



Q: Why do corners & thin parts of my prints warp?

A: Warping occurs due to the printed material cooling & shrinking. There are several techniques to reduce the effects of warping on a print.

## PRINT QUALITY

Thin features on the print warp off of the bed after several layers of printing.

## PROBLEM

The material creates tension on the thin features of the part as it cools and begins to pull away from the print surface.

Use Skirt/Brim: No

## CORRECTIVE ACTION

By slightly reducing the infill percentage, the internal tension can be relieved so the print does not curl as much. Adding a brim to the part gives the thin features more surface contact. The tension can be transferred to the brim and minimize the part curling away from the print bed.

- Use Skirt/Brim: Yes
- Skirt Layers: 1
- Skirt Offset from Part: 0mm
- Skirt Outlines: 15



Zoomed View

**Failed Print:** The warping has caused the edges to be scarred by the nozzle as it passes. The problem continues to compound as the print builds increases in height.



Failed Print



Fixed Print

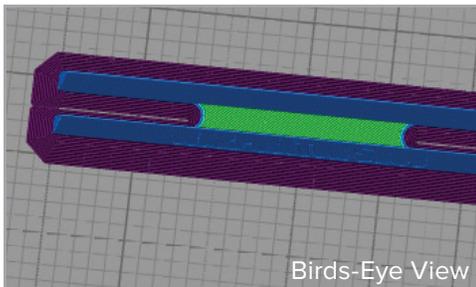


Zoomed View

**Fixed Print:** The brim easily breaks off of the print and with minimal cleanup, the print looks great. There is no warping.

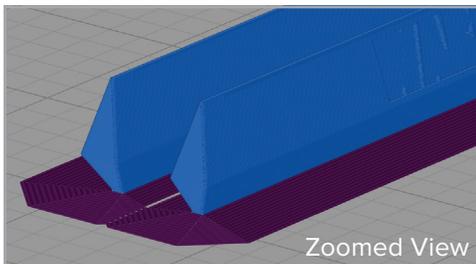


**Rule of Thumb:** Only make one adjustment at a time, so you can see the result of the change.



Birds-Eye View

**Model, Sliced:** The purple indicates the added brim around the solid part.



Zoomed View

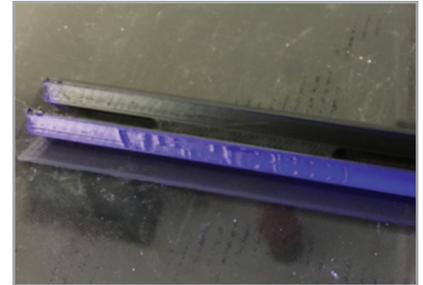
**Model, Sliced:** The brim is only 1 layer thick and can easily be removed with a hobby knife and deburring tool.

**KEY POINTS**

Reducing the tension in the print reduces the print warping. Maximizing the surface contact of the thin structures helps maintain bed adherence.



**Failed Print:** Thin edges of the print have curled away from the print bed while the thicker center of the part has stayed adhered to the surface.



**Fixed Print:** The brim remains attached to the build surface and provides a larger surface area for the small print features.

PRINT PROCESS SETTINGS	BEFORE FIX BAD PRINT	AFTER FIX GOOD PRINT
Material Type	PLA	PLA
Bed Temperature	80C	80C
Nozzle Size	.6mm	.6mm
Nozzle Temperature	190C	200C
Flow Rate (Extrusion Multiplier)	1.00	1.00
Extrusion Width	.68mm	.68mm
Print Speed	100mm/s	100mm/s
Layer Height	.2mm	.2mm
Number of Perimeters	2	2
Top Layers	5	5
Bottom Layers	5	5
Infill Percentage	20%	15%
Support Structures	none	none
Use skirt / brim	No	Yes
Skirt layers	n/a	1
Skirt offset	n/a	0mm

**OTHER NOTES**

Maintaining a stable room temperature is key to preventing warping. A cold room or a fan blowing on the part may cause a failed print.

