SPRINT 3D Printing

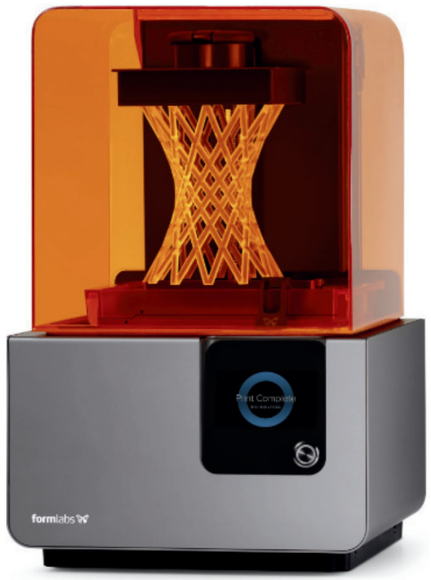
What Is 3D Printing?

3D printing, also called additive manufacturing, creates 3D objects layer by layer from a digital design. Unlike traditional manufacturing, which is best for mass production, 3D printing is great for making custom parts with less waste and lower costs.

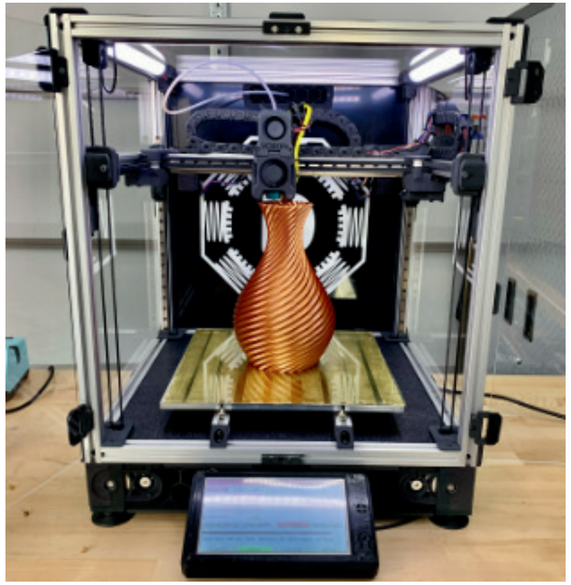
The origins of 3D printing date back to the 1980s when Charles Hull invented stereolithography, the first additive manufacturing process. Since then, people have used 3D printing to make all kinds of amazing things, like human tissues for medicine, rocket engines for space travel, and even houses built with giant 3D printers!

3D Printing in the Classroom

Schools use 3D printing to help students learn STEM skills, design prototypes, and bring ideas to life. The most common types are FDM and SLA because they are affordable, easy to use, and work well for different projects. FDM printers are great for beginners since they use plastic filament and are budget-friendly. SLA printers create high-detail models, making them useful for more advanced designs.



Fused Deposition Modeling (FDM)
Melts plastic filament and builds objects layer by layer.
Needs supports for overhangs and complex shapes.
Popular with hobbyists, educators, and small businesses. because it's affordable and easy to use.



