

Beyond PLA

**All sorts of stuff you can print on
your 3d printer**

Also:

Making prints stick / avoiding failed prints

Presentation by Rich Olson

www.nothinglabs.com / rich@nothinglabs.com

All the filament discussed

- Can print without a heated build platform!
- Are available in 1.75mm diameter
- Have less well understood safety than PLA
 - Print in a well ventilated area
- Might be more prone to clog your nozzle or jam your stepstruder
 - Be comfortable with taking things apart...
- Are experimental
 - Some are easy to print - some are really hard
 - Have patience, experiment, expect failed prints and learn from them
- Notes based on Replicator 2 / Makerware
 - Should be generally applicable to other printers / slicers

LAYWOO-D3

- Some kind of plastic with about 40% wood particles added
- Prints look like (and smell like) particle board
 - Parts don't feel like typical 3D prints
- Easy to print
 - Sticks well to blue tape / print up to about 90 mm/s
- Darkness changes with print temperature!
 - Prints lighter at 185c / darker at 240c
- Resulting print is softer / weaker than PLA
 - Hardens somewhat after printing
 - Filament is easy to break
- Available from matterhackers.com and others

LAYWOO-D3

Images from <http://www.tridimake.com> www.3dprintergear.com.au and www.soliforum.com

Wood bands generated by script adjusting temp.

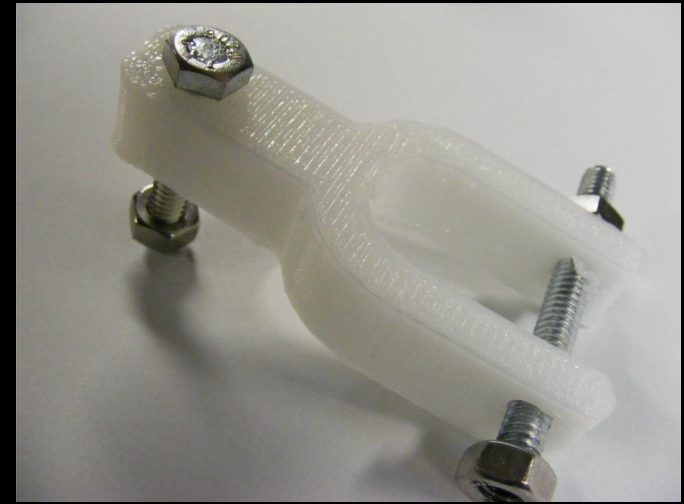


Taulman Nylon 618 / 645 Properties

- Very, very strong and tough compared to PLA
- Much trickier to print than PLA
- Nylon 618 = natural white
- Nylon 645 = fairly clear (difficult to work with)
- Can dye either with fabric dye
- Absorbs water from air - keep someplace dry
 - Very important in humid climates / seasons
 - Nylon can re-absorb moisture in 18 hours (Taulman)
 - Symptoms: Heavily steaming print nozzle, foamy prints that easily delaminate
 - Large zip lock bag with silica kitty litter should work

Images from <http://taulman3d.org>

Dyeing filament before print provides a tie-dye effect



Taulman Nylon 618 / 645 Printing

- Doesn't stick to bare blue tape that well
 - Easy solution: Blue tape + glue stick = good results!
 - Taulman suggested surface is Garolite LE
 - More prone to curling than PLA
 - Large/dense objects may need a heated build platform
- Print hot - around 250c
 - If it steams / bubbles more than a little - try a bit cooler
- Print slow - 44 mm/s or less to avoid delamination
 - If having delamination problems - go slower still
 - May need perimeter speed as low as 20 mm/s for best strength (custom profile)
- Consider turning off PLA cooling fan
 - Can create delamination issues / may or may not help with curling
 - From menu when printing or "doFanCommand = false"

Taulman Nylon 618 / 645 Printing

- Possible issues with “under-extrusion”
 - Nylon is slippery / stepstruder may have problems grabbing
 - Results in prints that delaminate / look crappy
 - Test: Pull on filament when loading - easy to stop feeding?
 - Try increasing stepstruder plunger / spring tension
 - Specifying a slightly smaller filament diameter than measured (maybe 1.68mm vs 1.75mm) may help
 - Results in more filament extruded (same effect as reducing feedStockMultiplier)
 - Printing slower may also help (as low as 20mm/s)
 - Try decreasing layer height (100 micron)
- “Stringing” can be reduced with custom profile
 - Increase all instances of retractDistance to 3.6
- Have not had great luck with Nylon 645
 - Very difficult to get to feed reliably / nozzle jams
 - Might be easier to print with larger nozzle diameter

