

3D Printing Procedures

Table of Contents

General Printing Procedure.	1
Tips for Printing PLA.	2
Tips for Printing ABS.	2
Tools Needed.	2
Storing Filament.	3
Preparing Sketchup Drawing for Printing.	3
Using NetFabb Basic to Make Repairs.	4
Setting Z Stop/Limit.	7
Leveling Z Axis.	8
Leveling Bed.	8
New Spool of Filament.	9
Calculate Remaining Filament.	9
How to Make ABS Juice.	9
Calibrating the Extruder.	10
Heating Bed.	10
Using Loops and Brims.	11
Determining Nozzle Temperature for Specific Filament.	11
Air Under Kapton Tape.	12
Jams.	12
Notes.	12

General Printing Procedure

- Raise Z 20mm
- Clean bed / extruder with denatured alcohol or acetone if really dirty
- Make sure extruder is off
- Turn bed on to 100C
- Wait 30 minutes for bed to fully heat
- Make sure Z axis butts against nuts on all-thread (right side comes loose)
- If you need to level bed, do so now using procedure
- Turn on extruder to expected temp
- Apply ABS juice to bed
- Wait for extruder to reach necessary temp
 - 230 for ABS
 - 190 for PLA

- If you need to set Z axis limit, do so now using procedure
- Place object
- Slice Object
 - Make sure slice settings are right
 - Recommend using Brim
 - Slice
 - Check Log for errors
- Extrude 10mm or so of filament to verify it is flowing
- Use the abs230_level.gcode to get fine tune bed leveling
- Start run
- Examine Loops of skirt as they print
 - Each loop needs to touch the other
 - should be a little flat, but not too much
 - If wavy, bed is too far away. If smashed flat, bed is too close.
 - adjust bed screws in 1/4 to 1/2 increments as necessary
 - Keep eyes on print for at least 4 layers, adjusting bed level as necessary

Tips for Printing PLA

Tips for Printing ABS

- Bed must be 100C
- Use correct temp for the ABS. If you don't know it for the spool in use, use the procedure to determine it (and write it on the spool).

Tools Needed

Here are the tools I always need:

- .1mm feeler gauge (or scraps of paper)
- IR thermometer
- locking scissors
- hobby knife
- miniature spatula
- phillips screwdriver
- wire cutters
- 2" painters tape (actually I just use the spool as a bed support these days)
- filament spool holder
- log book and pen
- paper towels
- denatured alcohol

- abs juice
- sharpie to put date and # on the print (I use a normal pen for final prints but I mark them all so I can find them in the log book).

and here are the tools I sometimes need:

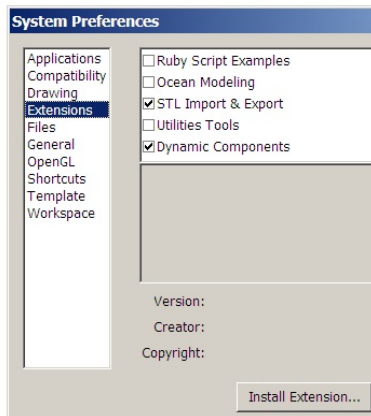
- dial indicator
- small wire brush
- monkey wrench
- digital caliper
- acetone

Storing Filament

- Filament needs to be stored in a low humidity location
- Put Crystal Kitty Litter (this specific brand) into an ice chest and keep open filament in the ice chest with the kitty litter
- I don't have a way to do this yet, but for long term storage, put filament into airtight bag with silicon desiccant.

Preparing Sketchup Drawing for Printing

You must have the Sketchup STL Export Extension installed. If it isn't, I keep the installable extension in f:\download\sketchup8. To verify if it is installed go to windows | preferences:



- Once your model is ready to export, select it.
- Click on File | Export STL
I've been using Millimeters/Ascii for export options so far.
- Upload the resulting STL file to cloud.netfabb.com to attempt to detect any errors

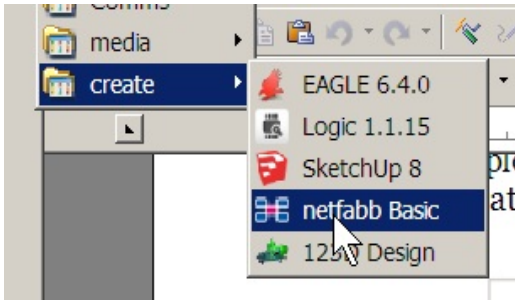
that will mess up the print:

- When you load the file into repetier for the first time, check it carefully for problems.
- I **strongly suggest** printing any file that is big at 1/4 scale first just to make sure there are no problems. No sense in wasting filament.

