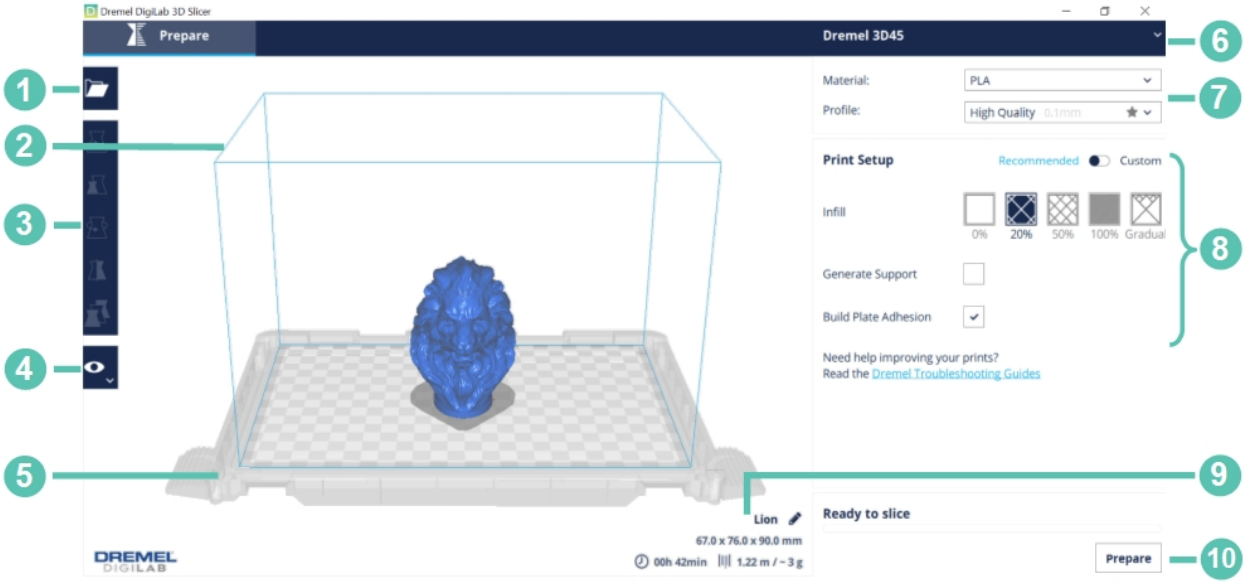
Digilab 3D Slicer Basics

Step 1: Familiarize Yourself with the Workspace



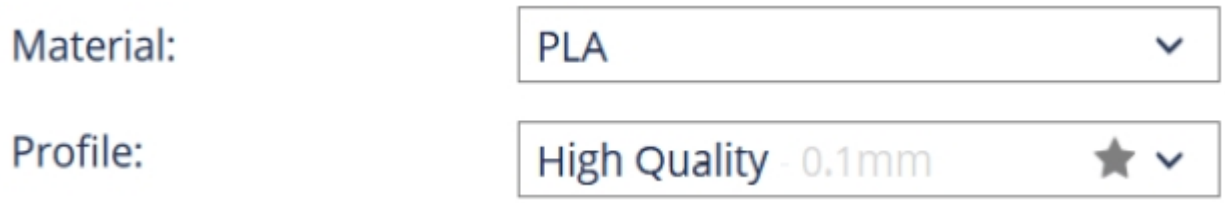
The figure above shows the interface of the Digilab Slicer with all features labeled. The features are described below:

1. **Open File:** Import the file to be printed.
2. **Build Volume:** The thin blue line represents the edge of the build volume, which is 10" x 6" x 6.7" (255 mm x 155 mm x 170 mm).
3. **File Preparation:** Move, scale, rotate, and mirror the file in the file preparation tab.
4. **View Mode**: Visualize the file as a solid, x-ray, or by layers based on the setup being used.
5. **Build Platform:** Represents the location to where the models adhere when printing.
6. **Printer Selection:** The selected printer should be 3D40.
7. **Material and Quality:** Ensure PLA is the material selected. Select the profile (quality) of the print.
8. **Print Setup:** Adjust infill, support, and build plate adhesion settings.
9. **Print Estimates:** Estimate print time after preparing the file.
10. **Prepare:** Prepare the file to be printed based on the print setup being used.

