

Intro to 3D Printing

Created by: Liam MacLean

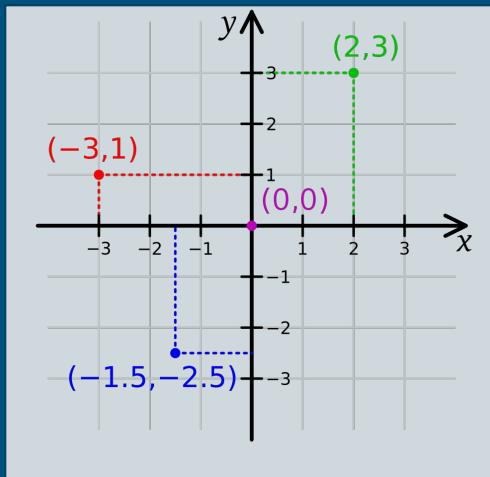


What is 3D Printing?

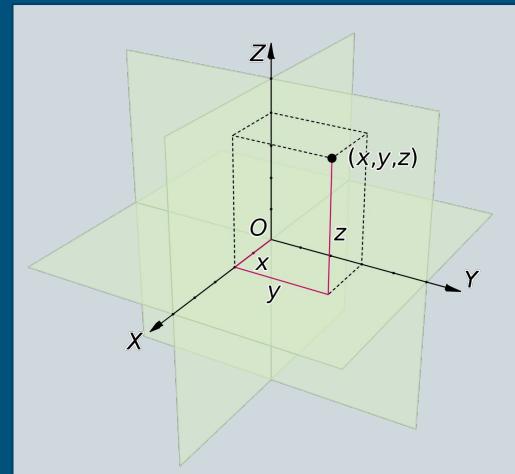
- First appeared in 1981
- Two (main) types of 3D Printing:
 - Fused Filament Fabrication (FFF, also called FDM) (The one we will be working with)
 - Works from bottom to top, laying down material
 - Build size is much larger, it is more versatile, and comparatively very fast
 - Worse at printing highly detailed object, especially at small sizes
 - Resin
 - Uses UV Light to solidify liquid resin into a solid shape
 - Better for details, requires less post-processing (work like sanding to finish the 3D Printing object)
 - Print size is much smaller, it takes much more time, and it is much less versatile

Quick Introduction to Coordinate Planes

- This is a 2D Coordinate Plane
- It has 2 dimensions: X and Y
- You can find any point on this plane with 2 numbers: the X coordinate and the Y coordinate

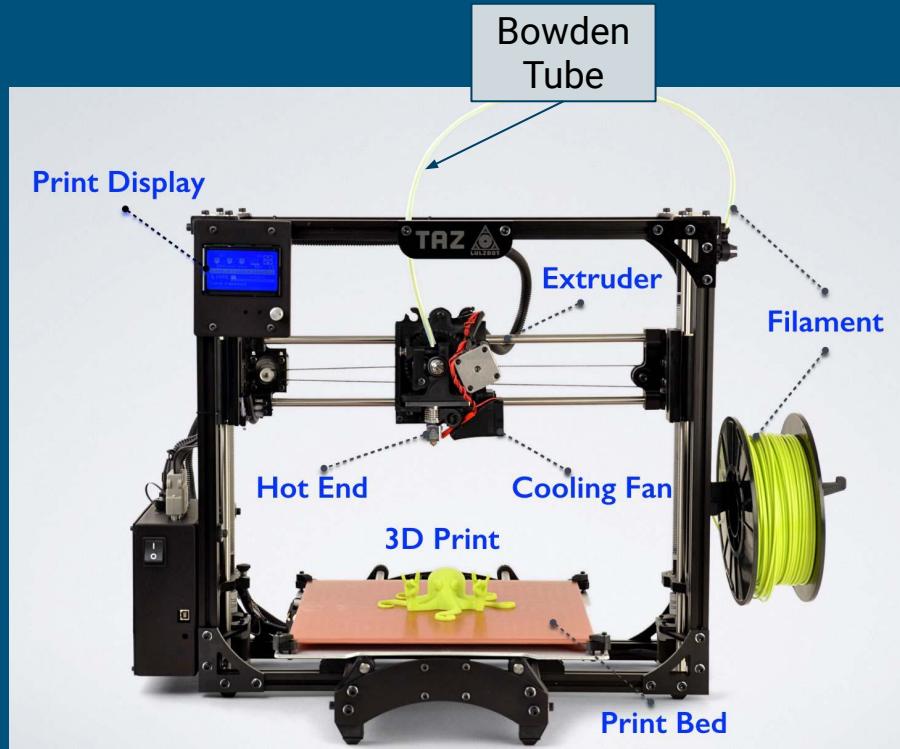


- This is a 3D Coordinate Plane
- It has 3 dimensions: X, Y and Z
- Just like the 2D Plane, you can find any point on this plane with 3 numbers: the X, Y and Z coordinates



Anatomy of the Printer

- **Extruder** - Feeds filament to the Hot End
 - Bowden tube - not present on all printers
- **Hot End** - Includes all parts that are involved in melting filament
 - Nozzle - Traditionally 4mm
 - Cooling Fans
- **Bed** - Where the print is built



How Does it Work?

- FDM Printers work on a xyz plane
- Printers Are Dumb
 - All an FDM Printer does is move a nozzle to a location and place a set amount of material
 - Most often this is done in continuous lines
- FDM Printers always work from bottom-up
 - When has the printer has finished with a layer (usually 2mm tall), the nozzle moves up (by the same amount as the layer height) and starts on the next layer.



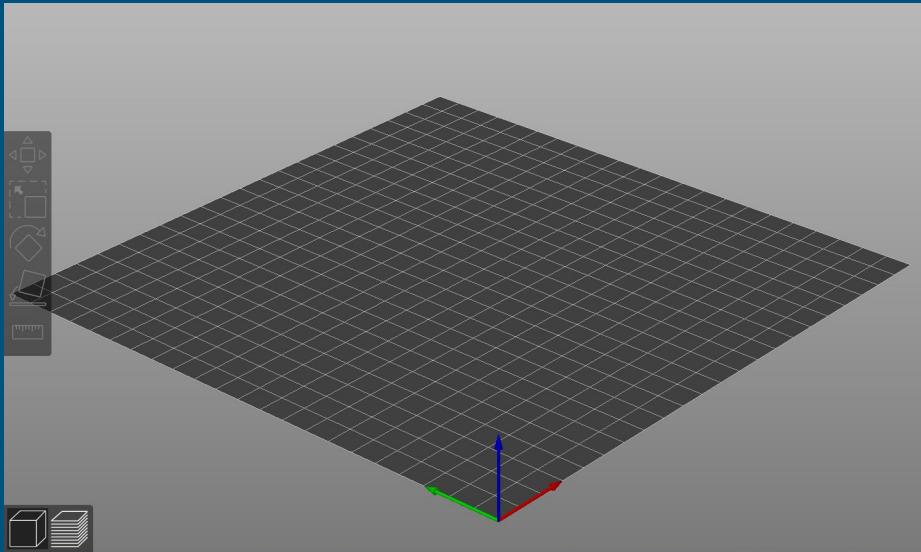
How Do We Print Things?

