3D Printing Quick Start
&
Resource Guide

JCPL

**JESSAMINE COUNTY
PUBLIC LIBRARY**

Simplify3D Software – JCPL 3D Slicing Software:

https://www.simplify3d.com/

3D Model Repositories

Thingiverse

http://www.thingiverse.com/

YouMagine

http://www.youmagine.com/

3D Modeling / CAD Software

TinkerCAD

http://tinkercad.com/

SketchUp Make

http://sketchup.com/

Autodesk Meshmixer

http://meshmixer.com/

Blender

http://blender.org/

Filament Resources

Hatchbox

https://hatchbox3d.com

Proto-Pasta

https://www.proto-pasta.com

Dremel Ideabuilder PLA Filament

https://3dprinter.dremel.com/3d-printer-filament

-    **Our Printers Require 1.75mm Diameter PLA Compatible Filament**   -

Simplify3D Quick Start Guide

The Configuration Assistant

When you open Simplify3D for the first time, you may want to open the Configuration Assistant. From the drop down menu select the Dremel Ideabuilder 3D20 to automatically configure all of your settings.
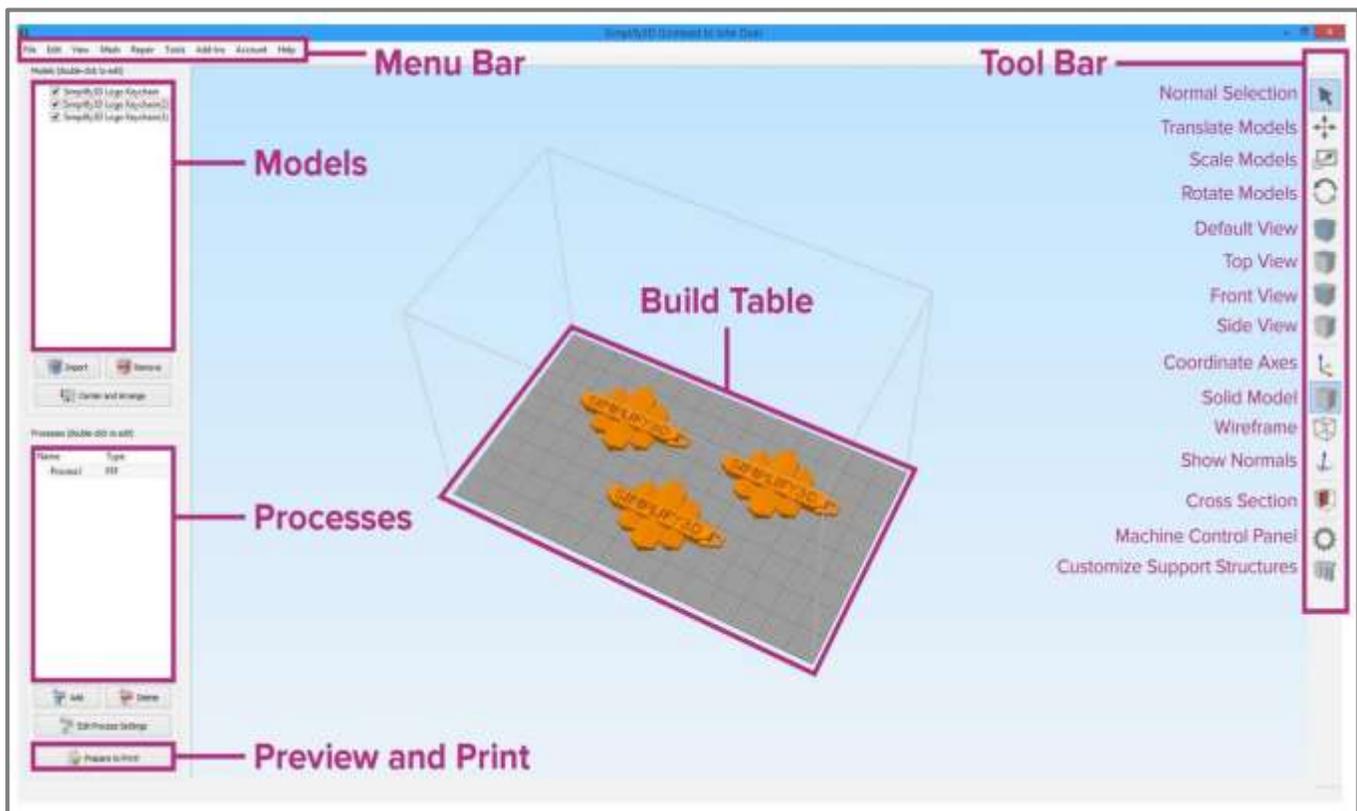
You can access the Configuration Assistant at any time to reset your profile settings.
Go to Help > Configuration Assistant.