Step 2: Select the Material and the Profile

Ensure PLA is the material selected.

Select the print profile (quality) for your print. The quality is determined by the layer height of the print. A smaller layer height will create a more detailed part, with a longer print time. A taller layer height will print faster, but with less details depending on the part being printed. As you go towards a lower quality, the print layers are more visible and details may go missing.



Step 3: Import the File

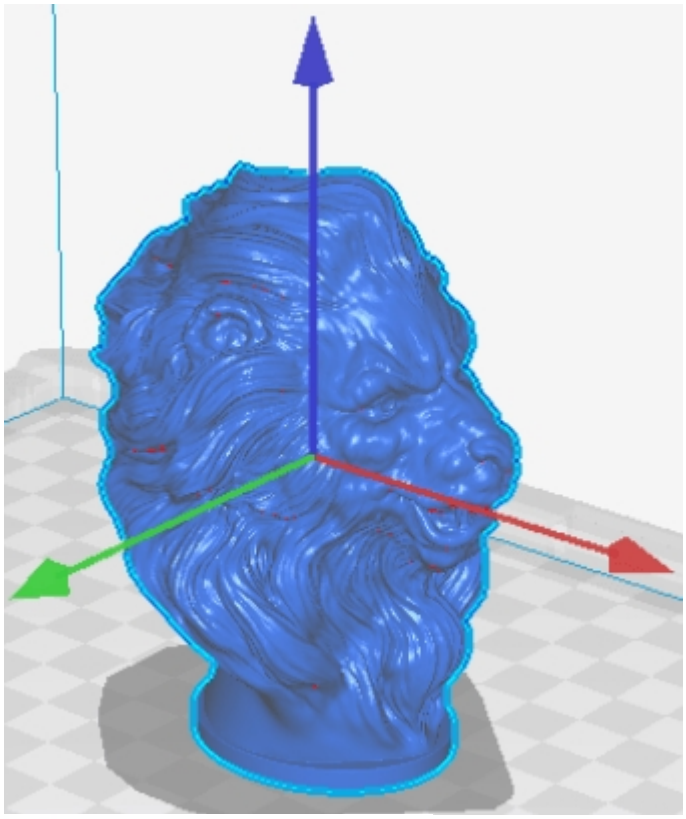
Import the **.stl** file using the Open File button.

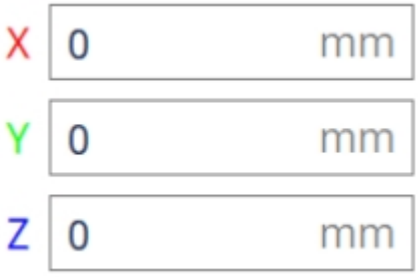
Note: If your file is not visible after importing, zoom in to the center of the build platform and check if the file is there. It may occur that the part has been imported in different units (millimeters instead of inches), therefore importing a very small file with the wrong dimensions. In this case, hold Control + A to select all the files on the build platform, and scale the part up by 2540% to convert the dimensions from millimeters to inches. Ensure the dimensions are correct after scaling the part up.

Step 4: File Preparation (OPTIONAL)

