

3D PRINTING On CR-10SPRO

CREATED BY THE DI CREW

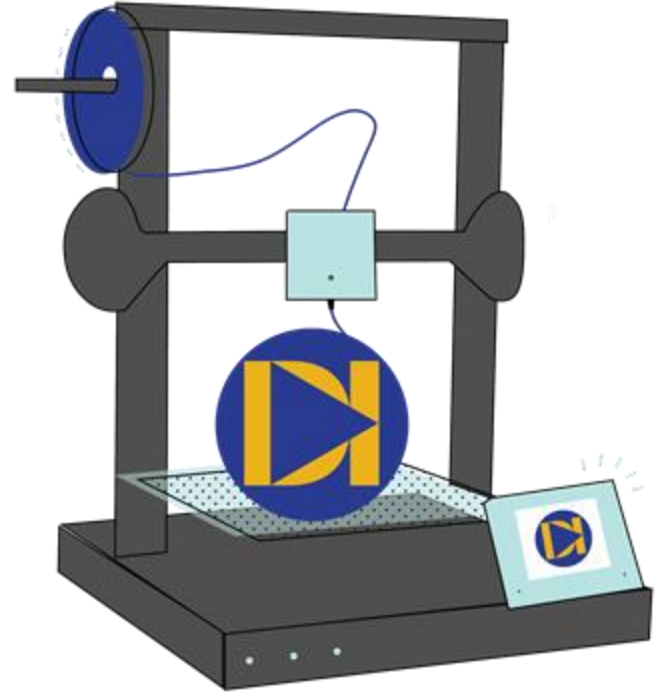


TABLE OF CONTENTS

Basic Information

- Machine info
- COVID Safety

File Preparation (CURA)

- File Compatibility
- Loading your file
- Model Transformations

Choosing your settings

- Layer Height
- Supports
- Infill
- Preview and Review

Using the CR-10s Pro

- Printer Set-up
- Printing

File Preparation



File Compatibility

- Cura is able to accept a variety of file types for its slicing capabilities. Below is a list of the most commonly used Cura file types. The full list can be accessed [here](#)

3D files:

- STL (.stl)
- OBJ (.obj)

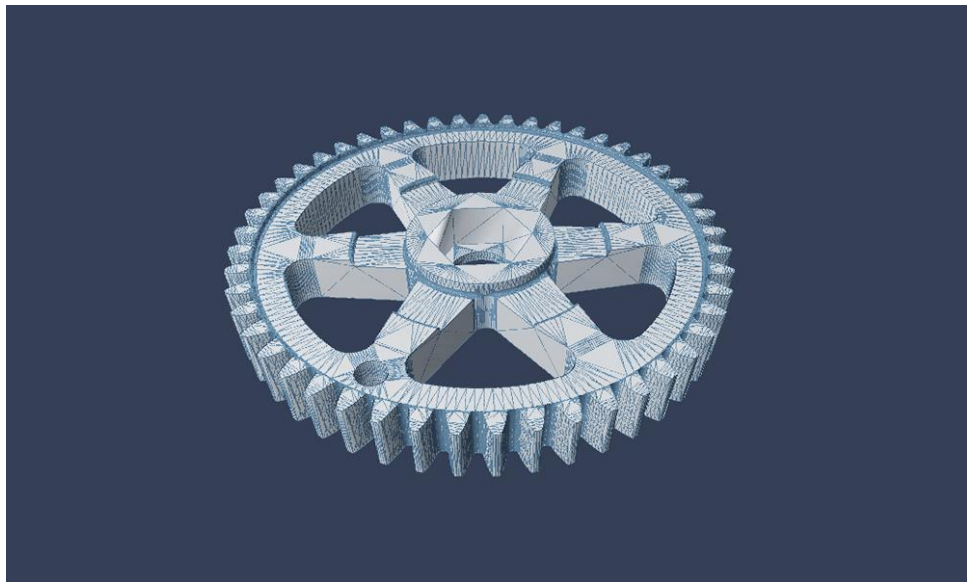
Image files:

- Jpeg (.jpeg)
- PNG (.png)

