

## Fused Filament Fabrication (FFF) material properties & considerations

<b>PLA</b> - (Polylactic Acid, a corn-based plastic)	<b>ABS</b> - (Acrylonitrile Butadiene Styrene, petroleum based)
Extruder temperature: 190-215°C (typical), up to 250°C	Extruder Temperature: 230-240°C (typical), up to 250°C
Heated Bed: not required, 50-60°C optional	Heated Bed: 110°C

Other specialty filaments:

<b>PET</b> - (Polyethylene terephthalate) <b>PETT</b> AKA t-glase	<b>HIPS</b> - (High Impact Polystyrene)
Extruder temperature: 207-235°C (212-224° optimum)	Extruder temperature: 230-235°
Heated Bed: 50-60° C optional (?)	Heated Bed: 50-60°C
<b>Nylon</b> - (i.e. Taulman 618 or 645)	<b>LayBrick</b>
Extruder temperature: 235-270°C (240-250° recommended)	Extruder temperature: 165-210°C (smooth/rough finish)
Heated Bed: 60-70°C	Heated Bed: not required, 40-60°C optional
<b>PVA</b> - (water soluble)	<b>LayWoo-D3</b>
Extruder temperature: 170-190°C	Extruder temperature: 175-250°C (light/dark color change)
Heated Bed: not required, 50-60° optional	Heated Bed: not required, 50-60°C optional
<b>BendLay</b> - (modified Butadiene; food-safe)	<b>TPU</b> (Thermoplastic Urethane) AKA Nijaflex, Filaflex
Extruder temperature: 210-240°C (230-240 recommended)	Extruder temperature: 170-230°C ( <b>Custom extruder required.</b> )
Heated Bed: not required, 50-60° optional	Heated Bed: not required (30-40°C)
<b>Polycarbonate</b>	
Extruder temperature: 300°C	

### Filament considerations:

- Plastic filament can, and will, absorb moisture from the air. Moisture in the filament will turn to steam as it enters the nozzle of the printer, and can cause gaps in the flow or create back-pressure or swelling of the filament, increasing the risk of jamming.
- Filament is normally shipped in sealed bags with a small pack of desiccant. When not using the filament, return it to a sealed bag with the desiccant (i.e. large heavy-duty/freezer-storage zip lock bag). Commercial filament storage containers are available specifically for this purpose = "**Filament Safe**" by Perception Engineering. (SKU 153502, 104513, 073874.) You can also use air-tight food storage containers and use off-the-shelf desiccant such as **DampRid** or "renewable" packs like **Eva-Dry** Wireless Mini Dehumidifiers. (search for desiccant on Amazon)

### Cooling and heating filament:

- ABS and some other filaments shrink as they cool. This can cause the print to warp as layers are laid down, or for the print to pull loose from the build plate. Heating the build plate reduces this problem, but drafts and working in a cool/cold environment can aggravate this issue. More printers are moving to an enclosed workspace to help stabilize the temperature of the work chamber.
- Active cooling: Printing very small areas or cross-sections or objects may not allow enough time for one layer of plastic to cool off before the next layer is laid down on top. This can cause sagging, loss of detail, and other issues. Some printers have a fan on the print head assembly to blow air on the object to cool it down faster. Using a small fan to blow across the object can also help, but may cause warping or layer separation.

### Getting the object to stick to the build plate:

- Level the print bed! Use the menu option to level the bed, and/or manually level it. Adjust the bed so that the distance from the nozzle and the build surface is just enough to be able to slide a piece of paper under the nozzle. Move the nozzle to the center and all sides of the bed and repeat this adjustment until it is at a uniform height. The paper should be able to move under the nozzle with just a slight amount of resistance - compare this to moving a pen or pencil across the paper. If you have two or more nozzles on your printer, check each nozzle separately. (If the nozzles are not all at the same height, you may need to refer to the manufacturer's instructions for adjusting the nozzle height.)
- Clean the print surface. Use acetone or isopropyl alcohol to remove all grease and residue from the bed.
- Use tape on the bed: If you are using Kapton tape or blue painters tape, clean the surface of the tape before printing. (This can make the surface of blue painters tape slightly fuzzy, which can improve the adhesion of the plastic to the tape - sometimes too well!) Some users lightly sand the surface of the Kapton tape to improve adhesion (a 3M abrasive pad may work better than sand paper. Clean the surface with alcohol after sanding.)
- If you have a heated bed, turn it on. Use the recommended temperature for that type of filament, but any heating will generally reduce shrinking and separation of the model from the bed.

