

SeeMeCNC Guides

Setting Up Slic3r

This guide will walk you through the steps of setting up Slic3r for slicing and printing on your SeeMeCNC printer.

Written By: SeeMeCNC



Slic3r

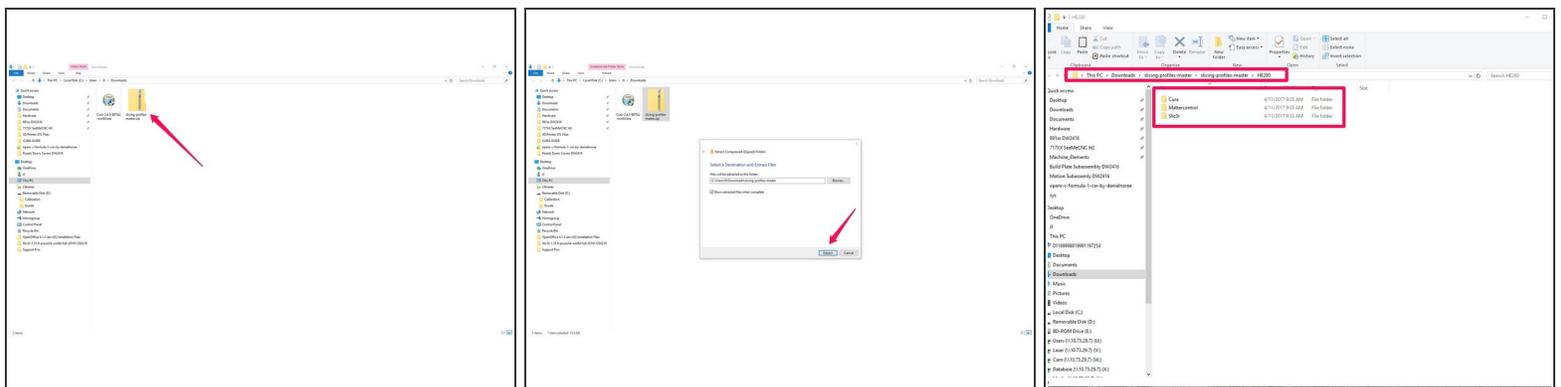
G-code generator

Step 1 — Download Slic3r



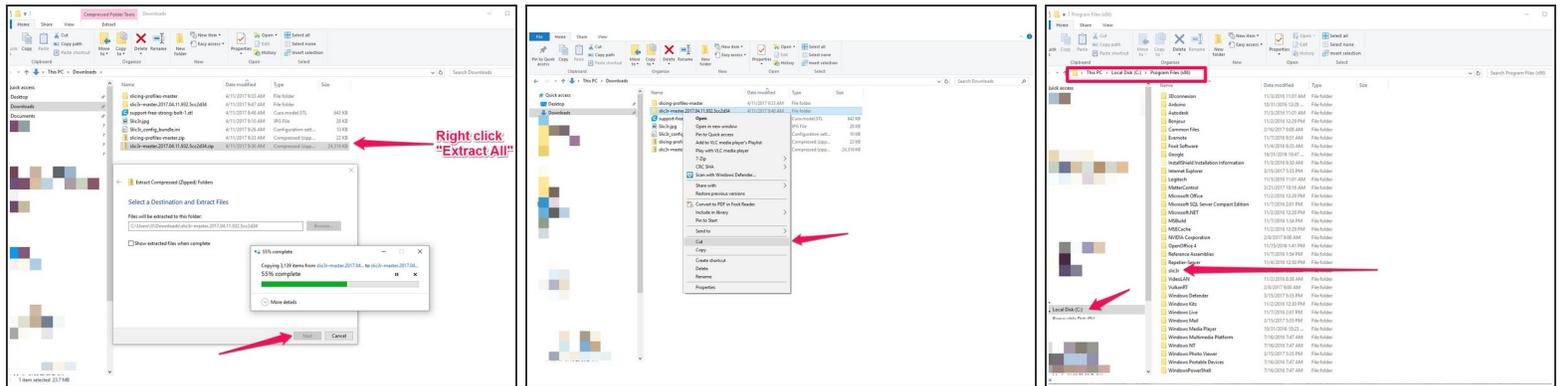
- Visit: [Slic3r](#)
- ① For this guide, version 1.3.0-dev for Windows is being downloaded and used. There is also a Mac OS X (10.7+) and Linux version available.

