229/508 User Manual

V022 2/2/2020 (!)

Table of Contents

Safety:

Your Acceptance of Responsibility:

There are a number of potential dangers when using the PolyPrinter. In careful hands these do not normally present large risks. However, you must accept the responsibility of managing those risks yourself, and in turn so must you also be responsible for, or transfer responsibility to anyone else using or who has access to the machine. Do not use or purchase the Polyprinter without full acceptance of this responsibility.

PolyPrinter Operation

By its nature, the PolyPrinter must use some very hot components.

- The heated bed and its supports can be hotter than boiling water. Treat it with as much caution.
- The "hot end" where the plastic is melted is even hotter. It is in the range of a stove element. It WILL cause immediate burns if contacted while hot.

There are also potential problems of a chemical nature:
- Acetone, which is (optionally) used occasionally to clean the bed surface, is flammable and its vapor may be harmful. It must be used only on a cool bed, and never poured or spilled. The room must have adequate ventilation to disperse the vapors. Acetone on a heated bed will vaporize rapidly, forming an explosive mixture.
- The ABS or other plastics used for printing may give off fumes and smoke-like particles. In general these are not deemed very toxic but some people may develop a reaction after intense or long term exposure.
- Keep small children, the uninitiated, and anyone else not able to be safe, away from the PolyPrinter when its hood or side panels are open.

The general safety precautions are:
- Ensure that the printer is used in a well-ventilated area. The combination of HEPA filter and the enclosed cabinet greatly reduces the smell and smoke-like particles that are produced during printing, but there is still some that could potentially build up if the printer is in a confined space with no ventilation.
- Keep the side doors and the hood closed while extruding or printing, and for a few minutes after printing.
- Do not touch the bed surface when hot. (Or when cold, because fingerprints prevent adhesion.)
- Do not touch the extruder head block or nozzle while it is hot. When printing, it is normally hot enough to cause immediate burns if contacted.

Mechanical Dangers:

The moving components of the PolyPrinter can hit, pinch and trap hands and fingers. Sometimes they can move surprisingly quickly, and with enough force to cause harm. Do not interfere with the moving parts. Be especially careful not to have a finger or hand become trapped between the head and the bed, since both can cause burns.

Misuse, abuse:
- Do not use the PolyPrinter as a bird cage, reptile habitat, pet dryer, cat-startler or anything else that could harm your pets. Using it to dry books, warm coffee, make cupcakes, and to dry insects is fine.
- Pets can be bothersome. Cats in particular recognize that the PolyPrinter is a reasonably-empty box. Cats will also think the top of the PolyPrinter was designed to warm their feet and tummies. That's OK. It has been specifically designed to take the occasional helpful feline's weight, since, in fact, during development their presence and opinion was not optional.

Part Cleaning

- Support removal is generally not too terribly difficult, but does depend entirely on the part, and the settings used for support. A good pair of small needle-nose pliers with grippy serrations in the jaws can be used to tear away a great percentage of the support, most of the time. Using the pliers can be safer than using a knife.
- If using a knife, be sure to cut with the knife pressure in a direction away from body parts or anything precious - a knife will very often shoot far past the cutting point when the plastic suddenly gives way.
- In general, it is best to have a good solid work surface to do parts cleaning upon. Protect it with a sheet of something durable but not precious, like a cutting board, cardboard, wood or a magazine.
- Try to orient the part being cleaned so that you are generally putting downward pressure on the tool, such that the desk surface takes all the force and is also what is hit by the tool when you cleave the plastic.
- Use finger guards or leather gloves to protect against minor nicks and cuts.
- Use tools that are just sharp enough to do the job. Sometimes a slightly duller tool will actually make support removal easier, because it does not dig into the plastic so much.

Parts of the PolyPrinter

Color-Coding:
The mechanical parts that you might normally need to interact with are yellow.

Main Components

- X Carriage - travels left and right
- Y Carriage (Printer Bed) travels for and aft.
- Z Height Adjuster - adjusts nozzle clearance to the bed for the first layer

Front Switches

At the lower left side there are two black switches. The top one controls the "Camera" light on the right side of the hood. (If equipped.) It's good for general lighting of the printing area, and especially helpful in giving the camera enough light to work with, especially if you want to monitor printing progress remotely in a room where the lights have been turned off.

The lower switch controls a light on the X Carriage to better be able to see printing in progress.

On the side of the same switch housing is a red Reset switch. This will interrupt any operations in progress, cancel printing and disconnect communications. It is not normally needed or used as part of operations. It is occasionally needed if USB connections have been disturbed.

Rear Panel:

- USB Connector
- Power Cord Connection
- Power Switch
  - You do not need to turn the PolyPrinter off when it is not being used. Leaving it on takes very little power, and maintains the USB connection so that it is able to print on demand.
- Fuse

Camera

The built-in camera (PolyPrinter 229 and 508) provides the opportunity to use a program on the connected computer to monitor printing. There are several methods for doing this:

- Create a Skype or other internet communication account just for the printer. Set it to auto-answer calls, and you can then call your printer from another computer or a smartphone. This is only suitable for situations when it is acceptable for the monitoring to be done by only one person at a time. It may also create privacy problems if the camera (which has audio too) allows someone to eavesdrop on the general area of the printer, unbeknownst to others.

- Set up a frame-capture program that periodically uploads an image from the camera to a web server. This is good for when the printer is shared such that a number of people are

interested in the status. An example can be found at https://www.yawcam.com/download.php where you can click on yawcam_install.exe

- You may want to try out several camera software packages e.g.: Digi-Watcher, Yawcam, iSpy - NOTE: we cannot vouch for these but they appear to be legitimate and applicable! (also IP Camera Adapter, DeskShare)

Testing the Camera

In Windows, hit the Windows key on the keyboard and start typing "Camera". You should be able to open the Camera app and see the picture (you may need to select the camera from the list of installed cameras).

Computer Requirements

There are three main activities typically involved with using a 3D printer:

- Designing parts and preparing the 3D files
- Processing the files before printing ("Slicing")
- Sending the files to the printer and printing them.

Each has its own effect on the requirements the computer needs to meet.

It is possible to use a relatively slow, older computer to do all of these things. It generally gets annoying to deal with a very slow computer though, so if you have a choice, and certainly if you have the budget to buy a new one, it is best to aim a higher than the minimum specifications.

Video

A good video card or the newer Intel integrated video will give a nicer experience with most 3D design software, since it generally involves a need for rapid refreshing of the display. In addition, the software interface to the printer, PolyPronter, updates a display of the printing process by default that takes up quite a bit of video performance, and can actually slow down printing and reduce print quality if on a computer with slow video, if not put into a mode ("Mini Mode") where that drawing is suppressed.

CPU

The slicing programs e.g. Kisslicer will use all available processor cores to speed up the slicing process. An old single-core machine like a Pentium 4, or a Centrino, will be much slower than a multicore chip like an Intel i7, which has eight effective cores. This is not too much of an issue for small parts, and is merely a question of the time you'd have to wait for the slicing process to complete. **A generally useful price point is an i5**, which will be slower than an i7 but usually

good enough.

RAM

The design programs and especially the slicing programs can use large amounts of memory to hold all of the 3D information when they process the files. Like the number of cores, this can be exchanged for time, but sometimes, and it is again much more of an issue with the larger parts, the amount of time required can rise dramatically, if the computer is bogged down moving information back and forth between hard drive and RAM because there is not enough RAM to work on it all at once. **Aim for at least 6 GB if you are looking at a new computer.**

Operating System

Windows 10 is preferred but it is possible to use Windows 7, or Windows 8 (requires a few extra steps during installation). We only recommend considering 64-bit versions.

Apple Mac, Linux: Will work but no install package yet. Each software package must be separately installed.

NOTE: *PolyPrinter is no longer formally supporting installations on 32-bit operating systems.*

System Specifications

Bare Minimum:
> Intel Core Duo, 4+ GB RAM, Intel integrated video. Windows 10 64-bit.
> Effects: Will need to use "Mini Mode" in PolyPronter while printing. Kisslicer may issue warnings about RAM, and will be slow to slice files.
> 64-bit operating system

Useable:
> Intel i3 or i5, 6+ GB RAM. There is a lot of variation in the video performance in this range, but if the computer is labeled as an "Entertainment" system it may be adequate. Windows 10 64-bit.

Better:
> Intel i7, 8+ GB RAM. If new, the video should be fine. Windows 10 64-bit

Power Saver Settings

It is very important that the computer not go "to sleep" during printing, while PolyPronter is sending commands to the printer. If that happens, often the USB connection will be dropped, and the print in progress will fail, and be unable to be continued.

You can set up a custom power setting for printing, if you wish.

You'll want Hibernation to be Never, and Sleep to be Never. It's OK for the hard drive to shut down, and for the CPU to use power-saving modes.

Computer Update Settings

Set Automatic Updates to require your approval before installation and especially rebooting.

Computers with Extremely Slow Graphics

A few of the slower laptops we've seen have such slow graphics chips that the drawing that PolyPronter does is actually slowed down so much that printing speed is reduced. If the printer seems sluggish, stopping briefly or going half-speed at times during a print, you may need to use "Mini-Mode" which closes most of the PolyPronter window. That button will change to Full Mode so you can open it up fully again to check progress or start another print.

