FABRICO ADITIVO E IMPRESSÃO 3D
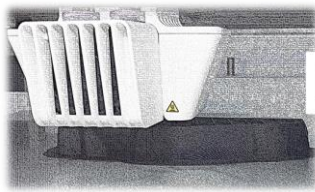

PROGRAMAÇÃO
DA IMPRESSÃO 3D

Carlos Relvas
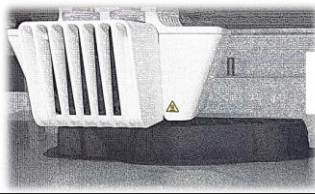
Índice

P

PROGRAMAÇÃO DA IMPRESSÃO 3D

SOFTWARE DE SLICCER

ULTIMAKER-CURA

Three stages of 3D printing

There are three basic stages to preparing files for 3D printing.

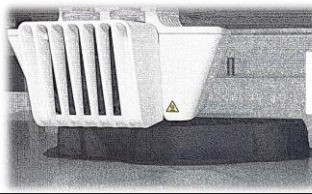1. **Modeling**: This is carried out in any 3D modeling application such as Tinkercad or SketchUp, which are just two of many example applications. These applications have their own file format and these enable you to open, edit, save, and export those 3D printer files from the application.

2. **3D file export**: Once you have created your model, it then needs to be exported as either an STL, OBJ, or 3MF file. These are the file formats that are recognized by Cura. They differ from the file formats that are native to the 3D modeling applications as they just hold the final geometry and not the individual primitives and editable content. Still, you can change the size of the 3D model, but not the geometry.

3. **Slicing file export:** The STL or OBJ file can then be imported into the Cura software where it is sliced and output as G-Code. This G-Code is just a text document (in essence) with a list of commands for the 3D printer to read and follow such as hot-end temperature, move to the left this much, right that much etc.

The first stage of the process requires 3D modeling, but if your modeling abilities are just in their early stages then you can pop along to sites such as Thingiverse, Cults, or MyMiniFactory and download millions of pre-made and print-ready models (more file repositories here). These are usually in the STL format and ready to be imported directly into the Cura software.

What does the Cura Software do?

Cura slices 3D models. It translates the 3D STL, OBJ or 3MF file into a format that the printer can understand. Fused filament fabrication (FFF) 3D printers print one layer upon another to build up the 3D object. Cura 3D takes the 3D model and works out how those layers are placed on the print bed and creates a set of instructions for the printer to follow — layer on layer.

Cura generates instructions for your 3D printer. They are called G-Code, a text document that ends with the file extension .gcode. Open the file and you'll actually be able to read through quite a bit of the code and understand what it's telling the printer to do.

Select/ Add a new 3D printer (machine)

On first loading Cura, you'll be asked to select a printer. If not, or if you want to set up a new printer, then select *Settings > Printer*.

You'll now be confronted with a selection of many printers. If you downloaded through the link at the top, then the listed printers will all be Ultimaker. For all other printers click Other and if you're lucky then your printer will be listed.
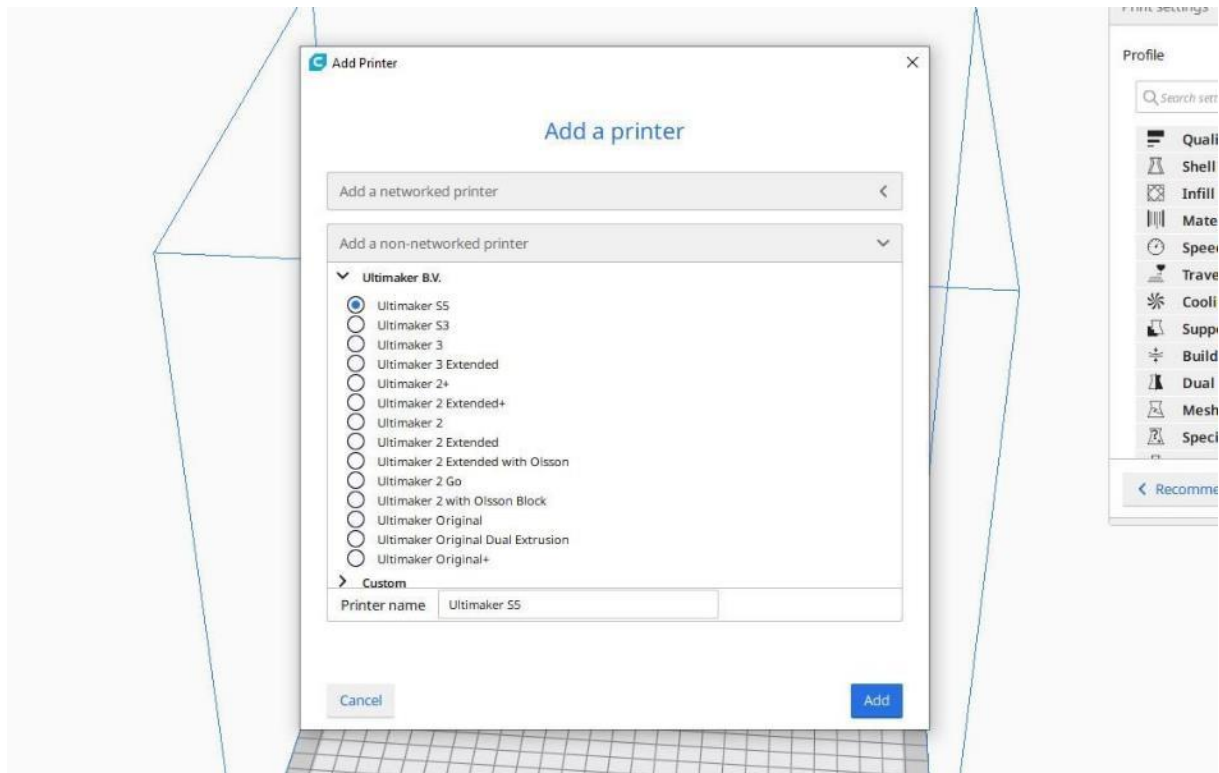