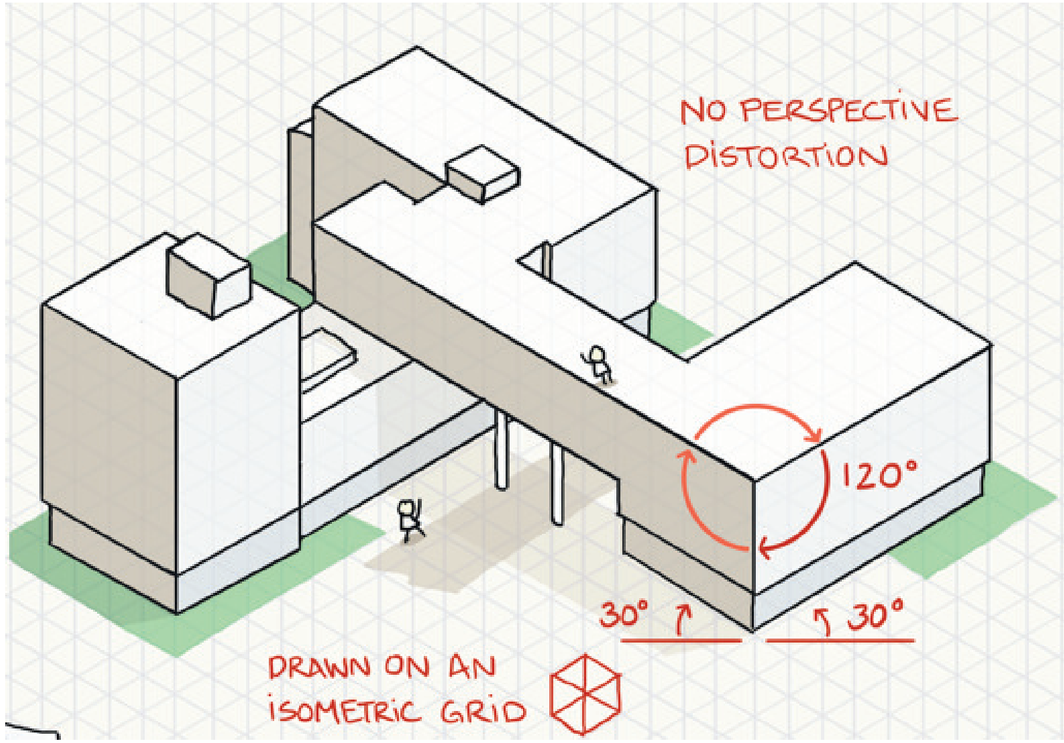
Stereolithography (SLA)
Uses a UV laser to harden liquid resin layer by layer.
Creates smooth, high-detail models.
Used by engineers, designers, and medical professionals for precision work.

Engineering Drawing

Sketching by hand is still an important part of the design process because it allows designers to quickly develop and explore ideas. Hand-drawn sketches help bring concepts to life.

To make objects look more realistic, we use 3D drawings to show depth and shape. Isometric drawings are one type of 3D drawing that use 120-degree angles to create accurate views.

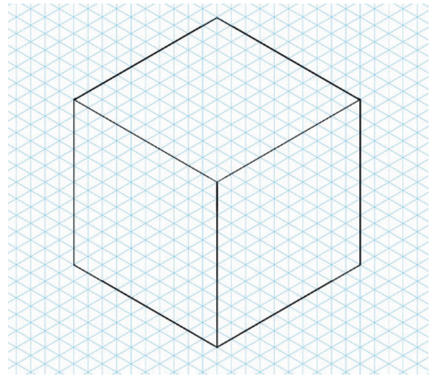
Using isometric grid paper helps guide the lines for these types of drawings making them an easy option for beginners. . Let’s practice drawing in 3D using this special grid!



Isometric Drawings

Steps:

1. Find a starting point on the grid and mark a dot where you want to begin.
2. Draw the front vertical edge of the object by following a straight-up line on the grid.
3. Use the grid lines to draw two slanted edges (one to the left, one to the right). These should follow the existing angles of the isometric grid.
4. Connect the lines to create the top, side, and front faces of the object.
5. Add details like cutouts, holes, or other features while following the grid lines.

