CADS Additive
Print better.

2nd edition 04/2023

USER MANUAL AM-STUDIO

CADS Additive GmbH Technologiepark 17, 4320 Perg, Austria
T +43 7262 20540 // office@cads-additive.com // www.cads-additive.com

TABLE OF CONTENTS

TABLE OF CONTENTS

1 ABOUT THIS DOCUMENT

AM-Studio software is used for 3D data preparation for part generation using additive manufacturing procedures.

The software guides through the process using the following steps:

- Import and repair of native CAD formats and neutral file formats
- Positioning, orientation and nesting of parts
- Marking on build plate and print object
- Generation of support geometries
- Determination of exposure strategies
- Time and cost calculation of building processes

The user manual describes the full range of functions of AM-Studio.

Disclaimer

This user manual has been checked for conformity with the described software.

Nevertheless, it is not possible to exclude deviations, so that it is not possible to guarantee complete conformity.

The information in the user manual is checked on a regular basis. Necessary corrections are included in subsequent editions.

1.1 Purpose

This user manual provides the target group with the information, procedures and operating actions required for the respective phase of use for the AM-Studio software.

The user manual is part of the product and must be kept accessible to the user at all times.

1.2 Target group

The user manual is intended for all users of the AM-Studio software.

To understand this documentation, experience in 3D data preparation is helpful, but not necessary.

1.3 Means of representation

The following means of representation are used in this user manual:

Lists

Lists without a specific order that must be followed are depicted as lists with bullet points or indents.

Action instructions

Action instructions are structured as follows:

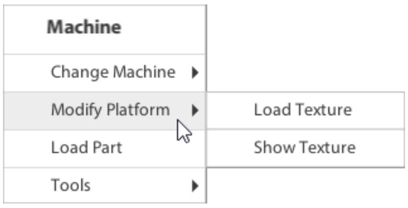
- Requirements for an action are identified with a tick ☑.
- The individual action steps are numbered.
- An interim result is identified with a white arrow with a black line ⇨.
- The final outcome of an action is identified with a white arrow in a black circle ➲.

Example:

☑ The **Orientation** action step is selected.

☑ An image file is loaded on the substrate plate.

1. Click on the substrate plate with the right mouse button.
   ⇨ A context menu opens.



2. Click on **[Modify Platform] [Show Texture]** in the context menu.

➲ The image on the substrate plate is hidden or displayed.

Action steps

In AM-Studio, the user is guided through the successive action steps, from the import of 3D models, to the actual printing process on machines from various manufacturers. These six action steps are highlighted in bold, green text, e.g. **Orientation**, **Build Process**.

Notes

Notes are highlighted with green margins and grey text. Notes usually contain important additional information or further information on a certain section.

Example:

Note

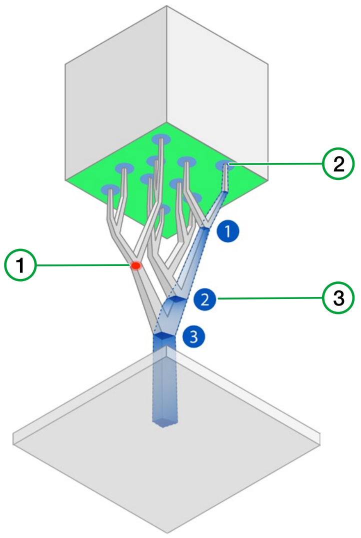
The height of the test specimen is updated automatically when a new, higher part is loaded onto the substrate plate.

Infographics

The infographics in this user manual usually have position numbers ① (circle with green borders and green writing) and reference lines. The individual positions are listed in a key below the infographic.

Numbers with filled-in circles (different colours possible) and white text, however, are part of the infographic.

Example:



1 Branch Point

2 Top Diameter

3 Diameter = Top Diameter + Summand x Branch Number

Information

Brief background information, e.g. information given in parameter tables, is identified with an info icon ⓘ.

Menu title and tabs

As a general rule, menu titles and tabs are written in a bold and italic font.

Example:

- "Click on the ***Build Strategy*** tab in the function panel."

Parameter

All programme elements where values can be entered, functions activated or points selected via a drop-down menu, for example, are classed as parameters.

In the text, parameters are highlighted in bold.
Example:

- "**Point Filter Tolerance**: Defines the width of the layer within which consecutive vectors are."

Cross-references

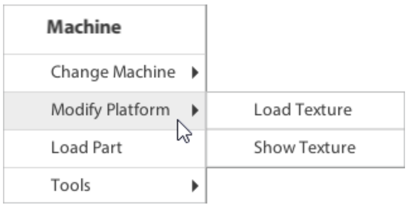
Cross-references to other chapters or sections of text in the user manual are written in green and are underlined. The cross-references are interactive, this means that when you click on a reference, the PDF document automatically jumps to the corresponding point in the text, e.g.

- *Step 1: Start*

Keys and buttons

- Keys on a computer keyboard are placed in angle brackets, e.g. <E> key, <Ctrl> key.

- Buttons in context menus and input windows are printed in bold and placed in square brackets, e.g. **[Save]**, **[Delete]**.
- If multiple buttons in a context menu need to be activated immediately one after the other, they will be written down in the corresponding order, e.g. "Click on **[Modify Platform] [Show Texture]** in the context menu".



1.4 Contact information

CADS Additive GmbH

Technologiepark 17

4320 Perg

Austria

+43 7262 20540

office@cads-additive.com

2 SECURITY

ATTENTION

Unsafe operating states caused by manipulation of the software

Manipulation of the software, e.g. by viruses, trojans or worms, can result in unsafe operating states in systems, machines and networks, which can lead to property damage.

- *Keep software up to date.*
- *Implement and continuously maintain a holistic security concept that reflects the current state of the art.*
- *Prevent unauthorised physical access to systems, machines and networks by locking systems.*
- *Restrict user rights of staff based on the function they perform.*
- *Segment and monitor networks.*
- *Make use of malware protection software.*
- *Check external storage media before use.*
- *Train staff regularly on current IT security threats and appropriate countermeasures.*
- *Make backups.*

3 NEW FEATURES AND FUNCTIONS

New functions and features of the program version AM-Studio 2023c are listed below.

- The hatching functions have been upgraded to add a further "lattice" hatch type.
- Machines which have laser combinations, are no longer listed separately in the general settings in ***ADD MACHINE***. For the variants SLM 280 and SLM 280 Twin, there is now, e.g. only one SLM 280 entry. A new "Laser Count" drop-down menu allows you to select the desired number of lasers.
- If a part was pre-deformed before saving a project, this was not visible when loading the project. The value of pre-deforming is now displayed in the model tree in "Simulation".
- If the info icons are deactivated, the display of the part dimensions now remains visible.

4 OVERVIEW

The AM-Studio software prepares 3D models for additive manufacturing.

The software ensures a successful printing process that produces the maximum possible number of parts in the shortest possible time and consequently optimises the utilisation of AM machines.

Operating concept

In AM-Studio, the user is guided through successive steps from importing the 3D models to the actual printing process on machines from various manufacturers.

These steps can be selected via the progress bar in the AM-Studio user interface.

Fig. 1: Progress bar

Action steps can be omitted. The progress bar can be used to return to completed action steps at any time.

The following action steps must be completed:

- Step 1: Start: Creation of a AM-Studio project and import the 3D models
- Step 2: Orientation: Positioning and orientation of the 3D models in the build chamber
- Step 3: Support: Adding structures to the 3D models to support the model during the build process while dissipating heat generated by the laser
- Step 4: Nesting: Duplication of parts and an efficient and non-contact arrangement of these copies or a large number of parts on the substrate plate
- Step 5: Build Process: Creation of the layers and laser paths for 3D printing and part simulation
- Step 6: Viewer: Verification of the laser paths and cost and time estimation

5 PROGRAMME START

Note

Before starting the software, AM-Studio must have been installed according to the document "AM-Studio_InstallationGuide".

AM-Studio can be started via the corresponding desktop icon or the start menu entry.

5.1 User interface

The user interface of AM-Studio can be divided into different sections.

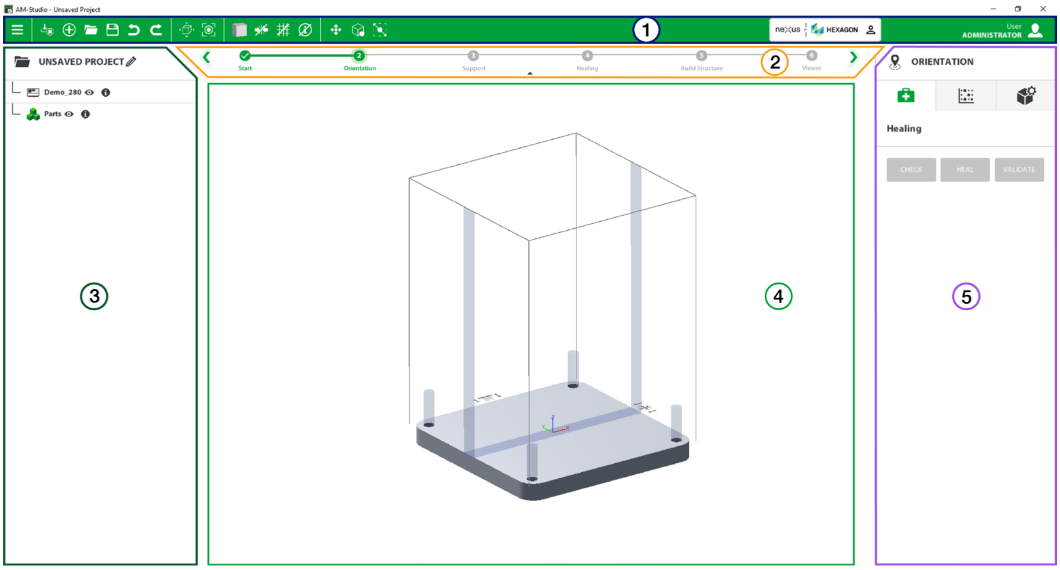


Fig. 2: User interface of the AM-Studio

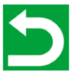
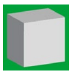
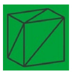
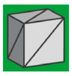
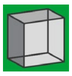
1 Menu bar
2 Progress bar
3 Model tree
4 Workspace
5 Function panel

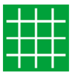
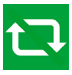
Menu bar

The selected user group is displayed on the right-hand side of the menu bar.

The following actions can be triggered via the buttons in the menu bar:

Icon	Action
	Import a part geometry (native CAD or 3D mesh model) into the current project
	Create a new project Clicking this icon displays a list of configured machine profiles from which the desired machine for the project is selected.
	Opening of a saved project

Icon	Action
	Saving of a current project
	Undo the previous action (Key combination: <CTRL> + <Z>).
	Restore the undone action (Key combination: <CTRL> + <Y>).
	Selecting the view of the build chamber Possible views: • Front • Rear • Top • Bottom • Left • Right • Isometric
	Set the focus on the selected part
   	Set the part display Possible values: • Shaded • Wire mesh • Shaded and wire mesh • Semi-transparent

Icon	Action
 	Display/no display of part collisions in red shading To display a collision, the option Activate Part Collision Module must also be activated in the settings under General.
 	Show/Hide a Grid on the Substrate Plate
 	Show/hide information symbols on the parts
 	Switch between Translation and Rotation
 	Toggle between local and global coordinate system
 	Switch between part origin and part centre point

Tab. 1: Buttons in the menu bar

Progress bar

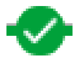
The action step to be processed can be determined via the progress bar.

Fig. 1: Progress bar

A specific action step can be selected in the following ways:

- By clicking
- By using the function keys <F1> to <F6>
- By using the green arrows on the right and left edges of the function bar.

The following symbols on the function bar show the processing status of the individual action steps.

Icon	Meaning
	Action currently selected for processing
	Action step completed
	Action step not yet processed

Tab. 2: Display of the action steps in the function panel

Function panel

The functions available in the respective action step can be selected via the function panel.

The function panel can be positioned freely by clicking and dragging its title bar. It can snap to the right edge of the model tree or to the right edge of the AM-Studio window.

The button in the title bar of a function panel that has not been snapped in can be used to hide and also show the contents of the function panel except for the title bar.

Workspace

The workspace shows the substrate plate with the loaded parts.

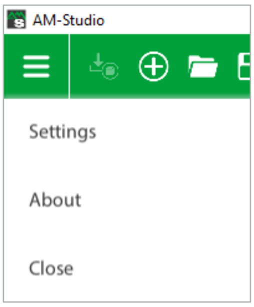
Model tree

All parts used in the project are listed in the model tree.

5.1.1 Setting the display

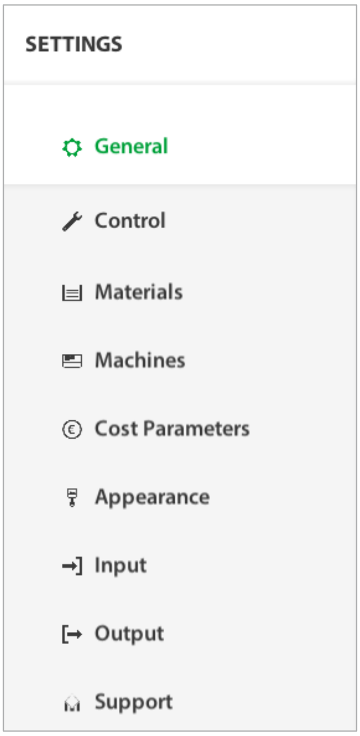
1. Click on the burger menu in the menu bar.

   ⇨ A drop-down menu opens.

2. Click on **Settings** in the drop-down menu.

   ⇨ A menu opens.



3. Click on the tab ***Appearance*** in the menu.
   ⇨ The setting options are displayed.
4. Define the parameters as desired.
5. Confirm the entries by clicking on **[SAVE]**.

➲ The changes are accepted.

Clicking on **[Close]** closes the ***Settings*** menu.

Note

All options for display settings are described in the chapter Setting (see *Appearance**).*

6 SETUP

The following settings must be specified before preparing 3D models for additive manufacturing for the first time.

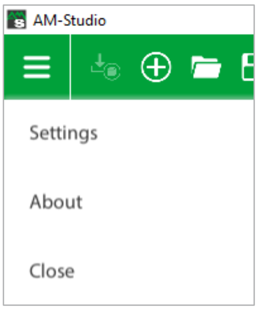
6.1 Creating a new material

A material profile must be created for the material used in additive manufacturing.

Creating a material profile

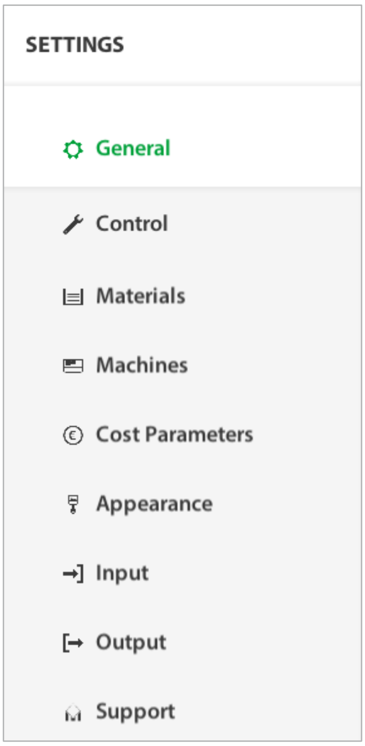
1. Click on the burger menu in the menu bar.

   ⇨ A drop-down menu opens.



2. Click on **[Settings]** in the drop-down menu.

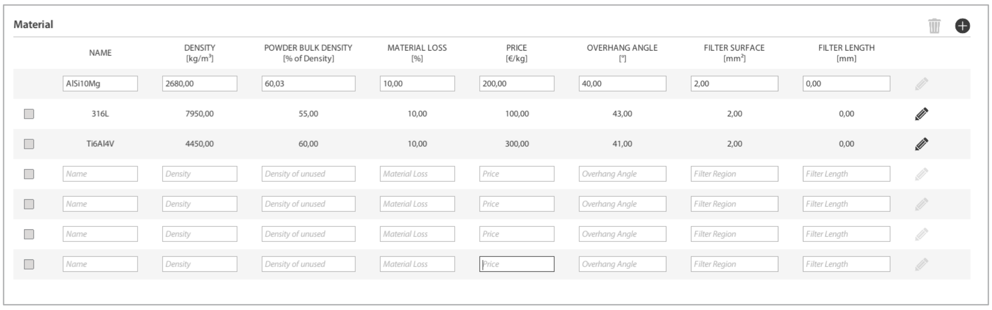
   ⇨ A menu opens.



3. Click on the ***Materials*** tab in the menu.

   ⇨ The setting options are displayed.

4. If no suitable material profile has been created, click on the button ⊕ to create a new material profile.



5. Fill in the input fields according to the material used.
6. Click **[SAVE]**.

➲ The new material profile is created.

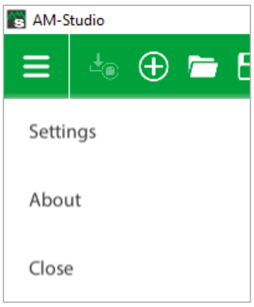
Clicking on **[Close]** closes the ***Settings*** menu.

6.2 Creating a new machine

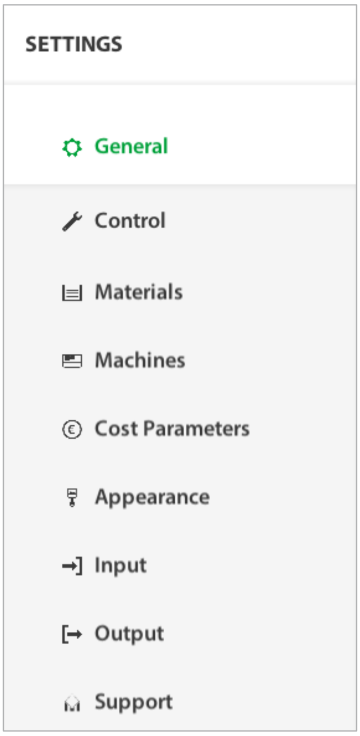
A machine profile must be created for the AM machine used.

Creating a machine profile

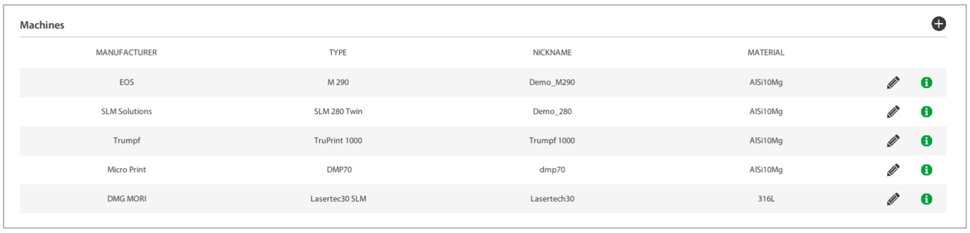
1. Click on the burger menu ☰ in the menu bar.
   ⇨ A drop-down menu opens.



2. Click on **[Settings]** in the drop-down menu.
   ⇨ A menu opens.



3. Click on the ***Machines*** tab in the menu.
   ⇨ The setting options are displayed.
4. If no suitable machine profile has been created, click on the button ⊕ to create a new machine profile.

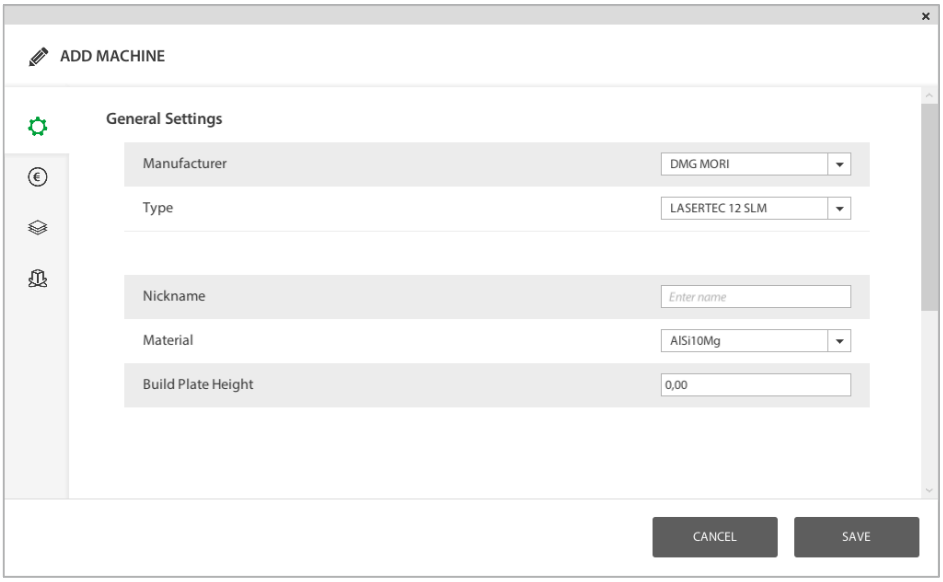


Machines

MANUFACTURER	TYPE	NICKNAME	MATERIAL
EOS	M 290	Demo_M290	AlSi10Mg
SLM Solutions	SLM 280 Twin	Demo_280	AlSi10Mg
Trumpf	TruPrint 1000	Trumpf 1000	AlSi10Mg
Micro Print	DMP70	dmp70	AlSi10Mg
DMG MORI	Lasertec30 SLM	Lasertech30	316L

   ⇨ A window opens.

5. Under ***General Settings***, define the general parameters of the AM machine.



- In Manufacturer, select the manufacturer of the AM machine used.
- In Type, select the AM machine used.
- In Nickname, enter a name for the machine profile.
- In Material, select the material used.
- In Build Plate Height, enter the thickness of the substrate plate.
  Depending on the AM machine, the thickness of the substrate plate reduces the possible build height.

6. Only visible for certain AM machines:
   Under **AM-LOCK**, activate the zero-point clamping system if required.
7. Under **Costs**, define the operating costs of the AM machine used.
8. Under **Build Processor**, define the build parameters of the AM machine used.
9. Click **[SAVE]**.

➲ The new machine profile is created.

Clicking on **[Close]** closes the ***Settings*** menu.

6.3 Restricting user rights

In AM-Studio, user profiles are defined with the following rights.

Rights	User profiles			
	Administrator	Process Developer	Process Engineer	Qualified Process Engineer
Displaying cost parameters	✓			✓
Editing cost parameters	✓			
Displaying build time calculation parameters	✓	✓		✓
Editing build time calculation parameters	✓	✓		
Editing machine types	✓	✓		
Deleting machine types	✓	✓		
Editing support parameters	✓	✓	✓	
Editing build strategy parameters	✓	✓	✓	

Tab. 3: User rights in AM-Studio

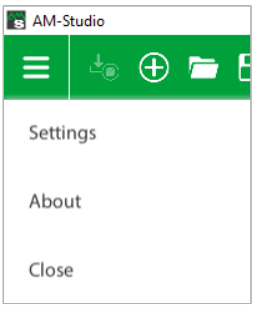
When the programme is started for the first time, the user profile "Administrator" is active.

Switching between user profiles can be protected with a password. If a password is set up, it is only possible to switch to another profile after entering the password correctly.

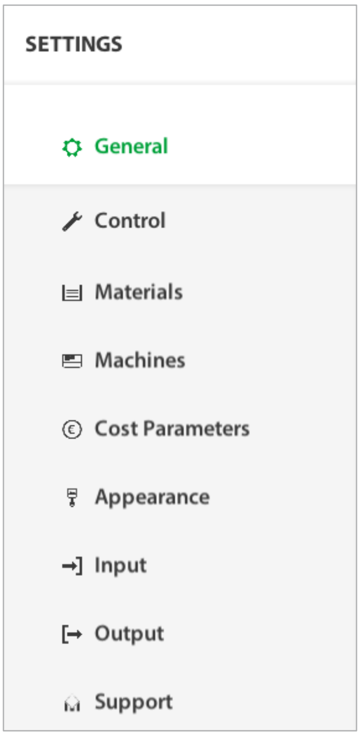
Setting a password for switching user profiles

1. Click on the burger menu in the menu bar.

   ⇨ A drop-down menu opens.

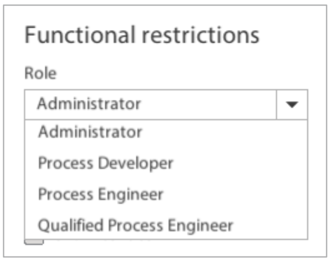


2. Click on **[Settings]** in the drop-down menu.
   ⇨ A menu opens.

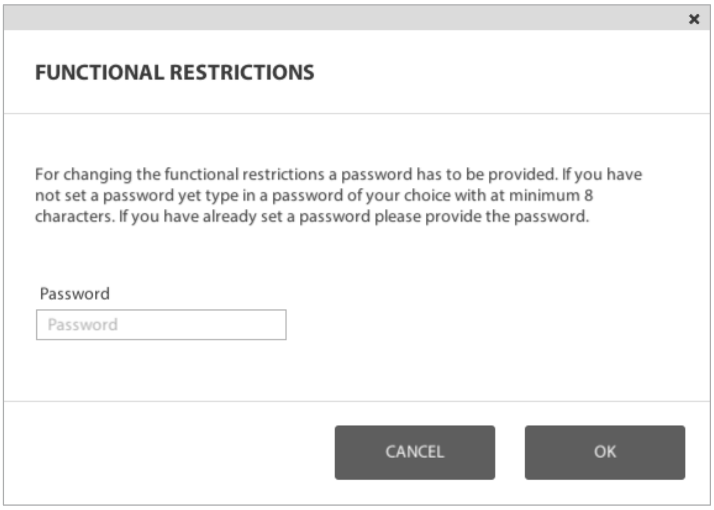


3. Click on the ***General*** tab in the menu.
   ⇨ The setting options are displayed.

4. Under **Functional restrictions**, select the desired user profile via the drop-down menu **Role**.



⇨ A window opens.



5. Under **Password**, assign a password for switching the profile.
6. Click **[OK]**.

➲ The selected user profile is active and a password for switching the user profile is set.

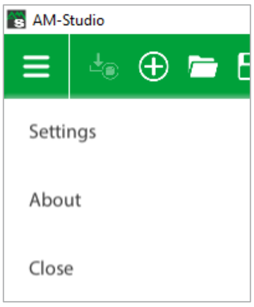
6.4 Activating the CAD Interface

ⓘ If a licence for the CAD Interface has been purchased, it must be activated in the settings.

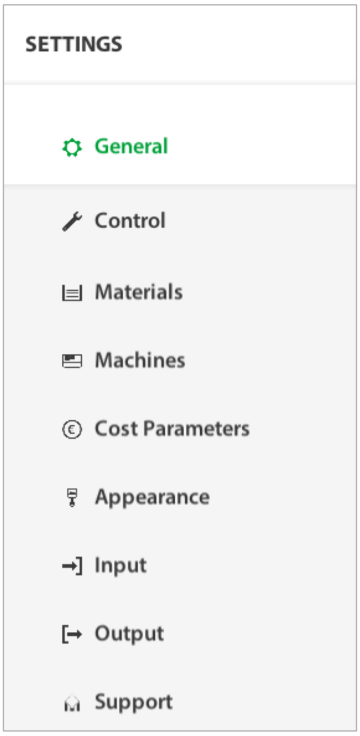
☑ A CAD interface licence has been purchased.

1. Click on the burger menu ☰ in the menu bar.

⇨ A drop-down menu opens.



2. Click on **[Settings]** in the drop-down menu.
   ⇨ A menu opens.



3. Click on the ***General*** tab in the menu.
   ⇨ The setting options are displayed.
4. Under **Licensed Features**, check the box for **CAD Interface**.

➲ The CAD Interface is activated.

7 STEP 1: START

This chapter contains information on the action step **Start**. The information serves as support when preparing a project.

Fig. 3: *Progress bar: Start action step*

AM Studio supports a range of file formats when reading in CAD data, including the following:

- STL
- etc.

Only in conjunction with CAD Interface

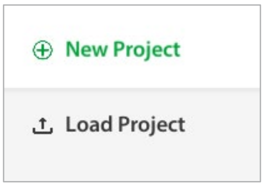
- Autodesk Inventor
- Dassault Systemes Catia
- PTC Creo/ProE
- Siemens NX
- STEP
- etc.

7.1 Creating a project

☑ The desired machine is created in ***Settings*** (see Creating a new machine).

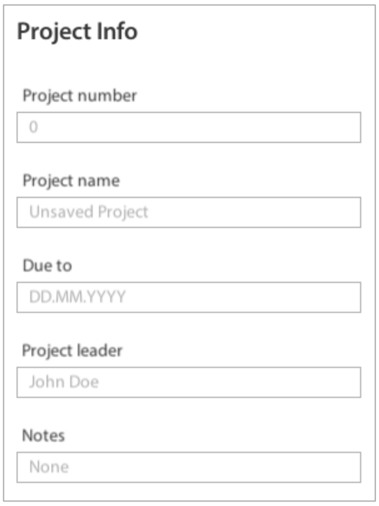
☑ The action step **Start** is selected.

1. Click on **[New Project]** in the left window area.



2. Select the desired machine in the list ***Machines***.

3. If necessary, enter additional data for the project in the input fields on the right side of the screen.



4. Click on **[Start Project]** in the footer.

➲ The project is created.

7.2 Saving a project

ⓘ A project that has been started can be saved before the project is completed. This allows work to be interrupted and then continued with a saved intermediate status.

1. Click on the burger menu in the menu bar.
   ⇨ A drop-down menu opens.

2. Click **[Save As...]**.
   ⇨ An Explorer window opens.
3. Navigate to the desired save location.
4. Click **[Save]**.

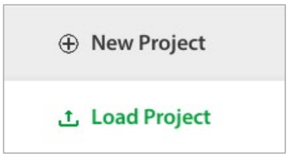
➲ The project is saved taking into account the currently selected action step.

7.3 Loading a project

ⓘ A project that has already been created and saved can be loaded.

☑ The **Start** action step is selected.

1. Click on **[Load Project]** in the left window area.



⇨ An Explorer window opens.

2. Navigate to the project to be loaded.
3. Select the project.
4. Click **[Open]**.

➲ The project is loaded

Note

Recently loaded projects appear under Recent projects and can be selected from there.

8 STEP 2: ORIENTATION

This section describes the possible operations in the **Orientation** action step.

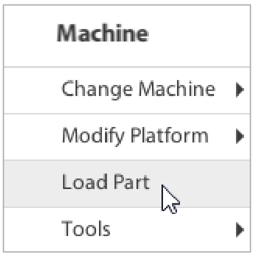
Fig. 4: Progress bar: Orientation action step

8.1 Loading a file

☑ A project has been created.

☑ The **Orientation** action step is selected.

1. Click on the substrate plate with the right mouse button.
   ⇨ A context menu opens.



2. Click on **[Load part]** in the context menu.
   ⇨ An Explorer window opens.
3. Navigate to the file to be loaded.
4. Select the file with a mouse click. Multiple selection is possible by holding down the <Ctrl> key.
5. Click **[Open]**.

➲ The file is loaded

Note *Alternatively, a part can also be imported into AM-Studio via the menu bar or with "Drag and Drop".*

8.2 Changing a project

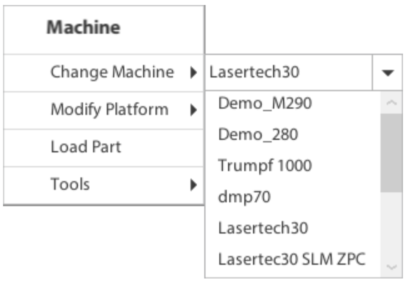
ⓘ Projects that have already been created can be adapted during editing.

Exchanging the machine

☑ Several machines are defined in the programme settings under ***Settings > Machines***.

☑ The **Orientation** action step is selected.

1. Click on the substrate plate with the right mouse button.
   ⇨ A context menu opens.
2. Click on **[Change machine]** in the context menu.
   ⇨ A drop-down menu appears.



3. Select the desired machine via the drop-down menu.

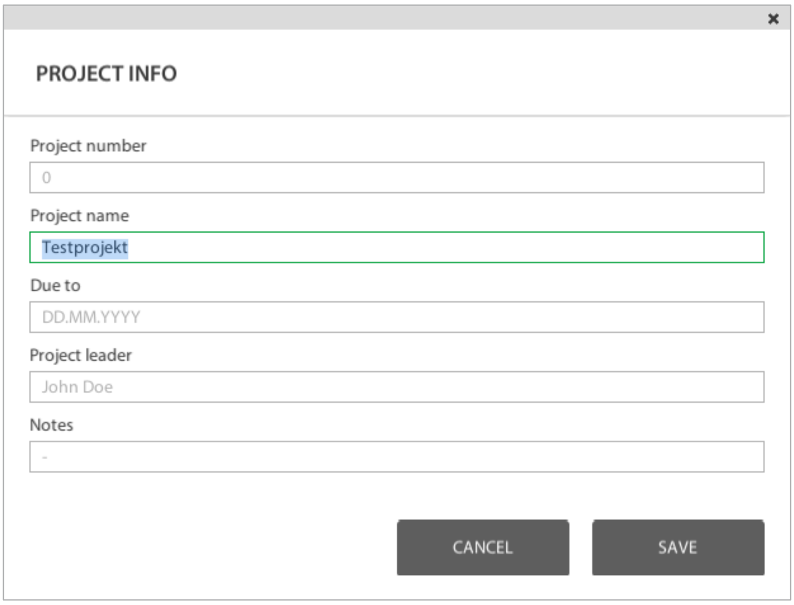
➲ The machine-specific parameters are applied to the project.

Changing the project name and metadata

☑ The **Orientation** action step is selected.

1. Navigate to the project name in the model tree.

2. Click on the pencil symbol behind the project name.
   ⇨ A window opens.



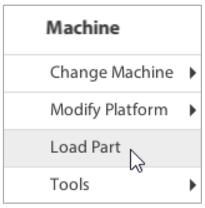
3. Enter the new project name and the new metadata in the input fields.
4. Click **[Save]**.

➲ The project and its metadata are changed.

Adding a part

☑ The **Orientation** action step is selected.

1. Click on the substrate plate with the right mouse button.
   ⇨ A context menu opens.



2. Click on **[Load part]** in the context menu.
   ⇨ An Explorer window opens.
3. Navigate to the desired file.
4. Select the file with a mouse click. You can select multiple files by holding down the <Ctrl> key.
5. Click **[Open]**.

➲ The file is loaded.

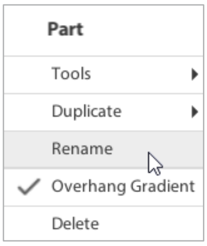
Note | *Alternatively, a part can also be imported into AM-Studio via the menu bar or with "Drag and Drop".*

Renaming a part

☑ The **Orientation** action step is selected.

1. Right-click on the part to be renamed. Alternatively, click on the part name in the model tree with the right mouse button.

   ⇨ A context menu opens.

2. Click on **[Rename]** in the context menu.

   ⇨ The name of the part can be changed in the model tree.

3. Edit the name as desired.

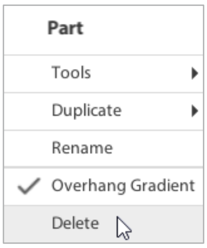
➲ The part is renamed.

Deleting a part

☑ The **Orientation** action step is selected.

1. Right-click on the part to be deleted. Alternatively, click on the part name in the model tree with the right mouse button.

   ⇨ A context menu opens.

2. Click on **[Delete]** in the context menu.

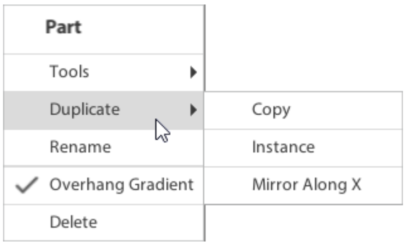
➲ The part is deleted.

Copying a part

☑ The **Orientation** action step is selected.

1. Click with the right mouse button on the part to be copied. Alternatively, click on the part name in the model tree with the right mouse button.

   ⇨ A context menu opens.

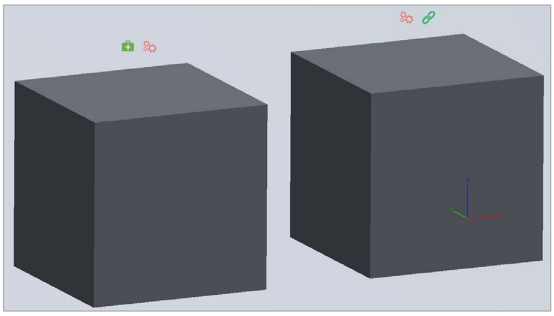


2. Click on **[Duplicate] [Copy]** in the context menu.

➲ The part is copied.

Create an instance of a part

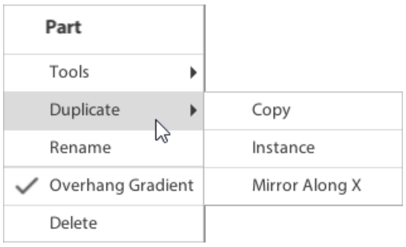
ⓘ Instances of a part are – unlike copies – permanently linked to the original. Any change to the original, such as the rotation of a part, also affects the instance of the part. Instances of parts are marked with a chain symbol in AM-Studio.



☑ The **Orientation** action step is selected.

1. Rick click the part of which an instance is to be created. Alternatively, click on the part name in the model tree with the right mouse button.

   ⇨ A context menu opens.



2. Click on **[Duplicate] [Instance]** in the context menu.

➲ An instance of the part is created.

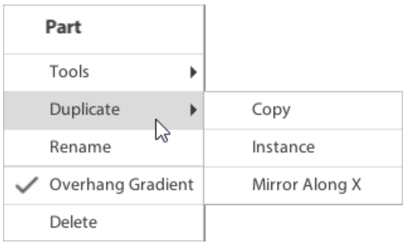
Mirror part along X-axis

ⓘ The function "Mirror Along X" creates a copy of a part mirrored on the X-axis.

☑ The **Orientation** action step is selected.

1. Right click the part of which a copy mirrored along the X-axis is to be created. Alternatively, click on the part name in the model tree with the right mouse button.

   ⇨ A context menu opens.



2. Click on **[Duplicate] [Mirror Along X]** in the context menu.

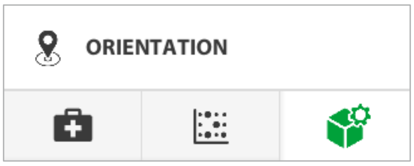
➲ A copy of the part mirrored on the X-axis is created.

Adjust the unit of measurement of a part

ⓘ AM-Studio uses the unit of measurement millimetres by default. If a part was designed in a different unit of measurement, the unit of measurement must be adjusted. AM-Studio automatically scales the part after the adjustment so that the part corresponds to the millimetre length dimension.

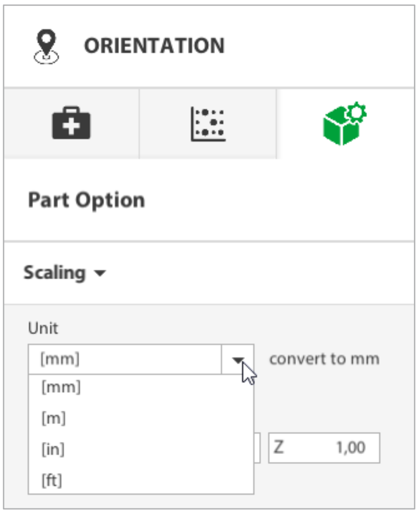
☑ The **Orientation** action step is selected.

1. Click on the part whose unit of measurement is to be adjusted.
2. Click on the tab ***Part Option*** in the function panel.



3. Open the ***Scaling*** menu.

4. Select the desired unit from the drop-down menu under **Unit**.



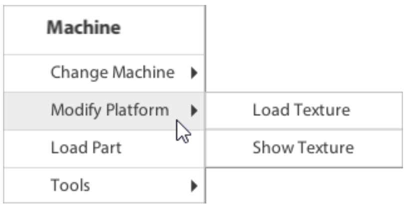
➲ The unit is applied to the part and the part is scaled.

Loading image files onto the substrate plate

ⓘ Image files can be loaded onto the substrate plate.

☑ The **Orientation** action step is selected.

1. Click on the substrate plate with the right mouse button.
   ⇨ A context menu opens.



2. Click on **[Modify Platform] [Load Texture]** in the context menu.
   ⇨ An Explorer window opens.
3. Navigate to the desired file.
4. Select the file with a mouse click.

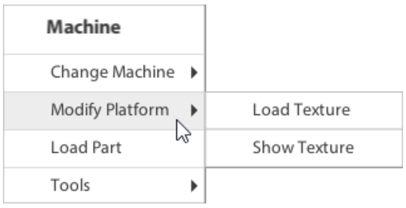
➲ The image file is loaded onto the substrate plate.

Hide and show image files on the substrate plate

☑ The **Orientation** action step is selected.

☑ An image file is loaded on the substrate plate.

1. Click on the substrate plate with the right mouse button.
   ⇨ A context menu opens.



2. Click on **[Modify Platform] [Show Texture]** in the context menu.

➲ The image on the substrate plate is hidden or shown.

8.3 Part view and cut plane

ⓘ With the "View cut" function, it is possible to cut through parts on the substrate plate and move through them layer by layer. This allows the internal structure of a part to be viewed.

Note

For better orientation in the working area, the rest position of the coater and the gas flow are shown symbolically on the substrate plate.

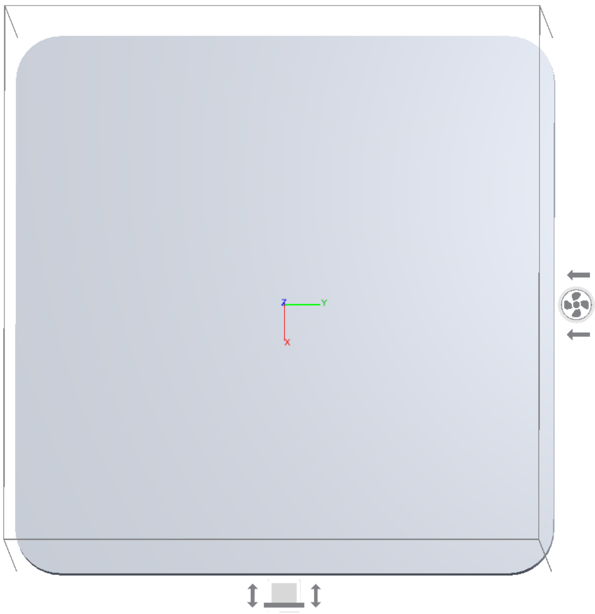


Fig. 5: Substrate plate from above with rest position of the coater and inflow side of the gas

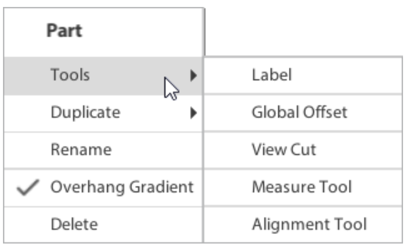
Icon	Description
	Rest position of the coater
	Inlet side of the gas

Tab. 4: Symbols for coater and gas flow

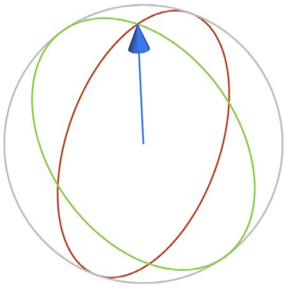
☑ The action steps **Orientation** or **Support** are selected.

1. Click on a part with the right mouse button.

   ⇨ A context menu opens.



2. Click on **[Tools] [View cut]** in the context menu.
   ⇨ The rotation gizmo is displayed.



3. Move the mouse pointer over the red or green circle.
   ⇨ The selected circle is highlighted in yellow.

Note

The circles are only selected when the mouse pointer hovers over the circle segment facing the viewer.

4. Click on the selected circle and keep the left mouse button pressed.
5. Move the circle.
   ⇨ The part is intersected according to the axis of rotation of the respective circle.
6. Move the mouse pointer inside the white circle.
   ⇨ The white circle is marked yellow.
7. Click into a free area inside the circle and keep the left mouse button pressed.
   ⇨ The cut plane can be moved freely along the X and Y axes.
8. Click on the arrow and keep the left mouse button pressed.
9. Move the arrow in the desired direction.
   ⇨ The part is traversed according to the selected rotation axis.
10. Press the <Esc> key on the keyboard.
    ⇨ The "View cut" function is terminated.

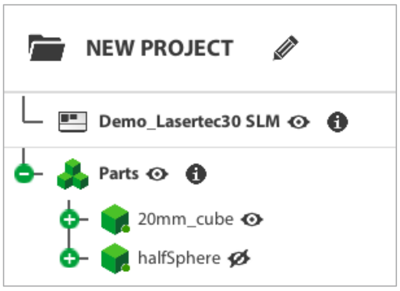
Note

The View cut function can also be controlled via a combination of keys and the mouse wheel. The part is cut through in defined steps.
1. [Ctrl] + mouse wheel cuts the part into thicker layers
2. [Ctrl] + mouse wheel cuts the part into thinner layers

8.4 Hiding and showing a part

☑ The **Orientation** action step is selected.

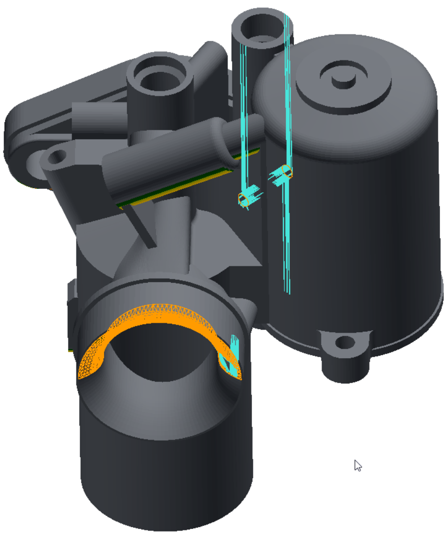
1. Navigate to the desired part in the model tree.
2. Click on the 👁 or ⊘ button.



➲ The part is hidden or shown.

8.5 Repairing a geometry

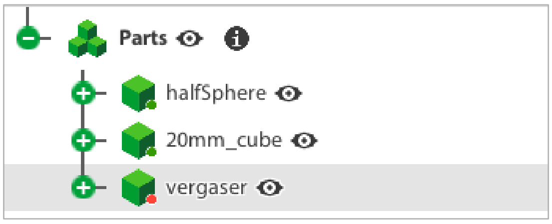
ⓘ The function "Healing" in the function panel can be used to repair errors in a part geometry. AM-Studio analyses parts loaded into the program and displays faulty areas in colour.



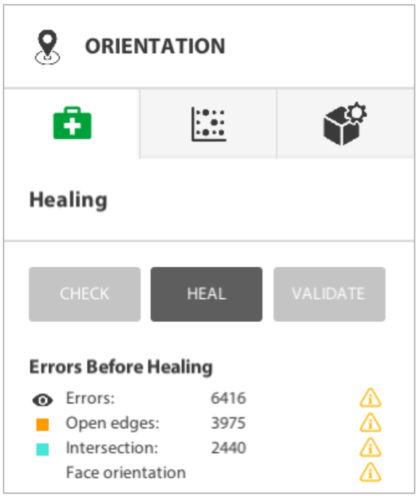
Note

Faulty parts are marked with a red first aid kit in the build cahmber. Fault-free parts have a green first aid kit.

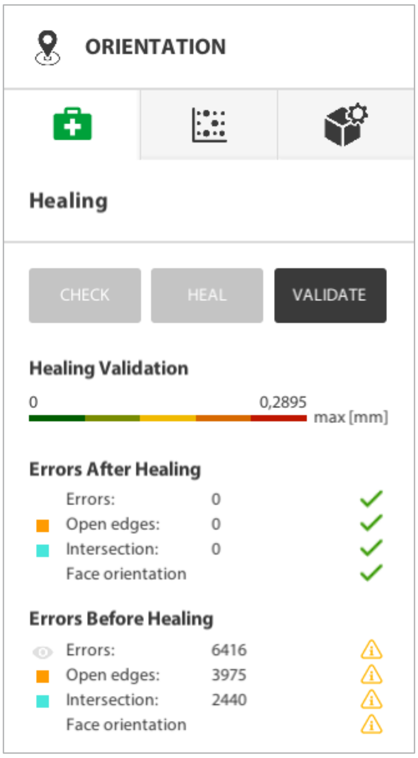
Faulty parts are marked with a red dot in the model tree. Fault-free parts have a green dot.



ⓘ Orange areas on the part indicate open edges in the part. Blue areas indicate defects in cut surfaces.



After the healing process, healed areas on the part are coloured. The quality of the healed areas can be evaluated using a scale, the Healing Validation. Successful repairs are marked with a green check mark ✓.

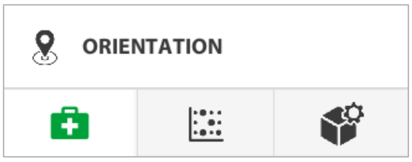


Repairing a geometry

☑ A file is loaded.

☑ The **Orientation** action step is selected.

1. Click on a part.
2. Click on the tab ***Healing*** in the function panel.



3. Click **[Check]**.

   ⇨ All errors in the part geometry are highlighted in blue or orange.

   ⇨ An error report is displayed in the function panel.

Note

*If the function "Auto Check Errors" is activated in the Settings under Input > Healing Options, each part that is loaded onto the substrate plate is automatically checked for errors in the geometry. The **[Check]** button is then greyed out. "Auto Check Errors" is activated by default in AM-Studio.*

4. Click **[Heal]**.

   ⇨ All detected errors in the geometry are repaired.

5. Click **[Validate]**.

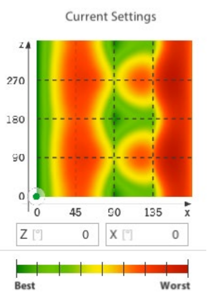
   ⇨ All differences between the repaired geometry and the original geometry are shown in colour.

