Trainers for Visually Impaired Students Introduce 3D Printing

Tutorial Module 3

Introduction to FDM 3D Slicer Software

Tutorial for the T4VIS-In3D trainer course

Published by the
T4VIS-In3D project consortium

Co-funded by the
Erasmus+ Programme
of the European Union

The project “T4VIS-In3D” was co-financed by the “ERASMUS+” Programme of the European Commission

This Tutorial is published by the T4VIS-IN3D project consortium.

Printed:
June 2021 by Berufsförderungswerk Düren gGmbH

The T4VIS-In3D Project Consortium:

Berufsförderungswerk Düren gGmbH (Project co-ordination)
Karl-Arnold-Str. 132-134, D52349 Düren, Germany, http://www.bfw-dueren.de

Fundacion Aspaym Castilla Y Leon
C/ Severo Ochoa 33, Las Piedras 000, 47130, Simancas Valladolid, Spain, https://www.aspaymcyl.org/

Hilfsgemeinschaft der Blinden und Sehschwachen Osterreichs
Jägerstrasse 36, 1200 Wien, Austria, https://www.hilfsgemeinschaft.at/

Instituttet for Blinde og Svagsynede, IBOS
Rymarksvej 1, 2900 Hellerup, Denmark, https://www.ibos.dk

Istituto Regionale Rittmeyer per i ciechi di Trieste
Viale Miramare 119, 34136 Trieste, Italy, http://www.istitutorittmeyer.it/

NRCB
24 Landos Str., Plovdiv, 4006, P. Box 11, Bulgaria, http://www.rehcenter.org

Content

1 General

Slicer software refers to all programmes that enable the conversion of object files into machine-specific instructions for the printer. The slicer software thus generates the necessary machine code for specific printers, which includes all instructions for moving the extruder, the printing plate as well as the temperature control. This information is generated layer by layer for the model. In the layered view, the model is displayed in slices. Hence, the name of this software.

Usually, every 3D printer is delivered with a slicer software that has already been optimally adjusted to the respective device by the manufacturer.

Nevertheless, it is necessary to learn how to use this software in order to achieve optimal printing results.

Due to the large number of slicer software on the market and their short update cycles, we will mainly focus on the functions of these software types that are particularly relevant for the creation of tactile models.

In general, there are freely available and proprietary slicer software. In the case of freely available slicers, a distinction must be made between open source slicers and freely traded slicers, which are being offered for download free of charge by manufacturers of 3D printers. These slicers optimally support the manufacturer's machines, but can also be adapted to 3D printers from other manufacturers. The most popular representative of this type is “Cura” by the Dutch company Ultimaker. This is the reason why Cura will be presented as an example in the following.

Other FDM slicers from different manufacturers include, for example:

1. Prusa Slicer https://github.com/prusa3d/Slic3r/releases
2. IdeaMaker https://www.raise3d.com/ideamaker/

Other freely available FDM slicers are e.g.:

1. Slic3r http://slic3r.org/
2. Craftware https://craftunique.com/craftware/
3. Repetier Host https://www.repetier.com/download-now/

Probably the most popular proprietary FDM slicer is:

Simplify3D https://www.simplify3d.com/

FDM slicers create a text-based file based on the G-code used to control CNC machines. This file contains all control commands and is related to the device

parameters of the 3D printer. Thus, a file created by a slicer can only be used for the same type of printer.

The following section shows an introductory part of a G-code file. Please note that the text after the semicolon is a comment explaining the respective command.

```
;Generated with Cura_SteamEngine
4.9.1
M104 S210
M105
M109 S210
M82 ;absolute extrusion mode
;Sliced at: Tue 15-06-2021 12:23:24
G21 ;metric values
G90 ;absolute positioning
M82 ;set extruder to absolute mode
M107 ;start with the fan off
G1 Z5.0 F1800 ;move Z to 5mm
G28 X0 Y0 F1800 ;move X/Y to min
endstops
G28 Z0 ;move Z to min endstop
G92 E0 ;zero the extruded length
```

1.1 Differences among FDM slicers

All slicers vary in their graphical user interface and their handling. Some also require a higher level of user knowledge to be able to use the software to its best advantage and thereby achieve optimum results.
Slicers often use different names for identical functions. The decisive factor, however, is the algorithm on which the creation of the G-code is based. The most widely used algorithm at present is probably that of Slic3r.

The GUI of the slicer Cura (version 4.9)

