Allevi Guide to Repetier-Host

Setting up Repetier-host

- 1. Install <u>Repetier-Host</u>.
 - **a.** Close window if Repetier-Host Server opens in your browser and continue.
- 2. Open Repetier-Host. Go to 'Config' > 'Printer Settings' and assign the following values in the appropriate sections.

Printer Settings					
Printer:	default			•	â
Connection Printe	er Extruder Printer Shap	eScrip	ts Advanced		
Number of Ex	truder:	2	\$		
Number of Fa	ns:	0	•		
Max. Extrude	r Temperature:	160			
Max. Bed Ten	nperature:	30			
Max. Volume per second		12	[mm³/s]		
Printer has	s a Mixing Extruder (one	nozzle	for all colors)		
Extruder 1					
Name:					
Diameter:	0.4	[mm]	Temperature Offset:	0	[°C]
Color:					
Offset X:	0		Offset Y:	0	[mm]
Extruder 2					
Name:					
Diameter:	0.4	[mm]	Temperature Offset:	0	[°C]
Color:					
Offset X:	0		Offset Y:	0	[mm]
Printer Settings					
Printer:	default			· 🔒	
Connection Printe	r Extruder Printer Shape	Scripts	Advanced		
Printer Type:	Rostock Printer (circu	ılar prin	t shape)		
Home X: 0	Home Y:	0	Home Z:	0	
Printable Radius:	45		mm		
Printable Height:	60		mm		

Note: For 'Number of Extruder', enter the number of extruders on your printer.

- **3.** Go to 'Config' > 'Units of imported Objects' and select millimeters.
- 4. Download the Slic3r Config Bundle file.
- 5. In Repetier Host, go to the Slicer tab and click "Configuration."



6. Go to "File" > "Load Config Bundle..." and select the Slic3r Config Bundle file you just downloaded.



Slicing your STL

You can design STL files on any CAD software, such as SolidWorks or Autodesk Fusion 360. Alternatively, you can find a host of STL files in the "Getting Started" folder. To begin we will use a predesigned STL file.

- 1. Load the <u>Pediatric-Bronchi.STL</u> file into Repetier-Host.
 - a. Move the object on the bedplate using the arrows highlighted on the left.
 - **b.** Save, copy, or scale object with the items on the right.
 - **c.** Assign the STL to an extruder by using with the extruder drop down menu.



- 2. Enter your print settings.
 - **a.** Select the "Slicer" tab to assign preconfigured Slic3r settings to slice the file.
 - **b.** Speed and layer height are defined.

🛞 Repetier-Host V2.1.3 - Pediatric-Bronchi.stl			- 🛛 ×	
File View Config Printer Tools Help				
Connect Load Log	_	Printer Settings	Easy Mode Emergency Stop	
3D View Temperature Curve	Object Placement Slice	er rint Preview Manual Control SD Car	d	
	Slice with Slic3r			
	Slicer: Slic3r		· 🗇 Manager	
			Configuration	
	Print Setting:	Allevi_0.2mm_6mms_0infill		
	Printer Settings:	Allevi_0.2mm	· ·	
	Filament Settings:			
	Extruder 1:	- Allevi -		
	Extruder 2:	- Allevi -	•	
	Extruder 3:	- Allevi -		
	Override Silc3r S C C C C C C C C C C C C C C C C C C	ettings Copy Print Settings to Override	20%	

- **3.** Select the configuration tab to adjust settings important for bioprinting. Print, filament, and printer settings are accessible under 'Settings.'
 - **a.** Layer Height Determines how thick or thin your layers are

evi2_0.1_6mms 🗸 🔚	Layer height			
Layers and perimeters	Layer height:	0.1	mm	
Skirt and brim	First layer height:	0.1	mm or %	
Support material	Use adaptive slicing:			
Speed	Adaptive quality:		75 %	
Advanced	Match horizontal surfaces:			
Output options				
Shortcuts	Vertical shells			
	Perimeters:	3	(minimum)	
	Spiral vase:			
	Horizontal shells			

b. Speed – Determines how fast the extruder will move during the print. Remember to change all the speeds in the column to the desired number

evi2_0.1_bmms V	Speed for print moves				
Layers and perimeters	Perimeters:	6	✓ mm/s		
Infill Skirt and brim	⊾ small:	6	✓ mm/s or %		
Support material	6 external:	6	✓ mm/s or %		
Speed	Infill:	6	✓ mm/s		
Multiple extruders	4 solid:	6	✓ mm/s or %		
Advanced Output options	⊾ top solid:	6	✓ mm/s or %		
Notes	4 gaps:	6	✓ mm/s or %		
Shortcuts	Bridges:	6	∽ mm/s		
	Support material:	6	✓ mm/s		
	6 interface:	100%	mm/s or %		
	Speed for non-print moves	100.76	✓ mm/s or 76		
	Travel	6	mm/s		

Recommendation: 2 mm/s is good for a small accurate design (example: lattice). 8-16 mm/s is good for a large design (example: ear). **Speed is a print parameter that is very dependent on material and pressure.**

c. Nozzle Diameter – Enter the number that accurately depicts your nozzle tip inner diameter

Plater Print Settings Plinte	r Settings 🗶					
allevi2_0.1mm V	Size Size					
General	Nozzle diameter:	0.15	mm			
Fxtruder 1	Limits					
W Extruder 2	Min:	0.1	mm			
	Max:	0.3	mm			
	Position (for multi-extrude	r printers)				
	Extruder offset:	x: 0 y: 0	mm			
	Retraction					
	Length:	0	mm (ze	ero to disable)		
	Lift Z:	0	mm			
	Only lift Z:	Above Z: 0		mm Below Z: 0	mm	
	Speed	40	mm/s			

NOTE: Your layer height must be less than your nozzle diameter to correctly slice NOTE: Once the changes have been made, remember to save and label your settings.

4. Once your settings have been saved, you are now ready to slice the file.

Slice with Slic3r

- 5. You can view the printer movement by viewing the g-code created by your slicer file. Note: If no g-code is generated, check that your nozzle diameter is thin enough.
 - **a.** Save your g-code file.



How does your gcode look? You can compare with our presliced file!