How to 3D Print an STL File

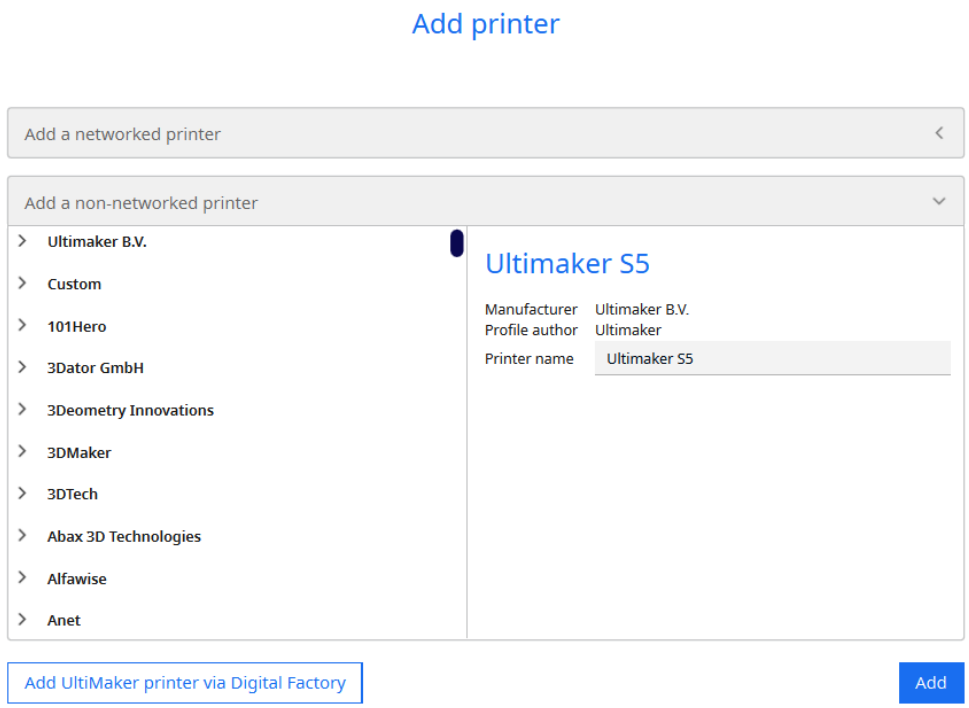
Download the sculpture

Download the STL file from our website for the sculpture you want to print.

You will also need to download a Slicer software if you don't already have one. Likely your 3D printer has a recommended slicer program that you can use. If you don't have one, we recommend UltiMaker Cura slicer software (https://ultimaker.com) as it is easy to navigate and as straightforward as 3D printing gets. This guide will be based on UltiMaker Cura but the settings referenced here should be available in a different software, just perhaps under a different name.

Slicing the STL

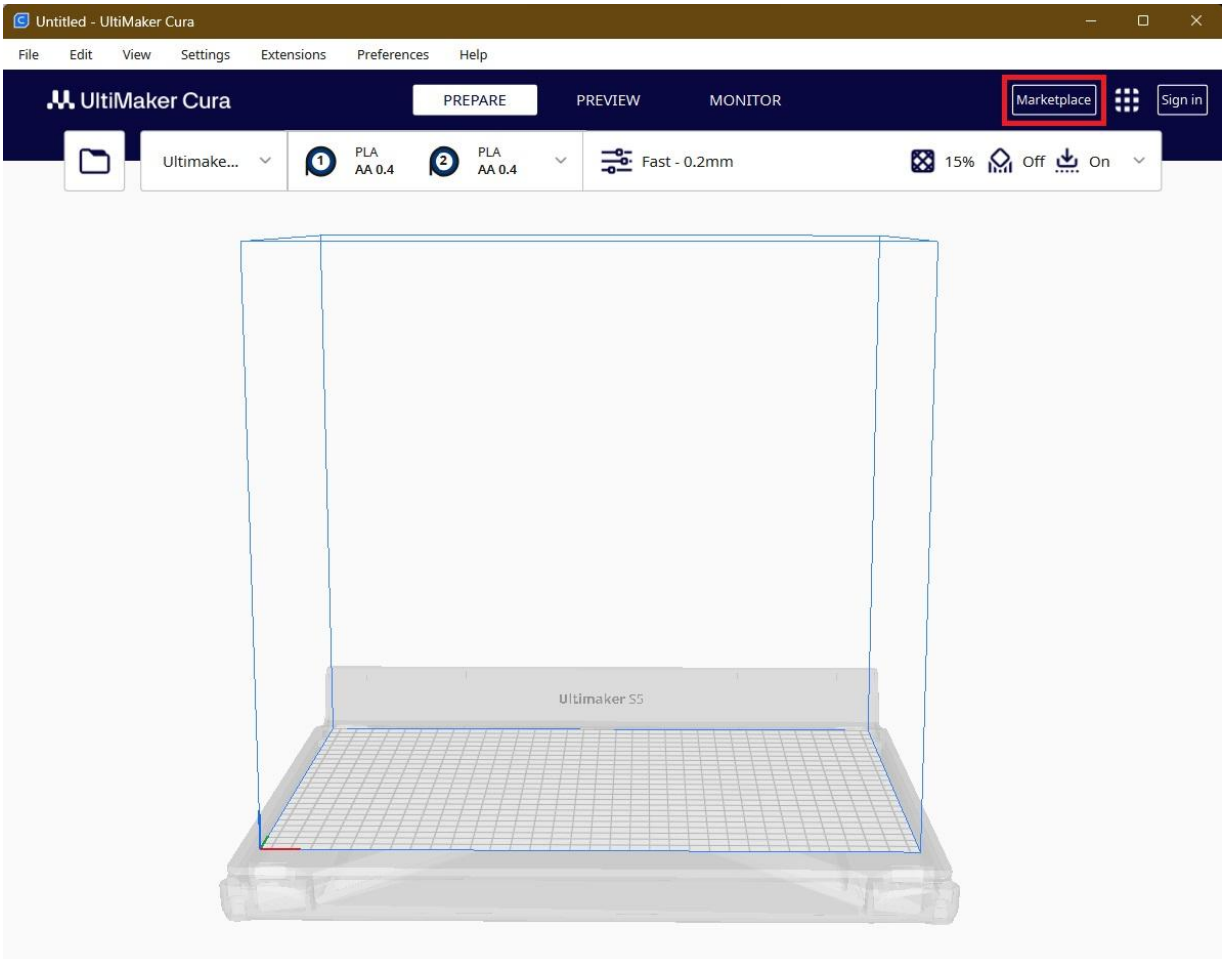
If you've downloaded the slicer software for the first time you'll need to add your 3D printer. In Cura it will prompt you to add your printer. Click the "Add a non-networked printer" list and scroll until you find your printer.



