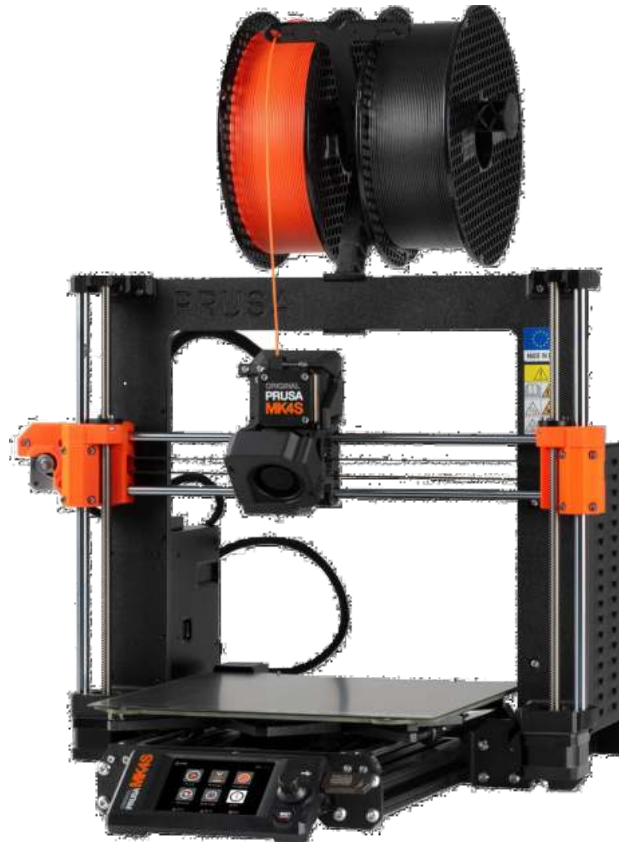


Prusa MK4 3D Printers



Introduction: Our 3D printers are capable of taking 3D models and printing them using various types of plastic, most commonly PLA. The process isn't the fastest, but these machines can create one of a kind custom objects.

General Printing		Filament maintenance	16
Getting a 3D model	1	Cleaning the print bed	19
Parts of the 3D printer	2	Cleaning the nozzle	19
Slicing your file			
Left sidebar	3		
Right Sidebar	6		
Supports	8		
Infill	9		
Starting a print	10		
Advanced			
Paint on Supports	12		
Support Styles	12		
Copy vs Instance	13		
Variable Layer Height	13		
Helper Discs	14		
Staff Use			

General Printing

Getting a 3D model

Many times you can find a 3D model ready to print from a variety of websites. There's often no need to reinvent the wheel with your project if your model already exists. Many times you can also modify those models to better suit your needs giving you a head start.

Regardless of where you find your model, it needs to be in an .STL, .3MF, or .OBJ

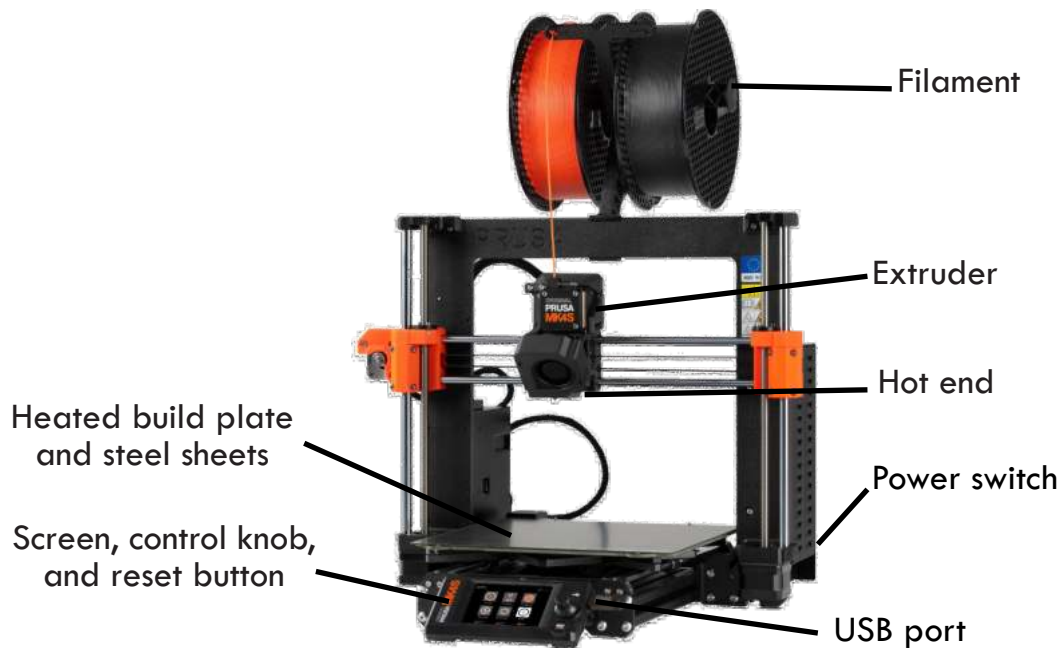
Below are a few places online where you can find ready to print models, though there are many more than listed here.

