



**Competency-Based Learning Materials (CBLMs)
on
CAD-CAM Design and Programming (Mechanical)
Light Engineering Sector**

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

Index

Introduction.....	01
How to Use This Educational Material.....	02
Authors Information.....	03
Icon List.....	04
Module 1: Understand Mechanical Drawing.....	05
Learning Outcome 1.1: Interpret Mechanical Drawing.....	06
Learning Outcome 1.2: Create Mechanical Drawing.....	43
Module 2: Perform CAD Modeling.....	59
Learning Outcome 2.1: Install CAD Software.....	60
Learning Outcome 2.2: Create 2D Drawing.....	80
Learning Outcome 2.3: Develop 3D Model.....	107
Learning Outcome 2.4: Create Mold Based 3D Model.....	130
Module 3: Select Cutting Tools and Workpiece.....	143
Learning Outcome 3.1: Understand Cutting Tools	144
Learning Outcome 3.2: Select Cutting Tools for Lathe Operations	154
Learning Outcome 3.3: Select Cutting Tools for Milling Operations.....	164
Learning Outcome 3.4: Selection of Cutting Tools According to Appropriate Consistency.....	171
Module 4: Perform CAM Programming.....	176
Learning Outcome 4.1: Install CAM Software.....	177
Learning Outcome 4.2: Use CAM Software Interface.....	186
Learning Outcome 4.3: Carry Out Milling 2D CAM Programming.....	196
Learning Outcome 4.4: Carry Out Milling 3D CAM Programming.....	242
Learning Outcome 4.5: Load and Run Program.....	258
Module 5: Carry Out CNC Milling Machine Operations.....	271
Learning Outcome 5.1: Prepare CNC Milling Machine.....	272
Learning Outcome 5.2: Download and Transfer Program.....	286
Learning Outcome 5.3: Operate CNC Milling Machine.....	291
Learning Outcome 5.4: Check and Measure Work Piece.....	296
Module 6: Carry Out CNC Lathe Machine Operations.....	302
Learning Outcome 6.1: Prepare CNC Lathe Machine.....	303
Learning Outcome 6.2: Download and Transfer Program.....	314
Learning Outcome 6.3: Operate CNC Lathe Machine.....	317
Learning Outcome 6.4: Check and Measure Work Piece.....	320

Introduction

Competency-based learning materials (student guides) for CAD CAM Design and Programming (Mechanical) is a document that is written/aligned with the applicable competency standards, provide training in line with industry requirements, and qualify trained individuals for relevant work through established competency-based assessments.

This document is owned by the Finance Division, Ministry of Finance, Government of the People's Republic of Bangladesh and is prepared under the Skills for Industry Competitiveness and Innovation Program (SICIP).

Government and private institutions will be able to use the information included in the competency-based learning materials for activities that benefit Bangladesh.

Document/Document Receiving Place:

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

Probashi Kallayan Bhavan Building (Level-16),
71-72 Old Elephant Road, Eskaton Road, Dhaka 1000.

Telephone: + 8802 551 38598-9 (PABX), + 8802 551 38753-5

Website: www.sicip-fd.gov.bd

How to Use This Educational Material

This module provides essential training materials and learning activities designed to help you develop the knowledge and skills required for CAD/CAM Design and Programming in the mechanical field. By completing this module, you will gain the competence and proficiency needed to perform effectively as a skilled technician.

The course is made up of the following 6 modules, which include the knowledge, skills, and attitudes necessary for a worker to be effective:

1. Understand mechanical drawing
2. Perform CAD Modeling
3. Select cutting tools and workpiece
4. Perform CAM programming
5. Carry out CNC milling machine operations
6. Carry out CNC lathe machine operations

As a trainee, you are expected to complete a series of tasks in order to achieve each learning outcome of the module. These tasks may be carried out in either real-world or simulated workplace settings for practice purposes.

To meet the learning outcomes, you must engage in the required training and practice activities that build the necessary skills and knowledge. Refer to the module's **Learning Outcome** page for detailed instructions on completing each task, along with the tools and materials needed.

This page acts as your guide to achieving competency. By mastering the information sheet, you will gain a clear understanding of how to perform each task. Once you are confident with the information, you should complete the **Self-Check**.

The **Self-Check** section in this learning guide is designed to follow the information sheet and allows you to assess your progress. After finishing it, review the **Answer Script** located at the end of the module to evaluate your understanding.

You must complete all tasks outlined in the **Information Sheet** and **Learning Outcome**, applying the knowledge you've gained to develop your practical skills. Always prioritize safety while working, and don't hesitate to ask your facilitator or trainer for clarification when needed.

After you have completed all the required activities in this learning guide, a formal assessment will be conducted to determine whether you have met the specified learning outcomes and are ready to proceed to the next module.

CBLM Development Committee

As part of the training program, **Competency-Based Learning Materials (CBLM)** play a vital role in supporting both trainees and trainers by promoting interactive and effective learning.

To develop this CBLM, a dedicated team was formed that included external experts and was led by head of the **PIU, SICIP-BITAC Project**. The team conducted a detailed task analysis based on the performance criteria outlined in the **Competency Standards (CSs)** for **CAD-CAM Design and Programming**, while also considering the needs of the trainees.

Following the task analysis, the components of the CBLM were developed through a series of consultation meetings and workshops. As a result, the **CBLM for CAD-CAM Design and Programming** was successfully created. Finally, a validation meeting was held with subject matter experts, and the material was formally validated on **21 October 2025**.

Experts Involved:

Industry and institutes experts who provided their valuable inputs to develop this Competency Based Learning Material:

Name	Organization	Designation
Md. Rezwan Munshi	BITAC	Assistant Engineer & Lead Trainer, (CAD-CAM Design and Programming)
Md. Humayun Kabir	BITAC	Executive Engineer (CAD-CAM Expert)
Dr. Md. Jalal Uddin PEng.	BITAC	Director (Head PIU, SICIP, BITAC)
Simata Kundu	Esquire Plastics Ltd.	Industry Expert

Icon List

Icon Name	Icon
Module Content	
Learning Outcome	
Performance Criteria	
Content	
Assessment Criteria	
Necessary Material and Instrument	
Information Sheet	
Self-Check	
Answer Script	
Activity/Work	
Job Sheet	
Assessment Plan	
Competency Review	

Module 1: Understand Mechanical Drawing



Module Content

Module Description: This module describes the knowledge, skills and attitudes required of the operator to perform mechanical drawing. It includes understanding mechanical drawings, interpreting mechanical drawings, projection method, recognize different views (Orthographic, Isometric, and Perspective), sectional views, dimension, tolerance, scale, signs and symbols in a standard mechanical drawing.

Minimum Time: 20 Hours.



Learning Outcome:

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

- 1.1 Interpret mechanical drawing.
- 1.2 Create mechanical drawing.



Performance Criteria:

- 1. Different types of mechanical drawings are interpreted.
- 2. Different views of mechanical drawing are identified and described.
- 3. Necessity of putting dimensions in different drawings is comprehended.
- 4. Drawing is created accurately.
- 5. Components, objects, parts, materials and assemblies are identified, drawn and described.
- 6. Base line or datum points are specified, as required.
- 7. Instructions are included in drawing.
- 8. Tolerance, limits and fits are identified in technical drawing.
- 9. Job specification and instructions are identified and followed.

Learning Outcome 1.1: Interpret Mechanical Drawing



Content:

- Types of mechanical drawings
- Visual elements
- Textual and dimensional information.
- Line types



Assessment Criteria

1. Be able to collect and interpret mechanical drawings.
2. Be able to identify signs and signals.
3. Be able to identify terms and abbreviations.
4. Be able to explain specifications job specifications.
5. Be able to explain tolerance, limits and fits are identified in technical drawing.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. Drawing paper
2. T Square, Set Square
3. Pencil, Eraser
4. Compass, Protractor
5. Scale
6. Learning manual



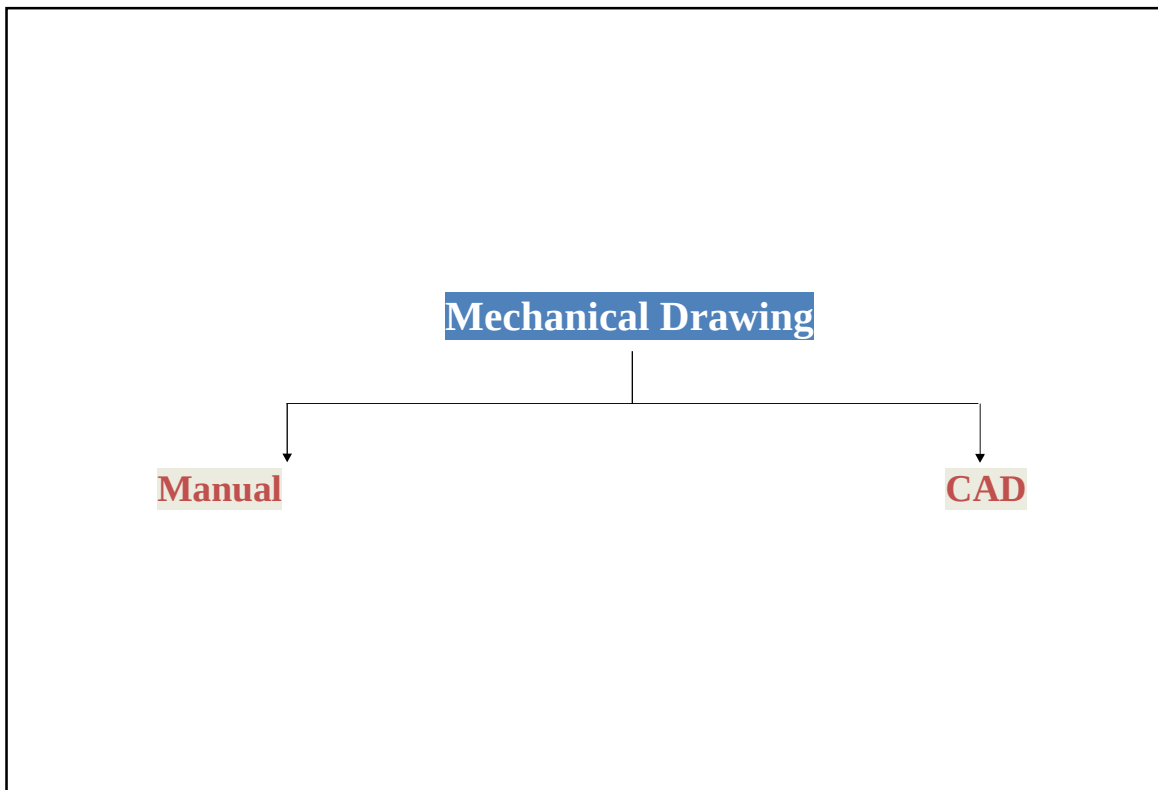
Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Interpret Mechanical Drawing	• Information sheet: 1.1-1 to 1.1-4 • Self-Check: 1.1-1 to 1.1-4 • Answer script: 1.1-1 to 1.1-4 • Engineering Drawing by Dr. Md. Quamrul Islam



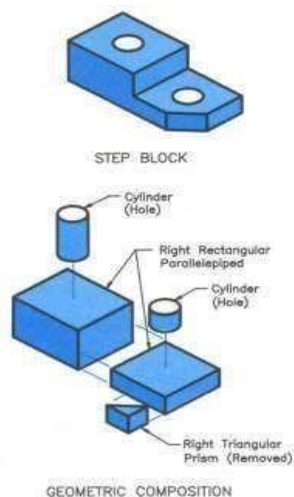
Information Sheet: 1.1-1

Learning Objective: Interpret Mechanical Drawings in the Workplace.

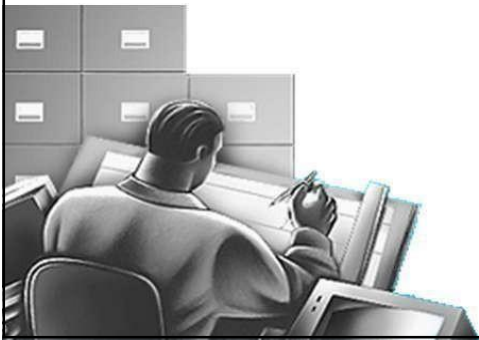


Purpose of Mechanical Drawing

1. An engineering drawing is not an illustration.
2. It is a specification of the size and shape of a part or assembly.
3. The important information on a drawing is the dimension and tolerance of all of its features.



Drawing Standard



Introduction

Standards are set of rules that govern how technical drawings are represented.

Drawing standards are used so that drawings **convey the same meaning to everyone** who reads them.

Standard Code

Country	Code	Full name
USA	ANSI	American National Standard InstituteJIS
Japan		Japanese Industrial Standard British
UK	BS	Standard
Australia	AS	Australian Standard
Germany	DIN	Deutsches Institut fürNormung
	ISO	International Standards Organization

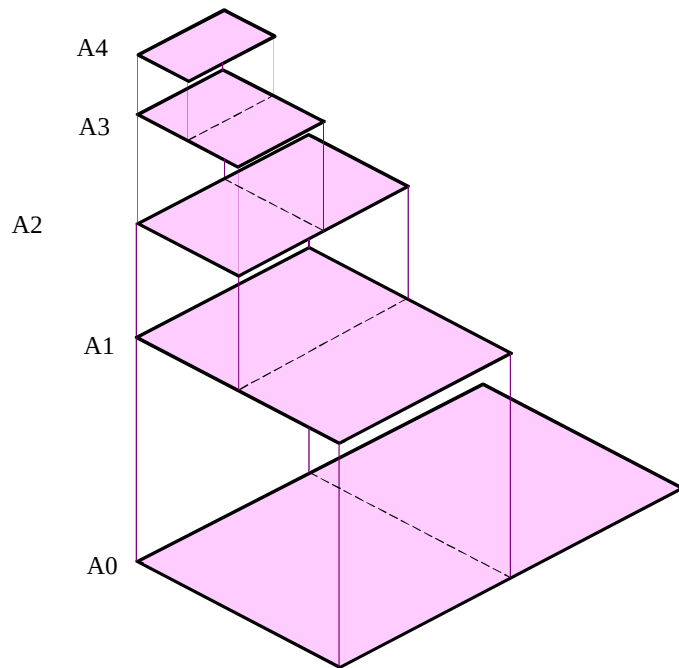
Drawing Sheet

■ Trimmed paper of a size A0 ~ A4.

■ Standard sheet size (JIS)

A4	210 x 297
A3	297 x 420
A2	420 x 594
A1	594 x 841
A0	841 x 1189

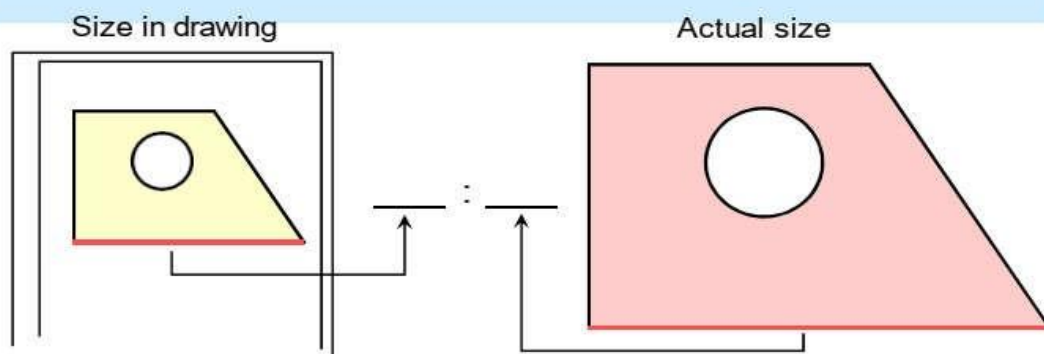
(Dimensions in millimeters)



Drawing Scales

Length, size

Scale is the ratio of the linear dimension of an element of an object shown in the drawing to the real linear dimension of the same element of the object.



Drawing Scales

Designation of a scale consists of the word “**SCALE**” followed by the indication of its **ratio**, as follow

SCALE 1:1 for full size

SCALE **X**:1 for **enlargement scales**

SCALE 1:**X** for **reduction scales**

Dimension numbers shown in the drawing are correspond to “**true size**” of the object and they are **independent of** the scale used in creating that drawing.

Basic Line Types

Types of Lines	Appearance	Name according to application
Continuous thick line		Visible line
Continuous thin line		Dimension line Extension line Leader line
Dash thick line		Hidden line
Chain thin line		Center line

Meaning of Lines

Visible lines represent features that can be seen in the current view

Hidden lines represent features that can not be seen in the current view

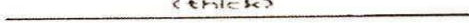
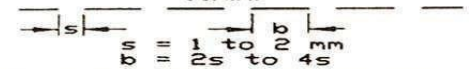
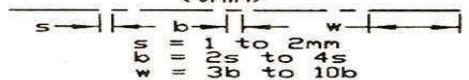
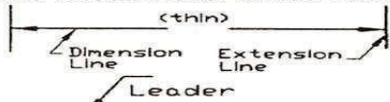
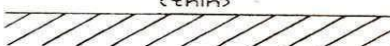
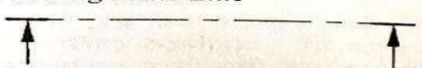
Center line represents symmetry, path of motion, centers of circles, axis of axisymmetric parts

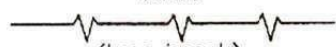
Dimension and Extension lines indicate the sizes and location of features on a drawing

Meaning of Lines

Section Lines solid thin lines that indicate cut surfaces **Cutting- Plane and Viewing-Plane Lines** indicate location of cutting planes for sectional views and the viewing position for removed partial views

Break Lines indicate only portion of object is drawn. May be random “squiggled” line or thin dashes joined by zigzags.

