

From Concept to Object: 3D Printing on the Creality Ender 3



Image courtesy of:
<https://www.creality3dofficial.com/products/official-creality-ender-3-3d-printer>

Jason Clor

July 7, 2021

Contents

Introduction	2
Overview	2
Instructions	2
1.0 Preparing your file	2
1.1 Install Ultimaker Cura	3
1.1.1 Navigating Cura's interface	3
1.1.2 Add Ender 3 to presets	4
1.2 Open the file	6
1.3 Modify the object's size, position and orientation	6
1.3.1 Translation	8
1.3.2 Scaling	9
1.3.3 Rotation	10
1.3.4 Multiplication	10
1.4 Configure print settings	11
1.4.1 Set material	11
1.4.2 Set nozzle size	11
1.4.3 Check print settings list	12
1.5 Slice, preview and save	13
1.5.1 Click "Slice"	14
1.5.2 Preview the results	15
1.5.3 Save the file to an SD card	15
2.0 Import your file to the Ender 3	16
2.1 Insert SD card	16
2.2 Initialize the Ender 3	16
2.3 Select your file	16
3.0 Printing	17
4.0 Post-processing	17
4.1 Remove the print	17
4.2 Trim excess material	17
4.3 Optional finishing	17
Troubleshooting	18
Support problems	18
Adhesion problems	18
Extrusion problems	18
Conclusion	18

Introduction

3D printing is an additive manufacturing process that enables users to create physical models of digital objects out of thermoplastics. In the past, plastics manufacturing has required the design of a specialty injection mold to enable factory production of desired prototypes, an commitment of considerable time and money; but a 3D printer such as the Creality Ender 3 enables the consumer to generate objects of similar materials and quality in their own home relatively inexpensively and in a matter of hours.

Overview

The following instructions are designed to enable a beginning user to prepare the digital file of a desired object (or “model”), import it to the Ender 3, print it and perform any necessary post-processing.

These instructions assume the user has already selected a design to print: either the sample file included on the specially formatted SD card included with the Ender 3, a model downloaded from one of many online 3D printing repositories (such as Thingiverse, Cults3D or MyMiniFactory), or an original design created using computer-aided design (CAD) software such as AutoCAD or Blender. In order to be printed on the Ender 3, a model must be saved in the stereolithography (STL) format.

It is also assumed that the user has successfully assembled their Ender 3, levelled the print plate and supplied the printer with filament. If these steps have not yet been completed, the user should first consult the assembly and calibration instructions supplied by the manufacturer.

Instructions

1.0 Preparing your file

Before a file can be used to print a 3D object, the chosen model must be processed using an application called a *slicer*. Slicing software analyzes an object’s geometry, divides it into the correct number of layers based on the printer’s resolution, and encodes a series of instructions for the printer using a language called Gcode.

A good, free slicer for beginners is Cura, which is distributed by 3D printer manufacturer Ultimaker.

1.1 Install Ultimaker Cura

Navigate to <https://ultimaker.com/software/ultimaker-cura> and download the most current version of Ultimaker Cura for your platform (Linux, MacOS or Windows). Once the download is complete, install and launch the application.

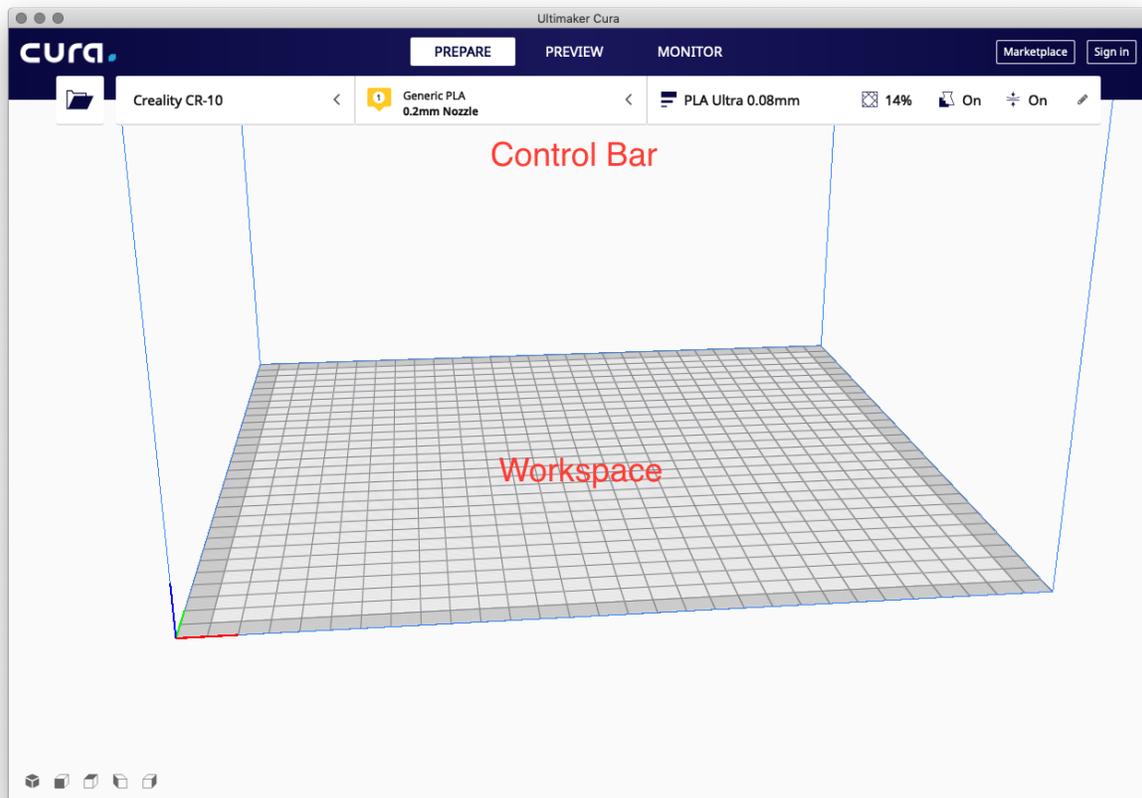
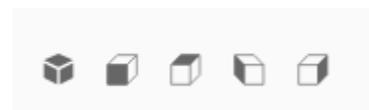


Image 1-1

1.1.1 Navigating Cura's interface

The main window of Cura is divided into a control bar and the main workspace, which displays a representation of the printer's print surface (Image 1-1). From the control bar, the user can load an object, change the printer preset, change the nozzle and material settings, modify print settings, or preview slicing before export. The workspace is used to position, rotate, scale and duplicate objects.

