Converting Your 3D Model into A Printer File (Slicing)

Purpose: This document will walk you through how to convert your 3D model into the file format needed by the 3D printer through a process known as slicing. This is done with FlashPrint, a slicing software developed by the manufacturer of our 3D printers, FlashForge.

Step 1:

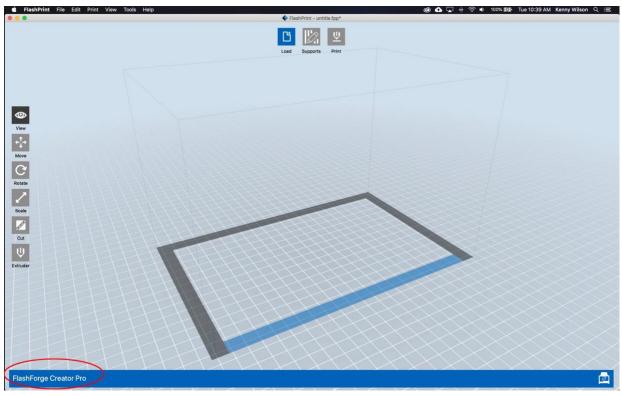
Open FlashPrint on the iMac machine located next to the 3D printers. The icon looks like this:



Step 2:

Confirm the printer is set to 'FlashForge Creator Pro' in the bottom left corner. If it is not defaulted on 'FlashForge Creator Pro', you can change it to Creator Pro by going to:

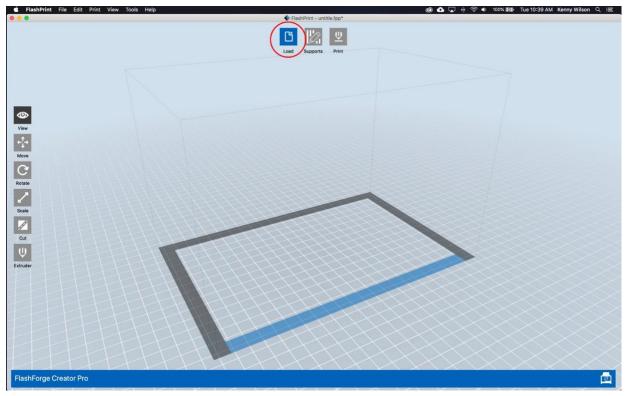
Print -> Machine Type -> Flashforge Creator Pro



Printer Confirmation

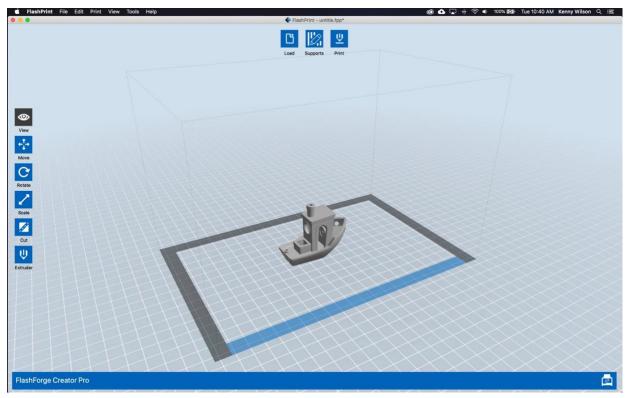
Step 3:

From the home screen, click on the button labeled 'load' located in the upper center of the window.



Load File

After you've imported a compatible 3D file, you should see your model on the print bed. The gird pattern represents the base of the print bed and the solid blue line indicates the front of the print space. The thin, grey lined box represents the total space, or volume, of the 3D printer.

