3D Printing Tutorial
Tinkerine Ditto Pro printers for
ELEC391

Engineering Services, Dept. of Electrical and Computer Engineering
University of British Columbia, Faculty of Applied Science

3D Printing with the Tinkerine Ditto Pro

In this exercise we will explore 3D printing CAD designed parts using the Tinkerine Ditto Pro. These 3D printers are available for ECE students enrolled in project courses, ECE research work, and other related UBC research.

Setup and Calibration

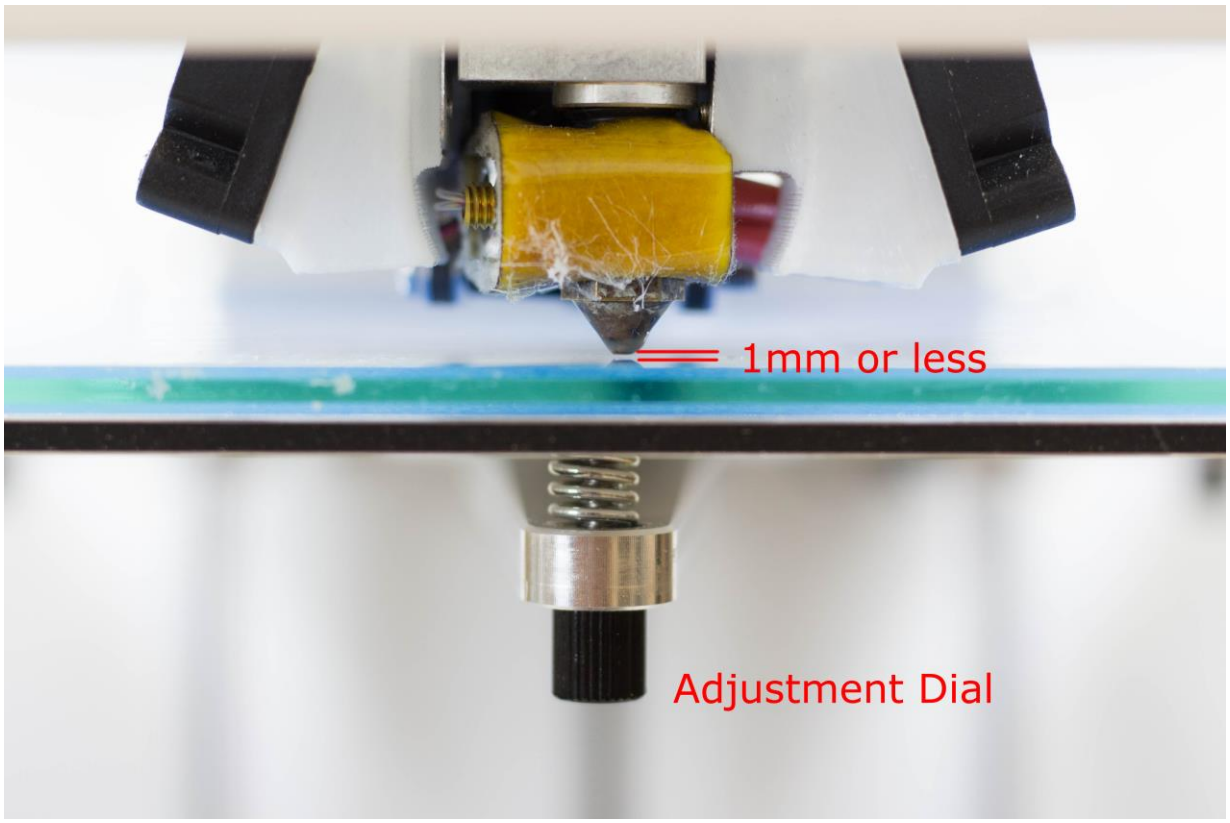
The first thing to do before 3D printing an object is to calibrate and setup the Tinkerine Ditto Pro 3D printer. Setup and calibration consists of 3 steps: leveling the printing bed, loading new filament, and preparing the build platform

1. Levelling the Printing Bed

This calibration wizard is used to make sure the printing bed is level. A level bed is important since any significant angles to the bed may cause malfunctions and misalignment during printing. To activate the bed level wizard, ensure that the Tinkerine Ditto Pro 3D printer is plugged in, and the power turned on.

