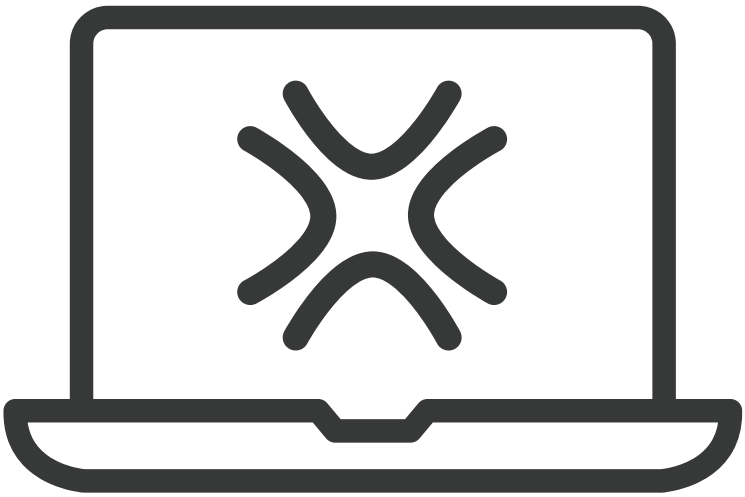
SINTERIT



Sinterit STUDIO

SOFTWARE VER. 1.10.7.1

Original user manual

Please read the manual before using the product.
For the most up-to-date manual,
visit our website: www.sinterit.com/support/

Version 02/2025/EN

Table of Contents

1. INSTALLATION

System requirements for installing Sinterit STUDIO Software:
64-bit processor, Windows 7 or higher, Minimum 1 GB of disk space, Minimum 2 GB of RAM, Graphics adapter compatible with OpenGL 3.0 or higher.

1. Plug the USB flash drive into the USB port on your computer.
2. Locate the **“Sinterit Studio”** folder.
3. Open **“SinteritStudioSetup.exe”** file.
4. Follow the on-screen instructions (Fig 1.1).

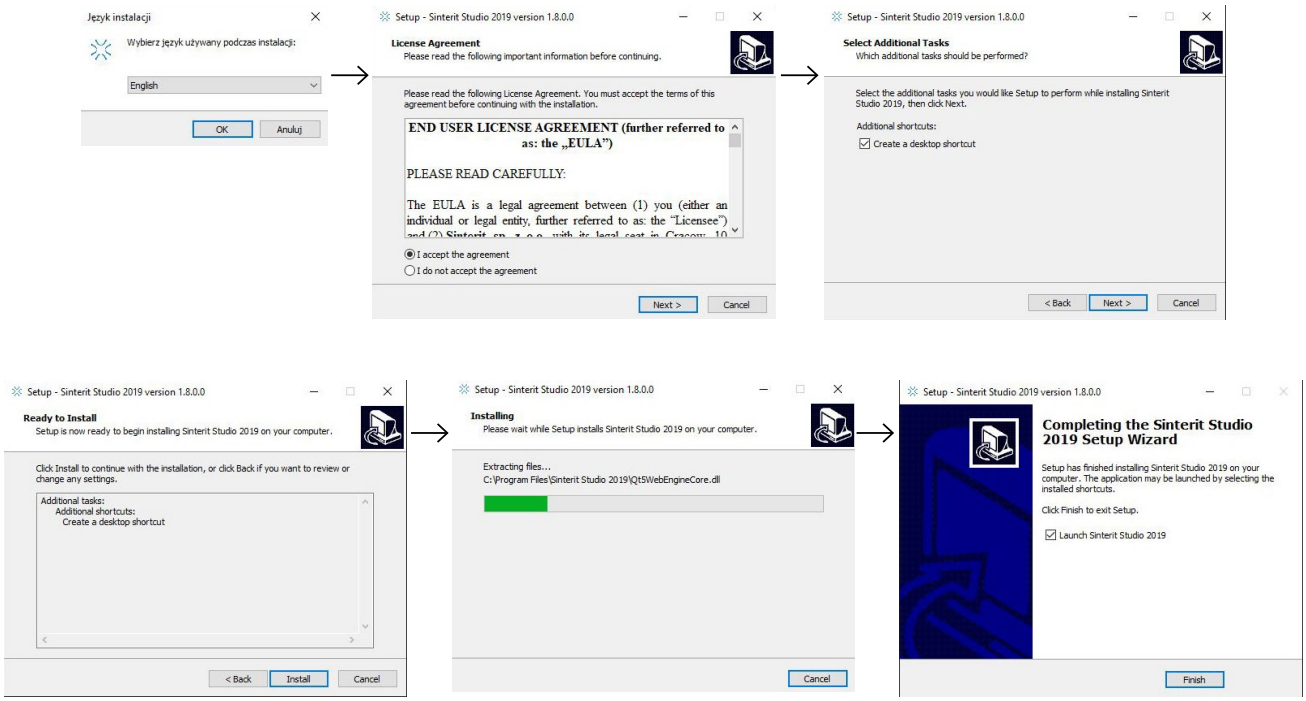


Fig. 1.1 On-screen instructions during installation.

2. TABS OVERVIEW

In order to prepare your models for printing, you must first complete the five steps. You will see them at the top of the window, displayed as tabs.

- PRESET - choosing printer model, powder type, layer height etc.;
- MODELS - arranging models on the PRINT BED;
- SLICE - slicing models into layers and saving the file for printing;
- PREVIEW - previewing layers before printing;
- PRINTERS - status overview of the connected printers.

The key features in the top navigation bar (Fig. 2.1) are:

- **File** - allows you to open a new file (New), open an already saved file (Open), add model files into the project (Import models), save a project in the *.sspf or *.sspfz format (Save, Save As...), open a *.scode file for printing (Load SCode) or exit the program (Exit);
- **Edit** - allows you to undo changes (Undo) or redo them (Redo), cancel recent change of powder type (Undo change material), and perform some basic model operations on the MODELS tab: (Select all), (Move model), (Remove model), (Duplicate model).
- **Settings** - allows you to customize the display (Display settings) and position of models (Editing settings); as well as import or export custom profiles (Export and Import custom materials). You can also change (Model colors), manually add a printer to the Printers tab (Add printer IP address) and (Import/export models) used in the project.
- **Help** - allows you to check for a software update (Check for update), update a printer (Check for Lisa X update, Update printer), view manuals (Manuals), use product key (Enter product key) or check basic information about software (About) and any required (Legal) disclosures.

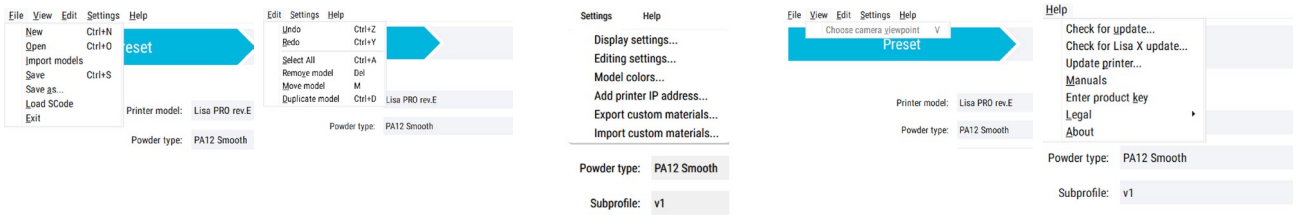


Fig. 2.1 Top navigation bar.

File types in Sinterit Studio:

- ***.sspf** - the basic project format in Sinterit STUDIO, it doesn't contain model files;
- ***.sspfz** - a *.sspf file compressed together with models used in the project. It is useful for transferring the project to an external device or sending it online;
- ***.scode** - a sliced file, ready to print with Sinterit SLS printers;
- ***.stl, *.fbx, *.dxf, *.dae, *.obj, *.3ds, *.3mf** - file formats supported by Sinterit STUDIO.

2.1 Preset

IMPORTANT
The settings in this section are **global.** This allows for setting up parameters for the whole build - these are essential for powder reusability and powder management during printing.

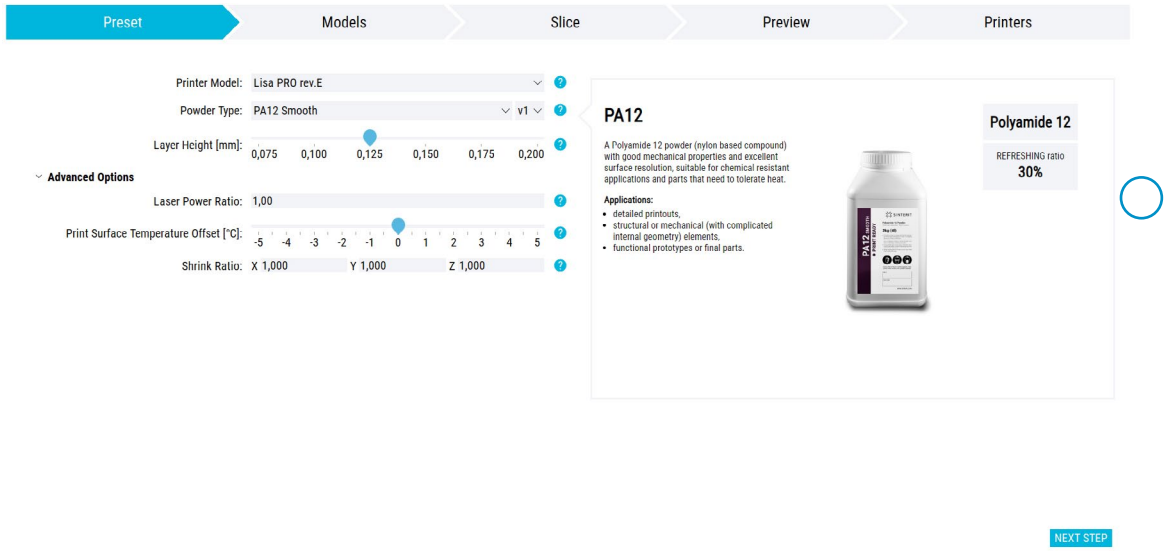


*Fig. 2.2 **Preset** step view.*

2.1.1 Basic settings

- **Printer Model** - choosing your printer model. Depending on your printer type, you will see a different list of available powders. For example, PA11 ESD is available when Lisa PRO rev. E is selected, but it cannot be chosen for Lisa rev. B.

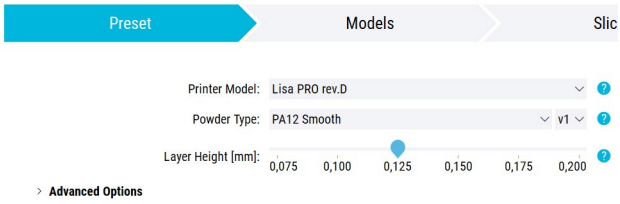


Fig. 2.3 Choosing the printer model.

- **Powder Type** - selecting powder type. Once the desired powder is selected, dedicated printing parameters appear in the other tabs. The selection of available materials depends on your software version and printer model. Select Archived materials to access the profiles for discontinued powder types.

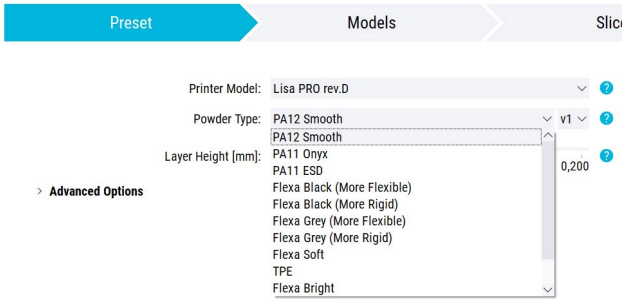


Fig. 2.4 Choosing powder type.

- **Subprofile** - Sinterit sometimes makes changes to the powder types available on the market. This setting allows the user to still use any powder on hand, of a previously available formulation, without disrupting their workflow.

