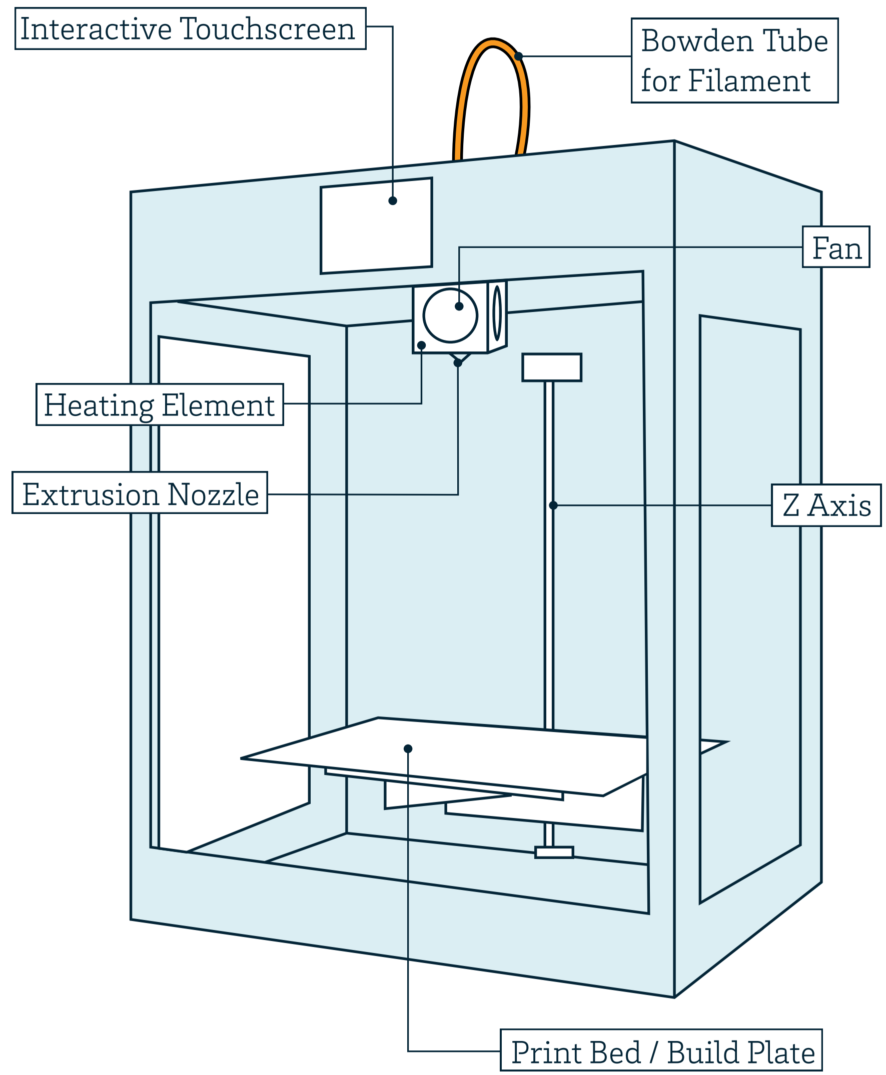
the Basics of 3D Printing

Setting up a model to 3D print requires a **slicing software** – think of it as a print menu. You can choose settings for your model and export the **G-code** file that tells the 3D printer what to do on each layer of your print. If you're interested, feel free to practice each setting while following along with this guide.

We based this interactive guide on Craftware, a free slicing software that is designed to be used with Craftbot 3D printers. If you are looking for a guide for a different slicing software, have questions about your new 3D printer, or need advice or troubleshooting help, don't hesitate to reach out to info@fluxspace.io and we would love to help you!



Get Started

Click on the folder icon on the upper left of your window to **Open/Import** a 3D model into Craftware. You can design your 3D model in a program like TinkerCAD or download it from thingiverse.com.

In the middle of the top of your Craftware window, you'll find three tabs:

CAD **MANUFACTURING** **GCODE**

The **CAD** tab is useful if you want to make any adjustments to your 3D object. You can use the top toolbars labeled **Object**, **Transformation**, **Printer**, **Support**, and **Primitives** to edit your model if needed. When you click on the toolbar, a popup window will appear to help you adjust the model - remember to hit **Apply** when you're done making your changes.

The **MANUFACTURING** tab is the tab you'll use most often. Here you'll select the settings that determine the quality, speed, strength, and density of your 3D model. When you're done setting up your 3D model, click **SLICE** in the bottom right corner.

The **GCODE** tab allows you to preview the print before it begins. You'll see path that the printer will take, the duration and material usage of the print. If you want to adust those metrics, go back and adjust the settings in **MANUFACTURING** and slice your model again!