Step 2 — Download SeeMeCNC Slice Settings



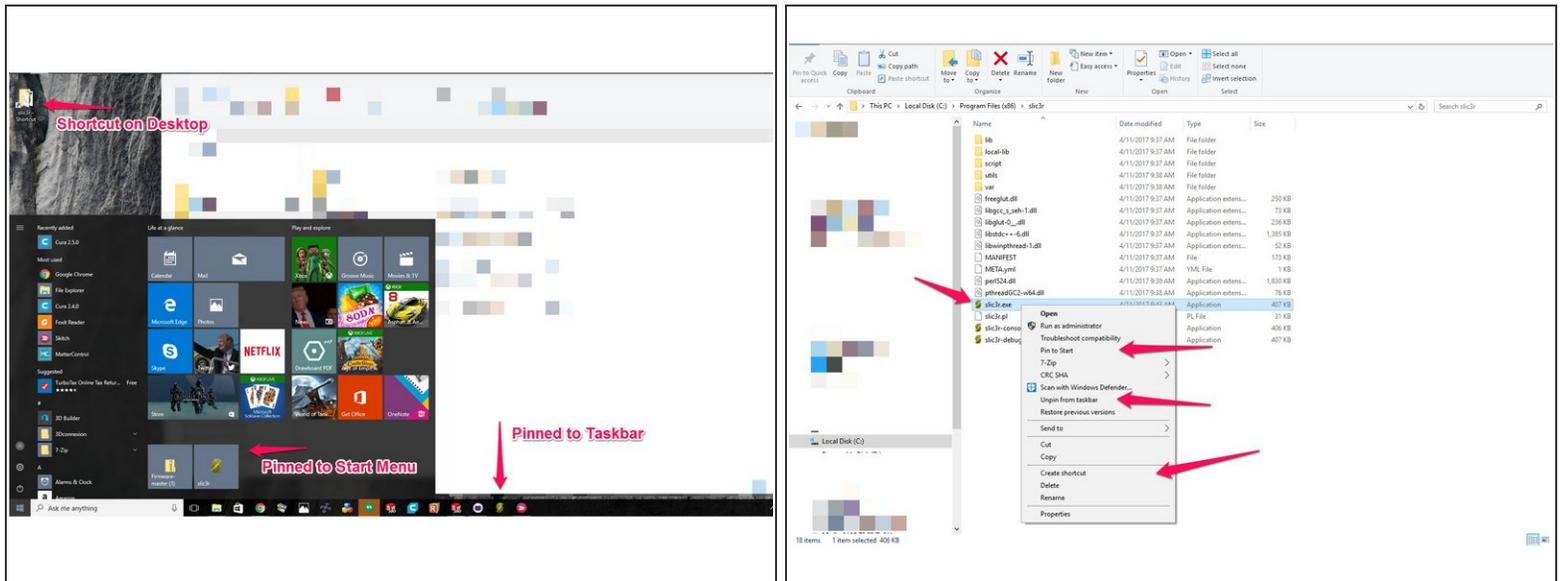
- Download and extract the [SeeMeCNC Slicing Profiles](#)
- Save/move them to a location where you can access them in the future steps / or to return to defaults.
- There are several different pre-configured slicing profiles for you to choose from. They have been broken into categories for hot end type (ERIS & HE280) and slicing engine (MatterControl, Slic3r, & Cura) . This guide will show you how to import the appropriate settings in future steps.

Step 3 — Relocate Slic3r (Windows)