If not, head along to the manufacturer's website and you may find that there's a custom version of the Cura software (or a Cura profile) ready to download. If not, then select Custom and Add Printer.
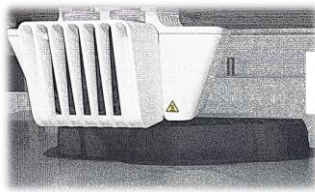
You'll now be shown the Add Printer screen and here you'll need to know a bit about your printer. Again, details should be found on the manufacturer's website. If you built the printer yourself, then you should know these details off by heart!
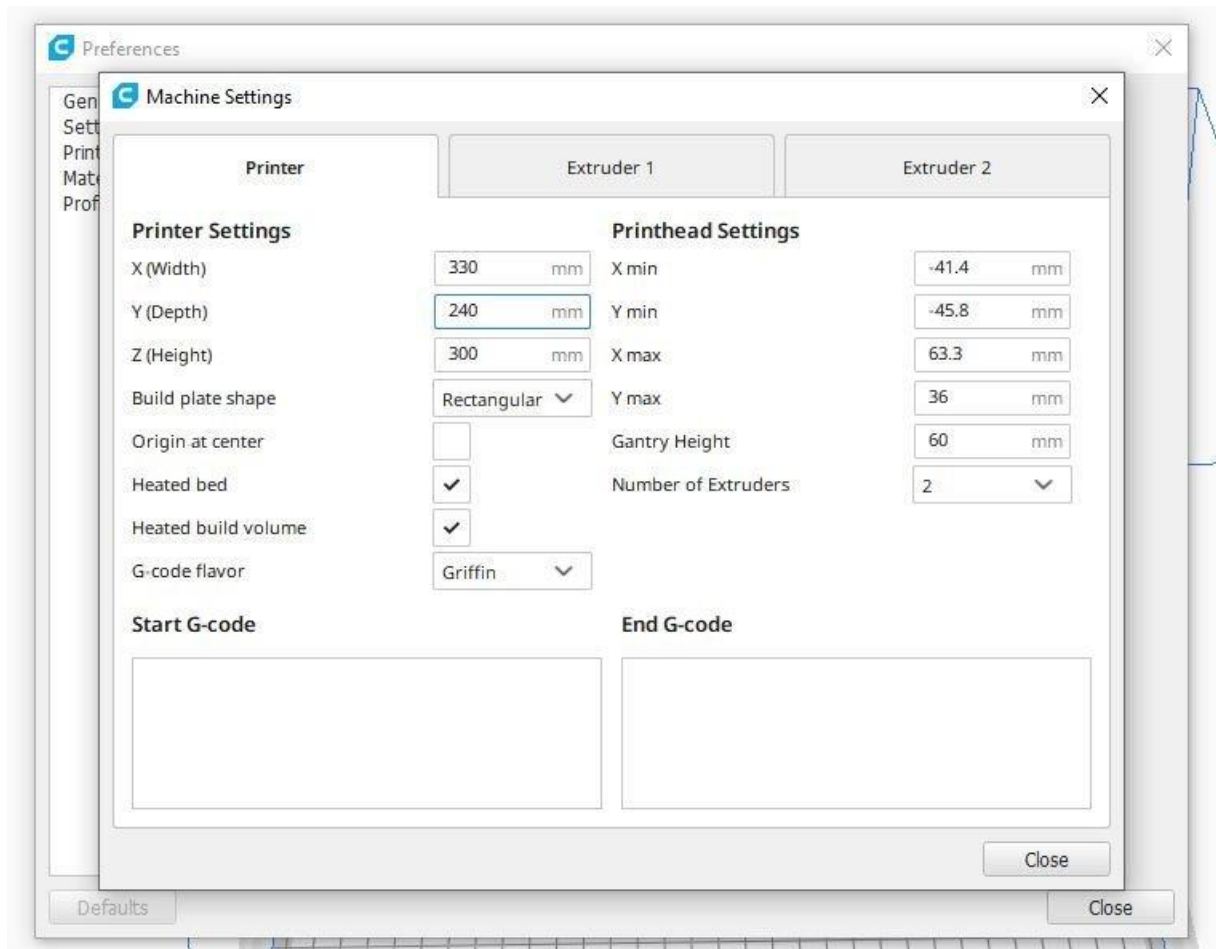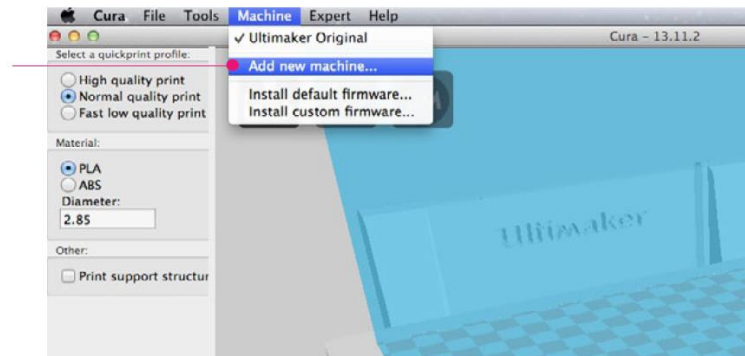

Just enter the settings for your 3D printer in the Cura Machine settings window.
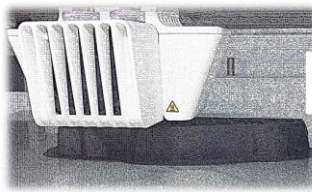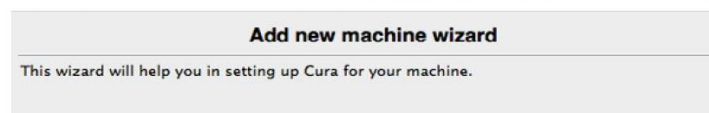
For using various 3D printers Cura has a build in function to work with different Ultimakers or 3D printers. So if you had the Ultimaker Original and now like to add the latest Ultimaker 2 in Cura in works as follows.

1. Under Machine in the menu bar you select **Add new machine**.
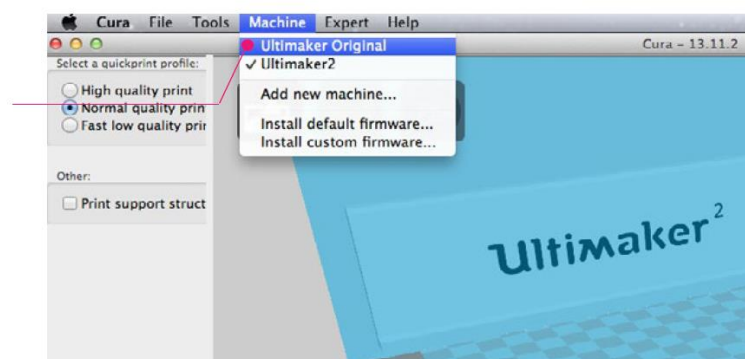

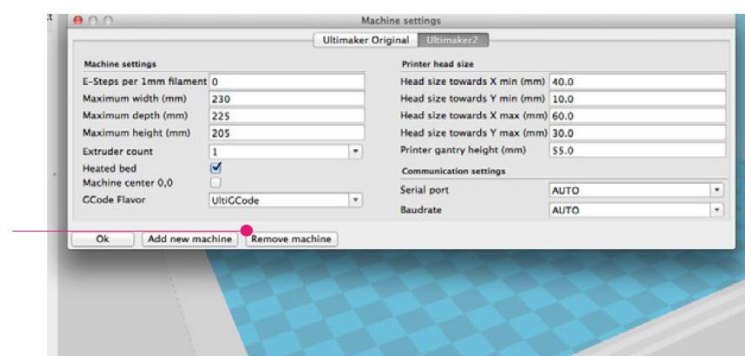**2.** Select next on the add new machine wizard step.

