Fusion3 F410 Basic Printing

This tutorial will instruct users in the basics of setting up and running a basic PLA print on the Fusion3 F410 printer.

Written By: Matt P. Lamparter

This document was generated on 2024-01-26 02:25:05 PM (MST).

INTRODUCTION

The Fusion3 F410 is a large format 3D printer. As with the other printers in the Maker-E, it prints primarily PLA. While it is capable of printing more exotic filaments such as TPU (flexible), nylon, and carbon fiber infused filaments you may not use them without first talking to a qualified Maker-E technician.

The print volume of the F410 is 14" x 14" x 12".

Step 1 — Before Printing

⚠ The 3D printers are not for printing boxes or any high-cost material print that can be easily achieved by other methods, such as laser cutting. If you believe your print is an exception, please email Matt Lamparter, ECE Lab Director and manager of the Maker-E, with a copy of your design and a justification via mlampart@bucknell.edu.

- For box creation using laser cutting, users can use this tool to generate SVG files: https://festi.info/boxes.py/

Step 2 — Turn on the printer
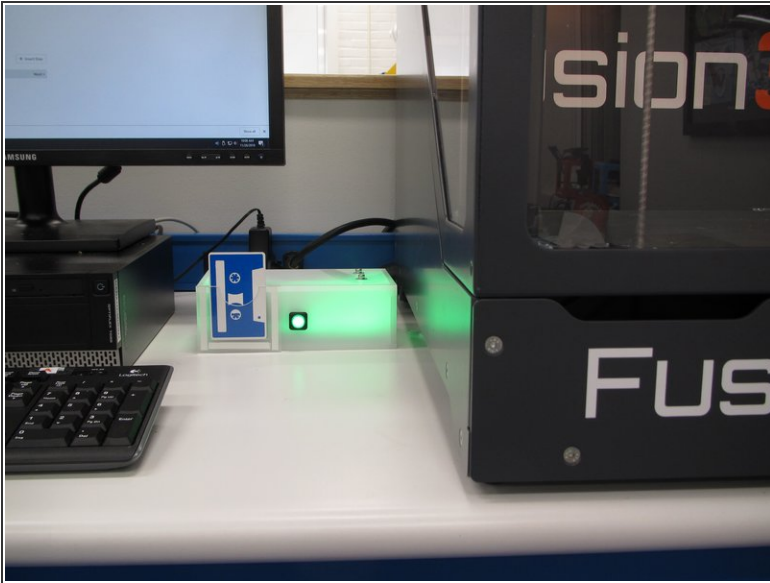
- Begin by placing your own card, or a training card, in the white Portal box beside the 3D printer. The box should illuminate green and the printer should turn on.

- ⚠ If you are not authorized to use the printer do not continue and first get trained. Unauthorized use of 3D printers will result is suspension from the MakerE.

- To confirm that the printer is on, make sure that the integrated LCD shows the home screen.

This document was generated on 2024-01-26 02:25:05 PM (MST).

Step 3 — Simplify3D software
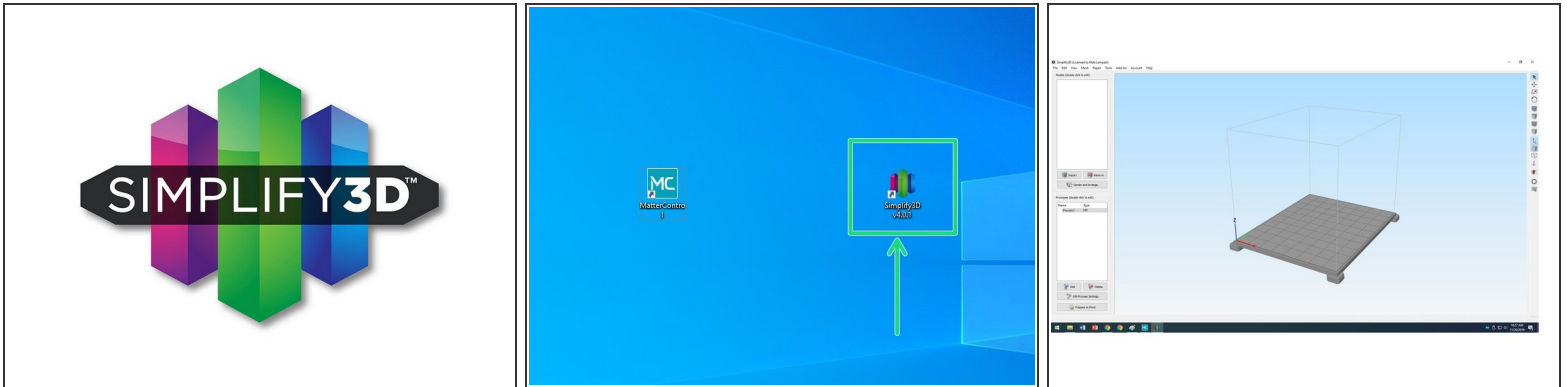
- The F410 printer uses Simplify3D, the same software utilized by the MakerGear M2 printer. If you have used the M2 printer previously, the steps for importing and setting up your object are identical. If you are new to Simplify3D, please read on for details.

- Once the printer powers up, continue by logging into the computer using the following credentials:
  - Username: .\MakerGear (note the .\ are required to properly log in)

  - Password: BucknellMakers2019

- Make sure to log in using the above credentials. DO NOT log in using your username and password. Since you will not be able to access any files in your private Netspace your file(s) should be accessible via public Netspace, email, physical media, etc.

