Trainers for Visually Impaired Students Introduce 3D Printing

Tutorial Module 6
Introduction SLA Slicing 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:

October 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 SLA Slicers

SLA slicers accomplish the same tasks as FDM slicers. They transform a polygon body from an STL file into a layer model that creates a print file according to the device parameters of the 3D printer used.

However, the specificity of SLA/DLP printing with liquid photopolymer requires different settings regarding device hardware, support structures and materials.

Some manufacturers equip their devices with their own slicer. Other manufacturers use slicers from other manufacturers. Some of these slicers have a free version and a paid version with an extended range of functions.
Well-known SLA slicers are, for example:

Name	Link	Free Version	Paid Version
Chitubox	https://www.chitubox.com/en/index	X	X
Lychee	https://mango3d.io/	X	X
PrusaSlicer	https://www.prusa3d.de/prusaslicer/	X	
Formware	https://www.formware.co/slicer/download		X

In this tutorial we explain how to use the free version of ChiTuBox. This slicer is supported by many well-known manufacturers and available for Windows and MacOS. The profiles of the device parameters are easily retrievable, so that a printer can be easily installed. In addition, the parameters for the materials are also provided. In order to download the free Chitubox version from the homepage, you first have to register as a user.

With the download you receive an installation file for the selected operating system. The following chapters explain the version for the Microsoft Windows® operating system.
An online version of the manual from the developer can be found at:
https://manual.chitubox.com/user-manual-basic/introduction

After installing and opening the programme, you will see a display of the programme version, which you can close by pressing the "X" button.

