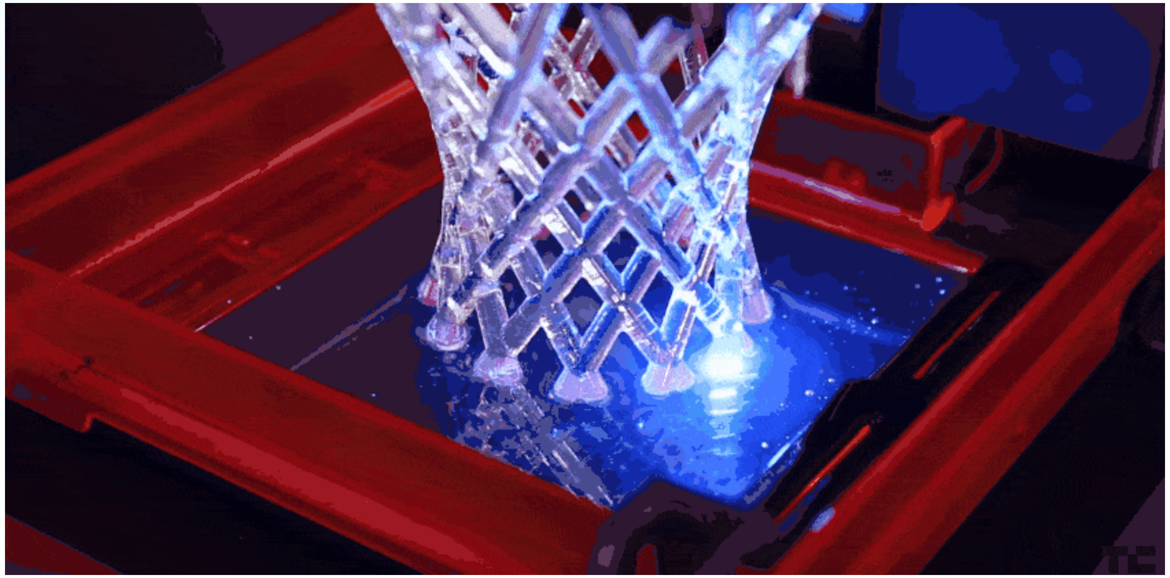
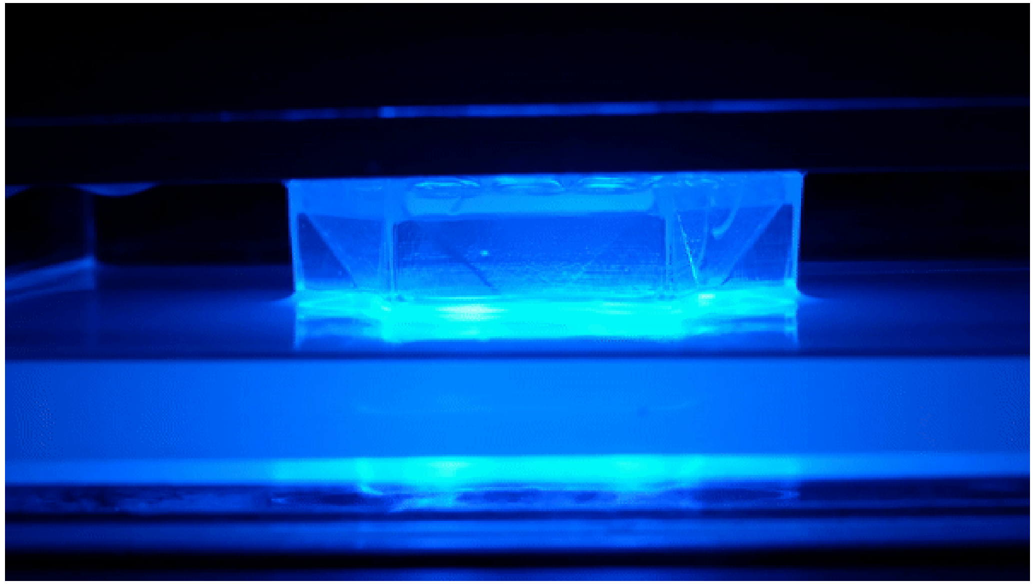
SLA,SLS,FDM,SGC & LOM

Dr. Prithvi C

Stereo Lithography

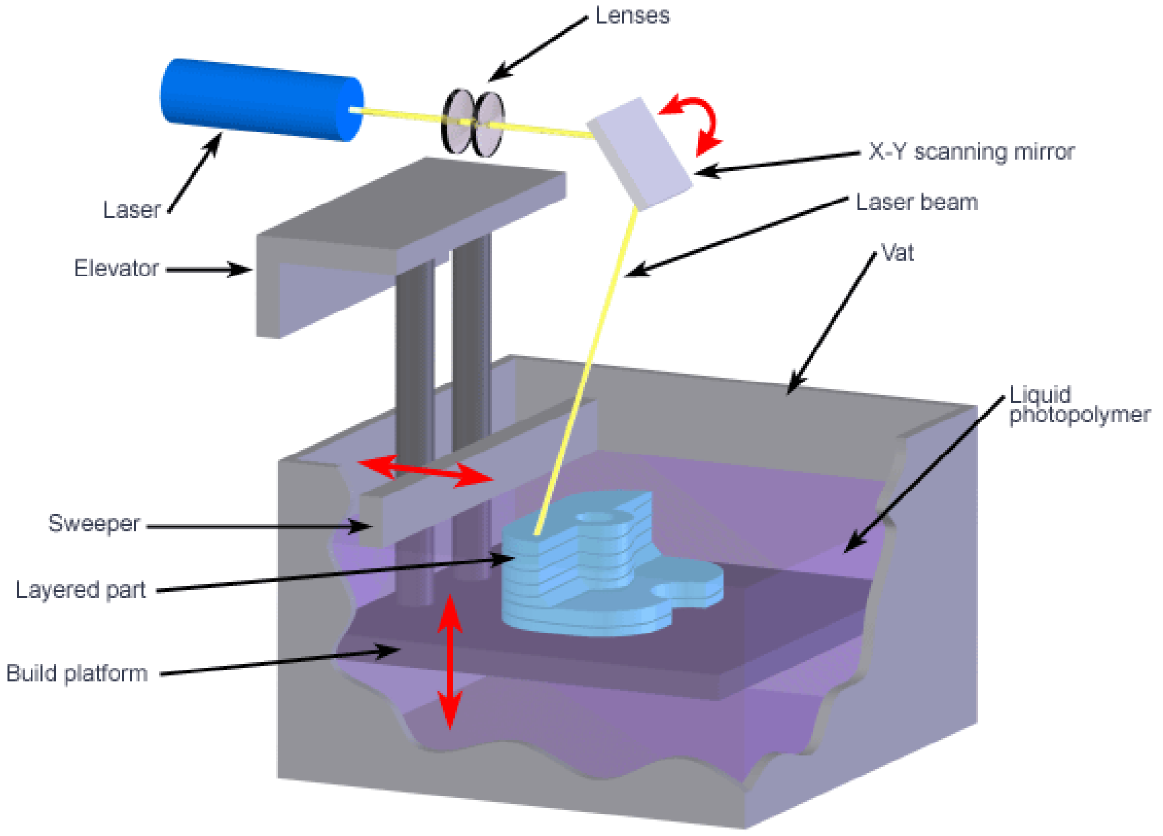
Stereolithography (SLA) is an additive manufacturing process that falls under the umbrella of 3D printing technologies. It is one of the earliest and most precise methods of 3D printing, widely used for creating high-resolution prototypes, models, and end-use parts.