Using the Slicer

Craftware includes several presets that will help you make easy decisions when you're learning how to 3D print. In the **MANUFACTURING** tab, the left menu includes these presets. At the top, select your printer and the type of filament you're using. Below that, you have a couple options for how to adjust your settings.

Smart **Easy** **Default** **Draft** **Fine** **Vase**

Smart and **Easy** are both simplified ways of adjusting many settings at once. In those tabs, you can use sliders to adjust your model settings. Click Reset to go back to the automated settings for both menus.

Default gives maximum flexibility and precision as you set up your 3D print. It includes every setting you might want to adjust for your print! If **Default** looks intimidating, use the other menus until you feel comfortable with each setting.

Draft and **Fine** limit the number of parameters you can adjust. You can also click Reset to use the standardized settings for both tabs.

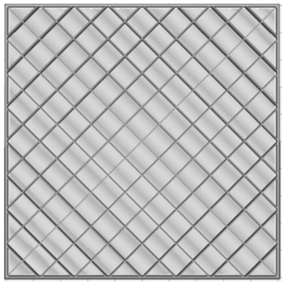
Vase Mode is a toggle option within the **Easy** setting menu, but you can also set up a Vase mode print within the **Vase** menu. Vase Mode is the a style of printing only the **Perimeter** of your 3D object. The result is a hollow model with a single layer exterior.

Material

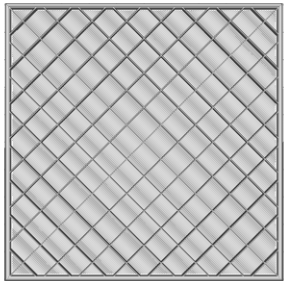
3D printer filament can be made out of different materials. Most frequently, filament is made of plastic, but you can also buy specialized filaments with interesting aesthetic or functional qualities (glow in the dark, conductive, etc).

Loop Count & Top / Bottom Shell Thickness

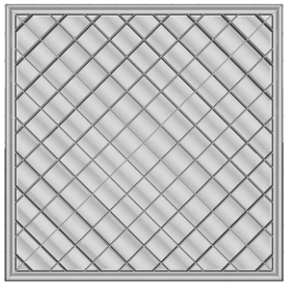
One of the custom controls available in Craftbot is to adjust the thickness of the walls, top, and bottom of your model. You might see these called Shells if you're using a different slicing software. Walls / Top / Bottom are measured in both millimeters and layers and can be adjusted independently of each other. For example, the tiles below have wall and bottom thickness as indicated, but a top thickness of 0 layers, so you can see what the inside of the model looks like. A greater number of layers will make your model stiffer and stronger, while fewer layers will make your model print faster and use less material.