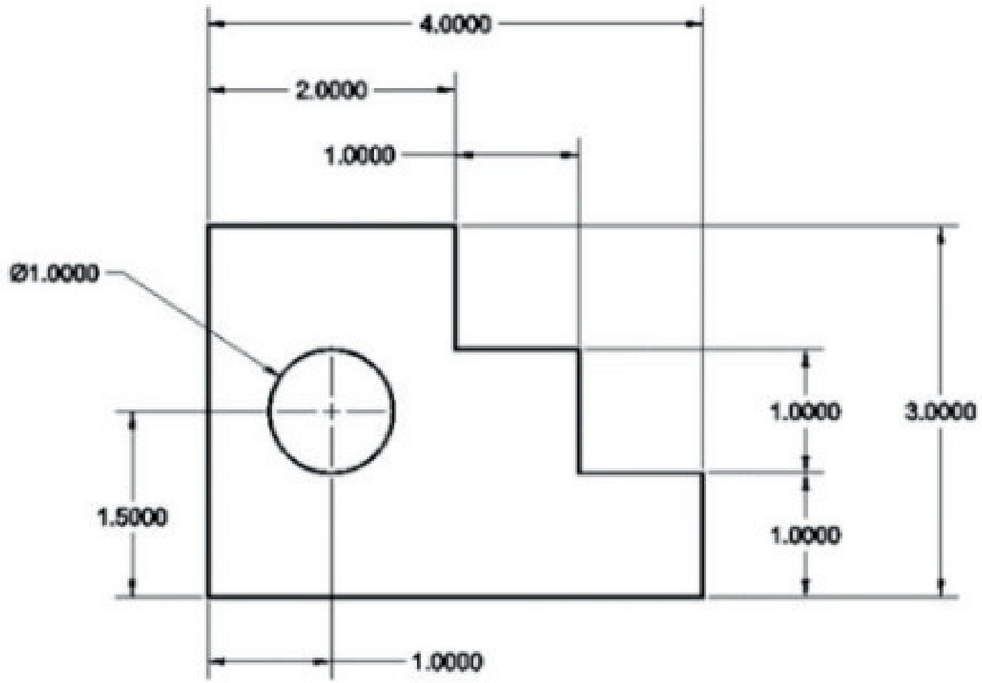


Dimensioning

Dimensions are the numbers that show the size and location of different parts of an object. These measurements are needed to build or create the object correctly.

Dimensions work alongside multi-view drawings to give a complete picture of the design. The way dimensions are placed on a drawing can change depending on the type of design being created.

Typically drawings are dimensioned using the metric system. Commonly Meters (m), Centimeters (cm) or millimeters (mm)

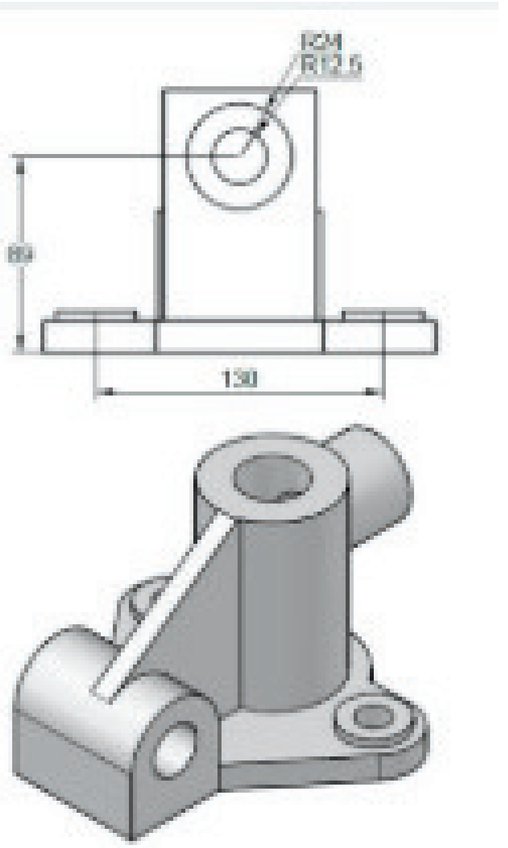


Computer Aided Design

Computer-aided design (CAD) software takes designs further by creating precise 2D drawings and 3D models. CAD is used for 3D printing because it creates the digital files that slicing software converts into instructions for the printer.

Popular CAD programs include Tinkercad, which is great for beginners, and AutoCAD, used for more complex projects. Other programs like Fusion 360 and SolidWorks offer advanced tools for professional designs.

If you don’t want to create a model from scratch, websites like Thingiverse, GrabCAD, and Cults3D allow users to download existing 3D designs that can be modified or printed directly.



Slicing Software

Slicing software turns a 3D model (like STL, OBJ, or 3MF files) into step-by-step instructions for a 3D printer. It slices the model into thin layers and creates G-code, which tells the printer how to build it.