➲ The geometry is repaired.

8.6 Part placement with Orientation Maps

8.6.1 Basics

The Orientation Maps in AM-Studio support the user in the orientation of a part on the substrate plate. The five criteria " Build Time" (Speed), "Support Volume" (Support), "Disortion" (Distortion), " Part Projection" (Shadow) and "Step Error" (Stairstep) make it possible to find the optimal orientation for each part. Up to three criteria can be superimposed. The result of this superposition is displayed in the orientation map "Current Settings". The colour ranges shown in the Orientation Maps help to evaluate the respective orientation of a part. An evaluation scale can be used to determine whether the orientation of the part is good or in need of optimisation.

Graphic	Description
	The quality evaluation of the respective orientation is based on colour charts. Green areas stand for good orientation and red areas for poor orientation. **Note** *The orientation maps shown here use the "Rainbow" colour scheme. In the programme settings it is possible to switch to the "Blue" colour scheme. In this colour scheme, white stands for optimal orientation and dark blue for poor orientation.*

Tab. 5: Reference table for Orientation Maps

A total of 6 Orientation Maps are available for the orientation of the part on the substrate plate.

Icon	Description
	"Speed" shows the orientation possibilities of the part when the weighting is completely on the build time.
	"Support" shows the orientation possibilities of the part when the weighting is completely on the supports. A good orientation means that less support material is needed, a bad orientation means that more support material is needed.
	"Distortion" shows the orientation possibilities of the part when the weighting is completely on the distortion of the part. The relative distortion tendency of the part is shown for the selected orientation. For example, if the laser has to process very long paths, the probability that the part will warp is generally higher.
	"Shadow" shows the orientation possibilities of the part when the weighting is completely on the shadow cast by the part. This orientation map is based on the area of the part projected perpendicular to the substrate plate. Depending on the orientation, the part casts more or less shadow on the substrate plate. The size of the shadow is used as a basis to calculate the number of parts that can fit on the substrate plate.

Icon	Description
	"Stairstep" shows the orientation possibilities of the part when the weighting is completely on the step error. This criterion takes into account how strongly the dimensional accuracy of inclined surfaces is influenced by the selected orientation. In the layering method, small stair-like steps (step defects) occur in the part due to technical reasons if the degree of inclination of a surface is too large.

Tab. 6: Description of the five criterion-related orientation maps

"Current Settings" is a calculation of a maximum of three selected Orientation Maps. Depending on the relationship between the selected Orientation Maps selected in the Magic Triangle, the representation in Current Settings is adjusted. If, for example, the weighting is entirely on the build time, Current Settings and this Orientation Map match. With the movable green point in Current Settings, an area can be selected that is suitable for an optimal positioning of the part. The part in the build chamber aligns itself following the movements of the green point. The positioning of the green point in Current Settings is transferred to all selected Orientation Maps.

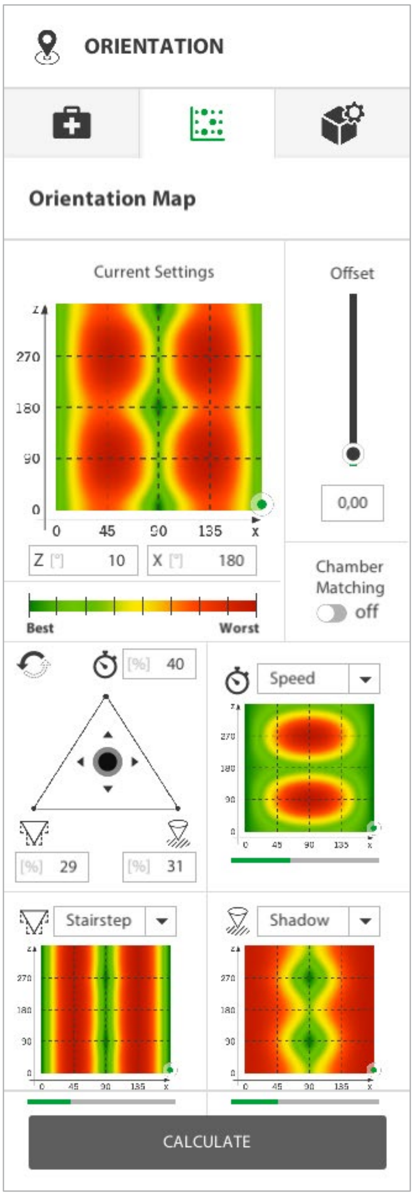
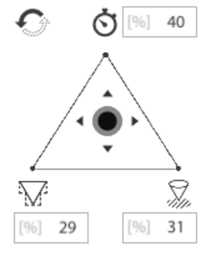


Fig. 6: The menu of the Orientation Maps

The Magic Triangle

Graphic	Description
	Magic Triangle can be used to define the weighting in terms of build time, support volume, warpage, shadow cast/part projection or step error in the orientation of the part. Depending on the position of the black dot, the corresponding Orientation Maps are overlaid. The result of this overlay is displayed in Current Settings. If the weighting of the criteria is to be done as precisely as possible, the corresponding percentage weighting values can also be entered manually.

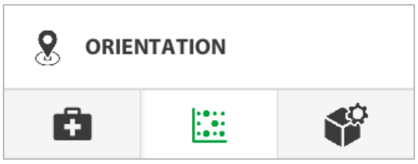
Tab. 7: Reference Table for Magic Triangle

8.6.2 Placement

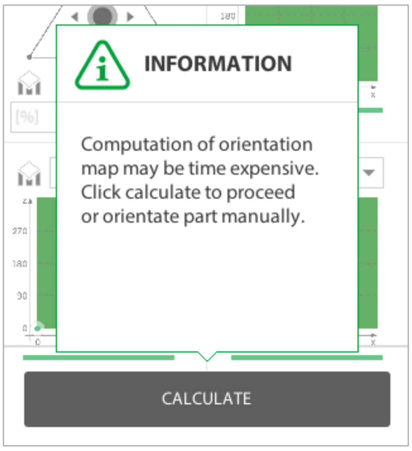
Placing the part with Orientation Maps

☑ The **Orientation** action step is selected.

1. Click on the part.
2. Click on the tab ***Orientation Map*** in the function panel.



⇨ If the parts are complex, a note opens that the calculation of the colour maps can be time-consuming.



3. Click on the **[Calculate]** button.

⇨ Four Orientation Maps are created.

4. Select the desired criteria for the three lower Orientation Maps via the drop-down menu.

⇨ The selected criteria - recognisable by their symbols - are distributed to the corners of the Magic Triangle.

5. In the Magic Triangle, drag the black dot within the triangle in the direction of the corner points to place the weighting in the orientation on the respective criterion.

⇨ Depending on the orientation of the black point within the Magic Triangle, Current Settings is composed of the three selected criteria.

6. Place the green point within Current Settings according to the desired orientation of the part in the build chamber.

➲ The part is oriented in the build chamber depending on the orientation of the green dot within Current Settings.

Note

It is possible to navigate to the local minima and maxima within Current Settings by holding down the <Shift> key.
For an exact weighting, numerical values can also be entered in the corresponding input windows at the orientation criteria.

Positioning a part with Chamber Matching

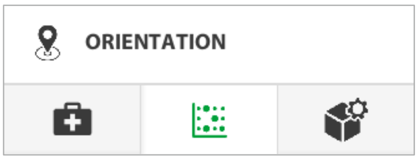
ⓘ If areas of a part are outside the build chamber, the part is coloured red. The function "Chamber Matching" can help to find a suitable positioning even for parts that are difficult to place in the build chamber due to their size.

☑ The **Orientation** action step is selected.

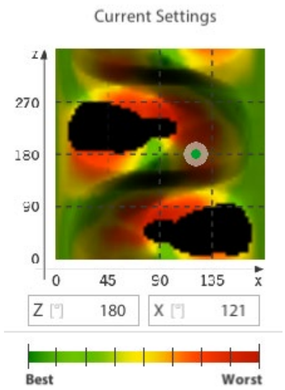
☑ A Orientation Map for the part is calculated.

☑ The part in the build chamber is coloured red.

1. Click on the tab ***Orientation Map*** in the function panel.



2. Activate the function "Chamber Matching" with the toggle switch .
   ⇨ The Orientation Map Current Settings changes to the mode "Chamber Matching".



Note

Part orientations in which parts of the part are outside the chamber are shown in black in Current Settings. If coloured areas are also displayed, it is possible to place the part completely in the build chamber. The same colour scale applies to these coloured areas as to other Orientation Maps.
If it is a darkened coloured area, there are a few Z-rotations in which the part lies within the build chamber.

3. Use the green dot in Current Settings to select an area that allows complete placement of the part.
   ⇨ The part changes colour from red to grey.

➲ The part is completely placed in the build chamber.

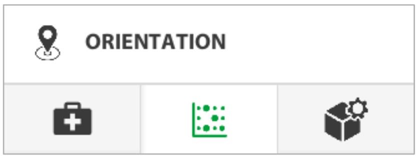
Specifying an offset value

ⓘ By entering an offset value, a defined distance in millimetres between part and substrate plate can be defined.

☑ The **Orientation** action step is selected.

☑ A Orientation Map for the part is calculated.

1. Click on the tab ***Orientation Map*** in the function panel.



2. Enter the desired offset value manually in the input field **Offset** or alternatively define it with the slider.

➲ A distance is created between the part and the substrate plate.

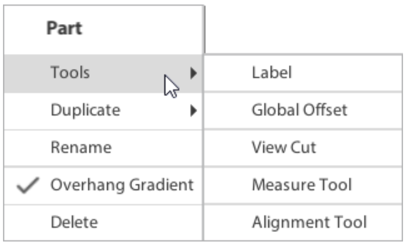
8.7 Rotation of a part

ⓘ In AM-Studio, there are different possibilities to manually rotate a part in the build chamber.

Rotation of a part with the Alignment Tool

☑ The **Orientation** action step is selected.

1. Click on a part with the right mouse button.
   ⇨ A context menu opens.



2. Click on **[Tools] [Alignment Tool]** in the context menu.
   ⇨ If the mouse is moved over the part, a turquoise vector arrow appears, which is perpendicular to the surface of the part below it (surface normal).
3. Click on the part.
   ⇨ The part is rotated.

➲ The surface selected with the arrow is rotated in the direction of the substrate plate and aligned perpendicular to the substrate plate.

Press the <Esc> key to exit the Alignment Tool.

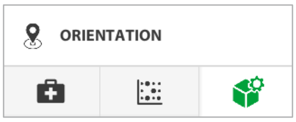
Note

Hold down the <Shift> key to rotate the part in the opposite direction when the arrow is clicked.

Rotating a part by entering values – Relative rotation

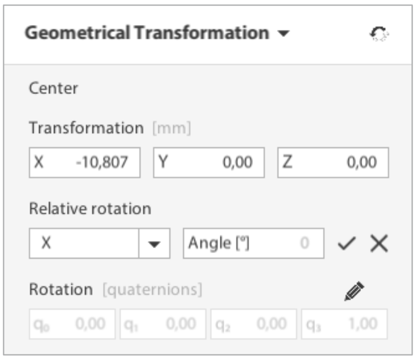
☑ The **Orientation** action step is selected.

1. Click on the part.
2. Click on the tab ***Part Option*** in the function panel.



3. Open the menu ***Geometrical Transformation*** .
4. Select the desired rotation axis from the drop-down menu under **Relative rotation**.

5. Enter the desired rotation angle in **Angle[°]**.



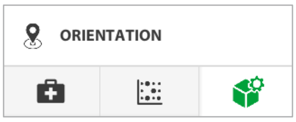
6. Confirm the entry by clicking on the check mark .

➲ The part is rotated accordingly.

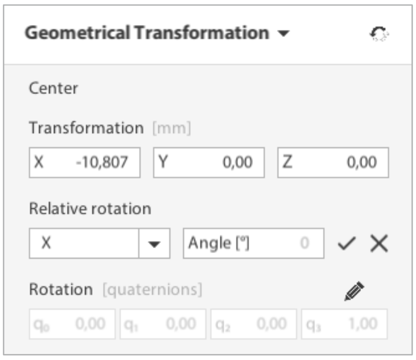
Rotating a part by entering values – Rotation

☑ The **Orientation** action step is selected.

1. Click on the part.
2. Click on the tab ***Part Option*** in the function panel.



3. Open the ***Geometrical Transformation*** menu.
4. Click on the pencil symbol at **Rotation** to enter quaternion values.



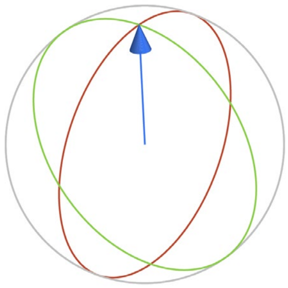
5. Enter the desired values.
6. Confirm the entry by clicking on the check mark .

➲ The part is rotated accordingly.

Rotating a part with the rotation gizmo

☑ The **Orientation** action step is selected.

1. Click on the part.
2. Press the <E> key on the keyboard.
   ⇨ The rotation gizmo is displayed.



3. Move the cursor over the circle with the desired rotation direction.
   ⇨ The selected circle is highlighted in yellow.
4. Click on the selected circle and keep the left mouse button pressed.
   ⇨ The part can be moved in the selected rotation direction.
5. Select the red circle to rotate the part around the X-axis.
6. Select the green circle to rotate the part around the Y-axis.
7. Select the blue circle to rotate the part around the Z-axis.

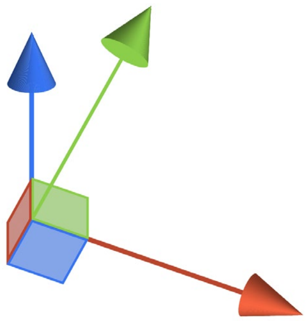
➲ The part is rotated accordingly.

8.8 Placing a part

Moving a part with the translation gizmo

☑ The **Orientation** action step is selected.

1. Click on the part.
   ⇨ Three arrows and a square appear above the part.



2. Click on the red arrow and hold down the left mouse button to move the part in the X direction.
3. Click on the green arrow and keep the left mouse button pressed to move the part in the Y direction.
4. Click on the blue arrow and keep the left mouse button pressed to move the part in the Z direction.

5. Move the part along the respective arrow until the part has reached the desired position.
6. Click on the corresponding side of the square and keep the left mouse button pressed to move the part along a plane.

   Blue: XY plane

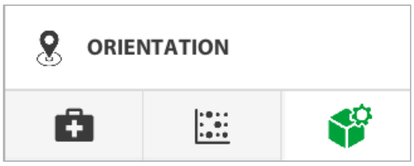
   Red: YZ plane

   Green: XZ plane

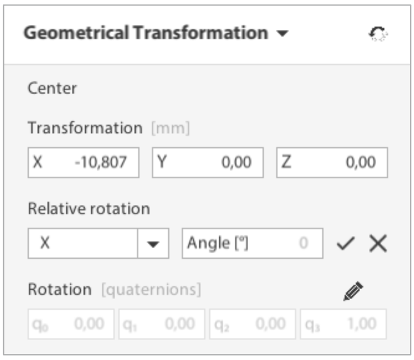
Moving a part by entering values

☑ The **Orientation** action step is selected.

1. Click on the part.
2. Click on the tab ***Part Option*** in the function panel.



3. Open the menu ***Geometrical Transformation***.



4. Under **Transformation** enter a value in the input field [X] to move the part in X direction.
5. Under **Transformation** enter a value in the input field [Y] to move the part in Y direction.
6. Under **Transformation** enter a value in the input field [Z] to move the part in Z direction.

   Confirm each of these entries by pressing the<Enter> key.

➲ The part is shifted according to the values entered.

8.9 Scaling a part

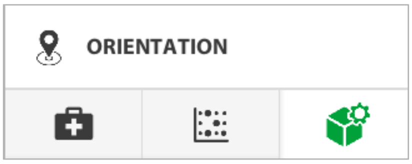
ⓘ A loaded part can be scaled in size.

Individual scaling

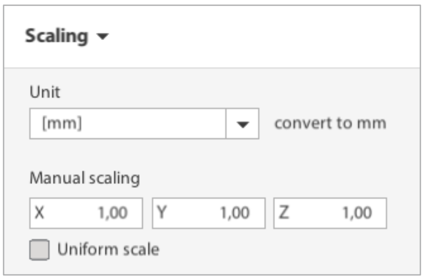
ⓘ With individual scaling, scaling factors can be entered separately for the X, Y and Z axes.Select the red circle to rotate the part around the Z-axis.

☑ The action step **Orientation** is selected.

1. Click on the part to be scales.
2. Click on the tab ***Part Option*** in the function panel.



3. Open the menu ***Scaling***.



4. Enter the desired factors by which the part is to be scaled in the input fields under **Manual Scaling**.
5. Enter the desired factor in the input field [X] to scale the part in the X direction.
6. Enter the desired factor in the [Y] input field to scale the part in the Y direction.
7. Enter the desired factor in the [Z] input field to scale the part in the Z direction.

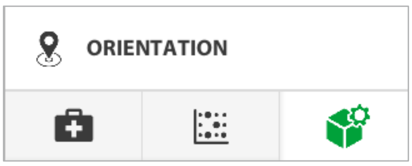
➲ The part is scaled according to the entered values.

Uniform scaling

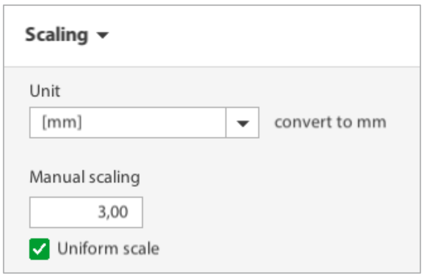
ⓘ With uniform scaling, a scaling factor is applied equally to the X, Y and Z axes.

☑ The **Orientation** action step is selected.

1. Click on the part to be scales.
2. Click on the tab ***Part Option*** in the function panel.



3. Open the menu ***Scaling***.



4. Set the check mark ☑ at **Uniform scale**.

   ⇨ The input fields for the X, Y and Z axes are combined in one field.

5. Enter the desired scaling factor.

➲ The part is scaled according to the entered value.

8.10 Automatic part placement

ⓘ From version 2021b of AM-Studio, it is possible to have parts automatically distributed on the substrate plate by an integrated algorithm.

Arrangement of parts with Auto arrange

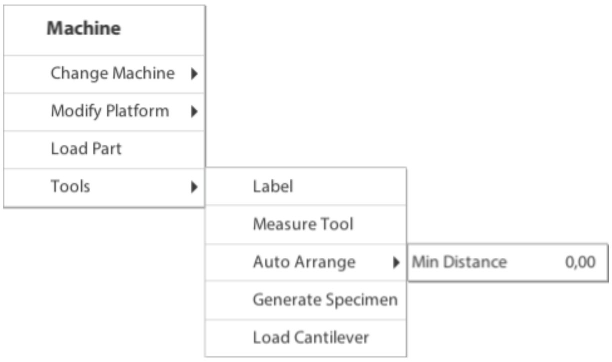
Note

The function "Auto arrange" uses the same algorithm as the automatic 2D nesting/packing.

☑ The **Orientation** action step is selected.

1. Click on the substrate plate with the right mouse button.

   ⇨ A context menu opens.



2. Navigate to **[Tools] [Auto arrange]** [**Min Distance]** in the context menu.

3. Click on **[Min Distance]** in the context menu and enter the desired minimum distance.

   The distance can be between

   - two parts
   - one part and the edge of the substrate plate
   - one part and a no-build zone

4. Click on **[Auto arrange]** in the context menu to start the process.
   ⇨ The algorithm tries to position all parts on the substrate plate to fit.

➲ The best possible arrangement of the parts is presented (regardless of whether all parts could be positioned on the substrate plate).

Note

If the entered values exceed the area of the substrate plate or if there is a danger of exceeding the area, a warning message is issued. With NO the process is aborted, with YES the process is continued.

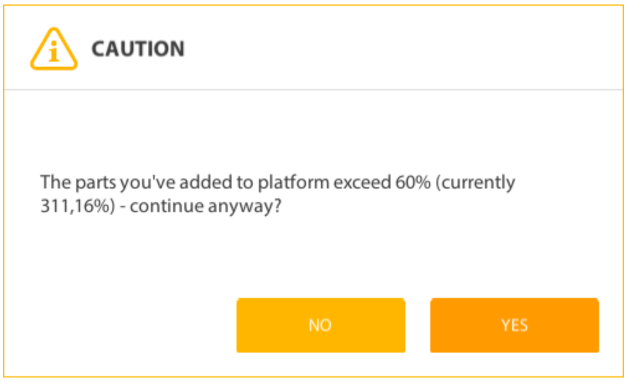


Fig. 7: *Warning for exceeded area*

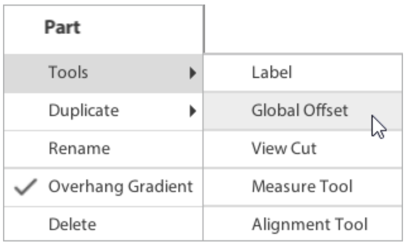
8.11 Allowance for a part

ⓘ In AM-Studio a global offset (Global offset) can be defined for all surfaces. This function can be used to compensate for the expected surface removal on parts caused by downstream processes, such as electro-chemical surface treatments.

Definition of a global allowance for a part

☑ The **Orientation** action step is selected.

1. Click on a part with the right mouse button.
   ⇨ A context menu opens.



2. Click on **[Tools] [Global offset]** in the context menu.
   ⇨ A window opens.

3. Enter the desired allowance to the part surface in millimetres in the input field.

Note

Alternatively to 3. the desired allowance can be defined in two other ways in 0.05 mm steps:

• with the arrows of the input field

• with the slider of the input field, which appears when the mouse pointer hovers over the activated input field.

4. Confirm the entry with a click on **[Apply]**.

➲ The part is provided with the desired offset.

Note

The Global offset function can only be applied globally to the entire part , not selectively to individual areas of the part.

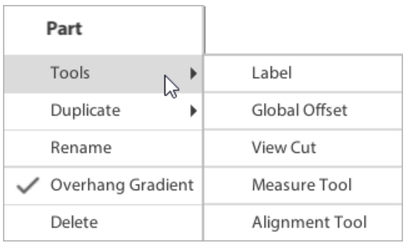
8.12 Measuring

ⓘ Distances and angles can be measured on the substrate plate and the part.

☑ The action step **Orientation** or **Viewer** is selected.

1. Click on a part with the right mouse button.

   ⇨ A context menu opens.



Note

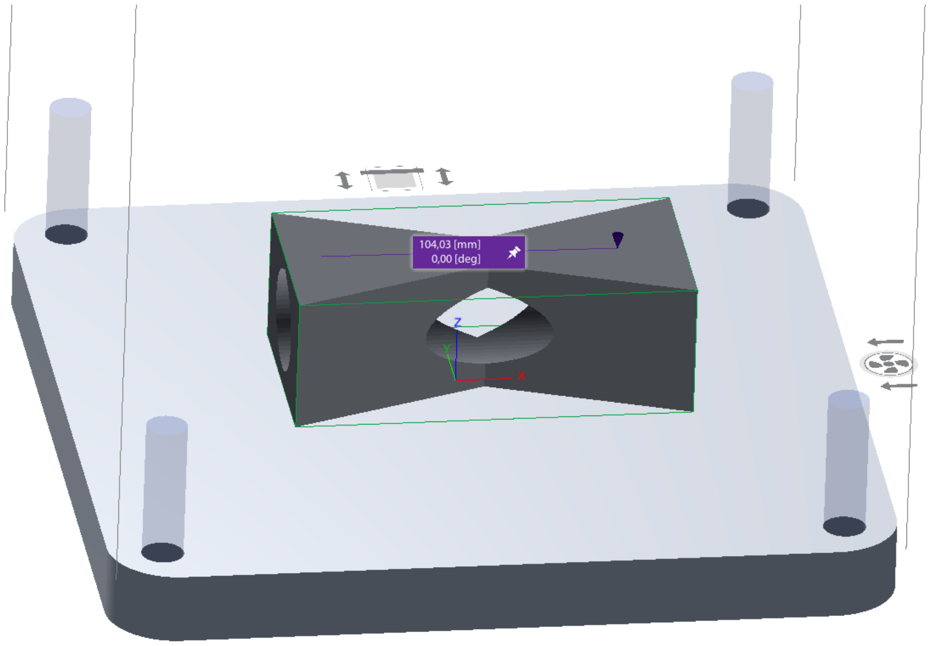
The measuring tool can also be accessed by right-clicking on the substrate plate. In the context menu that opens, also click on [Tools] [Measure Tool].

2. Click on **[Tools] [Measure Tool]** in the context menu.

   ⇨ The mouse pointer becomes a blue cone that automatically aligns itself according to the surface normal.

3. Select the starting point of the measurement with the blue cone and confirm with a mouse click.

4. Select the end point of the measurement with the blue cone and confirm with a mouse click.



➲ The distance and the angle between the 2 points are displayed.

Click on the icon to display the measurement.

Press the <Esc> key to close the Measure Tool.

Click on the icon to hide the measurement.

8.13 Density sample

ⓘ A test specimen can be generated to test the density of the building material. The height of the test specimen is based on the highest part in the build chamber.

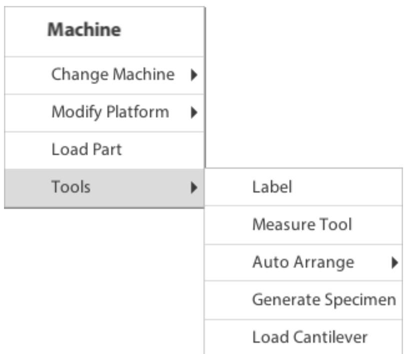
Note

The height of the test specimen is updated automatically when a new, higher part is loaded onto the substrate plate.

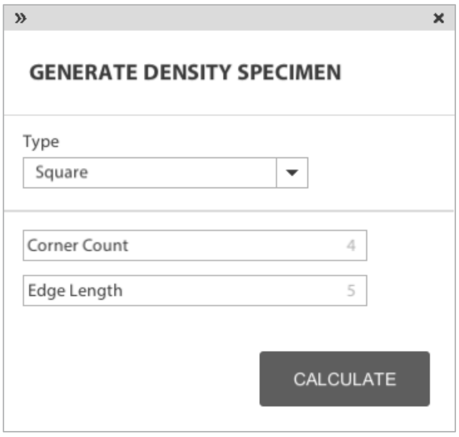
☑ The **Orientation** action step is selected.

1. Click on the substrate plate with the right mouse button.

   ⇨ A context menu opens.



2. Click on **[Tools] [Generate Specimen]** in the context menu.
   ⇨ A new window opens.



3. Select the desired shape of the specimen in the drop-down menu **Type**.

If **Square is selected:**

4. Enter the desired number of corners of the specimen in the input field **Corner Count**.
5. Enter the desired edge length in millimetres in the input field **Edge Length**.
6. Click **[Calculate]**.

➲ The generated sample is shown.

If **Circle is selected:**

7. Enter the desired number of corners of the specimen in the input field **Corner Count**.
8. Enter the desired radius in millimetres in the input field **Radius**.
9. Click **[Calculate]**.

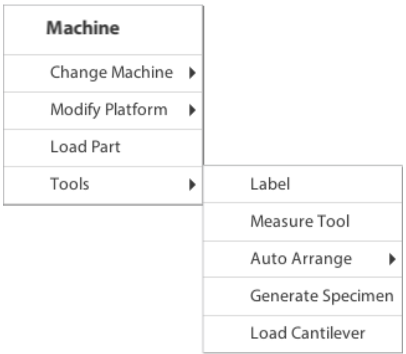
➲ The generated sample is shown.

8.14 Substrate plate labelling

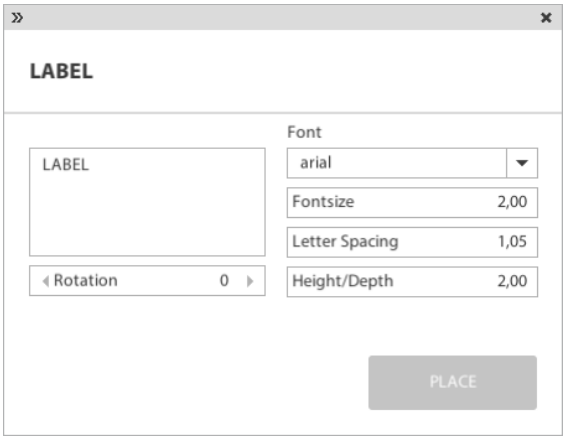
ⓘ With the substrate plate labelling function, a part consisting of text can be generated on the substrate plate. This function is useful to be able to clearly identify substrate plates later.

☑ The **Orientation** action step is selected.

1. Click on the substrate plate with the right mouse button.
   ⇨ A context menu opens.



2. Click on **[Tools] [Label]** in the context menu.
   ⇨ A window opens.



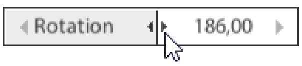
   ⇨ The mouse pointer becomes a blue cone.
3. Enter the text to be generated on the substrate plate in the **Label** input field.
4. Select the font from the drop-down menu **Font** .
5. Enter the font size in the input field **Fontsize**.
6. Enter the spacing between the individual letters in the input field **Letter Spacing**.
7. Enter the height of the font in the input field **Height/Depth**.
8. Enter the desired rotation angle of the text in the **Rotation**.

Note

As an alternative to 8., the desired rotation can be set in two other ways in 0.05 mm steps:

- *with the arrows of the input field*

- *with the slider of the input field, which appears when the mouse pointer hovers over the activated input field.*



9. Navigate with the mouse to the position where the text is to be generated.
10. Confirm the position with a mouse click.
    ⇨ A preview of the text on the substrate plate is displayed.
    ⇨ The **[PLACE]** button turns green and is thus enabled.
11. Click the **[PLACE] .**

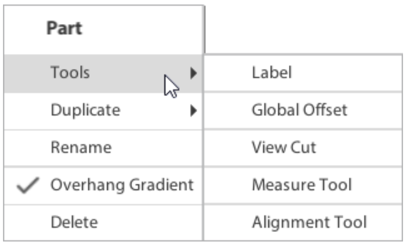
➲ The text is generated as a part.

8.15 Part labelling

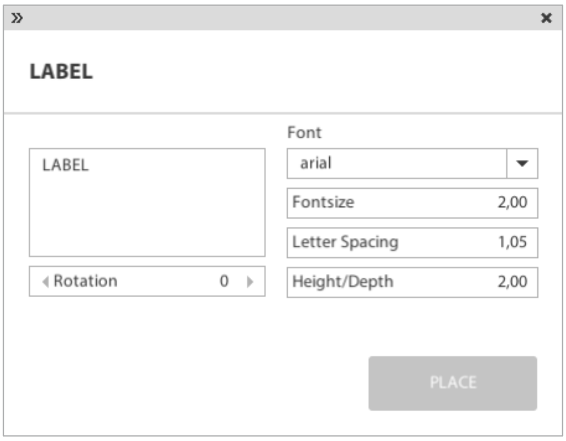
ⓘ Parts can be labelled with the "Label" function. A text is imprinted on the part.

☑ The **Orientation** action step is selected.

1. Click with the right mouse button on the part to be labelled.
   ⇨ A context menu opens.



2. Click on **[Tools] [Label]** in the context menu.
   ⇨ A window opens.



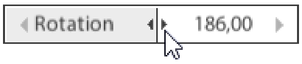
   ⇨ The mouse pointer becomes a blue cone.
3. Enter the text to be generated on the part in the **Label** input field.
4. Select the font from the drop-down menu **Font**.
5. Enter the font size in the input field **Fontsize**.
6. Enter the spacing between the individual letters in the input field **Letter Spacing**.
7. Enter the height or depth of the font in the input field **Height/Depth**.
   ⇨ A positive value creates a raised font.
   ⇨ A negative value creates an embossed font.
8. Enter the desired rotation angle of the text in the **Rotation**.

Note

As an alternative to 8., the desired rotation can be set in two other ways in 0.05 mm steps:

- *with the arrows of the input field*

- *with the slider of the input field, which appears when the mouse pointer hovers over the activated input field.*



9. Navigate with the mouse to the position where the text is to be created.
10. Confirm the position with a mouse click.
    - ⇨ A preview of the text on the part is displayed.
    - ⇨ The **[Emboss]** or **[Engrave]** button turns green and is thus enabled.

11 Click on the **[Emboss]** or **[Engrave]** button.

➲ The part is labelled.

Note

The labelling process can change the polygon mesh of the part in such a way that errors occur.
After the labelling process, the part should be checked and healed if necessary.
On highly curved surfaces, labelling is usually not possible.

8.16 Cantilever

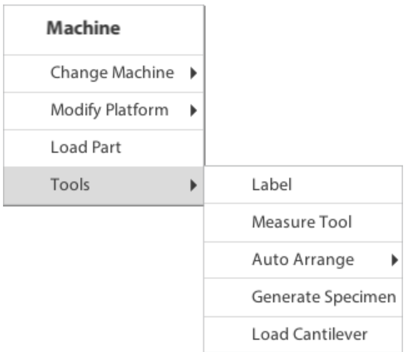
ⓘ In the Orientation action step, a Cantilever can be loaded onto the substrate plate. With the help of a printed Cantilever, the warping of a part can be measured. This value will be needed to calibrate the machine for the simulation and determine the pre-deformation of a part.

Fig. 8: Cantilever

Loading a cantilever onto the substrate plate

☑ The **Orientation** action step is selected.

1. Click on the substrate plate with the right mouse button.
   ⇨ A context menu opens.



2. Click on **[Tools] [Load Cantilever]** in the context menu.

➲ The cantilever is loaded onto the substrate plate.

Note

You can find information about the exact position of the cantilever on the substrate plate in the "Simulation Calibration" PDF file. This can be downloaded from the menu Settings > Machines > Add Machine > Simulation Calibration

9 STEP 3: SUPPORT

This chapter lists information on the **Support** action step, in which support geometries (Supports) are created.

Fig. 9: *Progress bar: Support action step*

Note

> *Support geometries for multiple parts can be generated at the same time (bulk support). To do this, multiple parts must be selected before you can click on [Generate].*

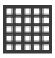
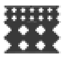
9.1 Support types

This chapter provides general information on the types of support geometry that can be generated with AM-Studio. Support geometries serve as a support between the part and the substrate plate or between part areas and the substrate plate. After completion of a part, the support geometries are usually removed again.

Note

> *Support geometries essentially have two tasks: On the one hand, they serve to securely fix the part in the build chamber, and on the other hand, they dissipate the resulting process heat into the build chamber floor (depending on the additive manufacturing process). Therefore, it is essential for a successful building process to use suitable support geometries.*

For the field of selective laser melting (SLM process), AM-Studio offers five different support types.

Icon	Support Type
	Rod Support
	Block Support
	Adaptive Cell Support
	Line Support
	Tree Support

Tab. 8: *Support types for the SLM process*

Rod Support

Rod supports are supports in the form of rods that are distributed differently over the area to be supported, depending on the setting.

Note

> *This type of support geometry is particularly beneficial on free-form surfaces (e.g. dental implants), as they follow the part surface very well due to the implemented samplers and are easier to remove after completion than block supports, for example.*

Block Support

Block supports are supports in the form of a grid consisting of regularly arranged rectangles or squares. The size of the rectangles or squares can be freely selected.

Note

> *This type of support is one of the oldest support geometries in the SLM process and is particularly suitable for flat support regions that are close to the process chamber floor.*

Adaptive Cell Support

Adaptive Cell Supports are supports in a cubic pattern. The size of the cells decreases at definable distances from the substrate plate towards the part. The reduction of the cells is made possible by a cross-shaped wall within the support structure.

Note

> *This type of support was developed especially for the SLM process, because the large cells make it much easier to remove powder than would be the case with block supports, for example. In addition, the volume and thus the amount of powder required is also much lower. This also makes this type of support more cost-efficient.*
> *Adaptive Cell Support has proven to be particularly efficient as a support geometry for very tall support structures, as this is where the advantages of the large cells are best utilised.*
> *This type of support is also preferable in conjunction with the chemical-electrochemical polishing process, Hirtising® (https://www.rena.com/de/technologie/prozesstechnologie/hirtisieren), as it has lower process times than the other support geometries.*

Line Support

Line supports are supports in the form of a thin wall along the centre of the area to be supported. This wall is usually crossed by several supporting walls (struts) at the transition to the substrate plate to increase stiffness.

Note

> *This type of support can be applied not only to individual lines or sharp edges, but also to surfaces. This results in a skeletonisation of the region. Since this type of support also enables the exact support particularly efficiently, edges remain intact and there is no rounding of the part geometry.*

Tree Support

Tree supports are supports in the shape of a tree. The lower part of the tree support represents the trunk and the upper part of the tree support represents the branching branches of a tree.

Note

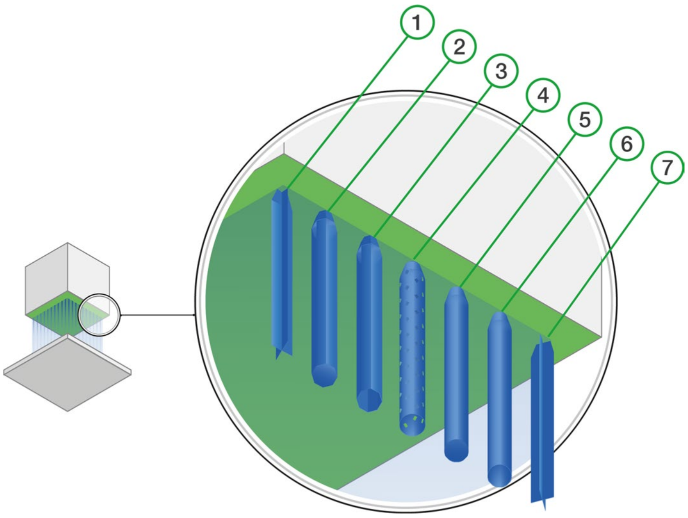
> *This type of support is particularly efficient in connection with topology-optimised/bionic part geometries, as it has less volume due to the many branches and therefore also incurs less costs. It can still be passed by parts very well, should a part-to-part support not be target-oriented.*

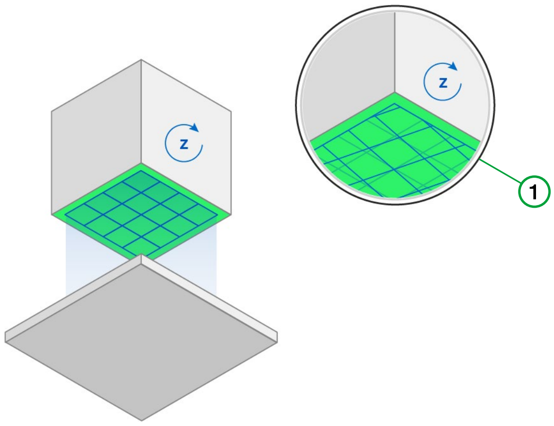
9.1.1 Rod Support

ⓘ This chapter contains information on the parameters that are defined under Rod Support in the action step **Support**.

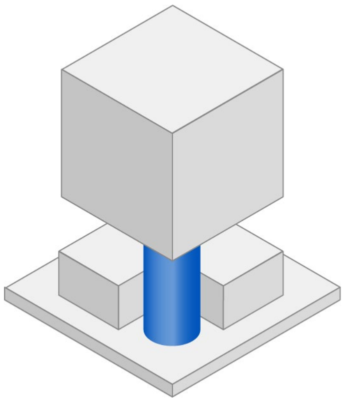
9.1.1.1 Rod Settings

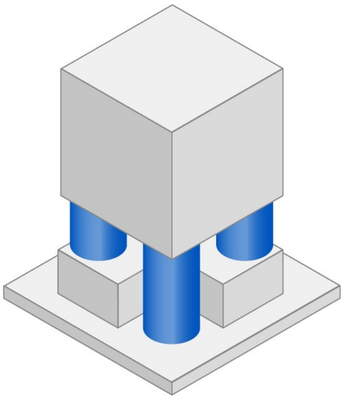
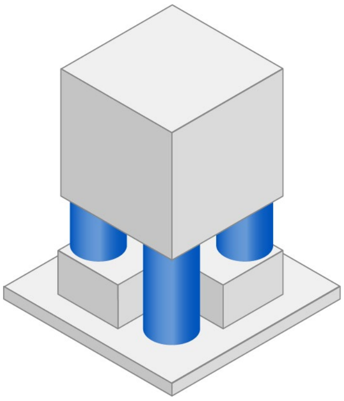
The following parameters are available to define the rod support geometry:

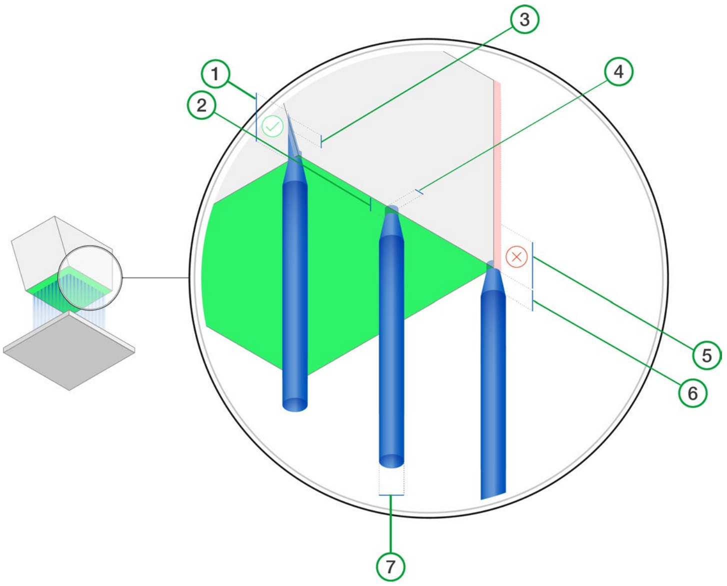
Parameter	Description
General	
Rod type	 **1** crossrod **2** solidoctagonrod **3** octagonrod **4** perftube **5** solidroundrod **6** roundrod **7** xrod Allows the selection of different rod types. • Crossrod: The individual Rod Supports have the shape of an X. The intersecting surfaces are aligned in the x and y directions. • Solidoctagonrod: The individual rod supports have the shape of a filled eight-sided prism. • Octagonrod: The individual Rod Supports have the shape of a hollow eight-sided prism. • Perftube: The individual Rod Supports have the shape of a hollow cylinder. The outer surfaces are perforated. • Solidroundrod: The individual rod supports have the shape of a filled cylinder. • Roundrod: The individual Rod Supports have the shape of a hollow cylinder. The outer surfaces are not perforated. • Xrod: The individual Rod Supports have the shape of an X. The intersecting surfaces are at a 45° angle to the x and y directions.
Rod Diameter	Defines the diameter of the individual rods of the Rod Supports.

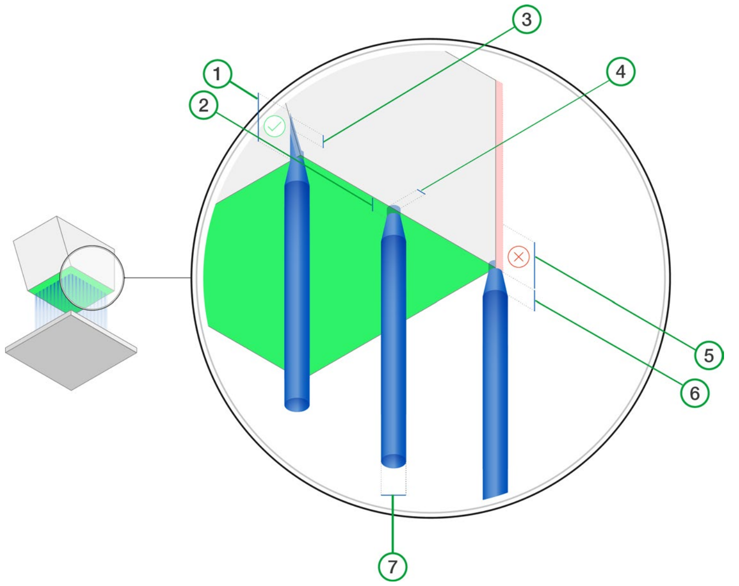
<table>
<tr><td>Rotation</td><td>

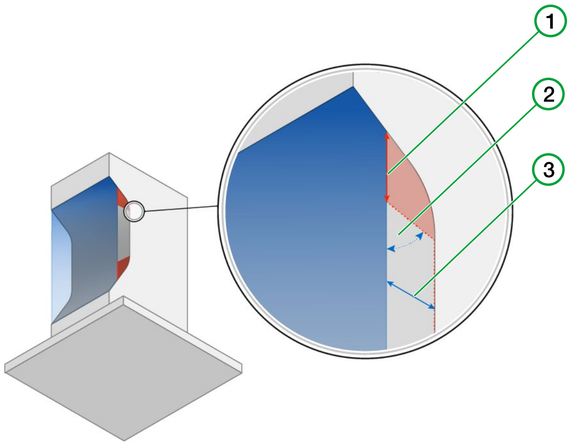
1 Rel. Gas Flow Angle

If this parameter is activated, it is possible to define a rotation of the supports relative to the gas flow direction.
Rel. Gas Flow Angle [°]: Defines the angle for the rotation of the supports. For example, an angle of 0° means that the supports are oriented in the direction of the gas flow.</td></tr>
<tr><td colspan="2">Build Support</td></tr>
<tr><td>Build on Platform</td><td>
If this parameter is activated, supports are generated between the part and the substrate plate.</td></tr>
</table>

Build on Part	 If this parameter is activated, supports are generated between part areas.
Combined	 This parameter is enabled by activating **Build on Platform** and **Build on Part**.

Tip Attachment	
	 **1** Max Extension Height **2** Intrusion **3** Intrusion **4** Tip Scale x RD **5** Max Extension Height **6** Relative Height x RD **7** Rod Diameter (RD)
Relative Height	Defines how long the tapered area at the tip of the rods is.
Tip Scale	Defines the scaling factor for the diameter of the tip adjacent to the part. Diameter = Scaling factor * Rod Diameter
Intrusion	Defines how far the support is to be built into the part.
Max Extension Height	If the rod is built at an edge where there is no intersection area parallel to the substrate plate, the rod can be built up to the **Max Extension Height**, as long as it meets the geometry.

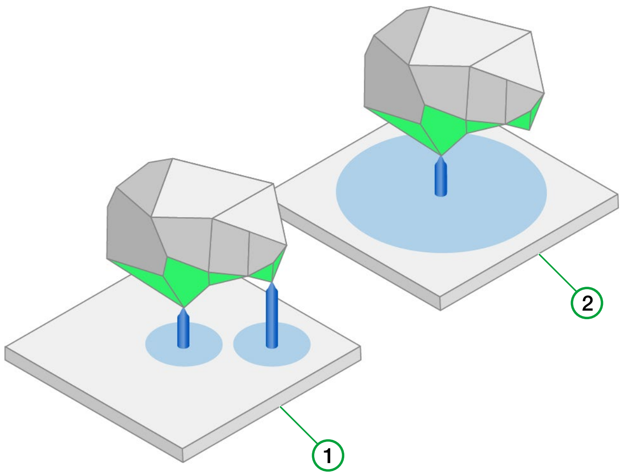
Base Attachment	
	 **1** Max Extension Height **2** Intrusion **3** Intrusion **4** Tip Scale x RD **5** Max Extension Height **6** Relative Height x RD **7** Rod Diameter (RD) Attention! This option is only relevant for supports that extend from the part to the same or another part. Support structures that extend onto the substrate plate are not affected by these parameters.
Relative Height	Defines how long the tapered area at the tip of the rods is.
Tip Scale	Defines the scaling factor for the diameter of the tip adjacent to the part. Diameter = Scaling factor * Rod Diameter
Intrusion	Defines how far the support is to be built into the part.
Max Extension Height	If the rod is built at an edge where there is no intersection area parallel to the substrate plate, the rod can be built up to the **Max Extension Height**, as long as it meets the geometry.

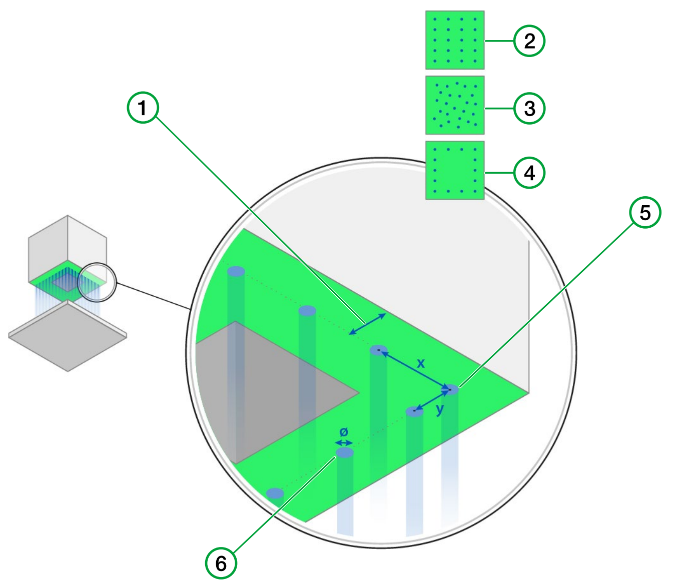
Check Part Clearance	
Check Part Clearance	 **1** Vertical Offset **2** Inclination **3** Part Clearance If this parameter is activated, parameters for the minimum distance between a vertical wall and supports can be defined. This can prevent parts and supports from colliding.
Clearance	Defines the minimum distance between a vertical wall and the supports.
Collision Volume	• **Vertical Offset**: Defines the vertical distance from a part area where the "Clearance" function is not applied. This creates a fictional wall between the part and the area where the "Clearance" function is applied. • **Inclination**: Defines the angle of the fictional wall from which the "Clearance" function is not applied.