Navigate to the Wizard menu using the scroll wheel. Once in the wizard menus, select Bed Level.

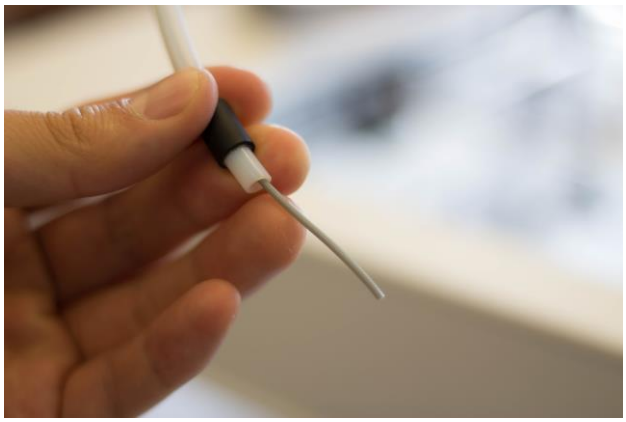
Follow the instructions on the display, ensuring that a small space is left between the printer head and the build platform at every point (about 1mm or less). To change the distance between the printing bed and the printing head, adjust the small black dials on the bottom of the bed.



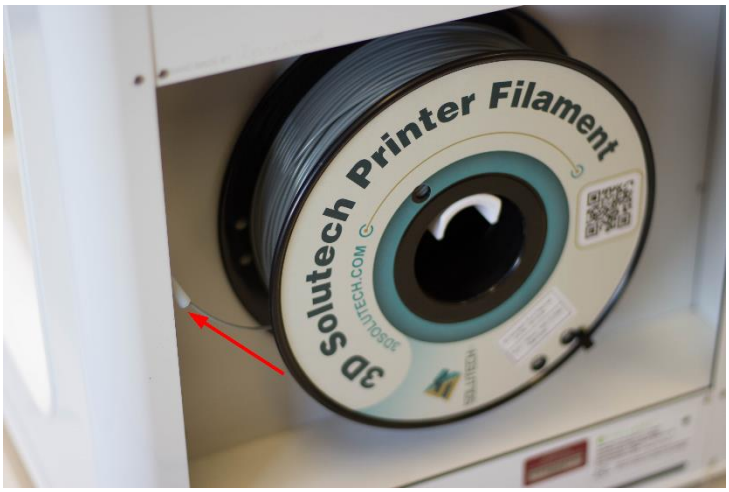
2. Loading New Filament

This calibration wizard is used when loading new filament into the printing head. In the wizard menu, select New Filament. Follow the instructions on screen. You may need to wait for the printing head to heat up.

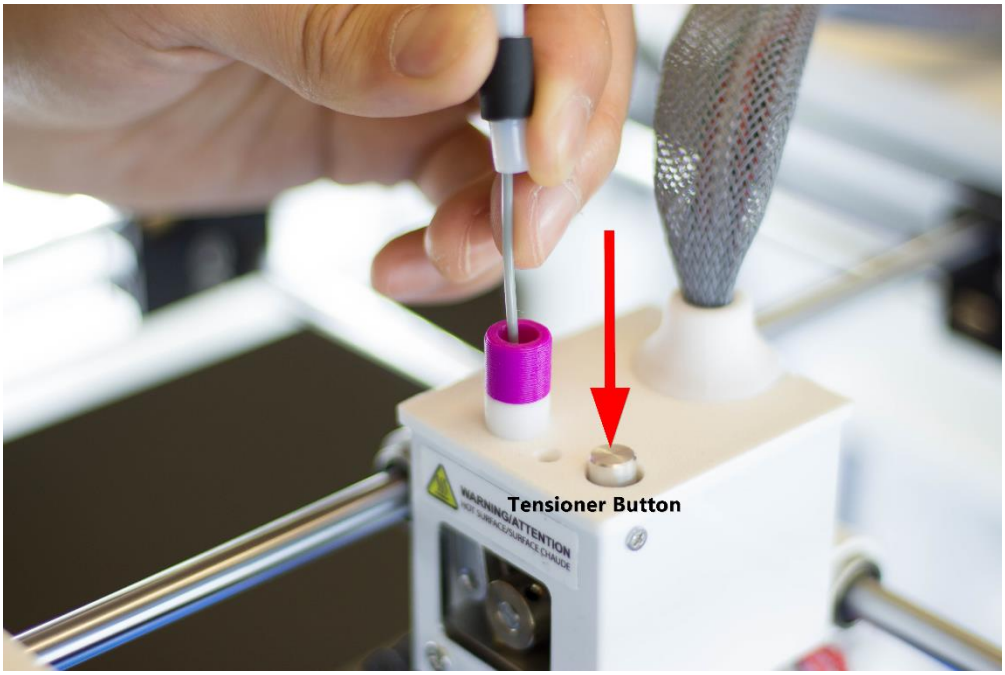
Remove any old filament and feed the new filament up through the guide tube and out the other side.



