Plastic 3D Printing Submission Guide
Ultimaker S7

D
AN
IELS
DIGITAL
FABRICATION

D
AN
IELS
DIGITAL
FABRICATION

Using this Manual

Each page in this manual follows a general layout of two columns. Read left, to right to the bottom of each column, then move to the next.

important keywords are highlighted in red

On images, areas of interest are highlighted with red arrows or squares/circles.

Table of Contents

The table of contents can be found on the next page. Chapter titles and sections are hyperlinked. You can click on a chapter title or a section to quickly navigate.

Hyperlinks

Hyperlinks are created throughout the manual, all title in table of content is hyperlink to their specific page. Additionally, underlined texts also contains hyperlink to URL or other pages in the manual.

About this Manual

This manual was revised on Mar. 27th, 2025

Contents

Ultimaker S7

The Ultimaker S7 stands out with its dual print head, enabling the use of diverse materials like PLA, ABS, Nylon, and specialty filaments. This feature facilitates complex multi-material prints, including those with water-soluble supports. Its heated bed and integrated camera for remote monitoring further enhance its functionality. Its fast print speed making this machine a versatile tool for both professionals and hobbyists.

Software:
Ultimaker Cura

System Requirement:
Windows 8, 8.1, 10, 11
Mac

Maximum Build Size:
330 x 240 x 300 mm
12.9 x 9.4 x 11.8 in

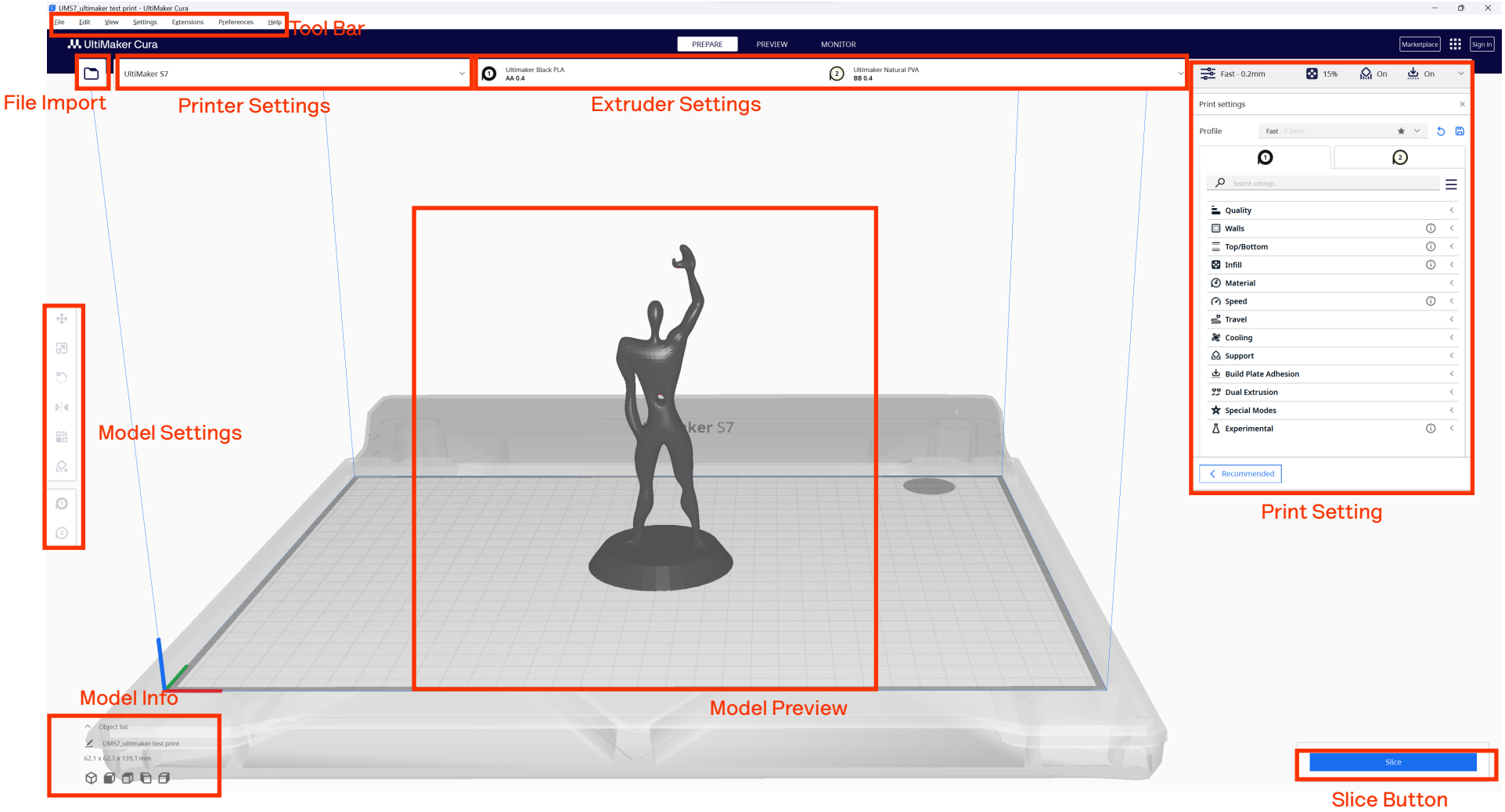
Model Materials:

PLA Tough Black & White (model)	$0.10/g
PVA (support)	$0.20/g

Layer Thickness (resolution, mm):
0.06
0.1
0.15
0.2
0.3

Ultimaker Cura UI

Ultimaker Cura is the primary slicer used to process files for 3d print, below are annotations for features of the UI.



Ultimaker Cura Download

https://ultimaker.com/software/ultimaker-cura/#downloads

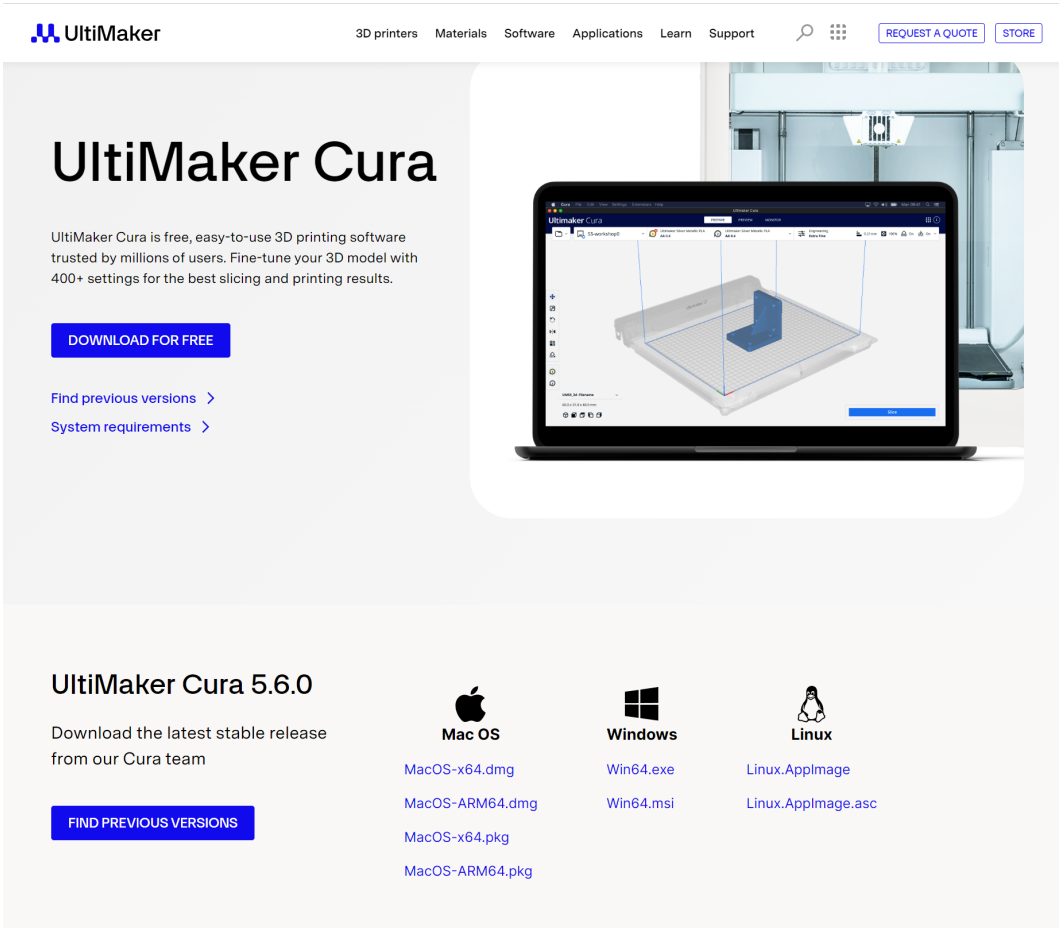
Ultimaker Cura is a free slicer that will process digital files for 3d printing, please click on the web link above to navigate to the downloads.

