Prototyping Workshops

- Prototyping - Level 1 - Operating the Laser Machines
  - Laser Cutting Workshop FAQs
  - Workshop Resource #1
  - Workshop Resource #2
  - ESS Training Resources
- Prototyping - Level 1 - Operating the 3D Printers
  - 3D Printing Workshop FAQs
  - Handy 3D Printing Videos for next level printing:
  - Workshop Resource #2
  - ESS Training Resources

Prototyping - Level 1 - Operating the Laser Machines

Workshop Signup Link

Click here to sign up for this workshop

View Slides

Click here to see the workshop slides

Laser Cutting Workshop FAQs

When are workshops available to take?

The ITLP's Prototyping workshops typically begin on the third week of each semester, and run until Fall break. Signups will be available at the start of the semeste

Are there any required prerequisites to taking the Laser Cutting workshop?

The only requirement to sign up for this workshop is taking the **ITLL Tour,** which departs from LaunchPoint each weekday at 5:15pm. This will give you an overall understanding of the ITLL's resources, and afterwards you will gain access to longer hours and a computer login, both of which are needed to take the workshop!

How long does the Laser Cutting workshop run?

This workshop typically will take 60-75 minutes of instruction, with the remaining time being used to run your engravings.

Are there any materials should I bring with me?

No! You will be using our desktop computers, and we will provide materials to make your own engraving.

Workshop Resource #1

Workshop Resource #2

ESS Training Resources

Prototyping - Level 1 - Operating the 3D Printers

Click here to sign up for this workshop

Click here to see the workshop slides

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)

Handy 3D Printing Videos for next level printing:

Designing and slicing for multi material 3D printing

How to design print in place foldable models like FAB365 - 3D design for 3D printing

How to Design a Print with Perfect Tolerance EVERY Time

Ultimate Guide to Connecting 3D Printed Parts | Pins, Fins, Slots, & Snaps

Learn 15 Print-in-Place Mechanisms in 15 Minutes

Prusa XL - Multi-Material Testing (TPU and PLA)

NEVER Let Your Parts Warp Again

6 Ways To Make Snap Fits for 3D Printing

Joining Features | Design for Mass Production 3D Printing

How-to Design Print in Place Hinges | Design for Mass Production 3D Printing

Design Compliant Lids for Perfect Tolerance Every Time | Design for 3D Printing: Lids: Vol. 2

Stop Using Slicer Supports | Design for 3D Printing

Workshop Resource #2

ESS Training Resources