Printing Guide For Qortal Case 3.0

\*PRINTING MATERIAL REQUIRED PETG\*

Layers/Perimeters Print Settings

* Layer height-0.2mm
* Perimeters-4

Infill Print Settings

* Infill/Fill Density-100%
* Infill/Fill Pattern-Rectilinear
* Top/Bottom Fill Pattern-Rectilinear

Skirt/Brim Settings

* \*NOTE\* on ALL Prusa printers we have had no need for skirts or brims. If you know your machine well you will be able to identify if you need skirt and or brim. For NEW individuals to 3D Printing I would recommend to at least use a brim. Or play with your machine for a few prints to see if you need the extra adhesion.

Support Material Settings

* \*NOTE\* on ALL Prusa printers we have had no need for Supports. If you know your machine well you will be able to identify if you need to implement supports. For NEW individuals to 3D Printing I would recommend to at least use supports on the SSD case. The only portion that may need supports is the posts that go into the bottom portion of the case. Highley recommend again to play with your machine for a few prints to see if you need supports on this small of overhangs.

Print Speed Settings

* Perimeters – MAX 45mm/s
* Small Perimeters-MAX 25mm/s
* External Perimeters- MAX 25mm/s
* Infill-Max 60 mm/s
* Solid Infill-MAX 60mm/s
* Top Solid Infill-MAX 40mm/s
* Support Material-MAX 40mm/s
* Bridges-MAX 30mm/s
* Gaps-Max 40 mm/s
* First Layer- Max 10 mm/s

Printing Temperature Settings

* Nozzle-250c
* Bed/Build Palate-90c

\*Please note these temperature values are based off of Prusament PETG. On Genuine Prusa’s\*

\*ATTENION\* If you are using a Genuine Prusa Printer Please Use The Stock Profile Of (0.20mm Quality) This Is What Our Print Farm Uses. \*Recommended but NOT Needed Is A Textured Sheet.