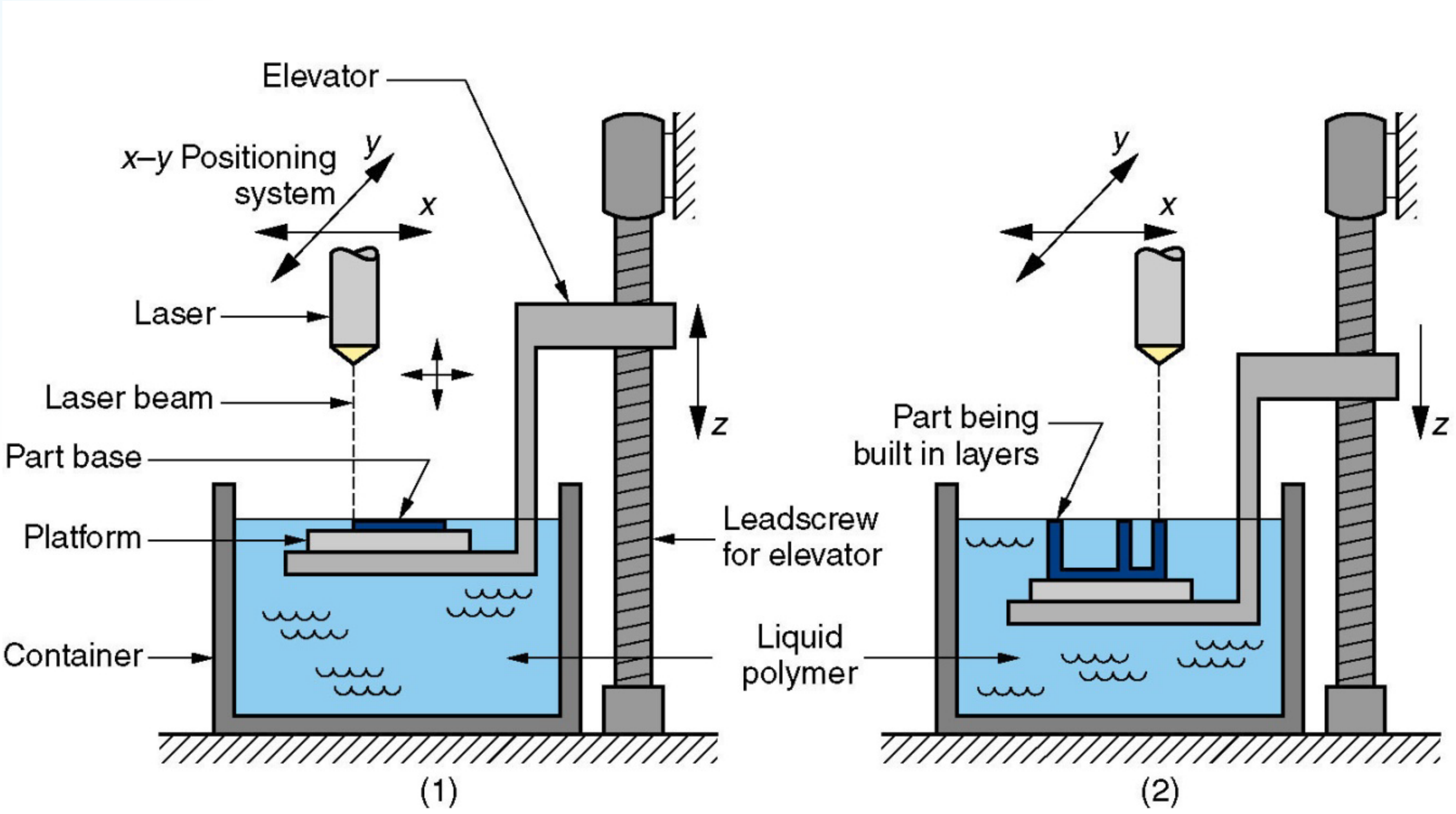
Working of SLA

- Liquid Resin: SLA uses a vat of liquid photopolymer resin as the printing material.
- UV Laser: A UV laser beam is directed onto the surface of the resin, curing (solidifying) it layer by layer.
- Layer-by-Layer Build: The build platform lowers incrementally after each layer is cured, allowing the next layer to be formed on top of the previous one.
- Post-Processing: After printing, the object is rinsed to remove excess resin and often cured further in a UV oven to achieve full strength.



Copyright © 2008 CustomPartNet

Working of SLA



Key Features of SLA

- High Precision: SLA is known for its ability to produce parts with fine details, smooth surfaces, and high dimensional accuracy.
- Material Variety: A wide range of photopolymer resins is available, including standard, tough, flexible, castable, and biocompatible options.
- Applications: Commonly used in industries like dentistry, jewelry, prototyping, and manufacturing for creating detailed models, molds, and functional parts.

Stereolithography Parts

Advantages and Limitations of SLA

Advantages

- High Resolution: Produces parts with excellent surface finish and fine details.
- Speed: Faster for small, intricate parts compared to some other 3D printing methods.
- Material Properties: Resins can be engineered for specific mechanical, thermal, or optical properties.

Disadvantages

- Material Limitations: Resins are generally more brittle and less durable than materials used in other 3D printing methods like FDM or SLS.
- Post-Processing: Requires cleaning and additional curing, which can be time-consuming.
- Cost: Resins and SLA printers can be more expensive compared to other 3D printing technologies.

Applications of SLA

- Prototyping: Rapid creation of detailed prototypes for design validation.
- Dentistry: Custom dental models, aligners, and crowns.
- Jewelry: High-detail wax patterns for casting.
- Medical: Surgical guides and anatomical models.
- Art and Design: Intricate sculptures and artistic creations.

Process Parameters

Laser Power:

- Determines the intensity of the UV light used to cure the resin.
- Higher power can increase curing speed but may lead to over-curing or reduced precision.

Layer Thickness:

- Typically ranges from 25 to 100 microns.
- Thinner layers result in higher resolution and smoother surfaces but increase print time.

Scan Speed:

- The speed at which the laser moves across the resin surface.
- Faster speeds reduce print time but may compromise detail and accuracy.

Process Parameters

Resin Properties:

- Viscosity, curing time, and mechanical properties of the resin affect the print quality.
- Different resins (standard, tough, flexible, etc.) are chosen based on the application.

Build Orientation:

- The angle at which the part is oriented on the build platform.
- Proper orientation minimizes support structures and improves surface finish.

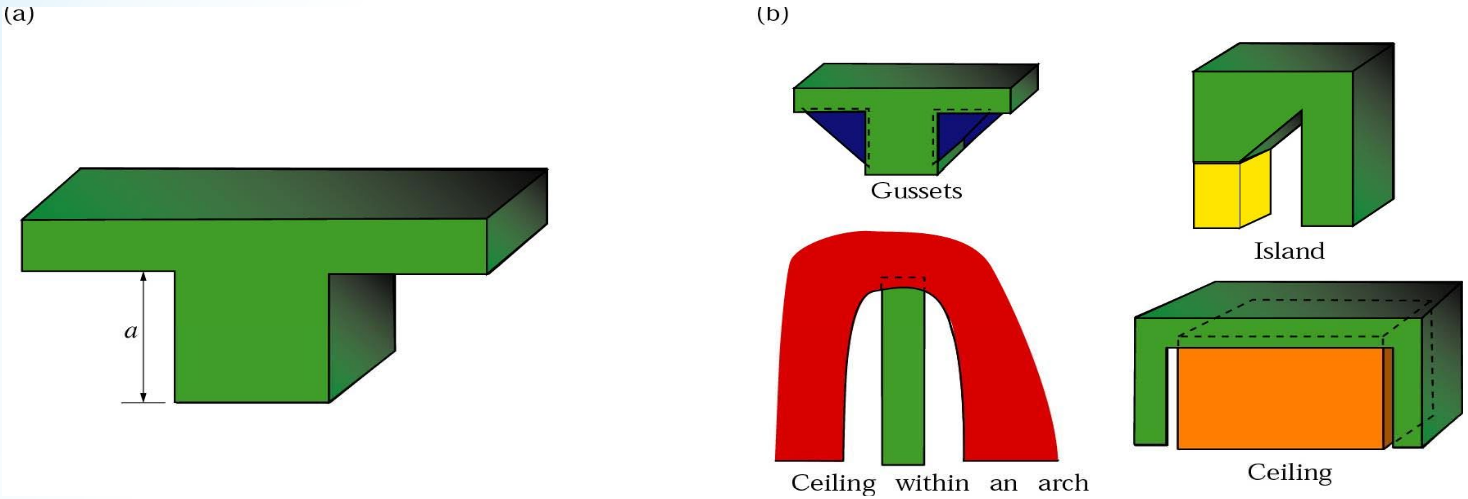
Support Structures:

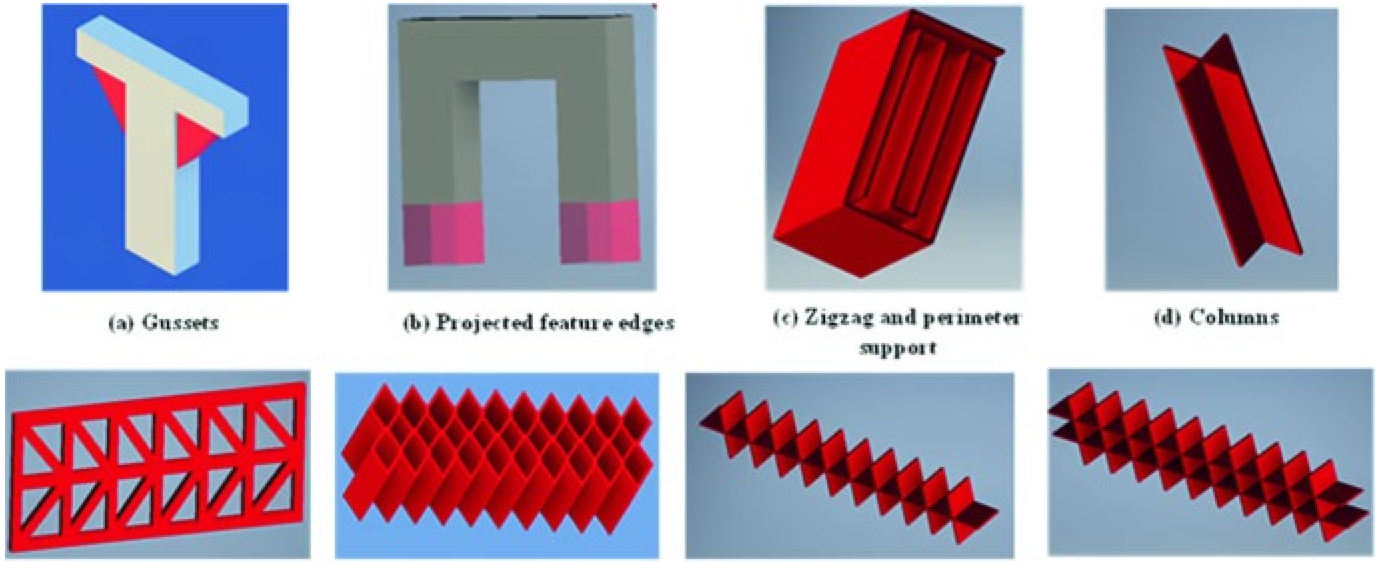
- Required for overhangs and complex geometries to prevent deformation during printing.
- Supports are removed during post-processing.

Post-Curing:

- Additional UV curing after printing to enhance the mechanical properties and stability of the part.

Common Support Structures





(a) Gussets
(b) Projected feature edges
(c) Zigzag and perimeter support
(d) Columns
(f) Perforated wall structures
(e) Perforated wall structures for various web-based support design
(g) Single web
(h) Webs

Process Details

Step 1: Preparation

3D Model Creation: A digital 3D model is created using CAD software and exported in STL or OBJ format.

Slicing: The model is sliced into thin layers using slicing software, which generates the toolpath for the UV laser.

Step 2: Printing

Resin Filling: The build platform is submerged in a vat of liquid photopolymer resin.

Laser Curing: A UV laser traces the cross-section of the first layer onto the resin surface, curing it into a solid.

Layer Addition: The build platform moves down by one layer thickness, and the process repeats until the entire object is printed.

Step 3: Post-Processing

Resin Removal: The printed part is removed from the build platform and rinsed in a solvent (e.g., isopropyl alcohol) to remove uncured resin.

Support Removal: Support structures are carefully removed using tools.

Post-Curing: The part is placed in a UV oven for additional curing to achieve full strength and stability.

Process Overview

Aspect	Details
Principle	Photopolymerization of liquid resin using a UV laser.
Layer Thickness	25–100 microns (adjustable for resolution vs. speed).
Laser Power	Adjustable to control curing speed and precision.
Resin Types	Standard, tough, flexible, castable, biocompatible, etc.
Build Orientation	Optimized to minimize supports and improve surface finish.
Post-Processing	Rinsing, support removal, and post-curing in a UV oven.
Applications	Prototyping, dental models, jewelry, medical devices, art, and design.

Part Build Time in STL

Time to complete a single layer :

$T_i = (A_i / vD) + T_d$

Where,

T_i = time to complete layer i;

A_i = area of layer i;

v = average scanning speed of the laser beam at the surface;

D = diameter of the “spot size,” assumed circular; and

T_d = delay time between layers to reposition the worktable.

Once the T_i values have been determined for all layers, then the build cycle time is:

$$T_c = \sum_{i=1}^{n_l} T_i$$

where T_c = STL build cycle time; and n_l = number of layers used to approximate the part

Numerical Problem

A prototype of a tube with a square cross-section is to be fabricated using stereolithography. The outside dimension of the square = 100 mm and the inside dimension = 90 mm (wall thickness = 5 mm except at corners). The height of the tube (z-direction) = 80 mm. Layer thickness = 0.10 mm. The diameter of the laser beam ("spot size") = 0.25 mm, and the beam is moved across the surface of the photopolymer at a velocity of 500 mm/s. Compute an estimate for the time required to build the part, if 10s are lost each layer to lower the height of the platform that holds the part. Neglect the time for postcuring.

Solution: Layer area A_i is same for all layers.

$A_i = 100^2 - 90^2 = 1900\ mm^2$.

Time to complete one layer T_i is same for all layers.

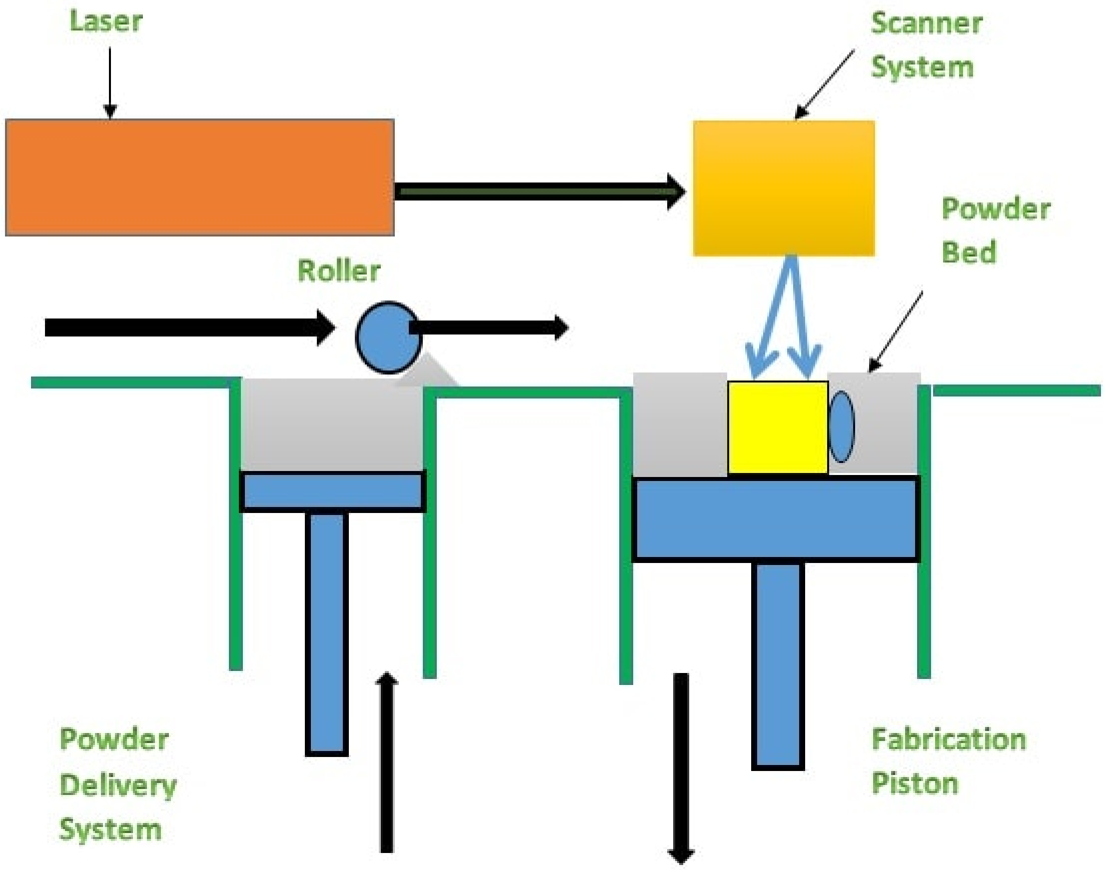
$T_i = (1900\ mm^2)/(0.25\ mm)(500 mm/s) + 10\ s = 25.2\ s$

