Beginner's Guide to Prusa MK4

Printer-Based Menu Controls

This is the home screen of 3D printer itself:

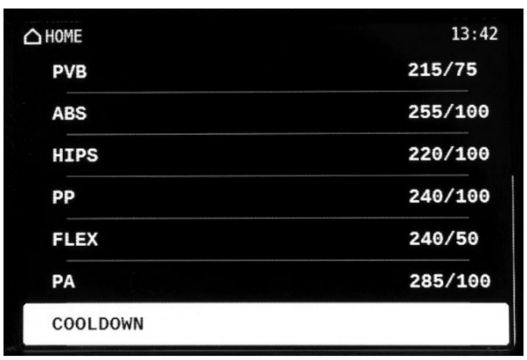
The upper right corner shows time, in the middle, there are 6 menu options, and at the bottom of the screen, you can see nozzle temperature, heatbed temperature, and type of material loaded into the printer. During a preheat, the temperatures will be in the format "current temperature/target temperature".

Most of the menu options are only useful for troubleshooting or testing purposes. As a beginner you should only ever need the **Print** and **Filament** menus.

No USB/Print

This menu item reads No USB when the USB slot is empty, and changes to **Print** when a USB stick is inserted. Choose the file you wish to print.

Preheat

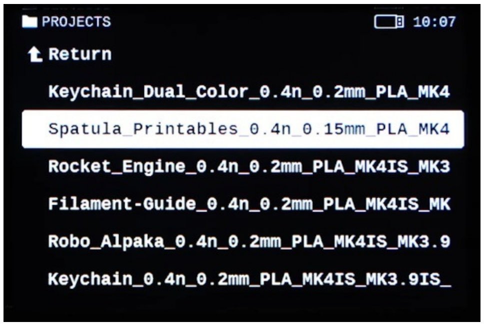


Through this menu, you can preheat the nozzle and the heatbed to a certain temperature, as per the chosen filament material. Under normal circumstances, **you don't have to enter this menu at all** - when printing, the correct preheat settings are always already included in the print file and, when loading a new filament, you choose the correct temperature through the **Load Filament** option.

Filament

Here, you can **Load** or **Unload Filament**, **Change Filament** (which is just the two previous options one after another) or **Purge Filament** (i.e. get rid of the remnants of the previous filament color - normally, this is already included in the **Load Filament** procedure, though).

Choosing a 3D model from the USB drive



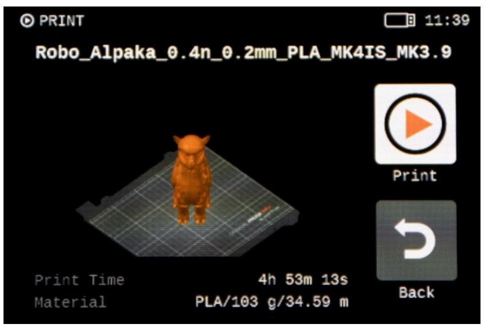
Insert the USB disc into the slot on the right side of the display box.

The **No USB menu** option will then change to **Print** and immediately after, the file folder on the USB drive will automatically open.

Note how the name of each file (called **G-code**) includes what printer is it made for ("MK4"), which material ("PLA"), layer height (0.2 mm), and the time it takes to print (from tens of minutes to several hours).

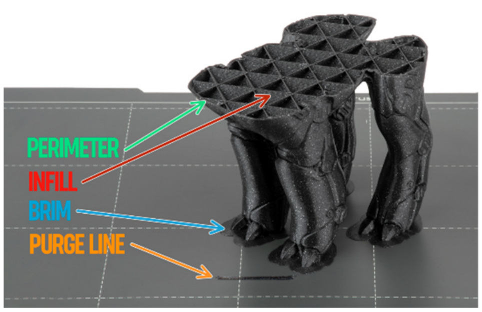
Starting the print process

Use the controlling knob to open the **Print** menu and select your model.



You will see a little preview of the model, and the estimated print time. Once you confirm by pressing the **Print** icon, both the print sheet and the nozzle will start heating up. No user input is needed from that point on – the printer will run through cleaning the nozzle, measuring the distance to the print bed, and preparing to print. During these steps, the heated nozzle may leak a bit of filament. It is normal and nothing to be concerned about.