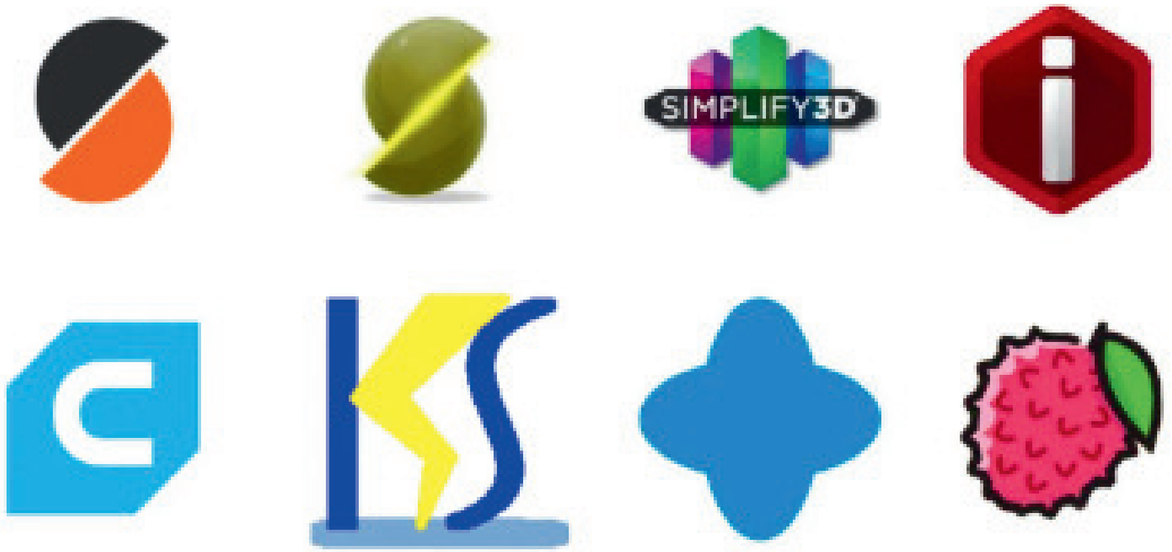
How Slicing Works:

- FDM (Fused Deposition Modeling): The software sets layer thickness, infill (how solid the object is inside), print speed, and support placement. This method is common in schools and for hobbyists.

- SLA (Stereolithography): The slicer controls resin layer thickness and directs a laser to harden specific areas. Supports are added for fragile or hanging parts.

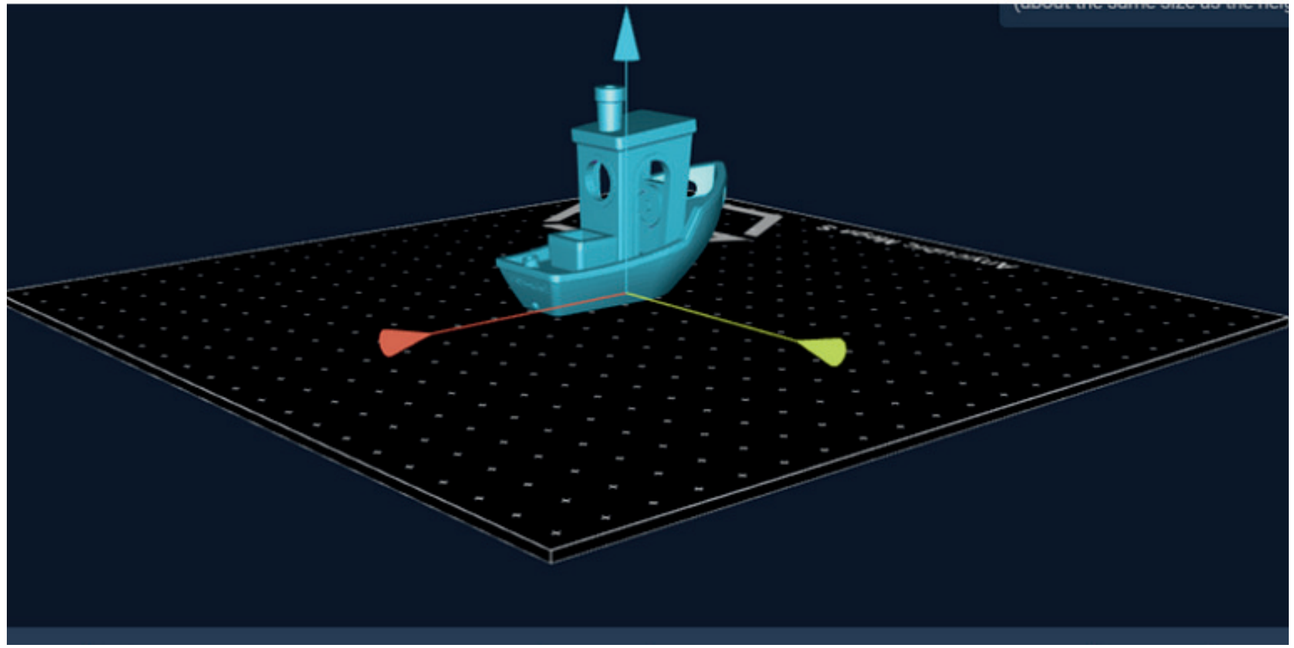
Uploading Your File

Download the appropriate slicing software for your 3D printer such as Cura, PrusaSlicer, or Simplify3D. This infomation can be found in your 3D printer's manual or website.