- Model
 - Created in any form of modeling software, in our case Sketchup
- Slicer
 - Slicer are programs that translate the model into a set of instructions (**GCode**) that the printer can understand
 - These instructions only tell the printer basic information: where to move the nozzle, how fast, for how long, how much material to let out, etc.
 - Because of this **PRINTER'S NEVER KNOW WHEN THEY'VE FAILED**. They will keep printing like nothing is wrong until they finish their instructions.



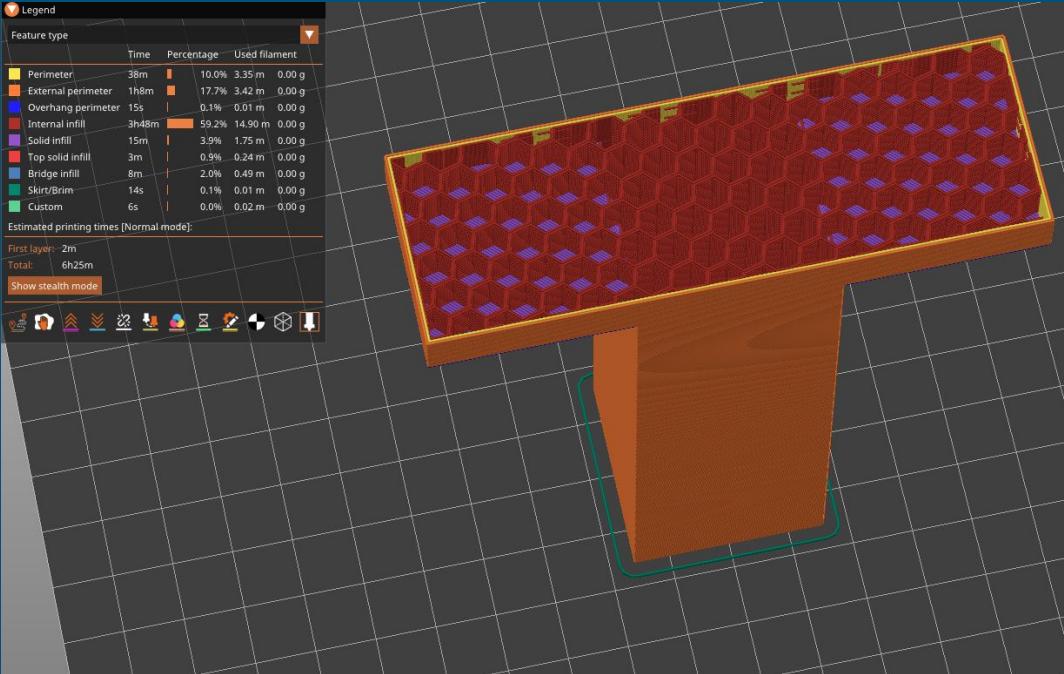
How Slicers Work

- It's all in the name!
 - The Slicer first cuts your model vertically into layers of a defined height (usually 2mm)
 - It then calculates the best path for the nozzle to follow for each layer
 - Finally it puts all of these instructions together along with other information for the printer to work
 - It then gives you a GCode file, which you can transfer to your printer to begin printing



