moved away from the shaft or a beam has been replaced and the fixture isn't connected anymore to the beam, than Reposition solves these problems.	
Add to STL	The fixtures will be converted to stl	
Export to File	This button prompts the user to save the beam as an STL file hence an automatic conversion will be done. The user thus can enter the name and the path in order to save an STL-file of this fixture.	

– Contact Properties

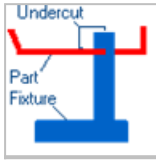
These are the properties of the Contact (the upper side of the pillar). The contact tab deals with the position, the shape and the removal of undercuts of the contact.



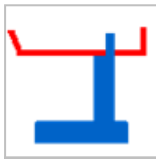
Position	Indicate	With indicate which point, point on wireframe or which center of a circle to use as a center of the contact. After clicking by mouse, the parameters (X, Y, Z) can be used to make the position round numbers.
	Indicate Direction	If the user selects the function 'indicate direction', upon indicating to point of contact an arrow will appear with which the user can direct in which direction the base will have to be attached to a beam or a baseplate.
	X	X-position of the contact.
	Y	Y-position of the contact.
	Z	Z-position of the contact.
Shape	Type	Shape of the contact. It can be a box or a cylinder.
	dX	X dimension of the box-contact (will change depending of orientation).
	dY	Y dimension of the box-contact (will change depending of orientation).
	Radius	The radius of the cylinder-contact.
	Tolerance	The tolerance of the STL representation of the cylinder-contact.
	Height	Extra height of the contact. The contact is automatically as high as the distance from the base plate until the part This height is added to that distance.
Remove Undercuts	Clearance	This allows the user to define some clearance. A 4 mm pin never fits in a 4 mm hole. Thus the pin will be a bit smaller or the hole a bit bigger. When we say a bit, we don't think of 1 mm but rather of 0.01 mm. This 0.01 mm is the clearance.
		Indicate whether you want to remove undercuts and in which direction.

1. Remove Undercuts

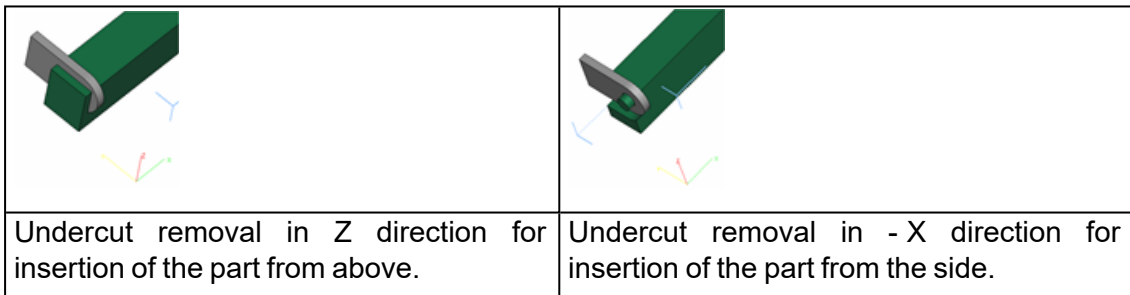
If we take a short look on a section of an assembly of part and fixture we see:



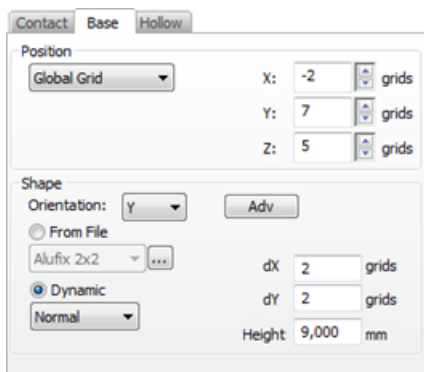
or without the undercut

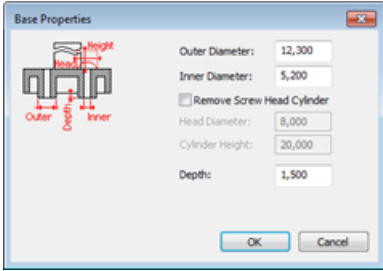


Thus you can see that a fixture with a big undercut cannot be taken out of the part. In the combo box, the user can choose the orientation (X, -X, Y, -Y, Z, -Z) of the undercut removal.



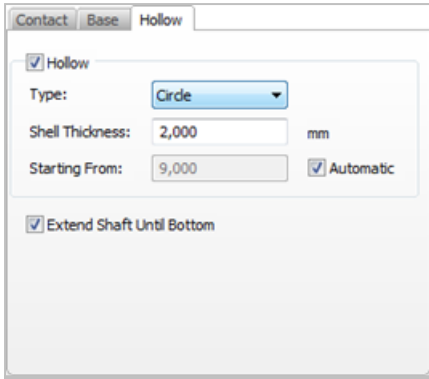
– Base Properties



Position	Global Grid	The fixture is placed on the global grid.	
	X	X position in grids.	
	Y	Y position in grids.	
	Z	Z position in grids.	
Shape	Orientation	The user can choose the orientation of the base. This indicates the direction of the fixtures (X, -X, Y, -Y, Z, -Z). This setting is related with the indicate direction feature (See)	
	Advanced base properties	<p>Advanced base properties. These give you more freedom for the design of the base, especially concerning the placement of the base on the base plate.</p> 	
	From file	<p>You can now create fixtures that will fit on every system by using STL-bases. When a base is not in the drop down list, you can add a base by using the – button.</p> <p>Create the wanted beam in the STL format, save the file so that the WCS is in the bottom corner of the STL.</p>	
Dynamic	Type	The user can choose from the combo box: normal, caps, Rexroth	
	dX	X dimension in grids.	
	dY	Y dimension in grids	
	Height	The height of the base of the fixture.	

– Hollow Properties

Dependent on how the fixtures are made it can be useful to make them hollow. When they are made with Stereolithography or LS, the Hollow version is a gain of time.

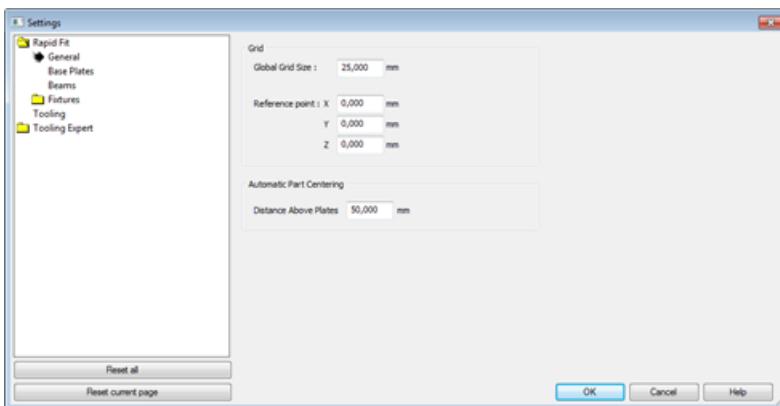


Hollow	Make the fixture hollow or not.
Type	The inner shape can be a circle or a square independent of the outer shape.
Shell Thickness	The thickness of the wall.
Starting from	This can be automatic (check-box). Then the fixture is hollow until a certain height. That height is a 'shell thickness' away from the position of the contact. If you wish, you can enter a height manually by checking off the Automatic check box and entering the height in the Starting from edit box.
Extend shaft	To improve the stability of the shaft, you can extend it to the bottom so it will make contact with the base plate.

Settings

Here, the user is able to define the default properties of RapidFit. To reach the Settings of RapidFit go to Menubar/Options/Settings/Modules/RapidFit

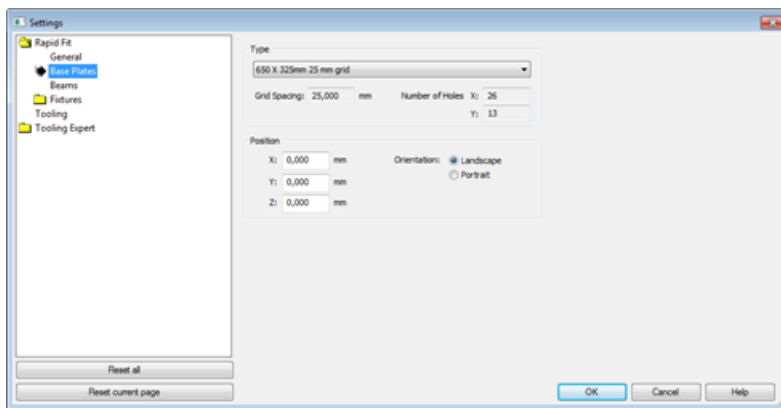
— General



Grid	One should know that there is a difference between the grid and the base plate. The grid, of which the reference point can be predefined in this General tab of the Settings, is virtual. You can imagine an infinite series of circles in x and y.	
	Global Grid Size	The distance between the centers of these circles

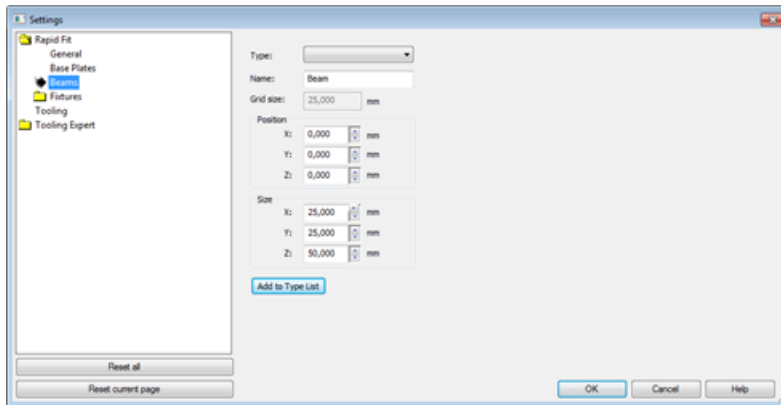
		of the grid.
	Reference Point	The Reference point is the position of the center point of one of the circles. When you make a new Base Plate, it will start from the circle on the reference point and extend in the positive x- and y-direction.
Automatic Part Centering	Distance Above Plates	Magics will translate the part centered above the platforms. The distance used between the part and the plates will be the given height Distance Above Plates in this menu.

– Base Plates



Type	This is the type of Base Plate. The drop – down menu lets you choose between existing Base Plates. They are filtered on the selected grid size.	
	Grid Spacing	The distance between the centers of these circles of the grid.
	Number of Holes	The amount of holes of the Base Plate.
Position	XYZ	The position of the base plates can be altered but only with reference to this grid, hence only in steps that are a multiple of the grid size.
	Orientation	The orientation plays the role of Portrait and Landscape in a print Setup of any program. If you use it on a square shaped Base Plate, only the text at the sides of the plate will tell you that you changed the orientation.

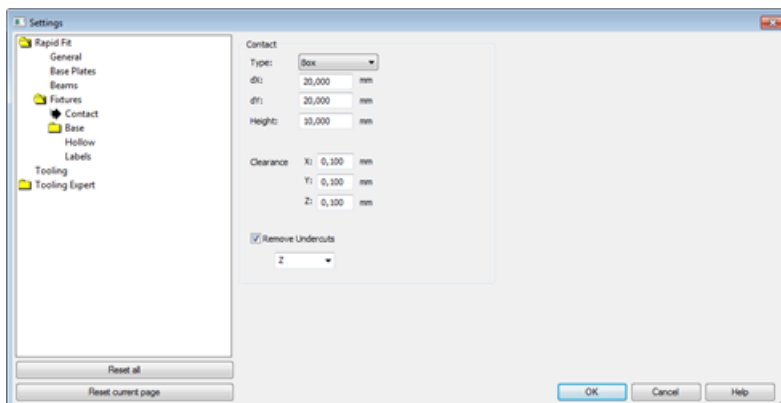
– Beams

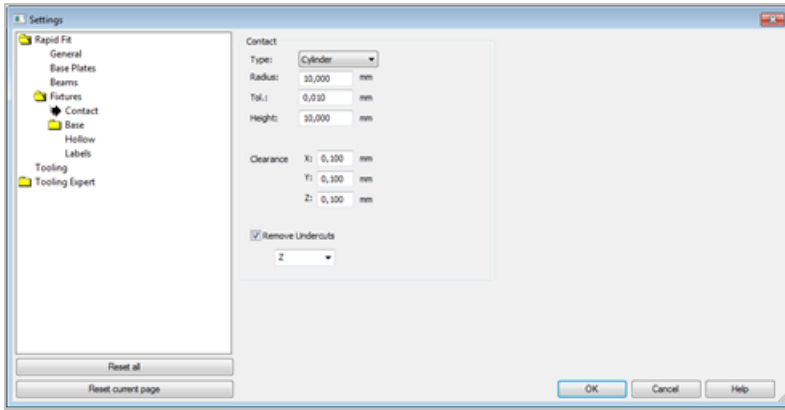


Type List	One can also chose predefined types of beams (particular size) from the type list.
Name	The name of the beam.
Grid Size	The Grid Size of the Base Plate. The Grid Size indicates the step of the movement on the base plate.
Position	The (x, y, z) position where the beam will appear.
Size	The size of the beam.
Add to Type List	If the user wants to work with a self-defined type of beam, a particular size (of a beam) can be saved by clicking Add to Type List and giving a name to the newly created type.

Fixtures

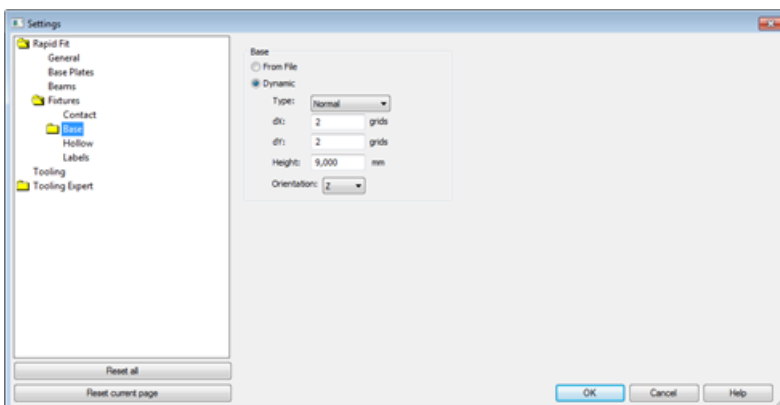
– Contact





Type	Shape of the contact. It can be a box or a cylinder.
dX	X dimension of the box-contact (will change depending of orientation).
dY	Y dimension of the box-contact (will change depending of orientation).
Radius	The radius of the cylinder-contact.
Tolerance	The tolerance of the STL representation of the cylinder-contact.
Height	Extra height of the contact. The contact is automatically as high as the distance from the base plate until the part This height is added to that distance.
Clearance	This allows the user to define some clearance. A 4 mm pin never fits in a 4 mm hole. Thus the pin will be a bit smaller or the hole a bit bigger. When we say a bit, we don't think of 1 mm but rather of 0.01 mm. This 0.01 mm is the clearance.
Remove Undercuts	Indicate whether you want to remove undercuts and in which direction.

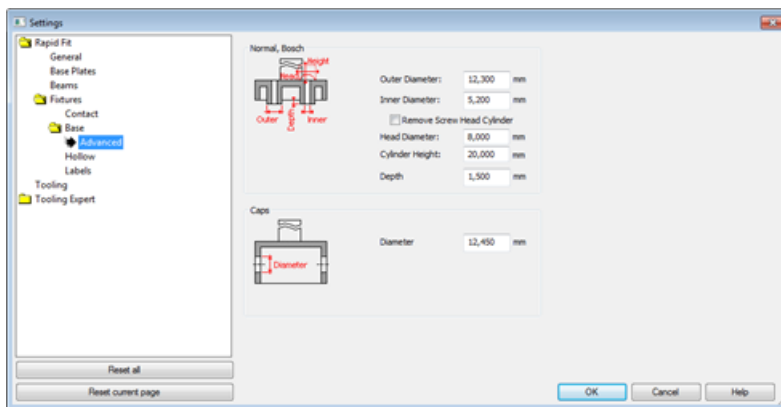
– Base



From file	You can now create fixtures that will fit on every system by using STL-bases.
-----------	---

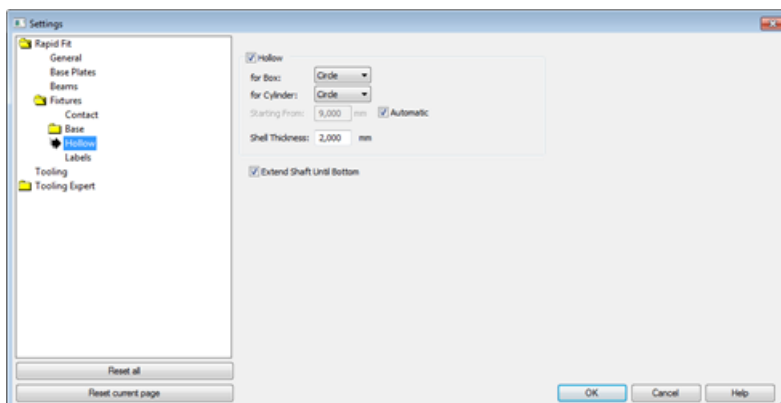
Dynamic	Type	The user can choose from the combo box: normal, caps, Rexroth
	dX	X dimension in grids.
	dY	Y dimension in grids
	Height	The height of the base of the fixture.
	Orientation	The user can choose the orientation of the base. This indicates the direction of the fixtures (X, -X, Y, -Y, Z, -Z). This setting is related with the indicate direction feature (See)

1. Advanced



Normal Bosch	These properties give you more freedom for the design of the base, especially concerning the placement of the base on the base plate.
Caps	These properties give you more freedom for the design of the caps.

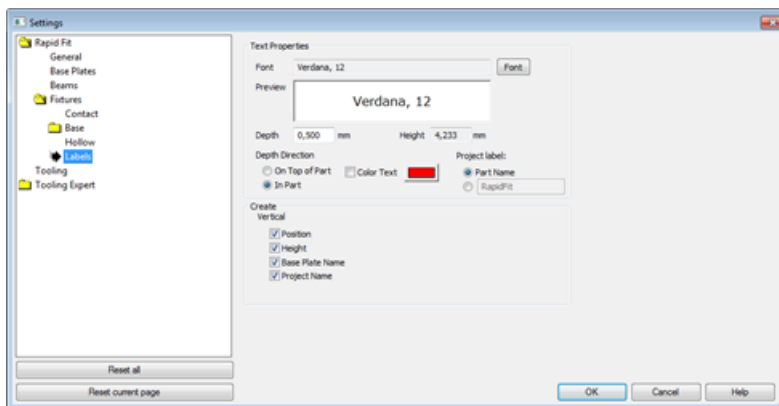
– Hollow



Hollow	Make the fixture hollow or not.
For Box	The inner shape can be a circle or a square.
For	The inner shape can be a circle or a square

Cylinder	
Starting from	This can be automatic (check-box). Then the fixture is hollow until a certain height. That height is a 'shell thickness' away from the position of the contact. If you wish, you can enter a height manually by checking off the Automatic check box and entering the height in the Starting from edit box.
Shell Thickness	The thickness of the wall.
Extend shaft	To improve the stability of the shaft, you can extend it to the bottom so it will make contact with the base plate.

– Labels




Text Properties	Font	The user can define the font of the labels.
	Preview	A preview is given of the chosen font.
	Depth	The depth of the label
	Height	The height of the text.
	Depth Direction	The text can be directed out of the base or into the base.
	Color Text	Colors the text of the label.
	Project Label	Displays the part name or a name defined by the user.
Create	Vertical	Displays the checked items on the base of the fixture.

File Operations

The information of this module can be stored in a Magics Project File.

Tools

– Center Part(s)

 The loaded part(s) are centered on the base plates.



Document Generation

The document generation, as explained before (The reader is referred to the Manual of Magics Base for further information on the topic of Document Generation), can also be used to generate reports of the rapid fit set-ups. These are the additional fields for documenting the RapidFit setups.

- General RapidFit tags

RapidFitProjectName	The name of the RapidFit file
RapidFitGridSize	The gridsize used (=distance between holes, default 25mm)
RapidFitGridSizeZ	If applied, the gridsize in the Z direction
BasePlatesRefPoint	The reference point of the baseplate setup
BasePlatesSetupXDimension	The X dimension of the total baseplate setup
BasePlatesSetupYDimension	The Y dimension of the total baseplate setup
BasePlatesSetupZDimension	The Z dimension of the total baseplate setup
BasePlatesSetupDimensions	The total dimensions of the total baseplate setup
FixturesBoundingBoxDimensions	The dimensions of the bounding box of all the fixtures

The Setup View allows inserting a picture of the complete set-up.

- Baseplate Tags

These tags are to represent the information of the baseplates. In case of using multiple baseplates, this information will be repeated once for each baseplate. To achieve this, the tags must be placed into a table. Magics will repeat this information in the table for each baseplate.

BasePlateID	The ID number of the baseplate
BasePlateName	The name of the baseplate
BasePlateNumHolesX	The amount of holes in the X direction
BasePlateNumHolesY	The amount of holes in the Y direction
BasePlateOrientation	The orientation of the baseplate (horizontal or vertical)
BasePlatePositionX	The X position of the baseplate
BasePlatePositionY	The Y position of the baseplate
BasePlatePositionZ	The Z position of the baseplate

Pictures of the base plates can be inserted via the Base Plate View part of the Materialise menu in the insert menu of Word.



– Beam Tags

These tags are to represent the information of the beams. In case of using multiple beams, this information will be repeated once for each beam. To achieve this, the tags must be placed into a table. Magics will repeat this information in the table for each beam.

BeamID	The ID number of the beam
BeamName	The name of the beam
BeamPositionX	The X position of the beam
BeamPositionY	The Y position of the beam
BeamPositionZ	The Z position of the beam
BeamDimensions	The dimensions of the beam
BeamSizeX	The X-dimension of the beam
BeamSizeY	The Y-dimension of the beam
BeamSizeZ	The Z-dimension of the beam

Pictures of the beams can be inserted via the Beams View part of the Materialise menu in the insert menu of Word.

– Fixture tags

FixtureName	The name of the fixture
FixtureGridPositionX	The X position of the fixture relative to the baseplate
FixtureAbsPositionX	
FixtureGridPositionY	The Y position of the fixture relative to the baseplate
FixtureAbsPositionY	
FixtureGridPositionZ	The Z position of the fixture relative to the baseplate
FixtureAbsPositionZ	
FixtureGlobalPositionX	The global X position of the fixture (using the origin as 0)
FixtureGlobalPositionY	The global Y position of the fixture (using the origin as 0)
FixtureGlobalPositionZ	The global Z position of the fixture (using the origin as 0)
FixtureBoundingBoxDimensions	The bounding box dimensions of the fixture
FixtureOwningPlateID	The ID of the baseplate on which the fixture is put
FixtureOwningPlateName	The name of the baseplate on which the fixture is put
FixtureClearanceX	The clearance used in X

FixtureClearanceY	The clearance used in Y
FixtureView	

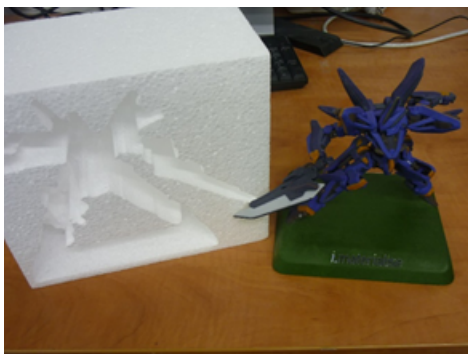
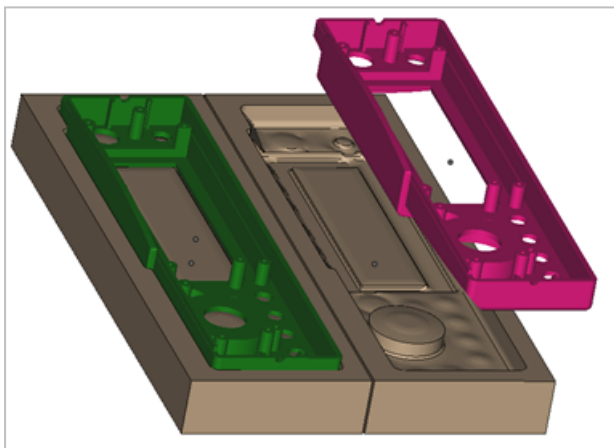
Pictures of the fixtures can be inserted via the Fixtures View part of the Materialise menu in the insert menu of Word.

2. FormFit



Very fragile parts (e.g. figurines) or large parts which can easily deform or get damaged during transport are often created with Additive Manufacturing. With the new FormFit function you can quickly create packaging files which can then be used to secure fragile, expensive, complex, large or unstable parts. The generated packaging file follows the shape of your object perfectly but avoids undercuts so that the stl file can be used to generate e.g. a foam which optimally supports your part.

Two options are available to create a FormFit file: one in which the bounding box dimensions of the file are defined relative to the part (e.g. to have 50mm of surrounding material around the object) and one in which the dimensions are absolute (e.g. the file should fit a known shape).





Chapter 5. Lattice

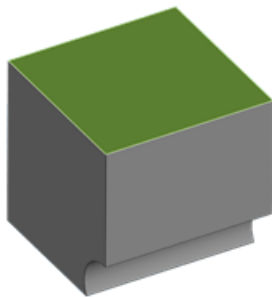
5.1. Mesh Lattice

All the operations in this group results in changes to the mesh geometry as part of the lattice operation.

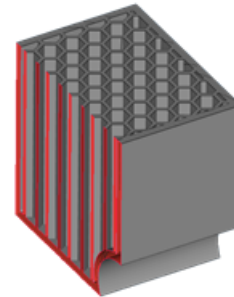
1. Honeycomb Lattice



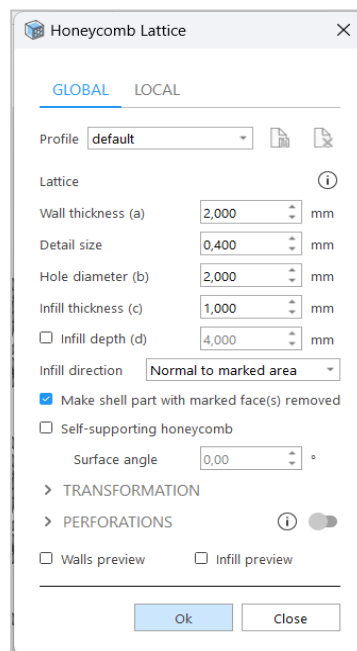
With this command you can easily create a hollowed part filled with a honeycomb infills; by marking the desired surface, an shelled part with an open side can be created to remove trapped material. The operation is similar to the traditional hollowing, where material is removed from the original part; the final printed part will become lighter, allowing you to save material and build time. In addition, the generated honeycomb lattice will let you keep maximal part strength and functional use.






Original part with marked surface to be removed

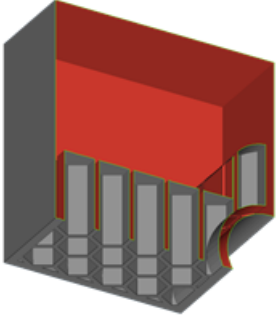


Resulting part with honeycomb lattice generated



Global/Local	When selecting Global , the honeycomb infills will be applied to the full part.
--------------	--

	<p>When selecting Local, the honeycomb infills will only be applied to the marked area.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  When selecting Local method ensure there is a marked region to ensure that a preview can be shown and the operation can be performed. </div>	
Profile	Profile selection	<p>A profile is can be saved to quickly switch between different sets of target parameters for the honeycomb lattice.</p> <p>A default profile default is available on initial load of the application. If the parameters are modified while a profile is selected, this profile will be marked with a star (*). The changes will become permanent if the profile is saved with the changes.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  The profile is unique for the Global and Local methods so they can be edited and maintained independently. </div>
	Save profile	Save the customized profile to a the Honeycomb profiles folder in the user's program data.
	Delete profile	Delete the selected profile from the Honeycomb profiles folder in the user's program data.
Wall Thickness (a)	This value displays the distance over which the triangles of the original shell get an offset in order to generate a hollow part.	
Detail Size	<p>This value displays the level of detail that will remain in the new shell. This value shall be the same as the smallest detail of the part. The smaller this value, the more triangles will be included in the new shell and the more detail can thus be incorporated.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;">  Remark: If the value is chosen too high, it is possible that the internal wall intersects with the external wall. </div>	
Hole diameter (b)	This value corresponds to the distance of the opposing corners of the hexagonal hole profile of the honeycomb(s).	
Infill thickness (c)	It's the thickness of the walls between the honeycomb(s).	
Infill depth (d)	Enable the honeycomb infills to be generated only for a specified depth in the part. The depth will be calculated as an offset from the part surface along the infill direction selected.	

		
Infill direction	Normal to marked area	Select the direction that the honeycomb infills will follow when being generated. The direction of the lattices corresponds to the average normal of the marked surface area.
	User defined	<p>The direction is defined by a vector: fill in the coordinates in the X, Y and Z field.</p> <div style="border: 1px solid #add8e6; padding: 10px; margin-top: 10px;"> <p>Note: You can also indicate line entity on scene after activating the Indicate line button: the lattices will be in the direction of the line. Otherwise, you can also indicate a triangle after activating the Indicate Triangle button, the lattices will be in the direction of the normal of the selected triangle.</p> </div>
Make shell part with marked face (s) removed	When this option is selected, the marked surface will be removed to create a shell part with an open side.	
Self-supporting honeycomb	When this option is selected, the resulting surfaces of the infills will be self-supporting so that no supports will be required to print the part successfully.	
	Surface angle	This value defines the self-supporting angle used to generate the honeycomb surfaces.

– Transformation

▼ TRANSFORMATION

X translation mm

Y translation mm

Rotation °

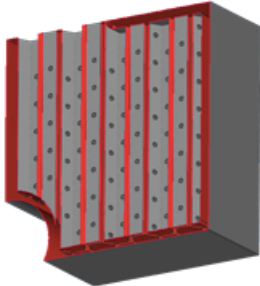
X translation	Move the honeycomb infills along the X axis of the lattice coordinate system
Y translation	Move the honeycomb infills along the Y axis of the lattice coordinate system
Rotation	Rotate the honeycomb infills around the Z axis of the lattice coordinate system
Align view	Click on this button to align the viewport to the top view of the lattice coordinate system to make it easy to understand how the lattice will be transformed

– Perforations

▼ Perforations
ⓘ
☑

Diameter (d) mm

Interval (e) mm

Perforations (toggle switch)	Add circular perforations to the infill walls to further drain the material that might be trapped in closed areas of the honeycomb lattice.	
		
	Diameter	The size of the perforation.
Interval	Distance between 2 consecutive perforations.	

– Preview

Walls preview
 Infill preview

You can preview the walls and/or of the infill of the honeycomb lattice that will be generated as a guide to defined the appropriate parameters to achieve your desired results quickly. The preview can also be used in combination with Multi-section in order to inspect the part inside, and how the lattice will be generated.



Currently, preview parameters are not saved as part of the honeycomb profiles.

2. Volume Mesh Lattice

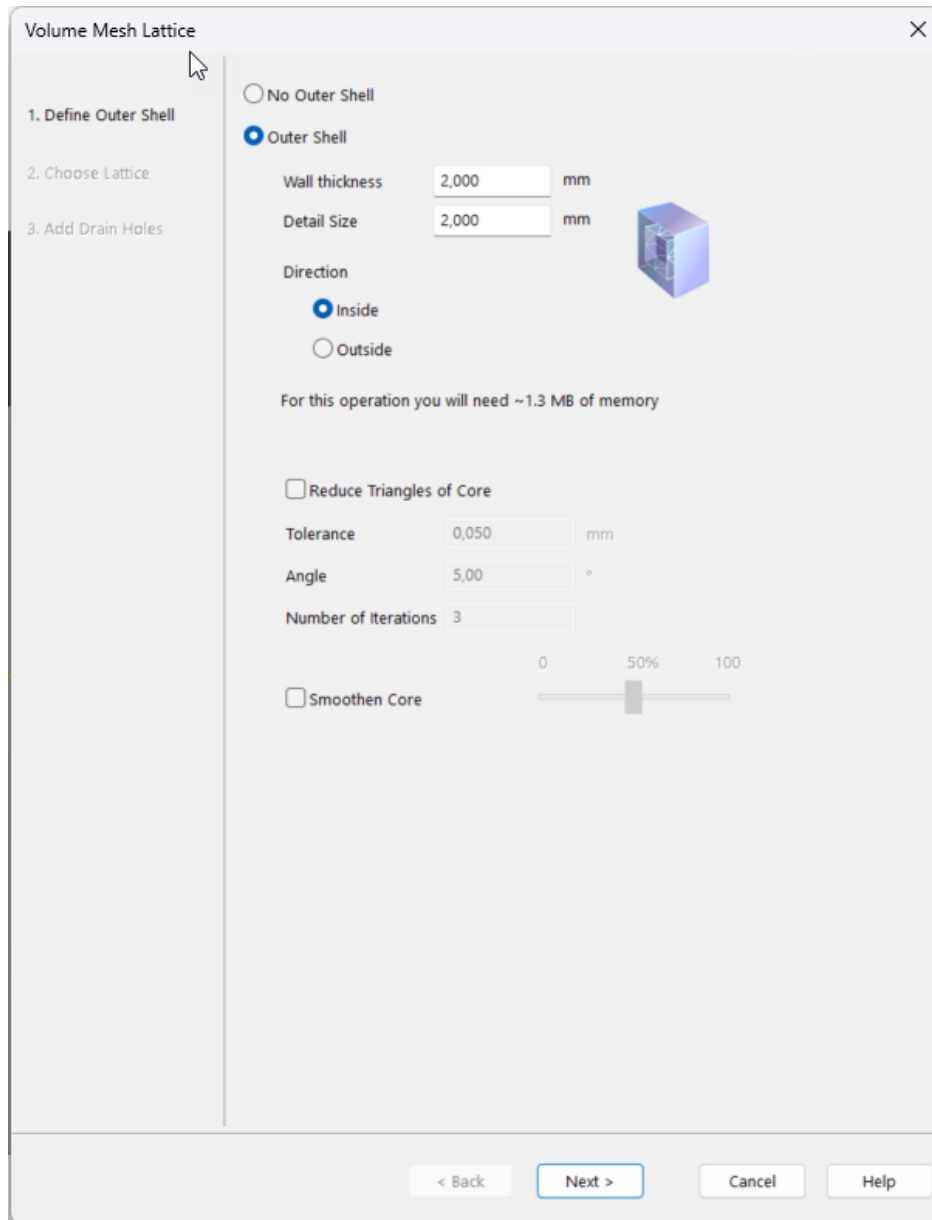


This operation can be used to hollow and fill the hollow region with mesh-based unit cells quickly. It is also possible to only fill the solid region with a mesh-based unit cell lattice.


The wizard consists out of 3 pages:

- Define outer Shell
- Choose Lattice
- Add Drain Holes

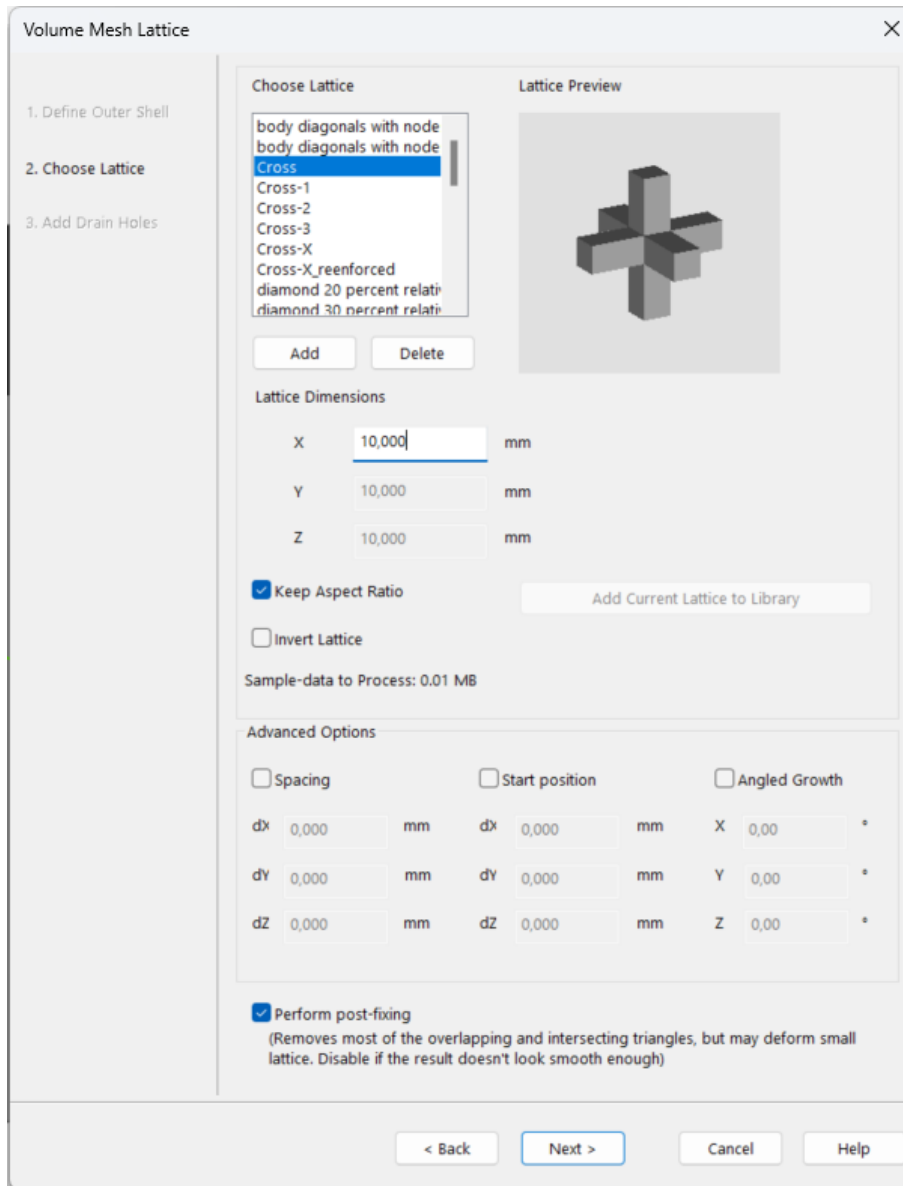
Define outer shell



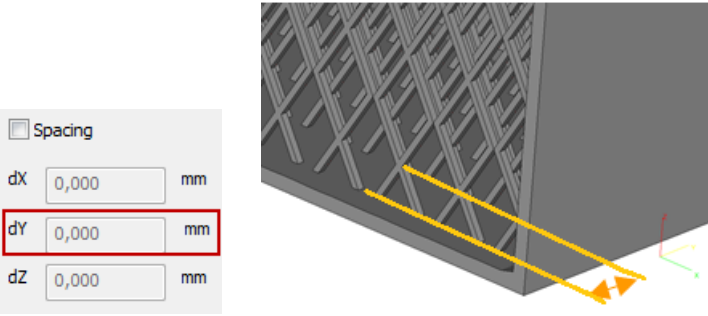
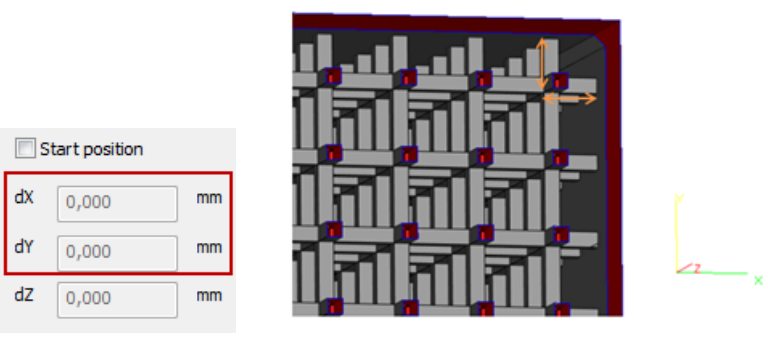

No outer shell	The solid region of the part will be filled with lattice and all the surfaces that belongs to the part will be removed.
Outer shell	A new shell will be created on the part and the hollow region in the part will be filled with lattice instead.
Wall Thickness	This value displays the distance over which the triangles of the original shell get an offset in order to generate a hollow part.
Detail Size	This value displays the level of detail that will remain in the new shell. Standard, this value should be the same as the smallest detail of the part. The smaller this value, the more triangles will be included in the new shell and the more detail can thus be incorporated.

	 Remark: If the smallest detail is chosen too high, it is possible that the internal will wall intersect with the external wall.
Direction	Here you determine if you want to create a new shell at the interior or at the exterior of the existing shell.
Memory requirements	While you set the parameters, Magics makes an estimation of the quantity of free RAM that will be needed during the calculation and of the number of triangles that will be created. You'll need to enter new values in the Wall Thickness and Smallest Detail fields to see a new estimation of the amount of RAM and new triangles. The amount of triangles can later on be reduced with the Triangle Reduction function. The memory requirements depend greatly on the value set for Smallest Detail.
Reduce triangles of Core	Because the hollow function creates a lot of triangles, you have the possibility to reduce these at once.
	Tolerance _____
	Angle _____ 1 Triangle Reduction
	Number of Iterations _____
Smoothen Core	By checking this option a smoothing will be performed on the created core.

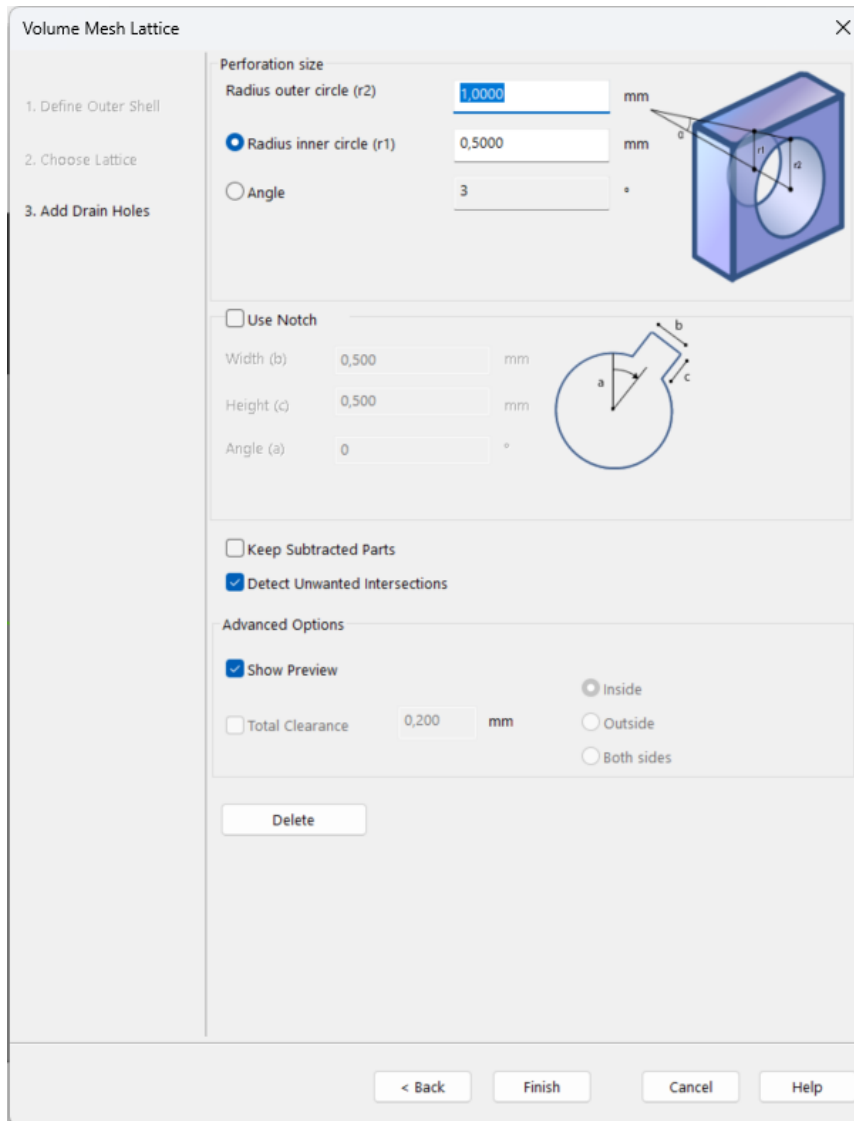
Choose Lattice



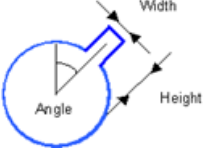
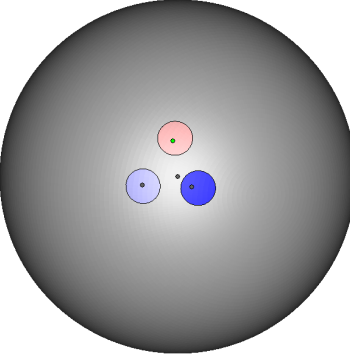
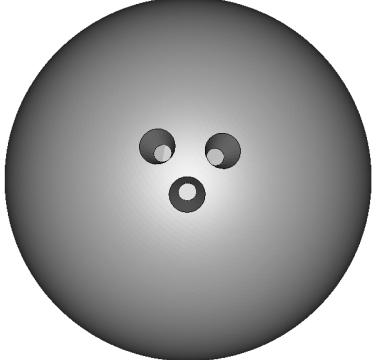
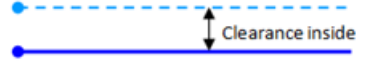

Choose Lattice	This shows a library with mesh -based unit cells that can be used for lattice generation.	
	Add	Add a unit cell to the library.
	Delete	Delete the selected unit cell from the library.
	Lattice Preview	A preview of the unit cell is shown.
	Add Current Lattice to Library	When a value in the Lattice dimensions parameters are changed with an existing unit cell selected, this button will be activated. When this button is pressed, the selected unit cell will be updated to the new unit cell with the new lattice dimensions.
Lattice Dimensions	Set the unit cell size along the X, Y and Z axes along the lattice coordinate system	

	<p>Keep Aspect Ratio</p>	<p>When this is checked the Y and Z length will be rescaled uniformly with the X length.</p>
	<p>Invert Lattice</p>	<p>If this set to True, then the lattice will be subtracted from the part while if it is false, the lattice will be unified to the part.</p>
	<p>Sample- data to Process</p>	<p>The estimated memory size of the mesh-based lattice.</p>
<p>Advanced Options</p>	<p>Spacing</p>	<p>Sets the spacing between the unit cells in the lattice structure along the lattice coordinate system.</p>
		
	<p>Start position</p>	<p>Sets the starting position of the center of the unit cell bounding box in the lattice coordinate system.</p>
		
	<p>Angled Growth</p>	<p>Sets the rotation angle of the lattice structure along with the unit cells about X, Y and Z axes in the lattice coordinate system.</p>
		

Add Drain Holes



Delete	Deletes the selected perforations. A perforation can be selected by clicking on the preview perforation cone on the 3D space.
Perforation size	The dimensions of the perforation (s). There are 2 possible combinations of dimensions: <ul style="list-style-type: none"> • Radius of outer circle (r2) and Radius of inner circle (r1) • Radius of outer circle (r2) and Radius of inner circle (r2)
	Radius outer circle (r2) Specifies the radius of the outer circle of the perforation.
	Radius inner circle (r1) Specifies the radius of the inner circle of

		the perforation.
	Angle	Specifies the angle of the perforation
Use notch		Define the use and dimensions of notch on the perforations.
	Width (b)	
	Height (c)	
	Angle (a)	
Keep subtracted parts		When True, the subtracted part(s) are preserved as parts in the scene, else the subtracted part (s) are unloaded automatically.
		
With subtractions		Without subtractions
Detect unwanted intersections		Collisions will be detected when a cone is added. Default on. (See Perforator/ Advanced options , page 137)
Advanced options		
	Show preview	If set to True, a preview of the perforation is shown when hovering the mouse cursor on the part to be perforated.
	Total clearance	Define an offset towards the inner, outer or both sides of the intersecting line between the parts. This way you introduce a little gap between the parts that result from the perforation.
	Inside	
	Outside	

	Both sides	
--	------------	--

3. Tetrahedron Lattice



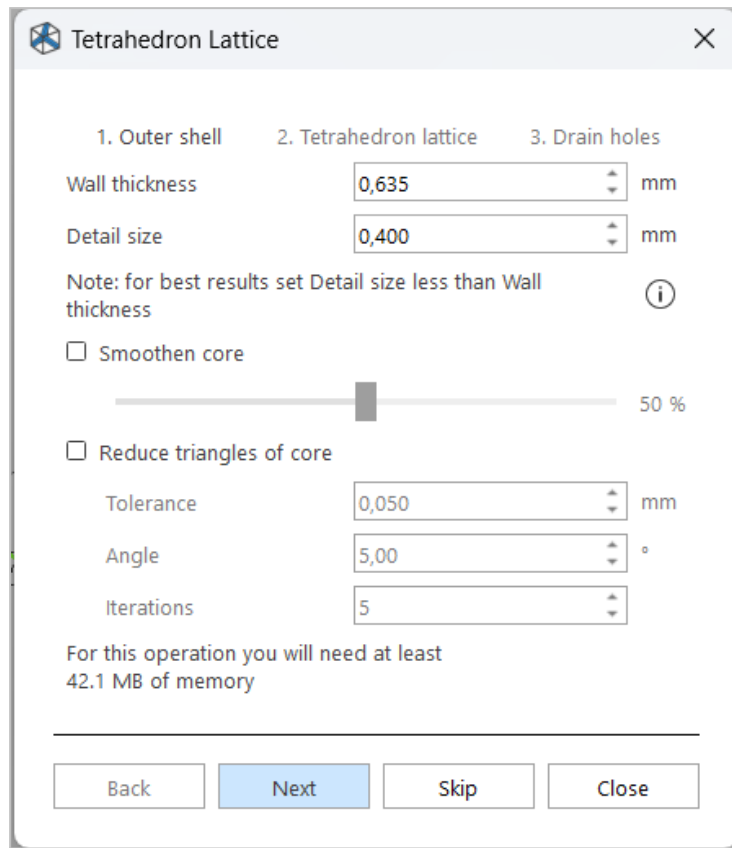
This operation can be used to hollow and fill the hollow region with tetrahedron lattice quickly with custom thickness and strut length. Tetrahedron lattice generally has lower triangle counts than a typical mesh-based lattice while also achieving self-supporting properties that makes it easier to manufacture with technologies that requires supports such as stereolithography. It is also possible to fill already hollowed region with the tetrahedron lattice.

The wizard consists out of 3 pages:

- Outer Shell
- Tetrahedron Lattice
- Drain Holes



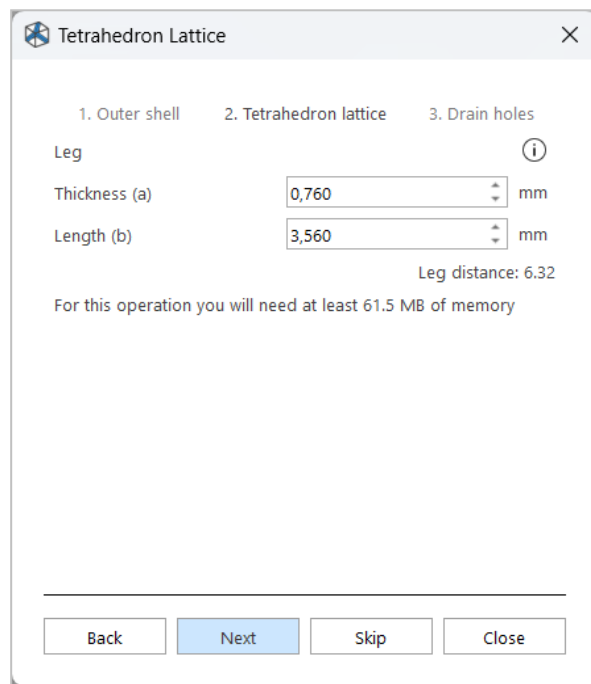
Outer Shell

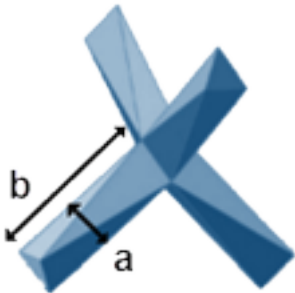


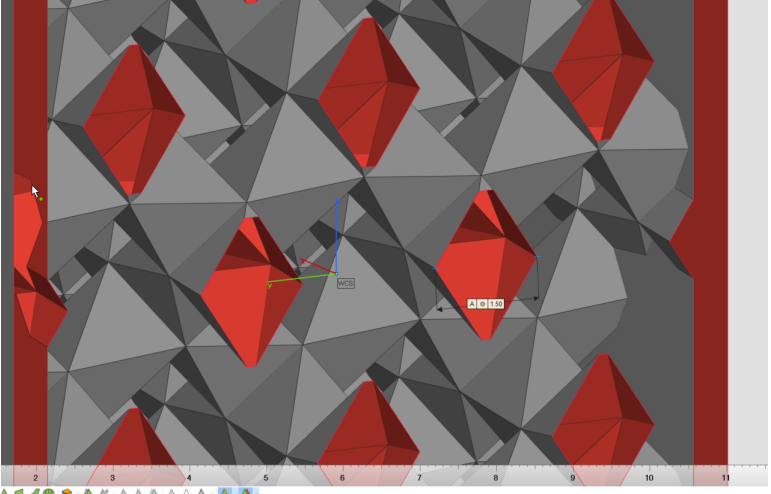
Outer shell	A new internal hollow shell will be created on the part . This step can be skipped if the part has hollow shells already.	
Wall Thickness	This value displays the distance over which the triangles of the original shell get an offset in order to generate a hollow part.	
Detail Size	<p>This value displays the level of detail that will remain in the new shell. Standard, this value should be the same as the smallest detail of the part. The smaller this value, the more triangles will be included in the new shell and the more detail can thus be incorporated.</p> <div style="border: 1px solid blue; padding: 5px; margin-top: 10px;"> <p> Remark: If the smallest detail is chosen too high, it is possible that the internal will wall intersect with the external wall.</p> </div>	
Reduce triangles of Core	Tolerance	1 Triangle Reduction
	Angle	
	Number of Iterations	
Smoothen	By checking this option a smoothing will be performed on the created core.	

Core	The strength of the smoothing can also be set.
Memory requirements	While you set the parameters, Magics makes an estimation of the quantity of free RAM that will be needed during the calculation and of the number of triangles that will be created. You'll need to enter new values in the Wall Thickness and Smallest Detail fields to see a new estimation of the amount of RAM and new triangles. The amount of triangles can later on be reduced with the Triangle Reduction function. The memory requirements depend greatly on the value set for Smallest Detail.

Tetrahedron Lattice



Tetrahedron Lattice	In this step, the specific tetrahedron parameters can be set to achieve the desired lattice structure.	
Leg	With these parameters, a specific tetrahedron unit cell will be generated which then will be used to fill the target region.	
		The point to point thickness of the tetrahedron strut at its thickest region.

		
	<p>Length (b)</p>	<p>The length of the tetrahedron strut from on connection point to the other.</p>
	<p>Leg distance</p>	<p>This value is calculated based on the leg thickness and leg length values. Where the leg distance shows the distance delta between the min and max sizes of the tetrahedron cell bounding box.</p>
<p>Memory requirements</p>	<p>While you set the parameters, Magics makes an estimation of the quantity of free RAM that will be needed during the calculation and of the number of triangles that will be created. Changing the leg parameters will result in the memory estimation to be recalculated.</p>	

Drain Holes

Tetrahedron Lattice

1. Outer shell 2. Tetrahedron lattice 3. Drain holes

Perforation type ⓘ

Radius outer circle (r2) mm

Angle (a) °

Show Preview

▼ NOTCH

Add notch ⓘ

Width (b) mm

Height (c) mm

Angle (a) °

▼ ADVANCED OPTIONS

Detect Unwanted Intersections

Keep Subtracted Parts

Total Clearance mm

Direction

Note: Hold left mouse button to translate selected preview(s)

The parameters on the drain hole page are identical to the parameter on the Perforator operation page with the identical behaviors. See **Perforator, page 135** for more information and detailed description of the parameters. The operation can be completed without adding any perforations.

5.2. Slice Based Lattice

All the operations in this group are slice-based operations of the corresponding mesh lattice operations. With slice-based operations, the file size can be reduced tremendously with improved slicing speed in the Materialise Build Processors.

1. Slice Based Volume Lattice



This operation is a slice-based version of the Volume Mesh Lattice operation where the final lattice is not generated as a physical geometry within the application but added as a parametric component that will be directly generated as slices during build processing by the Materialise Build Processor.



See for ***Volume Mesh Lattice, page 211*** detailed information of the parameters that are similar between the operations. This chapter only describes the behavior of the parameters unique to this operation compared to the other operation.

The wizard consists out of 3 pages:

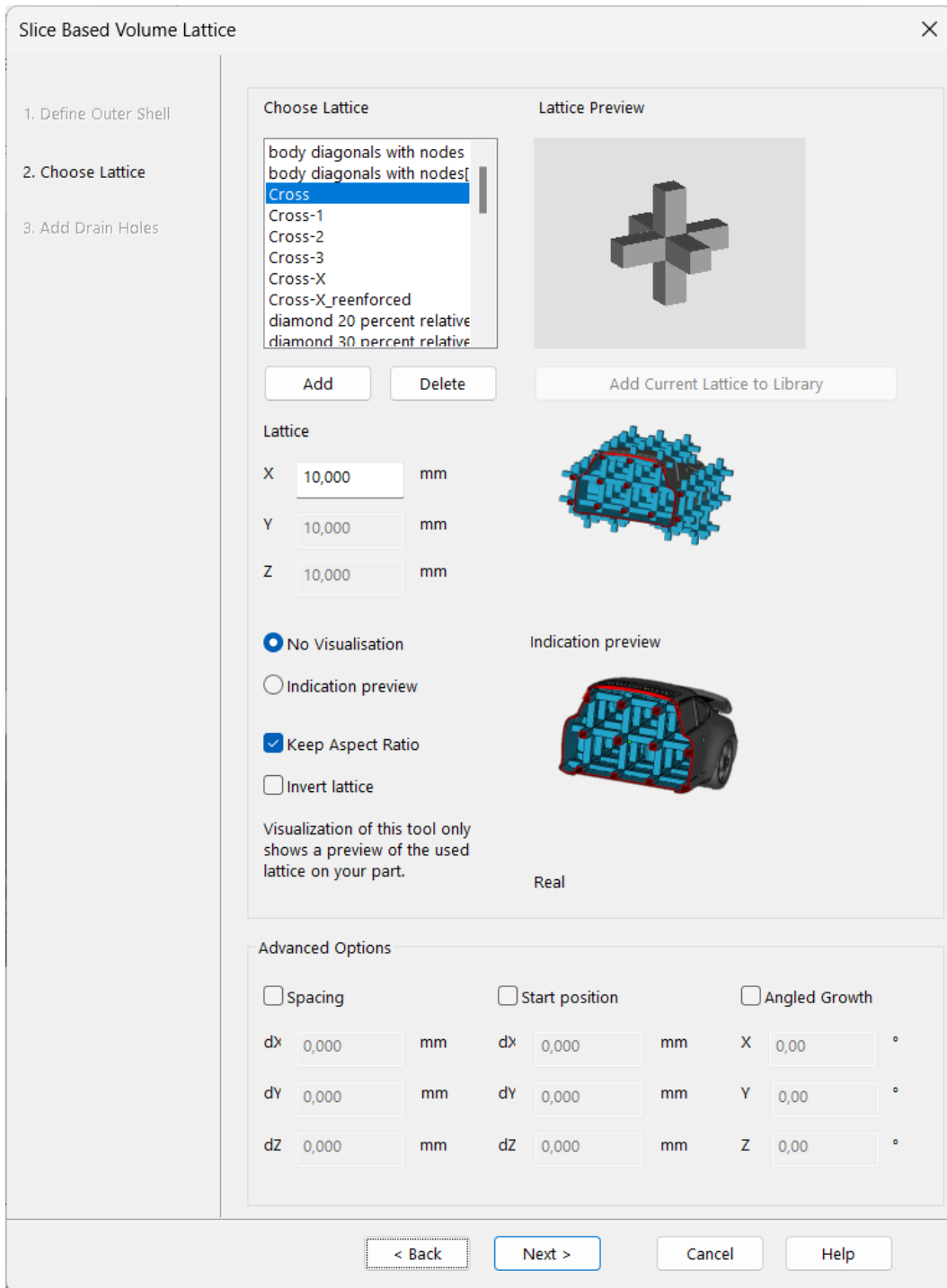
- Define outer Shell
- Choose Structure
- Add Drain Holes

Define outer shell

- See ***Define outer shell, page 212***



Choose Structure



*Note: Only the additional parameters in this section that are not already described in the Volume Mesh Lattice section are described here. For all other parameters, please see **Choose Lattice, page 214** for detailed description of the specific parameters.*

Lattice Dimensions	No Visualization	If this option is set, no preview of the lattice shown on the target part.
	Indication Preview	If this option is set, simple untrimmed preview of the lattice is shown instead of the original input part.



Add Drain Holes

- See **Add Drain Holes, page 216**

2. Slice Based Tetrahedron Lattice



This operation is a slice-based version of the Tetrahedron Lattice operation where the final lattice is not generated as a physical geometry within the application but added as a parametric component that will be directly generated as slices during build processing by the Materialise Build Processor. See **Tetrahedron Lattice, page 218** for more detailed information of all the parameters that are similar between these operations. This chapter only describes the behavior of the parameters unique to this operation compared to the other operation.

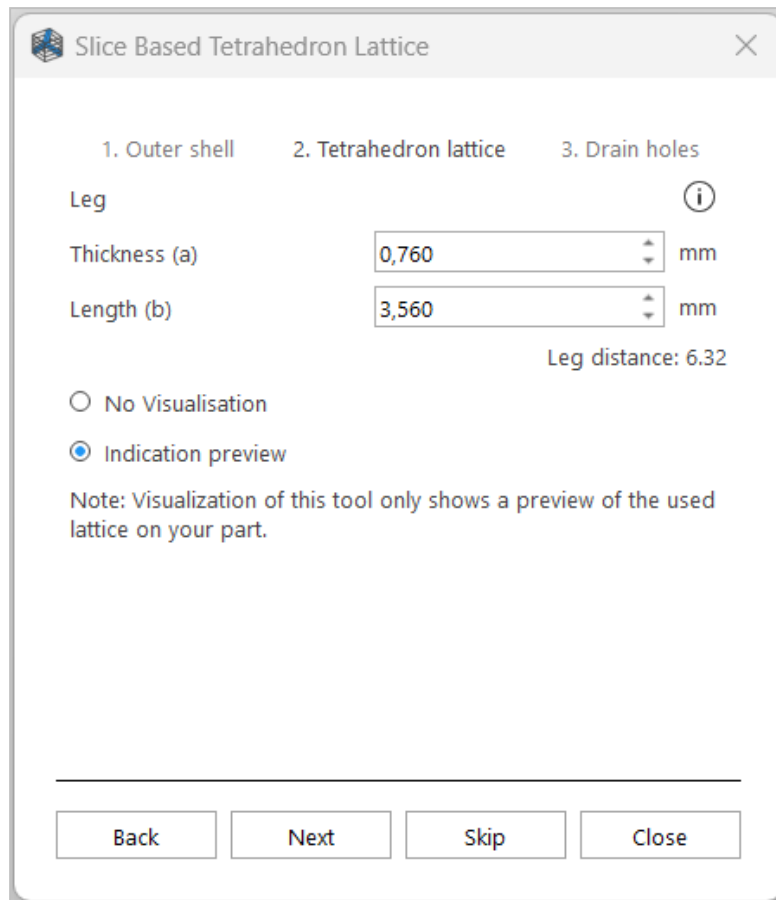
The wizard consists out of 3 pages:

- Outer Shell
- Tetrahedron Lattice
- Drain Holes

Outer Shell

See **Outer Shell, page 219** for more information on all the parameters.

Tetrahedron Lattice



*Note: Only the additional parameters in this section that are not already described in the Volume Mesh Lattice section are described here. For all other parameters, please see **Tetrahedron Lattice**, page 220 for detailed description of the specific parameters.*

Lattice Dimensions	No Visualization	If this option is set, no preview of the lattice shown on the target part.
	Indication Preview	If this option is set, simple untrimmed preview of the lattice is shown instead of the original input part.

Drain Holes

See **Drain Holes**, page 222 for more information.

5.3. Beam Lattice

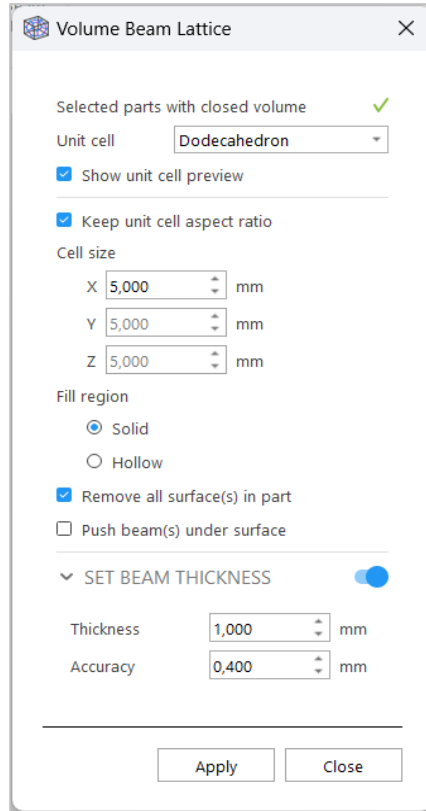
All the operations in this group works with beam lattices which is a lightweight representation of lattices that can be used to improve user experience while working with complex lattice structures. Beam lattice can be imported or exported through file formats such as 3mf and MatAMX and processed directly by selected Materialise Build Processors without needing to generate mesh geometry reducing the need to transfer large files.







1. Volume Beam Lattice



The 'Volume Beam Lattice' allows to fill up the solid or hollow region of the part with the selected basic unit cell structure. The basic unit cell structure is tiled in 3D dimension so that bounding box fits the specified master part till the surface.



Selected parts with closed volume	When at least 1 part with closed volume is selected, the red cross mark turns into green check mark accordingly
Unit cell	<p>All unit cells added to the BeamLatticeStructures library folder are available from the dropdown.</p> <div style="border: 1px solid blue; padding: 5px; margin: 5px 0;"> <p> Note: If you would like to add a custom unit cell to the library, save it in matpart format and locate in BeamLatticeStructures folder in the ProgramData folder. The unit cell can contain only beams (graphs) but not meshes. Once saved , the custom unit cell will appear in the unit cell dropdown list.</p> </div> <div style="border: 1px solid blue; padding: 5px; margin: 5px 0;"> <p> Example of custom unit cell: </p> </div>

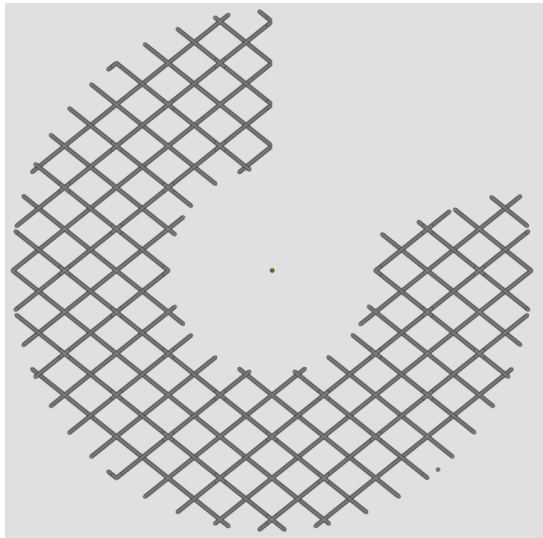
	Show unit cell preview	If ON, a preview of the single unit cell along with the set sizes and thickness will be shown per part.
Keep unit cell aspect ratio	If ON, X, Y and Z will be rescaled uniformly with the any length changes made to either of the input, preserving the aspect ratio. If OFF, X, Y and Z can be assigned separately with non-dependent values.	
Cell size	Specify the size of the unit cell structure in X, Y and Z dimensions. The cell size is based on the bounding box size of the unit cell without accounting for any thicknesses on the original unit cell. If the unit cell has non-zero positive beam node thickness, then the node thickness will also be scaled as per the scaling factor as the cell size.	
Fill region	Choose which region of selected part to fill up with unit cells	
	Solid	The solid region of the part will be filled up with the selected unit cell
	Hollow	The hollow volume will automatically be detected for selected input part and be filled with unit cells.
Remove all surface(s) in part	With this option ON, all the surfaces that exist in the part will be removed upon filling them with beam lattices. When filling the fill volumes, this can reduce an extra manual step needed to remove the surfaces. Currently, this option is not available when choosing the fill a hollow region instead.	
Push beam(s) under surface	With this option ON, the beams closest to the surface(s) will be forced to be right under the surface accounting for the final beam node(s) thickness. This will ensure lattices can be filled within a solid region without any protruding beam nodes through the surface which can result in bad surface finishing. Currently, this option is not available when choosing to fill a hollow region instead.	
Set beam thickness	The option to choose whether to assign thickness and accuracy immediately or create volume beam lattice with zero-thickness beams. If switched ON, then the specified thickness and accuracy will be set to all beams in the resulting beam lattice structure. If Off, the original thickness of the beam nodes on the unit cell will be used.	
	 Note: Thickness and accuracy can also be defined later in the flow with 'Edit beam lattice properties' operation.	
	Thickness	Default: 1 mm. Thickness (diameter) of a lattice beam.
	Accuracy	Default : 0.4 mm (recommended value is 40% of beam thickness). Accuracy of beam lattice when converted to mesh or printed slice based with Materialise Build Processors.



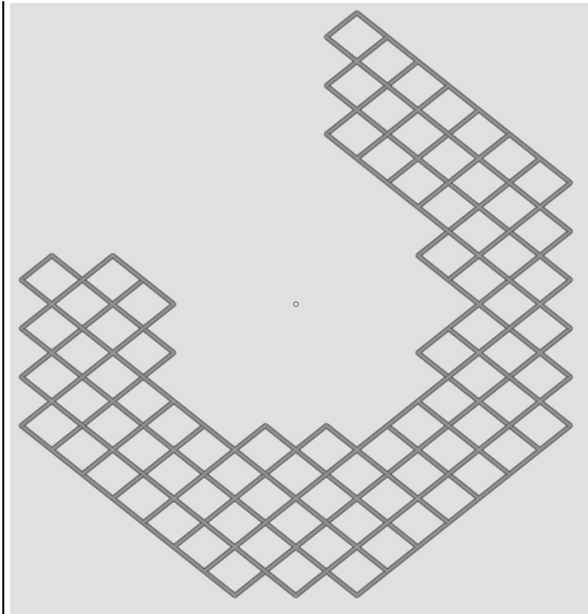
2. Filter Loose Beams



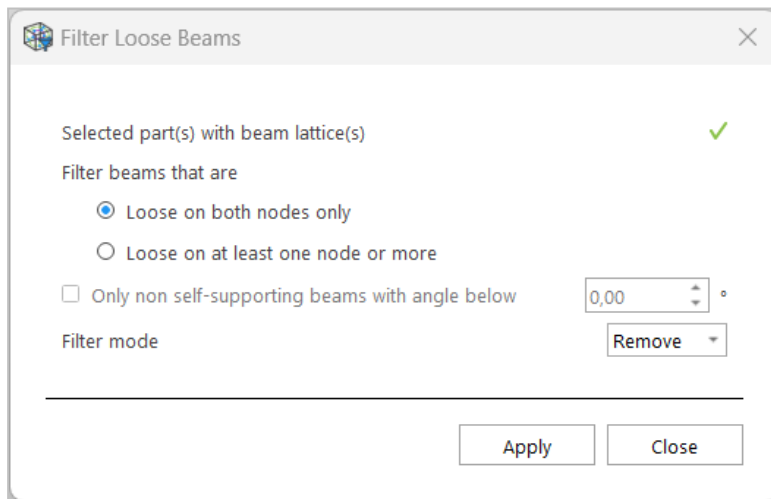
Filter beam lattices that has no connection to other beams or geometries in the model that will fail during printing quickly. Filtered beams can be removed or marked for further inspections and modifications. It is also possible to filter loose beams based on it angle relative to XY-plane to identify beams below the self supporting angle suitable for the machine.



Beam lattice with unfiltered beams



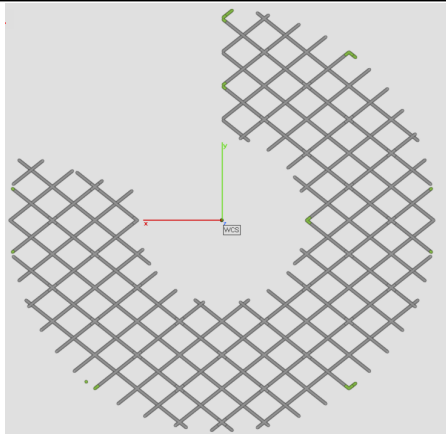
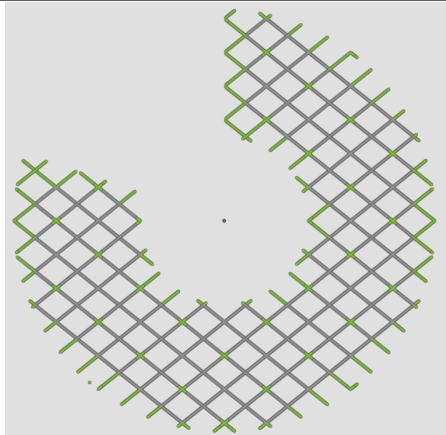
Fully filter beam lattice



Selected part(s) with beam lattice(s)	This is a status indicator if all the selected part(s) contains a beam lattice structure to be filtered.	
Filter beams that are	Loose on both nodes only	When this option is set, only beams that are not connected on both sides of the beam nodes to other beams or a solid region will be filtered

	Loose on at least one node or more	When this option is set, beams that are not connected on at least one of the side will also be filtered
Only non self-supporting beams with angle below		When this is set to ON, only loose beams that are at an angle lower than the specified self supporting angle will be filtered. This option is only available with the filter option "Loose on at least one node or more".
Filter mode	Remove	All the filtered beams will be immediately removed from the model.
	Separate	All the filtered beams will be separated to a separate part with a randomize color to visualize the filtered beams easily. The separated part can be unloaded if not needed.
	Mark	All the filtered beams will be marked immediately on the input model itself. Marked beams can then be deleted or separated as per the marking operations.

Filtering options

	<p>Mark beams loose on both ends only</p> <p>With the filter option "Loose on both ends only", only the beams that are not connected to any lattice or solid geometry on both sides of the beam nodes will be filtered.</p>
	<p>Mark beams loose on at least one node or more</p> <p>With the filter option "Loose on at least one node or more", the beams that are not connected to any lattice or solid geometry on at least one node will already be filtered along with the beams that are not connected on both end nodes.</p> <p>Filtering beams with this option can subsequently cause other beams to become loosely connected, hence this operation iterates over the beam lattice structure to find the most stable configuration with no loose beams at all upon the filtering step.</p>

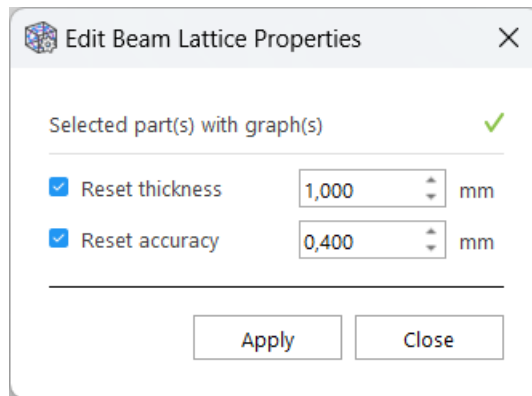
Note: In Magics, a beam node is considered to be connected in 3 different scenarios:

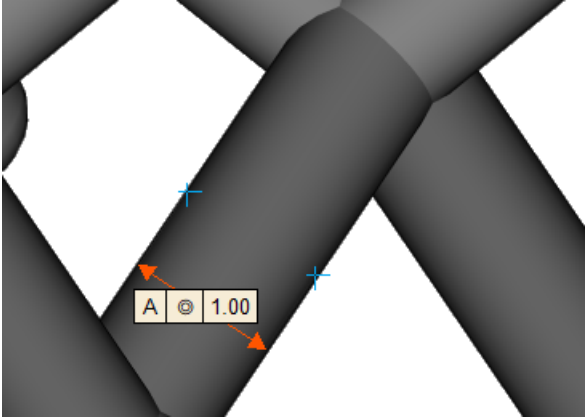
1. The beam node is shared with another beam in the same part.
2. The beam node is shared with a triangle node on triangle in the same part.
3. The beam node is located in the solid region of the part.

3. Edit Beam Lattice Properties



With this tool you can reset thickness and/or accuracy for beam lattices quickly or assign values for zero-thickness beams.



Select part(s) with graph(s)	If at least one part containing beam lattices is selected, the red cross mark will turn into green check mark accordingly
Reset thickness	<p>Default: 1 mm. All beams of the selected part will get uniform thickness specified in this field.</p>  <p>Note: thickness is the diameter of the beam.</p>
Reset accuracy	Default: 0.4 . Accuracy that will be used when converting beam lattices to mesh or directly by Materialise Build processor during printing.



Note: By default accuracy is always calculated as 40% of specified thickness. This is recommended ratio. However, it can always be overruled manually by the user.

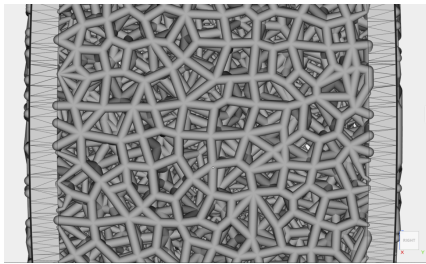
4. Convert Beam Lattice to Mesh



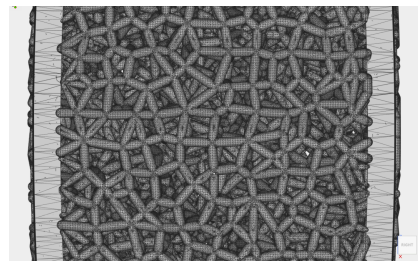
Convert beam lattices in selected part(s) to mesh geometry such that the part(s) can be edited and exported as mesh models. Only beam lattices with non-zero thickness and accuracy can be converted to mesh geometry.



Note: Converting beam lattices to mesh geometry can result in large amounts of triangles. Hence, it is advised to preserve the beam lattices as is as long as possible in your Design to Print workflow. If you have access to Materialise Build Processors, conversion of beam lattices to mesh geometry is not even required.



Original Beam Lattice



Converted Beam Lattice to Mesh

Convert Beam Lattice to Mesh
✕

Selected part(s) with valid beam lattice(s) ✔

Override thickness mm

Override accuracy mm

Estimated maximum memory is 30.0694 MB

REDUCE GENERATED TRIANGLES ●

Tolerance mm

Max angle °

Iterations

Selected part(s) with valid beam lattice(s)	<p>This is a status indicator if all the selected part(s) contains a valid beam lattice that can be converted to mesh geometry. A valid beam lattice is a lattice structure with non-zero thickness and accuracy beams in its structures.</p> <p>In the case beam lattice exists and the appropriate properties are overridden in this operation, the status might still show checked to indicate that the operation can still be applied.</p>
Override thickness	When input parts contains beam lattices with undesired final thickness, they can be quickly overridden by setting this to On, and setting a custom thickness value.
Override accuracy	When input parts contains beam lattices with undesired final accuracy, they can be quickly overridden by setting this to On, and setting a custom accuracy value.
Memory estimations	When one or more part(s) with valid beam lattice are selected, an approximation of the maximum RAM needed to perform the conversion is computed. This will be dependent on the complexity of the lattice structure and its properties.

Reduce generated triangles

Converting beam lattices to mesh can inadvertently result in large number of triangles. As such the standard triangle reduction operation is offered as a post processing step to reduce the number of triangles on the final model. The reduction is only applied to the new surfaces generated from the converted lattices, existing surfaces on the input part are left unchanged.

For more information on the triangle reduction parameters, see ***Triangle Reduction, page 108***

Chapter 6. Texture

6.1. Main

1. Select Texture

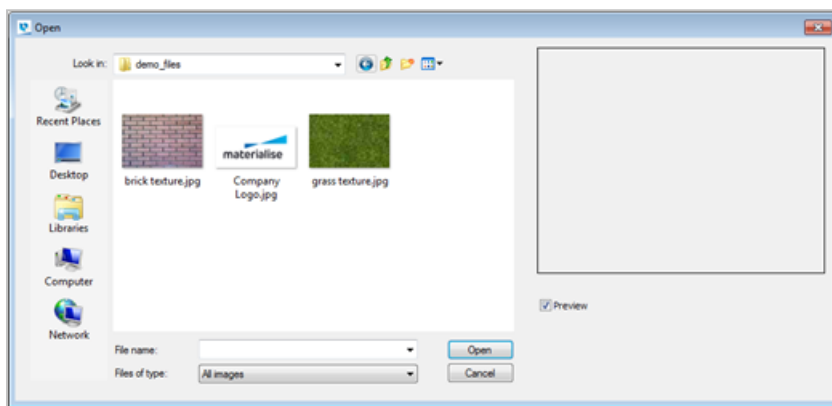


Select an existing texture in your design.

2. New Texture



Apply a new texture to a selected area or part.



You can browse to images to use as a texture. Any image file type is allowed. Select the image you want to use and click **Open**.

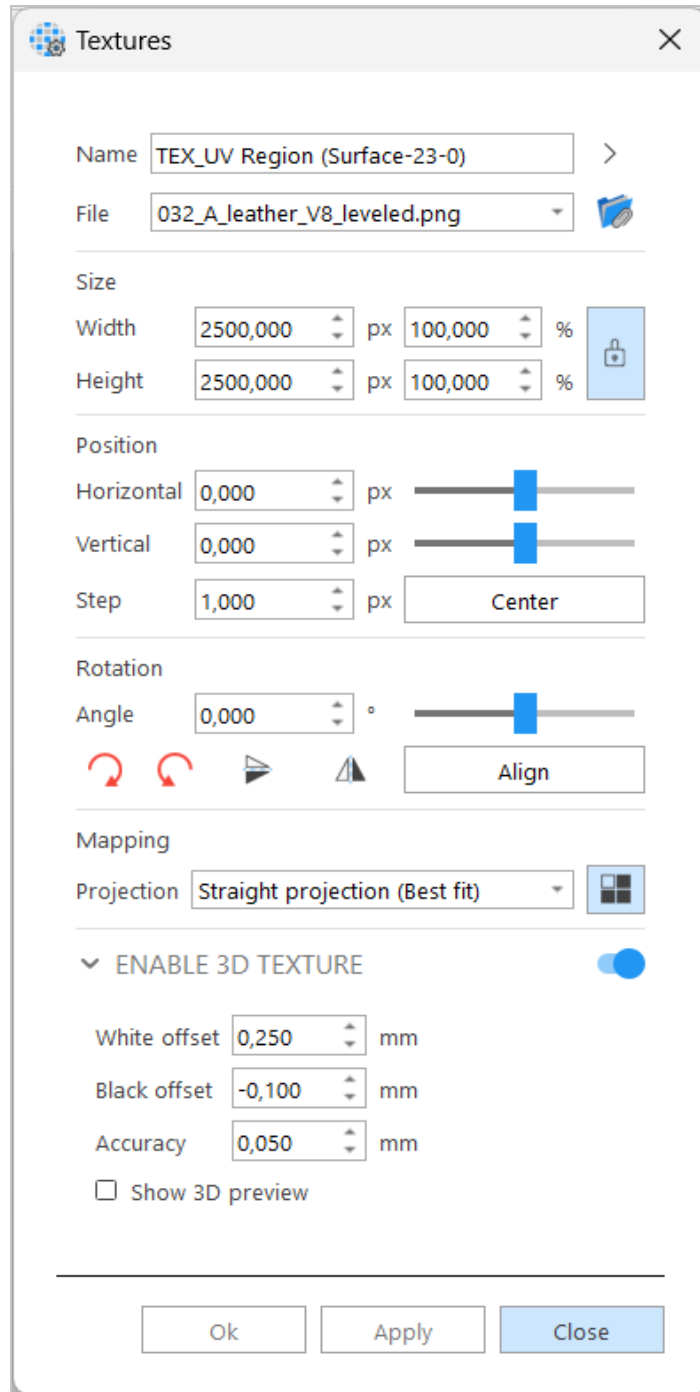
Next, the Textures dialog appears, allowing you to change the application of the texture.

- See **Edit Texture**, page 234 below.

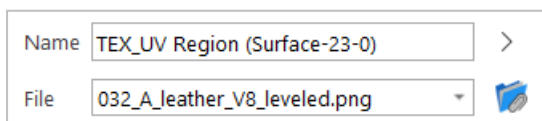
3. Edit Texture




In the Textures Dialog you can change the used image, and adjust the usage and appearance of the selected texture image. After making the desired changes, click **Apply** to stay in the Textures dialog after applying the changes. Click **OK** to apply and close the dialog. Click **Close** to close the dialog without applying the changes.

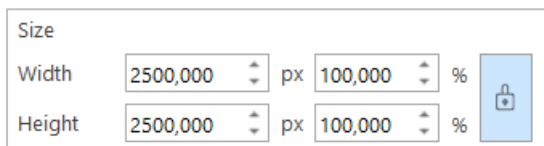


Texture




Name	Name of the surface where the texture is applied to. By default the name of the texture file used, but can be changed.
Next Texture (>)	Click to go to the next texture on the selected part.
File	Shows the file used. You can select any other loaded texture from the drop-down menu, or load a new texture using the  -button.

Dimensions



When applying a texture, the height and width of the loaded image is used. Change the units or the percentage by typing, or use the slider bar.

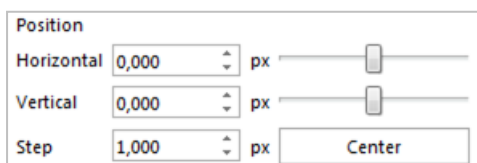
By default, the width/height ratio is locked. Click the  -button to change width and height independently.

Rotation



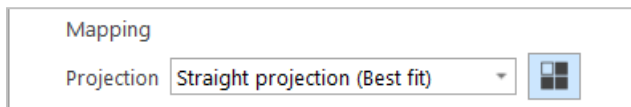
By default the rotation angle is 0 degrees. Enter a degree, or use the slider bar to interactively change the rotation. Align a texture to a specific edge by using the **Align Wireframe** button, then selecting the desired wireframe edge.

Position



Position allows you to change the actual placement of the image. By default, the image is centered. Use **Y+**, **X-**, **X+**, **Y-** to change the positioning, or use the slider bars. Click **Center** to center the image. Step lets you change the number of units that the position will be changed with each position change when using the buttons.

Mapping

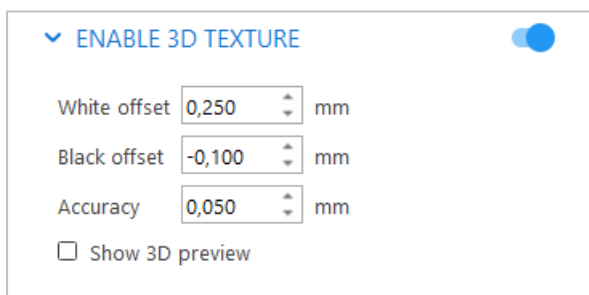


The mapping parameters can control how the images are mapped to the surface and if the image is tiled to fit the texture region. There are 2 main options for projections; Straight projection and Cylindrical projection.

Straight projection will work best with any open regions where the region is a smooth surface with the triangle normals and the total surface normal has minimum deviation of less than 30°. Cylindrical projection is good for cylindrical surfaces that has 2 closed contour(s).

The tile image toggle button allows you to enable or disable repetitive images across the texture region. Its by default set to enabled.

Enabling 3D Texture parameters



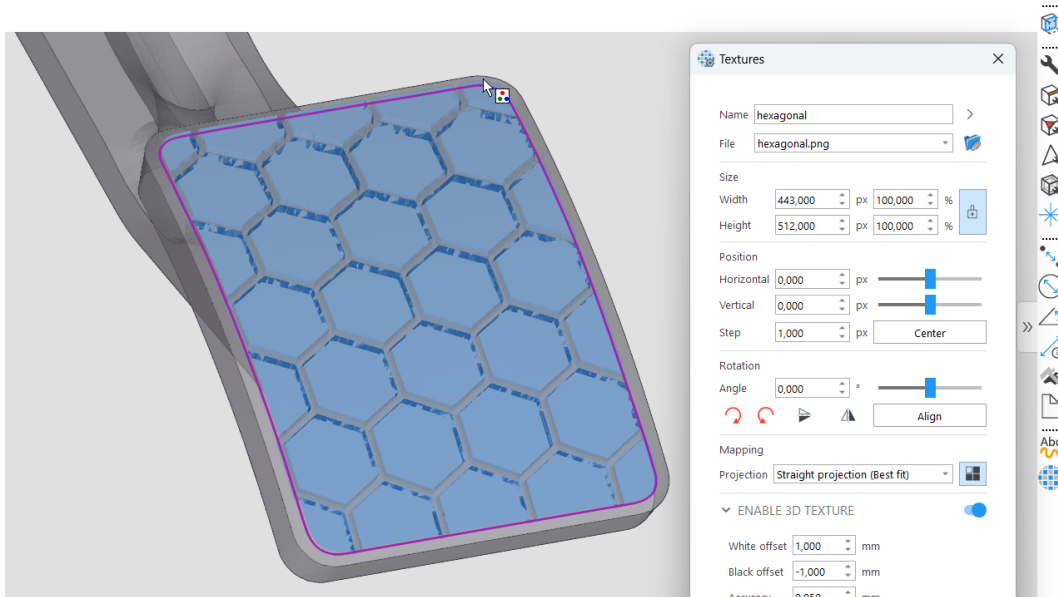
3D texture parameters makes it possible to define the white and black pixel displacements that shall be converted to actual printable geometry by the Materialise Build Processors when build jobs.



To benefit from this feature, you shall ensure that you have access to a build processors that can process 3D texture information from Magics RP.

The white and black offsets can be a positive or negative value where positive offsets results in displacement along the triangle normal and negative value will result in engraved displacement. While the accuracy parameter controls the resolution used by the Build Processor to generate the geometry. Lower accuracy values results in better quality geometries.

3D Visualization of the texture can be previewed with the "Show 3D preview" checkbox set to On as shown below.



4. Update Textures



Updates textures of the selected part(s).



Remark: more info about the status of the textures can be found in **Textures page, page 616**.

5. Copy Texture



Copy a selected texture.

6. Paste Texture



Paste a previously copied texture to a new area or part.

7. Delete Texture From Triangles



This option deletes the texture from selected triangles.

8. Delete Texture



This option deletes the texture of the selected part(s).

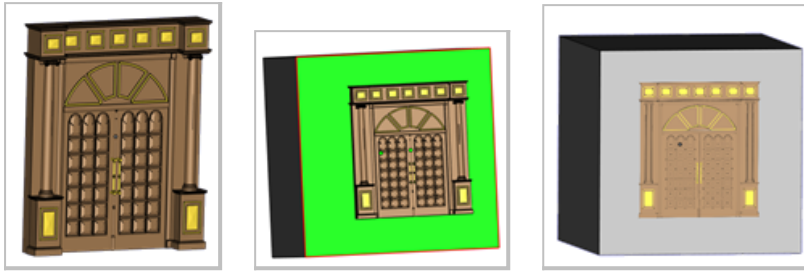
9. Part to texture



With this function, you can create a texture on a marked area by projecting another (textured) part onto that area. This can be useful when you want to preserve details that are too



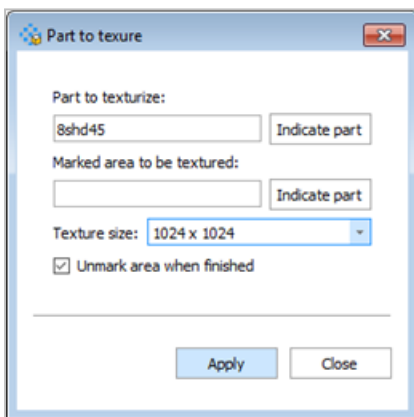
small or fragile to print.



For this operation, you will need two parts:

- The part which will be converted into a texture (part to texturize)
- The part onto which the texture will be applied (area to be textured).

Position the part to texturize on top of the area you want to texture, mark the area, and open the function. Use the “Indicate part” buttons to fill in the correct parts and click “Apply” to perform the operation.



Part to texturize	This part will be converted to a texture.
Marked area to be textured	This part should contain the marked area onto which the texture will be applied.
Indicate part	Click this button, then click a part to fill it in into the corresponding field.
Texture size	The bigger the texture size, the more detailed the texture will be, but the more time the operation will take. In general, the bigger the area you want to cover, the bigger the texture should be.
Unmark area when finished	The area needs to be unmarked to see the result, but if you want to re-use the marked area for another operation, this function can be disabled.

Apply	Perform the operation without closing the dialog.
Close	Close the dialog without performing the operation.



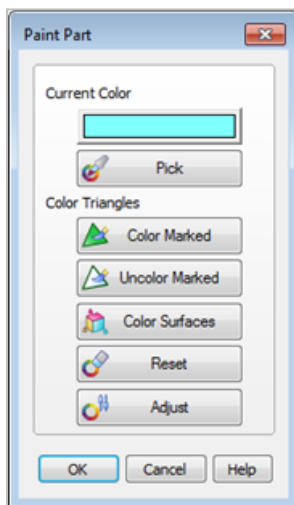
Remark: The fastest way to use Part to texture is to already position the parts correctly, mark the desired area, selecting the involved parts and then activating the function. This way, the correct parts will be filled in automatically. If only one part is selected, it will be filled in as “part to texturize”.


6.2. Color

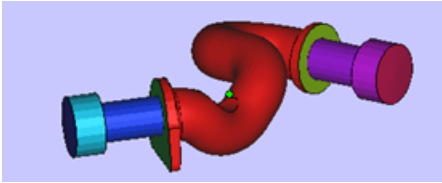
1. Paint Part



Parts and triangles can be painted. When you load a part it also has a color. This color is not a property of the part. It just acts like a background color to visualize the part. We call it the STL color. You can over paint this color with this paint function. When you use the paint functionality, the assigned colors can be saved if you save the part as 'STL (Colored)'.



	A dialog pops up and a color palette opens. You can choose a color. We call this color the paint color.
Pick	If you want to change the paint color to a color of some triangles on the part, you can push the pick button and click on a triangle with the desired color, or pick a color out of a texture. It is also possible to pick a color from a color per vertex map. The paint color will become the picked one.
Color Marked	When you push the Color Marked button the marked triangles are painted in the paint color.
Uncolor Marked	When you push the Uncolor Marked button the marked triangles get their STL color.

Autocolor	With the Auto button, each surface (amount of triangles surrounded by a wireframe) will get a separate color. 
Reset	The Reset button erases the colors. The part will get the STL color.
Adjust	Enables the user to adjust the brightness, contrast and gamma of the color.

2. Color Surfaces

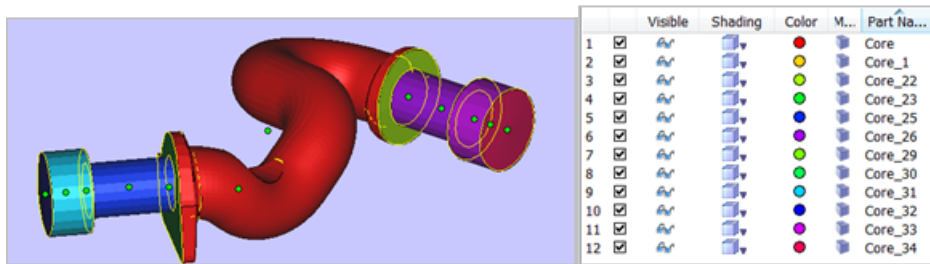


Apply a different color to each surface of the selected part(s).

3. Split part by color



All triangles with the same color will be split off the original and converted into a part. The new parts are listed in the part list.



6.3. Visibility

1. Toggle Textures Visibility



Toggle the texture visibility on and off, allowing to view the design without the applied texture(s).

2. Invert Textures Visibility



Inverts the texture visibility. If only one texture is used on a part, this function acts similar to the *Toggle Textures Visibility*. If multiple textures are used, visible textures will be made invisible, and invisible visible.

3. Triangle Colors



Show/Hide the painted colors (triangle & vertex color) of the part(s) in the current scene.



Chapter 7. Position

7.1. Transformation

1. Translate



Move the selected part(s) interactively or by inputting values.

Translate Part(s)
✕

Resulting coordinates

X mm

Y mm

Z mm

Relative translation

dX mm

dY mm

dZ mm

Enable snapping Size mm

Translate along line

Make copy Show preview

▼ TRANSLATION ORIGIN

Common for selection

Individual per part

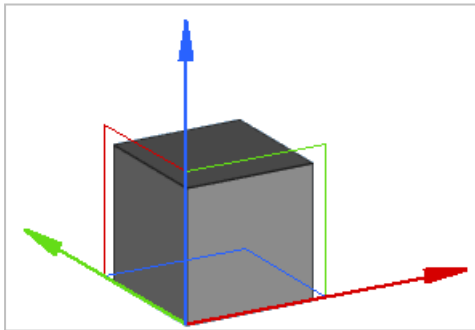
Define origin for selected parts:

	Min	Middle	Max	User	
X	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="text" value="0.000"/> mm
Y	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="text" value="0.000"/> mm
Z	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="text" value="0.000"/> mm

Note: Translate operation is available for Mesh, BREP and Slice part(s).

Resulting coordinates	Use the resulting coordinates fields if you want to move a part to a specific location.
Relative translation	Use the relative translation fields if you want to move the part a specific distance away from the current position.
Enable snapping	Enable if you want to translate with specified intervals. (For example: only move per 1 mm.) You can use the snapping option both for interactive translation and with coordinate fields.
Size	The snapping value used.
Translate	After enabling, select a line (part/triangle edge) along which you want to

along line	translate. This function can only be used when translating interactively.
Make Copy	Enable if you want to make a copy on the desired place and keep the original part on its place.
Show preview	When enabled, a preview will appear to show the result of the inputted values.
Define origin for selected parts	Define the translation origin of the part to select which point of the part should be moved (to which point in space).
	Multiple parts selected <ul style="list-style-type: none"> – Common for selection: all parts will be moved around the same origin – Individual per part: each part will be moved around its own origin
	Origin definition <p>Select between minimum, middle, maximum or user defined. You can also click on “Indicate point” and select a point directly on the scene. The gizmo will move to visualize your choice.</p>
To default position	Click to move the origin of the translation to the default position (see <i>Translate to Default Position, page 285</i>).
To default Z	Click to move the part to the default Z-height. X and Y will remain unchanged.
Apply	Apply the changes. The dialog box won't close, so you can easily perform the translation in multiple steps.
OK	Apply the changes. The dialog will be closed automatically.



To move interactively, click on an axis of the translation gizmo and drag to move the part along that axis. It is also possible to click on a plane of the gizmo to move the part within that plane.



The translate function will always take the active user coordinate system into account.



By default, if only one part is selected, the translation gizmo will appear in the minimum point* of the bounding box of the part. If multiple parts are selected, the gizmo will appear in the minimum point of the bounding box of all selected parts. All the parts will then be translated together, without changing the distances between the parts.

*The position of where the gizmo appears depends on what you select in “Define origin for selected parts”.


2. Rotate

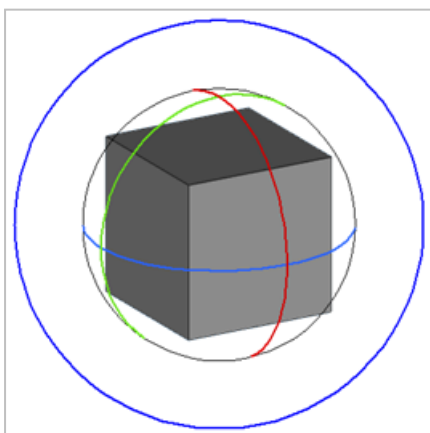


Rotate the selected part(s) interactively or by inputting values.

Note: Rotate operation is available for Mesh, BREP and Slice part(s).

Rotation Angles	Fill in the desired rotation angles in the X, Y and Z fields. The positive rotation sense is counter clockwise (CCK).
Enable snapping	Enable if you want to rotate with specified intervals. (For example: rotate per 45°) You can use the snapping option both for interactive rotation and with angle fields.
Size	The snapping value used.

Rotate around line	<p>After enabling, select a line (part/triangle edge) around which you want to rotate. You can then rotate interactively or by inputting a value.</p> <p>Rotation Angle <input type="text" value="0.00"/></p> 
Keep original Z position	<p>When enabled, the part's minimal Z-position will stay the same while rotating.</p>
Make copy	<p>Magics will create a copy on the desired place and keep the original part on his place.</p>
Show preview	<p>When enabled, a preview will appear to show the result of the inputted values.</p>
Rotation Center	<p>There are 3 options available:</p> <ul style="list-style-type: none"> - Center of selection: If multiple parts are selected, they will all rotate around the center of the selection, moving their position. - Individual part center: Each part will rotate around its own center, so their position won't change, only the orientation. - Custom rotation center: You can define a point around which the selected part(s) should rotate. With "Indicate point", you can easily select a rotation center. "Default center" resets the values to the center of the selection.
Apply	<p>Apply the changes. The dialog box won't close, so you can easily perform the rotation in multiple steps.</p>
OK	<p>Apply the changes. The dialog will be closed automatically.</p>



To rotate interactively, click on an axis of the rotation gizmo and drag to rotate the part along that axis. To rotate perpendicularly to the screen, use the outer (blue) circle. To rotate in an unrestricted way, click in between the axes of the rotation gizmo and drag.

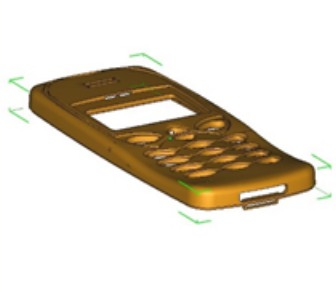


Remark: The interactive rotate function will always take the selected user coordination system into account.

3. Pick and Place Parts



This command allows the user to translate and rotate (around the axis perpendicular to the platform) selected parts on a platform by mouse movements. You can select the part by first clicking on the icon and then clicking on the part. The pick and place tags will appear. There are nine tags on a selected part in the pick and place mode:



- One translation tag: the filled green or white circle located in the center of the part.
- Eight rotation tags: the hollow green or white tags located on the corners of the bounding box.

This command allows easy positioning and nesting of the parts on the building platform.

If the cursor is positioned above the translation tag, the cursor will change to the translation cursor (☞). To translate the part, push the left mouse button. If several parts are selected, they will all move in the same direction over the same distance.