Gcode

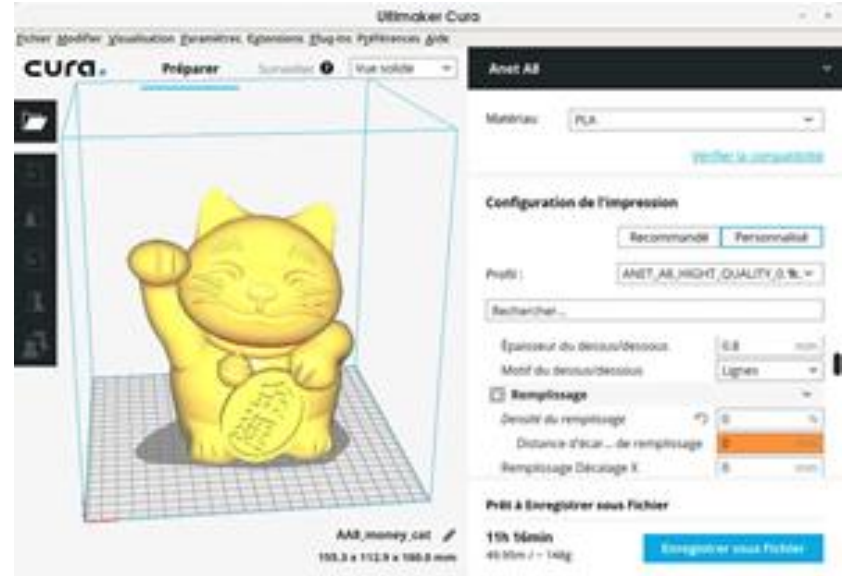
- Gcode (.gcode)

File Type



Loading your file

Open your file in Cura by pressing “ctrl+o”, by simply dragging and dropping the file into the CURA window, or by clicking on the folder icon in the top left of the screen.



Model Transformations



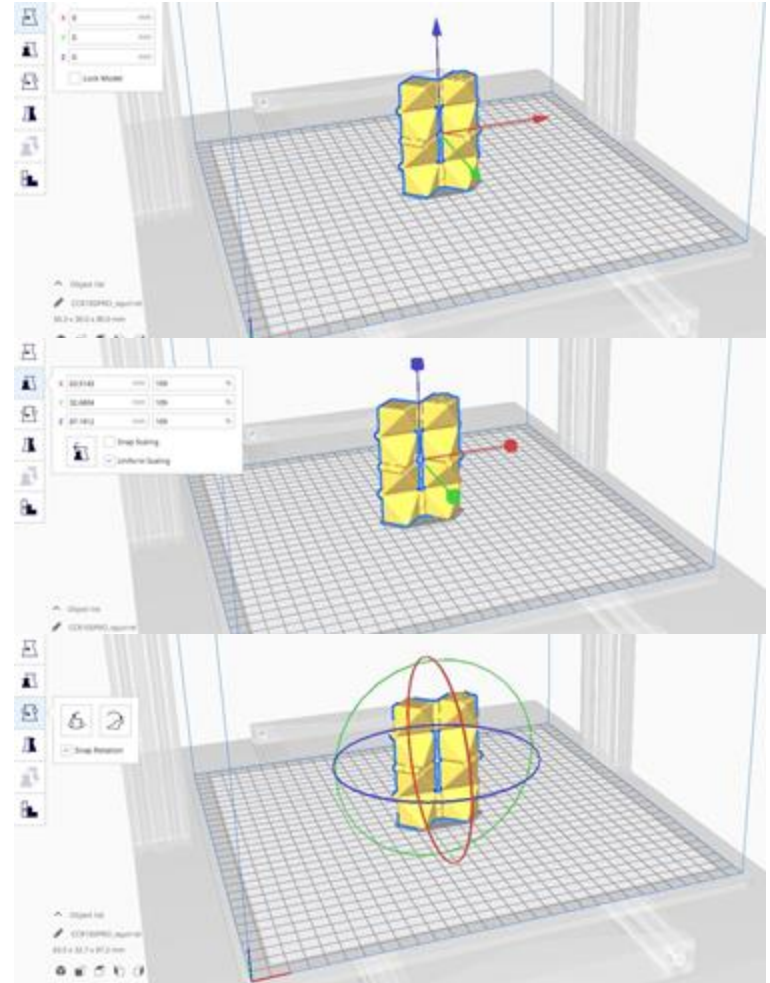
Orient your Model

“Move” tool. Setting each of the x, y, and z coordinates to 0 will center the model on the surface of the build plate.

“Scale” tool to change the dimensions of your model.
* Be sure to double check the dimensions due to the file moving process. Cura’s standard unit settings are in mm

“Rotate” tool to change the orientation of the object.

(You want the largest Flat surface on the Build Plate)



Choosing your settings



Layer Height

- **Layer Height** is the height of each layer of the print. This determines the resolution, or the quality of the print.

A print with a lower layer height, will have a higher quality, and thus, will appear smoother in appearance. And vice-versa

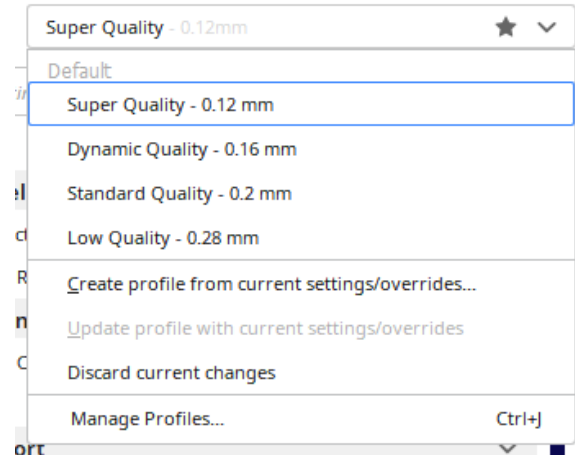
Layer height can be adjusted manually through the settings in the upper right corner of the CURA window. Preset profiles such as “standard, low, and super quality” can also be selected.