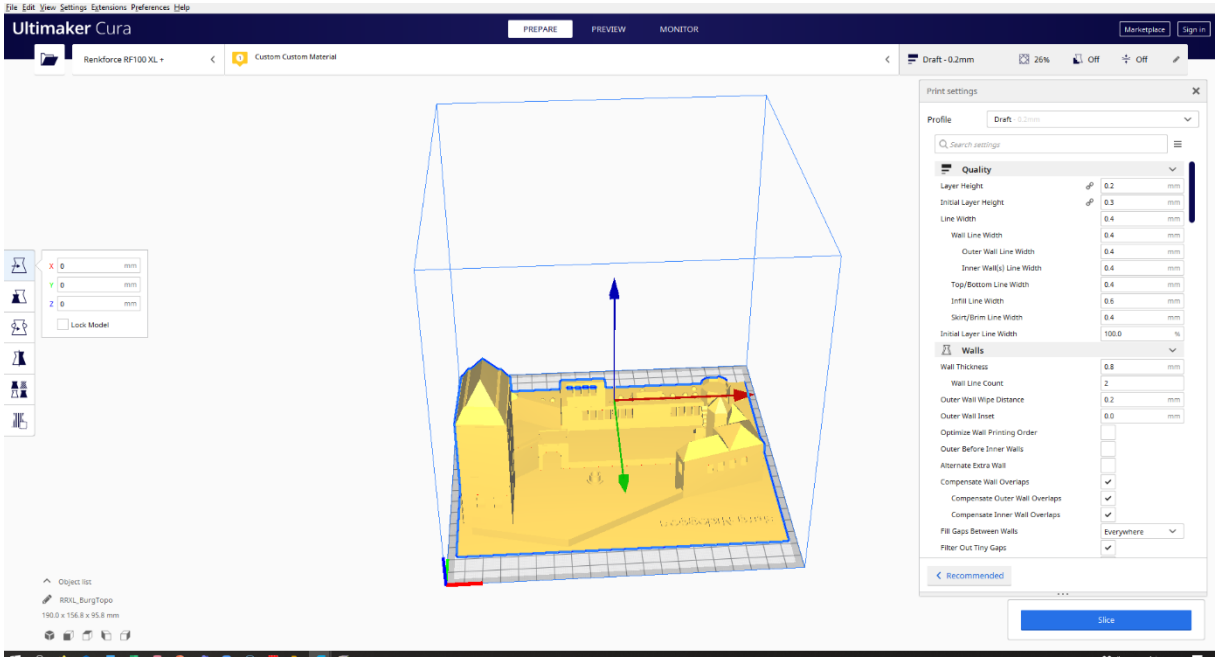

Figure 1 Ultimaker Cura

The GUI of Repetier Host FDM Slicer for comparison.

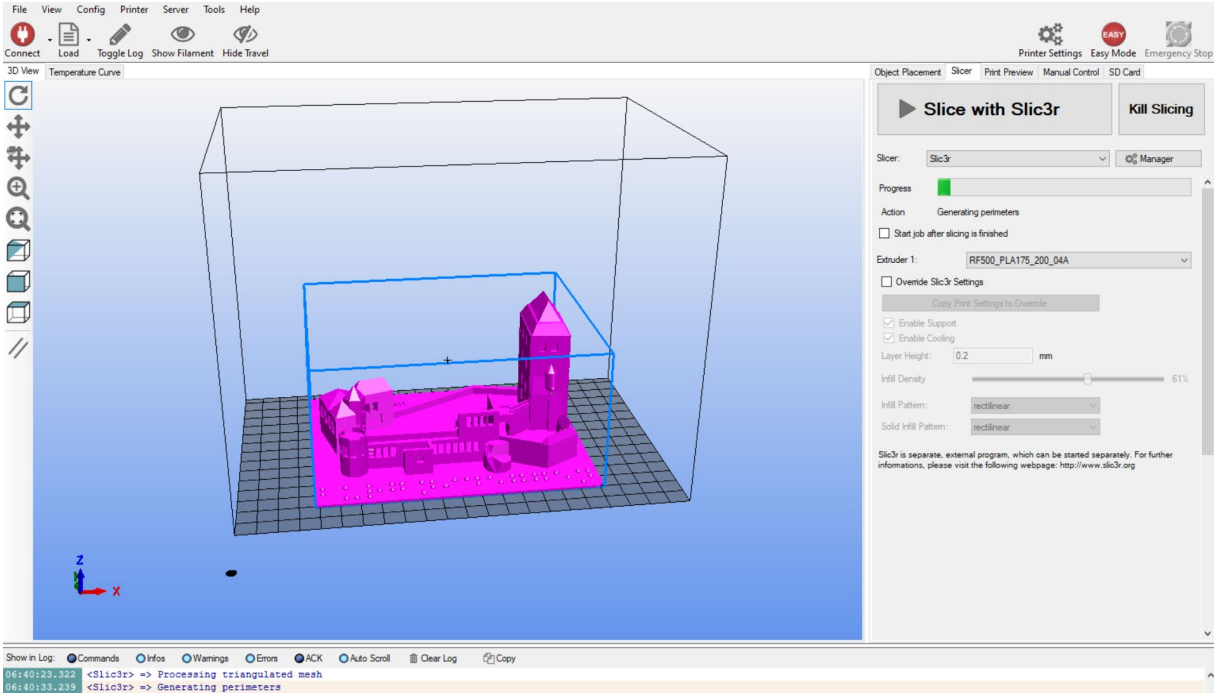

Figure 2 Repetier Host

1.2 Common features of FDM slicers

Although the various slicers have their own GUI, they usually have the following functions in common:

1. Building space mode to view the volume model and (after slicing) the layer model.
2. Setting modes to define parameters for:
   - Extruder and build plate temperature
   - Material extrusion
   - Wall thicknesses
   - Filling/ internal density (Infill)
   - Support material (Support)
   - Extrusion speed
   - Flow rate of the filament
   - Shape and nature of the build plate adhesion
3. Manipulation options of the model, such as:
   - Moving the model
   - Scaling of the model
   - Rotating the model
   - Duplicating the model

Usually only experienced users work with the G-code. An overview of the commands can be found at: https://duet3d.dozuki.com/Wiki/Gcode

2 Operation of an FDM slicer using the example of Cura

Since Cura is a widely used slicer that comes with 3D printers from many manufacturers, we will explain the functions of this type of software using the Cura slicer as an example.