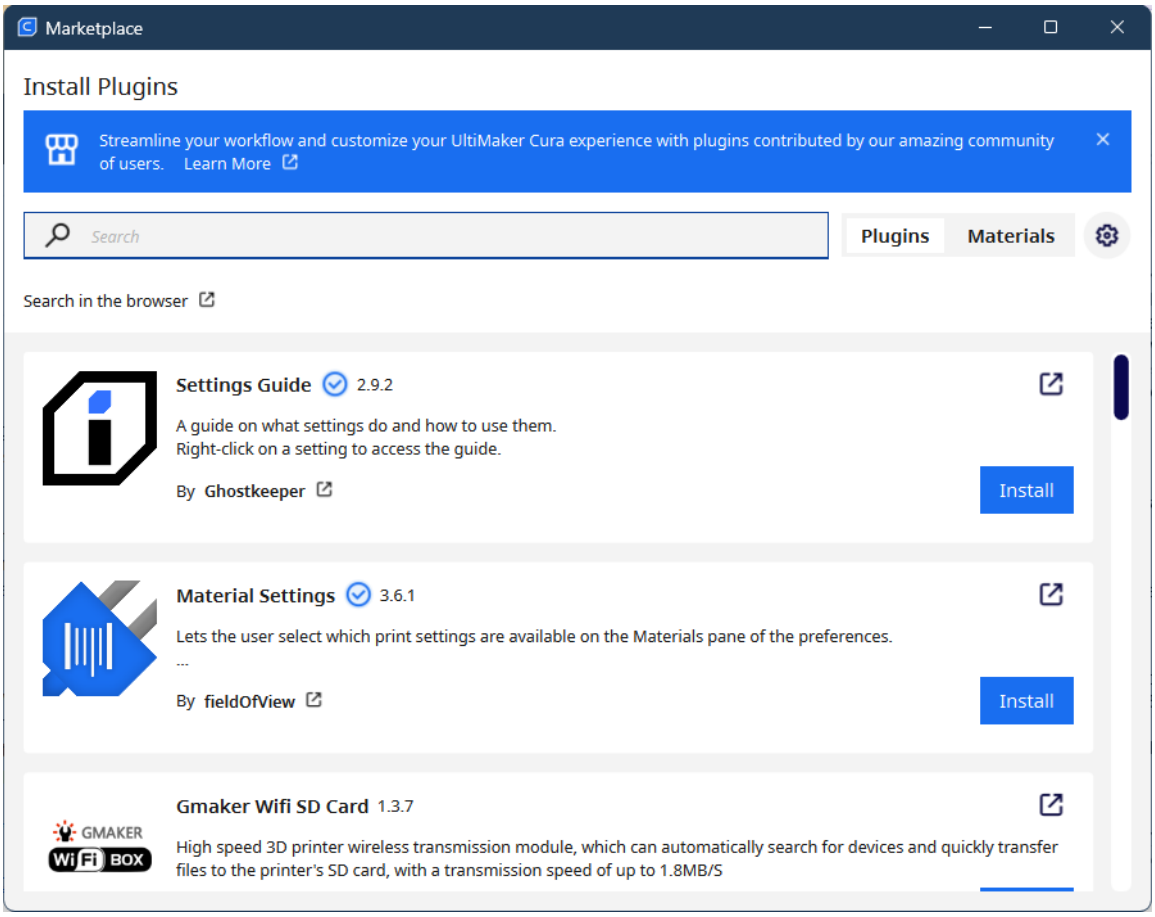
Adding Your Printer

Skip to the next section Importing the STL if you found your printer.