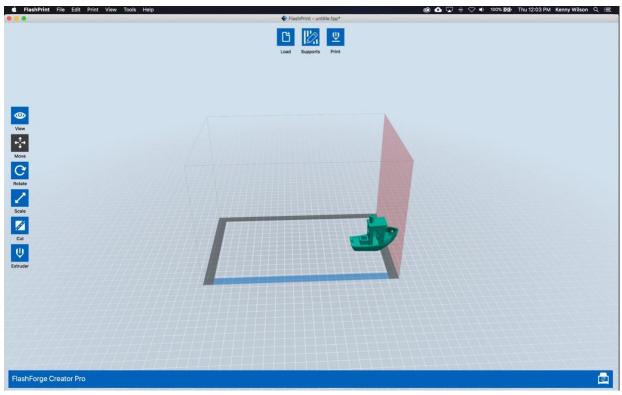


File Loaded

You cannot have models outside of this box. If you do have a model outside of this space, you'll see a red wall on the side of the box you are protruding out from. You will receive a warning if you attempt to continue to save the print file while having a model outside of this bounding box. Please do not attempt to continue printing when you have an item outside of the bounding box as it will cause damage to the 3D printer. A way to help correct this issue, especially when you have multiple models on the print bed, is to go to:

Edit -> Auto Layout All

You can select 'OK' when prompted with the spacing. If your model(s) are still protruding out of the bounding box after performing this, then you may have too many models packed into one print. Or your models may simply be too large and you'll need to scale them down to fit.



Model Out of Bounds

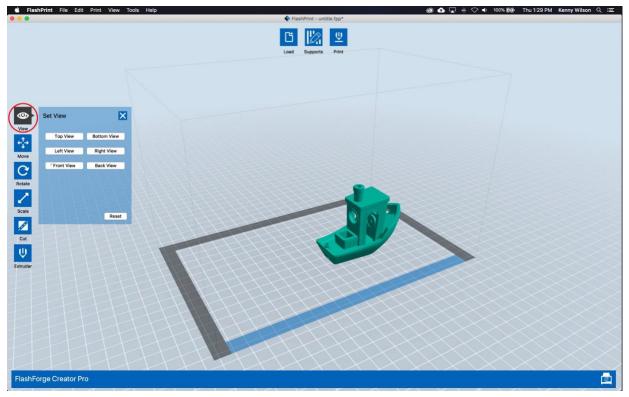
Step 4:

You can adjust the model if needed. Typically, the way your model is automatically placed into the print bed will be fine for printing, but some of the adjustments you can make include:

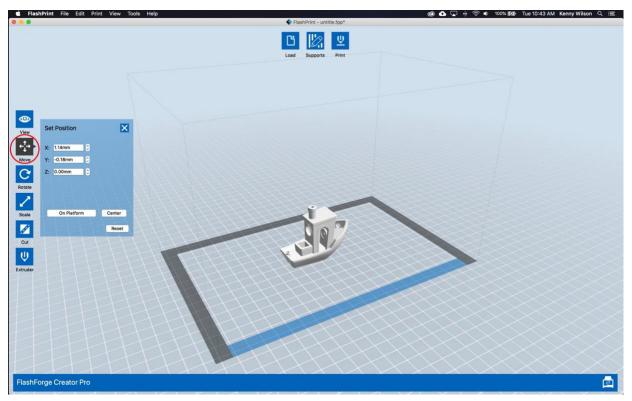
- View View allows you to change the view, or perspective, you're viewing the model and bounding box from. Additionally, pressing the right mouse button and dragging changes the view in free-form and pressing the left mouse button and dragging moves the ground plane up or down.
- Move Select to move the model to different locations on the print bed. Click on the model and move in the X and Y plane. To move along the z-axis, click the arrow next to the Z input box or enter a value
- Rotate This allows you to rotate your model amongst the X, Y, and Z planes. Once clicked you should see red, blue, and green circles that indicate the three planes. If you do not see this, make sure the check box next to 'Surface to Platform' is not checked. Click on one of the three circles and move your mouse to rotate the model. Additionally, you can enter values into the input boxes.
- Scale This allows you to adjust the scale, or size, of your model. Typically, you'll want to keep the 'Uniform Scaling' box selected. Click on the model and move your mouse to scale the model.
- Cut Cut allows you to cut your model into two or more pieces to fit within the bounding box. We advise not having a model size that require this. This is an advanced

feature. For more information about this, refer to the FlashPrint manual linked in the previous page.

