

Phantom Plastic Press v1 Printing Parameters (etching gearbox update)

Slicer: Cura 5.6.0

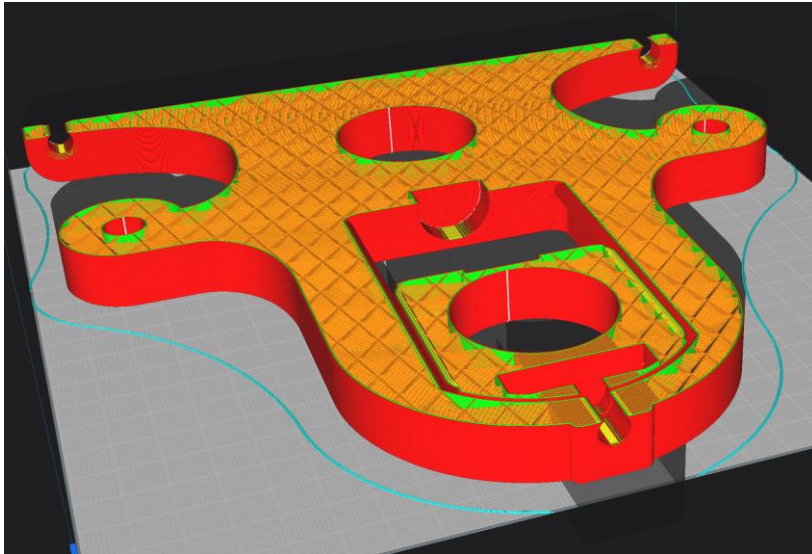
My 3d Printer: Anycubic Kobra 2

Filament: SunLu PLA +

Nozzle: 0.4

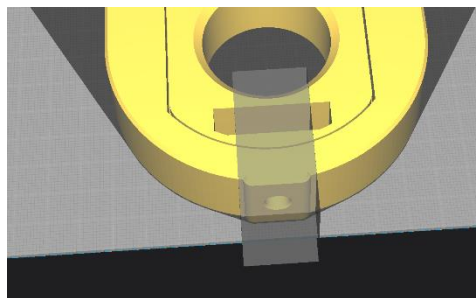
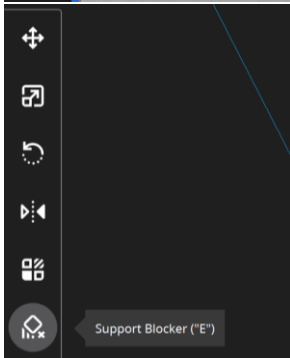
A lot of the other settings like speed and temperature will probably depend on your individual printers. You might have to test your tolerances and fit. I usually print 0.6mm lines with my 0.4mm nozzle for speed and better adhesion. Wall thickness noted below are the MINIMUM wall thickness. You can always use more walls or infill if you are worried about the strength of your parts.

Press body w/ Micro registration (Print 2)



Layer Height: 0.2
Wall thickness: 1.2mm
Bottom Layers: 5
Top Layers: 5
Infill type: cubic
Infill Density: 15%
Support: No

You can print either side up. Printing the side with the ruler marks facing up will make them easier to see. The spot where the threaded rod meets the bearing block is the weakest link in this press so we will print this spot with 100% infill. Follow steps below to see how to do this in Cura.



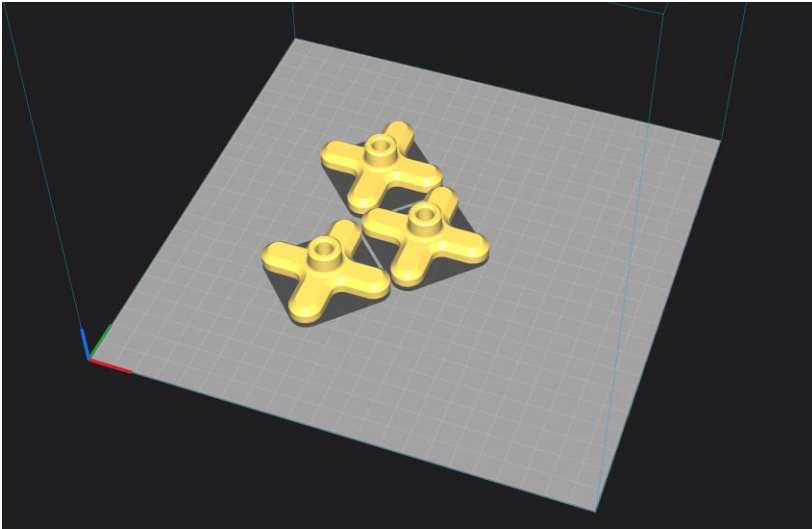
Click on the press body, and on the left hand side of Cura add a "support blocker" in the area where the threaded rod meets the bearing block.