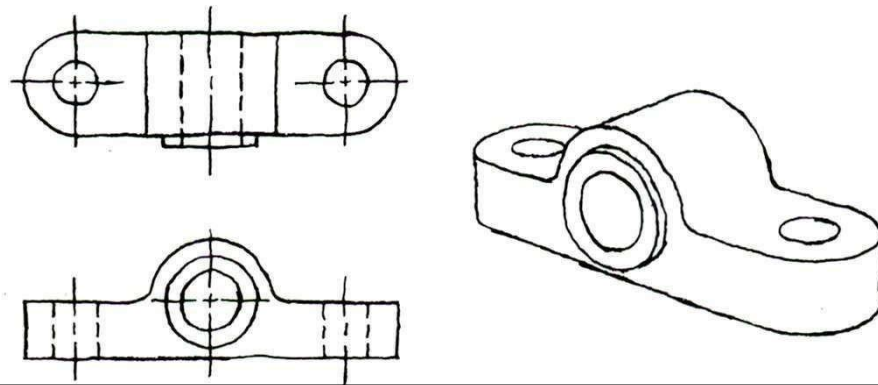
Sl.	Types of Line	Usage
1	Visible line / object line (thick) 	to indicate all visible outlines of an object. It shows the shape of an object.
2	Hidden line / dashed line (thin)  $s = 1 \text{ to } 2 \text{ mm}$ $b = 2s \text{ to } 4s$	to represent the hidden edge of an object. It must begin and end with a dash touching the visible lines. Dashes that show hidden lines usually touch each other at intersection.
3	Center line (thin)  $s = 1 \text{ to } 2\text{mm}$ $b = 2s \text{ to } 4s$ $w = 3b \text{ to } 10b$	to show the center line of holes, pitch line.
4	Extension line, dimension line and Leaders (thin)  Dimension Line Extension Line Leader	to show dimension of an object extension line, dimension line and leaders are used.
5	Section line (thin) 	to indicate the cut portion of an object.
6	Cutting plane line  (thick) or	to show the imaginary cutting of an object
7	ISO Cutting Plane Line 	

Sl.	Types of Line	Usage
8	Break line (thin)  (long break) (thick)  (short break)	to show a break on the object. It shortens the view of a long part.
9	Phantom line/repeat line (thin) 	to show the alternate position of an object or the position of an adjacent part.

Freehand Drawing

The lines are sketched without using instruments other than pencils and erasers.

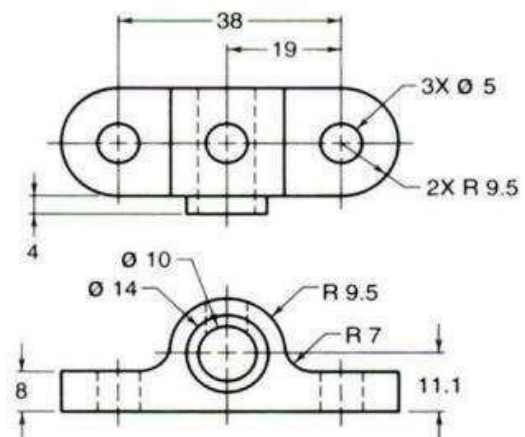
Example



Instrument Drawing

Instruments are used to draw straight lines, circles, and curves concisely and accurately. Thus, the drawings are usually made to scale.

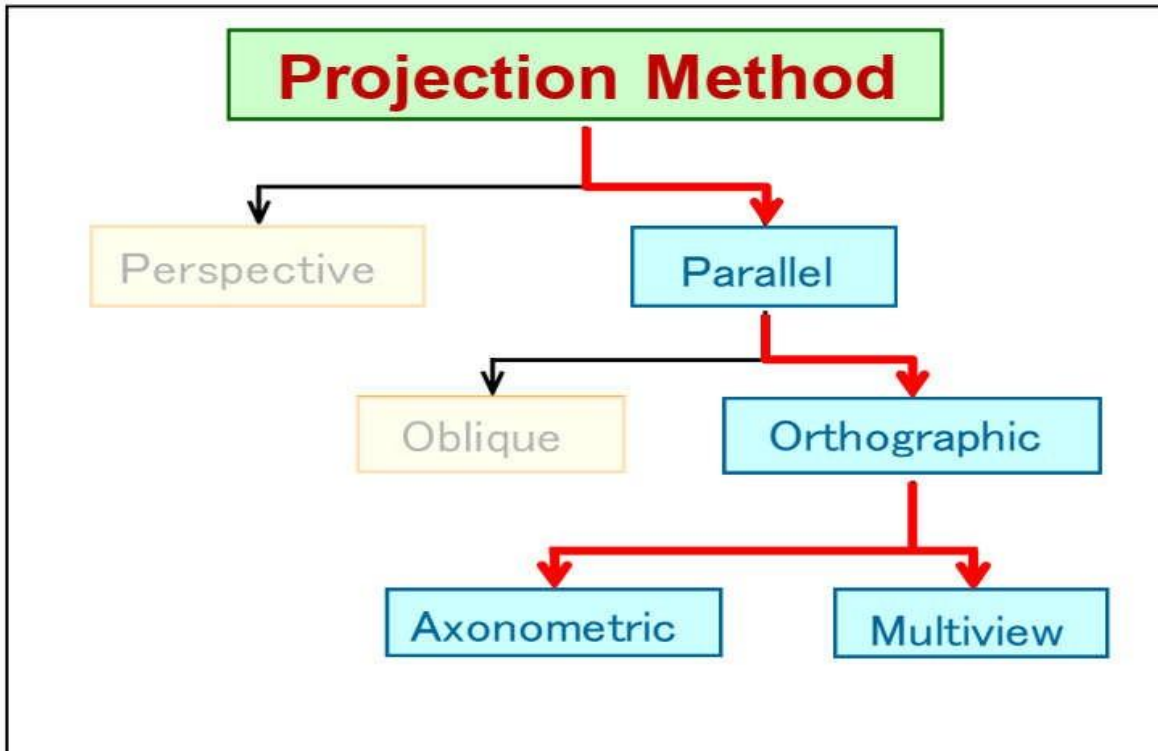
Example





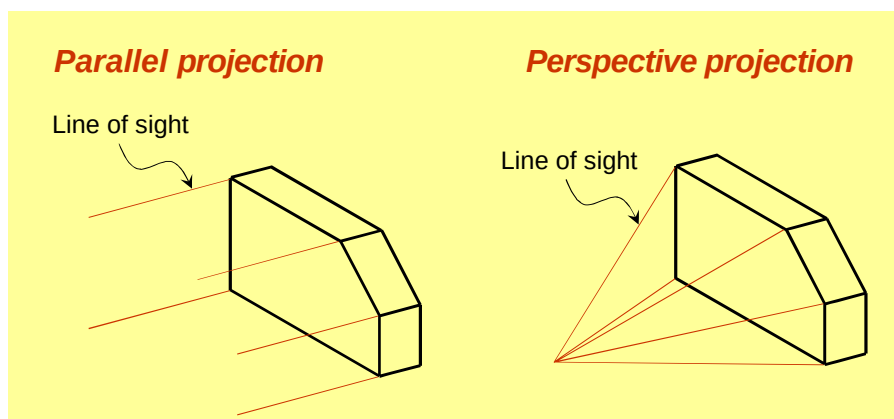
Information Sheet: 1.1-2

Learning Objective: Understanding the Projection Method.



Line of sight is an imaginary ray of light between an observer's eye and an object.

There are 2 types of LOS: Parallel and Converge

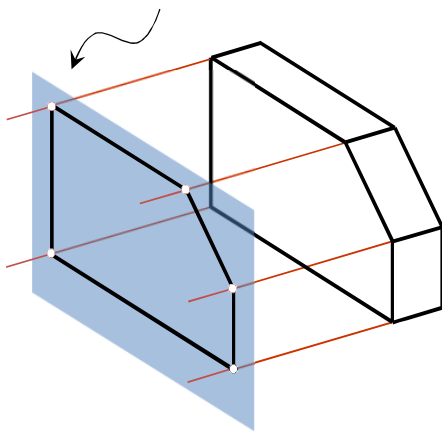


Plane of projection is an imaginary flat plane which the image is created.

- The image is produced by connecting the points where the LOS pierce the projection plane.

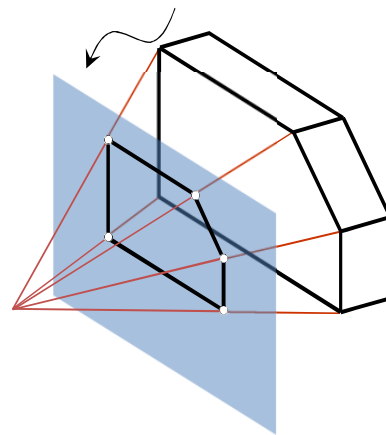
Parallel projection

Plane of projection



Perspective projection

Plane of projection



Disadvantage of Perspective Projection

- Perspective projection is **not** used by engineers for manufacturing of parts, because

- 1) It is difficult to create.
- 2) It does not reveal exact shape and size.





Information Sheet: 1.1-3

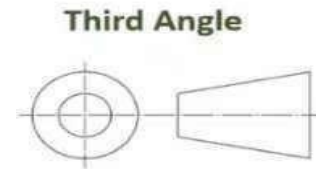
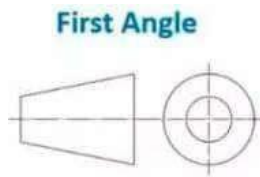
Learning Objective: Recognize Different Views.



Orthographic/Multiview Projection

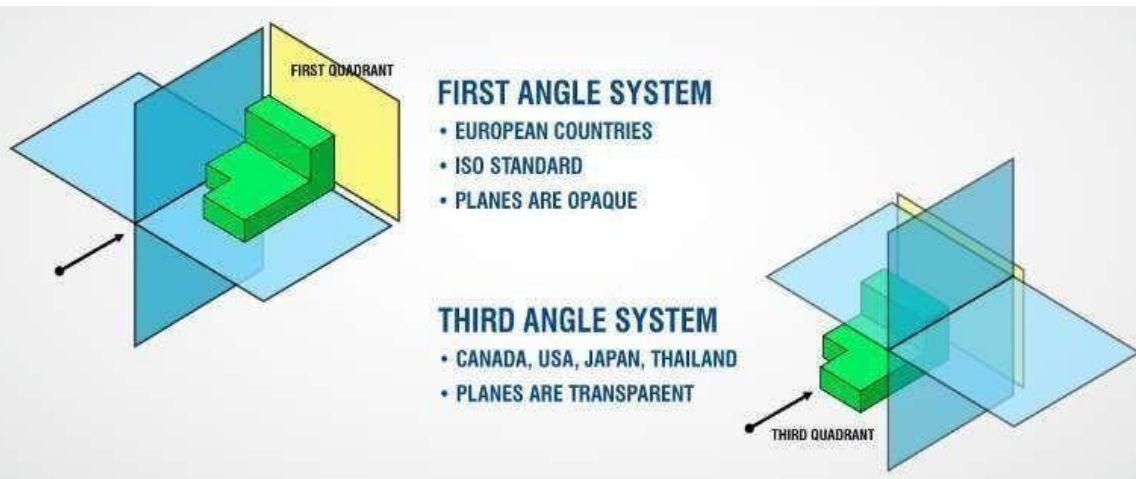
- Orthographic" comes from the Greek word for "straight writing (or drawing)." This projection shows the object as it looks from the front, right, left, top, bottom, or back, and are typically positioned relative to each other according to the rules of either "First Angle" or "Third Angle" projection.

Symbols for Third Angle (right) or First Angle (left).



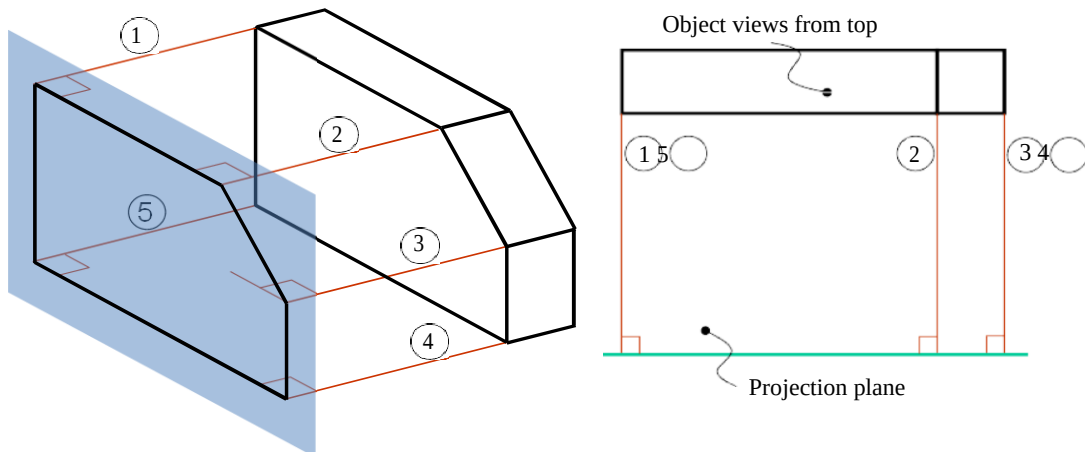
- First angle projection is the ISO standard and is primarily used in Europe. The 3D object is projected into 2D "paper" space as if you were looking at an X-ray of the object: the top view is under the front view, the right view is at the left of the front view.
- Third angle projection is primarily used in the United States and Canada. The left view is placed on the left the top view on the top.

Projection System

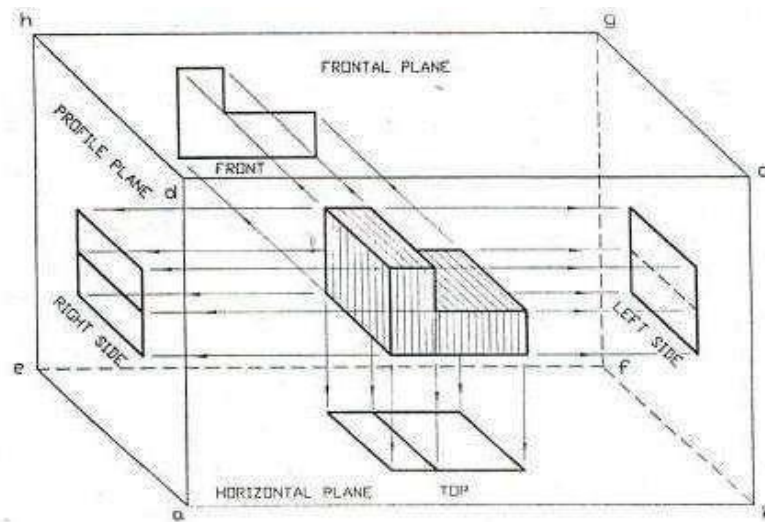


Meaning

Orthographic projection is a parallel projection technique in which the parallel lines of sight are *perpendicular* to the projection plane



Orthographic Drawing (1st Angle Projection)



Orthographic Drawing (1st Angle Projection)

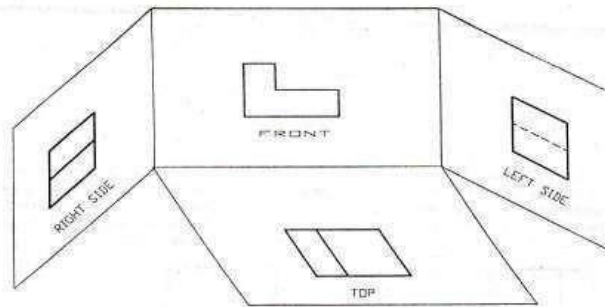
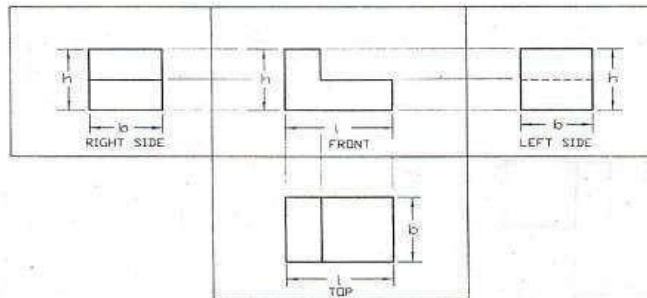
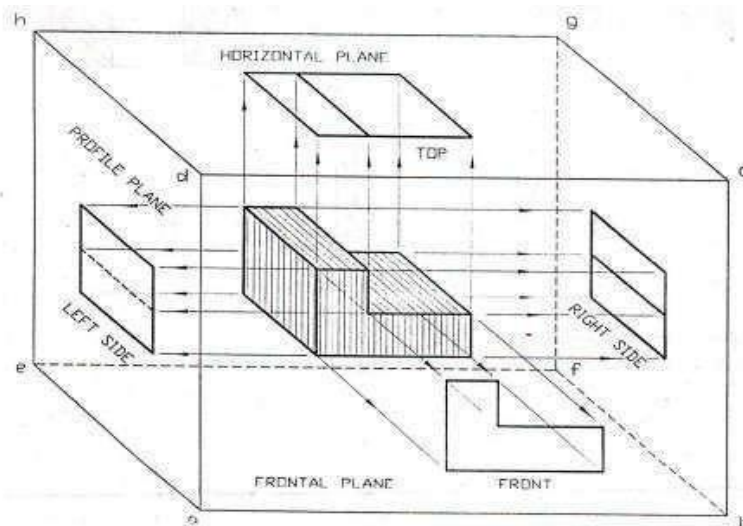


