

Printing 3D Models with CSDTs

Last updated on September 30th, Version 1

Download the Cura Slicer to follow along with the guide:

<https://ultimaker.com/software/ultimaker-cura>

For this guide, we are going to be working with the Adinkra CSDT. However, this guide would work with any other CSDT using CSnap!



Terminology

Here is a quick list of words in this guide that may or may not be familiar to you:

- STL: Standard Tessellation Language, one of the file formats that 3D printers use
- Model: The 3D representation of your design

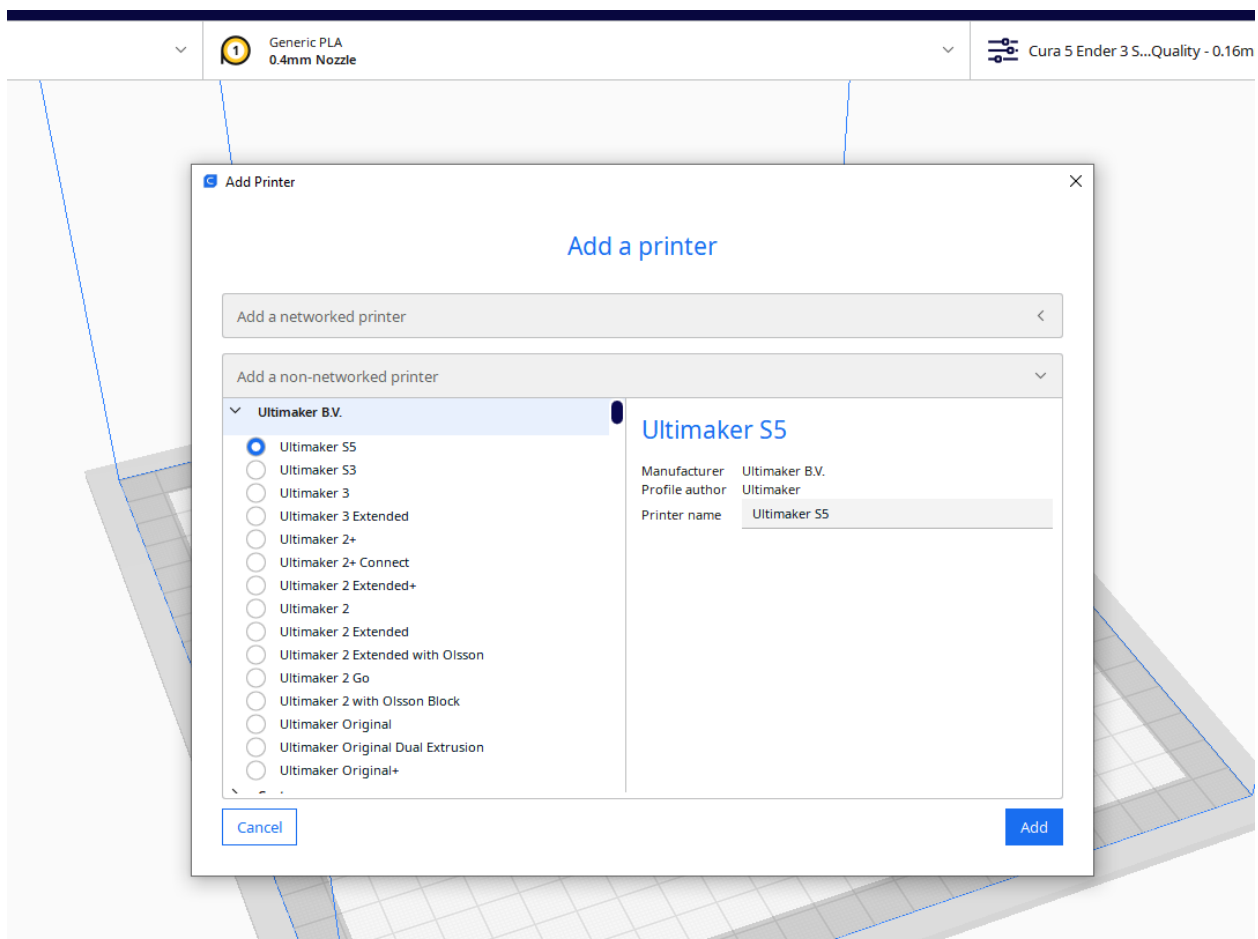
- Slicer: In order to print your design, software like Cura needs to slice your design into layers, so the printer knows where to go and where to print.
- FDM: Fused deposition modeling, this is the most common type of 3D printer. This one prints by melting some type of filament layer by layer on a moving bed / moving xyz axis.
- PLA Filament: Polylactic Acid, the most common type of filament for FDM printers. It is a biodegradable and food safe plastic, comes in a variety of colors and strengths
- SLA: Stereolithography, less common type of 3D printer, but allows for higher detail prints. Uses a photosensitive resin and hardens layer by layer by dipping the print bed into the stuff (one axis).