- Enable printing of a "raft" or "brim" during the slicing step (may not be available in all 3D printer software). A raft extends under the build area and can help with adhesion by creating a level platform under the object. A brim is a thin boarder added to the outside edge of your printed object to increase the surface area in contact with the bed. Both a raft and brim may require additional filing or sanding of your finished object.
- Use "glue" on the bed. Several methods have been suggested both by users and printer manufacturers:
  - **Glue Stick:** One manufacturer includes a water-soluble glue stick to help ABS stick to the glass plate of the print bed. Use a slightly moist paper towel to form a very thin layer on the surface where the print will be laid down. Use too much glue, and you may have problems removing the object from the surface when done.
  - **ABS Juice:** Make your own solution by adding some scrap ABS filament to a small container of acetone (make sure to pick a container that acetone doesn't dissolve!) Use a lint-free cloth or paper towel to apply a thin solution to the surface of the build plate. The acetone will quickly evaporate leaving a thin film of ABS plastic on the surface. If you get too much plastic in the solution, it will form a frosty surface and your part will be very difficult to remove. Thin the solution with more acetone so that it leaves only a slightly streaky film on the surface.
  - **"Aqua-Net Super Hold":** This apparently comes in a purple can. Clean the surface of your build plate and then lightly spray the surface of the build plate with the hair spray. This will form a surface layer that should make your print adhere to the print bed better.
- **Filament temperature:** Increasing the print temperature can improve adhesion of the first layer (some slicing programs allow you to set one temperature for the first layer and another for the rest of the job.)
- **Specialty coatings:** Several new products are just starting to show up on the market now or are at a pre-order stage.
  - **BuildTak** (Ideal Jacobs Corp. [www.buildtak.com](http://www.buildtak.com)) is a stick-on plastic film, much thicker than Kapton or blue painters tape. The coating is micro-textured to improve the adhesion of the filament to the layer, but will still release when cool. Make sure to level the bed before using this to avoid damage from the nozzles as you start your first print. Clean the surface to remove any oils from handling and installing the covering. High-temperature filaments such as nylon or polycarbonate may still need a thin coating of water-base glue stick to maintain adhesion. Estimated use can be up to 60 prints from one application.
  - **GeckoTek** ([www.geckotek3d.com](http://www.geckotek3d.com)) Like BuildTak, this is a special coating for the build plate that improves adhesion of the filament to the surface, but still allows easy release of the print when done. GeckoTek adds a twist by applying their material to a thin sheet of spring steel. This allows you to gently flex the plate to pop your 3D print off once the surface has cooled. They also are producing a selection of magnetic bed replacements made from aluminum with embedded magnets that holds the steel back to the plate, but lifts off when the job is complete. Magnetic beds may be heated or not, depending on the printer it is designed for. (FYI: the PowerSpec 3D-X uses the same bed as the Flashforge Creator or Creator X.)

## Getting the object to un-stick from the build plate:

**Removing the parts from the print bed:** Techniques will vary depending on the type of print bed you have (glass, aluminum, ceramic, granite, etc.) and if you have used tape or special coated-beds.