Figure 3.11: Views on Sides of Unfolding Box in First Angle Projection



Orthographic Drawing (3rd Angle Projection)



Orthographic Drawing (3rd Angle Projection)

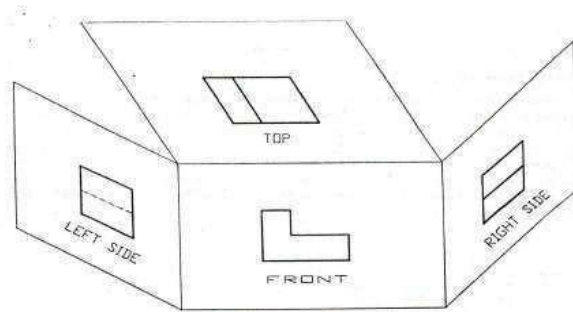
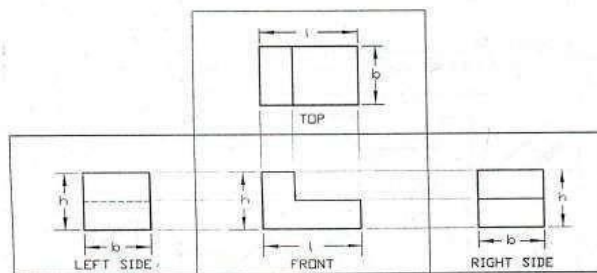
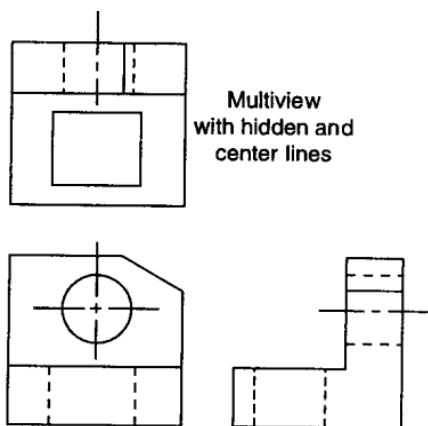


Figure 3.7: Views on Sides of Unfolding Box in Third Angle Projection

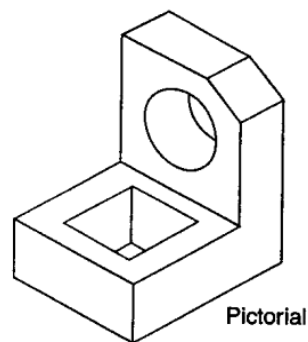


Orthographic/Multiview

Draw object from two / three perpendicular views

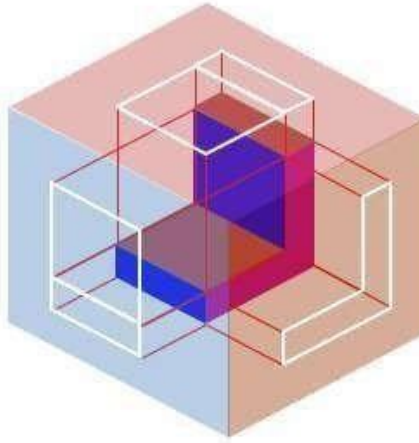


Multiview drawing.



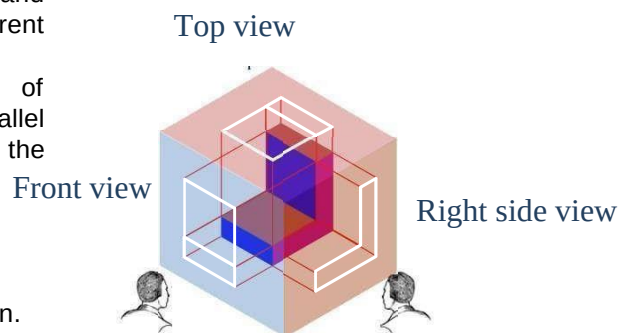
What it looks like pictorially

Generating Orthographic View: The Glass Box Approach



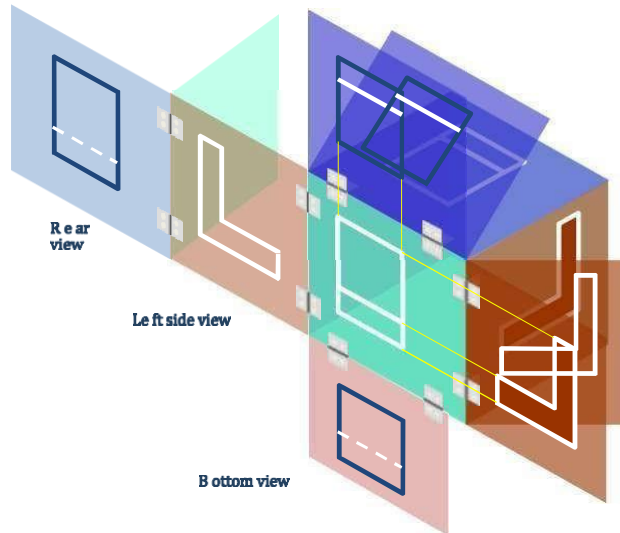
Generating Orthographic View: The Glass Box Approach

- Between the observer and the object, a transparent plane, or pane of glass, representing a plane of projection, is placed parallel to the front surfaces of the object.
- On the pane of glass is shown in outline how the object appears to the observer from that direction.
- If planes of projection are placed parallel to the principal faces of the object, they form a "glass box"



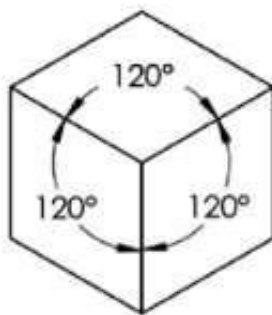
The observer is always on the outside, looking

Generating Multiview Projection: Unfolding the Glass Box

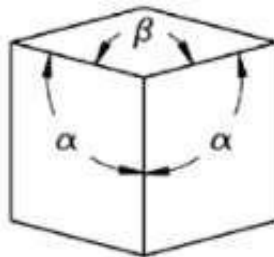


The drawings are done on a flat sheet of paper, so it is necessary to unfold the planes so that they will all lie in the same plane

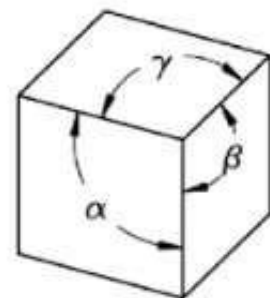
Axonometric Projection



Isometric



Dimetric



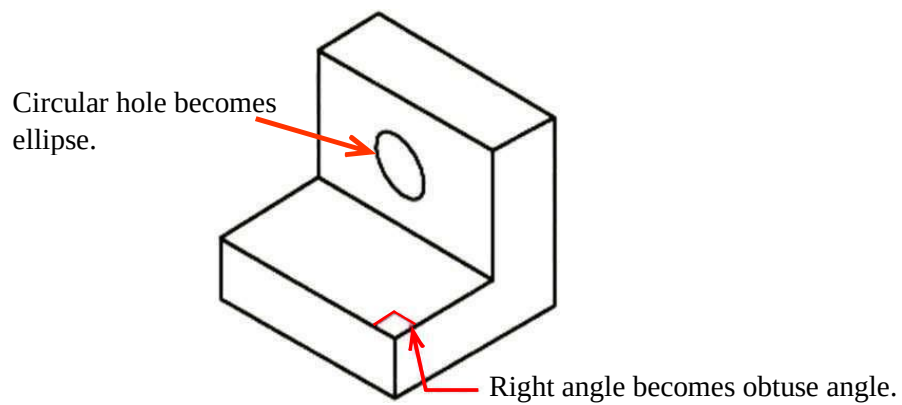
Trimetric

Axonometric (Isometric) Drawing

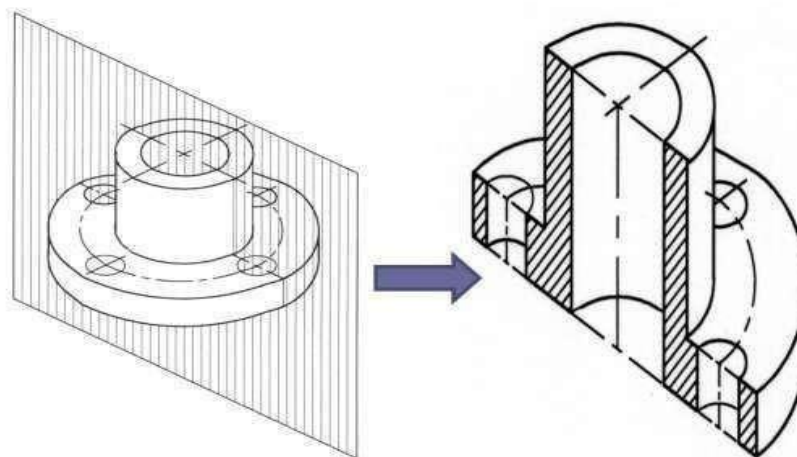
Advantage Easy to understand

Disadvantage Shape and angle distortion

Example Distortions of shape and size in isometric drawing



SECTIONAL VIEW



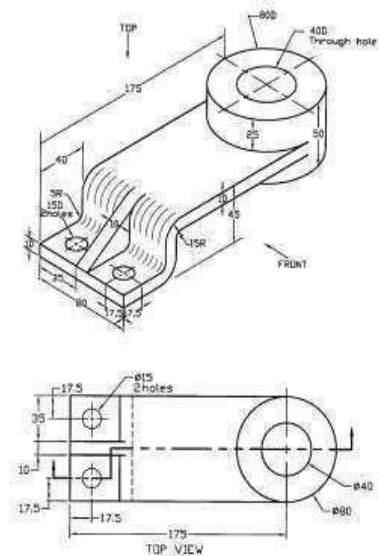
WHY SECTIONAL VIEW?



- ✓ In cases where lots of **hidden lines** are required to show the **interior detail** of an object , the **visualization** of the object becomes very **difficult**. To **remove** this difficulty sectional view is drawn.
- ✓ To represent the **interior construction or details of hidden features** clearly to the users **sectional view** is drawn.

SELECTING THE CUTTING PLANE

- The cutting plane is selected in a way, so that it can show highest number of internal features possible.

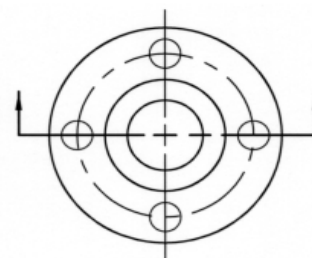


CUTTING PLANE LINE(125% THICKNESS)

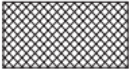
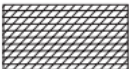
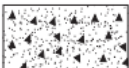


Cutting Plane Line (125% Thickness)

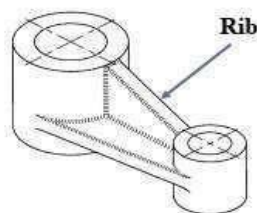
Cutting plane line represent **the line of action** of the cutting line



Conventional Representation of Materials

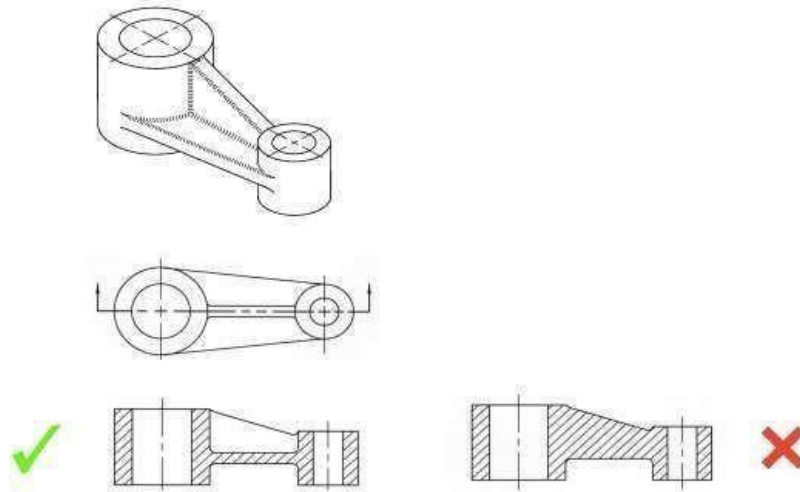
Type	Convention	Material
Metals		Steel, Cast Iron, Copper and its Alloys, Aluminium and its Alloys, etc.
		Lead, Zinc, Tin, White-metal, etc.
Glass		Glass
Packing and Insulating material		Porcelain, Stoneware, Marble, Slate, etc.
		Asbestos, Fibre, Felt, Synthetic resin products, Paper, Cork, Linoleum, Rubber, Leather, Wax, Insulating and Filling materials, etc.
Liquids		Water, Oil, Petrol, Kerosene, etc.
Wood		Wood, Plywood, etc.
Concrete		A mixture of Cement, Sand and Gravel

RIBS IN SECTION



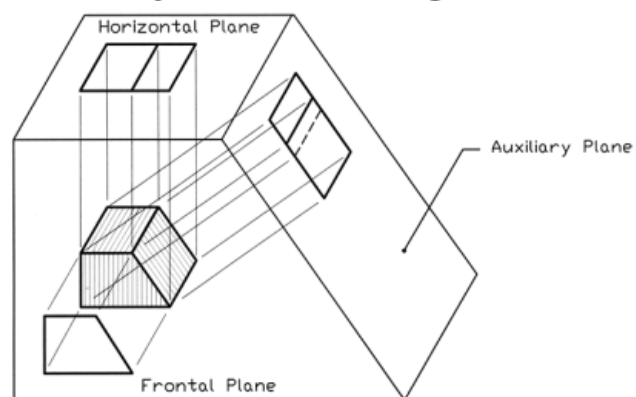
- When the cutting plane passes **longitudinally** through the center of any **rib or web** section lining is **omitted** there, because **ribs or webs** are not any **integral part** of the object and are only provided to increase the strength of the object.
- On the other hand, when the cutting plane **cuts** the ribs **transversely**, that is, at right angle to its length, it is always sectioned.

RIBS IN SECTION

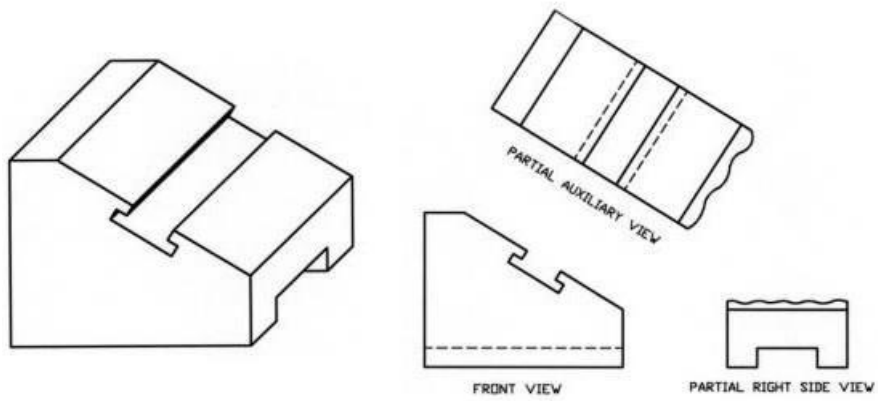


GENERATING AUXILIARY VIEW

- ☐ A picture plane is assumed parallel to the inclined surface called **auxiliary projection plane**.
- ☐ Orthographic projection on that plane is made.
- ☐ It gives the **true shape of the inclined surface** and preserves the objective of the drawing.