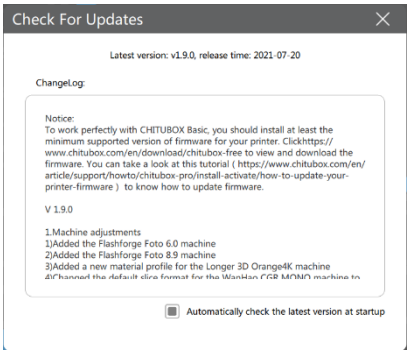

Figure 1 Version reference

2 Chitubox Graphical User Interface (GUI)

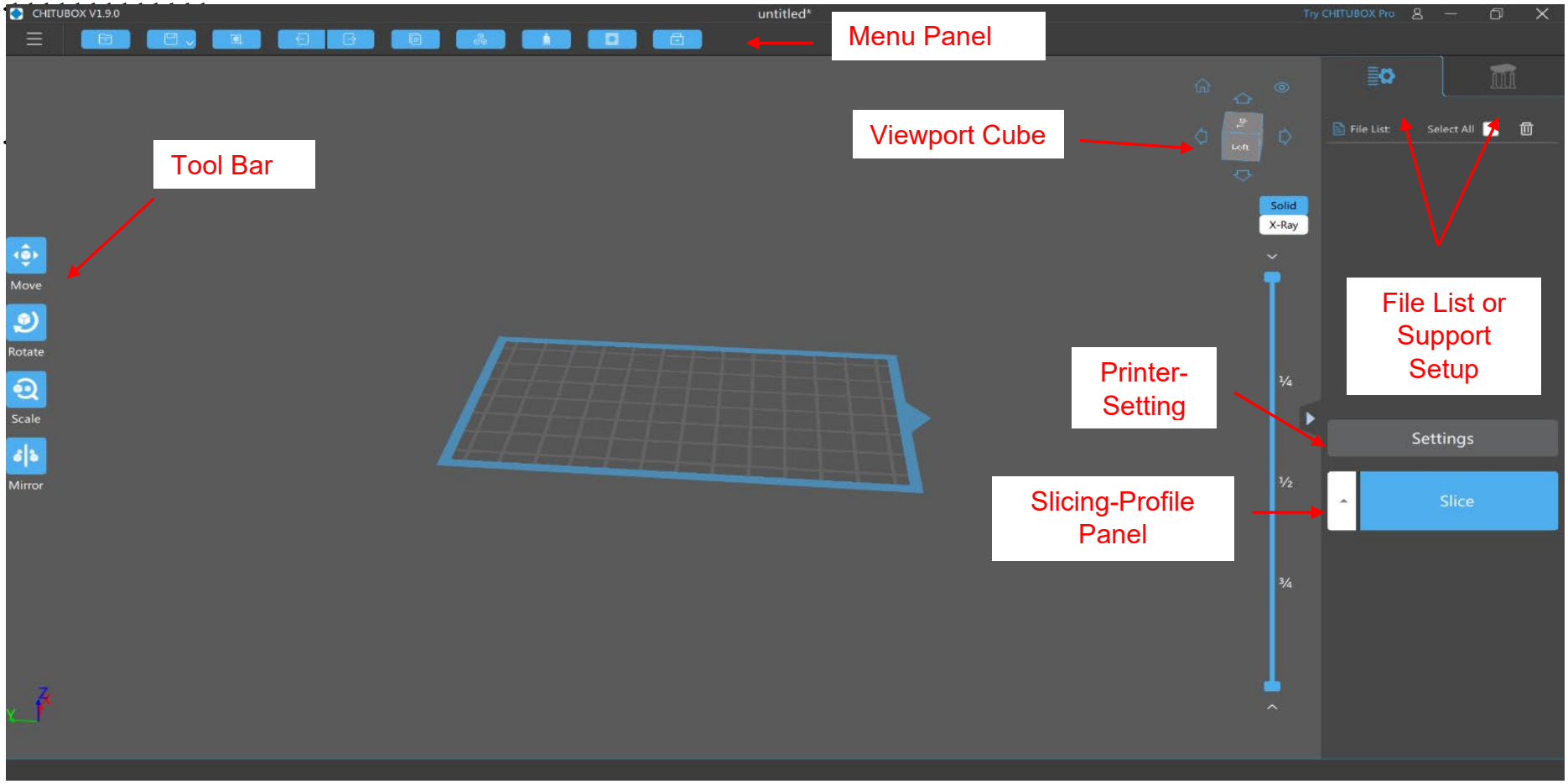

Figure 2 Chitubox User Interface

Chitubox is only available in English and Chinese.

If you compare Chitubox with Cura, the user interfaces initially look very different. However, the functions are also arranged in symbols and menu items.

As in Cura, the editing area forms the largest section of the image. Here, the image section can be zoomed in and out with the scroll key of the mouse. By keeping the left mouse button pressed, the image section can be shifted in all three axes.

2.1 Setup of the printer

When Chitubox is started for the first time, a printer must be added. To do this, select the "Settings" button with the left mouse button. The dialogue window for the settings opens. If no printer is visible in the left-hand area, it must be added. To do this, select the button  to choose the relevant printer from the list.

