# 3D PRINTERS: IMPORTANT

The 3D printer is **hot**

Touching the bed, nozzle, extruded plastic, or print can cause burns, even after the print job finishes.

The 3D printer **may not stop** for objects in its way

While there is a safety feature that should stop it when it meets resistance, the 3D printer is a piece of industrial equipment with unshielded moving parts powered by strong motors, and these motors may not always stop for people or objects in the way. Jamming or impeding any axis of the printer can damage or destroy the motors and controller. Make sure the printer has enough clearance on all sides and cables and other objects are moved out of the way, and keep your hands and hair clear while the printer is working.

Never print without the spring steel sheet

You shouldn't need to use removal tools with these 3D printers, because there is a springy steel sheet magnetized to the build plate. **NEVER PRINT WITHOUT THE SPRING STEEL SHEET MAGNETIZED IN PLACE.** It's in the right spot when pushed all the way back so the two silver bolts fit in the inside corners:
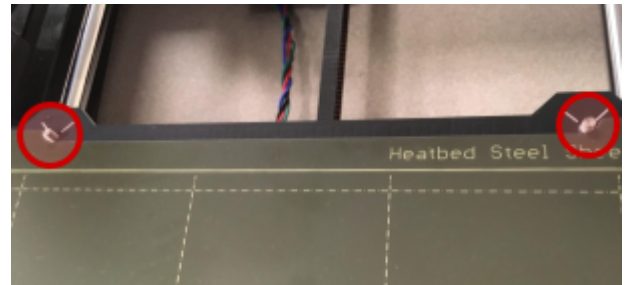
This can peel up to help remove prints. If it is absolutely necessary to use removal tools, a cautious use of force is recommended. Improper use of removal tools can also damage or destroy the print bed surface. **Only use removal tools as a last resort.**

Use proper material settings

Improper material settings will cause your print to fail and can permanently damage the printer's hot end.

Quality is not guaranteed

The strength, fit, food safety, etc of your printed projects cannot be guaranteed by the library.

Monitor your print

Be prepared to stop a failed print job. Plan to check on your print job frequently. Know how to pause, resume, and stop print jobs.

When in doubt, turn the printer off

Shut the printer off with the power switch (back of the control box on the left size)

3D Printer Training

Your printer training will be limited to the basics of both of the library's 3D printers. After you are trained, we encourage you to seek out tutorials that interest you to further develop your skills.

Step 1: Find a 3D model file

A model file for a 3D printer is just like an image for a regular printer. Our printers can print .STL and .OBJ files and they have a build area of 250mm x 210mm x 210mm (**9.84 in 8.3 in x 8.3 in**)

- Search thingiverse.com for free models (try looking for your favorite animal)
- If you've made your own 3D model, export an STL or OBJ from your 3D modeling program
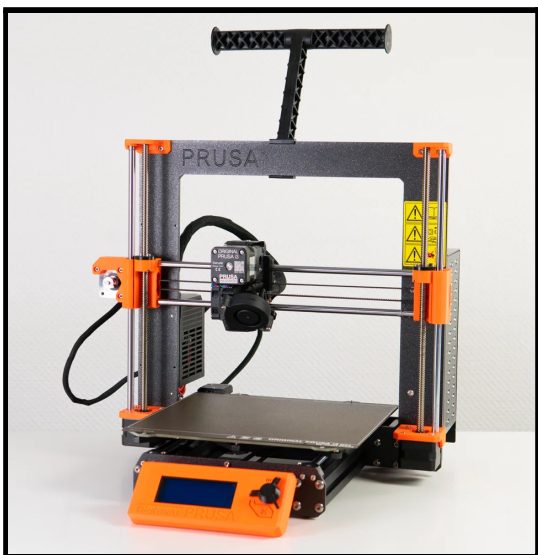
Step 2. Choose your filament

Filament is the string of plastic that the printer uses to make objects. Our printers are "FDM" models (fused deposition modeling) and **can only print with plastic filament**, although the plastic can have quite a bit of other material, like glitter, wood shavings, or metal powder, mixed in.
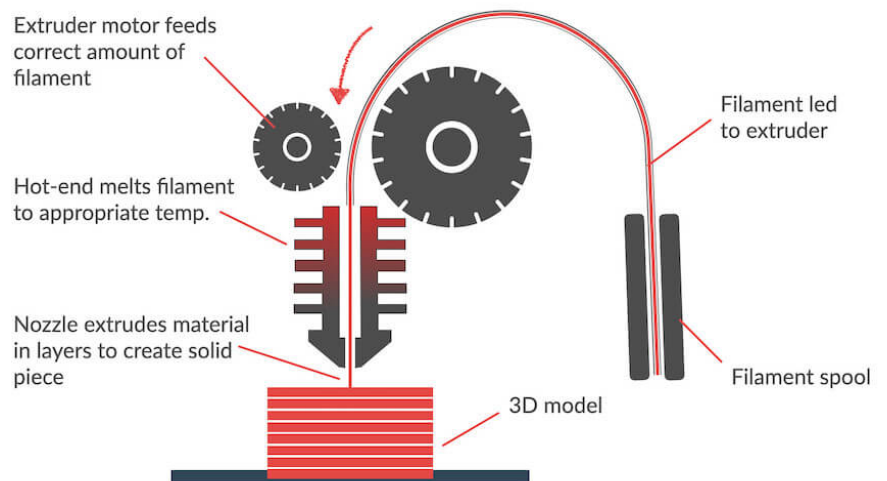
In an FDM printer, filament is pulled into the toolhead, melted, and extruded through the hot end, melting it and squeezing it out in thin layers, each sticking to the layer below and building the object from the bottom up.