- Launch the Simplify3D application by double clicking the shortcut on the desktop.

- After Simplify3D opens you should see the screen like the one shown here. If you encounter a login screen, please stop and see a technician.

Step 4 — Import Your File
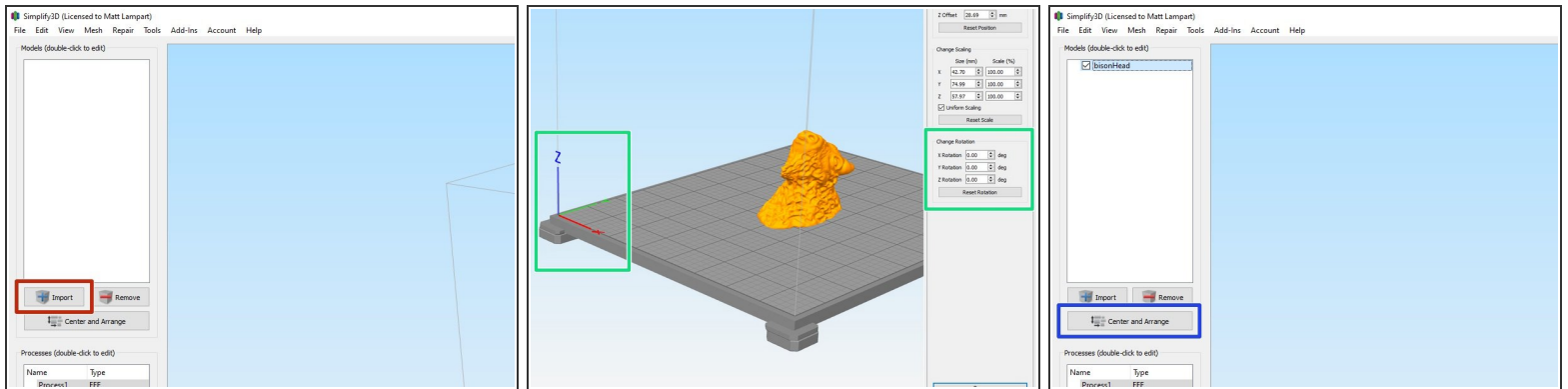
- If you are using the printer for the first time to pass your certification then you may download the bisonHead.stl file attached to the bottom of this guide. This file came from [Thingiverse](#), which is a great resource for free 3D object files.

- Click **Import** and navigate to where your STL file is saved. After you have imported your file, experiment with using the mouse to navigate the workspace. Zooming, rotation, and translation are all possible with the scroll wheel, left button, and right button, respectively.

- The orientation of your object may affect how much support material is needed. More on this later. The basic idea is to rotate your part so that as much flat surface area sits on the print bed as possible.

- You can change the rotation by double clicking the object then adjusting the rotation settings as needed. The bisonHead.stl object has a flat bottom which is already resting on the print bed. Thus, it does not require any rotation.

- If you rotated your model, click **Center and Arrange** to reposition your model in the center of the print bed and drop it flat on the surface.

This document was generated on 2024-01-26 02:25:05 PM (MST).

Step 5 — Defining support material
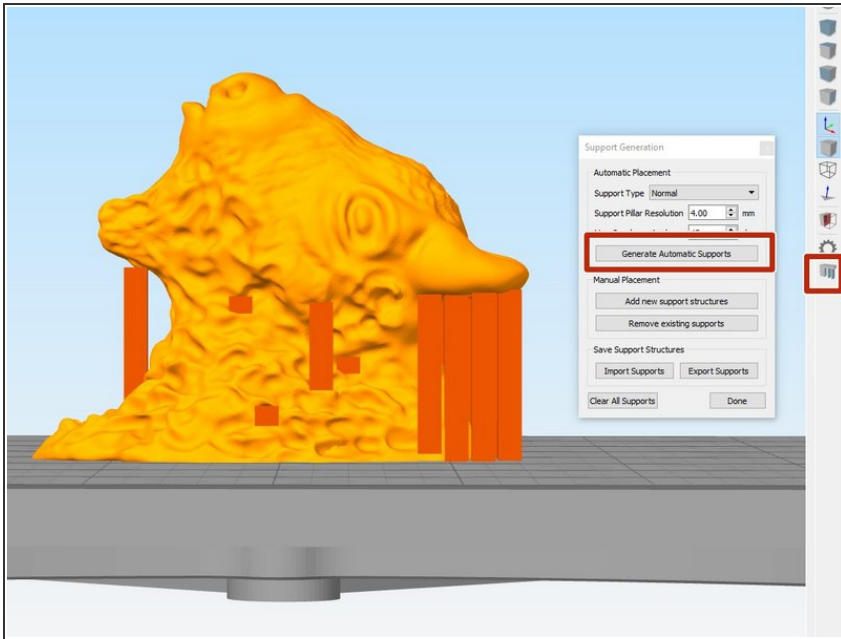
- Notice how the bison's horns and chin stick out from the head, with nothing underneath them? If the printer tried to print these as shown it would be extruding hot plastic in midair - which would promptly fall! We call this overhang.

- The solution is to define "support material" underneath these overhangs which will give the printer something to rest the overhangs on while they are being printed. Support material is generated in such a way as to provide good support while printing, but can be easily removed once the print finishes. Support material is temporary.

- Begin by clicking the stonehenge-looking icon on the right to bring up the **Support Generation** window. Click the **Generate Automatic Supports** button to have Simplify3D add supports where it deems necessary.