Taulman Nylon Safety and Availability

- Potentially emits CYANIDE at high temperatures
- Taulman's testing suggests this isn't a problem at printing temperature
 - Print in well ventilated area...
- Sold at www.taulman3d.com
 - Nylon 618 \$20 / 1lbs spool
 - Nylon 645 \$30 / 1lbs spool

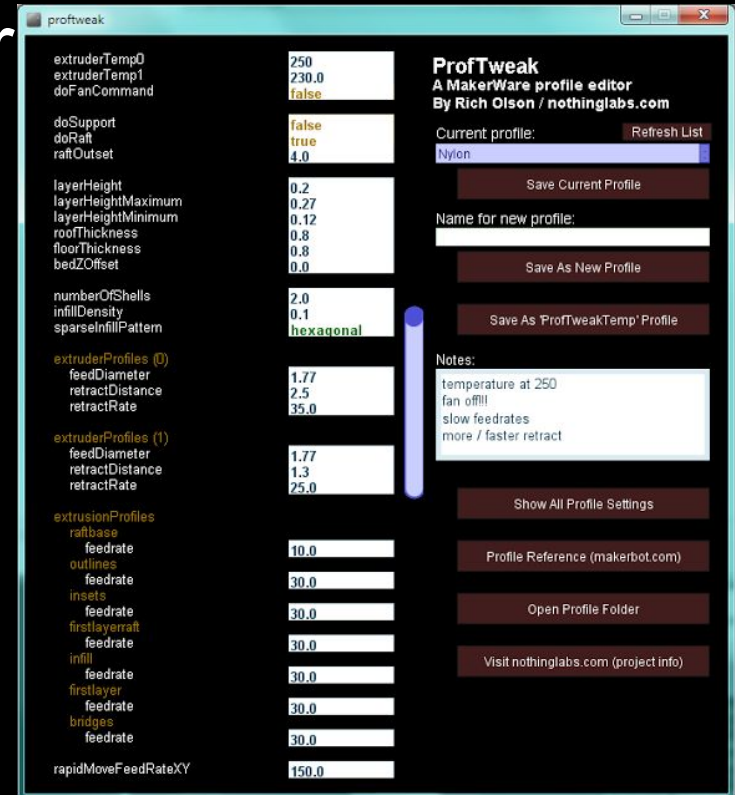
MakerWare Custom Profiles

- MakerWare doesn't really let you fully control print speed from the GUI (as of version 2.3)
 - Perimeter ("Outline") speed is locked at 40mm/s no matter how low you set it*
- You may need to tweak "deeper" settings for good results with some filaments
 - Really reduce all print speeds
 - Turn the fan off
 - Increase or decrease "retract"
 - Print with a larger raft, eliminate the anchor, etc...
- You can do all this stuff by creating a custom profile and using a text editor to adjust it (or...)

*Check `c:\windows\temp` (or equivalent) to see the `.config` file MakerWare generates for your print

ProfTweak - a MakerWare profile editor

- Something I tossed together for my own use
- Let's you edit common (and less common) profile variables
- Experimental - use at your own risk (but let me know if you find problems)
- Get it for free at <http://nothinglabs.com>



Print Troubleshooting

- **Avoiding air prints**
- **Making your print stick to the platform**
- **There's more filament to come!**

Problem: Air prints / extrusion fail

- Poor extrusion common cause of air and craptacular prints
 - “Click of death”
- TEST: Perform a filament load - you should be able to pull on the filament with moderate force and have it still continue feeding.
- Makerbot Replicator users:
 - If you have the stepstruder upgrade - make sure pieces haven't warped from heat
 - If you're still using the delrin plunger - get the stepstruder upgrade
- Verify your filament has consistent diameter
 - Some variance is OK / normal (1.65mm - 1.75mm seems typical for “1.75mm” filament)

Problem: Air prints / extrusion fail

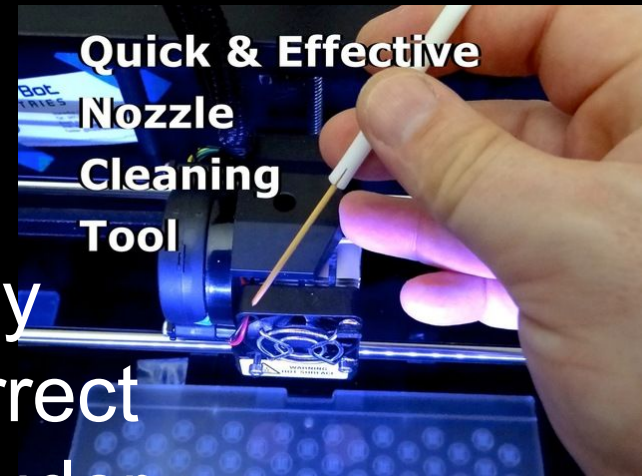
- Does the failure happen on the first layer or soon after?
 - Your build platform may be too close to the nozzle
- With the extruder heated can you push through filament by hand with a moderate amount of effort?
 - If not - your nozzle may be jammed. Consider replacing it (get a bunch on eBay cheap)
 - This test is subjective - try it on a printer that's working to get an idea of how much force is typical
 - Semi-jammed nozzles are common - might cause problems with some filaments - but not others