There are several different plastics that make a good 3D printing material, each with its own pros and cons. Depending on what you intend to use your print for, it's good to know a bit about your filament options, but:

If you're not sure, start with PLA

FDM – ADDITIVE MANUFACTURING

Extruder motor feeds correct amount of filament

Hot-end melts filament to appropriate temp.

Nozzle extrudes material in layers to create solid piece

Filament led to extruder

Filament spool

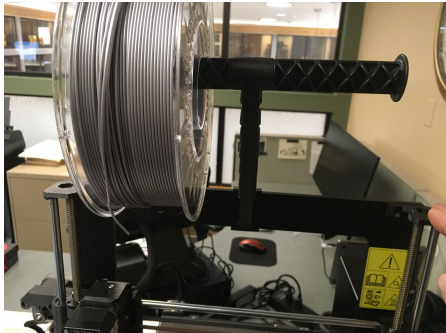3D model

Comparing Filaments

Filament type	Pros	Cons
PLA (polylactic acid)	- Low warp, easy and fast to print with - More eco-friendly (made with cornstarch) - Low-odor/pleasant smell - Huge variety of colors and other options (glow, glitter, etc) - Tends to be least expensive	- Not heat-resistant (don't leave a PLA print in a hot car or print a PLA coffee cup) - Brittle, thin pieces break quite easily - Carbonizes at a low temperature, which can create burnt bits that clog the printer nozzle
Co-polyesters (**nGen, PETG**)	- Combines some of the good qualities of PLA (easy to print with, no smell) and ABS (makes a strong, heat-resistant functional part) - Attractive, glossy surface finish - Very good bridging ability (can cross large gaps without sagging)	- Requires more heat and a slower print speed than PLA, making prints take longer - Can't be polished or sanded (it's too tough)
ABS (acrylonitrile butadiene styrene)	- Makes a tough finished part - Resistant to heat and sunlight - Can be polished with acetone	- Emits a strong chemical smell when printing - Prone to warping and splitting, poor bridging ability
Other common types	Several other filament formulas are available. Nylon makes incredibly durable parts for engineering applications, but can be finicky to print with and must be carefully stored. Polycarbonate is also very durable but can be quite challenging to print with.	
Specialty filaments	Specialty filaments can be common filaments (PLA, ABS) with a high percentage of additives like glitter, wood powder, metal powder, or ceramic. There are flexible filaments from firm to very soft (**TPU** is a flexible filament that prints well on our printers). There are also unusual or experimental formulas for unique applications, including filaments for printing dissolving supports or filaments that develop a foam texture after being washed.	

There are many other options out there besides these- check manufacturer & distributor websites for information about filament performance. The last page of this training has information about printer specs you can use when selecting filament.

Step 3. Loading the filament

Since we allow our patrons to print with the filament of their choice, we must purge the nozzle with cleaning filament to ensure that incompatible filaments don't mix, which can clog the nozzle. Loading and unloading filament is easy, just be sure to do so with cleaning filament before and after every job!

