# 3D Printing Workshop FAQs

## Key People for Support

**Kat Penzkover**, 3D Printing Engineer - Project Depot
**Sa'Von Thompson**, 3D Printing Technician - ITLL 1B12
**Engineering Support Student Staff** - LaunchPoint

### What is Layer Height?

The slicer (3DPrinterOS) will slice your solid object into layers and translate these layers into instructions for the 3D Printer (gcode).

**How Layer Height Works**:

- It controls resolution only on the vertical Z-axis, not the horizontal X/Y plane.
- Measured in millimeters (e.g., 0.12 mm, 0.2 mm, 0.28 mm).
- Set inside slicing software before you print

**Choosing the Right Size:**

- **0.10 mm (Fine):** Best for miniatures and high-detail display models.
- **0.20 mm (Standard):** The everyday choice for functional and cosmetic items.
- **0.28–0.32 mm (Coarse):** Best for quick drafts, rough prototypes, or large items

**Important Rules:**

- Stay within 25% to 75% of your nozzle diameter. For a 0.4 mm nozzle, do not go below 0.1 mm or above 0.3 mm easily.
- Thinner layers can improve layer bonding, while thicker layers drastically cut down total print time

### What are Perimeters?

This is another way to say Wall Thickness.

Wall thickness is the **size of the solid outer shell of a 3D-printed part.** For standard FDM printing with a 0.4 mm nozzle, use a minimum of 0.8 mm (two loops, or two **Perimeters**). Setting more perimeters will increase you wall thickness by the width of the nozzle (0.4mm). The default for the ITLL printers is 2 perimeters which is 0.8mm (3 perimeters = 1.2mm, 4 perimeters = 1.6mm...)

**Why It Matters**

- **Strength:** Thick walls make parts strong. Thin walls break or bend easily.
- **No Voids:** Matching your nozzle size stops the software from leaving weak gaps inside the wall.
- **Save Time:** Thicker walls use more plastic and take longer to print.

**Rules of Thumb**

- **Standard Models:** Use 0.8 mm to 1.2 mm (2 to 3 walls).
- **Strong Parts:** Use 1.6 mm to 2.4 mm (4 to 6 walls).
- **Minimum Limit:** Never go below 0.8 mm with a 0.4 mm nozzle unless you use special single-wall modes

## What is Infill?

In 3D printing, infill is the **internal geometric structure printed inside an object's outer shell.** Because most 3D printed objects are not printed completely solid to save time and material, the slicer software fills the empty space with a repeating lattice pattern that acts like a skeleton.

**Why Infill Matters**

- **Structural Support:** It stops flat top layers from sagging or falling into the empty middle during printing.
- **Strength vs. Weight:** It controls how strong, rigid, or heavy the final piece will be.
- **Time and Cost:** Less infill prints faster and uses less plastic filament. Most users agree that a partially hollow object with a smart infill is far more efficient than a 100% solid print.

**Key Settings**

**Density (%):** This tells the printer how packed the inside should be.

- *0% to 15%:* Best for display models, toys, and lightweight items.
- *15% to 50%:* Great for general use and standard functional prototypes.
- *50% to 100%:* Used for heavy-duty mechanical parts needing high strength.

**Pattern:** The shape of the inner grid. Common choices include:

- *Grid / Lines:* Fast to print, but only strong in a couple of directions.
- Cubic: Strong in all directions, easy to print
- *Gyroid:* A wavy, organic pattern that provides equal strength in all directions and works great with flexible plastics

## What is a Brim?

A brim in 3D printing is a flat, single-layer ring of extra plastic that surrounds the base of a model and directly touches its outer edge.  Acting like the brim of a hat, it widens the footprint of the print to dramatically improve bed adhesion and stop corners from curling or warping. I also like to call this a 'suction cup' added to the base of your print to help it maintain attachment to the bed.

### How a Brim Works

- **Wider footprint:** Adds several outer loops of filament tightly linked to the object's first layer.
- **More grip:** Increases the surface area touching the build plate so the print stays securely anchored.
- **Easy cleanup:** Peels away cleanly from the final object by hand after printing.

### When to Use a Brim

- **Warp-prone plastics:** Essential for filaments like ABS, Nylon, or large flat prints that shrink as they cool.
- **Small contact areas:** Keeps tiny bases, feet, or miniatures from breaking free mid-print.
- **Tall, skinny models:** Stabilizes vertical prints that might wobble and tip over from nozzle movement.
- **First layers with sharp corners**: Occasionally, there is some shrinkage due to cooling of a part and the stress is concentrated at the corners (such as the corners of boxes). This contraction can cause the part to peel up from the surface at the corners. A brim helps mitigate this problem.

### Brim vs. Skirt vs. Raft

- **Skirt:** A loop printed around the model without touching it; used only to prime the nozzle.
- **Brim:** Attached directly to the model's base on the build plate for adhesion.
- **Raft:** A thick, multi-layer web printed underneath the entire object.

## What is a Skirt?

A skirt in 3D printing is **a line of plastic printed in a loop around the outside of your model on the build plate, without touching the model itself**. It helps prime the nozzle, check filament flow, and preview the first layer before the actual print begins.

### Main Purposes of a Skirt

- **Prime the nozzle:** Ensures melted plastic is flowing evenly and builds proper pressure inside the hotend before your model starts.
- **Check first-layer quality:** Gives you a quick chance to spot bed-leveling or adhesion issues and cancel the print if the plastic is not sticking.
- **Clean the tip:** Catches any stray blobs or ooze hanging from the nozzle right before moving to the real print area.

**Customizable Settings**

- **Offset:** How far away the skirt sits from your actual print.
- **Outlines (Loops):** How many rings of plastic the printer makes. More loops mean a longer prime time.
- **Layers:** How high the skirt goes (usually just 1 layer is enough)

## What type of Support should I use?

Support material in 3D printing is **a temporary scaffolding built alongside an object to hold up overhangs, bridges, or steep angles that would otherwise sag or collapse in mid-air due to gravity or printing forces**.

**Why You Need Supports**

- **Overhangs:** Parts that stick out at an angle greater than 50 degrees from the vertical.
- **Bridges:** Horizontal gaps stretching between two raised points.

**Main Types of Support Structures**

- **Lattice / Grid:** Standard blocky, dense grids great for mechanical and heavy parts.
- **Tree / Organic:** Branching structures that touch the model in fewer spots. They use less plastic and leave fewer marks on artistic or curved prints.

**Common Material Approaches**

- **Standard Breakaway:** Printed using the exact same plastic as the object. You snap or cut them off by hand after printing, though they can leave rough marks.
- **Soluble Supports:** Printed from special materials like PVA filament that dissolve completely in water, or HIPS that dissolves in chemical baths. This leaves a smooth finish but requires a dual-extruder printer (check out the ITLL's Prusa XL for this type of printing)

Revision #4
Created 2026-02-23 13:07:20 MST by Rachel
Updated 2026-07-27 14:05:16 MDT by Kathryn