Competency-Based Learning Materials (CBLMs)
on
3D Printing Technology
Light Engineering Sector

Skills for Industry Competitiveness and Innovation Program (SICIP)
Finance Division, Ministry of Finance

Contents

Module-1: Understanding the Fundamentals of 3D Printing Technology	**3**
1.1 Learning outcome: Interpret overview of 3D Printing Technology	3
1.2 Learning outcome: Identify FDM Printing Technology	21
1.3 Learning outcome: Identify SLA Printing Technology	35
1.4 Learning outcome: Recognize the role of slicer software in 3D printing	46
Module-2: Performing 3d Modelling Using CAD Software	**52**
2.1 Learning outcome: Install CAD software	52
2.2 Learning outcome: Carry out 2D sketching for design creation	61
2.3 Learning outcome: Design 3D models using feature and surface tools	102
2.4 Learning outcome: Assemble 3D models	132
Module-3: Performing Fused Deposition Modelling (FDM) Printing	**145**
3.1 Learning outcome: Prepare the FDM printer for operation	145
3.2 Learning outcome: Configure print settings and parameters using slicer	157
3.3 Learning outcome: Monitor the 3D printing process	201
3.4 Learning outcome: Perform post-processing of printed parts	206
Module-4: Performing Masked Stereolithography (MSLA) Printing	**211**
4.1 Learning outcome: Prepare the MSLA printer for operation	211
4.2 Learning outcome: Configure print settings and parameters using slicer	219
4.3 Learning outcome: Monitor the 3D printing process	241
4.4 Learning outcome: Perform post-processing of printed parts	245
Module-5: Implementing Design Principles for Additive Manufacturing (DfAM)	**249**
5.1 Learning outcome: Understand the principles and advantages of DfAM	249
5.2 Learning outcome: Apply DfAM Principles to Enhance Product Design	261
Module-6: Perform Prototyping Using 3D Scanning Technology	**265**
6.1 Learning outcome: Conduct 3D scanning of physical objects	265
6.2 Learning outcome: Perform post processing of scanned files	305
6.3 Learning outcome: Print prototype	329
Module-7: Performing Basic Maintenance of 3D Printer	**333**
7.1 Learning outcome: Troubleshoot 3D printer issues	333
7.2 Learning outcome: Conduct preventive maintenance of the 3D printer	343

Module-1: Understanding the Fundamentals of 3D Printing Technology

Module descriptor:
This unit covers the knowledge, skills and attitudes required to understand the fundamentals of 3D printing technology. It specifically includes the tasks of interpreting overview of 3D printing technology, identifying FDM printing technology, identifying SLA printing technology, and recognizing the role of slicer software in 3D printing.

Nominal Duration: 10 hours

Learning outcome:
7.1 Interpret overview of 3D Printing Technology
7.2 Identify FDM Printing Technology
7.3 Identify SLA Printing Technology
7.4 Recognize the role of slicer software in 3D printing

1.1 Learning outcome: Interpret overview of 3D Printing Technology

Learning Objectives:

At the end of this section, a trainee will be able to know –

1. Definition of 3D printing technology
2. Brief history of the development of 3D printing technology
3. Basic steps of 3D printing process
4. Typical Materials Used in 3D Printing
5. Differences between additive and subtractive manufacturing
6. Common applications of 3D printing
7. Selection criteria for choosing an appropriate printing technology
8. Methods of 3D printing technology

1.1.1 Information sheet: Fundamentals of 3D Printing Technology

Definition of 3D Printing

3D Printing or, Additive manufacturing (AM) is defined by the ASTM society as a process of joining (adding) materials to make objects from 3D model data, usually layer upon layer, as opposed to subtractive manufacturing methodologies. Its early root was Layer manufacturing or LM, Rapid manufacturing or RM, Rapid tooling, Rapid prototyping or RP, Laser rapid prototyping etc.

Brief History of Development of 3D Printing

1981 – **Hideo Kodama** came up with a layer-by-layer approach for manufacturing, using a photosensitive resin that was polymerized by UV light.

1986 – **Chuck Hull** was granted the patent for first ever commercialized 3D printer for **Stereolithography** technique at 3D Systems Corporation.

1989 – **Carl Deckard** filed a patent for **Selective Laser Sintering (SLS)** RP process.

1989 – **Scott Crump** filed patent for **Fused Deposition Modelling (FDM)** which was issued to Stratasys.

2004 – **Dr. Adrian Bowyer** conceived the **RepRap** (Replicating Rapid-prototyper) concept of an open source, self-replicating 3D printer

2009 – First commercially available **BfB RapMan 3D printer** based on the RepRap concept was offered for sale

3D Printing Process Steps

Generally, the 3D printing process involves following steps -

1. **Design a 3D model**
   - Create a 3D digital model using CAD (computer-aided design) software, or find an existing design online. Ensure the model is manifold (watertight) and check for any errors that would prevent it from printing correctly.
2. **Export and prepare the file**
   - Export the model as an STL, OBJ, or 3MF file, which are standard file formats for 3D printing.
3. **Slice the model**
   - Open the 3D model in a slicing software (a slicer) and use the slicer to prepare the 3D model for printing. This software converts the 3D model into a series of thin, horizontal layers and generates a toolpath for the printer.
   - The output is a printing program (G-code file or any readable file) that must be compatible with the printer. This output file contains the specific instructions for the 3D printer, including temperature, print speed, UV power, exposure time etc.
4. **Set up the printer**
   - Load the chosen printing material into the printer.
   - Clean and level the build platform to ensure the first layer adheres properly.
5. **Print the part**
   - Transfer the printing program to the printer, usually via a USB drive or network connection. And start the printing process. Monitor the printing process to ensure it is running smoothly.
6. **Post-processing**
   - Once printing is complete, remove the finished object from the build platform.
   - Post-processing can be of different kinds depending on the printing technology, including removing supports, washing and curing, de-powdering, heat treatment or machining etc.
   - Perform any additional finishing steps, such as sanding, painting, or assembly, to achieve the desired final product.
7. **Inspection and application**
   - The final step is to validate the printed product by checking whether the product meets the specifications by visual, dimensional, or functional inspection.

Typical Materials Used in 3D Printing

	Amorphous	Semi-crystalline	Thermoset	Material extrusion	Vat polymerization	Material jetting	Powder bed fusion	Binder jetting	Sheet lamination	Directed energy deposition
ABS [Acryonitrile Butadiene Styrene]	X			X						
Polycarbonate	X			X						
PC/ABS Blend	X			X						
PLA [Polylactic Acid]	X			X						
Polyetherimide (PEI)]	X			X						
Acrylics			X		X	X				
Acrylates			X		X	X				
Epoxies			X		X	X				
Polyamide (Nylon) 11 and 12		X					X			
Neat		X					X			
Glass filled		X					X			
Carbon filled		X					X			
Metal (Al) filled		X					X			
Polymer bound	X	X		X						
Polystyrene	X						X			
Polypropylene		X					X			
Polyester ("Flex")							X			
Polyetheretherkeytone (PEEK)		X		X			X			
Thermoplastic polyurethane (Elastomer)				X			X			
Chocolate		X		X						
Paper									X|	
Aluminum alloys							X	X	X	X
Co–Cr alloys							X	X		X
Gold							X			
Nickel alloys							X	X		X
Silver							X			
Stainless steel							X	X	X	X
Titanium, commercial purity							X	X	X	X
Ti–6Al–4V							X	X	X	X
Tool steel							X	X		X

Differences Between 3D Printing and Subtractive Manufacturing

Strength of 3D Printing

- 3D printing can produce structures, such as complex or hollow interior sections, that are impossible to produce using non-additive processes.

- With the right printer, anything that can be designed in a CAD system can be 3D printed.
- It's possible to produce parts using multiple different materials.
- 3D printing also generates less waste than subtractive processes, since material doesn't need to be cut away and discarded.

Weakness of 3D Printing

- 3D printed products don't have much strength, heat resistance, or surface finish required for final parts. Additive processes generally result in tiny pores in surfaces. These weaken the part and can lead to contamination. That's why 3D printed products require additional processing after printing.
- These processes are applicable to a narrow range of materials.

Strength of Subtractive Manufacturing

- CNC machining can produce stronger parts with much better tolerances and smoother finishes than 3D printing techniques. Additionally, extremely durable metal parts can be produced using CNC machining.
- These processes can efficiently handle a wide variety of materials.

Weakness of Subtractive Manufacturing

- CNC machining can require substantial setup time. For this reason, subtractive manufacturing may be expensive for anything but high quantities of parts.
- Wastage of material is high.
- Complex or hollow interior sections are difficult to produce.

Applications and Impacts

Aerospace and automotive

- *Rapid prototyping:* Creating components for cars and aircraft to test design and functionality.
- *Final part production:* Manufacturing complex and lightweight parts for vehicles and spacecraft.
- *Supply chain resilience:* Producing spare parts locally, reducing the need for long supply chains.

Healthcare and medicine

- *Prosthetics and orthotics:* Creating customized and affordable prosthetics.
- *Surgical planning:* Printing models of organs or bones to help surgeons plan complex procedures.
- *Dental applications:* Manufacturing dental crowns, aligners, and other tools.
- *Bioprinting:* Research into creating biological structures and tissues.

Manufacturing and industrial

- *Prototyping:* Quickly and cost-effectively creating prototypes for product development.
- *Custom tooling:* Producing jigs, fixtures, and other tools to optimize production.
- *End-use parts:* Manufacturing functional, final parts for various industrial applications.

Construction

- *Architectural models:* Creating detailed scale models of buildings and projects.
- *Building construction:* 3D printing entire houses and structures using materials like concrete.

Consumer goods and education

- *Personalized products:* Creating custom gifts, jewelry, and food items.
- *Apparel:* Experimenting with fashion items like shoes, and dresses.
- *Educational tools:* Providing hands-on learning experiences and creating models for students.

Selection Criteria for Choosing an Appropriate Printing Technology

- What properties are required in final parts
- What materials are to be used along with availability and demand
- Printing cost vs market price
- What expertise is required to print
- Does the manufacturer or distributer of printer offer technical-support

1.1.2 Information sheet: Methods of 3D Printing Technology

Main Methods of 3D Printing

Seven main technologies of 3D Printing or Additive Manufacturing are –

1. **Binder Jetting**
2. **Directed Energy Deposition**
3. **Material Extrusion**
4. **Material Jetting**
5. **Powder Bed Fusion**
6. **Sheet Lamination**
7. **Vat Photo Polymerization**

The following are some common terms that are used when discussing about these processes.

Build chamber: the enclosed location within the 3D printer where the parts are fabricated.
Build platform/ surface: a base which provides a surface upon which the building of the part is started and supported throughout the build process.
Build space/ build volume: the total usable volume available in the machine for building parts.

Binder Jetting (BJT)

ISO/ASTM definition: "binder jetting, an additive manufacturing process in which a liquid bonding agent is selectively deposited to join powder materials."

It's also known as –

- ColorJet Printing or CJP (3D Systems Corporation)
- Digital Metal® (Höganäs AB)
- Ink Jet Printing or IJP
- Multi Jet Fusion™ (Hewlett-Packard Development Company)

First Used: In 1989, researchers at Massachusetts Institute of Technology (MIT)

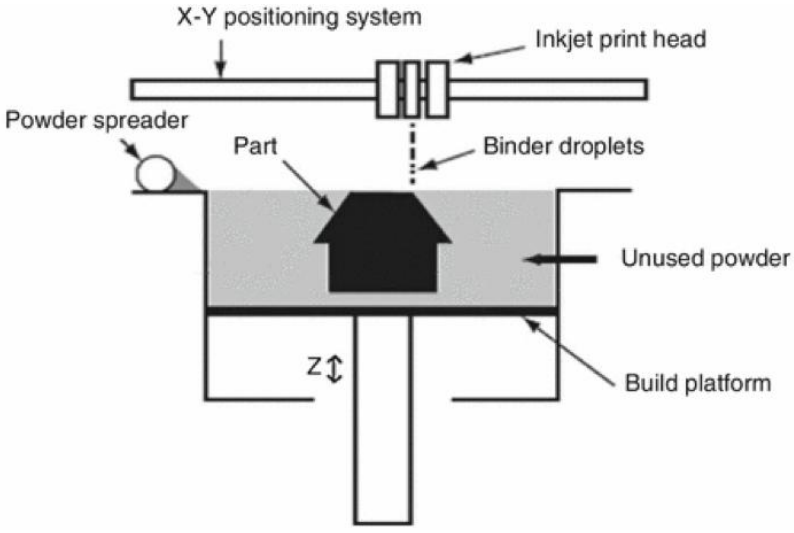


Figure 1.1: Example of a binder jetting system

Process: First, a thin layer of loose powder between 0.05mm and 0.5mm is smoothly spread flat over a build platform. Then, a traditional inkjet print head moves back and forth over this powder and deposits a binding material rather than an ink onto the areas that are to be made solid. This clear or coloured binder is initially a liquid and causes the powder material to bind together usually involving a chemical reaction.

This chemical reaction depends on the method and materials used and occurs either when the binder comes into contact with the raw powder, comes in contact with the air and evaporates, contacts another chemical that is mixed into the powder or is activated by heat. In the next stage of printing the build platform changes height, a new layer of powder is deposited, and the process is repeated layer by layer until the part is complete.

Advantages:

- Materials and binders used can be very inexpensive.
- Parts can be also created very quickly.
- Print heads can also be combined and different colours can be added to the binders in order to create full-colour 3D printed parts.
- Due to laying down material over the entire build platform at once, there is no need for support structures.
- It is used in creating sand cores and molds for casting and allows the mass production of traditional castings using AM technology.

Disadvantages:

- Printed parts are very weak and not very dense, especially when no external energy like heat is used to bind parts. These green parts generally need to be post-processed by infusing them with another material like Cyanoacrylate glue (super glue) or bronze to give the final part more strength.
- De-powdering a part (taking the final part out of the powder bed build area) can be very messy.
- As the powder is very fine and can be an inhalation hazard; special respiratory equipment is usually needed.

Materials Used:

Traditionally a gypsum powder and water/alcohol mix were some of the very first powder/binder combinations, but now a variety of materials can be printed such as metals like stainless steel alloys and copper, plastics, and glass.

Directed Energy Deposition (DED)

ISO/ASTM definition: “directed energy deposition, an additive manufacturing process in which focused thermal energy is used to fuse materials by melting as they are being deposited. "Focused thermal energy" means that an energy source (e.g., laser, electron beam, or plasma arc) is focused to melt the materials being deposited.”

It’s also known as –

- ➢ Direct Laser Deposition or DLD
- ➢ Direct Metal Deposition or DMD® (DM3D Technology, LLC)
- ➢ Electron Beam Additive Manufacturing or EBAM ™ (Sciaky, Inc.)
- ➢ Electron Beam Freeform Fabrication or EBF3
- ➢ Focused Ion Beam Direct Writing or FIBDW
- ➢ Laser Chemical Vapor Deposition or LCVD
- ➢ Laser Engineered Net Shaping or LENS® (Sandia National Labs)
- ➢ Wire Arc Additive Manufacturing or WAAM

First Used: From 1994-1997 in New Mexico USA, Sandia National Laboratories developed a new AM technology which they called LENS.

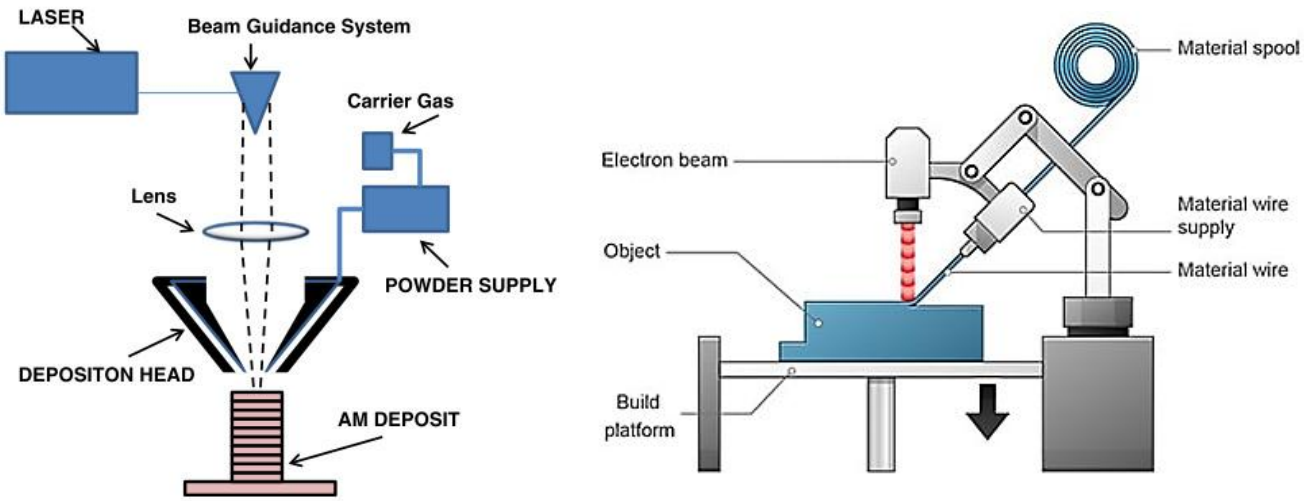


Figure 1.2: Example of a directed energy deposition system, LENS (Left) and EBF3 (Right)

Process: Directed energy deposition processes look very different between each method, but the premise is the same in each case. First, there is a focused area of intense energy, usually a thermal energy source like a laser, electron beam, or TIG welding torch. A feedstock material is introduced layer by layer into that intense energy area causing it to bond to the surrounding material. This feedstock can be introduced either through blowing powder into that area (cold spray process), by pushing a solid wire into that area, or by introducing a special gas into the build chamber. Into the intense energy area, the powder or wire melts and binds to the previous layers and then cools and solidifies.

Advantages:

- With the powder blowing method, powders can be changed or mixed mid-build thus creating multiple materials in a build, even creating a gradient between two different materials in the same build.
- Directed energy deposition processes are able to aid in the repair of damaged parts or to develop near-net shape parts that normally could not have been repaired using traditional methods. These can roughly add materials which are then cleaned up and machined to tolerance afterwards.



Figure 1.3: Near-net shapes are additively produced and then finished using subtractive method

Disadvantages:

- These processes require material to be deposited one spot at a time, and cannot do entire layers all at one time which is time-consuming.
- Build speeds are limited and slow.
- These processes do not lend themselves to create support structures, which limits the printing of certain geometries with overhangs.

- Resolutions are generally very low and have a rough surface finish that may need post-processing like machining to get tight tolerances.

Materials Used:
Almost any weldable metal can be additively manufactured using DED, including Aluminum alloys, Inconel, Niobium alloys, Stainless Steel and tool steel, Tantalum, Titanium alloys, Tungsten, Copper, Cobalt-Chrome superalloys, Nickel superalloys etc.

Material Extrusion (MEX)

ISO/ASTM definition: "material extrusion, an additive manufacturing process in which material is selectively dispensed through a nozzle or orifice."

It's also known as –

➢ Fused Deposition Modeling or FDM® (Stratasys Inc.)
➢ Fused Filament Fabrication or FFF

First Used:
In 1988, Scott Crump invented a new AM process based on candle wax and a hot glue gun while making a toy for his daughter in the kitchen. The next year he started the company Stratasys, which became one of the largest AM companies in the world.

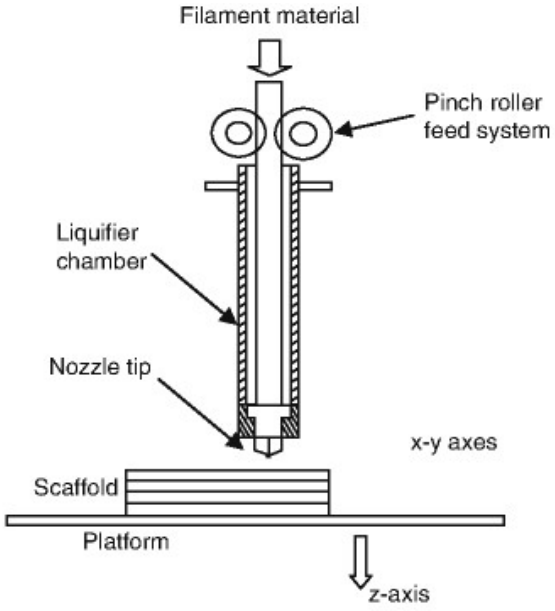


Figure 1.4: Example of a material extrusion system

Process:
The core principle of this technology is that any material in filament form is heated and then pushed through a nozzle and used to print layer by layer until the product is completed. The technology is scalable and is only limited by nozzle size and supporting machine structure. This supporting machine structure can take many different shapes such as a delta robot configuration or multi-jointed robot arms.

Advantages:
- The extensive range of materials can be used.
- The mechanics of this type of printing are fairly simple and easy to modify especially due to the availability of open source designs.
- Multiple print heads or extruders can be installed on the same machine thus enabling multi-material multi-color printing.

Disadvantages:

- This process is slow as only one nozzle operates at a time.
- Parts generally have anisotropic material properties, and the same part can exhibit different strengths depending on how it was printed.
- Layer heights are generally larger than other AM processes and are thus more visible and contribute to a higher surface roughness.
- Support materials and structures need to be used. Removing these supports is either a manual and labor-intensive process, or a process which requires dissolving and rinsing of parts in a chemical bath of some sort.

Material Used:

These printers use a plastic like ABS (Acrylonitrile butadiene styrene), PLA (Polylactic acid), PETG (Polyethylene terephthalate glycol), Nylon, TPU (Thermoplastic Polyurethane), PC (Polycarbonate), PVA (Polyvinyl Alcohol), HIPS (High Impact Polystyrene), ASA (Acrylic Styrene Acrylonitrile), PP (Polypropylene) or PEEK (Polyether ether ketone) etc., any material that can be pushed through a nozzle (heated or not). Other examples include cement, chocolate, ceramic pastes or slurries, metal clays and metal filled plastics (Stainless steel, Tool steel, Copper, Nickel alloys etc.), ground-up and blended food, or even biocompatible organic cellular scaffolding gel.

Material Jetting (MJT)

ISO/ASTM definition: "material jetting, an additive manufacturing process in which droplets of build material are selectively deposited."

It's also known as –

- ➢ Drop-On Demand or DOD
- ➢ Laser-Induced Forward Transfer or LIFT
- ➢ Liquid Metal Jetting or LMJ
- ➢ Multi-Jet-Printing or MJP (3D Systems Corporation)
- ➢ Nano Particle Jetting™ (XJet) or NPJ
- ➢ Polyjet® (Stratasys Inc)

First Used:

In 1984, Bill Masters patented first material jetting technology and founded the company Perception Systems, Inc.

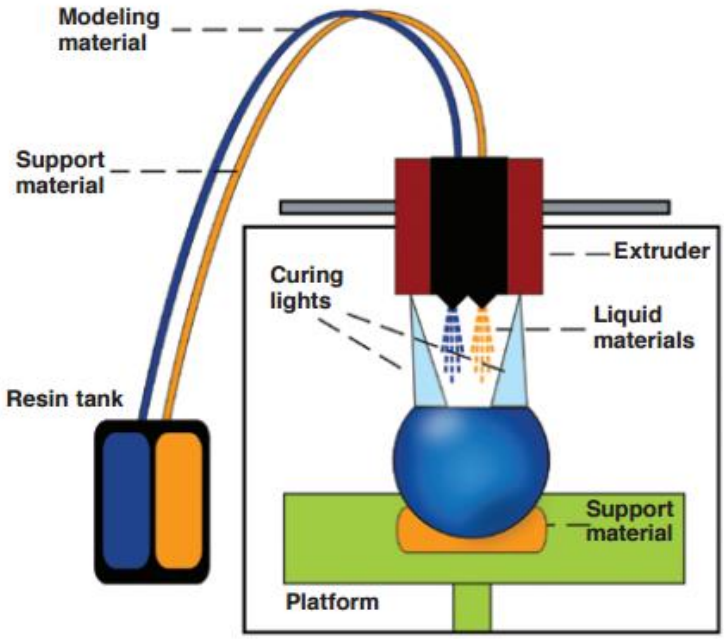


Figure 1.5: Material jetting example setup

Process:

Material jetting is very similar to the type of technology that exists in a standard home-based inkjet printer and is closely related to the binder jetting process. The main difference is that rather than printing ink or a binder, it prints the specific type of material that will make up the final part.

The material is deposited drop by drop in a very precise and fine detailed manner from a print head. The exact mechanism for depositing drops can be categorized under the following three main types, (a) PolyJet technology, (b) Nano Particle Jetting (NPJ), and (c) Drop-On Demand (DOD). Regardless of the drop creation process, once the drop is deposited, it then solidifies either through material cooling, external curing from a UV light source, or by evaporating a liquid transport material by using infrared light/heat.

Advantages:

- One big advantage of this technology is the ability to gang multiple print heads together. Having multiple print heads allows these machines to do unique things, such as print in different colours like traditional ink-jet printers, print faster by printing over the entire build surface in one pass, and print in multiple materials at the same time.
- The surface quality of these parts is usually quite high due to jetting very small droplets. Material jetting can produce details as small as 14 μm.

Disadvantages:

- The build time can be slow due to the nature of jetting very small amounts of material at a time over a small portion of the build area.
- Support structures are also required; thus, one print head is dedicated to jetting only support material.
- Removal of the support material is a manual step requiring one of the following methods: melting or dissolving away of the support material, spraying away the support material manually with water using a high-pressure wash, or removing them by hand.

Material Used:

The materials used in material jetting are alumina (aluminum oxide), zirconia, stainless steel, waxes, and UV-cured resins. The UV-cured resins are generally formulated from modified acrylic monomers. A

growing variety of types of print resin (Standard, Flexible, Simulated Polypropylene, Simulated ABS, Castable, High temperature, Transparent, Medical grade etc.) are available to suit various applications.

Powder Bed Fusion (PBF)

ISO/ASTM definition: "powder bed fusion, an additive manufacturing process in which thermal energy selectively fuses regions of a powder bed."

It's also known as –

- ➢ Selective Laser Melting or SLM
- ➢ Selective Laser Sintering or SLS® (3D Systems Corporation)
- ➢ Direct Metal Laser Sintering or DMLS® (EOS GmbH)
- ➢ Electron Beam Melting or EBM (Arcam AB)
- ➢ Laser Powder Bed Fusion or LPBF

First Used:

Powder bed fusion contains a range of technologies, all of which have core similarities, the main three are SLS, SLM and EBM. These originated from the research at the University of Texas at Austin in the early 1980s which was awarded a patent in 1989.

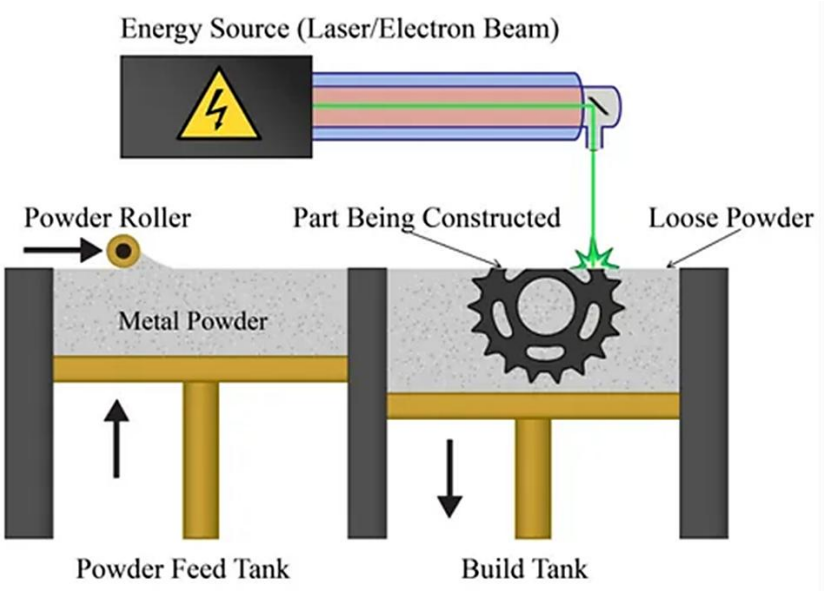


Figure 1.6: Powder Bed Fusion example setup

Process:

Powder bed fusion methods all start with a powder bed, another parallel to binder jetting. A thin layer of loose powder between 0.001mm and 0.2mm with an average of 0.02mm to 0.1mm is smoothly spread flat over a build platform. This layer of powder is then passed over by either a laser or electron beam which supplies significant heat to the powder. The powder is then either partially melted (sintered) or fully melted, to a point where the powder fuses to itself and to the layers below. Then the build platform changes height, a new layer of powder is deposited, and the process is repeated layer by layer until the part is complete. Some of the heat source that causes melting are computer-controlled CO_2 laser, Yb-fiber optic laser, infrared lamp, and electron beam.

Advantages:

- There is a wide range of materials that can be processed from plastic parts like Nylon to a wide range of different metals like copper and Inconel to even ceramics.

- The parts that are fully or even partially melted can have significant strength advantages over non-melted processes as the material properties can be close to that of stock material.
- Depending on the process and material, support structures may not be needed, as the powder bed becomes the support.
- Build speeds can be fast depending on materials and process.
- Processes that use lasers result in a high level of detail and fine features.

Disadvantages:

- All of these processes can only utilize a single material in the final part.
- Layer heights are a function of the powder diameter and thus have a medium resolution in the build direction compared to other processes.
- Printed surface finish is generally rough and will need some post-processing in order to achieve tight tolerances.
- To remove unsolidified material, it needs either air blasting, powder blasting, ultrasonic vibration, or mechanical methods to remove the powder, with certain geometries like deep narrow cavities being especially difficult.

Material Used:

PBF 3D printers support a wide range of materials like ceramic, metal (Aluminum and light alloys, Chrome-Cobalt alloys, Precious metals (gold, silver), Copper and alloys (bronze, brass), Inconel, Steel (mild steel, alloy steel, and stainless steel (316L and 17-4PH)), Titanium and alloys, Nickel and alloys etc.), and plastic (Nylon, Glass-filled nylon, Mineral-filled nylon, Polypropylene, TPU, Polyether ether ketone (PEEK), Polyetherketoneketone (PEKK) etc.), in the form of powder.

Sheet Lamination (SHL)

ISO/ASTM definition: “sheet lamination, an additive manufacturing process in which sheets of material are bonded to form a part.”

It’s also known as –

- ➢ Computer-Aided Manufacturing of Laminated Engineering Materials or CAM-LEM
- ➢ Laminated Object Manufacturing or LOM
- ➢ Plastic Sheet Lamination or PSL (Solidimension Ltd.)
- ➢ Ultrasonic Additive Manufacturing or UAM

First Used:

The LOM process was developed by a company named Helisys and in 1991 they started selling a machine that made 3D parts using rolls of paper and a CO_2 laser.

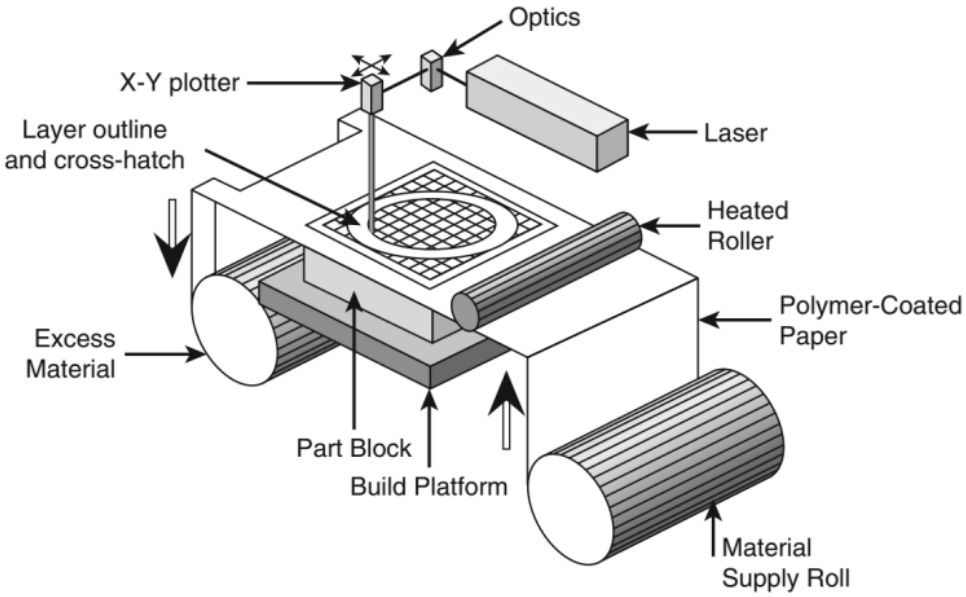


Figure 1.7: Sheet lamination example setup

Process:

This process starts with a single layer of solid material put across the build surface, be it paper, plastic, metal, or ceramic. This layer may or may not be bonded to the previous layers first as it depends on the process. Some processes bond the entire layer to the previous and then cut the 2D slice into the layer. The bonding between each layer occurs either by applying glue to the build surface, through melting a polymer that is embedded within the roll of printing materials, or by fusing layers via an atomic migration.

Advantages:

- LOM can do full-color prints which is relatively inexpensive since the raw material is normal paper, can be infused with materials to increase strength or color saturation.
- UC can print with metal, and has the ability to do multi-metal layers within the overall part.
- No support material is needed since each layer is already solid and can support itself.

Disadvantages:

- The layer height can't be changed without changing the thickness of the sheets of material being used.
- Regardless of the size of part being made, an entire sheet is consumed per layer. That's why the material waste can be high.

Material Used:

Sheet lamination can use a wide variety of materials, such as paper, almost all plastic sheet, ceramic (Alumina, zirconia, zirconia-toughened alumina (ZTA), silica, and a variety of oxide and non-oxide ceramics), metal (Aluminum, copper, stainless steel, titanium), and composites with thermoplastic matrix materials (Carbon Fibre, Fiberglass, Aramid Fibre (i.e. Kevlar), PBO (i.e. Zylon), along with metal fibres like steel, aluminum or titanium).

Vat Photopolymerization (VPP)

ISO/ASTM definition: "vat photopolymerization, an additive manufacturing process in which liquid photopolymer in a vat is selectively cured by light-activated polymerization."

It's also known as –

- ➢ Continuous Liquid Interface Production (CLIP)
- ➢ Stereolithography Apparatus (SLA)
- ➢ Digital light processing (DLP)
- ➢ Masked Stereolithography (MSLA)

First Used:

The first parts were made by Dr. Hideo Kodama of Japan using SLA in 1981.

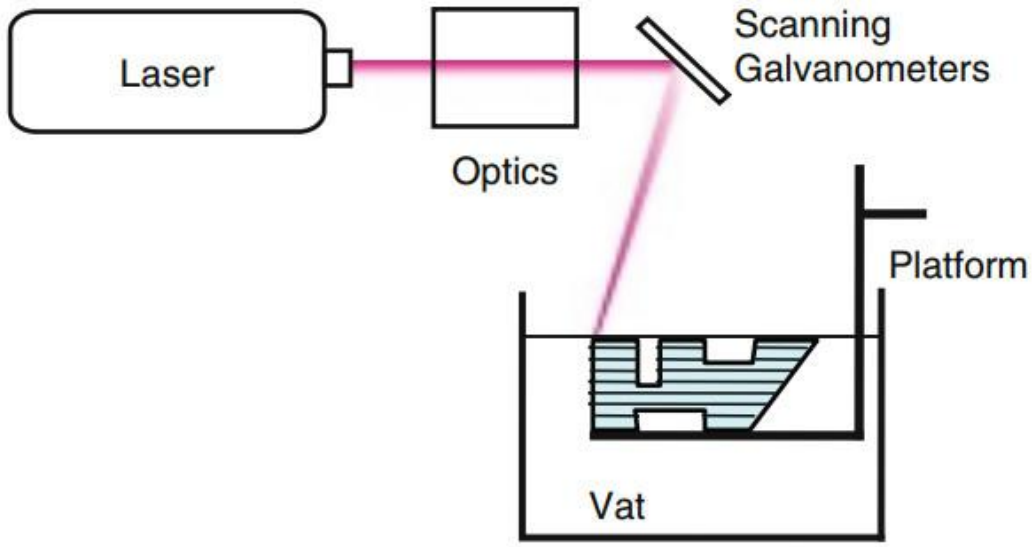


Figure 1.8: Vat Polymerization example setup

Process:

This process involves using a liquid resin as the main type of material. Specifically, this liquid resin has the special property of being able to become solid once it is exposed to light. This liquid resin is held in a container or vat, in which a flat build platform is partially submerged. This platform starts near the surface of the liquid, and then gets exposed to light. This light can be a UV laser (SLA), a digital light processing (DLP) projector, a UV light bulb filtered through an LCD screen (CLIP or MSLA). Once the resin is cured and made solid, the build platform either moves further into the vat, or partially comes out of the vat leaving the solid cured portion just under the surface and then the process is repeated.

After the printing is completed, the printed model must be washed using water, isopropyl alcohol (IPA), or tripropylene glycol monomethyl ether (TPM). Washing process can be done manually or in a washing station. After washing, the model is needed to be cured using UV light or direct sunlight.

Advantages:

- It is capable of very high detail surface finish.
- It can be very fast compared to other processes.
- Printing time is same for same object regardless of the number of parts to be printed at a time.
- Products are relatively stronger.

Disadvantages:

- A limited number of specially produced material can be printed.
- Depending on the geometry of the part, support structures are required and can be very complex and require manual removal afterwards.
- Post-processing is always required.
- Multi-color and multi-material printing are possible.

- Handling is messy and hazardous for health.

Material Used:

Vat Photopolymerization produces parts from UV-cured resins. The UV-cured resins are generally formulated from modified acrylic monomers. A growing variety of print resin are available to suit various applications, such as, Standard resin, Clear resin, Simulated ABS or Tough resin, Simulated Polypropylene or Durable resin, Heat-resistant resin, Flexible resin (rubber-like), Ceramic-filled resin (Rigid), Castable resin, Transparent resin, Medical grade resin etc.

1.1.3 Self-check

Write down the most appropriate answers for the following questions:

1. Define 3D Printing. Name alternative names of 3D Printing.
2. Write down about post processing in 3D printing process.
3. Write down the applications of 3D printing in engineering sectors.
4. Fill up the table given below:

Printing Technology	Form of printing material used	Type of printing material used
Directed Energy Deposition		
Material Extrusion		
Powder Bed Fusion		

1.1.4 Answer Key

1. Answer:

3D printing can be defined as a process of joining (adding) materials to make objects from 3D model data, usually layer upon layer, as opposed to subtractive manufacturing methodologies. Alternative names are Additive manufacturing, Layer manufacturing or LM, Rapid manufacturing or RM, Rapid tooling, Rapid prototyping or RP, Laser rapid prototyping etc.

2. Answer:

Post-processing

- Once printing is complete, remove the finished object from the build platform.
- Post-processing can be of different kinds depending on the printing technology, including removing supports, washing and curing, de-powdering, heat treatment or machining etc.
- Perform any additional finishing steps, such as sanding, painting, or assembly, to achieve the desired final product.

3. Answer:

Aerospace (fabricating low volume complex aerospace parts, aircraft wings, replacement parts in the aerospace industry, fabricating specialized parts, light weight structures, parts with minimal waste)
Automotive (Structural composite components, engine valves, turbocharger, spare and replacement parts)

Manufacturing and industrial (Prototyping, jigs, fixtures, functional parts for various industrial applications)
Construction (Architectural models, entire houses and structures)

4. Answer:

Printing Technology	Form of printing material used	Type of printing material used
Directed Energy Deposition	Powder, Wire	Metal Alloy
Material Extrusion	Filament	Plastic, Composite
Powder Bed Fusion	Powder	Metal Alloy, Plastic

1.2 Learning outcome: Identify FDM Printing Technology

Learning Objectives:

At the end of this section, a trainee will be able to know –

1. Basic components of FDM printer including hot end and extruder
2. Different types of FDM printing methods
3. General working principle of FDM printing
4. Typical Materials Used in FDM printing
5. Different 3D printing file formats
6. FDM printing slicer and slicing parameters

1.2.1 Information sheet: FDM printer and its components

Fused Deposition Modeling or FDM printing is under material extrusion category of 3D printing technology. It was first introduced by Stratasys Inc. In FDM printing technology, the printing material in the form of filament is selectively dispensed or extruded through a heated nozzle or multiple nozzles.

A typical FDM 3D printer always consists of some essential components, which include a frame, motherboard, heated bed/ build plate, extruder and hotend.

Frame: The main structure of the printer.

Motherboard: The control center that converts the 3D model into movements and functions like filament extrusion and temperature control.

Stepper Motors: These motors drive the movement of the printer's axes (X, Y, and Z) and the filament.

Power Supply: Power Supply provides the electrical power for all the components.

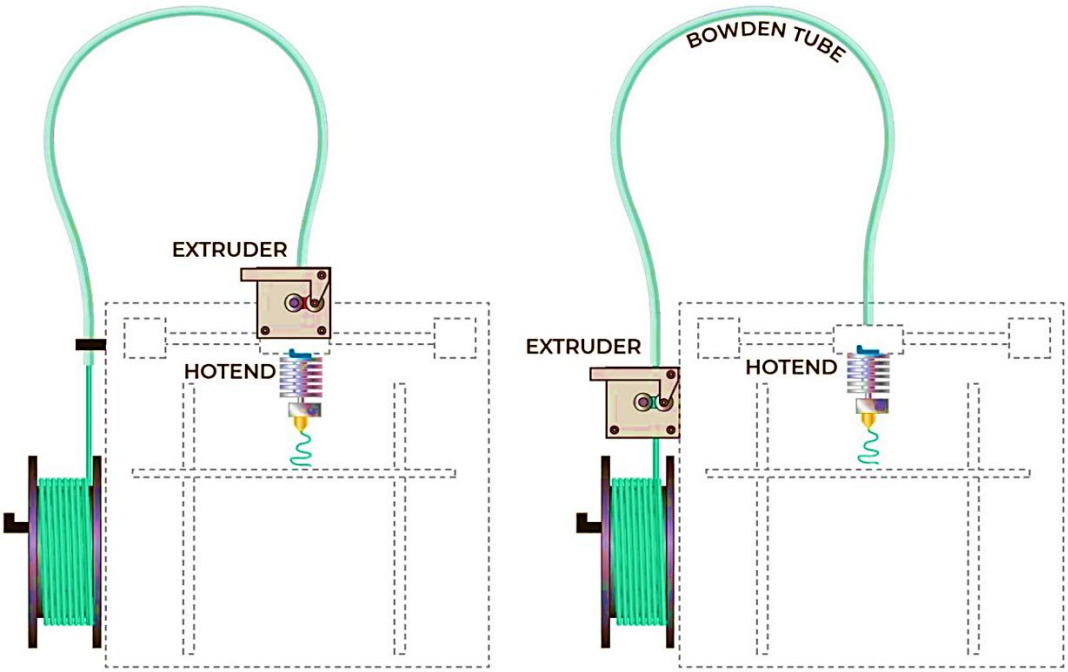
Build Platform/Print Bed/Heated Bed: The heated surface where the printer deposits the material used for printing. A heated bed can help reduce wrapping on large prints and help adhere parts better.

Filament: The raw material, typically a thermoplastic, that is fed into the printer from a spool.

Extruder: The mechanism that pulls the filament from the spool and pushes it toward the hot end. The extruder has mainly two parts; the stepper motor and the feeding system. This is either a Direct-drive extruder or a Bowden tube system.

1. Direct-drive extruder
   - **Mechanism:** The motor that pushes the filament is located directly on the print head, above the hot end.
   - **Advantages:** Provides excellent control over filament, especially for flexible filaments like TPU, due to the short distance between the motor and the nozzle. It can also offer better precision and retraction control.
   - **Disadvantages:** The added weight of the motor on the print head can limit the printer's speed and increase inertia.

2. Bowden tube system
   - **Mechanism:** The motor is separated from the print head and mounted on the printer’s frame. The extruder is connected with the print head via a PTFE tube through which the filament is pushed. This PTFE tube is known as the Bowden tube.
   - **Advantages:** The weight on the print head is significantly reduced, allowing for higher print speeds and reduced ghosting or ringing artifacts on prints.
   - **Disadvantages:** The extra length of the tube can lead to some filament grinding or slipping, and it can be less effective for flexible filaments.



(a) Direct-drive extruder (b) Bowden tube system

Figure 1.9: Different types of extruder setup

Hot End: Hot End melts the filament and extrudes it through a nozzle. A hot end generally consists of following components –

- **Nozzle:** The small tip where the melted filament exits. These nozzles come in various sizes, from as fine as 0.2 mm to as large as 1.2 mm or more, with common sizes being 0.4 mm and 0.6 mm. Smaller nozzles are best for high-detail, low-speed prints, while larger nozzles excel at faster printing of functional or large parts where strength is more important than fine resolution. The choice of nozzle size depends on the desired balance between print speed, detail, and strength.
  Standard size nozzles are only 12-14mm long. This type of nozzles has few models like E3D v5/v6, MK7, MK8, MK10 etc.
  FDM printer nozzles are also categorized by their material, such as brass, stainless steel, hardened steel, and ruby-tipped. The material is chosen based on the filament's properties (e.g., abrasiveness) and applications (e.g., medical and food-contact safe printing).



Figure 1.10: Different types of nozzle

- **Heating Block and Heater Cartridge:** Heats the filament to its melting point.
- **Thermistor:** A sensor that monitors and regulates the temperature.
- **Heat Break:** Prevents heat from traveling up to the cold end.

- **Heat Sink and Cooling Fan:** Cools the extruded plastic to solidify it quickly and improve overhangs and overall print quality.

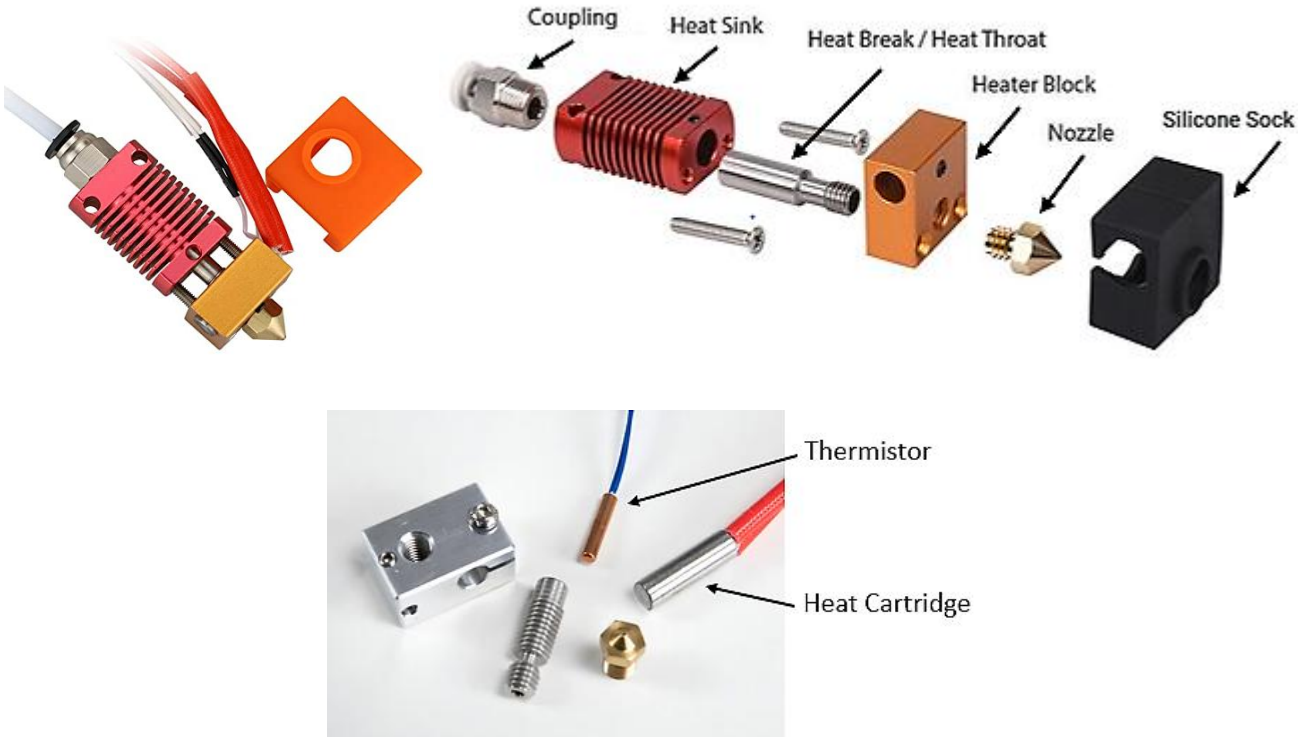


Figure 1.11: Hot end assembly and its components

Below is a typical example of desktop FDM printer (Model: Original Prusa i3 MK3S+, Manufacturer: Prusa Research a.s., Czech Republic).

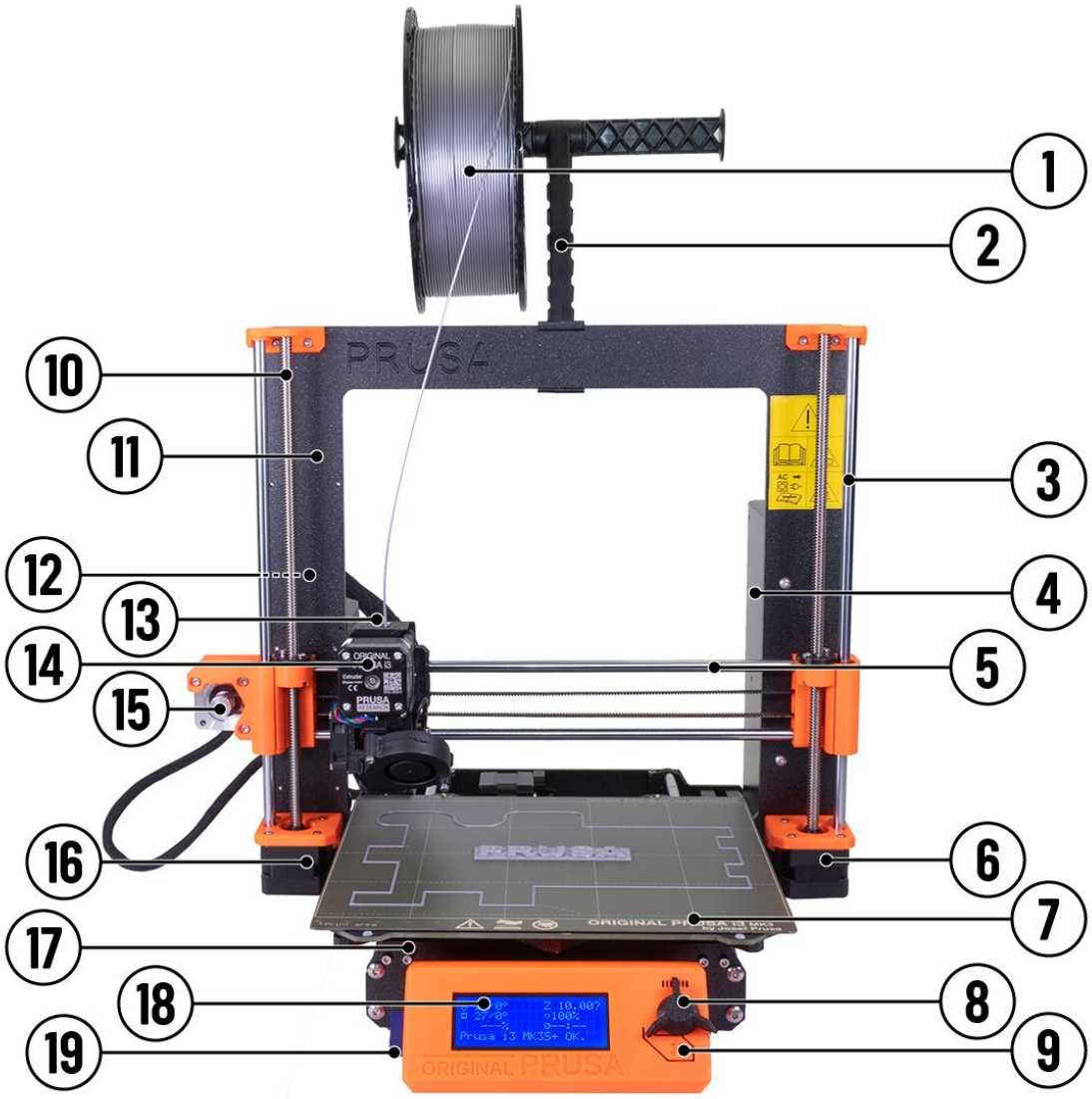


Figure 1.12: Original Prusa i3 MK3S+ printer description: (1) Filament spool (2) Spool holder (3) Z-Axis (4) Power supply (5) X-Axis (6) Z2 Stepper (7) Heatbed with spring steel sheet (8) LCD Knob (9) Reset Button (10) Threaded Rod (11) Printer Frame (12) USB Port (13) Einsy Rambo Board (14) Extruder Stepper (15) X-Stepper (16) Z1 Stepper (17) Y-Axis (18) LCD Panel (19) SD Card Slot

Mechanical and build components

- **Frame:** Milled dural frame for rigidity and stability.
- **Extruder:** Direct-drive extruder with Bondtech gears for reliable filament control.
- **Hot end:** Features an E3D V6 hot end, capable of reaching a maximum temperature of 300°C.
- **Build Plate:** Magnetic build plate with removable PEI spring steel sheets.
- **Nozzle:** Default nozzle is 0.4 mm, but other sizes are supported.
- **Y-Axis:** Aluminum extrusion Y-axis assembly with bearing clips for easier assembly.

Control and electronics

- **Controller:** Einsy Rambo 8-bit board co-developed with Ultimachine, featuring Trinamic 2130 drivers for sensorless homing on the X and Y axes.
- Sensors:
  - **SuperPINDA:** Advanced sensor for high-accuracy, automatic mesh bed leveling.
  - **IR Filament Sensor:** Detects filament run-out and pauses the print.

- **Other:** 3 thermistors and fan RPM sensors are included for monitoring.

- **Power:** Custom 240W Delta power supply with hardware-based Power Panic for recovering from power loss.
- **Connectivity:** Ethernet and Wi-Fi connectivity are available via an optional Raspberry Pi.

Other features

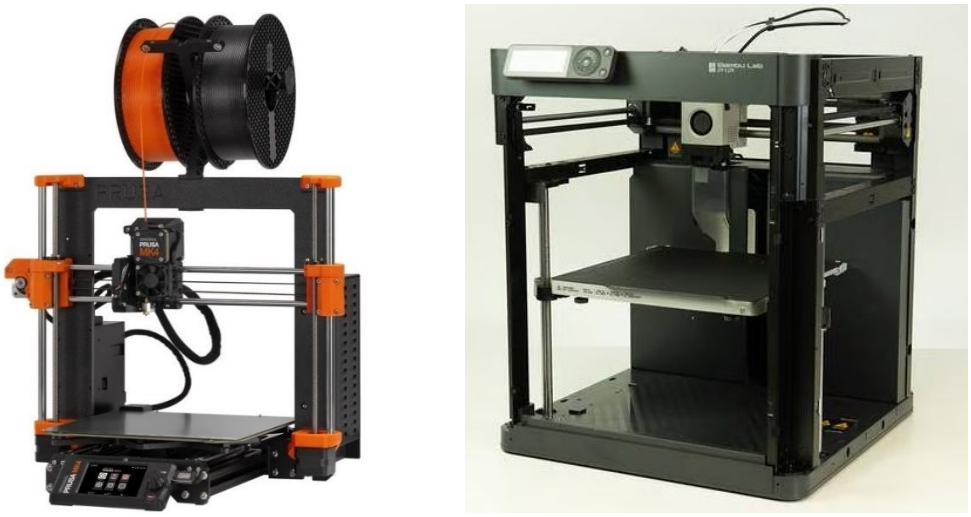
- **Build Volume:** 250 x 210 x 210 mm.
- **Materials:** Supports a wide range of thermoplastics including PLA, PETG, ASA, ABS, PC, Nylon etc.
- **Layer Height:** Ranging from 0.05 to 0.35 mm.
- **Filament:** Uses 1.75 mm diameter filament.

1.2.2 Information sheet: Types of FDM printers

FDM printers are mainly categorized by their movement systems (kinematics). Among them, the most common types are –

Rectilinear/ Cartesian Printer: They use a Cartesian coordinate system (X, Y, Z axes) to move the print head and/or print bed. Variations include:

- **Bed Slinger Printer:** The print bed moves along the Y-axis, and the print head moves along the X and Z axes.
- **CoreXY Printer:** The print bed moves on the Z-axis only, while the print head moves on the X and Y axes, supported by a system of two belts.



(a) Bed Slinger Cartesian Printer (b) CoreXY Cartesian Printer

Figure 1.13: Different Types of Cartesian Printers

Delta Printer: These printers have three arms that move the print head in a triangular or delta-like fashion, while the print bed typically remains stationary. They are often faster and good for printing tall objects.

Polar Printer: These printers use polar coordinates instead of Cartesian, which means their movement is based on radius and angle rather than X, Y, and Z coordinates.



(a) (b)

Figure 1.14: (a) Delta Printer, (b) Polar Printer

SCARA Printer: Also known as Selective Compliance Assembly Robot Arm, this type of printer is ideal for tasks requiring precise movements in an arc, which can be more efficient and faster than the linear movements. SCARA printers feature a printhead attached to a robotic arm powered by two motors. The motors are both connected to linkages that move the printhead around the XY-plane, while another motor controls motion along the Z-axis.



(a) (b)

Figure 1.15: (a) SCARA Printer, (b) Belt Printer

Belt Printer: Concerning the printhead's motion system, the belt printers are essentially CoreXY printers with the frame tilted at a 45-degree angle. Then, the print surface is replaced by a sort of conveyor belt, which moves one way or the other in the combined Z- and Y-axes. This belt can print a single infinitely long piece or it can continuously print many small parts.

1.2.3 Information sheet: Fundamentals of FDM Printing Technology

Working Principle

An FDM 3D printer works by depositing melted filament material over a build platform layer by layer until a part is completed.

FDM uses digital design files that are uploaded to the machine itself and translates them into physical dimensions.

Materials for FDM include polymers such as ABS, PLA, PETG and TPU, which the machine feeds as filament through a heated nozzle.

To operate an FDM machine, one first loads a spool of this thermoplastic filament into the printer. Once the nozzle hits the desired temperature, the printer feeds the filament through an extruder and a printhead (which includes a hot end and a nozzle).

This printhead is attached to a three-axis system that allows it to move across the X, Y and Z axes. The printer extrudes melted material in thin strands and deposits them layer by layer along a path determined by the design. Once deposited, the material cools and solidifies. In most of the cases, fans are attached to the printhead to accelerate cooling.

To fill an area within a layer, multiple passes are required, similar to coloring in a shape with a marker. When the printer finishes a layer, the build platform descends and the machine begins work on the next layer. In some machine setups, the printhead moves up. This process repeats until the part is finished.

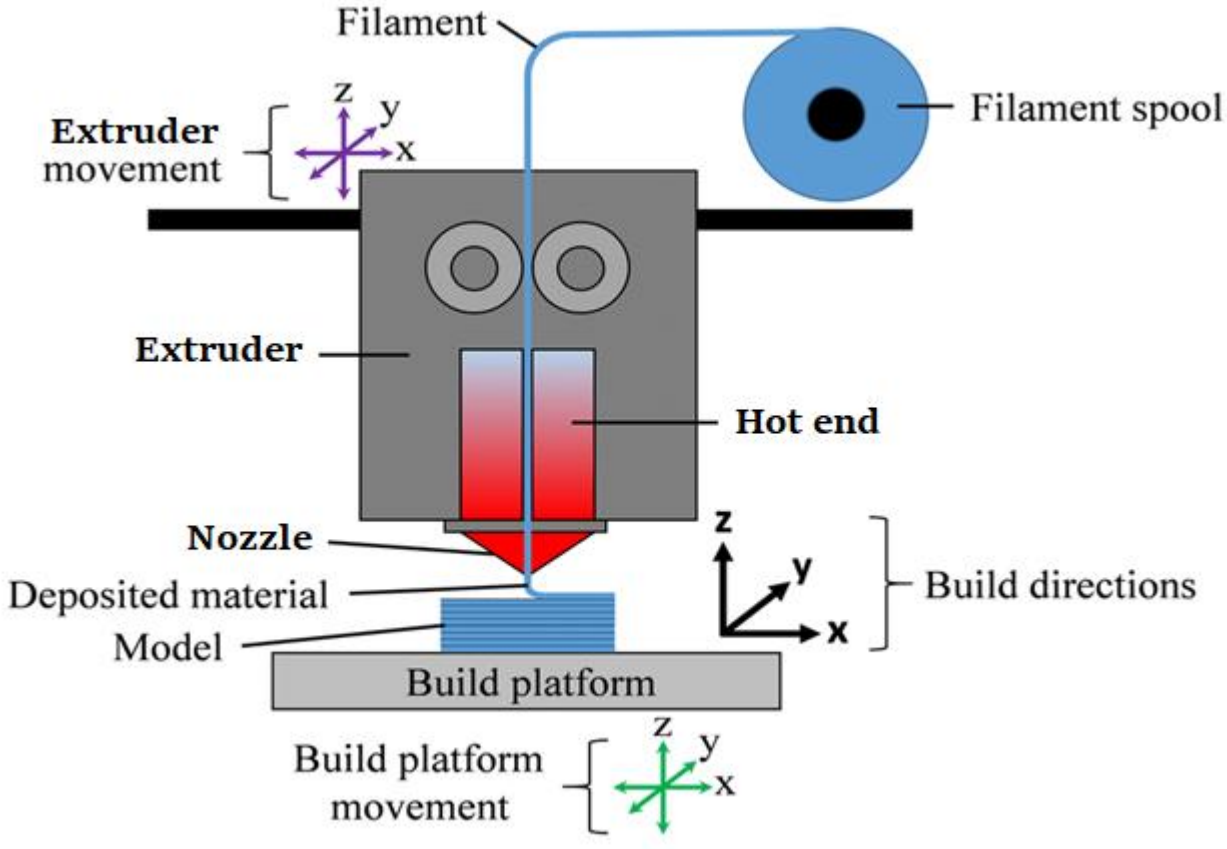


Figure 1.16: Schematic of a typical FDM printer

FDM Printing Materials

Printing Material	Advantages	Disadvantages	Typical Use	Printing Characteristics
PLA	• Easy to print • Detailed prints of small models • Trouble-free printing of large objects • Hard, low flexibility • Almost odorless • Affordable • Wide color selection	• Brittle • Low-temperature resistance (60 °C) • Difficult to post-process • Not suitable for outdoor use	• Prototypes • Toys • Figures • Jewelry • Architecture models	• Nozzle temperature: 215 °C • Bed temperature: 50 - 60 °C
PET/PETG	• High-temperature resistance • Easy to print • Low thermal expansion • Ductility and strength • Easy sanding • Almost odorless • Glossy surface • Perfect layer adhesion • Water and humidity resistant • Recyclable	• Not suitable for printing small models • Possible stringing • Bridging is problematic • Strong adhesion to the printbed • Supports can be difficult to remove	• Mechanical components • Holders and cases • Watertight prints (pots)	• Nozzle temperature: 240 °C • Bed temperature: 80 - 100 °C
ASA/ABS	• High impact and wear resistance • Very good temperature resistance • Suitable for outdoor use • easy to glue together • Can be smoothed with acetone vapors	• Difficult to print • Tendency to warp • Worse detail resolution • Unpleasant odor during printing	• Covers and protective cases • Prototypes • Replacement parts • Toys and figures	• Nozzle temperature: 245 - 265 °C • Bed temperature: 90 - 110 °C.
PC (Polycarbonate)	• High-temperature resistance • High impact and tensile strength • Good electric insulation properties	• High warping, especially with large models	• Functional prototypes • End-use components • Jigs and fixtures • Safety equipment	• Nozzle temperature: 275 °C • Bed temperature: 110 °C for the first layer, 115 °C for other layers

TPU (Thermoplastic Polyurethane)	• Flexibility and elasticity • Minimal warping • Good layer adhesion • Abrasion resistance	• More demanding to prepare and print • Must be printed slowly • Higher price • Must be stored in a dry environment	• Phone cover • Camera case • RC car wheel	• Nozzle temperature: 230 - 260 °C • Bed temperature: 45 - 65 °C
PA (Nylon)	• Mechanical properties • Chemical resistance • Flexible, but strong	• Must be stored in a dry environment • Somewhat difficult to print	• Screws • Nuts • Replacement parts • Gearbox	• Nozzle temperature: 240 °C • Bed temperature: 80 - 90 °C.
PP	• Ductile • Partially elastic • Temperature resistance	• Tendency to warp • Bad adhesion to the heated bed	• Living hinges • Flexible containers • Wearable components	• Nozzle temperature: 210 -230 °C • Bed temperature: 95 - 100 °C.
Composite materials (glass filled, copper filled, bronze filled, carbon filled etc.)	• High strength-to-weight ratio • Design flexibility • Customizable properties • Great look after post-processing • No warping	• Requires a hardened nozzle • May require a 0.6 mm nozzle	• Aerodynamic applications • Mechanical parts • Household items	• The printing characteristics of each material may vary depending on the plastic base

3D Printing File Formats

The most common 3D printing file formats include STL, OBJ, AMF, and 3MF. These are compatible with any 3D printing technology.

1. STL (Standard Triangle Language/ Standard Tessellation Language/ Stereolithography)

The most common format, it uses a tessellation of triangles to represent 3D geometry. It can be used for basic designs where color and material data are not needed. It's the most widely compatible format.

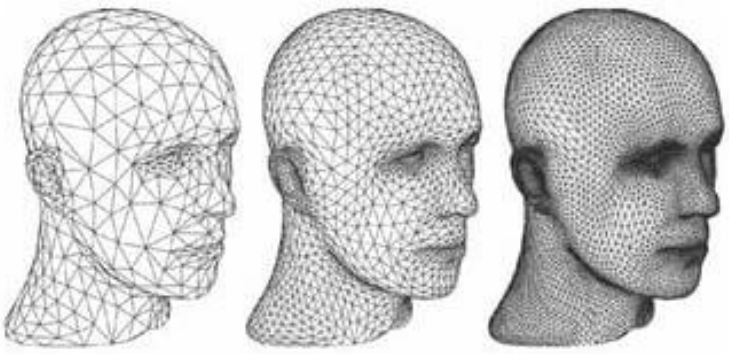


Figure 1.17: STL files with different mesh size

2. OBJ (Wavefront Object)
This file format can store geometry, color, and material data, but it is more complex and results in larger file sizes than STL. It is capable of capturing more detail than STL, especially with its ability to store color and texture data.

3. AMF (Additive Manufacturing File)
An XML-based format designed for 3D printing that can store more information than STL. It is used for advanced applications that require multiple materials or colors, though it is less widely supported than STL.

4. 3MF (3D Manufacturing Format)
A newer format specifically created with 3D printing in mind, aiming to overcome the limitations of STL. This file format is also XML-based. It can store comprehensive 3D model information beyond just geometry, such as colors, materials, textures, and print settings. Developed to overcome the limitations of older formats like STL, 3mf files are more versatile and can include features like multiple parts, custom supports, and model orientation in a single, compressed package. This makes them ideal for complex prints and for sending complete, ready-to-print data to slicers and printers.

Popular FDM Slicing Software / Slicer

A 3D printing slicer is a software that converts a 3D digital model into a series of instructions for printing that a 3D printer can understand. Some most used FDM slicers are –

- UltiMaker Cura
- PrusaSlicer
- Orca Slicer
- Bambu Studio
- Simplify3D
- Autodesk Netfabb

1.2.4 Information sheet: Basic FDM Printing Terminologies

Print Bed Calibration/ Leveling: The act or purpose of adjusting the gap between the nozzle and print bed to perform the printing correctly or more efficiently.

Filament: Typically, a thermoplastic formed into a continuous wire and wound onto a spool so it is compatible with a 3D printer's extrusion system. The thermoplastic filament can be made of ABS, PLA, TPU, HIPS, PETG etc.

Filament diameter: The size designation of a roll of filament. Usually 1.75mm or 3mm/2.85 mm (3mm and 2.85mm are in the same size filament category are used interchangeable in 3mm and 2.85 mm material extrusion 3D printers).

Infill: The area within a 3D-printed object that connects top, bottom and side layers. This creates a rigid structure and determines print durability.

Infill Density: a percentage value that determines how much of the interior volume of a 3D-printed object is filled with material. The value can range from 0-100%, recommended is 5-25%. The fill can be of different type; Honeycomb, Grid, Triangular etc.

Layer: Extruded plastic of a closed loop, represented as a two-dimensional drawing on the X-Y plane. When replicated over again in the Z direction, it produces a 3D object or multi-layered X-Y drawing.

Layer Height: Utmost determinant of quality for 3D printing, it defines the distance between lines of extruded plastic in the Z-direction. Material extrusion 3D printers typically print layers between 0.1mm and 0.3 mm high. A lower layer height translates to a smoother, higher quality print. A higher layer translates into a fast, low quality print.

Print Speed: The rate at which a 3D printer is capable of moving while extruding plastic. A print speed of 50mm/s will be successful on most FDM printers. A print speed of 20-30 mm/s will produce higher quality prints

Travel Speed: The rate at which the nozzle assembly will move to a new position when not extruding.

Nozzle/ Printing Temperature: The temperature of the hot end at which the filament if melted and extruded.

Bed Temperature: The temperature of the build plate, which is crucial for ensuring proper first-layer adhesion and preventing warping.

Resolution: The smallest movement a printer’s hot end can make within a single X-Y layer. Often indicates the produced quality of a printed model.

Retraction: Backwards movement of an extruder motor to reduce the amount of material stringing or oozing.

G-code: Coding language that the 3D printer understands. It is used to transmit instructions to a 3D printer's control system to tell the printer how to print the 3D model.

Slice: A horizontal layer of a digital object produced by a slicer program. Each slice contains coordinates for printing locations on the build surface, as well as instructions as to layer height, shell thickness and more.

Slicer: A type of program, such as UltiMaker Cura or Simplify 3D, that allows manipulation of a 3D model and converts the file type into a coordinate system (usually. G code) the printer follows to create a model.

Shell: The sidewalls of a 3D printed model, created by the exterior edges of every layer

Shell thickness: The total width of an outside wall of a 3D-printed part. Shell thickness should be a multiple of nozzle size. Two shell is typically best. An increased number of shells will lead to a stronger model.

Support: Additional removable structures that are printed to support overhangs or other parts of a model that do not make contact with the build plate during printing.

1.2.5 Self-check

Write down the most appropriate answers for the following questions:

1. What is the mother technology of FDM printing? Which company first introduced it?
2. Write down the advantages of Direct-drive extruder?
3. What are the differences of Bed slinger, Delta & Belt printer?
4. Which material is best suited for high temperature application? What are the nozzle and bed temperature of that material?
5. Write short note on terms given below:
   3MF, UltiMaker Cure, Layer height, Travel speed, Shell

1.2.6 Answer Key

1. Answer:

Fused Deposition Modeling or FDM printing is under material extrusion category of 3D printing technology. It was first introduced by Stratasys Inc.

2. Answer:

It provides excellent control over filament, especially for flexible filaments like TPU, due to the short distance between the motor and the nozzle. It can also offer better precision and retraction control.

3. Answer:

Bed Slinger Printer: The print bed moves along the Y-axis, and the print head moves along the X and Z axes. The print speed of this printer is relatively slow.

Delta Printer: These printers have three arms that move the print head in a triangular or delta-like fashion, while the print bed typically remains stationary. They are often faster and good for printing tall objects.

Belt Printer: Concerning the printhead's motion system, the belt printers are essentially CoreXY printers with the frame tilted at a 45-degree angle. Then, the print surface is replaced by a sort of conveyor belt, which moves one way or the other in the combined Z- and Y-axes. This belt can print a single infinitely long piece or it can continuously print many small parts.

4. Answer:

ASA and ABS are best suited for high temperature application. For these materials, the optimum nozzle temperature is 245 - 265 °C and bed temperature is 90 - 110 °C.

5. Answer:

3MF: Full form is 3D Manufacturing Format. This file format is XML-based, aiming to overcome the limitations of STL. It can store comprehensive 3D model information beyond just geometry, such as colors, materials, textures, and print settings.
UltiMaker Cure: A slicer for FDM printer.
Layer height: Utmost determinant of quality for 3D printing, it defines the distance between lines of extruded plastic in the Z-direction. Material extrusion 3D printers typically print layers between 0.1mm and 0.3 mm high. A lower layer height translates to a smoother, higher quality print. A higher layer translates into a fast, low quality print.
Travel Speed: The rate at which the nozzle assembly will move to a new position when not extruding.
Shell: The sidewalls of a 3D printed model, created by the exterior edges of every layer.

1.3 Learning outcome: Identify SLA Printing Technology

Learning Objectives:

At the end of this section, a trainee will be able to know –

1. Different types of SLA printing methods
2. Basic components of an SLA printer
3. General working principle of SLA printing
4. Post processing in SLA printing
5. Typical Materials Used in SLA printing
6. SLA printing slicer and slicing parameters

1.3.1 Information sheet: Fundamentals of VPP or SLA Printing

Vat photopolymerization (VPP) is one of the first and most extensively used 3D printing processes, and it has a considerable impact on a variety of industries. It was first introduced by Dr. Hideo Kodama of Japan in 1981. VPP involves selectively solidifying liquid resin layer by layer through controlled exposure to light. It enables the precise fabrication of three-dimensional objects with high resolution and complex geometries. It employs light-curing technologies to cure the resin.

Types of VPP or SLA Printing Methods

Depending on the curing light and the method of curing, VPP is classified into Stereolithography (SLA), Digital Light Processing (DLP), Continuous Liquid Interface Production (CLIP), and Masked Stereolithography (MSLA).

Stereolithography (SLA): SLA uses a UV laser to selectively cure liquid resin, tracing the cross-section of each layer. It is known for high resolution and smooth surface finishes.

Digital Light Processing (DLP): DLP uses a digital light projector to flash a complete image of a layer onto the resin at once, curing the entire layer simultaneously. This makes it faster than SLA.

Masked Stereolithography (MSLA) or, Liquid Crystal Display (LCD): Similar to DLP in that it cures entire layers at once, but it uses a UV light source and an LCD screen to mask the light, creating the pattern for each layer. It is often more affordable than SLA or DLP and suitable for desktop printers.

Continuous Liquid Interface Production (CLIP): CLIP is an advanced method that creates a continuous liquid interface between the build platform and the projector window, eliminating the need for discrete layer-by-layer separation and making the process much faster.

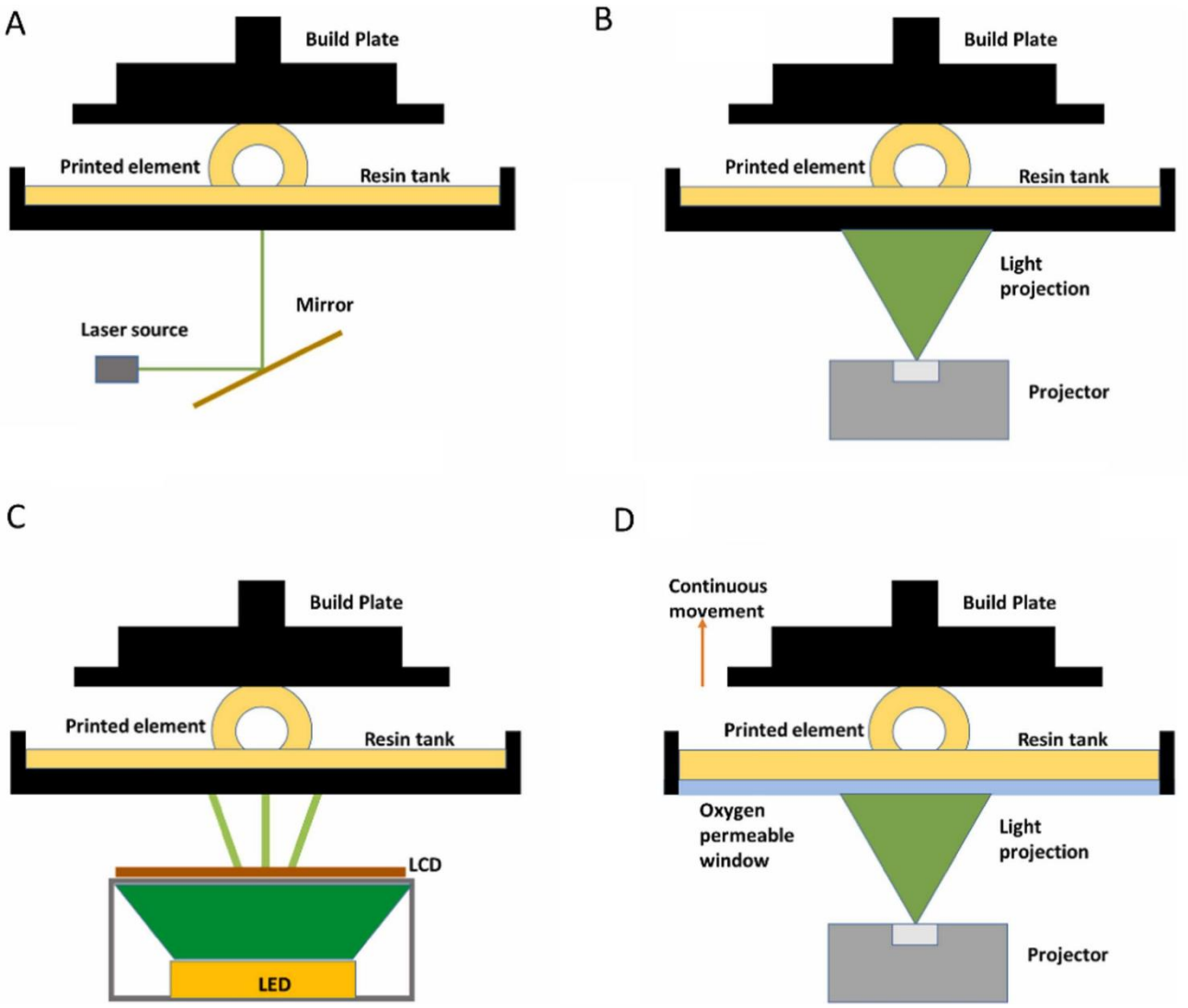


Figure 1.18: Different Types of Vat photopolymerization,
A) SLA, B) DLP, C) MSLA/ LCD, and D) CLIP

Basic components of an SLA printer

The basic components of an SLA printer are the photopolymer resin vat, the light source, and the build platform. The resin is a liquid that hardens when exposed to a light source, and the build platform moves the printed object, layer by layer, out of the vat.

Photopolymer Resin Vat: It's a container that holds the liquid photopolymer resin. The object is built layer by layer within this vat. The base of the vat is a FEP (Fluorinated Ethylene Propylene) film, which is a transparent flexible membrane and helps to peel away each layer after being cured.
Light Source: This provides the energy to solidify the resin. This can be a UV laser for Stereolithography (SLA), a projector for Digital Light Processing (DLP) or UV LED with an LCD screen (MSLA).

- **UV Laser (SLA):** A high-power laser is guided by a mirror to trace the shape of each layer onto the resin surface.
- **Projector (DLP):** A projector shines the image of an entire layer onto the resin at once, allowing for faster printing.
- **UV LED Array (MSLA):** An array of ultraviolet LEDs positioned below the resin vat provides the light necessary to cure the liquid resin. The uniformity and collimation of this light source are crucial

for consistent print quality. A high-resolution monochrome LCD screen is placed between the UV LED array and the resin vat. It acts as a digital mask, selectively allowing the UV light to pass through for each specific layer's cross-section, while blocking the light in unwanted areas.

Build Platform: A build platform moves the object being printed. It is lowered into the resin vat after each layer is cured, or sometimes raised from it, depending on the specific printer design.
Z-axis movement system: A mechanical system that precisely controls the up-and-down movement of the build platform to ensure correct layer height.

Optics: In many systems, a lens or mirror system is used to focus the light source and accurately project the image onto the resin layer.

Control system: This includes the software and hardware that translate a digital 3D model into instructions for the light source and movement system. It can also be used to control features like grayscale light for smoother surfaces.

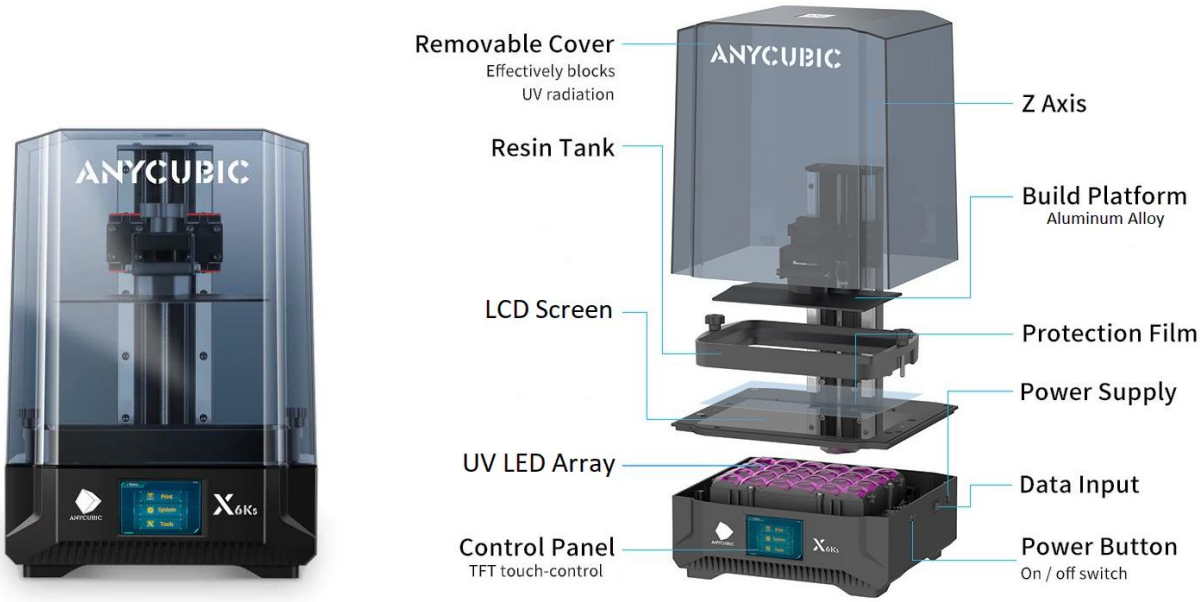


Figure 1.19: Basic components of a typical MSLA printer

Working Principle of SLA Printing

This process involves using a liquid resin as the main type of material. Specifically, this liquid resin has the special property of being able to become solid once it is exposed to light.

This light can be ultraviolet as in SL processes. Wavelengths of UV light used can be 365nm, 385nm, 395nm, 405nm etc.

This liquid resin is held in a container or vat, in which a flat build platform is partially submerged. This platform starts near the surface of the liquid, and then gets exposed to light. This light can be a UV laser (SL), a digital light processing (DLP) projector, a UV light bulb filtered through an LCD screen mask (MSLA), an LCD screen similar to home theater projectors (CLIP).

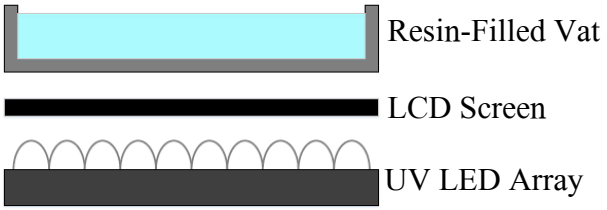
Once the resin is cured and made solid, the build platform either moves further into the vat, or partially comes out of the vat leaving the solid cured portion just under the surface and then the process is repeated.

After the printing is completed, the printed model must be washed using water, Isopropyl Alcohol (IPA), or Tripropylene Glycol Monomethyl Ether (TPM). Washing process can be done manually or in a washing station. After washing, the model is needed to be cured using UV light or direct sunlight.

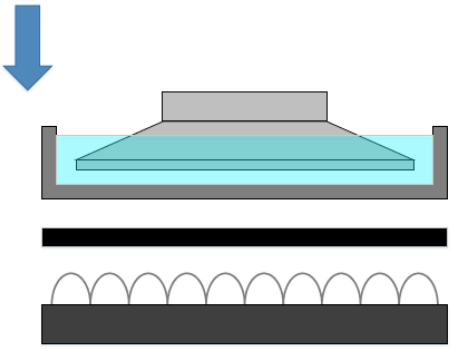
Given below is the step-by-step printing process of a typical MSLA printer.



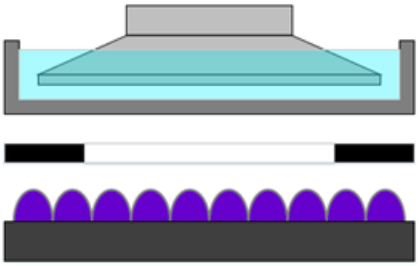
Step-1: Before printing starts



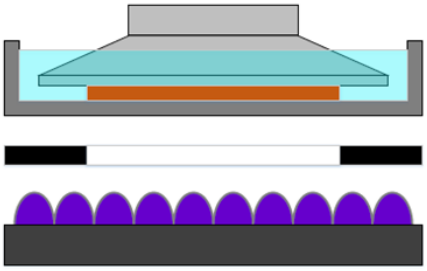
Step-2: Print starts, build plate descends and waits for a while



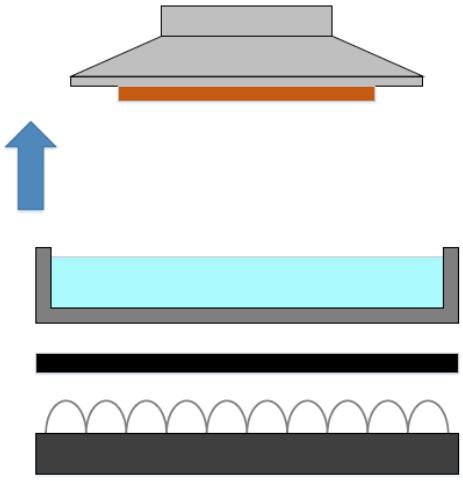
Step-3: UV light is active and LCD screen filter UV light to print a specific layer



Step-4: The target layer is printed



Step-5: UV light is off and Build plate is lifted to a distance to separate the printed layer from vat



Step-6: Step 2 to 5 is repeated until the print is completed.

Post Processing in SLA Printing

The post-processing steps for SLA printing are washing, support removal, curing, and optional finishing.

Washing

After printing, the part is washed in a solvent, typically isopropyl alcohol (IPA) or, Tripropylene Glycol Monomethyl Ether (TPM), to remove uncured liquid resin from the surface. then, the part is needed to be dried completely after washing, either by air-drying or using a heat gun.

Support Removal

The support structures must be carefully removed from the print after the washing. This can be done using flush cutters, a hobby knife, or tweezers. Removing supports while the resin is still slightly soft can make the process easier.

Curing

Exposing the part to UV light is required to fully cure the resin and achieve its final mechanical properties. This can be done using a dedicated UV curing station, a nail polish curing lamp, or even by placing it in direct sunlight for a longer period. Curing is a critical step to ensure the part is fully hardened and durable.

SLA Printing Materials

SLA Printing produces parts from photocurable resins. These photocurable resins are generally formulated from modified **acrylic monomers**. Another important additive used in these resins is **photoinitiators** are used to initiate curing reactions in the printing process. It is uniformly dispersed into the polymer resin, and once it is exposed to light, it will start cross-linking the polymer chains, resulting in solidification, or curing. **Coupling agents** are added to photocurable resins to ensure the uniform dispersion of the functional additives in the polymer resin. Other additives are added to these resins to improve specific properties such as strength, and refractive index.

A variety of resins are available, each with different properties for specific applications, including standard, engineering, flexible, castable, high-temperature, dental, and clear resins. The most common types of SLA resins are –

- **Standard Resins:** These are used for general-purpose applications and create parts with smooth surfaces, ideal for prototypes and visual models.
- **Engineering Resins:** Designed for functional parts, these resins come in different formulations for durability, such as Tough Resin, which is strong and resistant to impact, and ABS-like resin, which offers high strength and toughness.
- **Flexible and Elastic Resins:** These resins can be bent, stretched, and compressed, and some are formulated to simulate the feel of rubber or silicone for applications like grips, seals, and soft-touch parts.
- **High-Temperature Resins:** These resins have a high heat deflection temperature, making them suitable for applications that require thermal stability, like jigs, molds, and fixtures.
- **Ceramic-Filled Resins:** These are filled with ceramic particles to create parts that are rigid, strong, and heat-resistant, often used for tools, housings, and manifolds.
- **Dental and Medical Resins:** Biocompatible resins are used to print parts for medical equipment, dental models, and splints. Some are designed for long-term contact with skin or mucosal membranes, while others are optimized for creating clear dental aligners.
- **Clear Resins:** These resins are transparent, allowing for high-detail prints that are useful for visual models or parts where clarity is needed.
- **Castable Resins:** These specialized resins can be used for creating patterns that are then used in the investment casting process to produce metal parts.

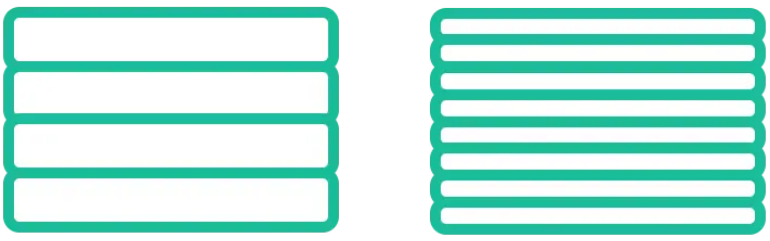
Popular SLA Slicing Software / Slicer

Some of the most used SLA slicers are –

- Anycubic Photon Workshop
- Chitubox
- Lychee Slicer,
- Formlabs PreForm, and
- PrusaSlicer

1.3.2 Information sheet: Basic SLA Printing Terminologies

Layer thickness: is also called layer height in some slicers, refers to the height of a single layer in a 3D print. This is a customizable setting that directly impacts the quality, detail, and speed of your prints. Layer thickness typically can be varied from 0.01 to 0.1 mm. Thinner layers require shorter exposure times but result in more layers overall, significantly increasing the total print time. Conversely, thicker layers shorten print times but reduce detail and make layer lines more visible.



Shorter 3D printing time

More visible layer lines

Longer 3D printing time

Less visible layer lines

Figure 1.20: Layer Thickness vs Print Quality

Normal Exposure Time: is also referred to as **layer exposure time**, **normal exposure time**, or **layer cure time** in slicers, determines how long the 3D printer exposes the resin to UV light to cure a single layer. This parameter is typically measured in seconds or milliseconds and is one of the most important settings for ensuring the quality, sharpness, and overall success of a 3D print.

Bottom Exposure Time: is also referred to as burn-in layer exposure time, initial exposure time, or base layer exposure in some slicers. Burn-in layer exposure time is the duration for which the printer cures the first few layers with a longer exposure. These initial layers help the model adhere strongly to the build plate, ensuring a stable foundation for the rest of the print.

Number of Bottom Layers: (also called the number of burn-in layers, bottom layer count, or base layers in some slicers) determines how many layers will be exposed with a higher **bottom exposure time**. These initial layers are exposed for a longer time to ensure a solid foundation that adheres securely to the build plate.



Figure 1.20: Bottom layers and normal layers

Lift Height / Z Lift Distance: refers to the distance the build plate moves up between layers to separate the printed model from the FEP film. The ideal lift height is one that allows the model to peel off the FEP without straining the printer. For general use, it is recommended a lift height of 10 mm for bottom layers and 7 mm for normal layers. This setting is conservative and works well in most cases.

Lift Speed: is also called platform lift speed in some slicers. Lift speed is the rate at which the build plate rises from the bottom to the top after each layer's exposure. It also defines the speed at which the model is peeled off the FEP film.

Retract Speed: is also known as platform lower speed in some slicers, refers to the speed at which the build plate lowers into the vat after each layer exposure. For very viscous resins, a lower retract speed is recommended. It helps reduce turbulence and allows resin to escape from under the build plate more efficiently. For most resins, however, the impact of retract speed on print quality is minimal.

Off Time: (also known as rest time after retract, light-off delay, exposure off time, or rest time before lift in some slicers) is a setting that introduces a delay after the build plate retracts to its lowest position. This pause allows time for the resin to fully settle before the next layer exposure begins. Only after this pause does the light source turn on for the next layer exposure.

Anti-aliasing: In resin 3D printing, LCD screens consist of black and white pixels. When the pixel is black, it blocks the UV light, preventing the resin from curing. When the pixel is white, the UV light passes through, curing the resin. Anti-aliasing adjusts the pixels at the edges of the model to grey, creating smoother transitions between layers and reducing the pixelation effect. This can help reduce

visible layers and pixel lines, making prints appear smoother. However, using anti-aliasing may lead to a slight loss of detail in the intricate parts of a model.

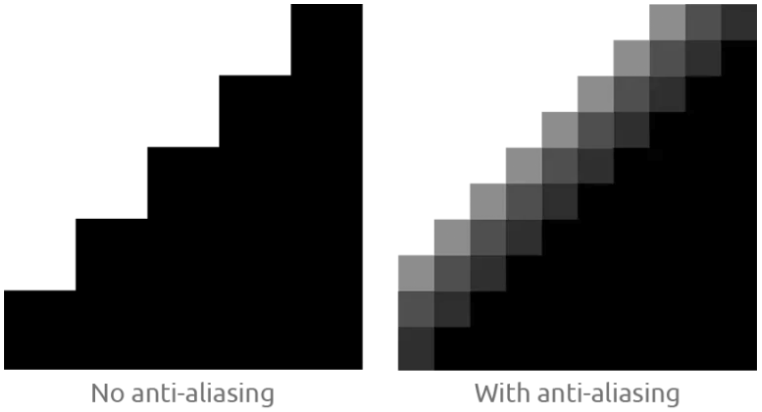


Figure 1.20: Anti-aliasing in SLA printing

UV power: is also known as LED power, Light PWM, or UV Light power, refers to the adjustable output power of the UV LEDs in some 3D printers. This setting determines how much UV light the LEDs produce, which in turn affects how quickly the resin cures. Higher UV power allows the resin to cure faster, reducing print time. By default, UV power is often set between 60–100%.

Supports: are temporary, printed components that prevent overhangs and bridges from collapsing during the build. They are crucial for ensuring the print's stability and accuracy, but they must be manually removed after printing, which can leave marks and requires post-processing like sanding. The design, including part orientation, is critical to minimizing the impact of supports on visible surfaces.

1.3.3 Self-check

Write down the most appropriate answers for the following questions:

1. What is the main difference between DLP and MSLA printing?
2. Write down about post processing in SLA printing process.
3. What is the role of photoinitiators and coupling agents used in resins?
4. Why are normal exposure time and bottom layers crucial?
5. Why is Lift Height used?

1.3.4 Answer Key

1. Answer:

Digital Light Processing (DLP) uses a digital light projector to flash a complete image of a layer onto the resin at once, curing the entire layer simultaneously.

Masked Stereolithography (MSLA) uses a UV light source and an LCD screen to mask the light, creating the pattern for each layer.

2. Answer:

Washing: After printing, the part is washed in a solvent, typically isopropyl alcohol (IPA) or, Tripropylene Glycol Monomethyl Ether (TPM), to remove uncured liquid resin from the surface.

Curing: Curing is a critical step to ensure the part is fully hardened and durable. This can be done using a dedicated UV curing station, a nail polish curing lamp, or even by placing it in direct sunlight for a longer period.

3. Answer:

Photoinitiators are used to initiate curing reactions in the printing process.

Coupling agents are added to ensure the uniform dispersion of the functional additives in the polymer resin.

4. Answer:

The **normal exposure time** is one of the most important settings in SLA printing for ensuring the quality, sharpness, and overall success of a 3D print.

The **bottom layers** help the model to adhere strongly to the build plate, ensuring a stable foundation for the rest of the print.

5. Answer:

The **lift height** is used to peel off the model from the FEP film without straining the printer.

1.4 Learning outcome: Recognize the role of slicer software in 3D printing

Learning Objectives:

At the end of this section, a trainee will be able to know –

1. Function and importance of slicer or slicing software
2. 3D model orientation and manipulation within slicer
3. Basic printing parameters used slicer

1.4.1 Information sheet: Function and Importance of Slicer

Slicing in 3D printing is the process of converting a 3D digital model into a series of thin, horizontal layers and generating a set of instructions for the 3D printer to build it. A software called a slicer or slicing software translates the 3D model (often an STL, OBJ, or 3MF file) into G-code, which tells the 3D printer exactly how to move, extrude material, and build each layer, from start to finish. This step is crucial because it bridges the gap between a digital design and a physical object by defining every aspect of the print, including resolution (layer height or layer thickness), internal structure (infill pattern and density), wall thickness and perimeters, support structures, adhesion methods, speed and temperature settings.

Functions of slicing software

Translation: A slicer converts standard 3D model files (like STL, OBJ, or 3MF) into a series of thin, horizontal layers.
G-code generation: the slicer then translates the layered model into G-code, a set of instructions that tell the 3D printer exactly how to move, extrude, and build the object layer by layer.
Parameter customization: Before g-code generation the users can customize parameters to define and adjust settings that control the printing process, such as:

- **Layer height** determines the thickness of each individual layer which affects print resolution and print time
- **Print speed** controls how fast the printer moves, affecting print time and quality. balancing precision and production time
- **Infill density and pattern** determine the part's internal structure and strength
- **Support structures** are added automatically or manually to print overhanging parts of the model that would otherwise print in mid-air.

Preview and estimation: Slicers also provide visual previews of the final print layer by layer and estimates the total print time, material consumption, and cost before printing begins.
Pre-print analysis: The **pre-print analysis of** the model's geometry can be done with the help of a slicing software to identify areas that may need support structures or pose printing challenges.

Importance in 3D printing

Makes printing possible: Without a slicer, the 3D printer would have no way to understand how to build the physical object from the digital design.

Ensures print quality and performance: The ability to customize settings allows users to balance factors like strength, detail, and speed, tailoring the print to their specific needs.
Optimizes material usage: By controlling the infill density and other settings, slicers help reduce the amount of material used, which can lower costs.
Enhances consistency: For manufacturing, slicers ensure that every part produced meets the same quality and specifications, a crucial factor for scaling up production.
Reduces print failures: Features like support generation for overhangs and model repair tools help ensure a successful print and prevent failures.
Facilitates advanced features: It provides the necessary control to implement features like custom support structures or material-specific settings (like temperature and flow rate) that are essential for successful printing.

1.4.2 Information sheet: Slicer User Interface and Basic Functions

The User Interface (UI) of every slicer is fairly simple and almost similar. Any slicer typically includes a central 3D model preview window, a left or right sidebar for general settings and printer profiles, and a tabbed panel for fine-tuning print, filament, and other settings. Here, the UI of UltiMaker Cura 5.9.0 is shown as an example.

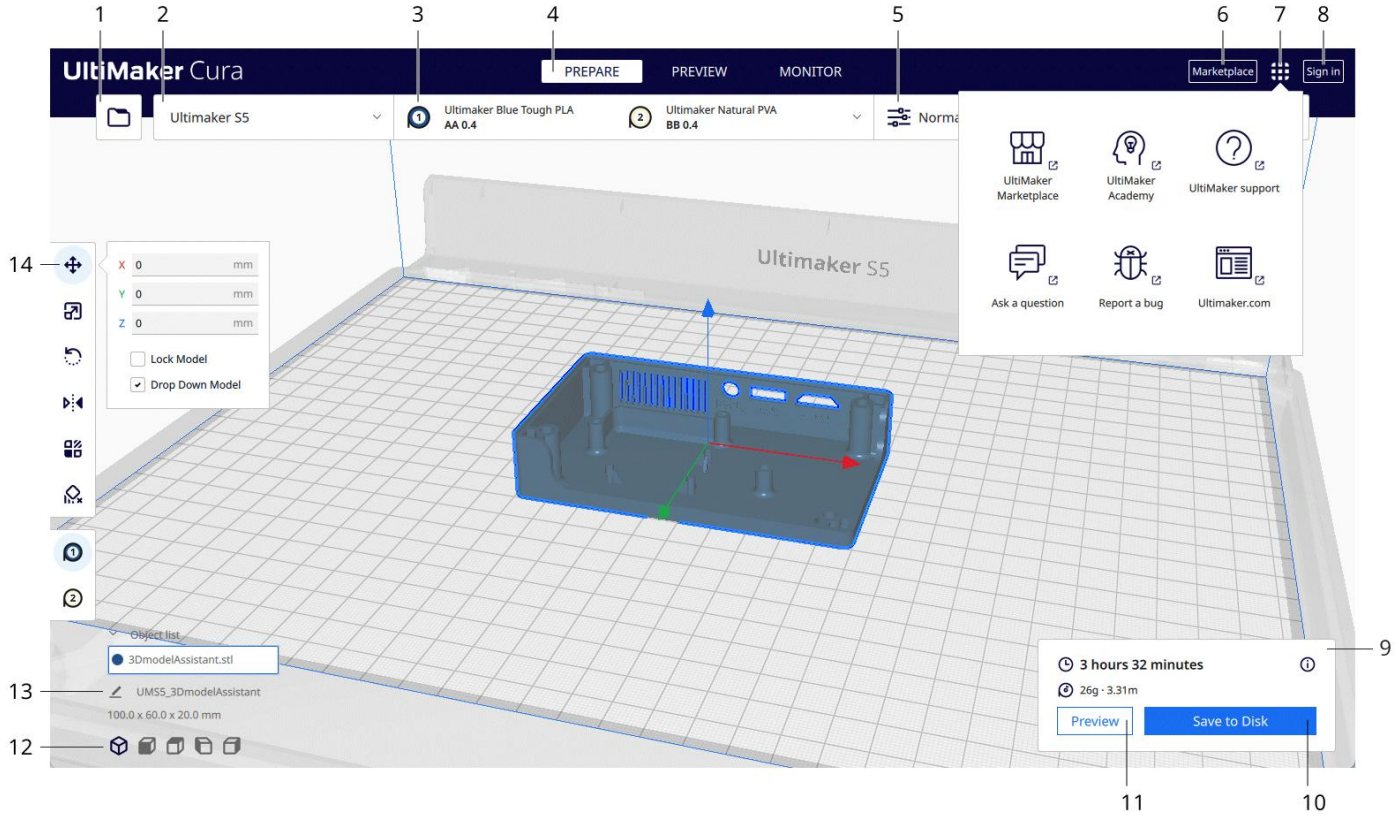


Figure 1.21: Typical UI of a Slicer

1. **Open file.** Opens a 3D file.
2. **Printer selection panel.** Displays the selected printer.
3. **Configuration panel.** Contains the material and print core or nozzle setup.
4. **Stages.** The prepare, preview, and monitor stage. The goal of the **Prepare stage** is to load and configure your 3D models for printing. **Preview stage** can preview demo the printed model when the model has

been sliced. If the printer is connected to a local network, then the **Monitor stage** shows the real time update of a print in progress.

5. **Print settings panel.** This panel contains the print profiles and all individual print parameter settings.
6. **Marketplace.** Contains plugins and material packages.
7. **App switcher.** Easily visit UltiMaker's additional platforms for support, e-learning, and more.
8. **UltiMaker account.** Sign in to your UltiMaker account or create a new one to gain access to cloud functionality, Marketplace plugins and print profiles, backups, and more.
9. **Action panel.** Contains an action button dependent on the current stage. This can contain the button to slice the model, but other options are available once the model is prepared.
10. **Save or share file.** After slicing, the file can be saved to a computer or a removable drive, or directly start the print via local network.
11. **Preview.** Proceeds to the next stage, the Preview stage.
12. **Camera position tool.** Easily positions the camera to one of the default viewing angles.
13. **Model information.** Contains the print job name (based on the first model loaded) and the total dimensions of all printable models on the build plate. If there are multiple models, these are shown in the **Object list.**
14. **Adjustment tools.** Move, rotate, mirror, scale, and more options to alter the appearance or orientation of the imported 3D model(s).

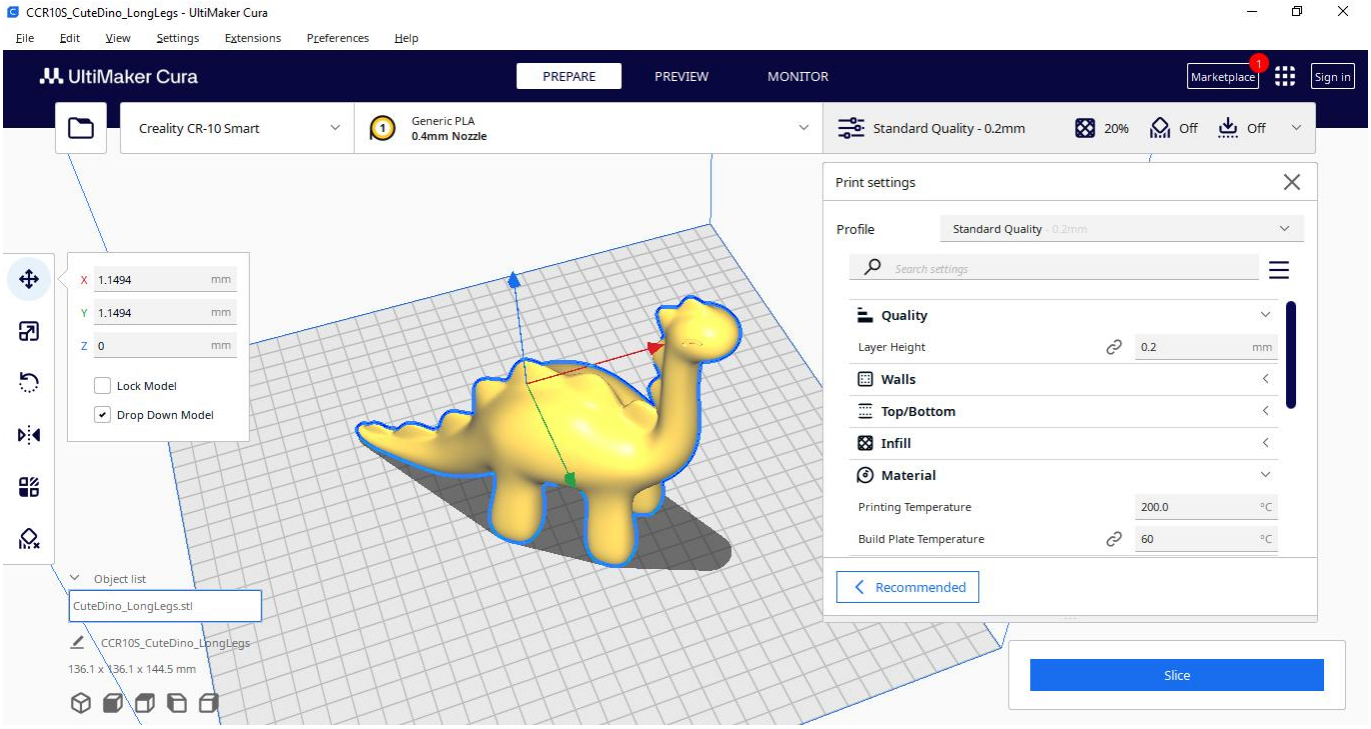


Figure 1.22: A 3D Model before Slicing

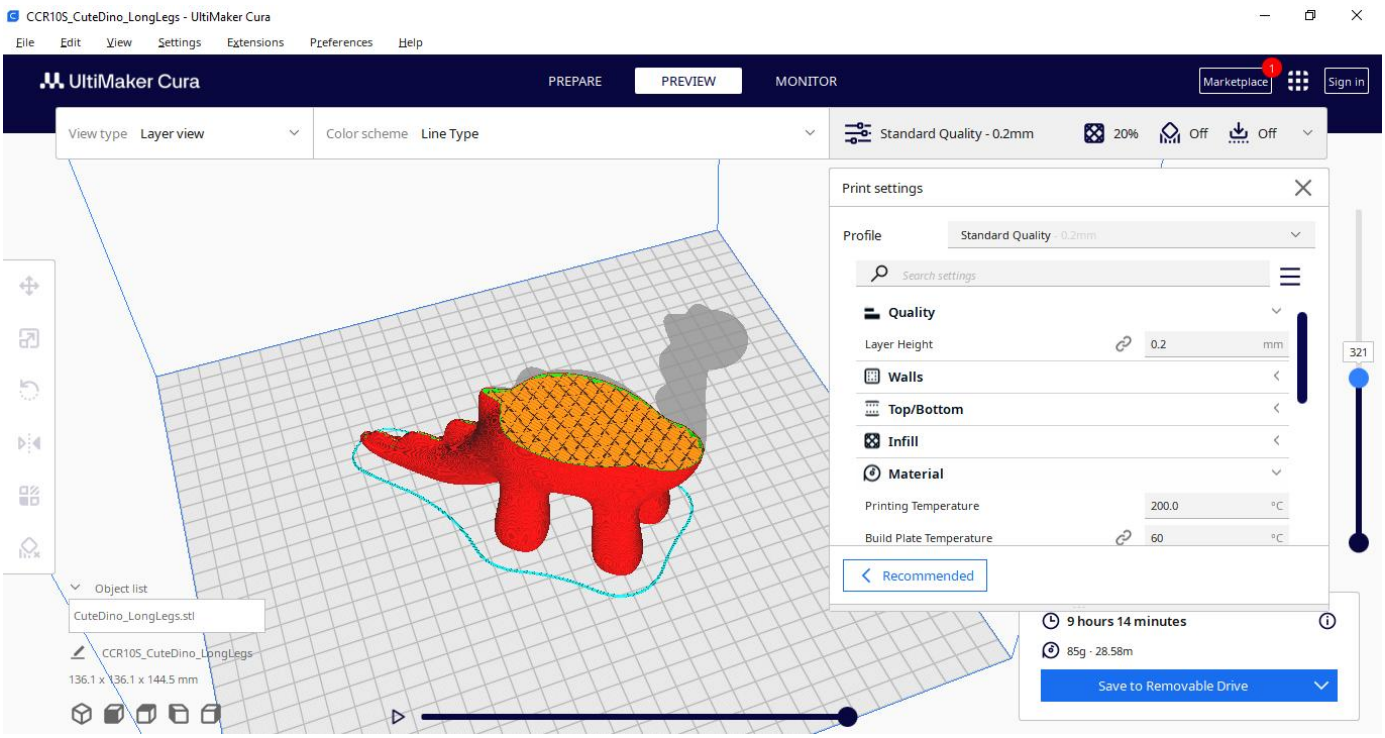


Figure 1.23: A 3D Model after Slicing

1.4.3 Self-check

Write down the most appropriate answers for the following questions:

1. Why is slicing crucial in 3D printing?

2. Write down the functions of pre-print analysis.
3. How can one optimize material cost using slicer?

1.4.4 Answer Key

1. Answer:

Because it bridges the gap between a digital design and a physical object by defining every aspect of the print.

2. Answer:

The pre-print analysis can identify areas of a 3D model whether it may need support structures or pose printing challenges with the help of a slicing software.

3. Answer:

By controlling the infill density and other related settings.

Module-2: Performing 3D Modelling Using CAD Software

Module descriptor:

This unit covers the knowledge, skills and attitudes required to perform 3D modelling using CAD software. It specifically includes the tasks of installing CAD software, carrying out 2D sketching for design creation, designing 3D models using feature and surface tools, and assembling 3D models.

Nominal Duration: 85 hours

Learning outcome:

After completing this module, the trainee must acquire the following competencies:

7.5 Install CAD software
7.6 Carry out 2D sketching for design creation
7.7 Design 3D models using feature and surface tools
7.8 Assemble 3D models

2.1 Learning outcome: Install CAD software

Content:

1. Installation process and licensing of SolidWorks software
2. Initial setup and required plugins or extensions installation and configuration within SolidWorks
3. Functionality test of installed software

Assessment Criteria:

1. System requirements for SolidWorks software are identified.
2. Installation process of SolidWorks software is followed.
3. Licensing and activation keys are entered.
4. Software is activated to ensure legitimate usage.
5. Required plugins or extensions are installed and configured within the SolidWorks software.
6. Functionality of the SolidWorks software is tested by opening a sample project or 3D model.

Necessary Equipment:

1. Desktop Compatible with software requirements (Must meet the minimum/ recommended specifications)
    i. Operating System: 64-bit Windows 10 or 11
    ii. Processor: Intel or AMD processor, 3.3 GHz or higher
    iii. RAM: Minimum 8GB, with 32GB or more recommended for larger assemblies and simulation
    iv. Storage: A Solid-State Drive (SSD) with at least 20GB of free disk space
    v. Graphics Card: A certified graphics card (NVIDIA Quadro or AMD Radeon Pro)
2. SolidWorks Software (2020 version or newer)
3. Stable internet connection
4. SolidWorks serial number or license
5. SolidWorks installer (online or offline version)
6. User manual or online help documentation

Learning activity:

Learning activity	Resources/Special instrumentation/References
Install CAD software	• Information sheet: 2.1.1 • Self-check: 2.1.2 • Answer key: 2.1.3

Information Sheet 2.1.1:

Introduction

Before performing 3D modelling, you must first install the CAD (Computer-Aided Design) software properly. CAD software is the primary tool used in designing, visualizing, and preparing models for 3D printing. A successful installation ensures the software runs smoothly, integrates well with system hardware, and provides all required features for modelling and exporting files in formats suitable for additive manufacturing.

Improper installation may lead to frequent system errors, missing features, or compatibility issues with your 3D printer or slicer software. Therefore, understanding the correct installation process is essential for an efficient workflow and professional results.