Windows 8

There is a problem installing the standard USB driver under Windows 8. You need to turn off a new "feature" in Win8:

From the Metro Start Screen, open Settings (move your mouse to the bottom-right-corner of the screen and wait for the pop-out bar to appear, then click the Gear icon).
Click More PC Settings.
Click General.
Scroll down, and click Restart now under Advanced Startup.
Wait a bit. Click Troubleshoot.
Click Advanced Options
Click Windows Startup Settings.
Click Restart.

 When your computer restarts, select "Disable driver signature enforcement" from the list. You can now load your modified driver. Reboot again once the driver is installed and all will be well.

References:
- ■ http://www.windows7hacker.com/index.php/2012/08/how-to-install-an-un-signed-3rd-party-driver-in-windows-8/
- ■ http://www.pjrc.com/teensy/windows8.txt

Finding Parts

The installation will have created a small samples folder under the main install folder (e.g. C:\Program FIles\PolyPrinter\Samples).

1. Get the STL file into a folder accessible (local HD, USB stick, LAN drive) to the local machine.
   ● Download from www.thingiverse.com which contains tens of thousands of potentially printable parts (and lots of other buildable items).
   ● Alibre, 123D, Sketchup etc. CAD program export
   ● Meshlab, Blender etc.

2. The downloadable STL files can have problems or oddities. When you load the part into Kisslicer, look for any red-colored triangles. If you see a lot of red areas, then those areas will probably not end up printing correctly. (There is a program called NetFabb that can will help you fix that but that's beyond the scope of this quick guide.
3. Make sure the orientation is the best for printing. Many STLs are oriented to make their designing convenient, but that is not necessarily the best way to print it.
There are two important aspects to the orientation:
>   1. Amount of support required. This affects printing time, cleaning time, and surface finish.
>   2. Strength. Printed parts are stronger within the layers than between layers.

Try to orient the part so that the biggest, flattest surface is touching the bed. Look for overhangs - any part of the object that sticks out horizontally, or whose underside slopes down will need support.

In Netfabb, rotation around any of the X, Y, or Z axes is easy. Select Part, Rotate from the menu or use the toolbar button, or use CTRL-R. Select the angle (usually 90 or 270) and the axis, and press the Rotate button.

When you have the object rotated the intended way, use Part, Export Part and Save as STL. Save it under a different name so you can keep the original e.g. if the original name was "xxxxx.stl", save it as "xxxx rot.stl".

Slicer Settings

We could probably write a small book about the slicing settings. Plastic, print choices, parts, all kinds of things can make you try different settings.

Here are the basics:

Kisslicer

Our installation package from our web site sets up Kisslicer with a good set of configuration files that generally meets the needs of a wide range of objects. You can extend the configurations by making copies of the supplied settings and modifying them according to your purpose.

Slicing Parts

The control program, PolyPronter, is set up to only load GCODE files, because that is all that the printer understands.

Kisslicer creates GCODE from STL files.

The usual steps, for printing a file using Kisslicer are:
- Open the STL file in Kisslicer.
- Slice the file and save it. This saves the GCODE file in the same directory as the STL file. Kisslicer will use the settings from the tabs in the bottom right corner of its main window.
- Open the file in PolyPronter. Make sure you open the *.GCODE and not the *.STL file.
- Print the file.

For most things, you can simply use:
Style Tab: Style: .**Normal**
Support Tab: Support: .**Normal**
Material Tab: Material: **ABS**
Printer Tab: Printer:**PolyPrinter**

On the Style tab, in the bottom left there is a Slider labeled Precision:
Fast
- When you just want to see what it's like, use full speed. When you're printing a "keeper", you will want to slow it down.
- For some parts, it won't make much difference, because it has to slow down on the interior fill, and for small things with lots of layers it slows down and turns on the fan to avoid making it all melty.

Precise
- Use this if you want the best finish and accuracy.

○ You can adjust the Precision slider from 0 to 100% Precision. If you're not sure. then 50% is a good place to start, because it's a good tradeoff between speed and quality. You can see Kisslicer's time estimate vary as you move this slider, if

you have already sliced the object with the current settings.

For scaling, look at the menu up top, "All Models"
    Inch->mm is a shortcut to 25.4 conversion

Objects with irregular bottoms, thin contact with the bed, or other things that prevent good adherence to the bed may need a Brim. When printing challenging parts with no support, you may need to use support up to a certain level to ensure good initial adherence and accuracy of any bottom curvatures.

To unload a part from Kisslicer, press the "X" on the small window (the Thumbnail) along the right side.

The first time you print a particular part that you may want more than one of, print just a single one, if there is any doubt about the settings being correct.

Steps

Get the STL file into a folder accessible (local HD, USB stick, LAN drive) to the local machine.

The free version of Kisslicer will not allow loading more than one STL at a time, so to load a new STL you must press the "X" on the small window along the right side, to remove the old one.

Press "Open", and find the STL file for the part you wish to print, and press "Open" to load it. The part should appear in the main window.

Check the file for oddities. Lots of STL files have reversed triangles and other problems. If it's all green-colored, it's fine. Red areas will probably not be sliced correctly. NetFabb will help you fix them up, but that's beyond the scope of this manual at this time.

Scaling

If it is surprisingly tiny, it may have been designed in Inches. Kisslicer has a quick Inch-to-mm scaling conversion for that. See the menu, All Models/Inch → mm.

You can adjust the size of the model to something you think is appropriate by typing over the "Height" field in the model's thumbnail window on the right-hand side. You can also use the menu, All Models/Scale By X function.

Orientation

Make sure the orientation is the best for printing. Many STLs are oriented to make their designing convenient, but that is not necessarily the best way to print it.

There are two important aspects to the orientation:

- ○ Amount of support required. This affects printing time, cleaning time, and surface finish.
- ○ Strength. Printed parts are stronger within the layers than between layers.

Try to orient the part so that the biggest, flattest surface is touching the bed. Look for overhangs - any part of the object that sticks out horizontally, or whose underside slopes down will need support.

Kisslicer has a few basic rotation features. Right -click on the thumbnail window and you will get a menu. Hover over "Transform Axes" and a submenu will appear. Slide over to it and you can select from several rotation options. Experiment with them.

Tip: Tall/wobbly prints should always be sliced to align the most stable direction with Y, since the bed moves in that direction.

Do the actual slicing

Press "Slice". For large models this can take quite a while. You can see the progress in the graphics window.

When it has sliced, there will be a time estimate on the lower right. You can easily scale the model by typing in a new height in the Upper right-hand window with the thumbnail of the part. Just type over the height. You can also rotate it on the bed.

Saving the GCODE

Once you are happy with all your settings (you may need to re-slice, if the time estimate has disappeared), then you can Save.

After you have approved the output file name (which normally needs no changes) a black text window will appear for a short while, as a post-processing script runs that improves the print quality a bit. When that window disappears, the GCODE file is ready to be loaded into PolyPronter.

Simplify3d

Please contact us at support@polyprinter.com to get a configuration file.

PolyPronter - Connecting to your printer

Find the icon for "PolyPronter" :-p , in your Start Menu. Click to launch PolyPronter. It should look something like this:
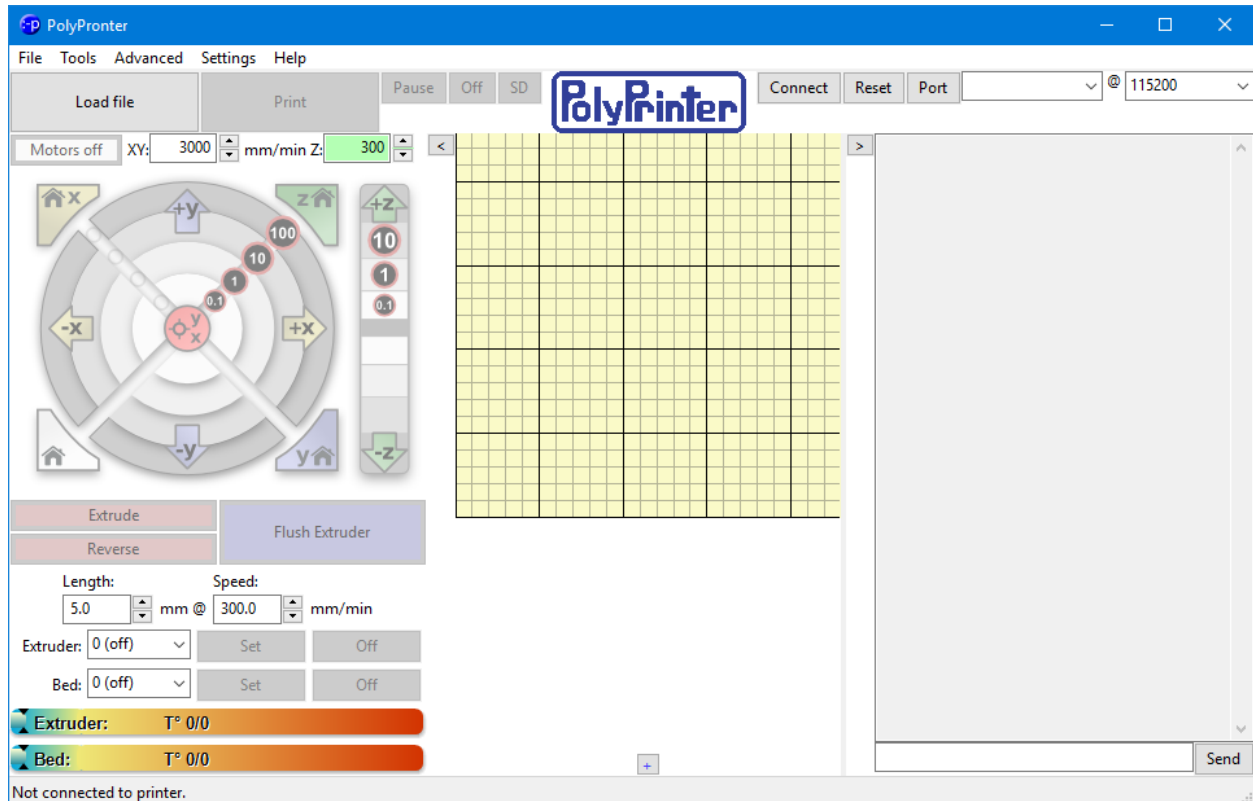
Connecting

Connect: Used to connect PolyPronter to the printer selected in the dropdown menu to the right of the Port button (ex. COM4). If there is nothing listed under the dropdown, try using Port (see below).