Fig. 2.5 Choosing powder profile.

- **Layer Height** - vertical distance between consecutive project slices. Adjustments will change the duration and accuracy of the process. Move the slider to make changes

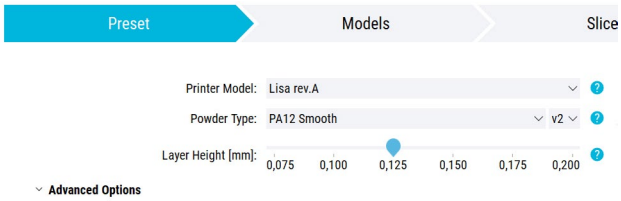
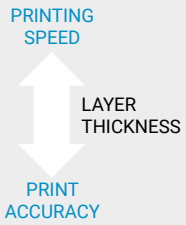


Fig. 2.6 Changing the layer height parameter.

IMPORTANT
Increasing the layer height from 0.100 to 0.125 [mm] reduces printing time but decreases the fidelity of the printed object.



2.1.2 Advanced options

Additional settings that allow you to better customize the printing process.

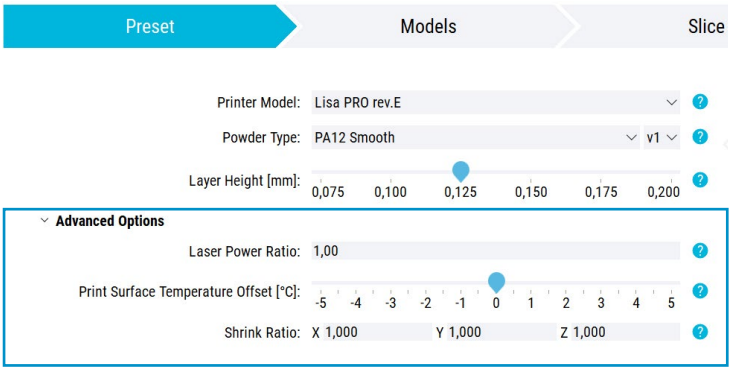
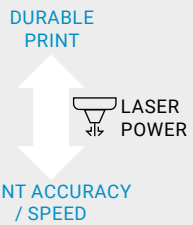


Fig. 2.7 Advanced options

- **Laser Power Ratio** - final laser power value would be multiplied by this factor. Allowed range: 0.5-3.0.

IMPORTANT
1.0 is the standard power for a specific powder type (100%). Increasing the power (e.g. to 1.3) enables to achieve greater durability of the printed object but also reduces precision (“spilling” of melted powder, lack of detail) and in some cases (TPU, more rigid) the printing speed.



- **Print Surface Temperature Offset [°C]** - selected temperature will be added to Print bed temperature for the whole build. It is recommended to increase temperature by +0.5 [°C] for highly utilized builds, or when cake is too powdery. When the cake is too solid it is recommended to decrease temperature by -0.5 [°C]. Decreasing the temperature can help with cleaning and setting for motion movable parts but also may develop an orange peel effect or even layer dislocation.
- **Shrink Ratio** - shrinkage ratio of the material. The models will be expanded along the width of the print bed so that after the shrinkage it will have the expected size. Parameter is used as dimension multiplier - higher value effects in bigger final parts and vice versa. It can be changed in the X, Y or Z axis. Allowed range: 0.9-1.1.

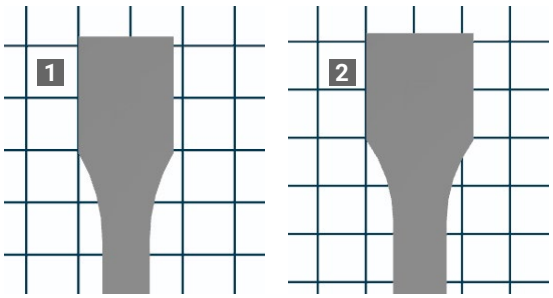


Fig. 2.8 Difference in applying shrinkage of 0.9 (1) and 1.1 (2) in the X axis.

- **Use short warmup** - tick to encode within the slice file the command to greatly shorten warmup time. Only available for PA12 Industrial projects, only with Sinterit STUDIO ADVANCED on Lisa X printers with firmware version 590 or later (⚙ **SETTINGS → SYSTEM INFO**), in rev. K and later with support for the feature (⚙ **SETTINGS → SYSTEM INFO → ACTIVE FEATURES**).

2.2 Custom Material Parameters (open parameters)

Additional parameters have been provided for users who are interested in developing current and new materials. From the **Powder Type** list, in the **Preset** step, select **Custom Material**... A new list named **Custom Material Parameters** will appear.

On the very bottom of the parameter list, you can click the button (Apply to all models) to update all extant models to the selected print settings. You can also choose to (Save) or (Delete material) without scrolling all the way up.

2.2.1 Basic settings

This section contains:
- **Material name** - custom material will be saved with the name set by the user,
- **Modify existing material** - to modify an existing material check the box and select the material you are interested in,
- **Nitrogen required** - use when material is exposed to oxidation. Due to nitrogen connection to the printer, the amount of oxygen during processing is minimized,
- **Refresh ratio [%]** - the parameter defines how much fresh powder has to be mixed with used powder to maintain its printing capability as print ready powder. For example with a 50% refresh ratio it is necessary to mix the same amount of fresh powder as used powder. Used powder in this case is defined as remaining powder from cake without printed parts volume. Residual powder in the feed bed and overflow powder is not counted but it should be added to the mixture,
- **Recoater Blade required** - tick to require the Recoater Blade to be installed before printing,
- **Intake fan RPM, Exhaust fan RPM** - in Lisa X there is a laser protective glass system that uses airflow to protect glass from vapours generated as the powder melts. The fans are controlled by RPMs set by the user in range of (0-12600). For flexible materials it is recommended to keep both intake and exhaust fans at the same 12600 RPM level, but for other materials, e.g. PA12 or PA11 it is recommended to instead lower the intake to 3700 RPM, while keeping the intake at max (12600 RPM).

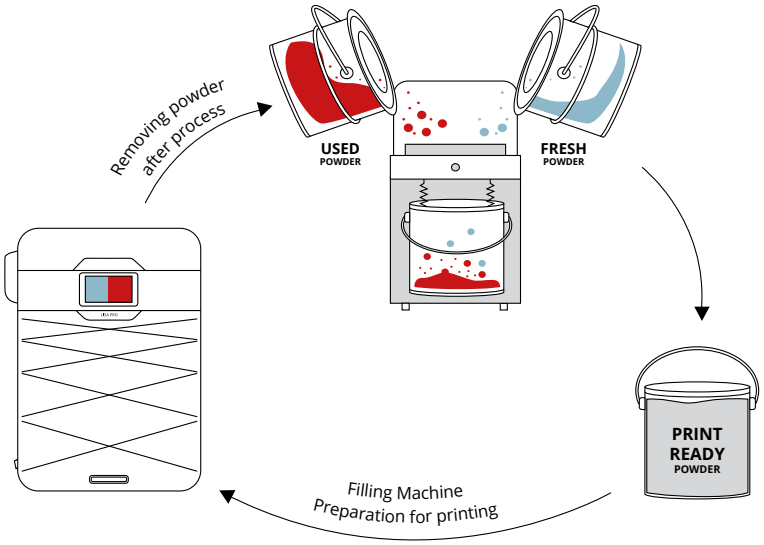


Fig. 2.9 Powder refreshing process.

- **Empty layer feed ratio** - Influencing factor on how much powder is required to cover one print bed layer without melted parts on the previous layer. Printer calculates amount of powder to be recoated via the following formula:

$$H\,[mm] = Z\,[mm] \times \frac{3}{4} \times \left(A + B \times \frac{X\,[mm]}{200\,[mm]}\right)$$

H - Vertical movement of feed bed before powder recoat [mm]
Z - Layer Height [mm]
A - Empty layer feed ratio
B - Full layer feed ratio
X - Total length of printouts on layer in X axis [mm]

The formula is calculated for each single printed layer due to the variable level of layer filling.

- **Full layer feed ratio** - Influencing factor on how much powder is required to cover one print bed layer with melted parts on the previous layer. Printer calculates amount of powder to be recoated via the formula below:

$$H\,[mm] = Z\,[mm] \times \frac{3}{4} \times \left(A + B \times \frac{X\,[mm]}{200\,[mm]}\right)$$

H - Vertical movement of feed bed before powder recoat [mm]
Z - Layer Height [mm]
A - Empty layer feed ratio
B - Full layer feed ratio
X - Total length of printouts on layer in X axis [mm]

The formula is calculated for each single printed layer due to the variant level of layer filling.

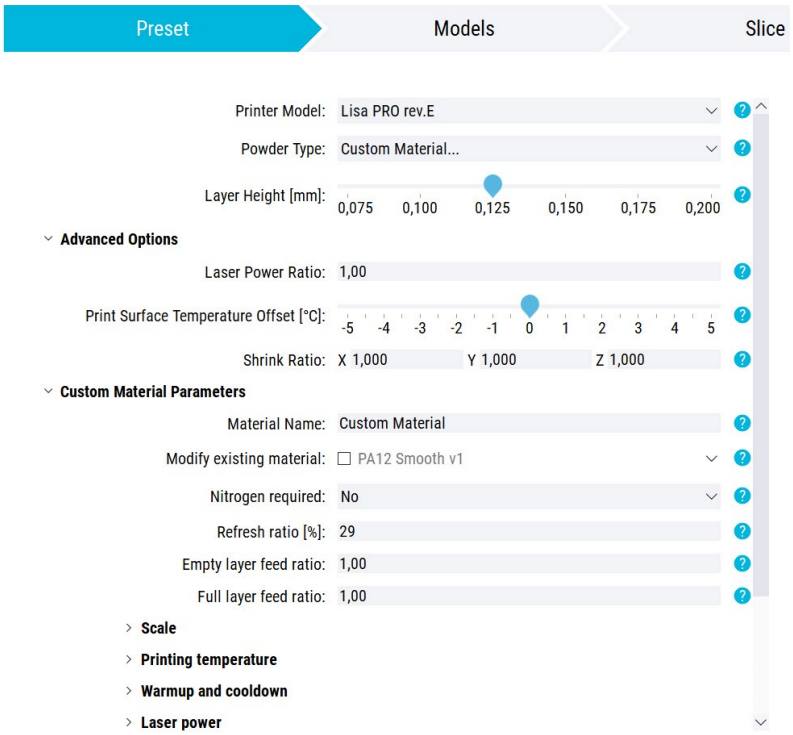


Fig. 2.10 Custom Material Parameters - basic settings.

- **Minimum layer time** - always wait at least that long before recoating two consecutive layers,
- **Wait time after recoating** - wait for an additional amount of time at the start of printing each layer,
- **Recoater parking position** - position for the recoater to stay in while the layer is being printed.

2.2.2 Scale

This section allows you to adjust the virtual size of printouts to balance shrinkage of models during printing.

- **Shrink Ratio** - Shrinkage ratio of the material. The models will be expanded along the width of the print bed so that after the shrinkage it will have the expected size. Parameter is used as dimension multiplier - higher value effects in bigger final parts and vice versa. It can be changed in the X, Y or Z axis. Allowed range: 0.9-1.1.

Fig. 2.11 Scale settings.

2.2.3 Printing temperature

This section allows to set targets for each heater group and to control piston temperature drop during printing.