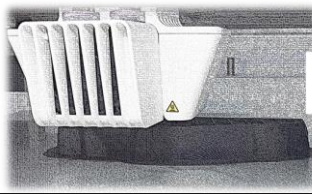**3. Select the machine** type you would like to select.
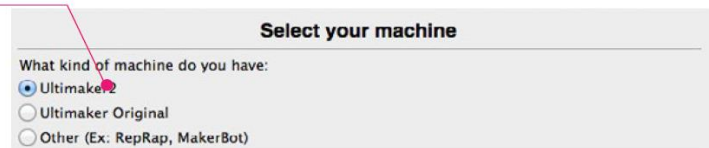
**4.** New you can switch by selecting the right machine, through machine in the Menu bar.

5. You can remove a 3D printer machine by going to File in the Menubar and select Machine settings, on the bottom of the pop up you find the function **Remove Machine.**
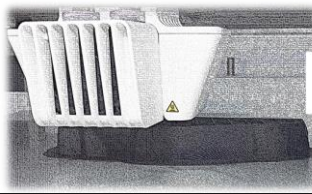
Cura software interface

This is the main 'quick print' screen of Cura. Here you can load and adjust 3D models, choose print profiles and upload files to the YouMagine library. Below you can see a quick overview about all the items in the interface. Later on in this chapter they will be explained in more depth
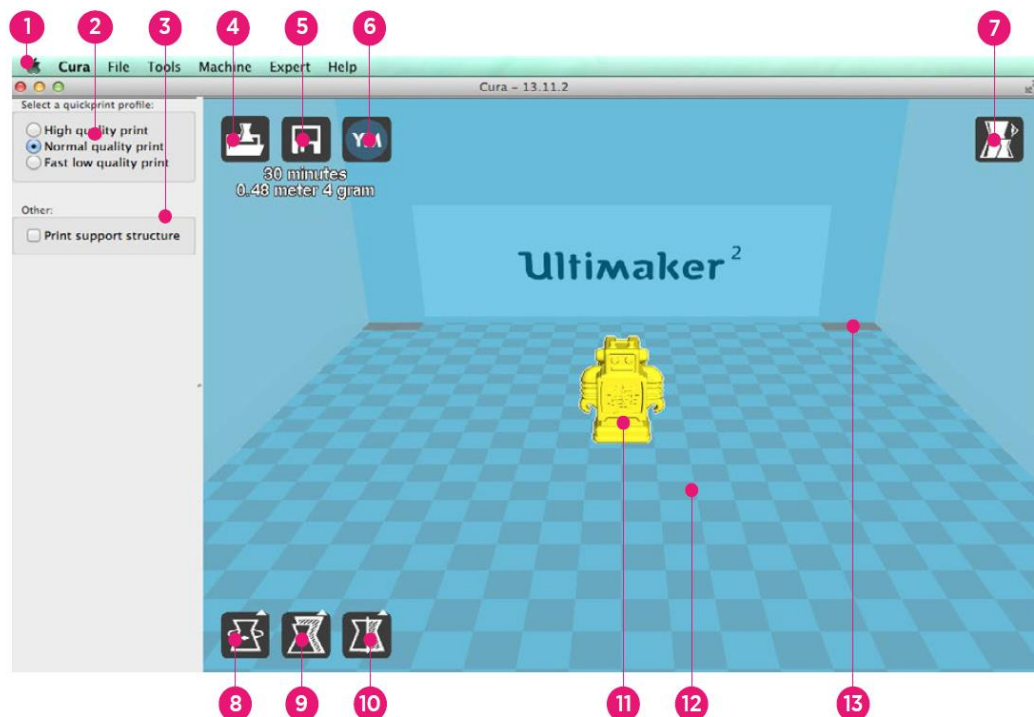
1. Menu bar In this bar you can change settings, machines and profiles.
2. Make a selection in 3 different quick print profiles.
3. The option to print with support structure.
4. A button which gives you the opportunity to load objects.
5. With this button you can save prepared files to your Ultimaker SD-card.
6. Through this button you can share 3D files on YouMagine.com.
7. A prepared model can be viewed in other modes to check it's printpath.
8. The option to change the rotation of the object you like to print.
9. The option to change the Scale of the object you like to print.
10. The options to Mirror the model you like to print.
11. The model you have loaded through the load file button.
12. This is a visualisation of the print area of your Ultimaker.
13. (ultimaker 2) The grey squares in the build area are the no go zones. In your Ultimaker 2 these are the metal clips were you can't print.

Adding a 3D printer model to Cura

The left icon on the top of the 3D interface is the Load file button.
Through this button you can search in your files to the following 3D file extensions: .stl .obj
.dae .amf
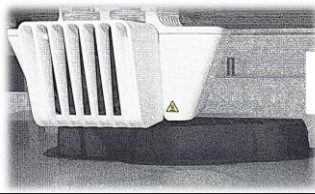
Once you have set up Cura for your printer, it's time to import a model into the Cura
software.