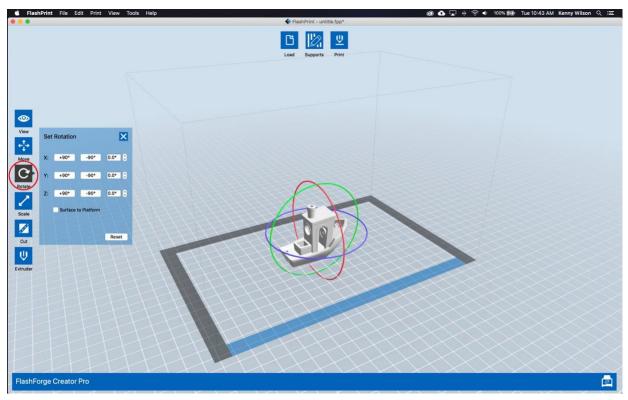
• Extruder – This allows you to choose which extruder, and filament, you want to use. It's defaulted on the right extruder. You can look at the color of filament loaded into the printer and decide which one you'd like by choosing left or right extruder.



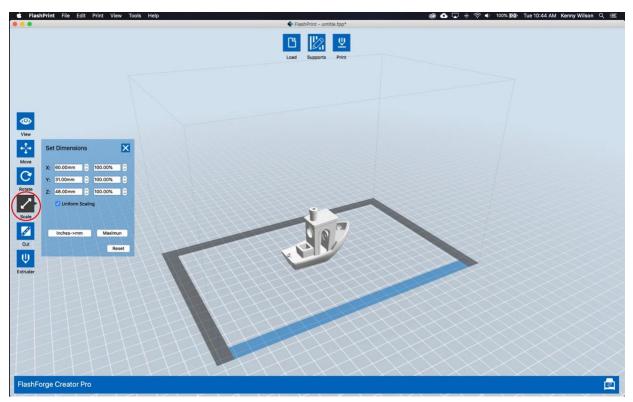
View Options



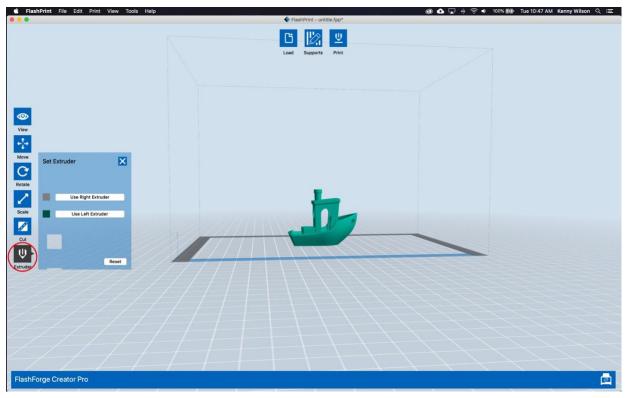
Move Options



Rotate Options



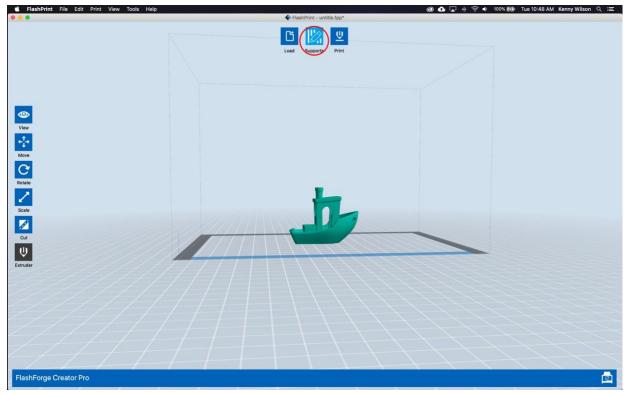
Scale Options



Choose Extruder

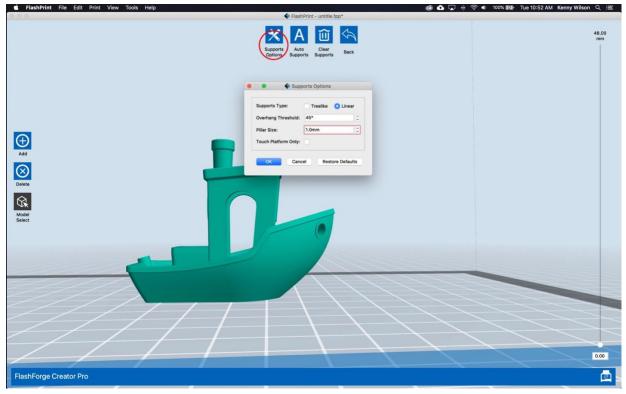
Step 5:

Next, you want to select the 'Supports' button in the upper center of the window.



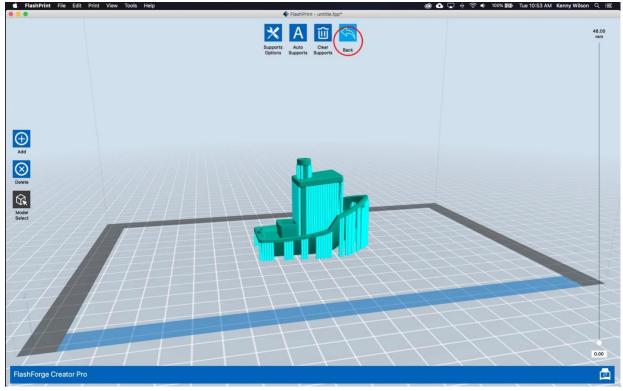
Supports

Then click 'Support Options'. You'll be presented with a dialog box that has a number of options to choose from. We recommend selecting the '**Linear**' support type. 'Overhand Threshold' refers to the angle at which supports will be added to the model. A typical range for this would be 20 to 45 degrees. More often than not, you can go with the defaults and then experiment with other options as you gain more experience. Click 'OK' when done.