The Interface Layout

After the printer configuration is complete, you will see the main Simplify3D software interface.

The major elements of the software interface are identified below:

Four Steps To Your First Print

1. Prepare

The first step of the 3D printing process in Simplify3D is to import the model you want to build.
You may obtain 3D models from a variety of online repositories, such as Thingiverse.com or Youmagine.com, or you can make your own model using CAD software such as TinkerCad or Sketchup. Save or download the 3D model file as an .STL file.

Click Import in the Models section and navigate to the folder where you saved your file. You can also drag-and-drop file(s) onto the Build Table. Click on Center and Arrange to move your models to the center of the build plate. Use Tool Bar options (listed vertically on the right side of the window) to view and manipulate your model.
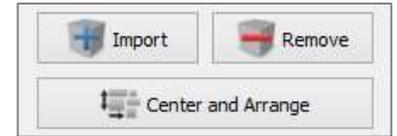
2. Process

Now you will define a Process that specifies how your model is to be constructed. Most models use a single process; models that are more complicated may use multiple processes. The first Process is configured automatically based on the printer selected in the Configuration Assistant. You can make any adjustments you choose, but "Process1" is a great place to start!

To make adjustments to your Process, double-click on the Process or click Edit Process Settings. This will open the FFF Settings window.*

The FFF Settings window contains all the settings that specify how your model is to be constructed:

- Select Profile: Loads a printer profile and the corresponding settings, which have been selected to ensure optimum quality.

- Auto-Configure for Material: Adjusts temperature, extrusion, and cooling settings based on the material you select.

- Auto-Configure for Print Quality: Adjusts layer height, infill density, and other quality settings based on the level you desire.

- Auto-Configure Extruders: This selection box appears if you selected a printer with multiple extruders. Select the extruders that you intend to use and they will be automatically enabled.
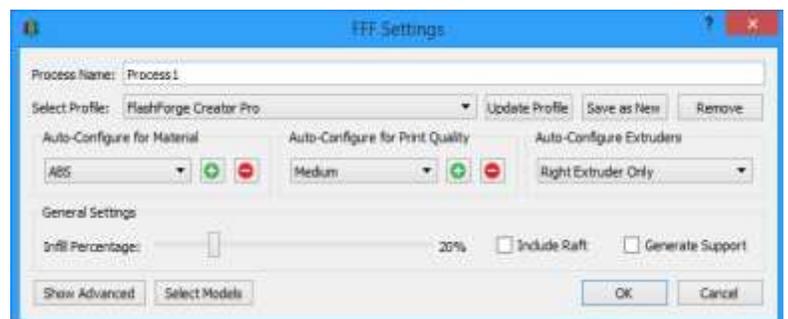
*The acronym FFF stands for Fused Filament Fabrication, and refers to 3D printing using filament-based materials.

- Infill Percentage: Adjusts the interior solidity of your 3D print. 0% is completely hollow and 100% is completely solid. A value in the range of 20-50% is frequently used.

- Include Raft: Generates a raft structure underneath your part. Rafts are used to improve the bond to your build platform and may help reduce warping on large models. A raft can also compensate for slight leveling issues if you have difficulty with first layer adhesion.

- Generate Support: Turns on the Support Generation Tool. Support structures are used when your model has steep overhangs or unsupported areas. Typically, overhangs greater than 45° benefit from support material. Simplify3D also allows you to customize the location of your support structures; simply click to add or delete. Support structures break away cleanly and easily, minimizing damage and ensuring high quality models.

Click OK when you have finished configuring your settings in the FFF Settings window. At any time, you can modify the settings by double-clicking on the Process in the list, or by clicking Edit Process Settings. Click Show Advanced to view additional options.