- Heated bed - Once the build is complete, software usually turns off the heated bed and nozzle. Allow the bed to cool and try removing the part once it has reached room temperature. If it still is stuck to the bed, use the pre-heat option to warm the bed; try removing the part when the bed is hot. Repeat the cooling-heating steps to break the seal between the bed and the plastic part.
- With glass, ceramic, granite and other hard, rigid materials, you can almost always use metal tools like a razor blade, putty knife, or paint scraper directly on the surface without too much fear of scratching it. Plastic scrapers are an option that can be used on most any type of bed.
- Aluminum, Lucite, or other soft beds can easily be scratched by steel. Using a protective layer such as Kapton or painters tape can reduce the risk if you need to pry or wedge a spatula tool under the part. Thin spring-steel spatulas such as the *Cricut Tools Craft Spatula* work well on most surfaces as long as you are careful and have a layer of tape to protect the bed.
- Pliers may work on small parts where you are not too concerned about abrading the surface finish (pegs and connecting tabs for example.) Try twisting the part or rocking it back and forth to release the part from one of the edges.

## Other common print issues:

- **Layer adhesion** - splitting between layers. Gaps or separation between the layers can be caused by several things:

- Filament temperature is set too low - try increasing the temperature or make several calibration prints at different temperatures to identify the best temperature setting for that specific filament. Remember, different types of filament will have different melting points, and even the same model of printer may have variations in temperature accuracy. Use the recommended temperature range as a starting point, and determine the temperature to use based on your own results.
- Layer thickness is set too high - try reducing the layer thickness if you are close to the size of your print nozzle.
- Work environment is cool or drafty - try covering the open sides of your printer or otherwise enclosing the print space. Use the heated bed to keep the build area warm.
- Print speed is too fast. Change the speed while extruding (usually in millimeters/minute) or increase the nozzle temperature to support accelerated travel.
- **Threads and surface burs** - both can be caused by "oozing" from the nozzle. This problem is common with multi-head printers as the nozzle that is not printing can slowly ooze and leave drops of plastic on the layer as it moves across the print. There are several solutions to reduce or minimize threads and burrs, including:
  - Active cooling - add a small fan to blow across the print or towards the filament nozzle. This cools the filament quicker as it leaves the nozzle, reducing or eliminating sagging and threads.
  - Slicing and printer software settings
    - Retraction settings in the slicing software. This pulls the filament back from the nozzle when not actively using that head, or when changing position to start a new thread.
    - Slicer has options under the Printer Settings to "Wipe while retracting" and to "Lift Z" (raise the nozzle from the current layer before moving.) Under the Print settings is an option to enable "Ooze prevention"
  - Filament temperature - if set too high, the filament will continue to melt and leak from the nozzle. Try reducing the temperature and/or do some temperature calibration tests with that spool of filament to identify the best print temperature to use.
  - Add a "skirt" or try some different printing software. MakerWare automatically adds thin "fences" one either side of the object when using dual extrusion. This helps eliminate burs forming by "wiping" the inactive nozzle on the fence during the print process. Adding a skirt around your object in the slicing software can do the same thing; set the skirt height to the same height of your object. Make sure that you keep the distance from the base far enough away to account for overhangs as the object prints.
  - Design a skirt - If you are developing your own models or modifying them, add a thin-wall object around the main part. Like a skirt or the MakerWare fence, ooze from the nozzle will tend to get wiped off on the outer shell rather than your model.

## Slice and print settings:

- **Layer height:** This is equivalent to the resolution of your print. Choosing a smaller layer height increases the level of detail of your finished print and increases the print time (more thin layers must be printed to equal thicker ones). Layer height is usually specified in 10ths of a millimeter;  $0.100 = 100$  microns. When choosing a layer height, you are limited to the size of the hole in your printer's nozzle and the precision of the z-axis movement. If you attempt to print a layer thickness greater than the nozzle size, the print will fail (you cannot make a layer 0.5mm thick if the hole is only 0.4mm - there will be a gap between the layers.) Layer height may also impact jamming if the layers are too thin or the filament temperature is set too cool. If you encounter frequent jamming or slipping in the extruder, try increasing layer height (i.e. use layer height or 0.15 or more if jamming occurs at 0.10mm.) The first layer is usually set thicker than the rest to improve the bond to the build plate surface.
- **Infill:** You have the option of making your objects hollow, solid, or anywhere in between. The slicing program can generate a "fill" pattern on the inside based on the percentage of fill you specify. The infill pattern can be rectangular, hexagonal, or some other style, limited by the software options. Infill adds support to make your object more rigid and to provide a solid structure to support any overhangs of the outer shell. Increasing the infill percentage will increase the print time and use more filament to print your object.
- **Support:** 3D printers cannot print in open air; there must be some sort of support structure to build the layer on. Layers can overlap the layer below, usually limited to no more than a 45 degree overhang. If your object has an area that "hangs out" more than this amount, you need some sort of external scaffolding to hold the layer up. Turning support on in the software identifies areas that extend out too far, and adds some sort of support, starting at the build plate and extending up to the layer that needs support.