- **Feed bed temperature** - allowed range: 0-150. temperature value that will be set as a target on Feed Bed surface. This temperature value should never be set as high as Print bed temperature, as it can lead to certain issues with the powder in the Feed bed.
- **Print bed temperature** - temperature value that will be set as a target on the surface of the Print Bed. Allowed range is 0-210 [°C]. Temperature should always be set at least a few [°C] lower than powder melting point. Rubber-like materials do not require temperatures close to melting point, but PA type materials usually do (typically around 5 [°C] below melting point temperature),
- **Cylinder temperature** - temperature value that will be set as target on the cylinder heaters. Allowed range is 0-180 [°C]. Temperature should always be set a few [°C] lower than powder melting point. Increase of this parameter value can reduce the parts' bending inside the chamber during printing,
- **Piston temperature** - temperature value that will be set as target on the piston heaters. Allowed range is 0-180 [°C]. Temperature should always be set a few [°C] lower than powder melting point. Increase of this parameter value can minimize first layer's curling effect, but setting it too high can result in powder melting or degradation,

- **Print chamber temperature** - temperature value that will be set as target on the side heaters. Allowed range is 0-140 [°C]. This temperature value should never be set as high as the Print Bed temperature, as it can lead to certain issues with the powder in the Feed bed. It aids in preheating the powder so its value should be set at a safe powder level,
- **Piston temperature reduction** - allows you to customise piston temperature changes at different heights of the print in progress (excluding warmup height).The piston temperature is important at the very start of printing - it prevents warping. Afterwards, it should be lowered, to limit the thermal degradation of the powder.

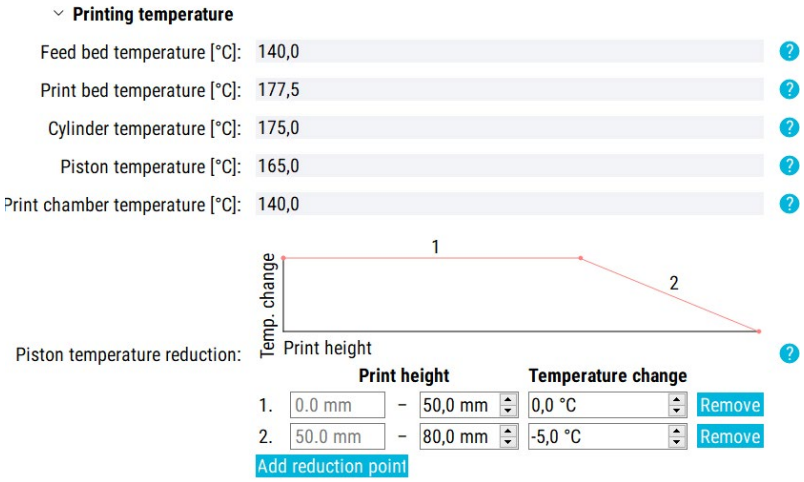


Fig. 2.12 Printing temperature section.

2.2.4 Warmup and cooldown

This section allows to manage time and height of warmup and cooldown:

- **Rising temperature warmup height** - amount of powder to be recoated before printing that is started before the print bed target temperature is achieved. To prepare part bed for printing, the target temperature during warmup is 1.5 °C higher than during printing. Rapid heating can cause problems with part bed locally overheating,
- **Rising temperature warmup time** - period of time over which to raise temperature from 50°C to the target temperature (does not include time to recoat the powder).
- **Constant temperature warmup height** - amount of powder to recoat before printing is started while temperature stays at the target temperature. It helps to stabilize temperature on part bed and make it even before printing starts,
- **Constant temperature warmup time** - period of time over which to keep the temperature at the target temperature (does not include time to recoat the powder).
- **Cooldown cover height** - amount of powder to be recoated when printing is completed while the temperature is kept at the target temperature,
- **Cooldown time** - period of time over which temperature settings would be proportionally decreased from the printing targets to the heaters' turning off without powder recoating. For materials that are printed in high temperatures, insufficient cooldown time can cause excessive warping and bending of printouts. After the cooldown is complete the printer still can be too hot (>50°C) to be opened.

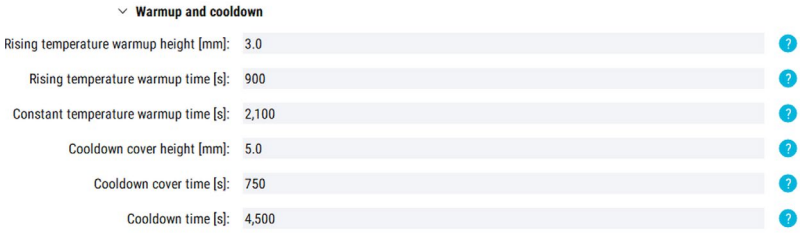


Fig. 2.13 Warmup and cooldown section.

- **Rising temperature warmup time** - period of time over which to raise temperature from 50C to the target temperature (does not include time to recoat the powder).

2.2.5 Laser power

This section allows to adjust parameters related to the power of the laser:

- **Energy scale** - parameter that increases the laser power used to melt select single model. Concerns both infill and perimeters. Works as a multiplier for all parameters that define final laser power,
- **Max energy per cm³, infill** - one of the parameters used to define laser energy on infill. Has a small impact on laser energy through the first layers but a marked effect on layers at the depths equal or higher to that defined by ”max depth - infill”. For example setting value to 260 from 250 with ”max depth infill” set to 0.7 increases infill laser power at 0.1 mm by 1.7% but on 0.7 mm by 3.4%,
- **Const energy, infill** - one of the parameters used to define laser energy on infill. Has high impact on laser energy through the first layers but a less significant effect on the layers at depth equal or higher to that defined by ”max depth - infill”. For example setting value to 0.6 from 0.5 with ”max depth infill” set to 0.7 increases infill laser power at 0.1 mm by 11.7% but on 0.7 mm by 3.4%,
- **Max power depth, infill** - maximum defined laser power would be used after reaching depth specified by this value. Before reaching this depth, laser power is gradually decreased. An insufficient value of this parameter results in excessively melted first layers of the infill surface. On the other hand, an overly high value results in the first layers of infill falling off,
- **Max infill energy multiplier per repeat** - if multiple repeats of infills are being drawn, you can draw those repeats with different laser power. This parameter accepts a semicolon-separated list of numbers. Each number is a multiplier for a given repeat of infills. E.g. „0.3;0.7” means that the first repeat of infill will be printed with 0.3 of laser power computed from the parameters above, the second one with 0.7 of the power, and all the following ones exactly at computed power.
- **Max energy per cm³, perimeters** - one of the parameters used to define laser energy on perimeters. Has a small impact on laser energy through the first layers but a marked effect on the layers at depth equal or higher to that defined by ”max depth - perimeters”. For example setting value to 260 from 250 with ”max depth perimeters” set to 0.7 increases perimeters laser power at 0.1 mm by 1.7% but on 0.7 mm by 3.4%,
- **Const energy, perimeters** - one of the parameters used to define laser energy on the perimeters. Has high impact on laser energy through the first layers but a smaller effect on the layers at depth equal or higher to that defined by ”max depth - perimeters”. For example setting value to 0.6 from 0.5 with ”max depth perimeters” set to 0.7 increases perimeters laser power at 0.1 mm by 11.7% but on 0.7 mm by 3.4%,
- **Max power depth, perimeters** - maximum defined laser power would be used after reaching depth specified by this value. Before reaching this depth, laser power is gradually decreased. Too low value of this parameter results in excessively melted first layers of perimeters. On the other hand, too high value results in falling off first layers of perimeters.
- **Max perimeter energy multiplier per repeat** - if multiple repeats of perimeters are being drawn, you can draw those repeats with different laser power. This parameter accepts a semicolon-separated list of numbers. Each number is a multiplier for a given repeat of perimeters. E.g. „0.3;0.7” means that the first repeat of perimeters will be printed with 0.3 of laser power computed from the parameters above, the second one with 0.7 of the power, and all the following ones exactly at computed power.

Fig. 2.14 Laser power section.

2.2.6 Laser movement and geometry

- **Drawing order** - when the repeat count of infills or perimeters is larger than 1, this parameter is used how to interleave drawings of infills vs. perimeters. When „Infill First, Interleave" or „Perimeters First, Interleave" is selected, drawing infills will be interleaved with drawing perimeters, starting with infills or perimeters respectively. When „All Infill First" or „All Perimeters First" is selected, all repeats of infill (or perimeters) are being drawn first before the repeats of perimeters (or infills) are being drawn. The other parameter that impacts the ordering of repeated models is „Repeated scanning strategy".
- **Perimeter repeats** - use perimeters more than once. The amount of perimeters used is defined via this parameter. The lines are printed one after another. Using more than one perimeter can strengthen models and improve details while using powders that require a high amount of energy. Most effective on rubber like materials,
- **Infill repeats** - use infill more than once. The amount of infill used is defined by this parameter. The Lines are printed one after another. Using more than one infill can strengthen models while using powders that require a high amount of energy. Most effective on rubber like materials,
- **Infill direction** - choose the desired angle of approach of the laser.
- **Repeated scanning strategy** - when the repeat count of infills or perimeters is larger than 1, this parameter is used to decide how to order repeated drawings of models. When „Repeat whole layer" is selected, then all the models will be printed once before we repeat drawing them again. When „Repeat each model" is selected, each model will be printed as many times as requested before we start printing another model. The order of drawing repeated infills vs. perimeters is controlled by the „Drawing order" parameter.
- **Number of perimeters** - number of perimeters around the infill. While using more then 1 perimeter, each line is printed closer to the model center with an offset defined by the offset between perimeters parameter,

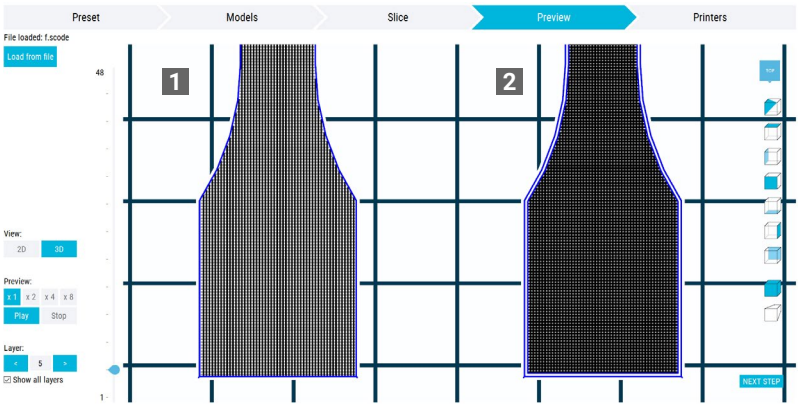


Fig. 2.15 The difference between a model printed with one perimeter line (1) and one printed with 2 perimeter lines with "Next perimeter offset" value set to 0.4 [mm](2).

- **First perimeter offset** - offset between the model wall and the midpoint of the first perimeter line. This parameter is used to improve the scale of the models. Increasing its value results in model size decrease by about twice the parameter value and vice versa,
- **Offset between perimeters** - offset between the midpoint of perimeter lines. Applicable if the number of perimeters is bigger than one. Only usable with the Number of perimeters option, does not apply to Perimeter repeats. Parameter change can result in quality improvement,
- **Infill offset** - gap between infill line end and perimeters. The length is measured between the focus of the laser beam used to print infill and perimeters. Adjusting the value can result in a better connection between the perimeters and the infill,
- **Hatch spacing** - separation between two consecutive infill lines, which is defined by the distance between the foci of the laser beams. It has a huge impact on the tensile strength of the printed model - typically, lowering this parameter improves the mechanical properties of the printout but at a cost of increasing print duration. This happens because with a lower value of this parameter, the lines of infill are partially overlapping due to the size of the laser dot greater than the parameter value.