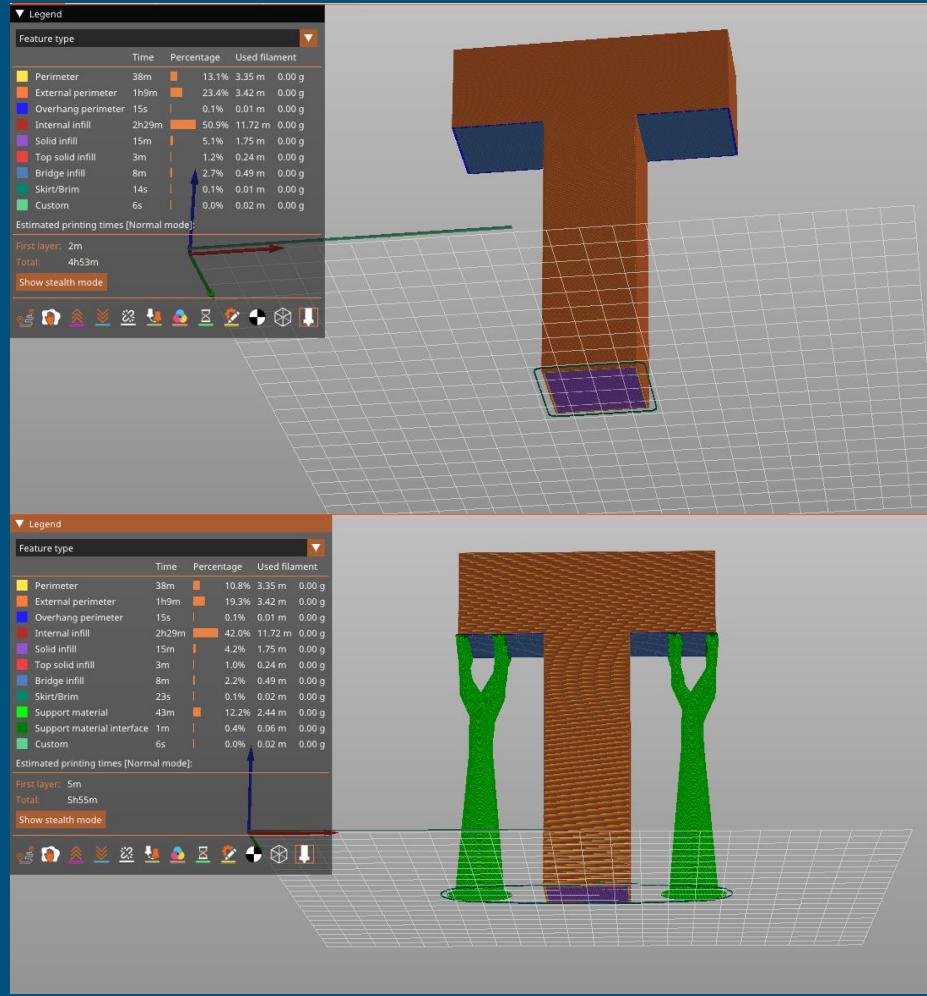
Lines that the Printer Prints

- Almost all FDM prints involve 3 main type of lines which the printer lays down
 - Perimeter - the outside (or sometimes inside) of the model
 - Traditionally laid down more slowly, because it is the visible part of the model
 - Infill - the inside of the model
 - Most 3D printed models aren't completely solid, they are usually only filled to 15-25% to save on material
 - This is usually laid down much more quickly as it is invisible in the completed model
 - Overhang and Bridges
 - 3D Printers always print from the bottom up
 - Parts that don't have anything below them are called overhangs or bridges.



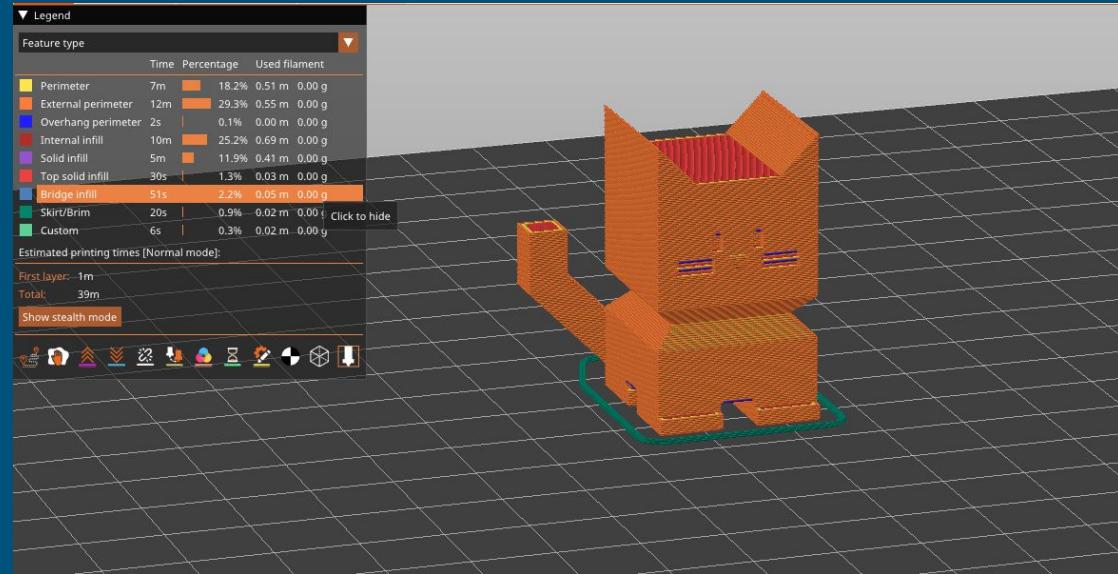
Overhang, Bridges and Supports

- When a part of the print doesn't have anything under it, it's called **overhang**
- Most printers can handle small amounts of overhang, but the quality is usually much lower
 - It is ideal in most cases to reduce the overhang in your prints as much as possible
 - Most printers are able to handle 45 degrees of overhang, use this when designing objects you intend to 3D print
- If you can't eliminate all major overhang, you use **supports**
 - **Supports** are printed parts that are designed to provide a base layer for overhang and that break off of the final product
 - Note how in the picture, the supports don't cover the entire overhang. That's because the slicer knows how much of an overhang our printer can print, and so optimizes the supports for that distance.



Designing for 3D Printers

- When possible, limit overhangs to 45 degrees
 - Note how the Calibration Cat's (CaliCat) tail is angled so that the printer is able to handle it without supports
- If bridges are needed, make them short and have solid bases on either side
 - Notice how CaliCat's belly has only a short bridge