Tab. 9: Rod Support: Rod Settings

9.1.1.2 Rod Point Sampler

The following parameters are available to define the actual rod geometry:

Parameter	Description
Min Point Sampler	
Min Point Sampler	 **1** Min Point Sampler **2** Local Domain Size If this parameter is activated, Rod Supports are generated in a local minimum of an area to be supported. This prevents geometries from being created loosely in the powder. The local minimum is an area and not a geometric point. Depending on the parameters selected, a single rod support or a series of rod supports in a Poisson distribution around the centre of gravity is generated at the centre of gravity of this area.
Min Distance To Outer Perimeter	Defines the minimum distance between the Rod Supports and the outer contour.
Seed	Defines the starting value for the first distribution calculation. This initial value, together with a random value, gives the next value of the Poisson distribution.
Min Distance Between Points	Defines the minimum distance between the individual rods of the Rod Supports.
Max Diagonal For Single Point Plateau	Defines the size of the local minimum area above which no single rod support supports the area, but a series of Poisson distributed rod supports.
Min Minima Fraction For Plateau	Defines how many points of the polygon grid may lie below or above the surface level and the surface still counts as a local minimum.
Max Plataeu Gradient	Defines the angle in degrees that the surface may be tilted and still be considered a local minimum.

Sample Large Plateaus	If this parameter is activated, the Min Point Sampler is applied to minimum surfaces. • **Min Distance Between Points On Plateau:** Defines the spacing of the rods on the minimum surface. • **Max Plateau Area Ratio**: Defines a ratio between the sides of the plateau that must be met to generate supports. • **Max Plateau BB Short Side**: Defines the maximum side length of the bounding box of the minimum area.
Additional Sampling Algorithms	
	 **1** Min Distance To Contour **2** Regular Grid Sampler **3** Poisson Disk Sampler **4** Contour Sampler **5** Grid Cell Size X Distance Between Points **6** Rod Diameter
Contour Sampler	
Contour Sampler	If this parameter is activated, **Rod Supports** are generated along the edges of the area to be supported.
Min Distance To Outer Perimeter	Defines the minimum distance between the **Rod Supports** and the outer contour.
Distance Between Points	Defines the minimum distance between the individual rods of the **Rod Supports**.
Regular Grid Sampler	
Regular Grid Sampler	If this parameter is activated, bars of the Rod Supports are generated at regular intervals for the selected area to be supported. A fictitious cell is created for each rod of the Rod Supports, the size of which can be

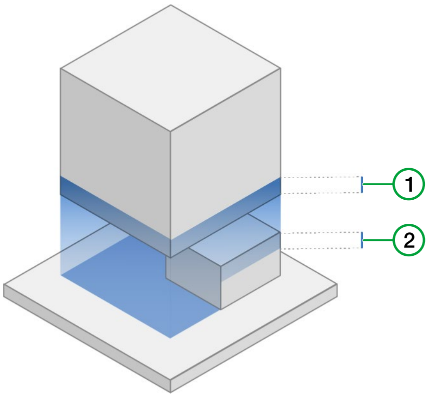
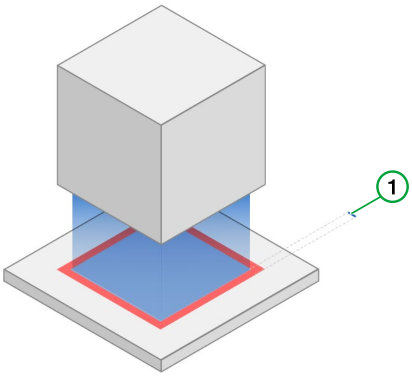
	adjusted. In this way, the minimum distance between the rods can be defined.
Min Distance To Outer Perimeter	Defines the minimum distance between the Rod Supports and the outer contour.
Grid Cell Size X	Defines the width of the fictional cells in the X-direction.
Grid Cell Size Y	Defines the length of the fictional cells in the Y-direction.
Check Distance To Input Points	If this parameter is activated, existing rods of other sampling alogrithms are taken into account when applying the Regular Grid Sampler. • **Min Dist To Input Points**: Defines the distance to existing rod supports.
Poisson Disk Sampler	
Poisson Disk Sampler	If this parameter is activated, Rod Supports are statistically generated and distributed based on a Poisson distribution for the selected area to be supported. The distribution calculation is based on the calculation of a random value with all random values from previous calculations. The resulting value in turn is used for the subsequent distribution calculation. 1. Calculation: Random value + value under **Seed** = value 1 2. Calculation: Random value + value 1 = value 2 3. Calculation: Random value + value 1 + value 2 = value 3.
Min Distance To Outer Perimeter	Defines the minimum distance between the Rod Supports and the outer contour.
Min Distance Between Points	Defines the minimum distance between the individual rods of the Rod Support.
Seed	Defines a starting value for the first distribution calculation. This initial value together with a random value yields the first Poisson distribution. With this function, the same result can be guaranteed with multiple assignments of the Poisson distribution.

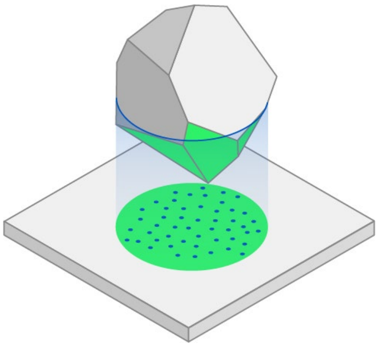
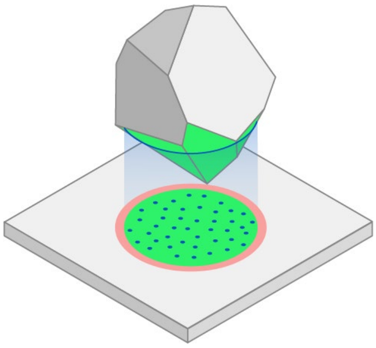
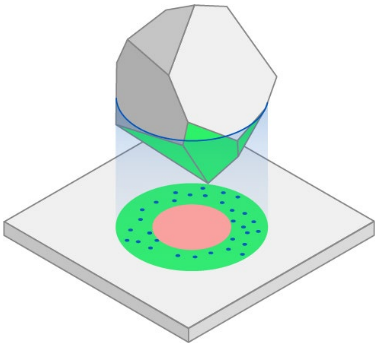
Tab. 10: Rod Support: Rod Point Sampler

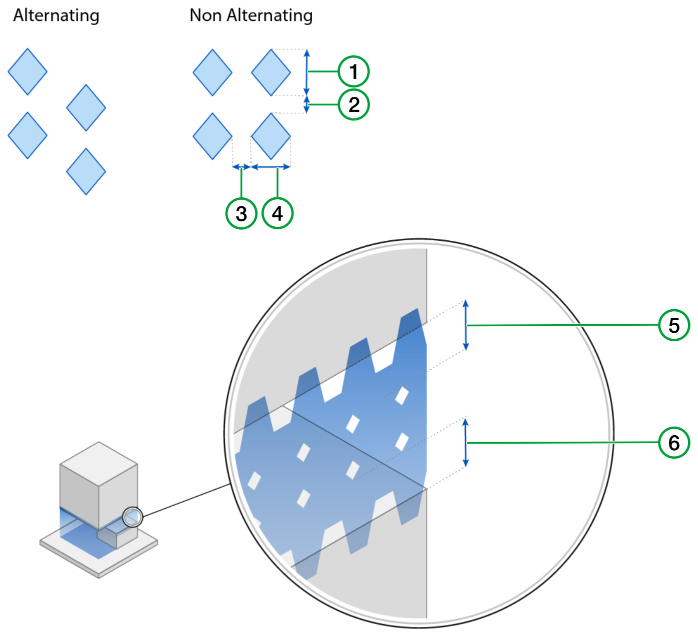
9.1.1.3 Contour

ⓘ A Contour is an additional support wall that encloses other support structures.

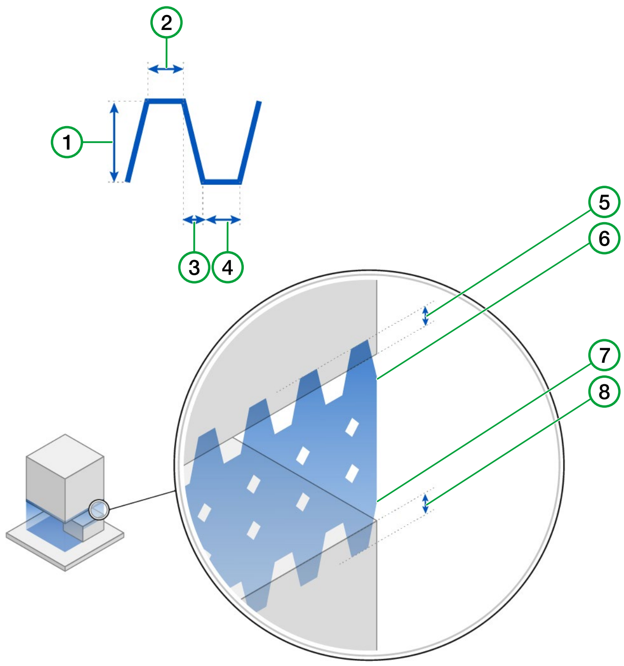
In the ***Contour*** menu, the following parameters can be defined to define the Contour geometry:

Parameter	Description
General	
Build Contour	When this parameter is activated, the selected support structure is rein-forced with a Contour at the outer edge.
	 **1** Intrusion Top **2** Intrusion Bottom
Top Intrusion	Defines how far the supports of the Contour should penetrate the part from the bottom.
Bottom Intrusion	Defines how far the supports of the Contour should penetrate the part from the top.
Offset	 **1** Offset Defines the distance of the Contour to the outer edge of the support re-gion.

Parameter	Description
Smooth Contour	
Smooth Contour	 If this parameter is activated, the sharp edges of the support structure are smoothed.
Max Support Angle	 Defines the angle between the part and the substrate plate up to which supports are generated. The distance between the support with Smooth Contour and the outer area of the support region can be increased. As a result, the area with a support structure becomes smaller.
Min Support Angle	 Defines the angle between the part and the substrate plate from which supports are generated.

Parameter	Description
Contour Perforation Pattern	
	 **1** Height **2** Height Clearance **3** Width Clearance **4** Width **5** Top Solid Height **6** Bottom Solid Height

Parameter	Description
Contour Perforation Pattern	If this parameter is activated, a perforation pattern can be defined for the Contour. The following parameters can be defined: • **Type**: Allows the selection between different shapes of the perforation (currently only "Simple Diamond"). • **Width**: Defines the width of the perforation. • **Height**: Defines the height of the perforation. • **Height Clearance**: Defines the vertical distance between perforations (height distance). • **Width Clearance**: Defines the horizontal distance between perforations (width distance). • **Border Clearance**: Defines the distance of the perforation to corners. ⓘ A weakening of the corners by perforations can thus be avoided. • **Top Solid Height**: Defines the distance between the part and the top row of perforations measured at the highest point of the perforation. • **Bottom Solid Height**: Defines the distance between the part and the lowest row of perforations measured at the lowest point of the perforation. • **Alternate Diamonds**: Enables/disables an offset of the perforations from each other.

<table>
<tr><th>Parameter</th><th>Description</th></tr>
<tr><td colspan="2">Tooth Pattern</td></tr>
<tr><td></td><td>


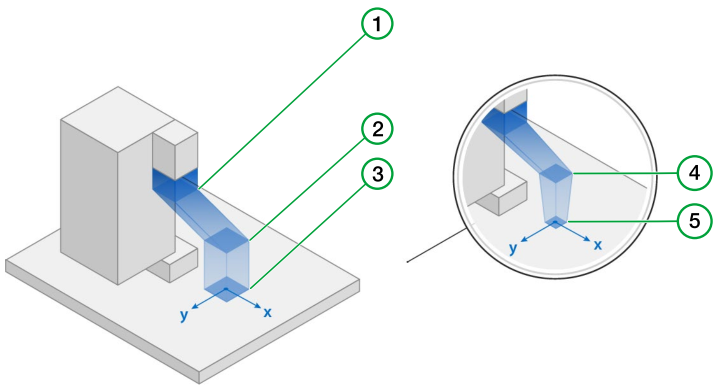
1 Height
2 Top Width
3 Crossing Width
4 Bottom Width
5 Top Intrusion
6 Top Teeth
7 Bottom Teeth
8 Bottom Intrusion
If parameters under Tooth Pattern are activated, depending on the setting, recesses in the form of teeth are generated in a Contour. These tooth patterns can be generated at the upper and/or lower edge.
</td></tr>
<tr><td>Top And Bottom Equal</td><td>If this parameter is activated, the structure of the teeth for upper and lower structures of the Contour is identical. The values are taken from the settings of the Top Tooth Pattern.</td></tr>
</table>

<table>
<tr><th>Parameter</th><th>Description</th></tr>
<tr><td>Top Tooth Pattern</td><td>If this parameter is activated, the following values can be defined for the structure of the upper row of teeth.
Bottom Width: Defines the bottom width of the teeth.
Crossing Width: Defines the distance between the end point of the top tooth width (Top Width) and the start point of the bottom tooth width (bottom width).
Top Width: Defines the top width of the teeth.
Height: Defines the tooth height between the part and the support plus the value of the intrusion.
Note
The values of the intrusion are defined in the menu Contour.</td></tr>
<tr><td>Bottom Tooth Pattern</td><td>If this parameter is activated, the following values can be defined for the structure of the lower row of teeth.
Bottom Width: Defines the bottom width of the teeth.
Crossing Width: Defines the distance between the end point of the top tooth width (Top Width) and the start point of the bottom tooth width (bottom width).
Top Width: Defines the top width of the teeth.
Height: Defines the tooth height between the part and the support plus the value of the intrusion.
Note
The values of the intrusion are defined in the menu Contour.</td></tr>
</table>

Tab. 11: Rod Support: Contour

Deformation

ⓘ Settings for the deformation of supports can be defined with the **Deformation** function. Vertically aligned supports can be moved using two planes.

Parameter	Description
	 **1** Upper Plane Z **2** Lower Plane Z **3** Platform Translation **4** Lower Plane Scale **5** Platform Scale
Activate	Activates the "Deformation" function.
Max Deformation Angle	Defines the maximum deformation angle of the support geometry.
Upper Plane Z	Defines the distance between the substrate plate and the top of the support area to be angled.
Lower Plane Z	Defines the distance between the substrate plate and the lower end of the support area that is to be angled.
Platform Translation X	Defines the offset of the lower area of the supports relative to the upper area of the supports in the X-direction.
Platform Translation Y	Defines the offset of the lower area of the supports relative to the upper area of the supports in the X-direction.
Lower Plane Scale	Defines a scaling factor to reduce/increase the support footprint at the level of the lower plane.
Platform Scale	Defines a scaling factor to reduce/increase the support footprint on the substrate plate.

Tab. 12: Rod Support: Deformation

Note

Besides entering the values manually, a deformation can also be performed interactively in the 3D view. The necessary command can be found in the context menu of the respective support structure. The tool for editing the deformation is visible when the Deformation function is activated. The positions of the Upper Plane and the Lower Plane can be defined by using the two arrows. The plane symbol moves the support along the X-position and Y-position on the substrate plate.

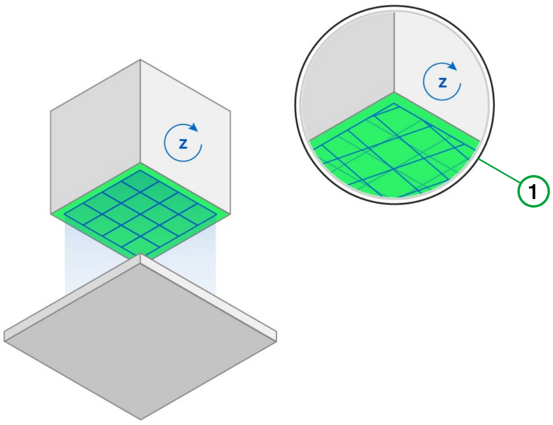
> *However, the angle for the maximum deformation and the values for the scaling of the support base planes – Lower Plane Scale and Platform Scale – must be set in the input fields.*

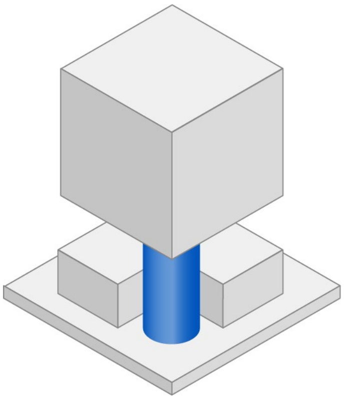
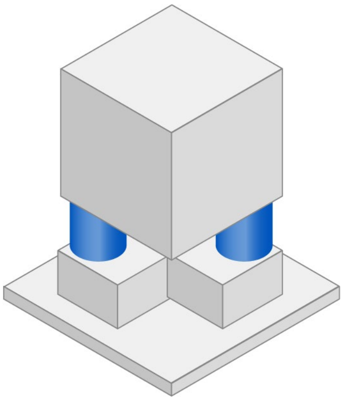
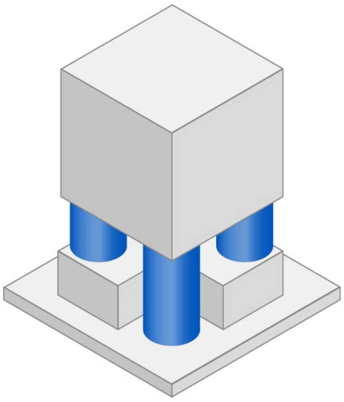
9.1.2 Block support

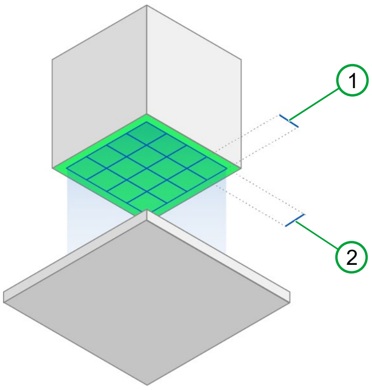
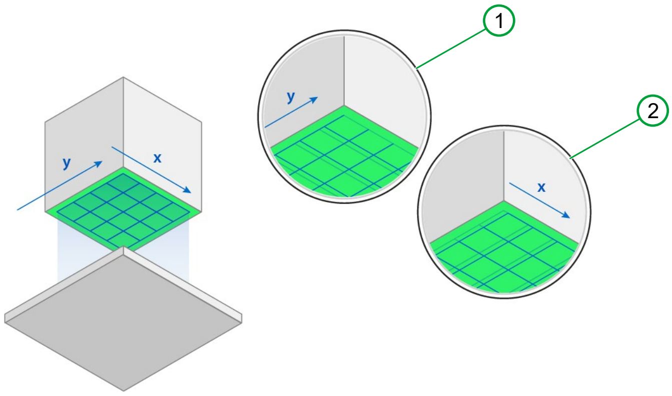
ⓘ This chapter contains information on the parameters that are defined under Block Support in the action step **Support**.

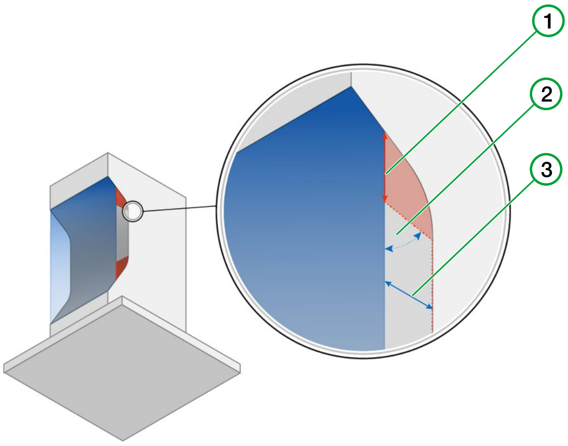
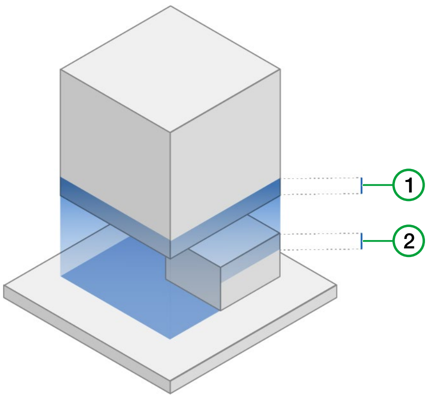
9.1.2.1 Block settings

The following parameters are available for defining the block support geometry:

Parameter	Description
General	
Rotation	 **1** Rel. Gas Flow Angle If this parameter is activated, it is possible to define a rotation of the supports relative to the gas flow direction. • **Rel. Gas Flow Angle [°]**: Defines the angle for the rotation of the supports. For example, an angle of 0° means that the supports are oriented in the direction of the gas flow.

Parameter	Description
Build Support	
Build on Platform	 If this parameter is activated, supports are generated between the part and the substrate plate.
Build on Part	 If this parameter is activated, supports are generated between part areas.
Combined	 This parameter is activated by activating **Build on Platform** and **Build on Part.**

Parameter	Description
Block Spacing	
	 **1** X Spacing **2** Y Spacing
X Spacing	Defines the size of the block cells in X-direction.
Y Spacing	Defines the size of the block cells in Y-direction.
Shift	
	 **1** Y Shift **2** X Shift
X Shift	Defines the displacement of the block cells in X-direction.
Y Shift	Defines the displacement of the block cells in Y-direction.

Parameter	Description
Check Part Clearance	
Check Part Clearance	 **1** Vertical Offset **3** Part Clearance **2** Inclination If this parameter is activated, parameters can be defined for the minimum distance between a vertical wall and supports. This can prevent parts and supports from colliding.
Clearance	Defines the minimum distance between a vertical wall and supports.
Collision Volume	• **Vertical Offset:** Defines the vertical distance from a part area where the Clearance function is not applied. This creates a fictional wall between the part and the area where the "Clearance" function is applied. • **Inclination:** Defines the angle of the fictional wall from which the "Clearance" function is not applied.
Intrusion	
	 **1** Intrusion Top **2** Intrusion Bottom
Top	Defines how far the supports should penetrate the part from below.

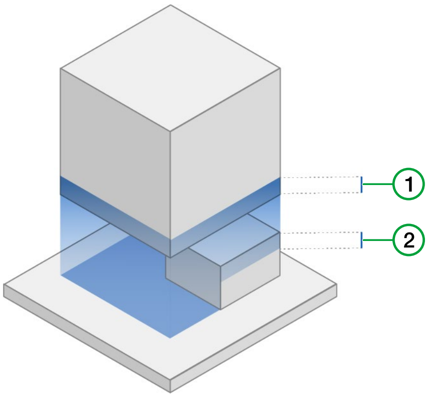
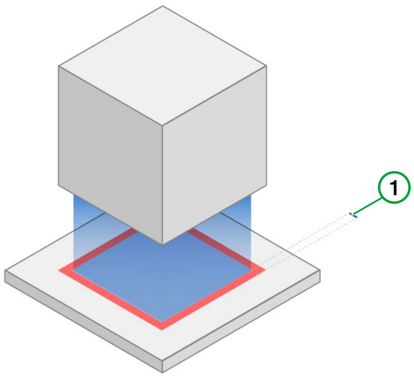
Parameter	Description
Bottom	Defines how far the supports should penetrate the part from above.

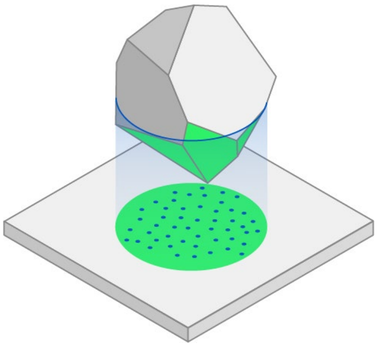
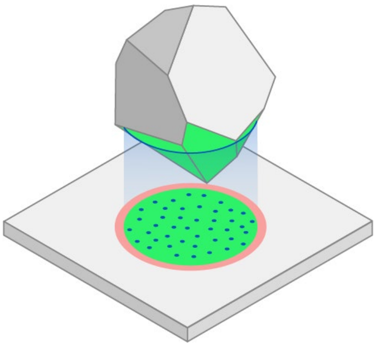
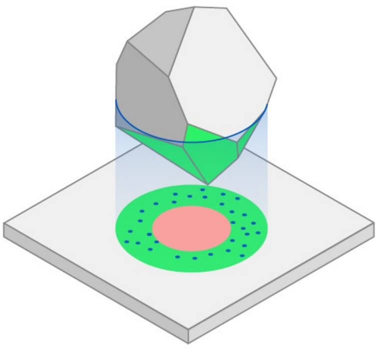
Tab. 13: Block Support: Block Settings

9.1.2.2 Contour

ⓘ A Contour is an additional support wall that encloses other support structures.

In the ***Contour*** menu, the following parameters can be defined to define the Contour geometry:

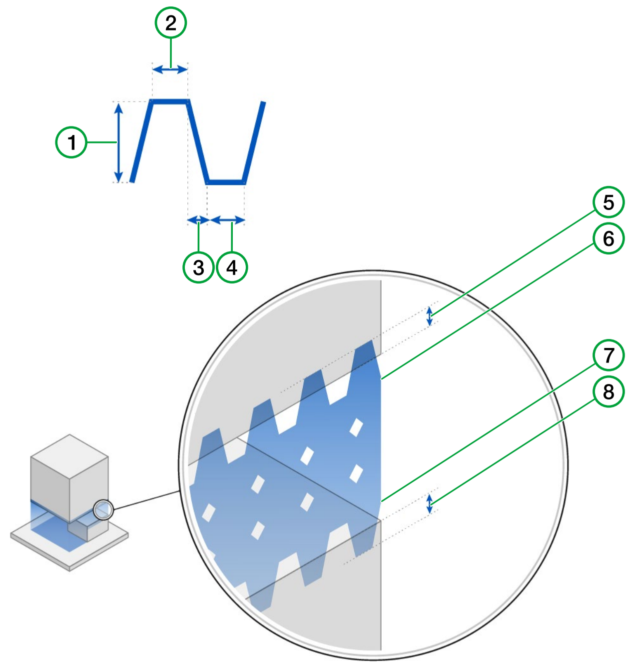
Parameter	Description
General	
Build Contour	When this parameter is activated, the selected support structure is reinforced with a Contour at the outer edge.
	 **1** Intrusion Top **2** Intrusion Bottom
Top Intrusion	Defines how far the supports of the **Contour** should penetrate the part from the bottom.
Bottom Intrusion	Defines how far the supports of the **Contour** should penetrate from above into an underlying part.
Offset	 **1** Offset Defines the distance of the Contour to the outer edge of the support region.

Parameter	Description
Smooth Contour	
Smooth Contour	 If this parameter is activated, the sharp edges of the support structure are smoothed.
Max Support Angle	 Defines the angle between the part and the substrate plate up to which supports are generated. The distance between the support with **Smooth Contour** and the outer area of the support region can be increased. As a result, the area with a support structure becomes smaller.
Min Support Angle	 Defines the angle between the part and the substrate plate from which supports are generated.

Tab. 14: Block Support: Contour

9.1.2.3 Tooth Pattern

ⓘ If parameters in ***Tooth Pattern*** are activated, depending on the setting, recesses in the form of teeth are generated in a Contour. These tooth patterns can be generated at the upper and/or lower edge.

<table>
<tr><th>Parameter</th><th>Description</th></tr>
<tr><td></td><td>


<table>
<tr><td>1 Height</td><td>5 Top Intrusion</td></tr>
<tr><td>2 Top Width</td><td>6 Top Teeth</td></tr>
<tr><td>3 Crossing Width</td><td>7 Bottom Teeth</td></tr>
<tr><td>4 Bottom Width</td><td>8 Bottom Intrusion</td></tr>
</table>
</td></tr>
<tr><td>Top And Bottom Equal</td><td>If this parameter is activated, the structure of the teeth for upper and lower structures of the Contour is identical. The values are taken from the settings of the Top Tooth Pattern.</td></tr>
<tr><td>Top Tooth Pattern</td><td>If this parameter is activated, the following values can be defined for the structure of the upper row of teeth.

Bottom Width: Defines the bottom width of the teeth.
Crossing Width: Defines the distance between the end point of the top tooth width (Top Width) and the start point of the bottom tooth width (bottom width).
Top Width: Defines the top width of the teeth.
Height: Defines the tooth height between the part and the support plus the value of the intrusion.

Note
The values of the intrusion are defined in the menu Contour.
</td></tr>
</table>

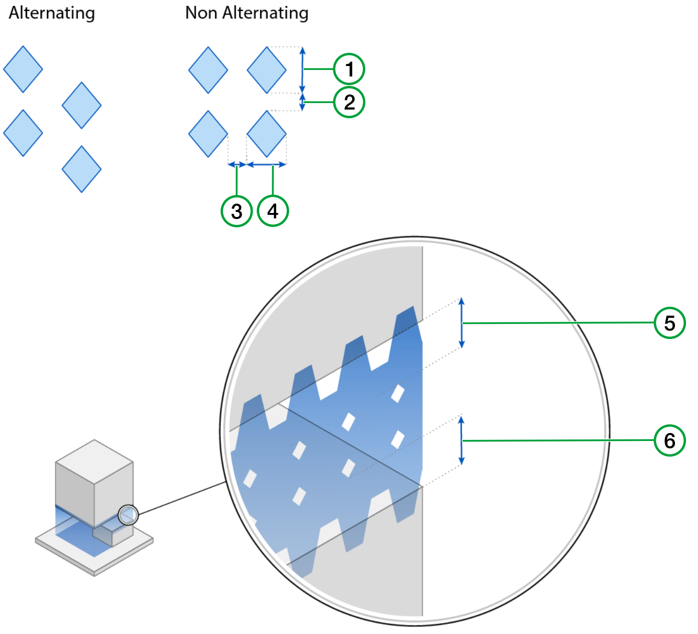
Parameter	Description
Bottom Tooth Pattern	If this parameter is activated, the following values can be defined for the structure of the lower row of teeth. • **Bottom Width:** Defines the bottom width of the teeth. • **Crossing Width:** Defines the distance between the end point of the top tooth width (Top Width) and the start point of the bottom tooth width (Bottom Width). • **Top Width:** Defines the top width of the teeth. • **Height:** Defines the tooth height between the part and the support plus the value of the intrusion. **Note** *The values of the intrusion are defined in the menu Contour.*

Tab. 15: Block Support: Tooth Pattern

9.1.2.4 Perforation

ⓘ A Contour can be perforated at Block Supports. This perforation allows e.g. loose powder to flow out of the support geometry better.

When the function is activated, the following parameters can be defined regarding the perforation of the Contour:

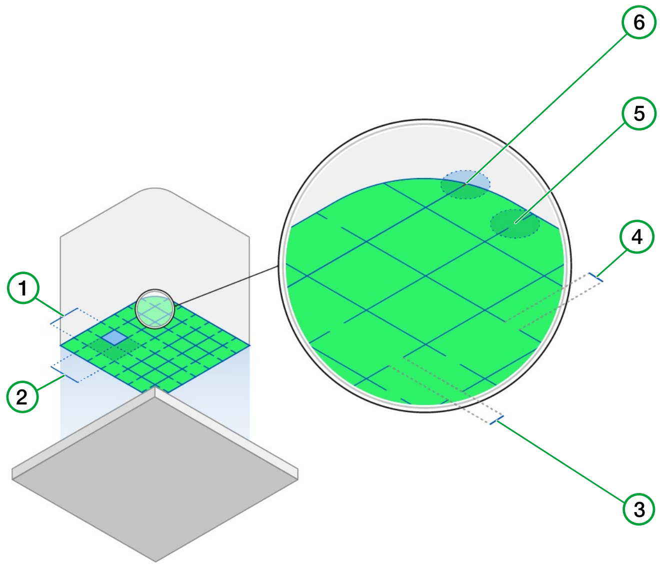
Parameter	Description
	 **1** Height **2** Height Clearance **3** Width Clearance **4** Width **5** Top Solid Height **6** Bottom Solid Height

Parameter	Description
Perforation	If this parameter is activated, a perforation pattern can be defined for the Contour. The following parameters can be defined: • **Type**: Allows the selection between different shapes of the perforation (currently only "Simple Diamond"). • **Width**: Defines the width of the perforation. • **Height**: Defines the height of the perforation. • **Height Clearance**: Defines the vertical distance between perforations (height distance). • **Width Clearance**: Defines the horizontal distance between perforations (width distance). • **Border Clearance**: Defines the distance of the perforation to corners. ⓘ A weakening of the corners by perforations can thus be avoided. • **Top Solid Height**: Defines the distance between the part and the top row of perforations measured at the highest point of the perforation. • **Bottom Solid Height**: Defines the distance between the part and the lowest row of perforations measured at the lowest point of the perforation. • **Alternate Diamonds**: Enables/disables an offset of the perforations from each other.

Tab. 16: Block Support: Perforation

9.1.2.5 Fracturing

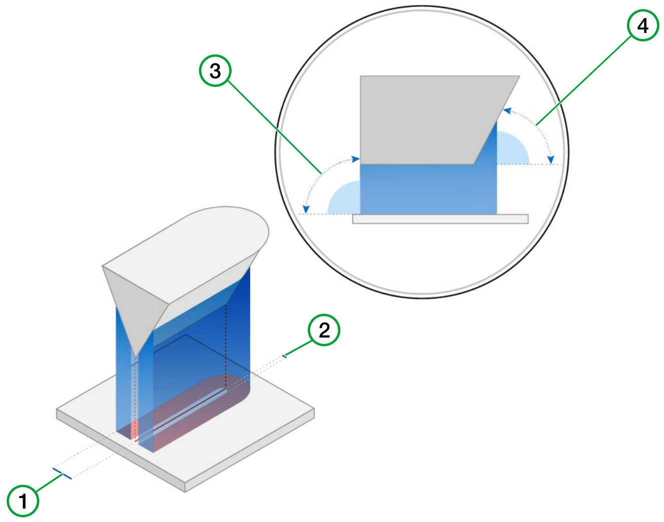
ⓘ If the "Fracturing" function is activated, the support geometry of a region can be divided into sub-blocks. This simplifies the manual removal of the supports considerably.

Parameter	Description
	 **1** X n Cells **2** Y n Cells **3** X Distance **4** Y Distance **5** ≥ min Segment Length, Facturing **6** < min Segment Length, no Facturing
Do Fracturing	If this function is activated, Block Supports are divided into sub-blocks according to the parameter values.
Min Segment Length	Defines the minimum length of an element before applying fracturing.
X n Cells	Defines the number of cells connected in the X-direction.
Y n Cells	Defines the number of cells connected in the Y-direction.
X Distance	Defines the X-distance between the individual cell blocks.
Y Distance	Defines the Y-distance between the individual cell blocks.

Tab. 17: Block Support: Fracturing

9.1.2.6 Line Thickening

ⓘ Parts for which only one edge is calculated as a support region due to their shape or positioning require a special support function. Since supports are usually not to be built directly onto an edge, the edge of the part is projected as a line onto the substrate plate. An offset value can now be defined for this line. Supports are then generated around this value. Since the offset value is subtracted from the thickness of the supports, the offset value must be smaller than the support thickness value.

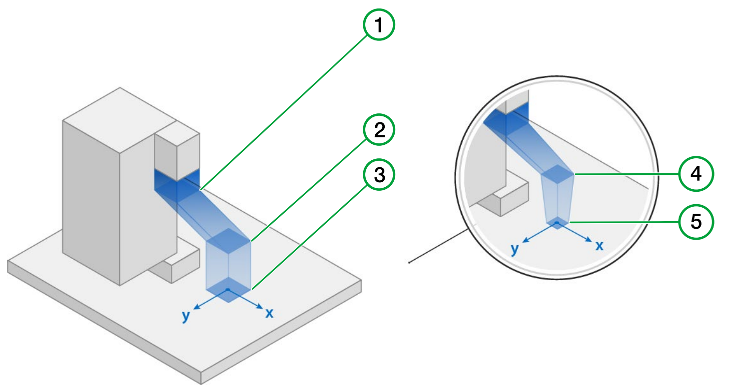
Parameter	Description
	 **1** Thickness **3** > Max Support Angle **2** Offset From Line **4** < Max Support Angle
Line Thickening	If this parameter is activated, the selected line element is revealed according to the parameters and supported by a Block Support.
Thickness	Defines the width of the support structure on each side of the part edge.
Offset From Line	Defines at what distance from the line area the support should be built.
Min Support Angle	Defines the minimum angle from which support structures are generated. The angle between the horizontal and the part is taken as a basis.
Max Support Angle	Defines the maximum angle up to which support structures are generated. The angle between the horizontal and the part is taken as a basis.

Parameter	Description
Filter Unconnected Blobs	In an unfavourable case, supports can be calculated that are not linearly connected to other supports and are therefore useless. If this parameter is activated, neither these individual supports are filtered nor generated. Since this process requires a lot of computing power and the creation of the individual supports is very unlikely, the process can be switched off.

Tab. 18: Block Support: Line Thickening

9.1.2.7 Deformation

ⓘ Settings for the deformation of supports can be defined with the "Deformation" function. Vertically aligned supports can be moved using two planes.

Parameter	Description
	 **1** Upper Plane Z **2** Lower Plane Z **3** Platform Translation **4** Lower Plane Scale **5** Platform Scale
Activate	Activates the function "Deformation"**.**
Max Deformation Angle	Defines the maximum deformation angle of the support geometry.
Upper Plane Z	Defines the distance between the substrate plate and the top of the support area to be angled.
Lower Plane Z	Defines the distance between the substrate plate and the lower end of the support area that is to be angled.
Platform Translation X	Defines the offset of the lower area of the supports relative to the upper area of the supports in the X-direction.
Platform Translation Y	Defines the offset of the lower area of the supports relative to the upper area of the supports in the X-direction.

Parameter	Description
Lower Plane Scale	Defines a scaling factor to reduce/increase the support footprint at the level of the lower plane.
Platform Scale	Defines a scaling factor to reduce/increase the support footprint on the substrate plate.

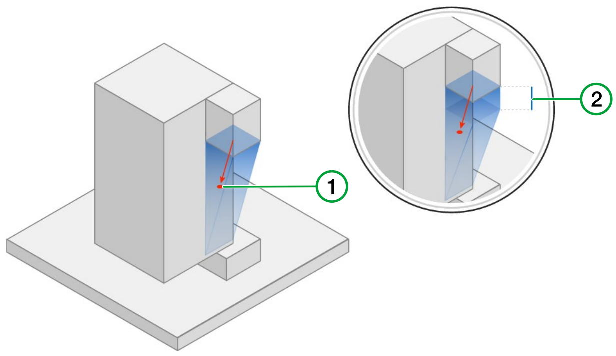
Tab. 19: Block Support: Deformation

Note

Besides entering the values manually, a deformation can also be performed interactively in the 3D view. The necessary command can be found in the context menu of the respective support structure. The tool for editing the deformation is visible when the Deformation function is activated. The positions of the Upper Plane and the Lower Plane can be defined by using the two arrows. The plane symbol moves the support along the X-position and Y-position on the substrate plate. However, the angle for the maximum deformation and the values for the scaling of the support base planes – Lower Plane Scale and Platform Scale – must be set in the input fields.

9.1.2.8 Gusset

ⓘ With the "Gusset" function, you can define a target point of the support structure for Block Supports. This target point can be on the part itself or on the substrate plate.

Parameter	Description
	 **1** Target Point **2** Vertical Offset
Activate	Activates/deactivates the "Gusset" function.
X	Defines the absolute X-position of the support.
Y	Defines the absolute Y-position of the support.
Z	Defines the absolute Z position of the support.
Scale	Defines a scaling factor. The cross-sectional area of the support in the target area is reduced/increased by this scaling factor. • Factor < 1 reduces the cross-sectional area. • Factor > 1 increases the cross-sectional area.
Do Vertical Top Offset	Activates/deactivates the function "Vertical Top Offset".

Parameter	Description
Vertical Top Offset	Defines an offset value in millimetres. The support is built on the part by this offset value vertically in the direction of the substrate plate.

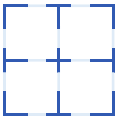
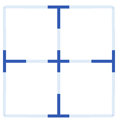
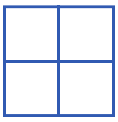
Tab. 20: *Block Support: Gusset*

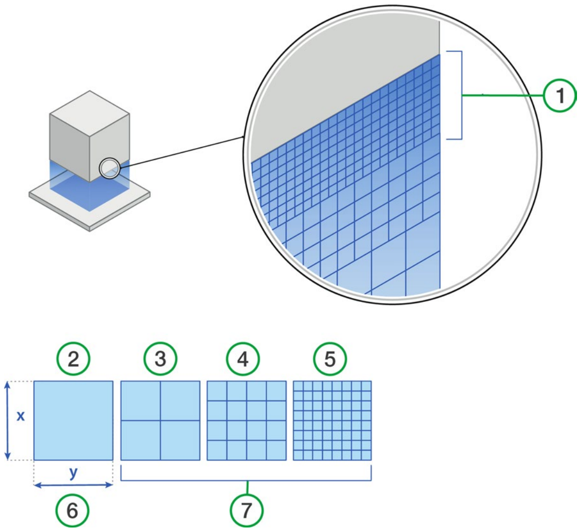
9.1.3 Adaptive Cell Support

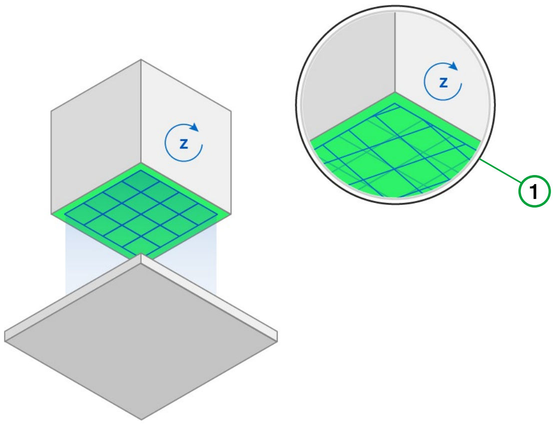
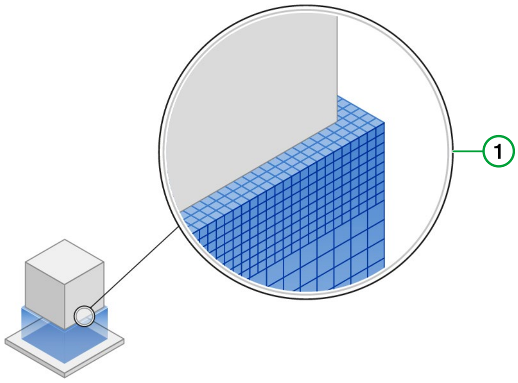
ⓘ This chapter contains information on the parameters that are defined under Adaptive Cell Support in the action step **Support**.

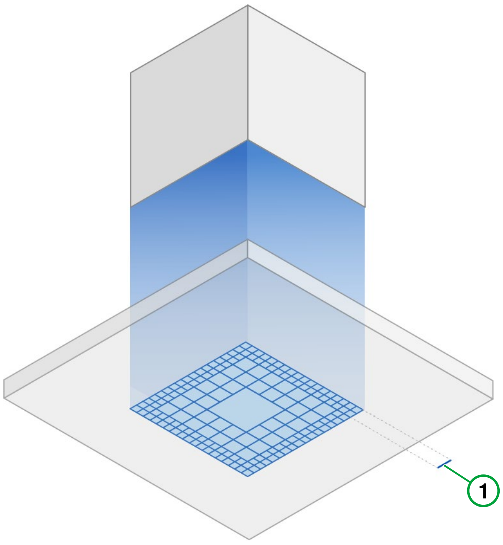
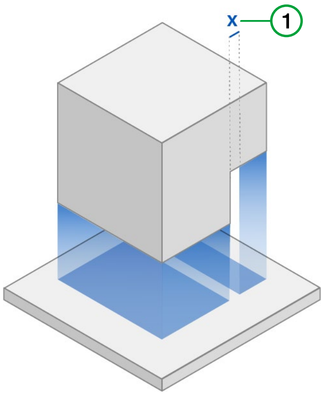
9.1.3.1 Adaptive Cell Settings

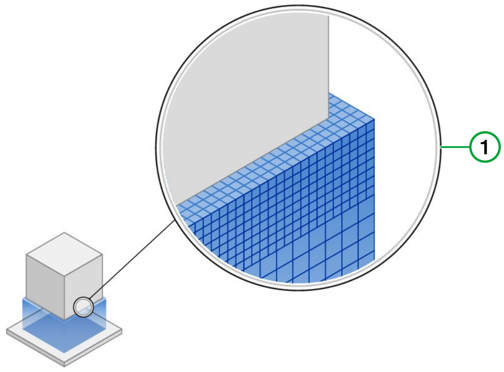
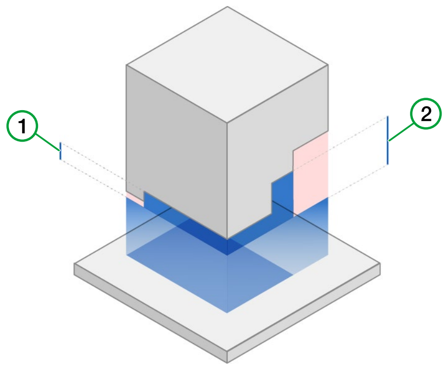
The following parameters can be used to define the support geometry of adaptive cells:

Parameter	Description
General	
Adaptive Cell Type	Allows the selection of a shape of the adaptive cells. default:  default_cross_attachment:  default_full_attachment: 

Parameter	Description
Adaptive Cell Settings	 **1** Min Num Smallest Cells **2** Level 0 **3** Level 1 **4** Level 2 **5** Level 3 **6** Largest Cell Size **7** Max Subdivisions • Largest Cell Size: Defines the size of the lowest horizontal adaptive cells. • Max Subdivisions: Defines the frequency with which the horizontal adaptive cells may be reduced in size. • Min Num Smallest Cells: Defines the number of smallest adaptive cells on a part.

Parameter	Description
Rotation	 **1** Rel. Gas Flow Angle If this parameter is activated, it is possible to define a rotation of the supports relative to the gas flow direction. • **Rel. Gas Flow Angle [°]**: Defines the angle for the rotation of the supports. For example, an angle of 0° means that the supports are oriented in the direction of the gas flow.
Border	
Extend Over Border	 **1** Extend Over Border If this parameter is activated, the supports may extend over the border of the area to be supported in X and Y direction. This may make the supports wider than the part.
Max Subdivisions	Defines the maximum number of reductions of the adaptive cells in the area of the support Contour.
Min Subdivisions:	Defines the minimum number of reductions of the adaptive cells in the area of the support Contour.

Parameter	Description
Smallest Cell Inwards	Defines the depth of the adaptive cells (based on the values under Max Subdivisions und Min Subdivisions).  **1** Smallest Cell Inwards
Clearance	
	 **1** Part Clearance
Smallest Cell For Clearance	Defines the size of the adaptive cells to a vertical wall of a part.
Part Clearance	Defines the distance between a vertical wall of a part and the supports to be generated. This prevents the supports from merging with a vertical wall of the part.
Part to Part	
Part to Part	If this parameter is activated, supports can be generated not only between the part and the substrate plate, but also between part areas.

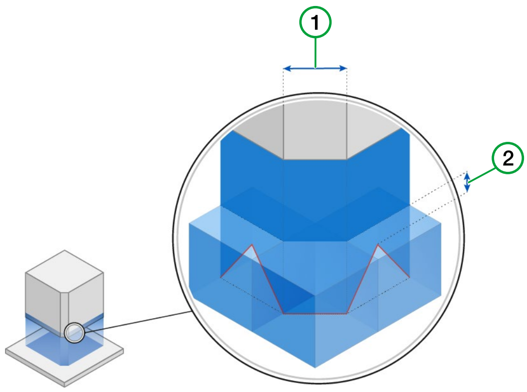
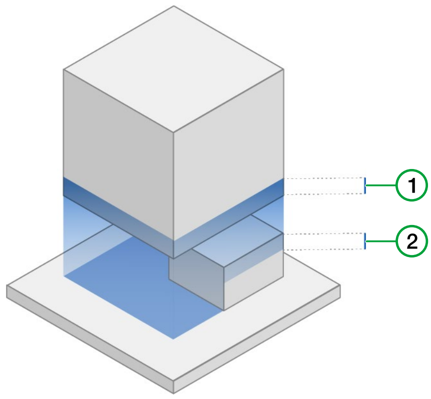
Parameter	Description
Extend Over Border	 **1** Extend Over Border If this parameter is activated, the supports may extend over the border of the area to be supported in X and Y direction. This may make the supports wider than the part.
Border Max Subdivisions	Defines the maximum number of reductions of the adaptive cells in the area of the support Contour.
Platform Max Subdivisions	Defines the maximum number of reductions of the adaptive cells in the substrate plate area of the support.
Attachment	
	 **1** Min Attachment Height **2** Max Attachment Height Attachments are generated between the supports and the part to close possible gaps. Gaps can occur because only complete adaptive cells are generated.
Attachment Intrusion	Defines how far the attachments penetrate the part.
Min Attachment Height	Defines the minimum length of the attachments.
Max Attachment Height	Defines the maximum length of the attachments.