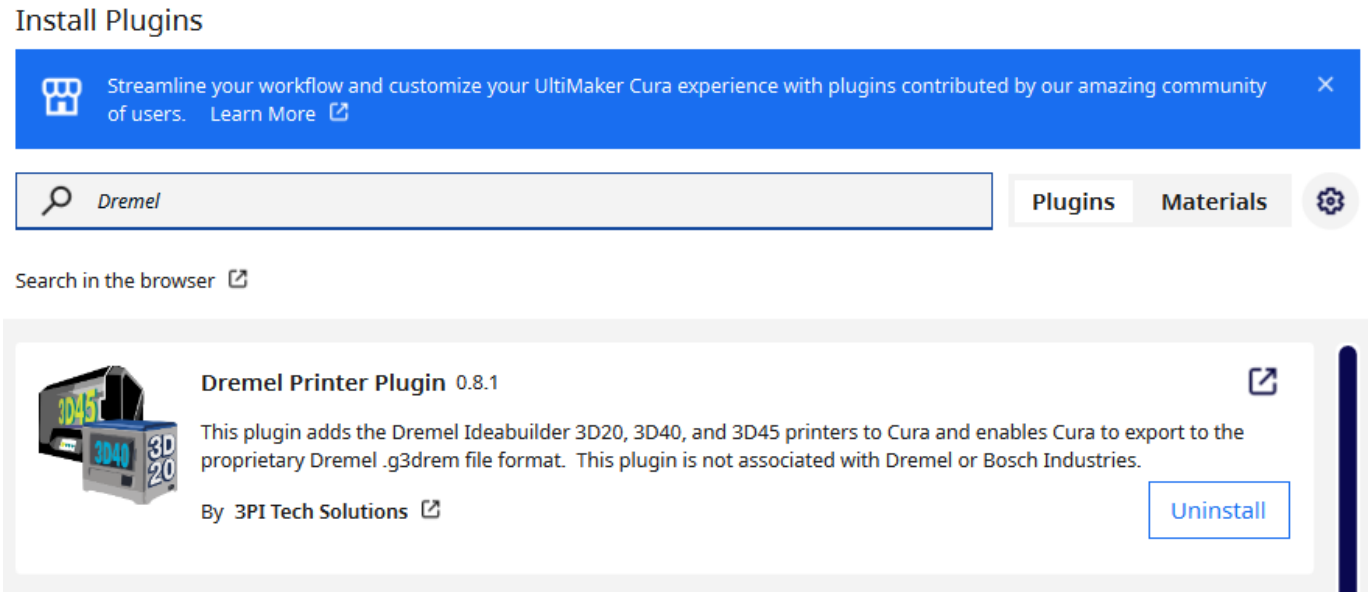
If you don't see your printer listed, choose the UltiMaker S5 and click Add. Once you are at the main Cura page there will be a Marketplace button at the top right of the screen as indicated by the red rectangle.



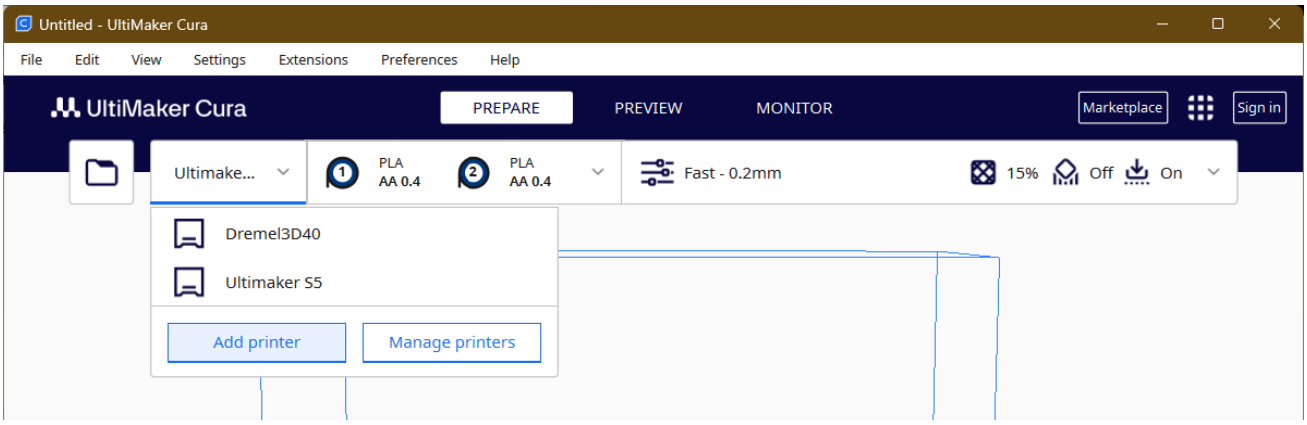
Click the Marketplace button and a window will open. Type the name of your printer in the search bar and you should be able to find a plugin for it.