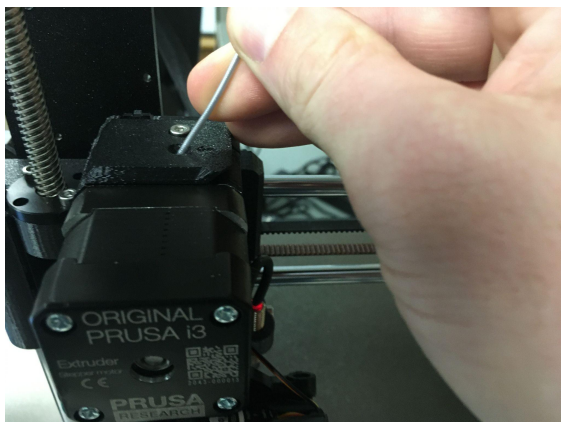|	
Loading the filament (before you print)	
	### Hang the filament up, trim the end, and turn the printer on Hang your filament spool or length on the spool hanger at the top of the printer frame and bring the filament to the toolhead so you know it will reach. Use the clippers to trim the end of filament to a point, the pointier the better. **Turn the printer on with the power switch located in the back right near the power cord.**
	### Autoload the Filament Using the GLCD (graphical LCD) controller on the front of the printer, press the wheel to open the menu. Scroll down until you see the Autoload filament option is highlighted, then click the wheel again. It will tell you that "Autoload is active, press button and insert filament."
	### Insert the Filament in the Toolhead Insert the cut end of the filament into the hole on top of the toolhead. You don't have to press hard. When you've inserted your filament, the screen will change to allow you to select what type of filament you've inserted. **ONLY USE 1.75 MM DIAMETER FILAMENT**, other sizes may damage the toolhead and put the 3D printer out of commission. **You shouldn't need to use force to load filament.**
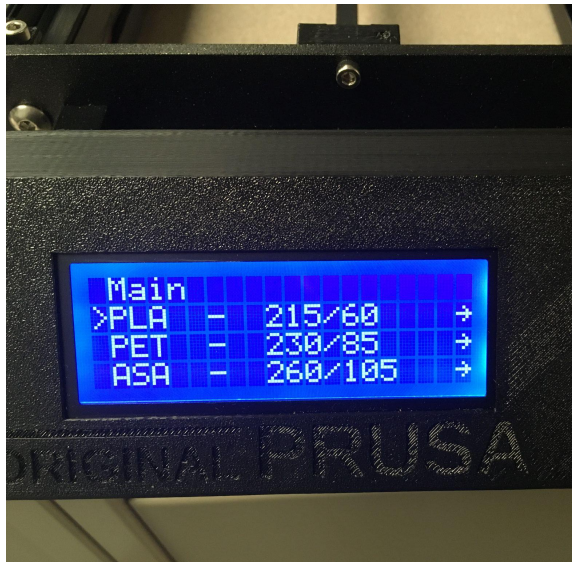
Choose Filament Type

Select the appropriate type of filament. The printer will heat up to the proper temperature based on this selection and then it will automatically extrude some filament, or pull the filament through the heating element and out of the nozzle. It's normal for melted plastic to come out of the bottom, simply remove it with tweezers and remember that the nozzle is hot.

The printer will ask if the extruded filament is the right color (it wouldn't be if the last user had left small amounts of their filament without running a cleaner through the toolhead). If it is, press the button to say, "Yes".

Now you're ready to print!

Step 4: Slicing the file

A **slicer** is specialized software that slices your model file into thin layers and turns each layer into a list of instructions called G-Code. G-Code tells the printer what path to follow, when to lift up, when to extrude, how hot to get, et cetera!

We use a free slicer that is customized for our printers called PrusaSlicer, or Prusa for short. You can download Prusa and explore it on your own.

Having your own copy of Prusa is a great way to estimate how long your jobs will take, and be as prepared as possible when you reserve time on the 3D printers.
https://tinyurl.com/yhefy2pc

Open the model file in Prusa

- Open **PrusaSlicer** from Desktop
- Click File, then Import, then Import STL, and select the .stl file you would like to print
- You can also click and drag an .stl file into the workspace in the Prusa window
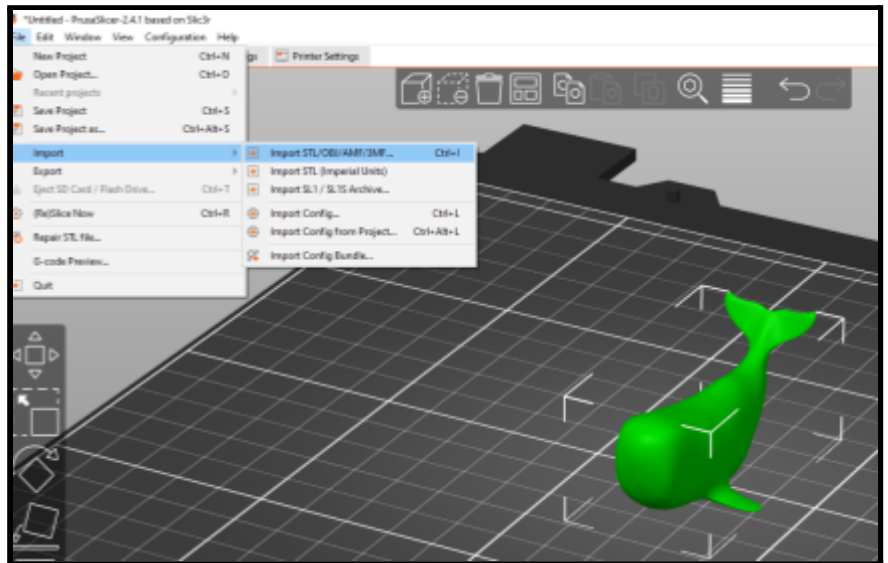
## Viewing and modifying models in the Prusa workspace

There are only a few options for modifying STL/OBJ files in a slicer- to make substantial modifications (for example, making a hole in an object bigger while leaving the rest of the object untouched) you would need the original 3D file and the software it was made in. But you can make a few simple modifications to help fit your models onto the printer bed.