The following image depicts the graphical user interface (GUI) of the Ultimaker Cura FDM slicer and various tool areas for adjusting the print result.

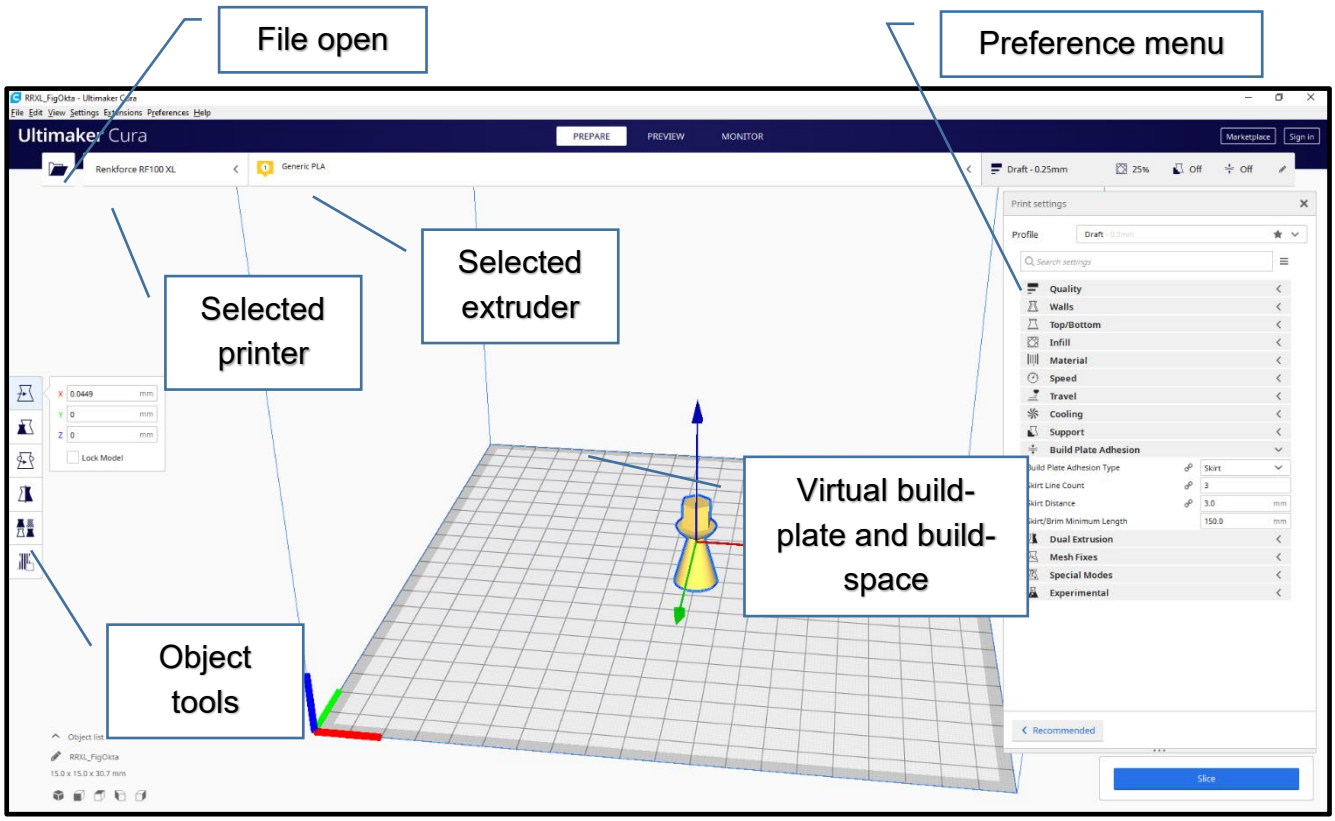

Figure 3 The GUI of the slicer Cura (version 4.9)

The view of the building area can be altered by clicking the right mouse button and using the scroll wheel.

By moving the mouse pointer into the view area of the building area and pressing the right mouse button, the building area can be rotated. By moving the mouse vertically, the building area can be turned upwards or downwards.

2.1 Loading objects and correct positioning

Cura automatically positions the model in the centre of the building room. In case no model is shown in the construction space after the loading process, this may be due to different measuring systems: If a model was created in the imperial measurement system, it cannot be displayed in Cura in the metric system; hence, there is a conflict.

By clicking on the model with the left mouse button, the object toolbar opens on the left side of the screen showing the following functions:

Move object

Scale object

Rotate object

Mirror object

Model settings

Support blocker

The last two options are primarily required by experienced users.

2.1.1 The Move function

When using the "Move" function, a text input field opens up and three arrows along the X (red), Y (green) and Z (blue) axes are displayed on the object.

