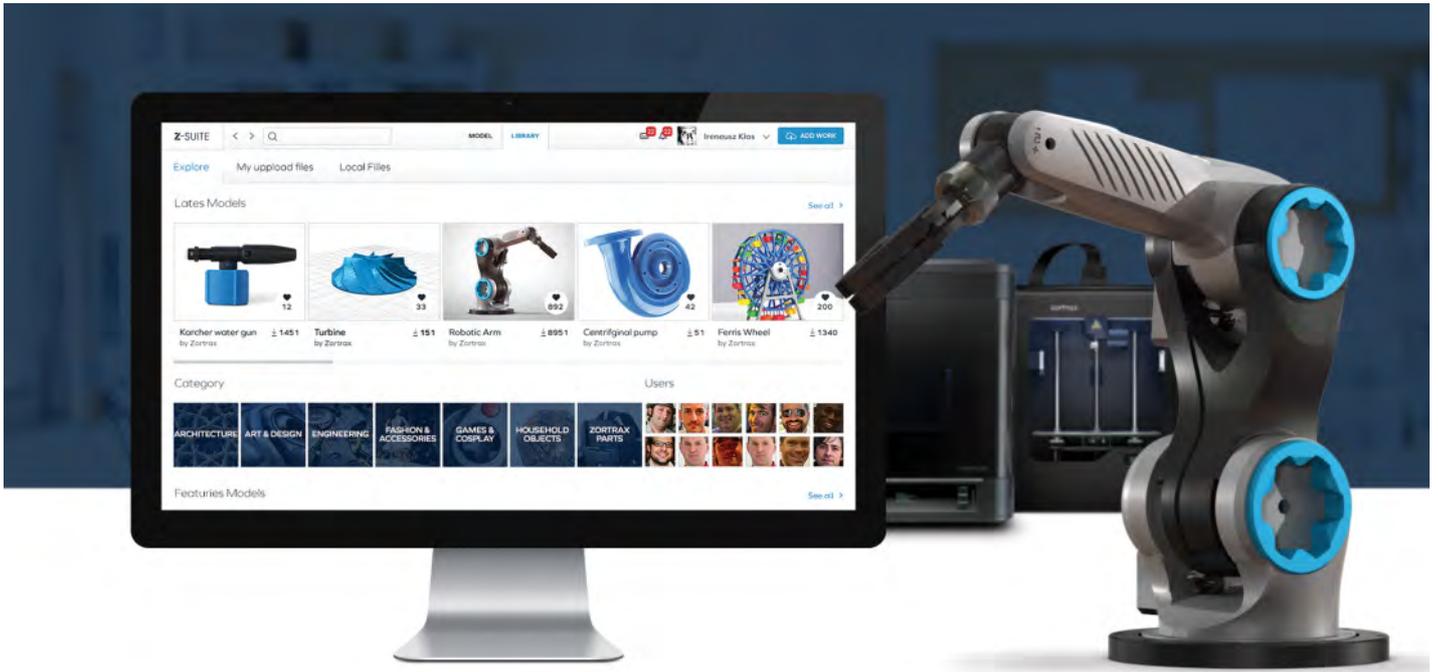


Printing at Skylab: Introduction to Z-suite and Zortrax M200



Zortrax M200 is a commercial 3D printer that started out as a Kickstarter project in 2013. The printer is currently (January 2016) one of the 'easiest to use' desktop FDM 3D printers on the market. Unlike many other 3D printers, Zortrax ecosystem is not open-source, with uneditable g-code (machine instructions), and limited control over advanced print settings. This means that the only possible way for using the Zortrax machines is through Zortrax's slicing software named 'Z-suite.'

DTU Skylab currently have 5 Zortrax M200 machines available for regular use. Please note the following:

- The build envelope is 200 x 200 x 180 mm
- Always check the type of the filament spool on the machine you are about to use. Please do not change filament spools, and contact a Skylab employee if a spool needs to be changed.
- You are always welcome to consult with Skylab workshop employees if you have any questions or doubts about using the equipment.
- If you notice that any equipment is out of order or malfunctioning, please do not try to fix it yourself and let an employee know about the issue.
- Reservations for a number of the 3d printers can be made on [planningpanda.com](http://www.planningpanda.com) (<http://www.planningpanda.com>). The remaining machines are meant for 'drop in printing.'
- You can download the Z-suite to your own computer <http://support.zortrax.com/downloads/> (<http://support.zortrax.com/downloads/>). During installation, you will be asked for a machine serial number; you can use this one: ZACEC8AAB

From Digital to Physical: How to print with Zortrax & Z-suite

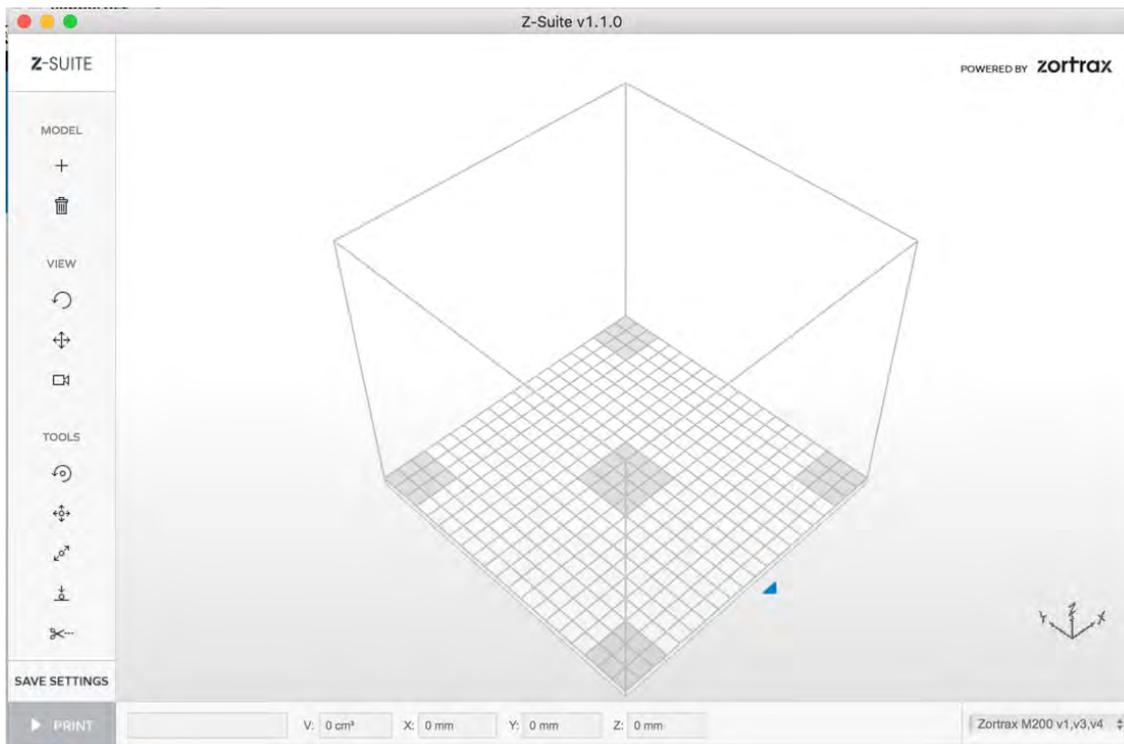
- Step 1: Getting started.
- Step 2: Workspace views
- Step 3: Workspace tools
- Step 4: Print settings
- Step 5: Prepare to print (slicing)
- Step 6: Start printer
- Step 7: Tips and tricks