During the 3D print process



The first thing to materialize is the **purge line**. The model itself starts with a few flat horizontal layers at the bottom. The pads around the feet are called a **brim**. Its body consists of an outer shell, with thickness determined by the number of layers, called **perimeters** (i.e how many times the nozzle runs around the model). The usual number is between two and four. Once the print builds up a bit, you will see that the inside of the model is not solid, but filled by an internal structure called the **infill**. The infill saves both material and printing time while still providing enough structural support.

After the print, let it cool down for a while. The print will come off a cold sheet much more easily. You can check the print sheet temperature on the display. Manipulating the sheet while it's still hot may result in slight warping of the bottom of the fresh print and, in some rare cases, damage to the sheet surface. Once the sheet is cooled down, gently lift it by the two closer corners.

To remove the print, just bend the sheet a little bit, it will pop off easily.

Printables.com: Introduction

If you aren't ready to try creating your own models yet, there are many online databases full of 3D objects which can be downloaded for free! Prusa has its own website and community for finding and sharing projects, many of which can be downloaded for free at Printables.com.

Printables.com is a pretty large database, meant for all 3D printer owners, both Original Prusa and other brands. It's reliable, easy to navigate, and with a lot of added features.

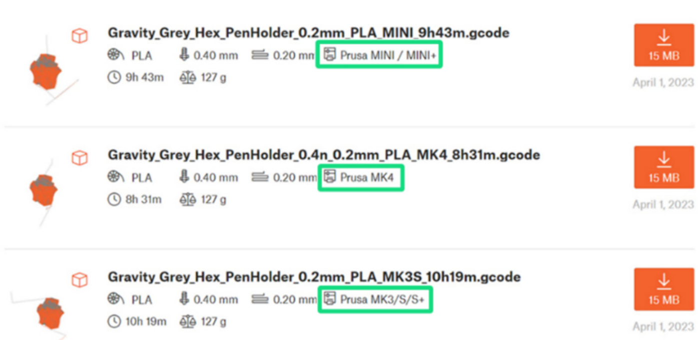
As a beginner, we recommend that you go straight to the main content of the website: the **3D Models database** itself. The basic navigation is pretty intuitive - feel free to click back and forth between the topic categories for a while, to get an idea of what the database offers: Art & Design, Household, Learning, Toys & Games - there is something for everyone.

You can also search by clicking on the magnifying glass icon in the top right corner.

> **Note:** Keep in mind that you can't just take a 3D model in any of the regular formats (.stl, .obj) and load it directly into your 3D printer. The file needs to be converted to a special instruction file called a G-code. This conversion process is called slicing, and it is done in software called a Prusa Slicer which is covered later in this document.

If you don't want to learn the slicing process yet, there are ready-made G-code files available as well - i.e. files that are already prepared for printing. Just download them on your USB drive and you are good to go.

Be careful though, each G-code is made for a specific printer type and won't work properly in any other. Also, each G-code file can only be printed from a certain printing material (there are specific temperature settings, etc.). That means, in your case, make sure you download only G-codes made for **MK4** printers and for **PLA** material. Filter the models using the buttons on the left. Choose only prints that include the "G-code Print Files", which are made for "Prusa MK4".



There might be more than one G-code included with each print, make sure to choose the right one - at the tiny spool icon there should be "PLA", and at the tiny printer icon "Prusa MK4". The name of the material and printer are also included in the G-code file name.

If you want to play it safe, start with smaller, simpler models (if something fails, at least it fails quicker and with less waste of material). To get a broader picture, browse through the lower section of the menu of each print. Check the instructions from the author, comments from other users, or, best of all, photos of their successful prints.