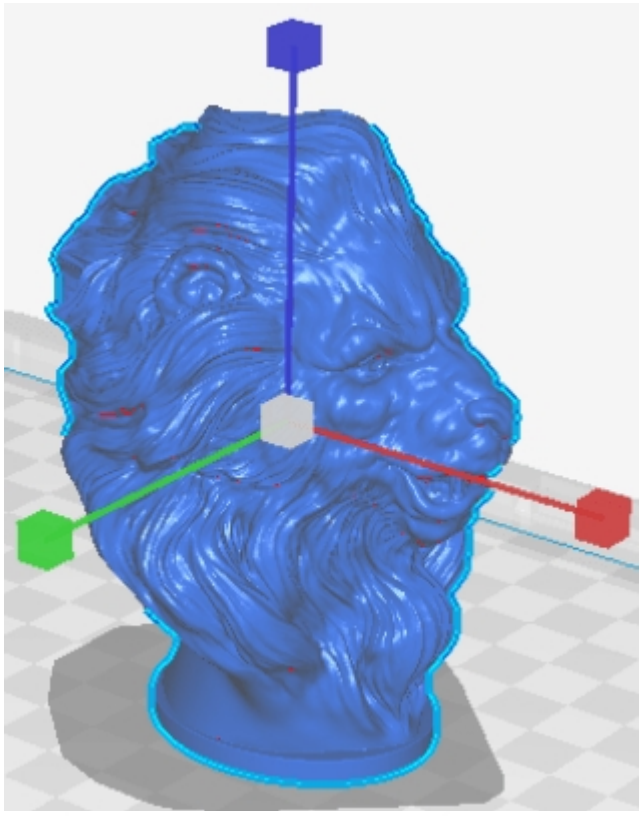
The following basic transformations can be performed to prepare the file:

Move: Move the part along the build volume by clicking the arrows on the part (shown in the left figure below) and dragging it or inputting the distance of the translation (shown in the right figure). Ensure the part is inside the build volume and in contact with build platform.



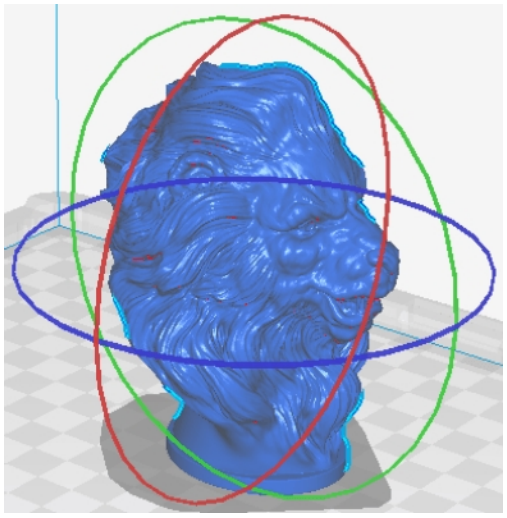


Scale: Scale the part by clicking on the file and dragging the scaling arrows (shown in the left figure), or by inputting the desired dimensions or scaling percentage (shown in the right figure). Check the uniform scaling option in order to make the scaling consistent in X, Y, and Z axes. Ensure the part is inside the build volume.



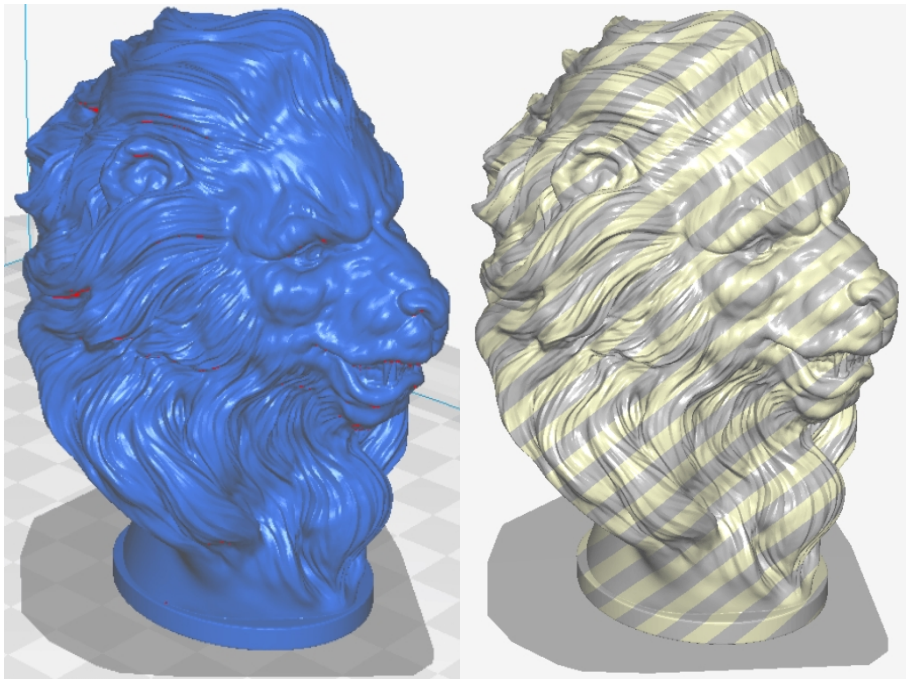
X	76	mm	100	%
Y	67	mm	100	%
Z	90	mm	100	%