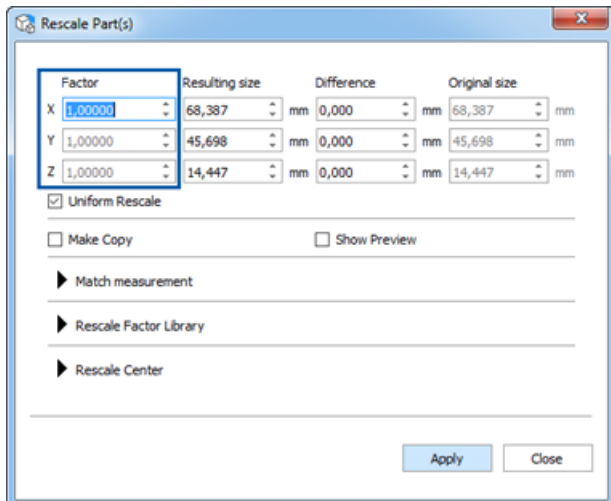
4. Rescale



A part can be rescaled with different factors in the three main directions or set to a certain size in the three main directions.

Note: Rescale operation is available for Mesh and BREP part(s).

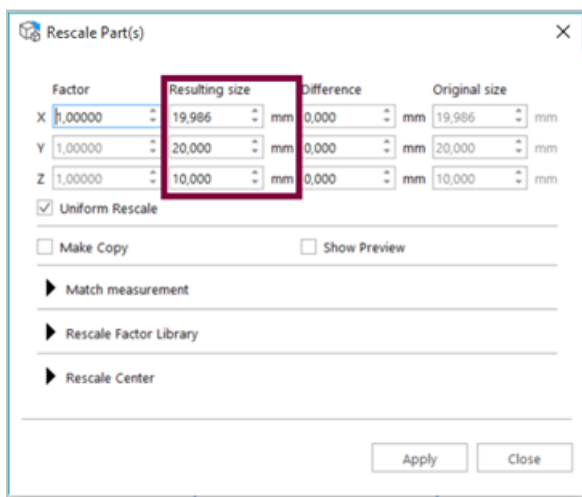
Rescale factor



The factor is a multiplying value for the dimensions in that direction. When the factor is 1, no rescaling is done, when the factor is 2, the size is doubled. A factor bigger than 1 will enlarge the part, a factor smaller than 1 will shrink the part.

Resulting Size

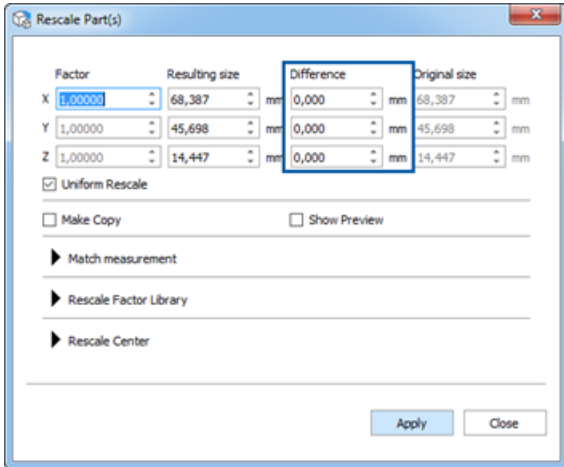
The resulting size column has a dual function. First of all it is showing the actual dimensions of the selected part but it also allows to set the dimensions of the part in the 3 main directions in absolute values.



Checkbox Options

Uniform Rescale	The rescale factor is identical in all directions.
Show Preview	Checking this box will show a preview of the rescale.
Make copy	Make a copy of the original part before resizing or rescaling

Difference Dimensions



If you want a part to become for instance 2mm bigger in the X-direction, enter 2 in the dX edit box. The corresponding factor(s) will change accordingly.

Rescale Factor Library

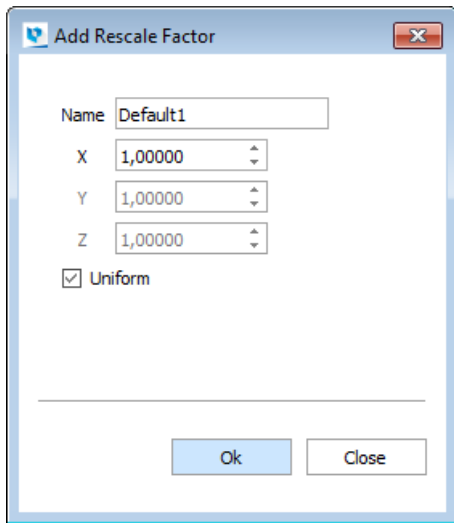


This library contains all stored rescale factors. This allows you to rescale your part very fast with the most common rescale factors. Two predefined ones (**inch->mm** and **mm->inch**) are provided to you. They can be easily accessed as **Convert inch to mm** and **Convert mm to inch** from the Quick Search as well.

You can edit and add new rescale factors very easily.

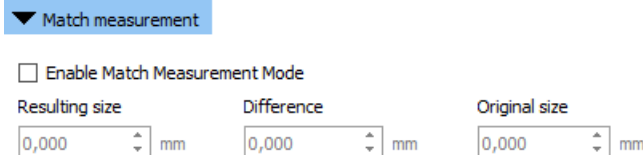
New	Create a new rescale factor. (see dialog box below)
Edit	Open the selected rescale factor so you can edit it.
Delete	Delete the selected rescale factor.

1. Add Rescale Factor



Name	The name of the rescale factor
Direction	Different directions can be specified to perform the rescale
Uniform Rescale	The rescale factor is identical in all directions

Match



A required value or indicated measurement can be used to rescale the part.

Size bounding box	The part can be rescaled in X, Y and Z direction. (Following the bounding box)
Measurement (M)	A defined measurement can be used to rescale the part.
Select Measurement	By clicking the 'Select Measurement' button a measurement can be selected.
Current value	The indicated value is shown.
New value	The desired value.

**Note: Match Measurement Mode is not available on measurement annotations made on BREP part(s) currently.*



Advanced options

1. Rescale Center

▼ Rescale Center

X mm

Y mm

Z mm

Rescale Each Part Around its Center

Keep Original Z Position

Rescaling is by default done around the center of each part individually. Rescale Center allows the user to define a rescale center different than the center of the WCS and enter the desired coordinates. Each part will now be rescaled around this new center. The original z-position can be maintained. You can also choose to add the scale factors to the part name.

5. Mirror

Mirror

Mirror about

YZ plane

XZ plane

XY plane

3 points plane

Plane perpendicular to indicated line

Plane coincident with indicated face

Plane parallel to screen

> MIRROR PLANE POSITION

Make copy

Preview

Apply Close

Note: Mirror operation is available for Mesh and BREP part(s).



When mirroring parts, you can choose to do this about:

- YZ - plane
- XZ - plane
- XY - plane
- 3-points plane
- Plane perpendicular to indicated line

- Plane coincident with indicated face
- Plane parallel to the screen

If 'Make Copy' checkbox is checked, a new part is created and the original input part is preserved with no changes. When several parts are selected, they will be mirrored around their common center, in case you accept the default option of mirroring around the center of the Parts.



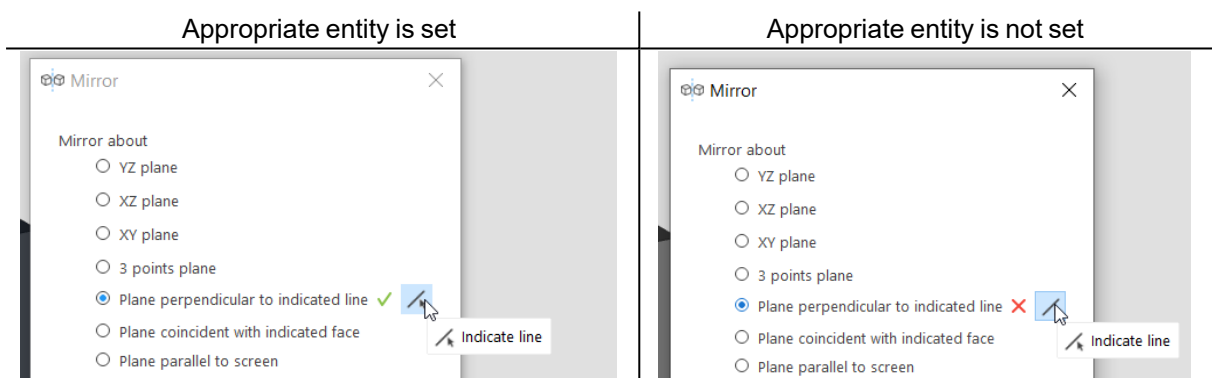
Note: The naming template for the mirrored part can be set in Settings > General > Naming

If the 'Preview' checkbox is checked, a preview is shown before any changes are made. Changes are only applied when you click on **Apply**.

Picking entities for fit planes

When any of the Mirror about method that requires a information to be picked is selected, you would need to specify the appropriate input for the Mirror operation to be performed. You are shown a check mark if the appropriate entity is already set.

The mouse mode to pick entity can be activated manually by clicking on the icon button for the entity picker.



Mirror Plane Position

▼ MIRROR PLANE POSITION

Through the center of selection

Offset distance mm

The real position of the mirror plane can be further modified by setting an offset distance along the plane normal. The plane normal is a vector direction perpendicular to the plane. To be able to switch the direction of the offset the switch direction button can be clicked.



Note: In the current version, some Mirror plane positions are not available for some specific Mirror method options.

If 'Through the center of selection' is checked the plane is automatically offset such that the plane is going through the center of the combined bounding box of all the selected parts.



Note: The current active user coordinate system is used as the main axis when computing the starting position of the mirror plane.

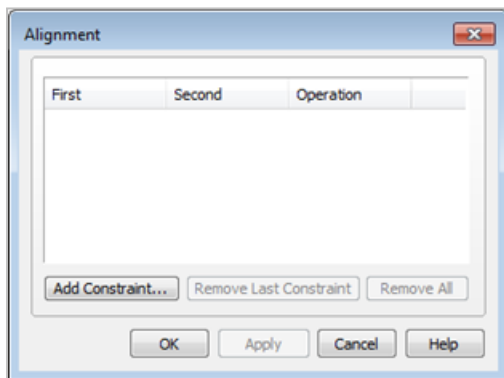
6. Alignment



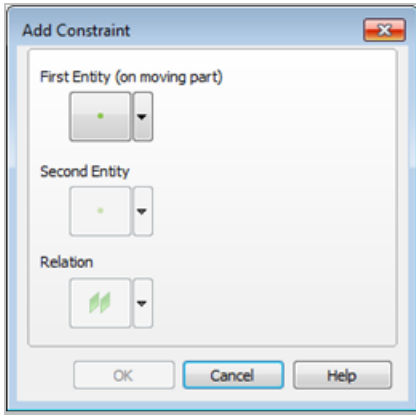
The alignment feature allows you to align one part to another. To align two parts you use one entity (point, line, circle...) on each part and you define the desired relation between the entities. The relation is called an operation.

Background Information:

A part has, in a 3D-space, 6 degrees of freedom (=dof): 3 translation (t1, t2, t3) and 3 rotation (r1, r2, r3) degrees of freedom. The directions 1, 2, 3, 1', 2' and 3' are two sets of 3 directions, perpendicular to each other. Each alignment action of two parts will decrease the amount of dof's with at least 1. At the end no degrees of freedom are left, then the parts are aligned.



Add Constraint		A dialog pops up where the user has to define the entities to align and the relation.
Remove Constraint	Last	Removes the last added constraint.
Remove All		Removes all the constraints.



The Add Constraint dialog box has three buttons:

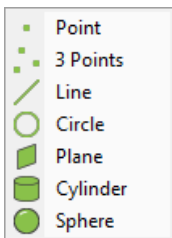
- The first and the second select the entities
- The third is used to select a relation between the entities.

At the right of each button, there is an arrow. Pushing these arrows pops up a list with the possibilities for this entity. Pushing the arrow next to the Relation button shows the possible operations for that particular combination of entities. If you agree with the constraint, you can add it to the list by clicking the OK button.

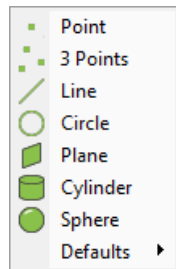
The Entities

There are six entities you can use to align parts. Each entity has its specific properties. The properties are shortly mentioned in the following table together with a brief explanation. There are eight (so called 'Defaults') you can use to align a part to the coordinate system.

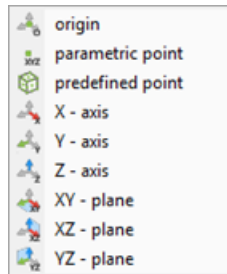
'First' Entities



'Second' Entities and Defaults



The Defaults



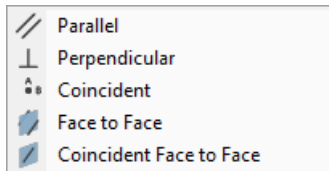
Entity	Property	Description
Point	Point Coordinates	Point properties are its coordinates. There is no direction on a point. Thus, when one of the entities is a point, only coincident can be chosen as a relation.
Line	Starting- and Endpoint	A line is defined by its starting- and endpoint. It also has a direction. Thus you can make a line parallel or perpendicular to another entity with a direction.
Circle	Centre point, Radius and	A center point, a radius and a normal define a circle. The normal is representing the plane containing the circle. It is shown on the

Entity	Property	Description
	the normal	screen because the user needs to know the direction of the normal in order to choose between 'Parallel' and 'Face to face'.
Plane	The normal of the plane and the point where you marked	A plane is defined by its normal and by the point you marked the plane with. The normal of a plane can be derived from the STL data. It follows the normal of the triangles of the plane (i.e. outside of the object)
Cylinder	Direction of the axis, radius and a point on the axis	The properties of a cylinder are the radius, the direction of the axis and a point on the axis. A cylinder is internally treated as a line (the cylinder's axis). Thus, a certain operation on a line or on a cylinder will result in the same change of dof's.
Sphere	Centre point and radius	The properties of a sphere are the center of the sphere and its radius. A sphere is treated as a point (the center of the sphere).



Remark: Using the align function for the creation of a new UCS will show the defaults and not the first entities like in the image above.

Relations



There are five possible relations. You can make two entities Parallel, Perpendicular, Coincident, Face-to-Face or Coincident Face-to-Face.

Parallel	This operation will make the two entities parallel, their normal pointing in the same sense.
Perpendicular	When for example two planes are perpendicular, there will be an angle of 90° ($\pi/2$ rad) between them.
Coincident	Coincident is chosen when you want two entities to be on the same location.
Face to Face	The two entities will be parallel, and their normal will point in the opposite sense.
Coincident Face to Face	This is a combination of 'coincident' and 'face to face'.

Advised Way of Working

Select 2 parts and push the alignment button (Tools Toolbar by default) or choose for Menu/Tools/Alignment to enter the alignment mode.

- Click on the Add Constraint button.
- Only the Select First Entity button is available. Click it and mark the first entity.
- The Select Second Entity button becomes available. Once you marked the second entity, the Relation button becomes available as well.
- From now on you can change the first entity (type can also be changed) and the second entity (and if desired its type) until you choose a relation.
- If the listed set of instructions aligns the parts as desired, push the OK button.
- Now you can proceed with another constraint.

If you want to exit, push the OK button on the dialog box or if you don't want to save the operations, close the dialog box with the Cancel button of the dialog box in the upper right corner of the window.

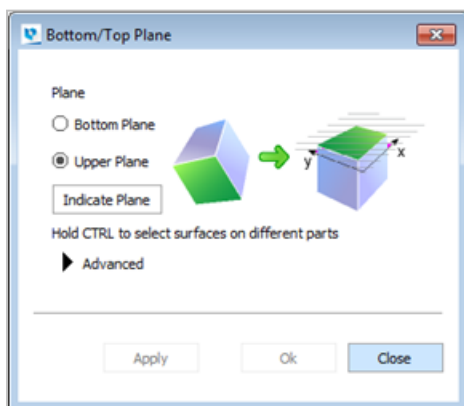
A closer look on the relations and entities

Strictly there are only two kinds of relations: an orienting and a locating operation. The Parallel, Perpendicular and Face to face are pure orienting operations. The directions of the two entities are opposite in the Face to face orientation. Coincident is a locating operation. When you choose Coincident, an orientation is added if the entities have a direction, as is the case for a plane. Coincident Face to face is again a combination. It's the same as Coincident and Face to face.


7. Bottom/Top Plane



This command allows easy orientation of the part by indicating a plane as the bottom/top plane. This plane will be automatically oriented parallel to the platform. The bottom/top plane window looks like this:



Indicate	The user selects one triangle and a whole plane (according to the plane selection)
----------	--

	<p>parameters) will be indicated by the default green marking color. The selected plane will be placed parallel to the platform (// XY-plane). E.g. Bottom plane selection in the following figure.</p>
<p>Plane</p>	

Advanced

▼ Advanced

Plane Selection Parameters

Surface Tolerance mm

Angle Deviation °

Positioning

None

Keep Original Z Position

Translate to Default Position

Translate to Default Z Position

Plane Selection Parameters	
Surface Tolerance	Indicates the maximum deviation in mm or inches that a related triangle may have to be part of the same plane that contains the selected triangle
Angle Deviation	Indicates the maximum angle in degrees between the normals of a related triangle and the selected triangle, in order to be part of the same plane.
Positioning	
None	No translation is done.
Keep Original Z Position	The part first will be rotated and next will be translated in such a way that the original minimum Z position remains the same.
Translate to Default Position	The part first will be rotated and next will be translated to the default part position.
Translate to Default Z position	The part will be translated to the default Z position.



The indicate place function is only performed when the part is loaded in standard mode. If the part is loaded in compact mode, the indicate plane function is going to act like indicate triangle.

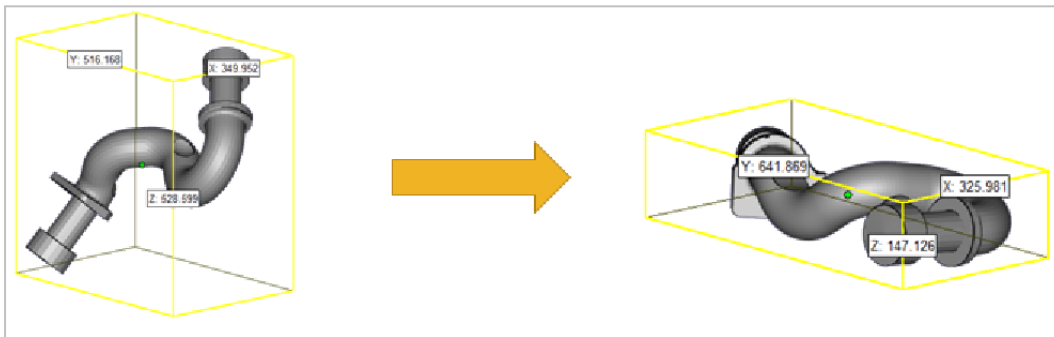


Use CTRL+click left mouse button so select multiple parts. Click a part a second time to deselect.

8. Minimize Bounding Box



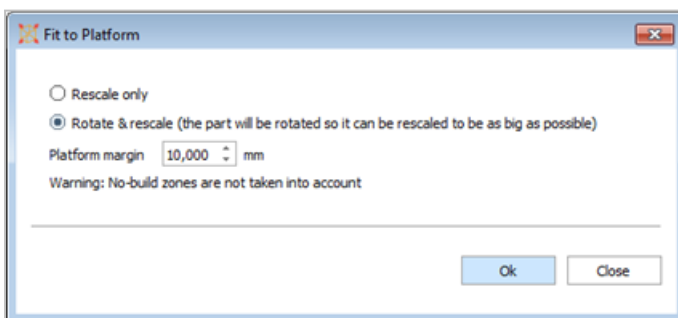
The part will be rotated in such a way that its bounding box is minimized. The scale of the part itself will not change. This function can help with quoting and efficient build preparation.



9. Fit to Platform



Fit to platform scales the selected part(s) so it fits in the build envelope of the active machine.



Rescale only	When this option is selected, the part will be rescaled without changing the orientation.
Rotate & rescale	The part will be rotated so it can be rescaled to be as big as possible. The algorithm takes the shape of the build envelope into account.
Platform	The distance between the platform edge and the part. The default value is taken

margin	from the Machine properties (Platform -> Automatic placement -> Platform margin).
--------	---



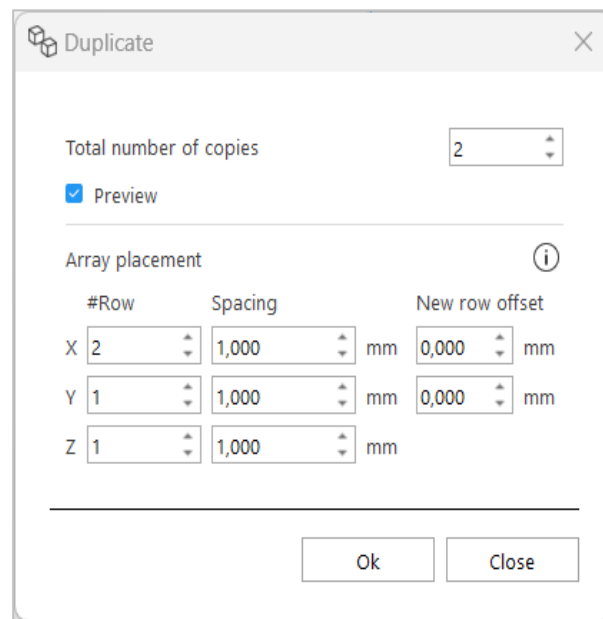
Remark: If multiple parts are selected, they will all be rescaled individually. If you want them to be rescaled as one, you will first have to merge them. The default part height (Machine properties -> Platform -> Default part position -> Minimum Z) will also be taken into account to ensure the part will fit into the build envelope.

7.2. Copies

1. Duplicate



This command duplicates the selected part(s) on the current active scene.



Note: Duplicate operation is available for Mesh, BREP and Slice part(s).

Total Number of Copies	Here you have to indicate the total number of parts (the original part included) you would like to have in the end.
Preview	When enabled, a preview will be shown for each of the copy with its placement in the scene.
Array Placement	
#Row	Its the number of row(s) that will be added along the specified axis. This ensures the copies are always placed in an array placement.
Spacing	The distance between 2 subsequent copies' bounding boxes along the main

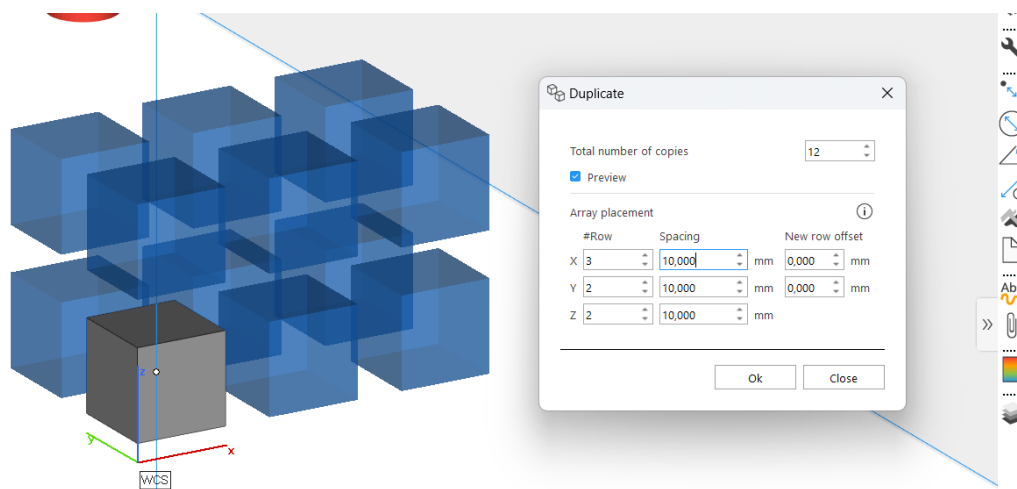
	axes.
New row offset	Sets the position offset of copies along the orthogonal axis of the of the copy to achieve a staggered array placement. For example, new row offset on X axis will have the new rows along X to be offsetted along the Y axis and vice versa.

Array placement

With array placement parameters it is possible to achieve part placement configuration on platforms quickly. The two primary placement configurations are:

LINEAR PLACEMENT

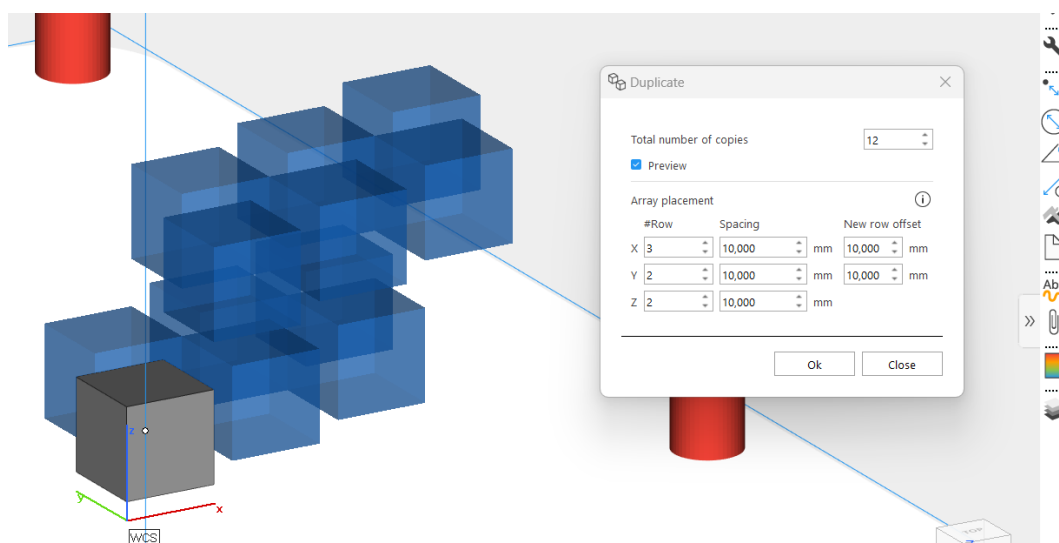
With linear placement, copies are placed with the same spacing interval along the specified axis.



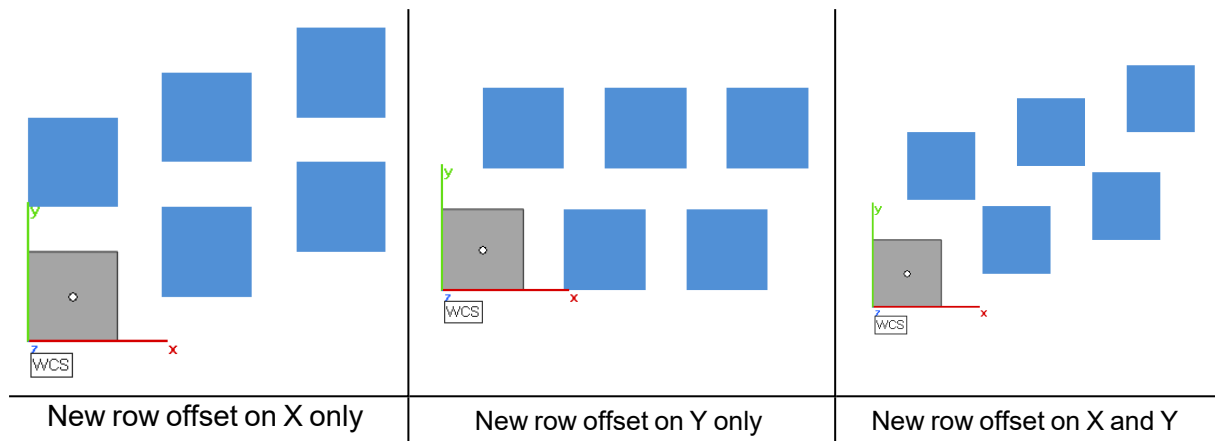
STAGGERED ARRAY PLACEMENT

It is also possible to achieve a staggered placement of copies on the XY plane quickly. This configuration could be useful for builds where the number of copies are limited and they shall not be placed in a linear array to reduce the overall build failure risks.

To achieve such configuration a non-zero new row offset value can be set for copies on the X axis or the Y axis.



Specifying a non-zero new row offset value for copies on the X axis, will result in each new row of the copies on the X axis to be offset along the Y axis and viceversa. The table below shows the different new row offset value configurations.



Unique copies and virtual copies

Based on the context of the operation, the operation is able to create copies as unique parts or virtual copies that can reduce the file sizes. In the Part scene, the duplicate operation will always result in new independent unique parts.

Meanwhile, in the Platform scene, the duplicate operation will generally create a virtual copy that shares the underlying geometry that is more performant overall.



There is a special case where parts with supports when duplicated along the Z axis, all new copies will be new unique parts. Being aware of it can ensure user can reduce the amount of deep copies created to ensure more performant app experience.

Part naming

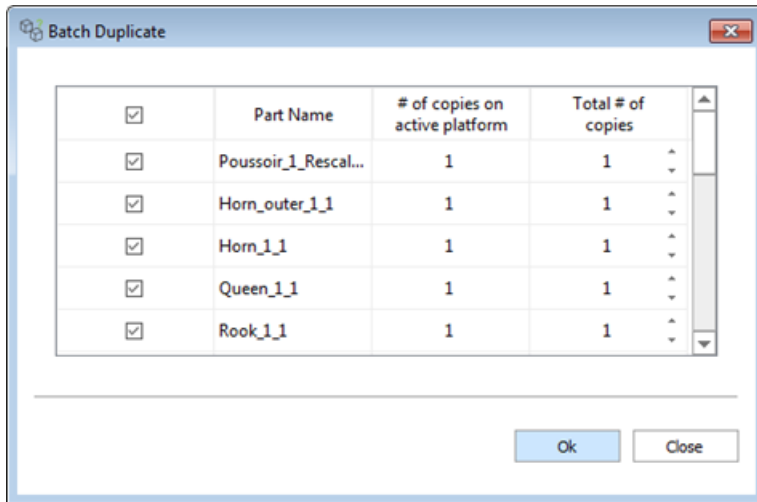
When new parts are created, their name is made unique based on the Naming template for Duplicate operation in the in the settings. See **Naming, page 519** for more information.

2. Batch Duplicate



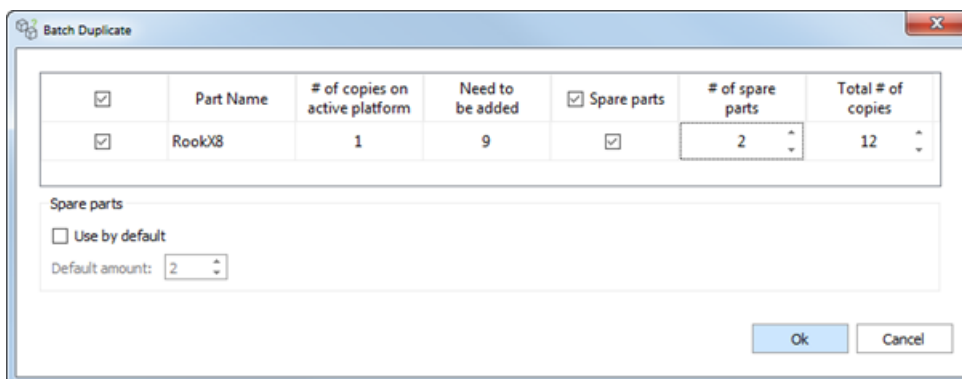
Batch Duplicate enables you to make a different amount of copies for multiple parts at the same time. It will help you save time by immediately placing the desired amount of duplicates of each part onto a platform.

When you have Streamics, Batch Duplicate will detect how many copies were ordered and immediately fill in the required number, taking into account already planned and built parts.



Dialog without Streamics

Open Batch Duplicate on the scene where you want to add copies. All the unique parts that are loaded in the Part Scene will be listed. Enable the checkbox of the parts you want to add. In “Total # of copies” you can input the desired number of copies. “# of copies on active platform” shows the amount of copies that are already present on the current scene. Press OK to make the copies. They will be placed through automatic placement.



Dialog with Streamics

When using Batch Duplicate in combination with Streamics parts, a few extra columns will appear: “Need to be added” will show you how many parts still need to be planned, minus the ones already on the platform. This value takes into account parts that are already built, parts that are planned on a build, and parts on other platforms in the same Magics session. It does not take into account platforms saved in Streamics. The formula looks like this:

Need to be added = ordered amount - parts built - parts planned - parts on platforms

Other extra columns are “Spare parts” and “# of spare parts”. This allows you to easily add some extra copies for fragile parts that might break. If you do this often, it might be interesting to enable “Use by default” and insert a fitting default amount. Whenever you open this Batch Duplicate again, the spare parts fields will be filled in and enabled. You can always disable them and change the amount of spare parts.



7.3. Automatic

1. Automatic Placement



This command will nest the loaded parts on the building platform. Please note that for nesting the parts in 3D, the Sintermodule (see Sinter Module, page 1) is required.

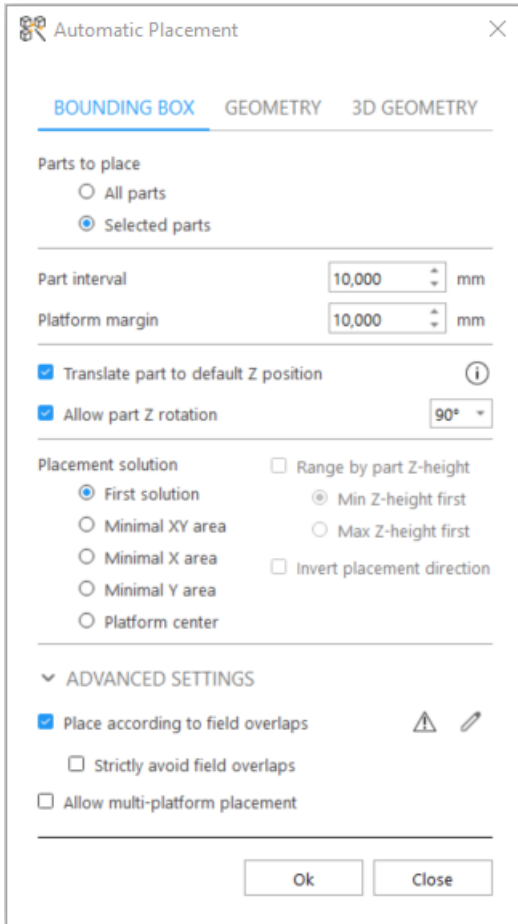
There are two options:

- Geometry based nesting
- Bounding box based nesting

Please recall, that while importing multiple parts you can also use the automatic placement algorithm to immediately position your parts on the platform. Automatic placement is also possible when the platform isn't big enough to load all the specified parts on the platform. A dialog box will appear indicating no solution is found, but a search is performed to find a solution outside platform borders. So even if the parts don't fit the platform, they are spread out to have a better overview.

Bounding box

Magics will nest the parts, representing the parts by their bounding boxes. This will result in a fast nesting, however the full surface of your machine will not be used due to the rough representation of the parts.

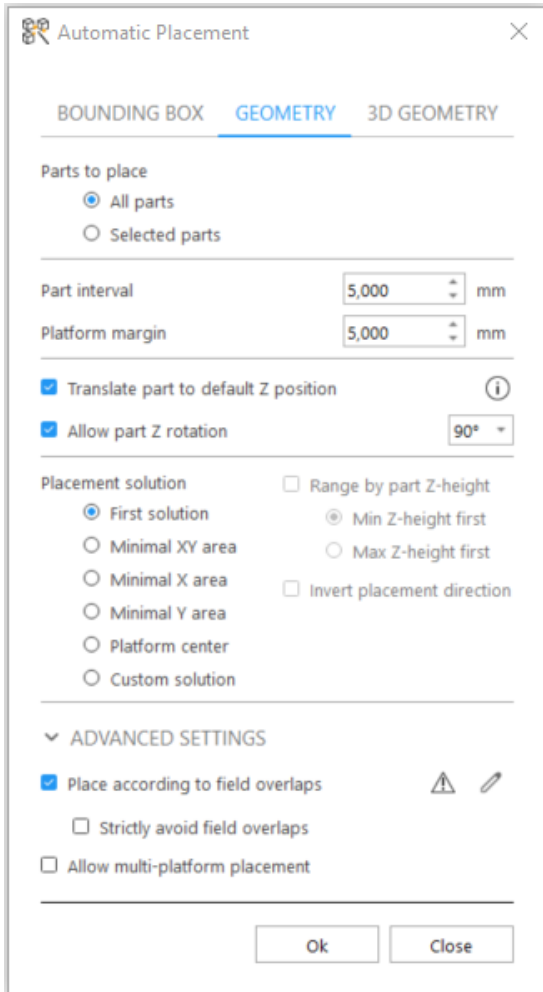


Parts to place	All Parts	Nests all the parts or only the selected parts.
	Selected Parts	
Part Interval	The minimum distance between two parts.	
Platform Margin	The minimum distance between (the bounding box of) a part and the edge of the platform.	
Translate part to default Z position	Moves the parts to the default z-height, but will remove the support.	
Allow part Z rotation	Allow part Z rotation will give Magics the liberty to rotate the parts while Autoplacing. The angle with which the parts can be rotated can be specified in the dropdown menu. Smaller angles will take longer to calculate, but might result in a denser nesting.	
Allow multi-platform placement	If this option is checked and the nested parts don't fit on the current Platform, Magics will create as many as needed new Platforms to nest all (or the selected) Parts.	
Placement solution		
First Possible Solution	With this option, Magics offers the first placement he finds for which all parts are nested on the platform.	


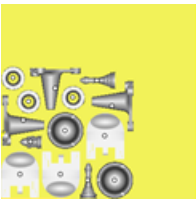
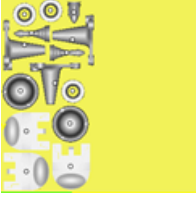

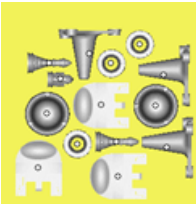
Minimal XY area	The total surface area of all the loaded parts is minimized.	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
			Min Z- height first	
Minimal X area	The delta-X of the total surface area of the loaded parts is minimized. Invert direction: The total surface area of the loaded parts is minimized on the opposite side of the platform.	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
			Min Z- height first	
		Invert placement direction	The total surface area of the loaded parts is minimized on the opposite side of the platform.	
Minimal Y area	The delta-Y of the total surface area of the loaded parts is minimized. Invert direction: The total surface area of the loaded parts is minimized on the opposite side of the platform.	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
			Min Z- height first	
		Invert placement direction	The total surface area of the loaded parts is minimized on the opposite side of the platform.	
Platform center	Parts are nested around the center of the platform. A circular shape is created.	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
			Min Z- height first	
Advanced settings				
Place according to field overlaps	When enabled, automatic placement algorithm will take scan fields into account and try to place each part into a single scan field and avoid field overlaps if possible. If a part doesn't fit into one scan field, then it will overlap into the minimal number of scan fields possible.			
Strictly avoid field overlaps	Force parts to not be placed into field overlaps. Parts that can't fit will not be placed.			
Allow multi-platform placement	If this option is checked and the nested parts don't fit on the current Platform, Magics will create as many as needed new Platforms to nest all (or the selected) Parts.			

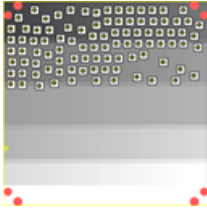
Geometry

A nesting based on bounding box can cause a waste of capacity in case of parts, which are having a big bounding box but a small projected area. Magics will nest the parts using the actual form of the parts and so increase the efficiency of the nesting.



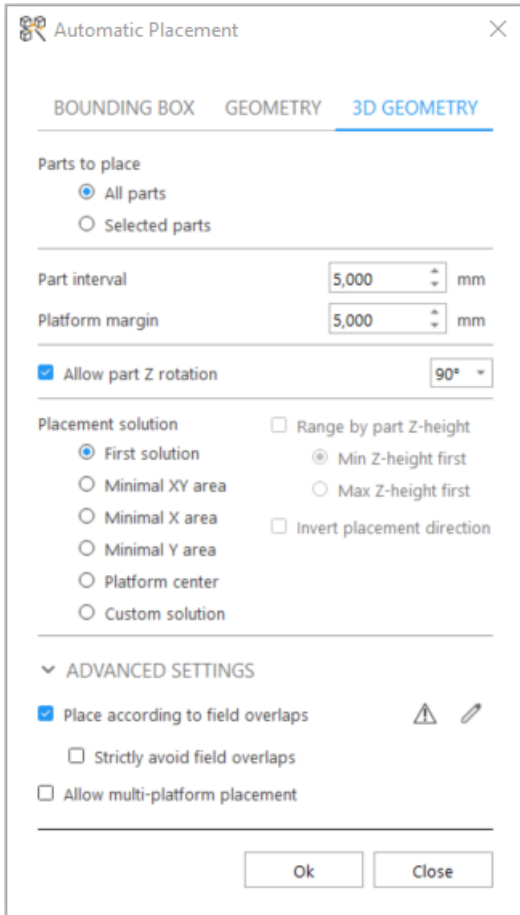
Parts to place	All Parts	Nests all the parts or only the selected parts.
	Selected Parts	
Part Interval	The minimum distance between two parts.	
Platform Margin	The minimum distance between (the bounding box of) a part and the edge of the platform.	
Translate part to default Z position	Moves the parts to the default z-height, but will remove the support.	
Allow part Z rotation	Allow part Z rotation will give Magics the liberty to rotate the parts while Autoplacing. The angle with which the parts can be rotated can be specified in the dropdown menu. Smaller angles will take longer to calculate, but might result in a denser nesting.	
Allow multi-platform placement	If this option is checked and the nested parts don't fit on the current Platform, Magics will create as many as needed new Platforms to nest all (or the selected) Parts.	
Placement solution		
First	With this option, Magics offers the first placement he finds for which all parts	


Possible Solution	are nested on the platform. 			
Minimal XY area	The total surface area of all the loaded parts is minimized. 	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
			Min Z- height first	
Minimal X area	The delta-X of the total surface area of the loaded parts is minimized. 	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
			Min Z- height first	
		Invert placement direction	The total surface area of the loaded parts is minimized on the opposite side of the platform.	
Minimal Y area	The delta-Y of the total surface area of the loaded parts is minimized. 	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
			Min Z- height first	
		Invert placement direction	The total surface area of the loaded parts is minimized on the opposite side of the platform.	
Platform center	Parts are nested around the center of the platform. A circular shape is created. 	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
			Min Z- height first	
Custom solution	With this option, you can add a grayscale image	Range by part Z-height	Max Z- height first	Arrange parts in the order of their

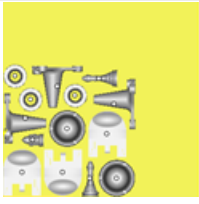
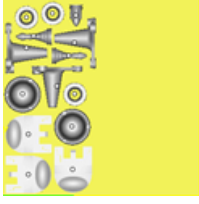
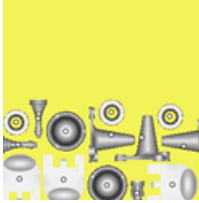
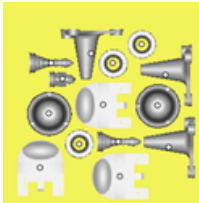
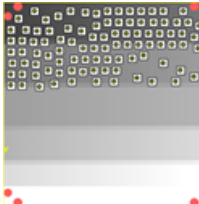
	to assign priority or penalty zones for Autoplacement.			
			Min Z- height first	height (either ascending or descending).
Advanced settings				
Place according to field overlaps	When enabled, automatic placement algorithm will take scan fields into account and try to place each part into a single scan field and avoid field overlaps if possible. If a part doesn't fit into one scan field, then it will overlap into the minimal number of scan fields possible.			
Strictly avoid field overlaps	Force parts to not be placed into field overlaps. Parts that can't fit will not be placed.			
Allow multi-platform placement	If this option is checked and the nested parts don't fit on the current Platform, Magics will create as many as needed new Platforms to nest all (or the selected) Parts.			

3D Geometry (for e-Stage users only)

Magics will nest the parts while making sure that the projection on the platform of the parts does not intersect.



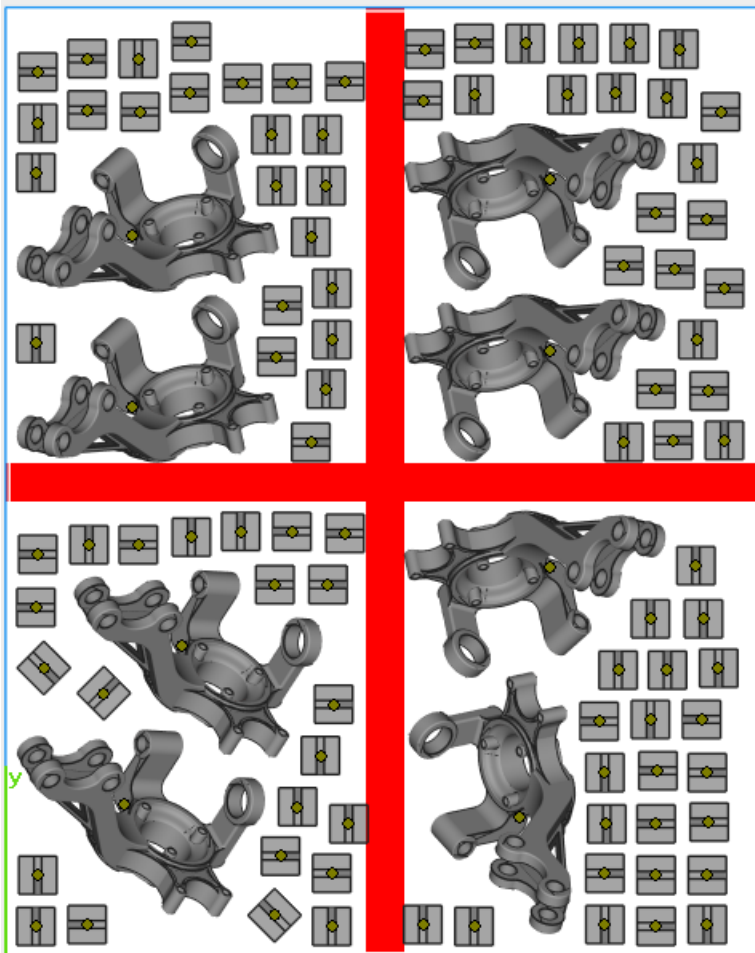
Parts to place	All Parts	Nests all the parts or only the selected parts.		
	Selected Parts			
Part Interval	The minimum distance between two parts.			
Platform Margin	The minimum distance between (the bounding box of) a part and the edge of the platform.			
Allow part Z rotation	Allow part Z rotation will give Magics the liberty to rotate the parts while Autoplacing. The angle with which the parts can be rotated can be specified in the dropdown menu. Smaller angles will take longer to calculate, but might result in a denser nesting.			
Placement solution				
First Possible Solution	With this option, Magics offers the first placement he finds for which all parts are nested on the platform. 			
Minimal XY area	The total surface area of all the loaded parts is minimized.	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either

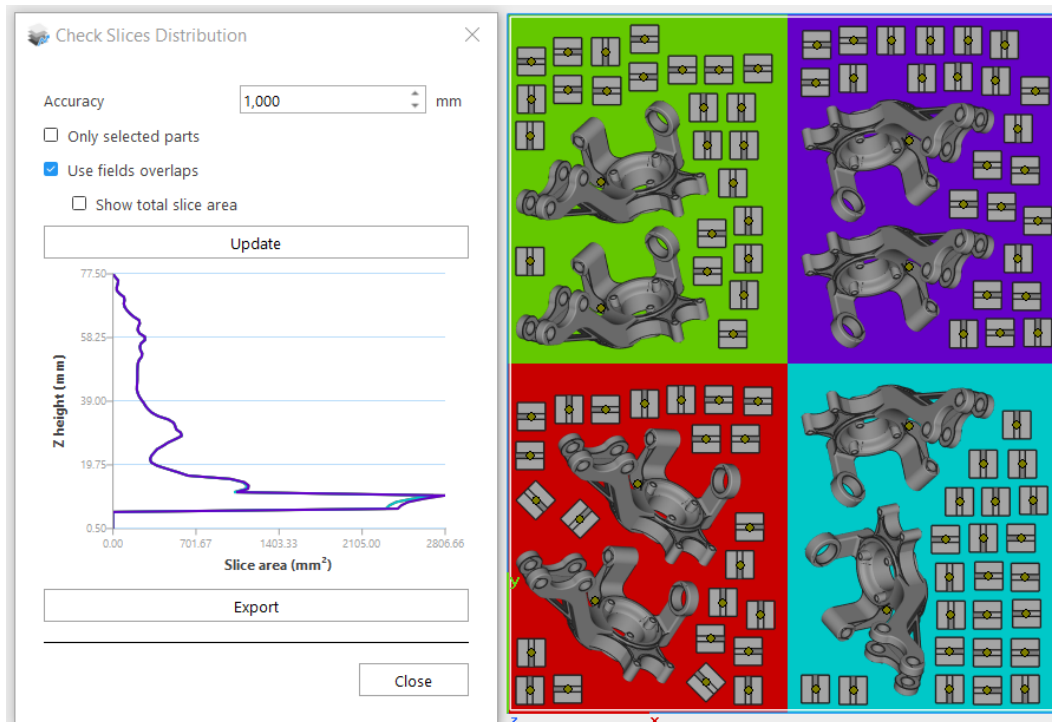
				
Minimal X area	The delta-X of the total surface area of the loaded parts is minimized. 	Range by part Z-height	Min Z- height first	ascending or descending).
			Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
		Invert placement direction	The total surface area of the loaded parts is minimized on the opposite side of the platform.	
Minimal Y area	The delta-Y of the total surface area of the loaded parts is minimized. 	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
			Min Z- height first	
		Invert placement direction	The total surface area of the loaded parts is minimized on the opposite side of the platform.	
Platform center	Parts are nested around the center of the platform. A circular shape is created. 	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
			Min Z- height first	
Custom solution	With this option, you can add a grayscale image to assign priority or penalty zones for Autoplacement. 	Range by part Z-height	Max Z- height first	Arrange parts in the order of their height (either ascending or descending).
			Min Z- height first	
Advanced settings				

Place according to field overlaps	When enabled, automatic placement algorithm will take scan fields into account and try to place each part into a single scan field and avoid field overlaps if possible. If a part doesn't fit into one scan field, then it will overlap into the minimal number of scan fields possible.
Strictly avoid field overlaps	Force parts to not be placed into field overlaps. Parts that can't fit will not be placed.
Allow multi-platform placement	If this option is checked and the nested parts don't fit on the current Platform, Magics will create as many as needed new Platforms to nest all (or the selected) Parts.



Outplace on platforms with field overlaps. If a platform has field overlaps enabled in Machine Properties (see details in 2 Machine Properties -> General info page), then all automatic placement options will take them into account and try to avoid them during placement. Magics will nest parts trying to evenly distribute them by slice volume between all available scan fields. Some parts might still end up in the overlap.





2. Orientation Optimizer

Introduction



The orientation tool was designed to help in analyzing and providing the best orientation for your part. It gives you the possibility to perform an analysis on your part(s) orientation or to automatically calculate the optimal part orientation based on specific criteria. The following criteria are implemented:

- Z-height (build height)
- XY projection
- Max XY section
- Support surface
- Support on marked

Depending on the technique that is used different parameters can play a role in the orientation of your parts:

For all techniques the *z-height* or *build height* is a very important factor. By reducing the build height, the build time can be reduced. The limited build height can also lead to the saving of expensive material.

In Stereolithography, two of the implemented criteria are of particular interest: to minimize the amount of *support surface* and to minimize the *XY projection area*. By orientating the part in such a way that the amount of support surface is minimized, less material and less finishing time is needed for the particular part. Currently, no attention is given to A or B-faces etc. so user interaction will be needed if this has to be taken into account. By minimizing the XY projection area, the parts are oriented in such a way that the total amount of parts on a Stereolithography platform can be maximized. Of course this contradicts with the minimum z-height criterion.

For Laser Sintering (plastics but more importantly in metal sintering) *large cross sections* should be avoided since an object can deform due to thermal stresses generated during the building process. Large cross sections typically generate a lot of thermal stress and should therefore be avoided.

For several techniques, support is needed in order to be able to build the part. There might be surfaces where the support structure is not wanted. For example, on surfaces where it is hard to remove the support in postprocessing.

The criterion *Support on marked* represents the percentage of the marked area that will need support. A Support on marked value of 10%, for example, implies that 10% of the marked area will need support. The lower the value, the better.



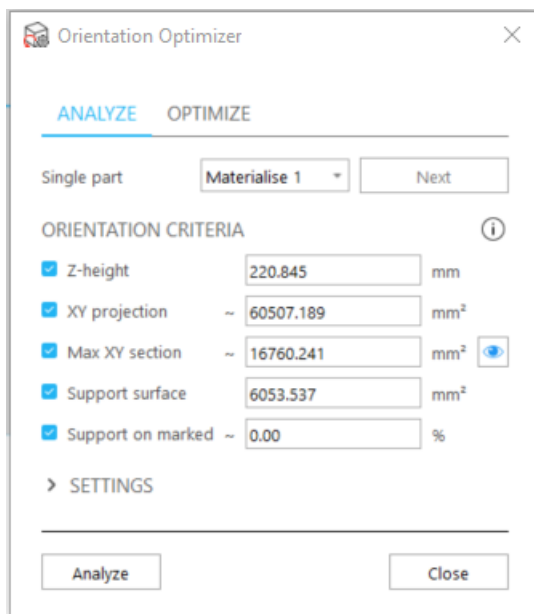
Remark: the parameters that are available within the Orientation optimizer depend on the type of license that is active. Z-height and XY projection are available for all users. Support surface and Max XY section are linked to other modules (SG and/or e-Stage for support surface and SG+ for the maximal cross section)



Support-related parameters and **Max XY section** do not take into account the beam lattice component(s) of the part.

Orientation optimizer: Analyze

The first page of the Orientation optimizer consists of the analysis tool. By pressing **Analyze**, the flagged criteria are calculated on the current orientation of the selected part. A user can observe the z-height, support surface, etc. in the current orientation of the part.

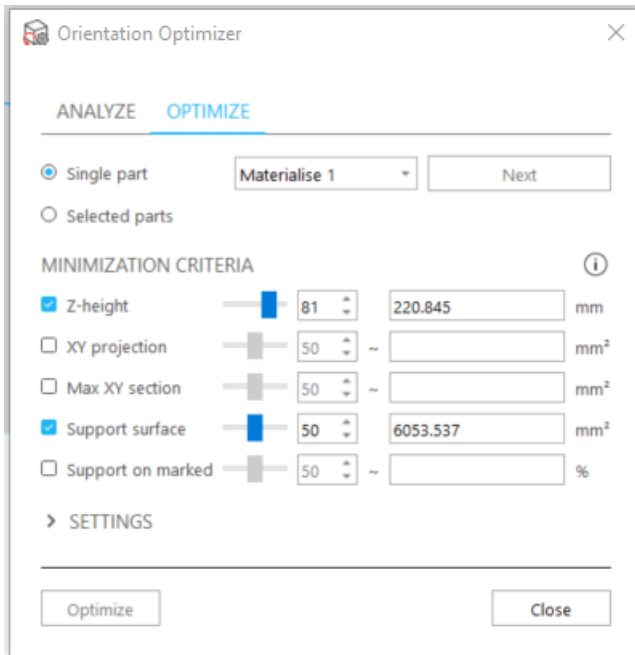


Current part	The analysis is performed part per part. With the “next” button you can navigate to the following part.
--------------	---

Orientation criteria	The Orientation criteria are used to analyze the current part(s) orientation.	
	Z-height	This gives you the possibility to check per part what the current z- height is (the distance from the platform to the highest point of the part)
	XY projection	This is the projection of the part in the XY plane (on the platform)
	Max XY section	The maximum cross sectional area is shown. This is found by slicing the part and calculating the cross section surface area in each slice.
		View Magics shows the maximal XY cross section on the part
	Support surface	This gives you the total amount of surface area (in mm ²) that needs support in the current orientation of the part.
	Support on marked	This represents how much area of the marked area that will need support (%)
Analyze	The analysis is performed/ executed on the selected part .	

Orientation optimizer: Optimize

The second page of the Orientation optimizer is the real optimization. The current part or multiple parts (the parts selected in the parts list) can be automatically oriented by using the criteria given below. The criteria which the user wants to use should be selected and with the sliders a relative weight of the parameters can be assigned (e.g. for metal machines, consider using the parameters Max XY section and Support surface together). Click "Optimize" to apply the optimal orientation based on both parameters. Afterwards, the sliders can be used to change the relative weight of the parameters, and the orientation of the parts will change on the fly. The more parameters are selected and the greater accuracy that is required (see Settings dropdown) the longer the calculation will take.



Single part	The optimization is performed on the current part	
Selected parts	The optimization is performed on all selected parts in the parts list	
Minimization criteria	Minimization criteria are the parameters that should be used for the optimization	
	Z-height	Orientate the part in such a way, that the height is minimal
	XY projection	Orientate the part with the lowest amount of XY-projection area.
	Max XY section	Orientate the part to have the smallest maximal cross section.
	Support surface	Orientate the part with the lowest amount of support area
	Support on marked	Orientate the part to have the smallest support area on the marked surface.
Optimize	This button will trigger the algorithm to find the orientation that fits the selected minimization criteria, display the part in the new orientation and show the values for each of the criteria for the selected part in the current part orientation.	

Settings



The optimization algorithm calculates all possible 3D orientations for the selected criteria for each part leading to a very large amount of possible solutions. To avoid lengthy calculations, the fast algorithm limits itself to 320 possible orientations whereas the accurate algorithm has a solution space of 1280 orientations.

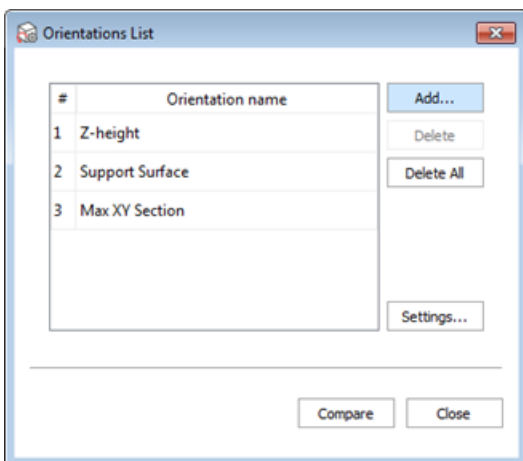
Optimize Algorithm	Fast	320 possible orientations are calculated
	Accurate	1280 possible orientations are calculated
Supported area selection angle	Use selection angle value from machine properties	The parameter that was set in the Machine Properties is used in the orientation tool.
	Specify manually	Specify the support area selection angle manually.

3. Orientation Comparator



This tool allows you to compare different orientation options and make a good judgement on different parameters. You can analyze the consequences linked to an orientation which can sometimes be hard to notice.

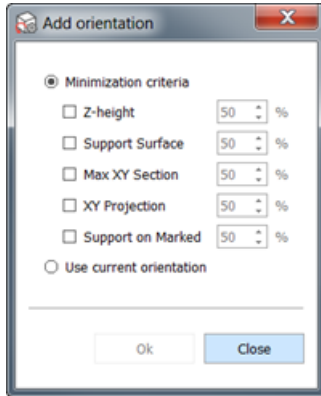
Orientation List



In this dialog, the orientations that will be compared, are listed. By default, the orientations with minimum z-height, support surface and max XY section are included. By clicking 'Add...', you can add an orientation to the list. It is also possible to rename orientations, delete them from

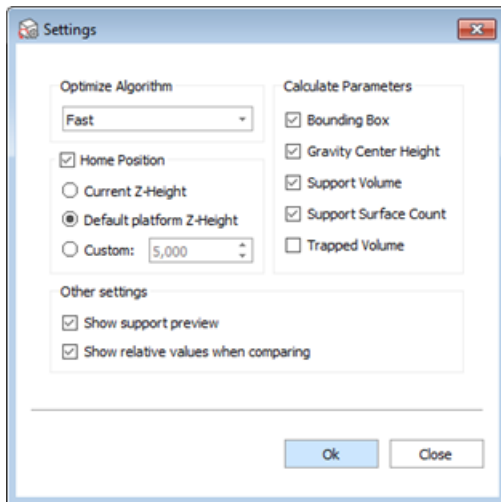
the list or change the setting of the comparator. The 'Compare' button will start the calculation and show you the comparison.

1. Add...



Adding orientations can be done in two ways. Either you can use the minimization criteria from the orientation optimizer or it is possible to use the current orientation of the part.

2. Settings




In this dialog, the comparator can be configured.

Optimize Algorithm	To avoid lengthy calculations, the algorithms is limited to an amount of orientations.	
	Fast	320 possible orientations are calculated
	Accurate	1280 possible orientations are calculated
Calculate Parameters	When the checkbox in front of a parameter is checked, this parameter will be calculated when pressing 'Compare'.	

	Bounding Box	The volume of the bounding box is calculated.
	Gravity Center Height	The z- coordinate of the gravity center is calculated.
	Support Volume	An estimation of the support volume is calculated. This estimation is based on the support preview in the supported area preview.
	Support Surface Count	The amount of support surfaces when entering SG is calculated.
	Trapped Volume	The sum of all trapped volumes is calculated.
Home Position	You can choose whether you want the lowest point of the part to be placed on a specific z-height during the calculation.	
	Current Z-Height	The current z-coordinate of the lowest point will be used.
	Default platform Z-Height	The default platform z-height that is defined in the machine properties will be used.
	Custom	You can define a custom value.
Other settings		
	Show support preview	When checked, this will show the support preview in the orientation preview.
	Show relative values when comparing	When checked and the comparing toggle is on, the parameters will be shown as relative values with respect to the reference orientation.

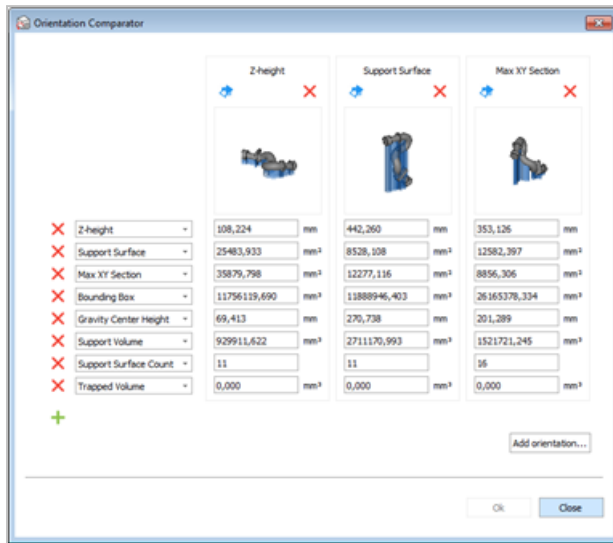
Orientation Comparator dialog

After clicking 'Compare' and some calculation time, the Orientation Comparator dialog opens. A preview of the calculated orientations is shown. When you click on the preview, the orientation of the part will change in the scene. This allows more in-depth inspection.

Below the preview, you can find the parameters that correspond with the orientation. It is possible to compare these values with color codes by pressing the Compare toggle . All orientations will be compared with the reference orientation. A green color indication for a certain parameter means that the corresponding orientation is scoring better than the reference orientation on this parameter. The red color stands for a worse score compared to the reference.

In this dialog, it is also possible to add more orientations by pressing 'Add orientation...'

When you found an orientation that you like, select the preview and press Ok. When pressed cancel, the part will go back in the orientation before entering the orientation comparator.



Support-related parameters and Max XY section do not take into account the beam lattice component(s) of the part.

4. Shape Sorter

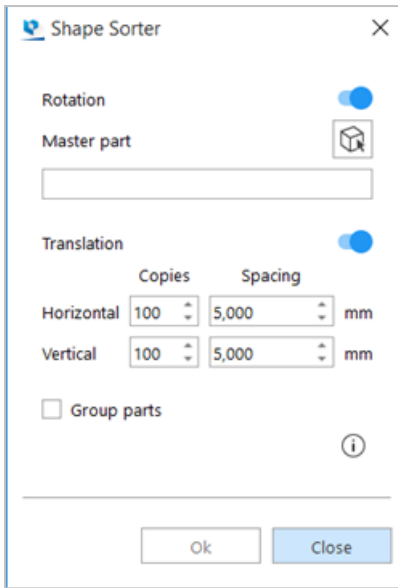



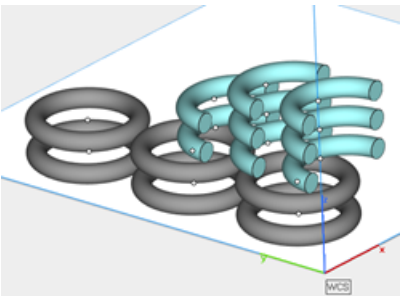
The shape sorter is designed to arrange parts with a similar geometry. The parts can be:

- Rotated only according to the orientation of the Master part
- Translated only, while keeping the orientation of the part
- Both Rotated according to the orientation of the master part and then translated



Remark: To utilize the shape sorter functionality you either must have a “Sintermodule” license or a “Support generation” license.



Indicate	Click the icon to indicate the master part you want to use. The 'master part' mouse mode will become active.	
Horizontal placement	Indicate the number of copies and the spacing between them that will be placed on the horizontal axes	
Vertical placement	Indicate the number of copies and the spacing between them that will be placed vertically	
Copies	Specify the total amount of parts to be placed in a single row or column. Exiting this amount will start a new row based on the master part.	
Spacing	The minimum distance between two parts.	
Group parts	All selected parts will be grouped together as one.	
		
Indication of master part	Result: Sorting based on position of master part on horizontal only	Result: Sorting based on position of master part on horizontal and vertical

Advised way of working

- Import multiple similar parts
- Place the part you want to use as master in the correct orientation

- Select all similar parts
- Assign the master part
- Parts are arranged according to the orientation of the master part

5. 3D nester



More information can be found in the 3D Nester module itself.

- See 1 3D Nester

7.4. Group

Sometimes people want to make sub-assemblies of parts.

With the grouping functionality parts can be placed in groups and these are handled as one part for a number of operations.

1. Group



The group functionality groups selected parts in a group. The parts you can group can be either single parts or already defined groups. Nested groups don't exist.

A group and a part that are grouped result in the part being added to the group.

A group and a group that are grouped result in a new group where the existing ones are combined.

2. Ungroup



The Ungroup function can only be performed on a group of parts.

All part in the group becomes individual parts again.

The ungroup functionality is disabled when you select a part that doesn't belong to a group.

3. Remove From Group

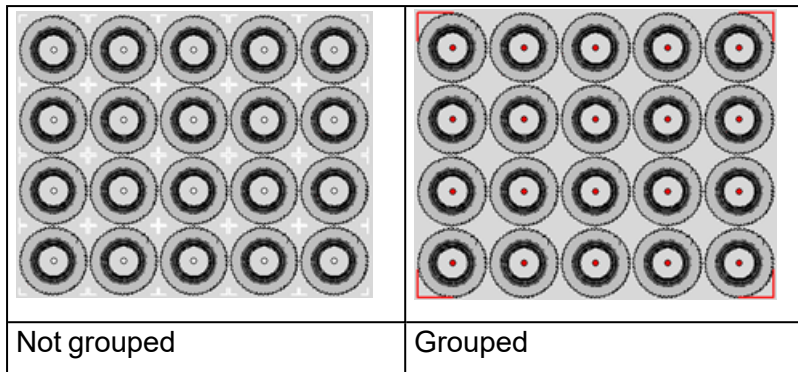


The remove from group will remove selected part from the existing group.

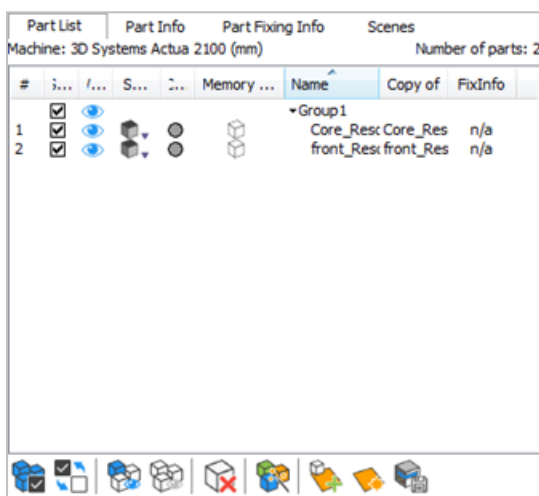
This option is only enabled when 1 or more parts of a group in the parts list are selected.

4. Grouping visualization

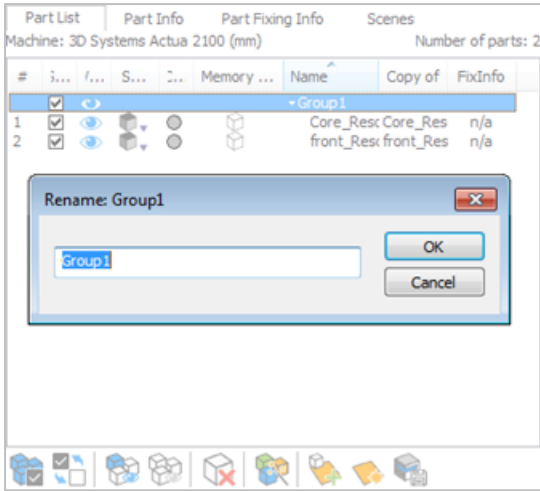
Parts that are part of a group can be recognized by the 'red' center point.



Within the part list the groups are displayed at the end of the list. By default, the naming of the groups start with “Group1”.



Remark: The name of the group can be changed by double clicking the name.

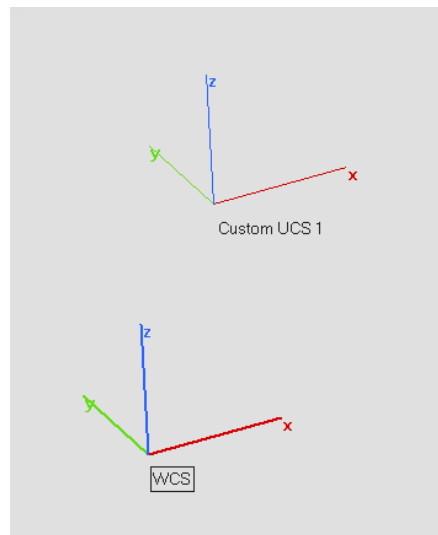
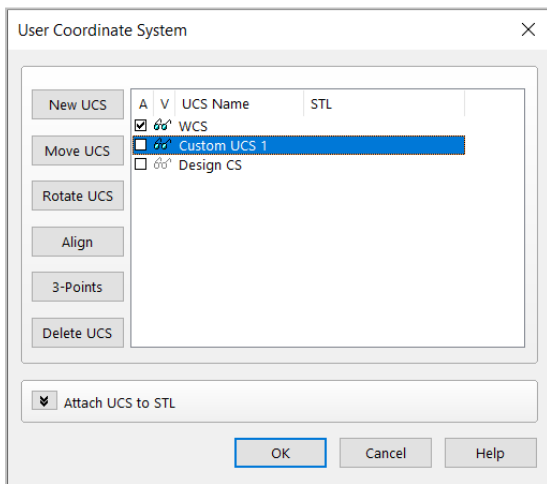


7.5. Coordinate System




1. User Coordinate System




When working in the mesh part scene, you can define custom coordinate system(s) and activate them instead of the World Coordinate system (WCS) to achieve advanced transformed outputs from some operation(s). Each mesh part scene can have its own set of unique set of unique User Coordinate System (UCS). You can save and load the UCS in a Magics project files through the Save Scene As or Save Project As commands.



Column	Description
Active (A)	If you have more coordinate systems, you can make one of them active. Internally, the program works as if the active coordinate system is the only one. Importing, exportin, creating primitive part(s), measuring, cutting, transforming, labeling and other operations are relative to the active coordinate system whenever necessary. Hence, only one coordinate system can be active at any time to ensure the outcome is predictable and

Column	Description
	<p>reproducible always. Thus, you can only see one checkbox indicated in the A (Active) column of the list. In the example above, WCS is active.</p> <div style="border: 1px solid #00aaff; padding: 10px; margin: 10px 0;"> <p> Note: When switching between scenes, the active coordinate system will always be reset to the World Coordinate System of the active scene.</p> </div> <div style="border: 1px solid #00aaff; padding: 10px; margin: 10px 0;"> <p> Remark: If you load a part, it will be loaded in the active coordinate system. If in a previous Magics session, you saved a part at position (10,10,10) in a particular coordinate system it will be loaded at (10,10,10) in the active coordinate system. If the two coordinate systems are different, the part will have a different absolute position in the World Coordinate System, the reference system of Magics.</p> </div>
Visualization (V)	<p>It can be useful to have a few coordinate systems shown on the screen. The second column of the list has glasses for the coordinate systems to show and grayed glasses for coordinate systems to hide.</p> <div style="border: 1px solid #00aaff; padding: 10px; margin: 10px 0;"> <p> Remark: If the Coordinate System indicator toggle button is toggled ON then it is set such that all coordinate system indicators shall be visible. However, this still relies on the individual coordinate system visibility to be toggled on.</p> </div>
UCS Name	The name of the coordinate systems can be changed by double clicking on the name. Name of the coordinate system is also shown below the CS indicator in the viewport.
STL	When an UCS is attached to a part, the name of the part is shown in this column. Only one mesh part can be assigned to each custom CS at once.

Button	Description
New UCS	New UCS is always created based on the currently focused UCS (highlighted in blue). If no UCS is currently focused, then the UCS will be created based on the WCS. Upon the creation, the new UCS will be in the same position as the CS it is based on. To modify the UCS, please use the command to Move, Rotate, Align the focused UCS.
Move UCS	The move command will allow you to specify the displacement of the UCS along X, Y and Z direction of the focused coordinate system.
Rotate UCS	The rotate command will allow you to specify the rotational displacement in degrees you want the UCS to rotate about the X, Y and Z-axis of the focused coordinate system.
Align	The align button will lead you to the align dialog. The align functionality allows you to align axes or planes (XY-plane, etc.) of the UCS to a plane, cylinder, etc.

Button	Description
	.
3-Points	<p>3-points method allows the focused UCS to be realigned based on 3 cartesian points indicated by the user. When the command is launched, the cursor pointer will change into 1, 2, and 3.</p> <ol style="list-style-type: none"> The first point you indicate will be the origin of the selected UCS. The second point will be the reference point for X-axis such that the direction of the X-axis is the vector between point 1 and 2. The Y-axis is drawn through the origin; parallel with a line that is drawn perpendicular on the X-axis through the third point indicated. The third point will be the reference point to determine the direction of Y-axis. The Z-axis is drawn through the origin, perpendicular on the XY plane.
Delete UCS	<p>Deletes the focused coordinate system.</p> <div style="border: 1px solid #00aaff; padding: 5px; margin-top: 10px;"> <p> Note: The WCS cannot be deleted. If you delete the active UCS, the WCS become active.</p> </div>

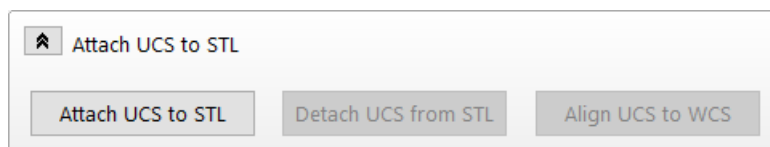


Note: The current active User Coordinate System and the World Coordinate System cannot be modified.

Attach UCS to STL

You can also attach a user coordinate system to a any of the selected mesh part. When the UCS is attached to a mesh part, transforming the mesh part will also result in the coordinate system being transformed appropriately.

If the UCS is attached to the part, exporting the part will result in a custom UCS file to be created as well in the same folder as the exported part file.



Attach UCS to STL	<p>You can attach a UCS to a part. When an UCS is attached to a part it will make the same movement as the part and the relative position of the UCS according to the part will stay fixed.</p> <p>To attach a UCS to a part, select the UCS (mark the row in blue) and select the part. When a UCS is attached to a part, it will be shown in the column STL.</p>
Detach UCS to	You detach an UCS from a part by selecting the UCS in the list, and pushing

STL	the button detaches STL.
Align UCS to WCS	When a UCS is attached to a part, you can make it coincide with the WCS. The attached part will make the same rotation and translation as the UCS. To do so, select an attached UCS and click the button.



Note: Placing the part on platform will not add the custom coordinate system to the platform scene as well since platform scene does not currently support custom coordinate systems.

2. Import UCS File



You can load a User Coordinate System (.ucs) file into mesh part scene to add the custom UCS to the list of coordinate system within the target scene.

7.6. Defaults

1. Translate to Default Z Position



This command will move the selected parts to the default Z position (see **Bottom/Top Plane, page 255**). If more than one part is selected, the entire group will be moved while their mutual position is kept. This option is also executed when pressing the Home button on your keyboard.

2. Translate to Default Position



This command will move the selected parts to the default position (see Bottom/Top Plane, page 1). If more than one part is selected, each part will be individually moved to the default position. The translation origin is fixed on the minimum X, Y, Z points of the part bounding box.

3. Original Position



Move all selected parts in the current scene to their original position.

4. Original in New Scene



Move all selected parts in the current scene to their original position AND open them in a newly chosen scene. Current scene will not be affected.

5. Save Current Position

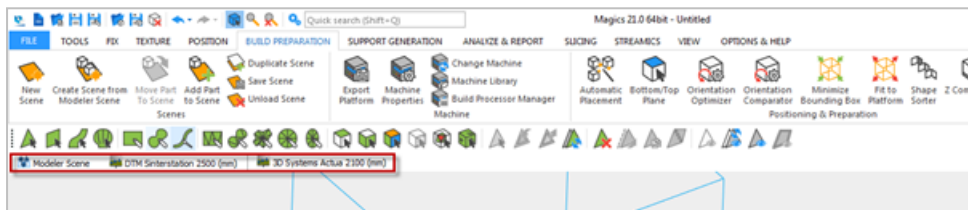


Save the current position as original position. The previous original position will be overwritten.

Chapter 8. Build Preparation

8.1. Scenes

With Scenes, platforms can be created to prepare your build. It gives you a workspace where you can orientate and position your parts in the same way as they should be built by the machine. A lot of parameters can be stored in such a scene. This enables you to create your own, customized platforms. Saving these machine profiles (.mmcf files) will ensure you can work in a fast and standardized way. You can also work on different platforms in parallel. The active scene is the one showed on the screen. You can switch between different scenes by clicking on the name of the platform in the main window (see picture below).



To orientate a part on the platform Indicate top or bottom view can be used. There is also a collision detection tool (see ***Collision Detection, page 455***) to see if the parts are well positioned on the platform. The automatic placement tool (see ***Automatic Placement, page 262***) makes it possible to easily and economically position different parts on the platform. You can calculate the build time and the cost of building some parts.

8.2. Scenes: Virtual Copies

This section will explain what virtual copies are and how you have to handle them. The goal of using virtual copies is to save memory. If you need e.g. 100 copies of a part, there will be a big difference in memory usage if you have to load (and save) 100 parts or only 1 part.

1. A Part and his Virtual Copies

The real parts are collected in a database. From this database you can create virtual copies on platforms. This virtual copy contains a reference to the real part and a translation matrix.

2. Edit a Virtual Copy

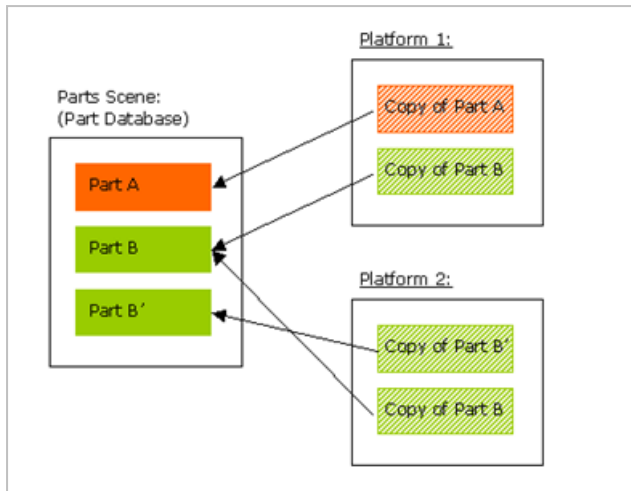
If you edit a real part (in the part scene), all virtual copies of that part, present in the current Magics session, will be edited in the same way.

Example

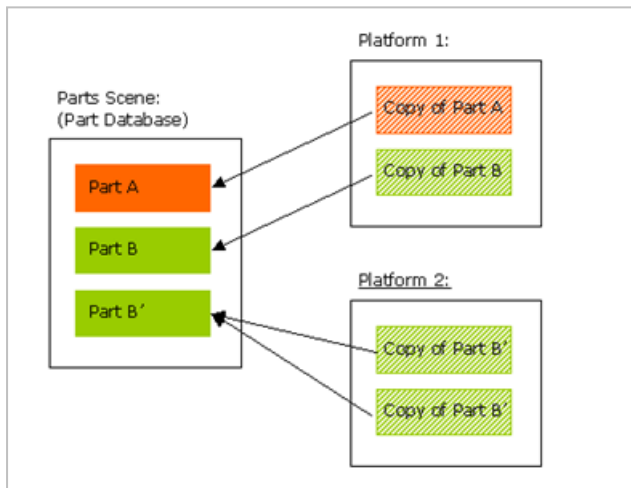
Suppose you select and edit one of the virtual copies of part B on platform 2.

Magics will ask if you want to apply the changes only on the selected copy or on all virtual copies of part B present on platform 2:

If you apply the changes only on the selected copy, this is the result:



A new part is created in the part database. The edited copy will refer to a new real part.
If you apply the changes on all virtual copies of that mother part, this is the result:



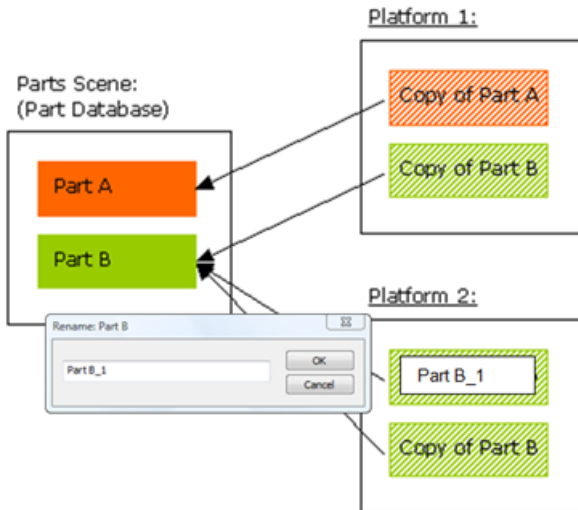
All the virtual copies of the same mother part, present on that platform, will refer to a new database part.

3. Naming of virtual copies

Virtual copies inherit the name of the mother part when it is created.

Within the platform scene however, virtual copies can be assigned with a unique/ different name.

Example



Editing virtual copies is only possible for virtual copies of the same mother part and present on the current platform! This is for safety reasons. If you want to edit all virtual copies of the same mother part, present on all the platforms, you should edit the STL files in the part scene.



Names of virtual copies change when the name of the mother part (in the part scene) is changed. Even if virtual copies are already renamed. The part status however stays the same.

4. Advised ways of working

A predefined scene also be opened by default if Magics opens. Choosing your default machines can be done in the machine library (see ***Edit machine properties, page 317***).

If you use Magics mostly for platform independent operation e.g. fixing STL files or creating RapidFit, we advise you to work in the part scene.

If you use Magics mainly for preparing platforms, you can follow one of the following method of working.

Open Magics with the default Platform Scene or create a new Platform Scene. Make the Platform, where you want to work on, active – by clicking on the platform name in the main window.

Load or create a part. In the background this part will be loaded in the Part Scene (part database) and immediately a Virtual Copy of that part is created on the platform. You can edit and prepare this part in the Platform Scene, like you are used to in previous version of Magics. Each operation you do on the copy in the Platform Scene will automatically be performed on the real part in the Part Scene.

Multiple Copies

If you want to build more than one copy of a part on that Platform, you can duplicate the part (see ***Duplicate, page 258***).

Most advised is to create the copies after you finished the preparation of the files. Otherwise, Magics will ask for each operation if you want to apply it on all copies or only on the selected ones. (If you select all copies, of course, the message won't be prompted).

8.3. Platform scene management

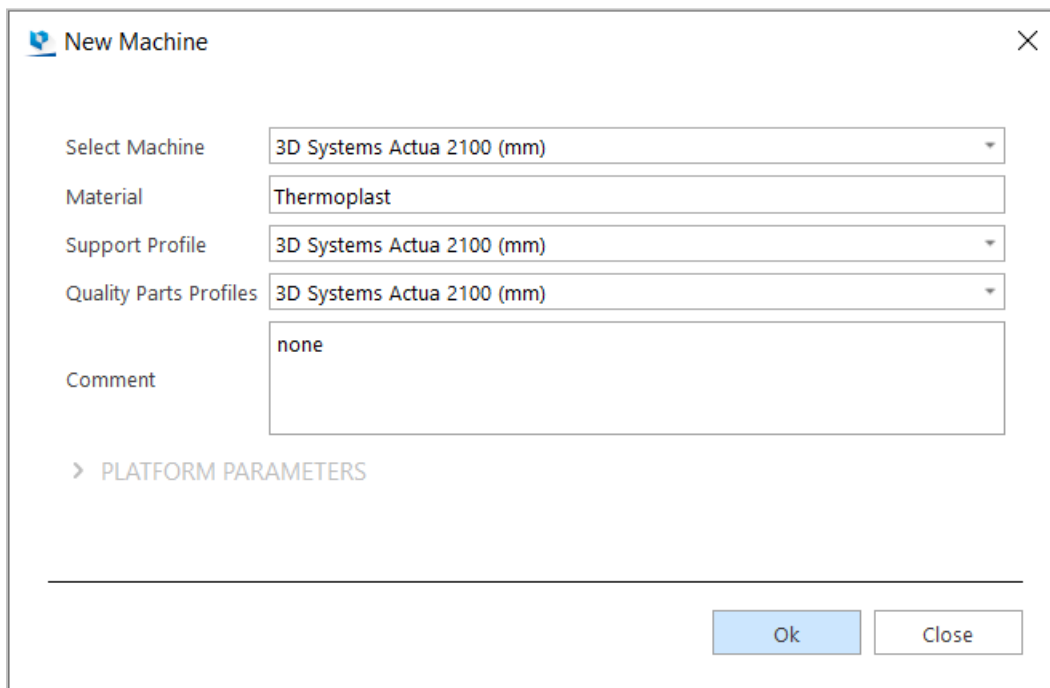
In this section, some common utility commands to create, modify and close platform scenes quickly are described in detail.

1. New Platform Scene

With this command, you can create a new platform scene by specifying the machine for the platform scene. The platform properties will depend on the properties of the machine you select.

It's also possible to select directly from this window a specific support profile previously generated.

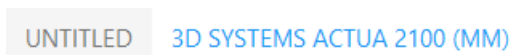
(See **Support Generation, page 346** chapter on how to generate support profiles). The new Platform Scene will be open with selected support properties ready to be applied.



The new scene will appear as the right most tab in the scene tab header.

If you create more than one platform scene for the same machine, the platform name will be the machine name extended with a number, e.g. *'Machine ABC (2)'*. You can organize machines via drag and drop.

Original Situation



Add New Scene

UNTITLED 3D SYSTEMS ACTUA 2100 (MM) (1) 3D SYSTEMS ACTUA 2100 (MM) (2)

2. Add Part to Platform



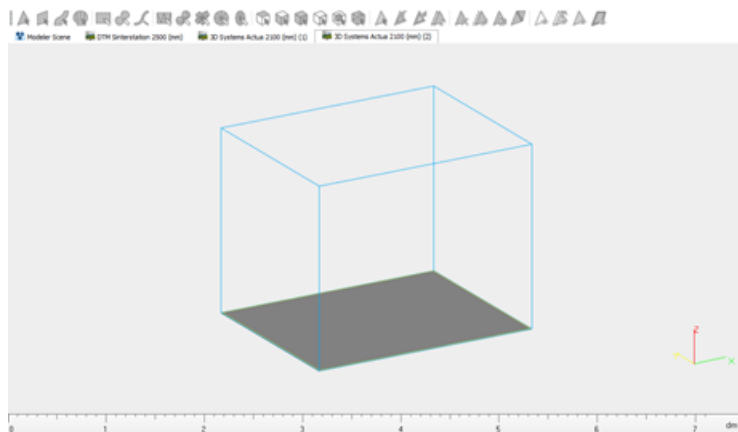
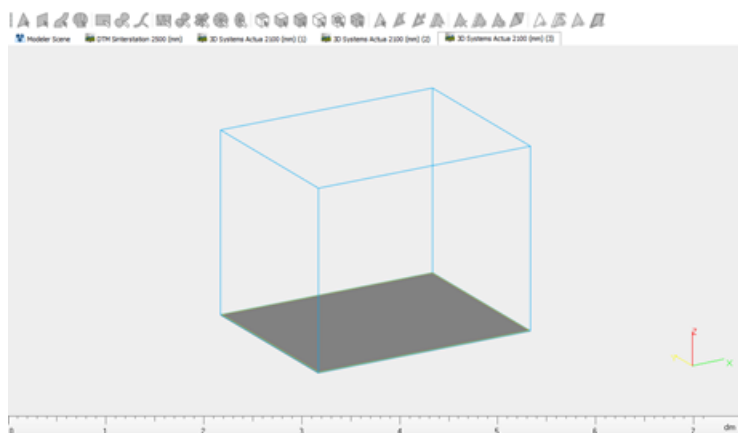
This command is only shown in the Mesh part scene to add the selected parts to a target platform scene. If an existing platform does not exist then the New Machine dialog is shown else you can specify which platform to add to from a pop-up menu.

- See ***New Platform Scene, page 289*** for more information on creating new platforms.

3. Duplicate Platform



This operation creates a Platform Scene, identical to the active Platform Scene. The Platform name will be the machine name extended with a number, e.g. '3D Systems SLA 250 (mm) (2)'.

Original Situation*Copy Scene*

If the platform contains parts, also the parts will be copied on the new platform.



4. Rename Scene



This operation can be found in the right mouse button context menu. It pops up a 'Rename Scene' dialog box in which you can change the name of the selected scene.

5. Save Scene As



This operation pops up a 'Save As' dialog box in which you can change the name and destination of the project you want to save. The scene is saved with all positioned parts, measurements, ...

6. Close Scene



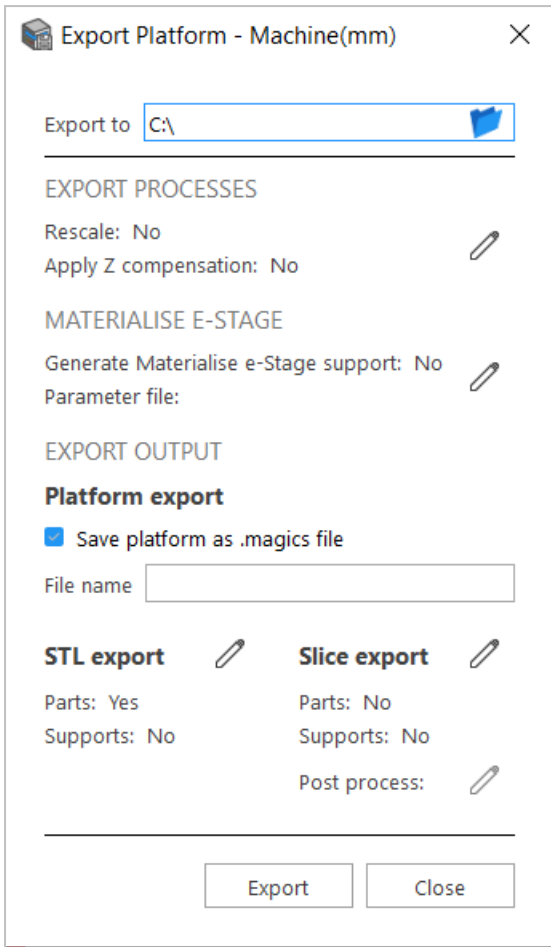
With this operation you can close the target platform scene. If there are parts present on the platform scene, Magics show dialog boxes to save both the scene and parts on it as well before closing the scene.

8.4. Machine


1. Export Platform



Exporting a platform slices the parts and the supports saved in the Magics project. This is the last step in work preparation. The result of the export platform operation is a file that can be sent to the RP-machine.



Export to	You can define the directory for the exported files.
Export processes	An overview is given of the operation(s) which are performed during the export of the platform. The export processes parameters are defined in the Machine Properties – Export processes (see Processes, page 311 for more information).
Materialise e-Stage	Materialise e-Stage is a separate software from Materialise, used to build fully automated and revolutionary support. Here you can define the profile that is used to generate automatic support during the export process.
Export output	Automatically export all the files you need together with the platform file.
	STL export Export STL files for all parts and support present on the active scene. The parameters are defined in the Machine Properties/Export platform page (see Export platform page, page 311 for more information).
	Slice export Export sliced files for all parts and support present on the active scene. The parameters are defined in the Machine Properties/Export slice page (see Export slice page, page 312 for more information).

 Remark: The file name is based on the specified format in the 'Export Platform Working folder' setting.

2. Machine Properties



This function will pop up the Machine Properties dialog of the active Platform. The machine properties contain all the information related to the chosen machine type.

General info page

Machine Properties: DTM Sinterstation 2500 (mm)

General info

- Parts placement
- Default Parts
- Z Compensation
- Build Time Estimation
- Cost estimation
- Support Generation mode
- Support Generation parameters
- Materialise e-Stage support para...
- Export platform
- Export slice
- Post-process slices
- Materialise e-Stage export

Machine name: DTM Sinterstation 2500 (mm)

Material: none

Comments: none

Build envelope

Platform shape: Rectangle

	X	Y	Z	
Size	381,000	330,000	300,000	mm
Rescaled platform	381,000	330,000	300,000	mm
Position	0,000	0,000	0,000	mm

Configuration

Show recoater direction

Axis: X

Direction: Positive

Show gas flow direction

Axis: X

Direction: Positive

Fields overlaps

Enable field overlaps

Color: ■

Axis: X

Position: 0,001 mm

Width: 0,001 mm

Add Delete

Laser parameters

Laser power: 0,020 W

Ask for laser power when estimating

Laser spot diameter: 0,200 mm

Slice visualization

Slice position: Middle

Save changes to: Active platform scene All platform scenes alike Machine library

Apply Ok Close

Machine Name	Each machine type has to have a name. If you are building with different parameters, it is best to have a machine type for each set of parameters (e.g. layer thickness).
---------------------	---



Material name	The material of the selected machine.
Comment	Comment on the selected machine.

– Build envelope

Platform Shape	The platform of the machine can be rectangular or circular.
Size	The X, Y and Z values of the platform size.
Rescaled platform	When you choose to rescale parts during export of the platform, the size of the build envelope is automatically adjusted in the opposite direction.
Position	The minimum X, Y and Z values of the platform position.

– Configuration

Show recoater direction	Enable the visualization of the recoater direction on the platform. This feature can be used for all machines presenting a recoater.	
	Axis	Specify along which axis the recoater moves.
	Direction	Specify if the recoater moves from left to right, from right to left, or if it is a double-side recoater.
Show gas flow direction	Enable the visualization of the gas flow direction on the platform. This feature can be used for laser melting (LM) technology.	
	Axis	Specify along which axis the gas blows.
	Direction	Specify if the gas blows from left to right or from right to left of the specified axis.

– Fields overlaps

Enable field overlaps	Activate and visualize the field overlaps on the platform without deleting the configured overlaps.
Overlaps list	Laser overlaps can be defined. Click on Add button to add an overlap that will be displayed on the platform, or select an existing overlap to edit it or delete it with the Delete button.
Color	The color used to visualize the overlap.
Axis	The axis of the overlap (X or Y axis of the platform).



Position	The position of the overlap.
Width	The width of the overlap.

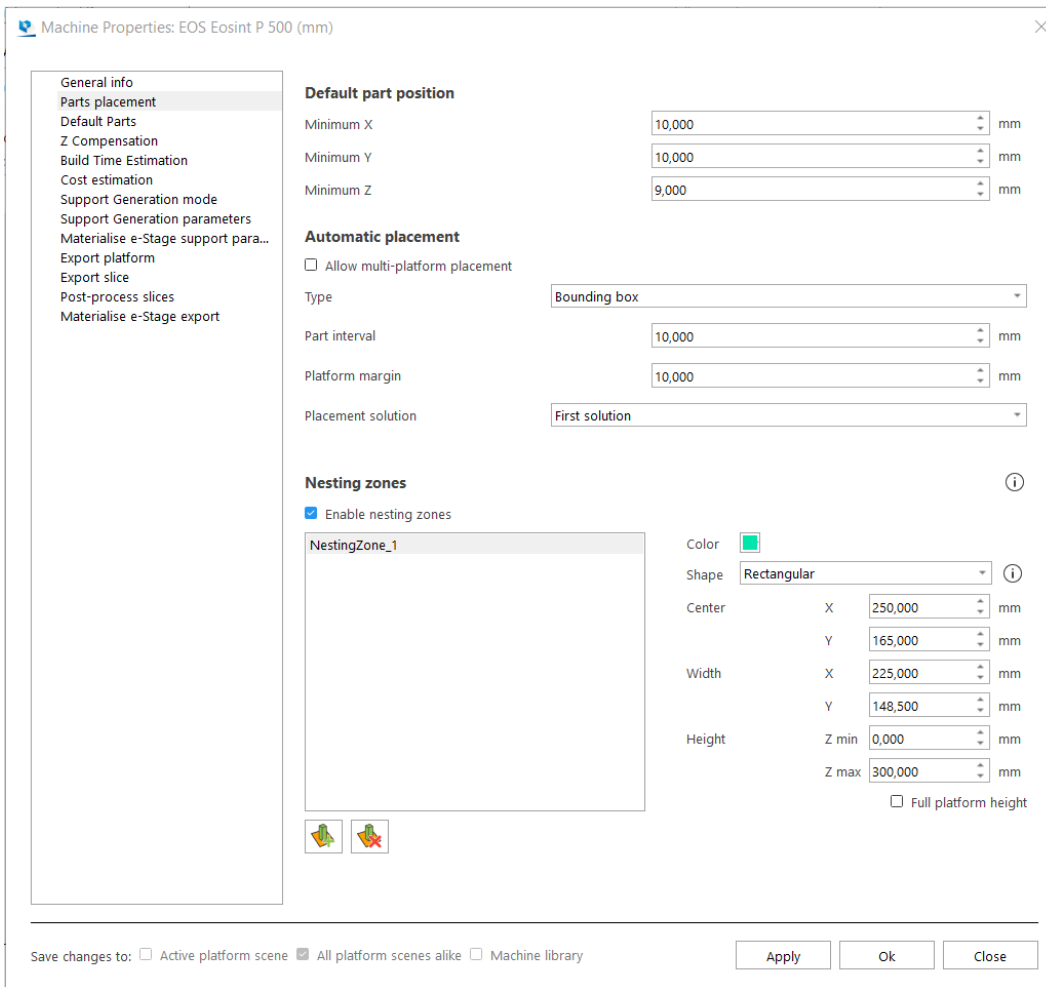
– Laser parameters

Laser power	Intensity of the laser (W).
Ask for laser power when estimating	When this checkbox is selected, a specific value of laser power can be input at the moment of running the build time estimation.
Laser spot diameter	The diameter of the laser spot.

– Slice visualization

Slice position	Select on which position the slices represent the part: top, middle or bottom of the slice.
----------------	---

Parts placement page



– Default part position



Minimum X, Y and Z	Coordinates that a part has by default on the platform.
--------------------	---

– Automatic placement

Allow multi-platform placement	When this option is selected and the parts do not fit on the current platform scene, Magics will create as many platform as needed to place all parts.
--------------------------------	--



Type: Bounding box	Part interval	The interval between parts.
	Platform margin	The margin of the platform.
	Placement	The placement solution that will be applied: <ul style="list-style-type: none">- First solution- Minimal XY area- Minimal X area- Minimal Y area- Platform center
	Range by part Z-height	When this option is selected, parts will be placed accordingly to their Z height; it is possible to start from the part with highest value or lowest value of Z height. <ul style="list-style-type: none">- Start with min Z-height first- Start with max Z-height first



Type: Geometry	Part interval	The interval between parts.
	Platform margin	The margin of the platform.
	Placement	The placement solution that will be applied: <ul style="list-style-type: none"> – First solution – Minimal XY Area – Minimal X area – Minimal Y area – Platform center – Custom solution*
	Allow part Z rotation	When checkbox is deselected, parts are only translated; when checkbox is selected, parts are translated and rotated of the specified angle during automatic placements.
	Range by part Z-height	When this option is selected, parts will be placed accordingly to their Z height; it is possible to start from the part with highest value or lowest value of Z height.
	*Custom options solution	 <p>Load an image that can be shown on the platform: the loaded image will be first converted to grayscale and then scaled to fill the platform.</p> <p>The darker areas of the image indicate higher priority, while the lighter areas indicate lower priority.</p> <p>Export the Platform image to get an image with the correct platform dimensions, that you can immediately edit.</p> <div style="border: 1px solid blue; padding: 10px; margin-top: 10px;"> <p> Note: lower accuracy will be faster, but the image used for the positioning of parts will be less detailed. Higher accuracy will give a more detailed image, but might be slower.</p> </div>

- Nesting zones

Define custom zones into which parts can be placed by 3D Nester.

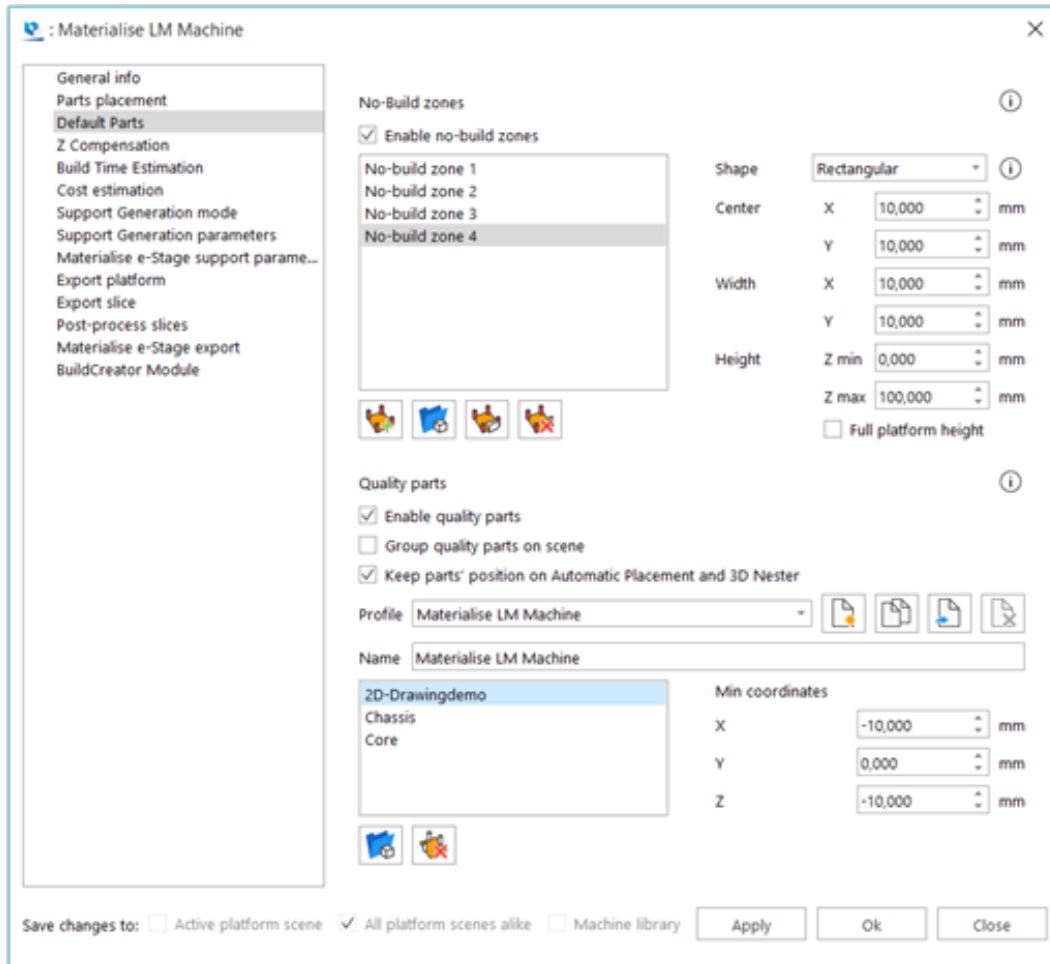
- Multiple zones can be created;

- Reorder zones by drag & drop on the list. During nesting parts will be placed into zones in the order of priority from top to bottom as they appear in the Machine properties.
- Zones can overlap. When zones overlap, a greater nesting density can be achieved, and the space can be used more efficiently.
- Use zones to assign parts to the regions of the platform with greatest print quality, for early breakout (top of the build), for strategic placement of parts with certain shape or size etc.