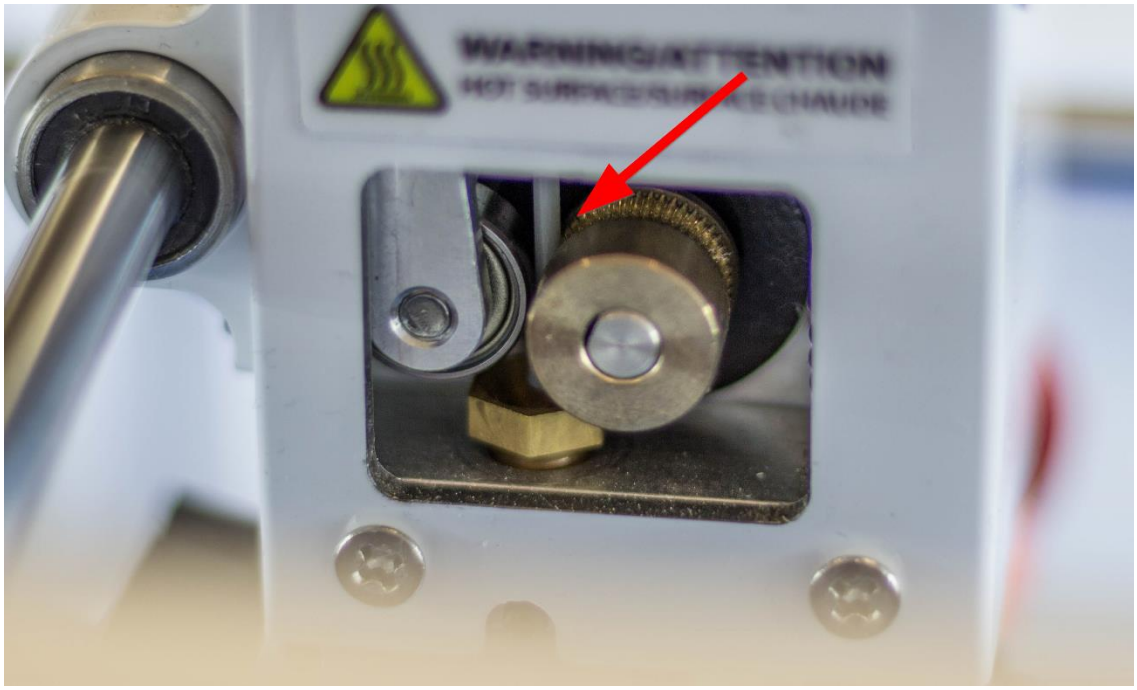
Ensure that the spool is hung neatly on the back of the printer, with the filament being spooled out clockwise and into the guide tube as shown.