Account is NOT REQUIRED to download Cura

Click on the installer that's compatible with your device

After the download has completed, run the installer

Follow the Installation wizard to complete installer



Checking Geometry

To ensure a successful print, please check the following for the geometry that you wish to print:

Select your geometry to check if your model is enclosed. If the command displays "_ open _____ added to selection" perform necessary edits to the geometry to enclose it.

Type "Show Edge" in the command bar to prompt the Edge Analysis Window to check for any naked edge.

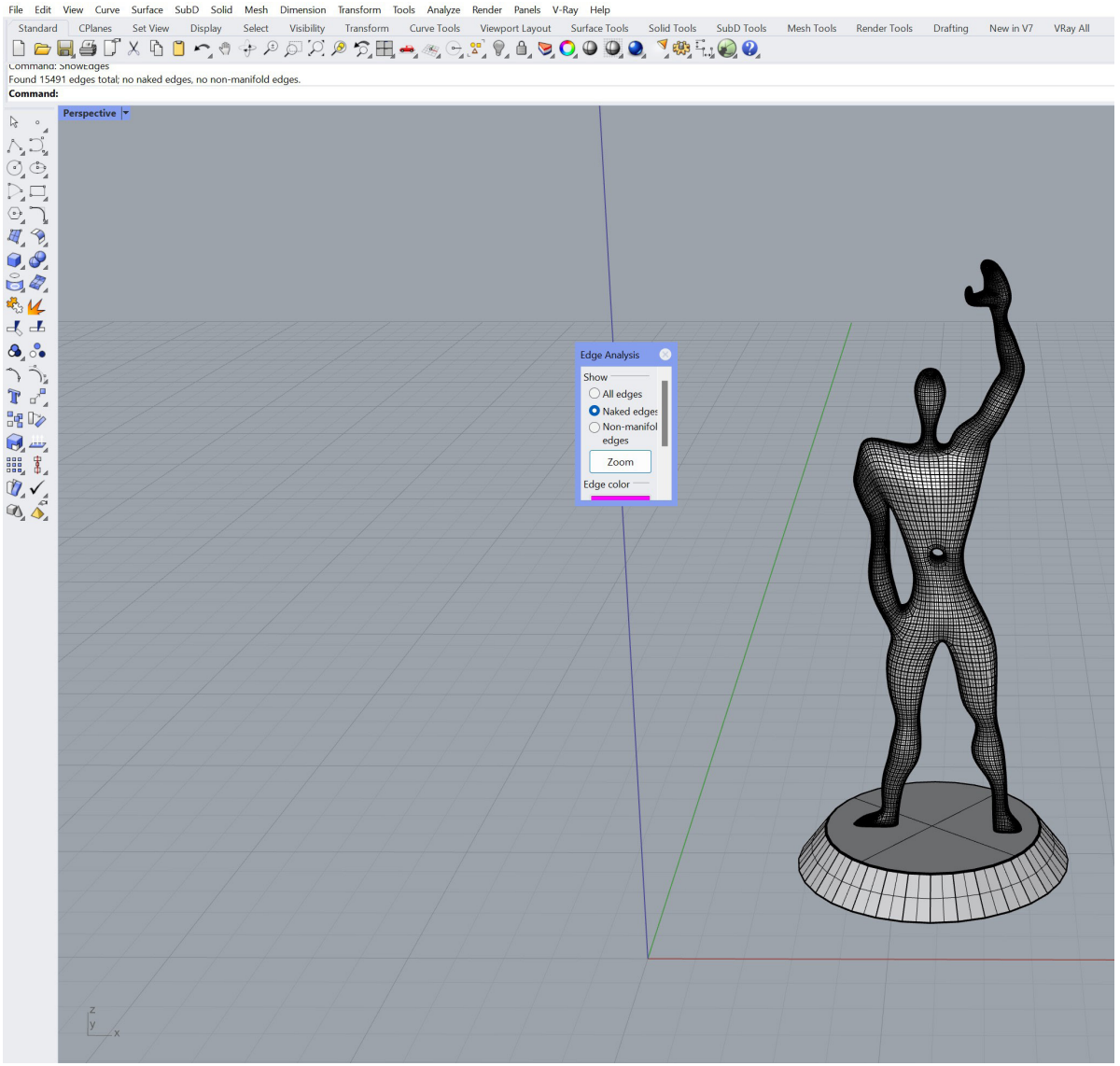
If your geometry has no unclosed-edges, the command bar will return the following: Found (#) of edges total; no naked edges, no non-manifold edges.

If this is the case, proceed to:
Generating the Mesh and Exporting File

If the command tab says the geometry has naked edges:

Select NakedEdge option in the prompt window to check for unclosed edges.

Recreate the surface so there are no naked edges.