Enable nesting zones	Activate and visualize the nesting zones on the platform.		
 Add nesting zone	Create a new nesting zone with default shapes (cylindrical or rectangular).		
 Delete nesting zone	Delete the selected nesting zone from the list.		
Shape: Cylindrical	Center	X, Y	The X and Y coordinates are used to define the center point of the cylinder section.
	Radius	Define the radius of the cylindrical nesting zone.	
	Height	Define the height of the cylindrical nesting zone.	
		Z Min	The starting height of the nesting zone.
	Z Max	The ending height of the nesting zone.	
Full Z height	The nesting zone is active for the full height of the build envelope. 'Height' will be disabled when choosing 'Full platform height'.		
Shape: Rectangular	Center	X, Y	The X and Y coordinates are used to define the center point of the rectangle section.
	Width	X, Y	Define the size of the rectangular nesting zone.
	Full X / Y width	X, Y	Automatically make the width of the zone equal to the width of the platform in X or Y direction.
	Height	Define the height of the rectangular nesting zone.	
		Z Min	The starting height of the nesting zone.
Z Max	The ending height of the nesting zone.		

	Full Z height	The nesting zone is active for the full height of the platform build envelope. 'Height' will be disabled when choosing 'full platform height'.
--	---------------	--


Defaults parts page






– No-Build zones

With the No-Build zone functionality you can indicate areas in which you don't want to build parts. These areas will be taken into account during nesting or automatic placement operations.

This feature can be useful in different situations. (e.g. metal machines, multiple lasers, ...)



Enable no-build zones	Activate and visualize the no-build zones on the platform.
 Add zone	Create a new no-build zone with default shapes (cylindrical or rectangular).

 Import part as zone	Import STL files to have customized shapes of no-build zones. More complex shapes can be used.		
	Min coordinates	The X, Y and Z coordinates are used to define the position of the STL on the platform. The minimum point of the part bounding box (for all axes) will be translated to the specified coordinates.	
 Load zones from another machine	Load no-build zones from another machine with defined no-build zones.		
 Delete zone	Delete the selected no-build zone from the list.		
Shape: Cylindrical	Center	X, Y	The X and Y coordinates are used to define the center point of the cylinder section.
	Radius	Define the radius of the cylindrical no-build zone.	
	Height	Define the height of the cylindrical no-build zone.	
		Z Min	The starting height of the no-build zone.
	Z Max	The ending height of the no-build zone.	
Full platform height	The no-build zone is active for the full height of the build envelope. 'Height' will be disabled when choosing 'Full platform height'.		
Shape: Rectangular	Center	X, Y	The X and Y coordinates are used to define the center point of the rectangle section.
	Width	X, Y	Define the size of the rectangular no-build zone.
	Height	Define the height of the rectangular no-build zone.	
		Z Min	The starting height of the no-build zone.
		Z Max	The ending height of the no-build zone.
Full platform height	The no build zone is active for the full height of the platform build envelope. 'Height' will be disabled when choosing 'full platform height'.		

- Quality parts

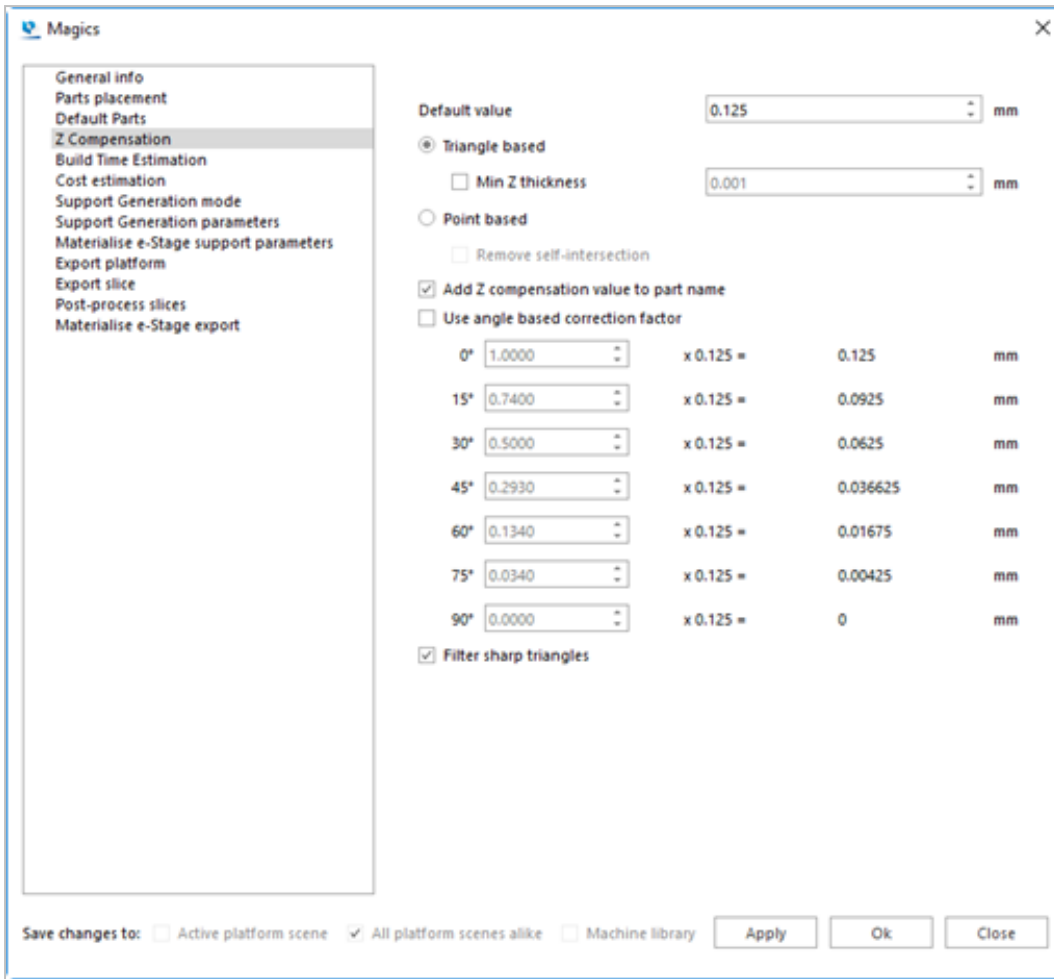


In the Quality parts tab you are able to define models that will be loaded automatically when opening a new scene. The position of these models can be specified.

Enable quality parts	Activate and visualize the quality parts on the platform. These parts are also added to the Part list.	
Group quality parts on scene	All quality parts added will be grouped in one group.	
Keep parts position on Automatic Placement and 3D Nester	Quality parts will remain in their position defined in Machine properties when running Automatic Placement or 3D Nester.	
Profile	Select and manage the profiles of quality parts.	
	Create new profile	Click this button to create a new profile.
	Duplicate profile	Click this button to create a copy of the current profile.
	Load profile from another machine	Click this button to load an existing profile from another machine.
	Delete profile	Click this button to delete the current profile.
	Name	Type the name of the profile.
 Import part	Create a quality part by importing an *.stl or a *.matpart file. When using *.matpart files, you are able to retain label planning information and support structures that are saved in the part loaded.	
	Min coordinates	The X, Y and Z coordinates are used to define the position of the quality part on the platform. The minimum point of the part bounding box (for all axes) will be translated to the specified coordinates.
 Delete part	Delete the selected quality part from the list.	

Z-Compensation page


For models built with Stereolithography and Laser Sintering, overcure may cause extra material to build up on down-facing surfaces. To avoid the time-consuming process of manually correcting these errors after the part is finished, the Z-Compensation function can be used.



Default Value	The value that is displayed by default when the user applies a Z-compensation on a part located on the respective platform.
Advanced Options	When you prefer to perform Z-compensation during export, you can choose the Z-compensation algorithm. (see Z-Compensation, page 319)

Build time estimation page

There are 2 methods available for the build time estimation: the first one is focused on Stereolithography technology, while the second one uses a self-learning algorithm that is based on teaching platform data that you can add over time.

 Remark: The laser parameters defined in the **General info page, page 293** will be taken into account during the build time estimation.

1. Stereolithography method

The screenshot shows the 'Magics' software window with the 'Stereolithography' method selected. The interface is divided into several sections for parameter configuration:

- Method:** Stereolithography
- Part parameters:**
 - Layer thickness: 0.250 mm
 - Border overcure: 0.180 mm
 - Hatch overcure: -0.020 mm
 - Hatch distance: 0.080 mm
 - Recoat time: 35.00 sec
- Support parameters:**
 - Layer thickness: 0.250 mm
 - Border overcure: 0.030 mm
 - Hatch overcure: -0.020 mm
 - Line spacing: 4.000 mm
 - Recoat time: 25.00 sec
- Resin parameters:**
 - Critical exposure (Ec): 10.00 mJ/cm²
 - Penetration depth (Dp): 6.50 mm

At the bottom, there are checkboxes for 'Save changes to:' with options 'Active platform scene', 'All platform scenes alike' (checked), and 'Machine library'. Buttons for 'Apply', 'Ok', and 'Close' are also present.

– Part parameters

Layer thickness	The thickness of the layers.
Border overcure	The overcure of the border.
Hatch overcure	The overcure of the hatching.
Hatch Distance	The interval distance of the hatchings.
Recoat Time	The time needed to put a new layer of resin/powder on top of the already made construction.

– Support parameters

Layer thickness	The thickness of the layers.
Border overcure	The overcure of the border.
Hatch overcure	The overcure of the hatching.
Hatch Distance	The interval distance of the hatchings.
Recoat Time	The time needed to put a new layer of resin/powder on top of the already made construction.

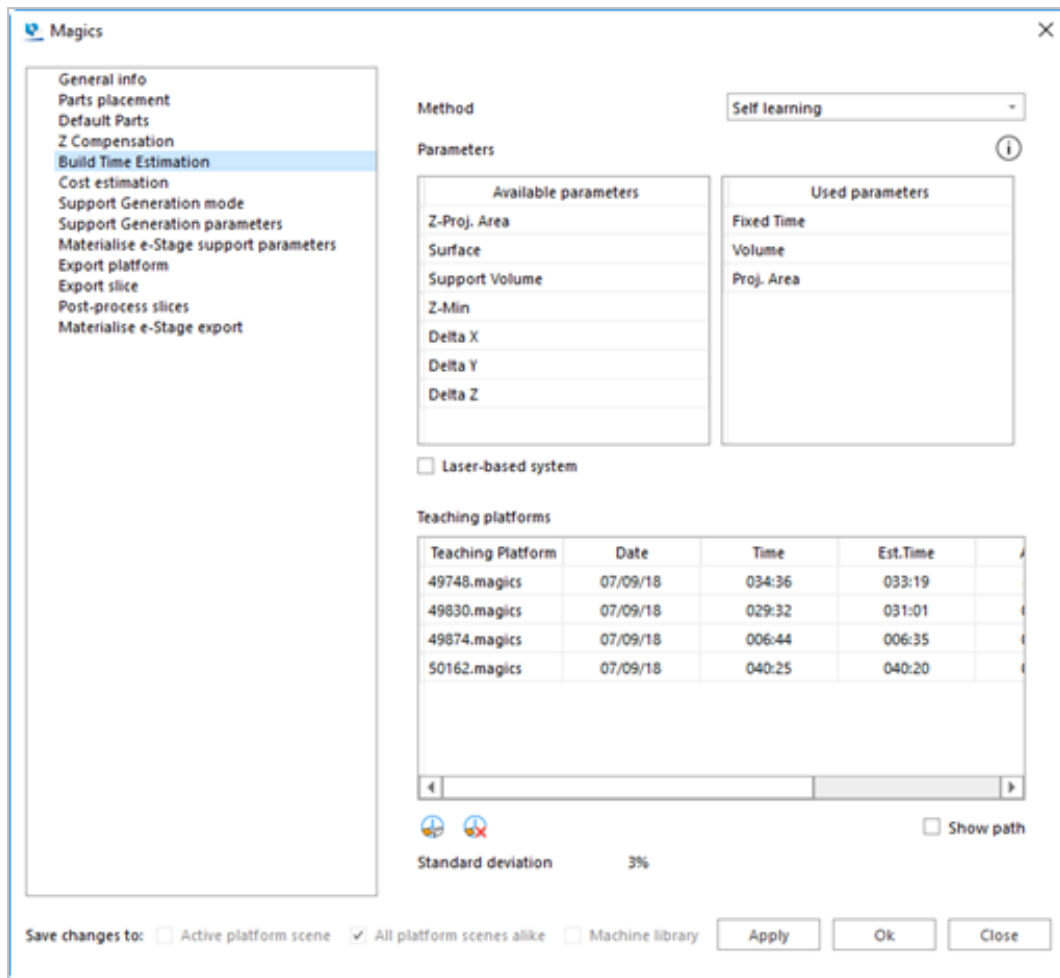
– Resin parameters

Critical exposure (E_c)	The critical dose of energy.
Penetration depth (D_p)	The curing depth of the laser.

2. Self-learning method

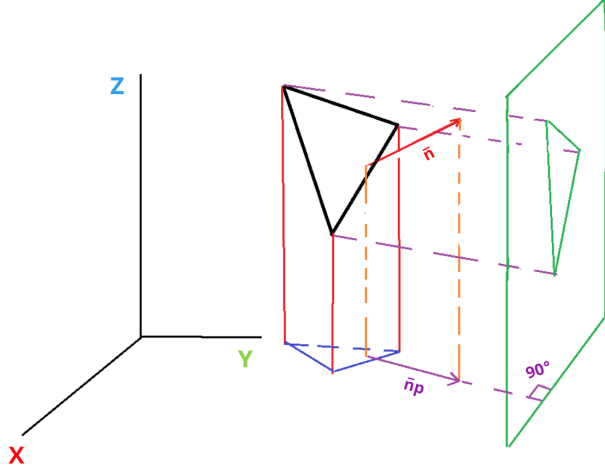
The self-learning method requires a list of teaching platform files. These platforms should be actually built on the machine, thus providing real information of printed builds. Putting together such a list will allow Magics to accurately estimate the build time of new platforms.

The platform and the appropriate parameters used to calculate the build time must be selected.



– Parameters

Fixed time	Often there is a fixed time in play, e.g. to heat up the build platform.
Volume	The combined volume of all parts on the build. E.g. for laser based machines, this is a crucial parameter as your volume has a crucial effect on how long it prints.
Proj. Area	The combined projected area of all parts on the platform. E.g. for FDM this is often related to how much support is created, for SLA it might have an effect on the recoater time.
Surface	The combined surface area of all parts on the build. E.g. for laser based machines, the surface is often lasered separately, increasing the build time based on the surface area.
Support volume	The combined volume of all supports on the build.
Z-min	The minimal Z height of the lowest part. E.g. this would take the layers into account where there is nothing or only supports
Delta X	The difference in X of the maximum of the closest part and minimum of the closest part.

Delta Y	The difference in Y of the maximum of the closest part and minimum of the closest part.
Delta Z	The height difference of the parts. This parameter takes layers into account and how much impact they have on the time.
Z-Proj. Area	<p>Total area of all triangle's projection on specific vertical surfaces. Normal of this surface is the projection of triangle's normal on XY-plane, or in other words, the vertical surface is perpendicular to the surface formed with triangle's normal and z-axis.</p> 

The two parameters lists show you the parameters that can be used for build time calculation of a platform. The list on the left contains all the parameters that may be included in the build time calculation, while the list on the right contains the parameters that will be included in the calculation. You can drag and drop parameters from one list to the other.

When the **Laser-based system** checkbox is selected, one extra option will appear on the Used parameters list; you can decide if the specific parameter depends or not on the laser power by flagging it in the Laser-based column.

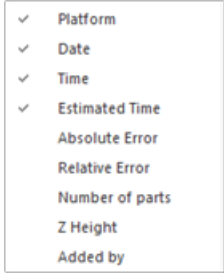


Note: You should have at least as much teaching platforms as there are parameters.

– Teaching platforms

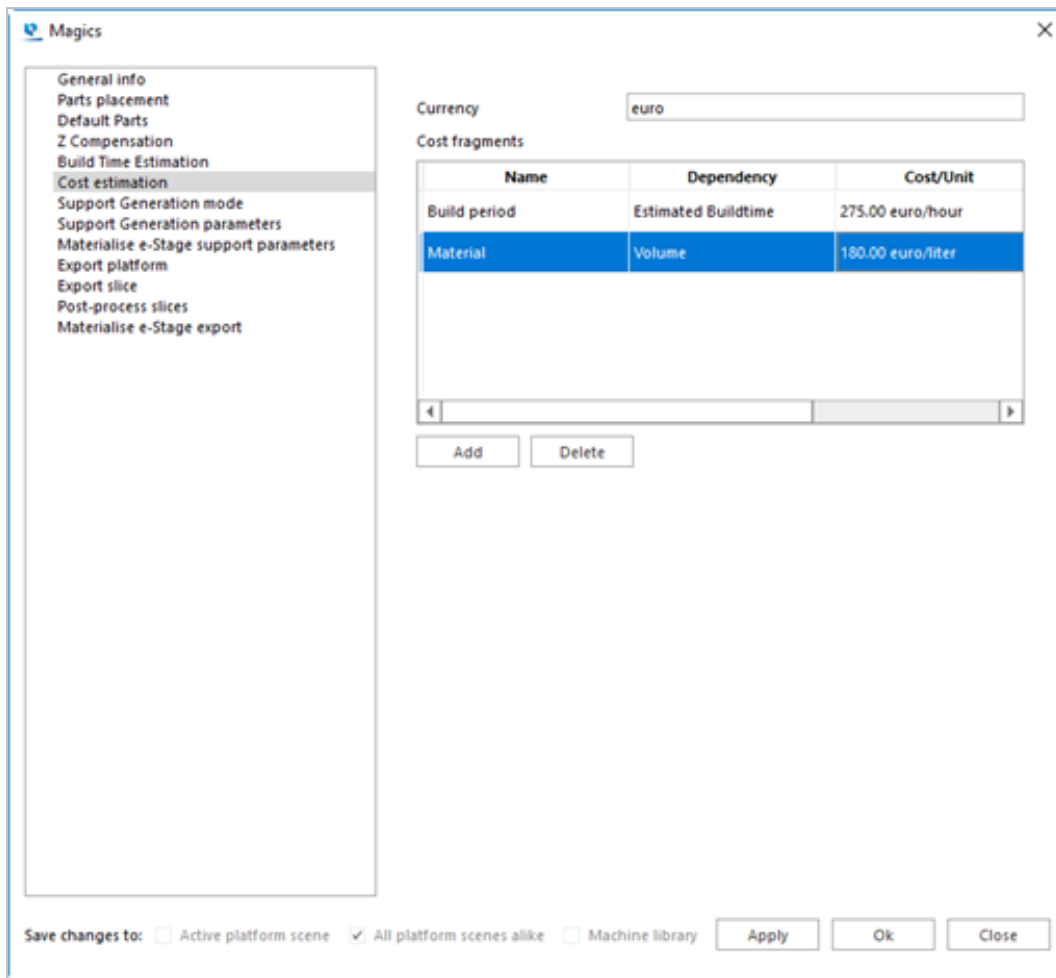
Here you can visualize and manage the data of your teaching platforms used for the build time estimation calculations.

Platform	The file name of the Magics project used as a teaching platform.
Date	The value corresponds to the date when the teaching platform has been imported.
Time	Insert the real build time of the teaching platform (hours and minutes). Double click on the field to edit the value.

Estimated time	The value corresponds to the estimated build time of the teaching platform.
Absolute and Relative errors	The absolute and relative errors give the statistical errors between the estimated and the actual build time of the teaching platform. Absolute error is difference in hours and minutes, while the relative error is expresses as a percentage of the real build time.
Extend list with additional data	<p>You have the possibility to add or remove additional columns; right mouse click on the table headers to display a list of additional columns.</p> 
Import platform	Click on this icon to import a Magics project as a teaching platform.
Delete platform	After selecting one or more teaching platforms from the list, click on this icon to delete the selected teaching platform(s) from the list.
Show path	Select this option to display in the Platform column the file path instead of the Magics project file name.
Standard deviation	This value is related to all the relative errors of the teaching platforms added to the table; it gives an idea of the average deviation of the value resulting from the build time estimation compared to the real build time.

Cost estimation page

Cost estimation is a primary need of a company. The cost estimation is done in the tools menu and is done based on parameters which are machine dependent and thus are defined in the Machine Properties.



The Currency can be entered at the top of the page. A cost can be added, edited or removed. To edit a cost fragment, just double-click on the desired field in the table.

- You can give the cost a name (appears in the column Name)
- The dependency determines the variable of the cost. It can be one of the following list. In the first column, you find the dependency, in the second column you find the unit in which this parameter is expressed.

Dependency	Unit
Estimated Build Time	Hour
Fixed Cost	Platform
Volume	Liter
Support Volume	Liter
Surface	cm ²
Delta X	mm
Delta Y	mm
Delta Z	mm

Dependency	Unit
Number of STL Files	Part
Bounding Box Volume	Liter

- Define the cost per unit for each fragment



Remark: When you choose Estimated Build Time, the build time has to be calculated as well and thus all the parameters for these calculations have to be correct. Magics will also ask you for the laser power.

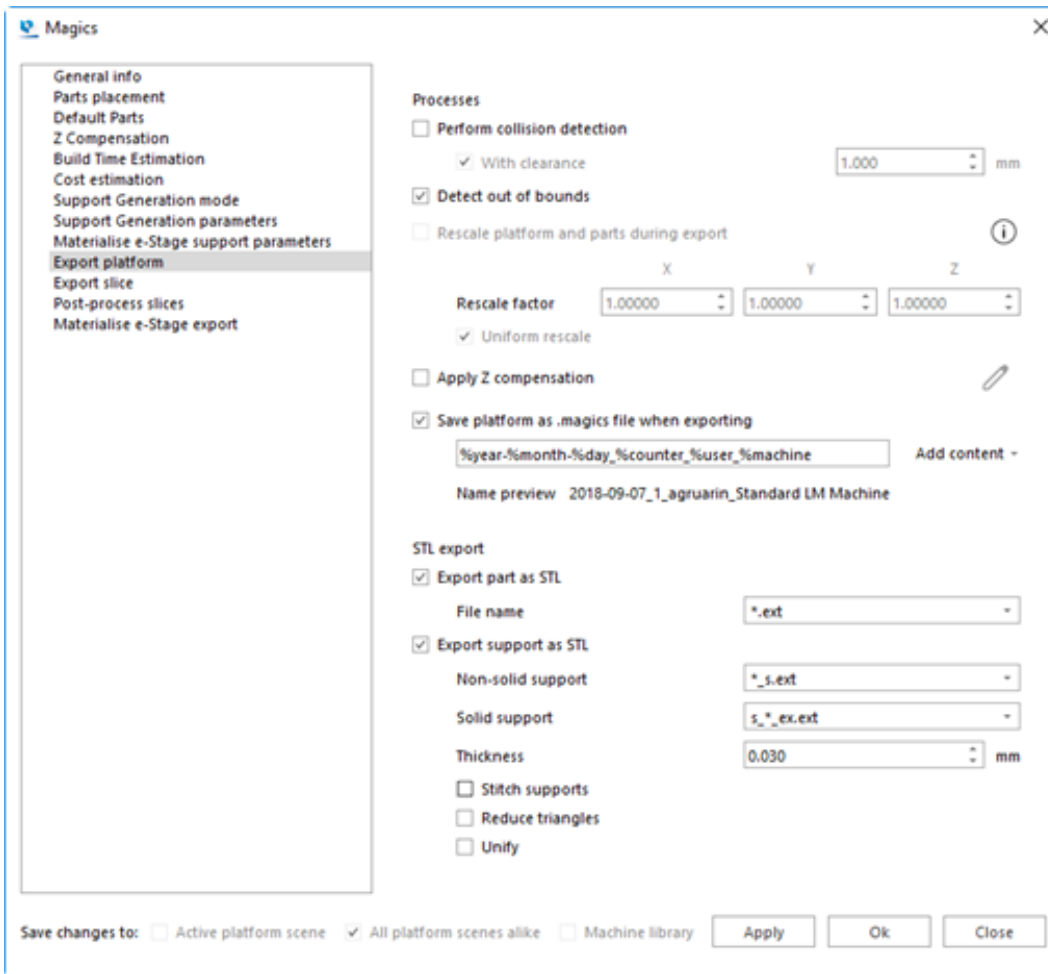
Support Generation mode

- See 3 Support Generation Mode

Support Generation parameters

- See 4 Support generation parameters .


Export platform page



– Processes

Perform collision detection	During export colliding parts can be detected.	
	With clearance	Use a defined clearance to detect collision between different parts.
Detect out of bounds	When selected, a check is performed to see that all parts within the platform scene are included within the platform bounds.	
Rescale platform and parts during export	When exporting, platform and parts can be rescaled by a given factor. (See Rescale , page 246 for more information)	
Apply Z compensation	You can apply Z compensation when exporting.	
Save Platform as .magics file when exporting	When exporting the platform is saved as a magics file.	
	Define the naming template by adding the predefined tags; you can see a preview of the resulting name for the current platform.	

- STL export

Export part as STL	When selected, the part is exported as STL.	
	Filename	Choose a default name convention.  Note: when saving the part, the symbol * is replaced by the part name
Export support as STL	When selected, the support is exported as STL. This also includes e-Stage supports.	
	Non-solid supports	Choose a default name convention for non-solid supports
	Solid supports	Choose a default name convention for solid supports
	Thickness	Define the thickness of your support.
	Stitch supports	A stitch operation is performed on the support before exporting, in order to reduce the bad edges.
	Reduce triangles	A triangle reduction is performed before exporting, in order to reduce the amount of triangles
	Unify	A unify operation is performed before exporting.

Export slice page

More information can be found under the 'Slicing' module.

- See **Machine Setup, page 480**

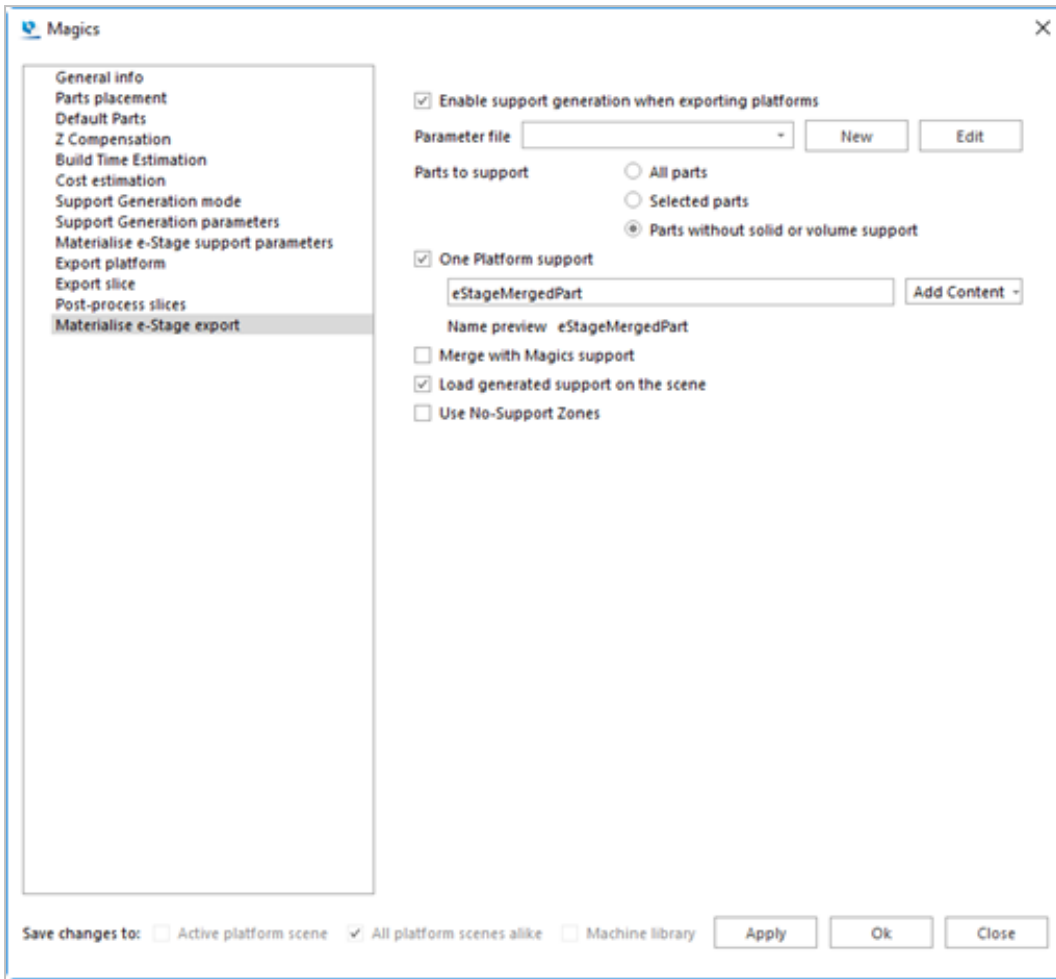
Post-process slices page

More information can be found under the 'Slicing' module.

- See **Post-process slices page, page 482**

Materialise e-Stage export page

Materialise e-Stage is a separate program used to build fully automated and revolutionary support. In Magics it is possible to activate Materialise e-Stage as an export process. You can find it therefore as an option in the machine properties. Each machine can have its own Materialise e-Stage parameter set. More information of the parameters, you can find in the manual of Materialise e-Stage itself.



<p>Enable support generation when exporting platforms</p>	<p>When selected, it activates Materialise e-Stage as an export process.</p>	
<p>Parameter file</p>	<p>A dropdown menu gives you the possibility to choose an e-Stage *.par file from the e-Stage parameter folder defined in the Magics Settings.</p>	
	<p>New</p>	<p>The e-Stage parameter window opens with the default e-Stage parameters. By default the parameter editor will propose to store the file in the e-Stage parameter folder. You can save it somewhere else but in that case you will not be able to select it in Magics.</p>
	<p>Edit</p>	<p>The selected parameter file opens in the e-Stage parameter window.</p>
<p>Parts to support</p>	<p>Define which parts of the platform will be supported by e-Stage support.</p> <ul style="list-style-type: none"> - All parts - Selected parts 	

	– Parts without solid or volume support	
One platform support	One platform is exported including all the supports.	
	Naming template	Add the content which has to be displayed in the project name.
	Preview	A preview is shown of the naming for .magics files that are created via the export platform functionality
Merge with Magics support	SG supports are merged together into one with e-Stage if SG supports are exported as STL.	
Load generated support on the scene	The generated support is loaded into Magics as an STL part.	
Use No-Support Zones	When this checkbox is selected, areas that are indicated as No-Support Zones will not be supported by e-Stage support.	



If you want to open a .par file that is older than your e-Stage version, a dialog box will pop up. When you want to update your e-Stage par file, the e-Stage parameter window will open with the parameters found in the file and the absent parameters with the default values from e-Stage.



If you want to open an e-Stage .par file that is newer than your e-Stage version, a dialog box will pop up. A message is shown that your e-Stage version cannot load this e-Stage par file.

Save changes in

Save changes in:

Active platform scene All platform scenes alike Machine library

Active Scene	Platform	Changes made in the machine properties are only saved in the active platform scene
All Platform Scenes Alike		Changes made in the machine properties will be saved for all active platform scenes.
Machine Library		Changes made in the machine properties are saved in the machine library.



Remark: To have a clear overview of all used parameters per machine a report can be generated. (File > Generate Report > 'Select the template corresponding with the used version of office' >Machine properties report)

3. Change Machine



If you want to change the machine of the active Platform Scene, you can do this with this operation. The Select Machine dialog will pop up and you can choose another machine and another support profile for this machine.

4. My Machines



The My Machines command is meant to setup Magics for the user's RP-machine. As a user you can choose to use one of the available machines or specify your own machine.

The machine selection consists out of two lists:

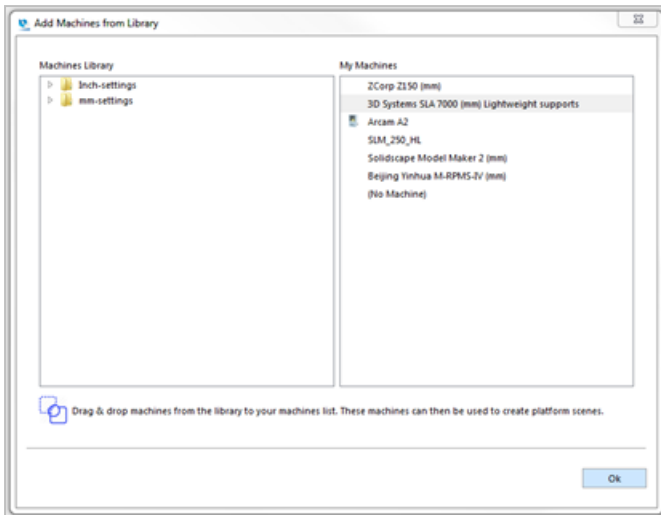
- My Machines: The machines that are used by the user, they are stored in a designated folder. This folder allows adding, copying, deleting and editing the parameters of the machines.
- Machine Library: This is a list of all machines installed in Magics. To edit the parameters of these machines, the user needs to first copy them to the folder "My machines".

The "My Machines" – list is empty in the beginning, and the user has to define the desired machines to work with. The user must add machines from the "Machine Library". Once added in the "My Machines" – list, the user can:

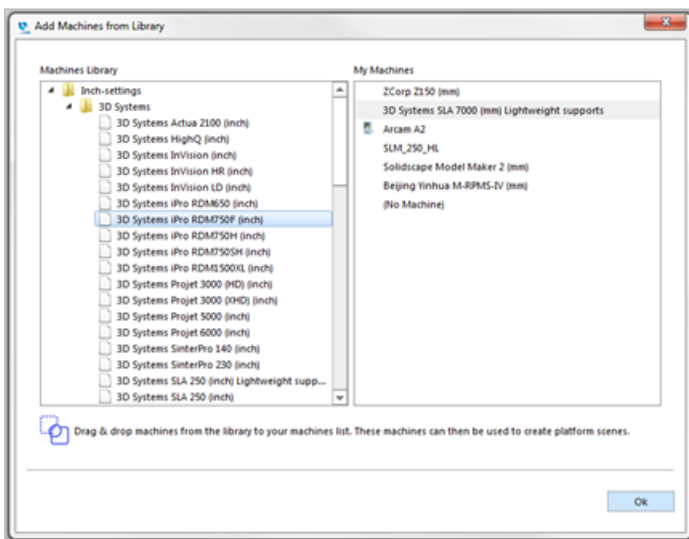
- Change the parameters (includes renaming) of the machines
- Copy the machines
- Delete machines

Advised way of working

1) The first time you want to use the Machine Library, the list is empty. The user must add machine files from the "Machine Library" to the "My Machines" via Add from Library. The following dialog pops up:

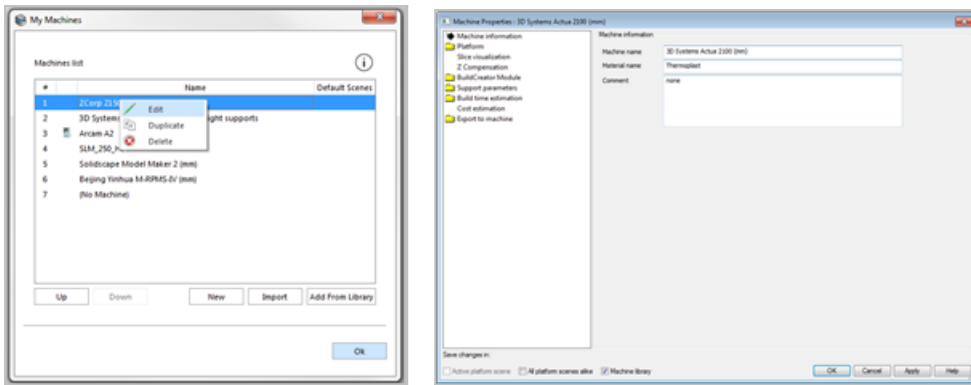


2) The user selects the required machines by browsing the predefined list in the “Machine Library”.

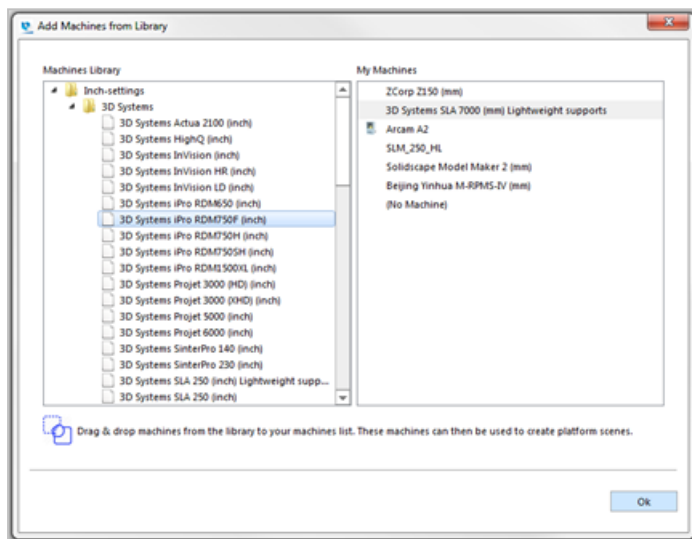


3) The machines are copied to the “My Machines” list by drag & drop.

4) Now, the user can perform the desired modifications on the machines present in the list by clicking the right mouse button on the desired machine and selecting ‘Edit’.



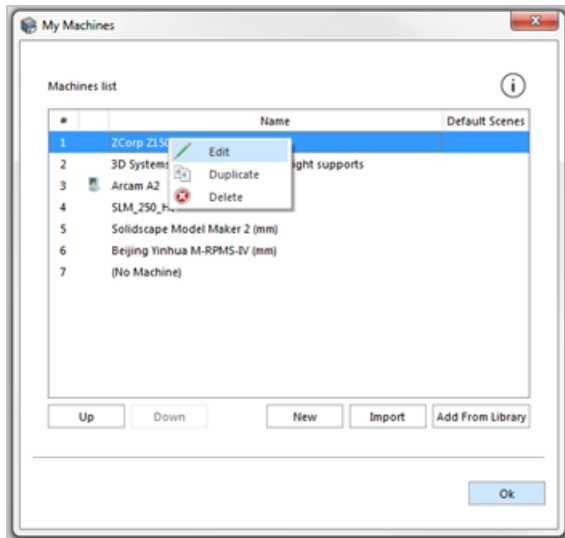
1. Add machines from library



The Machine library is structured in folders:

- Mm-files
- Brands
- List of all machines made by the manufacturer
- Inch-files
- Brands
- List of all machines made by the manufacturer

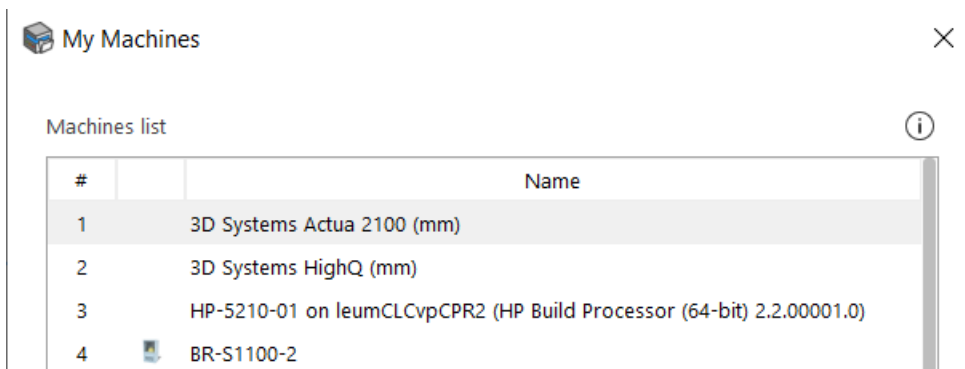
2. Edit machine properties



By clicking the right mouse button on a specific machine, the user accesses a contextual menu where he can:

- Edit the parameters of the machine: the Machine Properties dialog will be shown, where user is able to define the machine properties.
- Duplicate the machine: the selected machine will be copied to the “My Machines” list with "Copy" added to the name of the duplicate machine. The Duplicate option enables user to create new machines based on a selected one.
- Delete the machine: the selected machine can be removed from the “My Machines” list. Confirmation about this action is asked.

The top side of the dialog contains the “My Machines” list, while at the bottom user can find controllers for the table.



<p># (Priority)</p>	<p>This value defines the position of the machine in the list. The order defined here will be used when displaying the machines list in other Magics functions (e.g. see Change Machine, page 315)</p> <p>Double click the left mouse button to change value, or simply drag and drop a machine in the desired position.</p> <p>When a machine icon is displayed, that machine is a Build Processor machine.</p>
-------------------------	---

Name	It displays the name of the machine. To change it, double click the left mouse button on the machine name and simply type the new name.
------	---



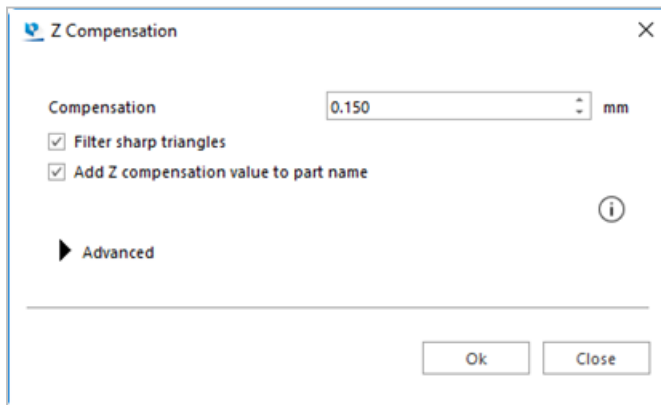
Up	Move the selected machine one position up in the priority order.
Down	Move the selected machine one position down in the priority order.
New	Create a completely new machine based on default parameters.
Import	It allows the user to import machine files. The opened dialog box filters on *.mmcf files.
Add from Library	It opens up the Add Machines From Library dialog, so the user can add new machines to the "My Machines" list.

8.5. Positioning & Editing

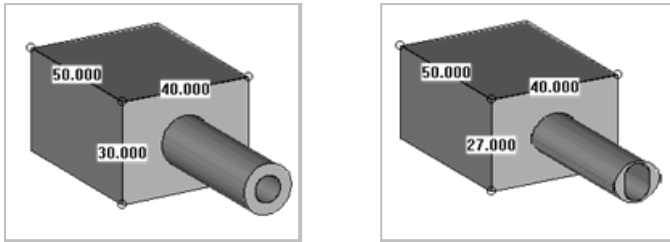
1. Z-Compensation



For models built with Stereolithography and Laser Sintering, overcure may cause extra material to build up on down facing surfaces. To avoid the time-consuming process of manually correcting these errors after the part is finished, the Z-compensation function can be used. To Z-compensate the selected parts, the user has to fill in a Z-compensation value in mm or inches and choose a Z-compensation algorithm in the following dialog:



The Z-compensation will classify all down facing surfaces that need compensation and offset them with the desired value. Appropriate modifications are made to adjacent triangles to keep the part consistent and error free.



The figures above display the result of a Z-compensation:

- The first figure represents the original part.
- The second figure represents the same part, but Z-compensated with a value of 3 mm.



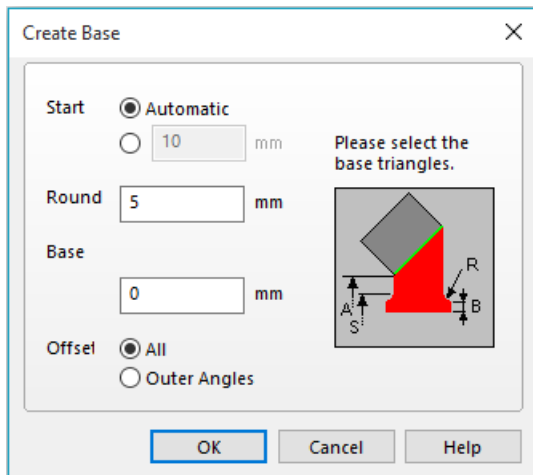
Remark: A Z-compensation value will in reality never be bigger than 0.5 mm and this case 3 mm is used for an educational purpose. If a part already has been Z-compensated, Magics gives a warning shown in the Z Compensated field on the 1 Part Info page.

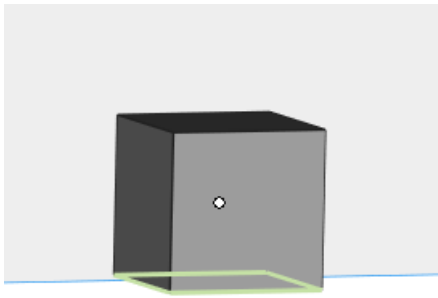
2. Create base



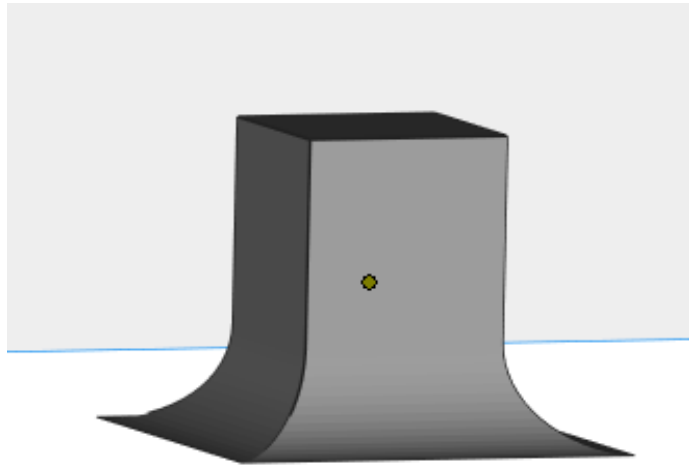
With this command, user could make an extrusion from the marked triangle(s) to generate a part base. It is available with SG+ license.

This function is for metal printing that uses wire cutting in the post processing steps. This kind of rounded edges base attached to the build platform could help conducting the heat and lowering the chance of printing failure.





Part with marked triangles



After applying

8.6. Part Pages

- See **Part Info page, page 585**

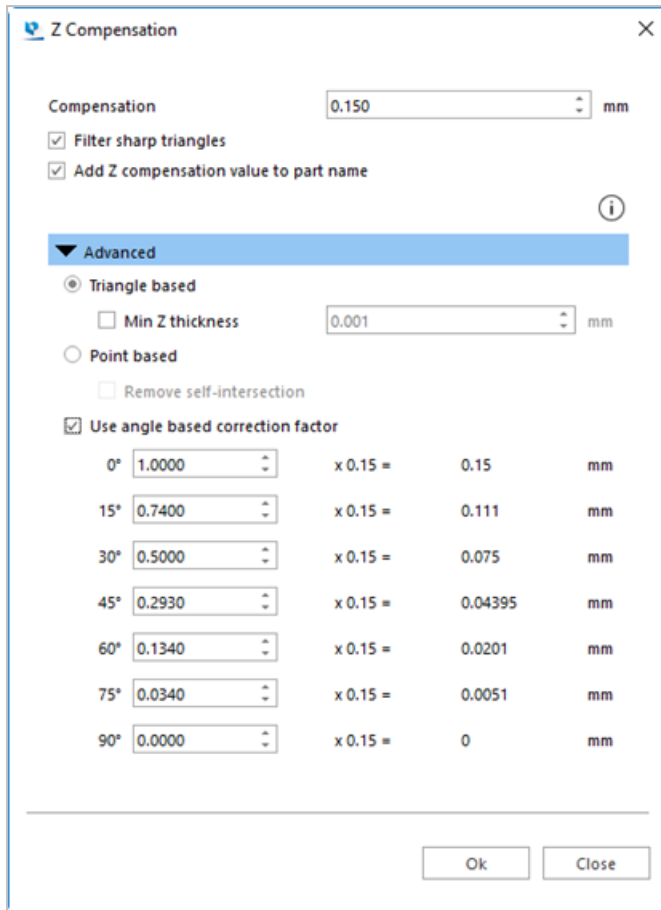
Filter sharp triangles	Triangles thinner as this distance will be marked or removed, depending on your choice.
Add Z-compensation value to part name	This will add the value of the Z compensation to the Part Name.



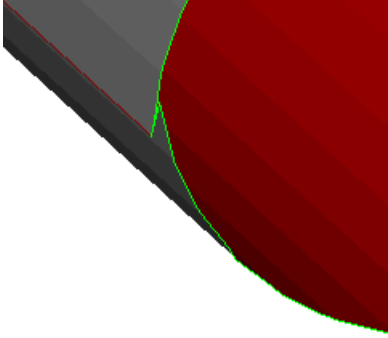

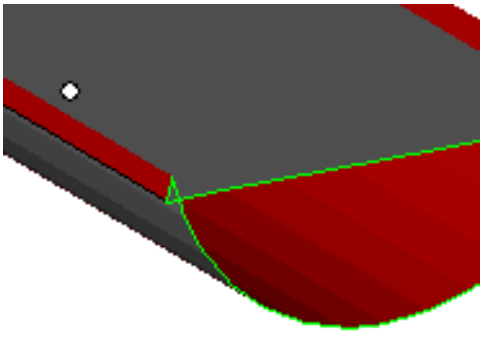
Remark: The parameters used for 'Filter Sharp Triangles' are linked to the ones in the fixing wizard.

Advanced Options

In the Advanced Options, you can choose which algorithm Magics should use to perform the Z compensation. Also extra options to refine the Z compensation are available.



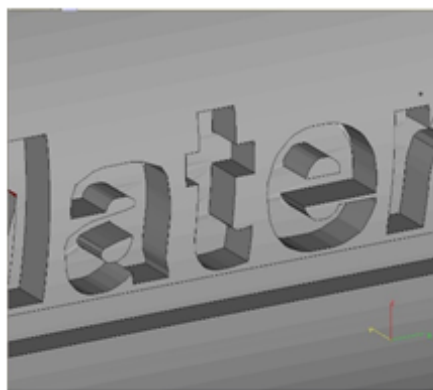
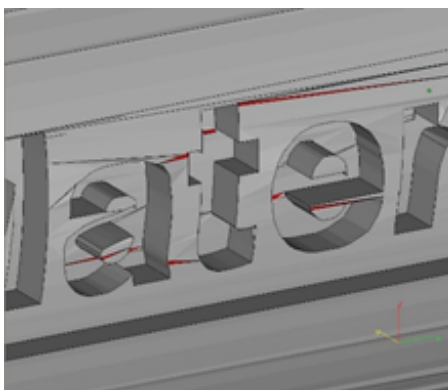
<p>Triangle Based</p>		<p>This algorithm will detect all down facing triangles and will move them in the Z direction over a distance determined by the entered Z compensation value.</p>
	<p>Min Z-thickness</p>	<p>Features thinner than – measured in the Z direction – this value will not be Z compensated.</p>

<p>Point Based</p>		<p>This algorithm will detect all down facing triangles and will move the points of those triangles in the Z direction over a distance determined by the entered Z compensation value.</p> 
<p>Remove Self-Intersections</p>		<p>Remove Self- Intersections: By moving the down facing surfaces upwards, sometimes self-intersections may occur. When 'remove self-intersections' is switched on, a post processing will remove these intersections. This might take a while if the file is big.</p>
<p>Use Angle Based Correction Factor</p>	<p>Checking this function will apply a Z-compensation depending on the angle of the down facing triangle. If the angle is orientated horizontal, a full Z-compensation is performed. If the angle approaches an upright position, almost no Z-compensation is performed. This can be useful if you work with very accurate machines.</p>	

In most cases, the triangle based algorithm will give much nicer results. The figures below show an example.

Point based Z compensation

Triangle based Z compensation



8.7. Sinter

1. 3D Nester

Workflow



- Import part(s)
- Define orientation
- Position large part(s)
- virtual copies
- Pre-nest parts with Subnester
- Protect small/ fragile part(s) with Sinterboxes
- Run 3D Nester
- Export platform



More information on the other items within the workflow can be obtained during one of our training sessions.

Introduction



Magics 3D Nester allows to automatically place parts in the 3D build volume and create an optimal load for a sintering machine(s), maximizing the number of parts in the build envelope and / or minimizing the build time.

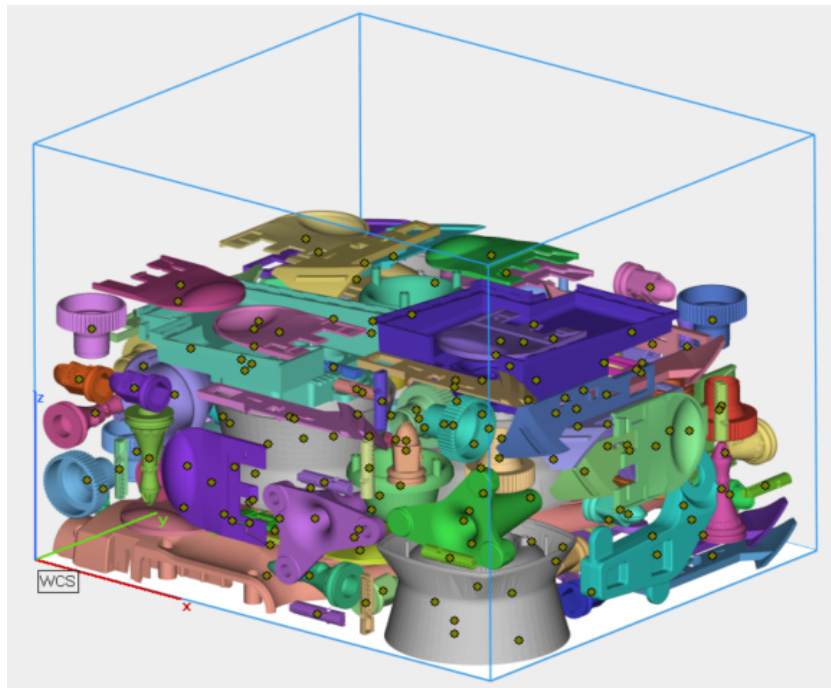
At the same time, the software ensures that none of the parts collide with other parts, with the build platform, or no-build zones.

A live preview is available during the nesting of the platform. Magics shows where parts are placed, which ones still have to be processed and / or which part(s) don't fit in the build envelope. (E.g. part is too large, platform is already fully loaded etc.)

There are two nesting methods in Magics: **Bounding box** and **Geometry**.

The **Bounding box** method uses the bounding boxes as the boundaries of the parts. After the nesting, the bounding boxes of the parts do not overlap. Any margins are calculated from the bounding boxes. This method is very fast but does not allow to achieve high density of packing unless the parts are all rectangular cuboid shape lying flat on the XY plane.

The **Geometry** method uses the geometry of the parts to calculate the rough boundaries of the models, and the margins are calculated from these more realistic boundaries. The Geometry method allows for tighter packing but takes more time due to a significant number of possibilities to calculate. The Geometry nesting method has a greater number of parameters in the UI.

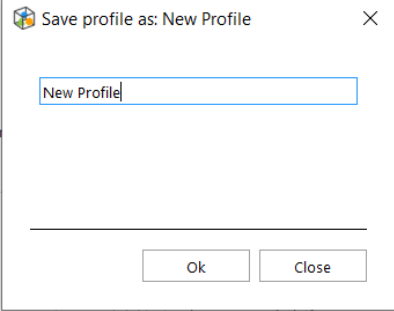


Overview of the main nesting settings

The main nesting settings include some of the basic settings that are likely to be changed every time the nester is run.

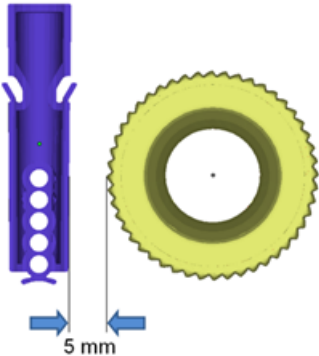
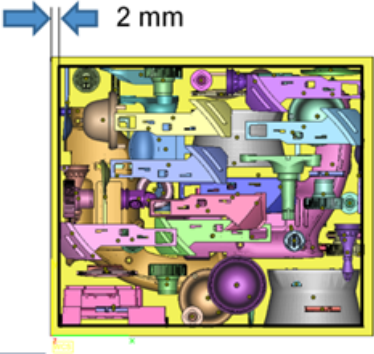
Profiles

Profile	<p>A profile is a "recipe" for the nester that automatically sets all parameters to the values they had when the profile was saved.</p> <p>A default profile "Magics Defaults" with default nesting values is always available for selection. If any value of "Magics Defaults" is changed, then a new setup can be saved to a new profile.</p> <p>If the parameters are modified while a profile is selected, this profile is marked with a star (*). The changes become permanent if the profile is saved with the changes.</p> <p>Profile: <input type="text" value="Magics Defaults*"/> 📄 🗑️</p>
----------------	---



Profiles can be easily transferred to other systems via the Magics Profile (Options & Help -> Customization -> Export14.4 Magics Profile). Share your Magics Profile file with another user to share the nesting profiles via Import Magics Profile.

Margins


Part interval	<p>This value refers to the minimal distance between two parts.</p> 
Margin to sides	<p>This is the minimal distance to the build envelope of the machine.</p> 


Parts selection

Selected parts	Only the selected parts in the part list are taken into account during nesting. Unselected parts are considered by 3D nester as 'locked parts' by default.
All parts	All parts loaded into the scene are taken into account during nesting.
Selected parts first	Part loaded in a scene, which are selected will be nested first. These are parts with a higher priority compared to the others.

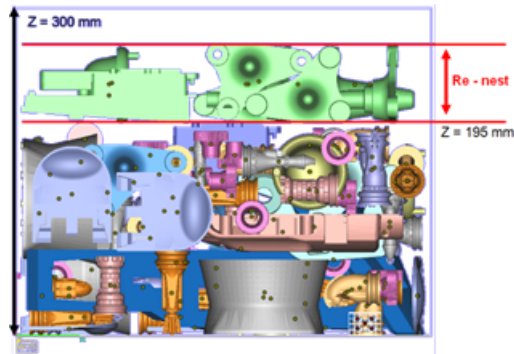
Stop criteria

Stop at	The nesting will be ended as soon as all parts are processed and packed inside
----------------	--

first solution	the build envelope.
Optimize until nesting density ...%	<p>Nesting will end when the predefined nesting density is reached. The nesting density is calculated according to the following formula:</p> $ND = \frac{\text{Volume all parts}}{Pa * Z pos_{\text{highest part}}} * 100\%$ <ul style="list-style-type: none"> • <i>ND</i>: Nest Density • <i>Pa</i>: Platform area • <i>Z pos</i>: Z position <div style="border: 1px solid #add8e6; padding: 5px; margin-top: 10px;"> <p> Parts are only taken into account when the center of the part lays inside the platform bounds</p> </div>
Manually end nesting process	<p>The user manually ends the nesting process based on the nesting height and nesting density visible. At this moment the current status of the nesting can be kept. If the nester has nested the maximum possible parts and optimized the build to the best possible state according to the settings chosen by the user, the nesting will end automatically.</p>
Stop after	When checked, the user can enter a max duration of nesting in hours, minutes and seconds.

 Independently of which stop criteria is chosen, the 3D Nester will always perform an interlocking test at the end.

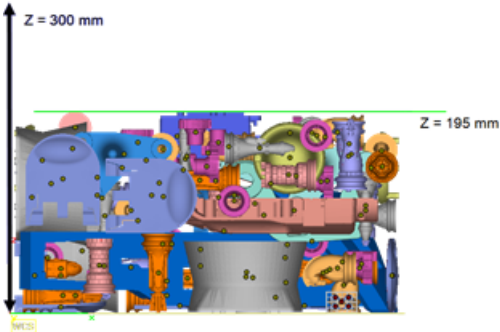
Re-nesting

Re-nest all parts above	<p>Nest additional parts above a specified Z height even if your build is already launched.</p> <p><i>Re-nest all parts above = 195 mm</i></p> 
--------------------------------	--

Overview of the Advanced Settings

Advanced settings include a variety of options for advanced use cases as well as more granular control over basic flows.

Nesting settings

Start from current platform configuration	When checked, the 3D nester will start from the current platform configuration: all parts that are within the build envelope will remain there. They might still move around during the optimization phase.	
Maximal used height	<p>The maximal used height is the height used during nesting. By default the height specified in the machine setup, of the selected machine is used. This value can be changed though obviously this value must be smaller than the real height of the build envelope.</p> <p><i>Maximal used height is 195 mm</i></p> 	
Optimize platform	Minimize height	Nest parts to achieve the minimal possible build height.
	Distribute in height	Distribute parts uniformly over the entire height of the platform.
	Optimize slice volume	Distribute parts uniformly over the entire height of the platform and make slice volume as similar as possible across all slices.
	Optimize slice volume and minimize height	Try to minimize the height of the build while keeping the slice volume as similar as possible across all slices
Use fast nesting	Faster nesting with lower density. Recommended for nesting too many parts that greatly vary in size and won't all fit on one platform.	

Freedom of parts

Default	<p>Specify which type of freedom of parts is used during nesting. The default value is applied to all parts.</p> <div data-bbox="384 1789 687 1973" style="border: 1px solid black; padding: 5px;"> <ul style="list-style-type: none"> Fixed Rotate 90° and translate Fix Z direction Fix XY directions <li style="background-color: #e0e0e0;">Fix bottom plane Fix bottom plane and XY Translate only </div>	
	Rotation angle	For "Fix Z direction" and "Fix bottom plane" free-

	dom of parts it is possible to specify any rotation angle value for the Z or pick from preset values.
Specify per part	Enable possibility to set values for each part individually in the 1 3D Nester which opens after confirming the nesting.

Default freedom of parts values:

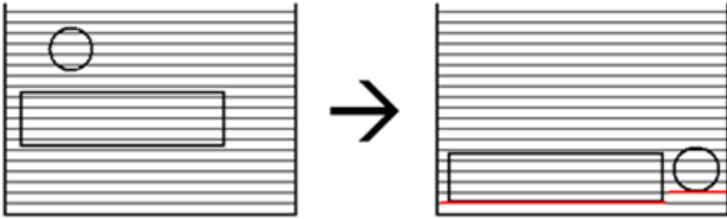
Fixed	The part will be completely fixed, meaning no rotation or translation will be done with the part.
Rotate 90° and translate	The part is rotated with steps of 90°, translation is free.
Fix Z direction	The part will only turn around the X and Y axis with angles of 180° and will rotate freely around the Z axis, translation is free. Around the Z-axis, the part can rotate by any number of degrees, select from the preset values (15°, 30°, 45° or 90°) or type the necessary rotation angle.
Fix XY directions	The part can rotate around any axis with steps of 180° only, translations are free.
Fix bottom plane	The part will only turn around the Z axis, translation is free. Around the Z-axis, the part can rotate by any number of degrees, select from the preset values (15°, 30°, 45° or 90°) or type the necessary rotation angle.
Fix bottom plane and XY	The part will only turn around the Z axis with steps of 180°, translation is free.
Translate only	The part will only be translated.

Rotation angle options

	Rotation Around axis			Translation Direction		
	X	Y	Z	X	Y	Z
Rotate 90° and translate	90°	90°	90°	Yes	Yes	Yes
Fix Z direction	180°	180°	15° 30° 45° 90° Other	Yes	Yes	Yes
Fix bottom plane	No	No	15° 30° 45° 90° Other	Yes	Yes	Yes

	Rotation Around axis			Translation Direction		
	Fix X-Y directions	180°	180°	180°	Yes	Yes
Fix bottom plane and X-Y	No	No	180°	Yes	Yes	Yes
Translate only	No	No	No	Yes	Yes	Yes
Fixed	No	No	No	No	No	No

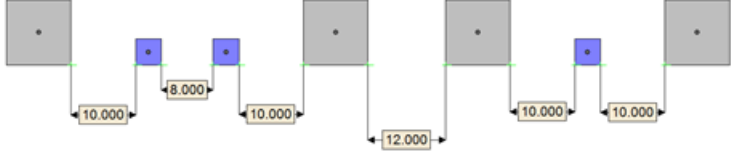
Part settings

Place on machine layer	Parts are slightly post-shifted so that the minimal z-coordinate of each part is located precisely on a layer of the machine.	
		
	As defined in Machine Properties	The layer thickness defined in the machine library is taken into account when placing parts at the layers
	Specify layer thickness	Manually specify the layer thickness to use when placing parts on the layers
Skip copies of the part that failed to nest	<p>Normally, 3D Nester goes through all copies of parts and tries to place them, even if other copies have already failed to be placed. Sometimes the algorithm can find a placement for such a part after several failed attempts, but the time needed for the calculations can be significant.</p> <p>To reduce the nesting time of platforms with copies, enable "Skip copies of the parts that failed to place" and specify the number of attempts the 3D Nester should make before skipping the remaining copies. The recommended number of attempts is 2-3.</p> <p>If a copy fails to nest, after a successful placement of another copy of the same part the counter of failed attempts is reset.</p>	


Special parts

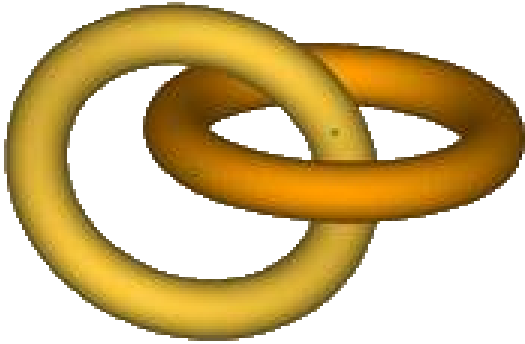
Special parts are user-selected parts that require a special approach during nesting. It is possible to identify special parts in the 3D Nester, if any of the special settings are selected in this section of the 3D Nester.

Special parts interval	<p>Define different part intervals based on the size of the parts.</p> <p>Large or dense parts preferably use a special part interval.</p>
-------------------------------	--

	 <ul style="list-style-type: none"> - Normal part interval = 8 mm - Normal - large interval = 10 mm - Large interval = 12 mm
<p>Nest special parts in the platform center</p>	<p>When checked, you are able to assign priority to special parts and nest them automatically in the platform center. Special parts are defined in the 1 3D Nester that will appear after the 3D Nester is started.</p>

Interlocking analysis

<p>Enable interlocking analysis</p>	<p>Check this option to enable interlocking check for the 3D Nester run.</p>	
<p>Interlocking candidates</p>	<p>Automatic</p>	<p>When this option is checked, the 3D nester will try to analyze all parts and identify the ones that can cause interlocking. This information will be taken into account during nesting.</p>
	<p>Select candidates manually</p>	<p>Option to manually indicate which parts are candidates for interlocking. Manually selected candidates are colored in orange.</p> <p>To select parts, click on individual parts or hold left mouse button and drag to select an area with parts.</p> <p>To unselect parts, click on a part again or select an area by holding Shift key and left mouse button.</p> <p>To select and unselect parts, make sure that the mouse mode for indicating parts is enabled.</p> 
<p>Resolve interlocking</p>	<p>Automatically</p>	<p>During nesting, if interlocking is detected, Magics will show a pop-up informing that there are interlocked parts, and will proceed to re-nest interlocked parts without actions from the user needed. In case of multi-platform nesting, after interlocking is resolved, nesting will continue on the next platform without interaction needed.</p>
	<p>Manually</p>	<p>If interlocking is detected during nesting, then the nesting process is paused, only interlocked parts are visible, and a dialog is shown.</p> <p>Options:</p> <ul style="list-style-type: none"> • Solve interlocking: Magics will re-nest interlocked parts and take into account the "Freedom of parts" that was assigned to the interlocked parts before

		<p>the start of the nesting (see Part settings -> Freedom of parts).</p> <ul style="list-style-type: none"> • Keep current state: nesting process will end without resolving interlocking. Use manual tools to resolve interlocking afterwards. 
--	--	---

Multi-platform nesting

Enable multi-platform placement	Place parts on multiple platforms in iteration of the 3D Nester
Nest only on opened platforms	The parts will only be nested on opened platforms. If this option is not enabled, then the nester will automatically create new copies of the same platform to continue nesting, until all parts are nested.

When multi-platform nesting is enabled, and the 3D Nester is running, then the buttons in the progress dialog work as follows:

- **Skip to next platform:** stop nesting the current platform and move to the next platform.
- **Keep status:** stop nesting and keep all nested parts and platforms as they are at the moment.
- **Cancel:** stop nesting and revert parts to the positions before the nesting operation began.

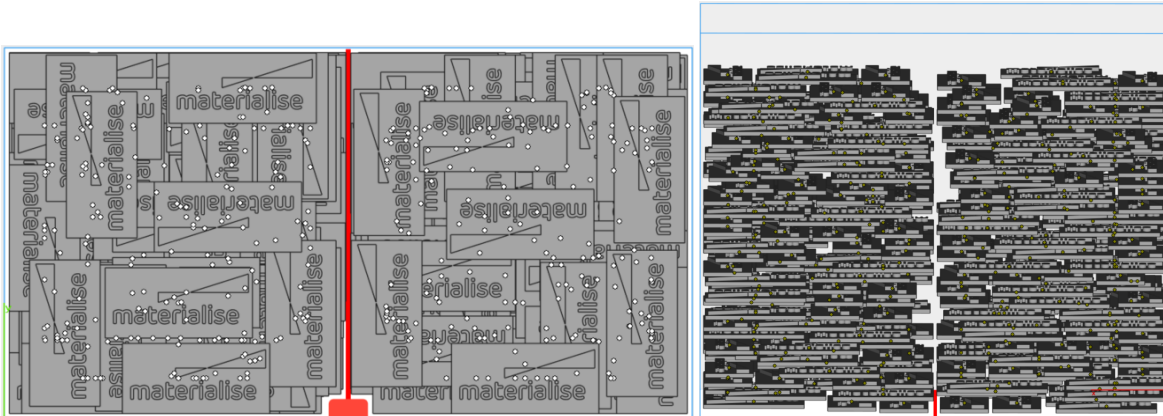
Multi-laser platform settings

When the field overlaps are enabled for the current platform in the 2 Machine Properties, it is possible to use Multi-laser platform settings in the 3D Nester dialog.

Nest according to field overlaps	Enable if the nester should account for the scan fields during nesting.
	<p>Strictly avoid field overlaps</p> <p>Enable to have all parts strictly avoid any field overlaps.</p> <ul style="list-style-type: none"> • It is possible to have only some parts strictly avoid field overlaps in the Settings per part that will appear after you click OK in the 3D Nester dialog.
	<p>Balance slice volume per scan field</p> <p>3D Nester will try to make the slice volume as even as possible in each scan field in each layer. This setting might make nesting slower, but the laser efficiency during printing can improve.</p>

Depending on the part size, orientation, and nesting settings, the parts will interact with the field overlaps as follows:

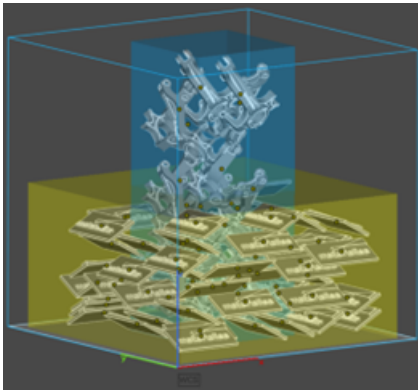
- If a part can fit, then place the part in one scan field without crossing into the overlap
- If a part can't fit into a single scan field, then place the part into a minimal possible number of scan fields crossing a minimal possible number of field overlaps.
- The nester will try to distribute parts between the scan fields evenly so that the build height is similar in all fields.



Custom nesting zones

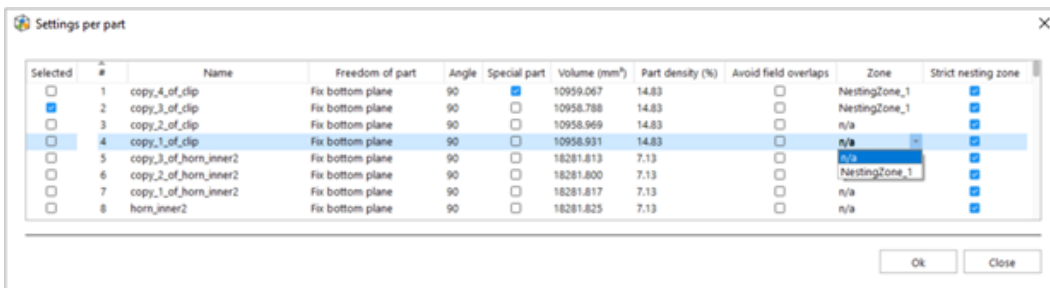
When Nesting zones are set and enabled for the current platform in the 2 Machine Properties, it is possible to use Custom nesting zones settings in the 3D Nester dialog.

Nest parts in zones	When enabled, then each part can be assigned to one of the set-up zones and during nesting parts will be placed in their assigned zones. If a part can't fit into its assigned zone, then it is placed in another available spot in the platform volume. Parts can be assigned to the specific zones in 1 3D Nester that will appear after you click OK in the 3D Nester dialog.
Strict nesting zones	When enabled, then all parts assigned to zones should be placed only into the zone that they are assigned to. If a part can't fit into the assigned zone, then it is not nested and is placed outside the platform.



Settings per part dialog

There are settings in the 3D Nester that, if enabled, will prompt the **Settings per part** dialog to appear after you click OK in the 3D Nester dialog.



Settings per part table shows all parts on the platform and can have the following columns:

Column	Description	What setting triggers the column to be shown
Selected	Indicates if the part is selected on the platform. If in part selection it is indicated that only selected parts should be nested, then the selection in this list will change which parts are nested.	Shown by default
#	Order number of the part	Shown by default
Name	The name of the part.	Shown by default
Freedom of part	Freedom of parts options are available to assign to each part individually. For example, it is possible to make specific parts fixed in their existing position. The freedom of part can be modified by double clicking on it and selecting the value from the drop down.	Freedom of parts -> Specify per part
Angle	Set custom angle for each part with "Fix bottom plane" or "Fix Z direction" freedom of part value.	Freedom of parts -> Specify per part
Special part	Select the checkbox to indicate that the part is a special part. Special part	Special parts -> Special parts interval;

	interval and / or nesting in the platform center will apply depending on the Special parts settings from the 3D Nester dialog.	Special parts -> Nest special parts in the platform center
Volume, mm3	The total amount of volume of the part	Shown by default
Part Density, %	Also known as packing factor. $PD = \frac{PVol}{BBVol} * 100\%$ <i>PD: Part density</i> <i>PVol: Part volume</i> <i>BBVol: Bounding box volume</i>	Shown by default
Strictly avoid field overlaps	Assign parts that should strictly avoid overlaps	Multi-laser platform settings -> Nest according to field overlaps
Zone	Double click to assign the part to one of the zones. "-" value means no zone was assigned, and the part can be nested in any zone or space that isn't assigned as a zone.	Custom nesting zones -> Nest parts in zones
Strict nesting zone	Check this option so that the part can be placed in the assigned zone only.	Custom nesting zones -> Nest parts in zones
Save parameters per part for the current Magics session	Parameters will not be reset to the defaults after running the Nester.	

3D Nester Progress

During the nesting, the progress of placing parts is shown in real time. The nesting is divided into 4 steps. For every step a progress bar is shown.

1. Step 1: **Preprocess parts**

Parts are being analyzed before the actual nesting is taking place. During preprocessing, the parts are removed from the platform build envelope, except for fixed parts and if Start from current platform configuration setting is enabled.

2. Step 2: **Nest parts**

The actual nesting of parts takes place in this step. During nesting the following information is displayed in the nesting progress dialog

- Number of parts that are already nested
- Total number of parts to be nested

- Parts out of platform: number of parts placed outside the platform boundaries (failed parts)
- Nesting height and the amount of time after which the last change in height occurred
- Nesting density and the amount of time after which the last change in density occurred
- A graph showing the evolution of nesting density and height in real time

The graph can be managed during nesting in the following ways:

Action	Result
Click + hold left mouse button and move the mouse	Selects a certain period of time from the chart to visualize in more detail
Scroll mouse wheel	Zoom in / out of the chart
Double left click	Zoom out to the main view of the chart
Hover with the mouse over the points on the charts	The density / height values at the indicated time on the chart are shown
Save as PNG	Exports a PNG of the chart
Export to CSV	Exports the values of the chart to a CSV file

3. Step 3: Nesting optimization

Depending on the chosen 'Stop criteria' and the Optimize platformvalue, the 3D nester further optimizes the placement of the parts.

4. Step 4: Interlocking test

During the final step an interlocking test is performed.

Interlocking means that 2 (or more) parts cannot be separated due to the positions they are nested in. (See Interlocking analysis in 1 3D Nester description)

Nesting can be stopped manually by clicking on **Keep status** or **Cancel**.

- **Keep status:** preserve the current state of the parts placement and stop nesting.
- **Cancel:** stop nesting and revert all parts to the positions before the latest nesting adoration started.

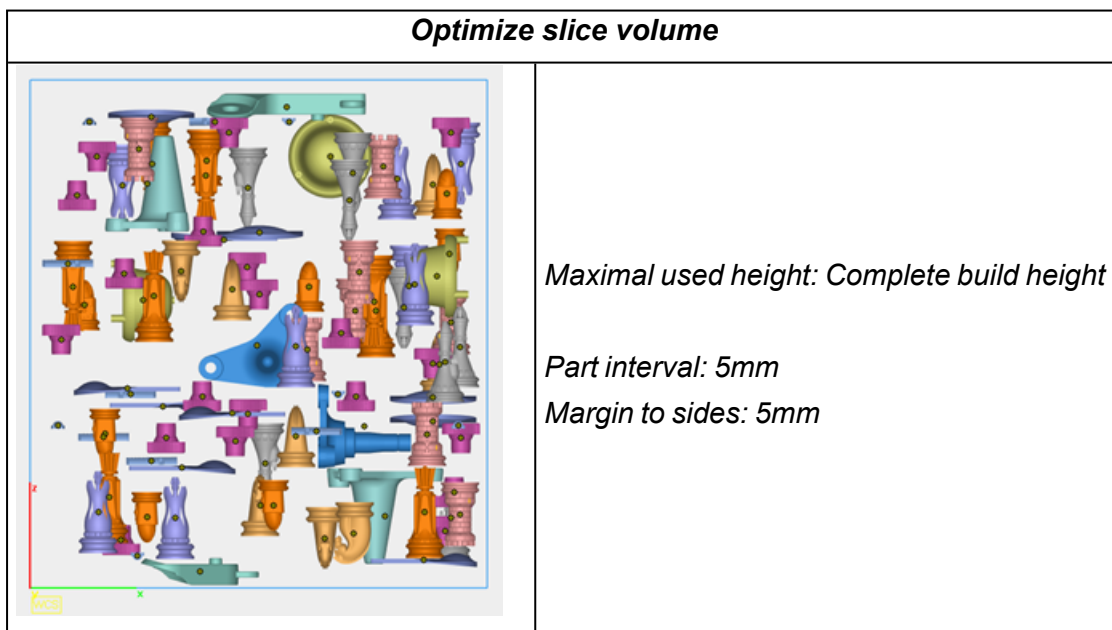
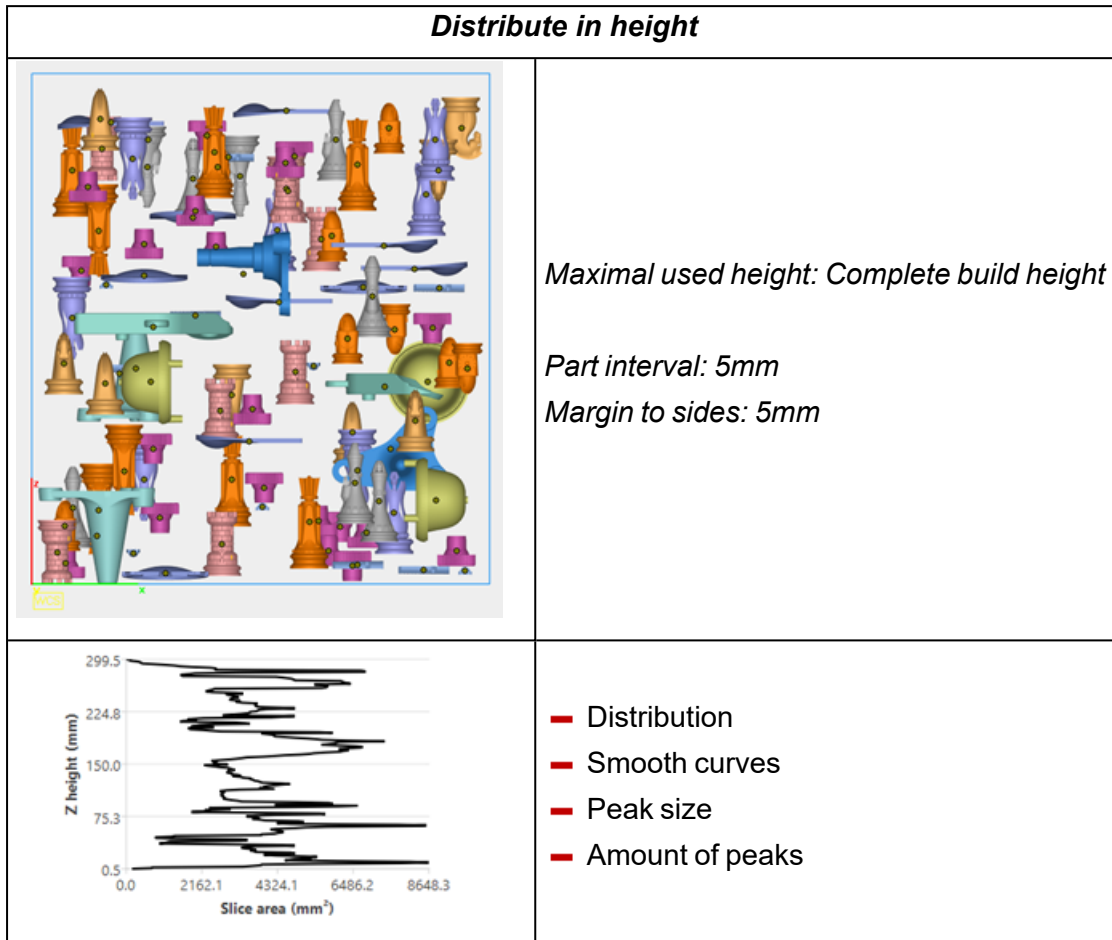
3D Nester - Nest by Bounding Box

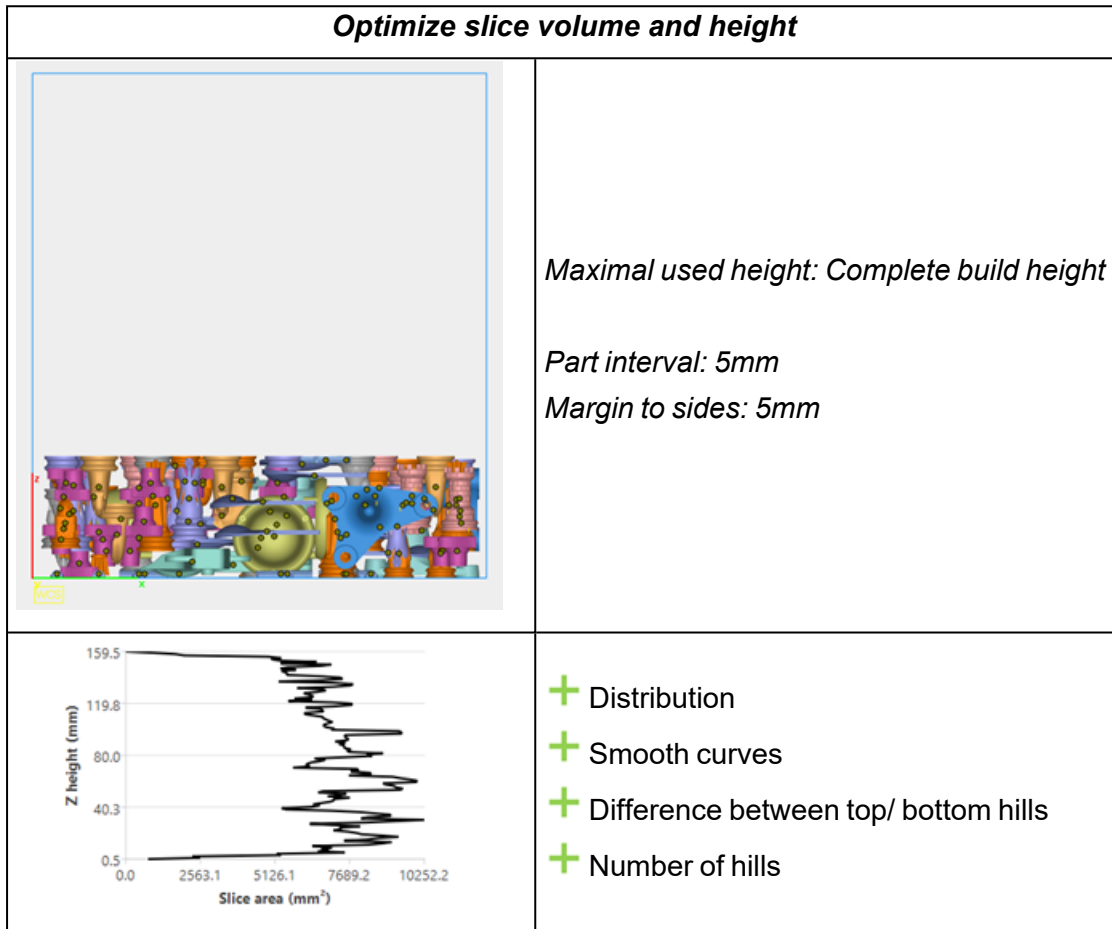
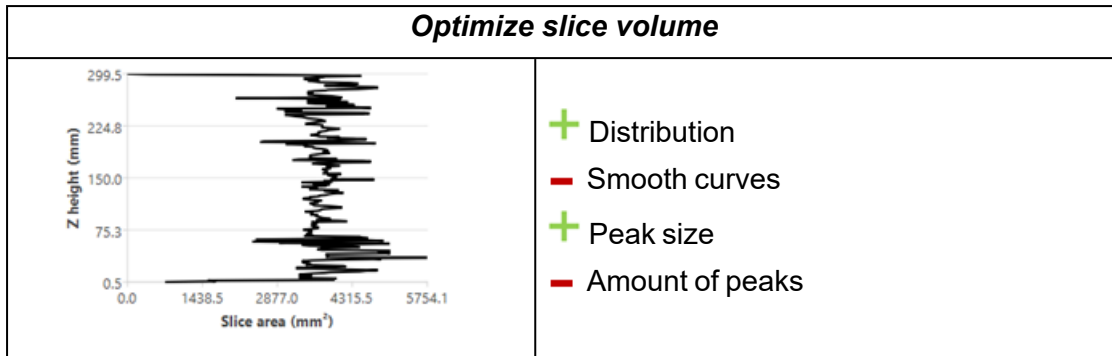
There are fewer parameters for the Bounding box nesting method. Refer to the Geometry parameters to understand the Bounding box parameters.

2. 3D Nester - Check slices distribution

The slice distribution chart gives you the possibility to analyze the surface area for each layer and distribution within your build. Less difference between the layers will increase the build quality.

There are 2 options within the 3D nester dialog that have an influence on the slices distribution graph. (3D Nester – Nesting settings – Optimization criteria)





Remark: Manual ending of the nesting procedure is needed when using one of the optimization methods.

3. Subnester



The Subnester is a pre-nester which can be executed before creating a Sinterbox around a group of parts. It will nest selected parts according to the set parameters. In this way small and fragile parts are nested together.

Workflow

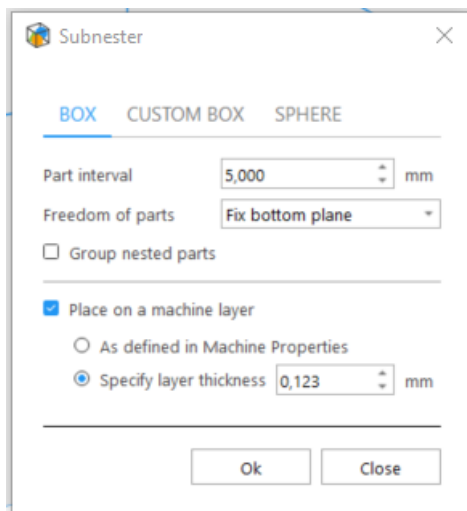


- Import part(s)
- Define orientation
- Position large part(s)
- Create virtual copies
- Pre-nest parts
- Protect small/ fragile part(s)
- Run 3D Nester
- Export platform

Subnester dialog

Launch Subnester command from the Sinter section of the Home ribbon.

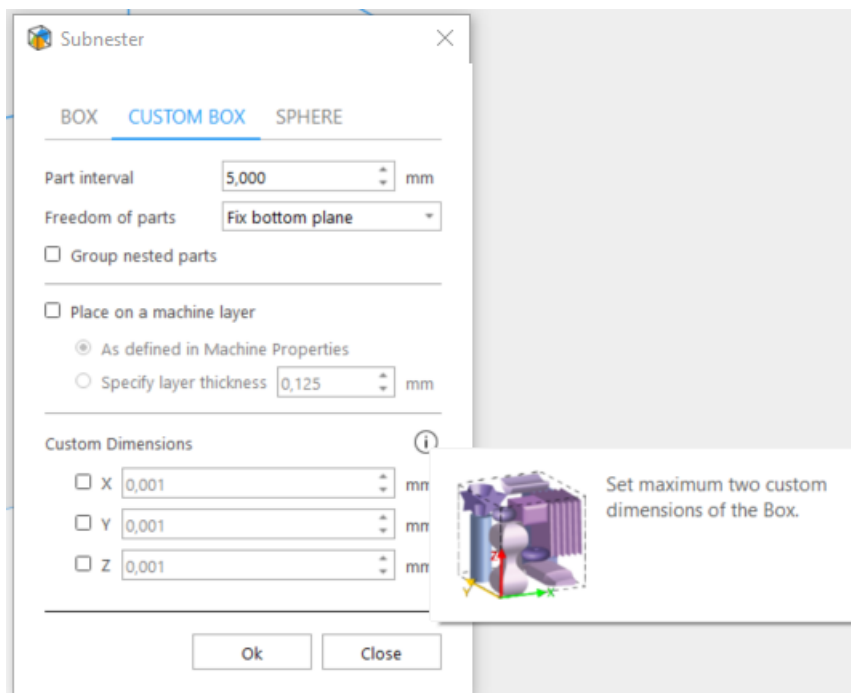
Subnester: Box



Magics will nest the selected parts in a box-based shape.

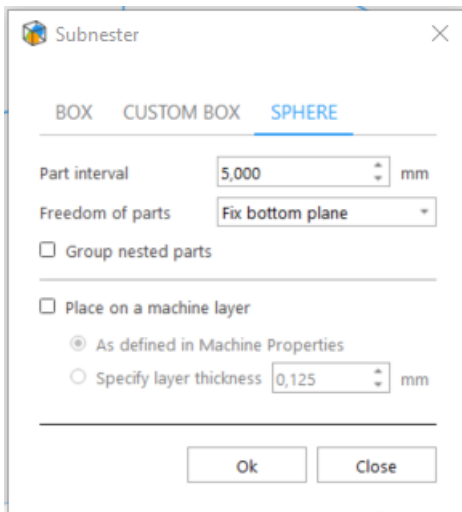
It is best to choose the box placement solution before creating a box-based Sinterbox.

Subnester: Custom Box



A Custom Box option allows to create a box with one or two predefined dimensions (X-Y-Z). The third dimension will be defined by the Subnester based on the space needed to nest selected parts.

Subnester: Sphere



The selected parts are nested in a sphere-based shape.

It is best to choose the sphere placement solution before creating a Freeform or Organic Sinterbox.

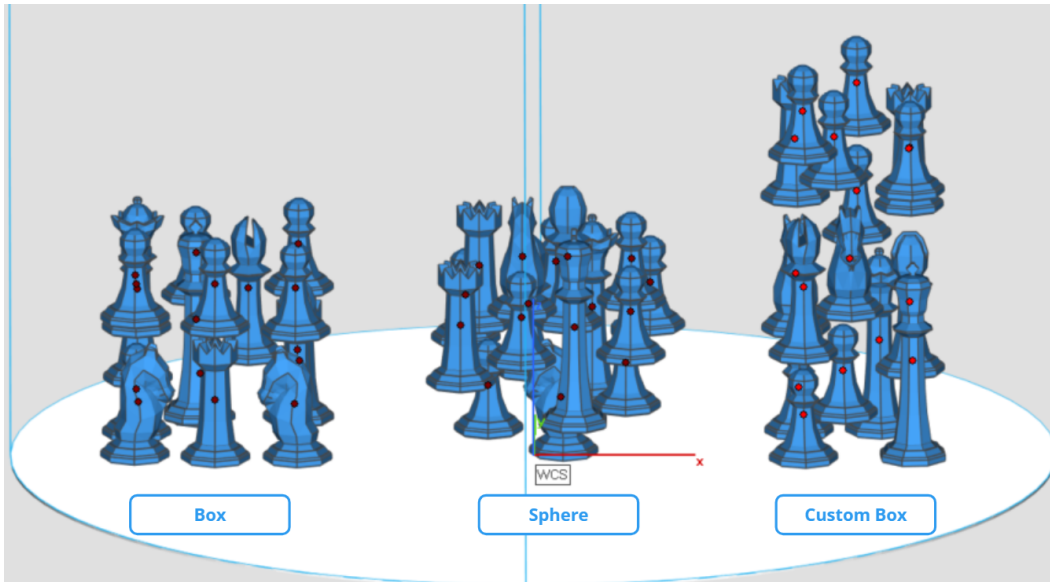
Subnester parameters

Part interval	This value refers to the minimal distance between two nested parts	
Freedom of parts	Choose the type restriction to translate and rotate your parts	
	Fix bottom plane	The part will only turn around the Z axis, translation is free
	Translate only	The part will only be translated
	Rotate 90° and translate	The part will be translated and rotated by 90°
Group parts	When checked, the pre-nested parts are placed in a group and are treated as one part.	
Place on a machine layer	Parts are slightly post-shifted so that the minimal z-coordinate of each part is located precisely on a layer of the machine.	
Get from machine properties	The layer thickness defined in the machine library is taken into account when placing parts at the layers	
Specify	Manually specify the layer thickness to use when placing parts on the layers	



In the subnester, parts are placed on the machine layer only relative to other parts in the group. In order to preserve the placement on the machine layer till the end of build creation, sinterbox parameter values and nesting parameter values need to be aligned with the layer thickness.

Subnester results examples



4. Sinterbox

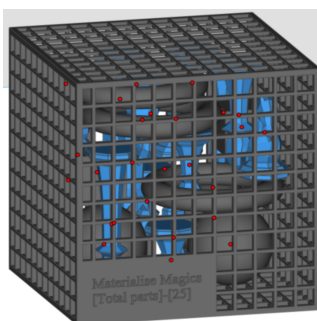


A Sinterbox is a cage-like enclosure that can be created around several selected parts, commonly grouped using a Subnester operation. The Sinterboxes can be used to protect small and fragile parts so that they don't get lost or damaged during the post processing phase.

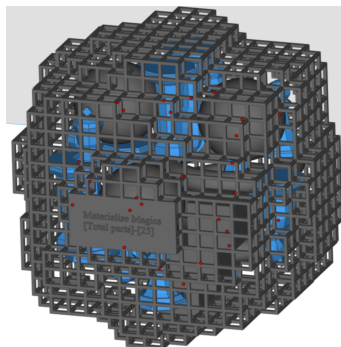
There are 3 different types of Sinterboxes that can be created in Magics:

- Rectangular (box-based)
- Freeform (box-based, following the shape of the parts)
- Organic (bag-like)

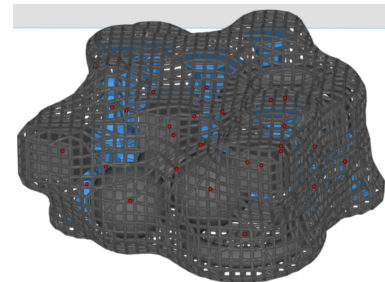
Rectangular



Freeform



Organic



Workflow



- Import part(s)
- Define orientation
- Position large part(s)
- Create virtual copies
- Pre-nest parts
- **Protect small/ fragile part(s)**
- Run 3D Nester
- Export platform

How to create a Sinterbox in Magics



To create a Sinterbox in Magics, follow these steps:


- Select the necessary parts
- Initiate the Sinterbox dialog from the **Sinter** section of the **Home** ribbon
- Select the type of the Sinterbox
- Specify parameters
- Preview the result (optional)
- Apply the Sinterbox. The resulting Sinterbox and the parts inside will be grouped so that they stay together during placement and nesting.

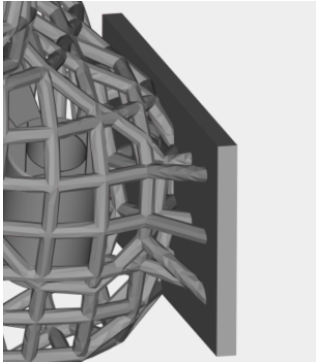


For faster rendering, any Sinterbox is generated in **Shade** instead of **Shade & Wire** mode. Adjust the shade mode of the Sinterboxes from the Part list or the toolbar as needed. For more information go to 1 View tools

Parameters and Options

Sinterbox parameters	Rectangular	Freeform	Organic
-----------------------------	-------------	----------	---------

Margin to part (a)	The minimum distance between (the bounding box of) a part and the edge of the Sinterbox.	+	+	+
Avoid holes and dents	Sinterbox will not go through large holes and into large dents, but wrap them at the level of the high points of the surrounding geometry.	n/a	n/a	+
Thickness (b)	The wall thickness of the box, except the cover thickness.	+	n/a	n/a
Cover thickness (c)	The wall thickness of the cover of the box.	+	n/a	n/a
Create internal walls	Creates walls in between parts within the Sinterbox. As a result, each part will have its own compartment in the Sinterbox. The thickness of the internal walls is the same as Beam thickness (e) that is defined for the perforations.  To calculate the position of the walls, the bounding box of each part is used.	+	+	n/a
Perforations				
Hole size (d)	Define the size of the perforations	+	+	+
Beam thickness (e)	The distance between two perforations.	+	+	+
Label				
Add content	'Part names' adds all the part names as text to the label. 'Amount of parts' adds the total amount of parts inside the Sinterbox to the label.	+	+	+
Content	Text input for the label content.	+	+	+
Font	Indicate the text style on the label.	+	+	+
Size	Indicate the size of the text in pt or mm.	+	+	+
Raised / Engraved	Indicate if the text of the label should be raised or engraved.	+	+	+
Height	Indicate label height or depth	+	+	+
Margin	Distance between the label text and the edge of the box.	+	+	+
Generate support	Create a more secure	n/a	n/a	+

<p>beams</p>	<p>connection of a label and the curved surface of the organic sinterbox with support beams.</p> 			
<p>Preview</p>				
<p>Preview</p>	<p>Automatically generate a preview of the Sinterbox.</p>	+	+	+

5. Toggle Nesting Density



Show /hide the nesting density of the current platform.

- See ***Toggle Nesting Density, page 471***

Chapter 9. Support Generation

9.1. Introduction

Magics RP is equipped with a module for support generation. The support generator allows you to generate supports for a whole platform and then edit the support on each surface individually. This support generation work routine allows you to easily adapt supports also after the part has been replaced on the building platform.

Support is only needed under certain surfaces. The selection is based upon the selection parameters from the Machine Properties (1: Support generation parameters). Magics selects these surfaces when entering the support generation module (2: Automatic support generation). Once you've arrived in the support generator module, Magics allows you to adapt the support to your needs (3: Modifying surfaces, support types and parameters). The support generation parameters are crucial during the initial automatic support generation but can be modified for each individual support. In the first place you can adapt the construction parameters, which are defined in the Machine Setup. This interactive change applies only to the active support. The active support is the one that is visible on your screen or when you made them all visible, it is the one with a different color (the bad edges color; default yellow). Secondly you can remove parts of the support in 3D or remove and if necessary redraw portions of the support in the 2D-edit window. At last you can save or export the support you've made.

Schematically a support is generated in the following steps:

1. Definition of the selection and the construction parameters in the Machine Setup;
2. Automatic support generation;
3. Modification of the support types and construction parameters;
4. 2D and 3D editing of the supports;
5. Saving and exporting the supports.

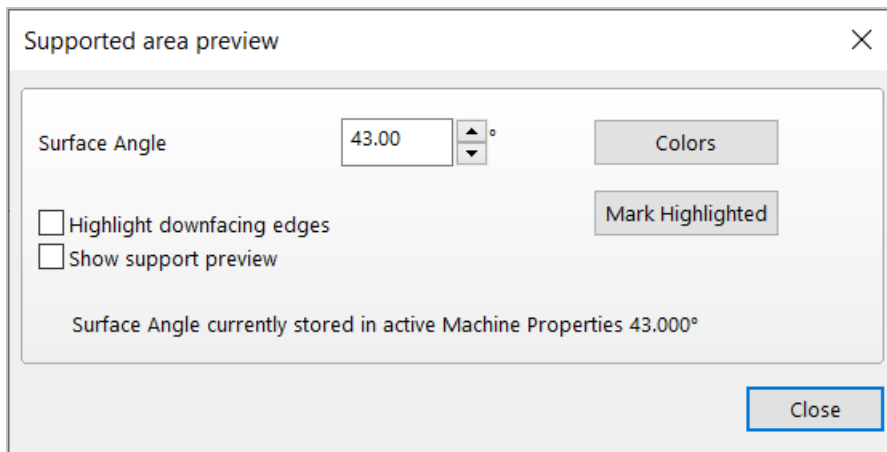
There are also special visualization options for the visualization of volume supports.

