Welcome to Prusa Slicer

By: JT Raymond

Oct. '23

1) you'll need to download the Prusa Slicer

- URL: https://www.prusa3d.com/page/prusaslicer_424/
- Download whichever version aligns with your OS.
- Throughout this tutorial, I'll be using the Windows version.

2) Installation Settings

- You will be prompted with this screen.
- Select only the boxes that I do, as we will not be using the other drivers.
- Our printers are Prusa i3 MK3S+, so I selected that box.
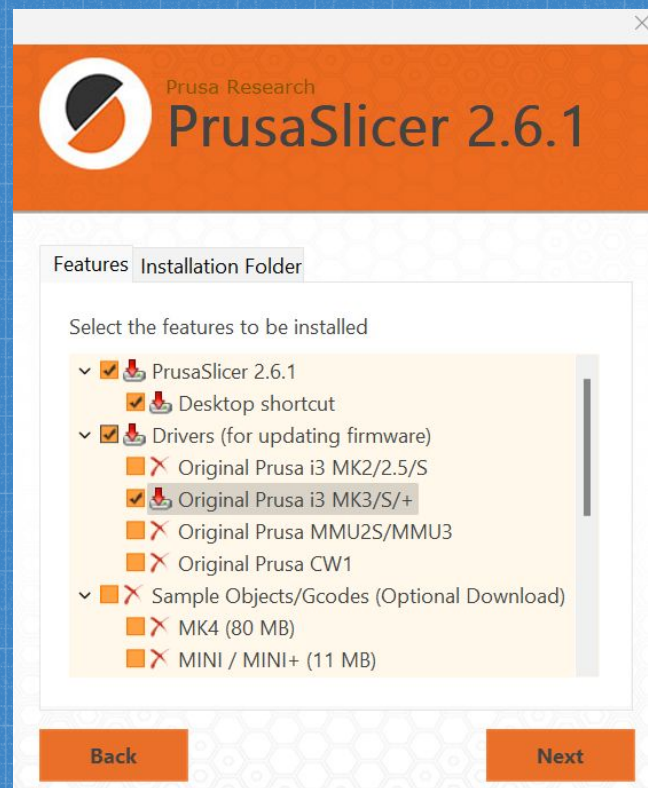
3) Finalizing Installation

- Follow all of the remaining steps to install the Prusa Slicer.
- Finally, click "finish."
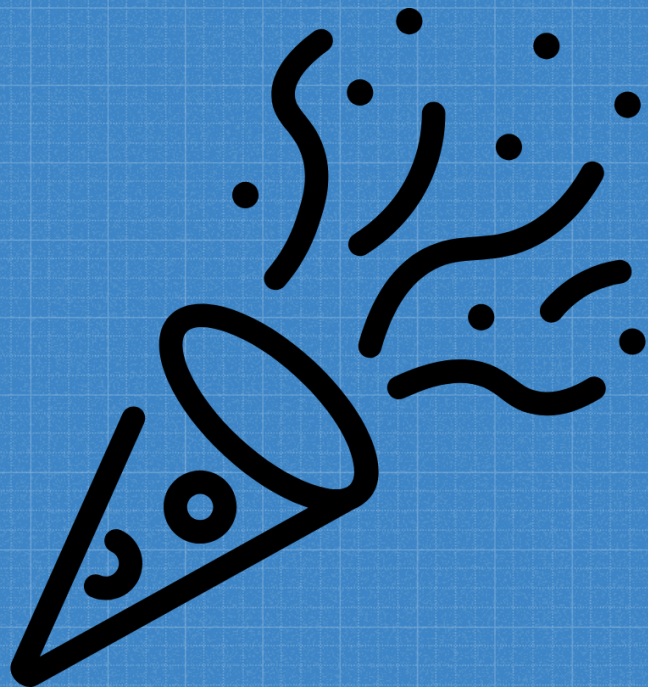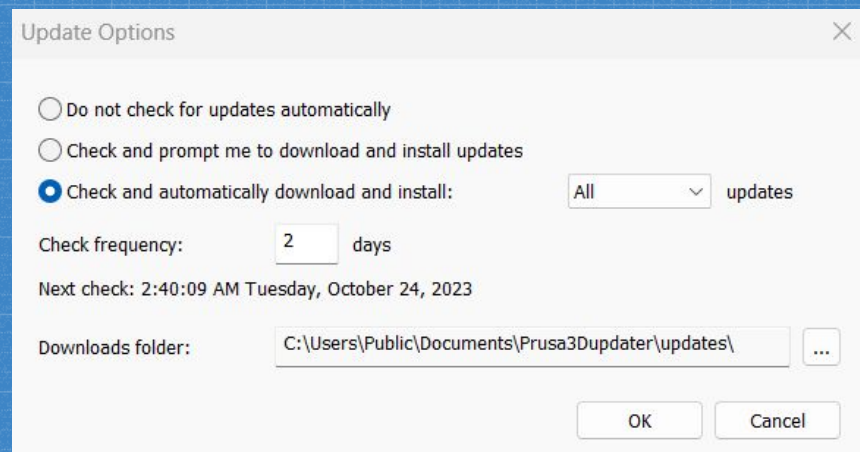- Congrats! You've downloaded the Prusa Slicer!

