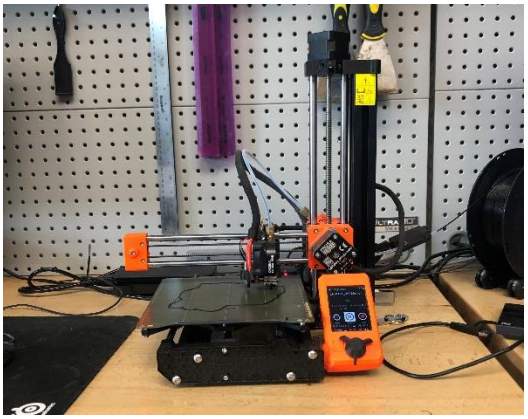
OT3d.org

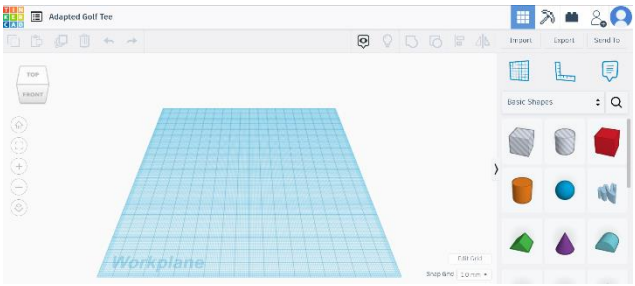
Start to Finish Guide

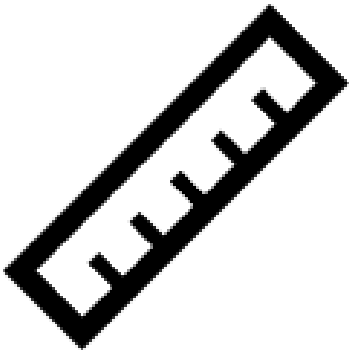
This document is designed to serve as a reference to walk you through the basic steps of the 3d printing process. Other useful guides and videos can be found at OT3d.org



This guide will focus on the process of 3d printing using an FDM printer, from basic file creation to print cleanup.

To start, we will need an object to print. For the purposes of building this object, I will use a free program called TinkerCad, which allows me to build an item by joining basic shapes together. A full walkthrough of how to use TinkerCad can be found using this QR code which links to a separate video and visual guide.



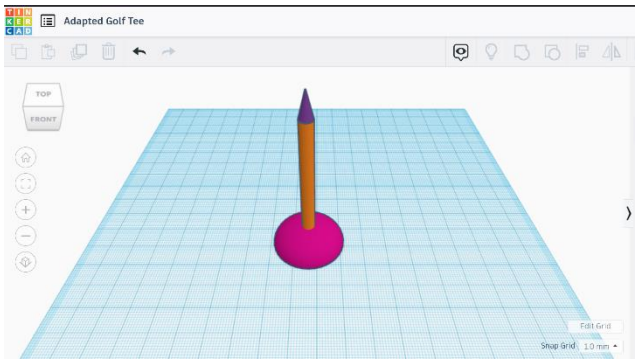


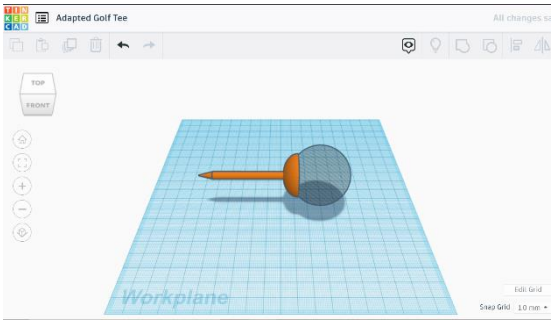
To provide an example for this guide, I will be making an adapted golf tee that makes placing the ball easier. To do so, I start out by taking a few measurements. I use a set of calipers to give me the dimensions of a standard golf tee, as well as the size of a standard golf ball.