View your model from different angles using the mouse

- Scroll with the scroll wheel: **Zoom**
- Click and drag with the left mouse button: **Orbit**
- Click and drag with the right mouse button: **Pan**

Move, scale, rotate, place on face (lay flat), and cut models

- Using the buttons on the left-hand menu, you can **move**, **scale** (resize), **rotate**, **lay flat**, and **cut** your model. Experiment with each and **Undo** with the button at the top (or CTRL + Z)

- Drag the blue, red, and green handles of the Move/Scale/Rotate tools, or type the values in the boxes at right under Object Manipulation, and press Enter.
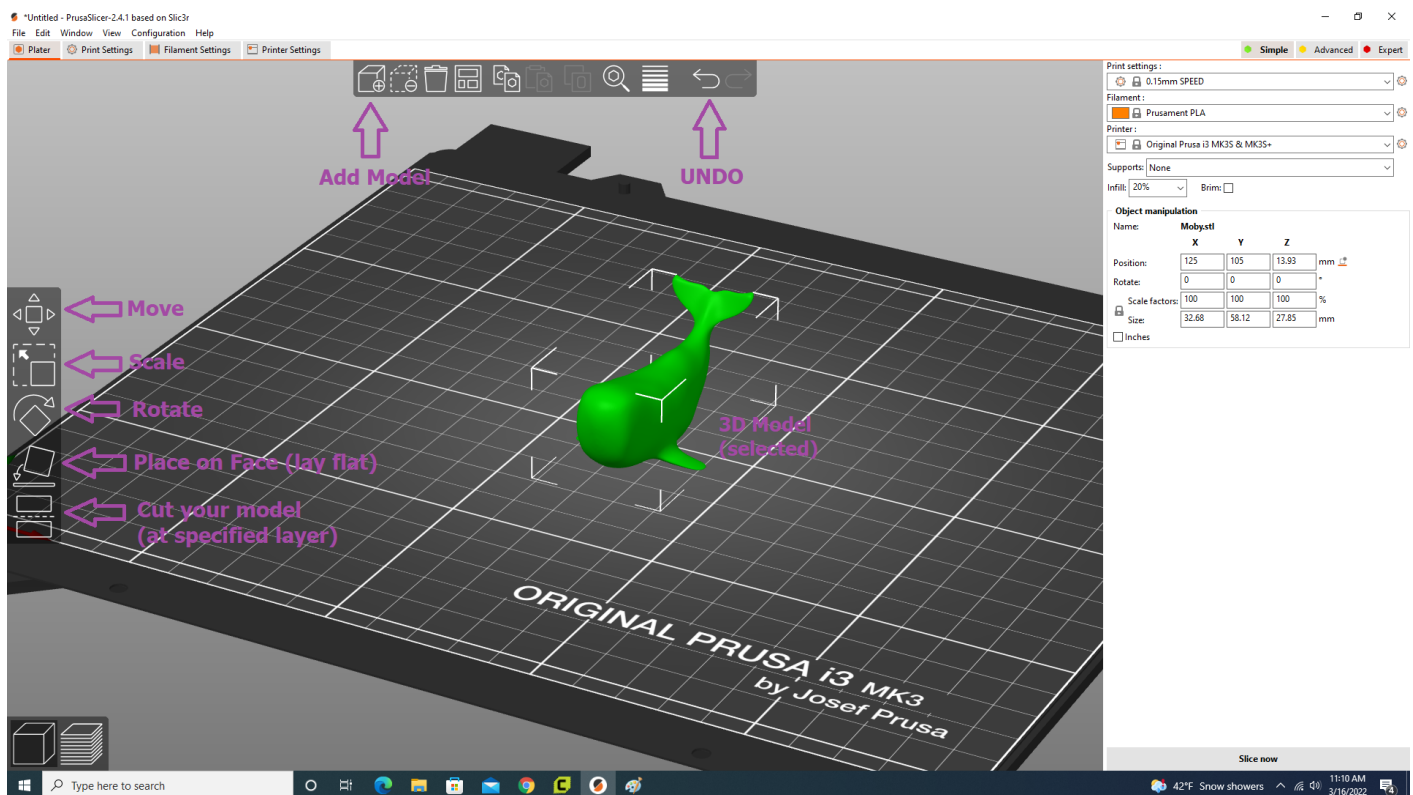
Add and delete models

- Multiple different models can be placed on the print bed with Add Model.
- Select a model and press Delete on the keyboard or right-click > Delete

Making models fit

- The print bed is shown in the Prusa Workspace
- Models always touch the print bed, but you may need to rotate the model to get the bottom side/flat side facing down
- If a model turns blue, it doesn't fit in the build area. Adjust the size and/or position of the model so it fits, note that it may be too tall and Prusa only shows the maximum height of a model if it is too tall.