1.1.1.2 To change the view of the workspace, left-click and drag the mouse to rotate and move the scroll wheel to zoom in and out.



1.1.1.3 Optionally, click on one of the cube icons in the lower left of the workspace (Image 1-2) to align the view with one of the orthographic directions (front, top, left or right).

Image 1-2

1.1.2 Add Ender 3 to presets

In order for Cura to export print instructions the Ender 3 can use, the printer must be first added to Cura's presets.

1.1.2.1 From the Settings menu, select Printer > Add Printer...

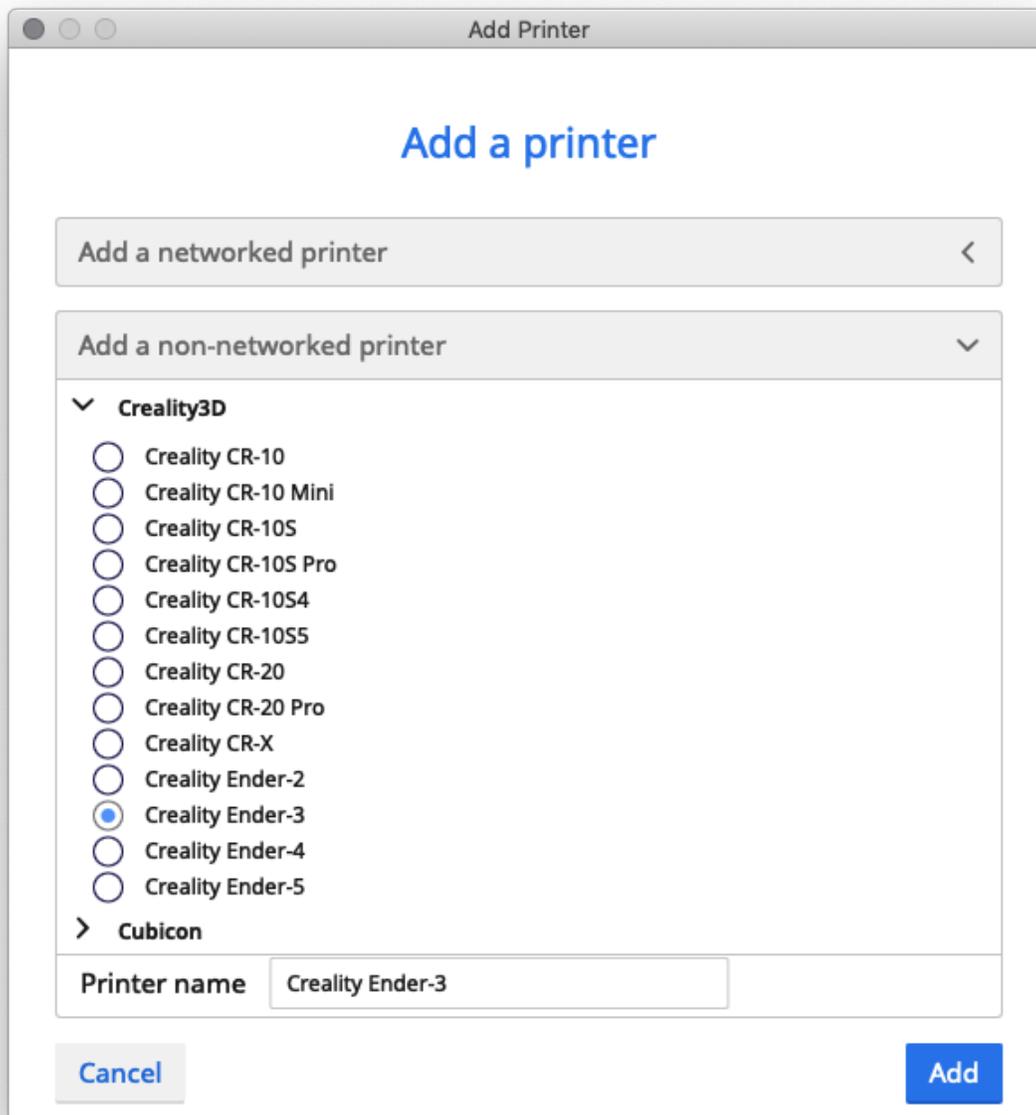


Image 1-3

1.1.2.2 In the modal that appears, scroll through the non-networked printer list to Creality3D, open the list, and select “Creality Ender-3” (Image 1-3).