Step 6 — Customizing support material
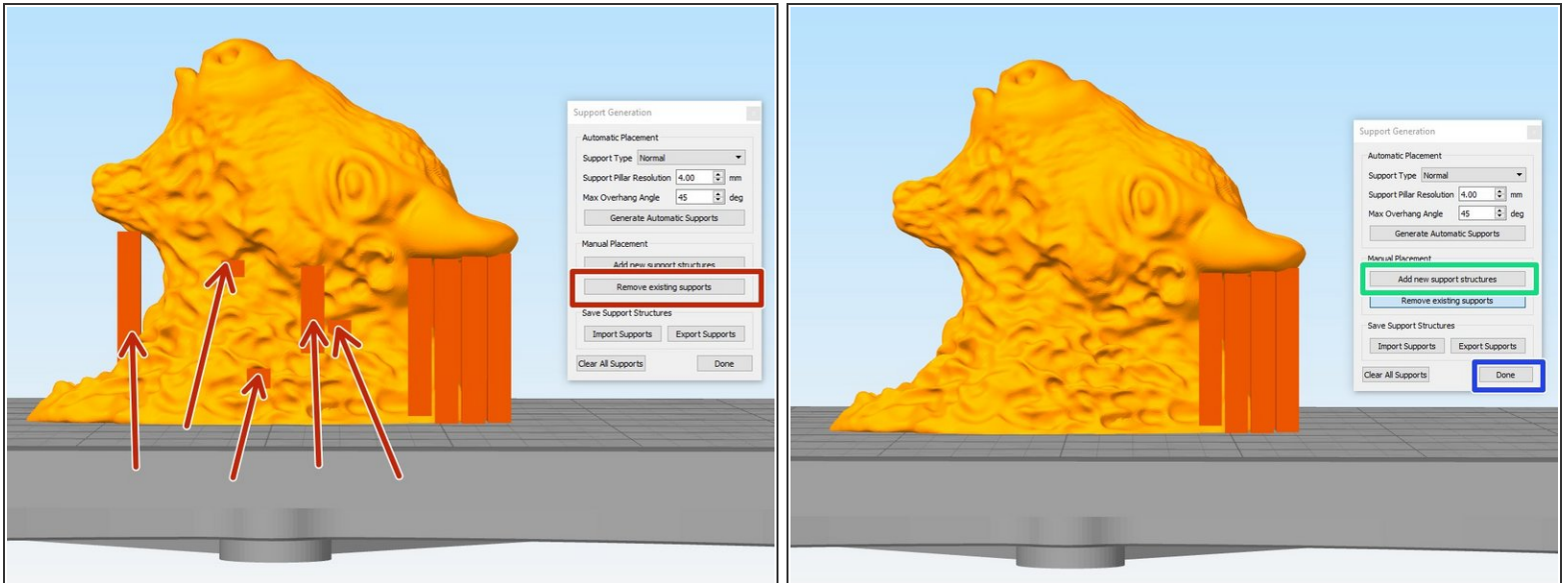
- Simplify3D does a pretty good job at placing support material, but it is important to examine the automatic placement and think about how the resulting model will print. In this example, there are several areas where support is not needed and will only make for difficult removal. These are called out in red.

- Click on the **Remove existing supports** button and then left click on all of the supports you wish to remove. Remember to check both sides of the model!

- If you wish, you can also add additional support structures in any place you feel Simplify3D neglected to place them. Click on the **Add new support structures** button and then left click where you would like to add support. You will see new, dark orange pillars appear where you click.

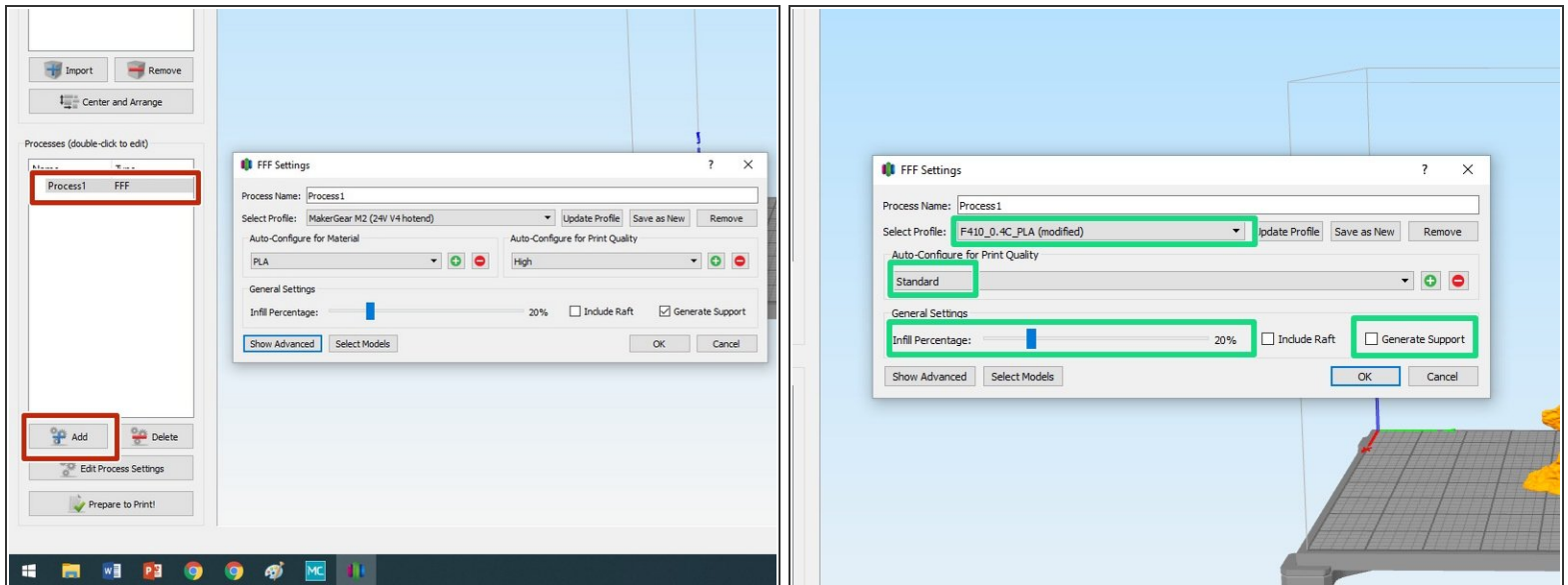- Click **Done** to exit support definition when you are satisfied.

