



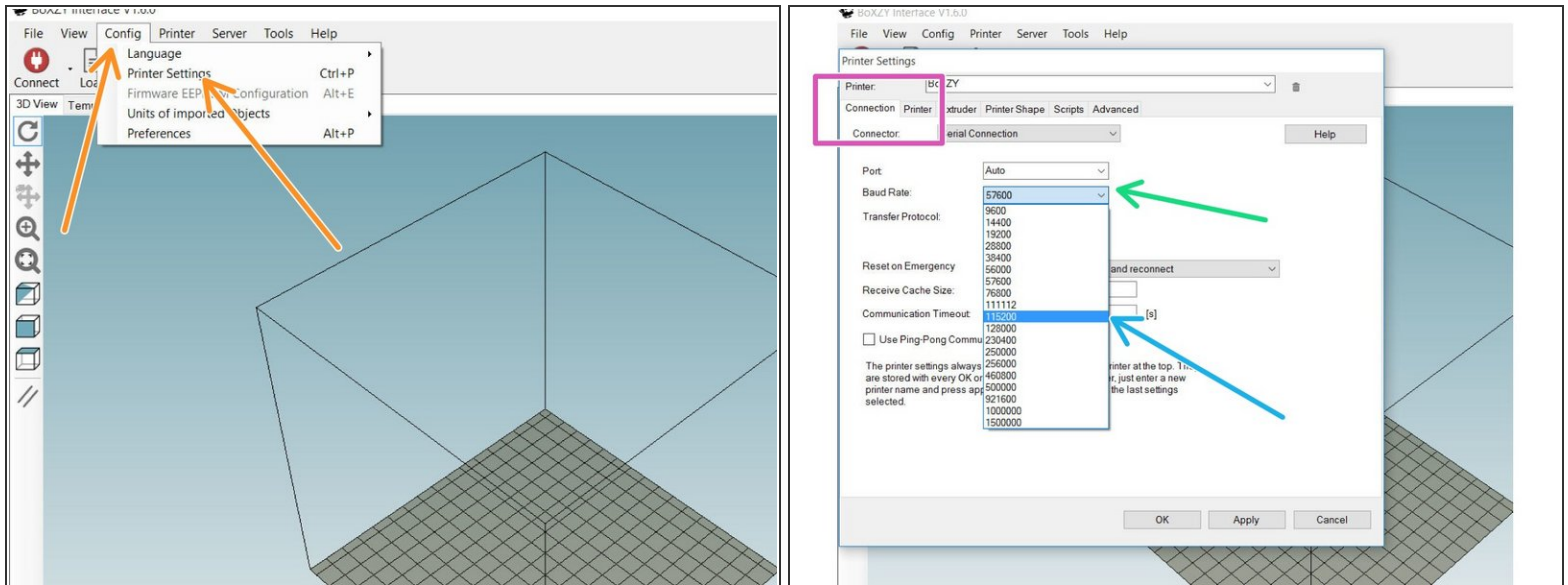
# **Penn State at Behrend (for computers without administrative access)**

In this tutorial we've included settings files for those of you who cannot run BoXZY as administrator. Import these settings when you run BoXZY. You will only have to do this once on accounts without administrative access.

Written By: Joel Johnson



## Step 1 — First things, first! Check your Baud Rate!



- Go to the "Config" menu and select the option "Printer Settings"
- Make sure you're in the "Connection" tab
- Look at "Baud Rate" and if it doesn't say 115200--then, it's wrong.
- If it's wrong, click the Baud Rate drop down menu and select 115200
- Click "OK" to save.