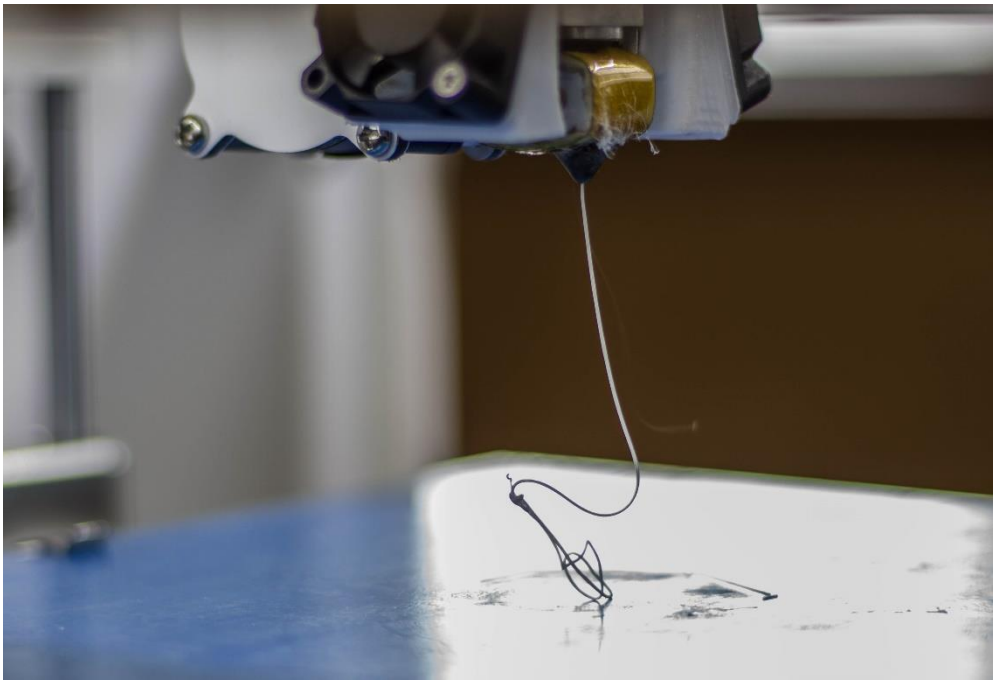
Press the tensioner button down to thread the new filament into the hot end of the printer head.



Note the position of the filament, make sure it is fed in between both gears.



Push the filament through the head until it comes out of the hot end. Trim off the excess filament. If the printer was previously loaded with a different colored filament, push the new filament through until you see the new color come out of the hot end.

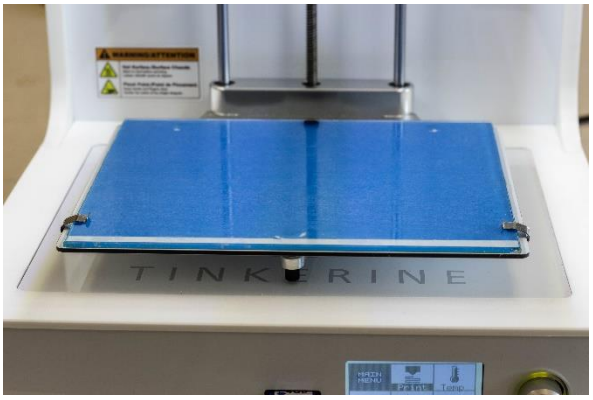


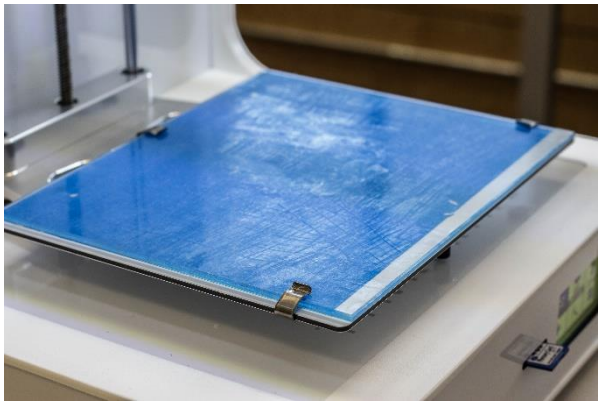
3. Preparing the Build Platform

First, while the printer is off, scrape off any dried glue or filament from the built platform using the tools provided.