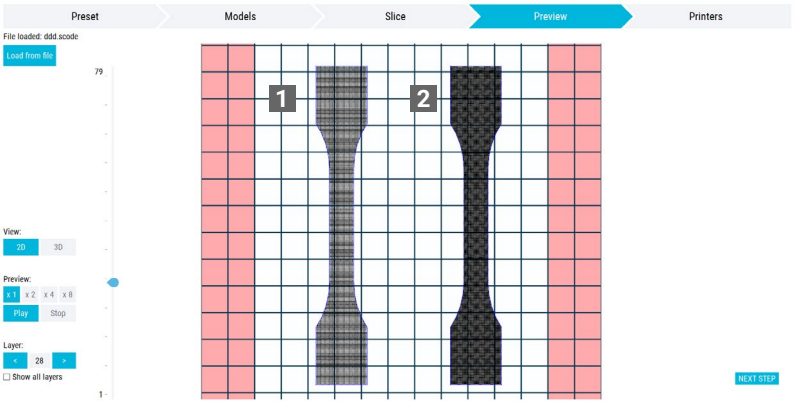


Fig. 2.16 The difference between the model with hatch spacing parameter set to 0.5 (left) and 0.3 (right). Right model is printed with much more infill lines.

- **Model shell wall thickness** - this parameter defines the maximum shell wall thickness. Greater shell thickness results in more durable printouts at the cost of printing time.
- **Laser power ratio inside shell** - this parameter controls printing on the inside of the shell wall (defaults to 1.0).

You can set it to 0 to print a hollow shell (assuming you leave an opening to remove any unsintered powder afterwards). Other values can allow you to print parts with different physical properties on the inside and on the outside of the shell.

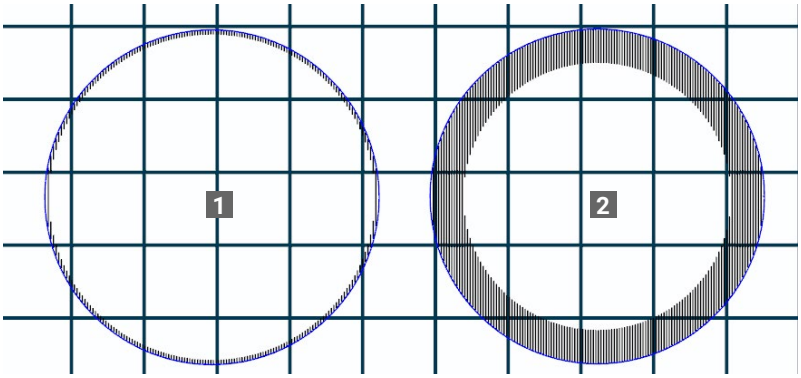


Fig. 2.17 The difference between the model with shell thickness parameter set to 1 (1) and 5 (2).

Fig. 2.18 Laser movement and geometry section.

2.2.7 Skeletons

This parameter is designed for the small details of the model that can be damaged. Skeletons are enabled by default and can only be turned off in the Models step. This section contains:

- **Skeleton wall laser scale** - this parameter can be used to enhance fine details that may fall off or break easily. Multiply laser power by this number when printing thin walls (walls that are printed with one laser infill line) at a distance bigger than 0.2 mm from the model surface,

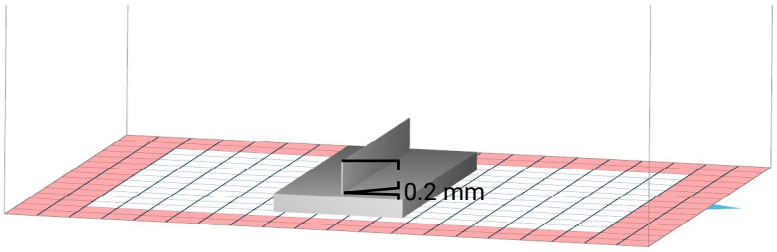


Fig. 2.19 Image illustrates the range of this parameter area of effect.

- **Surface skeleton wall laser scale** - this parameter can be used to enhance fine details that may fall off or break easily. Multiply laser power by this number when printing thin walls (walls that are printed with one laser infill line) at a distance less than 0.2 mm from the model surface,

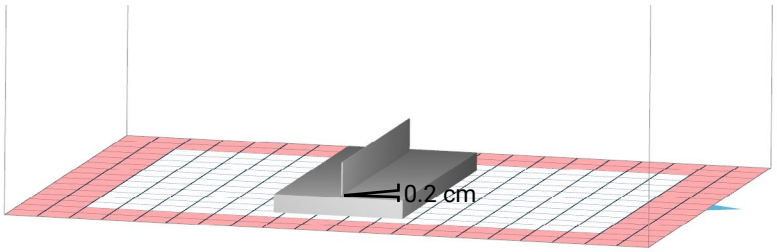


Fig 2.20 Image illustrates the range of this parameter area of effect.

- **Dot laser scale** - this parameter can be used to enhance fine details that may fall off or break easily. Multiply laser power by this number when printing single dots at a distance greater than 0.2 mm from the model surface,
- **Surface dot laser scale** - this parameter can be used to enhance fine details that may fall off or break easily. Multiply laser power by this number when printing single dots at a distance less than 0.2 mm from the model surface. Examples of this rule are sharp edges, extremely thin cylinders or tips of the cones.

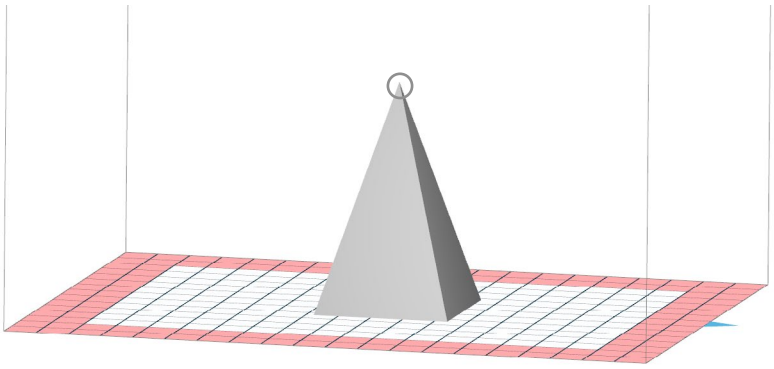


Fig. 2.21 Image illustrates the range of this parameter area of effect.

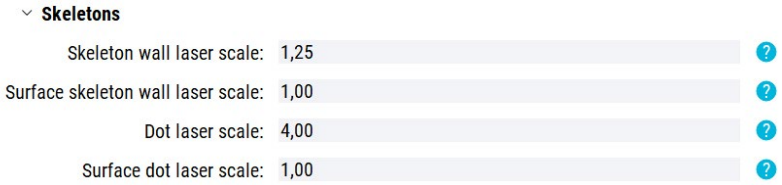


Fig. 2.22 Skeletons section.

Move on to the next step by clicking Next step (1) in the bottom right corner of the window or Models (2) at the top of the dialog. (Fig. 2.23)

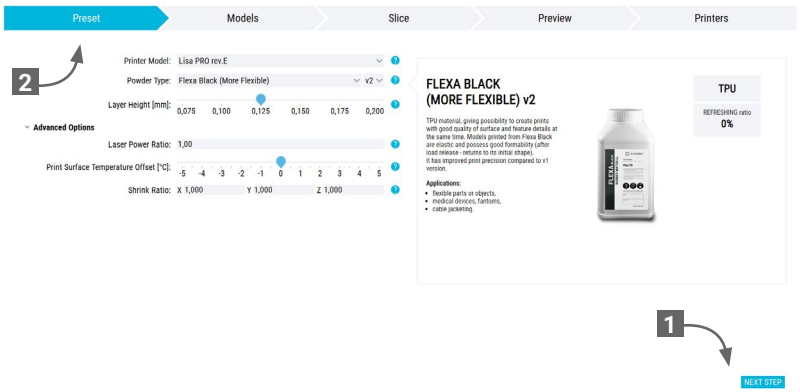


Fig. 2.23 Moving on to the next step.

2.3 Models

This step is a visualization of the alignment of the models in the Print Bed.

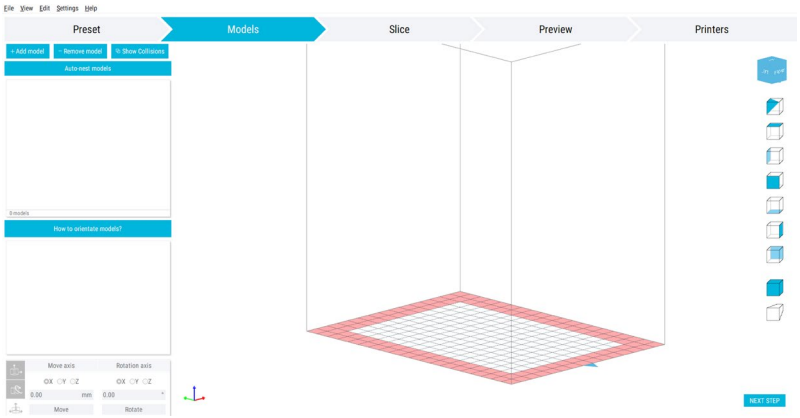


*Fig. 2.24 **Models** step view.*

Click the „How to orientate models?" button to view an article exploring the topic in detail.

2.3.1 Adding/removing model

- **+ ADD MODEL** - allows to add models to the Print bed. Supported file formats: *.stl, *.fbx, *.dxf, *.dae, *.obj, *.3ds, *.3mf)
- **- REMOVE MODEL** - allows to remove a single model from the Print bed. You can also select the model and use the delete key on the keyboard.

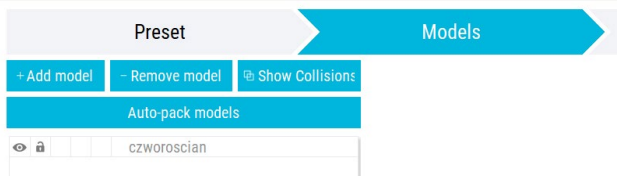


Fig. 2.25 Adding/Removing model.

2.3.2 Collisions

It may happen that you won't see the overlap of the models. You can check this easily. Just select the Show Collisions button. If the models overlap, collision icons (1) will appear next to the model names and the area where the contact occurs will be indicated in red (2) (Fig 2.26).

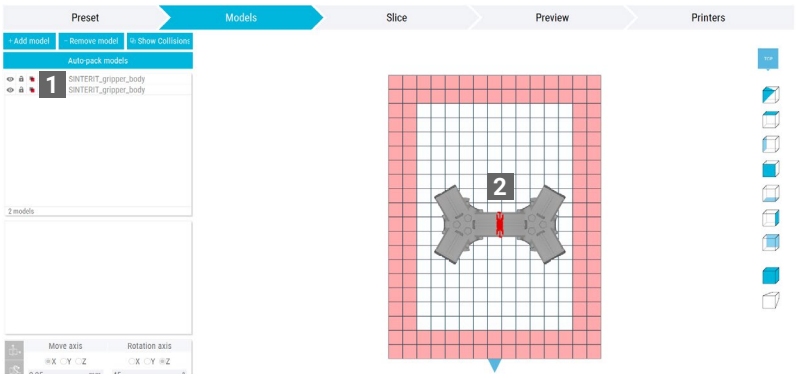


Fig 2.26 Collision of models.

2.3.3 Positioning in the red area

When placing the model, remember to avoid extending past the white area. Placing the model in the red area may lead to deformation or destruction of the printout. The program will inform you in two ways if this situation should happen: a red warning sign (1) will appear next to the model names and the fragment located within the red area will be highlighted in red (2).