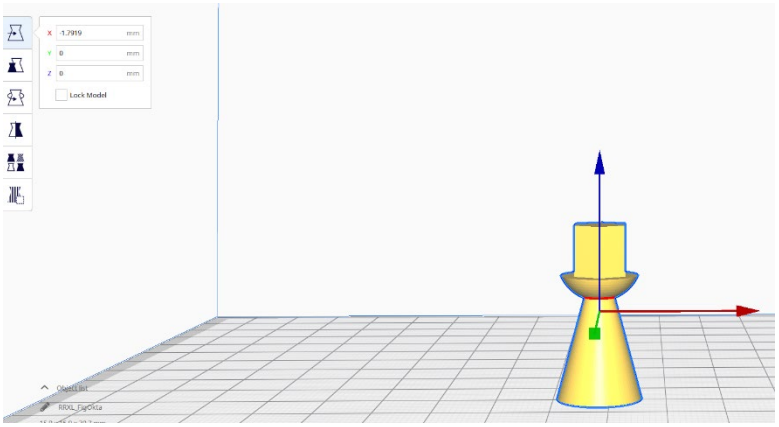

Figure 4 Move function

This function can also be executed by the key command “t”. Shifting the object can now also be done either by entering the values in the input field or by moving the mouse. To do this, the relevant axis arrow must first be clicked with the left mouse button and then moved by moving the mouse.

2.1.2 The Scale function

The function "Scale" allows resizing the model. This function can also be accessed by the key command "s".

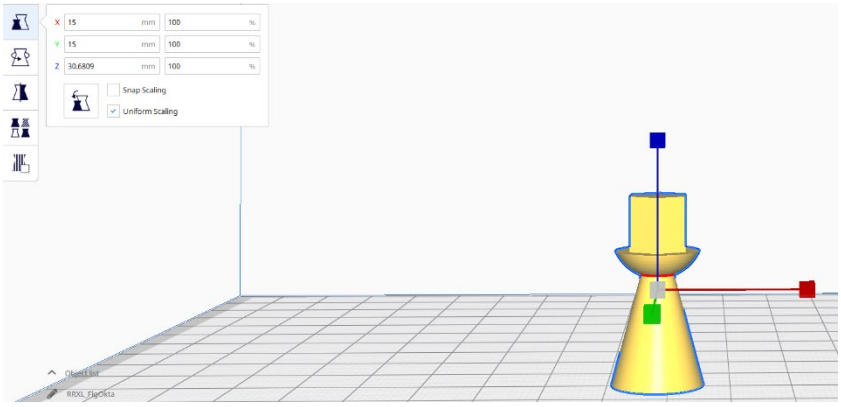

Figure 5 The scale function

Again, it is possible to alter the size using the mouse by moving the relevant axis arrow while holding down the left mouse button.
A more accurate way to scale the model is to enter the values in the input field. This can be done either by entering the percentage value or the absolute size. By default, a symmetrical size change is set. This means that when entering a percentage value

or size, the other two values are changed proportionally. If this is not desired, the "Uniform Scaling" check box must be deactivated.

2.1.3 The Rotate function

The rotate function can also be executed with the key command "r". This function allows the object to be rotated along all three axes. This function can only be performed by moving the mouse along an axis marker with the left mouse button pressed. During the rotation, the degree of angle reached is shown.

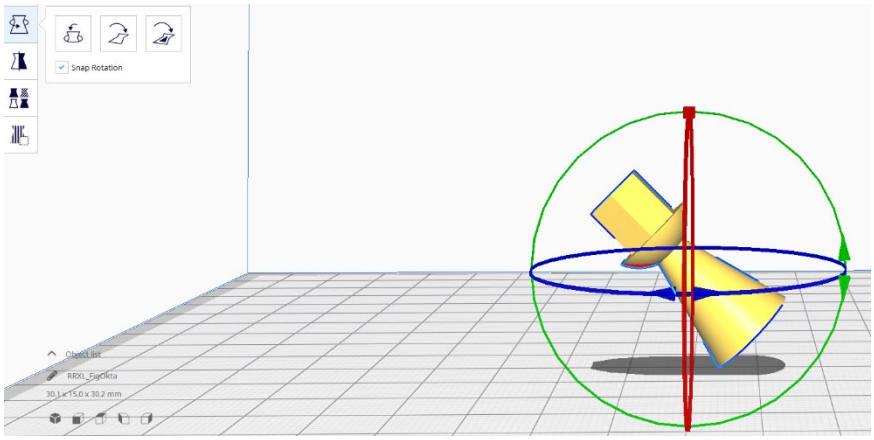

Figure 6 The rotate function

2.1.4 The Mirror function

This function allows models to be mirrored along an axis. This function can also be activated via the key command "m".

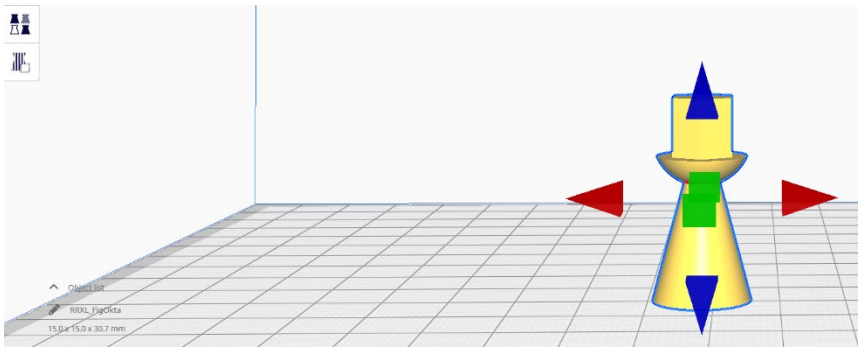

Figure 7 The mirror function

The mirroring process can be executed in one direction by clicking the left mouse button of the relevant axis arrow.

This function is particularly interesting to create mirrored counterparts of models. It is for example very easy to produce a right-hand rear-view mirror for the left-hand side.