- **Orientation:** Most software programs for 3D printing allow you to rotate your object within the build area. Reasons why you may want or need to adjust the orientation of the object include:
  - To place the largest or flattest portion level on the build plate
  - To minimize or eliminate overhangs
  - To orient holes or openings perpendicular to the bed

## Finishing and smoothing:

- Layer height - using a smaller layer height increases the detail level of your finished part and reduces the size of the stepping at the edge of the layers.
- Knives and scrapers - can remove threads and burs from the surface
- Sanding & buffing - This will take some experimenting, both with different types of abrasive paper and your technique. Plan on hand sanding, as machine sanding, flexible shafts, etc. are likely to generate too much frictional heat that will melt the surface, and embed grit and dirt as a result. Wet sanding will work better, but porosity in the surface can trap the grit and dirt from the process. You can use sanding blocks, popsicle sticks, pencils, etc. as support for the abrasive paper and to conform to the area being sanded. Plan on using a coarse, medium, and fine grit paper to achieve the best results (coarse = 100-220; medium = 320-400; fine: 400-600. To obtain a glossy finish you can use a felt polish pad or muslin polish wheel lightly charged with plastic polish or an extra-fine polish stick (available from hobby/jewelry supply). If using motorized polish wheels, keep the edge of the wheel with the layers separated and loose. Avoid build up of excessive polish compound -- you need very little to polish plastics. Keep the object moving (don't hold it in one place), and very lightly feather the surface with the wheel to avoid generating heat.
- Files - Medium to coarse wood files work fine on plastic parts (Fine may work, but will tend to clog faster.) Invest in a "file card" wire brush for cleaning out the teeth of your files. Files are available in all sorts of shapes and sizes. Flat files work well for squaring up edges and corners; use a flat file to smooth both flat and convex (curved) surfaces. Round and half-round files are easier to use on inside curves to smooth or enlarge an opening. Needle files are small versions of the files and available in all shapes sizes and coarseness.
- Heat polishing - a small-tip heat gun can be used to carefully melt the surface of your finished print. This works best on larger flat or rounded areas, as edges or small, thin areas will heat faster and deform during the process.
- Chemical finishing - If the plastic material is soluble in a solvent, objects may be "polished" by limited exposure to that chemical. Note that many of the solvents for the various plastics have known health hazards and in some cases are extremely flammable! Small details may be lost, as you are dissolving the exposed surface.
  - ABS is soluble in acetone. Objects can be dipped, sprayed, wiped or exposed to acetone fumes to partially dissolve the surface. As the acetone evaporates, the plastic will harden to a smooth shiny finish. Acetone is very flammable; use in well ventilated areas and avoid breathing fumes or prolonged exposure as it can be absorbed through the skin or lungs. See [Wikipedia](#) for more complete health and exposure references.
  - HIPS is soluble in limonene (citrus based solvent AKA GooGone) Spray, wipe or dip your object. Limonene is slow to evaporate and can leave the surface sticky until it completely reacts with the plastic or dries. Rinsing or washing the part to remove excess liquid may result in a cloudy, uneven or frosted finish. HIPS can be used similar to PVA (water-soluble) filament as support structure, and removed without harming most other plastics by submerging the object in limonene until all of the exposed HIPS has been dissolved. See [Wikipedia](#) for more detail on possible health and safety concerns.
  - PLA is soluble in tetrahydrofuran (THF), an industrial solvent. Polishing can be accomplished in much the same manner as acetone and ABS, by dipping or vapor smoothing. From the [Wikipedia article](#): *"THF is considered a relatively nontoxic solvent, with the median lethal dose (LD50) comparable to that for acetone. Reflecting its remarkable solvent properties, it penetrates the skin causing rapid dehydration. THF readily dissolves latex and is typically handled with nitrile or neoprene rubber gloves. It is highly flammable. One danger posed by THF follows from its tendency to form highly-explosive peroxides on storage in air."*