OT3d.org

Start to Finish Guide 2

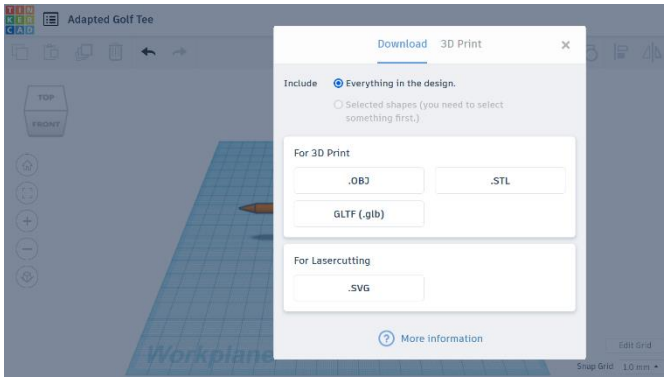
Once I have the measurements, I can start my design. Looking at a standard golf tee, I can break it down into a half sphere, a cylinder, and a cone. I can use the measurements I took to size these 3 objects in TinkerCad. Once that is done, I can then align the 3 parts so that they are centered and touching.



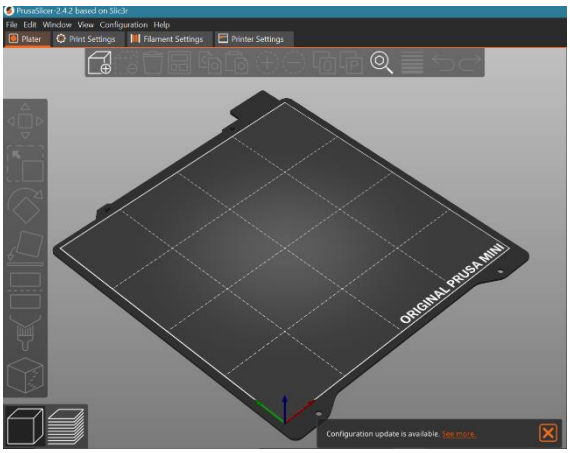


Highlighting all of the objects and selecting the 'Group' option creates an object that is roughly tee shaped. To create the divot for the ball, I will place and edit the dimensions of sphere to match the dimensions of a golf ball. I will then make it a 'hole,' align it with the top of the tee, and combine them to create a divot.

Now that my design is done, I can export the model that I created. TinkerCad allows me to export as a .stl or .obj, either of which will work for the next step. The exported file should download to your computer.

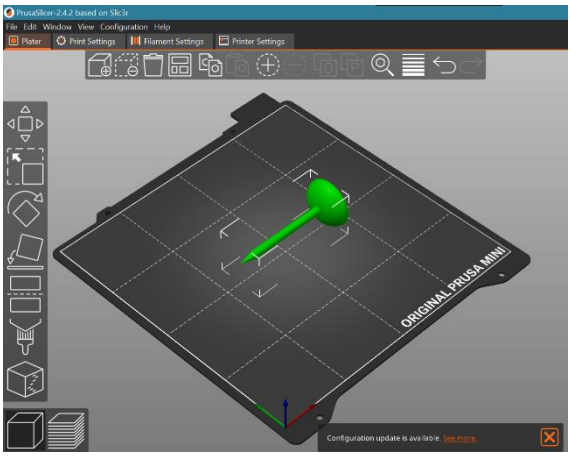


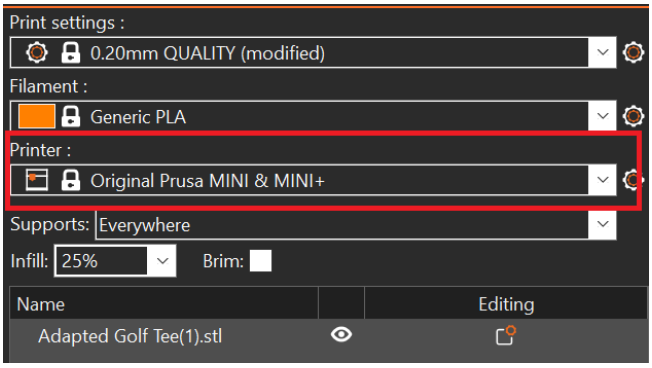
Helpful Hints: When downloading your slicer program of choice, you can set it as the default program of choice to open .stl and .obj files. This can also be done within your computer's settings.



Once the file has downloaded, we can open our slicer program. I will demonstrate the slicing process using Prusa Slicer, a free program with a good variety of features.

Once Prusa is open, I can then drag the file from my downloads into Prusa. The object will appear on the build plate. Up next, we will adjust the slicer settings.