1. **Printables** - A newer site created by Prusa, though it's usage extends beyond specifically their ecosystem.
2. **Thingiverse** - The largest repository of 3D models. The first place to check when looking for designs.
3. **Yeggi** - This is a search engine for 3D models that will search across multiple sites including those listed above. Some of these are free and others are paid models, so you may have to pay to get the design while those above are free.
4. **My Mini Factory** - A website with a large selection of accessibility aids, as well as other more general use prints.
5. **Cults3D** - A website filled with lots of high detail and high quality models. Many require payment, though there are some free models on the site.

However, if you are interested in designing from the ground up, or even modifying an existing model, there are a variety of tools you can use. Training lessons for many of these can be found on either YouTube or LinkedIn Learning.

- **Tinkercad** - This is a free browser based 3D design tool and is very easy to use. Designed for students first diving into 3D modeling, it provides a simple environment for combining shapes together to create new shapes. This is the platform we often start patrons with design.
- **Fusion 360** - This is a more advanced design program from the same company as TinkerCAD. Available on the Maker Studio computers, this software is free for personal usage with a valid license. It is great for mechanical parts, but also has some ability to do organic shapes as well. There is a learning curve, but it's fairly intuitive once you get used to it.
- **Blender** - This is a very advanced 3D design program capable of high quality CGI designs. It's a free program, but contains a very high learning curve. This is best geared toward sculptural and organic works.
- **HeroForge** - If you're interested in figure work, HeroForge is a browser based platform intended for tabletop miniature games, but you can scale up your creation to print at larger sizes for full size figures.
- **SculptGL** - A free browser based sculpting program. Takes a bit of getting started with, but you can create high quality sculpted models.
- **Onshape** - This is a free CAD program in a browser that works similarly to Fusion 360. There is a paid tier that gives private file storage and additional features.

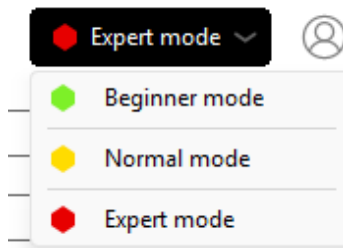
Parts of the 3D printer



Slicing your file

To slice your file into the code that the printer reads, we use a program called PrusaSlicer.

If you install PrusaSlicer at home, you can choose the setting that works best for you. However, we have all of our computers set for Expert mode which will give you the most tools and options to have available when slicing your design. Some of these options described below may not appear in Simple or Advanced.



Top Bar

Going across the top of your screen is a bar with a variety of tools. Most of the top bar is used for adding and arranging your models on the bed. Going from left to right we have:



- Add and Delete - Add will place additional 3D models onto your print bed. You can have multiple 3D models on one print bed at a time.

- Delete - This will remove the selected model(s) from the print bed.



- Delete all - Removes every file off the print bed.



- Arrange - These two tool options space all the models apart so there's room between them and nothing overlaps. Very useful tool to quickly arrange parts around the bed. The option on the left will arrange all of the models across multiple print beds if there are too many files to fit on one bed. The second one will arrange only the currently selected bed. You can further modify their positioning manually if desired.



- Copy and paste - These next two act like most programs. You copy the model and can paste additional copies. Once copied, the file stays in memory and will continue to paste additional copies until another object is copied.



- Add and remove instances - Similar to copy and paste, this will add additional copies of your model. Removing them can be done easily here as well. The difference between these two methods will be described later.



- Split into objects - This will take your model and cut it up into as many pieces as it can. Typically this does not work if the model was made as one solid piece. But if there are overlapping pieces that were not merged together, this will split them out. Note, the model may lose all the desired detail if this is used and the file wasn't designed properly for this operation.



- Split into parts - This is the method you should try first. If you have more than one piece in your model, this will take any non overlapping objects and break them apart into their own parts. This holds parts together better than splitting into objects, but only works if there are multiple models in one file.



- Variable Layer Height - This allows you to modify the layer height throughout your model. Typically used to smooth over the top layers or add more detail in particular locations. See the Advanced section of this guide on how to use this.



- Undo and redo - This will undo or redo actions taken in the program.

Enter a search term 

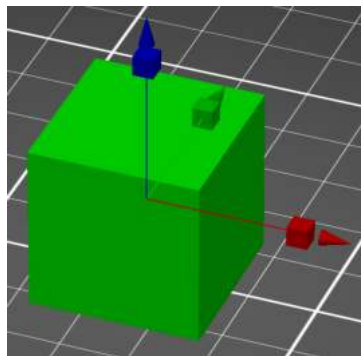
- Search - This sits above the current toolbar, but allows you to search for settings and options quickly when you can't find them inside the settings menus.

Left sidebar

The sidebar on the left is used to alter the model's properties from angle, size, and position. Going from top to bottom we have:

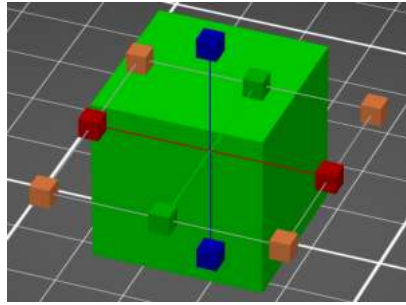


- Move - This lets you move the location of your object on the bed. Note: You can move along the X and Y directions all you want, but if you move your model up on the Z axis it will not stay there unless it is a piece of a collection of parts. Otherwise it always snaps to the bed itself.