Learning Objectives

After completing this information sheet, you should be able to:

1. Identify and verify the **system requirements** for the selected CAD software.
2. Download or prepare the **installation package** from legitimate sources.
3. Perform a **step-by-step installation** following manufacturer procedures.
4. Ensure the software runs correctly and is ready for modelling operations.

Key Knowledge

Computer/Laptop: A computer is a device that can be instructed to carry out sequences of arithmetic or logical operations automatically via computer programming. The word compute means to calculate accounts and Computer means a calculating device. But now a day's computer is not called only calculating device. Computer is such a device which receives information and analyses and presents that information through different media.

CAD: Computer-Aided Design (CAD) software allows users to create detailed 2D and 3D designs used in engineering, architecture, and manufacturing. For 3D printing, it is primarily used to create digital models that can be exported into file formats such as STL, OBJ, or 3MF. Examples of CAD software commonly used in 3D printing include:

- Autodesk Fusion 360
- SolidWorks
- TinkerCAD
- SketchUp
- FreeCAD
- Blender (for modelling)

Each of these programs has its own system requirements, licensing terms, and installation methods.

Operating System: Operating System is system software that manages computer hardware and software resources and provides common services for computer programs.

Software (SolidWorks): The SolidWorks application includes familiar Windows functions, such

as dragging and resizing windows. Many of the same icons, such as print, open, save, cut, and paste is also part of the SolidWorks application.

Types of Installation Files

- **Online Installer** – Downloads additional files during installation (requires internet).
- **Offline Installer / Full Package** – All files are pre-packed; suitable for systems without internet.
- **Educational or Trial Version** – May require an account sign-in for activation.
- **Licensed Version** – Requires a serial number or product key.

It is important to use **authentic installation sources** — avoid third-party or pirated downloads as they can contain malware and lead to licensing violations.

Pre-Installation Preparation

Before running the installer, you should:

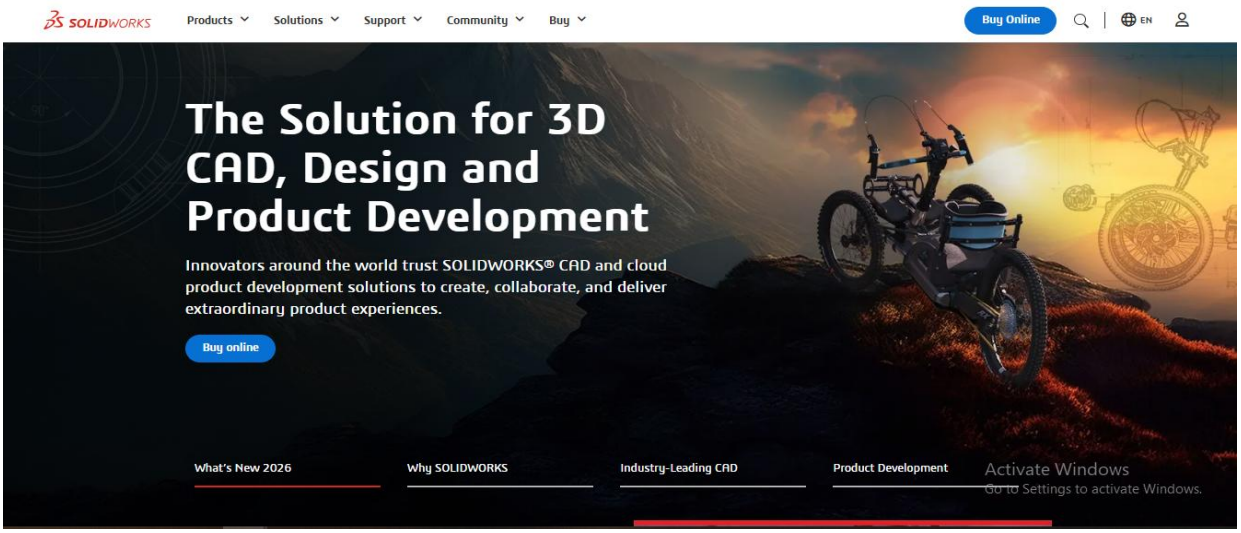
1. Close all open applications to avoid conflicts.
2. Log in as an **Administrator** (installation often needs admin privileges).
3. Check for **available storage space** and system updates.
4. Ensure a **stable power supply** if using a laptop — plug into AC power.
5. Temporarily disable unnecessary background programs or antivirus if they block installation files (but re-enable them afterwards).

Step-by-Step Installation Procedure

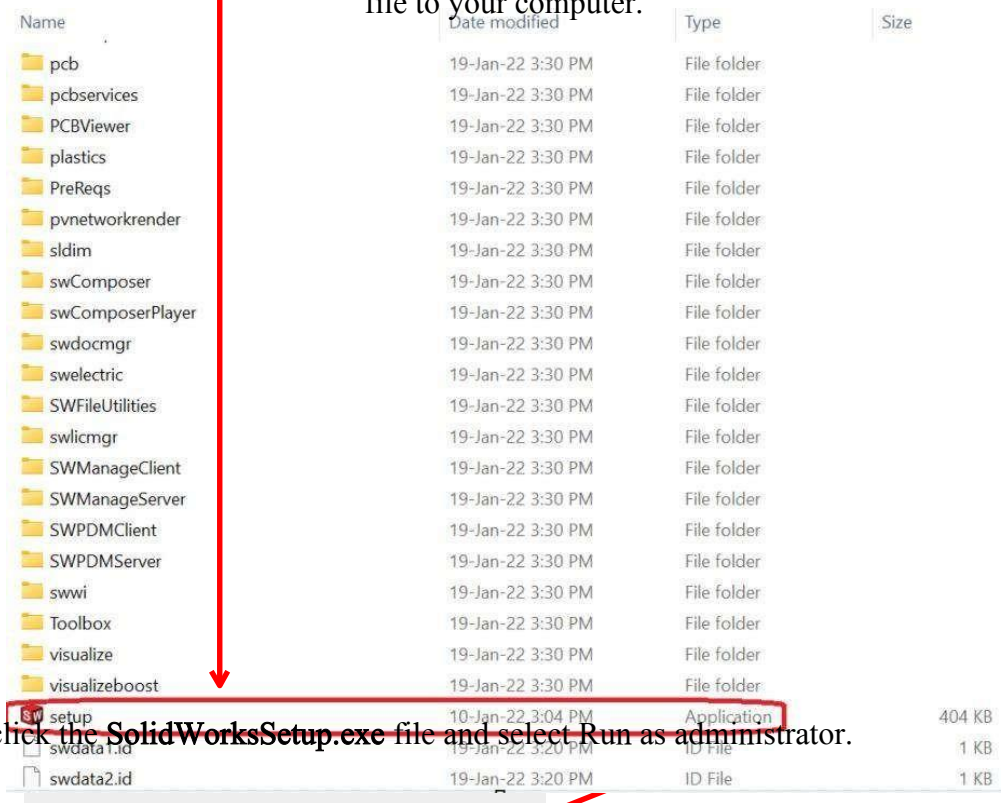
Below is a general procedure that applies to most CAD software. Specific details may vary depending on the brand and version.

Step 1: Download the file/files.

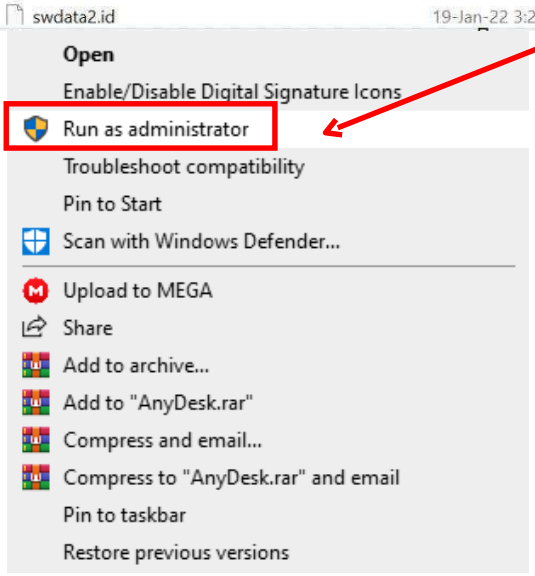
- Visit the official website (e.g. solidworks.com).
- Select the version suited for your operating system.
- If required, sign in or create an educational account.
- Save the installer to your Downloads folder or USB drive.



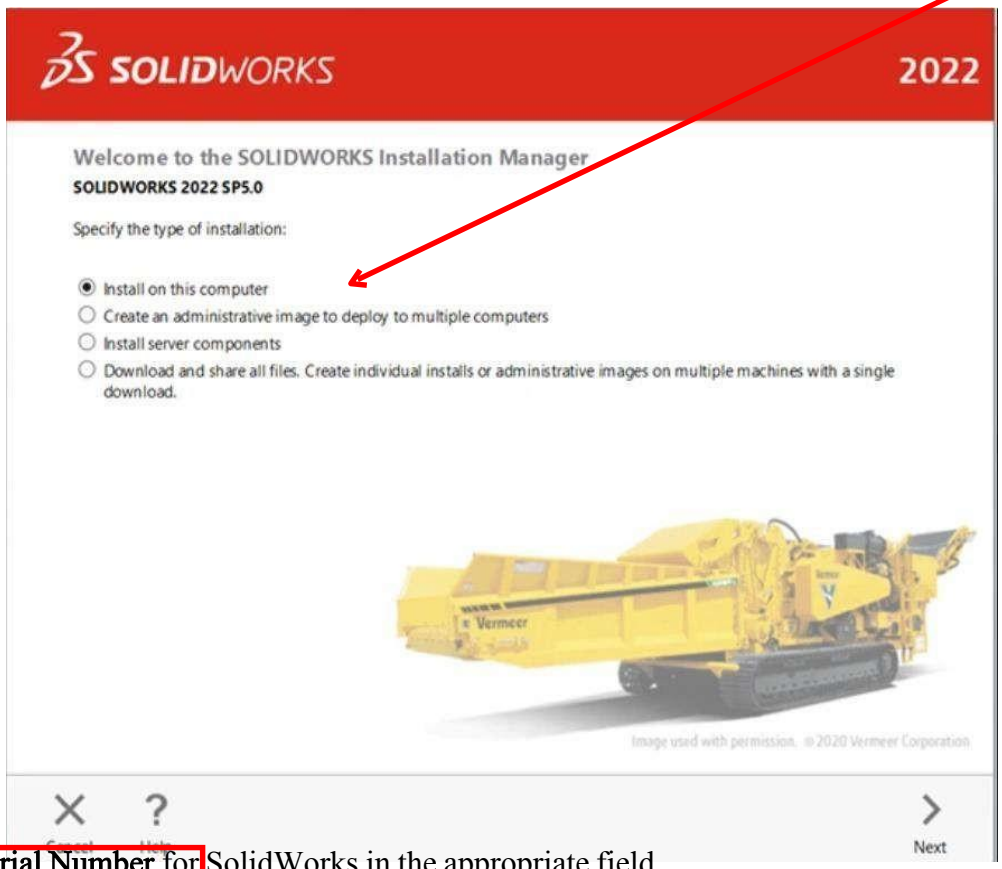
Step 2: Find the **SolidWorksSetup.exe** file to your computer.



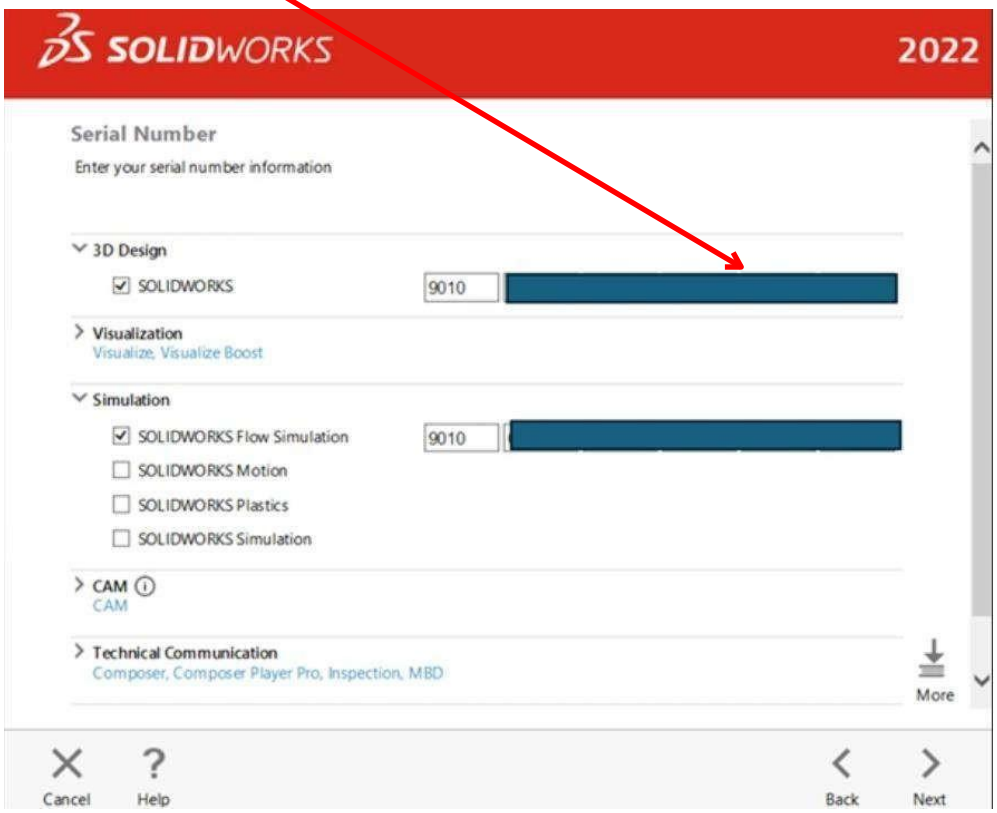
Step 3: Right on click the **SolidWorksSetup.exe** file and select Run as administrator.



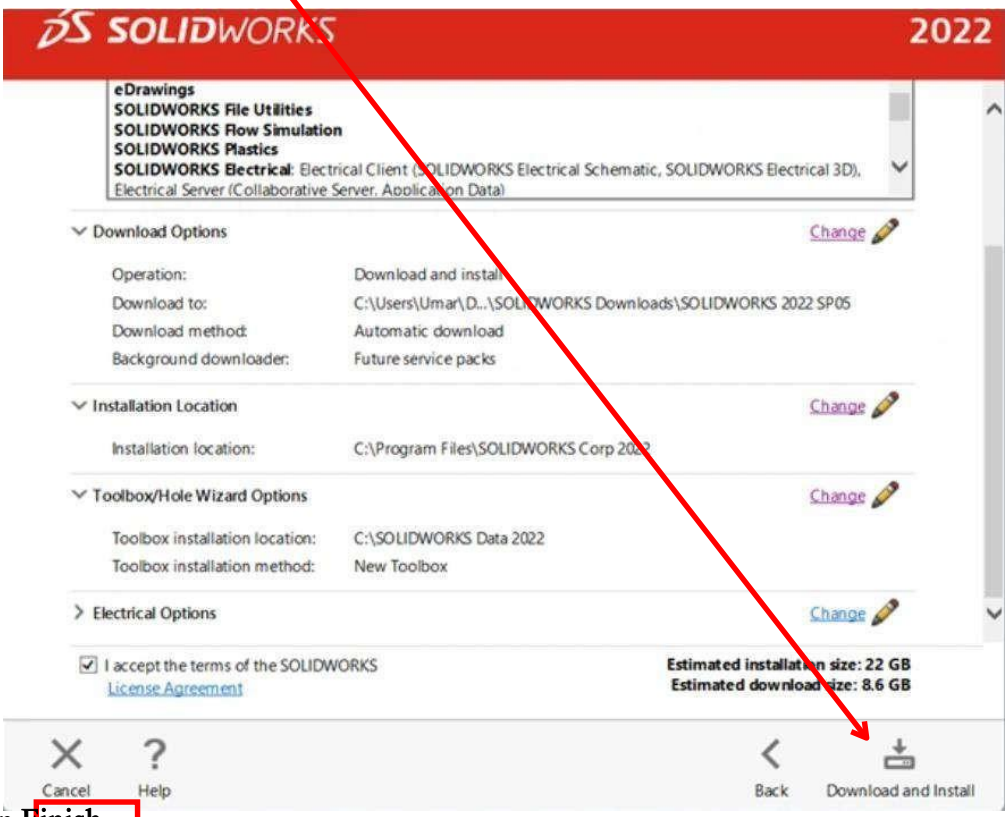
Step 4: When the SolidWorks Installation Manager window appears,select **Install on this computer.**



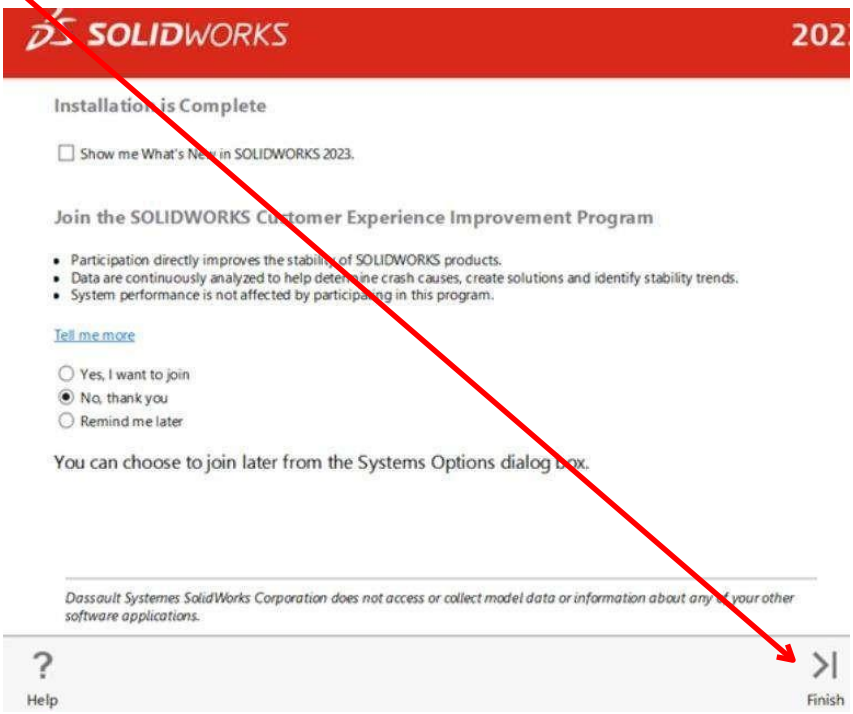
Step 5: Enter your **Serial Number** for SolidWorks in the appropriate field.



Step 6: Click on **Download and Install** to begin the process. The manager will download all the necessary files before beginning the installation.



Step 7: Click on **Finish.**



Step 8: Verify the Installation

1. Launch the program and start a new project.
2. Draw or model a simple object (like a cube or cylinder).
3. Save your project and attempt to **export** it as.STL.
4. Open the exported file in your slicer (e.g., Cura or PrusaSlicer) to confirm compatibility.

5. If everything runs smoothly, the CAD software is successfully installed and ready for use.

Safety & Good Practices

1. **Use Authorized Software Only** Installing pirated software is illegal and risky. Always use verified or academic versions provided by the instructor or training institution.
2. **Keep Your System Secure** Re-enable antivirus or firewall after installation to maintain system protection.
3. **Perform Regular Updates** CAD developers release patches to fix bugs and improve stability. Set the software to notify you of updates.
4. **Back up Files before** installing updates or add-ons, back up your CAD files and settings to prevent data loss.
5. **Optimize Performance**
    - Close unnecessary programs when running CAD software.
    - Update your graphics driver regularly.
    - Save files frequently to prevent data loss.
6. **Housekeeping Practices**
    - Organize project files in clearly named folders.
    - Delete temporary setup files to free disk space.
    - Keep your desktop and software interface uncluttered for efficient workflow.

Tips for Learners

- Take note of your **software version and build number** — instructors may require this for compatibility.
- If the software does not run after installation, check:
    - System requirements
    - Administrator privileges
    - Missing dependencies (like .NET Framework or Visual C++ Redistributables)
- Practice installing on different computers to become familiar with variations.
- Learn where the software stores configuration files — useful when transferring settings to another system.
- Always read release notes for new versions.

2.1.2 Self-Check

1. What is the importance of verifying **system requirements** before installing CAD software?
2. What are the main **types of installation files** used for CAD software?
3. What are the Pre-Installation Preparation?
4. List the general **steps to install CAD software** properly.
5. Why is it recommended to use **official sources only** when downloading CAD software?

2.1.3 Answer Paper

Self-Check Questions	Answer Key / Suggested Answer
1.	To ensure that the computer meets the hardware and software specifications required by the

Self-Check Questions	Answer Key / Suggested Answer
	CAD program; this prevents errors, crashes, and performance issues.
2.	1. Online Installer – downloads files during setup. 2. Offline/Full Installer – contains all installation files. 3. Educational or Trial Version – requires online sign-in.
3.	1. Close all open applications to avoid conflicts. 2. Log in as an Administrator 3. Check for available storage space and system updates. 4. Ensure a stable power supply if using a laptop — plug into AC power. 5. Temporarily disable unnecessary background programs or antivirus if they block installation files.
4.	1. Download the file/files. 2. Find the SolidWorksSetup.exe 3. Right on click the solidworkssetup.exe file and select run as administrator. 4. When the SolidWorks Installation Manager window appears, select install on this computer. 5. Enter your Serial Number for SolidWorks in the appropriate field. 6. Click on Download and Install to begin the process. The manager will download all the necessary files before beginning the installation. 7. Click on finish. 8. Verify the Installation
5.	To avoid viruses, malware, and illegal or corrupted software, and to ensure access to legitimate updates and technical support.

2.2 Learning Outcome: Carry out 2D sketching for design creation

Content:

1. Interface of SolidWorks software.
2. Basic 2D Sketch tools.
3. Basic relation tools.
4. Dimension and annotation.

Assessment Criteria:

1. **User interface of SolidWorks is correctly recognized and navigated**
   - Main interface components (Command Manager, FeatureManager Design Tree, Graphics Area, Menu Bar) are identified.
   - Required sketch tools and commands are accessed without assistance.

2. **Drawing workspace is set up appropriately within SolidWorks**
   - A new part file is created and an appropriate sketch plane is selected.
   - Document units, grid settings, and sketch environment are configured as required for the job.

3. **Basic 2D sketching tools are used to create foundational sketch elements**
   - Line, circle, point, rectangle, arc, polygon, spline, slot, pattern, split, mirror etc tools are applied correctly.
   - Sketch geometry is created according to design requirements.

4. **Geometrical constraints (relations) are applied to define relationships between sketch entities**
   - Horizontal, vertical, coincident, parallel, perpendicular, tangent, equal, concentric etc relations are correctly used.
   - Sketch is organized to reflect proper geometric behavior.

5. **Dimensions and annotations are added accurately as per job specification**
   - Smart Dimension tool is used to define size and positional relationships.
   - Dimensions follow standard drafting practices and do not conflict.

6. **Sketch errors or inconsistencies are identified and corrected**
   - Under-defined, over-defined, and conflicting dimensions or relations are detected.
   - Necessary adjustments, deletions, or corrections are performed to ensure a fully defined sketch.

7. **Final 2D sketch is saved and exported in appropriate file format for further processing**

Necessary Equipment:

1. A computer set
2. SolidWorks software (2020 version or above)

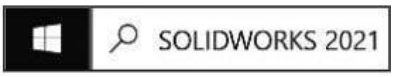
Learning activity:

Learning activity	Resources/Special instrumentation/References
Carry out 2D sketching for design creation	• Information sheet: 2.2.1 • Self-check: 2.2.2 • Answer key: 2.2.3 • Job Sheet: 2.2.4

Information Sheet: 2.2.1 (SolidWorks Interface)

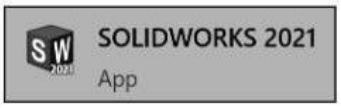
Learning Objective: Familiar with SolidWorks Interface.

1) Type **SOLIDWORKS 2021** in the Search window.

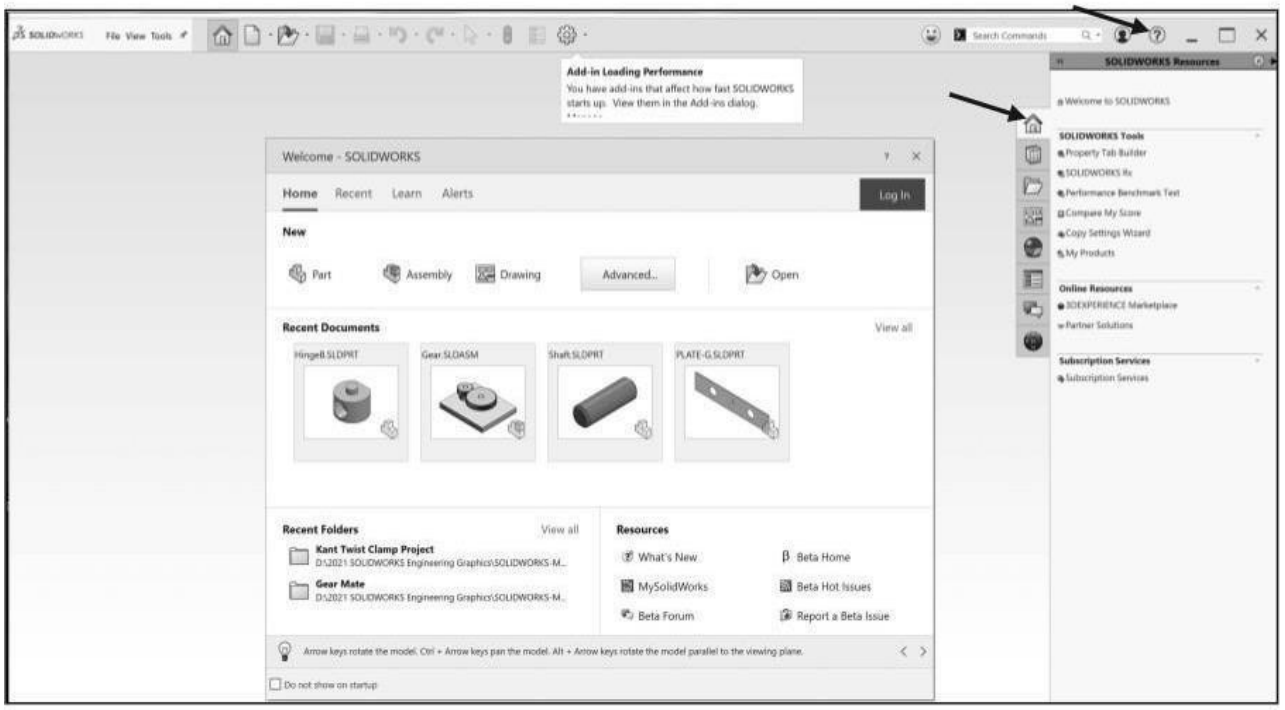


2) Click the **SOLIDWORKS 2021** application (or if available, **double-click** the SOLIDWORKS icon on the desktop). The SOLIDWORKS Welcome dialog box is displayed by default.



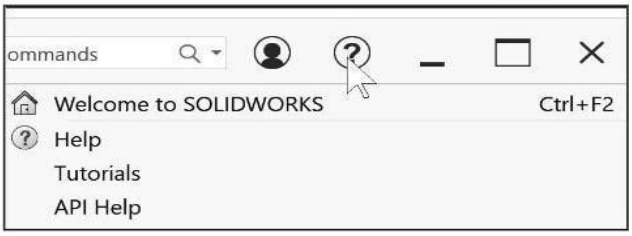


The Welcome dialog box provides a convenient way to open recent documents (Parts, Assemblies and Drawings), view recent folders, access SOLIDWORKS resources, and stay updated on SOLIDWORKS news.



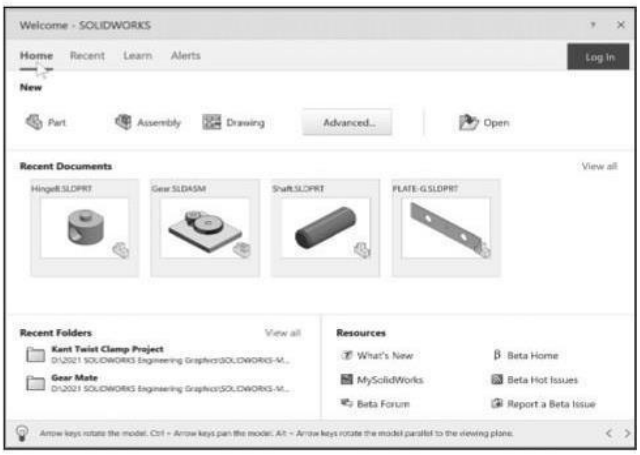
3) **View** your options. Do not open a document at this time.

4) If the Welcome dialog box is not displayed, click the **Welcome to SOLIDWORKS** icon from the Standard toolbar or click **Help** > **Welcome to SOLIDWORKS** from the Main menu.



Home Tab

The Home tab lets you open new and existing documents, view recent documents and folders, and access SOLIDWORKS resources (*Part, Assembly, Drawing, Advanced mode, Open*).



Recent Tab

The Recent tab lets you view a longer list of recent documents and folders. Sections in the Recent tab include *Documents* and *Folders*.

The Documents section includes thumbnails of documents that you have opened recently.

Click a thumbnail to open the document, or hover over a thumbnail to see the document location and access additional information about the document. When you hover over a thumbnail, the full path and last saved date of the document appears.



Close the Welcome dialog box.

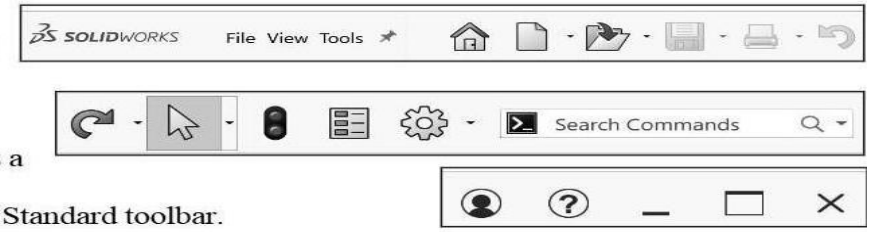
5) Click **Close** from the Welcome dialog box. The SOLIDWORKS Graphics window is displayed. Note: You can also click outside the Welcome dialog box, in the Graphics window.



View the SOLIDWORKS Graphics window.



Menu Bar toolbar



The SOLIDWORKS (UI) is designed to make maximum use of the Graphics window. The Menu Bar toolbar contains a set of the most frequently used tool buttons from the Standard toolbar.

The following default tools are available:

- **Welcome to SOLIDWORKS** - Open the Welcome dialog box, **New** - Create a new document; **Open** - Open an existing document; **Save** - Save an active document; **Print** - Print an active document; **Undo** - Reverse the last action; **Redo** - Redoes the last action that you reverse; **Select** - Select Sketch entities, components and more; **Rebuild** - Rebuild the active part, assembly or drawing;

File Properties - Summary information on the active document; **Options** - Change system options and Add-Ins for SOLIDWORKS; **Login** - Login to SOLIDWORKS; **Help** - access to help, tutorials, updates and more.

Menu Bar menu



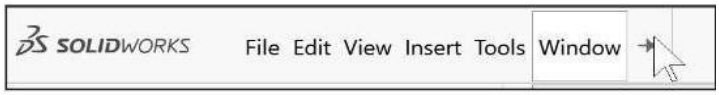
Click SOLIDWORKS in the Menu Bar toolbar to display the Menu Bar menu. SOLIDWORKS provides a context-sensitive menu structure. The menu titles remain the same for all three types of documents, but the menu items change depending on which type of document is active.



Example: The Insert menu includes features in part documents, mates in assembly documents, and drawing views in drawing documents. The display of the menu is also dependent on the workflow customization that you have selected. The default menu items for an active document are File, Edit, View, Insert, Tools, Window, Help and Pin.

The Pin option displays the Menu bar toolbar and the Menu bar menu as illustrated. Throughout the book, the Menu bar menu and the Menu bar toolbar are referred to as the Menu bar.







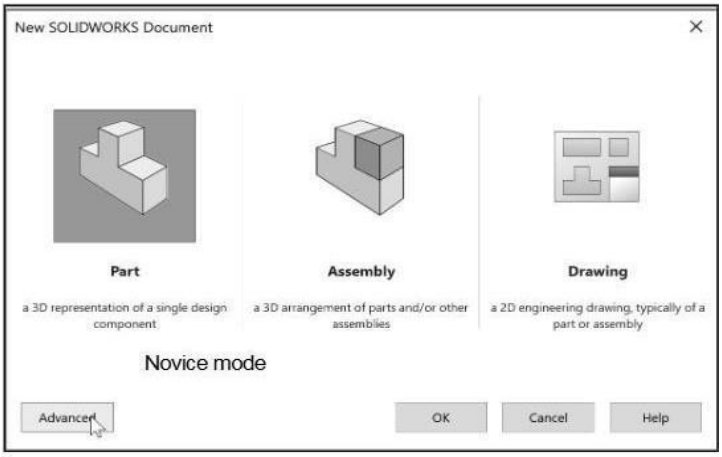
Drop-down menu

SOLIDWORKS takes advantage of the familiar Microsoft® Windows user interface. Communicate with SOLIDWORKS through drop-down menus, Context sensitive toolbars, Consolidated toolbars or the CommandManager tabs.

To close a SOLIDWORKS drop-down menu, press the Esc key. You can also click any other part of the SOLIDWORKS Graphics window or click another drop-down menu.

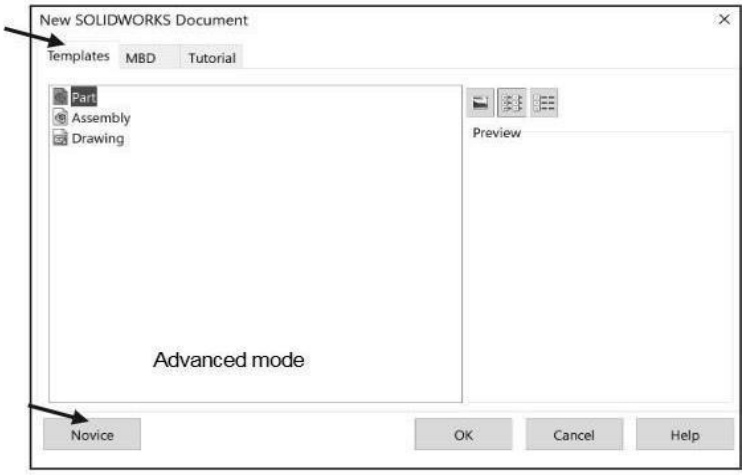


There are two modes in the New SOLIDWORKS Document dialog box: Novice and Advanced. The Novice option is the default option with three templates. The Advanced mode contains access to additional templates and tabs that you create in system options. Use the Advanced mode in this book.



Create a new part.

6) Click **New** from the Menu bar. The New SOLIDWORKS Document dialog box is displayed.



Select the Advanced mode.

7) Click the **Advanced** button as illustrated. The Advanced mode is set.

8) Click the **Templates** tab.

9) Click **Part**. Part is the default template from the New SOLIDWORKS Document dialog box.

10) Click **OK** from the New SOLIDWORKS Document dialog box.

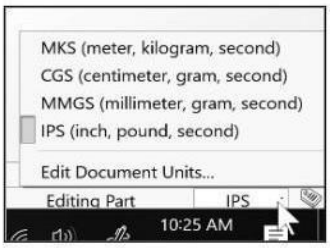
The Advanced mode remains selected for all new documents in the current SOLIDWORKS session. When you exit SOLIDWORKS, the Advanced mode setting is saved.

The default SOLIDWORKS installation contains three tabs in the New SOLIDWORKS Document dialog box: *Templates, MBD, and Tutorial*. The *Templates* tab corresponds to the default SOLIDWORKS templates. The *MBD* tab corresponds to the templates utilized in the SOLIDWORKS (Model Based Definition). The *Tutorial* tab corresponds to the templates utilized in the SOLIDWORKS Tutorials.

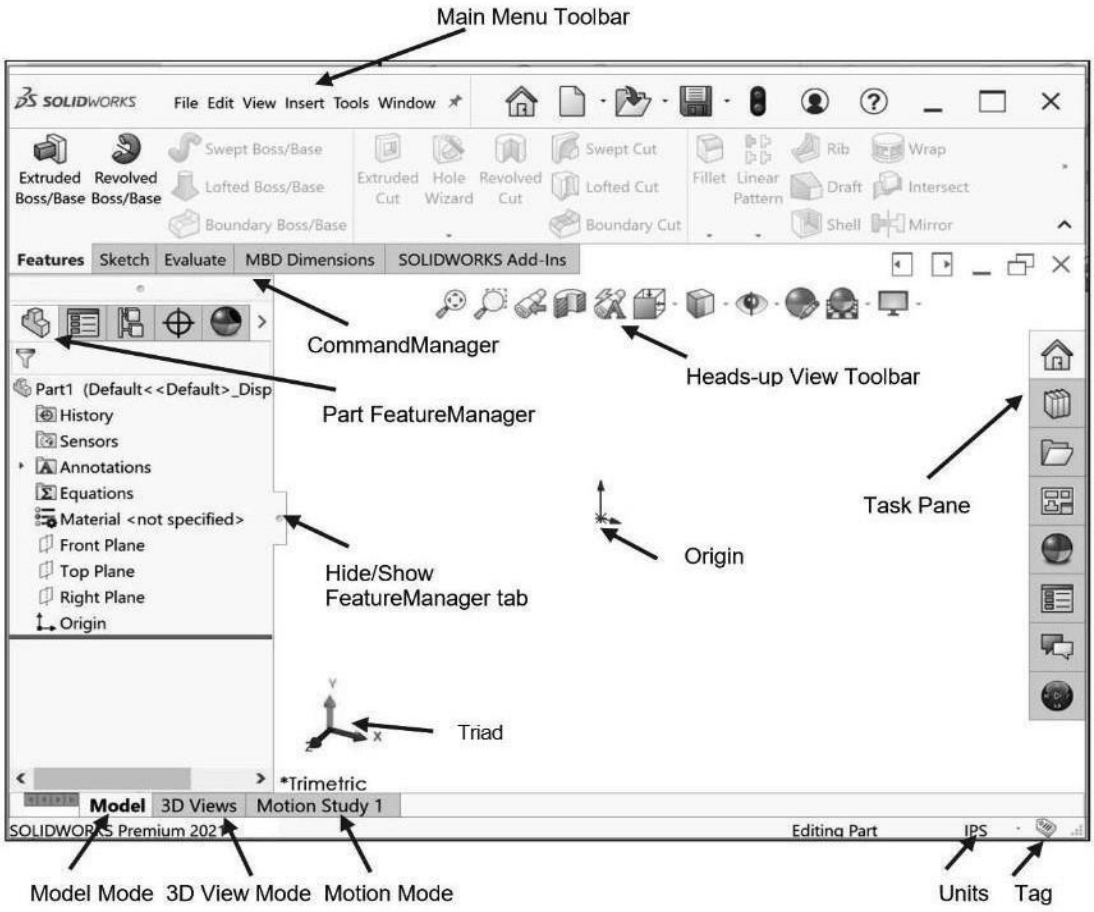
Part1 is displayed in the FeatureManager and is the name of the document. Part1 is the default part window name.

The Part Origin is displayed in blue in the center of the Graphics window. The Origin represents the intersection of the three default reference planes: *Front Plane*, *Top Plane* and *Right Plane*. The positive X-axis is horizontal and points to the right of the Origin in the Front view. The positive Y-axis is vertical and points upward in the Front view. The FeatureManager contains a list of features, reference geometry, and settings utilized in the part.

Edit the document units directly from the Graphics window as illustrated.



CommandManager and FeatureManager tabs will vary depending on system setup, version, and Add-ins.



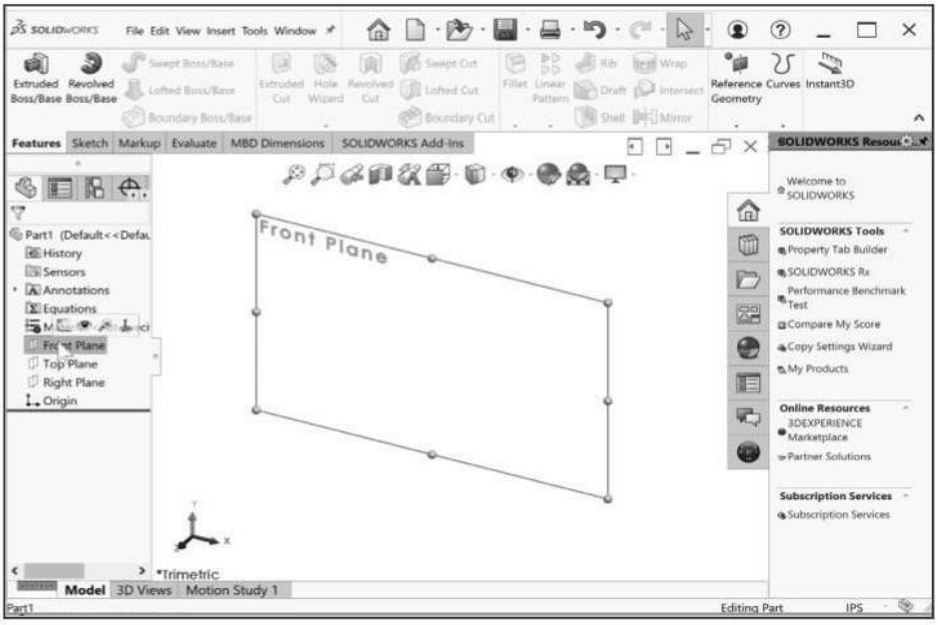
View the Default Sketch Planes.

11) Click the **Front Plane** from the FeatureManager.

12) Click the **Top Plane** from the FeatureManager.

13) Click the **Right Plane** from the FeatureManager.

14) Click the **Origin** from the FeatureManager. The Origin is the intersection of the Front, Top, and Right Planes.



Download all model files from the SDC Publications website (https://www.sdcpublications.com/downloads/978-1-63057-409-3). Open the Bracket part. Review the features and sketches in the Bracket FeatureManager. Work directly from a local hard drive.

Activity: Download the SOLIDWORKS folder. Open the Bracket Part.

Download the SOLIDWORKS folder. Open an existing SOLIDWORKS part.

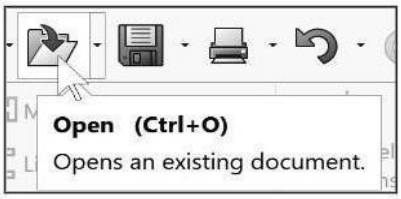
15) **Download** the SW-TUTORIAL-2021 folder.

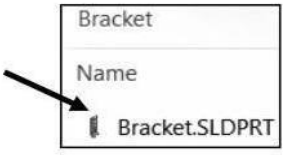
16) **Unzip** the folder. **Work** from the unzip folder.

17) Click **Open** from the Menu bar menu.

18) Browse to the **SW-TUTORIAL-2021\Bracket** folder.

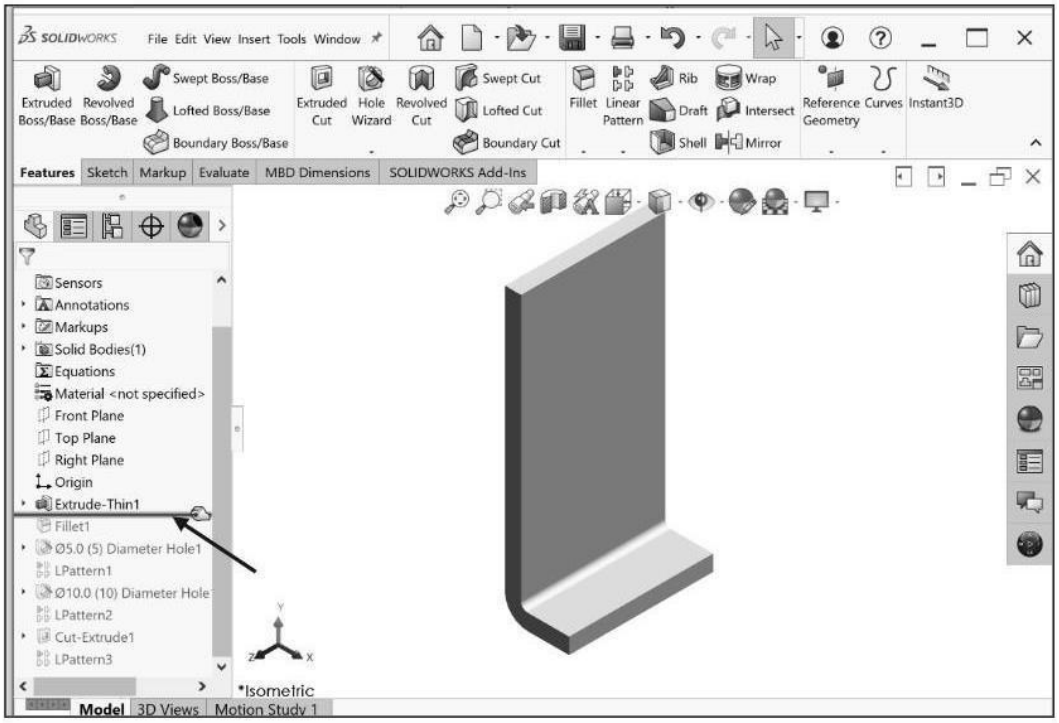
19) Double-click the **Bracket** part. The Bracket part is displayed in the Graphics window.





View the first feature in the Bracket Part.

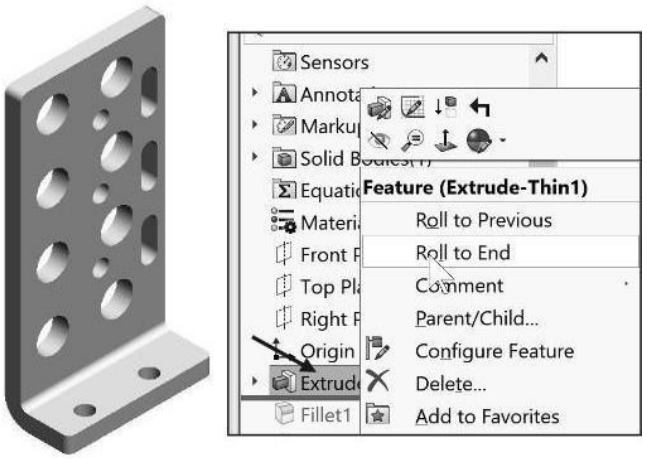
23) Drag the **rollback bar** up the FeatureManager above Fillet1. View the results in the Graphics window.



Return to the original Bracket Part FeatureManager.

24) Right-click **Extrude-Thin1** in the FeatureManager. The Pop-up Context toolbar is displayed.

25) Click **Roll to End**. View the results in the Graphics window.



Heads-up View toolbar



SOLIDWORKS provides the user with numerous view options. One of the most useful tools is the Heads-up View toolbar displayed in the Graphics window when a document is active.

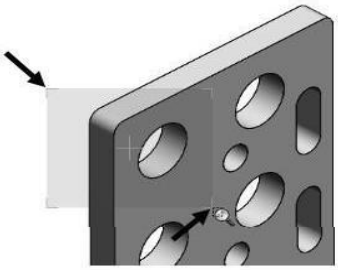
Dynamic Annotation Views : Only available with SOLIDWORKS MBD (Model Based Definition). Provides the ability to control how annotations are displayed when you rotate models.

In the next section, apply the following tools: Zoom to Fit, Zoom to Area, Zoom out, Rotate and select various view orientations from the Heads-up View toolbar.

Activity: Utilize the Heads-up View toolbar.

Zoom to Fit the model in the Graphics window.

26) Click the **Zoom to Fit** icon. The tool fits the model to the Graphics window.



Zoom to Area on the model in the Graphics window.

27) Click the **Zoom to Area** icon. The Zoom to Area icon is displayed.

Zoom in on the top left hole.

28) **Window-select** the top left corner as illustrated. View the results.



De-select the Zoom to Area tool.

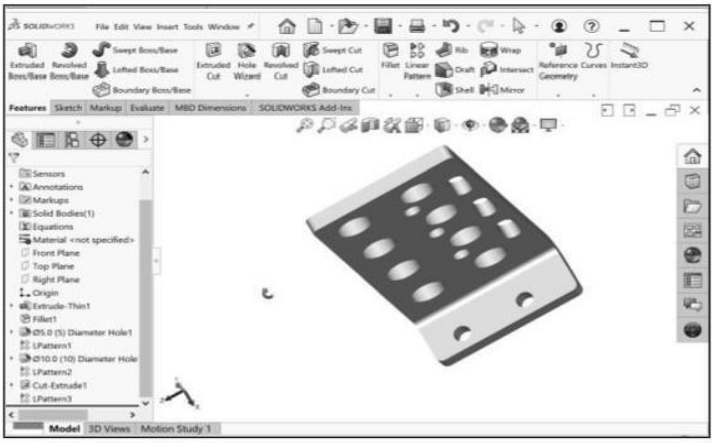
29) Click the **Zoom to Area** icon.

Fit the model to the Graphics window.

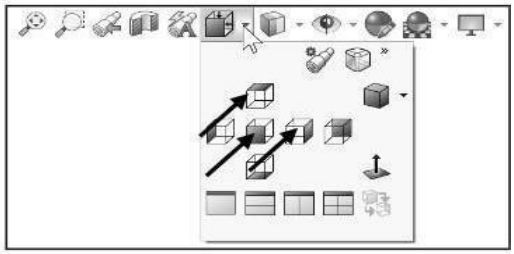
30) Press the **f** key.

Rotate the model.

31) Hold the **middle mouse button** down. Drag **upward**, **downward**, to the **left** and to the **right** to rotate the model in the Graphics window.



Display a few Standard Views.

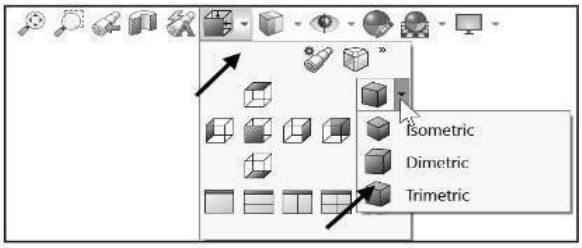


32) Click **inside** the Graphics window.

33) Click **Front** from the drop-down Heads-up view toolbar. The model is displayed in the Front view.

34) Click **Right** from the drop-down Heads-up view toolbar. The model is displayed in the Right view.

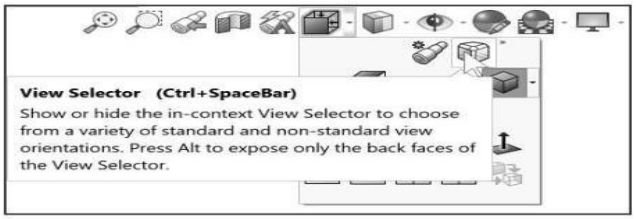
35) Click **Top** from the drop-down Heads-up view toolbar. The model is displayed in the Top view.



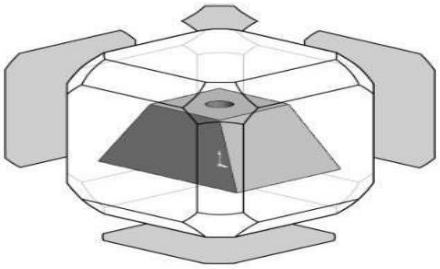
Display a Trimetric view of the Bracket model.

36) Click **Trimetric** from the drop-down Heads-up view toolbar as illustrated. Note your options. View the results in the Graphics window.

- *Section View*: Display a cutaway of a part or assembly, using one or more cross section planes.
- *Dynamic Annotation Views*: Only available with SOLIDWORKS MBD. Control how annotations are displayed when you rotate a model.



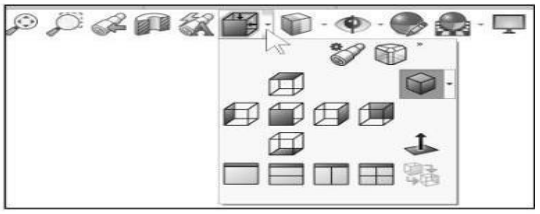
The Orientation dialog has an option to display a view cube (in-context View Selector) with a live model preview. This helps the user to understand how each standard view orientates the model. With the view cube, you can access additional standard views. The views are easy to understand and they can be accessed simply by selecting a face on the cube.



To activate the Orientation dialog box, press (Ctrl + spacebar) or click the View Orientation icon from the Heads-up View toolbar. The active model is displayed in the View Selector in an Isometric orientation (default view).

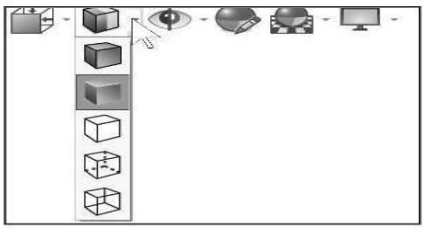
Click the View Selector icon in the Orientation dialog box to show or hide the in-context View Selector.

Press **Ctrl + spacebar** to activate the View Selector.



Press the **spacebar** to activate the Orientation dialog box.

- *View Orientation box*: Select a view orientation or the number of viewports. The options are *Top*, *Left*, *Front*, *Right*, *Back*, *Bottom*, *Single view*, *Two view - Horizontal*, *Two view - Vertical, Four view*. Click the drop-down arrow to access Axonometric views: *Isometric*, *Dimetric* and *Trimetric*.
- *Display Style*: Display the style for the active view. The options are *Wireframe*, *Hidden Lines Visible*, *Hidden Lines Removed*, *Shaded*, *Shaded With Edges*.

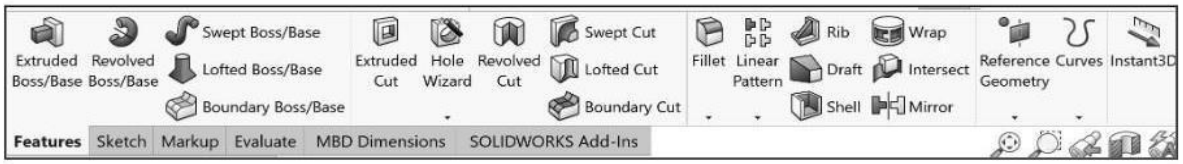


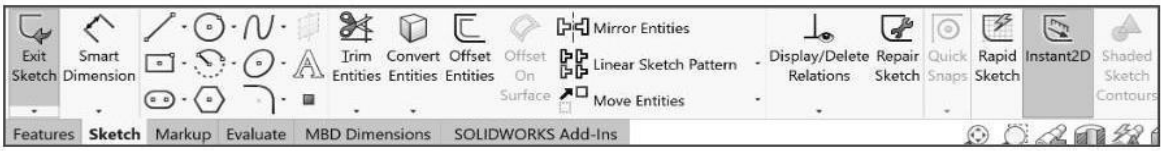
SOLIDWORKS CommandManager

The SOLIDWORKS CommandManager is a Context-sensitive toolbar. By default, it has toolbars embedded in it based on your active document type. When you click a tab below the CommandManager, it updates to display that toolbar. For example, if you click the Sketch tab, the Sketch toolbar is displayed.

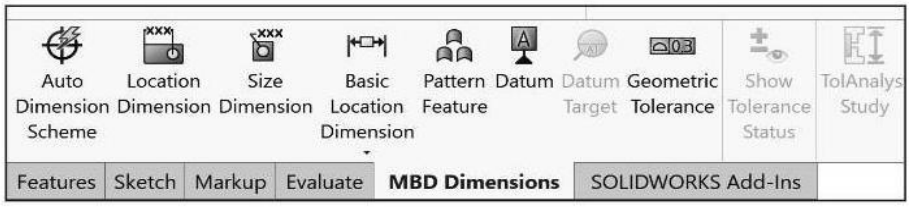
For commercial users, SOLIDWORKS Model Based Definition (MBD) and SOLIDWORKS CAM is a separate application. For education users, SOLIDWORKS MBD and SOLIDWORKS CAM is included in the SOLIDWORKS Education Edition.

Below is an illustrated CommandManager for a **Part** document. Tabs will vary depending on system setup and Add-ins.









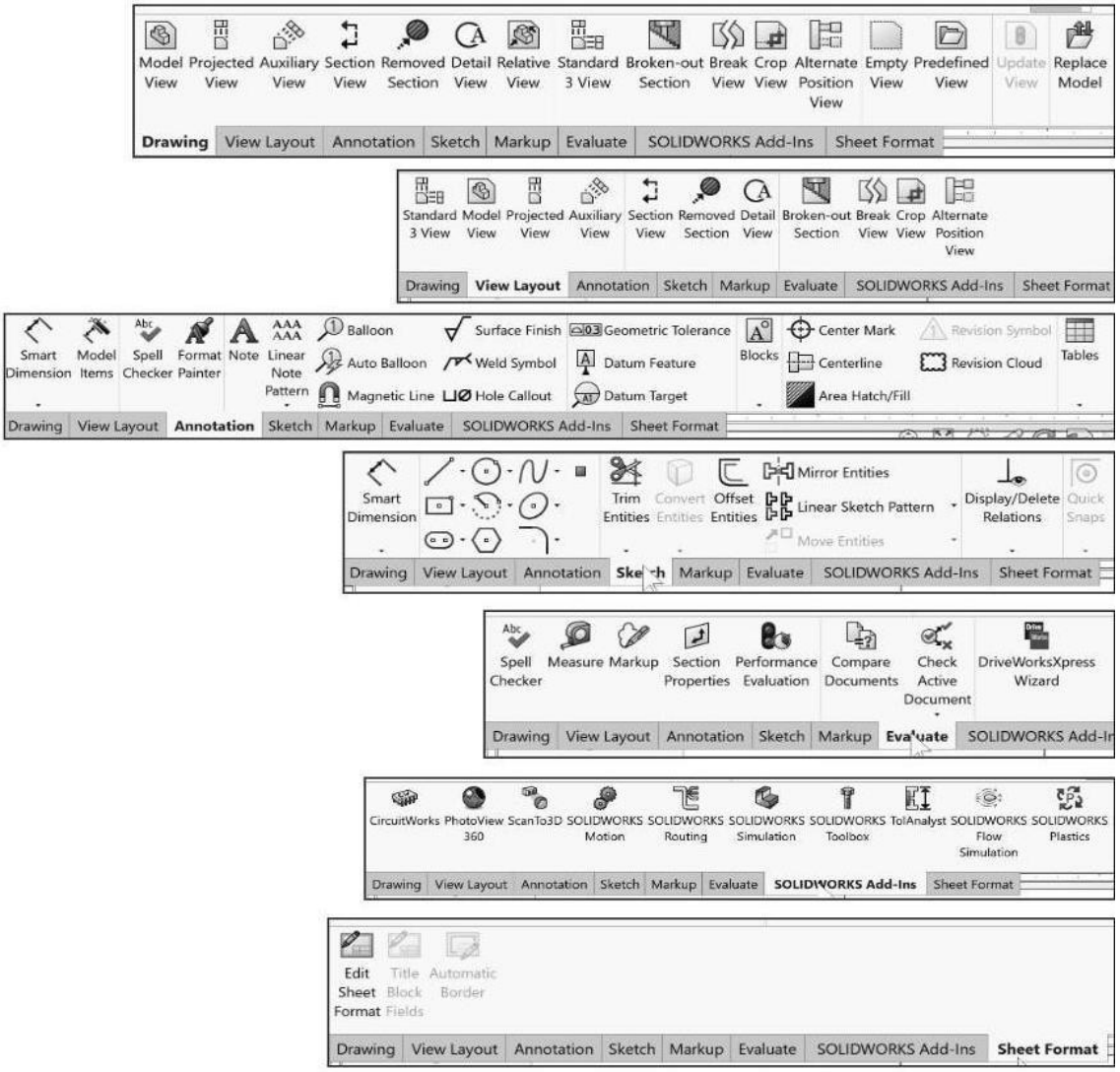


Set button size from the Toolbars tab of the Customize dialog box. To facilitate element selection on touch interfaces such as tablets, you can set up the larger Size buttons and text from the Options menu (Standard toolbar).

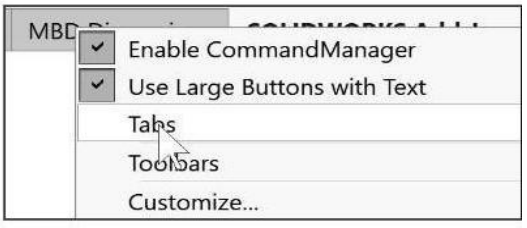


The SOLIDWORKS CommandManager is a Context-sensitive toolbar that automatically updates based on the toolbar you want to access. By default, it has toolbars embedded in it based on your active document type. The available tools are feature and document dependent.

Below is an illustrated CommandManager for a **Drawing** document. Tabs will vary depending on system setup and Add-ins.

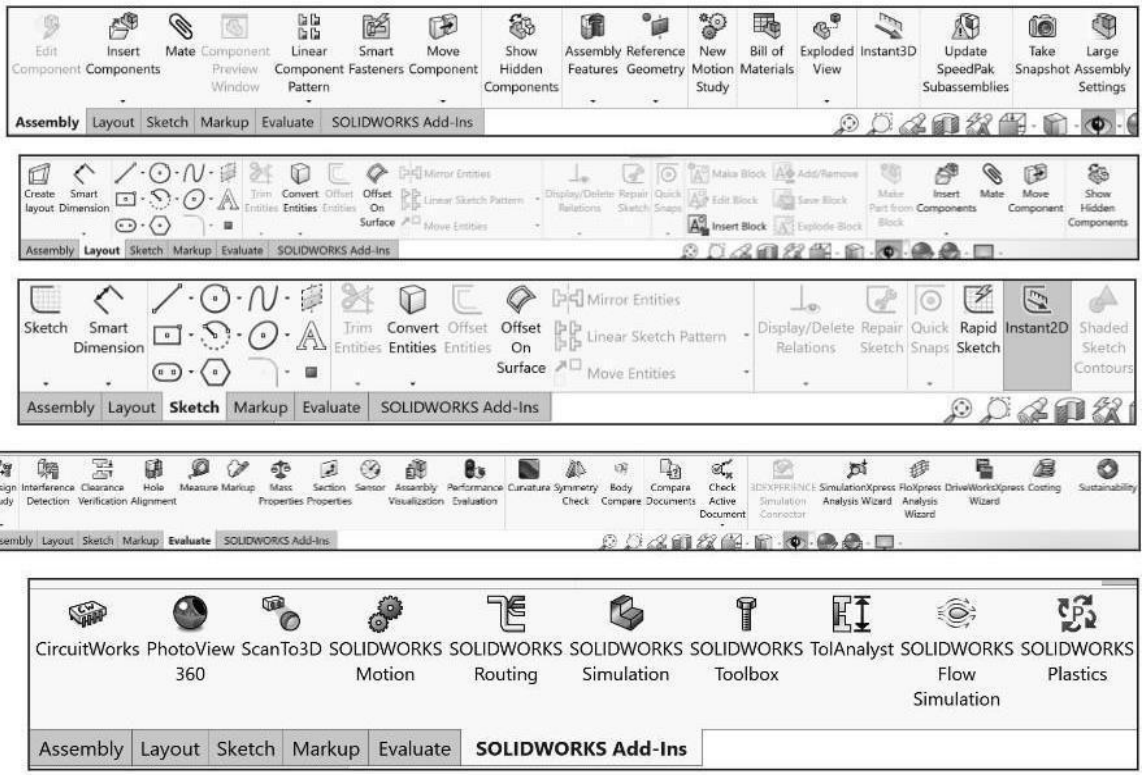


To add a custom tab, right-click on a tab and click Customize. You can also select to add a blank tab and populate it with custom tools from the Customize dialog box.

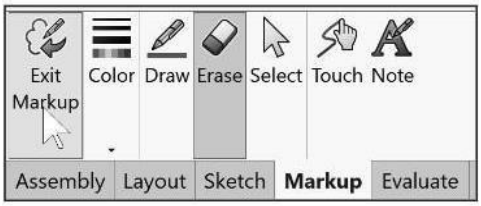


The SOLIDWORKS CommandManager is a Context-sensitive toolbar that automatically updates based on the toolbar you want to access. By default, it has toolbars embedded in it based on your active document type. The available tools are feature and document dependent.

Below is an illustrated CommandManager for an **Assembly** document. Tabs will vary depending on system setup and Add-ins.

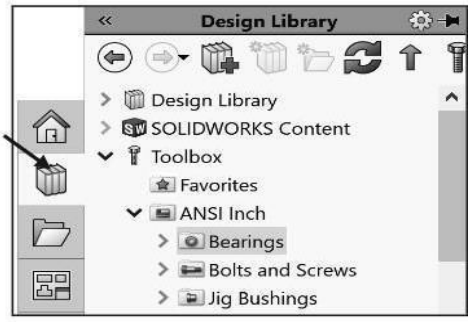


The Markup tab is displayed by default in the CommandManager on some systems. You can draw markups with a mouse on non-touch devices, display bounding boxes for markups, create markups in drawings, and use the context toolbar to access markup options.



The Markup toolbar displays different options depending on the device. **Draw** Draw and **Touch** Touch are **not** available for non-touch devices.

Design Library

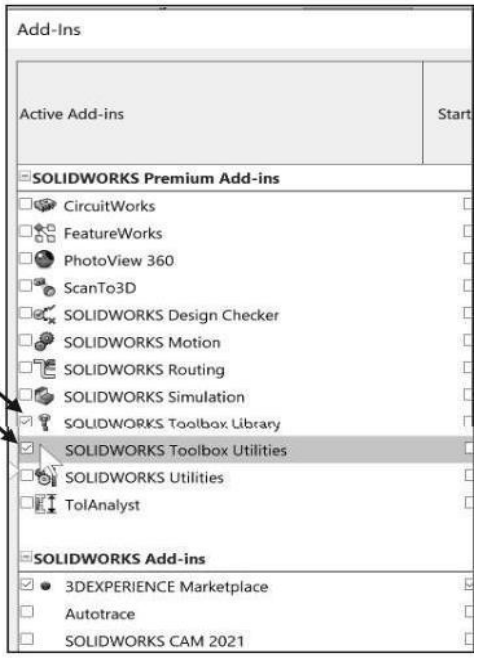


The Design Library contains reusable parts, assemblies, and other elements including library features.

The Design Library tab contains default selections. Each default selection contains additional sub categories.

The default selections are:

- *Design Library.*
- *SOLIDWORKS Content (Internet access required).*
- *Toolbox.*



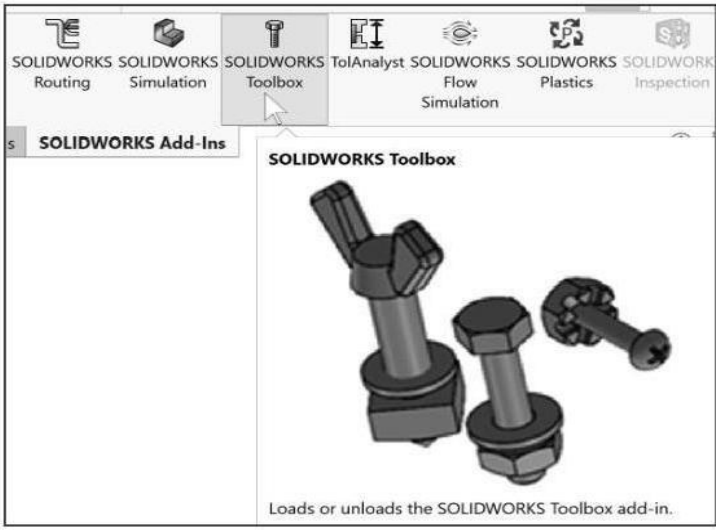
Activate the SOLIDWORKS Toolbox. Click Tools, Add-Ins.., from the Main menu. Check the SOLIDWORKS Toolbox Library and SOLIDWORKS Toolbox Utilities box from the Add-ins dialog box or click SOLIDWORKS Toolbox from the SOLIDWORKS Add-Ins tab.

To access the Design Library folders in a non-network environment, click Add File Location and browse to the needed path. Paths may vary depending on your SOLIDWORKS version and window setup. In a network environment, contact your IT department for system details.

File Explorer

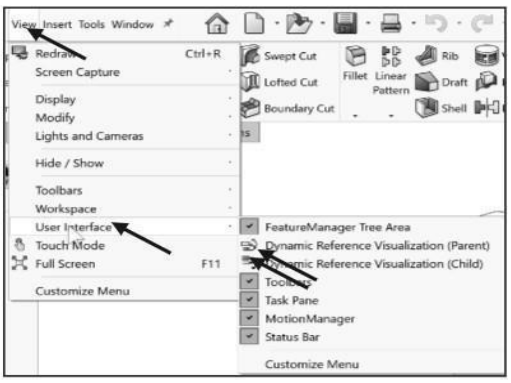
File Explorer duplicates Windows Explorer from your local computer and displays:

- *Recent Documents.*
- *Samples.*
- *Open in SOLIDWORKS.*
- *Desktop.*



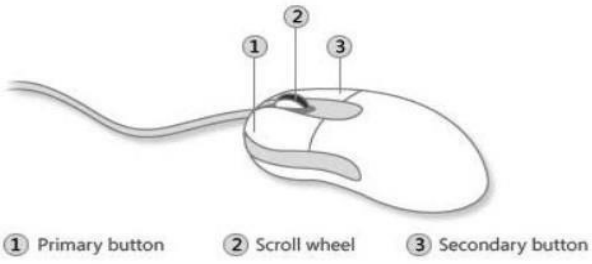
Use Dynamic reference visualization for a part, assembly and mates.

To display the Dynamic Reference Visualization, click **View ➢ User Interface ➢ Dynamic Reference Visualization Parent/Child)** from the Main menu bar.



Mouse Movements

A mouse typically has two buttons: a primary button (usually the left button) and a secondary button (usually the right button). Most mice also include a scroll wheel between the buttons to help you scroll through documents and to Zoom in, Zoom out and rotate models in SOLIDWORKS. It is highly recommended that you use a mouse with at least a Primary, Scroll and Secondary button.



Single-click

To click an item, point to the item on the screen, and then press and release the primary button (usually the left button). Clicking is most often used to select (mark) an item or open a menu. This is sometimes called single-clicking or left-clicking.

Double-click

To double-click an item, point to the item on the screen, and then click twice quickly. If the two clicks are spaced too far apart, they might be interpreted as two individual clicks rather than as one double-click. Double-clicking is most often used to open items on your desktop. For example, you can start a program or open a folder by double-clicking its icon on the desktop.

Right-click

To right-click an item, point to the item on the screen, and then press and release the secondary button (usually the right button). Right-clicking an item usually displays a list of things you can do with the item. Right-click in the open Graphics window or on a command in SOLIDWORKS, and additional pop-up context is displayed.

Scroll wheel

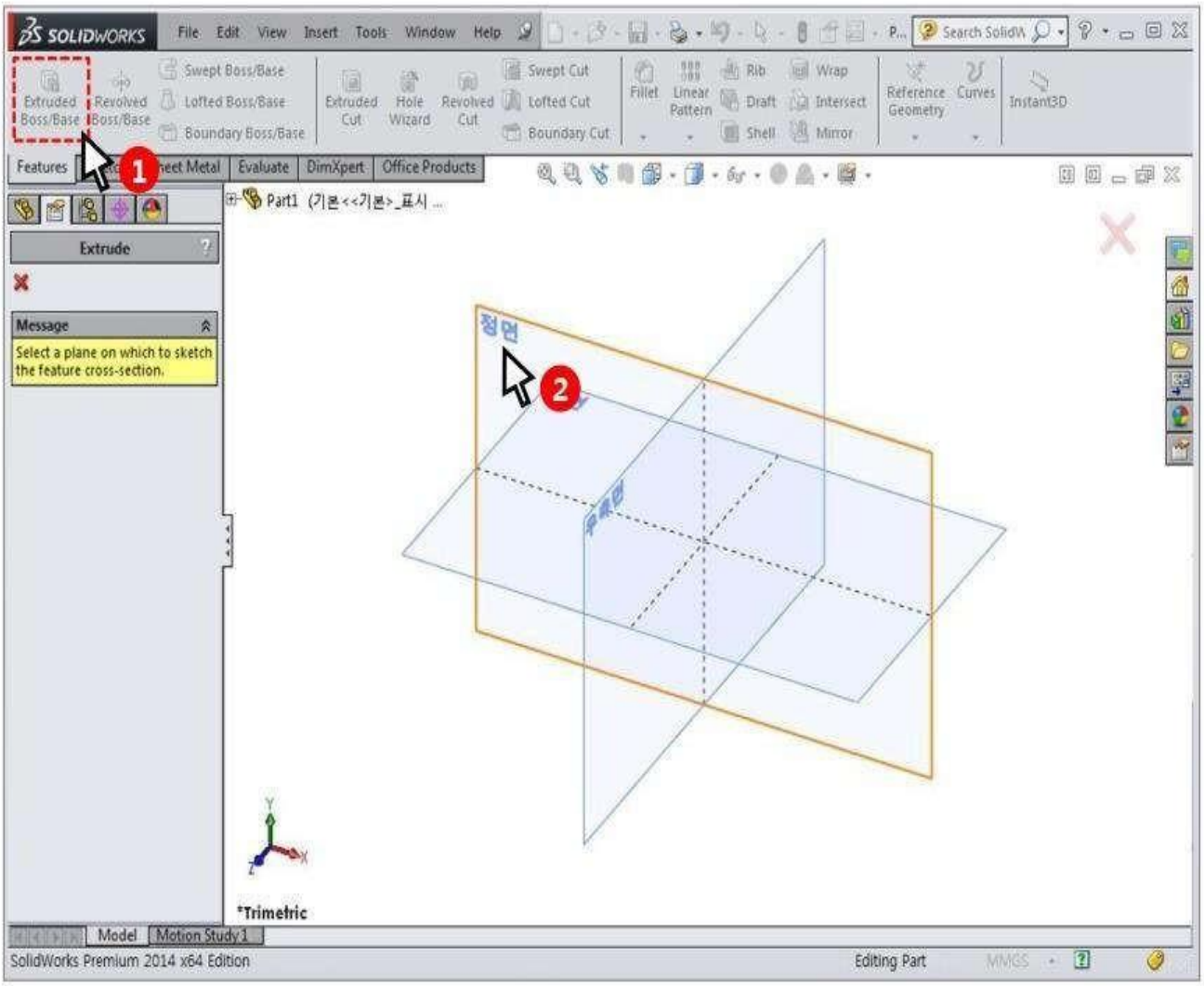
Use the scroll wheel to zoom-in or to zoom-out of the Graphics window in SOLIDWORKS. To zoom-in, roll the wheel backward (toward you). To zoom-out, roll the wheel forward (away from you).

Information Sheet: 2.2.1 (2D Sketch Tools)

Learning Objective: Selection of different planes and some Basic Sketch Tools (Line, circle, point, rectangle, arc, polygon, spline, slot, pattern, split, mirror etc tools)

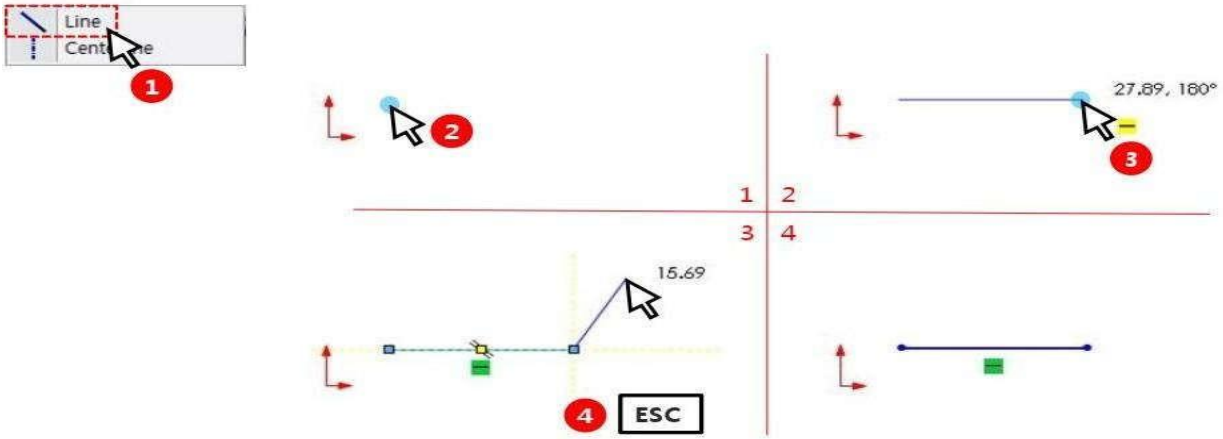
1.

Create a new sketch



2.

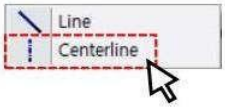
▣ **Line** : Sketches a line.

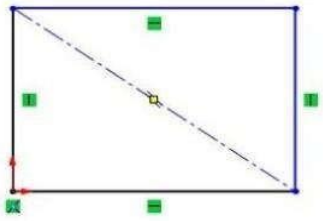


1. In an active 2D sketch, click Sketch tab > Line
2. In the graphics window, click to set the start point.
3. Do one of the following:
   - Continue clicking to create contiguous straight-line segments, or double-click to end the line.

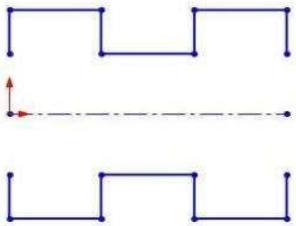
3.

▣ **Centerline** : Sketches a centerline. Use centerlines to create symmetrical sketch element, revolved features, or as construction geometry.

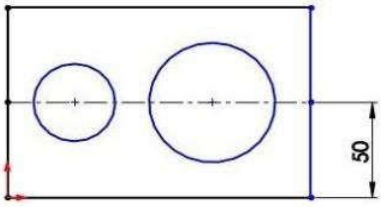




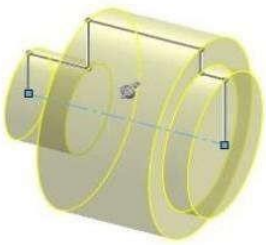
- Centerline to give constraints



- Centerline used for mirror entities



- Centerline to give dimensions

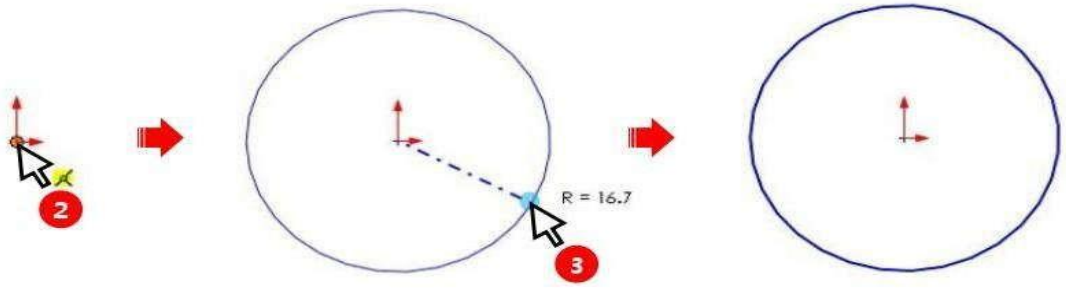


- Centerline for creating rotational features

4.

▣ **Circle** : Sketches a circle from the center point and radius.

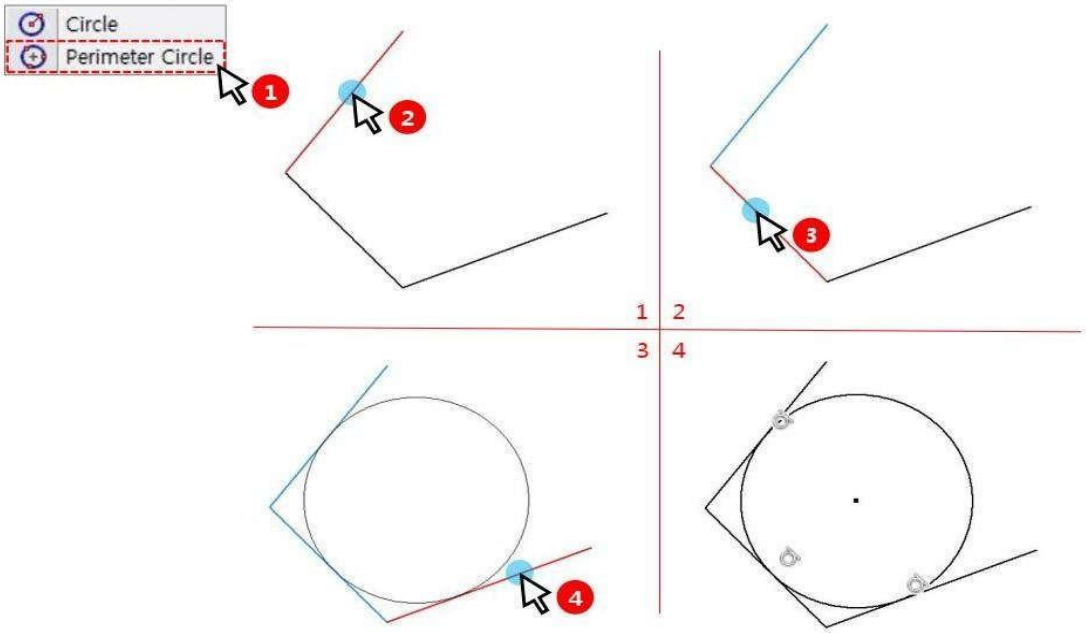




1. Click Sketch tab > Circle
2. Click in the graphics window to set the circle points or tangent lines.
3. Move the cursor to preview the circle radius or tangent lines.
4. Continue to create circles as needed,
5. To quit, press Esc or click another command,

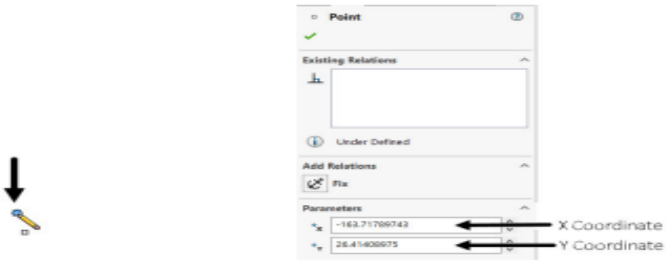
5.

▣ **Perimeter Circle** : Sketches a circle tangent to three lines.



6.

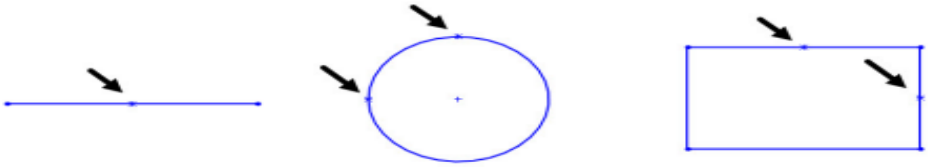
The Point Tool: This tool is used to place sketched point in the drawing area or on any sketch entity while drawing a sketch. ¬ Click on Point tool from the Sketch CommandManager to activate it ¬ Click anywhere in the drawing area to place Point and display the Point PropertyManager, as shown. ¬ Also, the coordinates of point at current location get displayed in the X Coordinate and Y Coordinate spinners, as shown. You can enter the required coordinates in these spinners to change it location.



➢ Now click on the Click Dialog button from the PropertyManager to close it and display the point placed, as shown.

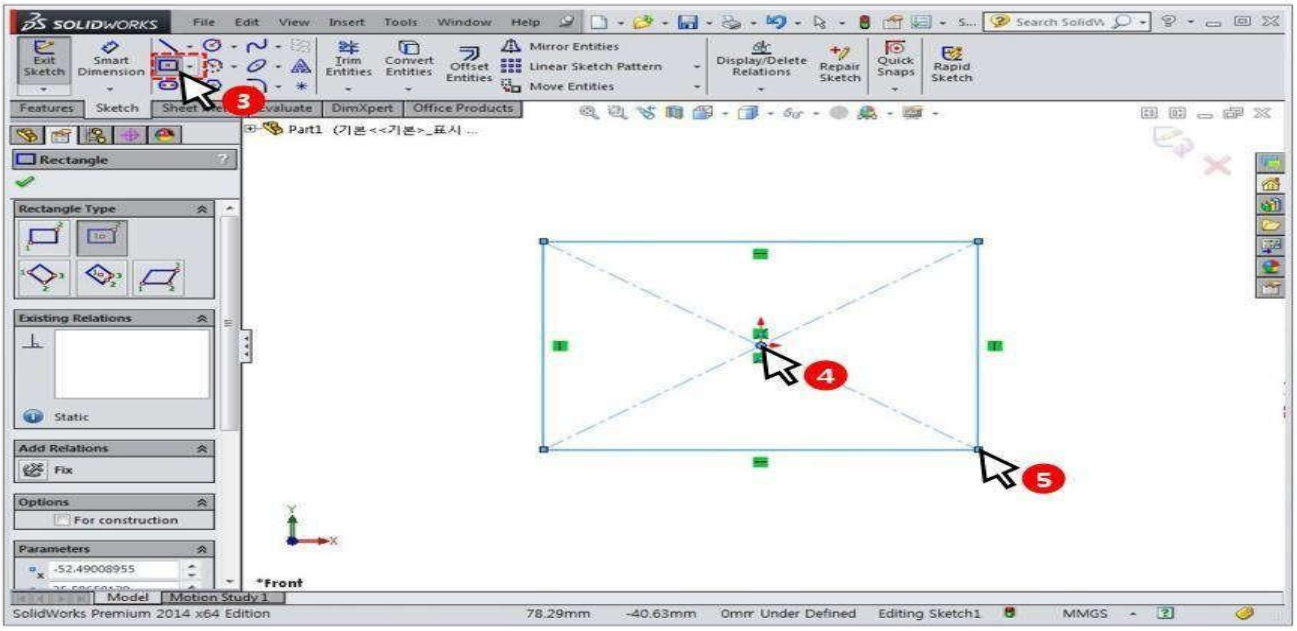


Similarly, you can place the point on other entities also, as shown.



7.

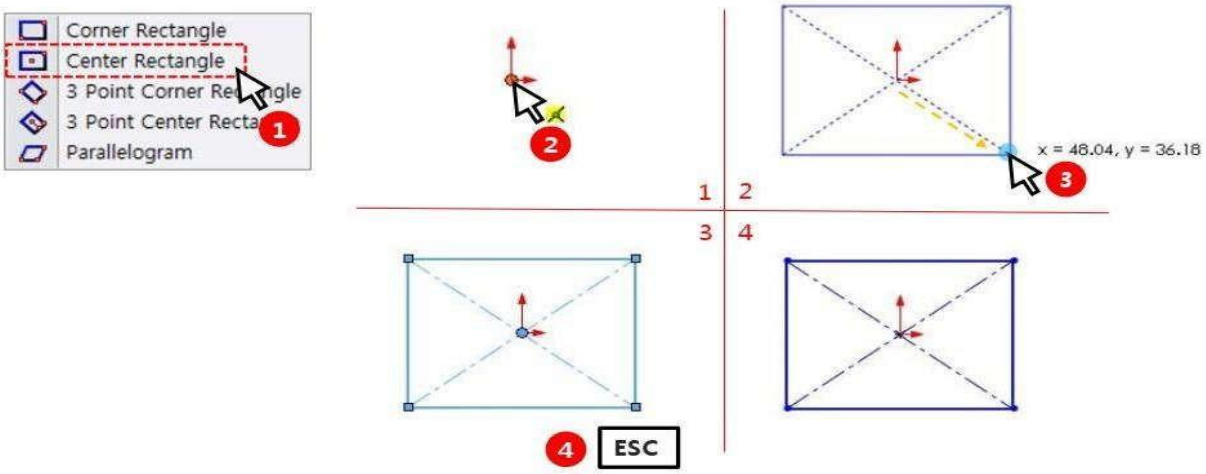
Rectangle



• Sketches a rectangle from the center.

8.

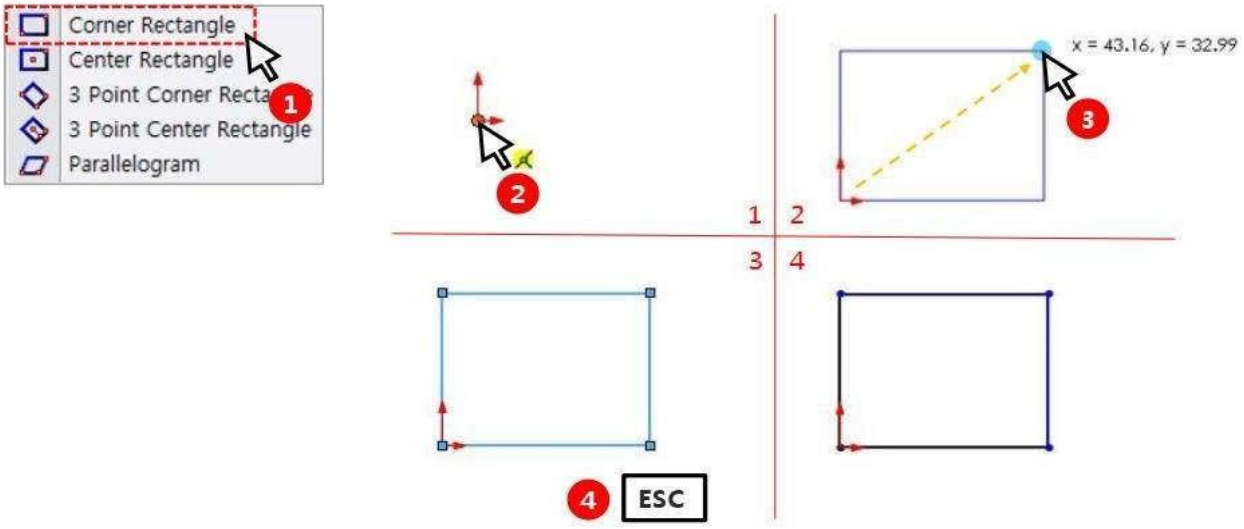
▣ Center Rectangle : Sketches a rectangle from the center.



1. Click Sketch tab > Center Rectangle
2. Click in the graphics window to set the center point.
3. Move the cursor and click to set the second points, depending on the type of rectangle you're drawing.
4. To quit, press Esc or click another command.

9.

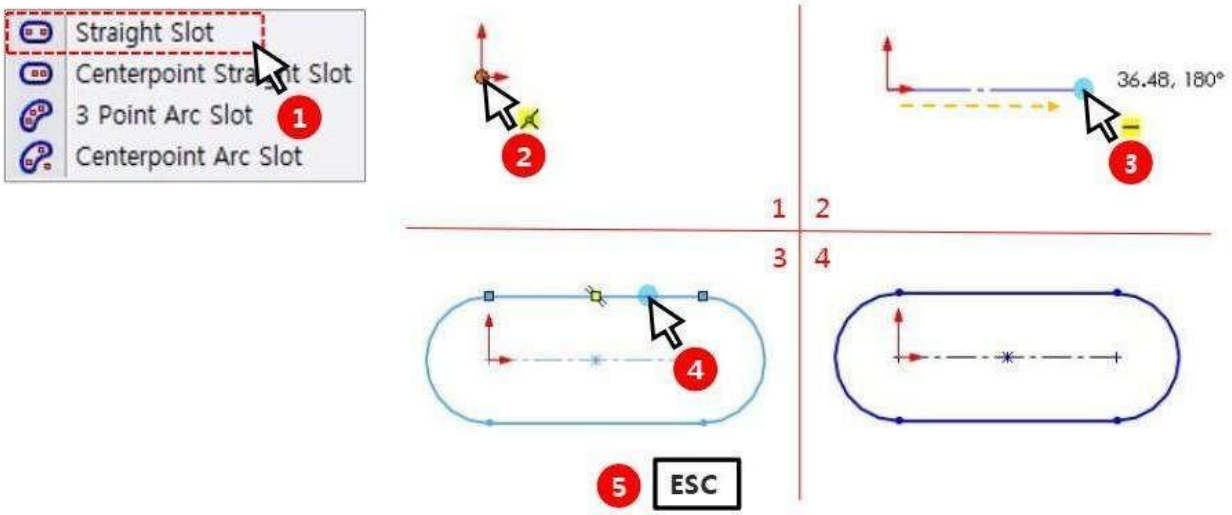
▣ Corner Rectangle : Sketches a rectangle from the corner points.



1. Click Sketch tab > Corner Rectangle
2. Click in the graphics window to set the first point.
3. Move the cursor and click to set the second points, depending on the type of rectangle you're drawing.
4. To quit, press Esc or click another command.

10.

▣ Straight Slot : Sketches a linear slot by defining the placement and distance of slot arc centers, and then the slot width.



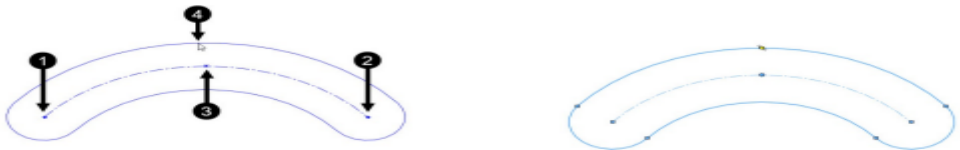
1. Click Sketch tab > Straight Slot
2. Click in the graphics window to set the first point.
3. Move the cursor and click to set the second, third points, depending on the type of slot you're drawing. Alternatively, enter the values in the edit field or right-click and enter a value.
4. To quit, press Esc or click another command.

11.

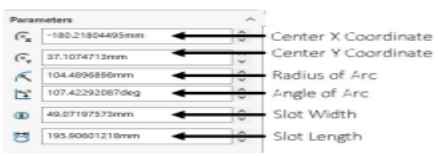
3 Point Arc Slot Tool

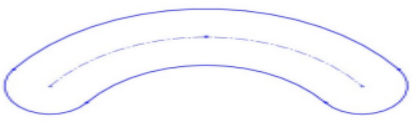
This tool is used to create a 3 Point Arc Slot by defining four points.

- Click on **Slot > 3 Point Arc Slot** tool from the **Sketch CommandManager** to activate it, as shown above.
- Click in the drawing area to define the point (1) as start point of the 3 Point Arc Slot, as shown.
- Move the cursor upto some distance and click to define point (2) as end point of the slot and display the preview of slot, as shown.
- Again, move the cursor upto some distance and click to define point (3) and radius of Slot, as shown.
- Now, move the cursor upto some distance and click to define point (4) for width of Slot and display the slot, as shown.



- Next, you can modify the properties of slot by entering required values in the active respective edit boxes under **Parameters** rollout of the **Slot PropertyManager**, if required, as shown.
- Now select the **ESC** key from keyboard to exit PropertyManager and display the 3 Point Arc Slot, as shown.



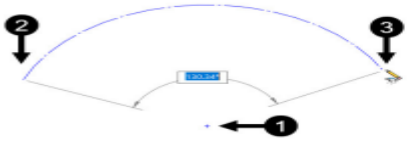


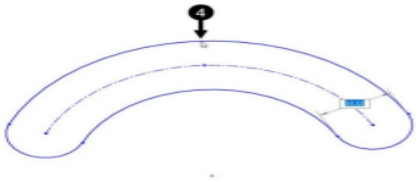
12.

Centerpoint Arc Slot Tool

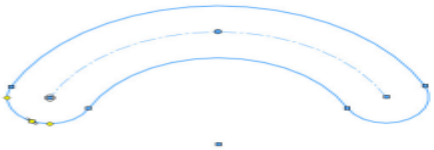
This tool is used to create a straight slot.

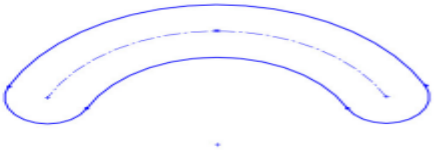
- Click on **Slot > Centerpoint Arc Slot** tool from the **Sketch CommandManager** to activate it, as shown above.
- Click in the drawing area to define the point (1) as center point of the Centerpoint Arc Slot, as shown.
- Move the cursor upto some distance and click to define point (2) as start point of the slot, as shown.
- Move the cursor upto some distance and click to define point (3) as end point of the slot and display the preview of slot, as shown.
- Again, move the cursor upto some distance and click to define point (4) for width of Slot to display preview of the slot, as shown below.





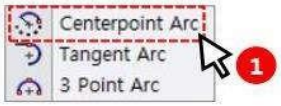
- Next, you can modify the properties of slot by entering required values in the active edit boxes under **Parameters** rollout of the **Slot PropertyManager**, as shown above.
- Now select the **ESC** key from keyboard to exit PropertyManager and display the Centerpoint Arc Slot, as shown.

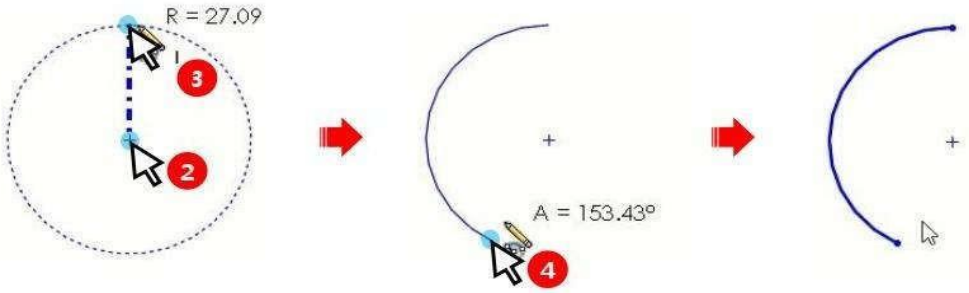




13.

Centerpoint Arc : Sketches a center point arc. Set the center point. Drag to place the arc starting, then to set its length and direction.

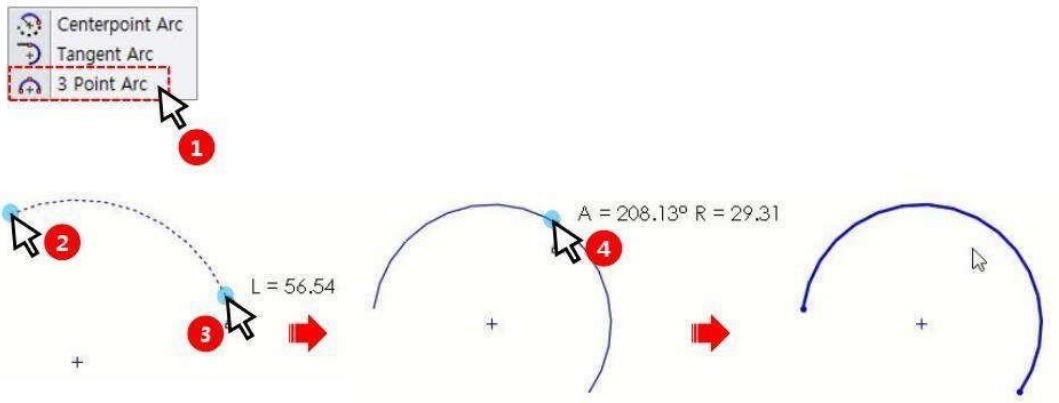




1. Click **Centerpoint Arc** .
2. Click to place the center of the arc.
3. Release and drag to set the radius and the angle.
4. Click to place a start point.
5. Release, drag, and click to set an end point.
6. Click .

14.

▣ **3 Point Arc** : Sketches arcs by specifying three points (start, end, and midpoint).



1. Click **3 Point Arc** .
2. Click to set a start point.
3. Drag the pointer , then click to set an end point.
4. Drag to set the radius.
5. Click to set the arc.
6. Click .

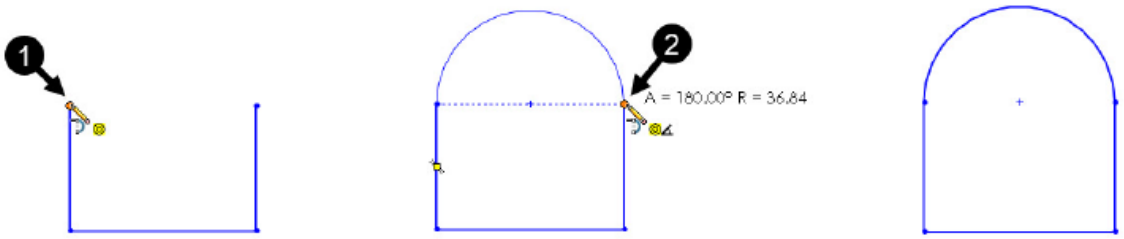
15.

Tangent Arc Tool

This tool is used to draw arc tangent to an existing entity.

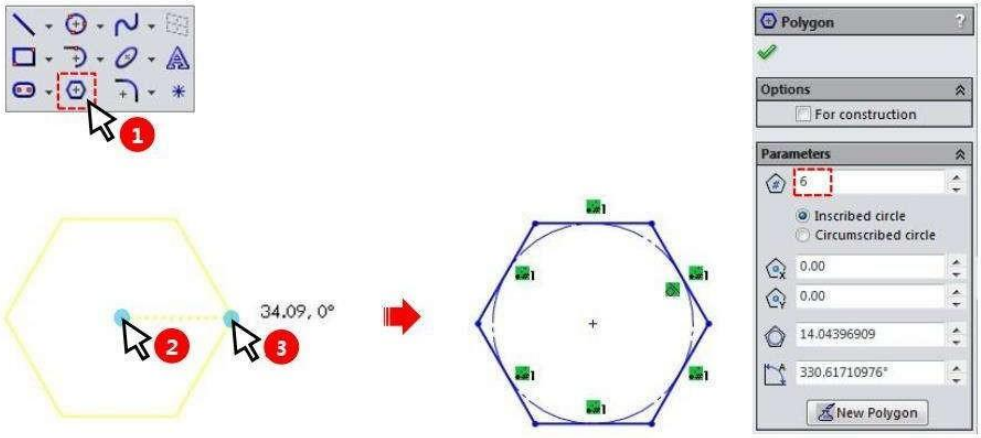
- Select **Arc > Tangent Arc** from the **Sketch CommandManager** to activate it.

  Alternatively, you can also select **Tangent Arc** tool from the **Arc Type** rollout of the **Arc PropertyManager**, as shown above.
- Click on the end point of the left line entity to define start point (1) of the arc, as shown
- Move the cursor slightly right and click on the end point of right line entity to define end point (2) of Arc, as shown.
- Press the **Esc** key twice from the keyboard to exit the tool and display arc created, as shown.



16.

▣ **Polygon** : Create equilateral polygons with any number of sides between 3 and 40.

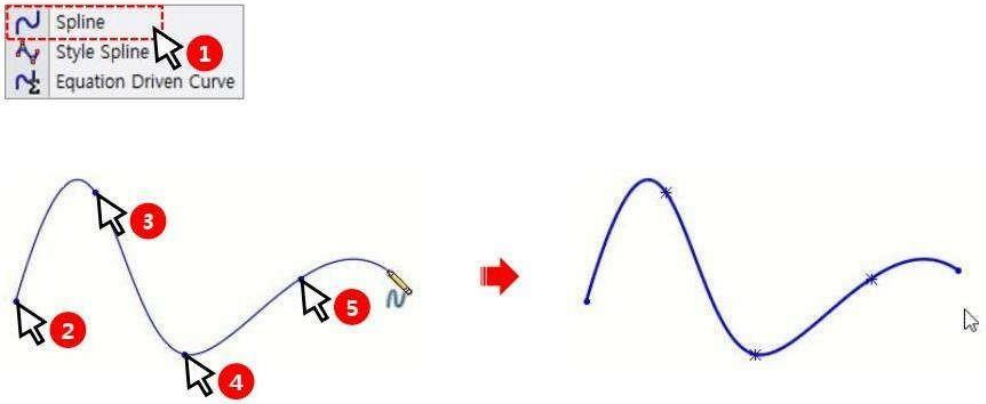


1. Click **Polygon** on the Sketch toolbar, or click **Tools** > **Sketch Entities** > **Polygon**.

   The pointer changes to .
2. Set the properties in the Polygon PropertyManager as necessary.
3. Click in the graphics area to place the center of the polygon, and drag out the polygon.

17.

▣ **Spline** : Sketches a spline. Click to add spline points that shape the curve.

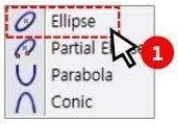


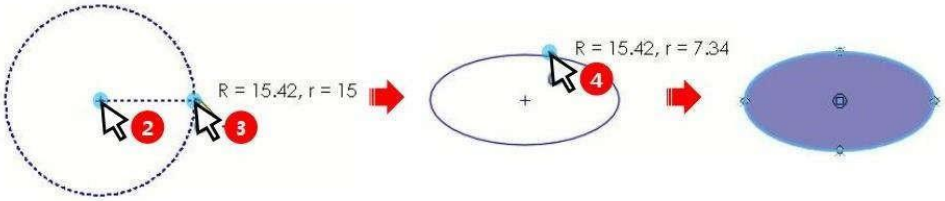
1. Click **Spline** (Sketch toolbar) or **Tools** > **Sketch Entities** > **Spline**.

   The pointer changes to .
2. Click to place the first point and drag out the first segment.
3. Click the next point and drag out the second segment.
4. Repeat for each segment, then double-click when the spline is complete.

18.

▣ **Ellipse** : Sketches a complete ellipse. Select the ellipse center, then drag to set the major and minor axes.





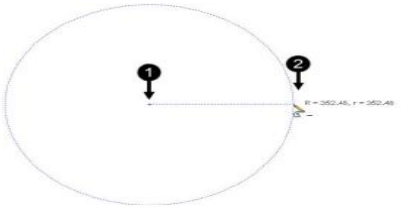
1. Click **Ellipse** on the Sketch toolbar or **Tools** > **Sketch Entities** > **Ellipse**.

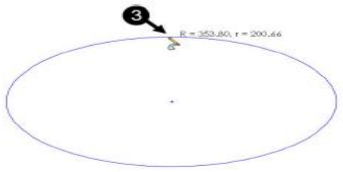
   The pointer changes to .
2. Click in the graphics area to place the center of the ellipse.
3. Drag and click to set the major axis of the ellipse.
4. Drag and click again to set the minor axis of the ellipse.

Partial Ellipse Tool

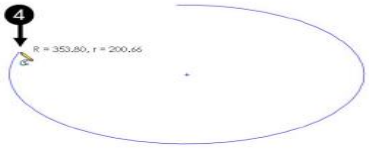
This tool is used to create a partial ellipse in the sketch environment.

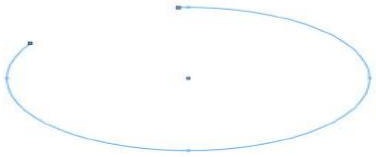
- Select **Ellipse > Partial Ellipse** tool from the **Sketch CommandManager**, as shown above.
- Move the cursor horizontally and click to define point (1) as the center of ellipse, as shown.
- Move the cursor vertically and click to define point (2) for the radius along major axis, as shown.
- Next, click to define point (3) for the radius along minor axis and start point of ellipse, as shown.



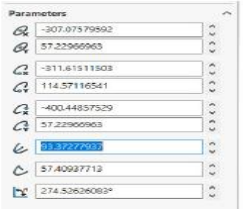


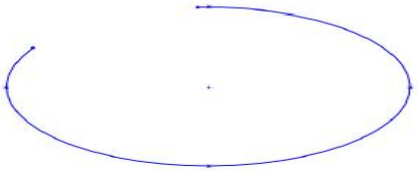
- Now click to define point (4) as the end point of the ellipse and display the preview of partial ellipse, as shown.





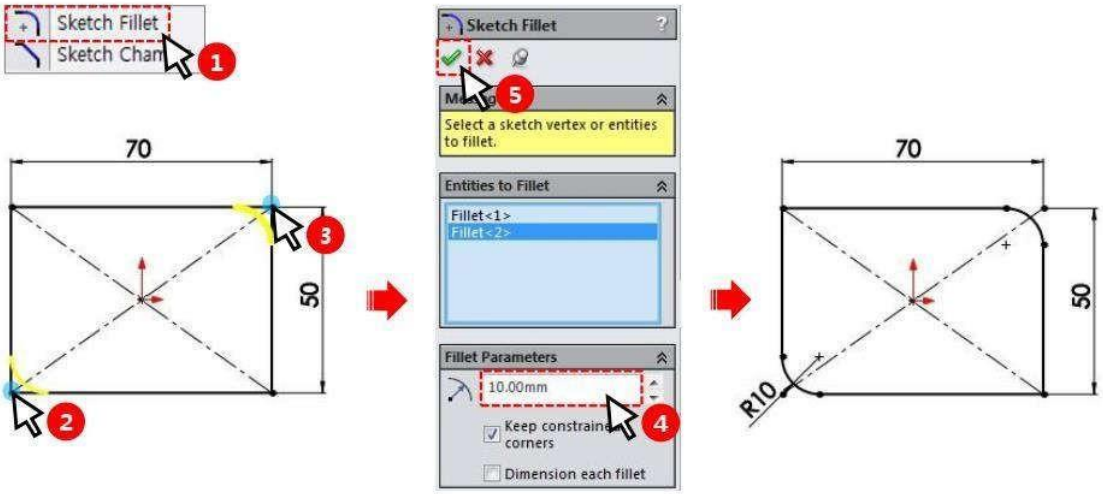
- Next, you can enter the required values in the respective edit boxes available under **Parameters** rollout of **Ellipse Property Manager** if required, as shown.
- Click on the button from the PropertyManager to exit it and display the ellipse, as shown.





19.

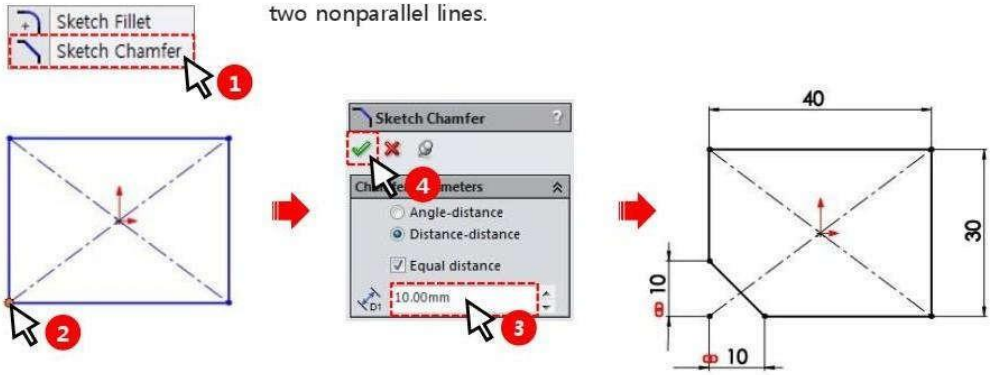
▣ Sketch Fillet : Sketches rounded corners (vertices) at the intersection of two selected lines.



1. In an open sketch, click **Sketch Fillet** on the Sketch toolbar, or **Tools** > **Sketch Tools** > **Fillet**.
2. Set the properties in the Sketch Fillet PropertyManager.
3. Select the sketch entities to fillet.
4. Drag the preview to adjust the fillet size if necessary.
5. Click ✔ to accept the fillet.

20.

▣ Sketch Chamfer : The Chamfer tool creates a beveled corner at the intersection of any two nonparallel lines.

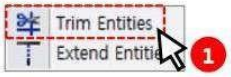


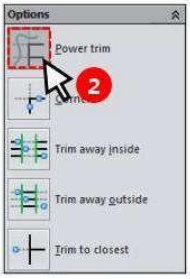
1. In an open sketch, click **Sketch Chamfer** on the Sketch toolbar, or click **Tools** > **Sketch Tools** > **Chamfer**.
2. In the PropertyManager, set the **Chamfer Parameters** as necessary.
3. In the graphics area, select the two sketch entities to chamfer. To select the sketch entities, you can:
   - Hold **Ctrl** and select two sketch entities.
   - Select a vertex.

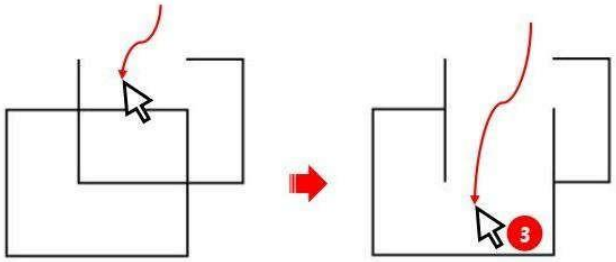
   The chamfer is applied immediately.
4. Click ✔ to accept the chamfer, or click **Undo** to remove the chamfer. You can undo a sequence of chamfers in reverse order.

21.

▣ Trim Entities (Power trim)







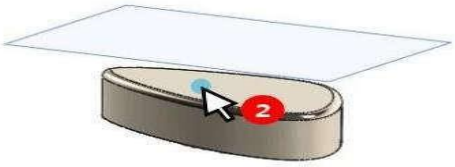
1. Right-click the sketch and select **Edit Sketch**.
2. Click **Trim Entities** (Sketch toolbar) or **Tools** > **Sketch Tools** > **Trim**.
3. In the PropertyManager, under **Options**, select **Power trim** .
4. Click in the graphics area next to the first entity, and drag across the sketch entity to trim.

   The pointer changes to as it crosses and trims the sketch entity. A trail is created along the trim path.
5. Continue to hold down the pointer and drag across each sketch entity you want to trim.
6. Release the pointer when finished trimming the sketch, then click .

22.

▣ Convert Entities : You can create one or more curves in a sketch by projecting an edge, loop, face, curve, or external sketch contour, set of edges, or set of sketch curves onto the sketch plane.