1.1.2.3 Click “Add.”

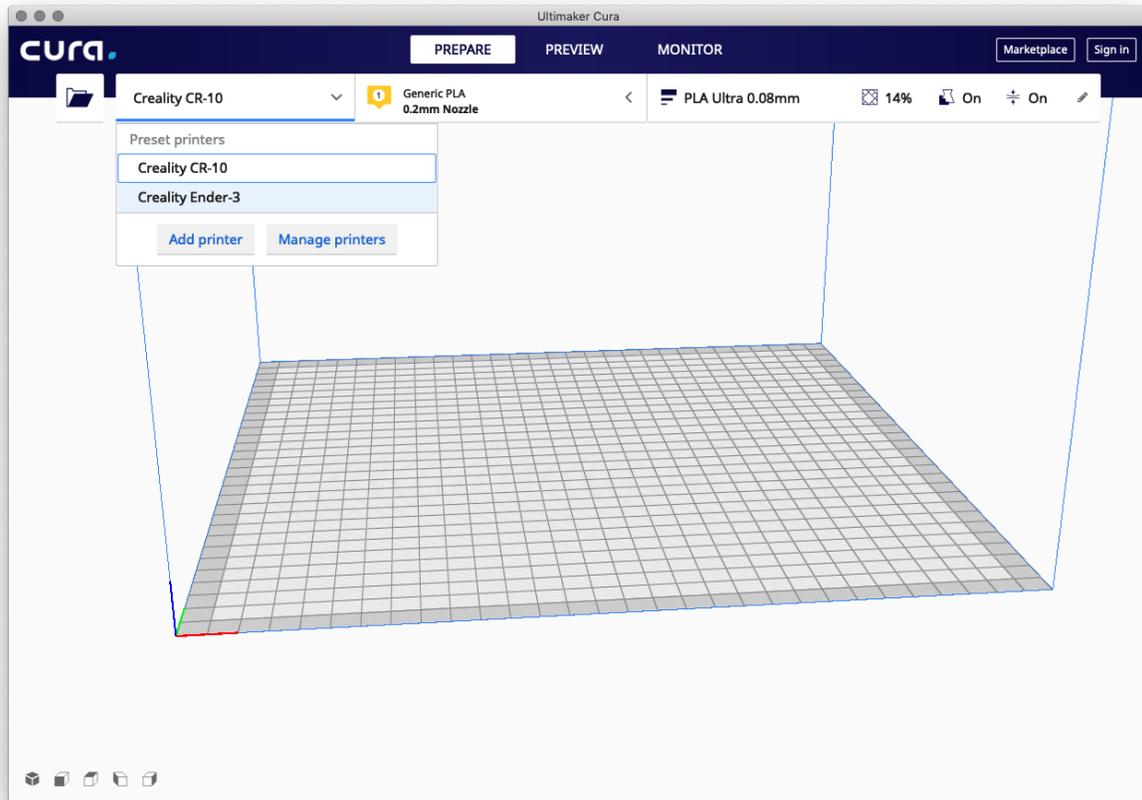


Image 1-4

1.1.2.4 In the main window, change the preset by clicking the box to the right of the folder icon in the control bar and selecting “Creality Ender-3” (Image 1-4).

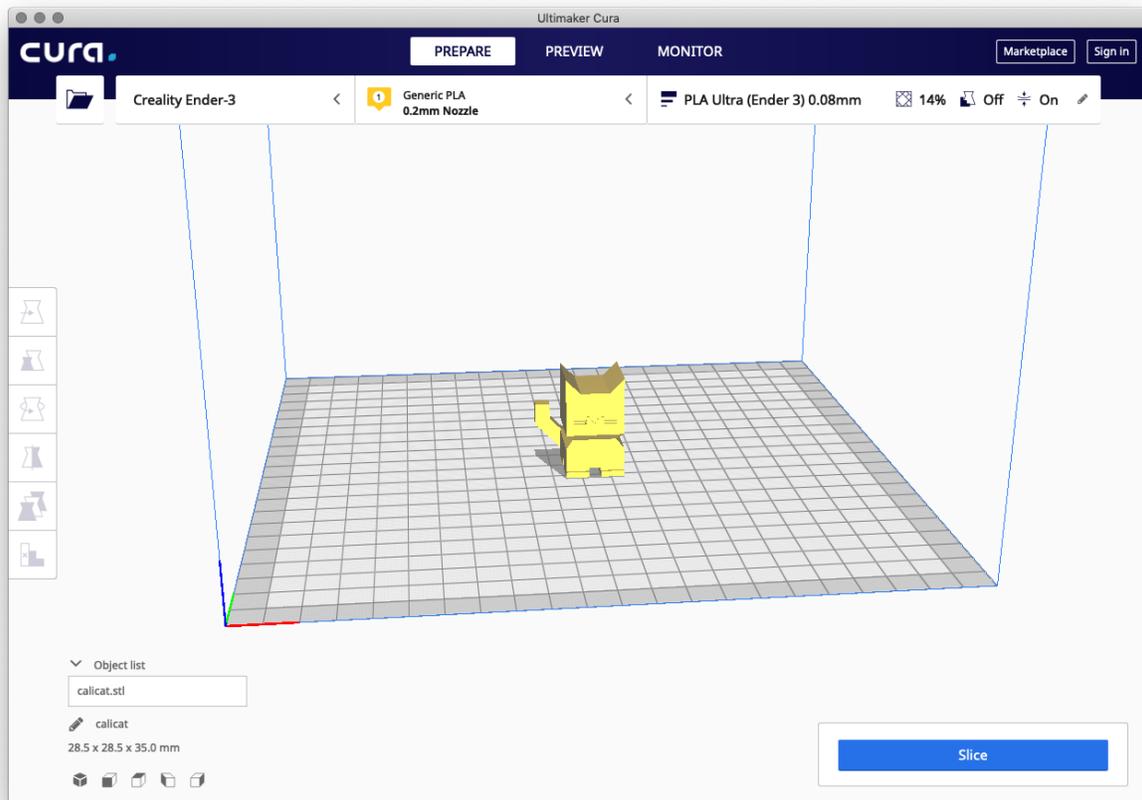


Image 1-5

1.2 Open the file

Click on the folder icon in the control bar (optionally, from the **File** menu, select **Open File(s)...**) and select the STL file you wish to print. Once loaded, the model will appear centered in Cura's workspace resting on the print surface (Image 1-5). (Note: a model may contain multiple objects.)

1.3 Modify the object's size, position and orientation

The model chosen to print may not be the correct size or in an ideal orientation for printing. Due to the additive nature of 3D printing, unsupported parts of a model's geometry (overhangs) will require printed supports to prevent sagging. Cura will insert these supports by default, but positioning an object in a way that requires fewer supports (or none) will decrease print time.