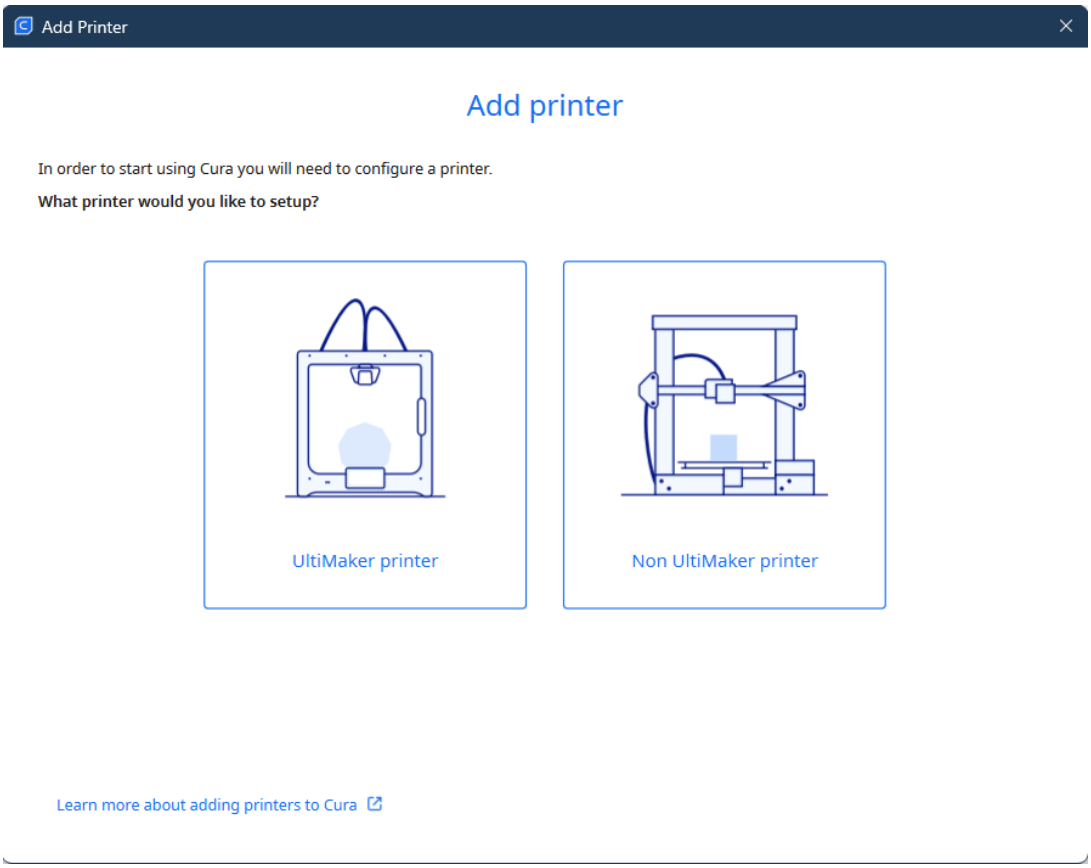


Make sure the plugin description says that it adds the printer to Cura. For example, we use a Dremel printer, and this is the plugin we use. Install your printers plugin and close the Marketplace window.

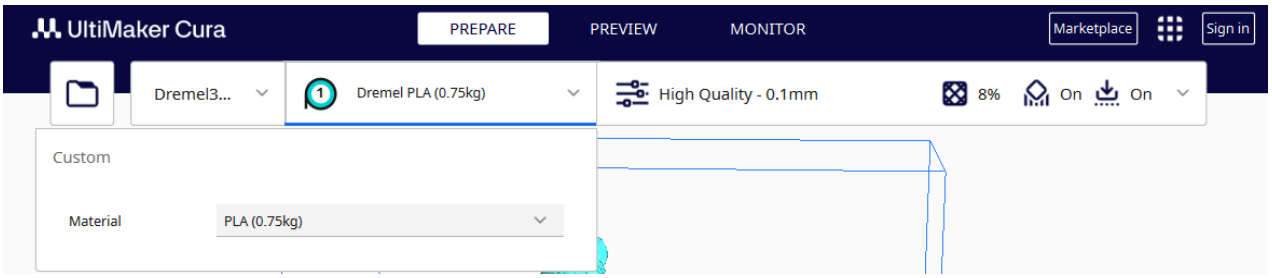


Back on the main window of the software, click the dropdown menu at the left side of the screen with the 3D printer name picked earlier. Choose “Add Printer”, and then choose “Non UltiMaker Printer” in the window that will open.