Sketch plane above part



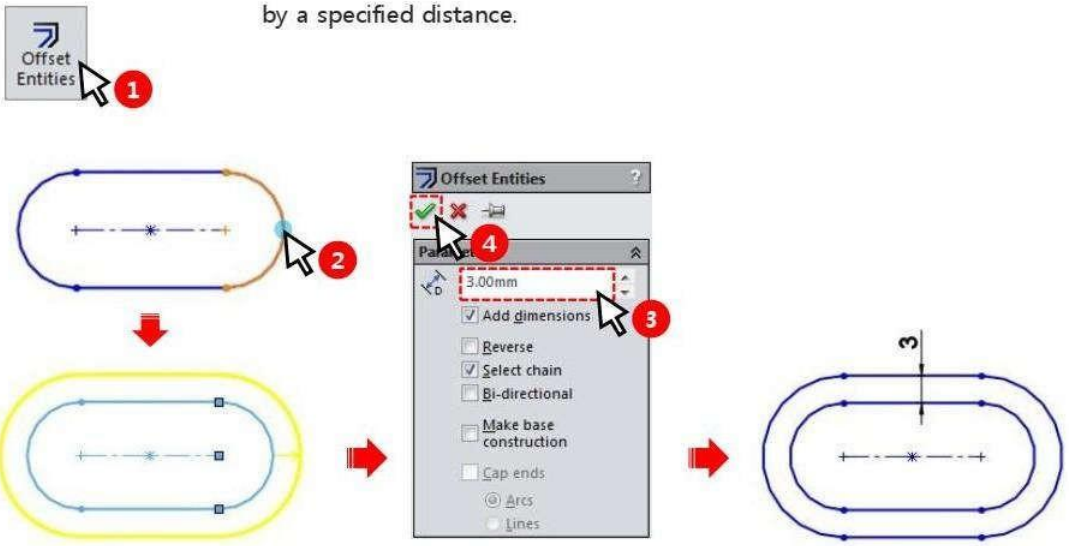
Loop of edges projected onto plane using **Convert Entities**

1. In an open sketch, click a model edge, loop, face, curve, external sketch contour, set of edges, or set of curves.

   You can also click an entity after clicking the **Convert Entities** tool.
2. Click **Convert Entities** (Sketch toolbar) or **Tools** > **Sketch Tools** > **Convert Entities**.
3. In the PropertyManager, click **Select chain** to convert all contiguous sketch entities.
4. Click .

23.

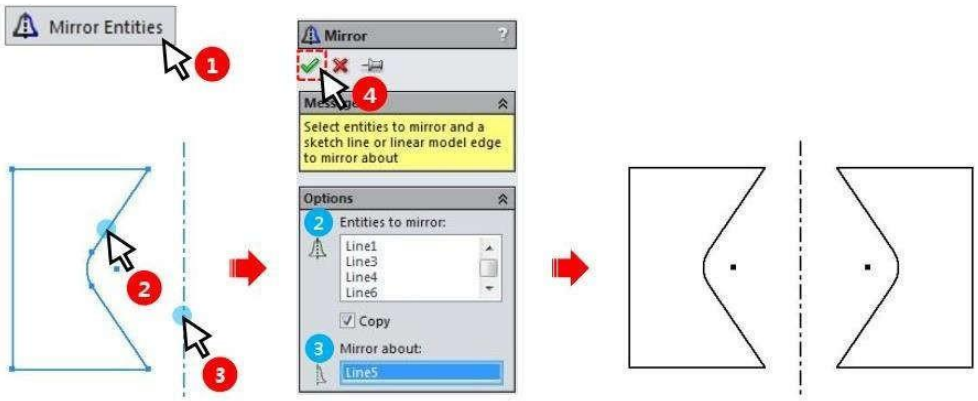
▣ Offset Entities : Offset one or more sketch entities, selected model edges, or model faces by a specified distance.



1. In an open sketch, select one or more sketch entities, a model face, or a model edge.
2. Click **Offset Entities** (Sketch toolbar) or **Tools** > **Sketch Tools** > **Offset Entities**.
3. Set the properties in the Offset Entities PropertyManager. When you click in the graphics area, the offset entity is complete. Set the properties before you click in the graphics area.
4. Click ✔ or click in the graphics area.

24.

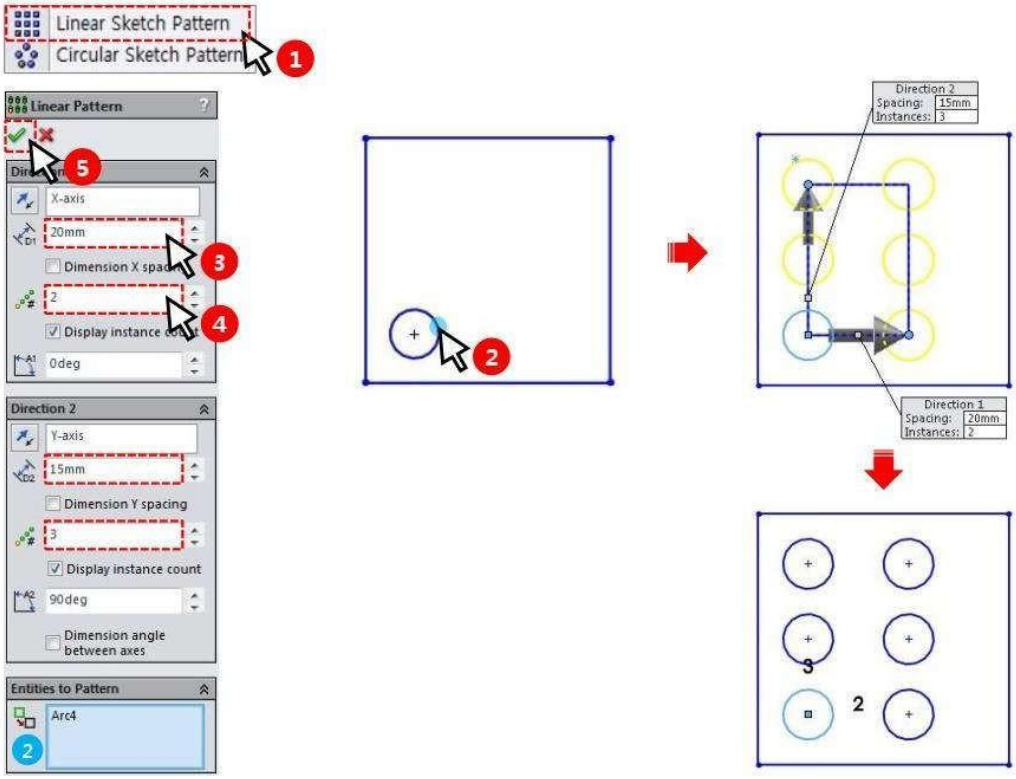
▣ Mirror Entities



1. In a sketch, click **Mirror Entities**
2. In the graphics window, select the geometry to pattern.
3. In the Mirror Pattern dialog box, click the selection tool under Mirror Line (2D) or Mirror Plane (3D) and then select a line or plane to about which to mirror the geometry.
   In the graphics window, a blue arrow indicates the line or plane.
4. Click Apply to finish the pattern and create more or click Done to create the pattern and exit the command.

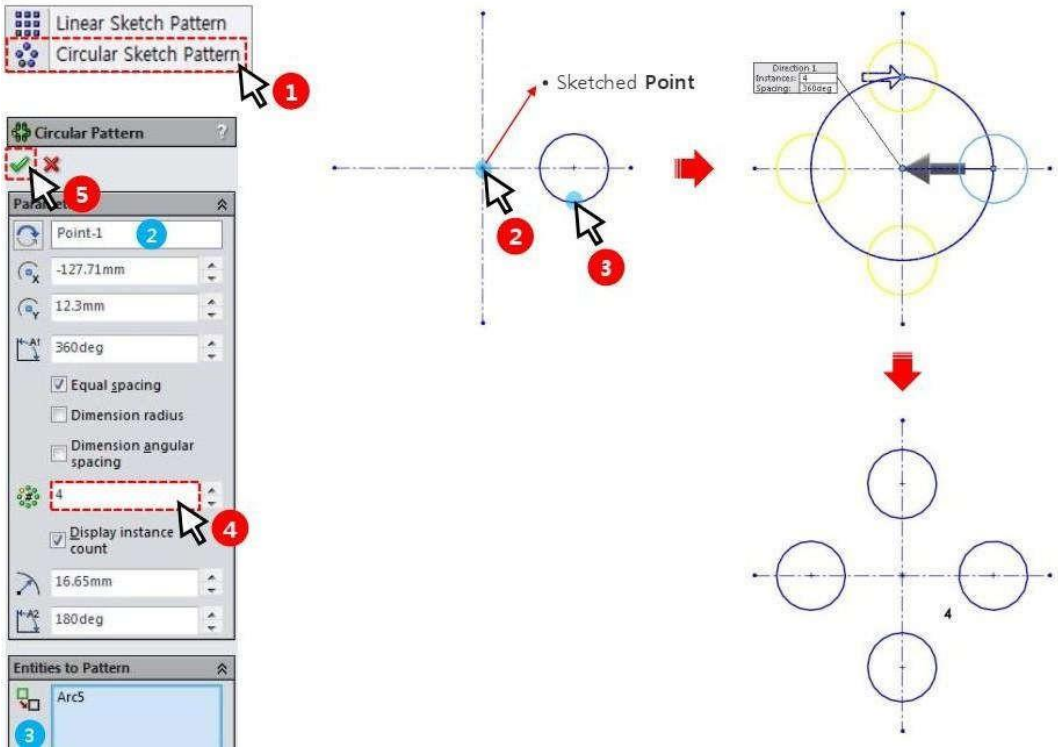
25.

▣ Linear Sketch Pattern



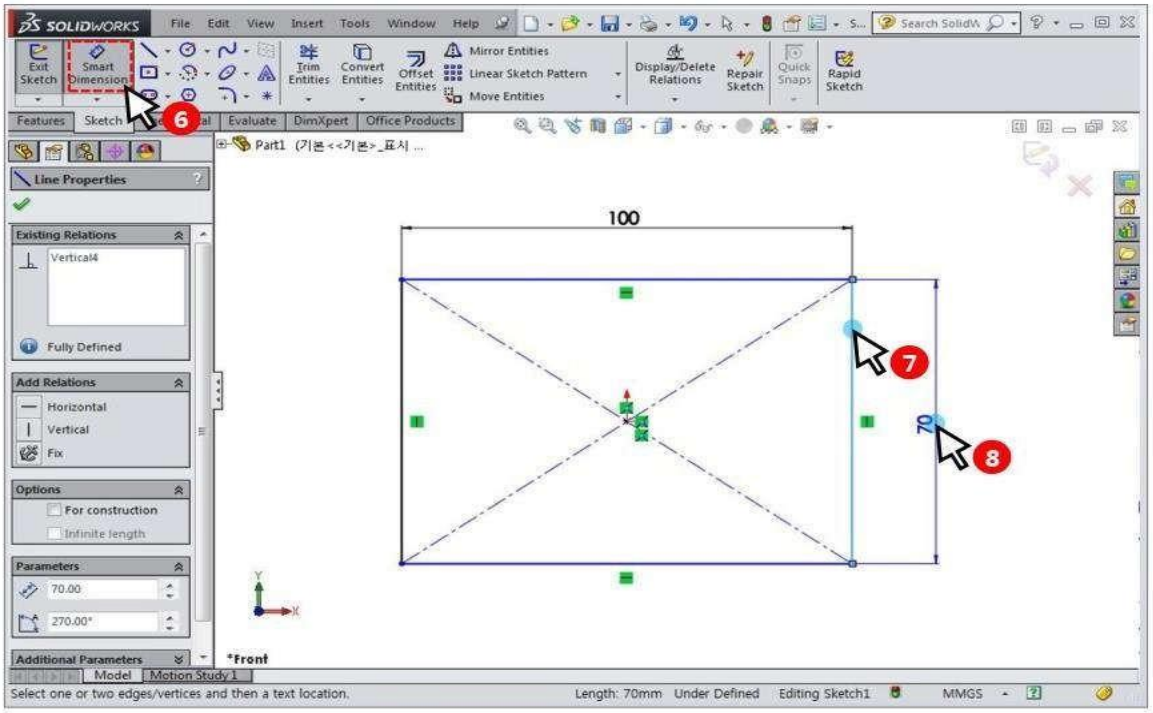
26.

▣ Circular Sketch Pattern



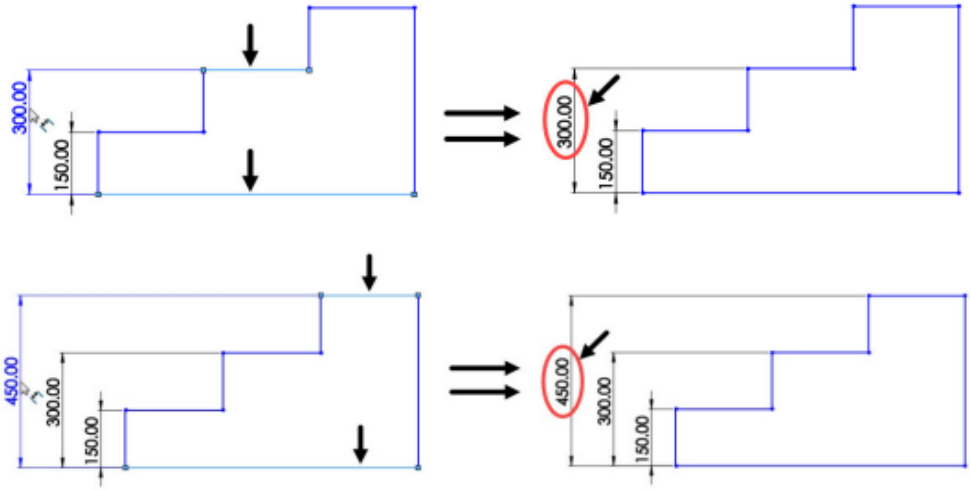
27.

Smart Dimension

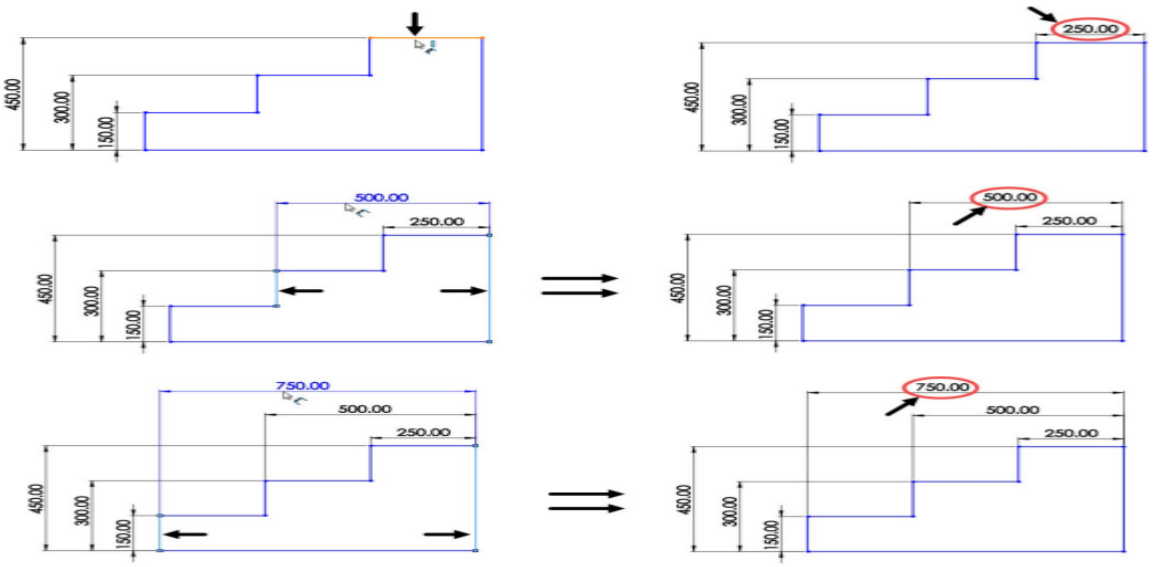


- Creates a dimension for one or more selected entities.

You can also generate vertical dimension by selecting two horizontal line entities or vice-versa, as shown.

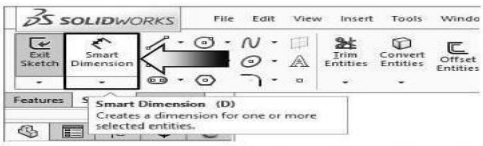


Similarly, you can generate horizontal dimensions by selecting two respective vertical line entities, as shown



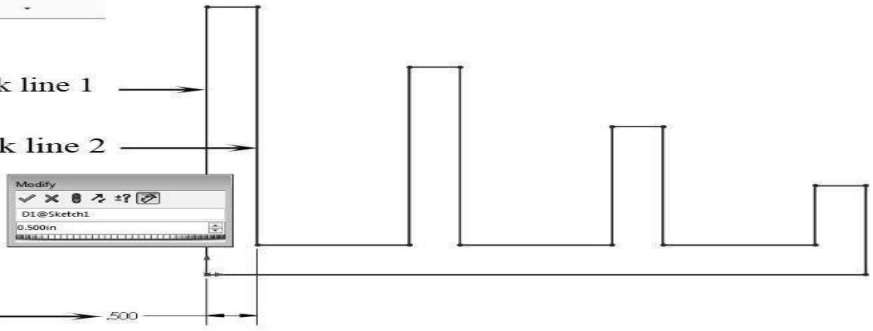
5. Adding the horizontal dimensions:

Select  from the Sketch toolbar - OR - select **Insert / Dimension**, and add the dimensions shown below (follow the 3 steps A, B and C).



A. Click line 1

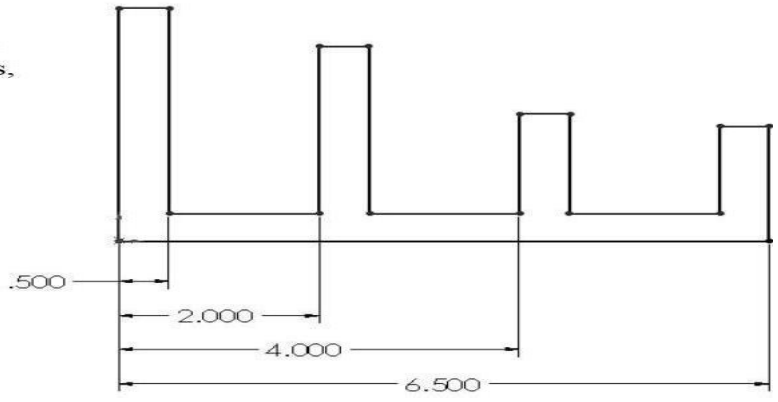
B. Click line 2



C. Place the dimension approximately here, type **.500** and press enter.

The Inch-Units is filled in automatically because it has been set previously to Inches, 3 decimal places.

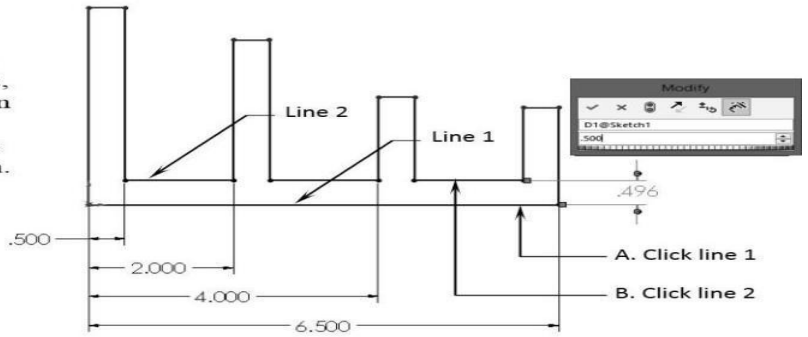
Continue adding the horizontal dimensions as shown here.



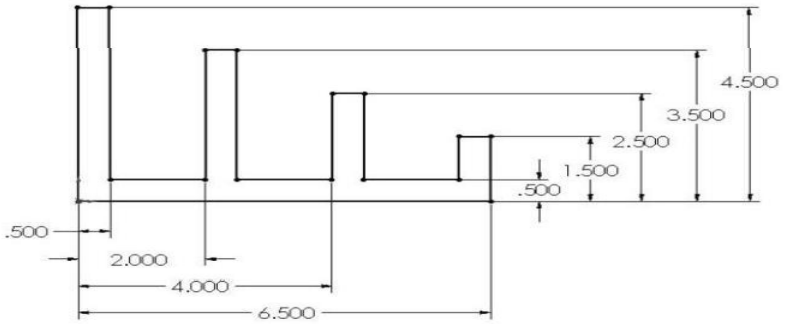
NOTE:
The color of the sketch lines changes from Blue to Black, to indicate that they have been constrained with dimensions.

6. Adding the Vertical dimensions:

With the Smart-Dimension tool still selected, click on line 1 and line 2, place the dimension approximately as shown, and change the value to **.500 in**.



Continue adding other dimensions until the entire sketch turns into the Black color.



The Status of a Sketch:

The current status of a sketch is displayed in the lower right corner of the screen.

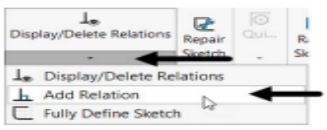
Fully Defined	=	**Black**	Fully Defined
Under Defined	=	**Blue**	Under Defined
Over Defined	=	**Red**	Over Defined

2.2.1 Geometric Relation Tools:

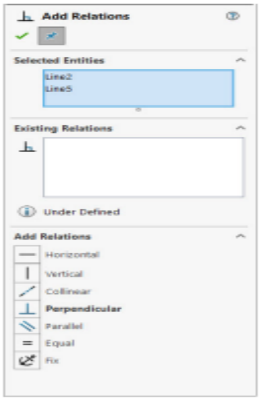
Geometric Relations / Constraints

The geometric relations/constraints are logical operations that are used to control the position and shape/size of the drawing, by applying relationships between sketch elements. Before moving to create a feature, it is considered a good practice to ensure that the sketch is fully constrained. The term, 'fully-constrained' means that the sketch has a definite shape and size. You can make a sketch fully-constrained by using dimensions and applying constraints. These relations/constrains can be applied after activating **Add Relation** option and displaying the **Add Relations** dialog box, as shown.

- Select the **Add Relation** option from the **Display/Delete Relations** flyout to display the **Add Relations PropertyManager**, as shown.
- Next, after clicking on any entity or entities, the respective geometric relations get visible in the PropertyManager, as shown.
- Next you can apply the required relation by selecting it from the dialog box.







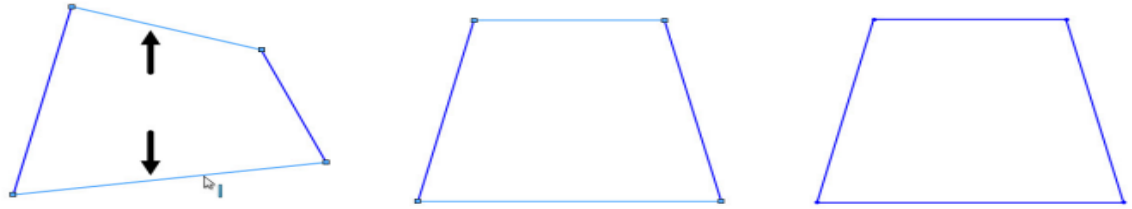
1.

Horizontal

This relation is used to make entities horizontal or apply horizontal relation among selected entities.

1. Select the **Add Relation** option (**Sketch** > **Display/Delete Relations** > **Add Relation**) to activate it and display the **Add Relations PropertyManager**, as shown above.
2. Select the line entity/entities, as shown.
3. Next click on the **Horizontal** button from the PropertyManager to apply and display entities with Horizontal relation, as shown.
4. Click on the **OK** button of the PropertyManager to exit it and click to display entities with horizontal relations, as shown.

You can also select an external entity such as an edge, plane, axis, or sketch curve on an external sketch that will act as a line to apply this relation.

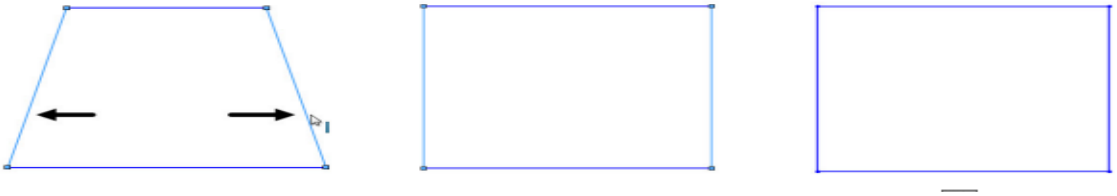


2.

Vertical

This relation is used to make entities vertical or apply vertical relation among selected entities.

- Select the **Add Relation** option to activate it and display the **Add Relations PropertyManager**, as shown above.
- Select the line entity/entities, as shown.
- Next click on the **Vertical** button from the PropertyManager to apply and display entities with vertical relation, as shown.
- Click on the **OK** button of the PropertyManager to exit it and click to display entities with vertical relations, as shown.



3.

Collinear

This relation is used to make the selected line to lie on the same straight/infinite line.

- Select the **Add Relation** option to activate it and display the **Add Relations PropertyManager**.
- Select the entities one by one, as shown.
- Next click on the **Collinear** button from the PropertyManager to apply collinear relation.
- Click on the **OK** button of the PropertyManager to exit it and display entities with collinear relation and symbol, as shown.

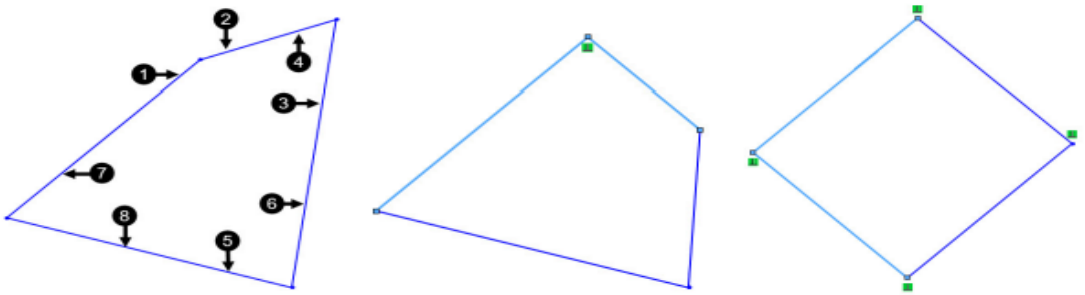


4.

Perpendicular

This relation is used to make two lines perpendicular to each other.

- Select the **Add Relation** option to activate it and display the **Add Relations PropertyManager**, as shown above.
- Click on line 1 & 2 one by one to apply Perpendicular Constraint among them, as shown.
- Next click on the **Perpendicular** button from the PropertyManager to apply perpendicular relation and display entities with symbol as shown.
- Similarly, you can apply perpendicular relation between other line by clicking them in the sequence 3 & 4, 5 & 6, and 7 & 8 as shown.
- Click on the **OK** button of the PropertyManager to exit it and display all entities with perpendicular relation and symbol, as shown.

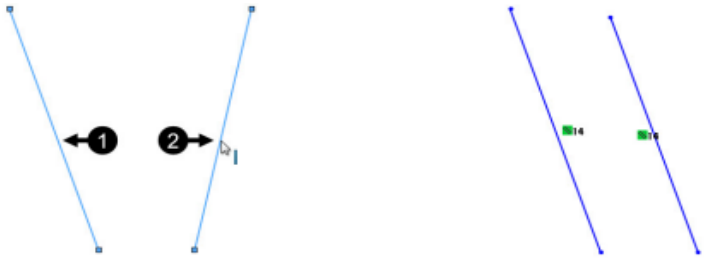


5.

Parallel

This relation is used to make two lines parallel to each other.

- Select the **Add Relation** option to activate it and display the **Add Relations PropertyManager**.
- Select the entities one by one, as shown.
- Next click on the **Parallel** button from the PropertyManager to apply parallel relation.
- Next, click on the **OK** button of the PropertyManager to exit it and display entities with parallel relation and symbol, as shown.



6.

Equal

This relation is used to make the dimensions of selected similar entities equal to each other like equal length for line entities and equal radii for selected arcs & circles.

- Select the **Add Relation** option to activate it and display the **Add Relations PropertyManager**.
- Select the entities one by one, as shown.
- Next click on the **Equal** button from the PropertyManager to apply equal relation.
- Next, click on the **OK** button of the PropertyManager to exit it and display entities with equal relation and symbol, as shown.



- Similarly, you can apply equal relation to line entities to make their length equal, as shown.



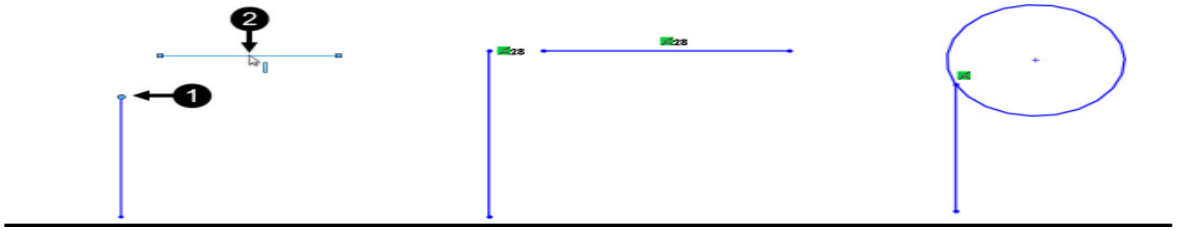
7.

Coincident

This relation is used to make a line entity coincident to selected point entity.

- Select the **Add Relation** option to activate it and display the **Add Relations PropertyManager**.
- Select the end point of the line entity and then select another line entity, as shown.
- Next click on the **Coincident** button from the PropertyManager to apply coincident relation.
- Click on the **OK** button of the PropertyManager to exit it and display entities with coincident relation and symbol, as shown.

Similarly, you can apply coincident relation between arc, circle, and ellipse with the selected point, as shown.



8.

Tangent

This relation is used to make an arc, circle, or line entity tangent to each other.

- Select the **Add Relation** option to activate it and display the **Add Relations PropertyManager**.
- Select the entities one by one, as shown.
- Next click on the **Tangent** button from the PropertyManager to apply tangent relation.

- Click on the **OK** button of the PropertyManager to exit it and display entities with tangent relation and symbol, as shown.



- Similarly, apply tangent relation among other entities, as shown.



9.

Midpoint

This relation is used to force a point to get aligned with the midpoint of a line entity.

- Select the **Add Relation** option to activate it and display the **Add Relations PropertyManager**.
- Click on the center point of the circle entity and then click on the line entity, as shown.
- Next click on the **Midpoint** button from the PropertyManager to apply midpoint relation.
- Click on the **OK** button of the PropertyManager to exit it and display entities with midpoint relation and symbol, as shown.



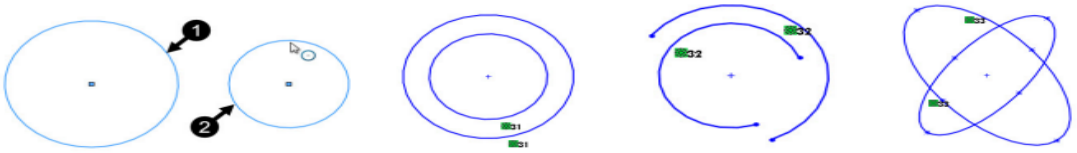
10.

Concentric

This constraint is used to make the center points of arcs, circles or ellipses coincident.

- Select the **Add Relation** option to activate it and display the **Add Relations PropertyManager**.
- Click on the two circles one by one, as shown.
- Next click on the **Concentric** button from the PropertyManager to apply concentric relation.
- Click on the **OK** button of the PropertyManager to exit it and display entities with concentric relation and symbol, as shown.

Similarly, you can apply coincident relation between arcs and ellipse, as shown.



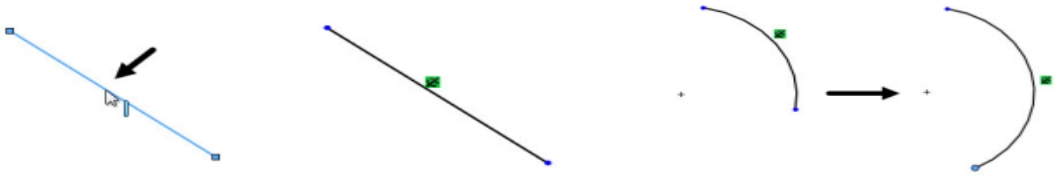
11.

Fix

This relation is used to fix the position of the selected entity.

- Select the **Add Relation** option to activate it and display the **Add Relations PropertyManager**.
- Click on the line entity, as shown.
- Next click on the **Fix** button from the PropertyManager to apply fix relation.
- Click on the **OK** button of the PropertyManager to exit it and display entities with fix relation and symbol, as shown.

If you apply fix relation to line or arc entities, you can change their sizes by dragging their endpoints, as shown.

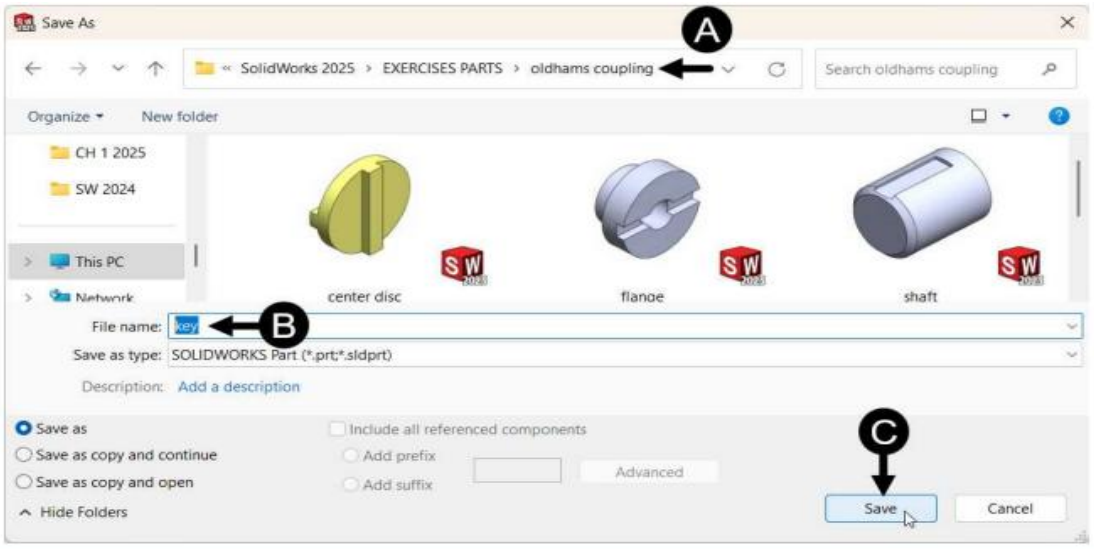


Save the sketch

Save

The **Save** button is used to save the model created in various SolidWorks environments, in the desired folder.

- After creating any model, click on the **Save** button to display the **Save As** dialog box, as shown.
- Browse to the desired folder and enter the desired name in the **File name** edit box, if required.
- Next, click on the **Save** button to save the file in the selected folder, as shown.



Note: When you click on the **Save** button, the **Save As** dialog box with the previously selected folder get displayed. If you click on the **Save** button the object gets saved in that folder. You can browse to any other folder/location and save the object there, if required.

2.2.2 Self-check

1. What happens to a sketch entity when it becomes "fully defined?"
2. What is the purpose of the "Convert Entities" tool?
3. What tool is used to remove excess lines or segments from a sketch?
4. What is the difference between a geometric relation and a dimension?
5. What tool should be used to make a copy of a sketch entity?
6. How to Save SolidWorks file?
7. Why do we need tangent relation?
8. How to use Parallel relation?

2.2.3 Answer key:

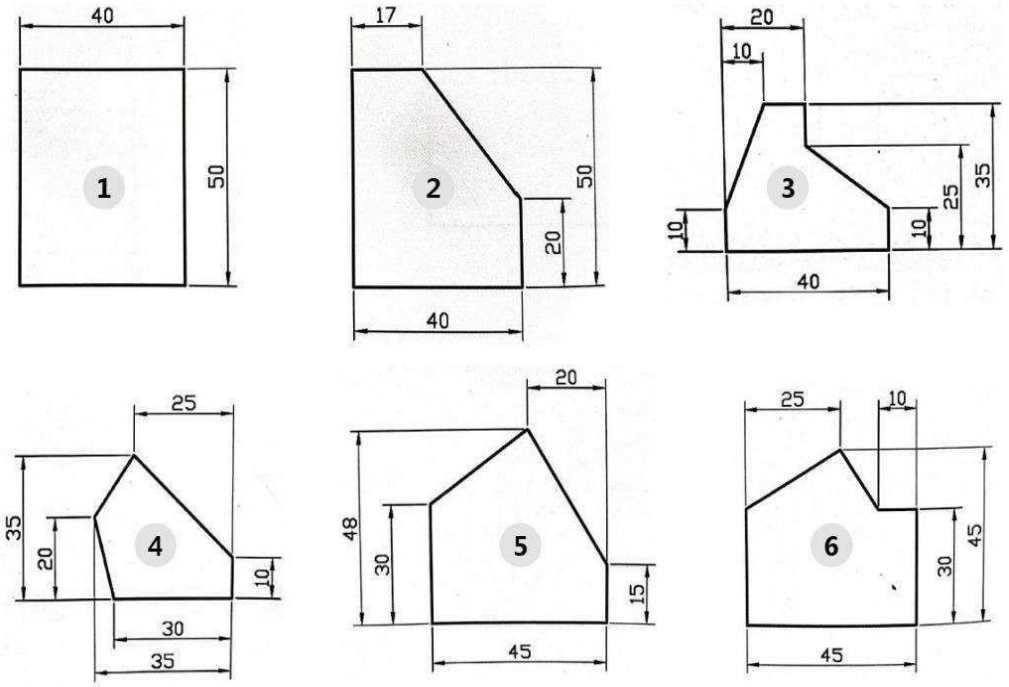
Self-Check Questions	Answer Key / Suggested Answer
1	When a sketch entity is "fully defined," its color turns black, indicating that all of its geometry is completely locked down by dimensions and relations.
2	The "Convert Entities" tool is used to project existing model geometry, such as edges, faces, and other sketches, into the active sketch.
3	The Trim Entities tool is used to remove excess lines or segments from a sketch.
4	**Dimensions** apply numerical values to sketch entities to control their exact size, location, and angles. Examples include defining a line to be exactly 50mm long or specifying a circle's diameter. **Geometric relations** (or sketch relations) define the geometric relationships between sketch entities without using a numerical value. Examples include making two lines parallel, two circles concentric, or a line tangent to a curve.
5	The Copy Entities tool is used to make a copy of a sketch entity.
6	After creating any model, click on the Save button to display the Save As dialog box. Browse to the desired folder and enter the desired name in the File name edit box, if required. Next, click on the Save button to save the file in the selected folder.
7	A tangent relation ensures a smooth, continuous transition between a line and an arc or between two curves. It is essential for creating clean geometry that avoids sharp edges and produces accurate, manufactural designs.
8	Select the entities one by one. Next click on the Parallel button from the PropertyManager to apply parallel relation. Next, click on the OK button of the PropertyManager to exit it and display entities with parallel relation and symbol.

2.2.4 Job Sheet: (A)

Job	Create 2D drawings of samples	
Materials/Resources	Sample Drawing	1 Pc
	Computer set	1 Set
	CAD software (SolidWorks)	

Procedure	• Trainee will collect required materials/resources. • Open CAD software (SolidWorks) • Set unit • Use Sketch Tools. • Give dimensions. • Make relations. • Trim unnecessary sketches. • Follow trainers/instructors • Save drawings in a folder.

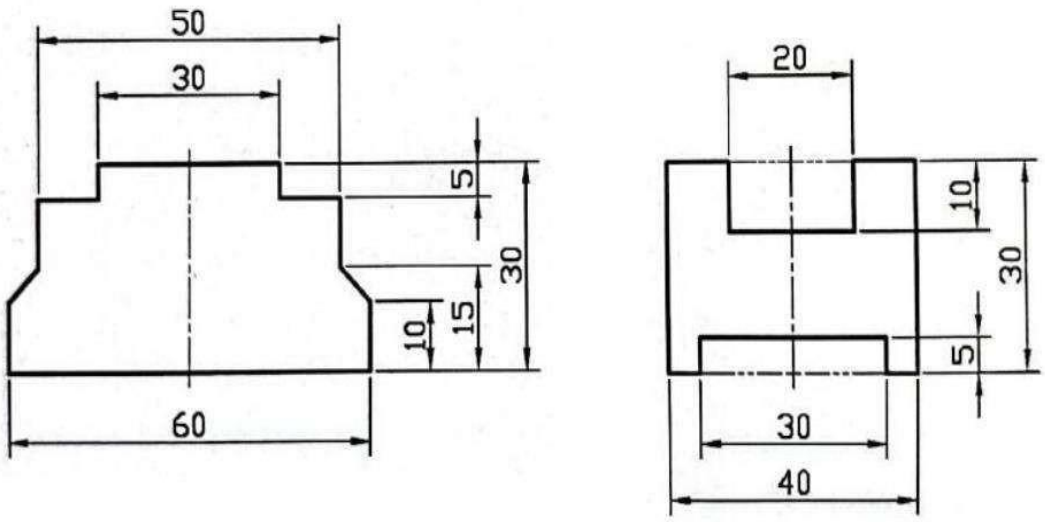
- **Sample (A)**



2.2.4 Job Sheet: (B)

Job	Create 2D drawings of samples	
Materials/Resources	Sample Drawing	1 Pc
	Computer set	1 Set
	CAD software (SolidWorks)	
Procedure	• Trainee will collect required materials/resources. • Open CAD software (SolidWorks) • Set unit • Use Sketch Tools. • Give dimensions. • Make relations. • Trim unnecessary sketches. • Follow trainers/instructors • Save drawings in a folder.	

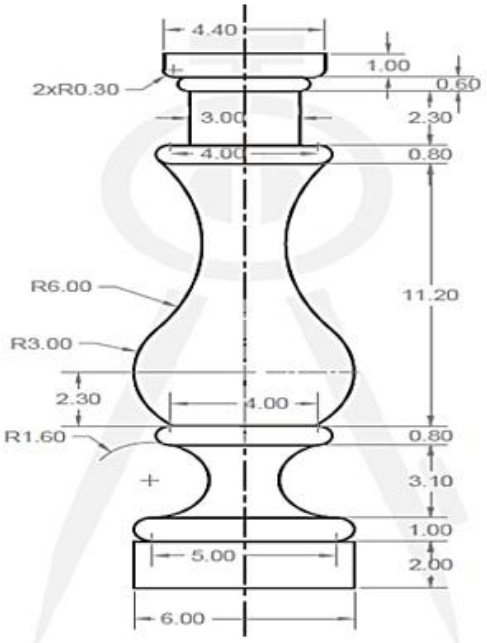
• Sample (B):

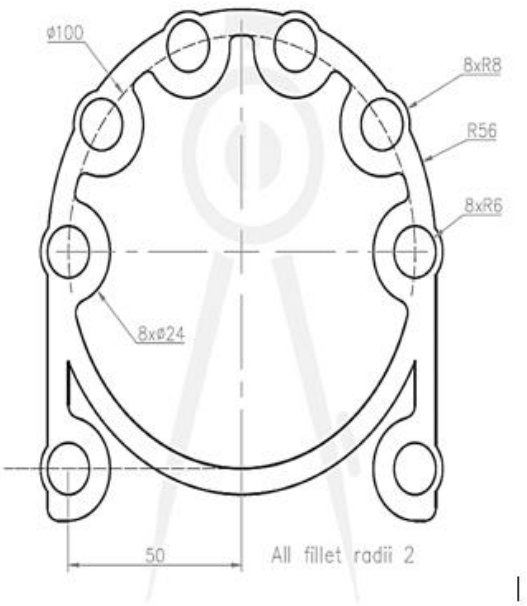


2.2.4 Job Sheet: (C)

Job	Create 2D drawings of samples	
Materials/Resources	Sample Drawing	1 Pc
	Computer set	1 Set
	CAD software (SolidWorks)	
Procedure	• Trainee will collect required materials/resources. • Open CAD software (SolidWorks) • Set unit • Use Sketch Tools. • Give dimensions. • Make relations. • Trim unnecessary sketches. • Follow trainers/instructors • Save drawings in a folder.	

- Sample (C)





2.3 Learning Outcome: Design 3D models using Feature and Surface tools

Content:

1. Introduction to 3D modelling using feature and surface tools.
2. Use of basic feature tools to create solid geometry.
3. Application of advanced features for complex model creation.
4. Use of surface modelling tools for design refinement.
5. Preparation of 3D models for export and manufacturing.

Assessment Criteria:

1. Basic feature tools are used to create solid models.
2. Advanced feature tools are applied to meet design requirements.
3. Surface tools are used to create or modify geometry.
4. Reference geometry is created and applied appropriately.
5. Features are edited and errors are resolved during modelling.
6. Final 3D model is verified for correctness and suitability.
7. 3D model is saved and exported in the required format.

Necessary Equipment:

1. A computer set
2. SolidWorks software (2020 version or above)

Learning activity:

Learning activity	Resources/Special instrumentation/References
Design 3D models using feature and surface tools	• Information sheet: 2.3.1 • Self-check: 2.3.2 • Answer key: 2.3.3 • Job Sheet: 2.3.4

Information Sheet: 2.3.1 (Feature Tools)

Learning Objective: Familiar with SolidWorks Feature Tools

▣ Features Toolbar

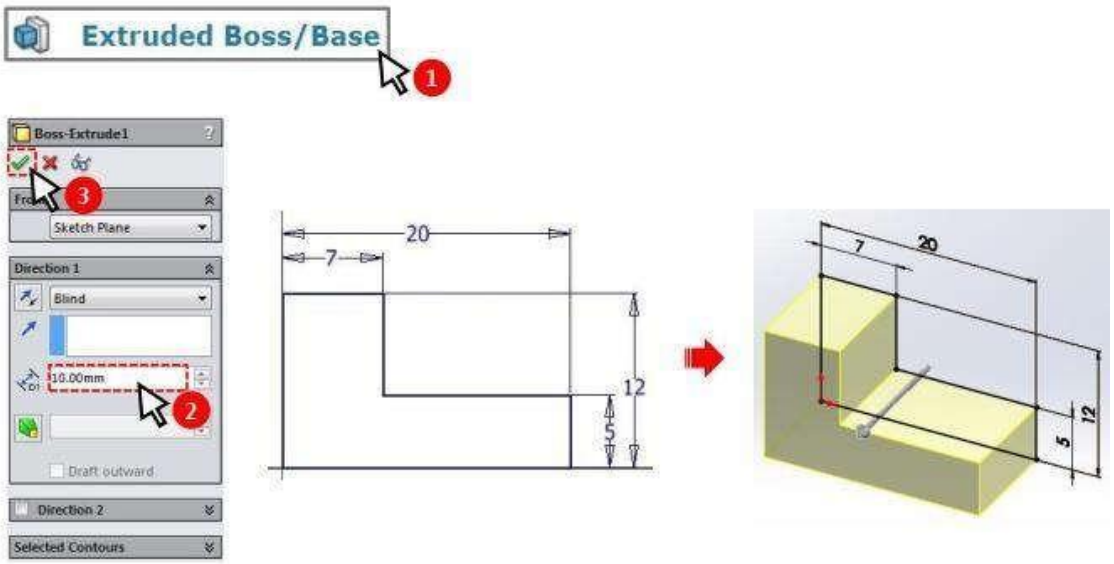
• The Features toolbar provides tools for creating model features. The set of features icons is very extensive so not all of them are included on the default Features toolbar.

• You can customize this toolbar by adding and removing icons to suit your working style and frequent tasks.

Extruded Boss/Base	Wrap	Scale	Intersect
Revolved Boss/Base	Live Section Plane	Shell	Combine
Swept Boss/Base	Model Break View	Draft	Join
Lofted Boss/Base	Instant3D	Move Face	Delete/Keep Body
Boundary Boss/Base	Suppress	Simple Hole	Heal Edges
Thicken	Unsuppress	Hole Wizard	Imported Geometry
Extruded Cut	Unsuppress with Dependents	Advanced Hole	Insert Part
Revolved Cut	Linear Pattern	Thread	Move/Copy Bodies
Swept Cut	Circular Pattern	Hole Series	Recognize Features
Lofted Cut	Mirror Feature	Dome	FeatureWorks Options
Boundary Cut	Curve Driven Pattern	Freeform	Grid System
Thickened Cut	Sketch Driven Pattern	Deform	Convert to Mesh Body
Cut with Surface	Table Driven Pattern	Indent	3D Texture
Fillet	Fill Pattern	Flex	Segmented Imported Mesh Body
Chamfer	Variable Pattern		
Rib	Split		

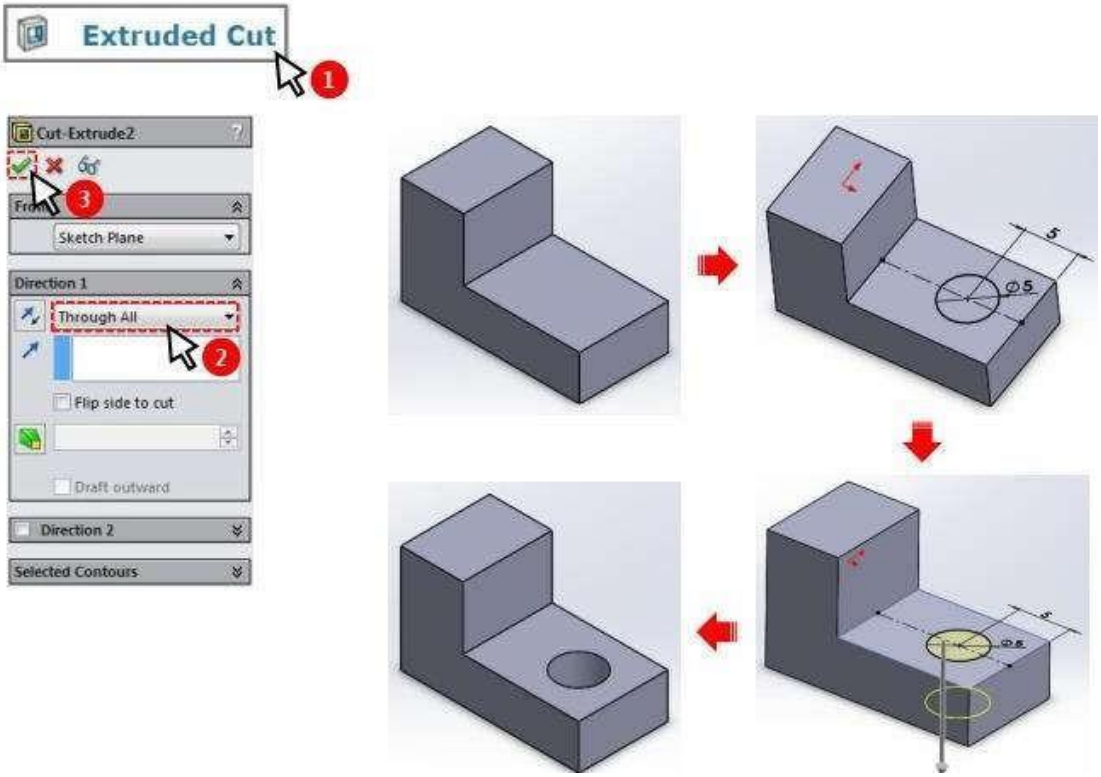
1.

▣ Extruded Boss/Base



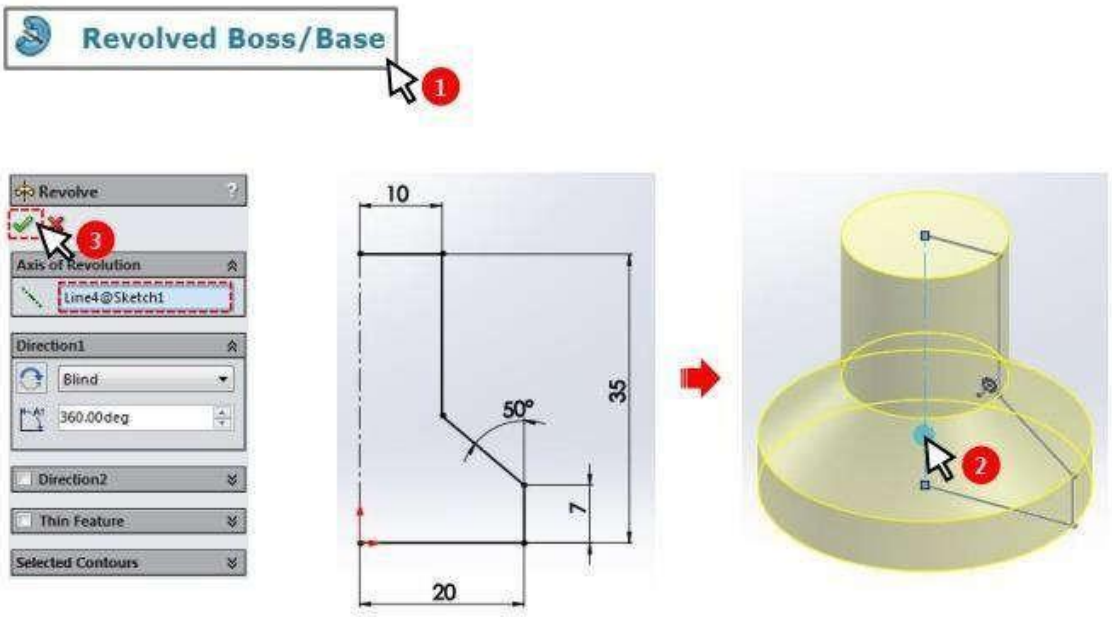
1. On the ribbon, click **Extruded Boss/Base.**
2. The profile to be extrude is automatically selected. (when there is only one profile)
3. Determine the distance and direction and complete the creation of the solid feature.

2.



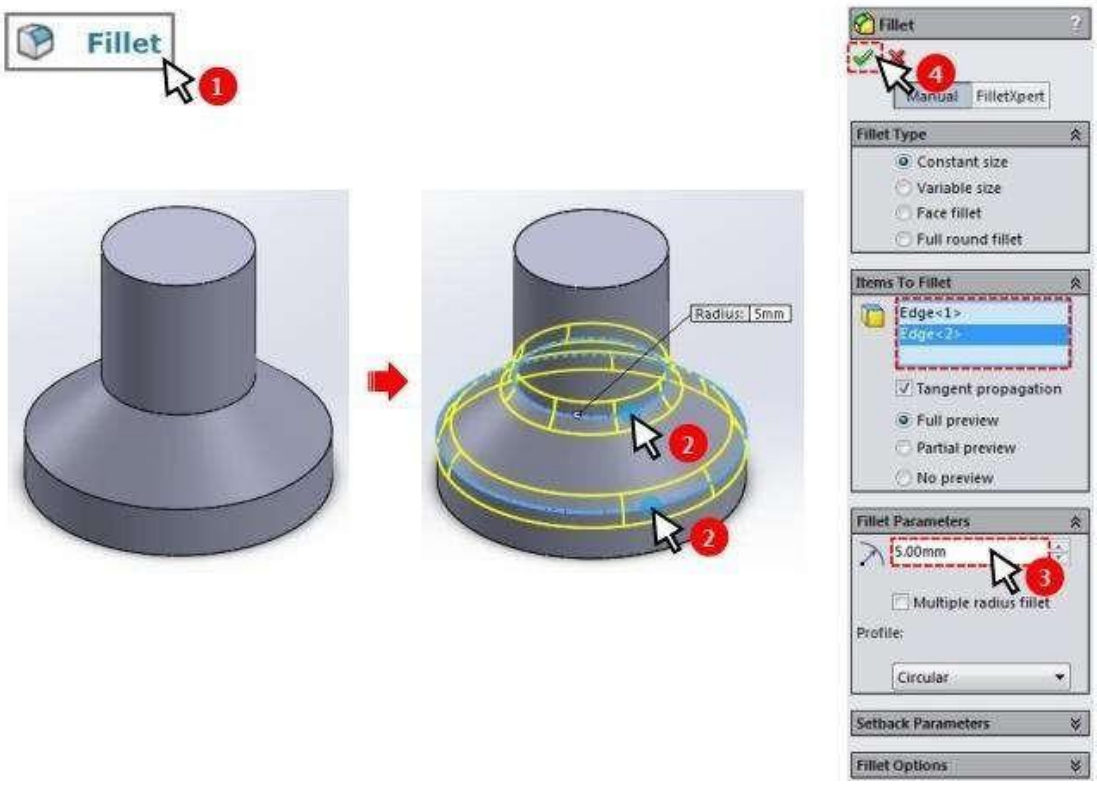
3.

Revolved Boss/Base



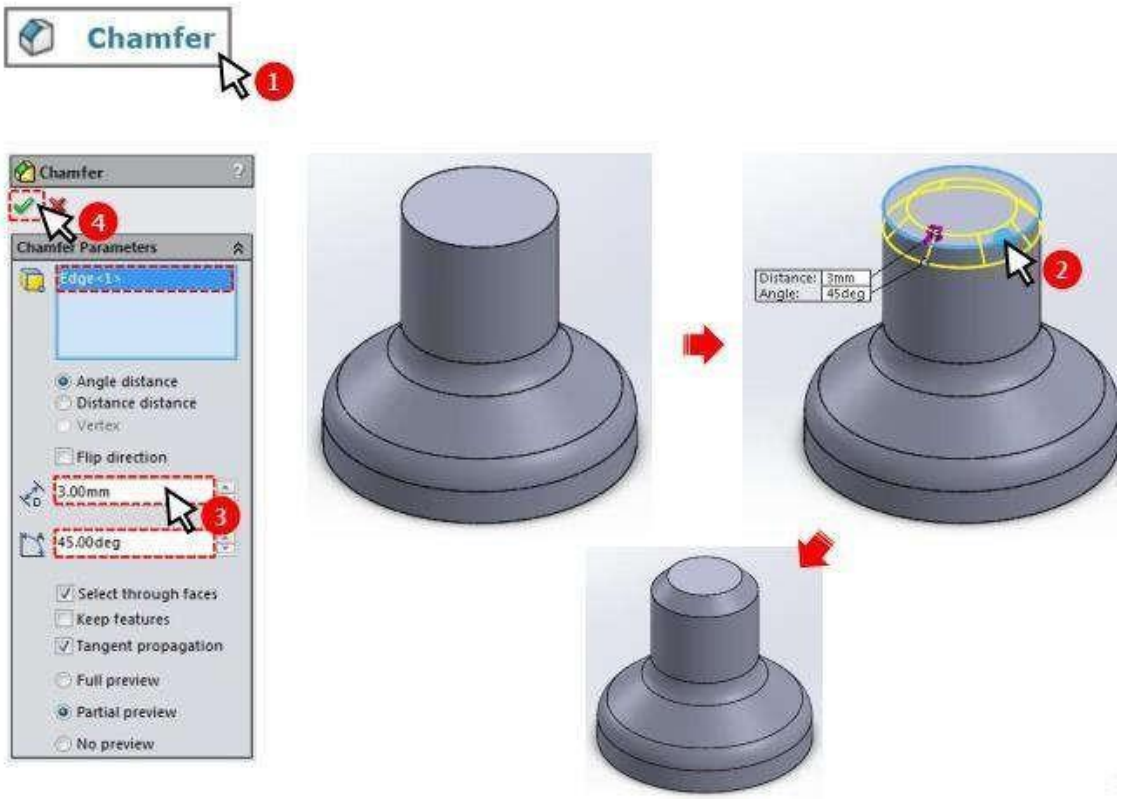
4.

▣ Fillet

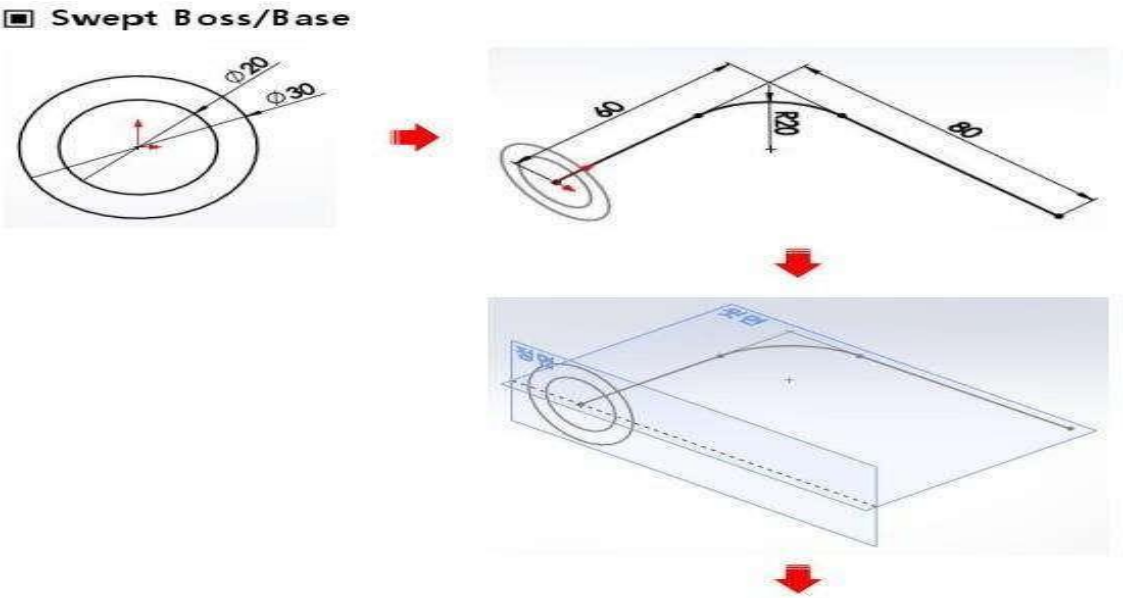


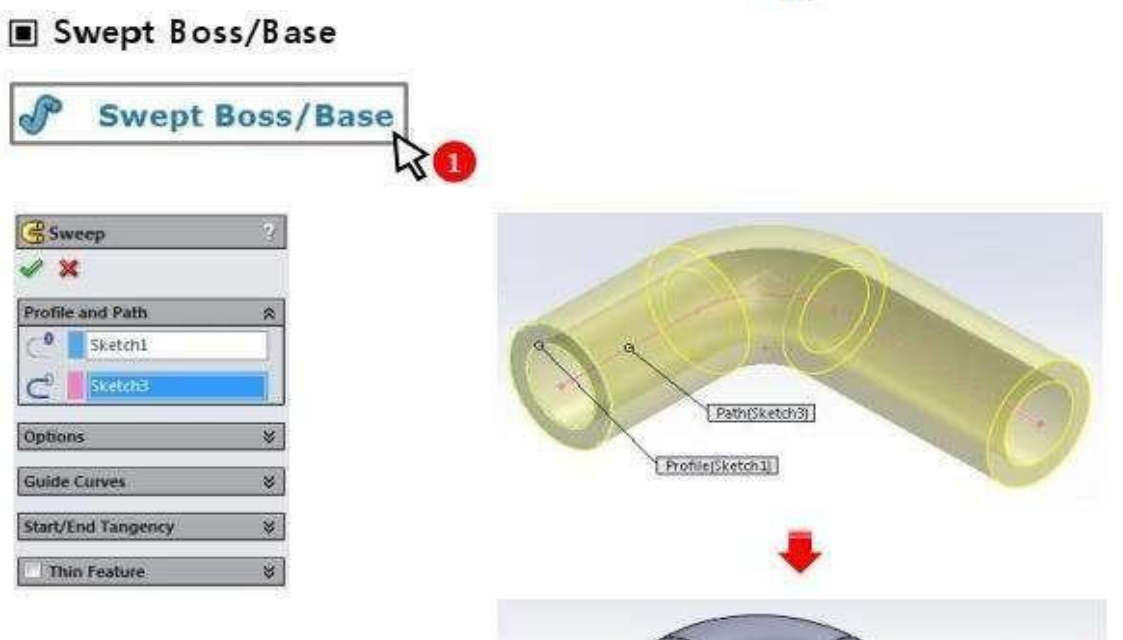
5.

▣ Chamfer



6.

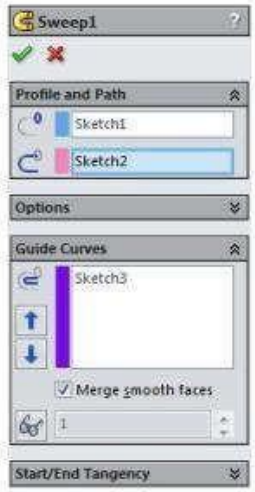

Swept Boss/Base
Ø20
Ø30
50
R20
80

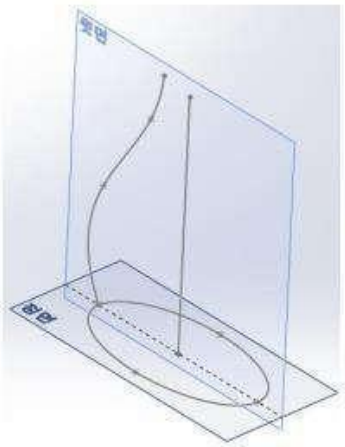

Swept Boss/Base
Swept Boss/Base
1
Sweep
Profile and Path
Sketch1
Sketch3
Options
Guide Curves
Start/End Tangency
Thin Feature
Path(Sketch3)
Profile(Sketch1)

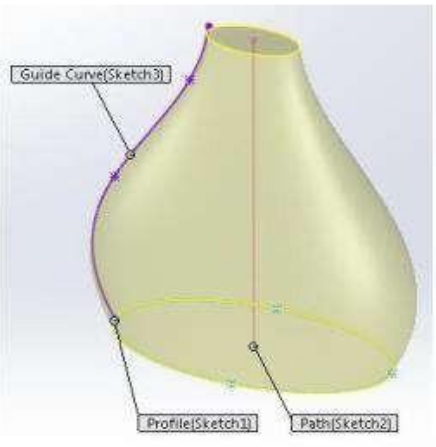
7.

▣ Lofted Boss/Base







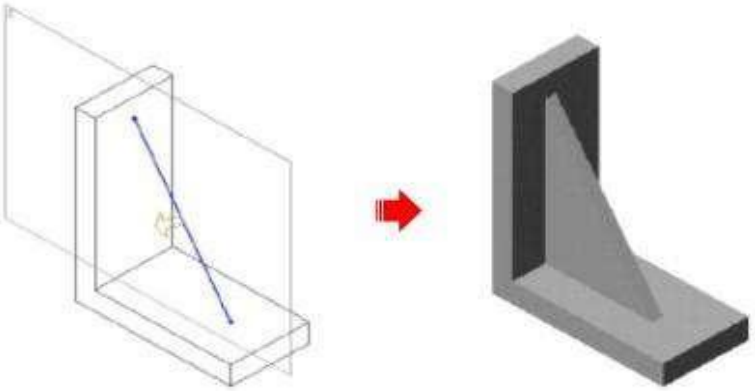


8.

▣ Creating Ribs

To create Ribs:

1. Sketch the contour to use as the rib feature on a plane that intersects the part, or is parallel or at an angle to an existing plane.



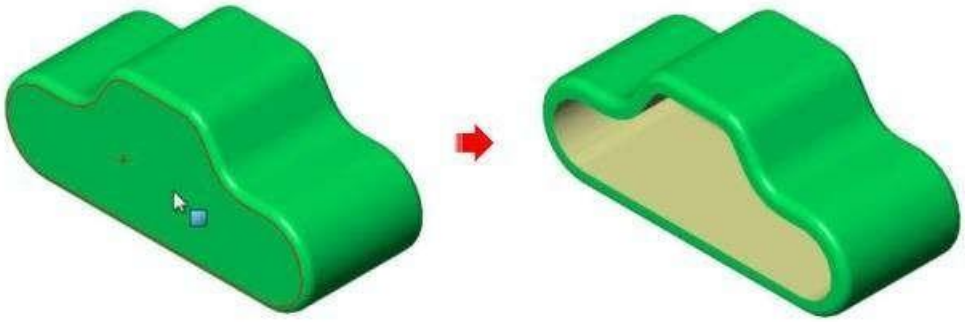
2. Click **Rib** (Features toolbar) or **Insert** > **Features** > **Rib**.
3. Set the PropertyManager options.
4. Click **OK** ✔.

9.

▣ Shell with Uniform Thickness

To create a shell feature of uniform thickness:

1. Click **Shell** (Features toolbar) or **Insert** > **Features** > **Shell.**
2. In the PropertyManager, under **Parameters:**
   - Set **Thickness** to set the thickness of the faces you keep.
   - Select one or more faces in the graphics area for **Faces to remove**.
   - Select **Shell outward** to increase the outside dimensions of the part.
   - Select **Show preview** to display a preview of the shell feature.
3. Click ✔.



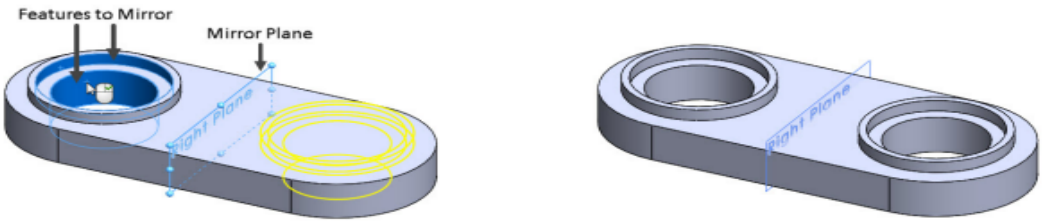
10.

Mirror **Mirror**

This tool is useful while designing a symmetric part by saving your time. By using this tool, you can copy individual features of the entire body. For creating a mirror feature in Part environment or 3D geometry, you need to have place or face to use it as the mirroring element. You can use default planes, model faces or sometime you need to create a datum/reference plane.

Mirror a Feature/Features of a Model

- Click on the **Mirror** tool from the **Features CommandManager** to activate it and display the **Mirror PropertyManager**, as shown.
- Select the Plane as the mirror plane, as shown.
- Select the features to mirror and display the preview of mirror features, as shown
- Click on the **OK** button of the PropertyManager to display the part will Mirror Feature, as shown.

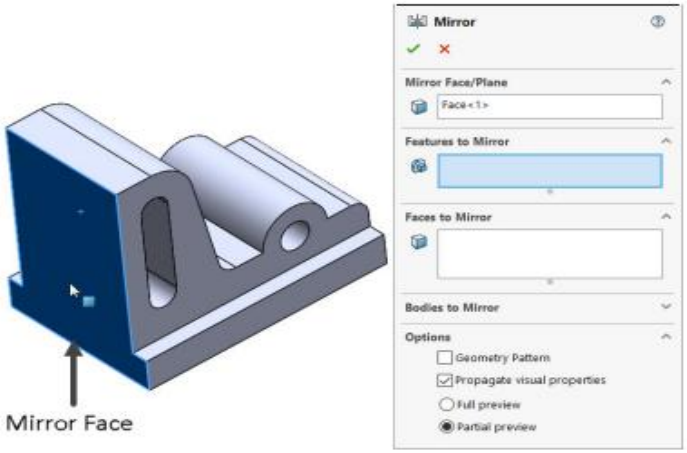


Now, if you make changes to original feature, the mirror feature will be updated automatically, as shown.

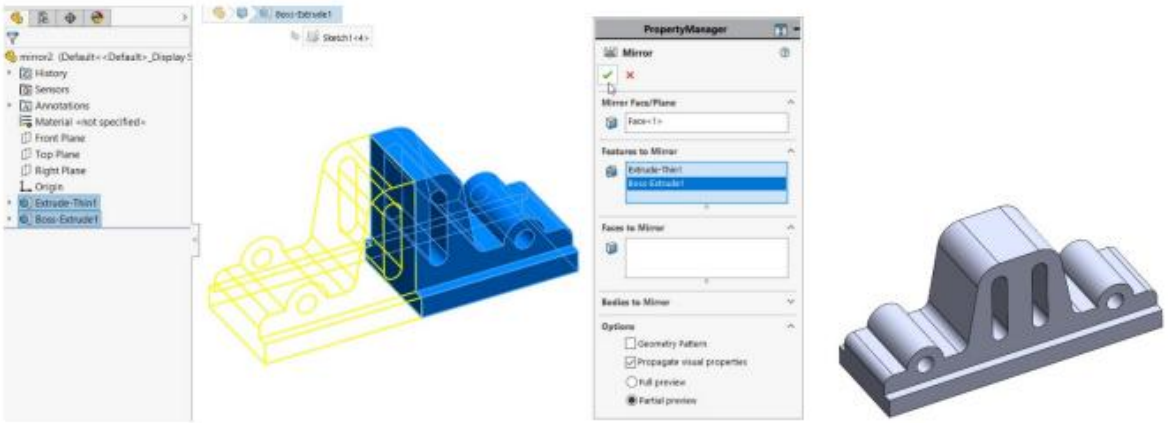
Mirror the Whole Model

If the part you are creating is completely symmetric, you can save more time by creating half of it and mirroring the entire geometry rather than individual features.

- Activate the **Mirror** tool.
- Click on the face of the model to select it as the mirror face, as shown.



- Click on all features from the **FeatureManager design tree** on the left side of the screen to display preview of mirror feature, as shown.
- Click on the **OK** button of the PropertyManager to display the Mirror Feature, as shown.



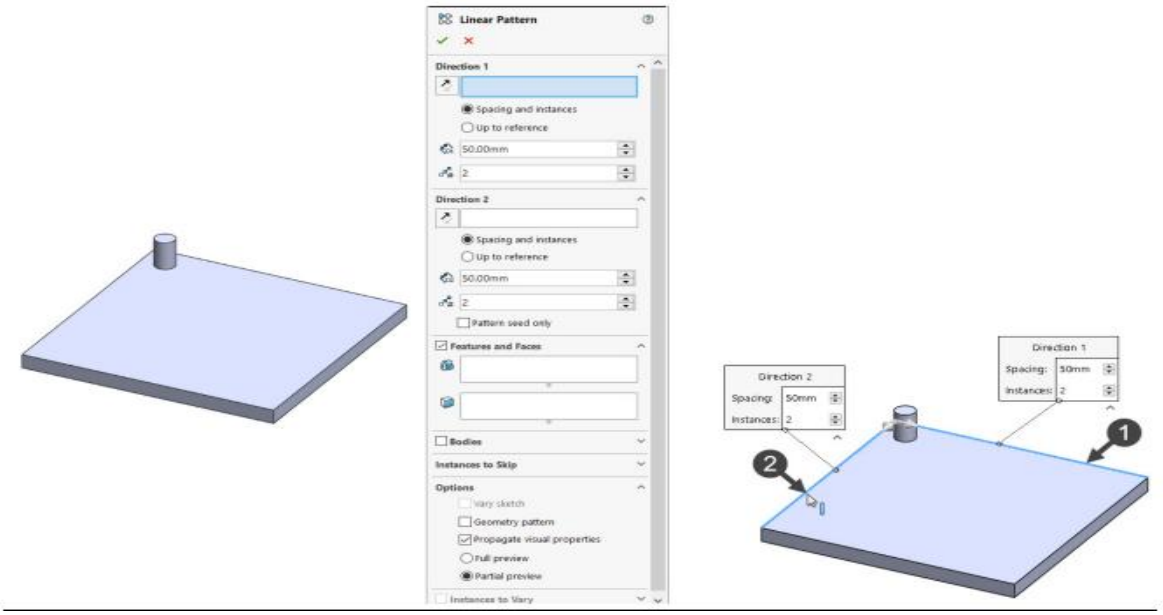
11.



Linear Pattern

The **Linear Pattern** tool is used to create multiple instances of selected feature or features with uniform space along a single or two linear paths.

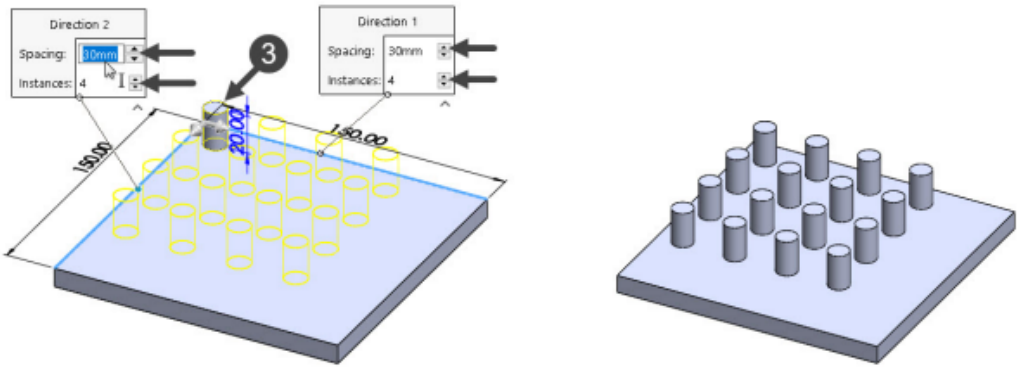
- Click on the **Linear Pattern** tool from the **Features CommandManager** to activate it and display the **Linear Pattern PropertyManager**, as shown.
- Click on the edges of the model to define both as Direction1 and Direction2 to pattern in both directions, as shown



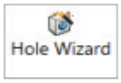
- Click on the feature to be patterned and display preview of pattern feature, as shown.
- Now enter or adjust the required spacing value and number of instances in their respective spinners available in the **Direction1** and **Direction2** callout attached with selected edges, as shown.

  Alternatively, you can enter these values in their respective edit boxes, available in the **Linear Pattern PropertyManager,** as shown above.

- Click on the **OK** button of the PropertyManager to display the Pattern Feature, as shown.



12.

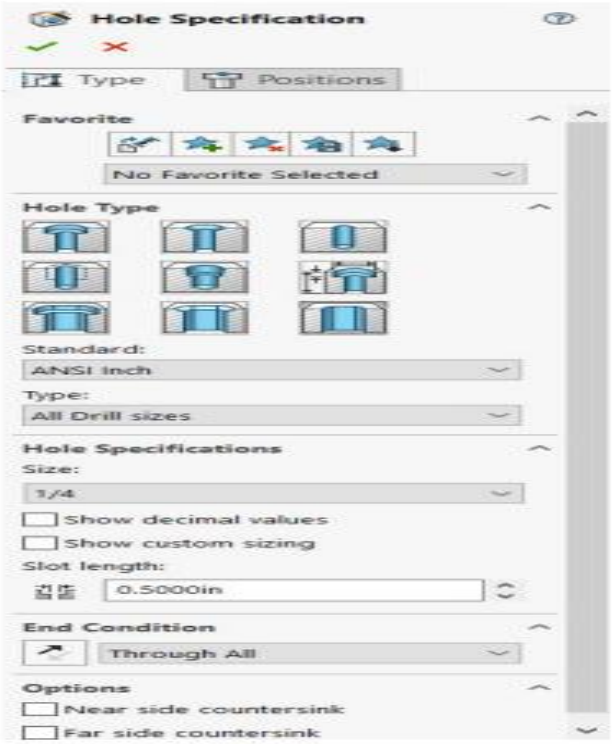


Hole Wizard

This tool is used to add standard holes like counterbore, countersink, tapped, drilled, and pipe tap holes. In SolidWorks there are various types of standard holes/slots grouped in the library. These options can be used for creating different types of holes. You can also create customized hole using this tool.

- Activate the **Hole Wizard** tool from the **Feature CommandManager** to display **Hole Specification PropertyManager**, as shown.

  Some of the important options available in the **Hole Specification PropertyManager** and procedure of creating various types of hole features are discussed next.



Information Sheet: 2.3.1 (Surface Tools)

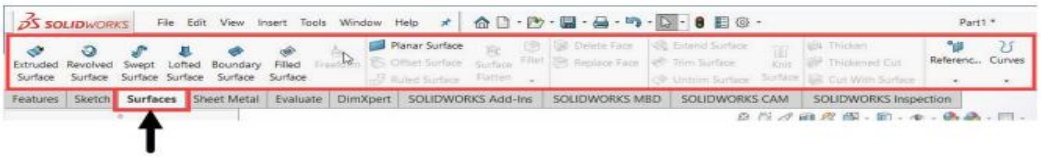
Learning Objective: Familiar with SolidWorks Surface Tools

SolidWorks surface tools can be used to create complex models/geometries using various standard tools like Extruded, Revolved, Swept, Lofted and so on. Surface models are used in automobile, aerospace, and plastic industries. The use of these tools is same as discussed earlier, the only difference is for creating a surface model, you need to select the required Surface button from the **Surfaces CommandManager**. In this chapter, you learn the basics of surfacing tools that are commonly used.

The main topics covered in this chapter are:

- Use of some standard tools for creating surface model
- Extruded Surface
- Revolved Surface
- Swept Surface
- Lofted Surface
- Boundary Surface
- Filled Surface
- Freeform
- Planar Surface
- Offset Surface
- Ruled Surface
- Surface Flatten
- Fillet
- Surface Editing Tools

Before creating a Surface model, you need to display the **Surfaces** tab with all the tool used in creating a Surface model. By default, the **Surfaces** tab might not be visible in the SolidWorks drawing environment. To start Surface model, you need to display all the tools used in creating it. Note that you can toggle between the visibilities of tabs similarly, as discussed in previous chapter.



The use of some of the tools in **Surface** tab is almost same as discussed in previous chapters.

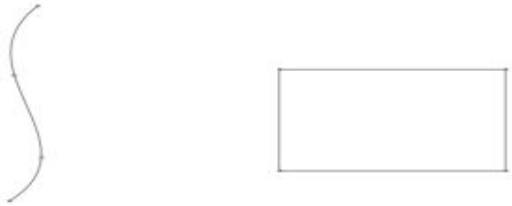
1.



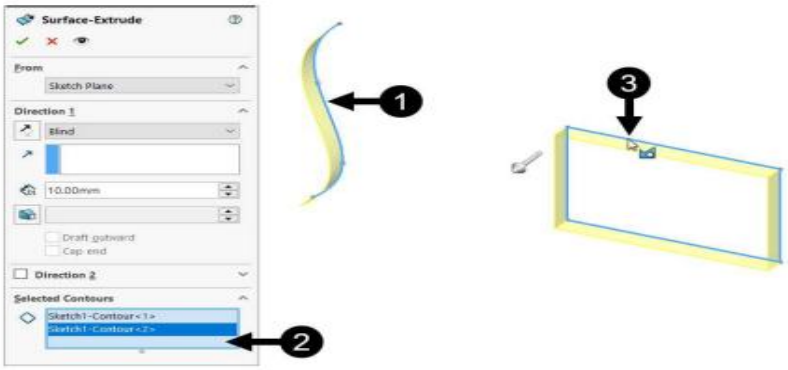
Extruded Surface

This tool is used to create an extruded surface model, by selecting an open or close sketch. To create an extruded surface model using this tool, follow the steps.

- Draw the open or close sketch first, as shown.



- Select the **Extruded Surface** tool from the **Surfaces CommandManager** to display **Surface-Extrude PropertyManager**, as shown.
- Click on the open sketches first to display preview of surface model, as shown.
- Next, click in the **Selected Contours** box of the PropertyManager and then click on the close sketch to display preview of another surface model, as shown



2.

Swept Surface

This tool is used to create a Swept surface model by sweeping an open or close sketch/profile, along a path. The steps to create a Swept Surface model are explained below:

- Draw the sketch for path on **Top** plane, as shown.

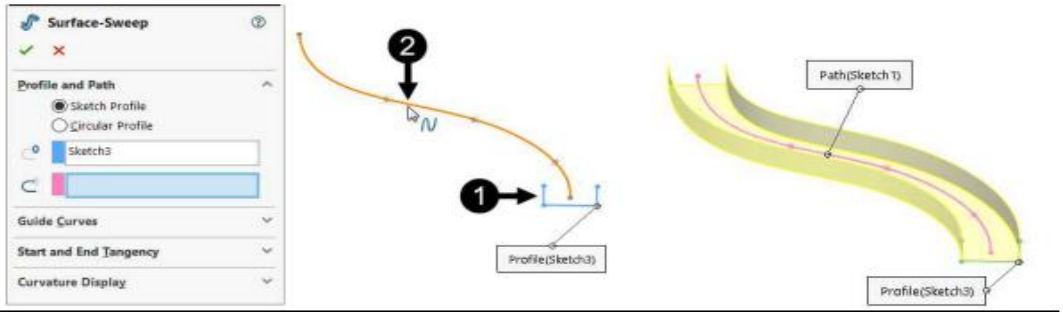


- Now, create a perpendicular plane along one of the endpoints of the above sketch, as shown.
- Next, draw another sketch for profile over it, as shown.





- Select the **Swept Surface** tool from the **Surfaces CommandManager** to display **Surface-Sweep PropertyManager**, as shown.
- Now, select the Profile and Path sketch one by one to display the preview of Swept Surface feature, as shown.



The use of options in this PropertyManager is same as of **Sweep PropertyManager** as discussed in chapter no 8. You can select the required options and enter the required values in it, if required.

- Click on the button of the PropertyManager to apply changes and display the Swept surface model, as shown.

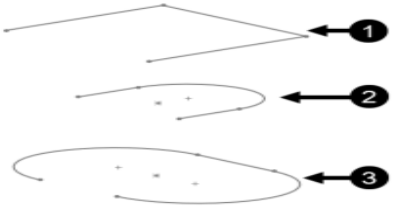


3.

Lofted Surface

This tool is used to create a Loft surface model by connecting two or more sketches created on different planes. The steps to create a lofted surface model are explained below:

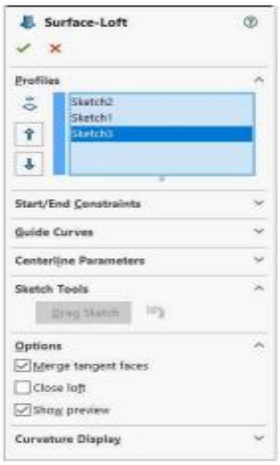
- Draw three sketches on three different planes, as shown.

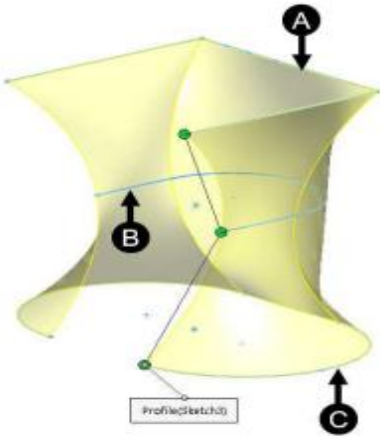


- Select the **Lofted Surface** tool from the **Surfaces CommandManager** to display **Surface-Loft PropertyManager**, as shown.
- Now, select the sketches one by one to display the preview of Swept Surface feature, as shown.

The use of options in this PropertyManager is same as of **Loft PropertyManager** as discussed in chapter no 8. You can select the required options and enter the required values in it, if required.

- Click on the button of the PropertyManager to apply changes and display the Lofted surface model, as shown.





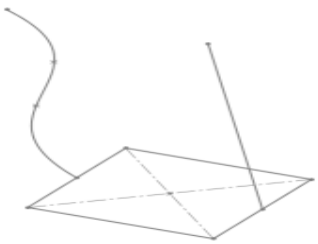


4.

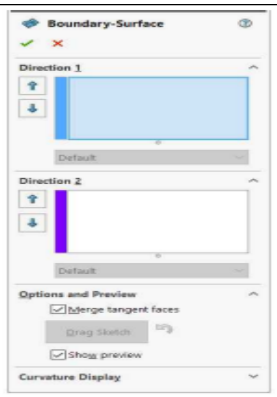
Boundary Surface

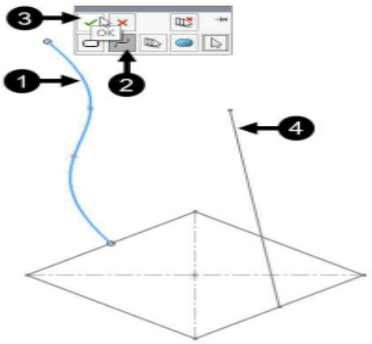
This tool is used to create surfaces that can be tangent or curvature continuous in both directions. This tools also works as Lofted surface tool but this tool delivers a higher quality results as compared to other. The steps to use this tool are explained below:

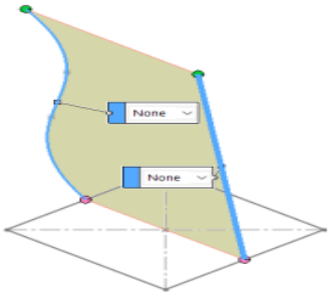
- Draw the sketches on two different planes, as shown.



- Select the **Boundary Surface** tool from the **Surfaces CommandManager** to display **Boundary-Surface PropertyManager**, as shown.
- Now click on one of the sketch entities of first sketch, as shown.
- Now, click on the **Select Open Loop** button of the **SelectionManager** displayed and then click on the **OK** button to select it, as shown.
- Similarly select another open sketch entity to display the preview of Boundary surface feature along the both selected open sketch entities, as shown.



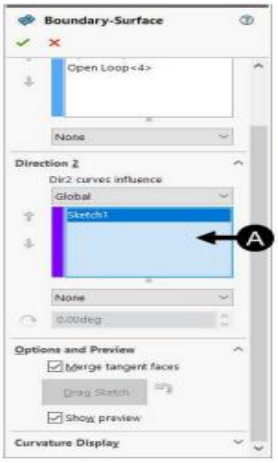


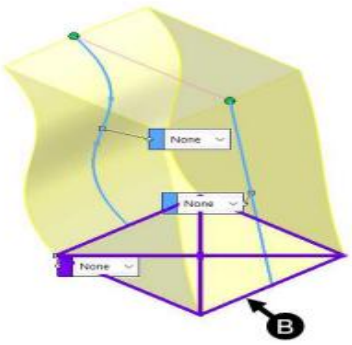


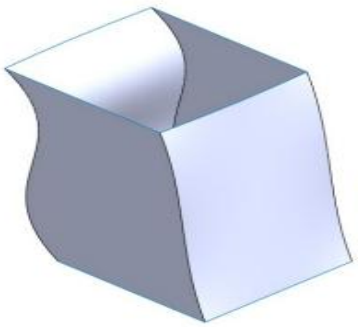
➢ Now click in the box under **Direction 2** rollout and then select another sketch to display preview of Boundary surface feature along it, as shown.

The use of options in this PropertyManager is same as of **Boundary PropertyManager** as discussed in chapter no 8. You can select the required options and enter the required values in it, if required.

➢ Click on the  button of the PropertyManager to apply changes and display the Boundary surface model, as shown.







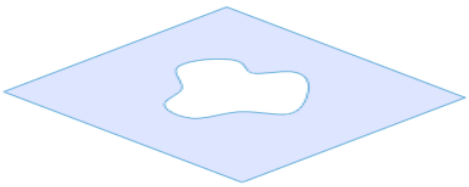
5.



Filled Surface

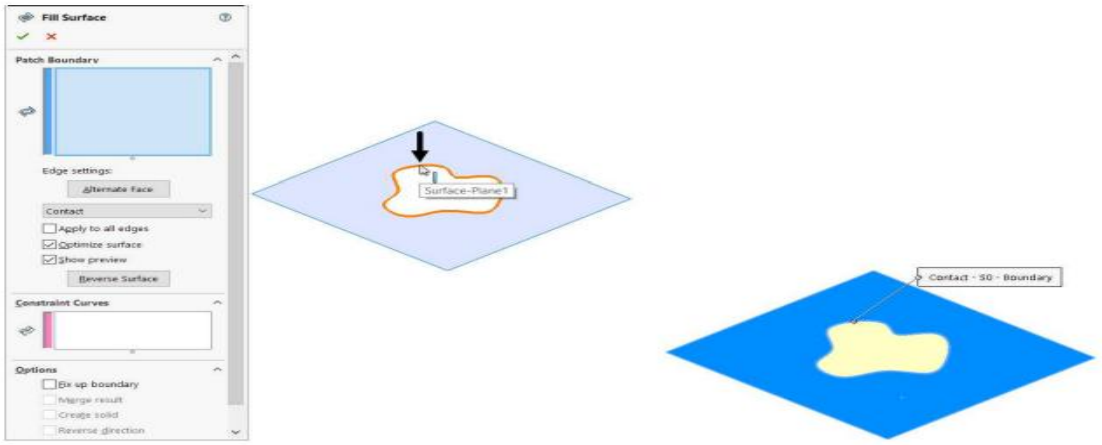
This tool is used to construct a surface patch to fill the gap in a surface model by selecting the edges, sketches or curves. The steps to use this tool are explained below:

➢ Create a surface model, as shown.



➢ Select the **Filled Surface** tool from the **Surfaces CommandManager** to display **Fill Surface PropertyManager**, as shown.

➢ Now select the boundaries of the area to create the surface patch along it and display preview of Filled surface, as shown.



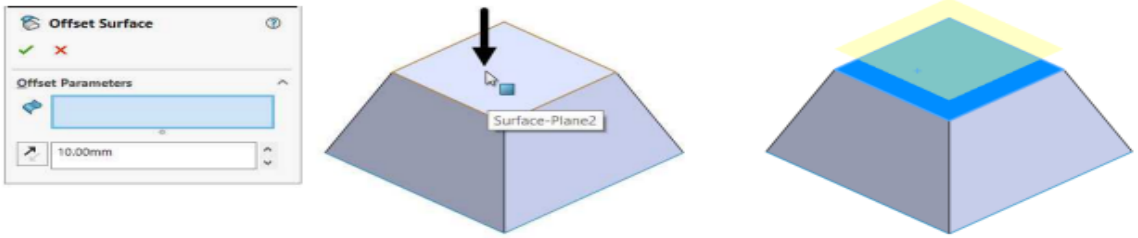
➢ Click on the ✔ button of the PropertyManager to apply changes and display the Filled surface model, as shown.

6.

Offset Surface

This tool is used to create a surface at an offset distance, from the selected surface/face. The steps to use this tool are explained below:

➢ Select the **Offset Surface** tool from the **Surfaces CommandManager** to display **Offset Surface PropertyManager**, as shown.
➢ Now click on the required surface to display preview of Offset surface along it, with the default distance entered in the PropertyManager, as shown.



➢ Enter the required distance in the **Offset Distance** spinner of the PropertyManager, if required.
➢ Now click on the ✔ button of the PropertyManager to apply changes and display the surface model with Offset surface, as shown.

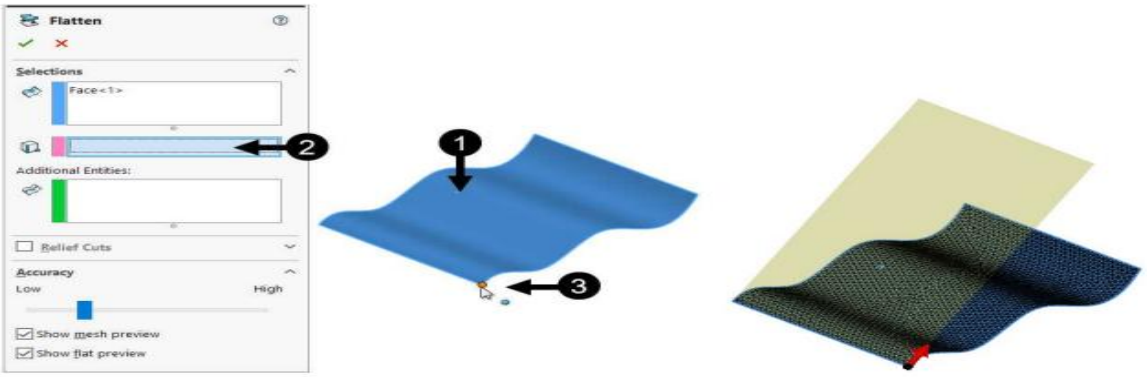
7.

Surface Flatten

This tool is used to flat the surface or set of surfaces to avoid deformation of some surfaces and faces when flattened. The steps to use this tool are explained below:

➢ Select the **Surface Flatten** tool from the **Surfaces CommandManager** to display **Flatten PropertyManager**, as shown.
➢ Click on the surface of the model that you want flatten, as shown.
➢ Now, click in another selection box of the PropertyManager and then select the vertex point or vertex of the surface model to display preview of Flatten surface, as shown.

The **Relief Cuts** check box is selected to apply relief in the surface.



➢ Now click on the button of the PropertyManager to apply changes and display the surface model with Flatten surface, as shown.

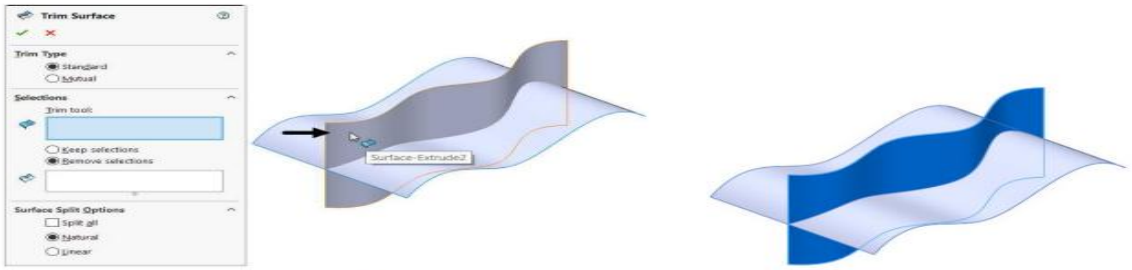
8.

Trim Surface **Trim Surface**

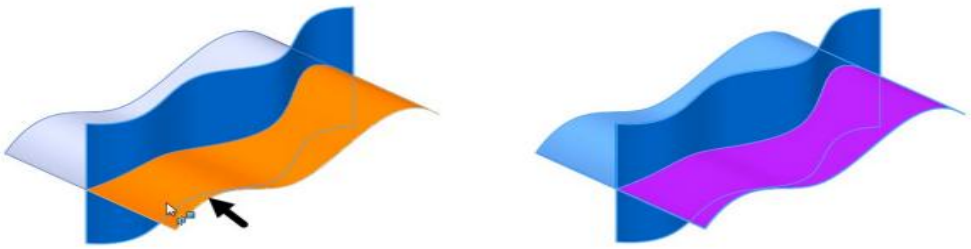
This tool is used to trim the unwanted or intersecting portion of the surfaces by using a surface, sketch, or plane as a trim tool. The steps to use this tool are discussed below:

➢ Select the **Trim Surface** tool from the **Surfaces CommandManager** to display **Trim Surface PropertyManager**, as shown.

➢ Select the surface to select it as the trim tool, as shown.



➢ Now click on the required side of the surface that you want to trim/remove, as shown.

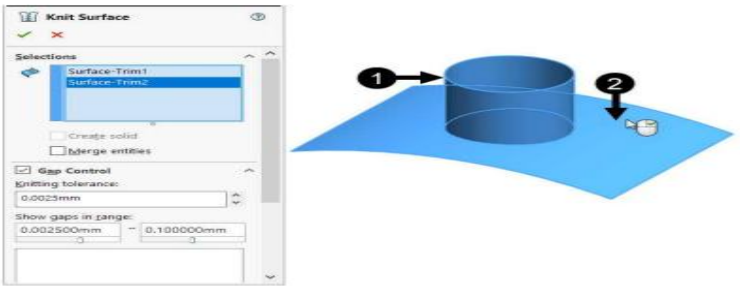


9.

Knit Surface

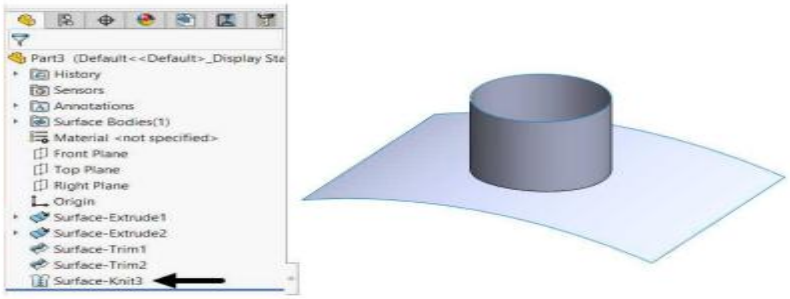
This tool is used to combine two or more faces and surfaces into single surface. The steps to use this tool are discussed below:

- Select the **Knit Surface** tool from the **Surfaces CommandManager** to display **Knit Surface PropertyManager**, as shown.
- Click on both surfaces one by one to select both, as shown.



- Now click on the button of the PropertyManager to apply changes and display the Knit surfaces, as shown.

Also, the **Surface-Knit** feature get displayed in the **FeatureManager Design Tree**, as shown.

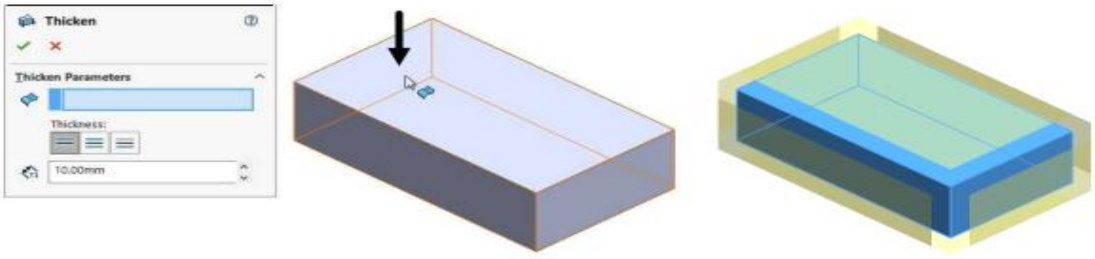


10.

Thicken Surface

This tool is used to increase thickness of surface/surfaces of the surface model. The steps to use this tool are discussed below:

- Select the **Thicken** tool from the **Surfaces CommandManager** to display **Thicken PropertyManager**, as shown.
- Click on surface to display preview of thicken surfaces, as shown.



- You can enter the required thickness in the **Thickness** spinner and select the required Thickness option from the PropertyManager, as shown above.
- Now click on the button of the PropertyManager to apply changes and display the surface model with Thicken surfaces, as shown.

11.

Save

The **Save** button is used to save the model created in various SolidWorks environments, in the desired folder.

- After creating any model, click on the **Save** button to display the **Save As** dialog box, as shown.
- Browse to the desired folder and enter the desired name in the **File name** edit box, if required.
- Next, click on the **Save** button to save the file in the selected folder, as shown.



Note: When you click on the **Save** button, the **Save As** dialog box with the previously selected folder get displayed. If you click on the **Save** button the object gets saved in that folder. You can browse to any other folder/location and save the object there, if required.

File Types in SolidWorks

Creo Parametric offers three main file types:

(*.**prt;*.sldprt**): This type of file has a geometry of individual part. The files created in Part environment. Sheetmetal environments will have this extension.

(*.**asm; *.sldasm**): This type of file is an assembly of one or more parts. In fact, it is a link of one or more parts.

.**drw**: The files created in the drawing environment have this extension.

2.3.2 Self-check

1. What is the difference between an **Extruded Boss/Base** and an **Extruded Cut**?
2. When would you use the **Revolve** feature instead of the **Extrude** feature?
3. What is the function of the **Fillet** tool versus the **Chamfer** tool?
4. How does the **Loft** feature create complex shapes?
5. What is the primary purpose of the **Shell** feature?
6. What does the **Mirror** tool allow you to do with a feature?
7. What is the purpose of a **Lofted Surface** in SolidWorks?
8. Why do we use the **Knit Surface** feature?

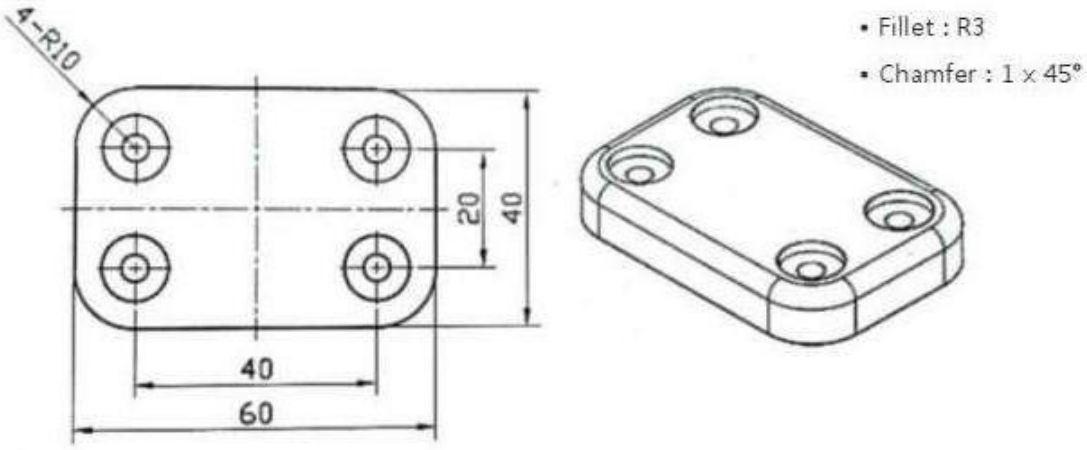
2.3.3 Answer key:

Self-Check Questions	Answer Key / Suggested Answer
1	An **Extruded Boss/Base** feature adds material to a part by extending a 2D sketch profile into a 3D shape. An **Extruded Cut** feature removes material from an existing part by extending a 2D sketch profile to create a cut.
2	We would use a **Revolve feature** for creating cylindrical or conical parts, or any object with rotational symmetry, such as a shaft, wheel, or bottle. This is often more efficient than creating and stacking many extruded circular sketches.
3	The **Fillet** tool creates a rounded edge by adding a tangent arc between two surfaces. Fillets are often used to reduce stress concentration on a part. The **Chamfer** tool creates a flat, angled or beveled edge between two surfaces. Chamfers are simpler to manufacture and are often used to create a lead-in for mating parts.
4	The **Loft feature** creates a complex, transitional shape by blending and transitioning between two or more sketch profiles or model faces.
5	The primary purpose of the **Shell feature** is to hollow out a part, leaving a thin, uniform wall thickness.
6	The **Mirror tool** allows you to create a symmetric copy of one or more features, a face, or even an entire body, across a selected plane or face.
7	A **Lofted Surface** creates a smooth transition between multiple profiles. It is used for complex shapes where simple extrudes or sweeps cannot achieve the desired form.
8	**Knit Surface** combines multiple individual surfaces into one connected surface body. It is essential when preparing surfaces to be thickened or converted into a solid.

2.3.4 Job Sheet: (A)

Job	Create 3D drawings of sample	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	
Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface Tools for 3D drawings. Follow trainers/instructors Save drawings in a folder.	

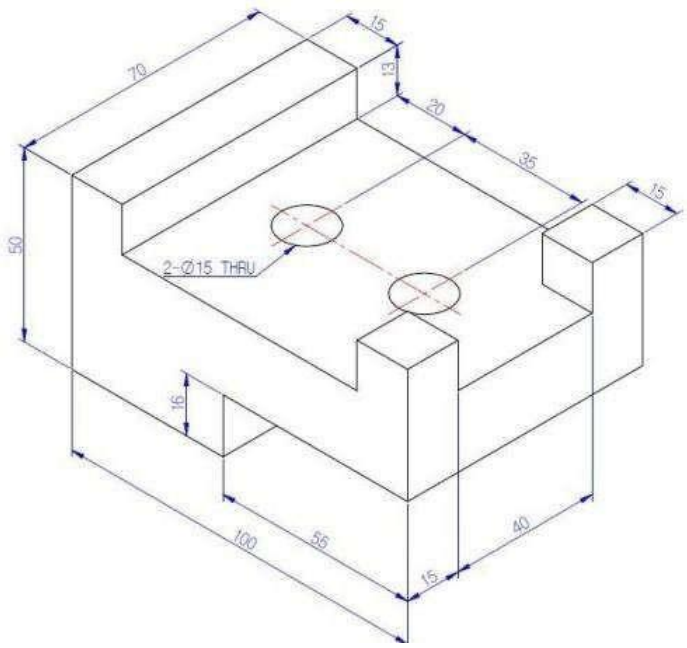
- Sample:

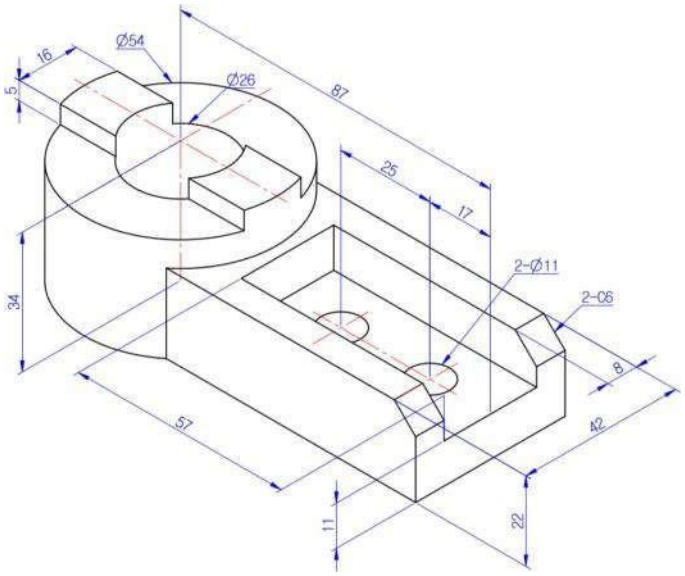


2.3.4 Job Sheet: (B)

Job	Create 3D drawings of sample	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	
Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface Tools for 3D drawings. Follow trainers/instructors Save drawings in a folder.	

- Samples:

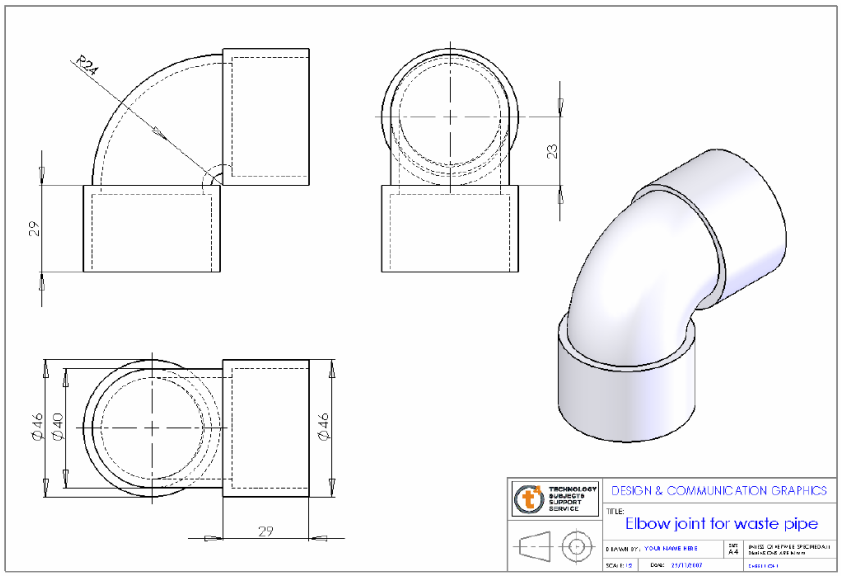




2.3.4 Job Sheet: (C)

Job	Create 3D drawings of sample	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	
Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface Tools for 3D drawings. Follow trainers/instructors Save drawings in a folder.	

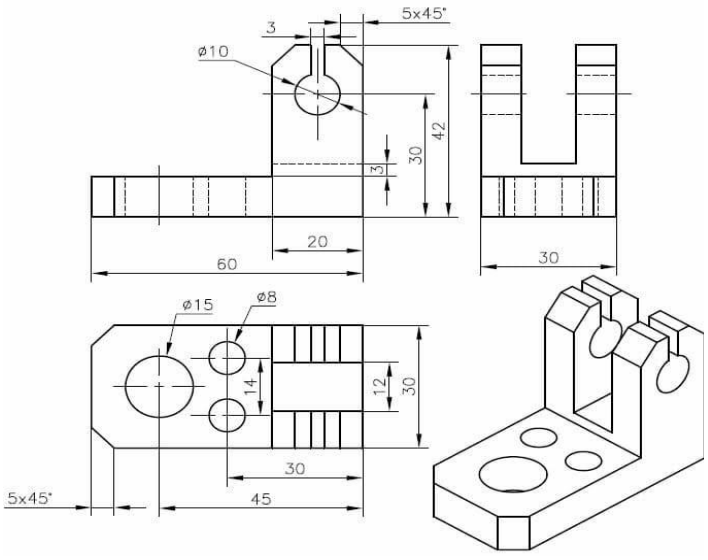
- **Samples:**

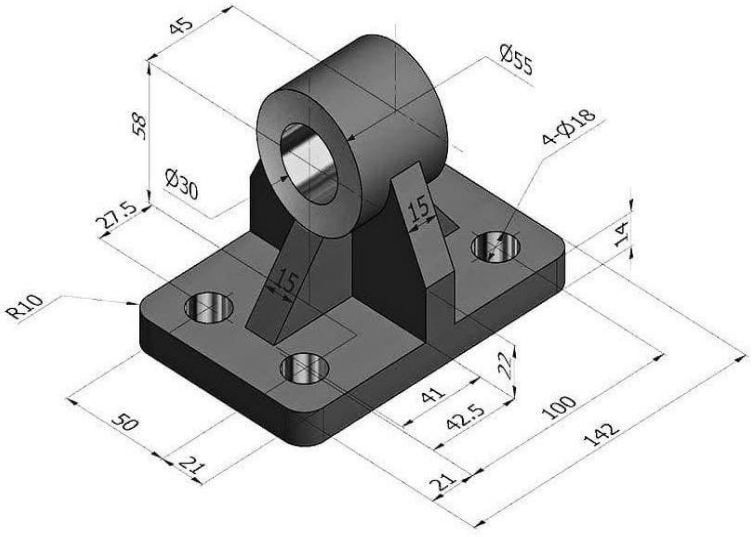


2.3.4 Job Sheet: (D)

Job	Create 3D drawings of sample	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	
Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface Tools for 3D drawings. Follow trainers/instructors Save drawings in a folder.	