Creating your model

1. Head over to the Adinkra Carving software to create your design.
2. Save your design!
3. Make sure that your design is 3d printing ready with the following modifications:
 - a. The background is white
 - b. The design has high contrast. Use dark pen colors and color changes to your sprite images.
 - c. Anything that you don't want printed is hidden (or deleted).
 - i. If you accidentally delete something important, you can click on the trash can below the stage to retrieve it
 - ii. If that is missing, just refresh the page to reload the last saved design.
4. Right click on the stage and select either **STL** or **Advanced STL**. Either option will convert your design to a STL, but Advanced has more options to modify.
 - a. If you choose Advanced, you have the following options to modify:
 - i. Base - Disabled by default. Determines if the model will have a back plate created as well, or not. It is recommended to select this one if your design lines don't connect to each other, otherwise, your 3d print would fall apart!
 - ii. Smooth - Enabled by default, allows for smoother curves

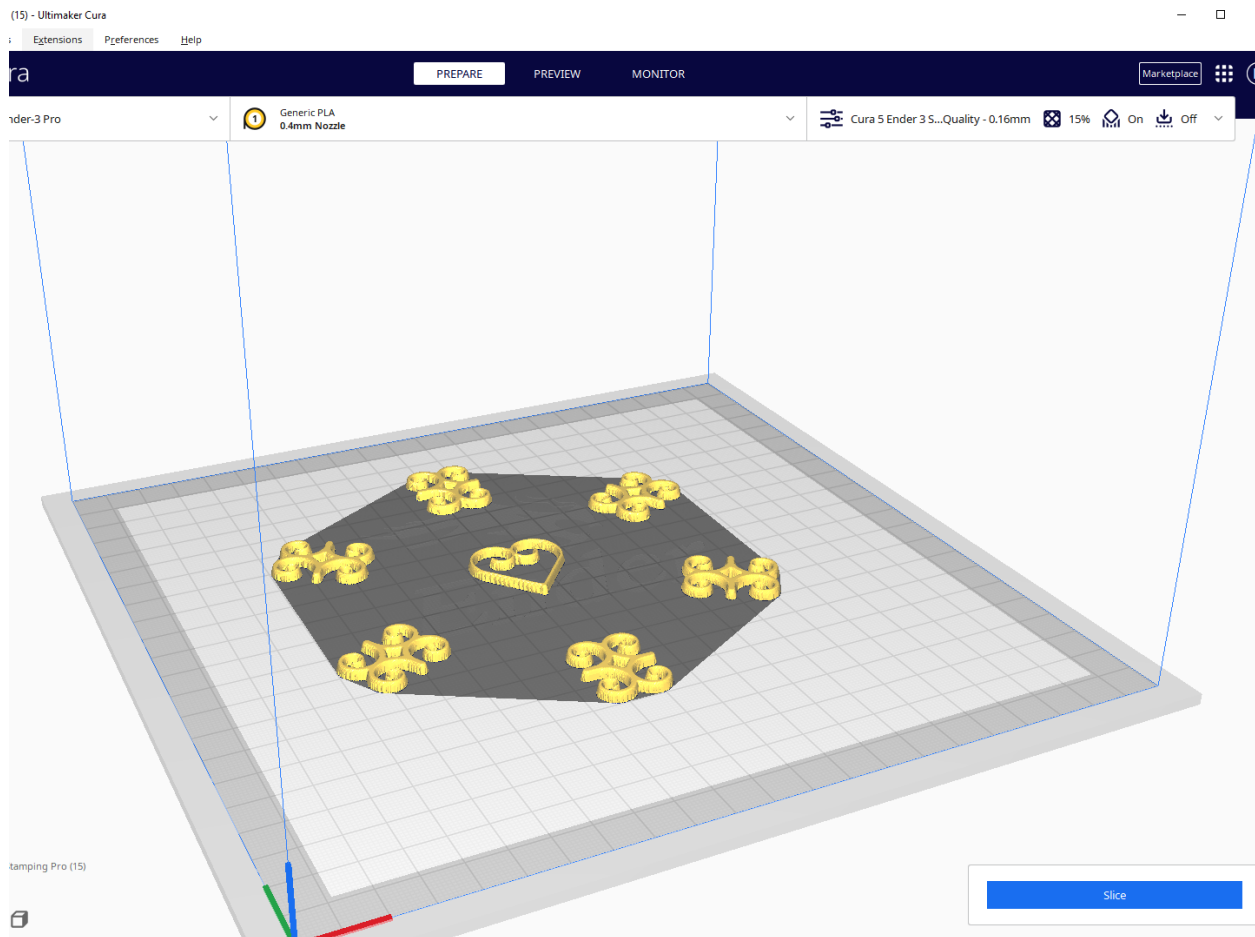
- iii. Negative - Disabled by default, you can use this to inverse your design (i.e. rather than creating a stamp, you can create a mold!)
 - iv. X, Y (Width, Height) - Determines the dimensions of your STL, default is 480x360.
5. Once it is downloaded, check your downloads folder to verify your STL model. You are now ready to import it into Cura.