Step 7
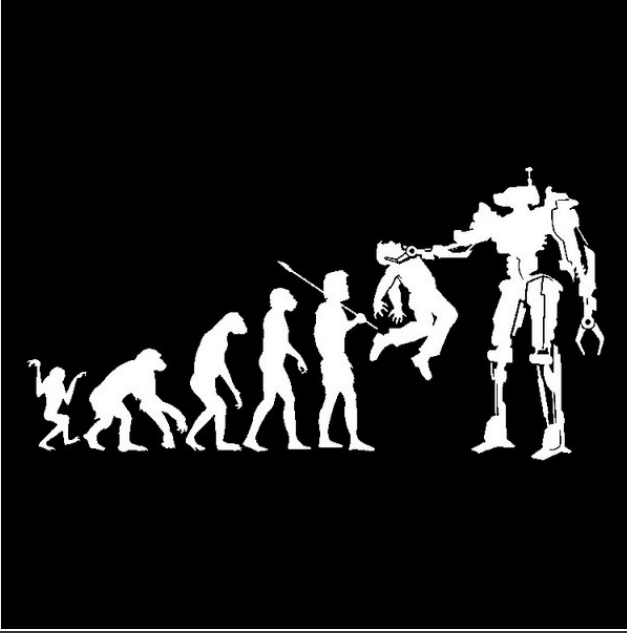
⚠ This picture is only here to get your attention!

⚠ **The information in Steps 7 and 8 is very important to avoid damaging the printer so read it carefully and ask a technician if you have any questions!**

This document was generated on 2024-01-26 02:25:05 PM (MST).

© 2024

bucknellmakers.dozuki.com/

Page 9 of 20

Step 8 — Defining the Print Settings

- ● Double click **Process1** in the Processes window to bring up the Print Settings dialog. If Process1 does not appear, click Add to generate a new process and open the dialog window.

- ● Adjust settings as shown:
  - ● Select the **F410_0.4C_PLA** profile. It is OK if "modified" or a number are appended to the end of that profile name.

  - ● Choose to print **Standard** quality. The trade off between Low and High quality comes in the form of quality vs. print time. Generaly, Medium results in good prints at reasonable speeds.

  - ● Set **Infill** to 20%. This parameter determines whether your part is completely hollow, solid, or somewhere in between. Increasing infill will add rigidity to your part if it must withstand heavy loads. However, this will also increase the print time for your part.

- ⚠ For the F410 printer, Infill MUST be set to a value between 15% and 85%. Values outside of this range will cause problems with the printer and cause your print to fail. Treat 85% as equivalent to 100% solid infill.

- ● Click **OK** when you are done.

This document was generated on 2024-01-26 02:25:05 PM (MST).

Step 9 — Generate a tool path
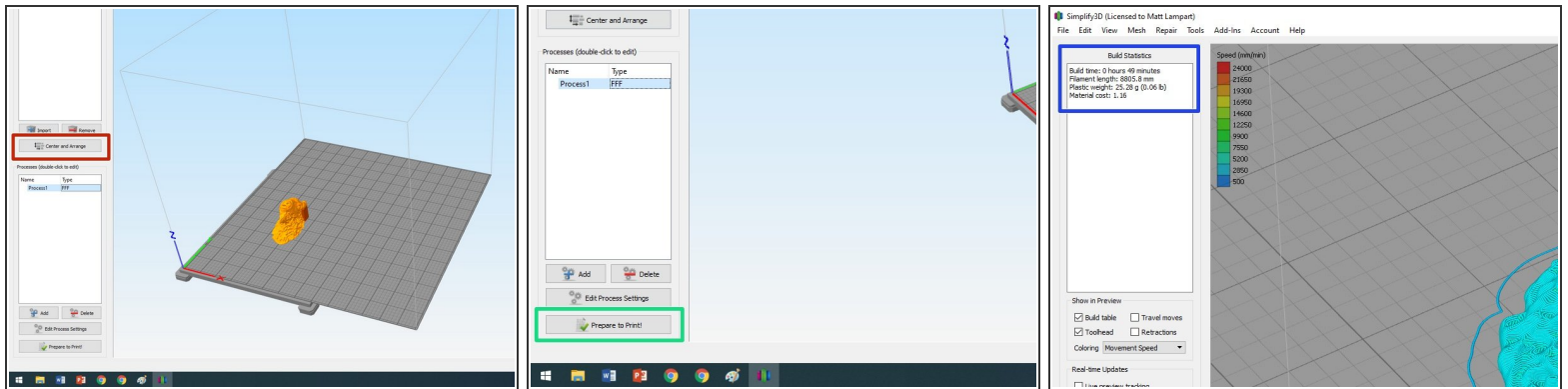
- ⓘ After clicking OK in the previous step you may have seen the print bed on the screen grow larger, depending on what print profile was last used. This is normal behavior.

