KLIC-N-PRINT 3D User Manual

June 13, 2020

Table of Contents

1. Introduction[1]

Thank you for purchasing a KNP3D! At KNK USA we are committed to providing the very best customer support for our products.

If you are new to 3D printing, remember that it's not like using a regular printer. It's not meant to be fast and it's not meant to be used as a production machine. 3D printing is a fabrication process and, with each print, you are essentially "growing an object." You will learn about the variables which affect each print (such as temperature, speed, layer thickness) and you will learn how these variables work in combination to produce amazing results!

So, let's get started! Before attempting to operate your KNP3D, please read Chapters 1 – 3. It's important to get the KNP3D set up properly, the plate leveled, and the proper settings loaded. Take note of anything marked **IMPORTANT** or in red type throughout this manual.

1.01 User Manual Options

It's not practical to print this entire manual because:

- It's a waste of paper and ink if you only ever need certain sections.
- The live links will not work.
- This manual is updated from time to time.
- You cannot search on individual words.

Also:

- Having the *Table of Contents* continuously viewable makes navigation easier. It's also useful to know how to search for specific words or terms. Here is a link to a tutorial which covers how to do these two things: How to Get the Most from your KNK UM.

1.02 Safety

Please be aware of the following safety guidelines when working with the KNP3D:

- Hot Elements: Do not touch the heated build plate or extruder nozzles while the KNP3D is printing. At the end of the printing process, wait for the printer to cool down before touching the extruder head or build plate. This can take around 15 to 20 minutes.
- Fire Hazard: Do not place flammable materials near the KNP3D.
- Pinch Points: Keep hands, long hair, jewelry, etc. away from the moving parts.
- Risk to Children and Pets: Please supervise children around the printer while it is heating, as well as printing. Also note that small printed items could pose a choking hazard to both children and pets, especially when inadvertently dropped onto the floor.
- Eye Protection: When removing support material on finished models, please use safety glasses to protect your eyes.

---

[1]

- Adequate Ventilation: When printing with ABS (acrylonitrile butadiene styrene) filament, a pungent odor will be emitted. Please operate the KNP3D in a well-ventilated room but avoid any direct air blowing on the printer as this can interfere with temperature control.

- Movement and Touching: Do not move the KNP3D or touch any circuitry while it is plugged in.

- Recommended Materials: Only use the recommended filament types mentioned in this manual. If you are interested in trying a different filament, please contact KNK USA first to determine if the KNP3D is designed to print that material.

1.03 Warranty

- **IMPORTANT**: If a KNP is damaged during the initial shipment to the customer, then your dealer should be notified as soon as possible. If damages are reported later than three weeks after delivery, then it will be the responsibility of the owner to ship the damaged KNP back to KNK USA for repair.

- The one-year parts and labor warranty covers parts for wear and tear. If warranty work is required, the owner is responsible for shipping costs to and from KNK USA. Discounted shipping will be provided by KNK USA.

- KNP3D owners outside the USA should contact their dealer for procedures regarding warranty work or shipping damage.

- It is also recommended that you retain the original box with packing materials in case you ever need to ship your KNP3D.

1.04 Support

- If you need additional assistance after reading this manual, here are several options for you:

  ◊ We have a KNP3D Facebook group at this link: https://www.facebook.com/groups/KNP3D/.

  ◊ We are also available by filling out a contact form at: https://knkusa.com/support-ticket-request/.

  ◊ If you purchased your KNP3D from one of our authorized resellers or distributors, please contact them as your first line of support.

1.05 Unpacking

- When unpacking the KNP3D we recommend placing the box on the floor.

  ◊ Open the top of the box and remove the cartons that surround the printer.

  ◊ Next, remove the printer by lifting it by the top metal frame. Do NOT remove the printer by lifting it by the cables.

  ◊ Take note that the two boxes at the bottom of the shipping box contain the power cable, USB cable and the filament guides.

  ◊ Place the KNP3D on a work surface. A 3D printer should be operated in a dust-free, non-humid room with a temperature of 65°F (18°C) or warmer. If printing ABS filament, also make sure the room has adequate ventilation.

  ◊ Open the box that contains the KNP3D extruder and remove the box from the printer.

  ◊ The build plate should now be visible. You'll notice a thin layer of Kapton tape covering the build plate. **Do not remove this film**. This is the surface material on which you'll eventually be printing.

- Next, the build plate needs to be raised.
  - ◊ This can be done by manually rotating the large threaded screw clockwise directly behind the build plate.
  - ◊ Raise the build plate until you can easily access any remaining packaging.

1.06 Contents

Please verify that you have received all of the following:
- KNP3D Printer
- USB Cable
- Power Cable
- 8 GB SD Card
- Hex wrench set
- 2 filament roll holders
- Sheet of Plastic Heat Bed Tape
- Sheet of Paper Heat Bed Tape

1.07 Other Tools and Supplies

Owners of 3D printers often find some of the following items useful. If you have other suggestions, please post at our user forum, linked in *Section 1.04,* and we'll include them in future updates!
- Safety Glasses – To protect your eyes while removing support material
- Cut-Resistant Gloves – To protect your hands while removing support material
- Paper Heat Bed Tape – Used to cover the build plate when printing with PLA
- Plastic Heat Bed Tape – Used to cover the build plate when printing with ABS
- Artist's Palette Knife or Metal Spatula – Used to remove printed object from the build plate
- Plastic Scraper – Used to apply tape to the build plate; aids in removing wrinkles
- Calipers – Used to measure the filament diameter (in case off-spec filament is suspected)
- Wire Cutters and Tweezers – Used to remove support material on printed objects
- Scissors – Used to snip off the end of filament (filament can have stringy ends after unloading)
- Lint Free Cloth – Used to wipe off dust
- Sandpaper – Used to smooth any rough edges on the printed object
- Files – Used to smooth any rough edges on the printed object

1.08 Parts of the KNP3D

Control Panel

SD Card Reader (on newer models the card reader is located at the bottom)

Y Axis Rods (2)

Z Axis Rods (2)

Build Platform

Extruders (2)

X Axis Rods (2)

X Axis Motor

Z Axis Limit Switch

Y Axis Motor

Build Plate

Build Platform Knobs (3)

Case

KLIC-N-PRINT3D

Front of KNP3D

Extruder Fans (2)

Nozzles (2)

Z Threaded Rod

Extruder Levers (2)

Interior of KNP3D

Extruder Cable

Filament Guide Tubes (2)

Filament Guide
Mounts (4)

Filament Spool

Spool Holders (2)

Power Outlet and
Power Switch

USB Port

Reset (do not press unless
instructed to do so.)

Back of KNP3D

2. Setting Up

2.01 Hardware Setup

- Install the extruder:

    (1) Start by manually turning the large threaded screw counterclockwise to lower the build plate all the way down.

    (2) Open the box which contains the extruder and remove it from the plastic sleeve.

Once positioned, bolt will pass through the mount and into the extruder here.

The other bolt will pass through the mount and be inserted here.

Bottom side of the extruder

One bolt will come up through this hole and into the bottom of the extruder.
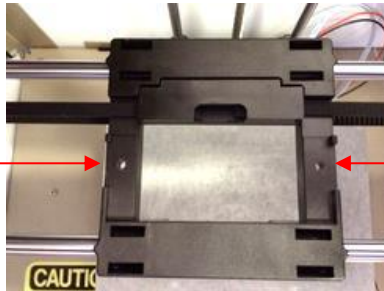
The other bolt will come up through this hole and into the bottom of the extruder.

Top side of the extruder mount

    (3) Place the extruder on the extruder mount, making sure the cooling fans are facing forward.

    (4) Once in place, locate the hex wrench and two silver bolts (found in the same accessory box as the extruder).

    (5) Tighten the two bolts on the bottom side of the extruder.
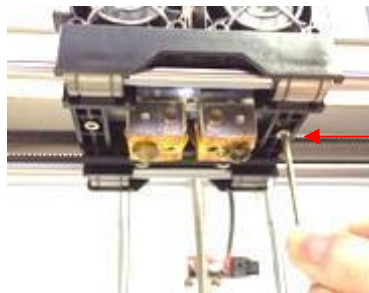
Screw the bolts on each side into place.

View from the bottom side of the extruder mount

11

- Attach the spool holders:

  (1) Open the two long boxes that were on the bottom of the shipping carton to locate two spool holders.

  (2) On the back of the KNP3D locate the two holes for mounting these holders.

  (3) Install by unscrewing the outermost nut on the spool holder and then inserting the holder into the hole and tightening the nut from inside the KNP3D.
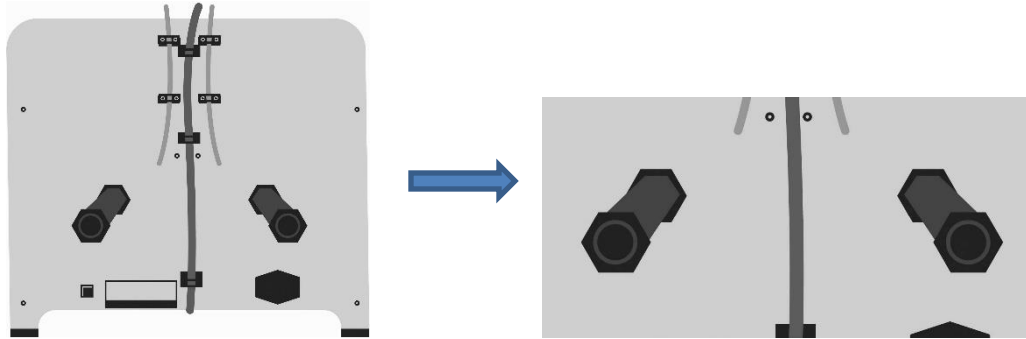
Most of the spool holder extends out from the back of the printer.

- Attach the filament guide tubes and filament:

  (1) Locate the two filament guide tubes.

  (2) Slip each tube into the fasteners on the back of the KNP3D and screw the fasteners securely:
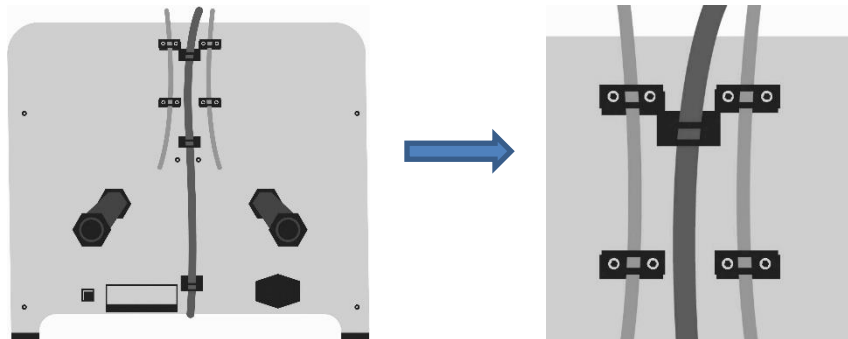
  (3) Place a roll of filament on the right spool of the KNP3D, making sure the filament will feed from the bottom side of the spools, straight up into the guide tubes:
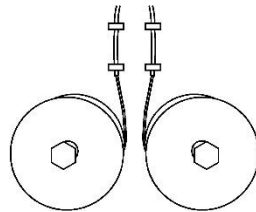
  (4) Loading the filament into the extruder will be covered in *Section 2.05.*

- Connect to an outlet:

  (1) Plug the power cable into the power port on the back of the printer.

  (2) Be sure the power switch is turned "off" before plugging the power cord into a power outlet.

(3) Turn off the power when not using the printer. Unplug from your power outlet if the printer will not be used for a long period of time.

2.02 Software Setup

There are a few pieces of software you'll need in order to create files that the KNP3D uses:

> **NOTE:** This manual is based on Slic3r 1.2.9. If you use a newer version, screenshots in this manual may not match up. Also, Slic3r profiles may need modifications.

2.02.1 Slic3r

- Slic3r is the tool you need to convert a digital 3D model into printing instructions for your 3D printer. It cuts the model into horizontal slices (layers), generates tool paths to fill them, and calculates the amount of material to be extruded.

- Windows Installation:

    ◊ Download Slic3r from this link: https://slic3r.org/download/. Choose either x64 or x86, depending on your computer: a 64 bit computer or an older 32 bit, respectively.

    ◊ Slic3r is a self-contained executable, requiring no installation. You simply download it, extract it, and it's ready to use. You will find there are two executable files: *slic3r* and *slic3r-console*. You only need to use *slic3r*.

    ◊ We recommend saving this *slic3r* file somewhere accessible, such as your **Desktop**, or creating a shortcut icon on the **Desktop**.

    ◊ The first time you open *slic3r*, the **Configuration Assistant** window will open. Click on **Cancel** as you will need to switch to **Expert** mode. This is covered in Chapter 3.

- Mac OSX Installation:

    ◊ Download Slic3r from this link: https://slic3r.org/download/.

    ◊ Double click the downloaded DMG file.

    ◊ Once **Finder** opens, drag the Slic3r icon into your **Applications** folder.

    ◊ The first time you open Slic3r, the **Configuration Assistant** window will open. Click on **Cancel** as you will need to switch to **Expert** mode. This is covered in Chapter 3.

    ◊ **Note:** You may receive a message that says "'Slic3r' can't be opened because it is from an unidentified developer." If so, then in your Mac's **System Preferences**, select **Security & Privacy** and click the lock icon that reads "Click the lock to make changes." Then set "Allow applications downloaded from: 'Mac App Store and identified developers'" to "Anywhere". Start Slic3r. Once running, if you'd like, you may reset Gatekeeper to "Mac App Store and identified developers".

2.02.2 KNP3D Converter

- KNP3D Converter is a post processing application. This will take the gcode files from Slic3r and convert them into x3g, a format the KNP3D understands.

- Download the KNP3D Converter (Windows or Mac) from this link:

  https://www.iloveknk.com/Support/KNP3D/Programs/

- Windows Installation:

  ◊ KNP3D Converter is a self-contained executable, requiring no installation. You simply download it, extract it, and it's ready to use.

  ◊ We recommend saving it somewhere accessible, such as your Desktop, or creating a shortcut icon on the Desktop.

- Mac OSX Installation:

  ◊ Double click the downloaded DMG file.

  ◊ Once **Finder** opens, drag the KNP3D Converter icon into your **Applications** folder.

- The KNP3D Converter can be used either of two ways:

  ◊ Open KNP3D converter, browse to locate the input .gcode file created in Slic3r, choose an output location (e.g. SD card) and click on **Convert.**

  ◊ Alternatively, after creating the .gcode file in Slic3r, open the folder where the file was saved. Drag and drop this file onto the KNP3D Converter icon and it will automatically generate the .x3g file and save it in the same folder.

2.02.3 Slic3r Profiles for KNP3D

- To get you started with appropriate settings for printing, some profiles are set up and available to download. A few things to know about these profiles:

  ◊ The profiles contain settings for the KNP3D itself, the filaments you can use, and some initial printing recommendations (temperatures, speeds, thicknesses).

  ◊ There are different profiles for fast printing (draft mode), medium printing (default mode), and slow printing (high quality mode).

  ◊ **Important:** There are no "perfect" profiles that work on every single type of object you might choose to print. For example, a small flat object will mostly likely need a different profile than a tall thin object. So, you will need to be doing test prints and adjusting settings as needed. When you find the best settings for a particular type of object, you can save those settings with a new profile name and build your own library of settings.

- Use the following link to download a profile bundle containing the settings to get you started**:**

  KNP3D Profile Bundle

- This is a zip file so, after downloading, extract the .ini file and note where you have saved it. Instructions for loading this file into Slic3r will be covered in *Section 3.02.*

2.03 Control Panel

- When you initially power up the KNP3D, a welcome screen will appear for a few seconds, followed by the **Main Menu**:

- To maneuver through the menu, press the buttons on the panel to the right of the display:

Move up the menu.

Go back to prior menu.

Go to next menu.

Select current option.

Move down the menu.

- **Main Menu** options include:

Small arrow indicates currently selected item in menu.

Browse SD card for a file to print. See *Sections 3.03 (step 10)* and *5.14*

Begin preheating filament and build plate. See *Section 5.02*

Go to **Utilities Menu**: See below and also *Appendix C.*
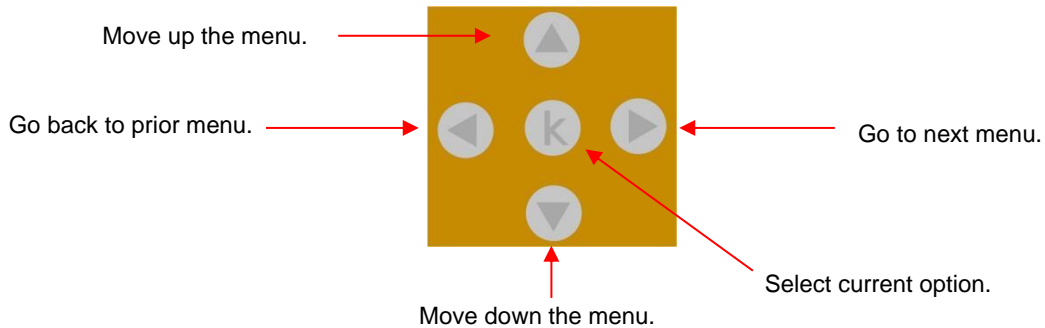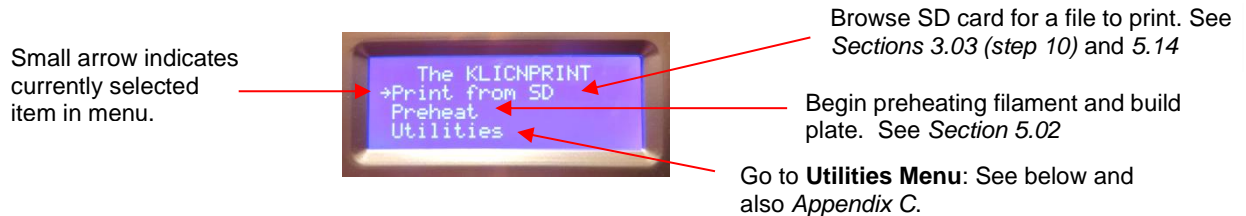
- **Utilities Menu** options include:

Check current temperatures

Load or Unload Filament See *Sections 2.05* and *5.05*

Change preheat settings See *Appendix C7*

Level Build Plate See *Section 2.04*

- Additional settings within the **Utilities Menu** are presented in *Appendix C.*

2.04 Leveling (Tramming) the Build Plate

- The leveling (tramming) procedure is <u>very important</u> for making sure your prints are successful. While "leveling" is the term used by most 3D printer owners and companies, the correct term is "tramming". The goal is to make sure the distance between the build plate and the nozzle is the same at all locations across the build plate.

- For this process, you'll need a single sheet of printer paper or even just a Post It note.

    (1) Turn on the power to the KNP3D. Place the sheet of paper on the build plate.

    (2) On the control panel, select **Utilities** and then select **Level Build Plate.** The current temperatures of the extruders and platform (build plate) will appear:

    (3) The extruder head will first move over to the right side and the build plate will rise up. Then the extruder will move to the center of the build plate. The display will now show this:
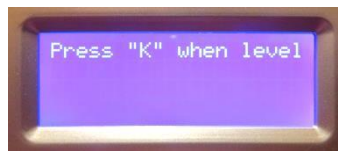
    (4) You can now manually move the extruder head in different positions over the build plate by sliding it left / right or pulling it forward / pushing it backward.

- Since this is a manual process, you can move the extruder head to as many positions as you would like. However, for this calibration, you will be moving the head to three different positions: over each of the three adjustment knobs (located below the build plate). In each position you will adjust the height so that it just touches the piece of paper.

    (5) To begin, move the extruder head above the front right knob.

    (6) If you cannot slide the paper between the extruder and the build plate, you have it too high and need to turn the knob to the left. If you can slip the paper between the build extruder and build plate, with room to move the paper up and down, you have the build plate too low so turn the knob to the right. Again, the goal is to have just a little bit of resistance when you slide the paper under the nozzle.

        ◊ Turn the knob to the right to raise the build plate (Right = Raise)

        ◊ Turn the knob to the left to lower the build plate. (Left = Lower)
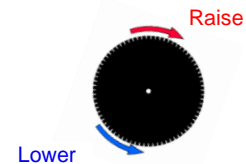
    (7) Once you have the front right knob adjusted, move the extruder head   to the front left and repeat Steps (5) and (6). You want to avoid having the nozzle dig into the plate so if, while moving, the paper suddenly has a lot of resistance, stop and lower the plate at that point.

    (8) After you have the front left knob adjusted, move to the center back. Repeat Steps (5) and (6).

    (9) Upon making adjustments at all three locations, repeat Steps (5) – (9) and adjust again. It usually takes three or four repeats at each location to get the tramming just right.

    (10) If you're satisfied that the plate is trammed and the paper will slide with just a little resistance in all three locations, press **K** on the control panel. Note that this calibration procedure should be repeated any time you've moved the printer or if you are having printing issues.

2.05 Loading Filament

(1) Before loading the filament, be sure to disconnect the guide tube from the top of the extruder by pulling the plastic tubing out of the hole in the acrylic plate.

(2) Feed the filament into the bottom of the guide tube until it comes out the other end. If you plan to use only one filament, the right side is considered the default. Remember that the spools should be loaded so that the filament feeds straight up into the guide tubes:
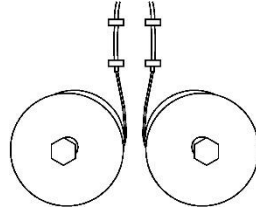
(3) Extend the filament several more inches so that you can easily hold it between your thumb and index finger. Use scissors to cut off and discard about one inch of the end.

(4) Next, on the control panel, go to: **Utilities > Filament Loading > Load Right:**

(5) The extruder head will now begin heating up to 230°C. This will be displayed on the control panel and, as the temperature increases, the progress bar will expand:
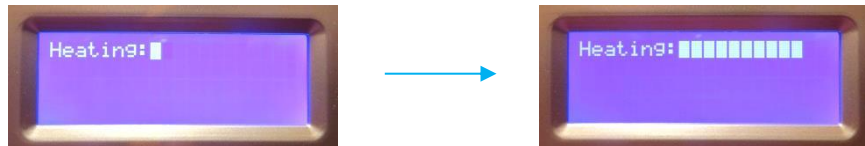
(6) When the extruder reaches the preset temperature, the KNP3D will play a little tune, prompting you to load the filament. The display will show:

(7) Press down on the small extruder lever on the right side of the right extruder and feed the filament firmly down into the hole on the top.
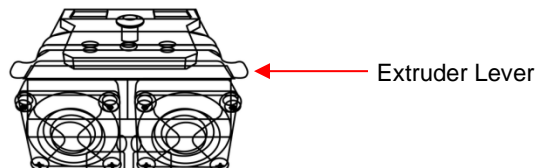
Extruder Lever

(8) Once in, let go of the extruder lever and the filament will begin to slowly feed into the extruder. Place the end of the guide tube into the hole in the acrylic plate. After a few seconds, you'll see filament streaming out of the nozzle.

(9) You will most likely see a different color for the first few seconds. Not to worry. Every machine is tested with a randomly selected colored filament before it's packaged for shipment. Once the proper color filament starts streaming out of the nozzle, allow it to continue feeding through for another 5 – 8 seconds.

(10) Press **K** to exit. Turn off the KNP3D. After a minute or so, you can remove and discard the extruded filament.

(11) If, at any time during the loading process you need to cancel, press the left button on the control panel and the option to cancel will appear.

(12) To learn how to unload filament, refer to *Sections 5.05* and *5.06.*

3. Your First Print

3.01 Overview of the 3D Printing Process

- 3D printing on a KNP3D uses an additive manufacturing process called Fused Filament Fabrication, allowing users to turn computer-designed 3D models into physical objects.

