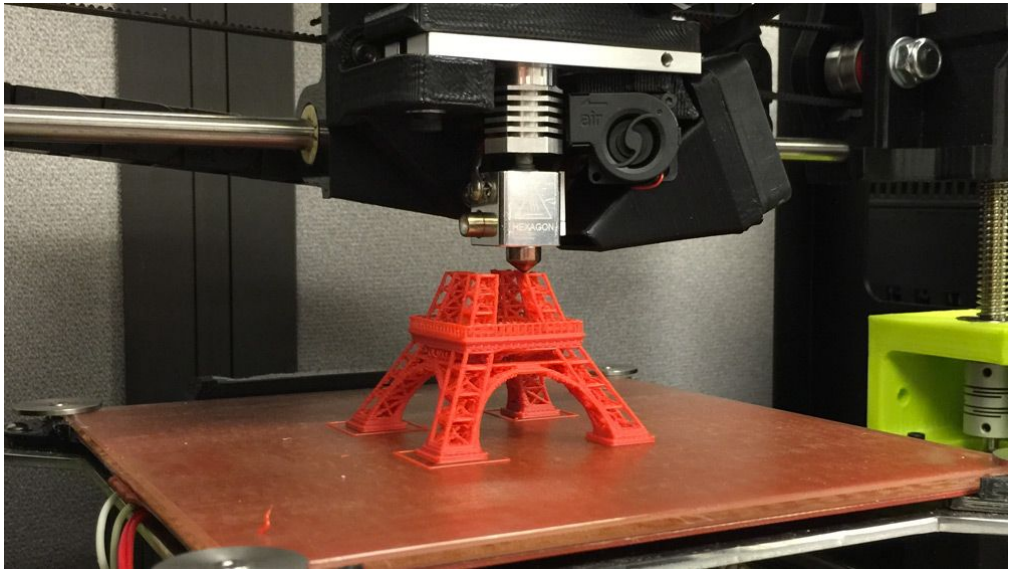
3D Printing

Kenny George

What is 3D printing

3D printing is form or rapid prototyping that allows for one off manufacturing of physical objects.



There are many types of 3D printing applications:

SLS - Selective Laser Sintering

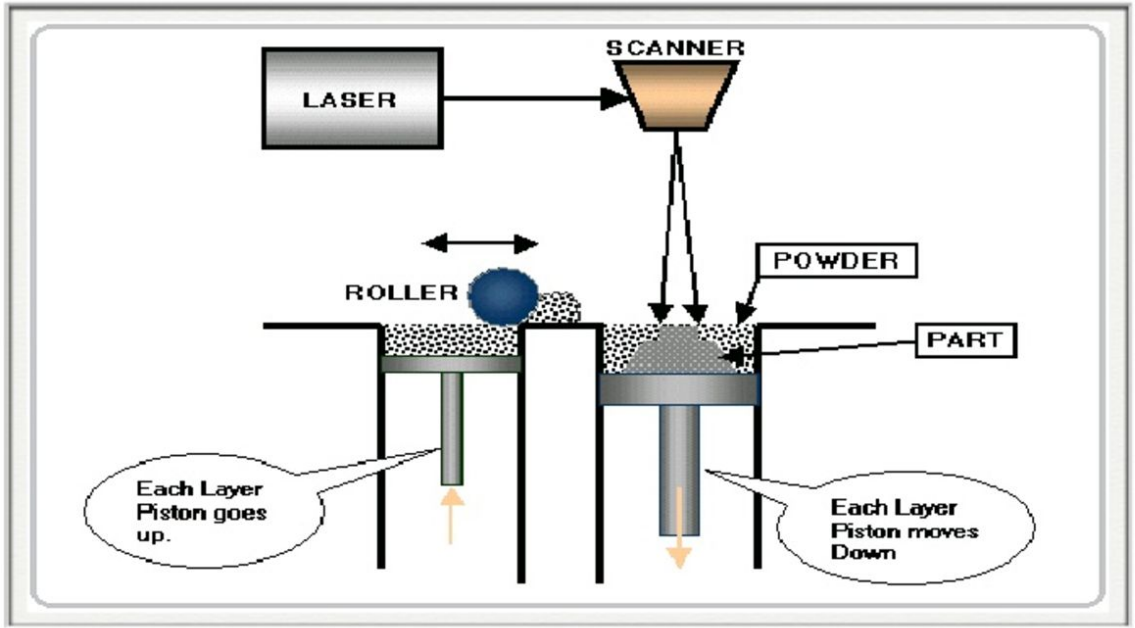
STL - Stereolithography

FDM - Fused Deposition Modeling

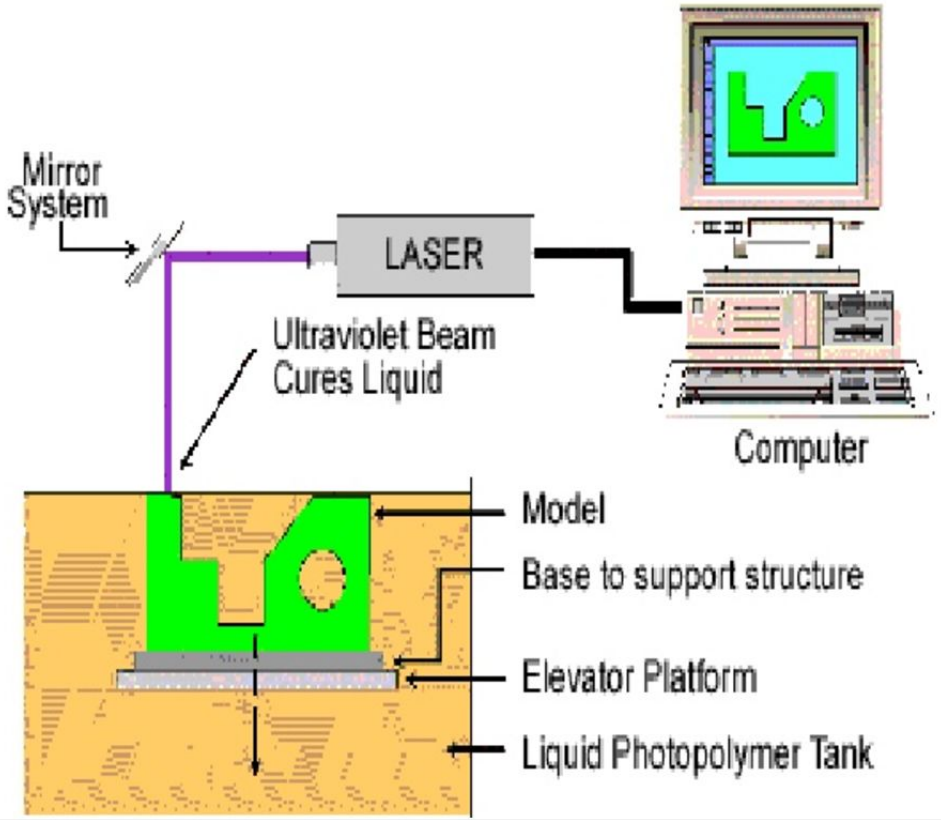
***THESE ARE ALL **ADDITIVE PROCESSES**

1 .SLS method

- Selective laser sintering (SLS) is an additive manufacturing technique that uses a high power laser (for example, a carbon dioxide laser) to fuse small particles of plastic, metal (direct metal laser sintering),ceramic or glass powders into a mass that has a desired 3-dimensional shape).

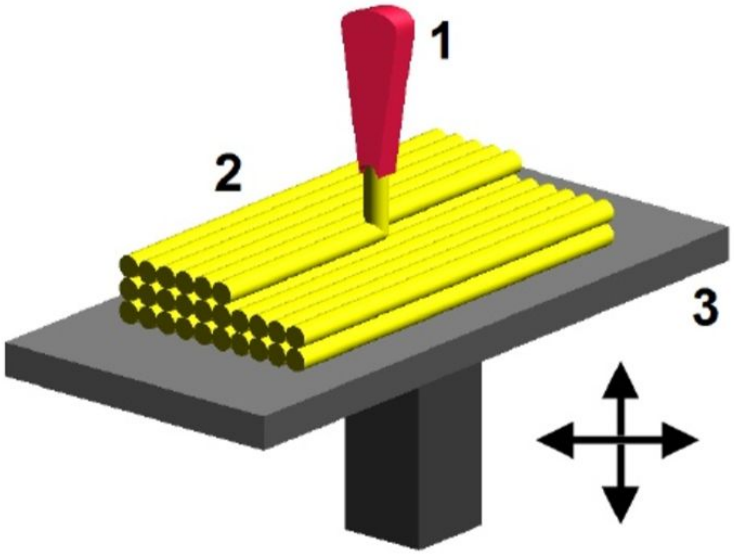


2 .STEREOLITHOGRAPHY



- Stereo lithography is an additive manufacturing process using a vat of liquid UV-curable photopolymer ”resin” and a UV laser to build parts a layer at a time.

3.Fused deposition modeling (FDM)



- **Fused deposition modeling** (**FDM**) is an additive manufacturing technology commonly used for modeling, prototyping, and production applications.
- The model or part is produced by extruding small beads of material which harden immediately to form layers.
  A thermoplastic filament or metal wire that is wound on a coil is unreeled to supply material to
  an extrusion nozzle head. The nozzle head heats the material and turns

FDM is the most common application used

PROS:

Cost effective

Quick

PLA is ecologically responsible

CONS:

Inconsistent

Not as accurate

Strategies for 3D printing

Necessary steps to consider

1:CAD to CAM

2. Printer constraints and material properties

3. Slicing and exporting Gcode

CAD to CAM

CAD stands for COMPUTER AIDED DESIGN

-This is the process of modeling your file on the computer.