Rotate: Rotate the file by holding and dragging on the colored lines around the file (See figure below). The angle will be changed by 15 degrees increment. Ensure that the part is in contact with the build surface.



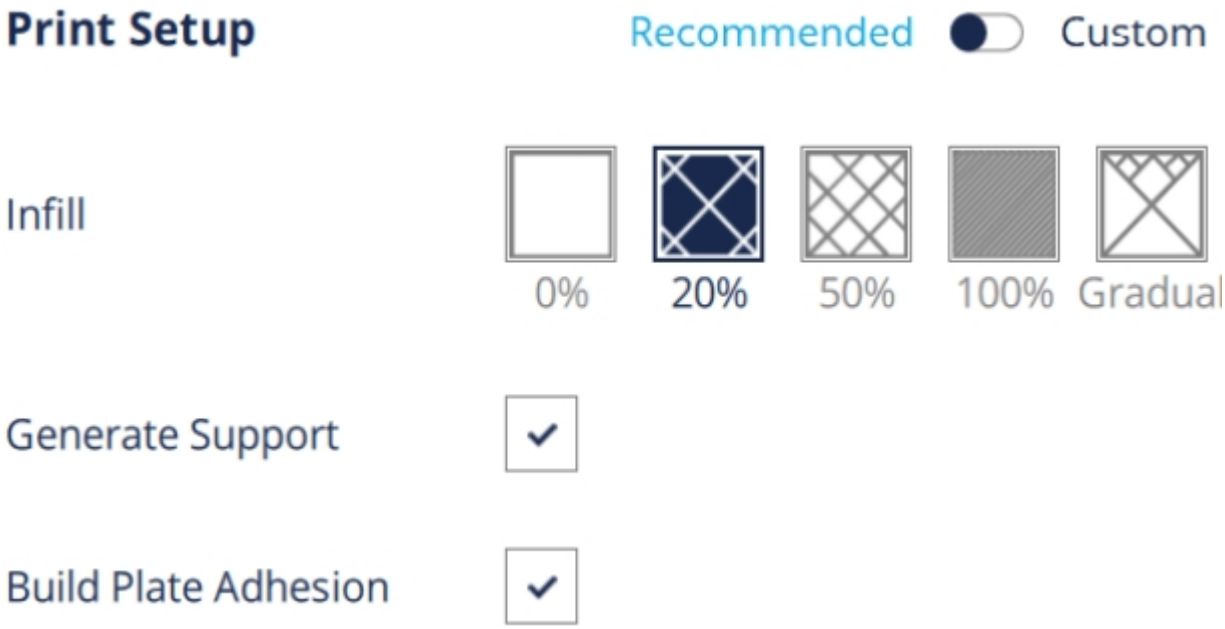
Mirror: Mirror the part across a certain plane by clicking on the arrows around the file.

Note: If the part appears not as a single solid color, it means that the part is not inside the build volume, and therefore will not be printed. Adjust the file size and placement until the solid color is achieved.