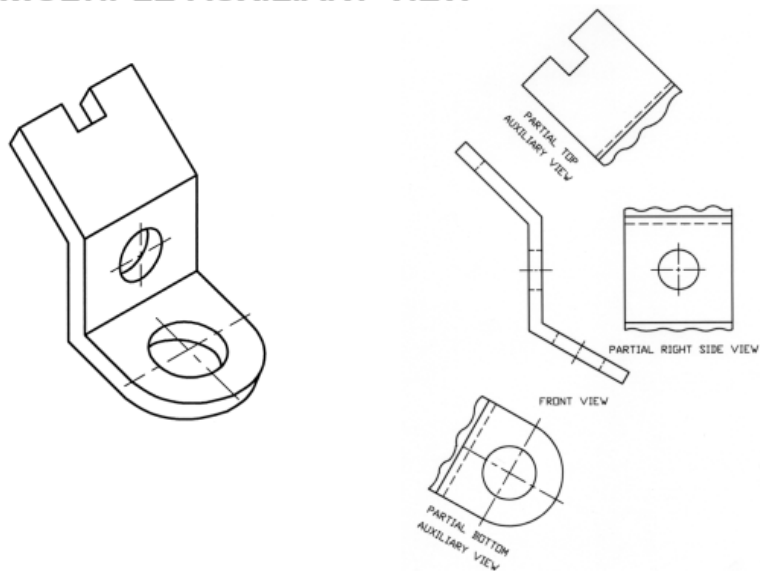


AUXILIARY VIEW

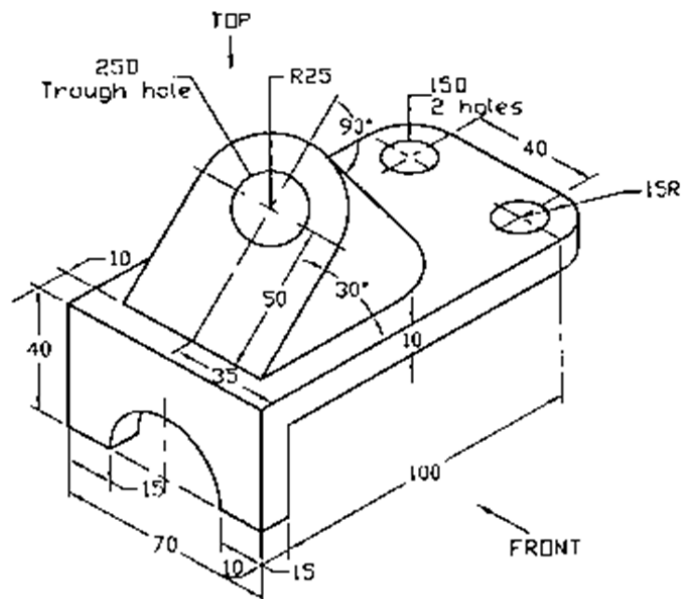


Auxiliary view

MULTIPLE AUXILIARY VIEW



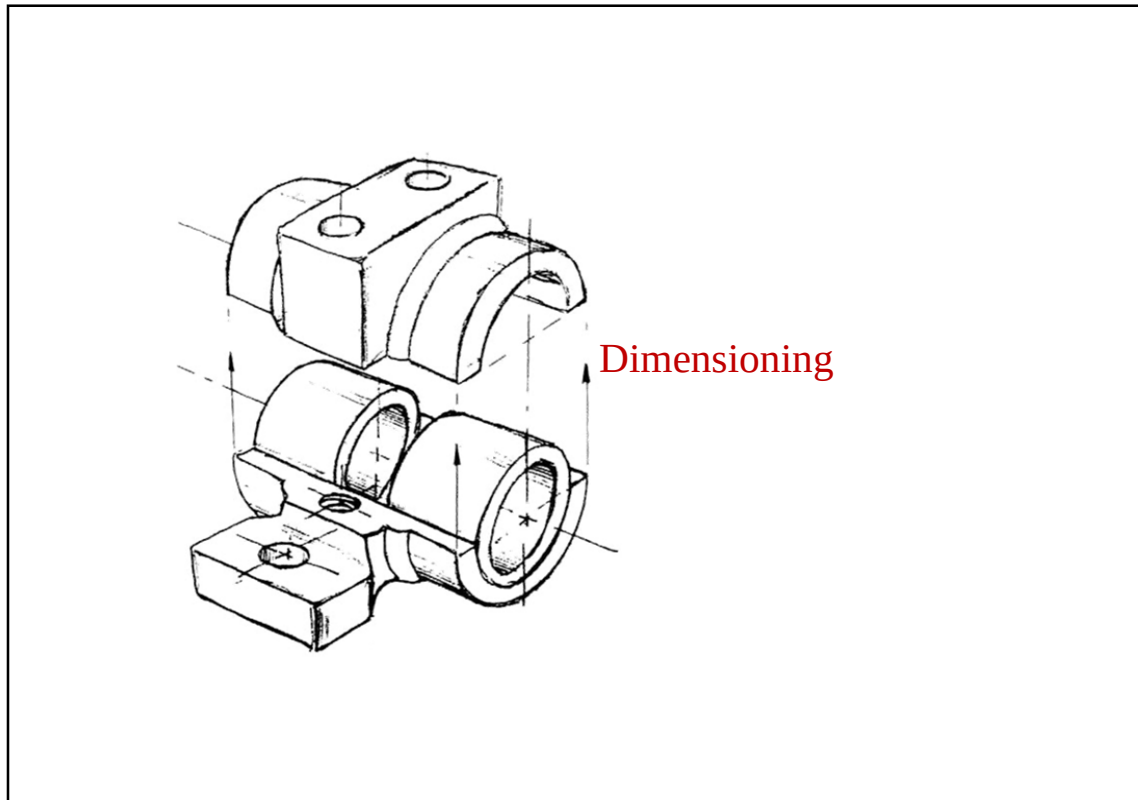
Auxiliary projection





Information Sheet: 1.1-4

Learning Objective: Understanding the Dimension, Tolerance, Scale, Signs and Symbols in Standard Mechanical Drawing.



Dimensioning Guidelines

The term “feature” refers to surfaces, faces, holes, slots, corners, bends, arcs and fillets that add up to form an engineering part.

Dimensions define the **size** of a feature or its **location** relative to other features or a frame of reference, called a datum.

The basic rules of dimensioning are:

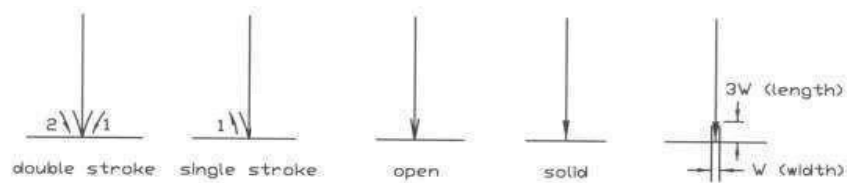
1. Dimension where the feature contour is shown;
2. Place dimensions between the views;
3. Dimension mating features for assembly;
4. Do not dimension to hidden lines;
5. Create a logical arrangement of dimensions;

□ Dimensioning is done basically by using-

- Arrowhead
- Extension line
- Dimension line
- Leader

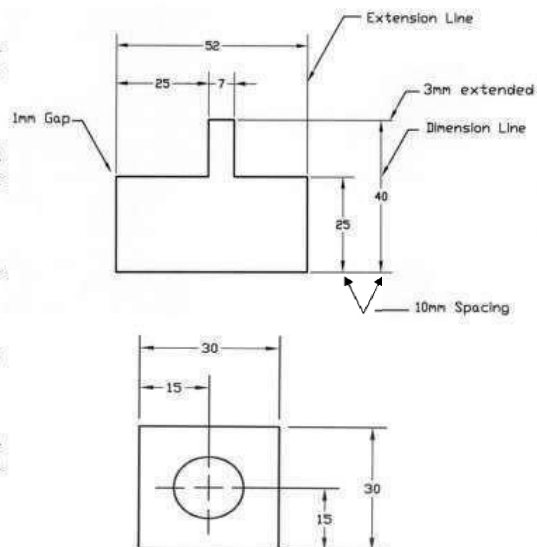
ARROWHEAD

- Usually drawn **freehand**.
- The approximate length of the arrowhead may be **3 mm**.
- The approximate ratio of the length to width of the arrowhead is **3:1**.



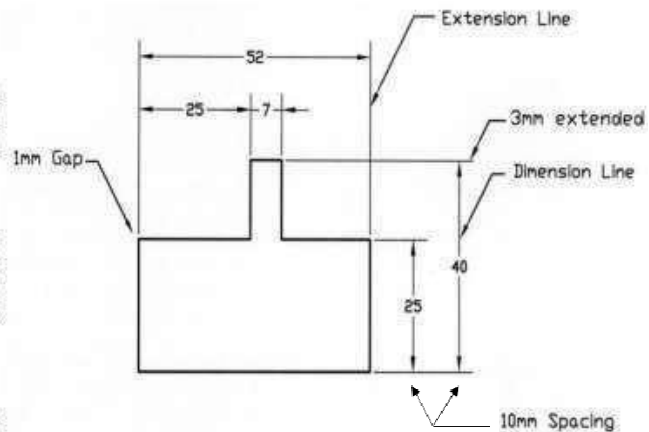
EXTENSION LINE (25% THICK LINE)

- A **gap of 1 mm** has to be kept in between the extension line and the visible line.
- An extension line should extend about **3 mm** from the outermost dimension line.
- Extension lines **may cross each other** without a break.
- **Centerlines** may be used as extension lines.
- Extension lines are usually drawn **perpendicular to dimension lines**.



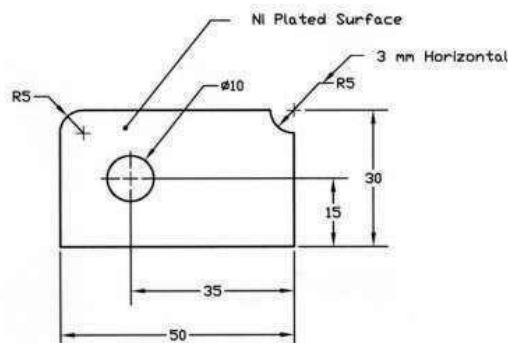
DIMENSION LINE (25% THICK LINE)

- Dimension line should be approximately 10 mm away from the visible line.
- The spacing between the consecutive parallel dimension lines may also be considered as 10 mm.
- Dimension lines should usually be placed outside the view unless it becomes necessary.
- When the space in between the extension lines is too small to insert dimension line completed with arrowhead, it may be provided outside the extension line.



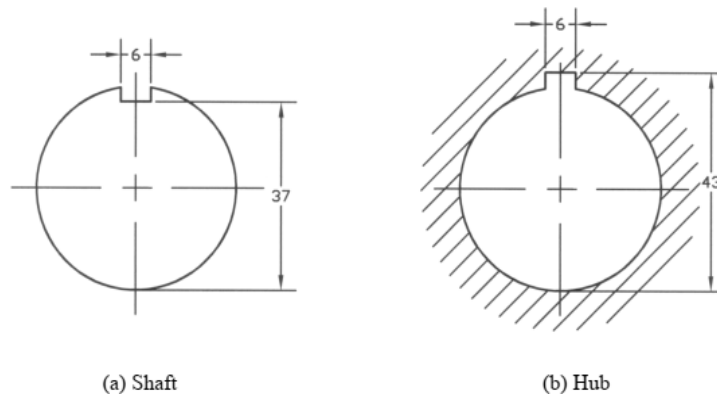
LEADERS

- Should always be inclined at an angle of 60° preferably and 45° occasionally with a 3 mm horizontal bar.
- Is either terminated by an arrowhead on a line or a small dot.
- Should not cross each other.
- To direct a circle or an arc the leader should be so drawn, if it is imagined to extend it must pass through the center of the circle or the arc.
- All notes and dimensions in a leader have to be provided in the horizontal direction.

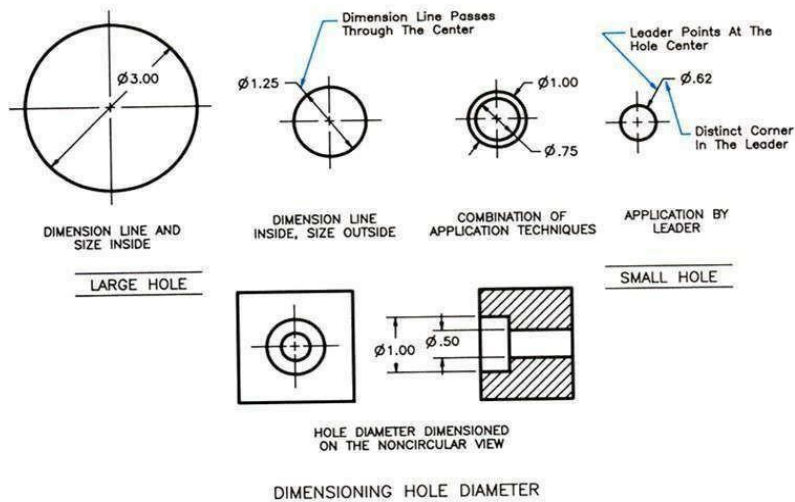


GENERAL DIMENSIONING SYMBOLS		
CURRENT PRACTICE	ABBREVIATION IN NOTES	PARAMETER
∅	DIA	DIAMETER
S∅	SPHER DIA	SPHERICAL DIAMETER
R	R	RADIUS
CR	CR	CONTROLLED RADIUS
SR	SR	SPHERICAL RADIUS
┐	CBORE or SFACE	COUNTERBORE SPOTFACE
∨	CSK	COUNTERSINK
⌞	DP	DEEP
○	—	DIMENSION ORIGIN
□	SQ	SQUARE
()	REF	REFERENCE
x	PL	PLACES, TIMES
⌒	—	ARC LENGTH
∠	—	SLOPE
⌢	—	CONICAL TAPER

DIMENSIONING IN KEYWAYS

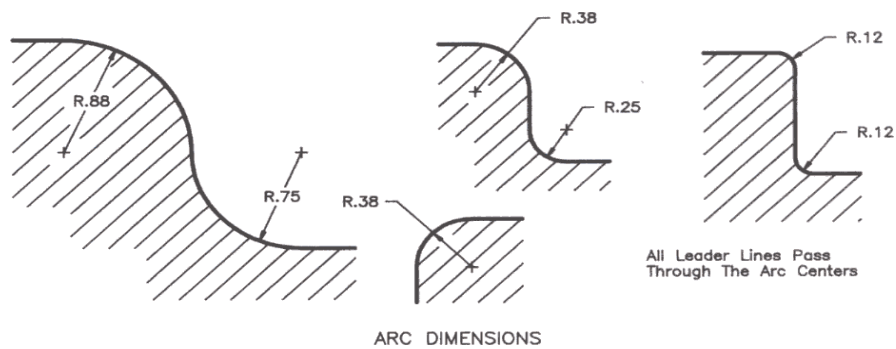


Dimensioning Holes



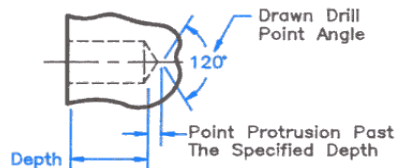
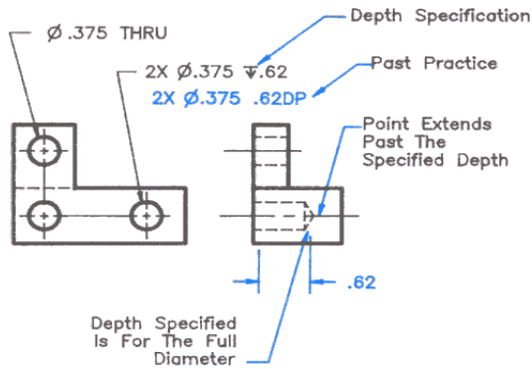
- Dimension the diameter of a hole.
- Locate the center-line.
- Use a notes and designators for repeated hole sizes.

Dimensioning the Radius of an Arc



Dimension an arc by its radius. Locate the center of the radius or two tangents to the arc.

Drilled Holes, Counter bores and Countersinks



- Use the depth symbol to define the depth of a drilled hole.
- Use the depth symbol or a section view to dimension a counterbore.
- Countersinks do not need a section view.

Drilled Holes, Counter bores and Countersinks

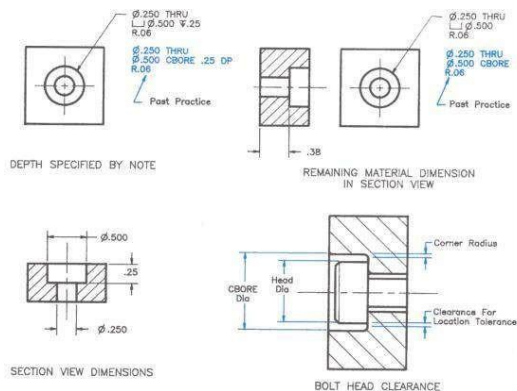
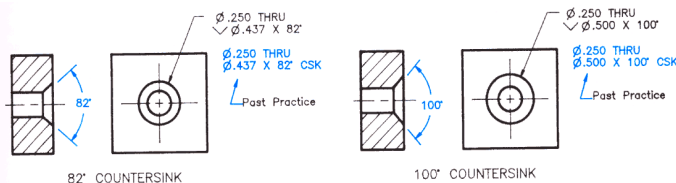


Figure 4-10. Counterbore dimensions provide a controlled depth.

- Use the depth symbol to define the depth of a drilled hole.
- Use the depth symbol or a section view to dimension a counter bore.

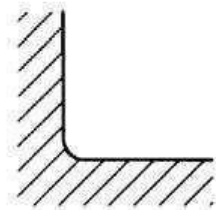


- Countersinks do not need a

Section view.