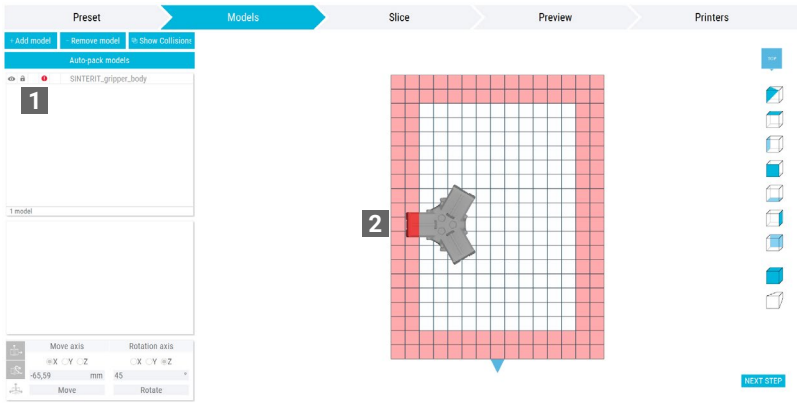


Fig 2.27 Positioning in the red area: warning sign (1) and highlighting the part of the object (2)

2.3.4 Visibility / Locking position

- **Visibility of the model** (1) - the model can be completely visible, transparent or hidden. This feature is useful when a large number of models makes it difficult to arrange them in a print bed.
- **Locking the model position** (2) - model can be locked so the object cannot be moved and rotated; or unlocked.

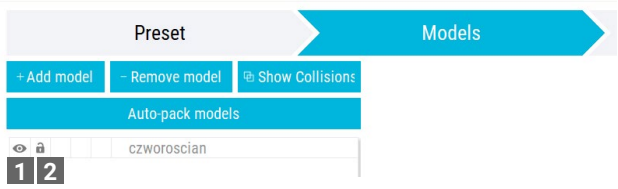


Fig. 2.28 Adding/Removing model.

2.3.5 Properties of the model

On the left side of the window there are tabs with properties (1) of the model. They show up when you click on the model (2).

IMPORTANT
Changes made in this section will change the properties of the **selected model only.** If you want to select more than one model hold down CTRL and select each model simultaneously.

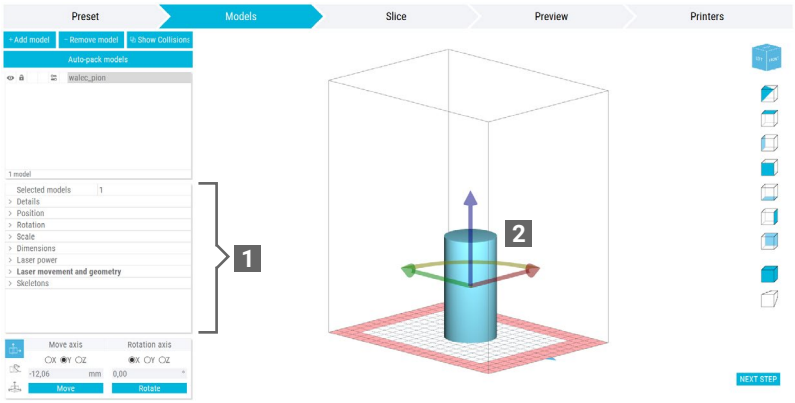
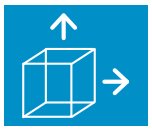


Fig. 2.29 Displaying model properties.

- **Selected models** - the number of selected models,
- **Details** - this tab is informational only. You will find out what is the location of the file (Path) and what is the number of triangles that the model is built of (Faces),
- **Position** - this parameter changes the position of the model in PRINT BED. Values can be inserted manually for each plane (X, Y, Z),
- **Rotation** - this parameter changes the rotation along the selected axis. The values can be entered manually for each axis (Pitch, Yaw, Roll) or after moving the mouse pointer over the selected plane (after switching to Rotation Axis),
- **Scale** - this parameter changes the size of the model. Sizes can be changed individually for each axis (X, Y, Z),
- **Dimensions** - this tab is informational only and shows the dimensions of the model,
- **Laser Power** - allows you to change e.g. energy scale and laser energy. Same parameters as in the Preset step. More information in the section *2.2.6 Laser power,*
- **Laser movement and geometry** - allows you to use perimeters, infill, make gaps between them etc. The parameters are the same as in the Preset step (More information in the section 2.2.6 *Laser movement and geometry*).
- **Skeletons** - allows you to make walls with the thickness equal or lower to that of a single laser line.This function is enabled by default and may be disabled only in the Models step. The parameters are the same as in the Preset step. For more information look up chapter: *2.2.8 Skeletons*.

2.3.6 Move/Rotation axis

In the bottom left corner of the window there is a panel dedicated to moving and rotating the model.



Hide / Show move manipulators - moving the model in three dimensions. Click the button in the bottom left part of the screen to reveal XYZ axes manipulators. By default, the left mouse button should be used, after moving the mouse pointer over the displayed axis. You can also input the desired value and accept it with the Move button.

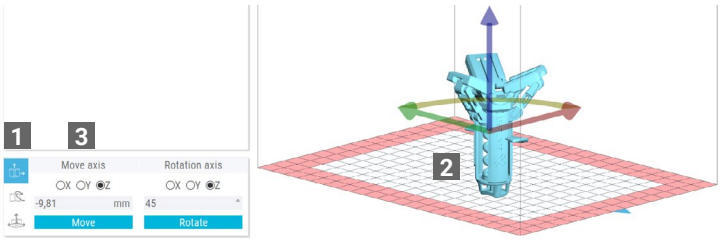


Fig. 2.30 Hide/Show move manipulators button (1), arrows representing the axes (2), entering the move value (3).

Rotation manipulators - click this button (1) to reveal the rotation manipulators. To change the orientation of the model, click on the selected axis and enter the appropriate value (2) (confirm with Rotate button) or click the axis in the model and move it manually (3).

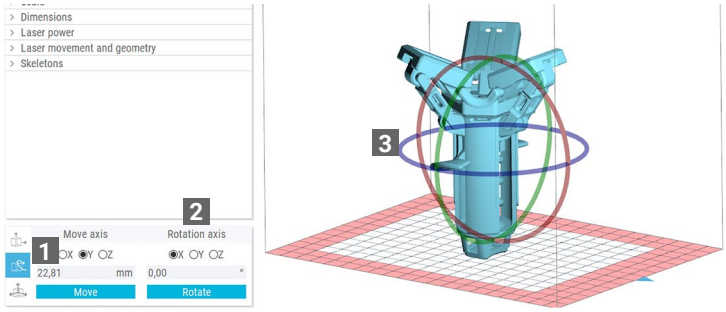


Fig. 2.31 Rotation manipulators button (1), entering the rotation value (2).

Local / Global coordinate system - to facilitate arranging models in Sinterit STUDIO Software, you can switch between global and local (for a given model) coordinate system. In the local system, the entered values add up. If you for example enter 30 degrees and click Rotate twice, the model will rotate a total of 60 degrees.

2.3.7 Context menu

Right-clicking a model (or the name of a model) shows the context menu (Fig. 2.32) which allows you to:

- **Duplicate Models** - you can copy a model multiple times by inserting the desired value in the box that appears. NOTE: The inserted number is the number of models after the duplication. So if you leave "1", the model won't be duplicated. You will find more information in chapter: *2.3.8 Duplicating models,*
- **Remove Models,**
- **Add Models,**
- **Move Models** - allows you to move the model to a selected edge of the safe print bed area: bottom, front, left, back, right,
- **Split Models into Submesh** - allows you to separate the model into individual mesh components,
- **Pack bed** - allows you to automatically arrange the maximum number of models in the Print bed. For more information check chapter *2.3.9 Auto-nesting,*
- **Rest Models** - allows you to change the model rotation settings and the placement of the model in a specific Print bed area,
- **View** - allows you to rotate the camera around the Print bed and the models inside. You can also change the view by pressing the desired place on the view cube or selecting the cube on the right. Both Perspective and Ortho cameras are available,
- **Model Properties** - allows you to copy the properties (rotation and scale) from one model to another.

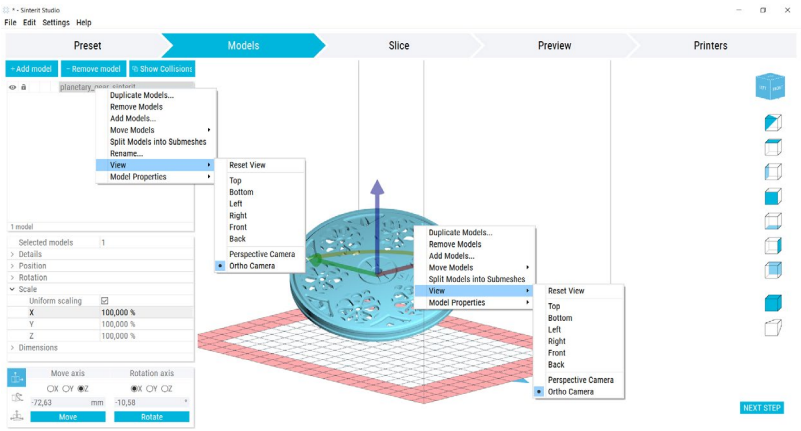


Fig. 2.32 Context menu of the model.

PERSPECTIVE CAMERA (1) - three-dimensional camera view, best for previewing the whole printing bed arrangement. To rotate the camera use the right mouse button.

ORTHO CAMERA (2) - orthogonal projection of the model on the plane (two-dimensional view in the work area). It is useful for accurately arranging objects in the working area. Particularly recommended with Z axis (top view). To rotate the camera use the right mouse button.

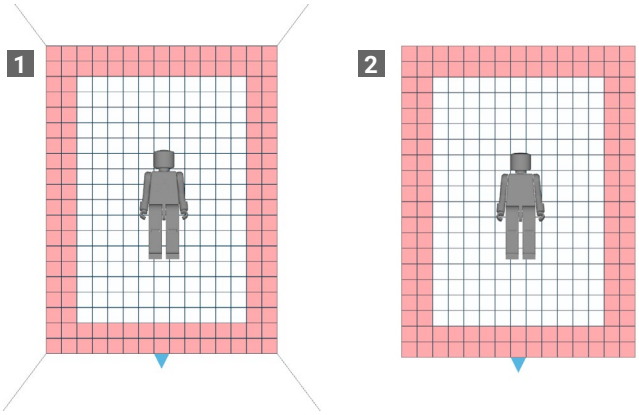


Fig. 2.33 Comparison of the Perspective Camera (1) and Ortho Camera (2) views in the Z axis.

2.3.8 Duplicating models

This is a very useful feature when you are printing multiple models at once. Allows you to duplicate the selected model in the specified amount in the three axes (XYZ).

1. Load the desired model (Models step -> Add model button),
2. Arrange the model according to the instructions from chapter: *3. Positioning of models,*
3. Open the context menu of the model (right-click on the model),
4. Select Duplicate Models...

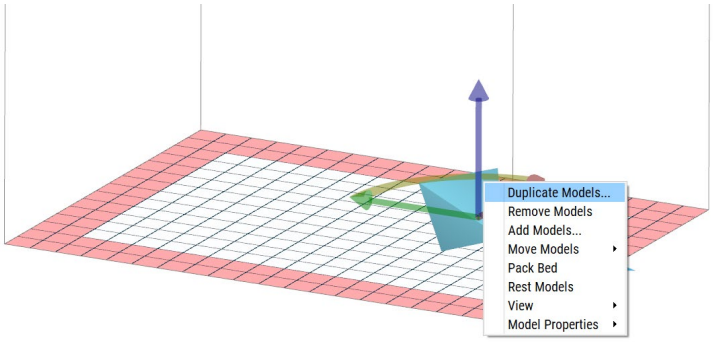


Fig. 2.34 Selecting Duplicate Models from context menu.

5. The ”linear pattern” window that appears contains input areas for you to fill in. The elements of the window mean:

- **Total number of instances** - decide in which axis you want the duplicate model to appear and enter the number of models at the selected axis symbol,
- **Gap** - the gap between duplicate models,
- **Dimensions** - the summed dimension in a given axis containing the dimension of the original model, the duplicated models and the gap between them.