Convert to Mesh

To export the geometry, first it needs to be converted to a mesh

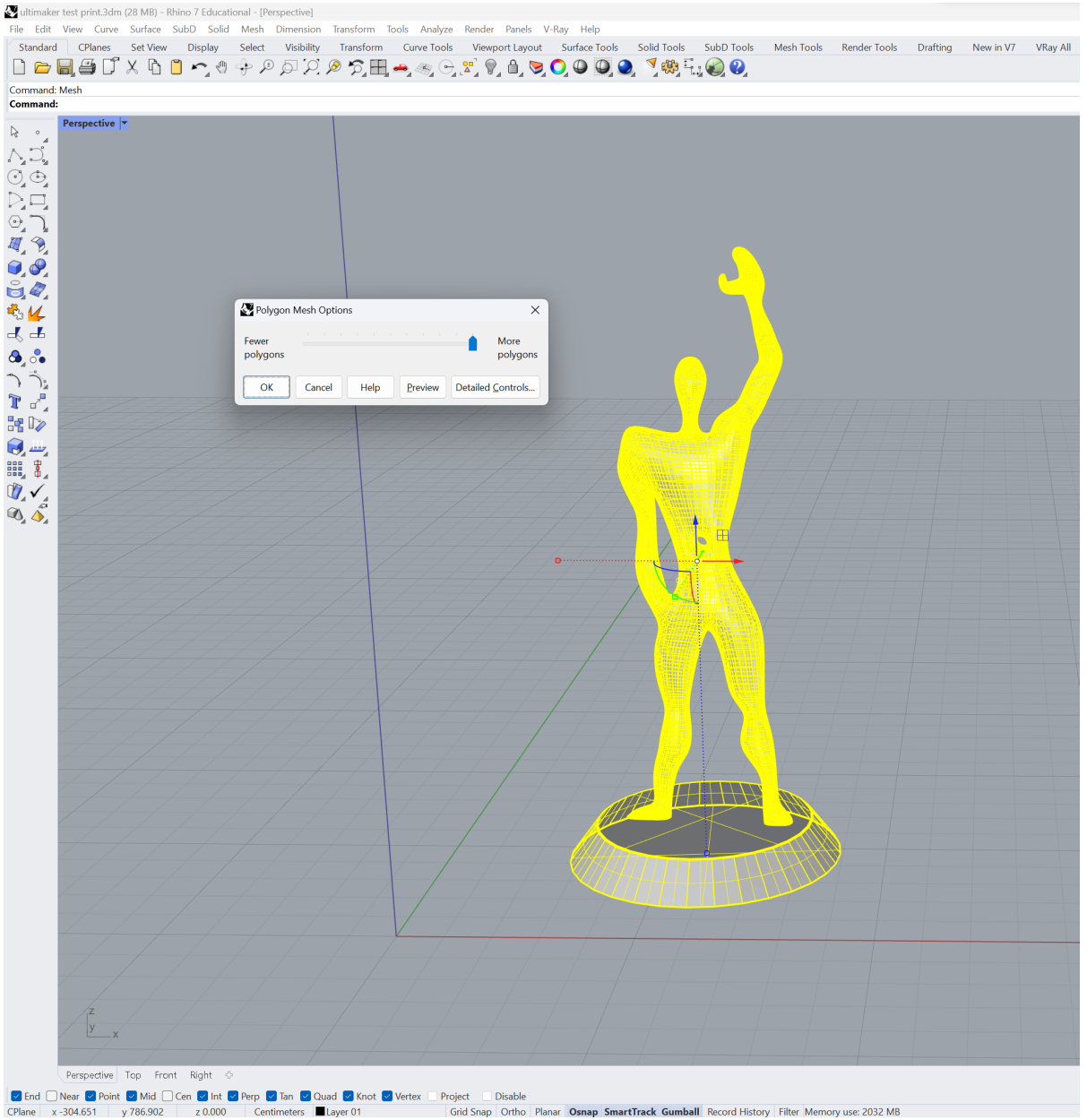
Type “Mesh” in the command bar, and select the geometries you wish to print.

After pressing Enter, the Polygon Mesh Options window will appear.

Drag the sliding bar all the way to the “More Polygons” option, and hit OK.

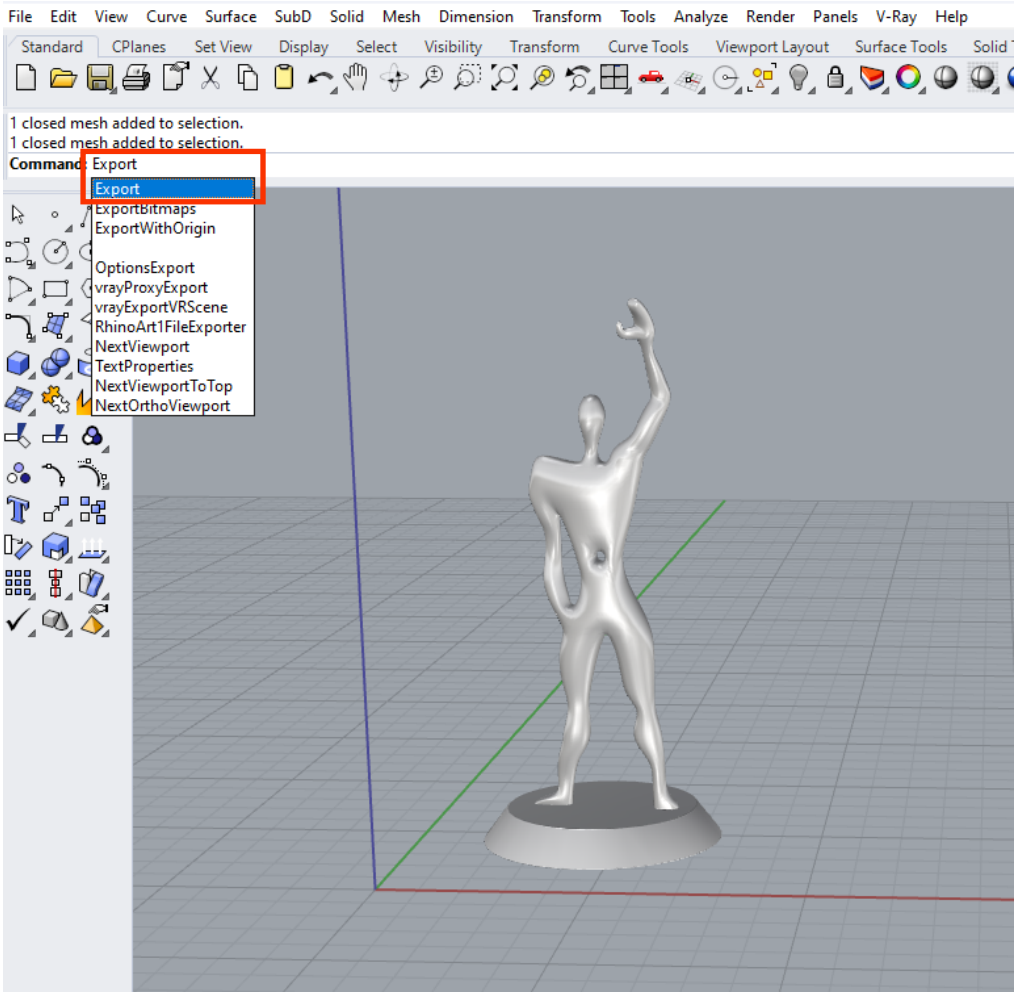
Move the Polysurface out of the way using the gumball to reveal the mesh.

To differentiate between the two. Under default “shaded” display settings, the mesh geometry will be either completely smooth or filled with isocurves, where as the polysurface will display surface edges.



Exporting file

Both format of STL and 3FM are supported formats for Cura to process, STL being the conventional method while 3FM slowly becoming the industry standard.



To Export for print

Select the converted mesh geometriesin rhino, and type “Export” in the command bar.

And you can choose from the two supported file formats.

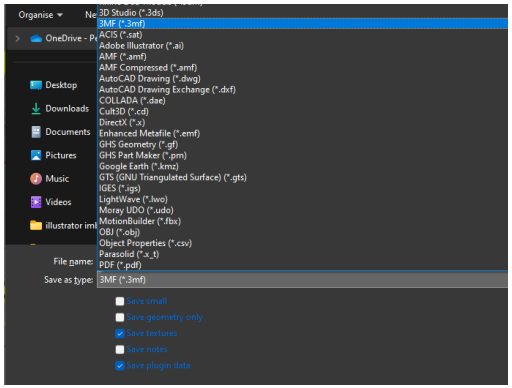
Exporting file cont.

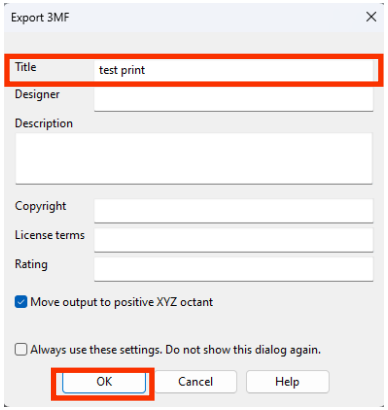
3FM

Select the converted mesh geometriesin rhino, and type “Export” in the command bar.

The file should be saved in the 3FM format, and after hitting save, the 3FM Export Option window will appear.

Name project title the same as your file name , and click OK.