- Any 3D program which will export in .stl, .obj. or .amf format can be used. The following are several programs to consider, if you are new to designing in 3D:

  ◊ Rhino3D

  ◊ Moment of Inspiration

  ◊ Or any number of other 3D design programs

- You can also locate .stl files on the Internet to download. Many of these are free files! To locate these, just search on "free stl files" at google.com. For now, a test file will be provided at a download link in *Section 3.03*.

- A program (typically referred to as a slicer), opens the .stl file and "slices" it into very thin layers, which is then translated into instructions that the printer understands. We recommend using Slic3r, which you downloaded in *Section 2.02.1.*

- There are different types of materials (aka filaments) which can be used for printing, such as PLA, ABS, nylon and more. These plastics are fed through the printer's extruder which melts the plastic and squirts it out of a nozzle (kind of like a miniature glue gun). The printer extrudes layer on layer until the object is complete.

- Below is a flow diagram of the process used with the KNP3D. Most steps involve just a click or two, so do not fret over the number of steps. Once you've performed this procedure a few times, it will be easy to remember.

```
┌─────────────────────────┐
│   Design or download    │
│      an .STL file       │
└───────────┬─────────────┘
            ↓
┌─────────────────────────┐
│ Open .STL file in Slic3r│
│  and adjust print settings│
└───────────┬─────────────┘
            ↓
┌─────────────────────────┐
│   Export file as .gcode │
└───────────┬─────────────┘
            ↓
┌─────────────────────────┐
│     Open .gcode file    │
│     in KNP Converter    │
└───────────┬─────────────┘
            ↓
┌─────────────────────────┐
│   Convert to .X3G and   │
│     save to SD Card     │
└───────────┬─────────────┘
            ↓
┌─────────────────────────┐
│ Insert SD Card into KNP3D│
│  and select file to print│
└─────────────────────────┘
```
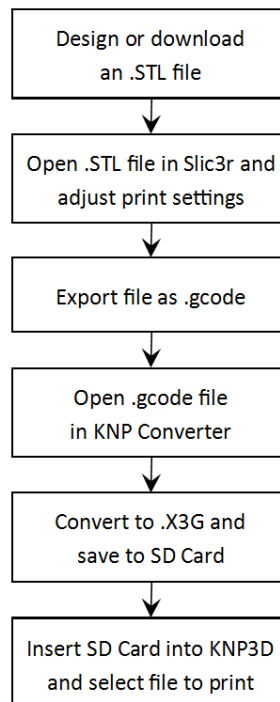
- We recommend you follow all of the steps in this chapter as a way to test your KNP3D. You can use either of the free PLA filaments that were shipped with your KNP3D. PLA (Polylactic Acid) is a popular filament to use because it readily sticks to the build plate, has a pleasant odor when heated, and is less likely to warp than the other commonly used thermoplastic, ABS (acrylonitrile butadiene styrene).

3.02 Load Profiles into Slic3r

- In this section, the profile bundle downloaded in *Section 2.02.3* will be loaded. As mentioned before, these profiles are starting points for prints. As you test and create your own successful models, you'll want to save your settings under new profile names. We also encourage you to share your results at our Facebook group: https://www.facebook.com/groups/KNP3D/.

- In *Chapter 4*, the most important features of Slic3r will be covered. Additionally, you can learn more about Slic3r by referring to its user manual at this link: Slic3r User Manual.

- The first time you open Slic3r, the **Configuration Assistant** window will also open. Click on **Cancel** as you will need to switch to **Expert** mode.

- To switch to **Expert** mode, go to **File > Preferences** (on a Mac, go to **Slic3r > Preferences**) and select **Expert** in the dropdown menu and mark all of the boxes. Then click on **OK**:
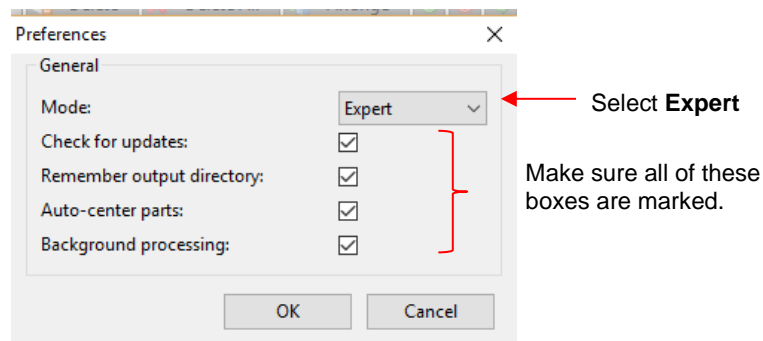
- You will need to restart Slic3r so that the menus will switch to the **Expert** mode.

- Once Slic3r has reopened, go to **File > Load Config Bundle** (note: not the top menu item, but the third one in the menu). Browse to locate the "*KNP3D Profiles*" file you downloaded in *Section 2.02.3*. Select it and click on **Open**.

3.03 Print a Test File

You're now ready to make your first print. We've created a simple 3D key fob to be used:

<div align="center">Download Key Fob</div>

(1) Download this .stl file and then open up Slic3r.

(2) On the **Plater** tab (top left), click the **Add** button. Browse and locate the downloaded .stl file and click on **Open**. The file name will appear under **Name**:

Note that the following screen was resized for easier screen capture for this user manual.

Click on **Add** and select the test file you downloaded.

Preview

Preview options

Select the same profiles shown here.

Click on **Export G-Code** when ready

File name

File information
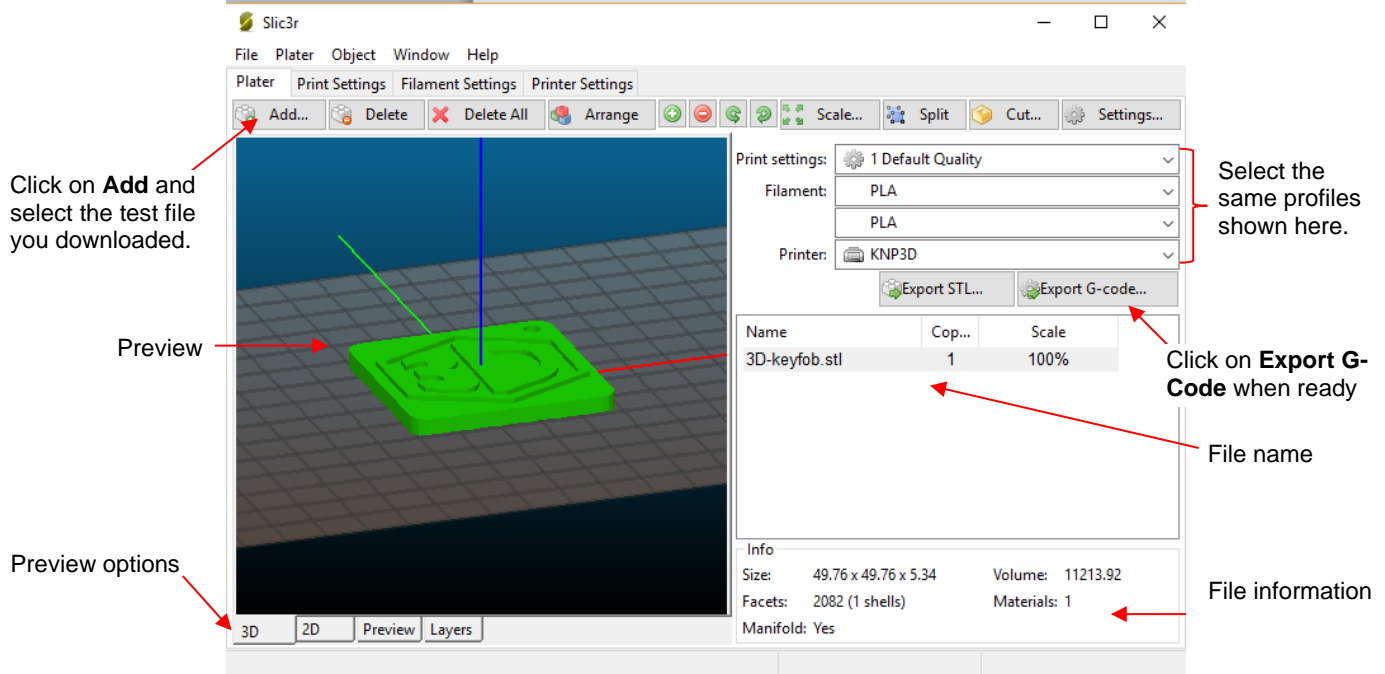
(3) On the right side, the profiles to be used in this print will need to be selected. Click on each drop-down arrow and select the **Profile** name shown in the screenshot above.

(4) Note that the upper profile under **Filament** is for the right extruder and the lower profile is for the left extruder. There might be situations in which you will need to select a different profile for the second extruder if using it for your print.

(5) Click the **Export G-code** button over on the right side. A window will open and you will be able to now click on **Save**. Note where the file will be saved and change the location, if desired.

(6) Next, we need to convert the gcode file to x3g. Open the KNP3D converter program and click the **Browse** button to the right of **Input File.** Locate and choose *KNP3Dcube.gcode* and click **Open**.

(7) Click the **Browse** button to the right of **Output File**. Here you can save the file directly to your SD card, or you can save it to a different directory and copy to your SD card later. For now, save to the SD card and click on **Save**.

(8) Now click on **Convert**. Once complete, a window will open letting you know the file has been successfully converted:

Gcode file selected here

Location for saving converted file

Click on **Convert**
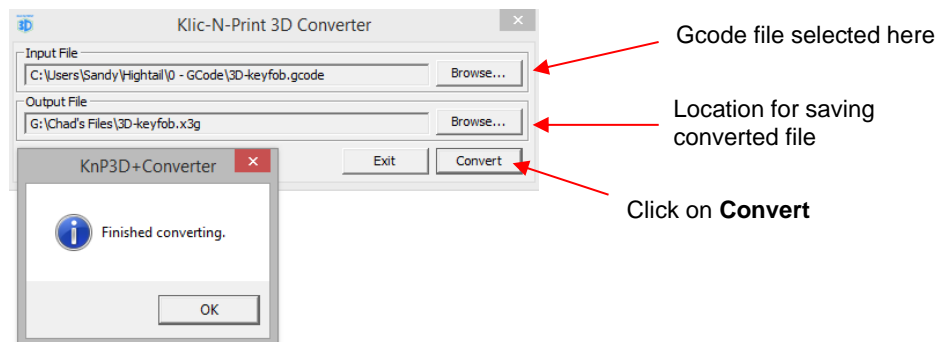
(9) Turn on the KNP3D.

(10) Insert the SD card into the 3D printer with the label facing forward:

Insert the SD card here with the label facing forward (on newer models the card reader is location at the bottom of the printer)

(11) From the KNP3D control panel, select **Print from SD** and then select the KNP3Dcube.x3g file.

(12) Your KNP3D should begin its first print after the build plate and extruder reach the preset temperature.

(13) Note that a skirt is always printed first to ensure the filament is flowing smoothly. If, after two passes, there is no filament or the filament is not sticking to the build plate, cancel the print by pressing the left key on the control panel. Refer to *Section 5.01.*

(14) The screen display during printing, which is called **Monitor Mode**, will provide the following information:

Percent completion

Current / Target temperatures

(15) When the printing is complete, the extruder will return to the back right corner. Allow to cool for a few minutes.

(16) To remove the model, use a sharp flat tool, such as a thin metal spatula or an artist's palette knife. Gently slip the tool under the outer edge of the model in several different spots. Then slip it under again and twist. The model should pop off (separate) easily from the plate.

(17) On the control panel, the following screen appears showing the length of time for that print, as well as the option to print another. If you are finished printing, turn off the power to the KNP3D.

Amount of filament used for last print.

Option to repeat the last print

Printing time for most recently completed print

(Note: the Print Time and Filament values above were for a different print, not the test object.

If you would like to print a quick file (handy when showing off your new KNP3D to friends and family members), try either or both of these two guitar pick files. They each print in about 5 minutes:

Thin Guitar Pick

Slightly Thicker Guitar Pick

4. More About Slic3r

4.01 Important – Read This!

Here's a list of the **key things** to check when using Slic3r:

- Do not automatically upgrade to the latest version of Slic3r without seeing an announcement by KNK USA at the User Forum. It's important that any changes made in Slic3r not impact the KNP3D in unexpected ways. KNK USA will thoroughly test all new stabilized versions before recommending them to KNP3D owners. The current version recommended is 1.2.9.

- Upon importing an STL file, always check the **Size** in the **Info** window. If the file being imported was designed in inches, then it will not print the correct size unless it is scaled in Slic3r, which uses mm as a default unit. This is covered in *Section 4.05.6.*

- Under the **Info** section of Slic3r, you will see a parameter called **Manifold**. Always make sure **Manifold** says "Yes", which indicates that edges are properly aligned in the model. Refer to *Section 4.05.10.*

- Before exporting to G-code, make sure the correct profiles are selected under **Print Settings, Filament**, and **Printer Settings** on the right side of the Slic3r window. Also remember that you need to adjust settings based on the type of object you're printing. This will be covered in the remainder of this chapter and in *Appendix B.*
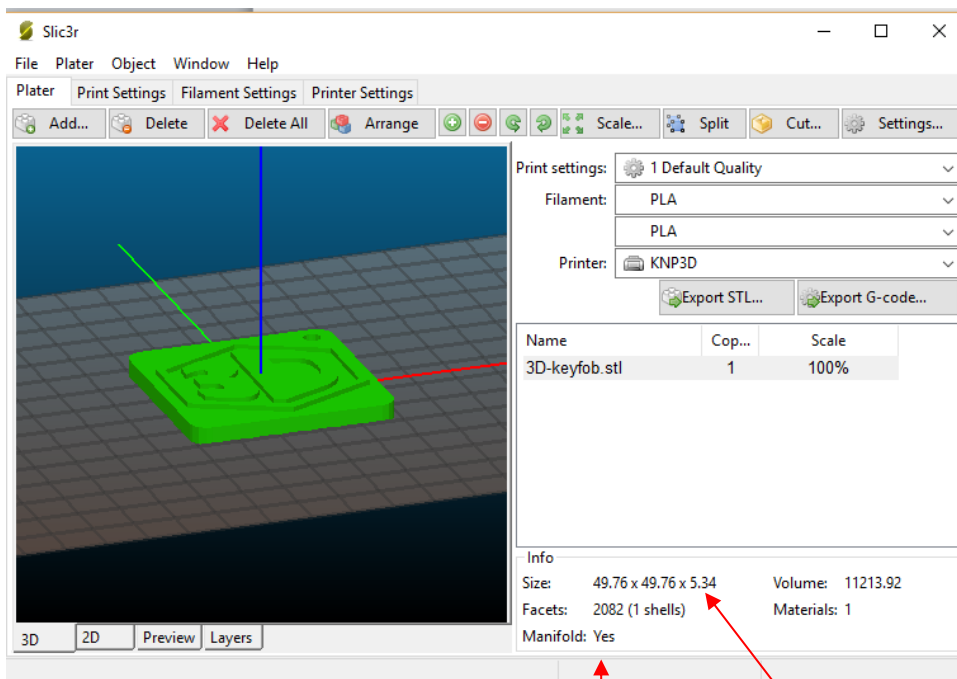
Are these the correct Profiles for the print?

Verify that **Manifold** says "Yes"

Check the **Size** which is shown in mm

4.02 Slic3r Tabs

4.02.1 Tabs Across the Top

- Notice the four tabs across the screen: **Plater**, **Print Settings**, **Filament Settings**, and **Printer Settings**:
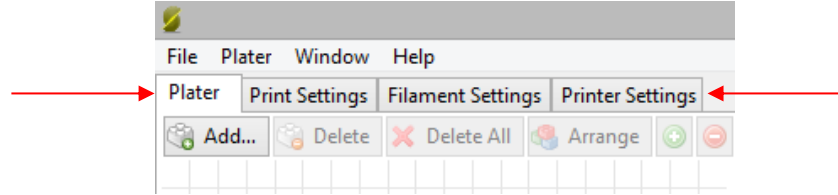
- Each of these tabs accesses certain settings that could, at one time or another, need to be changed from the defaults.   Some of these settings may then remain constant once changed. Others will need to be adjusted depending on the particular object being printed or filament being used.

- The functions of these tabs are:

   ◊ **Plater** tab is used when importing the .STL, .OBJ, or .AMF file and then making changes such as scaling, duplicating, and rotating. This tab will be used with each new model you wish to print. Refer to *Section 4.05* to understand the purpose of the icons in the **Plater** window.

   ◊ **Print Settings** are related to the print itself, such as the layer height, fill density, and various speeds. This tab will be used each time a different type of printing is needed, such as changing the quality of the print or adjusting settings related to the type of model.

   ◊ **Filament Settings** are related to the filament material, such as thickness, required temperature, cooling options. These settings will normally remain the same unless you change to a different type of filament, such as switching from PLA to ABS.

   ◊ **Printer Settings** are related to the KNP3D itself, such as the bed size, firmware type, and number of extruders.

4.02.2 Tabs Across the Bottom

- At the bottom left are four tabs which control how the preview appears:

- **The purposes of each tab are:**

   ◊ **3D**: This is the default mode when you open Slic3r. Once you open a file in Slic3r, a 3D simulation appears in the preview window. Note that the red, green, and blue lines represent the X, Y, and Z axes. Also, you can drag your mouse over the preview to change the angle of view:

The three axes intersect at the center of the object, thus the object will be cut at the origin on the build plate

3D simulation

3D selected
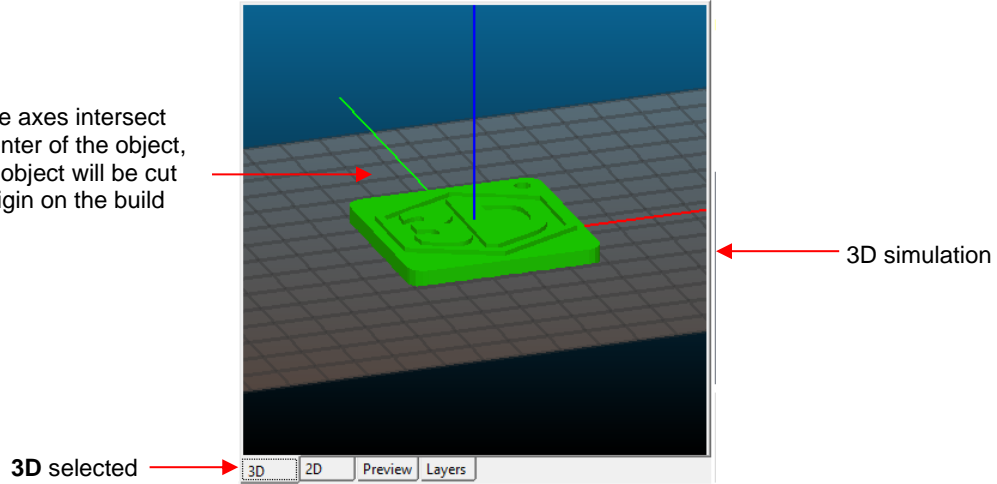
◊ **2D**: This is a 2-dimensional top view of the shape and many of the Slic3r screenshots in the rest of this chapter will appear in this mode because it was the only mode in prior versions of Slic3r:

2D top view

Object is centered with the origin (interesction of X=0 and Y=0)

2D selected
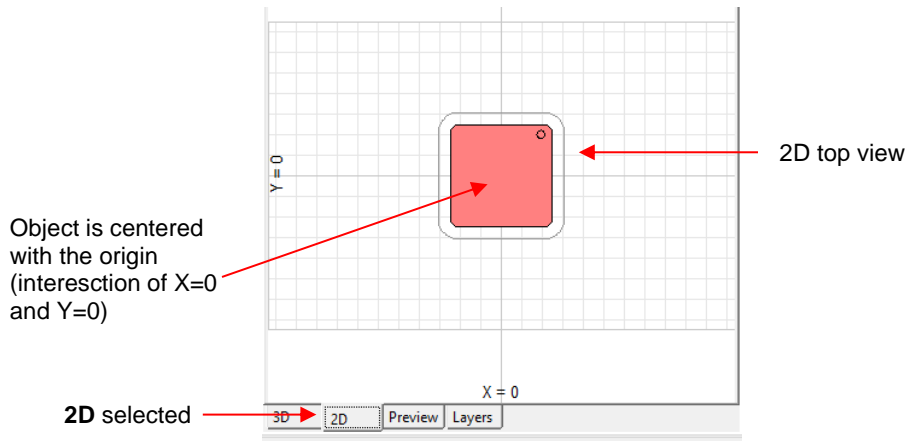
◊ **Preview:** This is another 3D preview with the addition of several things: (1) the skirt that is always printed first is shown and (2) a scroll bar on the right side allows you to see the print process:

Drag this bar to see the print status at various stages along the Z axis

The skirt is shown

Current Z axis height for the preview shown

Preview selected
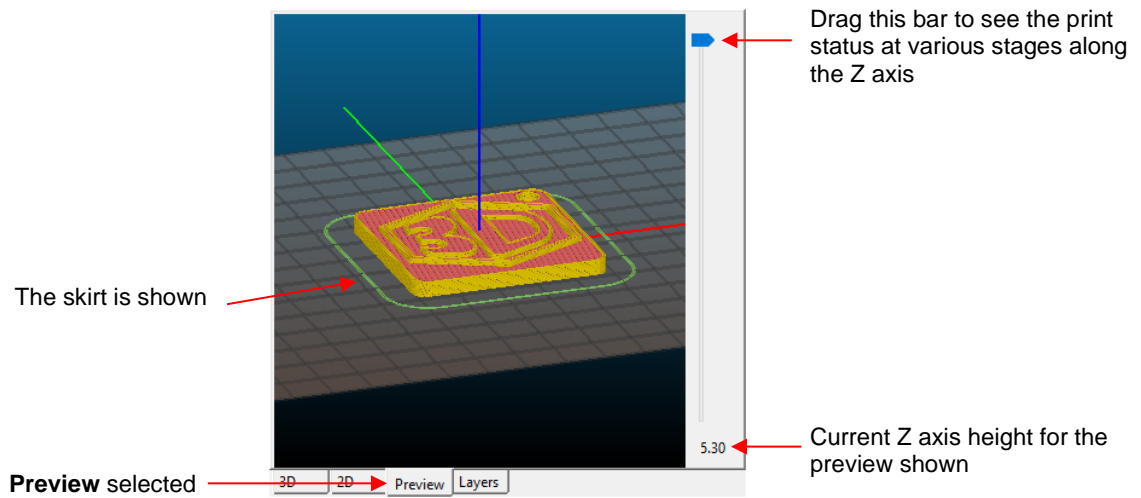
◊ **Layers:** This mode displays what is being printed at each Z axis height. Again, use the scroll bar on the right side to change the display. Here are several examples starting at the bottom and then scrolling up the Z axis:

0.30

| 3D | 2D | Preview | Layers |
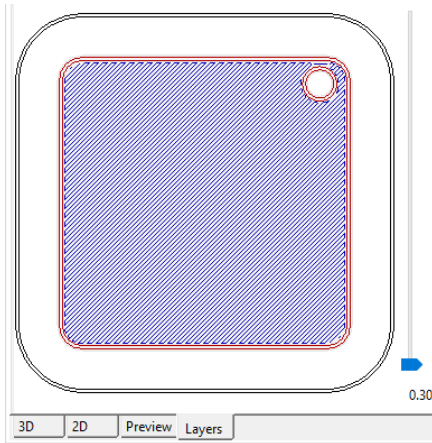
Initial bottom layers including
the skirt

0.50

| 3D | 2D | Preview | Layers |
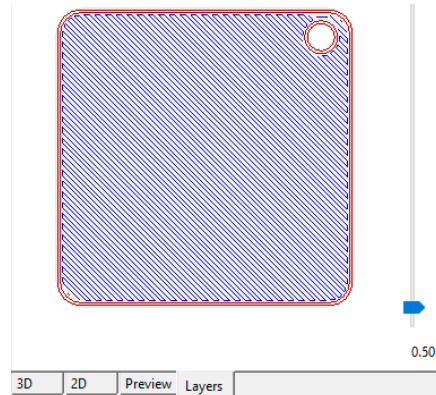
At 0.5 mm, bottom layers are printed in
opposite direction

0.90

| 3D | 2D | Preview | Layers |
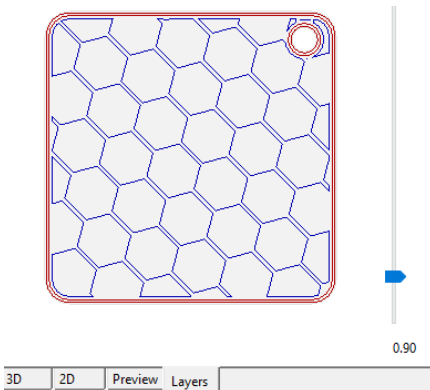
Honeycomb fill pattern begins

4.30

| 3D | 2D | Preview | Layers |
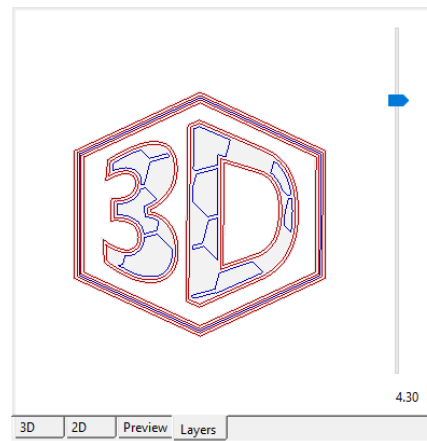
The "3D inside of a hexagon"
detail begins

4.02.3 The Other Tabs

- The line of tabs just above the **Preview** window are used to make changes to the object before exporting:

- These various functions will be covered in *Section 4.05.*

4.03 Understanding Profiles

- When you loaded the "*KNP3D Profiles*" file in *Section 3.02.2*, it added 10 different profiles and these profiles now appear in the dropdown menus on the right side of the main screen:

Different profiles available for each tab
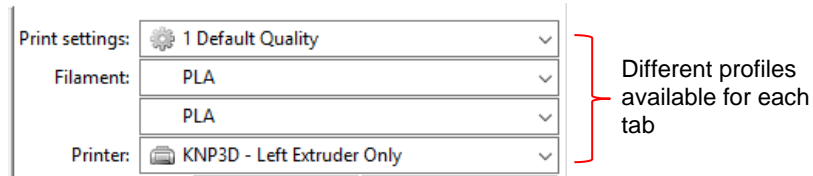
- A profile contains settings from one or more of these three tabs: **Print Settings**, **Filament Settings**, and **Printer Settings**. The 10 imported ones are for the following applications. Again, remember that you will be making adjustments based on your own experiences for different types of shapes.

