Printing

Markforged Onyx Pro (Industrial FDM 3D Printer)

Onyx/Fiberglass Printer Printing Procedures

IMPORTANT: Do not print with Onyx or Fiberglass until you have been trained on this equipment.

WARNING! Always confirm with SCiL Staff before using this printer!

Step 1: Obtaining and the model to print

First, you will need to make sure the file you would like to print is a .stl file or .step file. The slicer program only accepts .stl and .step files, so something like .obj will not work.

- To export a model as an stl file in Maya, first, go to Windows→Settings/Preferences→Plug-in Manager, then in the search bar, type in “stl”, and check the load and auto load boxes for the result titled “stlTranslator.mll”, then hit refresh, and close that window. Then, go to File, and instead of hitting export selection, click the box next to it. Under “General Options”, there is a drop down menu labeled “File type:”. Look for the “STLExport” option, then save and close that window. You can now export files from Maya in an stl format. Just make sure that when you do go to export the model(s), that you change the “Files of type:”drop down and select STLExport.
- If you used blender to make the model, you can export as an stl without additional setup.

The computer with access to the browser based slicer Eiger (eiger.io), is the Mac in the Makerspace.

Step 2: Setting up the model to print

Once you have navigated to the slicer, you should be in the Library screen of the site. It is recommended that you make a folder (and put your name as part of the folder name) to put the model(s) you are working with in, that way they don’t clutter up the Library.

Once in the folder you made, you can import your model. When you do, make sure that the Configuration is set to Markforged Smart Settings, and the Destination Folder is set to your folder.

The controls to navigate the slicer are as follows:

- Hold Left mouse button: moves the camera view via orbit of the model in the scene
- Click Left mouse button: while hovering over the model, selects a face that will rotate the model to make the selected face touching the build plate.

On the right side of the screen, make sure you have the Material set to Onyx, and the Reinforcement Material set to none, or if you have been authorized to do so, you can set it to Fiberglass.

Step 3: Adding Support Structure to the Model

The slicer will automatically add support structure where it is deemed needed. However, you can change how the supports generate. On the right side of the screen, there is a details panel with three menus: General, Settings, and Infill. For the most part you should not change any of the settings in either of these three menus.

Step 4: Slicing the Model

Once you have set up your model how you would like it, click the Save button at the bottom right of the screen. Eiger will then slice the model, as well as applying supports to the model. To see the support generation or specifics about the model, click the XRAY button near the top right of the screen. The support structure will be in purple, meanwhile, the model will be a transparent white. You can make the model visibly solid by clicking the Part button near the top middle of the screen, and you can do the same for the Support (and Fiber if applicable), by clicking their respective buttons to toggle between transparent or solid.

Note the build details panel on the left side of the screen, and the model’s respective costs.

WARNING REMINDER! Always confirm with SCiL Staff before using this printer!

Step 5: Printing the Model

Start by clicking on the Print button at the bottom right of the screen. This will take you to another page, in which you can choose how you would like to print the model. On the right side of the screen, you will see an option to set Printer Type and Select Printer. Make sure that the Printer Type is “Desktop Series (Onyx, Mark Two)”, and Printer Type is set to “Onyx Pro2”.

Once those have been set, click the Print button at the bottom right of the screen, and you can add it to the printer’s queue. From there, once the printer is not busy, you can start printing your model.

While the extruder nozzle heats up, apply a layer of glue(from a glue stick) on the build plate where the model will be printed. Apply enough glue for the plate to be sticky, but make sure it’s a smooth layer of glue, so that it does not gather and get run through or picked up by the extruder, as that will lead to gunking up the filament and causing uneven layers, which may lead to a failed print.

You may need to turn off the printer and turn it back on if the printer is having problems connecting to the laptop for printing. Double check the internet connection on the printer to make sure it's connected properly, and make sure the laptop is on the same internet connection.

When you start the print, make sure that you stay around to watch the first layer, to ensure that nothing goes wrong, or in the case of the print failing, you can stop it before it wastes more expensive filament.

Step 6: Cleaning up the build plate post print

Regardless of whether the print was successful or not, you will need to clean off the build plate once completed.

Revision #2
Created 2026-02-02 18:31:23 UTC by Mariah
Updated 2026-02-03 19:54:24 UTC by Libby Ellwood