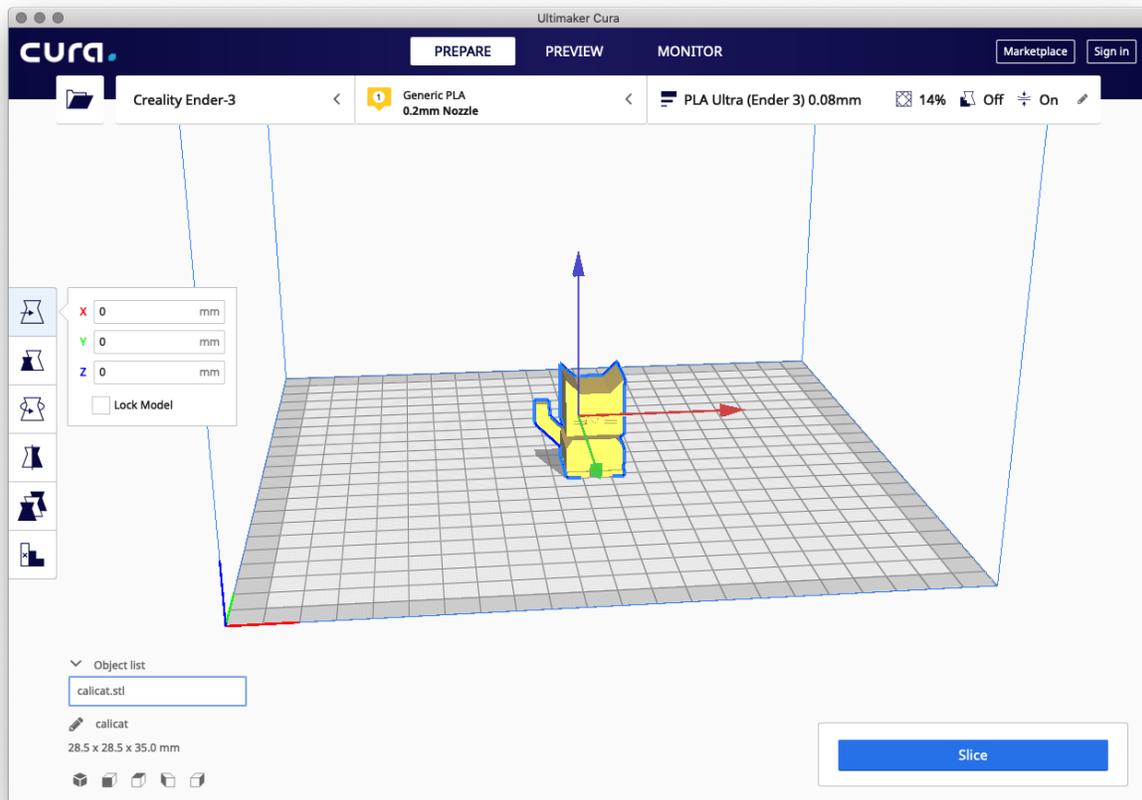


Image 1-6

In order to make changes to the attributes of a model in the workspace, click on it. A blue outline around the object indicates it is selected, and a set of icons representing transformation options appears on the left (Image 1-6).

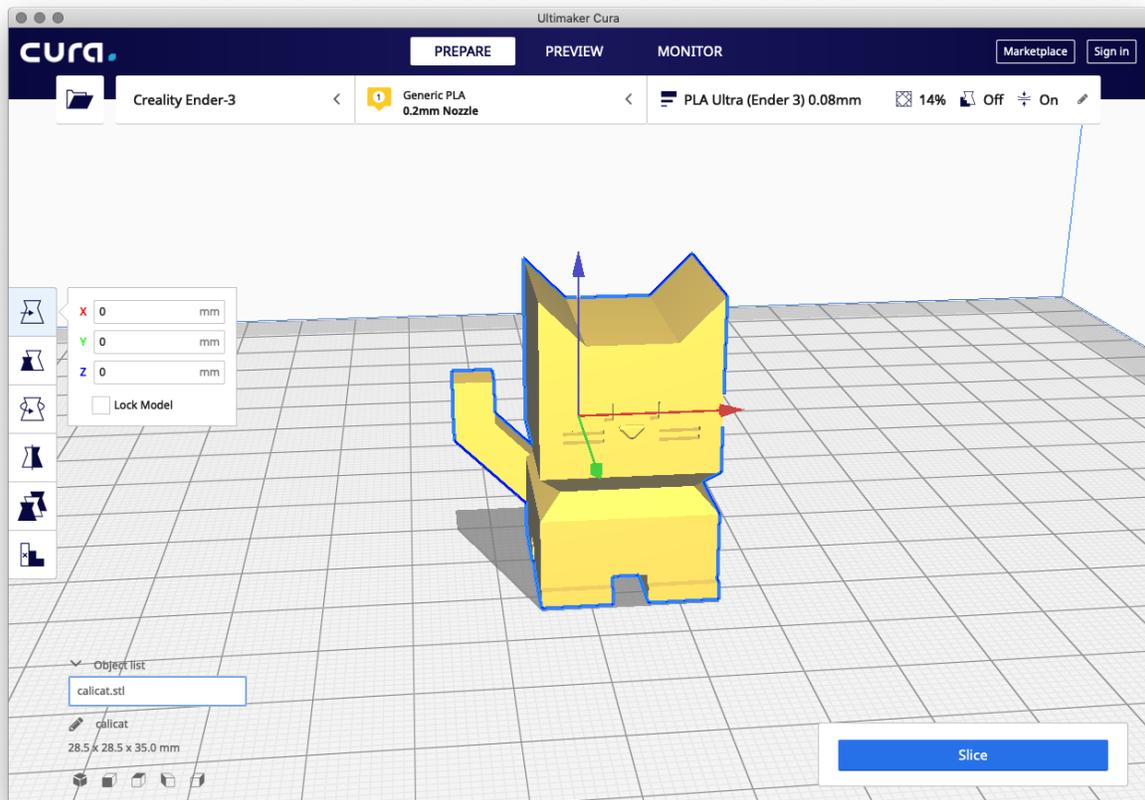


Image 1-7

1.3.1 Translation

To translate (move) an object, select the first transformation option (Image 1-7) or press 'T'. Then click and drag the object itself to move it on the plane parallel to the print surface. Optionally, click and drag one of the three colored arrows originating from the center of the object to move it only along the selected axis (x, y, z).