To import a model, you can either click on the floating folder icon on the left or
select *File > Open File(s)* from the top menu. Select an STL, OBJ, or 3MF file from your
computer and Cura will import it.
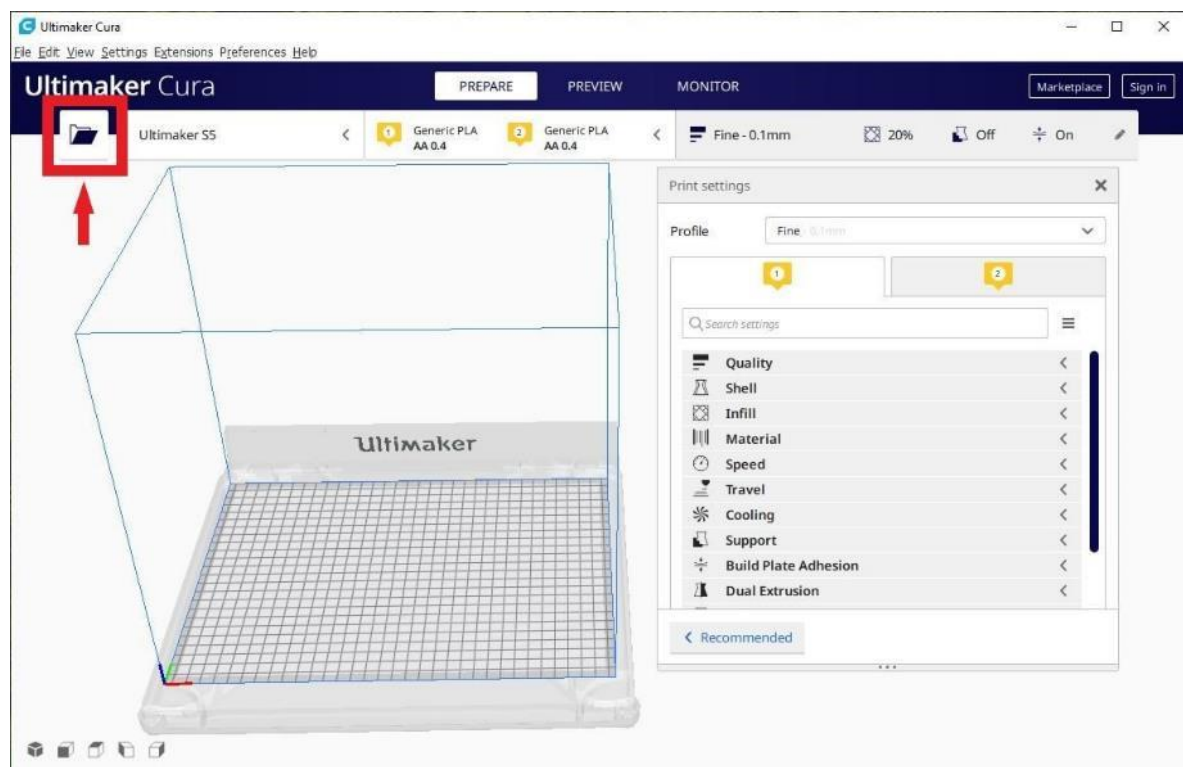
Wait a little bit and the model will appear on the Cura build area (the box in the center).
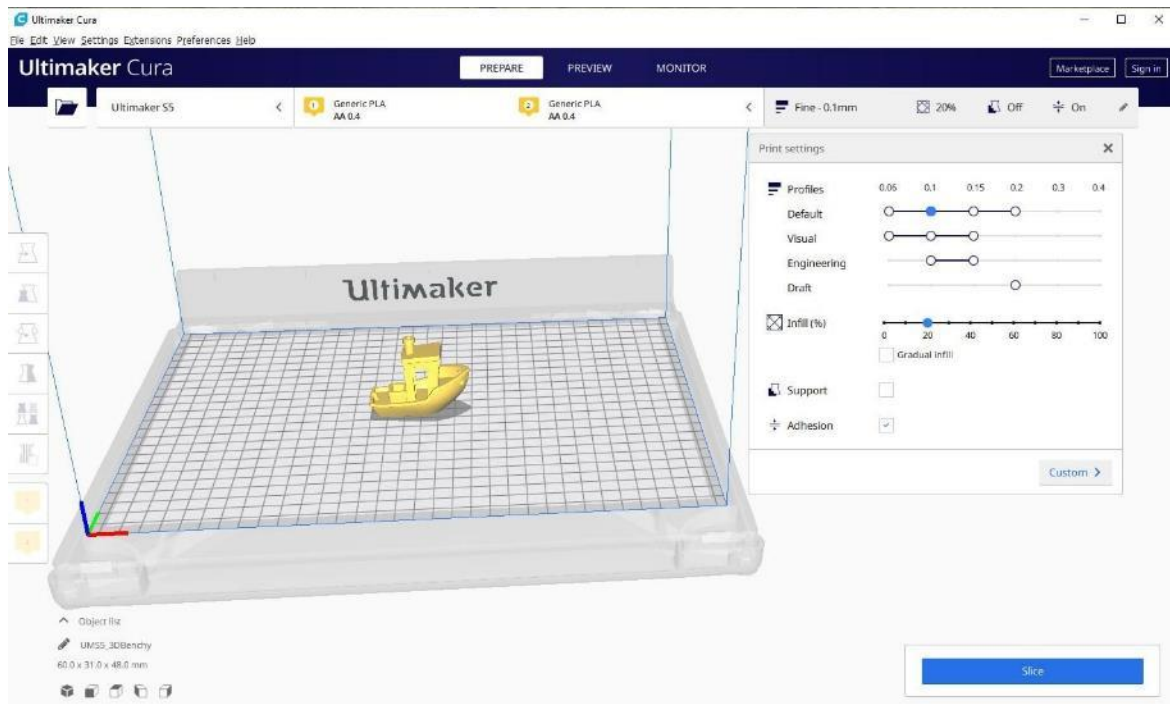
Basic orientation

The following mouse actions are used to work, navigate and view the 3D model.
You can use those orientation movements in the blue 3D interface.

Leftmouse button
Select objects. Hold and move
the mouse to drag
object on the 3D print area.

Scrollwheel button
Use the scroll wheel to zoom in
or out.

Rightmouse button
Hold and move the mouse to
rotate the viewpoint around the
3D model.

Rightmouse button + Shift
Hold and move the mouse to
pan the 3D view

shift

**Right and left mouse
button**
Hold and move the mouse to
zoom

Moving, scaling or rotating the model on the build platform

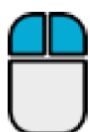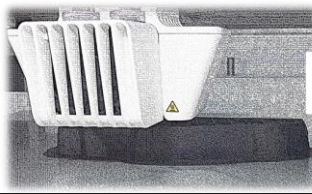Sometimes, you may want to move the model along Cura's build area because you don't want to print the model right at the center of the printer platform. At other times, the imported model might have the wrong orientation on the build area.

If your model needs adjusting, all you need to do is click on the model so that it is highlighted and then select one of the options from the tools on the left. Here you can quickly move, rotate, and scale the model.
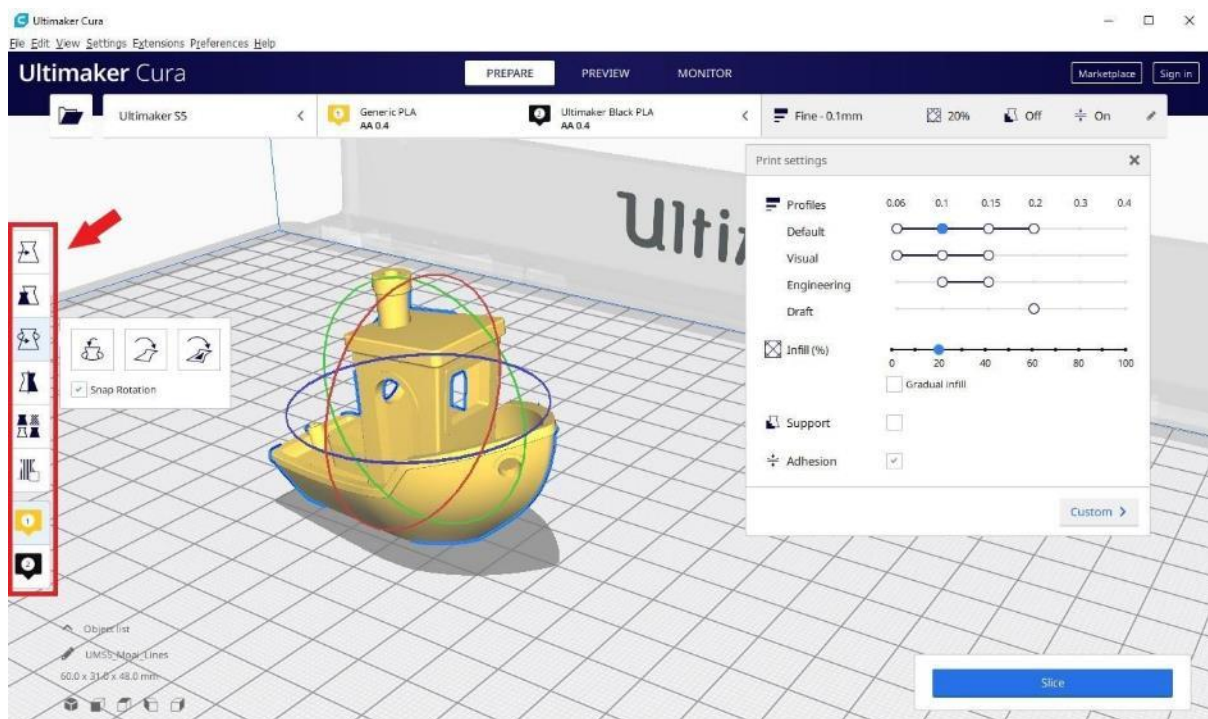
When you click on any of the Tool options in Cura, you'll see the arrows appear around the model. Just grab an arrow or hoop to make the change in the direction that you want. If you go wrong then you can just right-click and select Reset.