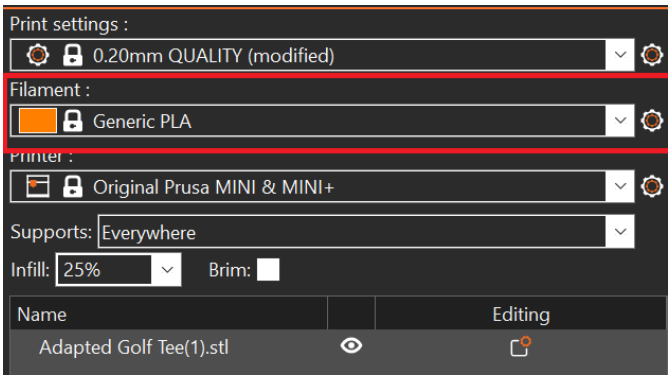


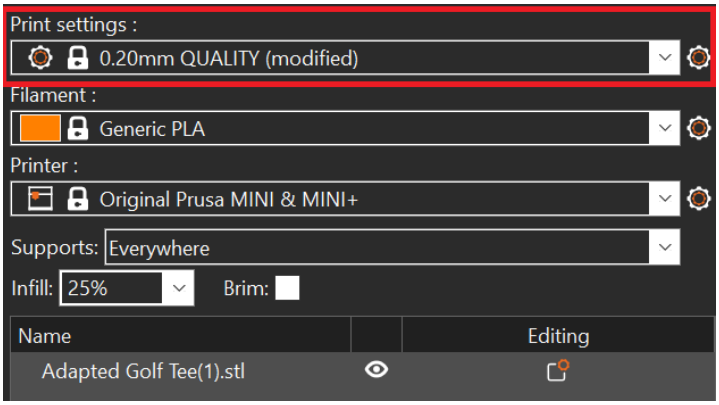


The first setting to check is the printer selection. This box will display the printer settings that have been previously downloaded. I will select the Prusa Mini from the drop down menu. The screen will then update to display the correct printer bed size.

Helpful Hints: If you do not see your printer in the printer selection box, you can click on the option to add a printer. Certain slicers will support different printers and I have found that using Prusa Slicer and Cura will provide the most options.

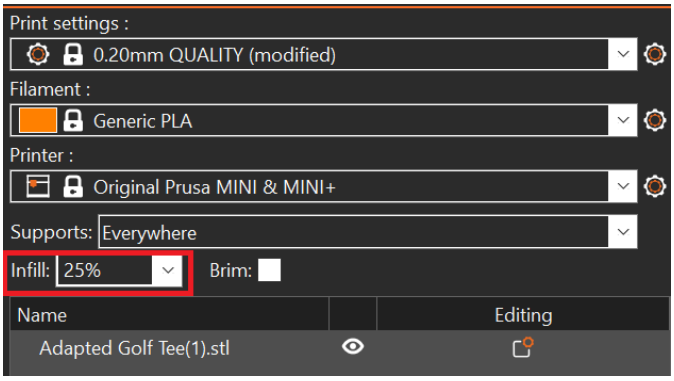
Next, we select the material with which we will be working. The general settings for this should be relatively accurate, but if you want to adjust these further you can go into the





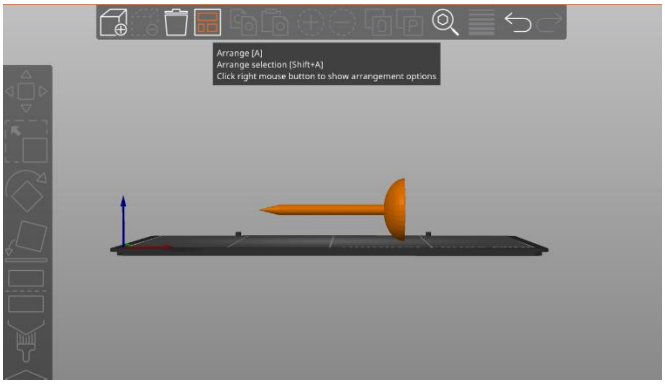
The tab on the top determines the speed and quality of our print. Faster prints tend to be slightly less refined, while high detail prints will take significantly longer. For most projects it is good to find a balance between the two, with .2 being a good starting point.

Up next, we will specify our infill percentage. Infill is the percent of the print interior that will be plastic. The more infill a print has, the more strength it will have. Less infill can lead to a weaker print but will print much faster. This print will take some force so we can set it to 50% for a good amount of strength.