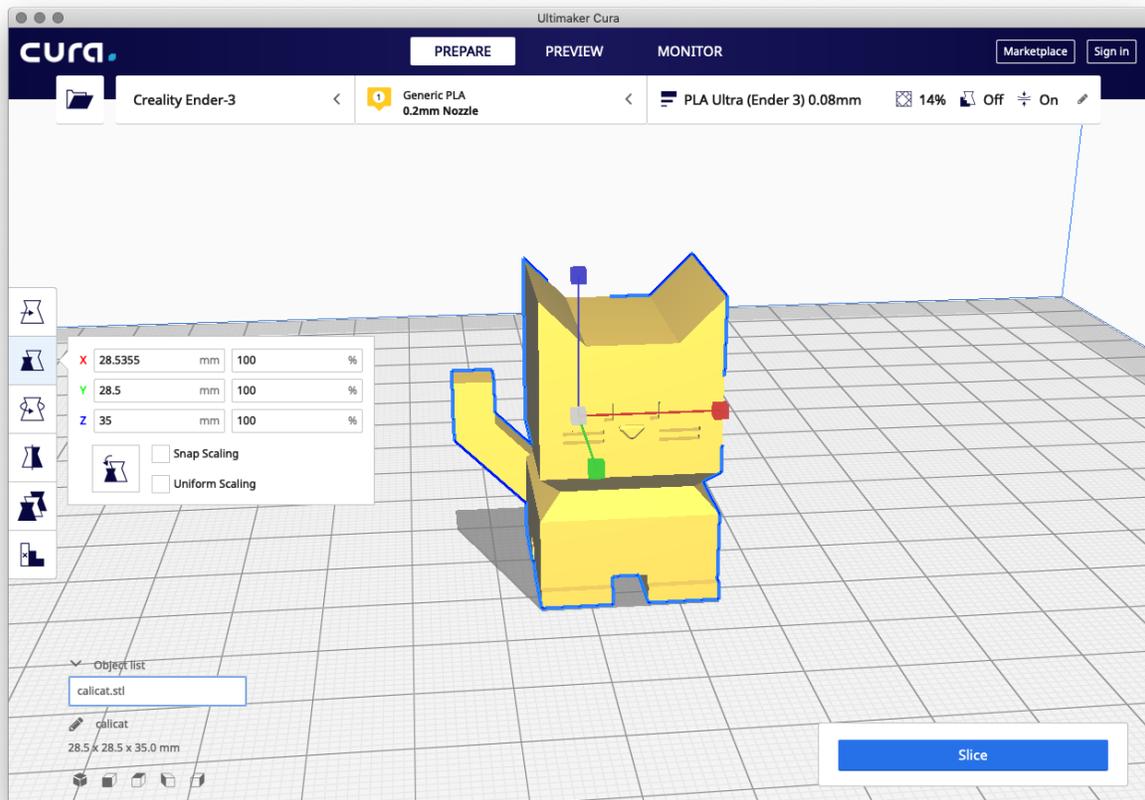


Image 1-8

1.3.2 Scaling

To scale an object, select the second transformation option (Image 1-8) or press 'S'. Then click and drag on the object to increase or decrease its size relative to the print surface. Optionally, deselect "uniform scaling," then click and drag one of the three axis indicators centered on the object to scale it only along that axis.

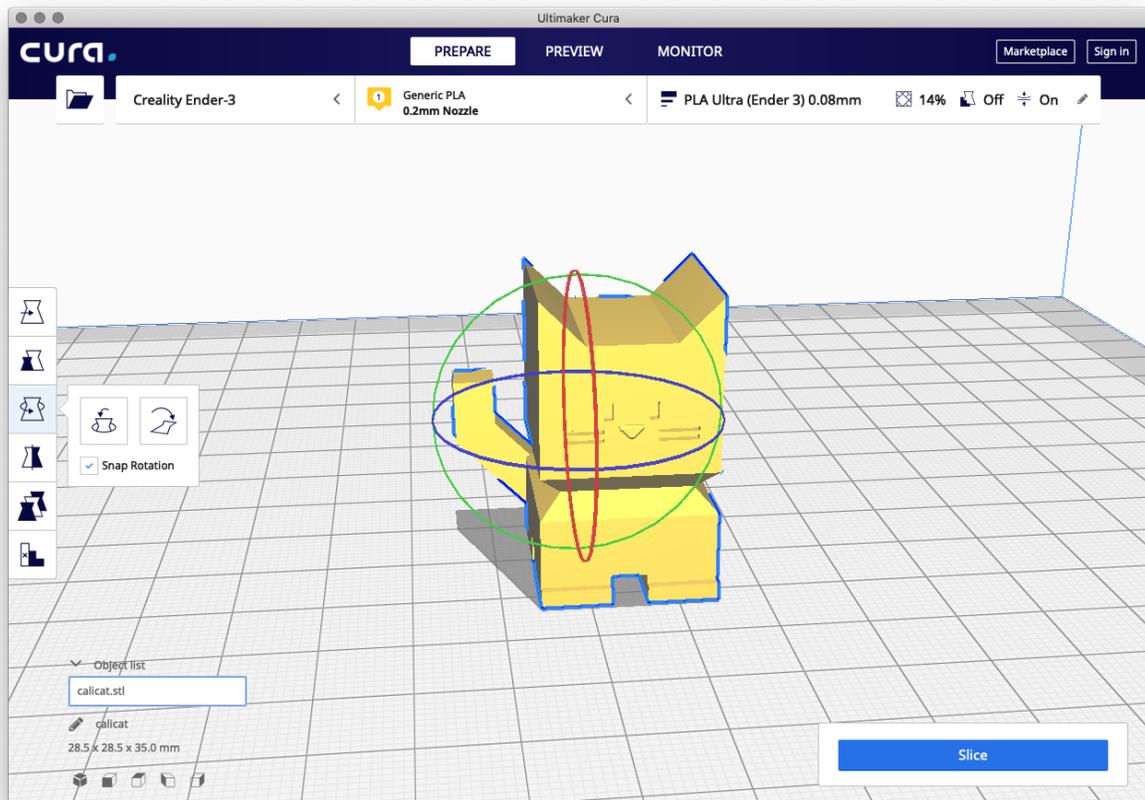


Image 1-8

1.3.3 Rotation

To rotate an object, select the third transformation option (Image 1-9) or press 'R'. Then click and drag the object to rotate it to the desired orientation. Optionally, click and drag the appropriate colored circle to rotate the object around a single axis.

1.3.4 Multiplication

To create additional copies of an object, right-click on it and select "Multiply Selected Model" from the drop-down menu, then enter the number of additional copies you want (Image 1-10). Cura will create these copies and distribute them evenly across the print surface.