Resize the support blocker so that it encompasses the top of the roller coupler and the hole where the threaded rod passes through.

Click "Per Model Settings" on the left hand toolbar and select "Modify settings for overlaps." Click on "Select settings" look up "infill" and check "Infill Density" and "Infill Pattern". Set these to 100% and Lines.



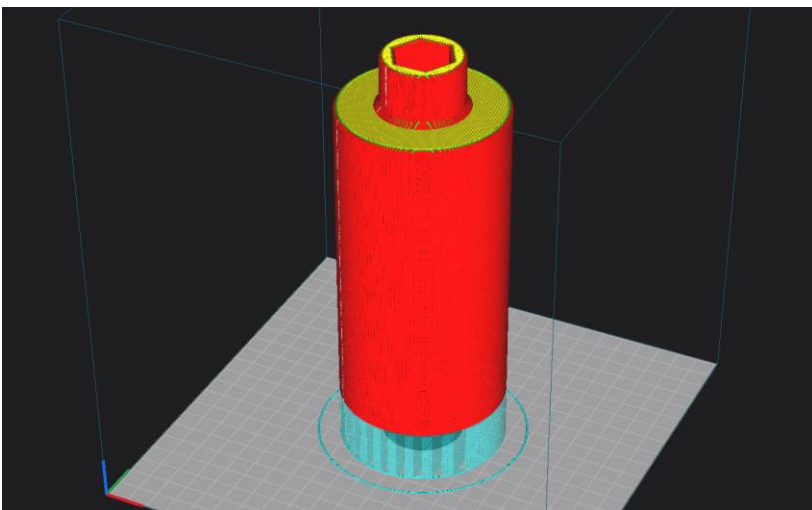
Pressure Knob (Print 3)



Layer Height: 0.2
Wall thickness: 1.2mm
Bottom Layers: 5
Top Layers: 5
Infill type: cubic
Infill Density: 15%
Support: No

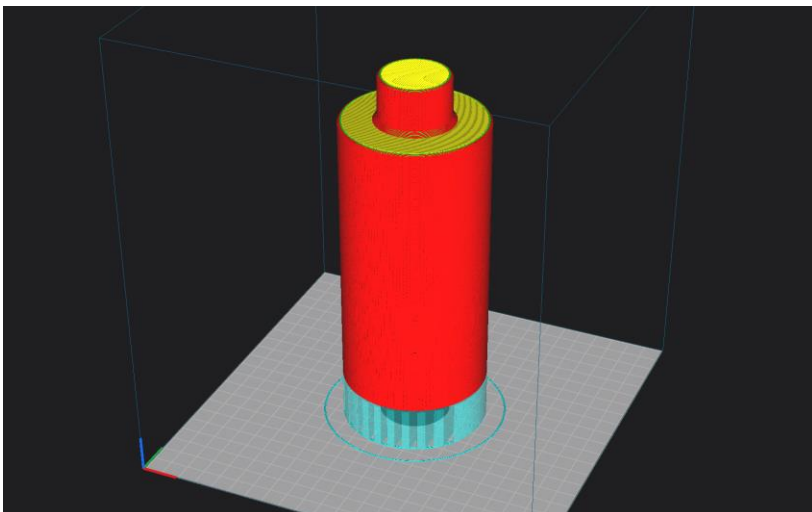
Print 3 of these, the 3rd one will help you attach the other 2

Drive Roller (bottom)



Layer Height: 0.2
Wall thickness: 1.8mm
Bottom Layers: 8
Top Layers: 5
Infill type: cubic
Infill Density: 15%
Support: YES