2.1.5 Positioning of oversized components

As soon as components are loaded that are larger than the building area, they are positioned outside of it. They also have a coarse shading.

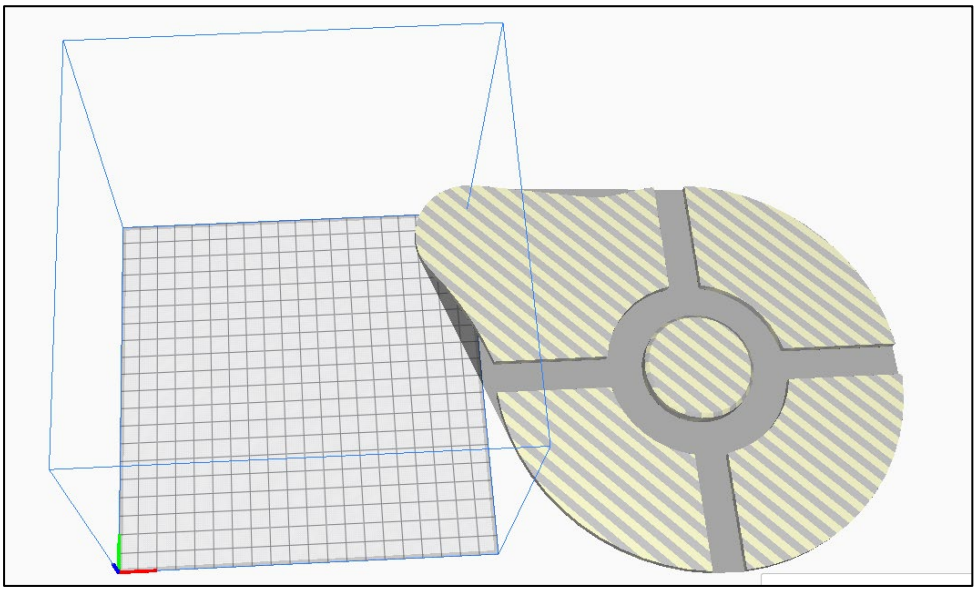

Figure 8 Positioning of oversized components

In order to be able to process these components, they must first be scaled down with the Scale function and then moved with the Move function. As soon as the model has been changed to the correct size and position, the hatching changes to a constant yellow colour.

2.1.6 The Context menu

The context menu is available as soon as the model has been clicked with the right mouse button. The context menu allows the following settings and functions (from top to bottom):

1. Centring of the component on the building platform. This function is also suitable for the optimal positioning of scaled-down models.
2. Delete the selected model;
3. Duplicate the selected model;
4. Select all models+
5. Arrange the models on the building platform. This allows for arranging as many components as possible.
6. delete all models from the building platform;
7. Reload all models;
8. Position all models at the former position;
9. Reset all model changes to the initial state;

10. Group models (For this, the models concerned must be selected, e.g. via "Select all models")Unite models. Here Cura tries to arrange the models on top of each other.
11. Ungrouping grouped selection

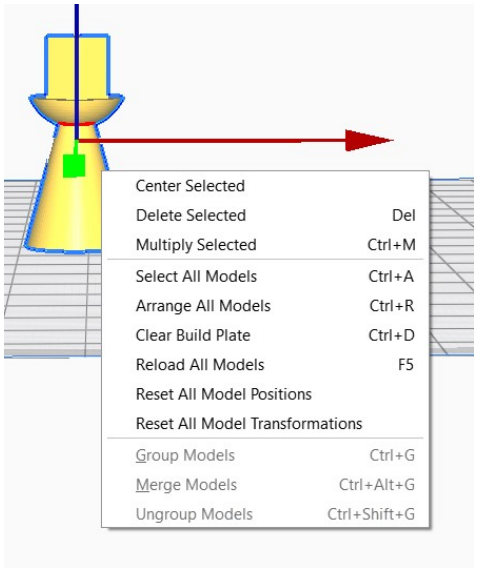

Figure 9 The context menue

2.2 Slicing a component

To slice a model, the following settings are needed:
1. Layer height
2. Infill density
3. Printing temperature of the nozzle and, if necessary, of the build plate
4. Necessary support structure
5. Adhesion of the build plate

To make these settings in Cura, you should use the print settings in the upper right window.

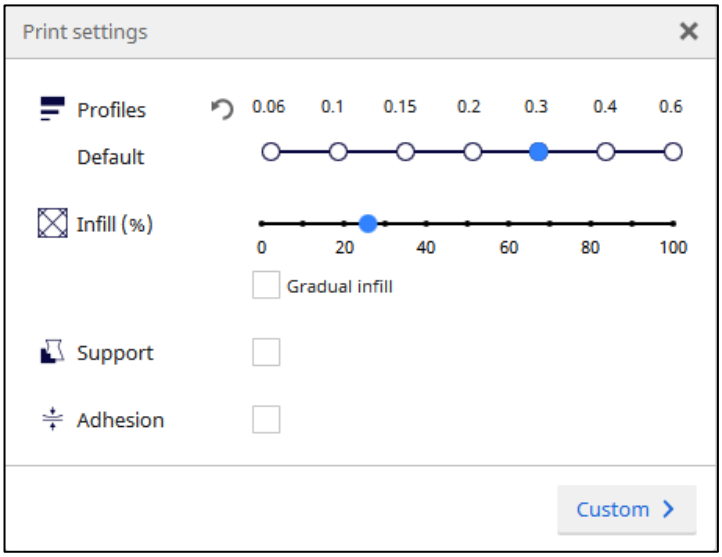

Figure 10 Default Setup

If this window is not visible, one of the icons in the upper right-hand side of the screen should be activated.