2 layers



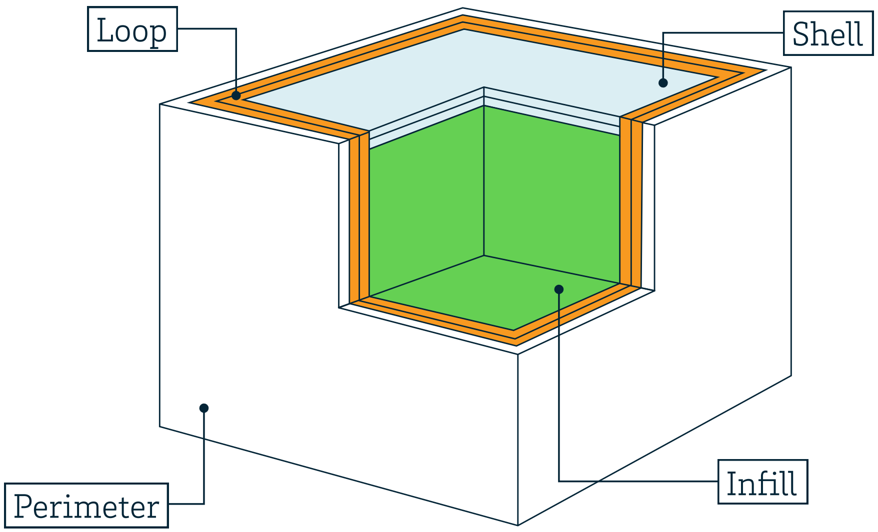
4 layers

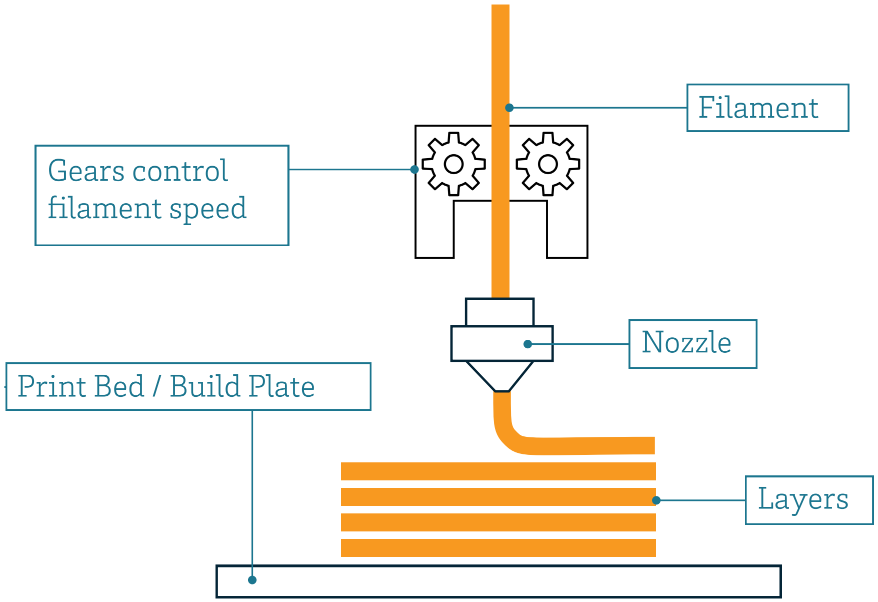


8 layers

Extrusion Width

Extrusion Width is measured as a percentage of the nozzle diameter. The nozzle is a small, usually brass colored, part of your 3D printer's extruder. The standard nozzle size is 0.4mm, but 3D printers usually came with nozzles of several different sizes! A smaller nozzle will let less material out, so walls and layers in your model will be thinner, it will take longer to print, but your print will be of a higher quality – imagine a high quality photo with small pixels. A larger nozzle will let more material through at once, making walls and layers thicker, reducing print time, and reducing overall quality.





Feed Rate & Speed

Feed Rate and Print speed are different names for the same metric, measured in millimeters per second. This is one of the most important and trickiest metrics to get right in order to have good 3D models. Printing too slow means that the plastic will burn instead of cooling down, while printing too fast causes filament to stretch out and leave streaks and holes in your print. Each material will have different speed requirements, and just like with temperature, you might need to do some research to determine the perfect speed for your printer.

Temperature

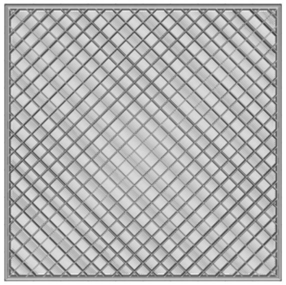
Different filaments melt at different temperatures, and these temperatures are usually printed on the spool of filament. Sometimes, you'll need to make little adjustments to accommodate your particular printer in order to get a perfect model. The two metrics used to adjust 3D printer temperature are: **filament temperature**, which refers to the temperature that the filament is heated to when it is being extruded through the printer head, and **bed temperature**, which refers to the temperature of the print bed, which helps the model with bed adhesion, and is usually much lower than printing temperature. If your bed plate isn't heated, filament will cool down too fast, and impact the shape and strength of your model. It's important for both the extruder and the print bed to be the right temperatures, so that they collaborate to prevent any model warping.

Retract & ZHop

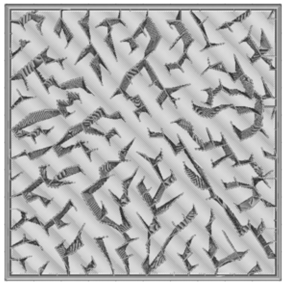
Sometimes, the 3D printer extruder head moves but does not print – like when it finishes one layer and moves to start printing the next layer – this is called travel. During this travel, a little bit of filament might be extruded and result in thin strings of filament stretching across your model. Retracting filament during travel removes the likelihood of this happening. You also have the option to add a Z hop, leaving some extra space between your model and the extruder head every time this travel occurs. This is a good option if you are printing something very fragile that you are worried might get knocked over during travel, but you generally don't need it.

Infill Pattern

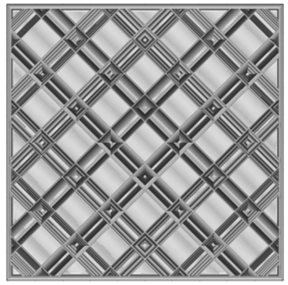
Slicing software like Cura generates infill patterns for the inside of your print. Each pattern has a unique combination of model strength, print time, and material usage.