Problem: Air prints / extrusion fail

- Having problems but don't want to take apart your bot at the moment?
 - Some hacks might get you printing
- Place a small amount of vegetable oil on the end of the filament before loading
 - This really works!
 - Don't ask me why
 - Re-apply as needed (even mid-print)
 - Use a high temperature oil like canola (no EVOO)
- Reduce print speed
 - This reduces the amount of strain on everything

Nozzle Unjamming

- Some jams can be dislodged by forcing an implement of the correct size through the top of the extruder while heated
 - Long 1.5mm allen wrench
 - Whphtomas's toothpick-based unjamming tool (image)
 - <http://www.thingiverse.com/thing:56597>
 - Anything else around 1.5mm and long enough
 - This doesn't always fully work - sometimes you just need a new nozzle



Problem: Prints not sticking / curling

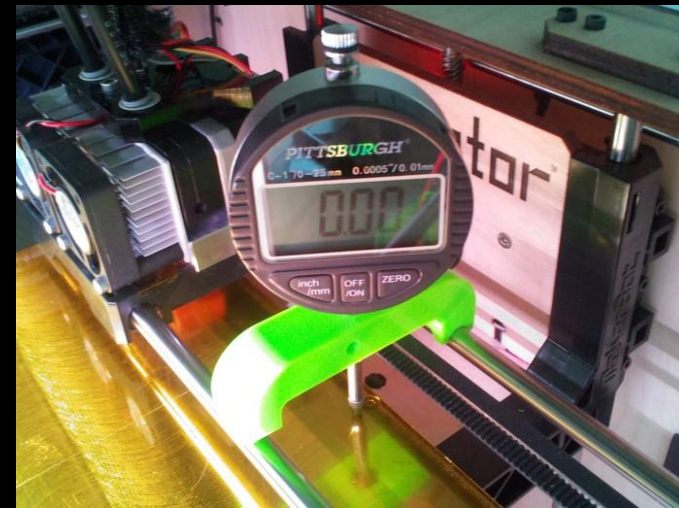
- Poor adhesion
 - Build plate not flat, level or at wrong height
 - Build surface material and cleanliness
- Curling (corners lift up)
 - Caused by plastic shrinking when cooling unevenly
 - Good adhesion can minimize
 - Minor issue with some materials (PLA) more of an issue with others (ABS, Nylon)
 - Some printers use heated build plates or chambers to reduce / eliminate
- Rafts, helpers discs and changing your print settings can help

Is your build plate flat?

- Non-flat build plate is common cause of adhesion / surface quality issues
- Leveling your build plate 3x times - to find the height in the center is too low or high?
 - Your build plate may not be flat
 - Acrylic build plates aren't great in this regard
 - Not that thermally stable - problem may develop over time
 - If your build plate isn't flat - it can never be level!
- You can use a dial-gauge to check how flat it is

Dial gauge

- Get a dial-gauge (\$15 ebay)
- Print an appropriate mount
 - <http://www.thingiverse.com/thing:23257>
 - many variants for different gauges / bots
- After leveling - push your print head aside - and move the dial gauge over the platform
 - Careful not to bump the Z-axis (it's easy)
- More than +/- 150(?) microns is enough to cause problems (note where issues are)
- Also useful for leveling

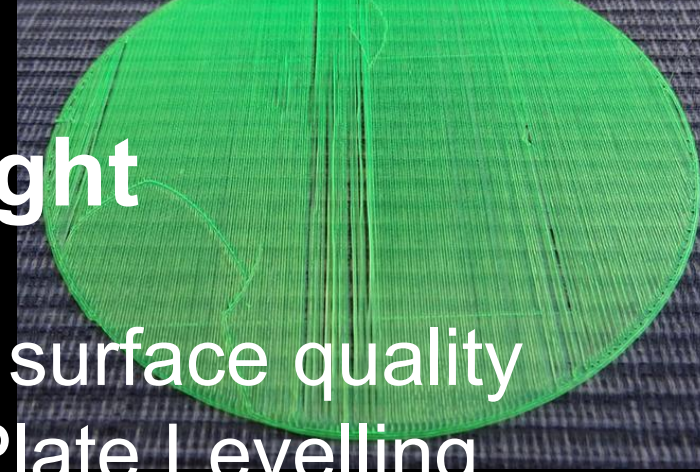


So your build platform isn't flat...