Step 1: Getting Started

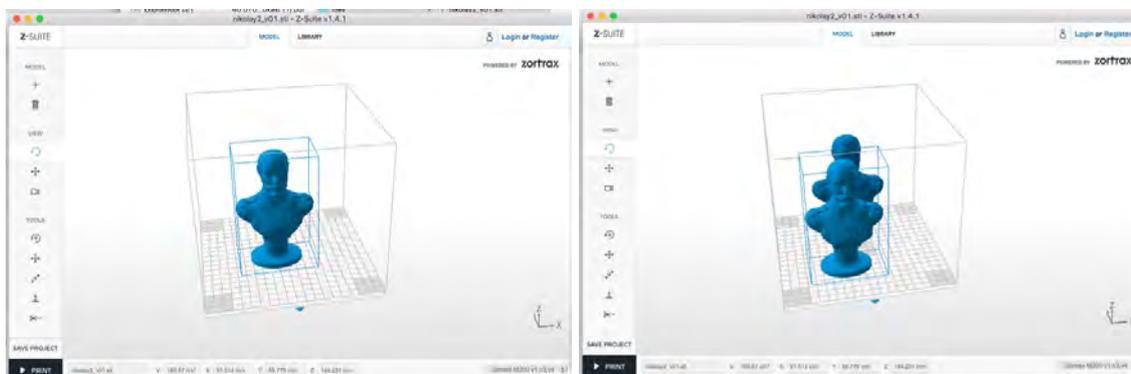
Z-suite takes a 3D CAD file and converts it to motion commands for the printer. As a prerequisite, you will need a valid 3D file in either of the following formats: .stl, .obj, .3mf. Please note that All CAD software can export models as .stl files, which is the standard 3d printing file format. For the rest of this tutorial, we will assume you are working with an STL file.

Import file

Locate the Z-suite software and open the program.



You can either 'drag and drop' the stl file on your workspace, or click the + icon and import the stl file from its location on your hard drive.

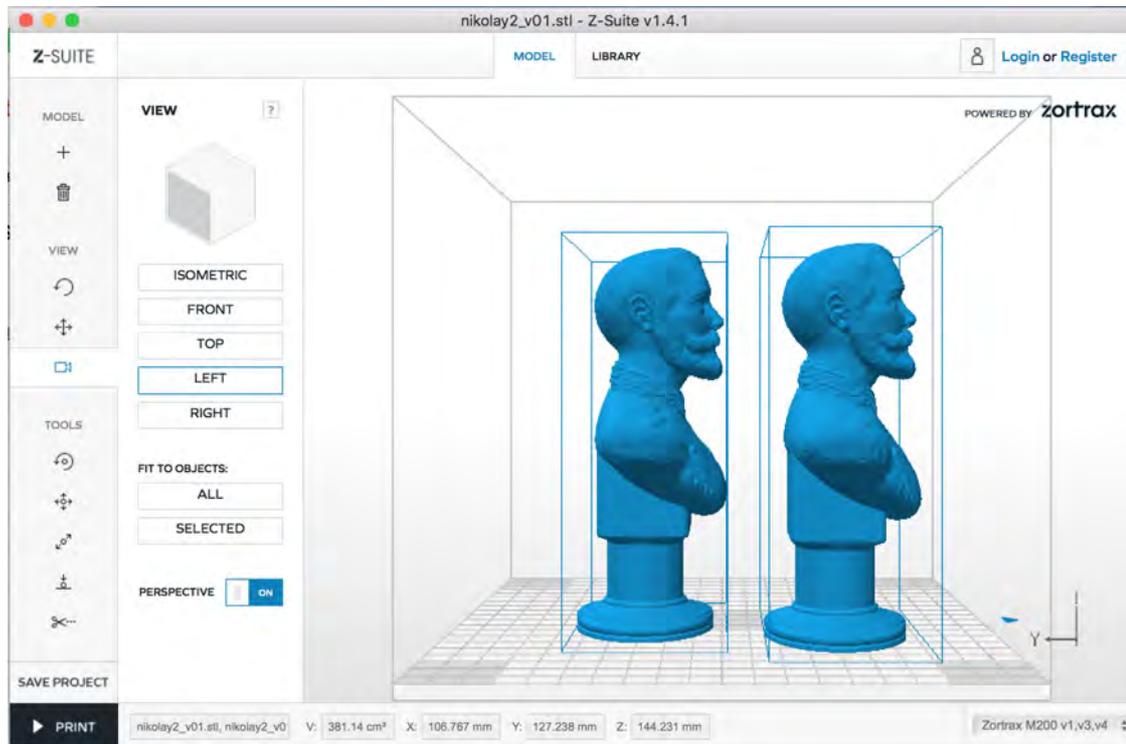


Multiple models can be added to the same workspace and models can be deleted by selecting a specific model. **When selected, a blue frame appears around the model. Use the right mouse button to select a model.**

Step 2: Workspace views

Z-suite has 3 different 'view tools'. When they are active, you can only manipulate the workspace view.

You can rotate and move the viewport using the controls under 'View'. The 'movie' view will help you set standard viewpoints