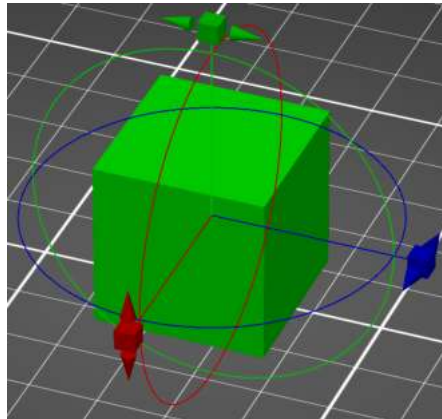




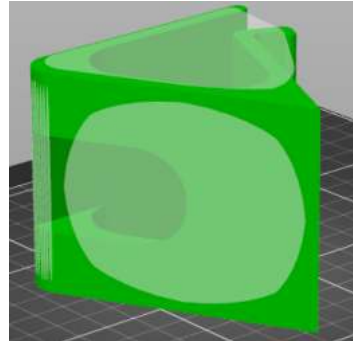
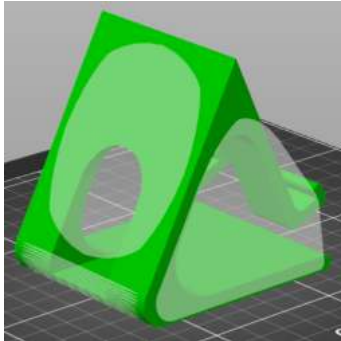
- Scale - This adjusts your model in one of three directions by using the red, green, and blue points, or in all directions using the orange points. Typically you want to use the orange points to resize.



- Rotate - This will rotate your model along the three axes. This can be useful in getting a better angle on your design or reducing support material needs. Inside the circle will lock into 22.5 degree rotations. Outside the circle is free form rotation.

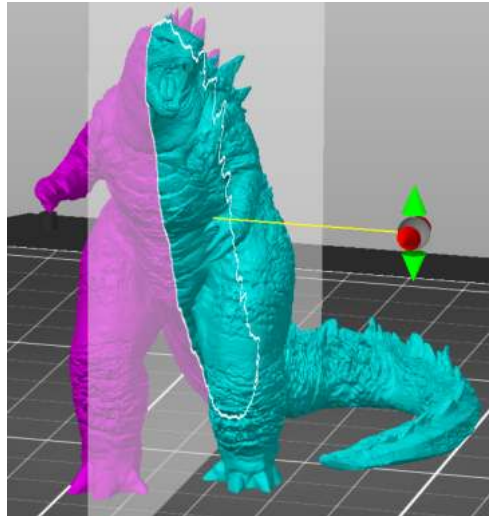


- Put on a face - This will auto rotate your model to put a software determined flat area right onto the bed. Useful for getting precise rotations to particular sides of a model to aid with printing or supports.





- **Cut** - This will cut your model into two pieces along a plane you determine. You can define the position and angle of the plane will cut and then arrange the parts. This is typically done to allow for prints that are too large to print or take too much time to be printed in pieces. They can be glued together after they are printed. This may not always be the best option for mechanical parts as the glued seam may weaken the overall model.



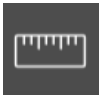
- **Paint on supports** - This allows you to define specific places to add or prevent support material. See the Advanced section of this guide for more information on this tool.



- **Seam painting** - This allows you to define the specific point that the seam of the print will be located. The seam is where it starts and finishes a layer and this can align that spot to a particular location to smooth out the appearance of a finished print.



- **Paint on fuzzy skin** - This will add a textured surface to the areas you choose. You can either add it with a brush or click on complete sides. This will rough up the surface which will lose out on detail, but can hide layer lines and improve texture if the project is meant to be held.

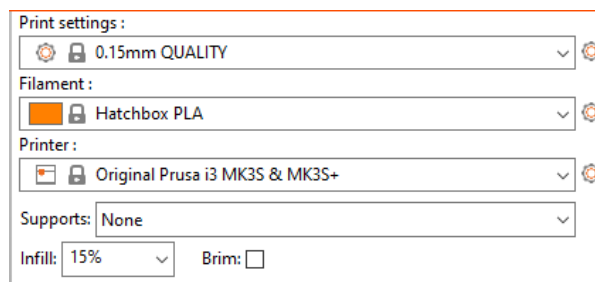


- **Measure** - This tool can be used to measure the distance between two points on a model. You can also then scale the model off this measurement by defining the needed distance between those two selected points.

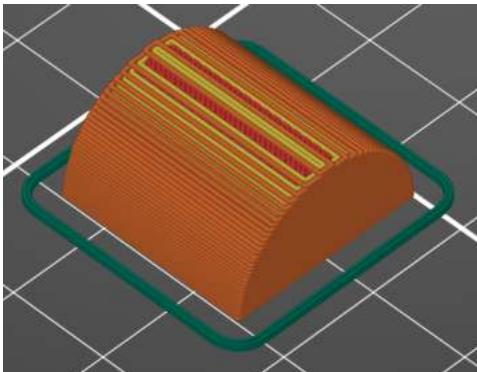
Right Sidebar

Here is where you can apply your print settings, define the size and position of your model more exactly, then slice and export your file.