- ⚠ Be sure that the print bed shown on the screen at this point is 14 squares wide by 14 squares deep! If the bed is smaller than that you have the wrong print profile selected and your print will fail and the printer may be damaged!

- ⚠ **If the printer breaks due to using the wrong profile, you will lose authorization to use the 3D printer and have to retake the certification.**

- Ensure that your part is still centered if the print bed enlarged. Click the **Center and Arrange** button again. Note that in this image the print bed has enlarged to the 14" x 14" bed size.

- Now click the **Prepare to Print** button. Simplify3D will show you a preview window detailing the toolpaths used to construct your object.

- The top left of the screen shows useful build statistics such as build time, filament usage, and cost of material.

Step 10 — Move SD card from printer to computer
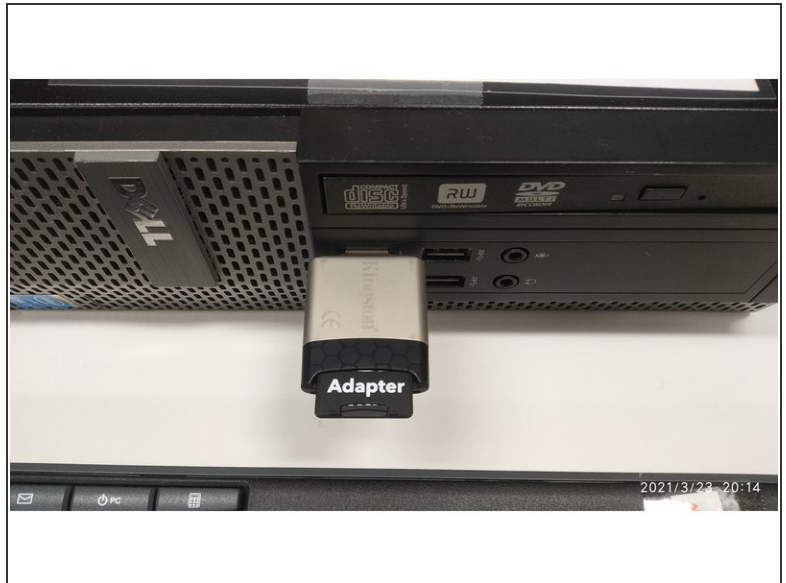
- Now you need to transfer the information to the printer. Remove the SD card from below the LCD interface on the front of the printer.

- Locate the SD/USB adapter which should be near the computer. Insert the SD card into the adapter.

- Now insert the SD/USB adapter into any of the USB ports on the front of the computer.

- ⓘ Recently we moved away from USB SD card holders to a laptop that has a built-in SD card reader on the front face near the right SHIFT key.

Step 11 — Save toolpath to SD card
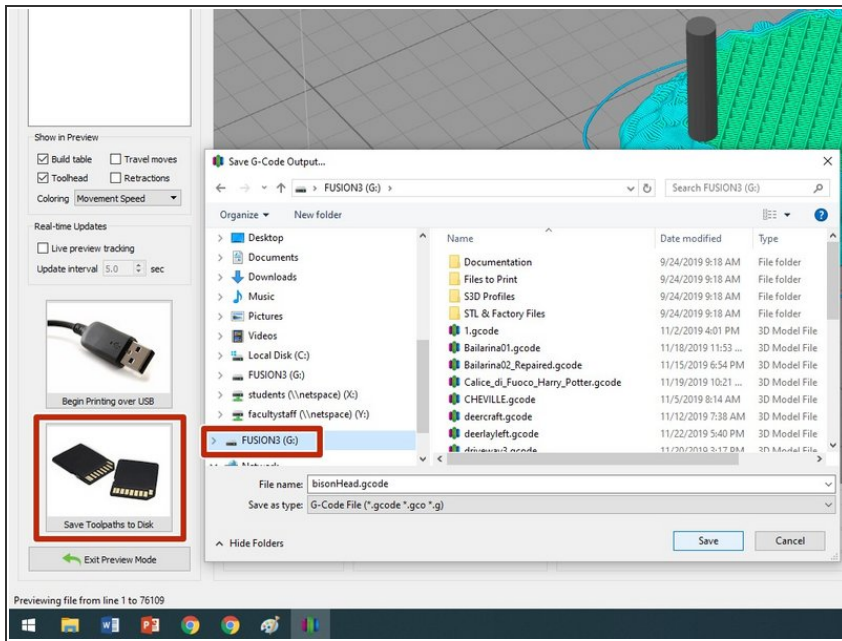
● Click **Save Toolpaths to Disk** and save your file to the SD card drive.

Step 12 — Move SD card back to printer

● Open a Windows Explorer window and locate the SD card drive. Right click on this drive and choose **Eject**. Wait until the drive disappears from Windows Explorer.

⚠ **Be sure to eject the SD card properly from Windows.** Failure to follow this step can cause the card to become corrupted which will prevent you from printing.

● Remove the SD card from the computer and insert it back into the printer's interface. Note that the orientation of the card should match the diagram on the front of the LCD interface.

