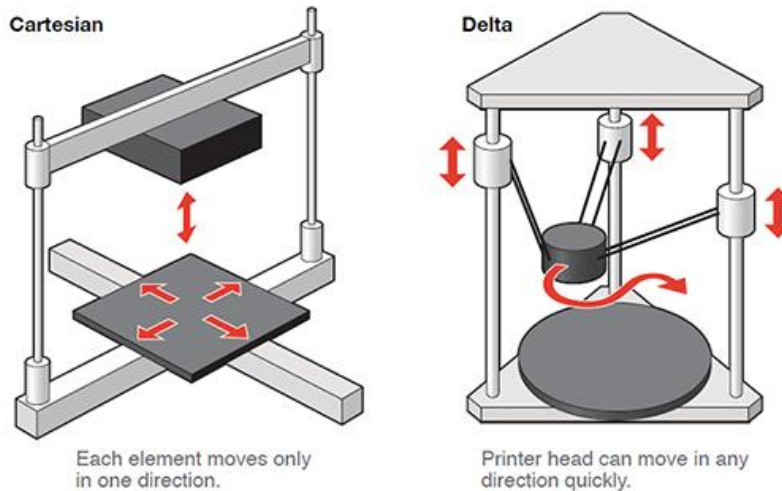


3D Printing Basics

Fused Filament 3D printer: machine that melts and deposits a filament on top or next to the same material, creating a joint by heat and/or adhesion

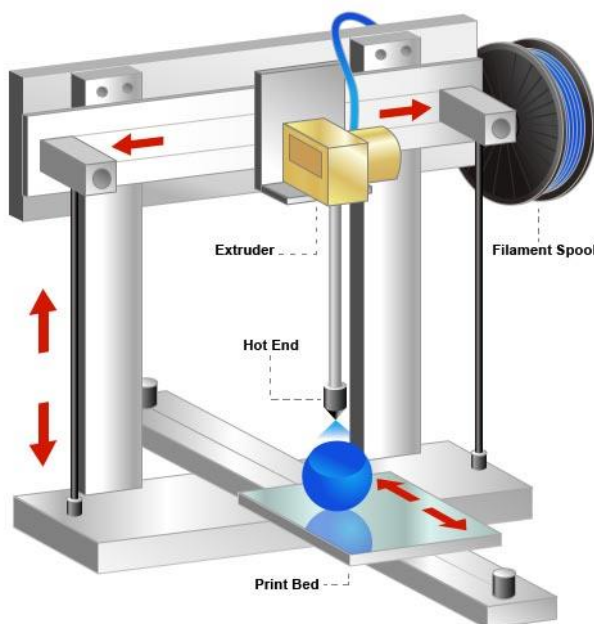


Basic Printer Parts:

EXTRUDER: part of the 3D printer that ejects material in liquid or semi-liquid form in order to deposit it in successive layers within the 3D printing volume.

HOT END: nozzle or print head that heats and melts the filament to the correct temperature.

PRINT BED: platform onto which the filament is laid. Some printers have a heated print bed that can be set to specific temperatures to improve print adhesion and reduce warping from the filament cooling too quickly.



FILAMENT - the base material that's used to 3D print objects via fused filament fabrication. Filament is usually a thermoplastic - such as ABS or PLA - that's fed to a print head as a solid, then heated to melting point for extrusion through a small nozzle. Filament is commonly available in spools of either 1.75 mm and 3 mm diameter widths.

PLA (Polylactic Acid) - **Extruding: 180-220 °C Heated Bed: 20-55 °C**
a bio-degradable plastic material that has gained traction with 3D printing for this very reason. It is offered in a variety of colors, including transparent, which has proven to be a useful option for some applications of 3D printing. However it is not as durable or as flexible as ABS.

ABS (Acrylonitrile Butadiene Styrene) - **Extruding: 220-235 °C Heated Bed: 80-110 °C**
a common plastic widely used on the entry-level FDM 3D printers. It is a particularly strong plastic and comes in a wide range of colors. ABS can be bought in filament form from a number of non-proprietary sources.

