

How to make G. Code for Prusa Mk4

1. Install Prusa slicer

https://www.prusa3d.com/page/prusaslicer_424/

2. Select Prusa Mk4 with 0.4 mm Nozzle

MK4 Family



Original Prusa MK4

☒ 0.4 mm nozzle

Alternate nozzles:

☐ 0.25 mm nozzle

☐ 0.3 mm nozzle

☐ 0.5 mm nozzle

☐ 0.6 mm nozzle

☐ 0.8 mm nozzle

3. Import your .stl file

File > Import > STL

4. Orient your part

Either do it manually or use place by surface, chose orientation that optimizes, layer orientation and support material used



5. Select Generic PLA

Filament :

  Generic PLA  

6. Enable Support

Supports:  Everywhere 

7. **Select Desired Infill**

Infill: 15% ▼

My recommendation, most parts 15-20 percent, extra strength 30 percent, don't go over 40 percent in most cases

8. **Slice**

Click slice now at bottom right and export file to somewhere you can locate later, please note the print time and amount of filament used, both stated in bottom right,

9. **Upload**

Upload your file to the website