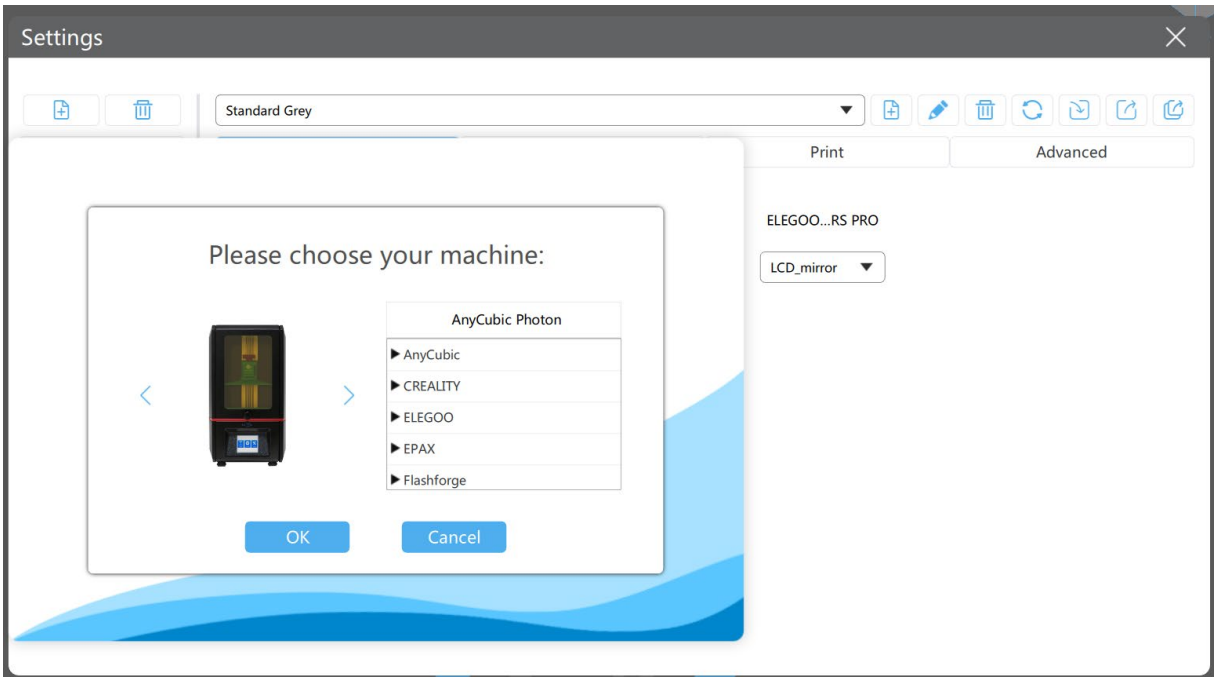

Figure 3 Selection of 3D Printer

After selecting the printer, all parameters are transferred to the settings. In the window, the areas "Machine", "Resin", "Print" and "Advanced" are available.

In the "Machine" area, the parameters of the printer such as the size of the building plate are saved. In the "Resin" area, the material designation and costs are entered. The "Print" area is particularly important. Here you enter the exposure times for the selected resin as well as the movement speeds of the build plate. This data is taken from the corresponding data sheets of the printer manufacturer.

Printing errors, such as insufficient adhesion of the exposed material to the building plate or cracks in the print, can be caused by incorrect parameters entered in this area.

2.2 Menu Panel

The Menu Panel contains the following features:

	Open File
	Save Project
	Screenshot and Video record
	Revoke (left), Redo (right)
	Duplicate
	Auto Layout
	Hollow
	Dig Hole
	Repair Panel

2.3 Tool Bar

The functions of the toolbar can be used to determine the positioning of the work piece on the building plate. By clicking the button with the left mouse button, the functions can be selected.
Moving the parts will be executed with the function “Move”. The part can be moved either with the mouse by holding down the left mouse button and moving the part. Alternatively, you can move the part by entering the values in the text fields of the X, Y or Z axis. By activating the "Put on the Plate" button, the component is placed directly on the building plate. The selecting the "Centered" function, the component is centred on the building plate. By activating the "Reset" button, all displacements are undone.

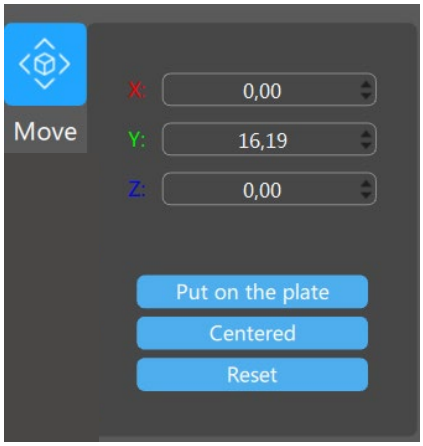

Figure 4 “Move” Function

Rotating the part along all three axes is done with the "Rotate" function. Activating the button opens a dialogue window. As with the "Move" function, it is possible to rotate the object along the three axes using the text field or the mouse. For rotations with the mouse, the respective axis circle must be clicked. By keeping the left mouse button pressed, the component can then be rotated along the selected axis.