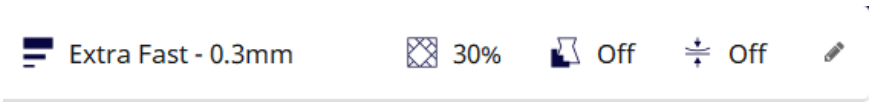

Figure 11 Setup toolbar

Since the setting options in the standard mode (Figure 10) are very limited, activate the "Custom" button to see all settings areas.

First, select the functions “Layer height” and “Layer width” from the menu.

1. Layer height. Here you select a height that is less than the nozzle thickness (usually 0.4mm). The more precise and fine the model quality should be, the lower the layer height. However, keep in mind that the printing process takes correspondingly longer. Make sure to set values that the printer is capable of. As a rule, the lowest height is 0.05mm. For braille printing, the layer height

should be at least 0.25mm, otherwise the top of the dots will be too sharp. The line width influences the smoothness of the surface. The smoother it should be, the smaller the width. Here, too, the device parameters of the 3D printer set the possible limits. In principle, however, the layer width cannot be smaller than the layer height.

2. The infill of the model is set to 15% by default. With large surfaces and pressure forces affecting them, this may be too low. A setting between 25% and 40% is sufficient in most cases to obtain a stable filling for tactile models.
3. The correct printing temperatures can be found in the data sheets of the filaments used.
4. Support structures depend on the structure and placement of the model on the build plate. Here, the 45° rule is to be applied as a simple guideline. As soon as the overhang corresponds to an angle greater than 45° to the build plate, support structures are required. This can be well illustrated by the example of the letters Y (45°) and T (90°). Support can be printed with normal filament or with water-soluble materials. However, 3D printers with two extruders are required for this.

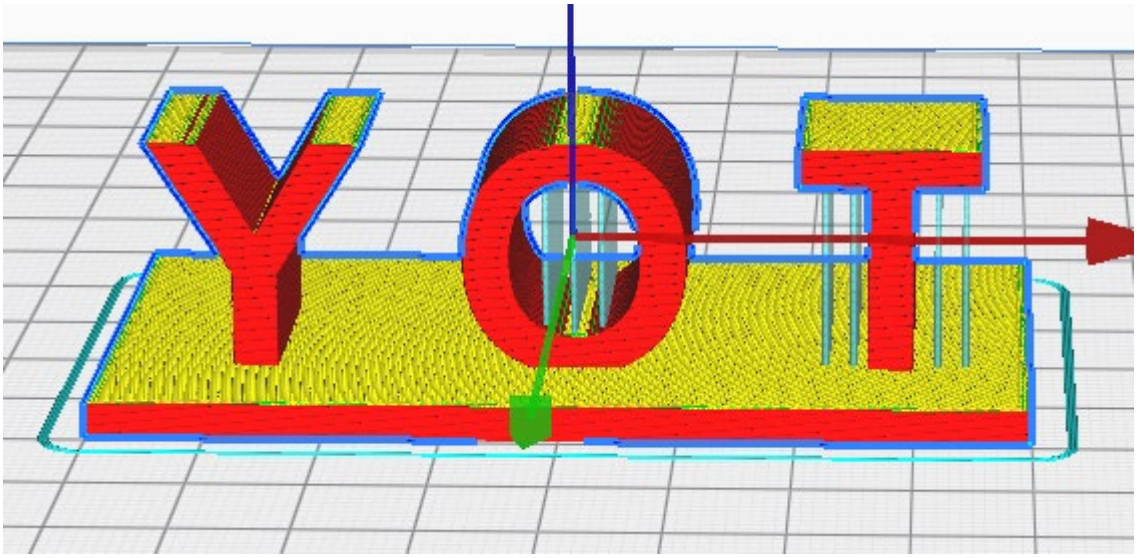

Figure 12 Support example 45° and 90° overhang

5. For a successful FDM print, it is extremely important that the first layer adheres to the build plate until the printing process is finished. If the component separates in places, the material will inevitably warp on the bottom of the workpiece. In extreme cases, the component becomes loose and is torn from the build plate by the extruder movement.

   Three options are offered for buildplate adhesion:
   a) Skirt
   b) Brim
   c) Raft

The Skirt option is not, technically speaking, a function that improves build plate adhesion. This function applies a border of filament before printing an object. The purpose of this function is to ensure that there is enough filament in the extruder nozzle when the object is printed.