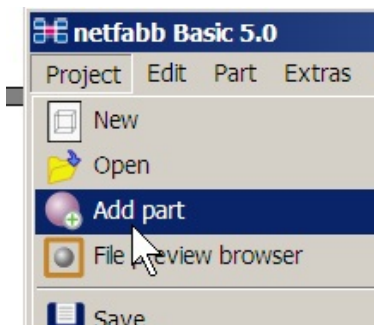
Using NetFabb Basic to Make Repairs

Rather than using cloud.netfabb.com, repairs can be made with the NetFabb basic program. At this point, I would only do this if cloud.netfabb.com can't make the repair.

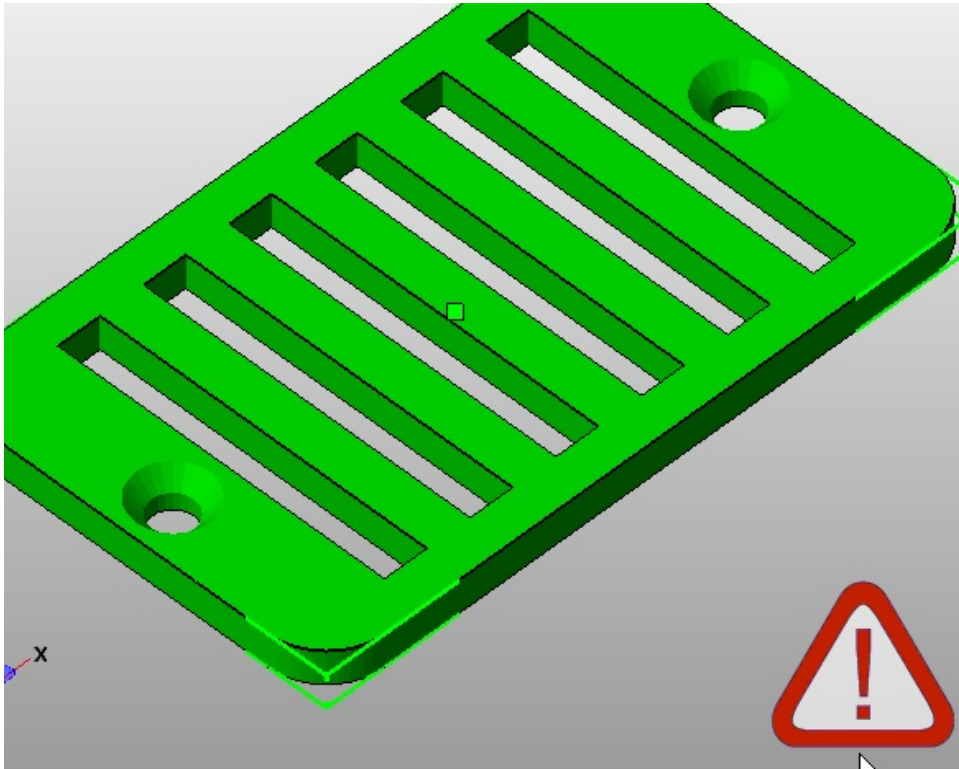
- Run NetFabb Basic:



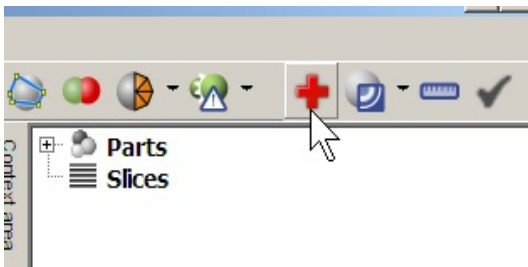
- Add the STL file generated by Sketchup:



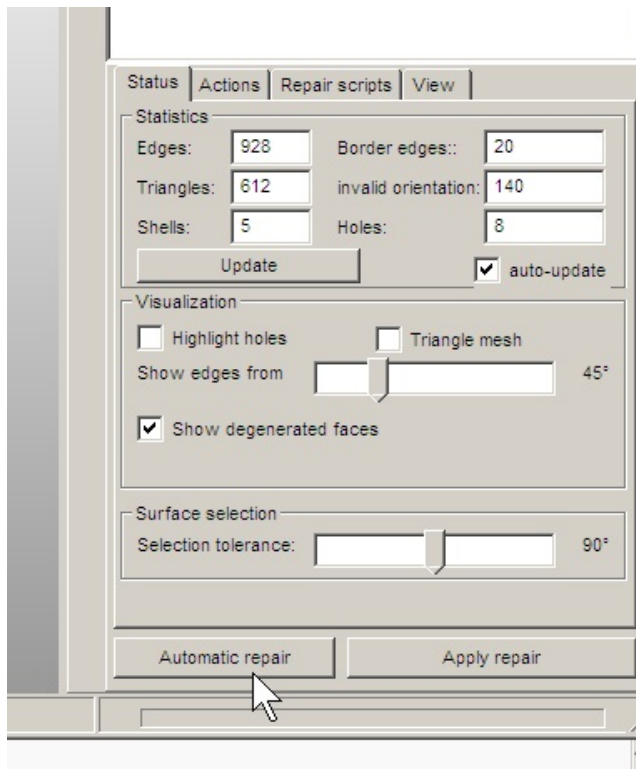
- If this symbol is displayed you've got a problem:



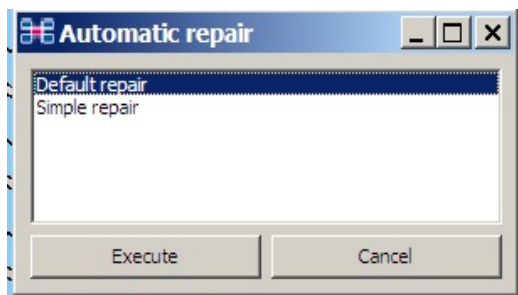
- Click on the repair button:



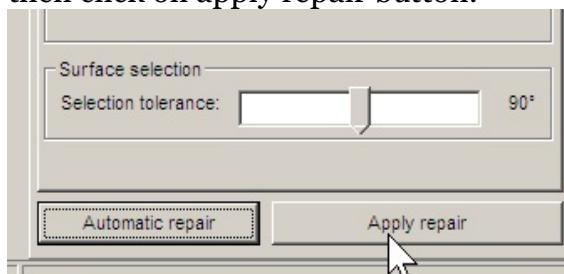
- Click on the Automatic Repair Button:



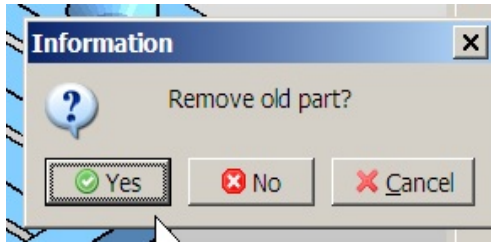
- Select Default Repair and Click on Execute Button:



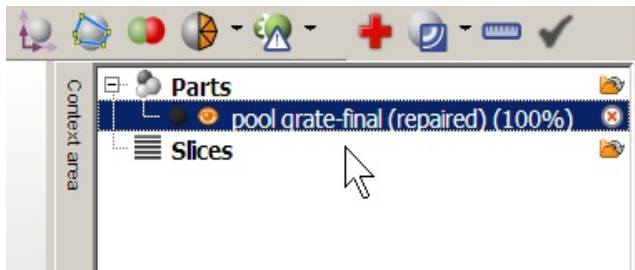
- then click on apply repair button:



- Click on Yes for Replace part:



- The broken part symbol should now be gone:



If it isn't you will need to do a manual repair which is beyond the scope of this document.