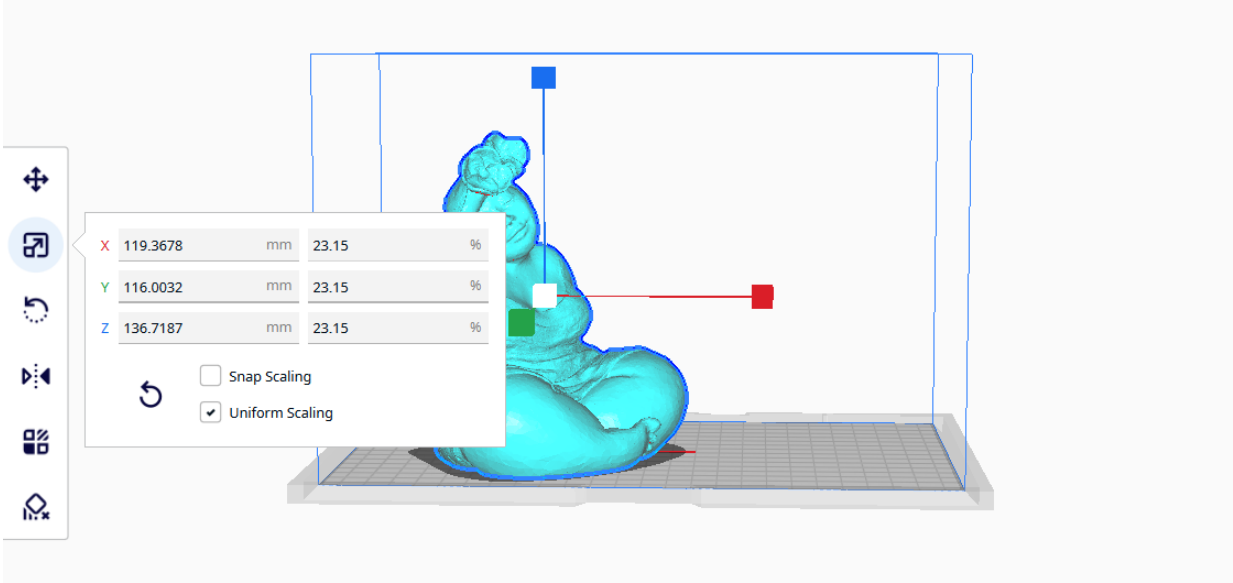


You should now be able to find your printer among the listed ones under “Add a non-networked printer”. Once you’ve added your printer make sure that the material selected is the one loaded in your 3D printer. You can change this by clicking the dropdown menu next to your printer’s name and then clicking “Material”. There should be different options depending on your 3D printer. If you don’t know what filament your 3D printer uses, check the sides of the box the filament came in. It’s likely that it is PLA.



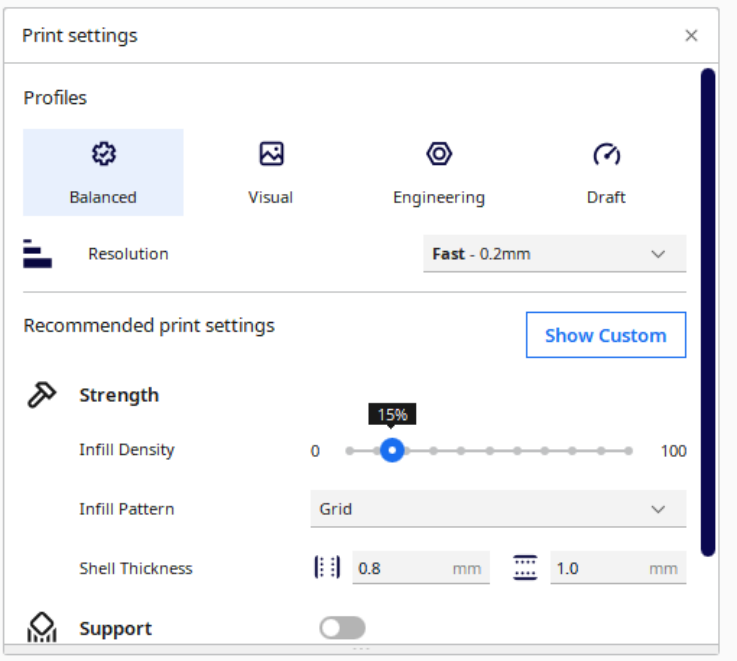
Importing the STL

Click the little file icon at the top left next to the name of your printer, and choose the sculpture STL file. Opening the STL file in the slicer program will give you the opportunity to resize the sculpture and position it on the buildplate. To resize it click the sculpture and then the second icon along the left side of the screen, called “Scale”. Make sure that “Uniform Scaling” is checked. You can now either drag one of the coloured handles to resize it or you can simply type the size in millimeters into one of the boxes and hit Enter. Once you have it at the size you want, right click the sculpture and choose “Center Selected”.

