# Design for Additive Manufacturing:

Slic3R Wiki (Volume Conservation FDM)

## Important factors to consider while 3D printing

* **Infill** – This is the amount of material present inside the volume. It is highly important for providing strength to the printed part. The infill structure can also be defined to provide desired qualities to the part
* **Layer thickness** – The thickness of each layer is highly detrimental in the quality and strength of part. While low layer thickness is good for accuracy, it also increases the printing time. With respect to manufacturing, the time is a crucial aspect.
* **Nozzle diameter** – The diameter of the nozzle can change a large number of factors. Smaller diameters may result in expansion of material after being released from the nozzle. This can produce significant deformation in many cases. Diameters of 0.3 and 0.35 were used by Tushar during experimentation at the MARS Lab, UTA. Choosing the nozzle diameter is a balance between and part quality.
* **Feed rate** – This is directly related to the manufacturing speed and material dispensing speed. A greater feed rate requires the machine to move quickly which can result in limited in-process optimizations. A fast feed rate may lead to discontinuous extrusion of strand. For the same input filament, increasing the feed rate can lead to thin strand. In the experiment conducted by Tushar, F100 led to a strand of 6 mm thickness and F500 led to a strand of 1 mm thickness.
* **Material properties** – This is the most important aspect of 3D printing. The material should be supported by the printer. In case of composites, the materials should be compatible with each other and have acceptable melting and softening temperature ranges. Highly viscous materials may not be able to flow through small nozzles leading to clogging. Non-Newtonian fluids can also pose a problem for injection molding as the viscosity changes with change in pressure which is inevitable in this process.
* **Nozzle to bed distance and layer height** – This refers to the distance from nozzle to bed and nozzle to topmost layer respectively. The initial nozzle to bed distance can be broken down into two cases:

NOZZLE

d

NBD

BED SURFACE

d = Nozzle Diameter; NBD = Nozzle to bed distance

* + NBD > d – In this case, the extruded strand is discontinuous due to less cohesive force
  + NBD ≈ d – In this case the cohesive forces are good which is likely to lead to a continuous strand
  + NBD < d – Here, the layer and strand get distorted which leads to unpredictable changes in the printed part
* **Calibration for Z** – Work area is defined by the calibration of Z-axis and the initial offsets. There are two ways for defining offsets:
  + Through the slicer application and the printer. This option is relatively easy as it requires minimal knowledge and can be done quite easily. However for small changes, to the machine everything needs to be reconfigured.
  + Through G-codes. This method is slightly hectic as it consists of going to specific files and changing the G-codes. However, it is still better as it provides high level of customization. Furthermore, the changes are stored making it easy to edit later.
* **Path planning** – This can be done through specifying via points or points needed to be printed. It is important to provide strand direction which can alter different mechanical properties of the printed part. The output of Prusa Slic3R consists of a file named “planning.cpp”. This file can be altered based on the Marlin Firmware for G-codes.

## Important parameters to account for while 3D printing

* **Strand distortion** – The cross sectional area of the strand is not a perfect circle. It is flattened at the top and bottom due to gravity and weight of layers above. However, this helps in increasing the area for bonding to other layers
* **Sagging** – There could be sagging in the material if the nozzle to bed distance is not calibrated properly
* **Pressure build-up** – In many cases, the nozzle does not dispense the material for a while and then dispenses it all at once. This pressure build-up can be avoided sometimes by squirting a little material before starting to print the part.
* **Nozzle material** – The nozzle material should not attract the material being extruded. If this happens then the material can form a droplet and stick to the nozzle leading to discontinuous strand.
* **Meniscus** – Viscosity of fluid may lead to the formation of a meniscus. While this is acceptable in larger parts, it might be an issue with small parts and parts requiring high precision.
* **Environment** – Material may require different environments for proper printing and curing. Factors such as oxidation, temperature, lighting etc. can have a significant effect on the part quality