You might want to print more than one model. With the model selected, right-click and select duplicate. Cura 3D will automatically reposition the models. If there's enough space to print two or more, then all models on the platform will be yellow. If there isn't enough space, then the model out of the print area will be shaded gray.
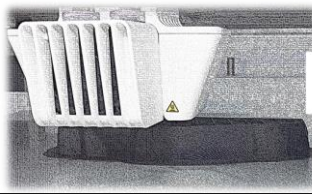
When you have loaded your model you can change the size or orientation, the following steps explain you the basic on how you can adjust you model on how you want it to be.

Rotating your object

The left icon on the bottom of the 3D interface is the rotation button. When you select and click it, you can rotate the model over it's XYZ axis. You see also more functions when you have selected the rotation button. the top icon's action, lays your model flat on the surface, to make sure your model is well attached to the build plate while printing. The second icon resets the 3D models rotation. By click-select one of the 3 orientation circles you adjust the rotation of the model. The rotation degree appears in the number around the model. When rotating and clicking shift you rotate per degree otherwise it's per 15 degrees.

Cura's settings panel

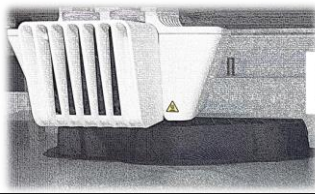Perhaps the most important part of the Cura window is the settings panel on the right. You need to choose the correct settings in this panel in order to get your desired print quality.

Cura's settings panel is divided into two sections. The topmost section is the Printer Settings and the next section is called Print Setup.

PRINTER SETTINGS

This section lets you select the right printer and material.
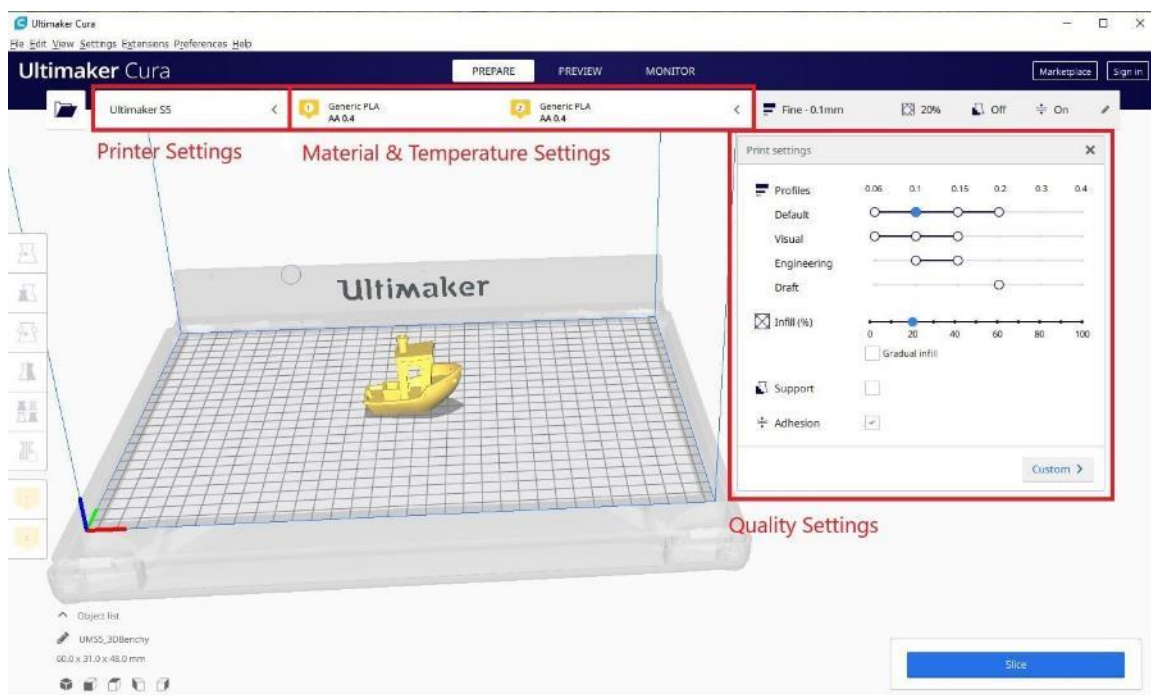
Printer: This is the printer that you selected in the first step. If you have more than one printer, then these can be set up, and then selected from this dropdown menu.

Material & Temperature: Quickly select the material and nozzle that your printer is using, and temperatures will be automatically adjusted.

PRINT SETUP

There are two options: Recommended and Custom

Recommended: The Recommended print options are calculated on the settings you input when initially configuring the Cura slicer for your printer. This option is a great choice when you're just starting out or you just want to see how the software and printer communicate. Options are limited under the Recommended header, but you can quickly adjust quality, infill, plate adhesion, and basic support structures.

Custom: This is where the fun really starts and will enable you to adjust the print settings – from quality through to speed. We'll look at this section and the options a little later.

The Recommended settings

LAYER HEIGHT
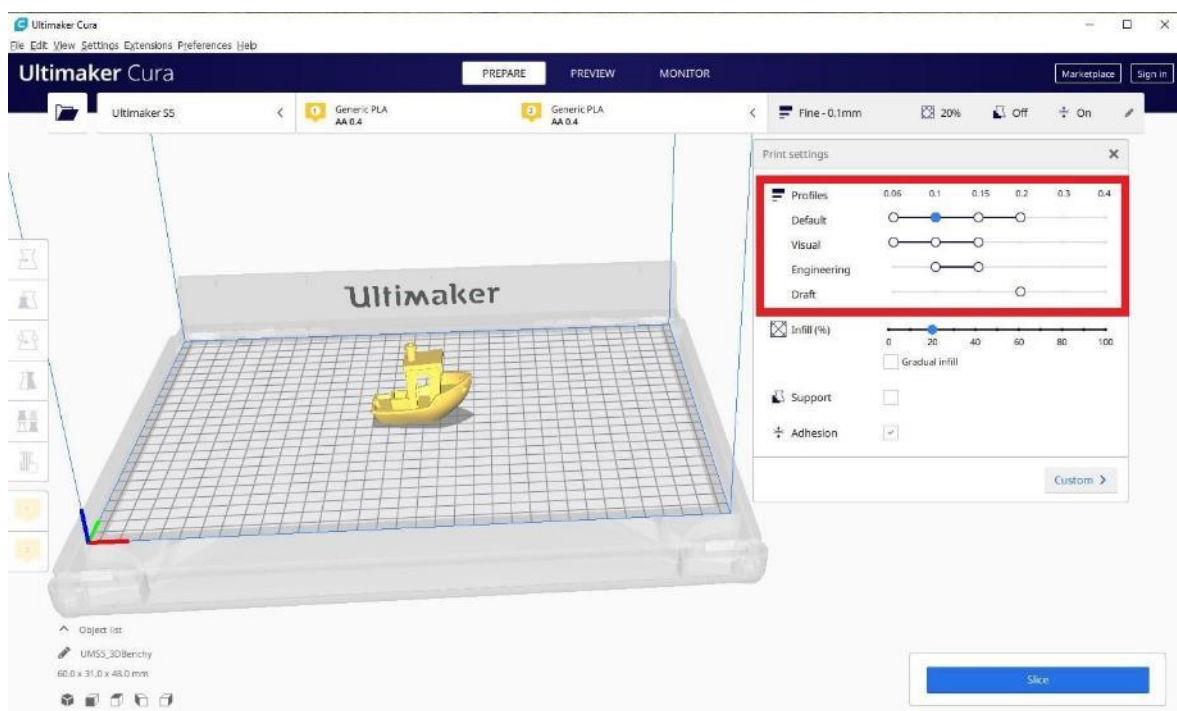
Change Layer Height according to the desired print quality