- Get a new one
 - Custom glass and aluminum drop-in replacements are available on ebay
 - I use a $\frac{3}{8}$ " glass sheet (covered with blue tape) with my Replicator 2 (custom cut for about \$35)
 - Look around online for info on cut size / options for your bot
- Flip it over
 - The other side may be flatter!
- Avoid the problem areas
 - Print around the problem
 - Strategically re-level for maximum usable space

Build Plate Level and Height

- Critical for print adhesion and surface quality
- Useful tool: Interactive Build Plate Levelling
 - <http://www.thingiverse.com/thing:39430>
- Actively adjust levelling knobs while printing / observe print quality
 - Stringy = too far / Bumpy = too close
- Use dial-gauge to note optimal height for future leveling (unique to bot / may change with time)
 - Set correct value over the screws - and you're done!
 - Makes leveling / height tweaks a bit more objective

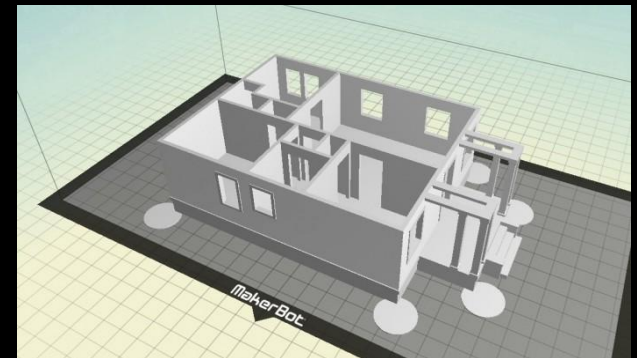


Printing Surface (Blue Tape)

- 2 x 3" wide strips cover full platform
 - Easier to work with than full sheets
 - Single strip down middle for small prints
- Cleaning tape with small amount of 95+% alcohol vastly improves adhesion
 - Cleans off any oil / grime
 - Also improves fresh / clean tape - so something else too?
 - Downside - print sometimes tears tape on removal
 - Tape is cheap / easy to replace
 - Much rather replace tape than have 2+ hour print peel up

Rafts, Helpers Disks and Print Settings

- Rafts increase effective surface area to improve adhesion
 - Makerware 2.2 rafts are improved / easier to remove
- Still curling up? Try a bigger raft!
 - Custom profile - increase "raftOutset" from 4 to 8
- Helper disks work same way
 - Place on corners
- More density = more curling
 - Reduce infill if $>10\%$,
 - Keep shells to 2
- Cooling fan may *increase* curling in some materials
 - Turn off from menu when printing or set "doFanCommand" false in custom profile



LAYBRICK

- Filament that prints like sandstone
 - Some kind of plastic with something added..
- Easy to print / sticks well to blue tape
- Parts are sandable / paintable / dyeable
 - Easy to clean up any print artifacts
 - Rit fabric dye in hot water results in nice matte colors
- Tends to hide extrusion lines
 - Almost invisible on 100 micron prints
- Surface changes with temperature / speed
 - Hotter (210c) = more rough
 - Cooler temps (170c) = smooth surface
 - Slower perimeter = smoother (25 mm/s = nice!)

LAYBRICK

- Softer / not as strong as PLA
 - Filament is easy to break
- Can make parts that really don't look printed
- Great option for organic / artistic prints
- Available from makergeeks.com
 - \$40 / 0.25kg (unspooled)
 - Don't try to spool unspooled filament (it'll end in tears)



Polyethylene Terephthalate (PET)

(matterhackers.com)

- Related to PETG used for pop bottles
- Easy to print
 - Sticks to blue tape. 230c / 90mm/s
- Rigid - but more brittle than PLA
 - Filament is easy to break
- Very clear / crystal-like
- Thick layer height = better clarity (try 340 micron)
- PET absorbs water from the air (hygroscopic)
 - Store in bag with silica cat litter when not in use
- Available from matterhackers.com
 - \$80 / 2.2lbs spool

Polyethylene Terephthalate (PET)

Image from bikealive.nl



Taulman T-glase “PETT”