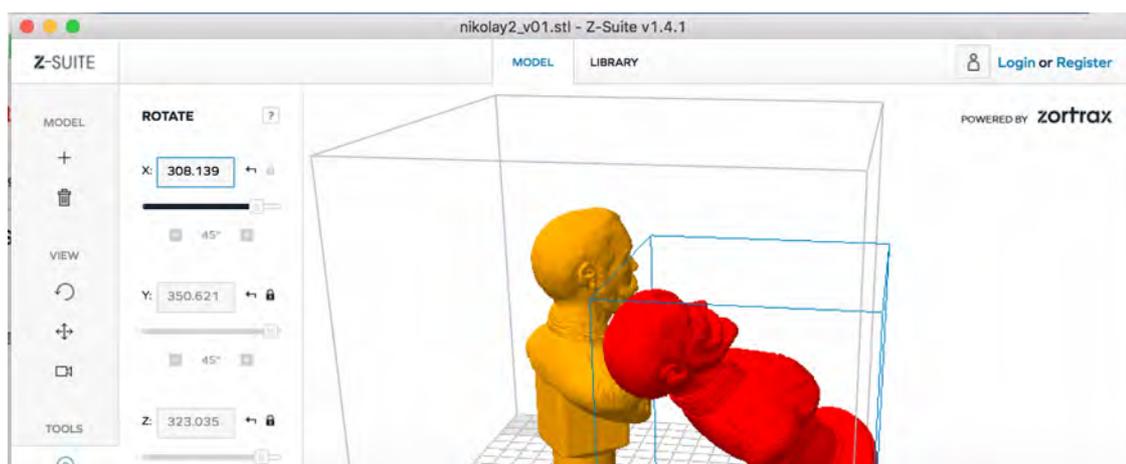
Step 3: Workspace tools

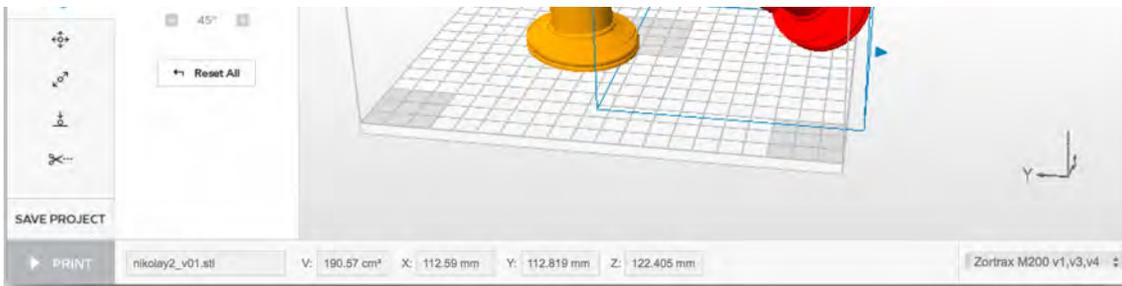
Your model will be imported to the workspace in the same orientation as it was exported from your CAD program. It is very important to move and orient your model for best printability, and workspace tools allows you to do basic manipulations on your model.

Rotate object

The rotate tool allows to rotate a selected model around the selected axis. Note that in the screen dump below the X axis is selected. The axis is selected in the **rotate menu**. The rotation is carried out by clicking and dragging with the mouse or by typing in a specific value in the rotate menu.

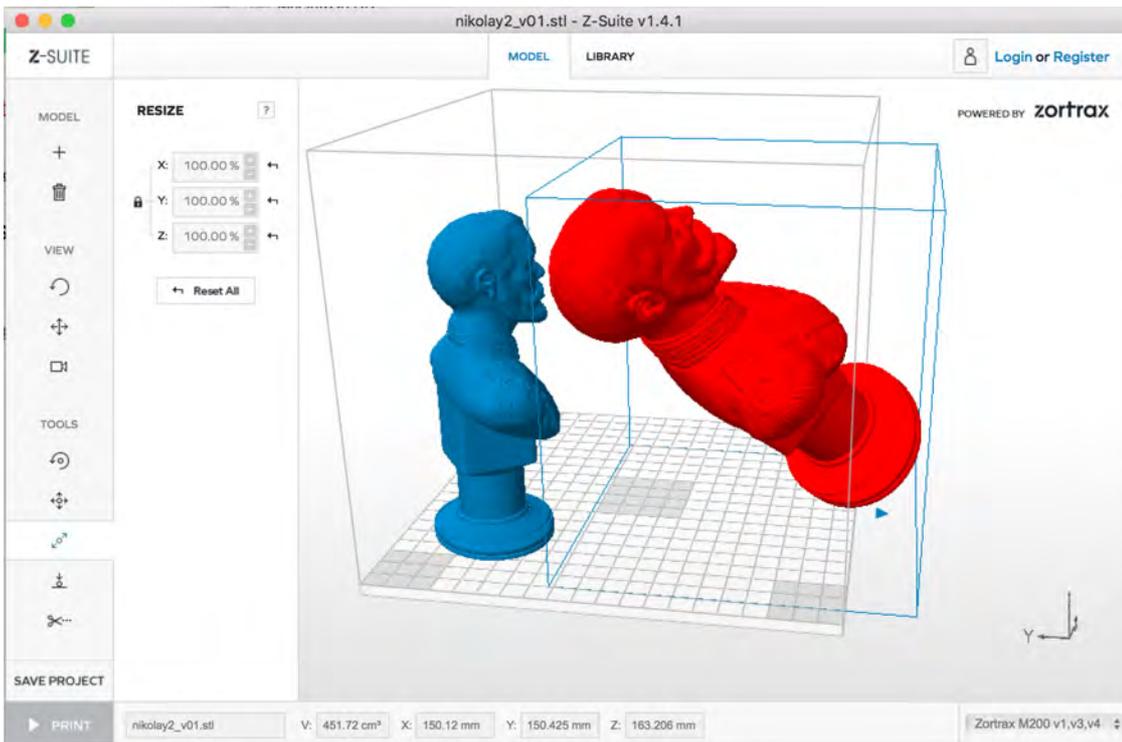
Pro tip: Note how the selected model have been **coloured red**. This is to illustrate that the model has been placed outside the print envelope. Further the second model is now an **orange colour**. This is to illustrate that the two parts are intersecting each other. In Z-suite a printable model is **displayed in blue**.





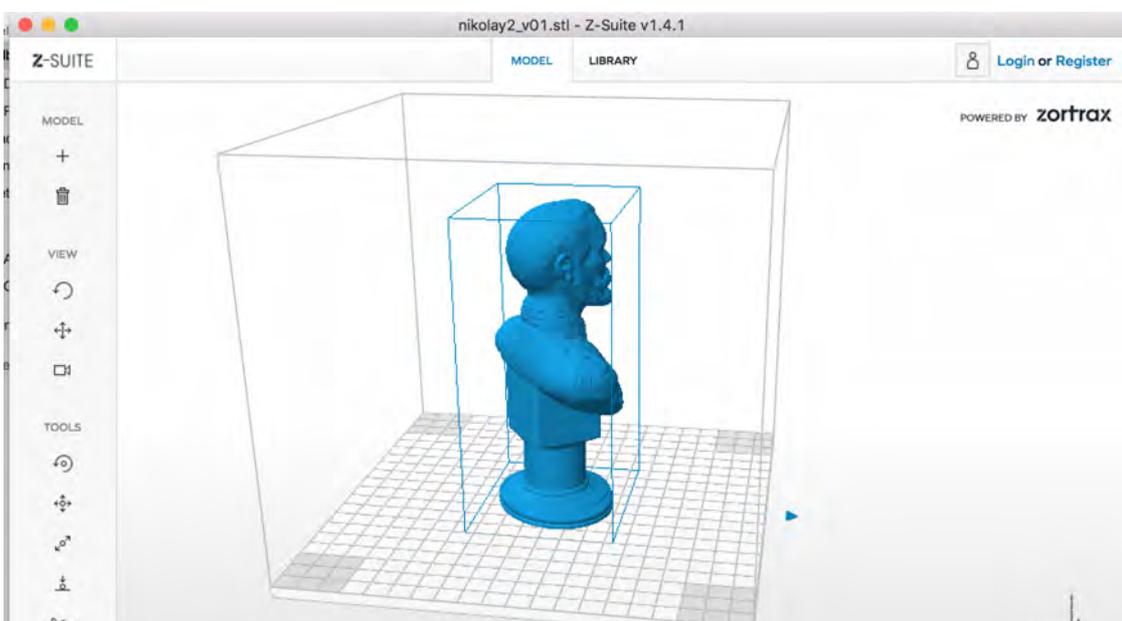
Move object

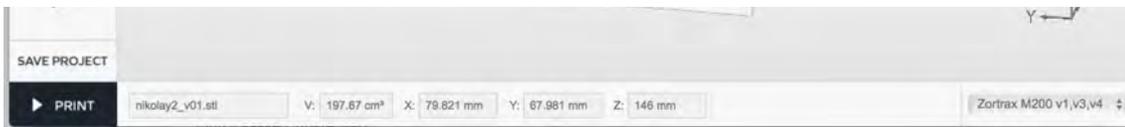
Similar to rotating, moving models can be done by typing a specific value in the move menu or by using the 'click and drag' functionality with your mouse. 'Center All' is self explanatory.