Reset: Not currently used for anything.

Port & 1st Dropdown: Used to scan for printers that are available for connecting. You can select any printers found with the dropdown menu

**2nd Dropdown:** This controls the baud rate. For polyprinters, this isn't important. In the event that you have more than one PolyPronter open, you can type something in here to better help you distinguish between which instance is tied with which printer. For example, typing "Blue" into this box and placing a strip of blue masking tape on or around the printer that it is connected to.

First Time Connecting Steps

The important part is the Port. Most of the time, it will already be filled in with the port that represents the USB connection to the printer. We'll assume it's correct for the moment.
Press the "Connect" button.
Some text should appear in the Console of PolyPronter, including the phrase: "Connected".
If it does not:, see below, in Problems and Solutions, PolyPronter Connection Problems.

The Big Two: Load and Print
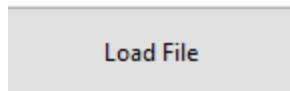
Load File: Used to select a Gcode file through a windows file explorer. Once you have opened a Gcode file, it will have appeared in the Graph (see below) portion of PolyPronter.
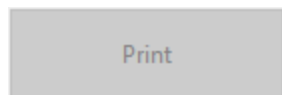
Print: Used to start printing the Gcode file that is queued. If the bed and extruder are not hot, then there will be a delay before the printer begins extruding.

Motor Controls
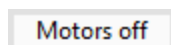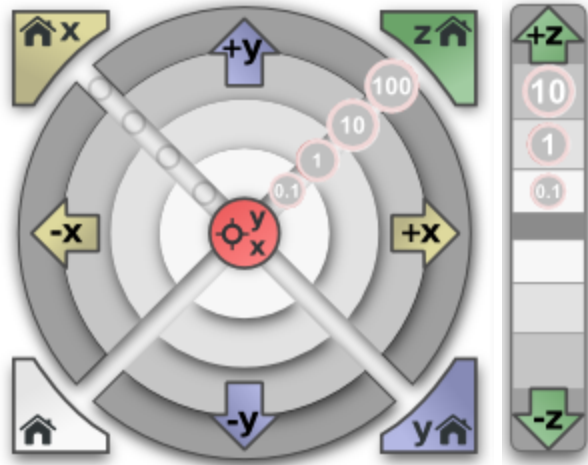
Motors off: Used to turn off the motors so you can move them manually.

 X Home: Sets the extruder to X0.

 Z Home: Sets the extruder to Z0.

 Home All: Used to move the nozzle to X Home, Y Home, and Z Home. (X0Y0Z0)

 Y Home: Sets the Extruder to Y0.

 Buttons on this side will raise the extruder, and increase Z height.

 Raises by 10 mm

 Raises by 1 mm

 Raises eases by 0.1 mm

 Lowers by 0.1 mm

 Lowers by 1 mm
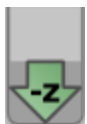
 Lowers by 10 mm

Extruding and Bed Controls
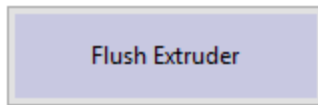
Flush Extruder Flush Extruder: Used to purify the filament after you've changed filament colors.

Extruder: 0 (off) ⌄ | Set | Off
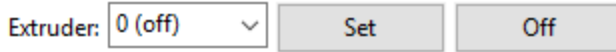
Dropdown: You can either select one of the three standard temperatures (0, 185, 230) or type your own number in (only for experts).

Set: Used to heat the extruder to the temperature listed in the dropdown. This is useful for some maintenance procedures

Off: Used to turn the extruder heater off after you've turned it on with Set.

Bed: 0 (off) ⌄ | Set | Off
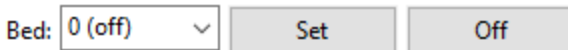
Dropdown: You can either select one of the three standard temperatures (0, 60, 110) or type your own number in (only for experts).

Set: Used to heat the bed to the temperature listed in the dropdown. This is useful for some maintenance procedures

Off: Used to turn the extruder bed off after you've turned it on with Set.

Extruder: T° 0/0
Bed: T° 0/0
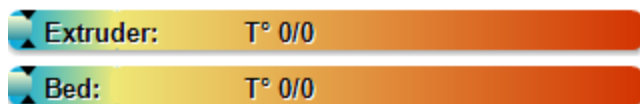
Temperature Gauges tell you (Current temperature)/(Set temperature) in celsius.
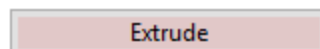
Extrude

Reverse

Extrude: Used to advance the extruder by a certain length (default 5 mm) at a certain speed (default 300 mm/min).
Reverse: Used to retreat the extruder by a certain length (default 5 mm) at a certain speed (default 300 mm/min).
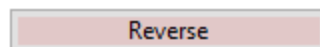
Graph   ## Console

```
\samples\Printable Bed Lattice.gcode, 16157
lines
2228.74mm of filament used in this print
The print goes:
- from 5.28 mm to 224.73 mm in X and is
219.45 mm wide
- from 0.00 mm to 224.72 mm in Y and is
224.72 mm deep
- from 0.00 mm to 0.25 mm in Z and is 0.25
mm high
Estimated duration: 1 layers, 0:19:48


                                      [ Send ]
```
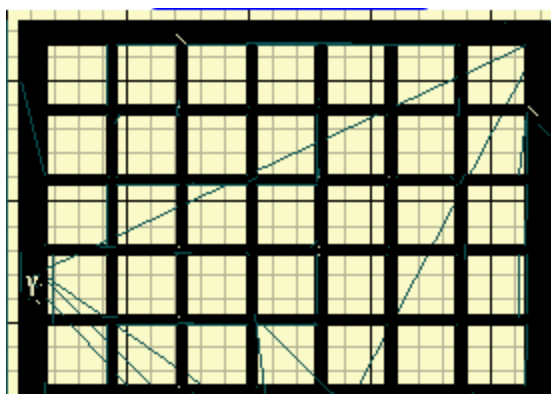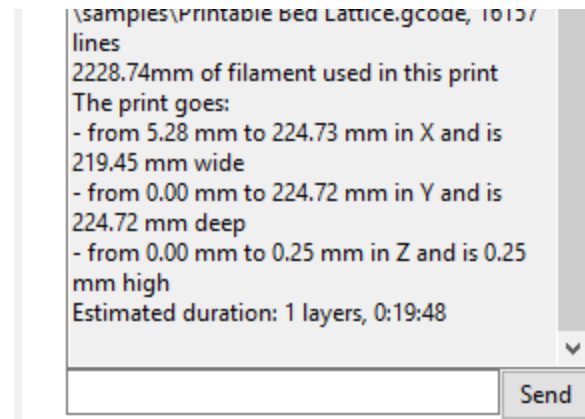
Graph: Shows an outline of the part and a general print progress bar.

Console: Displays feedback from the printer in text Form. Also allow the user to input some G-Code commands manually.

Installing the Filament

- If it's a new reel you will have to unpack it and find the end, (which should normally be threaded through a hole in the rim of the reel) taking care not to let it slip under any of the other turns of filament. Make sure you can loosen the end, but don't let it unravel. If there is no hole in the rim to secure the end with, hold onto the end. (You may need to discard the hub protecting discs and the filament retainer.)
- While securing the end of the filament, slip the reel over the reel mount so that the filament will come up through the hole in the filament guide arm, and continue up toward the filament pulley. Viewed from the front of the PolyPrinter, the end will be pointing upwards and will be on the left side of the reel, where it needs to go through the hole in the black filament guide arm. Feed it through that hole, and bring it forward toward the extruder.
- You don't need to put it over the pulley until you have fed it into the extruder, although you may do so.

Trimming the Filament

- Always trim the end of the filament before inserting it in the extruder.
- If the end of the filament is irregular, bent or blobbed, then you must trim it using the handy filament cutter that is mounted on the upper cross bar. Swipe the filament through the gap in the cutter, holding both sides.
- If the end is curled, straighten it out somewhat so it will be easier to insert.