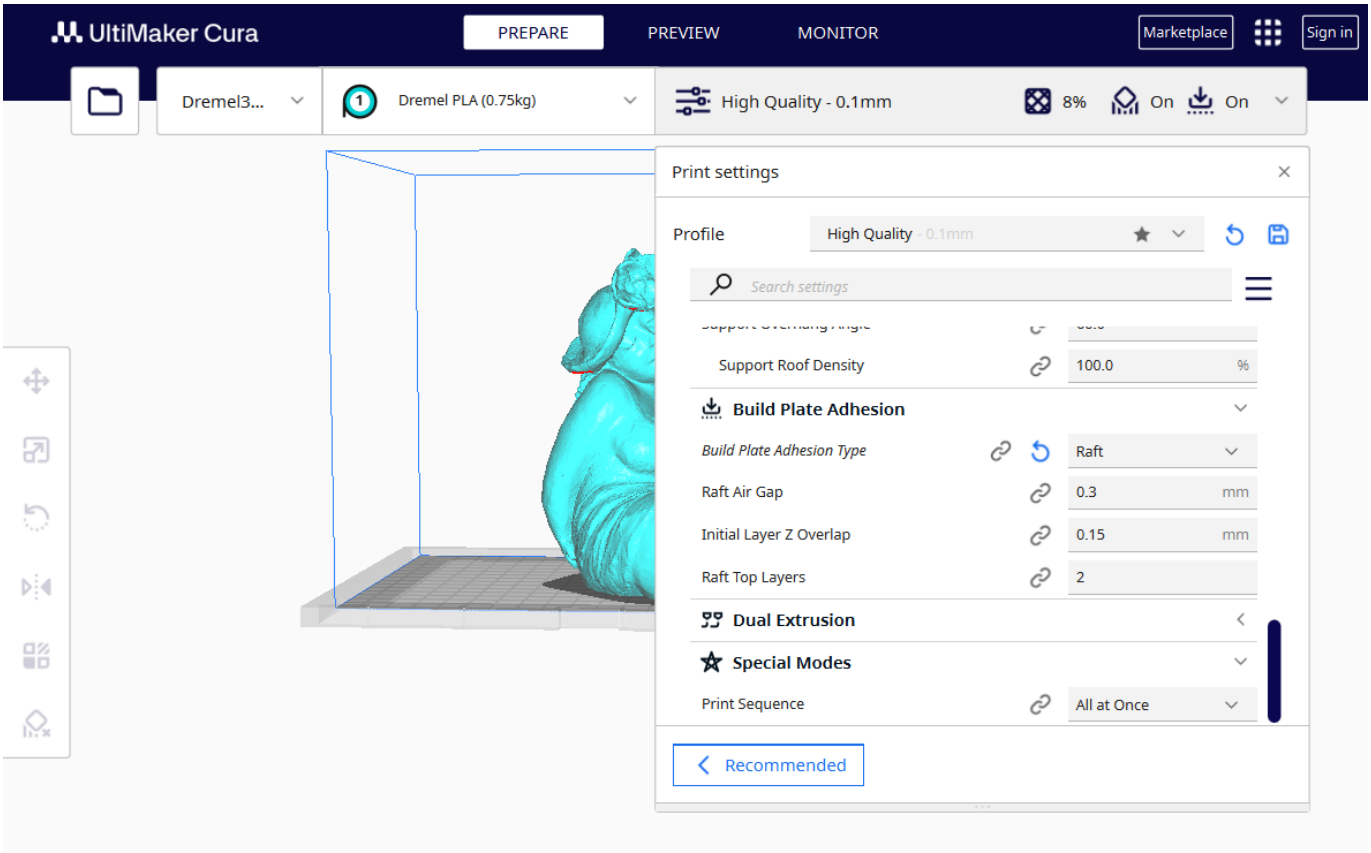


The sculpture should fit within the blue wireframe outline that indicates the maximum dimensions your printer can print. If it doesn’t, reduce the size of the sculpture.

Now you’ll need to change the print settings. This can get complicated but we’ll walk you through what you need to do. The drop down menu is at the top right of the screen. Click that bar and the “Print Settings” menu will open. Click the blue “Show Custom” button.



Your Print Settings should now look like this:



Profile/Quality
You can choose from the different Profiles available that will automatically determine a lot of the settings for you. For a good quality the print layer height should be between 0.1 and 0.2mm. The exact size will depend on your printer but as you can see next to “Profile” the software tells us that High Quality is 0.1mm. Choose the print quality you would like to use.

Walls
If you decide to print the sculpture hollow we recommend 3 walls but 2 should work okay if the print is smaller than 150mm. This is a good choice for decorative models that will be lightly handled.

Infill
This determines if the sculpture will be hollow or not, and also determines how much filament you’ll use. We typically choose Lightning as the pattern with a density of 10% to 15%. This lets the software figure out where internal support should go and takes less filament.

Support
We highly recommend making sure you have some type of Support selected. Change the option of “Support Structure” to Tree. If you don’t see “Support Structure” as an option, type “Tree” into the search bar under “Profile” Change the setting from Normal to Tree. Clear the search bar.
We typically choose Touching Buildplate to ensure that the supports are easy to remove from the finished piece.