HIPS (High Impact Polystyrene) – **Extruding: 220-230 °C Heated Bed: 50-60 °C**
a dissolve-able filament very similar to ABS in durability. HIPS is frequently used as support material because it is easily removed with Limonene solution, leaving the clean high-quality print that you want behind.

Printer Software:

CAD (Computer-aided design) - software that is used by architects, engineers, drafters, artists, and others to create precision drawings or technical illustrations. CAD software can be used to create two-dimensional (2-D) drawings or three-dimensional (3-D) models.

3-D CAD Beginner Software Examples - [TinkerCAD](#), [SketchUp](#)

STL / .stl (STereoLithography) - file format created by the CAD software that is the standard file type used by most 3D Printers. STL is a triangulated representation of a 3D CAD model. STL has several after-the-fact backronyms such as "Standard Triangle Language" and "Standard Tessellation Language."

SLICER - the software package used to divide a 3D model into flat layers, which are then printed one at a time. The output of a slicer is G-code that controls the path, speed, and temperature of the printer. Slicer software packages are available in both open-source and proprietary programs, and are an essential tool for successful 3D printing.

Slicer Software Examples – [CURA](#), [Slic3r](#), [Matter Control](#)

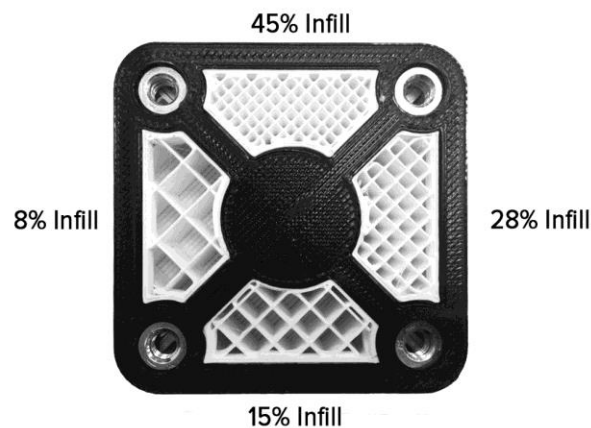
G-CODE (RS-274) - the common name for the most widely used numerical control (NC) programming language. It is used mainly in computer-aided manufacturing to control automated machine tools.

Printing Parameters:

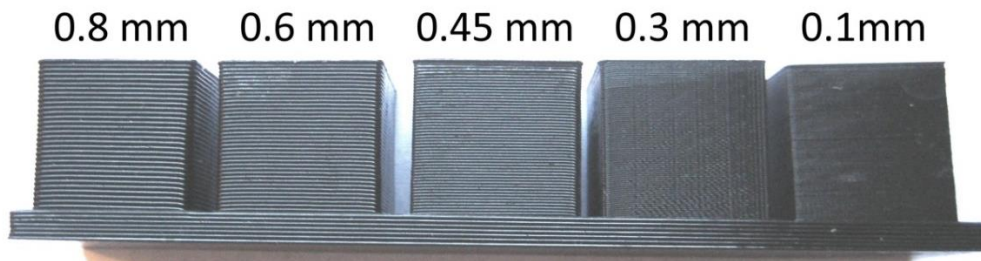
TEMPERATURE - optimal temperatures chosen for the extruder and print bed based on the type of filament used. Temperatures can be adjusted to improve print quality, but should always be set within the range suggested by the filament manufacturer (often printed on filament spool).

SHELLS / PERIMETERS - the number of times the outer most layer of the object is traced by the printer. The more shells on an object, the thicker and stronger the perimeter will be. Adding shells will also increase the print time significantly. Shells are also referred to as perimeters in some software and documentation.

INFILL - the structure that is printed inside an object. It is extruded in a designated percentage and pattern, which is set in the slicing software. Infill percentage and pattern influence print weight, material usage, strength, and print time.



LAYER HEIGHT - the thickness of each layer of filament placed during the print. Smaller layer height produces higher resolution, smoother prints, but takes significantly more time to print. Larger layer height will print somewhat stronger models more quickly, but will produce a lower quality finish. Your printer's layer height capability range will depend on the size of your hot end extruder.