Feeding Filament Into the Extruder

- Press down firmly on the left end of the yellow lever, so that it goes down, which moves the drive pressure ball bearing away from the filament drive wheel and lets the filament past. Push the filament down through the hole in the top of the lever, and ensure that it goes all the way down and into the filament tube that is just below the drive wheel. It should go into the tube under the drive wheel at least an inch (25mm).
- Push the filament, still holding the lever down, until it won't easily go any further. It should go in about two inches (5 cm), and if the head is up to temperature, continued steady pressure will make melted filament ooze out of the nozzle.
- Let go of the lever.

Closing the Filament Detector

- Hook the filament over the pulley and move the pulley and filament together up toward the switch.
- Its magnets should pull it into position with a firm click.
- PolyPronter should display "Filament OK".

Ensuring the Filament is Ready

- Using PolyPronter, set the head temperature to 230 C (ABS). That's as hot as a stovetop element. It will cause severe burns if you touch the head or nozzle, even momentarily.
- If you intend to print ABS, you can also set the bed warming to 110 C if you want to cut a little waiting time off the print. The bed gets hot enough to boil water. Don't touch it!
- Click the checkbox in PolyPronter labeled "Monitor Printer". It will start updating the temperature displays every few seconds.
- Once the head is up to temperature, press the "Extrude" button. The filament drive gear should turn with a little hum, some filament should be drawn into the extruder, and a bit of plastic may come out of the nozzle.
- Press the button a few more times, and soon there should be plastic coming out of the nozzle each time. If there is not, make sure that the end of the filament has actually gone down inside the filament tube If the end is curled, you may need to straighten it, cut it, (easy to do with the Filament Cutter) or flex and twist it around a bit to get the end into the filament tube.

Changing The Filament

Heat the head to at least 230 C.
Push down on the left side of the extruder lever to release the pressure on the filament. Pull up on the filament. The very tip of the removed filament can burn you - watch out.

If it does not want to come out, and you have only just heated the head up, wait a minute or so for the heat to fully warm the filament. You may set the head temperature to 260 C if necessary,

for a very stubborn one.

Once you have the filament out of the extruder, hold onto it and remove the spool with your other hand, and insert the free end into one of the holes in the rim of the spool, if there are any. (And if there are not, make at least one.) This will ensure that the filament doesn't become looped under itself or fall loose.

Inserting the new filament is done as described above.

Printing Your First Part

- Press "Load File" in PolyPronter, Find the "samples" folder under your software installation and choose the "Calib Steps and Hole.gcode" file.
- Press "Print". It will move the carriages a little, and then it will wait until the bed has headed up fully. This can take up to 13 minutes, depending on room temperature. If you are printing ABS, you should see the temperature display climbing up to 110 C as it heats up. The PolyPronter log window will show a new line every second or show, as it get updated temperatures while waiting. (Note: If a PolyPrinter employee didn't perform setup, it's advised to perform a "paper test" defined in "Adjustments" below.)
- Once the bed has heated up fully, the carriage will again move a bit, pause for a few seconds, and then printing should begin.

Getting parts off the bed

A good general procedure is to watch PolyPronter for the bed temperature to drop about 65 C and only then try to release the part.

The bed will cool down fastest if the hood is opened and the Y carriage is pulled forward all the way (the end of a successful print will normally cause the bed to move all the way forward on its own).

Parts remain bendable until they have cooled down below about 70 C, so you risk deforming them if you try to slowly peel or pry them off above that temperature.

If you let the bed cool down down even further, to 30 C, some parts (the larger ones, especially) often just come loose enough to pull easily off the bed.

Procedure
- If you have the PolyPrinter toolkit, the included separator tool, which can be slid flat under the part.

- If you only have a knife:
  - Get the curved part of the blade to hook under or into one corner of the part, without digging into the tape.
  - Twist the knife to cause upward force. Do NOT push the blade into the part or under it. Make the twist or an upward force do the work - such that the blade cannot suddenly shoot in its cutting direction and injure you. You should use a blade that is on the dull side to reduce the chance of such an injury. Parts will tend to almost instantaneously go from being very stuck to completely free.

Always clean the bed off completely

- Make sure you clean all the plastic off the bed before starting any print. Check the home corner too. But don't use your hands or fingers, - you could burn yourself, as well as get oils on the bed - and then the plastic will not stick there.
- You can often use the part you just took off (if it's not too hot to hold) to scrape off leftover bits.

Adjustments

Overall Z Height

This adjustment is very important to successful prints. It sets the thickness (squishiness) of the first printed layer. It needs to be in a fairly narrow range of height for that first layer to stick properly.
Once you have it set correctly it usually stays the same for quite a while, until something changes.
The PolyPrinter ships out only after it has been adjusted properly and has successfully printed a number of objects. However, it may have had a rough trip to your door, and may need a bit of a tune-up before you can print successfully.

How to tell if it's correct:

After each print

> Look at the bottom surface of the print. In areas where the part was flat in contact with the bed, you should see a mostly-shiny surface (if using the green PET tape) with visible lines from the way the first layer was extruded. These lines should not be so coarse as to feel like a nail file (head too high), nor virtually non-existent (head too low). About 90 percent of the surface should be shiny, so the lines are visible but not dominant.

If you see a significant deviation from the desired level of shininess, adjust the yellow Z knob just a little (e.g. ½ of a "flat" of the knob, or ½ hour if you imagine the pointer on the knob is an hour-hand of a clock).

If the print has a coarse and not very shiny bottom, turn the knob counter-clockwise.

If the lines are disappearing, from the plastic being mushed down into the bed, then turn the knob clockwise.

Make small adjustments, and only when you are sure it's needed.

Do a "Paper Test"

Paper Test Procedure:

1. Start the bed heating up to 110 if you normally print ABS. If you normally print PLA on blue tape, then you can leave the bed cold. (PLA on green, use 65C.)
2. Heat the head to 230C, to make sure any plastic on the nozzle does not affect the adjustment. If the nozzle has a lot of plastic residue on the bottom, use the brass brush to clean it off (once it's hot).
3. Raise head to 10 or more mm off the bed.
4. Turn off the motors. (Press the "Motors Off" button which is along the left side in PolyPronter, under the Load File button - If the printer has recently printed or moved its head they will remain "on" for about a minute.)
5. Move the extruder about half way across so it's roughly in the middle from left to right.
6. Move the bed so that the nozzle is roughly over the half-way point from front to rear.
7. Place a sheet of (new or used) photocopy paper (do not use heavy paper or card stock) on the bed, under the nozzle.
8. Press the Z Home button . The head should approach the paper on the bed, and should normally touch it.

If the paper is gripped (pull gently) by the nozzle pressing it into the bed:
    a. Raise the nozzle 0.1 mm. The paper should still be just a bit gripped.
    b. Raise the nozzle another 0.1mm and the paper should be completely free.
    c. If that is not the case, see "Adjustment", next.

If the paper is not at all gripped, see "Adjustment" just below.

Z Height Adjustment:

The Z Height adjuster is the yellow knob on the left X Carriage, hiding behind the left aluminum vertical support. (It's a little shy and does not like people watching it change.)

After each adjustment, you must press the Z-Home button (or hit Z on the keyboard) to make the printer move to find the new home position, and to let you see what difference the changed setting made.

Paper test, continued:
If the head does not even grip the piece of paper at all, or completely lets go when raised 0.1 mm ,(underneath the Z column) turn the knob counter-clockwise to reduce the clearance.

If the paper is still gripped, press the again. If the paper is still firmly gripped after raising a total of 0.2 mm, then increase the gap by raising the head, by turning the Z adjuster clockwise a little clockwise.

One full turn is 0.5 mm, which is quite a lot. If it's already very close to being correctly adjusted, turn it much less than a turn - 1/8th or ¼ turn at a time is a good starting point.

Bed Left-Right Z Height Check

Your PolyPrinter should not require much or any attention to the left-right height adjustment. But if you think it has drifted off (for example, prints not sticking on the right side or the left side, vs. the other side), you may need to adjust the "tilt" from left to right:

Since the Z Height detector switch is on the left side, we adjust the right-hand motor to match the left-hand motor.

Determining Whether Adjustment is Needed

1. Press the "X Home" button on PolyPronter. The X Carriage should go all the way to the left.
2. Press the "Z Home" button. The nozzle should approach and just barely touch the bed. It is OK if it has a small gap, for this procedure - in fact, it makes this a little easier to perform, so if there is no gap at all, create a very small one now, by pressing the "+Z .1" button in the top half of the vertical set of buttons to the right of the main circular controls in PolyPronter. Press it one or more times, enough to get a small visible gap between the nozzle and the bed. It doesn't much matter what the size of the gap is, as long as it's small enough for you to see whether it changes significantly, in the next steps.