Setting up Cura



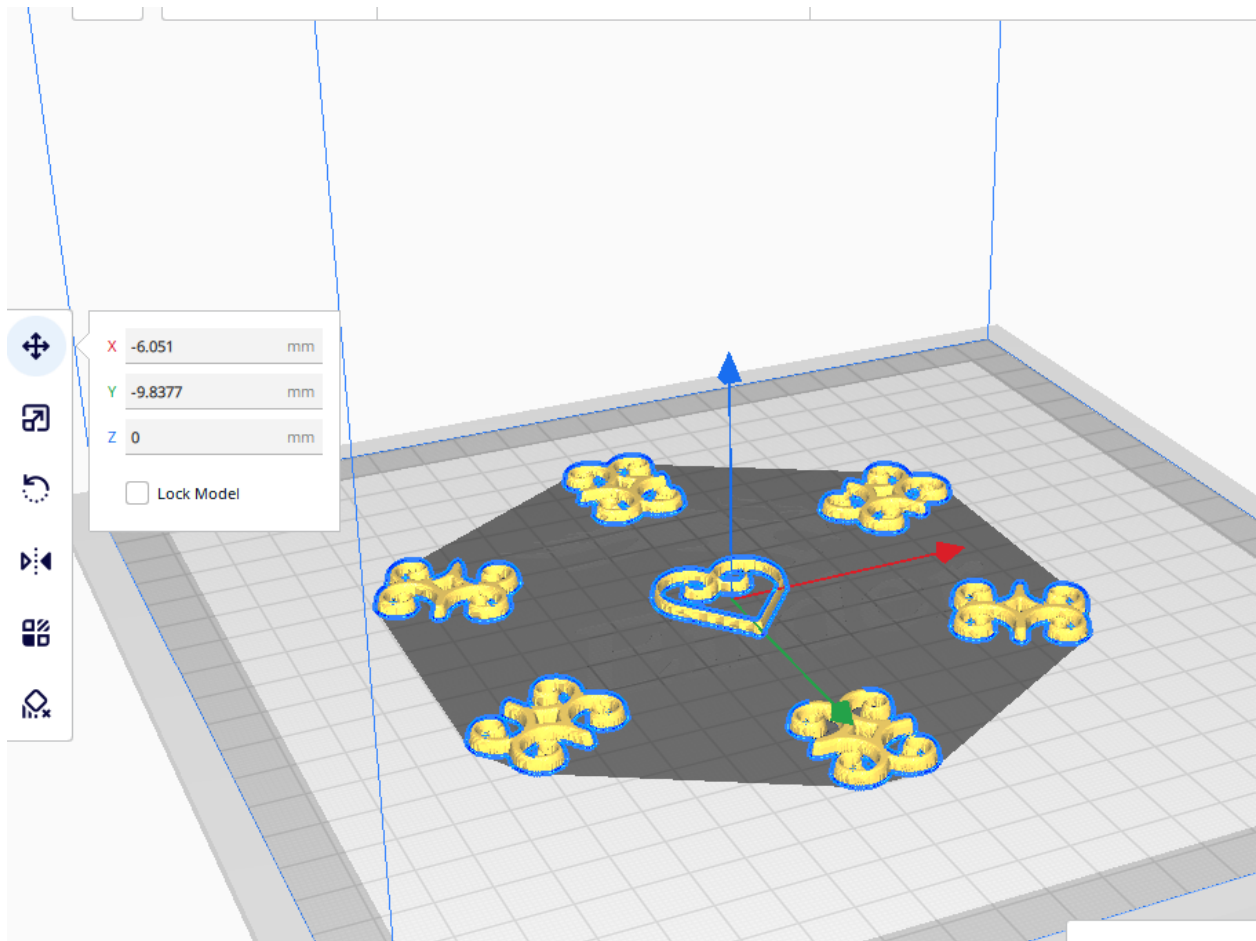
When you start Cura for the first time, it will take you through a walkthrough process to add your 3D printer. If not, go ahead and click on the 'Add Printer' option by the Folder icon in the top left, and select a 'Non-Networked' printer. Depending on your 3D printer, you may have to change additional options, as well as setting up the type of filament you would be using.