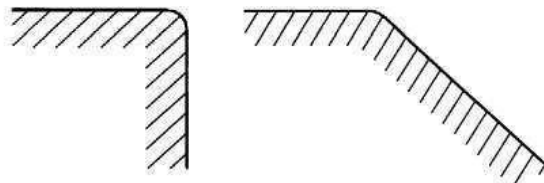
FILLETS

- The sharp **internal corners** are made rounded in order to avoid the possibility of **stress crack**.
- These rounded internal corners are called **fillets**.



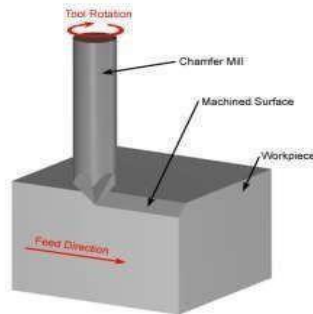
ROUNDS

- The **external sharp corners** are made rounded for **safety and appearance**.
- These rounded external corners are called **rounds**.

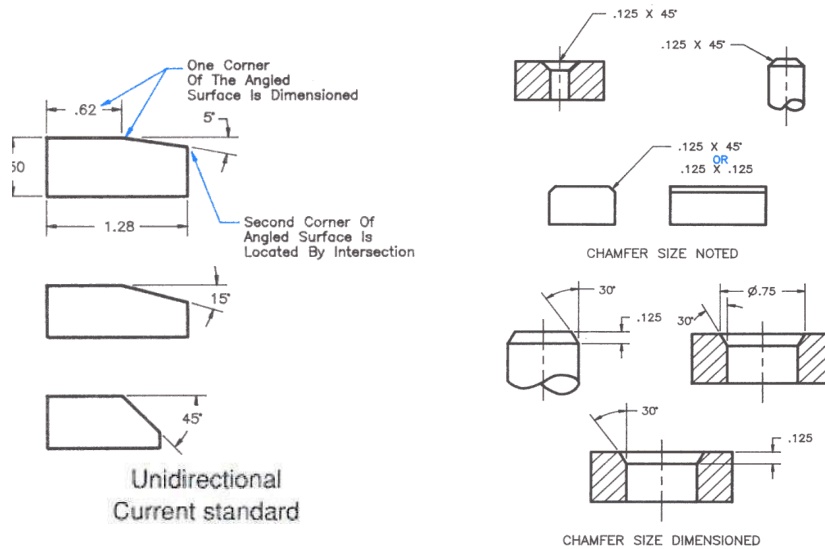


CHAMFER

- ✓ A **chamfer** is a beveled edge connecting two surfaces.
- ✓ A **beveled edge** refers to an edge of a structure that is not perpendicular to the faces of the piece.

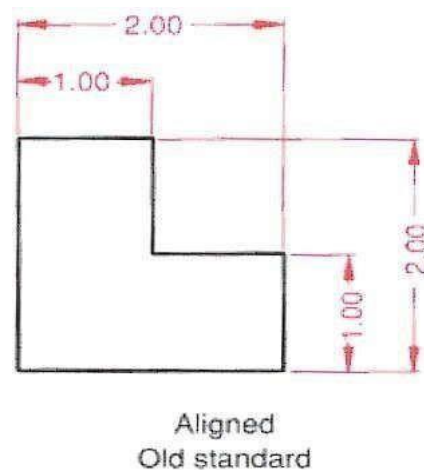


Angles, Chamfers and Tapers



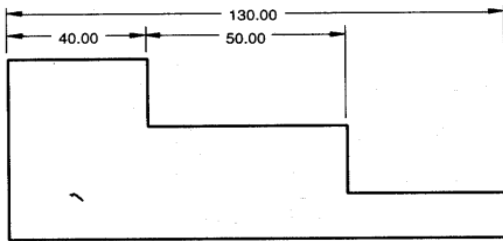
- Dimension the one vertex for an angled face, the other vertex is determined by an intersection.
- Chamfers are generally 45 degrees

Unidirectional or aligned dimensioning?

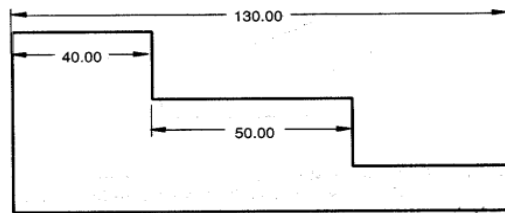


Grouping Dimensions

- Dimensions should always be placed outside the part

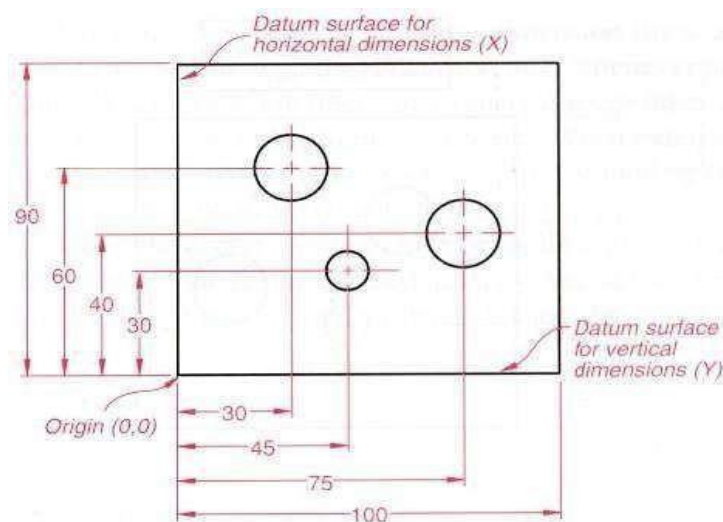


Yes



No

Where and how should we place dimensions when we have many dimensions? (cont.)





Self-Check: 1.1-1 to 1.1-4

1. What is the primary purpose of a mechanical drawing?
2. How does third-angle projection differ from first-angle projection?
3. Where is the right-side view typically placed in a third-angle orthographic drawing?
4. What is the function of a sectional view?
5. What is the difference between an isometric view and a perspective view?
6. What is the significance of a dimension written $50 \pm 0.1 \text{ mm}$ as



Answer Script: 1.1-1 to 1.1-4

1. A mechanical drawing is a technical blueprint that communicates the precise technical details, such as geometry, size, tolerances, and material, for the manufacturing, assembly, and inspection of a part or product.
2. In third-angle projection, the object is placed behind the viewing planes, so the views are arranged next to the face they represent. In first-angle projection, the object is between the viewer and the viewing planes, so the views are arranged opposite to the face they represent.
3. It is placed to the right of the front view.
4. A sectional view is used to expose internal features of an object that would otherwise be hidden from sight, making complex interiors easier to understand and dimension.
5. In an isometric view, parallel lines remain parallel and do not converge, providing a non-distorted, scalable pictorial drawing. In a perspective view, parallel lines appear to recede and meet at a vanishing point, which mimics how the human eye sees things but distorts dimensions.
6. This indicates that the nominal size of the feature is 50 mm, with a permissible tolerance of plus or minus 0.1 mm. The final part's size can be anywhere between 49.9 mm and 50.1 mm to be acceptable.

Learning Outcome 1.2: Create Mechanical Drawing



Content:

- Isometric drawing
- Orthographic drawing
- Auxiliary drawing



Assessment Criteria

1. Be able to understand Isometric to Orthographic drawing.
2. Be able to understand Isometric to Auxiliary drawing.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. Drawing paper
2. T Square, Set Square
3. Pencil, Eraser
4. Compass, Protractor
5. Scale



Activity/Work

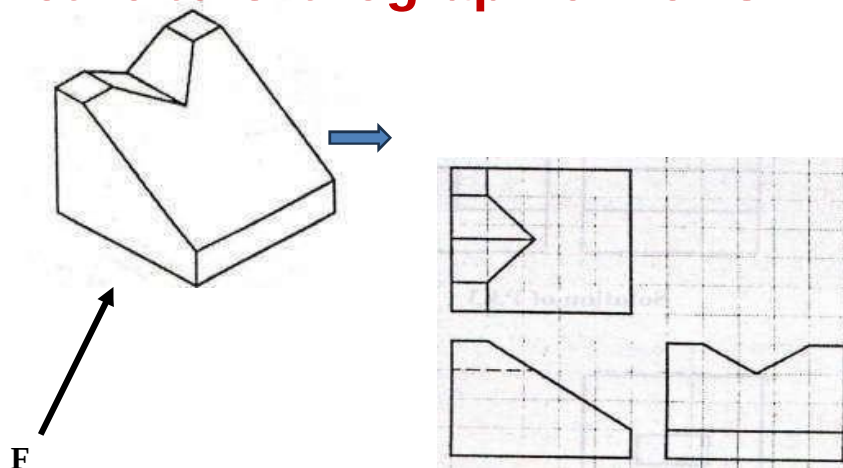
Activity/Work	Resource/Special Instruction/Reference
Create Mechanical Drawing	• Information Sheet: 1.2.-1 to 1.2.-2 • Self-Check: 1.2-1.2-2 • Job Sheet: 1.2 (a) to 1.2 (b) • Engineering Drawing by Dr. Md. Quamrul Islam



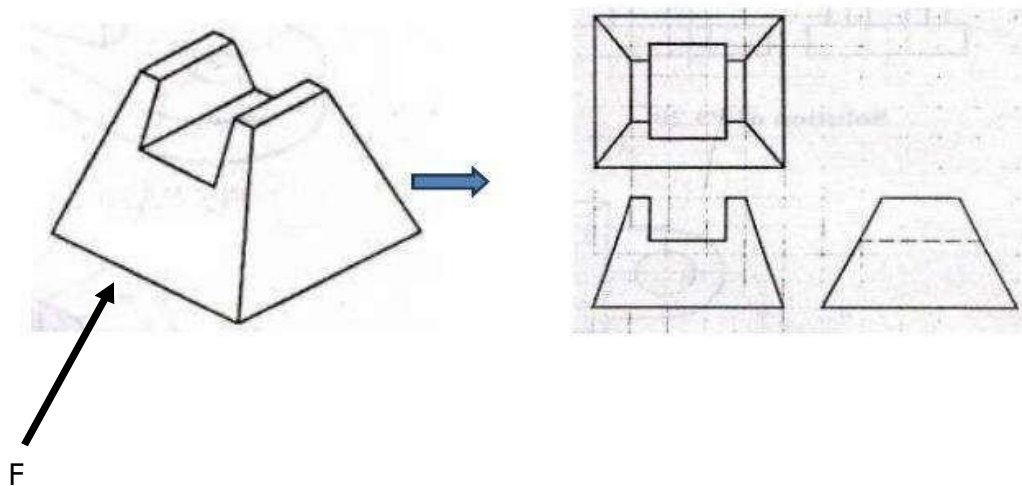
Information Sheet: 1.2-1

Learning Objective: Understand Isometric to Orthographic Drawing Transformation.