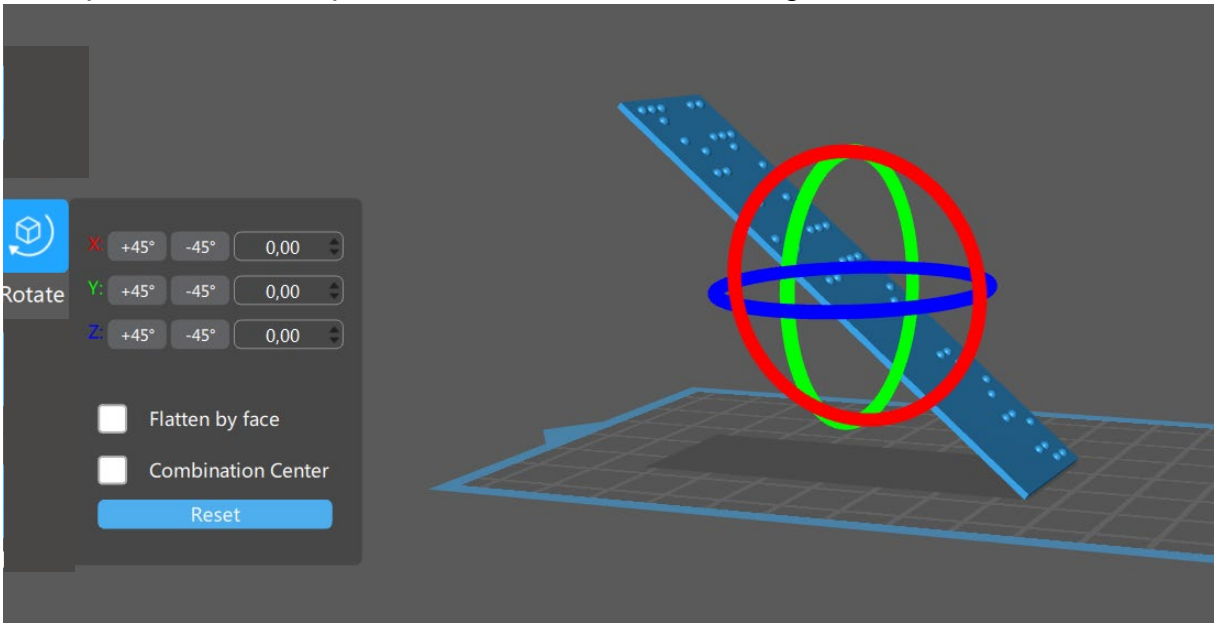

Figure 5 “Rotate” Function

Size adjustments of the component are made with the "Scale" function. With this function, too, the size changes can be made on the model either with the mouse or in the dialogue window by entering the desired values. The values can be entered either as absolute values in millimetres or as percentages. In the standard setting, size changes are carried out symmetrically on all three axes. This means that the component is changed proportionally on all axes. If you do not want to change the size proportionally, you must deactivate the 'Lock Ratio' check box.
To use the maximum size of the printer, select the "Scale to fit" button.

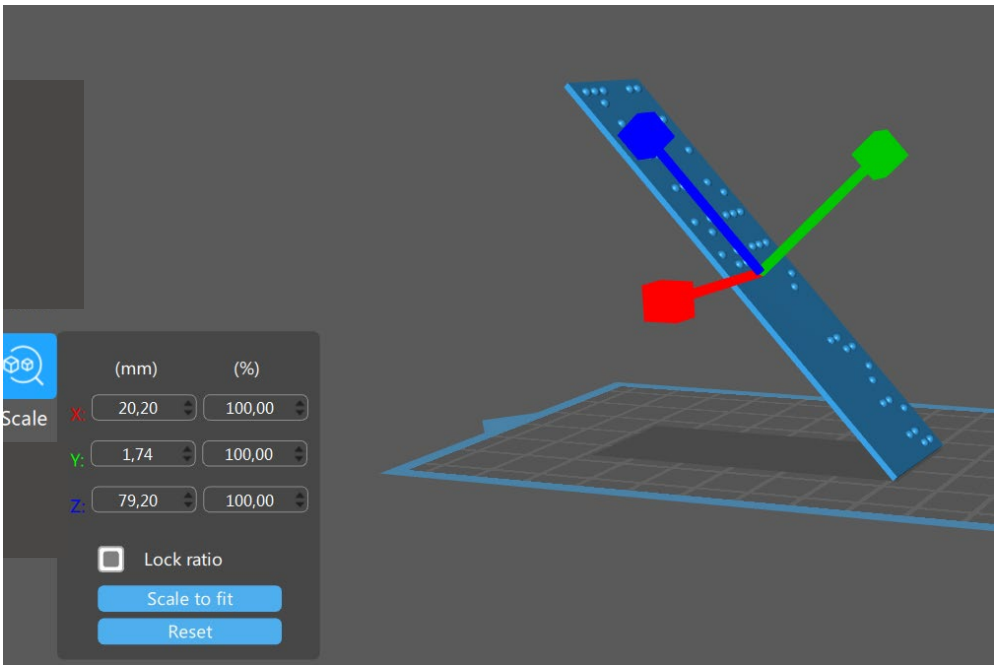

Figure 6 "Scale" Function

With the "Mirror" function, the component can be reversed.

The remaining elements of the user interface are explained using an example. For this, we take the Braille plate from Tutorial 5.

3 Positioning and slicing of components

With FDM printing, we have learned that components are positioned as flat as possible on the building plate and that the positioning requires as little support material as possible.

