

# Table of Contents



<b>Cura Settings</b>			
<b>Option</b>	<b>Value</b>	<b>Unit</b>	<b>Description</b>
<b>Quality</b>			
Layer Height	0.2	mm	
<b>Shell</b>			
<b>Infill</b>			
Infill Density	25	%	
<b>Material</b>			
Printing Temperature	230	°C	PETG
Build Plate Temperature	85	°C	
Enable Retraction	true		
Retract at Layer Change	true		
Retraction Distance	7	mm	Test: 2mm
Retraction Speed	70	mm/s	Test: 50 mm/s
<b>Speed</b>			
Print Speed	60	mm/s	Details ⇒ Slower
Infill Speed	50	mm/s	
Wall Speed	30	mm/s	
Outer Wall Speed	30	mm/s	
Inner Wall Speed	60	mm/s	
Travel Speed	80	mm/s	Faster/Slower? D:150
Initial Layer Speed	20	mm/s	More holes ⇒ Slower
<b>Travel</b>			
Combing Mode	Not in Skin		
Retract Before Outer Wall	true		
Avoid Printed Parts When Travel	true		
Avoid Supports When Traveling	true		
Z Hop When Retracted	true		
Z Hop Height	0.075	mm	
<b>Cooling</b>			
Enable Print Cooling	true		
Fan Speed	100	%	100 PLA
Initial Fan Speed	0	%	
Regular Fan Speed at Layer	3		
<b>Support</b>			
Generate Support	false		
Support Placement	Touching Build		
Support Overhang Angle	60	°	
<b>Build Plate Adhesion</b>			
Build Plate Adhesion Type	Skirt		
<b>Experimental</b>			
Tree Support	true		
Tree Support Branch Angle	40	°	
Tree Support Branch Distance	1	mm	

From:

<https://wiki.goatpron.de/> - **GoatPr[0o]n**

Permanent link:

[https://wiki.goatpron.de/projects:anycubic\\_i3\\_mega:cura\\_settings?rev=1553021699](https://wiki.goatpron.de/projects:anycubic_i3_mega:cura_settings?rev=1553021699)

Last update: **2019/03/19 18:54**