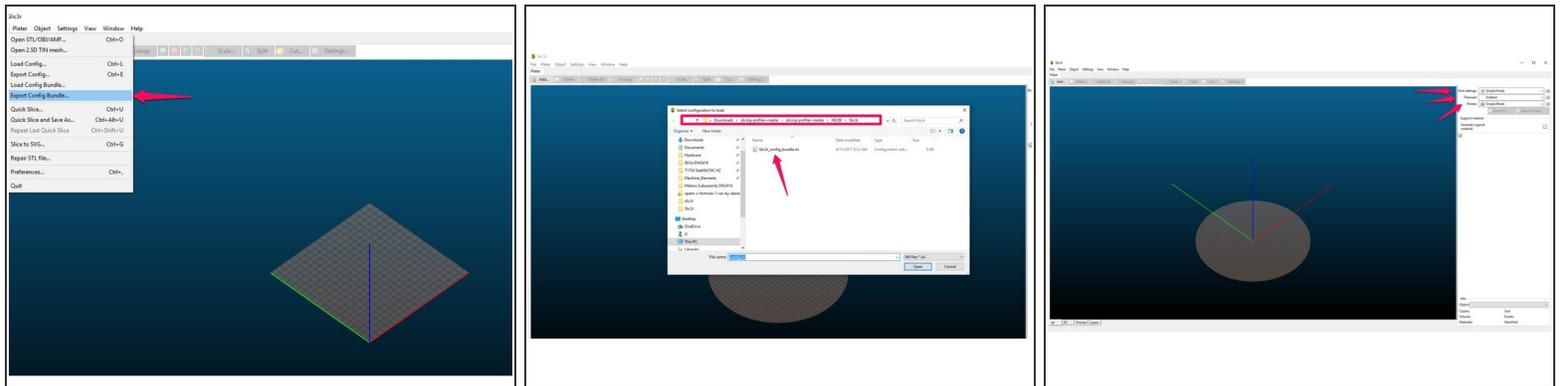
- Slic3r does not "install" on your computer. Rather you run an executable file each time you use Slic3r. For this step we will move the Slic3r files (for use with Windows) to your program files and create a shortcut / pin to taskbar.
- Right click on the "Zipped" Slic3r program folder that you downloaded and choose "Extract All"
- Choose "Next" to extract the files to the specified location.
- Right click on the newly unzipped folder and choose "Cut". We will be moving the folder to the Program Files (x86) folder (or other location that you would prefer)
- Navigate to the Program Files (x86) folder and paste the Slicer program folder. This is shown in the third image. The name of the folder can also be shortened to Slic3r using the rename function in Windows.

Step 4 — Create a Shortcut / Pin to Start or Taskbar



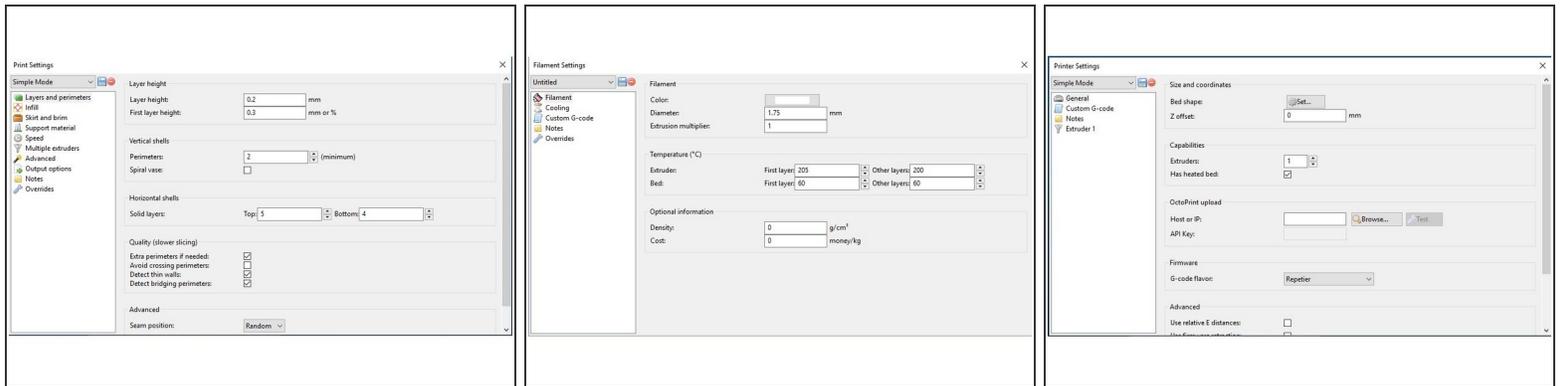
- For this step you can now choose to "Pin to Start", "Pin to taskbar", "Create Shortcut", or any combination of the three. (This is Windows Specific)
- Double click on the Slic3r folder.
- Right click on the file
- You can now choose to "Pin to Start", "Pin to taskbar", "Create Shortcut", or any combination of the three.
- This will make starting Slic3r more easily accessible.