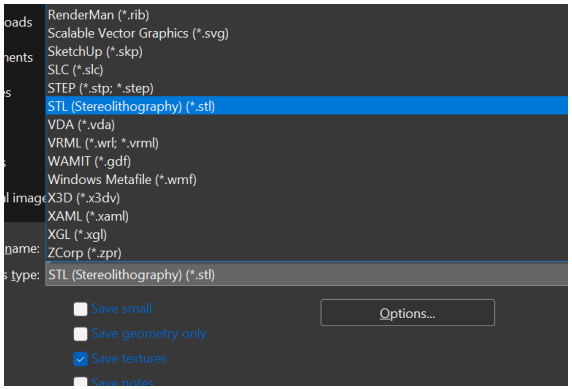


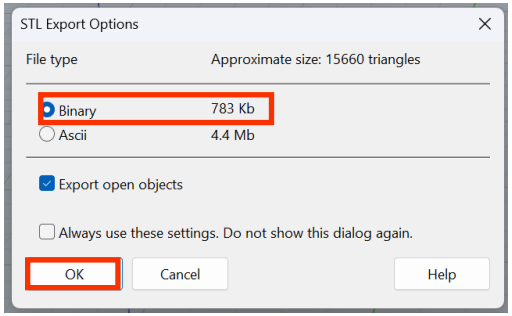
STL

Select the converted mesh geometries in rhino, and type “Export” in the command bar.

The file should be saved in the STL format, and after hitting save, the STL Export Option window will appear.

Make sure to choose the Binary Option, and click OK.





Ultimaker Cura: Workflow

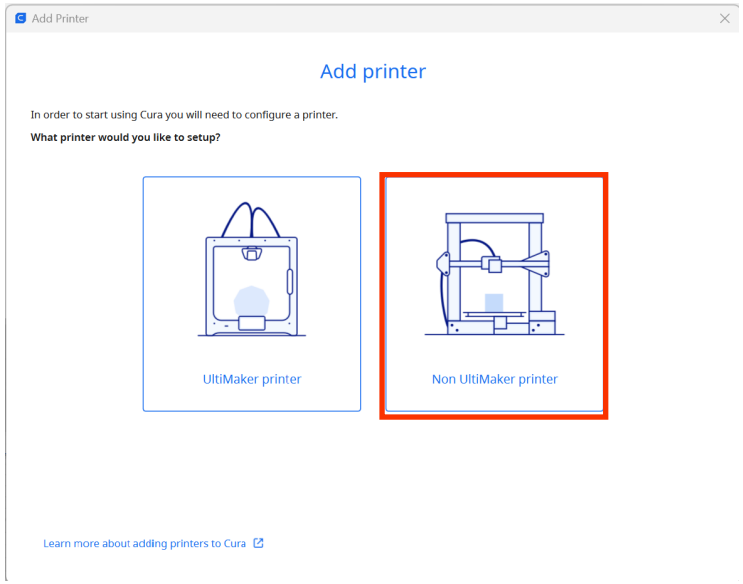
Setting the Printer

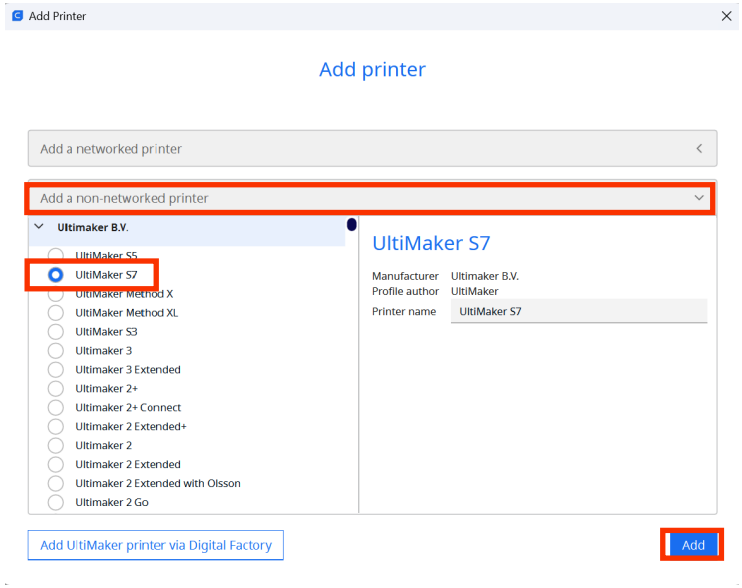
When first launching Cura, you will need to set up the printer configuration to match the machine we have here at DFAB.

This process is likely only required once, and your device will remember this setting. However, should you find the need to do this again, please refer to the step in this section.

Click on the Printer Settings drop-down on the top left of the UI

Add Printer > Non-Ultimaker Printer > Add a Non-Networked Printer > Select Ultimaker S7 > click Add





Import File

Files can be imported either using the File Import Icon on the top left of the UI to select the STL or 3FM file in folders.

Or

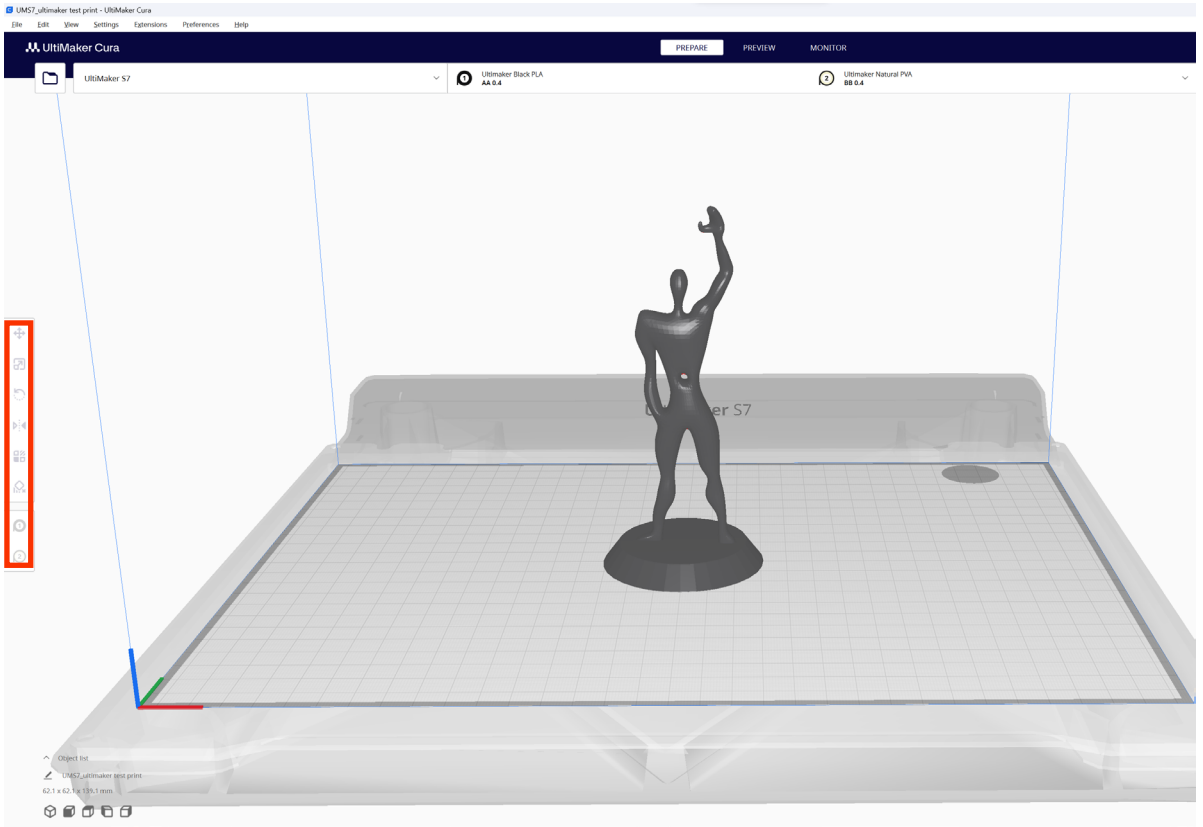
Simply drag the STL or 3FM file with the cursor inside the print plate in the model preview of the UI