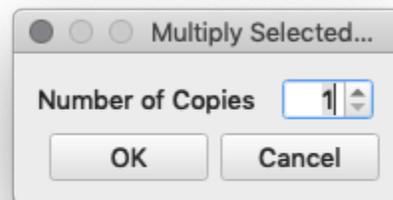


Image 1-10

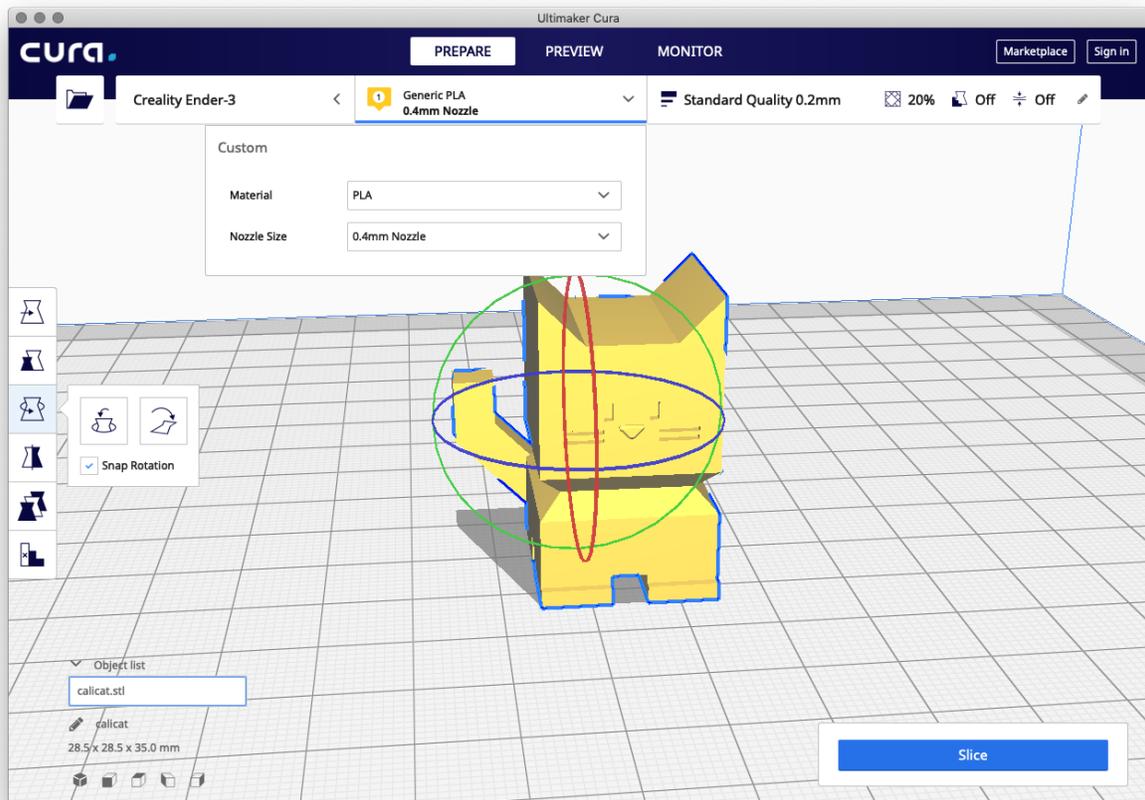


Image 1-11

1.4 Configure print settings

Once the model is the correct size and orientation for printing, confirm that printer settings are correct. Begin by clicking on the center section of the control bar to set material and nozzle size (Image 1-11).

1.4.1 Set material

The simplest and least expensive material for use with the Ender 3 is polylactic acid (PLA) filament; a small sample is included with every new printer. Assuming the printer has not been set up with a different material, select “Generic PLA” from the Material drop-down.

1.4.2 Set nozzle size

Using the factory-supplied nozzle, the Ender 3 prints at 0.4mm. Select this size from the drop-down menu.

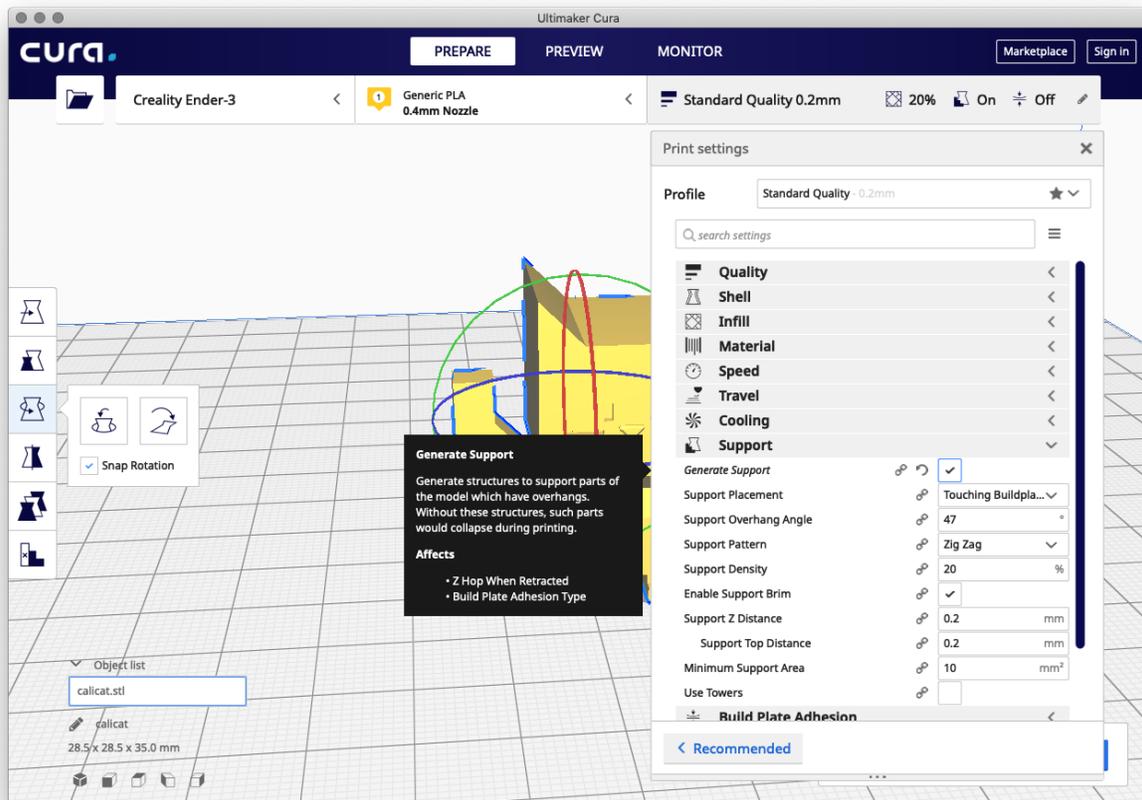


Image 1-12

1.4.3 Check print settings list

Click on the rightmost section of the control bar to open the print settings list (Image 1-12). These settings control everything from outer wall thickness of the object to infill percentage, print speed, nozzle and print surface temperature, fan speed, adhesion and support methods, and more.

For the purposes of a sample print, you should only need to confirm two settings: supports and adhesion.