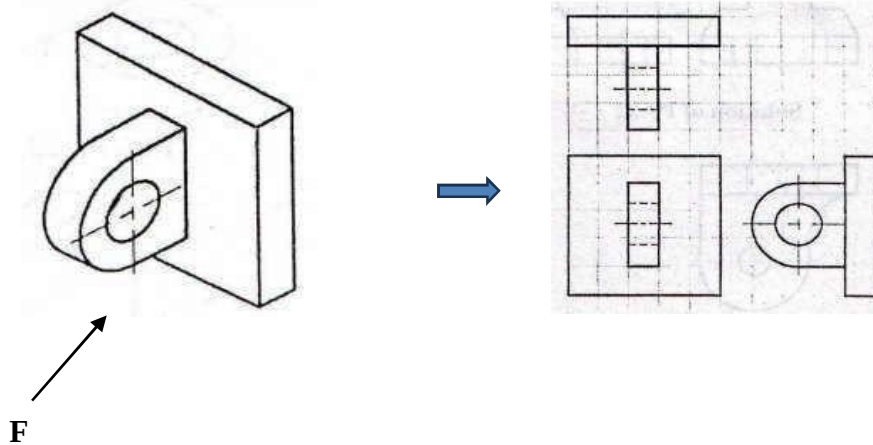
Isometric to Orthographic Views



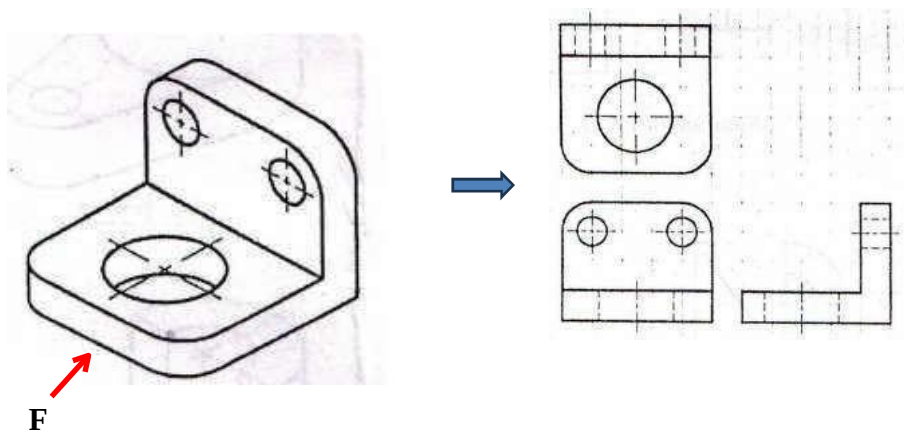
Isometric to Orthographic Views



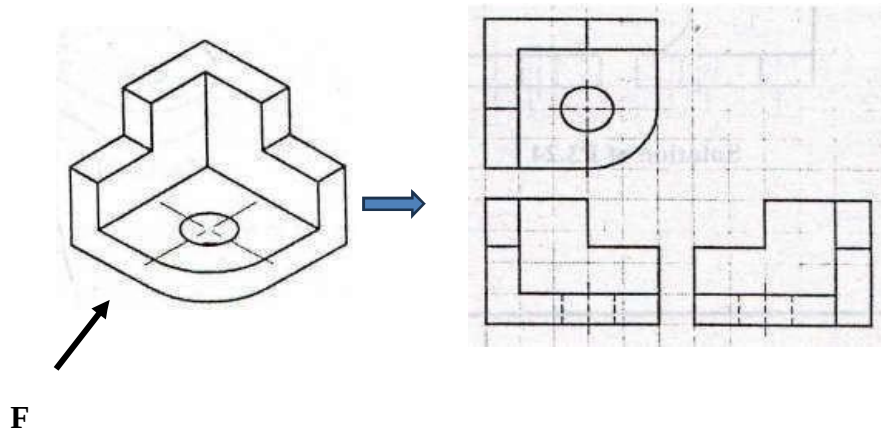
Isometric to Orthographic Views



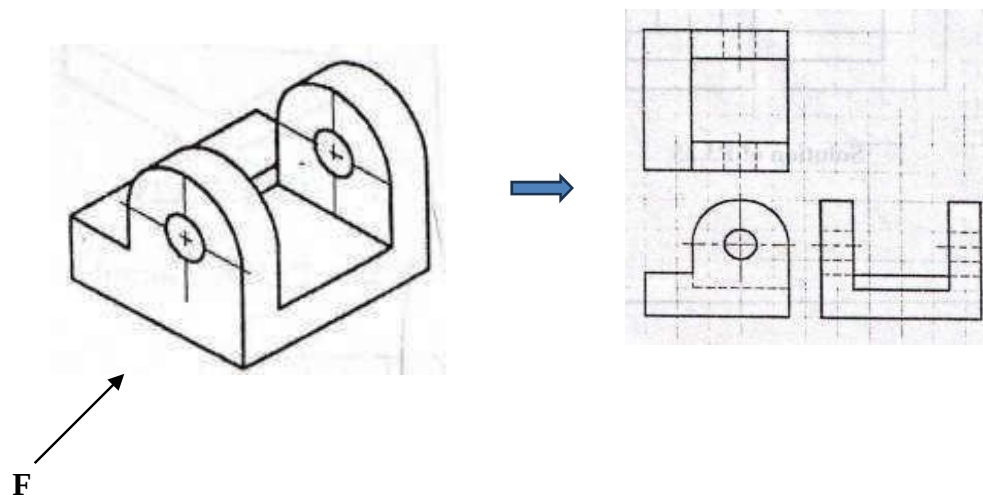
Isometric to Orthographic Views



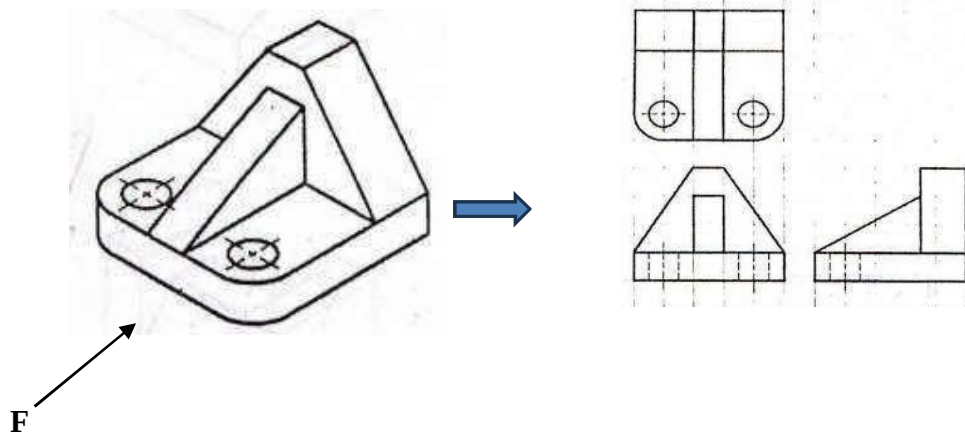
Isometric to Orthographic Views



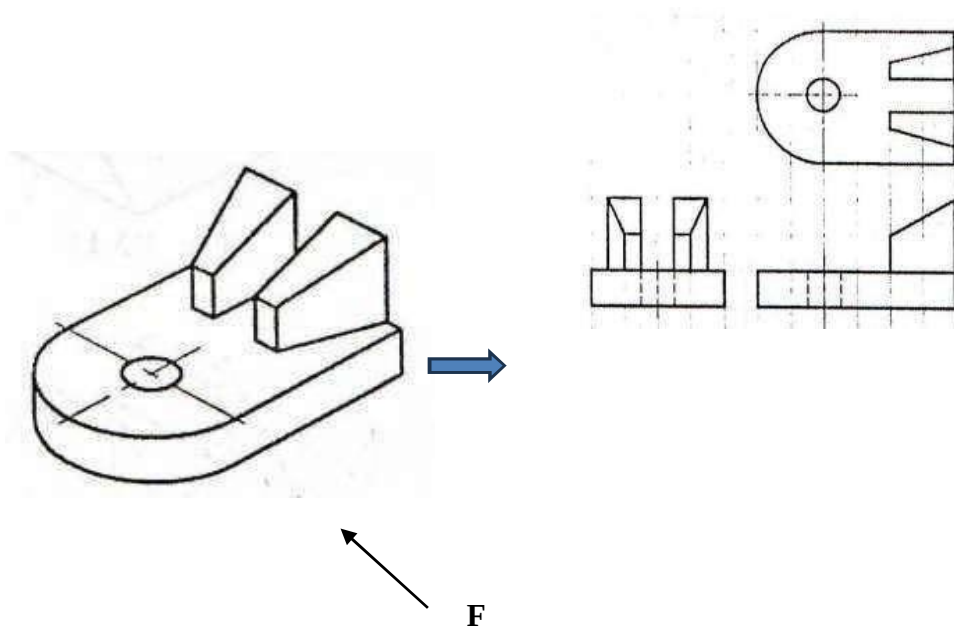
Isometric to Orthographic Views



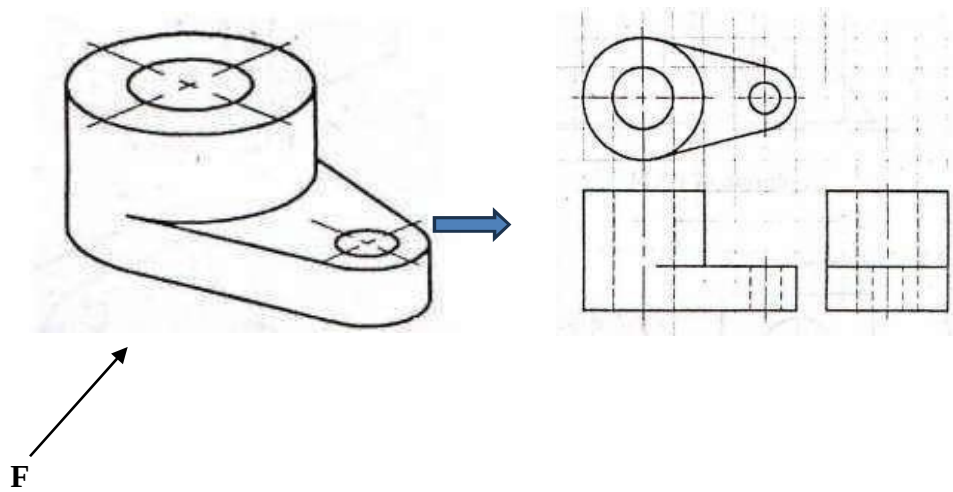
Isometric to Orthographic Views



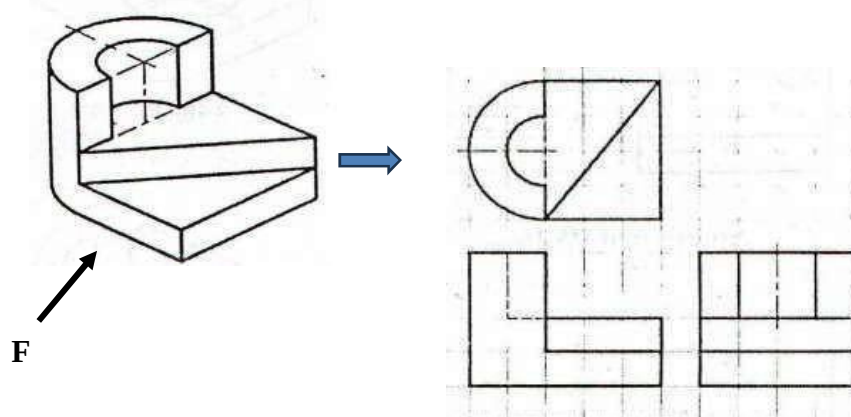
Isometric to Orthographic Views



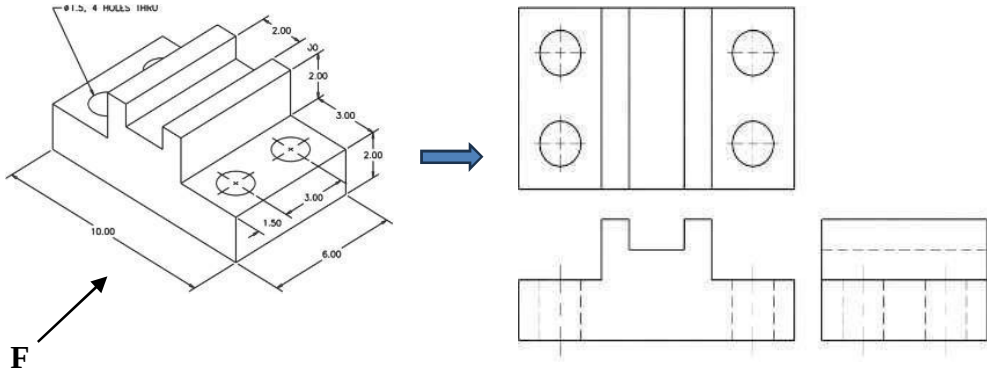
Isometric to Orthographic Views



Isometric to Orthographic Views



Isometric to Orthographic Views



The diagram illustrates the conversion of an isometric view of a mechanical part into its orthographic projections. The isometric view on the left shows a base plate with dimensions 10.00 (length) and 6.00 (width). It features a central rectangular cutout with a width of 3.00 and a depth of 2.00. On the right side of the base, there is a raised section with a width of 2.00 and a height of 3.00. A circular hole with a diameter of $\phi 1.5$ is located on the top surface of this raised section. A blue arrow points from the isometric view to the orthographic views on the right. The orthographic views include a top view (showing the circular hole and the cutout), a front view (showing the profile of the part), and a side view (showing the depth and the cutout). A black arrow labeled 'F' points towards the isometric view.

Isometric to Orthographic Views

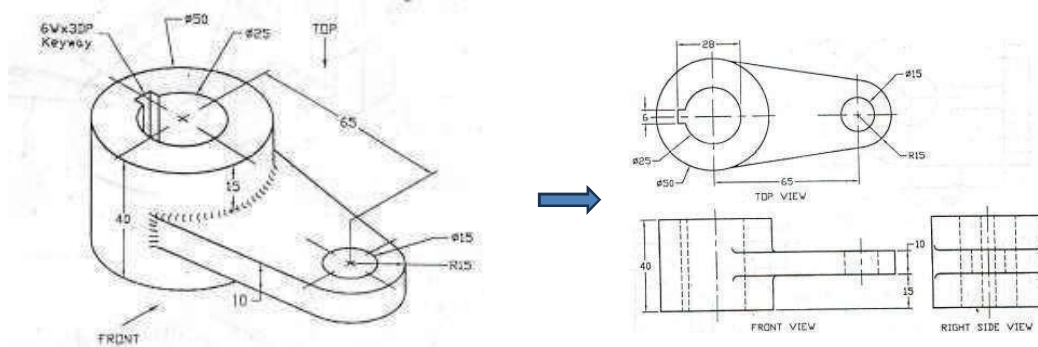
The diagram illustrates the conversion of an isometric view of a mechanical part into its orthographic projections. The isometric view on the left shows a part with a rectangular base, a sloped top surface, and a semi-circular end. Key dimensions and features include:

- Overall height: 50
- Base thickness: 10
- Top surface width: 35
- Sloped surface width: 20
- Sloped surface height: 30
- Top surface radius: R25
- Two $\phi 10$ drills (one on the sloped surface, one on the top surface)
- One $\phi 20$ hole (on the top surface)
- Isometric axes: X, Y, Z

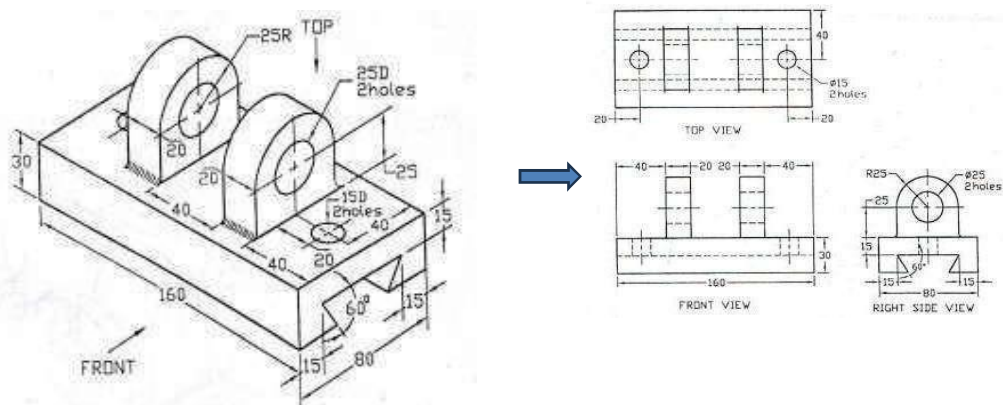
A blue arrow points from the isometric view to the orthographic projections on the right. The orthographic views are:

- TOP VIEW:** Shows the top surface with dimensions 35 (width) and 25 (radius). It includes the $\phi 20$ hole and the $\phi 10$ drill.
- FRONT VIEW:** Shows the front elevation with dimensions 50 (height), 10 (base thickness), 20 (sloped surface width), and 35 (top surface width). It includes the sloped surface and the $\phi 10$ drill.
- RIGHT SIDE VIEW:** Shows the right side elevation with dimensions 10 (base thickness), 21 (total width), 8 (sloped surface width), and 10 (top surface width). It includes the $\phi 10$ drill and the $\phi 20$ hole.