## 3D printer control & slicing software:

- **Repetier Software (RepetierHost)** *Windows, Linux, Macintosh*
  - Main site: [www.repetier.com](http://www.repetier.com)
  - Download: [www.repetier.com/download/](http://www.repetier.com/download/)
  - Documentation: [www.repetier.com/documentation/repetier-host/](http://www.repetier.com/documentation/repetier-host/)
- **Slic3r** gcode generator *Windows, Linux, Macintosh*
  - Main site: [slic3r.org/](http://slic3r.org/)
  - Download: [slic3r.org/download](http://slic3r.org/download)
  - Documentation: [manual.slic3r.org/](http://manual.slic3r.org/)
- **PronterFace AKA Prinrun** *Windows, Linux, Macintosh*
  - Main site: <https://github.com/kliment/Prinrun>
  - Download: follow links listed under "GETTING PRINTRUN" section of the site
  - Documentation: limited, some available on Wiki and other sites (i.e. [reprap.org/wiki/Prinrun](http://reprap.org/wiki/Prinrun))
- **ReplicatorG** *Windows, Linux, Macintosh*
  - Main site: [replicat.org/](http://replicat.org/)
  - Download: [replicat.org/download](http://replicat.org/download)
  - Documentation: [replicat.org/usage](http://replicat.org/usage) -- See main site for installation and other reference.
- **MakerWare** *Windows, Macintosh*
  - Main site: [www.makerbot.com/makerware](http://www.makerbot.com/makerware)
  - Download: [www.makerbot.com/makerware](http://www.makerbot.com/makerware) Scroll down and click on the "DOWNLOAD" box.
  - Support: [www.makerbot.com/support/makerware/troubleshooting/](http://www.makerbot.com/support/makerware/troubleshooting/)
- **Netfabb** 3D model repair *Windows, Linux, Macintosh*
  - Main site: [www.netfabb.com/](http://www.netfabb.com/)
  - Download: [www.netfabb.com/downloadcenter.php?basic=1](http://www.netfabb.com/downloadcenter.php?basic=1)
  - Documentation: [wiki.netfabb.com/Main\\_Page](http://wiki.netfabb.com/Main_Page)
- **MatterControl** *Windows, Macintosh*
  - Main site: [www.mattercontrol.com/](http://www.mattercontrol.com/)
  - Download: [www.mattercontrol.com/#jumpMatterControlDownloads](http://www.mattercontrol.com/#jumpMatterControlDownloads)
  - Documentation: [wiki.mattercontrol.com/](http://wiki.mattercontrol.com/)
- **Cura** *Windows, Linux, Macintosh*
  - Main site: [software.ultimaker.com/](http://software.ultimaker.com/)
  - Download: [software.ultimaker.com/](http://software.ultimaker.com/)
  - Documentation: [www.ultimaker.com/pages/support/manuals](http://www.ultimaker.com/pages/support/manuals)

## 3D design and/or modification software (free versions available):

- **AutoDesk 123D Design** - <http://www.123dapp.com/design>
- **Blender** - <http://www.blender.org/>
- **MeshLab** - <http://meshlab.sourceforge.net/>
- **SketchUp** - <http://www.sketchup.com/download>
- **DesignSpark** - <http://www.rs-online.com/designspark/electronics/eng/page/mechanical>
- **OpenSCAD** - <http://www.openscad.org/>
- **FreeCAD** - <http://sourceforge.net/projects/free-cad/>