Print settings

Your print settings are on the right side of the Prusa window. It is important to use the proper material settings for the filament you are using or your print is likely to fail.

Print Settings: Speed Dropdown

This chooses the resolution of your print, determined by the size of each layer, and is also effective for making prints take less time. Some options have a "SPEED" or "QUALITY" choice for this purpose also.

Generally, choose smaller layer sizes (0.05 mm) for higher detailed prints that will take longer, and larger layer sizes for less detailed prints that will finish faster (0.3 mm).
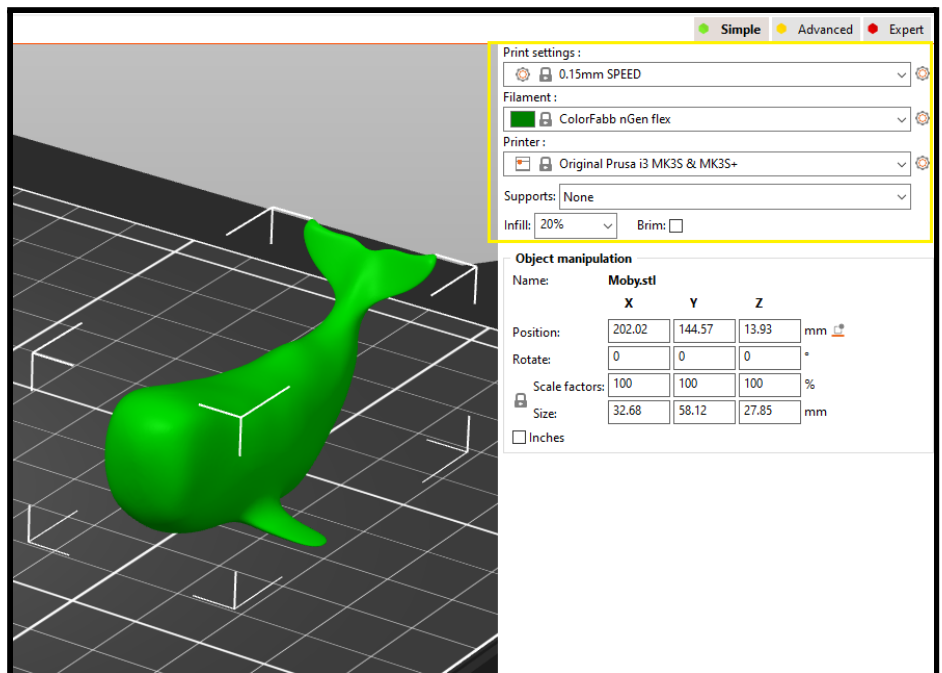
Filament Dropdown:

Each type of plastic has different requirements for temperature and extrusion speed. Choosing the wrong one can result in a failed print or, worse, a permanently clogged nozzle. Prusa comes preloaded with material profiles that set the nozzle temperature, bed temperature, extrusion speed, and several other options.

- Check the spool your plastic is on (or the bag it's in) for a label with the plastic type (PLA, ABS, PETG, etc).
- Pick a material preset from the dropdown. You can use any preset that matches your plastic type if you don't see your plastic's brand name.

Printer Dropdown:

This should always default to "Original Prusa i3 MK3S & MK3S+" and doesn't need to be changed.

Supports Dropdown:

Since the print is made from the bottom up, layer by layer, areas of the model that stick out into space with nothing underneath can't be printed accurately- the toolhead will extrude in midair and make spaghetti strands.

Take a look at your model in the build area. Are there parts that hang over mid-air above the build plate? You may need to use supports.

The easiest option to start with for supports is "Everywhere". The printer will automatically add a grid of removable supports under these areas. Printing supports will use more material and make your print take longer.

With experience, you'll be able to guess at the printer's ability to print overhangs depending on the plastic type and how severe the overhang is, and you can also try using Layer View with the Support Blocker to control exactly where supports are placed.

## The action figure guide to supports and bed adhesion

The printer will start printing the end of the gun and his fingers starting in the air- there's no way to print this successfully without supports

without supports

If he stands like this, he can be printed with *or* without supports- there are gradual overhangs and a bridge, but the printer can handle them

With supports & a brim (the brim helps hold the supports together, too)

Adding a brim will help keep the legs from breaking away from the bed during printing, since the bottoms of his feet are small compared to his height

OK

Infill Dropdown and Brim

The 3D printer will lay down a solid base and solid perimeters for your object, and you can choose how dense the plastic on the inside will be with the Infill percentage. Light infill prints much faster, while an object with 80% infill will feel heftier but be more solid. When in doubt, leave the infill at 20%.