Be sure to use the glass panes provided as the build platform, never print directly on the printing tray of the Tinkerine Ditto Pro. Once the built platform is clean, apply glue over the surface of the build platform. The glue will ensure that the filament sticks to the build platform.

Finally, ensure that the glass is firmly attached to the printing tray with the provided clips.





Preparing the .STL File

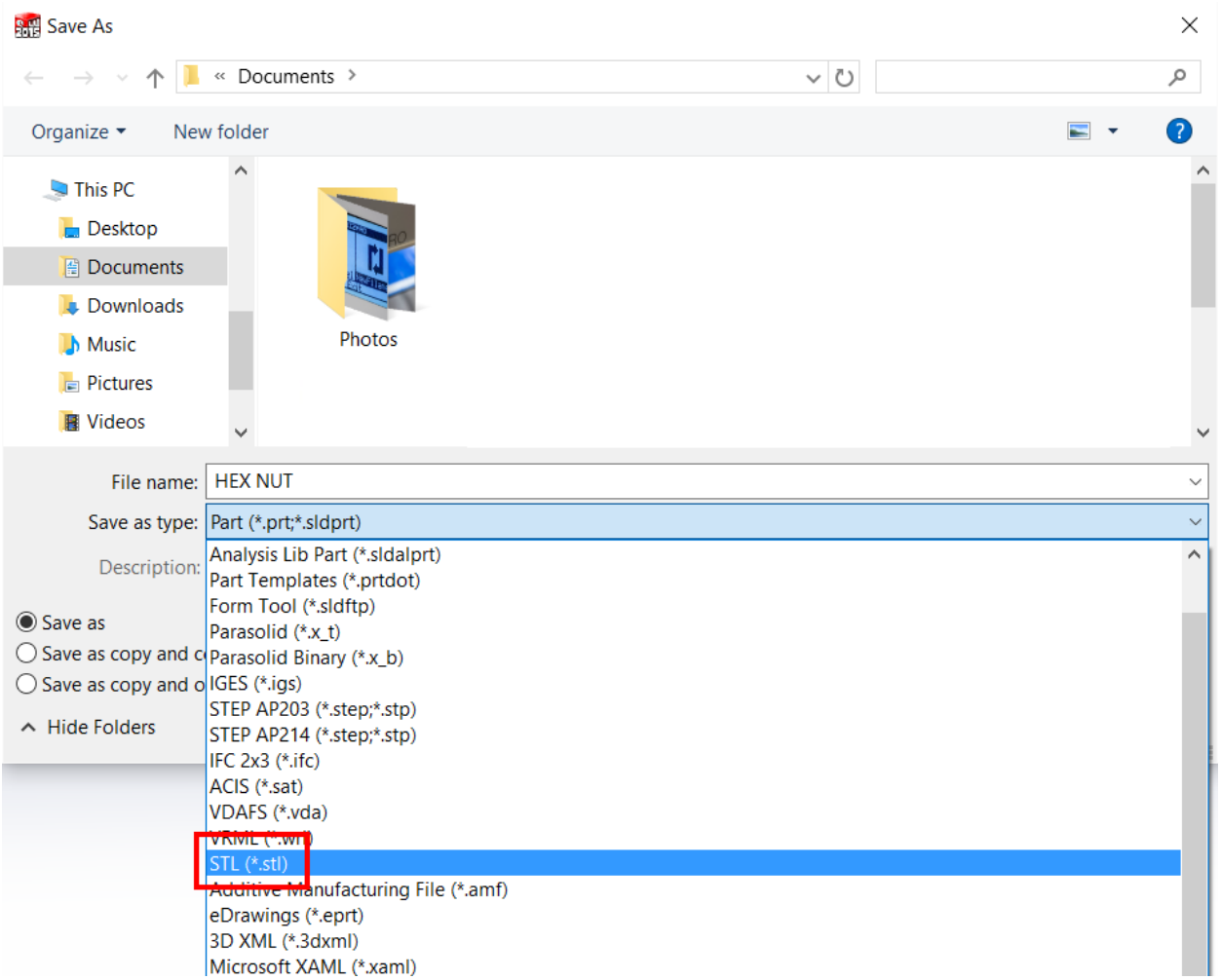
After the 3D printer is setup and calibrated, it is time to prepare our file for printing. After the part has been drafted in SolidWorks, it must be saved as a .stl file, then converted and sliced into a .g file before it is ready to be read and printed by the Tinkerine Ditto Pro.