- **Samples:**

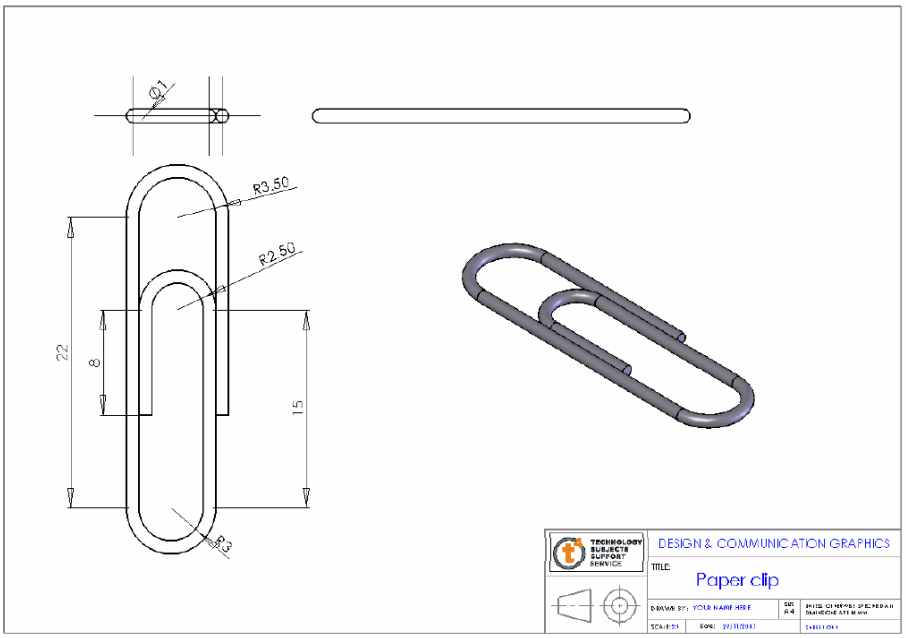




2.3.4 Job Sheet: (E)

Job	Create 3D drawings of sample	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	
Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface Tools for 3D drawings. Follow trainers/instructors Save drawings in a folder.	

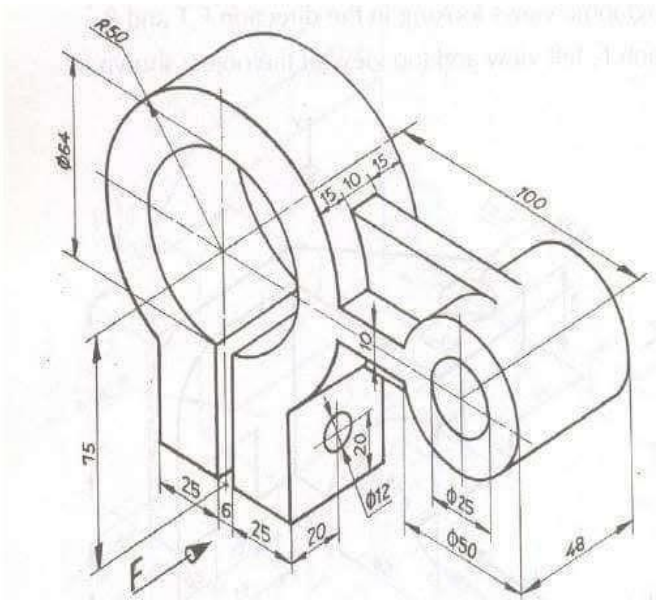
- **Sample:**

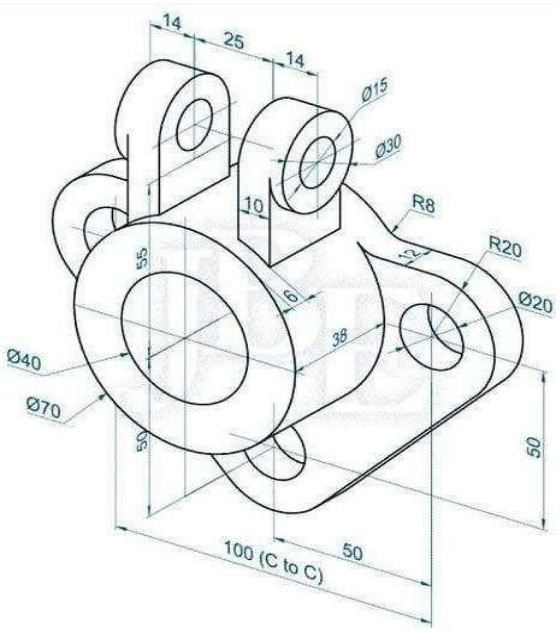


2.3.4 Job Sheet: (F)

Job	Create 3D drawings of sample	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	
Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface Tools for 3D drawings. Follow trainers/instructors Save drawings in a folder.	

- **Samples:**

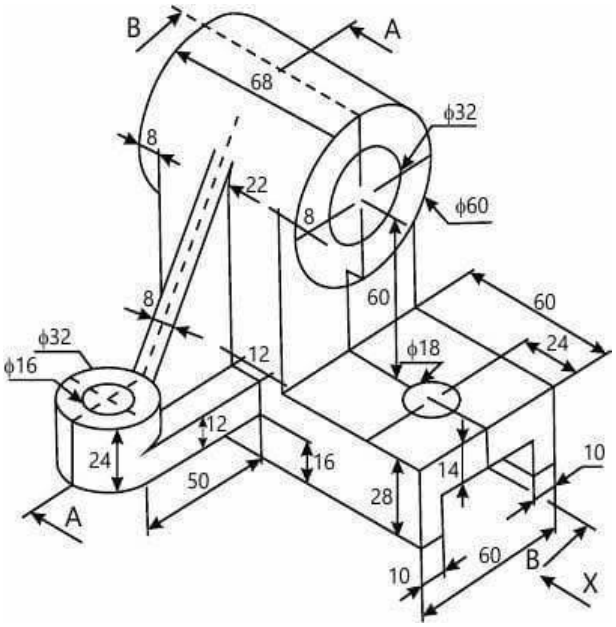


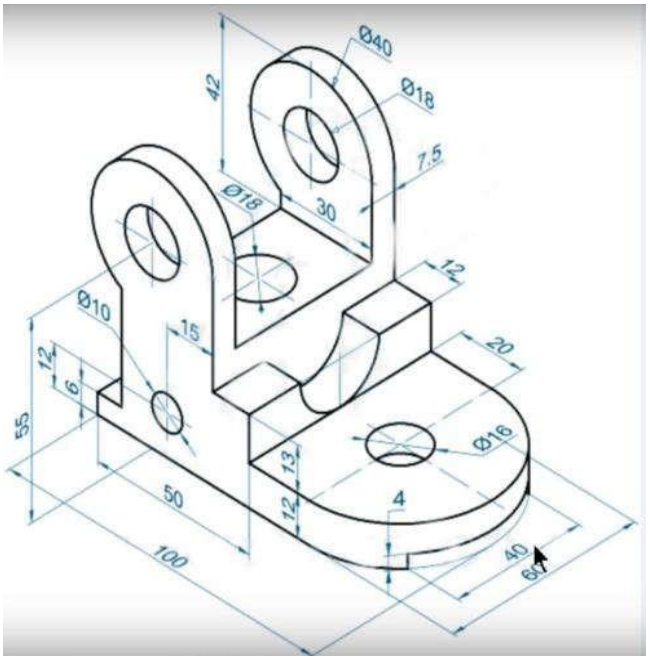


2.3.4 Job Sheet: (G)

Job	Create 3D drawings of sample	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	
Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface Tools for 3D drawings. Follow trainers/instructors Save drawings in a folder.	

- **Samples:**

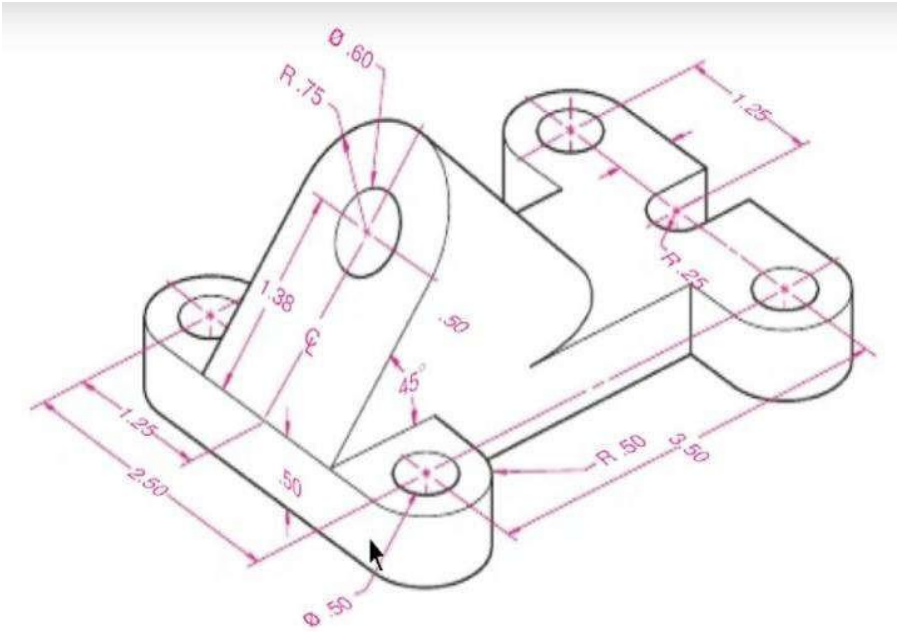




2.3.4 Job Sheet: (H)

Job	Create 3D drawings of sample	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	
Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface Tools for 3D drawings. Follow trainers/instructors Save drawings in a folder.	

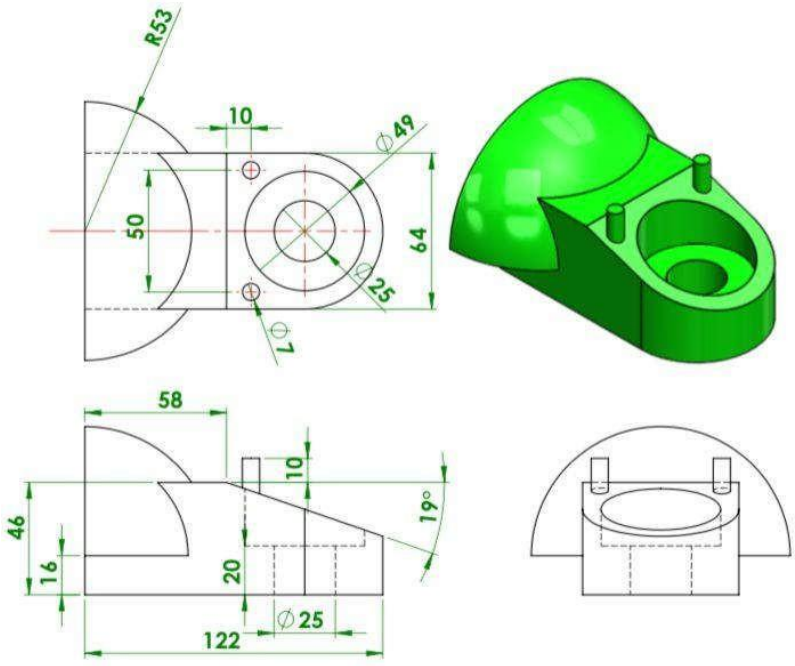
- **Sample:**



2.3.4 Job Sheet: (I)

Job	Create 3D drawings of sample	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	
Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface Tools for 3D drawings. Follow trainers/instructors Save drawings in a folder.	

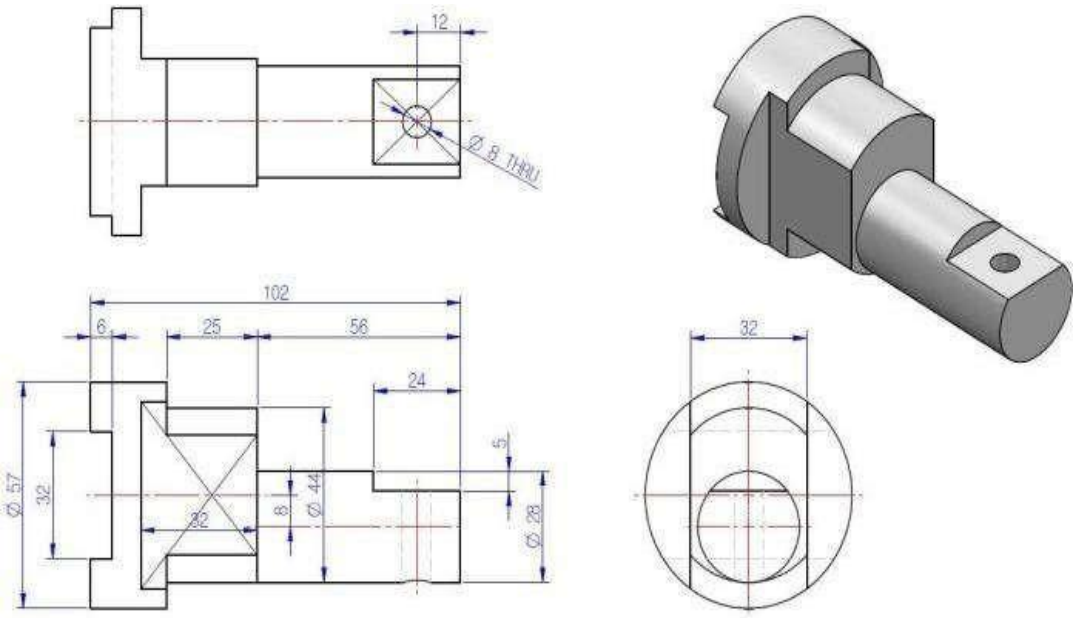
- **Sample:**



2.3.4 Job Sheet: (J)

Job	Create 3D drawings of sample	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	
Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface Tools for 3D drawings. Follow trainers/instructors Save drawings in a folder.	

- **Sample:**



2.4 Learning Outcome: Assemble 3D models

Content:

1. Introduction to assembly environment and workspace.
2. Inserting individual parts into the assembly file.
3. Applying mates to position and constrain components.
4. Checking alignment, motion, and fit of assembled parts.
5. Saving and preparing the assembly for further processing.

Assessment Criteria:

1. Assembly interface and tools are identified and navigated.
2. Required part files are imported correctly into the assembly.
3. Appropriate mates are applied to position components.
4. Component alignment and relationships are verified.
5. Assembly constraints are adjusted to meet design requirements.
6. Interference or motion issues are identified and corrected.
7. Assembly file is saved and exported in the required format.

Necessary Equipment:

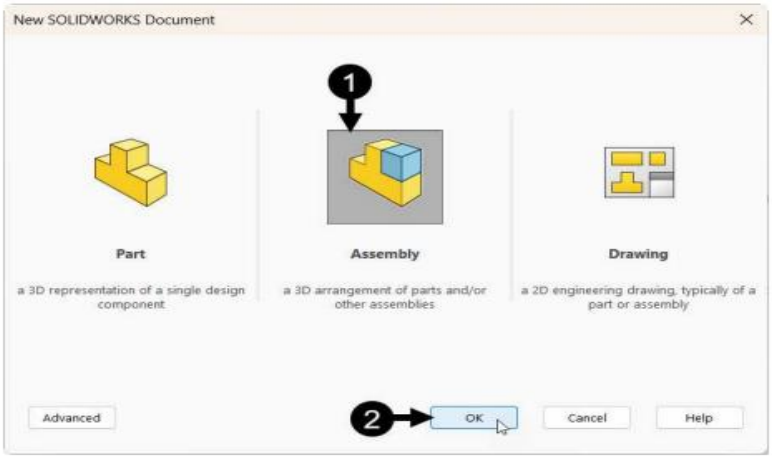
1. A computer set
2. SolidWorks software (2020 version or above)

Learning activity:

Learning activity	Resources/Special instrumentation/References
Assemble 3D models	• Information sheet: 2.4.1 • Self-check: 2.4.2 • Answer key: 2.4.3 • Job Sheet: 2.4.4

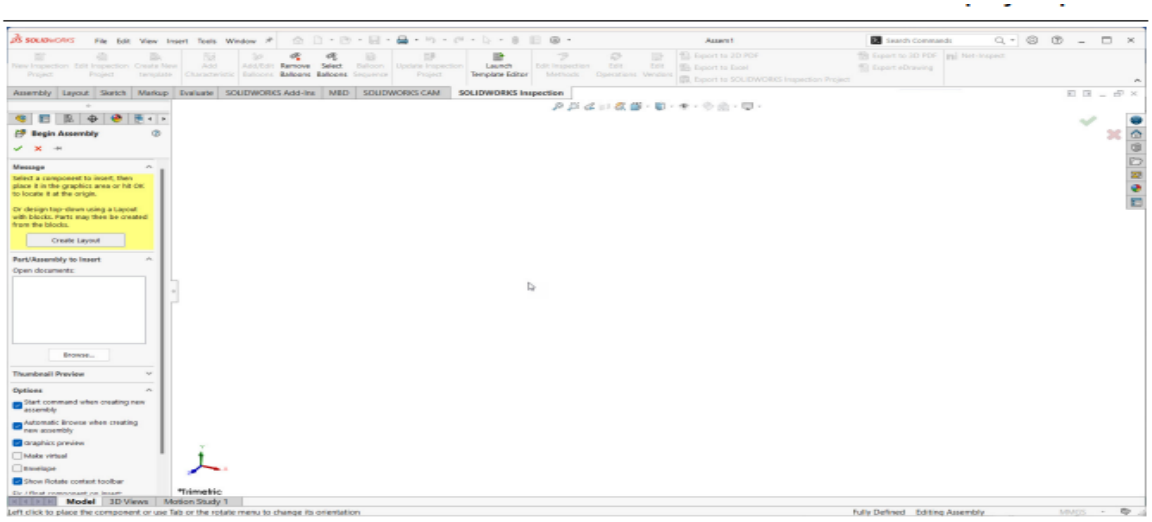
Starting an Assembly

To begin an assembly file, click on the **New** button from the **Quick Access Toolbar** to display **New SOLIDWORKS Document** dialog box. From the dialog box, select the **Assembly** button and click on the **OK** button to entering in the assembly environment, as shown. The Assembly environment with **Assembly** tab opened and **Begin Assembly PropertyManager** get displayed, as shown below.



Assembly Environment

The Assembly environment has tools to combine individual parts in an assembly. There are two ways to create an assembly. The first way is to create individual parts in part environment and then assemble them in the Assembly environment, known as **Bottom-up assembly design**. The second way is to start an assembly file and create individual parts in it, known as **Top-down assembly design**.



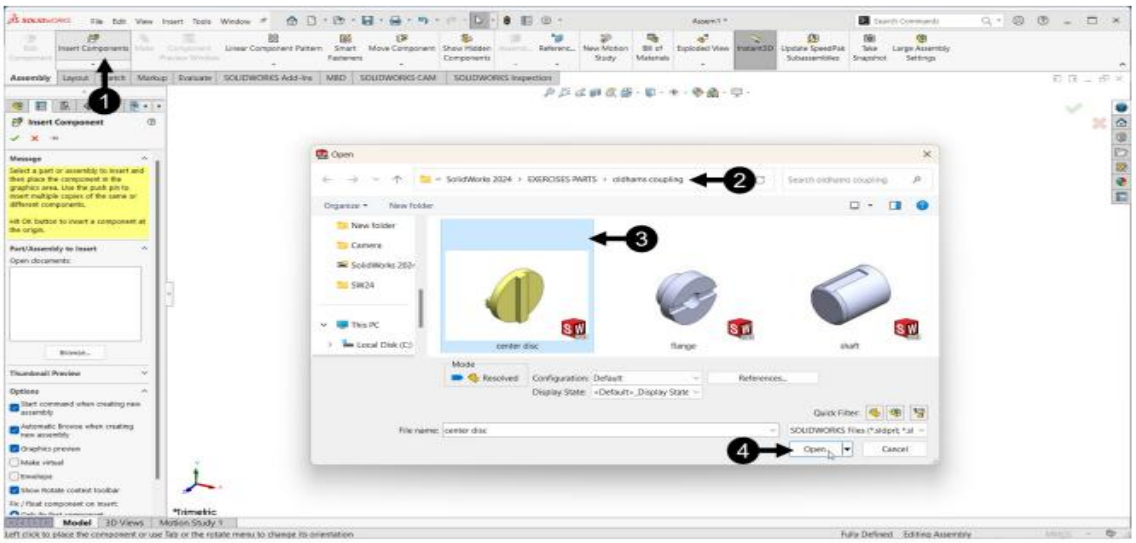
Note: Note that after entering in the assembly environment, by default the **Open** window get display automatically. You can then browse to the required folder and then select and import the required part file in the assembly environment, as shown.

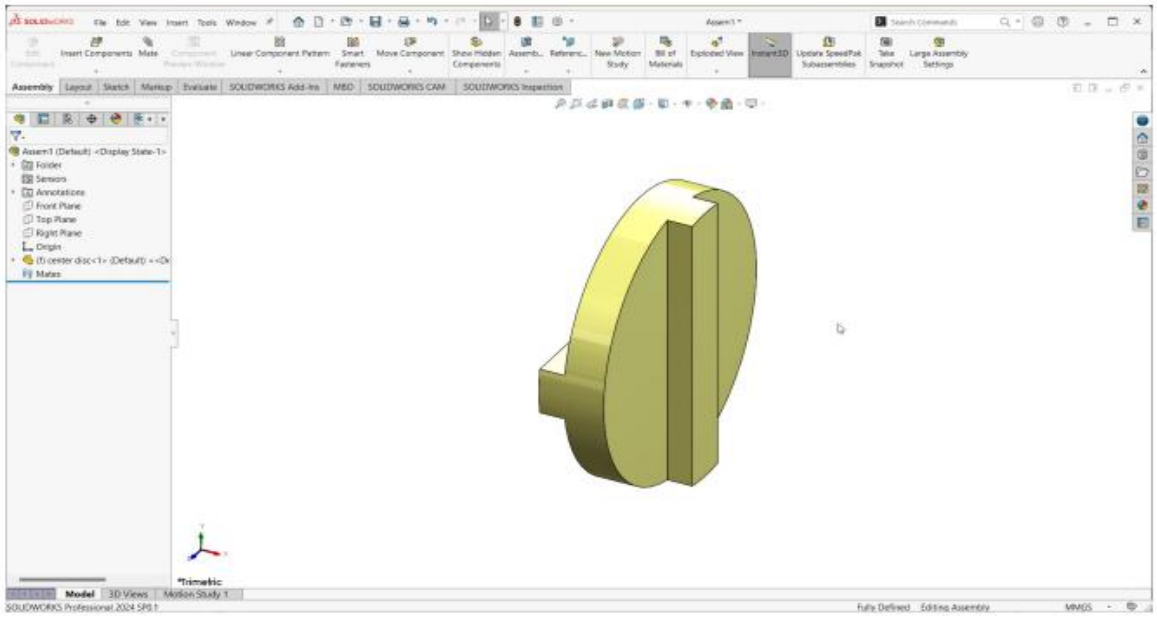
Inserting Components

There are two different methods to insert an existing part into an assembly. The first one is to insert using the **Insert Components** button. The second way is to drag it directly from Windows Explorer. In the second method, there is no need to open the components in **Open** window. You can simply drag-and-drop them into the assembly.

- Click on the **Insert Components** button from the **Assemble CommandManager** to display **Open** window, as shown.
- Browse to the required folder and select the required part file, as shown.
- Next click on the **Open** button to display it in the assembly environment, as shown.

  Alternatively, you can double click on the required file to import it.



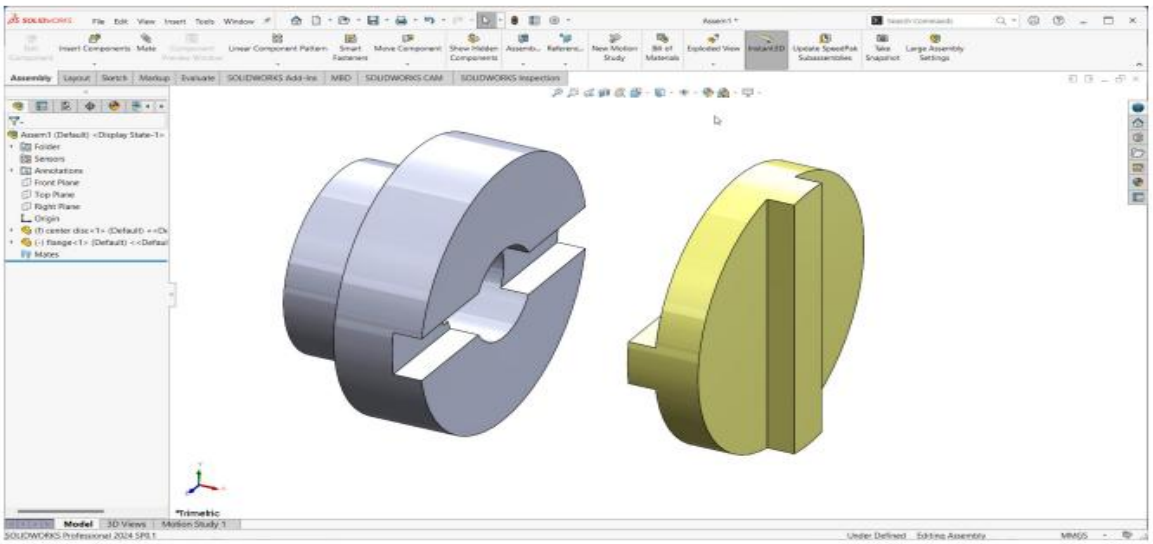


- After importing the first component, it will get fixed at the origin automatically, as shown. And then after inserting other components, you can assemble them with respect to it.

Note: As the initially inserted component get fixed automatically with the origin. So, the component gets fully defined automatically. Thus, its degrees of freedom in the drawing area get removed.

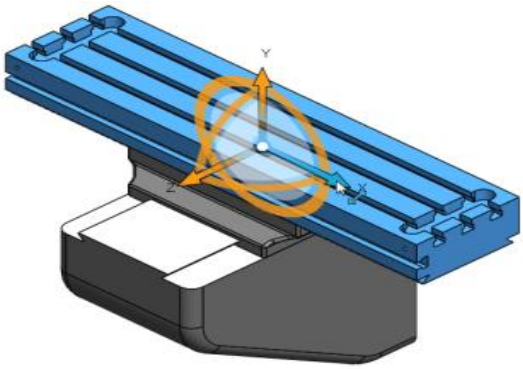
- Similarly, you can insert other components and you need to apply various mates to assemble them with respect to initially inserted or fixed component, as shown.

  These mates/constraints are discussed further in this chapter.



To move components

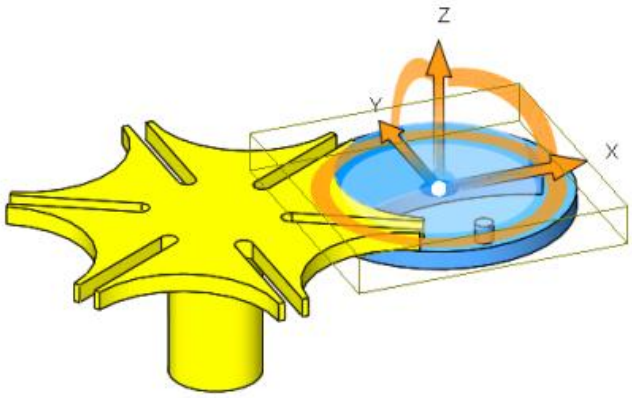
1. Select a component.



2. Drag elements of the triad:
   - Dragging an arm drags the component along the axis of the arm.
   - Dragging a ring drags the component along the plane of the ring.

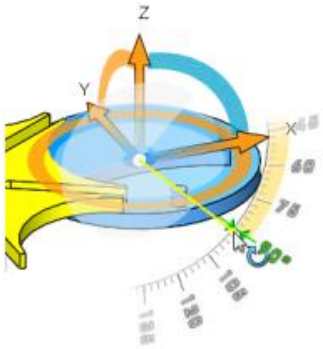
To rotate components

1. Select a component.



2. Select a ring and drag. Also:

- To snap, right-click the selected ring and choose Snap while Dragging. Close to the ring, the snap increment is 90°. The increment decreases as the pointer moves further away from the ring.



- To rotate in preset increments, right-click the selected ring and choose Rotate 90° or Rotate 180°.

- To enter specific increments, right-click the center sphere and select Show Rotate Delta XYZ Box.

Automatic Mate

After activating the **Mate** tool and selecting the entities, the system automatically applies the respective mate/constraint with respect to the selected entities. After selecting two flat surfaces; **Coincident** mate gets applied and after selecting two circles; **Concentric** mate get applied and so on.

➢ After inserting any component, select the **Mate** tool to activate it and display the **Mate PropertyManager**, as shown.

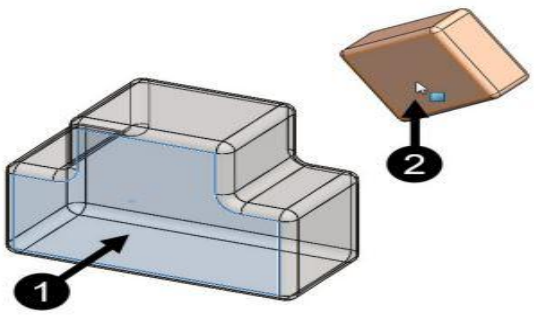
➢ Click on the two flat surface one-by-one to display preview of automatically applied **Coincident** mate between both, as shown.

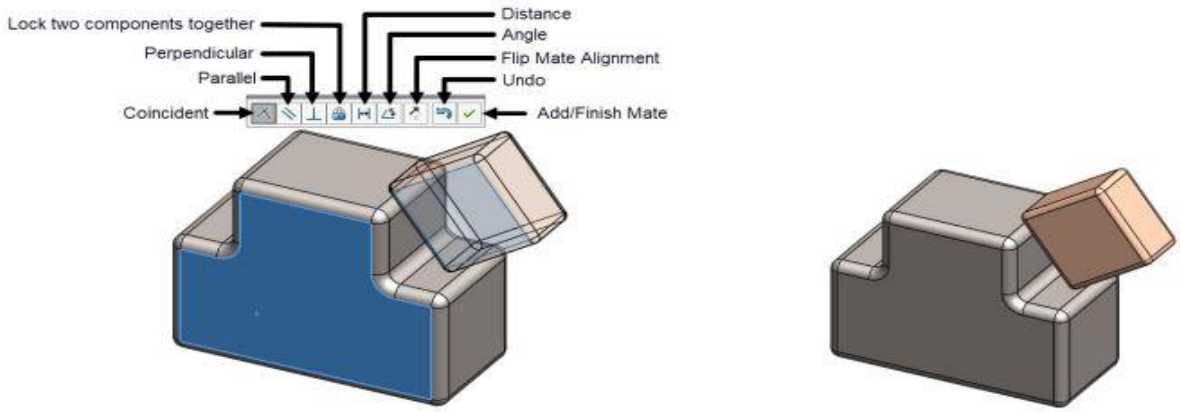
➢ Next to apply any change or apply any other mate, you can click on the buttons available in the **Mate** pop-up toolbar displayed, as shown.

The use of buttons in the **Mate** pop-up toolbar are displayed further in this chapter.

➢ Now click on the **Add/Finish Mate** button to apply mate and display components with applied mate, as shown.

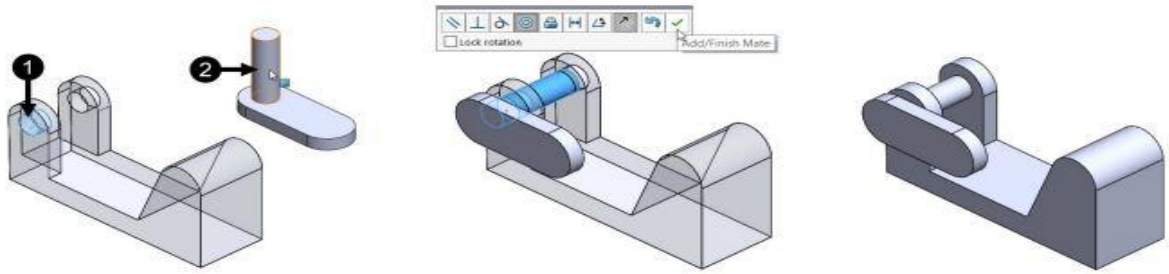






➢ Click on the **OK** button of the **Mate PropertyManager** to exit it.

Similarly, after selecting circular surfaces of two components, **Concentric** constrains will be applied automatically, as shown.



Note that, after activating the **Mate** tool and selecting two surfaces of two components, the **Mate** pop-up toolbar will also get display along with automatically applied respective mate, as shown above. And if required you can also select another mate from it. The use of other buttons in this pop-up toolbar are:

Flip Mate Alignment – This button is selected to flip the direction of applied mate.

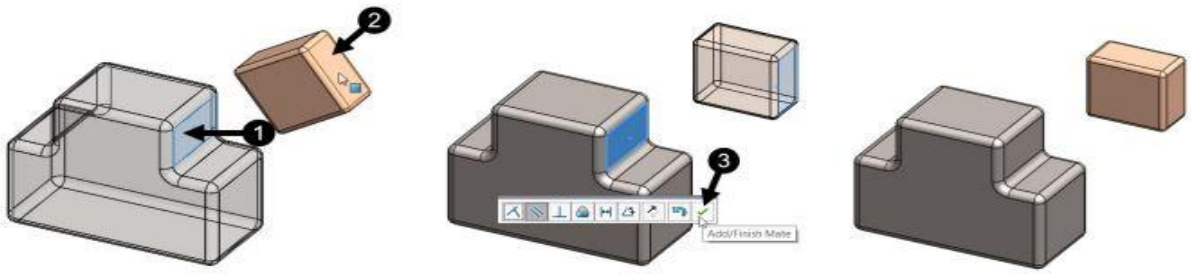
Undo – This button is selected to go back after applying any mate.

Add/Finish Mate - After applying mates, this button is selected to exit pop-up toolbar and display components with applied mates.

Parallel Mate

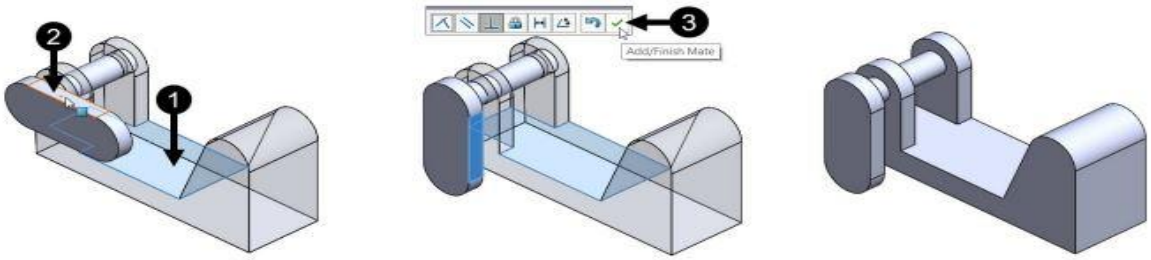
The **Parallel** mate makes an axis, face or edge of one part parallel to that of another part.

➢ Click on the **Parallel** button from the **Mate PropertyManager**, as shown above.
➢ Click on the two flat surfaces one by one to display preview of **Parallel** mate between them, as shown.
➢ Next click on the **Add/Finish Mate** button to apply mate and display components with applied mate, as shown.



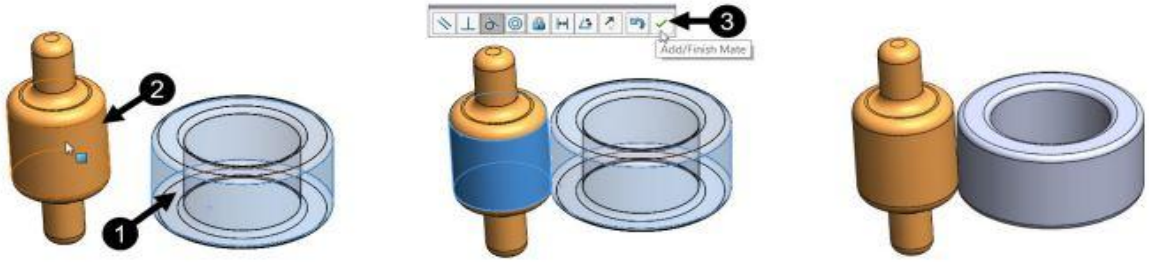
Perpendicular Mate

The **Perpendicular** mate is used to make two selected surfaces or two axes perpendicular to each other.



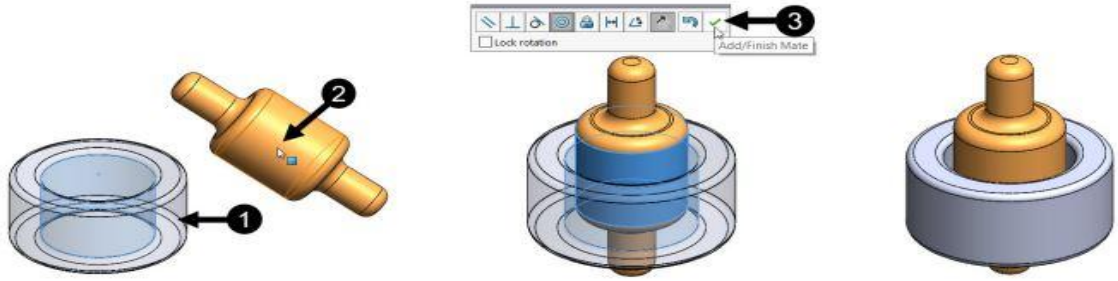
Tangent Mate

The **Tangent** mate is used to make the selected circular face tangent to the other selected face or plane.



Concentric Mate

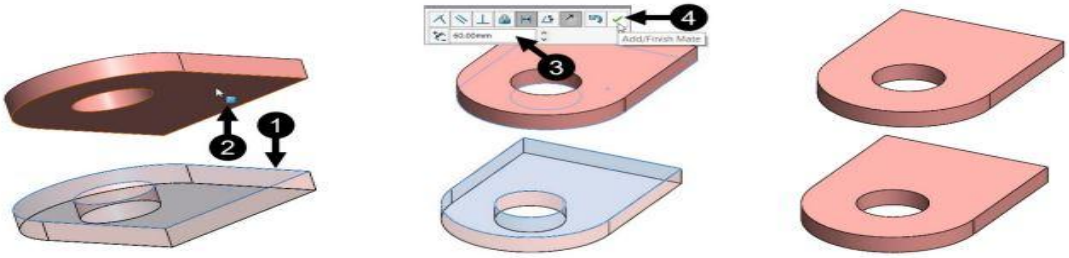
The **Concentric** mate is used to make the axes of two cylindrical faces coincide/concentric with each other.



Distance Mate

The **Distance** mate is selected to place the selected faces at the specified distance between them.

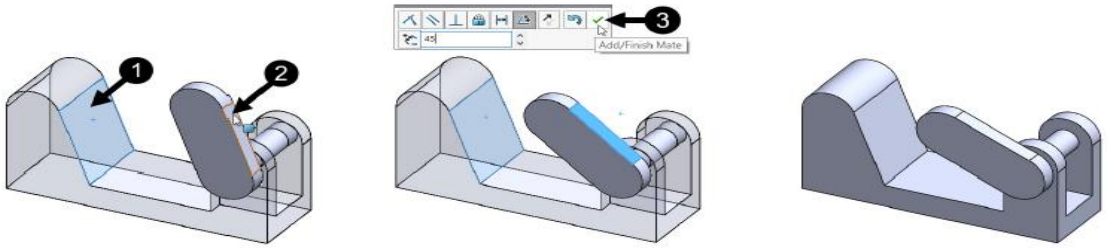
- Click on the **Distance** button from the **Mate PropertyManager**, as shown above.
- Click on two surfaces to place distance between them.
- Next enter or adjust the required distance value in the **Distance** spinner of the **Mate** pop-up toolbar, as shown.
- Next click on the **Add/Finish Mate** button to place the selected items at the defined distance value, as shown.



Angle Mate

The **Angle** button is used to position the selected faces at a specified angle.

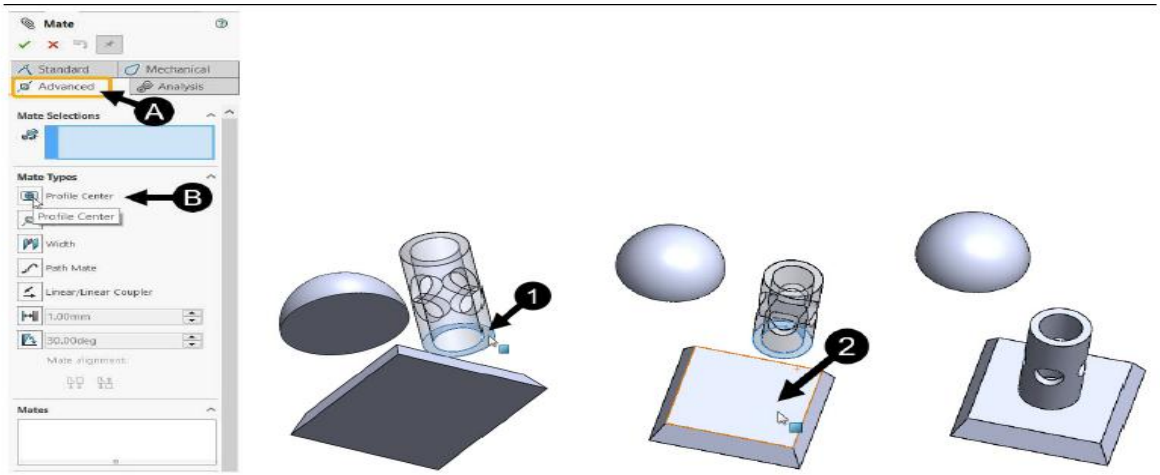
- Click on the **Angle** button from the **Mate PropertyManager**, as shown above.
- Click on two flat surfaces to position them at an angle with each other, as shown.
- Next enter or adjust the required angle value in the **Angle** spinner of the **Mate** pop-up toolbar, as shown.
- Next click on the **Add/Finish Mate** button to place the selected items at the defined angle value, as shown.



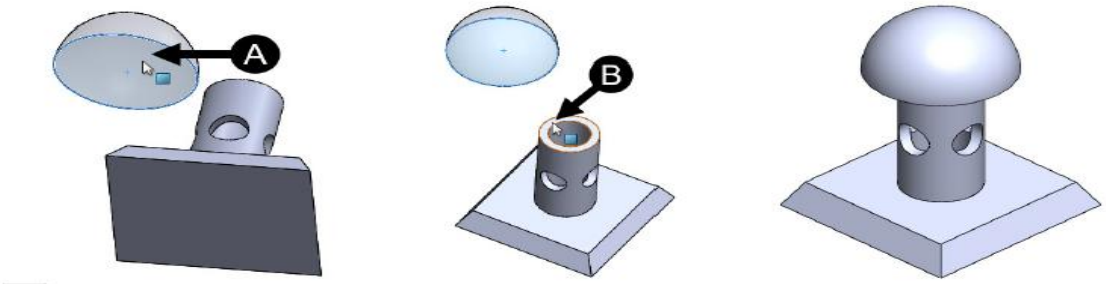
Profile Center

This button is selected to center align rectangular and circular components to each other and makes the components fully defined.

- Activate the **Mate** tool to display **Mate PropertyManager**, as shown.
- Next click on the **Profile Center** button under the **Advanced Mates** rollout, as shown.
- Click on planar faces of cylindrical and rectangular components to display preview of components, center align with each other.
- Next click on the **OK** button of PropertyManager to display components center align with each other, as shown.



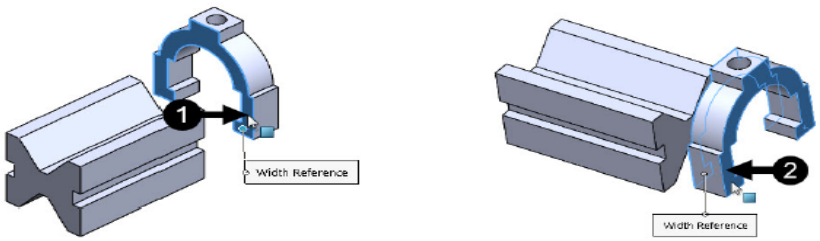
➢ Similarly, click on other two planar faces to center align them with each other, as shown.



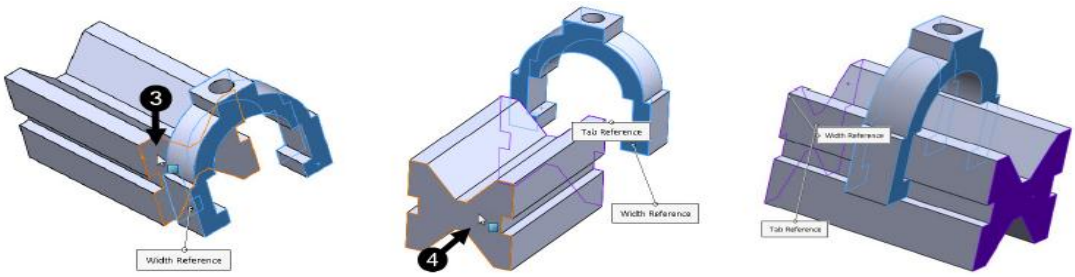
Width

This button is selected to constrain a component between two planar surfaces of another component.

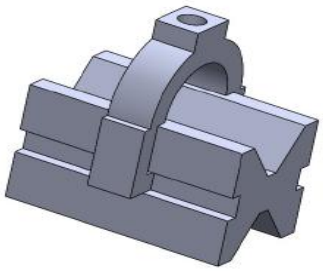
➢ Activate the **Mate** tool to display **Mate PropertyManager**, as shown.
➢ Next click on the **Width** button under the **Advanced Mates** rollout, as shown above.
➢ Click on the two planar surfaces of the component to mate it with another component, as shown.



➢ Next click on two planar surfaces of another component to display preview of components with Width mate, as shown.

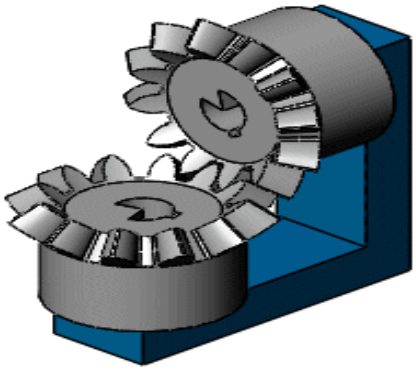


➢ Click on the OK button of the PropertyManager to display components with Width mate between each other, as shown.



Gear mates

Gear mates force two components to rotate relative to one another about selected axes. Valid selections for the axis of rotation for gear mates include cylindrical and conical faces, axes, and linear edges.



Also, you can mate any two components that you want to rotate relative to one another. You do not have to mate two gears.
Similar to other mate types, gear mates do not prevent interference or collisions between components. To prevent interference, use Collision Detection or Interference Detection.

To add a gear mate:

- Click Mate (Assembly toolbar) or Insert > Mate.
- In the PropertyManager, on the Mechanical tab, click Gear.
- Under Mate Selections, select the rotation axes on the two gears for Entities to Mate.
- Under Mate Type.

Option	Description
Ratio	The software assigns gear ratios based on the relative sizes of the cylindrical faces or circular edges you select. The values are parametric. You can override the values. To revert to the default value, delete the override value. The background color of the box is white for the default value, and yellow for the override value.

Option	Description
Reverse	Select Reverse to change the direction of rotation of the gears relative to one another.

- Click ok.

2.4.2: Self-check

1. What is the function of the **Insert Component** tool in an assembly?
2. What does the **Move or Rotate Component** tool allow you to do?
3. Why would you use the **Edit Component** option inside an assembly?
4. What is the purpose of the **Pattern Component** tool?
5. What does a **Concentric mate** do?
6. What is the purpose of a **Parallel mate**?
7. When should a **Width mate** be used?
8. What does a **Slot mate** help control in an assembly?

2.4.3 Answer key:

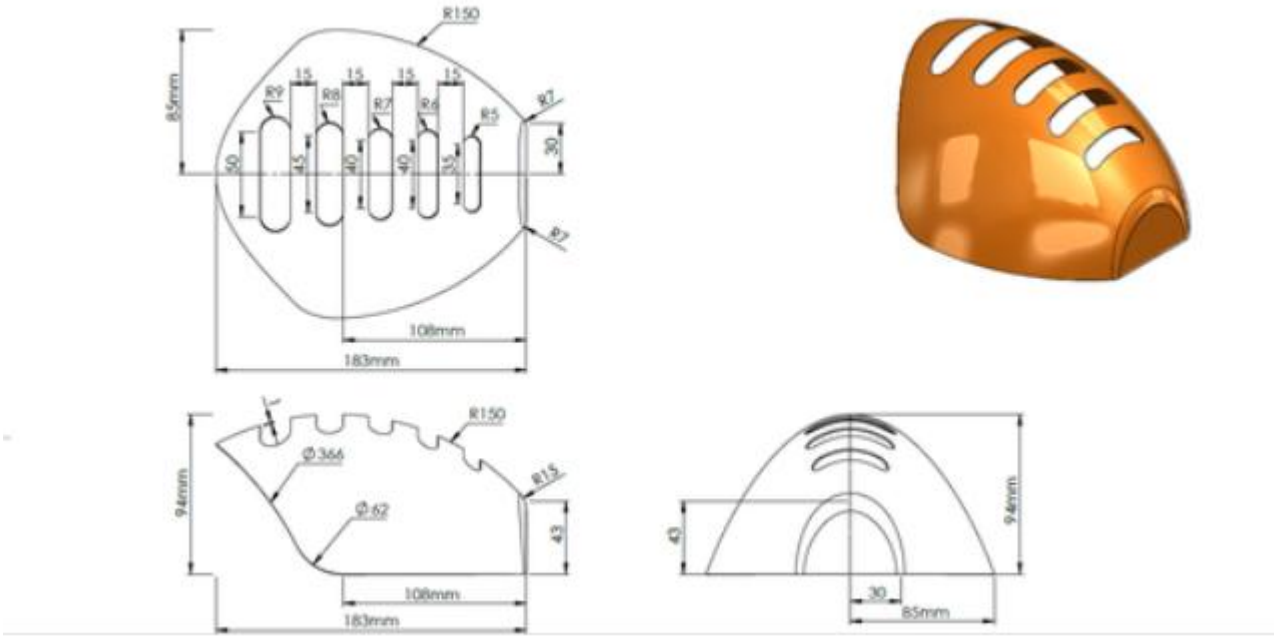
Self-Check Questions	Answer Key / Suggested Answer
1	It allows you to bring individual part files into the assembly workspace.
2	It lets you reposition or rotate a component before applying mates.
3	To modify a part directly within the assembly environment.
4	To duplicate a component in a linear or circular pattern.
5	It aligns two circular or cylindrical features to share the same center axis.
6	It keeps two faces or edges parallel to each other.
7	When you need to center a component between two faces or surfaces.
8	It constrains a part to move along a slot while preventing rotation.

2.4.4 Job Sheet: (A)

Job	**Create 3D drawings of sample**	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	

Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface/ Assembly/Mate Tools. Follow trainers/instructors Save drawings in a folder.

- **Sample:**

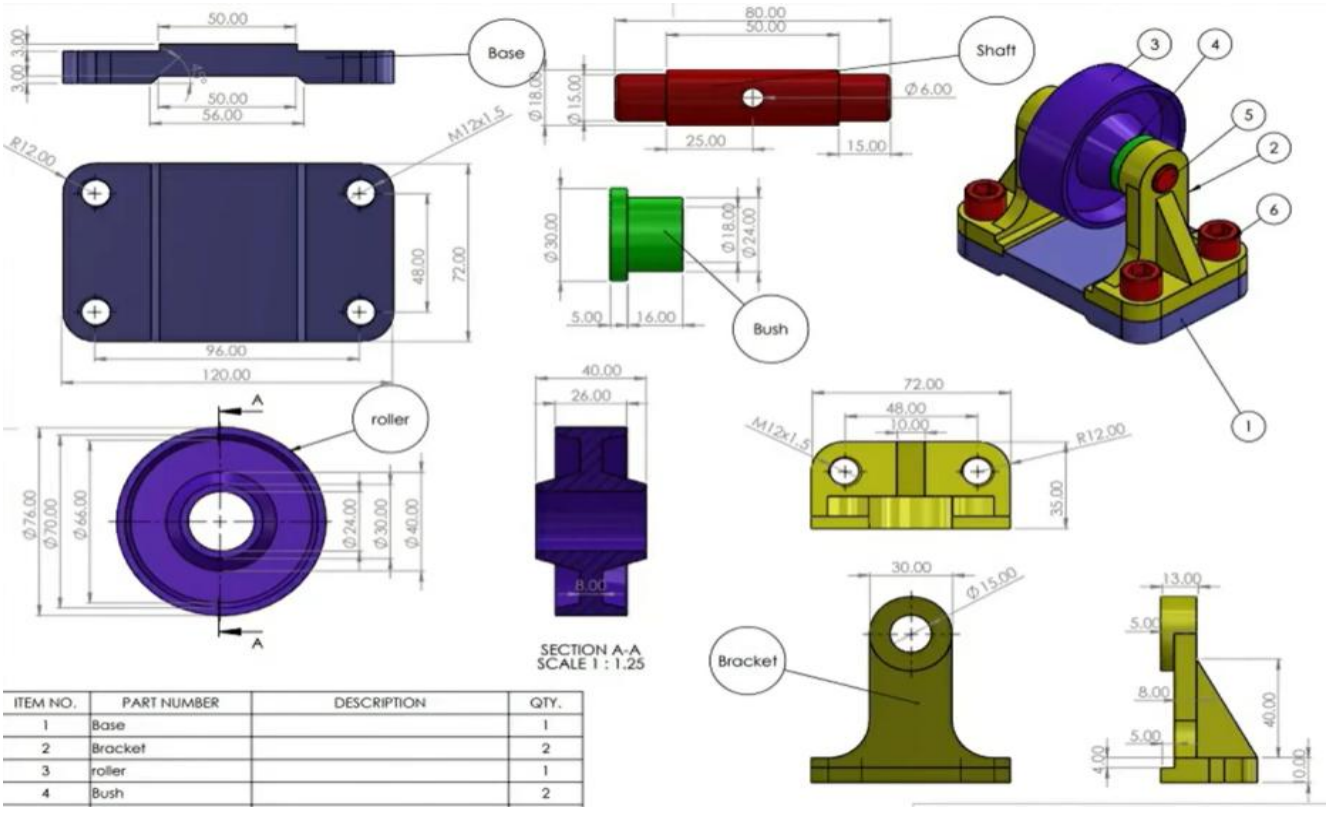


2.4.4 Job Sheet: (B)

Job	Create 3D drawings of sample	
Materials/Resources	Sample Drawing	1Pc
	Computer set	1Pc
	CAD software (SolidWorks)	

Procedure	Trainee will collect required materials/resources. Open CAD software (SolidWorks) Set unit Use Sketch Tools for 2D drawings. Give dimensions. Make relations. Trim unnecessary sketches. Use Feature/Surface/ Assembly/Mate Tools. Follow trainers/instructors Save drawings in a folder.

- **Sample:**



Module-3: Performing Fused Deposition Modelling (FDM) Printing

Module descriptor:
This unit of competency covers the knowledge, skills and attitudes required to perform fused deposition modelling (FDM) printing. It specifically includes the tasks of preparing the FDM printer for operation, configuring print settings and parameters using slicer, monitoring the 3D printing process and performing post-processing of printed parts.

Nominal Duration: 70 hours

Learning outcome:
3.1 Prepare the FDM printer for operation
3.2 Configure print settings and parameters using slicer
3.3 Monitor the 3D printing process
3.4 Perform postprocessing of printed parts

3.1 Learning outcome: Prepare the FDM printer for operation

Learning Objectives:

At the end of this section, a trainee will be able to know –
1. Build platform or heatbed cleaning and bed levelling
2. Filament loading and unloading
3. Z-offset Calibration and first layer print

3.1.1 Information sheet: Printer Interface Layout and Control

Throughout this module, we will be using a desktop FDM printer manufactured by Prusa Research a.s., Czech Republic, and the printer model is Original Prusa i3 MK3S+.

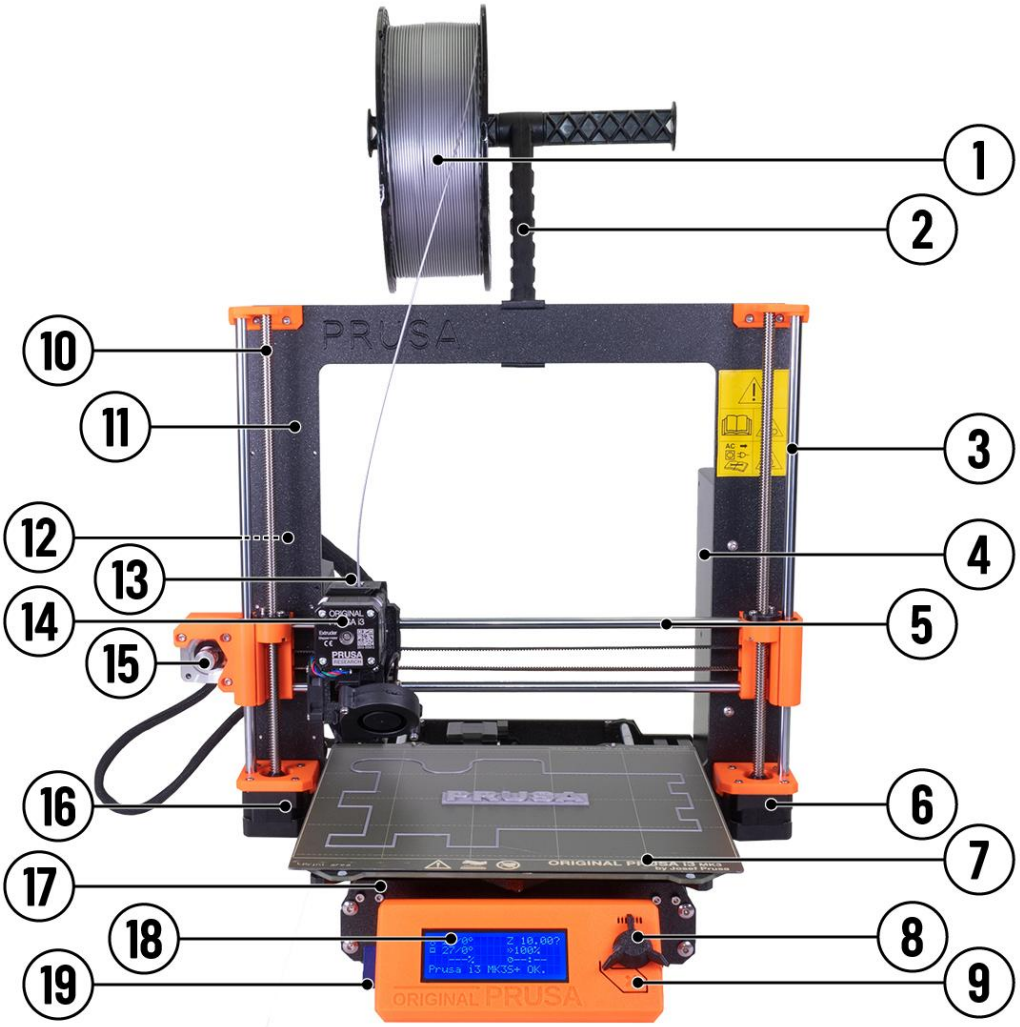


Figure 3.1: Original Prusa i3 MK3S+ printer description: (1) Filament spool (2) Spool holder (3) Z-Axis (4) Power supply (5) X-Axis (6) Z2 Stepper (7) Heatbed with spring steel sheet (8) LCD Knob (9) Reset Button (10) Threaded Rod (11) Printer Frame (12) USB Port (13) Einsy Rambo Board (14) Extruder Stepper (15) X-Stepper (16) Z1 Stepper (17) Y-Axis (18) LCD Panel (19) SD Card Slot

Mechanical and build components

- **Frame:** Milled dural frame for rigidity and stability.
- **Extruder:** Direct-drive extruder with Bondtech gears for reliable filament control.
- **Hot end:** An E3D V6 hot end, capable of reaching a maximum temperature of 300°C.
- **Build Plate:** Magnetic build plate with removable PEI spring steel sheets.
- **Nozzle:** Default nozzle is 0.4 mm, but other sizes are supported.
- **Build Volume:** 250 x 210 x 210 mm.
- **Materials:** Supports a wide range of thermoplastics, e.g., PLA, PETG, ABS, PC, Nylon etc.
- **Layer Height:** Ranging from 0.05 to 0.35 mm.
- **Filament:** Uses 1.75 mm diameter filament.

Control and electronics

- **Controller:** Einsy Rambo 8-bit board co-developed with Ultimachine, featuring Trinamic 2130 drivers for sensorless homing on the X and Y axes.

- Sensors:
  - **SuperPINDA (Prusa INDuction Autoleveling) Probe:** Advanced sensor for high-accuracy, automatic mesh bed leveling.
  - **IR Filament Sensor:** Detects filament run-out and pauses the print.
  - **Other:** 3 thermistors and fan RPM sensors are included for monitoring.
- **Power:** Custom 240W Delta power supply with hardware-based Power Panic for recovering from power loss.
- **Connectivity:** Ethernet and Wi-Fi connectivity are available via an optional Raspberry Pi.

Printer Control

This printer can be controlled in two ways, through LCD panel integrated with the printer or, from computer using Pronterface via USB cable. LCD panel is more reliable to use because there will be no lagging issue.



Figure 3.2: LCD Control Panel Main Screen

The main screen or, info screen of the LCD panel contains –

- Nozzle temperature (actual / desired temperature)
- Heatbed temperature (actual / desired temperature)
- Progress of printing in % - shown only during the printing
- Status bar (messages/ print file name)
- Z-axis position
- Printing speed
- Remaining time estimation

LCD Panel layout

- ❑ Info screen
- ❑ Live adjust Z (Shows during the printing process only)
- ❑ Tune (Shows during the printing process only)
  - Speed
  - Nozzle

- Bed
- Fan speed
- Flow
- Change filament
- Mode

❏ Pause print (Shows during the printing process only)
❏ Stop print (Shows during the printing process only)
❏ Preheat
- PLA - 215/60
- PET - 230/85
- ASA - 260/105
- PC - 275/110
- PVB - 215/75
- ABS - 255/100
- HIPS - 220/100
- PP - 254/100
- FLEX - 230/50
- Cooldown

❏ Print from SD
❏ AutoLoad Filament (Shows when filament is not loaded)
❏ Unload filament (Shows when filament is loaded)
❏ Settings
- Temperature
    - Nozzle
    - Bed
    - Fan speed
- Move axis
    - Move X
    - Move Y
    - Move Z
    - Extruder
- Disable steppers
- Filament sensor - On / Off
- F. autoload - On / Off / N/A
- Fans check - On / Off
- Mode - Normal / Stealth
- Crash detection - On / Off
- HW setup
    - Steel sheets
    - Nozzle d. - 0.25 / 0.40 / 0.60 / 0.80
    - Checks
    - Nozzle - warn / strict / none
    - Model - warn / strict / none
    - Firmware - warn / strict / none
- FS Action - Pause / Continue
- Fsensor Detection
- Mesh bed leveling
    - Mesh 3x3 / 7x7

    - Z-probe nr. 1 / 3 / 5
    - Magnets comp. - On / Off
  - Lin. correction
  - Temperature calibration - On / Off
  - RPi port - On / Off
  - Live adjust Z
  - Select language
  - SD card - Normal / FlashAir
  - Sort - Time / Alphabet / None
  - Sound - Loud / Once / Silent / Assist
  - Brightness
    - Level Bright - default 130 (50 - 255)
    - Level Dimmed - default 50 (0 - 130)
    - Mode - Auto / Bright / Dim
    - Timeout - default 10 (1 - 999)
- ❑ Calibration
  - Wizard
  - First layer calibration
  - Auto home
  - Belt test
  - Selftest
  - Calibrate XYZ
  - Calibrate Z
  - Mesh Bed Leveling
  - Bed level correction
  - PID Calibration
  - Reset XYZ calibration
  - Thermal Model Calibration
- ❑ Statistics
- ❑ Fail stats
  - Last print
  - Total
- ❑ Support
  - Firmware version
  - Fil. sensor v.
  - XYZ calibration detail
  - Extruder info
  - Sensor info
  - Belt status
  - Temperatures
  - Voltages
  - Dump Memory

3.1.2 Information sheet: Build platform or heatbed cleaning and bed levelling

Build platform or heatbed cleaning

To clean an FDM 3D printing bed, use **isopropyl alcohol** for routine cleaning of a clean bed, and **warm, soapy water** for deeper cleaning of stubborn residues like glue. Always wipe with a **microfiber cloth** or

lint-free paper towel and avoid touching the print surface with hands, and ensure the bed is dry before every print.

Routine cleaning

For everyday dust and fingerprints, the print bed can be wiped with a cloth dampened with isopropyl alcohol (70-95% IPA). A clean microfiber or lint-free paper towel will be good choice to use and allow the bed to dry completely. Spraying the alcohol directly onto the bed must be avoided, as it can seep into printer components. Instead, the alcohol should be applied it to the cloth first.

Deep cleaning

For tougher residue, like dried glue stick or hairspray, warm water with mild dish soap or detergent can be used. First, the surface is gently scrubbed with a soft sponge or microfiber cloth socked with soapy water. using harsh or abrasive pads should be avoided, as it can damage the top surface. Then, the bed is needed to be Rinsed completely with warm water and dried fully before next print. After a deep clean, IPA can be used for a final wipe down and to ensure no moisture remains.

Tips for all cleaning methods

Check your bed material: Some surfaces, like certain PEI sheets, may have specific cleaning instructions from the manufacturer.
Use a plastic scraper: For hard-to-remove filament buildup, a plastic scraper is a good option. Avoid metal scrapers, as they can scratch the surface.
Work on a cool bed: Ensure the bed is cool and powered off before cleaning to avoid burns or damage to the print surface.
Clean frequently: Regular cleaning after each print will help maintain proper bed adhesion for your prints.

Calibrate Z

Calibrate Z is located in the *LCD Menu - Calibration - Calibrate Z*. It is always done with the steel sheet on. It should be performed whenever the printer is moved to a different location. It saves the heights of all 9 calibration points in non-volatile memory. The stored information is used every time mesh bed leveling is called during a print. Before starting Z calibration, the nozzle must be cleaned by removing the plastic debris from the nozzle.

Mesh bed leveling

Mesh bed leveling can be found in *LCD Menu - Calibration - Mesh Bed Levelling*. This procedure is performed before every print. The SuperPINDA probe goes through a number of points in a grid pattern spread across the sheet plate (whether it is a powder coated or smooth PEI does not matter) and measures the distance to the sheet. These points are interpolated and used to create a virtual mesh of the bed. During printing, if the bed is slightly warped, the probe will still precisely follow the surface according to its measured mesh.

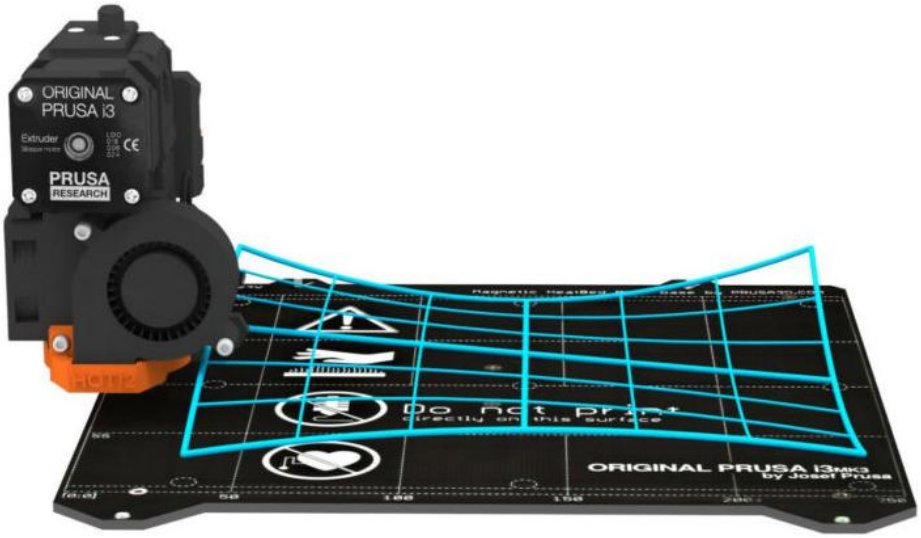


Figure 3.3: Visualization of mesh bed leveling

The Mesh bed leveling settings can be found within *Settings - Mesh bed Leveling*, on the LCD menu. There are 3 options:
Mesh (3x3 - 7x7)
Z-probe nr. (1 - 3 - 5)
Magnet comp. (ON / OFF)
Mesh: The default grid density is 3x3 (9 points). However, the grid density can be increased to 7x7 (49 points) for better bed levelling.
Z-probe nr.: This setting indicates how many times will each point be measured. It is suggested that to take either 3 or 5 measurements per point since the final value is calculated as an average value.
Magnet comp.: When 7x7 Mesh bed leveling is used, 3 of the 49 points are so close to magnets, that the reading can be off by a certain value. If the Magnet compensation is on, then the printer ignores readings from these 3 points and calculates them from the surrounding points.

3.1.3 Information sheet: Filament Loading and Unloading

Before loading a filament, it is advised to check whether the idler screw is about flush to protruding 0.5 - 1 mm with the plastic part when the filament is not loaded.

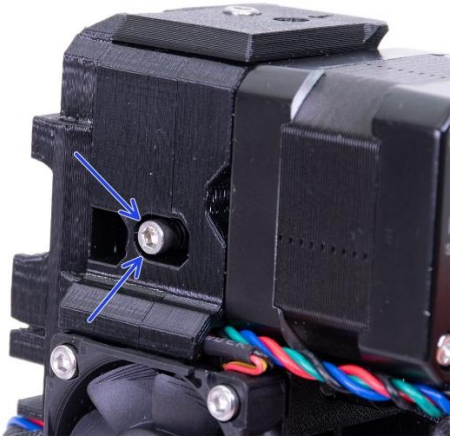


Figure 3.4: Idler Screw Tension Check

In this printer, the filament can be loaded using *AutoLoad Filament* option from LCD panel. To do that, the printer should be preheated to the specific temperature of the filament and then, the filament is needed to insert into the extruder. Everything is automated from this point. The filament tip should be nice and pointy. The Z-axis will rise in case the current Z coordinate is less than 20mm from the print bed. This will ensure that there is always enough space for cleaning the nozzle.

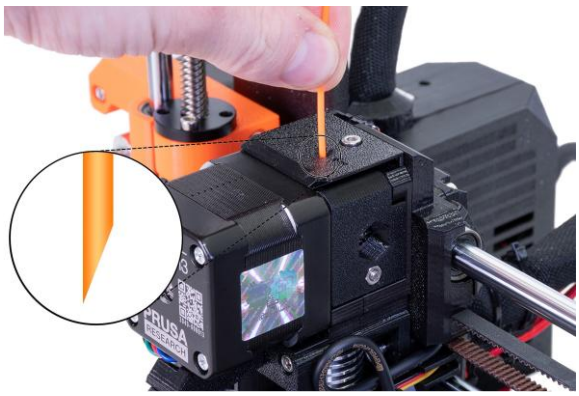
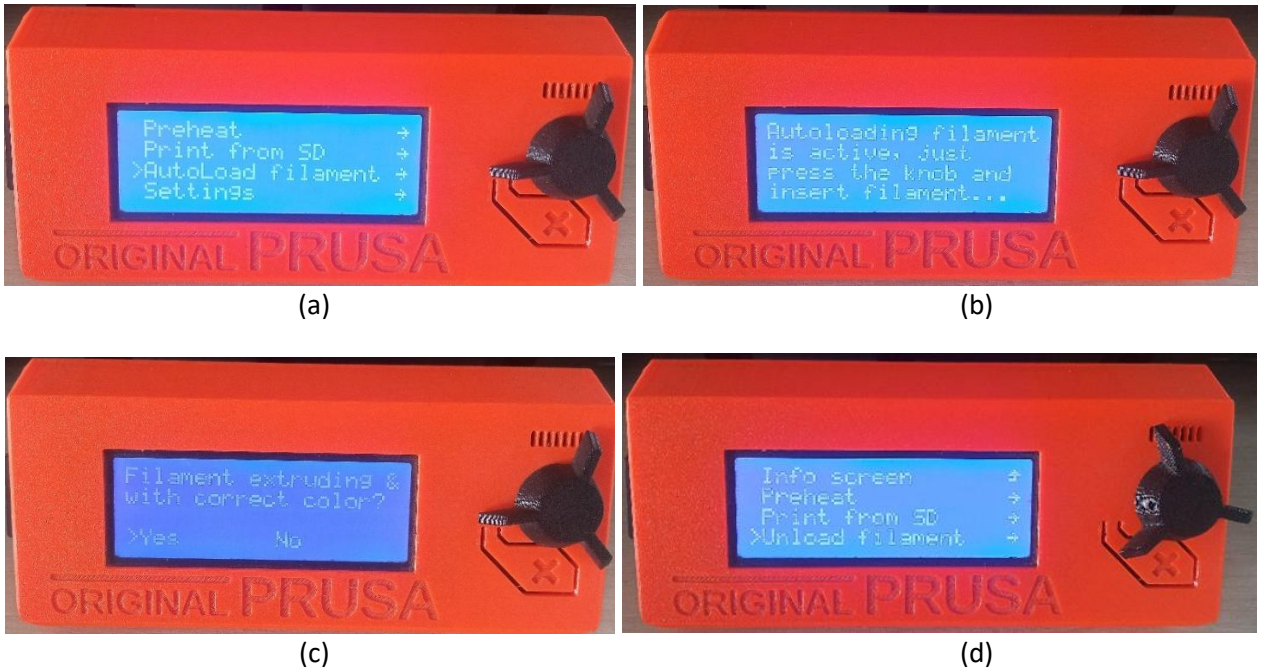


Figure 3.4: Loading the filament into the extruder

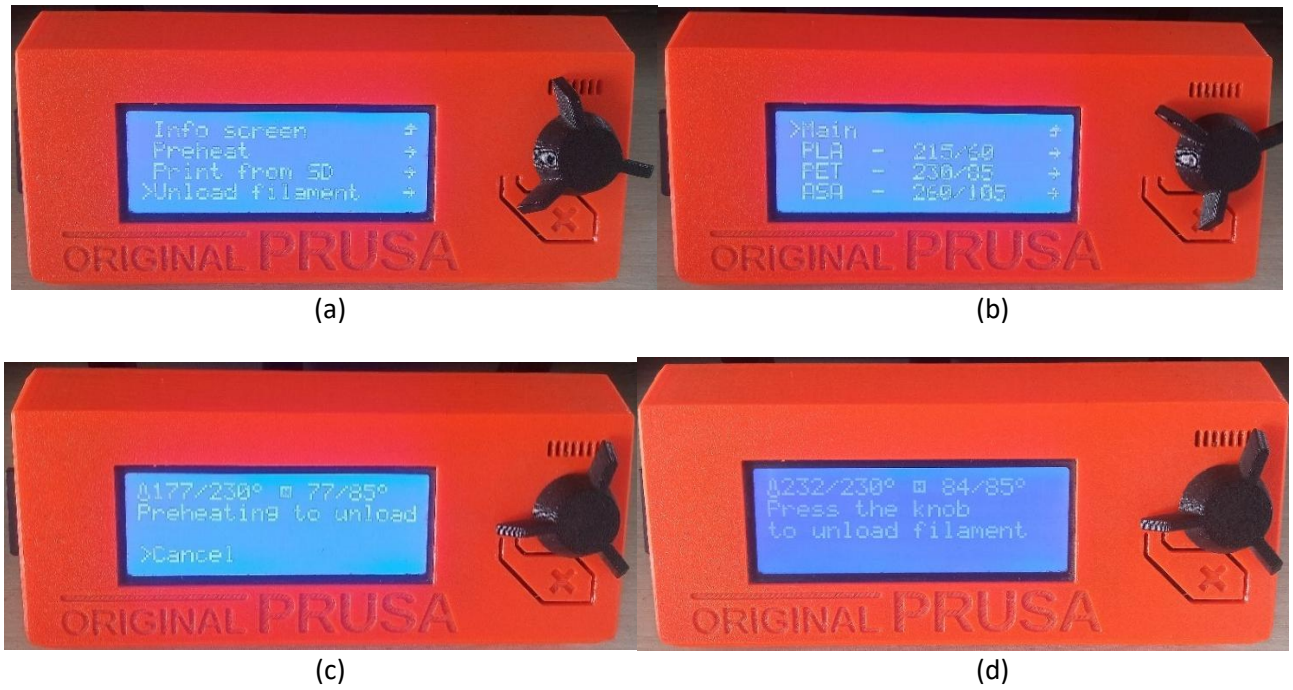


(a) (b)

(c) (d)

Figure 3.5: Steps of AutoLoad Filament

If a filament is loaded into the printer, the LCD panel will show the *Unload filament* option. To take out the filament loaded into the extruder, *Unload filament* option is needed to be selected. After that, the printer will check whether the nozzle is heated or not. If the nozzle is not preheated, the preheat menu will be automatically shown. From that menu, the desired filament type must be selected and the printer automatically starts preheating. After the printer reaches the target temperature, a prompt to unload the filament will be appeared. Then, just by pressing the control knob, the filament can be unloaded. If the nozzle was preheated, the filament is unloaded immediately.



(a) (b)

(c) (d)

Figure 3.5: Steps of Unload Filament

3.1.4 Information sheet: Z-offset Calibration and first layer print

The First Layer Calibration is used to calibrate the distance between the tip of the nozzle and the print surface. The aim is to adjust the nozzle height until the extruded plastic sticks nicely to the bed and you can see that it is being slightly squished.

As each type of steel sheet (smooth, satin, and textured) has a different thickness, it is imperative to run a different First layer calibration for each of them.

Generally, the first layer calibration should be performed every time there is a major change in the assembly. This includes changing the nozzle, extruder upgrades, location of the printer or other updates to any axis.

The first layer calibration can be run from the *LCD Panel - Calibration - First Layer Calibration*. During this process, the knob should be rotated to manually adjust the distance between the nozzle and the bed, while the printer is printing a zig-zag pattern. It is advisable to turn the knob counter-clockwise to bring the nozzle closer to the bed. Here, the aim is to adjust the nozzle height until the extruded plastic sticks nicely to the bed and it is being slightly squished. The value should not exceed -2.000 mm. If that happens (e.g. -2.500), the SuperPINDA probe must be moved slightly higher. By loosening the M3 screw on the probe holder, the height of the SuperPINDA probe can be adjusted by gently push or pull the sensor and then tighten the M3 screw again. After that, *Calibrate Z* must be re-run followed by the *First layer Calibration*.



Figure 3.6: First Layer Calibration Process

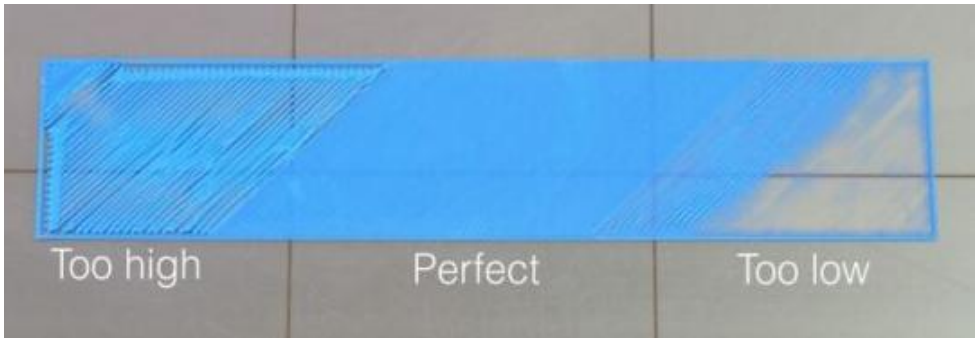


Figure 3.7: Fine Tuned of First Layer

Too high

If any gaps between the lines are appeared (left side of figure 3.7), then it means the nozzle is too high. In that case, the nozzle needs to be closer to the bed.

Too low

If the nozzle is set too low (right side of figure 3.7), the line squished completely flat and the end square will have ridges between the lines. When it is too low, the edges of the square can also start curling upwards. In that case, the nozzle needs to be away from the bed.

Just right

A correct adjustment will show an even surface (center of figure 3.7), with no gaps between lines, nor ridges.

3.1.5 Job sheet: Load a PETG/ABS filament in the printer and run first layer calibration to find Z-offset

Section	Content
Job Title	Load a PETG filament in the printer and run first layer calibration to find Z-offset
Personal Protective Equipment (PPE)	Safety gloves, protective goggles and safety shoes
Equipment, Tools, and Materials	1. FDM 3D printer (PLA/ABS/PETG compatible) 2. PETG Filament spool 3. SD card / USB drive / Wi-Fi transfer 4. Bed cleaning tools (scraper, alcohol wipes) 5. Allen key set
Procedure	1. Preparation - Connect the printer with power outlet and switch on the printer - Collect the PETG/ABS filament spool - Cut the filament in 45 degree approximately - Clean the heatbed and nozzle tip - position the Z-axis above 20 mm - preheat the nozzle for PETG/ABS from Preheat option of the printer LCD panel
	2. Load the filament - place the filament spool on the spool holder - insert the filament tip through the PTFE tube just before the extruder - Run AutoLoad Filament option from the LCD panel
	3. Run First Layer Calibration - Go to Calibration > First Layer Calibration from the LCD panel - run the first layer calibration by press the control knob - find out the problematic area during printing and correct by Z height adjustment.
	4. Test and record - run a single layer print to check whether the Z-offset is right - record the Z height value or Z-offset value

3.1.6 Self-check

Write down the most appropriate answers for the following questions:

4. What type of extruder does the Original Prusa i3 MK3S+ use, and what is its maximum hot end temperature?
5. How can the print bed of the Original Prusa i3 MK3S+ be cleaned for routine maintenance, and what cleaning methods should be avoided?
6. What is the function of the SuperPINDA probe?
7. What are the two ways the Original Prusa i3 MK3S+ printer can be controlled?
8. What are the three adjustable options within the "Settings - Mesh bed Leveling" menu on the LCD panel?

3.1.7 Answer Key

4. Answer:

The printer uses a direct-drive extruder with Bondtech gears and features an E3D V6 hot end capable of reaching a maximum temperature of 300°C.

5. Answer:

For routine cleaning, the bed can be wiped with a cloth dampened with isopropyl alcohol (70-95% IPA). Spraying alcohol directly onto the bed should be avoided, and harsh or abrasive pads should not be used for cleaning.

6. Answer:

The SuperPINDA probe is an advanced sensor used for high-accuracy, automatic mesh bed leveling.

4. Answer:

The printer can be controlled via the integrated LCD panel or from a computer using Pronterface via a USB cable.

5. Answer:

The three options are **Mesh** (grid density), **Z-probe nr.** (number of measurements per point), and **Magnet comp.**

3.2 Learning outcome: Configure print settings and parameters using slicer

Learning Objectives:

At the end of this section, a trainee will be able to know –

1. Basic print settings in slicer
2. Printing with support using slicer
3. Advanced Printing options
4. Calibration test related to the filament

3.2.1 Information sheet: Basic print settings in slicer

Throughout this module, we will be using PrusaSlicer which is a slicer based on the open-source project *Slic3r*. PrusaSlicer was developed by Prusa Research, mainly for their Original Prusa printers.

PrusaSlicer Interface

PrusaSlicer has a fairly simple user interface. Most of the screen is dedicated to the 3D view, which is used to manipulate 3D models. The top toolbar (1, 2) adds and removes objects. The left toolbar (4) has tools for manipulation with 3D models, such as the scale or rotate tool. The right panel is used mainly to select a print, filament, and printer profile (6, 7, 8) and to choose whether or not to generate supports or brim. The "Slice now" button (5) is used to generate G-code, which can then be exported to an SD card or sent directly to a printer.

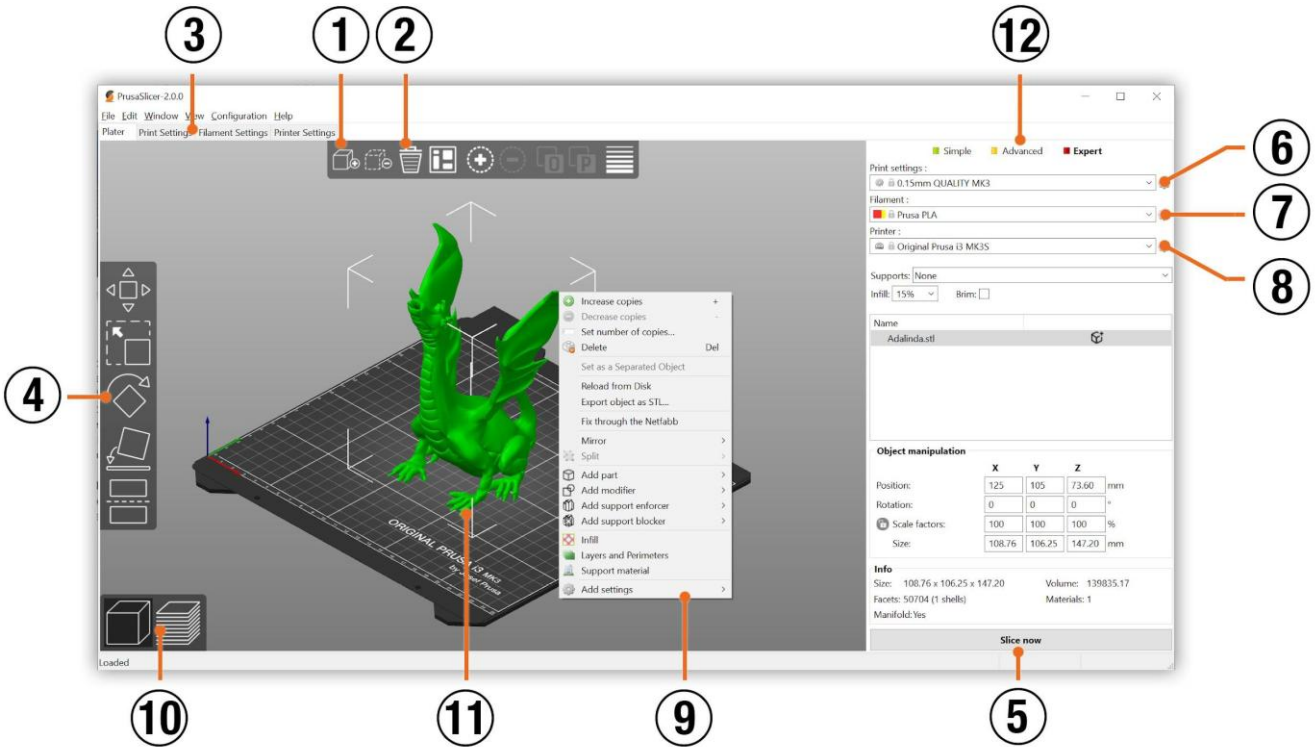


Figure 3.8: PrusaSlicer Interface

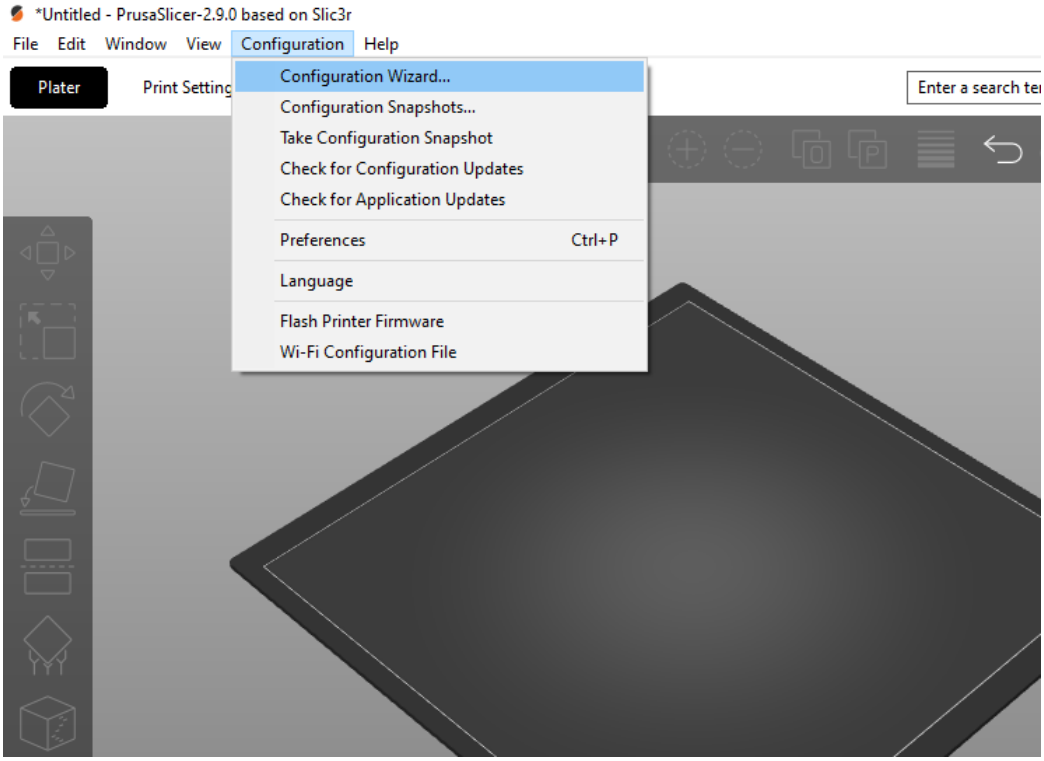
1. The **Add** button loads models into PrusaSlicer
2. **Delete** and **Delete All** buttons remove the model(s) from PrusaSlicer
3. Opens detailed settings of print, filament, and printer
4. **Move, Scale, Rotate, Place on Face** and **Cut** tools
5. Slice and generate the G-code button

6. Layer height and speed of a print
7. Material selection
8. Printer selection
9. **Right-click** on the model to open a context menu
10. Switch between 3D editor and layers preview
11. Model preview
12. Switch between Simple / Advanced / Expert mode

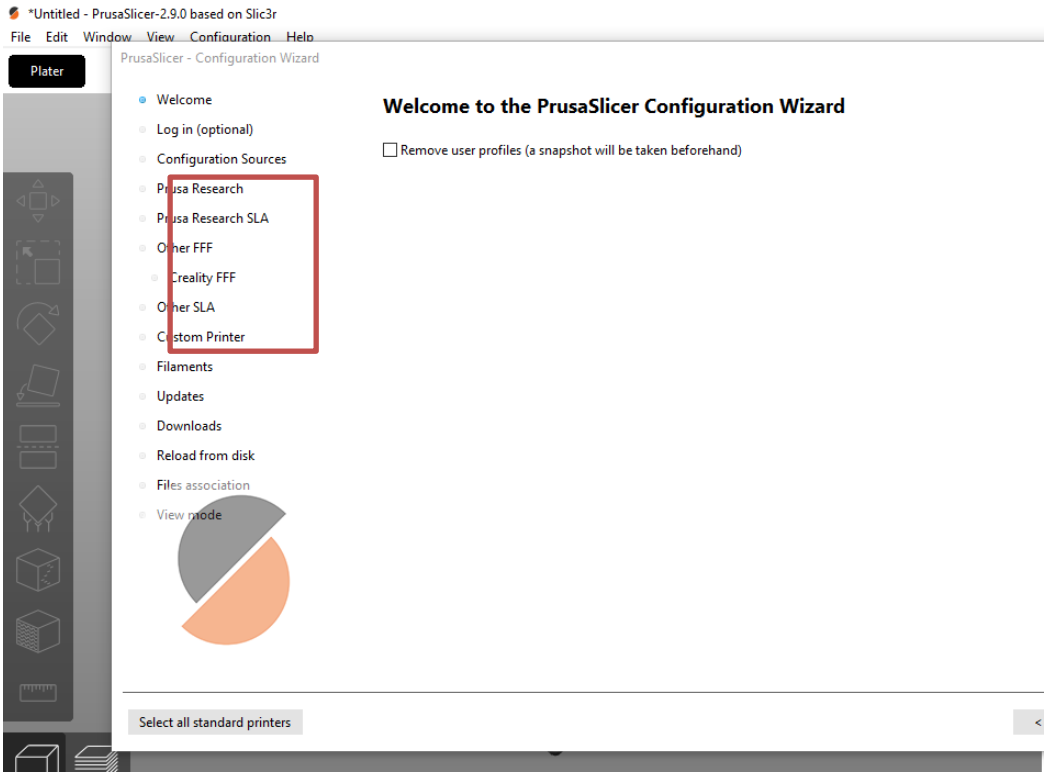
Model Printing with Basic Settings

Selecting printer, filament and profile

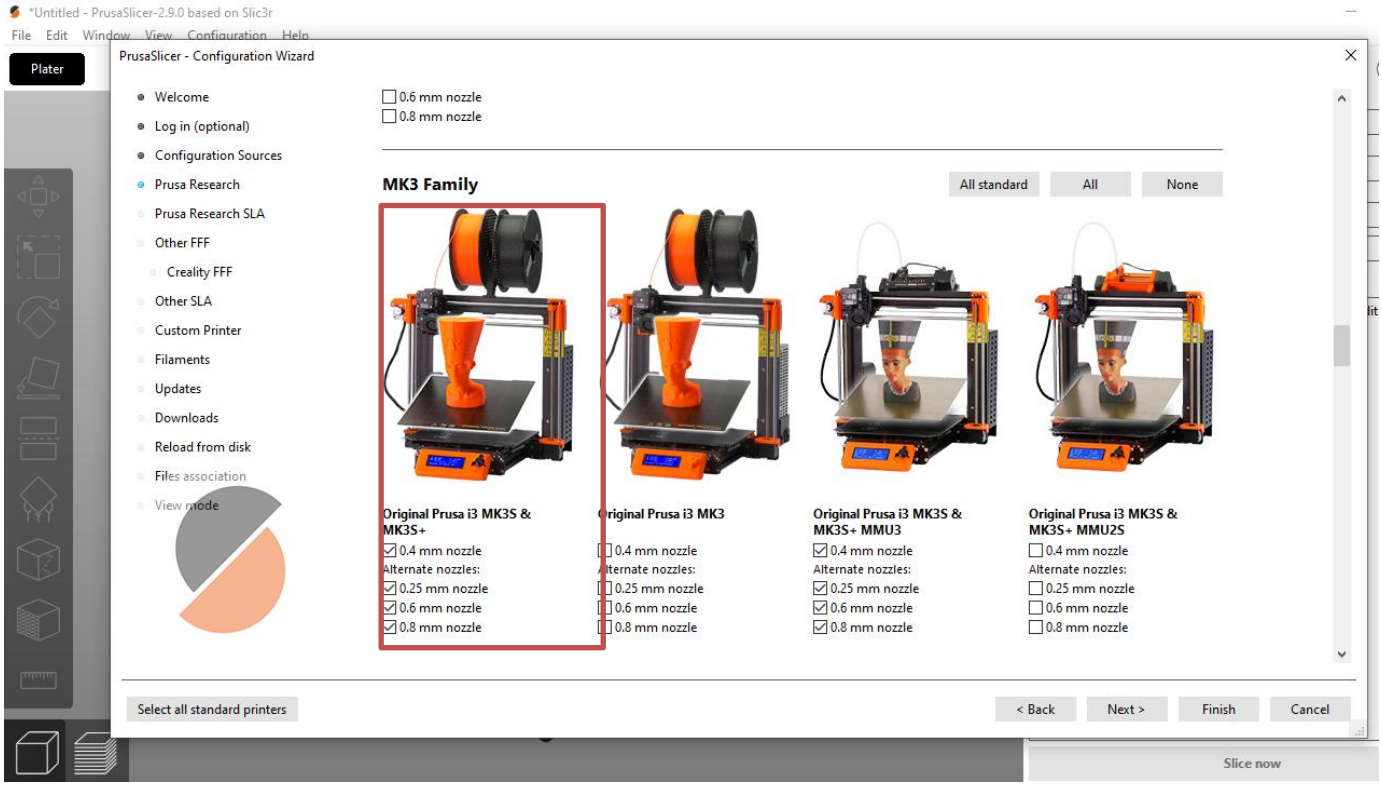
At first, the printer and the nozzle size should be imported from the Configuration tab of the top panel of the slicer. From the configuration tab, Configuration Wizard needs to be selected.



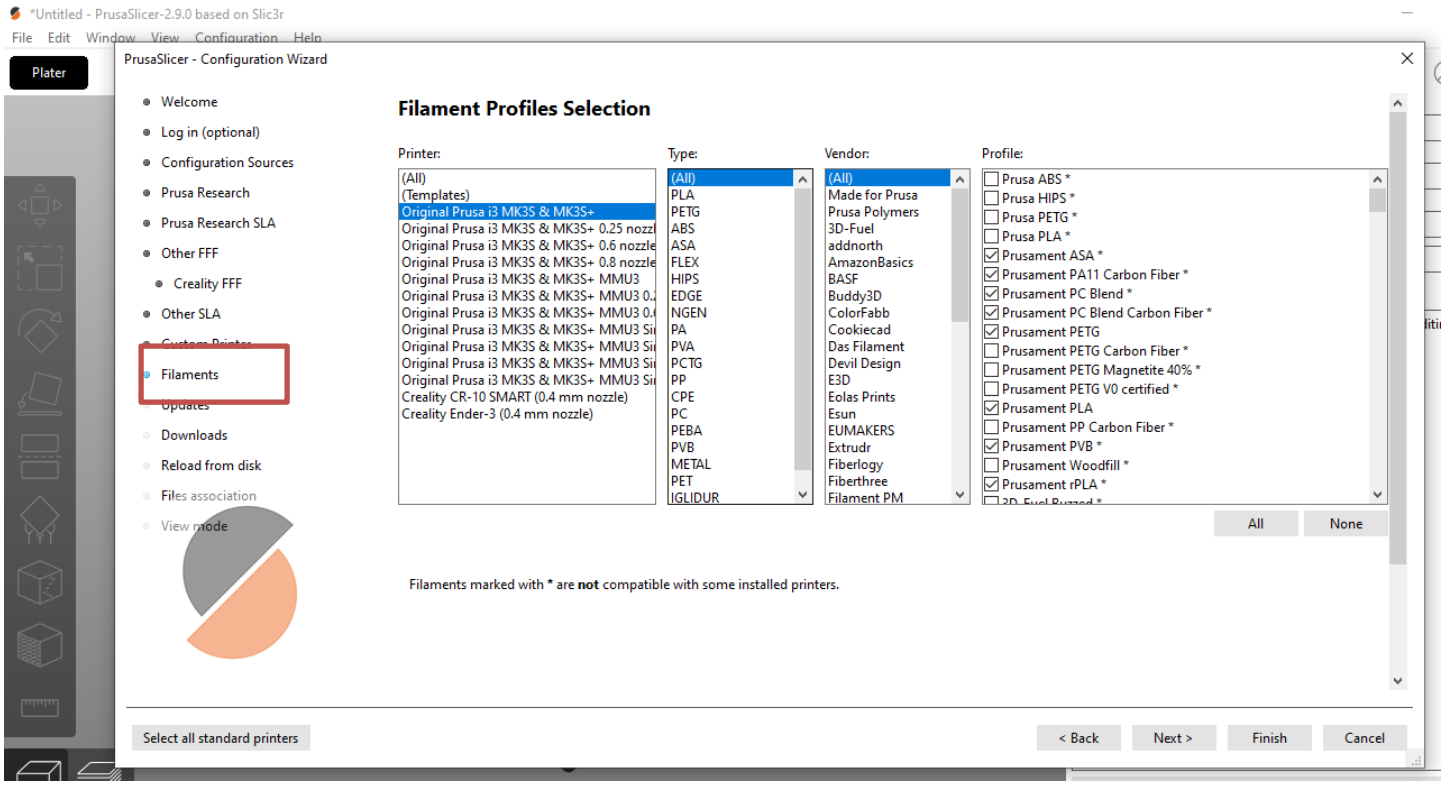
It will take while to open the Configuration Wizard.



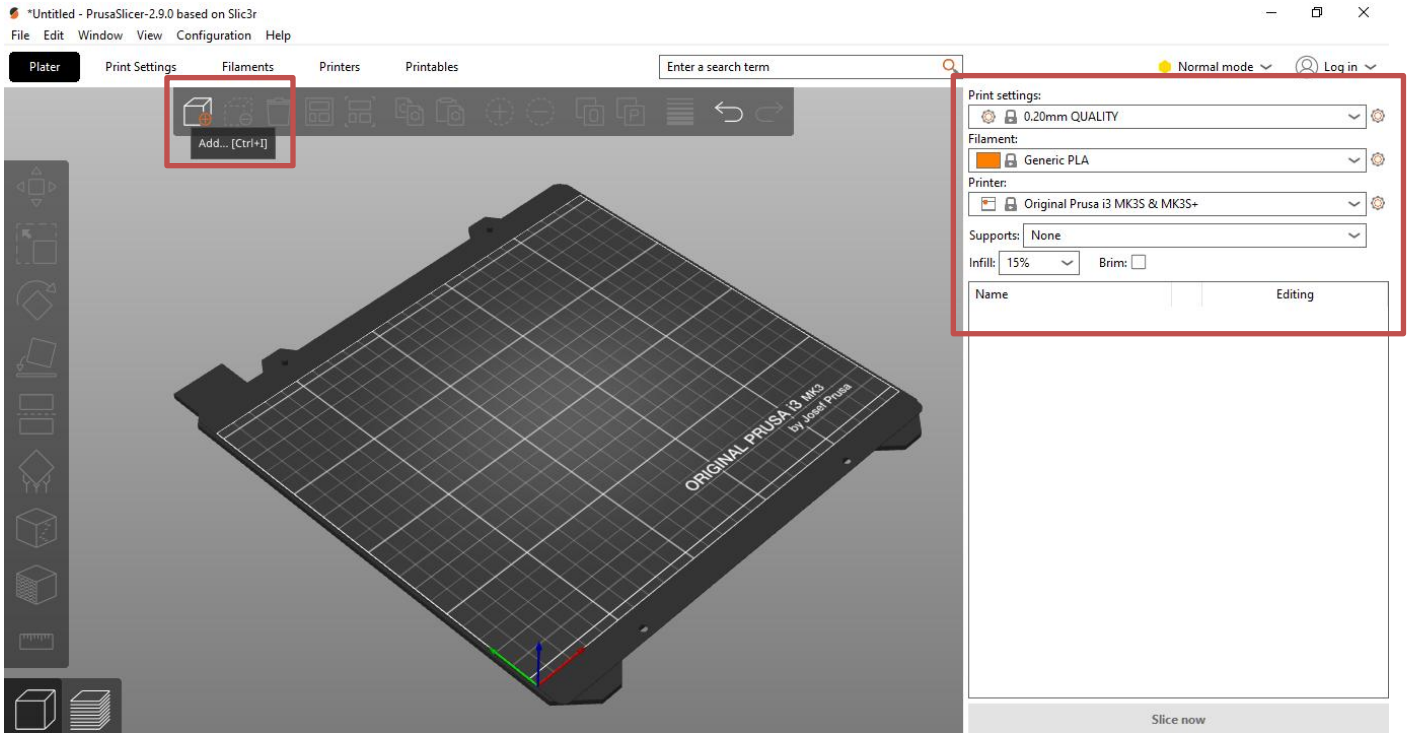
From the left side of Wizard, printer can be selected according the printer's brands. Here, Prusa Research is selected, as the printer used in this module is Original Prusa i3 MK3S+. to find the printer, it is required to scroll down. Along with the printer, the nozzle size must be selected.

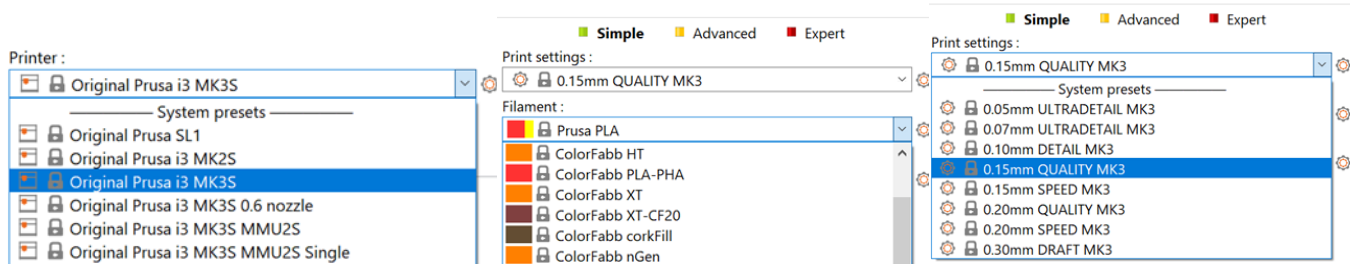


Similarly, the filament and profiles (default print parameters settings) compatible with the chosen printer need to be selected from the Wizard.



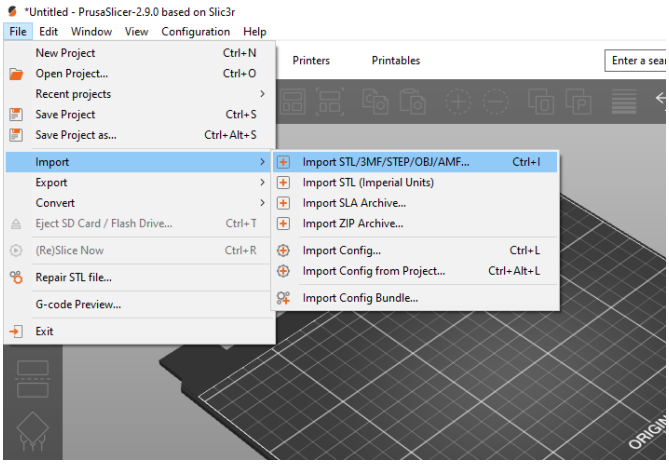
After closing the Configuration Wizard, the selected printer, filament and profile/ layer height can be found on the right panel of the slicer and selected from the drop-down menu in the right panel.

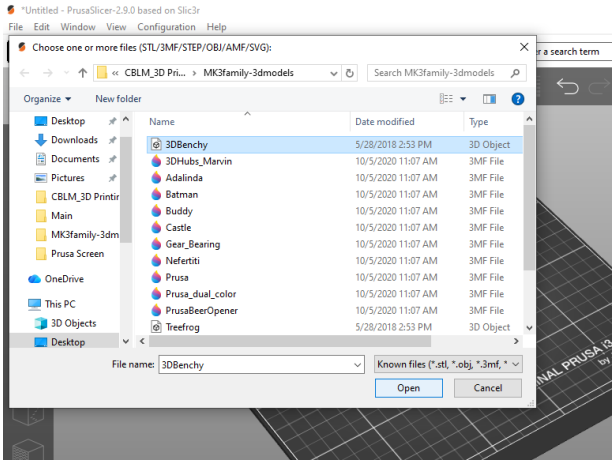




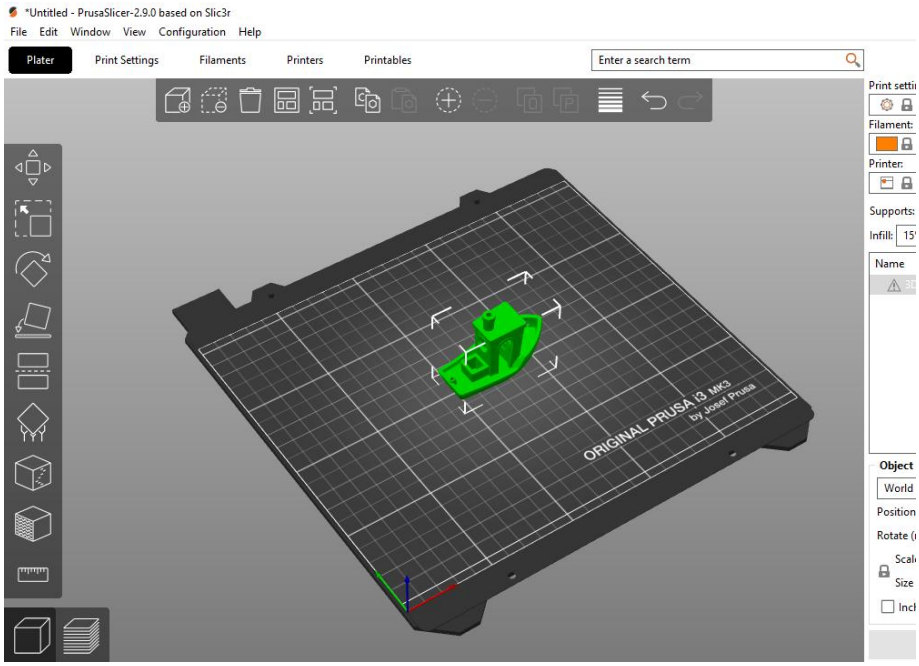
Model import

First, you have to import your model. You can either **drag and drop** it directly to the PrusaSlicer window, use the **Add button** in the top toolbar, or choose **File - Import**. Supported 3D formats are STL, STEP, 3MF, OBJ, and AMF.





After selecting the model from the desired folder, it will be appeared on the build plate of the slicer.

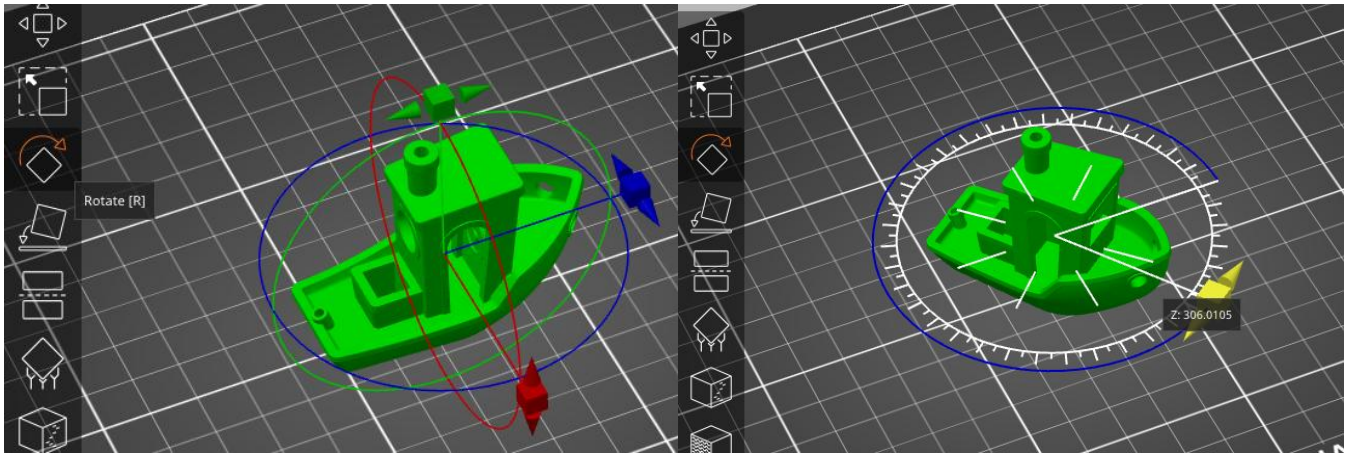


Mouse Gestures

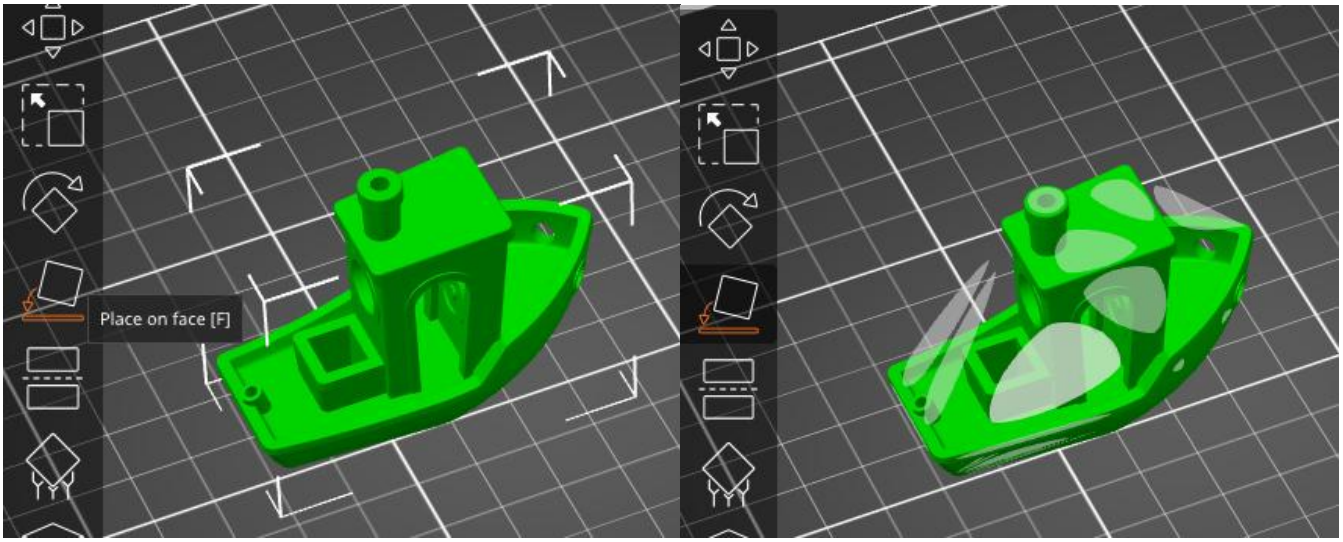
Left mouse button	-	**Rotate Build Plate & Manual positioning of 3D model**
Right mouse button	-	**Pan Build Plate**
Scroll wheel	-	**Pan & Zoom In-Out Build Plate**

Model orientation and manipulation

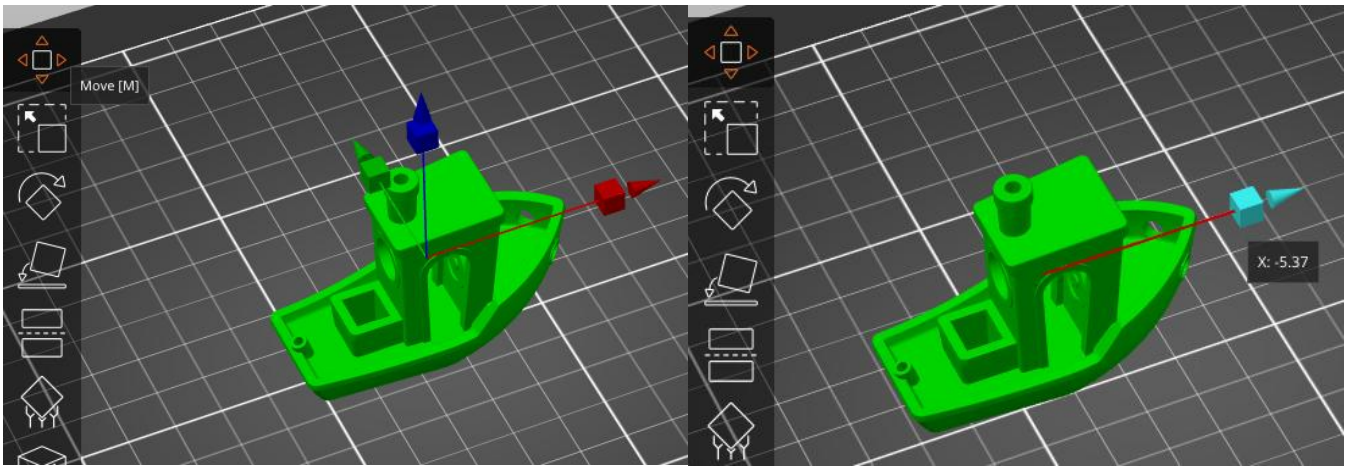
Your model may be oriented incorrectly after import. If that's the case you can use the **Rotate tool** from the left panel of the slicer to manually adjust to the object's orientation. When you pick a rotation axis, you can use the short guidelines to rotate by 5-degree increments or the long guidelines, which snap to 45-degree increments.