Number of layers $n_l = (80\ mm)/(0.10\ mm/layer) = 800$ layers

$$T_c = 800(25.2) = 20{,}160\ s = 336.0\ min = 5.6\ hr$$

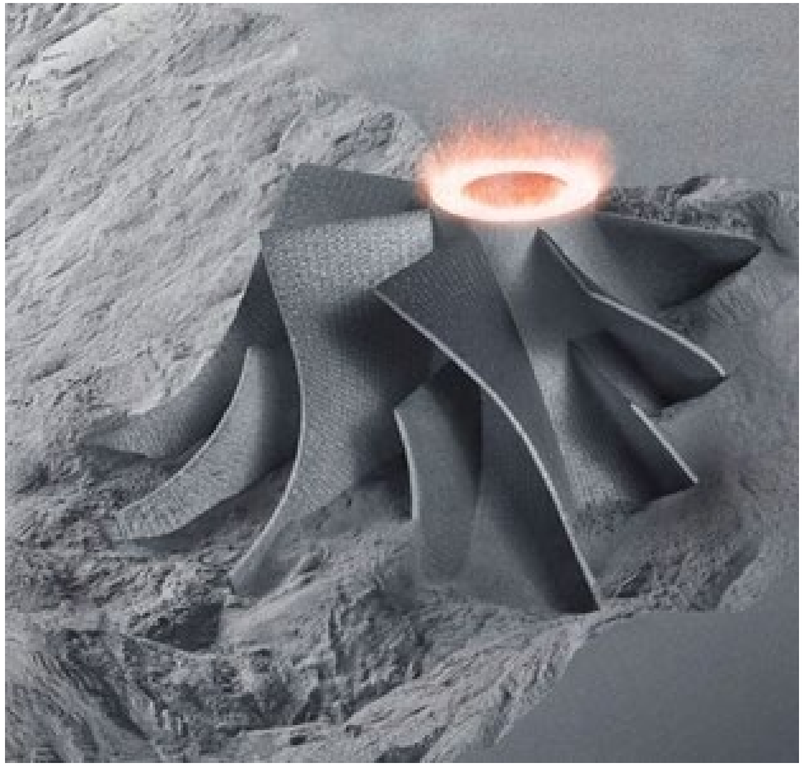
Selective Laser Sintering (SLS)

Selective laser sintering (SLS) is a 3d printing process (additive manufacturing) that uses high-powered lasers to sinter, or bind, finely powdered material together into a solid structure. In this process, a printer lays down an even layer of powder and then precisely sinters that layer, repeating the deposition and sintering process until the part is complete. The shape of the object is created by aiming a laser at the powder bed in specific points in space, guided by a digitally produced CAD (computer-aided design) file.



Selective Laser Sintering (SLS)

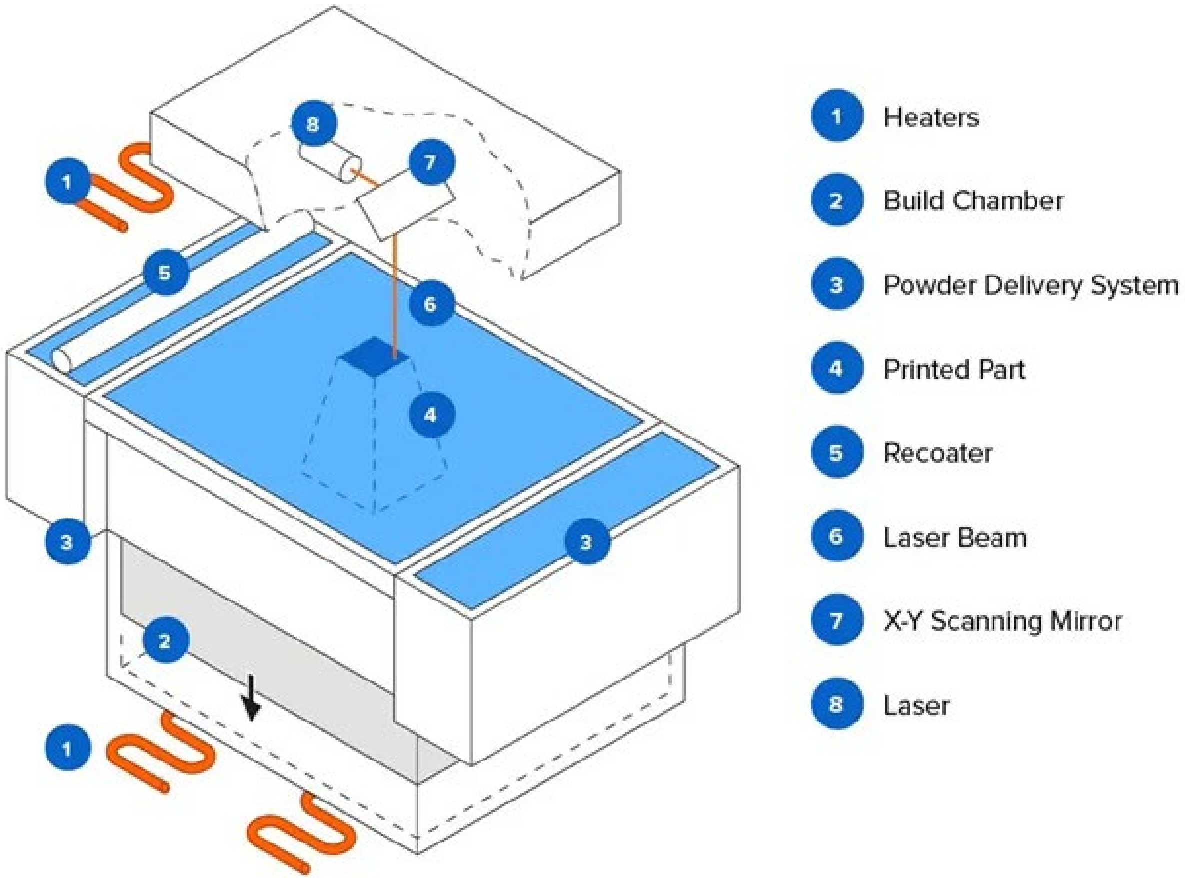
Unlike selective laser melting (SLM) which fully melts particles together, sintering causes an atomic reaction that fuses particles, turning powdered material into a solid structure. The term SLS is typically only used to refer to plastic and ceramic 3D printers — metal 3D printers using a similar process are referred to as DMLS or SLM machines.



SLS machines can produce high fidelity items, including precision, low volume parts in automotive and aerospace fields. SLS can produce large or geometrically complex, intricate, and highly accurate parts from a variety of materials.

Selective Laser Sintering - Working

- Preparation: A 3D model is created using CAD software and sliced into layers.
- Powder Bed: A thin layer of powdered material (e.g., nylon, metal, or ceramic) is spread evenly across the build platform.
- Laser Sintering: A laser selectively sinters (fuses) the powder particles according to the cross-section of the 3D model.
- Layer-by-Layer Process: The build platform lowers, and a new layer of powder is spread. The process repeats until the entire object is formed.



- Cooling and Post-Processing: After printing, the object is cooled, and excess powder is removed. Additional post-processing (e.g., sanding, dyeing, or coating) may be applied.

Advantages of using SLS

- No support structures: Unlike some other 3D printing processes, the part does not need any support structures since empty spaces are filled with unused loose powder making it self-supporting. This allows you the freedom to design empty hollow spaces, overhanging features and very thin features.
- High productivity: The process is one of the fastest 3D printing technologies since the lasers have a fast-scanning speed and the powders used only need a short exposure for fusing.
- Excellent mechanical properties: The SLS process produces very strong adhesion between layers so parts have good isotropic properties. This means that their tensile strength, hardness and elongation to break are similar across the x, y and z axes.
- Ideal for dying and colouring: Parts produced by SLS tend to have a porous surface – which could be an advantage or a disadvantage depending on your application. What it does mean is that they are an excellent choice for dying or colouring.
- Reduced product development time: Like all 3D printing technologies selective laser sintering allows engineers to prototype parts cost-effectively early in the design cycle, since it does not require tooling and involves minimal set up.

Disadvantages of using SLS

- Fewer materials: You don't get a wide choice of materials for SLS. Most projects use nylon-based materials, or polyamides, which are excellent engineering-grade plastics that can be used across a wide range of applications.
- Rough surface and porosity: The same porosity that makes SLS printed parts so great for colouring, also means that they have a relatively rough surface, are not leakproof and have a low impact strength, or brittleness.
- High shrink rate: Because the print powder is subjected to high temperatures for it to sinter, this does mean that as it starts to cool it shrinks which can produce a dimensionally less accurate part than other additive manufacturing technologies. Depending on the design, the shrinkage rate can be as high as 3 to 4%.
- Higher waste than other additive manufacturing: One of the key advantages with additive manufacturing is the minimal waste of material using the technology. Unfortunately, SLS does produce some waste since the powder in the chamber is preheated so that it will sinter with minimal exposure to the laser.

Applications of SLS

- Prototyping: Functional prototypes for testing and validation.
- Manufacturing: Production of end-use parts, especially for low-volume or custom products.
- Medical: Custom prosthetics, implants, and surgical guides.
- Aerospace: Lightweight, complex components for aircraft and spacecraft.
- Automotive: Custom parts and tooling.



Process Parameters

1. Laser Power: The energy output of the laser used to sinter the powder.

Impact:

- Higher laser power increases sintering depth and part density but may cause overheating or warping.
- Lower laser power may result in incomplete sintering and weak parts.