Print Settings	1 Default Quality	Medium settings; good for a first test print
	1 Faster - Lower Quality	For prototyping
	1 Slower - Higher Quality	For display objects or more accurate results
	Left Extr Only - Default Quality	
	Left Extr Only - Faster	Same settings as above but only the left extruder will be used
	Left Extr Only - Slower	
Filament Settings	ABS	
	PLA	Select based on which type of filament is being used.
Printer Settings	KNP3D	Always select KNP3D unless the left extruder will be used instead of the right.
	KNP3D - Left Extr Only	

- A profile can contain settings for just one tab or for more than one tab:

◊ For example, one could have a profile of settings pertaining to ABS which was set up under **Filament Settings**. One might also set up **Print Settings** more suited for ABS than for PLA. These two profiles could be saved under separate names - *ABS Print* and *ABS Filament*.

◊ Alternatively, both sets of settings could be saved under the same name and then would be located under both menus. Let's assume that the same name "*ABS Default Print Settings*" was used for saving both the changes made under **Filament Settings** and under **Print Settings**. That means this profile would then be available under the **Filament Settings** menu and under the **Print Settings** menu, but not under the **Printer Settings** menu (because it was never saved under that tab, using that name).
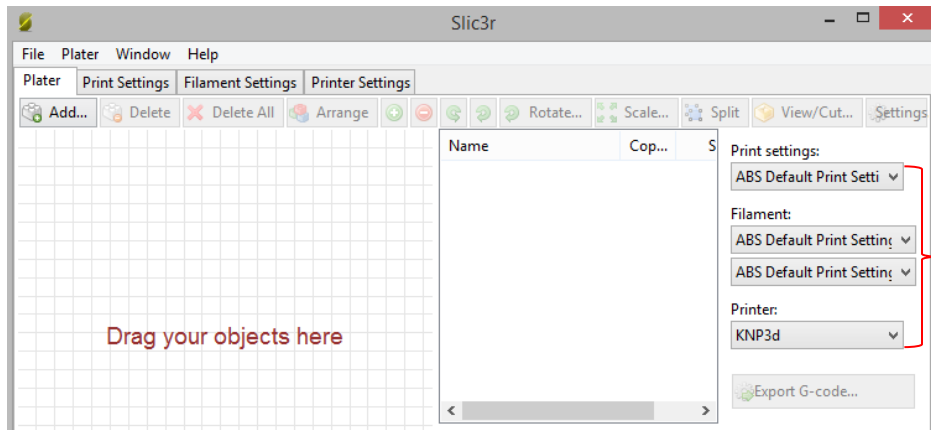
ABS Default Print Settings can be selected in these menus, but not under **Printer**.

4.04 Changing and Saving Settings

- The process of testing different variables is part of owning a 3D printer and getting the best results from it. It's usually advisable to change only one variable at a time and to take notes. You may find that instead of trial and error, you'd prefer to have some specific recommendations. Feel free to post on the user forum asking for help! We urge you to also post your .STL file (provided you have permission to do so) and a photo of your print with a description of any issues you had with that print. You will get the help you need and, at the same time, help future owners who read the forum and see your post.

- In this section, some typical changes will be made and saved, so that you'll understand how the process works. This same method is used for any changes under the **Print Settings**, **Filament Settings**, and **Printer Settings** tabs.

- You will see there are many settings available in Slic3r. Note that if you rest your mouse on the input window of any given setting, a popup window will appear describing the purpose of that setting. This is a great way to not only learn the function of that setting, but also to better understand the 3D printing process itself. Also take note that *Appendix B* and the Slic3r User Manual cover these settings in more detail.

- Try out the following example and make some changes under **Print Settings**:

  (1) Click on the **Print Settings** tab.

  (2) Select a profile to change. In this example, you'll be using the **Default**, as indicated by the upper blue arrow in the screenshot below.

  (3) Next, select from the submenu items which appear along the left side. The first one you'll be changing is part of **Layer and perimeters**. Thus, click on **Layer and perimeters** as indicated by the lower blue arrow.

  (4) The red arrow indicates a typical setting that might be adjusted. Let's say you wish to try a smaller **Layer height** which should improve the quality of the print. Change it from 0.2 to 0.15:

Select "*1Default Quality*"

Select **Layers and perimeters**

Change from 0.2 to 0.15.
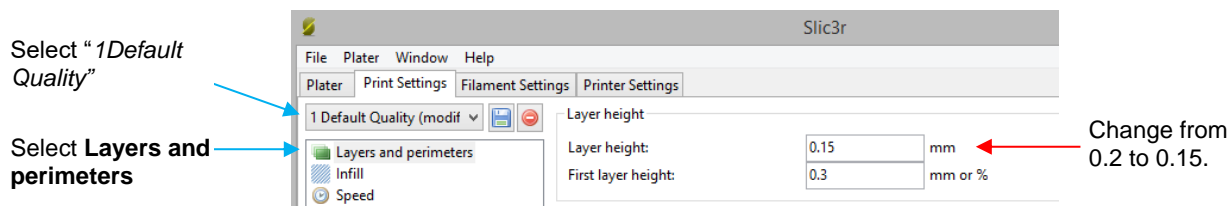
  (5) Now click on **Infill**, the next category in the left side menu. Decrease the **Fill density**, which will speed up the print a bit, to compensate for the lower **Layer height.**

Select **Infill**

Change the **Fill Density** to from 15% to 10%.
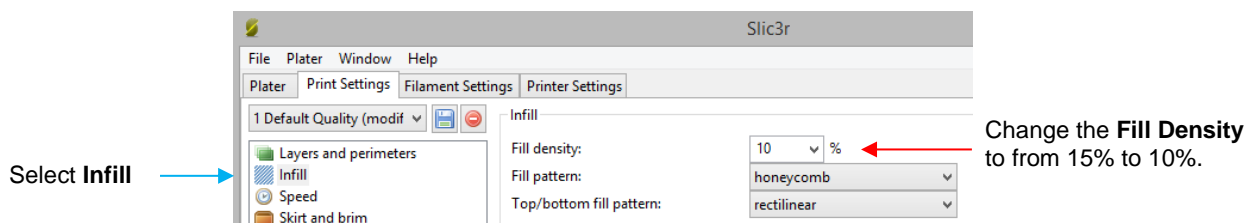
IMPORTANT: Once you make a change you need to save these settings on each tab, before moving to another tab. You can either overwrite the settings for this particular profile or you can save and enter a new **Profile** name, thus retaining the original profile. Normally, it's a good idea to use a new

name since the settings you may find that work well for one object might not work well for the next. Thus, you might want to name the **Profile** according to the object itself.

(6) So, at this point click on the small **Save** icon and give this new profile a new name, such as "*Dog Tag*" (assuming that's the object you're printing!):

Click here to save the new **Print Settings**

Input a new name and click on **OK**.
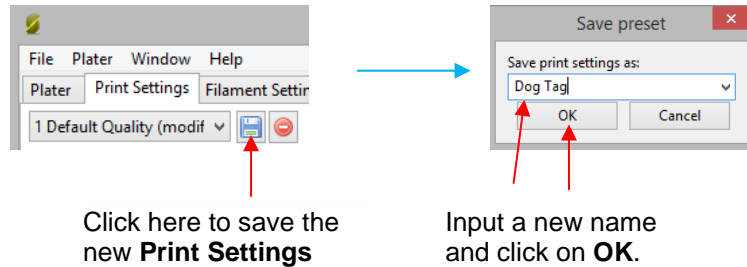
(7) Now let's assume you also want to make a **Filament Settings** change. For example, you might want to raise or lower the temperature. Click on that tab and then select the PLA profile:

Select **PLA**

Select **Filament**

Change the **Extruder Temperatures** to 210 and 200.
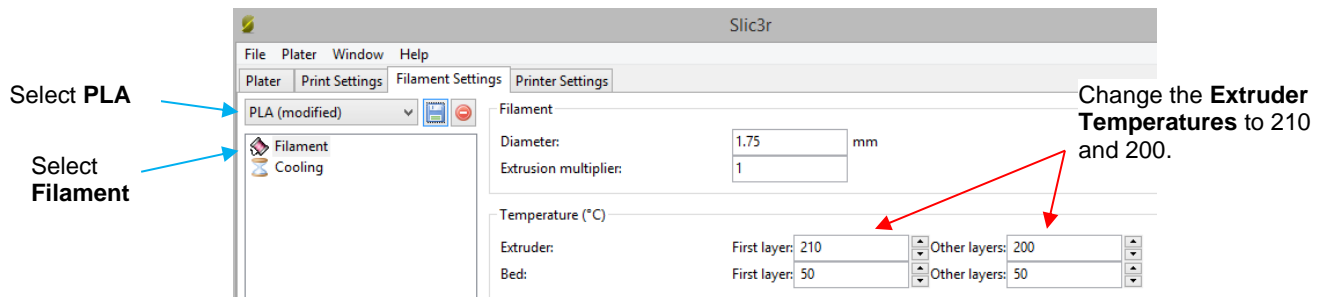
- Also, save these changes but this is the point at which you have a decision to make. Pick whichever one makes the most sense to you and, with experience, you'll develop your own preferences. Your options are:

  ◊ Replace the current PLA profile

  ◊ Save it using the same name as the one when you changed the **Print Settings** *("Dog Tag")*

  ◊ Create an entirely new name for this particular **Filament Settings** profile, such as "*PLA Lower Temp.*"

- If you wish to delete any profiles, go to the appropriate settings tab, select the profile, and then click on the small red icon (see below). If that profile name is used on other tabs, then it will need to be deleted separately from each one:

Click here to delete the "*Dog Tag*" profile from the **Print Settings** menu.
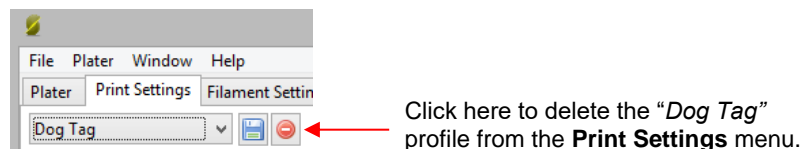
- As a safeguard, you should also save all of your settings by going to **File > Export Config Bundle,** and selecting a folder on your computer in which to save the configuration. Give it a new name, such as "*My KNP3D Profiles*".

- Note that this new "*My KNP Profiles.ini"* file can be loaded into Slic3r on another computer, saving you the time *of* re-entering changes to any settings. Go to **File > Load Config Bundle**, browse to locate the file, and click on **Open.**

4.05 Plater

- As stated earlier, the **Plater** tab is used when importing the .STL, .OBJ, or .AMF file and then making changes such as scaling, duplicating, and rotating. It also displays the profile selection menus, and the export options for G-code and STL.

- The following subsections explain the purpose of each button/icon you see while the **Plater** tab is selected.

4.05.1 Add

- The **Add** function is used to import .STL, .OBJ, and . AMF files into Slic3r.

- You are not limited to adding and printing only one file at a time. For example, in the screen shot below, two different STL files have been imported using the **Add** button. Both files will be printed and the two files together can be saved as a single STL file by clicking on the **Export STL** button on the right side.

- Shapes can also be moved to a different location on the grid by dragging them with your mouse.

Use the mouse to drag the shapes, as desired.
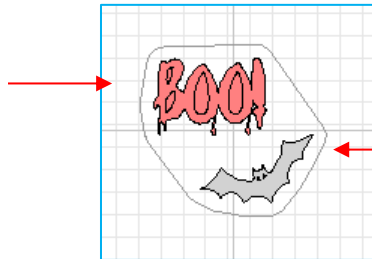
The thin border around the shapes is the skirt which will be printed first.

4.05.2 Delete and Delete All

- You can delete one or all of the STL files currently loaded. To delete just one of the files, highlight it and then click on **Delete.**

- **Delete All** will delete all loaded files without the need to highlight any of them.

4.05.3 Arrange
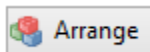
- This function will automatically arrange objects to take up less space. For example, let's say the two objects were dragged far apart, clicking on **Arrange** will bring them close together again:
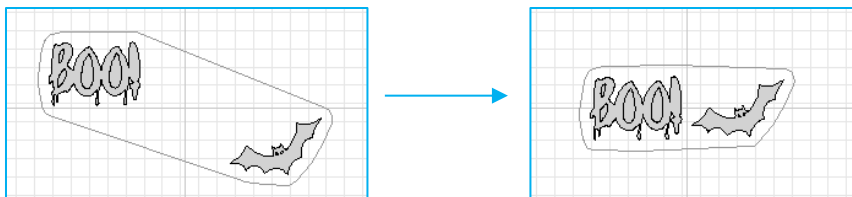
Arrange brings the shapes closer together.

30

4.05.4 Duplicate and Remove Duplicates

- The **Duplicate** function allows you to easily add more copies of the highlighted file. Just click on the green icon to add more copies and click on the red icon to delete copies. Warning, if you only have one left and you click on the red icon you will delete the file from Slic3r and will need to use **Add** to re-import.
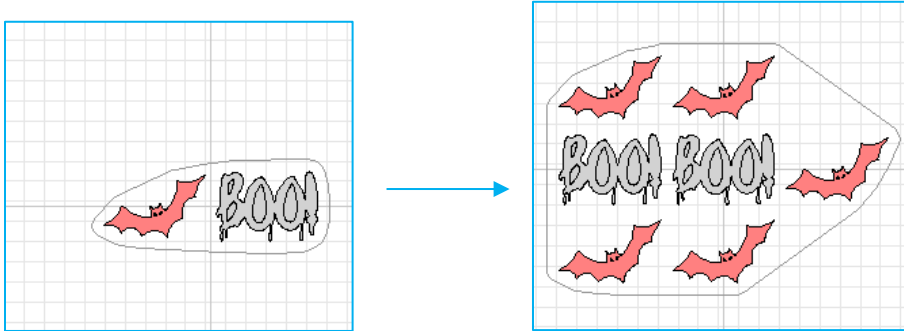
Duplicate was used to add 1 more "Boo!" and 4 more bats.

4.05.5 Rotate

There are three icons used for rotating:

- The first two rotate buttons will rotate a highlighted object in 45° increments, counterclockwise and clockwise as indicated by the icons.

- The **Rotate** button allows you to input the exact degree of rotation desired.

- When there are duplicates, the rotation will be applied to all:
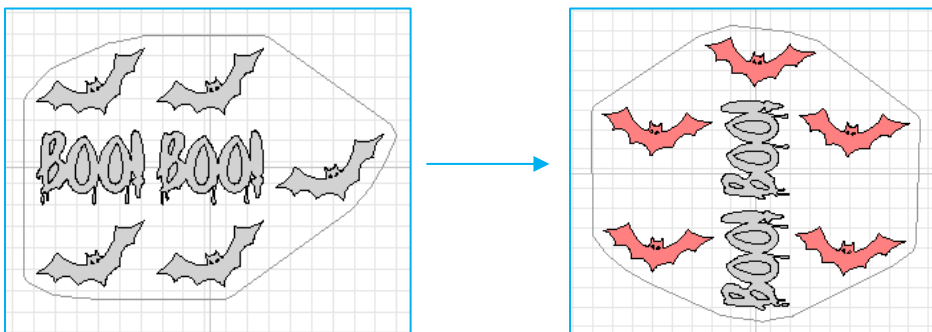
The "Boo!" shapes were rotated 90°counter-clockwise. The bats were rotated 30°clockwise.

- Note that after using this function you may need to manually drag and separate the shapes, in case any are overlapping as a result of the rotation.

4.05.6 Scale
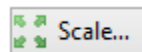
- The **Scale** function is used to resize an imported file. This can be VERY important due to the fact that imported objects into Slic3r are always considered to have been designed in mm.

- For example, if you designed a button that's 1 inch in diameter and ⅛ inch thick, that button will be imported as 1 mm in diameter and ⅛ mm thick! That's a tiny button! In order to resize the button to its original dimensions, click on **Scale** and input 2540:
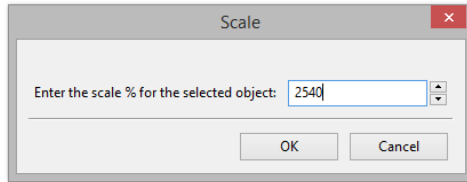
- If you designed the file in cm, then the scaling would need to be entered as 1000.

- If you wish to simply double the size of a shape, then highlight the file, click on **Scale** and enter 200.
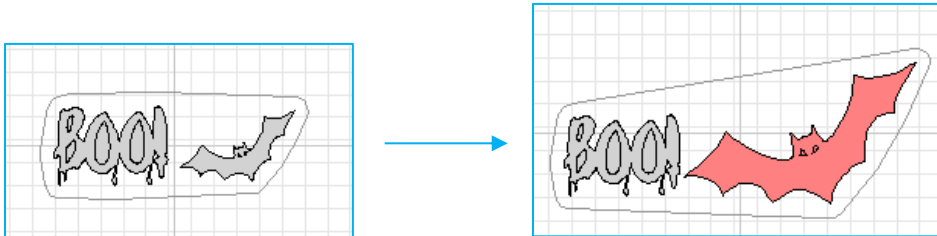
- Note that after using this function you may need to manually drag and separate shapes, in case any are overlapping.

4.05.7 Split

- **Split** is used to separate individual objects within an imported file. For example, let's suppose the bat and "Boo" were created together in a 3D design program and imported as a single STL file. Then let's also suppose you only want to print the bat. By highlighting the file and clicking on **Split**, the two objects are now separate:

Two objects are listed after applying **Split**.
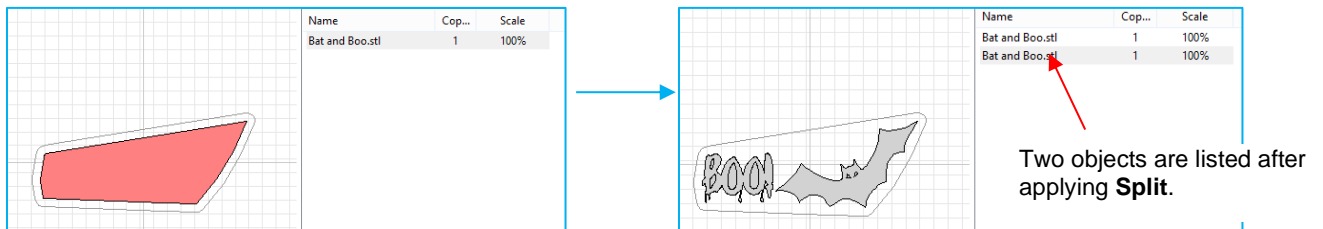
- The same file name is used for both objects since there's no way for Slic3r to know how to name them separately. To determine which is which, highlight one of the names and that object will also be highlighted:
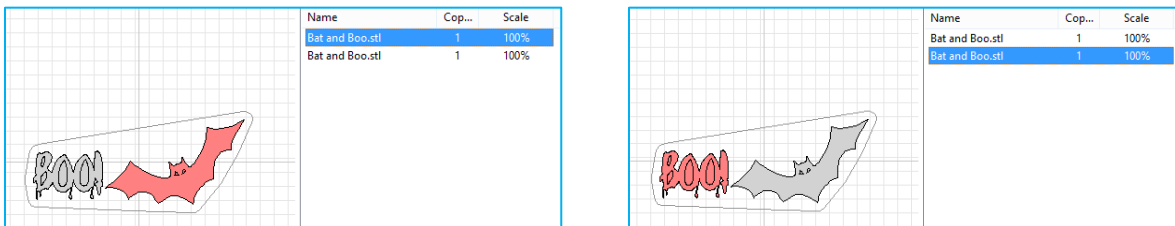
The top listing is the bat; the bottom listing is the "Boo!"

4.05.8 Cut `Cut...`
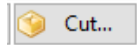
- The **Cut** function has several uses:

  ◊ You can see a 3D representation of the object that is similar to the 3D preview in the main window. Pressing and dragging the right mouse allows you to rotate the shape.

  ◊ You can slice off a desired amount of the object in the Z direction. This could be useful if you wanted to double the X and Y dimensions of a shape but not make it any thicker/taller. For example, when the bat was previously scaled to 200%, it also became twice as thick (increasing from 1.5 mm to 3.0 mm. To return the bat to its original thickness, the following settings could be applied:

Amount to cut off, in mm

Part to keep: Upper, Lower or both

Option to rotate lower part if keeping

Click here to complete the cut.
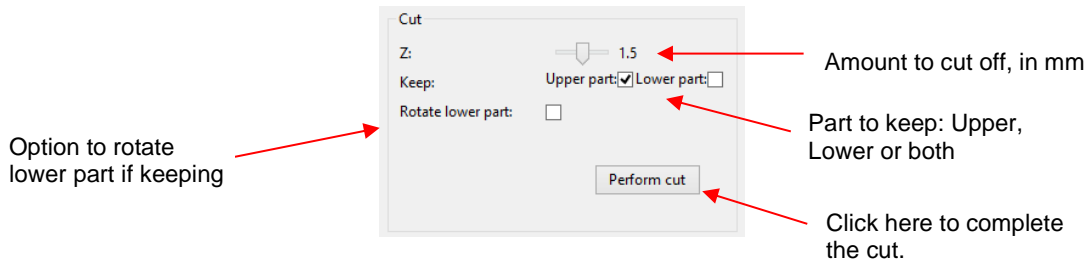
  ◊ Another application for the **Cut** function is when your object is too large for the build plate or shaped where the bottom will not have sufficient contact area with the build plate to insure a good print. The **Cut** function can be used to dissect the file into two or more sections for printing. Then, after the individual parts are printed, they can be glued together. Note that this is sometimes visible in items you purchase where you see a seam that joins the object together. This is the same thing!