3. Press the "Motors Off" button on the left upper side of PolyPronter. Also make sure that the Extruder and Bed "Off" buttons are pressed so you don't get burned while doing this procedure.
4. Then move the head to right manually and see whether the bottom of the nozzle moves away from or toward the bed as you move to the right.
5. Perform the "Paper Test" (see Z Height under Adjustments) on the right and left sides. (You can leave the head and bed cool for this, because you're not setting a final overall height.) What matters first of all is that it grip it the same left and right, Use the "0.1 mm " buttons to find a good height for this test. It doesn't matter whether it's correct overall, yet, as long as it shows you differences between left and right. Pick a spot about above the bed mounting rail, and centered front-to-rear, on each side to be your test spots.
6. If the clearance is the same left and right, perform the overall Z-height adjustment described above in "Z Height Adjustment".

Left-Right Adjustment

Loosen the two setscrews (with a 1.5mm (smallest) sized hex driver tool) that couple the right hand Z rod to the belt drive assembly at the top of the printer.

Make fine adjustments by slightly rotating the Z rods (most easily done down at the motor shaft). Hold the left-hand threaded rod so that it doesn't turn, and turn the right-hand threaded rod:
● counter-clockwise to decrease clearance
● clockwise to increase clearance

You should feel little "steps". Generally, you should only need a handful of steps. That is only a few degrees of rotation. Do a few, then test and decide if it's enough or too much. Take your time.

After you have made any adjustments, slide the head back and forth to check how the clearance changes. If you can't see it change, your alignment is good.

Tighten the two setscrews.

Adjust the overall Z height using the yellow adjustment knob, as needed.

Do a test print of the bed lattice, or the first couple of layers of some wide part that will show up the way the first layers are contacting the bed.

Bed Leveling (four corners)

The PolyPrinter's bed is generally quite stable over time, and should very rarely need a four-corner bed leveling (.e.g maybe once a year, usually not even that often)..

25

First, make sure this is really needed, by printing a test print. (Or checking relative grip of the nozzle to a piece of paper in each corner (about 1 inch (25mm) from the edges.)

See C:\Program Files\PolyPrinter\samples\Simple Test Square.gcode and use a lightish color so you can write on it with a Sharpie.
  - Look at the bottom to compare the effective Z height in each of the four corner areas of the bed.

Check all four yellow bed rail mounts for broken-ness or cracking, looseness, etc. because sometimes the problem is not really one of "adjustment".

Assessment of the Test Print

A four-corner adjustment is generally only needed if just ONE corner differs markedly from the other three. If the test print didn't work at all (Z-height adjustment), or shows only a left-right difference, fix those problems first.

If it appears to be a diagonal type of difference, one corner of the bed probably needs an adjustment, and then possibly also a bit of left-right, depending on what action is taken.

It's most useful to "map" all four corner areas (but about 1 inch or so from the very edges) relative to each other, to gauge whether e.g. three of them agree and the fourth is an outlier.

If it is found that on the left both are the same as each other, and on the right both agree with each other, but there's a difference between the left pair and the right pair, that would be a true left-right adjustment. In that case, see *Bed Left Right Adjustment*, above.

If the test print showed that just one corner was different from the other three, then you may be able to make an adjustment to just that one corner. See Adjusting a Bed Corner Height, below.

Adjusting a Bed Corner Height

Make sure you have read the above and have done a test print and have proven that only one corner has a problem. (See Bed Leveling, above.)

Important:

1. Only ever change one corner at a time. If one corner is a problem, leave the others

alone.
2. Keep a log of what steps you have taken - if you turn an adjustment screw, write down which direction and how much e.g fractions of a turn, or "hours" it was turned. That way you will be able to make better sense of the intermediate results. Number each step, and write that number on the previous test print, so the adjustment sequence number represents the correction you applied based on the interpretation of that print test. The log will also potentially allow you to restore the corner to its original setting, if you determine something else is actually the problem.
3. Make SMALL changes and do a print test each time.
4. If you feel that you need to adjust more than one corner, then first make sure that each side is made consistent front-to-back. Only worry about one side at a time. Once both are consistent, then you can make an overall left-right adjustment.
5. In almost no case should you end up needing to adjust more than two corners! If you are failing to see definite progress, something is unusual, or you might need to restore the adjustments to their original settings and approach it again.

How the adjustment system works:

Each corner is held at its height setpoint by two opposing vertical screws, and a horizontal clamping screw.

The horizontal clamping screw must be loosened (use the 2.0mm hex driver and hold the nut with the pliers) while making any adjustments. It can be left loose across a set of adjustments, and then retightened at the end, followed by a confirming test print afterwards to make sure nothing changed from tightening it.

The two opposing vertical screws locate the bed rail between them. It is very important to use the correct sequence of operations when making any changes.
● The lower screw is the actual adjustment screw. It's inside the rail, so only the access hole is visible on the top of the rail, next to the end..
● The upper screw snugs the rail down slightly so that it can't move upward and mess up the adjustment. It is a silver screw on top of the edge of the rail.

When raising a corner (to get tighter head contact)
1. You MUST first loosen the silver upper screw (2.0 mm hex driver) to give the rail some room to move upwards. Loosen it by at least as much as you plan on moving the lower screw. So in general, ½ turn or ¼ turn counter-clockwise.
2. Adjust the hidden lower screw by e.g. ¼ turn counter-clockwise at most. This will push the rail up just a bit.
3. Gently snug the upper screw down (clockwise) again to clamp the rail. You need only ensure that the rail isn't free to move up and down (doesn't have play or rattle if you try

to wiggle it up and down).

When lowering a corner (to get looser head contact)
1. Adjust the hidden lower screw by e.g. ¼ turn clockwise at most. This will give the rail room to move down.
2. Gently snug the upper screw down (clockwise) to clamp the rail. You need only ensure that the rail isn't free to move up and down (doesn't have play or rattle if you try to wiggle it up and down).

As you do each change write down how much you turned the hidden lower screw and what direction. This will help you make sense of the results.

Maintenance

Clearing Plastic from the Nozzle

Sometimes odds-and-ends of plastic will accumulate on and around the nozzle. If left there, they will gradually harden into a black mess. It's best to clear it away occasionally.

The one thing to watch out for is the bottom surface of the nozzle, which has the tiny hole the plastic extrudes through. The nozzle is a copper alloy, which is soft compared with any steel utensil you might be tempted to use. Never allow a steel object to touch the bottom face of the nozzle.

We therefore have a Brass Brush in the toolkit so that you can more safely snag bits of plastic off the nozzle or bed safely (your and the nozzle's safety).

Removing Plastic Scraps

In the course of printing, flushing filament, and clearing the bed, often small pieces of printed parts, coils of extruded filament, and other scraps are sometimes created, and they tend to accumulate in the printer.

Make sure it is kept away from the end-stop switches and the Y pulley and belt.

Extraneous plastic can interfere with switch action and carriage movement.

Re-taping the Bed

The PET tape on the bed will not be as effective if it becomes contaminated with oils from contact with fingers. The part lifter should normally allow contact to be avoided.

The tape also becomes less sticky when it gets scratched up due to many parts having been

printed and lifted from it.

PARAGRAPH OVER HERE IS WHEN YOUR TAPE SHOULD BE REPLACED AND SEGUE INTO PROCEDURE

Required Items:

- PET Tape
- Knife or scissors for cutting the tape
- Plastic credit card, hotel key card or such to act as a squeegee

Procedure:

1. Raise the extruder at least 100 mm by using PolyPronter (click on (+Z 10 mm) ten times).

2. Make sure the bed and extruder are cool and set to "off" - you could get burned if the bed or the extruder is hot.
3. Take off all of the old tape.
4. Align the first piece with the front edge of the bed, tilting it so only that front edge is touching. When it's line up right, take the credit card in one hand, keeping the tape tilted partly away from the bed with the other, and "squeegee" the card-hand-end toward that edge of the bed. If it doesn't lay completely flat (and it probably won't especially when you're still trying to learn how to do this), leave it alone temporarily. Now squeegee toward the hand still holding the tape, such that the card is at a bit of an angle and so is the line where the tape is being pushed down to the bed. Go all the way across and you can then let go of the tape.
    - If the tape is not aligned well at all, just pick it back off the bed and try again.
    - If it has wrinkles in it, you may need to use a fresh piece of tape.
    - If it just has some bubbles in it, you can pick up one end, peel it back halfway, and use the card to push it back down to the bed while keeping the end up away from the bed, so there's a clean line of contact. You may need to this for each half.
5. When you're satisfied with the first row of tape, do the same thing with the next, and continue on towards the back of the bed. Align one edge to the previous tape, keeping it taut. Tilt it so that only the near edge is making contact, and just barely. Don't bother laying it down unless the edge is lying immediately beside and not on top of the previous tape.
6. The rearmost section is probably most easily done through the side panel, with the bed slid partly back.
7. Leave some extra tape to each side at first, in case you need to lift it and re-squeegee it because of bubbles.
8. It can be helpful to clean the freshly-applied tape with Acetone (see next section), but it

often works without doing that.

Acetoning the Bed

Usually the tape is worn out before acetoning it would be useful, so it is relatively rare to need to do this.

However, if the tape is in good condition but not sticking (and if the Z height is correctly set) then you may find it useful to use Acetone to freshen it up:

- Fold a plain white paper towel over a number of times until it is around 2" x 3" - more like a pad than a towel. Warning: some towels, like the blue "Shop Towels" partially dissolve and then leave a residue on the bed, preventing the plastic from sticking - the opposite of what you're trying to achieve. We have found the Kirkland plain white towels from Costco to work well, but other brands of plain white towels will probably also work well.
- Pour a little acetone on one corner so that it just soaks in and wets that end of the pad. Hold the pad by the other end to avoid getting the acetone on your fingers. Wipe the bed with the wet end, covering the entire surface.
- When you are done with it, take the paper towel to an outside garbage if you can, to avoid a buildup of the vapor in your work or living area.