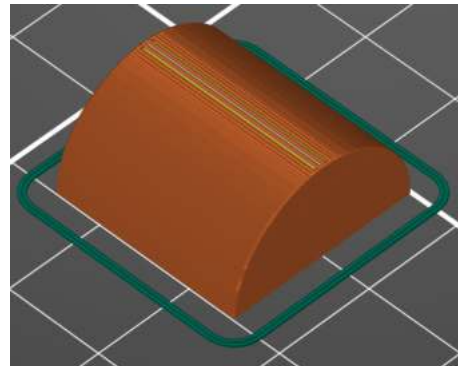
This section at the top allows you to apply settings to your model. These are the main parameters you want to adjust when printing to get the basic parts covered.



- **Print settings** - Here you'll define the layer height of your project. The sizes range from 0.1 mm to 0.2mm depending on the printer model. The smaller the number the more detail your finished project will have but will increase the print time. Printing a project at 0.2mm takes half the time it does to print at 0.1mm for instance. Depending on your model, some sizes may be better suited for the application. Straight and flat sides work best at 0.2mm but small details often require a smaller layer height to get the most out of the print. Note, the increased time does not often equate to more material used. The overall volume of the print is the same it just relates to the time required to print that volume. You can also go into the printer settings and dictate more specific and even smaller sizes but that requires experimentation to see how well those settings work.



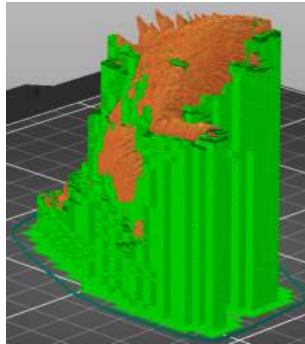
Printed at 0.2mm
11 minutes
5 grams



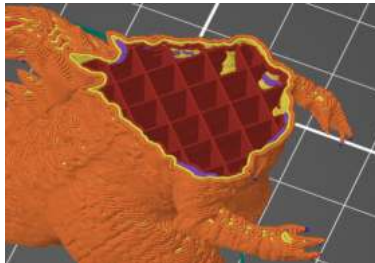
Printed at 0.1mm
22 minutes
5 grams

- **Filament** - Unless you're providing your own filament, this should be set for Hatchbox PLA as that is the filament the library uses.
- **Printer** - This ensures the code is written for the correct printer and makes sure the project doesn't fail immediately. As we only have one printer, you should not need to change this on our computers.

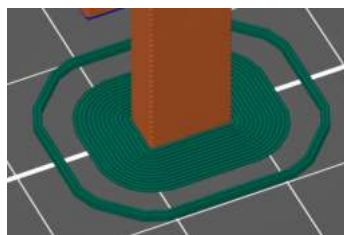
- Supports - This is where you can add support material. Supports are needed to help overhangs print correctly. The easiest solution is to click “Everywhere” in this drop down or potentially “Support on the build plate only.” Either option will generate supports, but depending on the shape of your model, one may be better than the other. See the Supports section for more information on support material.



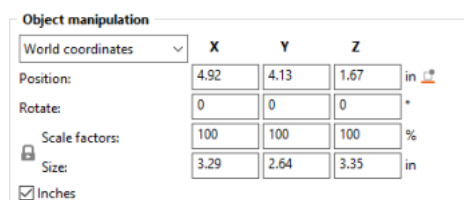
- Infill - Prints are not often printed completely solid or hollow, but have a certain amount of filament on the interior to help with structure and printing. By default it’s set for 15% and can be left alone unless you need something else. See the Infill section for more information on this feature.



- Brim - If your model is particularly tall and skinny or just has little contact with the bed in general, it can be useful to print a brim. This feature adds several rings of filament around the base of your model that can help with adhesion to help ensure your print doesn’t break off midprint. If your model has a wide and sturdy base you can often ignore this feature. It also is not advised for flexible models with a lot of tightly interconnected parts.



- Object Manipulation - Below the previous section is a set of boxes allowing you to alter the position, size, and rotation of your object with precise units instead of more freeform usage on the left sidebar. There is also a toggle to switch between millimeters and inches as your display unit of choice. By default in this area you will adjust all parts of the model at once if you change a box on the size. If you click the lock to unlock it, you can adjust just one direction of the model at a time independent of the others.



- At the very bottom of the right sidebar is your “Slice Now” button. This will take all of the settings you’ve applied above and generate the G-CODE that the printer reads. The sliced view will show you the supports, layers, and other settings. Once you’ve sliced it, that button switches to say “Export G-CODE” which then opens a dialog box to save your file on to a USB drive.



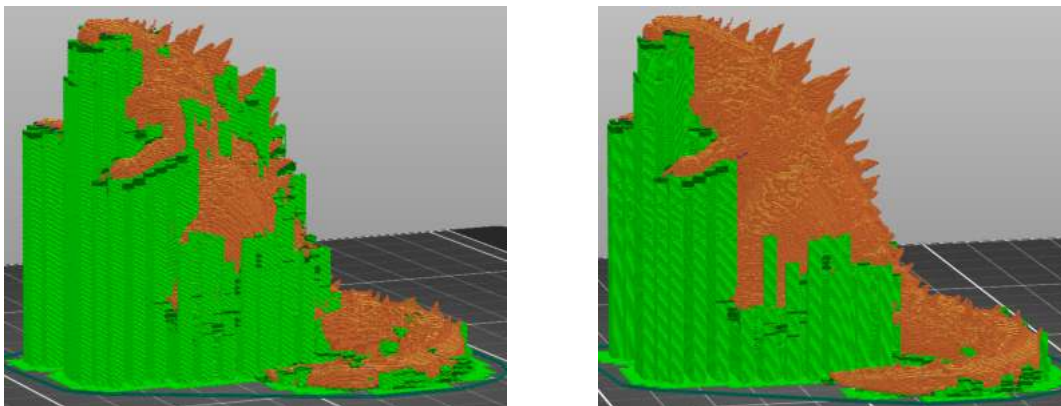
- This box at the bottom above the slice/export button shows some vital information. Namely the time it will take to print and the amount of filament used by the gram. The time is useful to know how long it will take. The used filament matters for the weight which is how we charge for a finished project. If you multiply the weight by \$0.05 you’ll get the estimated cost of your project. It typically is a little less than this estimate, but you can have an idea prior to starting what your project will cost at the end.