Before beginning the process, first install the 3D printer slicing software. Tinkerine provides their own slicer program called Tinkerine Suite for free, it can be installed at the following link:

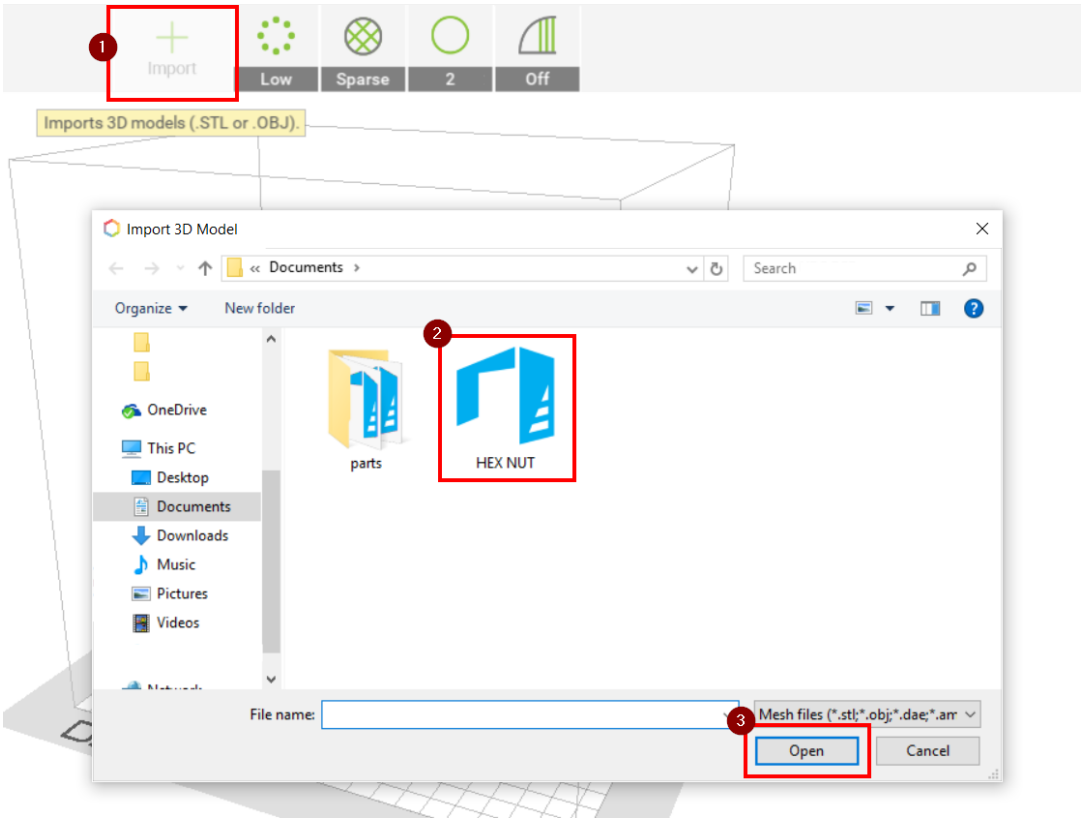
http://tinkerine.com/tinkerine-suite/

Once the slicer program has been installed, proceed to the next steps:

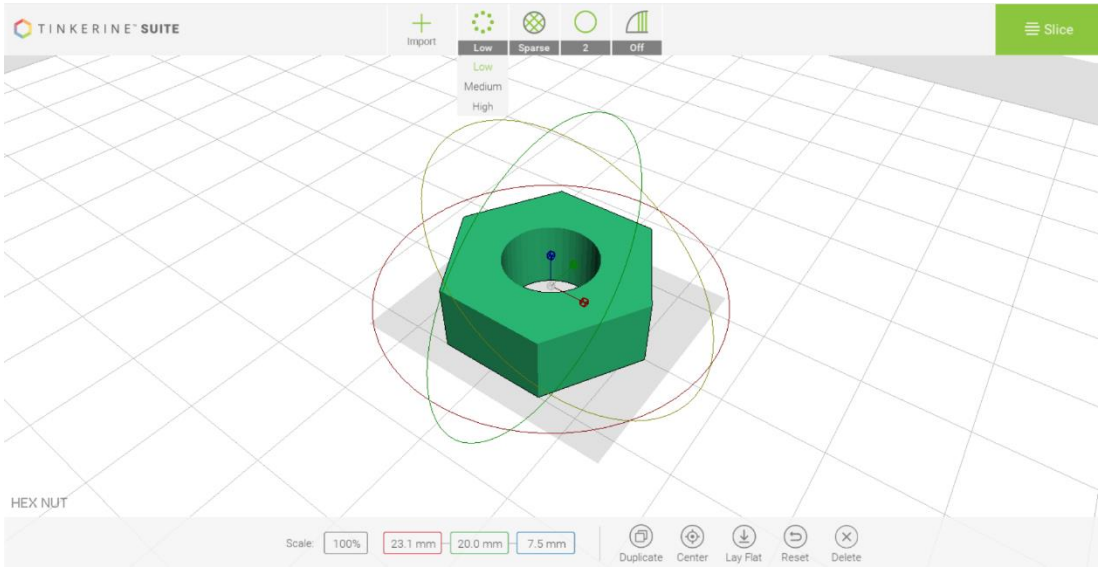
1. First, save the SolidWorks part as a .stl file. Select Save As, and choose .stl as the file type. STL is a file type that is universally used by most 3D printers and slicing software. It is the first step in preparing the SolidWorks part to be read by the 3D printer.



2. Once the part has been saved as an .stl file, we are ready to create a .g file using a slicer program. Open up Tinkerine Suite. To prepare our .stl file for printing, first import the .stl file into the program.



3. Once the file has been imported into Tinkerine Suite, the part should appear on the virtual printing bed. At this point there are a number of different options we can tweak, including Resolution, Infill, Wall, Support, Scale, etc.

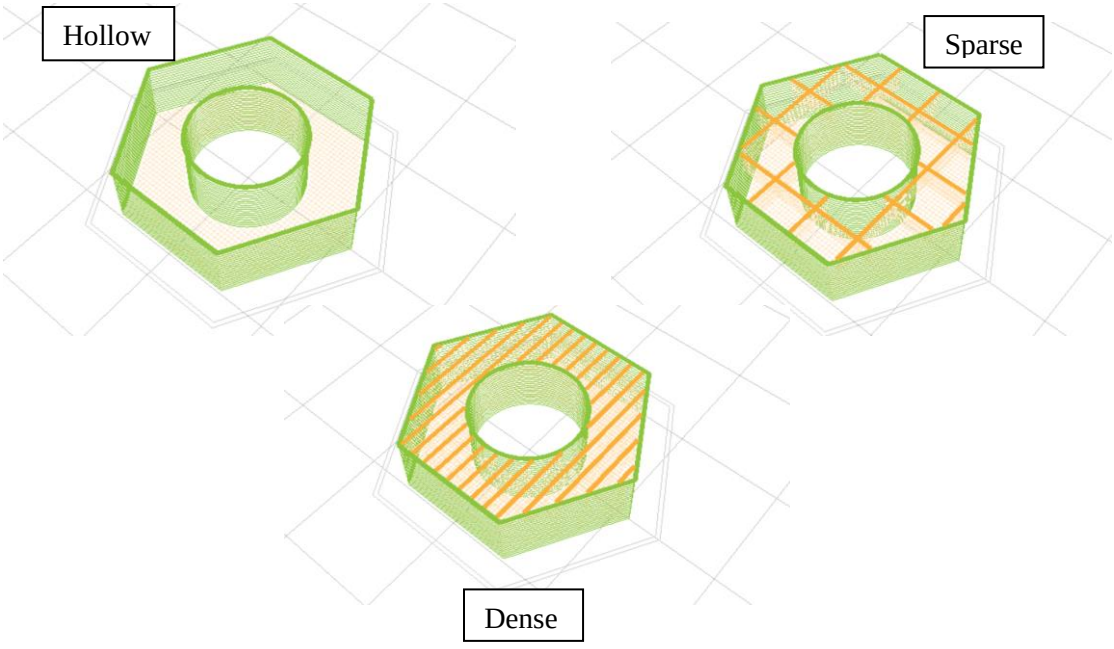


4. The resolution parameter controls the accuracy and precision of the printed material. The higher the resolution, the more precise the printing will be.

