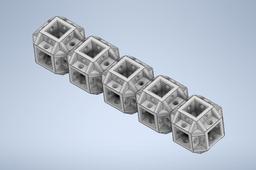
**Picking a nozzle for your 3D Printing project**

**[Martin Pirringer](https://grabcad.com/martin.pirringer-2)**

5 Nov, 2019 11:28 PM



Picking a nozzle for your 3D Printing project

1. **Step 1:**

Find out what nozzles your printer can handle most FDM printers can handle 0.25 through 0.8 sometimes with certain hotends bigger

1. **Step 2:**

Determine the needs for the print. Smaller nozzles will give a more accurate prints. Larger nozzles will create prints that are stronger and are getting printed much faster (most of the time) To determine the maximum size of the nozzle consider the following

1.) Layer height. For best results layer height should be between 25% and 50% of nozzle diameter. So if you are shooting for a 0.16 layer height the theoretical max nozzle size would be 0.64 and the minimum 0.32. Make sure that the layer height you pick is a multiple of a full step of your z-stepper(s) on most FDM printers that use a standard lead screw and stepper this is .040. So your layer height should be an integer multiple of 0.040 mm.

2.) Corners. You cannot print a "knife edge" the edge will always be a fillet with the radius of the nozzle opening plus extra filament. As the smallest sensible outside layer for consistent results is currently 110% nozzle diameter this for example will give you a radius or 0.22mm on a 0.4 nozzle (0.4 \* 1.1 / 2) This might be important when crowning gears with that in mind. So for example a gear tooth that can support a 0.4mm crown fillet will probably max out at a .7mm nozzle as a .8 will create a fillet of at least 0.44mm which might cause meshing problems.

So between those 2 you further eliminate nozzles unsuitable for that task

1. **Step 3:**

Establish max Volumetric flow for the remaining nozzle(s) to make the final decision use this https://grabcad.com/tutorials/dialing-in-a-filament-and-specifying-the-max-volumetric-e-xtrusion-value as a guide. In general the biggest nozzle that made it through the above steps will be the best to use due to causing the print head move at a slower pace which will cause stronger more accurate prints. Lets assume for example we have a filament with a max flow rate of 13 mm3/sec (typical for PLA on many standard V5 and V6 type extruders). If there are no limiting factors for accuracy like layer height or arcs then with a .4 nozzle the max layer height is .2 and an outside perimeter is .44 and an inside perimeter the max sane is about .8mm maybe 1 in some special cases. in case of the .44 1mm of extruded material with have .44\* .2 = 0.088mm3 per mm extruded so the max speed for the extruder for the outside perimeter will be 13/0.088 = about 147 mm/sec and for the .8 layer width aprox. 1/2 of that. Both numbers well above the 60mm/s recommended for most printers. So in that case the printer mechanics will determine the max speed(s) usually recommended by the printer manufacturer (you should not exceed them). Now with a .8 nozzle and a .4 layer height the max speed by extrusion for the .88mm width outside perimeter is .88 \* .4 = 0.352 now assuming flow tests also came out to 13 mm3/s in that case the max speed for an outside perimeter to produce a good quality print is 13/0.352 = about 37mm/sec and about 1/2 of that for the other perimeters. Now much slower head movements, shorter distances for acceleration and deceleration and with that better quality and layer adhesion and you might squeeze a couple of seconds out because the printer has less acceleration and deceleration distances. At this time a .7 or even .6 nozzle might make sense too as both are going to provide prints in the same time as the .8 with slighter sharper corners and lower layer height at the cost of slightly less layer adhesion and higher print head moves and acceleration. So at that point those factors should be evaluated.