SUPPORTS - the structures added by the slicer software that enable printing models with steep overhangs and cantilevered sections. Recommended when material will be deposited on a layer where a previous layer was not deposited or when there are angles ($>45^\circ$) that might cause undesired drooping or sag during a print. Support parameters can be adjusted within the printing software.

Optimal Printer Parameters based on Output Requirements:

| Strength | | Infill % | | | | | | |
|----------|------|----------|----|----|----|----|----|-----|
| [MPa] | | 10 | 30 | 50 | 70 | 80 | 90 | 100 |
| | 0.1 | 4 | 9 | 15 | 24 | 29 | 34 | 40 |
| Layer | 0.15 | 8 | 13 | 19 | 28 | 33 | 38 | 44 |
| Height | 0.2 | 10 | 15 | 21 | 30 | 35 | 40 | 46 |
| [mm] | 0.25 | 11 | 16 | 22 | 31 | 37 | 42 | 48 |
| | 0.3 | 11 | 16 | 22 | 31 | 36 | 41 | 48 |

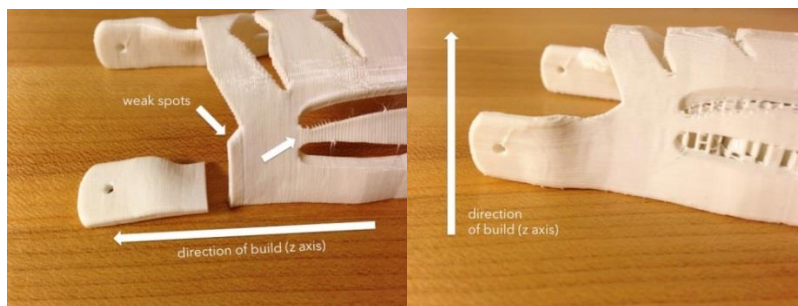
| Speed | | Infill % | | | | | | |
|--------|------|----------|----|----|----|----|----|-----|
| [mins] | | 10 | 30 | 50 | 70 | 80 | 90 | 100 |
| | 0.1 | 21 | 35 | 47 | 61 | 68 | 74 | 80 |
| Layer | 0.15 | 14 | 23 | 32 | 41 | 46 | 50 | 54 |
| Height | 0.2 | 10 | 18 | 24 | 31 | 35 | 38 | 41 |
| [mm] | 0.25 | 8 | 14 | 20 | 25 | 28 | 31 | 33 |
| | 0.3 | 7 | 12 | 16 | 21 | 24 | 26 | 28 |

| Cost | | Infill % | | | | | | |
|---------|------|----------|----|----|----|----|----|-----|
| [cents] | | 10 | 30 | 50 | 70 | 80 | 90 | 100 |
| | 0.1 | 5 | 9 | 13 | 16 | 18 | 20 | 22 |
| Layer | 0.15 | 5 | 9 | 13 | 17 | 19 | 21 | 23 |
| Height | 0.2 | 6 | 10 | 14 | 18 | 20 | 22 | 23 |
| [mm] | 0.25 | 6 | 10 | 14 | 18 | 20 | 22 | 24 |
| | 0.3 | 6 | 10 | 14 | 18 | 20 | 22 | 24 |

| Quality | | Infill % | | | | | | |
|---------|------|-----------|----|----|----|----|----|-----|
| | | 10 | 30 | 50 | 70 | 80 | 90 | 100 |
| | 0.1 | very high | | | | | | |
| Layer | 0.15 | high | | | | | | low |
| Height | 0.2 | medium | | | | | | |
| [mm] | 0.25 | low | | | | | | |
| | 0.3 | very low | | | | | | |

BEST \longleftrightarrow WORST

Print Orientation: Part orientation during printing plays a large role in the part's strength and utility. Like wood, printed objects have grain due to the layered fabrication process. Printed objects are strongest in the plane of the layer (the XY plane). Orientation also determines the amount of support material necessary. Objects can be designed to print in separate pieces or rotated before slicing for optimal strength.



Other 3D Printing Terms:

3DP 3D Printing

AM Additive Manufacturing

CAM Computer-aided manufacturing

CAE Computer-aided engineering

FDM Fused Deposition Modeling (Trademark of Stratasys)

FFF Freeform Fabrication or Fused Filament Fabrication (generic terms for FDM)