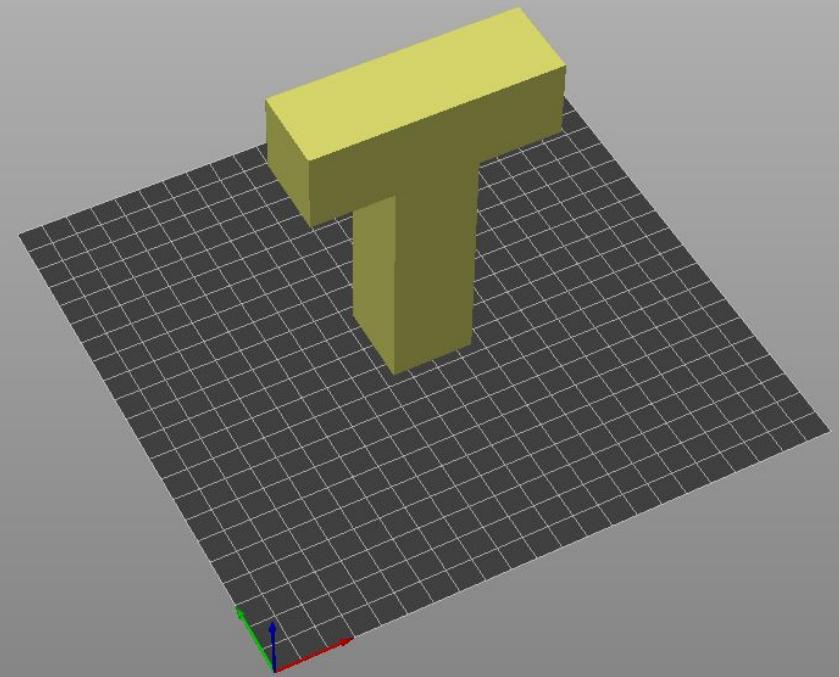


Orientation

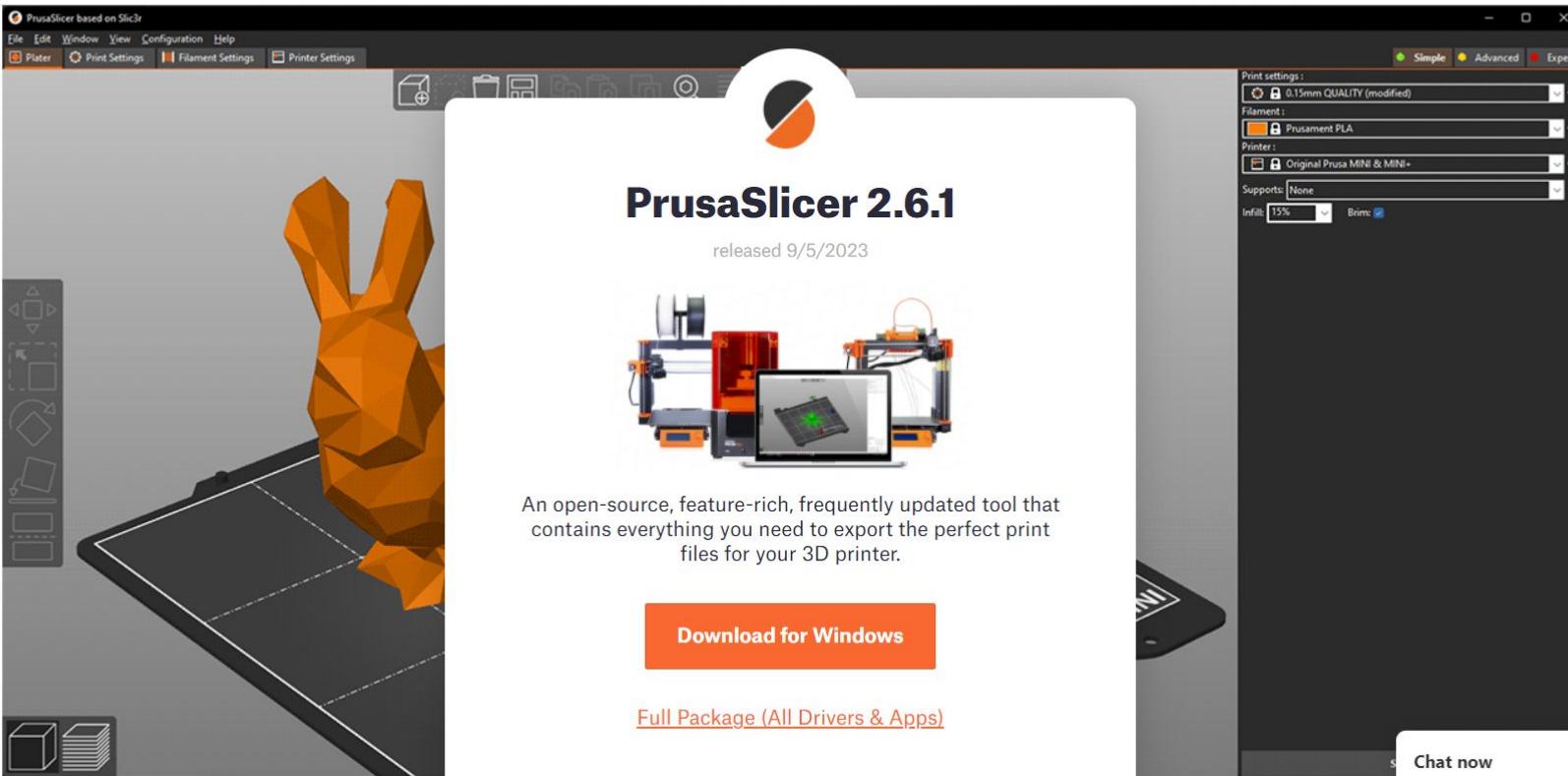
- It is usually best to avoid supports if possible
 - Higher likelihood of failure
 - More material
 - More time to both print and post-process (turning the printed object into its final form)
- To avoid supports, think about the **orientation** of your object
- Two main rules to follow when orienting your object for printing:
 - A large, flat surface in contact with the bed
 - Minimize the amount of overhang

How to Orient?

Using the principles we have discussed, what would be the best way to orient our capital "T" for printing?



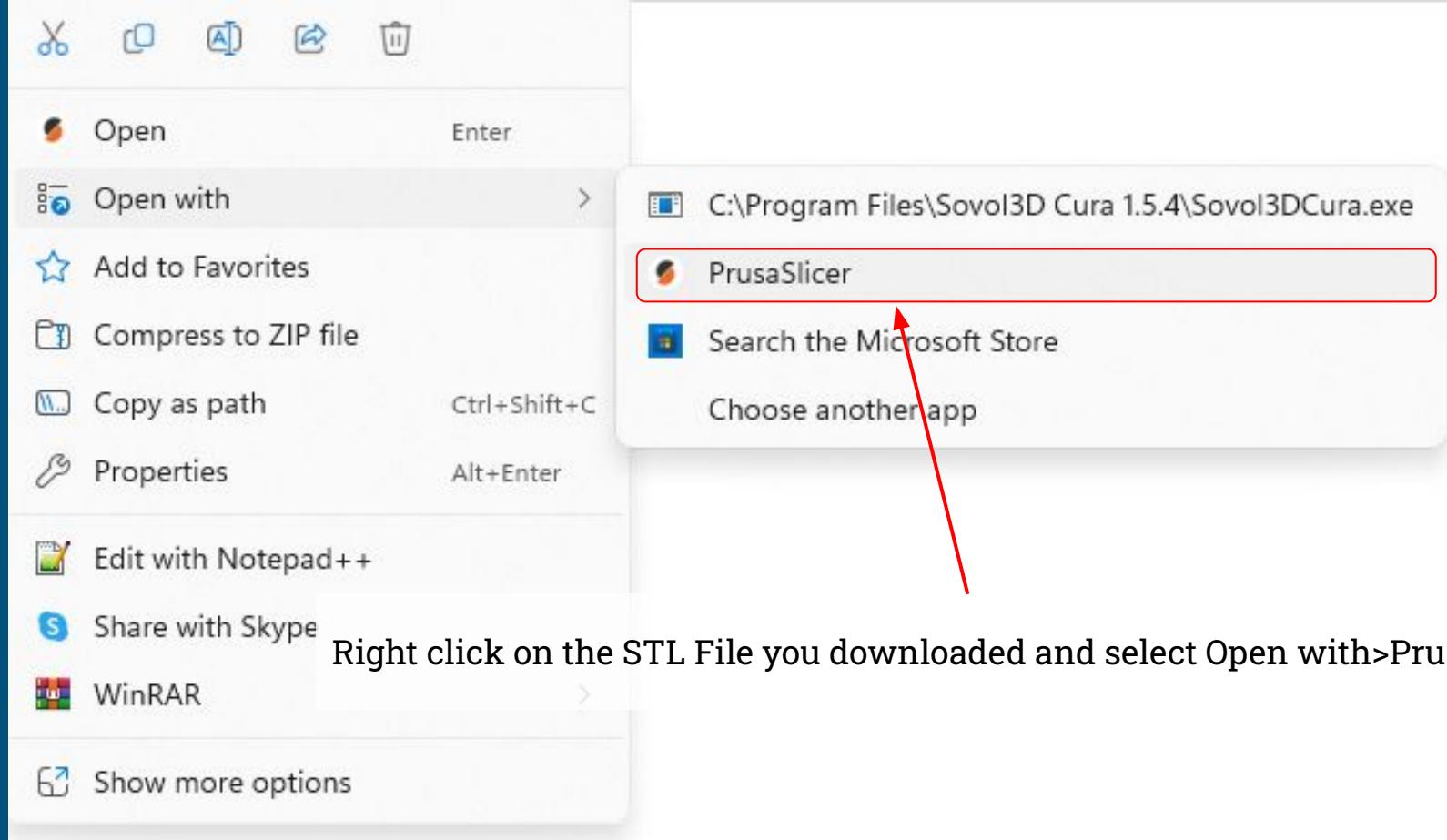
Let's Get Printing!