## Step 2 — Short clip showing what you're about to read



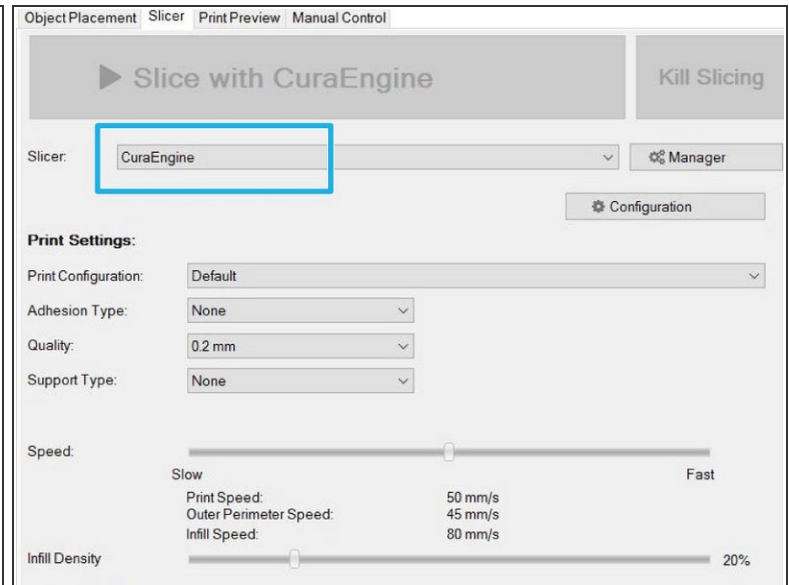
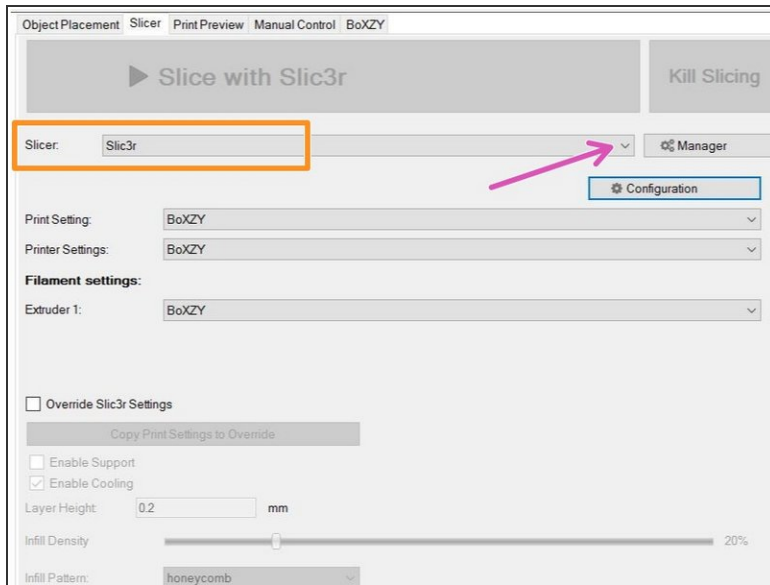
- Watch it before you read about it! It'll help you remember.
- ① After you start the video, you have option of watching the video full screen. Simply Right click the video and select "Open video in new tab". There's a little icon at the bottom right that allows you to expand the screen once you've opened the video in the new tab


## Step 3 — Download BoXZY's settings!!!



- [Download BoXZY's settings!!!](#)
- Unzip the folder and remember where you saved it.
- ① You will import 3 settings files. You'll import just one for Slic3r and you'll import 2 from CuraEngine. Cura engine stores its settings its print settings and its filament settings separately.


## Step 4 — Select your slicer !




 If you save your settings as you import these settings files, you'll only need to do this once, so be sure to save!