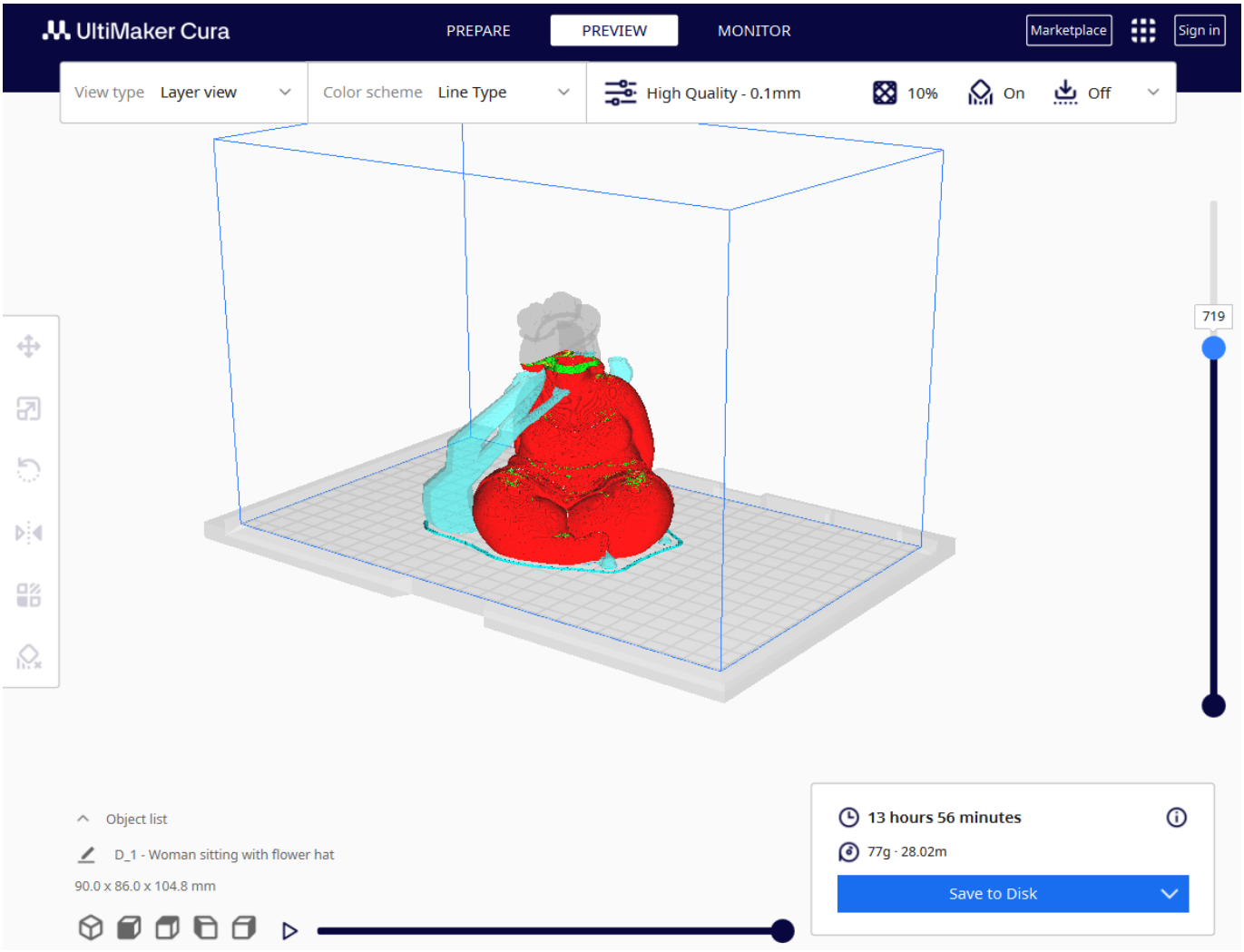
You can adjust the angle at which the supports will generate. We have found that 60° to 70° seems to work well, though if parts of the sculpture seem to be drooping you might find that reducing the angle to 50° solves the issue.

Build Plate Adhesion

We usually choose Skirt as the “Build Plate Adhesion Type”. If you find that your sculpture doesn’t seem to stay attached while printing you may want to select Brim or Raft instead. If that doesn’t help try lightly rubbing a gluestick onto the buildplate. This will make the surface a bit more tacky and gives the melted plastic something to hold on to. Make sure to wash the dried glue off between print jobs.

Finishing Touches

Once you’ve changed these settings, wait for the loading bar at the bottom right to finish. Cura is slicing the STL. Don’t worry if this takes a while. Once finished you should be able to see the supports that have been generated (shown here in light blue), the total number of layers the print will take as a slider on the right side, as well as the print time and total filament on the bottom right. You can drag the right slider to see how the printer will build the sculpture.