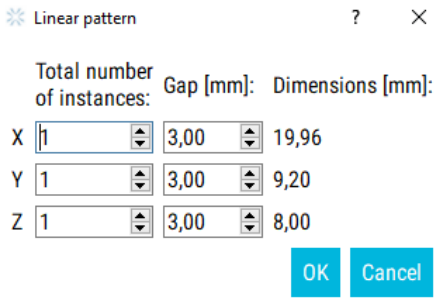


*Fig. 2.35 Linear pattern window (**Duplicate models**).*

The filled table shows that a duplicate model will appear in the Y-axis (i.e. there will be two models in the Y-axis) and the distance between them will be 10 [mm] (Fig. 2.36).

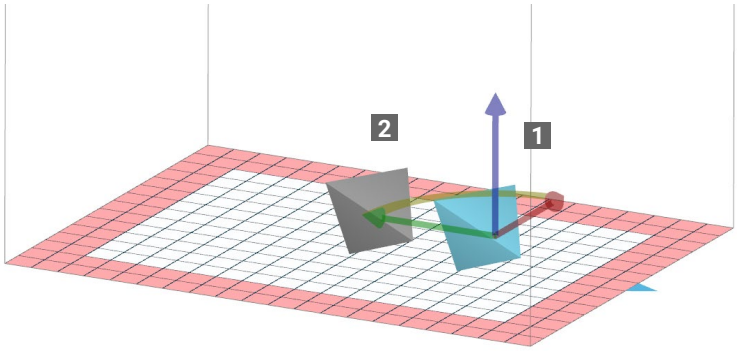


Fig. 2.36 Original (1) and duplicate (2) model.

IMPORTANT
There is a reason why the default gap between objects is 3 [mm]. Try not to reduce this distance to maintain good print quality. For more information see chapter: 3.8 Filling the build chamber.

2.3.9 Auto-nesting

Auto-nesting functionality provides automatic models arrangement in the printing area.
This tool will pack Pinting Area with prepositioned models, which can significantly shorten the time of build preparation.

1. Add model in Models step.
2. Rotate the model accordingly with section *3. Positioning of models.*

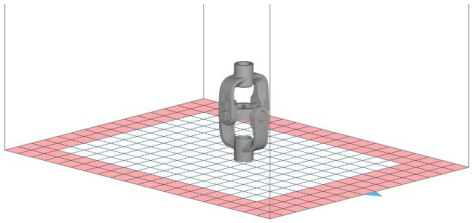


Fig. 2.37 Added and prepared model.

3. Duplicate the model accordingly with section *2.3.8 Duplicating models.* Do not worry about the models in the red area at this point.

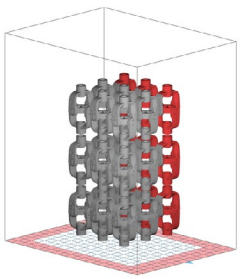


Fig. 2.38 Models after duplication.

4. Right-click on the screen and select Pack Bed. Now the models are not in the red area and there is no collision between them.

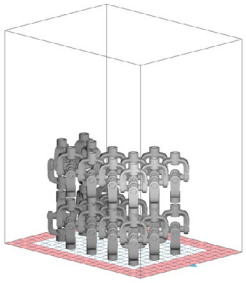


Fig. 2.39 Models after using the Pack Bed function.

2.4 Slice

This step involves slicing the models prepared in the earlier step into layers. Depending on the size of the file, this may take several minutes. Check the **“Generate report”** box to save results of this process. Press Slice and select a location to save the file.

IMPORTANT
Information displayed after the “slicing” process is necessary for further work with the printer.

Information required for preparing the Sinterit LISA/Lisa Pro/Lisa X/NILS 480 printer for printing appears in the dialog box.

Basic information:

- **SCode file** - file name,
- **Material** - used powder type,
- **Layer height,**
- **Estimated total print time,**
- **Estimated powder needed in Feed Bed** - estimated volume of powder required to be added in the Feed bed,
- **Refresh powder needed after print** - volume of Fresh powder required to be added after printing to Print ready powder.

Additional information:

- **Laser power multiplier** - laser power,
- **Total model layers count** - number of layers in the model,
- **Models volume,**
- **Estimated powder needed in Feed Bed (height)** - estimated amount of powder needed in the Feed Bed
- **Total print height,**
- **Estimated warmup time** - the time it takes the printer to warm up to the required temperature,
- **Estimated active print time** - the time during which the actual printing part happens
- **Estimated cooldown time** - the time it takes for the printer to cool down to a temperature that allows it to be opened,
- **Models** - numbers and names of sliced models contained in the project.

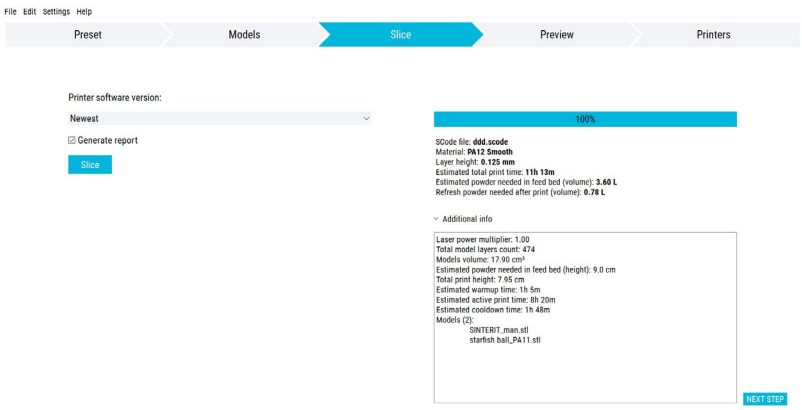


*Fig. 2.40 **Slice** step view.*

IMPORTANT
The *scode file, created at this step will later be sent to the printer. If you are not happy with the slicing or want to change something in the positioning/add a model/change the print settings you can do this and run the slicing again.

2.5 Preview

This tab allows previewing individual layers of the model after the "slicing" stage. This enables careful inspection of the sliced model and the detection of potential mistakes which are not visible at the stage of preparing the file. Depending on your preference, you can choose between **2D** (1) and **3D** views (2).

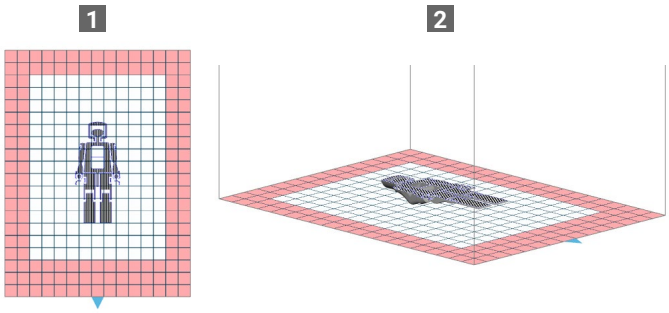


*Fig. 2.41 **2D** (1) and **3D** (2) view in the **Preview step**.*

You can check individual layers in two ways: by clicking the arrows (3) or moving the slider (4). If you want to see previous layers when verifying, check the Show all layers (5) box.

It is also possible to view the printing process of individual layers as an animation (Preview section) at the selected speed (6).

If you already have a *scode file, use the Load from file (7) button.

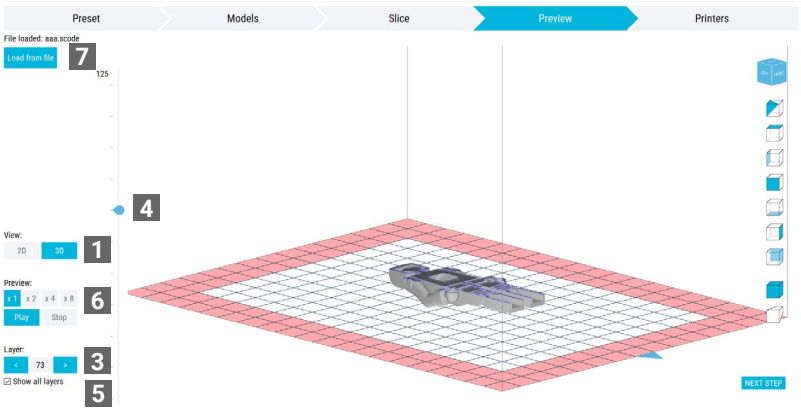


*Fig. 2.42 **Preview** step view.*

2.6 Printers

Here you can check the printing status and temperature inside Sinterit LISA/Lisa Pro/Lisa X/NILS 480 printers (1) connected via Wi-Fi (the instruction how to connect a printer to Wi-Fi network can be found in the printer's instruction manual). This allows you to constantly track the progress of the printing when you are in another room or building. The information you can expect to find at this stage are:

- **IP** - IP number of the printer,
- **S/N** - serial number of the printer,
- **Loaded file** - name of the loaded file,
- **...% - Printing** - printing progress in [%],
- **Time to finish** - how much time is left to finish printing
- **Surface temperature**

Some useful features are also available:

- **Camera View** - you can see what is actually taking place in the printer. The video output can be recorded to a local file (press **START RECORDING**).
- **Name printer*** - you can name the printer to make it easier to distinguish from the others,
- **Send SCode file** - allows you to send prepared file to the printer (WiFi connection required)
- **Update firmware** - you can update the firmware via Wi-Fi *(not available on Lisa X)*.
- **Abort print*** - if remote abort is enabled on the printer itself, user can remotely abort printing from Sinterit STUDIO (Lisa X and NILS only).
  **Only available with Sinterit STUDIO ADVANCED.*

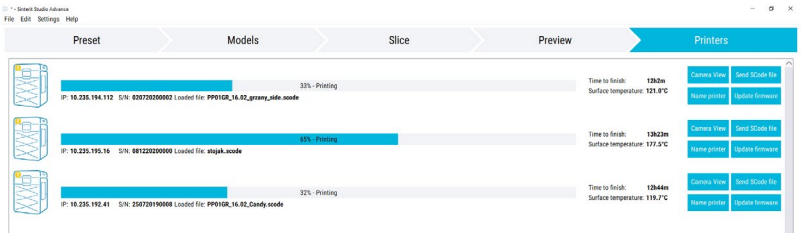


*Fig. 2.43 **Printers** step view.*

IMPORTANT
If the printer is not connected to a WiFi network, the file must be uploaded to the printer via a flash drive. Then load the files onto the flash drive and connect it to the printer at the required time. Follow the instructions on the printer screen.

3. POSITIONING OF MODELS

The first rule in arranging a print in the laser sintering technology is to make the cross-section of a solid model as small as possible which guarantees the best quality-to-durability ratio. In large cross-section surfaces there is an accumulation of heat inside the print, which may lead to internal stress of the material and result in the print edges curling up or down, especially in prints with right angles.

Sinterit STUDIO has several tools to facilitate the arrangement of models. In the Models tab, you can manipulate model settings - pan, rotate and scale. Try to always keep the models within the white rectangle shown in the view, this will allow you to get a properly sintered 3D print.

The tips below concern printing from **PA12 SMOOTH** and **PA11 ONYX** material. While using **FLEXA** or **TPE** powders, these rules are still valid, but don't have such a significant impact on the printouts.

3.1 Flat surfaces

In flat and thin surfaces, a lot of internal strain and shrinking occurs. Do not lay your models flat! The heat accumulating in the layers may cause deformation of your model.

The best solution for this kind of models is to print them rotated by 45 degrees in every axis. This will help to minimise the cross-section of the surface and release heat, which results in a better quality print.

EXCEPTION:
Flat surfaces of up to 12 cm^2 or consisting of only one layer (e. g. a booklet page).