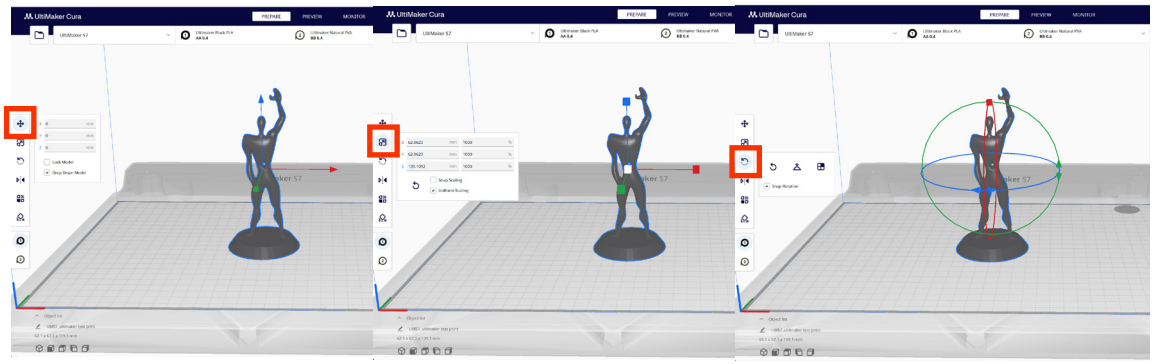
As displayed on the bottom left corner of the UI, Cura's base unit are in MM, make sure when creating the geometry in Rhino, the units are set to MM in properties.

Otherwise, you also have the option to scale the geometry in Cura via Model Setting located on the left of the screen.

First select the geometry you wish to alter, then you can either choose to Move, Scale, or Rotate the geometry.

There are other tools such as mirror, per model setting, and support blocker which you can use, but it's likely that you won't ever need to use them.





Slice Preview

When reach to the end of the workflow, slice the model via the Slice button located at the bottom right of the UI.

The slicing will take a few second, but if the print contains complex geometry or large volume of prints, this process will take slightly longer.

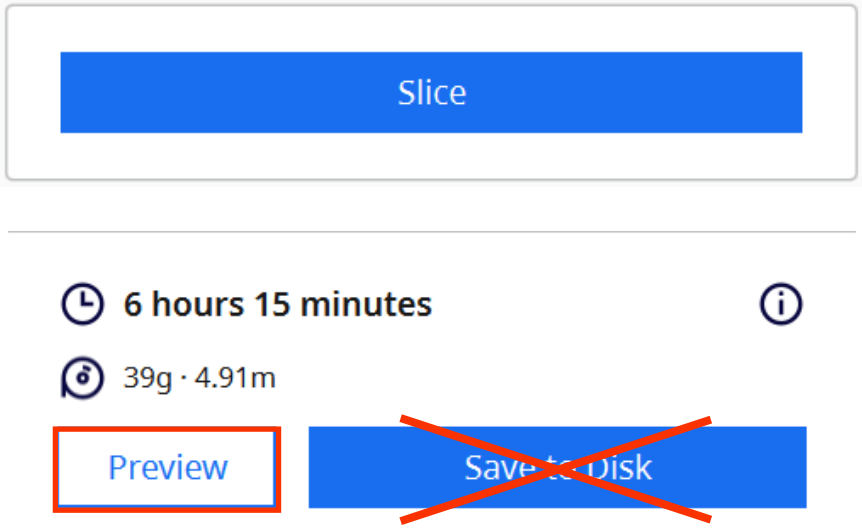
After slicing has complete, the estimated time and material needed will be displayed in the box that takes place of the Slice button in bottom right.

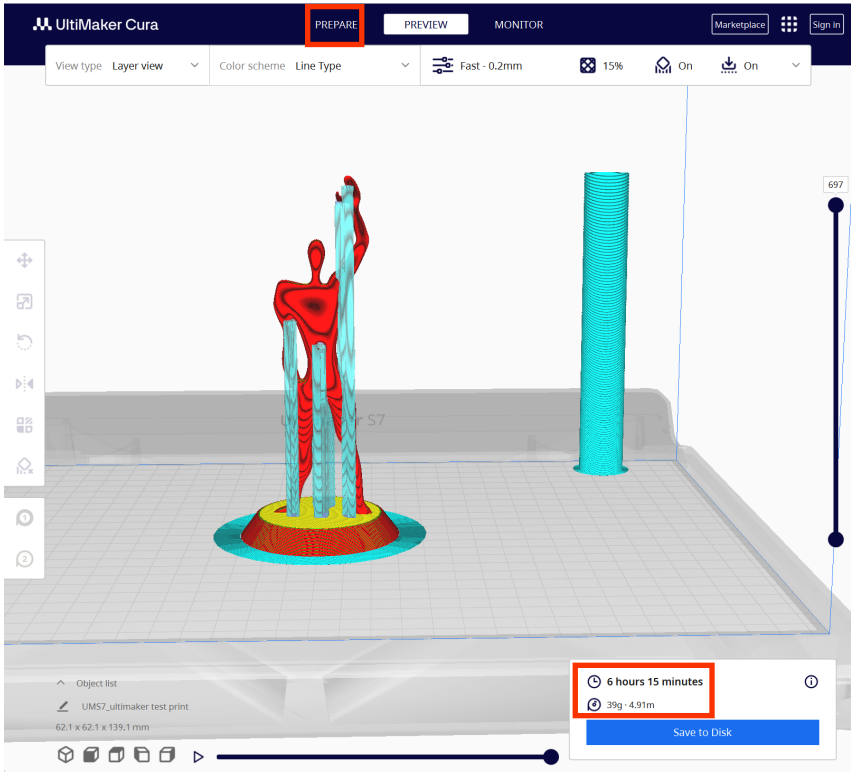
You may also click on Preview to see how the printed geometry, and the support structures.

If you wish to change certain settings, click on Prepare and make necessary changes.

The geometry will need to be sliced again for Cura to process your file.

DO NOT save your file via Save to Disk, for the correct saving process, please refer to Save for Submission