Support Options

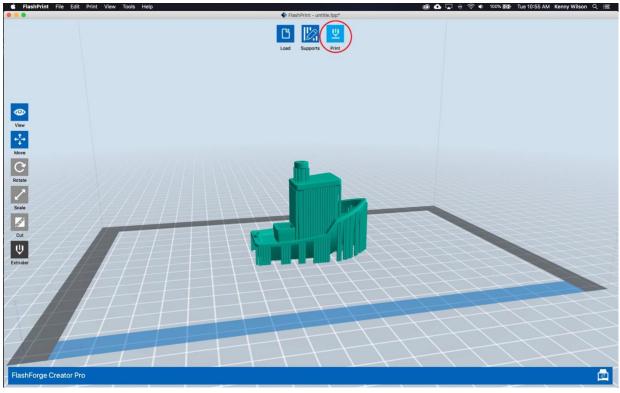
Next, click 'Auto Supports'. The software will automatically generate supports according to the options you selected. Finally, when you are happy with the support structure created, click on the 'Back' button to return to the previous FlashPrint options. You will be asked if you'd like to save the supports to the model at this point. You can click 'No'.



Exit Support Options

Step 6:

The next step is putting the final touches on the print file. Select 'Print' from the buttons in the upper center of the screen.



Print

There are again a number of very important options within this step so **please pay close attention to this step**. More than any other previous step, this step can be the difference between a successful print and a big disaster! Choose the following options in the top half of the dialog box:

- Machine Type FlashForge Creator Pro
- Material Right PLA
- Material Left PLA
- Supports Enable
- Raft Enable
- Resolution Standard

Once those options have been selected, click on the 'More Options' button.