Oiling the Rods

Roughly once per month, apply a very small amount of (e.g. a single drop per rod) of the green machine oil supplied in the Owner's Kit to the rods by soaking a Q-Tip, or a small piece of folded cloth with the oil. Rub it on the rods to get just a very slight film.

Checking:
A fingertip dragged along the rods should slide smoothly with very low friction. If it seems "grabby" then it needs re-lubrication, and possibly cleaning (see below).

If the fingertip comes away wet with oil, there is too much. It takes only a very thin film to provide sufficient lubrication. The fingertip will normally show glistening without being really wetted.

In either case, if there is a visible accumulation of dark or black oil residue on the rod(s), then wiping with a paper towel and re-oiling will freshen them up.

Checking the Belts

Warning: Belts do not normally need continual re-tightening. If they loosen repeatedly, something is changing. Please let us know so we can help figure out what.

Y Belt

Pull the bed all the way forward toward you (use Motors Off in PolyPronter if it doesn't want to move).
Strum the belt near the yellow attachment point. If you can feel or hear a low vibration or hum or tone at all, briefly, it's tight enough. There should be no actual slack.
 If there is slack, then turn the Y belt adjustment screw (Pictured above) ¼ turn at most, between checking for the belt vibrations until there is no slack.

X Belt

Move the X Carriage (extruder assembly) all the way to the left (X-home or move it by hand).
Strum the belt near the yellow attachment point. If you can feel or hear a low hum or tone at all, briefly, it's tight enough. There should be no actual slack.
 If there is slack, then turn the X belt adjustment screw (Pictured above) ¼ turn at most, between checking for the belt vibrations until there is no slack.

Adjusting Belt Tension

- X Belt Tension
- Y Belt Tension

Each tension adjustment is an easy turn of the respective screw adjuster. Righty-tighty.

The belts should certainly have no slack at all, and should have enough tension to make a noticeably quick vibration when "plucked".

- A belt that is too loose will cause pairing of the diagonal fill, such that instead of evenly spaced lines, you will see two lines close together, a space, then two lines close, etc.
- Belts that are just a bit too loose can cause inaccurate corners - edges that stick out a bit more than is acceptable for the given quality setting.

It is normal for the belts to get slightly less tension after an initial number of hours of printing. You may need to adjust them once, perhaps twice, in the first months of operation. If you need to tension them repeatedly, please contact us at support@PolyPrinter.com,.

Nozzle Problems

You can expect to replace the print head extruder nozzle once in a while:
- It may clog
- It wears out, enlarging the orifice and reducing print quality.

Clogged Nozzle

Diagnosing

If it is clogged, you will be unable to make plastic come out of the nozzle. This can happen (usually near the end of a large part, if Murphy's Law is operating normally) while printing, and you will typically hear the extruder clunking for a while, then perhaps going silent. The bed and X Carriage will still be merrily traversing the part but no plastic will be coming out. (This is called "printing air".)

Early Warnings

Sometimes before it fully clogs, it will partially clog, causing insufficient plastic to flow. If you have the head raised at least 30 mm and you extrude some plastic, it should normally fall straight down (unless it sticks to the head). If, however, it always curls up tightly and you cannot get it to extrude straight down, even when encouraged by snagging it with the brass brush and pulling on the end a bit, then the nozzle is partially clogged. Another sign of a partial (or complete) clog is a clunking sound when extruding filament like that at 300 mm/min (default setting).

Hardened Plastic

Some plastics can go brown and very hard if left in a full-temperature nozzle for an hour or more. Usually this is not a problem. But if you can see some plastic oozing out when you apply manual pressure to the filament, or if you try extruding (when it clunks) and that plastic is markedly darker in color than it started out, it may have hardened up. If any will flow at all, you can, with sufficient patience, usually force the hardened plastic slowly out until fresh plastic replaces it. It may help to push a bit through, remove the filament, cut off the end, and push a bit more through, repeatedly. It's still easier than changing the nozzle.

Sometimes it can be unclogged by carefully isolating one of the wavy strands of brass wire from the brass brush, and with the nozzle at printing temperature, getting the wire up the nozzle opening and working it around and in and out (it helps to try to push some filament through at the same time - you may need another person to do this more easily).

Sometimes, though, it is just plain clogged and becomes useless. See "Replacing the Nozzle", below.

Worn-out Nozzle

Diagnosis

If you see a section of some of your prints where layer after layer (typically on the right-rear corner or wherever filament has begun to be laid down on the outside after travel) the plastic

just doesn't seem to fill in properly, consistently, and perhaps curls up in place on the outside, it may be due to an increase in nozzle diameter due to wear and tear. Failing to fill well is another indication
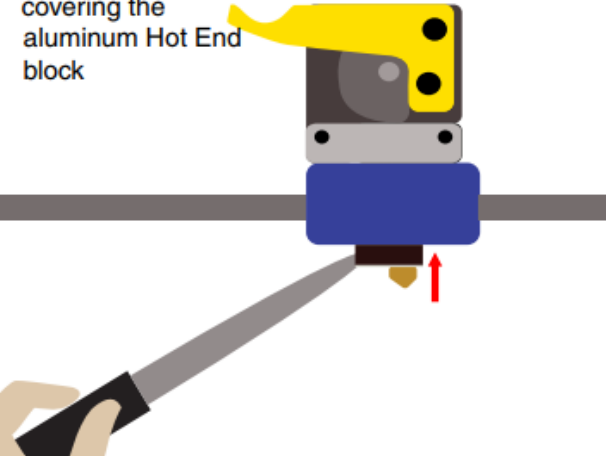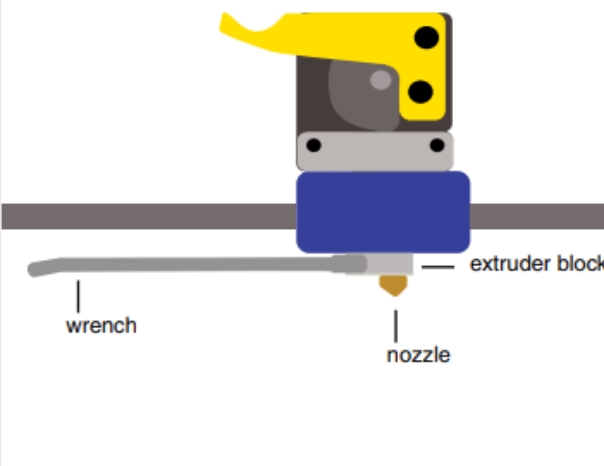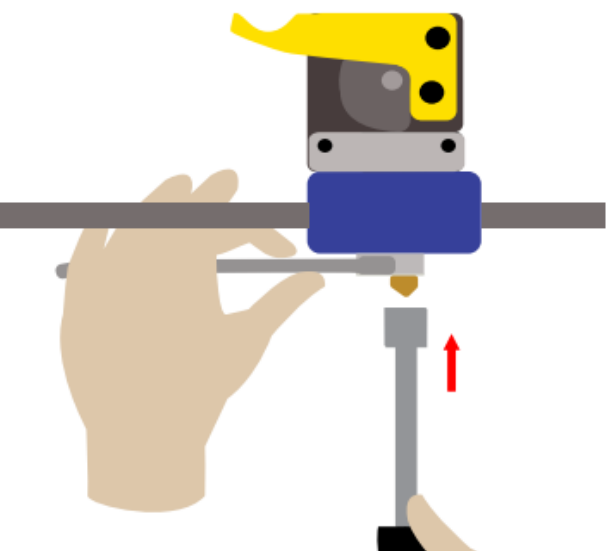
How to measure the nozzle diameter

Fortunately, the plastic can help with this - we don't actually need to measure the size of the hole in the nozzle directly.

1. Heat the head to e.g. 230 C, and extrude some plastic. Remove the plastic with the wire brush, gently. Wait a bit, and the nozzle will, as usual, slowly dribble a line of filament out.
2. Snag it with the wire brush, after a couple of inches has dribbled out - it's best if there is a good straight section.
3. Using a digital caliper, measure the diameter of this dribble, where it looks like it has a consistent diameter that represents the nominal diameter of the dribble. Be sure to use the wider part of the caliper jaws, not the very thin section at the tip of the caliper's jaws.
4. Rotate the dribble to see if it has some flatness to it, and take the average. (If it really has a lot of flatness, where for example it's twice as thick in one direction as the other, then the nozzle is probably clogged and will have to be cleaned or replaced.)

The standard nozzle diameter of 0.35 mm seems to produce about 0.40 mm measured diameter for this test. We have seen distinct problems with printing when the nozzle is worn to the point where this test gives measurements of 0.60 mm. Slight problems can be seen sometimes with a measurement of 0.50 mm. It does depend a lot on just what settings are being used. The numbers given are for 0.25 mm layer height and 0.35 mm extrusion width. To keep using a nozzle that measures oversize, you can increase the track width (Kisslicer Style, Extrusion Width setting). This will give a bit less horizontal resolution in your prints, but probably better overall quality, especially the exterior.