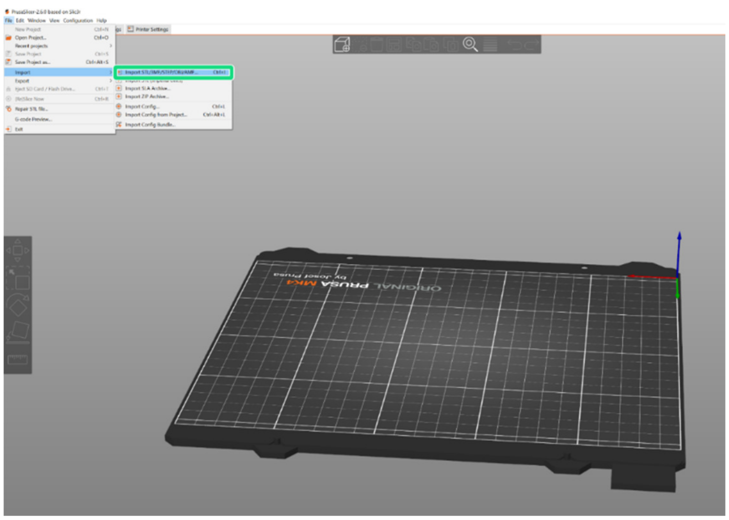
<u>Using Prusa Slicer</u>

Importing 3D data into the Slicer

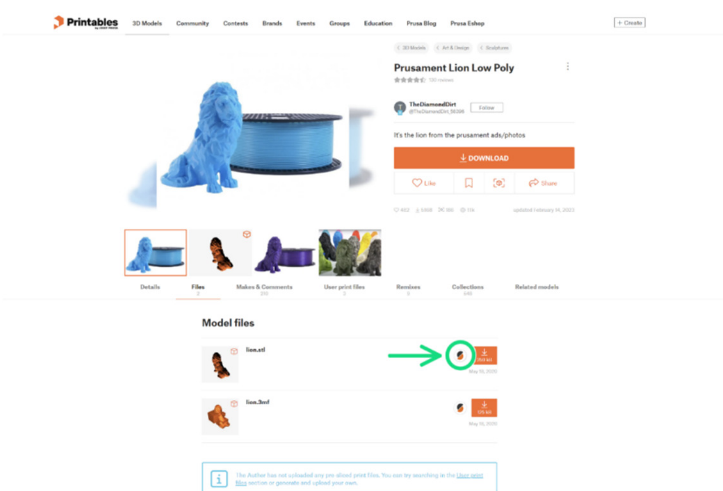
For the import, you have two main options - an **STL** file and a **3MF** file (don't worry about the AMF and OBJ options for now).

The 3MF format has a big advantage in storing the slicer settings, but since we want to demonstrate how to make those settings yourself, we will use the plain STL.

There are two ways of opening the model file in PrusaSlicer.



1) Download the file to your computer and then click on **File -> Import -> Import STL/OBJ/AMF/3MF** inside the Prusa Slicer software.



2) Use the PrusaSlicer button directly from the Printables website. If this option stays enabled during the Configuration Wizard, all you have to do now is click the button, and the model will automatically open in PrusaSlicer.

Note: If you can't see this button, you have to enable it in Printables.com by clicking your account icon in the upper right corner and selecting **Settings - > Display & language -> Show PrusaSlicer button.**

Once the object opens on the Slicer printbed, you can practice manipulating the view - **hold the left mouse button to rotate the print bed, zoom by scrolling mouse wheel, or hold the right button or mouse wheel to pan the view.**

Interface: Right side panel, Slice button

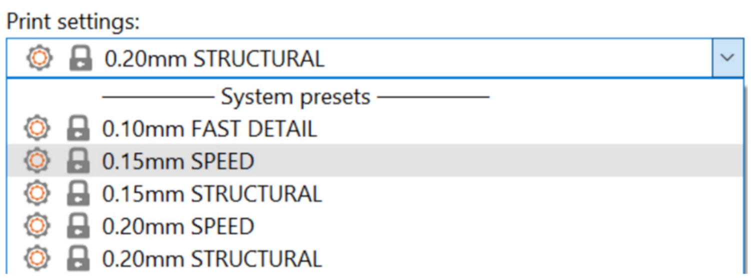
We start with the **panel on the right side of the screen**.

All the basic, must-have settings are placed in this part of the screen: selecting the layer height, print material, printer type, enabling supports, and choosing the infill density.

Down in the lower right corner, there is the actual **Slice** button.