Resize object

While resizing models within Z-suite is possible, it is always better to resize in the CAD program, for accurate dimensioning. In Z-suite, resizing is done in terms of percentages (relative values). The relationship between the different axis is locked. This is illustrated by the padlock symbol just left of the respective axis X, Y and Z. By clicking it the feature can be disabled and the single axis can be resized independently.





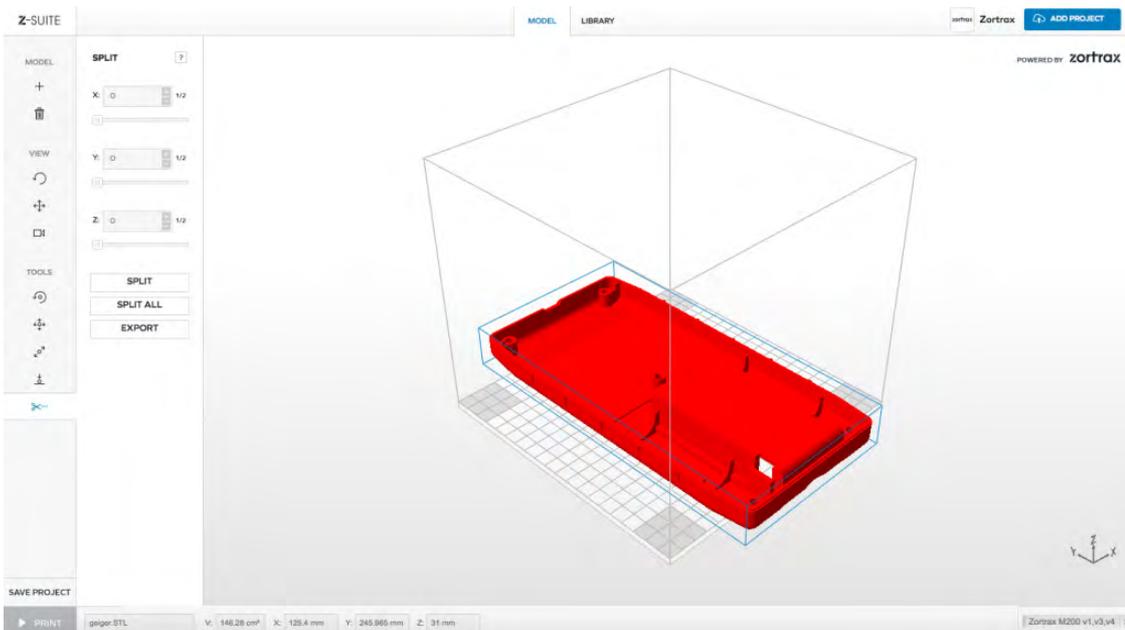
Auto arrange object

The auto arrange functionality tries to optimize the distribution of the models on the build plate in the case of multiple models in the build envelope. A single model will just be centered at the middle of the build plate.

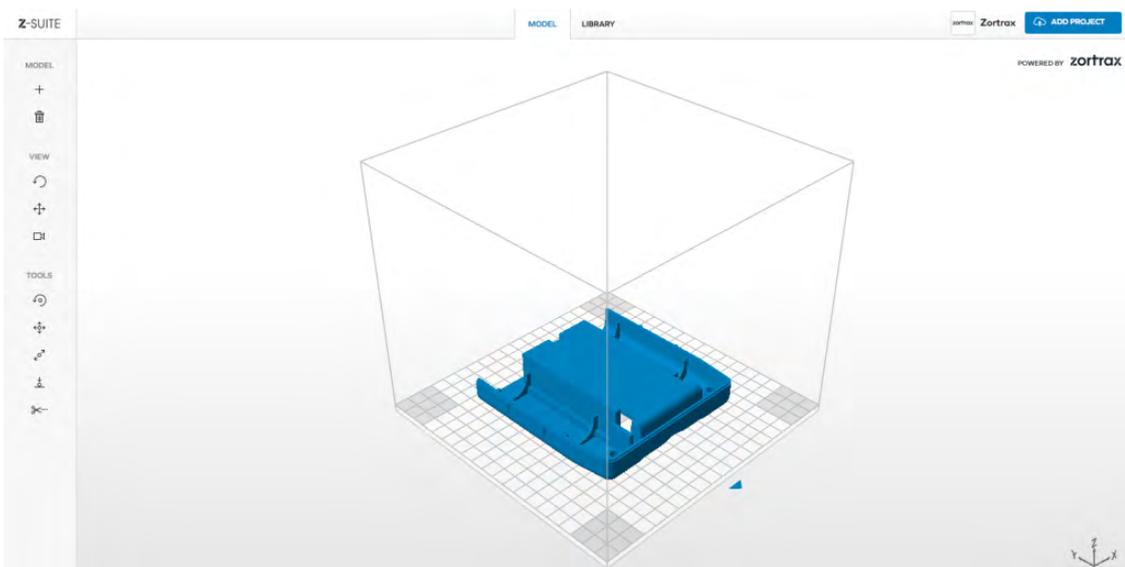
Split model

Z-suite contains a 'split model' feature which can be used if your models exceeds the print envelope of the printer. Again, we suggest splitting models in the CAD program to have a better control on the process.

NB: This feature often makes Z-suite crash in OS-X but works fine in Windows.



Choose the cutting plane and click Split at the desired dimensions. You can choose to split the models in half by clicking Half button. You can save both models separately by clicking Save.