9.2. Magics - Support Generation ribbon

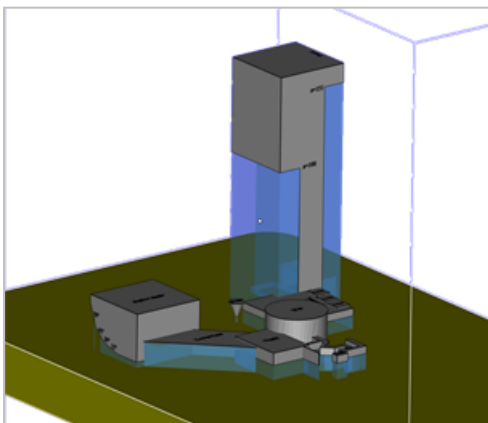
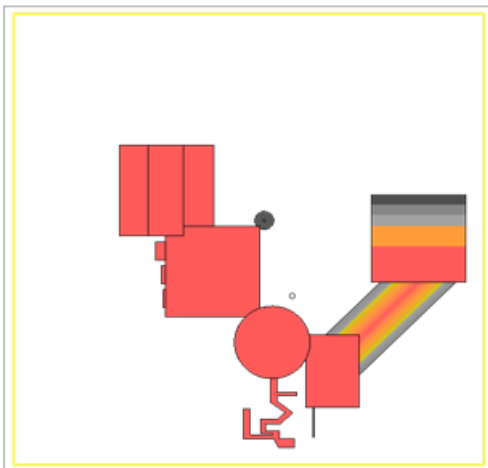
1. Supported Area Preview



Before generating your support within the support generation module, you can in advance visualize the areas that would need support. Based on the surface angle, the supported areas are indicated with color codes. Additionally, it is possible to highlight the down facing edges. When checking the 'Show support preview' checkbox, a provisional display of the support is visualized. At the same time of the visualization, you can change the surface angle or re-position your part based on the analysis of the surfaces and edges.



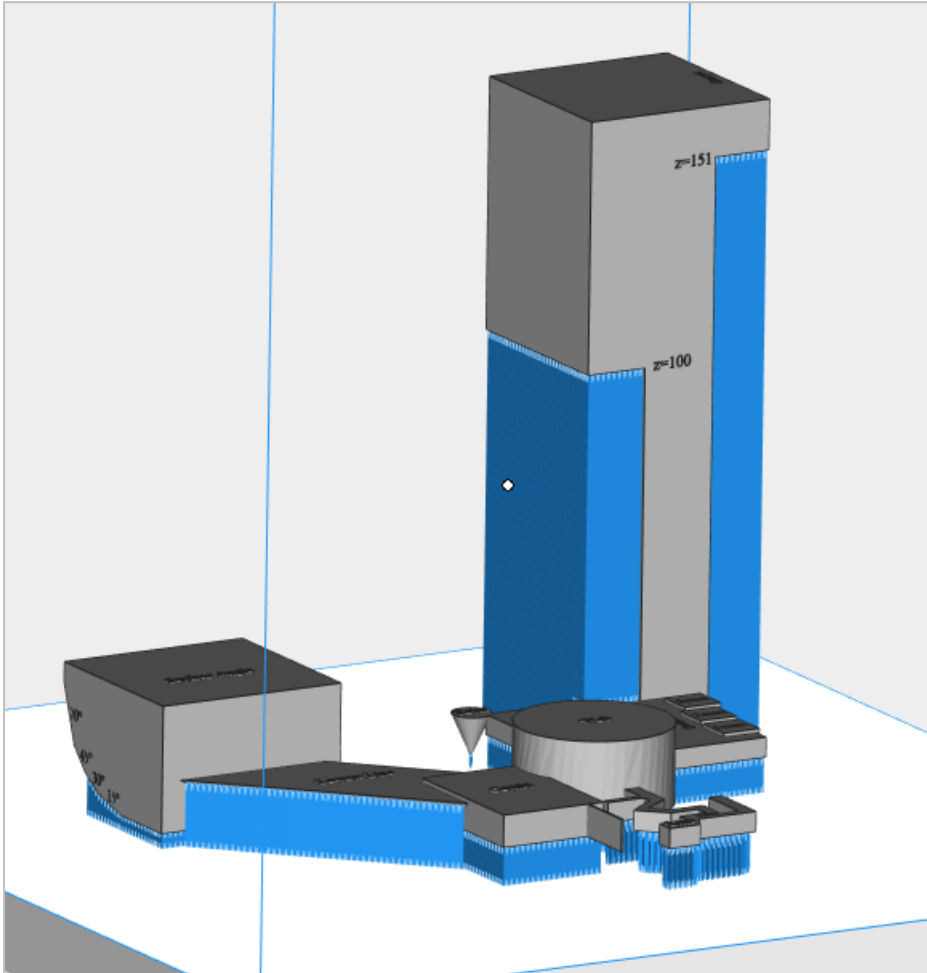
Surface Angle	Adjust the surface angle and see immediately on your part how the surfaces that need support change.
Mark highlighted	Marks the highlighted areas.



2. Generate Support



Easily create support structures on your part. Simply choose your platform, place your parts and generate your support.



3. Generate Support of Selected



Generate support for selected parts without accessing the support generation module. It will generate support for a mesh part.

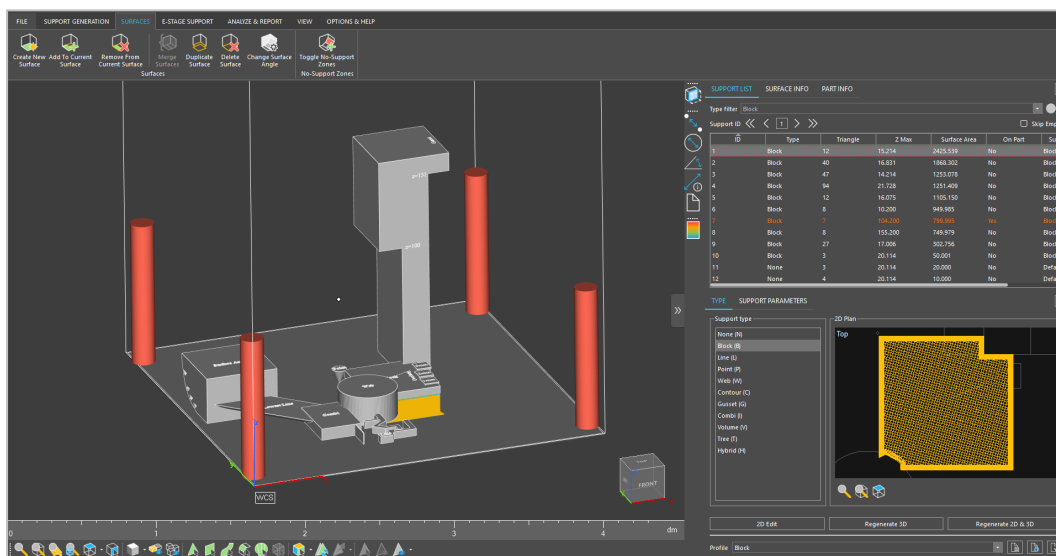
To generate the supports, Magics will utilize the support parameters which are found within the actual support profile without entering Support Generation mode.

4. Manual Support



Let Magics subdivide your part into the different surfaces that need support. Within the support generation module you can manually create the needed support structures for the already existing surfaces.

ID	Type	Triangle	Z Max	Surface Area	On Part	Sur
1	Block	12	15.214	2425.539	No	Block
2	Block	40	16.831	1868.302	No	Block
3	Block	47	14.214	1253.078	No	Block
4	Block	94	21.728	1251.409	No	Block
5	Block	12	16.075	1105.150	No	Block
6	Block	8	10.200	949.985	No	Block
7	Block	7	104.200	799.995	Yes	Block
8	Block	8	155.200	749.979	No	Block
9	Block	27	17.006	302.756	No	Block
10	Block	3	20.114	50.001	No	Block
11	None	3	20.114	20.000	No	Defai
12	None	4	20.114	10.000	No	Defai



5. Re-trim supports on platform



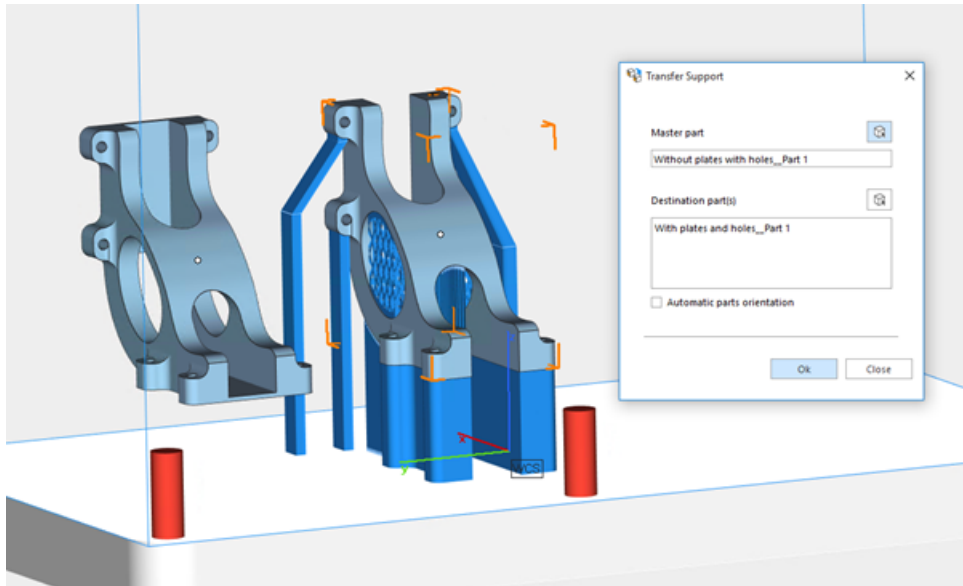
Initiating the command will re-trim supports with other parts or with the platform. During the placement of the supported parts on the platform it can happen that supports of one part intersects another part, in this case support needs to be re-trimmed with a part or with the platform.

6. Transfer Support



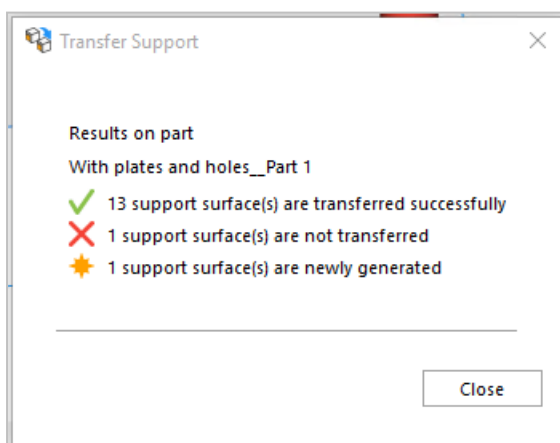
By clicking on transfer support button from the support generation ribbon user can select the “master part” (part with support) and the “destination part(s)”. Automatic orientation will change the orientation “destination part(s)” to “master part”. After clicking on “OK” button the algorithm will transfer all possible supports.

If the algorithm finds new support surfaces it will generate new surfaces based on automatic supports criteria in machine properties.



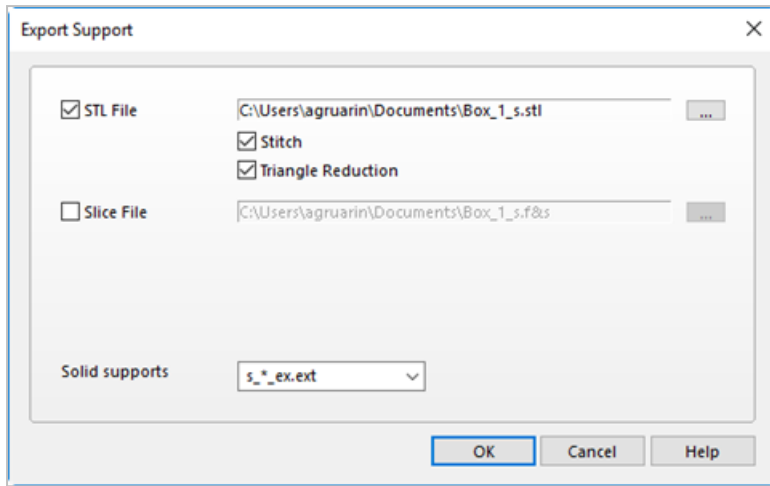
Results dialogue


This dialogue will show successfully, unsuccessfully and newly generated support surfaces. Unsuccessful supports will be highlighted as red.



7. Export support





STL File	Export supports as STL files.	
	Stitch	Open loops (contours with gaps in it) or gaps in the contours are closed.
	Triangle Reduction	During export a reduction of triangles is performed on the support.
Slice File	Export supports directly as slices.	
	 Remark: Type depends on specifications made in machine properties	
Solid supports	Select how the file name must be displayed for your machine.	

8. Unload support



Unload generated supports of parts that are selected.


9. Add No-Support Zones

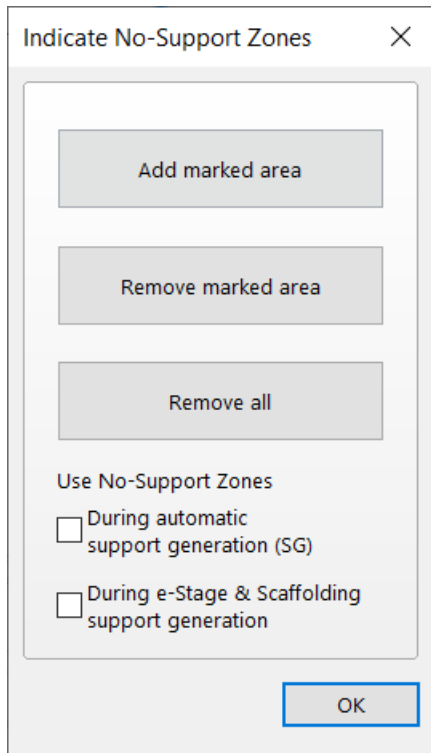
Some parts have surfaces that are very difficult to access and thus present complications for post-processing. Under these circumstances, it may be preferential to prevent supports from being generated in these zones, especially when achieving a higher quality surface finish outweighs a possible deformation due to lack of supports.



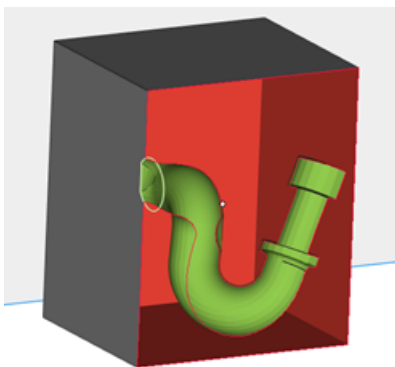
Important remark: In principle, these surfaces require supports to prevent surface deformation and prohibiting the generation of supports in these zones should be avoided as much as possible in combination with automatic support generation (like e-stage).

How to create 'no-support zones':

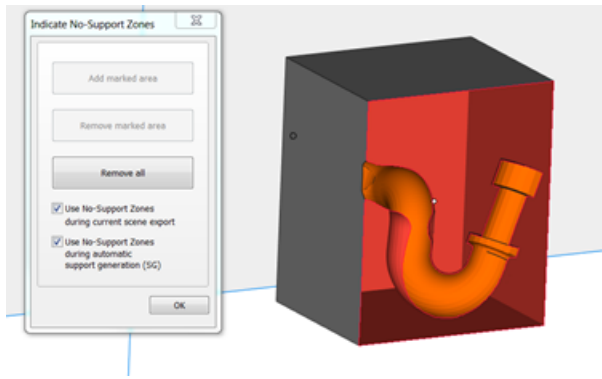
1. Press the 'add No-Support Zones' icon 
2. The 'Indicate No-Support Zones' window will pop up, and the cursor will change into a "marking" cursor. The view and marking tools can also be used to indicate the desired areas.



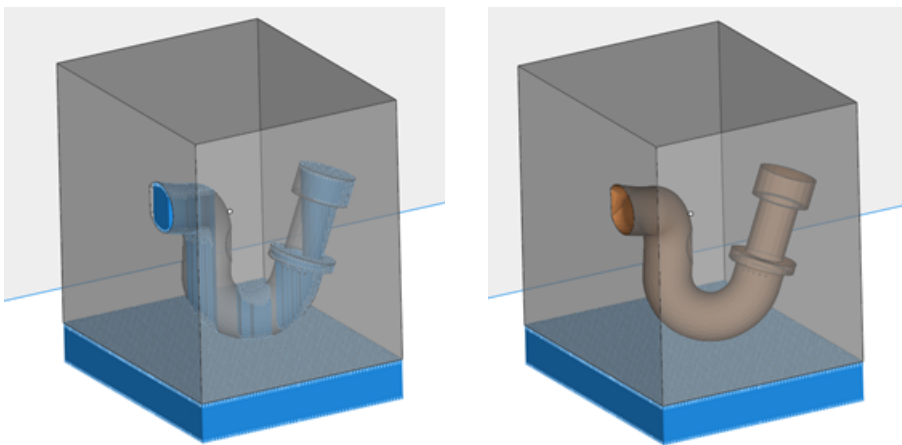
3. Mark the surfaces where no support is required.



4. Click on 'Add marked area' to define the marked triangles or lattice beams as 'No-Support Zones'. The marked areas will turn orange.



Depending on the checkboxes that are turned on or off in the 'Indicate No-Support Zones' window, SG or E-stage will not generate support on the No-Support Zones. You can see the difference below.



10. Toggle No-Support Zones



Toggle the visualization of the No-Support Zones.

11. Support Visibility



Toggle the visibility of the supports structures for all parts present in the scene.

- For support visualization settings, see paragraph Supports in Settings

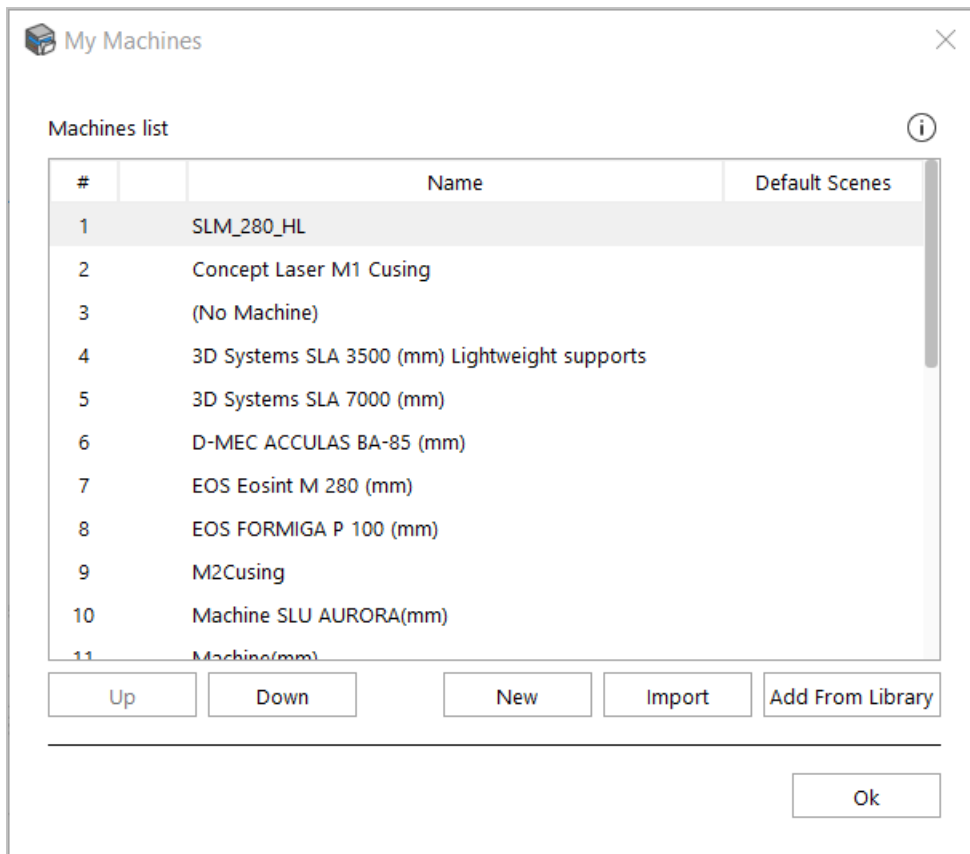
12. Baseplate Visibility



Toggle the visibility of the baseplate.

9.3. Automatic Support Generation

Automatic support generation can be done in different steps. The parameters for the support generation are defined like all other machine parameters and they are stored in support profiles.



Automatic support will create support surfaces based on the triangles of the part. For successful detection of the overhangs, the part needs to be fixed.

9.4. Machine Setup

Machine properties can be consulted by going to File/Machines/My Machines. Select the appropriate machine and then click with the right mouse button and select Edit option.

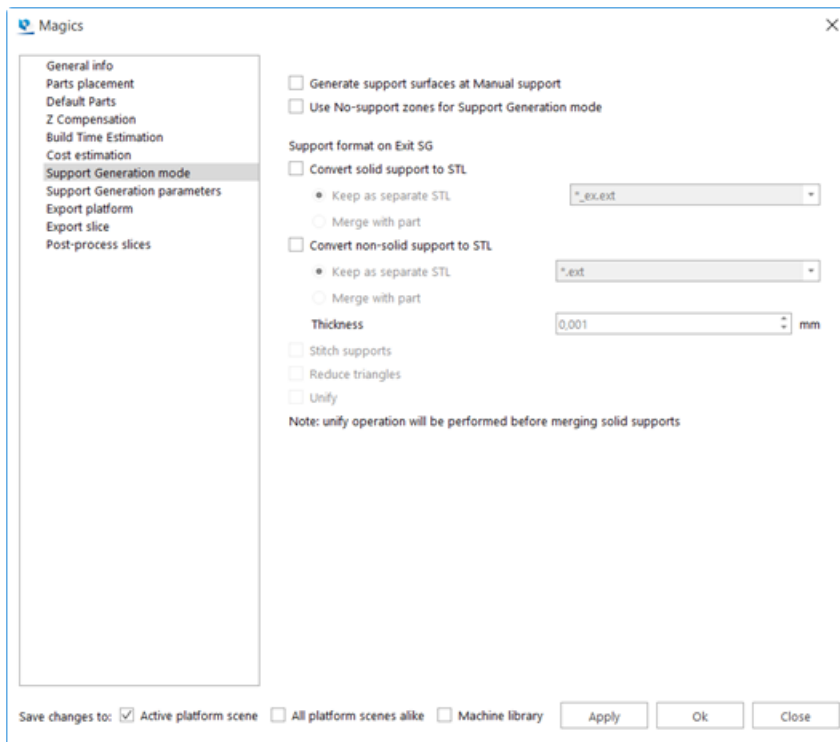
1. Build Time Estimation

- See **Build time estimation page, page 303**

2. Cost Estimation

- See **Cost estimation page, page 308**

3. Support Generation Mode



Generate support surfaces at Manual support is a useful feature for those who want to mark specific areas for support and do not want to wait too long for auto-generating of support surfaces which are not needed. Disable “Generate support surfaces at Manual support” to not calculate and generate these support surfaces in your support list.

If Use No-support zones for Support Generation mode is checked, No-support zones will be taken into account when generating the support surfaces.

Supports can be converted to STL on Exit SG. You can choose to create a separate STL with a specific naming convention or you can merge the STL with the STL of the part. There is the option to use various fixing operations.

4. Support generation parameters

Materialise LS Machine

General info
Parts placement
Default Parts
Z Compensation
Build Time Estimation
Cost estimation
Support Generation mode
Support Generation parameters
Materialise e-Stage support parameters
Export platform
Export slice
Post-process slices
Materialise e-Stage export
BuildCreator Module

Profile: Materialise LS Machine

Support type selection

Type: Non-solid

Point
 Point*
 Line
 Line*
 Block

Contour
 Web
 Gusset
 Combi

Default support type profiles

Type: Block

Profile: Block

Surface filters

Surface angle: 30,00 °

Max area: 1,000 mm²

Unconditional area filter

Min sharp edge detail: 0,100 mm

Max surface height: 10,000 mm

Automerge surfaces

Filter part to part support: Keep support on partial platform contact

Support type thresholds

Point support

Save changes to: Active platform scene All platform scenes alike Machine library

Apply Ok Close

Materialise LS Machine

General info
Parts placement
Default Parts
Z Compensation
Build Time Estimation
Cost estimation
Support Generation mode
Support Generation parameters
Materialise e-Stage support parameters
Export platform
Export slice
Post-process slices
Materialise e-Stage export
BuildCreator Module

Profile: Materialise LS Machine

Surface angle: 30,00 °

Max area: 1,000 mm²

Unconditional area filter

Min sharp edge detail: 0,100 mm

Max surface height: 10,000 mm

Automerge surfaces

Filter part to part support: Keep support on partial platform contact

Support type thresholds

Point support

Line support

Line* support

Gusset support

Max area: 10,000 mm²

Min thinness: 50,000

Max surface width: 12,000 mm

Point* on edge: 2,000 mm

No support on edge: 0,500 mm

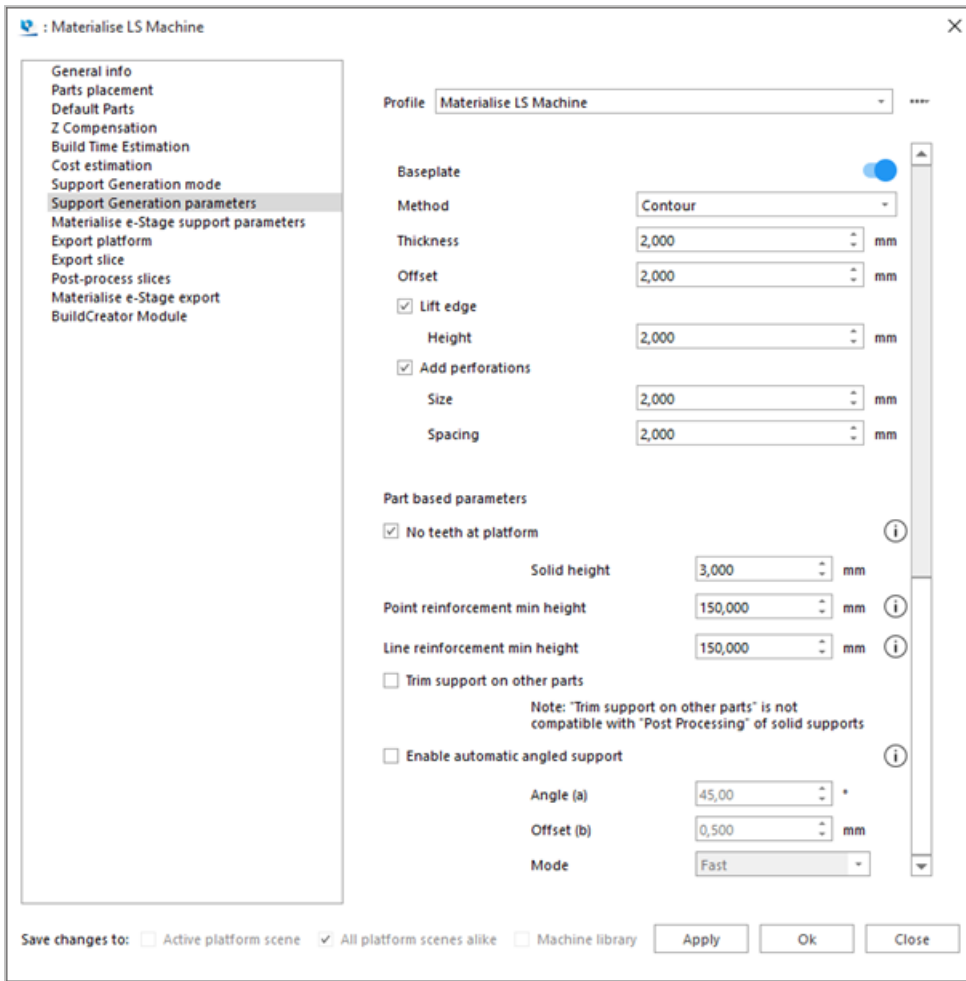
Min surface height: 100,000 mm

Search for remote walls

Max distance to wall (a): 10,000 mm

Save changes to: Active platform scene All platform scenes alike Machine library


Apply Ok Close

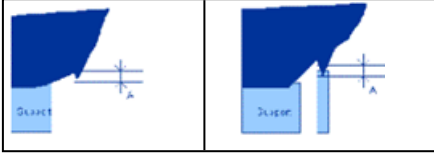


	Magics offers the possibility to manage different support parameters profiles for the same machine. This represents something useful when working with different kind of parts, different materials or different part size which might request different supports and parameters.	
Profile	Create new profile	Create a new support profile for the selected machine.
	Copy profile	Use an existing profile to create a new profile by copying. After copying the profile you can make the needed modifications. Make sure to save your modifications to the 'Machine library'. (Select 'Machine library' in the left down corner)
	Load profile	Load an already existing profile into your instance of Magics. Profiles generated on other systems can also be loaded.
	Rename profile	Add a new name for the profile
	Delete profile	Remove a profile from the list when it is no longer used
Support type	The user can define which types of support are generated automatically.	


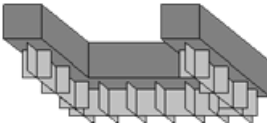
selection			
Default support type profiles	For every support type, you can define a default support type profile. These profiles can be created in Support Generation Mode.		
	Type	Select the type for which you want to choose the default.	
	Profile	The support type profiles in this dropdown, will be available in Support Generation Mode. The selected profile will be the default profile that is used.	
		Load profile from another machine	It is possible to load support type profiles that were created for another machine.
		Rename profile	Add a new name for the profile.
Delete profile	Remove a profile from the list when it is no longer used.		

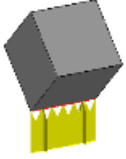
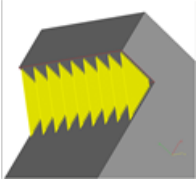
– Surface filters section

Surface angle	The Surface Angle defines which triangles are selected for support generation and which are not. It is defined as the angle between the horizontal plane and the surface. Surfaces whose angle to the horizontal plane is bigger than the selection angle are supposed to be self-supporting. This angle can be changed individually for each supported surface in the Support Generation Module (see Modifying Surfaces, Support, Types and Parameters).
Surface filter	Noise in the STL file can give rise to a huge amount of surfaces. The Surface Filter will filter out all surfaces which are smaller than this parameter, and which are supported by at least one other triangle. These surfaces will not be visible in the interactive support generator.
Unconditional surface filter	The unconditional surface filter will filter out all surfaces smaller than this parameter, also those that are not supported by other triangles.
Sharp edge filter	<p>Magics will support sharp down facing edges or points in the STL-file, in cases where there is no complete down facing surface available like for example in the following figures:</p>  <p>The surface area of a sharp edge support is 0. Consequently these sharp edges will be found at the end of the surface list in the Support Generation Module. This functionality is very sensitive to STL noise. STL noise can create a lot of situations in which sharp edge supports are placed. The Sharp Edge Filter will minimize the effect of STL noise and avoid</p>

	<p>unnecessary supports. The value of the Sharp Edge Filter determines the height of the details for which sharp edge supports will be generated. If this value is 0, all sharp edges will be supported. Typically you can set this parameter at the same value as the slice distance.</p> 
Maximum height	Surfaces higher than the maximum height will not be supported.
Automerge surfaces	Surfaces with small sharp triangles can be automerged.
Filter part to part supports	Part to part support will be filtered out with exceptions criteria, available options are: only keep support on partial platform contact, only keep support on full platform contact, only keep individual support segments on full platform contact, only keep individual support segments on full platform contact while avoiding internal support.

– Support type thresholds section



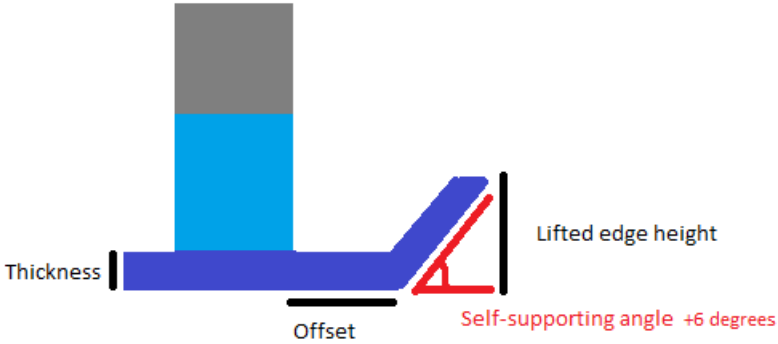
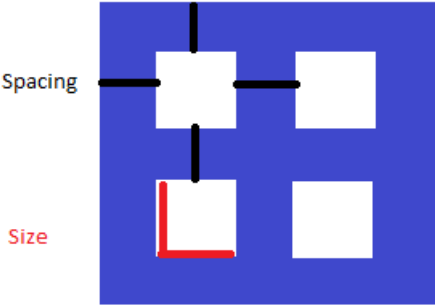
Point support	<p>A Point Support will typically be selected for very small surfaces, where a Block Support would be too small and unstable.</p> 	
	Max area	A Point Support will be selected if the surface is smaller than the 'Max Area' defined on this page.
Line support	<p>A line support is used for narrow down facing areas. This support type consists of a single wall down the center of the area and a number of crossing walls. A line support is not as strong, but easier to remove than classical block support. Since high line supports can become unstable, the user can reinforce them with an additional contour wall.</p> 	
	Min thickness	The thinness of the surface area must at least be equal to this value. The thinness indicates how narrow the shape of the surface is. It is calculated as the ratio of the square contour length and the area of the surface. With this parameter the user can influence the number of automatically generated line supports.
	Max surface width	In order to receive a line support automatically, the average width of the surface area may not exceed this value.


Line* support	Line* supports are a special kind of line supports. They support down facing edges. They do not support a surface (the surface is 0 mm ² and contains 0 triangles).	
		
	Point* on edge	When the edge is shorter than the given length, a point support will be set.
No support on edge	This filter will not put supports when the edge is shorter than the given length.	
Gusset support	Gussets will support on a sidewall, rather than on the platform or on a surface below.	
		
	Minimum surface height	Gussets are only interesting for surfaces that are positioned rather high (save resin). The surface must be higher than the Minimum Surface Height in order to receive a Gusset Support automatically.
Maximum distance to wall	This parameter determines how far the supporting wall can be from the supported surface.	
Search for remote walls	Create gusset supports between a surface and a wall that are not directly connected.	

– Baseplate

When enabled, a baseplate will automatically be generated for each part on the platform. This plate will also be printed and will be considered as support. It is often used in DLP printing, so parts can easily be removed from the platform, but it can also be useful in other technologies, like metal.

Switch	Toggles the baseplate on or off	
Method	Contour	The baseplate that is generated will roughly follow the contour of the parts.

	
Bounding Box	<p>The baseplate that is generated will be rectangular, according to the bounding box of the parts.</p> 
Thickness	<p>The thickness of the generated baseplate.</p> 
Offset	<p>The offset that will be given to the baseplate, from the edge of the parts.</p>
Lift edge	<p>When enabled, the edge of the baseplate will be lifted up. This can make it easier to remove from the platform.</p>
Height	<p>The height with which the edge is lifted.</p>
Add perforations	<p>When enabled, the baseplate will be perforated. This can be used to consume less material.</p> 
Size	<p>The size of the edge of the (square) perforations.</p>
Spacing	<p>The space between the perforations, as measured from the edge of one perforation to the edge of the next perforation.</p>

Baseplate Visibility button  can be found in the SUPPORT ribbon.

- Part based parameters section

No teeth at platform	Create teeth at platform at the bottom of a support when this is touching the platform, for easy support removal.	
	Solid height	When no teeth are generated, the solid height parameter determines a non-perforated support zone when perforations are applied to the support.
Point reinforcement min height	Point support reinforcement creates a box around the support; when a point support height is smaller than the defined value, the reinforcement will not be generated.	
Line reinforcement min height	Line support reinforcement creates a box around the support; when a line support height is smaller than the defined value, the reinforcement will not be generated.	
Trim supports on other parts	Supports will be trimmed with all other parts on the platform. It is advised to only turn this on if needed because it will slow down the generation of the supports.	

9.5. Support Parameters

1. General

Offset

TYPE **SUPPORT PARAMETERS** ...

Profile **Block** 📄 🔄 📄

Search parameters

- ▼ GENERAL
- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement

> BLOCK

(i)

XY offset mm

Z offset

Upper mm

Lower mm

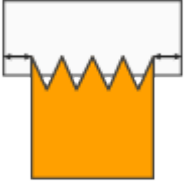
Vertical wall offset mm

No support

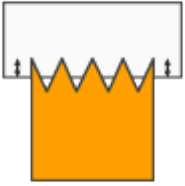
Wall offset (a) mm

Min height supporting wall (b) mm

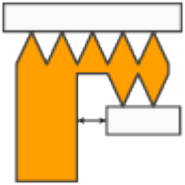
1. XY offset

<p>XY offset</p> 	<p>Determine the distance between the edge of the part and the border of the support.</p>
--	---

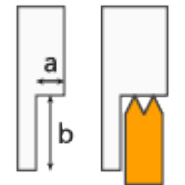
2. Z offset

<p>Z offset</p> 	<p>Determine the distance between the top and/or the bottom of the support and their respective surfaces. Positive values ensure the support penetrates the part. This is recommended to properly connect the support to the part.</p>
<p>Upper Z Offset</p>	<p>You can specify an offset in the supported surface.</p>
<p>Lower Z Offset</p>	<p>You can specify an offset in the supporting surface.</p>

3. Vertical wall offset

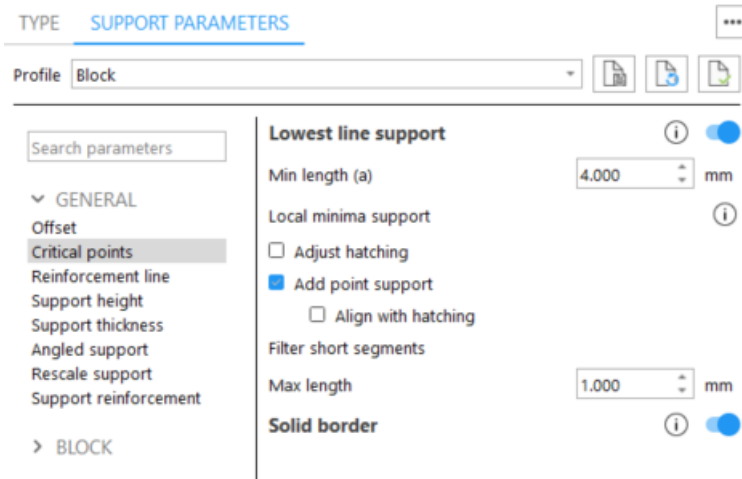
<p>Vertical wall offset</p> 	<p>In case a support is generated beside a vertical wall, the parameter Vertical Wall Offset makes sure a distance is kept between the vertical wall and the support. This avoids support being attached to the vertical wall and hence, unnecessary difficulties and marks on the part when removing the support.</p>
---	--

4. No support offset

<p>No support offset</p> 	<p>A (more or less) vertical wall gives support to another surface; in this case, no support needs to be generated for very small overhangs.</p> <p>Define the distance up till an overhang is self-supporting and the minimum height for a wall to be considered a supporting wall.</p>
<p>Wall offset (a)</p>	<p>Define the distance up till where the vertical wall is supporting.</p>

Min height supporting wall (b)	Define the minimum height of the wall to be a supporting wall.
--------------------------------	--

Critical points





1. Lowest line

<p>Lowest support line</p>	<p>Automatically generate an extra line of support to ensure the lower line (the imaginary line connecting the lowest points of the surface) is supported</p>
Min Length (a)	Lowest lines smaller than this length will be filtered out.

2. Local minima support

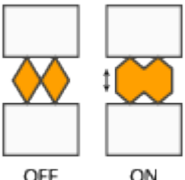
Local support minima	It can happen that some local minima are not supported. This can cause problems when building the part with some RP – techniques. When Local Minima is checked, a support will be placed. Local Minima is most convenient when building organic parts.
Adjust hatching	Move the hatching intersection to support the local minima (lowest points of the part)

	
<p>Add Point Support</p> 	<p>Add a point support to support local minima. When needed, rotate the point to match the hatching rotation.</p>
<p>Align with hatching</p>	<p>Will rotate the added point support to have the same angle of rotation as the hatching of the block support.</p>

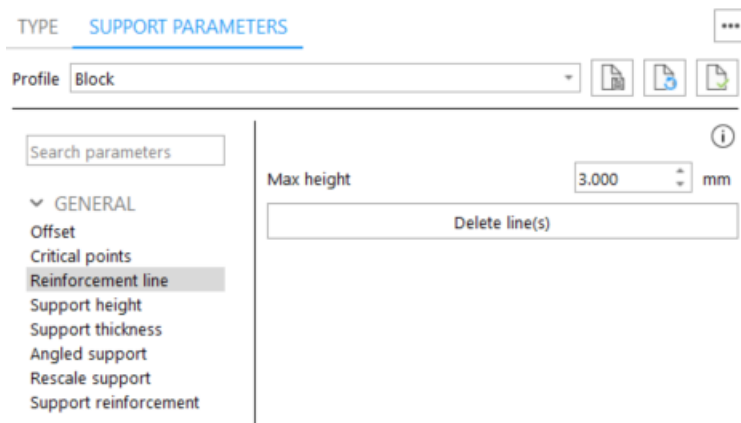
3. Filter short segments


Filter short segments	A support with a length less than the given one will be filtered out.
Max length	Supports shorter than max lengths will be filtered out.

4. Solid border

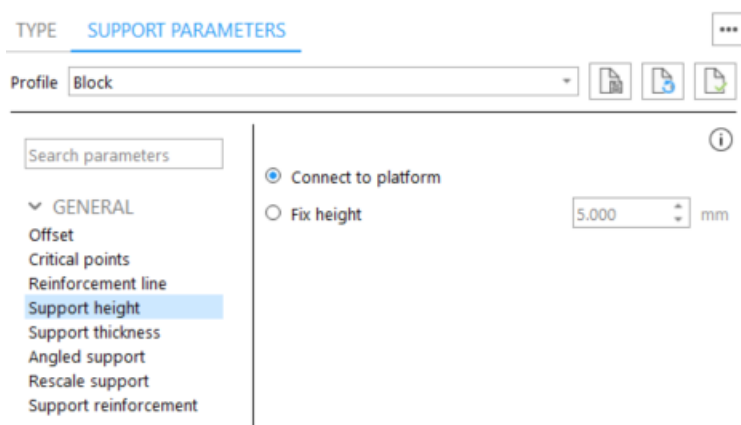
<p>Solid Border</p> 	<p>A border will connect support that otherwise would be loosely connected. It is used mostly for small supports with large teeth in comparison to the rest of the support.</p>
---	---

Reinforcement line

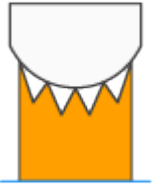
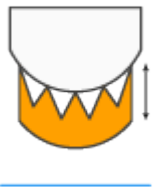


<p>Reinforcement line</p> 	<p>A line support that is manually drawn to reinforce an existing support that may be too fragile to stand on its own. To use this function, select "Draw reinforcement line" found in 2D edit tools.</p>
Max height	The Maximum height of the reinforcement line.
Delete line(s)	Will remove the drawn reinforcement line.




Support height



Support height	Adapt the height of the support
Connect to platform	The length of the support reaches till the platform

	
<p>Fixed height</p> 	<p>Customize the height of the support by defining a fixed height</p>

Support thickness

Profile **Block**   

Search parameters

- ▼ GENERAL
 - Offset
 - Critical points
 - Reinforcement line
 - Support height
 - Support thickness**
 - Angled support
 - Rescale support
 - Support reinforcement
- > BLOCK

General thickness

Upper teeth mm

Wall mm

Lower teeth mm

Separate thickness for block border

Upper teeth mm

Wall mm

Lower teeth mm

Separate thickness for lowest line

Upper teeth mm

Wall mm

Lower teeth mm

Thickness	Set the thickness for non-solid supports.
Upper teeth	The thickness of the upper teeth in mm.
Wall	The thickness of the wall in mm.
Lower teeth	The thickness of the lower teeth in mm.

Angled support

TYPE SUPPORT PARAMETERS

Profile Block

Search parameters

- GENERAL
 - Offset
 - Critical points
 - Reinforcement line
 - Support height
 - Support thickness
 - Angled support
 - Rescale support
 - Support reinforcement
- BLOCK
 - Hatching
 - Hatching teeth
 - Border
 - Border teeth
 - Fragmentation
 - Perforations
 - Heat sink

Support angle and rescale

Type Automatic angle

Interactive angle

Straight section

Top (a) 0.000 mm

Bottom (b) 0.000 mm

Shift (c) 0.000 mm

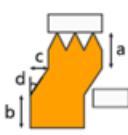
Angle (d) 0.00 °

Show current support

Preview support

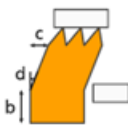
Regenerate 3D Regenerate 2D & 3D

Vertical angled support



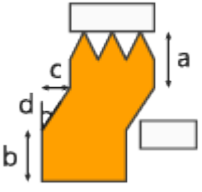
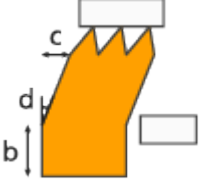
Angle a part to part support to ensure that it is not located on up-facing surfaces. Support will be angled starting from the lowest point of the surface, or from the defined distance (a) from the surface.

In-line angled support

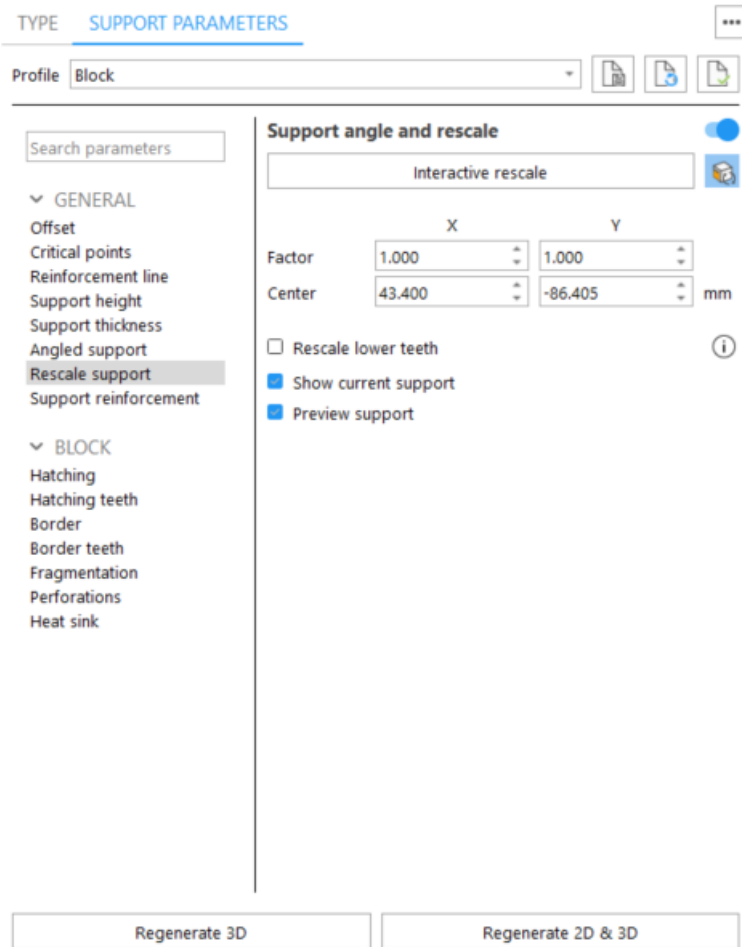


Angle a part to part support to ensure that it is not located on up-facing surfaces. Support will be angled from the connection point with the part.

Support angle and rescale	Create supports under a certain angle to prevent that the supports are located on up – facing surfaces
Vertical angled support	Angle a part to part support to ensure that it is not located on up-facing surfaces. Support will be angled starting from the lowest point of the surface, or from the defined distance (a) from the surface.

	
<p>In- line angled support</p> 	<p>Angle a part to art support to ensure that it is not located on up-facing surfaces. Support will be angled from the connection point with the part.</p>
<p>Top (a)</p>	<p>Define top straight part of the support.</p>
<p>Bottom (b)</p>	<p>Define bottom straight part of the support.</p>
<p>Shift (c)</p>	<p>Define the translation distance of the support on X and/or Y axis.</p>
<p>Angle (d)</p>	<p>Define the angle of support on X and/or Y axis.</p>
<p>Show current support</p>	<p>Shows current support before applying changes.</p>
<p>Preview support</p>	<p>Preview angled support without applying changes.</p>

Rescale support



Rescale support	Define a new projection area to be used when generating angled support.
Factor	Indicate on X and / or Y axis the multiplication factor of the projection area
Center	Indicate the center of scaling
Rescale lower teeth	Rescale support's lower teeth to synchronize the teeth number with the upper teeth.
Show current support	Shows current support before applying changes.
Preview support	Preview angled support without applying changes.

Support reinforcement

TYPE **SUPPORT PARAMETERS** ...

Profile **Block** 📄 📄 📄

Search parameters

GENERAL

- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement**

BLOCK

- Hatching
- Hatching teeth
- Border
- Border teeth
- Fragmentation
- Perforations
- Heat sink

Type **Manual** ⓘ

Border reinforcement

Direction from border ← → ↔

Reach (a) mm

Spacing (b) mm

Max height (c) mm

Min height (d) mm

Hatching reinforcement


Reach (a) mm

Spacing (b) mm

Max height (c) mm

Min height (d) mm

Regenerate 3D **Regenerate 2D & 3D**

<p>Support reinforcement</p> 	<p>Create a step-like base at the support bottom to increase its size and create a stronger connection with the platform.</p>
<p>Manual</p>	<p>Define parameters manually</p>
<p>Inner</p>	<p>Create extra support starting from the outer border, reaching inwards</p>
<p>Outer border</p>	<p>Create extra support starting from the outer border, reaching outwards</p>
<p>Internal</p>	<p>Create extra supports starting from the internal hatching, reaching both directions perpendicular to the hatching</p>
<p>Reach (a)</p>	<p>The distance between the existing support structure and the furthest newly created structure</p>
<p>Spacing (b)</p>	<p>The space between each line support</p>

Max height (c)	The height of the biggest “step”
Min height (d)	Set the minimum height of each line support
Automatic	Supports are automatically broadened during generation
Minimum length	All supports smaller than this minimum length will be broadened
Reach (a)	The distance between the existing support structure and the furthest newly created structure
Spacing (b)	The space between each line support
Max height (c)	The height of the biggest “step”
Min height (d)	Set the minimum height of each line support

2. Block

Hatching

TYPE **SUPPORT PARAMETERS** ...

Profile **Block** 📄 🔄 📄

Search parameters

- ▼ GENERAL
 - Offset
 - Critical points
 - Reinforcement line
 - Support height
 - Support thickness
 - Angled support
 - Rescale support
 - Support reinforcement
- ▼ BLOCK
 - Hatching**
 - Hatching teeth
 - Border
 - Border teeth
 - Fragmentation
 - Perforations
 - Heat sink

X hatching (a) mm

Y hatching (b) mm

Rotation angle (c) °

Filter hatching segments i

Min distance (d) mm

Min angle (e) °

Hatching levels

Number of levels

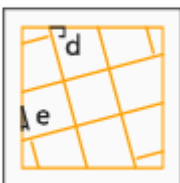
Hatching factor

Level height mm

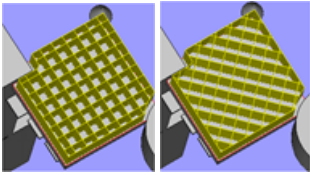

Overlap height mm

Solid borders mm

Regenerate 3D **Regenerate 2D & 3D**



Block support are made of a grid of X and Y lines, called hatching. Define the distance between these lines and the rotation angle around the Z axis.

X Hatching (a)	The distance between two X or Y hatching.
Y Hatching (b)	
Rotation Angle (c)	<p>The hatching on the first picture have no angle and they are parallel to the X- and Y-axes. On the second picture, they have an angle of 45°.</p> 
Filter hatching segments	<p>In some cases it might happen that very small hatches are created close to the support border. Remove the hatches that are closer to the border than the defined minimum distance and have an angle with the border smaller than the defined minimum angle.</p> 
Min distance (d)	Minimum distance of hatching stays away from the border.
Min angle (e)	Angle defines the minimum angle a rib has to make with the border in order not to be withdrawn with the clearance distance.

Hatching teeth

TYPE **SUPPORT PARAMETERS** ...

Profile **Block** 📄 🔄 📁

Search parameters

- GENERAL
 - Offset
 - Critical points
 - Reinforcement line
 - Support height
 - Support thickness
 - Angled support
 - Rescale support
 - Support reinforcement
- BLOCK
 - Hatching
 - Hatching teeth**
 - Border
 - Border teeth
 - Fragmentation
 - Perforations
 - Heat sink

Upper teeth ⓘ ☑

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Lower teeth ⓘ ☑

Use different values for lower teeth

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

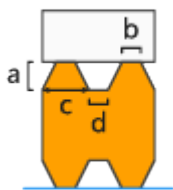
Keep teeth top in ends

Teeth synchronization ⓘ ☑

Teeth top at intersection

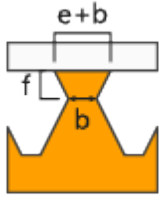
Teeth base at intersection

Upper teeth - You can specify whether you want Upper Teeth and/or Lower Teeth.



- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

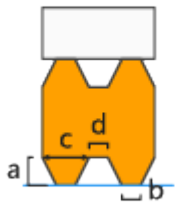
Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

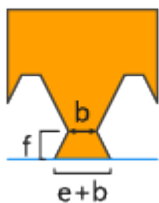
Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Lower teeth - Enable generation of lower teeth. Lower Teeth are only used if the support is trimming on another part. If the support is trimmed on the platform, there are no lower teeth.



- Use different values for lower teeth - Check it to set different values for lower teeth and break-off points of lower teeth
- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Keep teeth top in ends - You can decide to have a full tooth at the end of a support, instead of half a tooth.

Teeth synchronization - If switch is off, teeth won't be synchronized



Teeth top at intersection - Teeth of X and Y hatching intersect each other in the middle of the top of the teeth



Teeth base at intersection - Teeth of X and Y hatching intersect each other in the middle of the base of the teeth



Border

TYPE **SUPPORT PARAMETERS** ...

Profile **Block** 📄 🔄 📁

Search parameters

- GENERAL
- BLOCK
 - Hatching
 - Hatching teeth
 - Border**
 - Border teeth
 - Fragmentation
 - Perforations
 - Heat sink

Border ⓘ

Gusset border ⓘ


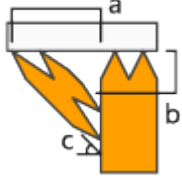
Length (a) mm

Interval mm

Notch (b) mm

Angle (c) °

Border	The borders of the support will be reinforced with a contour wall.
--------	--

 <p>OFF ON</p>	
Set separate thickness for border	Set a separate thickness for a non-solid border.
Upper teeth	Set the thickness of the upper teeth that reach the different upper surfaces.
Wall	Set the thickness of the wall in between the upper teeth and platform/lower teeth.
Lower teeth	Set the thickness of the lower teeth that touches lower surfaces.
Gusset border 	Create a gusset border to decrease the size of the block support. The gusset support will be attached to the surface and the block support.
Length	The length of the gusset is the length of the rib attached to the part.
Interval	The interval determines the distance between the gussets.
Notch	The notch parameter indicates how much the gusset is extracted from the corner.
Angle	The gusset angle, the angle between the overhang and the free border of the gusset, can be determined.

Border teeth

TYPE **SUPPORT PARAMETERS** ...

Profile **Block** 📄 🔍 📄

Search parameters

GENERAL
 Offset
 Critical points
 Reinforcement line
 Support height
 Support thickness
 Angled support
 Rescale support
 Support reinforcement

BLOCK
 Hatching
 Hatching teeth
 Border
Border teeth
 Fragmentation
 Perforations
 Heat sink

Upper teeth ⓘ

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Lower teeth ⓘ

Use different values for lower teeth

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

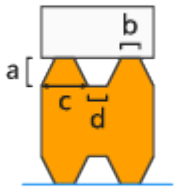
Top addition (e) mm

Waist Z-shift (f) mm

Keep teeth top in ends

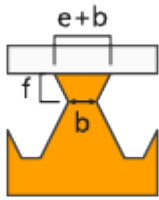
Regenerate 3D Regenerate 2D & 3D

Upper teeth - You can specify whether you want Upper Teeth and/or Lower Teeth.



- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

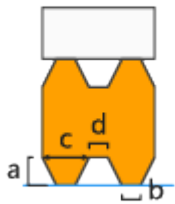
Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

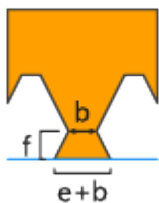
Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Lower teeth - Enable generation of lower teeth. Lower Teeth are only used if the support is trimming on another part. If the support is trimmed on the platform, there are no lower teeth.



- Use different values for lower teeth - Check it to set different values for lower teeth and break-off points of lower teeth
- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Keep teeth top in ends - You can decide to have a full tooth at the end of a support, instead of half a tooth.

Fragmentation

TYPE **SUPPORT PARAMETERS** ...

Profile **Block** 📄 🔄 📁

Search parameters

- ▼ GENERAL
 - Offset
 - Critical points
 - Reinforcement line
 - Support height
 - Support thickness
 - Angled support
 - Rescale support
 - Support reinforcement
- ▼ BLOCK
 - Hatching
 - Hatching teeth
 - Border
 - Border teeth
 - Fragmentation
 - Perforations
 - Heat sink

Hatching fragmentation ?

X interval (a) mm

Y interval (b) mm

Separation width (c) mm

Hatching fragmentation at crossing ?

X Interval (d) mm

Y Interval (e) mm

Separation width (f) mm

Border fragmentation ?

Separation width (g) mm

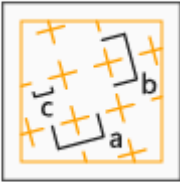

Fragmentate along the border

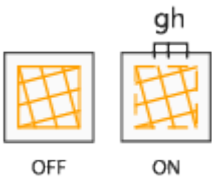
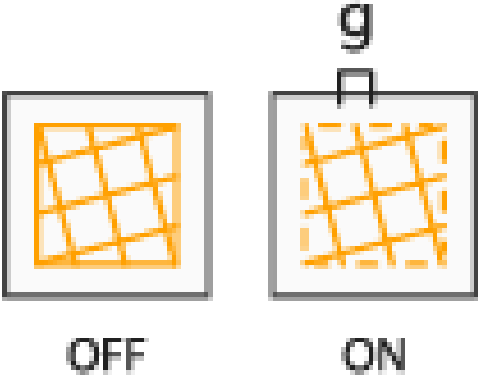
Interval (h) mm

Fragmentate at crossing with hatching

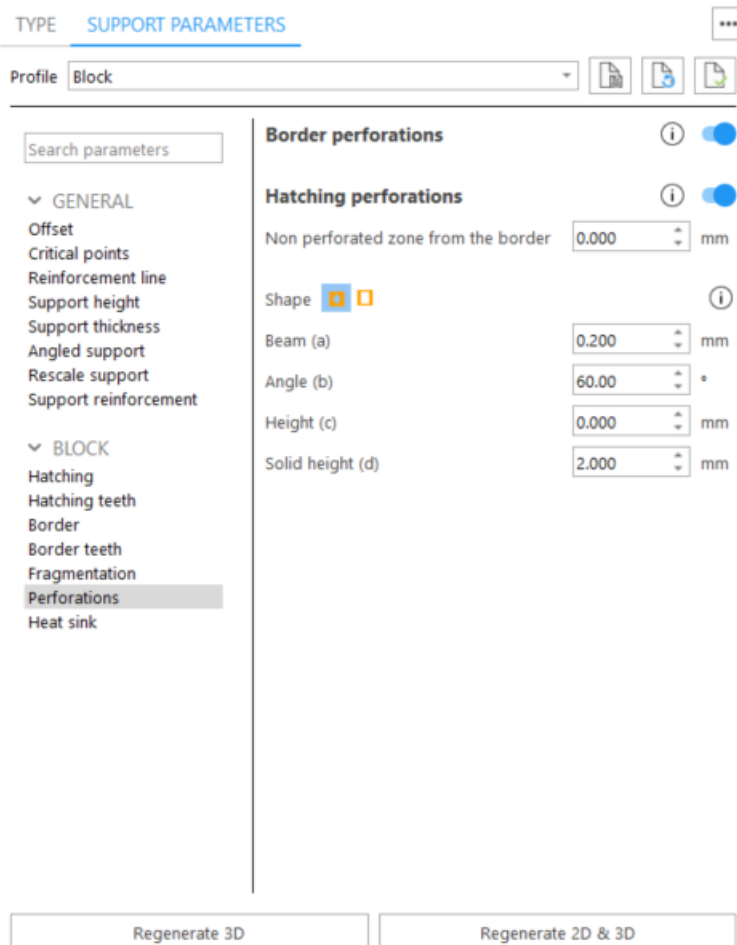
Regenerate 3D

Regenerate 2D & 3D

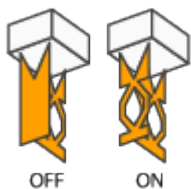
<p>Hatching fragmentation</p> 	<p>To facilitate the removal of the support, create gaps in the hatching by selecting fragmentation option</p>
<p>X Interval (a)</p>	<p>The interval of the gaps according the X direction.</p>
<p>Y Interval (b)</p>	<p>The interval of the gaps according the X direction.</p>
<p>Separation Width (c)</p>	<p>The width of the gaps in the hatching.</p>
<p>Hatching fragmentation at crossing</p> 	<p>To facilitate the removal of the support, create gaps at crossing of points X and Y hatching.</p>

X Interval (d)	The interval of the gaps according the X direction.
Y Interval (e)	The interval of the gaps according the X direction.
Separation Width (f)	The width of the gaps in the hatching.
Border fragmentation	Gaps in the border support will be generated
Separation Width (g)	Gaps width
Fragment along the border 	
Interval (h)	Distance between the end of a gap and the starting of the next
Fragment at crossing with hatching 	Gaps in the border support will be generated just where a crossing point (between hatching and support borders) is located

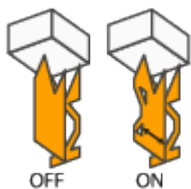
Perforations



Border perforations - To strengthen the support, do not add perforations for the border.

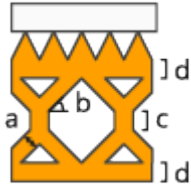


Hatching perforations - To strengthen the support, do not add perforations for the hatching.



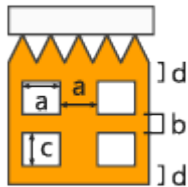
- Non perforated zone from the border - The hatching will not be perforated for the given distance, measured from the border to the inside. This will reinforce the support.

Shape - diamond



- Beam (a) - The beam thickness defines the thickness of the beams.
- Angle (b) - The perforation angle determines the angle of the perforations
- Height (c) - The height of the vertical part is set with this parameter.
- Solid Height (d) - The separate parameter Solid Height guarantees a good connection to the platform and the teeth.

Shape - rectangle



- Width (a) - Define the width of the rectangular holes.
- Interval (b) - Define the interval between the holes.
- Height (c) - Define the height of the rectangular holes
- Solid Height (d) - This parameter guarantees a good connection to the platform and the part.
- Perforate only (rows) - The support will only be perforated for the given rows, starting from the bottom. This enables the drainage of resin and ensures stable supports.

Trim hatching by border - If not checked, the hatching of the support are extended outside the border of the surface.

Heat sink

TYPE **SUPPORT PARAMETERS** ...

Profile **Block** 📄 🔍 📄

Search parameters

- ▼ GENERAL
- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement
- ▼ BLOCK
- Hatching
- Hatching teeth
- Border
- Border teeth
- Fragmentation
- Perforations
- Heat sink

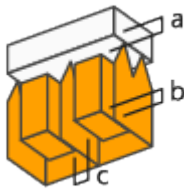
Solid volumes ⓘ 🔵

Offset

Upper (a)	<input type="text" value="2.000"/>	mm
Lower (b)	<input type="text" value="2.000"/>	mm
Inner (c)	<input type="text" value="2.000"/>	mm

Regenerate 3D

Regenerate 2D & 3D

<p>Solids supports</p> 	<p>Create solid support in between the already existing supports, to aid heat transfer from part to the platform</p>
Upper (a)	Distance between the part and heat transferring support
Lower (b)	Distance between the heat transferring support and the platform
Inner (c)	Distance between existing support and the heat transferring support

3. Line

Teeth

TYPE **SUPPORT PARAMETERS** ☰

Profile **Line** 📄 🔍 📄

Search parameters

GENERAL
 Offset
 Critical points
 Reinforcement line
 Support height
 Support thickness
 Angled support
 Rescale support
 Support reinforcement

LINE
Teeth
 Cross line
 Cross line teeth
 Perforations

Upper teeth ⓘ

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Lower teeth ⓘ

Use different values for lower teeth

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

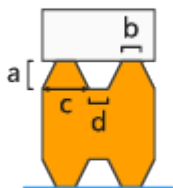
Top addition (e) mm

Waist Z-shift (f) mm

Keep teeth top in ends

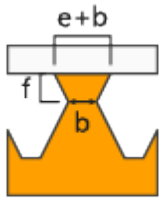
Regenerate 3D Regenerate 2D & 3D

Upper teeth - You can specify whether you want Upper Teeth and/or Lower Teeth.



- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

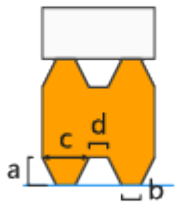
Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

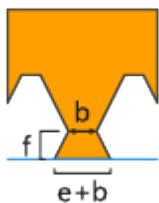
Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Lower teeth - Enable generation of lower teeth. Lower Teeth are only used if the support is trimming on another part. If the support is trimmed on the platform, there are no lower teeth.



- Use different values for lower teeth - Check it to set different values for lower teeth and break-off points of lower teeth
- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Keep teeth top in ends - You can decide to have a full tooth at the end of a support, instead of half a tooth.

Cross line

TYPE **SUPPORT PARAMETERS** ...

Profile **Line** 📄 📄 📄

Search parameters

- ▼ GENERAL
- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement
- ▼ LINE
- Teeth
- Cross line**
- Cross line teeth
- Perforations

Cross line ⓘ

Min rib length (a) mm

Max contact length (b) mm

Cross lines interval (c) mm


Sink cross lines ⓘ


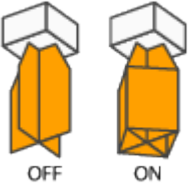
Distance mm

Line reinforcement ⓘ

Regenerate 3D

Regenerate 2D & 3D

	<p>Determine the length of the cross line, taking into account that large cross lines are more stable but harder to remove, and define the length of the contact between the cross line and the surface.</p>
Min Rib Length (a)	The length of the crossing lines.
Max Contact Length (b)	The contact length of the crossing lines with the support.
Cross lines interval (c)	The interval between two consecutive cross lines.
Sunken Cross Lines	To minimize the contact area of the support with the parts, the cross lines can be sunken, this means that they will not go till the part. They will stop at a certain distance before the part.

	
<p>Distance</p>	<p>The distance that a sunken cross line will stop before the part.</p>
<p>Line reinforcement</p> 	<p>You can decide to reinforce a support by adding an extra contour by checking the reinforcement checkbox. This is only done automatically if the height of the surface is bigger than the Reinforcement Height.</p>

Cross line teeth

TYPE **SUPPORT PARAMETERS** ...

Profile Line 📄 🔄 📁

Search parameters

- ▼ GENERAL
- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement
- ▼ LINE
- Teeth
- Cross line
- Cross line teeth
- Perforations

Upper teeth (i) 🔘

Height (a) 1.500 mm

Top length (b) 0.100 mm

Base length (c) 1.500 mm

Base interval (d) 0.200 mm

Add break off points

Top addition (e) 0.000 mm

Waist Z-shift (f) 0.001 mm

Lower teeth (i) 🔘

Use different values for lower teeth

Height (a) 1.500 mm

Top length (b) 0.100 mm

Base length (c) 1.500 mm

Base interval (d) 0.200 mm

Add break off points

Top addition (e) 0.000 mm

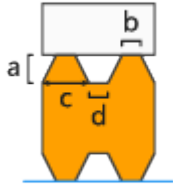
Waist Z-shift (f) 0.001 mm

Teeth synchronization (i) 🔘

Teeth top at intersection

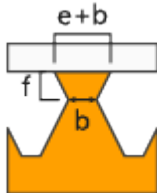
Teeth base at intersection

Upper teeth - You can specify whether you want Upper Teeth and/or Lower Teeth.



- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

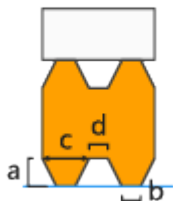
Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

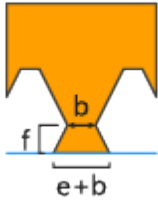
Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Lower teeth - Enable generation of lower teeth. Lower Teeth are only used if the support is trimming on another part. If the support is trimmed on the platform, there are no lower teeth.



- Use different values for lower teeth - Check it to set different values for lower teeth and break-off points of lower teeth
- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Teeth synchronization - If switch is off, teeth won't be synchronized



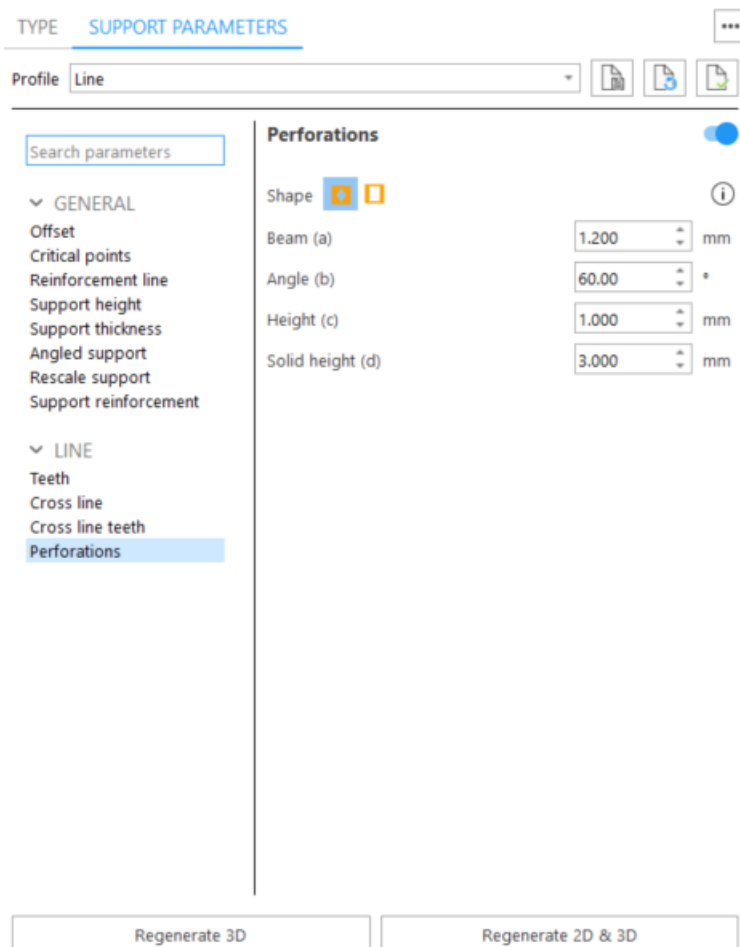
Teeth top at intersection - Teeth of X and Y hatching intersect each other in the middle of the top of the teeth



Teeth base at intersection - Teeth of X and Y hatching intersect each other in the middle of the base of the teeth

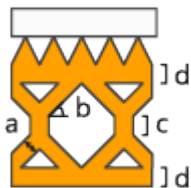


Perforations



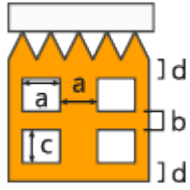
Perforations - To strengthen the support, do not add perforations for the border.

Shape - diamond



- Beam (a) - The beam thickness defines the thickness of the beams.
- Angle (b) - The perforation angle determines the angle of the perforations
- Height (c) - The height of the vertical part is set with this parameter.
- Solid Height (d) - The separate parameter Solid Height guarantees a good connection to the platform and the teeth.




Shape - rectangle



- Width (a) - Define the width of the rectangular holes.
- Interval (b) - Define the interval between the holes.
- Height (c) - Define the height of the rectangular holes
- Solid Height (d) - This parameter guarantees a good connection to the platform and the part.
- Perforate only (rows) - The support will only be perforated for the given rows, starting from the bottom. This enables the drainage of resin and ensures stable supports.

4. Line*

TYPE **SUPPORT PARAMETERS** ...

Profile **Line ***   

▼ GENERAL

- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement

▼ LINE*

- Teeth
- Cross line
- Cross line teeth
- Perforations

Upper teeth (i)

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Lower teeth (i)

Use different values for lower teeth

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Keep teeth top in ends

Regenerate 3D

Regenerate 2D & 3D

Line* is placed automatically under down-facing edges. It has the same parameters as 3 Line

5. Point

Rib

TYPE **SUPPORT PARAMETERS** ...

Profile **Point** 📄 🔄 📄

Search parameters

- GENERAL
 - Offset
 - Critical points
 - Reinforcement line
 - Support height
 - Support thickness
 - Angled support
 - Rescale support
 - Support reinforcement
- POINT
 - Rib**
 - Teeth

Rib ⓘ

Number of ribs: 2

Min rib length (a): 1.000 mm

Max contact length (b): 0.250 mm

Contact slope:

- Angle (c): 69.44 °
- Height (d): 1.000 mm

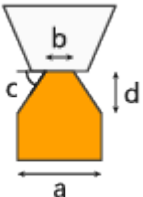

Sink ribs ⓘ

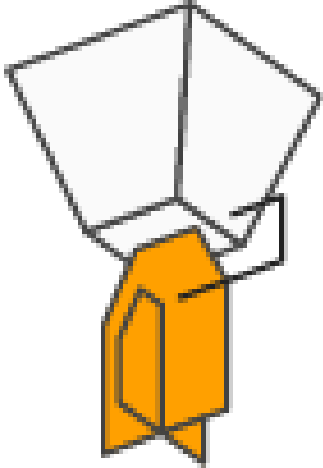
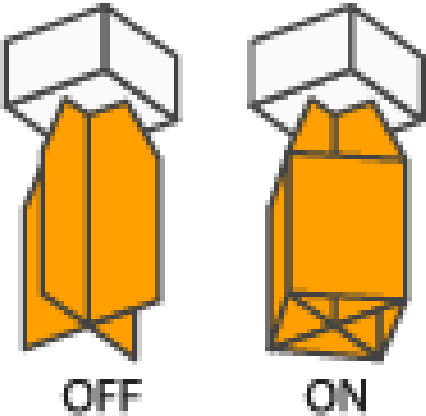
Number of sunken ribs: 0

Distance: 2.000 mm

Reinforce point support ⓘ

Regenerate 3D Regenerate 2D & 3D

	<p>Define the size of the support connection with the part and ensure support stability.</p>
<p>Number of ribs</p>	<p>Determine the number of ribs</p>
<p>Minimum Rib Length (a)</p>	<p>Determines the length of the ribs.</p> <div style="border: 1px solid #00aaff; padding: 10px; margin-top: 10px;"> <p> Note: you need a minimal length in order to have enough stability and to prevent the support from falling through the platform grid.</p> </div>
<p>Max contact length (b)</p>	<p>You also can define whether to support the complete surface or only a certain Contact Length.</p>

Contact slope - Angle (c)	You can choose the angle from the support to the part.
Contact slope - Height (d)	You can choose the distance from the support to the part.
Sink ribs	 <p>To minimize the contact area of the support with the parts, some ribs of the point support can be sunken. This means that they will not go till the part. They will stop at a certain distance before the part.</p>
Number of sunken ribs	Determine the number of the ribs to be sunk.
Distance	The distance that a sunken rib will stop before the part.
Reinforce point support	 <p>You can decide to reinforce a support by adding an extra contour.</p>

Teeth

TYPE **SUPPORT PARAMETERS** ☰

Profile **Point** 📄 🔄 📄

Search parameters

▾ GENERAL
 Offset
 Critical points
 Reinforcement line
 Support height
 Support thickness
 Angled support
 Rescale support
 Support reinforcement

▾ POINT
 Rib
Teeth

Lower teeth ℹ️ 🔵

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

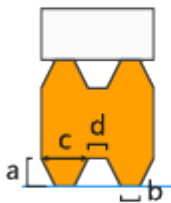
Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

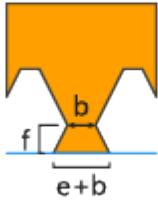
Regenerate 3D Regenerate 2D & 3D

Lower teeth - Enable generation of lower teeth. Lower Teeth are only used if the support is trimming on another part. If the support is trimmed on the platform, there are no lower teeth.



- Use different values for lower teeth - Check it to set different values for lower teeth and break-off points of lower teeth
- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

6. Point*

TYPE [SUPPORT PARAMETERS](#) ...

Profile Point 📄 📁 📁

▼ GENERAL

- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement

▼ POINT*

- Rib
- Teeth

Rib (i)

Number of ribs 2

Min rib length (a) 6.000 mm

Max contact length (b) 1.000 mm

Contact slope

Angle (c) 60.00 °

Height (d) 4.330 mm

Sink ribs (i)

Number of sunken ribs 1

Distance 1.016 mm

Reinforce point support (i)

Point* will be added automatically on down-facing vertices. Parameters are similar to 5 Point

7. Web

Web

TYPE **SUPPORT PARAMETERS** ...

Profile **Web** 📄 🔍 📄

Search parameters

- GENERAL
 - Offset
 - Critical points
 - Reinforcement line
 - Support height
 - Support thickness
 - Angled support
 - Rescale support
 - Support reinforcement
- WEB
 - Web**
 - Teeth


Number of ribs:

Manually calculate web radius

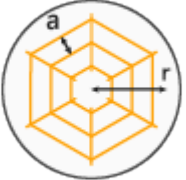
Radius (r): mm

Walls distance (a): mm

Regenerate 3D Regenerate 2D & 3D



Web support consists of a bundle of ribs, all directed to the center of the surface, and a number of circular walls that connect these ribs. With Automatic radius calculation option, the optimal radius for the web will be automatically defined.

	<p>Web support consists of a bundle of ribs, all directed to the center of the surface, and a number of circular walls that connect these ribs. With automatic radius calculation option, the optimal radius for the web will be automatically defined.</p>
<p>Number of ribs</p>	<p>This parameter specifies the number of ribs.</p>
<p>Manually calculate</p>	<p>If checked, define the radius of the web. If unchecked, the radius is</p>

web radius	automatically calculated - it is the local width decreased with the XY-offset value.
Radius (r)	This parameter specifies the length of the ribs from the center to the border of the surface.
Walls distance (a)	This parameter specifies the distance between 2 subsequent circular walls. It also determines the radius of the inner circular wall.

Teeth

TYPE **SUPPORT PARAMETERS** ...

Profile **Web** 📄 🔍 📄

Search parameters

GENERAL
 Offset
 Critical points
 Reinforcement line
 Support height
 Support thickness
 Angled support
 Rescale support
 Support reinforcement

WEB
 Web
Teeth

Upper teeth ⓘ

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Lower teeth ⓘ

Use different values for lower teeth

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

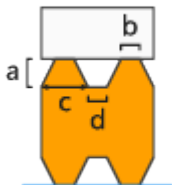
Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Keep teeth top in ends

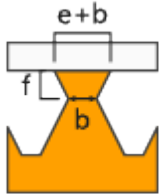
Upper teeth - You can specify whether you want Upper Teeth and/or Lower Teeth.



- Height (a)
- Top length (b)

- Base length (c)
- Base interval (d)

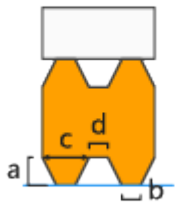
Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

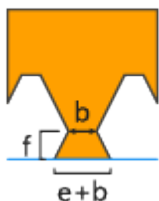
Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Lower teeth - Enable generation of lower teeth. Lower Teeth are only used if the support is trimming on another part. If the support is trimmed on the platform, there are no lower teeth.



- Use different values for lower teeth - Check it to set different values for lower teeth and break-off points of lower teeth
- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Keep teeth top in ends - You can decide to have a full tooth at the end of a support, instead of half a tooth.

8. Contour

Wall

TYPE **SUPPORT PARAMETERS** ⋮

Profile **Contour** 📄 📄 📄

Search parameters

▾ GENERAL
 Offset
 Critical points
 Reinforcement line
 Support height
 Support thickness
 Angled support
 Rescale support
 Support reinforcement

▾ CONTOUR
Wall
 Teeth
 Perforations

Walls distance (i) 1.500 mm


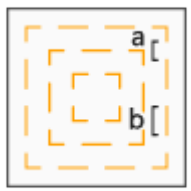
Fragmentation (i)

Separation width (a) 0.200 mm

Interval (b) 2.000 mm

Regenerate 3D

Regenerate 2D & 3D

<p>Walls distance</p> 	<p>Contour support consists of a bundle of walls placed one inside the other, that will follow the contour of the surface. Define the distance between two walls.</p>
<p>Fragmentation</p> 	<p>To facilitate the removal of the support, create gaps in the walls by selecting fragmentation option.</p>
<p>Separation width</p>	<p>Width of the gaps</p>

(a)	
Interval (b)	Distance between the end of a gap and the starting of the next one

Teeth

TYPE **SUPPORT PARAMETERS** ...

Profile **Contour** 📄 🔄 📁

Search parameters

GENERAL
 Offset
 Critical points
 Reinforcement line
 Support height
 Support thickness
 Angled support
 Rescale support
 Support reinforcement

CONTOUR
 Wall
Teeth
 Perforations

Upper teeth ⓘ

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Lower teeth ⓘ

Use different values for lower teeth

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

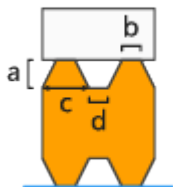
Top addition (e) mm

Waist Z-shift (f) mm

Keep teeth top in ends

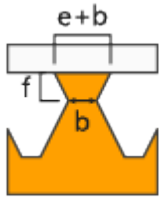
Regenerate 3D Regenerate 2D & 3D

Upper teeth - You can specify whether you want Upper Teeth and/or Lower Teeth.



- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

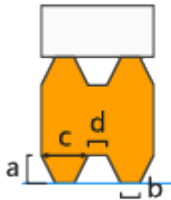
Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

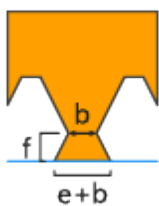
Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Lower teeth - Enable generation of lower teeth. Lower Teeth are only used if the support is trimming on another part. If the support is trimmed on the platform, there are no lower teeth.



- Use different values for lower teeth - Check it to set different values for lower teeth and break-off points of lower teeth
- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page

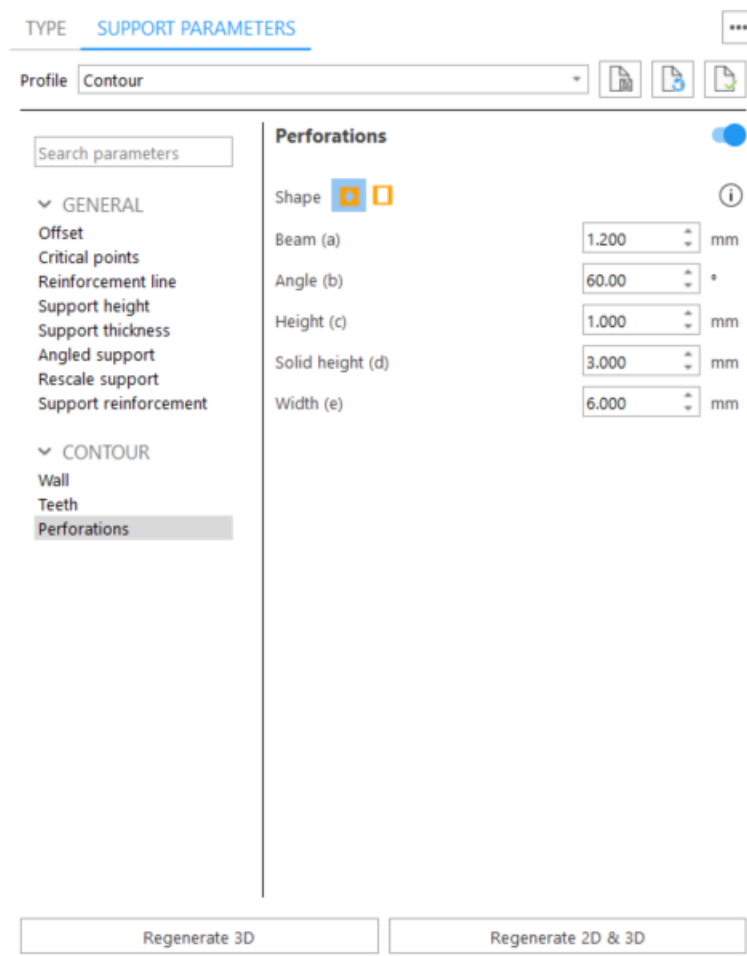


Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

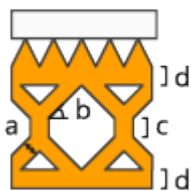
Keep teeth top in ends - You can decide to have a full tooth at the end of a support, instead of half a tooth.

Perforations



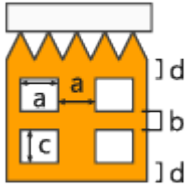
Perforations - To strengthen the support, do not add perforations for the border.

Shape - diamond



- Beam (a) - The beam thickness defines the thickness of the beams.
- Angle (b) - The perforation angle determines the angle of the perforations
- Height (c) - The height of the vertical part is set with this parameter.
- Solid Height (d) - The separate parameter Solid Height guarantees a good connection to the platform and the teeth.

Shape - rectangle



- Width (a) - Define the width of the rectangular holes.
- Interval (b) - Define the interval between the holes.
- Height (c) - Define the height of the rectangular holes
- Solid Height (d) - This parameter guarantees a good connection to the platform and the part.
- Perforate only (rows) - The support will only be perforated for the given rows, starting from the bottom. This enables the drainage of resin and ensures stable supports.

9. Gusset

Gusset

TYPE **SUPPORT PARAMETERS**

Profile **Gusset**

Search parameters

GENERAL
 Offset
 Critical points
 Reinforcement line
 Support height
 Support thickness

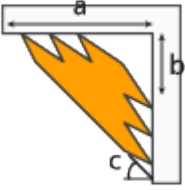
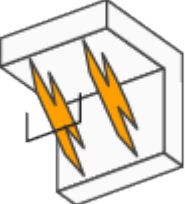
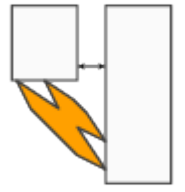
GUSSET
Gusset
 Teeth

Max length (a) 20.000 mm
 Min length (a) 0.000 mm
 Notch (b) 0.000 mm
 Angle (c) 45.00 °
 Gussets interval 1.000 mm

Remove gusset when self supporting
 Search for remote walls

Corner reinforcement

Regenerate 3D Regenerate 2D & 3D

	<p>Gusset supports are used to support overhangs which are large enough to have support, but not too large so that a Block Support is not needed. Gussets will support on a sidewall instead of on the platform or a lower surface.</p>
Maximal length (a)	The length of the gussets is the length of the rib attached to the part.
Minimal length (a)	If the overhang is smaller than the minimal length, support isn't necessary.
Notch (b)	The notch parameter indicates how much the gusset is extracted from the corner.
Angle (c)	The gusset angle, the angle between the overhang and the free border of the gusset, can be determined.
<p>Gusset interval</p> 	<p>The interval determines the distance between gussets.</p>
Remove when Self Supporting	The gusset support will be removed when the overhang is self-supporting.
<p>Search for remote walls</p> 	<p>Create gusset support between a surface and a wall that are not directly connected. Gusset support will be generated if the distance from the supporting wall to the end of the supported wall is smaller than the defined value in Machine Properties.</p>
Corner reinforcement	Will add a gusset on the corner of the part, gusset interval value will be ignored in this region.

Teeth

TYPE **SUPPORT PARAMETERS** ...

Profile **Gusset** 📄 🔍 📄

Search parameters

GENERAL
 Offset
 Critical points
 Reinforcement line
 Support height
 Support thickness

GUSSET
 Gusset
Teeth

Upper teeth ⓘ

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Lower teeth ⓘ

Use different values for lower teeth

Height (a) mm

Top length (b) mm

Base length (c) mm

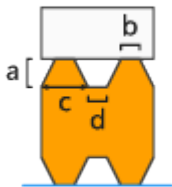
Base interval (d) mm

Add break off points

Top addition (e) mm

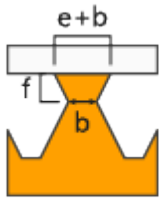
Waist Z-shift (f) mm

Upper teeth - You can specify whether you want Upper Teeth and/or Lower Teeth.



- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

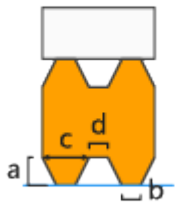
Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

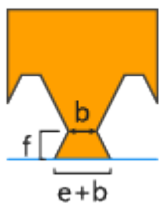
Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Lower teeth - Enable generation of lower teeth. Lower Teeth are only used if the support is trimming on another part. If the support is trimmed on the platform, there are no lower teeth.



- Use different values for lower teeth - Check it to set different values for lower teeth and break-off points of lower teeth
- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length




Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

10. Advanced Gusset

Advanced gusset

1. Advanced gussets

TYPE **SUPPORT PARAMETERS** ...

Profile   

- ▼ GENERAL
- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement
- > LINE
- ▼ ADVANCED GUSSET
 - Advanced gusset
 - Teeth

Length (a) mm

Trim on surface borders

Notch (b) mm

Angle (c) °

Top straight section (d) mm

Gussets interval mm

Rotation angle °

Reinforcement

Min lower connection height mm

Interval

Skipped gussets at start

Delete advanced gusset

Length (a)	The length of the gussets is the length of the rib attached to the part.
Notch (b)	The notch parameter indicates how much the gusset is extracted from the corner.
Angle (c)	The gusset angle, the angle between the overhang and the free border of the gusset, can be determined.
Top straight section (d)	Top section that is not angled, including teeth.
Gusset interval	Space in between different gussets
Rotation angle	Rotation of gussets around Z axis
Reinforcement	Create reinforcements for manually created gussets
Min lower connection height	The building height of support must exceed an adjustable threshold height
Interval	Space in between the reinforcement gussets
Skipped gussets at start	Start adding reinforcement gussets after skipping the specified amount.

Delete advanced gusset	Will permanently remove drawn advanced gussets
------------------------	--

2. Teeth

TYPE **SUPPORT PARAMETERS** ...

Profile Line 📄 🔄 📁

▼ GENERAL

- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement

> LINE

▼ ADVANCED GUSSET

- Advanced gusset
- Teeth

Upper teeth (i)

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Lower teeth (i)

Use different values for lower teeth

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

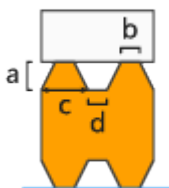
Waist Z-shift (f) mm

Keep teeth top in ends

Regenerate 3D

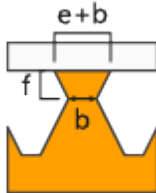
Regenerate 2D & 3D

Upper teeth - You can specify whether you want Upper Teeth and/or Lower Teeth.



- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

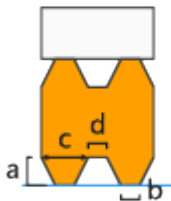
Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

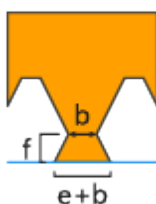
Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Lower teeth - Enable generation of lower teeth. Lower Teeth are only used if the support is trimming on another part. If the support is trimmed on the platform, there are no lower teeth.



- Use different values for lower teeth - Check it to set different values for lower teeth and break-off points of lower teeth
- Height (a)
- Top length (b)
- Base length (c)
- Base interval (d)

Break-off points - Create break-off points on teeth to break the supports on these points and easily remove it. Note that **b** value corresponds to the Top length parameter defined in Teeth page



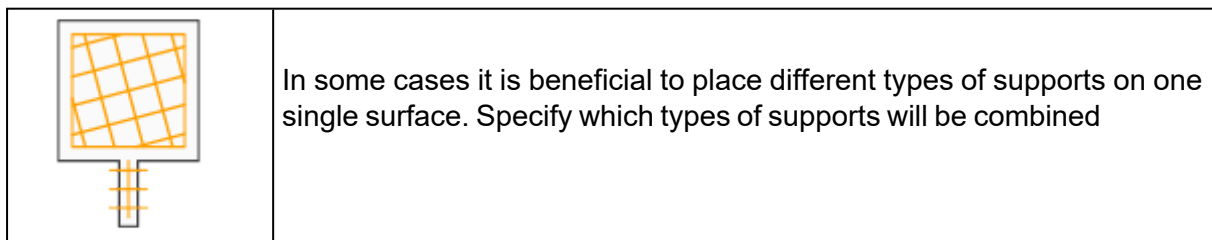
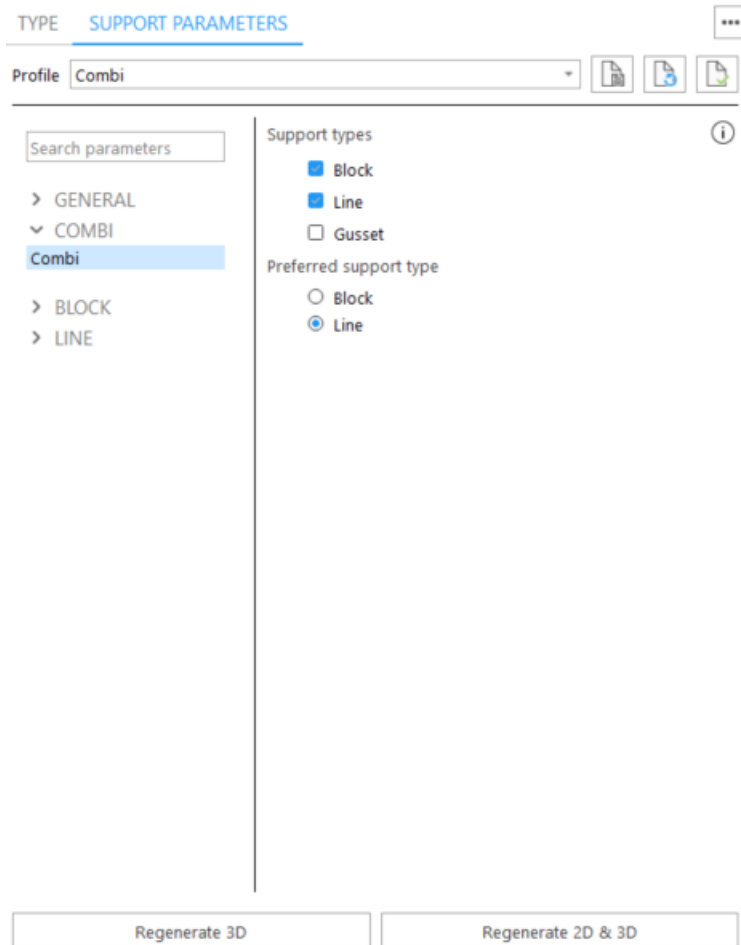
Top addition (e) - Broaden your teeth on top. Total length = Top addition + Top length

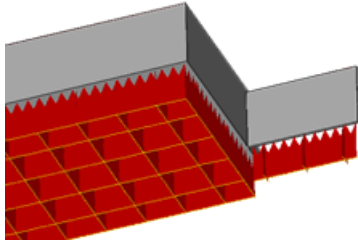
Waist Z-shift (f) - Move the break-off point of the tooth with this parameter in the Z-direction

Keep teeth top in ends - You can decide to have a full tooth at the end of a support, instead of half a tooth.

11. Combi

Combi






Block	<p>You can specify which types of supports will be combined. In the support generation mode you can also alter the parameters of the different support types. The default values are taken from the respective base support types.</p> 
Line	
Gusset	
Prefer Block	<p>It can be selected which support is preferred: block or line support.</p>
Prefer Line	

12. Volume

Volume

1. Volume

TYPE **SUPPORT PARAMETERS** ⋮

Profile   

- ▼ GENERAL
- Offset
- Support height
- Angled support
- Rescale support
- ▼ VOLUME
- Volume**
- Post-processing

Fragmentation (i)

Invert fragmentation

X Interval (a) mm

Y Interval (b) mm

Separation width (c) mm

Rotation angle (d) °

Start from Z height mm



Remove intersections (i)

Regenerate 3D

Regenerate 2D & 3D

Postprocess

Fragmentation	<p>When fragmenting volume support you avoid large cross sections in your support, this won't result in high thermal stress and increases build successes.</p> <p>Fragmentation can also be rotated, to avoid fragmentation slots which are parallel to the recoater of the machine.</p>
---------------	--

	
<p>Invert fragmentation</p> 	<p>X interval (a) Y interval (b) Separation width (c) Rotation angle (d)</p>

2. Post processing

Shell

TYPE **SUPPORT PARAMETERS** ...

Profile **Volume** 📄 🔄 📄

Search parameters

- ▼ GENERAL
 - Offset
 - Support height
 - Angled support
 - Rescale support
- ▼ VOLUME
 - Volume
 - Post-processing**

Shell 🔵

Wall Thickness mm

Detail Size mm

Internal lattice 🔵

Structure

X mm

Y mm

Z mm

Sample data to process: 898.1 MB

Shell	When selected the support will be hollowed and receive a shell	
	Wall Thickness	Thickness of the shell
	Detail Size	Allowed deviation for the wall thickness
	Sample data to process	Amount of memory necessary to perform this hollow operation

Internal lattice

Internal lattice	When selected the support will be converted to a lattice	
	Structure	The list shows the library of unit structures that can be used to create the lattice. The location of the library is given in your settings. One can manually add new structures to be used for the lattice creation
	X	Size of unit structure in X direction
	Y	Size of unit structure in Y direction
	Z	Size of unit structure in Z direction
	Sample data to process	Amount of memory necessary to perform this hollow operation

13. Tree

Tree

1. Spacing

TYPE **SUPPORT PARAMETERS** ...

Profile **Tree** 📄 📄 📄

GENERAL
 Offset
 Critical points
 Angled support

TREE
Spacing
 Branch
 Trunk
 Post-processing

Min distance between points mm
 Min distance between rows mm
 Max distance between points mm
 Place branches on surfaces smaller than the contact point

Allow only changes that preserve support structure
 Visualize post-processing

Min distance between connection points	Space between connection points
Min distance between rows	Space between rows of connection points
Max distance between	The maximum space between connection points

connection points	
Place branches on surfaces smaller than the contact point	When checked, will add branches even if the surface is smaller than the set top diameter of branch.
Allow only changes that preserve support structure	When checked will disable parameters that would regenerate entire structure, preserving manual changes to the tree.

2. Branch

TYPE **SUPPORT PARAMETERS**

Profile **Tree**

Search parameters

- GENERAL
 - Offset
 - Critical points
 - Angled support
- TREE
 - Spacing
 - Branch**
 - Trunk
 - Post-processing

Max branches per trunk: 1

Size

Diameter top (d1): 0.300 mm

Diameter bottom (d2): 0.500 mm

Break off point

Diameter (d3): 0.400 mm

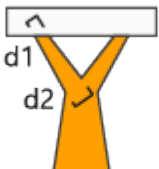
Distance from top (a): 5.000 mm

Break off point location

- On branch line
- Triangle normal
- Vertical

Allow only changes that preserve support structure

Regenerate

	Define the branch top and bottom diameters
Max branches per trunk	Number of maximum branches for each of the generated trunk.
Diameter top (d1)	Diameter of the branch at the top (where connected to the part)

Diameter bottom (d2)	Diameter of the branch at the bottom (where connected to the trunk)
Add break-off point	Add break off point at the extremity of your branch in contact with the part
Diameter (d3)	Break off point diameter
Distance from top (a)	Distance between your break off point and the part

3. Trunk

TYPE **SUPPORT PARAMETERS** ...

Profile **Tree** 📄 📄 📄

Search parameters

- GENERAL
 - Offset
 - Critical points
 - Angled support
- TREE
 - Spacing
 - Branch
 - Trunk**
 - Post-processing

Size ⓘ

Diameter top (d1) mm

Diameter bottom (d2) mm

Trunk height

Calculate automatically

Set max height mm

Set branch max angle °

Braces

Diameter (d3) mm

Generate automatically

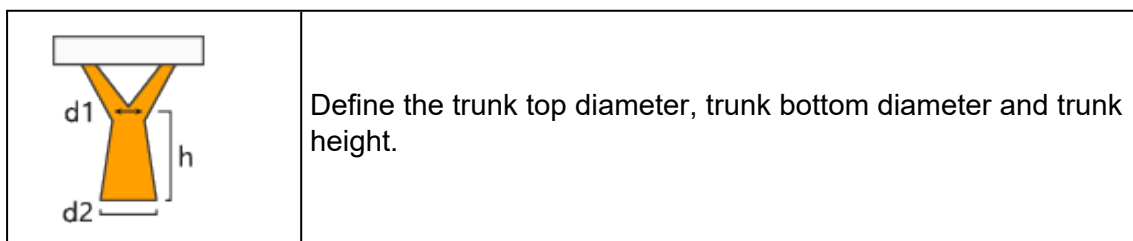
Offset from top and bottom (a) mm

Pattern

Zigzag

Cross

Allow only changes that preserve support structure



Diameter top (d1)	Diameter of the trunk at the top
Diameter bottom (d2)	Diameter of the trunk at the bottom
Calculate automatically	Calculates automatically the height of the trunk
Set max height	Set manual max height of the trunk
Set branch max angle	Set manual max angle of the branch

4. Critical points

TYPE **SUPPORT PARAMETERS** ...

Profile **Tree** 📄 🌐 📄

Search parameters

GENERAL
 Offset
Critical points
 Angled support

TREE
 Spacing
 Branch
 Trunk
Post-processing

Lowest line support

Min spacing mm

Min length mm

Allow only changes that preserve support structure

Lowest Line support	When lowest line is checked, an extra connection points will be placed so that this lowest line is correctly supported.
Minimal spacing	Defines the minimal spacing between the connection points along the lowest line.
Minimum length	Lowest lines smaller then this value (length) will be filtered out

5. Post processing

Parameters are similar to Volume support

TYPE **SUPPORT PARAMETERS** ...

Profile **Tree** 📄 🔄 📄

Search parameters

- ▼ GENERAL
 - Offset
 - Critical points
 - Angled support
- ▼ TREE
 - Spacing
 - Branch
 - Trunk
 - Post-processing**

Shell 🔵

Wall Thickness mm

Detail Size mm

Internal lattice ⚪

Structure

X mm

Y mm

Z mm

Sample data to process: 4.2 MB

Allow only changes that preserve support structure

Visualize post-processing

Visualize
post
processing

When one of the modes (Shell, Internal lattice) is switched on, post processing will be applied. If checked, post-processing will be executed in SG mode, if unchecked - post processing will be executed at exiting SG mode.