If the "Brim" box is selected, Prusa will have the 3D Printer lay down extra filament on the build plate around your object's first layer, which will later be broken off. You should use this option if your print is "tall and skinny," with a relatively small base on the build plate. Brim will help these models stick in place while they are being printed.

Slicing your Model:

When your model is set up to your satisfaction, it's time to slice it. Remember, slicing is the process the software does to create G code which can be read by the 3D printer. To slice, press the "Slice Now" button in the lower right corner. Prusa will switch to "Preview" mode so you can see how it intends to build your model layer by layer on the 3D Printer.
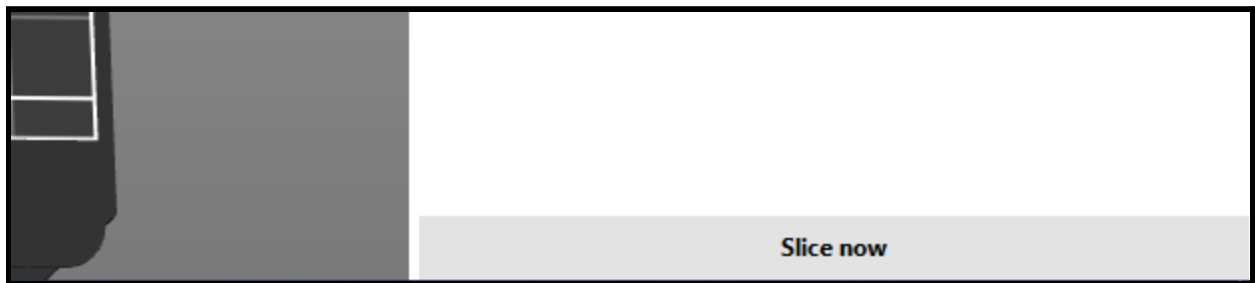

Estimates of the amount of material and time required will appear above the "Slice Now" button. If you need to switch back from "Preview" to further adjust your settings, click the cube in the lower left corner to return to "3D Editor View".
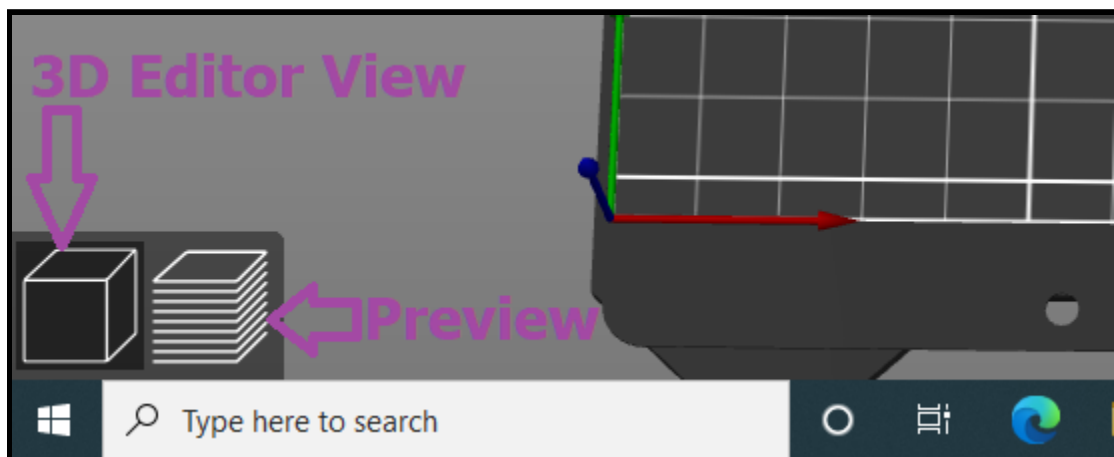
Save the G Code:

When you've finalized your print settings and have any necessary supports set up, you will save the G-Code to an SD card.

- Insert an SD card into the computer

- In the bottom right corner, click "Export G Code".