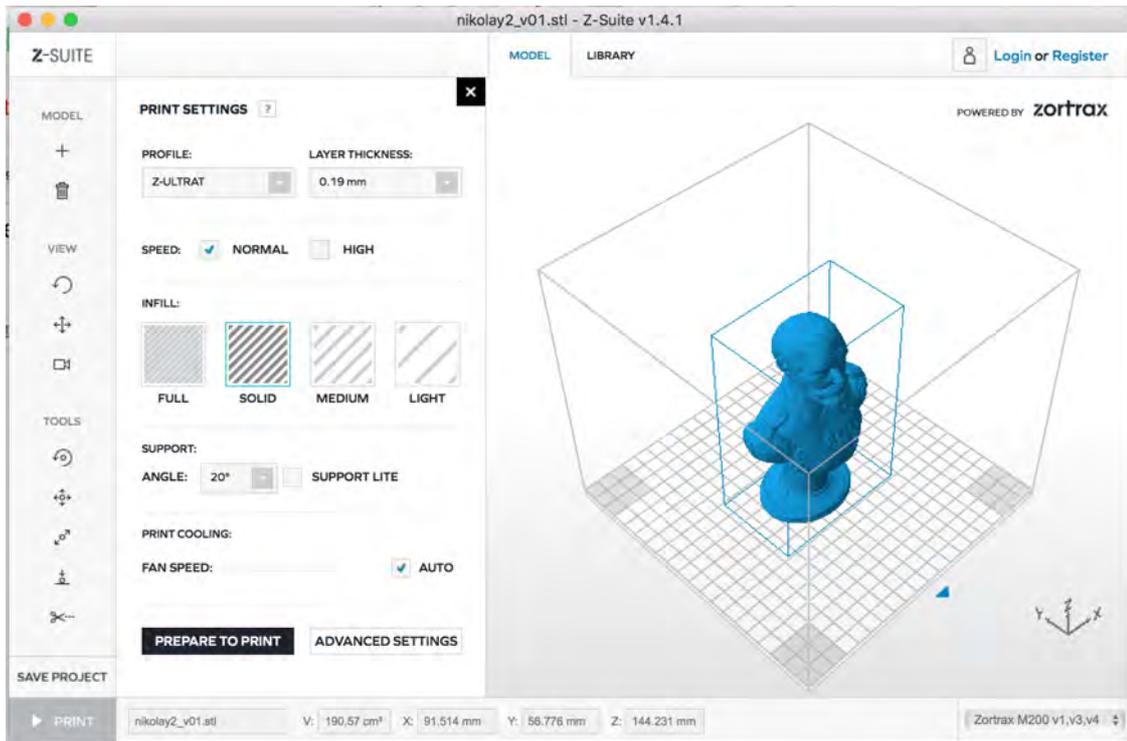




To start printing directly simply delete one half of the model and use e.g. the Auto arrange feature to place the model to be printed.

Step 4: Print settings

Once you are satisfied with the orientation and the placement of your models, you will move on to selecting the print parameters. Pressing the print button in the lower left corner will present these parameters.



Print profile, layer thickness and speed

Material

The drop Down menu under 'Profile' allows you to select the type of filament you will be using for the specific print job. In most cases you will be using the spool of filament already installed at the machine. **Please look at the spool on the machine you are about to use** and select the profile accordingly.

PROFILE:

Z-ULTRAT
Z-ULTRAT
 Z-HIPS
 Z-GLASS
 Z-PETG
 Z-PCABS
 Z-ABS



Layer thickness and print speed

Select the layer thickness you prefer in the dropdown menu. Finer layers provide better surface quality but the printing time and the material consumption will be increased. In many cases we recommend printing with thick layers (190 microns) to reduce print time and reduce chances of warping and eventually failed prints. The picture below intends to give you an impression of surface quality and also illustrate how the print time increases dramatically by choosing thin layers (90 microns).

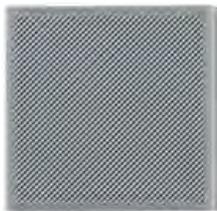


Layer height
190 microns
 (Full infill)
 (Normal speed)
Print time: 34 min

Layer height
90 microns
 (Full infill)
 (Normal speed)
Print time: 64 min

Speed. Z-Suite allows to choose between two different print speeds with simple checkboxes (see screen dump at page 10). High speed will reduce the printing time but it will also marginally reduce the print quality.

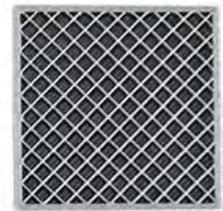
Infill



Full



Solid



Medium

Select the density of the model infill. Pictures below show what every type of an infill looks like. Full Infill allows you to print a dense model with the highest strength (but also long print time) and light allows for a light structure (and lower print time).

generally, the standard of support structure type might be generated from the printing, then redefining of mechanical part dimensions to meet the design aspect of 3d printing. The platform My3d Matter have done extensive research in how different types of infill influences 3d printed parts. For more information follow the link below:

<http://my3dmatter.com/influence-infill-layer-height-pattern/> (<http://my3dmatter.com/influence-infill-layer-height-pattern/>)

Support

As a rule of thumb, you should try to minimize the need for support structures by orienting your model in Z-suite or by tweaking the source file in the CAD program. Still in many cases, you will need the support structures to be able to print for model.



Support 20°



Support 40°

Z-suite allows to choose the slope angle for the support structures. 20° is the default option. It means that for all hanging parts of the model the support structure is generated at a 20° angle - measured from horizontal level. This is illustrated below for the settings 20°, 40° and 60°. The left illustration shows how support structure is only generated at a slope of 20°. The middle illustration shows support for slopes of below 40° and finally the right illustration shows support for 60° and below.

The tickbox 'Support lite' generates support structure without the outer perimeter. 'Support lite' consumes less filament and is easier to remove.

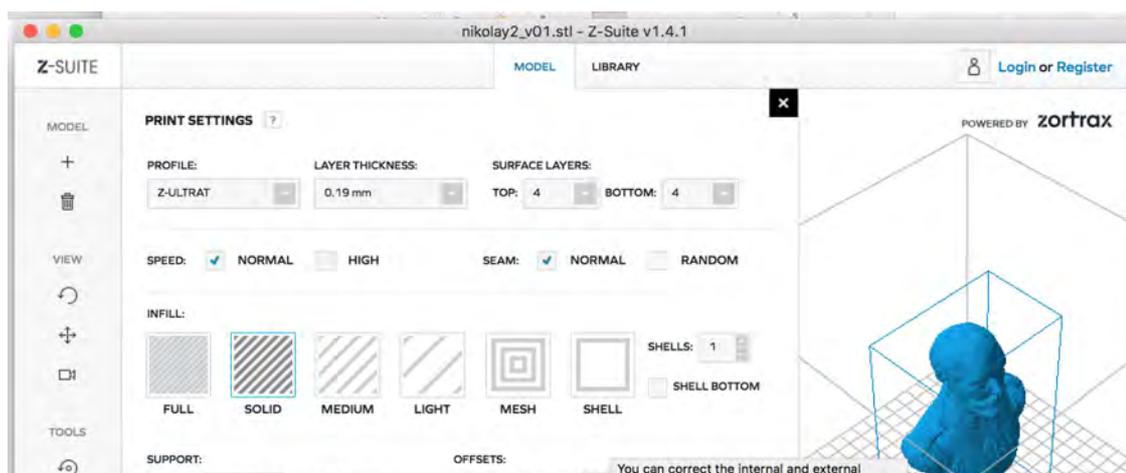


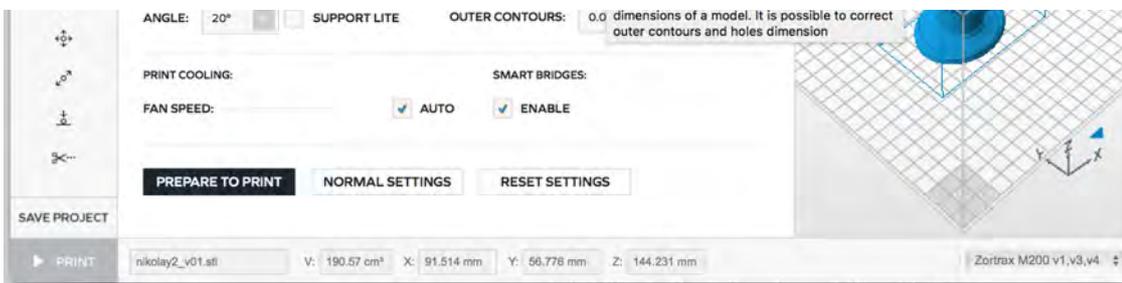
Print cooling

The feature allows you to set the fan speed, which cools down the printed models near the hot-end of the printer. Cooling is not useful for large prints without any thin sections, but needed for small and thin walled geometries. We recommend leaving the default option – AUTO. The printer will adjust the fan speed accordingly.