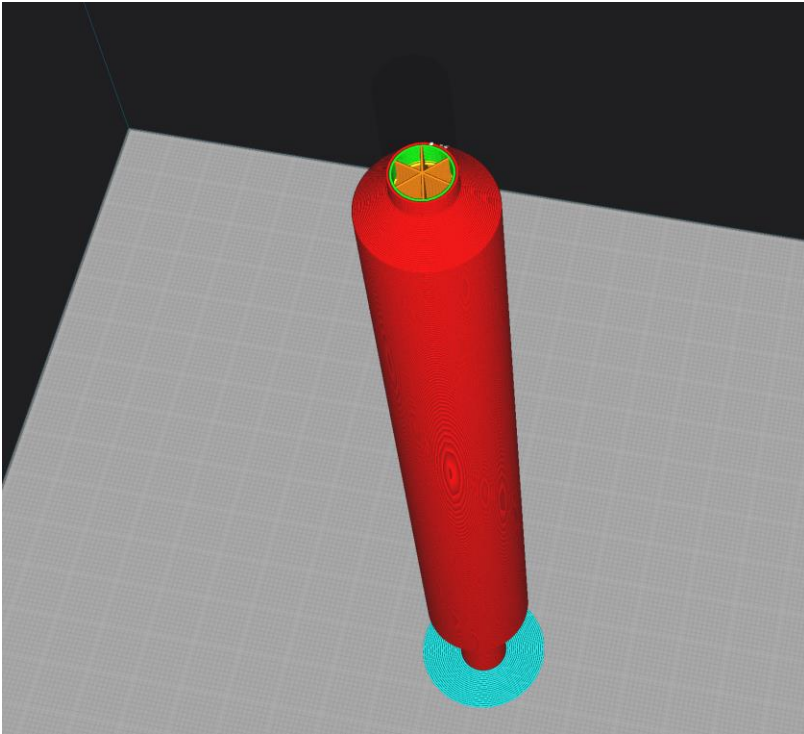
Idle Roller (Top)



Layer Height: 0.2
Wall thickness: 1.8mm
Bottom Layers: 8
Top Layers: 5
Infill type: cubic
Infill Density: 15%
Support: YES

Print this or another one with the hex hole. Your choice.

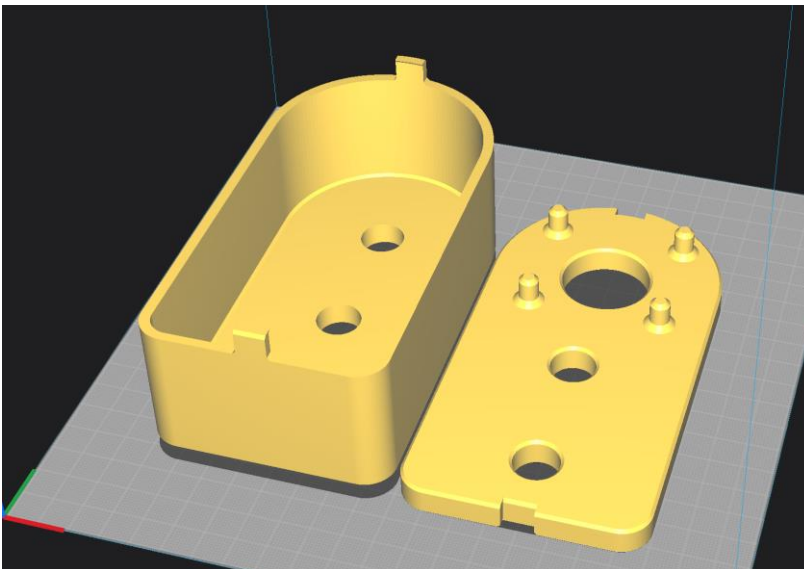
Press Bed Support (Print 2)



Layer Height: 0.2
Wall thickness: 1.2mm
Bottom Layers: 5
Top Layers: 5
Infill type: triangle
Infill Density: 15%
Support: no
Build plate adhesion: Brim

Triangle Infill is very important! Cubic doesn't support the thin parts well enough. Print with a Brim and 1 at a time if your build plate adhesion isn't as great.

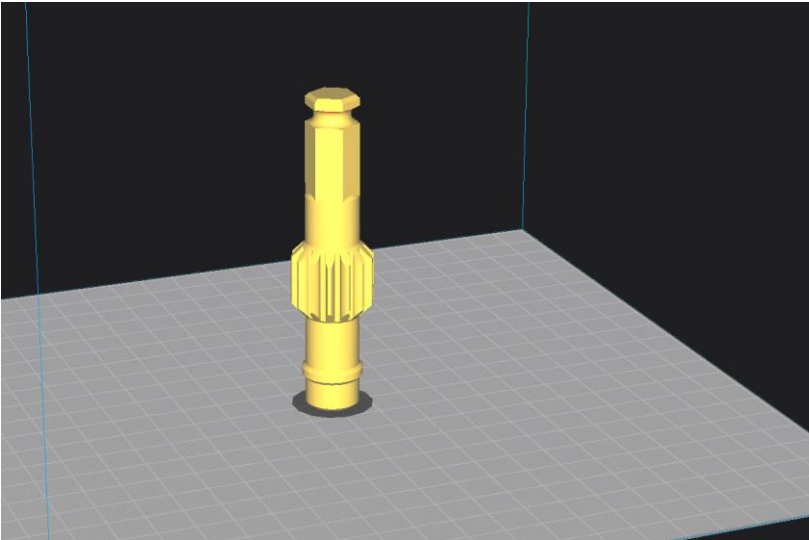
Gearbox cover



Layer Height: 0.2
Wall thickness: 1.2mm
Bottom Layers: 4
Top Layers: 4
Infill type: cubic
Infill Density: 15%
Support: no

I usually print the mounting plate and gearbox cover at the same time. These do not need to be printed super tough.