Let's quickly go through all the other important settings first:

Print settings

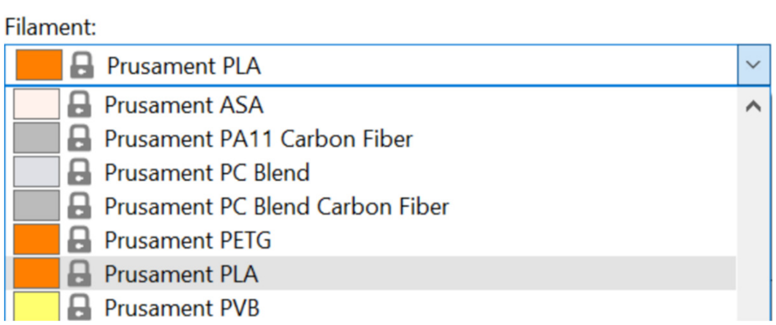


In this dropdown menu, you can choose from various print layer heights. Note that even though these presets are named after the layer height, each of them also includes many other different settings that influence the quality and print time.

The lower the layer height, the better the details will be - but the print process will take longer. The layer height has a big impact on the quality of the surface details. However, choosing small layer heights considerably increases the printing time. We recommend using a layer height between 0.1 and 0.2 mm.

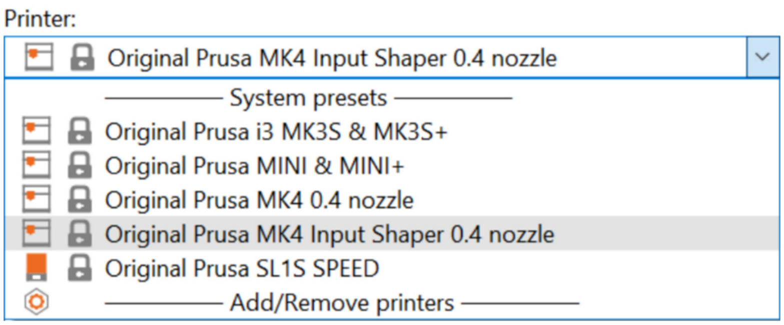
For most layer heights, there are two different profiles to choose from: **SPEED** prioritizes - you guessed it - print speed, while **STRUCTURAL** prioritizes the surface quality and overall toughness of the printed model. Neither of the profiles goes into an extreme, however - i.e. the SPEED profile still yields reasonably durable prints, and the STRUCTURAL one is by no means slow.

Filament



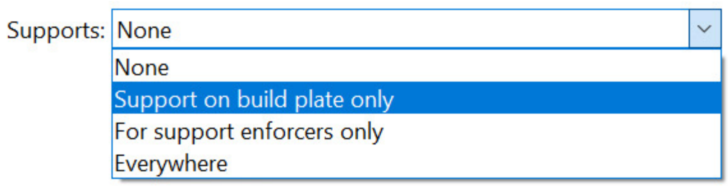
Choose your material. Different materials require different settings (temperature, print speed, etc.), so it’s important to choose the correct one. As long as you print from the spool you got with your printer, keep this setting on “Prusament PLA”.

Printer



Choose the correct printer: **Original Prusa MK4 Input Shaper** in this case.

Supports

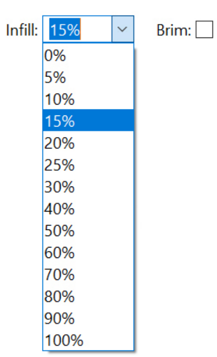


Choose whether slicer should generate **supports** - printed “scaffolding” which increases reliability when printing **overhangs** When the print is finished, you can just snap the supports off.

You may have already seen supports on some of the prints downloaded from Printables.com - now we will explain their purpose a bit further. Your 3D printer cannot reliably print into thin air. As the material builds up layer by layer, each layer has to have at least a bit of a solid base underneath. The parts of the model which are hanging or protruding “in the air” are called **overhangs**. Whether these overhangs are printable depends on their size and angle. 45° is OK (imagine arms of the letter “Y”), 90° (arms of the letter “T”) is not.

The printer can cope with the overhangs without the help of supports by slowing down and increasing cooling, then slowly building up the material in the air - this is called **bridging**. However, this only works to some extent and rarely looks perfect (there are saggy strings of filament, etc.). In the worst-case scenario, the print could also collapse altogether.