Step 5 — Configuring Slic3r



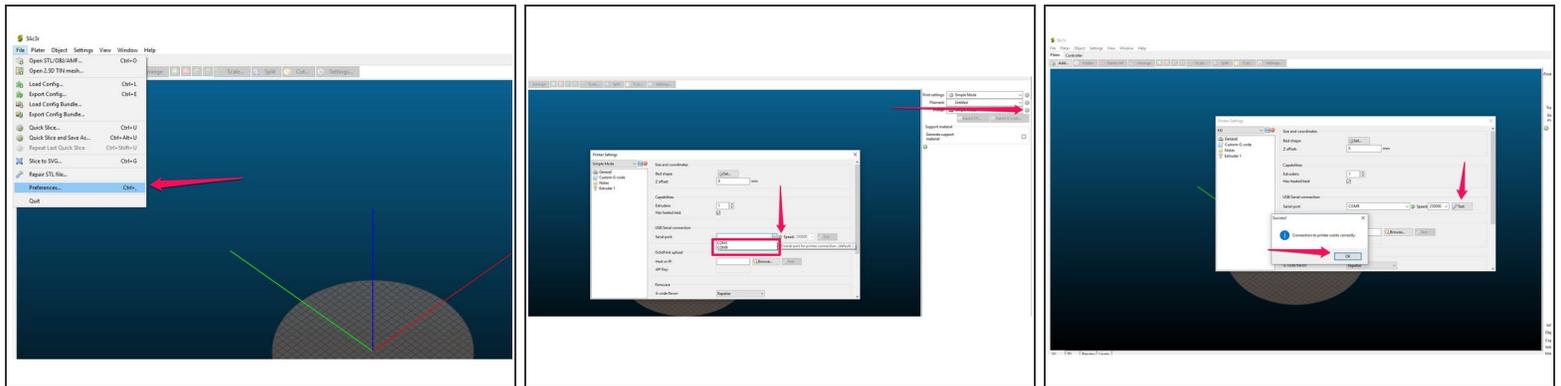
- Open Slic3r
- To import the Slic3r Configuration that you previously downloaded, Click on "Menu" in the upper left hand corner of the program. Choose "Load Config Bundle..."
- This will open a File Explorer dialog. Navigate to the file: Slic3r_config_Bundle.ini which is in the slicing profiles download that you downloaded previously.
- "Open" the file
- You will get a notification "3 presets successfully imported"
- To make the import setting take effect you will now need to close the program and then re-open it
- After closing and re-opening you will find that the Print Settings, Filament, and Printer now have updated profile names. It will also be evident that the bed has changed from rectangular to circular.