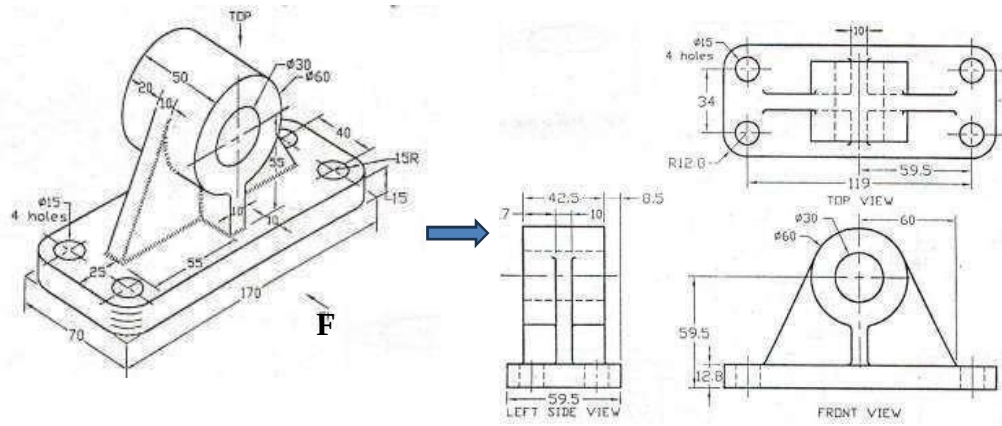
Isometric to Orthographic Views



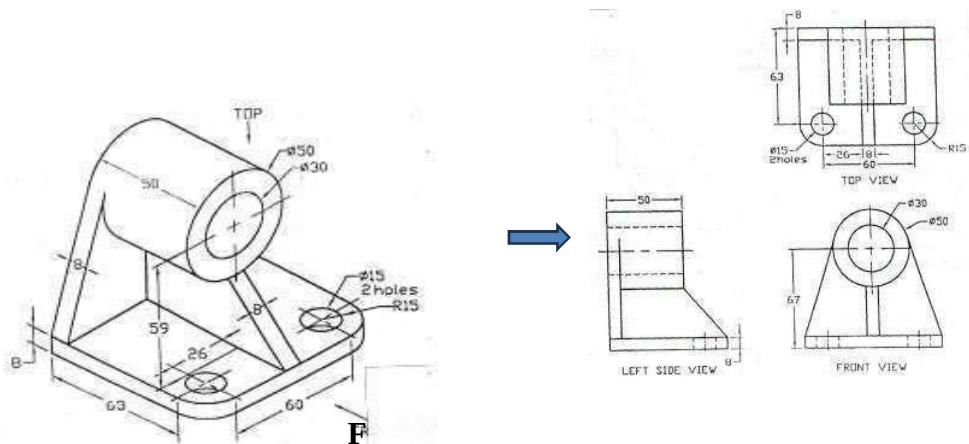
Isometric to Orthographic Views



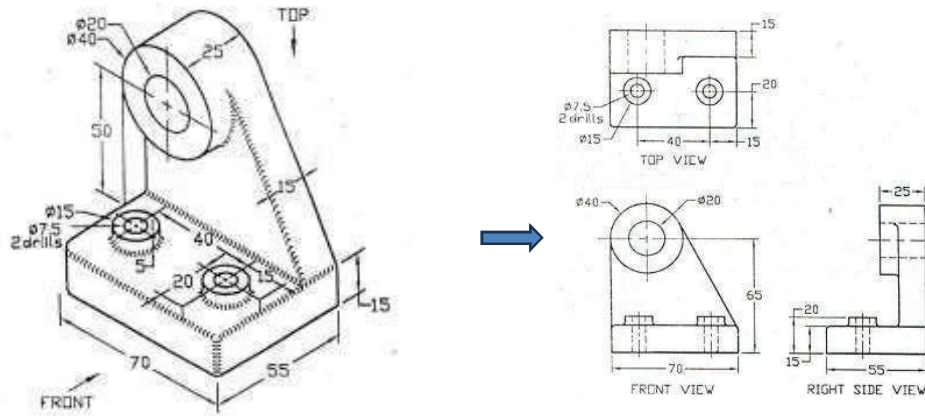
Isometric to Orthographic Views



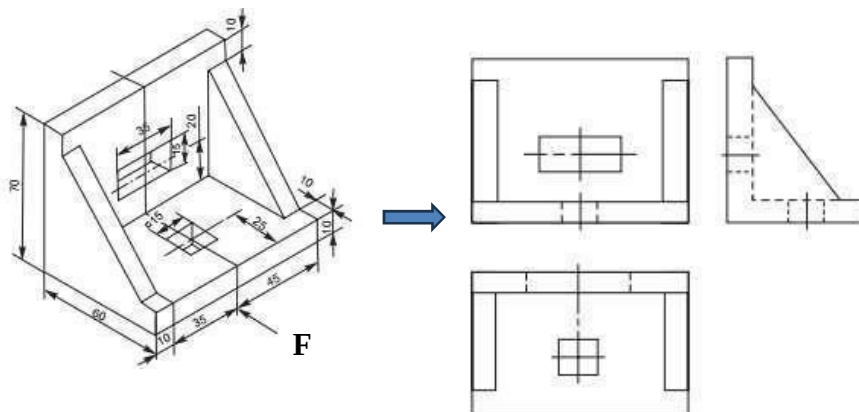
Isometric to Orthographic Views



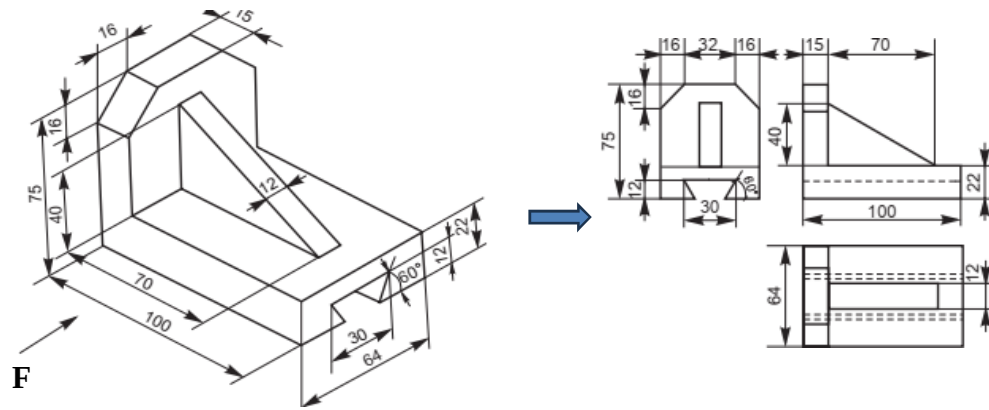
Isometric to Orthographic Views



Isometric to Orthographic Views



Isometric to Orthographic Views

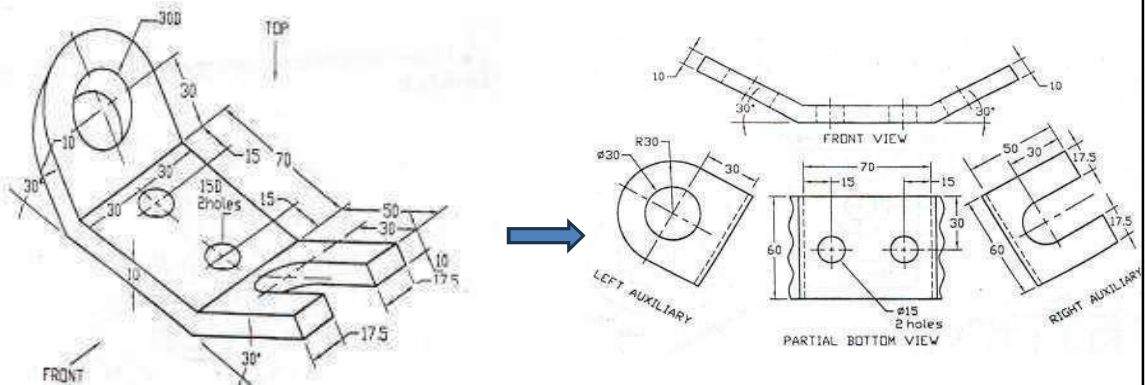




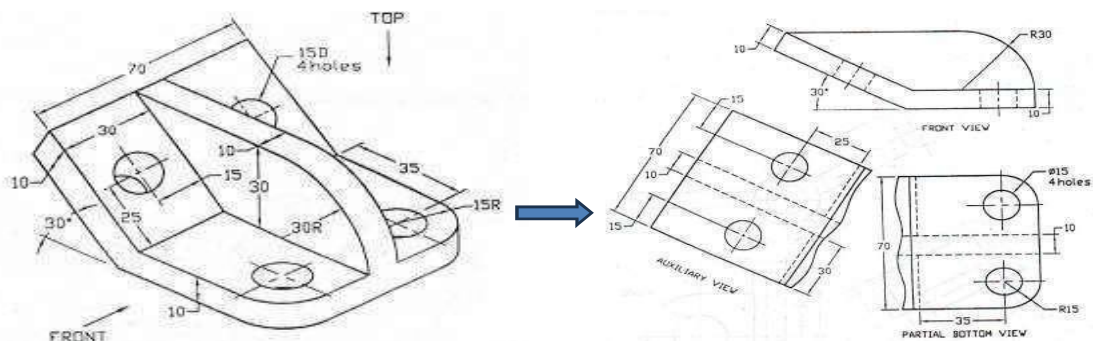
Information Sheet: 1.2-2

Learning Objective: Understand Isometric to Auxiliary Drawing Transformation.

Isometric to Auxiliary Views



Isometric to Auxiliary Views





Self-Check: 1.2-1-1.2-2

1. When is it necessary to draw an auxiliary view?
2. How are auxiliary views projected from a standard orthographic view?
3. What is the purpose of a reference line when constructing an auxiliary view?
4. How does an inclined surface appear in an auxiliary view?
5. How would you transfer the depth dimension from the side view to an auxiliary view projected from the front view?



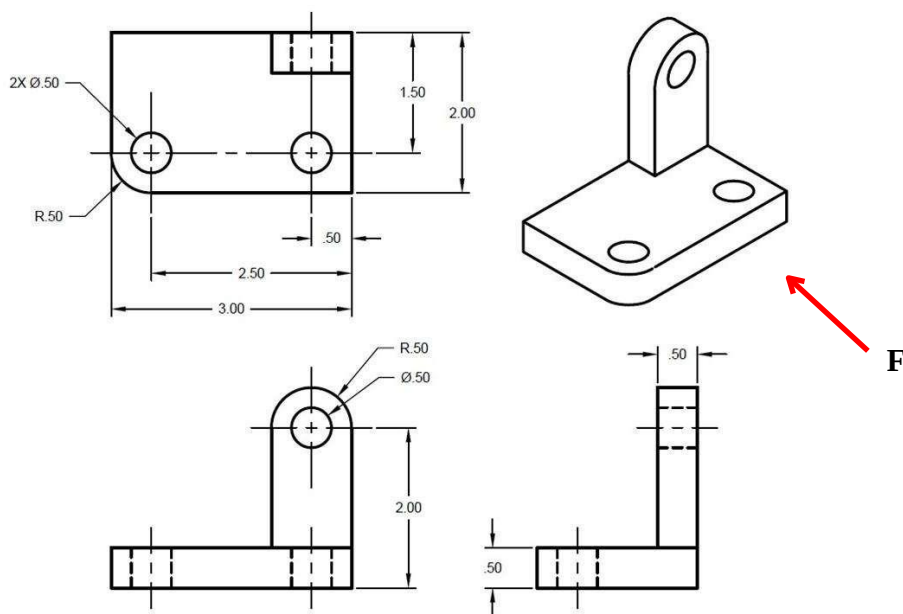
Answer Script: 1.2-1-1.2-2

1. An auxiliary view is necessary when an object has an inclined or oblique surface that does not appear in its true size and shape in any of the six principal orthographic views (top, front, right side, left side, bottom, and rear).
2. Auxiliary views are projected by following these steps:
 - a) Identify the inclined surface that you want to show in its true size and shape.
 - b) Draw a reference line on the standard orthographic view (usually the one where the inclined surface appears as an edge) that is parallel to the edge of the inclined surface.
3.
 - a) It is drawn parallel to the edge view of the inclined surface.
 - b) It acts as a datum or starting point from which to measure and transfer dimensions (such as depth, height, or width) from an adjacent standard view to the auxiliary view.
4. In an auxiliary view, an inclined surface appears in its **true size and true shape**, without any distortion or foreshortening.
5. To transfer the depth dimension:
 - a) **Select the source view:** Identify the side view, as this is where the true depth of the object is shown.
 - b) **Establish a reference line:** On both the side view and the auxiliary view, draw a reference line. A common practice is to place this line against the closest or farthest face of the object for consistency.

Job Sheet: 1.2 (a)

Job	Identify views of drawing from given sample	
Materials/Resources	Pencil	1 Pc
	Eraser	1 Pc
	A 4 Paper	1 Pc
	Scale	1 Pc
	Sample Drawing	1 Pc
	Tee Square	1 Pc
	Set Square	1 Pc
Procedure	Collect required materials/resources.	
	Fasten the drawing in drawing board and, interpret and visualize the drawing in context to perspective view, exploded view and hidden view.	
	Will measure angles and lines of shape of drawing by using geometric instruments if required and identify the different components and assembly in the drawing.	
	Measure all dimensions and compare with given dimension to identify the tolerance.	
	After identifying drawing all tools and materials to be cleaned and kept in the right place.	

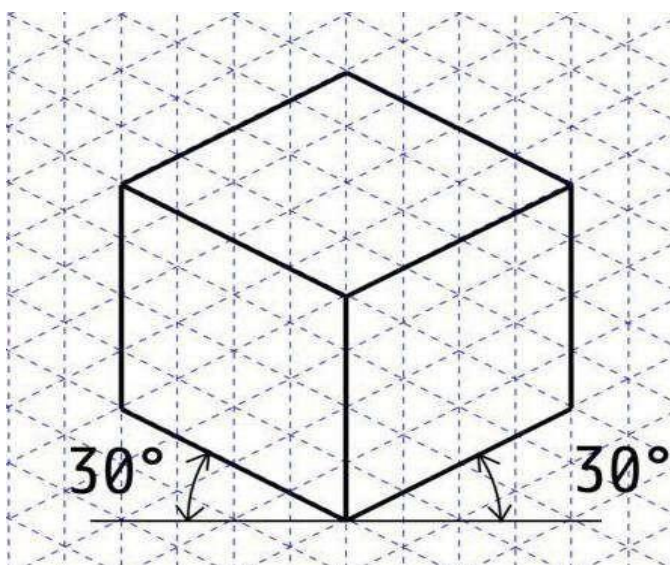
• Sample Drawing:



Job Sheet: 1.2 (b)

Job	Create an Isometric drawing of square block	
Materials/Resources	Pencil	1 Pc
	Eraser	1 Pc
	A 4 Paper	1 Pc
	Scale	1 Pc
	Sample Drawing	1 Pc
	Tee Square	1 Pc
	Set Square	1 Pc
Procedure	Collect required materials/resources.	
	Draw a straight line on drawing sheet.	
	Draw 30-degree angle to the horizontal using set square on the drawing sheet	
	In the same way draw 90-degree angle using set square on the drawing sheet	
	Follow trainers/instructors	

• Sample:





Assessment Plan

Module -1	Written test	Performance test	Observation	Oral test	Portfolio	Third party report	Discussion	Project	Weekly/Final report	Remarks
Learning Outcome 1	√		√	√						
Learning Outcome 2	√		√	√						



Competency Review

Here is a competency review for understand mechanical drawings.

Performance Criteria	Yes	No
Able to read and interpret different types of mechanical drawings (detail, assembly, weldment, etc.)?		
Correctly interprets general notes, specifications, and welding symbols?		
Can identify and define datums and their role in a tolerance scheme?		
Can generate different views (orthographic, isometric, sectional) from a 3D model?		
Correctly dimensions different geometric features (holes, slots, radii, angles)?		
Correctly uses symbols for diameter (Ø), radius (R), countersink, counterbore, and depth?		
Instructions are included in drawing?		

Module 2: Perform CAD modeling



Module Content

Module Description: This module describes the knowledge, skills and attitudes required of the operator to perform CAD modeling. It includes installing CAD software (SolidWorks), introduction to CAD software (SolidWorks) and interface, CAD software (SolidWorks) interface setup and CAD modeling (2D and 3D).

Minimum Time: 100 Hours



Learning Outcome:

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

- 2.1 Install CAD software (SolidWorks)
- 2.2 Create 2D drawing
- 2.3 Develop 3D model
- 2.4 Create mold based on 3D model



Performance Criteria:

1. Components of computer are identified and interpreted.
2. CAD software is installed as per standard operating procedure.
3. Screen display areas and basic parameters are set as per job specification.
4. Sketch tools are identified and selected for 2D drawing.
5. Sketch modify tools are identified and selected for 2D drawing.
6. 2D sketch relations are identified and explained.
7. 2D drawing is created.
8. CAD drawing is reviewed and modified, as necessary.
9. Features tools are identified and selected for 3D model.
10. Direct editing tools are identified and selected for 3D model.
11. 3D model is developed.
12. 3D model is reviewed and modified, if necessary.
13. 2D drawing is generated for review and finalizing the model.
14. Mold is created based on the 3D model.
15. Mold is converted to CAM supported files.
16. A separate folder is developed and the mold is saved for CAM programming.

Learning Outcome 2.1: Install CAD Software



Content:

- Check system requirements
- Find CAD software (SolidWorks) license key number
- Download and run the installer
- Navigate to the downloaded files and find the setup.exe or sldIM.exe file.



Assessment Criteria

1. Be able to Check system requirements.
2. Be able to find CAD software (SolidWorks) license key number.
3. Be able to download and run the installer.
4. Be able to navigate to the downloaded files and find the setup.exe or sldIM.exe file.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. A computer set.
2. Must meet the minimum and recommended specifications:
 - a) Operating System: 64-bit Windows 10 or 11
 - b) Processor: Intel or AMD processor, 3.3 GHz or higher
 - c) RAM: Minimum 8GB, with 32GB or more recommended for larger assemblies and simulation
 - d) Storage: A Solid-State Drive (SSD) with at least 20GB of free disk space
 - e) Graphics Card: A certified graphics card (NVIDIA Quadro or AMD Radeon Pro)
3. SolidWorks software with license key number.



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Install CAD Software	• Information Sheet: 2.1-1 to 2.1-1



Information Sheet: 2.1-1

Learning Objective: Install CAD Software (SolidWorks).

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 days 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: CAD means ‘computer aided design’ is mainly used for detailed engineering of 3D models or 2D drawings of physical components, but it is also used throughout the engineering process from conceptual design and layout of products, through strength and dynamic analysis of assemblies to definition of manufacturing methods of components.

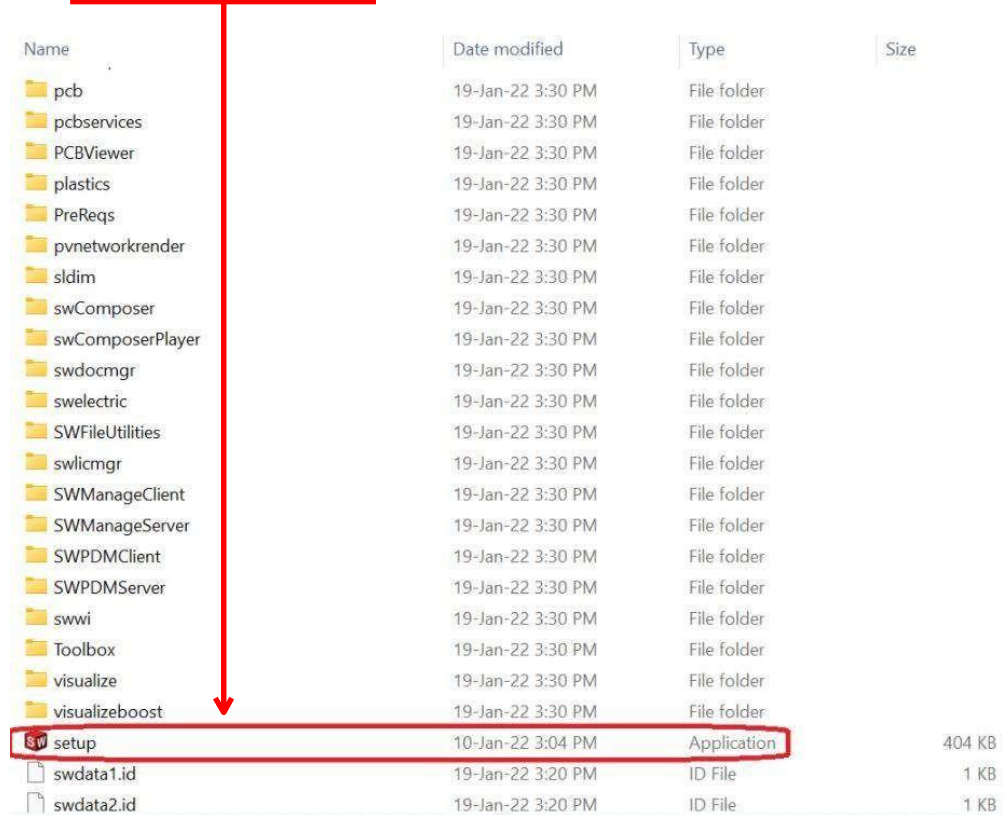
Operating System: Operating System is system software that manages computer hardware and software resources and provides common services for computer programs.

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.

Installation:

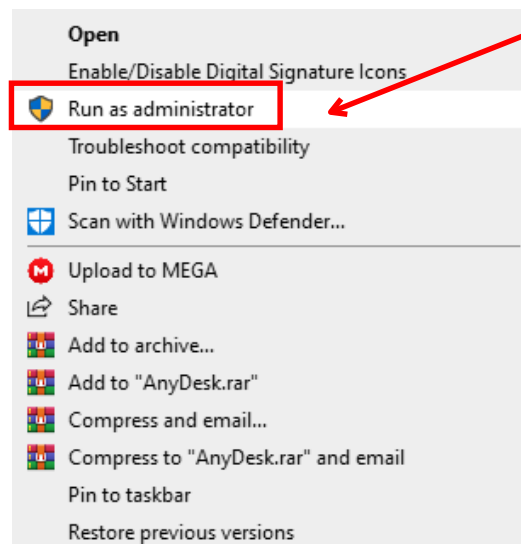
Step 1: Download the installation files.