In SLA printing, this is not possible due to the process.
When aligning a component in SLA printing, the cross-sectional area of the Z-axis, i.e. the contact area on the building plate, must be minimised. The suction forces acting on the transparent bottom surface of the resin tank must be minimised. This prevents the component from detaching from the building plate and sticking to the transparent film or bottom plate of the resin container. Even if this is not the case, the suction forces create tension in the model. Flat printed components therefore warp after drying and curing, as soon as the thickness exceeds 0.5 mm. This convexity cannot be corrected later. For this reason, components are printed at an angle to the building plate.

Figure 7 Warping of flat printed components

In the SLA process, the amount of time and material that is needed for the manufacturing process depends on the height of the component. Therefore, many users are tempted, despite the disadvantages, to position the components as flat as possible and directly on the building plate in order to save resin and enable a short printing time.

This can be demonstrated very clearly in the example shown in Figure 5. If the plate with the braille letters is positioned flat and directly on the building plate, printing with an Elegoo Mars Pro takes just 11 minutes and requires only 30 ml of resin. However, if the component is positioned as shown in the illustration and printed with the required support material, the manufacturing process takes 6.5 hours and requires 85 ml of resin. In that case, the part is easier to remove from the building plate and warping after drying can be avoided. However, removing the support material also requires additional time for finishing.

3.1 Correct positioning

We want to print the Braille test board from Figure 6. To do this, we will first rotate the board by 50° using the "Rotate" function.

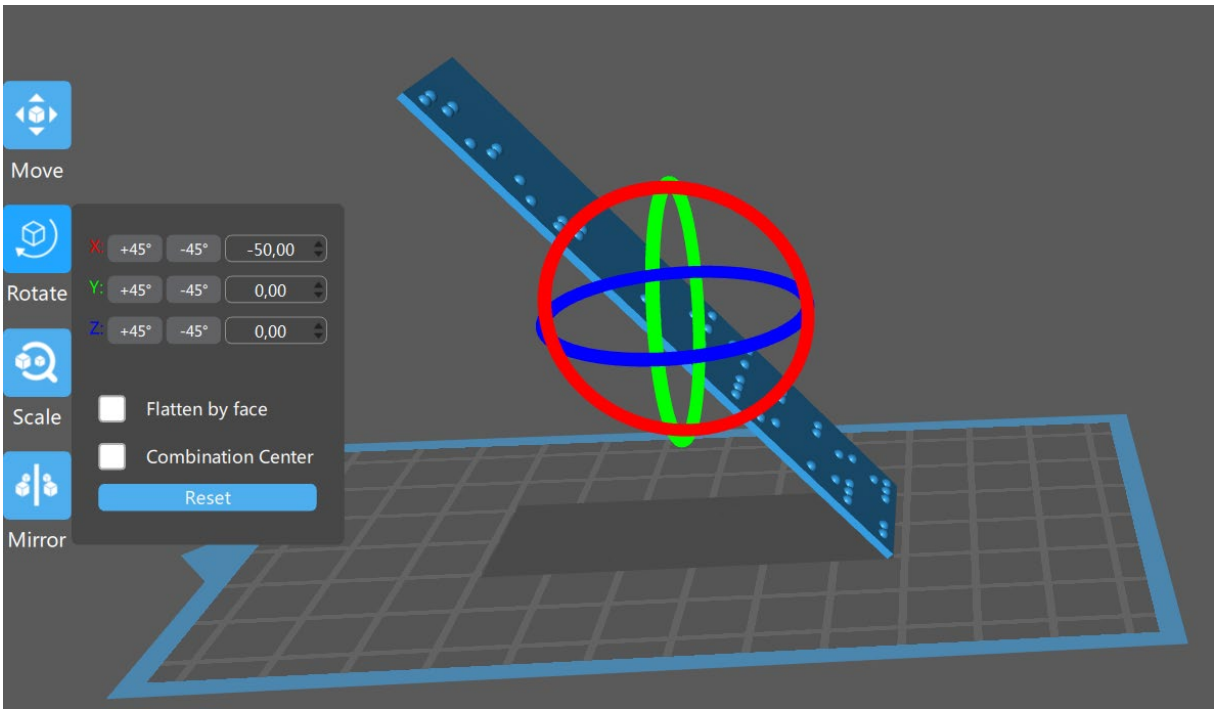

Figure 8 Positioning the component with the "Rotate" function

For this purpose, the component is first marked with a left mouse click. Subsequently, the "Rotate" function needs to be selected. Using the mouse or text input, the component can be rotated by 50° on the red X-axis. The Braille characters should be opposite the construction plate, i.e. pointing downwards. This should allow the resin to drip into the VAT and prevent residue from drying between the braille characters.