Typical Range: 30–200 watts, depending on the material and machine.

2. Scan Speed: The speed at which the laser moves across the powder bed.

Impact:

- Faster scan speeds reduce build time but may lead to insufficient sintering.
- Slower scan speeds improve part density and accuracy but increase build time.

Typical Range: 1–10 m/s, depending on the material and laser power.

Process Parameters

3. Layer Thickness: The thickness of each layer of powder spread on the build platform.

Impact:

- Thinner layers improve resolution and surface finish but increase build time.
- Thicker layers reduce build time but may compromise detail and surface quality.

Typical Range: 0.05–0.15 mm for polymers; 0.02–0.1 mm for metals.

4. Powder Bed Temperature: The temperature of the powder bed before and during sintering.

Impact:

- Higher temperatures improve sintering and reduce residual stress but may cause powder degradation.
- Lower temperatures may lead to poor layer adhesion and warping.

Typical Range: Just below the melting point of the material (e.g., ~170°C for nylon).

Process Parameters

5. Hatch Spacing: The distance between adjacent laser scan lines.

Impact:

- Smaller hatch spacing improves part density but increases build time.
- Larger hatch spacing reduces build time but may create voids or weak spots.

Typical Range: 0.1–0.5 mm, depending on the material and laser spot size.

6. Laser Spot Size: The diameter of the laser beam at the powder bed.

Impact:

- Smaller spot sizes improve detail and resolution but may slow down the process.
- Larger spot sizes increase build speed but reduce precision.

Typical Range: 0.1–0.5 mm.

Process Parameters

7. Scanning Strategy: The pattern in which the laser scans the powder bed (e.g., zigzag, raster, or contour).

Impact:

- Different strategies affect part strength, residual stress, and build time.
- Contour scanning improves edge accuracy, while raster scanning ensures uniform density.

8. Powder Material Properties: The characteristics of the powder material, such as particle size, shape, and flowability.

Impact:

- Smaller particles improve resolution but may reduce flowability.
- Spherical particles improve packing density and flowability.

Typical Particle Size: 20–80 microns for polymers; 10–50 microns for metals.

Process Parameters

9. Cooling Rate: The rate at which the part cools after sintering.

Impact:

- Rapid cooling may cause warping or residual stress.
- Controlled cooling improves part stability and mechanical properties.

10. Atmosphere Control: The environment inside the build chamber (inert gas like nitrogen or argon).

Impact:

- Prevents oxidation and degradation of the powder material.
- Critical for metal SLS to avoid contamination.

Optimization of Parameters:

Material-Specific Settings: Each material (e.g., nylon, aluminum, titanium) requires unique parameter combinations.

Trial and Error: Initial test prints are often needed to fine-tune parameters for specific geometries and applications.

Software Control: Advanced SLS machines use software to automatically adjust parameters for optimal results.

Data Preparation for SLS

1. 3D Model Creation

Software: Use CAD software (e.g., SolidWorks, Fusion 360, Rhino) to design the 3D model.

Requirements:

- Ensure the model is watertight (no gaps or holes in the mesh).
- Avoid overly thin features that may not print correctly.
- Consider SLS-specific design guidelines (e.g., minimum wall thickness, hole size, and overhang angles).

2. File Export

File Format: Export the 3D model in a compatible file format, typically STL (Standard Tessellation Language) or OBJ.

Resolution: Set an appropriate resolution for the STL file:

- Too high: Large file size, slower processing.
- Too low: Loss of detail, faceted surfaces.

Recommendation: Use a resolution that balances file size and detail (e.g., 0.01–0.05 mm tolerance).

Data Preparation for SLS

3. STL File Repair

Check for Errors: Use mesh repair software (e.g., Netfabb, Meshmixer, or Materialise Magics) to fix common issues like Non-manifold edges, Holes in the mesh, Intersecting or overlapping surfaces.

Importance: A faulty STL file can lead to printing failures or defective parts.

4. Orientation and Positioning

Optimize Part Orientation: Minimize overhangs to reduce the need for support structures (SLS typically doesn't require supports, but orientation affects surface quality and strength). Consider the build platform size and maximize part density for batch printing.

Layer Orientation: Align the part to optimize mechanical properties (e.g., strength along critical axes).

5. Nesting: Arranging multiple parts on the build platform to maximize space utilization.

Considerations: Ensure adequate spacing between parts to avoid fusion. Account for thermal expansion and shrinkage during cooling.

Software: Use SLS-specific software (e.g., Materialise Magics) for efficient nesting.

Data Preparation for SLS

6. Slicing: Dividing the 3D model into thin horizontal layers (slices) for the SLS machine to process.

Software: Use the machine's proprietary software or third-party tools (e.g., Materialise Magics, Simplify3D).

Layer Thickness: Set according to the desired resolution and material (typically 0.05–0.15 mm for polymers).

7. Parameter Settings

Machine-Specific Settings: Input process parameters such as: Laser power, Scan speed, Hatch spacing, Powder bed temperature.

Material-Specific Settings: Adjust parameters based on the material being used (e.g., nylon, TPU, or metal).

9. File Transfer

Format: Convert the sliced file into a machine-readable format (e.g., .SLI, .CLI, or proprietary formats).

Transfer: Send the file to the SLS machine via USB, network, or cloud.

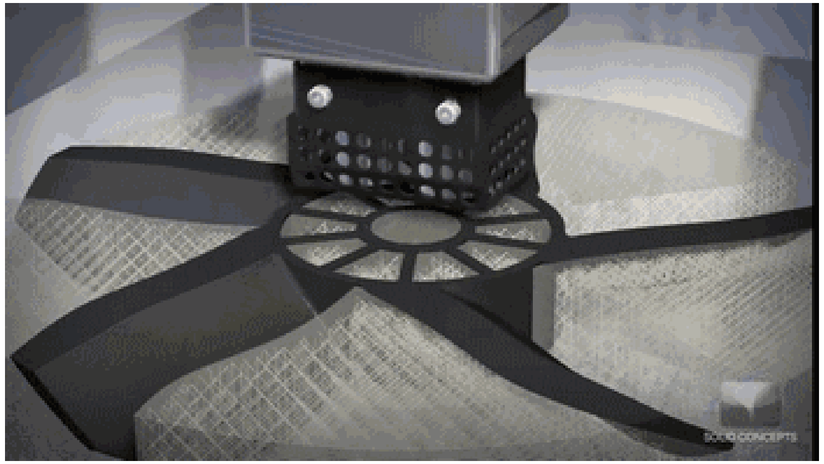
Tools and Software for SLS Data Preparation

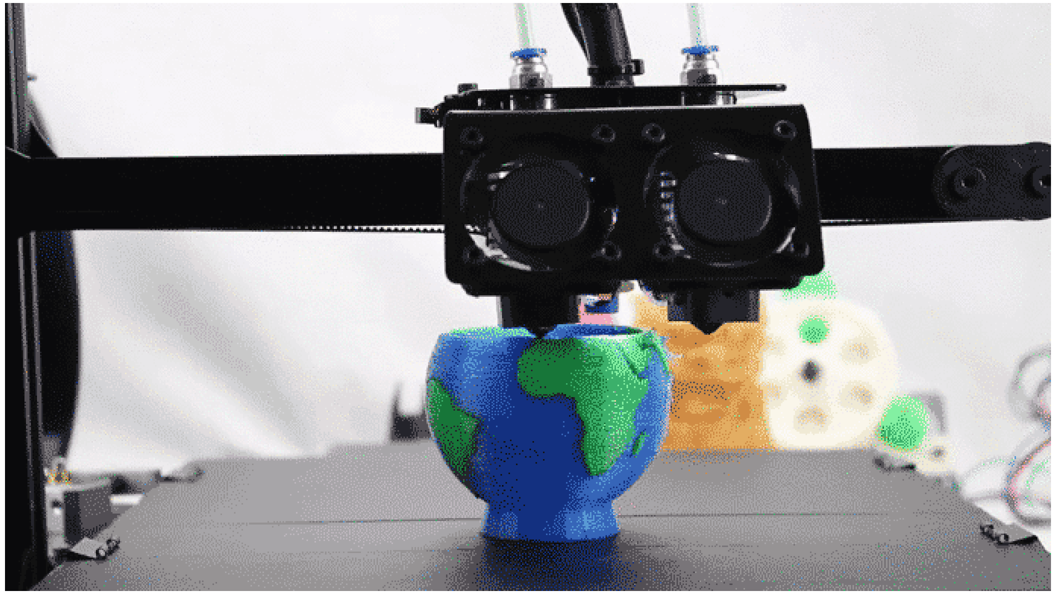
- ***Materialise Magics:*** Industry-standard software for STL repair, nesting, and support generation.
- ***Netfabb:*** Autodesk's tool for mesh repair and optimization.
- ***Meshmixer:*** Free tool for mesh editing and repair.
- ***Simplify3D:*** Slicing software with advanced parameter control.
- ***Machine-Specific Software:*** Most SLS machines come with proprietary software for slicing and parameter setting.

