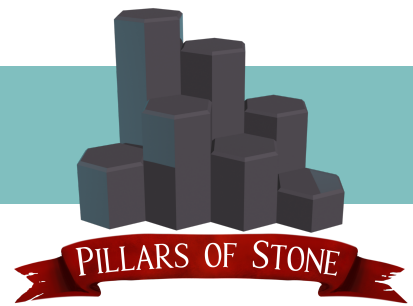


# 3D PRINTING GUIDE



## Recommended Settings

In general we recommend the following settings for printing Pillars of Stone on an FDM printer:

**Nozzle size:** .4mm

**Layer Height:** 0.2mm

**Infill:** 15-20%

**Supports:** Touching Build-plate only

**Speed:** 50 - 60 mm/s (this will depend on the capabilities of your specific printer)

I use a Creality Ender 3 pro, and a CR-10S for FDM printing.

If you don't mind losing just a little detail, but speeding up the print times, we've also had success with these settings:

**Nozzle size:** .6mm

**Layer Height:** 0.24mm

**Infill:** 15%

**Supports:** Touching Build-plate only

**Speed:** 50 - 60 mm/s (this will depend on the capabilities of your specific printer)

\*There is one piece that **REQUIRES** supports everywhere, and that is the TURTLE SHELL with WINGS.

\*The Rock Hovel dice tower can be difficult to print if your printer has trouble "bridging". If you are unsure, we recommend using supports in this case.

If you want to increase your detail on items such as the Initiative Tracker Clips, try using .12mm layer heights.