Step 6 — Getting to Know Slic3r



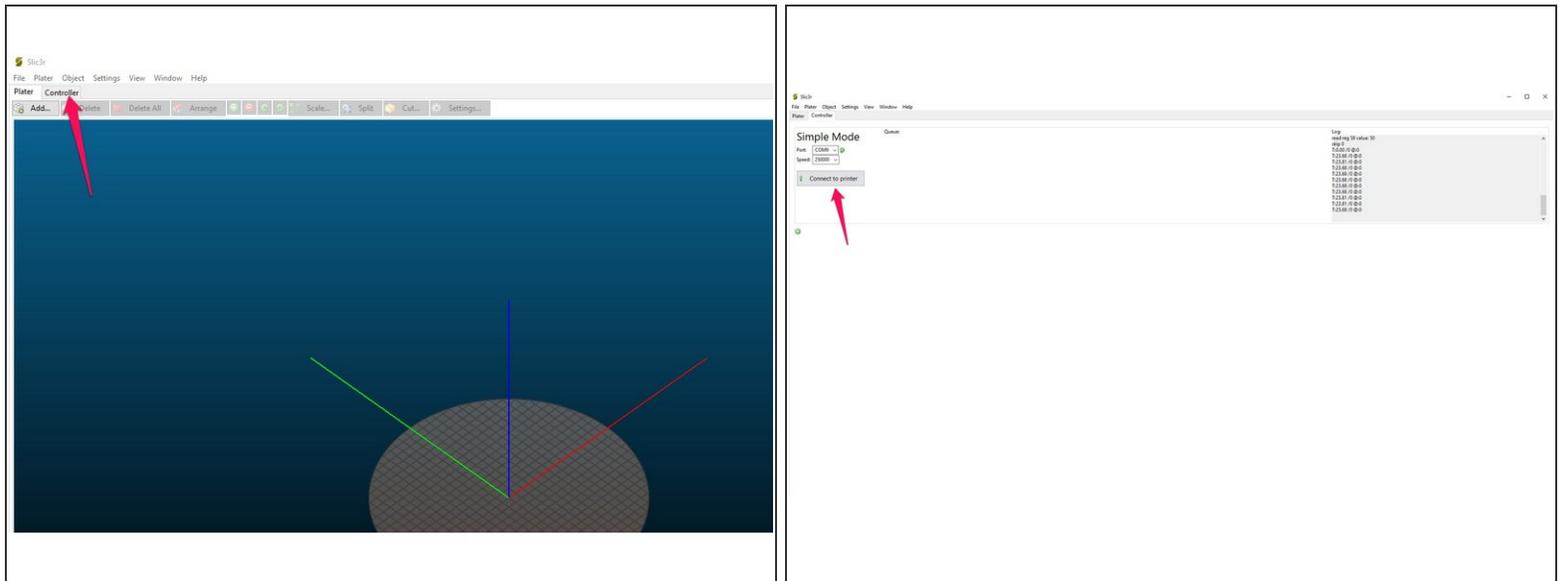
- Slic3r has three main settings divisions: Print Settings, Filament, and Printer
- Print Settings are as the name suggests, specific to things such as layer height, infill, speeds, support, etc.
- Filament Settings are settings that effect temperature & cooling settings
- Printer Settings are settings specific to your printer such as bed shape and size, Heated Bed (yes/no) Start and End GCODE, etc.
- For Printer Settings, be sure to set the active setting to your machine type. ie Rostock Max v3, Orion Delta, etc.

Step 7 — Advanced: Turning on USB Connection



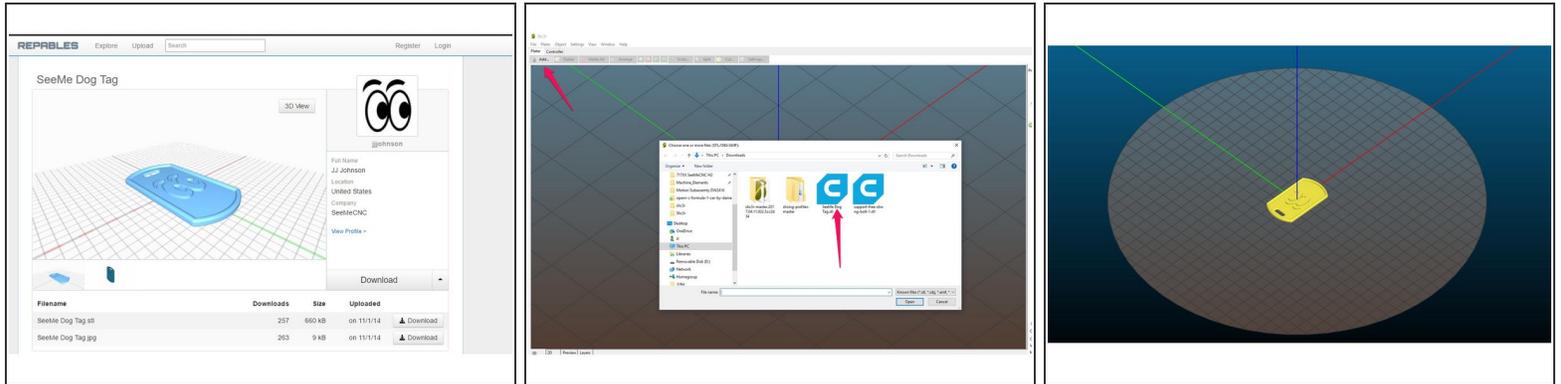
- The USB Connection is turned Off by default. Primarily, the best way to print is by using an SD card. This prevents issues such as USB latency and other issues that stop the flow of data from the computer to the printer.
- It is sometimes helpful to be able to directly control the printer via the computer though. This version of Slic3r has the capability, but it needs to be turned on first.
- In the upper left hand corner of the program choose "File" and then choose "Preferences"
- A dialog will pop up. Uncheck Disable USB / serial connection and then choose OK
- Slic3r will notify you that the program needs to be restarted for this to take effect. Close the program and re-open it.
- When it re-opens you will find that there is a tab labeled "Controller" Before Slic3r will be able to communicate we need to help it "find" your printer. Turn your printer on and make sure that the USB cable is connected to the Printer and Computer. Also make sure that there are no other programs open that will take over communication:
- Click on the gear icon in line with "Printer" (on the right hand side of the screen). This will open the printer setting. With your printer powered on and USB cable connected, choose the correct COM port for your printer, ensure that the Speed is set to 250000 and then click "Test"
- You should see a Success dialog. The computer and printer are now able to communicate with each other. Close the dialog, and then click the blue save icon in the upper left hand corner of the Printer Settings Dialog. Choose OK in the Save As dialog that appears.