- Rigid, tough and clear filament
 - Very stiff / about as tough as PLA (tougher than PET)
 - Unclear what PETT is vs PET (modifier?)
- Sticks to blue tape
 - Seems a bit picky about platform height - try moving platform a bit closer to nozzle if adhesion issues
- Can print at normal PLA speeds (90 mm/s)
 - 50 mm/s gives better interlayer adhesion
- Suggested print temp 212-224c (230c OK)
- ***Turning off the cooling fan may reduce curling***
 - From menu when printing or “doFanCommand = false”
- Thick layer height = better clarity (try 340 micron)
- Absorbs water from the air (hygroscopic)
 - Store in bag with silica cat litter when not in use

Taulman T-glase "PETT"

Available from <http://taulman3d.com/>

\$30 / 1 lbs spool



John Porter 3fproductdesign.com

PET+ (from MadeSolid)

- PET with some other kind of softener
- Claims “Easy like PLA. Strong like ABS” and not be prone to water absorption issues
- Is strong/tough stuff - needs few tricks for good results
 - Print hot - 260c to 270c and reduce inset and outline speeds to 30mm/s to avoid delamination
 - Reduce infill speed from 90mm/s to 60mm/s to prevent extruder skipping
 - Filament measured 1.65mm (correct for in slicer)
 - Reducing diameter in slicer a bit more (maybe 5%) to further increase amount extruded seems to result in stronger parts (or decrease feedStockMultiplier)
 - Cooling fan needs to be off!
 - Adhesion to blue tape isn't great - solved by applying glue stick
 - Had issues with roofs of some objects at 10% infill - better results with 20% infill
- Good option for tougher-than-PLA prints!

PET+ (from MadeSolid)

Available from Madesolid.com - \$35 / 1lbs (clear and various colors)

(Site has a nice demo video of PET+ vs. ABS)



Soft PLA / “Flex EcoPLA”

- What is it? PLA + some kind of softener.
 - Would be nice if they told you...
- Very soft and rubber-like
- Super tough
 - Very resistant to delamination
- Awesome - but tricky to print with!
- Adjusting infill density...
 - 5% infill = Rubber duck
 - 100% infill = Vulcanized rubber
- Possible uses:
 - Molds, drive belts, gripping surfaces, bath toys, tires

Soft PLA / “Flex EcoPLA”

pics from [petleh82 / youtube](https://www.youtube.com/user/petleh82), [BonsaiBrain / thingiverse](https://www.thingiverse.com/thing/1111111),
3dprintergear.blogspot.com



Printing Soft PLA / “Flex EcoPLA”

- Sticks OK (not great) to blue tape
 - Glue stick solves most adhesion issues
- Filament buckling / tangling inside stepstruder - multiple causes
 - Lack of support inside extruder
 - Latest Replicator 2 extruder upgrade has improved filament path that helps significantly:
<http://www.thingiverse.com/thing:53125>
 - Filament jams before getting to the extruder
 - Likes to wrap itself around filament spool holder
 - Sticks to guide tube and sides of spool / itself
 - Solution: Unspool amount of needed filament on floor - then feed directly into extruder bypassing guide tube
 - Semi-jammed nozzle can also cause buckling issue (even if other filaments print fine)
- Printing flexible filament on a bot with a “Bowden” style extruder is very problematic / may not be practical
- Printing slow (20mm/s) can also help with all the above issues

Printing Soft PLA / “Flex EcoPLA”

- Moderate overhangs can cause curling mid-print
 - **Increasing** number of shells may help (try 3 or 4)
 - Try decreasing speed especially perimeter (13 - 25mm/s)
- Previously loaded filament can result in jam early in print
 - Reload the filament before each print if needed
- Still jamming?
 - Increase distance between nozzle and platform (via knobs or set bedZOffset > 0)
 - Set anchorExtrusionAmount to 0
 - Reduce layer height to 100 or 150 micron
 - Reduce print speed
 - Increase temp. - maybe 260c (will cause more stringing)
 - Turn off or reduce retract (retractRate < 1.0)
- Various brand colors from makergeeks.com - “FormFutura” brand - \$45 / 1kg spool

FilaFlex and Ninjaflex

- Two new flexible filaments
 - NinjaFlex is softer than soft PLA
 - FilaFlex is even more soft
- NinjaFlex is TPE (thermoplastic elastomer)
- FilaFlex seems to be TPE (not totally sure)
- Both more rubber-like / less prone to deformation from stretching than soft PA
- Printing challenges are almost identical to Soft PLA - except may need to print even slower and hotter (15 mm/s 260c)
 - FilaFlex filament was only about 1.5mm (vs claimed 1.75mm) - had to correct in slicer for good results