Fusion Deposition Modeling (FDM)

Fused deposition modeling (FDM) 3D printing, also known as fused filament fabrication (FFF), is an additive manufacturing (AM) process within the realm of material extrusion. FDM builds parts layer by layer by selectively depositing melted material in a predetermined path. It uses thermoplastic polymers that come in filaments to form the final physical objects.

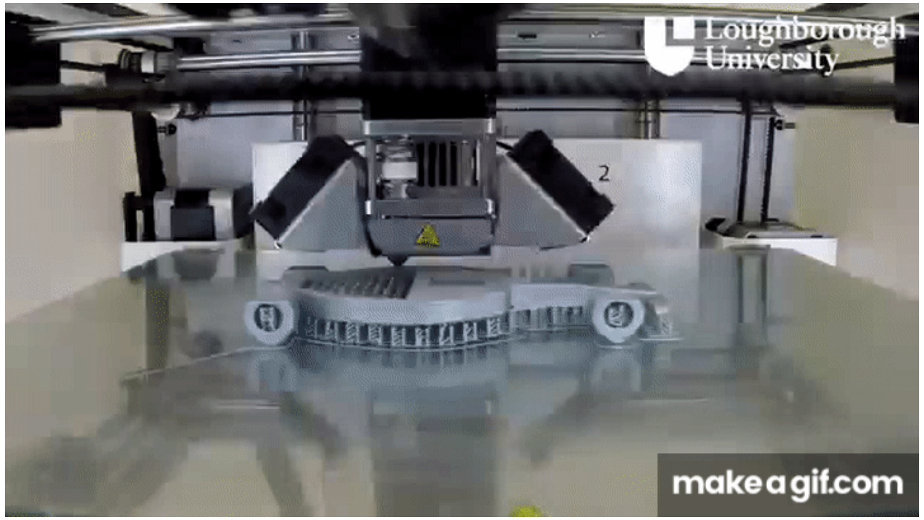
Composing the largest installed base of 3D printers worldwide, FDM is the most widely used technology across most industries, and likely the first process you think of when 3D printing comes up.



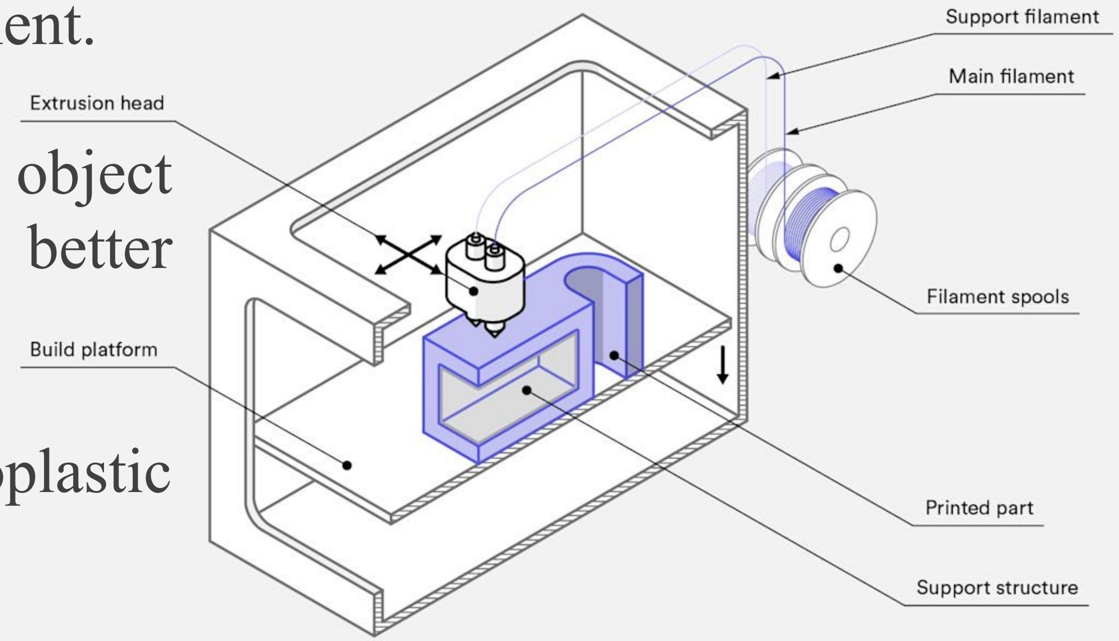


Working of FDM

- Material Preparation: A thermoplastic filament (e.g., PLA, ABS, PETG, or TPU) is fed into the 3D printer.
- Heating and Extrusion: The filament is heated to its melting point in the printer's extruder and then pushed through a nozzle.
- Layer-by-Layer Deposition: The molten material is deposited onto a build platform in precise layers, following a digital 3D model (typically in STL or G-code format).
- Cooling and Solidification: Each layer cools and solidifies, bonding to the previous layer to form the final object.
- Post-Processing: After printing, support structures (if used) are removed, and the object may be sanded, painted, or otherwise finished.



Components of an FDM Printer



- Extruder: Heats and extrudes the filament.
- Build Platform: The surface where the object is printed (may be heated for better adhesion).
- Filament Spool: Holds the thermoplastic material.
- Nozzle: Determines the layer resolution and extrusion width.
- Motion System: Controls the movement of the extruder and build platform (typically using stepper motors).

Materials Used in FDM

- ***PLA (Polylactic Acid):*** Easy to print, biodegradable, and ideal for beginners.
- ***ABS (Acrylonitrile Butadiene Styrene):*** Durable and heat-resistant but requires a heated bed and ventilation.
- ***PETG (Polyethylene Terephthalate Glycol):*** Strong, flexible, and resistant to moisture and chemicals.
- ***TPU (Thermoplastic Polyurethane):*** Flexible and rubber-like, used for soft parts.
- ***Nylon:*** Strong, durable, and flexible, but requires higher printing temperatures.
- ***Composite Filaments:*** Include materials like carbon fiber, wood, or metal particles for added strength or aesthetic effects.

Applications of FDM

- ***Prototyping***: Rapid creation of functional prototypes.
- ***Manufacturing***: Low-volume production of end-use parts.
- ***Education***: Teaching design and engineering concepts.
- ***Customization***: Creating personalized items like phone cases, toys, or tools.
- ***Medical***: Prosthetics, orthotics, and surgical guides.
- ***Aerospace and Automotive***: Lightweight, durable components

Advantages and Limitation of FDM

Advantages of FDM

- ***Cost-Effective***: FDM printers and materials are relatively affordable.
- ***Ease of Use***: User-friendly and widely accessible for hobbyists and professionals.
- ***Material Variety***: Supports a wide range of thermoplastics.
- ***Scalability***: Can produce small to medium-sized objects.
- ***Durability***: Printed parts are strong and functional.

Limitations of FDM

- ***Layer Lines***: Visible layer lines can affect surface finish and require post-processing.
- ***Limited Resolution***: Lower detail compared to technologies like SLA or SLS.
- ***Anisotropic Strength***: Parts are weaker along the layer lines.
- ***Support Structures***: Overhangs and complex geometries may require supports, which can be difficult to remove.
- ***Warping***: Thermal shrinkage can cause warping, especially with materials like ABS.

Data Preparation in FDM

1. ***3D Model Creation:*** Use CAD software (e.g., SolidWorks, Fusion 360, Tinkercad) to design the 3D model.

- Ensure the model is manifold (watertight) with no holes or non-manifold edges.
- Save the model in a compatible file format, such as STL or OBJ.

2. ***Exporting the 3D Model:*** Export the 3D model in **STL format** (most common for FDM).

Ensure the STL file has the correct **resolution**:

- High resolution: Smaller triangles, larger file size.
- Low resolution: Larger triangles, smaller file size.

Data Preparation in FDM

3. Slicing the Model: Use slicing software (e.g., Cura, PrusaSlicer, Simplify3D) to convert the 3D model into G-code, which contains instructions for the printer.

Key slicing parameters to configure:

- Layer Height: Thickness of each layer (e.g., 0.1 mm, 0.2 mm).
- Infill Density: Percentage of material inside the object (e.g., 20%, 100%).
- Print Speed: Speed of the extruder (e.g., 50 mm/s).
- Support Structures: Enable for overhangs or complex geometries.
- Build Plate Adhesion: Add a brim, raft, or skirt for better adhesion.
- Temperature Settings: Nozzle and bed temperature based on material.