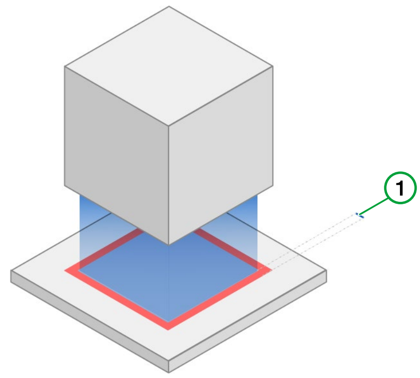
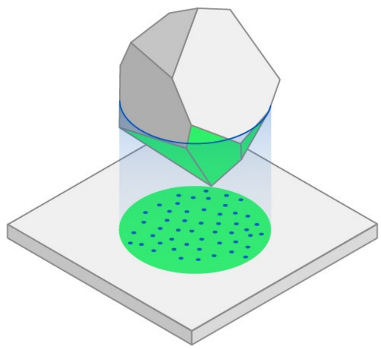
Tab. 21: Adaptive Cell Support: Adaptive Cell Settings

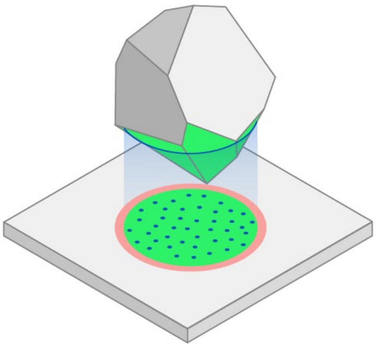
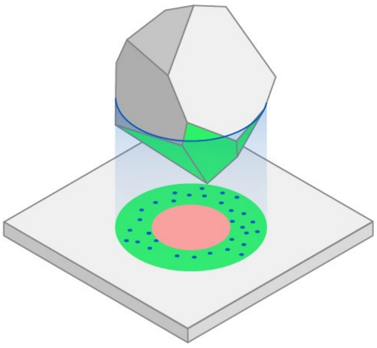
9.1.3.2 Contour

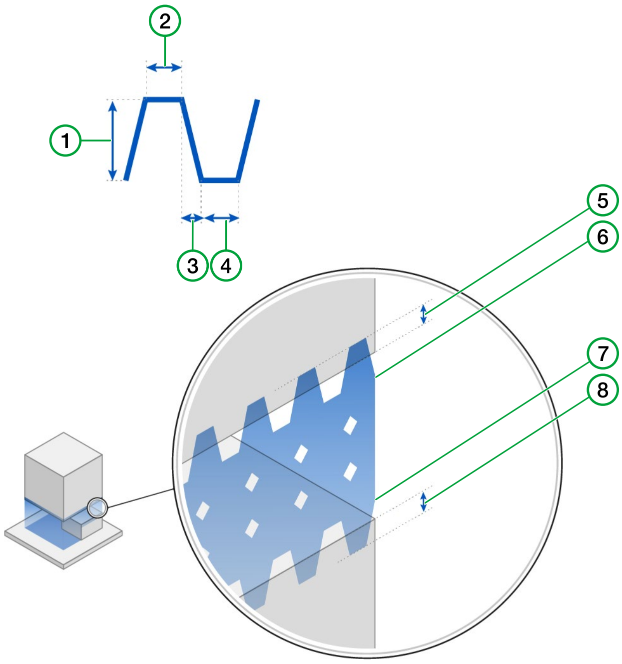
ⓘ A Contour is an additional support wall that encloses other support structures.

In the ***Contour*** menu, the following parameters can be defined to define the Contour geometry:

Parameter	Description
General	
Build Contour	 **1** Min Segment Length For Opening **2** Opening Distance From Cell Top When this parameter is activated, the selected support structure is reinforced with a Contour at the outer edge.
	 **1** Intrusion Top **2** Intrusion Bottom
Top Intrusion	Defines how far the supports of the **Contour** should penetrate the part from the bottom.
Bottom Intrusion	Defines how far the supports of the **Contour** should penetrate from above into an underlying part.
Min Segment Length For Opening	Defines the minimum length of a contour from which a perforation can occur underneath.

Parameter	Description
Opening Distance From Cell Top	Defines the distance between the teeth and the perforation.
Offset	 **1** Offset Defines the distance of the **Contour** to the outer edge of the support re-gion.
Smooth Contour	
Smooth Contour	 If this parameter is activated, the sharp edges of the support structure are smoothed.

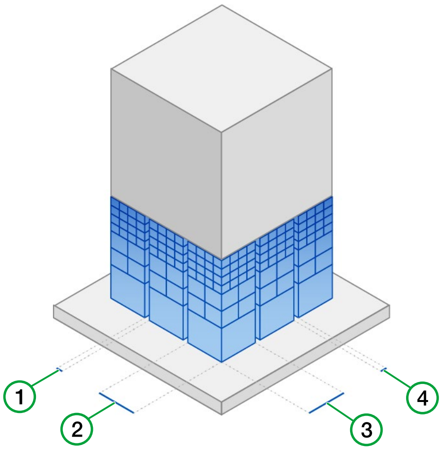
Parameter	Description
Max Support Angle	 Defines the angle between the part and the substrate plate up to which supports are generated. The distance between the support with **Smooth Contour** and the outer area of the support region can be increased. As a result, the area with a support structure becomes smaller.
Min Support Angle	 Defines the angle between the part and the substrate plate from which supports are generated.

Parameter	Description
Tooth Pattern	
	 **1** Height **2** Top Width **3** Crossing Width **4** Bottom Width **5** Top Intrusion **6** Top Teeth **7** Bottom Teeth **8** Bottom Intrusion
Top Tooth Pattern	If this parameter is activated, the following values can be defined for the structure of the upper row of teeth. • **Bottom Width:** Defines the bottom width of the teeth. • **Crossing Width:** Defines the distance between the end point of the top tooth width (Top Width) and the start point of the bottom tooth width (bottom width). • **Top Width:** Defines the top width of the teeth. • **Height:** Defines the tooth height between the part and the support plus the value of the intrusion.

Tab. 22: Adaptive Cell Support: Contour

9.1.3.3 Fracturing

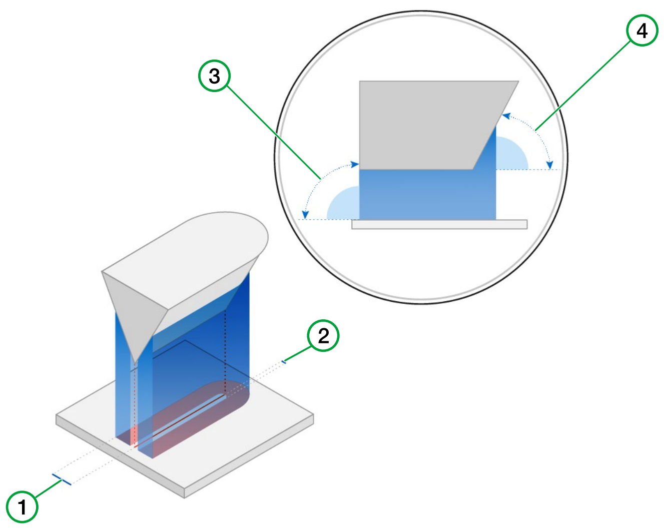
ⓘ If the "Fracturing" function is activated, the support geometry of a region can be divided into sub-blocks. This simplifies the manual removal of the supports considerably.

Parameter	Description
	 **1** Y Distance **2** Y n Cells **3** X n Cells **4** X Distance
X n Cells	Defines the number of cells connected in the X-direction.
Y n Cells	Defines the number of cells connected in the Y-direction.
X Distance	Defines the X-distance between the individual cell blocks.
Y Distance	Defines the Y-distance between the individual cell blocks.

Tab. 23: Adaptive Cell Support: Fracturing

9.1.3.4 Line Thickening

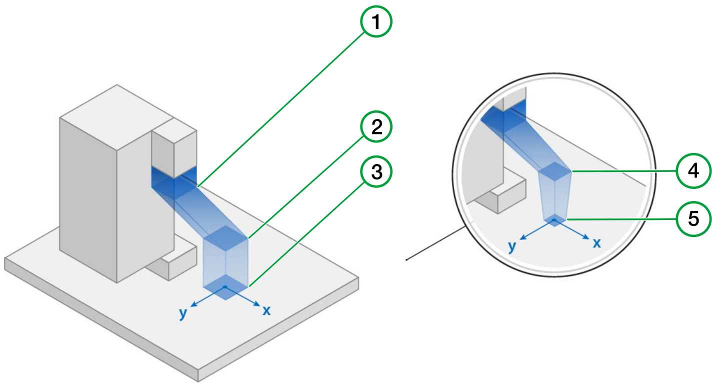
ⓘ Parts for which only one edge is calculated as a support region due to their shape or positioning require a special support function. Since supports are usually not to be built directly onto an edge, the edge of the part is projected as a line onto the substrate plate. An offset value can now be defined for this line. Supports are then generated around this value. Since the offset value is subtracted from the thickness of the supports, the offset value must be smaller than the support thickness value.

Parameter	Description
	 **1** Thickness **3** > Max Support Angle **2** Offset From Line **4** < Max Support Angle
Line Thickening	If this parameter is activated, the selected line element is revealed according to the parameters and supported by a Block Support.
Thickness	Defines the width of the support structure.
Offset From Line	Defines at what distance from the line area the support should be built.
Min Support Angle	Defines the minimum angle from which support structures are generated. The angle between the horizontal and the part is taken as a basis.
Max Support Angle	Defines the maximum angle up to which support structures are generated. The angle between the horizontal and the part is taken as a basis.
Filter Unconnected Blobs	In an unfavourable case, supports can be calculated that are not linearly connected to other supports and are therefore useless. If this parameter is activated, neither these individual supports are filtered nor generated. Since this process requires a lot of computing power and the creation of the individual supports is very unlikely, the process can be switched off.

Tab. 24: Adaptive Cell Support: Line Thickening

9.1.3.5 Deformation

ⓘ Settings for the deformation of supports can be defined with the "Deformation" function. Vertically aligned supports can be moved using two planes.

Parameter	Description
	 **1** Upper Plane Z **2** Lower Plane Z **3** Platform Translation **4** Lower Plane Scale **5** Platform Scale
Activate	Activates/deactivates the "Deformation" function.
Max Deformation Angle	Defines the maximum deformation angle of the support geometry.
Upper Plane Z	Defines the distance between the substrate plate and the top of the support area to be angled.
Lower Plane Z	Defines the distance between the substrate plate and the lower end of the support area that is to be angled.
Platform Translation X	Defines the offset of the lower area of the supports relative to the upper area of the supports in the X-direction.
Platform Translation Y	Defines the offset of the lower area of the supports relative to the upper area of the supports in the X-direction.
Lower Plane Scale	Defines a scaling factor to reduce/increase the support footprint at the level of the lower plane.
Platform Scale	Defines a scaling factor to reduce/increase the support footprint on the substrate plate.

Tab. 25: Adaptive Cell Support: Deformation

Note

Besides entering the values manually, a deformation can also be performed interactively in the 3D view. The necessary command can be found in the context menu of the respective support structure. The tool for editing the deformation is visible when the Deformation function is activated. The positions of the Upper Plane and the Lower Plane can be defined by using the two arrows. The plane symbol moves the support along the X-position and Y-position on the substrate plate.

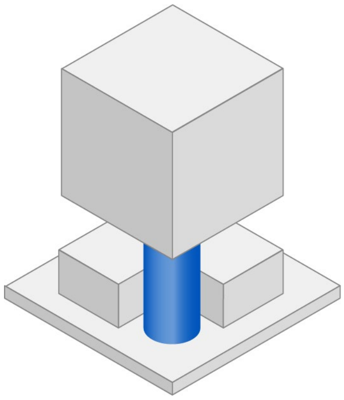
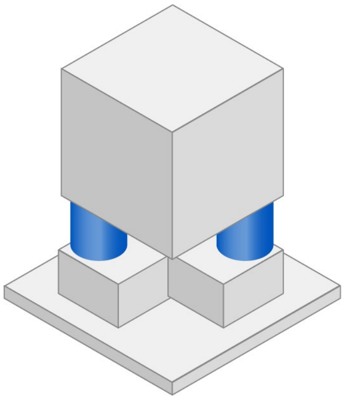
> *However, the angle for the maximum deformation and the values for the scaling of the support base planes – Lower Plane Scale and Platform Scale – must be set in the input fields.*

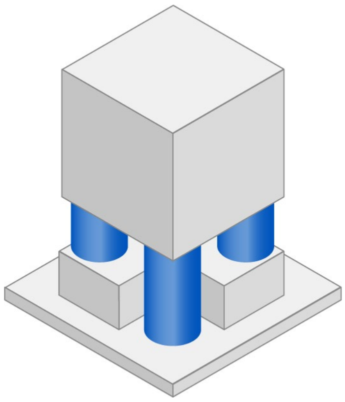
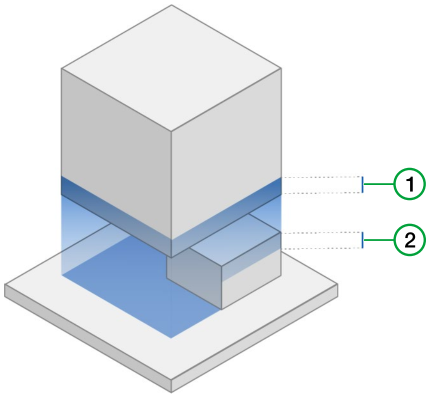
9.1.4 Line Support

ⓘ This chapter contains information on the parameters that are defined under Line Support in the action step **Support**.

9.1.4.1 Line Settings

The following parameters can be used to define the line support geometry:

Parameter	Description
Build Support	
Build on Platform	 If this parameter is activated, supports are generated between the part and the substrate plate.
Build on Part	 If this parameter is activated, supports are generated between part areas.

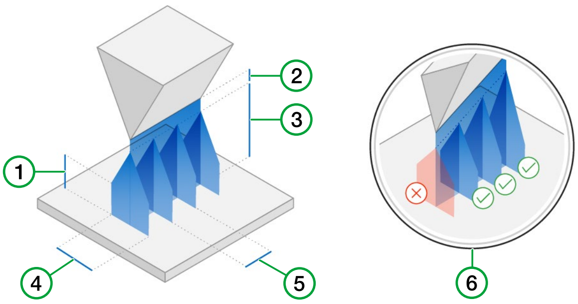
Parameter	Description
Combined	 This parameter is enabled by activating **Build on Platform** and **Build on Part**.
Intrusion	
	 **1** Intrusion Top **2** Intrusion Bottom
Top	Defines how far the supports should penetrate the part from below.
Bottom	Defines how far the supports should penetrate the part from above.

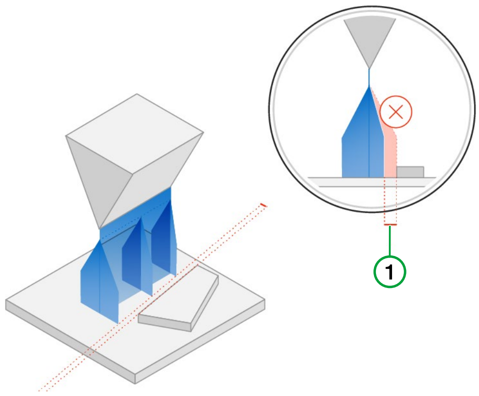
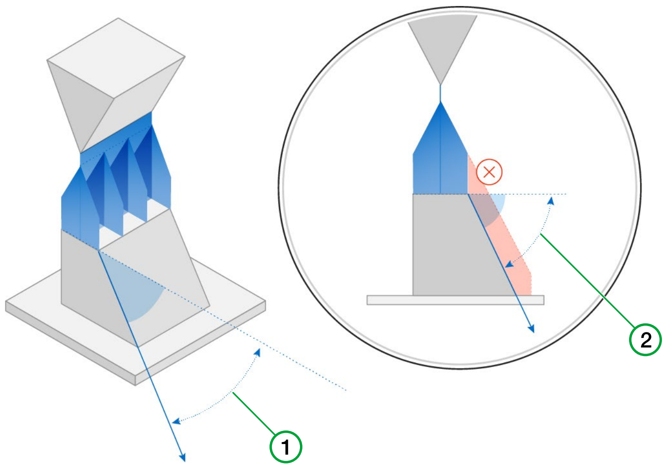
Tab. 26: Line Support: Line Settings

9.1.4.2 Struts

ⓘ Struts are support structures that reinforce line supports. Struts are at a 90° angle to the line supports and support them.

The following parameters are available for defining the Struts:

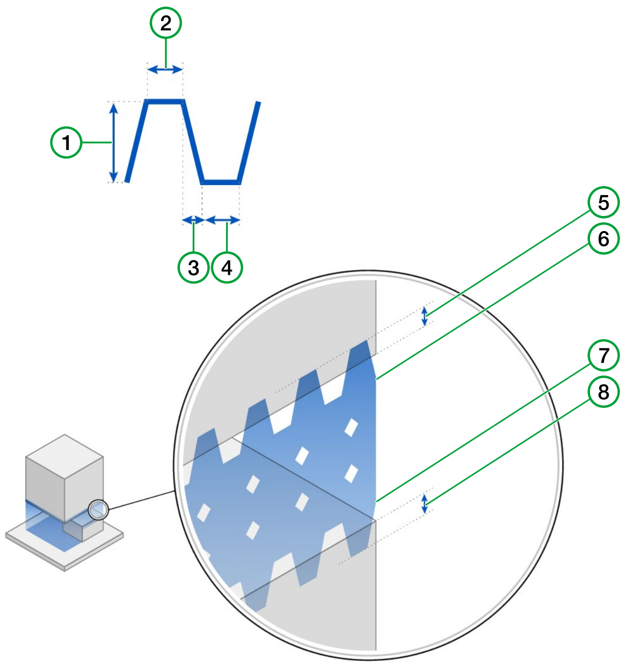
Parameter	Description
	 **1** Base Height **2** Top Distance (min & max) **3** Min Height **4** Width **5** Distance **6** x < min. height, no strut will be build
Line Has Struts	Activates/deactivates the "Struts" function.
Struts Distance	Defines the distance between the individual Struts.
Strut Min Height	Defines the height of the Struts. StrutsStruts that are smaller than the defined value are not generated.
Strut Min Top Distance	Defines the minimum distance between the top of the Struts and the part.
Strut Max Top Distance	Defines the maximum distance between the top of the Struts and the part.
Strut Base Height	Defines the height of the Struts up to which the Struts run upwards at a 90° angle to the substrate plate. After this height, the Struts bend and run upwards at an acute angle.
Strut Half Min Width	Defines the minimum width of the Struts halves at the substrate plate.
Strut Max Width	Defines the maximum width of the Struts at the substrate plate.

Parameter	Description
Strut Mesh Clearance	 **1** Mesh Clearance Defines the clearance between Struts and a nearby part.
Strut Max Slope on Part	 **1** Angle < x° **2** Angle < x° Defines the angle up to which Struts are generated.

Tab. 27: Line Support: Struts

9.1.4.3 Tooth Pattern

ⓘ If parameters in ***Tooth Pattern*** are activated, depending on the setting, recesses in the form of teeth are generated in a Contour. These tooth patterns can be generated at the upper and/or lower edge.

Parameter	Description
	 **1** Height **2** Top Width **3** Crossing Width **4** Bottom Width **5** Top Intrusion **6** Top Teeth **7** Bottom Teeth **8** Bottom Intrusion
Top And Bottom Equal	If this parameter is activated, the structure of the teeth for upper and lower structures of the Contour is identical. The values are taken from the settings of the **Top Tooth Pattern**.
Top Tooth Pattern	If this parameter is activated, the following values can be defined for the structure of the upper row of teeth. • **Bottom Width:** Defines the bottom width of the teeth. • **Crossing Width:** Defines the distance between the end point of the top tooth width (Top Width) and the start point of the bottom tooth width (bottom width). • **Top Width:** Defines the top width of the teeth. • **Height:** Defines the tooth height of the support. The teeth protrude into the part by the value Intrusion. **Note** *The values of the intrusion are defined in the menu Contour.*

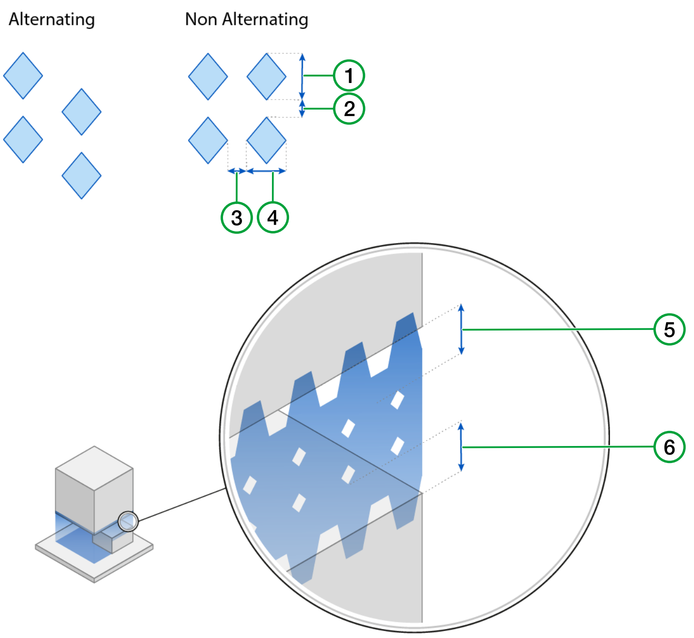
Parameter	Description
Bottom Tooth Pattern	If this parameter is activated, the following values can be defined for the structure of the lower row of teeth. • **Bottom Width:** Defines the bottom width of the teeth. • **Crossing Width:** Defines the distance between the end point of the top tooth width (Top Width) and the start point of the bottom tooth width (bottom width). • **Top Width:** Defines the top width of the teeth. • **Height:** Defines the tooth height of the support. The teeth protrude into the part by the value Intrusion. **Note** *The values of the intrusion are defined in the menu Contour.*

Tab. 28: *Line Support: Tooth Pattern*

9.1.4.4 Perforation

ⓘ A Contour can be perforated at Line Support. This perforation allows e.g. loose powder to flow out of the support geometry better.

When the function is activated, the following parameters can be defined regarding the perforation of the Contour:

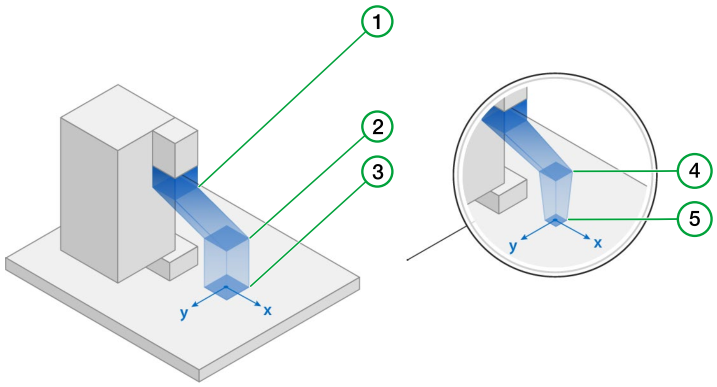
Parameter	Description
	 **1** Height **2** Height Clearance **3** Width Clearance **4** Width **5** Top Solid Height **7** Bottom Solid Height

Parameter	Description
Perforation	If this parameter is activated, a perforation pattern can be defined for the Contour. The following parameters can be defined: • **Type**: Allows the selection between different shapes of the perforation (currently only "Simple Diamond"). • **Width**: Defines the width of the perforation. • **Height**: Defines the height of the perforation. • **Height Clearance**: Defines the vertical distance between perforations (height distance). • **Width Clearance**: Defines the horizontal distance between perforations (width distance). • **Border Clearance**: Defines the distance of the perforation to corners. ⓘ A weakening of the corners by perforations can thus be avoided. • **Top Solid Height**: Defines the distance between the part and the top row of perforations measured at the highest point of the perforation. • **Bottom Solid Height**: Defines the distance between the part and the lowest row of perforations measured at the lowest point of the perforation. • **Alternate Diamonds**: Enables/disables an offset of the perforations from each other.

Tab. 29: Line Support: Perforation

9.1.4.5 Deformation

ⓘ Settings for the deformation of supports can be defined with the "Deformation" function. Vertically aligned supports can be moved using two planes.

Parameter	Description
	 **1** Upper Plane Z **2** Lower Plane Z **3** Platform Translation **4** Lower Plane Scale **5** Platform Scale
Activate	Activates the "Deformation” function.
Max Deformation Angle	Defines the maximum deformation angle of the support geometry.
Upper Plane Z	Defines the distance between the substrate plate and the top of the support area to be angled.
Lower Plane Z	Defines the distance between the substrate plate and the lower end of the support area that is to be angled.
Platform Translation X	Defines the offset of the lower area of the supports relative to the upper area of the supports in the X-direction.
Platform Translation Y	Defines the offset of the lower area of the supports relative to the upper area of the supports in the X-direction.
Lower Plane Scale	Defines a scaling factor to reduce/increase the support footprint at the level of the lower plane.
Platform Scale	Defines a scaling factor to reduce/increase the support footprint on the substrate plate.

Tab. 30: Line Support: Deformation

Note

Besides entering the values manually, a deformation can also be performed interactively in the 3D view. The necessary command can be found in the context menu of the respective support structure. The tool for editing the deformation is visible when the Deformation function is activated. The positions of the Upper Plane and the Lower Plane can be defined by using the two arrows. The plane symbol moves the support along the X-position and Y-position on the substrate plate.

However, the angle for the maximum deformation and the values for the scaling of the support base planes – Lower Plane Scale and Platform Scale – must be set in the input fields.

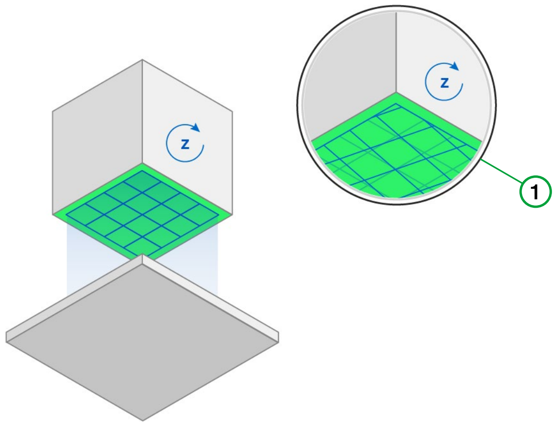
9.1.5 Tree Support

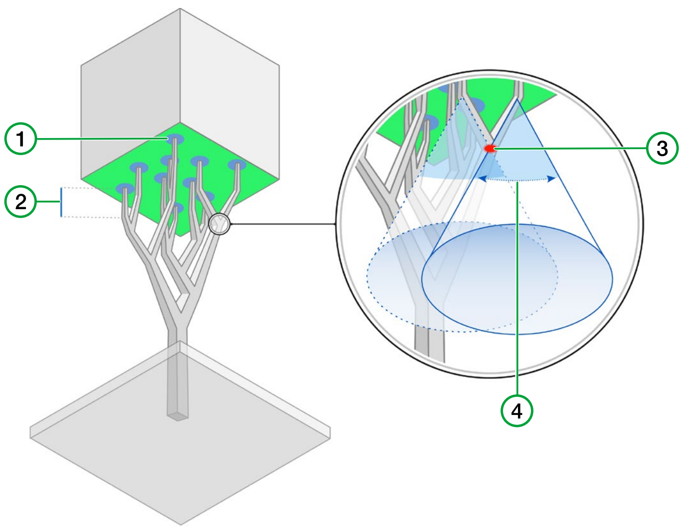
ⓘ This chapter contains information on the parameters that are defined under Tree Settings in the action step **Support**.

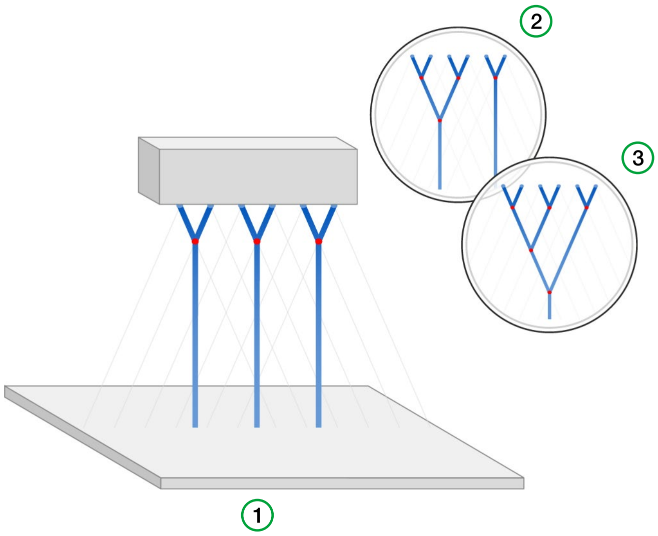
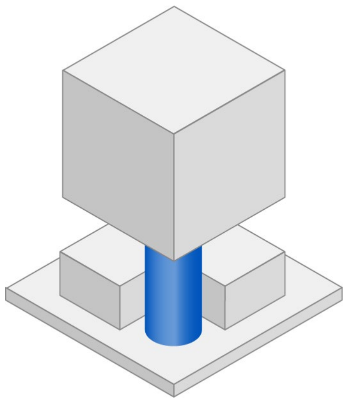
9.1.5.1 Tree Settings

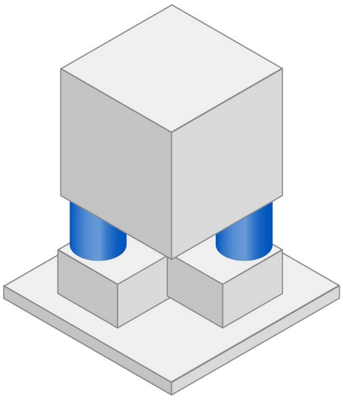
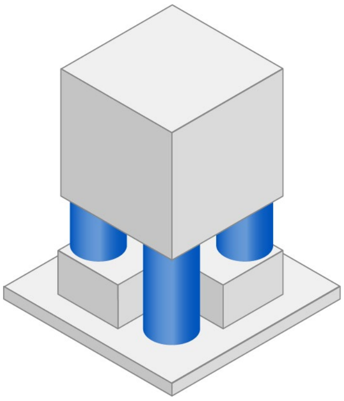
ⓘ The connection of the Tree Support to a part is based on the strategies of the Rod Support. For this reason, tree support and rod support share many parameters.

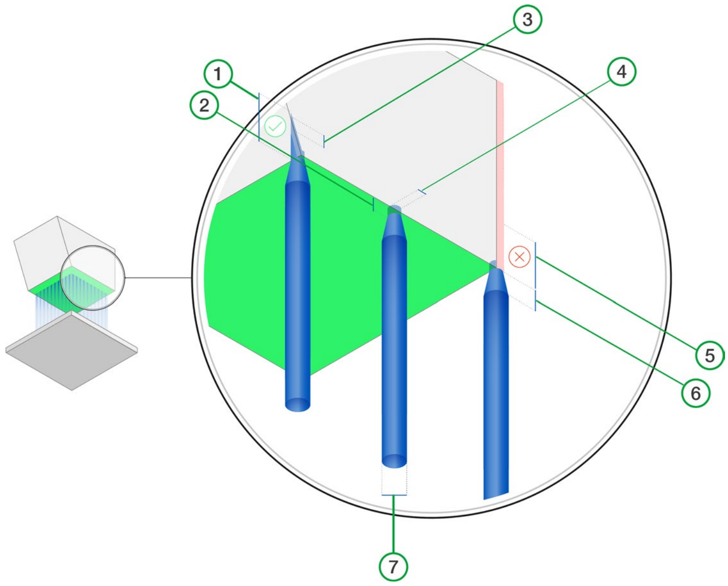
The following parameters are available for defining the Tree Support geometry:

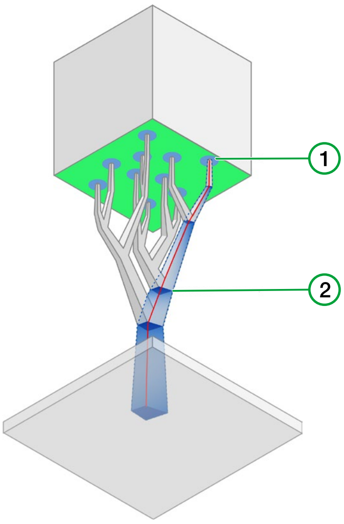
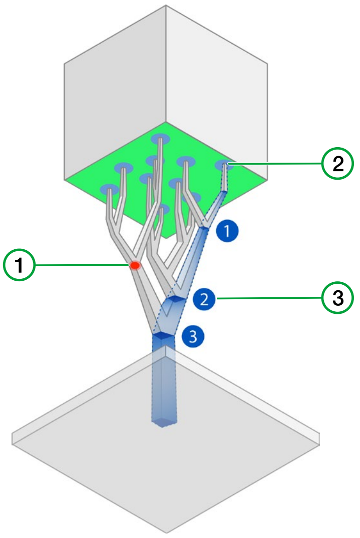
Parameter	Description
General	
Seed	Here a start value can be defined for the first distribution calculation. This start value, together with a random value, results in the distribution of the ties on the print object.
Region Rotation	 **1** Rel. Gas Flow Angle If this parameter is activated, it is possible to define a rotation of the supports relative to the gas flow direction. • **Rotation Angle [°]**: Defines the angle for the rotation of the supports. For example, an angle of 0° means that the supports are oriented in the direction of the gas flow.

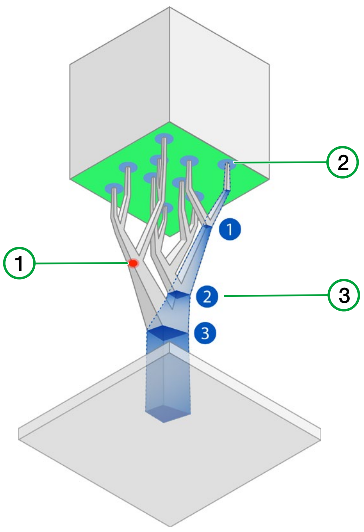
Parameter	Description
Tree Cone Apex Angle	 **1** Sample Point **3** Branch Point **2** Vertical Offset **4** Cone Apex Angle Defines the angle between the lower part of the Tree Support – the trunk – and the upper angled part of the Tree Support – the branches.
Attachment Vertical Offset	If this parameter is activated, the vertical length of the Tree Supports that directly connect to the part can be defined. • Attachment Vertical Offset: Defines an offset value for the vertical length of the 3d views.

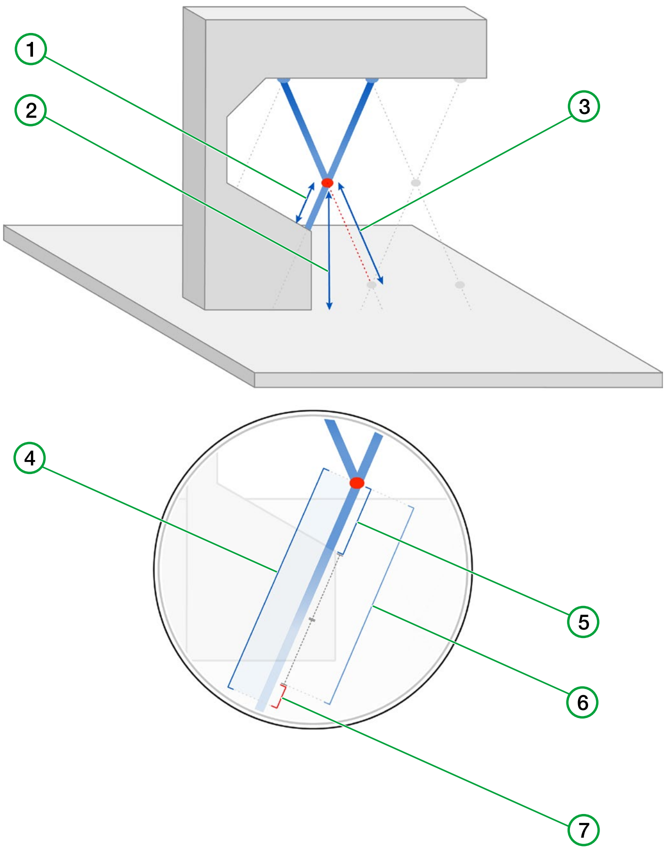
Parameter	Description
Max Attachments Per Trunk	 **1** Max Attachments = 2 **2** Max Attachments = 4 **3** Max Attachments = 6 Defines the maximum number of Tree Supports per trunk that directly connect to the part.
Build Support	
Build on Platform	 If this parameter is activated, supports are generated between the part and the substrate plate.

Parameter	Description
Build on Part	 If this parameter is activated, supports are generated between part areas.
Combined	 This parameter is activated by enabling Build on Platform and Build on Part.

Parameter	Description
Tip Attachment	
	 **1** Max Extension Height **2** Intrusion **3** Intrusion **4** Tip Scale x RD **5** Max Extension Height **6** Relative Height x RD **7** Rod Diameter (RD)
Top Diameter	Defines how big the diameter of the support at the tip of the Tree Support support is.
Relative Height	Defines how long the tapered area at the top of the rods is.
Tip Scale	Defines the scaling factor for the diameter of the tip adjacent to the part. Diameter = Scaling factor * Rod Diameter
Intrusion	Defines how far the support penetrates into the part.
Max Extension Height	If the rod is built at an edge where there is no intersection area parallel to the substrate plate, the rod can be built up to the **Max Extension Height**, as long as it meets the geometry.

Parameter	Description
Weighting	
Weight By Length Summand	 **1** Top Diameter **2** Diameter = Top Diameter + Summand x Length
Weight By Fork Summand	 **1** Branch Point **3** Diameter = Top Diameter + Summand x Branch Number **2** Top Diameter

Parameter	Description
Weight By Fork Factor	 **1** Branch Point **2** Top Diameter **3** Diameter = Top Diameter x Factor x Branch Number

Parameter	Description
Length	
	 **1** To Mesh Length **2** To Baseplate Length **3** To Branch Length **4** $L_{Algo} = L \times Factor + Summand$ **5** Length (L) **6** L x Factor **7** Summand The length of the branches of Tree Supports is determined considering three aspects: • Distance to the next mesh • Distance to the next branch • Distance to the substrate plate When building a tree support – starting from the support region downwards – the shortest possible connection is chosen. The formula for calculating the support length takes into account the parameters listed below.
Branch Length Factor	Defines the factor for the length of the branches.

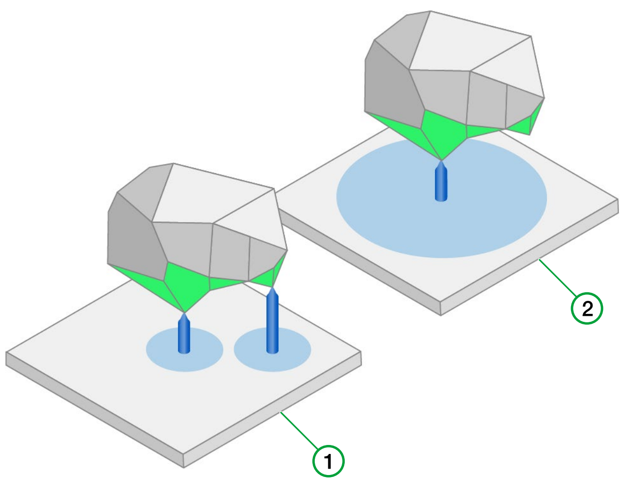
Parameter	Description
Tree To Mesh Length Factor	Defines the factor of the length from the tree to the mesh.
Tree To Platform Length Factor	Defines the factor of the length from the tree to the substrate plate.
Branch Length Summand	Defines the summand for the branch length.
Tree To Mesh Length Summand	Defines the summand for the length from tree to mesh.
Tree To Platform Length Summand	Defines the summand for the length from tree to substrate plate.

Tab. 31: Tree Support: Tree Settings

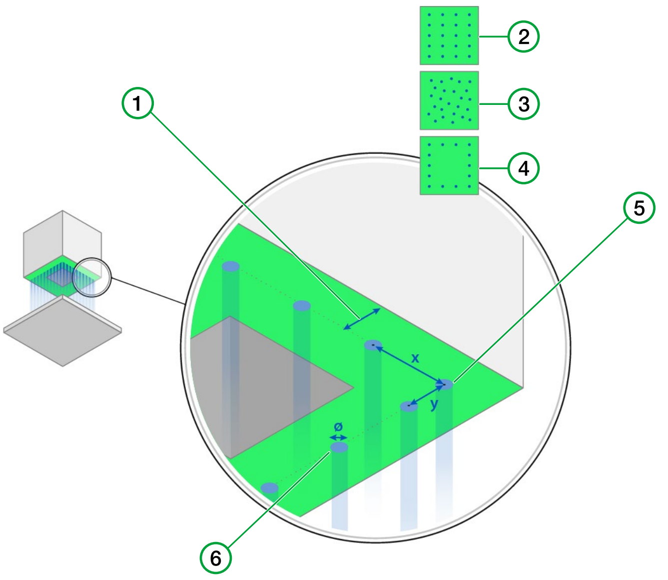
9.1.5.2 Tree Point Sampler

ⓘ In "Tree Point Sampler" the sampling methods can be defined that AM-Studio uses to distribute support region on a with support rods.

The following sampling methods and parameters are available:

Parameter	Description
Min Point Sampler	
Min Point Sampler	 **1** Min Point Sampler **2** Local Domain Size If this parameter is activated, rod supports are generated in a local minimum of an area to be supported. This prevents geometries from being created loosely in the powder. The local minimum is an area and not a geometric point. Depending on the parameters selected, a single rod support or a series of rod supports in a Poisson distribution around the centre of gravity is generated at the centre of gravity of this area.

Min Distance To Outer Perimeter	Defines the minimum distance between the Rod Supports and the outer contour.
Seed	Defines the starting value for the first distribution calculation. This initial value, together with a random value, gives the next value of the Poisson distribution.
Min Distance Between Points	Defines the minimum distance between the individual rods of the Rod Supports.
Max Diagonal For Single Point Plateau	Defines the size of the local minimum area above which no single rod support supports the area, but a series of Poisson distributed rod supports.
Min Minima Fraction For Plateau	Defines how many points of the polygon grid may lie below or above the surface level and the surface still counts as a local minimum.
Max Plataeu Gradient	Defines the angle in degrees that the surface may be tilted and still be considered a local minimum.
Sample Large Plateaus	If this parameter is activated, the Min Point Sampler is applied to minimum surfaces. • **Min Distance Between Points On Plateau:** Defines the spacing of the rods on the minimum surface. • **Max Plateau Area Ratio**: Defines a ratio between the sides of the plateau that must be met to generate supports. • **Max Plateau BB Short Side**: Defines the maximum side length of the bounding box of the minimum area.

Additional Sampling Algorithms	
	 **1** Min Distance To Contour **2** Regular Grid Sampler **3** Poisson Disk Sampler **4** Contour Sampler **5** Grid Cell Size X Distance Between Points **6** Rod Diameter
Contour Sampler	
Contour Sampler	If this parameter is activated, **Rod Supports** are generated along the edges of the area to be supported.
Min Distance To Contour	Defines the minimum distance between the **Rod Supports** and the outer contour.
Distance Between Points	Defines the minimum distance between the individual rods of the **Rod Supports**.
Regular Grid Sampler	
Regular Grid Sampler	If this parameter is activated, rods of the **Rod Support** are generated at regular intervals for the selected area to be supported. For each rod of the **Rod Supports**, a fictitious cell is created, which can be adjusted in size. In this way, the minimum distance between the rods can be defined.
Min Distance To Contour	Defines the minimum distance between the Rod Supports and the outer contour.
Grid Cell Size X	Defines the width of the fictional cells in the X-direction.
Grid Cell Size Y	Defines the length of the fictional cells in the Y-direction.

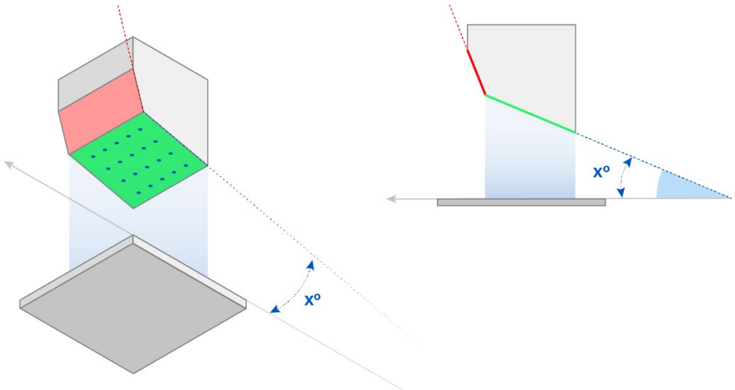
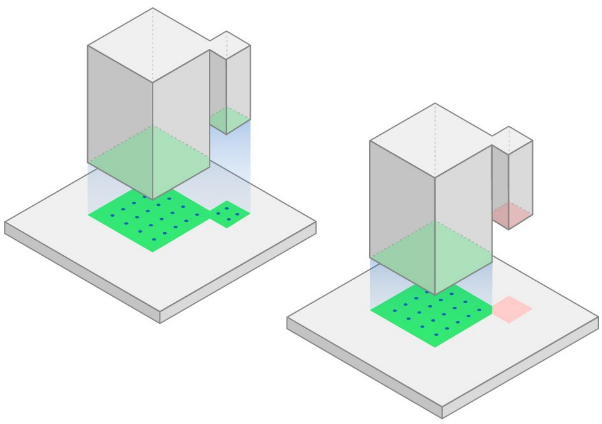
Check Distance To Input Points	If this parameter is activated, existing rods of other sampling alogrithms are taken into account when applying the Regular Grid Sampler. • **Min Dist To Input Points**: Defines the distance to existing rod supports.
Poisson Disk Sampler	
Poisson Disk Sampler	If this parameter is activated, Rod Supports are statistically generated and distributed based on a Poisson distribution for the selected area to be supported. The distribution calculation is based on the calculation of a random value with all random values from previous calculations. The resulting value in turn is used for the subsequent distribution calculation. 1. Calculation: Random value + value under **Seed** = value 1 2. Calculation: Random value + value 1 = value 2 3. Calculation: Random value + value 1 + value 2 = value 3.
Min Distance To Contour	Defines the minimum distance between the **Rod Supports** and the outer contour.
Min Distance Between Points	Defines the minimum distance between the individual rods of the **Rod Support.**
Seed	Defines a starting value for the first distribution calculation. This initial value together with a random value yields the first Poisson distribution. With this function, the same result can be guaranteed with multiple assignments of the Poisson distribution.

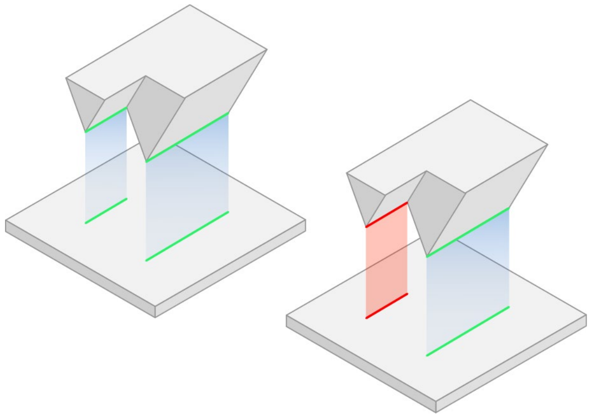
Tab. 32: *Tree Support: Tree Point Sampler*

9.2 Identification of support regions

ⓘ Before support structures can be generated, the support regions of parts must be calculated.

The areas that require a support are calculated automatically by AM-Studio when a part is loaded onto the substrate plate (**Orientation** action step) on the basis of the following parameters:

Parameter	Description
Overhang Angle	 Defines the angle between the part surface and the substrate plate from which a part surface is provided with a support structure. The angles of the support regions are coloured according to the colour scale of the "Overhang Gradient", which is pinned to the left of the function panel. **Note** *The colour scale is only visible when a part is selected.* *The tool Overhang Gradient must be activated in the context menu Part (action steps "Orientation" or "Support").*
Filter Region	 Defines a minimum area size from which supports are generated on a part area. Areas smaller than the defined value are not provided with a support structure.

Parameter	Description
Filter Line	 Defines a minimum line length from which supports are generated on a part. Lines shorter than the defined value are not provided with a support structure.

Tab. 33: Identification of support regions

Depending on the selected material, predefined values for the mentioned parameters are applied. However, the parameter values can be adjusted manually at any time.

ⓘ If the part is selected when you switch from the **Orientation** action step to the **Support** action step, or if it is selected in the **Support** action step, it appears in a semi-transparent view. The generated support regions shine through the part in green.

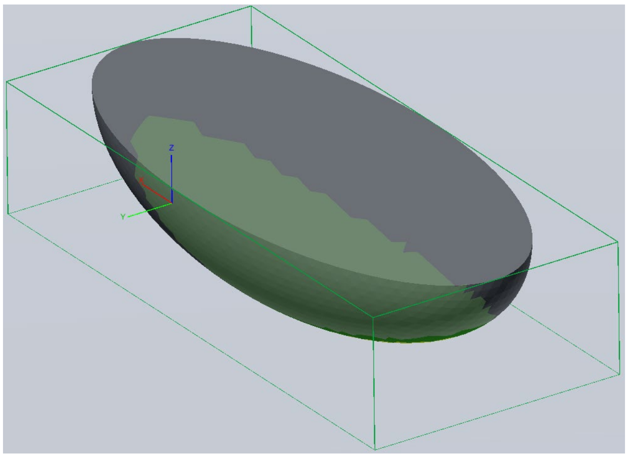


Fig. 10: Part in semi-transparent view with support regions (green)

ⓘ The angles of the support regions are coloured according to the Overhang Gradient scale (see above **Overhand Angle**).

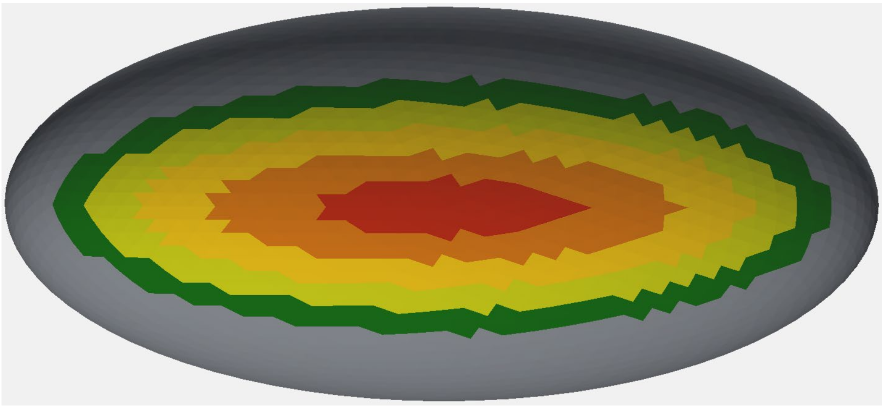


Fig. 11: Support regions on the part coloured according to the Overhang Gradient

ⓘ If the part is selected when you switch from the **Orientation** action step to the **Support** action step, or if it is selected in the **Support** action step, the ***Region List*** window opens automatically (see *list of the support regions*).