First, let's download PrusaSlicer.



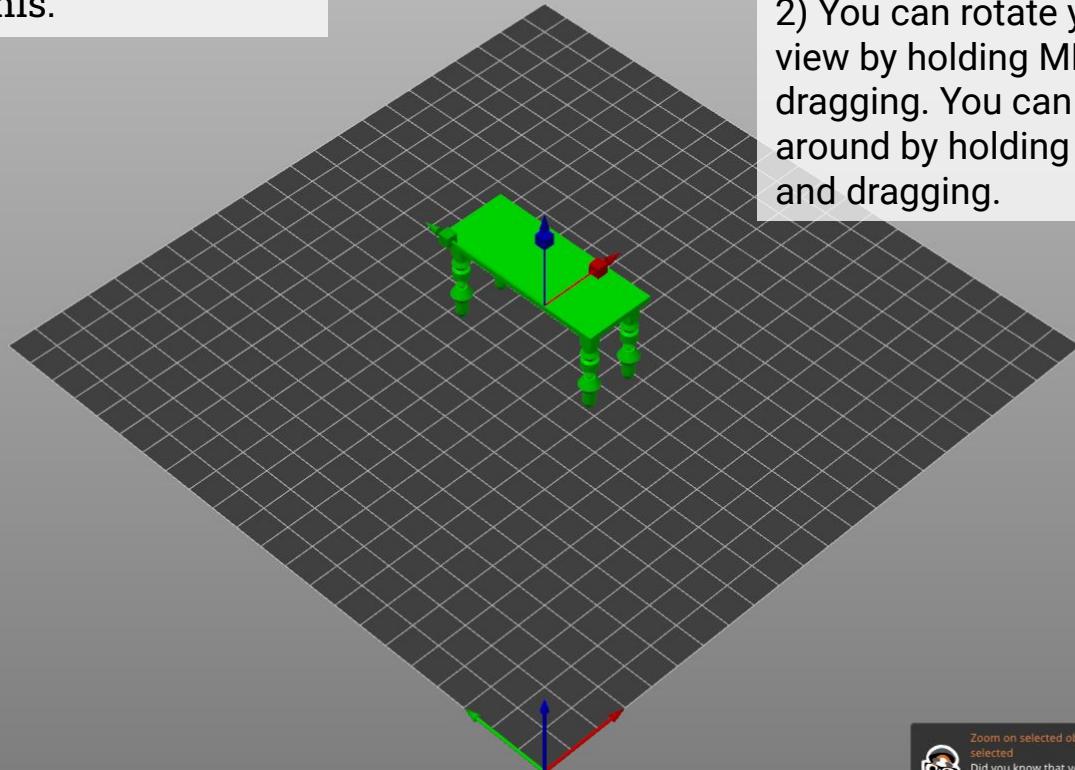
Log on to Sketchup and in the Menu of your table select Download>STL



Right click on the STL File you downloaded and select Open with>PrusaSlicer



1) Your screen should now look like this.



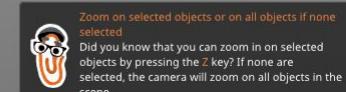
2) You can rotate your view by holding MB1 and dragging. You can move around by holding MB2 and dragging.

Print settings:

0.20mm QUALITY @MK3 - Copy
Filament:
Generic PLA - SV-06
Printer:
Sovol Octo * Sovol SV-06 - .4mm Nozzle
Supports: For support enforcers only
Infill: 15% Brim: <input type="checkbox"/>

Object manipulation

Name: table.stl	x <input type="checkbox"/>	y <input type="checkbox"/>	z <input type="checkbox"/>	
Position:	110	110	16.15	mm
Rotate (relative):	0	0	0	°
Scale factors:	100	100	100	%
Size (World):	24	60	32.31	mm
<input type="checkbox"/> Inches				



Slice now



Printer/Filament Settings

Print settings:

0.20mm QUALITY @MK3 - Copy

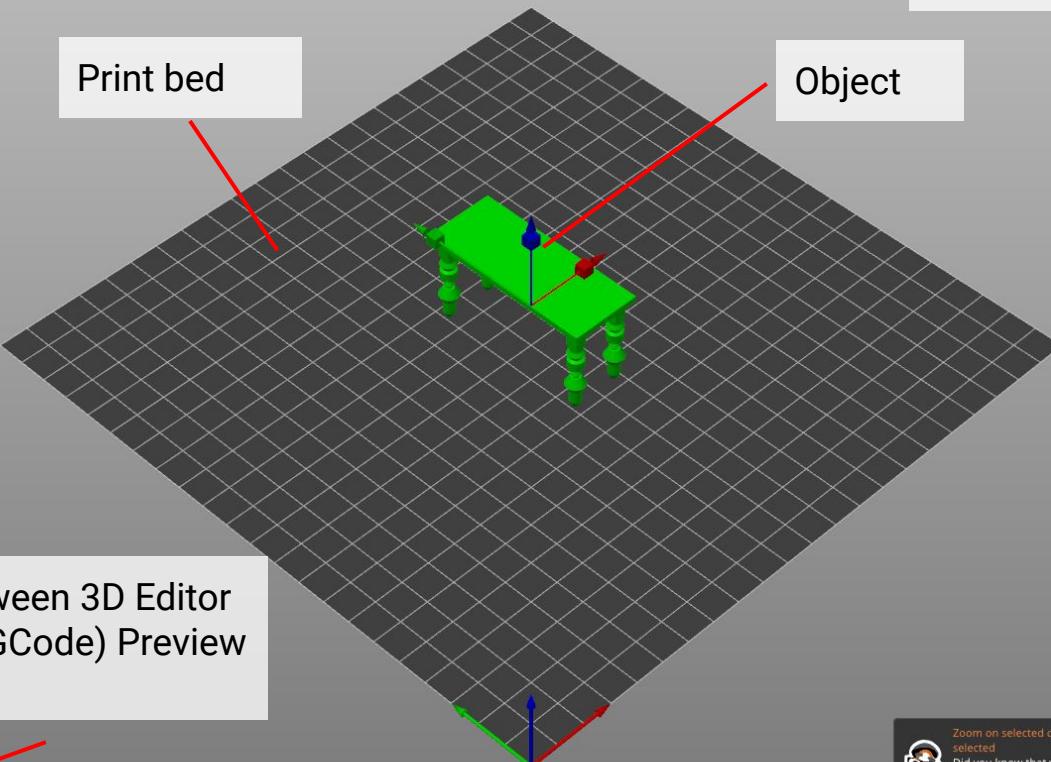
Filament: Generic PLA - SV-06

Printer: Sovol Octo * Sovol SV-06 - .4mm Nozzle

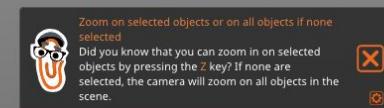
Supports: For support enforcers only

Infill: 15% Brim:

Tools



Switch between 3D Editor View and (GCode) Preview view



Click here to slice the object into GCode for the Printer

Slice now



These settings tell the slicer which printer and type of filament. For our demonstration we will use pre-made profile that can be downloaded from the bit.ly link. These settings can be edited in the upper left.

Print settings: This profile controls the general quality of print. You can reduce layer size and speed to improve quality at the cost of print speed, and visa versa.

Filament: This is the profile that controls the type material that the print will use. These settings include temperature of nozzle, amount of retraction, etc.

Printer: This is the profile that the specific printer containing settings like max print speed, nozzle width, etc.

These two settings menus deal with other aspects of the GCode.

Supports: This setting controls how the slicer creates supports for the object. There are four options:

Nothing: No supports

Support on build plate only: makes supports on all overhangs, but only starting from the print bed

For support enforcers only: this puts supports only on the places designated by the support brush (which we will discuss in a moment). This gives the most control over support placement.

Everywhere: this puts supports on all the overhangs.

Infill: This setting controls of the amount of the model that the printer will fill in. The more it fills in, the stronger it is, but the more time and material it takes. Usually 15% to 25% is plenty strong.

Print settings:

- 0.20mm QUALITY @MK3 - Copy
- Filament: Generic PLA - SV-06 0.20mm QUALITY @MK3 - Copy
- Printer: Sovol Octo * Sovol SV-06 - .4mm Nozzle

Object manipulation:

Name:	table.stl	X	Y	Z
Position:	110 110 16.15	mm		
Rotate (relative):	0 0 0	°		
Scale factors:	100 100 100	%		
Size [World]:	24 60 32.31	mm		
<input type="checkbox"/> Inches				

Legend

Feature type

	Time	Percentage	Used filament
Perimeter	5m	14.4%	0.25 m 0.00 g
External perimeter	7m	20.0%	0.30 m 0.00 g
Overhang perimeter	14s	0.7%	0.00 m 0.00 g
Internal infill	2m	6.0%	0.09 m 0.00 g
Solid infill	8m	25.9%	0.94 m 0.00 g
Top solid infill	2m	4.8%	0.11 m 0.00 g
Bridge infill	2m	7.3%	0.12 m 0.00 g
Skirt/Brim	15s	0.7%	0.01 m 0.00 g
Custom	6s	0.3%	0.02 m 0.00 g

Estimated printing times:

First layer: 19s

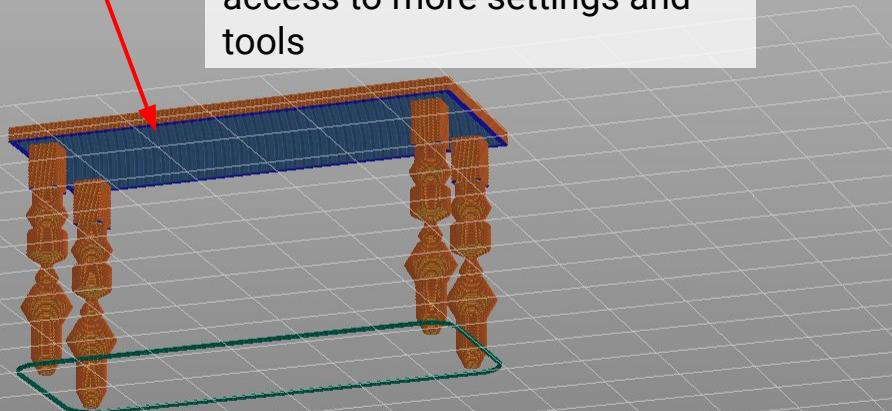
Total: 33m



1) Notice how we have a big overhang here. If we want to print this way we'll need supports

2) Click Advanced here to get access to more settings and tools

3) Now click here to switch back to object view

32.40
(162)

32.00

31.00

30.00

29.00

28.00

27.00

26.00

25.00

24.00

23.00

22.00

21.00

20.00

19.00

18.00

17.00

16.00

15.00

14.00

13.00

12.00

11.00

10.00

9.00

8.00

7.00

6.00

5.00

4.00

3.00

2.00

1.00

0.20

(1)

Simple

Advanced

Expert

Print settings:

0.20mm QUALITY @ MK3 - Copy

Filament:

Generic PLA - SV-06

Printer:

Sovol Octo * Sovol SV-06 - .4mm Nozzle

Supports: For support enforcers only

Infill: 15% Brim:

Object manipulation

Name: table.stl

Position: 110 110 16.15 mm

Rotate (relative): 0 0 0 °

Scale factors: 100 100 100 %

Size [World]: 24 60 32.31 mm

Inches

Sliced Info

Used Filament (m) 1.85

Used Filament (mm³) 4451.43

Estimated printing time:

- normal mode 33m

- stealth mode 33m

Export G-code



*Untitled - PrusaSlicer-2.6.0 based on Slic3r

File Edit Window View Configuration Help

Plater Print Settings Filament Settings Printer Settings

Simple Advanced Expert

Print settings:
0.20mm QUALITY @MK3 - Copy
Filament: Generic PLA - SV-06
Printer: Sovol Octo * Sovol SV-06 - .4mm Nozzle
Supports: For support enforcers only
Infill: 15% Brim:

Name: table.stl Editing

Object manipulation: Position (X: 110, Y: 110, Z: 16.15 mm), Rotate (relative: 0, 0, 0 °), Scale factors (100, 100, 100 %), Size [World]: 24, 60, 32.31 mm, Inches

Sliced Info: Used filament (m) 1.85, Used Filament (mm³) 4451.43, Estimated printing time: 33m - normal mode, 33m - stealth mode

Export G-code

1) Now select the table if it isn't already.
It should turn from yellow to green.

