SeeMeCNC Guides

Setting Up Simplify 3D

This guide will walk you through the steps of setting up S3D for slicing and printing on your SeeMeCNC printer.

Written By: Jim Carter

Process Name: Proces				Lindete Desfile	Cause as Mana
Select Profile: Roston	ck Max v3 PETG (modified)		Auto-Configure for Pr	Update Profile	Save as New Remove
PLA		3 0 0	Medium		3 0
General Settings					
Infill Percentage: —				— 25% 🗌 Includ	ie Raft 🗌 Generate Support
Show Advanced	Select Models				OK Cancel

INTRODUCTION

This guide has been created by user Jim Carter. Many thanks to Jim and all of our users who share their knowledge!

Many people chose to use slicing software that is free, these programs work well, but other chose to use Simplify3D, which is more tunable. This guide will show settings used for the Rostock Max v3, and may work for other printers in the lineup.

Remember: Your Mileage May Vary

Disclaimer: SeeMeCNC does not provide technical support for third party software such as Simplify 3D

Step 1 — Where Are All The Options?

ocess Name:	Process1					
lect Profile:	Rostock Max v3 PETG (modified)			Update Profile	Save as New	Remove
uto-Configure fo	or Material		Auto-Configure for Pri	nt Quality		
PLA		00	Medium		•	00
eneral Settings						
Infill Percenta	ge:O			— 25% 🗌 Inclu	ude Raft 🛛 Gene	rate Suppor
Show Advar	rced Select Models				ОК	Cancel

 First step is to make sure that the ADVANCED SETTINGS are shown.

Step 2 — Extruder Setup

Select Profile: Rostock MAX v3 (modifie	ed)	😒 🛛 Update Profile	Save as New Remove	Select Profile: Rostock MAX v3 (modified)		Update Profile Save as New Remove
Auto-Configure for Material		Auto-Configure for Print Quality		Auto-Configure for Material	Auto-C	onfigure for Print Quality
PLA	0 0	Medium	◎ ● ●	PLA	🖸 🗢 Me	dium 💿 💿 🗢
General Settings				General Settings		
Infill Percentage:		10%	Include Raft Generate Support	Infill Percentage:		10% Include Raft Generate Support
	s Infill Support Temperatu		pts Other Advanced	Extruder Layer Additions Infil	Support Temperature	Cooling G-Code Scripts Other Advanced
Extruder List (click item to edit settings)	HE280 Toolhea	ad		Layer Settings		ayer Settings
HE280	Extruder Toolhead Index			Primary Extruder HE280	Firs	t Layer Height 90 0 %
				Primary Layer Height 0.2000 0 mm	Firs	t Layer Width 110 C %
	Nozzle Diameter 0.50			Top Solid Lavers 4	Firs	t Layer Speed 50 0 %
	Extrusion Multiplier 0.90	0			Start	Points
	Extrusion Width O Auto	O Manual 0.50 C mm		Bottom Solid Layers 4 0		Jse random start points for all perimeters
	Ooze Control			Outline/Perimeter Shells 3 0	<u> </u>	Optimize start points for fastest printing speed Choose start point closest to specific location
	Retraction Retracti	ion Distance 6.50 0 m	m	Outline Direction: O Inside-Out O Outsid	e-in	x: 0.0 C Y: 0.0 C mm
		estart Distance 0.00 0 m	m	Print islands sequentially without optimiza Single outline corkscrew printing mode (va		
		ion Vertical Lift 0.30 0 m				
		ion Speed 110.0 C m				
	Coast at End Coastin		m			
Add Extruder	Wipe Nozzle Wipe Di		m			
Remove Extruder	mpe Nozzle mpe bi	3.00				

- The next few images will show the settings that are used for PLA that have fixed issues for many people. Copy each of the values to your printer's profile.
- The Extruder tab assumes that your printer has the stock .5mm nozzle installed.
- Pay special attention to the **Retraction Vertical Lift** section.

Step 3 — Nice Lines!