Sliced Info	
Used Filament (g) (including spool)	55.68 (300.68)
Used Filament (in)	717.58
Used Filament (in ³)	2.68
Cost	1.41
Estimated printing time:	
- normal mode	2h52m
- stealth mode	2h54m

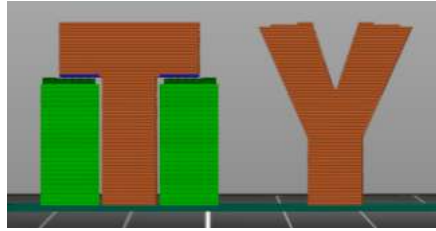
Supports

When 3D printing, overhangs must be supported from the bottom otherwise the filament will droop as it prints. The easiest thing to do is select “Everywhere” from the support drop down menu in the print settings. This will auto assign supports to everywhere the software determines supports are needed.

The second easiest is “Support on build plate only.” If you only support from the buildplate, the top side of your model will look cleaner over all, but if any parts need supporting that are not directly above the build plate, it won’t generate supports on it. This one takes a bit more finesse at times, but can help provide the support you need without as much extra clean up.

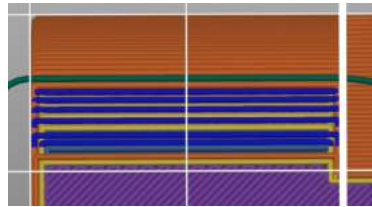


In the above example, the one on the left has supports everywhere which helps cover the spikes on the back, and some parts of the head. The one on the right is only supported from the build plate. There is less cleanup, but some spots on the spikes could need supporting. You would have to check the model and see what you think is needed.



Generally any part of your model that sits at 45 degrees and up does not need support material. Anything flatter than that will require it. If you look at the letters T and Y you can see that exemplified. The “Everywhere” setting may sometimes try to put the supports in that spot even if it doesn’t necessarily need them, so sometimes it’s worth looking at your model closely to decide if you need supports or not.

Typically, if you look at your sliced model without supports and see a lot of dark blue lines, this indicates areas the software recognizes will have trouble printing. Typically they end up having too much of a gap between the outer edge of the layer and the layer below it, resulting in drooping. If you see a lot of these, consider adding supports to resolve them.

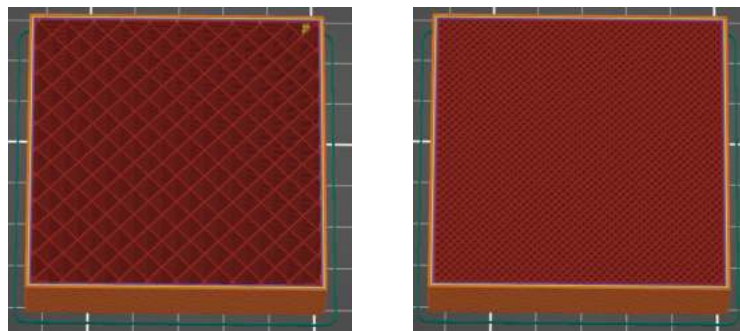


You can read more about generating unique and refined support material in the Advanced section of this guide.

Infill

The inside of 3D models are not solid but are also not hollow (unless you want them to be). Typically a model is filled with 15% infill, which is a percentage of the interior of the model that is filled with a pattern of filament to give it weight, structure, and stability. In most cases you can ignore modifying this part of your model, but there are reasons to consider it for both texture, weight, speed, and strength.

For instance, these are both the same model, sliced with all the same properties except infill. The one on the left is the default 15%, while the one on the right was modified to have 70% infill.



The one on the left would take 53 minutes to print and use 24 grams of filament. The one on the right would be 1 hour and 28 minutes using 50 grams of filament. However, the one on the right will be much more rigid and sturdy than the one on the left.

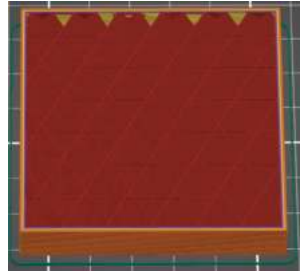
The following diagram will discuss a few more patterns and their benefits. They are all sliced using 20% infill.

Adaptive Cubic

50 minutes

21 grams

Useful for large prints with a lot of volume.

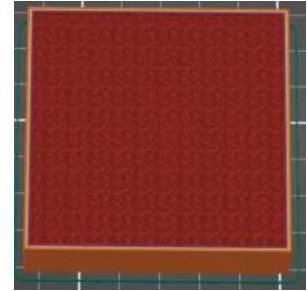


Gyroid

1 hour 10 minutes

24 grams

Strong in all directions, doesn't cross itself on the same layer.