  ◊ Yet another application, albeit one for an advanced user, involves cutting an object into two or more sections with the intent of using different print settings on those sections. This involves manually modifying the Gcode file, as well, which is why it's not for a beginner. But remember that the possibility does exist. For more information, refer to this excellent post by Richrap.

4.05.9 Settings `Settings...`
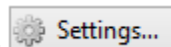
- In the settings function, changes can be made to certain print, filament, and/or printer settings as "one-time changes" only, without altering the current values for any of the configurations. This will be covered in specific printing scenarios later in this manual.

4.05.10 File Info

- To view the info on a file, highlight it first and then information on the file appears in the **Info** section at the bottom of the screen. Note that the **Size** is reported in mm and the **Volume** is reported in cubic mm. To convert to inches, you'll need to use a calculator and divide the dimension under **Size** by 25.4. To convert the volume to cubic inches, divide by 16,387. For cm, divide by 10 and by 1000, respectively.

- If the size of the object is incorrect, use the **Scale** function to change it. Refer to *Section 4.05.6*, under **Scale**.

- The **Manifold** parameter is especially important to check on any new STL file. It needs to indicate "Yes." A **Manifold** that says "No" will not export because there are issues with the design. You can

attempt to repair the model by going to **File>Repair STL File** and selecting the file to be imported again.

4.06 Important Menu Functions

- Most of the menu functions are just other ways to execute functions covered by the icons on the screen. There are, however, a few that are worth noting separately:

4.06.1 Export Configuration

- As covered in *Section 4.04*, if you create new profiles, you should back them up by using **File>Export Configuration.**

4.06.2 Repair Stl File

- As mentioned in *Section 4.05.10*, if the Manifold status says "No", then attempt to fix the file using **File>Repair STL File**.

4.06.3 Flip

- Occasionally you may need to flip a file, especially along the Z Axis. This will be evident when an imported STL file will print more successfully if it has a larger flatter side that is displaying as being on the top, versus the bottom:

- For example, let's say you imported a chair file and it appeared like this:
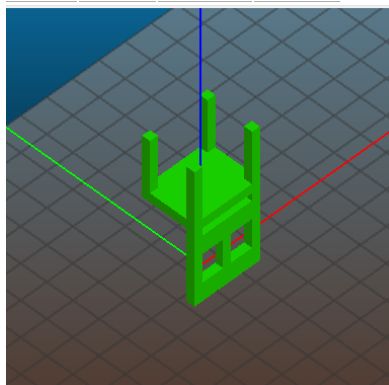
- Obviously, the chair is not going to print well in this position, even if the support material option is used. To flp the chair, go to **Object>Flip>Along Z Axis**. Now the chair is positioned on its four legs:
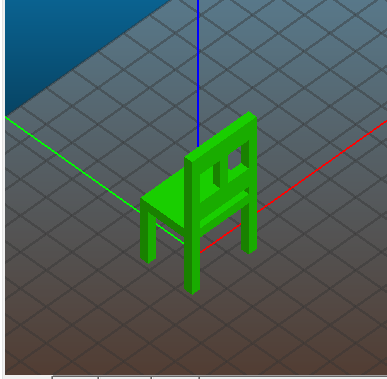
- Further, if you want the chair to print facing you, then apply **Object>Flip>Along Y Axis:**
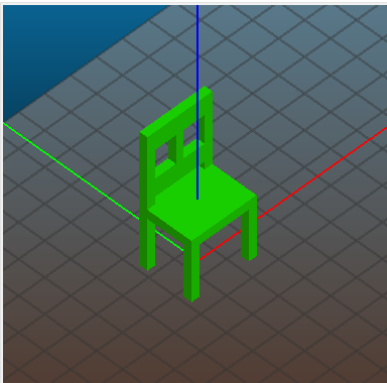
4.06.4 Export Object as STL

- If you modify the object, such as was done with the chair in the prior subsection and wish to save it in STIL format, go to **Object>Export Object as STL**.

4.07 More Information on Slic3r

- The Slic3r User Manual contains more details on functions and settings, as well as some useful general information about 3D printing.

- *Chapter 5* will cover some of the settings needed for specific applications.

- *Appendix B* contains screen shots from the submenus of the **Print Settings**, **Filaments Settings**, and **Printer Settings** tabs. Be sure to reference these screenshots when deciding upon changes.

5. More About Printing

5.01 Important – Read This!

Here's a list of the **key things** to remember when printing:

- The panic button is the **left arrow key** on the control panel. If something goes awry during a print, press it and an option to **Cancel** the print will appear:

After pressing left arrow key, option to **Cancel Print** appears.

- As shown in the screenshot above, there is also an option to pause a print. Refer to *Section 5.03* for instructions on when and how to use this function.

- Do not remove the SD card during the printing process. The print will be interrupted. This is the screen that will appear:
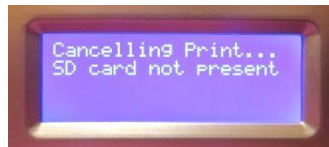
- When removing a roll of filament, try to keep the filament taut on the roll and tape the loose end of the filament to the spool. Allowing the filament to uncoil can later result in feeding issues when that filament is used again.

- PLA and ABS require different temperature settings. Make sure you select the correct profile under **Filament Settings**. More information on printing with ABS is covered in *Section 5.07.*

- Certain routine maintenance functions are recommended to keep your printer in top working order. Please check out these functions in *Chapter 7*.

5.02 Preheat

- Because there's an option to **Preheat**, one might think it's a requirement before every print. But it isn't. Whenever you send a file to print, the KNP3D will automatically preheat the extruder(s) and the build plate before beginning the print.

- One use for preheating is to save a little time. While you are finalizing your file in Slic3r, you can go ahead and begin preheating the KNP3D. It will be closer to target temperature once you are ready to load your file from the SD card.

- **IMPORTANT:** Do not use the **Preheat** function for an extended period of time as it can cause a filament clog inside the extruder.

- To initiate preheating, select it from the main menu:
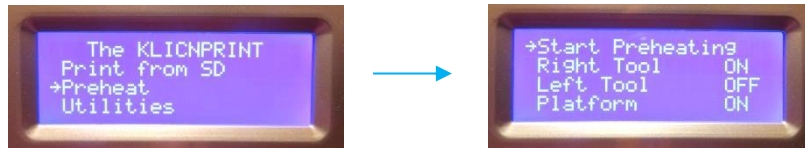
- If you wish to preheat the **Left Tool**, then move down in the above menu and press **K** to turn on the left extruder.

- Once you select **Start Preheating**, the screen will change to show the current temperature and target temperature:

Current and target temperatures are shown during preheat.

No target is shown for **L Extruder** because it was set to **OFF**.
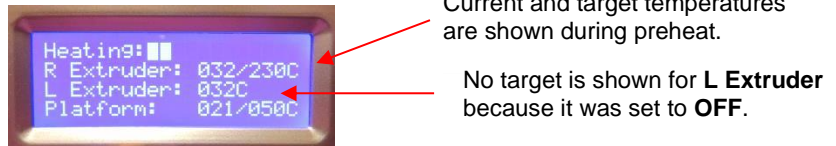
- To cancel the preheating, press the **Left** arrow key on the control panel to exit this window and go back to the main menu. Select **Preheat** again and then select **Turn Heaters Off**.
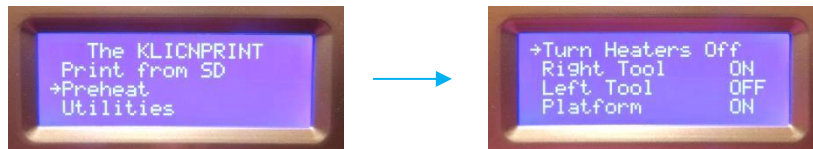
5.03 Pausing the Print

- There may be a reason to pause a print, such as the filament running out or discovering a small thread of filament caught in the object. The KNP3D can easily have printing paused and resumed. There are three kinds of pause modes available and are presented in this section.

5.03.1 Pause

- The **Pause** mode is used when you need to change a filament spool or, perhaps, remove a stray thread of filament from the build plate.

- At any time during the print, press the left arrow key on the control panel to bring up the following menu where you can scroll to and select **Pause** by pressing **K:**

Select **Pause** and the extruder will move to the back right corner while the build plate drops down.

- If you select **Pause** the following screens appear:
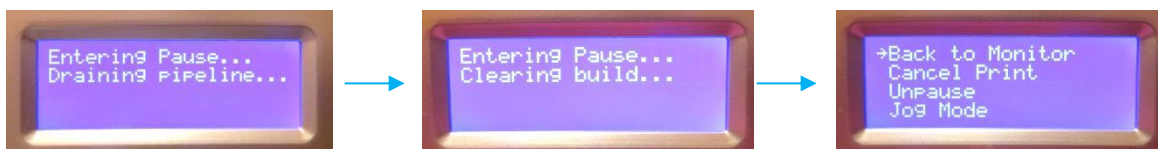
These screens appear as the extruder moves to the back right corner and the build plate drops.

This screen then appears. (This is just the first page of three.)

37

- In **Pause** mode, you have a number of options, as shown above in the far-right photo. Scrolling down will take you to second page of options. These options are covered in other sections of the manual, as relevant.

- Once you are ready to resume printing, select **Unpause** and a new screen appears as the extruder resumes its heating position (front left corner) and the build plate raises:

Scroll down to **Unpause** and press **K.**

- Then you will see the screen showing heating being resumed. Once the extruder(s) and build plate are at target temperatures, printing will start again from where it was paused.

- Note the **Pause** can be modified to keep the heat turned on (although this is not typically recommended. If you do want to keep the heat on, refer to *Appendix C.*

5.03.2 Pause at Zpos

- **Pause at Zpos** allows you to have the print stop at a certain Z location (height). This could be used if you are printing, say, a three-color design where a change in filament will be required at some point. It can be a little tricky to use because you need to know the exact Z axis location and then also take into account that you are starting at your calibrated build level which might be something like 0.3 mm.

- Once the print has begun, press the left arrow on the control panel to bring up the following menu where **Zpos** can be scrolled to and then selected by pressing **K**:

Selecting **Pause at ZPos** will stop the printer when it reaches a preset location on the Z axis.
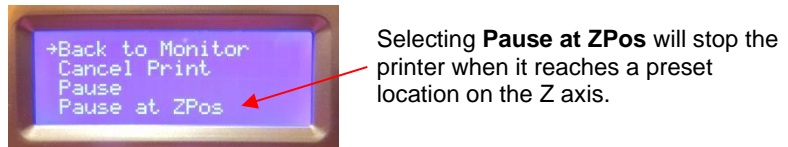
- The following window will open indicating the current Z axis location. Change that setting to the location where you want the KNP3D to pause the print. Naturally, this will need to be a higher value than what's shown in this window because a lower location has already been passed.

Current Z Axis location:

Change to a higher location on the Z Axis and the printer will go into **Pause** mode at this set point.

Use the **Up** button to increase the position.

Use **Left** arrow to exit without changes; use **K** to set new **ZPos** position.
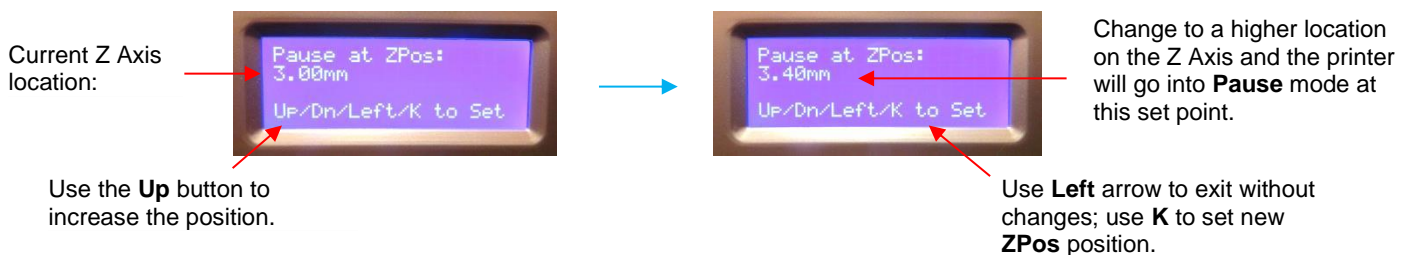
- When the build plate reaches the **Zpos** setting, the same thing happens as in the **Pause** mode. The extruder head moves to the back-right corner and the build plate will drop.   When you are ready to resume printing, select **Unpause**.

5.03.3 Cold Pause

- **Cold Pause** is used to pause the printing process for an extended period of time. The KNP3D will remain powered on, however all heaters, stepper motors, and LED lights will be shut down.

- To access the **Cold Pause** option during a print, press the left arrow to bring up the following menu. Press the down arrow repeatedly to scroll to the third page of this menu. Then select **Cold Pause**.

Scroll down

Press **K** to activate a **Cold Pause**.

- The **Cold Pause** will then proceed in the same manner as the regular **Pause**, other than you will see the LED lights turn off. You will have access to the same menu items.

- Once you are ready to resume printing, select **Unpause** and everything will proceed as described in *Section 5.03.1*.

5.04 Changing Temperature and Speed During a Print

- Being able to change the extruder temperature and the speed in the middle of the printing process can be very handy. Perhaps you meant to change the temperature in Slic3r before exporting the file. Perhaps you see something going wrong and want to make a change and try to save the print. Perhaps you just want to experiment with some changes without going back to Slic3r.

5.04.1 Changing the Speeds During a Print

- Once the print has begun, press the left arrow to bring up the following menu. Then use the down arrow to scroll to the second page. Select **Change Speed**:

Scroll down

Press **K** to enter **Change Speed** screen.

- The speeds can be increased or decreased in 5% increments. For the speed change to take effect, you must press **K**.

Press **Up** arrow twice to increase speeds by 10%.

Use **Up/Down** arrows to increase or decrease speed.

Use **Left** arrow to exit without changes, use **K** to set new speeds.
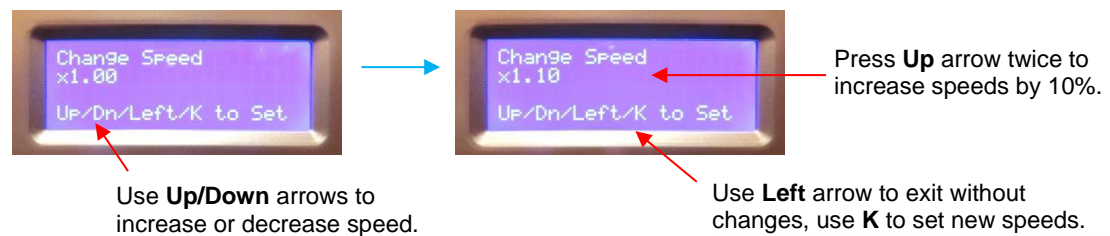
- You can always reenter this screen during the print and return to the original set speed or make another change.

5.04.2 Changing the Extrusion Temperature During a Print

- Once the print has begun, press the left arrow to bring up the following menu. Then use the down arrow to scroll to the second page. Select **Change Temperature:**

Scroll down

Scroll to and press **K** to enter **Change Temperature** screen.

- The temperature can be increased or decreased in 1°C increments. For this change to take effect, you must press **K**. You can always reenter this screen during the print and return to the original set temperature or make another change.

Use **Up/Down** arrows to increase or decrease temperature.

Press **Down** arrow five times to decrease temperature by 5°C.

Use **Left** arrow to exit without changes, use **K** to set the new temperature.

5.05 Unloading Filament While Not Currently Printing

- There are several reasons why you will need to unload the filament at some point:
  - ◊ Changing to a new or different spool of filament
  - ◊ Not planning to use your KNP3D for an extended period of time (remove filament and store in a dry location)
  - ◊ Needing to unclog the extruder.

- Unloading filament is very similar to the process of loading it. In fact, you want to begin by using the loading process so that any previously heated (swollen) filament up inside the extruder is pushed out and the extruder gear will have fresh unheated filament to grip when it begins to unload.
  - (1) Go to **Utilities > Filament Loading > Load right** and once the target temperature is reached and the little tune is heard, you should see filament streaming from the nozzle, unless the nozzle is clogged. Press **K**.
  - (2) Disconnect the guide tube from the top of the extruder by pulling the plastic tubing out of the hole in the acrylic plate.
  - (3) On the control panel, go to: **Utilities > Filament Loading > Unload right:**

  - (4) The extruder head will again begin heating up to 230°C. This will be displayed on the control panel and the progress bar will be displayed.

(5) When the extruder reaches the set temperature, the KNP3D will play a little tune, prompting you to load or unload the filament. The feeder inside the extruder will begin to push the filament back out. The display will show:

(6) If you are trying to clear a clog and get the filament flowing again, then immediately go back to **Loading Filament** on the control panel and proceed with those steps. Note that you may need to repeat the unloading/loading process several times.

(7) Otherwise, press down on the extruder lever on the side of the extruder and gently pull the filament out of the extruder.

(8) Press **K** to exit.

(9) Pull the filament out of the guide tube and secure to the spool using a piece of tape. You do not want the filament to unravel from the spool as this can cause feeding issues the next time that spool is loaded.

(10) If changing to a different color, remember that there will be residual filament in the extruder nozzle. Allow sufficient time during the loading of the new filament for the prior color to be fully extruded. If desired, a cleaning filament can be used before loading a new filament to not only ensure the prior color is removed, but also to keep the inside of the extruder free of clogs.

5.06 Changing Filament During a Print

- If you are running out of filament during a print or decide to change colors and the file hasn't been set up to use the left extruder, then you can access the **Load Filament** function using this method.

- Press the left arrow button during the print and then select the **Pause** option, as shown at the start of *Section 5.03.1.*

- The extruder head will move to the back-right corner. This position makes it a bit awkward to unload and load filament. Thus, select the **Jog Mode** first so that you can move the extruder to a more convenient location. Don't be concerned about losing the current position. It is stored in memory and will be used when the printing is resumed. Just make sure the nozzle is not positioned over the object you've already printed as there will be filament streaming down after the new spool is loaded.
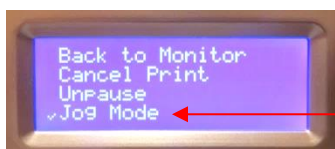
Scroll down to **Jog Mode** and press **K**.

- In **Jog Mode,** you use the X and Y screens to move the head to a different location:

X control window – press right button to go to Y control window.

Use the up/down buttons to move the extruder left and right.

Y control window – press **K** when ready to return to **Pause** menu.
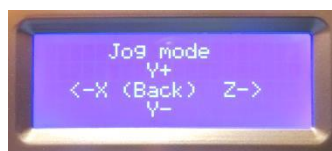
Use the up/down buttons to move the extruder forward and back.

- Once the extruder is in the desired location, press **K**. Then press the down arrow to go to the second page of the menu and select **Filament Loading**.

Press **K** to go to **Filament Loading**.

- Proceed with the instructions in *Section 5.05* for unloading the current filament. Then load the next filament using the steps in *Section 2.05*.

- Once the second filament is streaming from the nozzle, press **K** to stop the loading. Use a folded lint-free cloth to gently wipe away any filament hanging from the extruder nozzle.

- Press the left arrow button three times to return to the original menu where you can then select **Unpause**. The printing will now resume once the printing temperatures are reached.

5.07 Printing with ABS and Other Filaments

5.07.1 ABS

- ABS (Acrylonitrile Butadiene Styrene) is a petroleum-based plastic that has more flexibility than PLA. Thus, it is often a preferred choice when making objects that interlock with one another.

- ABS requires higher temperatures for printing. The preheat extruder temperature can remain at 230°C but the build plate preheat should be increased to 110°C. The temperatures for printing will be around 230 - 240°C while the build plate should be at 110°C. ABS is also vulnerable to warping due to air drafts in a room. You may need to even enclose as much of the printer as possible and there are some Your Tube videos and blog posts to provide suggestions.

- This higher heat requirement also means that finished ABS objects can be used in higher temperature environments than PLA objects.

- ABS emits a strong odor while heated. It can also emit toxic fumes if heated too high! Make sure your room is well ventilated.

- ABS doesn't print quite as smoothly as PLA and also has a propensity to warp or curl while on the build plate. It is recommended that KNK USA's Plastic Heat Bed Tape be used as the surface for printing versus the Paper Heat Bed Tape used for printing PLA.

- The profile bundle you initially downloaded contains a **Filament Settings** profile for ABS. Thus, on the **Plater** tab, be sure to switch to *"ABS Settings"* on the right side for either or both extruders, as applicable:

Change to *"ABS Settings"*
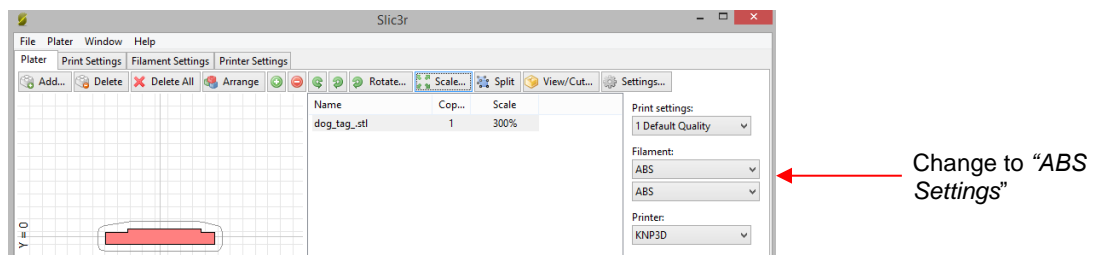
- As with PLA printing, finding the best combination of settings for a particular object, may take several experimental prints. You will most likely need to set up different **Print Settings** profiles for your ABS projects.

5.07.2 PVA Support

- PVA support material uses slightly lower printing temperatures than PLA. Click on the **Filament Settings** tab and change the profile for PLA to:
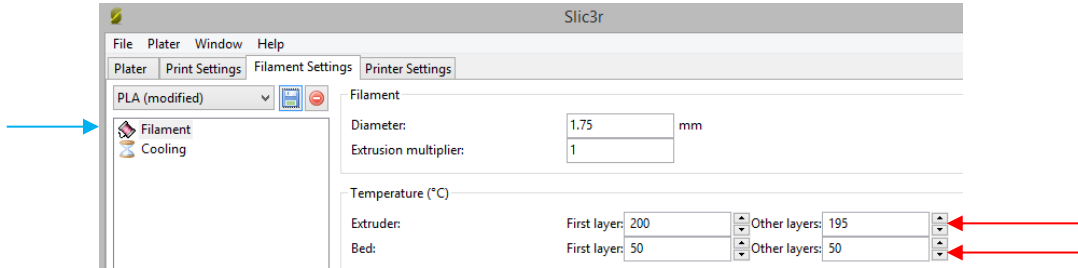
- <u>**IMPORTANT**</u>: Remember to click on the **Save** icon, but this time call these settings "*PVA*". Refer to *Section 4.04* for details on saving a profile:
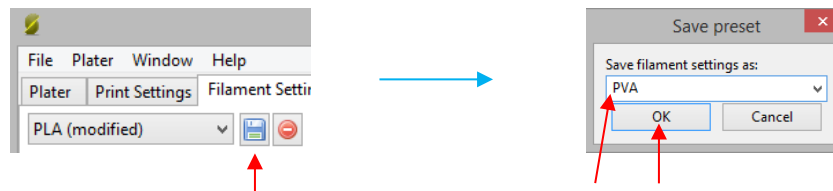
Click here to save the new **Filament Settings**

Input a new name and click on **OK**.