Setting Z Stop/Limit

- Home All Axes
- Turn motor off
- Raise Z 1mm
- Put sheet of paper under extruder
- Home Z axis
- Can you slide paper with just some friction?
 - If too loose, extruder is high. Turn limit screw about 1/4 turn left (left = lower)
 - If too right, extruder is low. Turn limit screw about 1/4 turn right (right = raise)
 - Home Z axis again and repeat until you get the right amount of friction.
- Slide extruder all the way right with paper under it - check for friction as you do this.
- If the paper gets too loose or tight the bed is out of level. Follow Bed Leveling Procedure.
- Move Bed to the rear checking for tightness. Again, if it isn't right, follow Bed Leveling Procedure

Leveling Z Axis

I have had issues with the Z axis getting out of whack - the nuts on the all-thread weren't being consistently turned the same amount. This problem was due to lack of lube on the all thread causing one stepper motor to bind up some times in which case the screws' height would go out of sync.

- Raise Z up high enough to get the old shipping restraint under the left side. Lower the extruder 1mm at a time until you exactly hit the restraint.
- Now try to slide the restraint under the right hand side (w/o changing height). It should fit exactly the same. If it does not, move the nut up or down on the all-thread until both sides are the same height.
- You **must** now level the bed.

Leveling Bed

Before Leveling Bed, do the Leveling Z Axis Procedure if you suspect the Z axis is not level.

I have two procedures: Quick and Dirty and Accurate:

Quick and Dirty Bed Level

- Set the Z Limit if you haven't already
- Move the extruder to the right with the paper under it.
- As you move, if the paper catches, lower the bed at that corner (tighten screw). Once you get to that corner, the paper should have the normal amount of friction on it.
- Repeat for right rear corner, then left rear corner, and come back to home position and make sure it is still reasonable.

Accurate Bed Level

- Home all axes
- Raise Z about 20mm
- Install dial indicator onto bracket
- Lower Z until dial indicator comes in contact with bed
- set zero or make note of indicator
- Note that dial indicator can't measure far left corner. That is a limitation of the bracket.

- Move X axis right 10mm at a time checking the indicator. Adjust the bed leveling screw as necessary.
- Once at the end of X axis, move toward the end of the Y and adjust that screw as you go.
- Then go back to the rear left corner, adjusting screw.
- Finally return to home, checking that everything is still the same.
- REMOVE DIAL INDICATOR!

New Spool of Filament

When you get a new spool of filament:

- Make sure the type of plastic is noted on the spool
- Weigh the spool, subtract the weight of the filament, to get the weight of the spool and mark it on the spool so you can calculate the weight of the filament left.
- Measure diameter of spool in 4 locations and average to get the real diameter of the filament. Write it on the spool as well.

Calculate Remaining Filament

- Determine weight of empty spool. Either you figured that out when the spool was new (total weight - weight of filament purchased, usually 1kg). Or weigh an empty spool that is the same.
- Weigh the spool + filament
- Subtract the weight of the spool
- To determine the weight of the filament in cubic centimeters:
 - PLA: $cc = \text{weight in grams} \times 1.25$
 - ABS: $cc = \text{weight in grams} \times 1.06$
- Now compare the weight of the remaining filament with the weight in cc's displayed by the slicer to determine if you have enough filament.

How to Make ABS Juice

- Put 1 cup of acetone into a denatured alcohol container (or other HDPE container - which acetone can't melt)
- Measure out 36" of 1.75 ABS filament
- Cut up into roughly 1" segments
- Place the cut filament into the bottle
- wait 24 hours

Calibrating the Extruder

- Go to Firmware settings, and make note of the current extruder steps (I write this in my log book):

The screenshot shows the Marlin Firmware EEPROM Settings interface. The 'Steps per mm' row has four input fields: X: 63.36, Y: 63.36, Z: 2272.72, and E: 543.83. The 'E' field is highlighted with a red rectangular box. Below it are 'Maximum feedrates [mm/s]' (X: 100.00, Y: 100.00, Z: 2.00, E: 14.00) and 'Maximum Acceleration [mm/s²]' (X: 2000, Y: 2000, Z: 30, E: 10000). At the bottom is an 'Acceleration:' field with the value 3000.00.