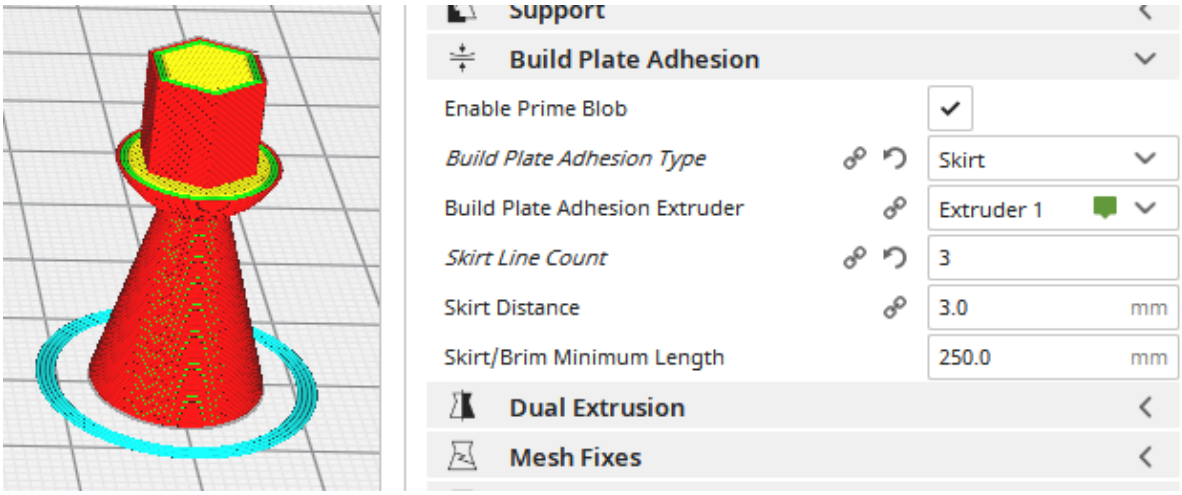

Figure 13 Skirt

The Brim function draws a thin single-layer border around the component. The adhesion surface is increased and the adhesive effect on the building board is improved. After printing, the brim can easily be removed from the component.

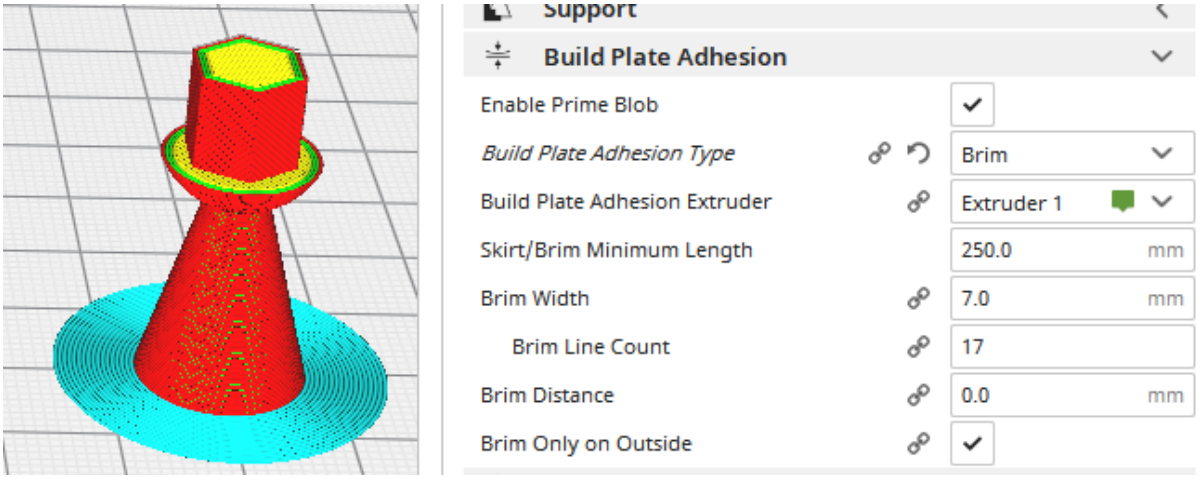

Figure 14 Brim

With the raft, a grid-like base is first applied underneath the component. This is particularly useful when a component is small or has a small contact surface or the height of the component is disproportionate to the contact surface. The raft increases this contact area and bonds the component to the raft grid. The raft can also be easily removed from the finished component.

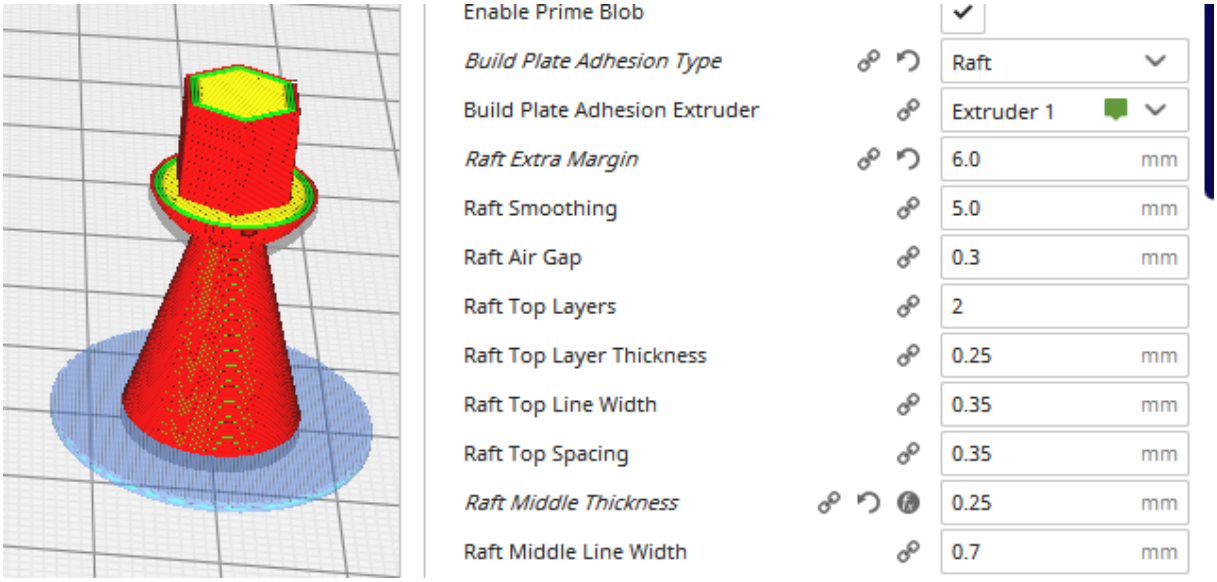

Figure 15 Raft

2.3 Evaluating the slicing process

One of the most common mistakes in FDM printing is made before the 3D printer starts printing. The result of the slicing process is not checked.