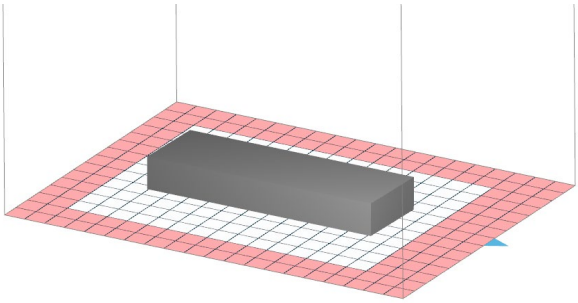


Fig. 3.1 ***Incorrect*** *arrangement of a flat model. In both cases, accumulation of heat may occur.*

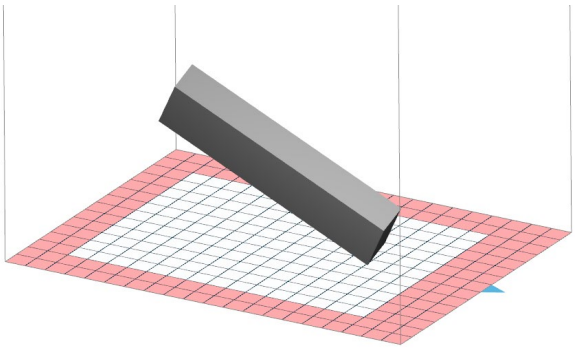


Fig. 3.2 ***Correct*** *arrangement of a flat model.*

3.2 Solid blocks and boxes

The main rule in arranging a printing of a dense model, as in the flat surfaces case, is to make the cross-section area as small as possible. In solid blocks and boxes, there is a significant accumulation of heat inside the block volume and local internal stress, which may deform the final product. The bending or curving of the block usually occurs at the corners.

3.2.1 Solid blocks

Solid blocks must be positioned so that no side exactly aligns with (is either parallel or perpendicular) the Print Beds walls. It is recommended to turn the model in all three axes, in the 15 to 85 degree range (45 degrees for each axis is optimal). Arranging the models at an angle diminishes the heat accumulation in the following layers.

With blocks with irregular angles or rounded surfaces, the rule of the smallest section surface possible also applies.

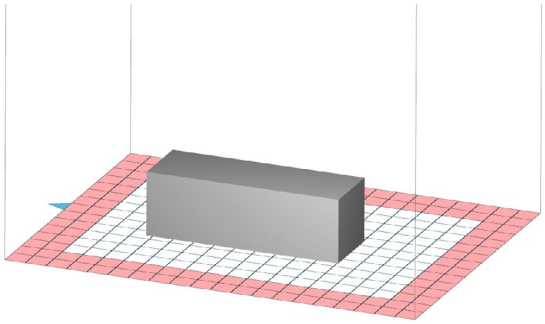


Fig. 3.3. ***Incorrect*** *arrangement of solid block.*

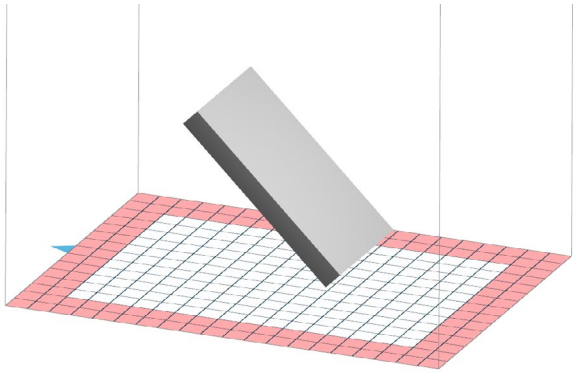


Fig. 3.4 ***Recommended*** *arrangement of the solid block..*

EXCEPTION:
For cylinders with smooth surfaces, you will get the best effect by printing them vertically, along the Z axis. However, it will not be a big mistake to arrange it at a 45 degree angle.

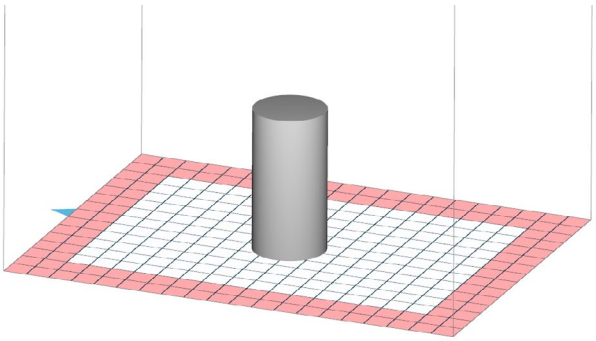


Fig. 3.5 ***Recommended*** *arrangement of the cylinder.*

3.2.2 Boxes

The arrangement recommendation for boxes and closed blocks is the same as for solid blocks. Additionally, be sure not to put such models, especially boxes, upside down and/or cover them with a lid if they come with one. Even if the model's sides are thin, the heat accumulated within the box may deform the print.

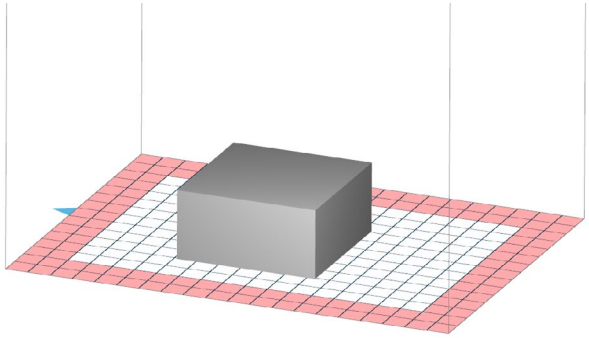


Fig. 3.6 ***Incorrect*** *arrangement of the box model.*

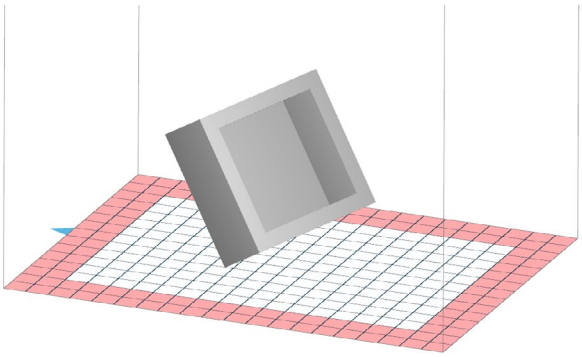


Fig. 3.7 ***Correct*** *arrangement of the box model*

3.3 Spheres, cylinders, pipe cylinders and other rounded objects

It is recommended to print cylinders and pipe cylinders with a smooth surface arranged vertically. However, sometimes this arrangement is not possible due to the size of the model. In such case you will have to rotate it (preferably at an angle of 45 degrees). If the rounded model has details you also need to rotate it.

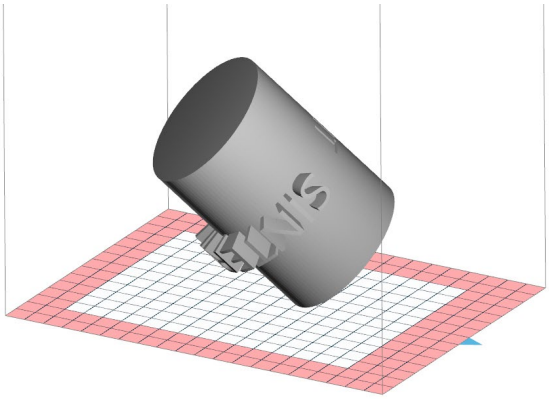


Fig. 3.8 ***Correct*** *arrangement of the cylinder with details.*

3.4 Sharp details vs. smooth edges

If the model has some detailing, please orient the detailed surface upward. The detailed surface will be sharp, while the bottom surface will be smoother.

3.4.1 Sharp details

If one of the surfaces contains detailed features and you want them well-visible, the model should be put in such a way that the detail is facing up. It is essential to keep the cross-section area as small as possible.

IMPORTANT
Flat models with sharp details should be arranged at 45 degrees at each axis, with the detail facing up. This angle will allow both correct printing of the flat surface and a defined and strong detail.

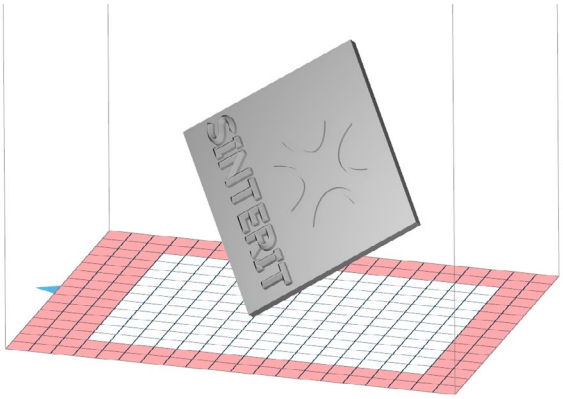


Fig. 3.9 Defined details, such as inscriptions, should be arranged face up.

3.4.2 Smooth edges

If you want to keep the detail smooth, arrange it upwards. Laying the part with the detail down will cause it to overrun.

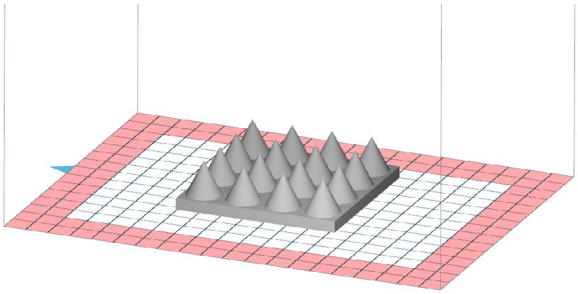


*Fig. 3.10 The **correct** positioning of the detail for a smooth finish.*

3.5 Openings and holes

If possible, any openings in the model should be laid flat (axes X and Y) and facing up (Fig. 3.11). Arranging them vertically may result in the opening shape change e.g. from round to oval and/or not retaining the intended size after printing.

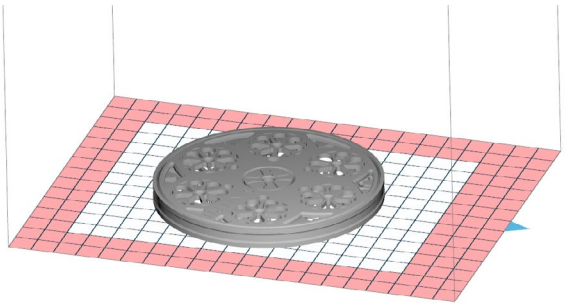


Fig. 3.11 ***Correct*** *arrangement of models with openings.*

In case there is no other way (the model is too big or the flat surfaces bend), the model with openings should be arranged at an angle in all three axes (Fig. 3.12). Please be aware that the round shapes may then be distorted.

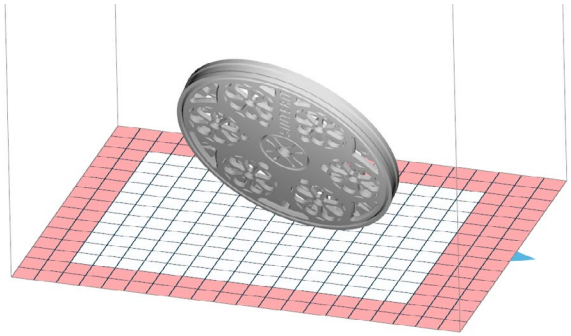
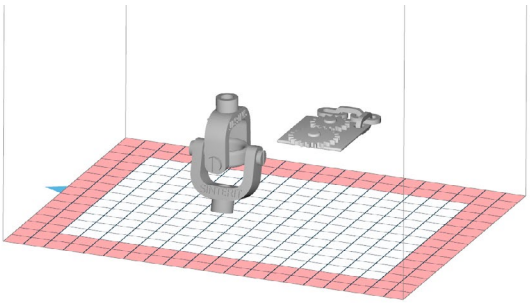


Fig. 3.12. Acceptable arrangement of models with openings.

3.6 Movable parts

If the model contains movable parts, please position it perpendicular/parallel to the printing chamber. This way, the joints will be the most accurate and if designed properly, the model should retain the intended articulation.



3.13 This arrangement should provide a movable model.

When the movable model is rotated, the joints would not be so accurate. This may make e.g the revolving joint immovable.