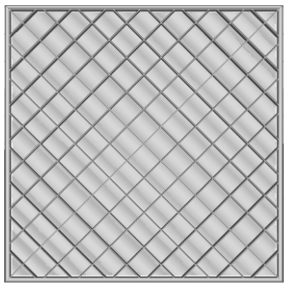
Zig Zag
Low 2D Strength



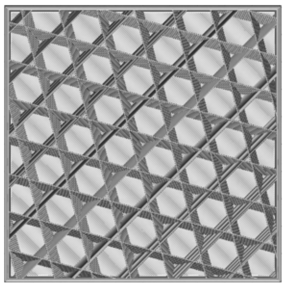
Lightning
Low 2D Strength



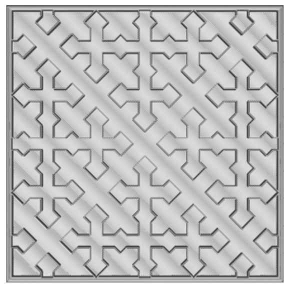
Octet
High 3D Strength



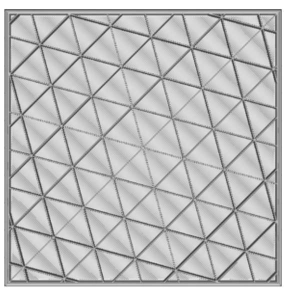
Grid
High 2D Strength



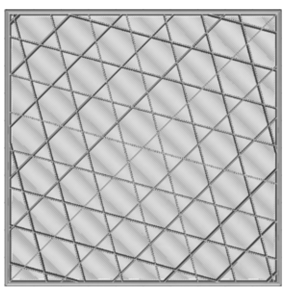
Cubic
High 2D Strength



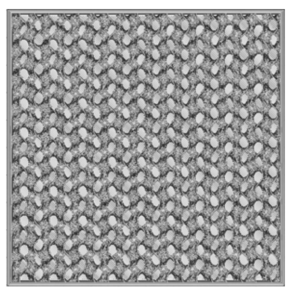
Cross 3D
High 3D Strength



Triangles
High 2D Strength



Tri Hexagon
High 2D Strength



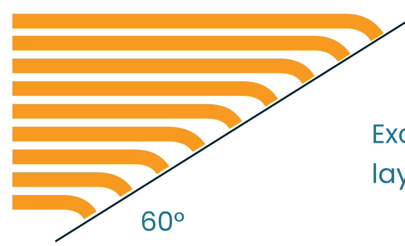
Gyroid
High 3D Strength

Adhesion

Adhesion helps your model to stick to the build plate. There are two main types of build plate adhesion. A **brim** is a thin layer surrounding your model to keep it in place as it prints. It can be removed with pliers or by hand. Brims are best for models that are flat on the bottom, and just need a little help gripping the build plate. **Rafts** are flat platforms that are printed underneath your model. Rafts provide a sturdy surface for support as your model prints, and are useful for stabilizing prints with small footprints.

Support

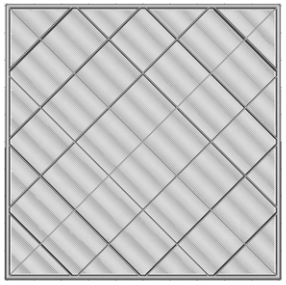
Supports are little columns used to hold up models with overhangs. Supports can be removed with pliers. In custom settings, you can define an overhang angle at which you want your print to be supported; the standard is 45 degrees from the vertical.



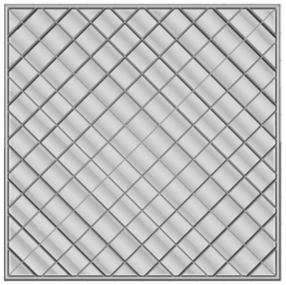
Excessive overhang: each layer of plastic will droop.

Infill Percentage

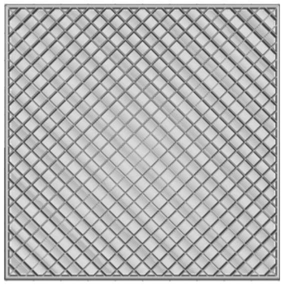
Infill Percentage refers to the density of your model. You can choose to print 100% solid models, but keep in mind that the print will take a very long time and use up a lot of material. High Infill Percentage models are useful for printing functional hardware or parts that you want to screw into. Low Infill Percentage models are perfect if you are prototyping, reducing material costs, or trying to save time on your prints.



5% infill



10% infill



20% infill

Layer Thickness

3D printers put material down one layer at a time. The thickness of each layer (the height measured in millimeters) impacts the overall quality of the print. A smaller layer height means that the quality of your model will be higher – similar to the effect of reducing layer height. A larger layer height means that your model will print much faster, but the quality will be lower, like a very pixelated photo.

Ironing

Ironing is a unique feature included in Craftware. Ironing adds one extra, very thin, layer to the top of your model, such that the top surface of your print is extra smooth. This step is personal preference: it doesn't make a significant impact when it comes to duration or material usage, so it's up to you if your model needs to look extra smooth on top!