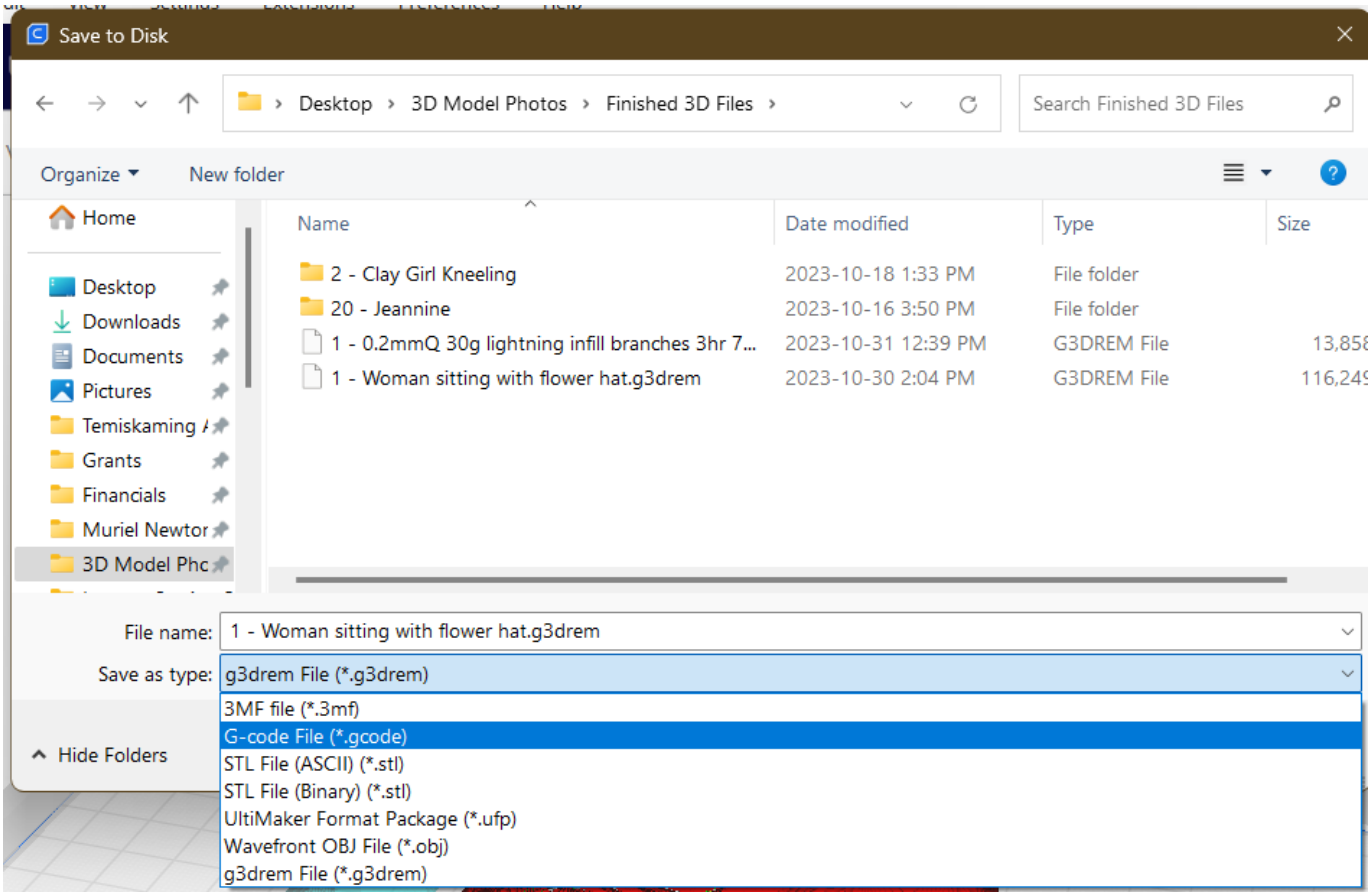


Pay attention to the amount of filament that will be used, as you don’t want your printer running out part-way through the print. There are a few things you can do to make sure this doesn’t happen:

- If you're unsure you can load in a fresh container of filament into your printer which should be more than enough;
- You can click the model and scale it down further. A smaller model will also reduce print time.

Printing

Click the blue "Save to Disk" button at the bottom right of the screen, or the little arrow next to it for more locations to save the file. When saving the file make sure that the file type is one your printer can read. Typically that will be the G-code but it's always good to double check.



Save the file to a USB, eject the USB from your computer and plug it into your 3D printer. Your printer's settings will differ but you should have the option to print. Choose this and navigate to the file you just saved. Print the file!

We recommend that you calibrate your 3D printer if it hasn't been used in some time. The instruction booklet that came with your printer will explain how to do this.

Some possible problems you may encounter

Sometimes the default settings of Cura end up interfering with the printer you chose. Typically Cura will give you a warning at the bottom of the screen about the issue, and if you type the setting it mentions into the search bar of the “Print Settings” menu it will likely show up highlighted and you can change it.

If your print doesn’t seem to be sticking to the printer’s buildplate you can try covering the surface with glue from a glue stick. It makes the surface a bit more tacky and gives the melted plastic something to hold on to.