14. Tree*

TYPE **SUPPORT PARAMETERS** ⋮

Profile **Cones/Trees *** 📄 🔄 📄

- GENERAL
 - Offset
 - Critical points
 - Angled support
- TREE***
 - Spacing
 - Branch**
 - Trunk
 - Post-processing

Max branches per trunk

Size ⓘ

Diameter top (d1) mm

Diameter bottom (d2) mm

Break off point ⓘ

Diameter (d3) mm

Distance from top (a) mm

Break off point location

- On branch line
- Triangle normal
- Vertical

Tree* will be placed automatically under down-facing edges. Parameters are similar to 13 Tree

15. Hybrid

Hybrid

TYPE **SUPPORT PARAMETERS** ...

Profile **Hybrid** 📄 📄 📄

- ▼ GENERAL
 - Offset
 - Critical points
 - Support thickness
- ▼ HYBRID
 - Hybrid**
- ▼ BLOCK
 - Hatching
 - Hatching teeth
 - Border
 - Border teeth
 - Fragmentation
 - Perforations
 - Heat sink
- ▼ TREE
 - Spacing
 - Branch
 - Trunk
 - Post-processing

Upper support height mm

Middle plate thickness mm

Upper support height	Defines the height of the upper part of the hybrid support structure
Middle plate thickness	Defines the height of the middle part of the hybrid support structure

16. Support on graphs

This feature is only available in combination with the Magics Lightweight Structures license. Supports will be automatically generated on graphs coming from 3-matic. All connected graphs will be treated as one support ID in the support list.

Support on graphs

TYPE **SUPPORT PARAMETERS** ...

Profile **Support on graphs** 📄 🔄 📄

Search parameters

▼ GENERAL

Offset

Support thickness

▼ SUPPORT ON GRAPHS

Support on graphs

Size

Support type

Point

Tree

Spacing 1.000 mm

Angle filter 15.00 °

Support graph edges

Max unsupported edge length 1.000 mm

Always support local minima

Regenerate 3D

Regenerate 2D && 3D

Support type: Point / Tree	You can use points or trees to support the graphs.
Spacing	Defines the space in between the points or trees.
Angle filter	Support is only wanted on the outer side of the structure. The angle filter is used to determine whether a point can be considered as internal. From every point, we draw a cone like shape (depending on the angle filter) down and see if this is intersecting with the structure. When it is intersecting we filter it out from the set of points that need support.
Support on graph edges	When the checkbox is deselected, only the endpoints of graphs will be considered as points that need support. When it is selected, intermediate support points will be created.
Max unsupported edge length	Is used to determine if and how many intermediate support points will be created on a graph.
Always support local minima	Local minima will always be supported independently of the spacing parameter.

Size

TYPE **SUPPORT PARAMETERS**

Profile **Support on graphs**

Search parameters

- GENERAL
 - Offset
 - Support thickness
- SUPPORT ON GRAPHS
 - Support on graphs
 - Size

Contact part (r1) mm

Contact platform (r2) mm

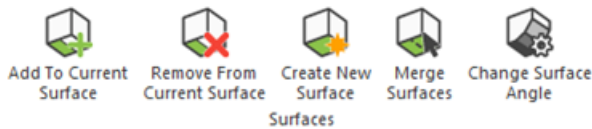
Regenerate 3D Regenerate 2D && 3D

Contact part (r1)	Diameter of the contact point to the part of the Tree type support.
Contact platform (r2)	Diameter of the Tree node connected to the platform.

9.6. Modifying Surfaces

1. Modifying the surfaces

In the ribbon of the support mode you can find tools to modify the selected surfaces.



Add Current Surface	To		The marked triangles will be added to the current surface. When you press Regenerate 2D & 3D, Magics will also generate support for these triangles.	
	Begin situation	Triangles to be added are marked	Triangles are added to support	
Remove From Current Surface	Begin situation		The marked triangles will be removed from the current surface. When you click Regenerate 2D & 3D, Magics won't generate supports anymore for the marked triangles.	
		Triangles to be removed are marked	Triangles are removed from surface	



Create New Surface	The marked triangles will become a new surface, which you can support.		
	Begin situation 	Triangles to be added as new surface are marked	Triangles are added as new surface
Merge Surfaces	Magics will merge the current surface with a given support.		
	Begin situation 	The middle surface will be merged with surface nr 1 	The surfaces are merged
Change surface angle	When a surface makes an angle smaller than the entered surface angle it is considered not self – supporting, so support is generated for the respective surface.		

2. Part Info page

– See **Part Info page, page 585**

3. Surface Info page

SUPPORT LIST **SURFACE INFO** PART INFO ☰

Support ID << < 1 > >> Skip Empty

DIMENSIONS

	Min	Max	Delta	
X	23.047	89.899	66.852	mm
Y	-159.675	-100.377	59.297	mm
Z	9.014	9.014	0.000	mm

SURFACE PROJECTION

Contour length	196.774 mm
Surface	2425.539 mm ²
Thickness	15.963

Dimensions	Min X Y Z	The dimensions of the surface.
	Max X Y Z	
	Delta X Y Z	
Contour Length	The contour length of the surface.	

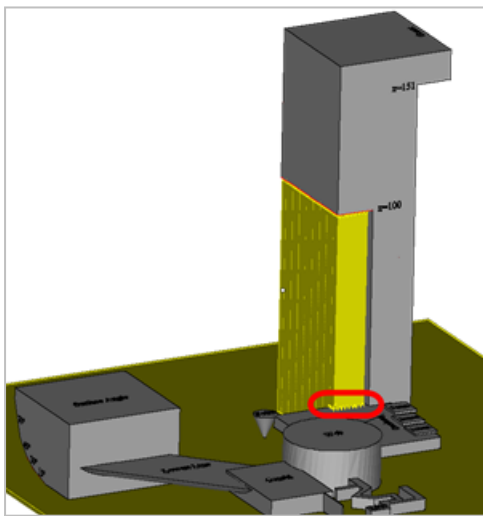
Surface	The surface area of the surface.
Thinness	This parameter represents the thinness (slenderness) of the surface.
Browsing buttons	Browsing buttons are used to browse through the different surfaces. The buttons are explained above.

4. Support List page

The support list gives you an overview of all surfaces created with their support type. It is also possible to enable a filter for the support list to only visualize supports per type.

ID	Type	Triangle	Z Max	Surface Area	On Part	Surface Profile
1	Line	181	31.396	593.335	No	Default
2	None	494	40.286	125.198	No	Default
24	None	60	44.947	1.251	No	Default
25	None	60	44.947	1.251	No	Default
26	None	60	44.947	1.251	No	Default

The red items in the list indicate that the support touches a lower surface of the part itself.



The columns of the Surface List can be selected by right-clicking on the title of the column; the following dialog will pop up:

- ✓ ID
- ✓ Type
- ✓ Triangle
 - X Min
 - Y Min
 - Z Min
 - X Max
 - Y Max
- ✓ Z Max
- ✓ Surface Area
 - Border Length
- ✓ On Part
- ✓ On Platform
- ✓ Surface Profile

ID	The unique number of the surface
Type	The support type given to this surface.
Triangles	Amount of triangles of the surface.
X Min	The starting dimensions of the surface.
Y Min	
Z Min	
X Max	The ending dimensions of the surface.
Y Max	
Z Max	
Surface Area	The surface area of the surface.
Border Length	The border length of the surface.
On Part	The support surface is attached to the part.
On Platform	The support surface is attached to the platform.
Surface Profile	A support profile, created by the user, is used.

A right mouse click in the surface list will pop up the following:

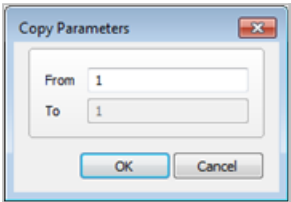
- Invert Selection
- Select All
- Select None

- Delete Surfaces
- Duplicate Surfaces

- Copy Parameters From
- Merge Selected Surfaces

- Save Order

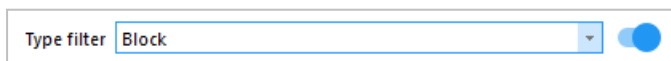
Invert Selection	The unselected surfaces will become selected and the selected surfaces
------------------	--

	will not be selected anymore.
Select All	All surfaces will be selected.
Select None	All surfaces will be deselected.
Delete Surface	All selected surfaces will be removed from the surface list.
Duplicate Surfaces	The selected surface will be duplicated. This gives the possibility to use different support types for one surface. (e.g. Volume support in combination with block support for the metal market)
Copy Parameters From	 <p>Copy the support type and all parameters from the given support number to the selected support numbers.</p>
Merge Selected Surfaces	The selected surfaces will be merged into one surface.
Save Order	The order is saved.

Browsing through the supports

The complete down facing surface will be split into multiple surfaces, which each have their support structure. You can browse through these surfaces in two ways :

- By enabling the type filter to visualize only the supports of a desired type
- By using the navigation buttons

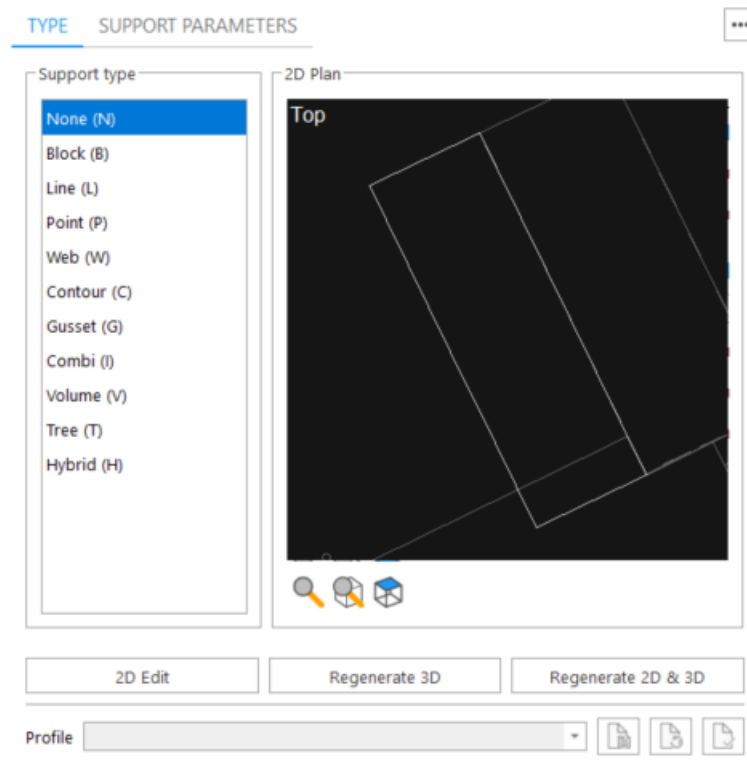


Type filter	This is a dropdown list. Click on the field to open the list of support types to choose from. Only the created support types will be visualized in the list, in alphabetical order.
Switch	Click once to enable the filter, and click again to disable it.

Support ID << < 1 > >> Skip Empty

<<	Return to surface one.
<	Return to the previous surface.
1	Window showing surface number.
>	Go to the next surface.
>>	Go to the last selected surface.
Skip Empty Supports	All surfaces which are selected by the surface angle but do not need supports following the selection parameters (no support offset, surface filter...), are skipped.

5. Support Toolbox



The Support Toolbox is a dynamic box, when selecting a surface the box will display the Toolpages regarding the selected surface.

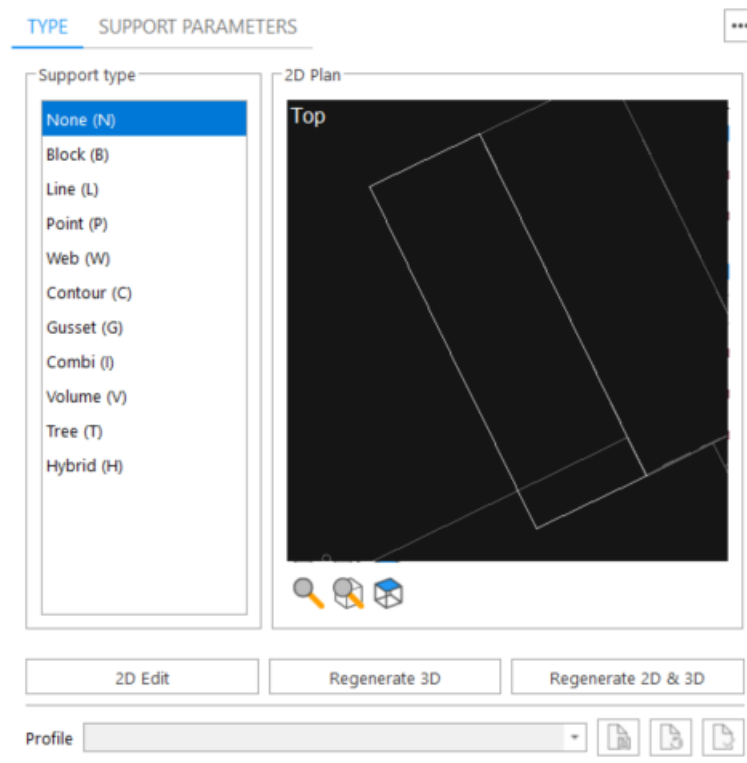
2D Edit	A new window appears with a 2D representation of the part, you can start 2D editing.
Regenerate 3D	The 2D ground plan will be used and the supports will be trimmed again on the part using the wanted parameters.

Regenerate 2D & 3D	The selected support type will be completely regenerated. The changes done in the 2D-editing mode will be ignored. When you changed a parameter (E.g. teeth) and you want to apply those settings after you did some 2D changes, press the Regenerate 3D button.
Postprocess	Cones and volume supports can be made hollow and/or supports can be filled with a lattice structure via post processing.
Save	Any type of support parameters can be saved.
Load	Load a set of support parameters that you have defined earlier.
Reset	Restores the parameters of the selected surfaces with the parameters from the current machine file.



Remark: The save and load functionality makes it possible to use different parameters for the same support types. The support parameter file is by default saved in the Support Library folder. (See Settings – Supports Library folder)

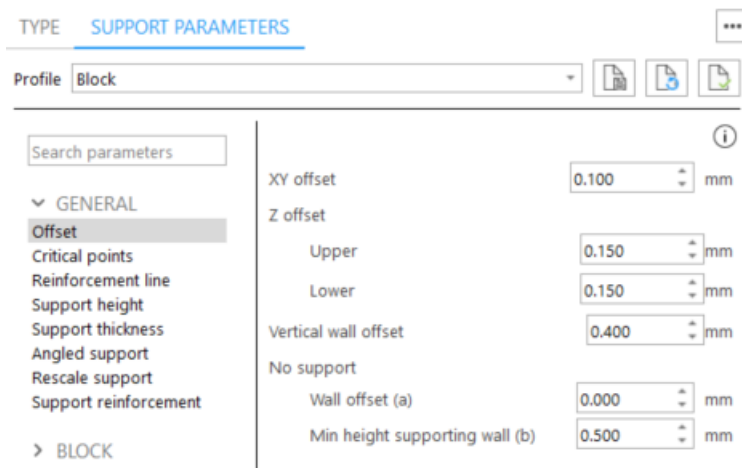
Type Toolpage



Type	The support type given to this surface. You can change the type of the active support simply by clicking on the dropdown menu and select the desired type. If the surface does not need any support, None is marked. E.g. from Block support to Contour support:
------	---

2D Plan	<p>This is a 2D overview to see where the surface with the support is located on the part. The top view of the part is displayed. The user is able to zoom and pan in this window by use of the mouse. To pan: click the mouse middle button somewhere in the window and pan. To zoom: press Ctrl and the right mouse button to zoom while moving the mouse, or simply scroll with the mouse middle button.</p>
Zoom	<p>Click to zoom in on the 2D plan, or draw the rectangular area you want to zoom into.</p>
Zoom on part	<p>Click to zoom until the part is completely visible on the 2D plan.</p>
Flip View	<p>Switch view of the part between top and bottom.</p>

General Toolpage



Detailed explanation on the common parameters can be found at the common support parameters from the machine properties.

- See **General**, page 363

Block Toolpage

TYPE **SUPPORT PARAMETERS** ☰

Profile **Block** 📄 🔄 📄

Search parameters

- GENERAL
 - Offset
 - Critical points
 - Reinforcement line
 - Support height
 - Support thickness
 - Angled support
 - Rescale support
 - Support reinforcement
- BLOCK
 - Hatching**
 - Hatching teeth
 - Border
 - Border teeth
 - Fragmentation
 - Perforations
 - Heat sink

X hatching (a) mm ⓘ

Y hatching (b) mm

Rotation angle (c) °

Filter hatching segments ⓘ

Min distance (d) mm

Min angle (e) °

Hatching levels

Number of levels

Hatching factor

Level height mm

Overlap height mm

Solid borders mm

Detailed explanation on the block parameters can be found at the block support parameters from the machine properties.

- See **Block**, page 373

Line Toolpage

TYPE **SUPPORT PARAMETERS** ...

Profile **Line** 📄 🔄 📄

Search parameters

GENERAL
 Offset
 Critical points
 Reinforcement line
 Support height
 Support thickness
 Angled support
 Rescale support
 Support reinforcement

LINE
Teeth
 Cross line
 Cross line teeth
 Perforations

Upper teeth ⓘ

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Lower teeth ⓘ

Use different values for lower teeth

Height (a) mm

Top length (b) mm

Base length (c) mm

Base interval (d) mm

Add break off points

Top addition (e) mm

Waist Z-shift (f) mm

Keep teeth top in ends

Regenerate 3D Regenerate 2D & 3D

Detailed explanation on the line parameters can be found at the line support parameters from the machine properties.

- See **Line**, page 386

Point Toolpage

TYPE **SUPPORT PARAMETERS** ...

Profile **Point** 📄 🔄 📄

GENERAL

- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement

POINT

- Rib**
- Teeth

Rib i

Number of ribs

Min rib length (a) mm

Max contact length (b) mm

Contact slope

Angle (c) °

Height (d) mm

Sink ribs i

Number of sunken ribs

Distance mm

Reinforce point support i

Detailed explanation on the point parameters can be found at the point support parameters from the machine properties.

— See **Point**, page 395

Web Toolpage

TYPE **SUPPORT PARAMETERS** ☰

Profile **Web** 📄 🔄 📄

Search parameters ℹ️

▼ GENERAL

- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement

▼ WEB

- Web**
- Teeth

Number of ribs

Manually calculate web radius

Radius (r) mm

Walls distance (a) mm

Detailed explanation on the web parameters can be found at the web support parameters from the machine properties.

- See **Web**, page 399

Contour Toolpage

TYPE **SUPPORT PARAMETERS** ...

Profile **Contour** 📄 🔄 📄

GENERAL

- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness
- Angled support
- Rescale support
- Support reinforcement

CONTOUR

- Wall**
- Teeth
- Perforations

Walls distance ⓘ mm

Fragmentation ⓘ

Separation width (a) mm

Interval (b) mm

Detailed explanation on the contour parameters can be found at the contour support parameters from the machine properties.

- See **Contour**, page 402

Gusset Toolpage

TYPE **SUPPORT PARAMETERS** ...

Profile **Gusset** 📄 📄 📄

Search parameters

GENERAL

- Offset
- Critical points
- Reinforcement line
- Support height
- Support thickness

GUSSET

- Gusset**
- Teeth

Max length (a) mm

Min length (a) mm

Notch (b) mm

Angle (c) °

Gussets interval mm

Remove gusset when self supporting

Search for remote walls i

Corner reinforcement 🔵

Detailed explanation on the gusset parameters can be found at the gusset support parameters from the machine properties.

- See ***Gusset***, page 406

Combi Toolpage

TYPE **SUPPORT PARAMETERS** ...

Profile **Combi** 📄 🔄 📄

Search parameters

- > GENERAL
- ▼ COMBI
 - Combi**
 - > BLOCK
 - > LINE

Support types i

- Block
- Line
- Gusset

Preferred support type

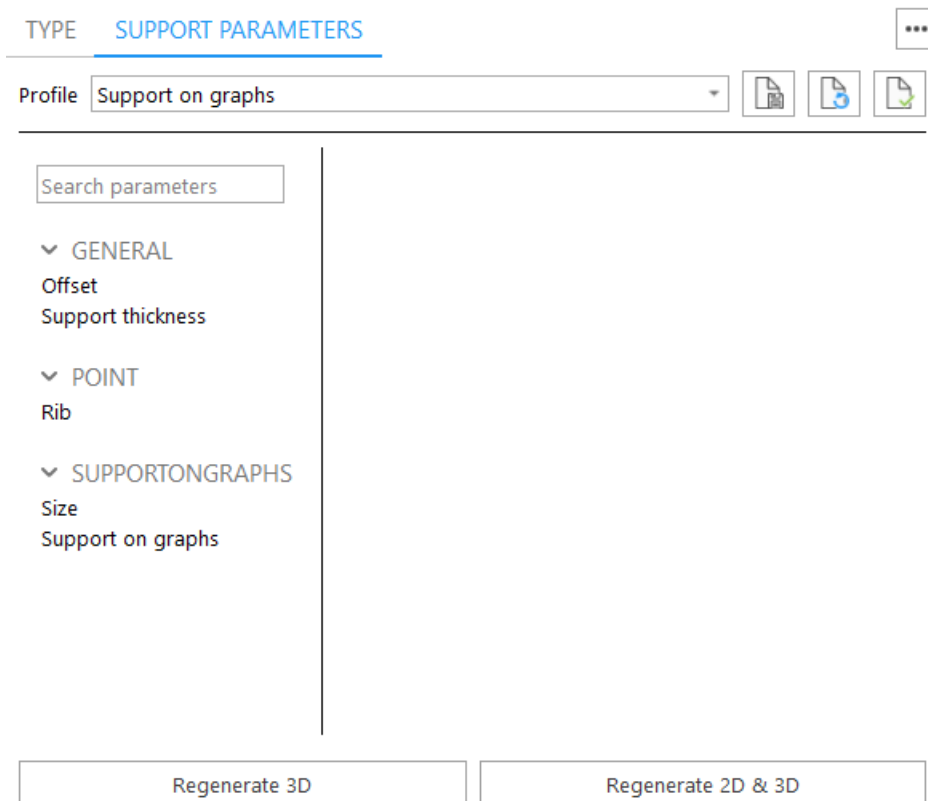
- Block
- Line

Regenerate 3D Regenerate 2D & 3D

Detailed explanation on the combi parameters can be found at the combi support parameters from the machine properties.

- See **Combi**, page 413

Support on graphs Toolpage



Detailed explanation on the ‘support on graphs’ parameters can be found at the ‘support on graphs’ support parameters from the machine properties.

- See **Support on graphs, page 422**

6. Support Types & Parameters

Magics will propose support structures based on the selection and construction parameters you have set in the Machine properties. In the support mode, you can change the proposed support type on a downfacing surface, for example from a line support to a block support. You can also adapt the support parameters of an individual support structure, for example lower the hatch distance of a block support.

9.7. 2D editing of supports

1. 2D and 3D Editing of Supports

Magics RP allows you to do 2D and 3D editing of supports.

The basic 2D operations are:

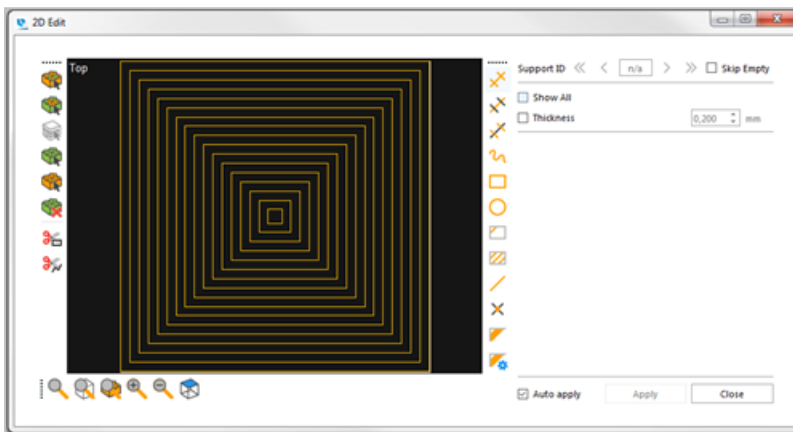
1. Selecting support structures or parts of support structures.
2. Deleting selections.
3. Cutting support.
4. 2D drawing of supports, which are then directly trimmed in 3D on the part.

The basic 3D operations are:

1. Selecting support structures or parts of support structures.
2. Deleting selections.

2D Editing

To start the editing in 2D, click the 2D Edit button in the Type Toolpage. A new window appears with a 2D representation of the part (see figure below).











1. Zoom toolbar

	Zoom: Click and drag to zoom in on the desired window.
	Zoom on part: This is an automatic zoom to make the entire part visible.
	Zoom on surface: This is an automatic zoom to make the entire selected surface visible. When a new surface is selected, the zoom on surface will be automatically performed.
	Zoom in and Zoom out: By clicking either of these buttons you zoom in/out around the center of the image with a fixed zoom factor.
	Flip view: Switch view of the part between top and bottom.

The regular zoom and pan buttons allow you also to zoom and pan in the 2D edit window.








2. Selection toolbar







While selecting or cutting, pan and zoom functions are available.

	Item: Selects one line of the active support. A second click will deselect it.
	Select Polyline: Selects a polyline of the active support. A second click will deselect it.
	Part to Part Items: Selects all lines of the active support that have a part to part connection.
	Support: selects the whole support.
	Deselect all: Deselect all selected supports.
	Delete selected: Delete all selected support structures.
	Cut: Click and drag on the 2D image to create a window selection. All support items inside the window will be cut and deleted.
	Polyline cut: Draw a polyline on the 2D image. Close the polyline or click the right mouse button to exit the command. All support items inside the polyline will be cut and deleted.


3. Drawing toolbar

While editing support, pan and zoom functions are available. This means you can zoom on a certain detail, indicate the first point of e.g. a line support, then zoom and pan to another detail and indicate the second point of the line support.

	Draw a line support without cross lines. Click to place points.
	Draw a single cross line support.
	Draw a line support with cross lines. Click to place points
	Draw a freeform line support. Click and drag to draw a freeform shape.
	Draw a line support in a rectangular shape. Click to define the rectangle corners.
	Draw a line support in a circular shape. There are 2 possible ways to create a circular support: <ul style="list-style-type: none"> - Draw circle by 3 points - Draw circle by center and radius Select the appropriate option to use at the right side of the 2D edit-box.
	Draw a single line support with direction, by indicating one point and the direction. The line will be trimmed on the border.

	<p>Draw multiple line supports with direction, by indicating one point and the direction. Define the number of lines and the distance between the lines with the parameters on the right side of the 2D edit-box. The lines will be trimmed on the border.</p> <p>Number lines <input type="text" value="3"/></p> <p>Lines interval <input type="text" value="1,000"/> mm</p>
	<p>Draw a reinforcement line to strengthen existing support.</p>
	<p>Draw a point support. Click to define the center and the direction of the ribs.</p>
	<p>Draw a single gusset support. 1st point sets the base side, while 2nd point sets the gusset length.</p> 
	<p>Draw multiple gusset supports by indicating the shape of the base side. Gussets will be placed perpendicular to the base.</p>

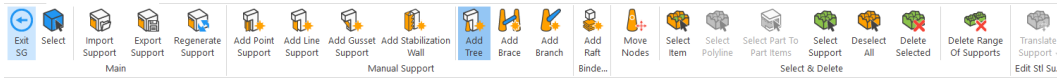
4. Support browsing and Apply

	Browse through the supports	
	First button	Return to surface one.
	Second button	Return to the previous surface
	Window	Surface number.
	Third button	Go to the next surface
Skip Empty Supports	All surfaces which are selected by the surface angle but do not need supports following the selection parameters (no support offset, surface filter...), are skipped.	
Show All	Shows all the supports	
Thickness	Visualize the thickness for the non-solid support. The default value corresponds to the laser spot diameter defined in the Machine Properties.	
<input checked="" type="checkbox"/> Auto apply	When the Auto apply checkbox is selected, all manual drawing of the support will be calculated and visualized in 3D automatically.	
<input type="button" value="Apply"/>	The manual drawing of the support will be calculated and visualized in 3D.	

You can find the 3D editing tools in the Support ribbon and in the Surface ribbon or Menubar/Tools.

3D Editing

1. The Support ribbon



Show / Hide e-Stage	Show the generated e-Stage support
Exit SG	Exit SG mode
Select	Select a part on the platform
Import support	
Export support	
Regenerate support	Regenerates all support of the selected part
Add point support	The user can draw point support in 3D.
Add line support	The user can draw line support in 3D.
Add gusset support	The user can draw gusset support in 3D.
Add Tree	The user can draw Tree support in 3D
Add Brace	Manually add a brace to a tree support with multiple trunks
Add Branch	Manually add branches to a tree support by adding more connection points on the part
Add Stabilization Wall	The user can add a vertical support wall to support thin and tall parts during FDM printing process
Add Raft	The user can add smear raft under the support in 3D
Move Nodes	Move nodes of a tree support
Select support	Selects entire support surface. A second click will deselect it.
Select part to part items	Automatically selects the items of a support which are going from part to part.
Select item	Select an item of the support. A second click will deselect it.
Select polyline	Selects a polyline of the active support. A second click will deselect it.
Deselect all	Deselects all selected support.
Delete selected	Deletes all selected support structures.
Delete range of supports	A dialog box that allows inserting a range of supports that you want to have deleted pops up.



Remark 1: Changing the active part without exiting SG

When the 'View all parts' is on, it is possible to change the active part in SG by launching "Select part" command and left clicking on the desired part in the scene.



Remark 2: Draw line, point, gusset on non-marked surface

It is possible to draw a line, point or gusset supports directly on the part surface without creating support surfaces. This is very useful to add additional supports without the need to first mark the area and create a support surface out of it.

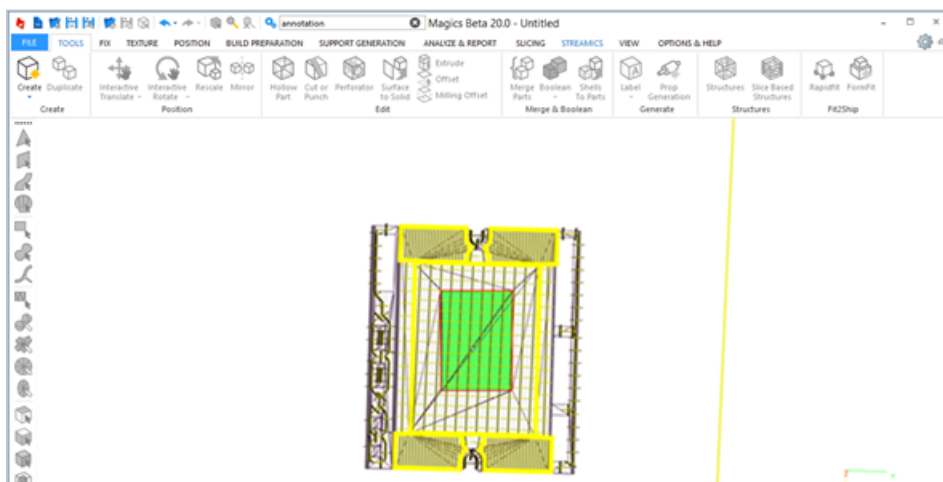
In order to manually draw supports please click on the specific type of support you would like to generate (point, line, gusset). In case of line support draw the path using points and once completed click on the right mouse button to generate support. If you are trying to generate gusset support please select first the starting point then the final point of your gusset.

Magics will automatically generate new surfaces in the support list.

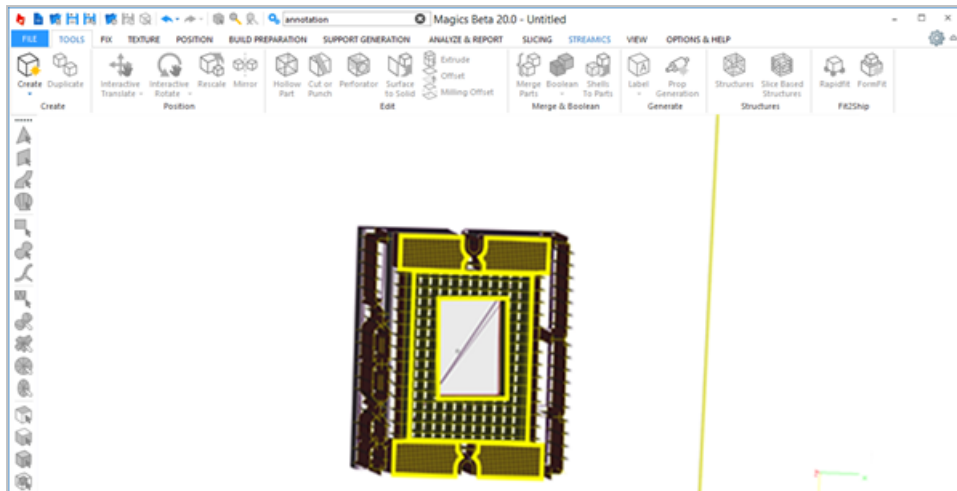
Please hold CTRL to add new support to the existing surface. Surface will be automatically updated. Notice that new type of support always starts new surface.

2. Using marking with remesh options in SG

Once you have automatically generated support surfaces in the SG module you have the possibility to mark triangles and add/remove them to/from triangles to certain surfaces or generate new surfaces made by the marked triangles. Marking with remesh can be used to select specific areas of the part that need additional support. This marked area can be used to create a new support surface to generate specific supports for this area. Please mark specific triangles using one of the marking with remesh functionalities (Window, Freeform, Polygon) and try to remove them from the current surface (see section below).



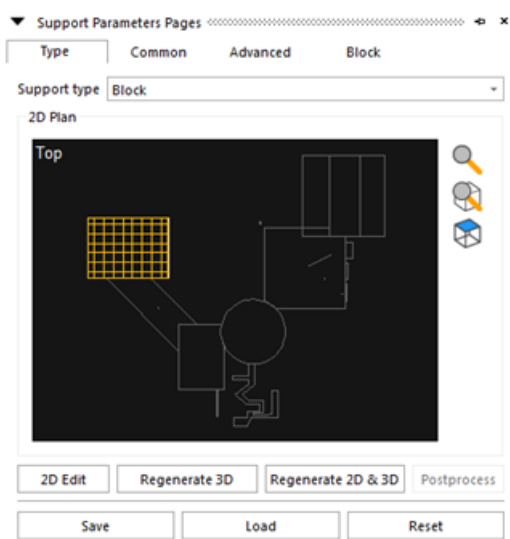
Already after marking you are able to see Magics remeshes triangles around the selected ones. After removing from the surface regenerate support 2D&3D to apply results.



This tool allows flexible manipulation of support surfaces in SG mode to create better customized support surfaces.

2. 2D Editing of Volume Supports

Type Selection



Here you can make a selection of the support type. There are only two options to place a volume support on a surface: either you place one or you don't. If you do the automatic support generation with the Select All None Support ON, Magics will select surfaces following the parameters set, but it will not place supports. You can use this buttons to place a support under the selected surfaces where you think it is necessary.

Surface Selection

Support ID << < 1 > >> Skip Empty

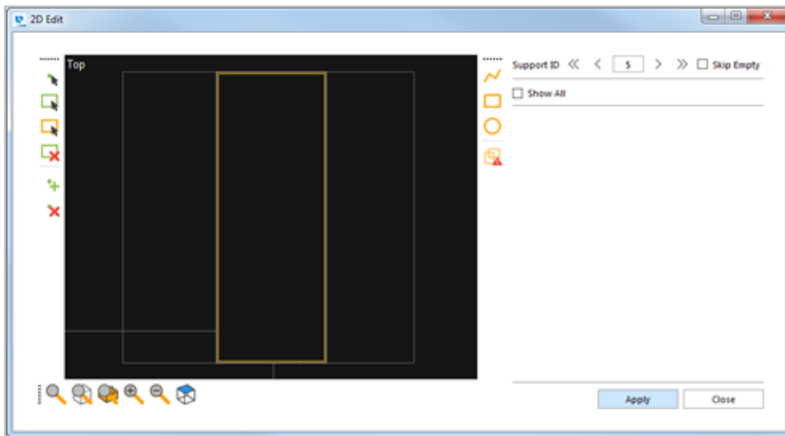
During the automatic support generation, a number of surfaces are selected based on the surface selection parameters. Default the biggest surface is catalogued as the first surface in the list. With the arrows you can inspect all the surfaces and supports.

From left to right:






«	Return to surface one.
<	Return to the previous surface.
1	Window showing surface number.
>	Go to the next surface.
»	Go to the last selected surface.
Skip Empty Supports	All surfaces, which are selected by the surface angle but do not need supports following the selection parameters (no support offset, surface filter...), are skipped.

2D edit dialog

To start the editing in 2D, click the 2D Edit button in the Type Toolpage. A new window appears with a 2D representation of the part (see figure below).






1. Zoom toolbar

	Zoom: Click and drag to zoom in on the desired window.
	Zoom on part: This is an automatic zoom to make the entire part visible.
	Zoom on surface: This is an automatic zoom to make the entire selected surface visible. When a new surface is selected, the zoom on surface will be automatically performed.
	Zoom in and Zoom out: By clicking either of these buttons you zoom in/out around the center of the image with a fixed zoom factor.
	Flip view: Switch view of the part between top and bottom.

The regular zoom and pan buttons allow you also to zoom and pan in the 2D edit window.




2. Draw Shapes

In the 2D edit mode, you can draw 2D volume supports yourself. You draw them in 2D and when you apply them (click on the Apply button), the volume supports are trimmed in 3D on the part. There are three default shapes: polyline, rectangle and circle. When you draw a freeform surface, mind the fact that no crossing lines are allowed.

	Draw a polygonal volume support.
	Draw a rectangular volume support.
	Draw a circular volume support.




3. Change Drawn Shapes

Once you have drawn a standard shape, you still can change the shape. You can add a point to the contour, you can delete a point or you can move the point.


	Select anchor point: Select a point of the contour. Hold and drag to move the point.
	Add anchor point: Add a point to the contour by clicking on it.
	Remove anchor point: Delete a point of the contour by clicking on an existing one.

4. Select and Delete

You can also select and delete support structures in the 2D window.

	Select Polyline: select a closed polyline of the active support.
	Deselect all: Deselect all selected supports.
	Delete selected: Delete all selected support structures.

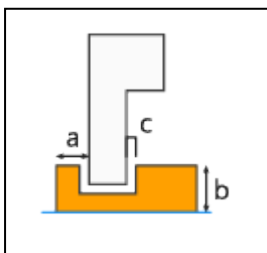
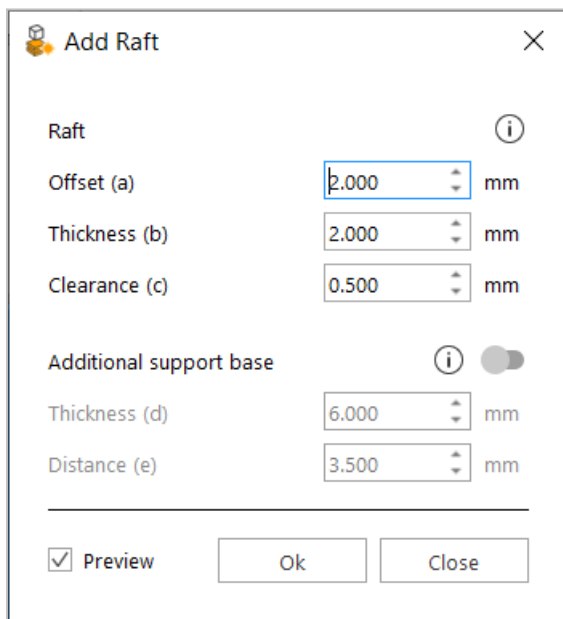
5. Detect Shape Intersections

	Since the generated supports are volume supports, the supports may not intersect. The intersection button does a check on this rule. If an intersecting contour is found the modifications to the support cannot be applied.
---	--

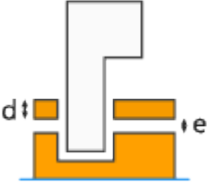
9.8. 3D editing of supports

1. Add Raft

The user can add smear raft under the support in 3D.




Generate a raft below the support and attached to the platform to prevent the part from smearing. Define the raft shape, the thickness and the clearance from the part.

Add Raft	
Offset (a)	Offset from the part to the margin of the raft
Thickness (b)	Thickness of the raft
Clearance (c)	Clearance between the part and the raft, to ease removal of the raft.
 <p>Additional support base</p>	<p>Add an additional support plate to establish a stable base for sintering. This base is generated above the raft at the desired distance, and is attached directly to the support.</p>
Thickness (d)	Thickness of the additional plate.
Distance (e)	Distance between the main base and the additional plate.

2. Stabilization Wall

The user can add a vertical support wall to support thin and tall parts during FDM printing process

 Add Stabilization Wall
✕

(i)

XY Offset (a) mm

Z Offset (b) mm

Support penetration mm

Extrude length (c) mm

Wall width (d) mm

Bridge width (e) mm

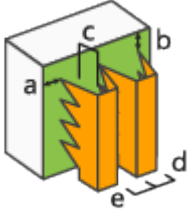
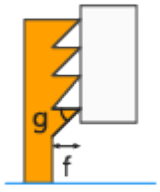
(i)

Teeth

Teeth height (f) mm





Teeth angle (g) °



Preview

	Vibrational forces within the machine can affect large, tall parts and may lead to dimensional inaccuracies. Mark the desired surface and add a stabilization wall to prevent it.
XY Offset (a)	Horizontal offset from the margins of the selected triangles.
Z offset (b)	Vertical offset from the margins of the selected triangles.
Support penetration	Penetration of the stabilization wall into the part.
Extrude length (c)	Distance from the part to the wall of the stabilization wall.
Wall width (d)	
Bridge width (e)	
	Teeth height (f) Teeth angle (g)
Preview	Preview instantly the effect of changed parameters before applying changes.

3. Create and edit tree support manually

To start a manual tree support, you have to press 'Add Tree'. After that, you can add connection points on the part using left mouse button click. First connection point will create a new support ID in the support list. To finish adding connection points and generate the Tree support, click the right mouse button. Following commands are used in the manual editing of tree supports:

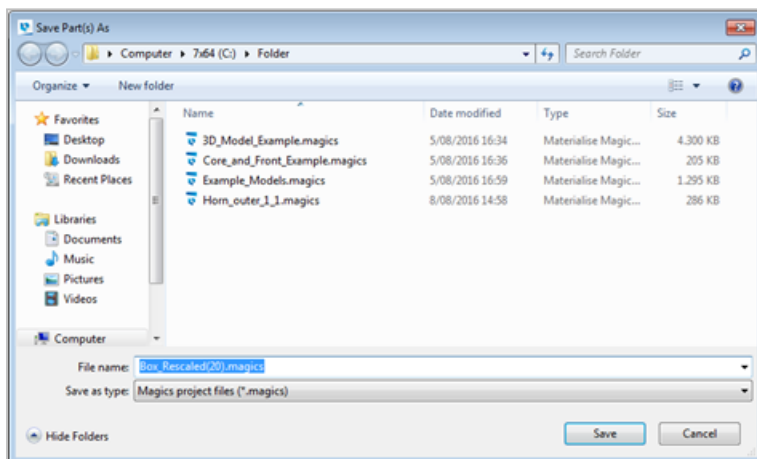
	Manually add a Tree support. Parameters used to create support are based on the default Tree profile in Machine Properties. Left mouse button click will add a new connection point, right mouse click will finalize the manual tree creation.
	Manually add branch on current Tree surface.
	Manually add brace on current Tree surface with multiple trunks by adding connection points on the part.
	Manually move nodes of the branches, braces and trunks of the selected Tree surfaces.

	
Select item 	Select one or more branches, braces, trunks.

9.9. Saving and Exporting Supports

1. Save Support

You can save the generated supports in the Magics application. This can be done by going to File/Save part as. The file is then saved as a .magics file. You can quit the program, and load the .magics project again. A dialog box appears to ask if you want to open the part with or without support (see figure below). Support made in Magics 9 or earlier versions was saved as .sup file. These support files can be loaded into the project by Modules/Support Generation/Load Support File(s).

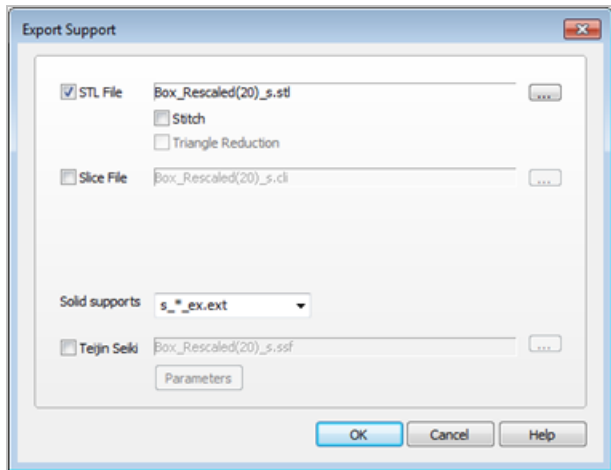


2. Export Support

You can also Export the generated supports towards different slice formats and the STL format. The choice of export file depends on your machine. To use the supports on your RP machine, you must use the Export functions, not the Save functions.

- If you also have the Slice Module (Slicing of Parts):

Go to File and select Export Support. Fill in a name, choose the directory in which all export files will be placed. Now you can set your slice parameters. Press OK and Magics starts generating the STL files and starts the slicing of both the part and the support. (see Slicing for the explanation of the parameters).



- If you do not have the Slice Module:

Go to Support Generation in the Modules Menu and select Export Support or in the support mode choose file in the menubar, and Export support. Now you can set your slice parameters. Press OK and Magics starts generating the STL file and the slice file of the supports.

9.10. Visualization of Support Structures

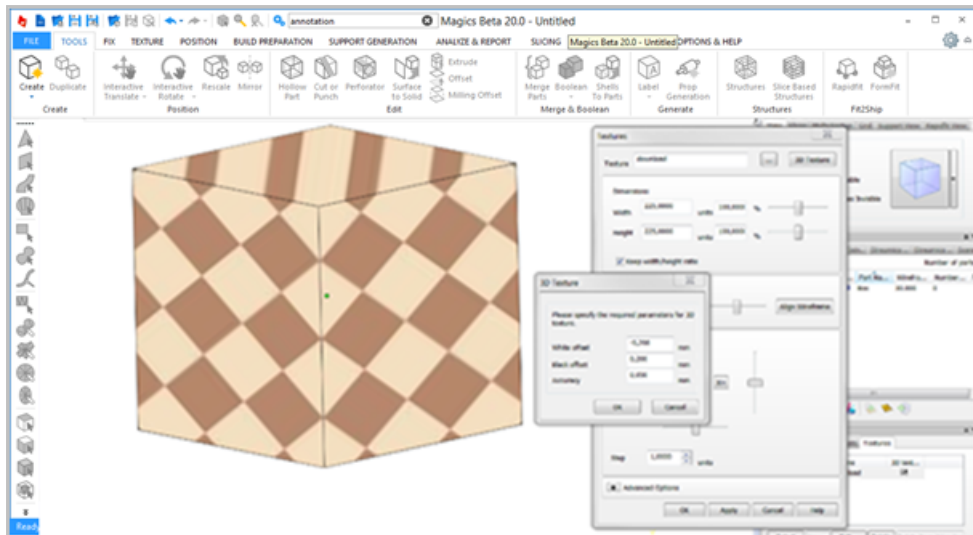
Visualization options of the support structures are defined in the Settings, on the page Supports (Settings/Visualization/Supports).

- See **Supports, page 534**

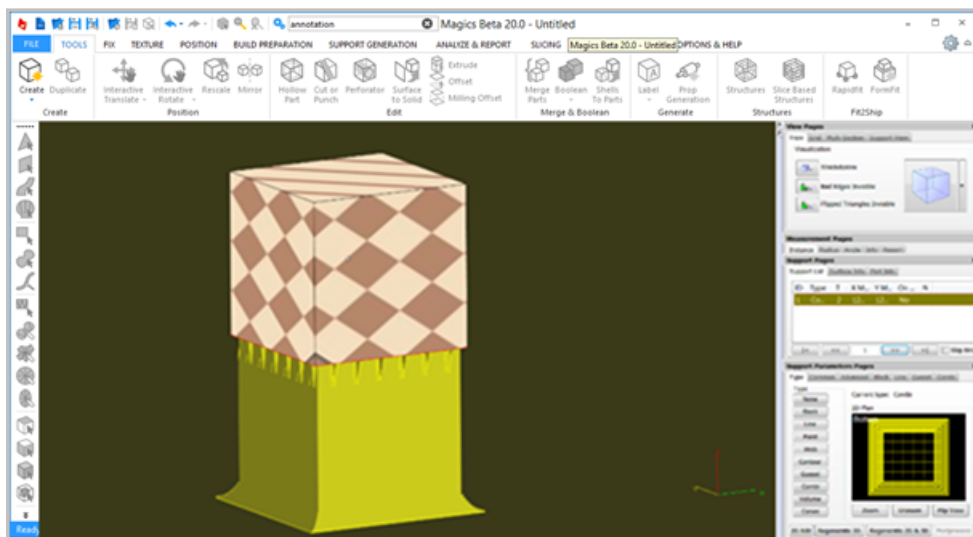
Show or hide the support of all parts present on the scene by using the command Support Visibility (see more information in paragraph Support Visibility).

9.11. Support on 3D Texture and Slice Based Lattice

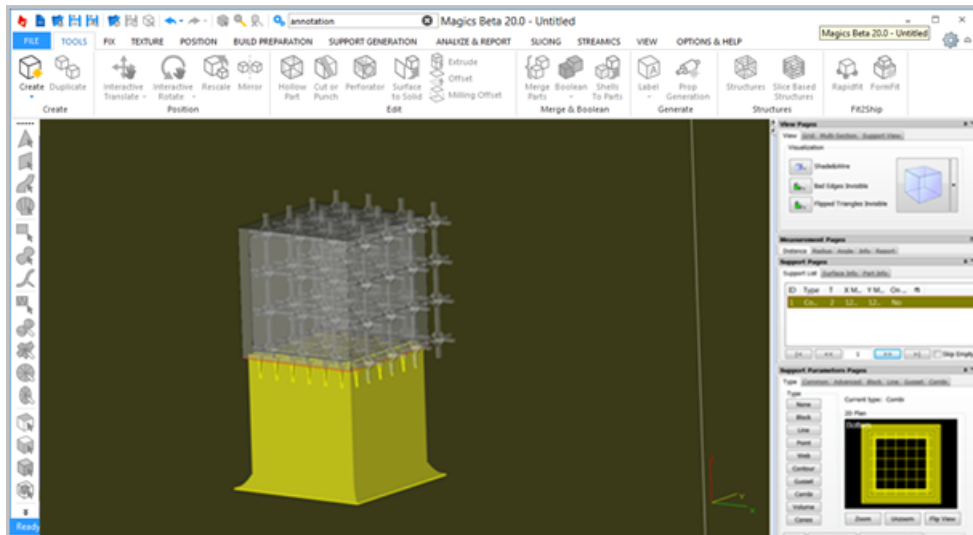
Thanks to our software solution, it's today possible to apply texture on your part in Magics which will be processed and considered as a 3D texture when processed by the build processor.



What you can see on Magics is than just a 2D preview on your part (for 3D texturing, see section above and BP). Support Generation module allows you to apply supports on a part where texture is applied.



Carefully consider that 3d texture offset is not taken into account by support. The user might compensate the gaps using z-offset in supports. Also when a slice based structure is generated SG allows to generate support on the part. This can help in order to obtain a complete preview part-support-structure before sending everything to the machine via BP.



All the SG module parameters are editable and applicable.

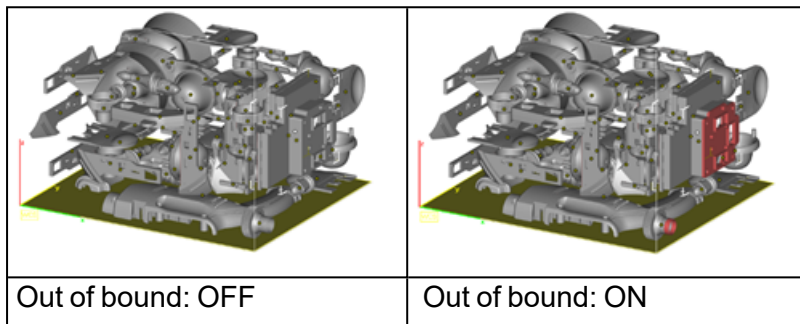
Chapter 10. Analyze & Report

10.1. Analyze Build

1. Out of bounds



The out of bounds function will color parts which are placed outside of the platform bounds. Any placement tool can be used without losing the color indications.

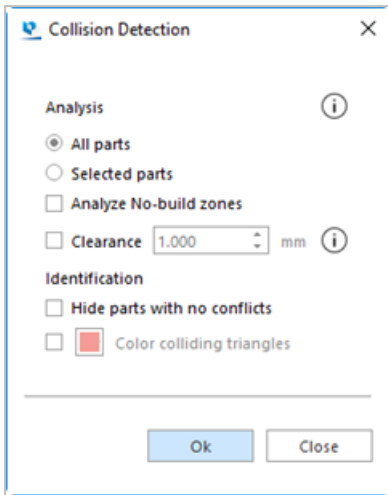


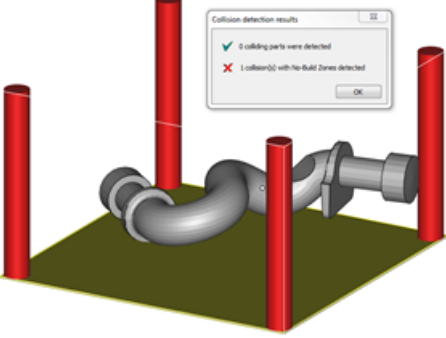
Remark: On round platforms, the whole part is colored when it (or its support) is out of bounds.

2. Collision Detection

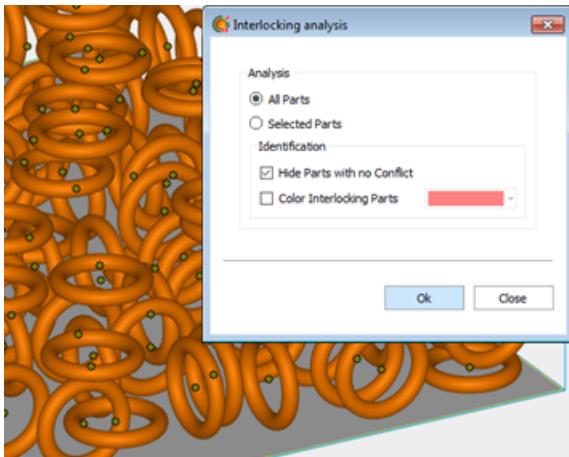


If several parts are loaded on the platform, Magics can detect if there is a collision. Collision can be detected between intersecting triangles or with an indicated clearance between different parts and supports. A message-box appears to tell you if there are or aren't colliding parts and/or supports. The involved triangles are marked (parts only, supports can't be marked).



Analysis	
All parts	Collision is detected between all loaded parts on the platform scene.
Selected parts	Collision is only detected between selected parts
Analyze No-Build-Zones	Collision is detected between the part(s) and active 'No-Build-Zones' 
Clearance	Define the spacing allowed between parts. If they are located within a distance smaller than this value, they will be identified as colliding parts.
Identification	
Hide parts with no conflict	Make parts where no collision is detected invisible after the analysis
Color colliding triangles	Color triangles where collision is detected

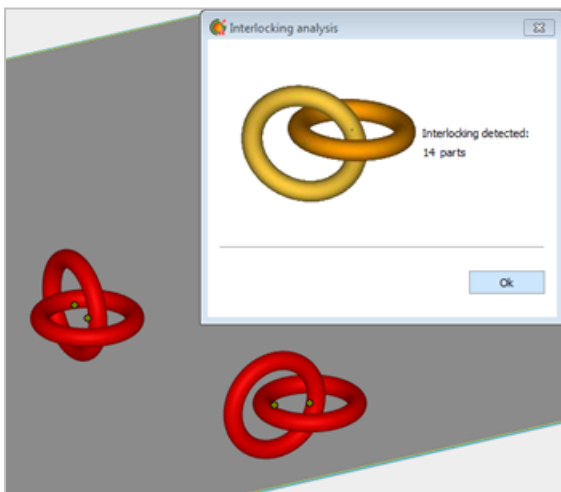
3. Interlocking Analysis



Interlocking means that 2 (or more) parts cannot be separated due to the positions they are nested in. When performing an Interlocking Analysis, the user can choose to analyze all parts or only the selected parts. One can choose how to identify the interlocked parts by checking or unchecking the boxes under Identification:

- 'Hide Parts with no Conflict' hides all parts that are not interlocked with one another.
- 'Color Interlocking Parts' will color the detected interlocked parts to the specified color.

If interlocking is detected, all suspected interlocking parts are visualized, while the other parts are hidden (if the user chose this option).



4. Wall Thickness Analysis



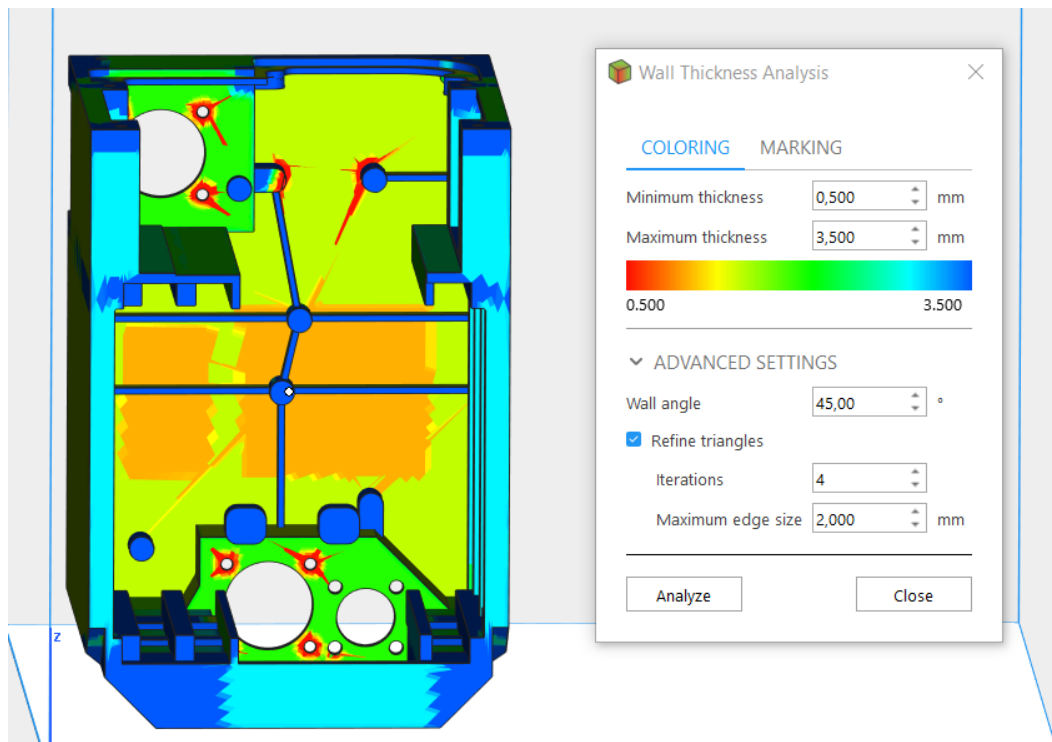
Wall thickness (as the name implies), is the thickness / depth / diameter of any section of a printed part.

Wall thickness analysis is a triangle based analysis. Therefore, each triangle is assigned a thickness value.

The thickness value is calculated by measuring the shortest distance between the triangle and the opposite triangle. There are multiple checks in place to classify two triangles to be opposite of each other. Do note that the triangles do not need to be directly opposite of each other in their normal direction to pass the check. This is because thickness can be measured in any orientation in the 3D coordinates.

Coloring

In the Coloring option of the Wall thickness analysis the triangles are colored corresponding to their wall thickness.



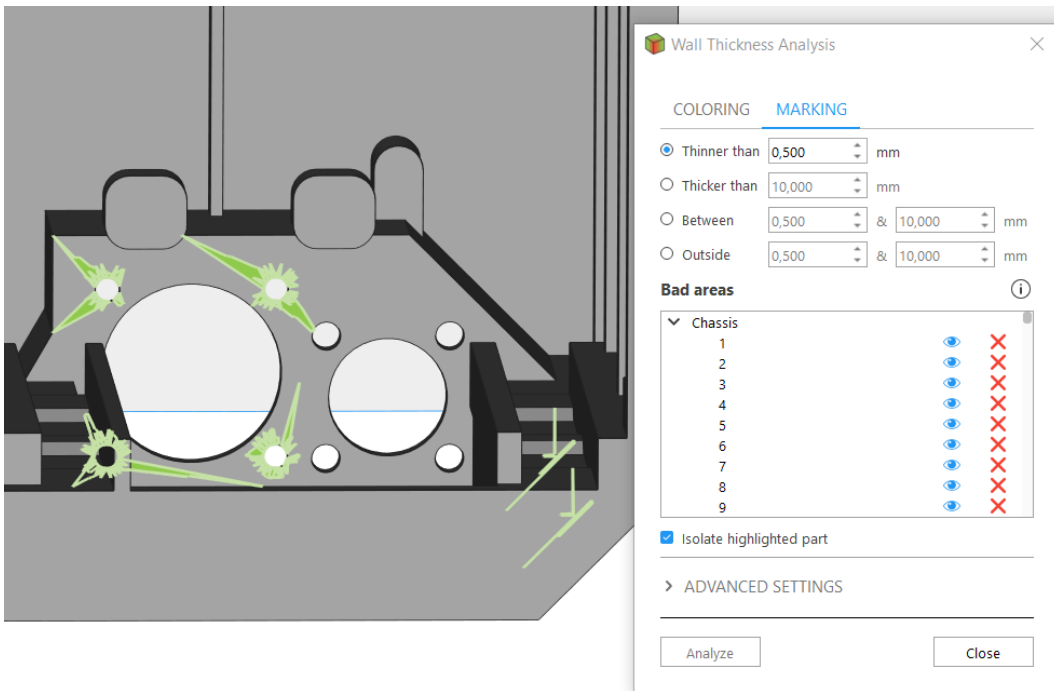
Minimum thickness	This parameter defines the minimum wall thickness needed. Every triangle with a local wall thickness smaller than the entered value, will receive the begin color.
Maximum thickness	The maximum thickness is the wall thickness for which you no longer expect to have problems. Local thickness greater than the maximum thickness does not need special attention and will have the end color. Triangles with local thickness situated between the minimum and the maximum thickness will have a color gradually changing from the begin color (minimum) to the end color (maximum wall thickness) over the color spectrum. The minimum and maximum wall thickness also forms the borders of the Wall Thickness Color Legend.
Color scale	In the color legend you can see which wall thicknesses corresponds to which color. Color range can be customized from the Settings -> Visualization -> Color -> Wall thickness analysis.

Analyze	Click the analyze button to run the analysis. Change the parameters and rerun the analysis as many times as needed without closing the dialog.
---------	--

 **Coloring** Wall Thickness Analysis option is also available for BREP parts.

Marking

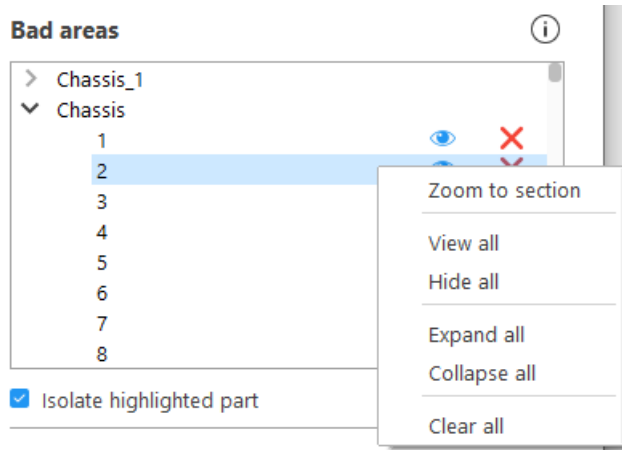
The Marking option identifies triangles where wall thickness is outside the acceptable threshold values based on the selected settings and marks them in groups as Bad areas. Each bad area can be examined individually from the results of the analysis.



Detect triangles of walls	Choose one of the detectors (Thinner than, Thicker than, Between or Outside) and enter the limiting value(s) in the edit box(es).
Thinner than	Search walls thinner than the entered value
Thicker than	Search walls thicker than the entered value
Between	Search walls between the entered values
Outside	Search walls outside the entered values
Analyze	Click the analyze button to run the analysis. Change the parameters and rerun the analysis as many times as needed without closing the dialog.

Marking results

After the analysis of the part(s) Magics shows a list of areas that match the WTA conditions withing the Bad areas section of the dialog.



Part	Part name is displayed. All bad areas for the part are shown in a tree under the part name and can be collapsed .
ID	Bad areas are numbered in order from 1 under each part.
Visible/ Invisible	Hide / reveal bad areas by clicking on the eye icon
Delete	Remove the indicated area from the list by clicking on the x icon
Isolate high- lighted parts	Only the selected part in the part list is isolated. To have a good overview all other parts on the platform are invisible.
Context menu - click right mouse button (RMB)	
Zoom to selection	Zoom to the area that is in selected state on the list
View all	Make all bad areas visible (mass-enable the eye icon)
Hide all	Hide all bad areas (mass-disable the eye icon)
Expand all	Expand the bad areas lists for all analyzed parts
Collapse all	Expand the bad areas lists for all analyzed parts
Clear all	Delete all bad areas that were identified
Other behavior	
Autozoom	Double-click on an area from the list to automatically zoom in on the bad area on the part. On a second double-click on the same area, the view point on the bad area is flipped 180 degrees.

Advanced settings

For a more detailed analysis it is possible to temporarily remesh the part to get a greater number of smaller triangles and a higher precision wall thickness analysis. To remesh for the analysis, enable Refine triangles and set the number of Iterations and Maximum edge size in the Advanced settings of the Wall thickness analysis dialog.

▼ ADVANCED SETTINGS

Wall angle °

Refine triangles

Iterations

Maximum edge size mm

Refine triangles parameters	
Refine triangles	If checked, the triangles that met the criteria are re-triangulated. The analysis is based on the newly created triangles.
Iterations	The max number of iterations for refining the triangles.
Max edge size	A triangle will only be split up in a smaller triangle if one dimension of that triangle is bigger than the maximum edge size. This parameter thus decides the accuracy with which you are going to visualize (and measure) the thickness distribution.

Refine triangles option applies default **Triangle reduction** and **Filter sharp triangles**, before the remesh.

To apply **Triangle reduction** and **Filter sharp triangles** without further remesh, set **Iterations** parameter to 0.

For **BREP** parts, **Wall thickness analysis** is performed on the underlying mesh. **Refine triangles** option can be applied to the underlying mesh to improve **Wall thickness analysis** results.

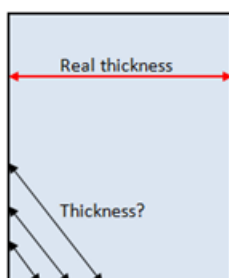


For the Coloring option, the triangles are re-triangulated temporarily to visualize results of the WTA. Once the dialog closes, the analyzed part gets back to its original mesh. For the Marking option, the part stays remeshed after closing the dialog, and the bad area triangles stay marked.

Wall angle

Defining a wall angle makes it possible to exclude certain areas/ surfaces from the analysis.

The following 2D drawing illustrates the case where you have a wall with two rectangular angles.



From a triangle point-of-view, the local wall thickness approaches zero when coming closer to the edge of the part. The real thickness though remains the same throughout the whole wall. Therefore we can exclude the area around these edges from the thickness calculations.

The wall angle is the parameter that avoids that the above areas are taken into account. When the wall angle is for instance set to 60 degrees, thickness calculations will only be done for edges whose angle is smaller than 60 degrees. These edges will be interpreted as functional edges of the part and not just 'wall borders'.

5. Detect Trapped Volumes



This functionality can be used to easily detect trapped volumes (cavities) in your part(s).

Before building parts with SL technology many users will try to identify if their parts consist out of any cavities. During building there will be a height difference between the resin within the cavity and the bin itself. This difference can lead to deformation of surfaces.

To pass this problem you can utilize the 'Detect Trapped Volumes' functionality to identify the trapped volumes on your part easily, once these areas are detected some perforation can be made to avoid deformation or parts can be repositioned to minimize the trapped volumes.

Another way to overcome this problem without creating perforations is to use the 'Detect Trapped Volumes' function to indicate the Z min and Z max values of the trapped volumes. These values can then be used within the machine software to adapt the build process.

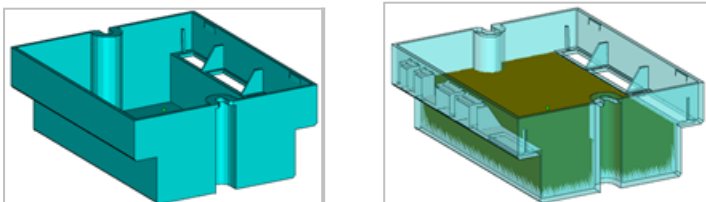
(Change the number of times that the recoater passes for each layer. More resin will be put in the trapped volume areas so that the resin level is the same at the inside and outside of the trapped volume.)

Type of trapped volumes

There are 2 types of trapped volumes that can be detected on your part, which are open volumes and closed volumes.

– Open trapped volumes:

These are cavities in your part which have a connection to the outside world. During the building process there might occur an issue due to a difference in height of the resin on the in- and outside of the part.

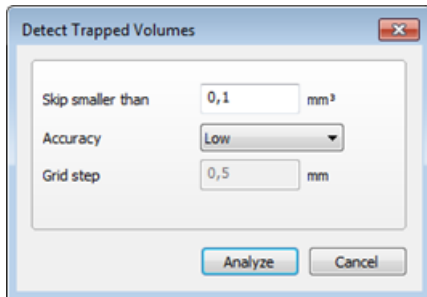


– Closed volumes:

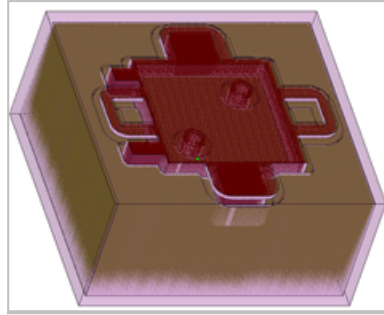
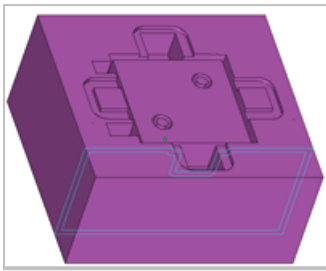
Internal cavities with a volume bigger than the specified value.



How does it work?



Skip smaller than	Eliminate cavities which are smaller than the entered value in mm ³	
Algorithm	Accurate	High accuracy, slow analysis
	Medium	Medium accuracy, medium speed
	Fast	Low accuracy, high speed
	User	User defined accuracy
Grid step	Depending on the chosen algorithm a predefined or user indicates step is used	
Analyze	The analysis of the trapped volumes is performed	



2 trapped volumes detected = 1 open volume + 1 closed volume (hollow)

Further specifications of the found trapped volumes are shown in the print screen below.

ID	Part	Z min	Z max	Estimated Volume	Type
1	front_Rescaled(0.8)	11.983	12.648	0.729	Opened
2	front_Rescaled(0.8)	11.263	19.983	8.720	Opened
3	front_Rescaled(0.8)	11.263	22.063	5.400	Opened
4	front_Rescaled(0.8)	11.263	19.983	6.540	Opened
5	front_Rescaled(0.8)	11.263	22.063	5.400	Opened
6	front_Rescaled(0.8)	11.263	19.983	4.360	Opened
7	front_Rescaled(0.8)	11.263	22.063	5.400	Opened
8	front_Rescaled(0.8)	11.263	19.983	4.360	Opened
9	front_Rescaled(0.8)	9.583	11.183	70.400	Opened
10	front_Rescaled(0.8)	11.743	11.983	21.120	Opened
11	front_Rescaled(0.8)	11.263	19.983	4.360	Opened
12	front_Rescaled(0.8)	11.263	19.983	4.360	Opened
13	front_Rescaled(0.8)	11.263	19.983	6.540	Opened
14	front_Rescaled(0.8)	11.263	22.063	5.400	Opened

Visible/ Invisible	Hide/ unhide trapped volumes
ID	Every trapped volume is indicated with a unique ID to easily identify them
Part	Part name is displayed
Zoom	Zoom in to the indicated trapped volume
Delete	Remove the indicated trapped volume from the list
Z min	The starting height of the trapped volume
Z max	The ending height of the trapped volume
Volume	The total volume of the cavity
Type	The type of trapped volume
Reanalyze selected parts	The list is updated by performing a new analysis on trapped volumes

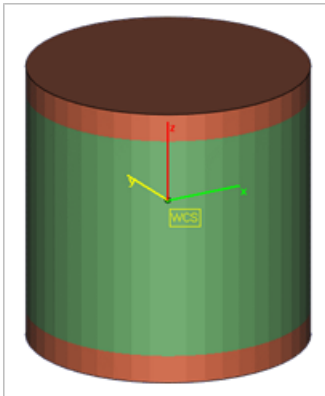
Isolate highlighted part	Only the selected part in the list is isolated. To have a good overview all other parts on the platform are visible
Autozoom	Zoom automatically to the trapped volume that is highlighted
Part level opacity	Use the slide bar to change the opacity level of your part

6. Build Risk Analysis



Visualize the risk of failure or distortion on the selected part(s) according to the slice distribution.

The risk is visualized by red, yellow or green. Areas with a high risk are shown in red, areas with medium risk in yellow, and areas with no risk are shown in green.



- For the settings, see **Build risk analysis, page 554**

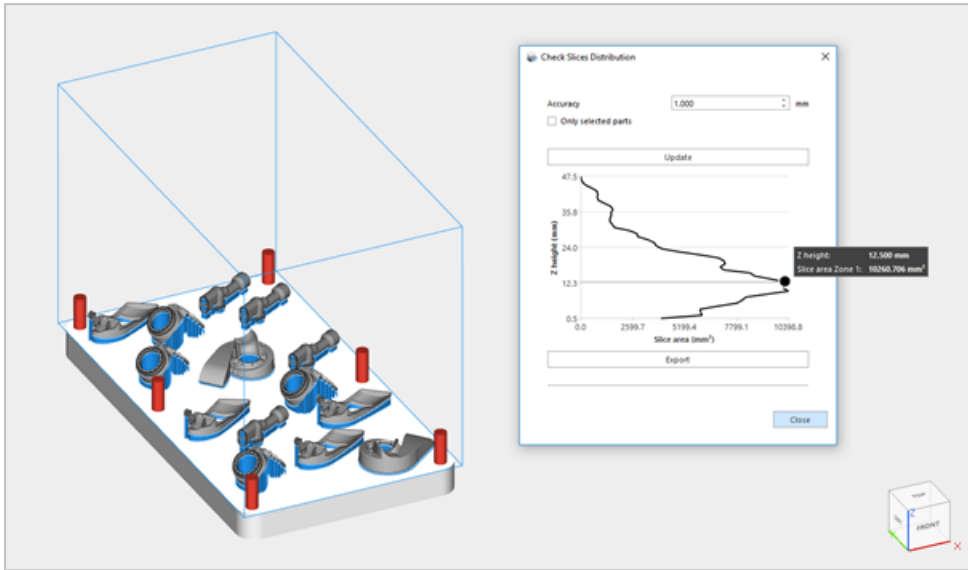
7. Check Slices Distribution



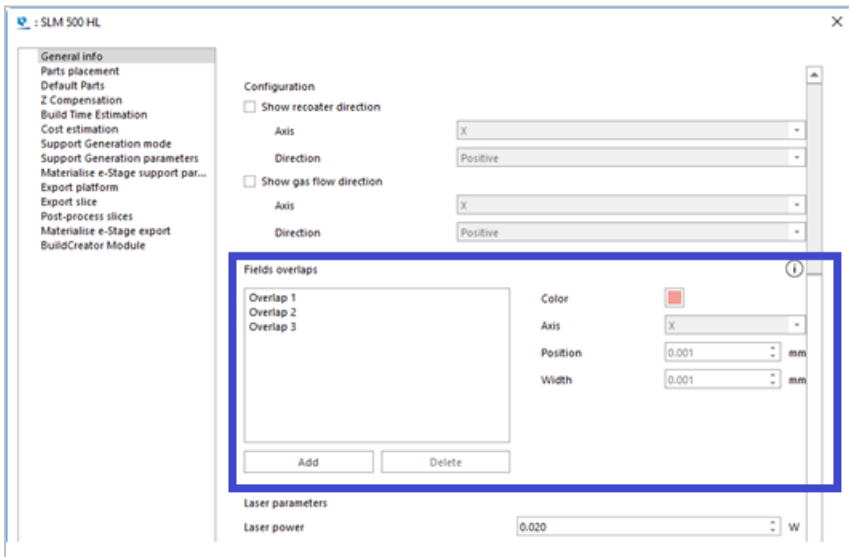
The slice distribution chart gives you the possibility to analyze the surface area for each layer and the distribution within your build. Hover over the chart with the mouse and visualize the values of Z height and slice area for each point.

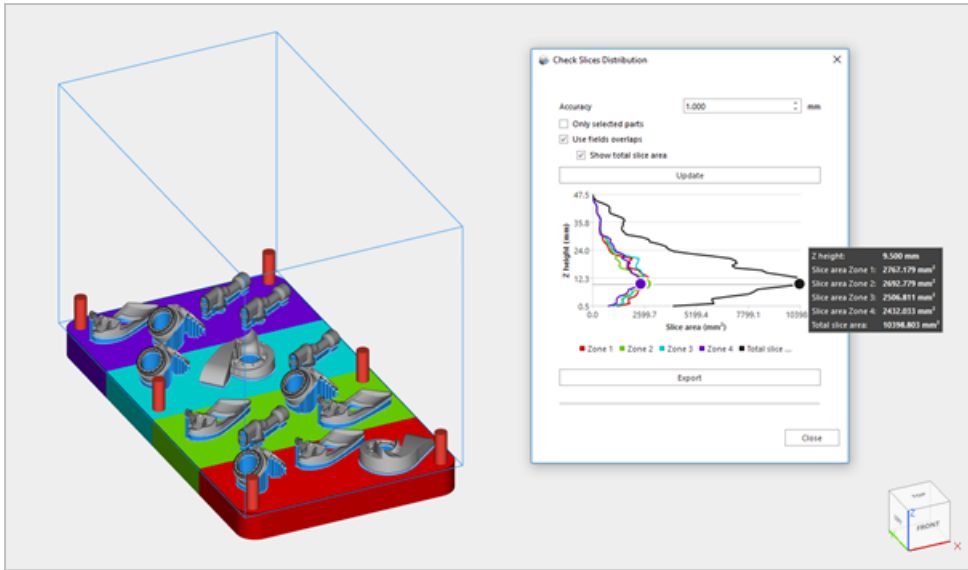
By adjusting the accuracy, you can control the distance between layers, which will be taken into account during the calculation; the bigger the accuracy, the further the layers will be between each other.

You are able to view the chart for the entire build or for the selected parts only. Supports structures are also taken into account in the slice area.



When fields are defined in the Machine Properties (see **General info page, page 293**), you can use those to split the chart data in multiple zones. This makes it possible to analyze the workload of multi-optic machines. Every zone of the chart represents the slice area of one scan field. You can decide to visualize or not the total slice area together with all the zones. The color on the chart corresponds to the color on the platform.





The data from the graph can also be exported to an Excel document.

Height (mm)	Total slice surface (mm ²)	Zone 1 (mm ²)	Zone 2 (mm ²)	Zone 3 (mm ²)	Zone 4 (mm ²)
0,500	0,000	0,000	0,000	0,000	0,000
1,500	0,000	0,000	0,000	0,000	0,000
2,500	0,000	0,000	0,000	0,000	0,000
3,500	0,000	0,000	0,000	0,000	0,000
4,500	1396,981	463,590	198,186	500,305	234,900
5,500	2669,143	637,865	685,160	649,411	696,706
6,500	3329,102	767,590	969,131	695,420	896,960
7,500	4068,768	973,128	1359,996	674,389	1061,255
8,500	5657,322	1219,249	1864,442	964,220	1609,412
9,500	7172,291	1689,107	2138,811	1447,336	1897,036
10,500	7084,237	1815,470	2003,419	1538,699	1726,649
11,500	7285,301	1913,116	2095,957	1546,693	1729,533
12,500	7568,379	1962,788	2236,434	1547,754	1821,403

For more information on the usage of the Slices Distribution together with the Sintermodule:

- See *3D Nester - Check slices distribution, page 336*

10.2. Estimate

1. Build Time Estimation



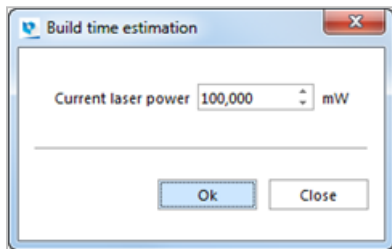
There are two ways to calculate build time:

- Stereolithography based: If the user is building with a Stereolithography machine, he can use the Stereolithography parameters of his machine. The program will take an artificial support structure into account (a grid below the part).

- Self-learning: In all other cases, self-learning build-time calculation is possible. The user can work with or without generated support structures.

In both cases, choosing the Build Time Estimation calculation function will display the results in the working area, for an easy overview on the estimation. This data is automatically updated in case transformations are applied to the parts present in the scene. To hide the results from the working area, you can just click again the Build Time Estimation button.

In the Machine Properties, under the Laser parameters options, you can specify a default value for the Laser power of the specific machine. Moreover, you can select if you want the software to ask you for the laser power each time the Build Time Estimation function is started. In case this option is selected, the dialog below will be shown when you start the Build Time Estimation function.



Remark: this dialog is available for:

- Laser parameters method (Stereolithography based machines)
- Self-learning method in combination with selection of Laser-based system option. This option can be specified in the Machine Properties, under the Self-learning settings.

Build Time calculation with Laser parameters

In case of SL-based build-time calculation you will get the following result:

	Build time estimation
Part scan time	143 hrs 41 min
Support scan time	11 hrs 27 min
Recoat time	06 hrs 44 min
Total time	161 hrs 53 min
Machine laser power	20.0000 mW

Five values are displayed: the scan time of the part, the scan time of all the supports, the total recoating time, the total building time and the machine laser power selected.

Self-learning Build Time Calculation

In case of self-learning Build Time calculation the following result will be displayed:

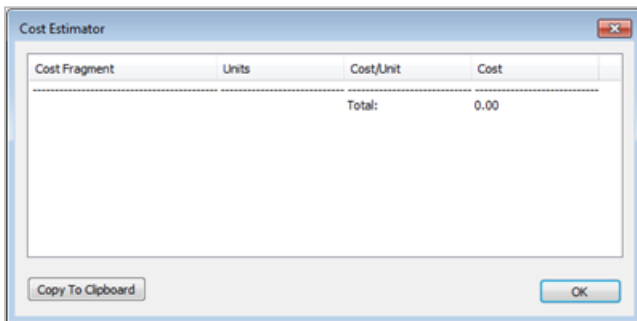
	Build time estimation
Total time	07 hrs 30 min

This build time is calculated for the current platform (all loaded parts) and is based on the teaching platforms which were specified in the Machine Properties.

2. Cost Estimator



The Cost Estimation is based upon parameters, which are machine dependent. Therefore, these parameters are defined in the Machine Settings. When you enter this function, the Select Machine dialog will appear, where you can choose a machine. The cost estimator calculates the cost of making the parts. For further details on the parameters used in the Cost Estimator, see Support Generation Manual.

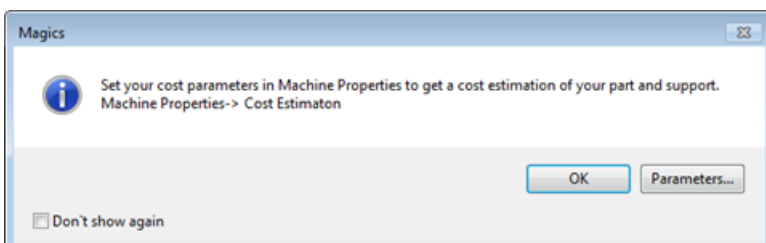


Remark: When you use Copy To Clipboard you can enter these data in Excel, Word, etc.

3. Material Cost Estimation



Show an estimation of the material cost of the selected part(s). To calculate the estimation, the cost parameters in Machine Properties need to be filled in. The following dialog will pop up:



Click **Parameters** to directly go to the Parameters in Machine Properties.

- See ***Stereolithography method, page 304***



Click **OK** to close the dialog. The estimation is made based on the available parameters and is shown:

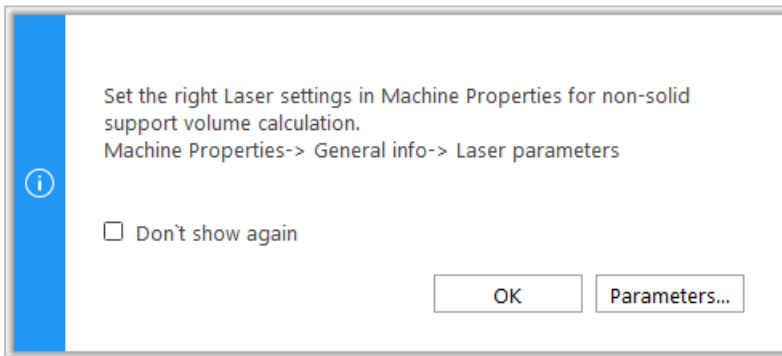
	Cost
Part(s) (materials)	22.50
Support (support)	112.11
Base Plate (support)	9.38
Total	134.61

4. Volume Estimation



Show an estimation of the materials volume of the selected part(s). When toggled the icon gets a blue background. Part(s) volume includes the mesh and the lattice components of the part(s).

To calculate the estimation, the laser spot diameter parameter in Machine Properties need to be filled in. The following dialog will pop up:



Click **Parameters** to directly go to the Parameters in Machine Properties.

- See Stereolithography method, page 1

Click **OK** to close the dialog. The estimation is made based on the available parameters and is shown:

	Volume
Part(s)	69910.852 mm ³
Support	0.000 mm ³
Base Plate	0.000 mm ³
Total	69910.852 mm ³

5. Toggle Nesting Density



Show/hide the nesting density of the current platform. Activating the nesting density gives an easy overview in the working area of the current volume utilization of the platform, the current nesting density and the build height of the nested build. When toggled the icon gets a blue background. This functionality is ONLY accessible when Sintermodule license is present.

Platform volume utilization	2.98 %
Current nesting density	4.59 %
Build height	195.00 mm

10.3. Measure

1. Measurement Distance Point to Point



Measure a distance point to point (CTRL+SHIFT+X).

Extra information on the measurement is shown in the Measurements pages.

- See **Measurements pages, page 604**

2. Measurement Thickness



Measure the thickness of an area. (CTRL+SHIFT+C)

Extra information on the measurement is shown in the Measurements pages.

- See Measurements pages, page 1

3. Add Real Measurements

- See **Add real measurement, page 611**

4. Measurements Quality



Measurements Quality brings you directly to the Measurements Quality dialog in the Settings.

- See **Measurements Quality, page 517**

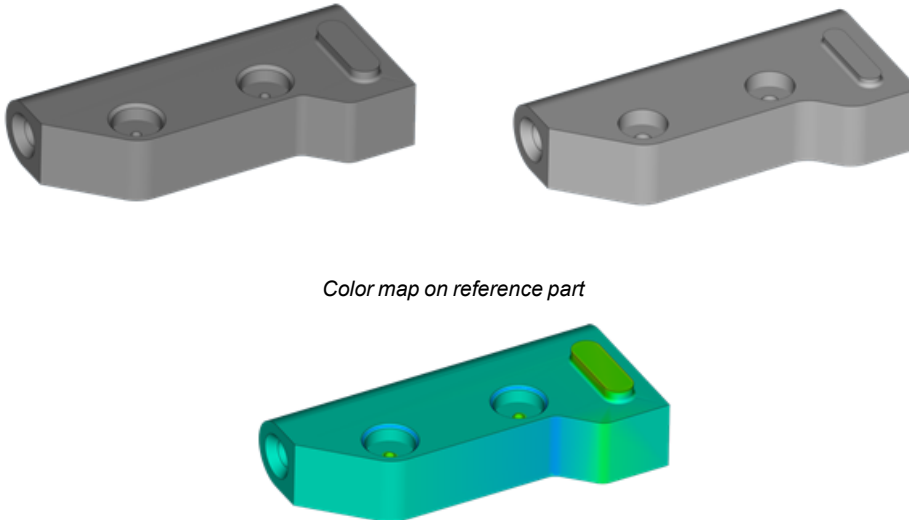
5. Part Comparison



The Part Comparison tool allows you to compare two meshes based on their triangle vertexes. The results are visualized in a color gradient map on the reference part, helping you to find the differences between the two selected parts.

Reference part

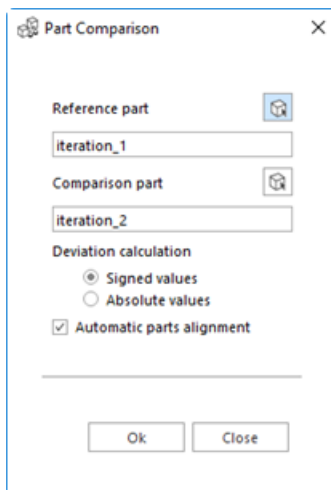
Comparison part



This tool can be used in various flows:

- Compare parts from different design iterations, to spot where the small differences are
- Analyze the shape deviation before-and-after repairing or editing the part mesh, to check if your part still lies in the tolerances you set
- Analyze the part deformation derived from the printing process or to compare the generated counter-deformed part that will compensate those distortions (in combination with Magics Simulation)

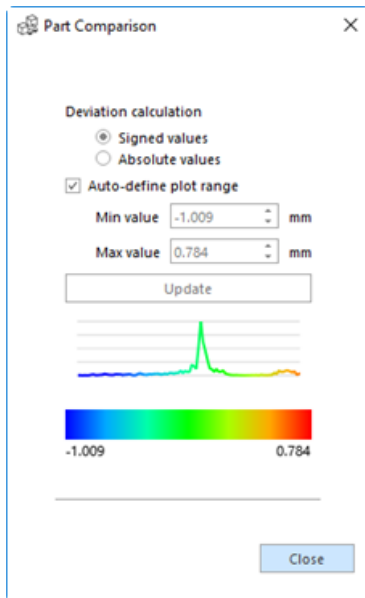
Settings



Reference part	Select the part that will be used as a reference during the comparison. The gradient color map of the results will be visualized on this part.
Comparison part	Select the part that will be used as a comparison part.
Deviation calculation	Select the analysis type used during the comparison. The type can also be changed on the spot later when visualizing the results.

	Signed values	Negative and positive deviations are used to differentiate between areas where reference part is inside the comparison part (negative) and where reference part is outside the comparison part (positive).
	Absolute values	No difference is made on values sign, only absolute values of deviation between both parts are calculated.
Automatic parts alignment	Enable this option when the 2 parts to be compared are not aligned with each other. An automatic alignment between the 2 parts will be performed before calculating the results.	

Results visualization



Deviation calculation	Select the analysis type used during the comparison. The type can also be changed on the spot later when visualizing the results.	
	Signed values	Negative and positive deviations are used to differentiate between areas where reference part is inside the comparison part (negative) and where reference part is outside the comparison part (positive).
	Absolute values	No difference is made on values sign, only absolute values of deviation between both parts are calculated.
Auto- define plot range	Enable this option to automatically visualize results on the part accordingly to the minimum and maximum deviation values present in the comparison.	
	Min. value	Manually define the minimum value of the color map. Deviations outside the range will be colored in grey.
	Max. value	Manually define the maximum value of the color map. Deviations outside the range will be colored in grey.

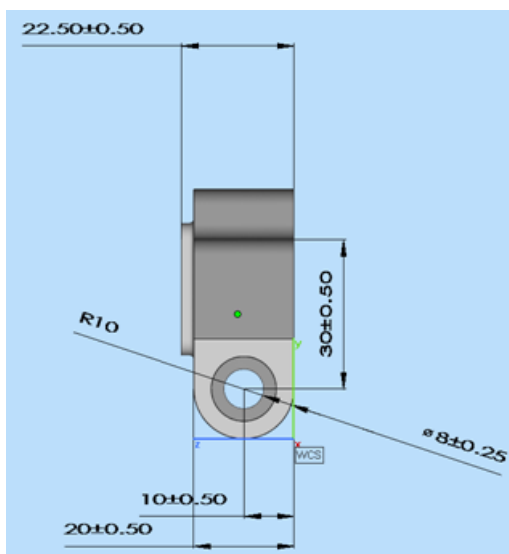
Update	Click on this button to update the chart based on top defined settings.
--------	---

10.4. Product and Manufacturing Information (PMI)

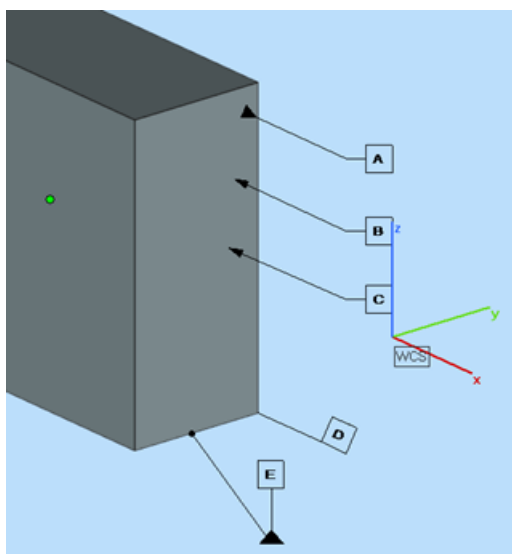
PMI is a set of data used to describe the technical specifications of a part that will be produced. This data can be added in 3D directly on the part, when the part is being designed in CAD, and saved together with the part in the proprietary file format. Magics has extended its import and can load not only the CAD part, but also the PMI information that has been added on the part, in the CAD software, during the design of the part. With it dimensions and annotations can be visualized. These measurements or annotations cannot be edited in Magics.


The following PMI elements can be loaded in Magics:

– Dimensions

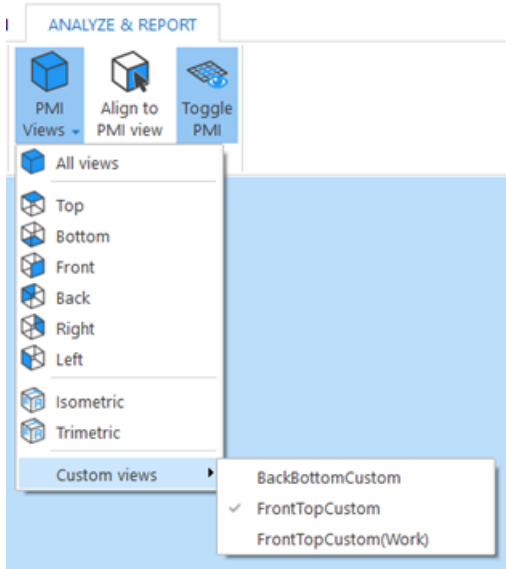


– Annotations



 **Note:** This extended import is possible only with the NX and Solidworks import modules and only if the part has, in fact, been saved with PMI information.

Under the “Analyze & Report” tab the PMI section can be found. It consists of three main commands as shown below.



PMI Views		Allows the user to choose a specific view for the part or show them all.
	All views	Shows all measurements and annotations that were placed on the part.
	Default views (Top, Bottom, ...)	Enables the user to switch between the default views of a part. Note: If the coordinates of the default views are different in the CAD software than in Magics, then these views will appear under Custom Views.
	ISO, Trimetric	Enables the user to quickly switch between Isometric and Trimetric views. The note from default views applies.
	Custom views	Shows the views that have been created by the user inside the CAD software.
Align to PMI view		Re-aligns the part to the previously selected view.
Toggle PMI		Shows/hides all PMI measurements/ annotations.



10.5. Report

1. Save Part(s) as 3Dpdf



Save the selected part(s) as 3Dpdf.

- See **Save Part(s) As 3D PDF, page 91**

2. Generate Report



Generate a Report in a document.

- See **Generate Report, page 77**

3. Create Report Template

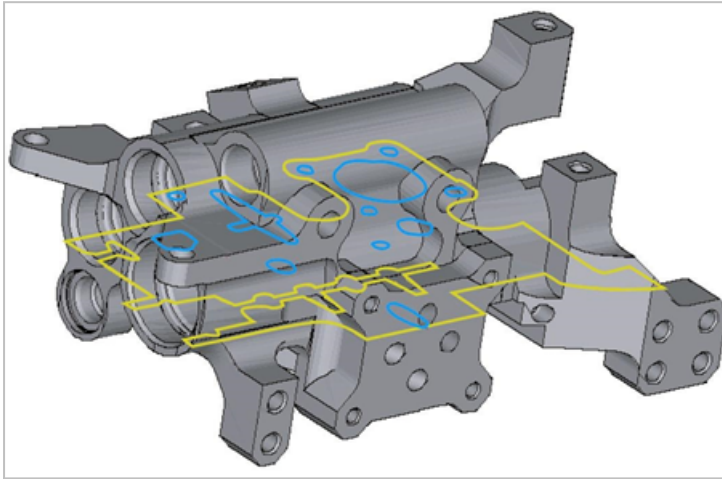


Create a report template in Microsoft Word or Microsoft Excel.

- See **Create Report Template, page 77**

Chapter 11. Slicing




Prepare sliced files for production on your RP machine. The Slice module generates files that can be sent directly to most RP machines. The preview function allows you to inspect the slices before the slice command is executed. In addition, the module can repair slices automatically.



11.1. Introduction

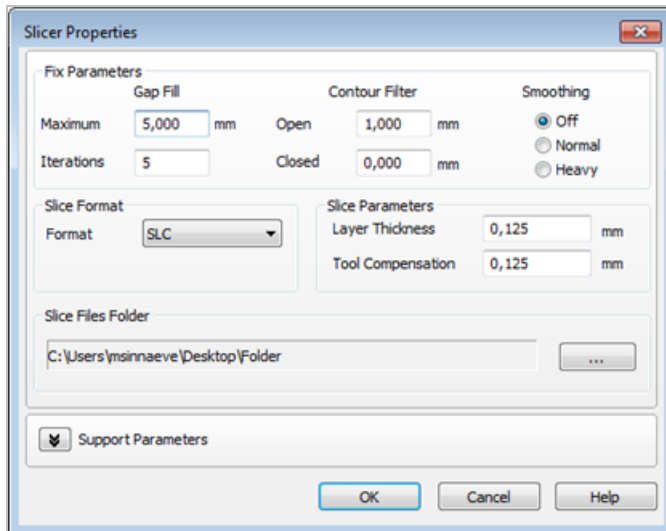
Magics allows you to output sliced files starting from STL-information. When you slice an STL-file, you make XY-cuts at different Z-positions that are related to the layer thickness used by the RP machine. If there are open contours, Magics' contour fixer automatically closes or fixes them. With the on-line slice preview, you can check the result before the slicing took place. The color of the contours indicates if you are working with a closed or an open contour.

11.2. The Slice ribbon

	The slice preview shows the contours of each slice
	Opens up the Slicer Properties Dialog Box, where you can set the parameters for the slices.
	Opens up the Slicer Properties Dialog Box, where you can set the parameters for the slices for the selected files.

1. Slicer Properties

Slice all, Slice selected and the parameter option in the preview slicer dialog box all bring you to the Slicer Properties dialog box, where you can set the parameters for the slices. The parameters set in the dialog are those of the currently selected machine, these parameters can be defined in the Machine Setup. (Menubar\ File\ Machine Set-up). The slicer properties dialog has two parts: the fix parameters and slice format part.



Fix Parameters

Gap Fill	Open loops (contours with gaps in it) or gaps in the contours are closed.	
	Maximum	This value displays the maximum gap that can be closed in a contour.
	Iterations	The contour fixer can work with iterations. Working with iterations results in more accurate fixing.
Contour Filter	The contour filter removes small or short contours automatically.	
	Open	All open contours shorter than this value are removed.
	Closed	All closed contours shorter than this value are removed
Smoothing	Overlapping vector points are united and the number of vectors is reduced. This reduces the size of the slice file but details may get lost. You can choose between Off, Normal and Heavy smoothing.	

Slice Format

Format	<p>The slicer supports the formats:</p> <ul style="list-style-type: none"> • CLI (Common Layer Interface from EOS), • CLI (float) (a CLI option for slicing at a higher resolution) • SLC (3D systems Layer Contour, SPI) • SSL (Stratasys), • F&S
Units	If you work with the EOS CLI format, a resolution is also asked. The default value is 0.05 mm.



CLI (float) has a resolution of 10^{38} pixels. Using CLI (float) it is possible to slice a file at a higher resolution than with the regular CLI. Please note that in this case, the size of the resulting file will be roughly twice the size of the file sliced with the regular CLI option.

Slice Parameters

Layer Thickness	Displays the layer thickness, as it will be used on the machine.
Tool Compensation	Displays the tool compensation (machine dependent). This is standard the radius of the used laser beam or the radius of particles of the used build material.

Slice Files Folder

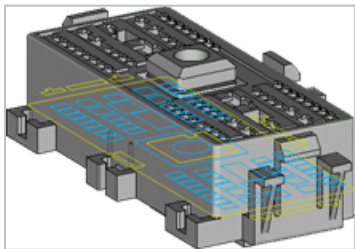
Slice Files Folder	Defines the output folder for the slice files.
--------------------	--

2. Preview Slicer

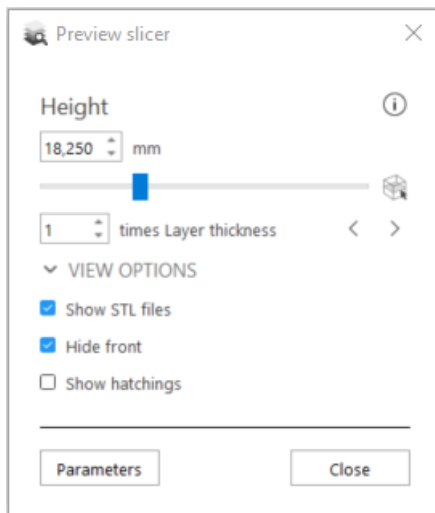
Preview Slicer command allows to see the contours of the slices. The contours are color-coded:


1. Yellow: external contours
2. Blue: internal contours
3. Red: open contours

The Preview Slicer also closes the gaps in open contours (contour fixer). The combination of the slice preview and the fixing tools, gives you a powerful tool for effective fixing. While you are fixing, you can use the slice preview to see if the contour fixer closes the open loops due to gaps between triangles. The slices are taken in the middle of a layer.



The Preview Slicer can be accessed from Slicing -> Basic ribbon.