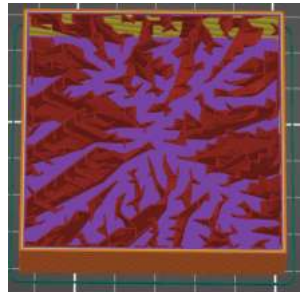


Lightning

1 hour

17 grams

Significantly faster with larger projects, uses notably less material, not as sturdy.

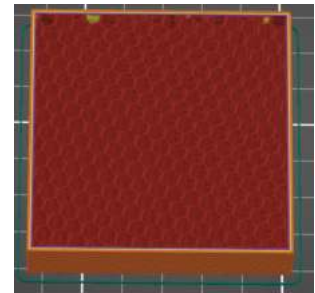


Honeycomb

1 hour 26 minutes

27 grams

Mechanically stronger, but higher time and material cost.



Starting a print

1. Take the G-CODE from the previous section and export it onto a USB flash drive.
2. Turn on the 3D printer using the switch in the back right of the machine.



3. Insert the USB drive side of the right screen housing at the front of the printer.



4. The printer will scan the contents of the card and display the most recently sliced file and a preview of what it looks like.



5. If this looks good, press on the knob to select the design and begin printing.



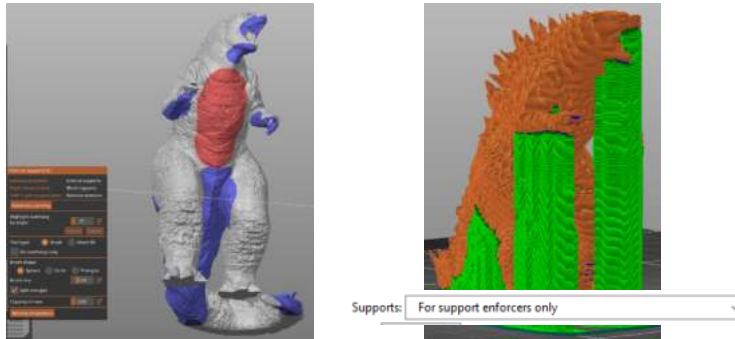
6. The printer will now begin calibration and then printing.

Advanced

Paint on Supports

With supports, the “Everywhere” setting might work for the vast majority of models, but occasionally, the software can be too eager with the amount of supports it prints. In this case, you can choose the paint on supports feature. This will let you determine exactly where you do and do not want supports to be.

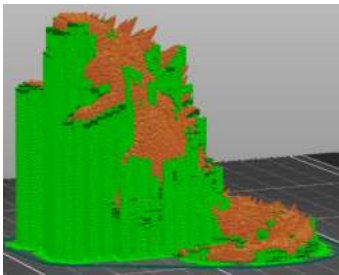
With this feature you can left click to place blue sections where you do want to place supports and right click to place red sections that will block supports from being placed.



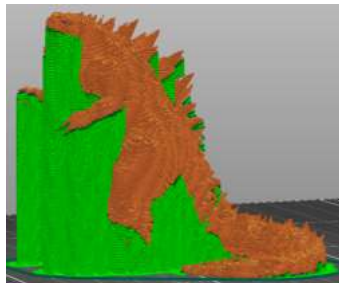
From the dropdown for supports, you will want to then choose the setting “For support enforcers only” to apply these settings as your support choice.

Support Styles

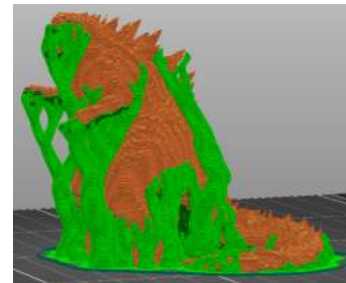
There are also different kinds of supports. The default is listed as Grid, but there is also Snug and Organic. Grid is very stable, however the supports tend to enclose a lot more of the object making cleanup more of a chore. Snug handles overhangs just as well, but there may be less support for tall and thin objects. Organic is a new support system that makes almost tree like branches around your object. These are easily removable, quicker to print than the others, and do not damage the surface. However, they are not as good when it comes to large flat areas that need supported and being a newer style, they may still need some development time on the software for any potential issues.