With the **[CALCULATE REGIONS]** button, support regions can be recalculated, e.g. if

- a part has been repositioned
- parameter values have been changed
- a new support type has been selected

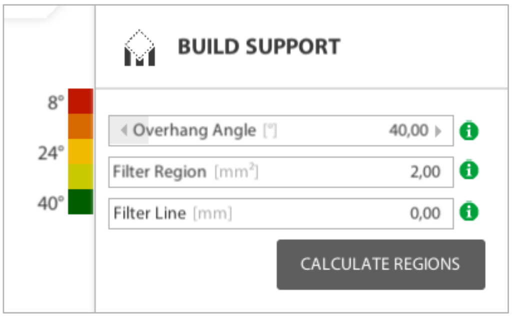


Fig. 12: Parameters for calculating support regions and colour scale of the Overhang Gradient

9.3 Creation of support regions

ⓘ When a part is loaded onto the substrate plate (**Orientation** action step), support regions are calculated automatically by AM-Studio . When you switch to the **Support** action step, these support regions become visible. If necessary, the support regions can be recalculated or generated manually.

Creation of support regions with CALCULATE REGIONS

☑ The **Support** action step is selected.

1. Click on a part.
   ⇨ Other parts are hidden.
2. Click on **[CALCULATE REGIONS]** in the function panel.

➲ All areas that require supports are coloured according to the "Overhang Gradient" colour scale.

Note

The areas are detected according to the criteria of the general support parameters (see Identification of support regions).

Manual creation of support regions

Note

> *In the model tree there is an overview of all automatically detected support regions for each part. The list of support regions is sorted by size in descending order.*

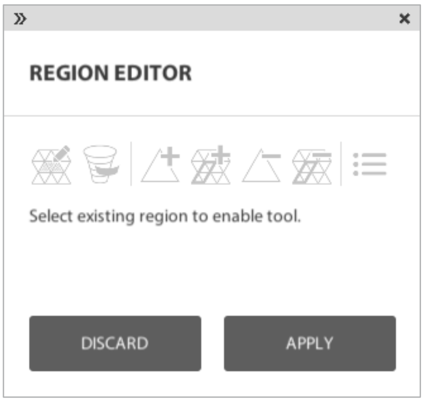
☑ The **Support** action step is selected.

1. Click on a part with the right mouse button.

   ⇨ A context menu opens.

2. Click **[Region] [Editor]**.

   ⇨ The Region Editor opens.



➲ The support regions can be edited manually.

The following functions are available for creating support regions:

Icon	Designation	Description
	Create Manually	Create manually defined support regions.
	Create From Overhang	Generation of manually defined support regions based on the overhang angle.

Tab. 34: Functions for creating support regions

In addition, the following functions are available to edit an already defined region:

Icon	Designation	Description
	Add Triangles	Adds triangles to the currently selected region.
	Merge Regions	Connects individual support regions to a large overall region.

Icon	Designation	Description
	Remove Triangles	Removes triangles from a region.
	Substract Regions	Removes a support region from another support region.
	Open Region List	Displays a sortable list of the support regions.

Tab. 35: Functions for editing support regions

Note

Refer to the information in the tooltips.

9.4 Creation of support structures

Automatic creation of supports

☑ The **Support** action step is selected.

☑ Support regions are calculated.

1. Click on the desired part.
   ⇨ Other parts are hidden.
2. Click on the icon of the desired support type in the function panel.
   ⇨ The icon turns green.
3. Define the parameters of the support type as desired or select a predefined support profile.
4. Click on **[GENERATE]** .

➲ Supports are created for the calculated areas.

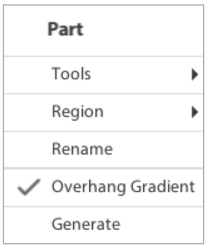
Generate supports for individual areas separately

☑ The **Support** action step is selected.

☑ Support regions are calculated.

1. Click on the desired part.
   ⇨ Other parts are hidden.
2. Click on the icon of the desired support type in the function panel.
   ⇨ The icon turns green.
3. Define the parameters of the support type as desired or select a predefined support profile.
4. Click on a support region.
   ⇨ The support region is marked.

5. Right click the marked support region.
   ⇨ A context menu opens.



6. Click on **[Generate]** in the context menu.

Note

Alternatively, click on [GENERATE] in the function panel.

➲ A support is generated for the selected support region.

Note

Several areas can be selected at once by holding down the <Ctrl> key.
To select all non-built support areas, click on Unassigned Surface or Unassigned Lines in the model tree.
If built-up support areas are selected again and another support type is chosen, the new supports are built into the existing structures.

Manual creation of individual Rod Supports

ⓘ Individual Rod Supports can be added to the part manually.

☑ The **Support** action step is selected.

☑ The support type rod support is selected in the function panel.

1. Click on the part.
   ⇨ Other parts are hidden.
2. Define the parameters of the rod supports as desired.
3. Navigate the cursor to the position on the part where a rod of the rod support is to be generated.
4. Press the <P> key on the keyboard.

➲ A single rod is generated at the current cursor position.

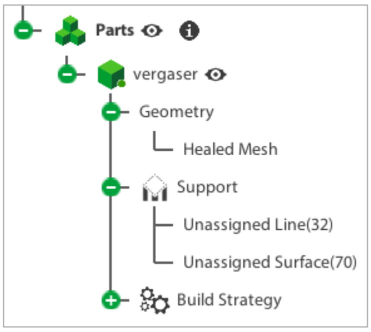
9.5 Deleting support regions

Deleting all support regions

☑ The Support action step is selected.

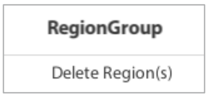
☑ Support regions are calculated.

1. Navigate to the desired part in the model tree.
2. Click on the item "Support" to open it.



3. Right-click on "Unassigned Surface" or "Unassigned Line".

   ⇨ A context menu opens.



4. Click on **[Delete Region(s)]** in the context menu.

➲ The support regions are deleted.

Deleting individual support regions

☑ The Support action step is selected.

☑ Support regions are calculated.

1. Click on a support region to be deleted on the part.

   ⇨ The support region is marked.

2. Right-click on the selected region.

   ⇨ A context menu opens.

3. Click on **[Region] [Delete]** in the context menu.

➲ The support region is deleted.

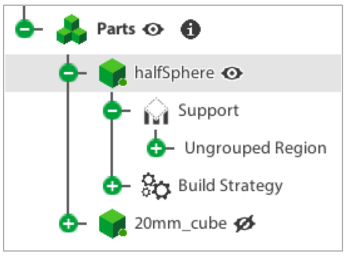
9.6 Deleting support structures

Deleting support structures

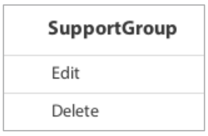
☑ The **Support** action step is selected.

☑ Supports are generated.

1. Navigate to the desired part in the model tree.
2. Click on the item "Support" to open it.
3. Click on the item "Ungrouped Regions" to open it.



4. In the model tree, right-click on the support to be deleted.
   ⇨ A context menu opens.



5. Click on **[Delete]** in the context menu.

➲ The support is deleted.

Note

Alternatively, a support structure can be selected directly on the part by clicking with the left mouse button (multiple selection possible by pressing the <Ctrl> key). Press the <Del> key to delete.
If several supports have been built on the region, make sure that the correct type is deleted.
Single, manually created Rod Supports are combined as a region and deleted as a whole in the manner described.
Supports in manually defined support areas are also deleted in the described manner.

Deleting separately generated Rod Supports

ⓘ Manually generated Rod Supports can be deleted individually.

☑ The **Support** action step is selected.

☑ Individual Rod Supports have been generated.

1. Click on the desired part.
   ⇨ Other parts are hidden.

2. Right-click on a rod that is to be deleted.
   ⇨ A context menu opens.

3. Click on **[Delete Support] [defaultRod]** in the context menu.

➲ The individual rod is deleted.

Note *Alternatively, a support structure can be selected directly on the part by clicking with the left mouse button (multiple selection possible by pressing the <Ctrl> key). Press the <Del> key to delete.*

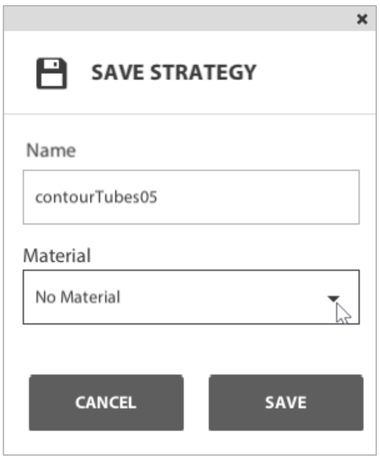
9.7 Managing support profiles

Creating a profile

Note *Support and build strategy profiles can be defined and saved depending on the material. If the filter button in the drop-down menu of the function panel is activated, only the profiles that match the material used are displayed.*

☑ The **Support** action step is selected.

1. Click on the icon of the desired support type in the function panel.
   ⇨ The icon turns green.
2. Define the parameters for the profile to be created as desired.
3. Click on the save symbol 🖫.
   ⇨ A window opens.



4. Enter a name for the profile in the **Name** input field.
5. Use the drop-down menu **Material** to select the material to which the profile can be applied.
6. Confirm the entry with a click on **[Save]**.

➲ The profile is created and appears in the drop-down menu of the function panel.

Loading a profile

☑ The **Support** action step is selected.

1. Click on the icon of the desired support type in the function panel.
   ⇨ The icon turns green.
2. Click on the filter icon to activate the filter function.
   ⇨ The profiles are filtered according to the material set in the settings.
3. Select the desired profile via the drop-down menu.

➲ The profile is loaded.

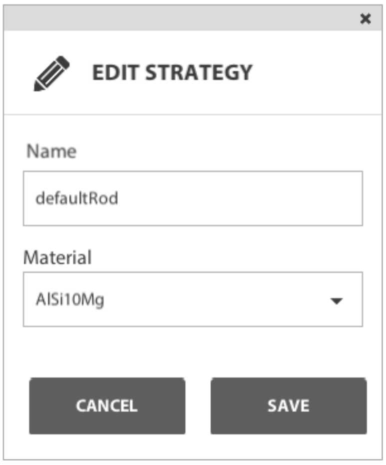
Editing a profile

☑ The **Support** action step is selected.

1. Click on the icon of the desired support type in the function panel.
   ⇨ The icon turns green.
2. Select the profile to be edited from the drop-down menu.

Note *Editable profiles are marked with a pencil symbol .*

3. Click on the pencil symbol behind the profile name.
   ⇨ A window opens.



4. Enter a name for the profile in the **Name** input field.
5. Use the drop-down menu **Material** to select the material to which the profile can be applied.
6. Confirm the entry with a click on **[Save]**.

➲ The profile is saved with the changes.

Removing a profile

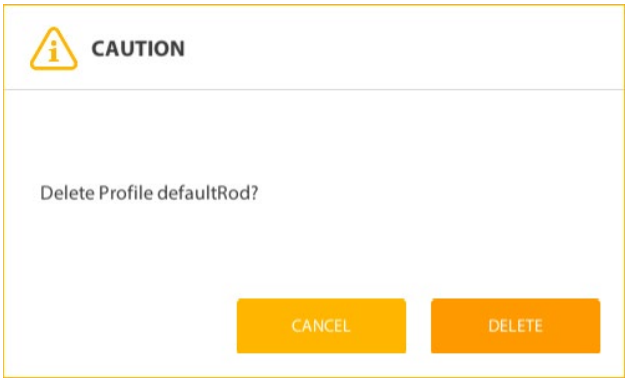
☑ The **Support** action step is selected.

1. Click on the icon of the desired support type in the function panel.
   ⇨ The icon turns green.
2. Select the profile to be deleted from the drop-down menu.

Note

Deletable profiles are marked with a bin icon 🗑.

3. Click on the bin icon 🗑 behind the profile name.
   ⇨ A warning message appears.

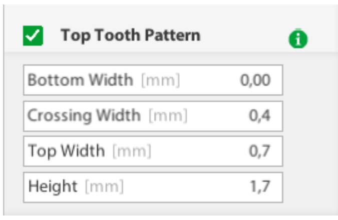
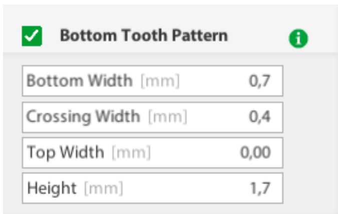


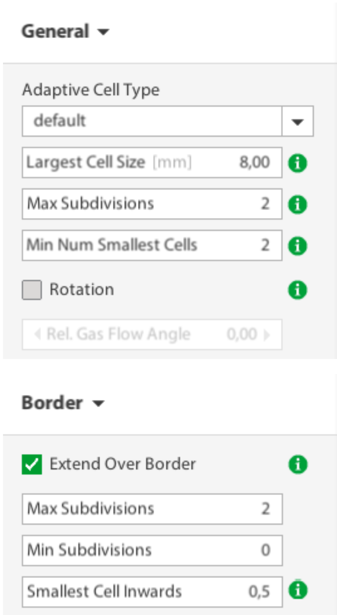
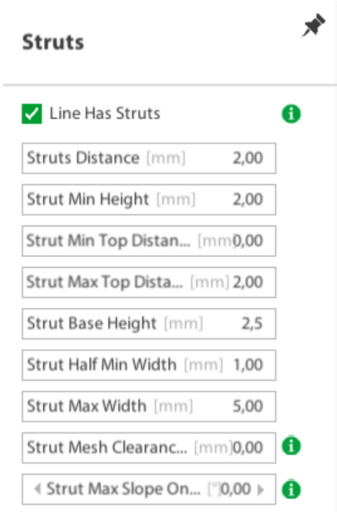
4. Confirm the deletion of the profile by clicking on **[Delete]**.

➲ The profile is deleted.

9.8 Conditions for support parameters

The following list lists conditions that must be fulfilled for error-free support generation.

Category	Subcategory	Condition	User Interface
All support types with the contour option "Tooth Pattern" (Rod, Block, Adaptive Cell, Line)	**Top Tooth Pattern**	Bottom Width + Crossing Width > 0.0 AND Top Width + Crossing Width > 0.0	
Support-Typen mit der Contour-Option "Tooth Pattern" (Rod, Block, und Line)	**Bottom Tooth Pat-tern**	Bottom Width + Crossing Width > 0.0 AND Top Width + Crossing Width > 0.0	

Category	Subcategory	Condition	User Interface
Adaptive Cell Support	**Cell settings**	Max Subdivisions (General) >= Max Subdivisions (Border)	
Line Support	**Struts**	Strut Max Width / 2 >= Strut Half Min Width AND Strut Max Top Distance >= Strut Min Top Distance	

Tab. 36: Conditions for support generation

10 STEP 4: NESTING

This chapter contains information about the **Nesting** action step.

Fig. 13: Progress bar: Nesting action step

If the same part is to be produced several times in one printing process, AM-Studio offers the following procedures for the efficient arrangement of several parts on the substrate plate:

- In manual nesting, the duplicates of the part are automatically placed in a matrix arrangement.
- With automatic 2D nesting, an attempt is made to utilise the build chamber as effectively as possible in order to produce the maximum number of parts in one printing process. An algorithm calculates the optimal distribution of the parts in the plane of the substrate plate.

10.1 Manual Nesting

This chapter describes the information on manual nesting in AM-Studio.

In manual nesting, a part is duplicated and distributed in a matrix arrangement on the substrate plate.

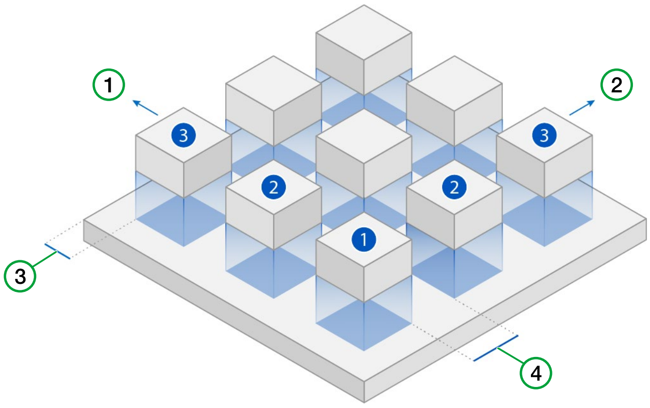


Abb. 14: Manual nesting

1 Y
2 X
3 To Edges
4 Between Parts Distance

Parameter	Description
Amount	
X	Defines the number of parts in the X-direction.
Y	Defines the number of parts in the Y-direction.
Fill Platform	If this parameter is activated, the substrate plate is completely filled with duplicates of the selected part.
Distance	
Between Parts X	Defines the minimum distance between the parts in the X-direction.
Between Parts Y	Defines the minimum distance between the parts in the Y-direction.

Parameter	Description
To Edges	Defines the distance of the parts to the edge of the substrate plate.

Tab. 37: Parameters for manual nesting

Creating a certain number of part duplicates

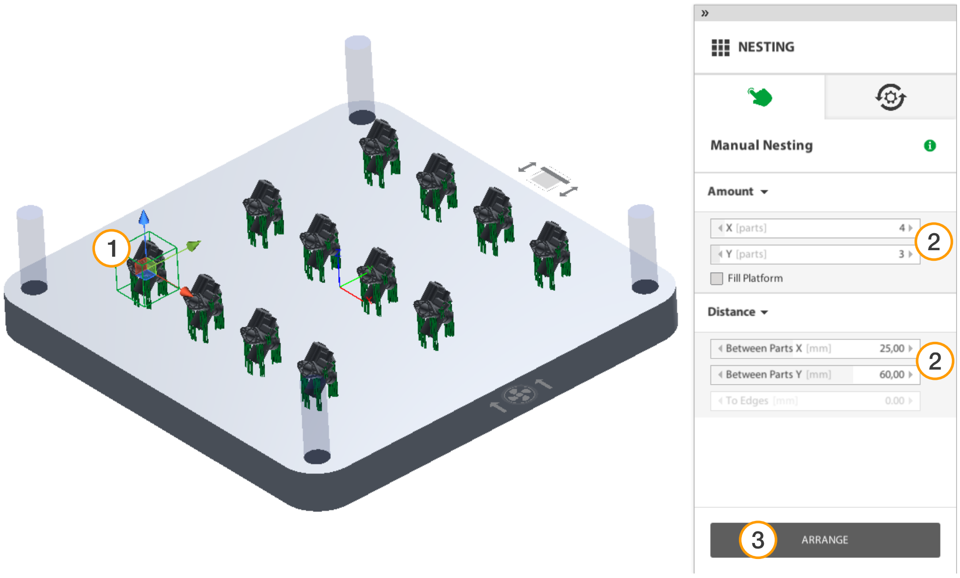


Fig. 15: Creating a certain number of part duplicates

☑ In the action step **Nesting**, the tab "Manual Nesting" is selected.

1. Select the part to be duplicated.
2. Set the desired values under **X**, **Y**, **Between Parts X** and **Between Parts Y**.
3. Click on the **[ARRANGE]** button.

➲ The part is reproduced in accordance with the selected parameters.

Filling the substrate plate with part duplicates

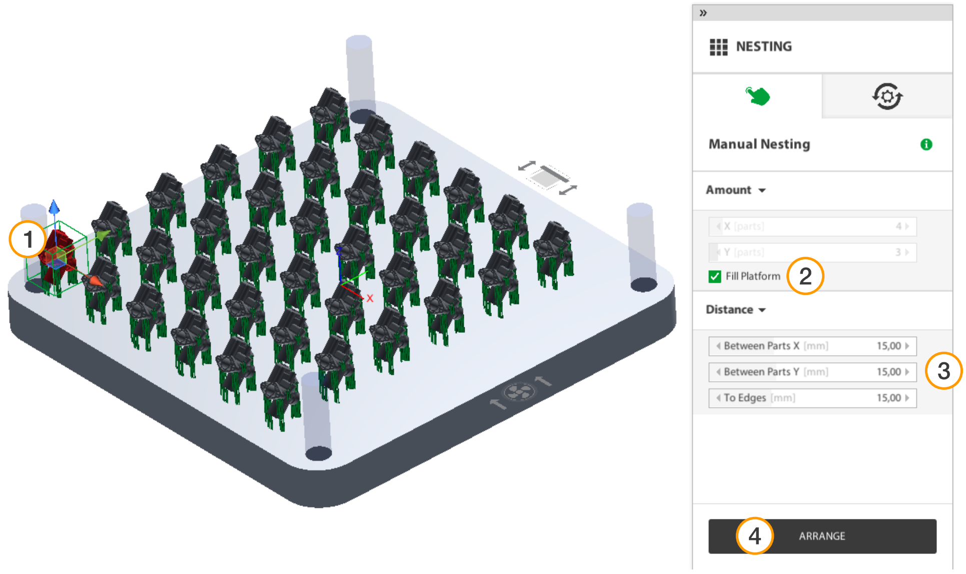


Fig. 16: Filling the substrate plate with part duplicates

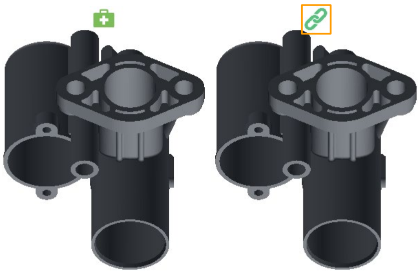
☑ In the action step **Nesting**, the tab "Manual Nesting" is selected.

1. Select the part to be duplicated.
2. Activate the parameter **Fill Platform**.
3. Set the desired values under **Between Parts X**, **Between Parts Y** and **To Edges**.
4. Click on the **ARRANGE** button.

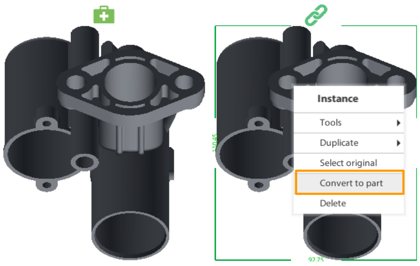
➲ The part is duplicated as often as possible on the substrate plate.

Note

Duplicates are initially instances of the original part, indicated by a chain symbol . 🔗*.*

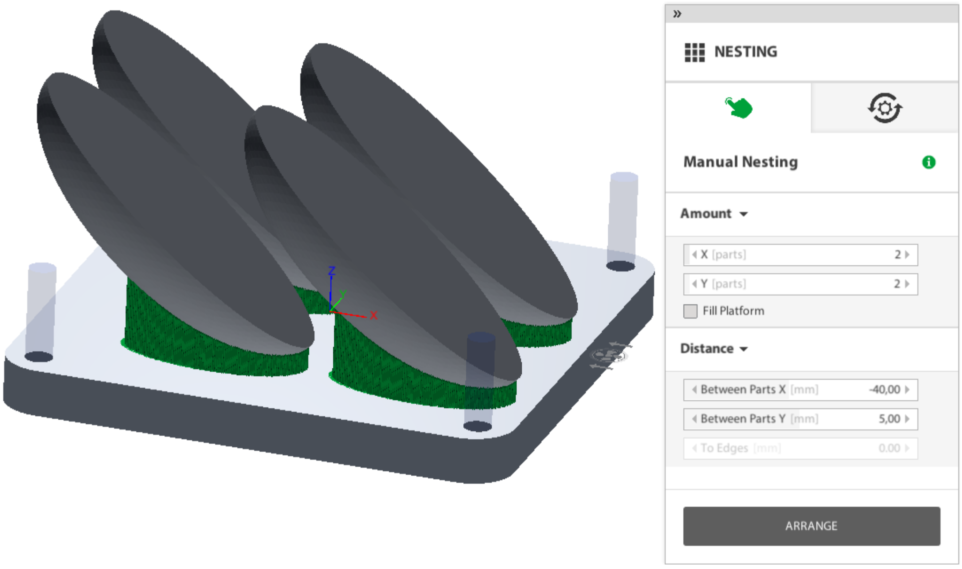


If the original part is changed, this change is adopted by all duplicates.
To change a duplicate independently of the original part, right-click on the duplicate and select "Convert to part" in the context menu.



Note

In order to optimally utilise the build chamber, Between Parts X and Between Parts Y can also take on negative values in order to push matching part duplicates into each other.



10.2 Automatic 2D nesting

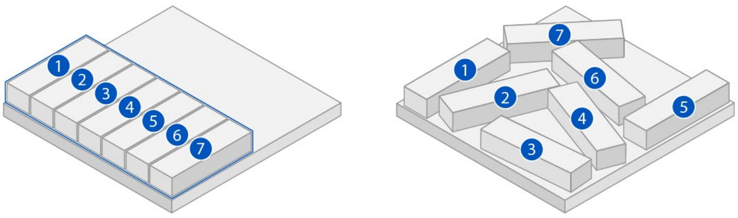
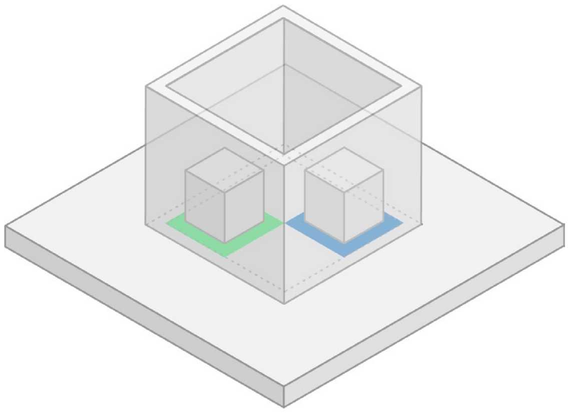
This chapter describes the information on automatic nesting in AM-Studio.

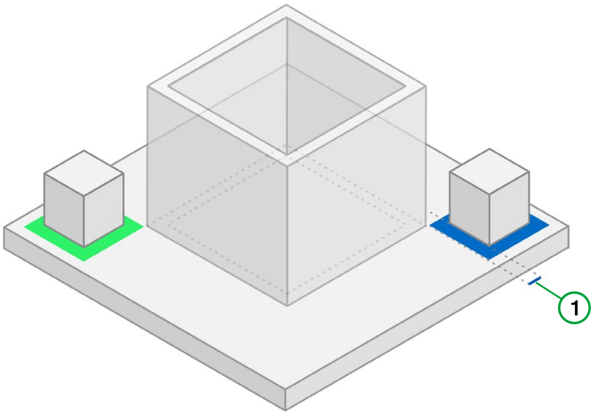
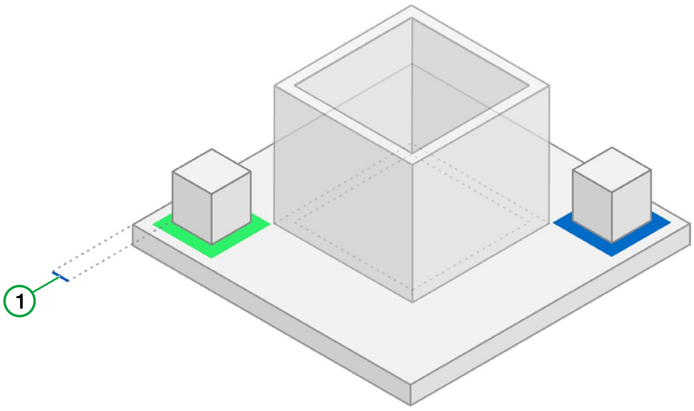
In automatic nesting, one or more parts are duplicated and automatically distributed on the substrate plate using an algorithm.

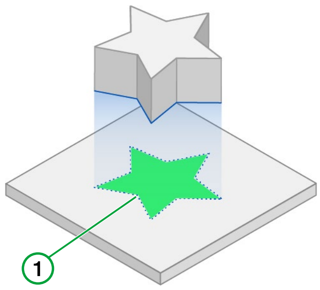
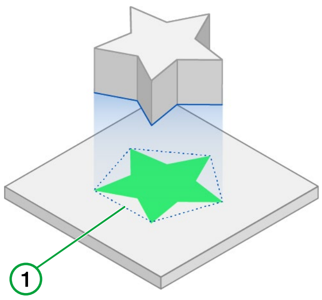
Note

2D nesting distributes the parts two-dimensionally on the substrate plate plane.
A vertical arrangement of the parts on top of each other with possible additional supports does not make sense.

Parameter	Description
	Three presets are available for the "Advanced Options" of automatic nesting: • **Quick Try** • **Normal Fit** • **Best Fit** These presets are graded according to precision and computational intensity. "Quick Try" offers the fastest results at the lowest precision, "Best Fit" provides the most precise results at the highest computational intensity.
Part List	
Name	Shows the name of a part located on the substrate plate.
No.	Defines the number of a part. This parameter is only displayed if **Fill Platform** is not activated. For **No.** > "1", duplicates of the original part are automatically created.
Ratio	Defines the percentage of a part in the number of all parts after nesting. This parameter is only displayed if **Fill Platform** is activated.
Z-Rot	When this parameter is activated, the algorithm can rotate the parts around their z-axis to achieve optimal utilisation of the substrate plate.
Fill Platform	If this parameter is activated, the algorithm tries to fill the substrate plate with the maximum possible number of duplicates.

Parameter	Description
Compactness	Defines the percentage filling of the substrate plate. This parameter is only editable if **Fill Platform** is activated. A complete filling of the entire substrate plate is often geometrically not possible. Experience shows that a compactness of more than 70 % should not be used. The following example shows a 50% filling of the substrate plate: 
Basic Operations	
Resolve Holes	If this parameter is activated, the algorithm tries to position parts within larger, hollow parts. 

Parameter	Description
Min Distance Between Parts	Defines the minimum normal distance between two parts.  **1** Min Distance Between Parts
Min Distance To Edge	Defines the minimum normal distance between the part and the edge of the substrate plate.  **1** Min Distance To Edge

Parameter	Description
Simplification Factor	Defines the degree of abstraction with which the algorithm simplifies the parts in order to be able to calculate the part placement more quickly. Example 1: Simplification Factor = 0.01 (Exact calculation):  **1** Calculated Shadow Example 2: Simplification Factor = 0.05 (Approximation):  **1** Calculated Shadow
Advanced Options	
An iteration consists of n generations and each generation has a certain population. A population is a randomly generated arrangement of the parts.	
Number Of iterations	Defines the number of development series.
Number Of Generations	Defines how many generations are created in a development series.
Population Size	Defines the number of individuals (arrangements) in one generation.
Translation Step	Defines the maximum displacement of individuals between generations.
Rotation Step	Defines the maximum rotation of individuals between generations.
Population Growth Factor	Defines the population size change between iterations (1.0 = unchanged).
Elitism	Defines how many individuals are carried over unchanged from one generation to the next (the best n individuals remain unchanged).

Parameter	Description
Tournament Size	Defines how many individuals can be considered as "parents" for the individual of the next generation (since only two may remain, comparisons are made from a value above "2").

Tab. 38: Parameters for automatic 2D nesting

Creating a certain number of part duplicates

☑ In the action step **Nesting**, the tab "Automatic Nesting" is selected.

1. Specify for each part how many times it should be present after nesting.
2. Under ***Basic Operations*** and ***Advanced Options***, set the parameters for the algorithm as desired.
3. Click on the **[ARRANGE]** button.

➲ The parts will be nested according to the selected parameters.

Filling the substrate plate with part duplicates

☑ In the action step **Nesting**, the tab "Automatic Nesting" is selected.

1. Activate the parameter **Fill Platform**.
2. Under **Ratio**, define the quantity ratio in which the parts are to be present after nesting.
3. Under **Compactness**, define the percentage coverage of the substrate plate with parts.
4. Under ***Basic Operations*** and ***Advanced Options***, set the parameters for the algorithm as desired.
5. Click on the **[ARRANGE]** button.

➲ The parts are duplicated as often as possible on the substrate plate.

11 STEP 5: BUILD PROCESS

This chapter describes the **Build Process** action step.

Fig. 17: Progress bar: Action step Build Process

In **Build Process**, there are the two areas in the function panel: ***Simulation*** and ***Build Strategy***.

Different end stages of parts in the build process can be simulated under ***Simulation***.

Under ***Build Strategy***, parameters for the following categories can be set:

- Gas flow
- Sections
- Build Job Scaling
- Slicing
- Hatching
- Parameterisation and assignment

11.1 Simulation

The **Build Process** action step offers a simulation function that calculates the deformation of a part during the build process.

With this knowledge, a pre-deformed part can be generated that will have the desired shape after the build process.

A simulation depends on the following factors:

- Machine
- Material
- Build strategy

Note

You can find information about the calibration of the AM machine in the "Simulation Calibration" PDF file. This can be downloaded from the menu Settings > Machines > Add Machine > Simulation Calibration.

11.1.1 Selecting a part stage

The following three stages can be simulated:

Icon	Designation	Description
	Part Finished	Simulates the finished part including the support structures and substrate plates. Use this stage if the part is still on the substrate plate during the subsequent heat treatment.
	Platform Removed	Simulates the finished part including support structures but without substrate plates. Use this stage if the part is removed from the substrate plate for the subsequent heat treat-ment. **Note** *This function is only available from the next update for users with a license that must be purchased separately.*
	Support Removed	Simulates the finished part after the support structures have been removed. Use this stage if: • the part is not going to be heat treated • the heat treatment will be carried out after the part has been removed from the sub-strate plate and the support **Note** *This function is only available from the next update for users with a license that must be purchased separately.*

Tab. 39: The three possible stages for the part simulation

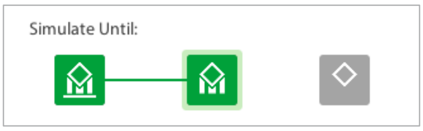
Selecting a part stage for the simulation

☑ The **Build Process** action step is selected.

☑ The ***Simulation*** tab is selected.

1. Click on the icon with the desired part stage in the function panel under “Simulate Until”. (Only "Part Finished" is available with the AM-Studio standard license)

   ⇨ The icon turns green and is connected to the icon in front of it with a line.



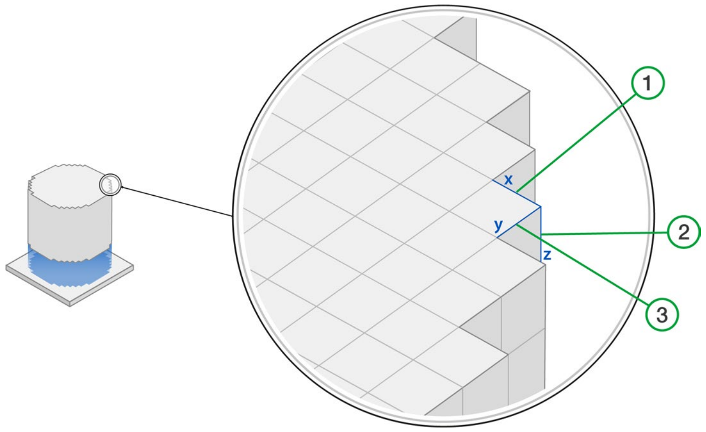
➲ The part stage that is to be simulated is selected.

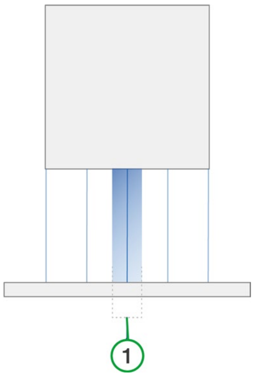
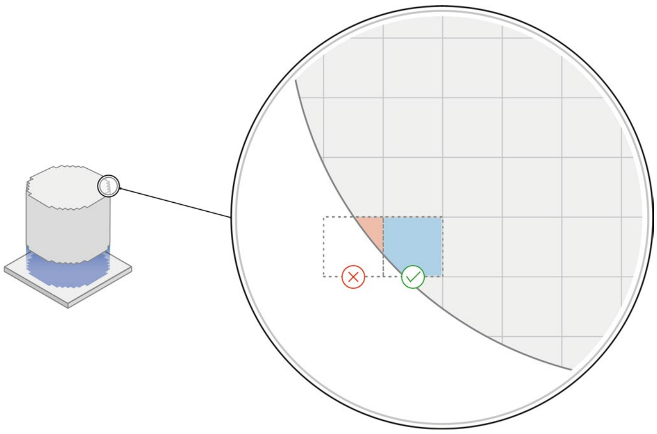
11.1.2 Voxel Mesh

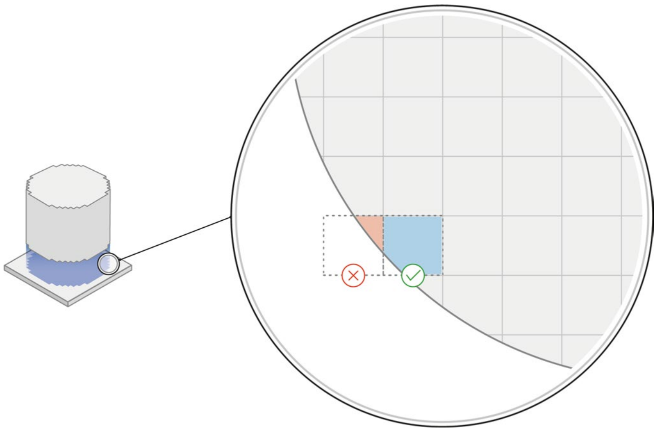
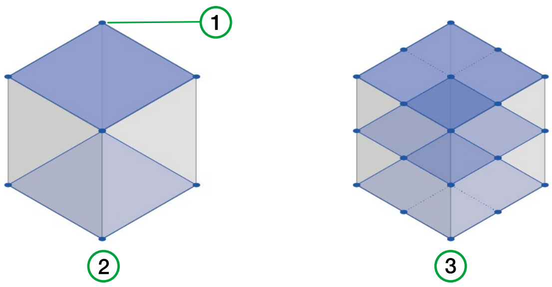
ⓘ In order to simulate a part, a Voxel Mesh must be generated. A Voxel Mesh splits a part into building blocks of a defined size.

ⓘ AM-Studio Calculates automatic values for each part that are then used as a basis to generate a Voxel Mesh. These values can be changed. The default values can be reloaded via a reset function.

The following table describes the parameters that can be defined during the generation of a Voxel Mesh.

Parameter	Description
Voxel Mesh	
Reset Values to Default	Clicking on the icon resets all entered values to the default values.
Uniform Voxel Size	When this parameter has been activated, the same values for the X, Y and Z direction are used to calculate the Voxels. As such, the Voxels are generated in the form of cubes. • **X/Y/Z Length**: Defines an identical value for the Voxel sizes in X, Y and Z direction
X Length	Defines the value for the Voxel length in X direction.  **1** X Length **2** Z Length **3** Y Length
Y Length	Defines the value for the Voxel length in Y direction.
Z Length	Defines the value for the Voxel length in Z direction.

Parameter	Description
Thickness	Defines the value for the thickness of the support structures. This value corresponds to the thickness of the laser melt pool.  **1** Support Thickness
Fraction of Part	Defines the percentage of the part that a Voxel covers. Voxel which do not reach the entered value, are not generated.  **Note** *The lowest possible value (as close to 0% as possible) should be selected in order to ensure that the part is completely covered with the Voxels.*

Parameter	Description
Fraction of Support	Defines the percentage of the support that a Voxel covers. Voxel which do not reach the entered value, are not generated.  **Note** *It is recommended that you set the value to 0% for support structures as if not, the Voxel Mesh of the support will be generated with holes.*
Precise Volume Fraction	If this parameter is activated, the number of sample points used for the calculation of the percentage of a part in a Voxel increases. The number increases from 8 (2 x 2 x 2) points to 27 (3 x 3 x 3) points.  **1** Voxel Data Point **2** Precise Volume Fraction Disabled **3** Precise Volume Fraction Enabled **Note** *When this function is activated, the calculation time for the Voxel Mesh increases but the results are more precise.*

Tab. 40: Parameter when generating a voxel mesh

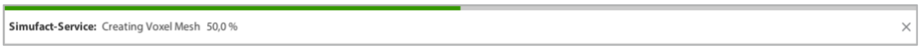
Voxel Mesh generate

☑ The **Build Process** action step is selected.

☑ The ***Simulation*** tab is selected.

☑ The desired values must be entered under ***Voxel Mesh***.

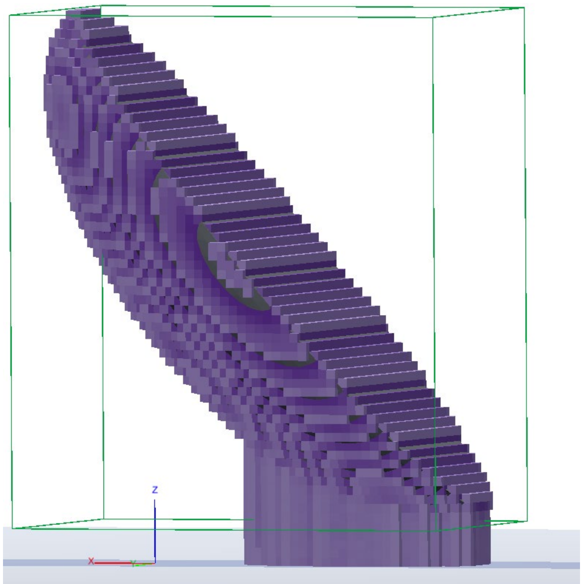
1. Click on the part that a preview needs to be generated for.
2. Click on **[Create]** in the ***Voxel Mesh*** window.
   ⇨ A bar that shows the progress of the Voxel Mesh calculation in percent is displayed.



➲ The Voxel Mesh is generated.

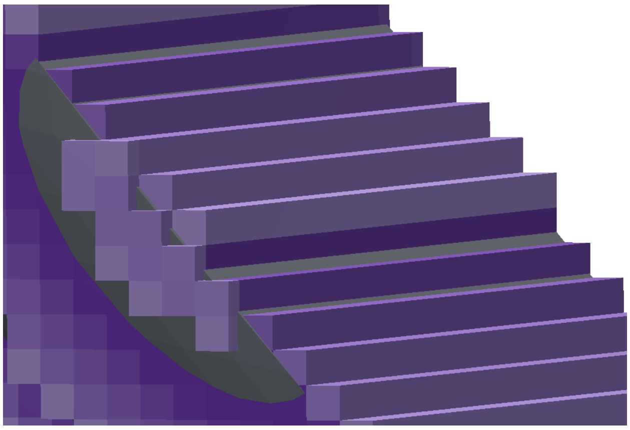
Display of the Voxel Mesh

ⓘ The generated Voxels used by AM-Studio during the part simulation can be viewed in the preview.



Note

The coloured shading of the Voxel shows what percentage of the part is in the respective Voxel. The darker the colour, the higher the percentage of the part in the Voxel. The lighter the colour, the lower the percentage of the part in the Voxel.
If the percentage falls below the limit, values defined in the parameters (standard value 5%), it will not be generated.
The following figure shows in detail Voxels in different levels of shading and points on a part that no Voxels have been generated for.



ⓘ The percentage values under **Fraction of Part** and **Fraction of Support** can be used to control whether a Voxel counts as valid or not.

For example, if the percentage value is set to zero, all Voxels that touch the part are generated.

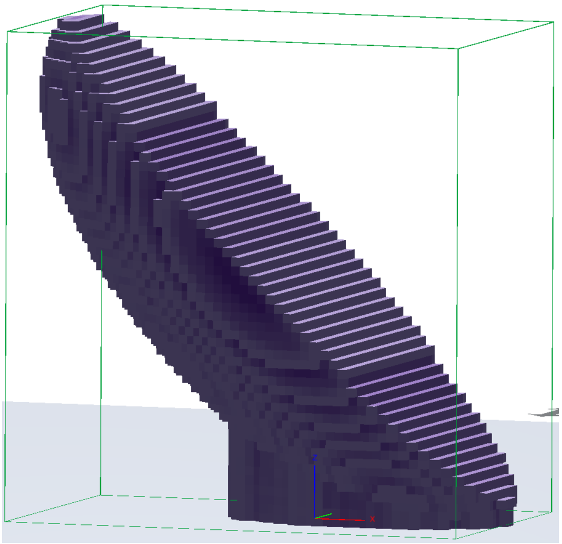


Fig. 18: Fraction of Part and Fraction of Support, each with a percentage value of zero.

If the percentage value is set to 100%, all Voxels are inside the part.

11.1.3 Starting the simulation

ⓘ A simulation can be started directly with no data needing to be defined in advance under ***Voxel Mesh***. AM-Studio then used the calculated default values. There is also no prerequisite for the generation of the Voxel Mesh.

ⓘ The simulation can be done with or without a *Simulation Calibration* of the machine being carried out first. If a machine *Simulation* is not carried out, the simulation is started with standard values instead.

☑ The **Build Process** action step is selected.

☑ The ***Simulation*** tab is selected.

☑ The desired *Simulation* is selected.

1. Click on the part that a simulation needs to be started for.

⇨ The part is surrounded by a green frame.

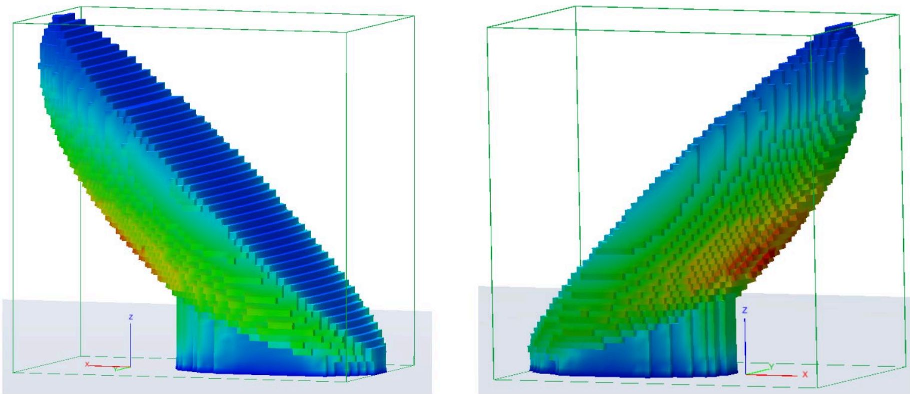
2. Click on **[Start]** in the function panel.

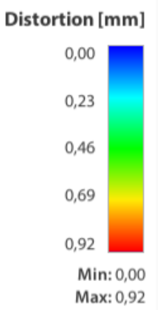
⇨ A bar that shows the progress of the simulation in percent is displayed.

➲ The part is simulated.

Display of a simulated part

ⓘ Once the simulation has been completed, the simulated part is displayed on the substrate plate. The Voxels are then coloured according to a scale that evaluates the deformation of the part.



Graphic	Description
	Thanks to the colour scale, areas of the part with minimal (blue) and maximal (red) deformation can be identified. The colour scale can be adjusted as follows by clicking: • Automatic [mm]: The min/max values from the simulation results are used. • Manual [mm]: The min/max values are entered by the user.

Tab. 41: *"Distortion" scale used to evaluate the deformation of a part*

11.1.4 Defining the pre-deformation (Manual Predeformation)

ⓘ With the "Manual Predeformation" function, a part can be pre-deformed. This pre-deformation should compensate any deformation effects that arise when a part is printed.

The simulated deformation value at a certain point on the part is multiplied with the given **Deformation Value** and in this way a new part is calculated.

The standard value for the **Deformation Value** is -0.95.

A **Deformation Value** of “1” generates the simulated, deformed part.

Note

The imported polygon mesh of the part is used to calculate the pre-deformation. As no curved, pre-deformed geometries can be generated on a triangle, points with very acute triangles (long triangles) do not provide satisfactory results.

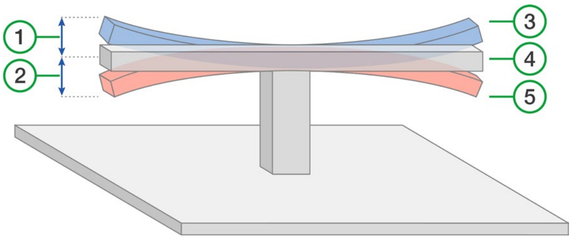


Fig. 19: Deformation

1 Deformation
2 Deformation Value · Deformation
3 Simulated part
4 Part
5 Pre-deformed part

☑ A part is simulated.

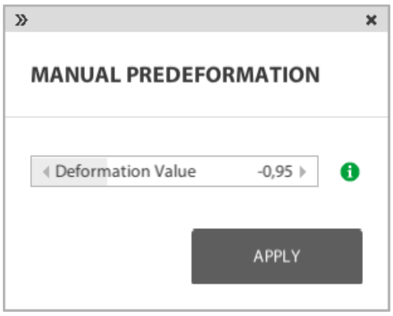
☑ Values for the pre-deformation that is to be applied are available.

Note

The "Manual Predeformation" input window opens automatically directly after a simulation. Only carry out steps 1 and 2 when the window is not open.

1. Right click on the part for which a pre-deformation needs to be defined.
   ⇨ A context menu opens.

2. Click on **[Tools] [Predeformation]** in the context menu.
3. An input window opens.



4. Enter the value for the desired pre-deformation.

➲ The part is pre-deformed.

Pre-deformation on the part

ⓘ The following figure shows how a pre-deformation can affect a part. The figure shows the part after a successful simulation in the **Orientation** action step.