- Open the Software and Upload Your File
- Launch your slicing software and ensure it is configured for your specific 3D printer model.
- Click "Open" or "Import" and select your 3D model file (STL, OBJ, or 3MF format) from your computer.

Modifying Your Model



After upload, the model will appear on the build plate in the virtual workspace where you can modify if needed.

Positioning:

- Drag the model to center it on the build plate or use alignment tools.

Scaling:

- Adjust the size of your model using the scale tool and entering new dimensions or percentages.

Rotating or Mirroring

- Use the rotate tool to orient the model for optimal printing (e.g., minimizing overhangs) or the mirror tool to flip

Supports

In FDM and SLA 3D printing, supports are temporary structures that hold up overhanging parts during printing. Without them, gravity can cause these parts to sag or collapse. Supports are usually needed for overhangs steeper than 45 degrees since the printer can't lay material properly without a base.

After printing, supports are removed by hand or dissolved if made of a special water-soluble material. Most 3D printer software can add supports automatically.

Adjusting Printer Settings

Material Type:

Select the specific material you are using. If supports are required, they may use a different material.

Layer Height:

Set the layer resolution. For example, use 0.2mm for standard quality or 0.1mm for finer detail.

Infill Percentage:

Adjust the infill density. 20% provides standard strength, while higher percentages increase durability.

Support Structures:

Enable supports for models with overhangs.

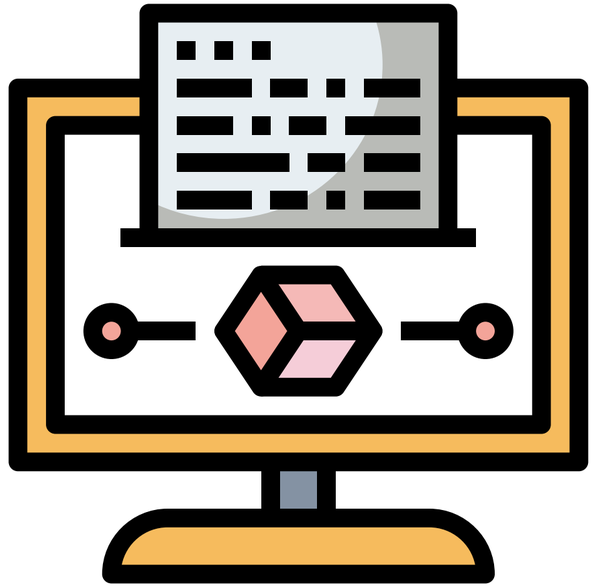
Choose the support type and placement as needed.

Adhesion:

Select an adhesion method (skirt, brim, or raft) to improve first-layer adhesion.

Slice, Save and Transfer

1. Once settings are configured, click the “Slice” button to generate a G-code file for your printer.
2. Look over the sliced layers to make sure there are no issues, such as gaps or unsupported areas.
3. Click “Save to File” or “Export G-code”, choosing the destination folder on your computer or directly saving to an SD card or USB drive, being sure to eject properly.
4. Insert the SD card or USB drive into your 3D printer.
5. Select the file from the printer’s menu to begin printing.



Engineering Design Process!

Diagnosing 3D Print Failures

When a 3D print fails, figuring out why is key to fixing the issue. Most failures come from weak layer adhesion, warping, voids, or incorrect print settings.

How to Analyze a Failed Print

1. Visual Inspection
2. Where did it break? (Along layers, corners, stress points?)
3. What does the break look like? (Smooth = tensile failure, jagged = shear failure, layers peeling = delamination)
4. Any visible defects? (Voids, gaps, rough layers?)

Preventing Common Failures

Common Failure Causes

- Interlayer delamination – Layers peeling apart
- Warping – Print curling due to uneven cooling
- Voids – Gaps in material reducing strength
- Under/Over extrusion – Too little or too much filament
- Bed adhesion failure – Print detaching from the bed

Avoiding Failures

- Choose the right material for strength and flexibility
- Improve design (avoid sharp corners, distribute stress)
- Keep the printer calibrated for consistent quality
- Adjust temperature settings for proper layer bonding and bed adhesion