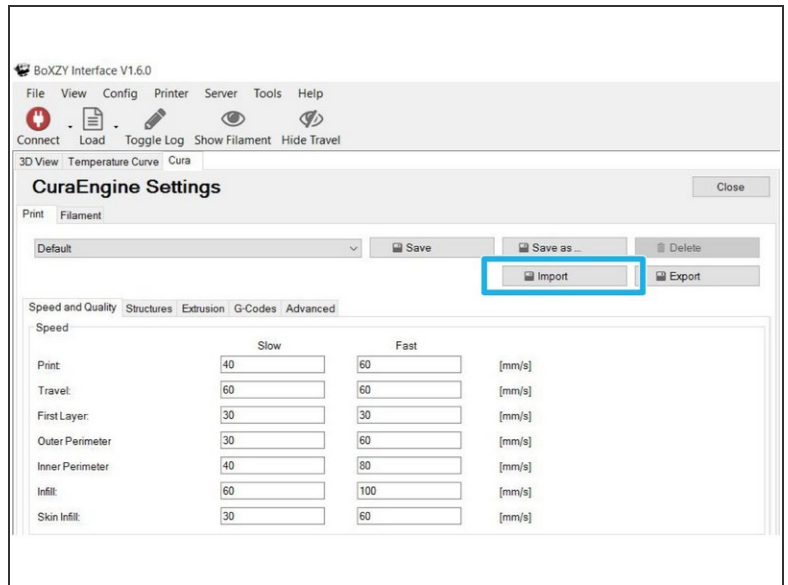
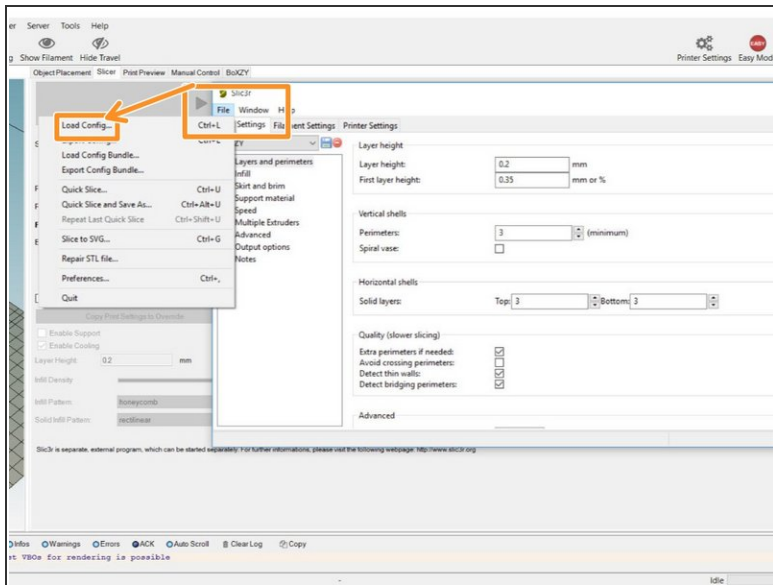
● Choose your slicer. You have two choices in the drop down menu:

- Slic3r
- CuraEngine

 You only change one settings file at a time. These instructions will take you through the process as if you were importing the Slic3r settings on one computer and the CuraEngine on another simultaneously. This is so you can follow along if you choose to either CuraEngine or Slic3r. You can follow the steps for both to have them both.