- Then, on the **Plater** tab, be sure to switch to **PVA** on the right side. Since PVA is often used as a support material for PLA, it is chosen, in this screenshot, for the second extruder only. For more information on <u>using the left extruder to print support material</u>, refer to *Section 5.10.*

Change to "*PVA* "
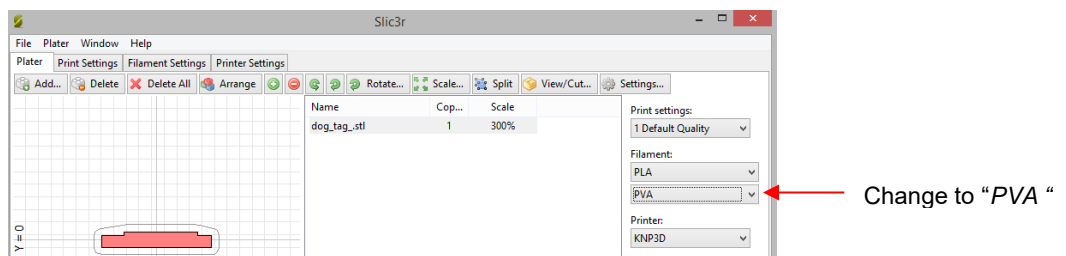
- Save these new settings by going to **File > Export Config Bundle,** and select "*My KNP3D Profiles*". These settings will be added to the same file with the other profiles you've downloaded into Slic3r and created yourself.

5.08 Printing Duplicates

There are several ways to print copies of your objects:

- Option 1: Select "Print Another Copy" from the screen that immediately follows the end of the printing process. This is the only option available for duplicating objects that are larger than half the size of the build plate since you wouldn't be able to fit two objects without overlapping them.

Select this option to have KNP3D immediately begin a repeat of the last print. →

- Option 2 – If you've already turned off the KNP3D, then just turn it back on and select the file again from the SD card.

- Option 3 - In Slic3r, use the **Duplicate** function to add copies of your object before exporting as a .gcode file. (Refer to *Section 4.05.4.*) The objects need to be small enough to fit within the boundaries of the build plate which are indicated by the gridded area on the **Plater** tab. Export the gcode file, convert to x3g, and now that file will print the duplicates as they are laid out on the grid.

- Option 4 – For really small shapes, the two extruders can print at the same time, making two copies of the same file. To do this, go to **Utilities > General Settings > Ditto Printing.** Use the Up arrow to change **Ditto Printing** from **OFF** to **ON**:

> **IMPORTANT**: don't forget to turn off **Ditto Printing** in case your next project is much bigger!

5.09 Skirt and Brim

- The **Skirt** function (under **Print Settings > Skirt and Brim**) prints an outline of your object at a set distance from the outside perimeter. The purpose of the **Skirt** is to make sure the filament is flowing smoothly from the extruder before the actual print begins. If there isn't a clean outline after the two loops are completed, stop the print by pressing the left arrow button to bring up the menu for canceling. Typically, it is recommended that 2 loops are used with a distance of about 6 mm from the object.

- The **Brim** function is used whenever an object has limited surface area contact with the build plate. **Brim** will add a set number of mm of material to connect the object to the build plate to provide stability. The usual recommendation is 4 – 6 mm.

5.10 Support Material

- Support material consists of thinly added layers to prevent collapse or even just sagging during a print. For example, if you are printing a chair or a swing set, or even the St. Louis Arch, there are large portions of the object that are "hanging over air".

- Once these types of objects are printed, the support material is removed using wire-cutters or other tools. (**IMPORTANT**: wear safety glasses and cut/puncture resistant gloves!) Some owners will use a dissolvable filament for the support material.

- When downloading 3D print files from the Internet, always check to see if the designer has mentioned whether or not support material has already been built into the design. If not, then Slic3r provides features to add support to a file.

- To use the **Support** function in Slic3r, click on the **Print Settings** tab and then, on the left side, select **Support Material**. Then check the box next to "**Generate support material**":
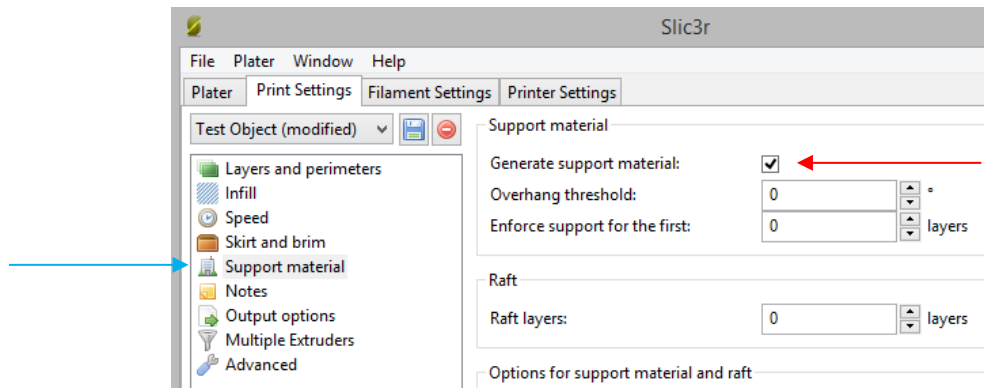
- Slic3r will then determine where support material might be needed for that design. After exporting the .gcode file, you may wish to turn off that setting or you can save it as a new print setting and name it something appropriate.

- A **Raft** will build a foundation beneath the object and is an alternative to **Brim**, covered in the prior section.

- If you are planning to use an alternative material (or a different color) from a spool on the left side extruder:

  ◊ Go to **Print Settings > Multiple Extruders** and make the following changes:

Will print support using left side extruder.

Option to turn on **Ooze Prevention**: Refer to *Section 5.11.3*
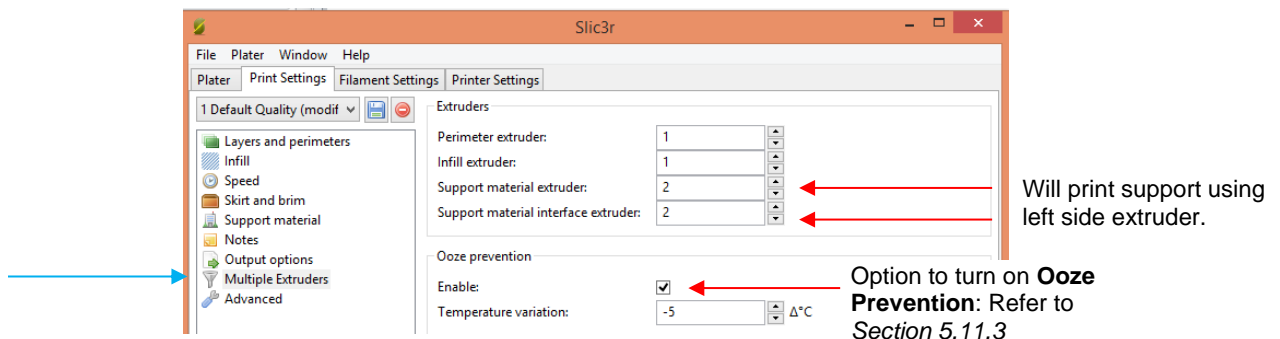
  ◊ Save the change under a new profile name, such as *"Default Print with Support"* (or whatever you prefer). Refer to *Section 4.04* for details on saving profiles.

  ◊ Go to **Filament Settings** > **Filament** and create a new profile if using an alternative filament for support. Change the temperature settings based on what is recommended for that particular support material. For example, PVA needs a slight lower temperature than PLA (Refer to *Section 5.07.2*).

  ◊ Proceed with the **Export G-code** step, making sure you have the correct profiles selected. Then perform the conversion to **.X3G.**

5.11 Two Color Printing

- To use both extruders during a print, it will be necessary to combine two .STL files into a single file. But even before doing that, the two files need to have the objects positioned so that, when combined, the different parts will print in the correct locations relative to one another. This will need to be done in the original design software.

- The following two test files can be downloaded and used as examples of this process:

<div align="center">

Inside Cylinder

Outside Cylinder

</div>

- The final object is a two-colored cylinder. Below are screenshots of the two .STL files and then the final combination:
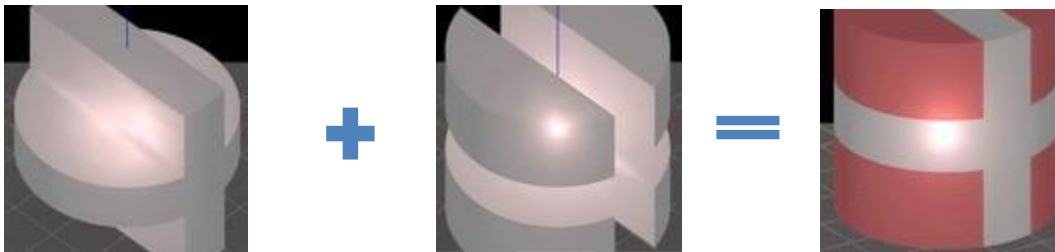
5.11.1 Generating the .AMF.XML File

When merging two files, you do not add them separately in Slic3r. Instead, use this process to combine them:

(1) Open Slic3r and go to **File > Combine multi-material STL files.**

(2) A window will open and you can then browse to locate the files you wish to combine.

(3) **IMPORTANT:** Select the file which will be printed with the filament that is loaded into the right/default extruder and then click on **Open.** In this example, the *cylinder-inside.stl* file is selected.

(4) Now you can select the second file (in this example, the *cylinder-outside.stl* file*)* which will be printed with the filament that is loaded into the left extruder. Click on **Open**. Note that these two didn't need to be located in the same folder.

(5) You will be prompted to open additional files. Click on **Cancel**, instead, and a new window will open.

(6) In this window, a name for the merged file is created based on the first file's name but with a new file extension - .amf.xml. Click on **Save**.

(7) Now, you can use the **Add** function to import this new merged file.

5.11.2 Assigning Extruders

The next step is to assign the extruders to the two parts of the file. To do this:

(1) First, select the file (under **Name**) and then click on the **Settings** button:

Then click on **Settings**

Highlight file name
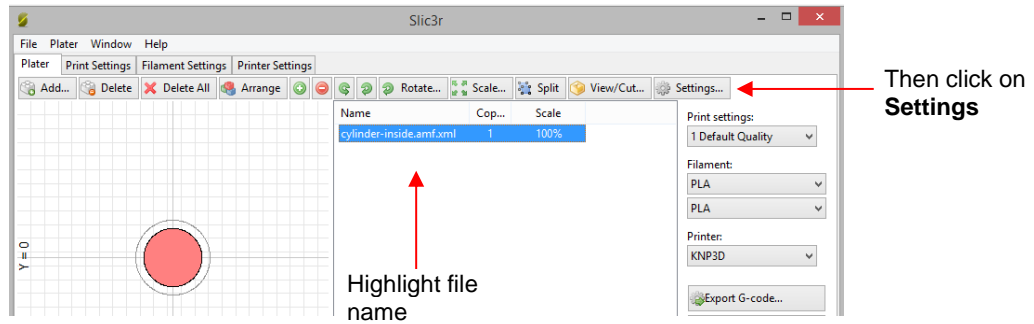
(2) Select the top file, indicated as "0" and you'll see that part turn green in the 3D preview. This helps you to make sure that the part that will printed on the default extruder is correct:

Select the first part

Click here to open a menu of settings

The selected part turns green
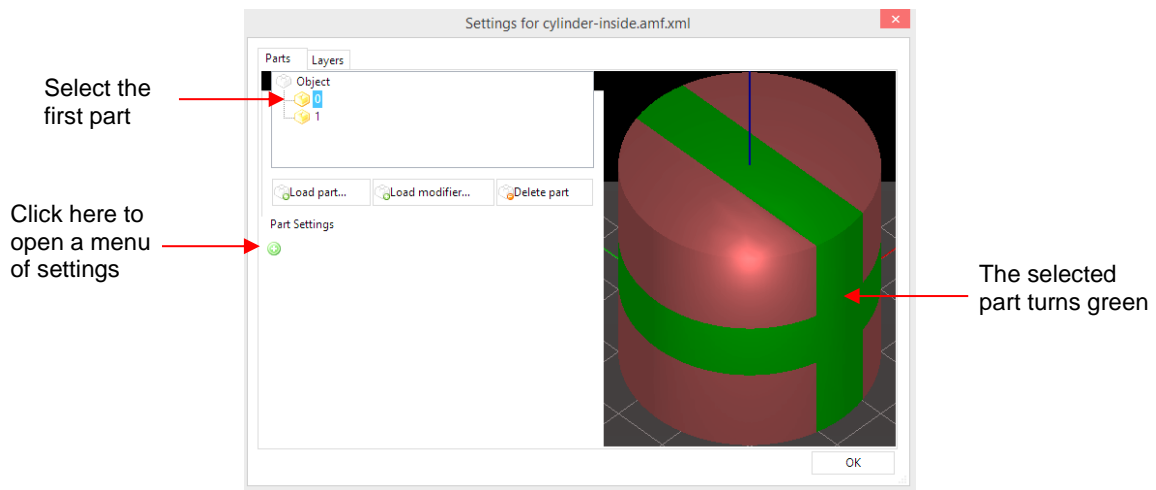
(3) Next click on the little green "+" icon to open a menu of settings and select the top choice, **Extruders > Default Extruder**:

Select the top option
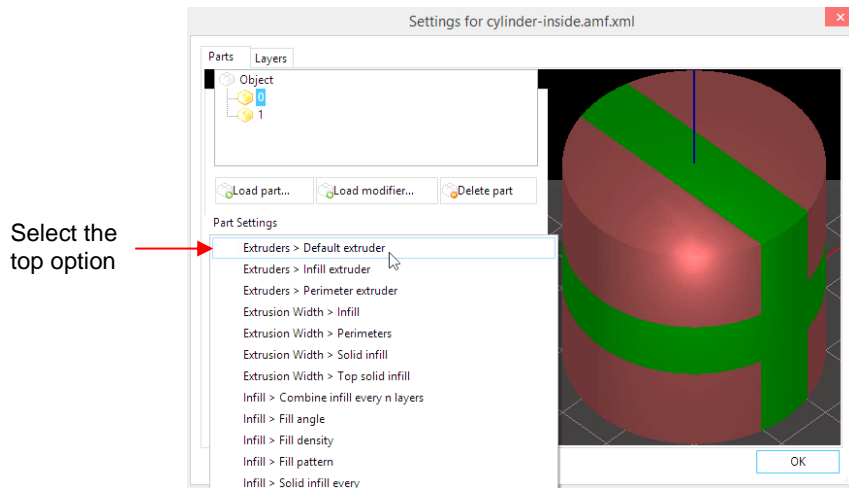
(4) The setting will now appear and the value can be changed to 1 for the right-side extruder:
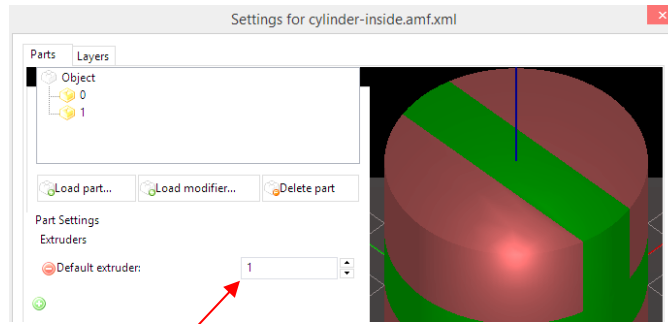
Change from 0 to 1.

(5) Then repeat with the other part by clicking on "1" under **Object**, clicking on the green "+" icon, and then changing from 0 to 2 for the left side extruder.

Second part is selected

Selected part turns green.

Change from 0 to 2.

Click on **OK** when finished.
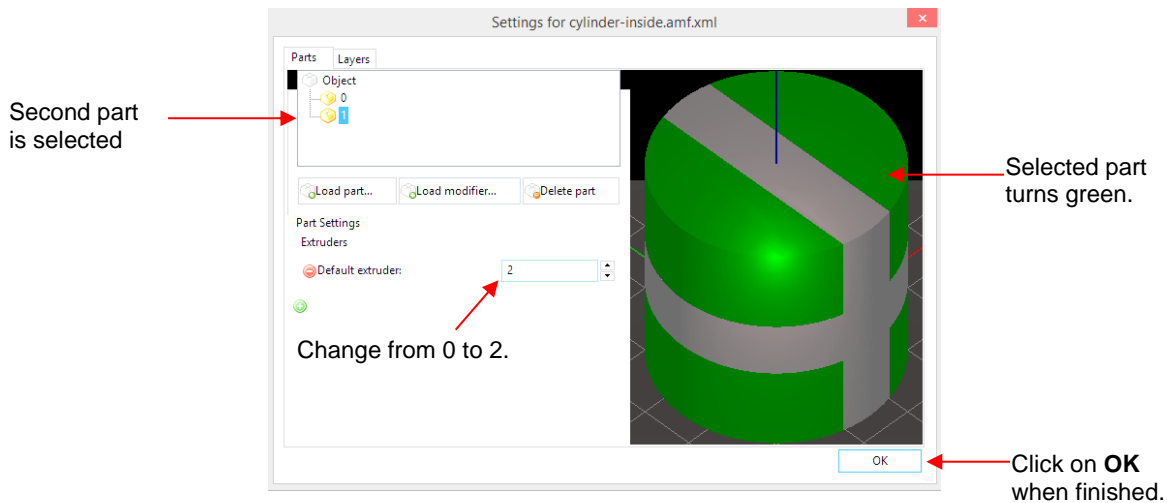
(6) Click on **OK** to have these settings made and to close the window.

- **IMPORTANT:** At the time of this writing, there's no way to switch the assignments of the two parts in order to use the opposite extruders. It will always print the upper "0" (first file in the merge) using the default/right side extruder and the lower "1" (second file in the merge) using the left side extruder. Thus, it's important to plan ahead or you'll need to switch the filament spools on the printer itself.

5.11.3 Ooze Prevention

- When using both extruders it's advisable to enable **Ooze prevention**. This setting does two things:
  ◊ Lowers the temperature of the idle extruder so that it doesn't leak filament while the other extruder is printing.
  ◊ Adds a skirt layer each time it changes to the default extruder. This "tall skirt" will then catch any extruded filament from either nozzle and keep it from sticking to the object being printed.

- To enable **Ooze prevention,** go to the **Printer Settings > Multiple Extruders** and mark the **Enable** box. Leave the **Temperature variation** setting (just below that) at -5.

Check this box.
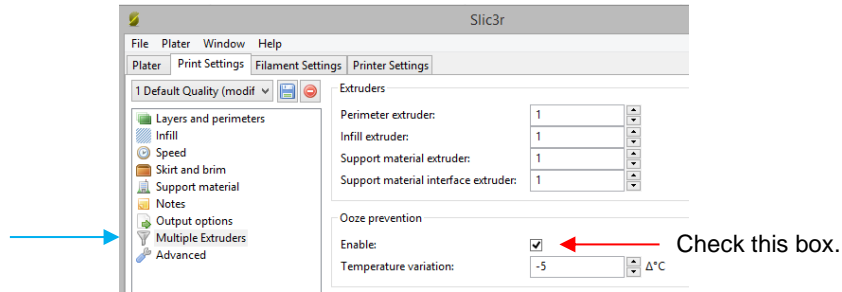
- Save the change under a new profile name, such as "Two Color Default" (or whatever you prefer). Refer to *Section 4.04* for details on saving profiles*.*

- Proceed with the **Export G-code** step, making sure you have the correct profiles selected (note that under **Filaments**, you now need two profiles selected. <u>Do not leave the second one set to "default".</u> Then perform the conversion to **.X3G.**

- During the print, you will notice the skirt is a bit messy and may not stick well to the base plate. This is perfectly normal. Remember that it's the part you'll be tossing out!

5.12 Using the Left Extruder Only

- Let's say you have red filament loaded into the left extruder and blue filament loaded into the right extruder. You want to print an object with red filament only, thus it would be handy to just use the left extruder without having to unload both and then load the blue filament into the right extruder. The original configuration bundle you downloaded contains the profiles for this situation.

- From the **Printer Settings**, select the "*KNP3D – Left Extr Only*" profile which has the required change under **Custom G-code**.

- Under **Print Settings**, select one of the profiles with "*Left Extr Only*" in the name. There are three options based on whether you want the default, fast, or slow settings. The necessary change under **Multiple Extruders** has been made to direct the left side extruder to be used. Then make any other settings changes that are best for your particular object.

- Click on **Settings** (the tab to the far right), and select the object and then select 2 from the dropdown menu. Then click on **OK**.

Select the object

Change to 2
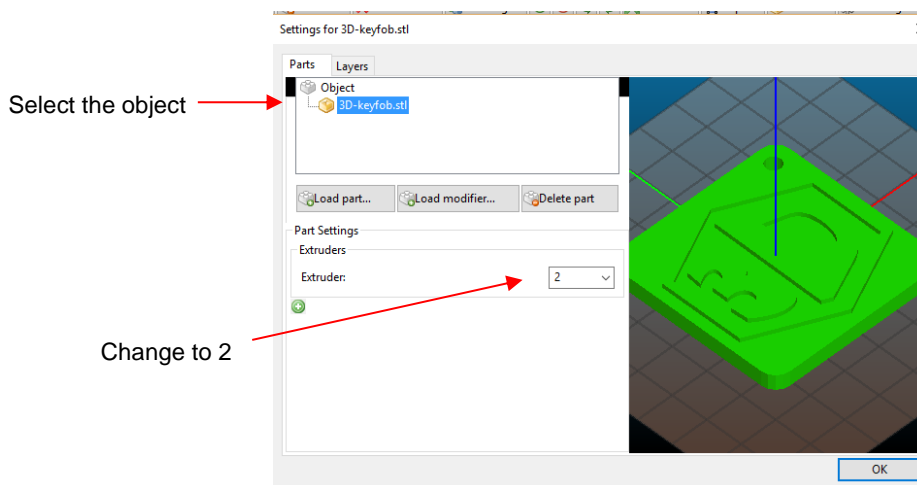
- Before exporting to G-code, make sure you have the correct profiles selected on the right side of the screen:
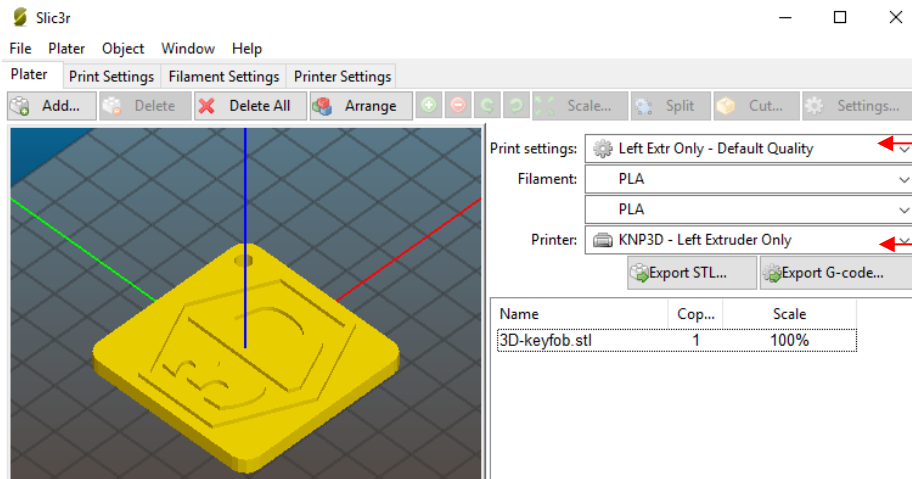
Make sure both the **Print Settings** and the **Printer** are using *Left Extr Only* profiles.