- Measure out 100mm on a drinking straw and cut to length, then slit open
- Use straw and mark both end of filament to measure out 100 mm on the filament
- Using the 1mm extruder feed, align first mark with top of extruder mechanism.
- Set extrude length to 100mm and click on extrude
- Mark where the top of the extruder really is on the filament
- Measure the difference
- Compute new Extruder Steps / mm:

$$\text{newSteps} = \text{expectedLen} \times \text{oldLen} / \text{actualLen}$$

If my current extruder steps / mm is 590.00, and the actual length of extrusion was 108.49mm, then the formula is

$$\begin{aligned} \text{newSteps} &= 100\text{mm} \times 590.00 \text{ s/mm} / 108.49\text{mm} \\ \text{newSteps} &= 543.829 \end{aligned}$$

- Put the new steps / mm into EEPROM.
- Run the test again to verify it is extruding the correct amount.

Heating Bed

The thermistor for the heated bed is on the bed heater, not the top of the bed, so it is not a very accurate measurement of the actual bed temperature.

Instead, I check the bed temp using an IR thermometer.

If you set the bed temp to 50C, then the thermistor reaches that temp quickly and cuts back the heater so it takes longer to get the top of the bed hot.

Instead, set the bed temp at 100C (the limit), and check the bed with IR thermometer until it gets to where you want it, then back off the temperature setting to where you really want it.

I am now in the habit of not setting the bed temp in the filament heat section of the slicer. There I have it set to 0 (don't do anything). I then manually control the bed heat myself before the run and during the run if I find it necessary to make a change.

Using Loops and Brims

Loops

You **MUST** have at least 2 loops. This give the printer a chance to get filament flowing smoothly and it gives you a chance to verify the extruder 1st layer will be the proper height and the printer bed is level.

Generally I use 3 loops. If I'm printing something big and am in a rush I go to 2 loops. But since big things take forever, that doesn't usually make sense, so 3 loops is the norm.

Brims

A brim gives the object a bigger foot print so it adheres better to the bed.

Prior to ABS juice, the only way I could get objects to stick to the bed was with a 5mm brim.

With ABS juice I really haven't needed brims, though if I were printing something with a small base that would be fairly tall I would still use them.

Determining Nozzle Temperature for Specific Filament

(this is from Triffid Hunter's Calibration Guide)

Each type of plastic, and each colourant for each type of plastic alters the ideal printing temperature. E.g., I can print opaque PLA at 165°C with fantastic results, but my translucent PLA prefers 180°C!

To find optimum temperature for each roll of filament:

- Choose a fairly simple model that's large enough that you can clearly see the infill while it's printing
- Make sure your hobbed bolt's teeth are clean of debris such as chunks of plastic
- Make sure your idler is tight! really tight! "it hurts my fingers to pull on it and I still can't move it" tight! A too-loose idler gives exactly the same symptoms as too low temperature.
- Start printing

- Lower temperature by 5° every 2-3 layers
- When infill starts being a row of dots instead of a line, increase temperature by 10°.
- Keep monitoring print, increase by 5° if your infill goes dotty again If you find that your prints are weak along the layer lines or even delaminate mid-print, you may need to go higher again.
- With ABS, wrapping your printer in a towel helps a LOT by keeping out draughts and breezes- but beware any PLA parts caught within!
- Store or remember that temperature for that type of filament

Air Under Kapton Tape

I seem to periodically get air under the kapton tape even though it seems to be attached well.

The stuff is pretty easy to peel back and then reapply it smoothly to get the air bubbles out.

Jams

If filament jams before hot end (right after hobb bolt), hobb bolt is too tight. Loosen a long ways then start calibration print. If filament flow is erratic, tighten bold 1/2 turn and keep monitoring until it is steady.

Notes

[Triffid Hunter's Calibration Guide](#)

Notes on how to Calibrate about anything that could use calibrating.