		2	Update Profile	Save as New Ren
Auto-Configure for Material		Auto-Configure for P	rint Quality	
PLA	00	Medium		0
General Settings				
Infill Percentage:			10% 🗌 Incl	ude Raft 🛛 🗌 Generate Su
Extruder Laver Additions Infill St	upport Tempera	ture Cooling	G-Code Scripts	Other Advanced
Layer Settings		First Layer Setting		
Primary Extruder HE280		First Layer Heig	ght 90 0 %	
Primary Layer Height 0.2000 0 mm		First Layer Wid	th 110 C %	
Top Solid Layers 4		First Layer Spe	ed 50 🗘 %	
Bottom Solid Layers 4 0		Start Points		
Outline/Perimeter Shells 3			start points for all points for all points for fastest	
Outline Direction: O Inside-Out Outside-In		Choose star	t point closest to spe	ecific location
Print islands sequentially without optimization		X: 0.0	0 Y: 0.0	0 mm
Single outline corkscrew printing mode (vase n	node)			

 On the Layer tab, pay attention to the settings for the first layer. The amount of top, bottom layers, along with the perimeters may vary depending on the print, as well as the layer height.

Step 4 — Skirt? Brim? Raft???

Select Profile: Rostock	MAX v3 (modified)				C Upr	date Profile	Save as Ne	ew Remove
Auto-Configure for Material					Auto-Configure for Print Qu	ality		
PLA			0	•	Medium			00
General Settings								
Infill Percentage:	-0				1	0% 🗌 Include I	Raft	Generate Suppo
Extruder I	ayer Additions	Infill	Suppo	rt Tempe	rature Cooling G-Co	de Scripts C	ther A	dvanced
	Use Skirt/Brim				🗆 Use Prime Pillar			
s	kirt Extruder	HE280			Prime Pillar Extruder	All Extruders	0	
s	ikirt Layers	1	0		Pillar Width	12.00 0 mm		
s	kirt Offset from Part	4.00	0 1	mm	Pillar Location	North-West	0	
5	kirt Outlines	6	0		Speed Multiplier	100 🔅 %		
	Use Raft				Use Ooze Shield			
	Raft Extruder	HE280		c	Ooze Shield Extruder	All Extruders	0	
	Raft Layers	3	0		Offset from Part	2.00 C mm	1	
	Raft Offset from Part	3.00	C r	nm	Ooze Shield Outlines	1. 0		
	Separation Distance	0.14	C r		Sidewall Shape	Waterfall	0	
	Raft Infill	85	0 9		Sidewall Angle Chang	e 30 0 deg	,	
	Disable raft base la				Speed Multiplier	100 0 %		
Hide Advanced	Select Models							Cance

 Under the Additions tab, the only settings to change are to the Skirt/Brim list. Printing with a raft on these machines seems counterproductive, and a waste of material. Since there is no purge line prior to the start of the print, set the number of skirt outlines to a minimum of 6. This gives the nozzle enough time to be primed and ready to go.

Step 5 — Density

	Rostock MAX v3 (modified	i)		Update Profile Save as New Remove		
Auto-Configure for	Material		Auto	o-Configure for Print Quality		
PLA		0	•	Medium		0
General Settings						
Infill Percentage	= ——•			10%	Include Raft	Generate Supp
Extru	ler Layer Additions	Infill Support	-	0. To 0.0.1	Scripts Other	Advanced
Extru		Support	Temperature	Cooling G-Code	Scripts Other	Advanced
	General			Infill Angle Offsets		
	Infill Extruder HE280			0 C deg	45	
	Internal Fill Pattern	tectilinear		Add Angle		
	External Fill Pattern	tectilinear	2	Remove Angle		
	Interior Fill Percentage	10 0 %				
	Outline Overlap	15 0 %				
	Infill Extrusion Width	150 0 %				
	Minimum Infill Length	5.00 0 mm				
	Print Sparse Infill Every	1 Clayers				
	Include solid diaphr	agm every 20 0	layers	Print every infill ang	la an asah launa	
				Print every min ang	e on each layer	

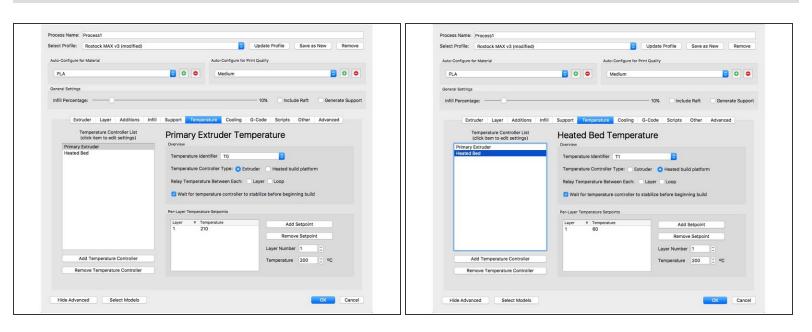
• The Infill tab is next. The % of infill used for your print will vary from print to print, but the other options below it are important.