NinjaFlex and FilaFlex



NinjaFlex - \$50 / 1lbs
from fennerdrives.com



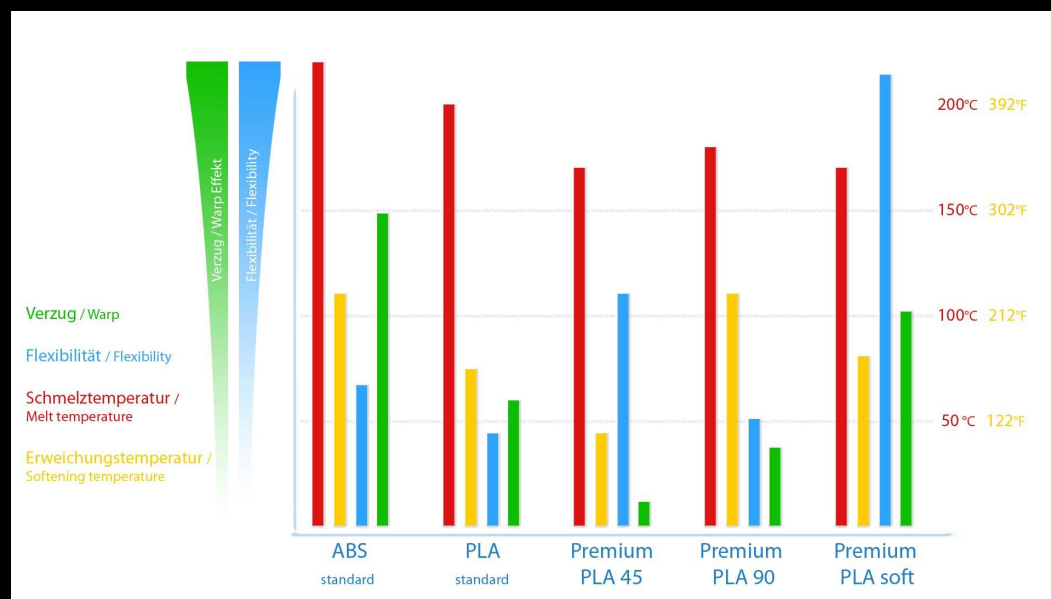
Filaflex - \$50 / 1lbs
from recreus.com

Obri-Tech PLA 45

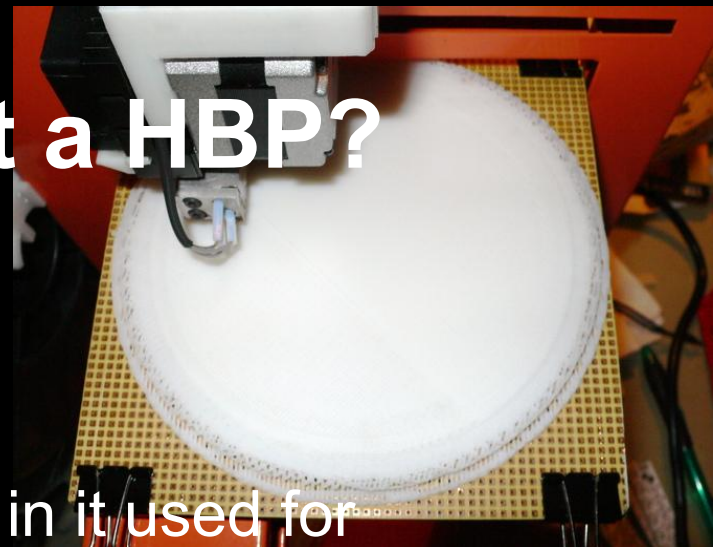
- Also PLA modified with softening agent
- Softer than regular PLA - but still hard
 - Parts deform / bend as opposed to break
 - Very resistant to delamination / tough
- Easy to print / sticks well to blue tape
- More resistant to curling than regular PLA
 - Works great for high-infill / solid prints
- “45”(c) refers to low softening temperature
 - Not good option for parts exposed to heat (>113f)
- Wide temp range / effects surface quality
 - 220c = textured translucent white
 - 150c = smoother / more transparent
- Currently only available in natural color

Obri-Tech PLA 45

- 3distributed.com (UK) for about \$50 + \$25 shipping for 1kg spool
 - No domestic sources I can find :(



Want to print ABS without a HBP?