Data Preparation in FDM

4. Generating G-code:

- The slicing software generates G-code, which contains:
  - Movement commands for the printer (X, Y, Z axes).
  - Extrusion commands (E values for filament flow).
  - Temperature settings.
- Save the G-code file to an SD card, USB drive, or send it directly to the printer.

5. Pre-Print Checks:

- Preview the Layers: Use the slicing software to visualize each layer and check for errors.
- Check for Overhangs: Ensure support structures are added where needed.
- Verify Print Settings: Double-check parameters like layer height, infill, and temperature.

Key Considerations in Data Preparation

1. ***File Formats - STL:*** Most common format for FDM. Represents the surface geometry using triangles. **OBJ**: Supports color and texture information. **3MF**: Modern format that includes metadata and supports multiple objects.
2. ***Orientation and Positioning:*** Orient the model to minimize overhangs and support structures. Position the model on the build plate to ensure stability and adhesion.
3. ***Support Structures:*** Use supports for overhangs greater than 45 degrees. Choose between **tree supports** (less material, easier to remove) or **linear supports** (more stable).
4. ***Infill Patterns:*** Common patterns include **grid**, **honeycomb**, and **triangular**. Adjust infill density based on the object's purpose (e.g., 20% for prototypes, 100% for functional parts).
5. ***Layer Height and Resolution:*** Lower layer heights (e.g., 0.1 mm) produce smoother surfaces but increase print time. Higher layer heights (e.g., 0.3 mm) are faster but result in visible layer lines.

Common Slicing Software and Troubleshooting

Softwares:

Cura: Free, open-source, and user-friendly.

PrusaSlicer: Optimized for Prusa printers but works with others.

Simplify3D: Paid software with advanced features.

MatterControl: Free software with built-in design tools.

Troubleshooting Data Preparation Issues:

- **Gaps in the Model:** Ensure the model is manifold and has no holes.
- **Poor Adhesion:** Adjust bed temperature, use a brim or raft, and level the build plate.
- **Stringing or Oozing:** Optimize retraction settings in the slicer.
- **Warping:** Use a heated bed, enclosure, or adhesive (e.g., glue stick).

Common Slicing Software and Troubleshooting

Softwares:

Cura: Free, open-source, and user-friendly.

PrusaSlicer: Optimized for Prusa printers but works with others.

Simplify3D: Paid software with advanced features.

MatterControl: Free software with built-in design tools.

Troubleshooting Data Preparation Issues:

- **Gaps in the Model:** Ensure the model is manifold and has no holes.
- **Poor Adhesion:** Adjust bed temperature, use a brim or raft, and level the build plate.
- **Stringing or Oozing:** Optimize retraction settings in the slicer.
- **Warping:** Use a heated bed, enclosure, or adhesive (e.g., glue stick).

Numerical Problem

A 3D model is to be printed using FDM with the following parameters: Total height of the object: 50 mm, Layer thickness: 0.2 mm, Print speed: 60 mm/s, Average length of each layer 200 mm. Calculate the total print time.

Solution:

1. **Number of layers**:

$$\text{Number of layers} = \frac{\text{Total height}}{\text{Layer thickness}} = \frac{50}{0.2} = 250 \text{ layers}$$

2. **Time per layer**:

$$\text{Time per layer} = \frac{\text{Length of layer}}{\text{Print speed}} = \frac{200}{60} = 3.33 \text{ seconds}$$

3. **Total print time**:

$$\text{Total print time} = \text{Number of layers} \times \text{Time per layer} = 250 \times 3.33 = 832.5 \text{ seconds}$$

Convert to minutes:

$$\frac{832.5}{60} = 13.875 \text{ minutes}$$

Numerical Problem

An FDM printer has a nozzle diameter of 0.4 mm and prints at a speed of 50 mm/s. The layer height is 0.2 mm. Calculate the volumetric extrusion rate in mm^3/s.

Solution:

1. **Cross-sectional area of extrusion**:

$$\text{Area} = \pi \times \left(\frac{\text{Nozzle diameter}}{2}\right)^2 = \pi \times \left(\frac{0.4}{2}\right)^2 = 0.1257\ \text{mm}^2$$

2. **Volumetric extrusion rate**:

$$\text{Extrusion rate} = \text{Area} \times \text{Print speed} = 0.1257 \times 50 = 6.285\ \text{mm}^3/\text{s}$$

Numerical Problem

A spool of filament has a diameter of 1.75 mm and weighs 1 kg. The density of the material (PLA) is 1.25 g/cm³. Calculate the total length of the filament on the spool.

Solution:

1. **Volume of filament**:

$$\text{Volume} = \frac{\text{Mass}}{\text{Density}} = \frac{1000}{1.25} = 800 \text{ cm}^3$$

2. **Cross-sectional area of filament**:

$$\text{Area} = \pi \times \left(\frac{\text{Diameter}}{2}\right)^2 = \pi \times \left(\frac{1.75}{2}\right)^2 = 2.405 \text{ mm}^2$$

Convert to cm^2:

$$2.405 \text{ mm}^2 = 0.02405 \text{ cm}^2$$

3. **Length of filament**:

$$\text{Length} = \frac{\text{Volume}}{\text{Area}} = \frac{800}{0.02405} = 33,264 \text{ cm}$$

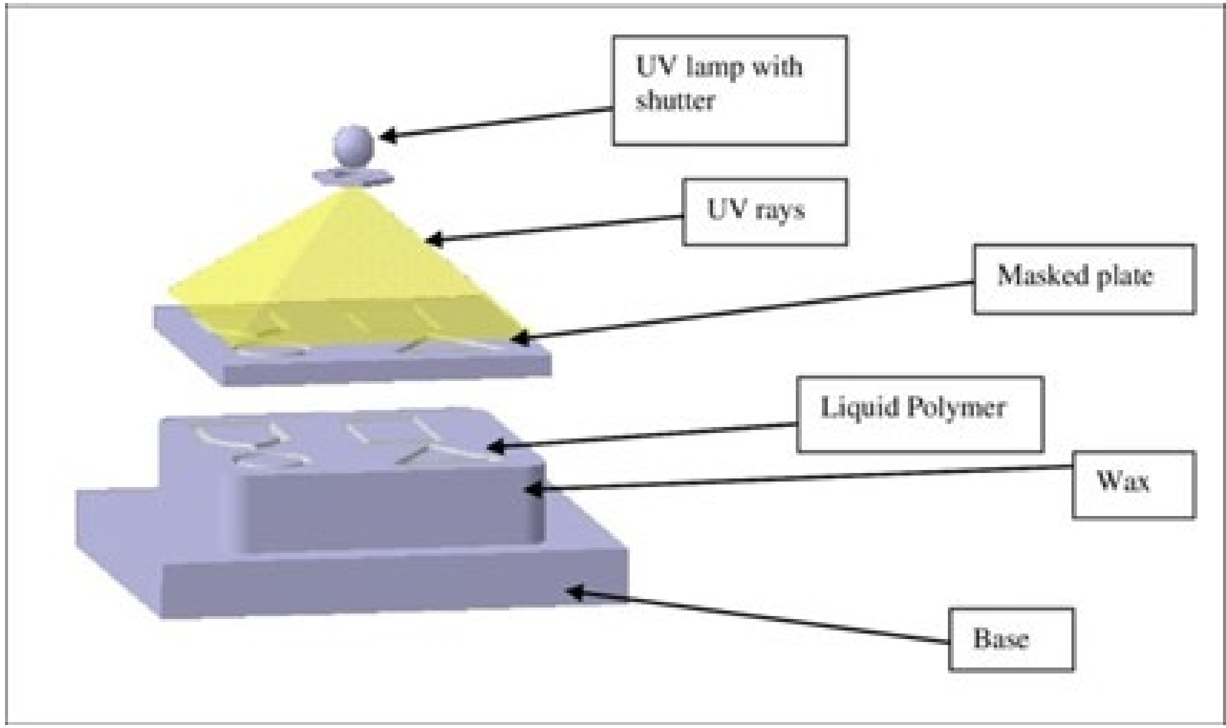
Convert to meters:

$$33,264 \text{ cm} = 332.64 \text{ meters}$$

Solid Ground Curing (SGC)

Solid Ground Curing (SGC) is a rapid prototyping and additive manufacturing technology used to create three-dimensional objects layer by layer. It was developed by Cubital Ltd. in the 1980s.