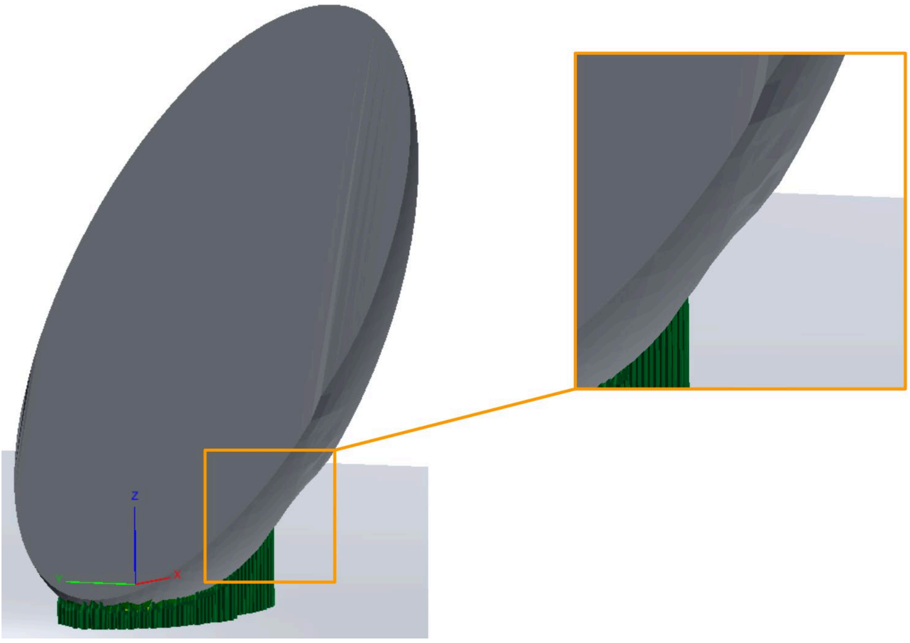


Fig. 20: Pre-deformation example

11.2 Build Strategy

11.2.1 Sections

With what are referred to as the ***Sections*** of the action step **Build Process**, the parts on the substrate plate can be subdivided in Z-direction to enable an effective preparation.

In order to achieve a faster print time, different parameter profiles can be assigned to the parts. For example, depending on the additive manufacturing system, certain areas of the part can be built with a greater layer thickness.

Note

> *Only integer multiples of the minimum layer thickness should be used for the build process, otherwise the number of coatings can increase significantly.*

Creating sections

☑ The **Build Process** action step is selected.

1. Click on the tab ***Build Strategy*** in the function panel.

2. Right-click on the part to be subdivided.

⇨ A context menu opens.

3. Click on **[Define Sections]** in the context menu.

   ⇨ A window opens.

4. Click on ⊕.

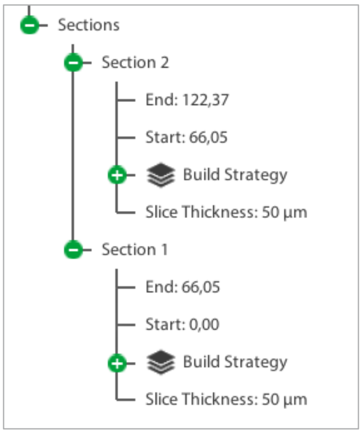
   ⇨ The part is automatically divided into two sections of equal size.

5. Enter the cutting height in the Z direction in the input field.

   ⇨ The cutting heights of the different sections are displayed in the model tree.

6. Select individual sections in the model tree.
7. In the function panel, click on **[ASSIGN TO SELECTED]** to link the selected sections with exposure strategies.

➲ The assigned exposure strategy and layer thickness are displayed in the model tree.



Note | *Using different layer thicknesses in one part generally allows for faster production times.*

11.2.2 Slicing

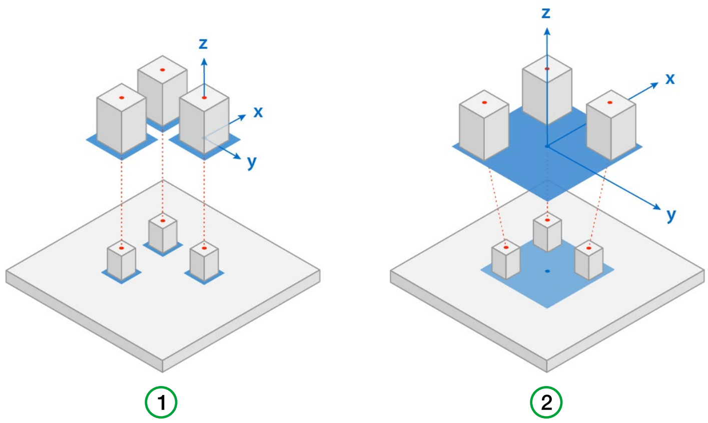
The ***Slicing*** area is in the **Build Process** action step, under the ***Build Strategy*** tab.

During the slicing, the part is sliced into horizontal layers with a defined layer thickness.

Slicing reveals the vector types used during the build process in each layer of the metal powder.

11.2.2.1 Scaling parameters

Parts can be scaled in AM-Studio to compensate for a shrinking of the part after cooling, for example.

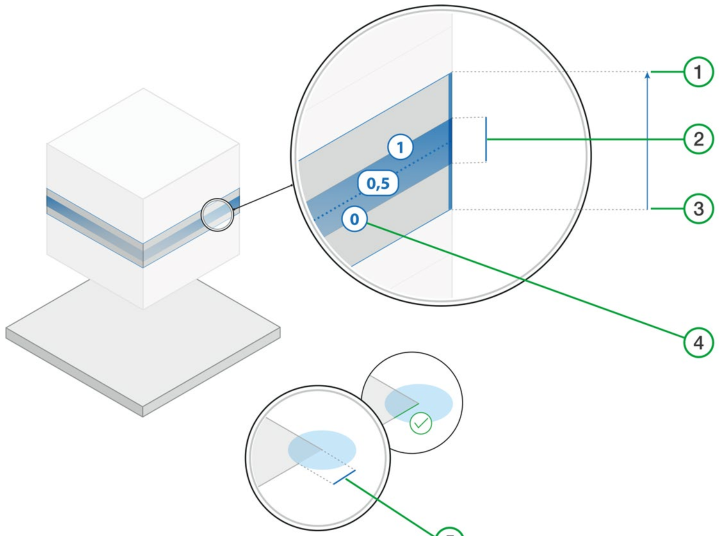
Parameter	Description
Scaling	
	 **1** Scale Around Center deactivated **2** Scale Around Center activated
Scale Around Center	If this option is activated, the parts are scaled around the centre of the substrate plate. This results in a shift of the parts on the X and Y axis. This prevents merging of part areas during upscaling. If this option is deactivated, each individual part is scaled around its own centre point.

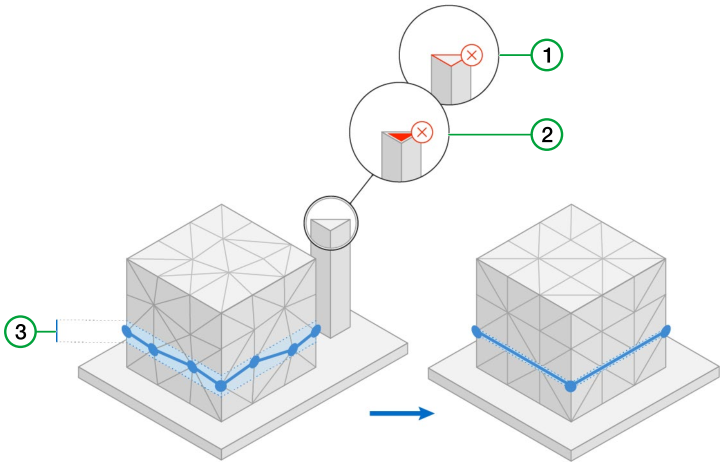
Parameter	Description
X	Defines the factor for scaling in the X-direction.
Y	Defines the factor for scaling in the Y-direction.
Z	Defines the factor for scaling in the Z-direction.

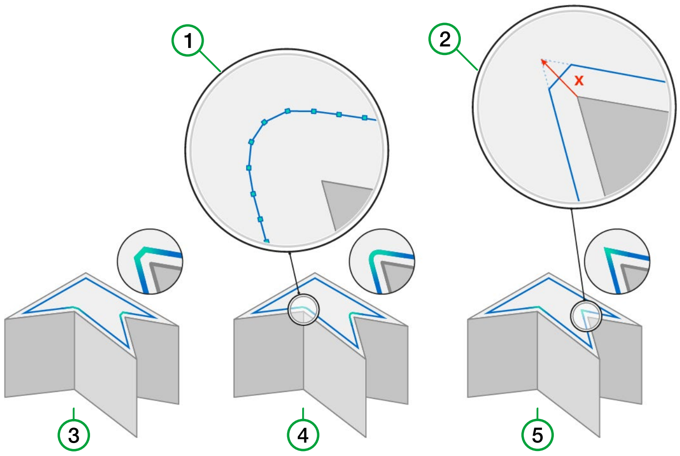
Tab. 42: Build Strategy: Slicing-Parameter "Scaling"

11.2.2.2 Part parameters

The filter settings for Slicing are described below. This is a maximum representation of all possible parameters. Depending on the type of machine used, not all parameters may be available.

Parameter	Description
General	
Slicing	 **1** Max Z **2** Slice Thickness **3** Min Z **4** Slice Height Factor **5** Gap Tolerance
Slice Thickness	Defines the thickness of the individual layers of metal powder.
Gap Tolerance	Defines the maximum length of gaps between vectors. Gaps shorter than the defined value are automatically closed.
Slice Height Factor	Defines the location within the layer for Slicing. The value "0" selects the lowest area of a slice, the value "0.5" the middle of the slice and the value "1" the top area of a slice.

Parameter	Description
Use Limits	If this parameter is activated, the Z-dimensions of the part can be defined between which the Slicing is to be carried out. This way, only a limited area of the part is analysed and time is saved during tests, for example. • **Max Z**: Defines the position on the Z-axis from which Slicing is to be terminated. • **Min Z**: Defines the position on the Z-axis from which Slicing is to be started. **Note** *No printing is possible for Min Z > "0".* *For Max Z < Max Z of all parts, printing is only possible up to that point.*
Filter	
Contour Filter	 **1** Min Area **2** Min Length **3** Point Filter Tolerance
Filter Area	If this parameter is activated, areas smaller than the area defined under **Min Area** are not exposed. • **Min Area**: Defines the minimum size of an area to be exposed. Areas smaller than the specified value are not exposed.
Filter Length	If this parameter is activated, areas with a shorter outer contour than defined under **Min Length** will not be exposed. • **Min Length**: Defines the minimum length of an outer contour from which exposure takes place. Areas that have a shorter outer contour than the defined value will not be exposed.

Parameter	Description
Point Filter	If this parameter is activated, consecutive vectors within a defined layer are combined into a longer vector. The total number of vectors will be reduced. • **Point Filter Tolerance**: Defines the width of the layer within which consecutive vectors are combined.
Offset Parameter	 **1** Arc Precision **2** Miter Limit **3** Offset Square **4** Offset Round **5** Offset Miter Under Offset Parameters, parameters can be defined for offset geometries that are created by shifting part edges inwards. The displacement creates gaps at inner corners within the offset geometry.
Miter Limit	Defines the factor that is multiplied by the value under **Beam Compensation**. The result defines the maximum distance of the outermost border to the actual edge of the part up to which the strategy **OFFSET_MITER** is applied. From a greater distance, the strategy **OFFSET_SQUARE** is automatically applied to the edge in question.
Arc Precision	For **Offset Strategy** = **OFFSET_ROUND**: Defines the deviation of the outermost border from the targeted offset geometry at a rounded corner. This deviation occurs because a rounded corner cannot be reproduced exactly using straight lines.

Parameter	Description
Offset Strategy	Defines the strategy for closing gaps at inner corners of the offset geometry. • **OFFSET_ROUND**: With this strategy the offset geometry is rounded at inner corners. • **OFFSET_MITER**: With this strategy, the offset geometry at inner corners is exposed as a peak up to a **Miter Limit** over defined distance. If the **Miter Limit** value is exceeded, the strategy **OFFSET_SQUARE** is automatically applied. • **OFFSET_SQUARE**: With this strategy, the offset geometry at inner corners is cut straight and thus the corner is angled.

Tab. 43: Build Strategy: Slicing Parameter "Part"

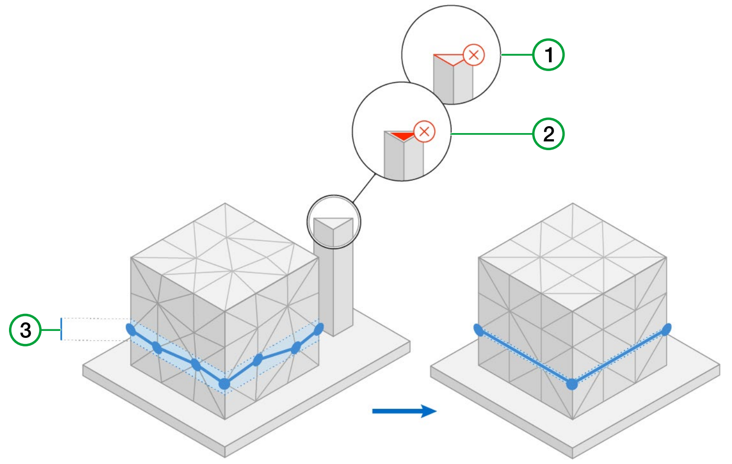
11.2.2.3 Support parameters

The settings for the exposure strategy of support structures can be defined under *Support*.

The following overview is a maximum representation of all possible parameters. Depending on the type of machine used, not all parameters may be available.

Parameter	Description
Skipped Layers	Defines the layer thickness with which a support structure is to be built up. • **Skipped Layers** = 0: The support structure is built up with the same layer thickness as the part. Example: Part: 30 µm, support structure: 30 µm • Skipped Layers = 1 – 4: The support structure is only printed in every second to fifth layer. Example: 30 µm part ≈ 60 µm support

Sort Strategy	
	Lets you select an exposure order of the support structure vectors. • **Nearest Neighbour**: With this method, the closest vector is exposed next. The settings in **Angle Tolerance** and **Orthogonal Direction Weight** have no function. • **Nearest Neighbour Direction**: With this method, the closest vector is exposed next. The values defined in **Angle Tolerance** and **Orthogonal Direction Weight** represent the weighting parameters. • **Direction Clustering**: This method determines a main orientation of the support vectors for block supports and adaptive supports the structures of which have two orthogonal orientations. The two directions are determined via the values under **Angle Tolerance** and **Orthogonal Direction Weight**. The support vectors are divided into three groups as a result of the defined values, which are exposed one after the other: – Group 1: All support vectors in the determined main orientation. Exposure of the support vectors from right to left. – Group 2: All support vectors orthogonal to the determined main orientation. Exposure of the support vectors from top to bottom. – Group 3: All other support vectors, e.g. support vectors in the contour. Exposure of the support vectors according to the method Nearest Neighbour. • Angle Tolerance: Defines the tolerance up to which support vectors are assigned to the first group. For example, with a value of "3", all support vectors with an orientation of ±3° to the main orientation are considered support vectors in the main orientation. • Direction Weight: Defines the tolerance up to which support vectors are assigned to the second group. For example, with a value of "3", all support vectors with an orientation of ±3° to the orthogonal orientation are considered support vectors of the second group.

Filter	
	 **1** Filter Min Area **2** Filter Min Length **3** Tolerance
Filter Area	If this parameter is activated, areas smaller than defined under Min Area are not exposed. • **Min Area**: Defines the minimum size of an area to be exposed. Areas smaller than the defined value are not exposed.
Filter Length	If this parameter is activated, areas with a shorter outer contour than defined under Min Length will not be exposed. • **Min Length**: Defines the minimum length of an outer contour from which exposure takes place. Areas that have a shorter outer contour than the defined value will not be exposed.
Point Filter	If this parameter is activated, consecutive vectors within a defined layer are combined into a longer vector. The total number of vectors will be reduced. • **Point Filter Tolerance**: Defines the width of the layer within which consecutive vectors are combined.
Remove Overlapping Contours	If this parameter is activated, contours that overlap are removed. • **Angle Tolerance**: Defines the angle between two support vectors from which the criterion for removing one of the two support vectors is considered to be fulfilled. Support vectors with a larger angle to each other are used for the removal. If the value under Tolerance is also exceeded, one of the two support vectors is removed. • **Tolerance**: Defines the distance between two support vectors from which the criterion for the removal of one of the two support vectors is considered to be fulfilled. Support vectors with a larger distance to each other are used for the removal. If the value under Angle Tolerance is also exceeded, one of the two support vectors is removed.

Tab. 44: *Build Strategy: Slicing Parameter "Support"*

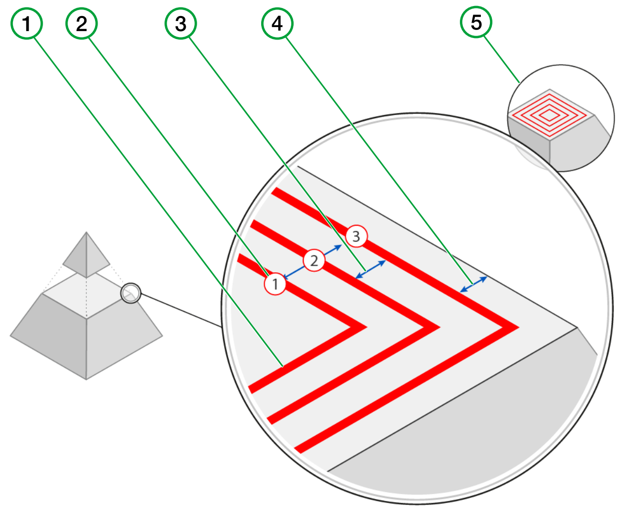
11.2.3 Hatching

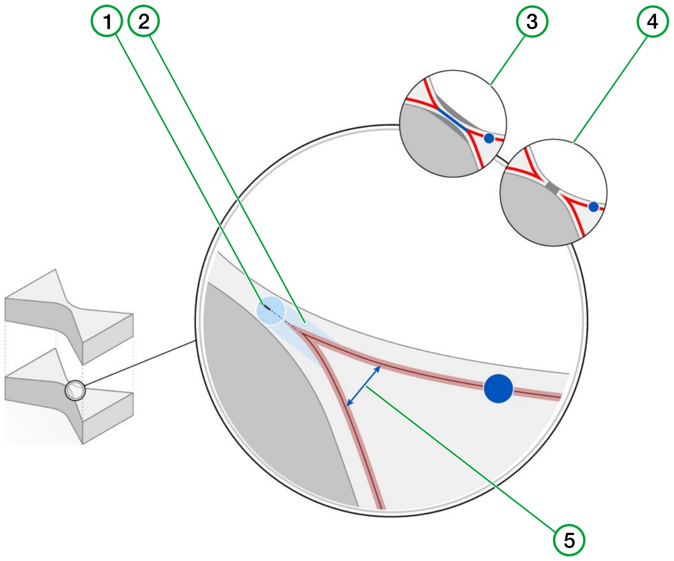
The *Hatching* area is in the Build Process action step, under the ***Build Strategy*** tab. The individual vectors that will be laser exposed and solidified by the AM machine are defined under ***Hatching***.

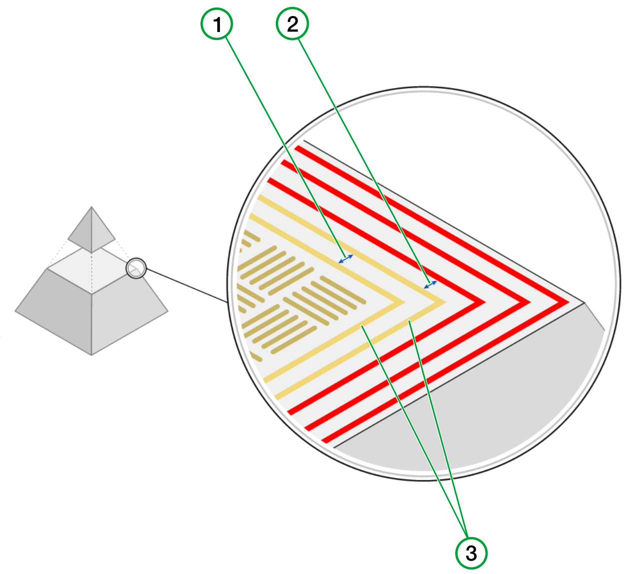
11.2.3.1 Volume Hatch

The following describes the hatching settings for the "Volume" item. The "Volume" is the inside of the part – the area inside a closed contour.

The following overview is a maximum representation of all possible parameters. Depending on the type of machine used, not all parameters may be available. The hatching settings for the different hatch types are described in detail under *Hatch types*.

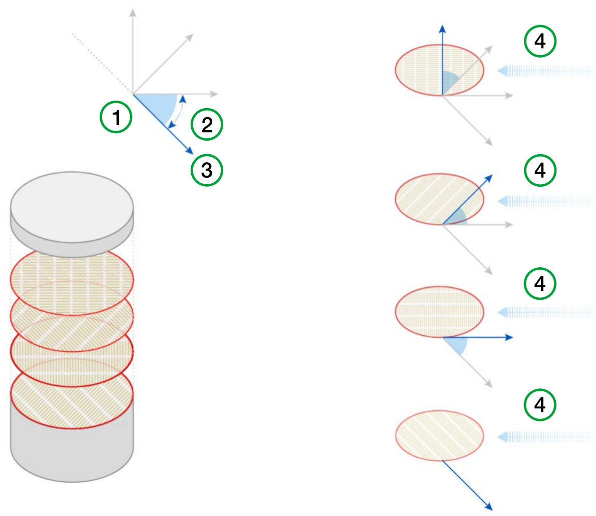
Parameter	Description
Borders	
	 **1** Number Of Border **2** Sort Borders Outwards **3** Border Distance **4** Beam Compensation **5** Fill Contour
Sort Borders Inwards	If this parameter is activated, the borders are generated from the outside to the inside.
Number of Border	Defines the number of borders to be generated.
Border Distance	Defines the distance between the borders.
Beam Compensation	Defines the inward indentation of the outermost border. This can be used to compensate for the width of the laser melt pool.
Contour Fill	If this parameter is activated, the entire exposure area is filled with borders.

Blocked Paths	 **1** Beam Compensation = Radius Correction Factor = 1.0 **2** Overdraw **3** Erase Blocked Path = Deactivated **4** Erase Blocked Path = Activated **5** Angle Threshold If this option is activated, parameters for Blocked Paths can be defined. Blocked Paths are areas that appear wider than desired during an exposure due to the width of the laser beam. If the area is not exposed, a gap is created in the part.
Traversal Strategy	Defines the exposure method of the blocked paths: • **CONNECT_UNIDIRECTIONAL:** The blocked path is exposed once. The exposure parameters can be defined in the submenu *Scanning* under Blocked Path - Volume. • **CONNECT_BIDIRECTIONAL**: The blocked path is exposed twice. The exposure parameters can be defined in the submenu Scanning under Blocked Path - Volume. • **MERGE**: The blocked path is exposed once. The exposure parameters correspond to those of the border.
Erase Blocked Path	If this parameter is activated, no vectors are set if the laser beam is wider than the part area to be exposed. This means that no connection is created between the areas and a gap is created in the part. This function is disabled by default.
Angle Threshold	Defines the angle of part areas from which blocked paths are generated. This causes the laser to leave the original path of the outermost border and continue to expose in the direction of the corner, so that this is also correctly exposed.

	Correction Factor Defines the length by which the inserted blocked path is shortened from the outside to the inside of a corner. The actual shortening is composed of the value for the **Beam Compensation** and the **Correction Factor** by multiplying the two values. Accordingly, if the value is "0", no shortening of the Blocked Path is made, so that exposure is made to the outermost edge of the part. This can lead to overexposure. With a value of "1", however, the Blocked Path is shortened from the outside to the inside by the **Beam Compensation**. With a value of "2" the Blocked Path is shortened from the outside to the inside by twice the **Beam Compensation**.
	Parabola Precision Defines the exposure accuracy for curved Blocked Paths.
Fill Contour	
	 **1** Fill Contour Distance **2** Fill Contour Offset **3** Number Of Fill Contours Under *Fill Contours* the settings for the vector type Fill Contour are defined.
Sort Fill Contours Inwards	If this parameter is activated, the Fill Contours are generated from the outside to the inside.
Number Of Fill Contours	Defines the number of Fill Contours to be generated.
Fill Contour Distance	Defines the distance between the Fill Contours.
Fill Contour Offset	Defines the distance between the outermost Fill Contour and the innermost Border.

Hatch	
Hatch types	The following hatch types are available for volume areas: • *Parallel* • *Stripes* • *Chess* • *Lattice* • *None*

Hatch Rotation

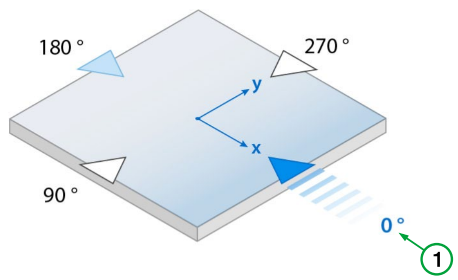


1 Limitation Window max. 180° **3** Angle

2 Angle Increment **4** Gas Flow

The hatch pattern can be rotated from layer to layer. This prevents each layer from being exposed in the same order and at the same angle to the gas flow.

- **Align with Gas Flow**: If this parameter is activated, the hatches are exposed against the direction of the gas flow.
- **Start Angle**: Defines the starting angle between the direction of the gas flow and the orientation of the hatches.
- **Limitation Window**: Defines the limited angular range in which the rotation of the hatch pattern from layer to layer takes place. If the parameter Align with Gas Flow is activated and the hatches are exposed against the gas flow, the rotation of the hatch pattern can only be performed in a range of 180°. The remaining 180° are skipped during the rotation so that an unwanted alignment of the hatches with the direction of the gas flow is avoided. If the parameter Align with Gas Flow is deactivated, the entire range of 360° is available.
- **Angle Increment**: Defines the size of the angle by which the hatch pattern is rotated after each layer.
- **Gas Flow Direction Angle**: Defines the direction of the gas flow.



1 Direction of the gas flow

Tab. 45: Build Strategy: Hatching parameter "Volume"

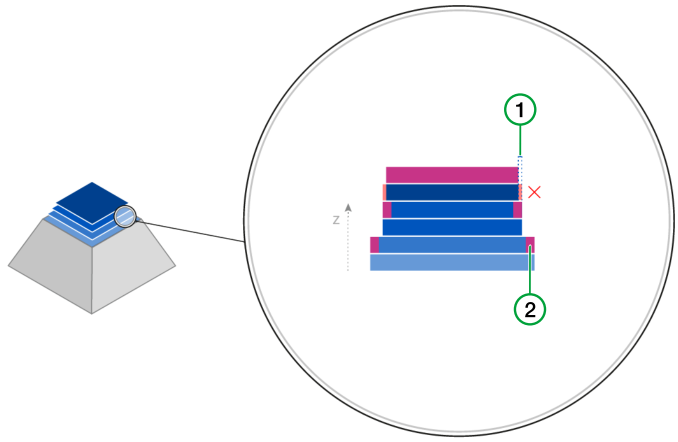
11.2.3.2 Up Skin Remelting Hatch

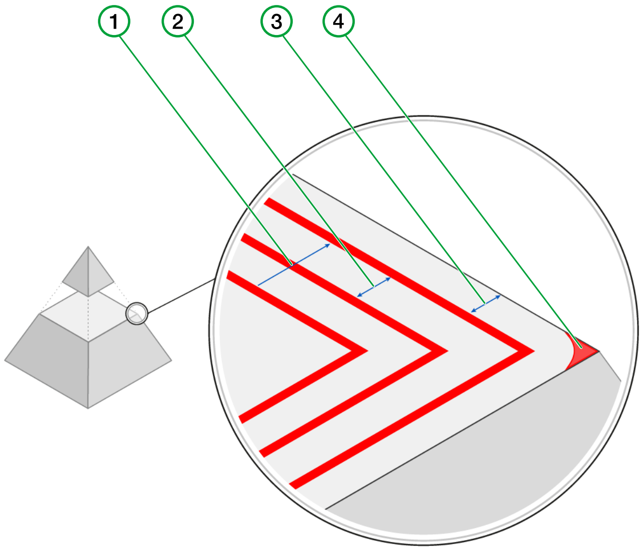
This chapter describes the hatching parameters for the ***Up Skin Remelting*** area.

Note

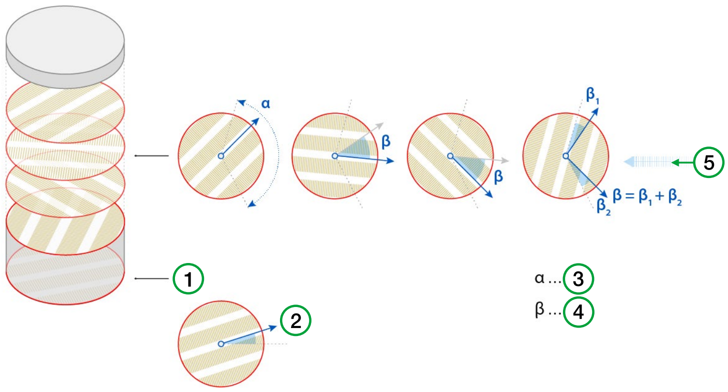
> *Difference between Up Skin Recoating and Up Skin Remelting:*
> *Up Skin Recoating follows a volume exposure. The coater first applies a new layer of powder (with the same Z-height of the substrate plate). Subsequently, the borders and hatches are exposed according to definition in Up Skin Recoating.*
> *Up Skin Remelting follows a volume exposure. In that case, no further coating with fresh powder is carried out. The borders and hatches are exposed directly on the volume layer.*

Parameterisation – Up Skin Remelting Hatch

Parameter	Description
General	
Tolerance	 **1** Tolerance **2** Up Skin Defines the minimum size of an area that is recognised as Up Skin. Areas smaller than the defined value are assigned to the volume.

Parameter	Description
Minimum Width	 **1** Inside Out **2** Border Distance **3** Offset **4** Minimal Up Skin Area Enabled Defines a limit value for an area where vectors are not created. This can be used to prevent the exposure of narrow areas.
Border	
Border	If this parameter is activated, a border is defined for this Hatch area (see *Borders*). • **Offset**: Defines the inward indentation of the outermost border. This can be used to compensate for the width of the laser melt pool. **Note** *For Up Skin, the term "Offset" is used instead of "Beam Compensation".*
Hatch	
Hatch types	The following hatch types are available for Up Skin areas: • *Parallel* • *Stripes* • *Chess* • *Lattice* • *None*

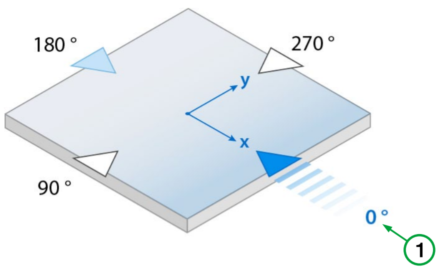
<table>
<tr><th>Parameter</th><th>Description</th></tr>
<tr><td>Hatch Rotation</td><td>



1 First Layer
2 Start Angle
3 Limitation Window
4 Angle Increment
5 Gas Flow

The hatch pattern can be rotated from layer to layer. This prevents each layer from being exposed in the same order and at the same angle to the gas flow.

- **Align with Gas Flow**: If this parameter is activated, the hatches are exposed against the direction of the gas flow.
- **Start Angle**: Defines the starting angle between the direction of the gas flow and the orientation of the hatches.
- **Limitation Window**: Defines the limited angular range in which the rotation of the hatch pattern from layer to layer takes place. If the parameter Align with Gas Flow is activated and the hatches are exposed against the gas flow, the rotation of the hatch pattern can only be performed in a range of 180°. The remaining 180° are skipped during the rotation so that an unwanted alignment of the hatches with the direction of the gas flow is avoided. If the parameter Align with Gas Flow is deactivated, the entire range of 360° is available.
- **Angle Increment**: Defines the size of the angle by which the hatch pattern is rotated after each layer.
- **Gas Flow Direction Angle**: Defines the direction of the gas flow.



1 Direction of the gas flow

</td></tr>
</table>

Tab. 46: Build Strategy: Parameters for the Up Skin Remelting Hatch

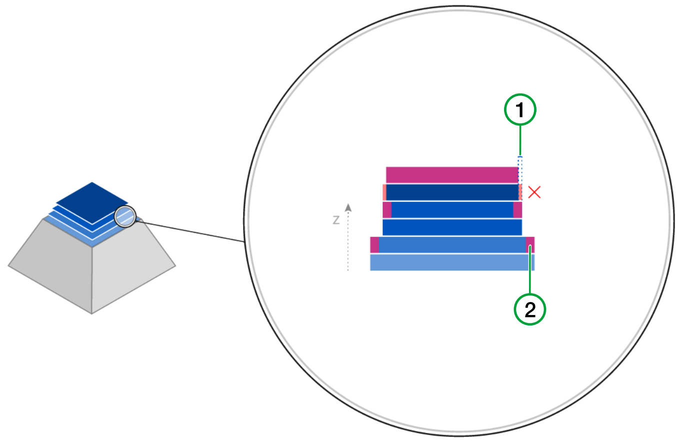
11.2.3.3 Up Skin Recoating Hatch

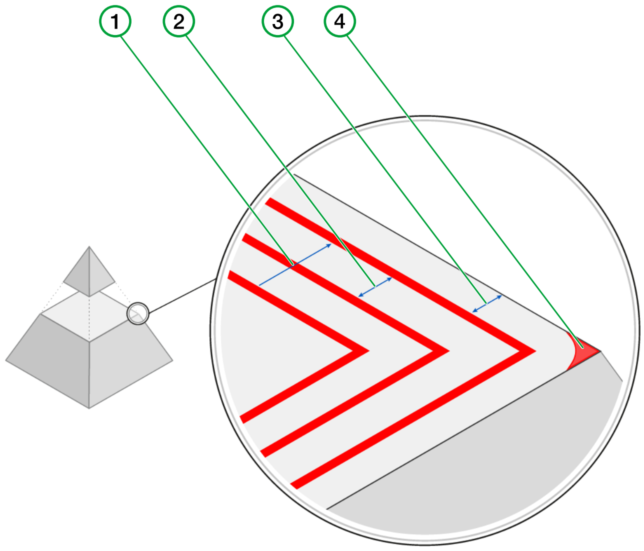
This chapter describes the hatching parameters for the ***Up Skin Recoating*** area.

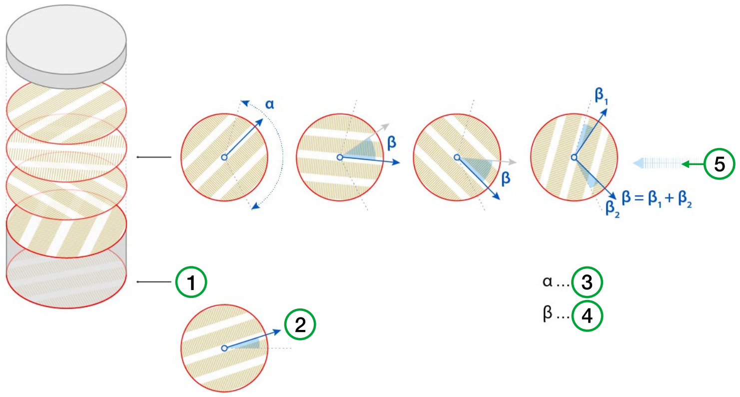
Note

Difference between Up Skin Recoating and Up Skin Remelting:
Up Skin Recoating follows a volume exposure. The coater first applies a new layer of powder (with the same Z-height of the substrate plate). Subsequently, the borders and hatches are exposed according to definition in Up Skin Recoating.
Up Skin Remelting follows a volume exposure. In that case, no further coating with fresh powder is carried out. The borders and hatches are exposed directly on the volume layer.

Parameterisation – Up Skin Recoating Hatch

Parameter	Description
General	
Tolerance	 **1** Tolerance **2** Up Skin Defines the minimum size of an area that is recognised as Up Skin. Areas smaller than the defined value are assigned to the volume.

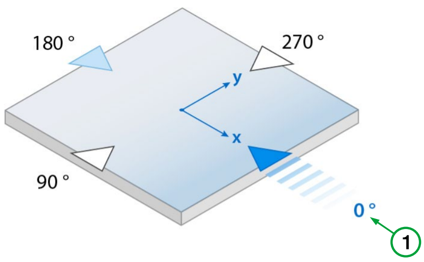
Parameter	Description
Minimum Width	 **1** Inside Out **3** Offset **2** Border Distance **4** Minimal Up Skin Area Enabled Defines a limit value for an area where vectors are not created. This can be used to prevent the exposure of narrow areas.
Border	
Border	If this parameter is activated, a border is defined for this Hatch area (see *Borders*). • **Offset**: Defines the inward indentation of the outermost border. This can be used to compensate for the width of the laser melt pool. **Note** *For Up Skin, the term "Offset" is used instead of "Beam Compensation".*
Hatch	
Hatch types	The following hatch types are available for Up Skin areas: • *Parallel* • *Stripes* • *Chess* • *Lattice* • *None*

<table>
<tr><th>Parameter</th><th>Description</th></tr>
<tr><td>Hatch Rotation</td><td>


1 First Layer
2 Start Angle
3 Limitation Window
4 Angle Increment
5 Gas Flow

The hatch pattern can be rotated from layer to layer. This prevents each layer from being exposed in the same order and at the same angle to the gas flow.

Align with Gas Flow: If this parameter is activated, the hatches are exposed against the direction of the gas flow.
Start Angle: Defines the starting angle between the direction of the gas flow and the orientation of the hatches.
Limitation Window: Defines the limited angular range in which the rotation of the hatch pattern from layer to layer takes place. If the parameter Align with Gas Flow is activated and the hatches are exposed against the gas flow, the rotation of the hatch pattern can only be performed in a range of 180°. The remaining 180° are skipped during the rotation so that an unwanted alignment of the hatches with the direction of the gas flow is avoided. If the parameter Align with Gas Flow is deactivated, the entire range of 360° is available.
Angle Increment: Defines the size of the angle by which the hatch pattern is rotated after each layer.
Gas Flow Direction Angle: Defines the direction of the gas flow.



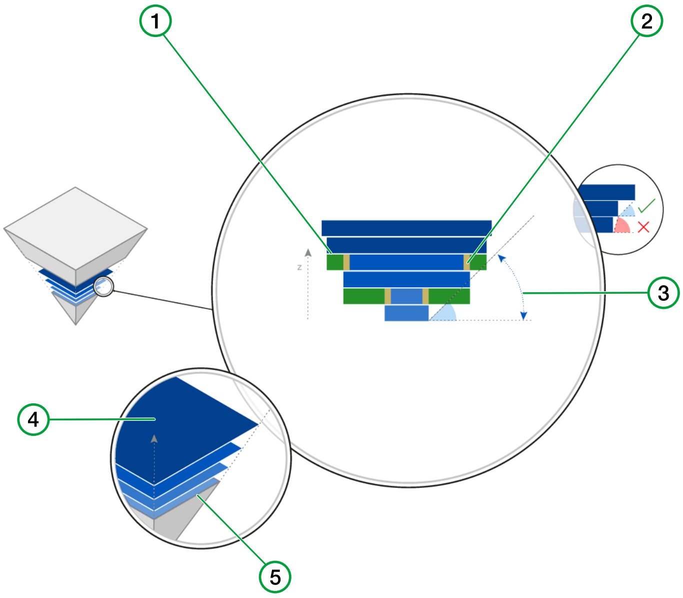
1 Direction of the gas flow
</td></tr>
</table>

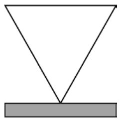
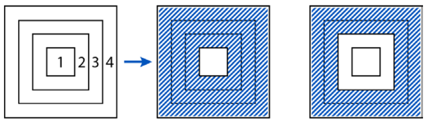
Tab. 47: Build Strategy: Parameters for the Up Skin Remelting Hatch

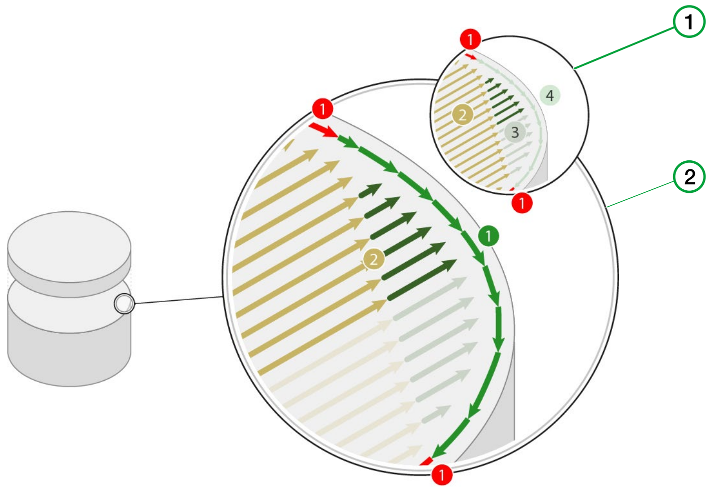
11.2.3.4 Down Skin Hatch

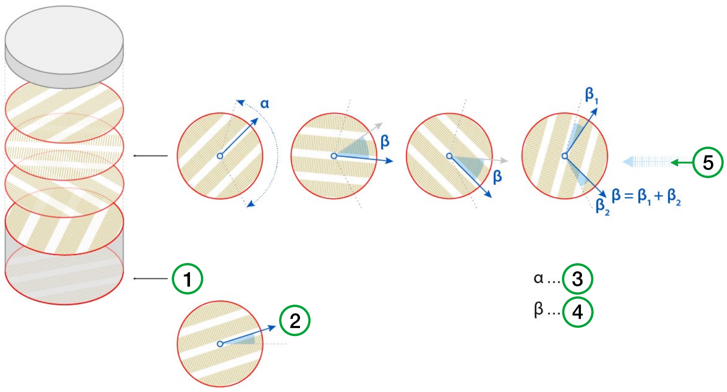
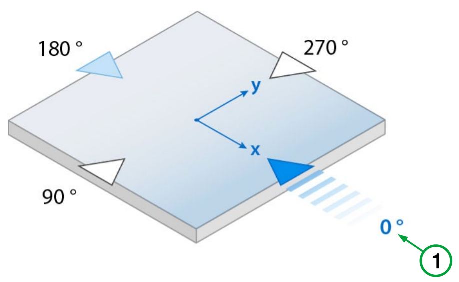
This chapter describes the hatching parameters for the ***Down Skin*** area.

Parameterisation – Down Skin Hatch

Parameter	Description
General	
	 **1** Down Skin **2** Overlap **3** Maximum Surface Angle **4** Layer Reference **5** Current Layer

Parameter	Description
Layer Reference	Defines the number of layers below the current layer that are used to calculate the down skin. For instance, if the value is "3", the horizontal surface of the current layer is compared to the horizontal surface of the layer that is three layers below (closer to the substrate plate). The difference between the two horizontal surfaces is defined as the down skin for the current layer. These are all newly added areas that have not yet been exposed below the current layer. The area that has already been exposed in the third layer and is not included in the difference is defined as Volume in the current layer. View from the side:  View from the top:  (The shaded area indicates the down skin, middle image: Layer reference: 3, right image: Layer reference: 2)
Maximal Surface Angle	Defines the angle from a layer of metal powder to the layer of metal powder above, from which this layer of metal powder above is recognised as a down skin.
Overlap	Defines the width of the overlap area between a down skin and a volume. In the overlapping area, two exposures are made (down skin and volume).

Parameter	Description
Down Skin Strategy V1	If this parameter has been activated, the "Down Skin Strategy V1" function is applied. This allows a support-free construction of overhangs with an inclination of ≥35°. This limiting angle can be reduced to up to 30° for small areas, depending on the area of the overhang. However, the resulting advantage of support geometry reduction causes an extension of the effective exposure time in the part hatch. Predefined build order (cf. graphic): Contour Hatch Down Skin Hatch Down Skin Contour  **1** Down Skin Specific Flag on **2** Down Skin Specific Flag Off
Hatch	
Hatch types	The following hatch types are available for Down Skin areas: • Parallel • Stripes • Chess • Lattice • None

Parameter	Description
Hatch Rotation	 **1** First Layer **2** Start Angle **3** Limitation Window **4** Angle Increment **5** Gas Flow The hatch pattern can be rotated from layer to layer. This prevents each layer from being exposed in the same order and at the same angle to the gas flow. • **Align with Gas Flow**: If this parameter is activated, the hatches are exposed against the direction of the gas flow. • **Start Angle**: Defines the starting angle between the direction of the gas flow and the orientation of the hatches. • **Limitation Window**: Defines the limited angular range in which the rotation of the hatch pattern from layer to layer takes place. If the parameter Align with Gas Flow is activated and the hatches are exposed against the gas flow, the rotation of the hatch pattern can only be performed in a range of 180°. The remaining 180° are skipped during the rotation so that an unwanted alignment of the hatches with the direction of the gas flow is avoided. If the parameter Align with Gas Flow is deactivated, the entire range of 360° is available. • **Angle Increment**: Defines the size of the angle by which the hatch pattern is rotated after each layer. • **Gas Flow Direction Angle**: Defines the direction of the gas flow.  **1** Direction of the gas flow

Tab. 48: *Build Strategy: Parameters for the Down Skin Hatch*

11.2.3.5 Hatch types

The selection of a hatch type and the settings of the specific parameter values can be found in the menu ***Hatching*** under the following menu items:

- Volume
- Up Skin Remelting
- Up Skin Recoating
- Down Skin

Hatches define the arrangement of vectors inside the part. These vectors provide the basis for the paths that the laser processes. This way, the Hatch types define the structure of the interior of the part.

Hatches can be arranged in different patterns.

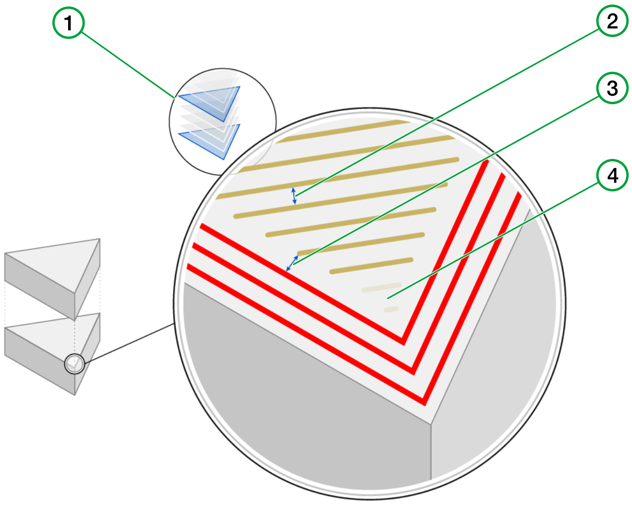
Hatch type	Description
	Parallel: When this Hatch type is selected, vectors are arranged in parallel.
	Stripes: When this Hatch type is selected, vectors are arranged in a striped pattern.
	Chess: When this Hatch type is selected, vectors are arranged in a chess-board pattern.
	Lattice: When this hatch type is selected, vectors are arranged in a grid pat-tern.
	None: If this Hatch type is selected, no vectors are created inside the part. The part is only built according to the parameters **Border** and **Fill Contour** and re-mains hollow inside.

Tab. 49: Hatch types

The parameters of the individual Hatch types are described below.

Parallel

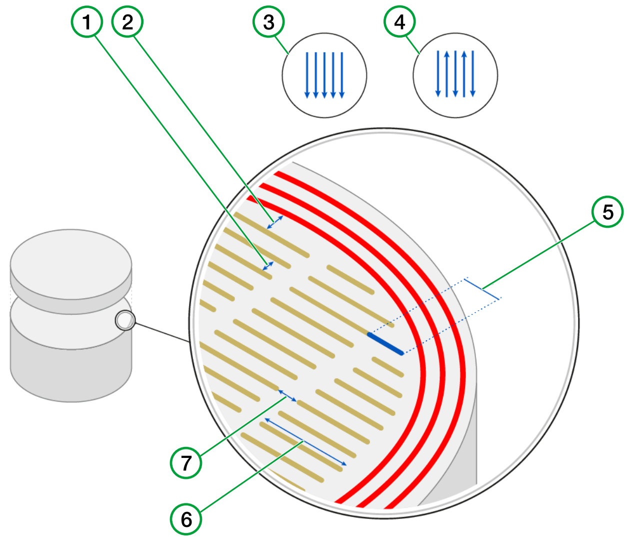
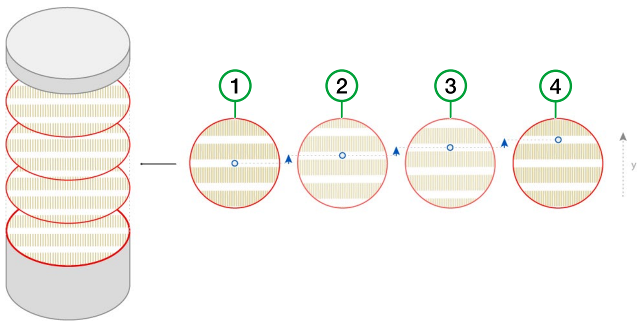
The following describes the settings for the Hatch vector type that is arranged in the Parallel pattern.

Parameter	Description
	 **1** Hatch Every x Layer **2** Hatch Distance **3** Hatch Offset **4** Filter Length
Reduce Jumps	If this parameter is activated, the vectors within the parallel pattern are re-distributed in the exposure sequence. This avoids frequent starting and stopping of the laser.
Alternating	If this parameter is activated, the vectors alternate in their exposure direc-tion.
Filter Length	Defines the minimum length of the Hatches. Hatches shorter than the defined value are not exposed.
Hatch Distance	Defines the distance between the Hatches.
Hatch Offset	Defines the distance between the innermost border and the outermost Hatch.

Tab. 50: Parameters for the Hatch Vector Type "Parallel"

Stripes

The following describes the settings for the Hatch vector type that is arranged in the Stripes pattern.