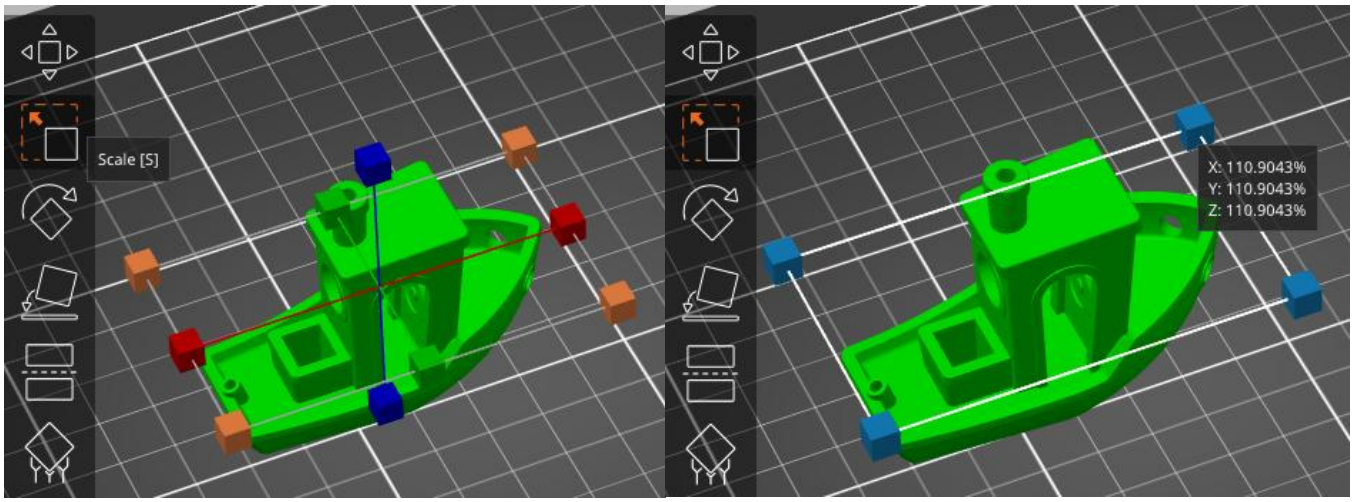
Alternatively, you can use the Place on Face tool. When you select it, several white planes will appear on your model. Clicking on the white plane will align it with the virtual print bed.



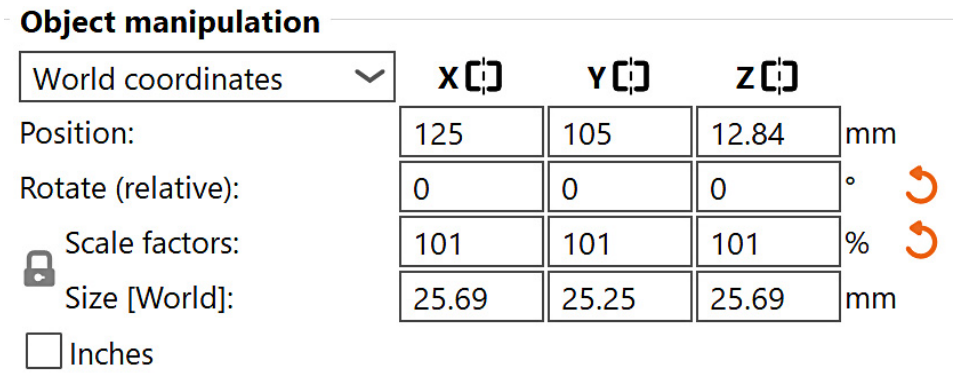
By default, the model will be placed at the center of the build plate. To place the model in any position, Move tool can be used. After selecting the Move tool three arrow directing to three axes (red arrow – x axis, green arrow – y axis and blue arrow – z axis) will be appeared on the model. By clicking on any arrow, the model can be placed to any location on the plate.



If the model needs scaling, a Scale tool is also available. By clicking and dragging the colored square, the model can be scaled up or scaled down.



Model manipulation is also possible from the down-right corner of the slicer.



Print Parameter Settings

The printing parameters can be manipulated or adjusted from the *Print Settings* tab of the top panel. There are three different mode of settings available in PrusaSlicer. These are Beginner mode, Normal mode and Expert Mode. Here, normal mode will be used to get the control over moderate number of print parameters.

Under Layers and perimeters tab, following are the basic parameters –

Layer height: This setting controls the height and the total number of the slices/ layers. Thinner layers give better accuracy but take more time to print.

First layer height When printing with low layer heights you might still want to print a thicker first layer to increase adhesion to the print bed. **Original Prusa print profiles always use 0.20 mm as the first layer height.**

Wall/ Perimeters Defines the minimum number of outlines that form the wall of a model. Original Prusa profiles always use a minimum of two perimeters.

The strength of a model is mostly defined by the number of perimeters (not the infill). If you want a stronger print, increase the number of perimeters.

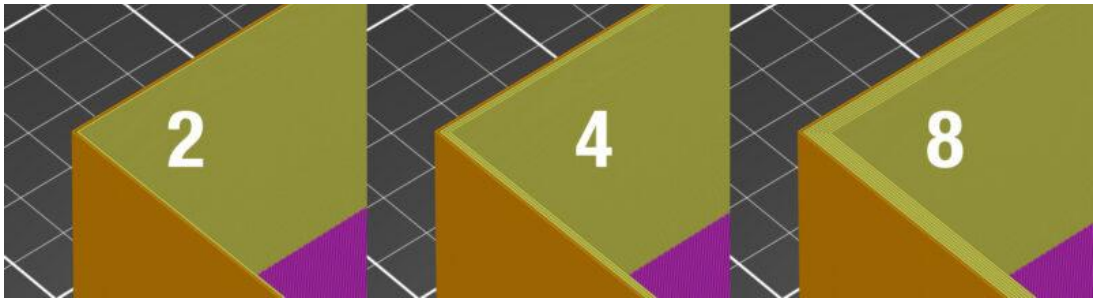


Figure 3.9: Model with different number of perimeters

Solid layers (Top & Bottom) The bottom and top layers of models are usually filled with solid layers.

when printing at low layer heights, you will need more solid layers to achieve the same top/bottom wall thickness (e.g. with 0.3 mm layer height use 3 top layers, with 0.1 mm layer height use 9 top layers).

Seam Position is the positions of perimeter starting point. There are four options available – Random, Nearest, Aligned and rear.

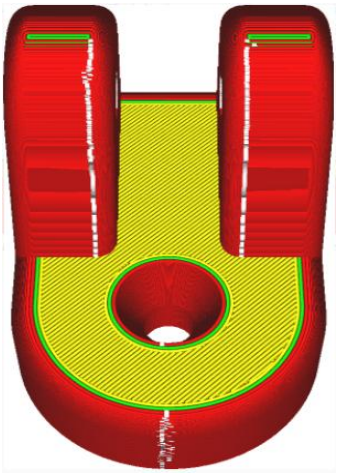


Figure 3.10: With Aligned mode, the seam is visible in one line.

Infill Objects are typically not printed solid, but not hollow either. A sparse infill pattern is used to fill the inside of the model. The main purpose of the infill is to provide an internal support structure for top layers, as well as to improve the mechanical properties of the print. On the other hand, we usually want to get away with as little infill as possible to reduce both print times and material consumption.

Under infill tab, following are the parameters –

Fill Density: This setting configures the density of the volume inside the print, which is a major factor in the strength of the final print. The fill density can also be adjusted in the right panel of slicer UI. Generally, values around 15-20% work well.

Fill Pattern: the fill pattern defines a structure that is used to fill the volume of the object. Various patterns are available, each of which has their own advantages. In PrusaSlicer, Gyroid is set by default.

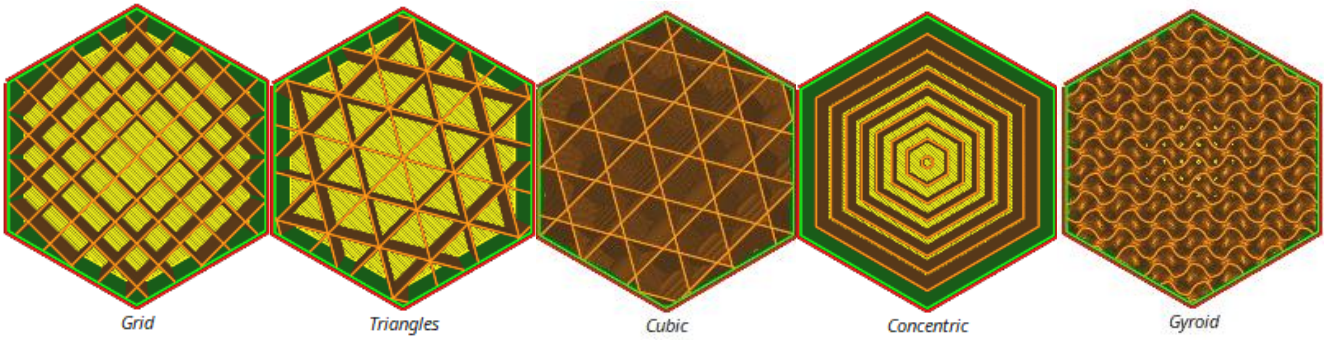


Figure 3.11: Fill Patterns

Under Skirt and brim tab, following are the parameters –

Skirt: A skirt is a plate adhesion option that extrudes an offset outline of the model on the first layer of the print. The skirt is extruded on the print bed before starting to print your model. Skirts serve a useful purpose because they help prime your extruder and establish a smooth flow of filament. The skirt also helps to remove unwanted colors and build pressure for material extrusion.

Brim: A plate adhesion option whose function is to reduce shrinkage of bottom print layers or better adhere a low surface area object by providing a larger base platform.

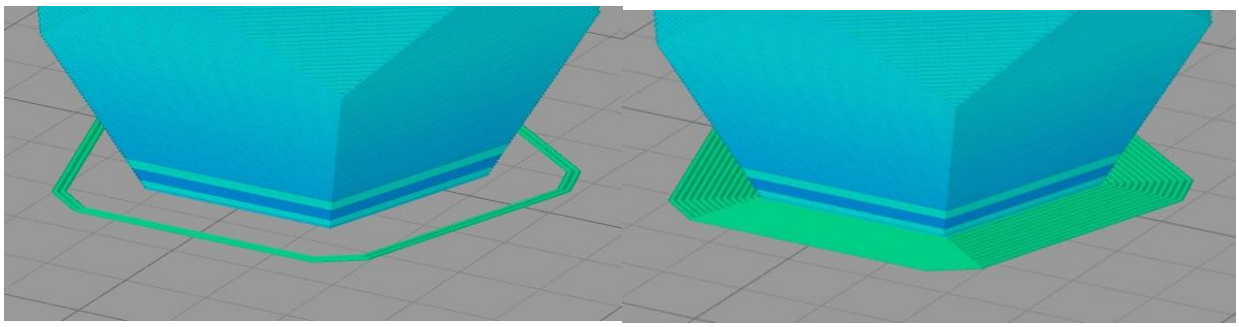


Figure 3.12: Skirt (Left) & Brim (Right)

Under speed tab, there are two types of speed - speed for print moves and speed for non-print moves. First set of speed can be called as print speed. Later set has two types of speed, one for travel along X and Y axis and another one is for Z axis travel.

Print Speed: The rate at which a 3D printer is capable of moving while extruding plastic. A print speed of 50mm/s will be successful on most FDM printers. A print speed of 20-30 mm/s will produce higher quality prints

Travel Speed: The rate at which the nozzle assembly will move to a new position when not extruding.

Filament Settings

The filament settings can be set from the *Filament* tab of the top panel. Under the filament tab, following are the most important parameters –

Nozzle/ Printing Temperature: The temperature of the hot end at which the filament if melted and extruded.

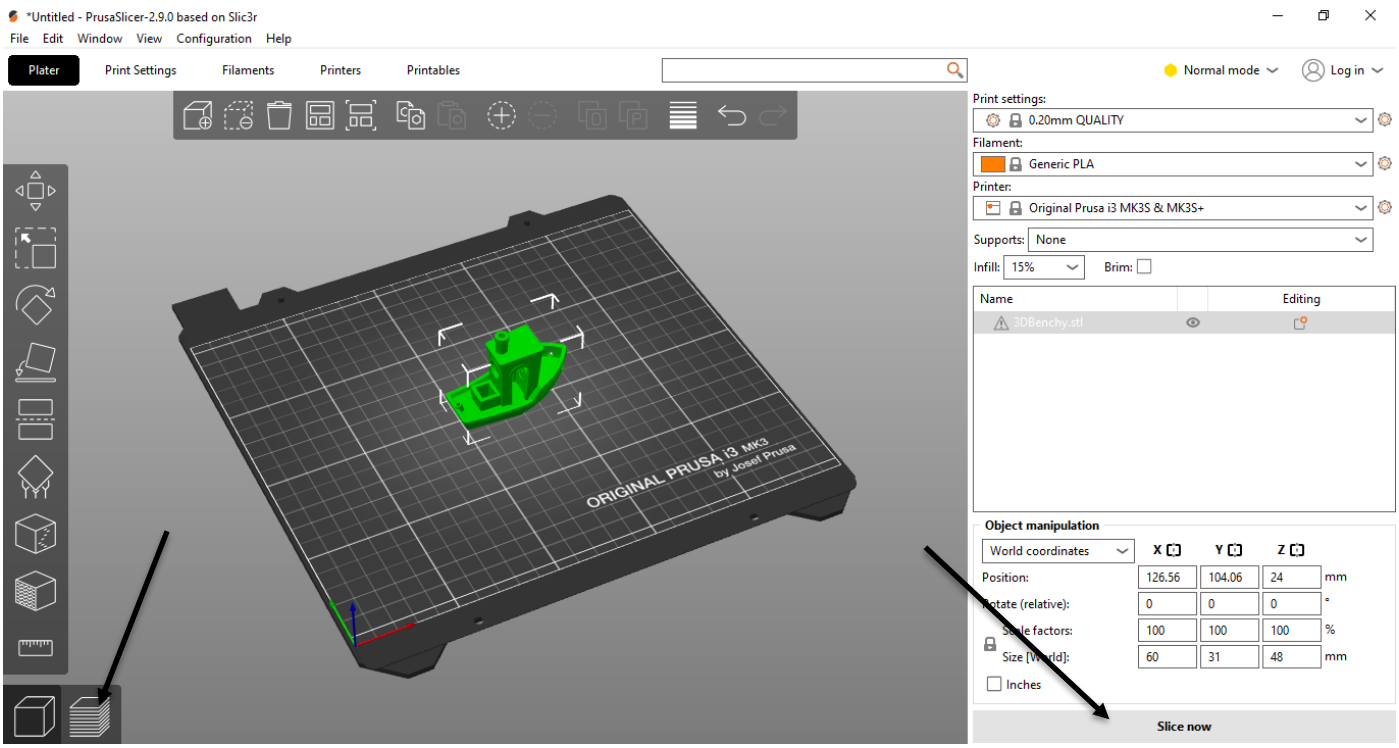
Bed Temperature: The temperature of the build plate, which is crucial for ensuring proper first-layer adhesion and preventing warping.

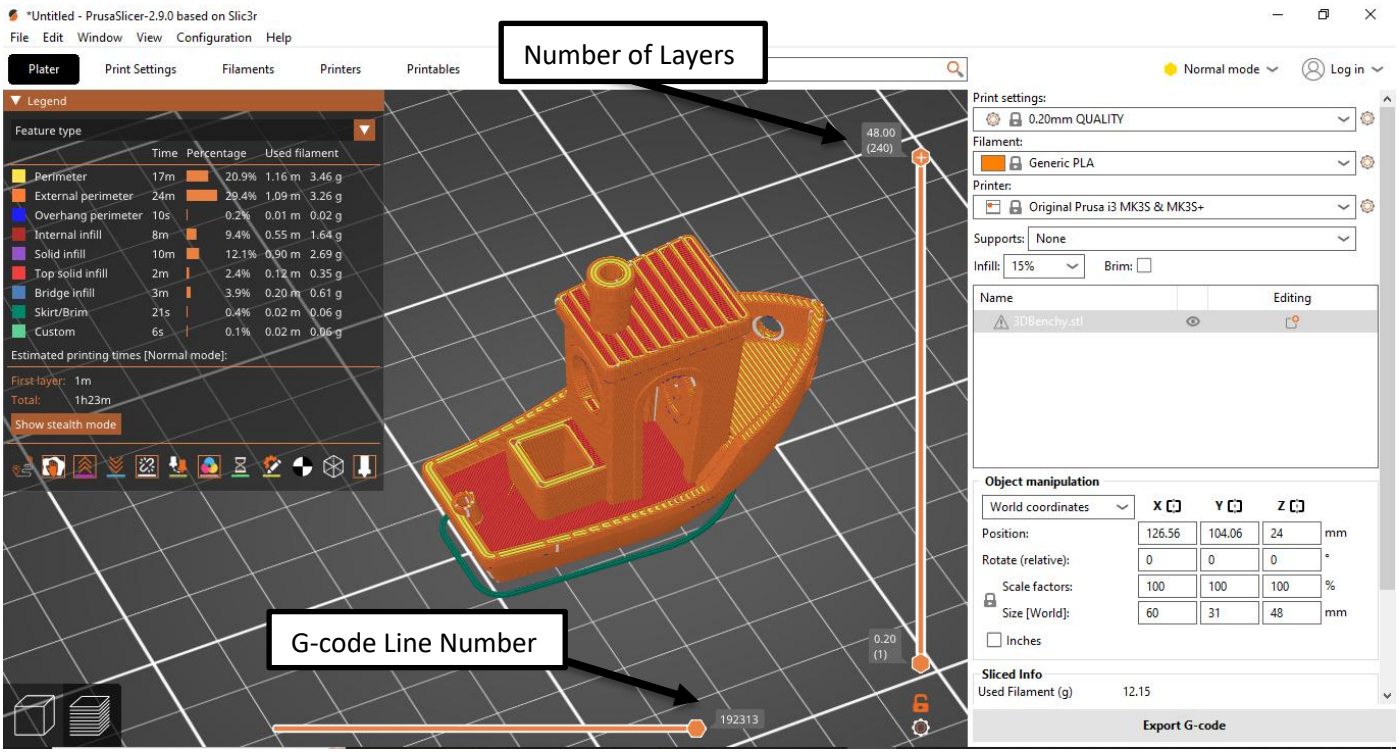
Cooling: if "*Keep fan always on*" is enabled, the nozzle fan will never be disabled and will be kept running at least at its minimum speed. This option is useful for PLA printing, but harmful for ABS printing. If "*Enable auto cooling"* is enabled, fan speed will be automatically adjusted according to layer printing time.

Retraction: Backwards movement of an extruder motor to reduce the amount of material stringing or oozing. Retraction length is the distance that the material is retracted from the nozzle tip, when a retraction is made. Retraction speed is the speed at which the material is pulled out of the nozzle and pushed back in, when making a retraction. These two values can be found by running calibration test for the filament that is going to be used.

Previewing G-code before print

After parameter settings, it's a good practice to always preview the G-code before printing it. When you click the "Slice now" button, you'll be automatically switched to the preview mode. Alternatively, you can use the switch between 3D view and preview mode in the lower-left corner. Using the slider on the right, you can adjust the preview and inspect individual layers. Check the overall shape of the model and make sure it matches the 3D object. Watch out for parts, that would start printing in mid-air, if you notice any, it's necessary to turn on support generation. If everything looks alright, you can proceed and select Export G-code.





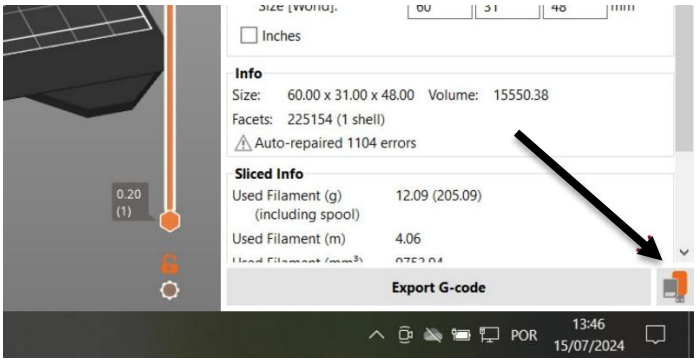
Exporting the G-code

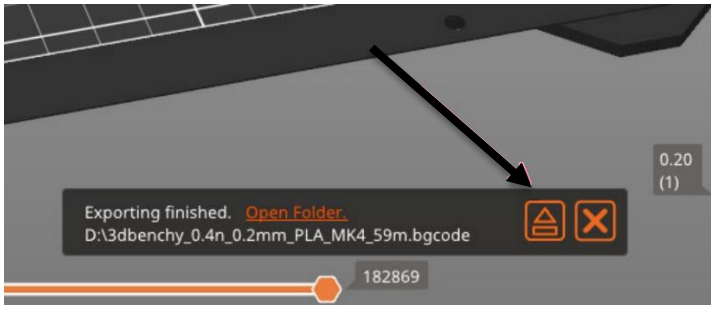
PrusaSlicer automatically detects if you insert an SD card, USB flash drive, or other removable media.

When detected, a new Export to SD/USB drive button will appear next to the Export G-code button (bottom right corner). Clicking it will open the file save dialog with the path initialized to the SD card or USB flash drive.

If multiple SD cards or USB flash drives are detected, the file dialog is opened at the last visited removable media.

Once a G-code is exported to a removable media, an Eject button is shown on the bottom right of the plater, which will safely disconnect your device.





3.2.2 Information sheet: Printing with support using slicer

Bridges and Overhangs

In 3D printing, overhangs are unsupported sections that slope downward from a single point. An overhang occurs when a layer extends beyond the layer below it. Without support from below, gravity can cause the material to droop or deform, leading to poor surface quality, sagging, or complete print failure. A widely used guideline is that overhangs at angles of 45◦ or less can generally be printed without issues. Angles steeper than 45◦ may require support material to print correctly, though the exact angle can depend on factors like material, temperature, and cooling.
It is imperative to know that if the vertical is considered zero degree, then overhangs at angles of 45◦ or less is better. And if the vertical is considered 90 degree, then overhangs at angles of 45◦ or more is better. Both essentially mean the same thing.

On the other hand, a bridge is a horizontal section of filament printed between two anchor points, often at a 90° angle, like the crossbar of an "H". The challenge of printing bridges is that the filament can sag, droop, or break because there is no support beneath it. Bridges are often printable for short distances (e.g., around 10mm) without supports. To improve print quality, increase cooling, slow down print speed, and consider using support structures for more extreme angles.

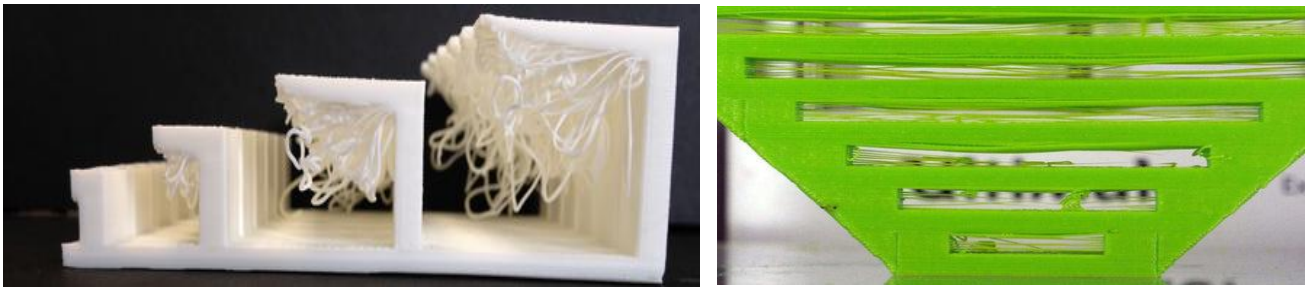


Figure 3.13: Overhangs (Left) & Bridges (Right)

Printing with support

If you print an object with a gradient lower than 45° (considering vertical = 90 degree), the resulting overhang would be preventing the object to be printed correctly. PrusaSlicer allows you to print such objects with support. Support material is an extra structure printed as scaffolding for the object. one can remove the support material after the printing is finished.

Choose the Print Settings tab (1) and click the Support Material option (2) in the left column. First, you have to check the Generate support material box (3). Next item – Overhang threshold (4) lets you set the minimal angle for the printing of the support material. Setting this item to zero lets the printer detect problematic parts automatically and print supports where needed.

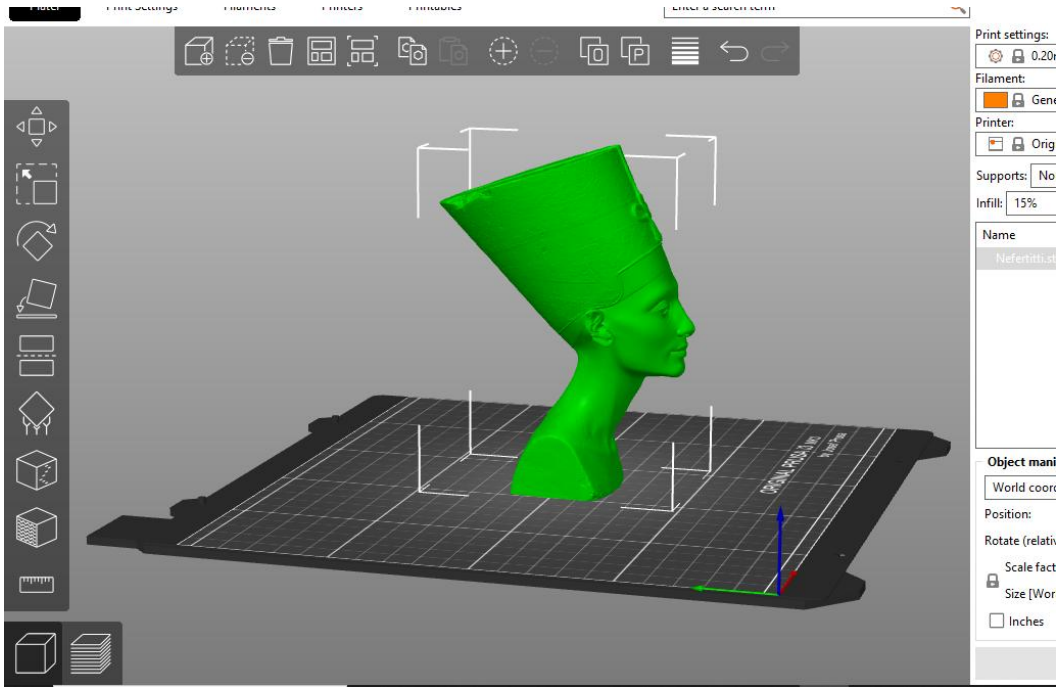
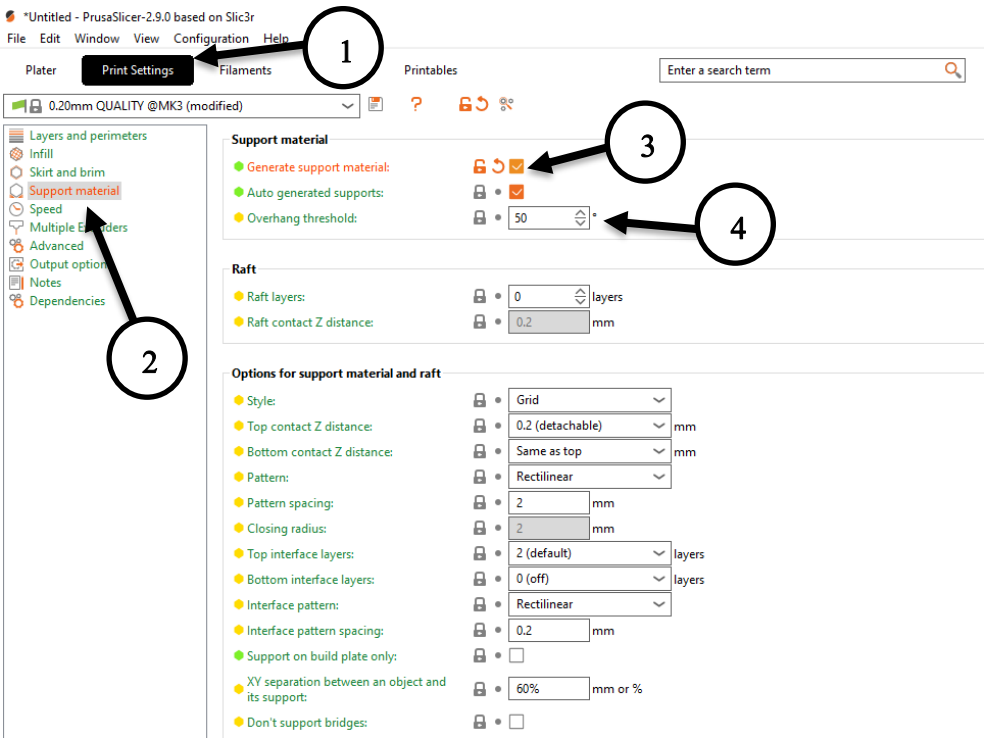
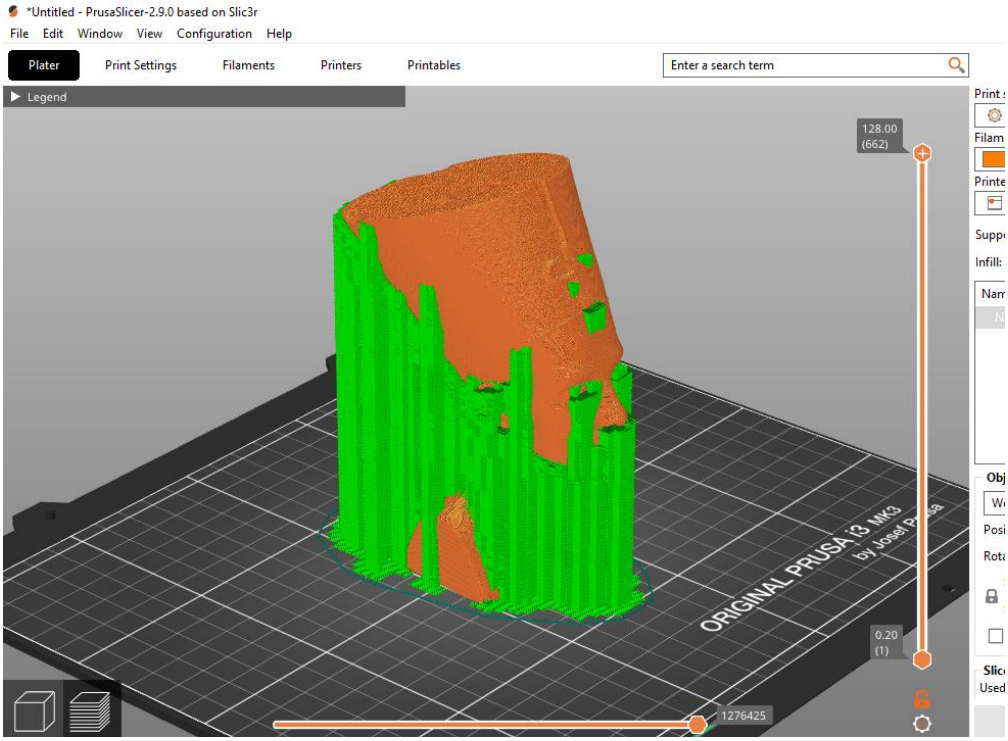


Figure 3:14: Model with overhangs



After applying the above settings, the most basic support structure can be found.



For more advanced support settings other parameters are required. Other support parameters are –

Generate support material

Supports are disabled unless a model has an overwrite for this setting

Auto-generated supports

When off, the overhang detection is off. Only support enforcers generate supports.

Overhang threshold

The Overhang threshold value represents the most horizontal slope (measured from the horizontal plane) that you can print without support material (90=vertical). Changing this value is a quick way to adjust the amount of generated supports. The lower the value, the less support will be generated.

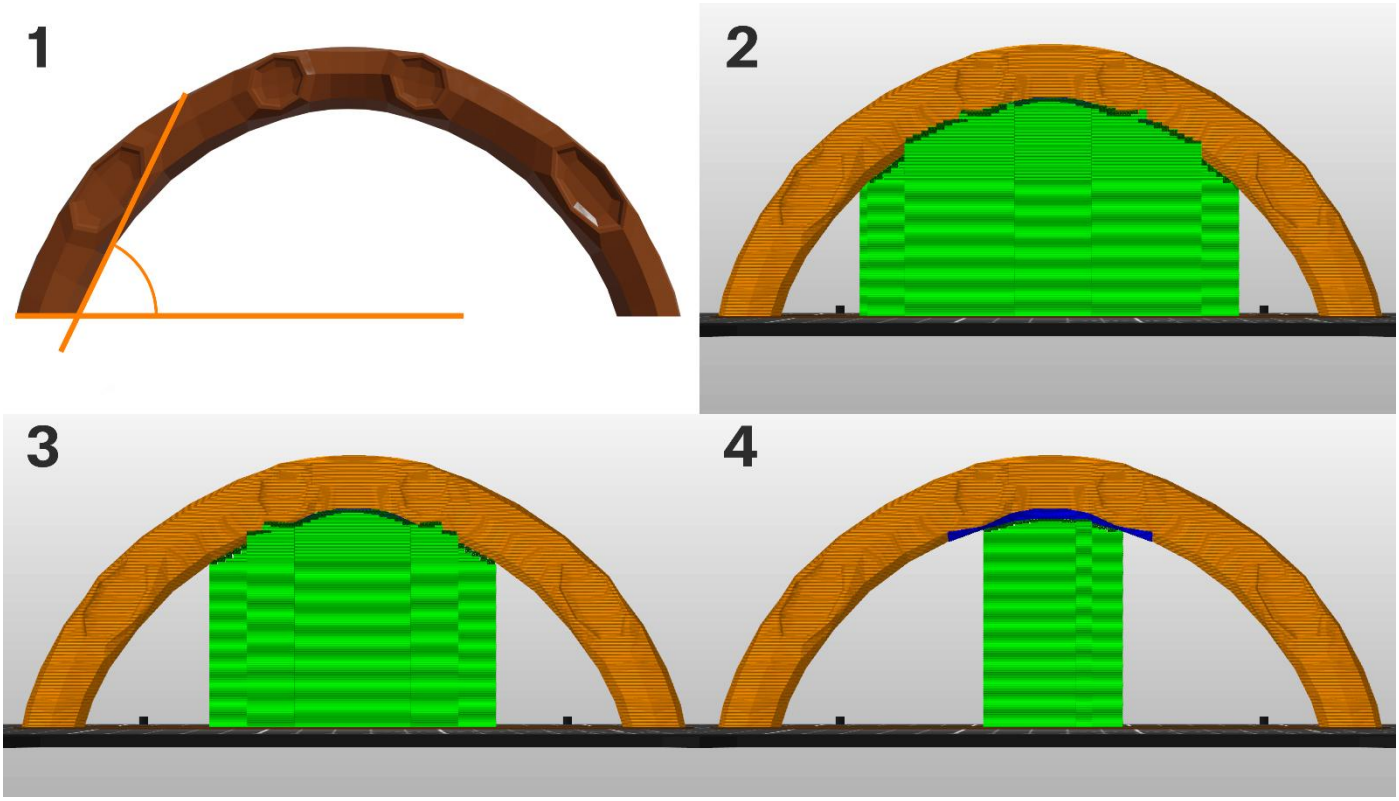


Figure 3.15: (1) Overhang threshold value and how it's measured, (2) Overhang threshold 55, (3) Overhang threshold 35 & (4) Overhang threshold 15

Supports everywhere

Supports can grow from both the print bed and any model surface when necessary.

This ensures that every overhang and part starting mid-air will get supported. However, small imperfections may appear on the surface where supports started growing from.

Supports from build plate only

Creates only supports that have their base at the print bed.

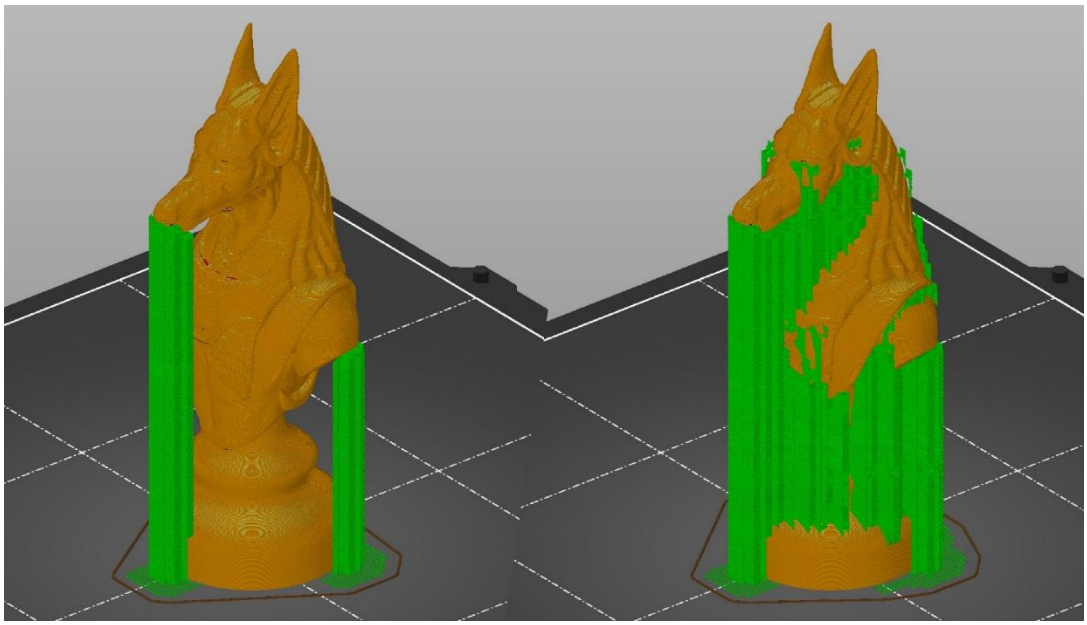
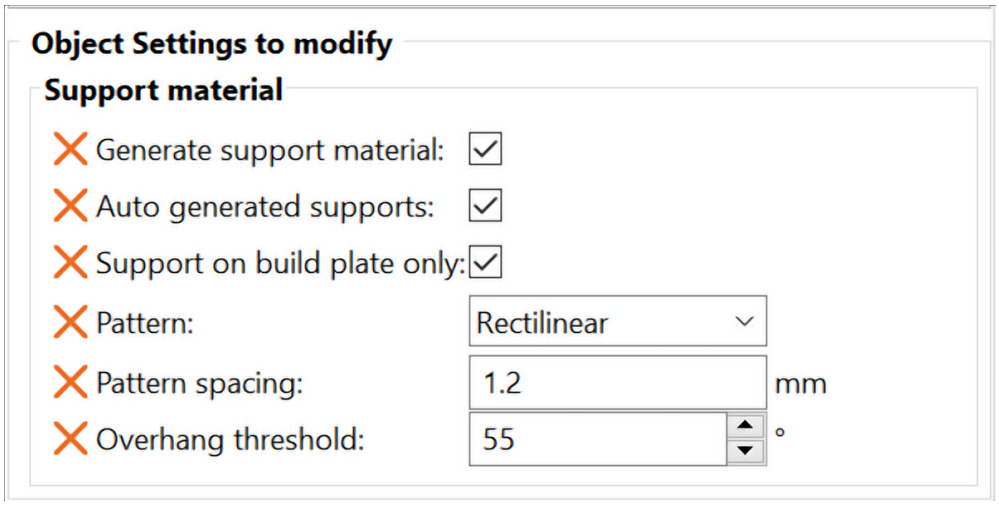


Figure 3.16 Supports from build plate only (left), supports everywhere (right)

Supports on/off for individual models

It's possible to print multiple objects at once each with its own support settings. By default, all models are affected by the global settings. You can overwrite them by **right-clicking** on a model in the 3D view and selecting **Support material**. Alternatively, you can right-click on the editing icon in the Object list.

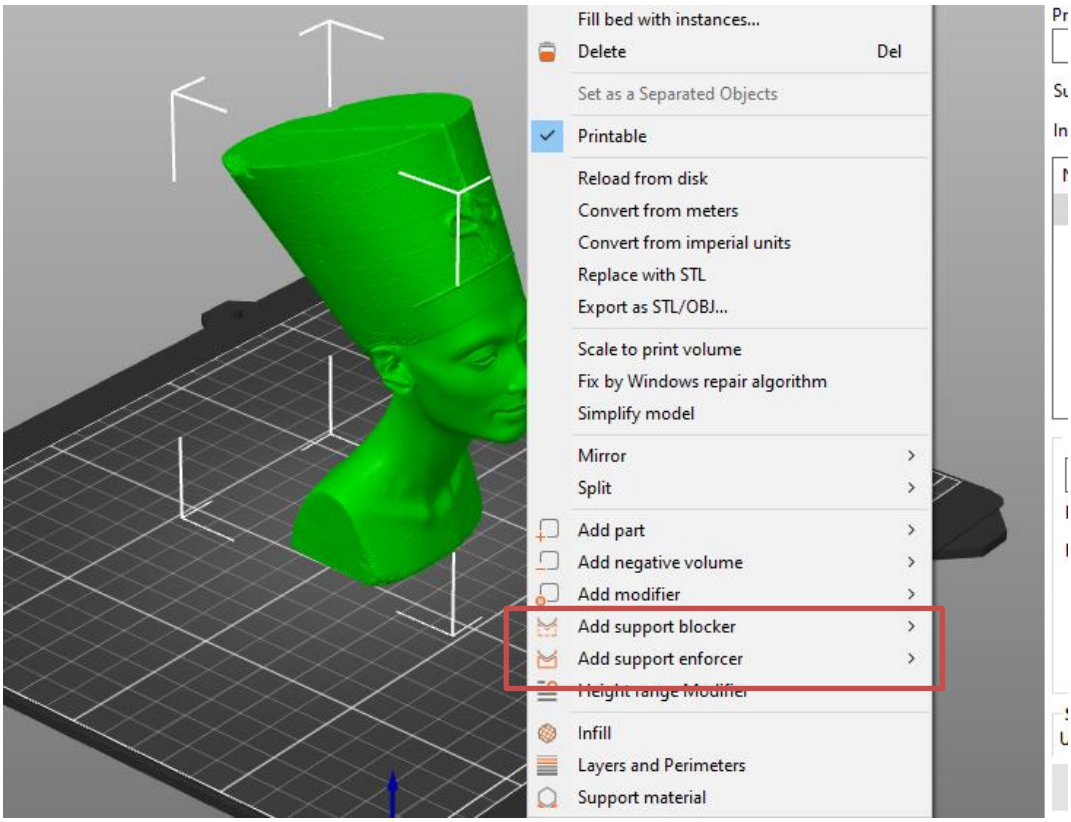
A new window will appear in the right panel called **Object settings to modify - Support material**. Changing these settings will affect only the relevant object and its instances. You can remove the settings overwrite by clicking on the red cross icon. When you do so, the global settings will once again be applied.



Support enforcer/blocker

The automatic supports can sometimes appear in hard-to-reach places, resulting in a tedious removal of support structures. If you think such a part of the model could be printed without supports, but neither the **Overhang threshold** setting nor selecting **Supports on built plate only** helped, you can adjust supports manually.

Right-click on a model and choose either a Support blocker or Support enforcer.



Support blockers and enforcers can be manipulated with the familiar Move, Rotate, and Scale tools. You can also quickly duplicate them with copy and paste.

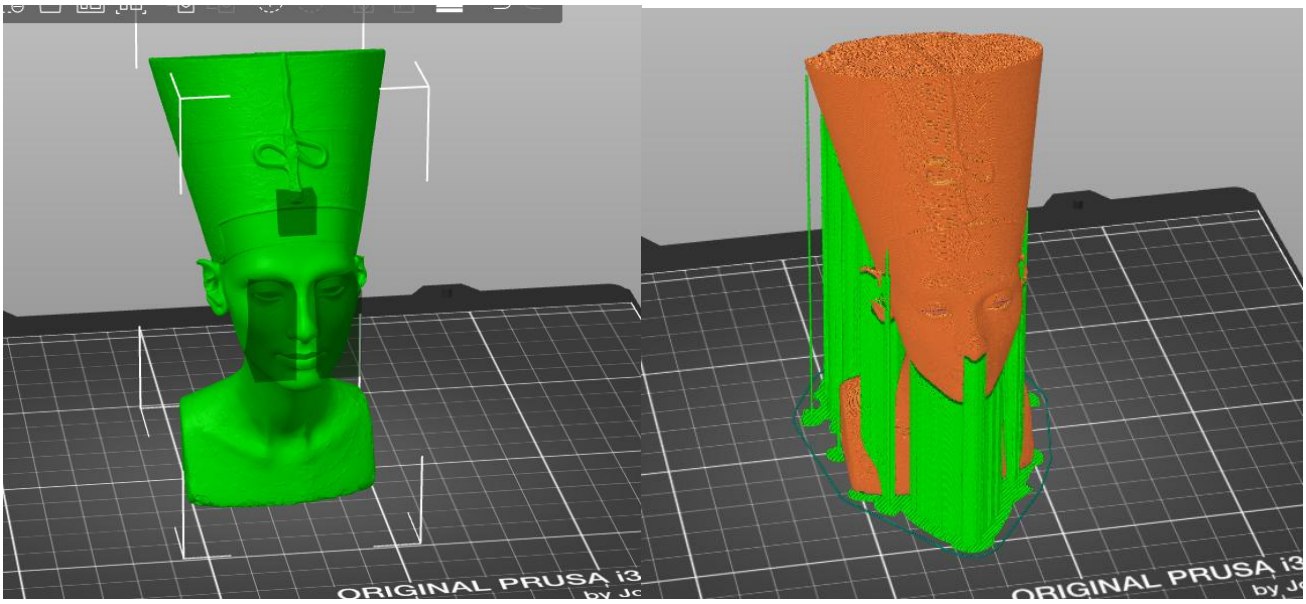


Figure 3.17: Support generation after support blockers applied

Support blocker

Support blockers are useful when you're generally happy with the automatic supports and you simply want to remove some of them. Focus on placing blockers where the supports meet the object, the rest of the support structure below will be removed automatically.

Support enforcers

Support enforcers are great in situations, where only a few parts of the object need to be supported, but the automatic algorithm places supports all over the model. In this situation, don't forget to switch the support mode to "For support enforcers only".

For support enforcers only

Automatic support generation is enabled only for areas marked with a support enforcer. This option is essentially a shortcut that turns **Print settings - Support material - Generate support material** on, but **Print settings - Support material - Auto-generated supports** off.

Raft layers

The object will be raised by this number of layers and support material will be generated under it. When the print finishes, you can grab the raft and peel it away from the part.

Raft contact Z distance

The vertical distance between the raft and the object. Ignored for soluble interfaces.

Style

Style and shape of supports. The default **Grid** supports have a very well-defined minimum area, making them very stable. However, they tend to leak out of regions they support along the walls of the object. The **Snug** supports maintain the shape of the overhang, so they do not leak to the walls. The trade-off is the possibility of lower stability of tall and thin support pillars.

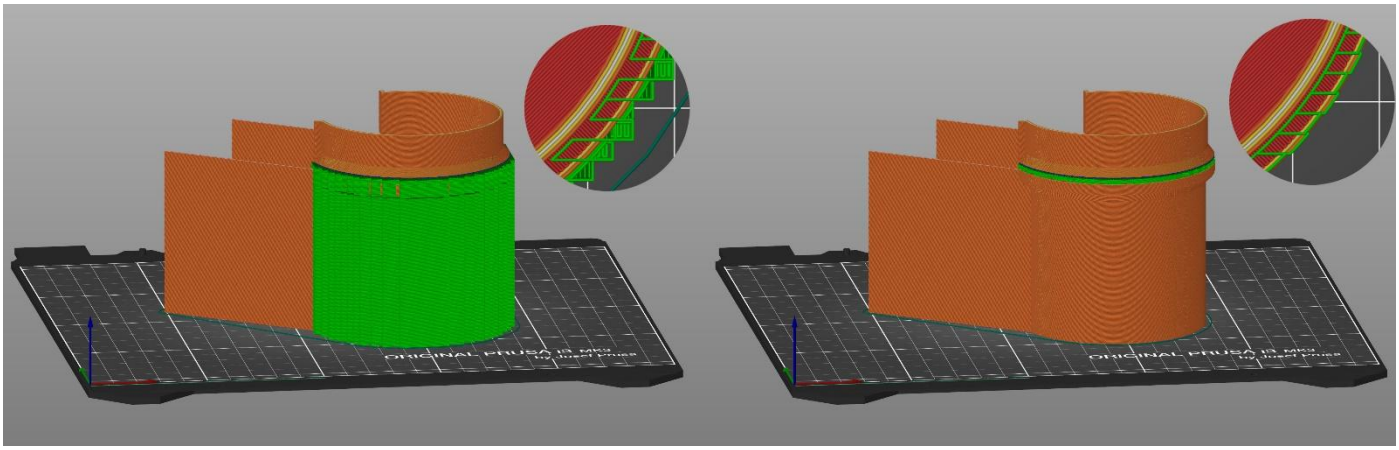


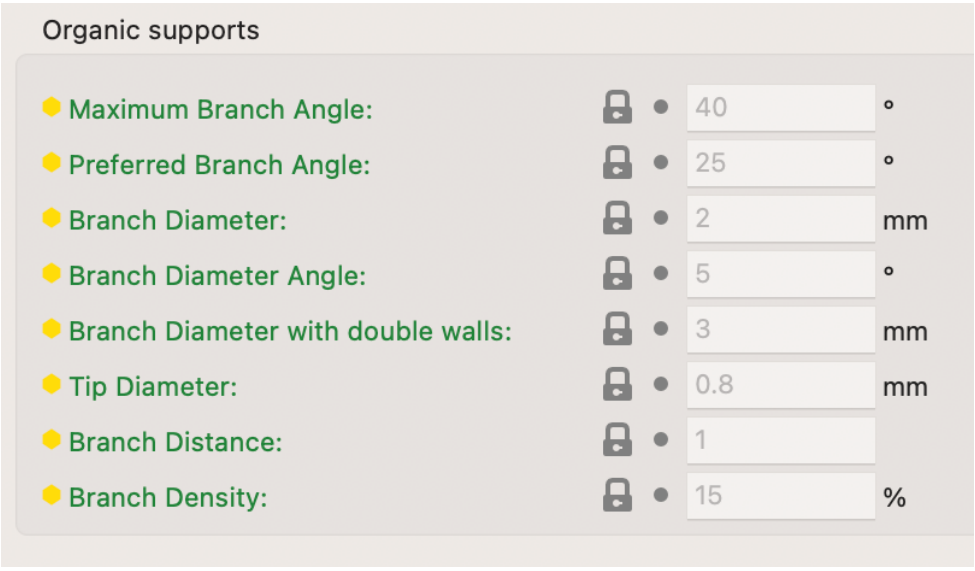
Figure 3.18 Grid supports (left), Snug supports (right)

Organic supports offer a completely different approach to supporting overhanging areas. Branches of support structures grow from the sides and automatically get denser beneath parts that need the support. The Organic supports are easily removable, do not scar the surface, and are fast and cheap to print.



figure 3.19 Grid supports (left), Organic supports (right)

Apart from the settings common for all support types, there are additional settings specific to the Organic supports (or, tree supports).



Organic supports settings

Maximum Branch Angle

The maximum angle of the branches, when the branches have to avoid the model. Use a lower angle to make them more vertical and more stable. Use a higher angle to be able to have more reach.

Preferred Branch Angle

The preferred angle of the branches, when they do not have to avoid the model. Use a lower angle to make them more vertical and more stable. Use a higher angle for branches to merge faster.

Branch Diameter

The diameter of the thinnest branches of organic supports. Thicker branches are sturdier. Branches towards the base will be wider than this.

Tip Diameter

Branch tip diameter for organic supports.

Branch Distance

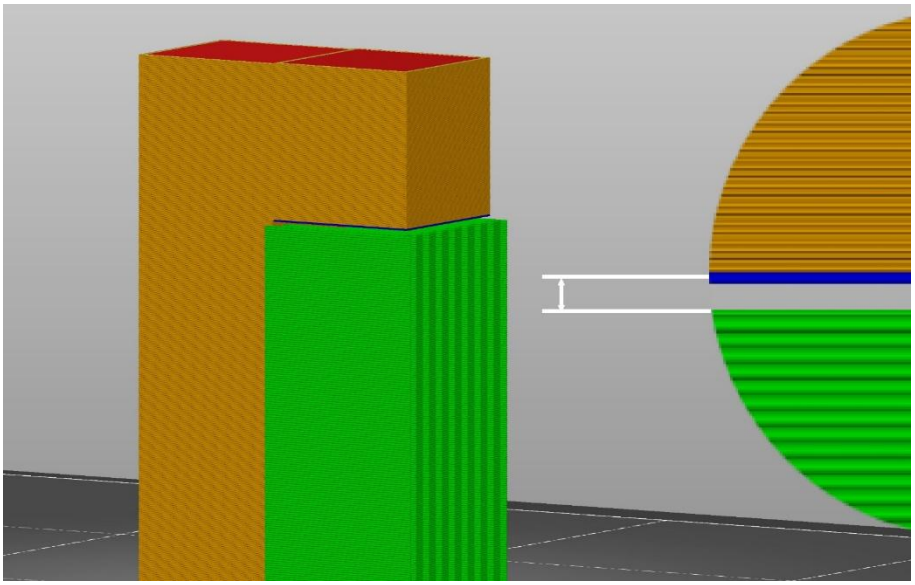
How far apart the branches need to be when they touch the model. Making this distance small will cause the tree support to touch the model at more points, causing better overhangs printing but making the supports harder to remove.

Branch Density

Adjusts the density of the support structure used to generate the tips of the branches. A higher value results in better overhangs but the supports are harder to remove. We recommend keeping the top support interfaces enabled instead of using a high branch density value if dense interfaces are needed.

Top contact Z distance

The vertical distance between the object and the support material interface. Setting this to 0 will prevent PrusaSlicer from using bridge flow and speed for the first object layer. Values between 50-75% of the layer height seem to work well.



Bottom contact Z distance

Similar to Top contact Z distance, but affecting the gap at the bottom of the support tower (when supports start on top of a model and not on the build plate).

Pattern

Rectilinear - default, usually the easiest to remove

Rectilinear grid - sturdier and less prone to breaking during printing compared to rectilinear infill, harder to remove

Honeycomb - sturdy, somewhat difficult to remove

Pattern spacing

Spacing between support material lines. Increase to make support removal easier. Support interface layers bridge over support material lines, if you increase this value too much, these bridges will start to sag and have an imperfect connection with the model.

The pattern spacing also sets the internal resolution of PrusaSlicer's supports grid. If you're using Paint-on supports or Support enforcers and you're unable to force supports exactly where you want them to be, try lowering the pattern spacing to get more resolution.

Pattern angle

Rotates infill pattern angle on the horizontal plane.

Closing radius

For Snug supports only. Gaps smaller than the closing radius will be filled in.

Top interface layers

Interface layers form a more uniform surface between the object and the support material. These are preferably set to a much denser pattern than the non-interface layers so that you can both maximize the surface area of the support in contact with the model to prevent sagging while minimizing print time and material used with a bigger non-interface pattern spacing.

Bottom interface layers

Similar as Top interface layers, but affecting the number of interface layers at the bottom of the support tower (when supports start on top of a model and not on the build plate).

Interface pattern spacing

Spacing between interface lines. Set to 0 to get a solid interface.

Interface loops

When enabled, the top layer of supports is covered with loops - continuous polylines.

XY separation between an object and its support

Sets the width of a gap between the model and supports. Bigger XY separation decreases the contact area of supports but makes them easier to remove and less prone to fusing to the model. If expressed as a percentage (for example 150%) it will be calculated over the external perimeter width.

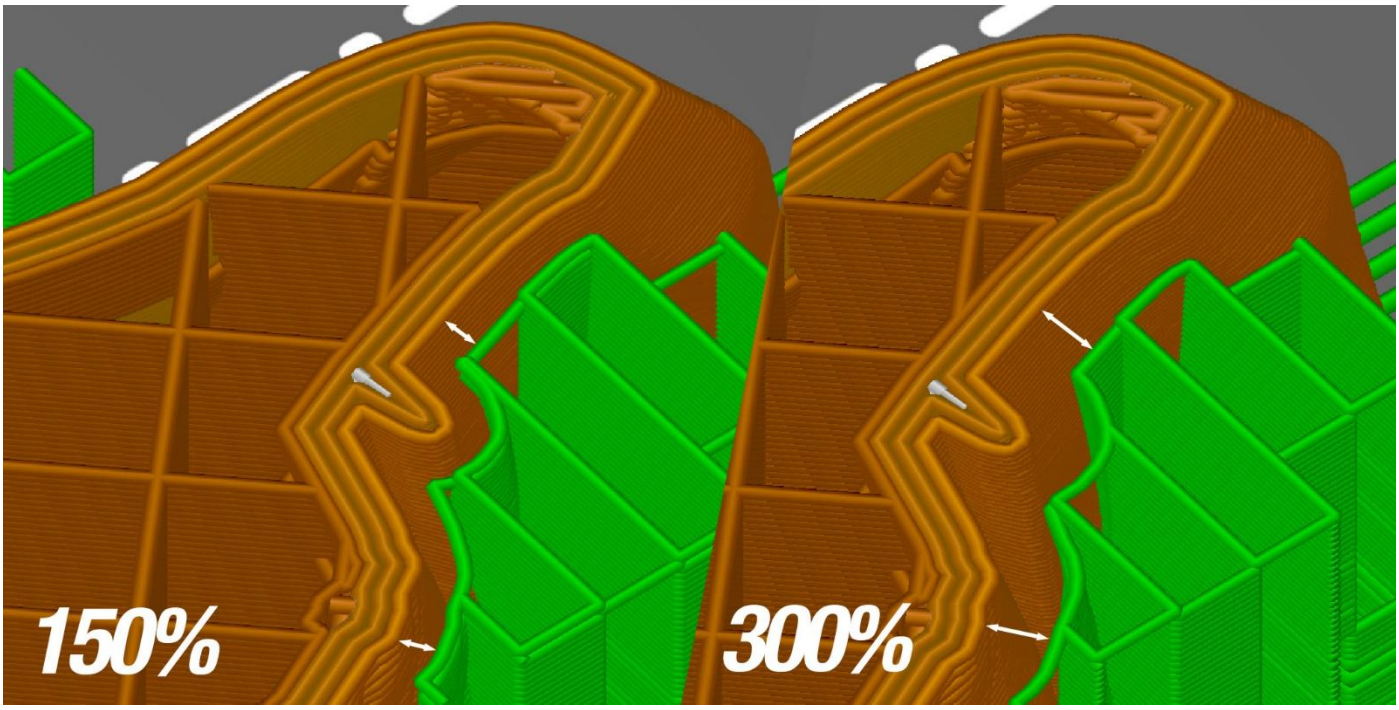


Figure 3.20: XY separation settings

3.2.3 Information sheet: Per model settings

With PrusaSlicer, you can print multiple models at once, each with slightly different settings. For example, with different infill or layer height.

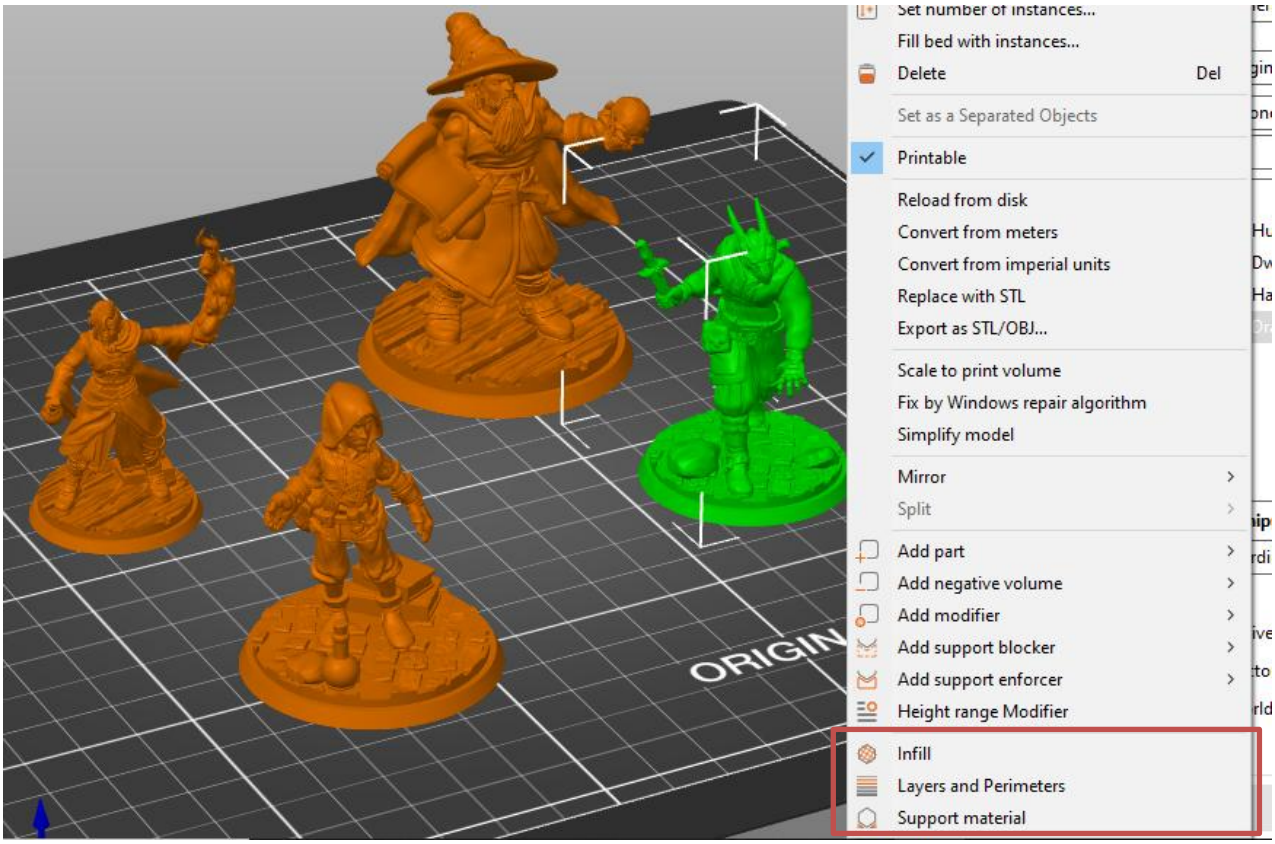
By Right-click on a model, following parameters can be picked for model-wise setting –

- **Infill**
- **Layers and perimeters**

- **Support material**

In the **Expert** mode, an additional **Add settings** option is available that lets you adjust settings such as speed, extrusion width, infill/perimeter overlap and more.

Alternatively, you can right-click on the editing icon in the Object list. A new window will appear in the right panel called **Object settings to modify**. Changing these settings will affect **only the relevant object and its instances**. You can remove the settings overwrite by clicking on the red cross icon. When you do so, the global settings will once again be applied.



3.2.4 Information sheet: G-Code & M-Code

Before going into any advanced printing technique, it is imperative to have the knowledge of g-code and m-code. This coding system is to give instructions to the printer for printing and other non-printing tasks. These instructions are interpreted by a software called firmware. Popular examples of firmware include Marlin, Klipper, and Repetier-Firmware, which vary in features, performance, and compatibility with different boards and processors. Original Prusa i3 MK3S+ printer's firmware is Marlin 1 (Legacy). The full set of g-code and m-code compatible with this firmware can be found in Marlin's website (https://marlinfw.org/meta/gcode/).

G-Code

G-Code means Geometric Code/ General Code/ Preparatory Functions. G-Code is used for Print-head

positioning, Print-head movement, expressing geometric entities etc. Simply, G-Code tells Print-head where to move, how fast to move and what path to take.

Below are some examples of most used G-Codes.

Code	Meaning	Example
G0	Linear Movements of Print-head with no extrusion (Used for initial movements and travel movements)	*See in the example program below*
G1	Linear Movements of Print-head with extrusion (Used for Printing)	*See in the example program below*
G4	Pause the print and waits for a given period of time	G4 S500; The meaning of the above command is, the G4 code will pause the printing for 500 seconds. After 500 seconds, the printing will start automatically.
G90	Absolute Positioning	*See in the example program below*
G91	Relative Positioning	*See in the example program below*
G28	Auto Home (X=0, Y=0, Z=0) Moves to the Home (X, Y & Z Coordinates specified by Manufacturer) from wherever the Print-head is currently at.	*See in the example program below*
G29	Performs Automatic Bed Levelling	*See in the example program below*
G92	Sets the current position to the values specified	G92 X0 Y0 Z0; The meaning of the above command is, the G92 code will set the nozzle's current XYZ position as X=0, Y=0, Z=0.

M-Code

M-Code is used to perform miscellaneous functions related to printer, except what G-Code does. Below are some examples of most used M-Codes.

Code	Meaning	Example
M104	Starts heating the hot end, but allows to run other commands afterwards simultaneously	*See in the example program below*
M109	Starts heating the hot end and wait until the desired temperature is reached	
M140	Starts heating bed, but allows to run other commands afterwards simultaneously	
M190	Starts heating bed and wait until desired temperature is reached	
M105	Displays the updated temperature until reaches the desired temperature	
M106	Sets the fan speed	

M107	Turns off the fan	

Program is a sequence of blocks describing the motions of a printer to print an object.

Block is a complete command given to the printer. A Block always starts with a G-code/ M-code commands followed by some **alphanumeric statements**. A Block must be ended with a **semicolon (;)**.

Alphanumeric statement consists of different letters and values. Every letter has a value after it. The meaning of the value depends on the meaning of letter.

Below are some examples of letters used in a program.

X → Positions along X axis

Y → Positions along Y axis

Z → Positions along Z axis

F → Feed rate (Print speed, Travel speed and all other kinds of speed in mm/min).
In Slicer, the unit of F is usually mm/s unless otherwise provided or mentioned.

E → Material to be extruded (mm)

S → Temperature value (°C), Fan Speed (0 means 0% speed and 255 means 100% speed)

T →Travel Speed / Acceleration

P → Print Speed / Acceleration

R → Retract Speed / Acceleration

Print G-Code Viewer

1. https://nraynaud.github.io/webgcode/
2. https://zupfe.velor.ca/

Below is a typical example of a program.

Program

Block -1	→	M140 S60;	Printer starts to heat the bed until 60°C.
Block -2	→	M105;	Printer display shows the real time update of the bed temperature.
Block -3	→	M190 S60;	Printer continues to heat the bed and does not go to the Block 4 until the bed heat is 60°C
Block -4	→	M104 S200;	Printer starts to heat the nozzle until 200°C.
Block -5	→	M105;	Printer display shows the real time update of the nozzle temperature.
Block -6	→	M109 S200;	Printer continues to heat the nozzle and does not go to the Block 7 until the nozzle heat is 200°C.
Block -7	→	M82;	Printer sets the absolute extrusion mode.
Block -8	→	G28;	Printer moves the Print-head to X=0, then Y=0 and then Z=0 position.
Block -9	→	G92 E0;	Printer sets the current position of the extruder motor as zero position.
Block -10	→	G90;	Printer sets the positioning system for X, Y & Z Axes as Absolute Positioning System.
Block -11	→	M106 S255;	Printer starts the fan with 100% speed.
Block -12	→	G0 Z2.0 F3000;	Printer moves the Print-head 2.0mm along the positive Z Axis with respect to Z=0. The speed of the movement is 3000 mm/min. There will be no extrusion.
Block -13	→	G0 X10.1 Y20 Z0.28 F5000.0;	Printer moves the Print-head 10.1mm along the Positive X Axis, 20.0mm along the Positive Y Axis, and 0.28mm along the Positive Z Axis with respect to X=0, Y=0, & Z=0. The speed of the movement is 5000 mm/min. There will be no extrusion.
Block -14	→	G1 X10.1 Y200.0 Z0.28 F1500.0 E15;	Printer moves the Print-head 10.1mm along the Positive X Axis, 200.0mm along the Positive Y Axis, and 0.28mm along the Positive Z Axis with respect to X=0, Y=0, & Z=0. The speed of the movement is 1500 mm/min. There will be extrusion from the position of Block-13 up to the position of Block-14. The extrusion amount or the length of filament used is 15.0mm.
Block -15	→	G0 X10.4 Y200.0 Z0.28 F5000.0;	Printer moves the Print-head 10.4mm along the Positive X Axis, 200.0mm along the Positive Y Axis, and 0.28mm along the Positive Z Axis with respect to X=0, Y=0, & Z=0. The speed of the movement is 5000 mm/min. There will be no extrusion.
Block -16	→	G1 X10.4 Y20 Z0.28 F1500.0 E30;	Printer moves the Print-head 10.4mm along the Positive X Axis, 20.0mm along the Positive Y Axis, and 0.28mm along the Positive Z Axis with respect to X=0, Y=0, & Z=0. The speed of the movement is 1500 mm/min. There will be extrusion from the position of Block-15 up to the position of Block-16. The extrusion amount is 30.0 - 15.0 = 15.0mm.
Block -17	→	M106 S0;	Printer turns off the fan.
Block -18	→	M104 S0;	Printer starts the cooling of the nozzle.
Block -19	→	M140 S0;	Printer starts the cooling of the bed.

3.2.5 Information sheet: Calibration test related to the filament

FDM printer calibration involves adjusting key settings to ensure accuracy, consistency, and quality in prints. Essential steps include bed leveling, setting the Z-offset, and calibrating the extruder's steps per millimeter (E-steps). You should also calibrate the flow rate and tune temperature, retraction, and speed for the specific filament you are using.

Here, a step-by-step calibration process discussed. This should be carried out after the printer calibrations that is discussed in learning outcome 3.1 is done.

Baseline Print

Before general calibration or before a significant modification is to be fitted, it is necessary to establish a baseline for comparison with later tests or before modifications. For this, a calibration cube should be downloaded from any 3D printing repository. “XYZ 20mm calibration cube” by iDig3Dprinting from Thingiverse.com is suggested. You can print this with the usual slicer settings at this moment.

Nozzle diameter: 0.4mm
Layer height: 0.2mm
Nozzle temperature / bed temperature: this value depends on filament that is to use
Retraction distance: default setting



Figure 3.21: Calibration cube

If there are no major issues, the next step can be continued. If there is a significant defect, then the printer calibration should be done again. Minor issues will hopefully be resolved with the subsequent tests.

E-steps Calibration

E-steps Calibration is required to determine the correct amount of steps Marlin firmware needs to send to the extruder stepper motor for accurate movement. E-steps calibration should be done when there has been

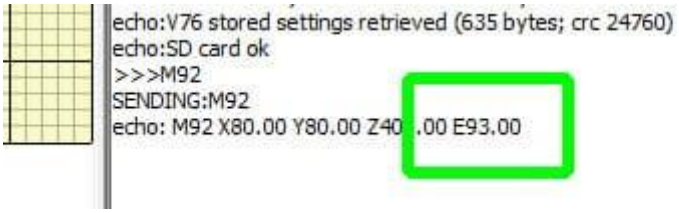
a change to the extruder/hot end. This calibration requires the printer to be directly connected with computer through a terminal software such as Pronterface.

E-steps Calibration can be done by sending simple g-code commands via Pronterface to extrude a set amount of filament, then measuring how much filament actually went through the system.

Firstly, we need to know the existing E-steps value. To find this, enter: M92

If you only receive an *ok* message from this, alternatively you can look for the **M92** line after entering: M503

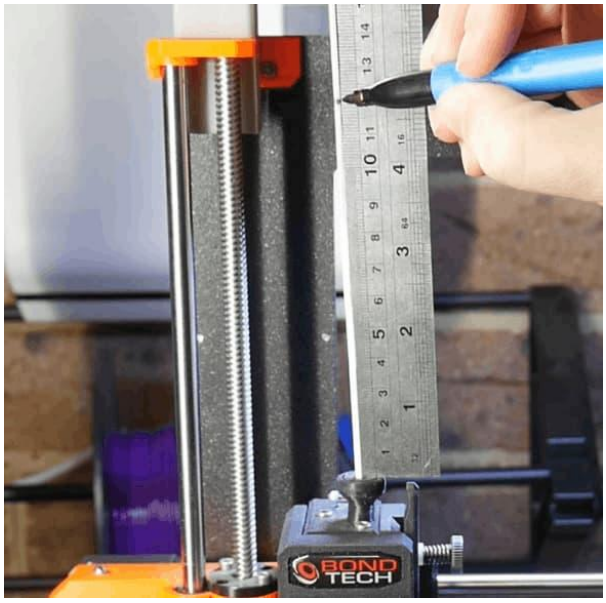
M92 is used to report or set the steps per mm for each axis. M92 by itself will report the current parameters. We want to make note of the number after **E**, in the example below, **93.00**:



Manually move the nozzle high enough above the bed to provide adequate clearance to extrude filament. Now heat up your hot end to whatever temperature you usually print with plus 10 degrees. Once the temperature is stable, enter: G91

G91 puts the printer in relative movement mode. This means requesting 100mm of filament adds 100mm to the current position, instead of moving to the specific position of 100mm.

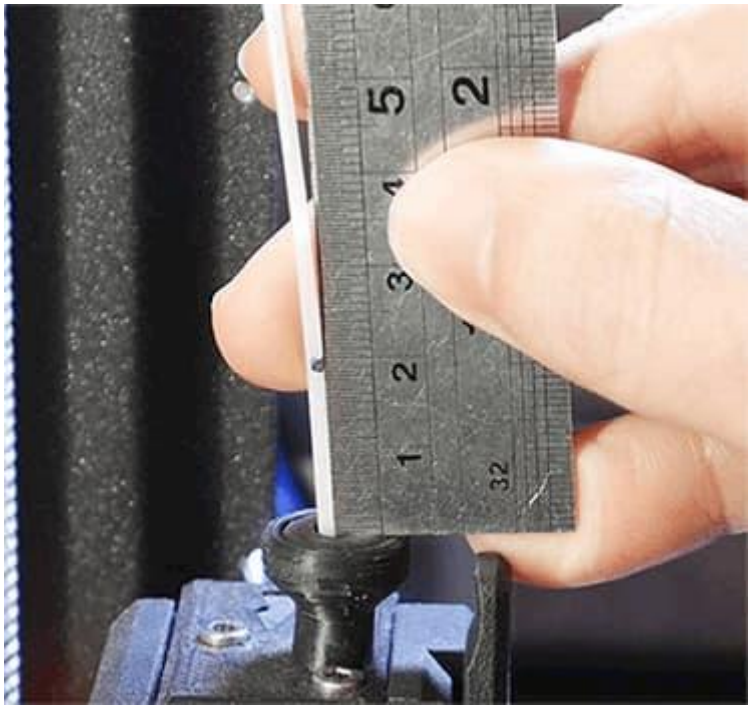
Now we take a permanent marker and put a mark 120mm from the entry to the extruder:



Next, we enter: G1 E100 F50

G1 sends a move command to the printer, in this case asking the extruder to advance 100mm at a speed of 50mm/min.

The filament will then very slowly go through the extruder (and hot end). Once the extrusion finishes, we measure the distance between the mark and the entry to the extruder.



Ideally, 20mm remains, which means exactly 100mm was extruded. If your distance is anything other than this, you need to calculate the correct E-steps.

Say, you have 19mm remains instead of 20mm, that means your extruded amount is 101.

the new E-step = (100*93)/101 = 92.08

Once you have determined the correct value, it must be saved to the firmware to take effect on subsequent prints. Although it can be hard coded into the firmware by recompiling Marlin, it is far easier to use g-code to achieve this.

In a terminal, enter: M92 E[your new value]

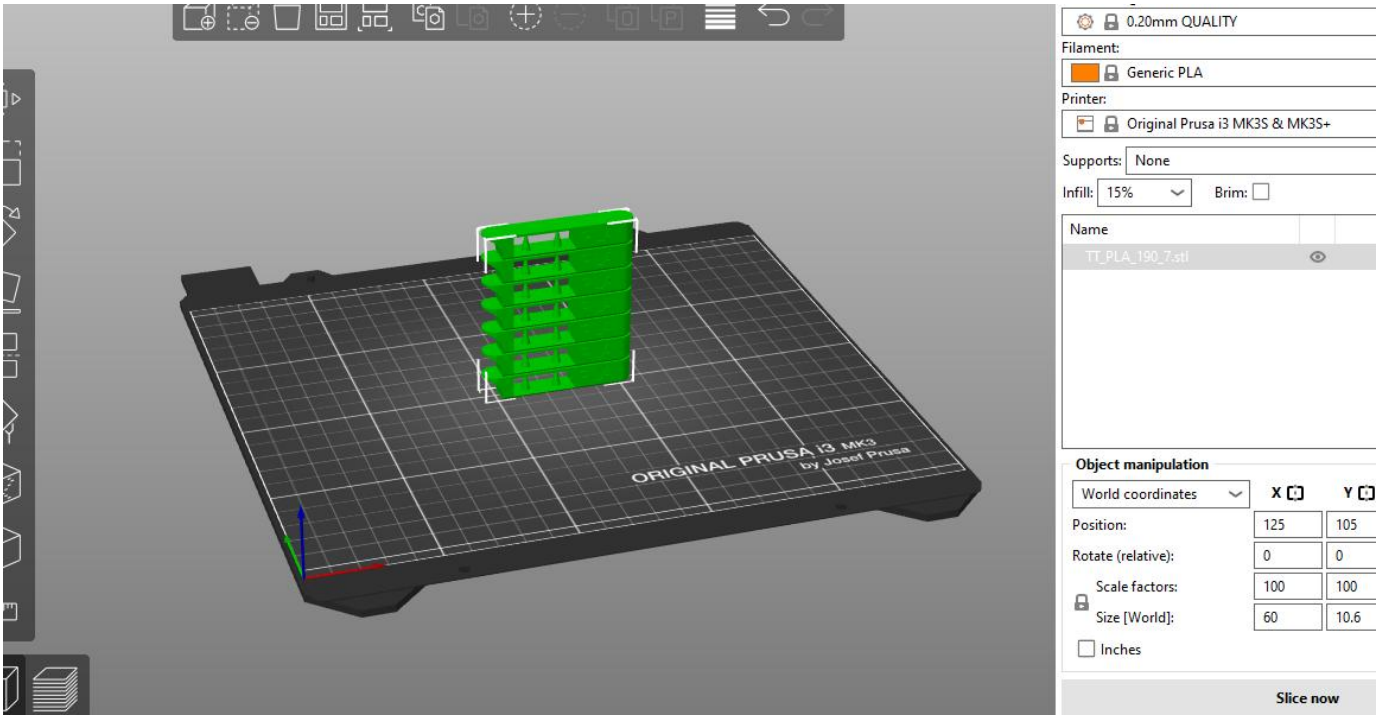
Obviously, you would substitute in your E-steps value after the E.

To save this value to printer's permanent memory or EEPROM (Electrically Erasable Programmable Read-Only Memory), enter this in terminal: M500

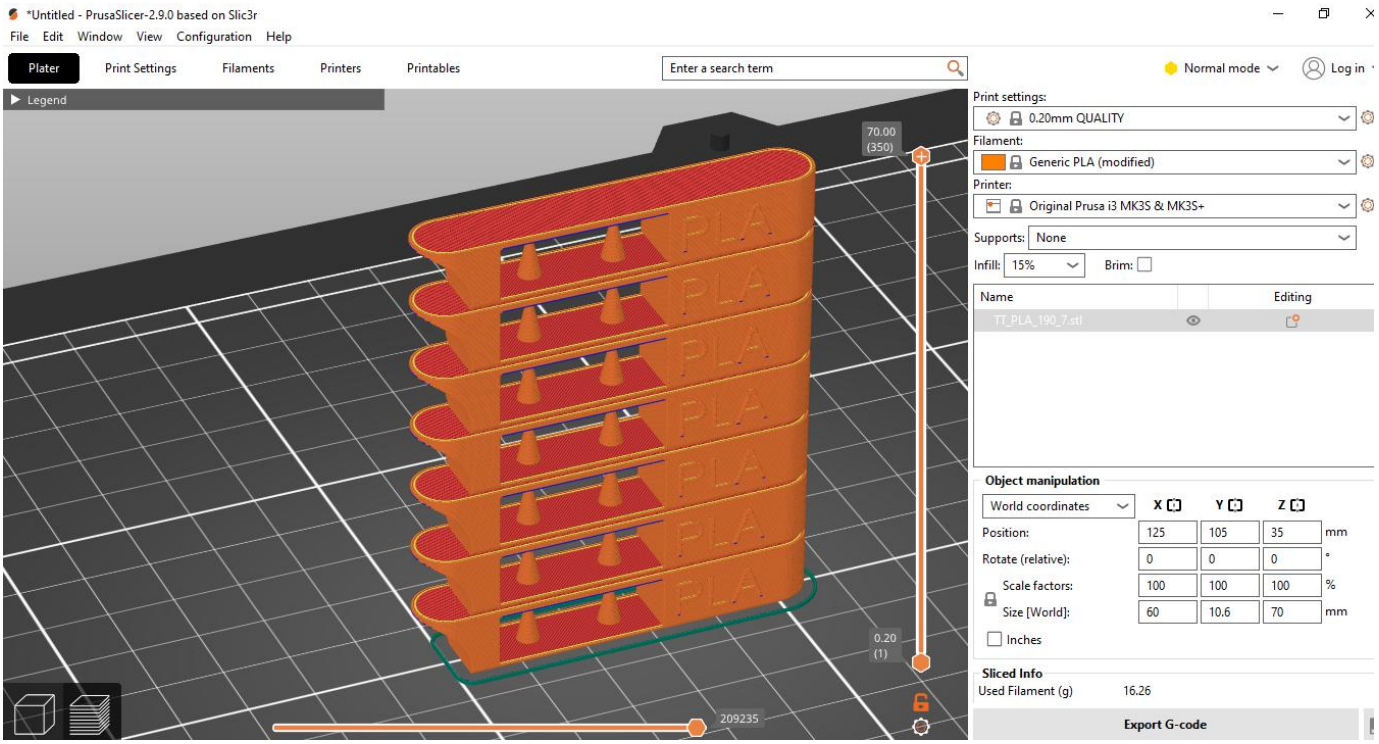
Temperature Test

The goal of this test is to set the ideal printing temperature for the hot end for a given filament. It should be done in case of initial calibration, any time the hot end is changed, or when trying a new type/brand of filament.

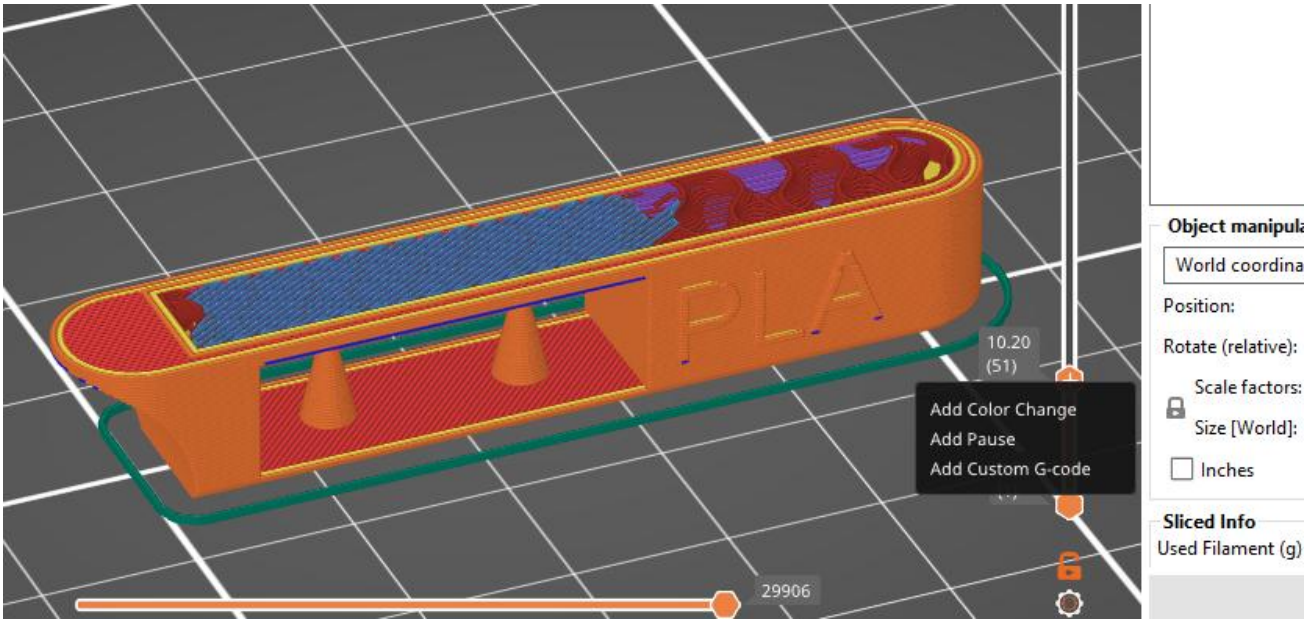
Go to Thingiverse.com, and download 3D object named “temperature tower”. Import it in PruseSlicer.



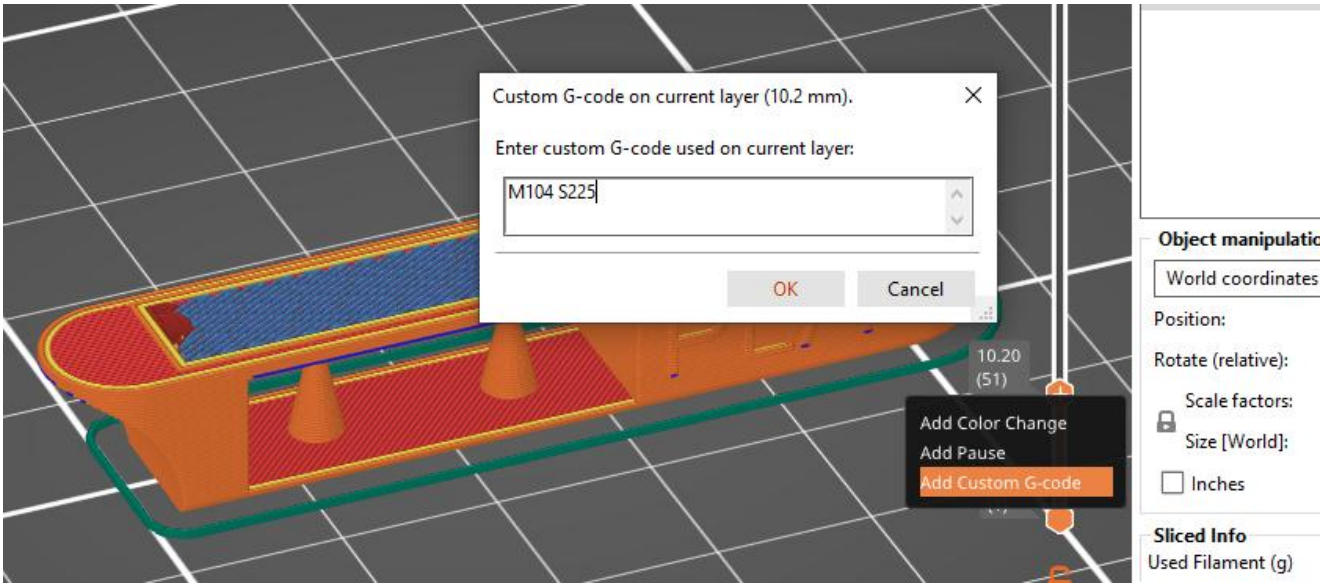
Go to Filaments > Filament > Temperature > Nozzle. Set the first and other layers temperature to some higher value such as for PLA filament it can be set to 230. Then slice the model and go to preview window.



Now, drag down the slider on the right side of the preview to the top of first segment. And right click on the plus sign of the slider. Three options will be appeared. Choose “Add Custom G-code” option.



A dialog box will be appeared. there enter: M104 S225. This will set the nozzle temperature to 225°C for subsequent layers. Do the same thing for every segment (not on the top layer) with 5°C decreasing.



After doing that, export the g-code. Print the model and find the segment with best print quality. Record and use that nozzle temperature value for any future print with that filament.

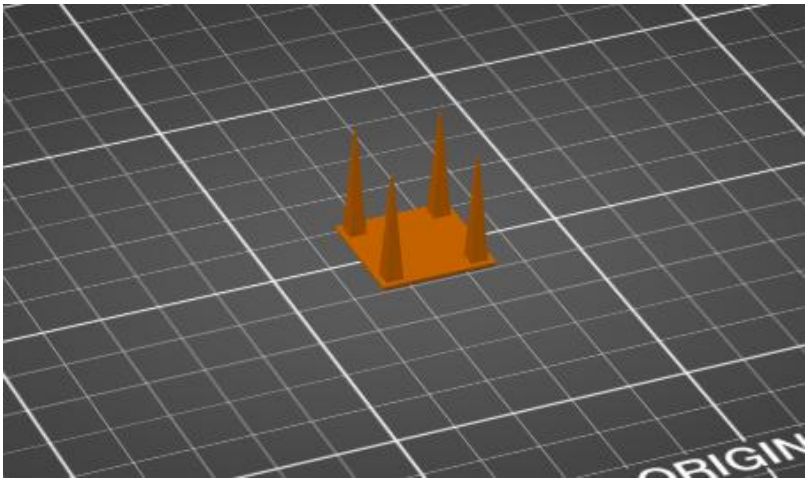


Figure 3.22: Temperature test

Retraction test

FDM works by melting plastic filament and extruding it accurately one layer at a time to build up 3D geometry. By its nature, the plastic will continue to ooze and drip out of the nozzle even when not pushed by the extruder. To combat this, our slicers use retraction, where the filament is withdrawn from the hot end, alleviating pressure and minimizing ooze. When properly tuned, this has the effect of removing stringing, the unwanted oozing of plastic between two points of the model. It should also be done in case of initial calibration, any time the hot end is changed, or when trying a new type/brand of filament. this is one of the toughest and most time-consuming calibration.

Go to Thingiverse.com, and download 3D object named “Retraction test” or, similar. Import it in PruseSlicer.



Go to Filaments > Filament Overrides > Retraction. Check mark the Retraction length and Retraction speed. Print the model with default set values. Observe the printed model whether stringing exists or not. If exists, change the values of these two parameters. Start with lower limit of 2mm for retraction length

and 20mm/s for retraction speed. Try every combination with 0.2mm increment for retraction length and 5mm/s increment for retraction speed, find the best result, record the values and use that values for any future print for that specific filament.

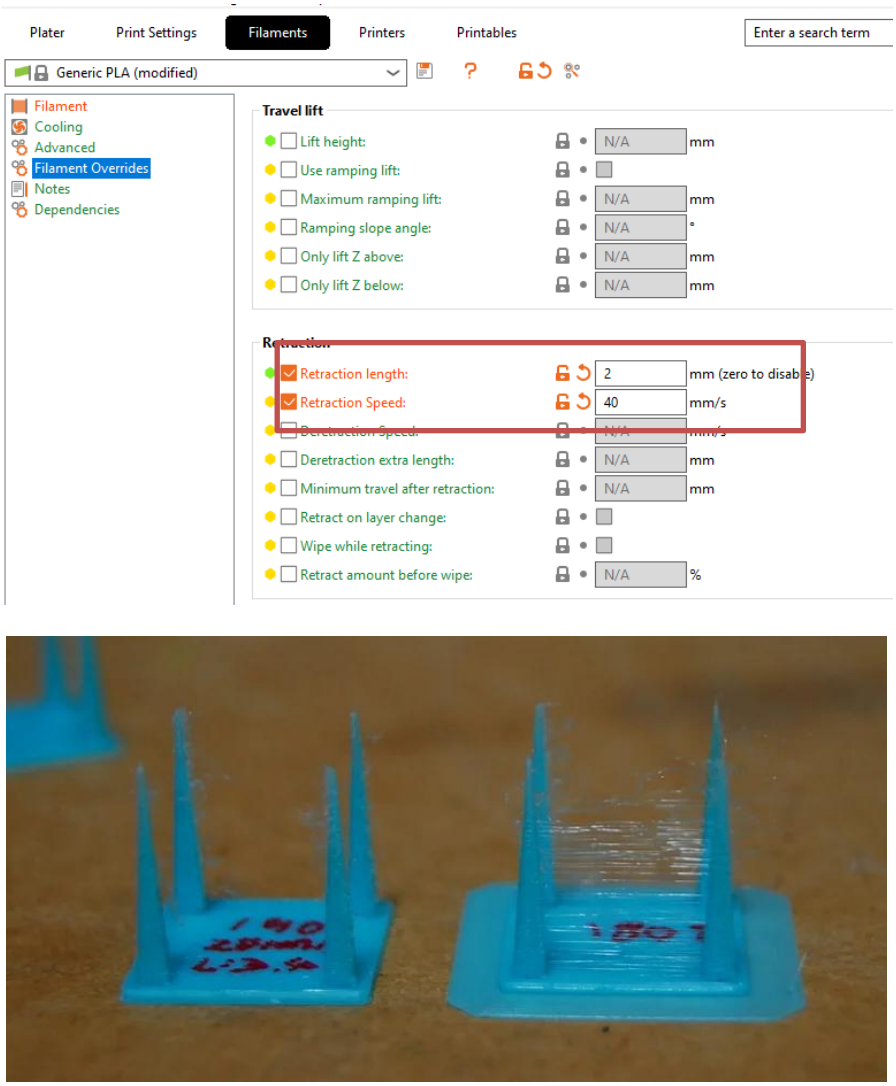


Figure 3.23: Print without stringing (left) and with stringing (right)

Acceleration and Jerk Calibration

We set a feed rate or movement speed in our slicer, but the printer does not instantly reach these speeds. Like a motor vehicle, it needs time to accelerate. If the distance of the movement is short, it may not even have time to reach the specified speed. Complementary to acceleration we have jerk (in PrusaSlicer). These settings have differences, but both are essentially responsible for making sure the printer does not come to a complete stop between each movement, but rather decelerates an appropriate amount depending on the angle of the next 'corner'. The aim of this calibration is to find the right compromise between printing speed and quality, specifically related to surface artefacts such as ghosting.

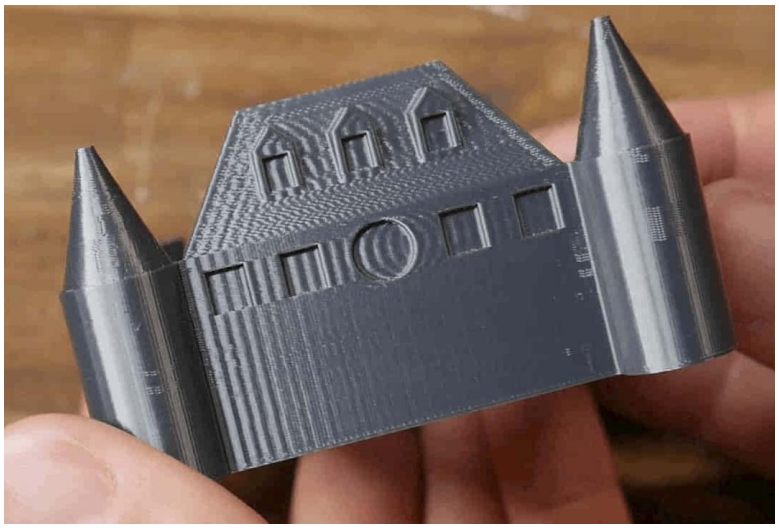
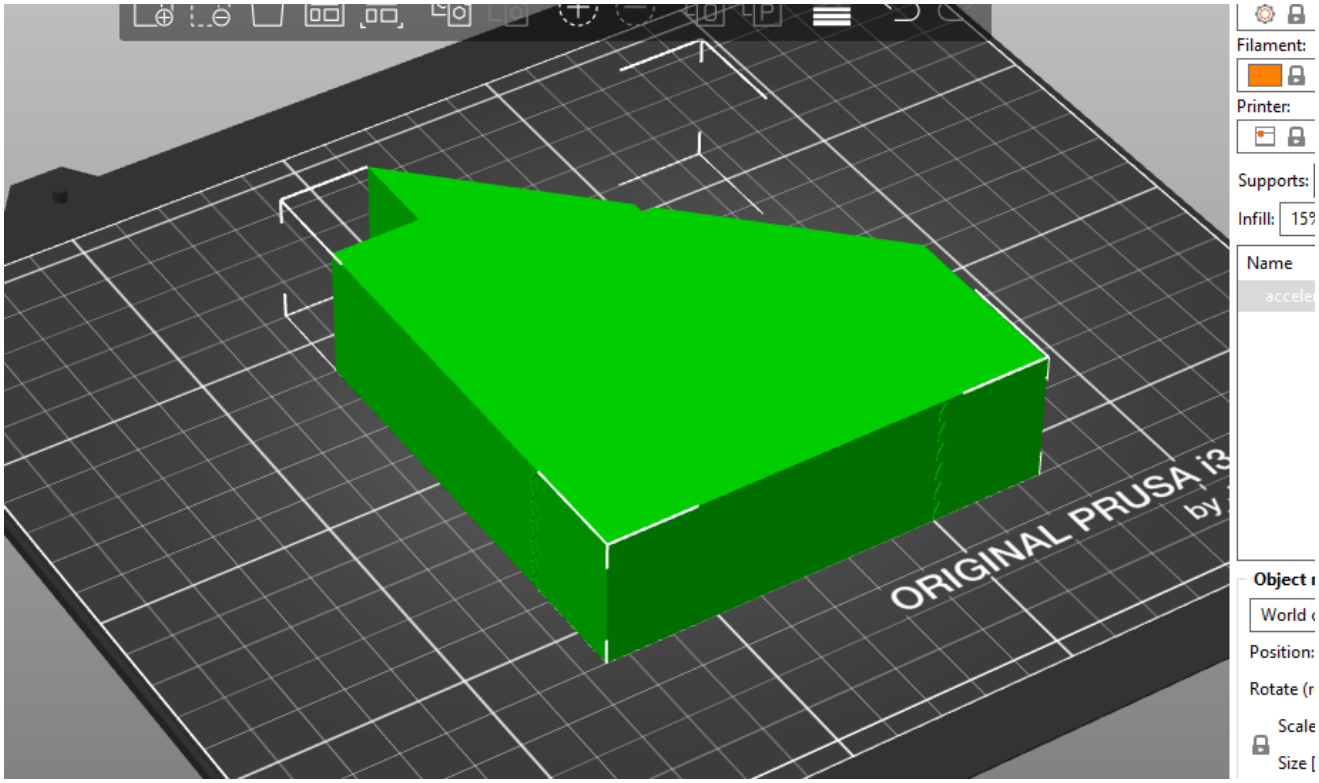
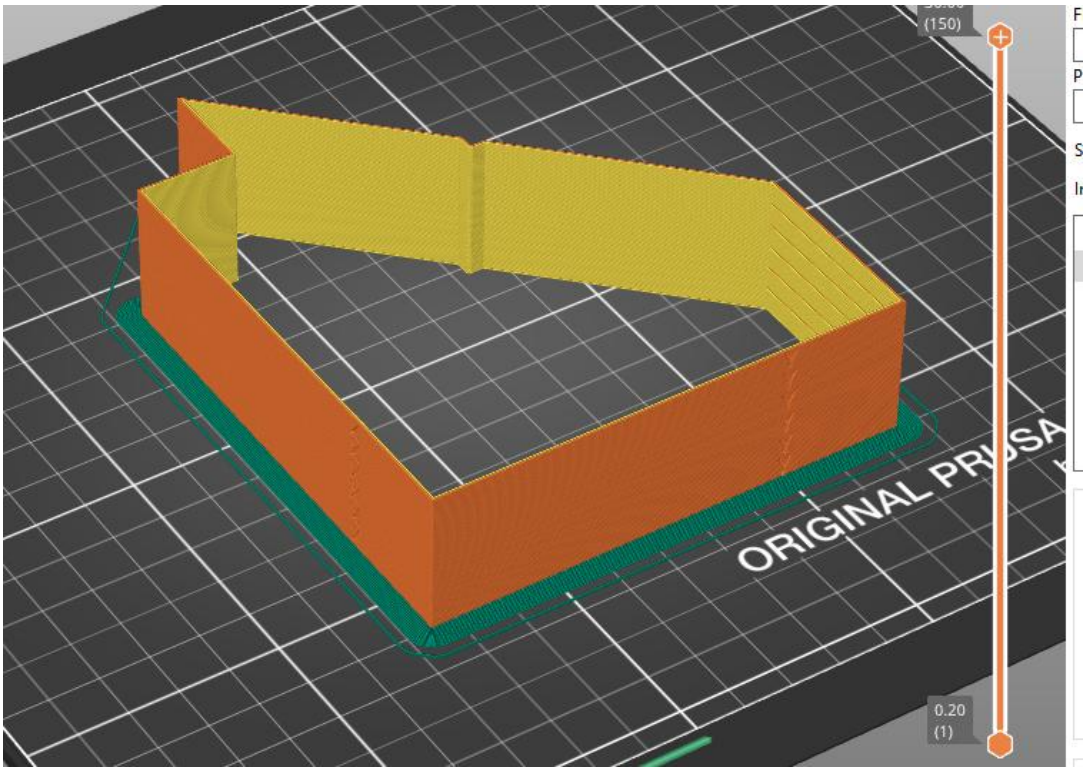


Figure 3.24: Ghosting in 3D print

Go to Thingiverse.com, and download 3D object named “Acceleration/ junction deviation test piece”. Import it in PruseSlicer.

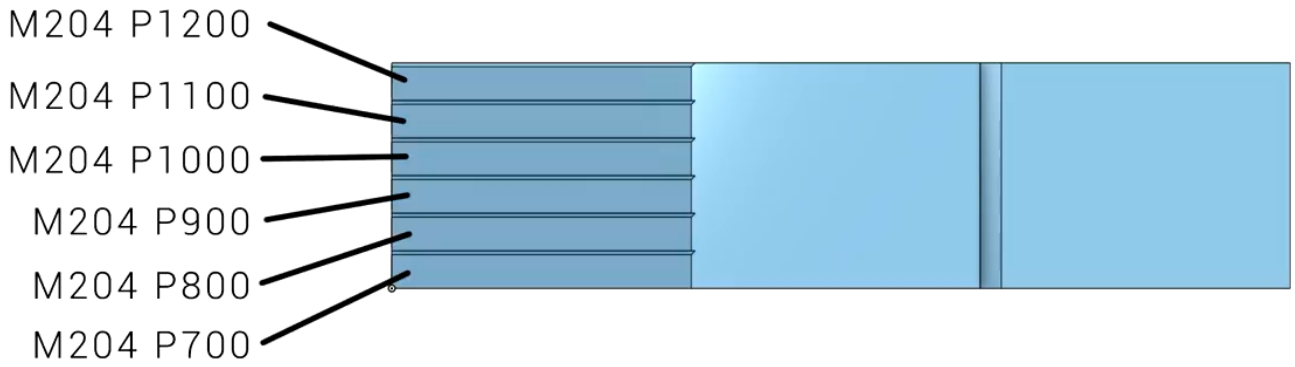


In the slicer, set the solid layers and infill to zero and brim width to 5mm. Then slice the model and go to preview window.



Now, similar to temperature test, we will edit g-code at 1st, 25th, 50th, 75th, 100th, and 125th layer.

We will enter different values for acceleration first and slice the model. Print and inspect model for least ghosting. After identify the acceleration value for least ghosting, record the value and use it for future print.



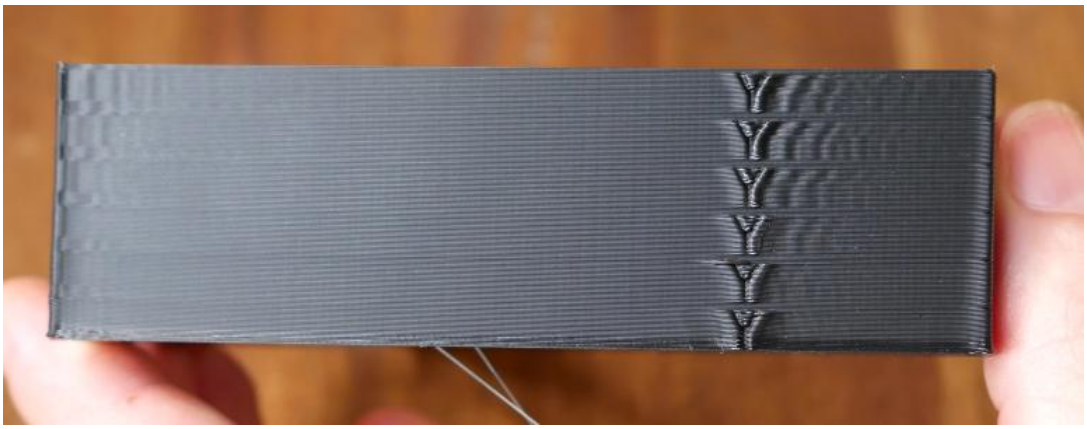
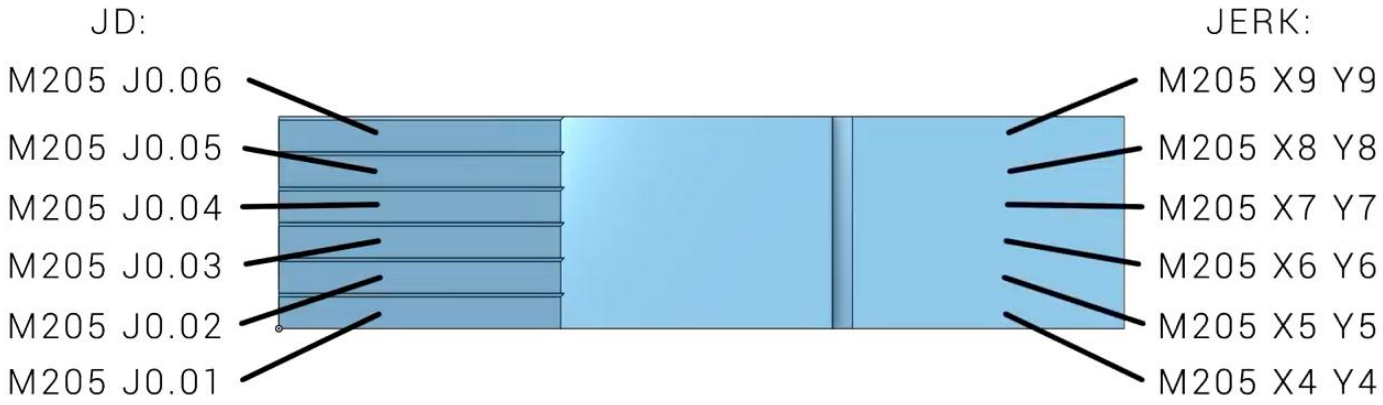


Figure 3:25: Second Y from the bottom has least ghosting

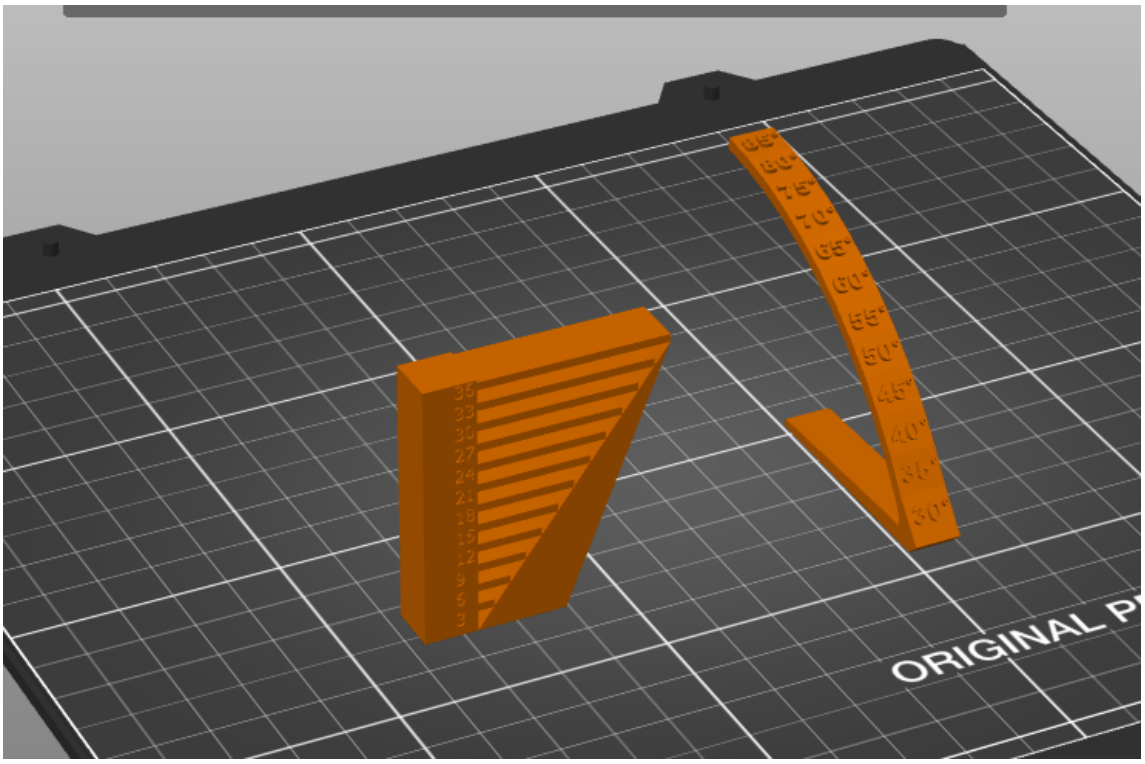
After finding the optimum value of acceleration, we will use it to find out optimum jerk value. The procedure will be exactly same as the acceleration test. Only the custom g-code will be different (as in figure)



Bridge and overhang test

We have discussed about bridge and overhang in information sheet 3.2.2. The aim of these tests is to find out the limiting overhang angle and the minimum distance of bridging to print without support. These tests should be done for every spool of filament.

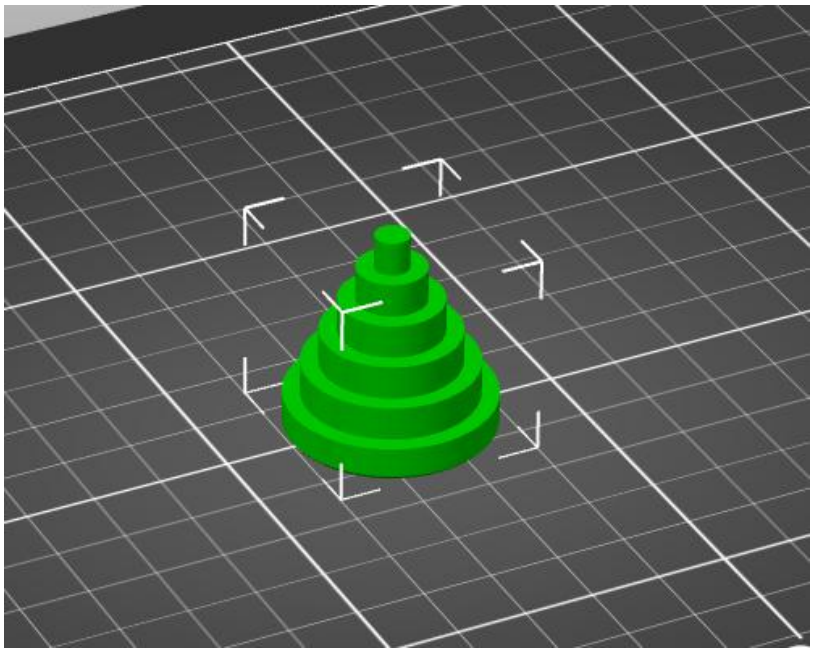
Go to Thingiverse.com, and download 3D object named “Bridge test” and “Overhang test” or, similar. Import it in PruseSlicer. Slice it with calibrated parameters and turn off support generation. Print the models and find out the distance and angle up to which the print quality is good without support. Record the value and use it for future print.