🖆 FlashPrint File Edit Print View Tools Help		🛞 🚯 🗔 🕂 🔅 🔹 100% 🌆 Thu 2:09 PM Kenny Wilson 🔍 ≔
0.0.0	FlashPrint - untitle.fpp*	
	Lood Supports Print	
	Print	
Set View	I want to: Preview Machine Type: FlashForge Creator Pro	
View	Material Right: PLA	8
Left View Right View	Material Left: PLA	8
Nove Front View Back View	Supports: Enable	
C Front View Back View Rotate	Raft: Enable Resolution: Low (Faster) Wall	
Rotate	Standard Brim High (Slower)	
Scale	Hyper More Optiona >>	
Reset	Layer Height Shells Infill Speed Temperature Others	
Cut	Layer Height: D.18mm	
	First Layer Height: 0.27mm	
Extruder		
2////////	Restore Defaul	
///////////////////////////////////////	OK Cancel Save Configu	ration
<i>Z//////</i>		
//////		
FlashForge Creator Pro		

More Options

You again have a number of options to carefully select here in the different tabs. Moving from the left tab to the right tab we have:

- Layer Height Change Layer Height to 0.20 mm, leave First Layer Height as is
- Shells You can leave this at the default value which is determined by your resolution selection
- Infill Infill determines the amount of filament used to fill-in enclosed spaces in your model. The more the infill, the stronger the model and the more material it has. This setting can change with different models, needs, and applications; however, we recommend a fill density of 15-20%, a fill pattern of hexagon, and a combined infill every 2 layers as a standard choice
- Speed You can leave this at the default value which is determined by your
- Temperature Set the extruder temperature to 220 and the platform to 60
- Others You can leave this alone

FlashPrint File Edit Print View Tools Help	FlashPrint - untitle.fr		වි ৰ) 100% 📾 Tue 10:59 AM Kenny Wilson Q :Ξ
		U I	
		rint	
	🔿 💿 🔶 Print		
	I want to: Preview Machine Type: FlashForge Creator Pro	0	
⊘ View	Material Right: PLA	8	
View	Material Left: PLA	8	
All and a second	Supports: Enable	8	
	Raft: Enable Resolution: Low (Faster)	Wall	
Rotate	Standard High (Slower)	Brim	
	Hyper More Options >>		
Scale	Layer Height Shells Infill Speed	Temperature Others	
	Layer Height: 0.20mm	Thickness of each laver. The less thickness of lavers, t	e more time
Cut dis	First Layer Height: 0.27mm	n will be used and the better model will be printed.	
Extruder			
		Restore Defaults	
	OX Can	cel Save Configuration	
FlashForge Creator Pro			

Layer Height

🖆 FlashPrint File Edit Print View Tools Help		🛞 🚯 🔽 🖶 🗢 🔹 100% 🗐 Thu 2:34 PM Kenny Wilson Q :Ξ
100	FlashPrint - untitle.fpp*	
	Load Supports Print	
	O • Print	
	I want to: Preview Machine Type: FlashForge Creator Pro	
View	Material Right: PLA	
View •	Material Left: PLA	
Move	Supports: Enable	
C	Raft: Enable	
Rotate	Resolution: Low (Faster) Wall O Standard Brim	
	High (Slower) Hyper	
Scale	More Options >>	
Scale	Layer Height Shells Infill Speed Temperature Others	
Cut	Perimeter Shells: 2	
	Top Solid Layers: 3	
Extruder	Bottom Solid Layers: 3	
	Restore Defaults	
	OK Cancel Save Configuration	
		\times
		\times
MATTIN	FRAXXXX	
FlashForge Creator Pro		🗖 🗖
		$\nabla $ \times \times ∇ ∇ \times \times \times \times \times

Shells

FlashPrint File Edit Print View Tools Nelp	🛞 🚳 💭 🔶 📢 100% 🗐 Thu 2:35 PM Kenny Wilson Q 💷
Lad	Supports Print
	Print.
I want to: 🗸 Preview	
Machine Type: FlashForge Creator View Material Right: PLA	Pro 0
Wachie Type: PlashForge Creator View Machie Type: View Material Right: PLA Material Left: More Supports:	0)
Move Supports: Enable	•
Raft: Enable Resolution: Low (Faster)	
Rotate Scale Cut Cut Cut Cut Cut Cut Cut Cut Cut Cut	Brin
Scale More Options >>	I Speed Temperature Others
Cut Fill Density:	15%
U Fil Pattern:	Hexagon
Extruder Combine Infil:	Every 2 Layers C
	Restore Defaults
	OK Cancel Save Configuration
FlashForge Creator Pro	