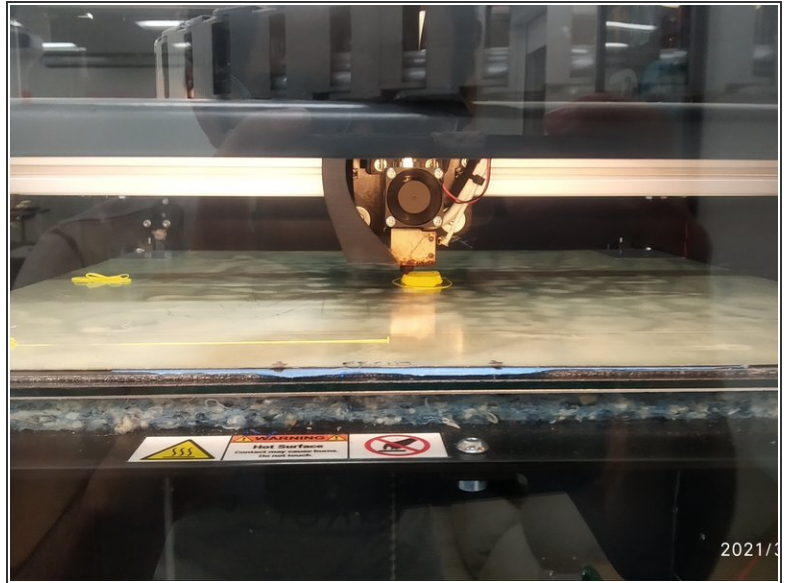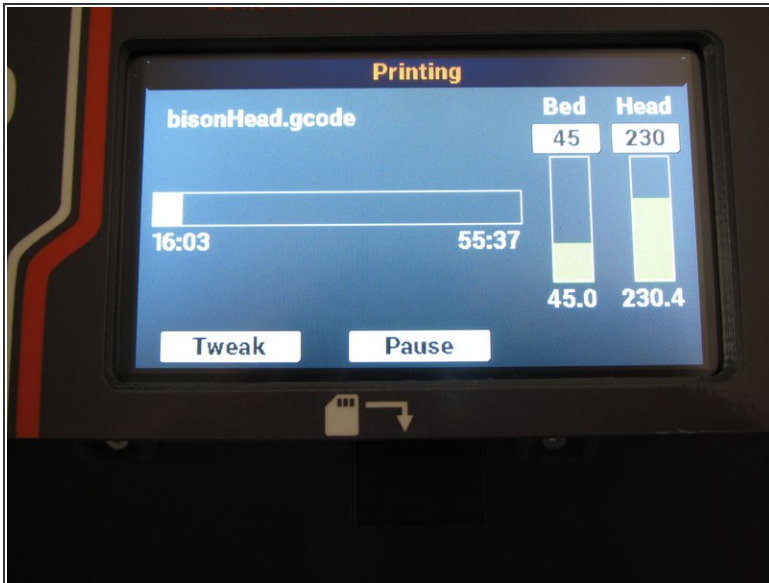## Step 13 — Make sure the print surface corners are not "lifted"
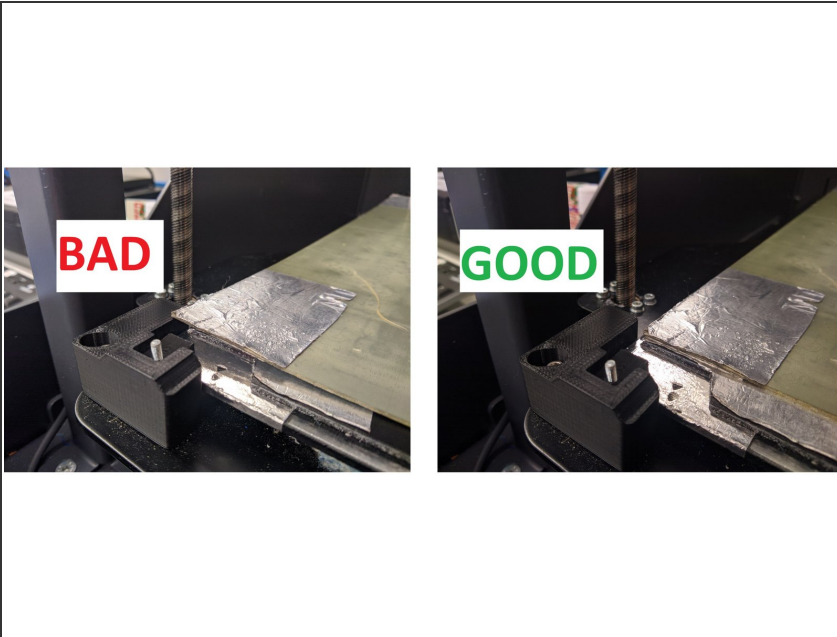
- Check if one of the corners are resting flat on the bed, because it is possible to have the surface slightly askew when attaching the magnetic print surface back onto the Fusion3 bed.

- If the corner is lifted, it will throw off the auto bed leveling at the start of a print which prevents the filament from adhering to the bed properly and can cause prints to fail and other bad things to happen to the printer.

- If it is not correct, lift the top sheet off the bed and straighten it. It may require a bit of force. If you are unsure or have any difficulties, ask for help from a tech.