As we have already discussed, 3D printers print an object by depositing layer after layer of material. The Layer Height slider in Cura controls the height of each individual layer. Here, the rule is: the lower the layer height, the better the print quality and vice versa. But note that setting a low value for Layer Height means that the print is going to take proportionally longer to complete. You need to make a trade-off between quality and print speed and pick your own sweet spot. 0.1 mm is a good starting point.

INFILL IN CURA
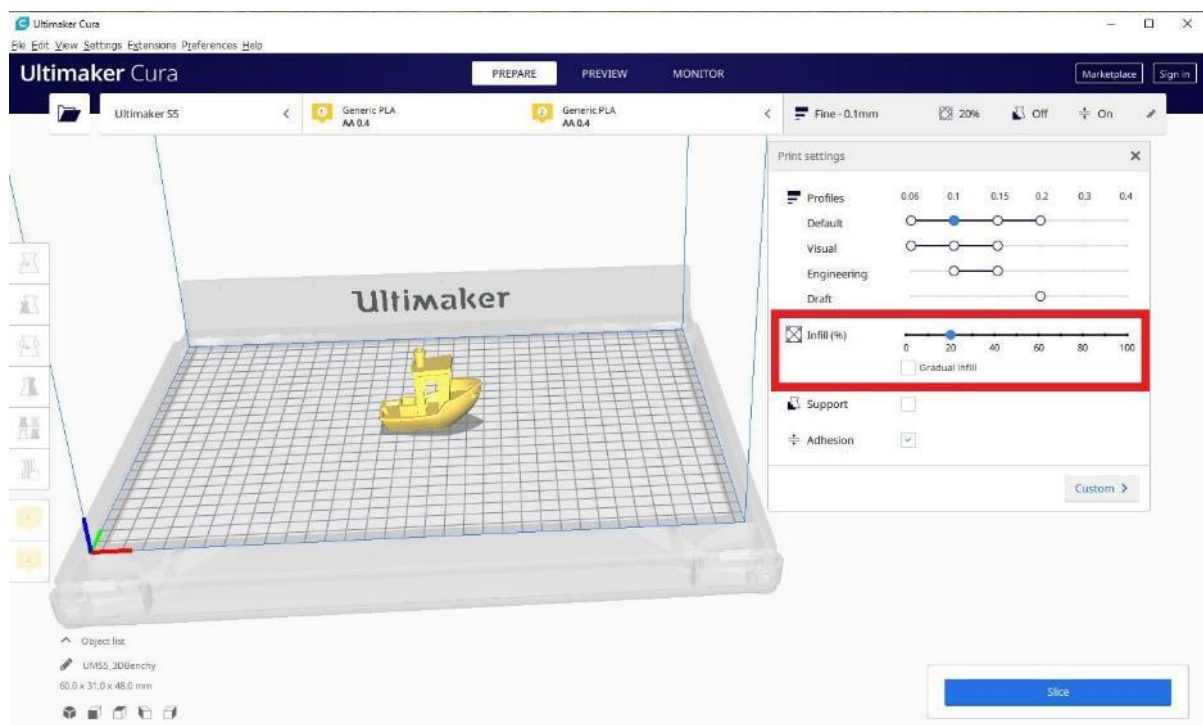

Change Infill Density to control the sturdiness of the model

The Infill slider controls the quality of the infill. Setting it to 0 % essentially means that you don't want any infill and want your object to be hollow. Anything in the range of 10% – 40% is known as a light infill. The 50% – 90% range is called a medium infill. Setting the slider to 100% will produce the strongest model. Light is a good starting point.

When the Infill slider is set to above 0%, a checkbox titled Enable Gradual appears. Checking this box will make Cura gradually increase the amount of infill towards the top of the model. This lets you use a low value for Infill and still get decent top quality. It's recommended that you check this box when using low values for Infill.

HELPER PARTS IN CURA

These are your support and adhesion settings – controlled by two checkboxes titled Generate Support and Build Plate Adhesion. If this is your first print, then switch both on. As a rule, if your 3D model has plenty of contact with the print platform then switch off Build Plate Adhesion. If your model has no overhangs switch off Generate Support in Cura's Helper Settings.

Basic settings

Quality

The height of each layer. For print quality and printing time this is the most important setting. Usual settings are 0.2mm for a low quality print. 0.1mm for a medium quality print. 0.06mm for a high quality print. And 0.02mm for an ultra-high quality print.
The thickness of the side shells, when printing a simple cube, this is the thickness of the side walls. Increasing it improves the strength of the part.
Retraction is pulling the filament back when moving over a gap in the print. This reduces the amount of thin lines between printed parts. These thin lines are called strings. Retraction is usually always enabled, unless you want to print faster or are printing with a material that does not allow retraction. Some models don't require retraction, this vase for example, because there are no gaps
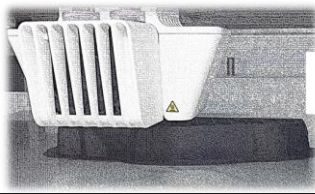
Fill

The bottom/top thickness is the outer shell thickness on the top and bottom. For example, when printing a simple cube, this is the bottom square and top square thickness that are put down. Increasing this will make a stronger part, and depending on your model, it will make for better solid tops.
Cura fills the internal parts of your model with a structure. This grid is made for strength and to support the top layers. The amount of infill you want is influenced by this setting. More infill produces stronger parts that take longer to print. If strength is not a requirement then this setting could be put on 5% for a low density infill that can still support the upper layers.