3. Preview

After you have finalized your Process settings, click Prepare to Print! This will initiate two important functions: Simplify3D will "slice" your model and transition to the Preview Mode. "Slicing" is the term used to describe the process of converting a digital model into line-by-line printer instructions called G-Code. The lightening-fast slicer in Simplify3D quickly prepares G-Code based on the variables you selected in the FFF Settings window.
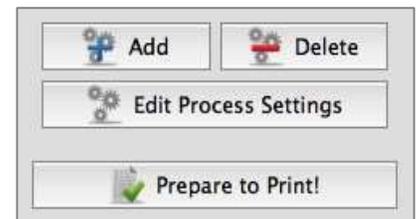
The animated Preview Mode allows you to inspect all aspects of your print operation prior to execution. The software reads the G-Code and displays line-by-line or layer-by-layer how your model is to be constructed.

A useful setting is to select Preview by Layer and place a checkmark in Single Layer Only.
Together, these selections will allow you to view one layer at a time.

If you want to make further adjustments to your model or process before printing, click Exit Preview Mode. This is a great time to ensure that your model is constructed properly.

The Preview Window also contains estimates about your print project. These Build Statistics appear in the top left corner of the Preview window.

- Build Time: How long it will take to finish your print.

    - Build Time is an estimation and can be extremely inaccurate

- Filament Length: How much filament it will take to build your model.

    - This is an estimation based on the elasticity of traditional PLA filament.

- Weight: How much the print will weigh when completed.

- This is an estimation based on the weight of traditional PLA filament.

4. Print

When you are satisfied with the Preview, there are two methods for how to start your print:

Begin Printing over USB: if you are using a USB connection, you have the option to view the build sequence real-time on your monitor by selecting Live Preview Tracking. For additional information on USB communication, access the Machine Control Panel by going to Tools > Machine Control Panel to view a broad range of information that is transmitted via USB.

Save Toolpaths to Disk: This saves your print file to a user-defined location, such as a folder on your computer or an SD card. The software will save a .gcode file and then automatically create a .X3G, .MakerBot or other print-type file if needed.

Additional Resources

For more information about any particular setting, use your mouse to hover over the setting to display a tooltip describing the functionality.

1. Printers

All of our printers are Dremel Ideabuilder model 3D20. For additional information go to the Dremel Ideabuilder website:

https://3dprinter.dremel.com

2. Filament

We stock standard PLA filament. Our printers are strictly PLA compatible; they are not capable of printing ABS, TPU, PETG, or any other exotic filaments. Hatchbox, Proto-Pasta, and AMOLEN make unique PLA compatible filaments, including Bronzefill, Copperfill, Rustable Iron, Silk & Marble, Woodfill, and PLA+ Nylon. These filaments, being PLA compatible are fully capable for use in on our printers. Our printers use 1.75mm diameter filament; they will not work with the larger 3mm diameter filament.

3. Resolution

The maximum resolution of our printers is .100mm (100 microns). Resolutions lower than 100 microns can result in machine faults and errors. Our printers will print as large as .300mm (300 microns). Resolutions larger than 300 microns will result in print failures.

4. Finishing Touches

Some models may require additional work after printing. Sanding and polishing are very popular options. If painting, we recommend Rust-Oleum Plastic Primer as a base coat.
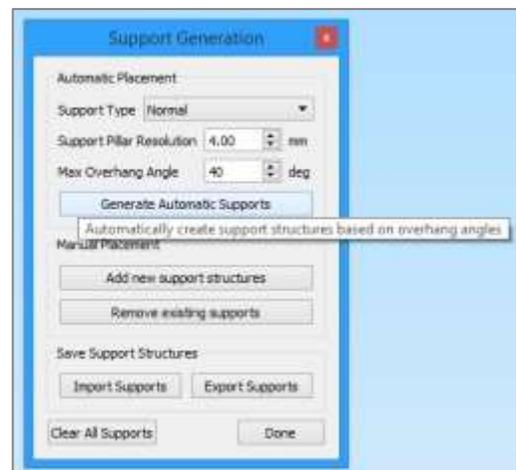
This will help to cover up some layer banding, which is a common surface imperfection with FFF manufacturing. Super glue can be used as an adhesive if models break or if needing to join multiple pieces together.