The rotation could, of course, be greater than 50°, but this is not necessary for a safe print. This angle of inclination enables warp-free printing and the attachment of sufficient support structures.

3.2 Applying the support structure

To apply the support structure, select the "Support settings" in the top right window by left- mouse click on it.

The window with the setting options for the support structures then appears.

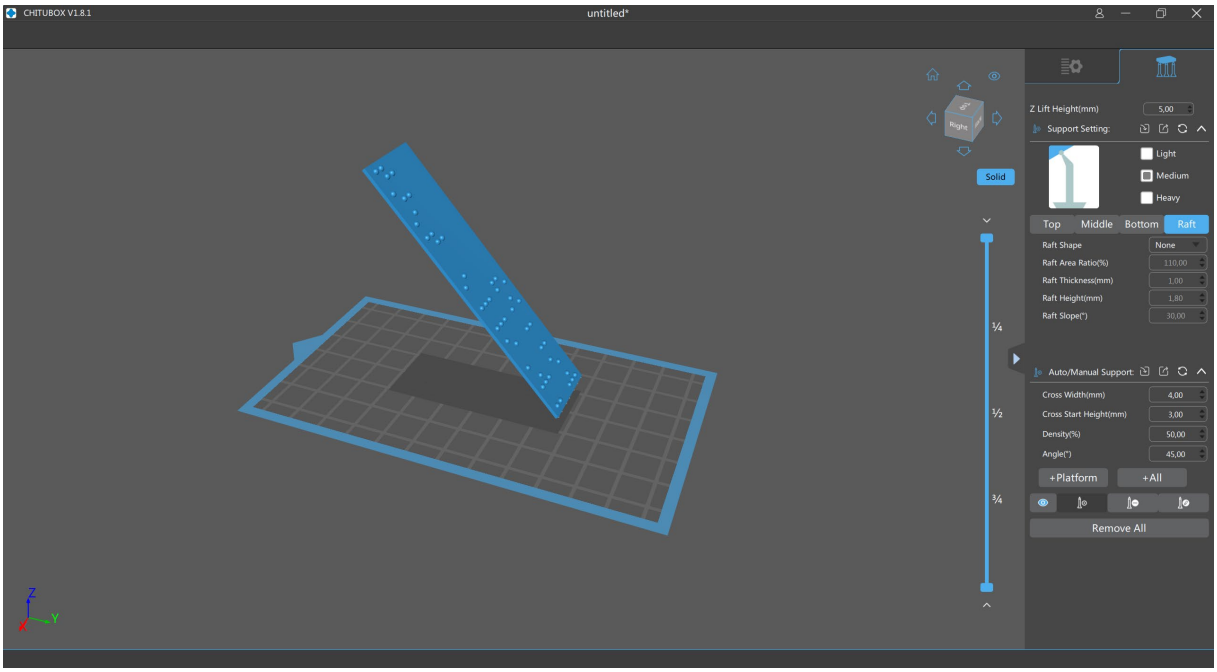

Figure 9 Chitubox-window with Support settings

The support settings are very versatile. In the upper area of the window, the thickness of the support structure and the shape of the contact surface to the component can be set. In the "Top", "Middle" and "Bottom" areas, these settings can be adjusted in different ways. Normally, the standard settings are sufficient when selecting the thickness. With the "Raft" function, a bottom plate can be added. This is useful for large components with extensive support structures. This makes it easier to remove the component in one piece.

In the lower area, the support structure can be added and removed manually. The "+Platform" button adds the support structure only to the components closest to the building plate. With the button "+All", the support structure is added to all overhangs. However, care must be taken that at least 55° have been set in the "Angle" setting.

In our example we choose the function "+All". The generated support structure is almost sufficient. However, to achieve a straight surface at the bottom edge, we have to manually add support structures along the edge. To do this, the function "Add" must be activated in the support structure bar.

Figure 10 Support structure bar, “Add” function activated

By left mouse click on the target area of the component, one column is added at a time.