2) We have some new tools now, select Paint-on supports

Paint-on supports [L]
Left mouse button: Enforce supports
Right mouse button: Block supports
Shift + Left mouse button: Remove selection
Automatic painting (highlighted)
Highlight overhang by angle: 0° Enforce Cancel
Tool type: Brush Smart fill
On overhangs only
Brush shape: Sphere Circle Triangles
Brush size: 2.00
Split triangles
Clipping of view: 0.00
Remove all selection

3) We could manually paint the places we think the object needs support, but instead lets let the Automatic Painting button handle it

*Untitled - PrusaSlicer-2.6.0 based on Slic3r

File Edit Window View Configuration Help

Plater Print Settings Filament Settings Printer Settings

Print settings:
0.20mm QUALITY @MK3 - Copy
Filament: Generic PLA - SV-06
Printer: Sovol Octo * Sovol SV-06 -.4mm Nozzle
Supports: For support enforcers only
Infill: 15% Brim: □
Name table.stl Editing
Paint-on supports

1) This is a bit messy. If we wanted we could manually paint it for better supports.

2) Make sure the Supports setting is set to "For support enforcers only."

3) Let's see what this support structure would look like when we slice it

Object manipulation
World coordinates X Y Z mm
Position: 110 110 16.15 mm
Rotate (relative): 0 0 0 °
Scale factors: 100 100 100 %
Size [World]: 24 60 32.31 mm
Inches
Sliced Info
Used Filament (m) 1.85
Used Filament (mm²) 4451.45
Estimated printing time:
- normal mode 33m
- stealth mode 33m
Slice now

*Untitled - PrusaSlicer-2.6.0 based on Slic3r

File Edit Window View Configuration Help

Printer Print Settings Filament Settings Printer Settings

Legend

Feature type

	Time	Percentage	Used filament
Perimeter	5m	4.8%	0.25 m 0.00 g
External perimeter	7m	6.6%	0.30 m 0.00 g
Overhang perimeter	14s	0.2%	0.01 m 0.00 g
Internal infill	2m	2.0%	0.09 m 0.00 g
Solid infill	8m	8.6%	0.94 m 0.00 g
Top solid infill	2m	1.6%	0.11 m 0.00 g
Bridge infill	2m	2.4%	0.12 m 0.00 g
Skirt/Brim	20s	0.3%	0.02 m 0.00 g
Support material	45m	45.9%	2.07 m 0.00 g
Support material interface	5m	5.2%	0.18 m 0.00 g
Custom	6s	0.1%	0.02 m 0.00 g

Estimated printing times (Normal mode):

First layer: 3m
Total: 1h39m
Show stealth mode

Print settings:

- Simple (Yellow)
- Advanced (Orange)
- Expert (Red)

Print settings:
0.20mm QUALITY @MK3 - Copy

Filament:
Generic PLA - SV-06

Printer:
Sovol Octo * Sovol SV-06 - .4mm Nozzle

Supports: For support enforcers only

Infill: 15% Brim:

Name: table.stl Editing
Paint-on supports

Object manipulation

World coordinates X Y Z mm
Position: 110 110 16.15 mm
Rotate (relative): 0 0 0 °
Scale factors: 100 100 100 %
Size [World]: 24 60 32.31 mm
 Inches

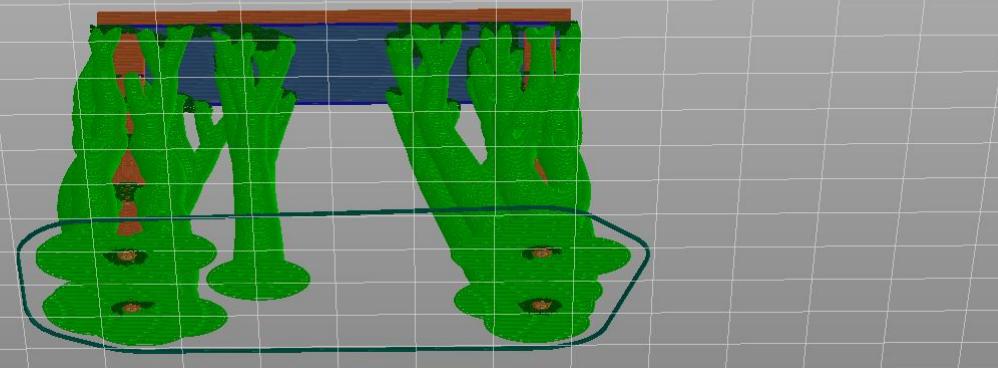
Sliced Info
Used Filament (m) 4.10
Used Filament (mm³) 9857.42
Estimated printing time:
- normal mode 1h39m
- stealth mode 1h40m

Export G-code

3D Editor View

2) Let's go back to the 3D Editor View and see if we can think of a better way.

1) Now it's even messier! Notice how we are going to use more filament and take three times as much time to print with supports. It's also much more likely to fail.



207501 298090

*Untitled - PrusaSlicer-2.6.0 based on Slic3r

File Edit Window View Configuration Help

Plater Print Settings Filament Settings Printer Settings

Print settings:
0.20mm QUALITY @MK3 - Copy
Filament: Generic PLA - SV-06
Printer: Sovol Octo * Sovol SV-06 - .4mm Nozzle
Supports: For support enforcers only
Infill: 15% Brim: □
Name: table.stl Editing
Paint-on supports

Object manipulation
World coordinates X Y Z mm
Position: 110 110 16.15 mm
Rotate (relative): 0 0 0 °
Scale factors: 100 100 100 %
Size [World]: 24 60 32.31 mm
Inches

Sliced Info
Used Filament (m) 4.10
Used Filament (mm³) 9857.42
Estimated printing time:
- normal mode 1h39m
- stealth mode 1h40m

Export G-code G

1) Let's try reorienting the table to avoid having to use supports.