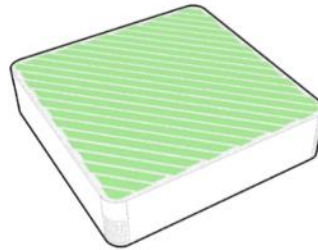O "Fill" descreve como uma parte sólida deve ser preenchida com material durante o processo de construção. São oferecidas três opções de enchimento com propriedades diferentes. Sólido, alta densidade e baixa densidade.
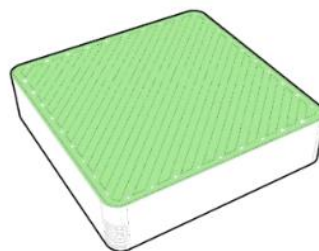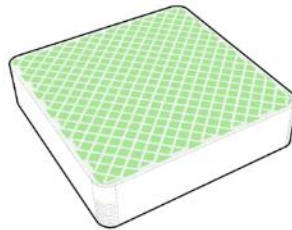

Sólido

Peças com interior sólido estão completamente densos. Esta é a opção mais pesada e que consome mais material.

Baixa densidade

Peças com interior baixa densidade são preenchidas com um padrão de trama cruzada. Esta é a opção mais ligeira e com menor custo de material.

Alta densidade

Peças com interior de alta densidade são preenchidas com um padrão de trama dupla. Esta opção proporciona maior rigidez e é mais leve que um interior sólido completo.

Print speed

Print speed sets the speed at which the print is put down. The default of 50mm per second is a bit low for an Ultimaker. But this is a safe starting point. People have printed up to speeds of 120mm per second. But this requires a well calibrated and tuned machine.

Support

Supports are structures printed below the print object to support parts that otherwise would be unprintable. There are 2 options, support structures that need to touch the build platform, or support structures that can also touch the top of your model.
The platform adhesion type is a setting to help the printed object stick on the printer bed. Large flat objects might get lose from the printer bed because of an effect called warping (curled up corners in the print). There is the option to use a raft, which is a thick grid under the object which scars the bottom of your print. Or a brim, which are lines around the bottom

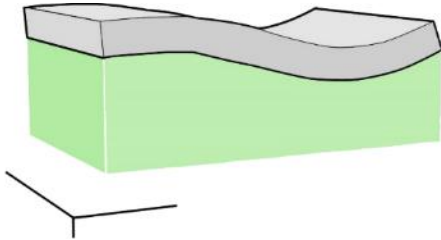of your object and because of the larger area the corners are kept down. Brim usually gives the best results as it does not scar the object. But it requires more space on the printer bed.

Suportes

FDM é efetuada numa plataforma de construção. Uma vez que os modelos são "construídos no ar", devem ser ligados à plataforma de suporte para evitar o colapso. Este acessório é conhecido como o "apoio" e é necessário para qualquer modelo construído utilizando esta tecnologia. Além de manter o modelo no lugar, também permite a construção de elementos salientes. Após o processo de construção estiver concluída, o suporte é removido. O material de suporte em FDM industrial é solúvel o que evita deixar marcas no modelo final.

Suporte externo
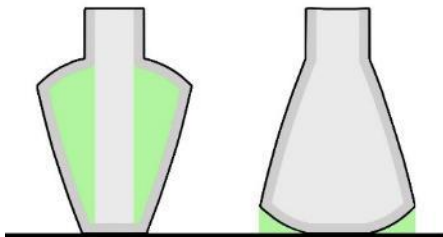
Para manter o seu modelo no lugar e evitar o colapso durante a impressão, os modelos com secções mais estreitas de 45 ° devem ser compatíveis. No exemplo, a parte inferior do vaso deve ser suportada, pois é mais estreita do que 45 °. O resto do desenho não necessita de suporte adicional, porque é mais larga do que um ângulo de 45 °.

Suporte interno
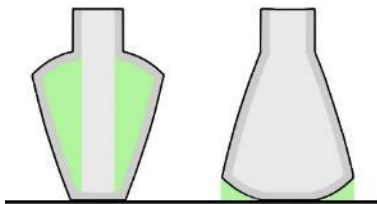
A regra dos 45 ° também se aplica dentro do seu modelo. Qualquer modelo com uma seção interna abaixo de 45 ° deve ser apoiada. No exemplo o modelo na parte superior deve ser apoiada para evitar o colapso durante o processo de impressão.

Advanced settings

Advanced settings are settings you usually only change once because you have special needs that do not match the defaults.

Nozzle size [1] : The size of the hole in your nozzle. The Ultimaker comes with a 0.4mm nozzle by default. Some people drill the nozzle to 0.6 or 0.8mm for faster printing at lower quality.

**Initial layer thickness** [2] : The thickness of the first layer. This first layer is set by 0.3mm by default. A 0.3mm layer gives a thick bottom layer which is easy to stick to the platform and allows for variation in the platform.

Cut off bottom [3] : Cut the bottom of the model, this effectively sinks the object into the printer bed. If your object does not have a lot of contact area with the printer bed then this feature could help you.

Travel speed [4]: The speed at which the printhead moves when it is not printing. The default is set on 150mm/s. But a well calibrated and oiled Ultimaker can go faster and up to 300mm/s. It is not uncommon for a printer to achieve 200mm/s or 250mm/s travel speed. But if you are seeing shifts in your print then the travel speed might be the cause.
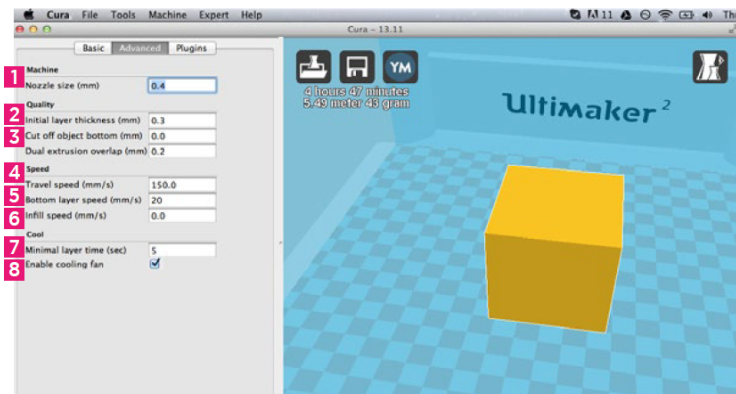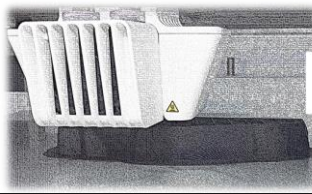
Bottom layer speed [5]: The speed at which the print head moves while it is laying down the first layer. This is done to make the print stick easier.

Infill Speed [6]: The speed at which infill lines are printed. If set to zero then same speed is used as for the rest of the print. A slight loss in outer quality can be expected if you use this to print a fast infill due to changes in nozzle pressure when switching between outside and infill parts.

Minimal layer time [7]: The minimal time spend on printing a single layer. If a layer takes less time to print then this configured time, then the layer is slowed down. This ensures that a layer is cooled down and solid enough before the next one is put on top.

Enable cooling fan [8]: The cooling fan is usually enabled and greatly improves print quality for PLA. For some other materials you might not want to use the cooling fan at all and this setting will disable it.