Suggested .2 for most prints



The print on the left has a HIGH layer height. The print on the right has a LOW layer height.

A glimpse into the advanced settings in CURA, including the layer height profiles that are available.



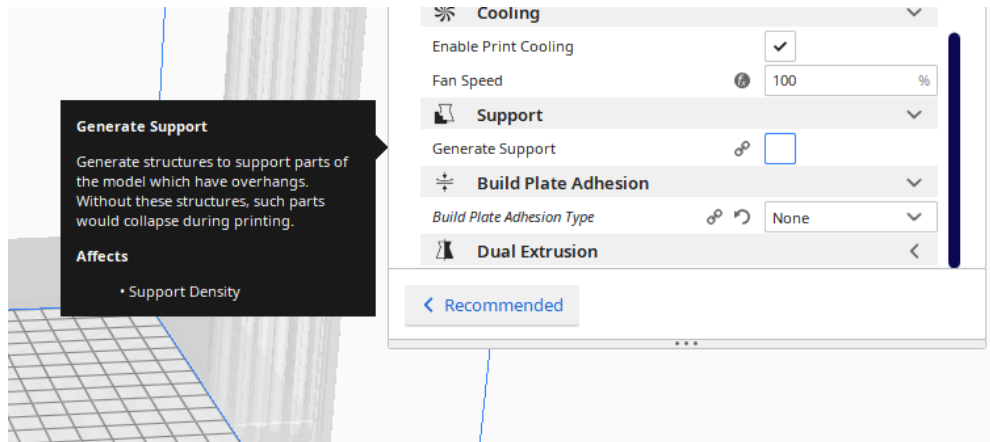
Supports

- **Supports** provide stability for overhanging components in your model.

Adding supports is required for many parts with large overhangs that are horizontal or close to being horizontal, roughly above an angle of 45 to 50 degrees.

Supports can be added under the advanced settings tab by scrolling downwards toward the bottom.

It is highly recommended to use a brim or raft when using supports



Infill

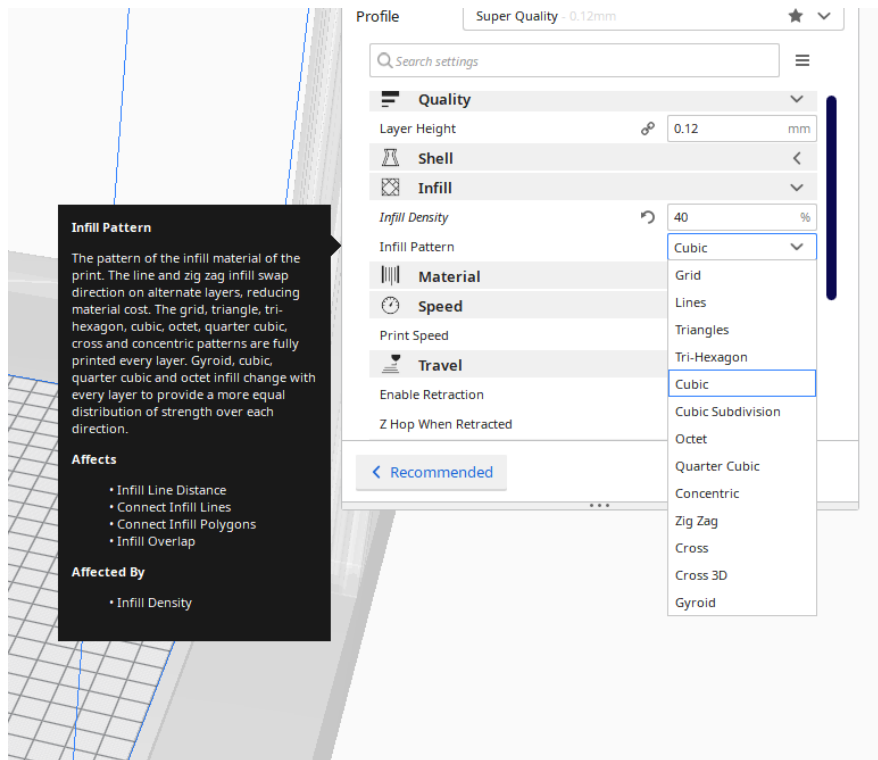
- **Infill** defines how much of the interior of your 3D model is solid.