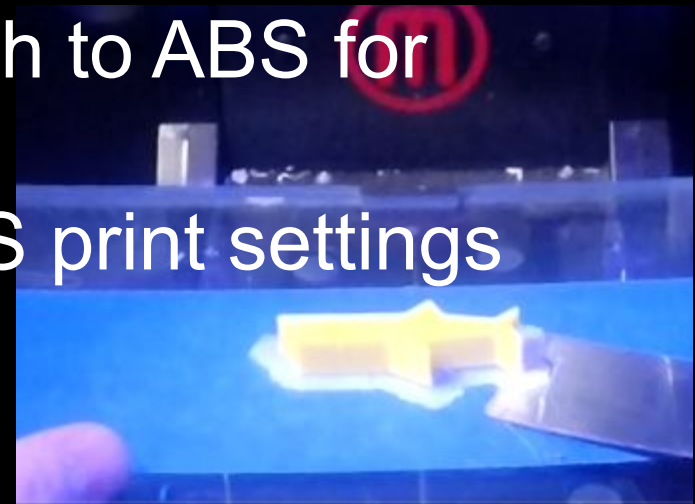


- Use perfboard!
 - Fiberglass with a bunch of holes in it used for electronics prototyping
- Tape it to your build plate
- Custom profile with “bedZOffset” set as appropriate
- Use a raft to deal with surface quality
- Info at

<http://www.thingiverse.com/thing:5428>

Want to print ABS without a HBP? (another way)

- Print raft in PLA - then switch to ABS for actual print
- Create profile based on ABS print settings
 - raftbasedensity to 1
 - raftbaseinterfacedensity to 1
 - raftmodelspace to 0
 - raftsurface thickness to .2
- This should give you a 1mm thick raft
 - Set z-pause to a height of 1mm to automatically pause for filament swap
- Works pretty well for objects up to moderate size



HIPS and Others..

- HIPS (high impact polystyrene)
 - Prints like ABS with same curling / adhesion problems - generally requires a HBP
 - Same tricks as ABS to print without HBP
 - Dissolves in Limonene solvent
 - Use support material / other interesting
- ColorFabb “PLA/PHA”
 - Supposed to be tougher than regular PLA - but I couldn't tell difference
- HIPLA / IMPLA (impact modified PLA)
 - Have not tried - hard to get in 1.75mm
- PVA
 - Have not tried - supposed to be water soluble



Buying PLA

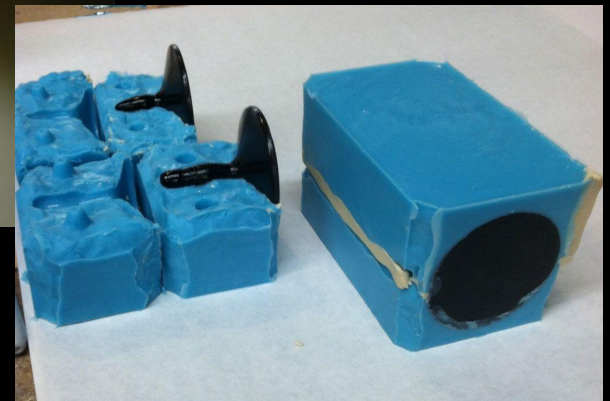
- Usually just check on Amazon to see if buyers are happy or not
- Bad filament
 - Inconsistent diameter / out-of-round (feed issues)
 - Bubbles (breakage)
 - I have generally not had PLA filament quality issues
- Filament coloring agents seem to effect print quality
 - Black filament often includes remains of other colors of filament
 - Have seen wide variance in darkness and print quality (curling) from same supplier
 - Some sellers offer “virgin black” at a premium

Other Stuff You Can Do

- Make silicone molds of your PLA print and cast in:
 - Urethane / epoxy
 - Lots of color / additive options (like fiberglass)
 - Low-volume production
 - Low temperature metal
 - Roto281F is nice lead-free alloy (easily melt with propane torch - cast into silicone mold)
 - “Easymold” silicone putty works great for simple stuff
- Lost PLA Casting
 - PLA burns out just like wax in a furnace
 - Traditional jewelry making techniques
 - Molten aluminum can directly vaporize PLA
 - Lower quality / larger “cope and drag” casts

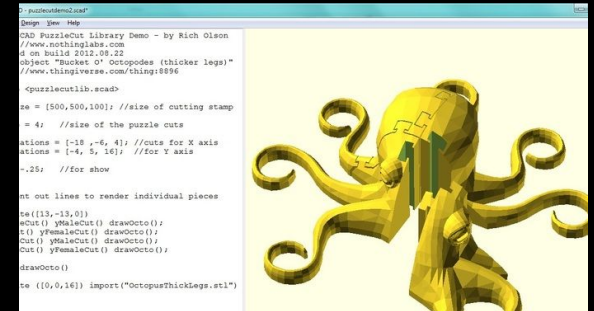
Other Stuff You Can Do

<http://boulderhackerspace.com>,
3dtopo.com, instructables.com /
“mepler”, hackerfriendly.com



