



3D PLATFORM TRAINING INSTRUCTION

PRINTING THE BREAKAWAY SUPPORT BOX

3DPT-0009

VERSION: NOVEMBER 2017

APPROVALS			
Name	Title	Signature	Date (YYYY-MM-DD)
B. Huebner	Author		2017-11-01
	Engineering		
	Service Manager		
	Manufacturing		

APPLICABLE MODELS	
Legacy	Work Series
<input checked="" type="checkbox"/> X1000	<input checked="" type="checkbox"/> 100
<input checked="" type="checkbox"/> X1000 CE	<input checked="" type="checkbox"/> 200
<input checked="" type="checkbox"/> WorkBench	<input checked="" type="checkbox"/> 300
	<input checked="" type="checkbox"/> 400

CONTROL STATUS
<input type="checkbox"/> Confidential
<input type="checkbox"/> Internal use only
<input checked="" type="checkbox"/> Uncontrolled

Table of Contents

Introduction & Purpose	2
Tools & Materials	2
Process	2
Printing the part	2
Prepare the printer	2
Prepare the part.....	2
Start print	2
Inspecting the part.....	3
Visual inspection:	3
Tool inspection:.....	3
Correcting the part	4
Wrapping up	4
Conclusion.....	4

Introduction & Purpose

The breakaway support box is a training tool designed to test the support settings and breakaway support on a small scale before printing a large part. This is critical to a large parts success and surface finish when support material is required.

This training instruction will guide the user through the steps of printing a model with breakaway support.

Tools & Materials

- Knife or razor scraper to remove completed prints.
- Calipers or micrometer.
- Print material (PLA material is recommended for calibration prints).
- Calibration chip gcode file
- SD card or network connection to the web interface (WorkSeries and later) to load print files.
- Needle Nose Pliers for support removal

Process

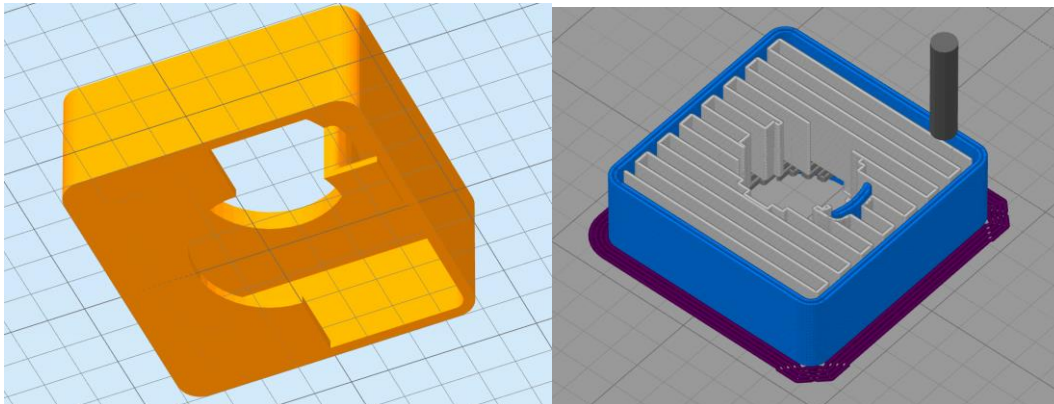
Printing the part

Prepare the printer

1. Prepare print bed. Clean the print area and apply adhesion agent if used.
2. Load material. Load print material and ensure extruder is extruding material properly.

Prepare the part

1. Prepare the gcode file. Orient the print within the slicer to minimize the use of support material. This will reduce print scarring after the supports have been removed. Add support material by selecting the support generation tool in the slicer. Optimize the support settings for the print.



2. Load file using SD card or web interface.

Start print

Print the gcode file and observe during the print. Ask these questions during the print.

- Is the support material too dense?
- Is the support material too sparse?
- Are there sufficient top layers to complete the top surface of the print?

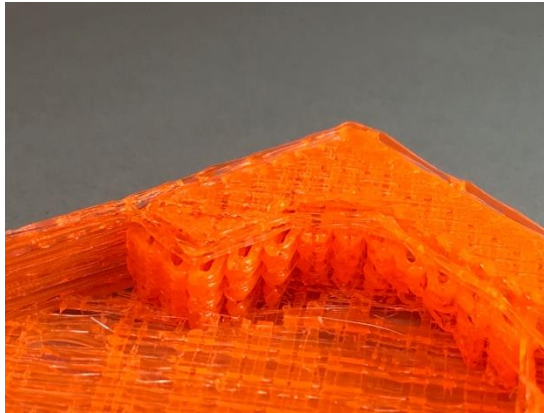
If the support material is too dense, the print will finish well but support removal will be difficult. If the support material is too sparse, the print may not finish well and will need to be reprinted. If there are not enough top layers to complete the top surface of the print, the support material will be ineffective.