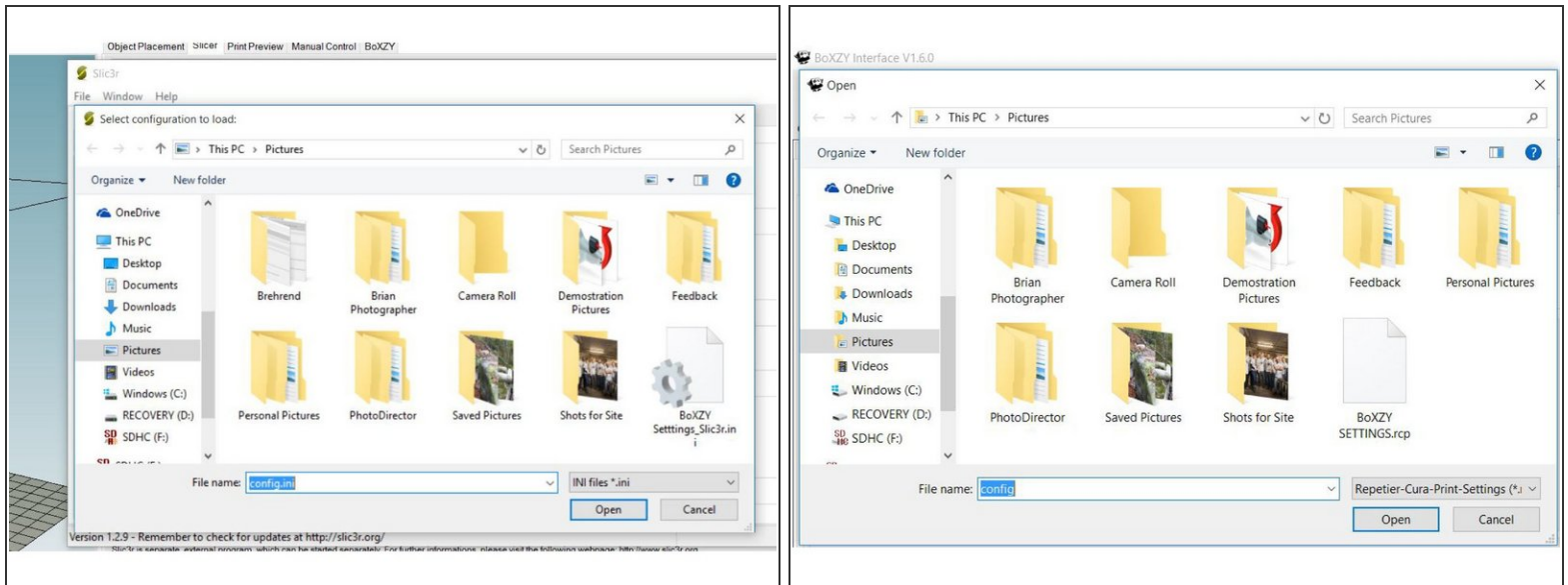
 You'll learn to import the Slic3r settings and the CuraEngine print settings first, and then after that you'll import the Filament settings for CuraEngine

## Step 5 — Locate your "Import" option



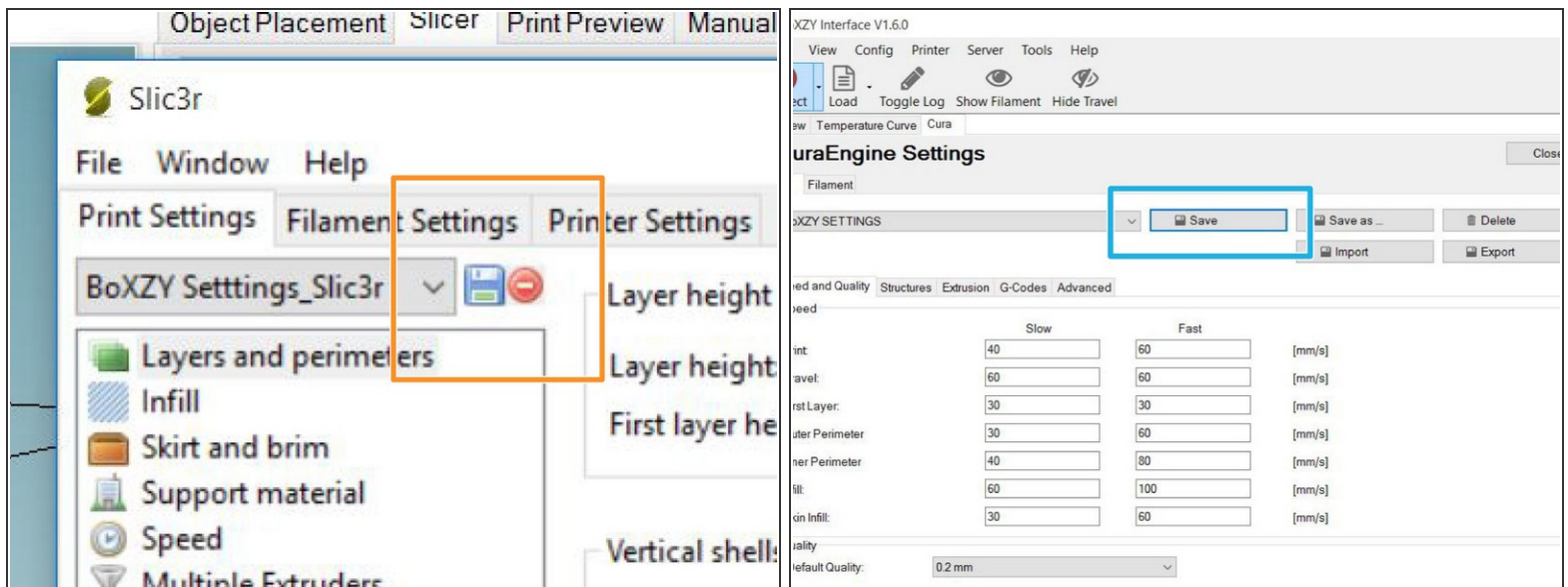
- Find the settings import option
  - For slic3r, go to "File" in the top right hand of the new window.
  - For Cura, you will immediately see an "Import" button. This is within the "Print" tab