4) Automatic Updates Setup

- When you open the "Configure Automatic Updates" program for the first time, you'll be met with this screen.
- You shouldn't have to change anything, but try to match mine.
- Click OK and exit the program.

5) Welcome to the Prusa Slicer!

- When you first open the slicer, you will be met with the following screen.
- It is important to familiarize yourself with the basics.

- For basic prints (almost all), you will stay between the "Plater" and the "Print Settings" menus (Top Left).
- The "Plater" will let you adjust you file orientation, scale, slice, and rotate features (You are on this now).
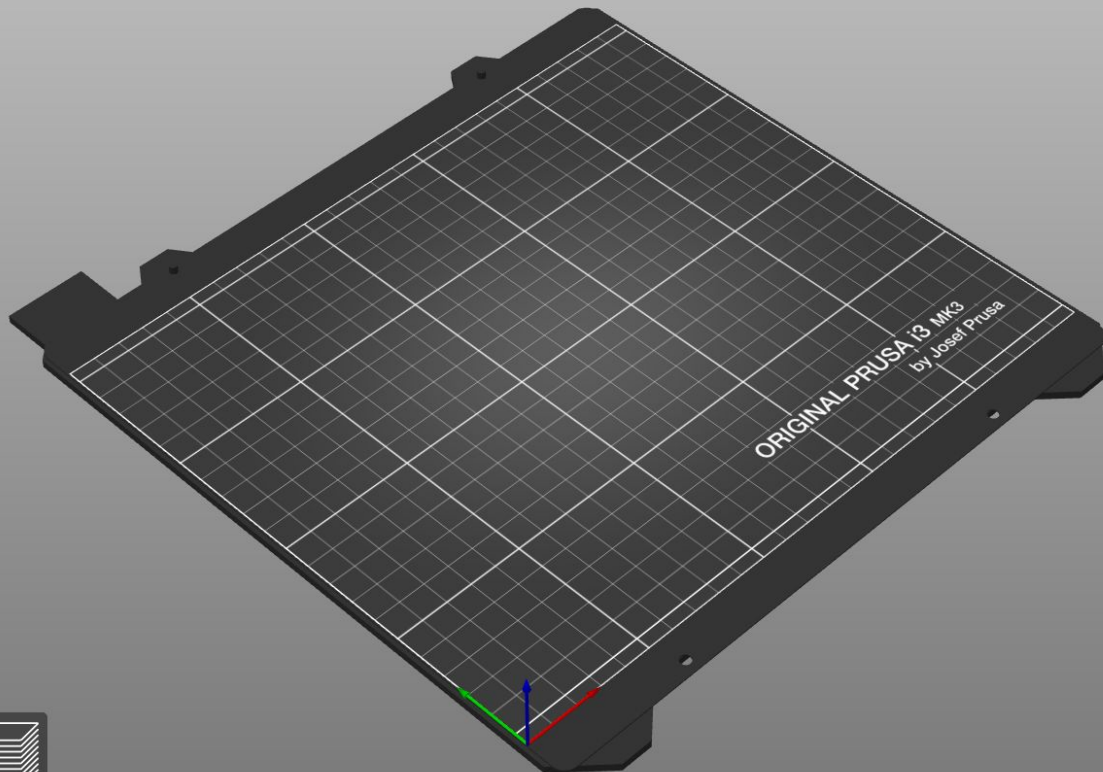
File   Edit   Window   View   Configuration   Help