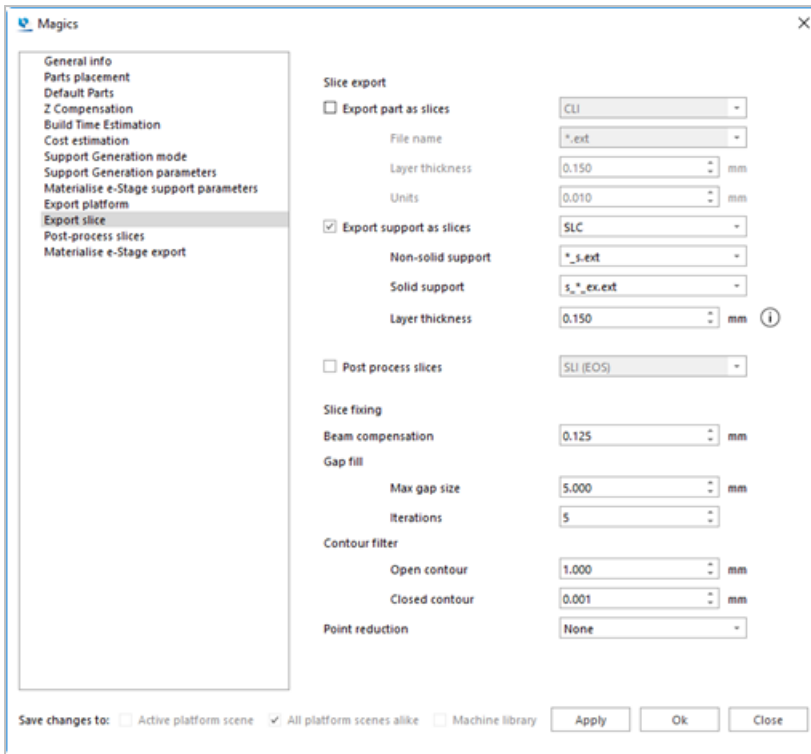


Height	This value displays the Z-position of the slice. You can change the position of the slice by typing in the desired value. Next press the enter-button. The slice height position should be a multiple of the Layer thickness. If you enter another number, this number will be rounded to the nearest layer position.
Slider	You can change the position of the slice using the slide bar.
	This button activates the Indicate Slice Pointer. You can indicate a point on the part and a slice preview will be displayed.
Slice Increment (Times Layer Thickness)	This value displays how many layers the increment is, which is used while you inspect slices with the slice preview.
Slice Walker < >	You can walk through the part and each slice preview will be made on top of each physical slice.
Parameters	On click on the button, the slicer properties dialog is displayed.
Show STL files	If this option is on, the slice is real-time displayed on the shaded image.
Show hatchings	A preview of the hatching is displayed on the screen. This allows you to predict if the machine will draw the hatching correctly on the part and if the hatching will be correctly trimmed.
Hide front	When this option is selected, only the part that is located behind the slice (as you see it on the screen) will be visible.

3. Machine Setup

Export slice page

The user is able to export the part and support as slices.



– Slice export

Export part as slices	Layer thickness	Displays the layer thickness, as it will be used on the machine.
	Format	The format of the part slices
	File name	The naming template of the file containing the slices. The character “*” will be replaced by the part name, while “ext” will be replaced by the slicing format chosen.
Export support as slices	Layer thickness	Displays the layer thickness, as it will be used on the machine.
	Format	The format of the support slices
	Units	If you work with the EOS CLI format, a resolution is also asked. The default value is 0.05 mm.
	File name	The naming template of the file containing the slices. The character “*” will be replaced by the part name, while “ext” will be replaced by the slicing format chosen.
Post process slices	Slices can be post processed. The format must be inserted.	

– Slice fixing

Beam compensation	The beam compensation takes into account the thickness of the laser beam to ensure the correct dimensions of the parts.	
Gap fill	Max gap size	Open loops (contours with gaps in it) or gaps in the contours will be closed. The maximum gap that must be fixed by stitching.
	Iterations	The number of iterations used when performing a stitch.
Contour filter	Open contour	All open contours are filtered out when the contour length is less than the entered value.
	Closed contour	All closed contours are filtered out when the contour length is less than the entered value.
Point reduction	Point reduction is used to reduce the amount of vectors. Successive vectors that lay on one line are merged. This reduces the size of the slice file but details may get lost.	

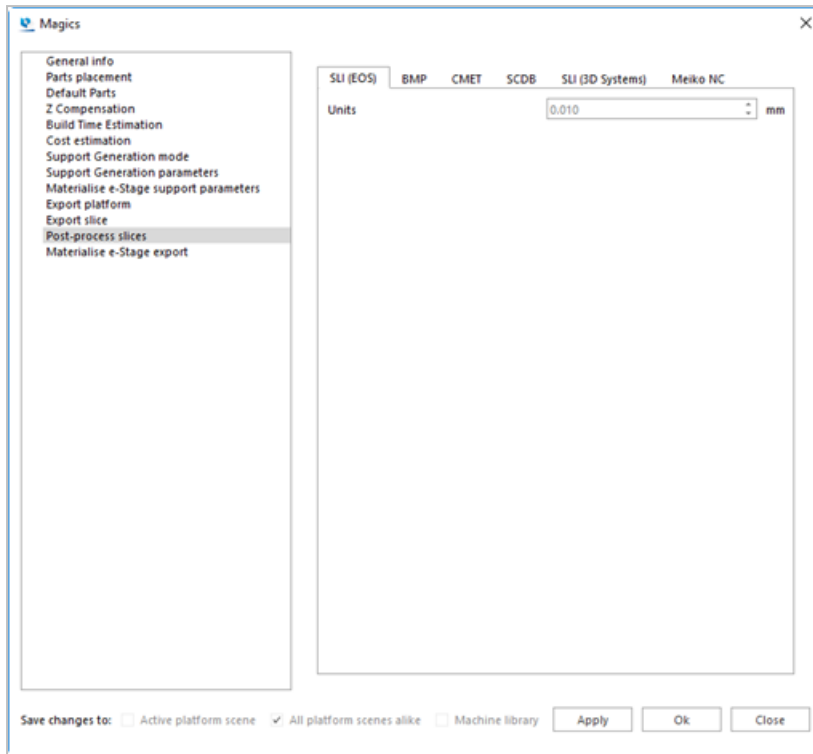
Post-process slices page

Post processing is an extra conversion to transform the slice format to a machine dependent slice format. It is then a two-step process: first the file is sliced to the format determined in the Slice properties dialog (see the section “Slicing”), and then this format is converted to the format you determine here.



Magics can post process to:

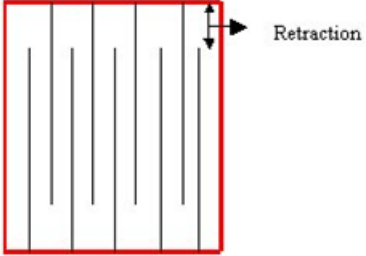
- SLI (3D systems) (License for C-Tools needed)
- SLI (EOS)
- SCDB (License for C-Tools needed)

Different parameters are present depending on the specific format requirements.



Resolution	<p>Unit size</p> <p>A position in the slice format is determined by 2 numbers</p> <ul style="list-style-type: none"> - The unit size - A unit value <p>The final position is determined by multiplying those 2 values. For example: when the unit size is 0.01 mm and the unit value is 1254, the position is $(0.01 \times 1254) = 12.54\text{mm}$</p> <p>It is clear that to keep a maximum of details, the unit size should be as small as possible. For the regular CLI slicing format the only restriction is that the maximum value for the unit value is 65536 (2^{16}). So the maximum distance you can cover with a unit size of 0.01 mm is 655.36 mm. ($=0.01 \times 65536$). If you have a bigger machine, the solution is to make the unit size bigger. (For example the EOS 700 has a platform with a length of 700mm, so a bigger unit size must be used to cover the whole platform).</p> <p>Thus, the first restriction for the unit size is: $\text{Unit size} > \text{max distance machine} / 65536$</p> <p>In the slice format, the layer thickness is also expressed in the unit size. This results in the restriction that $\text{Layer thickness} / \text{unit size}$ is an integer.</p> <p>So we should minimize the unit size keeping this in mind:</p> <p>$\text{Unit size} > \text{max distance machine} / 65536$</p> <p>$\text{Unit size} = \text{layer thickness} * N$ (where N is an integer)</p> <p>When a user works in inch, the unit size stays in mm, so the factor 25.4 is</p>
------------	---

	<p>introduced.</p> <p>Resolution</p> <p>An SL machine internally works with a resolution (the amount of defined positions per millimeter). Some machines use a unit size.</p> <p>Both values are linked: Resolution [1/mm] = 1 / Unit size [mm]</p> <p>A value is used to express the position of a point. The position is retrieved with the following formula:</p> <p>Position [mm] = value / resolution [1/mm] = value x unit size [mm]</p> <p>The value that is used is mostly expressed with 2 bytes, so it has a maximum value (2¹⁶=65536). This results in the fact that the higher the resolution, the smaller the covered area because the maximum value is reached faster.</p> <div style="border: 1px solid #00aaff; border-radius: 10px; padding: 10px; margin: 10px 0;"> <p> CLI (float) has a resolution of 10³⁸ pixels, so for practical purposes there is almost no limit. Using CLI (float) it is possible to slice a file at a higher resolution than with the regular CLI. Please note that in this case, the size of the resulting file will be two or more times bigger than with the regular CLI option.</p> </div> <div style="border: 1px solid #00aaff; border-radius: 10px; padding: 10px; margin: 10px 0;"> <p> Based on the differences, CLI or CLI (float) format option may suit the user's needs better depending on the platform size, expected precision, or resulting file size requirements.</p> </div>	
Hatching	<p>The hatchings are the hatches the laser makes to cure the volume inside the part. There are different parameters:</p>	
	X Hatch	The distance between 2 hatches in the X-direction.
	Y Hatch	The distance between 2 hatches in the Y-direction.
	Alternate Layers	When switched off, each layer is hatched in both directions (X and Y). When switched on, a layer is only hatched in one direction (e.g. X-direction), the next layer will be hatched in the other direction (e.g. Y-direction).
	Hatch Offset	This is the distance between the border and the hatches. This parameter is to compensate the thickness of the laser beam.
	Hatch Filter	Hatches shorter than this length will be ignored.
	Save first	By determining what is saved first, you

		can determine if the laser first scans the borders or the hatching.
Hatch style		A cured resin will shrink and this shrinkage will cause internal stress, this internal stress can cause deformations. Using special techniques of hatching, the internal stress can be minimized. This will minimize the deformations.
	Alternated	Instead of always hatching the part from e.g. left to right, alternated will hatch one layer from left to right en the next layer from right to left.
	Retracted	<p>The hatches are connecting the 2 borders of the part with each other. Because the cured resin of the hatches will shrink, they will pull at the borders and a deformation will occur.</p>  <p>To prevent that the hatches are connecting the 2 borders, you can use the retracted-option. The hatches will only be connected to one border and leave a space at the other border so the hatch is not connecting the 2 borders anymore and so the deformation will be minimized.</p>
	Staggered	When staggered is checked, the hatches will move each layer. In this way, the hatch comes between the 2 hatches of the layer below. You can compare it with building a wall with bricks, they are not placed perfectly on top of each other but they are moved a half brick each layer.
Skinfills	To improve the quality of the bottom and upper layer, you can apply extra hatching, called skinfills.	
	Angle	When the angle of the surface is lower than the parameter 'angle', skin fills will be applied.
	Number	The amount of layers below/above the upfacing/downfacing layers that should contain skinfills.
	Hatch	The distance between the hatches of the skinfill.
Merge	When slicing a platform with multiple parts, each part is sliced separately.	



	Merging these slice files, they will become 1 big SLI-file.	
	Merge all parts	All the SLI-files of the parts will be sliced in one SLI-file
	Merge all Supports	All the SLI-files of the supports will be sliced in one SLI-file
	First	To decide if Magics should first merge or hatch. When you choose to hatch before merging, the result is that the laser will hatch the parts one by one. When you first merge before hatching, the hatches will be calculated for all parts together, so it can be that multiple parts are hatched at the same time.
	Original files	To decide whether Magics should keep the original files it used for merging or not.

Chapter 12. Materialise Software

12.1. Streamics

1. Open Streamics client



Opens the associated Streamics Clients. The client can be associated in the settings.
(Settings – Modules – Streamics Client)

2. Save Selected Part(s) to Streamics



Saves the selected part to the Streamics Control System.

3. Save Platform to Streamics



Saves the active platform to the Streamics Control System.

4. Create Quote in Streamics



Create a quote from the selected part(s) in Streamics.

5. Create Order in Streamics



Create an order from the selected part(s) in Streamics.

6. Reset revision(s)

Reset revision(s) command can be called from the search. When applied, Magics will assign new GUID(s) to the part(s). After saving the part(s) to **Streamics**, new orphan part(s) will be created.

12.2. CO-AM

1. Magics integration with CO-AM

The CO-AM Software Platform is a cloud-based solution that allows manufacturers to scale their digital supply chain with:

- Modular solutions (e.g. order management, MES, quality module, Materialise Apps)
- Partner ecosystem of 3rd party software and hardware systems

CO-AM improves collaboration, manufacturing repeatability, quality compliance and end-to-end data security, and now Magics can be a part of the CO-AM workflows through the seamless integration.

2. CO-AM settings page in Magics

There is a CO-AM settings page in Settings -> Modules -> CO-AM.

To start working with CO-AM parts in Magics, first go to your CO-AM account on the web, and initiate a connection. Until then, you will see that your Magics instance is not connected to any CO-AM accounts as shown below.



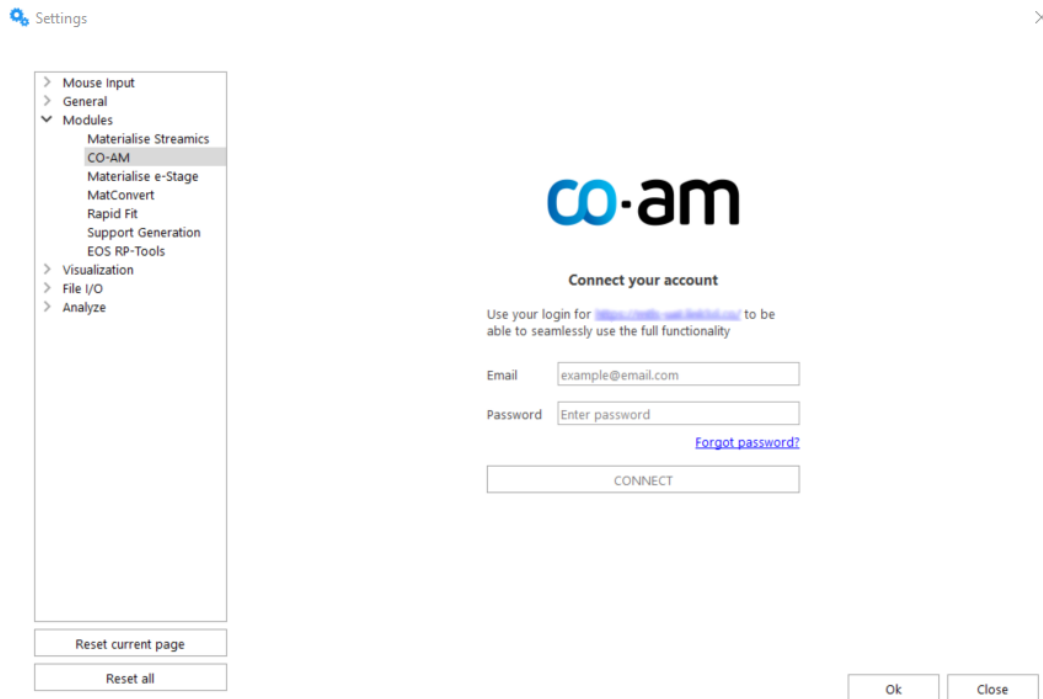
3. Connect with CO-AM

To connect Magics to a CO-AM account, go to the CO-AM web portal and initiate the connection. Consult CO-AM documentation or your customer success representative for more information on how to do it.

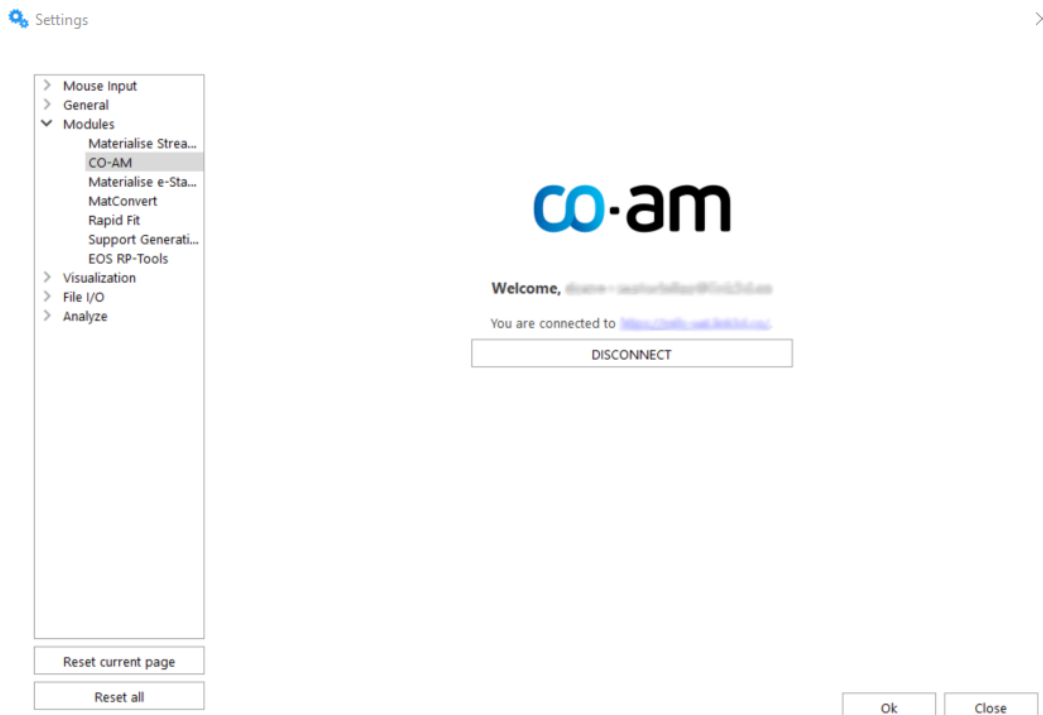
When you see a pop-up in your browser with the title "Open Materialise Magics xx.x?" and agree to open Magics, then Magics will be launched and the dialog with the log-in form will open on Settings -> Modules -> CO-AM page.

4. Log in with CO-AM account

When you see the log-in form, use the same account credentials as in your CO-AM web account and click the **Connect** button.



When you successfully connect an account, you will see the account and the CO-AM server you are connected to.



After connecting the account click the **OK** button in the settings dialog to apply this CO-AM account to your Magics instance.

5. Disconnect CO-AM account

When you click on the **Disconnect** button on the CO-AM settings page, then you will see the log-in form again. To finish disconnecting, click **OK** in the Settings dialog.

6. Using Magics with multiple CO-AM accounts

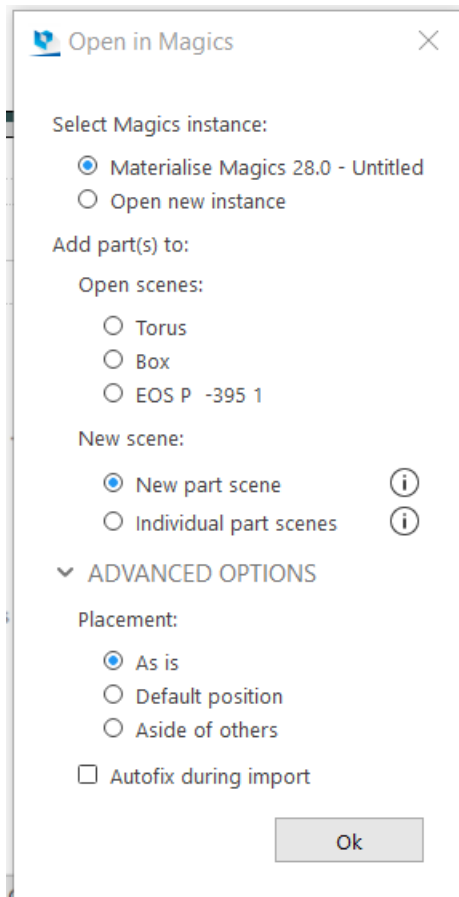
If you are working with multiple CO-AM web accounts on the same or different servers, when you initiate a connection from a different account than the one you are already logged in with, then Magics will open a new instance for each account and ask you to log in.

7. Open CO-AM part(s) in Magics

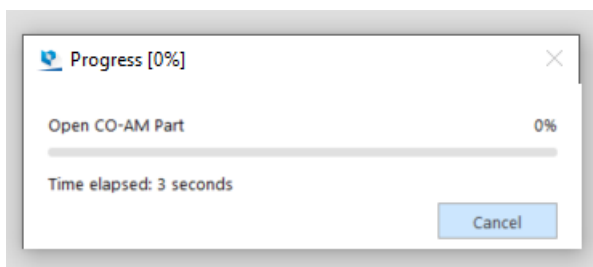
In CO-AM interface there are multiple pages that give the option to open parts in Magics.

- Locate "Open in Magics" button in CO-AM.
- Depending on the flow, select one or more parts and click "Open in Magics".
- If there is one or more Magics instances already opened, then Magics will show a dialog for selecting an instance and scene for the part(s).

Select Magics instance	Open new instance	Select this option to open a new Magics instance.
	{Instance name}	Select this option to open parts in a Magics instance that is already opened.
Add part(s) to	Open scenes	All scenes available for CO-AM parts are listed in the order they are open in the selected Magics instance. All mesh part scenes are shown. Platform scenes created from CO-AM are shown.
	New scene	New part scene: All selected parts will be opened in a single part scene in Magics. Individual part scenes: All selected parts will be opened in individual part scenes in Magics, one part per scene.
Advanced options	See 2 Import Part for details	



Click **OK** to start loading the parts. Magics will show a progress dialog while it is opening selected parts.

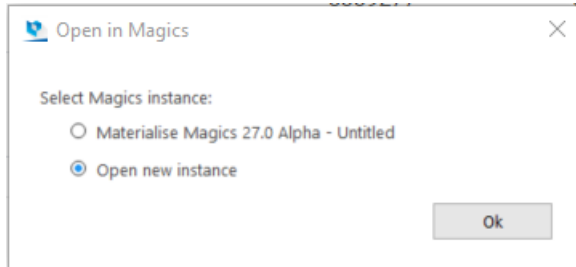


8. Open CO-AM platform in Magics

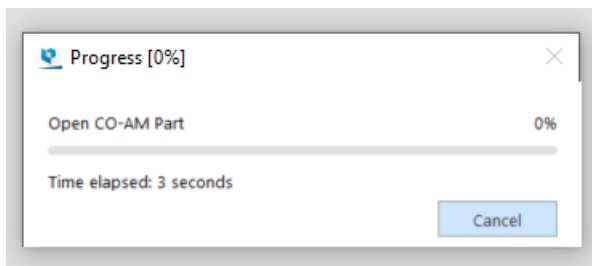
In CO-AM interface there are multiple pages that give the option to open a platform in Magics.

- Locate "Open in Magics" button in CO-AM for platforms on the Machine platforms list, in created jobs etc..
- Select the platform and click "Open in Magics".
- If there is one or more Magics instances already opened, then Magics will show a dialog for selecting an instance and scene for the platform opening.

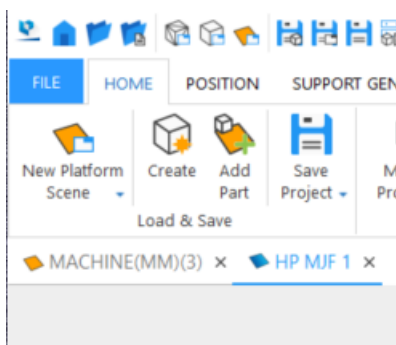
Select Magics instance	Open new instance	Select this option to open the platform in a new Magics instance.
	{Instance name}	Select this option to open the platform in a Magics instance that is already opened.



Click **OK** to start loading the platform with parts. Magics will show a progress dialog while it is opening parts on the platform.



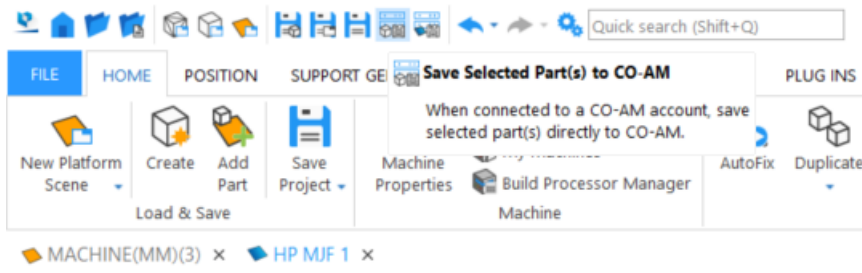
CO-AM platforms can be distinguished from other platforms in the same instance of Magics by the blue color of the icon in the platform scene tabs (i.e. "HP MJF 1" on the image below).



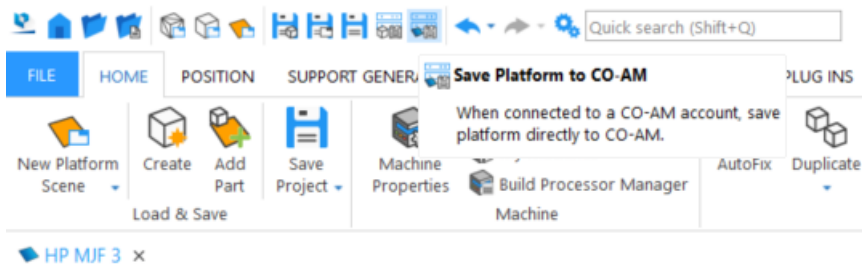
9. Save parts and platforms to CO-AM

When Magics is connected to a CO-AM account, then **Save selected part(s) to CO-AM** and **Save Platform to CO-AM** buttons appear in Magics interface:

- (1) (2) in the quick access bar
- (3) in the part context menu
- (4) under **File** -> **Save** menu



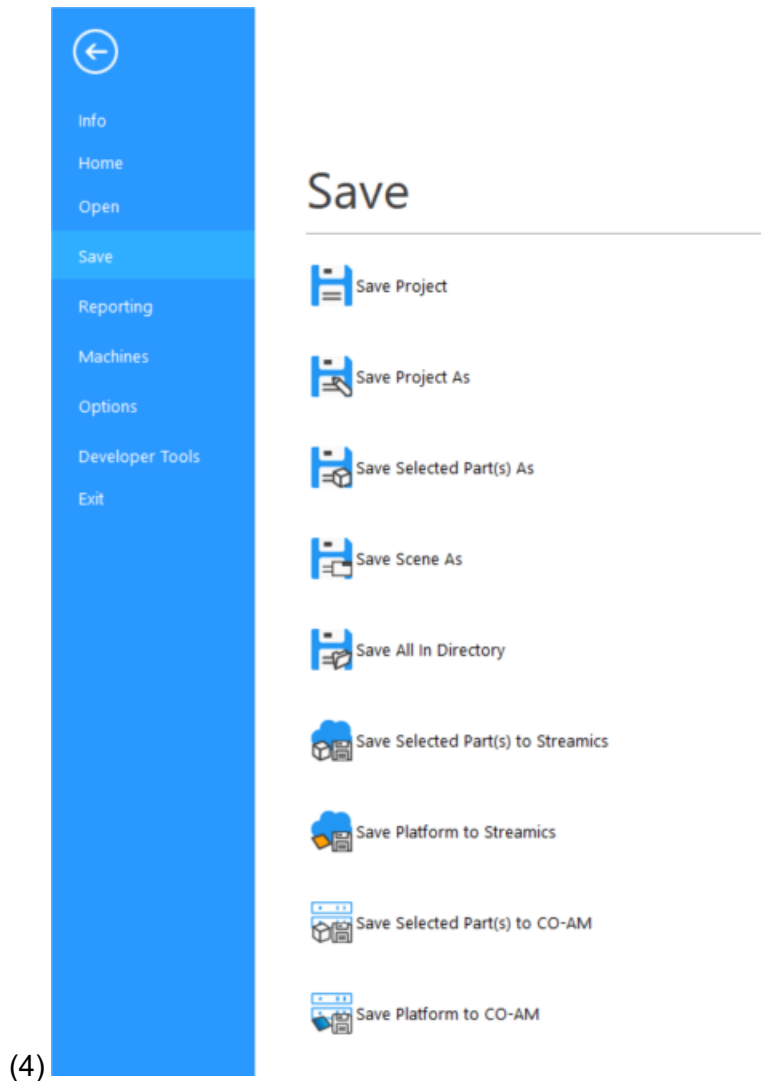
(1)



(2)



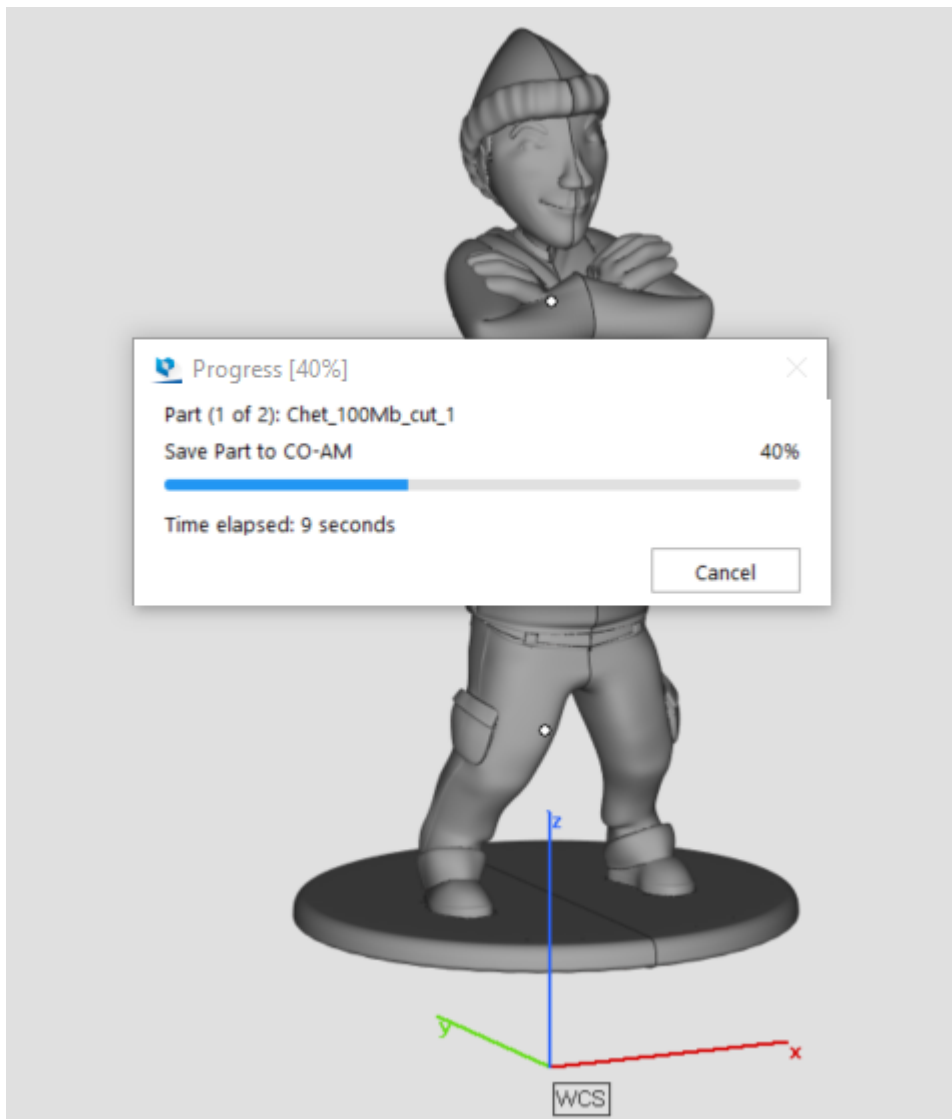
(3)



If user made any changes to the part since you opened or last saved the part to CO-AM, then the **Save selected part(s) to CO-AM** button will be disabled.

It is possible to save one or more CO-AM parts at a time.

When Magics is saving a platform or part(s) to CO-AM, then a progress dialog will be shown:



Given part meshes were changed on the CO-AM platform in Magics, when user clicks **Save Platform to CO-AM** button, then changes to the parts will be saved automatically.



If user applies "Change Machine" command in Magics, then the platform disconnects from CO-AM, and the metadata and traceability with CO-AM get lost. Only machines originating from CO-AM can be saved back to CO-AM.

10. Supported Magics commands for CO-AM parts

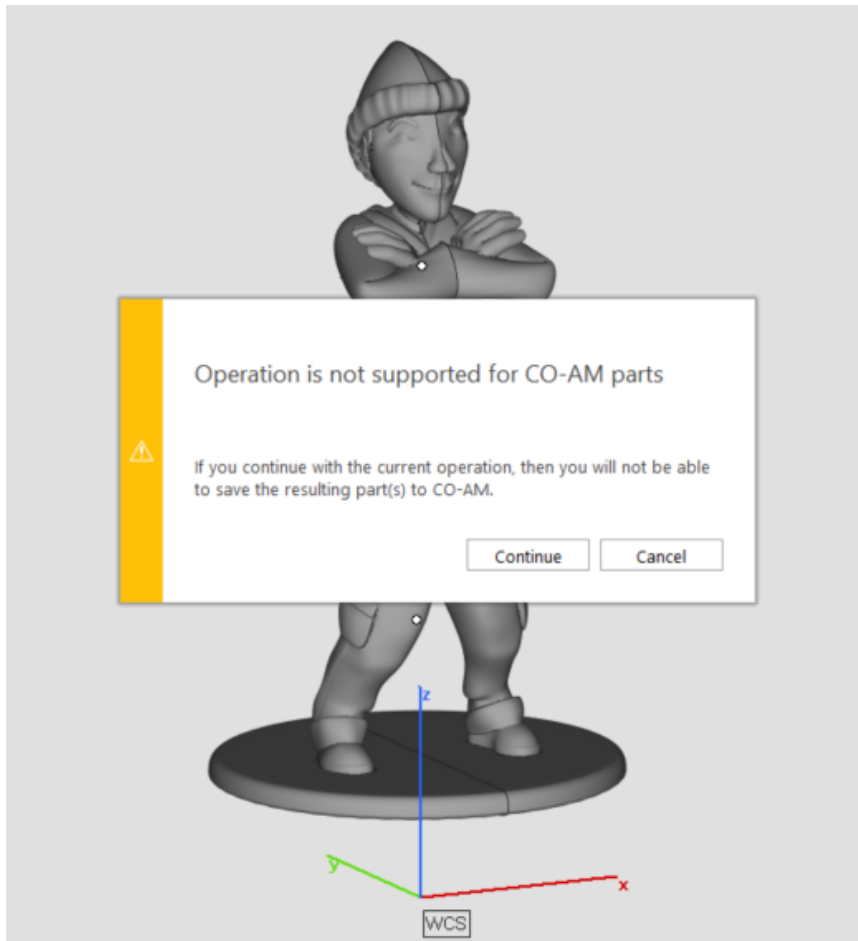
Part commands

You can use most Magics commands on a part scene.

You cannot save to CO-AM duplicates of the original CO-AM part that you create through the following commands:

- Duplicate
- Rotate with “Make copy” enabled
- Translate with “Make copy” enabled
- Mirror with “Make Copy” enabled
- Rescale with “Make Copy” enabled

You will see a warning when you try to perform a command that is not supported or that will break the link with CO-AM.



Platform commands

Most platform preparation commands work with CO-AM platforms.

Primary limitations for platform preparation in Magics 27:

- Sinterboxes cannot be saved to CO-AM
- Labels can only be saved as stls
- Parametric supports are converted to stl when saved to CO-AM

11. Connection of Parts and Platforms with CO-AM

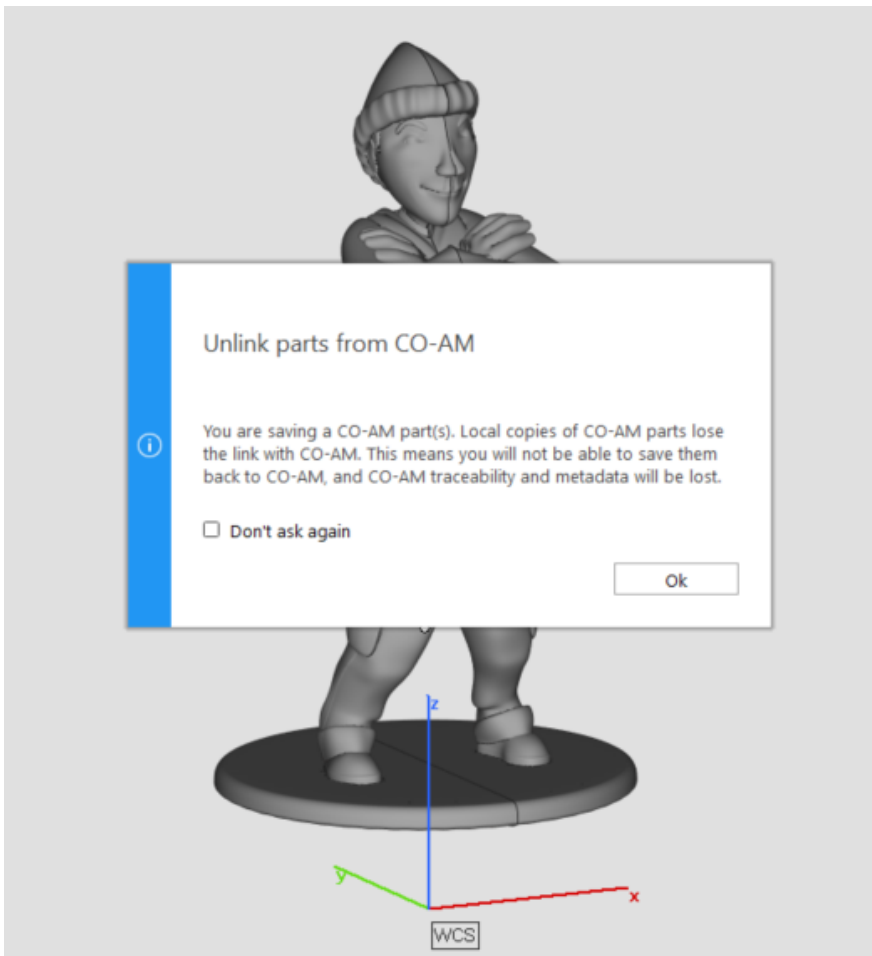
Parts opened in Magics from CO-AM maintain a connection with CO-AM for traceability and auditability.

The connection is maintained in the following cases:

- While the instance of Magics with the part or platform is open
- In the Emergency mode in case of losing a Magics license
- When parts or platforms are captured for autorecovery in case of a crash

A part loses the link with CO-AM if you perform one of the following operations:

- When you save a CO-AM part locally. The local copy of the part will not have the link with CO-AM anymore, you will lose CO-AM traceability and metadata, and will not be able to save the part to CO-AM.
- Merge a CO-AM part with another CO-AM part (cutting a part and saving the cut parts to CO-AM will result in having multiple CO-AM parts on the same scene)
- Boolean - unite
- Boolean - intersect parts
- Create new composed part



A platform loses the link with CO-AM if you save a CO-AM platform locally. The local copy of the platform will not have the link with CO-AM anymore, you will lose CO-AM traceability and metadata, and will not be able to save the platform to CO-AM.

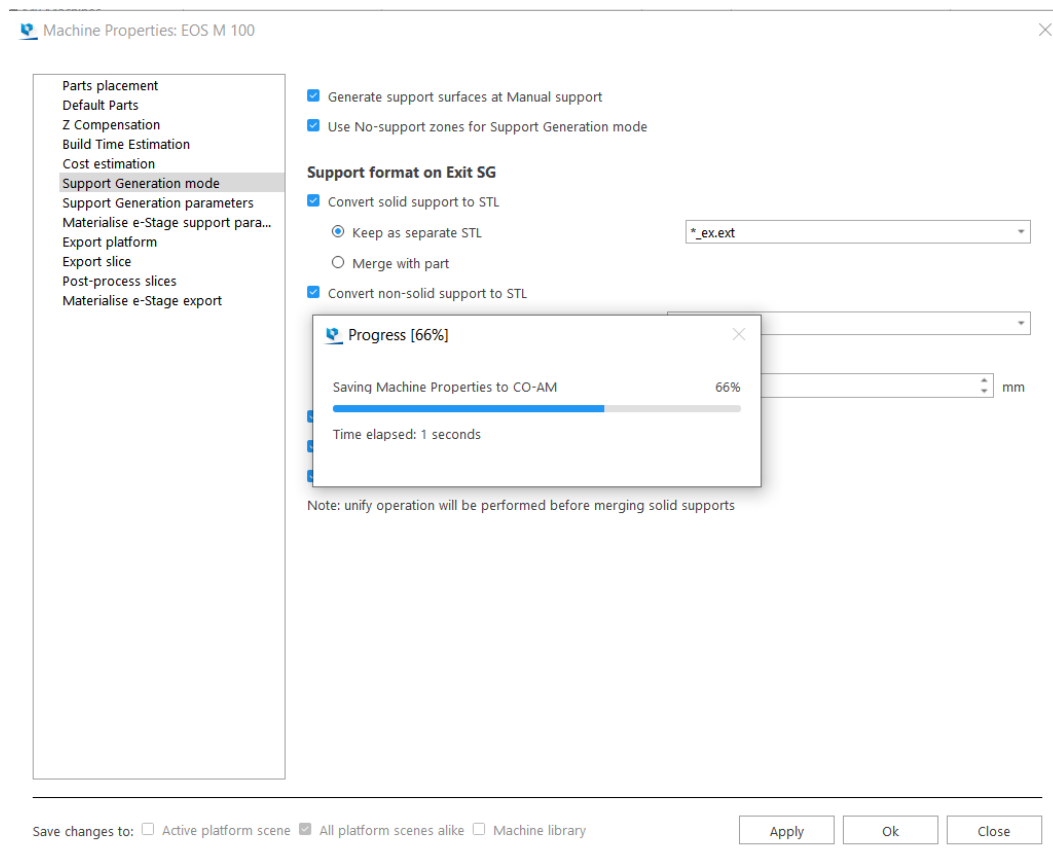
12. CO-AM Machine Properties

Machine Properties dialog contains physical machine parameters usually found on the General info page (Build envelope, Configuration, Fields overlaps etc.) and build preparation parameters (Parts placement, Support Generation, Export platform etc.).

General info parameters are managed from CO-AM side. Build preparation parameters on the other hand are controlled from Magics.

When a machine is opened from CO-AM for the first time, then the physical machine parameters are received from CO-AM. The rest of the parameters are generated from Magics defaults.

Given the Machine Properties dialog is open and one or more parameters are changed, when the parameters are applied to the current scene, then they are also saved to CO-AM.



When Machine Properties are changed in Magics and applied to the current scene, then the other scenes in the current instance of Magics with the same CO-AM machine will be updated with the same parameters.



When the machine with updated parameters is reopened in Magics from CO-AM, then the latest saved parameters are applied to the scene in Magics.



Some parameters are not saved and are set to the defaults every time the machine is opened from CO-AM.

Magics will use the default parameters for Cost estimation every time this machine is loaded from CO-AM and will not save your changes on this page to CO-AM.

Save changes to: Active platform scene All platform scenes alike Machine library

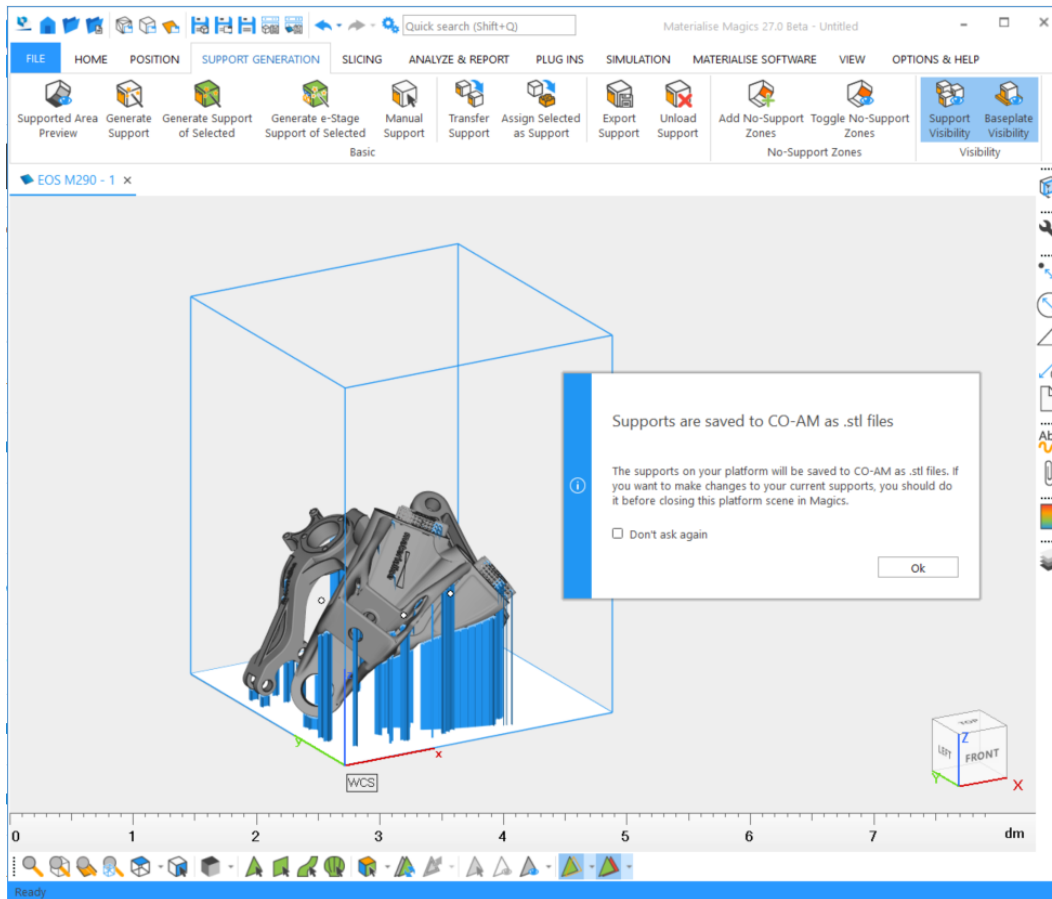
Machine Properties pages overview for CO-AM machines:

Parameters page	CO-AM integration support
General info	Managed from CO-AM side. Parameters page is hidden
Parts placement	Managed from Magics and saved to CO-AM
Default Parts	Can be changed in Magics for the current scene but can't be saved to CO-AM in Magics 27
Z Compensation	Managed from Magics and saved to CO-AM
Build Time Estimation	Can be changed in Magics for the current scene but can't be saved to CO-AM in Magics 27
Cost Estimation	Can be changed in Magics for the current scene but can't be saved to CO-AM in Magics 27
Support Generation mode	Managed from Magics and saved to CO-AM
Support Generation parameters	Managed from Magics and saved to CO-AM
Materialise e-Stage support parameters	Managed from Magics and saved to CO-AM
Export platform	Managed from Magics and saved to CO-AM
Export slices	Can be changed in Magics for the current scene but can't be saved to CO-AM in Magics 27
Post-process slices	Can be changed in Magics for the current scene but can't be saved to CO-AM in Magics 27
Materialise e-Stage export	Managed from Magics and saved to CO-AM

13. Supports on CO-AM platforms

To be able to generate supports for parts on CO-AM platforms, it is necessary to manually choose set Support generation parameters in the Machineproperties(see4 Support generation parameters).

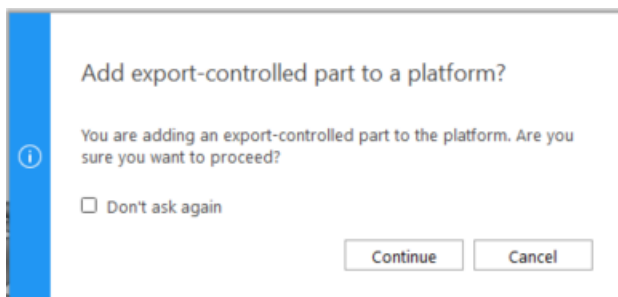
Platforms with supports can be saved to CO-AM as usual, but in the current version of Magics supports will be converted to stl supports during saving, so after reopening a saved platform with supports from CO-AM the supports can not be modified, and need to be regenerated again if necessary.



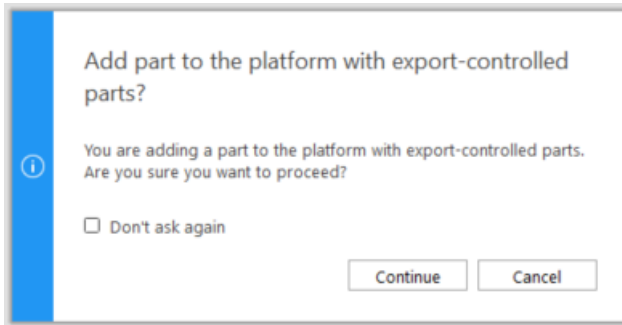
14. Export-controlled parts

CO-AM as a platform supports workflows with exported-controlled parts. Export-controlled parts are parts the production of which falls under the International Traffic in Arms Regulations (ITAR).

Magics is aware when ITAR parts are opened from CO-AM. If an ITAR part is being added to a CO-AM platform without any other ITAR parts, then Magics will notify the user about it in a message.



If a non-ITAR part is being added to a platform with ITAR parts, Magics will also notify about it.



These messages can be dismissed from being shown again. To re-enable the messages, go to *Options & Help -> Settings -> General -> Warnings* and select the corresponding checkbox.

12.3. Machine Manager

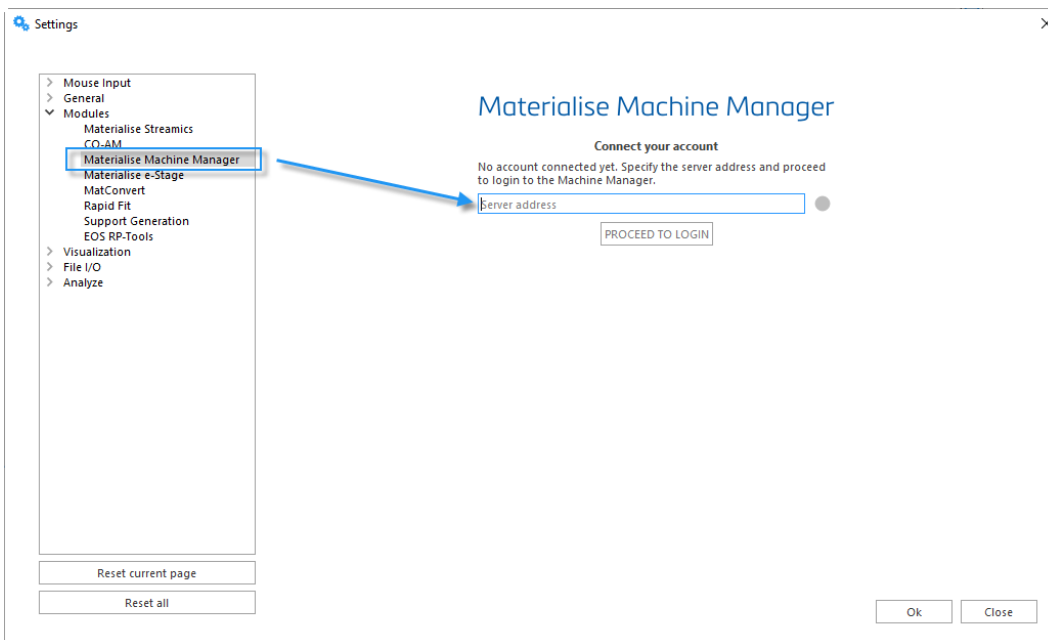
1. Magics integration with Machine Manager introduction

The Machine Manager is the home for the next generation of Build Processors. It is a cloud-based solution to manage machines, process parameters and build jobs. This new generation of Build Processors increase the versatility and scalability of your production as well as the productivity of your applications.

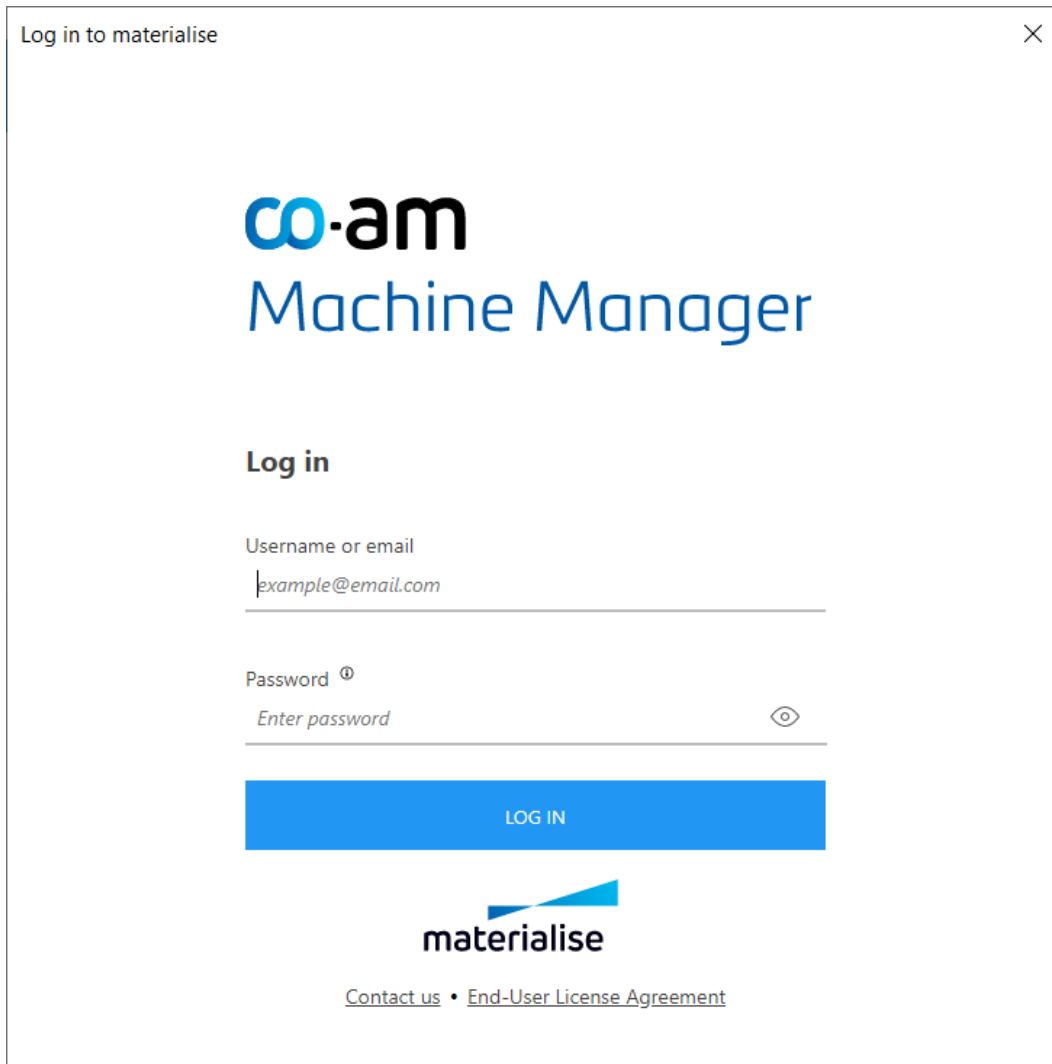
To enable an end-to-end work flow the integration of Magics with the Machine Manager is crucial. Below you can find out how this integration works.

2. Connect and Login to Machine Manager

To start working on a Machine Manager machine, first go to Settings page in Magics, under the Modules category, select Materialise Machine Manager page, where the server address needs to be added.



After "Proceed to login" button was clicked, the login page will be prompted:



The screenshot shows a web browser window titled "Log in to materialise". The page features the "co-am Machine Manager" logo at the top. Below the logo, there is a "Log in" section with two input fields: "Username or email" containing "example@email.com" and "Password" with a placeholder "Enter password" and a visibility toggle icon. A blue "LOG IN" button is positioned below the password field. At the bottom of the page, the "materialise" logo is displayed, along with links for "Contact us" and "End-User License Agreement".

After you have pressed the "Log in" button, you will have access to the machines available on the server. A stable connection to the server is needed to work seamlessly on a Machine Manager machine in Magics.

3. Machine selection

A machine or a New platform can be opened following the familiar work flow, also described in 1 New Platform Scene chapter.

Available machines on the selected server will be present in the Select Machine drop down list together with local library machines and Build Processor System machines.

On the New Machine / New Platform / Change machine dialog Support profile can be selected from available ones on the server.

After a platform was opened, you can proceed with data and build preparation operations as on a regular local library machine or Build Processor System machine.

4. Build preparation

Support generation profiles

Compared to local library machines and Build Processor System machines, Machine Managers machines store the support generation profiles of the machine on the server, together with the instance of the machine. Users with access to the machine will be able to use previously saved support generation profiles.

Saving support generation profiles to Machine Manager server

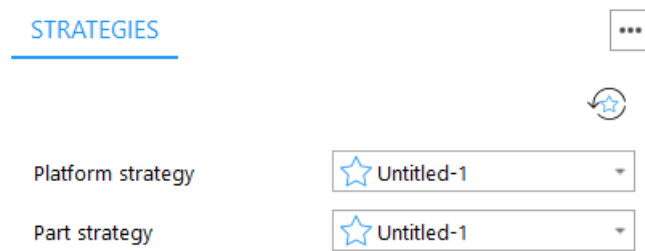
Support generation profiles can be saved on Machine Manager server from Magics by selecting "Machine Manager" check box of the "Save changes to" options on the Machine properties dialog.

Changes made on Part placement, Z compensation, Support generation mode, Support generation parameters, Materialise e-Stage support parameters, Export platform, Materialise e-Stage export pages in Machine properties will be saved to Machine Manager together with the machine.

5. Strategy assignment

Default strategies

Strategies are defined on Machine Manager terminal. When opening a Machine Manager machine in Magics, then the "Strategies" tool sheet will be available on the platform scene in the Tool pages with relevant strategies for the opened machine:



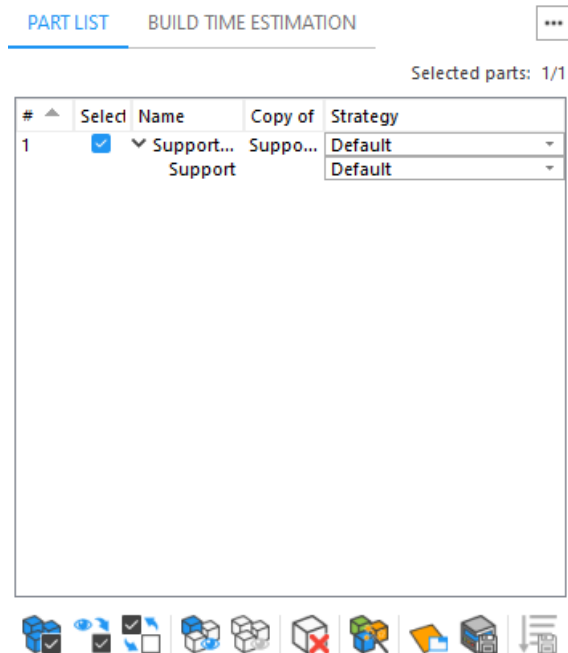
Available strategies represent the default strategies for platform and parts on the platform.

Star icon in front of the strategy name in the drop down list indicates the strategies marked "Set as default preferred" on Machine Manager.

"Reset to defaults" button will reset selections of the strategies to the ones marked "Set as default preferred" on Machine Manager.

Overriding default part and support strategies

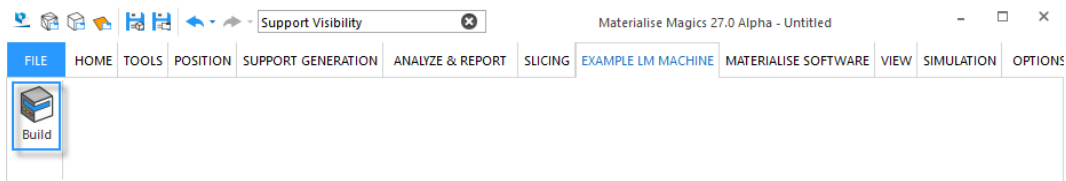
For each of the parts on the platform it is possible to select an overriding strategy. Available strategies to override part and support strategies can be found in Strategy column on the Part list tool sheet. Supports of a part are treated as one entity and the processing strategy will be applied to the merged support surfaces.



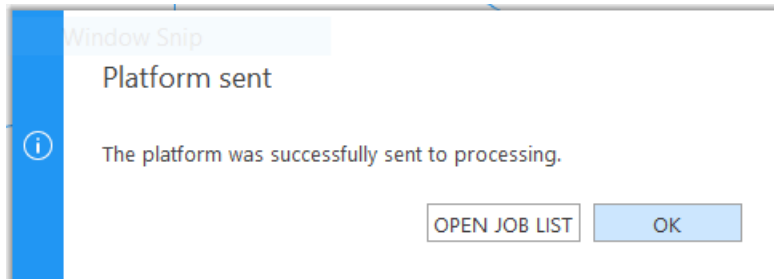
Sending for processing on Machine Manager

After strategies have been assigned, the platform is ready to be sent for processing to Machine Manager.

For an opened Machine Manager machine, a new ribbon is available, carrying the name of the machine. On the ribbon, the "Build" button will initiate upload of the platform to Machine Manager.



With a successful upload and creation of a processing job on the Machine Manager, a confirmation message will be prompted in Magics:

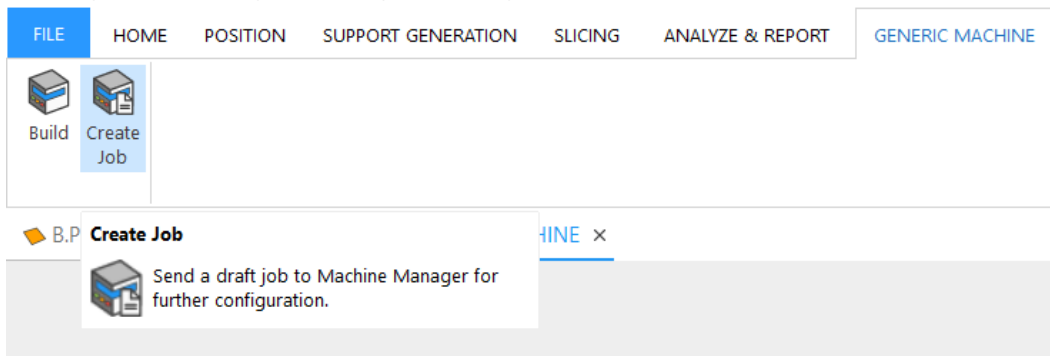


"Open job list" button will redirect to the Job list page on Machine Manager portal, where progress of the processing can be checked.

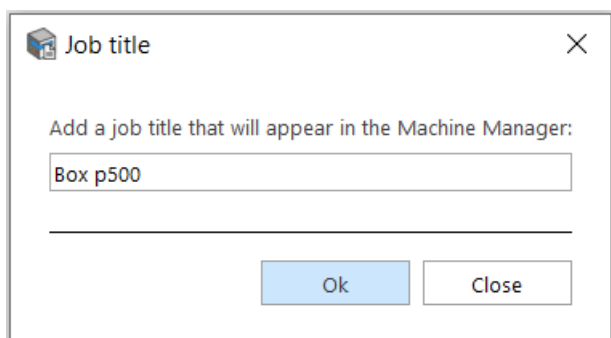
Redirect to Machine Manager for Strategy assignment

An alternative way to assign strategies is to create a draft job and get redirected to the Machine Manager terminal.

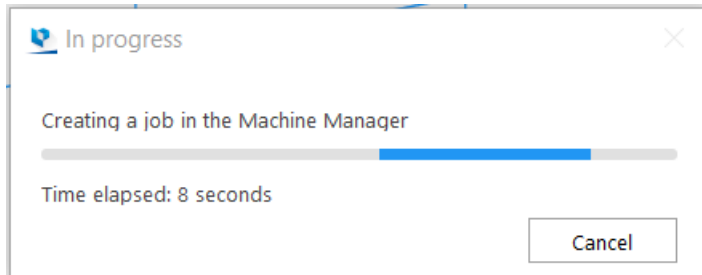
Under the machine ribbon, click "Create job"



In the dialog that appears, assign a job title for the Machine Manager:

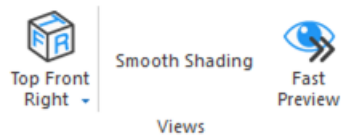


The job will take some time to upload to the Machine Manager, after which the Machine Manager page with the draft job will open in the browser. Proceed to assign strategies in the Machine Manager.



Chapter 13. View

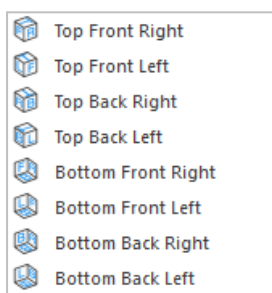
13.1. Views



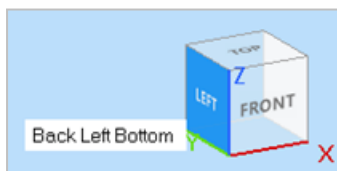
1. ISO views



This dropdown allows you to access the following views:



These views are also accessible by hovering over the corners of the interactive view cube, situated at the bottom right of the scene.



2. Smooth Shading

When Smooth Shading is enabled, the variations in color will be shown more gradually and no longer as separate triangles. Note that only the visualization of the part changes, the number of triangles and the accuracy of the STL are not changed.

3. Fast Preview



Fast Preview allows the application to only render wireframes or points of parts. It will speed up the visualization of 3D views.



13.2. Elements

1. Toggle Grid



Toggles the visibility of the grid.

- For grid settings, see paragraph **Grid, page 538**.

2. Rulers



Toggle the visibility of the Rulers. Rulers have been introduced into Magics to give the user the possibility to estimate the dimensions in which he is working. The rulers can be placed at the bottom of the working area and/or in the left side of the working area.

These settings can be defined in the Rulers visualization parameter window (Settings > Visualization > Rulers).

3. Coordinate System



The Coordinate System is called WCS (World Coordinate System) and it is the default coordinate system in which you work. The origin is set in (0,0,0).

Sometimes, for example when the part is zoomed in, or when the part is positioned far from the origin, this coordinate system is not visible on the screen. Therefore, you can switch on the Orientation Indicator.

4. No-build zones

No-build Zones		Toggle to show the No-build zones, if present.
Field Overlaps		Toggle to show Field overlaps, if present.
Platform Visibility		Toggle the Platform Visibility. (F11)

5. Part Dimensions



Activating the part dimensions gives a view on the bounding box with the dimension of each rib indicated.

6. Gravity Center



This command visualizes the gravity center of the selected part with a green dot.



7. Combined Bounding Box

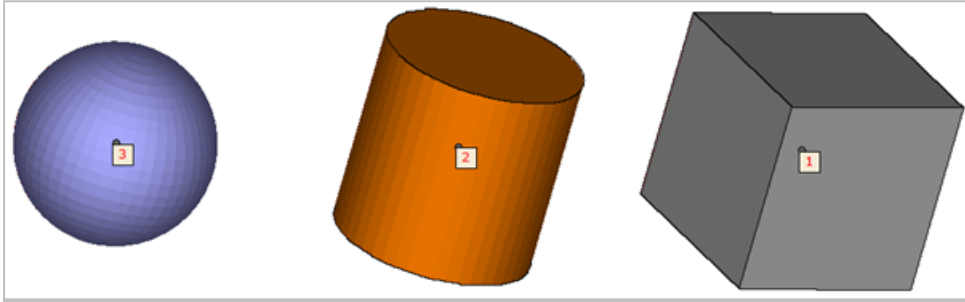


This command toggles the visibility of the bounding box of all selected parts. In addition, every minimum and maximum value of x, y and z are shown.

8. Tag ID



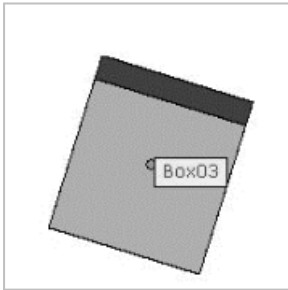
Every loaded part receives an ID in Magics, with this functionality it can be visualized.



9. Tag Names



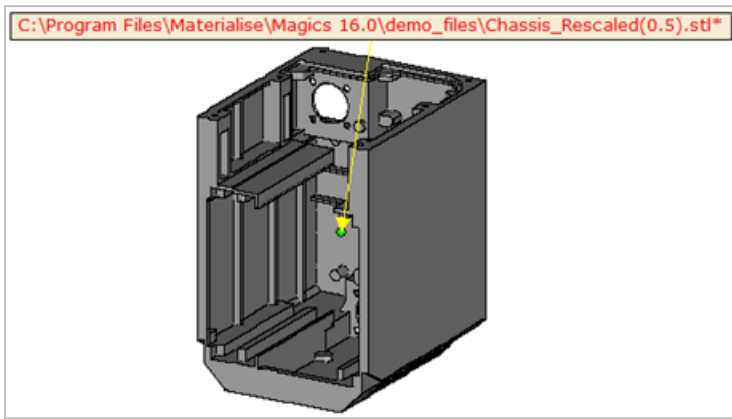
This command pasts the name of the part to the selection tag of the part. (F10)



10. Tag Path



By selecting this, it shows the path of the part nears its selection tag. (F12)

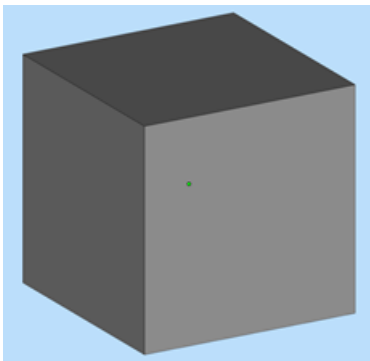


11. Selection Points

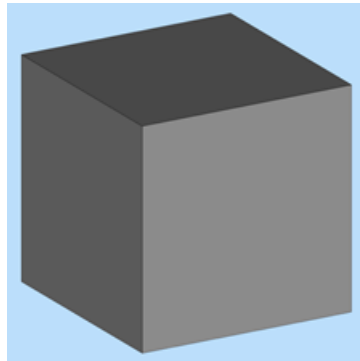


This command hides or shows the selection points on the parts. This can be useful if the you want to take screenshot of the parts.

On



Off



12. Toggle Graph Rendering



Toggles rendering of graphs.

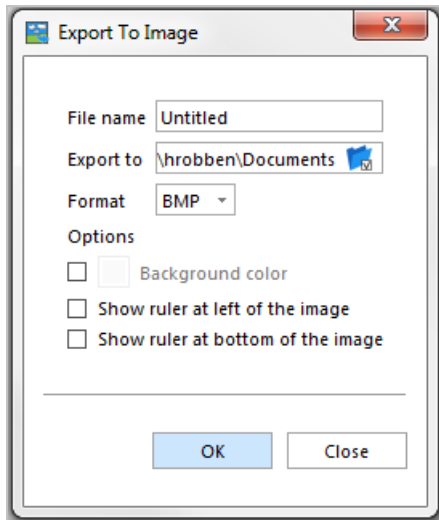
This command can be used to switch the lattice graph rendering between 3D beam rendering or line rendering. For more information, see graph rendering settings in **Graphs, page 534**

13.3. Export View

1. Export to Image



The current view can be exported as Bitmap, GIF, JPEG, TIFF, PNG file. You can enable the rulers and choose a background color. When no background color is selected, the color of the scene will be used.



2. Copy Viewport



To make screenshots of the Main Window, you can use this feature.

3. Print



This command starts the Magics print wizard that leads you to the Page Setup and the standard Windows Print dialog box.

- See ***Print***, page 91



Chapter 14. Options & Help

14.1. Settings



In the Settings dialog you can alter all parameters of your Magics Software.

1. Mouse Input

Customize Mouse Input

The mouse buttons are fully customizable by the user.

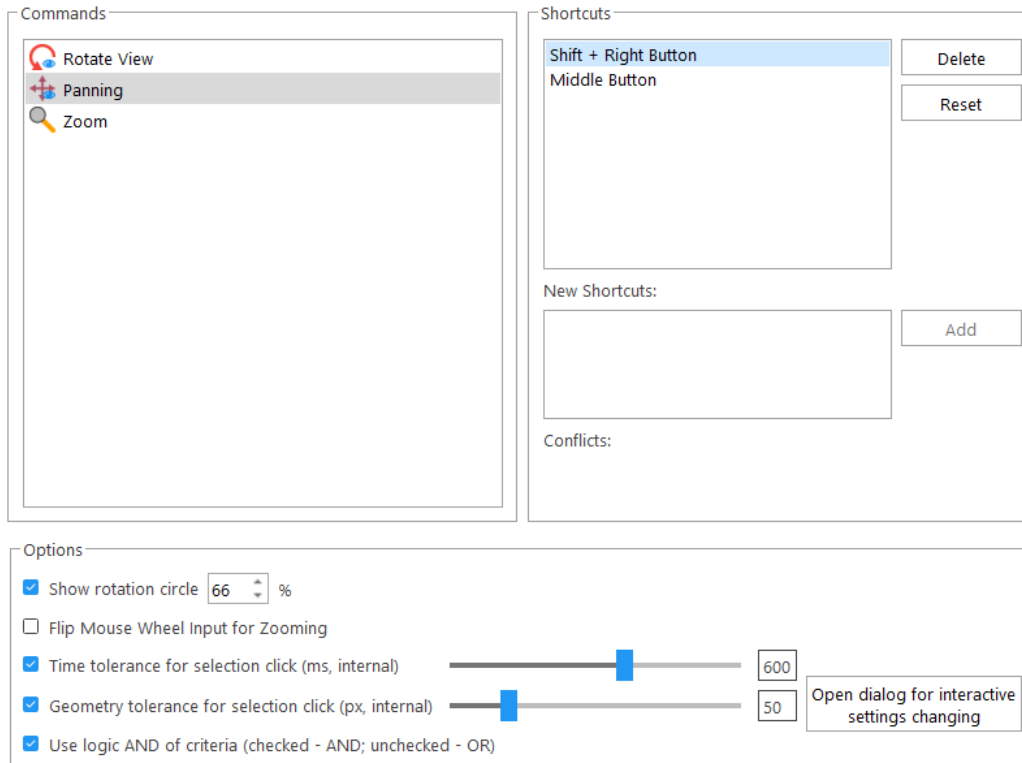
Command	This list contains the available functions for the mouse.
Shortcuts	This list represents the current shortcuts for the selected function in the Function List.
Delete	Deletes the selected shortcut.
Reset	The shortcuts for the selected function will be reset.
New Shortcut	By clicking in this textbox and clicking on the desired combination of (mouse) buttons, the shortcut will appear in this box.
Add	Adds the shortcut to the Shortcuts List.
Conflicts	This message alerts whether a shortcut for the selected function is in conflict with another shortcut of another function.



Apply	Designates the shortcut to the selected Command, the user is able to define more shortcuts without leaving the dialog.
-------	--

ADVISED WAY OF WORKING

- Select a command from the Command list. E.g. Panning. Panning already has a shortcut: SHIFT+Right Button and the middle mouse button. We'll create a new one.



- Activate the New Shortcut textbox by clicking in it and push your desired shortcut.
- Click on the **Add** button, the shortcut is added to the Shortcut List.
- Click **OK**. You can now use your newly created shortcut.

"FLIP MOUSE WHEEL INPUT FOR ZOOMING" CHECKBOX

Allows the user to change the direction of the zooming behavior by the use of the mouse wheel.

"SHOW ROTATION CIRCLE" CHECKBOX

Allows the user to decide whether or not to show the rotation circle on screen during rotation of the view. The size of the rotation circle can be adjusted to fit the user needs.



Selection & Snapping

Some functions in Magics RP (for example measuring and aligning) require that you snap to a feature of the part: a point, a line, a plane, a circle, a cylinder, and sphere.

Appearance

Size

Points

Wireframe Surface

Sections Grid

Bad Edges Gravity Center

Triangle Vertices Slices Preview

Manual Points Beams

Lines, Circles

Wireframe Beams

Sections

Bad Edges

Snap to

Points

Lines, Circles

Triangles, Cylinders, Spheres

Appearance	<p>This sets the changes the visualization preferences fo snapped entities when the mouse is hovered over the appropriate entity.</p> <ul style="list-style-type: none"> – Size : Defines the radius of the point picker preview circle in pixels. – Color: Defines the general highlight color of all the snapped entities such as points, lines and circles.
Points	<p>This sets the list of entities that can are allowed to be used as an anchor for a point picker .</p> <ul style="list-style-type: none"> – Wireframe: If On, point pickercan snap to visible wireframe lines on parts. – Sections: If On, points picker can snap to visible section lines on parts. – Bad Edges: If On, point picker can snap to visible bad edges on parts. – Triangle Vertices: If On, point picker can snap to triangle vertices of all visible triangles on parts. – Manual Points: If On, point picker can snap to manually created points from the points toolpage. – Surface: If On, point picker can snap to visible faces that are on parts. – Grid: If On, point picker can snap to the grid if the grid visibility is toggled on. See ,1 Toggle Grid. – Gravity Center: If On, point picker can snap to the gravity center

	<p>indicator if the gravity center visibility is toggled on. See, 6 Gravity Center</p> <ul style="list-style-type: none"> – Slices Preview: If On, point picker can snap to the slice lines on slice parts. – Beams: if On, point picker can snap to the beam lattices
Lines, Circles	<p>This sets the list of entities that can are allowed to be used as an anchor for a point picker .</p> <ul style="list-style-type: none"> – Wireframe: If On, line and circle picker can snap to appropriate visible wireframe curves on parts. – Section: If On, line and circle picker can snap to appropriate visible section curves on parts. – Bad Edge, If On, line and circle picker can snap to visible bad edges on parts. – Beams: if On, line picker can snap to beam lattices.
Snap to	<p>This sets if the mouse pickers should actively snap to appropriate entities while on hover with highlights or to perform the snapping only on click.</p> <p>If you are experiencing slow downs while accessing some mouse modes, you are adviced to set the Snap to options to off so that snapping is only done on click and not on hover.</p> <ul style="list-style-type: none"> – If On, the pickers willactively seek for anchors on hover and show a point picker preview for appropriate snapping entity – If Off, the pickers will only seek for anchors on click and no preview will be shown on hover.

2. General

Unit Size

Units

mm Inch

Use machine units for the platform scenes

Automatic Unit Conversion

inch -> mm if maximum size < mm

mm -> inch if maximum size > inch

1. Units

You can choose between mm and inches. You will have to select the units before you load the STL file. If the STL file was originally in millimeters, you have to choose mm. If the file is in inches, you will have to choose Inch. If several parts are loaded, some in millimeters and some in inches, the Unit Conversion has to be used otherwise parts are out of proportion. The program always remembers the last used unit and takes this as default the next time you start up the program.

2. Automatic Unit Conversion

The automatic unit conversion is used to avoid mistakes due to the units you work in. It may be that you are working in millimeters, and that you load a part whose dimensions are expressed in inches. A part of 2inch*2inch*2inch, will then become a part of 2mm*2mm*2mm. The size of the part is not correct anymore.

Because 1inch is 25.4mm, the dimensions of the part expressed in millimeters are bigger than those when the part is expressed in inches. A part of 2inch*2inch*2inch, is as big as a part of 50.8mm*50.8mm* 50.8mm.

When you are working in millimeters and you load a part and the dimensions are very small (you can define 'very small' in the options –see the figure above), it may be that the part you loaded was originally expressed in inches. Magics will then multiply the dimensions with 25.4 (inch to mm conversion), so the part will now be expressed in millimeters. When you are working in inches and you load a part and the dimensions are very big (you can define 'very big' in the settings – see figure above), it may be that the part you loaded was originally expressed in millimeters. Magics will then divide the dimensions with 25.4, so the part will now be expressed in inches.

Marking

Plane Marking	
Surface Tolerance	0.500mm
Angle Deviation	10.000°
Brush Marking	
Size of Brush	10
Vertical Marking	
Angle Deviation	10.000°
Horizontal Marking	
Angle Deviation	10.000°

1. Plane Marking

These parameters describe how accurate the program should search for triangles in one plane.

– See **Marking tools, page 568**

2. Brush Marking

Size of Brush	Brush Marking allows marking triangles on an active part by a brush. The user can define the size of the brush.
---------------	---

3. Vertical Marking

Angle deviation	Vertical Marking allows marking triangles on an active part vertically. The user can define the deviation.
-----------------	--

4. Horizontal Marking

Angle deviation	Horizontal Marking allows marking triangles on an active part horizontally. The user can define the deviation
-----------------	---

Measurements Quality

Tolerance type

Basic

Absolute Tolerance mm

for values \leq mm

Relative Tolerance %

Advanced

Profile

Measurements quality		
Basic	A simple quality measurement is performed based on the parameters: relative tolerance and absolute tolerance.	
	Absolute tolerance	Specify the tolerance for the absolute deviation from the real measurement compared to the STL measurement.
	For values	Only check for values that are smaller or equal to the entered value.
	Relative tolerance	Specify the tolerance for the relative deviation from the real measurement compared to the STL measurement
Advanced	An advanced quality measurement is performed based on a defined profile.	
	Profile	Specify which profile has to be used to check the quality of the measurement(s). The profile is an *.xml document saved in the hidden 'Advanced Tolerance' folder (make visible in File Explorer > View > Hidden items). (Route if Magics is installed in the default location: <i>C:\ProgramData\Materialise\Magics\Settings\Advanced_Tolerance</i>)

Adjust ranges for advanced parameters in notepad.

Measurement types:

- **A** = absolute
- **R** = relative
- Ranges are the values in which the tolerances need to change. For example: A = 3000 means 0.3 mm R = 0.3 means 3%.



Undo and Recovery

Enable Undo

Maximal Undo Filesize MB

Enable Auto Recovery

Enable Undo	By default this option is On. A temporary file with the history of the operations performed in an active application session is created and maintained throughout the lifecycle of the session. This enables undo and redo operations. When you close Magics, the undo file of that session is erased.
Maximal Undo File size	You can enter a maximum size of the file where the operations you do are saved. When the file size gets too big, the descriptions of the first actions you did in Magics will be removed. The default value is 1000 MB.
Enable Auto Recovery	After a crash, Magics will ask you if you would like to go back to the previous status, if Auto Recovery is enabled. You can only recover when you enabled undo. During a crash the undo file will not be erased in this case. An Auto Recovery will search for the undo file of your last session when you start up Magics again. When the option is OFF, you will not be able to recover the partial undo file to be reloaded back into the session.



The Undo and Auto Recovery files can be found in the sub-directory 'Magics RP Undo' of the regular Windows temporary directory.



When you enabled the Auto Recovery option it is possible that during a crash this recovery file gets corrupted. When Magics wants to start it will prompt if you want to recover. If the recovery file is corrupt, Magics might not be able to recover or function correctly if this happens. Do not choose to recover the next time or go to the standard Windows Temporary '\Magics RP Undo'-folder and delete the file.

History

Magics RP keeps a track of the history of every part once an operation is performed. Enabling this option can slow Magics down. By default this option is ON.

Enable History



User process Flow Improvement

Magics will collect user experience feedback to ensure future improvements and best software quality.

I like to participate in user experience feedback for future improvements


Yes No

Languages

You can change the language of you Magics installation. The language change is only applied after restarting Magics.

Change Preferred Language

English
Chinese
English
French
German
Italian
Japanese
Korean
Spanish



Naming

In this section the settings to related to part naming can be found and changed.

- Always use English names for newly created parts
- Always use a default English name for the Build Job

Custom names after operations

Duplicate
Mirror
Rescale
Shells to Parts
Copy/Separate Marked
Merge
Cut or Punch
Hollow
Z Compensation
FormFit

Naming template

Choose a predefined or create a custom one

Always use English names for newly created parts


If this option is set to true, then newly created part names will not be translated to the application language instead be preserved in the English language. The default value is false and intended to be changed by the user who need the part names to always be in latin characters.

Always use a default English name for the Build Job

If this option is set to true, when a platform is sent for processing via the "Build" button, then the default Build Job name in the Build dialog will not be translated to the application language and instead will be set to "Untitled" and shown in English language. The Build Job name can be changed by the user.

Custom names after operations

This setting group is for you to customize the name of result part names after specific operations. You are advised to follow the placeholders correctly to achieve the most desired output part naming.

Custom names after operations	For every function in this list you can specify the used naming convention
Naming template	<p>A suffix can be selected from the list or customized suffixes can be created, keeping some rules in mind.</p> <p>Symbols are standing for:</p> <ul style="list-style-type: none"> * : part name #: index number \$p: parameter value \$P: special parameter value (currently only available for Rescale operation) <p><i>Example</i></p> <p>Simple type the desired notation into the combo box. Magics will save this format into its preferences</p> <div data-bbox="491 1496 815 1603" style="border: 1px solid #ccc; padding: 5px; margin: 10px 0;"> <p>Naming template</p> <p>Choose a predefined or create a custom one</p> <p>mirrored_of_*</p> </div> <div data-bbox="496 1704 1386 1823" style="border: 1px solid #00aaff; border-radius: 10px; padding: 10px; margin-top: 10px;"> <p> Remark: Use only a '*' if no suffix is requested after the operation.</p> </div>

Warnings

Turn warning messages on/ off for particular functions/ actions to complete.

Show the 'Never ask again' message for

Change memory state of parts	<input checked="" type="checkbox"/>
Part interval is too small in 3D Nester	<input checked="" type="checkbox"/>
Overwrite .magics file in Export Platform	<input checked="" type="checkbox"/>
Overwrite STL files during Export Platform	<input checked="" type="checkbox"/>
Bad edges collision with no build zones	<input checked="" type="checkbox"/>
Delete bounding box measurements	<input checked="" type="checkbox"/>
Delete Build Risk Analysis profile	<input checked="" type="checkbox"/>
Delete profile quality parts	<input checked="" type="checkbox"/>
Delete SG profile	<input checked="" type="checkbox"/>
Refine and smooth texture shift	<input checked="" type="checkbox"/>
Delete selected template in Measurements Report	<input checked="" type="checkbox"/>
Clear template list in Measurements Report	<input checked="" type="checkbox"/>
Save to Streamics on exit Magics	<input checked="" type="checkbox"/>
Save to Disk on exit Magics	<input checked="" type="checkbox"/>
Volume Estimation parameters warning	<input checked="" type="checkbox"/>
Part Cost Estimation parameters warning	<input checked="" type="checkbox"/>
Build risk analysis	<input checked="" type="checkbox"/>
Overwrite machines in MyMachines folder	<input checked="" type="checkbox"/>
Loading support surface profile of different type	<input checked="" type="checkbox"/>
New Materialise e-Stage version detected dialog	<input checked="" type="checkbox"/>
Show gravity center warning	<input checked="" type="checkbox"/>
Compact Memory State: Show support preview warning	<input checked="" type="checkbox"/>
Generate Report Selected Parts Warning	<input checked="" type="checkbox"/>
Show warning before part unloading	<input checked="" type="checkbox"/>
Bad edges Label	<input checked="" type="checkbox"/>

Reporting

Show data of virtual copies separated.

- Enabled: Report tags show information of each virtual copy separated.
- Disabled: Report tags show information only about original parts which copies are present; amount of copies on active scene is shown near the part name but doesn't influence on the part parameters.

Show data of virtual copies separated

Enabled - report tags show information of each virtual copy separated.

Disabled - report tags show information only about original parts which copies are present; amount of copies on active scene is shown near the part name but doesn't influence on the part parameters.

Measurements font size in reports:

Small

Medium

Big



Project Template

You can set the path to the Magics project template file with default scenes and parts for the new project command.

- See **New Project, page 56** for more information for creating a new untitled project with default scenes.

Use default project template

Set the path to the Magics project file with default scenes and parts

Always use latest machine properties from machine library for platform scenes

Template will be used next time new project command is applied

Use default project template	If this is set to true, then the Magics project template that is set in the next input is used as the template for a new project always. Default is set to False. You will be shown a message dialog when calling the New Project command when this setting is set to false.
Path to Magics project template	Path to a Magics project file in the local device to be used as the default template project. The template path is set to a file in the ProgramData created during the installation of Magics. If the path is set to non-Magics file then the field will be reverted to the default path automatically.
Always use latest machine properties from machine library for platform scenes.	If this is set to true, then when starting a new project from the template, if the template has platform scenes in it then the latest machine properties from the machine library will be used for the new platform scenes.

3. Modules

Support Generation

Will allow you to set the support mode (Manual – Automatic) when switching to the support module (SG).

Switch part in SG mode

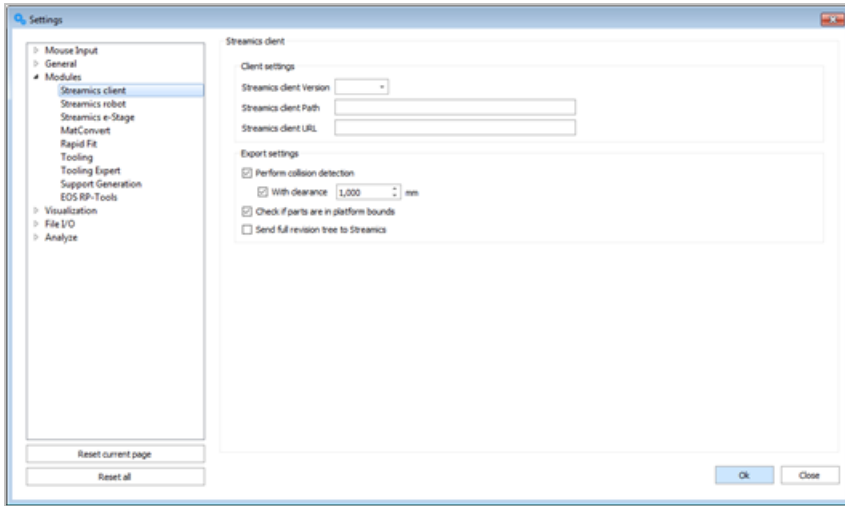
Generate support

Manual support

Always give the option in a dialog (where user can choose which mode to use)

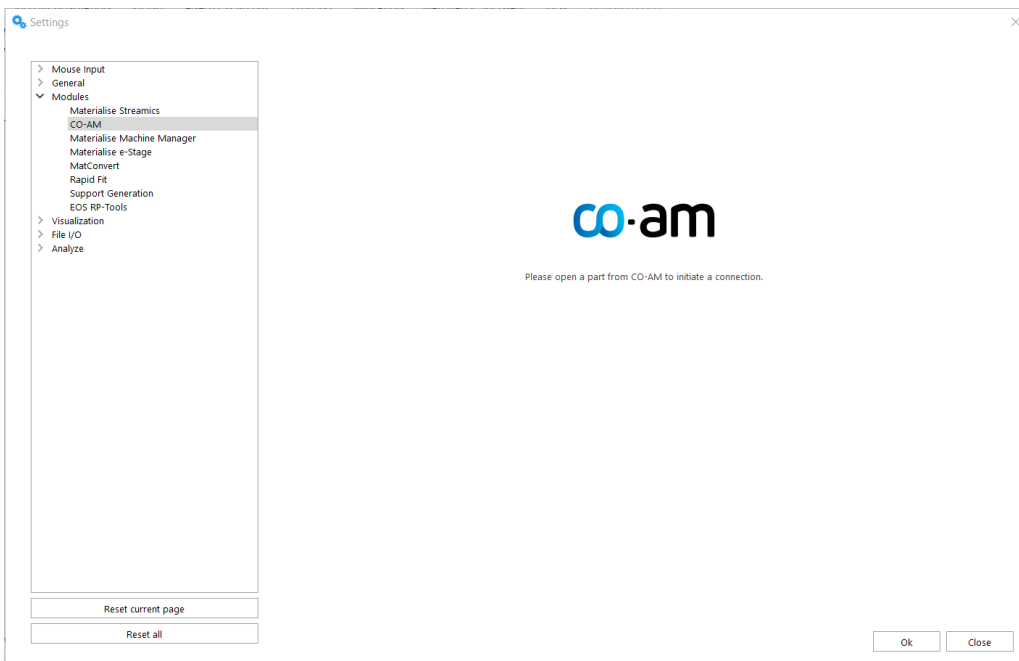
Streamics Client

Here you can change the settings of the Streamics Client. It also enables the user to perform a collision detection and out of bounds detection check before uploading platforms to Streamics. In addition there is an option to change the revision settings between full mode and limited mode.

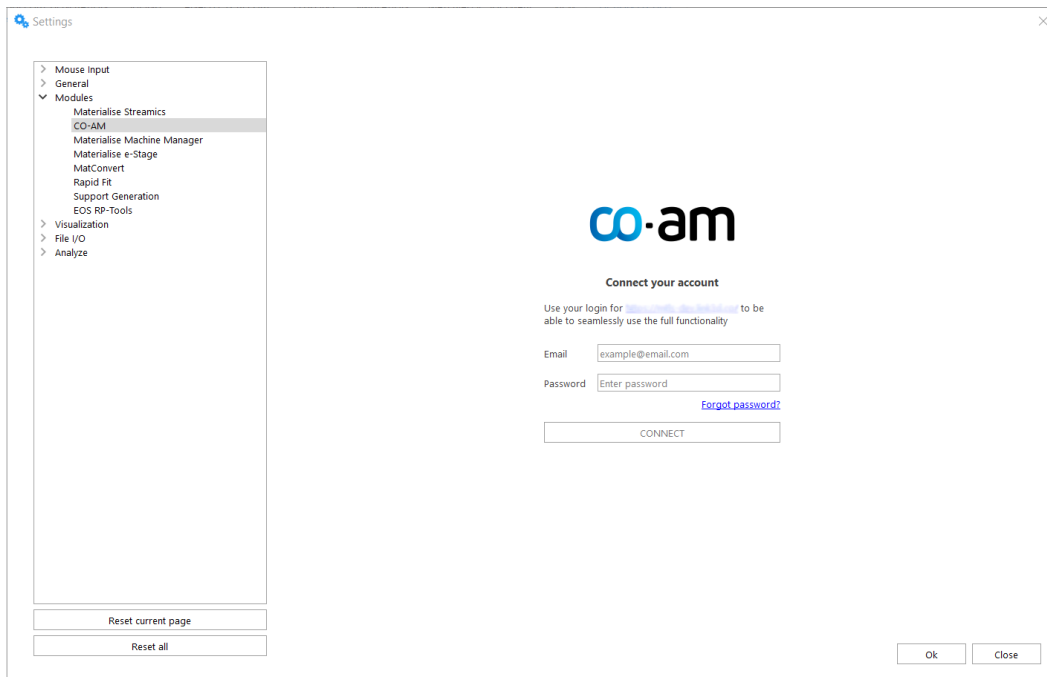


CO-AM

If user is not connected to any CO-AM accounts, then a message is shown to initiate a connection by opening a part or a platform from the CO-AM website.

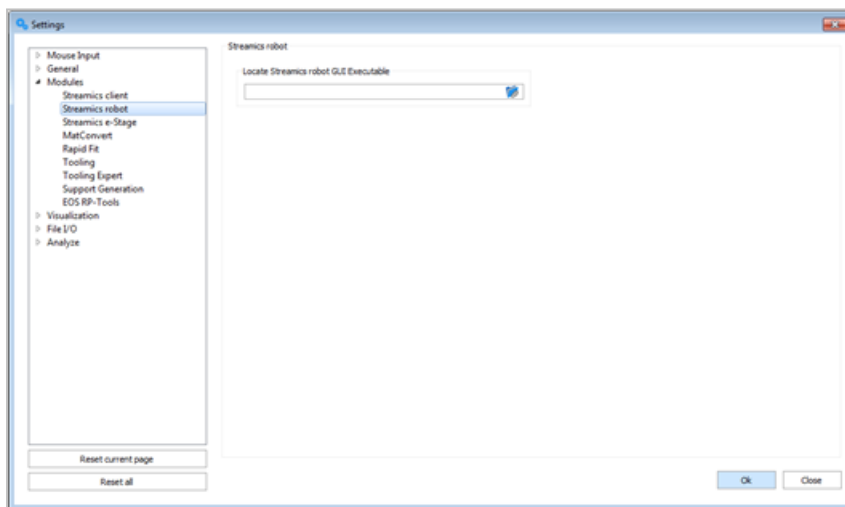


Once a connection is initiated from CO-AM, the user will see a login form for linking a CO-AM account.

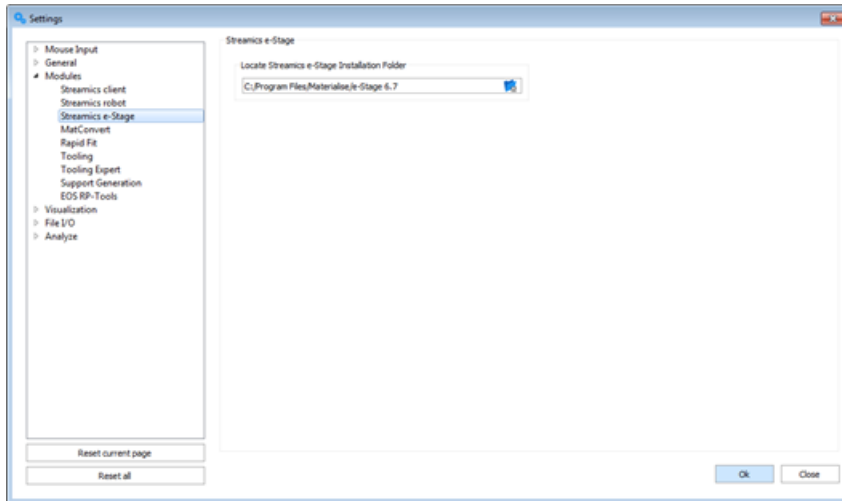


Materialise Robot

In this tab you can specify the installation folder of the Materialise Robot that you want to use for processing Robot content from Magics.



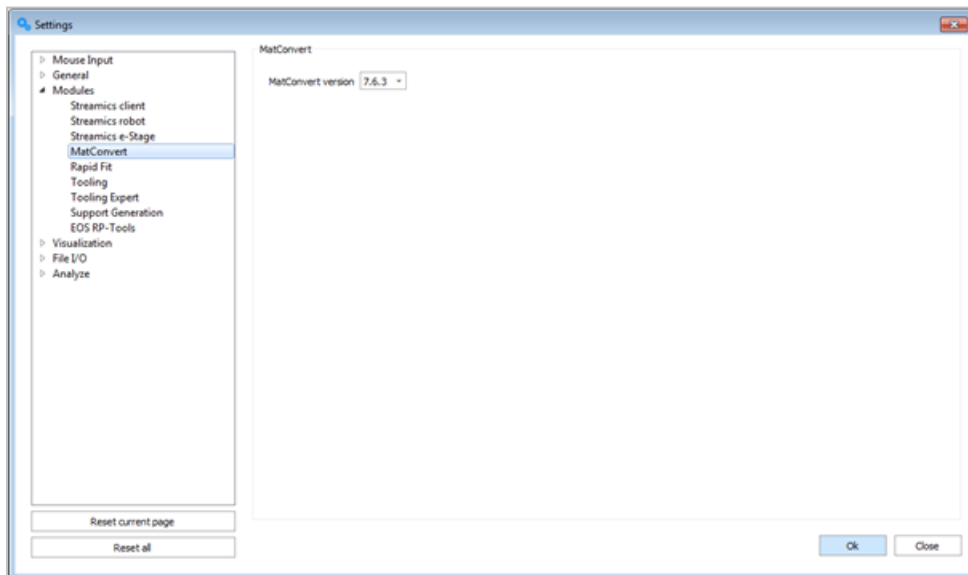
e-Stage



Locate e-Stage folder	Location of e-Stage. The e-Stage link is only active in Magics when the path to e-Stage is defined properly.
-----------------------	--

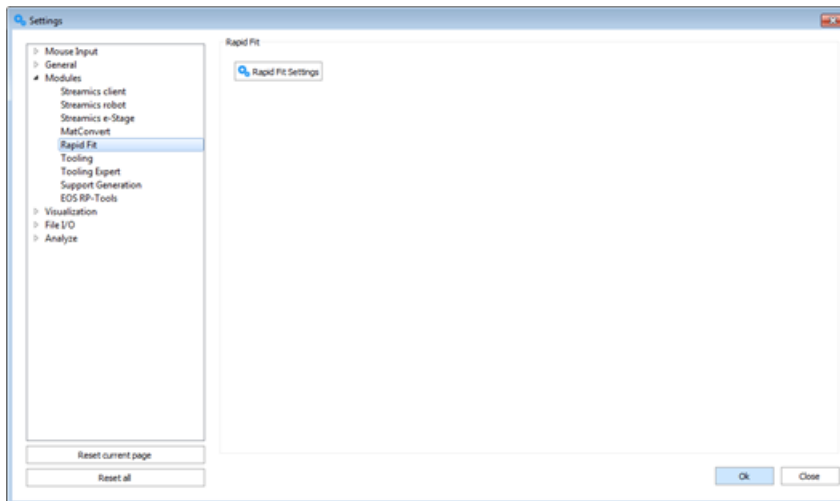
MatConvert

Magics uses MatConvert to bridge the gap between CAD files and 3D printers by importing nearly all standard CAD formats and packages. If you have multiple MatConvert versions installed on your pc, this setting will allow you to switch between different versions of MatConvert. This also allows you to upgrade your MatConvert to the latest release without installing a new Magics.

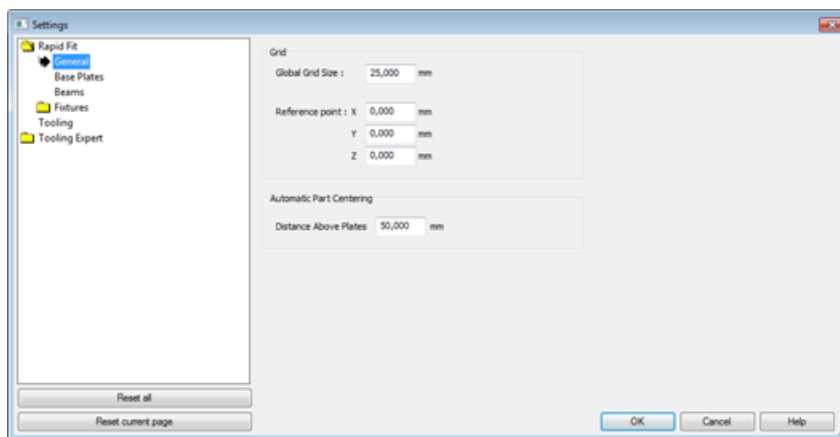


For MatConvert versions and supported file formats, please go to: <https://help.materialise.com/magics-release-documentation/matconvert>

Rapid Fit

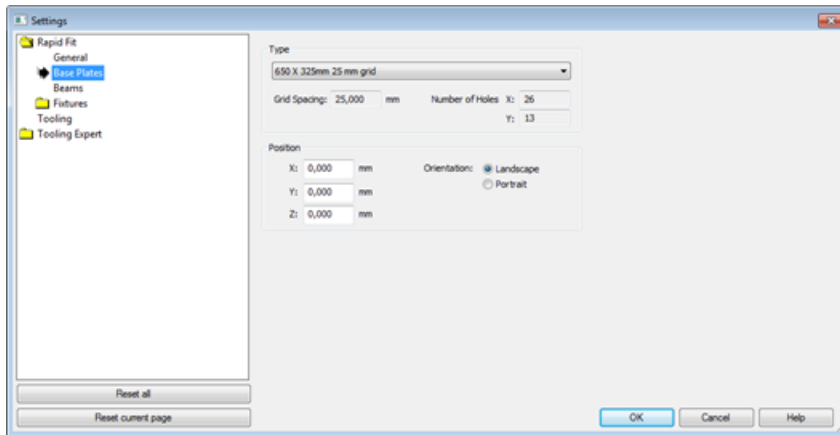


1. General



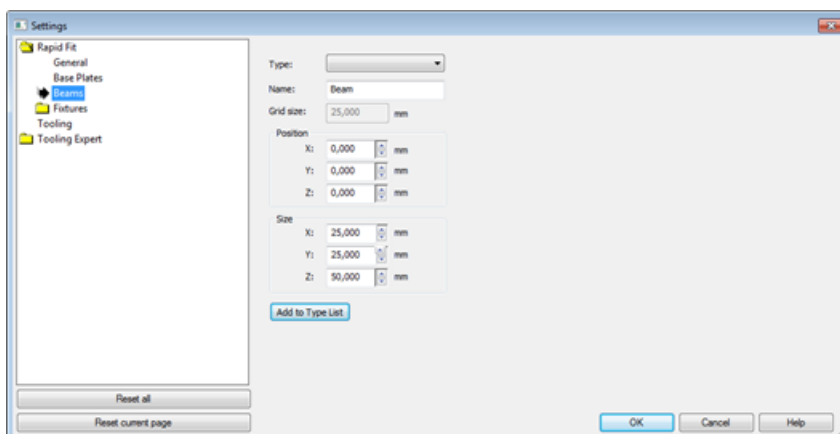
Grid	Global grid size	The size of the grid
	Reference point	The position of the reference point
Automatic Part centering	Distance Above Plates	The distance between a plate and part in z-direction.