Inspecting the part

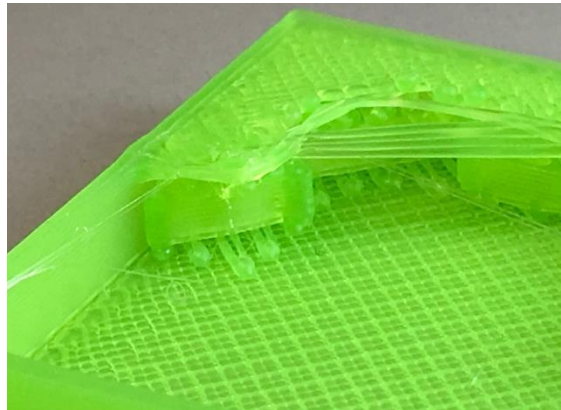
Visual inspection:

Look at the breakaway support cube after removal. There are visual and tactile clues to indicate whether the support material has been properly printed. There are three possibilities

1. Support material is too dense. The print will come off of the bed easily. Support material will print well but removal will be extremely difficult.



2. Support material is too sparse. The print will come off of the bed easily. Support material may have collapsed on itself or printed well. Top surfaces will have sagging and pillowing as a result of not having sufficient support material.
3. Not enough top surfaces. The print will come off of the bed easily. Support material will print well. The print looks unfinished. The tops must bridge across the open spaces between support columns. Too few top layers will result in a rough looking top surface.



Tool inspection:

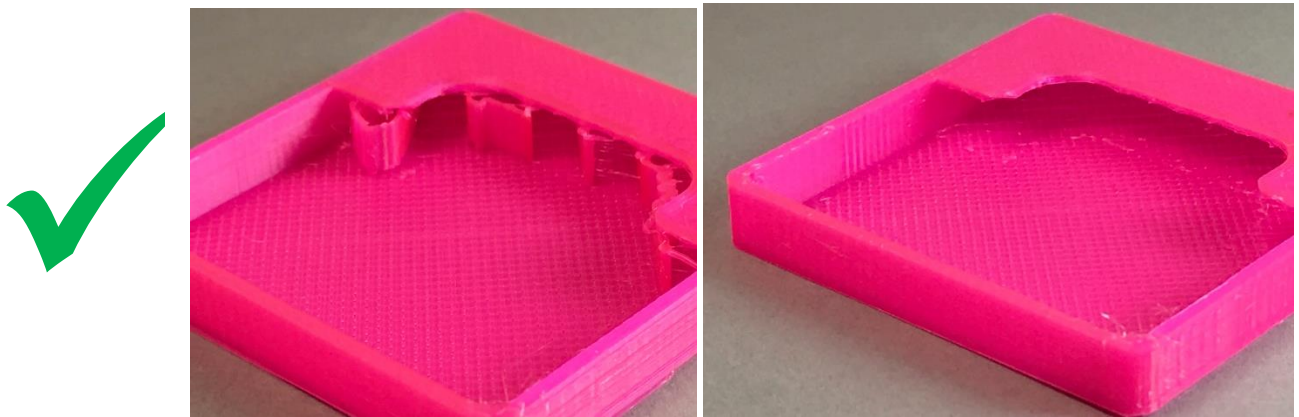
1. Use a set of calipers to measure the outer walls of the box and the Z height. Exact dimension will depend on which gcode file is printed, but will be displayed on the machine control once the print is complete (look for current z height).

Correcting the part

Corrections will depend on what is found during inspection.

1. **Support material is too dense.** Adjust the support infill density. This will reduce the number of support pillars used to generate the support material and aid in easily removable supports.
2. **Support Material is too sparse.** Adjust the support infill density. Increasing the density will result in a greater number of support pillars used to generate the support material. This will aid the print in completing successfully.
3. **Not enough top surfaces.** If the top surfaces have pillowing and a rough look, add more top surfaces to the print. This will give the print adequate layers to recover after bridging over the support material.

If changes are made within the slicer, reprint the breakaway support cube until the desired result is achieved.



Wrapping up

The breakaway support cube is a great tool to get started printing with support material. Try a few different geometries before printing large parts and save your results.

Conclusion

The breakaway support box is a simple but effective tool to introduce support material in printing. This print demonstrates the importance of optimizing breakaway supports, and provides a foundation for printing with support material.