Step 6 — Lean On Me

Select Profile: R	ostock MAX v3 (modified)		Update Profile Save as New Remove
Auto-Configure for N	faterial		Auto-Configure for Print Quality
PLA		00	Medium 💿 💿 🧲
General Settings			
Infill Percentage:	-0		10% Include Raft Generate Supp
Extrude	er Laver Additions	Infill Support Tempe	rature Cooling G-Code Scripts Other Advanced
Extruct		initii Support Tempe	(b) An example of the horizon from the formation of the formation for the horizon of the horizon of the formation of the horizon of the ho
	Support Material Generation	torial	Automatic Placement Only used if manual support is not defined
			Support Type Normal C
	Support Extruder HE28	80 0	
	Support Infill Percentage	30 0 %	Support Pillar Resolution 4.00 0 mm
	Extra Inflation Distance	0.00 0 mm	Max Overhang Angle 45 0 deg
	Dense Support Layers	3 0	Support Infill Angles
	Dense Infill Percentage	70 0 %	0 0 deg 45
	Print Support Every	1 C layers	-45 Add Angle
	Separation From Part		Remove Angle
	Horizontal Offset From P	Part 0.30 0 mm	
	Upper Vertical Separatio	n Layers 1 0	
	Lower Vertical Separation	n Layers 1 0	

 Support!!!! This section is very important when using support for your print where applicable. In this screenshot the support is turned off, but copying each setting value shown will ensure a great support structure, that produces great overhang/bridging layers. Pay close attention to the Support Infill Angles section.

Step 7 — Temps

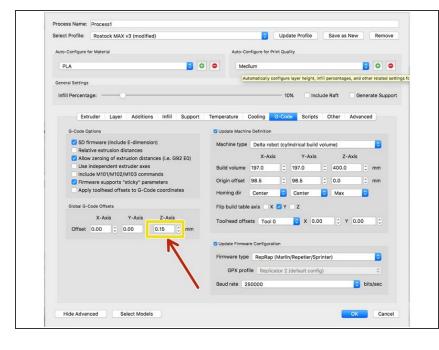


- Located under the Temperature tab, you will find settings for the extruder, and heated bed.
- Different types of PLA run different temps. Try starting at 210c, and work your way up or down from there depending on your material.
- Setting the heated bed to 60c will yield great adhesion results when coupled with an aerosol hairspray of your choice.

Step 8 — Is It Windy In Here?

Process Name: Select Profile:	Rostock MAX v3	tere altitle de	O Update Profile Save as New Remo
Select Profile:	ROSTOCK MAX V3	modified)	opdate Prome Save as New Remo
Auto-Configure fo	or Material		Auto-Configure for Print Quality
PLA		00	Medium 💿 💿
General Settings			
Infill Percenta	ge:		10% Include Raft Generate Sup
Extr	uder Layer A	dditions Infill Support Tempera	ature Cooling G-Code Scripts Other Advanced
Per-Layer	Fan Controls		Speed Overrides
Laye v	Fan Speed	Add Setpoint	Adjust printing speed for layers below 15.0 C sec
1	1 0 2 60 3 100	Remove Setpoint	
			Allow speed reductions down to 20 0 %
		Layer Number 1	Fan Overrides
		Fan Speed 60 0 %	
			Increase fan speed for layers below 60.0 0 sec
			Maximum cooling fan speed 100 0 %
			Bridging fan speed override 100 0 %
Fan Option			
🗹 Blip f	an to full power wh	en increasing from idle	

- The cooling settings for PLA are important. This printer comes with an amazing part fan setup that is tried and true. There is NO need to print ducts to redirect the air flow, or relocate the fans, or even change them out. The are designed to create a flow of air that hits perfectly every time.
- So if you have modified your cooling... CHANGE IT BACK TO STOCK!
- The 3 stage cooling setup shown works for almost every print.



Step 9 — GCode???