Plater   Print Settings   Filament Settings   Printer Settings

Simple   Advanced   Expert

Print settings:
0.05mm ULTRADETAIL

Filament:
Prusa PLA

Printer:
Original Prusa i3 MK3S & MK3S+

Supports: None

Infill: 15%   Brim:

Name	Editing

ORIGINAL PRUSA i3 MK3
by Josef Prusa

Slice now

6) Print Settings Menu Basics

- The "Print Settings" menu will look different.
- It will be confusing if you don't already know what you're doing.
- If you follow these steps, you won't need to vary too far from the basics to see success.

- The following few slides will show the basic print settings that you will need to know.

PLEASE NOTE you will have to set these again for EVERY print you do in a new session.

File   Edit   Window   View   Configuration   Help

Plater | Print Settings | Filament Settings | Printer Settings   Simple   Advanced   Expert

0.05mm ULTRADETAIL @MK3

- Layers and perimeters
- Infill
- Skirt and brim
- Support material
- Speed
- Multiple Extruders
- Advanced
- Output options
- Notes
- Dependencies

Layer height

Layer height: 0.05   mm
First layer height: 0.2   mm

Vertical shells

Perimeters: 3   (minimum)
Spiral vase: ☐

Recommended object thin wall thickness for layer height 0.05 and 2 lines: 0.89 mm , 4 lines: 1.77 mm , 6 lines: 2.65 mm

Horizontal shells

Solid layers: Top: 14   Bottom: 10
Minimum shell thickness: Top: 0.7   mm   Bottom: 0.5   mm

Top shell is 0.7 mm thick for layer height 0.05 mm.
Bottom shell is 0.5 mm thick for layer height 0.05 mm.

Quality (slower slicing)

Extra perimeters if needed: ☐
Extra perimeters on overhangs (Experimental):   ☐
Avoid crossing curled overhangs (Experimental): ☐
Avoid crossing perimeters: ☐
Avoid crossing perimeters - Max detour length: 0   mm or % (zero to disable)
Detect thin walls: ☐
Thick bridges: ☑
Detect bridging perimeters: ☑

File   Edit   Window   View   Configuration   Help

Plater | Print Settings | Filament Settings | Printer Settings

0.05mm ULTRADETAIL @MK3 (modified)

- Layers and perimeters
- Infill
- Skirt and brim
- Support material
- Speed
- Multiple Extruders
- Advanced
- Output options
- Notes
- Dependencies

Infill

- Fill density: 10% %
- Fill pattern: Gyroid
- Length of the infill anchor: 2.5 mm or %
- Maximum length of the infill anchor: 12 mm or %
- Top fill pattern: Monotonic Lines
- Bottom fill pattern: Monotonic

Ironing

- Enable ironing:
- Ironing Type: All top surfaces
- Flow rate: 15 %
- Spacing between ironing passes: 0.1 mm

Reducing printing time

- Combine infill every: 1 layers

Advanced

- Solid infill every: 0 layers
- Fill angle: 45 °
- Solid infill threshold area: 0 mm²
- Bridging angle: 0 °
- Only retract when crossing perimeters:
- Infill before perimeters:

10

Plater   **Print Settings**   **Filament Settings**   **Printer Settings**

🚩🔒 0.05mm ULTRADETAIL @MK3 (modified)   ⌄   💾   ❓   🔒 ●   🔍 ⚙

- ☰ Layers and perimeters
- ⬡ Infill
- ⬡ Skirt and brim
- ⬡ Support material
- 🕑 Speed
- ⍦ Multiple Extruders
- ⚙ Advanced
- ↪ Output options
- 📄 Notes
- ⚙ Dependencies

Skirt

🟡 Loops (minimum):   🔒 ● [1   ⬍]

🟢 Distance from brim/object: 🔒 ● [2   ] mm

🟡 Skirt height: 🔒 ● [3   ⬍] layers

🟡 Draft shield: 🔒 ● [Disabled   ⌄]

🔴 Minimal filament extrusion length: 🔒 ● [4   ] mm

Brim

🟢 Brim type: 🔒 ● [Outer brim only ⌄]

🟢 Brim width: 🔒 ● [0   ] mm

🟡 Brim separation gap: 🔒 ● [0.1   ] mm

File Edit Window View Configuration Help

Plater | Print Settings | Filament Settings | Printer Settings

0.05mm ULTRADETAIL @MK3 (modified)

- Layers and perimeters
- Infill
- Skirt and brim
- Support material
- Speed
- Multiple Extruders
- Advanced
- Output options
- Notes
- Dependencies

Support material

Generate support material: ☑
Auto generated supports: ☑
Overhang threshold: 50 °
Enforce support for the first: 0 layers
First layer density: 90 %
First layer expansion: 3 mm

Raft

Raft layers: 0 layers
Raft contact Z distance: 0.1 mm
Raft expansion: 1.5 mm

Options for support material and raft

Style: Grid
Top contact Z distance: 0.1 (detachable) mm
Bottom contact Z distance: Same as top mm
Pattern: Rectilinear
With sheath around the support: ☐
Pattern spacing: 4 mm
Pattern angle: 0 °
Closing radius: 2 mm
Top interface layers: 2 (default) layers
Bottom interface layers: 0 (off) layers
Interface pattern: Rectilinear
Interface pattern spacing: 0.2 mm
Interface loops: ☐
Support on build plate only: ☐

Support on build plate only: ☐
XY separation between an object and its support: 60% mm or %
Don't support bridges: ☐
Synchronize with object layers: ☐