- Choose the SD card from the file explorer. Make the file name unique to you so you can find it later.
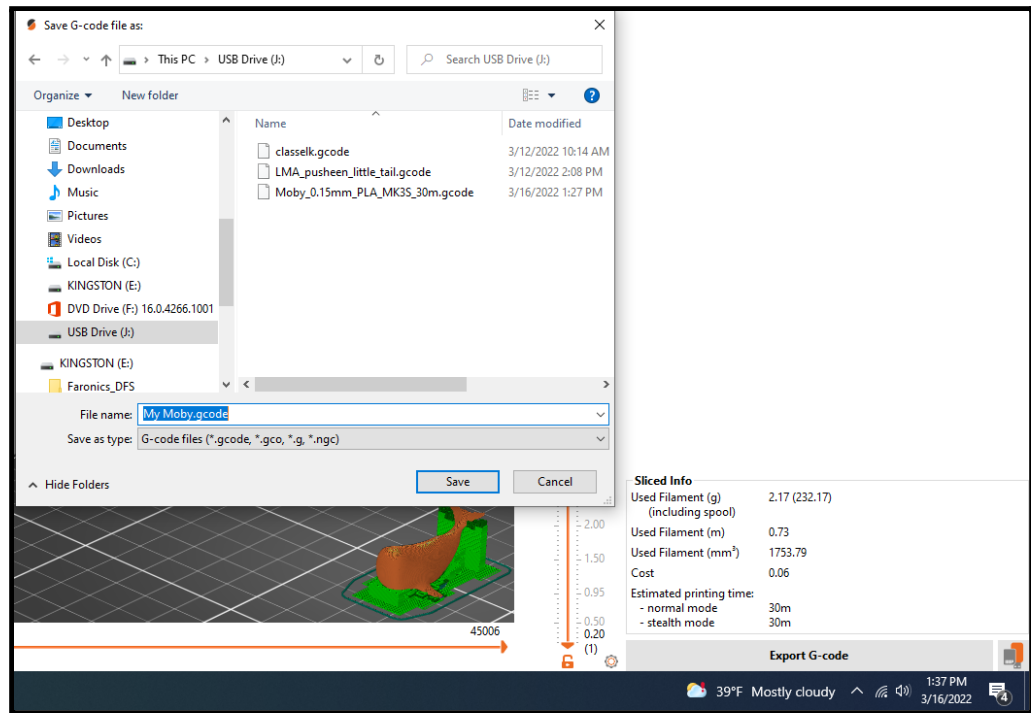

- Once you've double-checked your settings, click the Save button, then click Eject in the File Saved dialog box.
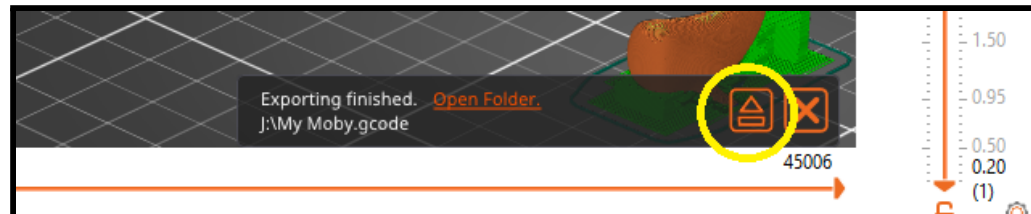
Why SD cards?

These printers can be connected directly to a computer and print over USB, but the USB connection has to stay up during the entire print job, even if the job is 6 hours long! Saving the gcode file to an SD card and walking it over to the printer ensures a stable connection that's hard for people to accidentally interrupt.

Step 5: Start the job
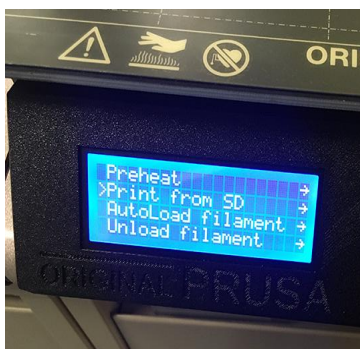
- Remove the SD card from the back of the computer and insert it, label facing away, into the printer.

- Open the GLCD menu and scroll to Print from SD

- Turn the wheel right to scroll through the files on the SD card. Click to select the job.

- That's it! Everything else is automatic

- Even more automatically, inserting the card should immediately bring you to the menu where you choose your .gcode file, and you're off!

During printing

Keep an eye on the printer as it goes through its warmup sequence and the print job begins. Print jobs are most likely to fail on the first layer or two.

Printer warmup sequence

Start heating: the printer will go to its home position and begin heating the nozzle to a medium temperature to prepare for cleaning. If the nozzle is set to a high temperature after a filament change, the nozzle will cool down to the cleaning temperature.

Nozzle cleaning: The printer will extrude some filament onto the build plate to clean off any excess plastic and get things flowing, then travel to specific areas of the bed to check the bed alignment with a probe. If the bed is not perfectly level, it will adjust the print job automatically to compensate.

Finish heating: The toolhead will move and start heating the bed and nozzle to the correct temperature. This can take awhile if the printer started cold. You can see the current temperature of the bed and nozzle on the GLCD Info Screen.

Once the printer has preheated, leveled the bed, and cleaned the nozzle, it will start printing.

Be ready to stop the job

Watch your print carefully at this early stage! If the plastic starts to snag on the nozzle, or the first layer looks like it isn't sticking right, it's best to pause or stop the job:

- Click the scroll wheel to enter the menu
- If you want to check the print without ending it, choose **Pause Print**. If you like the way it looks, choose **Resume Print** from the GLCD menu.
- If the print looks like it's failing, scroll down to **Stop Job** and click to select.
- Remove the failed print (including the skirt) by peeling up the bed on the build plate or in extreme circumstances with the putty knife or metal spatula.