Ultimaker Cura: Print Setting

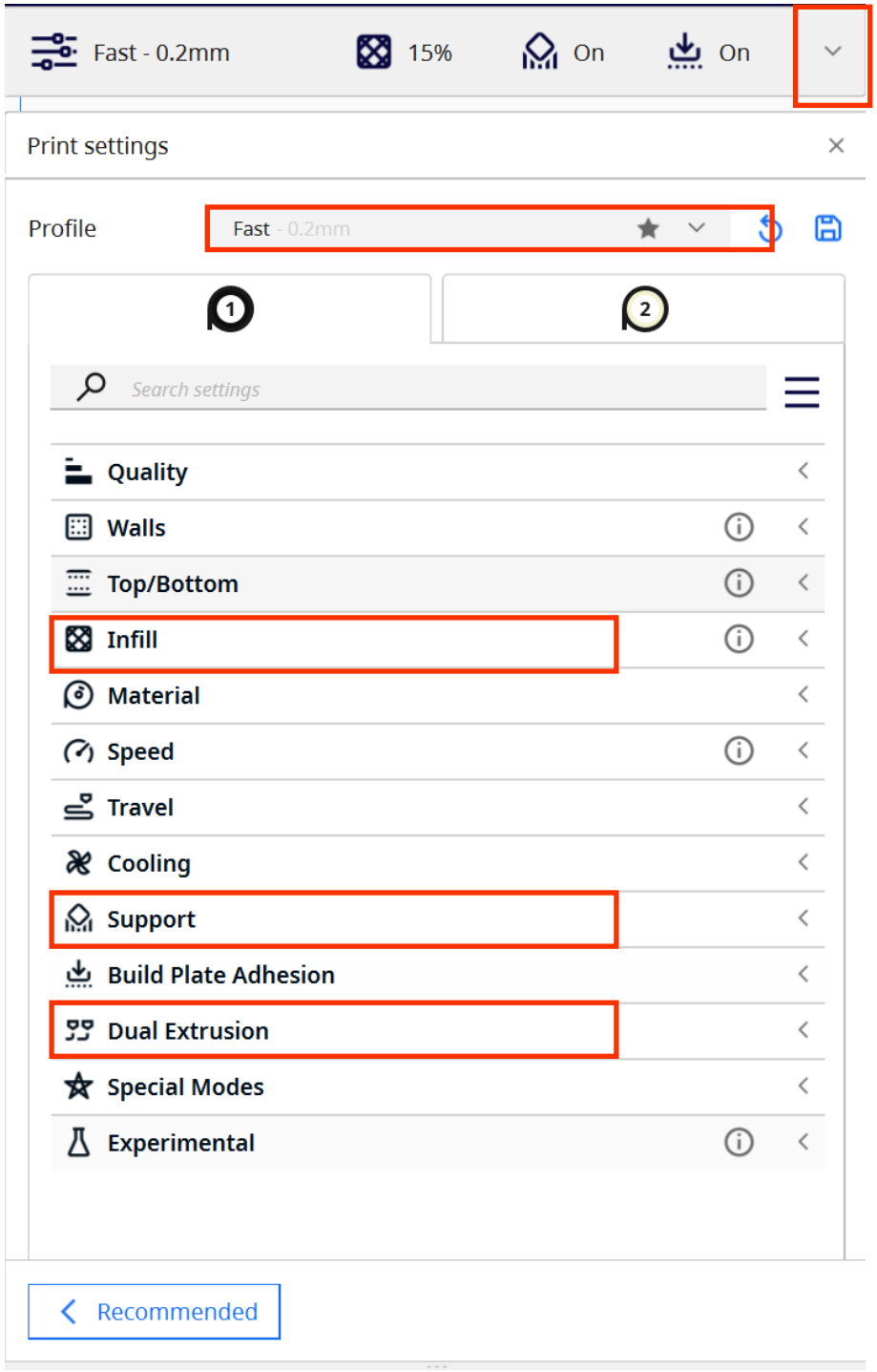
There are additional settings which are necessary to improve your file for printing in many ways; such as reduce the amount of time to print, reducing the cost, enhance the structural integrity your geometry, etc.

If the drop-down is not displayed on your UI, simply click the drop-down arrow

There's a whole list of settings to choose, but there will only be a few section that should/can be changed:

Material
Profile:Print quality
Infill
Support
Dual Extrusion

Others are better left to the default setting.



Printing Material

The materials can be used to print are Black and White Ultimaker Tough PLA.

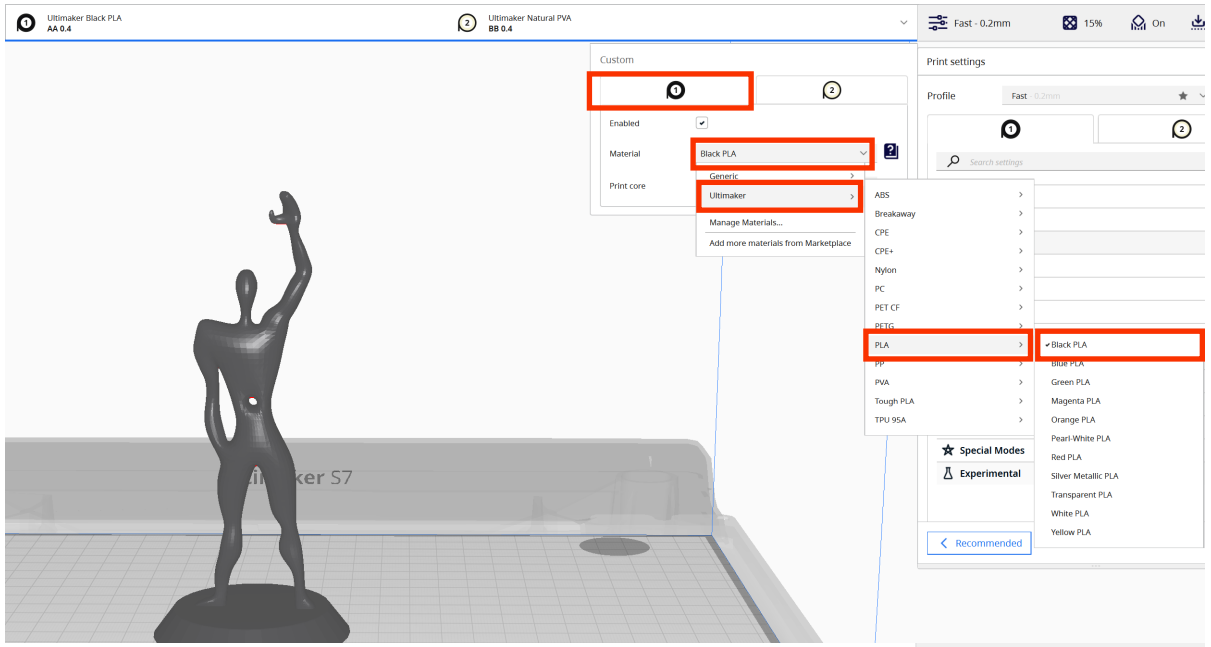
Additionally, there's also the PVA* material which is exclusively used to print supports, as they can be melted away during post print treatment offered at DFAB.

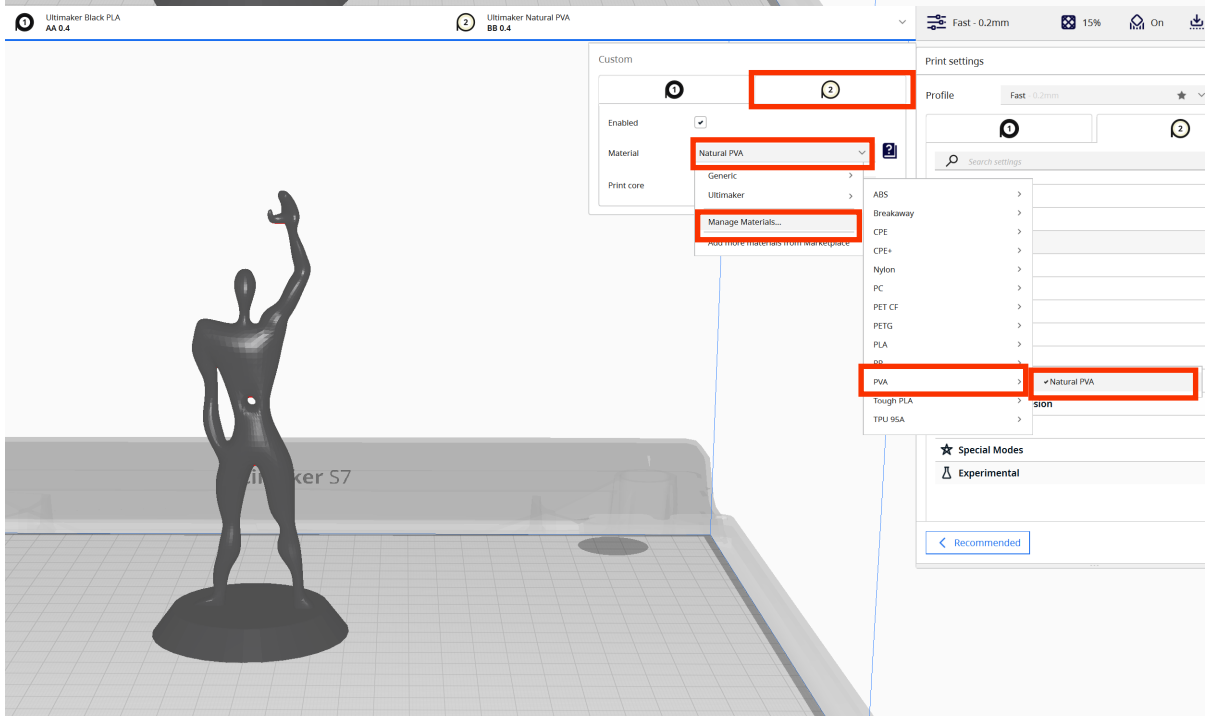
These material can be chose from the Extruder Settings drop-down.