Organic supports

Maximum Branch Angle: 40 °
Preferred Branch Angle: 30 °
Branch Diameter: 2 mm
Branch Diameter Angle: 3 °
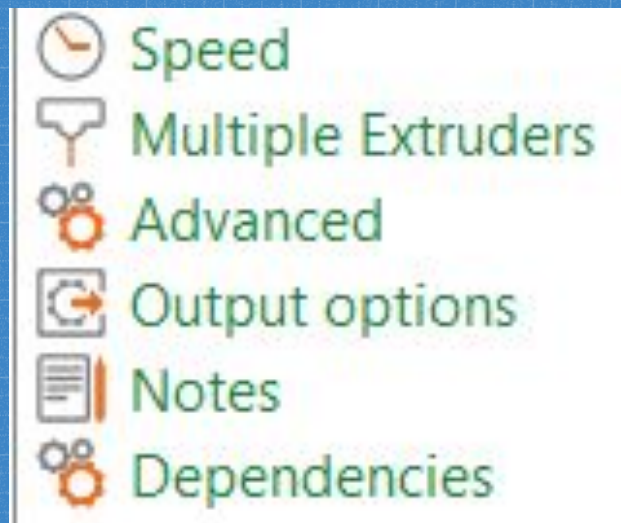Branch Diameter with double walls: 3 mm
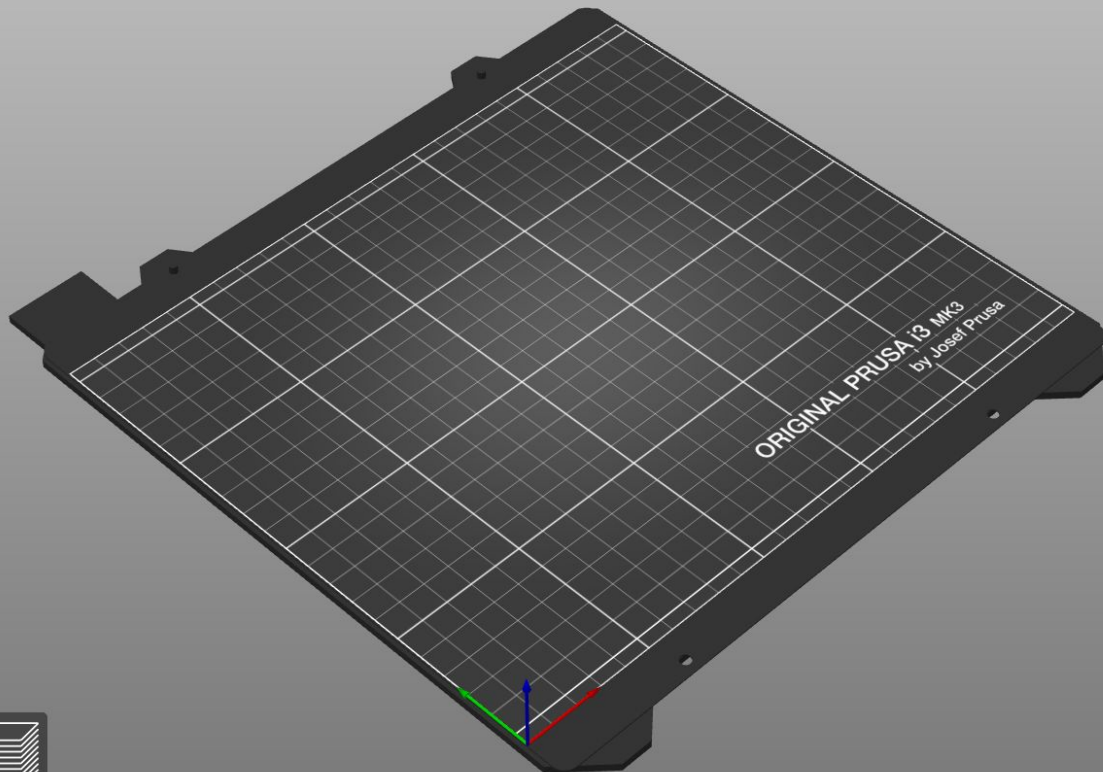Tip Diameter: 0.6 mm
Branch Distance: 1
Branch Density: 30 %

12

5) Prusa Slicer Basics Follow-Up

- For the remainder of the sub-menu settings, you won't have to change anything to see successful prints.

File   Edit   Window   View   Configuration   Help

Plater   Print Settings   Filament Settings   Printer Settings

Simple   Advanced   Expert

Print settings:
0.05mm ULTRADETAIL

Filament:
Prusa PLA

Printer:
Original Prusa i3 MK3S & MK3S+

Supports: None

Infill: 15%   Brim:

Name	Editing

Slice now

ORIGINAL PRUSA i3 MK3
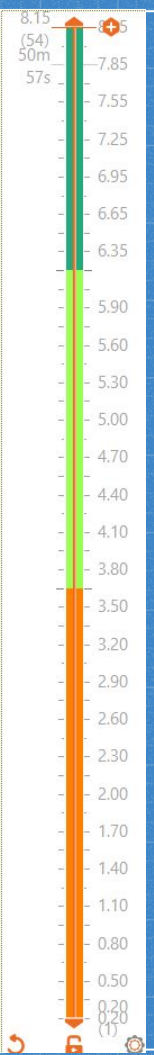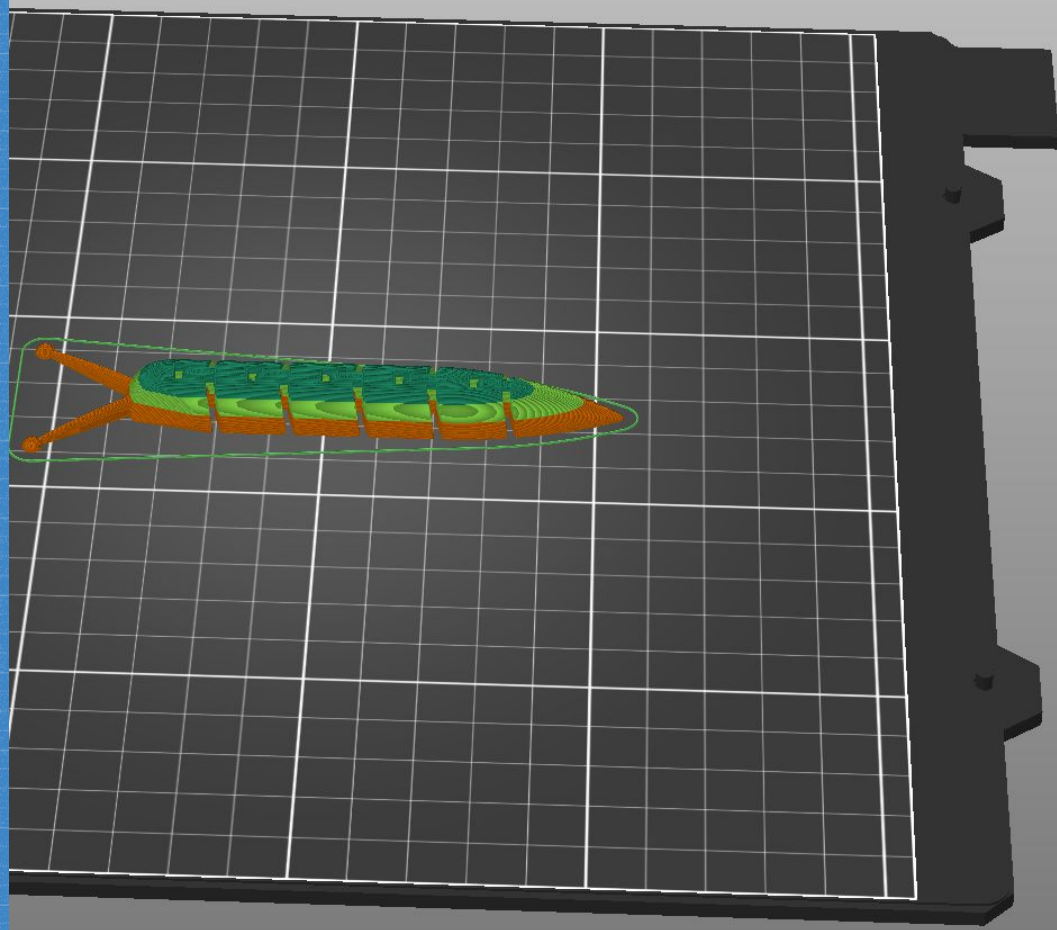by Josef Prusa

7) Plater Menu Basics

- When you want to add an STL file into the slicer, you will select the "cube with a plus" icon.

- After clicking that, you will be greeted by your computer's file browser. From there, you need to select the file you'd like to print, then click "Open."
- All other options in the "Plater" menu are self-explanatory and can be used to organize and move objects.

Color Changes While Printing

- This should be rarely used, but may be fun to experiment with.
- While dragging through the layers, like normal, if you click the little "+," you will see a new color added into your sliced model. For example, you can see the 3 different colors on the slug.

- By doing this, your Prusa printer will automatically pause the print when it finishes a certain color, then ask you to manually remove and load new filament.
- This will occur as many times as you request a filament change.

Happy Printing! :)