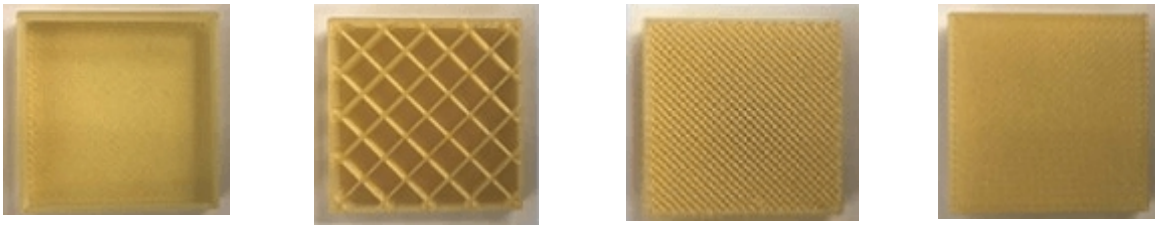


Step 5: Adjust print setup

Infill, support, and build plate adhesion settings can be adjusted under Print Setup.



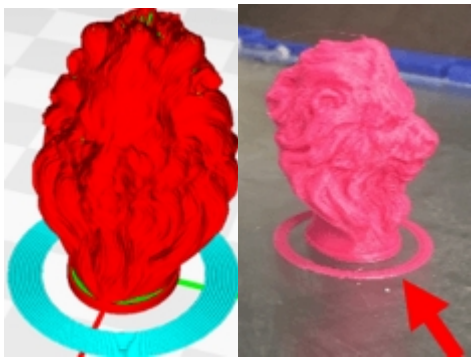
Infill: The infill is the internal support material in the part. The infill percentage can be adjusted from 0% (empty) to 100% (solid). Higher infill results in *a stronger* and *heavier* part, with a *longer* print time. Smaller infill results in a *lighter* part with a *faster* print time. Gradual infill will gradually increase the amount of infill towards the top of the model.



0% 20% 50% 100%

Support: Support structures (encircle in yellow below) will be printed where there are overhangs on the part. Without these structures, the model would not be able to be printed. Those supports can be removed after the part has been printed (see picture on the right).

Build Plate Adhesion: This setting enables printing a skirt under recommended settings (see picture below), which will add a flat area around or under the object that can be easily removed after printing.



For advanced users, select Custom under print setup. That enables a wider range of print settings to be adjusted, creating the optimal print setup for your print.

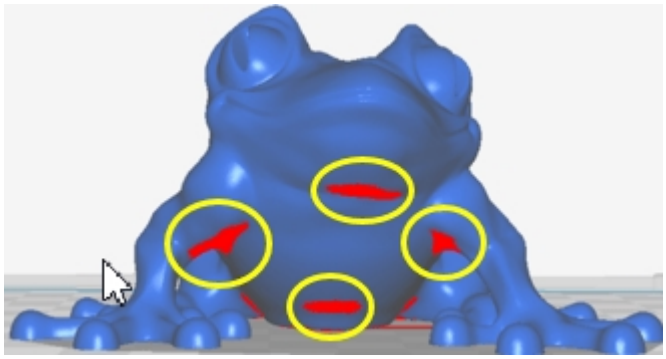
Step 6: Prepare Button

After the print setup has been adjusted, the file is ready to be sliced. Hit the prepare button and the software will create a file that can be 3D printed by the Dremel 3D printer.