1.4.3.1 Click to open the “Support” menu and check “Generate Support” if it is not already selected (Image 1-12).

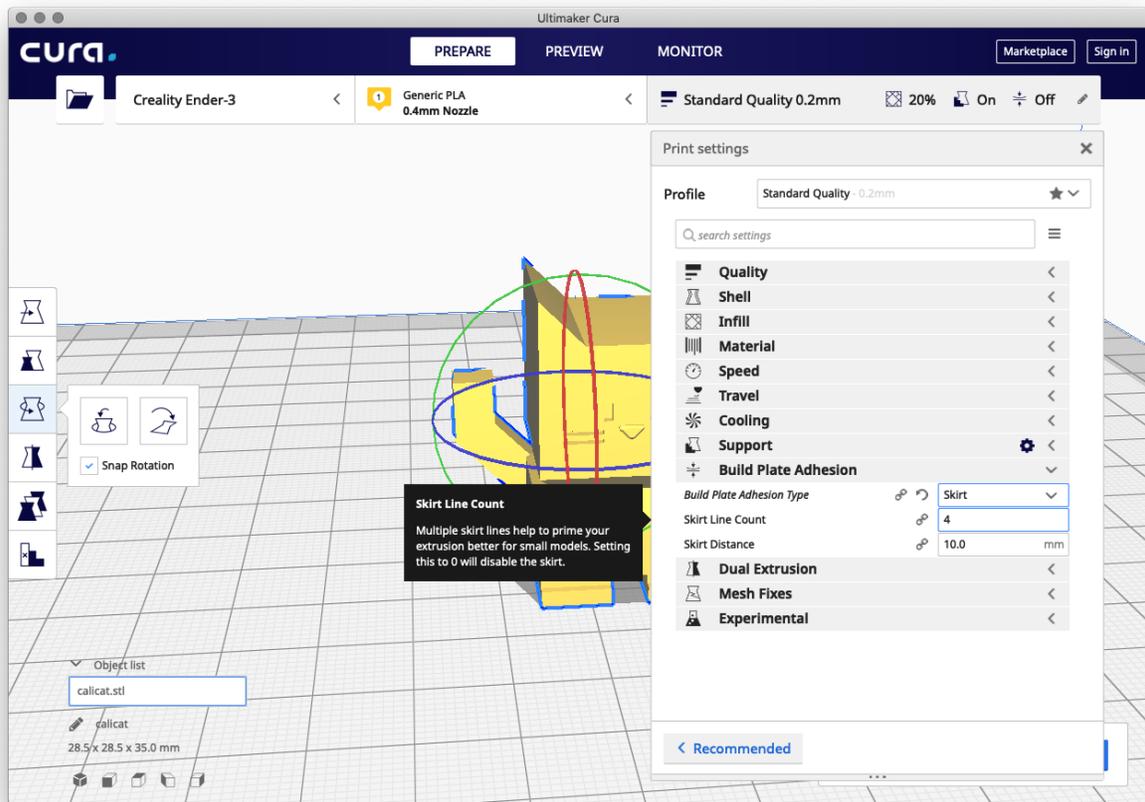


Image 1-13

1.4.3.2 Click to open the “Build Plate Adhesion” menu and select “Skirt” for “Build Plate Adhesion Type” (Image 1-13).

1.5 Slice, preview and save

The model should now be properly scaled and oriented, and the proper print settings made, in order for Cura to perform the slicing and generate Gcode to instruct the Ender 3 to print the object.

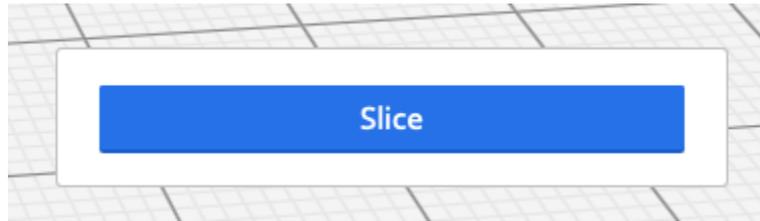


Image 1-14

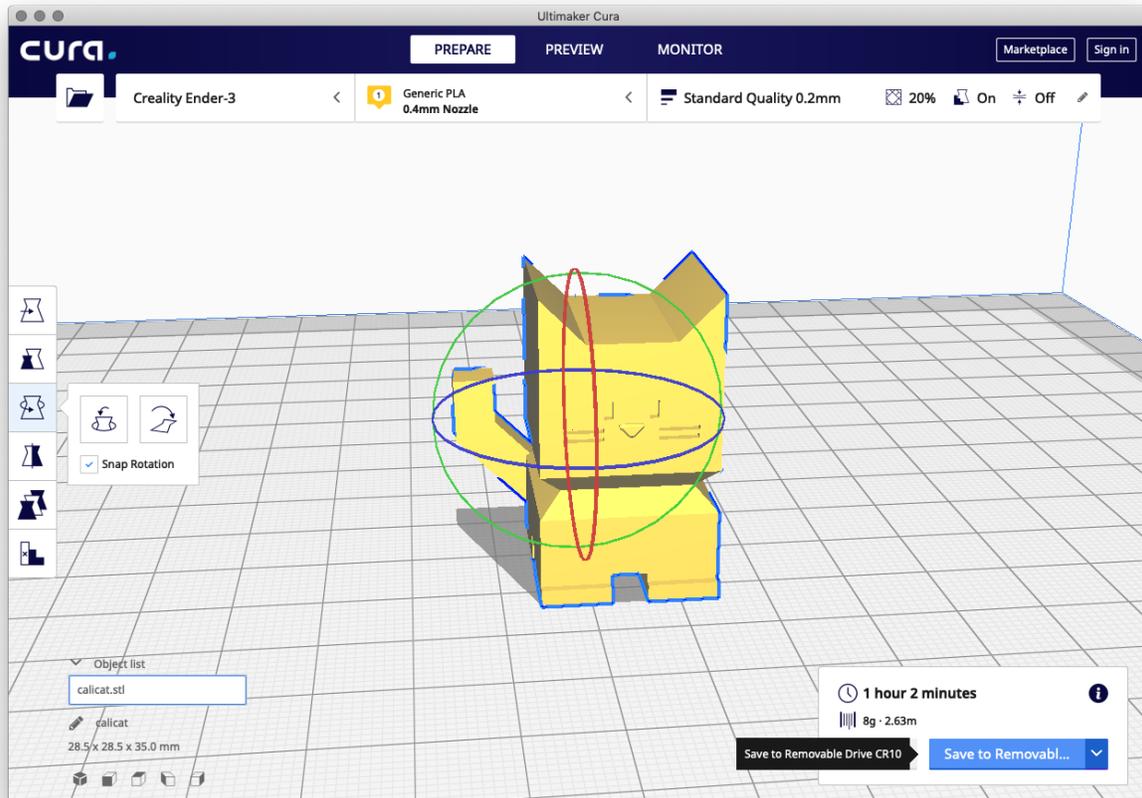


Image 1-15

1.5.1 Click “Slice”

Click on the “Slice” button in the lower right corner of the workspace (Image 1-14). When Cura completes the slicing process (this may take several minutes), an estimate of print time and filament amount will be displayed (Image 1-15).