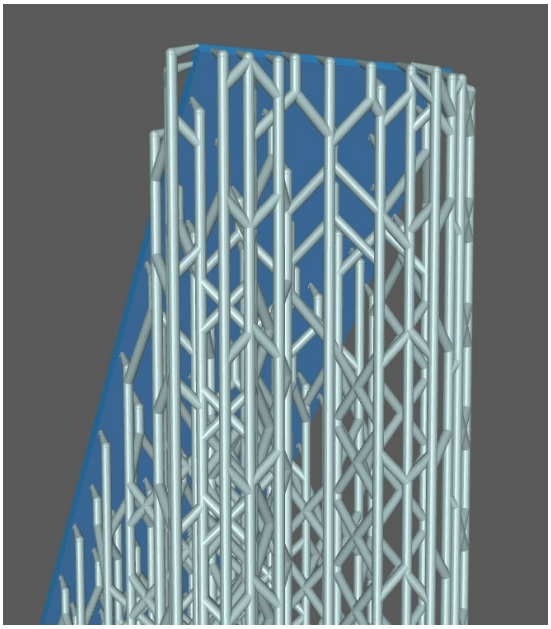

Figure 11 Manually added support structure on the edge

To remove all support structures, select the "Remove all" button. To remove individual structures, select the middle button with the minus sign in the support structure bar. Subsequently, mark the support structure to be deleted with the left mouse button. This is then shown in red. The support structure is then deleted with the remove button.

3.3 Slicing the component

Once all the support structures are in place, return to the model view by selecting  .

Activating the "Slice" button starts the process. The progress of the slicing operation is indicated by a progress bar at the bottom of the screen. Even with smaller models, the slicing procedure requires more time compared to the FDM process.

By pressing the "Save" button, the file is saved and can be transferred to the SLA printer.

4 Hollowing the model

The SLA process does not use infill. There is thus the option to print the parts solid, which increases the weight and the resin consumption. Therefore, it is possible to hollow out models. Hollowed out models must provide for the resin to drain inside the model. This is why at least two drainage holes of at least 1mm diameter must be included in the model. At least one of these holes must be inserted at the lowest point of the model. If the resin in a hollowed-out model cannot drain off due to a lack of drainage holes, the model will burst. This is caused by outgassing of the resin. To be clear, it is not a question if the model will burst, but only when. Leaking resin is a health risk. Therefore, this risk must be eliminated.

To print the models with overhang in a stable way, the support material is also inserted inside the hollowed model. To ensure this, the following steps must be followed:

1. positioning the model
2. hollowing out the model
3. inserting drainage holes
4. creating support material
5. slicing the model

In our example, we position a model of a church on the building plate so that the drainage holes can be inserted in a sufficiently large part of the model. In the following example, this is the gable roof.

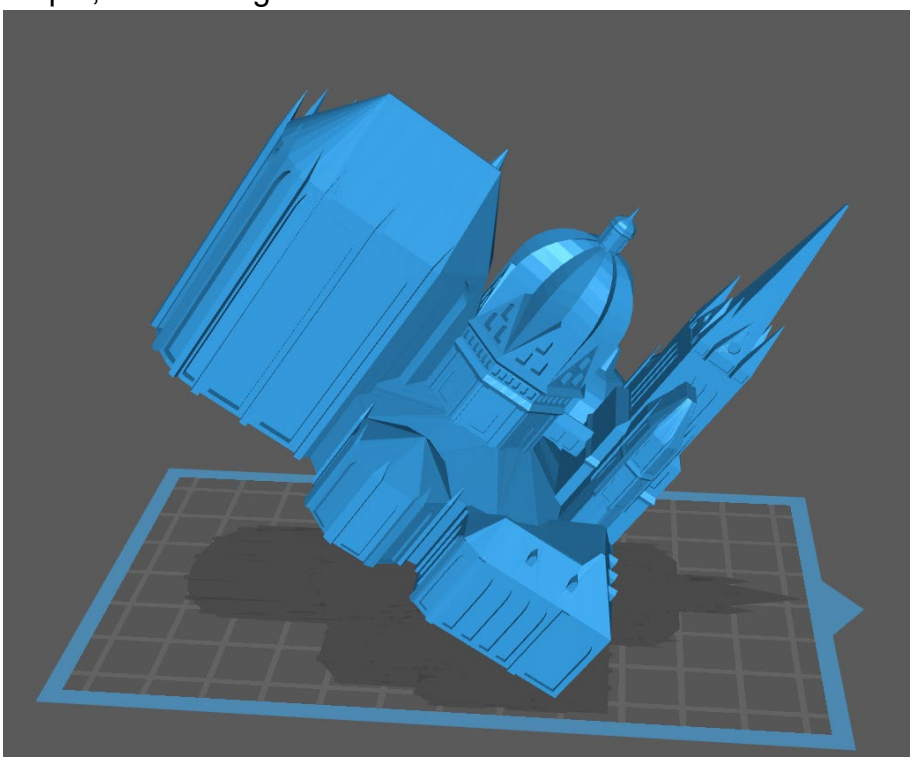

Figure 12 Positioning the model

Activating the button  , opens a dialogue window.