5.13 Printing a Solid Object as a "Vase"

- The **Spiral Vase** feature in Slic3r will print a shape with solid bottom, but otherwise hollow. This is a quick way to turn any solid shape into a vase.

  ◊ **Print Settings > Layers and Perimeters**
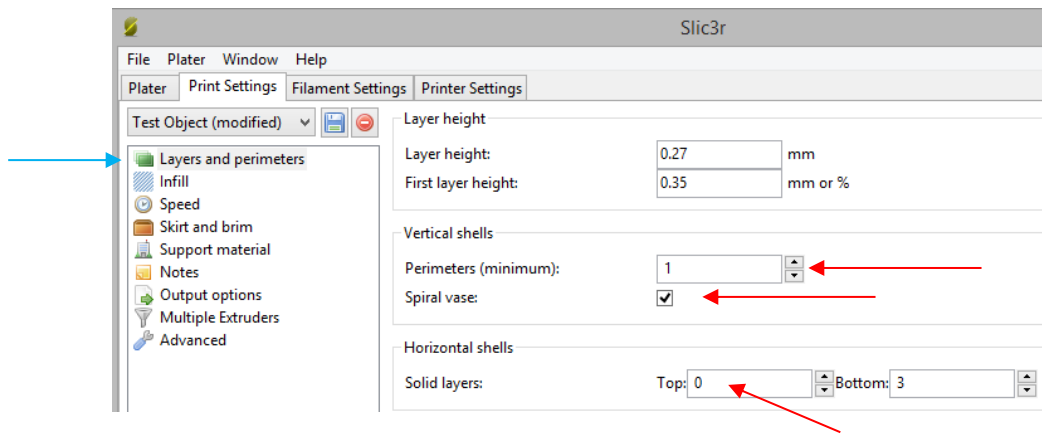
  ◊ **Print Settings > Infill**
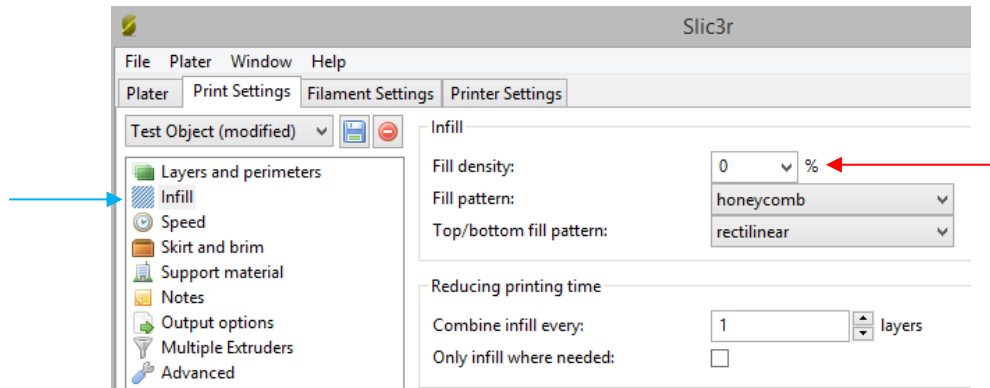
- You should then save these settings under a new profile name, such as "*Vase.*" Refer to *Section 4.04* for details on saving profiles. Then select this profile under **Print Settings** and proceed to export as G-code.

5.14 Organizing Files on the SD Card

- Because only three file names from the SD card can be displayed on the screen, it can become tedious maneuvering through pages of file names. Remember that the list of files will be in chronological order based on when they were saved to the SD card. Thus, when you first view the contents on the SD card, you can use the **UP** arrow to move to the bottom of the list.

- There will be occasions when you want to print a file that's not the most recent one. For that reason, it is recommended that you immediately begin organizing your files into folders and subfolders, so that they will be easier to locate at a later time.

- In the following example, three folders have been set up. Within "My Files", two additional subfolders have been created: "Fun Files" and "Craft Files". Then "Craft Files" is opened to a list of files, as shown.

Top level menu on SD Card   Subfolders under My Files   Files inside "Craft Files"

- One issue about using folders: if you have mistakenly entered the wrong subfolder and **Exit** out, you may find that you cannot get back to the top level. In that situation, just eject the SD card and push it back in. The main menu will be back again.

- You may also quickly learn that file naming choices can be very important. If you are testing changes in settings, then either include the modified profile name in your file name or include a specific setting that's been modified. This will help you track changes you've made and allow you to more easily delete files, later on, that are no longer needed. To prevent file names from being too long, you can also simply add numbers after each file name and then, in your notebook, record what changes are associated with this particular file number.

6. Post Processing

6.01 Removing the Object

- After the printing has stopped, wait a few minutes for the object to cool. If you attempt to remove the shape too soon, it can warp or be damaged.

- To remove the printed object from the build plate, use a sharp flat tool, such as a thin metal spatula or an artist's palette knife. Gently slip the tool under the outer edge of the model and twist. The model should pop off (separate) easily from the plate.

- For shapes with a large footprint, it's advisable to slip the spatula or knife under several spots around the perimeter before attempting to pop it off. **IMPORTANT**: Be very careful to not bump the limit switch that is positioned behind the Z threaded rod.

6.02 Removing Support Material

- Having the right tools always makes jobs easier to complete. For removing support materials, try using needle nosed pliers to twist the support material away from the object. Tweezers can also be used to pick off small pieces.

- Always use safety glasses and safety gloves when removing support material. Both can be purchased for low prices at Harbor Freight Tools.

6.03 Sanding the Object

- Sanding by hand (using regular sandpaper) can be used to smooth rough spots on both PLA and ABS printed objects, although ABS objects are prone to whitening.

- A set of diamond grit needle files also work well for smoothing, especially inside tight corners. They come in different sizes and shapes and a set is very inexpensive at places like Harbor Freight Tools.

- Some owners of 3D printers use acetone slurries to smooth objects printed with ABS. Because of the safety precautions required, directions for this process will not be covered in this user manual.

6.04 Gluing Printed Objects

- Objects cut in half can be glued together using Super Glue or other glues designed for plastic adhesion. It's advisable to print a few small shapes cut in half to test the glue's strength and make sure you are satisfied with the fit and the resulting seam.

6.05 Other Finishing

- Printed objects can be painted with acrylic paint. Alcohol inks also seem to work well with PLA printed objects.

- For an extra shiny finish, check out this product: XTC-3D.

7. Maintenance

We suggest the following procedures in order to keep your printer in good operating condition. Note that the recommended frequency of these procedures is listed in each section. In some cases, the frequency depends on how much the printer is being operated:

H: high usage (printing every day)

M: medium usage (printing once or twice per week

L: low usage (printing occasionally)

7.01 Build Plate Surface Check

- Frequency - check before every print

- Keep the build plate clean and free of wrinkles. After every print, make sure all remnants of filament are removed after the printer is at room temperature. Make sure the tape that is adhered to the plate is wrinkle-free. Sometimes wrinkled tape can be pulled up and then squeegeed back down again using a plastic scraper.

- If the tape on the plate is torn or if wrinkles cannot be removed, replace the tape. Remember that paper tape is recommended for printing PLA and plastic tape is recommended for printing ABS.

- If the build plate has any kind of grease or oil, it can prevent the filament from sticking during a print. If this is the case, it can be wiped down with rubbing alcohol or another degreaser. However, please use the following steps:

    ◊ Turn off the power to the KNP3D.

    ◊ Make absolutely certain the printer is at room temperature.

    ◊ Make sure the degreasing agent is thoroughly wiped off the plate.

    ◊ Do not leave the degreasing agent near the printer. Store it elsewhere before turning the printer back on again.

7.02 Build Plate Leveling/Tramming

- Frequency - **H**: once a week; **M**: once a month; **L**: after every 15 – 20 prints. Also, check any time the printer has been moved.

- Refer back to *Section 2.04* for instructions on leveling/tramming. If you find that adjustments are not being needed, then you can decrease the frequency for checking.

7.03 Extruder Cleaning

- Frequency - as needed

- The nozzle on the extruder can be cleaned using a lint-free cloth. Make sure the printer has had time to cool to room temperature. If there are any strands of filament attached, then you can carefully use tweezers or a craft knife to remove.

- If a clog is suspected:

◊ On the control panel, select **Utilities > Filament Loading > Unload Right** (or **Unload Left**) and allow to run for a few seconds. Then go back and select **Load Right.** This process may need to be repeated several times.

◊ A cleaning filament can also be used to clear a clogged extruder. Use the normal preset temperatures.

◊ If the clog is still present, either contact KNK USA for assistance or refer to this instruction file for disassembling the extruder to clear the clog: Clearing a Clog

7.04 Cleaning the Rods

- Frequency - **H**: once a month; **M**: once a month; **L**: once a month

- Using a clean lint-free cloth, gently wipe both the smooth and the threaded rods inside the printer. Be sure to clean off any buildup around the linear bearings. Do not apply any form of lubricant or cleaning chemicals to the rods unless instructed by KNK USA.

- When cleaning, make sure the axes move smoothly with very little friction.

- The frequency of cleaning may need to be higher if the printer is operated in an environment with open windows or around any other activity generating dust.

7.05 Filament Check and Storage

- Frequency - as needed

- Retain the silica packets which come with every roll of filament. Store them with the opened rolls of filament in a dry, room-temperature storage area, such as a cabinet or drawer.

- Unraveling and subsequent uneven rewinding can lead to filament breakage the next time it is used in the printer. Try to avoid unraveling by either securing the loose end of the filament through the small holes in the spool or by taping the loose end to the spool. If you use tape, be sure to cut off that part of the filament before loading into the printer since any adhesive stuck to the filament could cause clogging in the nozzle.

- It's a good idea to check the filament diameter on any new spool of filament, especially if you have purchased a different brand, different color, or different type. Calipers can be purchased on Amazon or at Harbor Freight Tools. Measure in several locations along the first few feet and then input an average diameter under the **Filament Settings > Filament** screen in Slic3r.

7.06 Checking the Frame

- Frequency - **H**: once a month; **M**: once a quarter; **L**: after every 50 or so prints. Also, check any time the printer has been moved.

- Make sure the bolts which hold the frame together are all still tight. If any can be moved by hand, tighten them.

- To check for proper alignment, print a test square, and verify that all four corners are right angles. Here are two five-minute test files for you to use (one is a 3 inch square and the other is a 75 mm square). Don't forget that the skirt, which will print first, will have rounded corners. Wait for the square itself to print before judging whether or not the printer is out of alignment.

[3-inch-Square](#)

[75-mm-Square](#)

7.07 Miscellaneous

- Frequency - as needed

- Watch for and discard loose filament strands falling onto the floor. These could cause choking in pets and/or small children.

- Keep the inside of the printer dust-free. Wipe down with a lint-free cloth any time you see dust accumulating.

- When cleaning the printer, make sure the belts are in good condition and tight. It is unlikely that belts will loosen before many hours of printing. But it's quick to check while cleaning the rods.

- Back up any important print files on your SD card. Even though the life cycle of a card is about 10 years, you could inadvertently damage it while inserting or removing from your printer or your computer. Or your dog might get hold of it!

> **IMPORTANT:** If you have any questions or concerns about the condition of your KNP3D, please contact KNK USA using the following support ticket link:
>
> [Ticket Request](#)

Appendix A Troubleshooting

The KNP3D starts printing without dispersing any filament.

- *Verify that the filament is loaded properly by going to **Utilities>Load Filament.** Refer to Section 2.05 for additional instructions.*

_____

The Kapton tape on the build plate has torn.

- *Remove the Kapton tape from the top surface only. Note that it is also sandwiched between the build plate and the platform beneath. Do not try to remove it from there. Then replace with paper build plate tape or 3M blue painter's tape (for PLA) or plastic build plate tape for ABS.*

_____

Object will not stick to the build plate while printing.

- *Make sure the build plate is clean and there are no wrinkles in the tape which covers it.*

- *Go through the **Level Build Plate** procedure again (Refer to Section 2.04). The plate may need to be raised slightly.*

- *If your shape has a small size footprint, you may need to rotate it or add a brim or support. Refer to Sections 5.09 and 5.10.*

- *If printing with PLA, make sure you are using a paper build plate tape. If you printing with ABS, make sure you are using the plastic build plate tape and have the build plate temperature raised to 100°C.*

_____

The object isn't filled in well and the skirt is wiggly versus a smooth extrusion.

- *This is indicative of the build plate being too low. Go through the **Level Build Plate** procedure again (Refer to Section 2.04). Remember to check each location several times.*

_____

The skirt around the object is very wide and flat.

- *This is indicative of the build plate being too high. Go through the **Level Build Plate** procedure again (Refer to Section 2.04). Remember to check each location several times.*

_____

The filament is "blobbing up" on the build plate during the print.

- *Go through the **Level Build Plate** procedure again (Refer to Section 2.04). The plate may need to be lowered slightly.*

_____

When the print begins, the build plate doesn't rise and the printing process begins anyhow.

- *Go back to Slic3r and verify that that correct profile is selected. This can happen if the wrong **G-Code Flavor** is selected under **Printer Settings > Firmware Flavor**. It needs to be set to **Sailfish (Makerbot).***

———————————————————————

At the end of the print, the object is stuck to the plate.

- *Use a thin flat metal object, such as an artist's palette knife or a metal spatula to remove the object. Gently pry in several locations around the front of the object to loosen it. Then try popping it off by twisting the metal device.*

- *If necessary, preheat the bed back to 50°C. This should also help loosen the object as it warms up again.*

———————————————————————

After selecting the file to print, the KNP3D goes back to the main menu and doesn't start heating or printing.

- *Contact your KNP3D dealer or KNK USA.*

———————————————————————

The nozzle appears to be clogged with filament.

- *On the control panel, select **Utilities > Filament Loading > Unload Right** (or **Unload Left**) and allow to run for a few seconds. Then go back and select **Load Right.** This process may need to be repeated several times.*

- *A cleaning filament can also be used to clear a clogged extruder.*

- *If the clog is still present, either contact KNK USA for assistance or refer to this instruction file for disassembling the extruder to clear the clog:* Clearing a Clog

———————————————————————

The extruder is just moving around in a tiny circle making a blob. It's not actually printing the object.

- *Go back to Slic3r and check the dimensions of the object. Mostly likely, it was designed in inches whereas Slic4r assumes all imported designs are in mm. Refer to Section 4.05.6 to use **Scale** to convert the design.*

———————————————————————

There is a clicking noise coming from the extruder as it is printing.

- *This indicates the filament is not melting to a high enough temperature for proper extrusion. First try increasing the temperature in Slic3r under **Filament Settings > Filament**. But do not exceed 220°C for PLA or 260°C for ABS. Otherwise, decrease the printing speeds (go to **Print Settings > Speed**) so that the filament has more heating time inside the extruder.*

———————————————————————

The tape on the build plate won't stay down during the print. It curls up after a few minutes of printing.

- *Cut the tape a little larger than needed and then pinch the excess at the corners, just beneath the plate. If the tape is already cut to size, then cut some masking tape strips to fold around the corners and then pinch those ends, just beneath the plate. The masking tape binding should prevent the build plate tape from curling up.*

- *With PLA printing, where the bed temperature is only ~50°C, you can use masking tape to wrap around the sides (or at the corners) and press to the bottom of the build plate. Remove the masking tape if printing with ABS (due to the higher build plate temperature).*

_____

The left nozzle is higher than the right one <u>and</u> it's resulting in printing issues on the left side.

- *Contact KNK USA and we will assist you in making an adjustment to that nozzle.*

_____

The filament from the left nozzle isn't sticking to the plate but the filament from the right side is sticking well.

- *Go back to Slic3r and verify that you have the correct profile selected for the left nozzle, on the right side, under **Filaments**. Do not leave it set to "default."*

_____

Appendix B Slic3r Settings

- There will be situations in which changing settings within Slic3r is needed. This appendix contains screenshots from the various settings windows in Slic3r with an indication of which settings are typically adjusted.

- As you try different settings, you may wish to save these as new profiles, especially if you plan to use them frequently or if you will want to return to the original profile before the changes were made. The directions for saving a profile are in *Section 4.04.*

- Remember that when saving changes you may or may not want to overwrite an existing profile. It is often advisable to create new names based on the type of object being printed.

- The blue arrows indicate where to locate the settings window in Slic3r. The red arrows indicate settings to consider changing based on a particular situation. The number next to a red arrow corresponds to a number in the notes below that screenshot.

- In the note sections, comments are followed by initials in parentheses. These initials correspond to the source for the recommended settings, such as (KU) means the recommendation came from KNK USA. A list of these sources is below. If you find other sources that you feel should also be included, please submit them to sandy.mccauley@gmail.com

- In the note sections, a listing containing slashes: X / Y / Z means that those settings are recommended for high quality / medium quality / low quality printing.

- As with all settings, you may find your own combinations that work well! We welcome your sharing those settings at our user forum: http://knkusa.com/forums/.

Resources for the Slic3r Settings

- (KU): KNK USA (info supplied by Ron Smith and Chad Youngblut)

- (SM): Slic3r User Manual - http://manual.slic3r.org/

- (AR): Alessandro Ranellucci's (founder of Slic3r.org) presentation at a 2013 workshop –

   https://www.youtube.com/watch?v=i4zPHULdm9k

- (CB): Charles Bell's book: Maintaining and Troubleshooting 3D Printers, sold by Apress

- (JH): Joan Horvath's book: *Mastering 3D Printing*, sold by Apress.

- (PL): 3Dprintedlife video on Slic3r settings – https://www.youtube.com/watch?v=o1HPeovBclc

- (RR) Richraps's three posts on "Slic3r is Nicer" –

  ◊ http://richrap.blogspot.com/2012/01/slic3r-is-nicer-part-1-settings-and.html

  ◊ http://richrap.blogspot.com/2012/01/slic3r-is-nicer-part-2-filament-and.html

  ◊ http://richrap.blogspot.com/2012/01/slic3r-is-nicer-part-3-how-low-can-you.html

- (TB): Team Budmen various blog posts – http://blog.teambudmen.com/2013/

B1 Print Settings

- **Print Settings** are related to the print itself, such as the layer height, fill density, and a host of different speeds. This tab will be used each time a different type of printing is needed, such as changing the quality of the print or adjusting settings related to the type of model.

**Print Settings > Layers and Perimeters**
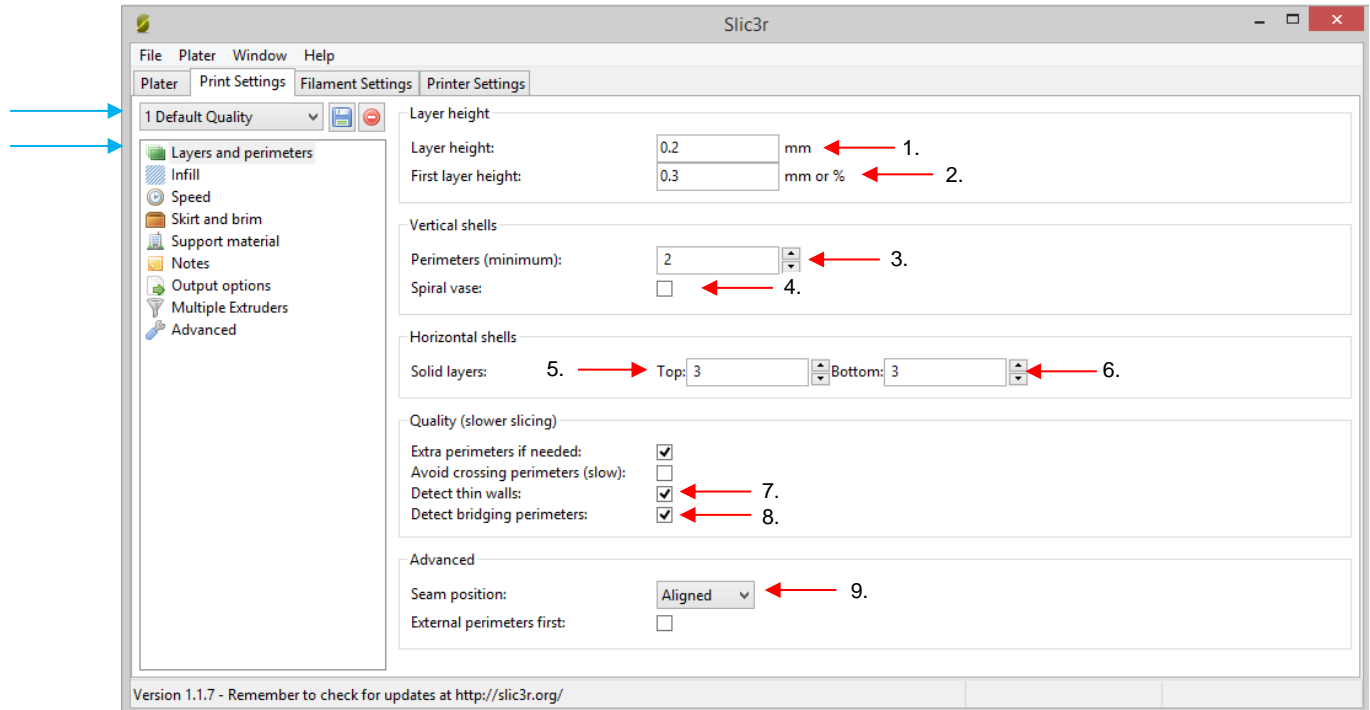
1. Use 0.1 / 0.2 / 0.3 (KU); maximum at **Nozzle Diameter**: use 0.4 mm (SM); use larger for prototyping / fast printing; smaller for display objects or more accurate tolerance (TB); Use an even multiplier of the Z motor step size, must be < **Nozzle Diameter** (JH)

2. Use 0.15 / 0.3 / 0.35 (KU); should be ≥ **Layer Height** but maximum = **Nozzle Diameter** (AR)

3. Use 2 unless printing a very thin object; then use 1 (KU); Use ≥ 2 (SM); Use 1-3 (AR) (RR); use 2-3 (PL); increase for strength; decrease for prototyping / fast printing or for decorative objects (TB)

4. Refer to *Section 5.13*

5. Increase to 4 to improve top layer appearance (KU); use ≥ 2 unless printing with **Spiral Vase** (SM); use 3 (RR); use 3 – 4 (PL); Increase for strength; decrease for prototyping / fast printing or for decorative objects (TB)

6. Use ≥ 2 (SM); use 3 – 6 (RR); use 3 – 4 (PL); Increase for strength; decrease for prototyping / fast printing or for decorative objects (TB)

