**Make sure that your printer is calibrated and printing accurately.**

The 3D printed parts were designed to be well fit together because of this they have tight tolerances. Make sure that your printer can reliably create parts within 0.01” or 0.25mm accuracy.

**Filament Usage**

It will take approx 1.2kg of PLA to print all necessary parts (all parts excluding 3D printed belt tensioners and end caps) with ABS or PETG it will be around 1kg due to lower filament density.

**Material Selection**

We used 3D Fuel PLA to make our own plasma table and with all the parts we ship out. PLA was chosen because it is cost effective, prints great, and is quite rigid compared to most other filaments. Since the parts will live most of their lives under a static load, we can sacrifice the brittleness of PLA in favor of its rigidity.

ABS and PETG are also usable, cost effective filaments, all three being relatively around the same price. One might choose to use ABS instead because of its great heat tolerance, exceptional print quality, and not as brittle as PLA, although being less rigid. One might choose to use PETG because of its excellent toughness. PETG parts can take significantly more abuse than PLA or ABS parts, and will rarely break along layer lines. Because of this it is not as rigid and does not print as well as either ABS or PLA.

**Print Settings**

All parts are printed with 3D Fuel PLA 2.85mm filament, 1.75mm filament will also work.

0.8mm nozzle

0.4mm layer height

220C nozzle 60bed

40mm/s, 100% cooling fan

85% grid infill and 4 perimeter walls 6 top and bottom layers

**Key Points**

* It is not advised to use 100% infill on PLA as parts will become far more brittle with negligible strength gains. 80% seems to be the best. ABS and PETG parts work well with 100% infill.
* Parts need a minimum of 3mm thick walls. Adding material to walls makes the parts far stronger than an equivalent amount used as infill. Even a part with 100% infill and 2 perimeter walls will be weaker than a part with 80% infill and 5 perimeter walls because of the print orientation.

**Other Settings**

Many people have trouble printing PETG and ABS. These materials cannot be printed like PLA; they need to be **HOT** and use minimal cooling.

We print PETG at 260C nozzle and 85C bed with 30% fan. Using clear PETG will also make it easy to tell if you are getting the right temperatures. The printed part should be as clear as the filament; if it is cloudy you need to print it hotter.

With ABS it is a similar story. 275C nozzle, 125C bed and 0% fan work best for us. If your printer cannot handle these temperatures stick with PLA it will work out just fine.