- The tab labeled GCode should really have a different title, but follow these settings for the v3.
- The Z-Axis Offset will vary for everyone. If after running a full calibration on the printer before your first print with these settings you find that the nozzle is a bit too close to the bed, start low and work your way up .01mm at a time in the called out section.

Step 10 — I Forgot My Lines...

Select Profile: Rostock MAX v3 (modified)	Update Profile Save as New Remove	Select Profile: Rostock MAX v3 (modified)	Update Profile Save as New Remove
Auto-Configure for Material	Auto-Configure for Print Quality	Auto-Configure for Material	Auto-Configure for Print Quality
PLA O	Medium O O	PLA	Image: Medium Image: Medium
General Settings		General Settings	
Infill Percentage: O	10% Include Raft Generate Support	Infill Percentage:	10% Include Raft Generate Support
Starting Script Layer Change Script	Retraction Script Tool Change Script Ending Script	M106 S0 ; turn off cooling fan M104 S0 ; turn off extruder M140 S0 ; turn off bed G28 ; home axes	ange Script Retraction Script Tool Change Script Ending Script
		M84 ; disable motors	
Post Processing Export file format Standard G-Code (.gcode)	8	M84 ; disable motors Post Processing Export file format Standard G-Code (.gcode	a) E
-) Star Wars C	Post Processing	
Export file format Standard G-Code (.gcode)) Star Wars 0	Post Processing Export file format Standard G-Code (.goode	3g files only) Star Wars 0
Export file format Standard G-Code (.gcode) Add celebration at end of build (for .x3g files only)) Star Wars C	Post Processing Export file format Standard G-Code (good Add celebration at end of build (for .x	3g files only) Star Wars 0

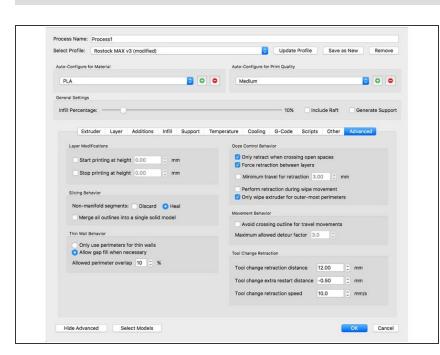
 Under the Scripts tab, make sure that the following lines of code are present for the starting and ending script tabs shown.

Step 11 — The Others?

Select Profile: Rostock MAX v3 (mod	ified)	CUpdate Profile Save as New Remove
Auto-Configure for Material		Auto-Configure for Print Quality
PLA	6 0	Medium 💿 💿
General Settings		
Infill Percentage:		10% Include Raft Generate Suppo
Extruder Layer Additi	ons Infill Support Temp	perature Cooling G-Code Scripts Other Advanced
Speeds		Filament Properties
Default Printing Speed	60.0 0 mm/s	Filament diameter 1.7500 🗘 mm
Outline Underspeed	50 0 %	Filament price 15.00 C price/kg
Solid Infill Underspeed	80 0 %	Filament density 1.25 C grams/cm*3
Support Structure Underspeed	80 0 %	Bridging
X/Y Axis Movement Speed	150.0 C mm/s	Unsupported area threshold 50.0 C sq mm
Z Axis Movement Speed	60.0 0 mm/s	Extra inflation distance 0.00 0 mm
Dimensional Adjustments		Bridging extrusion multiplier 95 0 %
Horizontal size compensation	0.00 0 mm	Bridging speed multiplier 70 C %

- When navigating through the Other tab, print speeds may vary for some prints, but 60mm/s works very well.
 Sometimes when more detail is needed, 40mm/s works great.
- Make sure in your general options for the program your Speed Display Units are set to mm/s and not mm/min. As 60mm/min is really, really slow. These settings can be found in the preferences for S3D

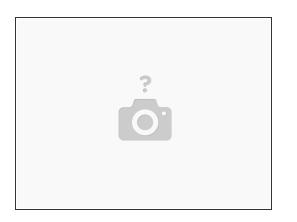
 Then just copy all the rest of the values. The \$\$\$ and weight of the filament affects 0% of the printing.



Step 12 — Head Of The Class

• The Advanced tab is last. Copy these settings as well.

Step 13 — Slice & Hit Print!



• That's all for this guide. Give these settings a shot and see how they work for you. We'd love to hear your feedback.