Scale test

A **scale test in FDM printing** primarily refers to using **specific test models to measure and fine-tune your printer's dimensional accuracy and ensure that printed objects match the intended digital scale.** These tests help account for FDM-specific issues like material shrinkage and extrusion width variations.

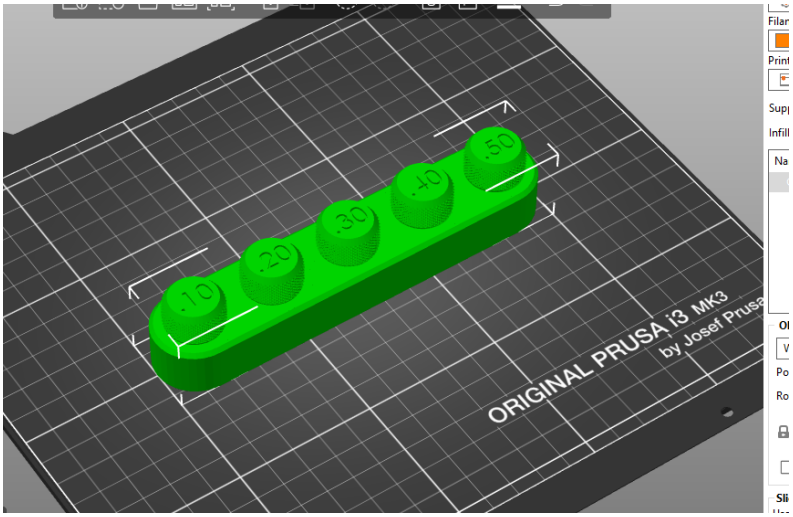
Go to printables.com, and download 3D object named “Simply Dimensional Accuracy Test” or, similar. Import it in PruseSlicer. Slice it with calibrated parameters and print the model. Compare the design dimension with printed model. Calculate the percentage of dimensional deviations. If the deviation is in X and Y direction and uniform, it can be solved by the parameter *Horizontal Expansion* in PrusaSlicer. Otherwise, we have remodel the CAD design with the consideration of % deviation.



Clearance test or Tolerance test

A tolerance test in FDM printing is a procedure to check a 3D printer's dimensional accuracy by printing a specialized calibration model, which often includes features with progressively smaller gaps or clearances. After printing, you measure the model to see which clearances the printer can reproduce accurately, allowing you to determine the right settings and tolerances for future prints and ensure parts will fit together properly.

Go to printables.com, and download 3D object named “Tolerance gauge” or, similar. Import it in PruseSlicer. Slice it with calibrated parameters and print the model.



After printing, physically test the fit. Some holes will be easy to fit a peg into, indicating a larger, looser tolerance. Others will require some pressure to fit, representing a tighter tolerance. Some of the smallest clearances will be stuck or won't fit at all, showing the limits of your printer's accuracy. Compare the results to the model's design to determine the appropriate clearance for your specific printer, material, and project. You can then use this information to adjust the tolerances in your CAD software for subsequent prints.

3.2.6 Job sheet: Print a 3D model with basic parameter settings

Section	Content
Job Title	Print a 3D model with basic parameter settings
Personal Protective Equipment (PPE)	Safety gloves, protective goggles and safety shoes
Equipment, Tools, and Materials	1. 3D model in STL/3MF format 2. FDM 3D printer (PLA/ABS/PETG compatible) 3. Slicer compatible with printer 4. PLA Filament preferable 5. SD card / USB drive / Wi-Fi transfer 6. Bed cleaning tools (scraper, alcohol wipes)
Procedure	1. Printing program generation - Download standard 3Dbenchy STL file from any online repository - import the file in the slicer - perform orientation and manipulation as per requirement - Input the parameters' value Layer height – 0.2mm Perimeters – 2 Infill – 20% Infill pattern – Gyroid Nozzle temperature - suggested value from slicer Bed temperature – suggested value from slicer Other parameter – default value - slice the file - save and transfer the print program to a SD card / USB drive
	2. Printer Preparation - Connect the printer with power outlet and switch on the printer - Collect the PETG/ABS filament spool - Cut the filament in 45 degree approximately - Clean the heatbed and nozzle tip - preheat the nozzle for the collected filament from Preheat option of the printer LCD panel - connect the SD card / USB drive to the printer
	3. Load the filament - place the filament spool on the spool holder - insert the filament tip through the PTFE tube just before the extruder - Run Unload Filament to take out the existing filament - Run AutoLoad Filament option from the LCD panel
	4. print and inspection - run the print program - observe the first layer adhesion - after print completion, remove the print carefully - inspect the printed model

3.2.7 Job sheet: Print a 3D model with support parameter settings

Section	Content
Job Title	Print a 3D model with support parameter settings
Personal Protective Equipment (PPE)	Safety gloves, protective goggles and safety shoes
Equipment, Tools, and Materials	1. 3D model in STL/3MF format 2. FDM 3D printer (PLA/ABS/PETG compatible) 3. Slicer compatible with printer 4. PLA Filament preferable 5. SD card / USB drive / Wi-Fi transfer 6. Bed cleaning tools (scraper, alcohol wipes)
Procedure	1. Printing program generation - Download NEFERTITI STL file by Prusa Research from printables.com - import the file in the slicer - perform orientation and manipulation as per requirement - Input the parameters' value Layer height – 0.2mm Perimeters – 2 Infill – 20% Infill pattern – Gyroid Nozzle temperature - suggested value from slicer Bed temperature – suggested value from slicer Other parameter – default value Support style – Organic Support blocker - eyes and the design in front the tiara - slice the file - save and transfer the print program to a SD card / USB drive
	2. Printer Preparation - Connect the printer with power outlet and switch on the printer - Collect the filament spool - Cut the filament in 45 degree approximately - Clean the heatbed and nozzle tip - preheat the nozzle for the collected filament from Preheat option of the printer LCD panel - connect the SD card / USB drive to the printer
	3. Load the filament - place the filament spool on the spool holder - insert the filament tip through the PTFE tube just before the extruder - Run Unload Filament to take out the existing filament - Run AutoLoad Filament option from the LCD panel
	4. print and inspection - run the print program - observe the first layer adhesion - after print completion, remove the print carefully

	- inspect the printed model

3.2.8 Job sheet: Perform temperature test with ABS filament

Section	Content
Job Title	Perform temperature test with ABS filament
Personal Protective Equipment (PPE)	Safety gloves, protective goggles and safety shoes
Equipment, Tools, and Materials	1. 3D model in STL/3MF format 2. FDM 3D printer (PLA/ABS/PETG compatible) 3. Slicer compatible with printer 4. ABS Filament 5. SD card / USB drive / Wi-Fi transfer 6. Bed cleaning tools (scraper, alcohol wipes)
Procedure	1. Print preparation in slicer - Go to Thingiverse.com, and download 3D object named "temperature tower". - import the file in the slicer - perform orientation and manipulation as per requirement - Input the parameters' value Layer height – 0.2mm Perimeters – 2 Infill – 20% Infill pattern – Gyroid Bed temperature – suggested value from slicer Other parameter – default value - Go to Filaments > Filament > Temperature > Nozzle. Set the first and other layers temperature to some higher value such as for PLA filament it can be set to 230 - slice the model and go to preview window.
	2. Nozzle temperature setting and slicing - in preview window, drag down the slider on the right side of the preview to the top of first segment. - right click on the plus sign of the slider. Three options will be appeared. Choose "Add Custom G-code" option. - A dialog box will be appeared. there enter: M104 S225. This will set the nozzle temperature to 225°C for subsequent layers. -Do the same thing for every segment (not on the top layer) with 5°C decreasing. - slice the model - save and transfer the print program to a SD card / USB drive
	2. Printer Preparation - Connect the printer with power outlet and switch on the printer - Collect the ABS filament spool - Cut the filament in 45 degree approximately - Clean the heatbed and nozzle tip

	- preheat the nozzle for the collected filament from Preheat option of the printer LCD panel - connect the SD card / USB drive to the printer
	3. Load the filament - place the filament spool on the spool holder - insert the filament tip through the PTFE tube just before the extruder - Run Unload Filament to take out the existing filament - Run AutoLoad Filament option from the LCD panel
	4. print, inspection and record - run the print program - observe the first layer adhesion - after print completion, remove the print carefully - inspect the printed model with various print temperature segment - find out the segment with best print quality - identify the temperature for that segment - record the value for future use.

3.2.9 Self-check

Write down the most appropriate answers for the following questions:

1. What should be used for routine cleaning of the print bed?
2. What two main factors define the strength of an FDM printed model?
3. What is the function of the "Place on Face" tool in PrusaSlicer?
4. What are the mouse gestures used to move and view the build plate within PrusaSlicer?
5. What is the primary purpose of a "skirt" during printing?
6. What is the primary difference between G0 and G1 G-Codes?
7. What software is required to connect the printer to a computer for the E-steps calibration process?
8. What visual defect is the retraction test designed to eliminate, and what causes this defect?
9. What is "ghosting" (also known as surface artifacts) caused by, and which two calibration tests help minimize it?

3.2.10 Answer Key

1. Answer:

Isopropyl alcohol (IPA) applied to a lint-free cloth.

2. Answer:

The number of perimeters and the infill density.

3. Answer:

When selected, clicking on a white plane that appears on the model will automatically align that face with the virtual print bed.

4. Answer:

The left mouse button rotates the build plate, the right mouse button pans the build plate, and the scroll wheel allows for both panning and zooming in/out.

5. Answer:

To prime the extruder and ensure smooth filament flow before the model starts printing.

6. Answer:

G0 commands linear movements of the print head *without* extrusion (travel moves), while G1 commands linear movements *with* extrusion (printing moves).

7. Answer:

A terminal software such as Pronterface.

8. Answer:

The test eliminates "stringing," which is caused by melted plastic oozing or dripping from the nozzle during non-print movements.

9. Answer:

Ghosting is caused by the printer's inability to instantly reach or stop at specified speeds during quick direction changes. The **Acceleration** and **Jerk** (Junction Deviation) calibrations help minimize it.

3.3 Learning outcome: Monitor the 3D printing process

Learning Objectives:

At the end of this section, a trainee will be able to know –

1. Real-time Monitoring techniques of 3D printing
2. Troubleshooting at first layer

3.3.1 Information sheet: Real-time Monitoring techniques of 3D printing

Real-time monitoring of the FDM printing process involves using sensors and software to track the print in situ and detect defects as they happen. Common methods include optical monitoring with cameras and laser profilometers, acoustic and vibration sensing, and monitoring temperature, filament flow, and extrusion. Advanced systems can integrate this data with digital twin technology or machine learning models to analyze the data, identify issues like warping or delamination, and trigger automated actions like pausing or adjusting the print.

Monitoring methods

- **Optical and vision systems:** Use cameras (standard or specialized like endoscopic) and laser scanners to capture real-time images and 3D data of the printed part, allowing for visual inspection and defect detection.
- **Acoustic and vibration sensing:** Acoustic sensors can detect anomalies like nozzle clogging, while vibration sensors can monitor the machine's mechanical stability, which may indicate an issue like layer shifting.
- **Temperature sensing:** An infrared (IR) thermal camera can be used to monitor the temperature field of the printed object to identify potential issues related to improper cooling.
- **Filament and extrusion monitoring:** Sensors can track filament flow to detect blockages or jams in the extruder or hot end, and monitor filament availability.
- **IoT and digital twins:** Internet of Things (IoT) sensors collect data that can be sent to a remote location or a digital twin model. This allows for real-time quality assessment, remote monitoring, and predictive analysis.

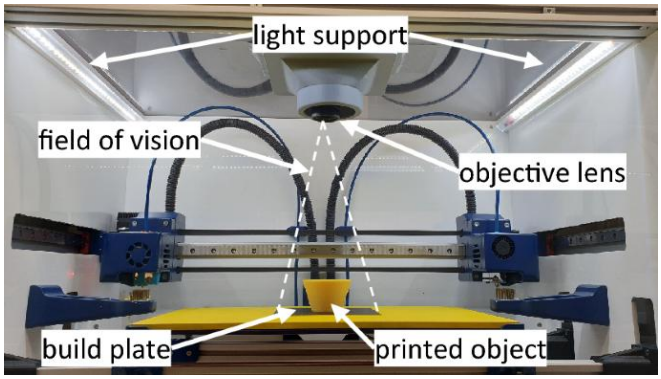


Figure 3.26: 3D-printer with camera mounted on top

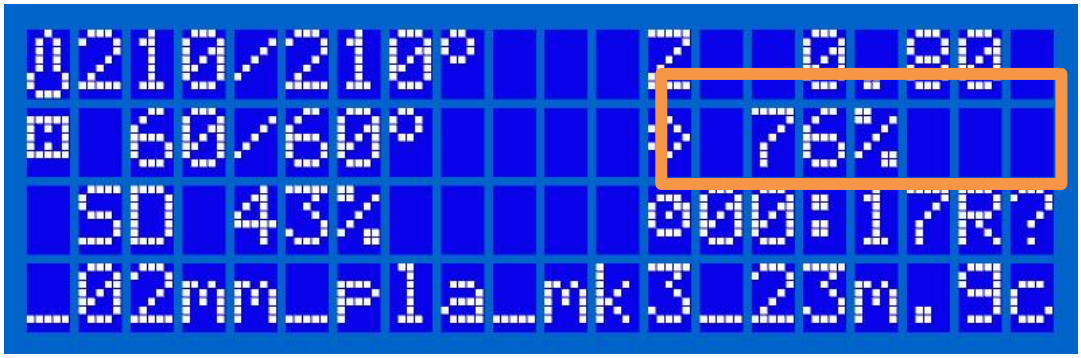
3.3.2 Information sheet: Troubleshooting at first layer

The first layer is the essential one because it is the base of the printed object. Therefore, if it isn't perfect, the chance of print failure increases. Many common 3D printing problems stem from a poor first layer.

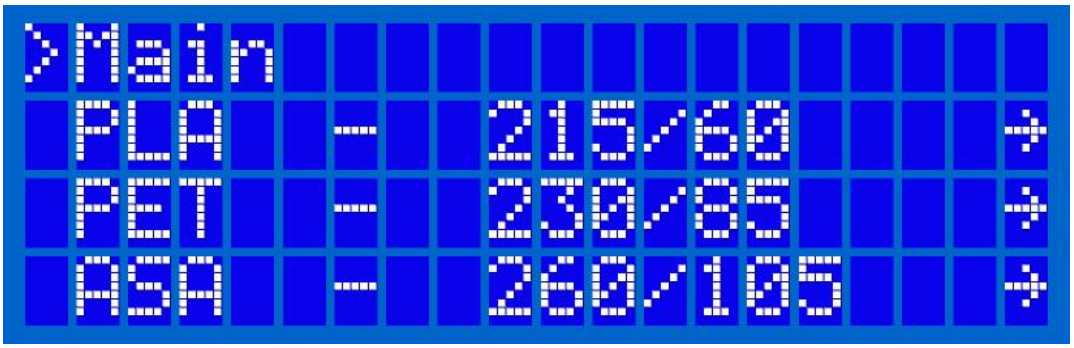
There are a couple of things that can go wrong when printing your first layer, so let's take a look at what can be done to avoid these problems.

There is a possibility that after "First Layer Calibration" and "Live Z" adjustment discussed in learning outcome 3.1, first layer adhesion may not be successful. In that case, following techniques and troubleshooting can be applied –

Decrease the printing speed
By decreasing the printing speed, layer adhesion problem can be remedied. The easiest way how to do it is by rotating the knob during the printing process, lowering the percentage.
Counterclockwise = decrease speed, Clockwise = increase speed
It is suggested decreasing the speed to about 75% for the first three layers, and then returning it to normal.



Use recommended printing temperatures
Make sure to use the recommended nozzle and heatbed temperatures. PrusaSlicer will configure them correctly based on the selected material, so you don't need to adjust the temperatures manually on the printer itself. If you are experimenting with new materials that don't adhere well, you can try to increase the heatbed temperature by 5-10 °C. This way the plastic will stick a bit better.



Brim
Before you consider applying extra adhesion materials onto the bed, consider using the **Brim option** in PrusaSlicer, which increases the surface area of the first layer.

Bed Level Correction
If the width of the line varies across its length, Bed Level Correction may have to be done.

Together with the **Mesh Bed Leveling**, the **Bed Level Correction** is an important feature designed to allow users to compensate for even the slightest imperfections in the first layer. It allows to virtually

raise or lower the heatbed on the Left, Right, Front and Back sides, essentially enabling you to have a different Live Adjust Z value on each of the four sides of the heatbed. This feature is most beneficial if you frequently need to use the entire area of the heatbed.

The adjustment limit is +/- 100 microns (µm). However, even 20 microns can make a huge difference, so when using this function, **make small** incremental changes. The same way as Live-Z adjust, negative values lower the nozzle closer to the heatbed.

Test model

Download "Bed level test" from printables.com and slice the test mode using the Prusa Slicer's default PLA settings, 0.20 mm layer height.

Procedure

- Print the g-code prepared in the previous step with a PLA filament (ideally light and opaque color). The print is just one layer high and takes up most of the printable surface.
- Evaluate where the layer is too squished and full of scars and where there are gaps in between the lines. In Figure 3.27, the lines are way too squished in the top-right corner, and the nozzle is too far away in the bottom-left.
- Go to the *LCD Panel>Calibration>Bed level correction*. There, you can adjust Left/Right/Front/Rear values (understood from your point of view, as you are looking at the printer's LCD).



- Edit the values, we suggest going in steps of +/- 10. These values will be **added to the baseline Live Z value set in step 1. Remember, a negative value will decrease the distance between the nozzle making the layer more squished. A positive value will increase the nozzle-bed distance, curing the scars on your first layer.**
- Repeat the procedure until you are satisfied with the result (Figure 3.28).

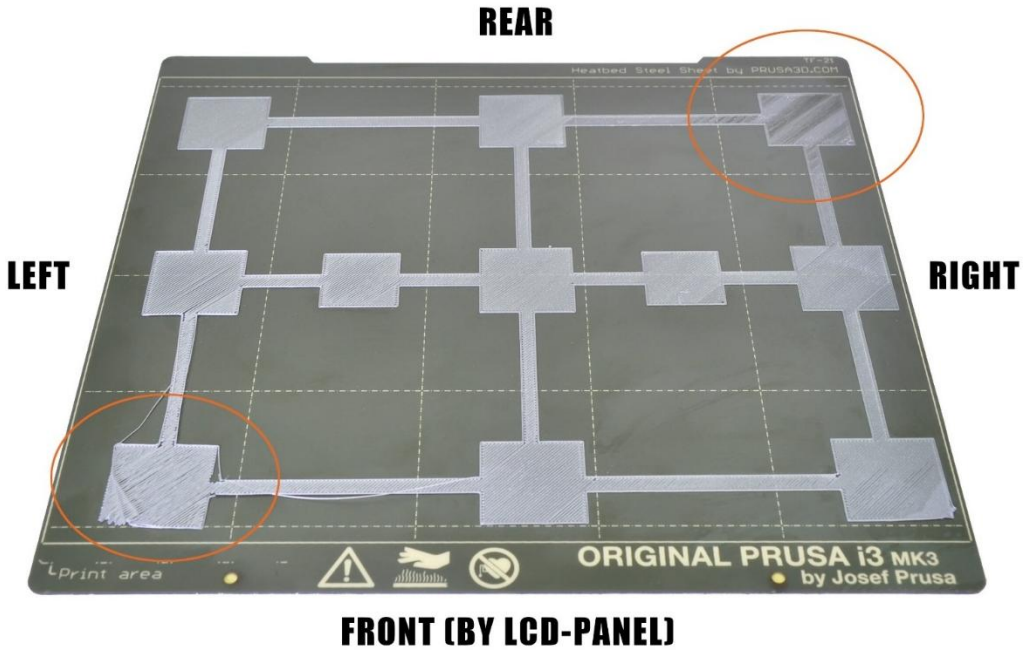


Figure 3.27: Without bed level correction

In the example above you want to adjust a minus-value on the left, bringing the nozzle closer and a plus-value on the right, moving the nozzle away. You may also need to adjust the distance front and back, but do a test print after adjusting the left and right side.

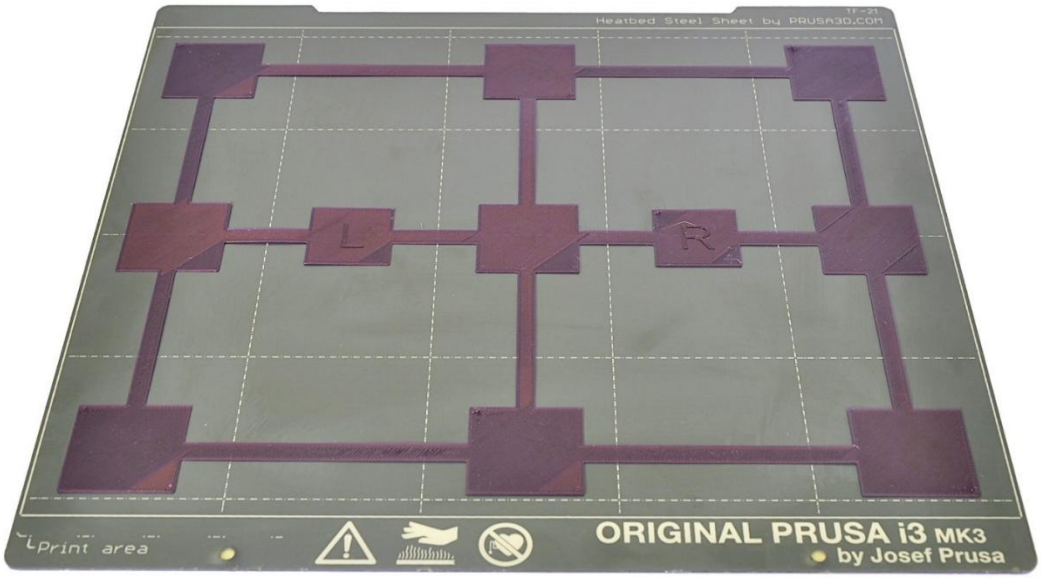


Figure 3.28: After bed level correction

For example, let's say your Live adjust Z value is -1.000, and that the layer is too squished in the rear, and that you see gaps between the lines on the right side.

- **Rear:** you need to enter a positive value (at the end it will be 25 for example) to make the layer less squished
- **Right:** you need to enter a negative value (at the end it will be -10 for example) to make the layer more squished

3.3.3 Self-check

Write down the most appropriate answers for the following questions:

1. What is the primary reason of achieving a perfect first layer is essential for FDM printing?
2. In real-time monitoring, what specific issue can acoustic sensors help detect?
3. What simple, immediate adjustment can be made using the LCD knob *during* the printing of the first few layers to improve adhesion?
4. What is the purpose of the "Bed Level Correction" feature in the printer's LCD menu?
5. When using the "Bed Level Correction" feature, what effect does entering a negative value have on the nozzle-to-bed distance?

3.3.4 Answer Key

1. Answer:

The first layer acts as the base of the printed object; if it is imperfect, the chances of total print failure increase significantly.

2. Answer:

Anomalies such as nozzle clogging.

3. Answer:

Decreasing the printing speed by rotating the knob counterclockwise (suggested reduction to 75% for the first three layers).

4. Answer:

To virtually raise or lower specific areas (Left, Right, Front, Rear) of the heatbed to compensate for slight imperfections and achieve uniform first layer squish across the entire bed.

5. Answer:

A negative value decreases the distance between the nozzle and the bed, making the layer more "squished.".

3.4 Learning outcome: Perform postprocessing of printed parts

Learning Objectives:

At the end of this section, a trainee will be able to know –

1. Support removal techniques
2. Surface finishing and surface treatment

3.4.1 Information sheet: Support removal

Tools Required

Tweezers/ Pliers: Standard or needle-nose pliers for grasping and twisting off supports.
Flush cutters/ snips: Ideal for cutting supports close to the model or reaching difficult areas.
Hobby knife/ Craft knife/ scalpel: For carefully cutting away smaller pieces and cleaning up support contact points.
Sandpaper: To smooth out any leftover marks or imperfections.
Heat gun: Can be used to smooth out minor marks left by supports.
Rotary tool kit: Using mechanical tools is best suited for large, simple models where you don't have to worry about accidentally cutting off some detail.
Solvents: like warm water, citric acid, and isopropyl alcohol for soluble support materials

Step-by-step guide

- **Let the print cool:** Wait until the print is completely cool to avoid damaging the model while removing supports.
- **Remove large outer supports:** Start with the largest parts of the support structure that are not directly attached to the model. You can often use a "twist and pull" method with pliers to snap these off.
- **Use flush cutters:** For any supports that are tough to break off or in hard-to-reach areas, use flush cutters to snip them as close to the model as possible.
- **Clean up details:** Use a craft knife or a small scraper to carefully remove any small remnants, bumps, or marks where the supports connected to the model.
- **Sand and finish:** If there are still visible marks, use sandpaper to sand the area smooth. For some materials, a heat gun can also help to melt away minor imperfections, but be careful not to damage the print.
- **For Breakaway Supports:** Starting from the edges or looser sections, use a pair of pliers or your hands to remove the supports. Avoid using extreme pressure to prevent damage.
- **For Soluble Supports:** Dissolve the supports in warm water (for PVA) or limonene or citric acid (for HIPS). PVA and HIPS support should fully dissolve in 12 to 24 hours; however, delicate or smaller parts may require a shorter soaking time, as little as a few minutes, depending on their size. Check the supports periodically during the soaking process to see if they've sufficiently loosened or dissolved.

3.4.2 Information sheet: Surface finishing and surface treatment

Surface finishing

FDM prints typically have visible layer lines and a rough texture. These methods help achieve a smoother, more professional finish.

- **Sanding:** The most common method. Start with a low-grit sandpaper (e.g., 100-200 grit) to remove major imperfections, then gradually move to finer grits (up to 1000 or even 5000) for a smooth finish. Wet sanding is recommended to prevent heat buildup and reduce airborne dust.
- **Vapor Smoothing:** A chemical solvent (e.g., acetone for ABS or ASA) is vaporized in a sealed container, which gently dissolves and smooths the outer layer of the print, providing a glossy, injection-molded look. This process requires good ventilation and safety precautions.

  The simplest method is to pour acetone into a large container (which can be plastic, but glass is recommended), then place the prints on a platform above the acetone. Close the container lid for 10-20 minutes so that the vapor can melt the outer layer of the parts. You want the vapor to be able to escape the container rather than build up, though. So, if the lid seals tightly, consider drilling a few holes beforehand. If you don’t have an appropriate container, you can apply small amounts of acetone with a brush onto the surface of your 3D prints.



Figure 3.29: Vapor Smoothing with Acetone

- **Epoxy or Filler Coatings:** Applying a layer of epoxy resin or a specialized filler (like XTC-3D, an epoxy designed for 3D prints) can hide layer lines and create a smooth, paintable surface.



Figure 3.30: Epoxy Coatings

- **Media Blasting/Sandblasting:** Abrasive materials (like glass beads or sand) are forcefully projected at the part to achieve a consistent, matte surface finish, which is faster than manual sanding for large batches.

- **Tumbling/Polishing:** Placing parts in a mechanical tumbler with an abrasive medium can polish the surface, especially effective for metal-filled filaments.

Enhancing Appearance and Functionality

- **Priming and Painting:** After smoothing and cleaning, a plastic-specific primer is applied to ensure the topcoat paint adheres well and to further hide any remaining layer lines.

- **Assembly and Joining:** Large models are often printed in smaller parts and assembled using glues (superglue, epoxy) or by chemically "welding" them together using solvents like acetone for ABS.
- **Annealing:** A heat treatment process where the part is heated and slowly cooled to a specific temperature to reduce internal stress and improve the mechanical strength and heat resistance of the material (primarily ABS, PLA, and Nylon).
- **Electroplating:** Coating the plastic part with a thin layer of real metal (e.g., copper) to give it a metallic appearance and significantly increase its strength and conductivity.
- **Hydro Dipping:** Also known as water transfer printing or water transfer imaging, hydro dipping is the process of applying printed graphic designs to solid objects using special water transfer paper. Hydro dipping is only used to enhance an object's aesthetics and does not change the feel or dimensional properties. However, the possibilities are infinite with the graphics you can add to a part, so this is one of the best ways to make your 3D prints eye-catching.



Figure 3.31: Hydro Dipping

3.4.3 Self-check

Write down the most appropriate answers for the following questions:

1. Which tool is ideal for cutting support structures very close to the FDM model or reaching difficult areas?
2. What safety precautions and environment are necessary for performing vapor smoothing with a chemical like acetone?
3. How do soluble support materials like PVA or HIPS differ in their removal process compared to standard "breakaway" supports?
4. What is the primary purpose of "annealing" as a post-processing treatment for FDM prints?
5. What is the difference in finish between using a coarse sandpaper (e.g., 100-200 grit) and a fine sandpaper (e.g., 1000-5000 grit)?

3.4.4 Answer Key

1. Answer:

Flush cutters (or snips).

2. Answer:

Good ventilation is required, and safety precautions must be taken; the process should ideally use a glass container with a lid that allows vapor to escape.

3. Answer:

Soluble supports are dissolved in a liquid (water for PVA, limonene for HIPS) rather than being physically twisted or cut off.

4. Answer:

Annealing is a heat treatment process used to reduce internal stress within the material and improve the part's mechanical strength and heat resistance.

5. Answer:

Coarse grit removes major imperfections, while fine grit is used to achieve a smooth, professional finish.

Module-4: Performing Masked Stereolithography (MSLA) Printing

Module descriptor:
This unit covers the knowledge, skills and attitudes required to perform masked stereolithography (MSLA) printing. It specifically includes the tasks of preparing the MSLA printer for operation, configuring print settings and parameters using slicer, monitoring the 3D printing process and performing post-processing of printed parts.

Nominal Duration: 50 hours

Learning outcome:
4.1 Prepare the MSLA printer for operation
4.2 Configure print settings and parameters using slicer
4.3 Monitor the 3D printing process
4.4 Perform postprocessing of printed parts

4.1 Learning outcome: Prepare the MSLA printer for operation

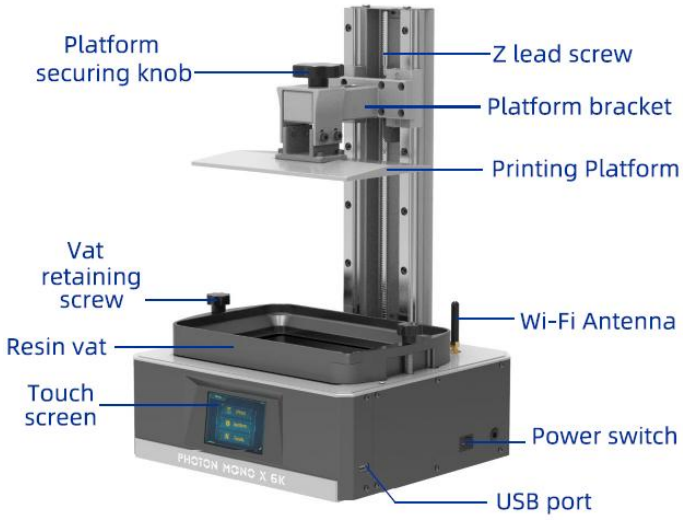
Learning Objectives:

At the end of this section, a trainee will be able to know –

1. Printer interface control
2. Build platform or heatbed cleaning and bed levelling
3. Resin loading and print run

4.1.1 Information sheet: Printer Interface Layout and Control

Throughout this module, we will be using a desktop MSLA printer manufactured by Shenzhen Anycubic Technology Co. Ltd, China, and the printer model is Anycubic Photon Mono X 6K.



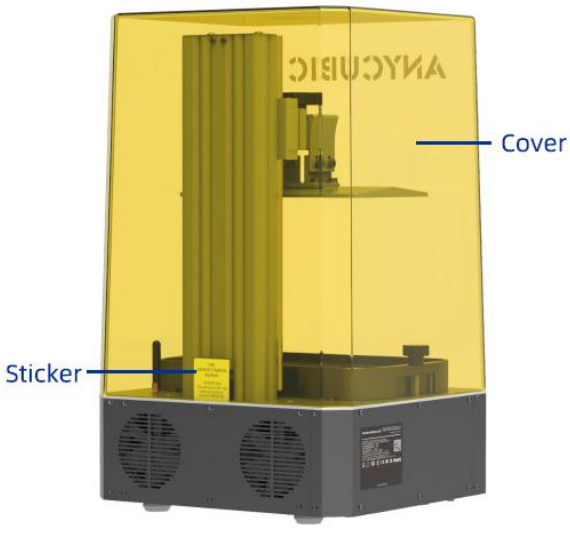


Figure 4.1: Anycubic Photon Mono X 6K

Anycubic Photon Mono X 6K is a large-format resin 3D printer known for its high resolution, achieved through a 9.25-inch 6K monochrome LCD screen, and its relatively fast printing speeds. The printing technique applied in this printer is LCD Shadow Masking.

Key Features and Specifications

- **High Resolution**: The printer features a 9.25" monochrome LCD screen with a **6K resolution** of 5760 x 3600 pixels, resulting in an XY resolution of **34.4 µm**. This allows for the production of highly detailed models with sharp edges and corners.
- **Large Build Volume**: It offers a build volume of **197 x 122 x 245 mm** (L*W*H), providing enough space for large objects or batch printing of smaller items.
- **Light Source**: A powerful 40-LED matrix UV light source is used with rated power of 120W. The printing material should be 405nm UV curable resin.
- **Printing Speed**: The use of a high light-transmittance monochrome screen and its powerful light source allows for print speeds of up to **60 mm/h**.
- **Enhanced Adhesion & Stability**: The build platform is **laser-engraved** to improve model adhesion and reduce the chance of prints falling off or warping. It also uses **dual linear Z-axis rails** for increased stability and precision in vertical movement.
- **User Protection**: A replaceable anti-scratch protective film is included to safeguard the valuable LCD screen, extending its lifespan. The printer also has a removable cover to block UV radiation and a safety feature that can stop printing if the cover is lifted.
- **Software and Connectivity**: The printer is compatible with the free, self-developed **Anycubic Photon Workshop** slicer software (version 3.0 or later). It supports data input via USB Type-A 2.0 and Wi-Fi.

Printer Control

The Anycubic Photon Mono X 6K is controlled via its 3.5-inch color TFT touchscreen. This printer UI is organized into three main sections: **Print**, **System**, and **Tools**.

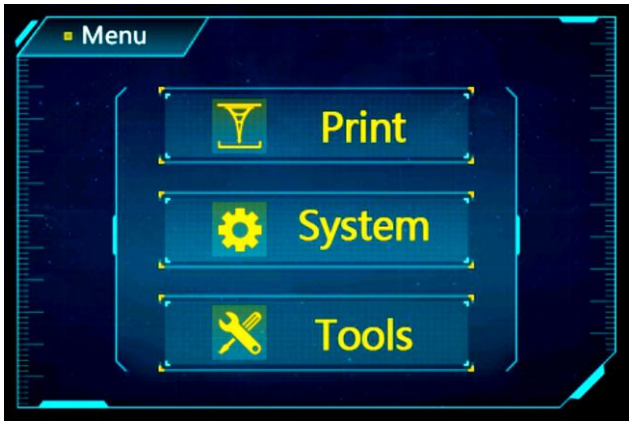


Figure 4.2: Printer UI

Print: This menu is used to select and start print jobs.

- **File Selection:** Allows browsing files stored on the connected USB drive. The printer reads sliced files with the .pwmb format.
- **Start/Pause Printing:** Initiates the selected print or temporarily pauses a running job, at which point the platform will rise to allow for adding or changing resin.
- **Real-time Monitoring:** Displays the current printing progress, number of layers completed versus total layers, and estimated remaining print time.

Tools: This section contains essential functions for maintenance, calibration, and machine settings.

- **Move Z:** Manually controls the movement of the Z-axis (up and down) using specific distance increments (0.1mm, 1mm, 10mm) for tasks like leveling or removing the resin vat.
- **Z=0 (Return to Zero):** Resets the current position as the new zero point for the Z-axis, which is a crucial step in the manual leveling process.
- **Detection:** Allows users to test the UV light source and LCD screen by exposing a test image for a preset time, helping to diagnose potential hardware issues.
- **Exposure:** Manually exposes the screen for a set time, useful for quick curing of the resin in the vat for cleaning purposes.
- **Lock Icon (Door Detection):** Enables or disables the safety function that pauses printing if the printer's cover is removed.
- **Horn Icon (Sound):** Turns the screen beeping sound on or off.
- **Gear Icon (Settings/UV Power):** Enters a sub-menu to adjust the UV power output percentage (e.g., setting it to 100% for transparent resins or 80% for opaque resins).

System: This menu provides access to general machine information and settings. “Language” helps to changes the display language of the UI. “Info” displays firmware version, serial number, and other system details. “Factory Reset” resets all settings to their original factory defaults.

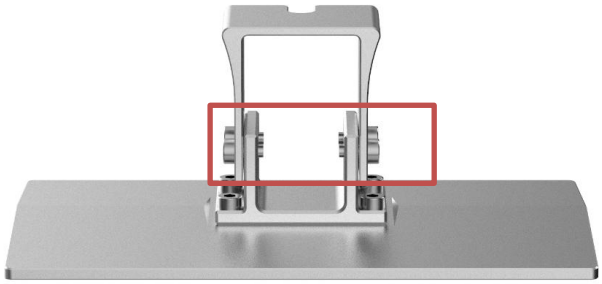
4.1.2 Information sheet: Build platform cleaning and bed levelling

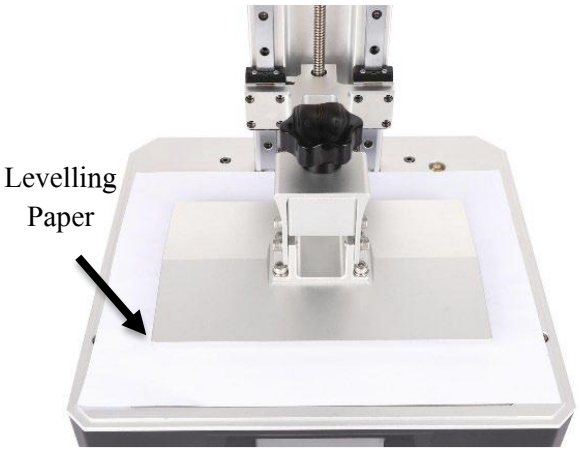
Build platform cleaning

- **Wear gloves:** Protect your hands from resin and cleaning solvents by wearing gloves.
- **Remove loose resin:** Use the scraper provided with your printer to gently remove any uncured or partially cured resin from the build plate. Wipe away any remaining debris with a dry paper towel.
- **Apply isopropyl alcohol:** Mist the build plate with isopropyl alcohol (IPA) or wipe it with a cloth dampened with IPA.
- **Clean the surface:** Use a clean, lint-free cloth or paper towel to wipe the entire surface of the build plate. For build plates with textured surfaces, a soft-bristle brush can be used to clean crevices.
- **Address stubborn resin:** If resin is still stuck, apply more IPA and let it sit for a minute to soften the cured resin. Use a plastic scraper and a low angle to gently scrape off the softened resin. Do not use metal scrapers on the build plate surface or the LCD screen, as this can cause damage.
- **Dry completely:** Wipe the build plate dry with a fresh paper towel to remove any remaining IPA and resin residue. Ensure the surface is completely dry before reattaching it to the printer or starting a new print.

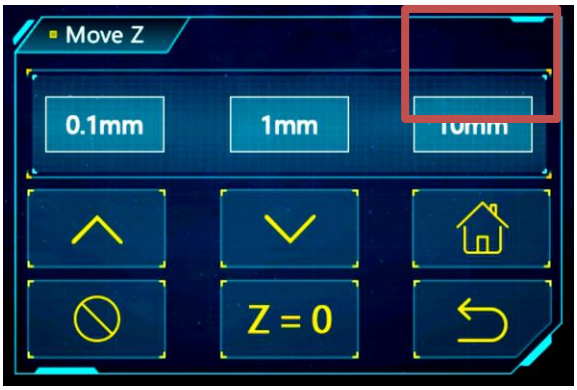
Bed Levelling

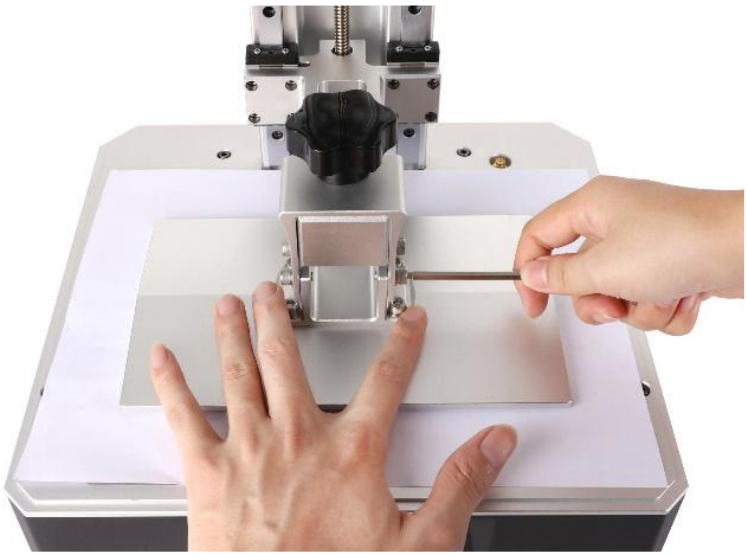
After cleaning the bed, the four side-screws of the bed are needed to be loosen and then the bed is installed on the platform bracket. After that, the resin vat is removed from the printer and a levelling paper or a A4 paper is placed on the LCD screen.



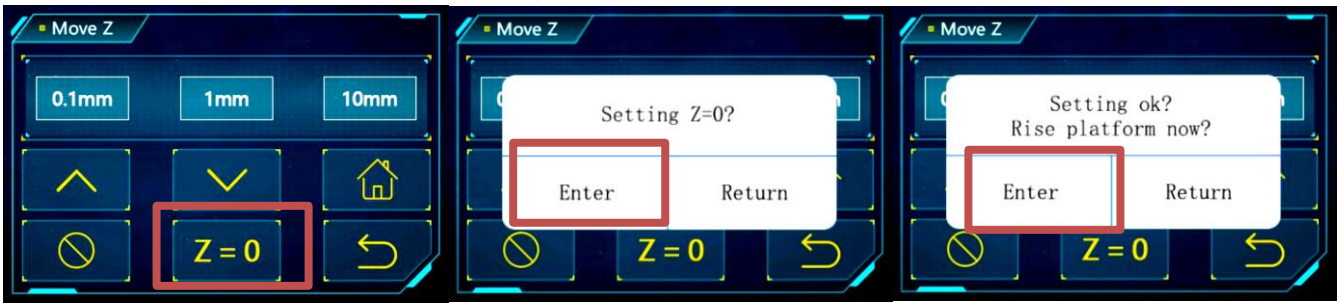


From the touch screen, *Tools > Move Z > Home icon* is selected. Now, Z axis will descend and stop automatically. After the descend, the platform is gently pressed against the LCD screen and the side-screws are tightened.





Then from the touch screen, *Tools > Move Z > Z=0* is selected and a prompt will be appeared for confirmation. After confirmation, this will save the current position of the bottom of the platform as zero position. From *Tools > Move Z*, now Z axis can be ascended and the levelling paper needs to be taken out.



The bed levelling is finished and now the resin vat can be installed in the printer.

4.1.3 Information sheet: Resin Loading and Print Run

Pouring Resin

Pouring resin into an SLA printer vat is a simple process, but requires careful handling to ensure safety and prevent spills or contamination.

It is imperative to always wear chemical-resistant gloves, such as nitrile or neoprene, when handling liquid resin or resin-coated surfaces to prevent skin irritation or allergic reactions. A well-ventilated area must be ensured before printing to avoid the buildup of fumes and vapors.

For a new print

- **Check the Vat:** Ensure the resin vat is securely installed in the printer and free of any debris or cured resin bits.
- **Mix the Resin:** Gently shake the resin bottle for about 30 seconds to one minute before pouring to ensure the pigments and components are evenly mixed, as they can settle over time.
- **Pour Slowly:** Unscrew the bottle cap and carefully and slowly pour the resin into a corner of the vat.
- **Monitor the Max Fill Line:** Most resin vats have a maximum fill line indicated on the side. Do not overfill the vat, as the build plate will displace a significant amount of resin when it lowers, which could cause a messy and potentially damaging spill.
- **Remove Bubbles:** After pouring, you can wait a few minutes for any air bubbles to rise to the surface and pop, or gently mix the surface with a plastic scraper or silicone spatula to help dissipate them.
- **Close the Printer Cover:** Close the printer's UV cover to protect the resin from ambient light, which can prematurely cure it.

Refilling Mid-Print

If the print requires more resin than the vat can initially hold, the print can be paused to add more resin.

- **Pause the Print:** Pause the print via the printer's control panel or associated software. The build plate will lift out of the resin.
- **Pour Carefully:** Slowly pour the additional resin into the vat, being careful not to spill any inside the printer's chassis. Do not exceed the maximum fill line.
- **Resume Printing:** Close the printer's cover and resume the print.

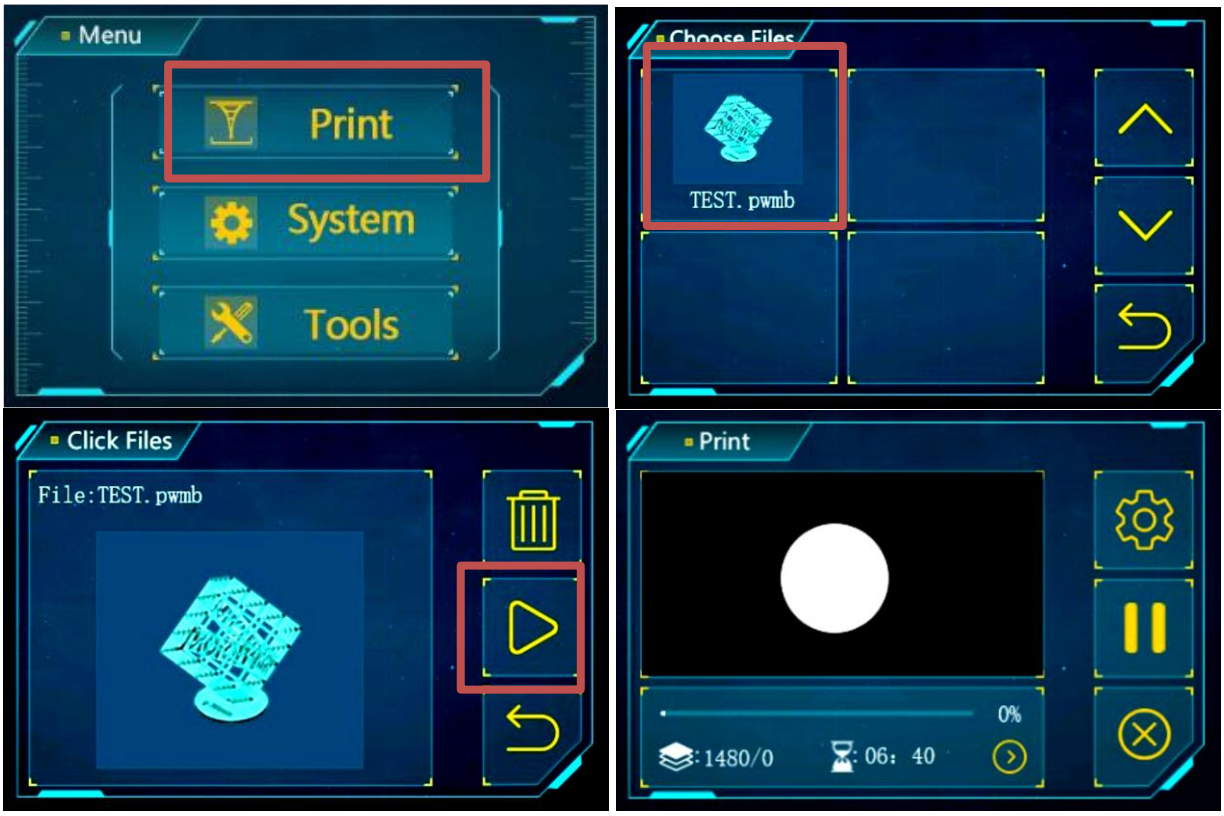


Figure 4.3: Resin pouring process

Print Run

A USB memory with print program should be inserted first. By selecting *Print* from the printer UI, the print program can be accessed. From there, the desired program can be selected. After selecting the print

program, another page will appear with play button to start the print. By tapping the button, the printing can be started.



4.1.4 Self-check

Write down the most appropriate answers for the following questions:

1. What is the specific printing technique used by the Anycubic Photon Mono X 6K printer?
2. What type of material is required for use with this printer's light source?
3. What hardware safety feature does the printer have related to its cover?
4. When cleaning the build plate, what type of scraper should be used, and what type should be avoided?
5. If a print requires more resin mid-print, what happens to the build plate when the job is paused?
6. When performing bed leveling, what item is placed on the LCD screen before the build plate is brought down to the home position?

4.1.5 Answer Key

1. Answer:

LCD Shadow Masking.

2. Answer:

405nm UV curable resin.

3. Answer:

A safety function can be enabled to stop the printing process if the cover is lifted.

4. Answer:

A plastic scraper should be used; metal scrapers should be avoided to prevent damage.

5. Answer:

The build plate will automatically lift out of the resin to allow for refilling.

6. Answer:

A leveling paper or an A4 paper.

4.2 Learning outcome: Configure print settings and parameters using slicer

Learning Objectives:

At the end of this section, a trainee will be able to know –

1. Basic print settings in slicer
2. Printing with support using slicer
3. Advanced Printing options

4.2.1 Information sheet: Basic print settings in slicer

Throughout this module, we will be using Anycubic Photon Workshop slicer developed by Shenzhen Anycubic Technology Co. Ltd for their Anycubic resin printer.

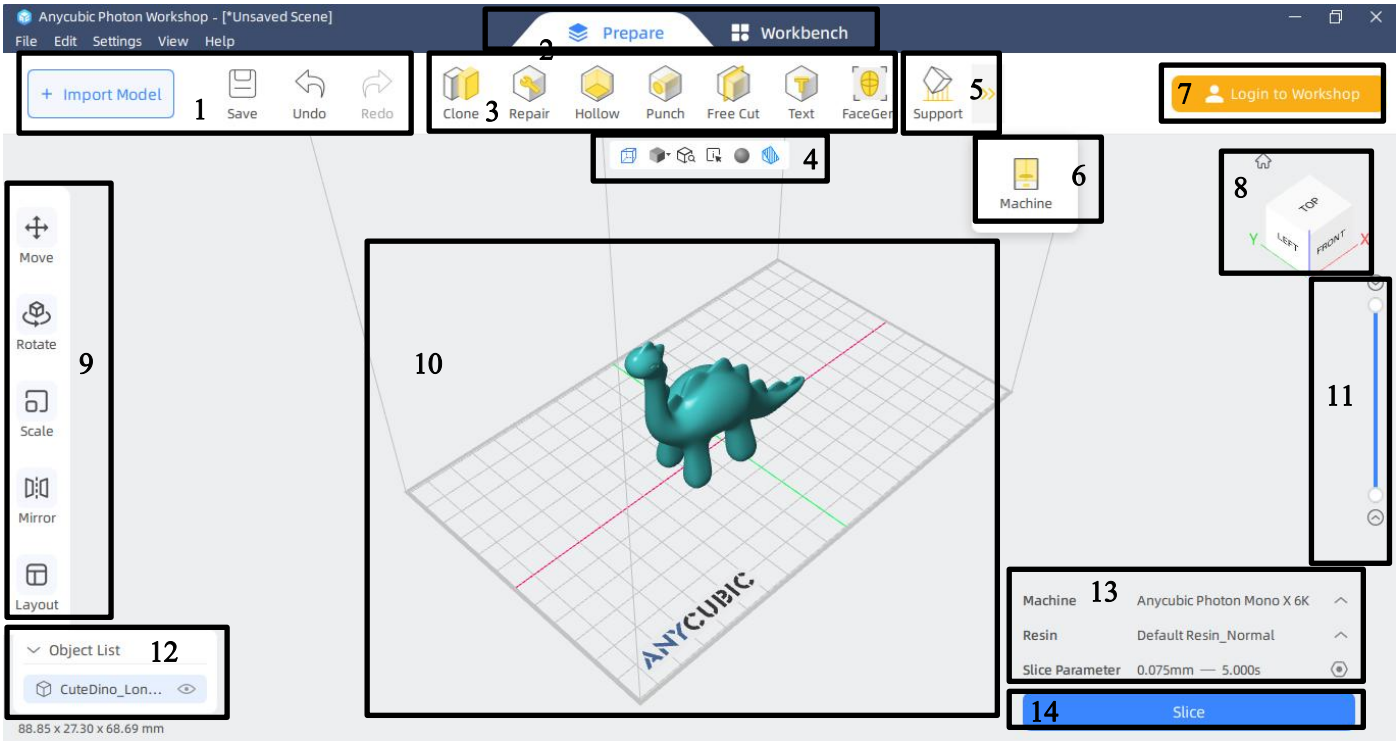


Figure 4.4: Anycubic Photon Workshop Interface

1. Open/Save, Undo/Redo: Standard file operations.
2. Prepare/Workbench Interface: Switch between modes.
3. Edit Functions: Tools for modifying sliced files (e.g., Clone, Repair, Hollow, Punch, Cut, Text).
4. View Mode: Switch between different visual representations.
5. Support Settings: Configure support structures.
6. Machine Settings: Configure printer and resin parameters.
7. Anycubic Cloud: Log in to the cloud service.
8. View Switcher: Toggle between different camera views.
9. Object Editing: Tools for manipulating individual models (Move, Rotate, Scale, Mirror, Layout).
10. 3D Model Preview: Visualizes the model.
11. Layer Slider: Preview each layer of the model.
12. Object List: Displays all loaded models.

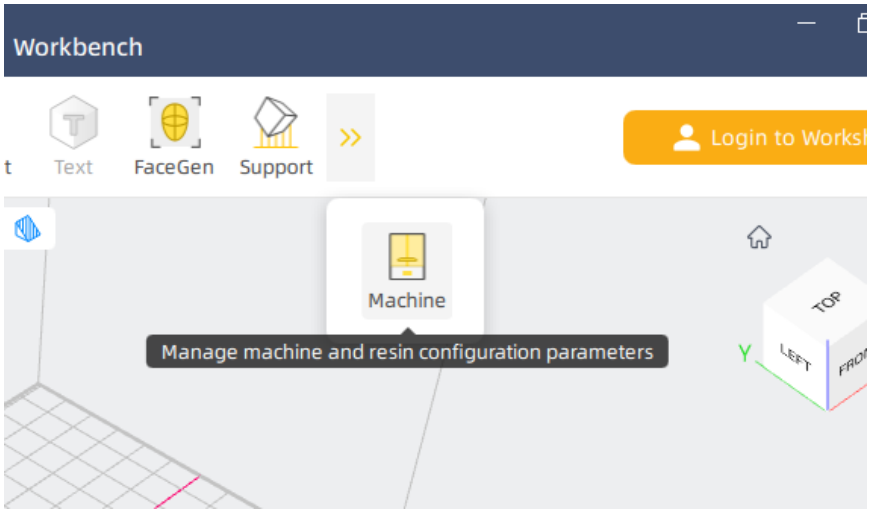
13. Machine Configuration: Set printer, resin, and slice parameters.

14. Slice Button: Initiate the slicing process.

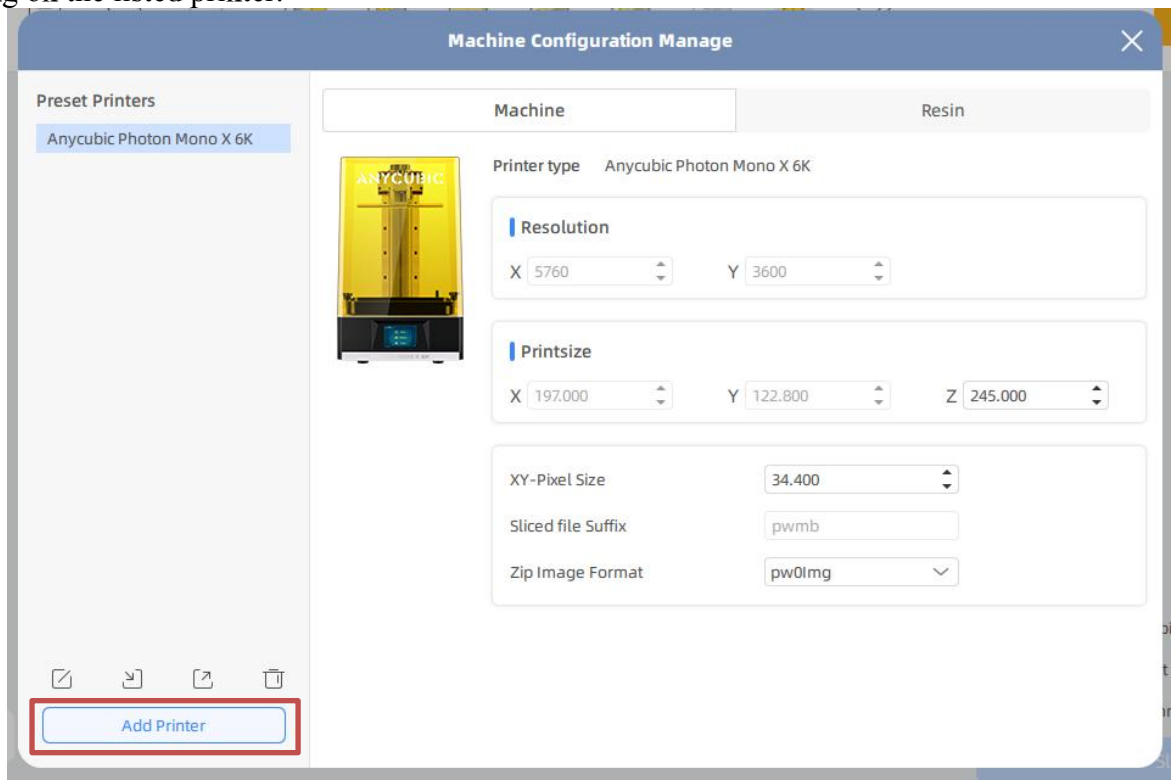
Model Printing with Basic Parameter Settings

Selecting printer and resin

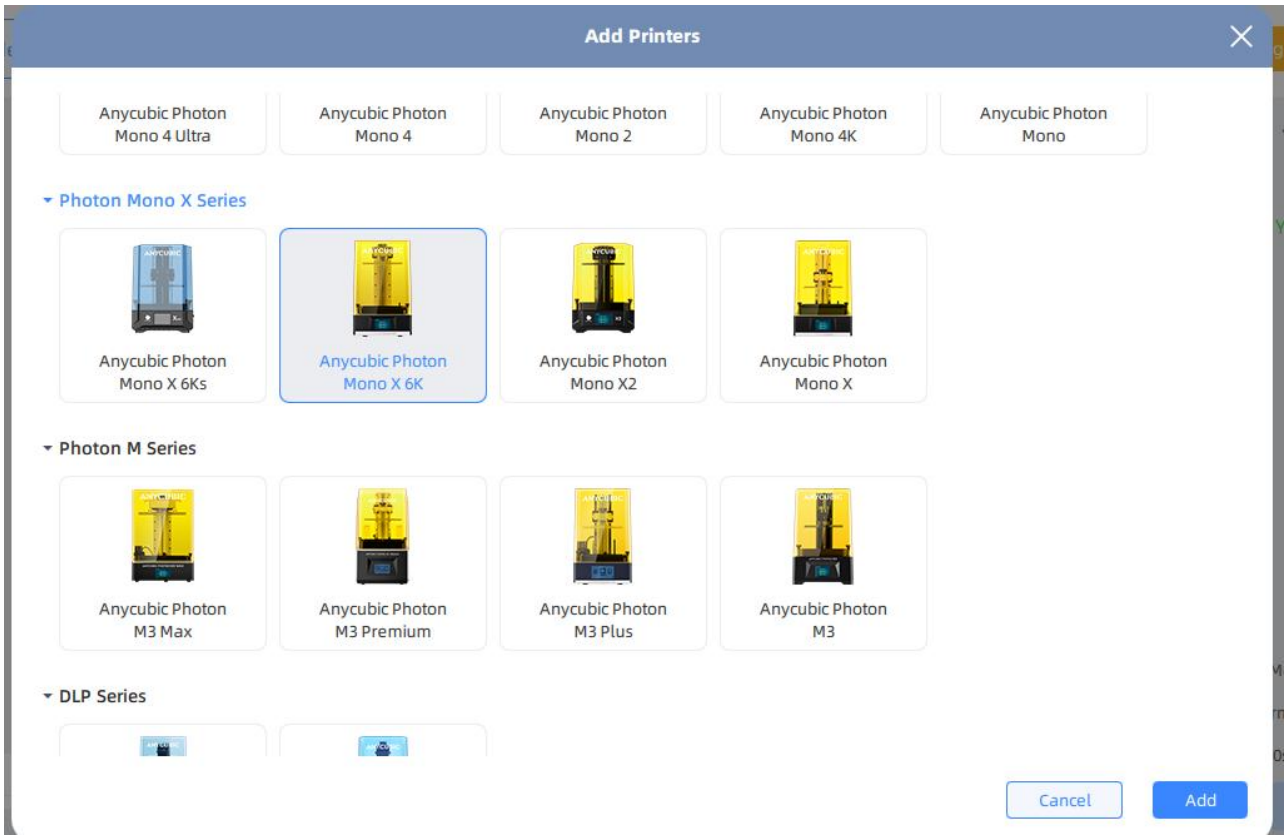
To start slicing, we need to select the printer in which the model is going to be printed and the resin with which the model will be printed. From the *Machine* settings, we can configure printer and resin parameters.



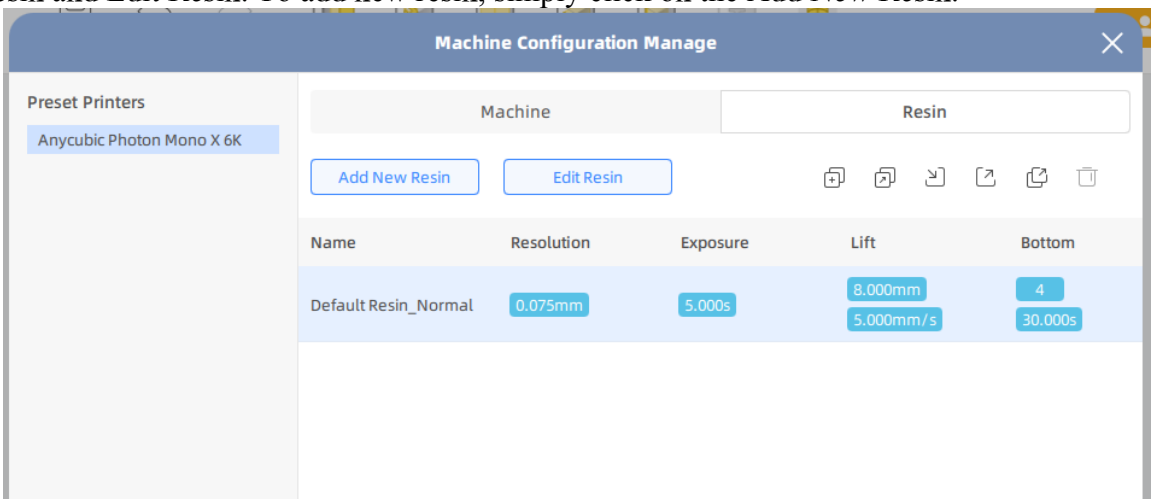
From the *Machine* settings, we can get *Machine Configuration Manage* window. There are two tabs in that window, *Machine* and *Resin*. From *Machine* tab, we can add, delete and modify any Anycubic resin printer or, we create custom printer. To add new printer, simple click on *Add Printer* button. There is list of already added printers on the left side of the tab named *Preset Printers*. Already added printers can be selected by clicking on the listed printer.



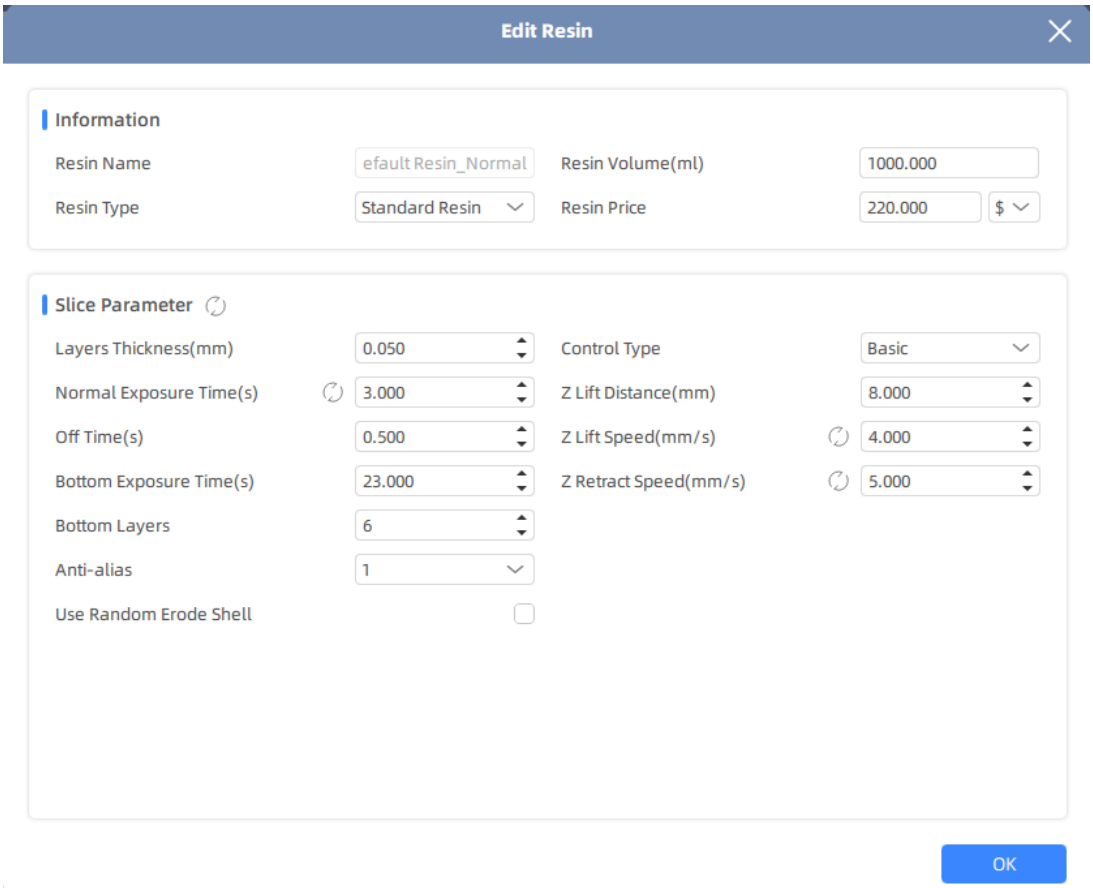
If we click on the *Add Printer* button, a list of printers will appear. From there, we can select our desired printer and click add button from bottom-right corner.



Added printer will be appeared on the *Preset Printers* list. From there, we will be able see the printer's key specifications. Now, to add resin, we have to go the *Resin* tab. There are two options available, *Add New Resin* and *Edit Resin*. To add new resin, simply click on the *Add New Resin*.



To edit resin, double left-click on the resin that needs editing. An *Edit Resin* will be appeared which shows all resin related slice parameters.



Layer thickness: Thinner layers require shorter exposure times but result in more layers overall, significantly increasing the total print time. Conversely, thicker layers shorten print times but reduce detail and make layer lines more visible. Examples of different layer thickness are given below –

100 µm: A faster option suitable for larger mechanical parts, quick prototypes, and some dental applications. Layer lines are visible, but the faster printing speed makes it a practical choice for less detailed models.

50 µm (0.05 mm): The most commonly used layer thickness, offering a good balance between print quality and speed. Layer lines are not noticeable without close inspection, making this setting ideal for miniatures, dental models, and prototypes.

20–30 µm: This setting is perfect for intricate parts, jewelry, and detailed miniatures. At this layer height, layer lines are almost invisible, but print times increase by 1.5 to 3 times compared to 50 µm.

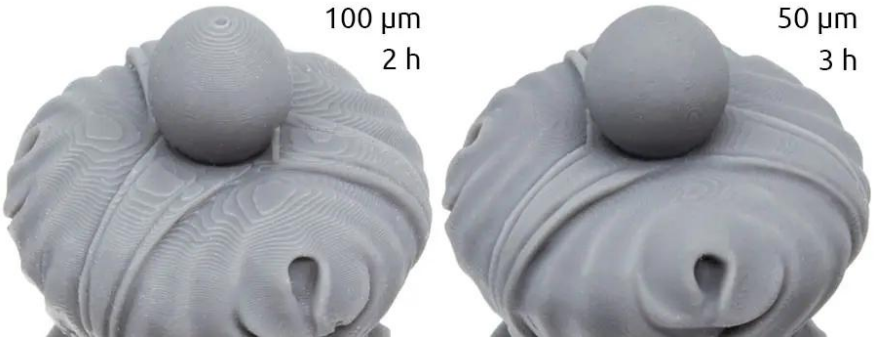


Figure 4.5: Layer thickness vs print quality and print time

Normal exposure time: Getting the exposure time right is essential because it directly impacts:

- **Print Quality:** Correct exposure ensures well-defined details and smooth surfaces.
- **Print Success:** Proper curing minimizes failed prints caused by under- or over-exposure.

- **Sharpness:** Accurate exposure times prevent light bleed, preserving fine features and intricate designs.

By adjusting this setting to suit your resin and printer, you can significantly enhance your printing results and avoid common issues like excessive curing or incomplete layer adhesion.

Finding the ideal exposure time for your resin requires testing with several calibration parts. Start by printing with different exposure times and evaluate which gives the best results for your specific setup. Check your resin manufacturer's website for recommended exposure settings. While these won't be perfectly accurate, they provide a solid starting point. From there, print additional calibration parts using exposure times that are 10–20% longer and 10–20% shorter to fine-tune the settings.

Off time: Typically, a rest time of 0.5 to 1 second is sufficient for the resin to stop moving. However, for viscous resins, a longer rest time is needed, while liquid resins may require a shorter rest time.

Bottom exposure time: The longer the burn-in layer exposure, the better the model will stick to the build plate, preventing it from shifting or detaching during the printing process. Proper burn-in settings are crucial for ensuring that the first few layers cure well and create a strong base for the model.
Recommended exposure times for burn-in layers can vary widely depending on your printer type and model.
When using color LCD 3D printers, we suggest making the burn-in layer exposure around 8–12 times longer than your normal layer exposure time.
Bottom layers: The ideal number of burn-in layers depends on the raft thickness of your model. Generally, the number of burn-in layers should be at least two layers higher than the number of layers in the raft. This ensures the entire raft is well-cured and gives extra curing to the first two layers of supports. This process helps create a stronger base for the supports, ensuring they adhere well to the raft and improves overall adhesion to the build plate.

We recommend using 2–4 layer rafts for optimal adhesion. Here's how to set up your burn-in layers depending on your layer thickness:

For 50 μm layers: Use a 100–200 μm raft and cure 4–6 bottom layers with longer exposure times.
For 20 μm layers: Use a 40–80 μm raft and cure 4–6 bottom layers with longer exposure times.
Without a raft: If you're printing directly on the build plate, use only 2–4 bottom layers. There's no need to cure more layers.

These settings help ensure a solid foundation for your print, increasing the chances of success, especially with complex models.
Z lift distance: The ideal lift height is one that allows the model to peel off the FEP without straining the printer. Thinner FEP films, such as 50 μm, may require a higher lift than thicker films like 127 μm. Similarly, larger printers (e.g., 10-inch) or loose FEP might also benefit from a higher lift. Resin properties can also affect the required lift.

For general use, we recommend a lift height of 10 mm for bottom layers and 7 mm for normal layers. This setting is conservative and works well in most cases.
Z lift speed: There are two main approaches to lift speed, and opinions can differ. Most manufacturers recommend using slower lift speeds to gently peel the model from the FEP, avoiding unnecessary stress. This is especially helpful for beginners who want to reduce the complexity of the printing process. We recommend 40–60 mm/min for normal layers and 5 mm/min for bottom layers.
Z retract speed: Follow the retract speed recommended by your 3D printer manufacturer, typically between 150–400 mm/min. This speed usually does not significantly affect print quality. A slower retract speed will extend the overall 3D printing session.

For very viscous resins, a lower retract speed is recommended. It helps reduce turbulence and allows resin to escape from under the build plate more efficiently. For most resins, however, the impact of retract speed on print quality is minimal.

Anti-alias: A higher grade of anti-alias value could enhance the ability to smooth the edges for each layer during printing, thereby resulting better surface of the printer objects. A higher grade of anti-alias value also means longer slicing time and larger files.

When anti-alias value is larger than 1, you can set the grades of gray level and image blur according to the requirement.

- **Grey Level/Offset:** This parameter controls the starting point and contrast of the grey levels. This needs to be tuned to the specific resin used, as different resins have different sensitivities to UV light intensity. The higher the gray level, the brighter the pixels of anti-alias. That means less smooth surface.
- **Image Blur/Radius:** This setting determines how many pixels away from the edge the anti-aliasing effect is applied. The higher the grade of image blur, the more blurry or smooth the surface will be. Use this cautiously; high blur levels can reduce detail and potentially cause under-curing.

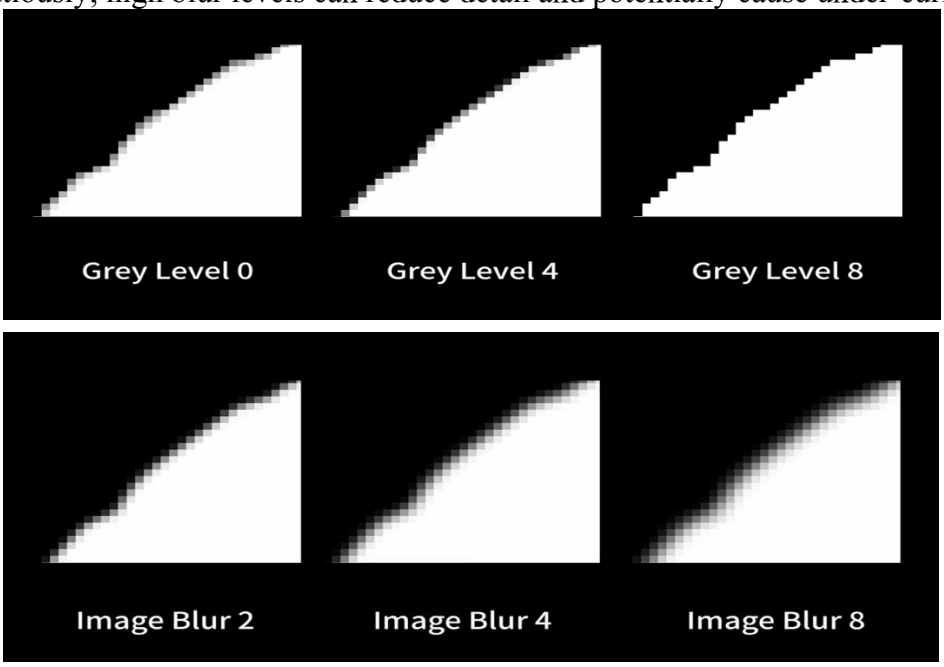
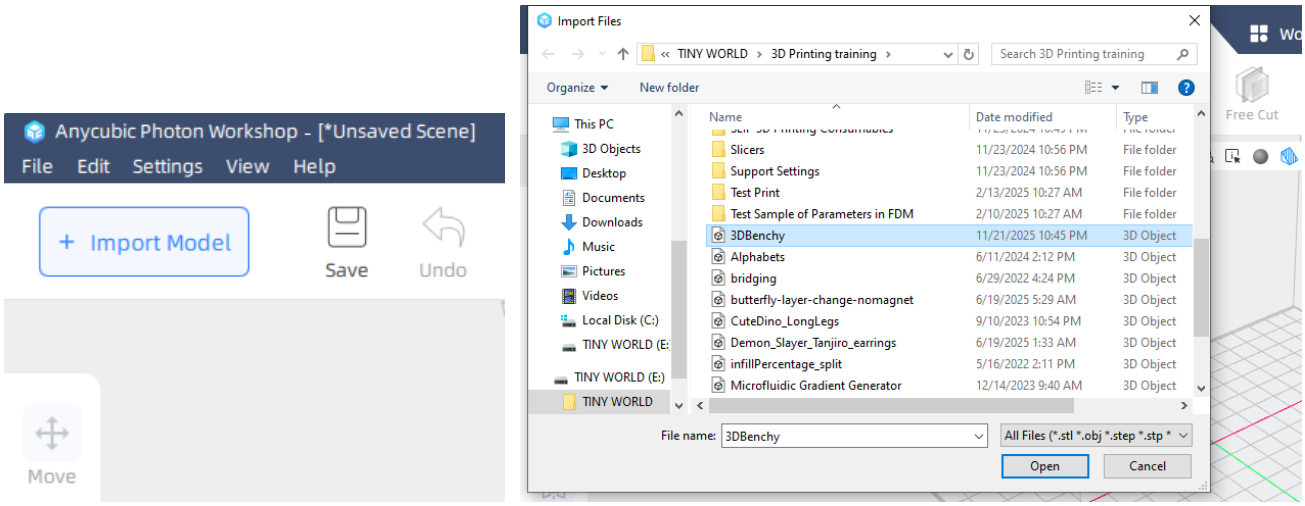


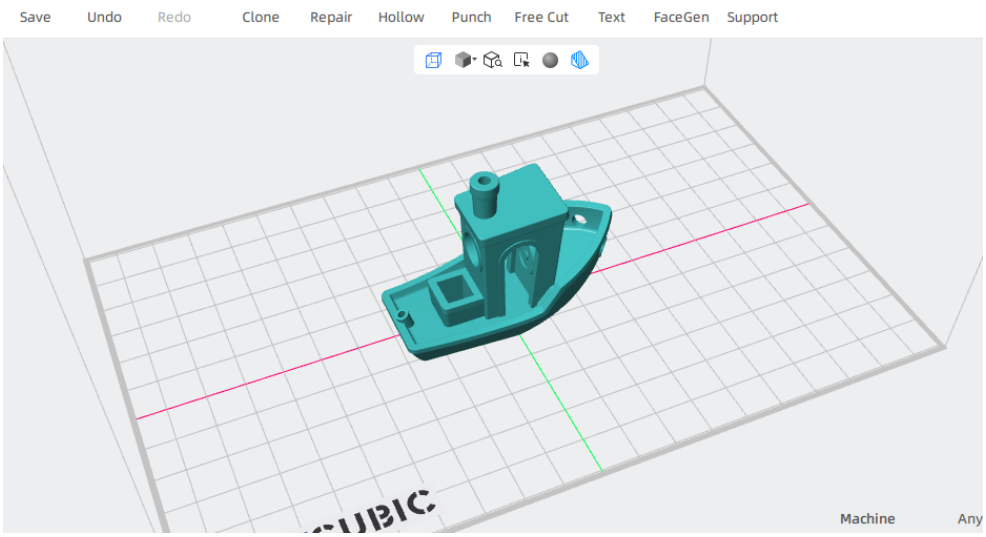
Figure 4.6: Grey Level and Image Blur

Model import

After selecting the appropriate parameter settings, we can add model in the slicer. It can be done by clicking on *Import Model.*



After selecting the model from the desired folder, it will be appeared on the build plate of the slicer.

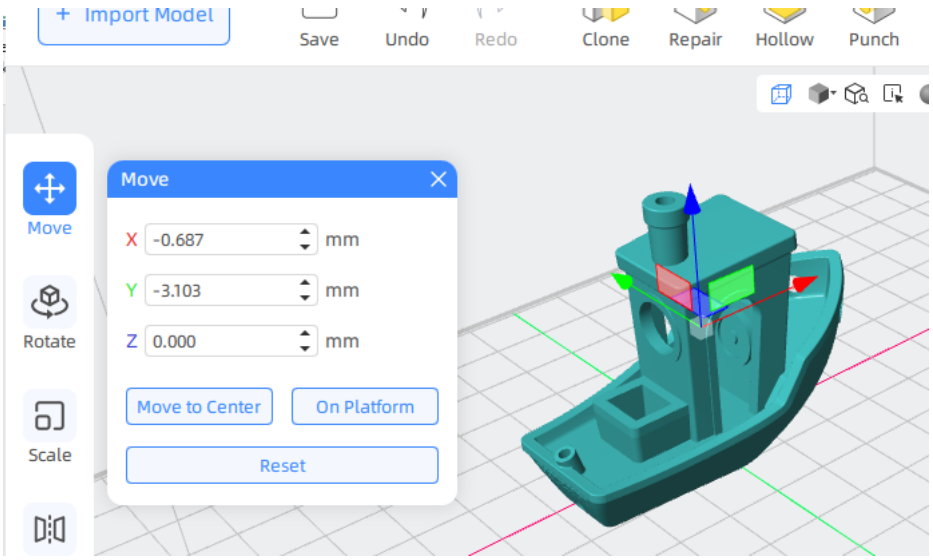


Mouse Gestures

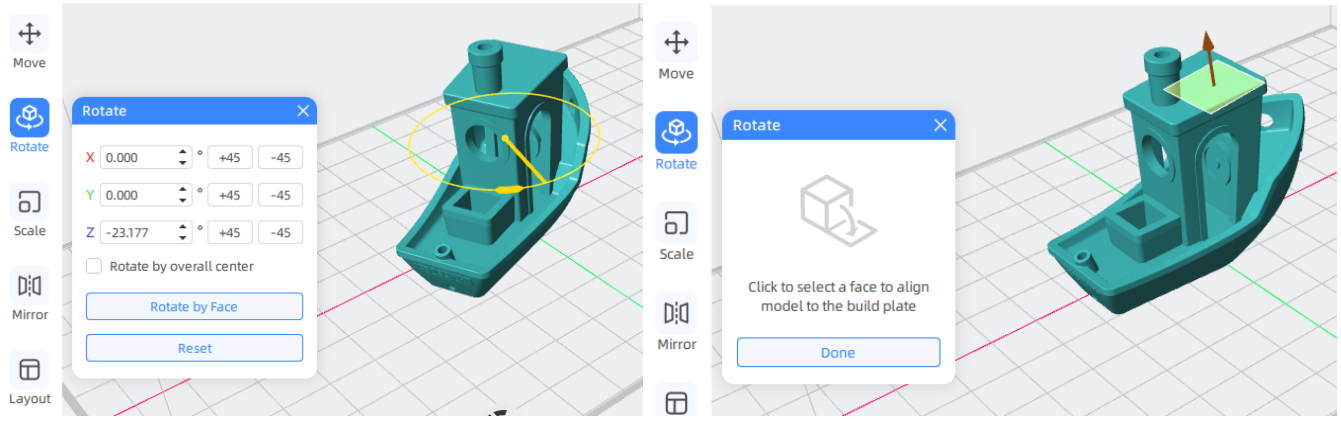
Left mouse button	-	**Pan Build Plate & Manual positioning of 3D model**
Right mouse button	-	**Rotate Build Plate**
Scroll wheel	-	**Pan & Zoom In-Out Build Plate**

Model Orientation and manipulation

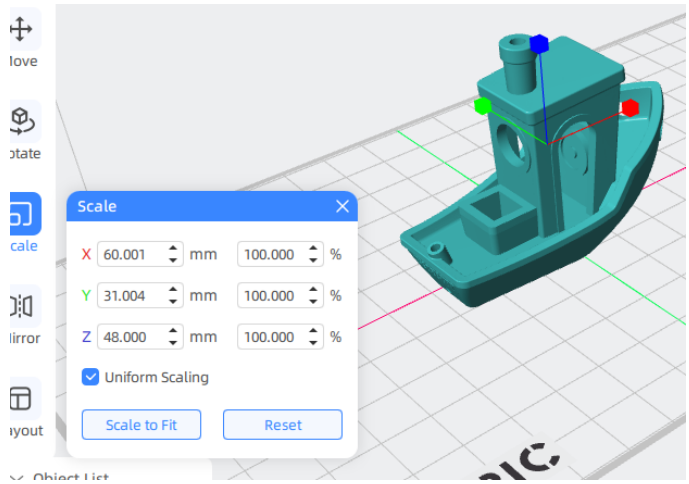
By default, the model will be placed at the center of the build plate. To place the model in any position, Move tool can be used. After selecting the Move tool three arrow directing to three axes (red arrow – x axis, green arrow – y axis and blue arrow – z axis) will be appeared on the model. By clicking on any arrow, the model can be placed to any location on the plate. *Move to Center* option can place the model at the center of the platform. *On Platform* option is required if the model is suspended in the air.



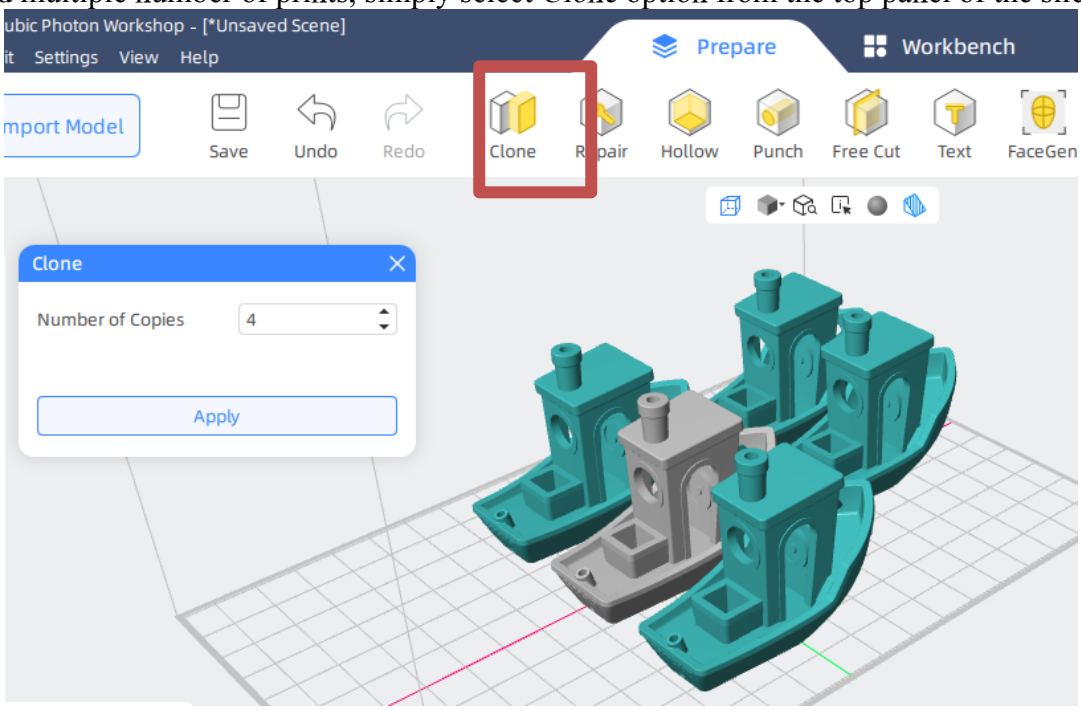
Your model may be oriented incorrectly after import. If that's the case you can use the **Rotate tool** from the left panel of the slicer to manually adjust to the object's orientation by picking a rotation axis. *Rotate by Face* helps to place the model on the build platform along the plane that is selected over the model.



If the model needs scaling, a Scale tool is also available. By clicking and dragging the colored square, the model can be scaled up or scaled down.

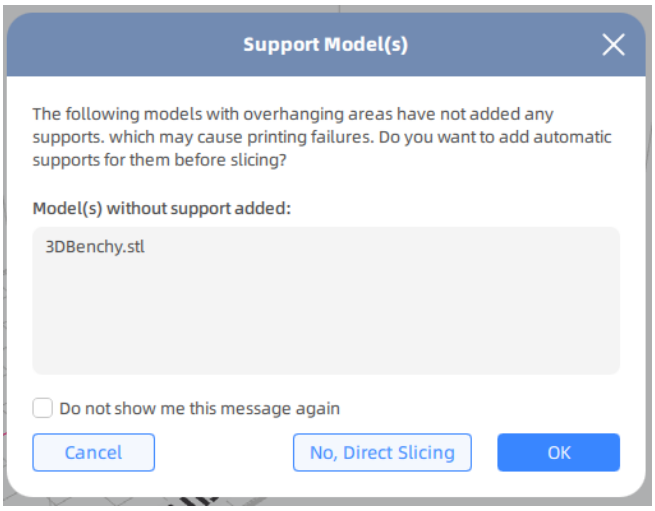


If you need multiple number of prints, simply select *Clone* option from the top panel of the slicer.

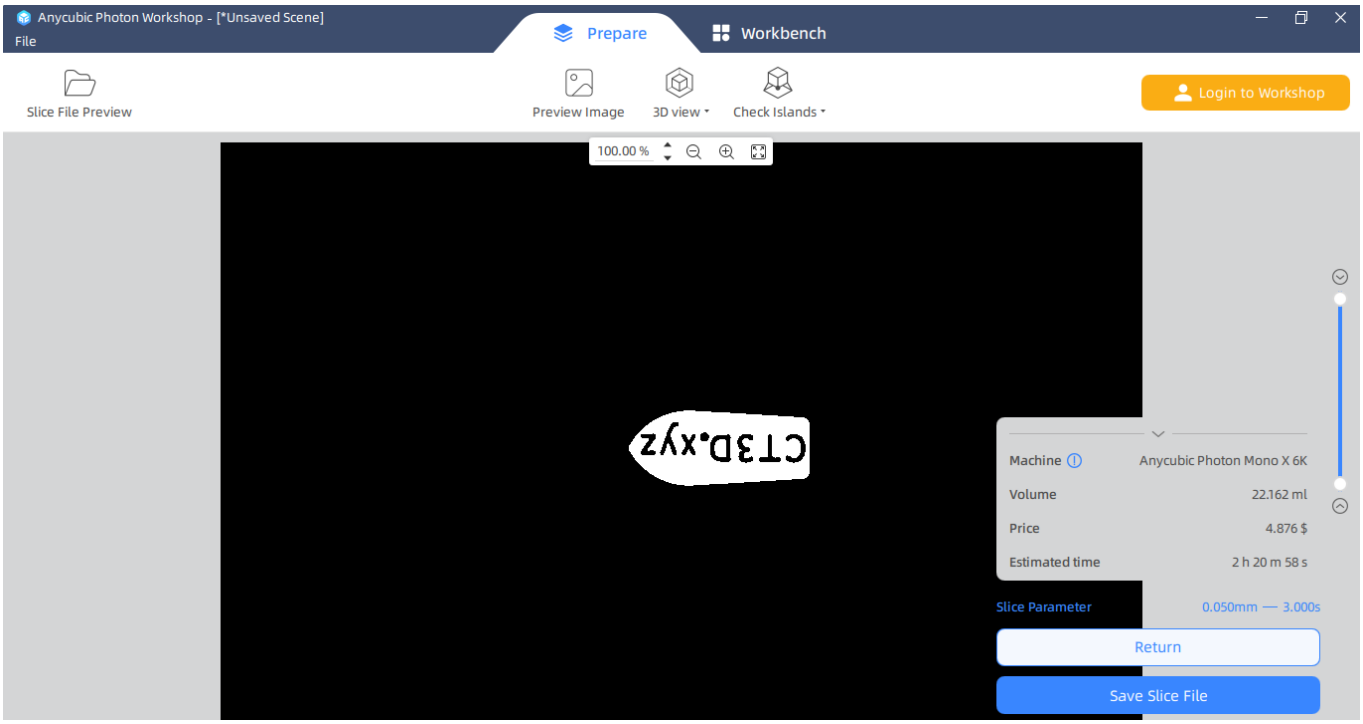


Previewing Sliced View before Print

After model orientation, manipulation and parameter settings, click on the *Slice* button on the lower-right corner of the slicer UI. A prompt will be appeared to select support creation. If there is not requirement for support, then select *No, Direct Slicing*. If required, then select *OK*. For the model here, support is not necessary.



After slicing the model, the slicer will automatically open the preview window with the sliced model.



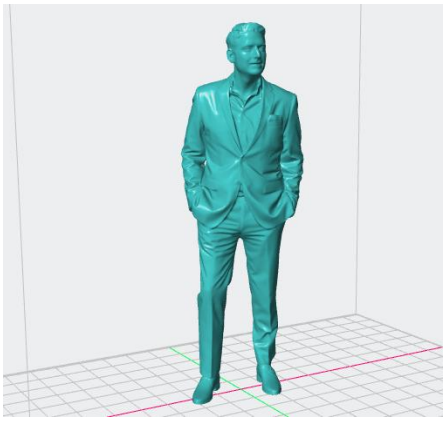
By sliding the slider on the right side of the slicer, the layer by layer view can be accessed. Moreover, the preview window provides the estimation of print time and resin used. If everything seems alright, then we can simply select *Save Slice File* in a USB drive. Then transfer the file to the printer and print.

If we want to change any of the parameter or settings, we can simply go back to the main UI by clicking *Return*.

4.2.2 Information sheet: Printing with support using slicer

In 3D printing, overhangs are the main reason for printing with support. But there is a major difference between support structures used in FDM printing and MSLA printing. In case of MSLA printing, Support structure is more complex but much more efficient than that of FDM printing.

Here, we will be using a model to create support structure and learn the support parameters.



If we print this model in current position, then it will take considerable amount of time to print. This is because the more layers we have, the more layers need to be cured. That means print time is approximately the multiple of exposure time and layer number. The lying position of our model is better condition to print is short time. As in lying position the print area will increase, the risk of layer failure will also increase.

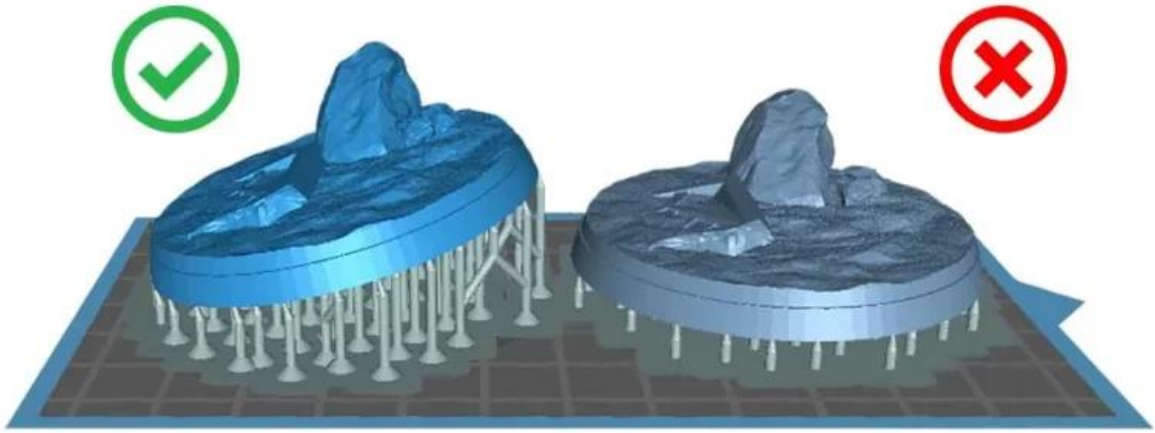
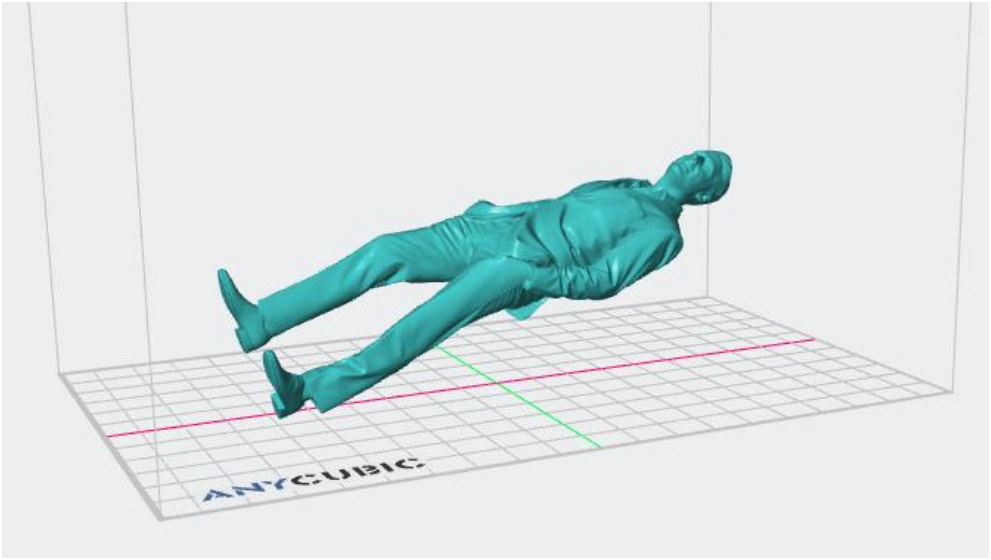
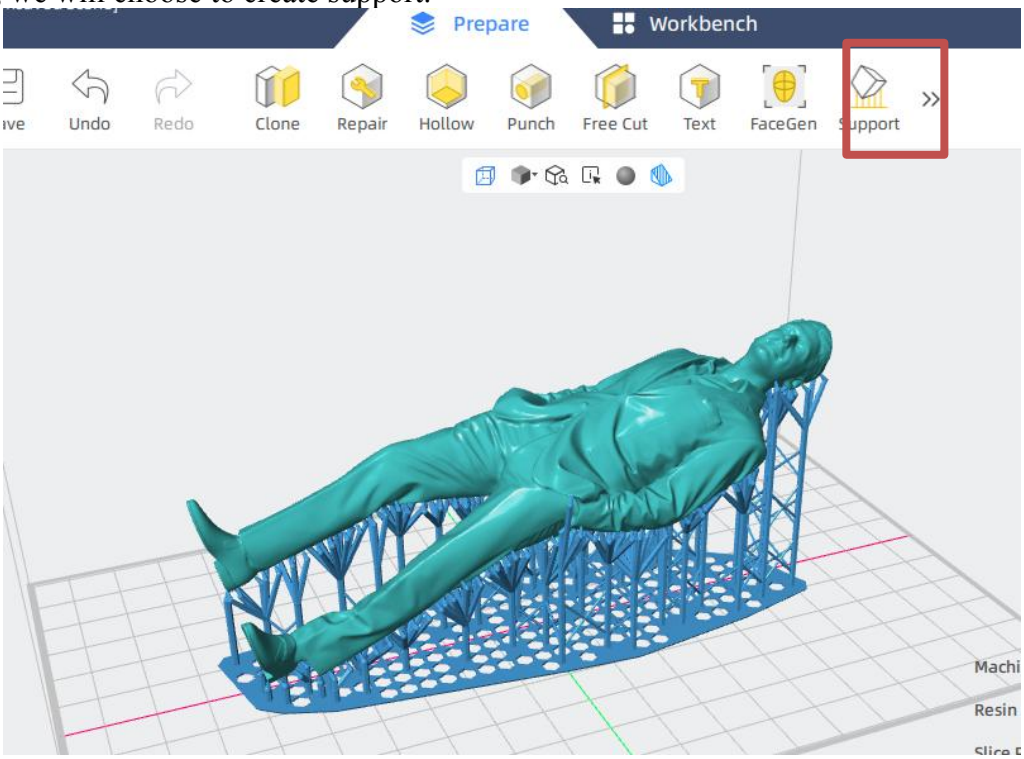


Figure 4.7: Minimizing cross-section by orienting in angular position

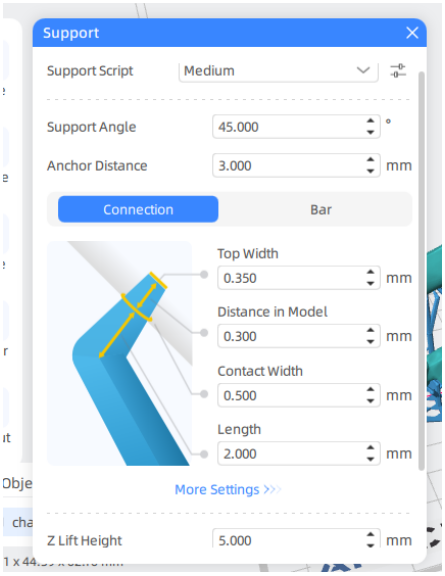
Now, we will set the model in lying position but with 15 degrees inclination.

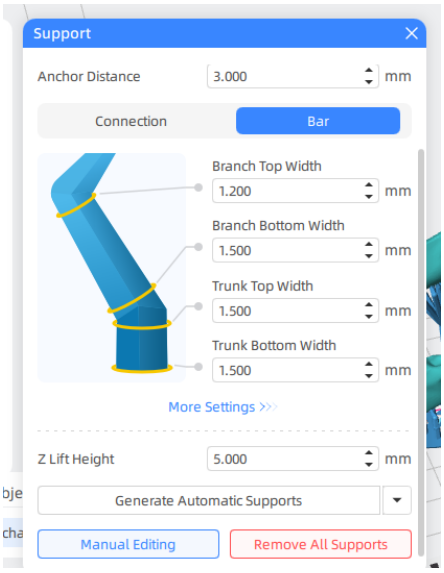


After model orientation, manipulation and parameter settings, click on the *Slice* button on the lower-right corner of the slicer UI. A prompt will be appeared to select support creation. If there is not requirement for support, then select *No, Direct Slicing*. If required, then select *OK*. Here the support is required. So, we will choose to create support.



Now, to change the existing support settings, we can select *Support* option from the top panel of the slicer.





Support Script: There are three support scripts.

- Light: Contact area between the support and model is small, and the support is easy to remove. This is best for small prints.
- Moderate: this is a default setting for support structure. It is recommended to use first.
- Heavy: Support contact with the model are is large and solid. This ideal for large prints.

Support angle: the larger the support angle, the more the supports.

Anchor Distance: the distance between the supports, the shorter the distance the more the supports.

Z Lift Height: Raise the models before adding supports to avoid bottom of the models being destroyed during printing.

Connection

Distance in model: the length of the support bar inserting into lower surface of the model. An appropriate connection length can ensure that the support top is rough enough and easier to remove.

Top width: The width of the contact point that touches the lower surface of the model. The larger the width, the larger the contact are between the support and the model.

Contact width: this is the width of the contact that touches the lower surface of the model.

Length: The distance between the lower surface of the model and the contact point widened to middle width.

Bar

Branch top width: The width of where branch contacts the model

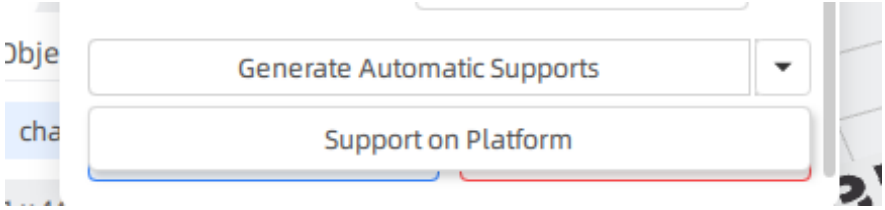
Branch bottom width: the width of where branch contacts the trunk

Trunk top width: the width of where trunk contacts all branches.

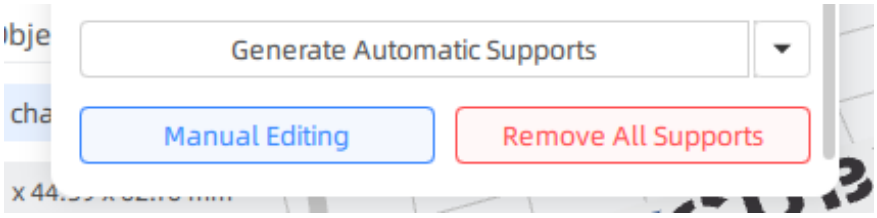
Trunk bottom width: the width of where trunk contacts platform or base.

Generate Automatic Supports

Under Generate Automatic Supports, there are two options support everywhere (*Generate Automatic Supports*) and Support on Platform which create supports on the model where the supports are directly connected with Platform.

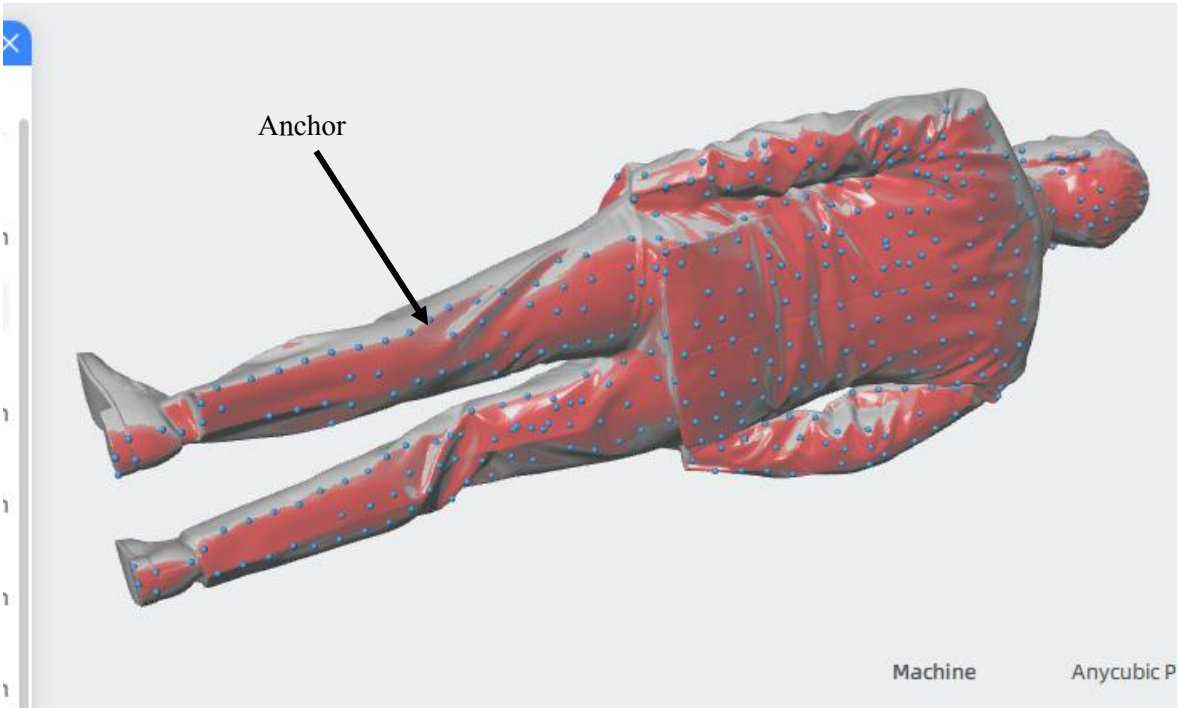


If manual editing of support is required, that also can be accessed from the below of *Generate Automatic Supports* option.



When manual editing is selected, eight options of support editing will be appeared. Among them, editing the anchor points of the support structure is more effective and rigorous.

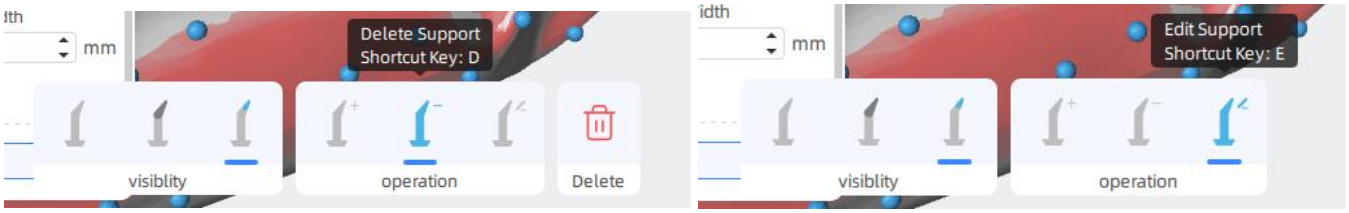




Here, the blue dots are anchors. If we want to create support at any point, first we have to select *Add Support* option and then we simply have to click on our desired position. After that, a new blue anchor will appear.



Similarly, we can delete or re-position an anchor by selecting either *Delete Support* or *Edit Support*.

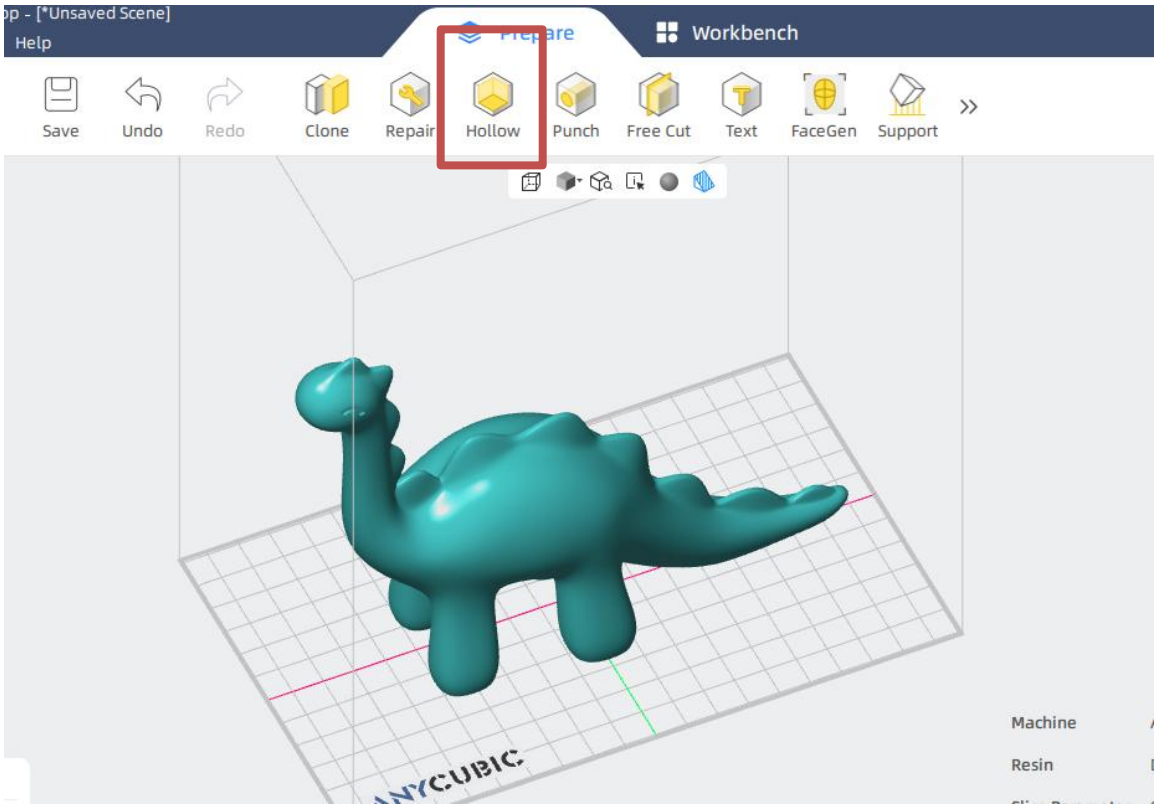


After support editing, we have to slice again. Now in the preview window, we will check the layers by keeping the *Check Islands* option *On.* This option help to find out the places with missing support. If any point requires support and the automatic support generation misses that point, then *Check Islands* option will show that by marking red. This can be solved by adding anchor at that point manually.

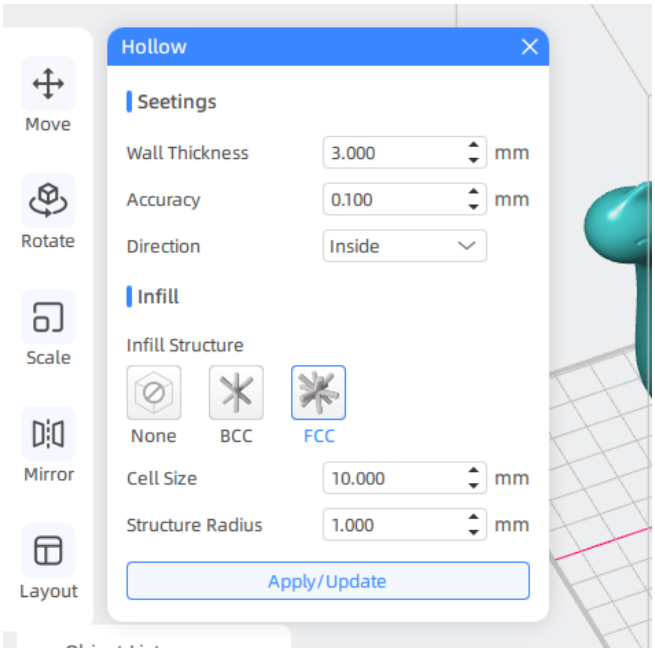


4.2.3 Information sheet: Advanced Printing options

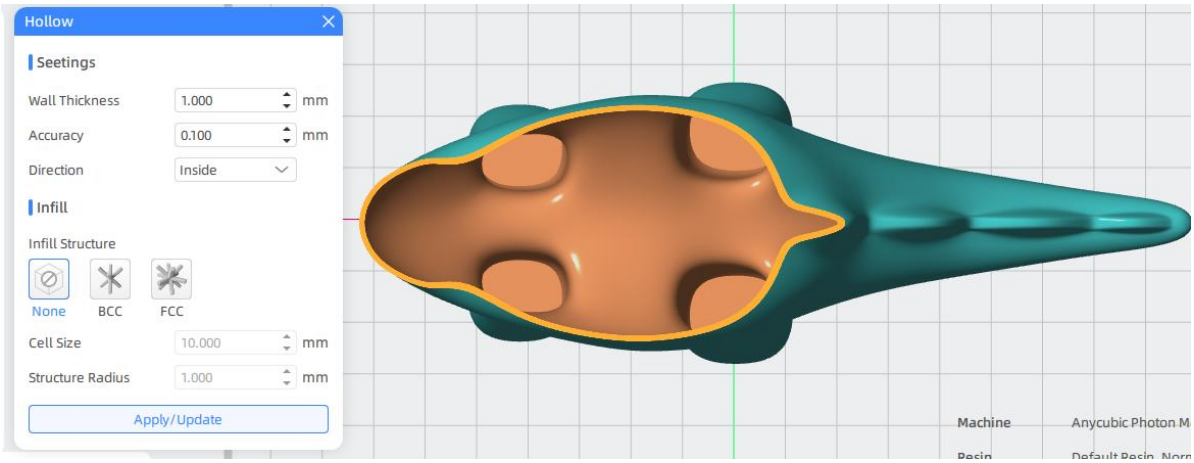
In case of large model, it is imperative to use hollow and infill settings to reduce resin consumption. We will try to learn these settings through an example.