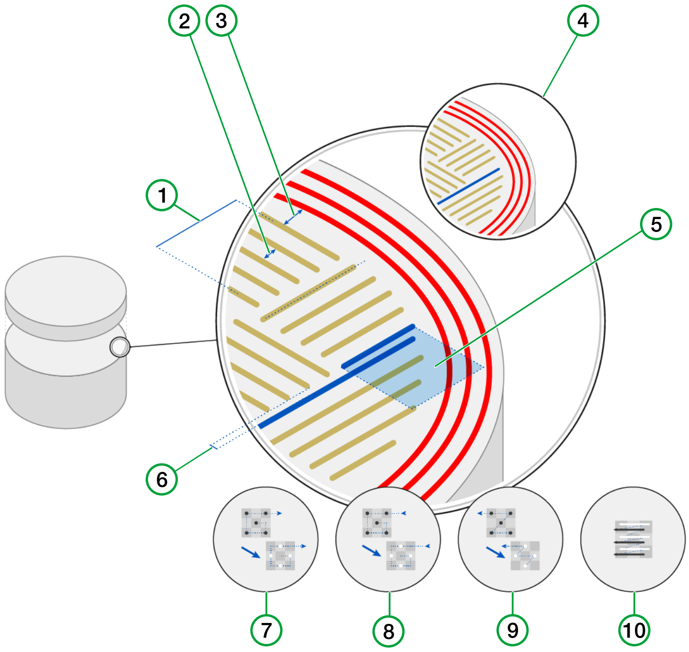
Parameter	Description
	 **1** Hatch Distance **2** Hatch Offset **3** Alternating disabled **4** Alternating enabled **5** Merge Length **6** Stripe Length **7** Stripe Offset
Alternating	If this parameter is activated, the vectors alternate in their exposure direction.
Shift Factor	Defines how often the entire Hatch pattern shifts in the Y direction. A shift takes place after every layer. A value of "3", for example, means that the Hatch pattern has returned to its original position after three layers. The Hatch pattern was therefore shifted twice. After the two shifts, the Hatch pattern is congruent with the original Hatch pattern. After the fourth layer, it is then shifted twice again. Example: Shift Factor = 3:  **1** 1st Layer = Original **2** 2nd Layer = 1st Shift **3** 3rd Layer = 2nd Shift **4** 4th Layer = Original

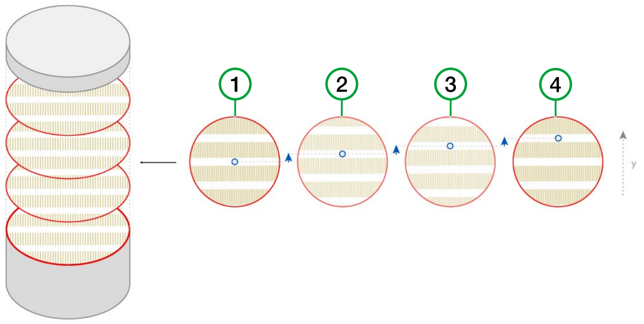
Parameter	Description
Merge	If this parameter is activated, vectors of a defined length are combined. • **Merge Length**: Defines the length from which Hatches are combined. Hatch vectors shorter than the defined value are merged with another Hatch vector.
Filter Length	Defines the minimum length of Hatches. Hatches shorter than the defined value are not exposed.
Stripe Offset	Defines the distance between Hatch patterns.
Stripe Length	Defines the length of the Hatch pattern.
Hatch Distance	Defines the distance between the vectors within the Hatch pattern.
Hatch Offset	Defines the distance between the edge of the Hatch pattern and the contour.

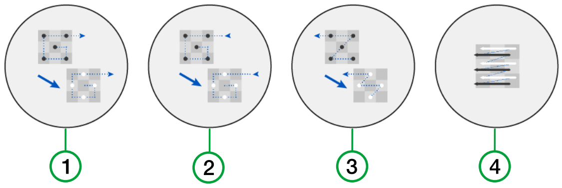
Tab. 51: Parameters for the "Stripes"

Chess

The following describes the settings for the Hatch vector type that is arranged in the Chess pattern.

Parameter	Description
	 **1** Number Of Vectors **2** Hatch Distance **3** Hatch Offset **4** Hatch Distance x Proximity Filter Factor = Vector removed **5** Minimal Field Size 2 (b + l) **6** Field Offset **7** Spiral Out **8** Spiral In **9** Field Based **10** Line Based
Alternating	If this parameter is activated, the vectors alternate in their exposure direction.

Parameter	Description
Shift Factor	Defines how often the entire Hatch pattern shifts in the Y direction. A shift takes place after every layer. A value of "3", for example, means that the Hatch pattern has returned to its original position after three layers. The Hatch pattern was therefore shifted twice. After the two shifts, the Hatch pattern is congruent with the original Hatch pattern. After the fourth layer, it is shifted twice again. Example: Shift Factor = 3:  **1** 1st Layer = Original **2** 2nd Layer = 1st Shift **3** 3rd Layer = 2nd Shift **4** 4th Layer = Original
Merge	If this parameter is activated, vectors of a defined length are combined.
Minimal Field Size	Defines the minimum perimeter of a field within the Hatch pattern. Fields that have a smaller perimeter than the defined one are merged with an adjacent field.
Proximity Filter Factor	Defines the factor that is multiplied by the value under **Hatch Distance**. This value defines up to which distance between two Hatch vectors within merged fields the shorter Hatch vector is removed. This can be used to prevent double exposure of vectors. For example, with a **Hatch Distance** of 0.1 mm and a **Proximity Filter Factor** of "2", all Hatch vectors that are at a distance of 0.2 mm or less from a longer Hatch vector will be removed.
Filter Length	Defines the minimum length of a Hatch. Hatches shorter than the defined value are not exposed.
Field Offset	Defines the distance between the individual fields of the Hatch pattern.
Number of Vectors	Defines the number of Hatch vectors within a single field of the Hatch pattern.
Hatch Distance	Defines the distance between the Hatches.
Hatch Offset	Defines the distance between the innermost border and the outermost Hatch.

<table>
<tr><th>Parameter</th><th>Description</th></tr>
<tr><td>Field Order Strategy</td><td>

1 Spiral Out
2 Spiral In
3 Field Based
4 Line Based
Defines the exposure order for the individual fields of the Hatch pattern.
The Hatch pattern is fictitiously divided into black and white fields and therefore resembles a chessboard.

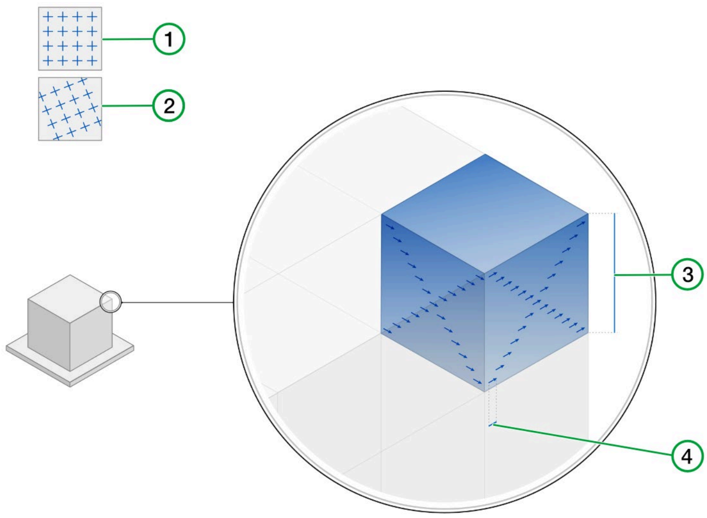
Order Line Based: With this method, the fictitious chessboard is exposed in rows from right to left. First all black squares of the first row are exposed and then all white squares. Only when a line of squares has been completely exposed is the neighbouring line of squares exposed. This process is repeated for all lines of fields.
Order Field Based: With this method, the fictitious chessboard is exposed by colours from right to left. First all black fields of the first row are exposed and then the second and all remaining ones. Only when all black fields have been completely exposed are the white fields exposed. The process for the white squares is then the same as for the black squares, i.e. the first line is first exposed from right to left before the second and all remaining lines of squares are exposed.
Order Spiral In: With this method, the fictitious chessboard is exposed in a spiral from the outside to the inside. First, all black squares are exposed in a spiral from the outside to the inside. Only when all black squares have been exposed are the white squares exposed in a spiral from the outside to the inside.
Order Spiral Out: With this method, the fictitious chessboard is exposed in a spiral from the inside to the outside. First, all black squares are exposed in a spiral from the inside to the outside. Only when all black squares have been exposed are the white squares exposed in a spiral from the inside to the outside.

</td></tr>
</table>

Tab. 52: Parameters for the Hatch vector type "Chess"

Lattice

The following describes the settings for the hatch vector type that is arranged in the lattice pattern.

Parameter	Description
	 **1** Cell Base **3** Cell Size **2** Cell Base Rotation [°] **4** Vector Length
Cell Size	Defines the size of the cells which are generated when hatching.
Vector Length	Defines the length of the vectors.
Cell Base Rotation	Defines the rotation angle of the cell.
Build Order	
SCANLINE	When selected, the vectors are processed in the scanning direction.
STRIPES	When selected, the vectors are processed in strips.
STRIPESSTRICT	When selected, the vectors are processed strictly in strips.

Tab. 53: Parameters for the "lattice" hatch vector type

None

By selecting this Hatch strategy, the corresponding area is not filled by vectors. The part is processed according to the **Border** and **Fill Contour** parameters, but is otherwise hollow.

11.2.3.6 Using the preview function

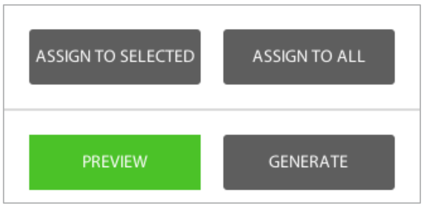
ⓘ The Preview function in the Build Strategy area makes it possible to pre-calculate the individual layers of a part in order to view the arrangement of the vectors in advance.

☑ The **Build Process** action step is selected.

☑ The ***Build Strategy*** tab is selected.

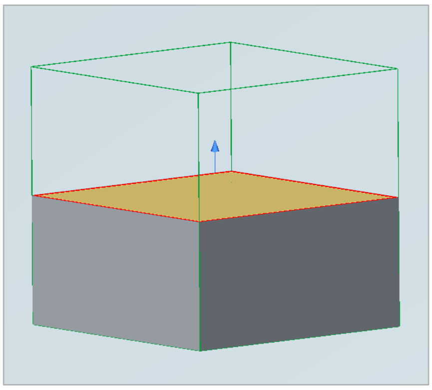
☑ Slicing and Hatching parameters are defined for a part.

1. Click on the part to be displayed in Preview.
2. Click on **[Preview]** in the function panel.



⇨ The part is displayed in the preview view.

3. Click on the arrow in the middle of the part and keep the left mouse button pressed.



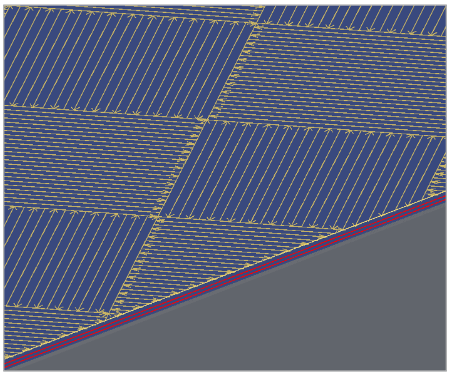
4. Move the arrow in the desired direction.

⇨ The part will be moved through in layers.

➲ The arrangement of the vectors per layer becomes visible.

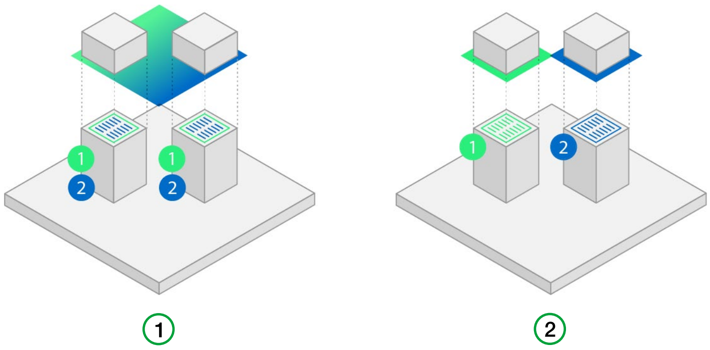
Note

The arrangement of the vectors in the part is only visible from a certain zoom factor.



11.2.3.7 Scanning

In ***Scanning***, the exposure method, the scan sequence and various scan parameters can be defined.

Parameter	Description
Scanning	
Mode	 **1** Platform **2** Part Defines the exposure method. • **Platform**: In this method, the vectors of all parts on the substrate plate are exposed according to the defined Scan Order of the vector types, e.g. first all hatches and then all borders of a part or several part areas. • **Part**: With this method, the exposure of the part areas takes place one after the other and for one layer of metal powder at a time, regardless of the vector types. As soon as the first part is completed for the layer, the second part is exposed in the same layer.
Scan Order	Here is a list of all vector types. Their order in Scanning can be set. Parameters that are not selected or are not available due to the selected machine type or Build Strategy are greyed out in the list. The following parameter settings from the respective menu items are available as a maximum: • **Volume**: Border, Fill Contour, Hatch • **Up Skin Remelting**: Border, Hatch • **Up Skin Recoating**: Border, Hatch • **Down Skin**: Border, Fill Contour, Hatch • **Blocked Path**: Border • **Blocked Path**: Down Skin • **Support**: Border

Parameter	Description
Scan Parameter	Defines settings for the exposure types of the individual vector types. • **Index**: For a machine with several lasers, selects the laser that exposes the selected vector type. • **Power**: Sets the laser power for the selected vector type. • **Speed**: Sets the exposure speed for the selected vector type. • **Focus**: Sets the focus diameter of the laser beam. – A value of "0" corresponds to optimal focusing. – A value of e.g. "1" widens the laser beam and thus reduces the laser power at the incident point. • **Id**: Shows the identification number of the vector type.

Tab. 54: Build Strategy: Scanning parameters

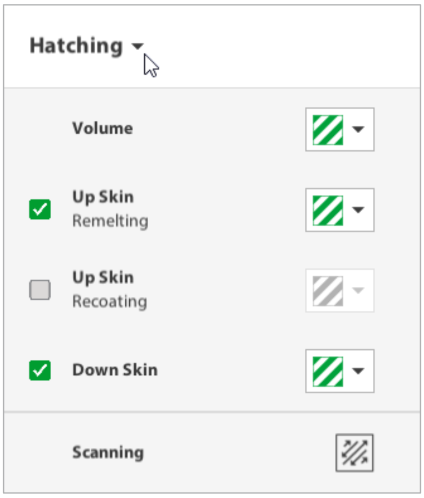
Set exposure sequence

☑ The **Build Process** action step is selected.

☑ The ***Build Strategy*** tab is selected.

☑ The selected machine or Build Strategy should support Hatching in AM-Studio.

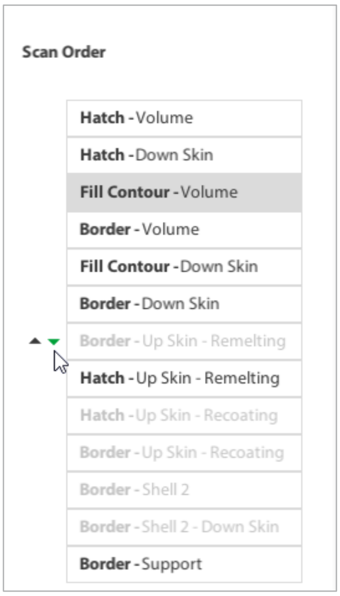
1. Open the menu item ***Hatching*** in the function panel.



2. Click **[Scanning]**.

   ⇨ The menu ***Scanning*** opens.

3. Click on a vector type in the menu area **Scan Order**.
   ⇨ The vector type is highlighted.



4. Use the up and down arrows in the centre of the table to move the desired vector types up or down in the order.

➲ The exposure order is set.

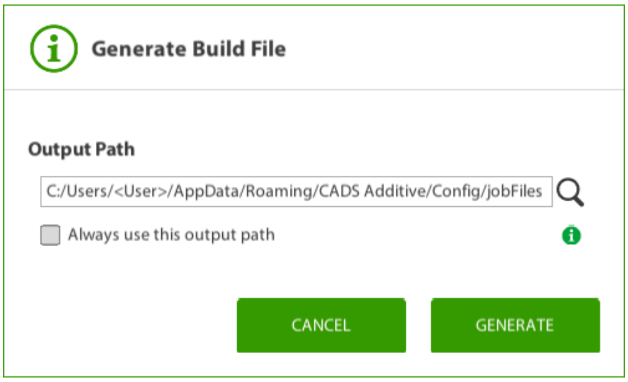
11.2.4 Generating a Build File

☑ The **Build Process** action step is selected.

☑ The ***Build Strategy*** tab is selected.

☑ All parameters are defined.

1. Click on **[GENERATE]** in the function panel.
   ⇨ A new window opens.



2. Navigate to the directory where the build file is to be saved.
3. Assign a file name to the build file.

4. Click on the **[GENERATE]** button in the window.
   ⇨ The build file is generated.

➲ The action step **Viewer** is called up.

11.2.5 Build processors

11.2.5.1 3D MicroPrint GmbH

AM-Studio is compatible with 3D MicroPrint printers.

The build files that can be read by 3D MicroPrint printers are exported via the **Titan**.Core build processor.

In order to use the high resolution of the 3D MicroPrint printers, the part must be resolved in detail by corresponding hatch vectors.

The build files can be exported separately into their own Common Layer Interface (CLI) files depending on the vector type.

11.2.5.2 DMG MORI Additive

Lasertec SLM machines from DMG MORI Additive can be controlled in two ways:

1 **Hatch with Titan.Core**: The AM-Studio software generates the complete contour and hatch information. In this case, no further calculation takes place in the machine. The building process can be started directly. This variant uses the CADS Additive **Titan**.Core which has extended functionalities compared to variant "Hatch on Machine". The slicing and hatching process with CADS Additive **Titan**.Core is significantly more efficient and faster and can be defined using the parameters in the chapters ***Slicing*** or ***Hatching***.

   The full support of this variant is only given with a CELOS operating system of version 1.9 or higher.

2 **Hatch on Machine**: The AM-Studio software generates layer data with contour information. The necessary offset contours and hatches are generated directly in the machine in preparation for the building process. The main part of the parameters for defining the contours and hatches is stored in the machine. This variant therefore only allows a subset of the possible parameters from chapter ***Slicing***. This corresponds to the classic data preparation using the DMG MORI Additive Build Processor on the machine.

Selection of the Build Processor

In the ***Build Strategy*** tab, the drop-down menu of the function panel can be used to select either variant 1 (Hatch with Titan.Core) or 2 (Hatch on Machine).

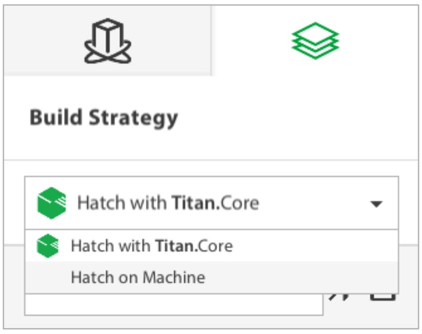


Fig. 21: Selection of the control variant

11.2.5.2.1 DMG CLI exchange format

Data preparation according to variant 1 and 2 is transmitted to the machine in the same way.

1. Data preparation in AM-Studio
2. Slicing (or Hatching) of the prepared data in the corresponding layer thickness
3. Export of the layer data into the DMG MORI Additive CLI format
4. Exit the AM-Studio environment
5. Starting the CELOS environment on the Lasertec SLM machine
6. Import of the exported AM-Studio data in DMG MORI Additive CLI format
7. Start of the print job

Note

DMG MORI Additive printers with CELOS v1.7.x are not able to read binary CLI data. The option "ASCII" in the AM Studio prompt must be selected.
As of v1.8.x, CELOS is also able to read binary CLI data. This allows smaller amounts of data and should be used preferentially.
Starting with CELOS v1.8.2 it is possible to use Titan.Core. For machines with older CELOS software versions, please refer to the notes above.

11.2.5.3 EOS GmbH

AM-Studio supports the machines of the manufacturer EOS GmbH.

Prepared data can be transferred to an EOS machine in the following ways:

- Directly using the EOSprint API interface of EOS GmbH
- Via a neutral exchange format (CLI), if no EOSprint licence is available on the user's machine

11.2.5.3.1 EOSprint API

When an EOS machine profile is selected or created, the system checks whether a corresponding EOSprint licence is available on the user's computer.

If AM-Studio finds such a licence, it communicates directly with the available EOS material database and the connected EOS machinery via the EOSprint API interface.

AM-Studio fully integrates these two parts and allows the data preparation to be provided directly as a task in the EOS machine. It is not necessary to switch to the EOSprint application.

Note

If no EOSprint licence is available, the EOS CLI exchange format can be used to transfer the prepared data to an EOS GmbH machine.

11.2.5.3.2 EOS CLI exchange format

If there is no EOSprint licence on the user's machine, the prepared data can be transferred to an EOS GmbH machine as follows:

1. Data preparation in AM-Studio
2. Slicing of the prepared data in the corresponding layer thickness
3. Export of the layer data into the EOS CLI format
4. Exit the AM-Studio environment
5. Starting the EOSprint application of EOS GmbH
6. Import of the AM-Studio data into EOSprint
7. Definition of the material parameters in EOSprint
8. Creating a task in EOSprint and starting the printing process

11.2.5.4 SLM Solutions Group AG

SLM Solutions Group AG machines can be controlled directly and fully with AM-Studio.

The Titan.Core and Basic Core internal processors are available as build processors. They can be selected from a drop-down menu under ***Build Strategy*** in the function panel.

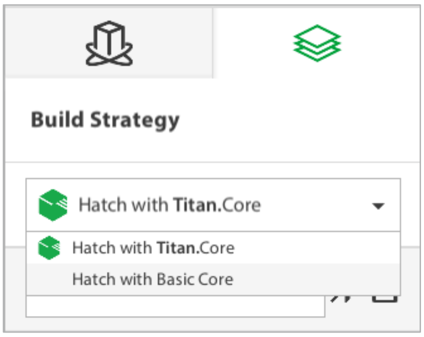


Fig. 22: Selection of the control variant

Details on the individual parameters can be found under *Slicing* and *Hatching*.

11.2.5.4.1 SLM Solutions file format

For selected SLM Solutions Group AG machines, the prepared data can be automatically extracted from AM-Studio as an *.SLM-Build file (see *Generating a Build File*), which can then either be sent to the AM machine via a data transfer or be exported in a neutral data format (CLI).

11.2.5.5 Trumpf GmbH

The scope of functions of AM-Studio includes data preparation for machines of the manufacturer Trumpf GmbH.

Prepared data can be transferred to a Trumpf machine in the following ways:

- Directly using the TruTops Print API interface of Trumpf GmbH.
- Via a neutral exchange format (CLI) if no TruTops Print license is available on the user's computer

11.2.5.5.1 TruTops Print API

If a Trumpf machine profile is created in the AM-Studio software and the TruTops Print API is used for this, the option **License** must be activated in the machine settings under ***Build Processor*** and the corresponding material and strategy must be selected.

AM-Studio then checks whether a corresponding licence is available on the user's machine.

If AM-Studio finds a licence, it communicates directly with the Trumpf API via the TruTops Print 0PI interface.

In order to use parameters and strategies, these must be created in TruTops Print.

Data preparation takes place directly in AM-Studio. To generate the hatches, the parts and supports are transferred to the Trumpf API. The Trumpf API generates a WZA file that is automatically read in by AM-Studio and displayed in the **Viewer**.

Note

> *If no TruTops Print licence is available, the CLI exchange format can be used to transfer the prepared data to a Trumpf GmbH machine.*

11.2.5.5.2 TruTops Print API integration

If a corresponding licence is available, a material and an associated Build Strategy can be assigned in AM-Studio.

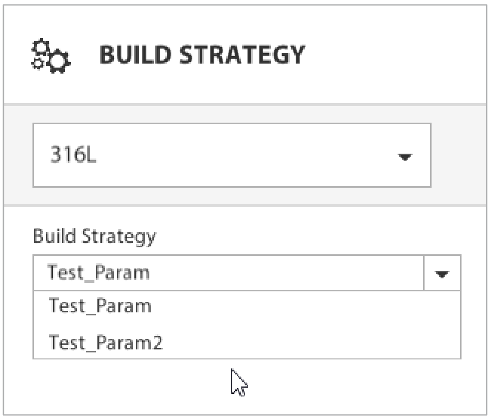


Fig. 23: Assigning material and build strategy

The layers calculated by the TruTops Print API can be viewed via the layer preview or after creating the hatches in the **Viewer**.

11.2.5.6 Eplus3D

AM-Studio supports machine manufactured by Eplus3D Tech Co.

The so-called epi files are exported via the CADS Additive **Titan**.Core build processor.

12 STEP 6: VIEWER

This chapter describes the **Viewer** action step.

Fig. 24: Progress bar: Viewer action step

The defined build strategy can be checked.

In addition, the costs and time required for the build process can be determined.

12.1 Management of the layer data

In the **Viewer** action step, the layer data generated by the Slicing/Hatching algorithm can be checked.

For this purpose, the generated laser paths are loaded and visualised.

The range of functions depends on the selected machine type.

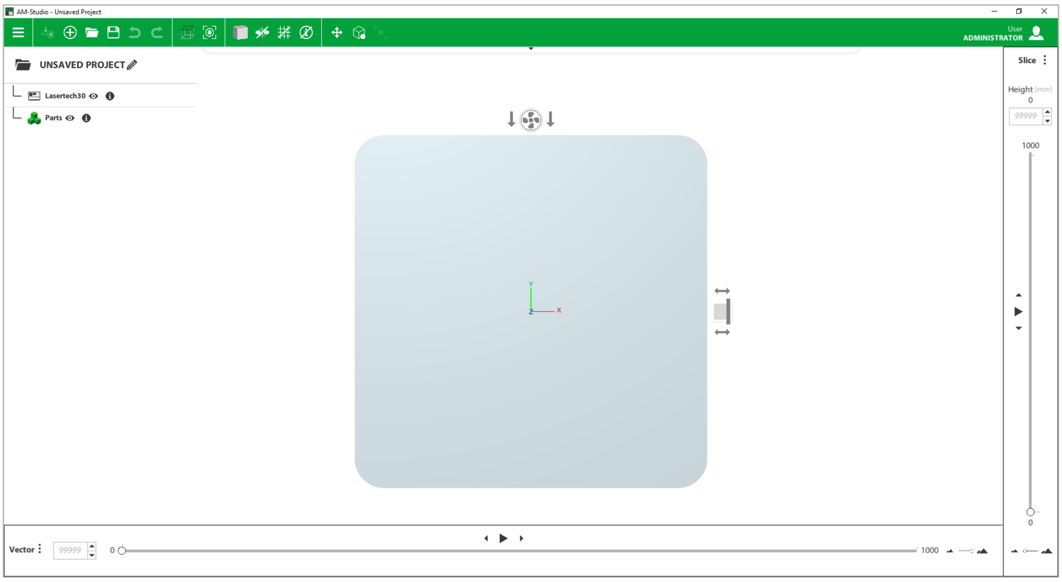


Fig. 25: Visualisation of the layer data in the Viewer

Loading of layer data when passing through the action steps

When all action steps up to the **Build Process** have been run through, a build file is generated with a click on the **[GENERATE]** button of the function panel. After the layer data has been calculated, AM-Studio switches to the action step **Viewer** and the layer data is displayed.

Loading layer data manually

☑ The **Viewer** action step is selected.

1. Click on the substrate plate with the right mouse button.
   ⇨ A context menu opens.

2. Click on **[Load Build File]** in the context menu.
   ⇨ An Explorer window opens.
3. Select the desired layer data file.

➲ The layer data is loaded and the first layer is displayed.

12.2 Evaluating the laser paths

The sliders at the bottom and right side of the window in the **Viewer** action step can be used to navigate through the individual layers and vectors of a layer data file.

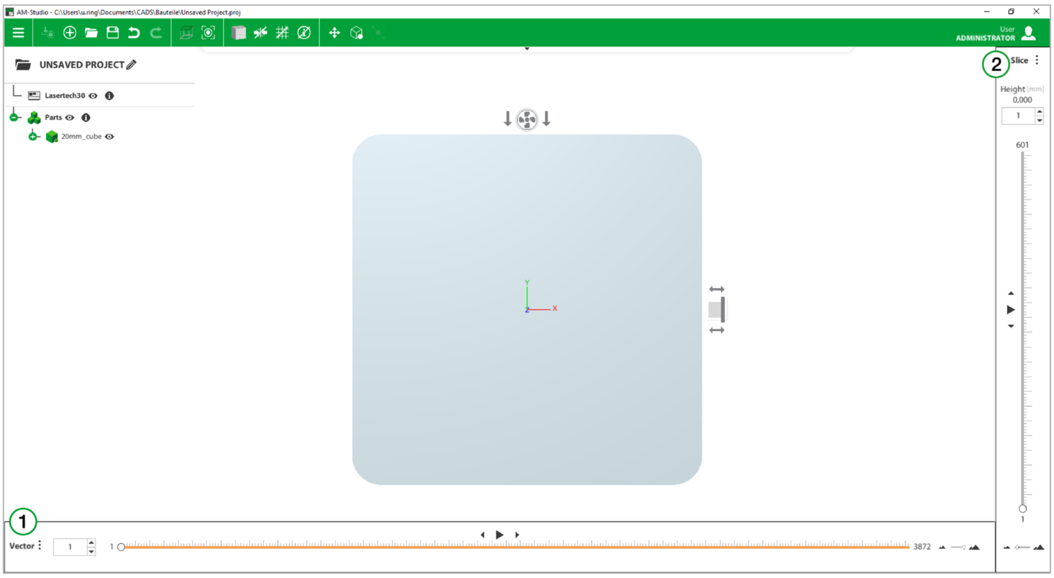


Fig. 26: Sliders for navigation

1 Display of the vectors and their processing order

2 Selection of a specific layer of the layer data file

Note

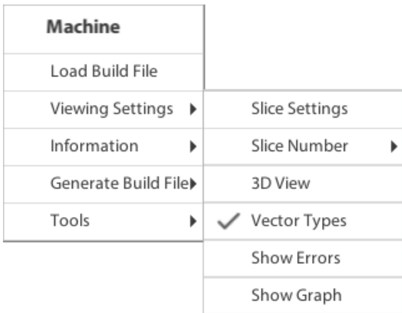
Both sliders can be animated by clicking on [Play] ▶.

Show/hide vector types

☑ The **Viewer** action step is selected.

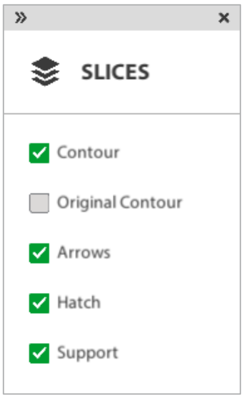
1. Click on the substrate plate with the right mouse button.

   ⇨ A context menu opens.



2. Click on **[Viewing Settings] [Slice settings]** in the context menu.

   ⇨ A window opens.



3. Check each vector type to show it or uncheck ☑ it to hide vector types.

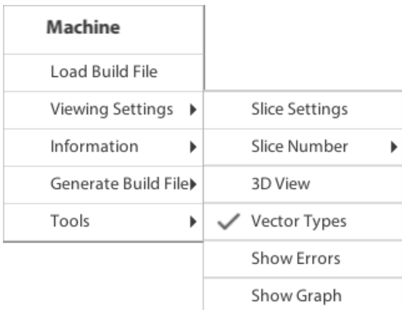
➲ The vector types are shown or hidden.

Coloured representation of the different vector types in the lower slider.

☑ The **Viewer** action step is selected.

1. Click on the substrate plate with the right mouse button.

   ⇨ A context menu opens.



2. Click on **[Viewing Settings] [Vector types]** in the context menu to activate the coloured display of the vector types.

   ⇨ The function is activated and marked with ✓.

➲ The lower slider shows the colours of the vector types.

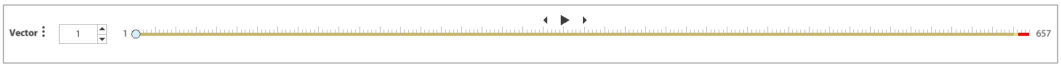


Fig. 27: Coloured representation of the vector types in the lower slider (in the example: orange, yellow, red).

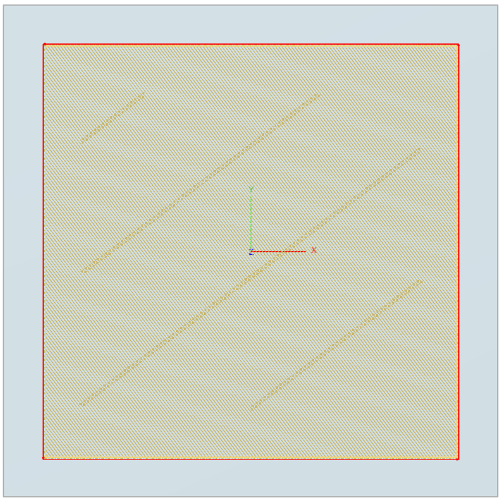


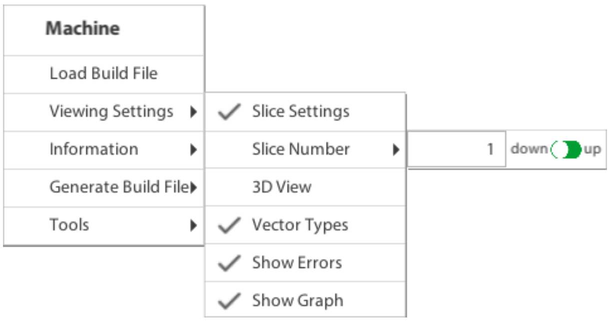
Fig. 28: Coloured representation of the vector types on the part (in the example: orange, yellow, red)

Display of vector information of several layers in a 3D view

☑ The **Viewer** action step is selected.

1. Click on the substrate plate with the right mouse button.

   ⇨ A context menu opens.



2. Click on **[Viewing Settings] [Slice number]** in the context menu.
3. Enter the number of layers to be displayed in the input field.
4. Use the toggle switch to select whether the displayed layers should be above (up) or below (down) the currently selected layer.

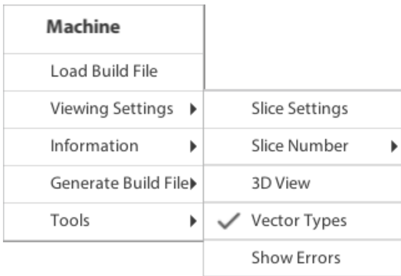
➲ The main window changes to a 3D view. The layer data of the selected layers are displayed interactively.

Showing a 3D view

☑ The **Viewer** action step is selected.

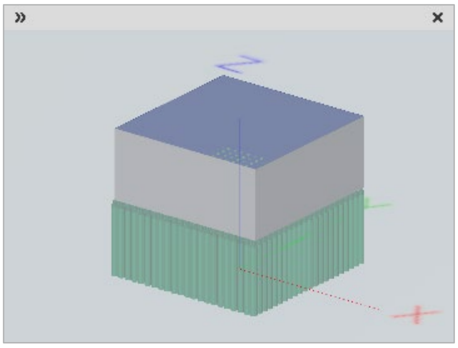
1. Click on the substrate plate with the right mouse button.

   ⇨ A context menu opens.



2. Click on **[Viewing Settings] [3d view]** in the context menu.

➲ A window opens showing the substrate plate with the parts. A layer schematically represents the current layer in the 3D model.



Note

If a layer data file was loaded without an associated AM-Studio project, no parts are displayed on the substrate plate.

12.3 Advanced functions of a layer data file

In addition to the vectors of a specific layer, the viewer can display further information.

Which information can be displayed in detail depends on the machine manufacturer.

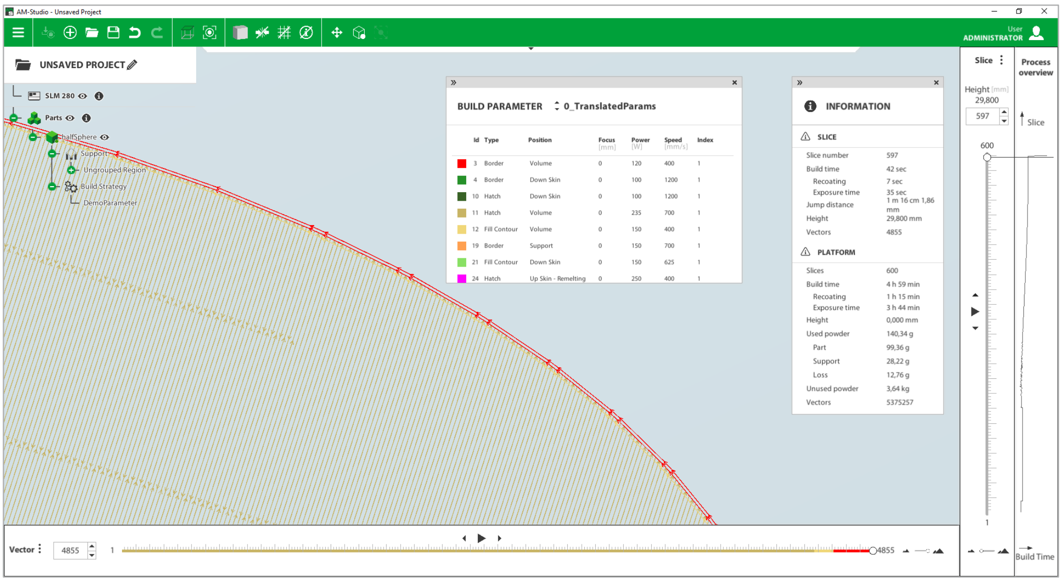


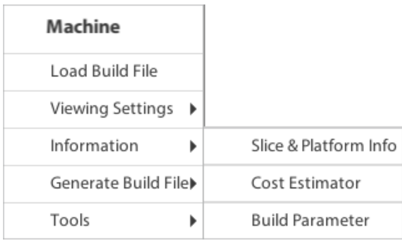
Fig. 29: Extended information in the Viewer

Display of laser parameters

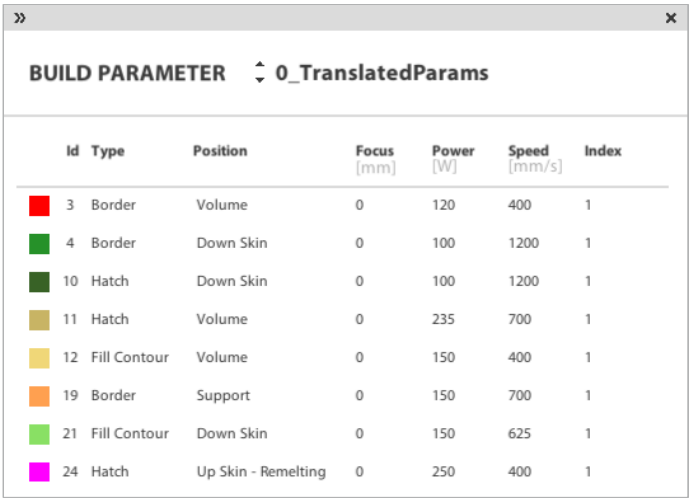
☑ The **Viewer** action step is selected.

1. Click on the substrate plate with the right mouse button.

   ⇨ A context menu opens.



2. Click on **[Information] [Build Parameter]** in the context menu.
   ⇨ A window opens.



BUILD PARAMETER ⁝ 0_TranslatedParams

	Id	Type	Position	Focus [mm]	Power [W]	Speed [mm/s]	Index
	3	Border	Volume	0	120	400	1
	4	Border	Down Skin	0	100	1200	1
	10	Hatch	Down Skin	0	100	1200	1
	11	Hatch	Volume	0	235	700	1
	12	Fill Contour	Volume	0	150	400	1
	19	Border	Support	0	150	700	1
	21	Fill Contour	Down Skin	0	150	625	1
	24	Hatch	Up Skin - Remelting	0	250	400	1

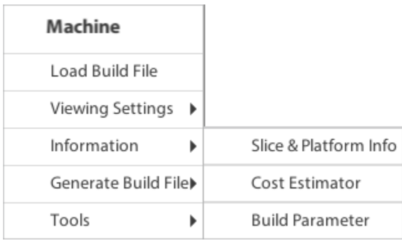
➲ Extended information on the individual scan parameters of the different Hatch types is displayed.

Note | *This information is not available for all machine types.*

Displaying extended layer information

☑ The **Viewer** action step is selected.

1. Click on the substrate plate with the right mouse button.
   ⇨ A context menu opens.



2. Click on **[Information] [Slice & platform info]** in the context menu.
   ⇨ A window opens.

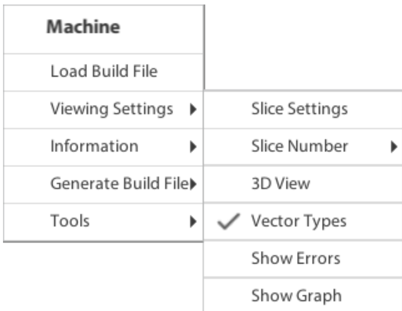
➲ Advanced information for the current layer and for the entire layer data file is displayed, e.g. the build time or the number of scan vectors.

Note | *This information is not available for all machine types.*

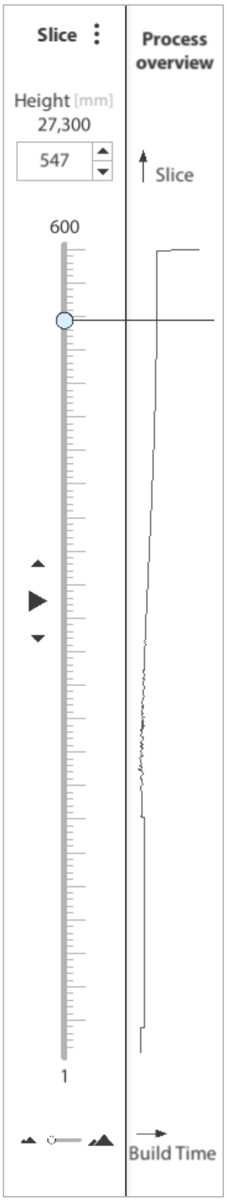
Viewing the build time for each layer

☑ The **Viewer** action step is selected.

1. Click on the substrate plate with the right mouse button.
   ⇨ A context menu opens.



2. Press the **[Viewing Settings] [Show graph]** button.
   ⇨ The graph ***Process overview*** is displayed next to the right slider.



➲ The build time for each layer is shown.

Note

This information is not available for all machine types.

The graph also shows how much energy is put into each layer.
If the build times for all layers are almost the same, a continuous and homogeneous energy input into the powder bed can be assumed.
If the build times for the different layers differ greatly, this leads to different cooling times for the individual layers. This can lead to thermal problems and failure of the build process.

12.4 Cost calculation

AM-Studio enables the calculation of the expected build time of a project.

The build time is estimated by simulating the build process on a virtual machine. This virtual machine uses the same algorithm as a real machine.

The build time in combination with the machine hourly rate and the material costs enables the calculation of the project costs.

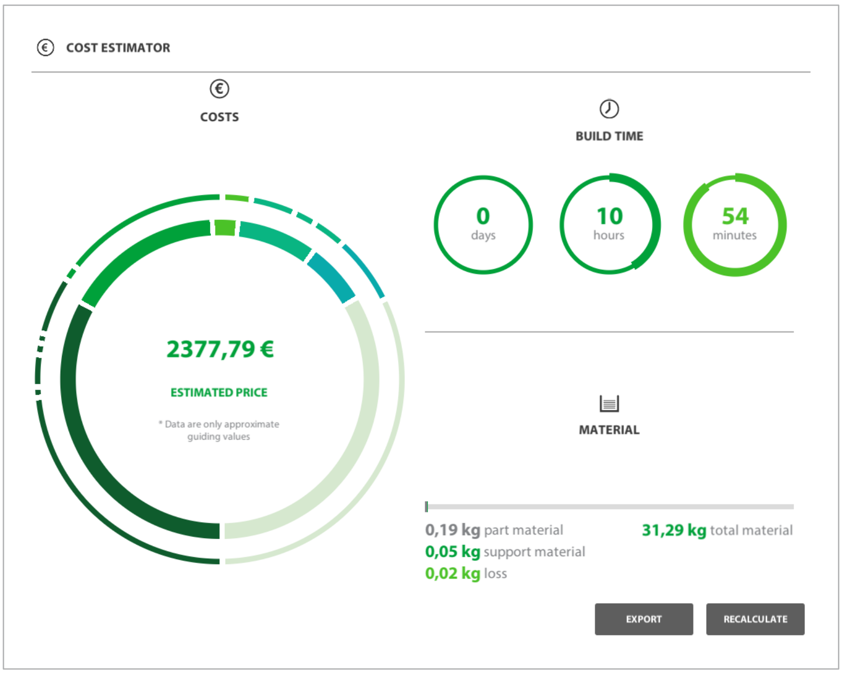


Fig. 30: Cost calculation in AM-Studio

Note

The estimation of the construction time is only available for the machines of the manufacturers EOS GmbH and SLM Solutions Group AG.
The cost calculation for other machines is only based on the material costs.
The costs shown refer to the production of the entire project.
It is not possible to calculate the costs for a single part.

Machine hourly rate

If the machine hourly rate is known, it can be stored directly in the settings of the machine. For this purpose, click on the tab ***Machines*** in the menu ***Settings*** and open the editing window by clicking on the pencil symbol ✎. Click on the ***Costs Parameters*** tab in the menu and enter the machine hourly rate under **Cost Parameters**.

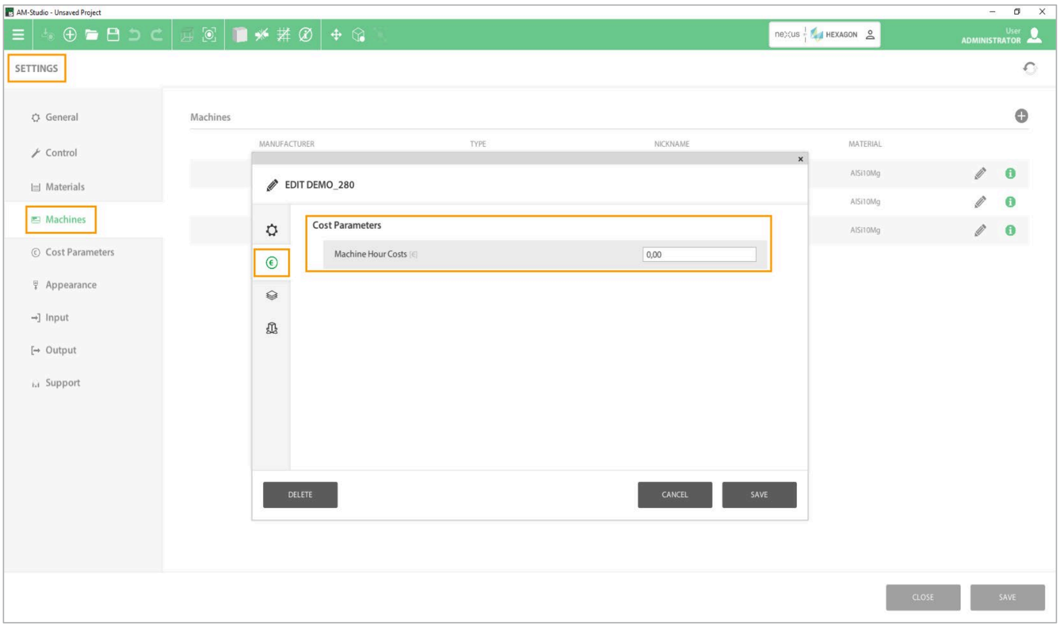
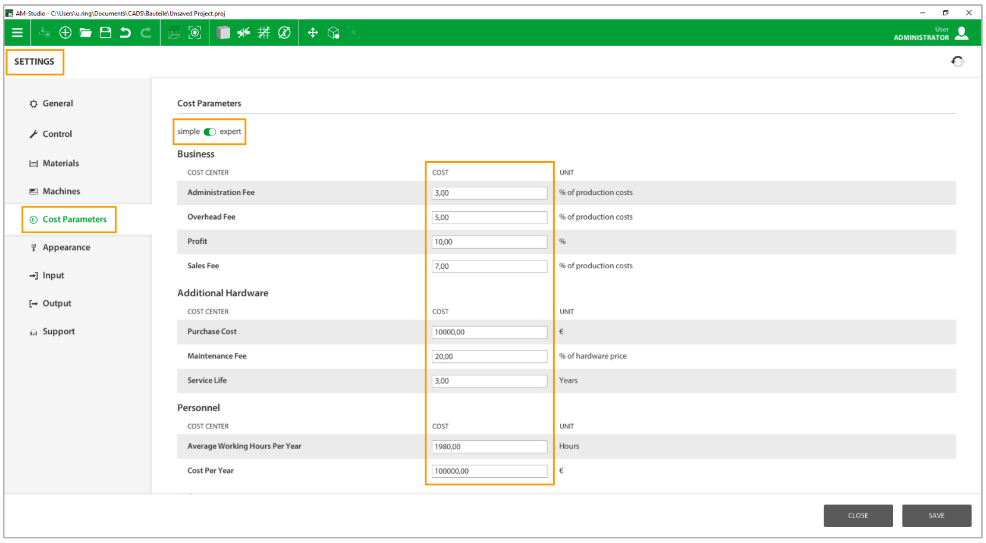


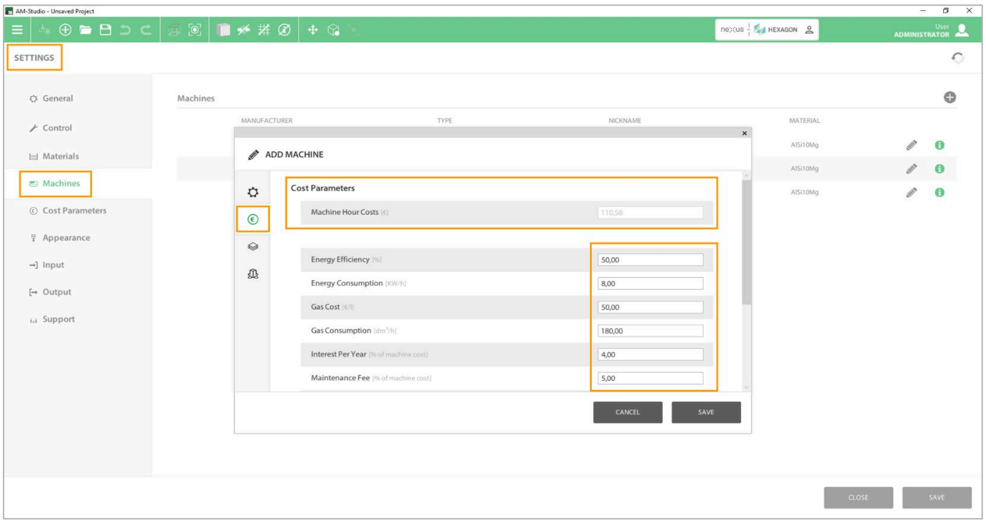
Fig. 31: Setting the machine hourly rate

If the machine hourly rate is unknown, it can be calculated in AM-Studio. To do this, click on the ***Cost Parameters*** tab in the ***Settings*** menu.