The most common mistakes made are:

1. Selecting the wrong printer or parameters such as temperature or speed.
2. The component is not positioned (evenly) on the build plate.
3. Support structures are missing or have not been added sufficiently.
4. The components do not have the correct infill.
5. No or wrong build plate adhesion is used.
6. The surface of the component is not closed.

After activating the "Slice" function by clicking on the respective button, the calculation process is started. Once the slicing process is finished, the following window becomes appears:

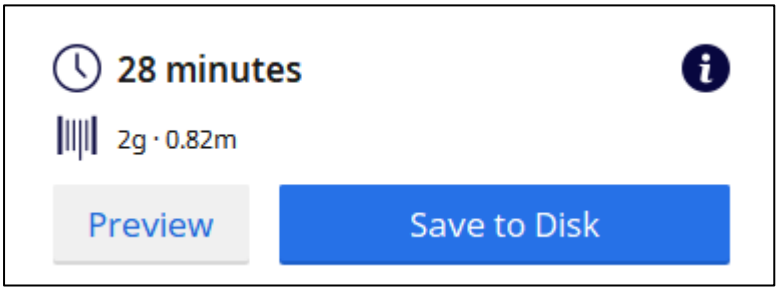

Figure 16 Post process- window

In addition to the estimated printing time and material consumption, the pop-up field also comprises a preview button as well as a button for saving the G-code file. The Preview button provides a layer-by-layer preview of the object to be printed as it has been generated in the G-code for the FDM printer.

The slide bar on the right side of the preview allows for a layer-by-layer view of the sliced component. If the component is not completely displayed on the build plate in the first layer, it has been positioned incorrectly or is uneven on the bottom side. In this case, you should think about inserting a raft and support.

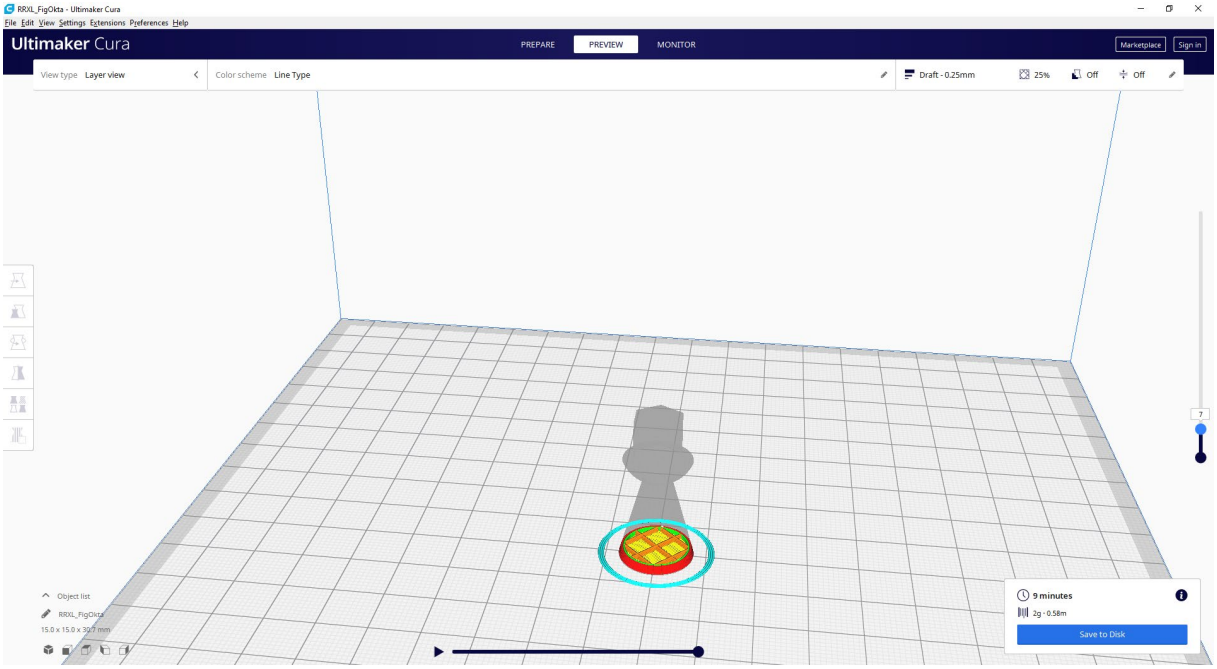

Figure 17 Print preview

If the support and build plate adhesion have been set, it is possible to determine whether they have been generated correctly and sufficiently. In addition, it is possible to detect at an early stage whether the wall thicknesses and surfaces have been sufficiently dimensioned.

3 List of figures