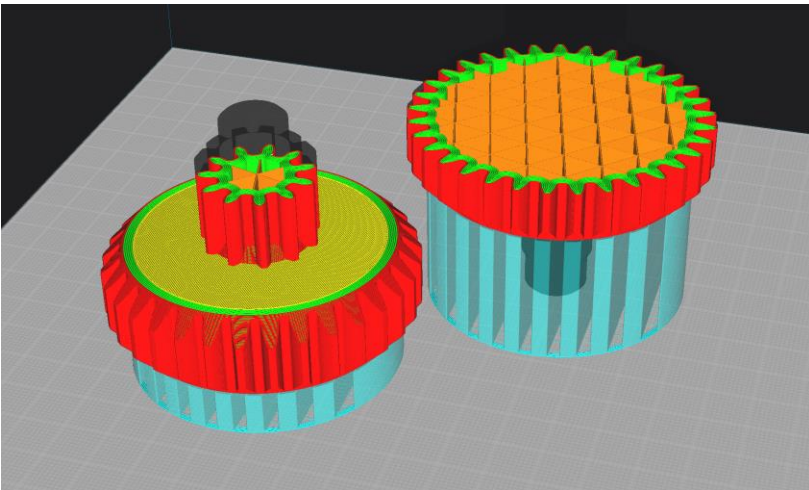
Crank Gear



Layer Height: 0.2
Wall thickness: 1.2mm
Alternate extra wall: Yes
Bottom Layers: 4
Top Layers: 4
Infill type: lines
Infill Density: 100%
Support: No
Infill Before Walls: No
Build plate adhesion: Brim

Print this solid, as it is most likely to shear when under high pressure or when the press is dropped or bumped.

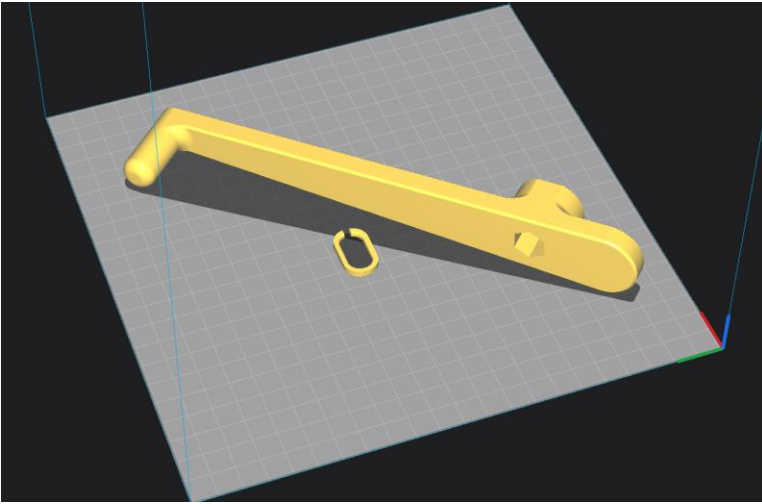
Middle gear and hex gear



Layer Height: 0.2
Wall thickness: 2 mm
Alternate extra wall: Yes
Bottom Layers: 5
Top Layers: 5
Infill type: Triangle
Infill Density: 15%
Support: yes
Infill Before Walls: No

Use enough wall thickness until the gear's teeth are solid. This is especially important if you are planning to print etchings.

