

Prusa Slicer: Quick Start Guide

Download Prusa Slicer https://www.prusa3d.com/page/prusaslicer_424/
(already installed on Makerspace computers)

Configure Slicer

- Add ONLY Prusa MK3S+ Printer with 0.4 mm nozzle
- Change NO other settings (*just keep clicking next, then finish*)

Create your model

- Save your SolidWorks or other solid file as a **.STL**

Set up your print

- Open Prusa Slicer, drag in your .STL parts
- Navigation: Left click – Rotate, Right Click – Pan, scroll wheel - zoom
- Use ctrl+c, ctrl+v to copy paste if you need multiples
- Use the side toolbar buttons to move your model around
- Place on Face – most useful. Just select the bottom of your part

Orienting your part

The biggest consideration when setting up to print is how you choose to orient your model. Orientation affects the model strength, surface finish, fit and tolerance of features, and whether supports are needed.

Simple parts designed specifically for 3D printing can ideally be printed without supports and with minimal compromises on strength and finish.

Print Settings – Set the layer height (the thickness of each layer)

- Leave on 0.15mm Quality for most prints
- Thicker layers print faster, less resolution

Filament – Leave on Generic PLA unless you're using another filament

Printer – Must be set to the Original Prusa i3 MK3S & MK3S+

Supports - provide material for model layers to build off of

- Basic – Set to **Support on build plate only**, unless your model is very complex and requires support **Everywhere**.

Infill – percentage reflects how solid the part is. This affects print time!

- For most parts, you can leave this at 15-30%
- For extra strength, add more Perimeters instead of just increasing infill
 - Print Settings > Layers and perimeters

- Prusa FFF
- Prusa MSLA
- Other Vendors
- Custom Printer
- Filaments
- Updates
- Reload from disk
- Files association
- View mode

MK3 Family



Original Prusa i3 MK3S & MK3S+

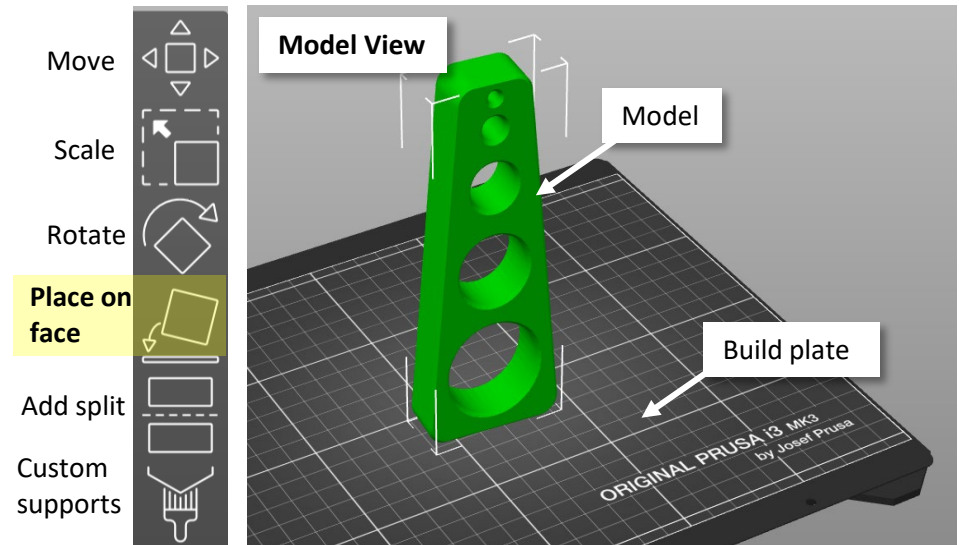
☒ 0.4 mm nozzle

File name:

Save as type:

Description: [Add a description](#)

Go to options to adjust resolution of .STL file if needed



Print settings :

0.20mm QUALITY

Filament :

Generic PLA (modified)

Printer :

Original Prusa i3 MK3S & MK3S+

Supports:

Infill: Brim: ☐

Support on build plate only
For support enforcers only
Everywhere

Support Material

Support is needed under overhangs so the layers of the part have something to adhere to below them. Adding support results in a rougher surface, so avoid designing parts with overhangs if possible.

- Needed for overhangs greater than 50 degrees or unsupported bridges
- Advanced – Add only supports you know you will need using the custom supports tool, set to **for support enforcers only** and draw on supports



Slice Now

- Once you've set your print settings, click **Slice Now** to slice the model
- The window switches to the sliced view, you need to toggle back to model view to move or re-orient your parts

Check the sliced model

- ✓ Any unsupported overhangs? (will be marked with blue)
- ✓ Any support material you don't want?
- ✓ Any parts off the build area or too close to each other?
- ✓ Drag the height slider to look inside the model layer by layer
- ✓ Do you like to look of the top and bottom infill?
- ✓ All good? Export G-Code

Export G-Code

- Confirm your print time is ok
- Once complete, click Export G-Code and save the file
- For printing off an SD card, save G-Code directly to your SD card
- For saving or sending your Build file for approval, Save Project as
 - Save as .3MF file (this saves all your settings and models)

Slice now

Sliced View

Infill Pattern

Perimeters

Top solid infill

Support

Drag the height slider to see each layer

Toggle between **model** and **sliced view**

Estimated printing time:
- normal mode 38m
- stealth mode 38m

Check the total printing time

Export G-code

Advanced Adjustments / Tweaks

For advanced users looking to customize print settings even further

- Set to **Expert Mode** and go to **Print Settings** on the top tabs
- **Layers and perimeters** – add perimeters for stronger models
- **Infill** – change fill density, pattern, top/bottom infill
- **Support Material** - adjust overhang threshold (angle from horizontal)
- **Filament Settings, Printer Settings** – leave these alone

For additional resources, Prusa Research has extensive online guides:
https://help.prusa3d.com/article/modeling-with-3d-printing-in-mind_164135

Plater Print Settings Filament Settings Printer Settings

Layers and perimeters

Infill

Vertical shells

Perimeters: 2

Fill density: 15%

Fill pattern: Gyroid

Length of the infill anchor: 2.5

Support material

Generate support material: ☒

Auto generated supports: ☒

Overhang threshold: 40