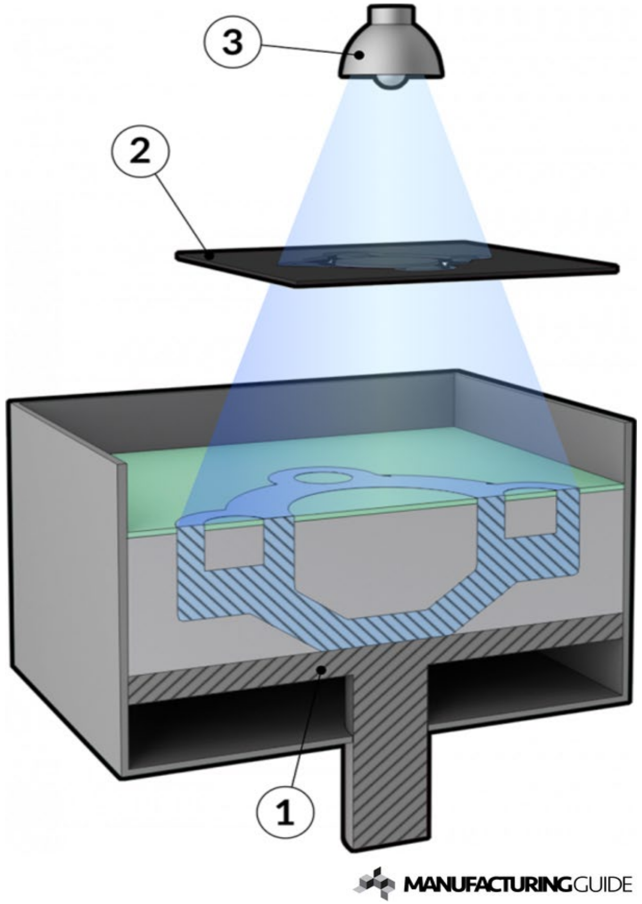
It is a photopolymer-based additive manufacturing process where a photosensitive resin is hardened layer-by-layer using UV light passed through a mask, allowing for high-throughput production of multiple parts simultaneously



Solid Ground Curing (SGC)

Objects built using liquid photopolymer, i.e. plastic which reacts to light. An adjustable construction platform [1] is initially placed in its highest position, only covered by a thin layer photopolymer.

CAD model first layer is drawn on a glass plate [2]. The surface to be solidified, is transparent and the rest of the glass sheet are covered with color using an electrostatic process similar to the laser printing. The glass plate is then placed between the canvas with liquid plastic and UV spotlight [3] which then lights up. Areas where the UV light passes through the glass plate and hits the liquid solidifies, while non-illuminated surface remains liquid. The glass plate is removed to be colored according to the next layer, and the non-solidified photopolymer is sucked away from the canvas.



Water-soluble liquid wax is spread over the work to fill and solidify in the cavities created by the non-solidified resin. The surface is then milled to obtain the required surface finish and thickness. New photopolymer is spread out and the process is repeated with a newly stained glass plate for each layer.

Solid Ground Curing (SGC)

Key Features:

- High Throughput: SGC allows for the simultaneous production of multiple parts, making it suitable for batch production and rapid prototyping.
- Wax Support: Wax replaces liquid resin in non-part areas, providing support for each layer and allowing for complex geometries.

Applications:

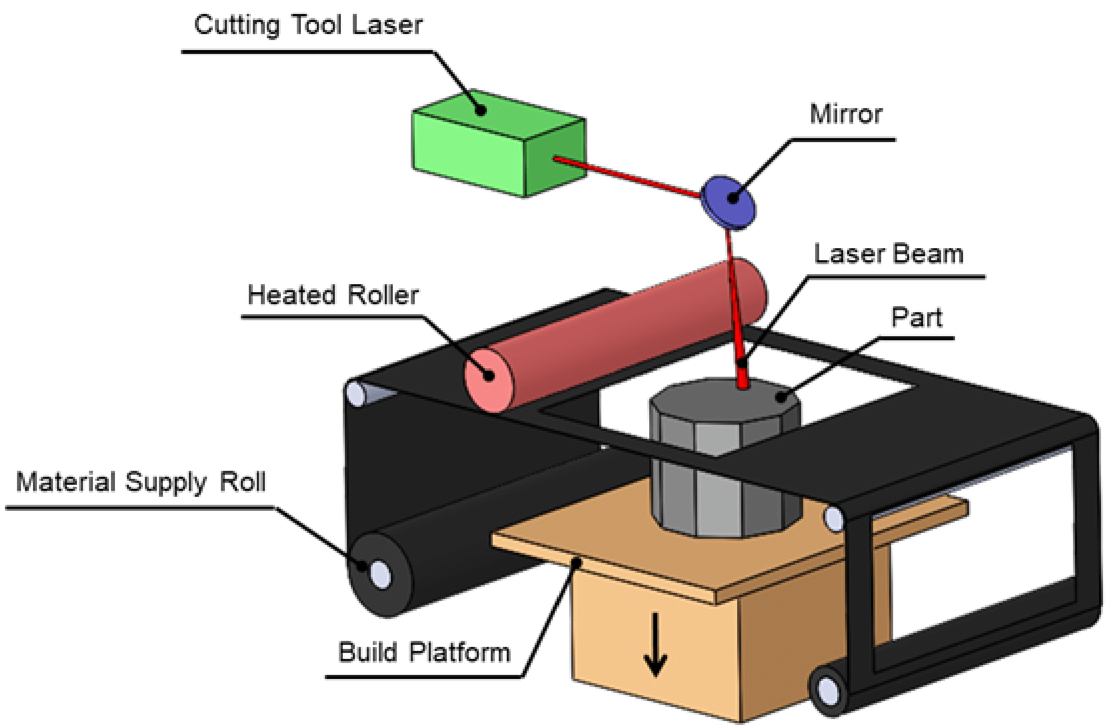
- SGC is used for producing models, prototypes, patterns, and production parts.
- It is particularly well-suited for applications requiring high-throughput production and complex geometries.

Limitations:

- It is a complex and expensive process, which has limited its widespread adoption compared to other additive manufacturing technologies like stereolithography (SLA) or fused deposition modeling (FDM).

Laminated Object Manufacturing (LOM)

Laminated Object Manufacturing (LOM) is an additive manufacturing (3D printing) technology that creates 3D objects by layering and bonding sheets of material, typically paper, plastic, or metal foil. The process involves cutting, stacking, and laminating these sheets to form a solid object.



LOM technology uses adhesive-coated paper, plastic, or metal laminates as a 3D printing medium. These sheets of material are glued together layer-by-layer and cut into shape using a knife or with laser cutting.

Key Steps in LOM

1. ***Material Preparation:*** A roll or stack of sheet material (e.g., paper, plastic, or metal foil) is fed into the machine. The sheet material is coated with a heat-activated adhesive on one side.
2. ***Layer Deposition:*** A new layer of sheet material is positioned onto the build platform or the previous layer. A heated roller or laminator applies heat and pressure to bond the new layer to the stack below, activating the adhesive.
3. ***Cutting the Layer:*** A laser or blade cuts the outline of the current cross-sectional layer based on the 3D CAD model data. The laser or blade also cuts a grid or crosshatch pattern in the excess material surrounding the part. This excess material acts as a support structure during the build process.
4. ***Layer Stacking:*** After cutting, the build platform lowers, and a new sheet of material is fed onto the stack. The process repeats: bonding, cutting, and stacking layers until the entire object is complete.
5. ***Excess Material Removal:*** Once all layers are bonded and cut, the excess material (support structure) is removed manually or mechanically. The final object is revealed, often requiring additional finishing, such as sanding or sealing, to improve surface quality.

Key Components of LOM

- ***Sheet Material:*** Typically paper, plastic, or metal foil, pre-coated with adhesive.

- ***Heated Roller:*** Bonds each new layer to the stack using heat and pressure.

- ***Cutting Tool:*** A laser or blade that precisely cuts the layer contours and support grid.

- ***Build Platform:*** Moves downward after each layer to accommodate the next sheet.

Advantages and Disadvantages of LOM

Advantages of LOM:

- ***Cost-Effective:*** Uses inexpensive materials like paper.
- ***Fast Build Times:*** Suitable for large parts due to rapid layer bonding.
- ***No Need for Supports:*** Excess material acts as a built-in support structure.
- ***Durable Parts:*** Laminated objects can be strong and stable.

Disadvantages of LOM:

- ***Material Waste:*** Generates significant waste from excess material.
- ***Limited Material Options:*** Primarily paper or plastic, limiting applications.
- ***Post-Processing:*** Requires manual removal of excess material and finishing.
- ***Lower Accuracy:*** Compared to other additive manufacturing methods like SLA or SLS.

Applications of LOM

- **Prototyping**: Quick and cost-effective for large prototypes.
- **Sand Casting Patterns**: Used in foundries for creating molds.
- **Visual Models**: For conceptual design and presentation purposes.






Thank You