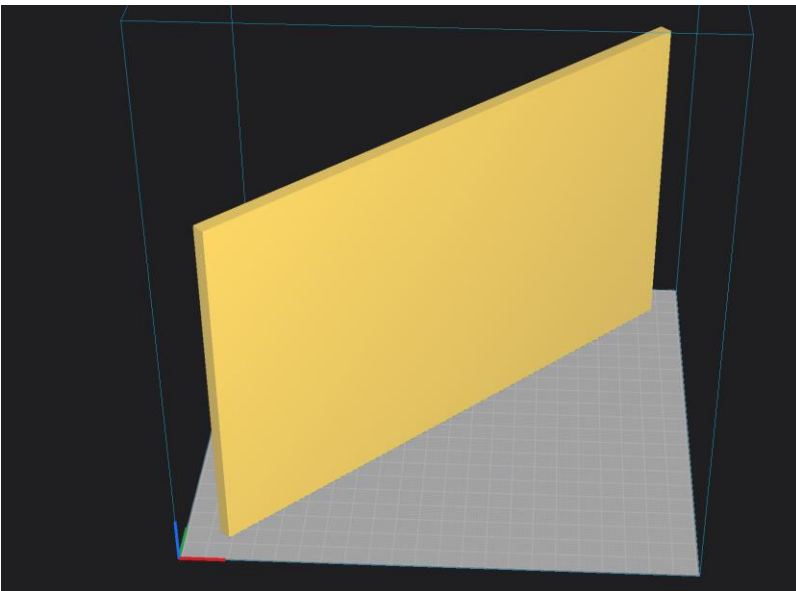
Crank Handle and lock pin



Layer Height: 0.2 (.12 looks better but optional)
Wall thickness: 1.2mm
Alternate extra wall: Yes
Bottom Layers: 4
Top Layers: 4
Infill type: cubic
Infill Density: 15%
Support: No
Infill Before Walls: No
Build plate adhesion: brim if you are having adhesion issues

Very sturdy and practical. Print it on its side.

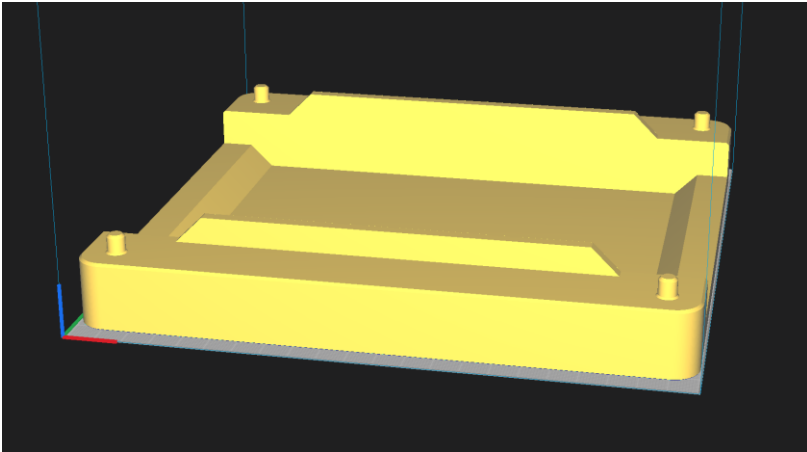
Press Bed (optional)



Layer Height: 0.2
Wall line count: 4
Alternate extra wall: Yes
Bottom Layers: 4
Top Layers: 4
Infill type: cubic
Infill Density: 25%
Support: No
Infill Before Walls: No
Build plate adhesion: skirt

Good Luck printing this one. After the brim is printed I usually tape it to the print bed.

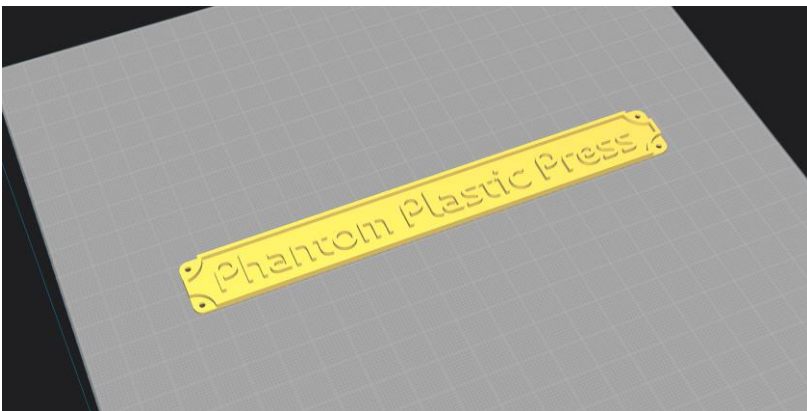
Base (Optional)



Layer Height: 0.3
Wall line count: 2
Alternate extra wall: doesn't matter
Bottom Layers: 4
Top Layers: 4
Infill type: cubic
Infill Density: 10%
Support: No
Infill Before Walls: No
Build plate adhesion: skirt

This can be the base for the press or you can skip this and screw the press halves to a $\frac{3}{4}$ inch thick piece of wood.

Nameplate



Layer Height: 0.2
Wall line count: 2
Alternate extra wall: doesn't matter
Bottom Layers: 4
Top Layers: 4
Infill type: cubic
Infill Density: 10%
Support: No

You can print this with multi colors to do a cool text effect or even do a relief roll to ink it up and let it dry. I would appreciate it if you put the nameplate on the base of your press.