Step 8 — Communication / Function Test



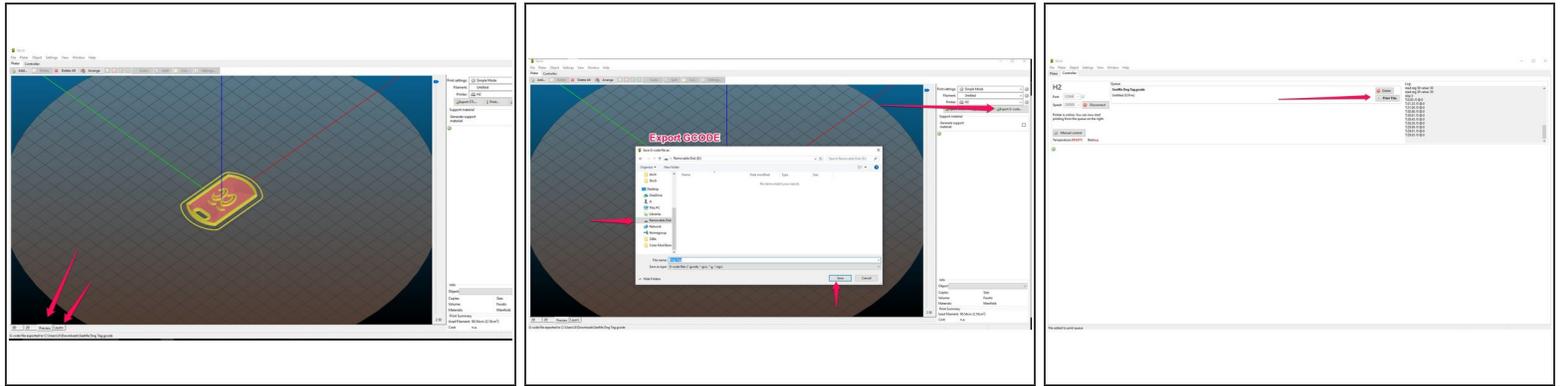
- Click on the Controller tab in the upper left of Slic3r
- Click on Connect Printer to connect
- Your printer should connect automatically. You can now Choose: "Manual Control"
- Click on the XYZ button in the bottom left hand corner of this dialog. This will home the printer and confirm the communication.
- You can test other functionalities of the program such as setting heats, homing, calibration (G29) etc.

Step 9 — Running a First Print



- For the sake of testing this guide will use the [SeeMeCNC Dog Tag](#) for the first print.
- Click on the "Add" button in Slic3r and choose the file to import
- The STL is imported and centered in virtual build area.

Step 10 — Running a First Print Continued



- You can view the slice preview in Slic3r by clicking on the Preview and/or Layers tab near the bottom of the virtual build area. This provides you with the opportunity to view the toolpathing before actually running the print.
- Now it is time to print. There are two main options for sending the GCODE to the printer. 1) Export the GCODE to the SD card (preferred method) or 2) Printing direct from Slic3r. Both are highlighted below
- Export to SD Card: Click on "Export G-code..." near the upper right of Slic3r. This will open a dialogue for navigating to the SD card. Save the GCODE to the SD card. You can then eject the SD card and transfer it to your printer.
- Printing from Slic3r: Click on the "Print..." button near the upper right of Slic3r. This will take you to the "Controller" tab of Slic3r where you can then connect to the printer (if not already connected) and print the part by clicking the "Print This" button.

Step 11 — Print!



- Your print should now be in action!