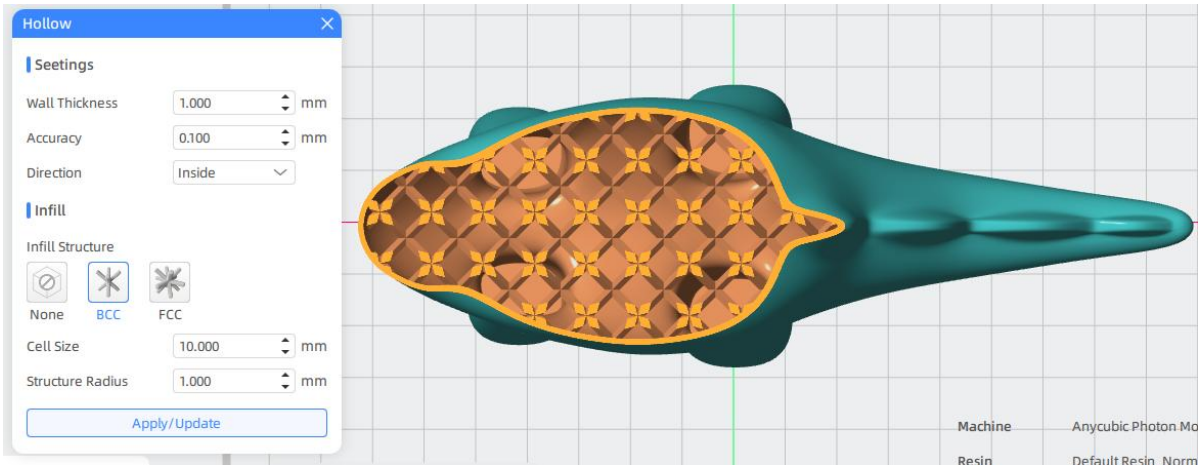
Select Hollow option from the top panel of the slicer. Hollow and infill settings will appear in a small window.



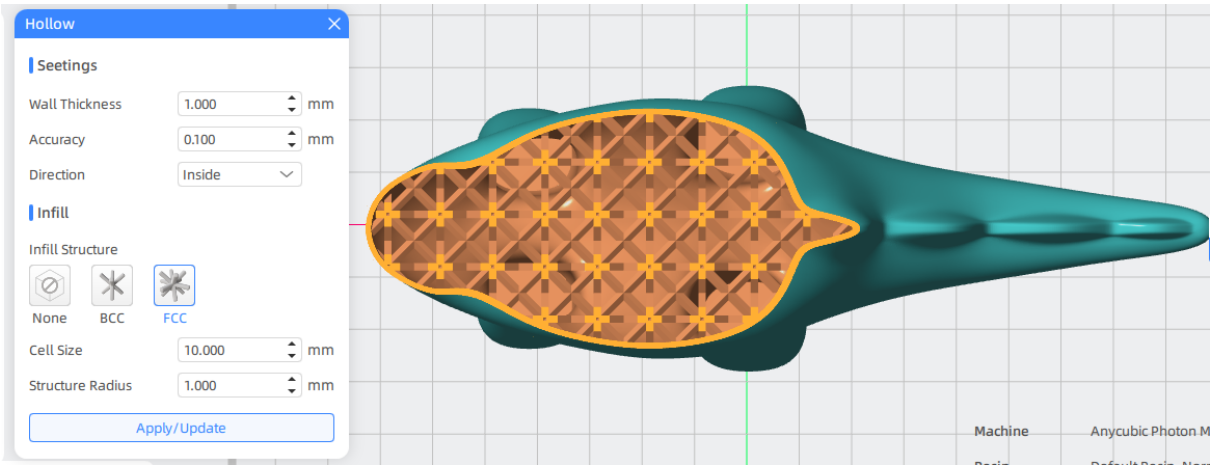
When wall thickness is 1mm and infill structure is selected “none”, the inner portion will be hollow with wall thickness 1mm.



When wall thickness is 1mm and infill structure is selected “BCC” with cell size 10mm and the structure radius 1mm, the inner portion will be as below –

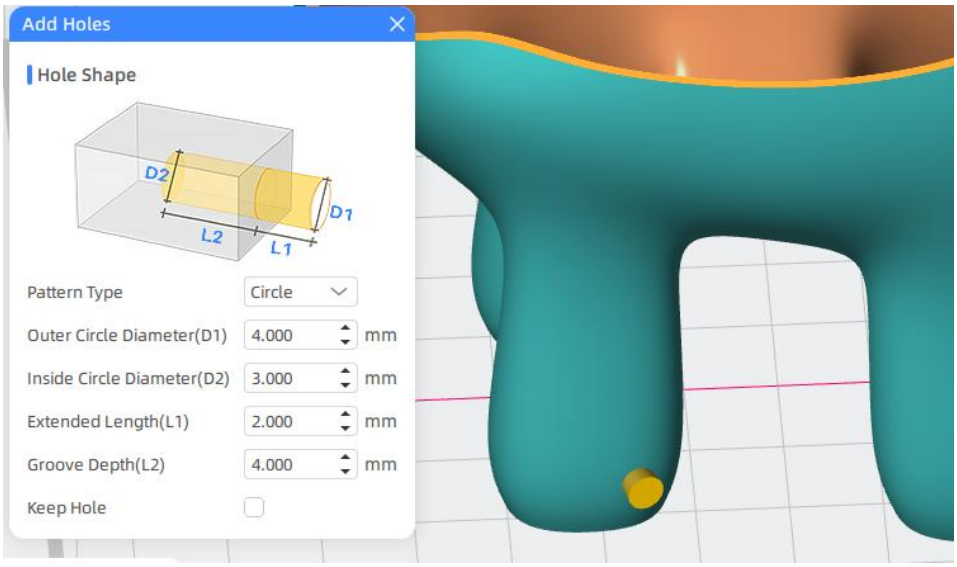


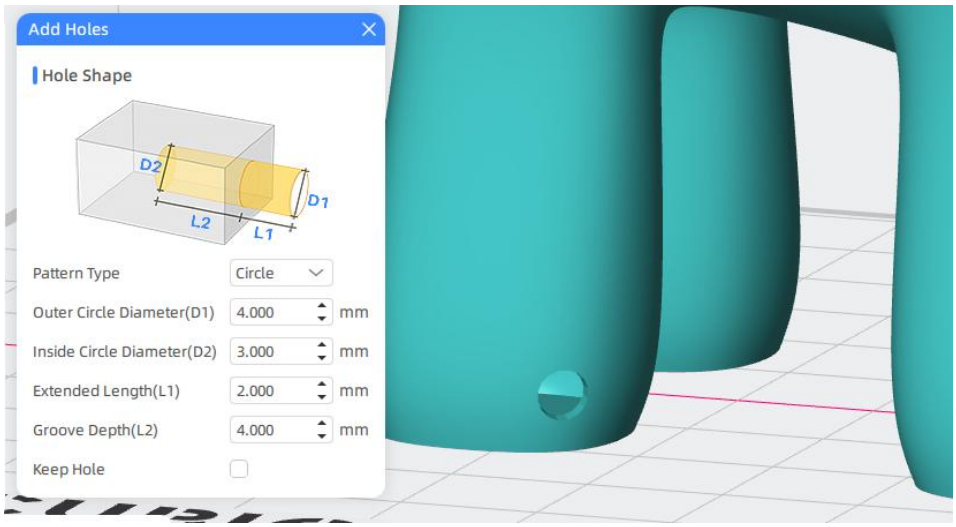
When wall thickness is 1mm and infill structure is selected “FCC” with cell size 10mm and the structure radius 1mm, the inner portion will be as below –



It is imperative to know that the larger the cell size, the lesser the infill, on the other hand, the larger the structure radius, the more the infill. BCC structure is lighter than FCC structure.

After selection of proper hollow and infill settings, a small hole must be created at the side or the bottom of the model to avoid the print failure due to vacuum drawing. When the print is completed, the resin inside the model can be discharged. To create a hole, select *Punch* option from the top panel of the slicer. It is important to note that the groove depth L2 must larger than the hollow wall thickness.


Add Holes
Hole Shape
D2
D1
L2
L1
Pattern Type
Circle
Outer Circle Diameter(D1)
4.000
mm
Inside Circle Diameter(D2)
3.000
mm
Extended Length(L1)
2.000
mm
Groove Depth(L2)
4.000
mm
Keep Hole


Add Holes
Hole Shape
D2
D1
L2
L1
Pattern Type
Circle
Outer Circle Diameter(D1)
4.000
mm
Inside Circle Diameter(D2)
3.000
mm
Extended Length(L1)
2.000
mm
Groove Depth(L2)
4.000
mm
Keep Hole

4.2.4 Job sheet: Print a 3D model with basic parameter settings

Section	Content
Job Title	Print a 3D model with basic parameter settings
Personal Protective Equipment (PPE)	Safety gloves, protective goggles, face mask and safety shoes
Equipment, Tools, and Materials	7. 3D model in STL/3MF format 8. MSLA 3D printer 9. Slicer compatible with printer 10. Standard resin 11. SD card / USB drive / Wi-Fi transfer 12. Bed cleaning tools (scraper, alcohol wipes) 13. Washing and curing station
Procedure	1. Printing program generation - Download standard 3Dbenchy STL file from any online repository - import the file in the slicer - perform orientation and manipulation as per requirement - Input the parameters' value Layer thickness – 0.05mm Normal exposure – suggested value from manufacturer Bottom exposure - suggested value from manufacturer Off time - suggested value from manufacturer Z Lift height – 8mm Z lift speed – 4 mm/s Retract speed – 5 mm/s - slice the file - save and transfer print program to a SD card/USB drive
	2. Printer Preparation - Connect the printer with power outlet and switch on the printer - Check and clean the bed and vat - level the bed and install the vat - Collect the resin and pour the resin - Connect the SD card / USB drive to the printer
	3. print and inspection - run the print program and observe the first layer adhesion - after print completion, remove the print carefully - wash and cure the printed part. - inspect the printed model

4.2.5 Job sheet: Print a 3D model with support parameter settings

Section	Content
Job Title	Print a 3D model with support parameter settings

Personal Protective Equipment (PPE)	Safety gloves, protective goggles, face mask and safety shoes
Equipment, Tools, and Materials	7. 3D model in STL/3MF format 8. MSLA 3D printer 9. Slicer compatible with printer 10. Standard resin 11. SD card / USB drive / Wi-Fi transfer 12. Bed cleaning tools (scraper, alcohol wipes) 13. Washing and curing station
Procedure	1. Printing program generation - Download standard 3Dbenchy STL file from any online repository - import the file in the slicer - perform orientation and manipulation as per requirement - Input the parameters' value Layer thickness – 0.05mm Normal exposure – suggested value from manufacturer Bottom exposure - suggested value from manufacturer Off time - suggested value from manufacturer Z Lift height – 8mm Z lift speed – 4 mm/s Retract speed – 5 mm/s - create support with medium script, anchor distance 2.5mm and Z lift distance 8mm. - use wall thickness of 1mm and BCC infill structure with cell size 8mm - punch a appropriate hole on the model - slice the file - save and transfer print program to a SD card/USB drive
	2. Printer Preparation - Connect the printer with power outlet and switch on the printer - Check and clean the bed and vat - level the bed and install the vat - Collect the resin and pour the resin - Connect the SD card / USB drive to the printer
	3. print and inspection - run the print program and observe the first layer adhesion - after print completion, remove the print carefully - wash and cure the printed part. - inspect the printed model

4.2.6 Self-check

Write down the most appropriate answers for the following questions:

1. What are two consequences of using a thinner layer thickness compared to a thicker one?
2. What is the function of the off time parameter, and when might a longer duration be necessary?
3. How do "Bottom exposure time" settings prevent print failures?
4. What is the recommended "Z retract speed" range, and when might a lower speed be required?
5. What is the mouse gesture used to rotate the build plate within the slicer interface?

6. When in the preview window after slicing, what two estimations does the software provide?
7. What is the main reason of orienting a model to an angular position for MSLA printing?
8. Why is it imperative to create a small hole when printing a hollow model?
9. In the hollow settings, how does adjusting the "structure radius" affect the infill?
10. Which infill structure (BCC or FCC) is described as being lighter?

4.2.7 Answer Key

1. Answer:

Thinner layers provide better detail/quality but significantly increase the total print time.

2. Answer:

It is a rest time for the resin to stop moving; longer rest time is needed for more viscous resins.

3. Answer:

Longer bottom exposure times create a stronger bond to the build plate, preventing the model from shifting or detaching during the print process.

4. Answer:

The recommended range is 150–400 mm/min; a lower speed is recommended for very viscous resins.

5. Answer:

The right mouse button.

6. Answer:

Estimated print time and estimated amount of resin used.

7. Answer:

To minimize the cross-sectional area of each layer to reduce print time and decrease the risk of layer failure.

8. Answer:

To avoid print failure caused by vacuum drawing and to allow uncured resin inside the model to be discharged after printing.

9. Answer:

The larger the structure radius, the more the infill.

10. Answer:

BCC structure is lighter than FCC structure.

4.3 Learning outcome: Monitor the 3D printing process

Learning Objectives:

At the end of this section, a trainee will be able to know –

1. Troubleshooting during printing

4.3.1 Information sheet: Troubleshooting during printing

Layer separation (delamination) in MSLA printing is primarily caused by insufficient adhesion between cured layers, which is most often linked to a low or inconsistent ambient temperature, incorrect exposure settings, excessive lift speeds, or improper model orientation.

Here are the key causes of layer separation:

Environmental and Material Issues

- **Low Ambient Temperature:** Resin is highly sensitive to temperature. If the environment is too cold (below the manufacturer's recommended range, often 20-30°C or 77°F-95°F), the resin becomes more viscous (thicker). This increased viscosity can prevent layers from flowing together and bonding properly, and can also lead to higher pulling forces that exceed the layer adhesion strength, causing the print to split.
- **Temperature Fluctuations:** Sudden changes in temperature during a print can cause different layers to cool and shrink at different rates, creating internal stress that pulls the part apart.
- **Expired or Poorly Mixed Resin:** Old or improperly stored resin may not cure consistently or bond effectively. Ensure the resin is well-shaken before use and stored correctly.
- **Resin Debris in the Vat:** Cured resin or other debris in the material tank can block the UV light or interfere with the build plate's movement, causing inconsistencies in layer curing and potential separation.

Printer Settings and Operation

- **Insufficient Exposure Time:** If the exposure time for regular layers is too low, the resin layers will not fully cure and bond to the previous layer, leading to a weak print that can easily delaminate.
- **Excessive Lift Speed:** If the build plate lifts too quickly from the FEP film, the force can be strong enough to pull the freshly cured layer apart from the rest of the model. Slowing down the lift speed allows for a gentler peel process.
- **Rapid Transition Between Layers:** A sharp drop in exposure time between the highly-exposed base layers and the regular layers can create a weak transition point where the layers are prone to separating. Using **transition layers** in your slicer software can help gradually reduce the exposure.

One possible remedy for layer separation is introducing transition layers along with two-stage motion control.

Transition Layer Count is also called **faded layers** in some slicers. When using transition layers, the exposure time gradually decreases over several layers, allowing a smoother shift from the prolonged exposure of bottom layers to the standard exposure used for the rest of the print.

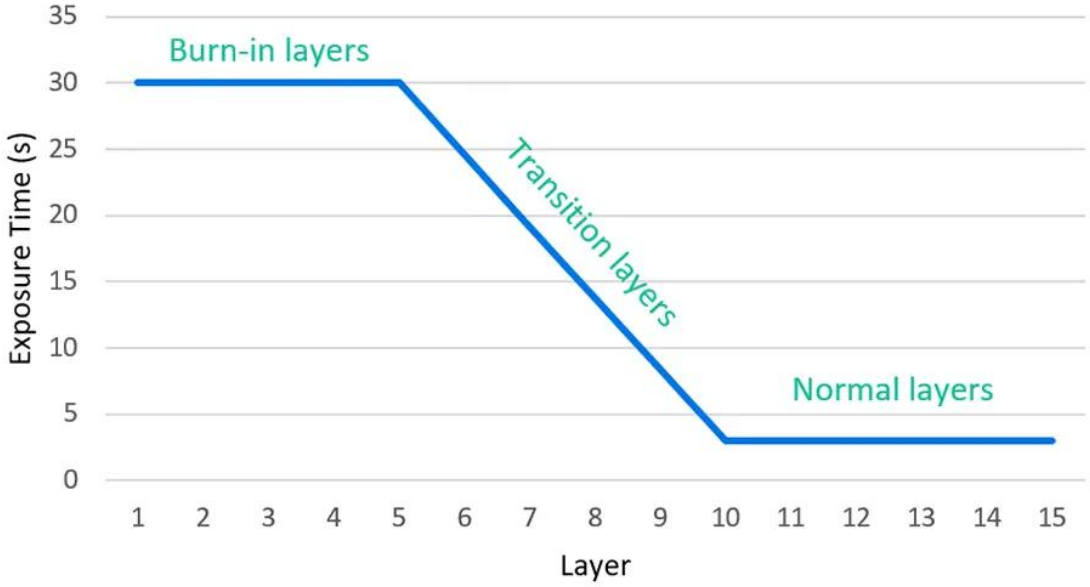


Figure 4.8: Understanding transition layers

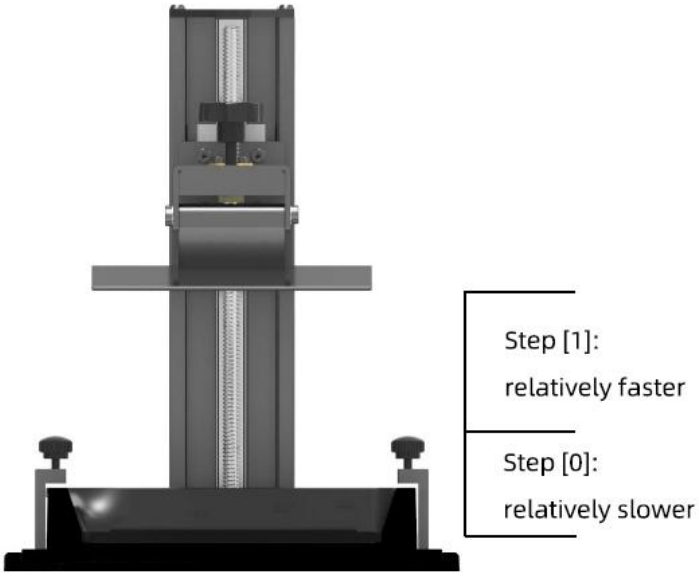
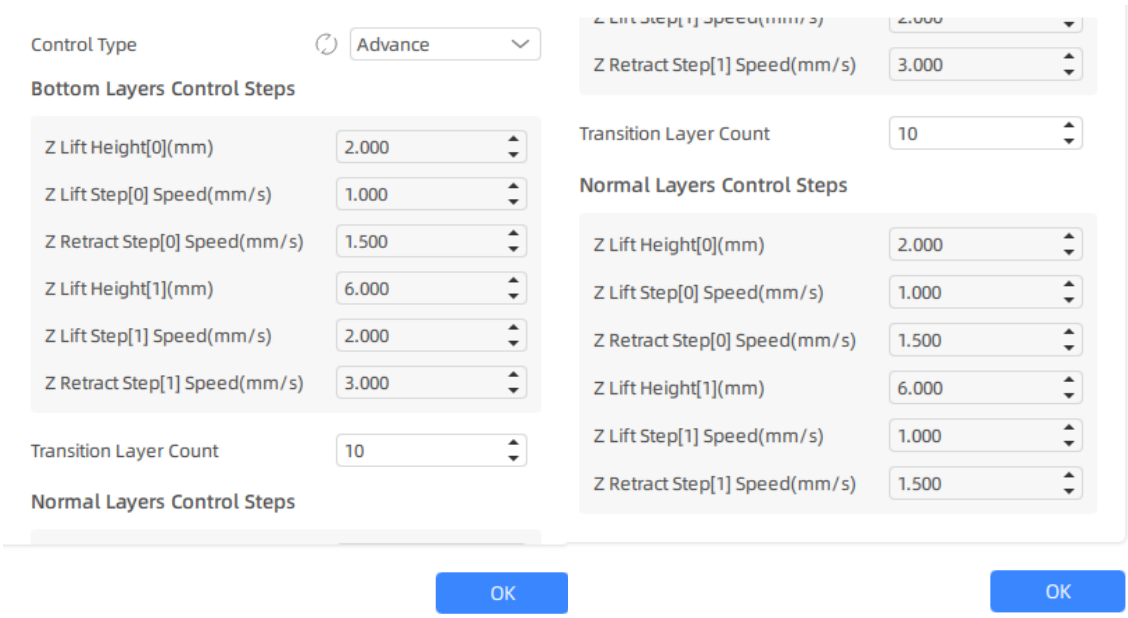


Figure 4.9: Two-stage Motion control representation

In Anycubic Photon Workshop, transition layer and two-stage motion control settings can be found in Machine > Machine Configuration Manage > Resin > Edit Resin > Control Type > Advance.



Bottom layers control: To set Z lift height, Z lift speed and Z retract speed of the bottom layers

Normal layer control: To set Z lift height, Z lift speed and Z retract speed of the normal layers

Transition layer count: The more the transition layers, the longer the time transition costs.

Step [0]: The stage when printing platform is moving near the curing face. The speed of this stage is relatively slow to avoid affecting the printing.

Step [1]: The stage when printing platform is moving away from the curing face. The speed of this stage is relatively fast to shorten the printing time.

Using these above settings, the print should be run again. To get a good result, there is a possibility of trial and error.

4.3.2 Self-check

Write down the most appropriate answers for the following questions:

1. What are two common environmental or material issues that can cause layer separation?
2. What printer setting issue can lead to insufficient adhesion between layers?
3. What is the primary purpose of introducing "transition layers" in the slicer settings?
4. In the two-stage motion control settings, what is the difference in speed between Step [0] and Step [1]?

4.3.3 Answer Key

1. Answer:

Low ambient temperature and using expired or poorly mixed resin.

2. Answer:

Insufficient exposure time for regular layers.

3. Answer:

To create a smoother shift in exposure time between the highly-exposed bottom layers and the standard exposure time for the rest of the print, avoiding a weak transition point.

4. Answer:

Step [0] is relatively slow (moving near the curing face), while Step [1] is relatively fast (moving away from the curing face).

4.4 Learning outcome: Perform post-processing of printed parts

Learning Objectives:

At the end of this section, a trainee will be able to know –

3. Washing, curing and support removal
4. Surface finishing and surface treatment

4.4.1 Information sheet: Washing, curing and support removal

Washing and curing are essential post-processing steps in MSLA printing that remove excess, uncured resin and solidify the model to achieve its full mechanical strength and desired surface finish.

Stage 1: Washing the Print

The primary goal of washing is to remove all residual, uncured liquid resin from the surface of the printed part. Failure to do this will result in a sticky, messy print with obscured details.

Materials Needed:

- **Personal Protective Equipment (PPE):** Nitrile gloves and safety glasses are mandatory.
- **Washing Solution:** Isopropyl alcohol (IPA) with 90% concentration or higher is standard. Alternatives include denatured alcohol, TPM (tripropylene glycol monomethyl ether), or specific commercial resin cleaners.
- **Containers:** Two sealable containers (or a dedicated wash station).
- **Tools:** Soft-bristled brush (like an old toothbrush), a scraper, and paper towels.

Step-by-Step Washing Process:

- **Remove from Build Plate:** Carefully use a scraper or spatula to gently remove the print from the build platform, allowing excess resin to drip back into the vat.
- **Initial "Dirty" Wash:** Place the model (supports still attached for structural integrity) into the first container of used or "dirty" washing solution. Agitate the part for 30-60 seconds to remove the bulk of the resin.
- **Second "Clean" Wash:** Transfer the print to a second container filled with fresh, clean washing solution. Agitate gently for another 1-3 minutes, using a soft brush to clean fine details and internal channels if the model is hollowed with drain holes.
- **Drying:** Remove the print from the solution and let it air dry completely in a well-ventilated area, away from direct UV light.
- **Important:** The print must be completely dry before moving to the curing stage. Any residual alcohol or moisture can cause a white, cloudy, or ashy surface finish when cured. An air gun or fan can speed up this process.

Stage 2: Curing the Print

Curing completes the polymerization process, hardening the print to its final strength and ensuring it is safe to handle without gloves.

Methods and Best Practices:

- **Dedicated UV Curing Station:** The most consistent method involves a dedicated UV curing station, often with a rotating turntable to ensure even exposure from all angles. These machines typically use a 405 nm UV light source. Cure times vary by resin type but typically range from 2 to 10 minutes.
- **Support Removal:** Supports are generally easier to remove after washing but before the final cure, as the material is still slightly pliable. Use flush cutters to snip them off close to the model to minimize scarring.

After proper washing and curing, your printed part will be hard, tack-free, and ready for optional post-processing steps like sanding, priming, and painting.

4.4.2 Information sheet: Surface finishing and surface treatment

MSLA printing naturally produces parts with an exceptionally smooth surface finish, often comparable to injection-molded parts, due to its low layer heights. However, post-processing steps (surface treatment) are necessary to achieve a professional aesthetic, remove support marks, and provide specific functional properties like durability or optical clarity.

Default Surface Finish (As Printed and Cured)

- **Appearance:** The "as-printed" surface is typically semi-transparent or matte with minimal, almost unnoticeable layer lines.
- **Irregularities:** The main imperfections are small "nibs" or connection points where support structures were attached.
- **Texture:** The surface feels very smooth to the touch, though microscopic imperfections scatter light, making clear resins appear slightly hazy.

Surface Treatment Techniques

After washing and curing, several treatments can be applied to enhance the finish.

1. Manual Sanding and Polishing

Sanding is the most common method for removing support marks and achieving a smoother, more uniform surface.

- **Process:**
  - Start with a coarse grit sandpaper (e.g., 200-400 grit) to remove visible support artifacts and major surface imperfections.
  - Progressively move to finer grits (600, 800, 1000, and up to 2000 or 3000 grit) to gradually smooth the surface and remove previous sanding scratches.
  - **Wet sanding** (using water or mineral oil) is highly recommended as it reduces dust, prevents the plastic from melting due to friction, and helps achieve a finer, smoother finish.
  - For a high-gloss, reflective finish, use a polishing compound and a soft cloth or a rotary tool with a buffing wheel after the final fine sanding stage.

2. Media Blasting (Bead Blasting)

This industrial process uses compressed air to propel fine media (like glass beads, sand, or ceramic) at the part's surface.

- **Result:** It creates a uniform, smooth, and satin-like matte finish by abrading the surface evenly. It is especially effective for complex geometries with hard-to-reach internal areas that are difficult to sand manually.

3. Priming and Painting

For custom colors or to hide minor imperfections, painting is a versatile option.

- **Process:**
  - Sand the print to a smooth finish and clean it thoroughly to ensure proper paint adhesion.
  - Apply a **filler primer** in several thin coats. The primer helps fill in minor layer lines or small holes and provides a consistent base for the paint.
  - Once the primer is dry, you can apply spray paint or acrylics in multiple thin, even layers.
  - A final clear coat or varnish can be applied to protect the paint job and provide a desired gloss or matte finish.

4. Clear Coating for Optical Clarity

For clear resin prints intended for optical applications (like lenses or fluidics), a clear coat is essential to restore transparency lost during sanding.

- **Process:** After meticulous fine sanding and polishing, a clear liquid polymer or a clear spray coating is applied. This coating fills micro-grooves and smooths out any remaining texture, maximizing light transmission and creating a glass-like appearance.

5. Electroplating

For functional parts requiring high strength, durability, or a metallic look, electroplating can deposit a thin layer of metal (like nickel or copper) onto the part's surface. This significantly strengthens the part and improves resistance to wear and UV exposure.

4.4.3 Self-check

Write down the most appropriate answers for the following questions:

1. When is the best time to remove support structures from an MSLA print, and why?
2. What is a potential result of moving a wet print (with residual IPA/moisture) directly to the curing stage?
3. What is the primary purpose of applying a "filler primer" to an MSLA print before painting?
4. Why is "wet sanding" highly recommended over dry sanding for smoothing resin prints?
5. What specific treatment is used for clear resin prints to restore the transparency that may be lost during sanding?

4.4.4 Answer Key

1. Answer:

After washing but before the final cure, because the material is still slightly pliable and less likely to scar.

2. Answer:

It can cause a white, cloudy, or ashy surface finish when cured.

3. Answer:

To help fill in minor layer lines or small holes and provide a consistent base for the paint.

4. Answer:

It reduces airborne dust, prevents the plastic from melting due to friction, and helps achieve a finer finish.

5. Answer:

Applying a clear coat or a clear liquid polymer to the surface.

Module-5: Implementing Design Principles for Additive Manufacturing (DfAM)

Module descriptor:
This unit of competency covers the knowledge, skills and attitudes required to implement design principles for additive manufacturing (DfAM). It specifically includes the tasks of understanding the principles and advantages of DfAM and applying DfAM principles to enhance product design.

Nominal Duration: 10 hours

Learning outcome:
5.1 Understand the principles and advantages of DfAM
5.2 Apply DfAM Principles to Enhance Product Design

5.1 Learning outcome: Understand the principles and advantages of DfAM

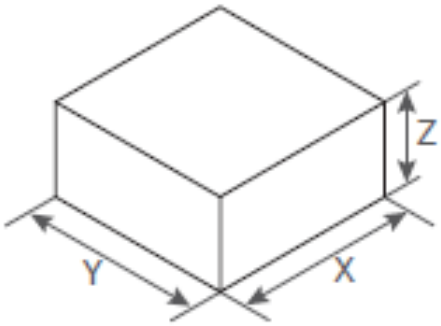
Design for additive manufacturing (**DfAM** or **DFAM**) is a general type of design methods or tools by which functional performance, manufacturability, reliability, and cost can be optimized subjected to the capabilities of additive manufacturing technologies.

All of the following principles differ greatly for each technology category. Some are not a concern; others are a major concern. Before designing for AM, one needs to know which process s/he is designing for, and if possible, what machine it will be built upon. Each machine and even different materials differ on some of these aspects.

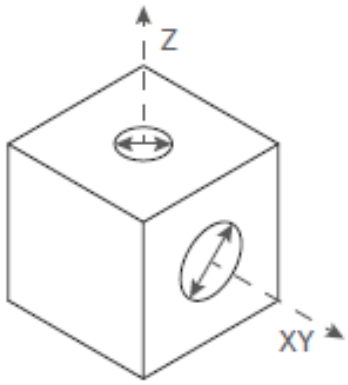
5.1.1 Information sheet: DfAM related to process constraints

Feature size

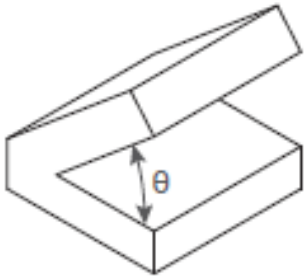
a) The minimum producible feature size
   i. Minimum part dimensions



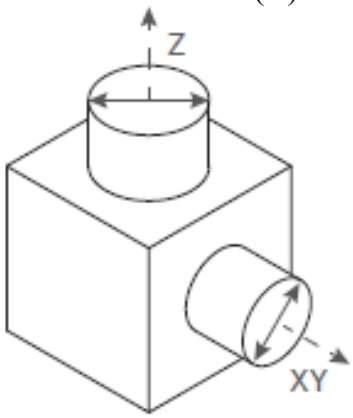
   ii. Minimum hole diameter

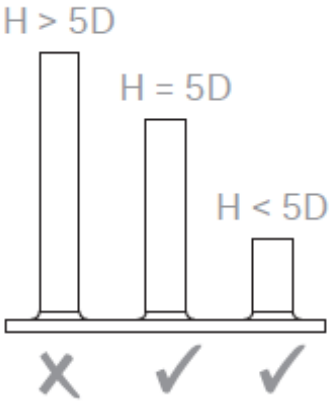


iii. Minimum unsupported overhang angle



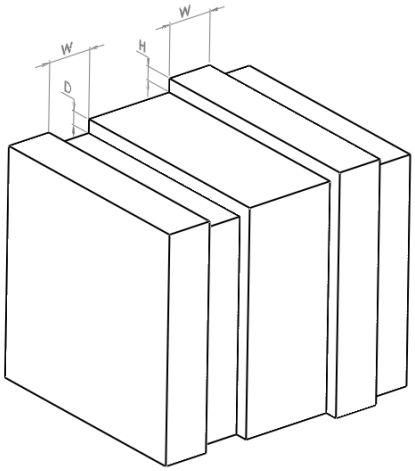
iv. Minimum post diameter (**Important note**: Avoid printing posts with heights (H) more than five times their diameter (D). Tall posts are more susceptible to shear on layer lines.)





v. Minimum engraved feature size

vi. Minimum embossed feature size



vii. Minimum wall thickness

viii. Minimum clearance between moving parts

b) Machine's minimum X, Y, and Z resolution
c) Dimensions of print must be multiple of layer height.
d) Nozzle diameter (in case of FDM printing)
e) Pixel size of LCD (in case of LCD based SLA / MSLA)

5.1.2 Information sheet: DfAM related to support and orientation

Supports, Overhangs & Orientation

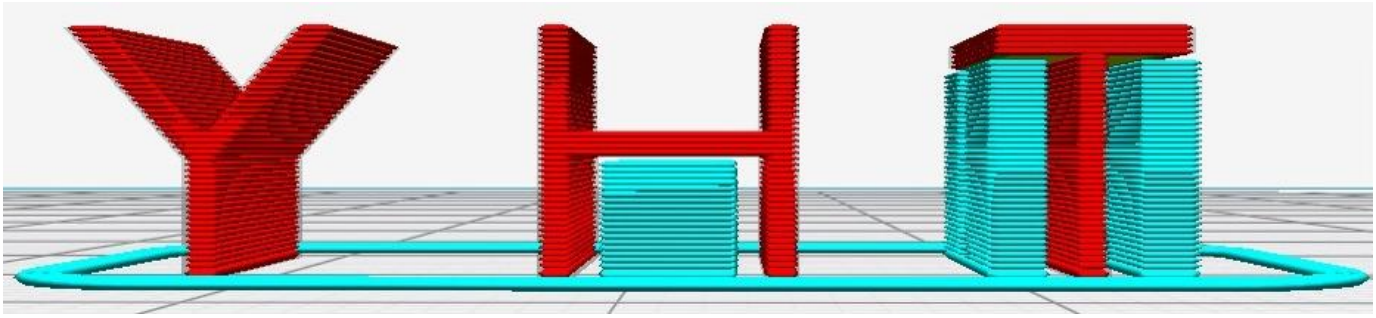


Figure 5.1: Overhangs, Bridging and Supports

a. How supports are designed or generated and removed needs to be thought of in the design process.
b. Generally, there is a critical angle (typically 45 degrees) that allows no support to be needed such as in the letter Y.

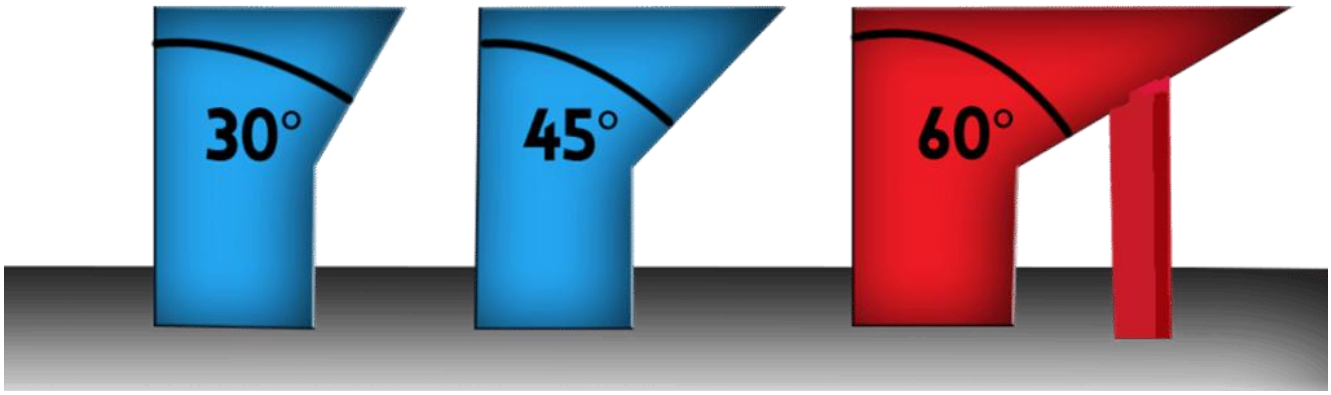


Figure 5.2: Overhangs and criteria for applying support

c. Some need supports for all bridges of a certain length such as the middle of a capital H, but it's optional depending on the quality and usage.
d. Others need supports for overhangs such as at the ends of a capital T. By changing or re-orientating the design, one can minimize the need for supports, and change how the supports are removed. From the left of figure given below:
    i. Large overhangs requiring extensive support material (shown in blue)
    ii. Modified design with additional tapered material to reduce supports, increasing the part mass and possibly requiring post-process machining / wire erosion

iii. Angled at 45 degrees - mostly self-supporting except for one local minimum. Down-skins and up-skins will exhibit different surface roughness
iv. Inverted with short supports under the bottom face - shorter build time, but post-process finishing of the supported face will be needed
v. Solid attachment to the bed with a stock allowance for EDM removal (in case of metal printing) - residual stress could be a concern here
vi. A similar approach, but with smaller attachment regions to reduce stress build-up - this is likely to be the most efficient design from a manufacturing standpoint
vii. A final alternative (not shown) is to lay the part flat on the plate. This reduces the build height, but also limits the number of parts that can be nested on the build plate, and will be prone to greater residual stress.

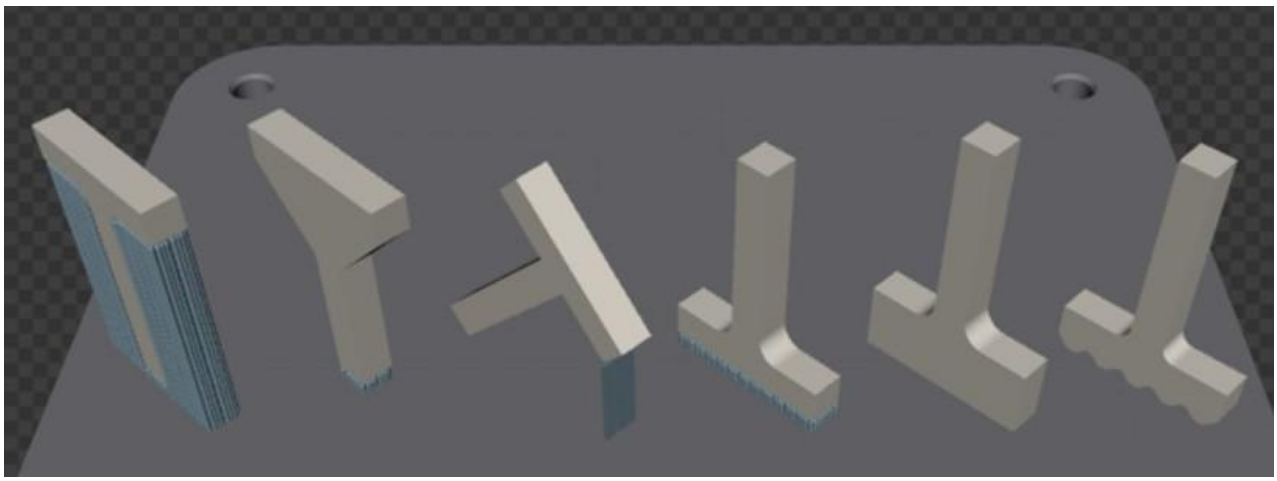


Figure 5.3: Same part in different orientation

e. Whilst a horizontal overhang of 0.3 – 1 mm can self-support, this is not recommended. Meanwhile, overhangs of more than 1 mm will definitely need re-design or support. Fillets and chamfers can be added to components to eliminate overhangs.

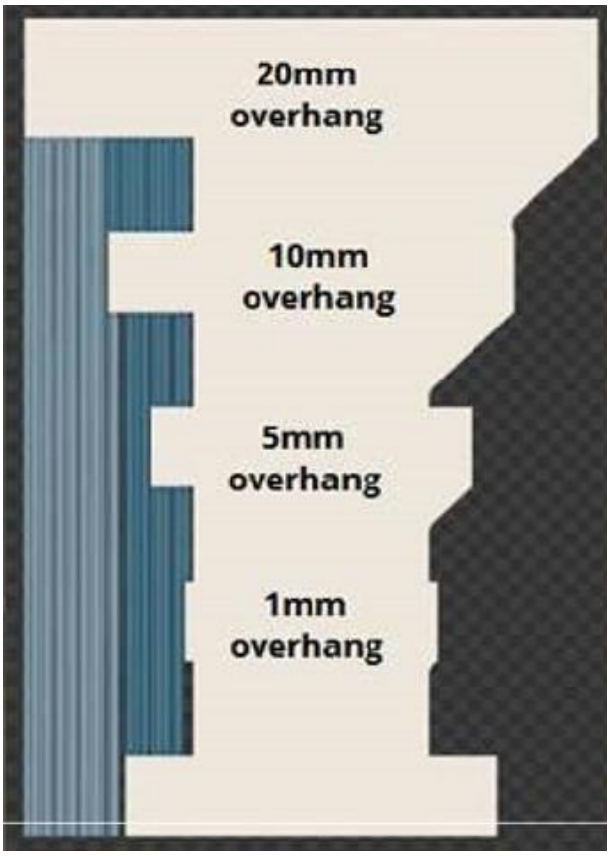


Figure 5.4: Different size of overhangs

f. Primary supports are those that are developed using CAD software along with the component, and designed as sacrificial structures that will be removed once the build is finished. These supports are in the control of the designer. Secondary supports are created within slicing software and can be managed via parameters, but lack control.

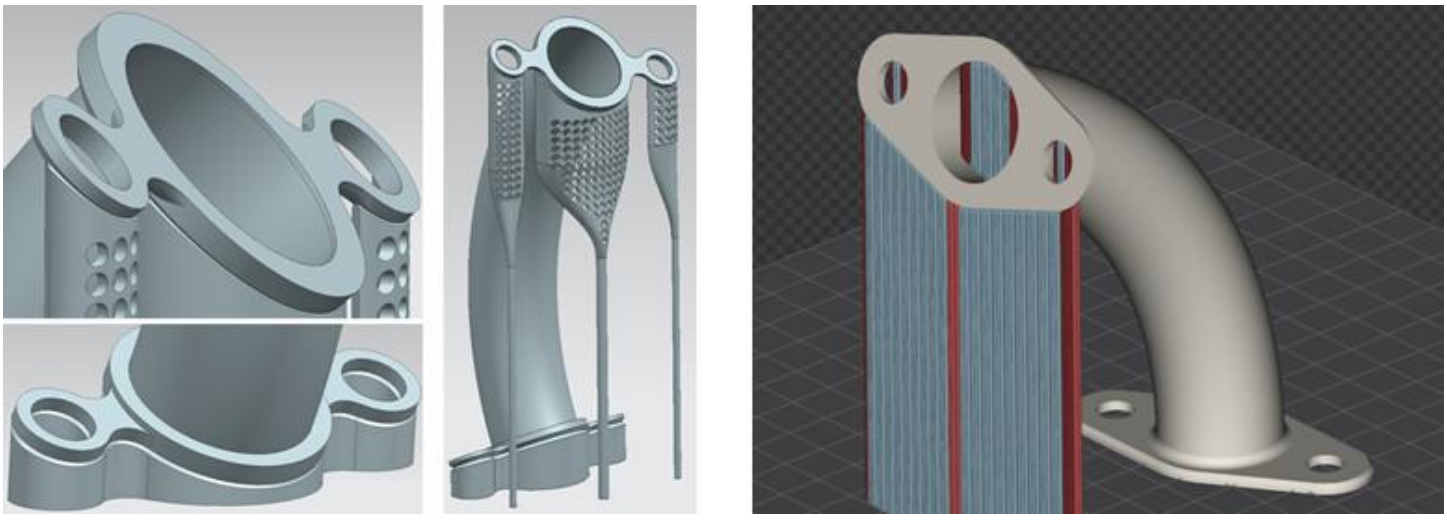


Figure 5.5: Primary supports, developed in CAD (left) and secondary supports, developed in slicing software (right).

5.1.3 Information sheet: DfAM related to Hole orientation and redesign

a) The best orientation to get a perfect hole is in horizontal position, or parallel to the build surface.
b) Lateral holes or holes on the wall needs supports to be printed. Even after using supports, these holes tend to distort or be pressed down. Since lateral holes are unlikely to be perfectly round, it often makes sense to change their shape so that they are self-supporting. In some cases, a teardrop or diamond shape may be acceptable for the finished feature.

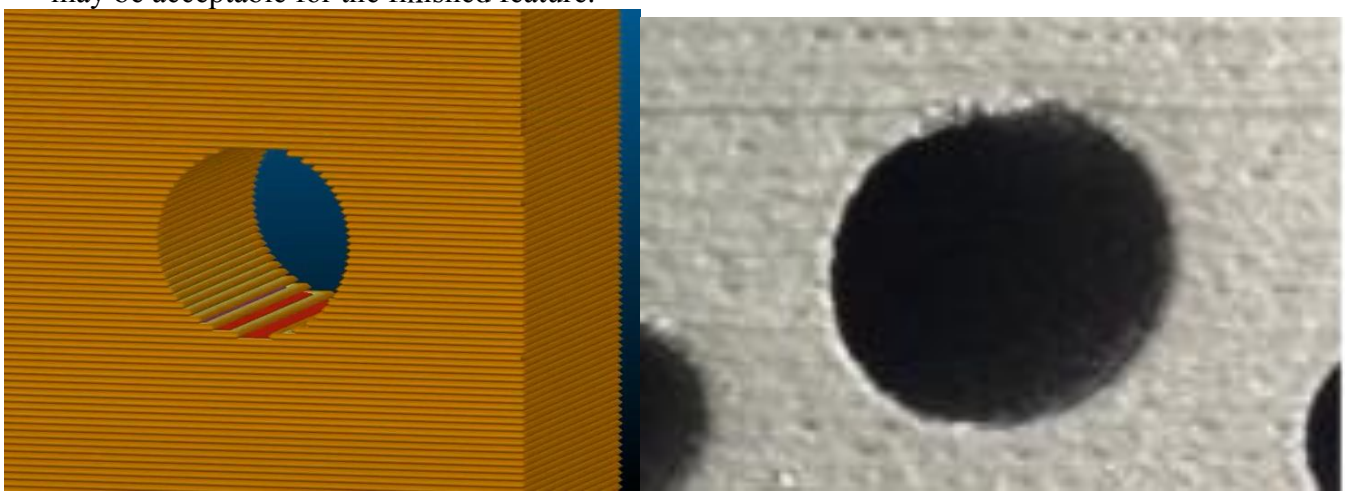


Figure 5.6: Lateral holes tend to distort or be pressed down

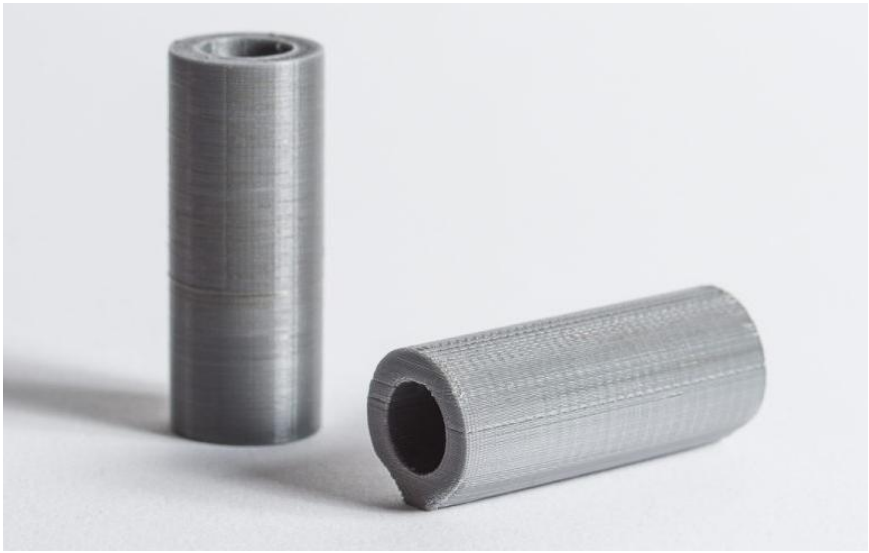


Figure 5.7: Wrong orientation results poor quality and appearance.

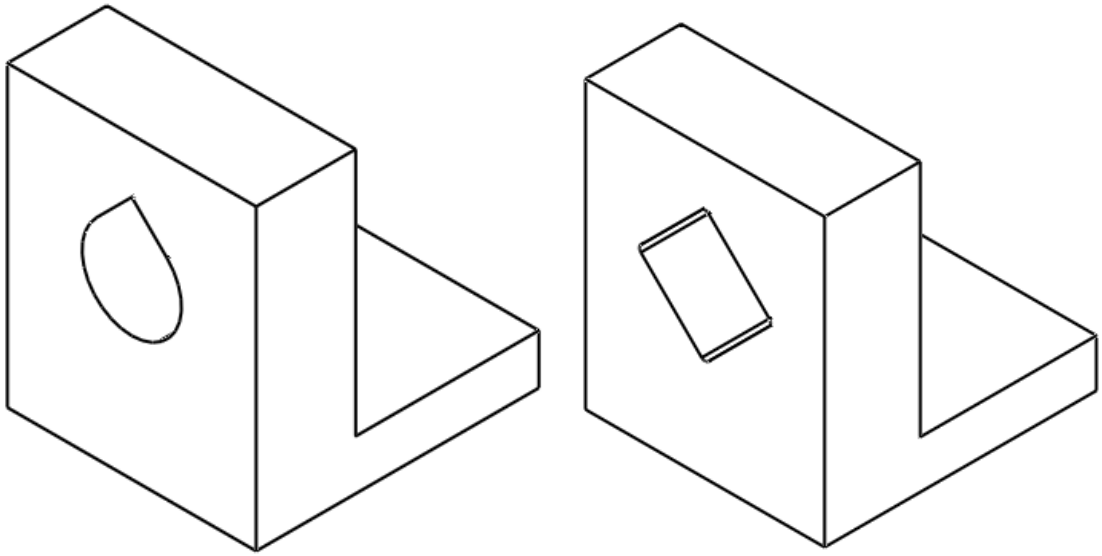


Figure 5.8: Teardrop shaped or diamond shaped hole

c) To achieve tolerance for push fit, two hole-designs given below known as tightening or compliant mechanisms can be considerable solutions.

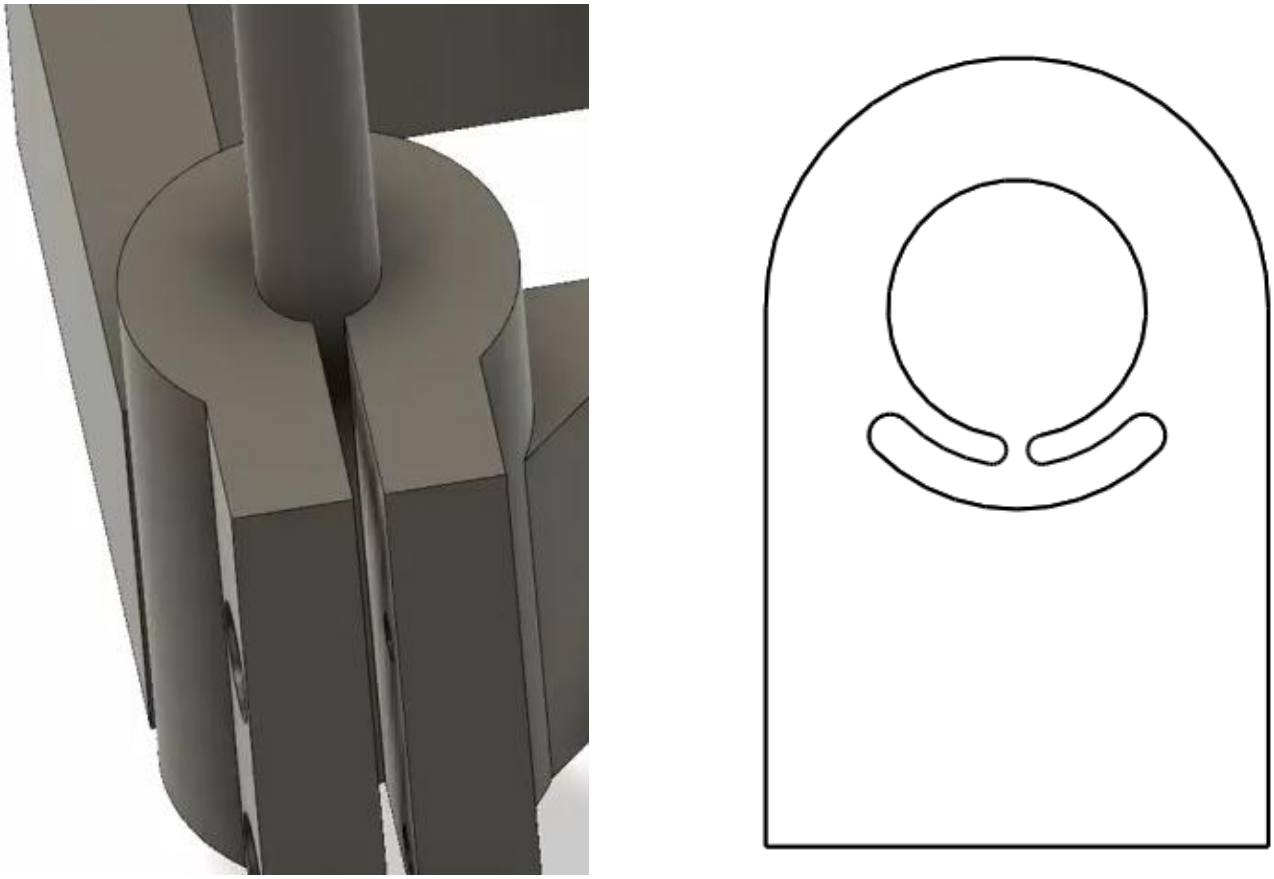
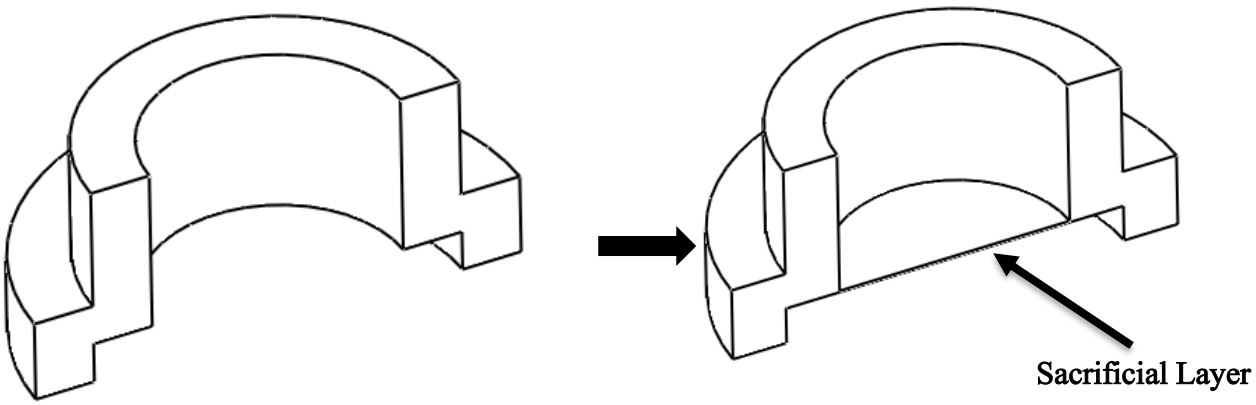


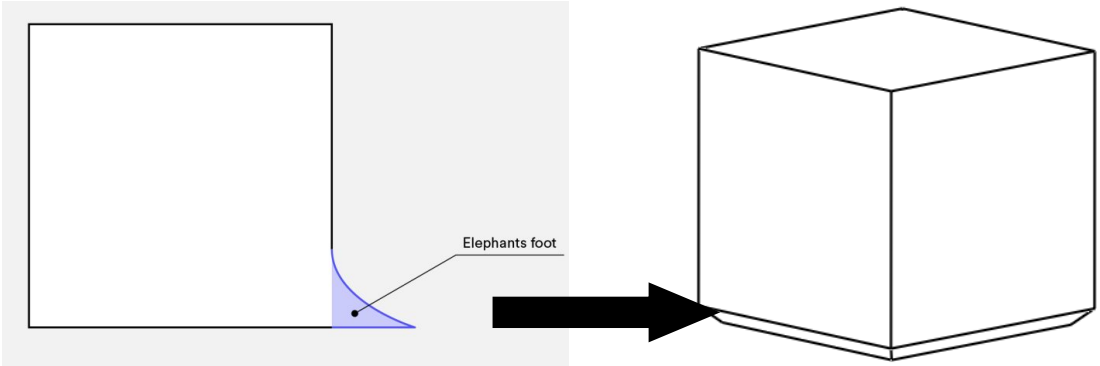
Figure 5.9: Hole design for fit

d) To print without support, horizontal holes can be redesigned with a sacrificial layer as below.



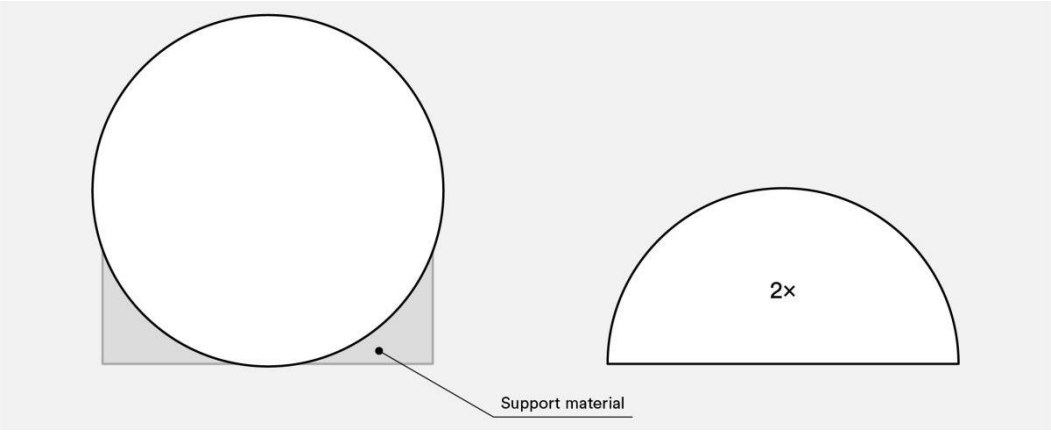
5.1.4 Information sheet: Miscellaneous DfAM Principles

Use chamfer to avoid elephant's foot



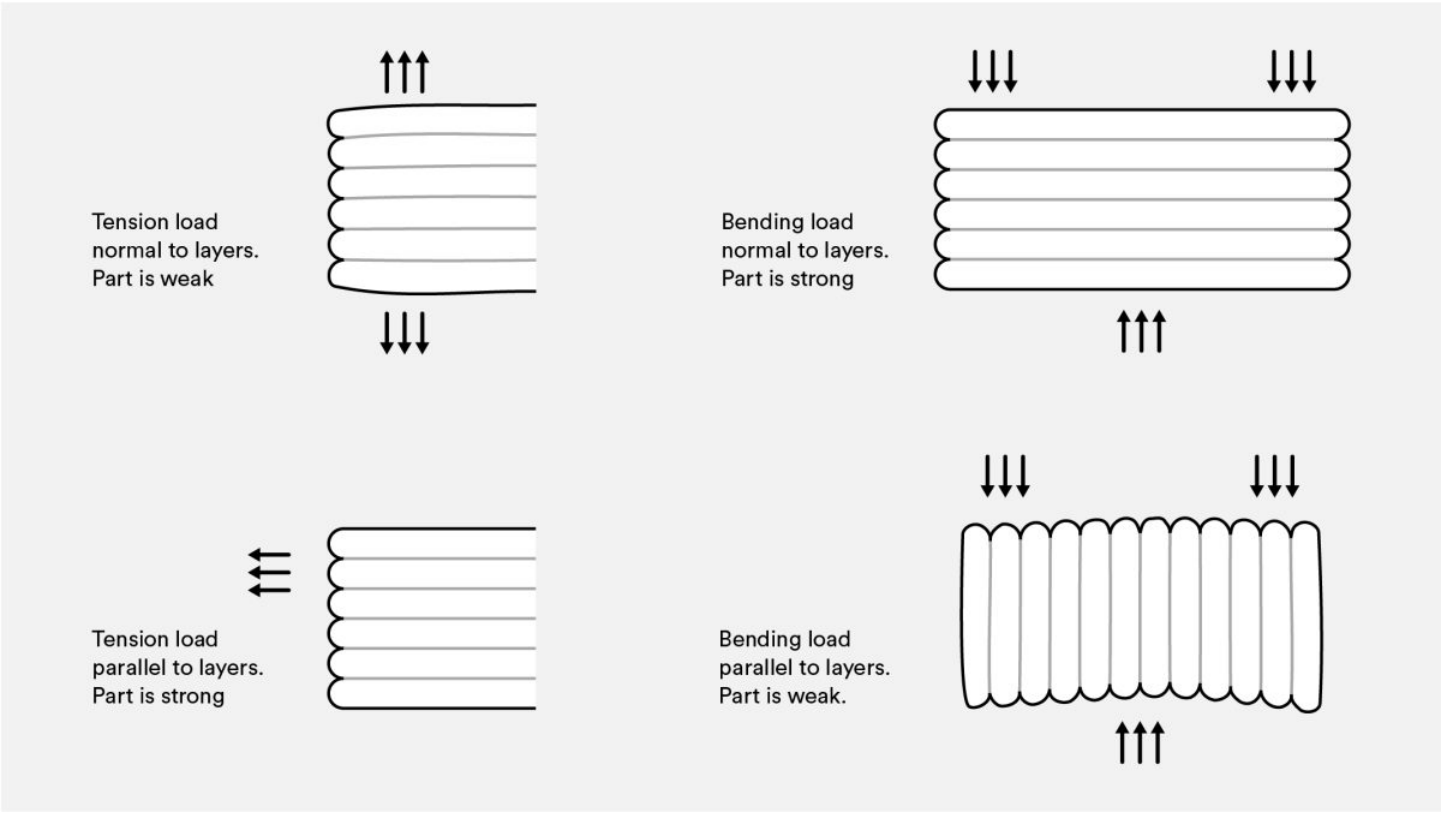
Splitting model

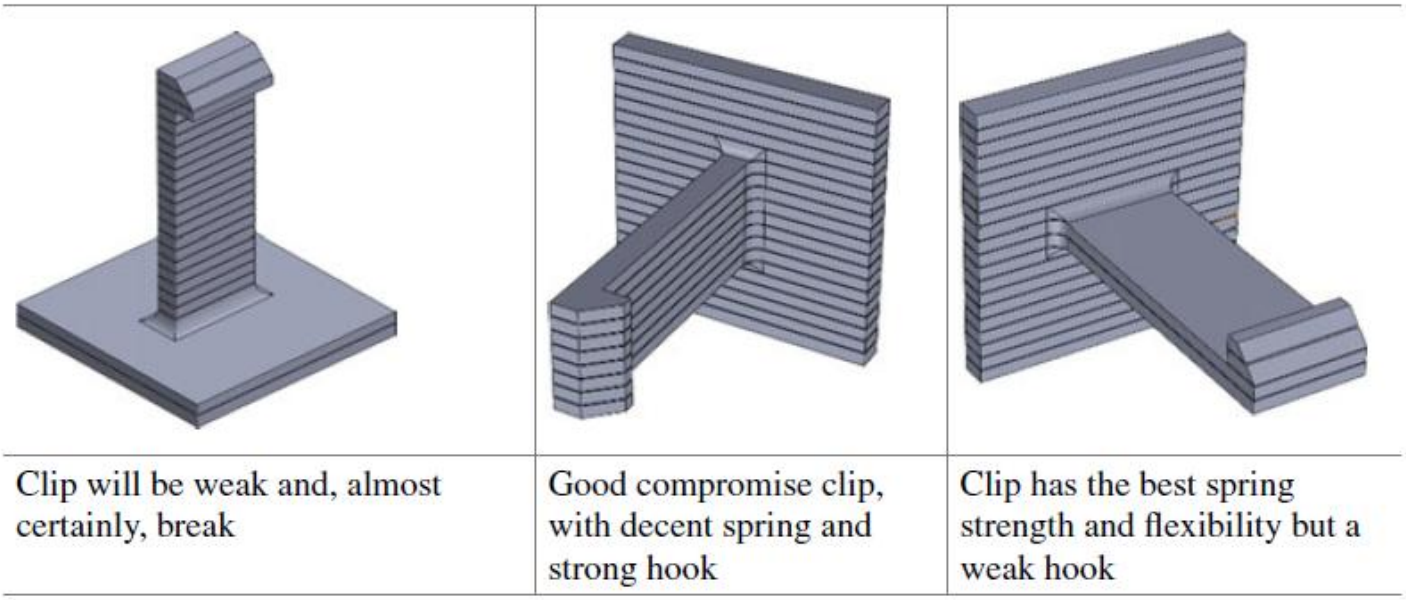
Splitting a model can often reduce its complexity, saving costs and time. Overhangs that require a large amount of support may be removed by simply splitting a complex shape into sections that are individually printed. If desired, the sections can be glued together once printing is completed.



Orientation vs. Mechanical Property

Due to the anisotropic nature of FDM printing, understanding the application of a component and how it is built are critical to the success of a design. FDM components are inherently weaker in one direction due to layer orientation.





Clip will be weak and, almost certainly, break	Good compromise clip, with decent spring and strong hook	Clip has the best spring strength and flexibility but a weak hook

Figure 5.10: Orientation vs. Mechanical Property

5.1.5 Information sheet: Advanced DfAM methods

Parts consolidation

Due to the constraints of traditional manufacturing methods, some complex components are usually separated into several parts for the ease of manufacturing as well as assembly. This situation has been

changed by the using of additive manufacturing technologies. Some case studies have been done to shows some parts in the original design can be consolidated into one complex part and fabricated by additive manufacturing processes. This redesigning process can be called as parts consolidation. The research shows parts consolidation will not only reduce part count; it can also improve the product functional performance. The design methods which can guide designers to do part consolidation can also be regarded as a type of DfAM methods.



Figure 5.11: 3d Printed Jet Engine Fuel Nozzle (right). Before 3D printing, this fuel nozzle (left) had 20 different pieces. Now, just one part, the nozzle (right) is 25% lighter and five times more durable. Source: GE Aviation.

Topology optimization

Topology optimization is a type of structural optimization technique which can optimize material layout within a given design space. Compared to other typical structural optimization techniques, such as size optimization or shape optimization, topology optimization can update both shape and topology of a part. Additive manufacturing processes can be applied to fabricate these topology optimization results. However, it should be noticed, some manufacturing constraints such as minimal feature size also need to

be considered during the topology optimization process. Since the topology optimization can help designers to get an optimal complex geometry for additive manufacturing, this technique can be considered one of DfAM methods.

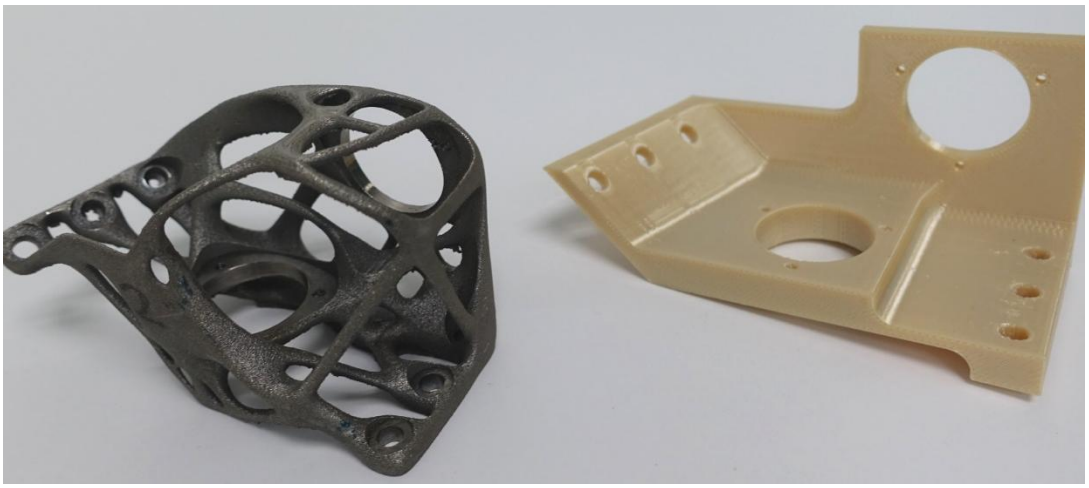


Figure 5.12: A bracket redesigned & 3D printed for the U.S. Army Aviation and Missile Research Development and Engineering Center (AMRDEC)



Figure 5.13: Conventional design of the steel cast bracket (left) that was environmentally assessed against the corresponding topology-optimized design of the EOS titanium AM-made bracket (right). Source: Airbus Group Innovations.

5.1.6 Self-check

Write down the most appropriate answers for the following questions:

1. Why should DfAM principles be followed in 3D printing?
2. What is a primary support and a secondary support?
3. Why does orientation affect mechanical property of an FDM printed object?

5.1.7 Answer Key

1. Answer:

Because by using DfAM, the functional performance, manufacturability, reliability, and cost can be optimized.

2. Answer:

Primary supports are those that are developed using CAD software along with the component, and designed as sacrificial structures that will be removed once the build is finished.
Secondary supports are created within slicing software and can be managed via parameters.

3. Answer:

Due to the anisotropic nature of FDM printing. FDM components are inherently weaker in one direction due to layer orientation.

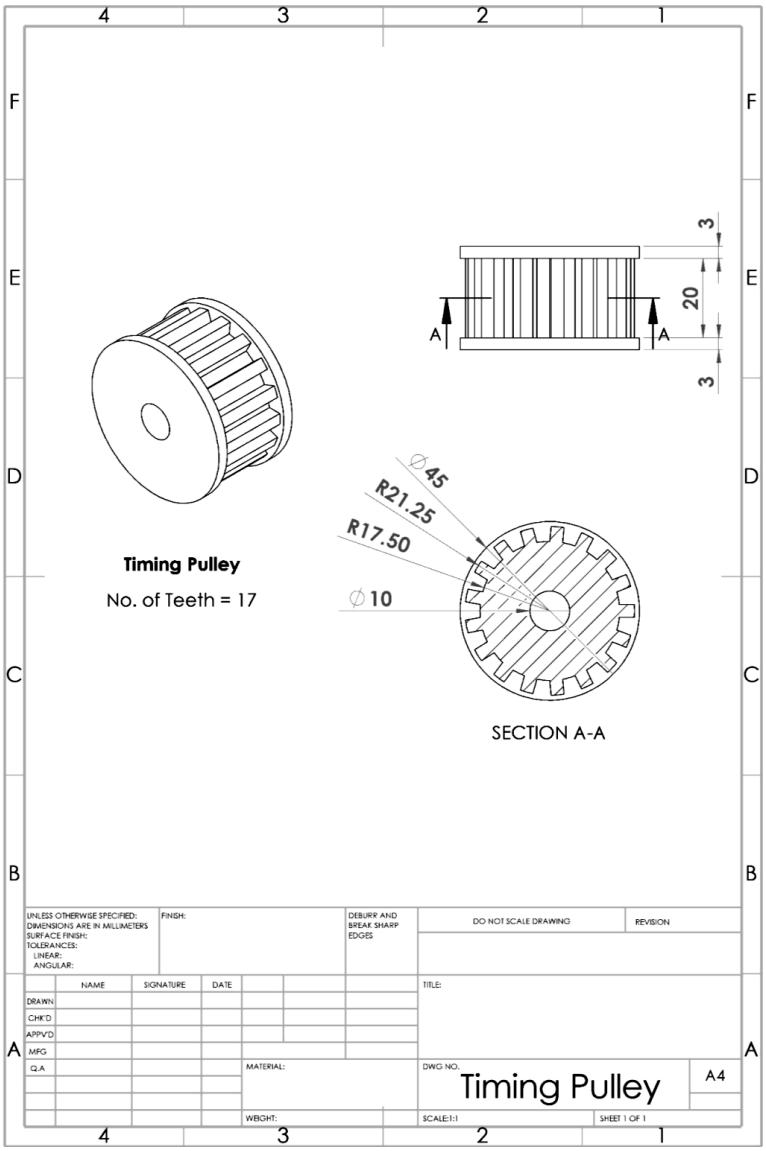
5.2 Learning outcome: Apply DfAM Principles to Enhance Product Design

5.2.1 Job sheet: Re-design a 3D model by modifying overhang angle and print without support

Section	Content
Job Title	Re-design a 3D model by modifying overhang angle and print without support
Personal Protective Equipment (PPE)	Safety gloves, protective goggles and safety shoes

Equipment, Tools, and Materials	14. 3D model drawing sheet (provided) 15. FDM 3D printer (PLA/ABS/PETG compatible) 16. Slicer compatible with printer 17. PLA Filament preferable 18. SD card / USB drive / Wi-Fi transfer 19. Bed cleaning tools (scraper, alcohol wipes)
Procedure	1. Design and re-design - first design the model as per drawing sheet - re-design the model in such a way that there will be no overhang angle = 45 degree - convert the model into STL format
	2. Printing program generation - import the STL file in the slicer - perform orientation and manipulation as per requirement - Input the parameters' value Layer height – 0.2mm Perimeters – 2 Infill – 20% Infill pattern – Gyroid Nozzle temperature - suggested value from slicer Bed temperature – suggested value from slicer Other parameter – default value - slice the file - save and transfer the print program to a SD card / USB drive
	3. Printer Preparation - Connect the printer with power outlet and switch on the printer - Collect the filament spool - Cut the filament in 45 degree approximately - Clean the heatbed and nozzle tip - preheat the nozzle for the collected filament from Preheat option of the printer LCD panel - connect the SD card / USB drive to the printer
	4. Load the filament - place the filament spool on the spool holder - insert the filament tip through the PTFE tube just before the extruder - Run Unload Filament to take out the existing filament - Run AutoLoad Filament option from the LCD panel
	5. print and inspection - run the print program - observe the first layer adhesion - after print completion, remove the print carefully - inspect the printed model

Drawing Sheet

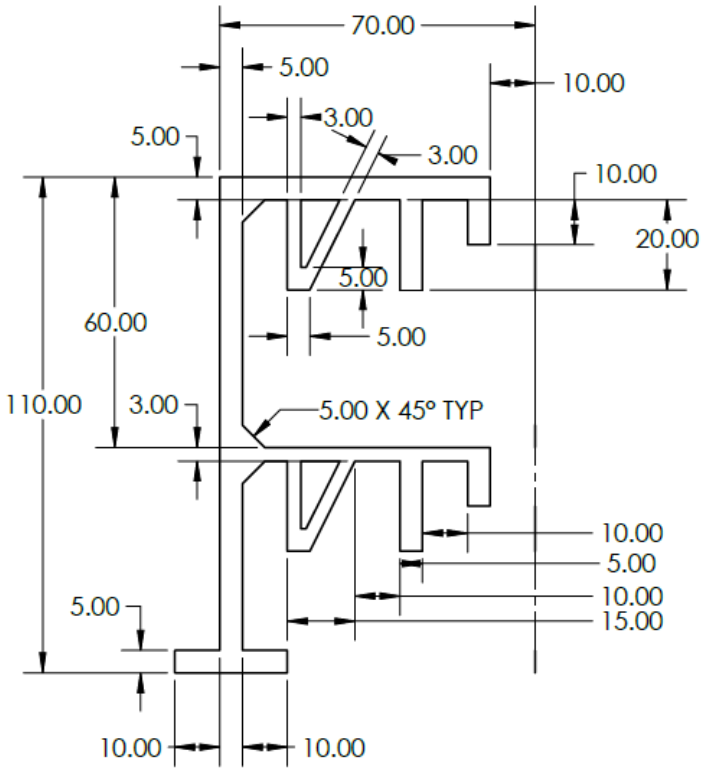


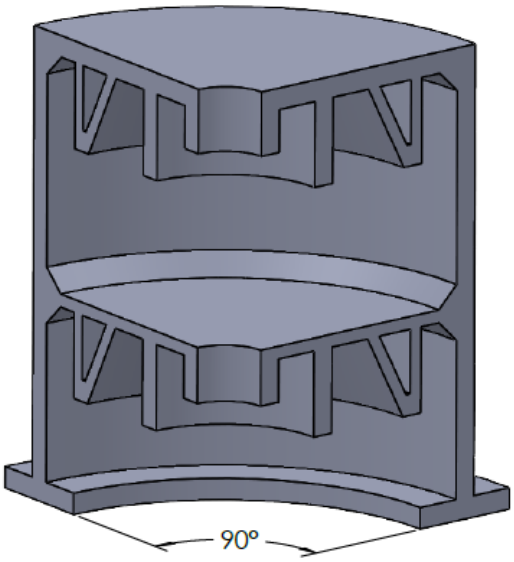
5.2.2 Job sheet: Re-design a 3D model by splitting and print without support

Section	Content
Job Title	Re-design a 3D model by splitting and print without support
Personal Protective Equipment (PPE)	Safety gloves, protective goggles and safety shoes
Equipment, Tools, and Materials	1. 3D model drawing sheet (provided) 2. FDM 3D printer (PLA/ABS/PETG compatible) 3. Slicer compatible with printer 4. PLA Filament preferable 5. SD card / USB drive / Wi-Fi transfer 6. Bed cleaning tools (scraper, alcohol wipes)
Procedure	1. Design and re-design - first design the model as per drawing sheet - re-design the model by splitting into three segments, - convert the segments into STL format
	2. Printing program generation - import the STL files in the slicer

	- perform orientation and manipulation as per requirement - Input the parameters' value Layer height – 0.2mm Perimeters – 2 Infill – 20% Infill pattern – Gyroid Nozzle temperature - suggested value from slicer Bed temperature – suggested value from slicer Other parameter – default value - slice the file with "Add Custom G-code" setting and a use G4 code to pause and assemble the segments mid-print. - save and transfer the print program to a SD card / USB drive
	3. Printer Preparation - Connect the printer with power outlet and switch on the printer - Collect the filament spool - Cut the filament in 45 degree approximately - Clean the heatbed and nozzle tip - preheat the nozzle for the collected filament from Preheat option of the printer LCD panel - connect the SD card / USB drive to the printer
	4. Load the filament - place the filament spool on the spool holder - insert the filament tip through the PTFE tube just before the extruder - Run Unload Filament to take out the existing filament - Run AutoLoad Filament option from the LCD panel
	5. print and inspection - run the print program - observe the first layer adhesion - after print completion, remove the print carefully - inspect the printed model

Drawing Sheet





(All Dimensions are in mm)

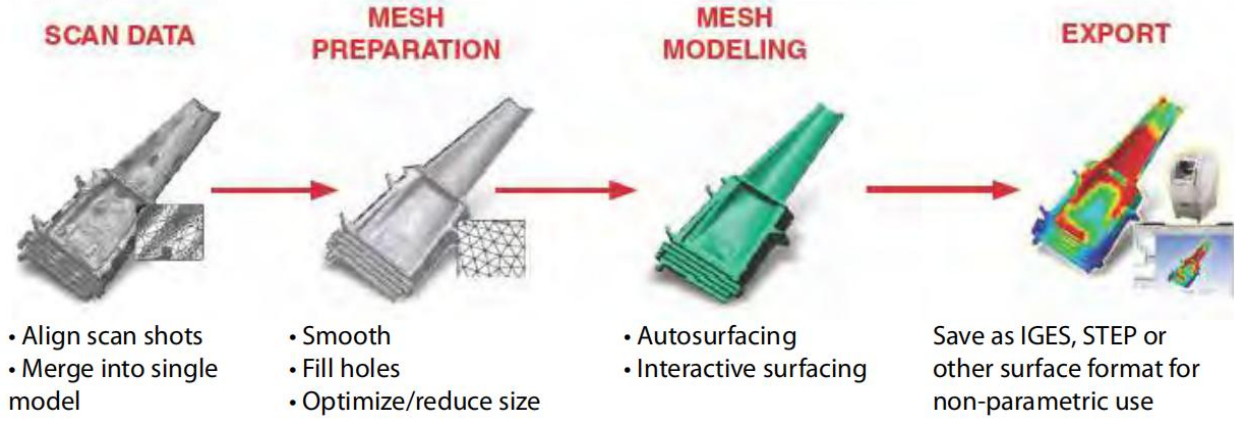
Module-6: Perform Prototyping Using 3D Scanning Technology

6.1 Learning Outcome: Conduct 3D scanning of physical objects

1. Information Sheet: Reverse Engineering (RE)

Reverse engineering (RE) is the process of reversing the typical steps of product design and manufacturing. Instead of starting with a design, RE begins with an already manufactured physical item and uses it to derive the necessary design information.

The main goal of RE physical parts is to create a **3D CAD (computer-aided design) model** from a representative sample piece. This resulting CAD model can then be used to manufacture more parts, or for documentation and 3D printing. For reverse engineering, **3D scanning** is often the preferred method because it can easily obtain **100 percent coverage** and capture fine details, making it ideal for non-geometric or complex contours like aircraft airfoils. This technique also gathers a large amount of data quickly, enhancing productivity.



1. Information Sheet: When Reverse Engineering is Necessary

RE is primarily needed when a 3D CAD model for a physical part does not exist or is unavailable to the manufacturer. By using RE, manufacturers can avoid designing a part from scratch, preventing the need to "re-invent the wheel".

Common scenarios requiring reverse engineering include:
• **Legacy and Historic Parts:** Creating a CAD model for old or historic designs that were never originally modeled in CAD.
• **Unavailable Intellectual Property:** Acquiring mounting and clearance geometry for aftermarket parts (e.g., automotive add-ons) when the original manufacturer (OEM) will not provide the intellectual property.
• **Maintenance and Repair:** Developing solutions for broken components in older aircraft (AOG condition - aircraft on the ground) or industrial machinery where replacement parts are no longer in production.

• **Service Life Extension:** Supporting programs (like SLEP for military aircraft) intended to keep aging equipment operational longer than originally planned.

1. **Information Sheet: 3D Scanning**

3D scanning is a technique that uses a 3D scanner to capture the shape of a physical subject, which can be an object, an environment, or a person. The scanner collects data from the surface of the physical object to accurately describe its shape in a digital, three-dimensional format. The result is a 3D file of the object that can be saved, edited, and 3D printed.



1. **Information Sheet: Point Cloud and Texture**

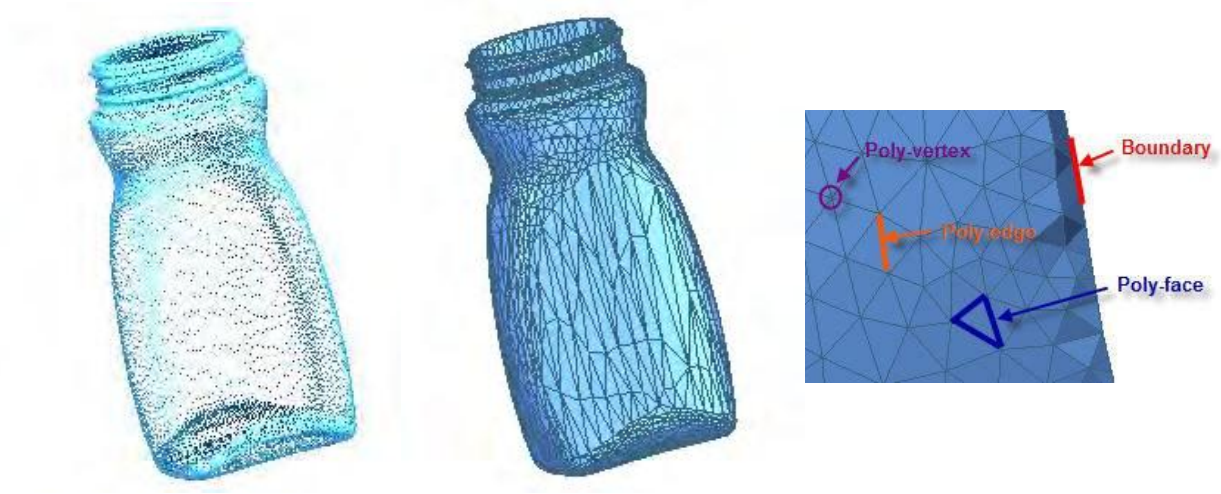
3D scanning typically collects millions of X-Y-Z data points that are processed into a **"point cloud,"** creating a detailed digital twin of the object. Some scanners can also simultaneously collect color data, known as **texture**.



Point Cloud with Textures

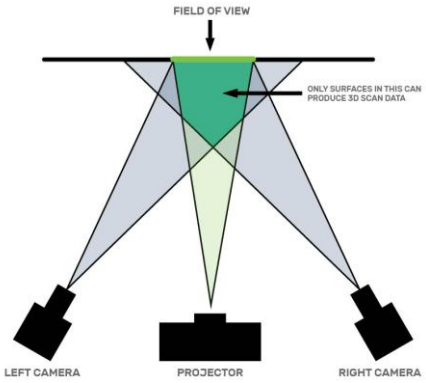


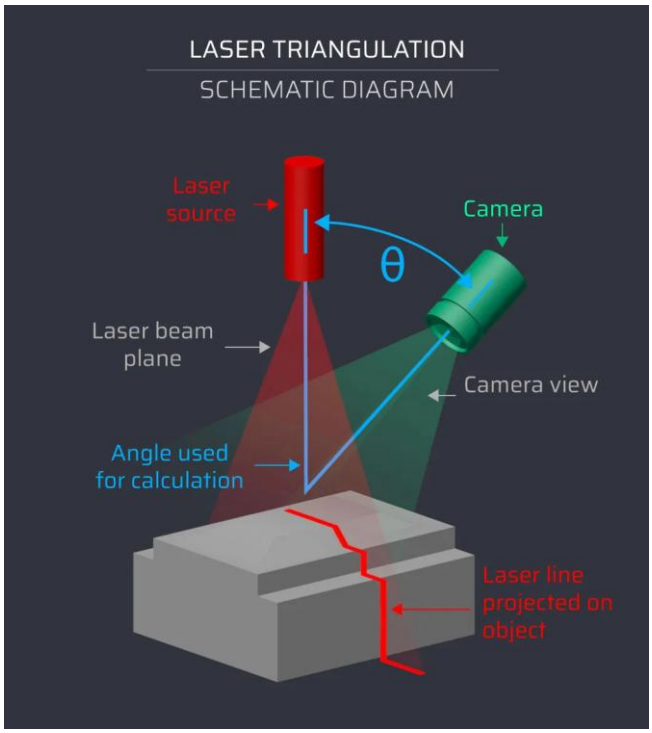
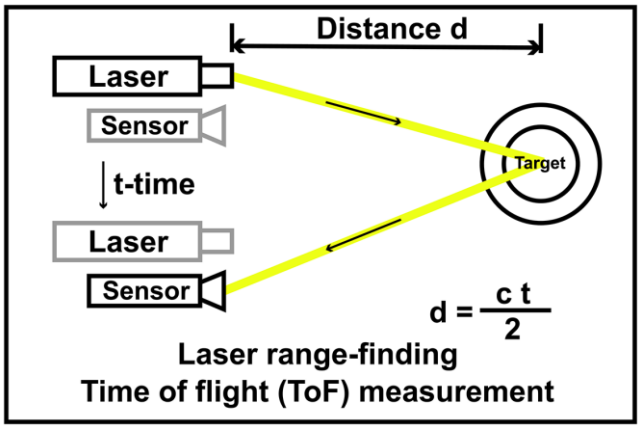
Point Cloud without Textures

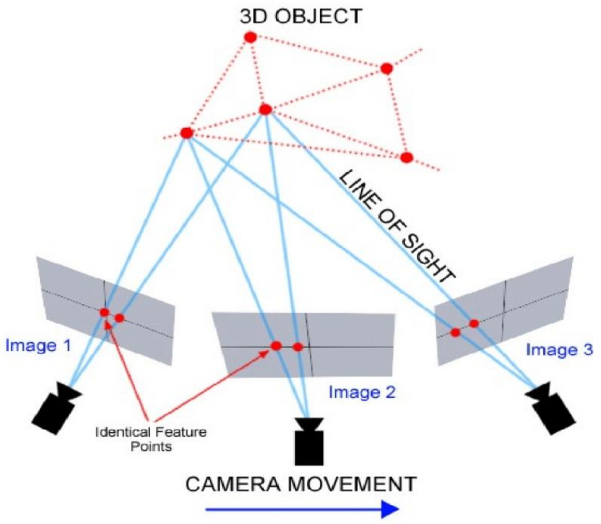
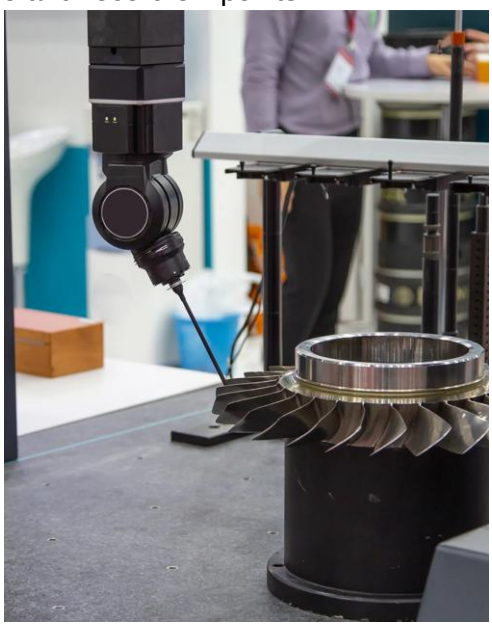


1. Information Sheet: Types of 3D Scanning Technologies

Different physical principles are utilized by various 3D scanning technologies:

Technology	Principle	Key Characteristics
Structured Light	Projects a precise series of shifting light patterns (often blue or white light) onto the surface. Cameras capture the distortion of the pattern, and triangulation principles are applied to calculate 3D coordinate measurements. 	High precision accuracy, speed, and resolution. Blue light (LED technology) is popular as it helps capture data on shinier/darker surfaces and filters ambient light.
Laser Triangulation	Projects a laser beam (line or point) onto the object. Sensors measure the deformation of the reflected laser light, and trigonometry is used to calculate the distance and map the surface.	High resolution and accuracy. **Sensitive to dark, transparent, or shiny surfaces.**

		
Laser Pulse (Time-of-Flight / Lidar)	Measures the time (Time of Flight, ToF) it takes for a casted laser pulse to hit an object and return to a sensor, calculating the exact distance. 	Best suited for scanning very large objects and environments (e.g., land surveying or entire buildings). Can be quite slow.
Photogrammetry	Analyzes multiple 2D photographs of a static subject taken from different viewpoints. Computer vision and computational geometry algorithms reconstruct the subject in 3D by detecting corresponding pixels.	Known for acquisition speed, ability to pick up colors/textures, and reconstructing large-scale subjects.

		
Contact-based/Digitizing	A physical probe, often attached to an articulated arm (like a Coordinate Measuring Machine or CMM), physically touches the subject's surface to sample and record 3D points. 	Known for high precision and ability to scan transparent or reflective surfaces. Slower speed, and inability to work well with organic, freeform shapes.

1. **Information Seet: 3D Scanner Button Identification**



No.	Description
1	Working distance indicator
2	Zoom in / Zoom out
3	Brightness up / Brightness down
4	Press: Preview / Scan / Pause Press and hold: bring up the interface menu
5	USB port
6	Power input

2. Information Sheet: Hardware Set-Up and PC Requirements

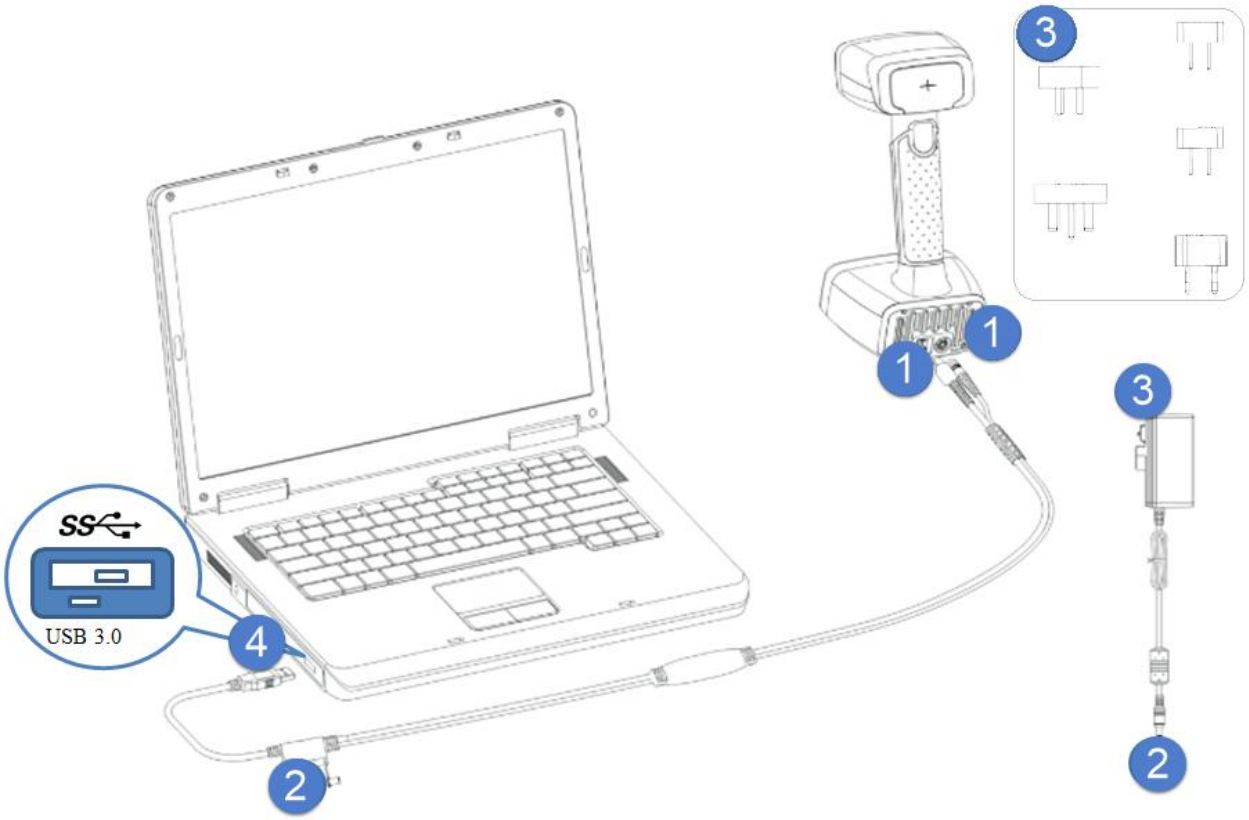
Before connecting the scanner, ensure your computer meets the specifications, especially regarding the graphics card.

PC Requirements:

• **Operating System:** Windows 10 64-bit is required.

• **Graphics Card:** An Nvidia card is required; the **Nvidia GeForce GTX 660 or Nvidia Quadro P1000** is required, while an **NVIDIA GTX serial card higher than GTX1080** is recommended.
• **USB Port:** At least one USB 3.0 port is necessary.
• **Multiple Graphics Cards:** If your computer has multiple graphics cards, you must access the NVIDIA Control Panel, go to Manage 3D Settings > Program Settings, and set the preferred graphic processor for the EXScan software to the **NVIDIA processor**.

1. **Information Sheet: Hardware Connection Steps**



1. **Connect the main cable:** Insert the pin plug into the scanner's circular connector port.
2. **Plug into the scanner USB:** Plug the other end of the cable into the scanner's USB port.
3. **Connect Power Adapter:** Plug the power adapter into the connection cable's DC in port.
4. **Connect to Power Outlet:** Plug the power cable's other end into a power outlet.
5. **Connect to PC:** Connect the other end of the connection cable to a **USB 3.0 port** on your computer.

1. **Information Sheet: Scanning software installation and licensing**

Download and Installation:
1. **Download the software:** Go to https://www.einscan.com/support/download/, select the Einscan HX model, and download the software installer. Registration is required for download.
2. **Install the software:** Double-click the installation package and follow the instructions to complete the installation. Administrator rights are required for this process. A shortcut will be created on your desktop.

1. **Information Sheet: Firmware Update (if required)**

1. When the software opens, it detects the device firmware.
2. If the scanner firmware is incompatible with the software version, a message will appear, and you must click **"Upgrade"**.
3. The upgrade takes about 6 minutes. **Do not disconnect the scanner or unplug the power** during this process.
4. After a successful upgrade, power off the device, reconnect it, and open the software again.

1. **Information Sheet: Device Activation (Initial Use)**

Device activation is mandatory during initial use.
• **Online Activation (Recommended):** If your computer is connected to the internet, click "online activation".
• **Local Activation:** If you have network connectivity issues, you can use "Local activation" by importing the **.ple license file**, which can be found in the USB drive.
• *Note:* If activation fails, you can send a request for an activation file to einscan_support@shining3d.com, ensuring you include your serial number.

1. **Information Sheet: Safety Warnings and Precautions**

When operating the EinScan HX, be mindful of the following safety and hardware considerations:
• **Laser Safety:** The EinScan HX uses a **Class II laser** when operating in **Laser Scan mode**. While this class is generally considered eye and skin safe under normal use, it is advised to **avoid staring directly at the laser beam** for extended periods. Keep the laser beam away from children.
• **Handling:** Handle the scanner with care, as **bumping or dropping** the device may damage the electronic and optical components. Do not touch the camera lens.
• **Calibration Board:** The calibration board must be protected and kept clean. Ensure there are **no scratches or stains** on the black surface with white circles. Only clean the board with clear water; do not use alcohol or chemical liquids. Use only the calibration board matched to your device's serial number.
• **Environment:** If texture scanning is used (Rapid Scan mode), strong light nearby can cause calibration failure. If operating outdoors, set up a shelter or cover to avoid direct sunlight.

1. **Information Sheet: Initial Calibration**

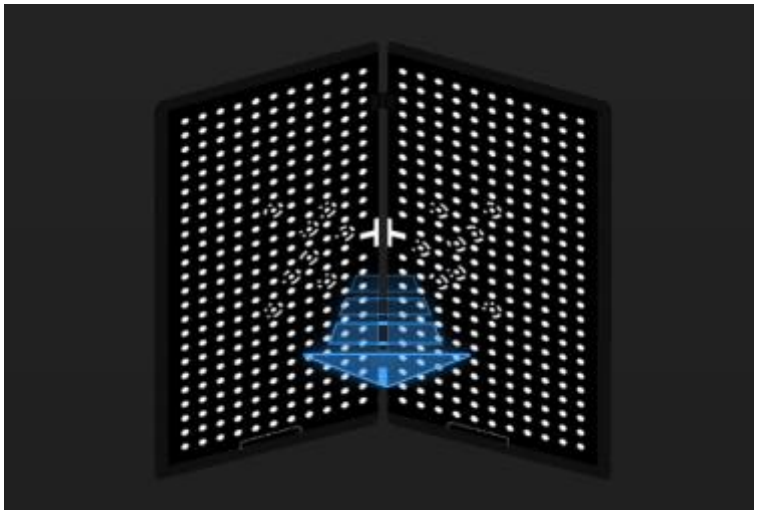
The EinScan HX utilizes a combination of structured light (LED) technology for fast scanning and laser triangulation for better accuracy and performance on challenging surfaces.
Calibration ensures the device scans with optimal accuracy and quality. You are guided through the **STANDARD CALIBRATION** first during initial use.

Calibration is required when:
- The scanner is used for the first time.
- The scanned data is incomplete or its quality drops.
- The scanner is severely shaken or jolted, for example, during transport.
- The accuracy drops during scanning, such as frequent alignment failures or frequent indications of tracking lost.
- The scanner has not been calibrated for more than 14 days. In this situation, the software will prompt you to calibrate the scanner.



Calibration Steps:

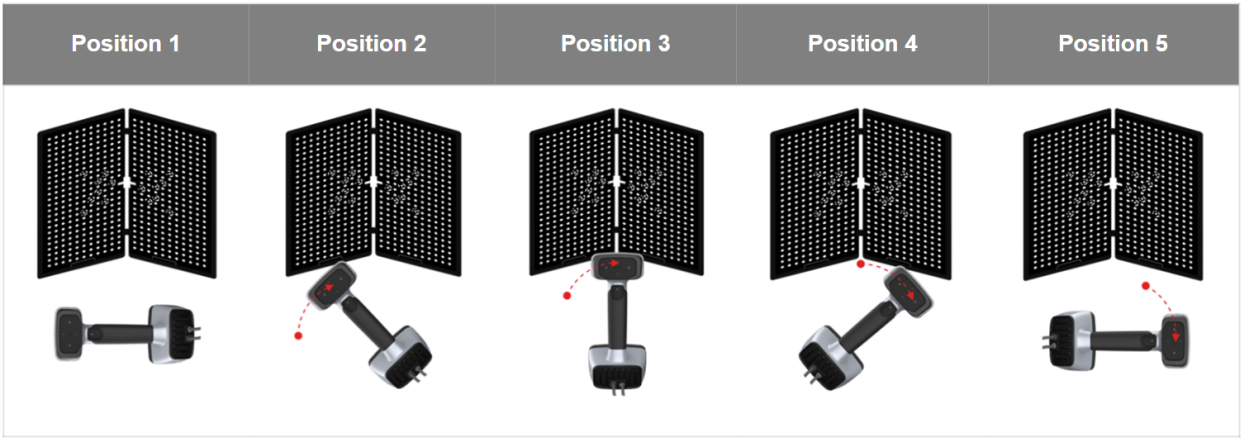
1. **Preparation:** Take out the calibration board and the position paper. Ensure the calibration board is clean, without scratches or stains on the black surface with white circles, and use only the board matching your device's serial number.



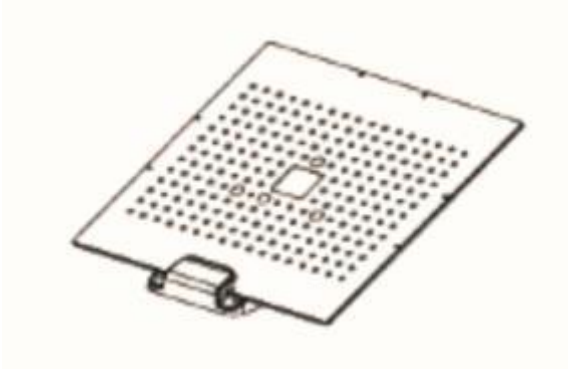


2. **Standard Calibration:**

◦ Put the calibration board vertically onto the position paper.

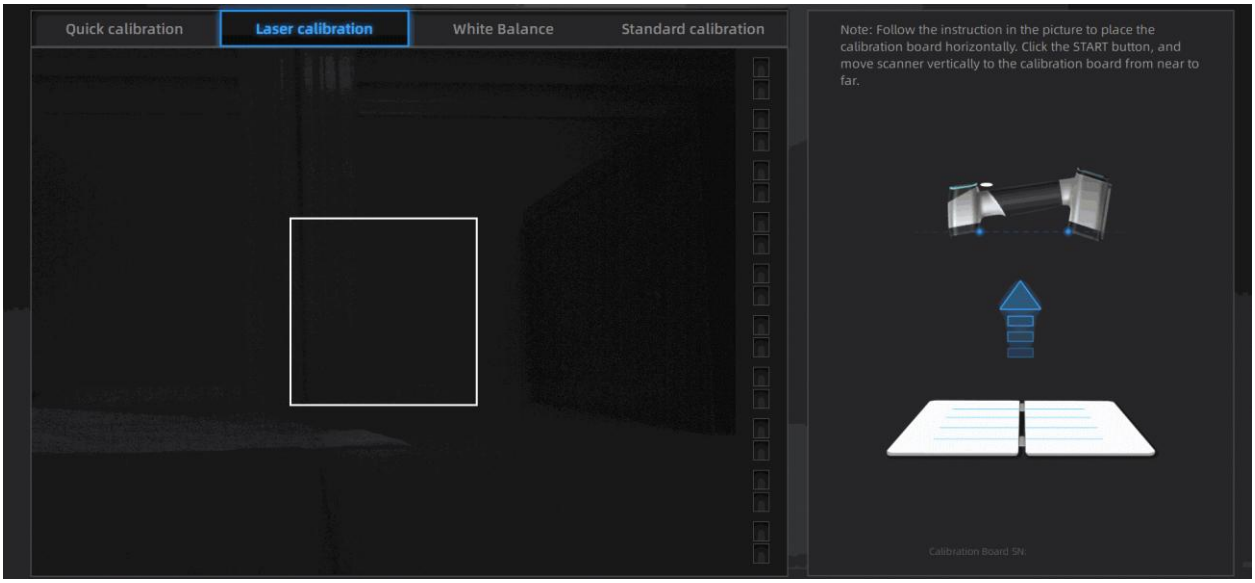


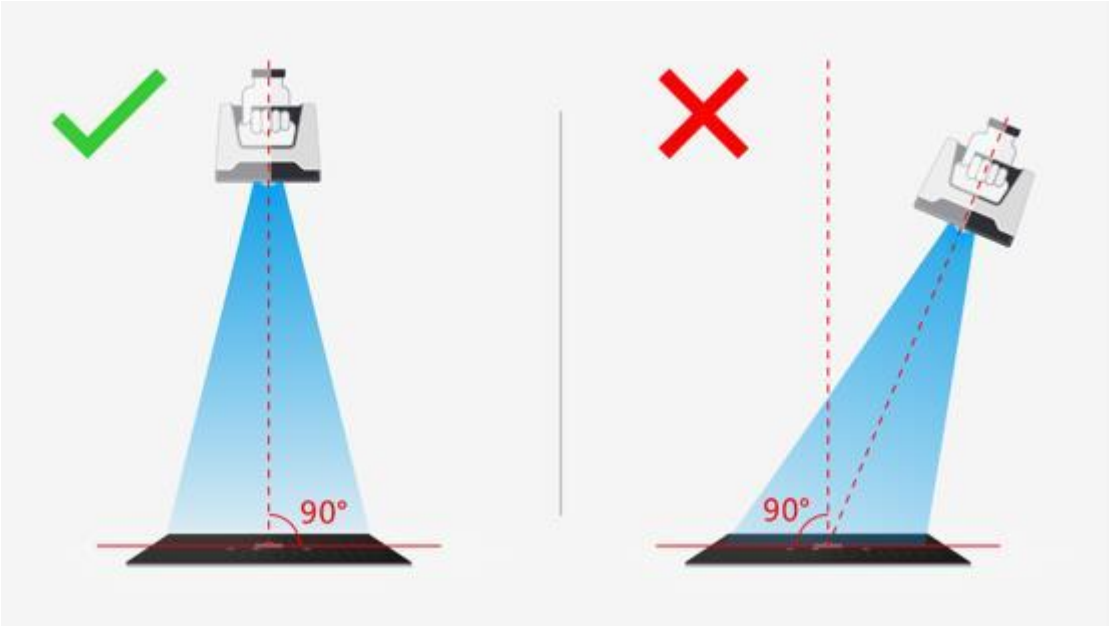
◦ Follow the on-screen instructions to capture **5 different positions** of varying heights.



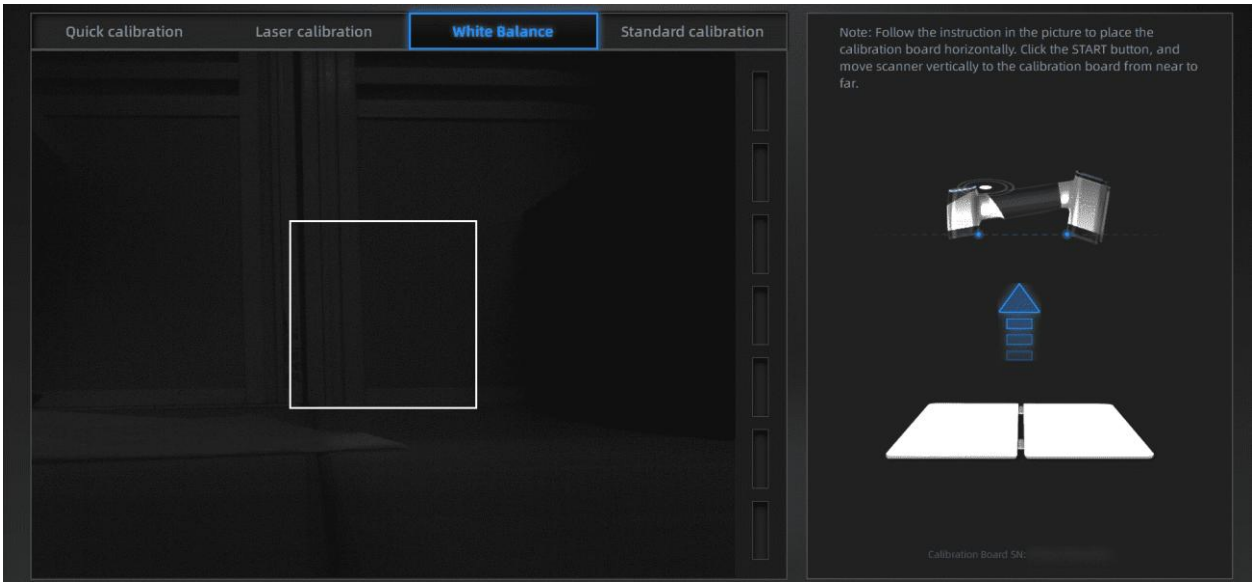
◦ While holding the scanner, keep it upright to the center of the board, moving it closer or farther until all the required distances are captured (the distance color bar should show green).

3. **Laser Calibration:** After Standard Calibration succeeds, the software moves to **LASER CALIBRATION**.
    1. Place the calibration board horizontally with the white side facing up.
    2. Pay attention to the camera window on the left and direct the camera at the middle of the calibration board.
    3. Hold still and press on the scanner to start calibration.
    4. Move the scanner up and down slowly until all distance indicators turn green.





4. **White Balance:** Next, the software moves to **WHITE BALANCE**.



◦ Put the **back (white) side of the board** on a stable, level table.

◦ Point the scanner upright to the board and move it up and down until a photo is taken at the right distance. This step is important for capturing accurate color texture, and is suggested every time the texture camera is plugged in or the lighting environment changes.

Once all calibrations succeed, a pop-up confirmation will appear, and you will enter the scan mode. You are now ready to start scanning.

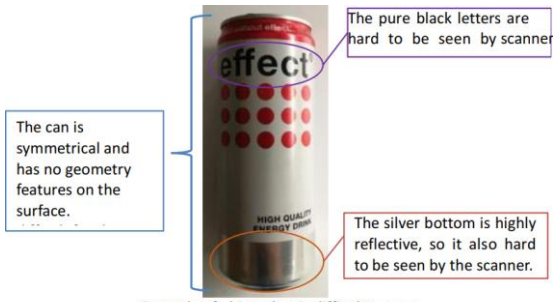
1. **Information Sheet: Preparation: Object and Surface Handling**

Proper object preparation is essential for achieving good scan results, especially regarding surface reflectivity and geometry features:

Surfaces and Material

The suitability of preparation depends on the chosen scan mode (Rapid Scan uses LED/structured light; Laser Scan uses laser triangulation).

Object Type	Preparation Strategy	Scan Mode Recommendation
Black, Transparent, Shiny, or Highly Reflective	Apply **washable or vanishing scanning spray**. Scanning systems commonly face issues with surface reflectivity, color, or finish.	The **Laser Scan** mode is specifically capable of scanning black and shiny objects, often without needing powder.
Objects with Less Features or Repetitive Features	Place markers on the objects. You can also mark or draw on the surface to add features.	Rapid Scan supports **Feature Alignment** or **Hybrid Alignment** (using features and markers).
Soft Material/Moving Parts	Avoid scanning moving or vibrating objects. Soft, non-rigid parts are unsuitable for contact probing and can deflect when touched.	The shape of the object must be maintained without any changes during the process.
Lattice Structures	Not recommended for scanning if they contain many small deep holes.	-



Example of object that is difficult to scan

Object size for scanning, referencing limitations and corresponding strategies:

The size of the object is a major consideration when choosing the appropriate 3D scanning technology and workflow

1. Object Size Categories and Scanner Suitability

Scanning technologies are often selected based on the scale of the workpiece:

• **Small to Medium-Sized Objects:** Objects ranging from $30\times30\times30\text{mm}$ up to approximately 4m in length are typically handled by handheld or fixed-mode structured light and laser scanners, such as the EinScan Pro series and EinScan HX.