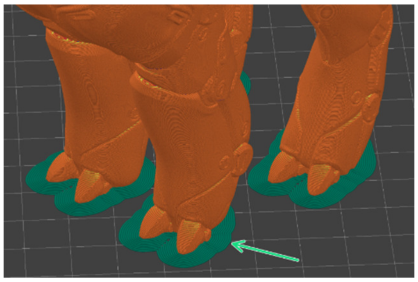
Infill



This option allows you to change the **infill** density (pattern can be changed elsewhere in a different menu). For now, you can leave the preset value (15 or 20%) as it is.

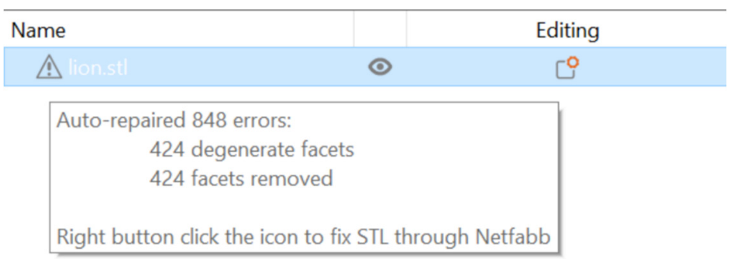
3D models are rarely printed fully solid on the inside, with 100% density. The print would take ages, consume loads of material and the resulting object would be heavy as a brick. Instead, most prints are hollow, with supporting mesh filling the cavity - the **infill**.

Brim



This option will generate a thin layer around the base of the model, which will improve the adhesion to the print bed. This is a handy option for models that don’t have enough of a grip in proportion to their size.

Repairing damaged models



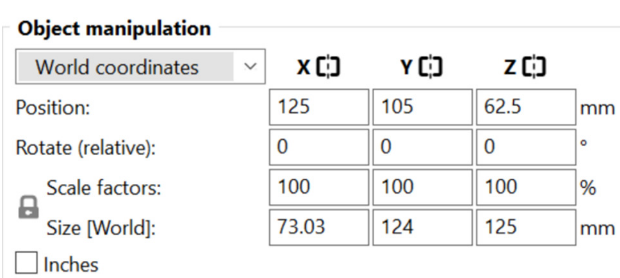
Below the dropdown menus, there is a large window with a list of models that are currently on the print sheet. Note the triangular warning sign next to the model’s name - this means there are some errors in the model file.

Typically, when you see this error the triangular mesh which forms the model surface is not fully closed, or perhaps there are some holes, stray vectors, and nodes. PrusaSlicer can repair such errors automatically.

In the Windows version, you can right-click the triangle to **Fix through Netfabb** and the warning will disappear. You can perform the same function by right-clicking the model itself.
Another option is in the upper left corner of the File menu - choose **Repair STL file**. PrusaSlicer will save the file again in an **OBJ** format, which is less prone to errors.

Note: Be careful, as in some cases, an intentional feature might be interpreted as an error, i.e. Netfabb would delete a hole that was actually supposed to be a part of the design.

Object manipulation



This menu basically duplicates the left icon bar functions **Move**, **Rotate,** and **Scale**.

Instead of manipulating the model with your mouse, **you can insert the exact numbers** in each of the X, Y, Z axes: position coordinates, rotation angles, and either percentage of size increase/decrease, or precise measures.

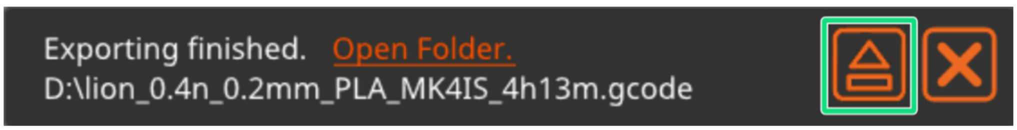
Note: Make sure you keep the ratios locked (using the padlock icon), so all the axes scale uniformly, otherwise the object will deform.

Slice now

Finally, you can now slice and export your first model. Click the "**Slice now**" button in the lower right corner and the program will **have slice the imported file and generate the G-code**.