-There are many applications free to students for CAD

1. Tinkercad - Part of Autodesk, web based, free, very simple for quick production
2. Fusion 360 - Part of Autodesk, more complex, but more accurate with the ability to size parts to function (great for virtual machining)
3. Maya and 3DS Max - Also Autodesk, Robust 3D modeling engines
4. Blender - Freeware, very comprehensive, albeit confusing interface

CAD to CAM

CAM stands for Computer Aided Machining

-This is simply the process of converting the 3D model from the application to a file format that can be read by the 3D printer application. This process is going to be specific to each 3D modeling software.

-In most instances you are looking to Export your files as a **.STL** File

Printer Constraints and Material Properties

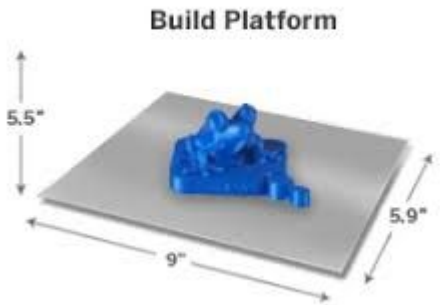
Before exporting your .STL file it is important to consider:

-Scale of print (it is best to do this in modeling software)

-capacity of printer (what are the XXY limitations of your printer)

-Material printer material (ABS, PLA, PET, HIPS)

-Extrusions, holes, curves, and other irregularities



SLICING AND GCODE

-Once you have an .STL (or multiple) .STL's, you will need to use a printer utility, called a **SLICER** to convert the print to a language that the printer can understand.

-The majority of 3D printers are **OPEN SOURCE** and will read gcode, a universal numerical controlling language spoken by 3D printers, CNC mills and other desktop fabrication devices.

-Some printers, like **MAKERBOT** use closed source applications, where they convert the file to their own proprietary language. For those applications you will use their recommended slicing software.

SLICING and GCODE

Free Slicing softwares

-Slic3R

-Cura

SLICING and GCODE

When slicing your STL’s for printing: here are some considerations to keep in mind:

-Can you fit multiple prints on the build plate?

-Can you arrange prints to minimize travel? This will cut down on print time and prevent temperature shifts between layers.

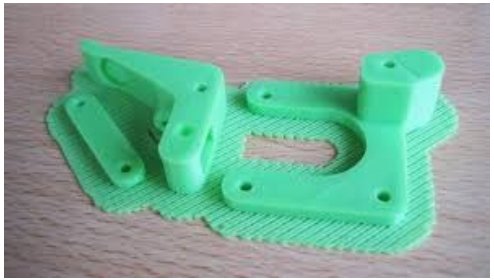
-What side will print best as bottom (best first layer adhesion)

-Can extrusion be pointed upwards and holes be vertical to reduce the need for supporting materials?

SLICING AND GCODE

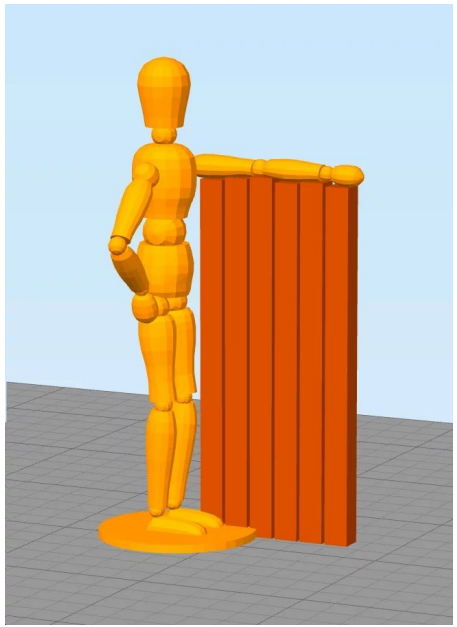
Supporting materials:

Raft: Raft actually is the horizontal grid of filament that is located under the bottom of the 3D models as shown in the above pictures. The purpose of adding a raft is to ensure that the printed model can attach to the print bed more firmly to prevent the warping issues. Also, raft is useful if the bottom part of the 3D model is small because raft can increase the contact surface between the model and the print bed, which means can let the model stick on the print bed better.



SLICING AND GCODE

Supports: Supports are vertical scaffolds intended to be removed and discarded: They are added when there is a hole or extrusion that will collapse under gravity.



SLICING AND GCODE

Best practices are to always print with a raft unless you have a large flat surface area as your first layer. Use your best judgement on supports though slicing softwares will generally calculate where they are needed an add for your.

**Rafts and supports are not a native function of slicing so make sure that you select that you want to add them in your slicing software.

SLICING AND GCODE

Factory recommendation for printer will default your extruder temperature and bed temperature (if applicable) based on printing material. Once you become more familiar with printing, you may wish to adjust temperature to promote less warping, better adhesion, or adjust for seasonal temperature shifts.

-Extrusion speed, retraction speed, and gantry speed are also adjusted to refine resolution and improve print quality.

SLICING AND GCODE

Once settings are all entered correctly, **ALWAYS DOUBLE CHECK!!**

-Then you are ready to slice and export GCODE as recommended by your slicing software and add to media device to load in printer.