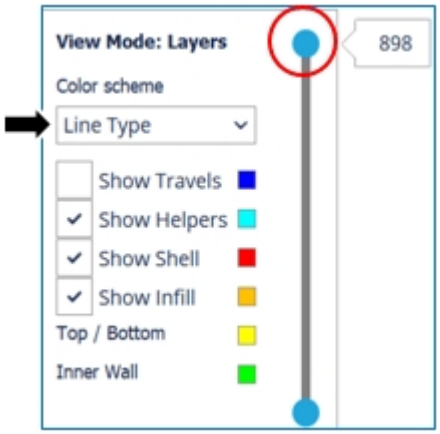
Step 7: Review the file

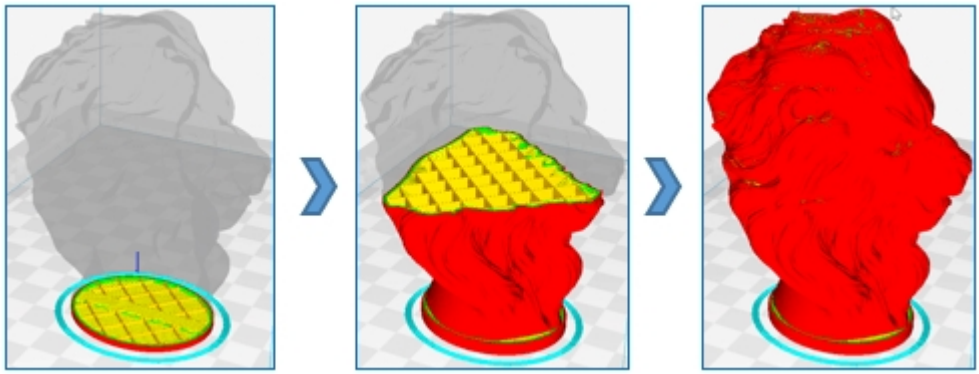
The last step prior to saving the sliced file is to review it to ensure there are no issues. Under View Mode, there are three options to visualize the file: solid, x-ray, and layers. Solid and layers view modes are described below:

Solid View Mode: Utilize the solid view mode to ensure there are no gaps or missing sections within the part. Parts with a width smaller than 0.4mm will not be printed. This mode will also assist you deciding whether the part requires support or not. Surfaces that appear red (see picture below encircled in yellow) may require support structures when printing.

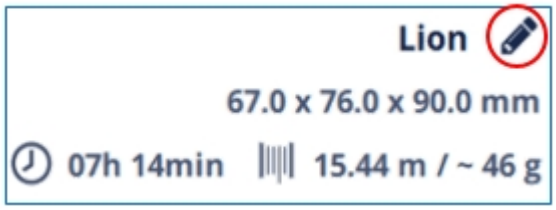


Layers View Mode: This mode is crucial to ensure there are no missing layers, gaps or missing sections. Select Line Type under color scheme to identify the different line types. Hold and drag the blue dot (encircled in red below) on the vertical line to visualize how the layers are being printed.

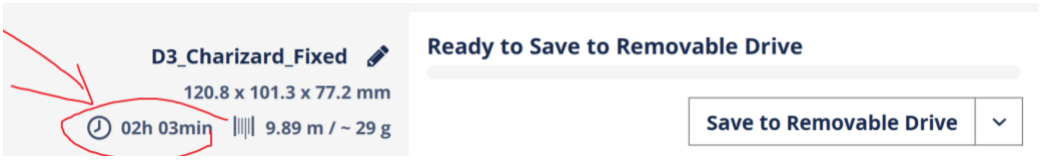




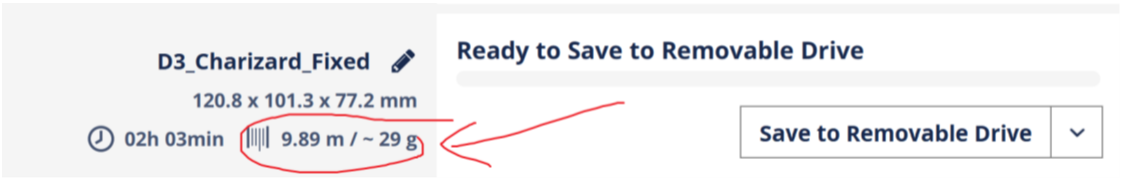
Also, review the print estimates. The software will estimate the print time, filament necessary and part weight. You can rename the file by clicking on the pencil symbol (encircled in red bellow) next to the name. as displayed by Dremel Digilab 3D Slicer (see circled area in below image) to



Estimated Time That is Needed to Print: The estimated time to print is displayed by the Dremel Digilab 3D Slicer software (see circled area in below image).



Amount on Filament on Roll/How Much Filament Do I Need: You can determine whether you have enough filament to complete your object before building by comparing the length of filament needed as displayed by Dremel Digilab 3D Slicer (see circled area in below image) to the length of filament on spool indicated by the length gauge. If 3D40 runs out of filament during a build, it will pause until more filament is added.



Step 8: Save file to USB drive

Click on the Save to Removable Drive Button and the file will be saved to the available USB drive on your computer. Hit the down arrow if you need to choose a specific location.