7. Uncheck to speed up print time but watch first few layers. (JH)

8. Uncheck to speed up print time but watch first few layers. (JH)

9. Try the "*Random*" setting with artistic/decorative objects (PL)

Print Settings > Infill
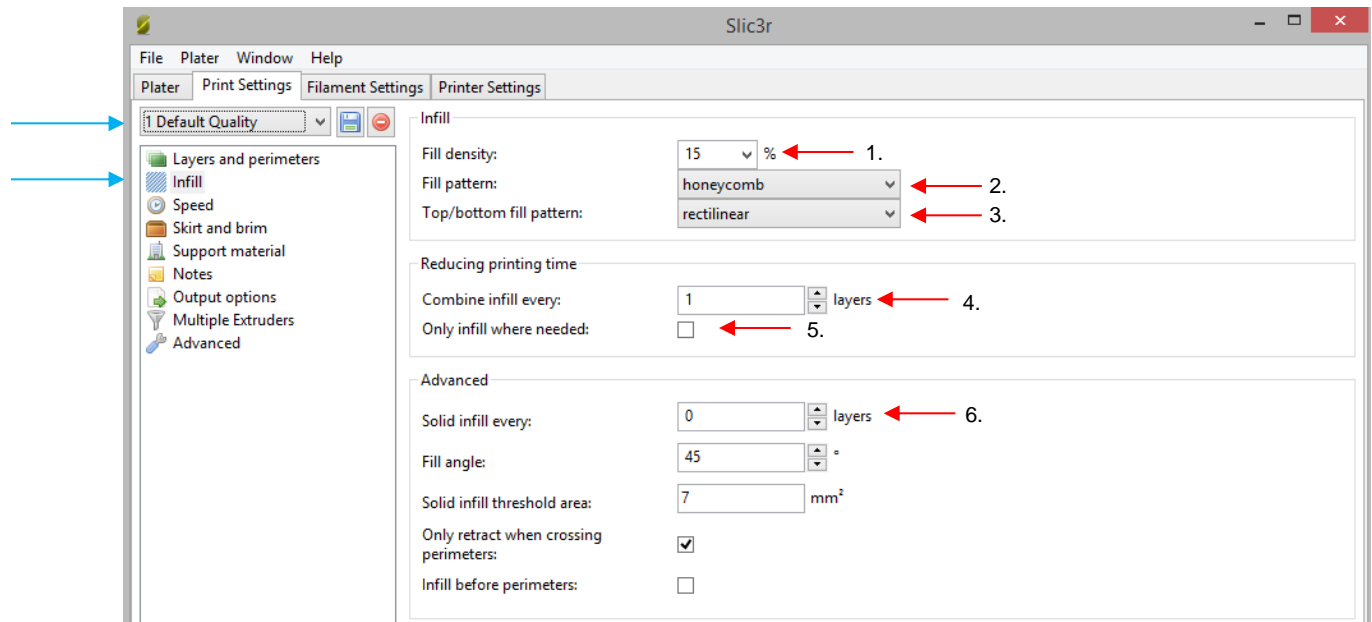
1. Can lower to 10, but use 25 – 35 for stronger objects (KU); use ≥ 20% to support flat ceilings and use 40% when more mechanical strength is needed (SM); use 10 – 20% for display objects (TB); If object is really small, may need to increase so that there's enough material present (JH)

2. Normally select "*Rectilinear*", "*Line*", or "*Honeycomb*" (which is the strongest). (SM)

3. *"Rectilinear"* normally selected based on appearance (KU)

4. Increase in order to speed up print time (SM)

5. Turn on to infill/support ceilings only (AR); if used, prints faster but object won't be as strong (KU)

6. Increase if more strength is needed (PL); Increase when printing short objects (RR)

Print Settings > Speed
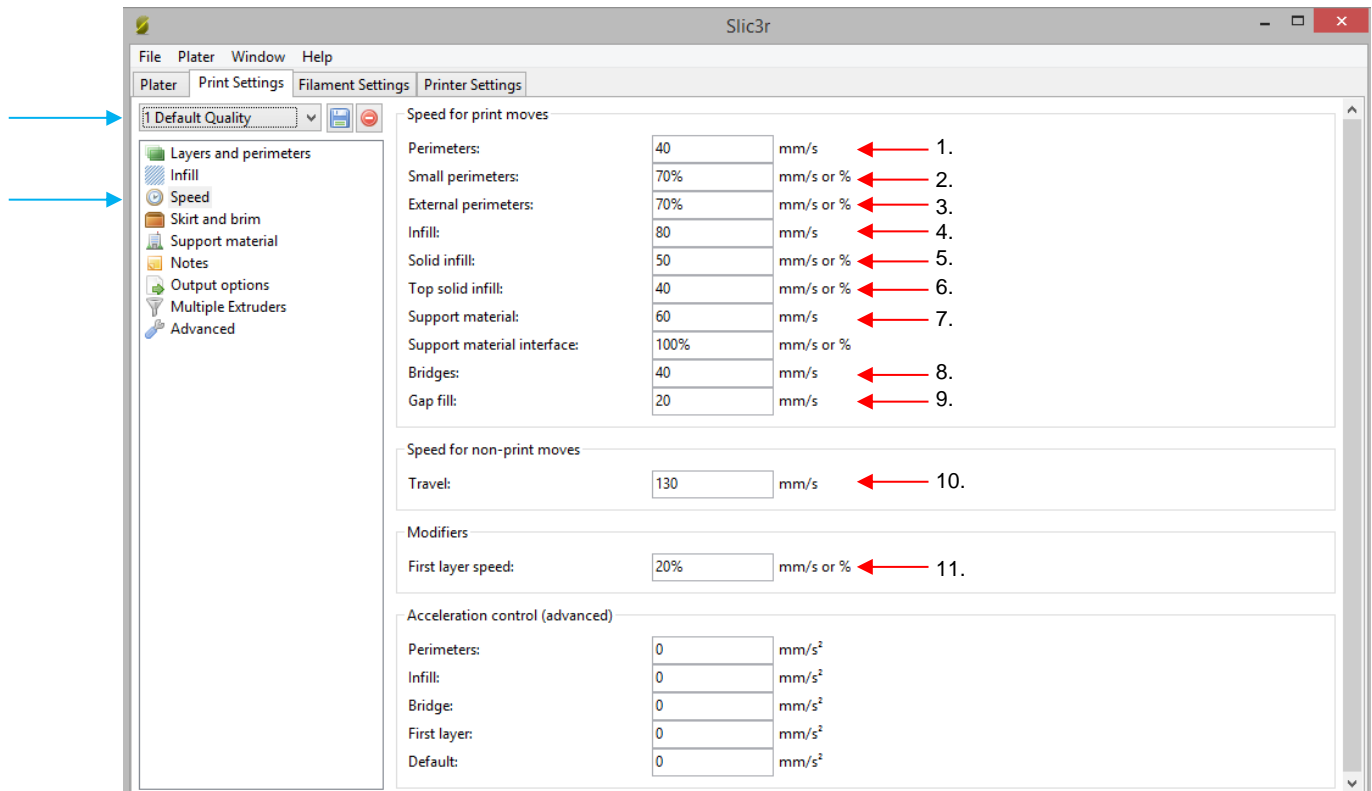
1. **Perimeters:** Normally use 40 – 50 but lower to 30 if **Layer height** is 0.1.(KU); always slower than **Infill speed** (SM)

2. **Small Perimeters:** Decrease if there are small holes, islands, or small details (SM) (AR)

3. **Ext. Perimeters**: Decrease to have a cleaner surface (SM)

4. **Infill**: Use 40 – 80 (KU); as fast as possible without compromising the integrity of the fill. (SM)

5. **Solid Infill:** Use 40 – 50 (KU); use slightly slower than **Infill** but faster than perimeters. (SM)

6. **Top Solid Infill**: Same or lower than **Solid Infill**, but lower to 20 if using a **Layer height** of 0.1 (KU)

7. **Support Material:** Usually as fast as possible (SM)

8. **Bridges**: Usually slower than **Perimeters**. Too slow will result in sagging (too much material), too fast will result in broken strands (too litte material) – need to experiment.(SM) (JH)

9. **Gap Fill**: Usually slow so that extruder doesn't begin to oscillate. (SM)

10. **Travel:** Usually as fast as possible to minimize ooze (SM)

11. **First Layer Speed:** Important that first layer speed is slow (KU); use 30% – 50% (SM); always slower than other layers (AR); use 20% – 25% (PL); decrease if prints are not sticking (CB) (JH)

Print Settings > Skirt and brim
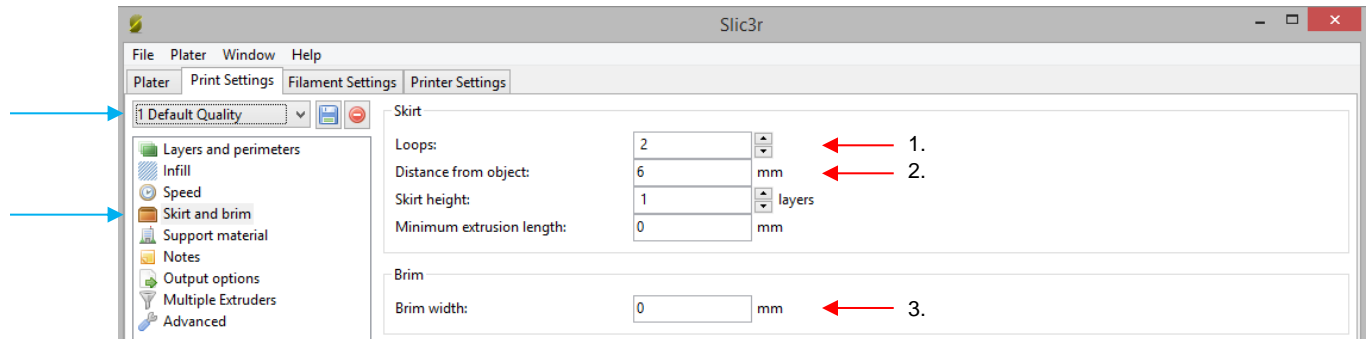
1. Use 2 (KU); Use 1 or 2 (PL) (RR); Use 2 but increase if switching from a dark colored filament to a light one (JH)

2. Use 6 (KU); Use 2 – 5 (PL); Use 6 (RR); Use 2 (JH)

3. Use 4 – 6 when object has low surface area contact with build plate (KU) (SM) (PL)

Print Settings > Support Material
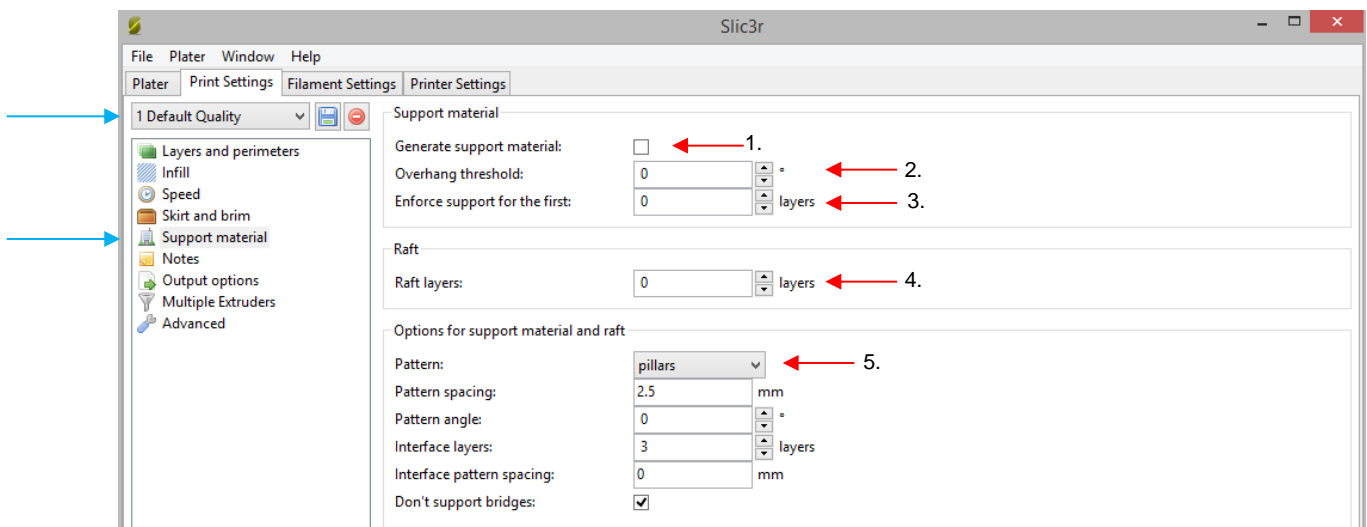
1. Turn on where there's overhang that will need support (KU) (CB) (PL)

2. Keep at 0; allow Slic3r to determine the best settings (SM)

3. Set as desired (SM)

4. Used to build a mesh under the object (PL)

5. Pillars is preferred, but use **Honeycomb** if having adhesion issues (KU)

Print Settings > Notes
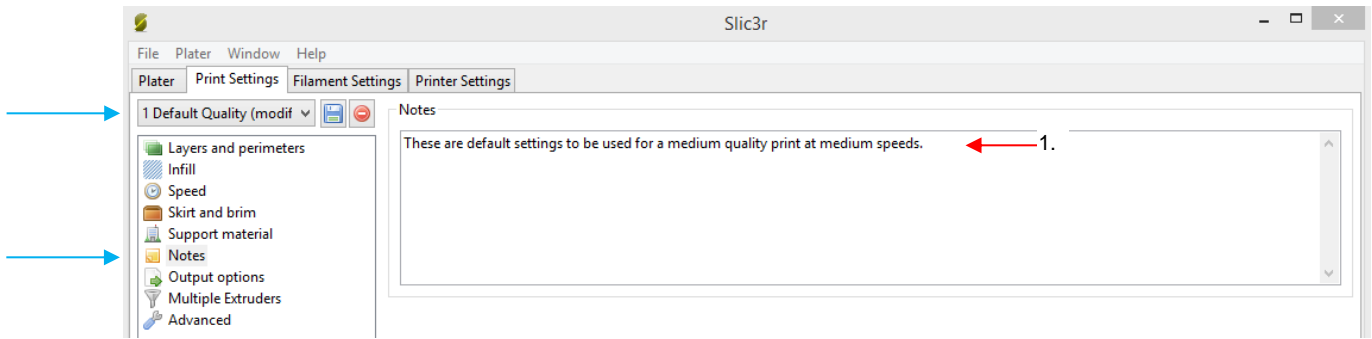
1. Use the **Notes** window for making any comments about the profile, such as what types of shapes print well with these settings. Maybe even list some specific file names you've used with this profile.

Print Settings > Output Options
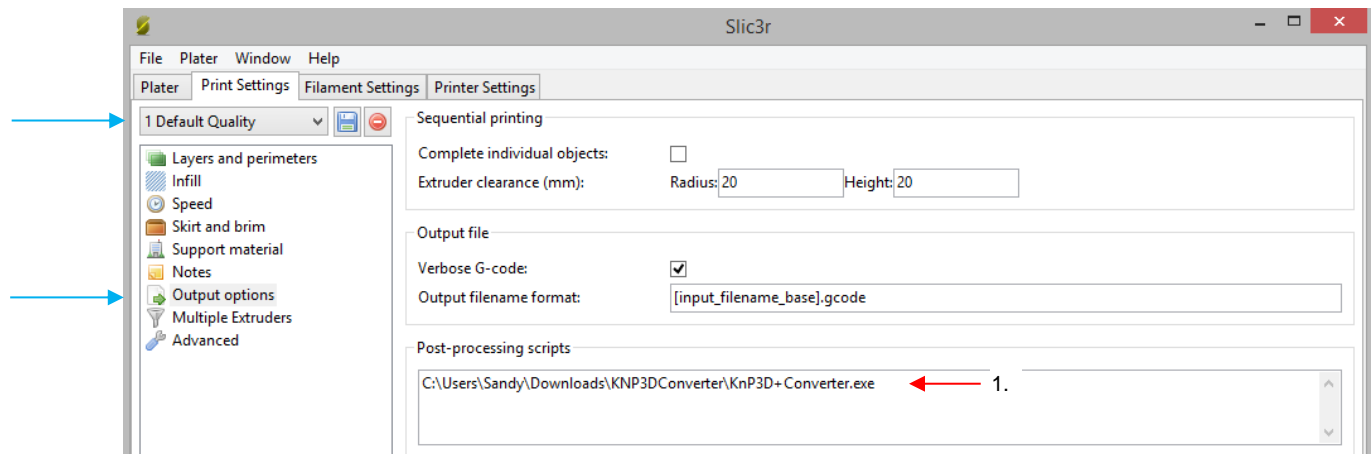
1. Add any post-processing scripts that you find available and applicable.

   ◊ One example is the script shown above: It will allow one to by-pass the second step of launching the **KNP Converter**. In other words, when the **Export to G-code** button is pressed, a G-code file is created and KNP Converter will also run and create the X3G version. Both files are saved in the same folder as the G-code file folder you select.

   ◊ To set up this script for yourself, just enter the path where you have the **KNP Converter** program saved. Don't forget that you will need to set this up for every **Print Settings** profile you plan to use, but it's easy to copy/paste from one profile to the next.

Print Settings > Multiple Extruders
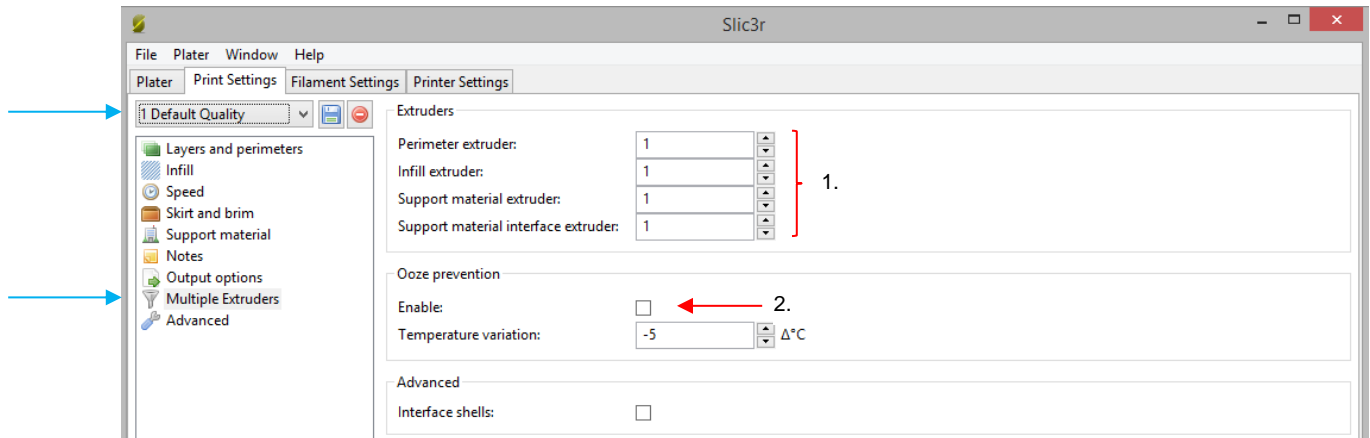
1. These are set to 2 when using the left extruder only. (KU) (Note: this is already set in the "*Left Extr Only*" profiles.)   When printing with a support material loaded into the left side, set the third and fourth settings only to 2. Refer to *Section 5.10.* (KU)

2. Turn on for dual extrusion prints. Refer to *Sections 5.10* and *5.11.3.* (KU)

Print Settings > Advanced
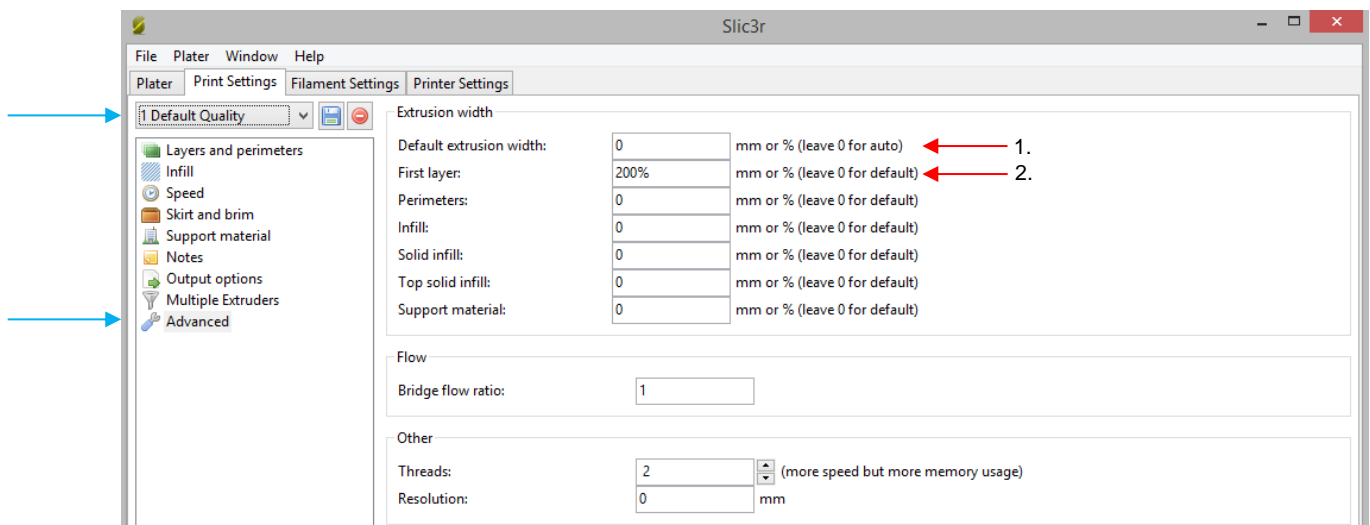
1. Increase to 110% when using left nozzle only and if that nozzle is a bit higher than the right nozzle. (KU)

2. Use 200% (KU); Needs to calculate to be larger than **Nozzle Diameter** (SM); ≥ 200% (AR)

B2 Filament Settings

- **Filament Settings** are related to the filament material, such as diameter, extrusion temperature, cooling options. These settings will normally remain the same unless you change to a different type of filament, such as switching from PLA to ABS.

Filament Settings > Filament
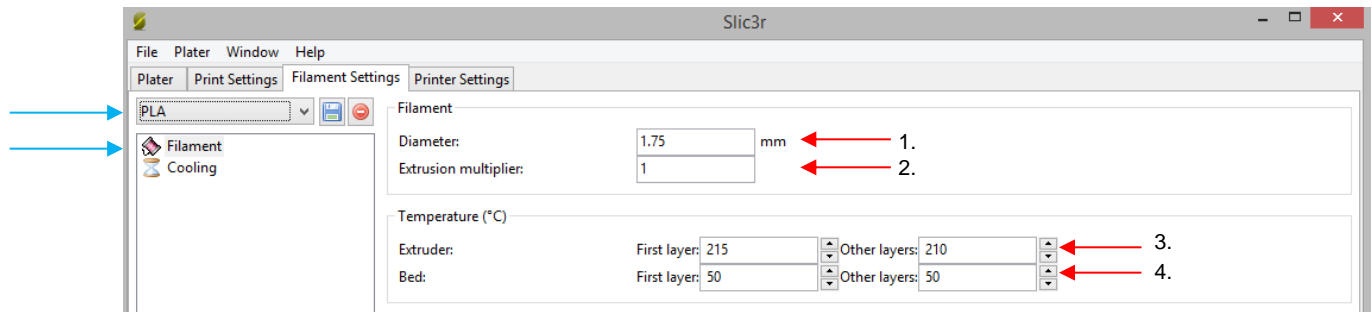
1. Use calipers to measure filament size as variance can occur. (JH); Adjust if variance is more than 0.01 – 0.02mm. If set too high: layer adhesion issues, too low: clumps and strings (CB)

2. For fine tuning extrusion flow rate in ± 0.05 amounts (SM); Also can use if filament is off-spec. Range: 0.9 – 1.1 (AR);

3. Increase if gaps in infill, decrease if threading. (KU); ABS requires 235 and 230, as set up in ABS profile (KU); **First layer** is ~ 5°C higher than **Other layers** (SM) (AR); Reduce if **Layer height** is ≤ 0.1 mm. (RR); Raise by 5°C if filament extrudes unevenly or not sticking to build plate. (CB)

4. Set to 110 and 110 (the same) for ABS, as set up in ABS profile (KU); Raise by 5°C if object is not sticking to build plate or is lifting on several sides. (CB)

Filament Settings > Cooling
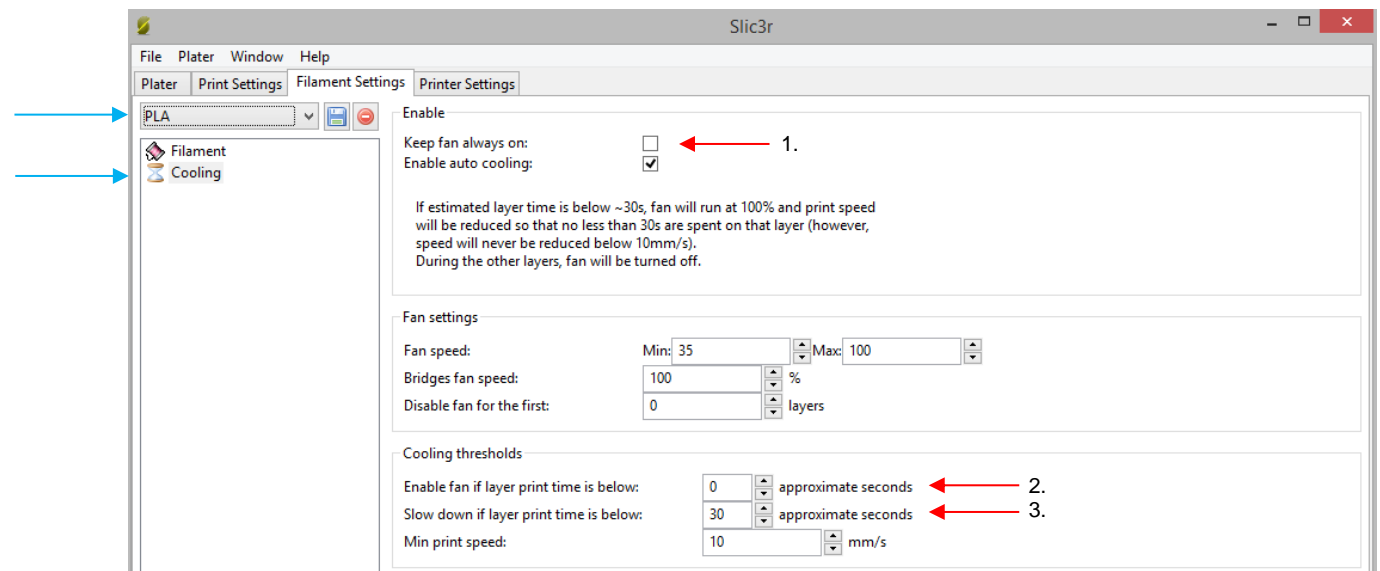
1. KNP3D does not have a cooling fan, thus leave unchecked. (KU)

2. Same case; leave set to 0 (KU)

3. Increase to 40 or 50 if printing an object like a sphere where there is a small contact area on the build plate and the object expands outward.