Advanced settings

In most cases, basic settings are good enough to achieve good quality prints. Advanced settings extends some of the options as explained below.





Surface layers

Allows you to set the number of layers in the top and bottom part of the print. In most cases you will be fine by using 4 layers. It is not recommended to go below a value of 4 layers as it increases risk of unsatisfactory surface quality. For challenging prints using up to 8 layers can help improve the surface appearance in challenging geometries.

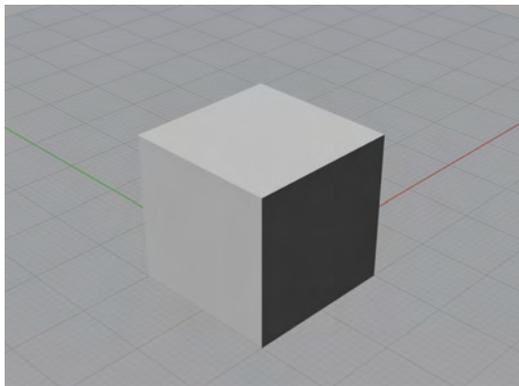
Seam

Allows you to choose the starting point of the new layer: fixed or random. In order to obtain a smooth surface finish, the random feature is recommended for parts where surface appearance is important.

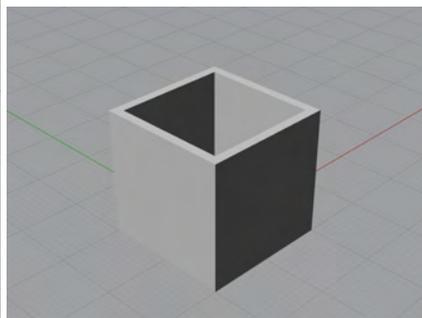
Advanced infill options

Mesh: Prints the model with 0% infill with horizontal surfaces (top and bottom) and the walls. This basically means that you will print a hollow structure. E.g. a hollow cube.

Shell: Prints the model with 0% infill and no horizontal surfaces (top and bottom). This option allows to choose the number of perimeters of the model wall. This can be useful for e.g. printing a vase.



Choosing 'mesh' will result in hollow cube

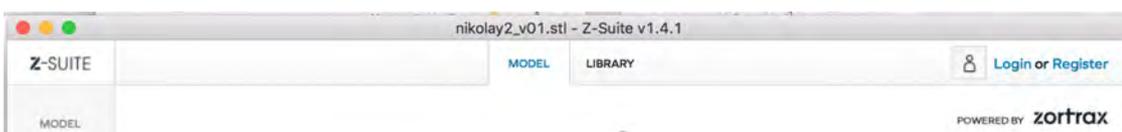


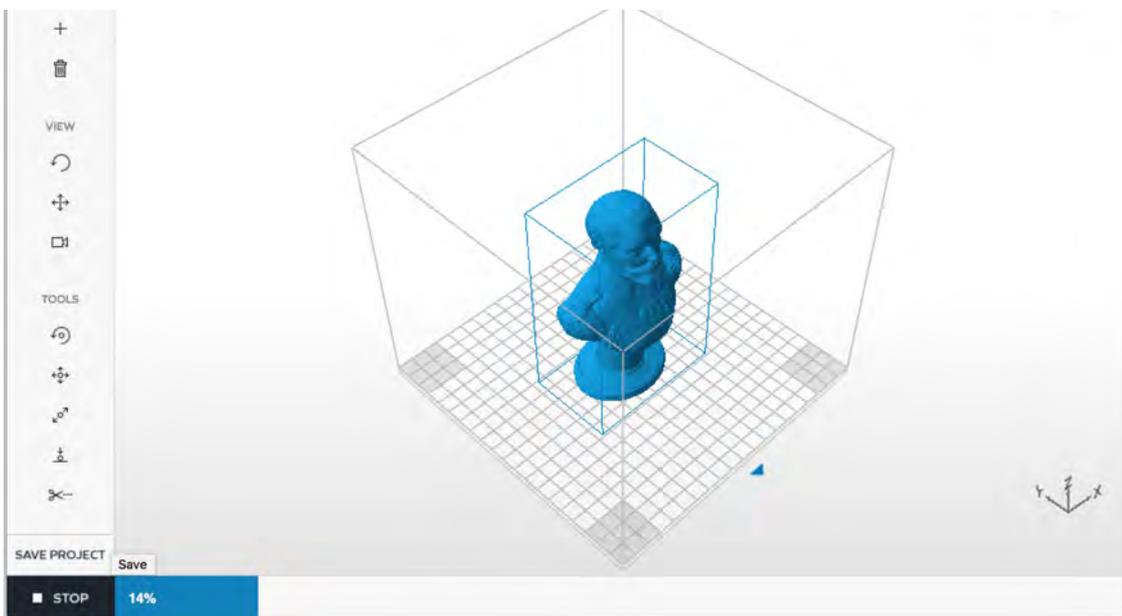
Choosing 'shell' will produce, well... a shell

Offset

This feature helps to correct the internal and external dimensions of a model. It is possible to modify outer contours and holes dimension. This can be helpful as e.g. holes tend to turn out smaller on a 3d printed part than what was defined in the Cad software.(see the tips at the end)