Replacing the Nozzle

Quick guide with pictures:

Changing a nozzle

Nozzle

10 mm wrench   7 mm Nut Driver   Pliers (optional)

Tools Required

1. Heat nozzle to 230 C
2. Raise the extruder to about 200 mm using the "Z+" button in PolyPronter
3. Use the pliers to push up the silicone sock covering the aluminum Hot End block

4. Use the wrench to stabilize the Hot End block, holding it parallel to the rail. IMPORTANT: do not use the wrench to turn, just keep it firmly in place

extruder block

wrench

nozzle

5: Push the nut Driver up to engage the Nozzle.

Continued on the next page.

6. Carefully unscrew the nozzle using the nozzle tool, keeping the Hot End block from turning with the wrench.

7. Use the pliers to carefully remove the nozzle from the Nut Driver. CAUTION: the nozzle will still be extremely hot. Place on a metal or glass surface to prevent melting or burning.

8. Place a clean nozzle into the Nut Driver, and gently screw it into the extruder, continuing to stabilize the Hot End block with the wrench.

9. Using the pliers, carefully slide the sock back down over the Hot End block, using the Pliers.

More in-depth text guide:

This requires using two wrenches (supplied) to loosen the old nozzle, and to tighten the new one.

Warning: Failing to use a second wrench to counter the nozzle wrench torque can result in loosening or failure of the filament tube. If after reading this section, you do not feel confident that you can do this yourself, find someone who is more "handy" and whom you can blame if there is trouble afterwards.

Required Items:
- 10 mm wrench (open-end is used to counter torque)

- 7 mm nut driver
- 1 Paper towel
- Optionally, one or two disposable gloves (to avoid skin irritation from glass fibers)

Steps:
Removing the nozzle
1. Let the bed cool down to 45 C or lower. Let the head cool down as well, though you'll heat it back up again when actually turning the nozzle.
2. Raise the head most of the Z travel, e.g. at least 150 mm so that there is room for the nut driver below the nozzle.
3. Move the X Carriage to roughly the center of its left-right travel.
4. Place the paper towel so that it covers the whole print bed.
5. Whenever you move the "sock" - the black silicone-covered insulator that wraps around the Hot End, small bits of glass fiber will fall down. The paper towel is to cover the bed in the hope of catching them all. They can cause skin irritation, so don't lean on the paper towel during these next steps. Do NOT blow them off the bed, because glass fiber particles are much worse when breathed in than when just touching skin. Wipe with a damp, slightly soapy cloth. (You'll need to acetone or retape the bed afterwards.)
6. (Here's where you need the head temperature to be low.) Push the black "sock" that surrounds the heated aluminum block (the "head") up to make enough room to be able to slide the 10 mm wrench onto it. You may need to pull it up on each corner in turn, because it tends to hang on the corners of the block. You may be able to maneuver the wrench in such a way as to get it on the hot end without raising the sock at all, by keeping the wrench at an angle, pointing the open end upward at an angle. At this point, you just need to make sure that you'll be able to get the wrench on there during some later steps.
7. An alternative is to pull the sock completely off., but this is generally more trouble, getting it back on, and it makes a bigger mess.
8. The wrench doesn't need to fully overlap the aluminum block - it needs just enough overlap to firmly counteract the torque you'll be applying to the nozzle.

9. Clean any old hardened plastic from the nozzle itself. Sometimes it can build up quite a layer of accumulated plastic. But it tends to harden and become brittle, so it is often not too difficult to remove when cold. You will need the nozzle to be relatively free of this extra plastic, in order to put the wrench on it properly (the circular or "box" end). Try it now, when the head is cold and you can clean it, before going on to the next steps where you'll actually be using it. If you cannot clean it well enough to fit the wrench on while the head is cold, you'll need to clean it with the brush while it's hot.
10. Once you are able to test-fit the 10 mm wrench onto the aluminum block and the 7 mm nut driver onto the nozzle, you must heat the head up to 200 C. (230, 240 are OK too, but using the minimum temperature that melts the plastic that may be in the threads is all that's needed, and you may as well be dealing with the minimum temperatures so that your recovery time from the burns will be quicker.)
11. Once the head is warm, clean the nozzle further if you were not able to clean it adequately before. The brass brush will usually work well. Watch out for flying bits of plastic or brass wires coming loose from the brush. Use safety goggles, or keep well back, and only inspect when you're not actively brushing.
12. You should now be ready to actually remove the nozzle. It's going to be hot, so again, make sure the paper towel is in place on the print bed, because if the nozzle drops the paper towel will prevent it from melting the tape where it lands.
13. Here's the one tricky and important part:
    ○   Slide the 10 mm open end wrench onto the aluminum block from the left. Slip the 7 mm nut driver onto the nozzle. (If you have a good feel for and experience with this type of thing, feel free to use different orientations.)
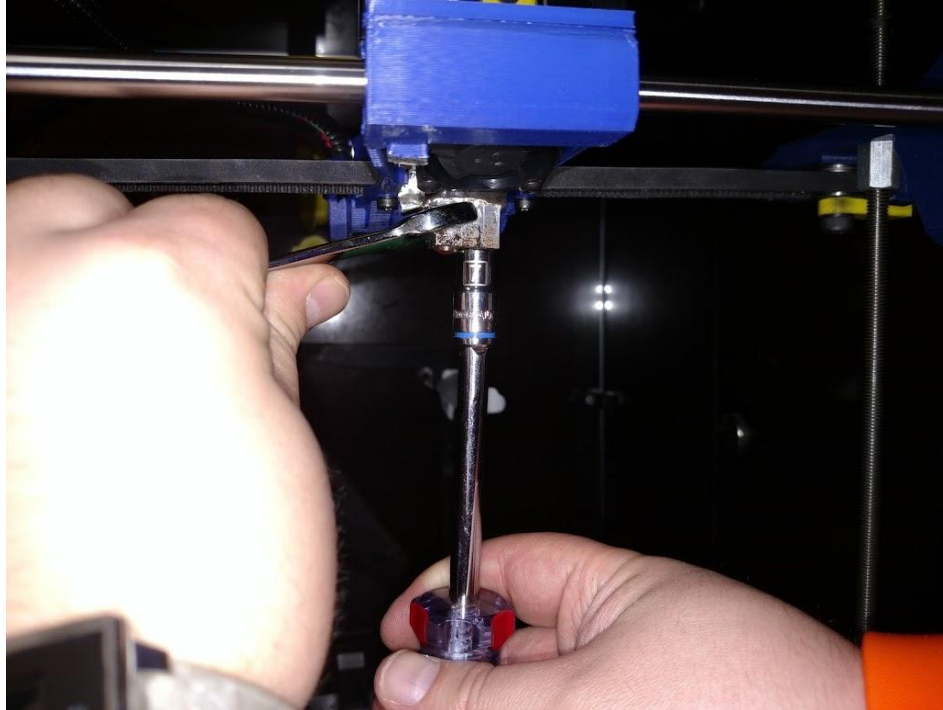
○ The purpose of the wrench is to counter the twisting force on the aluminum block (and therefore the relatively delicate Filament Tube) that will be applied to it through the effort of loosening the nozzle. All you need to do is keep it from trying to twist. You will therefore resist the nozzle-twisting force with a matching force.

○ If you take a long time with these next steps, the wrench will become too hot to work with and you will need to remove it and wait for it to cool down before trying again.

○ The nut driver on the nozzle needs to rotate with the near side traveling toward your left. Slowly apply a twisting torque on the nozzle. Match that twisting force using the 10 mm wrench - thumb pressing near the aluminum block, fingers pulling on the left end.

○ The nozzle should fairly easily break free and become easy to turn. At that point, you can remove the 10 mm wrench, and also turn off the head heat.

○ Spin the 7 mm nut driver a few times to remove the nozzle. Just leave it in the end of the nut driver to cool down. Do not try to remove it until it has cooled down - the nozzle will burn you if you try to grab it.

Installing the new nozzle

14. Make sure the head is at least 180 C, so any the plastic in the threads will be soft enough to give way.

○ Take the new nozzle, and insert it into the nut driver, with the threaded end exposed.

○ Put the nozzle's threaded end up to the hole in the hot end, align it and center it. Spin it back and forth a bit to find the thread starting position. It should spin in a few turns easily by hand. If it becomes crooked and will not spin further, back it

out and try again - it may need a somewhat different starting orientation.
- ○ When you have spun it a few turns by hand, you can go on to the next step.
15. Turn the head heat back on, to 200 C, so that the nozzle can be tightened properly in the case of plastic still being in the threads of the head.
16. Place the 10 mm wrench back on the aluminum block, ready to counter the tightening torque you will be applying to the nozzle.
17. Place the 7 mm wrench box end on the nozzle, and tighten it. (If wrench handle is to the right, then it goes way from you.) It should come close to the aluminum, and then stop wanting to go further. It takes a firm but not large amount of force on the wrench to tighten it so that it seals to the aluminum block. Remember to balance the torque you apply to the nozzle with counter-torque on the 10 mm wrench.
18. Let the head cool again, to 40C or below.
19. Replace the "sock". You will probably need to start it on all four corners and then pull it up on each corner in turn. It should hang a bit below the block but not as low as the nozzle.
20. Carefully fold the paper towel up so as to capture all debris left by these operations, and dispose of it so that you do not spread the glass fibers around.