Infill

FlashPrint File Edit Print View Tools Help	🔶 Flas	Print - untitle.fpp*	005	J ++ ◇ #0 100% (82) Thu 2:38	i Phill Kenny Wilson 🔍 📖
	Load	Supports Print			
		Print			
_ \ \	I want to: 🔍 Preview				
	Machine Type: FlashForge Creator Material Right: PLA	Pro	0		
	Material Left: PLA		0		
Move	Supports: Enable		0		
C	Raft: Enable		0		
Verv Verv Move Cotate Relate	Resolution: Low (Faster) • Standard High (Slower) Hyper	Wall Brim			
Scale	More Options >>				
	Layer Height Shells Infi Print Speed:	60mm/s			
	Travel Speed:	80mm/s			\approx
Extruder				$<\!\!<\!\!<\!\!<\!\!<\!\!<\!\!<\!\!<\!\!<\!\!<\!\!<\!\!<\!\!<\!\!$	>>>>
		Restore De	faults		
		OK Cancel Save Conf	figuration		>>>>
					\times
PETTTT	THE P				$\times\!\!\times\!\!\times$
	FTT	$\rightarrow \rightarrow $	\times	\times	\times
FlashForge Creator Pro					a
					$\sim \sim \sim \sim \sim$

Speed

K FlashPrint File Edit Print View Tools Help		🔕 🕼 🗔 🕂 🗢 🗢 100% Thu 2:38 PM Kenny Wilson Q 😑
	FlashPrint - untitle.fpp*	
	Load Supports Print	
	• • • • • • • • • • • • • • • • • • •	
	I want to: J Preview	
	Machine Type: FlashForge Creator Pro	
View	Material Right: PLA 0	
+++	Material Left: PLA 0	
Vew *** Move	Supports: Enable 0	
C	Raft: Enable 0	
Rotate	Resolution: Low (Faster) Wall	
	Standard Brim High (Slower) Hyper	
Scale	More Options >>	
	Layer Height Shells Infill Speed Temperature Others	
Cut	Left Extruder: 220°C	
	Platform: 60*C	
Extruder		
	Restore Defaults	
	OK Cancel Save Configuration	
		\times
MARKAM	X M X X X X X	
FlashForge Creator Pro		🖻 🔤

Temperature

Step 7:

The final step is to save the g-code file to the SD card. After making all your changes to the options, select the 'OK' button in the bottom of the dialog box.

Seat FlashPrint File Edit Print View Tools Help	🕲 🌰 🔽 🕂 🔿 🕸 100% 💯 Thu 2:38 PM Kenny Wi	ion Q 🖽
	FlashPrint - untitle.fpp*	
	Load Supports Print	
	•••• • • • • • • • • • • • • • • • • •	
	I want to: 🗸 Preview	
	Machine Type: FlashForge Creator Pro	
View	Material Right: PLA O	
View	Material Left: PLA 0	
Move	Supports: Enable 0	
C	Raft: Enable O	
Rotate	Resolution: Low (Faster) Wall Standard Brim	
	High (Slower)	
Scale	More Options >>	
	Layer Height Shells Infill Speed Temperature Others	~~~
Cut	Left Extruder: 220°C C	
	Platform: 60°C C	>>
Extruder		$<\!\!<\!\!<\!\!<$
	Restore Defaults	\leq
		>>
	OK Cancel Save Configuration	\sim
		\sim
		$\langle \rangle$
		\sim
MALTHAN	XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX	
FlashForge Creator Pro		

Save G-Code File To Disk

Another box will pop up and this is where you save your file. Make sure the SD card from the 3D printer is inserted into the iMac machine and that you save your file to this SD card. You can rename if you want but it is not necessary. Finally, click 'Save'. FlashPrint will take a few moments to generate your file and when finished, you'll have a file named 'YourFileName'.x3g on the SD card. Eject the SD card from the iMac and insert it back into the 3D printer.

Please ask the DLMC Media Mentors if you have questions or need help.