Resolution Level (All other parameters unchanged)	**Print time and filament used**
Low	Print Time 00h:03m Filament Usage 1g
Medium	Print Time 00h:05m Filament Usage 1g
High	Print Time 00h:09m Filament Usage 1g

Notice that the higher resolution comes at the sacrifice of printing time. A low resolution part will print much faster than a high resolution part. Generally, for simple parts without complex details (such as our hex nut), low resolution is adequate.

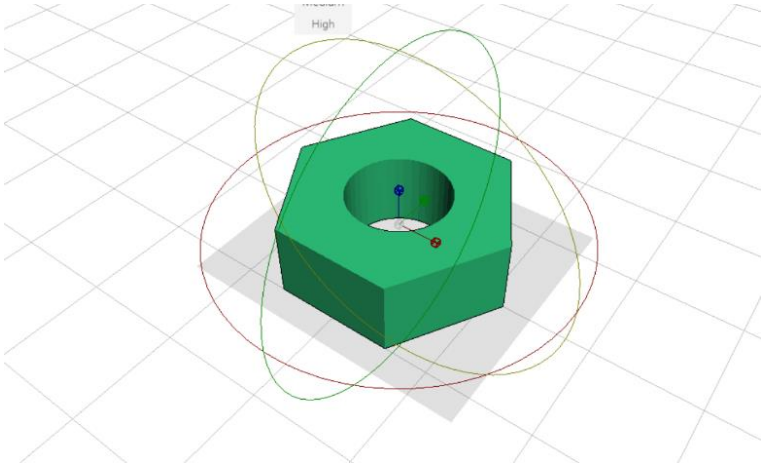
5. Infill describes the how the solid components are filled. Hollow means that there is no material inside the solid components of the part. Sparse will print a grid of support material inside the solid parts, and Dense will print a more compact matrix of support material. Similarly to resolution, a denser infill will increase printing time, but it will also add to the mass of filament used. Generally it is a good idea to have at least some infill, as hollow parts will be prone to failure during and after printing.



6. Wall is a parameter that controls how thick the printed walls will be of hollow/infilled components. Thicker walls translate to higher structural integrity but again comes at the cost of printing time and mass of filament used.

7. The Support parameter controls whether or not support material will be printed. Support material is an important consideration when dealing with overhanging components of a part that is to be 3D printed. When parts are overhanging, there is the potential for the part to not print properly. This phenomenon is especially prevalent when the angle of overhang exceeds 60° with the horizontal. In these scenarios it is good practice to print with support material but again, the print time and filament used will be increased as a result.

8. Once the desired parameters have been set it is imperative to consider the orientation of the part. Since the Tinkerine Ditto Pro 3D printer prints in layers of melted filament, the orientation of the printed part will have a tangible impact on printing time as well as the mechanical properties of the part. It is good practice to always orient the part such that circular holes/bores are oriented up/down instead of through sideways. Also, a flat/wide orientation is preferred to a tall/skinny orientation. The figure below shows the most ideal orientation for the hex nut we designed earlier.



9. Prepare file as a .g file by clicking the “Slice” button. Save this .g file to an SD card. We are now ready to print our part

Printing the .g File

After the 3D printer has been calibrated and the .g file prepared, it is finally time to print our part.

Simply insert the SD card into the Tinkerine Ditto Pro. Power on the device and navigate to the print menu. Find the file to be printed and push the button to begin the printing process. After a brief heat up period, the printer should start printing the part. Be sure to monitor the printing carefully and look out for any malfunctions by the printer itself or the filament spool. Once the part has finished printed, remove the part using the scraper.