1. In the settings, select the expert mode for cost calculation.
2. Enter the cost parameters.



3. Switch to the cost area of the machine ***(Settings*** > ***Machines*** > ***Cost Parameters***) and enter the cost parameters there as well.



➲ AM-Studio Calculates the resulting machine hourly rate for the specific machine from the sum of these cost parameters.

Note

A complete list of all cost parameters can be found in the chapter "Settings":

1. *machine-related cost parameters (see Cost Parameters (Cost Parameters)).*
2. *general cost parameters (see Cost Parameters)*

13 SETTING

13.1 General

General program settings are defined in ***General***. The individual menu items are explained below.

Preferred Output (Display Monitor)

Under ***Preferred Output***, the monitor/screen can be set on which AM-Studio is displayed by default. The number of the monitor corresponds to the setting in the operating system.

Fig. 32: Selection of the display monitor

Directories (File paths)

Under ***Directories***, the file paths of the following programme parts are displayed:

- Support profiles (see *Step 3: Support*)
- Layers and layer data or layer information
- Material database for the manufacturer Eos GmbH with values for available materials

These file paths are created by AM-Studio during the programme installation.

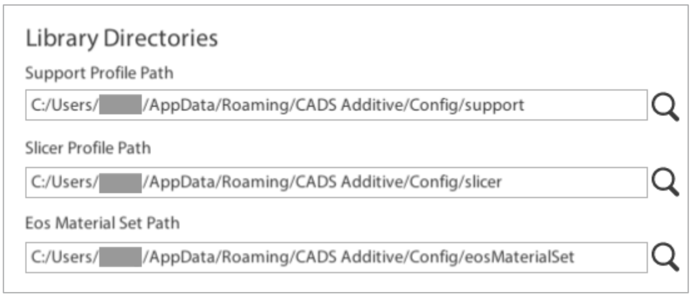


Fig. 33: File path management

Functional restrictions **(User roles)**

Under ***Functional restrictions***, the role of a user is defined. By defining a user role, the rights of certain groups of persons in AM-Studio are determined according to their area of responsibility. The selection of the respective user role is determined via the drop-down menu "**Role**" .

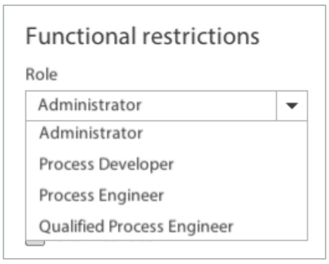


Fig. 34: Defining user roles

In AM-Studio, user profiles are defined with the following rights:

Rights	User profiles			
	Administrator	Process Developer	Process Engineer	Qualified Process Engineer
Displaying cost parameters	✓			✓
Editing cost parameters	✓			
Displaying build time calculation parameters	✓	✓		✓
Editing build time calculation parameters	✓	✓		
Editing machine types	✓	✓		
Deleting machine types	✓	✓		
Editing support parameters	✓	✓	✓	
Editing build strategy parameters	✓	✓	✓	

Tab. 55: User rights in AM-Studio

When the programme is started for the first time, the user profile "Administrator" is active.

Switching between user profiles can be protected with a password. If a password is set up, it is only possible to switch to another profile after entering the password correctly.

Licensed Features

The programme licences activated in AM-Studio are displayed under ***Licensed Features***.

The "Base Product" licence is activated by default when a licence for AM-Studio has been purchased. This base licence cannot be deactivated.

If a licence for the "CAD Interface" has also been purchased, this option can be activated with the check mark ✅. This CAD licence allows native files of different CAD programs to be imported into AM-Studio.

Fig. 35: Licence function

LIcense Server

The host name and port are displayed under ***LIcense Server*** . The display is for information purposes only. No settings can be made by the user.

Fig. 36: Licence server information

Auxiliary (high system requirements)

Under ***Auxiliary***, additional functions of AM-Studio can be activated. These additional functions are very CPU-intensive and can impair the performance of AM-Studio depending on the system used.

- If the option "Activate Part Collision Module" is activated, parts that are outside the substrate plate or that collide with other parts are displayed in red.
- If the option "Render Arrows in Slice Viewer" is activated, all paths that the laser processes when building up the layers are displayed in the preview of the part generation under **Build Strategy**.

Fig. 37: Additional functions (high system requirements)

In addition, the number of threads made available to the Build Processor by the system can be set manually.

13.2 Control

Using the menu item ***Control***, the mouse and keyboard behaviour can be set for navigation in the program and for program operation.

Mouse navigation

Preset profiles for mouse control can be selected via a drop-down menu under "**Mouse Control**".

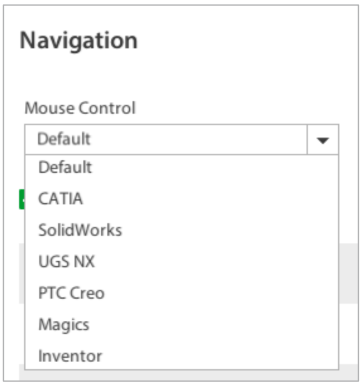


Fig. 38: Mouse navigation control profile

These presets take into account the mouse control of various CAD programs and can make it easier for users of these CAD programs to work with AM-Studio.

Control profiles from the following manufacturers are currently supported:

- Default (AM-Studio)
- CATIA
- SolidWorks
- UGS NX
- PTC Creo
- Magics
- Inventor

For each profile, the assignment of the mouse buttons or the respective combination of mouse button and button/key combination can be taken from a table.

Default (AM-Studio)

Selection	Left Mouse key
Rotation	Right Mouse key
Zoom	Mouse wheel
Translation	Middle Mouse key

CATIA		
Selection		Left Mouse key
Rotation		Middle Mouse key + Left Mouse key
Zoom		Middle Mouse key + Left Mouse key - Left Mouse key
Translation		Middle Mouse key
Solidworks		
Selection		Left Mouse key
Rotation		Middle Mouse key
Zoom		Middle Mouse key + Shift or Mouse wheel
Translation		Middle Mouse key + Ctrl
UGS NX		
Selection		Left Mouse key
Rotation		Middle Mouse key
Zoom		Middle Mouse key + Left Mouse key
Translation		Middle Mouse key + Right Mouse key
PTC Creo		
Selection		Left Mouse key
Rotation		Middle Mouse key
Zoom		Middle Mouse key + Ctrl
Translation		Middle Mouse key + Shift
Magics		
Selection		Left Mouse key
Rotation		Right Mouse key
Zoom		Right Mouse key + Ctrl Mouse wheel
Translation		Right Mouse key + Shift Middle Mouse key
Inventor		
Selection		Left Mouse key
Rotation		Middle Mouse key + Shift
Zoom		Mouse wheel
Translation		Middle Mouse key

Tab. 56: Mouse profiles in AM-Studio

The behaviour of the zoom function can be inverted with the option "Invert Zoom Direction". This function is activated by default in AM-Studio.

Fig. 39: Invert Zoom Direction

The following functions are controlled with the mouse:

Function	Description
Selection	Selection of a part on the substrate plate
Rotation	Rotation of the substrate plate in the working area
Zoom	Zoom in or out
Translation	Moving of the substrate panel in the working area

Tab. 57: Explanation of the mouse control

Keyboard Shortcuts

Under ***Keyboard Shortcuts***, predefined keyboard assignments can be selected or own key assignments can be defined. For each profile, the keyboard assignment or the respective key combination can be taken from a table.

A drop-down menu under "**Key Control**" allows you to choose between the predefined keyboard profiles "Default" and "Magics" or to define your own keyboard profile with "Userdefined".

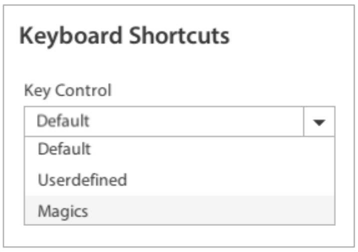


Fig. 40: Selection of keyboard profiles

The following functions can be controlled via a key or a key combination:

Function	Description
Focus	Centres and enlarges the view of the selected part
Move	Activates the translation gizmo to move the part.
Rotate	Activates the rotation gizmo to rotate the part.
View Cutting Plane	Activates/Deactivates the "View Cut" function
Activate Alignment Tool	Activates/Deactivates the align tool"
Activate Measure Tool	Activates/Deactivates the measuring tool

Function	Description
Toggle Part Origin/Center	Switches the rotation point of the part to the part origin or centre.
Toggle Local/Global - Translation or Rotation	Switches between local and global coordinate system
Manual Rod	Places a single Rod Support at a selected part region
Snap to Grid	Activates/deactivates a grid square on the substrate plate. When moving, a part snaps into the respective positions of the grid.
Undo	Undoes the last action
Redo	Repeats the last action
Save	Saves the project
Copy	Copies a part
3d view	Pastes a copied part
Duplicate	Copies a part and pastes it on the substrate plate
Select all Parts	Selects all elements on the substrate plate
Load Part	Opens an Explorer window to load parts
Open Project	Opens an Explorer window to open a project
Export Project	Opens the Export Options dialogue box
Export Log Files	Opens an Explorer window to specify a location for log files
Workflow Steps	Jumps to the respective action steps

Tab. 58: Key control

For the keyboard profiles "Default" and "Magics", the functions listed are permanently assigned to a key/key combination. For the keyboard profile "Userdefined", the following functions can be assigned to a key/key combination:

- Focus
- Move
- Rotate
- View Cutting Plane
- Activate Alignment Tool
- Activate Measure Tool
- Toggle Part Origin/Center
- Toggle Local/Global - Translation or Rotation
- Manual Rod
- Snap to Grid

Function	Keyboard profile "Default"	Keyboard profile "Userdefined"	Keyboard profile "Magics"
FUNCTION	KEYS	KEYS	KEYS
Focus	F	F	F
Move	W	W	T
Rotate	E	E	R
View Cutting Plane	C	C	C
Activate Alignment Tool	A	A	A
Activate Measure Tool	M	M	M
Toggle Part Origin / Center	Z	Z	Z
Toggle Local / Global - Translation or Rotation	X	X	X
Manual Rod	P	P	P
Snap To Grid	S	S	S
Undo	Ctrl + Z	Ctrl + Z	Ctrl + Z
Redo	Ctrl + Y	Ctrl + Y	Ctrl + Y
Save	Ctrl + S	Ctrl + S	Ctrl + S
Copy	Ctrl + C	Ctrl + C	Ctrl + C
Paste	Ctrl + V	Ctrl + V	Ctrl + V
Duplicate	Ctrl + D	Ctrl + D	Ctrl + D
Select All Parts	Ctrl + A	Ctrl + A	Ctrl + A
Load Part	Ctrl + I	Ctrl + I	Ctrl + I
Open Project	Ctrl + O	Ctrl + O	Ctrl + O
Export Project	Ctrl + E	Ctrl + E	Ctrl + E
Export Log Files	Ctrl + Shift + Alt + E	Ctrl + Shift + Alt + E	Ctrl + Shift + Alt + E
Workflow Steps	F1 - F6	F1 - F6	F1 - F6

Tab. 59: Overview Keyboard Assignments "Default", "Userdefined", "Magics"

13.3 Materials

The properties of the building materials used can be defined in this submenu.

You can enter the following properties:

- **Name**: Name of the material
- **Density**: Density of the material
- **Powder Bulk Density**: Bulk density of the powder component
- **Material Loss**: Degree of material loss
- **Price**: Price of the material in Euro per kilogram
- **Overhang Angle**: Slope angle of the material
- **Filter Surface** surface area
- **Filter Length**: Filter length

Material

	NAME	DENSITY [kg/m³]	POWDER BULK DENSITY [% of Density]	MATERIAL LOSS [%]	PRICE [€/kg]	OVERHANG ANGLE [°]	FILTER SURFACE [mm²]	FILTER LENGTH [mm]
☐	AlSi10Mg	2680,00	60,03	10,00	200,00	40,00	2,00	0,00
☐	316L	7950,00	55,00	10,00	100,00	43,00	2,00	0,00
☐	Ti6Al4V	4450,00	60,00	10,00	300,00	41,00	2,00	0,00
☐	17-4PH	7770,00	60,00	10,00	400,00	45,00	2,00	0,00
☐	CoCr	8300,00	60,00	10,00	500,00	45,00	2,00	0,00
☐	CuCr1Zr	8920,00	60,00	10,00	600,00	45,00	2,00	0,00
☐	IN-625	8440,00	60,00	10,00	700,00	45,00	2,00	0,00
☐	IN-718	8150,00	60,00	10,00	800,00	45,00	2,00	0,00

Fig. 41: Settings for materials

A new material can be added by clicking on the plus symbol ⊕ and the corresponding values can be entered.

☐	Name	Density	Density of unused	Material Loss	Price	Overhang Angle	Filter Region	Filter Length

Fig. 42: Input field for new material values

To delete one or more materials from the list, the respective line must be selected by placing a check mark ☑ to the left of the line.

Material

	NAME	DENSITY [kg/m³]	POWDER BULK DENSITY [% of Density]	MATERIAL LOSS [%]	PRICE [€/kg]	OVERHANG ANGLE [°]	FILTER SURFACE [mm²]	FILTER LENGTH [mm]
☑	AlSi10Mg	2680,00	60,03	10,00	200,00	40,00	2,00	0,00
☐	316L	7950,00	55,00	10,00	100,00	43,00	2,00	0,00

Fig. 43: Remove material

If you click on the waste bin symbol, a warning appears before the material is finally deleted. Clicking on **[CANCEL]** ends the process, clicking on **[Delete]** deletes the material from AM-Studio.

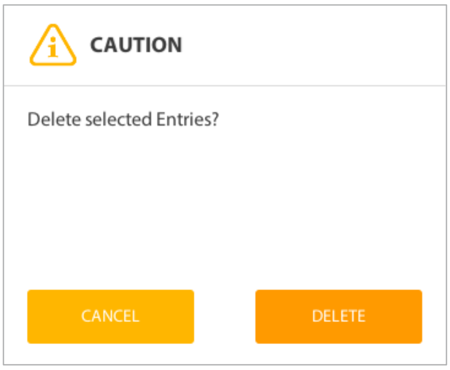


Fig. 44: Confirming or rejecting a deletion process

Clicking the pencil icon ✎ on the right allows you to change the material values in the individual input fields.

Fig. 45: Changing material values

13.4 Machines

You can make settings for AM machines in the ***Machines*** sub-menu. AM machines that have already been created are listed in a table.

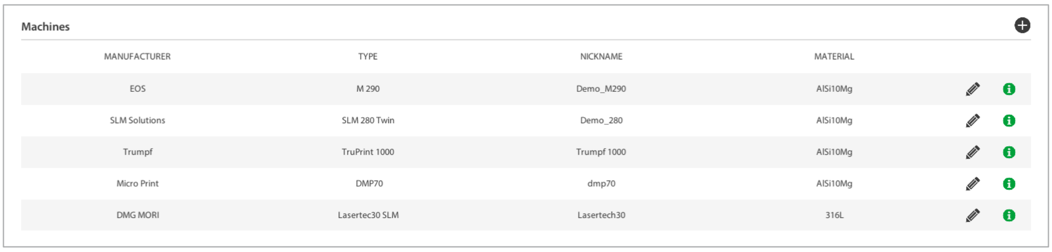


Fig. 46: AM machines

For each AM machine, the following data is displayed in the table:

- MANUFACTURER: Manufacturer of an AM machine
- TYPE: Type designation of the AM machine
- NICKNAME: freely selectable designation of an AM machine
- MATERIAL: material used by the respective machine

Creating a new AM machine

A new AM machine can be created by clicking on the plus symbol ⊕. After clicking on the plus symbol, the ***ADD MACHINE*** window opens. In this window, tje general settings for the machine (General), settings for the cost parameters of a machine (Cost Parameters) and settings for the Build Processor used can be defined.

General Settings (General > General Settings)

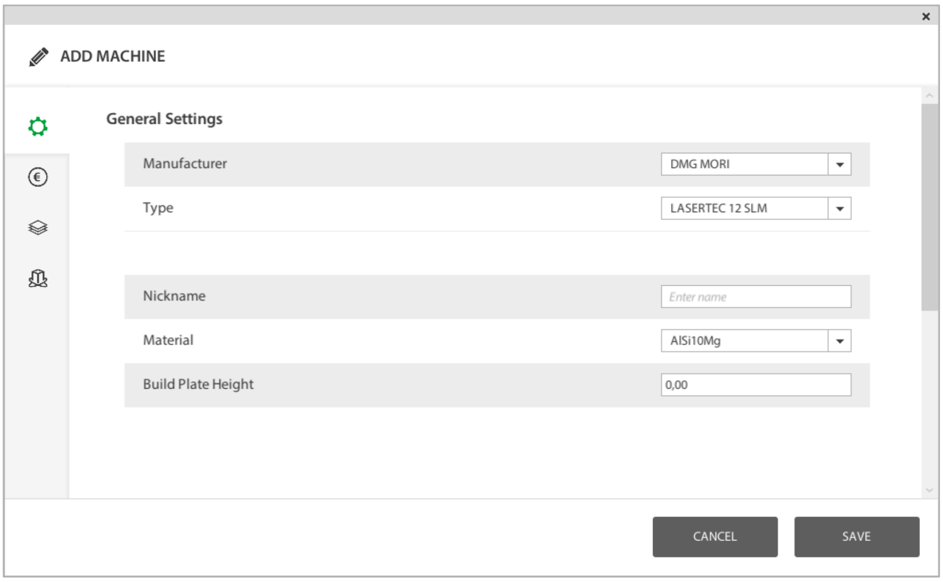


Fig. 47: General settings

The ***General*** menu allows the following settings:

Setting	Description
Manufacturer	Selection of a manufacturer via a drop-down menu
Type	Selection of a machine type via a drop-down menu
Nickname	Assignment of a freely selectable designation for a machine
Material	Selection of the material used via a drop-down menu
Build Plate Height	Defining the substrate plate height
Laser Count	Allows you to define the number of lasers in use. This parameter is only available for machine types which are compatible with this function.
AM-LOCK	Activating "AM-LOCK"

Tab. 60: General settings

Note

The option AM-LOCK can only be activated for certain machine types. The option is only visible in the menu if it is available.

If the option "Custom" is selected under **Manufacturer** , a machine can be completely redefined. The following parameters can be set**:**

Setting	Description
Type	"Add new Template" creates a new machine profile
Type Name	Assignment of a freely selectable designation for a machine
Plate Size	Defining the dimensions of the substrate plate for X-, Y- and Z-axis

Setting	Description
Point Zero	Defining the zero point on the substrate plate via a drop-down menu • Center: Zero point in the middle of the substrate plate • Front Left: Zero point at the front left of the substrate plate • Front Right: Zero point at the front right of the substrate plate • Back Left: Zero point at the rear right of the substrate plate • Back Right: Zero point at the rear right of the substrate plate
Recoater Position	Setting the rest position of the coater via a drop-down menu • Left: Rest position of the coater left • Right: Rest position of the coater right • Front: Rest position of the coater front • Back: Rest position of the coater back • Back Right: Rest position of the coater rear right
Gas Flow Position	Determining the direction of the gas inflow via a drop-down menu • Left: Gas flowing in from the left • Right: Gas flowing in from the right • Front: Gas flowing in from the front • Back: Gas flowing in from the back • Back Right: Gas flowing in from the back right
Build Plate Shape	Defining the shape of the substrate plate via selection criteria • Rectangle: Representation of the substrate plate as a rectangle • Circle: Display of the substrate plate as a circle
Laser Count	Entering the number of lasers in the machine
Material Type	Specifying the material type via a drop-down menu • Metal: Material type metal • Plastic: Material type plastic
Nickname	Assignment of a freely selectable designation for a machine
Material	Selection of the building material used via a drop-down menu according to the building materials created in Materials
Build Plate Height	Entering the substrate plate height
AM-LOCK	Activate "AM-LOCK"

Tab. 61: Custom Machine settings

Cost Parameters (Cost Parameters)

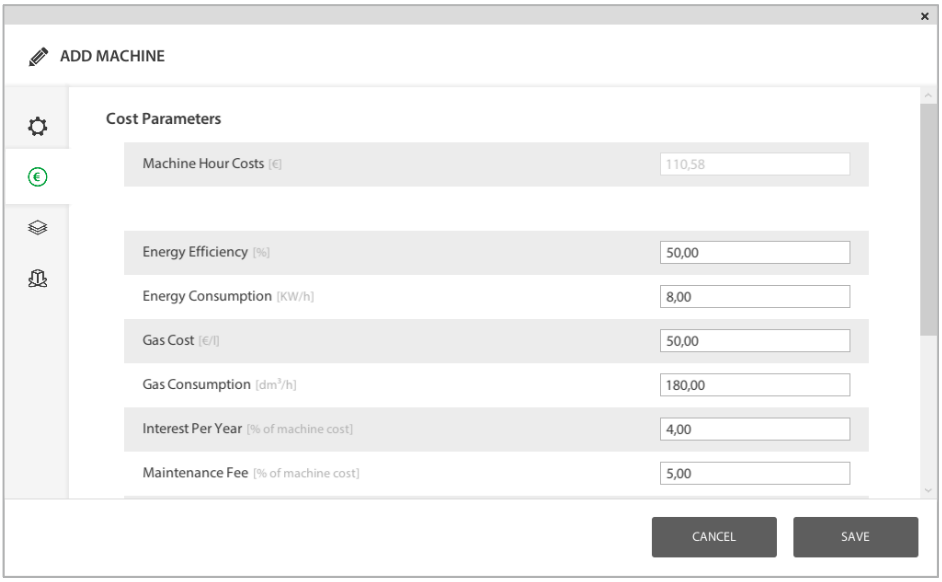


Fig. 48: Machine cost settings

Note

The advanced settings for the cost parameters are only displayed if "expert" is selected in the Settings > Cost Parameters menu item.

simple expert

If "simple" is selected, only the ***Machine Hour Cost*** *parameter is displayed.*

In the ***Cost Parameters*** menu, you can enter the following values:

Settings	Description
Machine Hour Costs	Machine costs as hourly rates/costs per machine hour
Energy Efficiency	Energy efficiency
Energy Consumption	Energy consumption
Gas Cost	Gas costs/petrol costs
Gas Consumption	Gas consumption
Interest Per Year	Interest per year
Maintenance Fee	Maintenance costs
Other Fee	Other charges
Purchase Cost	Acquisition costs
Unexpected Machine Costs	Unexpected machine costs
Room Cost	Space costs
Service Life	Running time/operating life

Settings	Description
Space Requirement	Space requirements

Tab. 62: Menu item Cost parameters

Build Processor

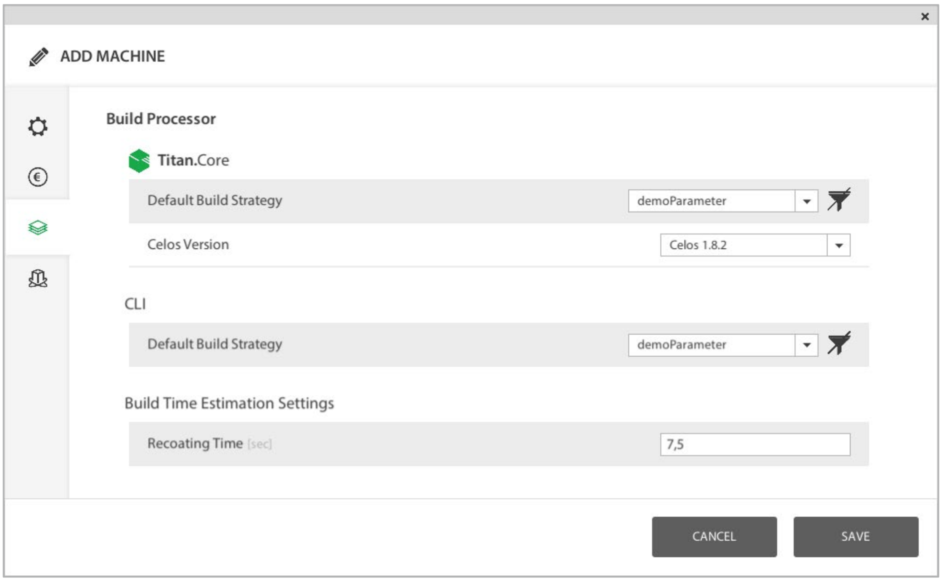


Fig. 5: Menu item Build Processor

The setting options under the menu item ***Build Processor*** differ depending on the selected manufacturer.

DMG MORI

Menu item	Setting	Description
Titan.Core	**Default Build Strategy**	Selection of a build strategy via a drop-down menu
	Celos Version	Selection of a Celos version via a drop-down menu
CLI	**Default Build Strategy**	Selection of a build strategy via a drop-down menu
Build Time Estimation Settings	**Recoating Time**	Input of coating time

Tab. 63: Settings Build Processor: DMG MORI machines

Eplus3D

Menu item	Setting	Description
Titan.Core	**Default Build Strategy**	Selection of a build strategy via a drop-down menu
Build Time Estimation Settings	**Recoating Time**	Input of coating time

Tab. 64: Settings Build Processor: Eplus3D machines

EOS

Menu item	Setting	Description
EOS	**License**	Activation of an existing licence with a check mark ✅
	Hostname/IP address	Entering the host or IP address
	Material Set	Selection of the building material via a drop-down menu
	Default Part Parameters	Selection of the part parameters
	Default Support Parameters	Selection of the support parameters
CLI	**Default Build Strategy**	Selection of a build strategy via a drop-down menu
Build Time Estimation Settings	**Recoating Time**	Input of coating time

Tab. 65: Build Processor: settings EOS machines

Micro Print

Menu item	Setting	Description
3D MICRO PRINT	**Default Build Strategy**	Selection of a build strategy via a drop-down menu
Build Time Estimation Settings	**Recoating Time**	Input of coating time

Tab. 66: Settings Build Processor: Micro Print machines

SLM Solutions

Menu item	Setting	Description
Titan.Core	**Default Build Strategy**	Selection of a build strategy via a drop-down menu
Build Time Estimation Settings	**Jump Delay**	Input of Jump Delay
	Jump Speed	Input of Jump Speed
	Mark Delay	Input of Mark Delay
	Polygon Delay	Input of Polygon Delay
	Polygon Edge Level	Input of Polygon Edge Level
	Recoating Time	Input of coating time
	Minimum Scan Time	Input of the minimum scan time
	Minimum Exposure Time Start Layer	Input of the minimum exposure time of the start layer
	nPrev	Input of nPrev
	nPost	Input of nPost

Tab. 67: Settings Build Processor: SLM machines

Trumpf

Menu item	Setting	Description
Trumpf	**License**	Activation of an existing licence with a check mark ☑
	Material Set	Selection of the building material via a drop-down menu
	Default Strategy	Selection of a build strategy via a drop-down menu
	Build File Version	Selection of a build file version via a drop-down menu
CLI	**Default Build Strategy**	Selection of a build strategy via a drop-down menu
Build Time Estimation Settings	**Recoating Time**	Input of coating time

Tab. 68: Build Processor: settings Trumpf machines

Simulation Calibration

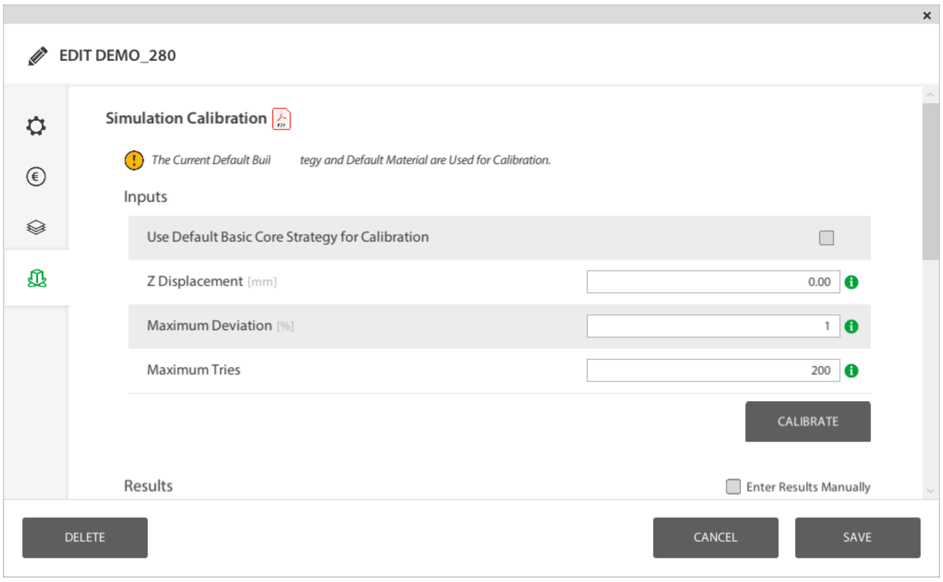


Fig. 49: Simulation Calibration menu item

ⓘ The simulation calibration calibrates the AM machine based on the pre-set building material, the pre-set Default Build Strategy and the deformation values measured on the Cantilever. Based on this data, AM-Studio runs through several simulations until it finds a simulation falls within the specified tolerances for the measured deformation value. The resulting strain values for the X, Y and Z dimensions are stored for the machines used.

Note

The calibration values for the set material and Default Build Strategy are stored.

The parameters for the calibration of the inherent strain values are defined in the ***Inputs*** sub-menu:

Setting	Description
<Checkbox>	The checkbox is only displayed if the machine offers an alternative Build Strategy. The text displayed with the checkbox also depends on the manufacturer. If this option has been activated, the alternative Build Strategy is used for the calibration.
Z Displacement	Defines the value of the cantilever deformation.
Maximum Deviation	Defines the maximum permissible deviation in percent. If the simulation value in one of the iteration steps of the calibration is less than the given **Z Displacement** value, the simulation is successfully completed. The inherent strain values are then determined and displayed under **Inherent Strain X/Y** and **Inherent Strain Z**.
Maximum Tries	Defines the maximum permissible number of iteration steps. If this number is reached, the simulation is successfully completed. The inherent strain values are then determined and displayed under **Inherent Strain X/Y** and **Inherent Strain Z**.

Tab. 69: Simulation Calibration - Inputs settings

The inherent strain values can be determined directly in the ***Results*** sub-menu:

Setting	Description
Enter Results Manually	If this option has been activated, the **Inherent Strain X/Y** and **Inherent Strain Z** values can be modified manually.
Inherent Strain X/Y	Defines the inherent strain in X/Y direction. The value of this field can only be modified if the **Enter Results Manually** option has been activated.
Inherent Strain Z	Defines the inherent strain in Z direction. The value of this field can only be modified if the **Enter Results Manually** option has been activated.

Tab. 70: Simulation Calibration - Results settings

Edit and delete AM machine

An AM machine that has been created can be edited by clicking on the pencil symbol behind the respective machine. An editing window opens in which the settings made when the machine was created can be edited.

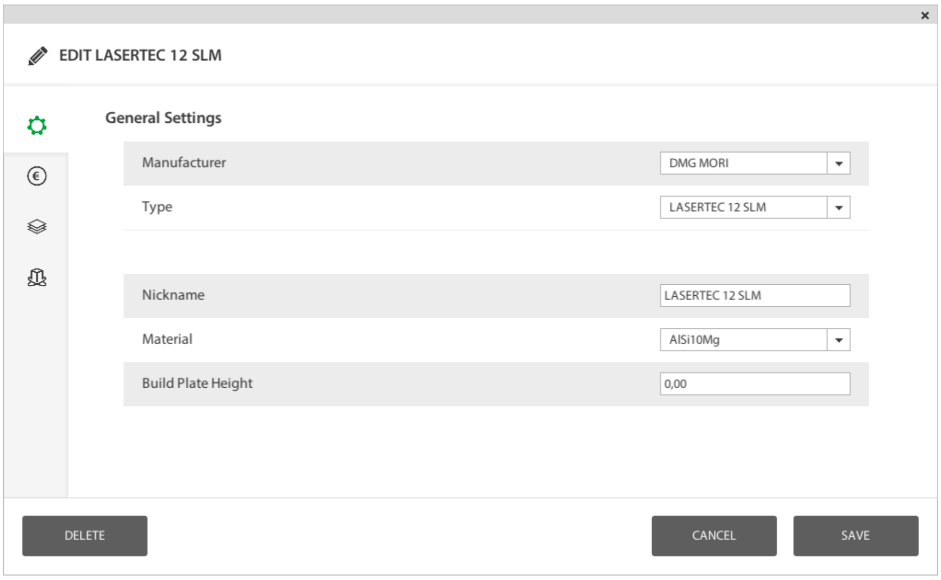


Fig. 6: Editing window for editing the machine settings

The selected AM machine can be deleted by clicking on the **[Delete]** button in the editing window. A warning appears before the machine is finally deleted.

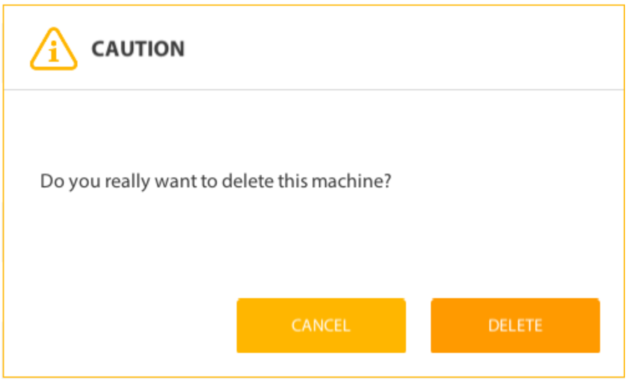


Fig. 50: Warning when deleting an AM machine

Clicking on **[CANCEL]** ends the process, clicking on **[Delete]** deletes the machine.

Viewing machine information

Clicking on the information symbol behind a machine opens the "***MACHINE INFORMATION***" window. Here you can view technical information about the selected machine. Click on **[Close]** to close the window again.

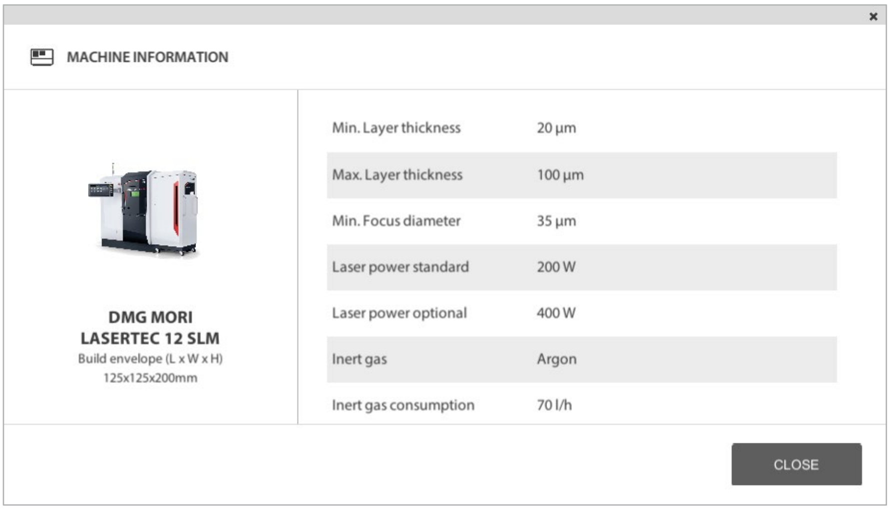


Fig. 10: Machine information

13.5 Cost Parameters

Under ***Cost Parameters***, costs can be defined for different criteria. When defining the cost parameters, "simple" and "expert" are available as two possible view options. If the option "simple" is selected, only the business criterion **Profit** is displayed, all other criteria are hidden. If the option "expert" is selected, all cost-relevant criteria can be viewed and edited.

The cost centres are divided into the following areas:

- ***Business***T - Business: business-related costs
- ***Additional Hardware***: Costs due to additional hardware
- ***Personnel***: Personnel costs
- ***Software***: Software costs
- ***Usage***: Usage costs

Cost centre	Description	Unit
Business		
Administration Fee	Administrative costs incurred	% of production costs
Overhead Fee	Overheads incurred	% of production costs
Profit	Profit margin	%
Sales Fee	Sales charges incurred	% of production costs
Additional Hardware		
Purchase Cost	Acquisition costs	€
Maintenance Fee	Maintenance costs	% of hardware costs
Service Life	Useful life	Years

Cost centre	Description	Unit
Personnel		
Average Working Hours Per Year	Average working time per year	Hours
Cost Per Year	Costs per year	€
Software		
Purchase Cost	Acquisition costs	€
Maintenance Fee	Customer support costs	€
Service Life	Useful life	Years
Usage		
Efficiency	Efficiency of the machine	%
Daily Use Of Equipment In Hours	Daily use of the machine	Hours
Average Working Days Per Year	Average number of days the machine is in use	Days
Personnel For Machine Related Personnel Expenditure	Working time for machine-related personnel expenses	% (100% = 1 full-time equivalent)
Personnel For Post-Processing Related Personnel Expenditure	Working time for rework-related personnel expenses	% (100% = 1 full-time equivalent)

Tab. 71: Cost parameters in the "expert" representation

13.6 Appearance

Appearance can be used to adjust the appearance of the user interface.

- Via the drop-down menu "**Buildplate Style**" you can choose between a 3D and a 2D representation of the substrate plate.
- The drop-down menu "**Camera Projection**" can be used to select between an orthographic and a perspective camera perspective.
- The colour scheme used in the Orientation Maps (see Basics), "Rainbow" or " Blue", can be selected via the drop-down menu "**Color Chart Orientation Map**".

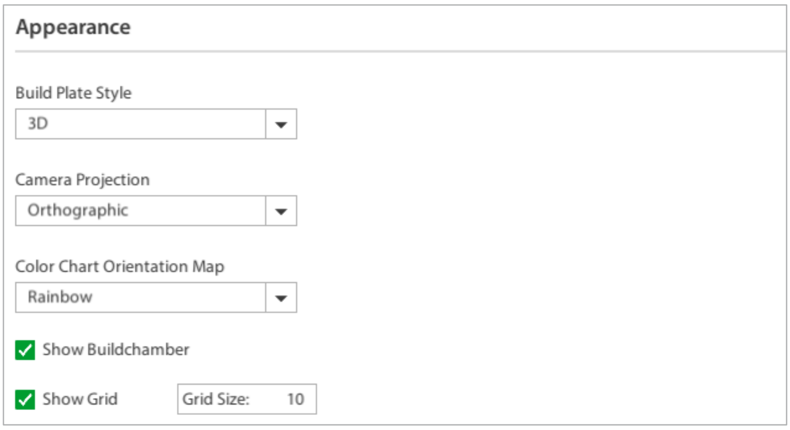


Fig. 51: *Settings in the "Appearance" menu*

- By setting a check mark at "**Show Buildchamber**" the view of the build chamber restriction can be activated.
- By setting a check mark at "**Show Grid**" the view of a square grid on the substrate plate can be activated. If "**Show Grid**" is activated, the size of the grid squares in millimetres can be specified in the "**Grid Size**" input field.

The settings under *Color* enable

- the selection of a cross-programme display mode
- colour settings for substrate plate, parts and support

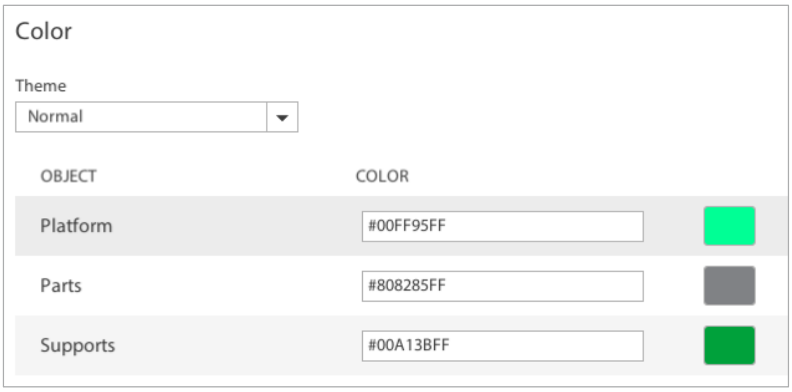


Fig. 52: *Colour settings*

The drop-down menu **"Theme"** can be used to select how the user interface is displayed:

- "Normal" for the display of AM-Studio in a light design.
- "Dark" for the display of AM-Studio in a dark design.

You can define your own colours for the substrate plate, parts and supports. The colours can be defined by entering hexadecimal values or with the ***Color Picker***. The ***Color Picker*** opens with a click on the colour fields of the objects.

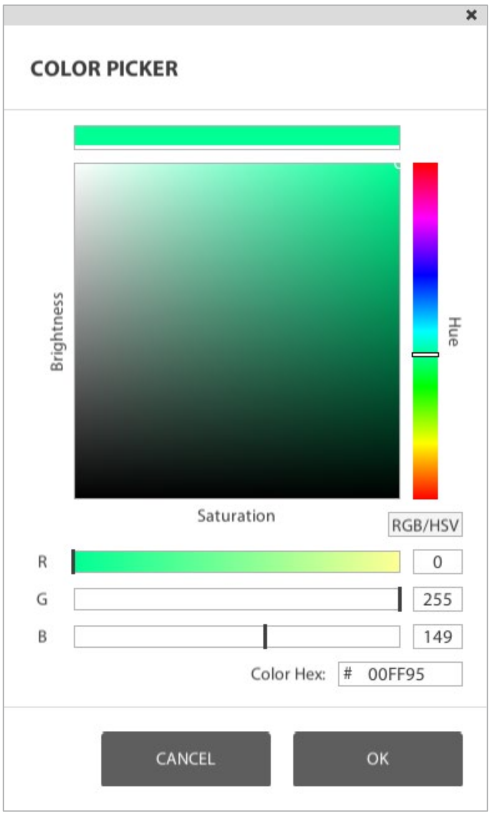


Fig. 53: Colour definition via the Color Picker

13.7 Input

The behaviour of AM-Studio when importing parts can be defined in the menu ***Input***.

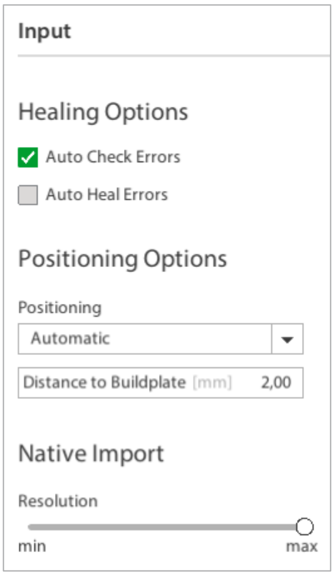


Fig. 54: Settings in the Input menu

Healing Options

In ***Healing Options***, the functions "**Auto Check Errors**" and "**Auto Heal Errors**" can be activated.

- If "**Auto Check Errors**" is activated, AM-Studio automatically checks parts for errors during import. Faulty parts are marked on the substrate plate with a red first aid kit, in the model tree they are marked with a red dot. Faulty areas are highlighted in colour (see Repairing a geometry).
- If "**Auto Heal Errors**" is activated, the geometry of faulty parts is automatically repaired during import. The function "**Auto Heal Errors**" is deactivated by default in AM-Studio.

Positioning Options

Under ***Positioning Options***, the drop-down menu "**Positioning**" can be used to select between three positioning options when importing parts.

- If the option "As in Part" is selected, parts are positioned on the substrate plate according to the data stored in the 3D object.
- If the option "Automatic" is selected, the first imported part is positioned on the set zero point of the substrate plate. When importing several parts, parts are positioned next to each other.
- If the option "Standard" is selected, all parts are positioned on the set zero point of the substrate plate during import.

The option "**Use Distance to Buildplate**" can only be activated or deactivated with the option "As in Part". If "**Use Distance to Buildplate**" is activated, an offset value for the distance of the part to the substrate plate can be entered manually in the input field "**Distance to Build Plate**".

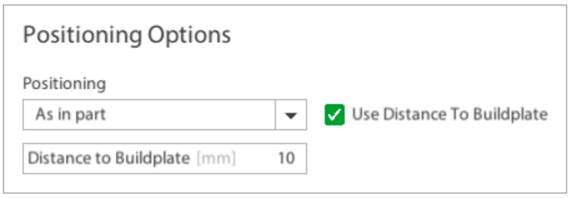


Fig. 55: Settings in the Positioning Options menu

The value set in "**Distance to Build Plate**" is always used for the options "Automatic" and "Standard" during import.

Native Import

Under ***Native Import***, a slider can be used to define the resolution in which native data is imported into AM-Studio. The level of resolution can be set gradually in a range between minimum resolution (min) and maximum resolution (max).

13.8 Output

In ***Output***, you determine where projects created in AM-Studio are saved on the computer.

Path Options

AM-Studio creates a standard output path for projects during installation. Clicking on the magnifying glass symbol opens an explorer window with which a new output path can be defined.

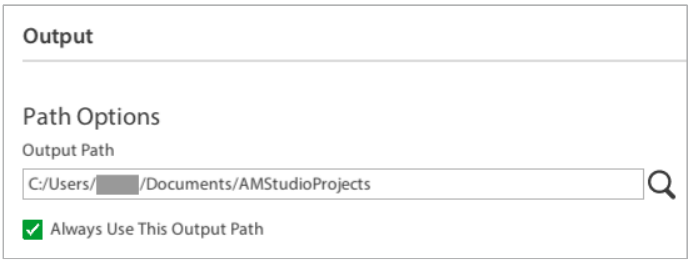


Fig. 56: Menu for the output path

If the same output path is always to be used, a check mark can be placed in the option "**Always use this output path**".

13.9 Support

In ***Support***, options for support structures can be activated or deactivated.

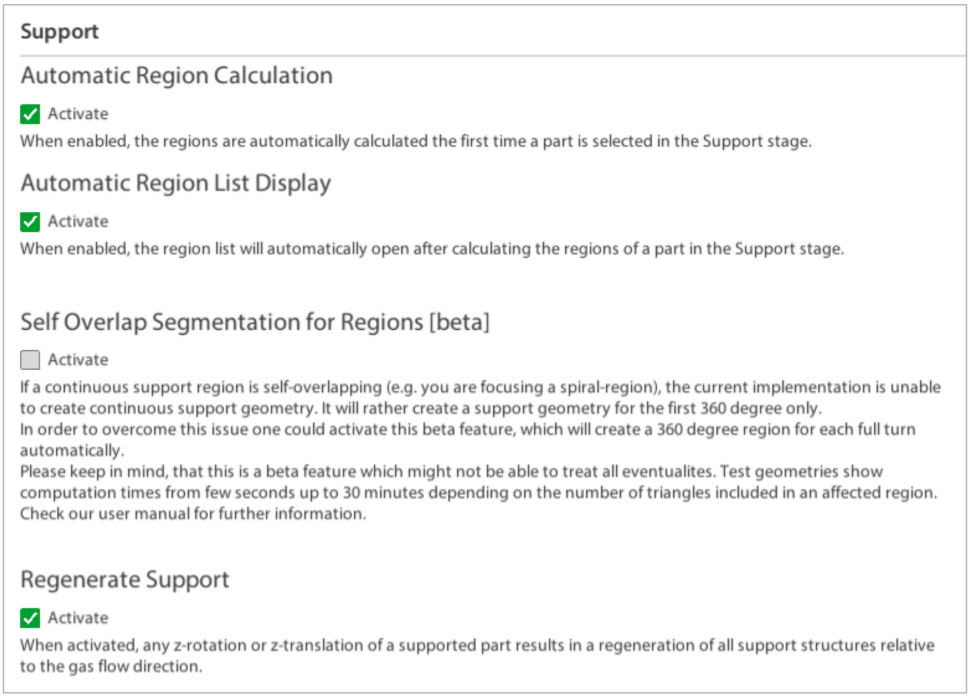


Fig. 57: Support options

Automatic Region Calculation

By setting a check mark ✅ next to **Activate**, the "Automatic Region Calculation" function can be activated. If the Automatic Region Calculation function has been activated, support regions are automatically generated when a part is first selected.

Automatic Region List Display

By setting a check mark ✅ next to **Activate**, the "Automatic Region List Display" function can be activated. If the Automatic Region List Display function has been activated, in the **Support** action step the list of support regions opens automatically after the part regions have been calculated.

Self Ovelap Segmentation for Regions

By setting a check mark ✅ next to **Activate** the "Self Overlap Segmentation for Regions [beta]" function can be activated.

Regenerate Support

By setting a check mark ✅ at **Activate** the function "Regenerate Support" can be activated.