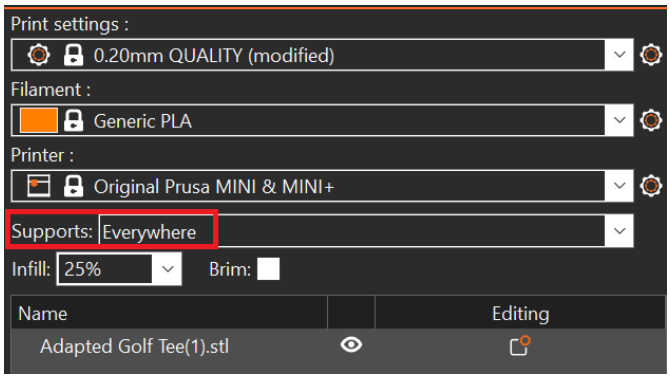
Helpful Hints:

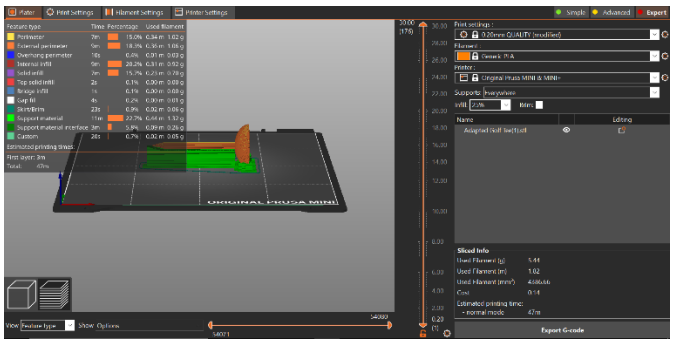
For most prints, you will often print in the 10-60% range. I tend to start with 25% as a good baseline, 50% for things that need to be strong, and 10% for quick prototypes.



Up next, let's orientate the object on the print bed. If I positioned it golf ball side down, it would have a large base which is good, but for this print I will orient it on its side so that each layer will have a longer and stronger connection for impacts.

Because part of the print is floating mid-air, it will need something to support these parts. The supports tab allows you to add supporting material under those overhangs to allow for such positioning.



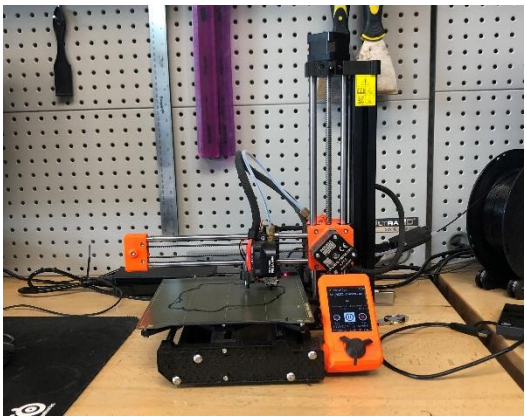


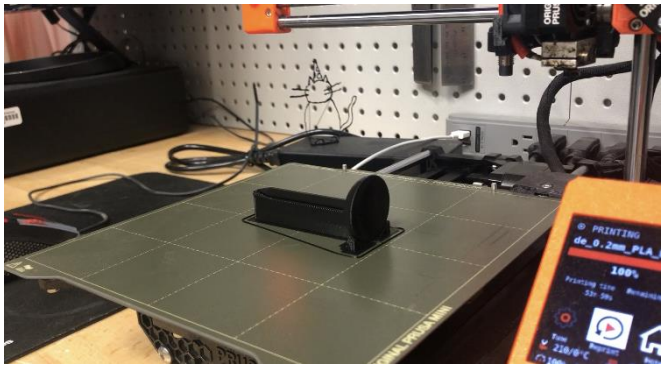
Once we are satisfied that all settings are correct, we can slice our file. This cuts the 3d object into layers that the 3d printer will follow. We can then save this as a .gcode file using the same button.

Helpful Hints: Different printers will use different means of transferring gcode. On the Prusa mini, I will use a usb drive, but other printers like the Ender 3 will use a micro SD card. Be aware of what your printer uses to save headaches.

Start to Finish Guide

Now that we have our file sliced and saved, we can start our print. First, I will turn the printer on and insert the flash drive. I can then navigate the printer's screen and select the print option by pressing the dial. The printer will then preheat and start printing. Watch for the first few layers, if possible, to make sure the print is sticking to the print bed.





Once the print is complete, you can pull it off of the build plate. If you used supports, using tools like flush cutters and sandpaper can help to clean them off and make the print look professional.

After cleaning up the print, you can complete any additional finishing that you would like. Be sure to turn off your printer and clean up any supports or stray materials to save time when you use the printer next.

This guide and more like it can be found at OT3d.org.

Happy Printing!