Step 14 — Start your print
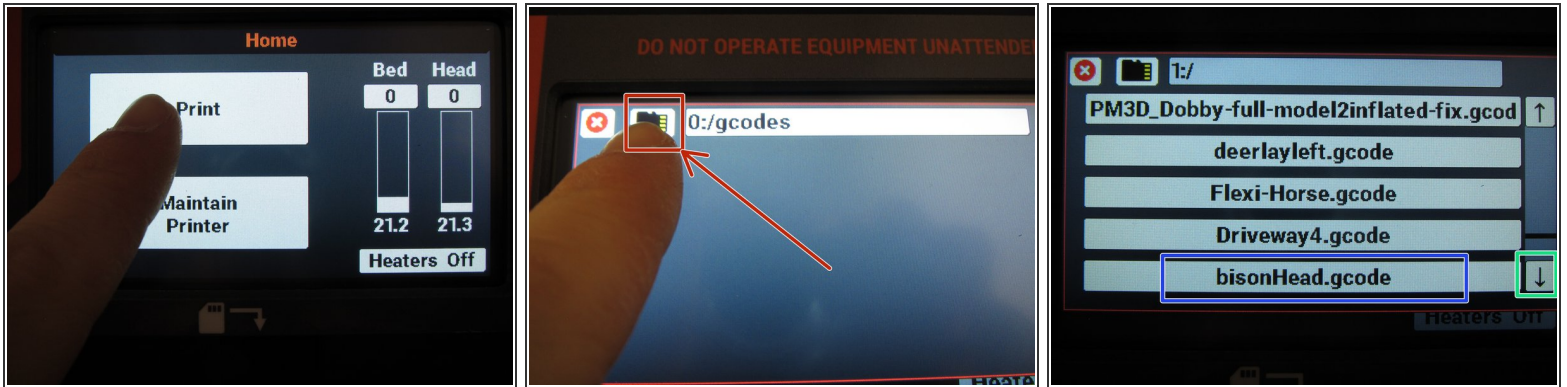
- All of the bullet points on this step reference the printer's touch LCD interface.

- Press **Print**, then press **Select File**.

- The next screen will show you a small SD card icon and the line **0:/gcodes**. Press the **SD card icon** to change to the path **1:/gcodes**.

- You will now be presented with a list of gcode files stored on the SD card. Use the **down arrow** key on the screen to navigate to the bottom of the list.

- Find your gcode file and press it.

- A File Information screen will appear. Press **Print** on this screen. The nozzle and bed will begin to heat up.

This document was generated on 2024-01-26 02:25:05 PM (MST).

© 2024   bucknellmakers.dozuki.com/   Page 15 of 20

Step 15 — Print initialization

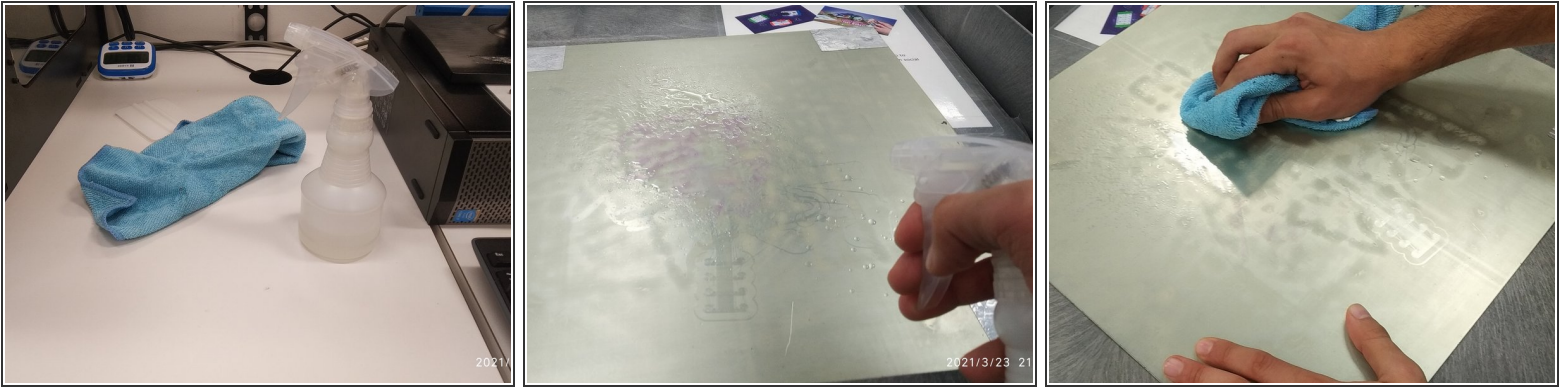⚠ Do not leave the printer until the print starts!

- After the bed and nozzle are at the correct temperature the printer will run through several initialization steps, including extruding, leveling the bed by touching each of the four corners, and wiping the nozzle on the bed.

- Once your print starts the LCD will display elapsed time (on the left) and time remaining (on the right). Note that the time remaining value will initially be significantly longer than what Simplify3D estimated. After several minutes this value will update to the correct time.