## Step 6 — Click Import and select your BoXZY settings file.



- Locate your file, select, and open your settings file
  - For Slic3r, it's an .ini file extension, labeled BoXZY\_Slic3r.ini
  - For Cura, it's an .rcp file extension, labeled BoXZY\_CuraPrint.rcp
- ① Make sure you've unzipped the file first!

## Step 7 — Save your settings!



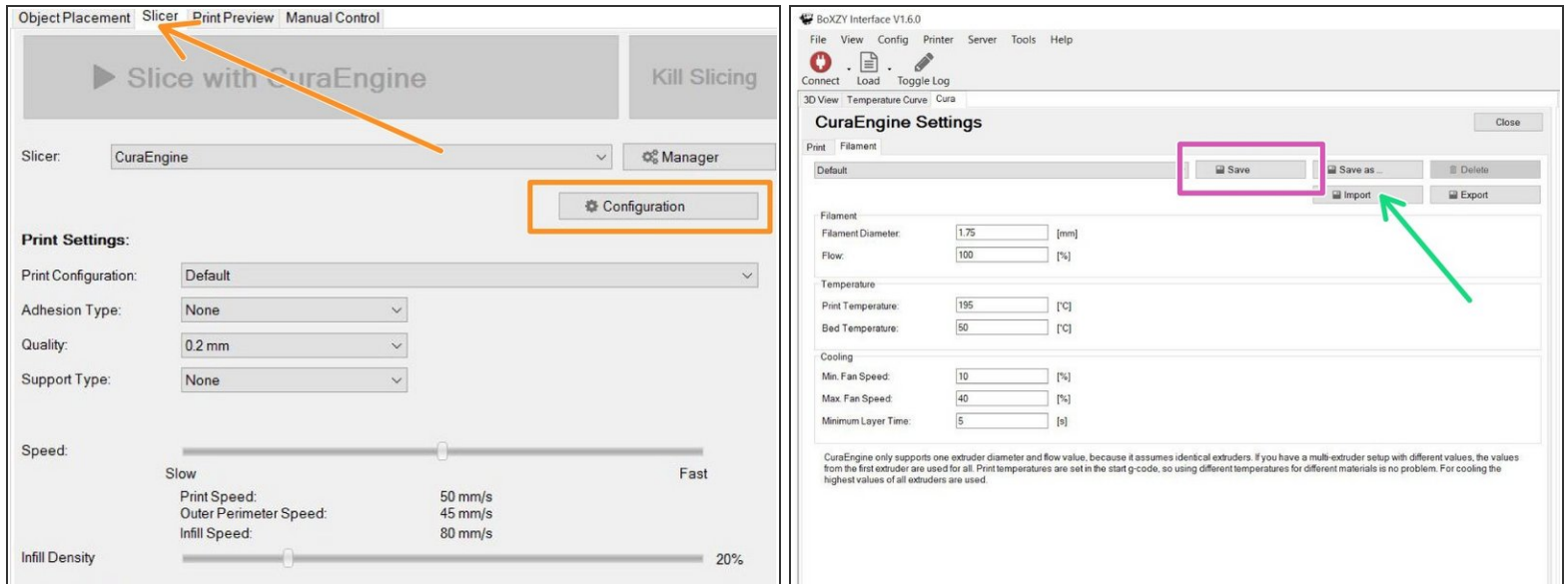
- Save for Slic3r or CuraEngine
  - Click save the save icon for Slic3r
  - Click the "Save" button for CuraEngine

⚠ Make sure you save!!!