Few things I've done with 3d printing

- **Elastostraps**
 - Generate 3d printed hook / loop fasteners of any size using OpenSCAD
- **Puzzlecut**
 - Cut STL files into smaller jigsaw style pieces for larger prints using OpenSCAD
- **3d Printed Bike Shifter**
 - Biked 2013 Seattle-to-Portland (STP) ride
- Details on all projects at ***nothinglabs.com***

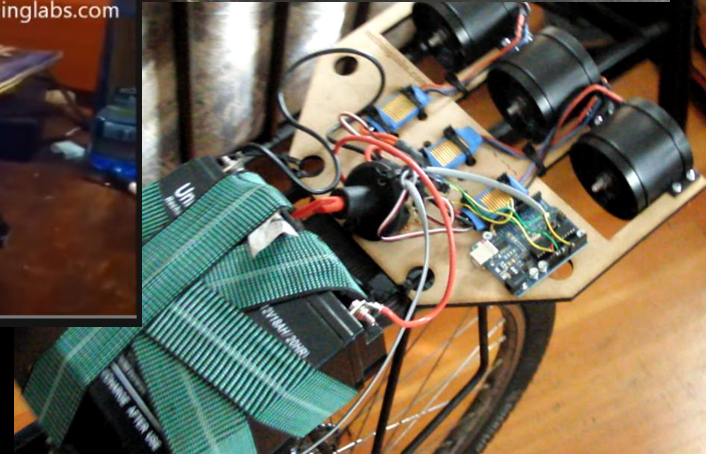
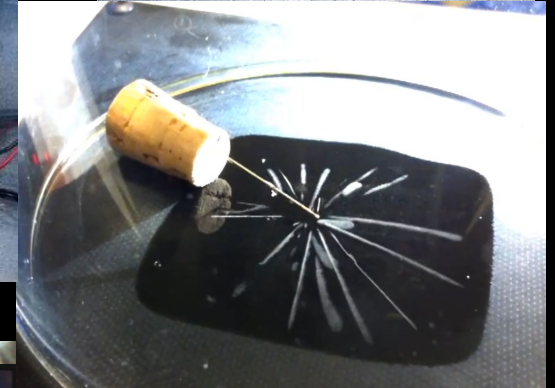
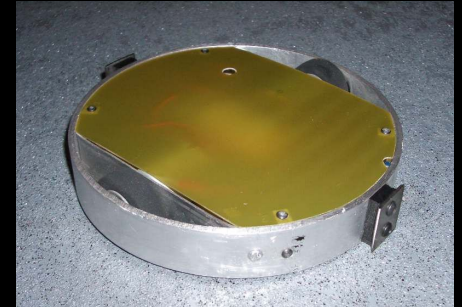
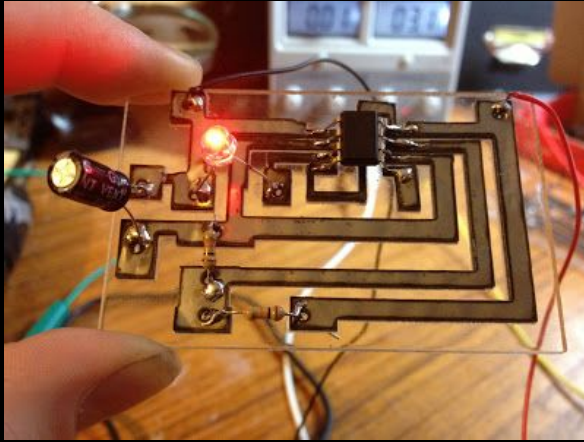


In conclusion...

- You can print lots of different things on your 3d printer even without a HBP
- PLA is great for ease of printing / consistency
 - But your project might call for something different and awesome....
- If you're 3d printing - you should be playing with experimental filaments!

Other Projects by me (Rich Olson) on Nothinglabs.com:

Electronic Cloud Chamber, Arduino-based Laser Show, Puzzlecut OpenSCAD script, money shredding alarm clock, 3d printed elastic hook/loop fasteners, lasercut circuit boards, alternative 3d printing filament guide, EDF "Jet" bike, translational drift combat robot control system, and some iPhone apps





Further Unabashed Self Promotion Rich Olson / nothinglabs.com



- rich@nothinglabs.com
- Twitter [@nothinglabs](https://twitter.com/nothinglabs)
- facebook.com/nothinglabs
- Have lots of random skills / resources
 - Robotics / electronics, laser cutting, 3d printing, embedded development, [iPhone dev](https://www.apple.com/iphone), making things work



Atmel