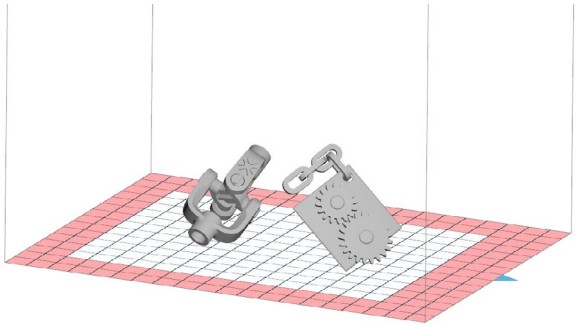


Fig. 3.14 ***Incorrect*** *arrangement, which may cause sticking of moving parts to surfaces.*

3.7 Temperature management

If you are printing more than one element at a time and they differ in height in the Z axis, the best practice is to arrange them flush with one another at the top. This will reduce the possibility of an ”orange peel” effect and eventual curving of the model.

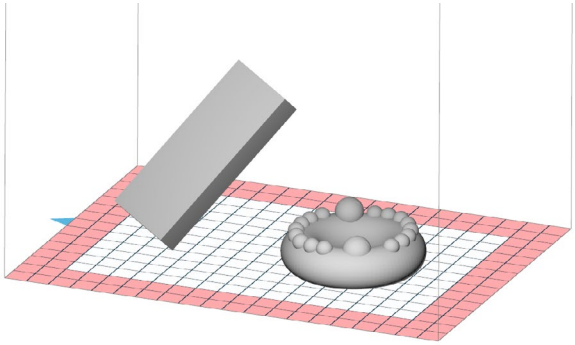


Fig. 3.15 ***Incorrect*** *arrangement. The possibility of defects.*

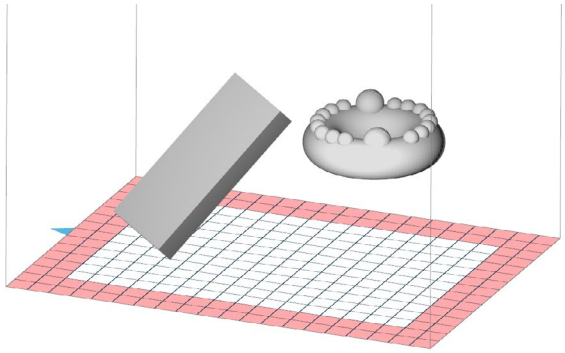


Fig. 3.16 ***Correct*** *positioning considering temperature management.*

3.8 Filling the build chamber

If you want to completely fill the printer work space, the first thing is to follow the instructions from the previous sections depending on the models used. However, it should be noticed that the number of models and their volume in the chamber will significantly influence the duration of the printing process. To fill the available space by placing more models vertically in the build chamber, keep the minimum distance between them at 3 [mm] so that the printouts do not stick together or warp.

When printing a large number of different models, it is recommended to print layers consisting of the same models. Printing different models on the same layer can cause some defects. However, if you don't mind small defects such as lines, you can mix models on layers.

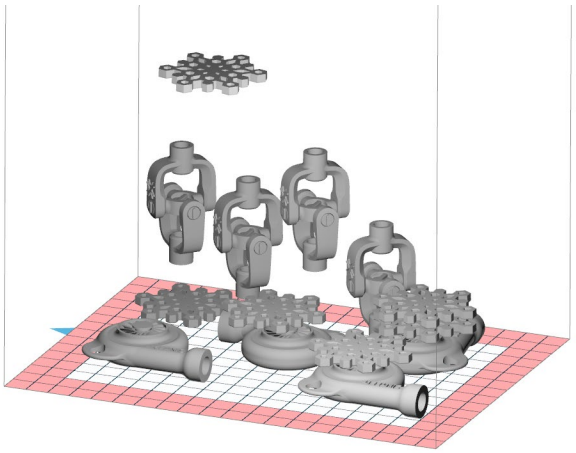


Fig. 3.17 ***Incorrect*** *arrangement of models in the print chamber.*

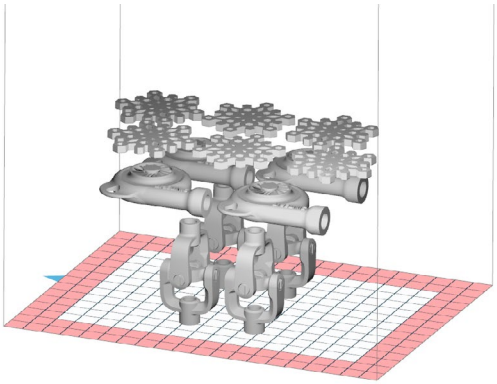


Fig. 3.18 ***Correct*** *arrangement of models in the print chamber.*

TIP

After the models are arranged, remember to always check if the objects don't collide with each other by using the CHECK COLLISIONS button.

3.9 Summary of positioning rules

- When arranging your prints, optimise the arrangement to follow as many of the above tips as possible.
- Models of different types printed on the same layer affect each other and cause small defects, e.g. lines, due to different exposure lengths of the layers. If you want to avoid such defects, try to stack only identical models on the same layers.
- Try to keep the layers similarly filled. If this is not possible, stack the longest layers higher, not at the bottom of the Print bed.
- You can skip some tips to reduce printing time or increase productivity, but this may result in lower quality.
- Finally, always make sure that the models don't collide with each other using the Show collisions function.
- If you have any concerns or questions about the arrangement of your print, contact Sinterit After-Sales: support@sinterit.com.

4. UPDATING THE SINTERIT PRINTERS USING SINTERIT STUDIO

It is possible to update Sinterit LISA/Lisa PRO/Lisa X/NILS 480 internal firmware so that it works with the latest available Sinterit Studio Software software. If you are not sure if you have the latest software version, you can check it by selecting **Help** - > **Check for update…**

To update the printer, follow these steps:

1. Select **Help** -> Update printer.
2. Choose the printer model you want to update (Fig. 4.1).
3. Insert the USB flash drive into the USB port on your computer, then click Create Update USB Drive. The process may take a few minutes(Fig. 4.1).

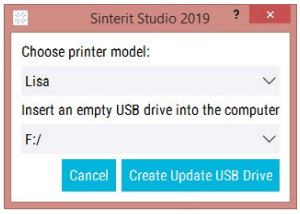


Fig. 4.1 Creating update files.

4. After copying the files a message will appear that you can remove the USB flash drive, then plug it into the USB port on the printer that is turned off. Turn on the printer and follow the instructions on the screen.

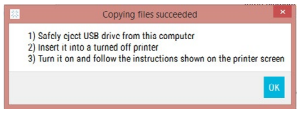


Fig. 4.2 Message after copying files.

5. UNLOCKING SINTERIT STUDIO ADVANCED

To have access to the expanded version of the software - Sinterit STUDIO ADVANCED - please contact our sales team. Once purchased, Sinterit STUDIO ADVANCED lets you work with the open parameters. To unlock new features in the software and on the printer:

1. Register your printer on our website www.sinterit.com/support/register-your-printer/.
2. You will receive a license key and activation files to the email address you provided.
3. In the Sinterit STUDIO Software select **Help**.
4. Choose **Enter product key**.
5. Enter your individual license code. The one you received in the email.
6. You should see new features (open parameters). You will find more information in the chapter: *2.2 Custom Material Parameters (open parameters).*
7. Save the file or files (depending on your printer) attached to the email to a flash drive.
8. Insert the USB flash drive into the USB port on the printer.
9. On the screen you will find the message, that an update has been detected.
10. Accept the update installation on the printer screen.
11. After a moment, you will see a message on the screen that you can reset the printer to complete the upgrade.
12. Turn off the printer at the power switch. Wait a few seconds and turn the printer back on.

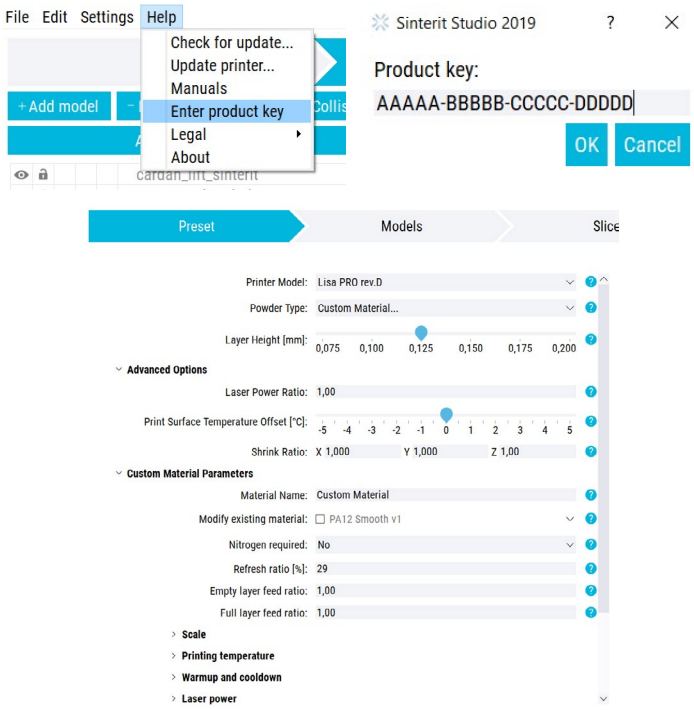


Fig. 5.1 Unlocking Sinterit STUDIO ADVANCED.

6. HARDWARE REQUIREMENTS

System requirements for Sinterit STUDIO Software

- 64-bit processor,
- Windows 7 or higher,
- Minimum 1 GB of disk space,
- Minimum 2 GB of RAM,
- Graphics adapter compatible with OpenGL 3.0 or higher.

7. TECHNICAL SUPPORT

If you have any questions or doubts, please contact our After-sales department.

- e-mail: support@sinterit.com
- phone: +48 570 702 886

For a list of distributors and technical support in each country, please visit our website www.sinterit.com

8. GENERAL LEGAL INFORMATION

Where this manual refers to Sinterit or the Company or “us/our”, this means Sinterit sp. z o.o. with its legal seat in Krakow, registered by the District Court for Kraków-Śródmieście in Krakow, XI Commercial Division of the National Court Register under number: 535095, NIP (tax number): 6793106416, with the share capital for date of publication of this manual of PLN 106,600 (say: one hundred six thousand and six hundred).

This document contains material protected under copyright and industrial property laws. In particular, this means that the document may not be reproduced or modified without the consent of Sinterit.

This manual serves to assist you in the correct use of the device, perform basic maintenance, and, if necessary, solve simple problems, allowing you to maintain the device in good condition.

This manual contains content exclusively for the provision of information and for use by individuals who have been professionally trained in the operation and maintenance of the equipment described below.

The information contained in this document is intended for use only with the product made by Sinterit and called Sinterit STUDIO and Sinterit STUDIO ADVANCED software.

Due to the constant development of Sinterit products the information contained in this manual as well as any specifications and markings issued or placed on Sinterit products by the Company are subject to change without notice.

9. DISCLAIMER

Sinterit is not responsible for any use of this information about other products.

Although every effort has been taken to provide accurate information about the product, Sinterit disclaims, to the widest extent permitted under the applicable law, any and all liability for any incorrect information or omission, and for anything that may result from such errors or omissions. Sinterit reserves the right to correct any and all errors and omissions at any time.

Further limitations or exclusions of Sinterit liability may result from the applicable laws or agreements entered into with the buyer of the products.

10. TRADEMARKS

The Sinterit logo is a registered trademark of the Company.

11. SOFTWARE LICENSE AGREEMENT

Sinterit grants the buyer a non-transferable license without a right to sublicensing to use Sinterit STUDIO Software under the terms and conditions set forth in agreement between the buyer of the given Sinterit 3D Printer and the Company.

SINTERIT Sp. z o.o.
ul. Nad Drwina 10/B-3, 30-741 Krakow, Poland
www.sinterit.com