Generate a G-code file with Cura

The model is now print-ready and all you need to do is to export the file from Cura to either an SD card or send it directly to the printer. Cura will now handle everything converting the 3D STL or OBJ into the G-code file required by the printer.

1. **Save the 3D print file**: Click either Save to file, Save to SD or Send to Printer button on the bottom right of the window.

2. **Estimate of time for 3D print**: Cura will give you a rough estimate on the length of time it will take for your printer to print the piece.

3. **Start the 3D print**: If tethered, sit back and wait for the printer to fire up and start printing. If you save to SD, then eject the SD card from your computer and transfer to your printer. Select print, select the file, and go.
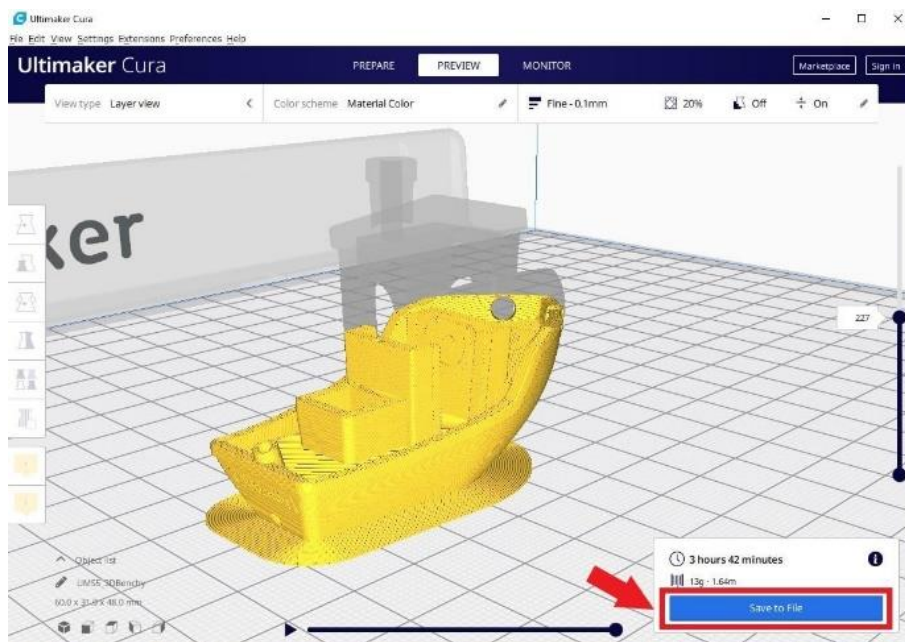
Printing the First Layer
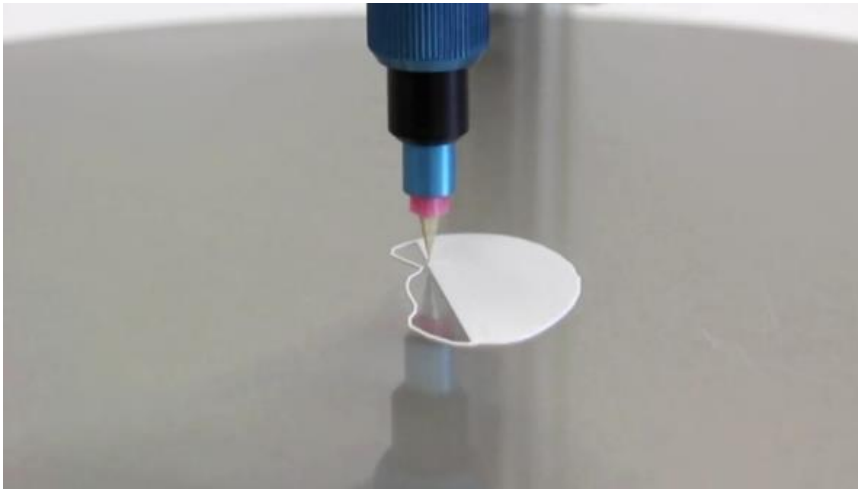
For fused filament fabrication (FFF), properly printing the first layer is important. Often blue painter's tape is said to b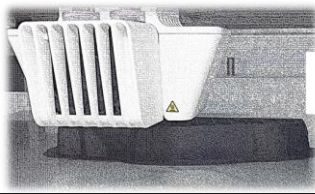e beneficial for making part removal easy and to help the first layer stick. However, if the bed is heated, the tape may not stick. Tape peeling from the build tray or overlapping can provide an uneven surface, which can cause dimensional instabilities and other problems. Also, using a heated bed, even for materials that do not require one, slows the cooling rate. This can prevent excessive shrinking, warping, and delamination from the build tray. Here are some suggestions to help you make the first layers stick.

- ✓ **Check operating temperatures:** Different materials have different melt and glass transition temperatures. Even switching can alter these temperatures. Heating elements in the bed may not achieve the proper temperatures. Putting a cloth or thermal insulating material over the bed until you are ready to print can help get those few extra degrees that might make the difference in firs- layer quality.

- ✓ **Check the build tray:** Make sure the build tray is clean. Oils or small debris can affect the adhesion of the first layer.

- ✓ **Level the build tray**: An offset build tray can cause not only a poor first layer, but can cause the print to fail completely later in the printing process.

- ✓ **Turn off the fan:** This will help to reduce rapid cooling that can lead to poor bonding

- ✓ **Surface finish:** Hairspray and sandpaper can help parts stick to the build tray, but there is another way. This technique comes with a caution as it can weld the part to the surface of the build tray if not done properly. Using acetone on a failed print and wiping the print on the build-tray surface will put down a layer of the polymer you are about to print with, and provide an easier surface to stick to. A little acetone and a light layer to keep parts from becoming welded to the build tray.

- ✓ **Slow the printer speed for the first layer:** It is not desirable to slow the speed, but it can help ensure the material is properly heated and pressed into the build tray.

✓ **Use the brim feature:** Brim is when a larger base layer is printed like a base flange around the base geometry that provides more surface area to be in contact with the bed. This can help the print to stick to the build surface and reduce warping. This is also suggested for tall prints with minimum surface contact with the build tray.

✓ **Adjust the Z-offset or flow rate**: By adjusting your Z-offset you can flatten the first layer, increasing the contact surface between the first layer and build tray. This can be done by increasing the flow rate. However, adjusting the flow of material may cause more problems with accuracy, and may not allow the materials to reach proper temperatures. Adjusting the Z-offset is preferred. After making adjustments, watch the print to see if the nozzle is dragging through the material. Dragging is a sign that the offset or flow is not set properly, and can also contribute to poor adhesion and poor prints. This technique is like adding a small brim on the part without turning on the feature. If this works you may want to consider a brim. Make sure the adjusted properties are only for this first layer otherwise it could result in a poor or failing print.

✓ **Thicker first layer:** Making a thicker, or lower-resolution, first layer can be great in case you are printing on an uneven surface, on top of an existing piece, or have problems removing it from the build tray. Thicker layers allow extra material for post processing to fix any surface finish problems that might have occurred during removal. Again, make sure the adjusted properties are only for this first layer.