The new nozzle may be slightly longer or shorter than the previous one. You will need to check the Z Height adjustment.

Problems and Solutions:

Printing Stopped - "Check Filament"

This means that one of the following conditions probably exists (in order of decreasing probability):
- No filament present - it has run out, broken etc.
- Filament not unwinding easily from spool - check for strands crossing each other
- Filament sensor (by magnet force) not closing all the way
- Filament sensor needs adjustment (if senses most of the time but not all the time)
- Broken part on filament sensor
- Filament slipped out of position
- Broken connection or wire at filament sensor
- Magnet came loose

Most Likely: Look for a problem with the filament binding up on the roll, or having run out. Clear up the problem, close the filament switch, and printing should resume.

Printing Stopped - no messages

If this happens to any print, it's really helpful to do <ALT>-PrtScn on PolyPronter at the time,

paste the screenshot into MS Paint, and then send that plus the STL and the exact GCODE being printed, to support@polyprinter.com for diagnosis. As well, it's good to know whether the printer is responsive through direct commands with PolyPronter e.g. raising the head a bit.

Head-Contacting-Bed Error

When starting a print

Check the bed tape for holes or tears at the front left corner of the bed, especially, or wherever the head got to. (Bed-contact messages imply there's a hole or tear.)

Otherwise, possibly someone adjusted the head too low and it needs to be raised. But only do that if there's no other apparent reason for the message.

General Head-height checking

A basic test is to position the X and so the head is in the middle and let it (with nozzle hot) pinch a piece of paper. (Check for messages at that point).

Raise 0.2 to 0.3 mm and the paper should be virtually free by 0.3mm. Check a few areas around the bed - it should be fairly consistent.

If there's a large variation, the bed has become unlevel. That normally doesn't happen for no reason, and the cause should be found. One quick thing is to try pushing down on each bed mounting rail in case it got pulled up. If it snaps back down it will probably be level again (and the bolt should really be tightened).

PolyPronter Connection Problems

If using PolyPronter, hit "Disconnect" first. Then hit the red reset button on the lower left side of the switch housing that's in front of the left vertical aluminum bar.

Then hit "Connect" on PolyPronter.

If it still fails then try a more robust procedure of
- Close PolyPronter
- Try another USB port(Listen for Window's "A device has been connected sound)
- Turn off the printer for 10 seconds
- Open PolyPronter and try "Connect" again.

If it's still no good, proceed to "Check the Port" below.

Check the Port

Open Device Manager, and look at the "Ports (COM & LPT)". It should list "USB Serial... COMxx). Make sure PolyPronter has that port selected. You may need to press the Port button

on PolyPronter, or restart PolyPronter to get the correct port number to appear.
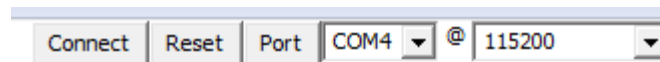
Try pressing the Reset button   , then press Connect again.

- If you see "M104" then the serial port is incorrect. Check the dropdown box and select any alternate port it has there. You may need to go to the WIndows Device Manager and see what serial ports are listed. it will likely be the only one listed. If pressing the Port button doesn't add the correct port to the list, you can edit the displayed COM… port text directly to match what's shown in Device Manager.
- If you do have the correct port, and nothing happens, open the front cover, and press the red Reset button on the Polyprinter, and then try the "Connect again".
- If still nothing happens, disconnect the USB cable, open the front cover, and press the red Reset button on the Polyprinter, reconnect the USB cable, and then try the "Connect again".

If it says "Connected..." you are all set. Try making the carriage move to make sure.

If the issue is still prevalent, please contact Support@polypronter.com

Parts are not sticking to the bed

- Ensure that the Z height is correct, and that it is level left-to-right.
- If the bed tape is scratched up, it may be time to change the tape.
- If people have touched the bed very much, it will get finger oils on it and the parts will not stick well.
  - You can sometimes freshen the bed up with acetone.
  - You can simply re-tape the bed instead of using acetone.
- Some parts simply don't have enough area on the bottom layer to stick well. See below:

## My small parts tend to get knocked loose during printing

First, see above for parts not sticking in general. But if all of that is in order:

Thin Parts
Some parts have almost no contact with the bed. If they also don't require much support, then there maybe so little bed contact that they come loose or tip over during printing, when they become taller.
Solution: Use a Brim (Support tab - choose Normal with Brim. A 0.25 or 0.5 mm thick brim of 5-10 mm width usually helps a lot.)

If it is a part you have designed, you can add one or two semicircles of one layer thickness to opposite corners (for example) to provide some additional area. These "mouse ears" are easy to peel off after printing.
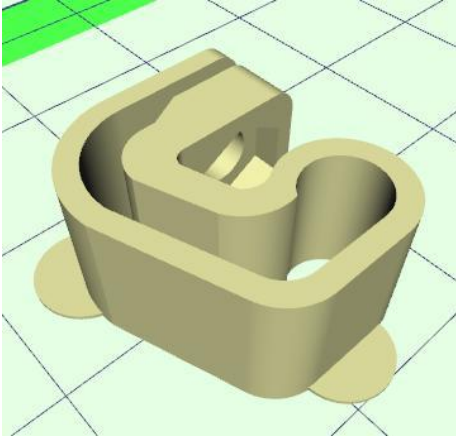
This also helps prevent corners peeling up, if the ears are made large enough.

Tips

You can make your own glue by dissolving some scrap filament in some acetone. The ABS thickens it up a bit (you decide how thick) and that lets it stay in place and fill in gaps a bit.

Acetone all by itself works well too, if the parts fit closely together. Brush it on both, push, hold, and they just melt together.

Any type of "krazy" glue, Cyanoacrylate, works well too (but can make a white film that shows up on dark or clear parts).

For strong yet flexible bonds, one of the"goop" type products will work well e.g. Automotive GOOP. Kiwi Shoe-Patch is the best.

Warranty

We want all PolyPrinter owners to be happy with their purchase. We feel that including a reasonable-duration parts warranty is one thing that can help. We want to be responsible for providing a machine that operates as it was intended and designed to do.

For a period of twelve months after a new PolyPrinter has been unpacked and begun to be set up, PolyPrinter will send replacement parts free of charge, if the part was found to be defective. Polyprinter will pay shipping both ways for such parts (if we want the old one back)..

Unfortunately it is not economical to provide on-site labor to install replacement parts, and shipping the entire PolyPrinter is very expensive. Therefore, this warranty is offered with the understanding that the customer must be prepared to install parts, or find someone local to do so. We will gladly try to assist with advice or instruction in replacing the parts. The entire PolyPrinter can be disassembled and reassembled with hand tools.

If you prefer to have PolyPrinter perform the labor, then you may send your machine to us at your expense (both ways) and we will do the repair, with no labor charge.

Alternatively, only at our discretion, we may deem that there is some reason why the printer must be returned to PolyPrinter for repair. (This is why it is important to keep the box and all packing materials somewhere.) We will in that case pay for shipping in both directions as part of the warranty.

More Formally:

The PolyPrinter has a 12-Month "carry-in" warranty.

The warranty covers:
- failures due to defects in manufacturing
- design problems preventing functionality promised to at the time of purchase
- two-way shipping of replacement parts
- labor at a designated repair facility
- forum, email, and live telephone support for diagnosis and repair
- breakage or wearing out of vital things that are not nominally wear items:
    - bearings
    - rods
    - belts
    - heaters
    - electronics

- ○ printed bits
- ○ wiring
- ○ fans

It does not cover:
- Obsolescence
- Mistreatment
- Consumables or wear items. (e.g. bed tape, nozzles.)
- Shipping the entire printer
- Cosmetics
- Damage
- on-site labor

After the Warranty Expires

We will generally allow renewal of the Warranty. This will maintain coverage of the abovementioned items, and also the Telephone support privilege that goes along with it.

Continuous Improvement vs New Models and Versions

We intend that the PolyPrinter will be continually evolving. We intend to keep improving it in minor and major ways for a long time. Each PolyPrinter represents the "state of the art" at the time it was manufactured. The existence of newer PolyPrinters with improvements does not by itself imply that your PolyPrinter has a defect that must be covered by the warranty. The warranty only covers things that were not manufactured correctly or could not fulfil their intended function to the degree they were specified to at the time.

That being said, however, this does not preclude us from offering customers the ability to self-upgrade their machines by printing replacement parts or purchasing updated replacement parts to keep their machines up to date. We will be happy to assist in this process with information and advice so that all our customers can have the best practical machines.

We intend to also have generational changes that may represent a new starting point, such that improvement will no longer be made to a given model. That is a normal part of a product development cycle. We therefore cannot predict for how long we will improve a particular generation in a manner that is compatible with existing PolyPrinters.

Open Source Software

The PolyPrinter comes with several open-source packages. The installation puts the license files in place at the time. Access to source code is available upon request.

The Gcode M115 gives information on the open source software that is part of the PolyPrinter itself.