2. Base Plates



Type	The size of the base plate	You choose a Plate Type out of the list, which is filtered in baseplates linked to the chosen grid size. The Base Plates are standardized. The type mentions the size of the plate in X and Y and also the Grid size.
	Grid spacing	The grid Parameter is a property of a Plate Type and can thus not be changed. It is the distance between the centers of the threaded holes. The distance between the side edge and the first row of holes is half the grid Parameter. Therefore the size of the plate is a multiple of this grid parameter.
	Number of holes	The Number of holes in the plate is the size (here 325) divided by the grid spacing (here 25).
Position	This is the position of the hole with lowest X and Y coordinate. The orientation can be compared with Landscape and Portrait of a page setup in other programs.	

3. Beams



Type	You can choose the type of beam you want
Name	The name of the beam can be changed



Grid size	The Grid Size of the Base Plate. The Grid Size indicates the step of the movement on the base plate.
Position	The (x, y, z) position where the beam will appear.
Size	The default size of the beam
Add to Type List	You can add the current defined beam type to the Type list at the top.

EOS - RP Tools

Deprecated from Magics version 27.0.

Visualization

Themes

Assign theme

Light theme

Dark theme

All the customized colors will be saved in the currently applied theme. To restore the default system colors for the currently applied theme, click the "Reset current theme colors" button.

Light Theme:

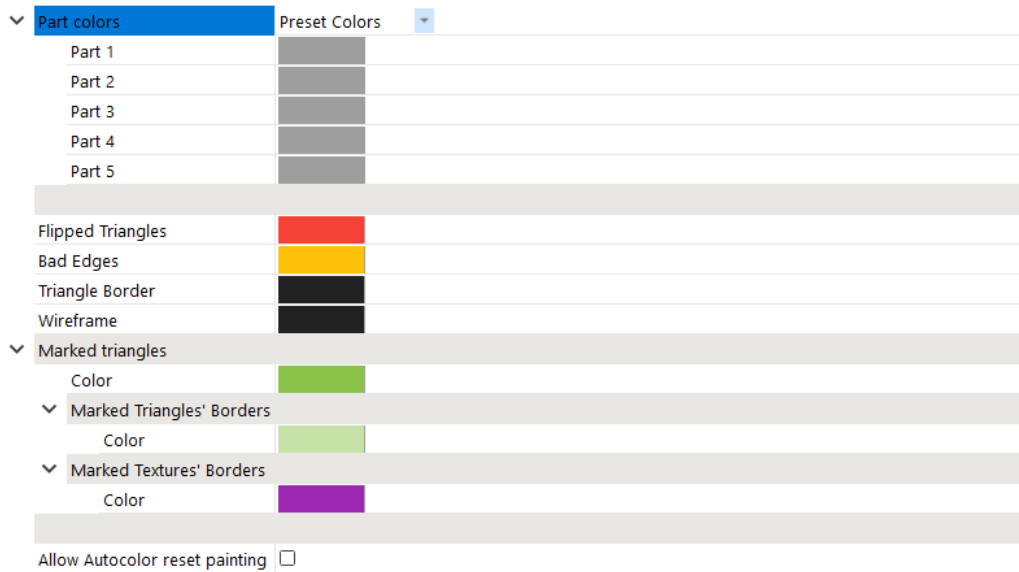
Embrace a clean and bright interface with our Light theme. Ideal for well-lit environments, this theme enhances readability and reduces eye strain.

Dark Theme:

For those who prefer a sleek and stylish look, our Dark theme is the perfect choice. This theme reduces glare, enhances contrast, and is particularly well-suited for low-light conditions.

Colors

1. STL Colors



When you select random colors, random colors are used for the different parts you import.

When you select Preset colors, the colors of this window are used.

Part 1 to Part 5	These five buttons represent the five colors as they will be assigned to the loaded parts. The first part you load will get the first color. The second part will get the second color and so on.
Flipped Triangles	The color of the flipped triangles.
Bad Edges	The color of the bad edges.
Triangle Border	The color of the triangle borders.
Wireframe	The color of the wireframe.
Marked Triangles	The color of the marked triangles.
Marked Triangles' Borders	The color of the marked triangles' borders.
Marked Borders' Thickness	The color of the marked borders' thickness.
Marked Textures' Borders	The color of the marked textures' borders.

Marked Textures' Thickness	The color of the marked textures' thickness.
Allow Autocolor reset painting	When checked, Autocolor is allowed to reset painting.

2. Support Colors

Current Support	
Normal Support	
Selected Support	
No support zone	
Support preview	
Triangle Border	
Edges	
Surface	
Support that bumps into surface (in support list toolpage)	
Downfacing Edges	
Other Parts	
Other Supports	
Shrinkline preview	

Supported Area preview

End color range

Begin color range

Reference

3. Slice Preview Colors

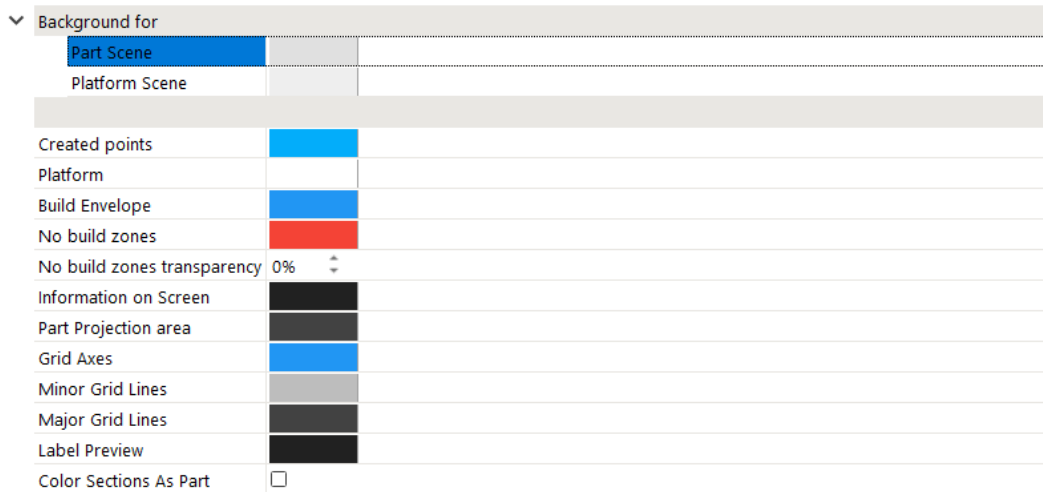
Borders	
Additional Borders	
Normal Hatching	
Support	
Upfacing	
Downfacing	
Open Contour	

4. Out of bounds Colors

Color parts on the side of the platform	<input checked="" type="checkbox"/>
Color parts above and below the platform	<input checked="" type="checkbox"/>
Color	
Highlight parts in Part List (internal only)	<input checked="" type="checkbox"/>

Color parts on the side of the platform	When activated the parts that exceed the bounds of the platform are colored.
Color parts above and below the platform	When activated, the parts that exceed the bounds of the platform above and below are colored.
Color	Specify the color of the parts that are out of bounds

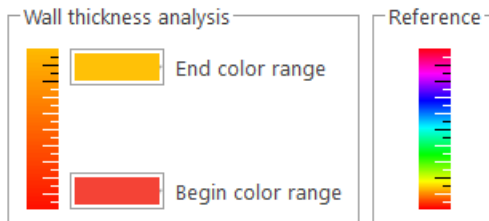
5. Other Colors



The user can define the colors of the background for the Part Scene, the Annotation Scenes and the Platform Scenes. Furthermore, he can choose the color of measurement lines, created points, the build envelope and the grid.

The user can choose to have sections colored as part.

6. Wall Thickness Analysis Colors



End color range When showing result as gradient coloring, this defines the maximum thickness color in the legend.

Begin color range When showing result as gradient coloring, this defines the minimum thickness color in the legend.

Reference The color gradient reference.

Renderer

Modern video cards have specialized hardware to make rendering faster. Magics uses Direct 3D rendering. This will make rendering a lot faster because the CPU is not specialist in rendering and a 3D card is.

The principle is that the computer sends the triangles to the memory of the video card and the card will do the rendering of the part, following the instructions of Magics.

This has directly 2 consequences:

- To have optimal results, the memory on the video card must be large enough (1 MB STL needs approx. 1,5MB ram on the 3D-card).
- Each time the part has been changed, the whole list of triangles must be send again to the video card. This can cause delays depending of the size of the part. When you have a huge amount of triangles (millions of triangles), delays can be avoided by switching back to software rendering. Each 3D-card has a limit.



Remark: We recommend that you try each mode and see what the results are. Not only the hardware is important, but also the drivers. If you're having problems, try again with other (more recent) drivers.

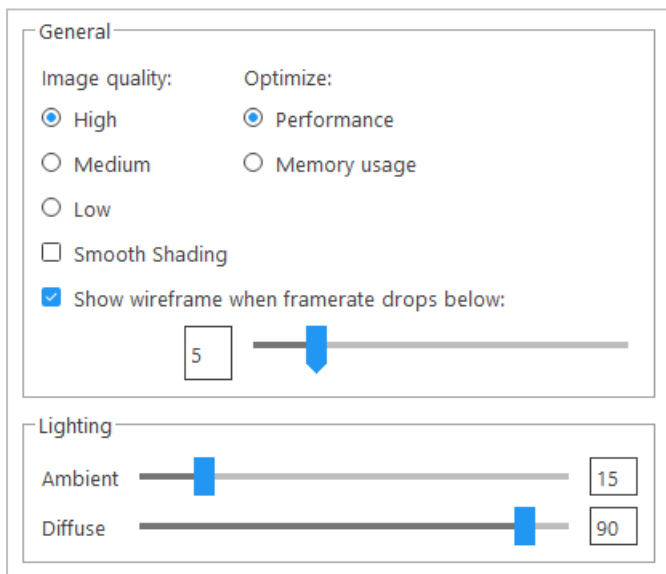
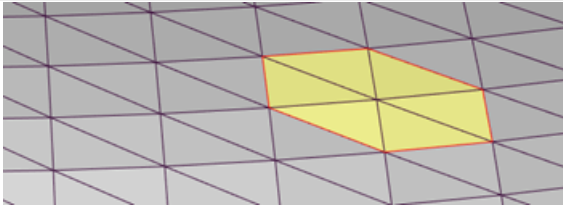
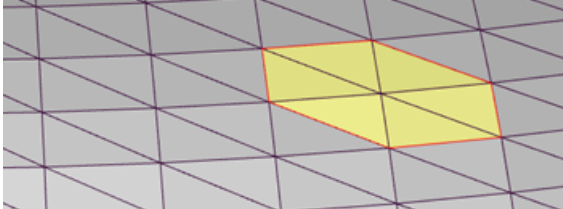
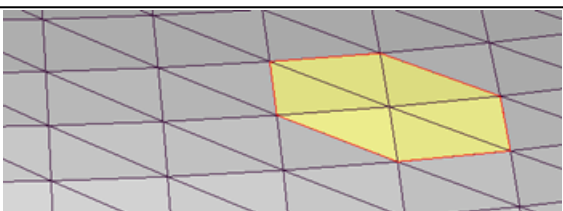
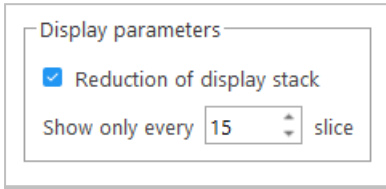


Image quality	High	
	Medium	
	Low	
Optimize	Performance	A copy of the triangles is saved on the memory of the graphics card, this should significantly speed up the rendering on most systems.
	Memory usage	The memory usage is limited when this option is flagged. The performance will be less though when working with large parts this option is best to choose.
Smooth shading		The part can be visualized using smooth shading. The variations in color will now be shown more gradually and no longer as separate triangles. Note that only the visualization of the part changes, the number of triangles and the accuracy of the STL are not changed.
Use Material Editor		<p>OFF: When you click on the colored round in the part list sheet , a color palette appears.</p> <p>ON: When you click on the colored round in the part list sheet, the material editor dialog appears. Besides the color, you can also select the material properties.</p> <p>The colors can be described in 2 ways: RGB (red, green, Blue) and HLS (Hue, Lightness, Saturation).</p>
Show wireframe when framerate drops below		This flag allows the application to only render wireframes or points of parts when rotating the view thus interaction speeds up in the 3D-views.
Lighting	Ambient	This is the reaction to the ambient light that exists everywhere in the scene, this light is scattered equally in all directions on the surface.
	Diffuse	This is the reaction to directional light; it reflects more in one direction.



Slice Stacks

To speed up the process of visualizing slice stacks after import, the amount of slice stacks displayed at once can be specified in the settings.



Reduction of display stack	Indicate how many slices of the full stack should be displayed.
----------------------------	---

Supports

- View surface border
 - Highlight surface border
 - Hidden line
- View support edge
- Show perforations
- Cut support with section view

View surface border	The border of the surface where the selected support is trimmed on is visualized in yellow.
Highlight surface border	The border of the surface where the selected support is trimmed on is visualized with a thicker line.
Hidden Line	The line showing the surface border is only visible when it is in the front.
View support edge	Edges of support are shown in light blue for better visualization.
Show perforations	Perforations are shown
Cut support with section view	When enabled, the section also influences the supports.

Graphs



Render by default as	Beam (thickness)	Beam thickness will be rendered by default
----------------------	------------------	--



	Lines (no thickness)	Beam thickness won't be rendered with this setting. The beams will be visualized as lines
Color of graphs with no thickness	Defines the color of beams without thickness	

Annotations and Measurements

Measurement

Arrows: Closed Filled

Width: 10

Height: 10

Grip Size: 5

Extension Lines

Draw Features

Text

Aligned

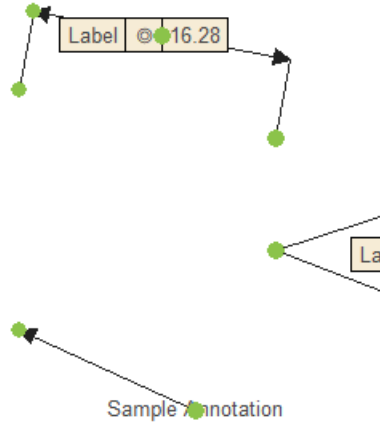
Show X, Y and Z distances

Show Label Only

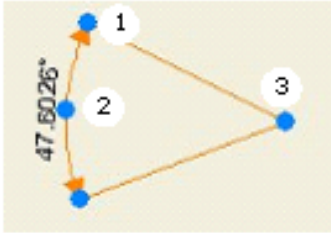
Show Tolerances

Show Label and Icons

Number of Decimals: 2



1. Measurement

Arrows	Arrows can be placed at the end of the line that indicates the distance between two features. An open, a closed, or a closed filled arrow can be chosen. The color of the arrows can be adapted.
Width	The width of the arrows can be adapted.
Height	The height of the arrows can be adapted.
Extension Lines	You can choose to draw extension lines or not. The color can be adapted.
Grip Size	<p>When you select a measurement, some grips (rounds) will appear on the measurement's lines. They allow changing the position of the measurement lines. The grip (1 in the figure) on the intersection of the line that indicates the length and an extension line allows you to turn the former around the axis between the features. The grip (2 in the figure) in the middle of the line that indicates the length allows you to extend or shorten the extension lines. The grips (3 in the figure) that mark the feature (in the case shown below it is a point), can be dragged to the same feature (in this case a point), positioned somewhere else on the part. The measurement value will be adapted. You can define the size and color of the grip.</p> 
Draw Features	A point is indicated with a cross, a line and a circle will be colored in the selected color, the axes of a cylinder will be drawn and the contours of its two discs will be colored in the indicated color, a sphere will be marked with three circumpolar circles. You can define the color of the features.

2. Text



Aligned	Checked	
	Not Checked	
<input type="button" value="Font"/>		You can adapt the font of the text.
Show X, Y and Z distances	Checked, the measurement value is not aligned.	
Show Label Only	Only show labels	
Show Tolerances	Show Tolerances	
Show Label and Icons	Show both label and icons	
Number of decimal	Specify the amount of decimal to take	

places	into account.	
--------	---------------	--

Grid

Hide from bottom
 Axis visible
 Auto extend
 Total grid size mm
 Z height mm
 Minor grid line mm
 Major grid line x 1.000 mm

Hide from bottom	The grid will be hidden when rotating the view below the XY plane.
Axis Visible	The axes form the intersection of the grid with the XZ and YZ planes through the origin. By default they are part of the major and minor grid. To see these axes, you can check this box. The axes will turn light blue.
Auto Extend	When this box is checked, the grid will be adapted depending on the Z projection of the loaded parts.
Total Grid Size	Here you can set the total grid size.
Z height	You can put the grid at a certain height under or above the XY plane.
Minor grid line	You can define the grid size here. The minor grid is colored light gray.
Major grid line	If you would like to have a second less detailed grid, you can fill in a number n here. The lines of the major grid will lie n times the lines of the minor grid. The major grid is colored dark grey.

Rulers

The rulers can be placed at the bottom of the working area and/or in the left side of the working area.

Show ruler at bottom of the viewing window
 Show ruler at left of the viewing window



Wireframe

Angle °

Angle	A line of the wireframe is drawn when the angle between 2 triangles is exceeding a certain value. The default value is 30°. Here, you can adjust this value.
-------	--

View cube

 Show view cube Lock view cube rotation

On-screen position

 Top-left Top-right Bottom-left Bottom-right

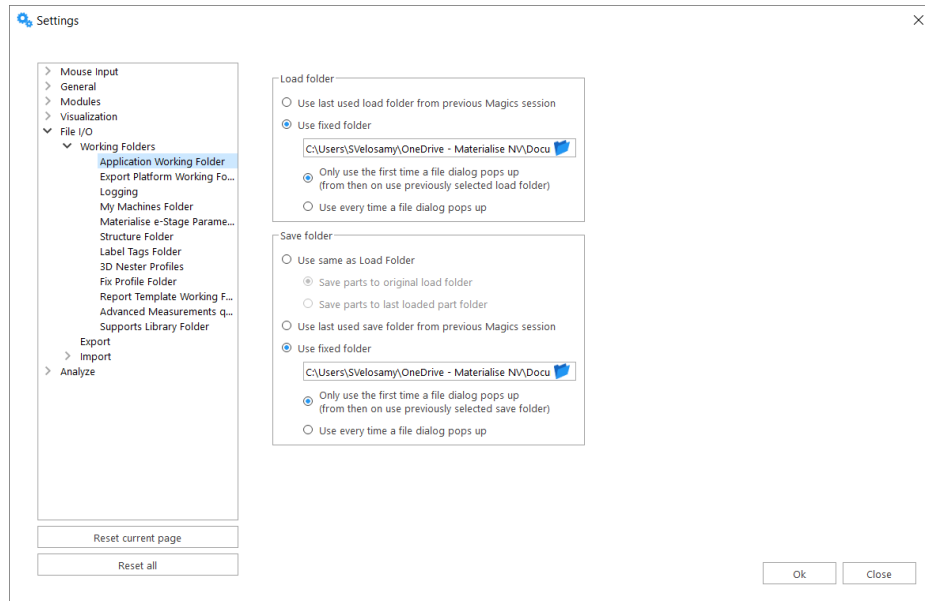
Show view cube	Toggle the visibility of the view cube on the scene.
Lock view cube rotation	When selected, the view cube will not be linked anymore to the rotation of the viewport, and it will always remain in the default ISO view front-left-top.
On screen position	Select the preferred corner of the scene to position the view cube. By default, it's positioned in the bottom-right corner.


4. File I/O

Working Folders

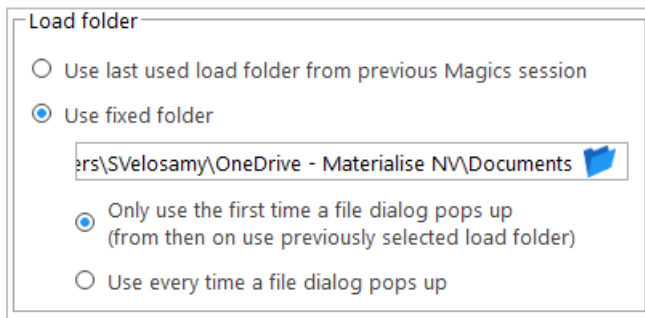
1. Application Working Folders

Application working folders settings allows the user the specify the default folders that will be accessed or shown when a Windows Explorer dialog is launched from Magics for file input output.



 You can browse your device and network directories and set a folder as default folder.

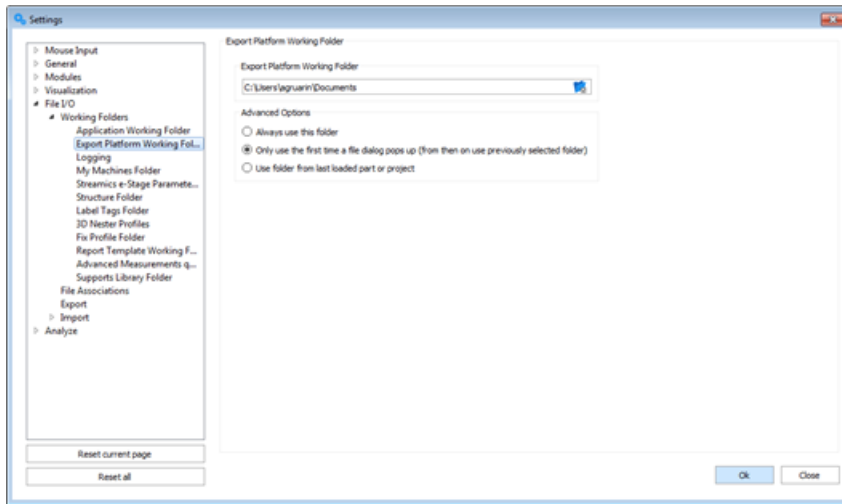
Load folder is the folder used when loading files into Magics by default.




Option 1	Always uses the last used folder for loading files from the previous Magics session if exist. Else uses the User documents folder for the first time.	
Option 2	User can specify a folder to be the main fixed folder when loading files. The default folder path will be the User documents folder.	
	Option 2a	After the first execution of file loading, the subsequent operation will show the folder location of the previously loaded file till the end of the current Magics session.
	Option 2b	Windows Explorer dialog will always point to the default fixed folder set.

2. Export Platform Working folder

Export platform working folder allows users to set the default folders to be used when exporting platforms and its content a build job folder.



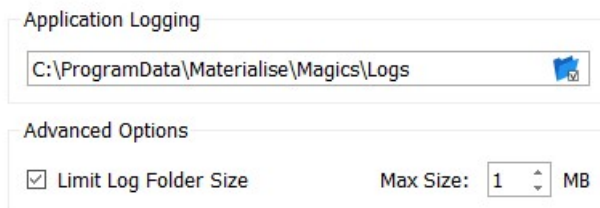
 You can browse your device and network directories and set a folder as default folder.

— Advanced Options


Option 1	Magics always proposes this folder when exporting a platform.
Option 2	Magics only proposes the chosen folder the first time a file dialog pops up. On subsequent operations a file dialog pops up, the previously selected folder is used.
Option 3	The folder where the last loaded file was from is used.

3. Logging

Application logging will allow you to specify where you will store the log file and if log file size will be limited and with how much.

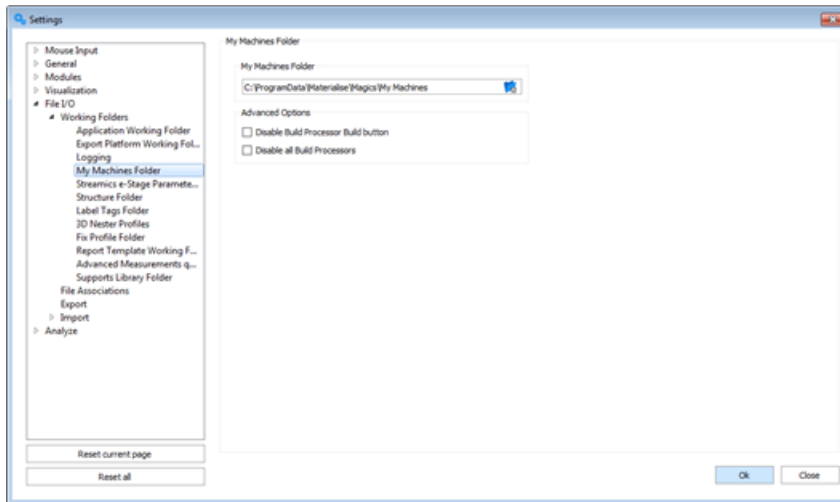



Magics keeps a log file. In this file all performed operations and actions are written.

	Here you can define where you would like to save the log file.
Limit	You can define the maximum size for the log folder or not.

Log Folder Size	
Max Size	Set the maximum size for the log folder. If the size is reached, the first actions of the Magics session will be erased, so that the latest actions can be written in the file.

4. My Machines Folder

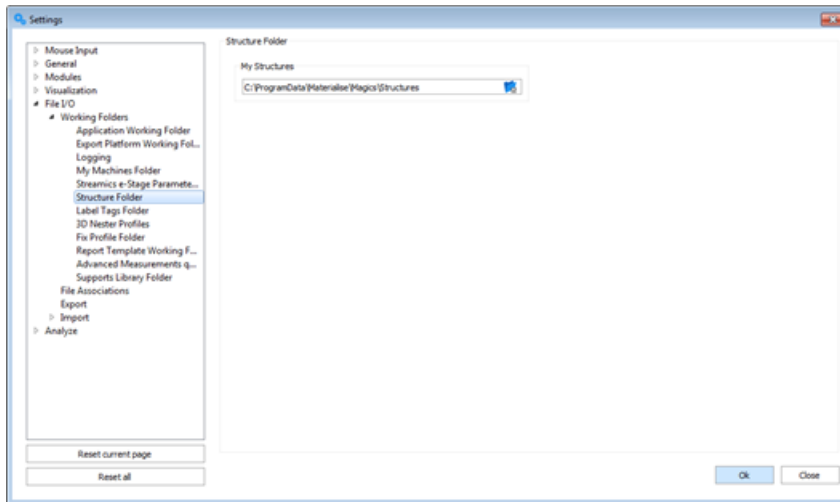



 Here you can define my machines folder. The My Machines Folder contains the machine files that are used by the user. (Machine Setup)

— Advanced Options

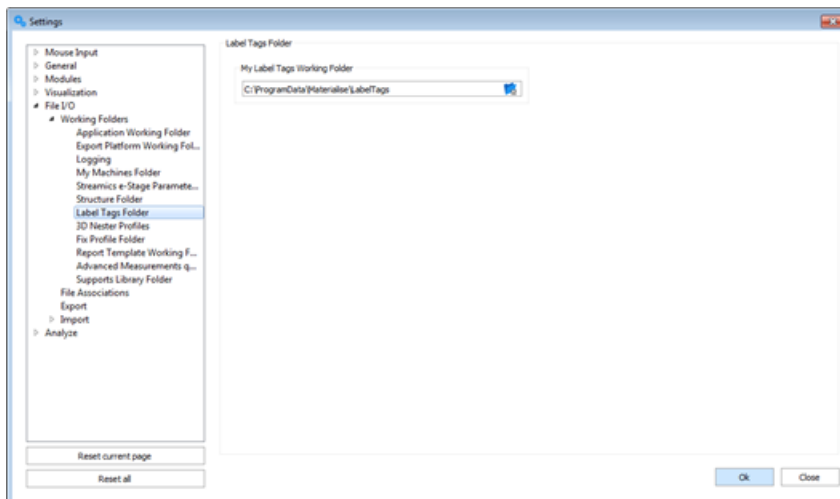
Disable Build Processor Build Button	Disable the Build Processor Build button.
Disable all Build Processors	No longer use any of the defined build processors.


5. Structure folder



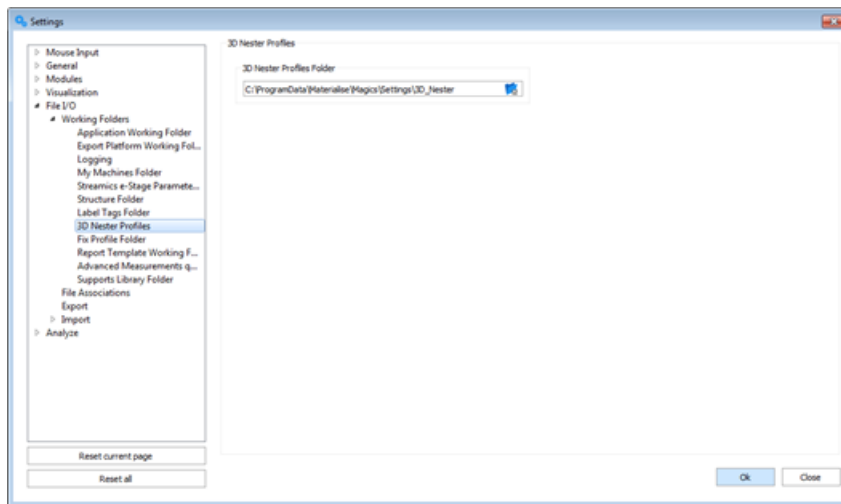
 Here you can define the structure folder. This folder contains all structures which can be used by the user.


6. Label Tags folder



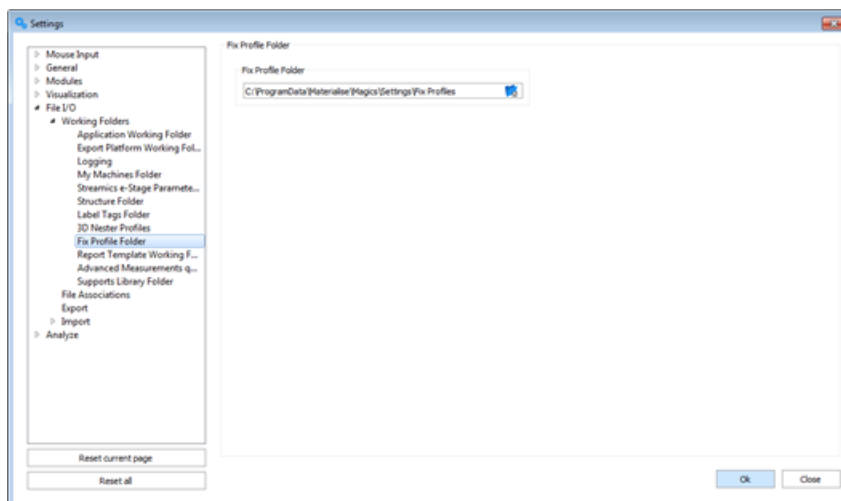
 Here you can define the label tag folder. This folder contains all label tags which can be used by the user.


7. 3D Nester Profile




 Here you can define the 3D Nester profile folder. This folder contains all 3D nester profiles which can be used by the user.

8. Fix Profile Folder

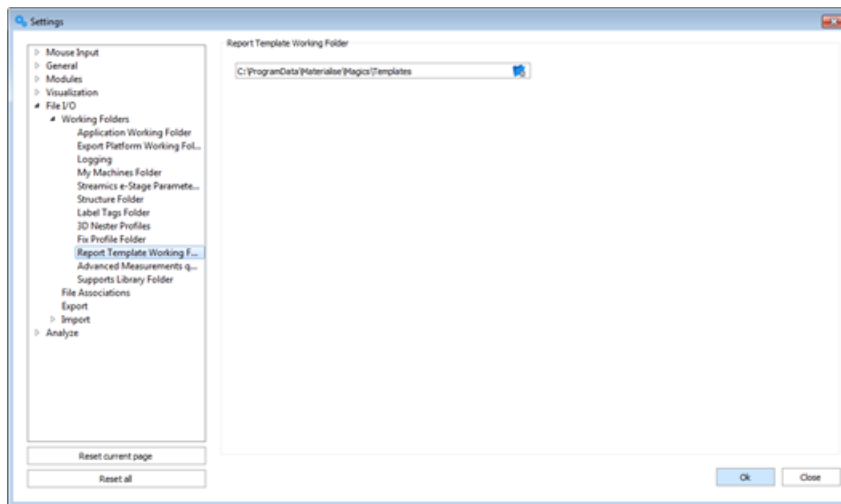



 Here you can define the Fix Profile Folder. The Fix Profile Folder contains the fix profile that is used by the user.

9. Honeycomb Profile folder

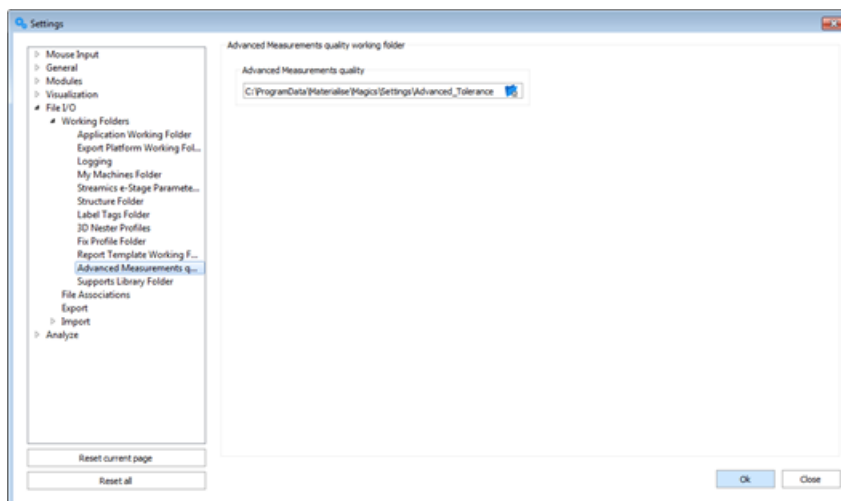
 Here you can define the Honeycomb Profile folder path. This folder shall contain all the honeycomb profiles that is used by the user.


10. Report Template Working Folder



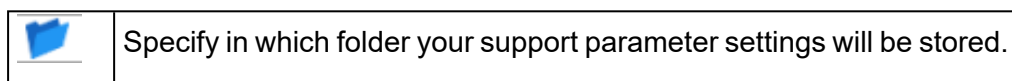
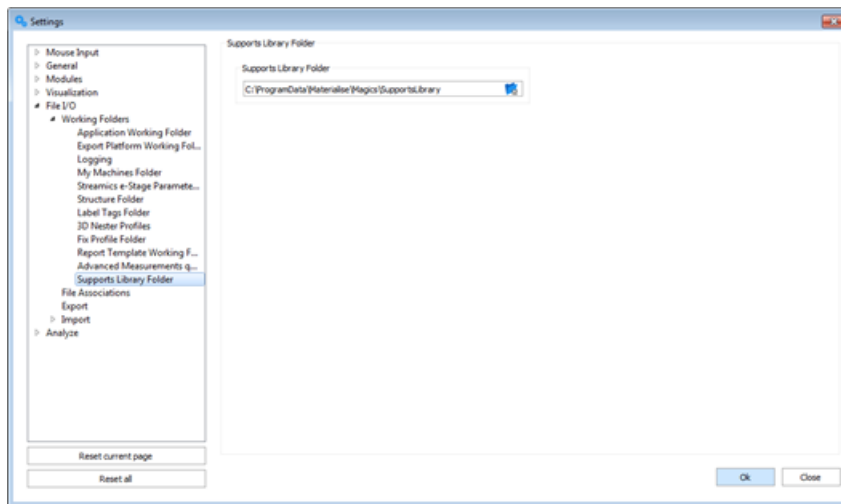
 Here you can define the Report Template Folder. The Report Template Folder contains the templates that are used by the user.

11. Advanced measurements quality



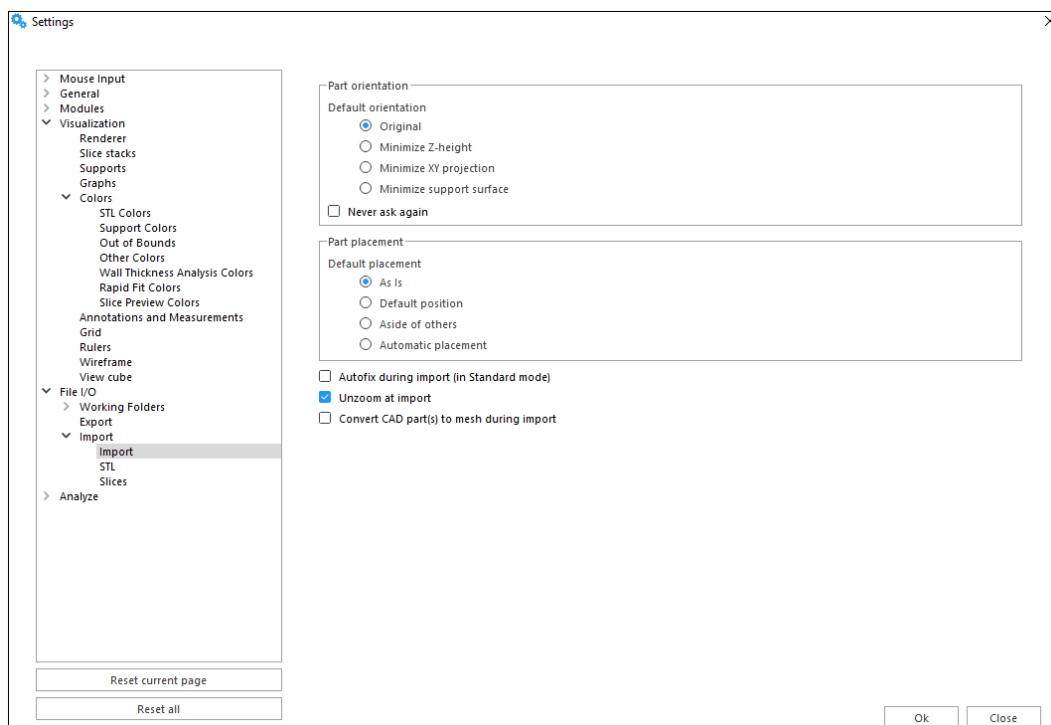
 Here you can define the advanced measurement folder. The advanced quality folder contains the quality documents that are used by the user.

12. Support Library Folder



Import

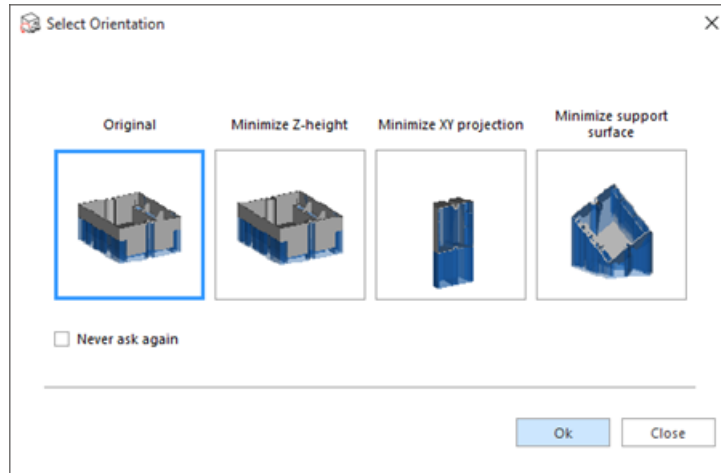
1. Import



Within this window you can control preferred option for opening CAD files either as BREP or STL by switching the checkbox 'Convert CAD part(s) to mesh during import'. If you are not keen on working with BREP models in Magics you can check this option on. In this case CAD files will be converted to mesh during import.

Part orientation

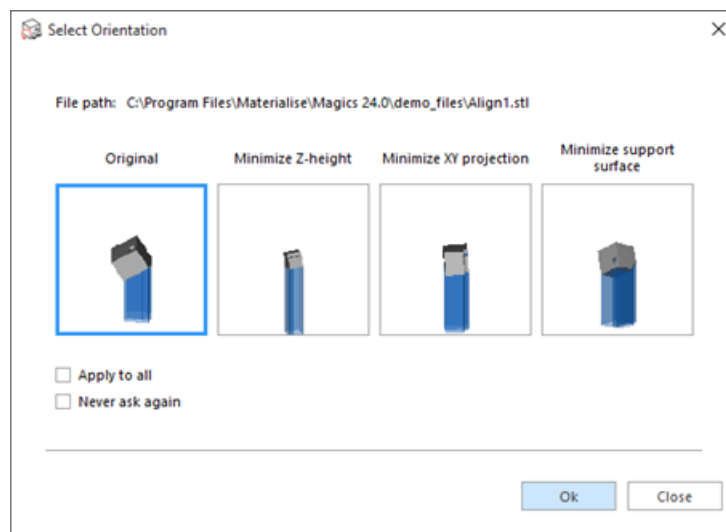
Part orientation dialog is shown when importing parts directly to a platform scene. Through this dialog you can quickly specify the desired orientation for your parts based on four main options when loading your parts.



Select the desired orientation by clicking on the preview, then clicking “Ok”. If “Never ask again” is enabled, the selected option will be remembered as the default one in the Settings page.

Original	The original orientation of the part will be used.
Minimize Z-height	The part will be rotated so the Z-height is minimized.
Minimize XY projection	The part will be rotated so the XY projection is minimized.
Minimize support surface	The part will be rotated so the support surface is minimized. This option is only available with a Support Generation license.
Never ask again	The selected option will be used without prompting the user to choose an orientation.

When importing multiple parts at once, the dialog will appear for each part with the origin file of the part shown above:




The “File path” can help you identify which part exactly is being imported.

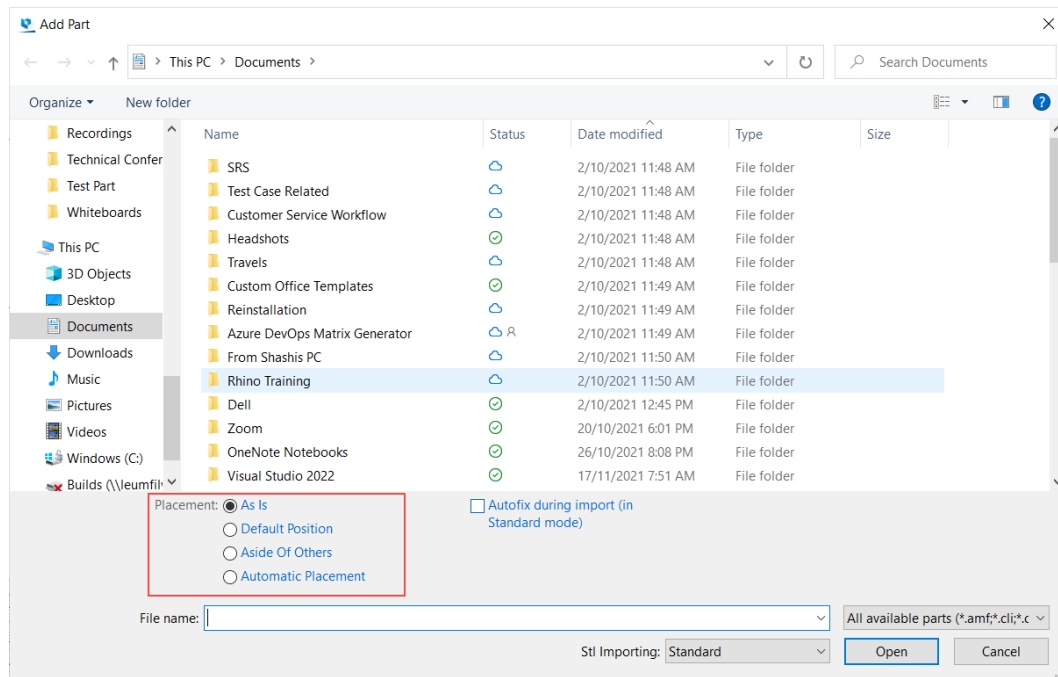
By checking “Apply to all”, the selected orientation will be applied to all parts that are being imported at once.

Part placement

Part placement options allows users to set a default way to arrange parts when importing parts to any of the active scenes through the "Import Part" command.

As Is	The original position of the part will be preserved as is.
Default Position	The part will be placed on the default position defined based on the target scene. For platform scenes this is the 'Default position' set in Machine properties, while for Part scenes this is by default 0,0,0 relative to active coordinate system.
Aside of others	Each individual part will be placed next to each to previously loaded parts as they are being loaded.
Automatic Placement	<p>The part will be placed according to the Automatic Placement method defined in the machine file. Parts that are already on the platform will not move, but will be taken into account while placing the newly imported parts.</p> <div style="border: 1px solid blue; padding: 5px; margin-top: 10px;">  Note: This option is not available in Part Scenes, if set this setting will be ignored and Parts will be loaded As is. </div>
Unzoom at import	After importing the file, the view will change to focus on the part.

When using the "Import Part command, the options are available on the bottom panel of the Windows Explorer dialog:



Autofix during import (in Standard mode)

If this is set to true, then all mesh parts that are loaded in the standard mode will be fixed automatically.

Unzoom at import

If this is set to true, the active scene the part was loaded in will be automatically unzoomed such that all visible parts are within the screen view.

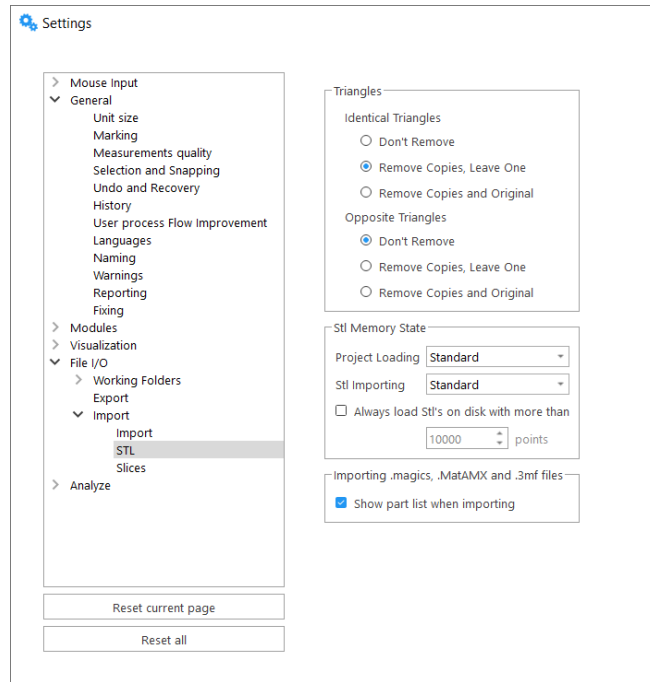
Convert CAD part(s) to mesh during import

If this is set to true, even if a CAD file can be loaded preserving the BREP parts, it will be loaded as a mesh part automatically.



Note: This setting is not used when importing parts directly to the BREP part scene, where parts from CAD file(s) are always loaded as BREP parts based on available import license.

2. STL



– Identical Triangles

Identical Triangles	Identical triangles have their normals in the same direction. You can choose to leave these triangles, leave one of the two triangles or remove them both.
Opposite Triangles	Opposite Triangles have their normals in opposite directions. You can choose to leave these triangles, leave one of the two triangles or remove them both.

– STL Memory State

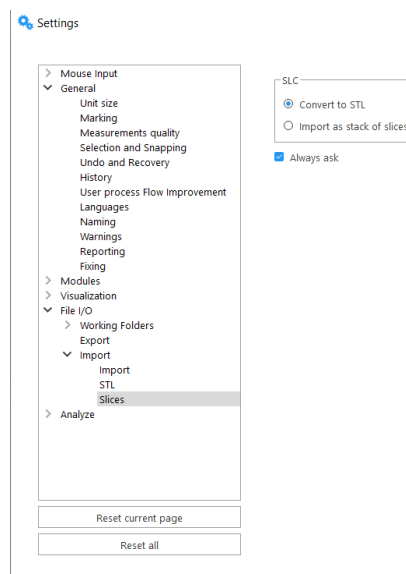
.magics Project Loading	You can define in which memory state you want to load a project. This memory state becomes the default when importing a project.	
	Standard	This is the standard memory state of a STL file. Magics knows the placement of the triangles and their mutual dependencies. The user is able to perform actions on STL level (E.g. deleting triangles).
	Compact	The STL resides in the memory as read-only, therefore it uses far less memory than the Standard memory state. Magics does not know the placement of the triangles nor their mutual dependencies. The user is not able to perform actions on STL level.
	On Disk	The STL is saved on disk and unloaded from the memory. The STL will stay in the project but the user cannot perform any actions on it.

	As Saved	The Project will be loaded as previously saved.
STL Importing	You can define in which memory state you want to load a project. This memory state becomes the default when importing a project.	
	Standard	This is the standard memory state of an STL file. Magics knows the placement of the triangles and their mutual dependencies. The user is able to perform actions on STL level (E.g. deleting triangles).
	Compact	The STL resides in the memory as read-only, therefore it uses far less memory than the Standard memory state. Magics does not know the placement of the triangles nor their mutual dependencies. The user is not able to perform actions on STL level.
	On Disk	The STL is saved on disk and unloaded from the memory. The STL will stay in the project but the user cannot perform any actions on it.
Always load STL's on disk	An STL with more triangles than defined by the user, will always be loaded on disk.	

– Importing .magics, .MatAMX and .3mf files

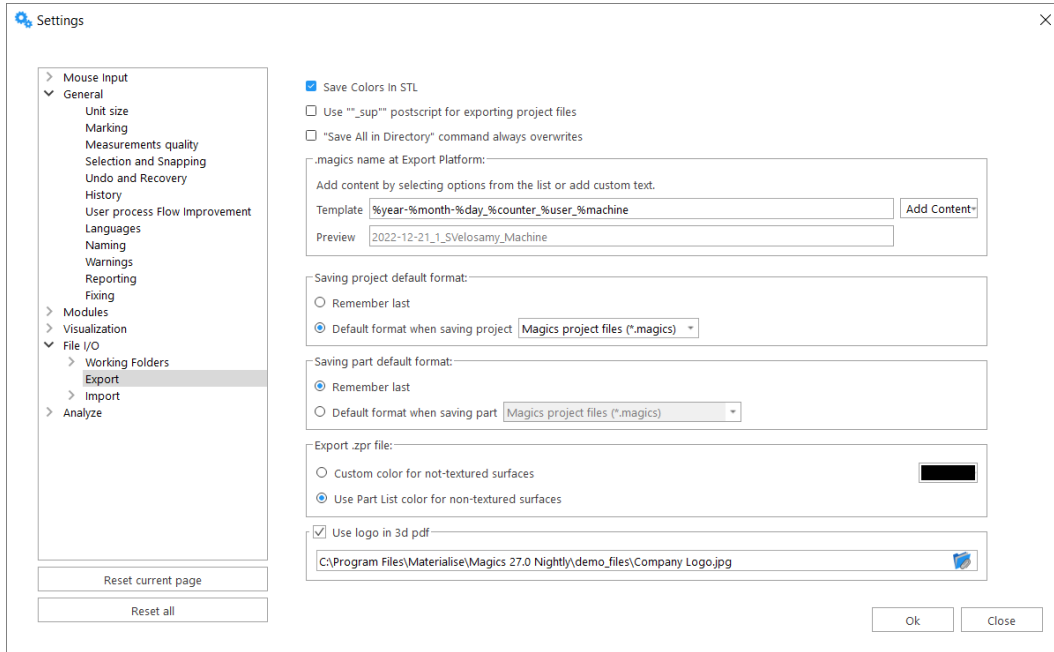
Show part list when importing	Check this option when you want to see a list of the parts saved in the imported .magics, .MatAMX, or .3mf file. In that list you can check the parts you want to be loaded. For more information, see 2 Import Part
-------------------------------	--

3. Slices




Convert to STL	.slc file will be imported as mesh
Import as stack of slices	.slc file will be imported as slices

Export

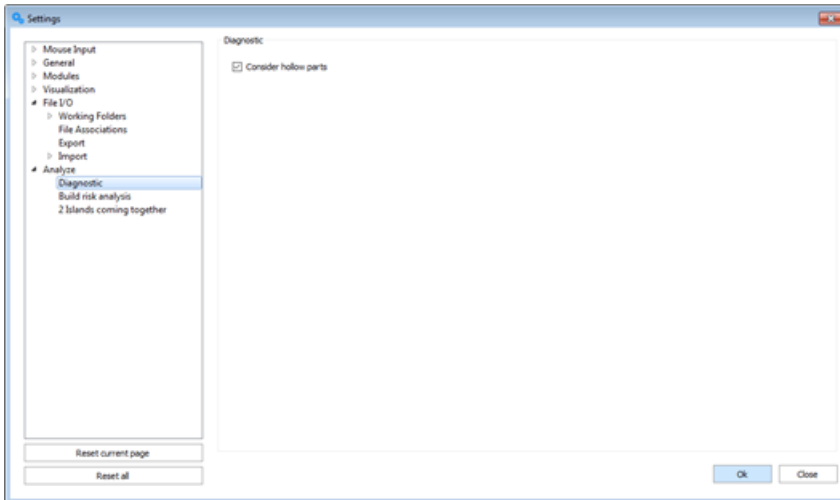


Save Colors in STL	When this option is checked, the parts will by default be saved as colored STL (Menu/Tools/Paint Part)	
Use _sup postscripts	When the user has created support for a certain file, the extension ' _sup' will appear behind the name of the part at the moment the user saves the part.	
Save All in Directory	Checking this option will overwrite parts without a warning message.	
.magics name at Export platform		
Template	Add the content which has to be displayed in the project name.	
Preview	A preview is shown of the naming for .magics files that are created via the export platform functionality	
Add Content		
	Year (YYYY)	Actual year the platform is created
	Month (MM)	Actual month the platform is created
	Day (DD)	Actual day the platform is created
	Build counter	Counts the amount of builds per day per

		platform
	User name	Displays the name of the system user who created the platform
	Machine name	The name of the platform that is created
	Project name	The name of the Magics platform
	<p>Example</p> <p>Template <input type="text" value="%year-%month-%day_%user_%machine"/></p> <p>Preview <input type="text" value="2012-07-03_ebielen_3D Systems SLA 250 (mm)"/></p> <div style="border: 1px solid blue; padding: 5px; margin-top: 10px;">  Remark: As seen in the example above, additional separation characters can be used. </div>	
Saving project default format		
Remember last	The default project saving format will be the last one used.	
Default format when saving project	Specify which format will be selected by default when saving a project	
Saving part default format		
Remember last	The default part saving format will be the last one used.	
Default format when saving part	Specify which format will be selected by default when saving a part	
Export .zpr file		
Custom color for non-textured surfaces	Select the color non-textured surfaces should receive when printed	
Use Part List color for non-textured surfaces	The printed color of non-textured surfaces will be the part color from the part list, as shown in the scene.	
Use logo in 3d pdf	Selected image will be used as the logo in 3d pdf file.	

5. Analyze

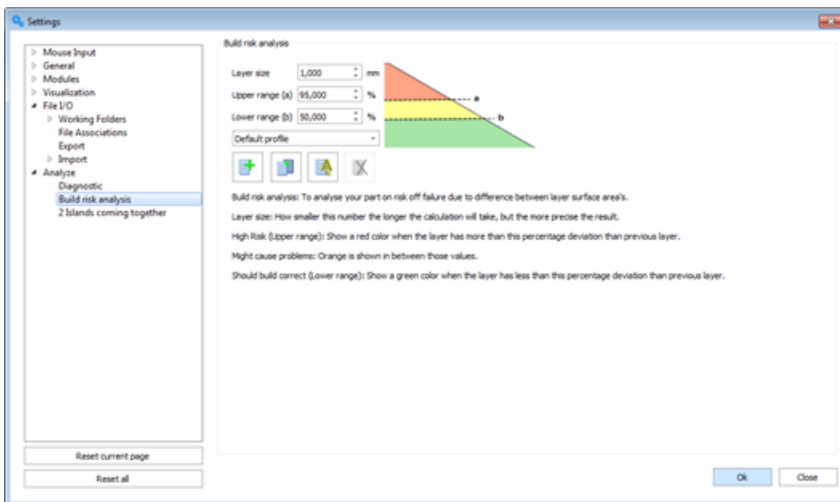
Diagnostic



Consider hollow parts	During automatic fixing, hollow parts are taken into consideration
-----------------------	--

Build risk analysis

To analyze your part on risk of failure due to differences between the layer surface areas.



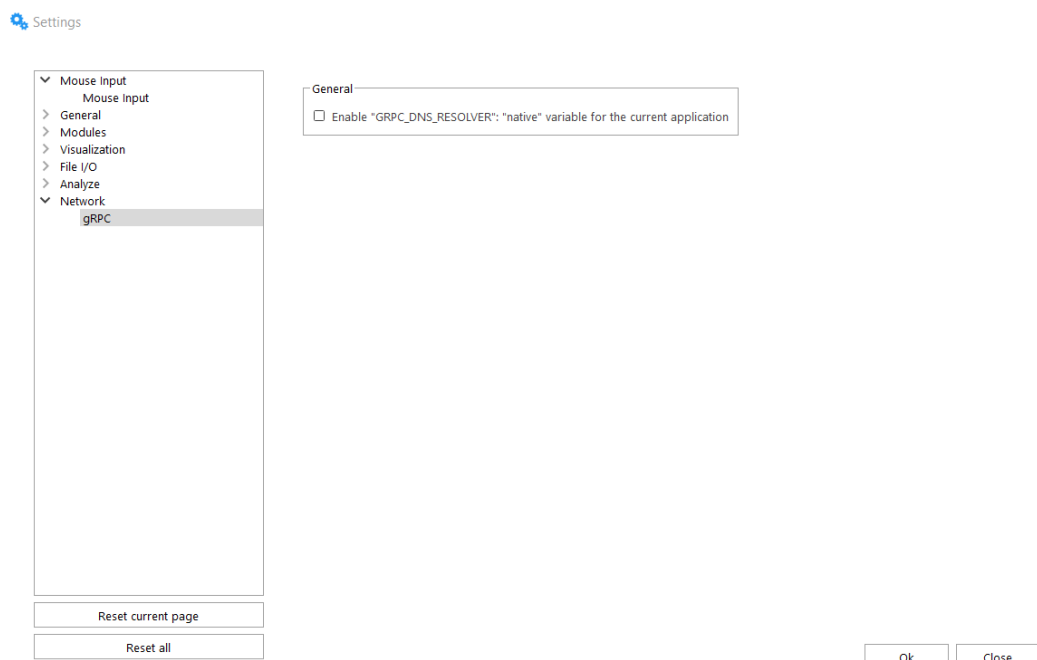
Layer size	The smaller the size, the longer the calculation will take, but the more precise the result will be.
Upper range (a) (red)	High risk: show a red color when the layer has more deviation than the specified percentage, compared with the previous layer.
(orange)	Might cause problems: Orange is shown for deviations that are between the upper and lower range.
Lower range	Should build correct: show a green color when the layer has less deviation

(b) (green)	than the specified percentage, compared with the previous layer.
Default profile	Select a previously created custom profile to load or save the settings, or change the default profile.

14.2. Network

gRPC

The gRPC settings page under the Network section has a general setting to enable "GRPC_DNS_RESOLVER" : "native" variable on the local PC. When enabled, it make the connection with services using a gRPC connector, such as CO-AM, more stable. Connection issues most commonly occur when using a VPN.

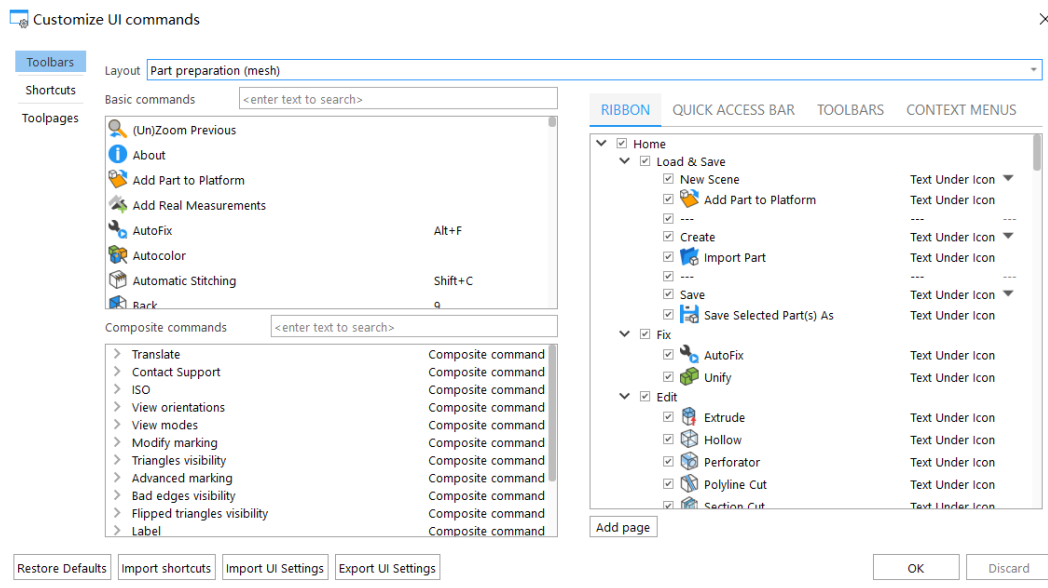


If using CO-AM, it is recommended to enable this option when the following error is present in the Magics log: *"Error code: 14, message: DNS resolution failed for {environmentName}: C-ares status is not ARES_SUCCESS qtype=AAAA name={environmentName} is_balancer=0: DNS server returned answer with no data, details:"*

14.3. Customize UI



In this dialog you can customize the ribbons, quick access bar, toolbars, context menus, toolpages and shortcuts.



Import UI Settings allows you to import UI settings file in .uimpf format. Once imported, the existing customization will be completely replaced with the new one.

Example: if in next Magics version a new command is added and placed by default in a specific place. When you import a profile created from a previous Magics version, Magics will not try to position this new command somewhere in the imported profile, but the new command will be available only via Quick search bar.

Export UI Settings allows you to export UI settings file in .uimpf format.

The saved profile contains information about position & visibility of commands in different UI areas (ribbons, quick access bar, toolbars, context menus), commands shortcuts, composite commands, position & visibility of toolpages, visual style

14.4. Magics Profile

A Magics profile is a profile that contains all user customizable settings in Magics. Each user can have its own Magics profile. These files have the extension *.mpf. It replaces or includes all previous profiles. It should include all user configurations of Magics.

This includes

- General settings
- RapidFit Files
- Fix profiles
- Report templates
- Machine parameters
- Streamics e-Stage parameters
- Structures
- 3D Nester profiles



1. Import settings of previous version

If you install a new Magics, it will look for existing settings on your computer. If it finds settings of the same Magics version, it will use these settings. If it finds settings of an older version, Magics will propose to upgrade these settings. This actually works very similar to importing a Magics profile.

Magics will import the general and customized settings of the previous version. Doing so, it will also refer to the correct location for machine files, e-Stage parameters, document generation templates and fix profiles.

1st situation

You are a new Magics user. The installation of Magics is performed for the first time.

Summary:

If you start Magics for the first time, Magics will search for existing settings. Because it is the first time that you start Magics, it is not able to find any settings of a previous installation. Default settings are created with all the default toolbars, Toolpages and shortcuts. You can either use these default settings to work with or you can customize them.

(See The Toolbars & The Shortcuts)

2nd situation

You already worked with Magics in the past. Previously you worked with Magics X and now you want to work with the latest version of Magics.

Summary:

Like in the first situation Magics will search for existing settings at startup. Because it is the first time that you install the latest version of Magics, Magics isn't able to find up to date settings. But old settings can be found. In this case Magics is going to give you the opportunity to upgrade your settings to the latest Magics version. If you want to take advantage of this opportunity, Magics will load your own customized settings. If you don't choose to do an upgrade, Magics will load the default settings.

3rd situation

You performed a re-installation of the same version of Magics. NO older version of Magics is installed.

Summary:

At startup Magics will be able to find up to date settings. (The same version of Magics was installed before) Magics will start up with the same settings as before. This can be the default settings (when no changes are made) or customized settings (when new toolbars, shortcuts are created).



4th situation

You performed a re-installation of the same version of Magics. Also an older version of Magics is installed.

Summary:

What happens when you perform a reinstallation of a Magics version, but you had in the past an older version installed? Since the same version of Magics already was installed on your computer, Magics will find the profile and will start up with this profile.

2. Import Magics Profile



A Magics profile can be imported from a certain location. When you import a *.mpf file, you will be asked what you want to import from the *.mpf file.

By default Magics will import the complete configuration and the configuration files will be stored in the default locations defined Magics.

The 'Advanced' options give you the possibility to choose what you want to import from the *.mpf file, and where you want to import it.

- General settings and RapidFit files will be placed in a predefined folder.
- For the fix profiles, report templates, mmcf files, and e-Stage par file, the user can choose between:
 - Importing the files in the path from profile
 - Importing the files in the folders assigned in the Magics settings.
 - Only referring to the path from profile and not importing the files.

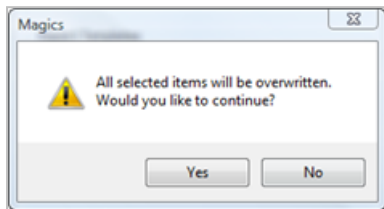
Do you want to import selected profiles?

<input checked="" type="checkbox"/> General Settings	<input checked="" type="checkbox"/> Rapid Fit Files
Fix Profiles	
<input type="checkbox"/> Use path from profile	<input checked="" type="checkbox"/> Import files
Report Templates	
<input type="checkbox"/> Use path from profile	<input checked="" type="checkbox"/> Import files
Machine Parameters	
<input type="checkbox"/> Use path from profile	<input checked="" type="checkbox"/> Import files
Materialise e-Stage Parameters	
<input type="checkbox"/> Use path from profile	<input checked="" type="checkbox"/> Import files
Structures	
<input type="checkbox"/> Use path from profile	<input checked="" type="checkbox"/> Import files
3D Nester Profiles	
<input type="checkbox"/> Use path from profile	<input checked="" type="checkbox"/> Import files
Macros	
<input type="checkbox"/> Use path from profile	<input checked="" type="checkbox"/> Import files
Measurement Quality Profiles	
<input type="checkbox"/> Use path from profile	<input checked="" type="checkbox"/> Import files

Yes No Help



Remark: A warning message appears when Magics detect that there is already an existing profile. This warning message is shown for each selected file type. (Fix profile, report templates, machine parameters & e-Stage parameters)



1st situation

You want to configure seven Magics installations of seven computers in the same way, but the users can have their own customization, like toolbars, background color, etc...

Summary:

You install Magics on one computer. You configure Magics like you want it to be. When you are ready configuring Magics, you export a Magics Profile.

You transfer the profile to each other computer you want to configure and you import the profile.

2nd situation

You want to restore your settings but the machine files and e-Stage parameters are on a shared network drive and you don't want to overwrite these files.

Summary:

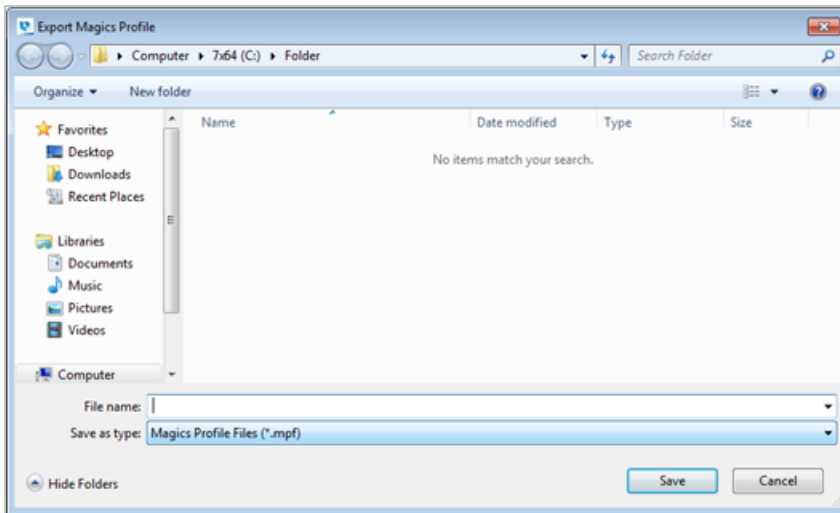
You have to make sure you have a .mpf file with settings you want to roll back to. During import you open the advanced options and you disable 'import files' for the 'Machine Parameters' and for the 'e-Stage Parameters'. Also you enable 'Use path from profile' for both 'Machine Parameters' and 'e-Stage Parameters'. If you continue the import of the profile, Magics will point to the correct network drives, but it will add settings to this location.

3. Export Magics Profile



You can create a .mpf file in the 'Options' menu with the function 'Export Magics Profile ...'. Magics will export all the configuration listed above to this file.

The export Magics profile option gives you the possibility to export your Magics profile to a certain directory. Magics will only save relevant files. (mmcf, e-Stage par file, stl files, *.dot files and *.xlt files) in the .mpf file.



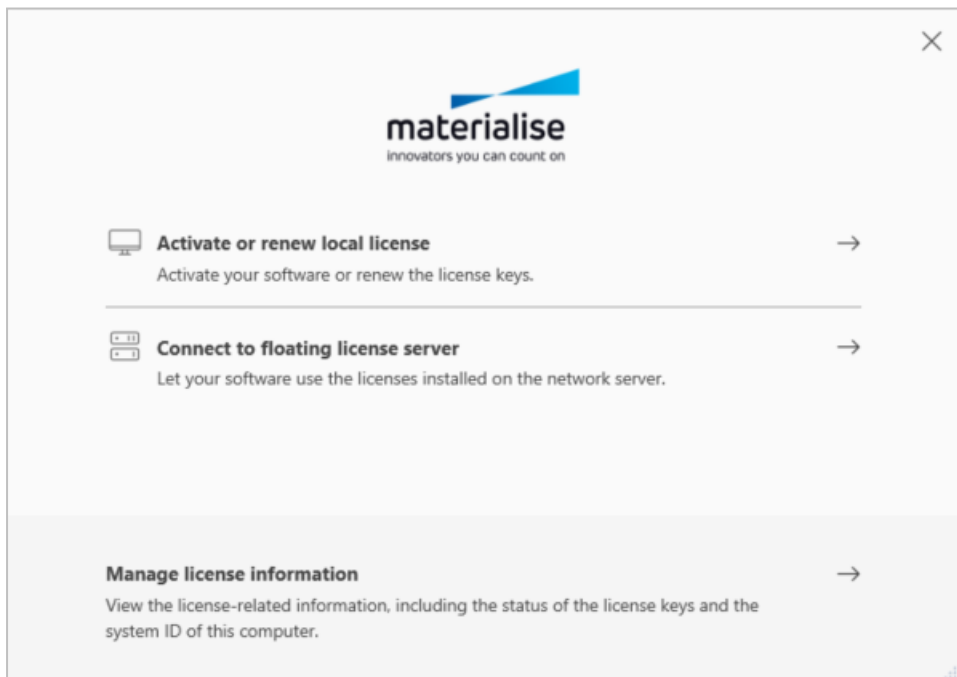
14.5. Licenses

1. Licenses



In the Registration dialog you can display your current license situation and contact information, request key files and register new modules.

- For details, please go to: <https://help.materialise.com/>



14.6. Help

1. Online Support



Direct user to the [Materialise Software Help Center](#) or send a support request automatically out of Magics to our customer support team.

Support Request - Required information

Fill out the support request form, select your nearest Materialise office and you can send an e-mail directly.

Please make sure the required fields are properly filled in. Our customer support team will then have all information to contact you as soon as possible.

Support Request

Select Office *
First Name *
Last Name *
Company *
E-mail Address *
Phone number
Details: *

* required fields

Additional information

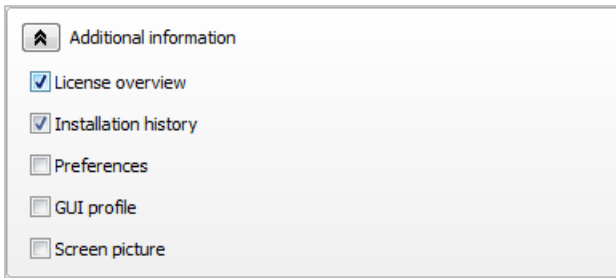
- License overview
- Installation history
- Preferences
- GUI profile
- Screen picture

Submit Close

Support Request - Additional information

To make sure that our customer support team has enough information regarding the problem some additional information can be included. You can select to add additional information to

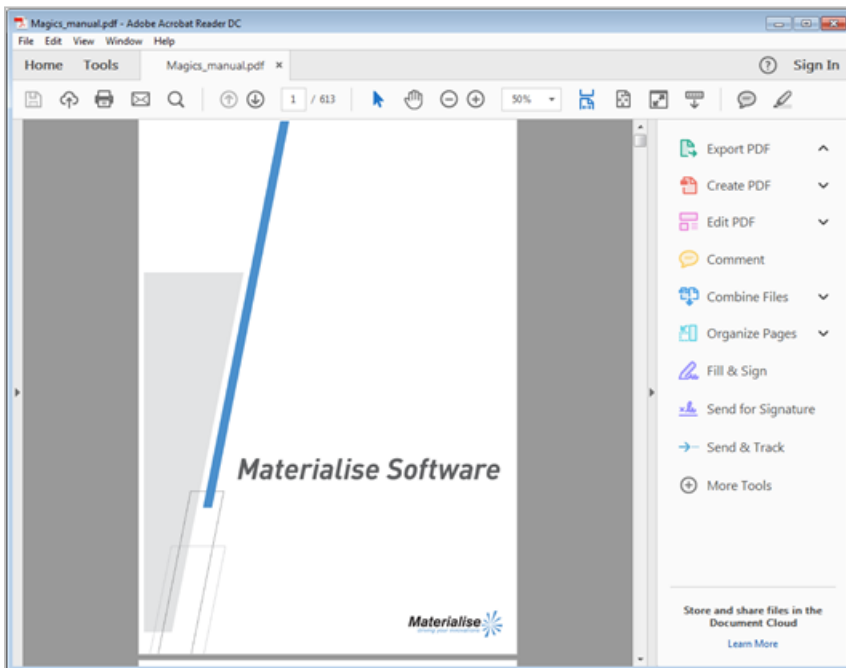
your support request which will give the customer support team more information about your Magics configuration. By sharing this information it is more likely that your support case will be solved more swiftly.



2. Manual



The manual of Magics offers you a clear and easy way to search for more information about a function. In almost each dialog is a Help button present to open directly the Magics manual.



3. What's New



Gives an overview of the new features in the Magics software, compared to the previous version.

14.7. About

1. About Magics



Gives the version details of the Magics software, and the performance parameters of the computer the software is running on.



14.8. Logging

1. Show log



Shows the log of the last performed actions by the Magics software. From the moment Magics runs, all the performed actions are written down in a log file. This file is automatically saved as a *.log file. Its name * is composed in the following way: 'Magics_year_month_date_time of first operation (hour, minutes, and seconds)'.

```

Log Window
10:02:42| TID15768|P|107|Operation parameters:
10:02:42| TID15768|D|107|236|Exit SG Module
10:02:42| TID15768||107|236|Operation time: Total (without pre operation) 93ms (pre operation 0ms, undo and autorecovery 93ms, operation 0ms). Total (with pre o
10:02:42| TID15768|S|107|236|Succeeded
10:02:42| TID15768|P|10056|Operation parameters: params.allowed_to_keep_as_stl_nonsolid = true;
10:02:42| OperationConvertSupportToStl:._PreOperation - there are no supports to convert
10:02:57| #SelectionMouse Click duration: 72 ms; Rectangle size: (0 x 0) px; Determined like click: true
10:02:57| TID15768|D|34|237|Part Selection
10:02:57| TID15768||34|237|Operation time: undo and autorecovery 63ms
10:02:57| TID15768||34|237|Selected Part via Mouse Click
10:02:57| TID15768|S|34|237|Succeeded
10:02:58| #SelectionMouse Click duration: 94 ms; Rectangle size: (0 x 0) px; Determined like click: true
10:02:59| #SelectionMouse Click duration: 94 ms; Rectangle size: (0 x 0) px; Determined like click: true
10:03:00| #SelectionMouse Click duration: 78 ms; Rectangle size: (0 x 0) px; Determined like click: true
10:03:02| TID15768|P|100|Operation parameters: params.m_enter_sg = true;
10:03:02| TID15768|D|100|239|Generate Support
10:03:02| TID15768||100|239|Operation time: Total (without pre operation) 578ms (pre operation 0ms, undo and autorecovery 94ms, operation 484ms). Total (with p
10:03:02| TID15768|S|100|239|Succeeded
10:03:02| ... on items:
10:03:02| ... - Box
10:03:02| TID15768|P|10054|Operation parameters: params.manual_support = true;
10:03:02| TID15768|D|10054|240|Generate Support
10:03:02| TID15768||10054|240|Operation time: Total (without pre operation) 140ms (pre operation 0ms, undo and autorecovery 0ms, operation 140ms). Total (with
10:03:02| TID15768|S|10054|240|Succeeded
10:04:12| TID15768|P|107|Operation parameters:
10:04:12| TID15768|D|107|241|Exit SG Module
10:04:12| TID15768||107|241|Operation time: Total (without pre operation) 78ms (pre operation 0ms, undo and autorecovery 78ms, operation 0ms). Total (with pre o
10:04:12| TID15768|S|107|241|Succeeded
10:04:12| TID15768|P|10056|Operation parameters: params.allowed_to_keep_as_stl_nonsolid = true;
10:04:12| OperationConvertSupportToStl:._PreOperation - there are no supports to convert
10:05:09| SupportParametersVersion is 10
10:05:09| SupportParametersVersion is 10

```

In the Settings (Settings > File I/O > Working Folder > Logging) you can define where you would like the files to be saved.

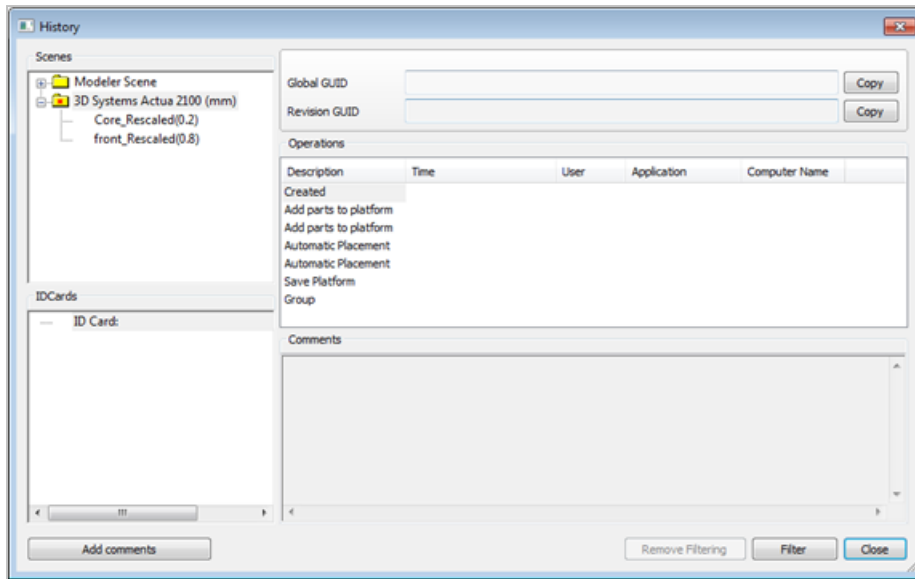
2. Show history



The history functionality lets you trace the full history of your part and platform. All operations performed in Magics are stored in a magics project file.

History information that is stored:

Description	Performed operations
Time	Time the operations is performed
User	Who performed the operation
Application	Magics full build number
Computer Name	Name of the computer used
Comment	Parameters of the operation



Remark: Some operations have an influence on the platform's history.

Example:

The cut & punch operation will add a part to the platform.



Chapter 15. Toolbars

Toolbar menus can be created by the customer according to his needs. The Marking Toolbar is by default present.

15.1. General toolbar



1. View tools

Zoom



To zoom in on a region, this region has to be defined by means of a box (drag from the left upper corner to the right bottom corner). When the mouse button was pressed, but no rectangle was drawn, the Zoom In 25% function will be applied. Zooming in and out can also be done using the mouse scroll.

Unzoom part



The zoom factor will be set so that all the active parts are displayed.

Unzoom platform



The zoom factor will be set so that the entire platform is displayed on the screen in the current view.

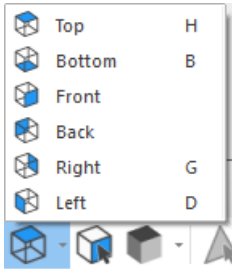
Home View



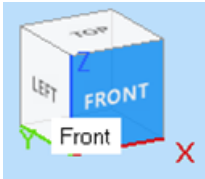
This button combines the Unzoom function and the Top Front Left ISO view option which allows the user to go back to the initial position with the best possible overview.

Standard views

The dropdown menu will display the list of standard views; by clicking on one view, the viewport will be updated to the selection.



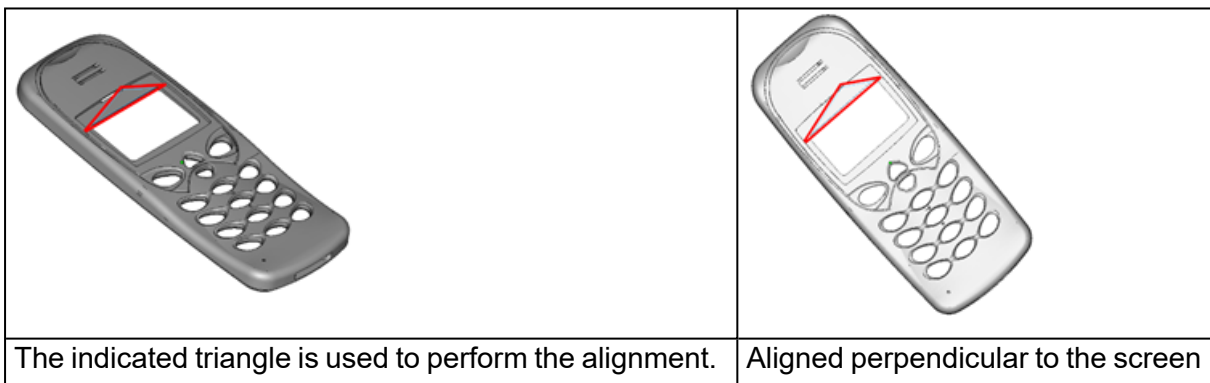
These views are also accessible by hovering over the faces of the interactive view cube, situated at the bottom right of the scene.



Indicate View Orientation

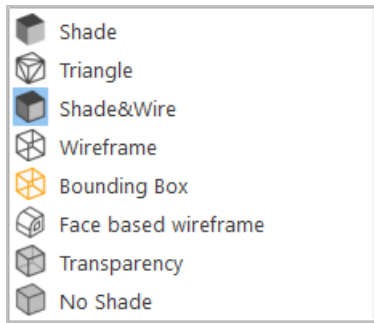












By selecting one triangle of a part, the indicate view orientation will automatically update the viewport so that the triangle normal is perpendicular to the screen.



Shade modes

The shade modes are the ways to visualize a part.



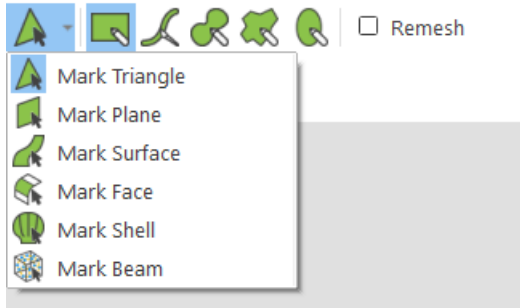
Shade		This visualization mode will display the part with shades according to the direction of the triangles.
Triangle		In Triangle mode, the triangles will be displayed upon the shaded part.
Shade&Wire		This mode will show a combination of the Shade mode and the Wireframe mode.
Wireframe		This visualization mode shows the edges of the object. This representation is deduced from the STL file. It has been tried to approach the normal wireframe representation as good as possible. But due to the limited information, STL errors and noise in the STL file, abnormalities in this representation may show up. A line of wireframe is drawn when the angle between 2 triangles exceeds a certain value. You can change this value and so adapt the wireframe view (Settings > Visualization > Wireframe).
Bounding Box		This mode will only show the bounding box of the part. This mode is quick in visualization.
Slice Preview		This mode will give you a preview of the slices.
Face based wireframe		If part wireframes are shown then the wireframe will be based on the face structure of the part obtained from the CAD file if exists.
Transparency		Parts are shown in a transparent mode.
Support Transparency		Supports are shown in a transparent mode.
No Shade		Parts are shown without rendering shadows.

2. Marking tools

When performing certain operations that requires input triangles or faces, the user would need to mark the required input entities. The part must first be selected before any entity can be marked! To indicate that an entity is marked the target entity will turn to default rendering color set in the settings (Default: Green). The marked entity and its edges color can be defined in the

Settings. When applying one of the options below, keep CTRL pressed to mark through your part, equally marking the entity through other entities relevant entities as well.

In the general toolbar, the user can select what they would like to mark: triangles, planes, surfaces, shells and others. When selecting these commands, a toolbar with marking options will appear below the ribbons.



Within this toolbar, the user can switch between what they want to mark, but they can also select how they would like to mark: rectangular/free-form/brush/... selection. Additionally, more options may be shown depending on the selected tools and methods.

What can be marked?



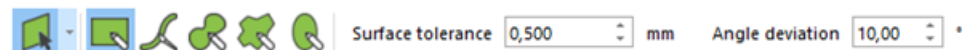
Mark Triangle: (un)mark individual triangles. Triangles can be marked one by one. Marked triangles can be unmarked by indicating them with the Mark Triangle cursor again.



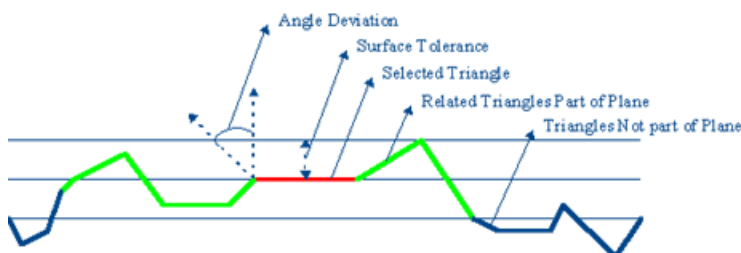
Note: The “Remesh” option is only available when “Mark Triangle” is selected.



Mark Plane: (Un)mark planes. By clicking one triangle, a whole plane can be selected or unselected. The plane is not necessary perfectly flat: the tolerance can be defined by the plane selection parameters, which will appear in the marking toolbar when selecting “Mark Plane”. These you can define in the Settings (Settings > General > Marking).



The indicated triangle will be the reference to mark the plane. Triangles that differ too much from the reference, will not be marked anymore. The user selects one triangle with the indicate plane cursor and an entire plane is marked. In order to do so, two tolerances have to be defined:



Surface Tolerance	Indicates the maximum deviation in mm or inches that a related triangle may have to be part of the same plane that contains the selected triangle
Angle Deviation	Indicates the maximum angle in degrees between the normals of a related triangle and the selected triangle, in order to be part of the same plane.



Mark Surface: (Un)mark surfaces. You have the possibility to mark a surface at once. A surface is defined by the wireframe (Settings > Visualization > Wireframe), which can be seen as the black lines on the parts, when the “Shade & Wire” or “Wireframe” view is activated. These black lines indicate the zones where two adjacent triangles have an angle more than the active value. (This value can be changed in Settings > Visualization > Wireframe, see **Wireframe, page 539**). Using the Mark Surface option, you will thus mark a smooth surface which ends at the sharp edges.



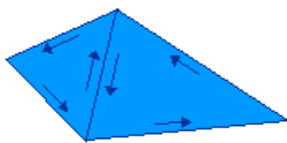
Mark Face : (Un)mark triangles that belongs to the underlying face structure(s). Face structures are usually obtained when importing parts from CAD files or the BREP Part scene. The same command can also be used in the BREP part scene to mark individual face(s). The faces can be identified by setting the shading mode on the part to enable "Face based wireframe" rendering coupled with "Shade & Wire" mode.



Note: Mark Face method is only method that is currently available for marking faces on BREP part(s).



Mark Shell: (Un)mark shells. A shell is defined as a limited collection of triangles correctly connected to each other. A triangle is part of a shell when the direction of rotation of the vectors of two adjacent triangles is opposite:



Direction of rotation of triangles in a shell



Mark Beam: (Un)mark beams from beam lattice structures. A beam lattice structure is made up of one or more beams and with this method, all the beams that falls within the region of the marking region will be marked quickly.

How can you mark?

Clicking: There is no separate option for marking through clicking. You can mark with a single click when “Rectangular Selection”, “Freeform Selection” or “Brush Selection” is active.



Note: Clicking selection is the only selection that is available for marking faces on BREP part(s).



Rectangular Selection: Make a rectangular selection by click-and-dragging. Hold Alt to make a square selection.



Brush Selection: Make a selection by painting through click-and-dragging. Hold Ctrl and scroll to change the brush size.



Note: “Remesh” is not available when using the Brush Selection method.



Freeform Selection: Make a freeform selection by click-and-dragging.



Polygon Selection: Make a polygon selection by clicking to place points and right-clicking to close the loop. Hold Alt to snap at 45° angles.



Ellipse Selection: Make an elliptical selection by clicking to place 3 points. Hold Alt to make a circular selection.



Note: All triangles that are touching or within the selection will be marked. So triangles (or other entities) don't have to be completely within the selection to be marked.

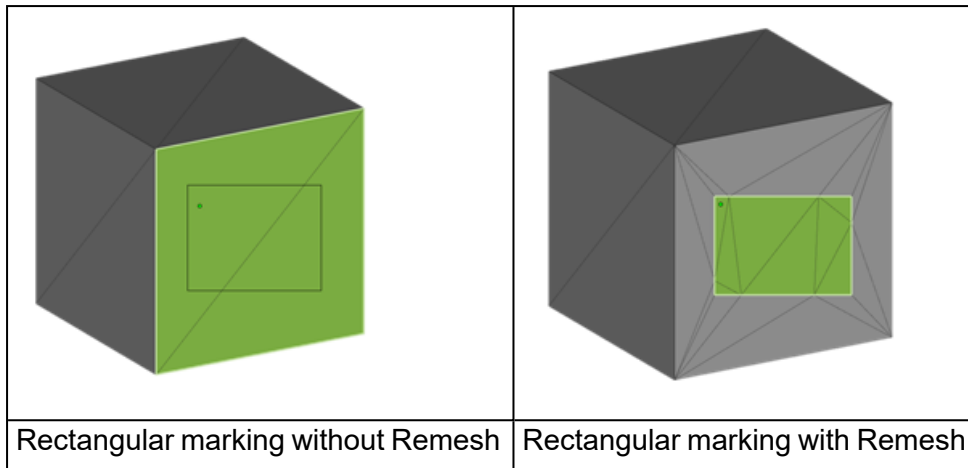
Additional tips and options for marking

Unmark: You can unmark entities (triangles, planes, surfaces, shells) by clicking them again. If you want to unmark using another type of selection, hold Shift.

Mark through: You can hold Ctrl to mark through a part.

Mark flipped triangles: By default, you can only mark triangles that are not flipped. To mark flipped triangles (when the red side is visible), hold X.

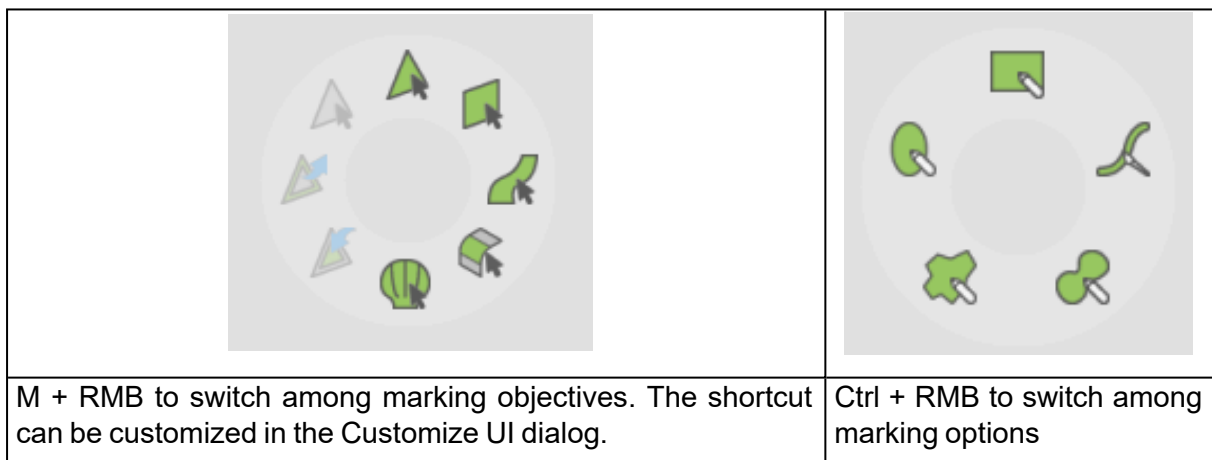
Mark with Remesh: When marking triangles, it is possible to enable “Remesh” in the marking toolbar. When “Remesh” is enabled, new triangles will be created to allow for a precise marking.



Note: You can also remesh parts with this tool without marking, by holding Shift.

Confine marking: This option is available for Mark plane, Mark surface and Mark shell. When this option is on, marking or unmarking will be limited with the boundaries that defined by the previously marked area or the invisible triangles.

Context menus: these are introduced for an easy switch under the marking mode.



Advanced marking

1. Mark Color



When you use this button, you can select any area of triangles with the same color. When pressing "Shift" when clicking on a colored triangle, all triangles of the part with the same color will be marked (they do not have to be connected).

2. Mark Texture



Click a texture on your part to mark all the triangles it covers.

3. Mark Horizontal



Mark all horizontal triangles.

4. Mark Vertical



Mark all vertical triangles.

5. Mark Contour



Mark all triangles that are part of the selected contour.

6. Select Noise Shells



Mark all noise shells of the selected part(s).

Invert Marked



With the toggle button the selection of marked triangles is inverted: all unmarked triangles turn green (or the defined marked triangles color) and vice versa.

Modify marking

1. Expand Selection



The group of selected triangles will become bigger. All unmarked triangles lying next to a marked triangle will be marked.

2. Shrink Selection



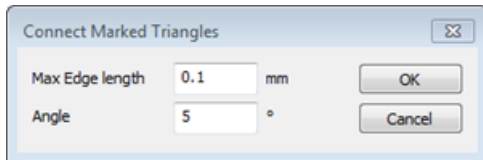
The group of selected triangles will become smaller. All triangles lying at the border of the group will be unmarked.



3. Connect Marked



When marking triangles, it can be difficult to select the very thin triangles. With this function, Magics will mark thin triangles between 2 marked triangles. When you click on this icon, you need to enter two parameters.



Max edge length	When the distance between 2 marked triangles is smaller than the given value, the triangles in between will be marked.
Angle	If the two selected triangles are too inclined towards each other- if their normals have an angle bigger then the value – the connect function will not work.

Delete marked



Delete all the marked entities on all the selected part(s).

Copy marked



The marked triangles are copied, a new part is created in the part list.

Separate marked



The marked triangles are separated from the original part and stored in a separate part.

Unmark All



All triangles will be unmarked.

Hide Marked



The marked triangles are hidden.



Make All Visible



Make all triangles visible.

Invert Triangle Visibility



Make invisible triangles visible and vice versa.

3. Errors visualization

Bad edges visibility

1. Bad Edges Visible



Defects in the STL file can be detected graphically. All edges that are not shared by exactly 2 triangles will be displayed in yellow on the screen.

2. Bad Edges Hidden Line



Hide the indication of the bad edges that are situated behind or inside the part.

3. Highlight Bad Edges



Bad edges are sometimes difficult to spot. The bad edges are drawn with thick lines to be seen easily.

Flipped triangles visibility

1. Flipped Triangles Visible



Defects in the STL file can be detected graphically. All triangles with a normal pointing into the screen are displayed in the Flipped Triangles color, which is by default red. Remaining red triangles in the shading indicate defects in the STL file (triangles with wrong-oriented normal, missing or overlapping triangles).

2. Flipped Triangles As Normal



Shows the flipped triangles as normal triangles.

15.2. Additional tools

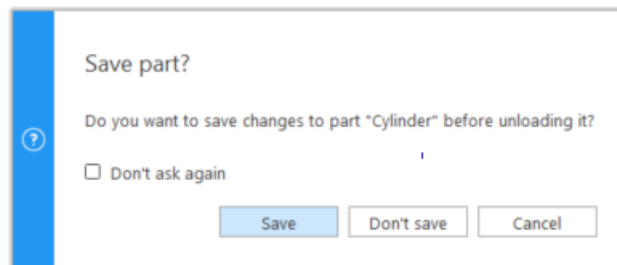
1. Unload Part



This command removes the selected parts from the active scene. If the user has selected several parts, these parts are removed at once. The Unload function does not affect any of the existing properties of the active scene or machine. The default shortcut for this command is CTRL + U.

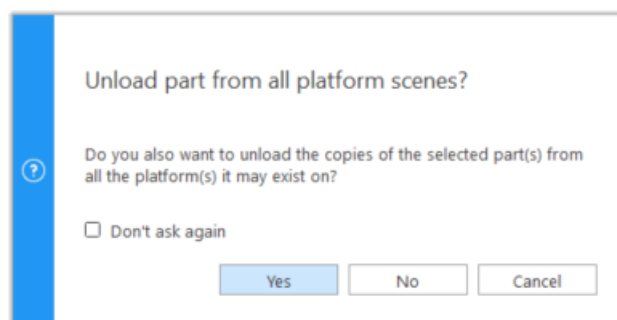
Prompt to save the part before unloading

The user is prompted to save the parts that will be unloaded if the part has been changed since the initial import. On save the save selected part(s) as dialog to save the part to file of the user's choice. This prompt will be shown when unloading part(s) from any scene.



Prompt to unload part from all platform scene

When user is unloading a part from the part scene, if one or more copies of the part may exist on one or more platforms, an additional prompt is shown to the user to unload all the parts from all possible platforms. If user chooses yes, then the part will be removed from the current active scene and all other platform scene that it may exist on.



2. Border marked Triangles



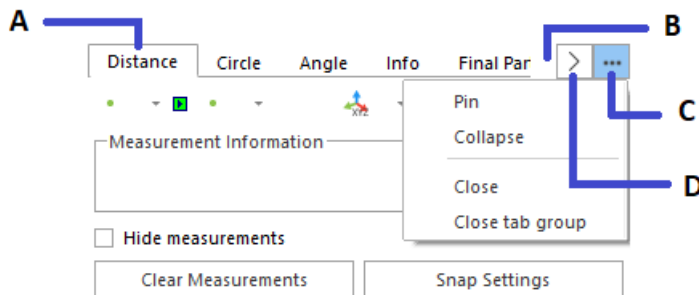
The border of the marked triangles is indicated by a red line or not shown.

This command is not present by default in the toolbar, but it can be added via the Customize UI dialog.

- See **Customizations, page 35** for more information

Chapter 16. The Toolpages

The toolpages are panels that contain content and actions about a single topic – e.g. measurements too to determine a distance. You get easy access to relevant content and/or actions grouped on a single topic per toolpage. You can decide which toolpages to visualize, depending on specific needs: dock a toolpage on the side of the scene, create toolbars containing single or grouped toolpages, leave toolpages floating. Toolpages can be collapsed or expanded by clicking on the specific toolpage tab or by clicking on the option in the menu button.



A: tab – B: tab bar – C: menu button – D: scroll button

The menu button offers you few options to control the visibility of the toolpage:

- Pin: pinning a toolpage (or a toolpages group) automatically expands it. When pinned, a toolpage (or a group) can't be collapsed manually or automatically by the auto-collapse feature
 - This option is available only for docked toolpages
- Collapse/Expand: click this option to manually collapse or expand a toolpage
 - This option is available only for docked and floating toolpages
- Close: click this option to close the active toolpage. You can visualize again the toolpage by enabling its visibility via the Customize UI window (see **Customize UI window, page 46** for more information)
- Close tab group: click this option to close the entire toolpages group. You can visualize again the toolpage by enabling its visibility via the Customize UI window (See Customize UI window, page 1 for more information)

When a toolpages group contains multiple tabs and these cannot be visualized all, a scroll button appears on the right side of the tab bar. Click to open a menu list of all toolpages present in the group and select the toolpage to activate.

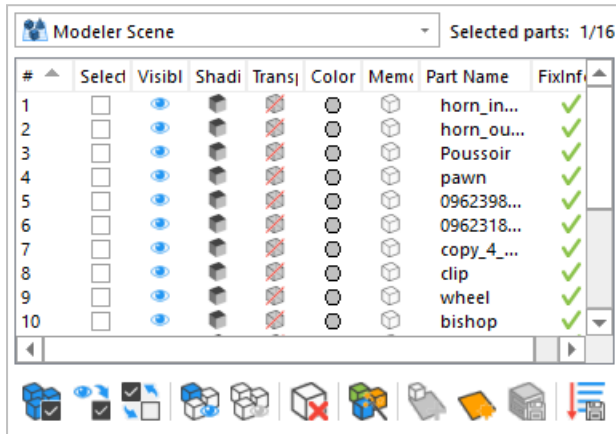
For some toolpages, Magics determines the height automatically (e.g. Part List or Surface List in the Support Generation module). This is dependent of:

- The space left over
- The space needed

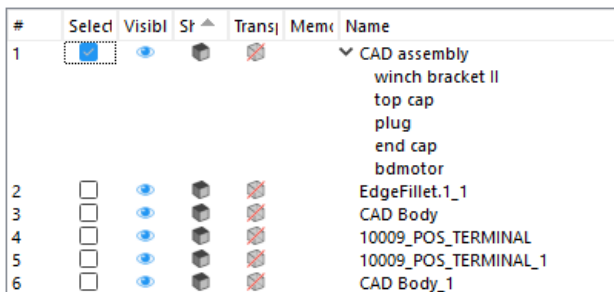
16.1. General Pages


1. Part List Page

The Part List Toolpage keeps track of the parts in the Part Scene and of the virtual copies in the Platform Scenes.