Increasing the infill increases the rigidity and stability of the model, but will use more material and take much more time.

Different types of Infill can change rigidity or print time.

Infill can be managed from the advanced settings menu.

Suggested to never go below 4%

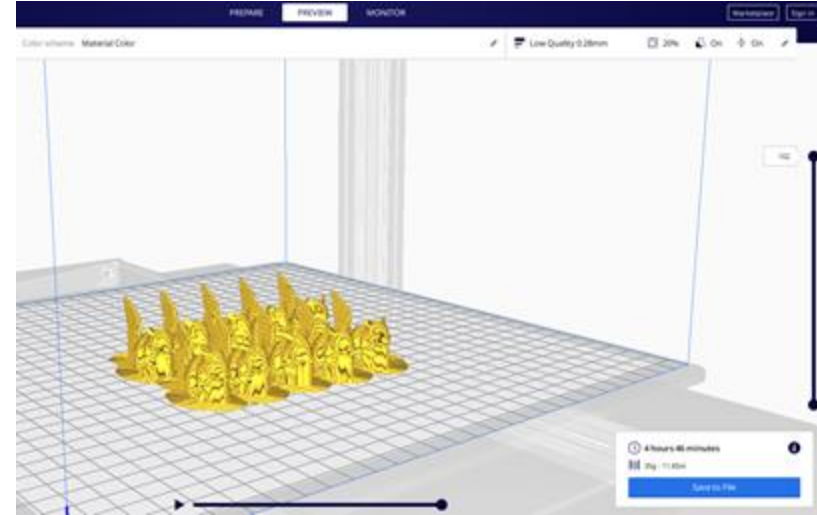


Preview And Review

Preview and review your work

Once you are satisfied with your print settings, slice your model with the “slice” button in the lower right corner of the window. You will receive a time and weight estimate for your print. Click the **“PREVIEW”** tab at the top of the screen to get a view of the layers, supports, and printed rendering.

What once was the **“Slice”** command has changed into the **“Save to File”** or the **“Save to ejectable device”** button. Save the gcode to MicroSD card in the ejectable card reader device. Eject this device.



Using the Creality CR-10s Pro



PRINTER SETUP

Power up Machine

- Power the machine on using the switch on the right side on the back of the machine.

Upload your file

- Insert the MicroSD card in the slot on the right side front.



Printing

- With a level bed and your filament properly loaded and ready to extrude, use the back key to get back to the printer's home screen. Press the Print icon to get to the list of jobs stored on the SD card.
- Select your G-code file from the list of prints and start printing.
- Make sure first layer prints without error (approximately 15-minute wait)
- Make sure to check in on your print every so often. If a print fails, catching it earlier can save time, money, and can even prevent machine damage.



PRINT SETUP

(DI CREW ONLY)



Printer Set up

Level the Build Plate (DI CREW ONLY)

Level the build plate- This is very important seeing as the build plate platform is very sensitive on these machines. Precision and care must be used in this critical step.

Select **Settings > Leveling >**
and then press **Z-Home**

this will re-home the z position of the machine
in case the person before you had adjusted
it during their print.



Printer Set up

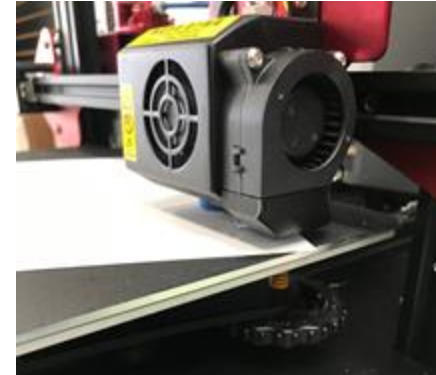
Manually Level the build plate (DI CREW ONLY)

After waiting for the z-home process, select **AUX leveling** to begin the manual leveling process.

Follow the on-screen prompts to assist the machine in the leveling process.

At each position marked on screen, slide a standard piece of computer paper (.2mm shim) under the nozzle. The paper should **not** slide loosely between the extruder nozzle and the build plate, but **should** provide a scratching resistance while still allowing movement.

Use the nobs under each corner of the build plate to adjust the build plate heights accordingly at each of the 4 corner positions.



Printer Set up

Load your selected Filament (DI CREW ONLY)

- Select **Settings** then select **Temp**
- Set the nozzle temperature to 200* C and wait for the machine to heat up.
- Heat up filament before removing filament (extrude 10mm before pulling the filament from the machine).
- Insert the filament into and through the **Filament Sensor**
- Keep feeding the filament until you feel it has reached the nozzle, and you will see your color of filament begin to ooze out of the extruding nozzle.

Pro-tip: Cut the tip of the filament at a sharp angle to encourage a smooth passage through the filament sensor.