When in doubt, cut the power

If things are going REALLY badly, especially if the printer is making a loud noise, seems jammed, or has caught on an object or person, turn the printer off by flipping the main power switch.

This will de-energize the motors instantly so you can move the X and Y axes by hand if needed.

Note: The bed and nozzle will remain hot for several minutes even if the power is off.

Step 6: Removing the print

Stop the job (or wait for it to stop)

When the print finishes, the bed will move to the back of the printer and remain there until the bed has cooled. When the bed moves forward again, you may remove the print from the bed.
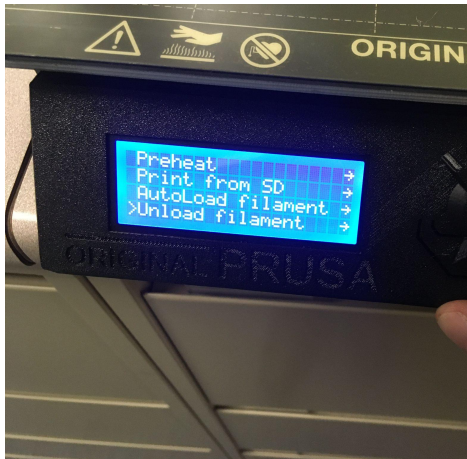
You can stop the print and move the bed forward before the bed cools- but be cautious as the bed and nozzle will still be hot.

Select **Stop Job** from the first page of the GLCD menu

Step 7. Unloading the filament and cleaning the nozzle

Since we allow our patrons to print with the filament of their choice, we ask that you remove your filament when done and purge the nozzle with cleaning filament to ensure that incompatible filaments don't mix, which can clog the nozzle. Not only is it easy, it's the responsible thing to do with a 3D printer that is shared among the community.

Unloading the filament and cleaning the nozzle (after you print)	
	Select "Unload Filament" to Start Auto Unload Using the GLCD (graphical LCD) controller on the front of the printer, click the wheel to open the menu. Scroll down and select **Unload Filament.** The printer will automatically start the process for unloading your filament, and you may need to tell it what type of filament is loaded as in the "Autoload Filament" section.
	Remove your filament When the nozzle is fully heated, pull your filament out from the top of the toolhead. **You shouldn't need to use force.**
	Insert the cleaning filament Cleaning filament is a flexible white filament that stays melted across a wide temperature range, so it's bad for printing parts but great for flushing old filament out! It is stored in the tool drawer near the printer. Feel free to use as much as you like- cleaning filament is good for the nozzles. Use the clippers to trim the cleaning filament to a point- the pointier, the better. Follow the same instructions for loading filament from earlier, starting with choosing **AutoLoad Filament** from the GLCD.
	On the GLCD, choose **Unload Filament**. Remove the cleaning filament and turn the printer off with the main power switch in the back.

Other Notes:

Finding & making 3D models

- Browse thingiverse.com and download something interesting!
- **Tinkercad** is a great place to start if you've never done any 3D modeling. If you are ready to try creating your first 3D model, sign up for Tinkercad at tinkercad.com, then go to Learn > Starters. After completing a lesson, you can export and download a model file using the Export button.
- **Autodesk Fusion 360** is a professional CAD program that is (relatively) easy to start with and great for 3D printing. A version with a few missing features is available for free personal use. autodesk.com/products/fusion-360/
- **Blender** is a very popular free modeling software that's especially good for artistic modeling blender.org

Buying filament

Filament is relatively inexpensive ($20-$40 for a 1kg roll) and widely available. We buy our filament from matterhackers.com. If you are buying your own, here are the relevant specs to help you determine if filament is compatible and select the right diameter. *Note: Printing with carbon fiber filament is not recommended by the manufacturer and **only 1.75 mm diameter filament can be used***.

Printer specs **Original Prusa i3MK3S+**

- **Build Area:** 250mm x 210mm x 210mm (9.84 in 8.3 in x 8.3 in)
- **Filament diameter:** 1.75 mm
- File types: STL, OBJ, AMF, 3MF
- **Feeder system:** Direct drive
- **Nozzle size:** 0.4 mm
- **Max. hot end temperature:** 300 °C
- **Max. heated bed temperature:** 120 ℃
- **Print bed material:** Removable magnetic PEI spring steel sheet
- **Bed leveling:** Automatic
- **Technology:** Fused deposition modeling (FDM)
- **Year:** 2020
- **Mechanical arrangement:** Cartesian XZ-head
- **Manufacturer:** Prusa Research
- **Connectivity:** SD, USB-Type B
- **Print recovery:** Yes
- **Filament sensor:** Yes
- **Filament materials:** Up to nylon, polycarbonate
- **Recommended slicers:** PrusaSlicer (available for Windows, Mac, Linux)