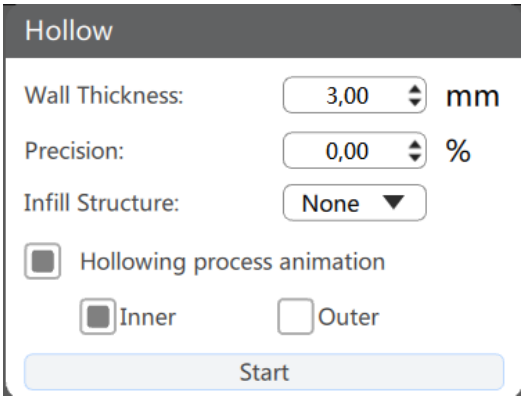

Figure 13 Dialogue window "Hollow"

In this dialogue window you can specify the wall thickness. It should not be less than 1 mm, even for small models. For models with overhangs of 90°, an infill structure can also be specified. In our case, this can be skipped. By activating the "Start" button, the model is hollowed out.

Subsequently, select the "Dig Hole" button. 
In this window, you can define:

1. the shape of the hole
2. the diameter of the hole
3. the depth of the hole

By activating the "Add a hole" button, you return to the window.

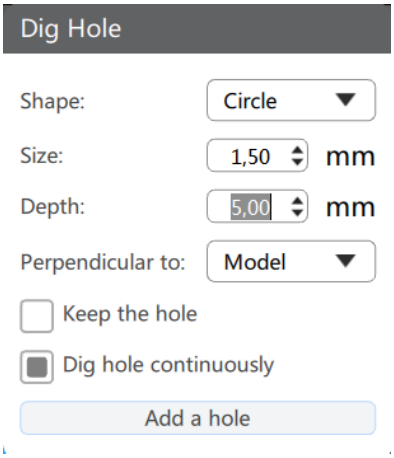

Figure 14 “Dig Hole” Window

Use the mouse pointer to position the green marker at the desired location. The hole is inserted by clicking the left mouse button. The second hole is inserted at the bottom of the pediment in the same way.

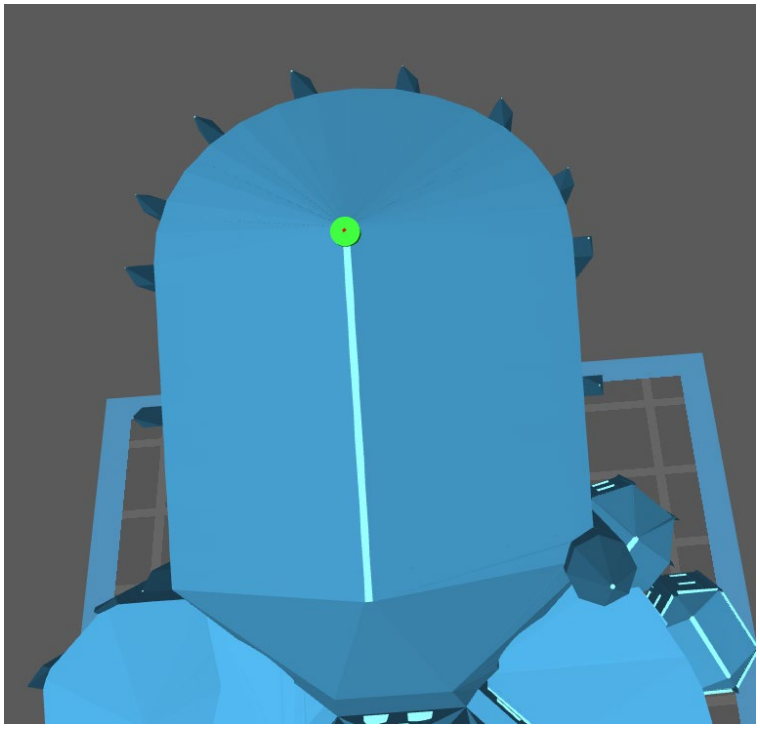

Figure 15 Positioning of the first hole

The support structure is then created as described in chapter 3.2 and the model is sliced.

5 List of Figures