The example image shows the process to load Black PLA into extruder 1:
Click spool 1 > material > ultimaker > PLA > Tough Black PLA, and set the print core to AA0.4

To load PVA for supports:
Click spool 2> material > ultimaker > PVA > Natural PVA, and set the print core to BB0.4

*The PVA option will require additional settings in the Print Settings drop-down





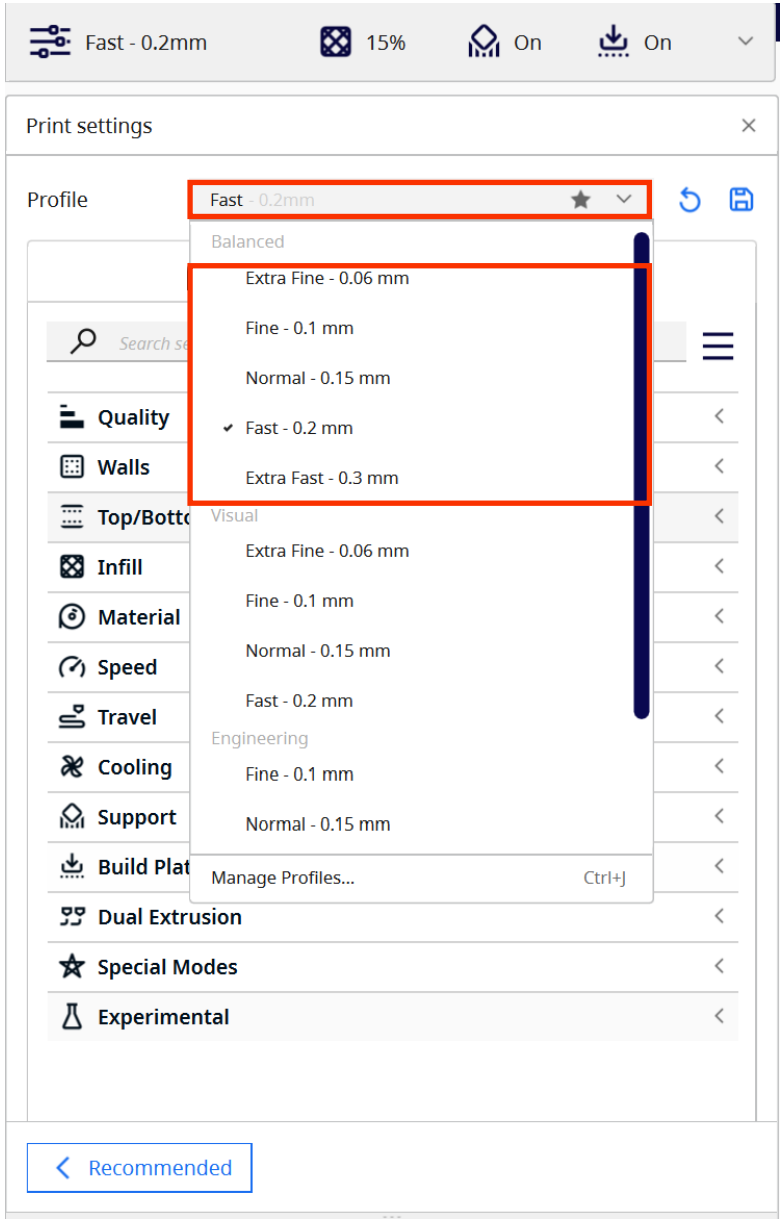
Profile: Print Quality

Choosing a profile simply means to choose the quality of the print.

Click on the Profile drop-down, there are options to choose from with varied layer height.

By selecting a thinner layer height, the completed print will be finer in detail, but the printing time will take longer.

The default is recommended to be set as 0.2mm, this setting is quite balanced in terms of cost, time, and resolution.



Infill

Infills setting controls the amount of filament used vs the structural integrity of the print.

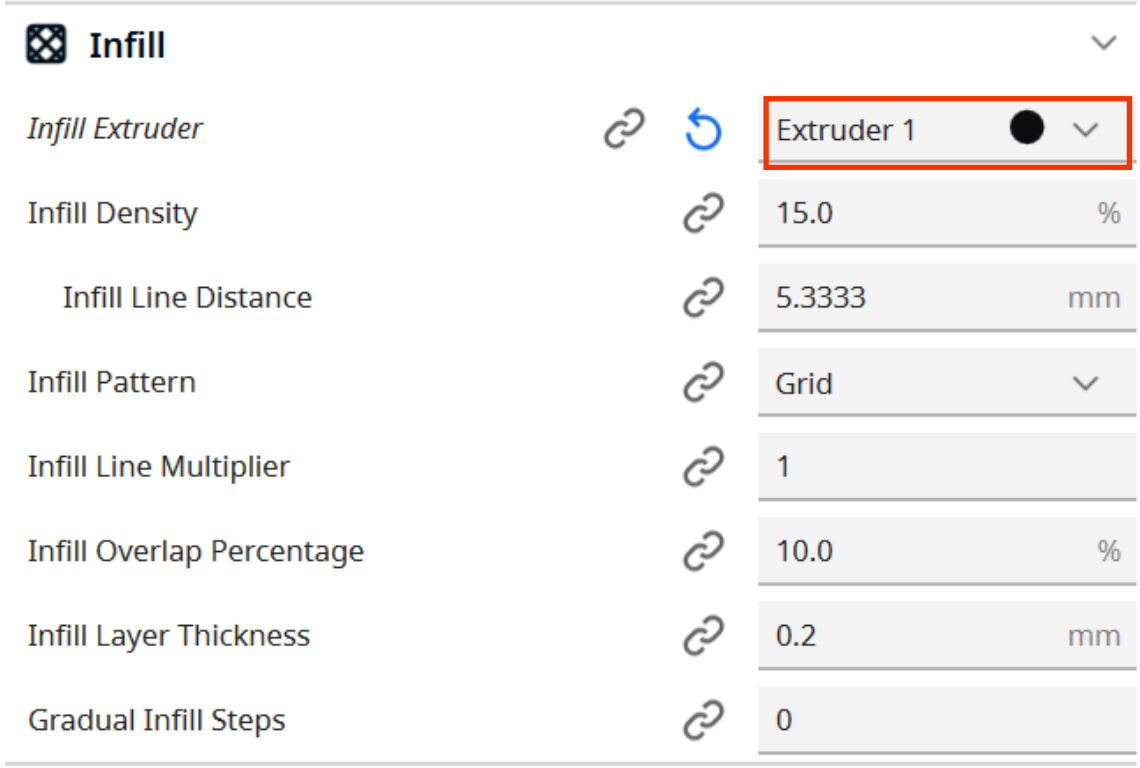
Always select the same extruder that's printing the PLA plastic, NOT the PVA support.

A solid infill will be intuitively stronger, but will require more material, meaning more cost.

In the context of making a architectural model, a 15% infill density will suffice.

Make sure the extruder matches the material loaded for printing (PLA, spool 1)

Infill pattern is another option that can be altered, different pattern will vary in printing time, amount of material required. The recommended option for the infill pattern is Grid.