◦ **Fixed Scan Mode (Turntable):** This mode is ideal for **small objects** that fit onto a $\Phi\ 150\text{mm}$ disc and require high accuracy and high resolution. Larger objects can still be scanned in fixed mode if manual or feature alignment is used instead of coded targets on the turntable.

◦ **Handheld Rapid/Laser Scan Modes:** These modes are generally recommended for objects that are **larger** than what the fixed mode turntable can accommodate (up to 4m length).
• **Large-Scale Subjects/Environments:** For scanning **very big objects, environments**, landscapes, or monuments, technologies like **Laser Pulse 3D scanners** (often terrestrial laser scanners) or **Photogrammetry** are typically employed.

2. Resolution and Maximum Size Relationship

When scanning in handheld modes with high resolution, the size of the object that can be successfully scanned is limited.
• **Resolution Constraint:** Theoretically, the **maximum scan size is related to the point distance (resolution)** by the formula: Maximum size of scan $\approx \text{point distance} \times 8192 / \text{mm}$.
• **Practical Constraint:** In practice, the size of the object that can be scanned also depends heavily on the capabilities and memory of the **computer's graphics card**. A higher resolution captures finer details but leads to larger files and longer processing times, potentially overwhelming the hardware.

3. Strategies for Large Objects (Volumetric Accuracy)

When scanning objects too large to be captured in a single frame, device relocation and alignment strategies are critical to maintain accuracy across the whole part:
• **Relocation/Leapfrogging:** For larger parts, the entire object cannot be scanned from a single device location, necessitating **device relocation** (or "leapfrogging").
• **Markers and Global Accuracy:** Using **Marker Alignment** is the recommended method for scanning **large objects** to mitigate the cumulative errors that accumulate from merging large amounts of data, thus ensuring higher global accuracy of the complete scanned data.
• **Global Marker File (GMF):** For scanning large-size objects requiring high accuracy, especially thin-wall objects that need accurate merging of opposite sides, a **Global Marker File** (GMF) captured by a photogrammetry system can be used to establish a highly accurate Markers Frame.

4. Objects Too Small or Complex

• **Small Parts for Fixed Mode:** The Fixed Scan Mode with a turntable is designed for **small to medium-sized objects**.
• **Lattice Structures:** Scanning is **not recommended** for **lattice structures with many small deep holes** due to voids and non-line-of-sight issues.
• **Thin-Wall Objects:** For thin-wall objects, external markers or a Global Marker File should be used to help merge the sides accurately.

1. **Information Sheet: Scanning Modes (Rapid and Laser, Fixed and Turntable)**

The EinScan HX offers two main handheld modes: Rapid Scan (LED) and Laser Scan (Laser Triangulation).
Rapid scan



In rapid scan mode, the scanner can scan faster, and capture color texture. There are four alignment modes: markers, features, hybrid alignment and texture. If the object to be scanned has rich features or texture, and is not easy to place markers, this mode can be used as a priority. For specific settings and explanations of alignment modes, please refer to Project settings > Rapid scan.

Laser scan

The scanned data of Laser Scan boasts high accuracy and high quality. In this mode, the scanner has strong adaptability to the material of the object to be scanned. For specific settings, please refer to Project settings > Laser scan.



Feature	Rapid Scan (LED/Structured Light)	Laser Scan (Laser Triangulation)
Light Source	LED	Laser
Accuracy	0.05 mm	**0.04 mm**
Speed	15 FPS (Non-texture) / 10 FPS (Texture)	**55 FPS**
Alignment	Markers, Features, Hybrid, Texture	**Markers only** (default)
Texture/Color	**Yes** (Texture Scan required)	No

Resolution in Rapid and Laser Scan

Resolution	Rapid Scan	Laser Scan
High Detail	A point distance of 0.7 mm by default	A point distance of 0.2 mm by default

Medium Detail	A point distance of 1.0 mm by default	A point distance of 0.5 mm by default
Low Detail	A point distance of 1.5 mm by default	A point distance of 1 mm by default

Fixed Scan Mode

In Fixed Scan Mode, the scanner and the object remain stationary during a single scan acquisition.

• **Accuracy:** The single shot accuracy for Fixed Scan (with or without a turntable) is specified as 0.04mm.
• **Speed:** A single scan typically takes less than 0.5seconds (for EinScan Pro EP and 2X Plus) or less than 1second (for EinScan Pro 2X).



Fixed Scan with Turntable Mode



The Turntable Scan Mode (referred to as Fixed Scan with Turntable) utilizes a **rotary table** to simplify the process of gaining access to the entire workpiece.

• **Hardware and Size:** This mode is perfect for objects that fit onto a Φ 150mm disc. However, larger objects can still be scanned if the automatic alignment using the turntable's coded targets is not required. The rotary table can be dynamically indexed to the reference system via its own encoder, establishing a dynamic reference system that provides access to all sides of the part and removes the need for re-fixturing or "leapfrogging" the part or scanner.

• **Workflow:** Before scanning, the user must set the **turntable steps** (between 2 and 180) for a 360 ◦ full rotation, with 8 steps being the default and recommended setting. Using more steps helps capture more complete data at various angles but does not increase accuracy.

1. **Information Sheet: Types of allignment**

Alignment	Description	Note
 **Feature Align**	1. Choose **Feature Align**. 2. Select the project to be aligned in the fixed window and the floated window. 3. Click **Apply** to align them.	Objects with repeated features, like a round or a ring, or that with small size are not suitable for this mode.

 **Manual Align**	1. Choose **Manual Align**. 2. Manually choose at least 3 common feature points on the data in the fixed window and the floated window respectively. 3. Click **Apply** to align them.	• The chosen points should not be in a line. • Manual feature alignment is a supplement to auto feature alignment, and can be used when it fails.
 **Markers Point Align**	1. Choose **Markers Point Align**. 2. Select the project to be aligned in the fixed window and the floated window. 3. Click **Apply** to align them.	The two projects should have at least 3 markers in common.
 **Markers Point Manual Align**	1. Choose **Markers Point Manual Align**. 2. Select the project to be aligned in the fixed window and the floated window. 3. Manually choose at least 3 common markers on the data in the fixed window and the floated window respectively. 4. Click **Apply** to align them.	The chosen markers should not be in a line.
Texture	To align scanned data of an object that has many colorful patterns on its surface. This alignment is suitable for a flat object or a large curved object.	
Turntable Coded Target Alignment	Applicable for scanner with turntable. This is the easiest alignment method and is highly recommended for achieving high accuracy and high resolution on small objects. It relies on the scanner recognizing coded targets on the turntable surface to calculate the object's new position after each rotation step. At least 4 common targets must be visible between neighboring scans.	

1. Information Sheet: Brightness and Litin Setup for Scanning

The efficiency and quality of 3D scanning are significantly influenced by the proper setup of **brightness (exposure) and ambient lighting conditions**.

Here are brief notes on setting up brightness and handling lighting for different scanning technologies:

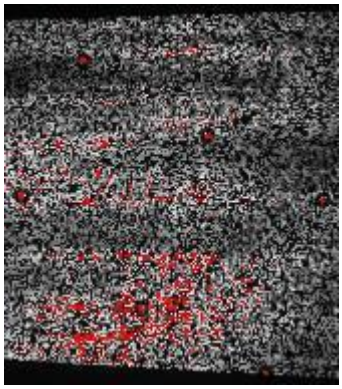
1. Brightness Setup (Exposure Adjustment)

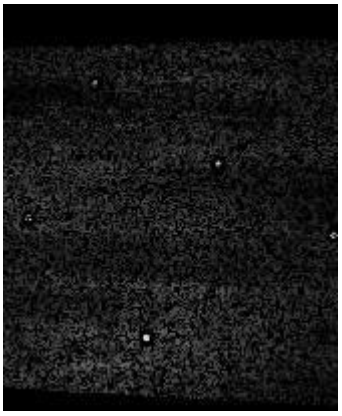
Brightness, often referred to as exposure or sensitivity, must be carefully balanced to ensure the scanner captures adequate detail without introducing errors.

- **Adjustment Method:** Brightness is typically adjusted in **Preview mode** before recording data. You can adjust the brightness using the **+/- buttons** on the scanner or a **slider in the software**.
- **Goal:** The optimal setting is the **lowest brightness** where the scan features (such as laser traces and markers) are **clearly visible**.
- **Identifying Errors:**
    - **Overexposed areas** appear **red** in the preview/rangefinder. Overexposure captures more noise.
    - **Underexposed areas** appear **black**. If the markers or laser traces are too dark, the data captured may be insufficient.
- **Special Considerations:** When scanning an object with **high contrasting texture** (e.g., white and black), **HDR (High Dynamic Range)** mode can be turned ON, though this will increase the time taken for each single scan.

Too bright **Proper** **Too dim**







2. Lighting Conditions and Challenges

Ambient lighting conditions are crucial, especially for non-contact scanning methods like Structured Light and Laser Triangulation.

Condition / Surface Type	Impact on Scan	Preparation / Mitigation

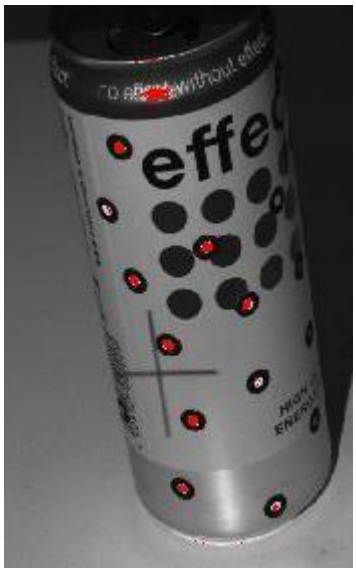
Ambient Light	Structured light scanners (using LED/Rapid Scan) are sensitive to lighting conditions and have trouble working outdoors in broad daylight.	If using **texture scanning (Rapid Scan mode)**, **strong light nearby can cause calibration failure**. If operating **outdoors**, you must **set up a shelter or cover to avoid direct sunlight**.
Shiny / Reflective Surfaces	Highly reflective (silver, shiny) surfaces are challenging for non-contact scanning. Scanning can have many issues with surface reflectivity.	**Apply washable or vanishing scanning spray** (matte finish coating) on the parts to make them more acceptable for scanning.
Dark / Black Surfaces	Dark or black surfaces can present a problem for certain scanning technologies, such as laser triangulation.	**Apply scanning powder/spray** before scanning. However, the **Laser Scan mode** (Laser Triangulation) on the EinScan HX is specifically capable of scanning black objects, often **without needing powder**.
Texture Quality (Color)	If using a color camera (Color Pack add-on) for texture capture, the lighting environment is key for high-quality texture data.	**White Balance calibration** should be performed every time the texture camera is plugged in or when the lighting environment changes to ensure accurate color texture capture.
Light Halo/Reflection	Light reflections can cause "light halo" artifacts on reflective textures.	To avoid light halo with reflective texture, scan with the scanner positioned at around **30 degrees from the surface normal**, and try to avoid light source reflections in the room.

3. Technology-Specific Brightness Notes

2. **Structured Light (LED/Rapid Scan):** Structured light scanners, particularly those using **blue light (LED technology)**, are popular because blue light assists in capturing data on shinier/darker surfaces and **filters ambient light**.
3. **Laser Scanners (Laser Triangulation):** For laser line scanners, the lines and markers should be clearly visible in the preview, and the lowest brightness setting that allows this is optimal.

4. Information Sheet: Marker Application (For Alignment/Tracking)

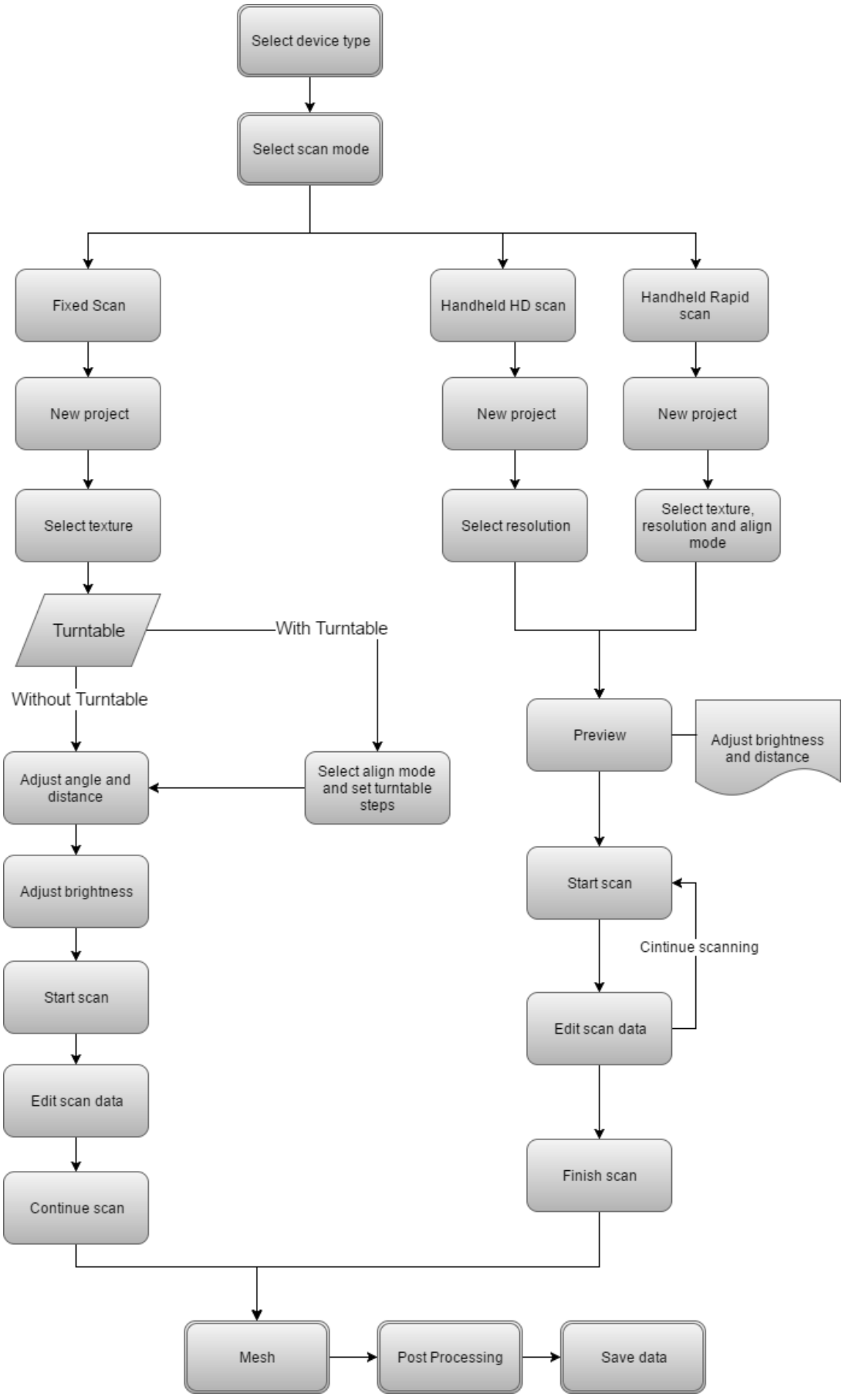
If relying on marker alignment (Marker Alignment, Hybrid Alignment, or Laser Scan mode), markers must be applied strategically to the object:





1. Place markers in a **random, non-linear pattern**.
2. Ensure markers are stuck on a **flat surface area** and kept flat.
3. Use **only the markers provided** with the device, as other markers may result in bad accuracy or may not be detected.
4. For optimal tracking, aim to capture **at least 3 markers** (Laser Scan) or **4 markers** (Rapid Scan and other handheld modes) in each frame/scanning field of view.
5. If scanning thin-wall objects or large items, a **Global Marker File (GMF)** can be imported to help merge sides accurately and maintain high global accuracy.

1. **Information Sheet: Scanning Workflow**

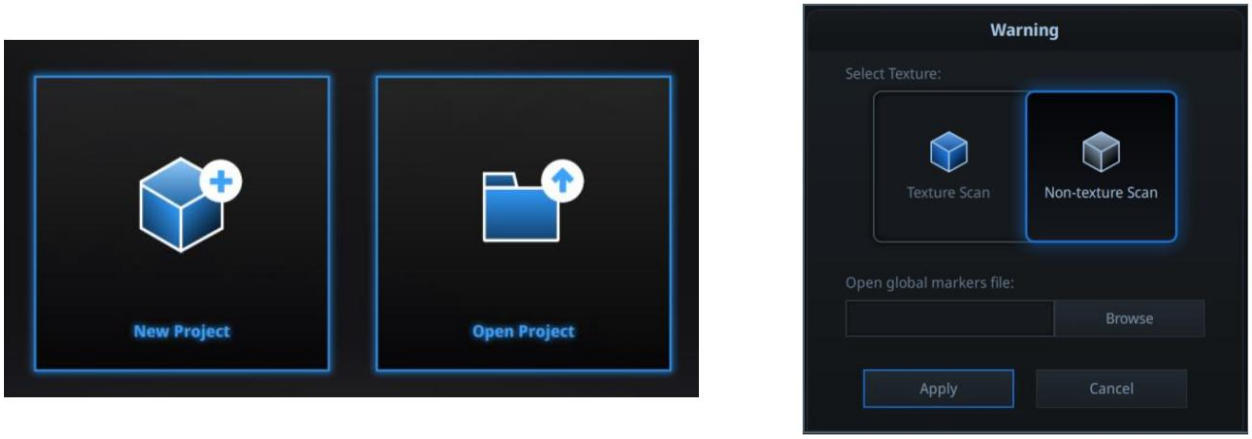

Select device type
Select scan mode
Fixed Scan
Handheld HD scan
Handheld Rapid scan
New project
New project
New project
Select texture
Select resolution
Select texture, resolution and align mode
Turntable
With Turntable
Without Turntable
Preview
Adjust brightness and distance
Adjust angle and distance
Select align mode and set turntable steps
Adjust brightness
Start scan
Cintinue scanning
Start scan
Edit scan data
Edit scan data
Finish scan
Continue scan
Mesh
Post Processing
Save data

1. **Project Setup and Data Acquisition**

The initial phase involves selecting the scan mode, setting parameters, and capturing the raw data (point cloud).

A. Create a Project and Select Scan Mode

1. **New Project:** Click "New Project" and specify the project name and save location.



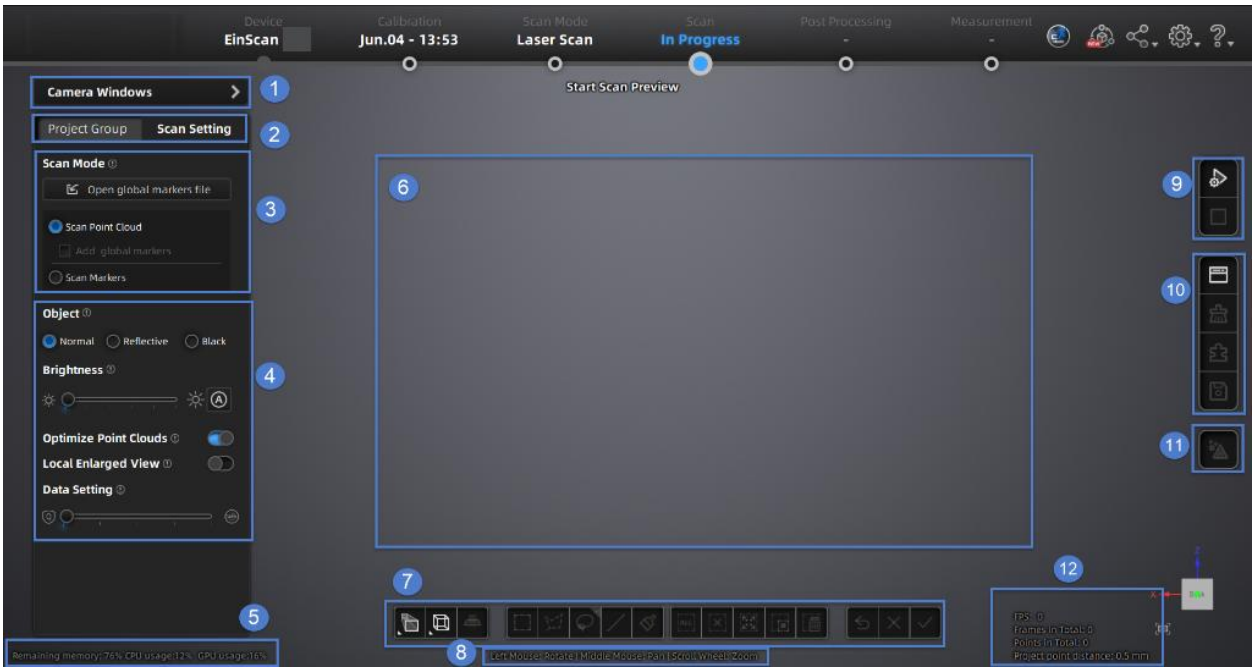
Create a project, fixed scan

2. **Select Scan Mode:** Select **Rapid Scan (LED)** for speed and texture capture, and **Laser Scan (Laser Triangulation)** for high accuracy on difficult surfaces like black or shiny objects. Also select fixed or turntable mode.
3. **Resolution:** Select the desired resolution (point distance), typically ranging from 0.05 mm up to 3.0 mm, depending on the mode. **Higher resolution** captures better detail but results in **larger files and longer processing times**.

4. **Alignment Settings (Rapid Scan Only):** Set alignment modes
   ◦ **Markers Alignment:** Uses markers for reliable tracking, recommended for large objects to maintain high global accuracy.
   ◦ **Feature Alignment:** Aligns based on geometric features; requires rich features on the object.
   ◦ **Hybrid Alignment:** Automatically switches between features and markers, allowing markers only on regions with limited geometry.
   ◦ **Texture Alignment:** Uses color/texture features for alignment, requiring **Texture Scan** to be selected.

Information Sheet: User Interface of Scanning Software

Function preview¶



① Camera window¶

To display the actual scanning scenarios so that the user can set scanning parameters.

② Project group and scan setting¶

To manage your project group and set scanning parameters. See more details in Project management and Scan settings.

③ Scan mode¶

In rapid scan mode, you can scan by opening global markers file.
In laser scan mode, you can scan point clouds or markers.

④ Parameter settings¶

See more details in Scan settings.

⑤ Remaining memory, CPU usage and GPU usage¶

- Remaining Memory: To display the percentage of remaining memory.
- CPU Usage: To display the CPU Usage of the computer in real time. You may need to close other unrelated software if it is too high.
- GPU Usage: To display the GPU Usage of the computer in real time.

⑥ Preview/Scanning window¶

To preview the model and check the scanned model.

⑦ Data editing¶

To edit data after scanning. See more details in Data editing.

⑧ Keyboard shortcuts¶

To change the perspectives and move the model by the composition of keys.

⑨ Buttons¶

Click  to preview the scanning; click  to start scanning; click  to pause scanning.

Click  to generate point clouds after scanning.

⑩ Function¶

To import the project file and to align, delete and save the model data.

⑪ Mesh¶

See more details in Mesh model.

⑫ Others¶

Display FPS, frame amount, point amount of the project and other information.

B. Scanning Controls (Preview, Distance, and Brightness)

1. **Preview Mode:** Before recording data, use **Preview mode** (click the trigger/Play button) to ensure optimal settings. Preview mode does not start scanning.



2. **Adjust Distance:** Ensure the scanner is at the **Optimal Distance** (approximately 470 mm for Laser Scan; 300–470 mm for Rapid Scan, depending on resolution). The distance is optimal when the **rangefinder color bar turns green** (Red is too close, Blue is too far).
3. **Adjust Brightness:** Double-click the Play button to enter exposure adjustment. Adjust using the +/- buttons on the scanner or the slider until the laser traces and markers are **clearly visible** but not overexposed (avoiding red areas) or underexposed (avoiding black areas).
4. **Start Scan:** Click "Start" or press the Play button to begin recording data.
5. **Tracking and Recovery:** If a **"Track lost"** alert appears, immediately point the scanner back at any previously scanned area to recover tracking and continue scanning.

C. Point Cloud Generation and Initial Editing

1. **Stop Scan and Optimize:** Click "Pause" (Play button). Click **Stop** to generate the optimized point cloud. This process includes realignment by marker position recalculation.
2. **Point Cloud Selection/Deletion:** Use the editing tools to remove unwanted data (noise or overspray).
   - **Selection:** Use **SHIFT + Left mouse button** to select redundant data (selection turns red).
   - **Deselection:** Use **CTRL + Left mouse button** to deselect.
   - **Delete:** Click the delete button or press the **DELETE** key. You can only **Undo** the most recently deleted data.

3. **Cutting Plane (Laser Scan):** This tool allows the user to define a plane to delete data on one side, useful for removing a scanning base during acquisition.

2. **Information Sheet: Distance and Brightness Adjustment**

Maintaining optimal distance and exposure (brightness) is crucial for acquiring high-quality data.
1. **Preview Mode:** Before starting the recording, enter **Preview mode** (using the trigger or software button) to adjust settings without saving useless data.
2. **Optimal Distance:** Adjust the distance between the scanner and the object until the **rangefinder color bar turns green**. Red indicates too close, and blue indicates too far.
   ◦ For **Laser Scan**, the Optimal Distance is approximately **470 mm**.
   ◦ For **Rapid Scan** (high resolution 0.25–0.5 mm), the Optimal Distance is approximately **300 mm**.
3. **Brightness Adjustment:** Adjust brightness by double-clicking the Play button/trigger and using the +/- buttons on the scanner or the slider in the software. The **lowest brightness** where the laser traces and markers are clearly visible is optimal. Overexposed areas appear red, and underexposed areas appear black.

1. **Information Sheet: Alignment and Movement Tips**

• **Continuous Movement:** To maximize scan efficiency and minimize potential misalignment, movement of the scanner should be **continuous and uniform**.
• **Perpendicularity:** During scanning, ensure the scanner is kept as **perpendicular to the surface** as possible.
• **Feature Alignment Strategy (Rapid Scan):** When starting a scan using Feature Alignment, hold the scanner on the object for about **3 seconds** before beginning to move.
• **Plane Detection (Rapid Scan):** When using Feature or Hybrid Alignment, enable the **Plane Detection** setting (recommended ON). If the software detects less geometry, it will warn you ("not enough features to align").
• **Texture Alignment Strategy:** Texture alignment relies on color and texture to align scans. If selected, move **evenly and steadily**, and avoid sudden rotations or changes in angle. Since the texture is captured separately from the 3D data, in areas with sharp edges, texture-data can sometimes be misaligned; try scanning from a different angle in these cases.

1. **Information Sheet: Typical Issues and Troubleshooting**

Tracking Loss ("Track Lost")

Tracking loss occurs when the scanner cannot match the current field of view with the previously recorded data. This happens if:
1. There are insufficient common features (Feature Alignment).
2. There are insufficient markers (Marker Alignment/Laser Scan).
3. The scanner is moving too fast.
4. The distance is outside the optimal range.
When the "Track lost" alert appears, immediately point the scanner back at **any previously scanned region** to recover tracking and continue the scan.

Data Voids and Holes

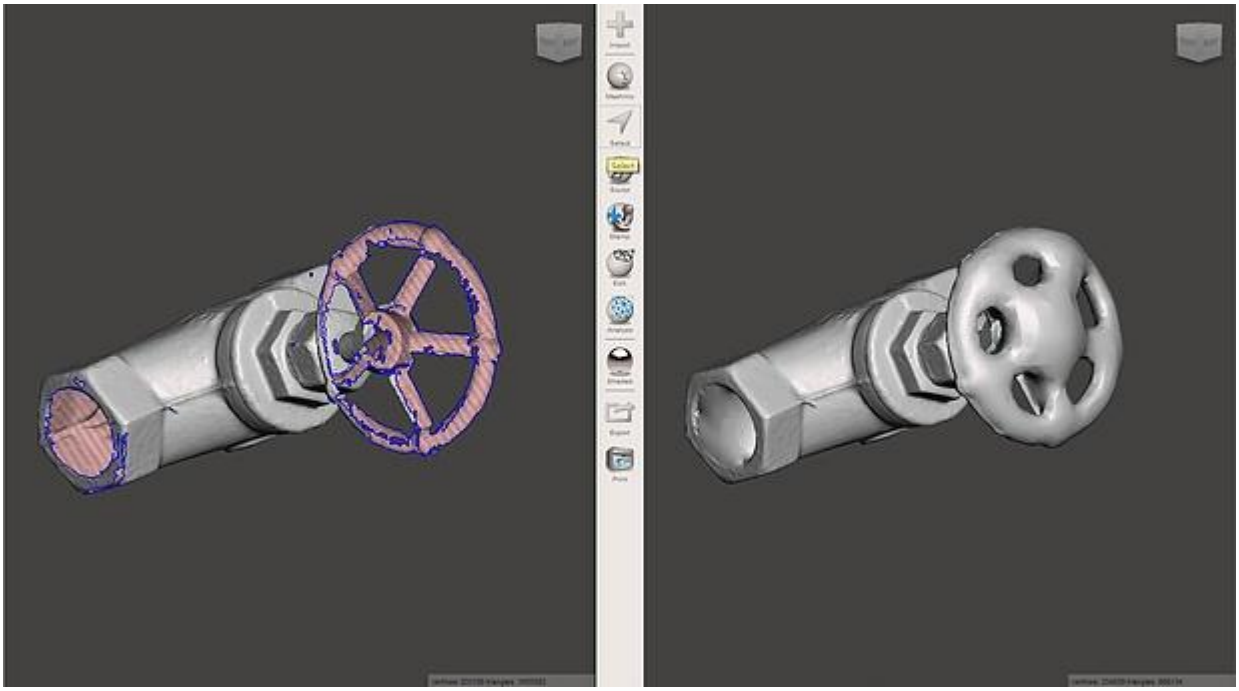
Voids are common in deep features like holes or slots due to the non-line-of-sight nature of structured light and laser scanning.

Texture Misalignment or Missing Texture

• If the texture is missing (appears as blue areas, usually on scan frame edges), it means texture information was missed. **Rescan the area from a different path**.
• If the texture is visually misaligned with the 3D data, check that the camera is properly locked, and redo the calibration.
• If you edit the mesh (e.g., hole filling or simplification), the texture rendering will be affected. Use the **Texture Remapping** tool to reapply the texture information correctly onto the modified mesh.
• To avoid **light halo** with reflective texture, scan with the scanner positioned at around **30 degrees from the surface normal**, and try to avoid light source reflections.

1. **Information Sheet: Meshing (STL Generation)**

Meshing converts the dense point cloud data into a polygonal mesh, which is usually necessary for downstream activities like 3D printing and CAD modeling.
1. **Create Mesh:** Click **Mesh Model**.
2. **Mesh Type Selection:**



◦ **Watertight:** Automatically fills all holes. This is the required output for direct **3D printing**. The Watertight option only keeps the largest connected data section.
◦ **Unwatertight:** The model retains any holes or voids that occurred during scanning. This option is faster than Watertight meshing.

3. **Texture and Meshing:** If texture was captured, it is separate from the 3D data. The texture will still be displayed on areas where holes are filled during meshing, but missing texture data will appear black.

1. **Information Sheet: Post Processing and Measurement**

After meshing, you enter the post-processing phase, which includes editing the geometry, measuring features, and alignment refinement. ***Primary post processing is done in 3D scanning software, however for advanced post processing a mesh editor software is required.***

A. Mesh Editing Tools

The Mesh Editing menu provides tools to refine the polygonal mesh structure:
• **Data Simplification:** Reduces the number of polygons and overall file size, thereby lowering the level of detail.
• **Hole Filling:** Tools include **Manual Fill** (picking the edge of a hole to fill it), **Auto Fill** (fills all holes smaller than a set perimeter), and specific fill methods like **Curvature, Tangent, or Flat**.
• **Smooth:** Reduces surface noise, but may smooth small details and sharp edges.
• **Sharpen:** Used to increase the level of detail.
• **Texture Remapping:** If mesh editing (like simplification or hole filling) affects the texture rendering, this function is used to reapply the texture information correctly onto the modified mesh.

B. Measurement and Alignment Modification

The Measurement menu offers tools often used in reverse engineering:
• **Create Features:** Allows the creation of fundamental geometric features such as **Points, Lines, and Planes** by fitting them to the mesh data using methods like 3 Points Fit or Best Fit.
• **Exact Movement:** Allows manual input of translation (mm) and rotation (degrees) to match the data origin to a specified coordinate system.
• **3-2-1 System Movement:** This tool aligns the data using the metrology standard of Datum Structure (Plane-Line-Point alignment), which removes the six degrees of freedom and repositions the data to the global coordinate system. This is particularly useful for establishing a foundational coordinate system for reverse engineering.
• **Dimensioning:** Calculates **Distance** between points, **Surface Area**, and **Volume** (only for watertight meshes).

1. **Information Sheet: Exporting the Final File (STL and Other Formats)**

The final step is saving the data for subsequent applications.
1. **Export Data:** Click **Save** to export the data. The default file format is **STL**.
2. **File Format Options:**
   ◦ **STL (Stereolithography):** A mesh format recommended for **3D printing** (if watertight) and general **Reverse Engineering**.
   ◦ **OBJ (Object):** Exports the mesh, plus separated **texture (JPG)** and a matching file (MTL), commonly used for artistic applications and 3D rendering.
   ◦ **ASC:** Exports the optimized **point cloud** data, often used if post-processing is required in specialized third-party software.
   ◦ **PLY and 3MF:** Mesh formats offering low storage usage.
   ◦ **P3:** Saves the marker positions, used for Global Marker Files.
3. **Scale Data:** The system allows scaling the data volume, though the level of detail and triangle count remain unchanged.
4. **Third-party Export:** EXScan Pro allows direct export of the meshed STL data to specific external software:
   ◦ **Geomagic Essentials:** Used as a bridge between scanning and design, enabling the extraction of features to create solid models.

◦ **Solid Edge SHINING 3D Edition:** A comprehensive platform that supports reverse engineering, CAD functions, and convergent modeling.

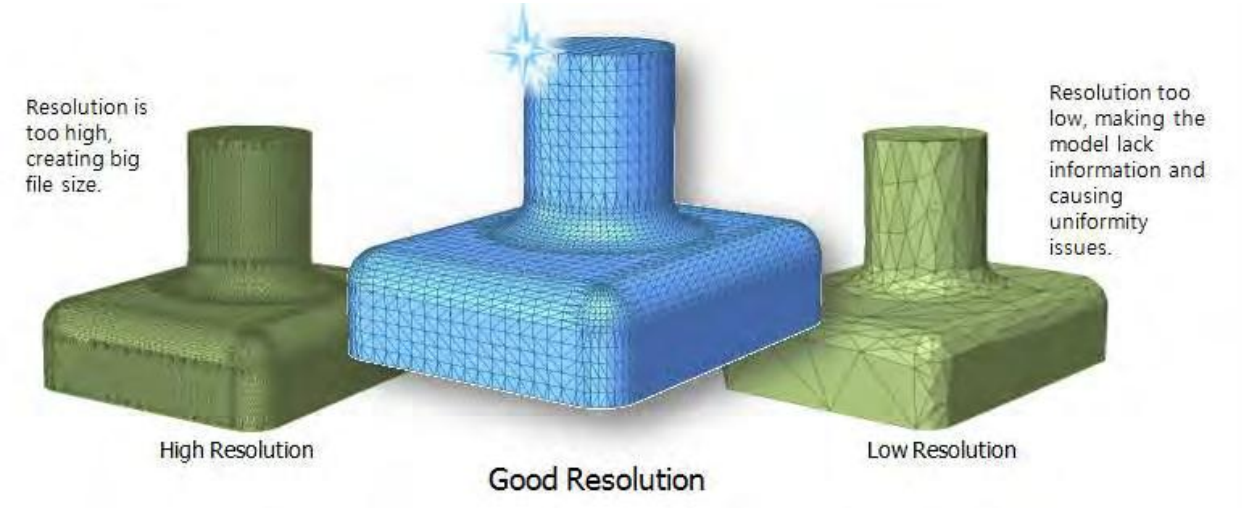
The entire scanning and mesh generation process results in a polygonal mesh (like an STL file). This mesh acts as the detailed digital surface data, which is then used as a reference to create a true CAD model in downstream reverse engineering processes. This transition from raw scan data (point cloud) to mesh (STL) to a usable CAD model requires specialized software tools to handle the creation of geometric features and free-flowing surfaces (NURBS)

Information Sheet: Post-Processing

• Use the scanner's editing tools to delete unwanted noise or disconnected data (overspray) using SHIFT + Left mouse selection.
• When meshing the data, select the **Watertight** option to automatically fill all holes and prepare the model for 3D printing. Note that Watertight meshing only keeps the largest connected data chunk.

1. **Information Sheet: Managing Large Data Files**

3D scanning quickly gathers a lot of data, resulting in large datasets (point cloud or mesh data) that can challenge computer performance and storage.
Strategy:
• **Data Simplification:** After meshing, use the **Data Simplification** tool to reduce the number of polygons, which decreases the file size and polygon count while maintaining the level of detail.
• **Resolution Selection:** Choose a lower resolution setting before scanning if the object does not require high detail, as higher resolution leads to larger files and longer processing times.



The EXScan software guides the user through a sequential workflow from raw scan data acquisition to the creation and export of a final STL or other desired file formats, encompassing crucial post-processing steps often used in reverse engineering (RE).
The following guide details the primary software functions within EXScan, drawing on the procedures described for the EinScan Pro series and the EinScan HX.

Job Sheet: Perform 3D Scanner Calibration

Equipment, Tools, and Materials

- EinScan HX 3D Scanner or equivalent
- Calibration board (must match device serial number)
- Position paper / calibration layout sheet
- Power adapter and cables
- USB 3.0 port computer
- EXScan Pro scanning software
- Clean, stable worktable
- Soft cloth (for dust removal; *no chemicals*)

Procedure

Follow the steps below **exactly as instructed** for correct scanner calibration.

1. Preparation

1. Ensure the scanner, calibration board, and PC are clean and dry.
2. Verify that the calibration board matches the scanner serial number.
3. Place the **position paper** flat on the worktable.
4. Launch **EXScan Pro** software — the system will automatically prompt calibration if needed.

2. Standard Calibration

1. Position the calibration board **vertically** on the position paper.
2. Hold the scanner perpendicular to the board.
3. Move closer or farther until the **distance bar shows green**.
4. Follow on-screen prompts to capture **five different positions/heights**.
5. At each position, hold the scanner steady and allow the software to register the frame.

3. Laser Calibration

1. Place the calibration board **horizontally**, white side up.
2. Aim the scanner's **left camera window** at the center of the board.
3. Press the scan button to begin calibration.
4. Slowly move the scanner **up and down** until all distance bars are green.
5. Once complete, the software will automatically continue to the next stage.

4. White Balance Calibration

1. Place the **white side** of the calibration board flat on the table.
2. Hold the scanner upright, perpendicular to the board.

3. Move the scanner up and down until the system captures the white balance frame.

4. White balance should be repeated whenever lighting conditions change.

5. Confirm Calibration

1. When all three processes (Standard, Laser, White Balance) succeed, a confirmation window will appear.
2. Click **Finish** to save calibration results.
3. The scanner enters **Scan Mode** and is ready for use.

Job Sheet: Perform Scanning of Rectangular Box Object

Equipment, Tools, and Materials

- EinScan HX 3D Scanner or equivalent
- Computer with EXScan software
- Power adapter and USB 3.0 cable
- Calibration board (if calibration is required before scanning)
- Turntable (optional, if using Fixed Scan with Turntable)
- Soft, matte scanning spray (if object is glossy or reflective)
- Stable worktable with non-reflective surface
- Rectangular box object (100–300 mm recommended)
- Adequate indoor lighting (avoid direct sunlight)

Procedure

1. Preparation

1. Inspect the rectangular box object; ensure it is clean and dry.
2. If the surface is shiny, apply a **washable scanning spray** to improve data capture
3. Position the object on a stable table or turntable.
4. Launch **EXScan** and select **New Project**.

2. Select Scan Mode

Choose based on object size and material:

- **Rapid Scan (LED)** – if object has rich features or matte surface.
- **Laser Scan** – preferred for shiny, black, or uniform surfaces.
- Optional: **Fixed Scan with Turntable** if the object fits within φ150 mm and high accuracy is required.

Set parameters:

- **Resolution**: 0.7–1.0 mm for general geometry
- **Alignment Mode**:
    - **Feature Alignment** (recommended for box-shaped objects)
    - **Hybrid Alignment** if markers are used

(See scan mode and alignment details on pages 16–19)

3. Adjust Distance and Brightness

1. Enter **Preview Mode**.
2. Move scanner until the **distance bar turns green** (≈300–470 mm depending on mode).
3. Adjust brightness/exposure so no area is too dark (black) or overexposed (red)
4. Ensure at least one face of the box is clearly visible before starting.

4. Start Scanning

1. Click **Start Scan**.
   Move the scanner **smoothly and continuously** around the object.
2. Keep scanner **perpendicular** to surfaces for highest accuracy.
3. Maintain visibility of edges and corners to avoid "Track Lost" events
4. Cover all six faces by rotating the object or using the turntable.
5. Pause and resume scanning as needed.

5. Generate Point Cloud

1. Click **Pause**, then **Stop** to generate the optimized point cloud.
2. Software will recalculate alignment markers and refine the point cloud.

6. Point Cloud Editing

- Select unwanted data using **SHIFT + Left Mouse**
- Deselect with **CTRL + Left Mouse**
- Delete noise or overspray
- Use **Cutting Plane** (Laser Mode) to remove table surface data

7. Mesh Generation

1. Click **Mesh Model**.
2. Choose:
    - **Watertight Mesh** – fills holes, suitable for export/3D printing
    - **Unwatertight Mesh** – preserves holes for downstream editing
3. Inspect the mesh for missing areas.
4. Use **Hole Filling** or **Smooth** if needed

8. Save and Export

1. Save the project file.
2. Export in required format:
    - **STL** (default for 3D printing and CAD import)

    - **OBJ/PLY** for color texture or 3D visualization
3. Verify that the rectangular box geometry is complete.

Job Sheet: Perform 3D Scanning of a Small Gear Using Turntable

Equipment, Tools, and Materials

- EinScan HX or equivalent compatible 3D scanner
- Motorized turntable (φ150 mm recommended)
- PC with EXScan Pro installed
- USB 3.0 cable & power adapter
- Matte scanning spray (if gear is metallic, shiny, or black)
- Clean tweezers or soft gloves
- Non-reflective table surface
- Small mechanical gear (30–80 mm diameter)

Procedure

1. Preparation

1. Inspect the gear for oil, rust, dust, or grease. Clean if necessary.
2. If shiny or black, apply **washable scanning spray** for optimum surface capture (see surface preparation, page 14)
3. Place the turntable on a stable, vibration-free table.
4. Launch **EXScan Pro** and connect the scanner.
5. If prompted, **perform calibration first** (required if >14 days or accuracy has dropped, page 10)

2. Select Scan Mode

1. Click **New Project**.
2. Select:

- **Fixed Scan → Turntable Mode** (recommended for small objects with complex geometry)

3. Set turntable rotation steps:
    - **8 steps (recommended)**
    - More steps if gear has many small teeth needing better coverage (page 18)
    .
4. Choose resolution:
    - **High resolution (0.2–0.5 mm)** depending on gear tooth detail.

3. Gear Positioning

1. Place the gear **centered** on the turntable.
2. Ensure the gear sits flat and stable—use small clay tack if required.
3. Ensure the gear does not exceed the turntable's φ150 mm capture zone.

4. Adjust Distance & Brightness

1. Enter **Preview Mode**.
2. Adjust the scanner distance until the **distance bar turns green** (optimal range 300–470 mm).
3. Adjust brightness so no part of the gear appears overly red (overexposed) or black (underexposed)
4. Verify the teeth edges appear crisp in preview.

5. Start the Turntable Scan

1. Click **Start Scan**.
2. The turntable rotates automatically through each step, capturing full 360° data.
3. Observe alignment:
    - Turntable mode uses **coded targets** for automatic perfect alignment
4. After one full rotation, check if the bottoms of the gear teeth or inner profile require additional angle passes.

6. Flip & Scan Underside

1. Pause scan.

2. Remove the gear, flip it upside down, and re-center it on the turntable.
3. Resume **Fixed Scan with Turntable**.
4. Capture another full rotation.
5. Software will automatically align two scans using feature alignment or coded targets

7. Point Cloud Optimization

1. Click **Stop** → generate point cloud.
2. Remove overspray data from table surface using:
    - **SHIFT + Left Mouse (select)**
    - **Delete key** (page 27)
      3D scanning CBLM_2
3. Use **Cutting Plane** if excessive base-plane data appears in Laser Mode.

8. Mesh Generation

1. Click **Mesh Model**.
2. Select:
    - **Watertight Mesh** for closed gear model
    - **Unwatertight Mesh** if internal voids must remain unfilled
3. Use **Hole Filling** or **Sharpen** to enhance fine tooth geometry
4. Inspect for fidelity of tooth profiles and root fillets.

9. Save & Export

1. Save project file for future editing.
2. Export as:
    - **STL** for reverse engineering or 3D printing
    - **OBJ/PLY** for color and visualization
3. Verify file scale and orientation.

Job Sheet: Perform 3D Scanning of Thin-Wall Object

Equipment, Tools, and Materials

- EinScan HX or equivalent scanner
- EXScan Pro software
- USB 3.0 cable & power adapter
- **Scanning markers** (official, manufacturer-provided)
- **Auxiliary objects (1–2 pcs)** with flat surfaces
- Matte scanning spray (if surface is reflective or glossy)
- Non-reflective table surface
- The thin-wall object (e.g., plastic shell, bent sheet metal, casing)
- Soft gloves/tweezers for handling markers
- Stable indoor lighting

Procedure

Thin-wall objects have **poor geometric features**, **sharp edges**, and **little depth**, which makes normal feature alignment fail. Therefore, **marker alignment** with auxiliary objects is required.

1. Preparation

1. Clean the thin-wall object to remove dust or oil.
2. If the surface is shiny or smooth, apply a **light coat of matte scanning spray**.
3. Select **Rapid Scan Mode** since it supports multiple alignment methods and works well with markers (CBLM page 16) .
4. Place the turntable/table surface clean and non-reflective.

A. METHOD 1 — For Thin-Wall Objects (Medium Size)

(Use 2 auxiliary objects on sides)

2. Prepare Auxiliary Objects

1. Take **two auxiliary blocks/objects** with stable flat sides.
2. Place **multiple markers** on:
    - Top surface
    - Lateral sides

Auxiliary objects provide stable features for marker alignment.

3. Place Markers on Thin-Wall Object

1. Stick **different markers** on **lateral sides only** of the thin-wall object.
2. Ensure markers:
    - Are not colinear
    - Are flat and bubble-free
    - Are 25–50 mm apart
    - Are visible from the scanner's viewpoint

4. Scanning Setup

1. Open EXScan → **New Project**.
2. Select **Rapid Scan**.
3. Set **Alignment Mode = Marker Alignment**.
4. Choose resolution (0.7–1.0 mm typical).
5. Enter **Preview Mode**.
6. Adjust:
    - Distance until bar is **green** (page 27).
    - Brightness so image is not overexposed (page 20).

5. Position All Objects

1. Place the **scanned object upright** (standing on its thin edge).

2. Place one auxiliary object on the left side, one on the right side.
3. Maintain approximately **5–10 cm gap** between them.
4. Ensure 3–4 markers from each side remain visible in each frame.

6. Perform Scanning

1. Start scanning.
2. Capture:
    - Front surface
    - Transitional areas (where auxiliary markers on left/right sides appear)
    - Back surface via marker continuity
3. Once both sides are captured, **remove auxiliary objects**.
4. Scan:
    - Lateral edges
    - Any missed areas

If “Track Lost” occurs, point back to a previously scanned area with clear markers.

7. Generate Point Cloud

1. Click **Pause → Stop**.
2. EXScan will recompute marker positions and align all frames automatically.
3. Delete noise using SHIFT + click → DELETE.

8. Mesh Generation

1. Click **Mesh Model**.
2. Choose:
    - **Watertight** → closed object
    - **Unwatertight** → to preserve open edges
3. Apply Hole Filling / Smooth / Sharpen if necessary (page 29).

B. METHOD 2 — For Small Thin-Wall Objects

2. Prepare Auxiliary Object

1. Take one flat auxiliary block.
2. Place markers on its side surface.

3. First Scan (Front Side)

1. Place thin-wall object **on top** of auxiliary object.
2. Select **Rapid Scan → Marker Alignment**.
3. Scan entire front surface.

4. Second Scan (Back Side)

1. Turn the object upside down.
2. Click **New Project Within Group** (multi-scan workflow).
3. Scan back side completely.

5. Align Both Projects

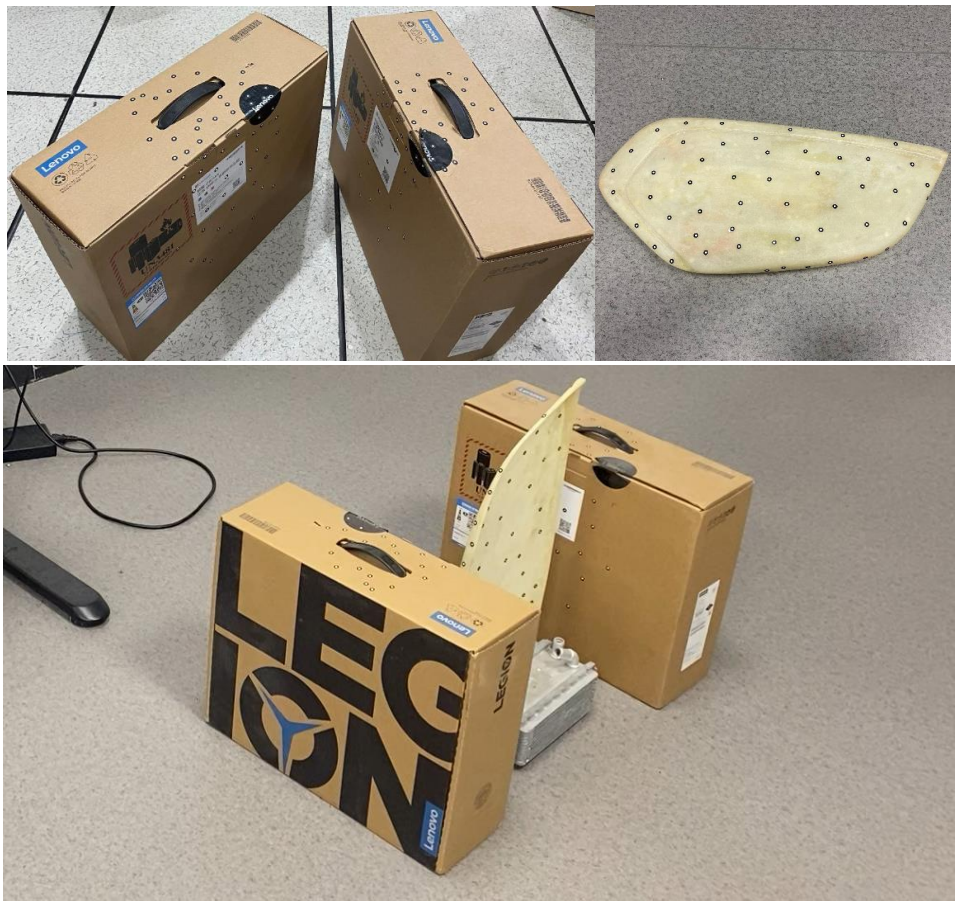
1. Click **Align**.
2. Try **Feature Align** first.
3. If not successful → use **Manual Align**:
    - Select 3 common points across two projects
    - Points must **not be colinear**

6. Generate Point Cloud & Mesh

Same as method 1.

Specification Sheet

Setup of auxiliary objects and thin wall object



Self-Check Questions (Learning Outcome 1: Perform 3D Scanning)

1. Explain why scanner calibration is required before performing a 3D scan.
2. Which scanning mode is most suitable for shiny or reflective objects, and why?
3. Why is matte scanning spray applied on glossy or reflective surfaces before scanning?
4. What does the green distance bar indicate during scanning, and why is it important?
5. When should marker alignment be used instead of feature alignment during scanning?
6. Why must brightness be adjusted correctly in preview mode before scanning?
7. Describe the proper workflow for scanning an object that requires capturing both the front and back sides.
8. What is the most commonly used export file format for scanned meshes, and what is it used for?
9. What factors can cause “Track Lost” during handheld scanning?
10. Describe the ideal environmental conditions needed for successful 3D scanning.

Answer Key

1. Scanner calibration ensures accurate internal camera alignment and correct depth measurement, resulting in precise scan data.

2. Laser Scan mode is preferred for shiny or reflective objects because it provides stable tracking and accurate geometry capture on low-feature or reflective surfaces.

3. Matte scanning spray reduces surface glare and reflection, allowing the scanner to capture accurate geometry without losing data.

4. A green distance bar indicates the scanner is at the optimal distance from the object, ensuring accurate geometry capture and stable tracking.

5. Marker alignment is used when the object has few geometric features, smooth surfaces, or thin walls where feature alignment would fail.

6. Brightness must be adjusted to prevent overexposure or underexposure, which can cause missing or distorted scan data.

7. First scan the front side, then flip the object and scan the back side. Afterwards, align both scans using feature or manual alignment tools.

8. STL is the most commonly exported format; it represents the mesh and is used for post-processing, editing, or preparing the model for 3D printing.

9. Track Lost can occur due to sudden scanner movement, insufficient surface features, excessive reflections, or loss of alignment markers.

10. Indoor, evenly lit environment with no direct sunlight or strong reflections; scanning surface should be stable and non-reflective.

6.2 Learning Outcome: Perform post processing of scanned files

Information Sheet: Installation & Licensing of Geomagic Design X

1. System Requirements (Pre-Check)

Before installation, ensure your PC meets minimum requirements:

- **OS**: Windows 10/11 (64-bit)
- **RAM**: 16–32 GB recommended
- **GPU**: NVIDIA graphics card with updated drivers
- **Storage**: SSD with at least 10 GB free
- **Internet**: Required for activation
  USB Port: Needed if using hardware dongle license

2. Download the Installer

1. Visit **3D Systems Customer Portal** or use the official download link provided by your reseller.
2. Log in with your customer account.
3. Download the **latest Geomagic Design X installer** (EXE).
4. Save it to your computer.

3. Install Geomagic Design X

1. Right-click the installer → **Run as Administrator**.
2. Accept the license agreement.
3. Choose the installation path (default recommended).
4. Select:
   - **Full installation**
   - Required components and drivers
5. Click **Install** and wait for the process to complete.
6. Restart your computer when prompted.

4. Launch the Software

- Open **Geomagic Design X** from the desktop or start menu.
- On first launch, the **Licensing Wizard** will appear automatically.

5. License Activation Methods

Geomagic Design X supports three main licensing modes:

A. Online Activation (Most Common)

Used when you have a serial number / activation ID.

1. In the Licensing Wizard → choose **Online Activation**.
2. Enter:
   - **Product Activation Code**

    - User credentials (if required)
3. Click **Activate**.
4. The software connects to the licensing server and confirms activation.
5. Restart Design X.

Note: Internet connection required during activation.

B. Offline Activation (No Internet PC)

Used when the workstation is offline.

1. Licensing Wizard → **Offline Activation**.
2. Save the **request file (.req)**.
3. Move it to an internet-enabled PC.
4. Upload the file to the **Geomagic Licensing Portal**.
5. Download the **license file (.lic)** generated.
6. Transfer it back to the offline PC.
7. Load the license file into Design X → **Activate Offline**.

C. Dongle / Hardware Key License

Some installations use a USB dongle.

1. Plug the USB dongle into the PC.
2. Install the **dongle driver** if prompted.
3. Start Design X → it automatically detects the dongle.
4. No further activation required.

Note: The software only runs while the dongle is connected.

6. Verify License Status

Inside the software:

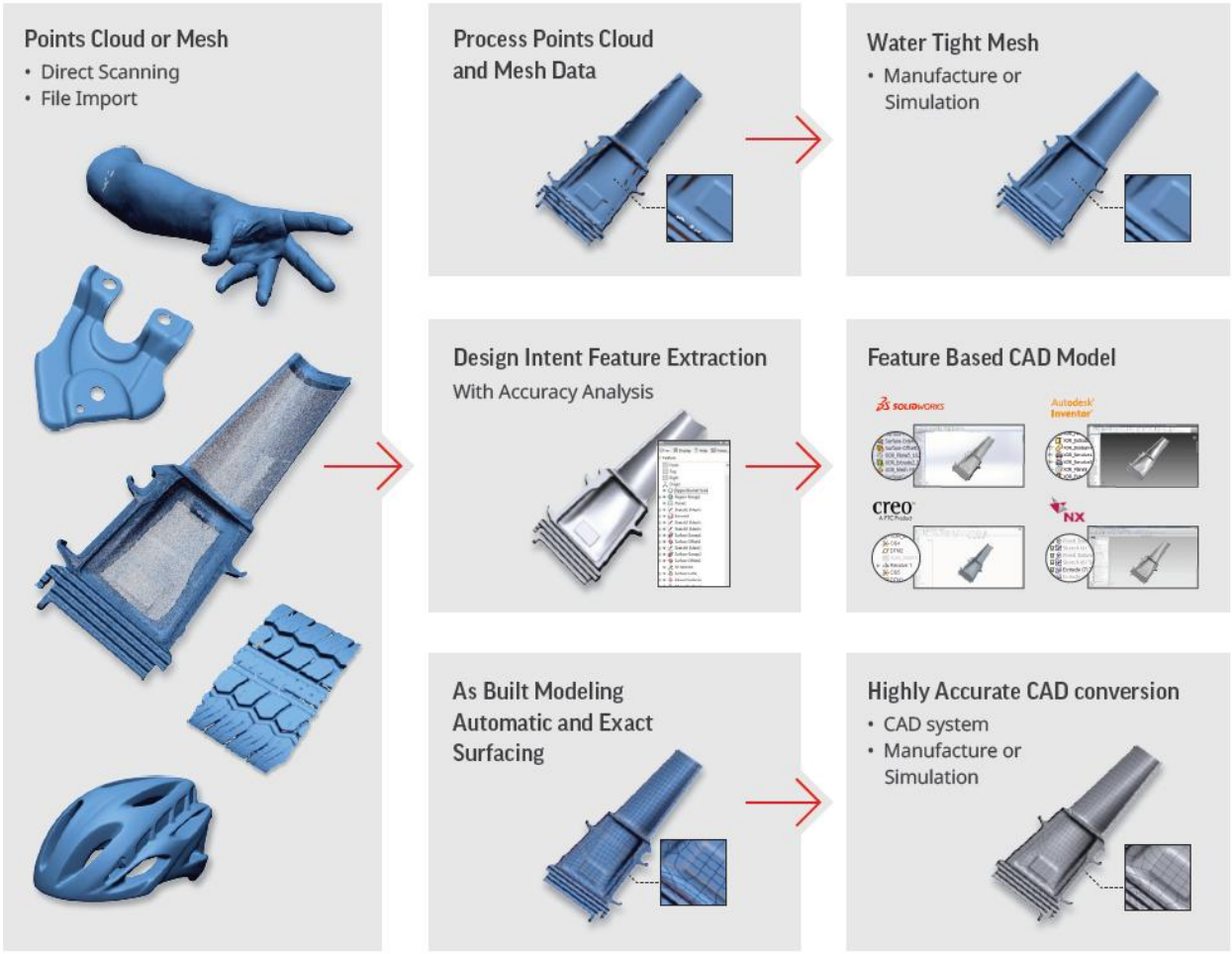
- Go to **Help → About → Licensing**
- Check:
    - Activation validity
    - Expiry date (for subscription licenses)
    - License type (floating, node-locked, dongle)

7. Deactivation (When Changing Computer)

If moving to a new PC:

1. Go to **Help → Licensing → Deactivate License**.
2. Confirm deactivation.
3. Reinstall and activate on the new PC.

1. **Information Sheet: Scan-To-CAD Reverse Design Workflow Using Geomagic**



The process of taking a polygonal STL file through post-processing in Geomagic and converting it into a CAD-ready STEP file is known as **Reverse Design** or **Scan-To-CAD**. This workflow uses Geomagic Design X, a software solution designed to create fully parametric CAD models from 3D scan data. The fundamental steps involve preparing the mesh data, extracting geometric design intent, generating solid or surface geometry, and finally exporting the resulting CAD model.

1. **Information Sheet: Import and Initial Data Handling (STL Mesh)**

The process begins by importing the STL file, which is a common format for polygonal mesh data.
1. **Import:** Use the Insert > Import command in Geomagic Design X to bring the STL file into the application. Importing scan data is registered as a feature in the history tree. \text{Placeholder Image: The following image shows imported scan data in the application.} \text{}
2. **Mesh Terminology:** The imported STL file is a mesh defined by basic entities:
   - **Poly-Face:** Triangles enclosed by three poly-edges.
   - **Poly-Edge:** The connecting edge between two poly-vertices.
   - **Poly-Vertex:** A point representing a position in 3D space.
   - **Boundary:** Any opening in the mesh where triangles are missing.

1. **Information Sheet: Pre-Processing and Mesh Editing**

Before creating precise CAD features, the mesh typically requires optimization and healing, especially if it contains defects or noise. Geomagic Design X offers comprehensive mesh operation tools.
• **Alignment:** If the STL originated from multiple scans, they must be aligned. Even a single scan should be aligned to an ideal design coordinate system (Global Coordinate System) using tools like the **Align Wizard™** (Tools > Align > Wizard).
• **Healing and Defect Removal:** Use the **Healing Wizard** (Tools > Mesh Tools > Healing Wizard) to automatically detect and correct various abnormal poly-faces, such as folded, tangled, or reversed triangles, or small noisy clusters.
• **Hole Filling:** Fill missing holes (boundaries) in the mesh using the Tools > Mesh Tools > Fill Holes command. This ensures a watertight model, which is often crucial for downstream manufacturing applications.
• **Optimization/Simplification:** Depending on the desired result (e.g., preparing for FEA or modeling high-quality surfaces), tools can be used to optimize the mesh structure:
◦ **Decimate Meshes** (Tools > Scan Tools > Decimate Meshes) reduces the number of poly-faces while preserving geometric features.
◦ **Global Remesh** (Tools > Mesh Tools > Global Remesh) globally regenerates poly-faces with uniform poly-edge lengths, improving mesh quality.

1. Information Sheet: Converting Mesh to CAD Model (Reverse Modeling)

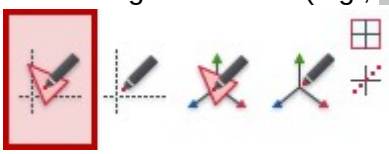
This phase involves transforming the optimized mesh into a **Solid Body** or **Surface Body**. The strategy chosen (Feature Modeling or Surface Fitting) depends on the part's geometry.

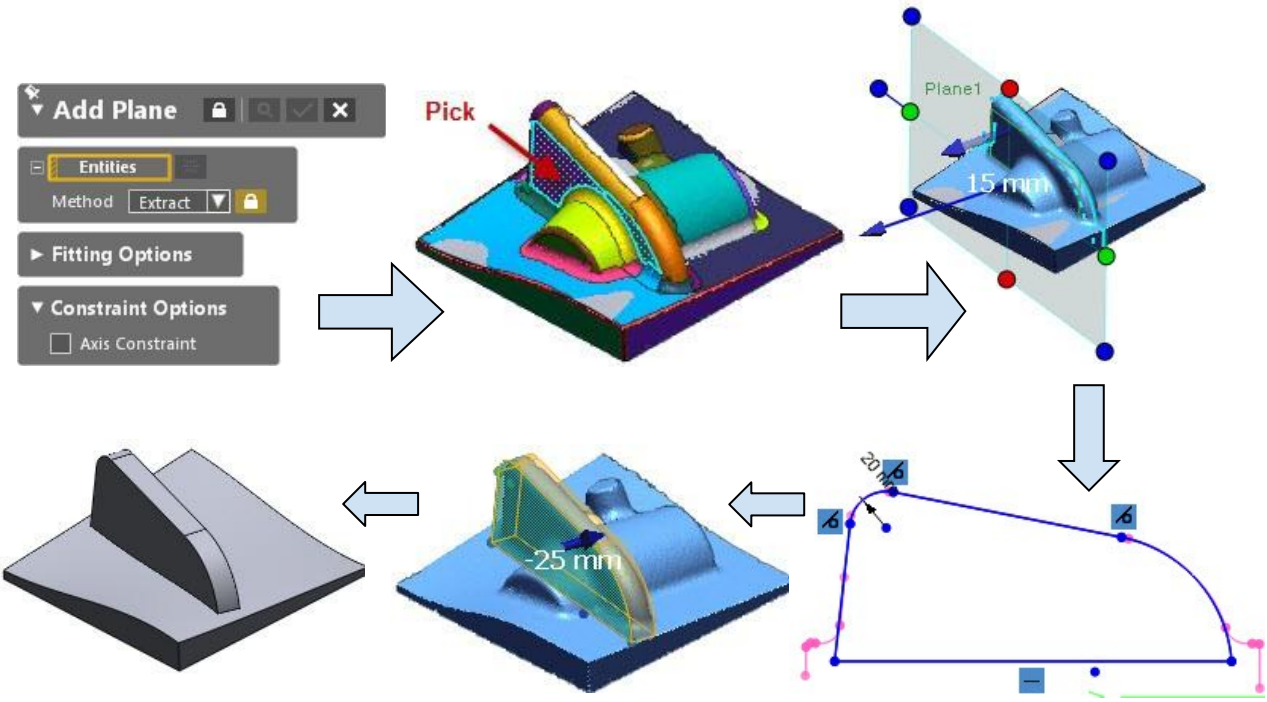
A. Parametric Feature Modeling (For STEP Solid Body Output)

This method focuses on capturing the original design intent of the part.
1. **Region Segmentation:** The mesh is analyzed and segmented into regions corresponding to geometric features (e.g., planes, cylinders) using **Region Group** mode or **Auto Segment** (Tools > Region Tools > Auto Segment).

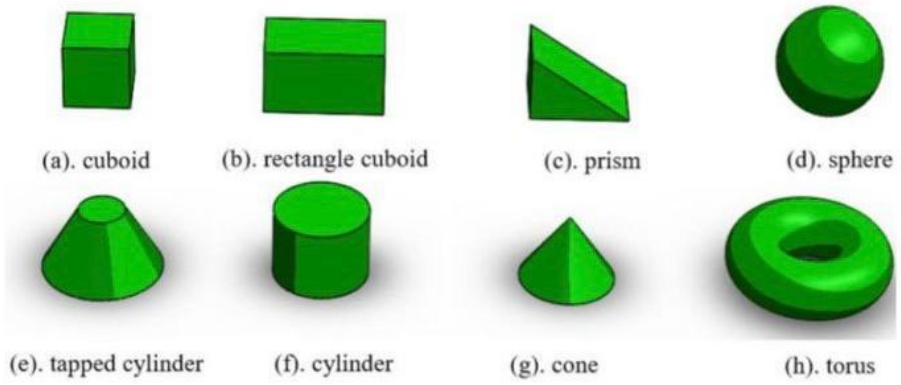


2. **Feature Extraction:** Use extracted regions to generate CAD geometry (referred to as a feature). This can be done using standard CAD features (Insert > Solid or Insert > Surface) or automated wizards:
◦ **Modeling Wizards:** Tools like the **Extrusion Wizard** or **Revolution Wizard** intelligently compute profiles, axes, and paths directly from the mesh regions to generate solid/surface features.
◦ **Sketching:** Create 2D sketches based on mesh sections using **Mesh Sketch** (Insert > Mesh Sketch), which are then used for solid modeling commands (e.g., Extrude, Revolve, Loft).





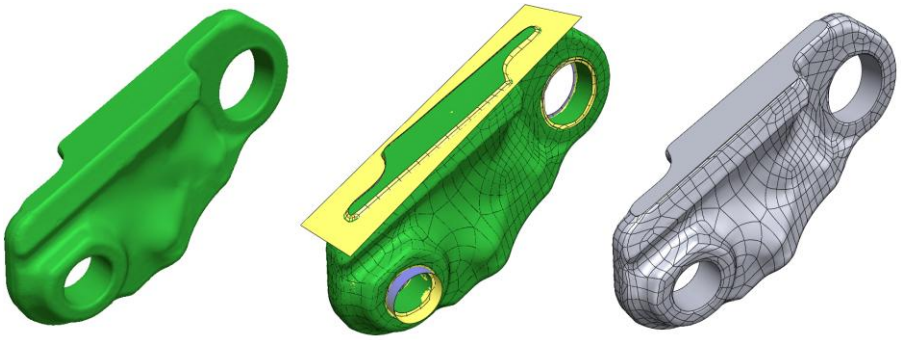
◦ **CAD Primitive:** The mesh can be approximated using cad primitive feature to create similar shape.



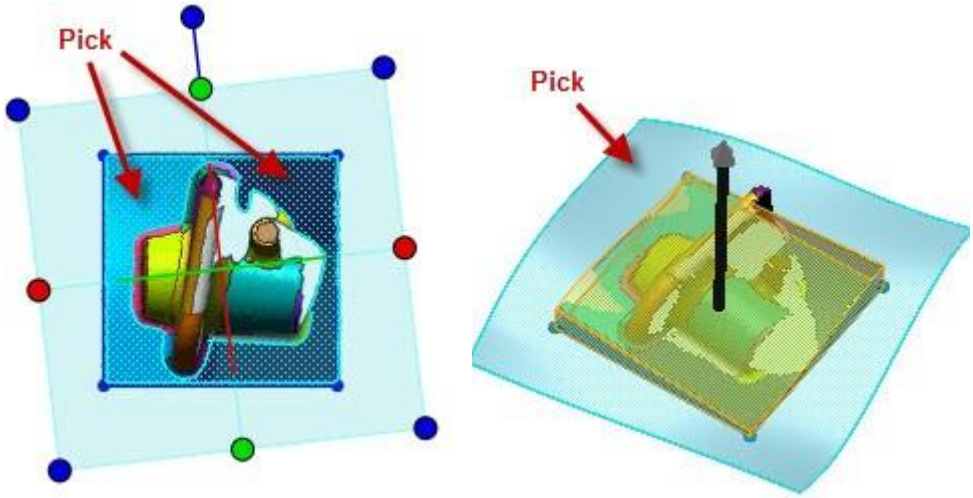
B. Fitting Surface Modeling (For STEP Surface Body Output)

This method is used primarily for complex freeform shapes, providing a verbatim copy of the scan.

• **Auto Surfacing:** Automatically generates a surface body by fitting surface patches onto the mesh (Insert > Surface > Auto Surfacing).



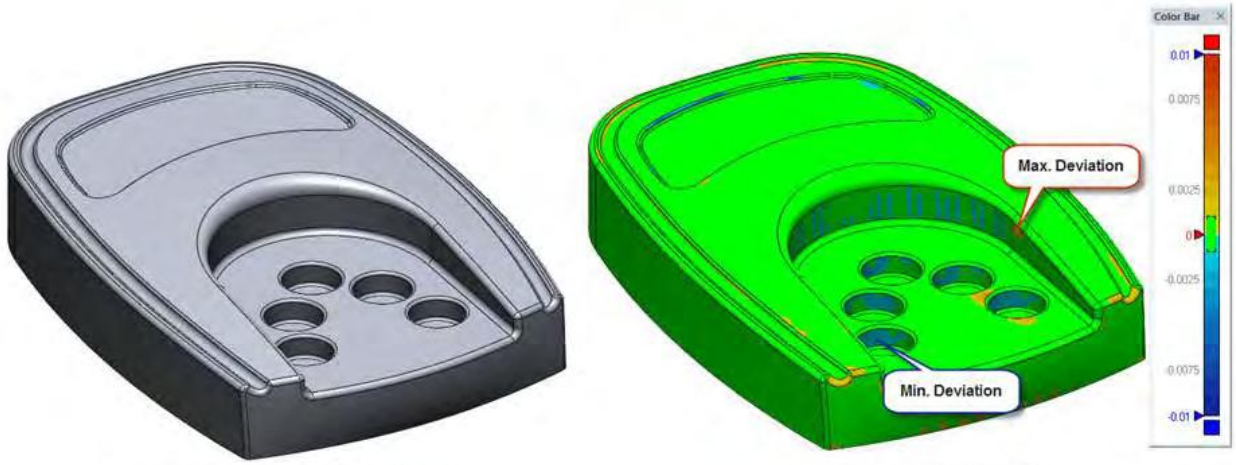
• **Boundary Fit/Mesh Fit:** Allows interactive generation of surfaces by fitting patches to selected mesh areas or boundaries.



The final model must result in a **Solid Body** (a body with closed surfaces enclosing a volume) for optimal use in CAD downstream applications.

1. **Information Sheet: Model Verification**

Throughout the reverse modeling process, the **Accuracy Analyzer™** is used to verify that the generated CAD features deviate minimally from the original mesh data, ensuring the model is within allowable design tolerances.



Reverse Designed Model

Accuracy Verification

Exporting to STEP

Once the parametric model (Solid Body or Surface Body) is complete and verified, it can be exported to the STEP format.
1. Go to File > Export.
2. Select the relevant Solid Body or Surface Body entity.
3. Choose **STEP (.stp)** as the output file format.
STEP is a standard file format that supports the transfer of geometry (points, curves, and surfaces).
Alternatively, Geomagic Design X offers **LiveTransfer™** to send the parametric model, including its full modeling history, directly into supported CAD systems such as SolidWorks or Siemens NX.



Job Sheet: Align Multiple Scans in Geomagic Design X

Equipment, Tools, and Materials

- Computer with **Geomagic Design X** installed
- Imported scans (STL, OBJ, PLY, or point cloud formats)
- Multiple scan datasets representing different sides of the object
- Mouse and keyboard
- Stable workstation with recommended GPU (CBLM page 32)
  3D scanning CBLM_2

Procedure

Multiple scans are aligned to produce a complete model when the object has occluded areas or requires scanning from several orientations. Geomagic Design X provides both **automatic alignment** and **manual alignment** workflows.

---

1. Import Multiple Scan Files

1. Open **Geomagic Design X**.
2. Click **Insert → Import**.
3. Select all scan files (e.g., front scan, back scan, top scan).
4. Imported datasets appear as separate mesh bodies in the Model Tree.
5. Rename each scan for clarity (e.g., “Front Mesh”, “Back Mesh”).

---

2. Pre-Process Each Scan (Optional but Recommended)

1. Remove noise using **Tools → Mesh Tools → Select/Remove Noise**.
2. Repair defects using **Healing Wizard** (CBLM page 35)
   3D scanning CBLM_2
   .
3. Use **Fill Holes** only for structural errors; avoid modifying shared edges.

This ensures clean data before alignment.

---

3. Use the Align Wizard (Automatic Best-Fit Alignment)

1. Go to **Tools → Align → Align Wizard**.
2. Select the **Reference Mesh** (the fixed dataset).
3. Select **Other Scan(s)** as Floating Meshes.
4. Choose alignment strategy:
   - **Best-Fit Alignment** (default)
   - **3-Point Alignment**
   - **N-Point Pairs Alignment**
   - **Global Registration** (for multiple scans)

5. Click **Apply**.

Best-fit uses iterative closest point (ICP) to automatically align all common geometry.

---

4. Manual Alignment (If Automatic Alignment Fails)

Thin-walled or low-feature objects often require manual alignment.

4.1 3-Point Manual Alignment

1. Go to **Tools → Align → Manual Alignment**.
2. Select **Reference Mesh** and **Floating Mesh**.
3. Pick **3 matching points** on each scan:
    - Choose distinct geometric points
    - Avoid points on straight lines
4. Click **Apply** to generate initial alignment.
5. Use **Best-Fit** after manual alignment to refine.

4.2 Global Registration

1. Go to **Tools → Align → Global Registration**.
2. Set:
    - **Max error tolerance**
    - **Overlap percentage**
3. Run alignment to refine all scans simultaneously.

(Global alignment is similar to marker-based recomputation in EXScan, page 27)

3D scanning CBLM_2

.

---

5. Inspect Alignment Accuracy

1. Use **Tools → Accuracy Analyzer** (CBLM page 38) to view deviation heat map.
2. Verify alignment error is within acceptable tolerance:
    - ≤ **0.2 mm** for high-accuracy reverse engineering
    - ≤ **0.5 mm** for general modeling
3. Adjust alignment if error is high:
    - Add more manual alignment points
    - Re-run Best-Fit
    - Remove noisy regions causing mismatch

6. Merge Aligned Scans

After alignment is validated:

1. Go to **Edit → Combine** or **Tools → Mesh → Merge Meshes**.
2. Choose:
    - **Merge into Single Mesh**
    - **Maintain separate regions** (optional)
3. Apply smoothing cautiously to avoid losing fine geometry.

---

7. Final Mesh Optimization

Before exporting:

1. Use **Global Remesh** to create uniform triangles (CBLM page 35).
2. Use **Decimate Mesh** to reduce polygon count if needed.
3. Use **Hole Fill** or **Sharp Edge Enhance** if gaps remain.

---

8. Save and Export

1. Save the project file (*.xmp*).
2. Export final mesh:
    - **STL** → for CAD or 3D printing
    - **OBJ/PLY** → if color required

Job Sheet: Repairing Mesh Errors in Geomagic Design X

Equipment, Tools, and Materials

- Computer with **Geomagic Design X** installed
- One or more imported mesh files (from STL/OBJ/PLY scans)
- Mouse and keyboard
- Stable workstation with suitable graphics card (NVIDIA recommended; CBLM page 32)
  3D scanning CBLM_2

Procedure

Mesh repair is essential before converting scans into CAD-ready geometry. Scanned files often include **holes, noise, non-manifold edges, intersecting faces, flipped triangles, or rough surfaces**. This workflow follows the recommended Repair → Optimize → Validate sequence.

---

1. Import Mesh

1. Open **Geomagic Design X**.
2. Go to **Insert → Import**.
3. Choose your STL/OBJ/PLY mesh file.
4. The imported mesh appears in the Model Tree.

5. Inspect mesh visually by rotating and zooming.

---

2. Identify Mesh Errors

Use the integrated error-detection tools:

1. Go to **Tools → Mesh Tools**.
2. Inspect mesh for:
    - Cracks or open boundaries
    - Noise clusters
    - Folded, tangled, or reversed faces
    - Self-intersections
    - Small missing areas
    - Texture misalignment (if textured scan)

Your CBLM notes these as common defects, especially after handheld scanning or hole-filled STL creation (page 29)

3D scanning CBLM_2

.

---

3. Automatic Repair Using Healing Wizard

1. Navigate to **Tools → Mesh Tools → Healing Wizard**.
2. Select the target mesh.
3. Choose automatic options:
    - **Remove Small Components**
    - **Fix Non-Manifold Edges**
    - **Repair Self-Intersections**

- **Fix Inverted Normals**
- **Fill Defect Holes**

4. Click **Apply**.

The Healing Wizard automatically detects and repairs errors identified on CBLM page 35 (folded, tangled poly-faces, noise, reversed triangles, etc.)

3D scanning CBLM_2

.

---

4. Manual Repair (For Critical Defects)

4.1 Remove Noise

1. Use **Select → Brush/Polygon Select**.
2. Highlight floating or undesired mesh clusters (red highlight).
3. Press **Delete**.

(Noise removal technique referenced on page 27)

3D scanning CBLM_2

.

4.2 Repair Holes

1. Go to **Tools → Mesh Tools → Fill Holes**.
2. Select hole edges.
3. Choose fill method:
   - **Curvature Fill** → for organic shapes
   - **Flat Fill** → for planar areas

- **Tangent Fill** → for blended surfaces

4. Apply fill.

Hole filling tools described on page 29.

3D scanning CBLM_2

4.3 Remove Bad Regions

1. Use **Mesh Doctor** or **Defect Detection**.
2. Highlight regions with:
    - Overlapping surfaces
    - Self-contact
    - Crumpled geometry
3. Delete or remesh problem zones.

4.4 Smooth/Sharpen Surface

1. Go to **Tools → Mesh Tools → Smooth**.
2. Set:
    - Strength: low to medium
    - Preserve edges: ON
3. Apply to reduce noise.
4. For sharp mechanical edges:
    - Use **Sharpen Tool** to restore crispness.

(Note: Over-smoothing removes fine detail — CBLM cautions about surface smoothing on page 29.)

3D scanning CBLM_2

---

5. Global Remesh & Decimation

To ensure a uniform mesh suitable for CAD conversion:

5.1 Global Remesh

1. Go to **Tools → Mesh Tools → Global Remesh**.
2. Set uniform triangle size.
3. Apply to regenerate mesh with consistent poly-edge lengths (page 35).
   3D scanning CBLM_2

5.2 Decimate Mesh

1. Go to **Tools → Scan Tools → Decimate Meshes**.
2. Reduce polygon count while preserving shape.

(Useful for large meshes; referenced in CBLM page 35.)

3D scanning CBLM_2

---

6. Validate Results Using Accuracy Analyzer

1. Go to **Tools → Accuracy Analyzer**.
2. Compare repaired mesh against original data (if point cloud is available).
3. Identify areas of high deviation.
4. Re-tweak mesh repair as needed.

Accuracy Analyzer described in CBLM page 38.

3D scanning CBLM_2

---

7. Save and Export

1. Save the project (*.xmp*).

2. Export repaired mesh:
    - **STL** → for downstream CAD
    - **OBJ/PLY** → for texture
3. Ensure file name matches version & revision.

Job Sheet: Convert Scanned STL File to STEP

Equipment, Tools, and Materials

- Computer with **Geomagic Design X** installed
- Clean, repaired **STL mesh** file from 3D scanning
- Mouse and keyboard
- Stable workstation with recommended GPU (CBLM page 32)
  3D scanning CBLM_2
- Optional: Point cloud and original mesh for accuracy comparison

---

Procedure

This workflow converts a raw scanned mesh (STL) into a **fully parametric CAD model** or **surface-based STEP file** using the Reverse Design workflow in Geomagic Design X.

---

1. Import the STL File

1. Open **Geomagic Design X**.
2. Go to **Insert → Import**.
3. Select the scanned **STL** mesh.
4. The mesh appears as “Scan Data” in the Model Tree.
5. Verify mesh orientation and scale.

(STL import explained on CBLM page 34)

3D scanning CBLM_2

---

2. Inspect and Repair the Mesh (If Needed)

Use mesh editing tools:

1. **Healing Wizard**
    - Tools → Mesh Tools → Healing Wizard
    - Fix holes, inverted triangles, self-intersections (page 35).
2. **Fill Holes**
    - Tools → Mesh Tools → Fill Holes
3. **Smooth / Sharpen**
4. **Simplify Mesh** if polygon count is too high.

A clean mesh is required before surface/feature extraction.

---

3. Align Mesh to Global Coordinate System

1. Use **Align Wizard**
    - Tools → Align → Wizard
2. Fit reference geometry:
    - Best-fit plane
    - Cylinder axis
    - 3-2-1 Alignment
3. Ensure X-Y-Z alignment matches intended CAD orientation.

(Alignment fundamentals in CBLM pages 35–36)

3D scanning CBLM_2

Choose One of Two Conversion Methods

METHOD A — Parametric CAD Modeling (Preferred for Mechanical Parts)

(Yields a parametric, editable STEP Solid)

4A. Auto Segment the Mesh

1. Go to **Tools → Region Tools → Auto Segment**.
2. Mesh gets divided into regions:
    - Planar
    - Cylindrical
    - Fillet
    - Freeform

(Defined on CBLM page 36)

3D scanning CBLM_2

5A. Extract CAD Features

Use the region data to create solid CAD features:

1. **Fit Plane / Cylinder / Sphere / Cone**
2. **Mesh Sketch**
    - Insert → Mesh Sketch
    - Extract profiles using spline, line, circle
3. **Parametric Features**:
    - Extrude

- Revolve
- Loft
- Sweep

4. Add finishing features:
    - Fillets
    - Chamfers
    - Shells
    - Mirror / Pattern

(All supported by DX CAD tools in pages 36–37.)

3D scanning CBLM_2

6A. Validate Accuracy

1. Use **Accuracy Analyzer**
    - Tools → Accuracy Analyzer
2. Compare Solid Model vs Mesh.
3. Deviation tolerance:
    - ≤**0.2 mm** high accuracy
    - ≤**0.5 mm** normal scanning

(Accuracy Analyzer explained on page 38.)

3D scanning CBLM_2

7A. Convert to Solid & Prepare for Export

Ensure:

- All surfaces are closed
- Model forms a **Solid Body** (not open)
- No gaps or floating surfaces exist

METHOD B — Auto Surfacing (Best for Freeform / Organic Parts)

(Yields a Surface-based STEP file)

4B. Use Auto Surfacing

1. Go to **Insert → Surface → Auto Surfacing**.
2. Select the entire mesh.
3. Choose level:
    - **High Precision** → more patches, smoother transitions
    - **Balanced** → faster, good general quality

Auto Surfacing fits **NURBS patches** over the mesh automatically (page 37).

3D scanning CBLM_2

5B. Check and Edit Surfaces

1. Inspect continuity.
2. Fix misaligned patches with **Boundary Fit** or **Surface Edit tools**.

6B. Validate Surface Accuracy

Use **Accuracy Analyzer** as in Method A.

Final Export

8. Export as STEP File

1. Go to **File → Export**.
2. Select:
    - **Solid Body** (Method A)
    - **Surface Body** (Method B)
3. Choose **STEP (*.stp / *.step)**.
4. Select:
    - **AP203** (geometry only)
    - **AP214** (geometry + colors)
5. Save the file.

(STEP export referenced on page 38–39.)

3D scanning CBLM_2

Specification Sheet (Technical Requirements)

Input Requirements

- Clean STL file with repaired mesh
- Correctly aligned scan data
- Uniform polygon distribution for best NURBS fitting

CAD Conversion Requirements

- For parametric modeling:
    - Region segmentation required
    - All sketches constrained

    - All surfaces must form closed solid
- For Auto Surfacing:
    - High-quality mesh required
    - Min. deviation ≤0.5 mm

Accuracy Requirements

- Use Accuracy Analyzer for validation
- Deviation must be within tolerance
- No gaps, floating surfaces, or misaligned patches

Output Requirements

- STEP file must be:
    - Watertight for solid
    - Continuous for surface
    - Correct orientation and scale
    - Named using project standard

Self-Check Questions (Learning Outcome 2: Post-Processing of Scanned Files)

1. What is the purpose of using the Healing Wizard in Geomagic Design X?

2. Explain why hole filling is necessary before converting a mesh to a CAD model.

3. What types of mesh defects commonly occur in scanned STL files?

4. Why is aligning multiple scans important in the post-processing process?

5. Describe how manual alignment is performed when automatic alignment fails.

6. What is the goal of using Global Registration during multi-scan processing?

7. Why is Global Remesh or Decimation used after mesh repair?

8. What is the purpose of performing deviation (accuracy) analysis after creating the CAD model or surface model?

9. When exporting the final model, what is the difference between exporting a mesh (STL) and a STEP file?

10. What issues might occur if a mesh is not fully repaired before exporting it for downstream CAD work?

Answer Key

1. The Healing Wizard automatically detects and fixes mesh errors such as holes, non-manifold edges, self-intersections, and inverted triangles, improving mesh quality before CAD conversion.

2. Hole filling ensures the mesh becomes watertight and structurally complete, which is required for operations like remeshing, auto surfacing, and exporting to CAD formats.

3. Common defects include noise clusters, holes, cracks, non-manifold edges, self-intersecting faces, flipped (inverted) normals, and uneven triangle distribution.

4. Aligning multiple scans ensures different views of the object merge correctly into a single, accurate model, covering hidden surfaces and maintaining dimensional continuity.

5. Manual alignment involves selecting matching point pairs (usually three non-colinear points) on two scans to create an initial alignment before refining with best-fit or global registration.

6. Global Registration improves the alignment of all imported scans simultaneously by minimizing overall deviation and optimizing overlap between datasets.

7. Global Remesh and Decimation create uniform polygon sizes and reduce unnecessary mesh density, improving processing speed and ensuring cleaner surfaces for CAD modeling.

8. Deviation analysis compares the mesh and CAD model to verify accuracy, highlight errors, and ensure the generated CAD geometry matches the original scanned shape within acceptable tolerance.

9. STL exports a mesh-only format used for 3D printing and basic visualization, while STEP exports a solid or surface CAD model with parametric or NURBS surfaces suitable for engineering and manufacturing workflows.

10. Unrepaired meshes may cause CAD conversion failures, inaccurate surfaces, poor feature extraction, holes in the CAD body, and errors during further modeling or manufacturing steps.

6.3 Learning Outcome: Print prototype

Job Sheet: 3D Print Scanned STL and Check Dimensional Accuracy

Equipment, Tools, and Materials

- Repaired and finalized **STL file** from 3D scanning
- Slicing software (Cura / PrusaSlicer / Simplify3D)
- FDM 3D printer (PLA/ABS/PETG compatible)
- Filament spool (PLA recommended for prototypes)
- SD card / USB drive / Wi-Fi transfer
- Digital caliper (0.01 mm resolution)
- Bed cleaning tools (scraper, alcohol wipes)
- PPE: safety glasses, heat-resistant gloves

Procedure

1. Import Scanned STL to Slicer

1. Open the slicing software.
2. Import the post-processed STL file.
3. Check the model orientation and scale.
4. Inspect model thickness and overhangs using **layer preview**.

---

2. Configure Print Parameters

1. Select material type (e.g., PLA).
2. Set:

- **Layer height**: 0.15–0.2 mm
- **Infill**: 20–40%
- **Walls**: 2–3 perimeters
- **Supports**: as required
- **Nozzle temp**: 200–210°C (PLA)
- **Bed temp**: 60°C

3. Slice the model and verify toolpaths.

---

3. Generate and Transfer Print File

1. Export **G-code** from slicer.
2. Transfer file to the printer via:
   - SD card
   - USB
   - Wi-Fi
3. Select the file from the printer's menu.

---

4. Prepare Printer and Start Print

1. Clean the build plate.
2. Preheat nozzle and bed.
3. Ensure filament extrudes smoothly.
4. Start the print and observe the **first layer adhesion**.
5. Monitor printing:
   - Nozzle flow
   - Warping

- Layer shifting
- Stringing

6. Allow print to complete and cool.

---

5. Remove Support and Clean Prototype

1. Remove part carefully with a scraper.
2. Detach supports without damaging edges.
3. Perform light surface finishing if required (trimming, sanding).

---

6. Dimensional Accuracy Verification

1. Identify critical dimensions from original scan/design.
2. Use digital caliper to measure:
   - Outer dimensions
   - Inner holes
   - Wall thickness
   - Features such as bosses, ridges, or slots
3. Take **three repeated readings** for accuracy.
4. Compare measurements with:
   - The digital STL dimensions
   - Required tolerance (typical FDM ±0.5 mm)
5. Record deviations.

---

7. Evaluate and Document Deviations

1. Accept part if all dimensions are within tolerance.
2. If deviations exceed limits:
    - Adjust slicer settings (shrinkage compensation, flow rate, wall count)
    - Check printer calibration (steps/mm, bed leveling, nozzle condition)
3. Finalize and store dimensional accuracy report.

Module-7: Performing Basic Maintenance of 3D Printer

Module descriptor:
This unit of competency covers the knowledge, skills and attitudes required to perform basic maintenance of 3D printer. It specifically includes the tasks of identifying and troubleshooting 3D printer issues, and conducting preventive maintenance of the 3D printer.

Nominal Duration: 10 hours

Learning outcome:
7.1 Troubleshoot 3D printer issues
7.2 Conduct preventive maintenance of the 3D printer

7.1 Learning outcome: Troubleshoot 3D printer issues

Learning Objectives:

At the end of this section, a trainee will be able to know –
1. Troubleshoot related to FDM printer and printing
2. Troubleshoot related to MSLA printer and printing

7.1.1 Information sheet: Troubleshoot related to FDM printer and printing

Common FDM 3D printing issues can typically be resolved by adjusting print settings, performing regular maintenance, and ensuring proper calibration. The most frequent problems involve poor adhesion, material flow issues, and print quality defects.

Poor Bed Adhesion / Warping

Symptoms
- Print detaches mid-print;
- corners lift or curl up.

Common Causes
- Bed not level or dirty.
- Incorrect bed temperature.
- Filament shrinks as it cools.

Solutions
- Re-level the bed and adjust Z-offset.
- Clean the bed with IPA or warm soapy water.
- Use adhesives (glue stick, PEI sheet, hairspray).
- Use a heated bed and enclosure to maintain consistent temperature.
- Use a brim or raft in your slicer.



Figure 7.1: Warping

Clogged Nozzle

Symptoms

- No or inconsistent filament extrusion;
- clicking sounds from extruder motor.

Common Causes

- Debris/burnt filament in the nozzle.
- Heat creep (filament softens too high in hot end).
- Nozzle too close to bed on first layer

Solutions

- Heat the nozzle and manually push filament through.
- Perform a "cold pull" to clear contamination.
- Clean with an acupuncture needle.
- Replace the nozzle if severely blocked.

Under-Extrusion

Symptoms

- Gaps in walls/layers;
- weak, thin, or missing sections;
- spongy print

Common Causes

- Clogged nozzle (partial).
- Incorrect flow rate/E-steps.
- Filament tangled or skipping in extruder gear.

Solutions

- Clean the nozzle.
- Calibrate E-steps (steps per mm) and flow rate (extrusion multiplier).
- Ensure filament spool feeds smoothly and is dry.

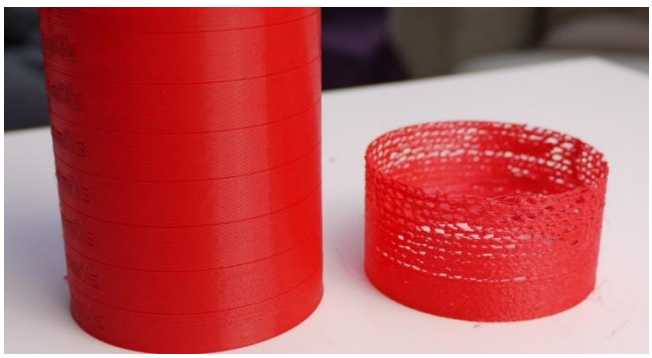


Figure 7.2: Under-Extrusion

Stringing / Oozing

Symptoms

- Fine threads of plastic left between different parts of a print during travel moves.

Common Causes

- Incorrect retraction settings (distance/speed).
- Nozzle temperature is too high.
- Wet or low-quality filament.

Solutions

- Tune retraction settings (increase distance/speed).
- Lower the nozzle temperature in 5°C increments.
- Ensure filament is stored dry.

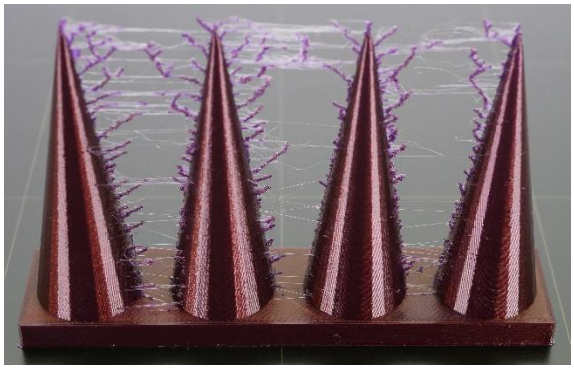


Figure 7.3: Stringing / Oozing

Layer Shifting

Symptoms

- Layers are misaligned, resulting in a skewed or offset print.

Common Causes

- Loose X or Y axis belts.
- Stepper motors overheating or skipping steps.
- Mechanical obstruction blocking movement.

Solutions

- Tighten the belts and check pulleys.
- Ensure smooth movement of axes and remove obstructions.
- Add cooling fans or heatsinks to motors if they are overheating.



Figure 7.4: Layer Shifting

Elephant's Foot

Symptoms

- Bottom layers bulge outward, making the base wider than intended

Common Causes

- Nozzle too close to the bed.
- Bed temperature is too high for the first layers

Solutions

- Adjust the Z-offset slightly higher.
- Lower the bed temperature by 5-10°C after the first layer prints.
- Use "Elephant's Foot Compensation" or "Horizontal Expansion" settings in the slicer.

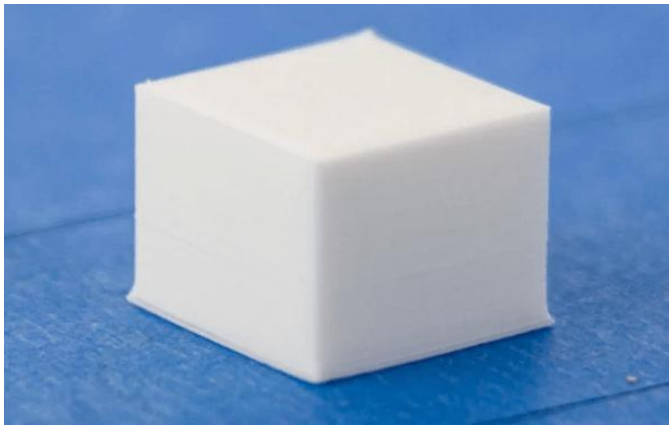


Figure 7.5: Elephant's Foot

Overheating / Sagging

Symptoms

- Drooping layers
- deformed vertical surfaces
- melted appearance on small features.

Common Causes

- Hot end temperature is too high.
- Insufficient cooling (fan speed too low).
- Minimum layer time is too low.

Solutions

- Lower the nozzle temperature.
- Increase the part cooling fan speed.
- Reduce print speed to allow layers to cool.
- Print multiple parts simultaneously to increase layer time.

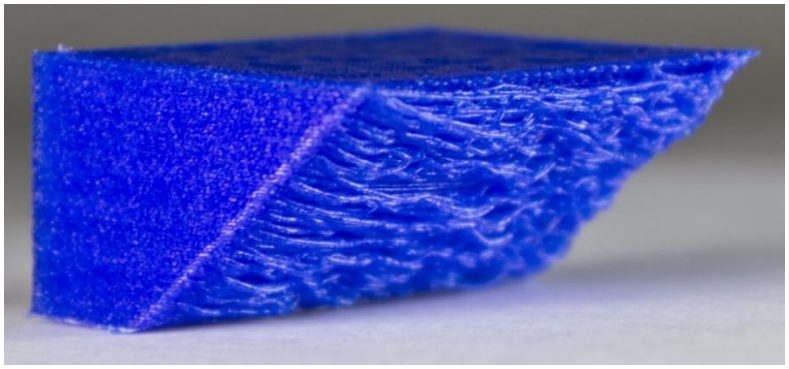


Figure 7.6: Overheating / Sagging

7.1.2 Information sheet: Troubleshoot related to MSLA printer and printing

Common issues in MSLA (Masked Stereolithography) printing relate primarily to adhesion failures, print quality defects, and material inconsistencies. Effective troubleshooting usually involves precise calibration, environment control, and careful post-processing.

Print Fails to Adhere to Build Plate

Symptoms

- Model remains stuck to the FEP film (bottom of the vat)
- build plate comes up empty

Common Causes

- Build plate is not leveled correctly.
- Insufficient bottom layer exposure time.
- Dirty build plate or FEP film.
- Resin temperature is too low.

Solutions

- Re-level the build plate using the manufacturer's specified procedure.
- Increase the bottom exposure time (burn-in layers) in the slicer settings.
- Clean the build plate and FEP with IPA; avoid touching surfaces with bare hands.
- Ensure the ambient temperature is within the resin's recommended range (e.g., 20–30°C).

Layer Separation / Delamination

Symptoms

- Horizontal cracks or splits within the print
- model breaks apart easily

Common Causes

- Insufficient exposure time for normal layers.
- Excessive Z-axis lift speed.
- Rapid temperature fluctuations during print.

Solutions

- Increase normal exposure time to ensure sufficient layer bonding.
- Reduce the Z lift speed in the slicer to minimize peeling forces.
- Add transition layers to smooth the exposure shift between bottom and normal layers.
- Maintain a stable printing environment temperature.

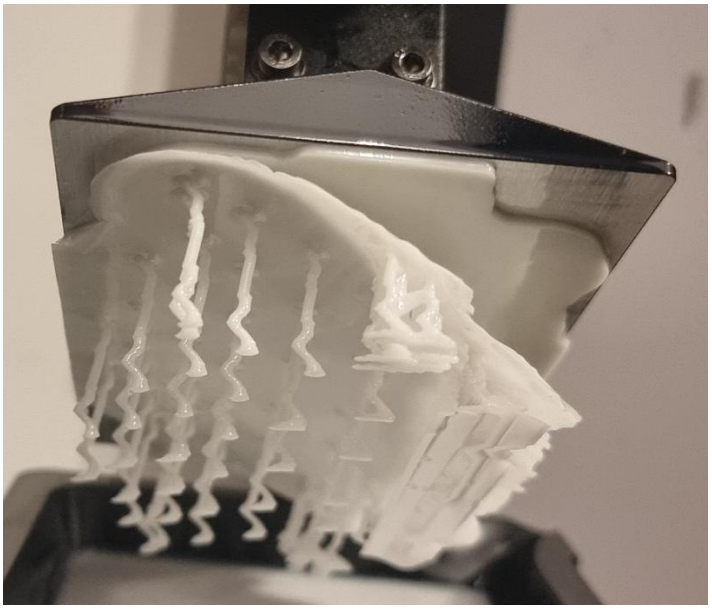


Figure 7.7: Layer Separation / Delamination

Incomplete Prints / Voids

Symptoms

- Specific sections of the model are missing or deformed
- holes appear in solid sections

Common Causes

- Debris in the resin vat blocking UV light.
- Suction cups forming in hollow parts.
- Worn or cloudy FEP film.

Solutions

- Filter the resin after a failed print to remove cured bits.
- Add drainage/ventilation holes to hollow models to prevent pressure differentials.
- Replace the FEP film if it is damaged or heavily clouded.

"Bloating" / Loss of Detail

Symptoms

- Features appear rounded or melted
- dimensions are inaccurate
- holes fill in

Common Causes

- Overexposure time (light bleed).
- Anti-aliasing settings are too high or misconfigured.
- Models are oriented poorly for fine detail features.

Solutions

- Decrease the normal exposure time gradually.
- Adjust anti-alias "Grey Level" and "Image Blur" settings.
- Orient the model to minimize large cross-sections parallel to the build plate.

Weak Supports / Support Failure

Symptoms

- Supports print correctly, but the model detaches from the supports mid-print

Common Causes

- Support contact points are too small.
- Inadequate support density for the model's weight.

Solutions

- Increase the size and depth of support contact points in the slicer.
- Add more supports manually to heavy or delicate areas.
- Decrease Z lift speed to reduce stress on supports.

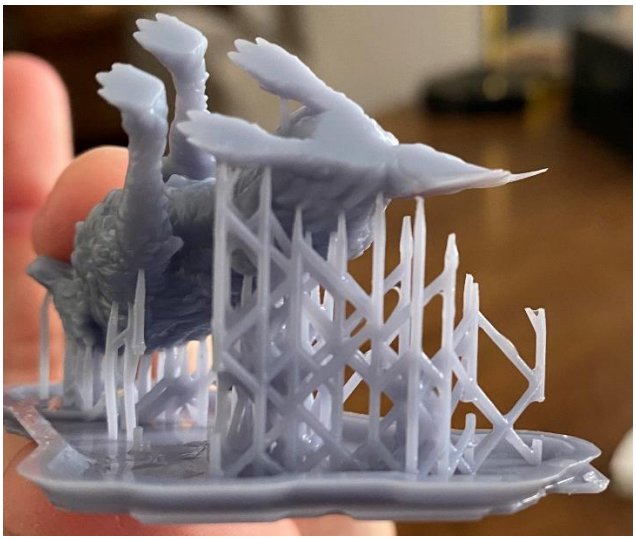


Figure 7.8: Support Failure

Sticky/Tacky Surface After Curing

Symptoms

- Print surface remains sticky or greasy even after post-processing.

Common Causes

- Insufficient washing time or contaminated IPA.
- Print was not completely dry before curing.

Solutions

- Re-wash the part in fresh, clean IPA.
- Ensure the print is completely dry before putting it into the UV station.
- Increase the curing time in a 405nm UV station

7.1.3 Self-check

Write down the most appropriate answers for the following questions:

1. What is the recommended cleaning agent for both FDM build plates and MSLA build plates/FEP film?
2. What FDM printing issue causes fine threads of plastic to be left between printed parts?
3. What are two common causes of MSLA print failures where the model stays stuck to the FEP film instead of the build plate?
4. In FDM printing, what mechanical issue causes layers to become misaligned?
5. Why is adding ventilation holes necessary when printing hollow models in MSLA printing?
6. What is the effect of having the FDM nozzle too close to the bed during the first layer?
7. What MSLA parameter adjustment can help fix layer separation caused by high peeling forces?
8. What post-processing error causes an MSLA print surface to remain sticky or tacky after washing and curing?
9. What is the primary solution for FDM models that are warping or lifting from the build plate?

7.1.4 Answer Key

1. Answer:

Isopropyl alcohol (IPA).

2. Answer:

Stringing (or oozing).

3. Answer:

The build plate is not leveled correctly, or the bottom exposure time is insufficient.

4. Answer:

Loose X or Y axis belts.

5. Answer:

To prevent the formation of a vacuum effect which can cause print failure.

6. Answer:
Poor adhesion, a clogged nozzle, or an "elephant's foot" effect (bulging base).

7. Answer:
Reducing the Z lift speed.

8. Answer:
Insufficient washing or curing time, or residual moisture on the part before curing.

9. Answer:
Ensuring proper bed leveling and temperature, and using adhesives or a brim/raft.

7.2 Learning outcome: Conduct preventive maintenance of the 3D printer

Learning Objectives:

At the end of this section, a trainee will be able to know –

1. Preventive maintenance related to FDM printer and printing
2. Preventive maintenance related to MSLA printer and printing

7.2.1 Information sheet: Preventive maintenance related to FDM printer and printing

As we are using Original Prusa i3 MK3S+ printer throughout the course, there are some mandatory regular maintenance suggested by the manufacturer.

It is recommended to regularly take some time to maintain your printer. This should be done even if the printer is working perfectly. The inspection and maintenance of its various components should be done every couple of hundred hours of print-time.

Tools required

Besides the usual tools, such as Allen keys, needle-nose pliers, and IPA with paper towels, you might need:

- **Lubricant:** the best option is the Prusa lubricant. One is sent with the printer, but it can also be found on internet. Other recommended lubricants are Super Lube 210XX, GLEIT-μ HF 400, and Mogul LV 2-EP. We do not recommend lubricants with a very high viscosity.
- **Compressed air:** also known as canned air. Be sure to not hold it upside down while spraying - otherwise, you could damage any electronic components while trying to clean them.
- **Brush:** a small brass brush is an ideal tool for cleaning the Bondtech filament pulleys. If you don't have a brass brush any brush with hard bristles will do the job. Alternative: You can use the acupuncture needle that comes with the printer or a toothpick.

Axes check

- **Smooth rods:** scratches/grooves - take each rod between two/three fingers and move them slowly from one end of the rod to another, while rotating the fingers around the rod. If you notice any scratch, verify it with your fingernail.
- **Bearings:** Clean and lubricate the bearings and rods as shown this guide. Despite cleaning and lubricating the smooth rods, it can happen that the axes' movement is still not smooth. In this case, please remove the bearings from the printer and lubricate them from the inside with a pea-sized amount of lubricant.
- **Belt pulleys:** any debris around the belt pulleys and belts should be immediately removed to ensure smooth operations. Also, check if the set screw of the belt pulley is firmly secured on the flat side of the motor shaft.
- **Idler screw** (M3x40 with the spring on it) is not overtightened or too loose. Generally, the head of the screw, on the left side, should be about flush with the printed parts of the extruder.
- **Y- and X-axis belts:** Both of the belts should be tight enough to sound like a low bass note when plucked. On the MK3/S/+, the belt status can be checked in the *LCD menu - Support - Belt status*. Recommended values are between 260 and 290.

Fans

The Original Prusa i3 MK3/S/+ can spot if any of the fans stop spinning. However, that does not mean you should not check them during maintenance. Dust or plastic build-up can decrease their efficiency or even damage them.

A cotton swab can also be used to clean off dust and debris from the propeller or canned air will clear the dust. Tweezers can be used to take out any plastic strands. Hold the propeller of the fan to prevent it from spinning while cleaning it with canned air. Otherwise, it can get damaged. Clear away any debris that may be stuck and check that it can rotate freely.

Extruder driver gear

The MK2.5/S and MK3/S/+ printers are equipped with two Bondtech gears, made of hardened carbon steel. One is the pulley gear, secured on the extruder's motor shaft by a set screw. The other is the idler gear, which spins freely mounted on the idler door. If you have had a clog, they can also have collected residue grinding the filament which will, in turn, reduce their grabbing force on the filament.

- Regular cleaning and lubrication will:
- Reduce wear and noise.
- Increase the grabbing force on the filament.
- Lower friction between the two pulley gears.
- Protect against rust.

Electronics

On the MK3/S/+ - Check and reconnect the electrical connectors on the EinsyRAMBo board every 600 - 800 hours of printing.

Check if there are any pinched or worn cables on the printer.

Open the electronics casing and blow compressed air in between the cables to get rid of dust.

Check the connection of all of the wires on the mini RAMBo board or EINSY RAMBo board. Especially pay attention to the cables from the PSU (power source) and heaters. Make sure they are inserted all the way in and well-tightened.

Users of the Original Prusa i3 MK3/S/+ should lightly tug the power and heatbed connectors. If any movement is felt, then re-tighten the screws.

Nozzle change

Putting an exact number on how long a nozzle will last is very difficult, as it depends on how much you print, what types of filament you use, and how often you change between filament types. **It is not the first thing you should look to if you are experiencing extrusion issues**, but if the printer has been in use for more than 3-6 months and you are getting poor print quality, even after doing the mentioned maintenance, you may want to consider replacing the nozzle.

Firmware and software

You should always be using the latest stable firmware version that we have published for your specific printer model. You can check the firmware version you have in the *LCD menu - Support*.

7.2.2 Information sheet: Preventive maintenance related to MSLA printer and printing

Preventive maintenance for an MSLA (Masked Stereolithography) printer focuses on keeping the resin and optical components clean, maintaining the build platform, and ensuring smooth mechanical operation.

Daily/Per-Print Maintenance

- **Clean the Build Platform:** After every print, scrape off any cured resin and wipe the platform clean with isopropyl alcohol (IPA) and paper towels to ensure proper adhesion for the next print.
- **Inspect and Filter Resin:** Check the resin tank (vat) for debris or cured fragments before each print. Use a plastic scraper (not metal, which can damage the film) to remove debris or use the printer's built-in cleaning utility. If significant contamination occurs, filter the resin through a fine mesh filter back into its bottle.
- **Check the FEP Film:** Inspect the transparent film at the bottom of the resin tank for scratches, cloudiness, or damage. A damaged film must be replaced immediately to prevent resin leaks that could damage the internal components, especially the screen.
- **Wipe Down Exterior:** Keep the surrounding work area and printer housing clean to prevent dust or spills from entering the machine.

Monthly/Quarterly Maintenance

- **Deep Clean:** Perform a more thorough cleaning of the printer's interior using compressed air and a soft brush to remove dust and resin particles. Wipe down all surfaces with a lint-free cloth dampened with IPA.
- **Calibrate and Test:**
  - **Platform Leveling:** Regularly check and re-level the build platform to ensure it is parallel to the screen, as an unlevel bed is a primary cause of failed prints and poor adhesion.
  - **Light Source Inspection:** Check the UV light source and any mirrors or lenses for dust or resin residue. Clean them gently with a lint-free cloth to maintain consistent exposure and print quality.
- **Lubricate Moving Parts:** Clean and apply a light coating of the manufacturer-recommended lubricant (such as lithium grease or light machine oil) to the Z-axis lead screw and guide rails to ensure smooth vertical movement.
- **Check Components and Cables:** Inspect all cables for signs of wear or damage, particularly near moving parts. Check the structural components for any signs of cracking or deformation.

Long-Term/Annual Maintenance

- **Replace Consumables:** The FEP film and the UV light source/LCD screen (depending on printer type) will degrade over time and require replacement. Follow the manufacturer's guidelines for expected lifespans (e.g., FEP film replacement may be needed after a few months of heavy use).
- **Update Software:** Regularly check the manufacturer's website for new firmware and slicing software updates to benefit from bug fixes and performance improvements.
- **Overall Performance Test:** Conduct a comprehensive test of the printer's functionality to ensure it is meeting performance standards.

7.2.3 Self-check

Write down the most appropriate answers for the following questions:

1. What is the recommended frequency for inspecting and maintaining FDM printer components?
2. What is the recommended status value range for the X and Y belts on the Original Prusa i3 MK3S+?
3. What is the primary purpose of lubricating the Z-axis lead screw and guide rails on both FDM and MSLA printers?
4. What must be done with leftover liquid resin before it can be disposed of according to regulations?
5. How can FEP film degradation be mitigated through preventive maintenance?

7.2.4 Answer Key

1. Answer:

Every couple of hundred hours of print-time.

2. Answer:

Between 260 and 290.

3. Answer:

To ensure smooth vertical movement and prevent wear.

4. Answer:

It must be fully cured (solidified) using a UV station or sunlight.

5. Answer:

By proactively inspecting the film for cloudiness/scratches and replacing it based on manufacturer guidelines or heavy use.

.