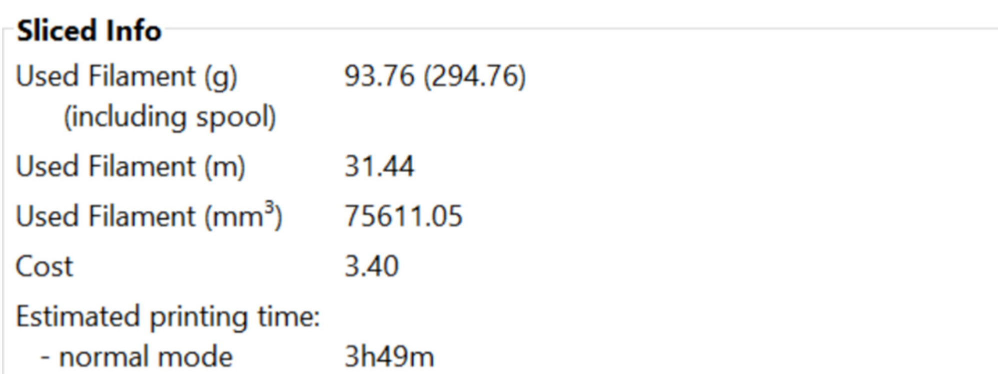
If you want to print right away, just click on **Export G-code**, download the file on your USB drive and you are ready to take it to the 3D printer and insert it for printing.

Note the icon on the right - if you have a USB drive inserted, you can click it to upload the G-code there directly.



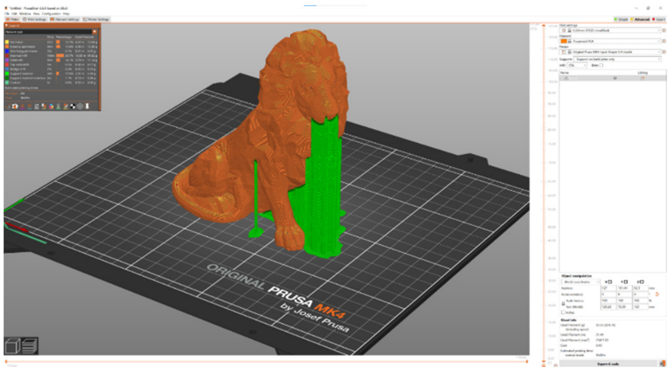
A small pop-up window will appear in the lower right corner - you can click this icon to safely eject the medium. It's a small but nifty feature. Basically, you don't have to click anywhere outside of the PrusaSlicer!

Note: Before you export your G-code, it's a good habit to **double-check everything first** - make sure you didn't forget anything (typically adding supports), the settings are correct (material and printer) and the print time matches your expectations.



In the lower right corner, there are now some stats about how many meters/grams of filament and how many hours this print will take.

In this example, at 0.20mm SPEED settings, a 100% sized lion will take less than 4 hours. If you don't want to wait that long, you can print a smaller lion faster. In the lower-left corner, switch back from **Preview** into the **3D editor view** - this will show you the same situation as it was before you hit **Slice now**.

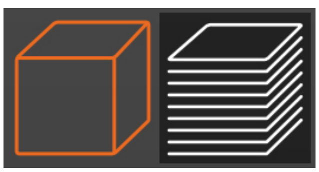


You can make the model smaller using the **Scale** (S) button on the left (or the **Object Manipulation** on the right). If you scale him down to 60% size and hit **Slice now** again, you will see that the print will now only take about an hour and 20 minutes!

Now, back to the **Preview** - we see the model exactly as it will be printed - including infill, supports, etc. When you zoom closer, you even see the individual lines of filament.

Everything is **color-coded** - green supports, orange external shell, scarlet infill. Don't worry about all the colors now though, this color-coding is just informative.

You can inspect the individual layers by **dragging the slider on the right side**. Note the checkered infill structure inside - it is called the **rectilinear** infill.



Using these icons in the lower-left corner of your screen, **you can toggle between the 3D editor view** (i.e. how did the scene look like before clicking **Slice Now**) and the sliced **G-code Preview**. You can also press Ctrl+5 or Ctrl+6.

When exporting G-code, always also save your project as a **3MF file**! This means you will have all your slicer settings saved in an editable form. Go to the top left corner of the screen and click **File** - **Save Project.**

Always double-check everything in the **Preview** before exporting G-code! You can prevent some errors, like forgetting to turn on the supports, choosing the wrong kind of filament, etc.