Grid
4 hours 22 minutes
35 grams

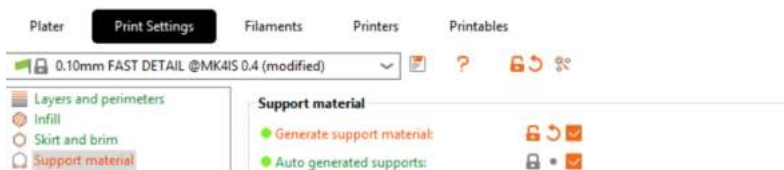


Snug
3 hours 24 minutes
30 grams



Organic
3 hours 7 minutes
32 grams

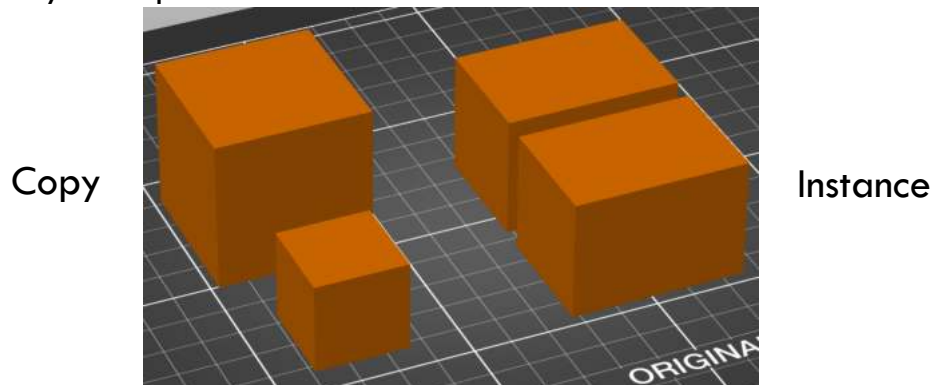
You can modify these settings by going to the Print Settings tab at the top, choosing the Support material option, and finding the drop down beside Style in the third option group. If you’re using the organic supports, you can further refine and tweak them in the Organic Supports section.



Copy vs Instance

To get multiple copies of your model, there are two ways to go about it. You can copy and paste the model multiple times, or you can use an instance.

The key difference between the two is how those copies can be altered. Copy and paste creates a new object. If you alter one of these, the other will remain as it was. However, with instances, if you change the size, change the face touching the bed, or cut the model in half, it will happen to all versions created through instances. Typically, if you need multiples of your project at different sizes, copy and paste is the better method. However, if you want identical pieces in higher quantities, instances can be a great solution to help you make changes quickly to a variety of objects.



Variable Layer Height

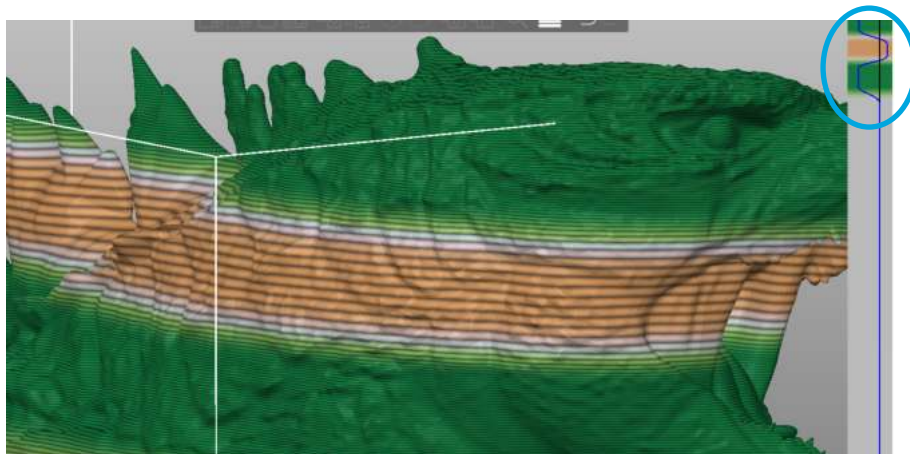
Layers are seen most at the top edge of a rounded surface or on a slope. If your model is more organic or has a notable spot where the layers are more prominent, you can vary the layer heights to best suit your project at each point. For instance, the straight sided body of this model are going to print fine, but the top edge of the head and jaw will show the layers more and could use some extra detail.

Note you cannot use Organic supports with variable layer height, just Grid and Snug.

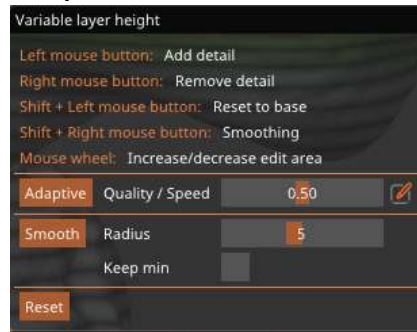
1. Set your layer height in Print settings to be the base layer height you wish to print at.
2. Click on the Variable Layer Height button on the top bar.



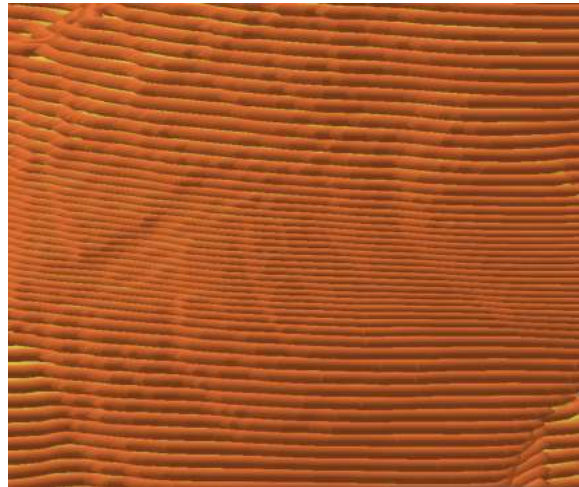
3. Using the slider on the right side, left click to increase the layer height, right click to decrease it.