For a BREP assembly, all its components have a tree structure representation and act as a single part entity in the Part List .



 **Note:** BREP Part(s) assemblies are only available in the Part List Toolpage in the Part Scene in Magics.

Selected parts:	Number of selected parts / Total parts in the scene
-----------------	---

#	Number of the part. Can be used for ordering.
Selected	When the checkbox of a part is checked, then the part is selected.
Visible	Show/hide the part by clicking the “eye” icon.
Shading	The user is able to define a separate visualization for each part. (Changing the visualization of a virtual copy on a Platform Scene will change the visualization of all virtual copies of the same mother part).

	<i>Visualization mode of parts</i>
	Hide Hide the respective part.
	Shade Shade will display the respective part with shades according to the direction of the triangles.
	Wireframe Wireframe shows the edges of the respective part.
	Shade&Wire The respective part is showed in a combination of the shade mode and the wireframe mode.
	Triangle* The triangles of the respective parts will be displayed upon the shaded part.
	Bounding Box This mode will only show the bounding box of the respective part.
	Transparent This mode will show the respective part as transparent.
	No Shade This mode shows the part without shading.
	<i>Visualization mode of slices</i>
	Slices* This mode will only show the slices of the respective part.
	Bounding Box This mode will only show the bounding box of the respective part.
Transparency	Toggle the transparency of the part
Color*	The color in the circle represents the color of the matching part. Clicking this circle leads you to the material editor dialog or to the color palette. (Changing the color of a virtual copy on a Platform Scene will change the color of all virtual copies of the same mother part).
Memory State*	The user is able to define a separate memory state for every part. (Changing the memory state of a virtual copy on a Platform Scene will change the memory state of all virtual copies of the same mother part).
	Compact The STL resides in the memory as read-only; therefore it uses far less memory than the Standard memory state. Magics does not know the placement of the triangles nor the mutual dependencies of the triangles. The user is not able to perform actions on STL level.
	Normal This is the standard memory state of a STL file. Magics knows the placement of the triangles and the mutual dependencies of the triangles. The user is able to perform actions on STL level (E.g. deleting triangles)
	On Disk The STL is saved on disk and unloaded from the memory. The STL will stay in the project but the user can't perform any actions on it.
Part Name	This column contains the names or the path of the loaded parts. In case of a Platform Scene, this column will show the name of the virtual copies. The

/Copy of	name of a virtual copy is the same as the name of the mother part. If the name doesn't fit in the column a pop up, containing the full part name, will be shown when hovering over the name.
FixInfo*	This column contains the fix status of the part. Double clicking on "n/a" will diagnose the part, double clicking again will perform an autofix on the part.
Extend with columns	Here you have the possibility to add or remove additional columns to the part list overview

**this parameter is unavailable for BREP Part(s) and assemblies*

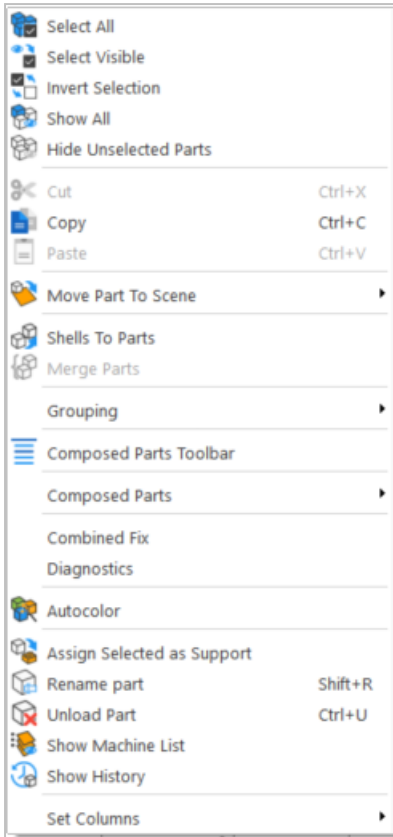
Select All	Selects every part in the list.
Select Visible	Selects all visible parts on the current scene.
Invert Selection	All selected parts are unselected and the unselected parts are selected. The Invert Selected functionality works on the Selected (S) column. Invisible parts will thus become visible.
Show All	All invisible parts are made visible.
Hide Unselected	Hides all unselected parts.
Unload Selected Parts	Unloads all selected parts.
Auto Color*	This button colors the parts as if they were newly loaded in Magics.
New Scene**	The Select Machine dialog will pop up, where you can select the machine of the newly created Platform.
Export platform **	This function is only available when a platform scene is loaded. It gives the possibility to export all loaded parts from Magics in one mouse click.
Save Order**	Saves the new order of parts after drag 'n dropping them in the list.

**this function is unavailable for BREP Part(s)/assemblies*

*** this function is available only for Platform Scene(s)*



Remark: When "Out of Bounds" is enabled, parts that are out of bounds will be also be marked in red in the Part List.



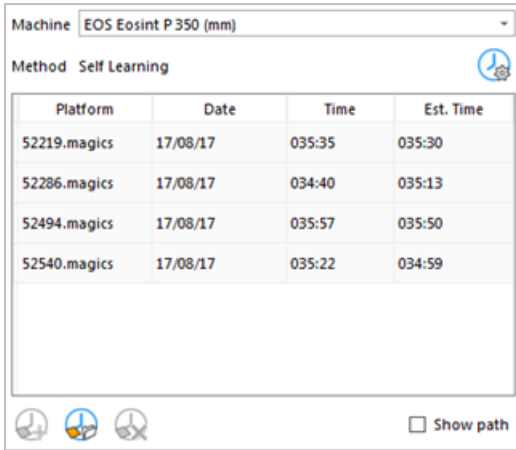
Select All	Selects every part in the list.
Select Visible	Selects all visible parts on the current scene.
Invert Selection	All selected parts are unselected and the unselected parts are selected. The Invert Selected functionality works on the Selected (S) column. Invisible parts will thus become visible.
Show All	Visualize all parts
Hide Unselected	Hides all unselected parts.
Cut Part(s)	Selected virtual copies of a platform can be cut to the clipboard.
Copy Part(s)	The selected parts or virtual copies are copied to the clipboard.
Paste Part(s)	Parts are pasted from the clipboard in the active scene.
Shells to Parts	This function will split the selected parts, if they consist of more shells, into different parts. Now, each shell will represent one part.
Merge Selected Parts	The selected parts are merged into one STL.
Grouping	The grouping functionality places parts in groups so that these are handled as one part for a number of operations.

	Group	Create a group of selected parts
	Ungroup	All parts within the group become individual parts again.
	Remove from group	Remove selected parts from an existing group
Diagnostics	Runs Diagnostics	
Combined Fix	Executes the Combined Fix	
Autocolor	This button assigns random colors to the parts that are newly loaded in Magics.	
Assign Selected as Support	Assign selected parts as support.	
Rename part	Rename selected part(s). If multiple parts are selected, only same prefix and suffix will be added to the parts.	
Unload Selected Parts	Deletes all the selected parts from the workspace.	
Show history	Trace all steps performed of your part and platform in a history overview	

Set Columns	Pops up a sub menu where the user can define the visualized columns.	
	#	Part order.
	Selected	Shows the part is selected or not.
	Visible	Shows the visibility of the part.
	Shading	Shows the visualization of the part.
	Transparency	Shows the part is transparent or not.
	Color	Shows the color of the part.
	Memory state	Shows the Memory State of the part in the Part List.
	Name	Shows the part name of the parts.
	Path	Shows the path of the part in the Part Name column.
	Copy of	Shows the master part.
	Surface	Shows the Surface of the parts in the Part List.
	Volume	Shows the Volume of the part in the Part List.
	Number of Triangles	Shows the amount of triangles used in the Part List.
	Number of Points	Shows the amount of points in the Part List.
Number of Invisible Triangles	Shows the amount of invisible triangles in the Part List.	
Wireframe angle	Shows the wireframe angle.	
	Support	Shows the part is a support or not.
	Solid	Shows the part is solid or not.
	FixInfo	Shows the fixing info of the part.
	Attribute	Shows the attribute of the part: Master, Shaping unite, Shaping subtract, Shaping Intersect, Auxiliary or Zoning.


2. Build Time Estimation page

The Build Time Estimation Toolpage keeps track of the settings for the build time estimation calculations; all machines stored in the My Machines library, including the Build Processor machines, are available in the toolpage.



Machine	This is a dropdown list containing all machines stored in the My Machines library. The machines are displayed following the priority assigned in the My Machines library.	
Method	The method chosen for build time estimation calculations is displayed. Click on the icon on the right side to edit the settings for the method chosen; the Machine Properties dialog will be automatically open.	
List of teaching platforms	The list is enabled only when the build time estimation method chosen is Self-learning. Here you can visualize and manage the data of your teaching platforms used for the build time estimation calculations.	
	Platform	The file name of the Magics project used as a teaching platform is displayed.
	Date	The value corresponds to the date when the teaching platform has been imported.
	Laser	This value shows if it is a laser-based system.
	Time	Insert the real build time of the teaching platform (hours and minutes). Double click on the field to edit the value.
	Estimated time	The value corresponds to the estimated build time of the teaching platform.
	Extend list with additional data	<p>You have the possibility to add or remove additional columns; right mouse click on the list headers to display a list of additional columns.</p> <div style="border: 1px solid gray; padding: 5px; width: fit-content;"> <ul style="list-style-type: none"> ✓ Platform ✓ Date ✓ Time ✓ Estimated Time Absolute Error Relative Error Number of parts Z Height Added by </div>

	Add current platform	Click on this icon to automatically add the current scene as a teaching platform.
	Import platform	Click on this icon to import a Magics project as a teaching platform.
	Delete platform	After selecting one or more teaching platforms from the list, click on this icon to delete the selected teaching platform(s) from the list.
	Show path	Select this option to display in the Platform column the file path instead of the Magics project file name.

 Remark: Teaching platforms that miss teaching information are displayed in red in the teaching platforms list.

16.2. Part Pages

1. Part Info page

The properties of the parts in the Part List are displayed. All dimensions are displayed in the selected units. Select one part on the scene to visualize its part information on this toolpage.

Dimensions			
	Min	Max	Delta
X	67.241	85.140	17.899 mm
Y	71.542	87.140	15.598 mm
Z	77.350	104.399	27.049 mm
Volume			1818.279 mm ³
Surface			1259.653 mm ²
<input checked="" type="checkbox"/> Automatic update		<input type="button" value="Update"/>	
Mesh info			
# Triangles	9152	# Marked	0
# Points	4575	# Invisible	0
Extra info			
Status	Not changed		
Z Compensated	No		

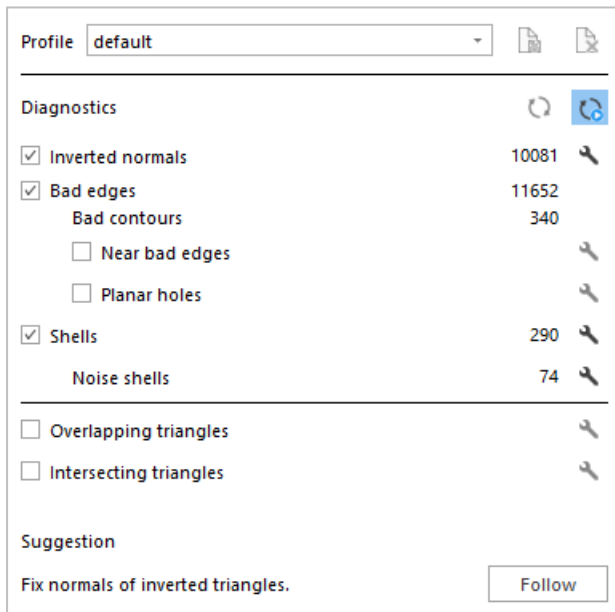
Dimensions	The dimensions of the part. The minimum and maximum coordinates (X, Y, Z) of the part. The delta value is the difference between the minimum and maximum.	
	Volume	The volume of the part.
	Surface	The surface of the part.

	Automatic update	If checked, the Volume and Surface will be updated automatically.
	Update	Manually update all the information by clicking on this button; by default the Volume and Surface information has to be updated.
Mesh info*	# Triangles	The amount of triangles of the part.
	# Points	The amount of points of the part.
	# Marked	The amount of marked triangles of the part.
	# Invisible	The amount of invisible triangles of the part.
Extra info*	Status	The status of the STL-part. If no modifications are made to the loaded part, the status is Not Changed. In the other case, the status is Changed.
	Z Compensated	Indicates if the part is z-compensated or not.

*These properties are not available for BREP Part(s)

2. Part Fixing Info page

The Part Fixing Information Toolpage will guide you through the essential steps to fix a corrupt STL.






Diagnostics

1. Introduction

The Diagnostics section is the key-step in the part fixing process. In this step you can always determine what is wrong with the STL-file. Based on the Diagnostic, a suggestion will be given as a guideline through the fixing process.

2. Advised way of working

- Check the items you want to analyze
 - You can auto fix the item by pressing the Fix button  on the right.
 - A Full Analysis will get you the best results
 - Press the “Refresh” icon  if the “Autorefresh” option  is off
 - The errors are shown
 - Look at the “Suggestion” section: An advice is given on the available data.
 - Items that are not checked are not taken in account to determine the advise
 - Press the “Follow” button to perform the advised action.
 - For the errors that cannot be fixed in this way, go to Fixing Pages to do a manual fix.
- See ***Fixing Pages, page 590.***

3. Tips and tricks

Change the advice

You can influence the advice with the check boxes. When unselecting a checkbox, the advice will not take this parameter in account. When Magics keeps on sending you to a certain fixing step, you can skip it this way.

Do I need a full analysis?

A full analysis is giving you the best result but consider that

- Each analysis takes time (especially the overlapping triangles and intersecting triangles)
- In the beginning you often do not need all information (especially the overlapping triangles and intersecting triangles)
- Depending for what you're going to use your STL-file afterwards, you may not need to repair intersecting and overlapping triangles.

Errors explained

1. Inverted Normals

In the STL format, a normal indicates the outside of a triangle. When the normal points to the wrong direction (the inside), the triangle needs to be inverted to have a watertight STL. This triangle is then called a flipped triangle.

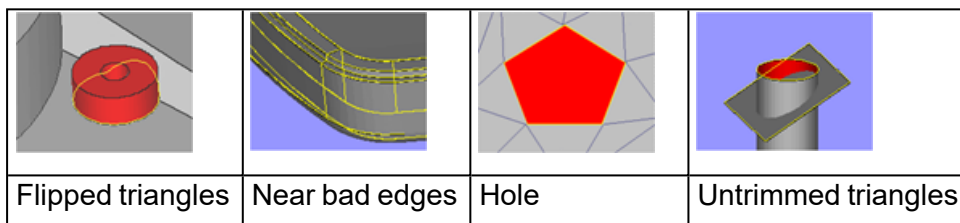
2. Bad edges

To have a correct STL file, all edges of each triangle should be connected properly to a neighbor. If an edge is not connected properly, the edge is called a bad edge and is indicated with a yellow line. A group of connected bad edges will make a bad contour. The STL file will be sliced in a subsequent step. To process the slicefiles correctly, every slice needs to be closed. This is why Bad edges need to be fixed.

3. Bad contours

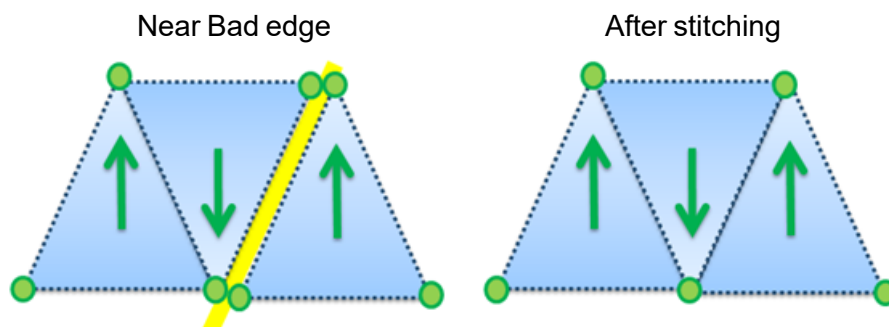
A group of bad edges connected to each other form a bad contour. E.g. the hole below has 1 bad contour that consists of 5 bad edges.

Some examples of common problems



– Near bad edges

Near bad edges are bad edges that are near other bad edges. These are mainly caused by 2 surfaces that are not well connected. You can recognize them as long yellow lines on the part. You can fix them easily by stitching. Stitching is an automatic operation that will unite two neighboring triangles which both have a bad contour right next to each other.

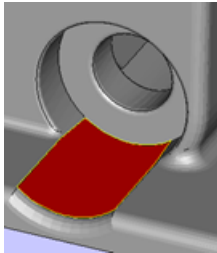


– Planar hole

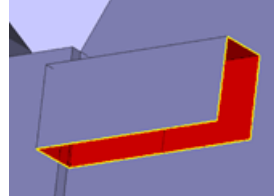
A hole consists of missing triangles. Use fill hole to fill it up. Magics is only able to recognize planar holes, which are recognized by the open contour which lies more

or less in one plane. Holes caused by more irregular contours will not be recognized by Magics and be shown as a bad contour.

Planar hole



Bad Contour



4. Intersecting triangles

Intersecting triangles are triangles cutting each other. It can happen sometimes that the STL surface has intersections. Depending of the application of the STL file, it's advised to remove the intersections. You can remove them with the Unify function on the Triangles Page.

- For Rapid Prototyping: This depends of your post processor. Normally, intersecting triangles are not a problem, however, some (older) slicers require a perfect shell without intersections.
- FEA: It is REQUIRED that there are no intersecting triangles.

5. Overlapping Triangles

An STL-file sometimes has overlapping triangles. These triangles can be removed with the tools in the double surfaces page.

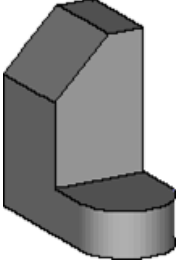
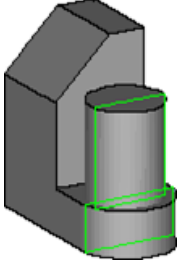
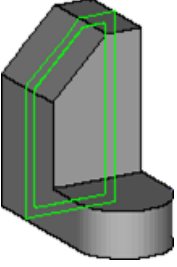
2 triangles are considered as overlapping as:

- The distance between them is smaller than the given tolerance. (E.g. 0,1 mm or 0,005 inch)
- The angle between the normal of the triangles is smaller than the given angle. (E.g. 5°)

Take into account that some "false alerts" may occur. When the triangles comply with the parameters, they will be marked as overlapping triangles, even when they are part of the geometry.

6. Shells

A shell is a collection of triangles connected to each other. Normally a part has only one shell because every triangle of the part is (indirectly) connected to every other triangle. Parts with:

1 shell	2 shells	2 shells
		
Every triangle is connected to each other.	The cylinder is not connected properly to the part. The overlap between the two shells can cause build failure. This can be solved with a Unify.	The part is hollow. The inner shell is not connected with the outer shell. This is normal with hollow parts.

7. Noise shells

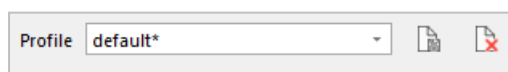
Some shells have no geometrical meaning and are considered as noise (waste) that we can throw away. However, it is recommended to look at these shells first before removing them. Even a shell of a few triangles can be important.

16.3. Fixing Pages

For all the possible errors, please see **Errors explained, page 587**.

1. Profiles

Maybe you're having STL's coming from different sources and for each "type" of STL file you need different settings for optimal fixing. With profiles, you can easily store your settings in profiles and load them again.





A default profile with default fixing parameters is always available for selection. This is called "default". If any value of "default" is changed, this setup can be saved to a new profile.

If the parameters are modified while a profile is selected, this profile will be marked with a *. The changes will become permanent if the profile is saved again.



2. Autofix page

Profile  


Invert normals

Stitch near bad edges

 Max gap size mm

 Iteration

Remove noise shells

Fill holes 

Only planar holes

Unify shells

Filter sharp triangles

Here you can decide yourself what actions you want.

Invert normals	Magics will reorient the normal of the triangles automatically.
Stitch near bad edges	Two bad edges (yellow lines) which are close enough to each other can stitched automatically by pulling the open edges towards each other. This way, you get a watertight stl. As soon as the bad edges are too far apart (Tolerance parameter), stitching will deform the design and the bad edge is treated as a hole.
	Max gap size Here you indicate what distance a point may be moved to fix the near bad edge.
	Iteration To get better results, the stitching is done in iterations, starting with a small tolerance and ending with the given tolerance
Remove Noise Shells	The automatic removal of detected noise shells, these noise shells make no geometrical sense.
Fill holes	Magics will only fill a contour when it recognizes it as a hole. Some contours are not holes.
	Only planar holes Magics will only fill a contour when it recognizes it as a planar hole. Some contours are not planar holes.
	Planar The hole will be filled as a planar hole
	Freeform Complex shaped contours are better filled using the freeform algorithm. Grid: The triangle size of the surface that is used to fill the contour
Unify shells	This will remove all internal geometry and intersecting triangles. This operation will only be done if the geometry allows it.

Filter Sharp triangles	Sharp triangles will be removed to improve surface quality.
------------------------------	---

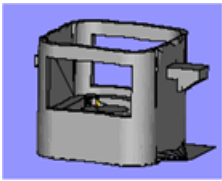
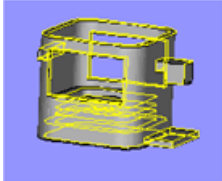
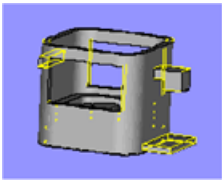
3. Near Bad Edges page

Profile  



Max gap size mm




Iteration

Magics will reposition points of triangles so that surfaces fit 100% correctly. For a Manual stitch, 2 parameters are needed.

Max gap size	The Stitch function will close gaps between all bad edges if the gap is smaller than this value. If the value is too small, not all near bad edges will be solved; if the value is too high, the geometry will be changed.	
Iteration	How many times the stitch algorithm should be applied. To avoid errors caused by high tolerances, Magics can stitch in iterations, starting with a stitch with a very small tolerance and ending with the given tolerance.	
		
Original file	Max gap size was too small. Still near bad edges visible	Max gap size was too high. Deformations of the part

4. Hole page

Profile  

Method   

Mark new triangles

Treat as one hole

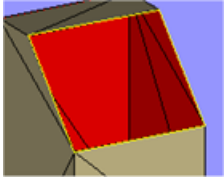
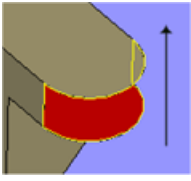
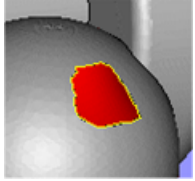
Part name	Length (mm)	# Edges
Body		
1	1.6929	3
2	0.0841	3
3	6.2523	350
4	0.0872	4
5	1.1099	38
6	0.1851	14
7	1.4758	64
8	0.3384	10
9	0.185	14

Introduction

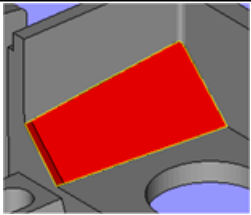
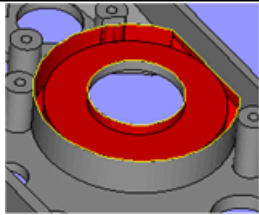
In some mesh parts some triangles may be missing that are typically considered a hole. A hole is a special kind of bad contour; It's a bad contour with no triangles on the inside.

First you'll need to identify what kind of hole you're dealing with:

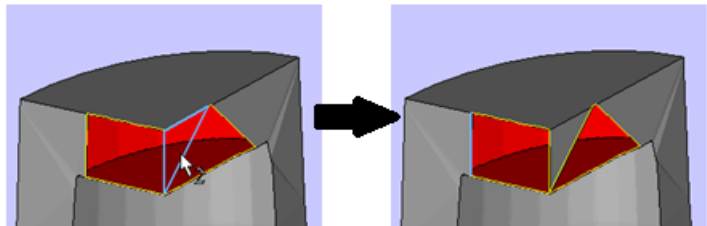
– Fill type

Planar	Ruled	Freeform
A simple hole which can be solved automatically	For correct fixing, the user should determine a direction (or rule) how the hole should be fixed	A smooth surface will be generated to fill up the hole
		

– Single or Multi-contour hole

Single Contour	Multi - Contour
The hole is a simple hole, consisting of 1 contour	The hole consists of multiple contours that must be connected
	

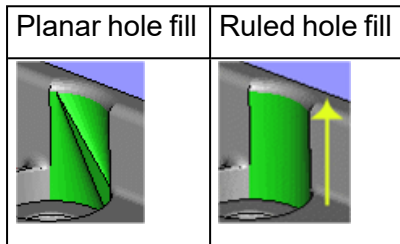
– Triangle creation

Create triangle
By creating a triangle or a bridge, you can manually draw new triangle(s) which connect two bad edges. This way, complex holes can be subdivided more simple holes


Fill type

Planar: Use this hole filling-type for simple holes. The hole will be filled as planar as possible, with respect to the shape of the contour

Ruled: Use this hole filling for holes that shouldn't be filled as planar as possible but where the triangles should be positioned parallel to a certain direction. This extra indication of the "filling direction" helps Magics to understand the geometry and has to be indicated manually.

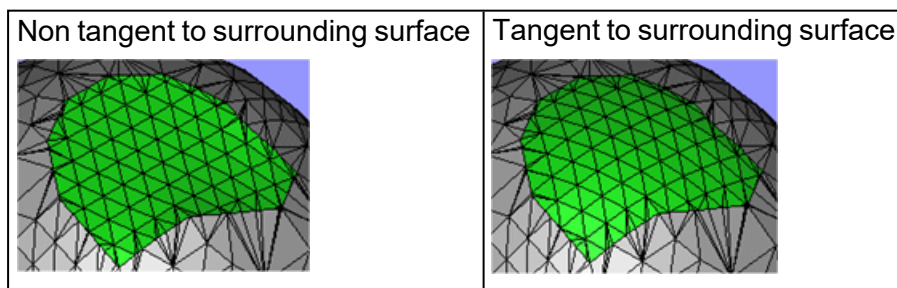


Filling direction: In what direction the triangles have to cross the hole to fill the hole correctly.

- You can select the X, Y or Z direction
- Select User defined option and click on Indicate line button to indicate a line on the wire-frame or a bad edge as filling direction

Freeform: For freeform hole filling, Magics will put a grid of triangles in the bad contour to ensure a fluent surface.

- Grid size: the size of the triangles
- Automatic: When Automatic is switched on, the generated grid will be generated according to a selected setting between Very coarse and Very fine.
- Tangent: When Tangent is switched on, the generated surface hole fits perfectly on the surrounding triangles.



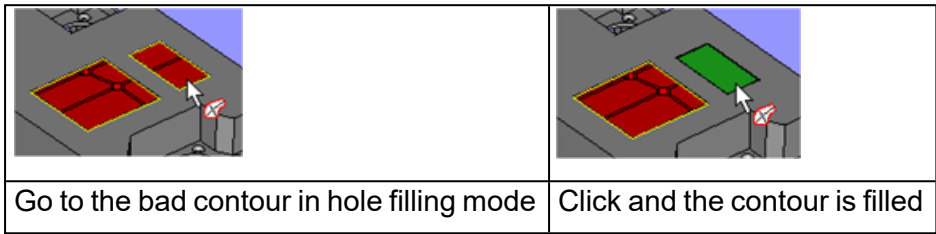
Single or Multi-contour hole

Single Contour



The easiest way to fill Single Contour holes is to use the "Fill hole mode".

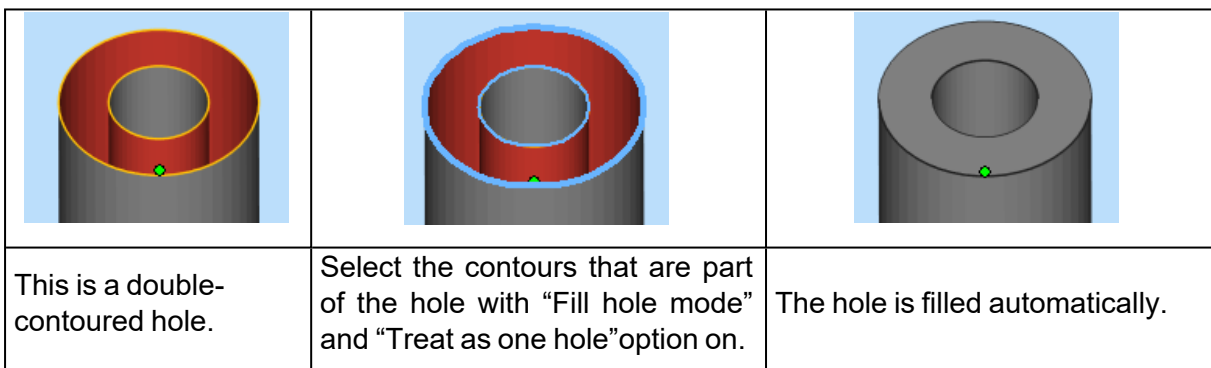
- Be sure you choose the right hole filling mode (Planar, Ruled or Freeform)
- Go with the cursor to the bad contour of the hole and click.
- New triangles will be created automatically to fill up the hole.



You can also fix single contour holes by first selecting the holes with the 'Select Holes' mouse mode or by selecting the hole from the holes list in the toolpage and then clicking the 'Fill Selected' command.

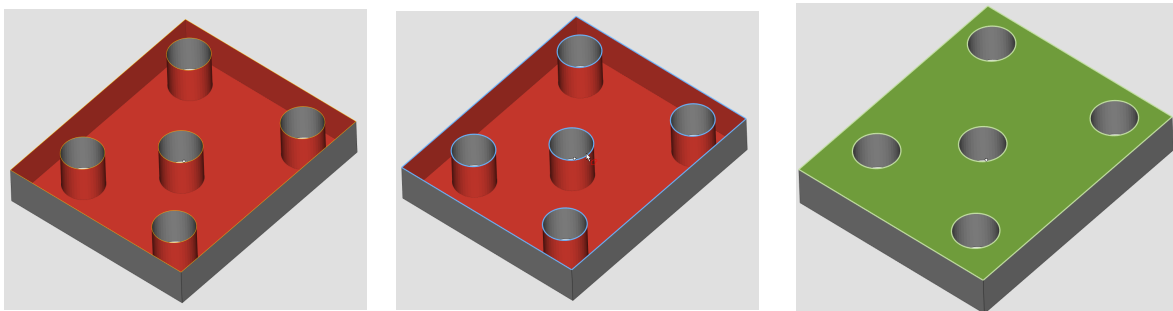
Multiple Contour - Double contour holes

To fill double-contoured holes, the combination of 'Fill hole mode' and 'Treat as one hole' functionality may be used. This command and parameter combination uses the selected contours to close the gap in between the 2 different contours quickly.



Multiple contour - Multi-contour holes with more than 2 contours

To fill a hole that has more than 2 contours, the combination of 'Select Holes' and 'Fill Selected' with the 'Treat as one hole' parameter set to True.

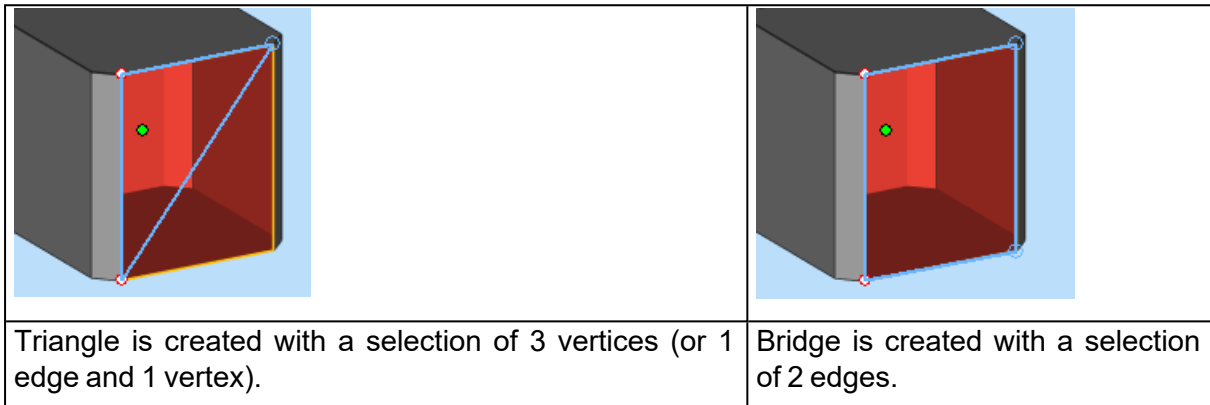


This is a hole with more than 2 contours.	Select the contours that are part of the same hole with the 'Select	The hole is filled automatically taking into account the cylindrical fea-
---	---	---

Holes' moused mode or by focusing on the holes in the holes list in the toolpage. Then, set the 'Treat features on the model as one hole' option on and click the 'Fill selected' command.

Create triangle

The create triangle function is used to help fixing complex holes. With a selection of point(s) or edge(s), triangle or bridge could be created.



Tips and tricks

1. The force of Planar Holes

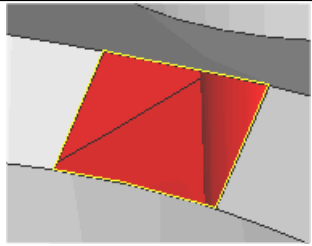
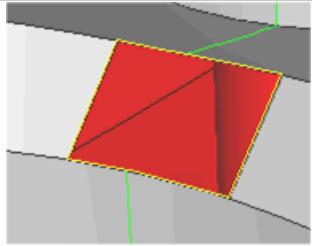
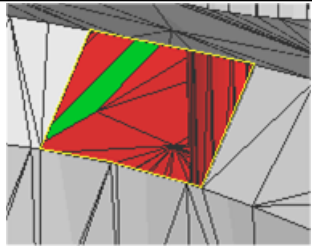
You'll be amazed how many non-planar holes the Planar Hole filling can handle. Because the Planar Hole filling is very fast, it does not harm to try to fill the hole with a Planar Hole filling first. If it's not ok, use "undo".

2. Auto Multiple Contour hole filling

When filling a single hole and Magics finds another contour in the inside of the bad contour, Magics supposes that the hole is in fact a multi contour hole. He will propose to use the found contour and perform a multi contour hole.



- How to recognize a hole?

	You can see the inside geometry through the hole. You will see the other side of the part
	When taking a section through the hole, there's no line drawn in the hole
	In triangle view, the triangles on the other side are not triangular. The marked triangle is not triangular



- Use the hole list to find holes.

Part name	Length (mm)	# Edges
▼ 1 Bad		
1	189,829	41
2	189,829	41
3	18,4781	8
4	18,4807	9
5	30,3295	52
6	30,3147	57
7	30,3294	53
8	33,1925	54
9	33,1718	54

In this list all the holes are shown. You can sort them by clicking on the column headers. You can select holes by clicking on them in the table, and zoom into them by double clicking.



5. Triangle page

Profile  

Filter sharp triangles

Max width mm

Min angle °

Overlapping triangles

Max distance (a) mm

Max angle (b) °

Triangles selection

Intersecting triangles

Identical triangles

Opposite normals

Same normals

Filter Sharp Triangles

Max width	Triangles thinner than this distance will be marked or removed, depending on your choice.	
Min angle	The thin triangle will only be selected when the angle it makes with its neighbors is bigger than the given angle. This is easy to filter only thin triangles of folds and leave thin triangles of curves untouched.	
Action	Collapse	The thin triangle will be removed and its neighbors will be connected to each other.
	Mark	The thin triangles will be marked.

Overlapping triangles

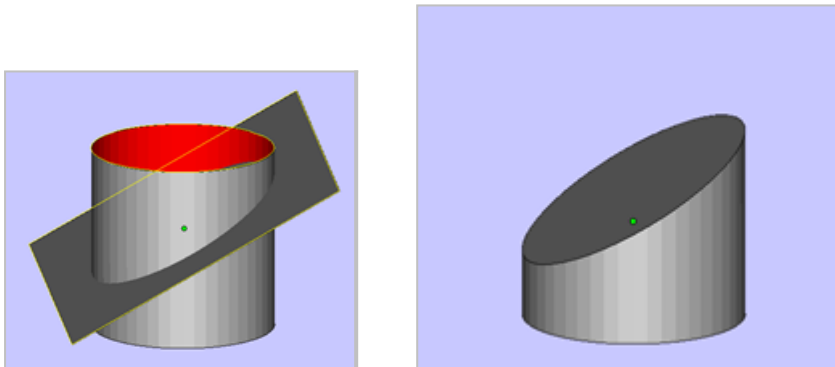
Overlaps (causing double surfaces) must be removed for some applications (E.g. Milling or FEA).

Max distance	The maximum distance between two triangles (surfaces) in order to be considered as double triangles (surfaces).
Max angle	If two triangles are inclined more than this angle, they are considered not to be double triangles.

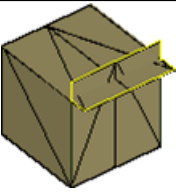
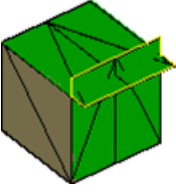
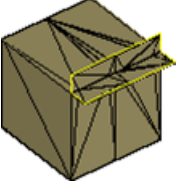
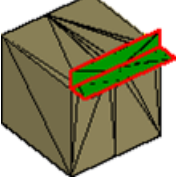
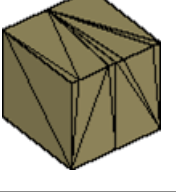
Triangles selection	Opposite normals	Triangles with their normal in the opposite direction.
	Same normals	Triangles with their normal in the same direction.
	All overlapping	Opposite normals and Same normals

Intersecting triangles

To solve the bad edges that are still left, more creative fixing is needed. Some possible errors you can still encounter are trimmed surfaces. In this case, a surface is sticking outside of the design and needs to be cut off (see picture below)

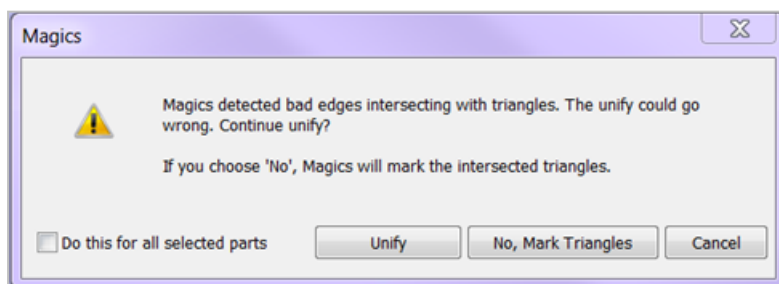


Unify	This will remove self-intersections and trim surfaces automatically. It's not advised to continue when a bad edge is intersecting a triangle because it will produce a corrupt part.
-------	--

Trim marked	This will trim the marked triangles	
		2 surfaces on this cube are not trimmed correctly. They are too long and intersecting. You can detect these intersections with the button detect intersecting.
		Mark the triangles that you want to trim
		Press Trim marked, the triangles are re-triangulated so there's an edge on the intersection line.
		Before After
		Mark the triangles you do not need anymore and delete them.
	The result.	

- Tips and Tricks

What does the message: "Magics detected bad edges intersecting with triangles. The Unify could go wrong. Continue Unify?" mean?




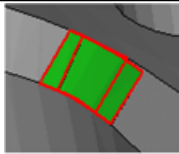


The Unify function won't have a good result if a bad edge is intersecting a triangle. Try to manually remove the bad edges that are intersecting the triangles. There are multiple ways doing this:

- Continue anyway the Unify

However it can damage your part, you can always try to continue. Sometimes the damage can be solved very easily with filling up the holes. Sometimes you'll need to undo and use an alternative way

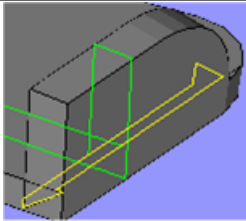
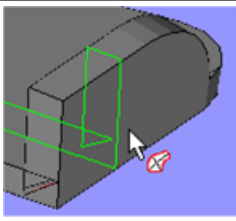
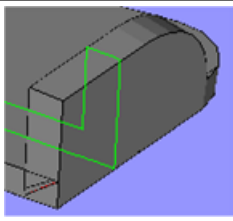
- By making a hole

Delete triangles so that the bad contour becomes a hole that you can fill. Trim marked can help here to re-triangulate the intersecting triangles so you can minimize the surface you need to delete.

	
This is the intersection	Mark all the triangles that are involved in the problem
	
Create a hole by deleting the marked triangles	Fill the hole with the hole filling functions

- By closing the bad contour

Fill the bad contour as if it was a hole. (A Freeform Hole fill will fill almost all bad contours). The internal geometry can be removed with Unify.

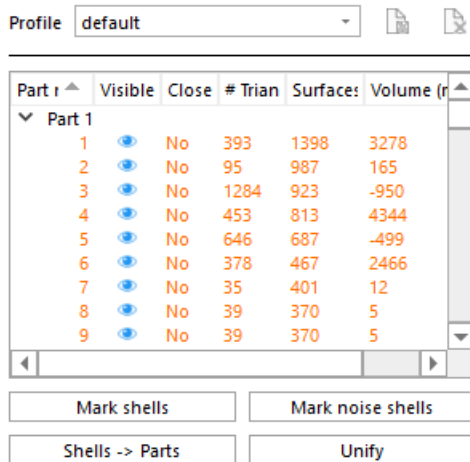
		
This is the trimming error.	With a fill hole the contour is filled with triangles	After a Unify, the internal geometry is gone.

Identical triangles

Opposite normals	The triangles have their normal in opposite directions. When you check this box, only the identical 'opposite' triangles are removed. You can choose to leave one of the two triangles or remove them both.
------------------	---

Same normals	The triangles have their normal in the same direction. When you check this box, only the identical triangles are removed. You can choose to leave one of the two triangles or remove them both.
--------------	---

6. Shell page



Introduction

It can happen that your part consists of multiple shells. Use this tool to manipulate the shells. Because this is not really an error, there's no automatic way to solve this problem.

In detail

– Shell list

In this list all the shells are shown. You can sort them by clicking on the column headers.

You can select shells by clicking on them in the table. To easily find the shells on your part, double click the items from the list. Shells can be quite small and therefore make it difficult to find them.

– Actions

Mark noise shells	The noise shells will be selected.
Unify	This will remove self-intersections and trim surfaces automatically. It's not advised to continue when a bad edge is intersecting a triangle because it will produce a corrupt part.
Shells -> Parts	This will make a part of each shell.

7. Point page

Manual Point Fixing

Create point by snapping


Create point by coordinate

X: mm

Y: mm

Z: mm

Show Points



User points

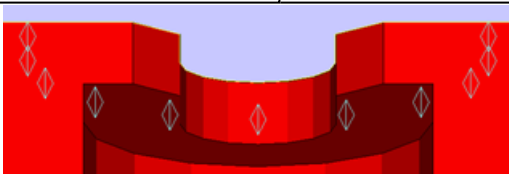


The user creates user points by using the Create button. The coordinates you need to create those points can be retrieved from the info box in the measuring Toolpage (these values are copy-pasteable via the right mouse button)

Free points



These points are created when all the triangles, which are using this point, are deleted. When all these connected triangles are deleted, Magics will keep the points so you can use them again later.

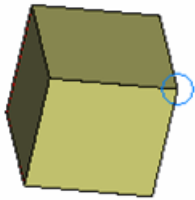
Manual Point Fixing	Show Points	Check this to show the currently present points. You can only see and use the points in Magics when they are visible
		
	Create point by snapping	Add new points manually on a surface.
	Create point by coordinate	Use X – Y – Z fields to enter the coordinates of the newly created point.
	Add point	A new point will be created using the given coordinates.
	Move point	Move selected part point.
	Delete point	Select points manually to remove them.
Delete all points	All present points are removed.	



To snap to these points the following selection has to be made in the settings overview of Magics: “Points with Triangles” & “Points without Triangles”. This can be activated via Options – Settings – General – Snapping.

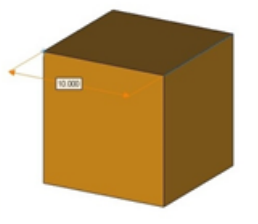
16.4. Measurements pages

Magics recognizes different features: a point, a line, a plane, a circle, a cylinder and a sphere. You select a feature by moving the pointer of the mouse. Magics will snap (the feature will be marked when you move over it) to all features of the type you selected. For example points - see figure - are marked with a round. In the Settings you can choose which features Magics should recognize. You may for example determine that you only want to snap to points that are in a section, or on a wireframe.

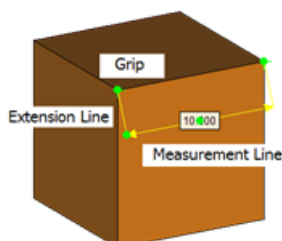


By clicking the left mouse button, you will select the marked feature. When all features of the measurement are selected, they will be marked if the draw feature in the measurement part of the settings is checked.

For example a point is indicated with a cross (see figure), a line with a line (see figure), and a sphere with 3 circumpolar circles. When the draw feature in the measurement part of the settings window is not on, the feature may be selected but is not indicated in a special way.



To select a measurement, you click on the icon ‘select parts’ in the main toolbar () or in the measurement Toolpage. The mouse pointer gets a green round to show you are in the selection mode. Click on the measurement value to select the measurement. When the measurement is selected some grips will appear. In the figure, the measurement with value 10 is selected. There is a grip in the middle of the measurement line and on one of the intersections of the extension line with the measurement line.



It is possible to adapt a measurement by dragging one of the feature indicators to a same feature positioned elsewhere on the part. This way you get a new measurement. To do so you first have to select a measurement.



In the Settings window, you can indicate how you would like to display the measurement (with or without arrows and extension lines, the size of the grips....).

If you are not satisfied with the position of the measurement value on the screen, you can change this position. First, select a measurement with the mouse. If you select the grips at the cross points of the extension lines, you can turn the measurement indication line around the axis that connects the selected features. When you select the grip in the middle of the measurement line, you can make the extension lines longer or shorter.

Selected measurements can be deleted with the Delete button on the keyboard.

To delete one or more measurements (but not all), one has to follow these two steps:

- Select the dimensions you want to delete (keep the shift button down to select several dimensions)
- Hit the Delete button on the keyboard.

All measurements can be deleted at once by clicking the Clear Measurements button in the respective Toolpage.



Hide measurements	When checked, the made measurements are hidden. Uncheck to see the measurements.
Clear Measurements	Deletes all measurements.
Snap Settings	Brings you to the Settings window. You can indicate how you would like to display the measurement.

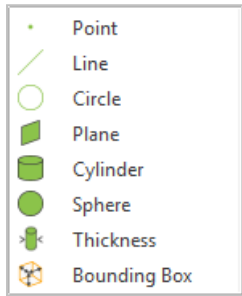
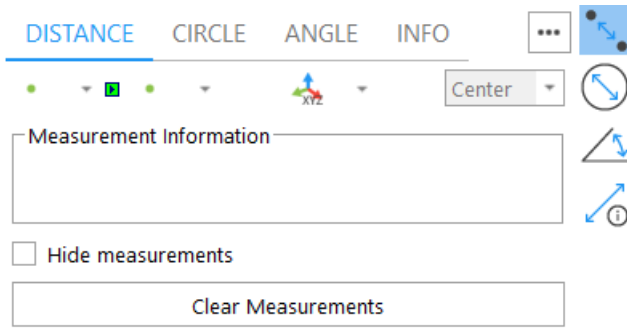
1. Measurements on BREP parts

Currently the options for measurements on BREP parts are allowed for distance, circle and angle measurement with only a subset of entity types compared to measurements on mesh parts. The supported entities are:

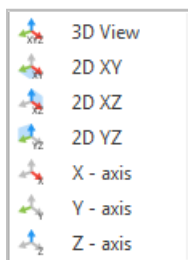
- i. Point(s)
- ii. Line(s)
- iii. Circle(s)
- iv. Plane(s) - (Planar Face)
- v. Thickness
- vi. Bounding box

2. Distance page

The Distance Toolpage allows you to measure the distance between several features.



Feature	Measurement Description
Point	The length from the second feature to the point.
Line	The length of the perpendicular through the second feature on the line.
Circle	The length of the perpendicular through the second feature on the circle.
Plane	The length of the perpendicular through the second feature on the plane. If you select a plane, Magics will snap to a triangle in that plane.
Cylinder	The length of the perpendicular through the second feature on the middle line of the cylinder.
Sphere	The length from the second feature to the center of the sphere.
Thickness	Measuring thickness is a special way of measuring: you do not have to select a second entity. When clicking on the part, Magics will measure the distance perpendicular on the triangles surface through the inside till it reaches another triangle (=the other side of the part). When measuring thickness, the snapping on the STL-surface is switched on automatically.
Bounding box	Measures the bounding box of the indicated part.



Snapping restrictions

3D View	The measurement is allowed in 3D
2D XY	The measurement is restricted to the 2D XY plane.
2D XZ	The measurement is restricted to the 2D XZ plane.
2D YZ	The measurement is restricted to the 2D YZ plane.
X-axis	The measurement is restricted to the 1D X axis.
Y-axis	The measurement is restricted to the 1D Y axis.
Z-axis	The measurement is restricted to the 1D Z axis.



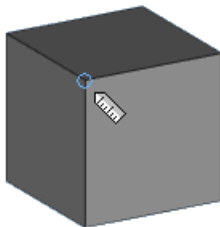
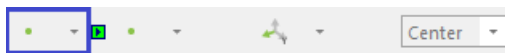
Remark: The measurement is dynamic: When you have selected the first feature, Magics will look for the second feature as you move your mouse over the part. The measurement value will change as you snap to features at different positions.

Advised way of working

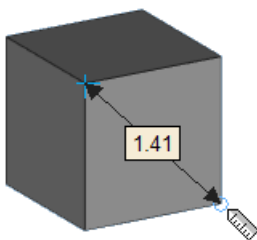
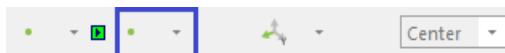
- Choose the restriction of the measurement. This is possible in 1, 2 or 3 dimensions.



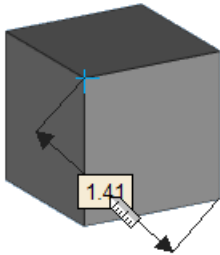
- Choose the first feature in the menu (by clicking on it), snap the feature on the part and click on it to select it.



- Choose the second feature in the second menu (by clicking on it) and snap to the feature on the part and then click on it to select it.



- Choose where you want to display the measurement on the screen, by dragging the extension lines of the measurement.



- When you click a last time, the measurement will be fixed.

3. Circle page

Radius

Measurement Information

Radius: 0.7071 mm
 Diameter: 1.4142 mm
 Center: 0.0000 0.0000 0.5000

Hide measurements

Clear Measurements

Snap Settings

Circle
 3 Points
 Sphere

Feature	Measurement description
Circle	Select an arc. The radius (or diameter) of the arc will be displayed.
3-Points	Select three points. The radius (or diameters) of the circle defined by these three points will be displayed. Be aware that it is possible to indicate three random points. This can result in a non-existing arc! It is advised to use the radius of an arc measure function when possible. Use this function only when the arc is not recognized by Magics as a feature.
Sphere	Select a sphere. The radius (or diameter) of the sphere will be displayed.
Radius or Diameter	You can choose whether you want to measure the radius or the diameter.

4. Angle page

Measurement Information

Angle: 135.00 deg (135°0'0")

Corner -0.5000 -0.5000 -0.5000

Hide measurements

Clear Measurements Snap Settings

- 3 Points
- Line
- Plane

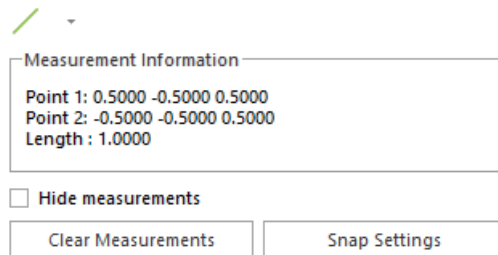
Feature	Measurement Description
3 points	<p>Select three points. Two imaginary lines will be drawn between those points: between points 1 and 2, and between points 2 and 3. The resulting angle is the angle between those two lines defined by the three points.</p> <div style="border: 1px solid #00aaff; padding: 10px; margin-top: 10px;"> <p> Remark: It is advised to measure the angle between two line features when possible. Only use this function when the lines can't be recognized by Magics as features.</p> </div>
Line	Select the two intersecting lines. Both lines will be highlighted, and the intersection point will be drawn. The resulting angle is the angle between those two lines.
Plane	Select two planes.
Defaults	You can also measure the angle between a line or plane and an axis or plane of the coordinate center.

- 3D View
- 2D XY
- 2D XZ
- 2D YZ
- X - axis
- Y - axis
- Z - axis

Snapping restrictions	
3D View	The measurement is allowed in 3D
2D XY	The measurement is restricted to the 2D XY plane.
2D XZ	The measurement is restricted to the 2D XZ plane.

Snapping restrictions	
2D YZ	The measurement is restricted to the 2D YZ plane.
X-axis	The measurement is restricted to the 1D X axis.
Y-axis	The measurement is restricted to the 1D Y axis.
Z-axis	The measurement is restricted to the 1D Z axis.

5. Info page



When you snap to a feature, or you select it, some coordinate information of the features is given. This information allows you to draw exactly the same feature on the same point in space.

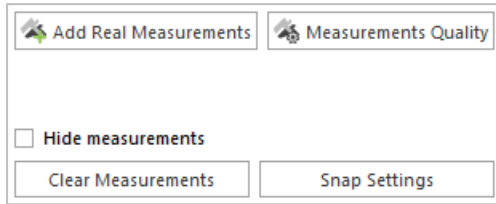
Point	The X, Y and Z coordinate of the point will be displayed.
Line	The X, Y and Z coordinate of the beginning and end points will be displayed, together with the length of the line.
Circle	The X, Y and Z coordinate of the center of the circle is given, together with its radius.
Triangle	The X, Y, Z coordinates of the corner points and the direction coefficient of the normal will be displayed.
Cylinder	The X, Y and Z coordinate of the middle point of the bounding discs is given, together with the radius.
Sphere	The X, Y and Z coordinate of the middle point of the sphere is given, together with its radius.
Section	The Length and surface of the selected contour is given.

Select	Measurements can be selected now. A measurement is selected when on the label green dots are visible. When active the measurement or label position can be changed.
Hide Measurements	When checked, the made measurements are hidden. Uncheck to see the measurements.
Clear Measurements	Deletes all measurements.
Snap Settings	Brings you to the Settings window. You can indicate how you would like to display the measurement.



6. Final Part page

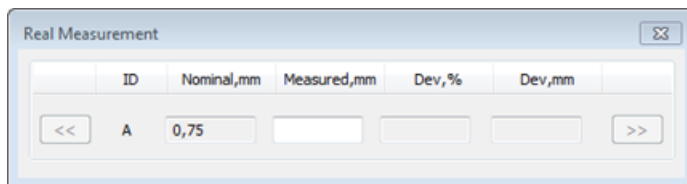
Within the final part Toolpage real life measurements of the part, can be added next to the measurements made in Magics.



Add real measurements	Add measurements made in real life to the existing ones in Magics to compare them.
Measurements quality	Brings you to the measurements quality settings where you can adjust the used tolerance.

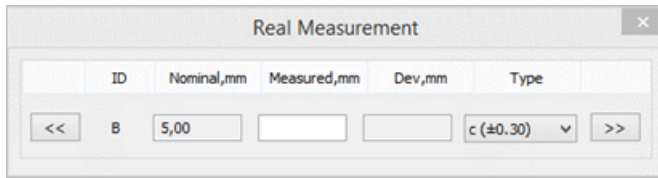
Add real measurement

1. Basic measurement quality




ID	Every measurement is indicated by a unique ID
Nominal, mm	Displays the value of the active measurement, measured in Magics
Measured, mm	Enter the real life measurement which matches the one in Magics
Dev, %	Shows the deviation in percentage between the measurement made in Magics and the one in real life.
Dev, mm	Shows the deviation in millimeters between the measurement made in Magics and the one in real life.
Browsing buttons	Navigate easily between the different measurements

2. Advanced measurement quality



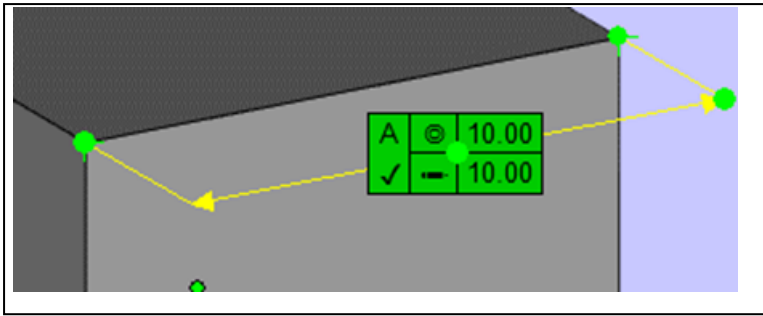
ID	Every measurement is indicated by a unique ID
Nominal, mm	Displays the value of the active measurement, measured in Magics
Measured, mm	Enter the real life measurement which matches the one in Magics
Dev, mm	Shows the deviation in millimeters between the measurement made in Magics and the one in real life.
Type	Choose the type of deviation. This is based on the selected measurement profile within the settings.
Browsing buttons	Navigate easily between the different measurements

 Remark: Via 'Settings' > 'General' > 'Measurements quality' it can be defined which measurement quality is used.

Analyzing measurements (Showing basic measurement quality)

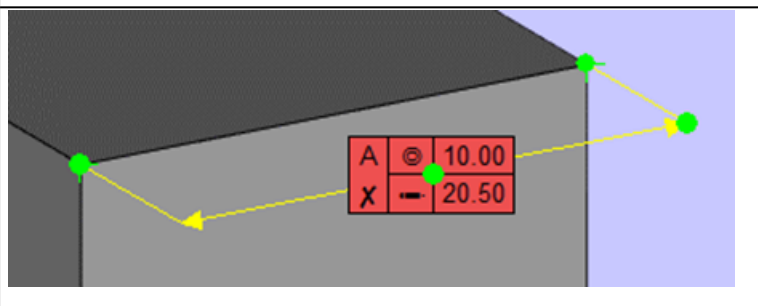
There are 3 possible outcomes after entering the real life measurements to Magics. Depending on the defined tolerance in the settings, the measurements will get one of the following color codes. (Default are 'Relative tolerance (Dev,%) = 0,3%'; 'Absolute tolerance (Dev, mm) = 0,2mm)

Green = Dev, % < 0.30% and Dev, mm < 0.2mm



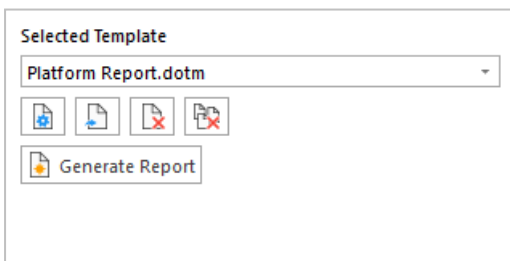
Red = Dev, % > 0.30% and Dev, mm > 0.2mm

ID	Nominal,mm	Measured,mm	Dev, %	Dev,mm
A	0,75	100,00	13233,33	99,25



7. Report page

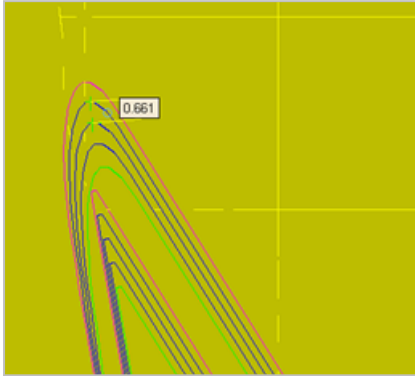
The Report Toolpage allows you to generate a measuring report.



Selected Template	The template you've selected.
Create report template	Create a customized template report. A 'start here' template is available to start from.
Load report template	Load an available template.
Remove selected template from the list	Erases the selected template
Clear template list	The entire template list will be erased.
Generate report	Create the actual report

8. Measurements on slices

Once parts are viewed as slices, measurements can also be made on them. All available measurement tools can be used to measure.

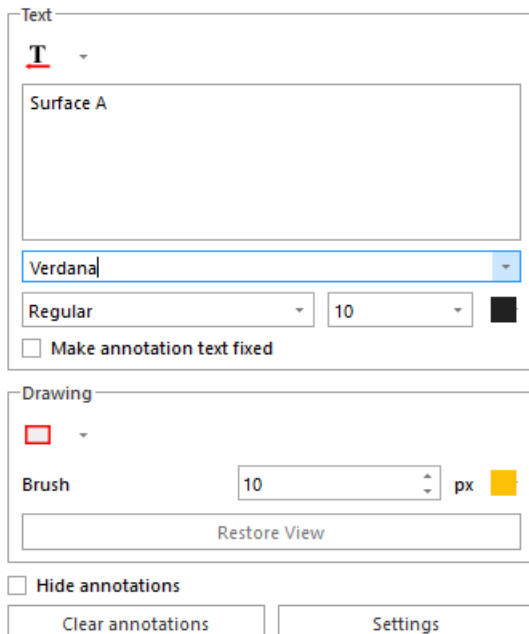


16.5. Annotation Pages


Adding extra information to the *.magics project has never been so easy. Now, it is possible to add texts and drawings; files can be attached and textures can be printed on parts. For keeping a well-organized overview of your project, working with annotation scenes is advised.

1. Annotations page

The Annotations Toolpage makes it easy to add a text to the Magics project or to draw in the active scene.



– Text section

Text dropdown	Choose between adding a simple textbox or a referring textbox with an arrow.
Annotation content	Once the text annotation is created on the scene, add the desired annotation content to this text box. When selecting one existing text annotation on the scene, use this text box to modify its content.
Font parameters	Change the font, font style, font size and text color for the selected text annotation.
Make annotation text fixed	<p>By default, a Text and arrow annotation is fixed to the part. When rotating, panning or zooming the scene, the positions of the text annotation will move accordingly. Select this option to block the position of the text annotation on the scene.</p> <div style="border: 1px solid #00aaff; border-radius: 10px; padding: 10px; margin-top: 10px;">  Note: this option is available only for Text and arrow annotation. </div>

– Drawing section

Drawing dropdown	Choose between drawing a rectangle, an ellipse or to make a drawing by freehand.
Brush	Change the thickness or the color of the brush. These options can be edited before drawing a shape or to modify the properties of the selected drawing on the scene.
Restore View	A drawing is not fixed to the part. When rotating, panning or zooming the scene, the drawing will not move simultaneously. You can restore a certain view point by selecting the relevant drawing and clicking Restore View button.

– General options

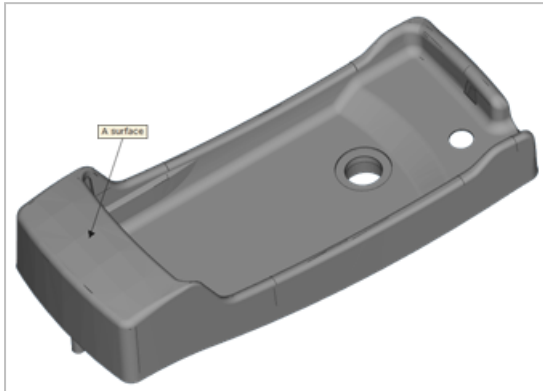
Hide annotations	When checked, the created annotations are hidden. Uncheck to see the annotations.
Clear annotations	All created texts and drawings annotations will be removed.
Settings	The general settings of the annotations can be changed.

Advised way of working

If you want to create an annotation fixed to a certain point, choose Text and Arrow. Snap a point on the part and click on it to select it. A preview of the annotation will appear. Start typing

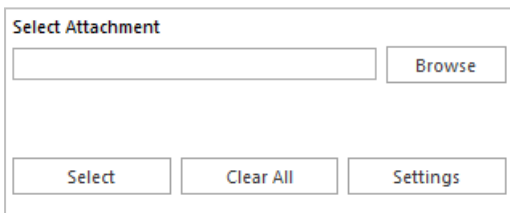


the desired content for the annotation. If you need to change the position of the annotation, drag the preview to the desired location and release the left mouse button.



2. Attachments page

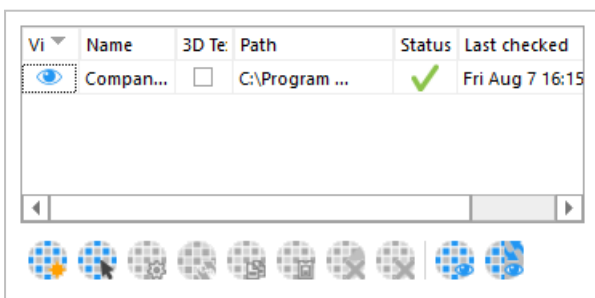
The Attachment Toolpage makes it easy to attach a file to the Magics project.



Browse	With the Browse button you can browse to the file that has to be attached.
Select	By clicking the Select button an annotation can be selected.
Clear all	All texts, drawings and attachments will be removed.
Settings	General settings can be changed.

3. Textures page

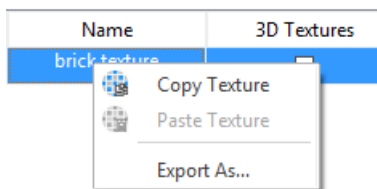
Texturing is the printing of an image on selected triangles of a part. You can read, create and save textured parts as ZPR, VRML or Magics files.



Visibility	Click the “eye” icon to show/hide the texture
Name	Name of the textured surface. By default this is the name of the file used for the texturing, but it’s possible to change this name.
3D Textures	Enable this checkbox if the texture should be converted to a 3D texture while slicing
Path	The location of the used texture. No path is shown when the texture isn’t applied through the Magics Texturing feature.
Status	Reflects whether the texture was found on the specified path and whether it was edited since it was reloaded in Magics. <u>Green checkmark</u> : texture found and up to date <u>Pencil</u> : Texture found, not up to date <u>Red cross</u> : texture not found
Last checked	Date & time of the latest attempt of loading the texture.

New Texture	Add a new texture
Select Texture	A texture can be selected on the part.
Edit Texture	A texture can be edited
Update textures	Updates the selected texture(s).
Copy Texture	Copy a texture
Paste Texture	Paste a copied texture
Delete Texture	A selected texture can be removed.
Delete Texture from triangles	A texture can be deleted from selected triangles.
Toggle Texture Visibility	Toggle the visibility of all textures
Invert Textures Visibility	Invert the visibility of textures.

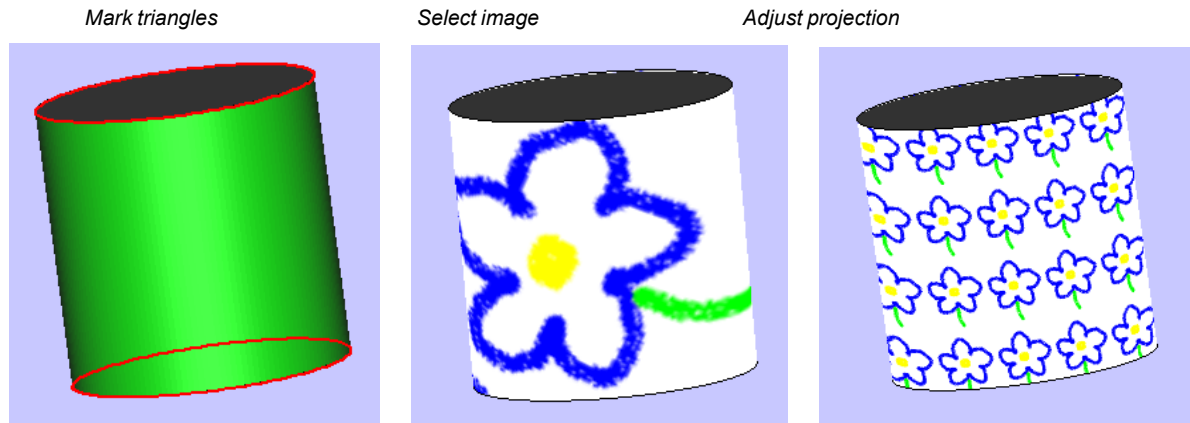
Context menu



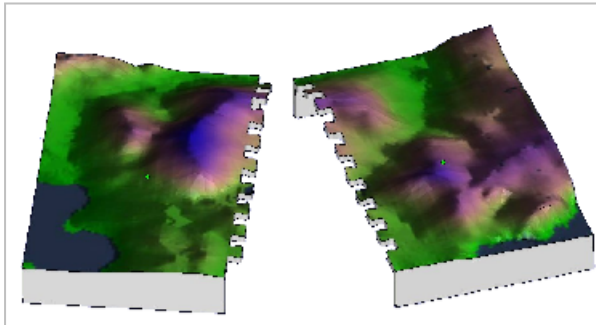
Feature	Description
Copy	Copy (CTRL+C) the selected part texture
Paste	Paste (CTRL+V) the previously copied texture to (a) selected triangle(s).
Export As...	Save (CTRL+S) the selected texture as an image file.

Advised way of working

First, you have to mark the triangles where you want to put the texture on. Then you create a new Texture in the texture Toolpage. Follow each step of the Toolpage to adjust the projection. Each change in the texturing dialog will be visualized in real-time.



Remark: After texturing, a part can be edited and/or fixed without loss of the texture. The figure below shows a textured part that is cut with a teeth cut.

**4. Beam Lattice page**

Observe beam lattice properties in the Beam Lattice toolpage.

Part/Lattice name	Name of the part with beam lattice
# Beams	Number of beams of selected part with lattice
Min. Thickness	Min diameter of a beam
Max. Thickness	Max diameter of a beam
Accuracy	Accuracy for beam to mesh conversion

If beam lattice thickness =0, such lattice isn't suitable for printing. It's advised to define thickness with **Edit Beam Lattice Properties** for slice base printing or during conversion with **Convert Beam Lattice to Mesh**

16.6. Slices Pages

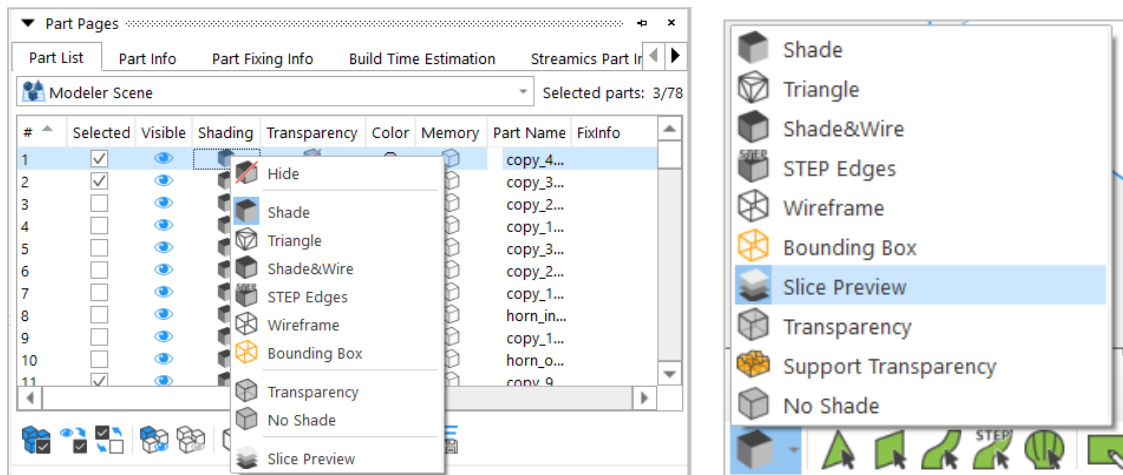
The Slice Toolpage allows the user to visualize the slices of the part. The slices can be visualized in 2D and 3D view. Measurements can also be taken on the slices.

Activate slices

The Slice Toolpage comes available when slices are present in the viewport. The view mode of each individual part can be controlled through the Part List.

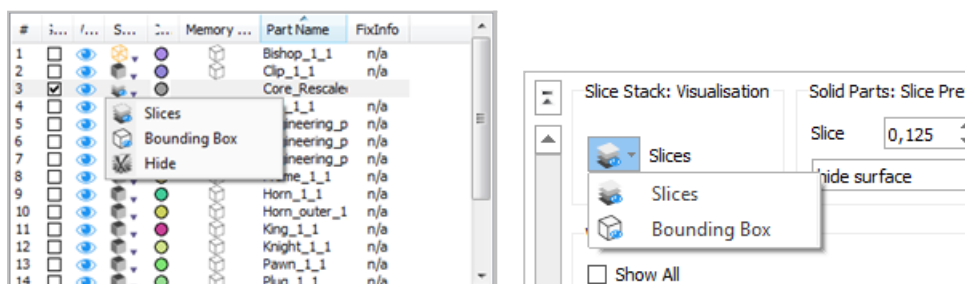
– STL parts (blue icons)

To activate slice preview on STL parts, turn their visual style to 'Slice Preview' in the part list. A shortcut to change the visual style of all STL parts at once can be found in the Shade modes button from toolbar.

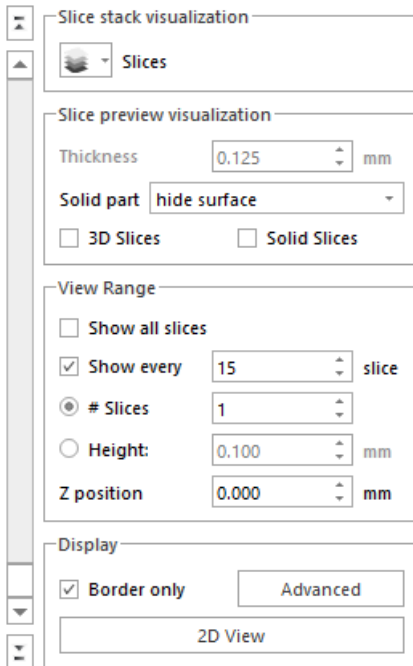


– Imported Slice Stacks (green icons)

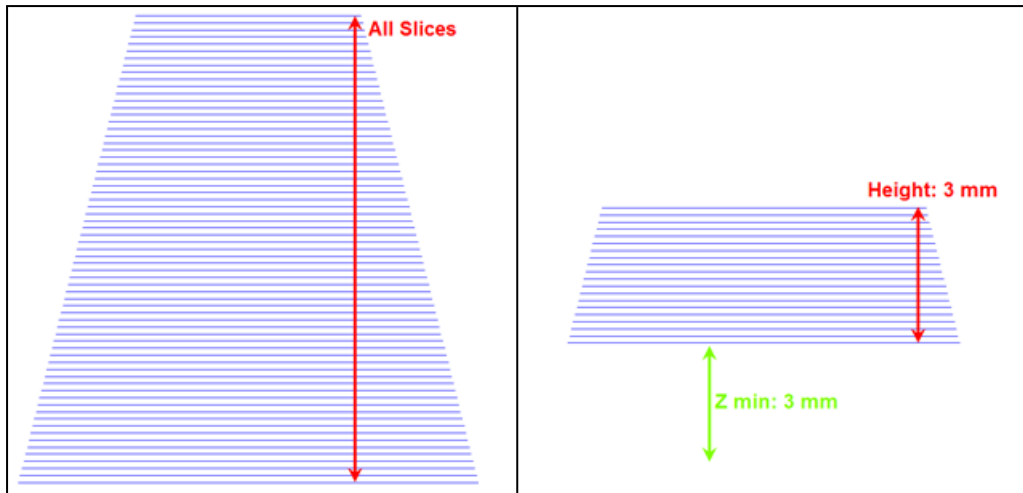
To show the slices inside an imported slice stack, turn their visual style to 'Slices' in the part list. Slice Stack is activated when importing a slice file into Magics. A shortcut to change the visual style of all Slice Stacks at once can be found in the Slices page.



Visualize slices



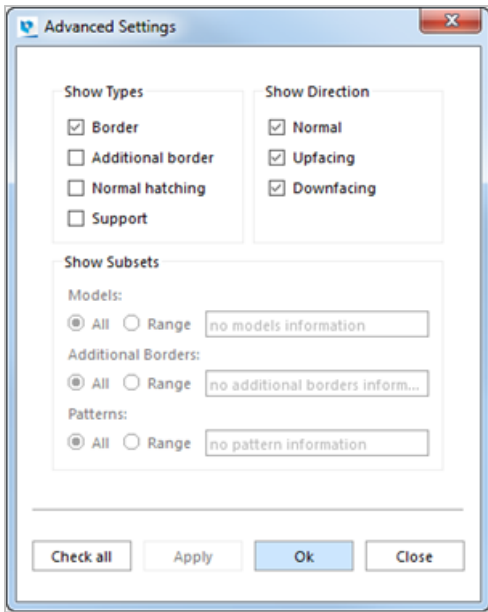
Slice Stack: Visualization (affects only Slice Stacks)		
	Slices	Turns all Slice Stacks in the drawing to view mode 'Slices'
	Bounding Box	Turns all Slice Stacks in the drawing to view mode 'Bounding Box'
Solid Parts: Slice preview (affects only Solid parts = STL parts)		
	Slice	<i>On platform scene</i> Shows the current layer thickness specified in the machine file
	Show solid part	The remainder of the solid part is still shown below as solid
View Range		
	Show All	Visualize all slices of the imported part(s)
	Slices	Set the number of slices in the view range
	Height	Set the height of the view range
	Z min	Set the bottom of the view range. This value can also be manipulated with the slider on the side



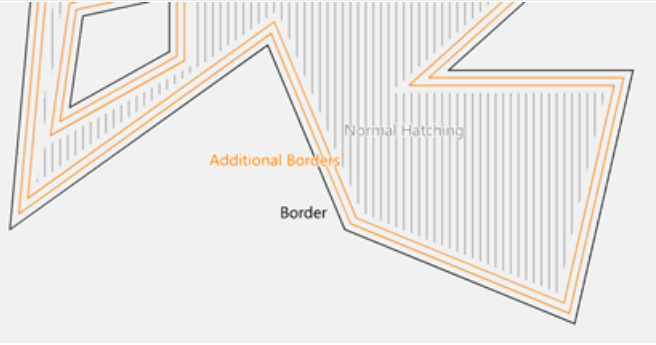
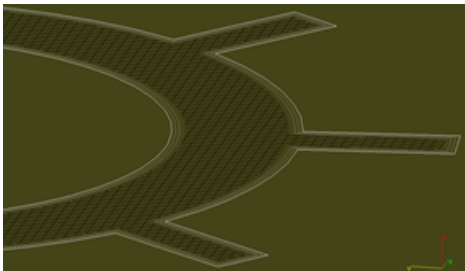
Display		
	Solid Slices	Show all the slices filled.
	3D Slices	Show the slices thick and filled. If 3D Slices is turned on, all other information in the slice stack is hidden.
	Borders only	Turned on: only the border (=contour) is shown. Turned off: all available information is shown. This can be controlled more in detail through the Advanced View Settings
	2D/ 3D View	2D: Locks the viewport to TOP view 3D: Releases the viewport and enables orbiting

Advanced View Settings

A slice stack can contain a range of information. If that information is present in the source file, this dialog allows more control over the set of information that is shown.

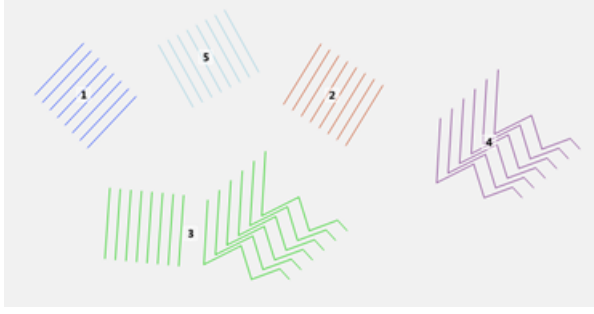
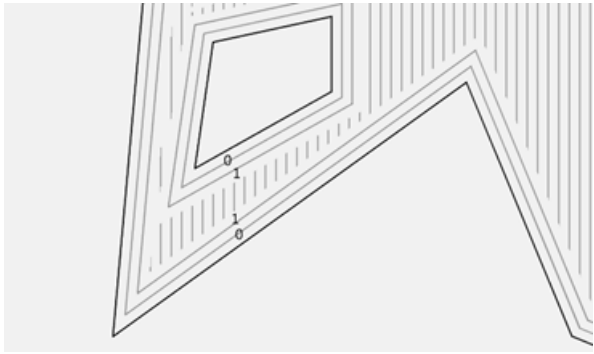
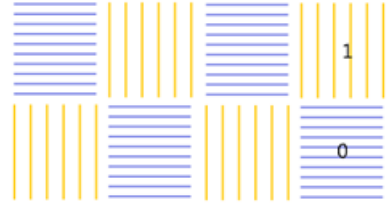


Show Types

	
Border	Turn border on/off
Additional Borders	Turn additional borders on/off If the border is offset a number of times, it's called Additional Borders.
Normal Hatching	Turn normal hatching on/off Normal Hatching is the collection of vectors used to fill the layer during the scan process. <i>Hatching example</i> 
Support	Turn support on/off Sliced support

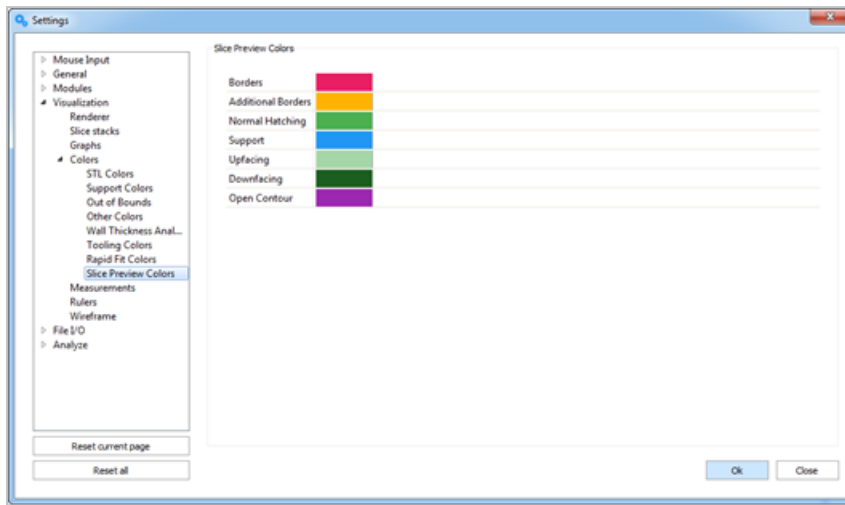
Show Direction	
Normal	Turn normal vectors on/off All slice information that is not upfacing nor downfacing
Upfacing	Turn upfacing vectors on/off The result of slicing a horizontal region without geometry on top of it.
Downfacing	Turn downfacing vectors on/off The result of slicing a horizontal region without geometry below it.

Show Subsets	
Models	If the slice stack kept information about the original models (parts) before slicing, they are indexed from 0 to the amount of models.

	
	<p>All Show all the models</p>
	<p>Range A range can be entered as:</p> <ul style="list-style-type: none"> - a list: 2,5,6,9 - a range: 3-8 - combination of both: 2,6-9 <p>and will only show the models with that index</p>
<p>Additional Borders</p>	<p>The additional borders are numbered starting from 0 for the outer most additional border, inwards.</p> 
	<p>All Show all the additional borders</p>
	<p>Range Show only a subset of the additional borders</p>
<p>Patterns</p>	<p>When certain hatches are collected into groups, they are called patterns.</p> 
	<p>All Show all the patterns</p>
	<p>Range Show only a subset of the patterns</p>

Color Settings

All color settings regarding the slice visualization can be adapted here.



Remark: Open contours overwrites other colors.

A graphic consisting of two overlapping triangles. The top triangle is light blue and points downwards. The bottom triangle is a darker blue and points upwards. They meet at a horizontal line.

materialise
innovators you can count on

Part III: Extra Information

Chapter 1. Windows Configuration Info

In this chapter, topics related configuring Windows systems to enable or disable some additional features for Magics RP. You may use this as a guide to configure your systems to enable certain work flow or features on the application.

1.1. Enabling long path support on Windows

Windows by default only supports file paths with a maximum of 256 characters. However, you may have a folder names or file names that causes the overall length of the file path to exceed 256 characters. When such files are attempted to be loaded into the application, the application will fail to load the file without any warning currently.

As such you are advised to enable the support for long paths on your system to ensure such random failure due to file path lengths does not block you from loading files into the application. To learn more about long paths in Windows system, you are advised to read this [article by Microsoft](#).



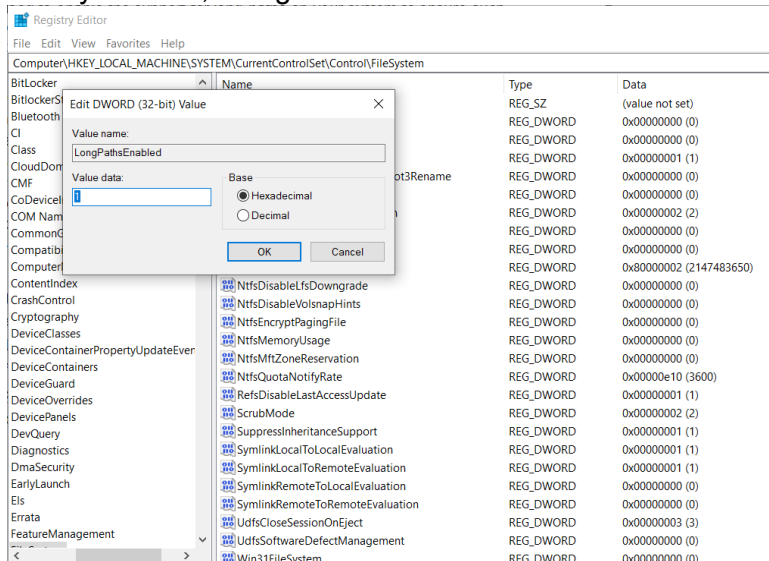
Administrator privileges are required: If you do not have the administrator privileges over your system then you would not able to make the necessary system changed. Please contact your IT team.



Note: This guide is only applicable if you are on Windows 10, version 1607 or later. If you are not, you would need to update your system before proceeding.

1. Modify your Windows Registry to enable Long paths
 - i. Run Registry Editor as an administrator.
 - ii. Navigate to the folder,

"HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Control\FileSystem"
 - iii. Modify the field, "LongPathsEnabled" and set the value to 1.



- iv. Close the Registry Editor.

2. Reboot your computer.

If all goes well you should now be able to load files that have full path length exceeding 256 characters into Magics successfully.



Note: When your windows file path exceeds 256 characters, Windows operation system will create a shorter reference of the path as the filepath which is then correctly consumed by application if long path support are enabled. However, any additional features that relies on the original file path text will not work as intended.

For example, "Add file name to part name option" when importing BREP Parts will only be able to keep the shorterned reference name generated by Windows and not the original file name.

Currently long path support is only available for a limited number of file formats that are native to the Magics application as shown below.

- i. Magics Project file (.magics)
- ii. Materialise AM Exchange file (.matamx)
- iii. STL file (.stl)
- iv. STL Zip file (.mgx)
- v. Materialise 3-matic Project file (.mxx)
- vi. MDCK file (.mdck)
- vii. Parasolid kernel file (.x_t & .x_b)*

For all other file formats support will be extended as per need, if you are interested in other file format getting similar support feel free to reach or to our customer support.

**Parasolid kernel file supports long path only when loaded into the BREP Part scene*

1.2. Setting the default application to open specific file formats

During the installation of a new Magics application, you will be prompted to associate a handful of 3D file formats to the installed application version.

- See ***Installation, page 16***

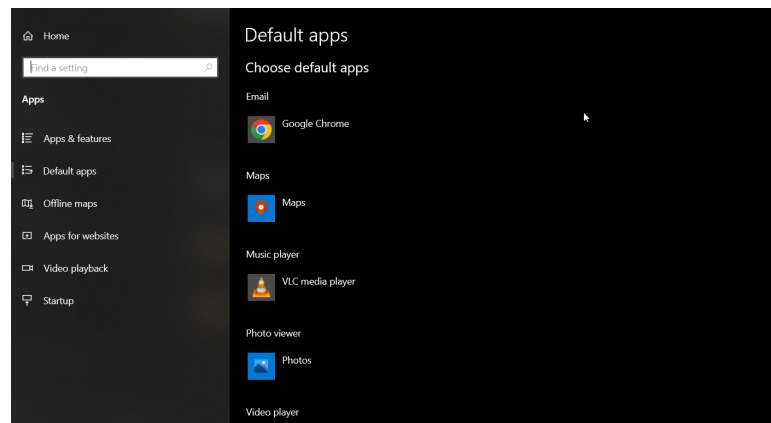
Installation, page 16

- i. Through Windows Settings
- ii. Through File Properties page of respective files

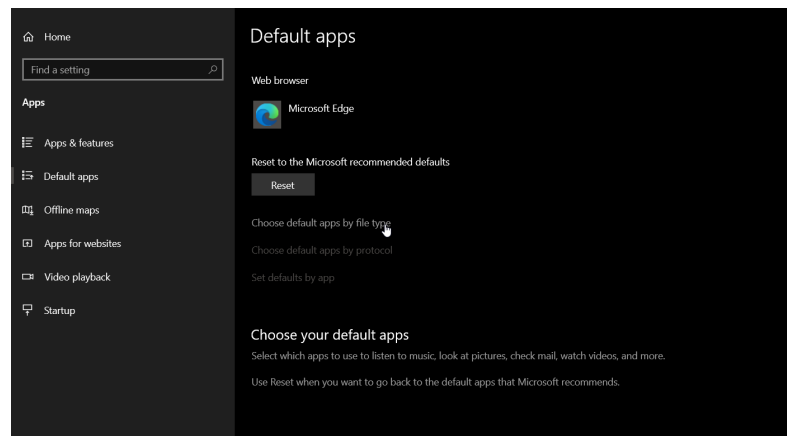
Both of these method should be compatible with Windows versions of Windows 10 build 1907 and newer. You are advised to first attempt method 1 before attempting method 2.

1. Windows Settings

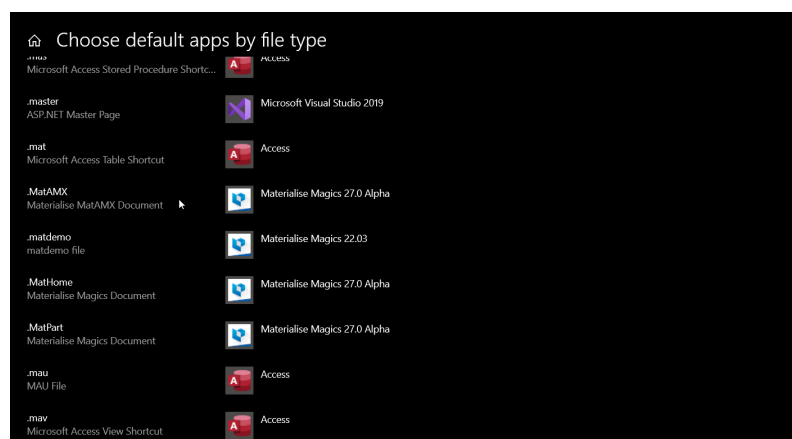
- i. Go to Windows Settings > System > Apps & Features > Default apps page.



- ii. Locate and click the Choose default apps by file type button

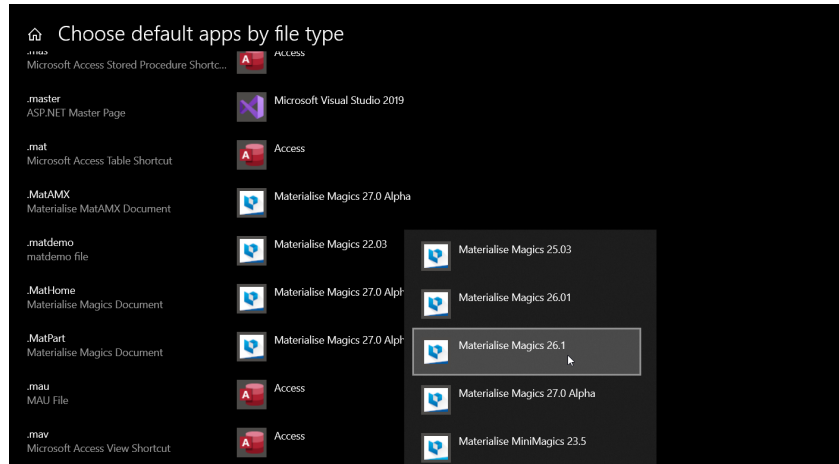


- iii. Select the file format that you are interested in from the list and click on the app button.



- iv. When the "Choose An app" pop-up is shown all the default apps installed on your PC that can be set for the file format will be shown. Click on the desired one to set it as the

default application.



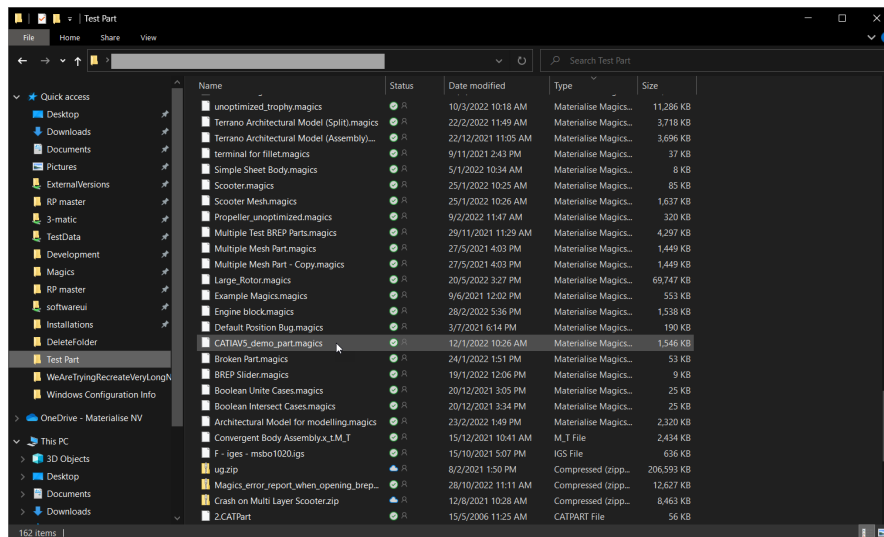
If everything went as per the steps, now the file format should have been set correctly, and double clicking the file should open in the application set as the default app.



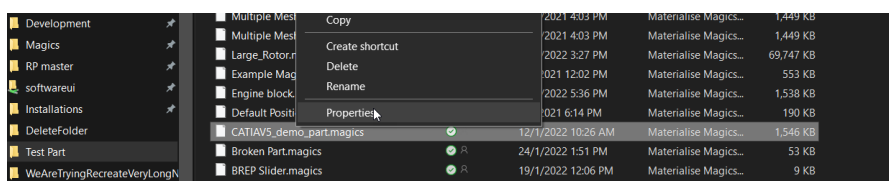
If the desired version of the application is not found on the "Choose an app" pop up window then proceed to method 2 to set your default application.

2. File properties page

- i. Locate a file with the type of file format you would like to set default app to in your File explorer

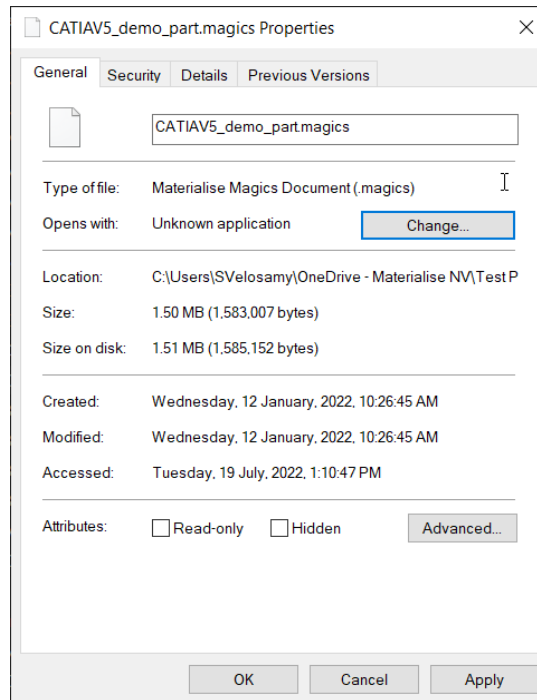


- ii. Right mouse button (RMB) click on the file, locate and click on the Properties button on the context menu

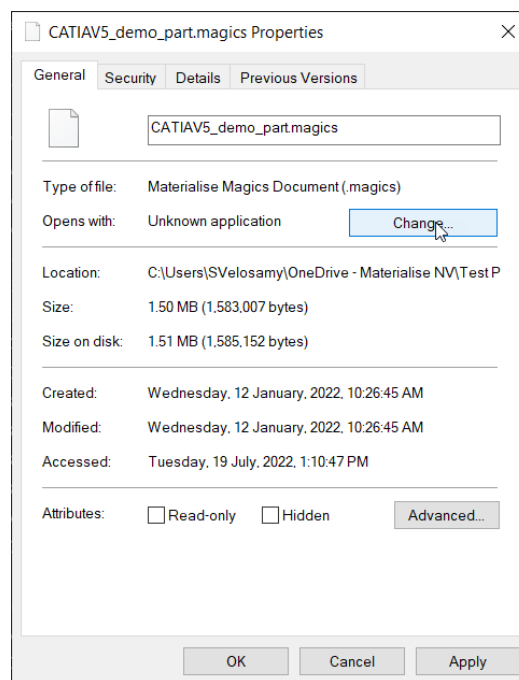


- iii. In the "General" tab of the File properties window, locate the "Opens with"

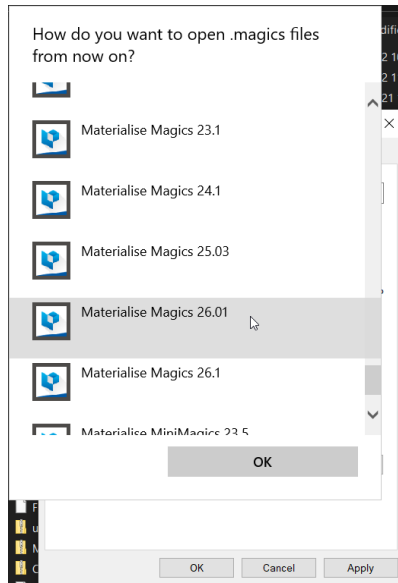
column




iv. Click the "Change" button and the file application list pop up will be shown



v. Select the appropriate application from the list of applications on the pop-up and click "Ok"

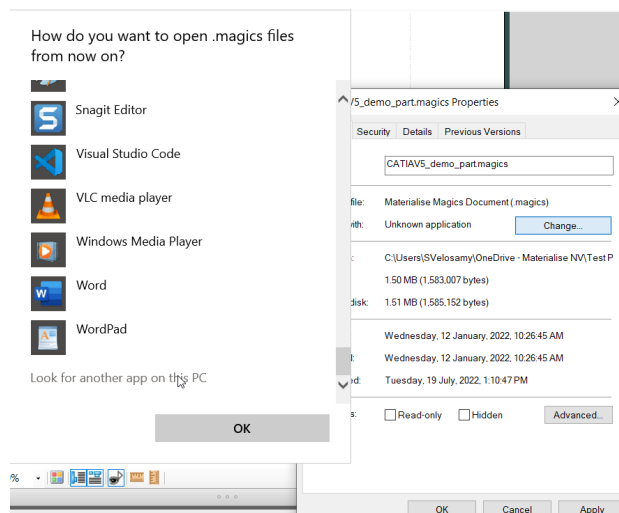


vi. On the file properties page, click "Apply".

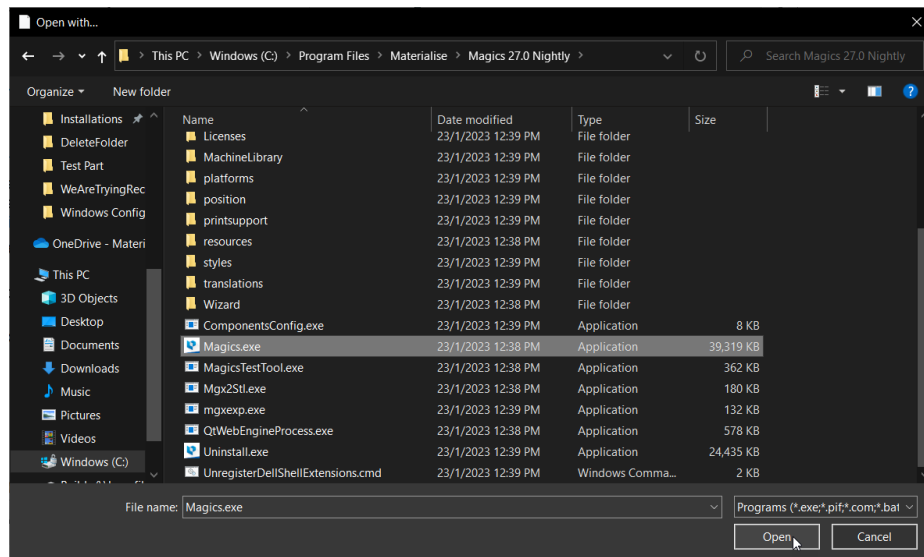
 If in step V, you are not able to find the appropriate application from the displayed list then see below.

When an appropriate application is not listed automatically by Windows then you would have to manually define the path the executable of the installed application that you would like to use. Follow this steps from step v onwards from the previous steps.

v. Select the "Look for another app on this pc" and the Windows File explorer windows will be shown



- vi. Navigate to the installed location of the application which is usually in the Program Files folder of your C:// drive
- vii. Locate the application executable with an ".exe" extension, select it and click "Open"



viii. On the file properties page, click "Apply"



If none of these methods work for you, please contact your sales application engineer for assistance.



Chapter 2. Contact Info

Europe (Headquarters)

Technologielaan 15

3001 Leuven

Belgium

Phone +32 16 39 66 11

software.support@materialise.be

United Kingdom

AMP Technology Centre

Advanced Manufacturing Park

Brunel Way, Catcliffe

Sheffield, S60 5WG

Phone +44 1143 997 845

software.support@materialise.co.uk

China

Baoshan District

Hutai Road 2999

1F Building no 1

Shanghai 200444

P.R.China

Phone +86 21 583 124 06

software.support@materialise.com.cn

Asia – Pacific

Unit 5-01, Menara OBYU

No. 4, Jalan PJU 8/8A, Damansara

Perdana

47820 Petaling Jaya

Selangor Darul Ehsan - Malaysia

Phone: +603 7724 1415

software.support@materialise.com.my

Website:

<http://software.materialise.com/>

USA & Canada

44650 Helm Court

Plymouth, MI 48170

USA

Phone: +1 734 259 6445

Phone (toll-free): +1 888 662 5057

software.support@materialise.com

Germany

Friedrichshafener Str. 3

82205 Gilching Germany

Phone: + 49 8105 77 859 20

software.support@materialise.de

Japan

Yokohama Portside Bldg. 2F

Sakae-cho 8-1

Kanagawa-ku, Yokohama

Phone +81 45 440 4591

software.support@materialise.co.jp