Importing into Cura



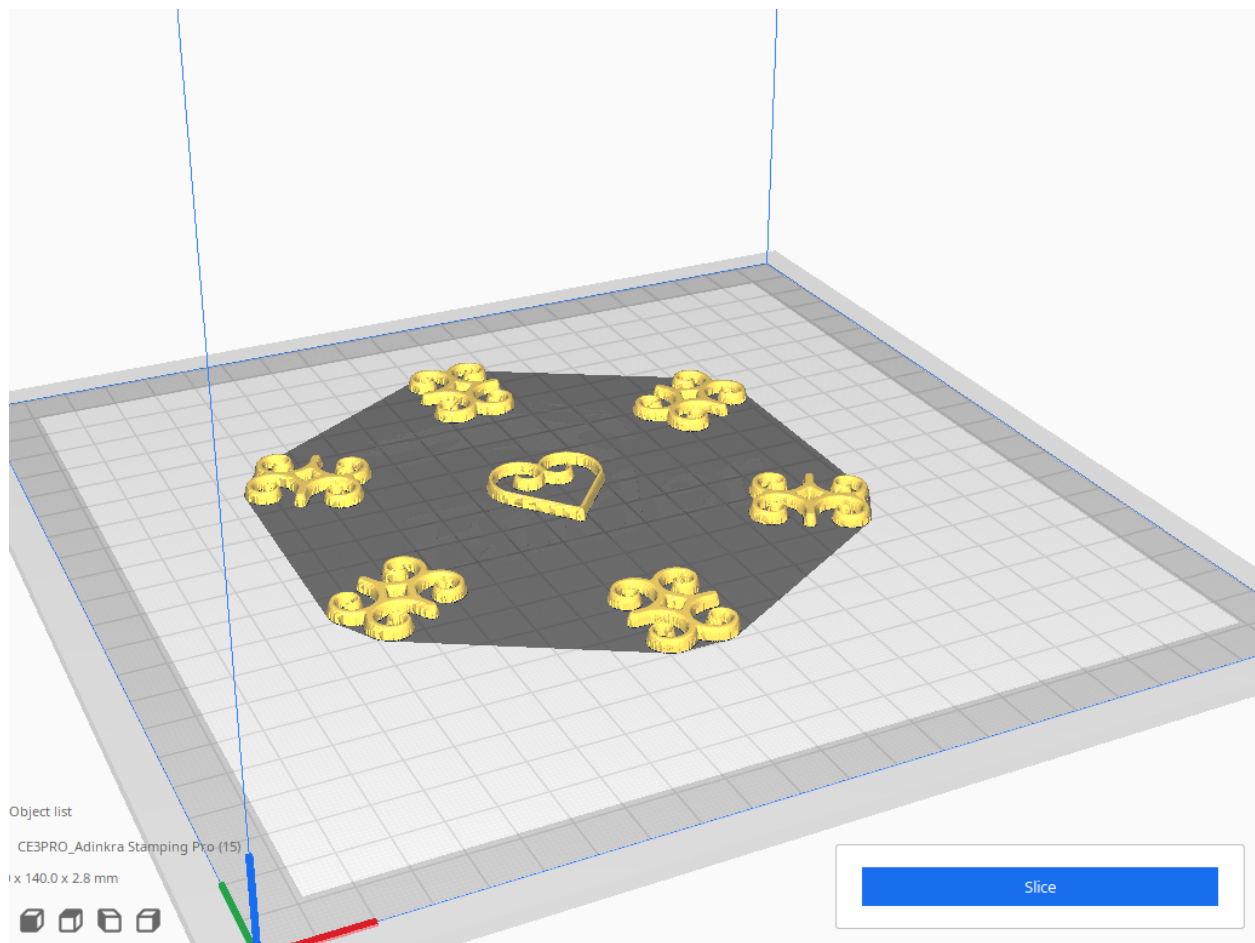
To import your model, go ahead and either drag and drop it into the software, or click on the File icon in the top left of the screen and select your model from your files. (I have imported a model created from the Adinkra Stamping software in this example).

Modifying your model



You can make minor adjustments to your model using the controls on the left. The most important ones are: Move, Scale, and Rotate.

Export for printing



Once you are happy with your positioning and scaling, go ahead and hit **Slice**. This will generate the GCode (set of instructions) to use on the 3D printer. You can save it to a flash drive / SD card and plug it into your printer for printing, or send it via Wi-Fi depending on the type of printer you have.

Printing Tips and Tricks

- Make sure to calibrate your 3D printer before printing.
 - The bed needs to be level and close enough to the extruder head to allow for a successful print. There isn't a best way to do this other than trial and error. Just look up your printer and see if there is a leveling guide to follow.
- Glue is your friend!

- Depending on the model you are printing, you may need to add some glue stick to your build plate in order to make sure the print actually stays on. Elmer's glue stick works just fine, but you can use 3d print bed adhesive if you so choose
- Double check your filament
 - Not all filaments will print the same! Especially so, not all PLA filaments are created equal. Depending on the company, the filament could require different build plate temps, different extruder temps, a thinner or wider nozzle, or a different extruder head altogether.
- Experiment with temps
 - Filaments have their recommended temps on the packaging (usually), but this doesn't always mean that these temps would work for you. Depending on your current environment's temp and humidity, your filament could require more or less heat to ensure the best prints possible.

If you run into any issues, or have any questions about 3D printing, contact us at csdtdevelopers@umich.edu