4. You can adjust some settings to try and smooth the model out with the sliders in the corner.



5. Slice your file as you would normally. The layers will be modified as you intended.

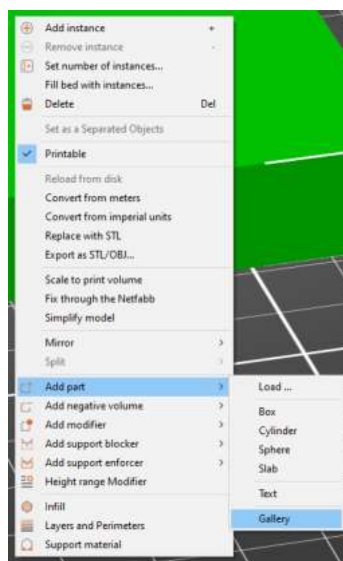


- a. Note: The layer adjustments are by layer not by model. If you have multiple pieces on the bed, it may be better to print them separately unless they all are benefitting from the same variable layer structure.

Helper Discs

Occasionally with wide flat objects with a lot of surface area, the edges of the print can raise up over time the longer the print lasts. A brim can help with these, but one solution has been to use helper discs at the corners of your object. These extend the area at the stress points to help give adhesion where it's needed and reduce extra clean up at the end of the project.

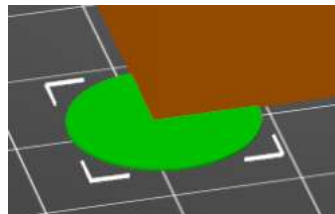
1. Right click on your model and scroll down to Add Part > Gallery.



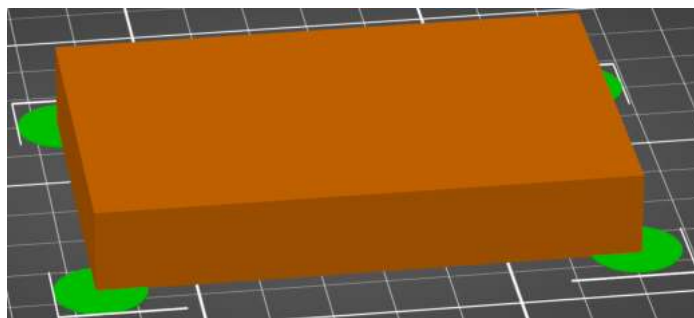
2. Select “helper disc” from the display and press OK.



3. A disc will appear on the bed beside your model. Move the disc to overlap one corner.



4. Add additional discs through the Gallery, copying, or instances so that each corner point touching the bed has a helper disc.



5. Once your project is completed, the single layer discs can be torn off by hand.

Staff Use

Filament maintenance

1. From the menu screen find Filament. Press the knob to select.



2. Select you want to load filament.



3. Scroll to the appropriate type of filament being used (PLA is most common). Press the knob to select.



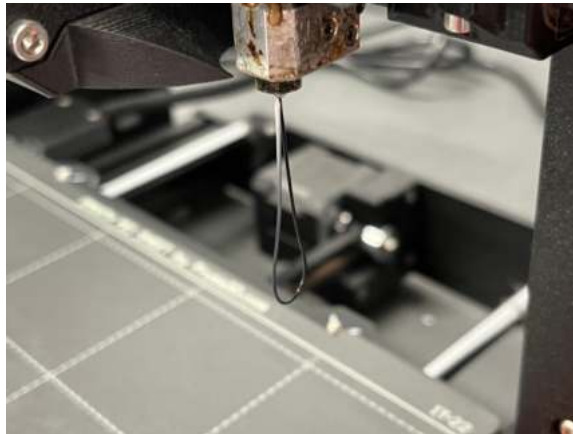
4. Cut the end of the filament at a sharp angle using the flush cut pliers. This angle will help guide the filament through the extruder more smoothly.



5. Insert the filament into the hole on the top of the extruder and press the knob.



- a. Sometimes you can insert the filament directly into the extruder and it will begin the loading process without going through the menu.
6. The printer will heat up and once it does so will begin extruding filament.



7. Check the color. If it is not the desired color, select “Purge more” and it will continue to purge filament. Repeat this step until the color is as intended and then select Yes.



To unload the filament, you'll follow a similar approach.

1. From the Filament menu select Unload Filament.



2. The printer will heat up to the required temperature. Once it reaches this temperature, it will purge a bit of filament, then remove the filament from the extruder.

When the printer runs out of filament mid print, follow these steps to replace the spool.

1. The machine will stop printing and begin beeping. Press the knob to eject any remaining filament from the extruder.
2. Take a new roll of filament, cut the end at a sharp angle, and insert it into the hole of the extruder. Press the knob to confirm it has been loaded.
3. If using the same filament as before just confirm same color and it will resume printing. If using a different filament, you may need to press no a few times to ensure the old color has been purged out of the hot end.

Cleaning the print bed

The main piece of maintenance to perform on the 3D printers is cleaning the print bed. Often the removable steel sheet gets covered in finger prints with use. When they build up to be noticeable, remove the sheet from the printer and use a bit of 90%+ isopropyl alcohol on a paper towel to clean it. Then return the steel sheet back to the bed.



Cleaning the nozzle

Occasionally, the nozzle will get coated in filament, typically due to a failed print job, but sometimes it occurs just through normal usage. To clean this off, follow the steps below.

1. Go to Settings > Move axis > Move Z > and then scroll the knob to the right to raise the height. Raise the Z axis so the nozzle sits a few inches above the plate.
2. Return back to the settings menu and navigate to Temperature. Change the nozzle temperature to 250 by rotating the knob to the right.
3. Once the nozzle has reached temperature, use a brass bristle brush and gently scrub away the filament off the nozzle. Be careful near any cables not to be too aggressive.
4. Press the reset button to end the heating process.