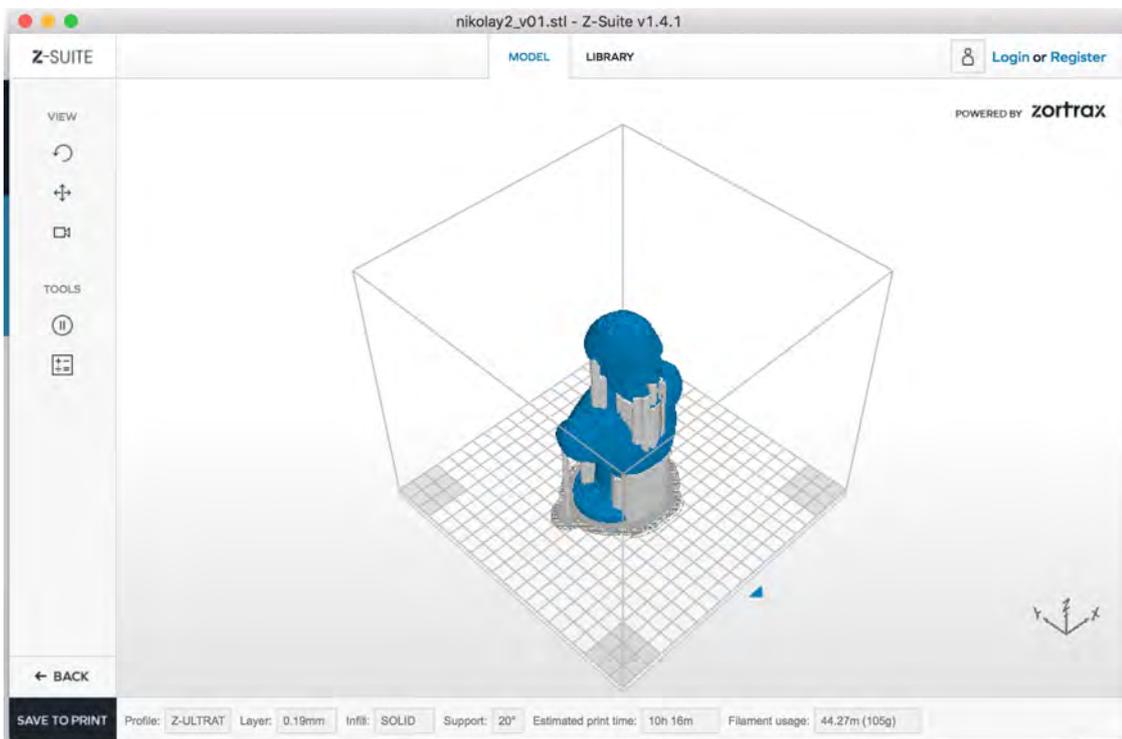
Step 5: Prepare to print (slicing)





When you are sure that all settings are optimised, you can press the print button one more time. The software will now start slicing your file and generate a 'g-code'. This process might take a few minutes and the process bar can seem to be stalling or completely frozen a few times during the process.

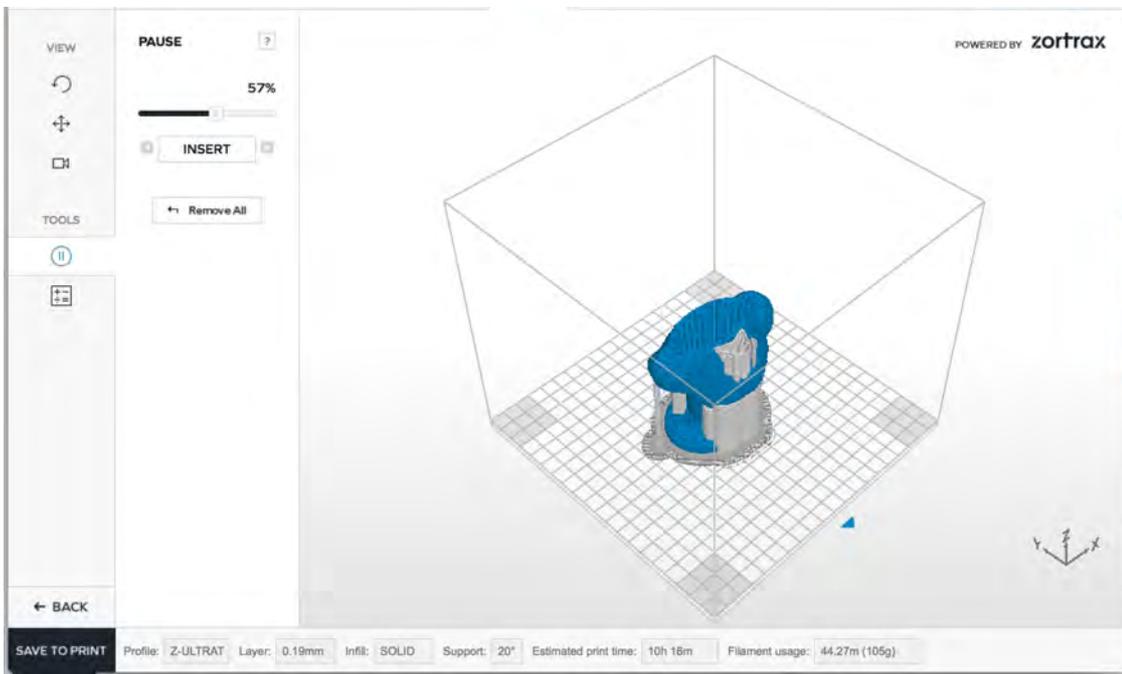
When the slicing process is completed a "Save to print" icon in the lower left corner. You can now save your 'g-code' on your hard drive or directly on the SD card. Before doing so you still have a few nice features available worth considering.



Inspect sliced design

Before saving you have the possibility to inspect your sliced model. The 3 different view modes are still present and you can rotate the design as desired. Further in the 'tools' section you can scroll through the layers of the design to inspect how e.g. the support structure have been generated. Also this allows you to pause the print process at a specific percentage level. This allows you to e.g. insert a nut or a screw inside the hollow structure and incorporate it into the 3d printed part (more on how to incorporate nuts, bolts and screws this in the tips/links section later in this document).





When you are happy about your setup press the “Save to print” icon in the lower left corner. You can now save your ‘g-code’ on your hard drive or directly on the SD card.

Insert SD card

You need to save your g-code on a SD card in order to transfer it to the printer. The previous user have probability left one of Skylabs SD cards sitting in the printer. Otherwise you might find one in the drawer below ‘printer 3’.

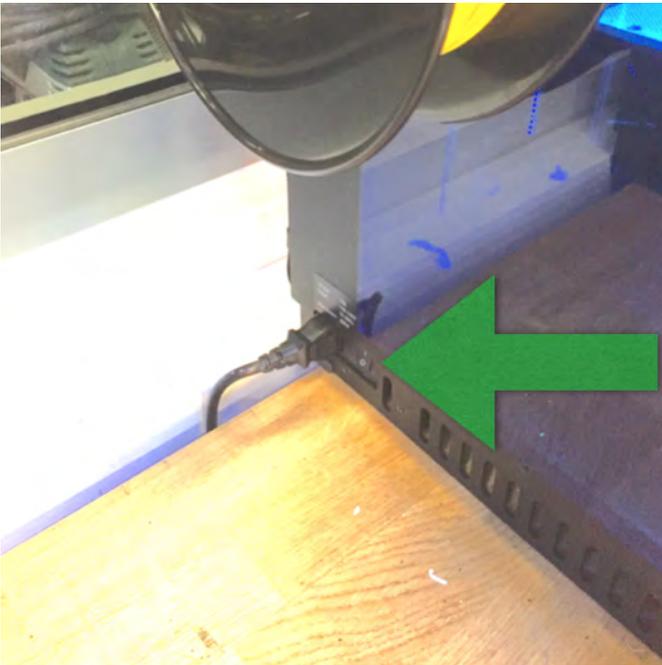
The PC’s at DTU Skylab has a USB to SD card reader readers similar to the one shown below. Use this device in order to save your g-code and transfer it to the 3d printer.