2) We have a number of tools to do this. We could manually rotate the table with the Rotate tool

3) Alternatively, we could use the "Place on Face" tool to automatically rotate the object to one of its faces.

4) With the "Place on Face" tool selected, click on the top of the table to place that face against the bed of the printer.

5) Finally, with the "Place on Face" tool and table unselected, click Arrange to place the table optimally on the bed.

*Untitled - PrusaSlicer-2.6.0 based on Slic3r

File Edit Window View Configuration Help

Plater Print Settings Filament Settings Printer Settings

Print settings:
0.20mm QUALITY @MK3 - Copy
Filament: Generic PLA - SV-06
Printer: Sovol Octo * Sovol SV-06 - .4mm Nozzle
Supports: For support enforcers only
Infill: 15% Brim: □
Name: table.stl Editing
Paint-on supports [L]

Left mouse button: Enforce supports
Right mouse button: Block supports
Shift + Left mouse button: Remove selection

Automatic painting
Highlight overhang by angle: 0° Enforce Cancel

Tool type: Brush Smart fill
On overhangs only

Brush shape: Sphere Circle Triangles
Brush size: 2.00

Split triangles
Clipping of view: 0.00

Remove all selection

1) We still have these painted support points on the object. Let's get rid of them before we slice.

2) With the object and the Paint-on supports tool selected, click "Remove all selection."

3) Now that we've dealt with that, let's slice the object!

Object manipulation
World coordinates X Y Z mm
Position: 110 110 16.15 mm
Rotate (relative): 0 0 0 °
Scale factors: 100 100 100 %
Size [World]: 24 60 32.31 mm
Inches
Slice now

*Untitled - PrusaSlicer-2.6.0 based on Slic3r

File Edit Window View Configuration Help

Printer Settings Filament Settings Printer Settings

Legend

Feature type

	Time	Percentage	Used filament
Perimeter	5m	14.4%	0.25 m 0.00 g
External perimeter	7m	19.9%	0.31 m 0.00 g
Overhang perimeter	2s	0.1%	0.00 m 0.00 g
Internal infill	2m	6.0%	0.09 m 0.00 g
Solid infill	12m	34.2%	1.05 m 0.00 g
Top solid infill	2m	4.7%	0.11 m 0.00 g
Skirt/Brim	17s	0.9%	0.02 m 0.00 g
Custom	6s	0.3%	0.02 m 0.00 g

Estimated printing times:
First layer: 3m
Total: 34m

Print settings:
0.20mm QUALITY @MK3 - Copy

Filament:
Generic PLA - SV-06

Printer:
Sovol Octo * Sovol SV-06 - .4mm Nozzle

Supports: For support enforcers only

Infill: 15% Brim:

Name: table.stl Editing

Object manipulation

World coordinates X Y Z mm
Position: 110 110 16.15 mm
Rotate (relative): 0 0 0 °
Scale factors: 100 100 100 %
Size [World]: 24 60 32.31 mm
Inches

Sliced Info
Used Filament (m) 1.85
Used Filament (mm³) 4404.93
Estimated printing time:
- normal mode 34m
- stealth mode 34m

Export G-code

Simple Advanced Expert

1) That's much better. Notice how we have almost no overhang, so we don't need supports. We also have a large, flat surface against the print bed, which will make the print much more stable.

2) We've also cut down on both material used and print time with this better orientation.

*Untitled - PrusaSlicer-2.6.0 based on Slic3r

File Edit Window View Configuration Help

Printer Print Settings Filament Settings Printer Settings

Legend

Feature type

	Time	Percentage	Used filament
Perimeter	5m	14.4%	0.25 m 0.00g
External perimeter	7m	19.9%	0.31 m 0.00g
Overhang perimeter	2s	0.1%	0.00 m 0.00g
Internal infill	2m	6.0%	0.09 m 0.00g
Solid infill	12m	34.2%	1.05 m 0.00g
Top solid infill	2m	4.7%	0.11 m 0.00g
Skirt/Brim	17s	0.9%	0.02 m 0.00g
Custom	6s	0.3%	0.02 m 0.00g

Estimated printing times:
First layer: 3m
Total: 34m

Print settings:
0.20mm QUALITY @MK3 - Copy

Filament:
Generic PLA - SV-06

Printer:
Sovol Octo * Sovol SV-06 - .4mm Nozzle

Supports: For support enforcers only

Object manipulation

World coordinates

X	Y	Z
110	110	16.15

Position:

Rotate (relative):	0	0	0
Scale factors:	100	100	100

Size [World]:
24 60 32.31 mm

Sliced Info
Used Filament (m) 1.85
Used Filament (mm³) 4440.93
Estimated printing time:
- normal mode 34m
- stealth mode 34m

Export G-code

59088

Save G-code file as:

USB Drive (E)

Organize New folder

Name Date modified Type Size

No items match your search.

fa23-edwards-e1 3D Printing Tuto

Creative Cloud This PC Local Disk (C) USB Drive (E) Network

File name: table_0.2mm PLA MK3S_34m.gcode

Save as type: G-code files (*.gcode, *.gco, *.g, *.ngc)

Save Cancel

1) There are many ways to send GCode from the Slicer to the Printer. The most universal, which we will learn today, is using an SD card. Click here to open the saving window.

2) Make sure your SD card is selected here. If you have multiple drives or SD cards plugged in to the computer, make sure you pick the right option.

3) After you've chosen the right drive and given the GCode file the name you want, click save. It will now give you an option to eject the card directly from the Slicer, do so and insert your SD card into the printer.

Finally, on your printer select the file and print. The exact method will depend on your printer, but in most cases it will be Print from Media/Storage>File Name.

Make sure to keep an eye on your print. Failures often happen in the first layer, when it fails to adhere to print bed.



Let's Print from the Web!

- One of the easiest and coolest uses for the 3D printer is printing stuff that other people have made!
 - There is a huge database of free files that other people have modeled and uploaded for free, from modular hydroponics systems to concrete molds
- For the rest of class, let's explore some of the things we can make, and in doing so think about problems and solutions that come with printing different models
- Check out these sites and see if there is anything you want to print with the rest of our time:
 - www.thingiverse.com
 - www.printables.com