- After you are sure the 3D printer is extruding properly (the filament is sticking to the bed and not making a 'birds nest') you can leave the printer. Check back regularly so you don't waste filament if the print fails.
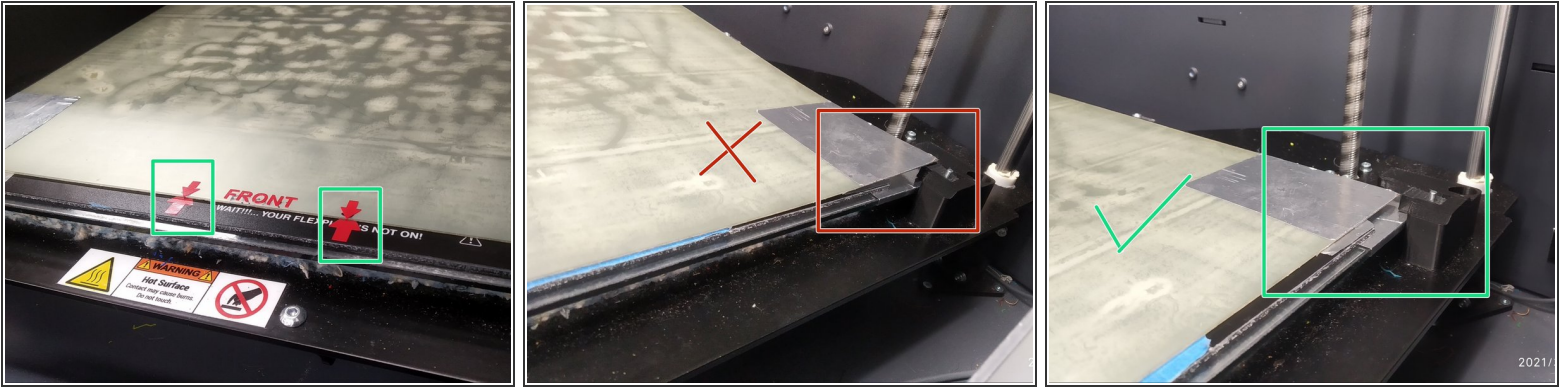
Step 16 — Part removal
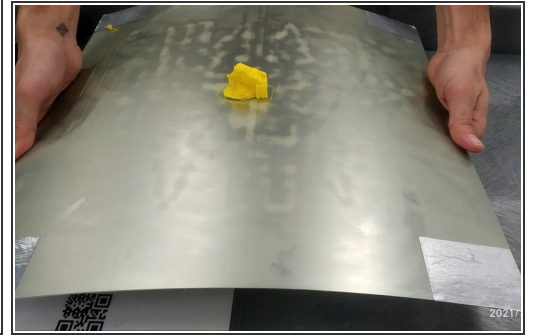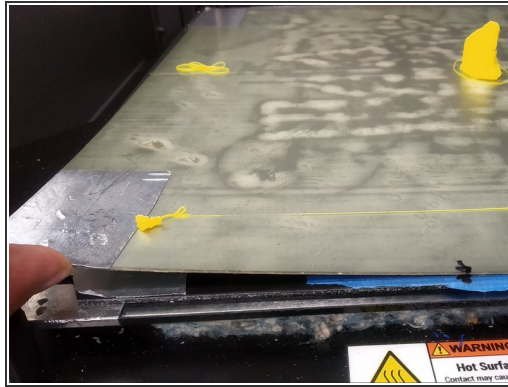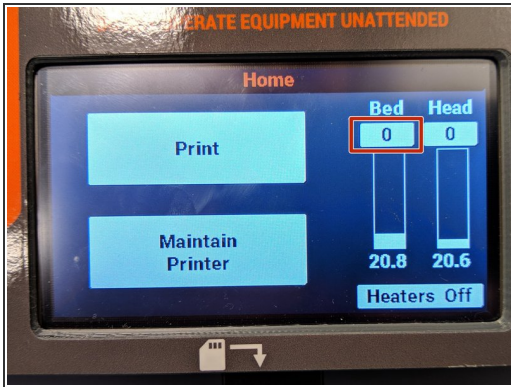
- When the print finishes the print bed will move to the bottom of the printer and the print head will move back out of the way.

- Lift up the thing metal bed from one of its edges to remove it (note that you will need to pull it up with a little force because it is magnetically held in place).

- If your part is small, you can probably remove it directly by holding it. Otherwise, flex the bed to remove your print.

- ⓘ Note that we do not use metal spatulas, found in other makerspaces, to remove parts. These tools can easily mar the print surface and cause problems with future prints. Bending and flexing the print bed should be sufficient to lift a few edges of your part. Then you can **gently** use a flat blade screw driver to pop your off of the bed.

- ⚠ Do not hack at or scrape at the bed with any tools! Doing so can damage the print bed. Damaging the printer will cause you to lose your certification to use 3D printers.

This document was generated on 2024-01-26 02:25:05 PM (MST).

© 2024   bucknellmakers.dozuki.com/   Page 17 of 20

Step 17 — Cleaning the print surface

- After you have removed your part, be sure to remove any remaining filament. Often a "skirt" or ring will be printed around the perimeter of your part. This skirt and all remaining filament should be removed from the print bed.

- ⚠ DO NOT clean it while attached to the printer as this can cause the bed to go out of alignment.

- You should also clean off the glue remaining on the bed. Use the spray bottle filled with water to apply a liberal amount of water to metal bed. If the correct Avery glue was used the glue should appear purple, making it easy to identify the areas to clean.

- Use the microfiber cloth to remove glue from the bed. Use as much water as necessary to completely remove all of the glue.

- You should leave the print bed looking pristine for the next user. Be a considerate Maker! Remember the User Agreement you signed!

This document was generated on 2024-01-26 02:25:05 PM (MST).

© 2024   bucknellmakers.dozuki.com/   Page 18 of 20

Step 18 — Replacing the bed

- Place back the bed on the printer.

- Make sure the edge with the 2 arrows is at the front and the arrows are aligned with the arrows on the underlying sheet.

- ⚠ Make sure the edges are placed correctly on the underlying sheet (i.e., they are not held up by the corners like the middle picture)

- Make sure the corners are not lifted (Refer to the picture in step 12).

Step 19 — Remove makerspace card

- When the print bed is clean, and you have removed all filament debris from inside of the enclosure, be sure to close the lid.

- Remove the makerspace card from the Portal box to shut down the printer.

- ⊡ Make sure to report any issues you encountered either by talking to a MakerE tech or leaving a note near the printer.