Step 6: Start your print

The printer has an on/off switch on the left side of the printer.





Insert SD card and start print



The SD card (with your g-code on it) is inserted into the printer. Orient the card so that the connection pins are pointing towards you.

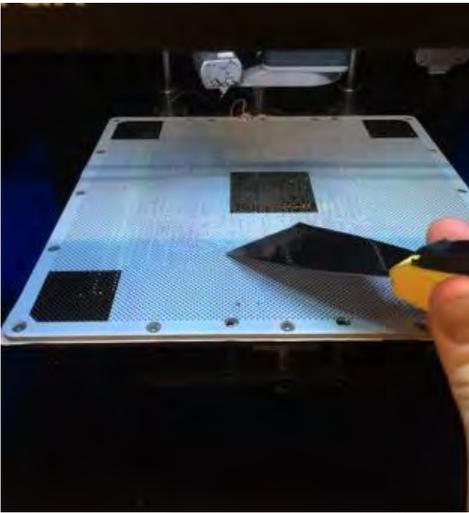


Use the knob to navigate the printer menu. To start a print simply chose 'models' and select for file (usually the top-most one).

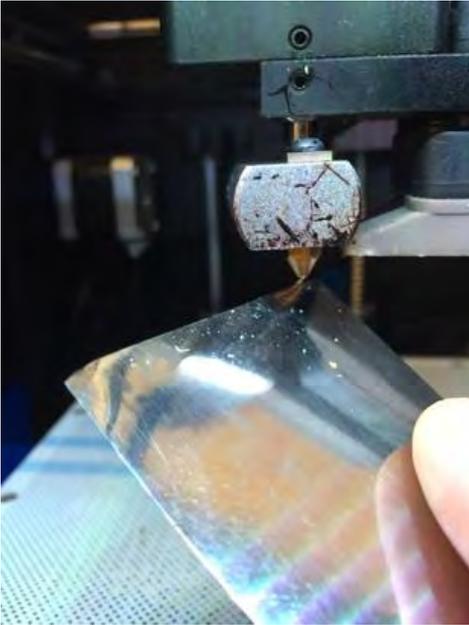


The printer starts when it has finished heating the hot-end and print bed. The display shows the process and time remaining.





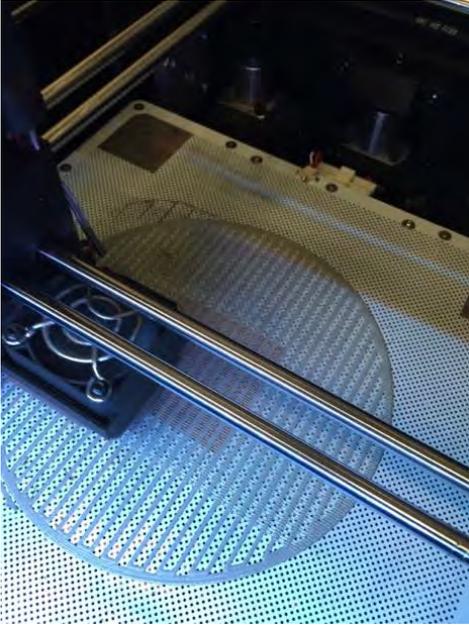
Before the print starts clean of any old filament residues on the nozzle and print bed with a spatel/scraper.



Ensure that there is enough filament left on the spool. If you are in doubt you can use the scale on the desk and check the weight of the spool (the spool itself weighs around 100 g). Z-suite provides you an estimate of filament consumption in grams (and print time) use this information to see is there is enough filament.



Place all side covers on the printer when in operation and do not put an open ventilation nozzle close to the printer (it creates a draft which introduces stresses to the print).



Watch the first 5 layers of the print – if they turn out well there is a good chance that the remaining part of the print will be successful.

The final 3d prints are often slightly smaller than the CAD model dimensions. If the tolerances are important you have to estimate the approximate shrinkage and add a margin (usually from 0.2mm to 0.3mm) to each dimension. This should be done during the initial cad modelling. Holes and outer contours can though be slightly modified in the **offset feature in the advanced**

If your print is warping, there is no need to complete the print job. Chances are really high that it will fail. Stop the printer and consider an alternative setup for your print job (see below for inspiration)

Warping

Warping is likely if you are printing a large model. There are different ways to tackle this

Ensure that all side covers are in place and close the air vents above the printer to avoid any

Don not print in high speed and orient the part with as little surface area in contact with the build plate as possible (be smart about how the part is oriented – support structure can be a great help but also a hassle to remove.)





Ensure that the print bed is clean – remove any residues from earlier prints and wipe it with ethanol.



Ensure that the print bed is correctly calibrated. *The printers at Skylab is calibrated weekly. If you think that the bed need calibration please ask a Skylab employee before you try anything yourself :-)*

Useful links for your 3d printed project

- Tip on how to insert bolts, nuts and screws in a 3d print: <https://www.fictiv.com/resources/starter/how-to-choose-the-best-fasteners-for-3d-printed-parts> (<https://www.fictiv.com/resources/starter/how-to-choose-the-best-fasteners-for-3d-printed-parts>)

- Tips on how to sand, prime and glue 3d printed parts (remember that the ABS part of the article is the one relevant for you):

<http://blog.fictiv.com/posts/ultimate-guide-to-finishing-3d-printed-parts> (<http://blog.fictiv.com/posts/ultimate-guide-to-finishing-3d-printed-parts>)

- Painting ABS printed parts:

<http://support.zortrax.com/painting-the-print/> (<http://support.zortrax.com/painting-the-print/>)

- Glueing together ABS printed parts:

<http://support.zortrax.com/gluing-the-print/> (<http://support.zortrax.com/gluing-the-print/>)

- Short video on how to remove support structure:

<https://www.youtube.com/watch?v=dxOV4OAifzY> <https://www.youtube.com/watch?v=dxOV4OAifzY>



<https://www.youtube.com/watch?v=dxOV4OAifzY>

- Extensive article on infill and patterns as well as 3d printing layer heights:

<http://my3dmatter.com/influence-infill-layer-height-pattern/> <http://my3dmatter.com/influence-infill-layer-height-pattern/>

Details about the Zortrax M-200

PRINTING	
Print technology	LPD™ - Layer Plastic Deposition
Build volume	200 x 200 x 185 mm [7.87 L x 7.87 W x 7.28 H in]
Layer resolution settings	Advanced: 25-50* microns [0.000984-0.0019685 in]
	Standard: 90-400 microns [0.003543-0.015748 in]
Wall thickness	Minimal: 400 microns Optimal: 800+ microns
Resolution of single printable point	400+ microns
Filament Diameter	1.75 mm [0.069 in]
Filament Type	Z-Filaments™
Nozzle diameter	0.4 mm [0.015 in]
Minimum single positioning	1.5 microns
Positioning precision X/Y	1.5 microns
Z single step	1.25 microns
Extruder maximum temperature	380° C [716° F]
Heated platform maximum temperature	110° C [230° F]