B3 Printer Settings

- **Printer Settings** are related to the KNP3D itself, such as the bed size, firmware type, and number of extruders.

Printer Settings > General
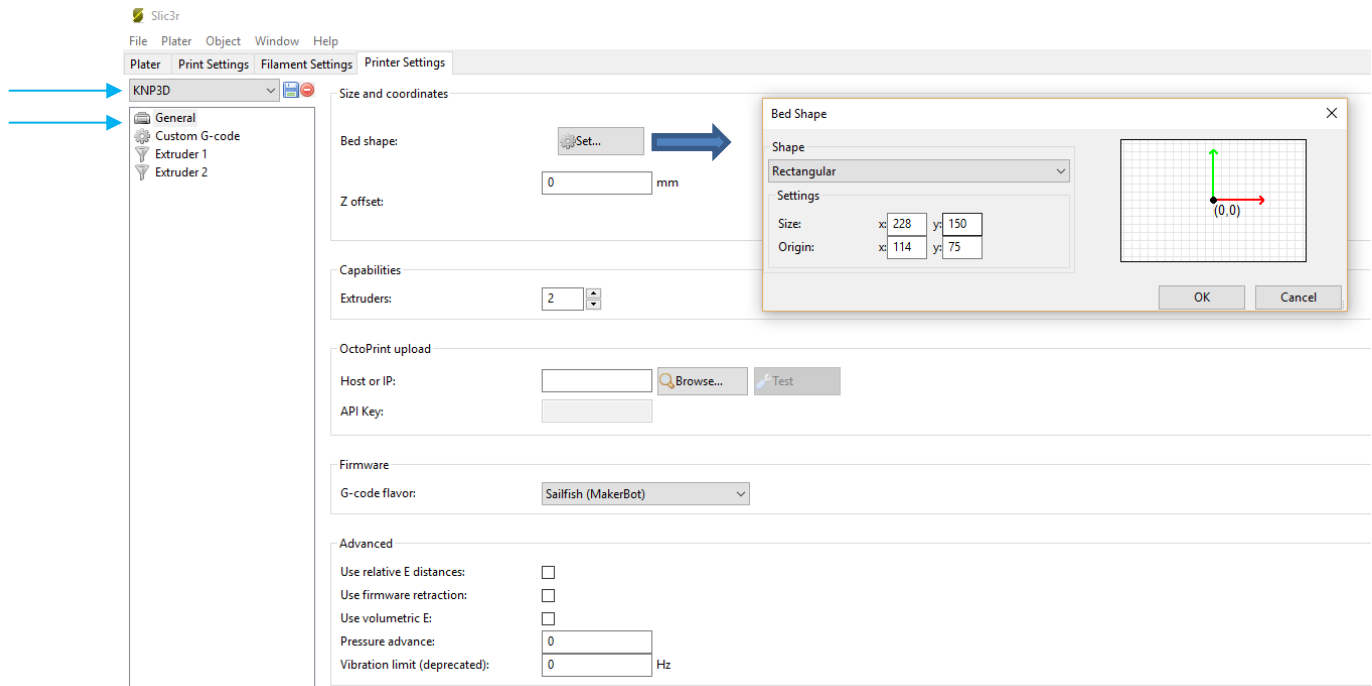
No changes recommended.

Printer Settings > Custom G-code
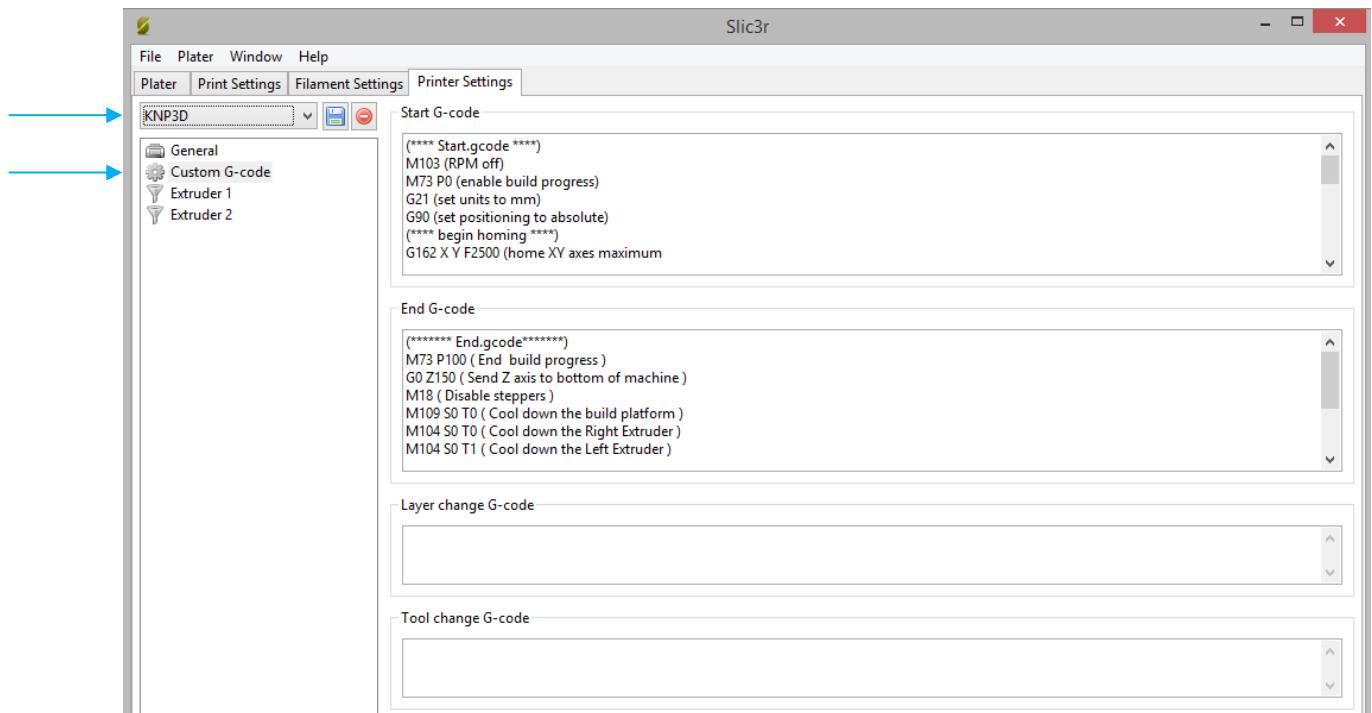
- A small change was made for the "*Left Extr Only*" profile. Otherwise, no changes needed unless you wish to study on your own and try out.
- G-code explanations available at http://www.reprap.org/wiki/G-code

Printer Settings > Extruder 1 (or Extruder 2)
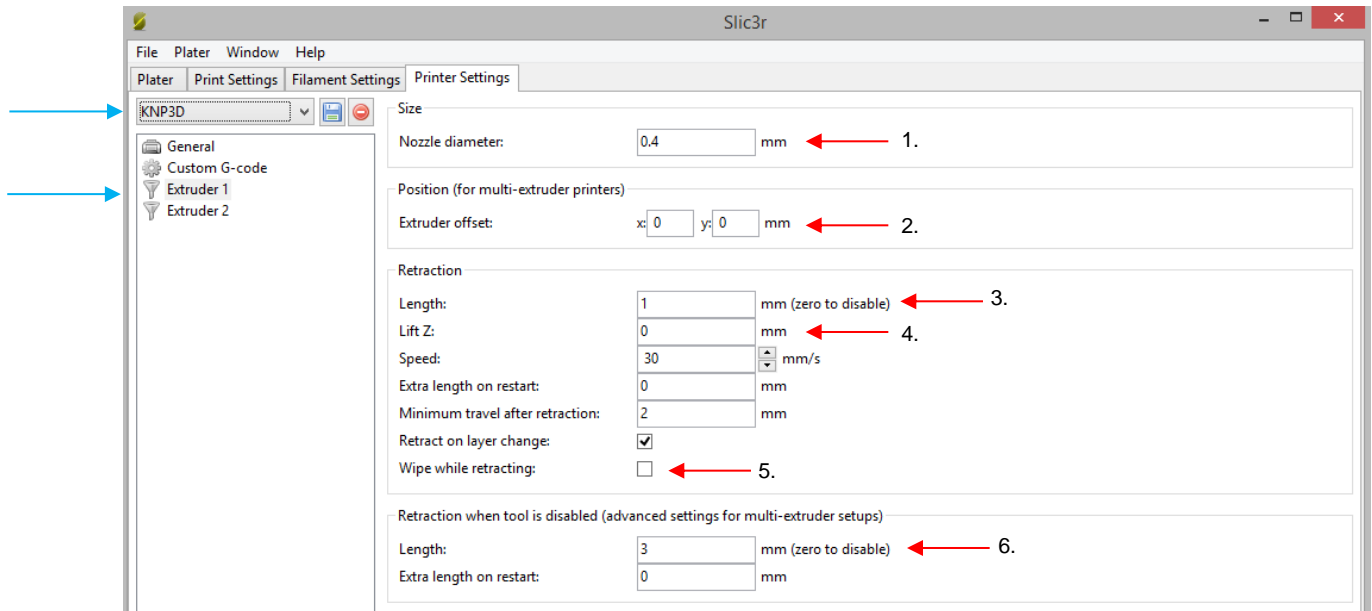
1. Only change if installing a different sized nozzle (KU).

2. Set within the firmware. KNK USA will advise if you are installing a different sized nozzle.(KU)

3. Usually 1 – 2 mm (SM); Can have a strong effect on print quality, increase if stringing is present, usually ≤ 2mm. (JH); Use to reduce oozing (CB)

4. Usually 0.1 mm (SM); Try 0.3 mm on shapes with small surface contact (RR); Use to reduce stringing (CB)

5. Turn on when a very clean object is needed; increases print time (KU)

6. Usually set to 2 or 3 but not higher than 5 (KU)

Appendix C Control Panel - Utilities

Map of the Menus on the KNP3D Control Panel While Idle

Refer to location in parenthesis for information on each function.

Print from SD Card	(5.09)
Preheat	(5.02)
Utilities	(C1 - C10)

Monitor Mode	(C1)
Filament Loading	(2.05)
Preheat Settings	(C7)
Level Build Plate	(2.04)
Home Axes	(C2)
Bot Statistics	(C8)
Filament Odometer	(C9)
General Settings	(C12)
Profiles	(C10)
Home Offsets	(C3)
Jog Mode	(C11)
Enable Steppers	(C3)
Blink LEDs	(C4)
Calibrate Nozzles	(C4)
Restore Settings	(C4)
Eeprom	(C4)
Version Information	(C5)

Ditto Printing	(4.02.4)
Override GC Temp	(C12)
Pause W/Heat	(C12)
Sound	(C12)
Heat LEDs	(C12)
LED Colour	(C12)
Accelerate	(C12)
Extruders	(C12)
Extruder Hold	(C12)
HBP Installed	(C12)
Tool Offset Sys	(C12)
Check SD Reads	(C12)
P-Stop Control	(C12)

Map of the Control Panel Menus During Printing

Press left button during a print to access the following menus.	

Back to Monitor	(C13)
Cancel Print	(5.01)
Pause	(5.03.1)
Pause at ZPos	(5.03.2)
Change Speed	(5.04.1)
Change Temperature	(5.04.2)
Set Cooling Fan Off	N/A
Print Statistics	(C14)
Cold Pause	(5.03.3)

Back to Monitor	(C13)
Cancel Print	(5.01)
Unpause	(C5.03.1)
Jog Mode	(5.06)
Filament Loading	(5.06)
Change Speed	(5.04.1)
Change Temperature	(5.04.2)
Print Statistics	(C14)

C1 – Utilities Menu Page 1

- On the control panel, select **Utilities** on the main menu:

Check current temperatures. Refer to *Section C6.*

Change preheat settings Refer to *Section C7.*

Load or Unload Filament Refer to *Section 2.05*

Level the build plate Refer to *Section 2.04*

```
Monitor Mode
→Filament Loading
Preheat Settings
Level Build Plate
```

C2 – Utilities Menu Page 2

- To access the following additional settings, press the down arrow on the control panel after accessing the menu shown in *Section C1*:

Raises build plate and moves extruder to origin; displays current temperatures

Info on current filament usage: Refer to *Section C9.*

Info on KNP3D usage: Refer to *Section C8.*

More settings: Refer to *Section C12.*

```
^Home Axes
Bot Statistics
Filament Odometer
General Settings
```

70

C3 – Utilities Menu Page 3

- To access the following additional settings, press the down arrow on the control panel after accessing the menu shown in *Section C2*:

Set up preheat profiles. Refer to *Section C10.*

Use arrow keys on control panel to move extruder head. Refer to *Section C11.*

^Profiles
Home Offsets
Jog Mode
Enable Steppers

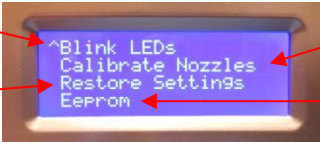Do not change unless a different sized nozzle is used. These are set for 0.4 mm.

Toggles stepper motors on/off – do not modify!

C4 – Utilities Menu Page 4

- To access the following additional settings, press the down arrow on the control panel after accessing the menu shown in *Section C3*:

Toggles on/off blinking of the LED lights.

Restores all settings to factory defaults.

^Blink LEDs
Calibrate Nozzles
Restore Settings
Eeprom

Calibrates nozzle lines for both X and Y – Do not modify!

Do not access Eeprom!

C5 – Utilities Menu Page 5

- To access the following additional settings, press the down arrow on the control panel after accessing the menu shown in *Section C4*:

Displays current firmware version.

^Version Information
Exit menu

Return to main menu.
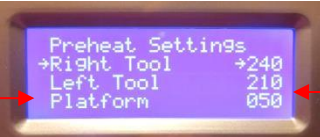
C6 – Monitor Mode before Print Begins

- This function displays the current temperatures of both extruders and the build plate.

- To access **Monitor Mode**, select from page 1 of the **Utilities** menu (refer to *Section C1*):

The KLICNPRINT
R Extruder: 060C
L Extruder: 028C
Platform: 022C

C7 – Preheat Settings

- Use this function to change the preheat settings to new temperatures.

- To access **Preheat Settings**, select from page 1of the **Utilities** menu (refer to *Section C1*).

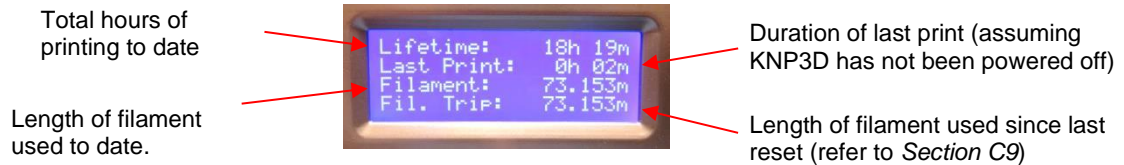(1) Use up/down arrows to select the tool or the platform. Press **K**.

Preheat Settings
→Right Tool   →240
Left Tool   210
Platform   050

(2) Use up/down arrows to change the temperature. Then press **K**. Finally, press left arrow to exit menu.

C8 – Bot Statistics

- This function presents info on the KNP3D's and filament usage.

- To access **Bot Statistics**, select from page 2 of the **Utilities** menu (refer to *Section C2*).

Total hours of printing to date

Length of filament used to date.

Duration of last print (assuming KNP3D has not been powered off)

Length of filament used since last reset (refer to *Section C9*)

C9 – Filament Odometer

- This function presents info on filament usage and ability to reset.

- To access **Filament Odometer**, select from page 2 of the **Utilities** menu (refer to *Section C2).*

Length of filament used to date.

Press **K** to reset **Trip** to 0.0.

Length of filament used since last reset

## C10 – Profiles (for Preheating)

- This function allows one to set up preheat settings and name them. This is useful if you are changing filament types frequently as it will allow you to select a particular profile for that material without needing to change the **Preheat Settings**.

- You also have the option to change **Home Offsets**; however this should only be done if changing to a nozzle with a different diameter. For the 0.4 mm nozzle these offsets should remain at their default settings of X: 151.998, Y: 74.995 and Z: 0.0.

- To set up a profile, first make any changes to the **Preheat** settings, as presented in *Section C7*.

- Next, to access **Profiles**, select from page 3 of the **Utilities** menu (refer to *Section C3).* Select it and the window of **Profile** names will open:

Select a profile and press **K**.

- Upon selecting a profile, four options appear:

Change **Preheat Heatings** and **Home Offsets** to what is assigned in this profile

Change the name of this profile.

Display the **Preheat Settings** and **Home Offsets** currently assigned to this profile.

Current **Preheat Settings** and **Home Offsets** will now be assigned to this profile.

72

- Selecting **Display Config** will bring up a window with these settings. The down arrow will need to be used to access the Z Offset and temperatures:
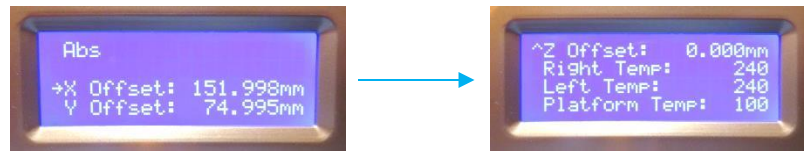
C11 – Jog Mode

- This function allows one to use the arrow keys on the key panel to move the extruder head or build plate to a different position.

- To access **Jog Mode**, select from page 3 of the **Utilities** menu (refer to *Section C3).* Then press the right and left arrows to maneuver between the X, Y and Z control windows. In each window, use the up/down keys to change the location and the **K** key to go back to a prior window:
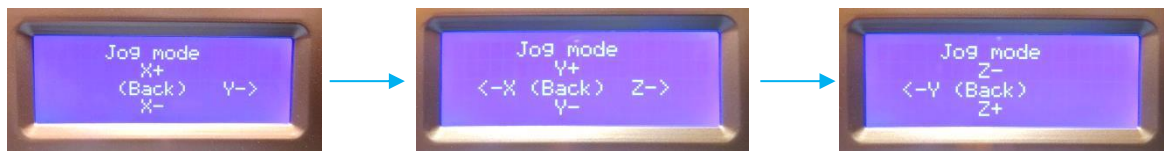
X control window   Y control window   Z (build plate) control window

C12 – General Settings

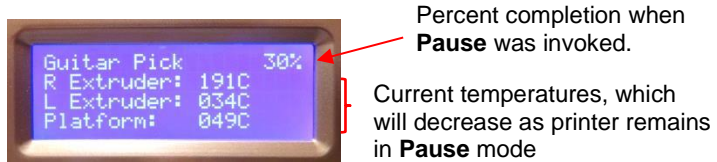| Function	Setting to Use	Comment
Ditto Printing	OFF (or ON)	When turned on, left extruder will print duplicate object at the same time as right extruder. Objects must be less than 1" wide or printing will overlap.
Override GC Temp	OFF (or ON)	Normally leave set to OFF. If ON, print using preheat settings instead of temperatures set inside file on SD card.
Pause w/ Heat	OFF (or ON)	If turned ON, heating continues while in Pause mode.
Sound	ON (or OFF)	Toggles on/off the tune played when reaching target temperature, end of print, etc.
Heat LEDs	ON (or OFF)	Toggles on/off the lights, which change color as the extruders are heating up.
LED Colour	WHITE (or any of 7 other options)	Select which color you prefer for internal LED lights
Accelerate	ON (or OFF)	Leave set to ON; Acceleration allows for higher quality at faster speeds.
Extruders	2	Number of extruders on KNP 3D: leave set at 2

73

Extruder Hold	OFF (or ON)	Leave set to OFF; for use with 3mm printers only
HBP Installed	YES (or NO)	Heated Bed Plate installed? Leave set to "YES"
Tool Offset Sys	NEW (or OLD)	Leave set to NEW; needed for dual extrusion printers
Check SD Reads	YES (or NO)	Leave set to YES; if turned on, errors in SD file will be checked during the print and corrected if possible.
P-Stop Control	OFF (or ON)	Leave set to OFF; not applicable to KNP3D

C13 – Monitor Mode after Print Begins

- If the printing is underway, the **Monitor Mode** displays this information:

Percent completion

Current / Target Temperatures

- If the printing is paused, the **Monitor Mode** displays this information:

Percent completion when **Pause** was invoked.

Current temperatures, which will decrease as printer remains in **Pause** mode

C14 – Statistics

- During a **Pause**, the statistics displays this information:

Length of print at time of **Pause**

Amount of filament used at time of **Pause**

Z axis position at time of **Pause**

Line of code processed at time of **Pause**

Appendix D Specifications

Printer Size: 22.4" x 19.7" x 23.2" (570 mm x 500 mm x 590 mm)

Printer Weight: 39.7 lbs. (18 kg)

Print Size: 6" x 6" x 9" (152 mm x 152 mm x 229 mm)

Technology: Fused Filament Fabrication

Number of Extruders: 2

Nozzle Diameter: 0.4 mm

Minimum Layer Thickness: 50 microns (0.00196")

Maximum Nozzle Temperature: 280°C

Filament: PLA, ABS, PETG, HIPS, PC, Nylon, Flexible

Filament Used: 1.75 mm

Power Requirement: 110 - 220V / 350W

HMI: LCD Interface

Connectivity: USB, SD

Operating Systems: Windows, Mac OSX

Software: Slic3r, KNP Converter

Acknowledgements