Support

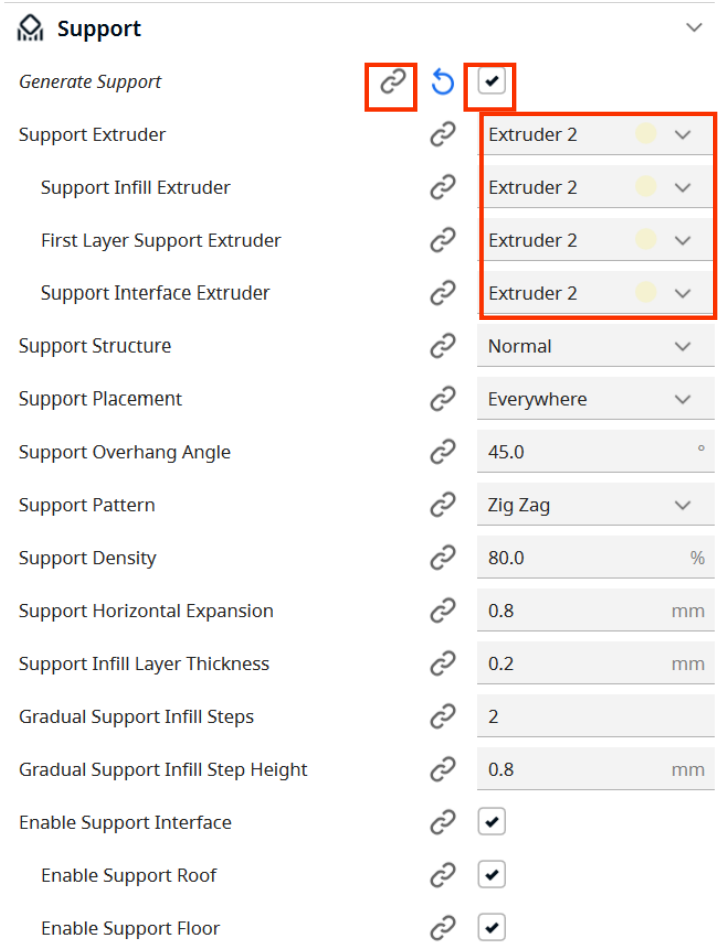
Some geometry with overhangs will require supports, but Cura can automatically determine where supports are needed.

Simply check the Generate Support Box, and click on the Link icon, Cura will configurate the support setting itself base on the imported geometry.

If you want to print the support as PVA which can be melted away during post process, support extruder must be changed to Extruder 2.

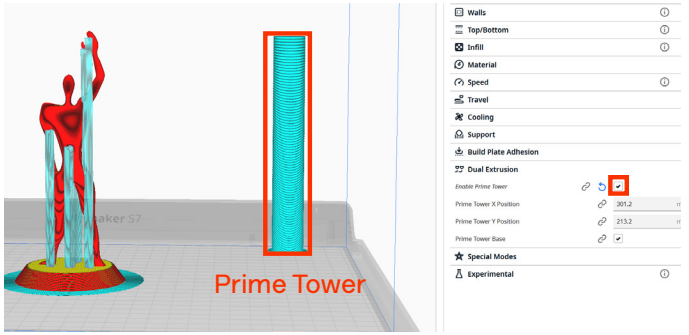
Sometimes depends on the geometry, supports can be easily peeled away, meaning they can be printed as PLA. To print the support as PLA, change the extruder to Extruder 1.

See next page for a comparison between the two options.



Dual Extrusion

If dual extruder is selected, ie printing with both PLA and PVA, the prime tower needs to be enabled to purge materials when alternating between extruders



Support cont.

Melt away vs Break off

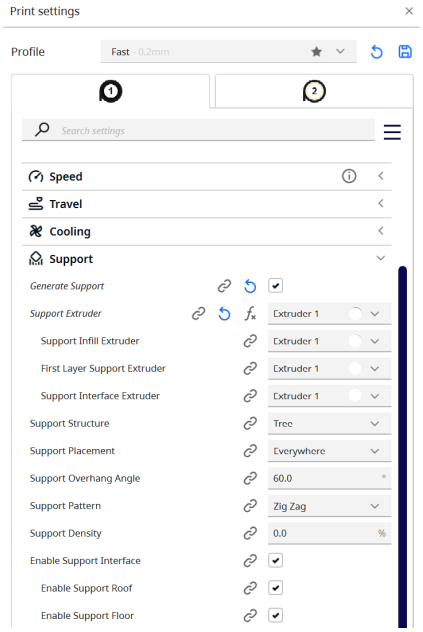
As previously mentioned, the support structure can be printed in PVA material that can be melted away.

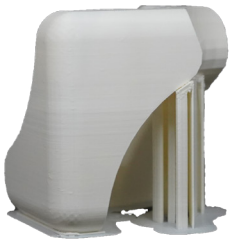
Or

It can also be printed in the same PLA material as the model itself, and can be broke off with tools or even your bare-hands.

Here's a simple typology model to better demonstrate the difference bewteen each support their generation process.

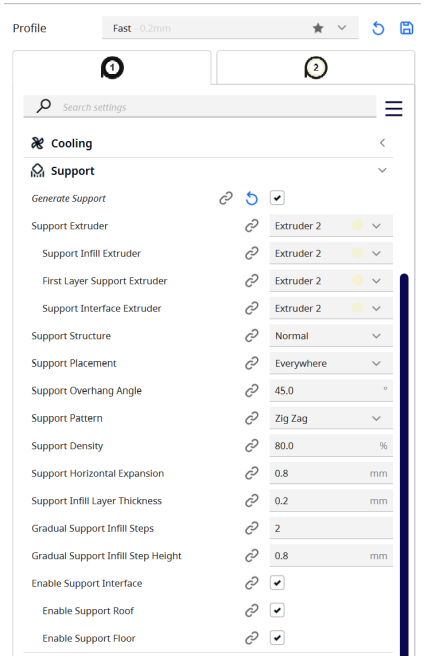
Not all geometry requires PVA, sometimes print the support and geometry in the same material can be a better choice based on cost and time.





Model: White Tough PLA

Support: White Tough PLA



Model: Black Tough PLA

Support: PVA

Print Esimate

Slice the geometry when ready, and the estimated time and material required will display on the bottom left.

PLA Black & White(model)	0.82/m
PVA (support)	1.63/m

Refer to the cost of the material/m, you can calculate the cost of your print.

If the cost is beyond your expected value, refer back to sections in the Print setting to find methods to reduce the cost. Whether its reduce the density of the model via the infill setting, or reduce to overall quality of the model via Profile layer height.

Save for Submission

When satisfied with all settings and configuration, save your file as a Cura project:

Tool bar > File > Save Project > file convention 3MF

All files are to be submitted via FTP and an email should be sent to

3dprinting@daniels.utoronto.ca

notifying the DFAB office there has been a new submission.

Always include in the email

First and Last name of the student
File names

* Failure to do so will result in your print not be processed and queued for printing

Submission: File naming convention

Copy and past your .stl AND .3mf files into the newly created folder, named as:

lastname_firstname_date

with following format:

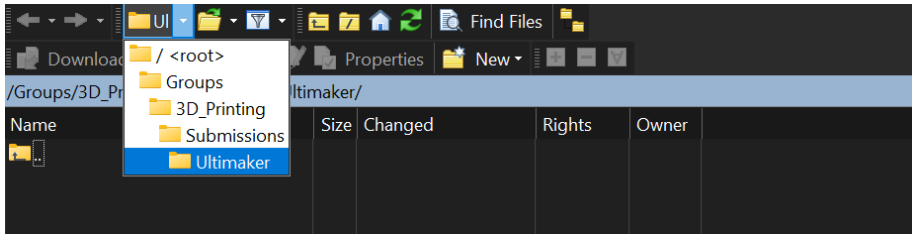
lastname_firstname_date.stl
lastname_firstname_date.3MF

Both files must be present for file submission or the print will not be processed.

Upload to submission folder

Upload the entire folder to the submission folder on FTP:

roots > Groups > 3D_printing > Submissions > Ultimaker



You can follow the instruction on how to download the FTP on the Daniels Website

https://www.daniels.utoronto.ca/search?s=ftp

End Of Manual

Go back to Contents