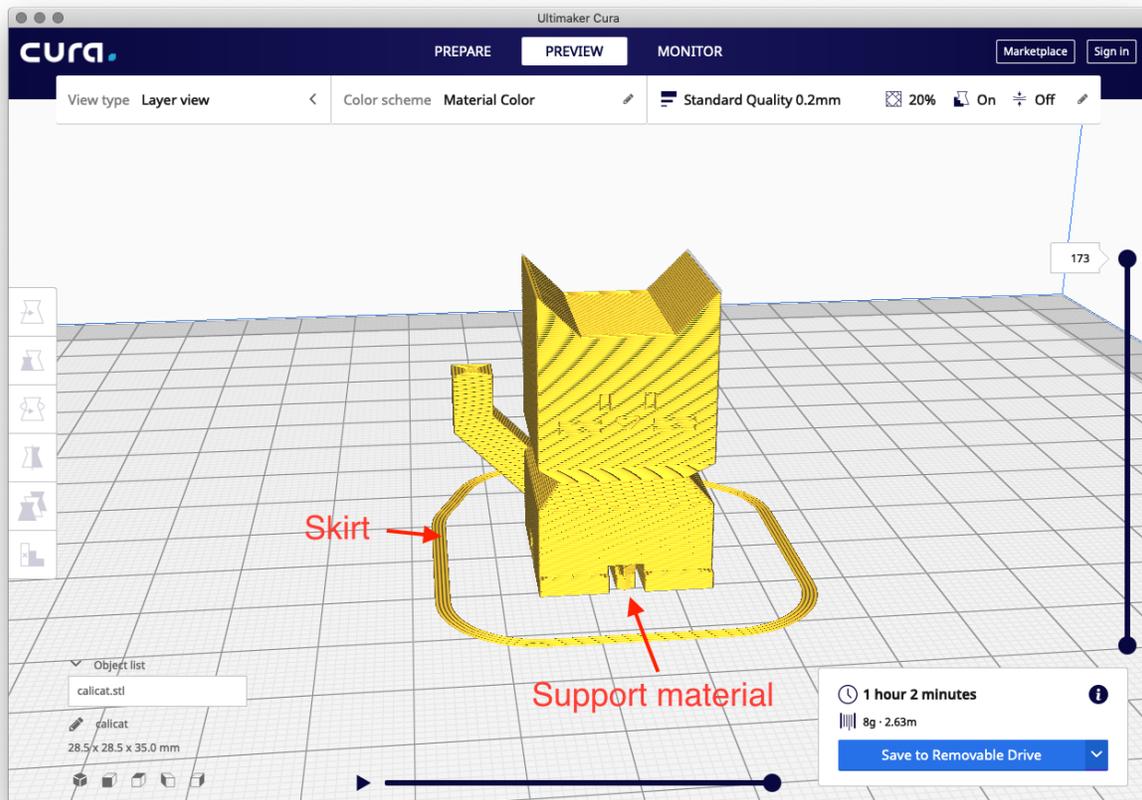


Image 1-16

1.5.2 Preview the results

Click on “Preview” beneath the print estimates (or, optionally, at the top center of the window) to see a preview of the print layers (Image 1-16). This preview will include additional material on the print surface surrounding the model (the skirt, which aids in adhesion) and any support material necessary to extrude overhanging parts of the model. The preview can be zoomed and rotated using the same controls as the preparation view.

1.5.3 Save the file to an SD card

If there are no obvious problems with the slicing, insert an SD card into your computer using an adapter and click on “Save to Removable Drive” in the lower right of the workspace. If you prefer, you can click on “Save to File” without inserting an SD card and manually transfer the resulting file to a card later.

3.0 Printing



Once printing begins, the Ender 3 will execute the instructions encoded in the print file. Progress can be monitored from the control screen, but no further action is required. When the print is complete, a tone will sound.

4.0 Post-processing

Once printing is complete, the finished object can be removed from the print surface.

4.1 Remove the print

Use the removal tool included with the printer (Image 4-1) to scrape the printed object free from the print surface. If the finished print adheres too firmly to the print surface to remove safely, separate the plastic print surface from the metal print plate and carefully flex it until the printed material comes free.



Image 4-1. Courtesy of <https://toolguyd.com/extra-tools-youll-need-for-3d-printing/>

4.2 Trim excess material

Supports and adhesion materials (such as brims or rafts), as well as any other excess plastic, may need to be removed from the object using a tool such as the flush cutters included with the Ender 3 (Image 4-2). With proper calibration, however, this material will be loose enough to remove by hand.



Image 4-2. Courtesy of https://support.formlabs.com/s/article/Advanced-Support-Removal-Techniques?language=en_US

4.3 Optional finishing

A printed object can be further finished by sanding, coating with filler to conceal print lines, or painting.

Troubleshooting

Variations in filament quality, printer manufacture, and printing environment can cause problems that negatively affect the quality of the printed product.

Support problems

In the event support material is insufficient for the model's overhangs (or missing altogether), check the print settings in Cura. Detailed information about the setting options can be found in documentation for Cura.

Adhesion problems

A variety of conditions can prevent extruded filament from adhering to the print surface, resulting in a print that comes loose prior to print completion (often resulting in a failed, "spaghetti" print, example at right). Adjustments to nozzle and print plate temperature can increase adhesion by ensuring filament is in a liquid state when it is placed on the surface. These adjustments can be made either in Cura or on the Ender 3's control screen once printing has begun. See Ender 3 documentation for more details.

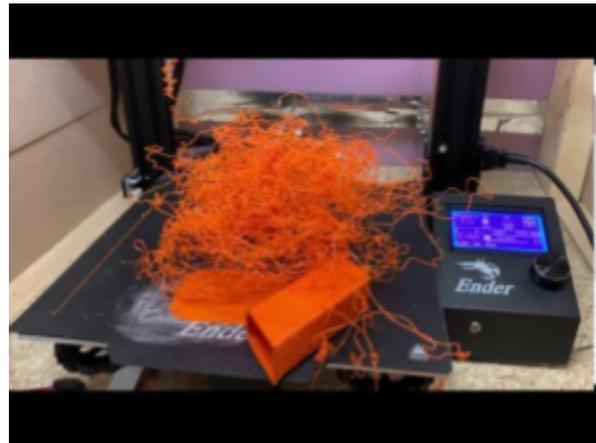


Image courtesy of <https://www.nikkoindustries.com/blogs/news/5-most-common-problems-with-the-creality-ender-3-troubleshooting-guide>

Extrusion problems

Gaps in layers or missing layers can be the result of extrusion problems, which may be the result of poor quality filament, insufficient nozzle temperature or extruder malfunction. In the event of these problems, change filament, adjust nozzle temperature, and check the extruder mechanism using steps laid out in the Ender 3's documentation.

Conclusion

Completed successfully, these instructions should demonstrate both the simplicity and the flexibility of the Creality Ender 3 in enabling a newcomer to the field of 3D printing to generate a physical object from a digital design. Ideally, this demonstration will serve as the foundation for an exploration of the limitless possibilities of both 3D design and simple additive manufacturing, encouraging further experimentation with new materials and original modeling concepts.