⚠ Don't click the round, red circle with the white line in it that's right next to the save button. That's the delete button...you should leave that alone. Human factors wasn't that button-designer's strong suit.

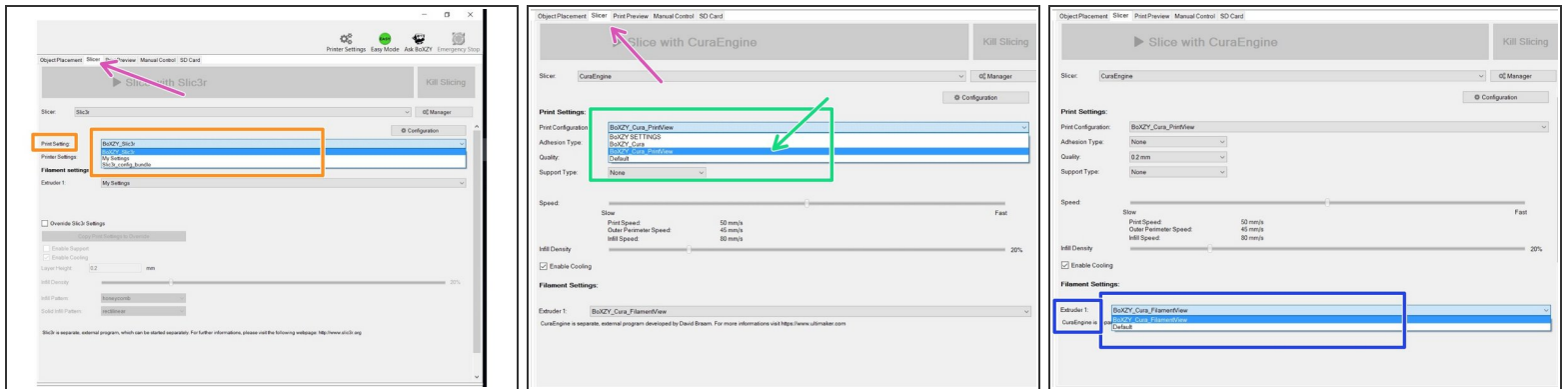


## Step 8 — Import CuraEngine Filament settings



- You'll repeat the same steps for importing settings you followed to import the CuraEngine print settings. Only, now, you'll import from the "Filament" tab to the right of the "Print" tab. This is within the CuraEngine settings after you click the "Configuration" button in the main slicer window
- Click "Import"
- Select BoXZY\_Cura\_FilamentView.rcf from where you saved the file you downloaded with BoXZY's settings.
- Make sure you click "Save" after you've imported the filament settings file.

## Step 9 — Run your imported settings and then print!



- Return to the "Slicer" tab.
  - ✦ This is in the main Slicer tab where you started
- You'll need to check that these settings are what you are actually running when you print.
- If you're using Slic3r for this print, then you'll only need to check one drop down menu called "Print Settings". You should see "BoXZY\_Slic3r" selected. If not, you should open the drop down menu and select it.
- If you're using CuraEngine, then you will need to check two drop down menus. The first is at the top. The first menu is called "Print Configuration:". "BoXZY\_Cura\_PrintView" should be selected, if it's not you should select it.
- Now, you just need to check the "Filament Settings". It's at the bottom. You'll see another drop down menu called "Extruder 1:". You should see "BoXZY\_Cura\_FilamentView" selected. If it's not, you should select it.