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

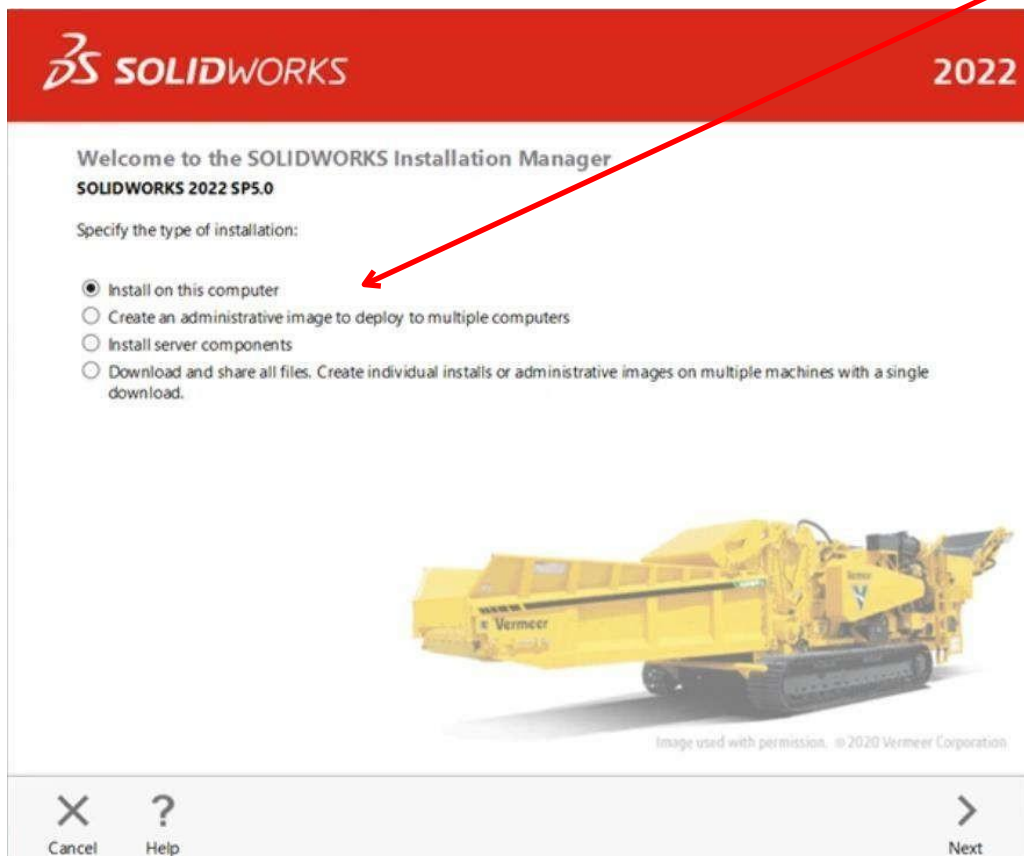


Name	Date modified	Type	Size
pcb	19-Jan-22 3:30 PM	File folder	
pcbservices	19-Jan-22 3:30 PM	File folder	
PCBViewer	19-Jan-22 3:30 PM	File folder	
plastics	19-Jan-22 3:30 PM	File folder	
PreReqs	19-Jan-22 3:30 PM	File folder	
pvnetworkrender	19-Jan-22 3:30 PM	File folder	
sldim	19-Jan-22 3:30 PM	File folder	
swComposer	19-Jan-22 3:30 PM	File folder	
swComposerPlayer	19-Jan-22 3:30 PM	File folder	
swdocmgr	19-Jan-22 3:30 PM	File folder	
swelectric	19-Jan-22 3:30 PM	File folder	
SWFileUtilities	19-Jan-22 3:30 PM	File folder	
swlicmgr	19-Jan-22 3:30 PM	File folder	
SWManageClient	19-Jan-22 3:30 PM	File folder	
SWManageServer	19-Jan-22 3:30 PM	File folder	
SWPDMClient	19-Jan-22 3:30 PM	File folder	
SWPDMServer	19-Jan-22 3:30 PM	File folder	
swwi	19-Jan-22 3:30 PM	File folder	
Toolbox	19-Jan-22 3:30 PM	File folder	
visualize	19-Jan-22 3:30 PM	File folder	
visualizeboost	19-Jan-22 3:30 PM	File folder	
setup	10-Jan-22 3:04 PM	Application	404 KB
swdata1.id	19-Jan-22 3:20 PM	ID File	1 KB
swdata2.id	19-Jan-22 3:20 PM	ID File	1 KB

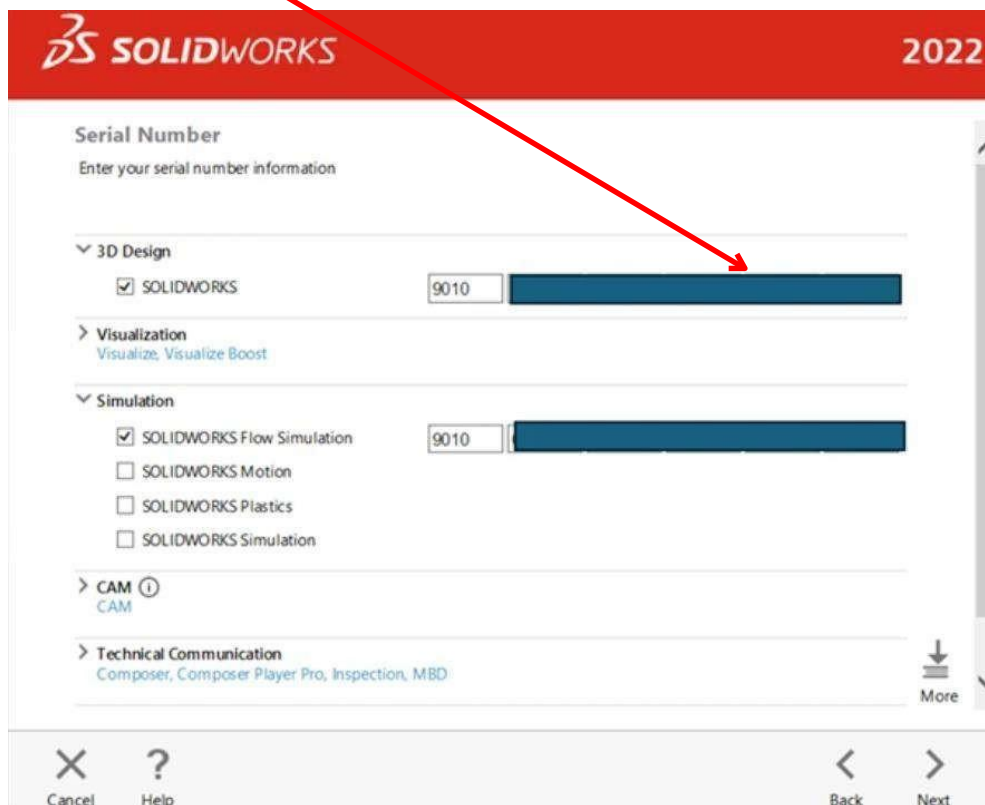
Step 3: Right on click the **SolidWorksSetup.exe** file you downloaded and select **Run** as administrator.



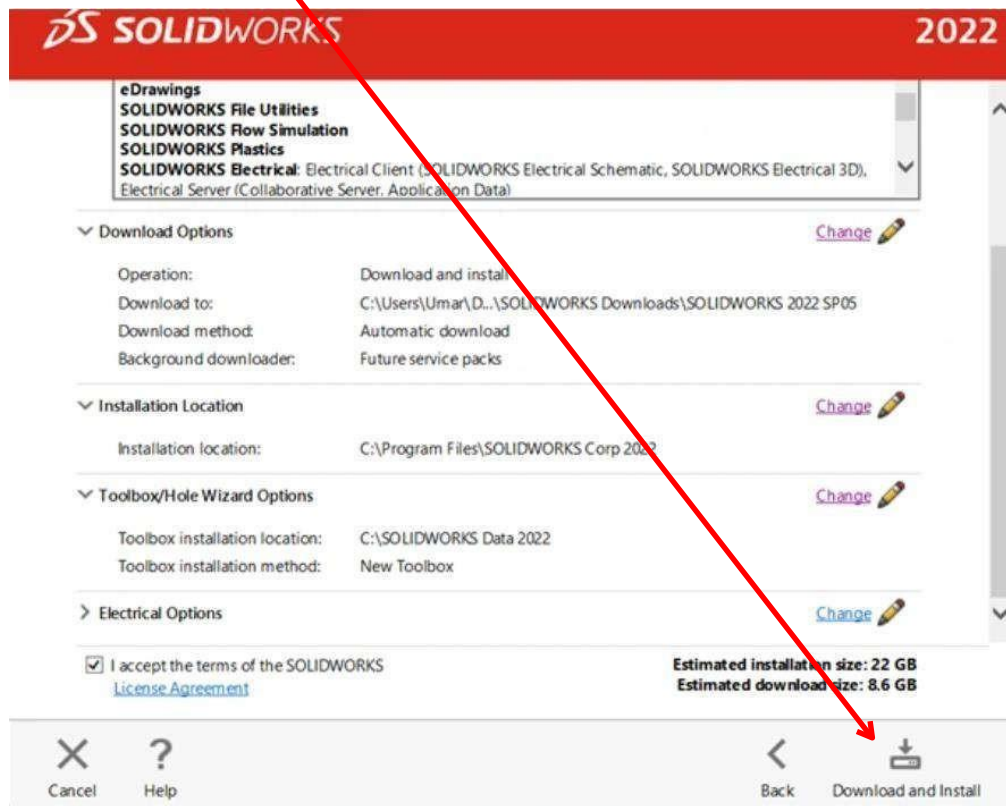
Step 4: When the SolidWorks Installation Manager window appears, select **Install on this computer.**



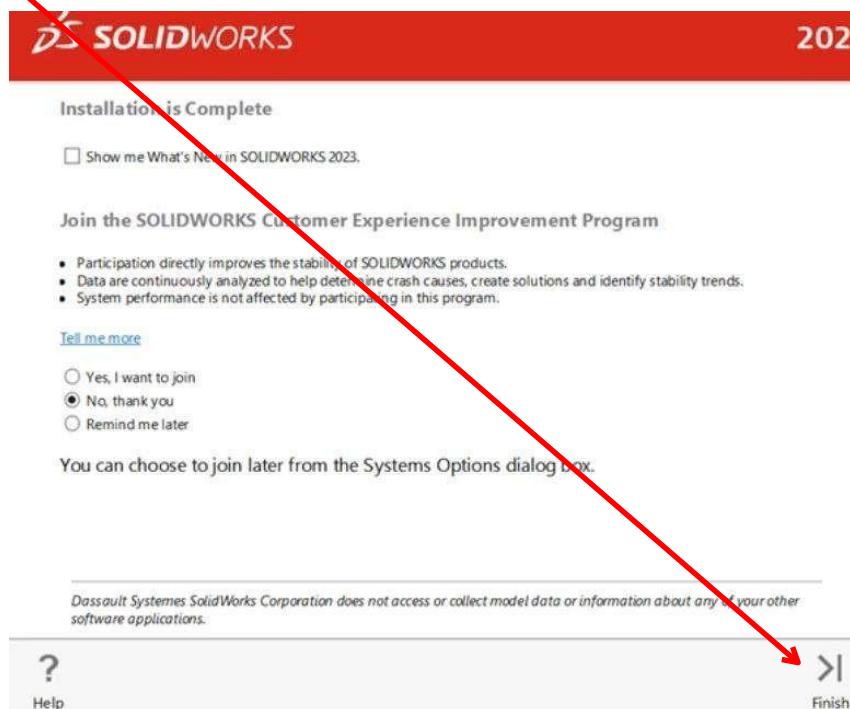
Step 5: Enter your **Serial Number** for SolidWorks in the appropriate field.



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**.



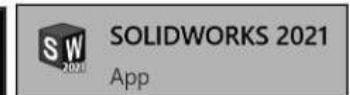
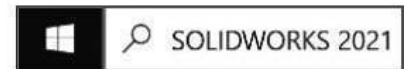
Step 8: Once the installation is complete, reboot your computer to finalize the process



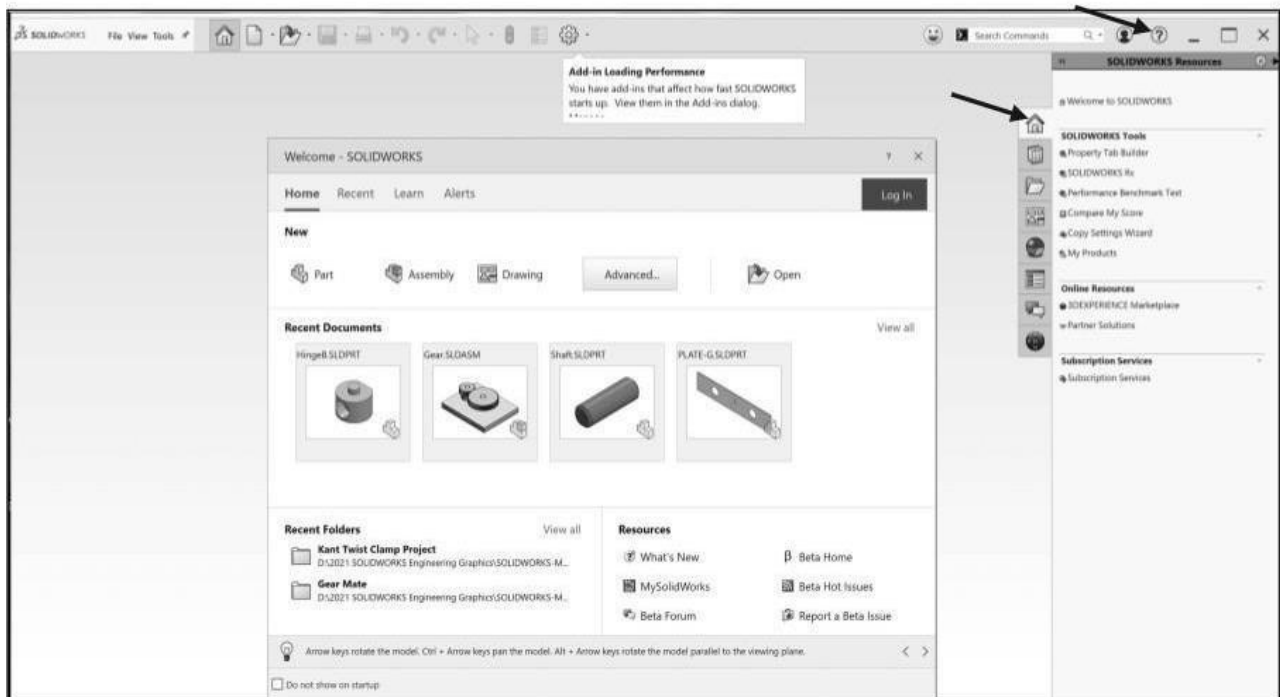
Information Sheet: 2.1-2

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.

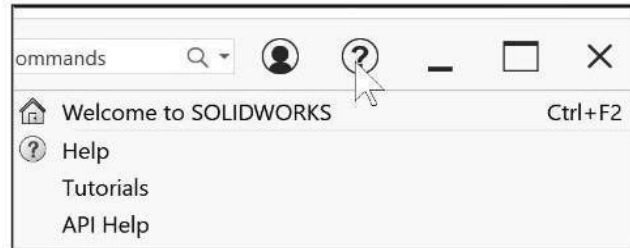


The Welcome dialog box provides a convenient way to open recent documents (Parts, Assemblies and Drawings), view recent folders, access SOLIDWORKS resources, and stay updated on SOLIDWORKS news.



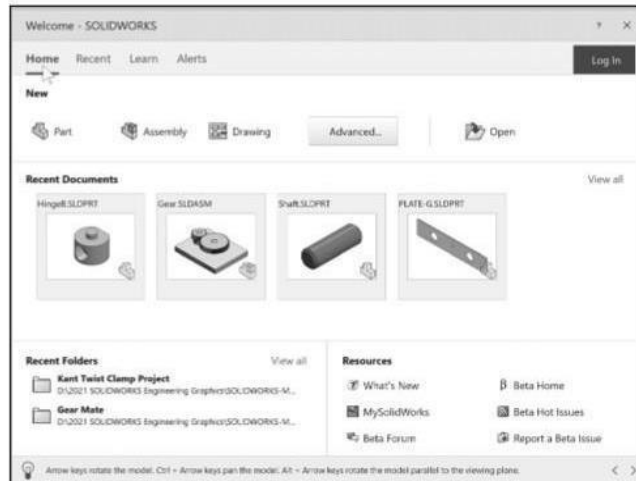
- 3) **View** your options. Do not open a document at this time.

- 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.

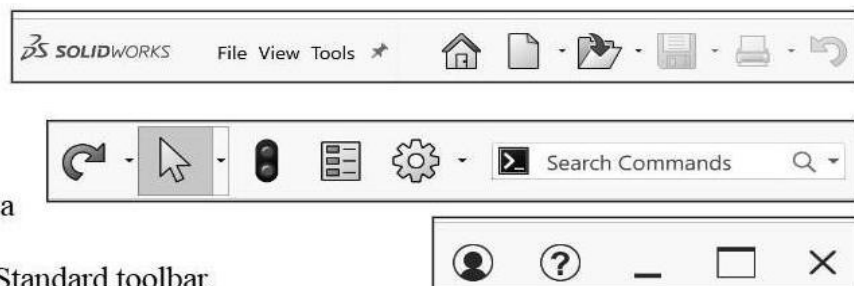


View the SOLIDWORKS Graphics window.



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.

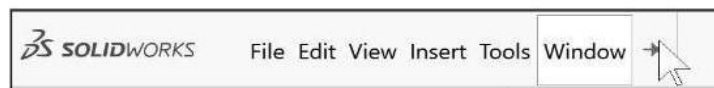
Menu Bar menu

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.

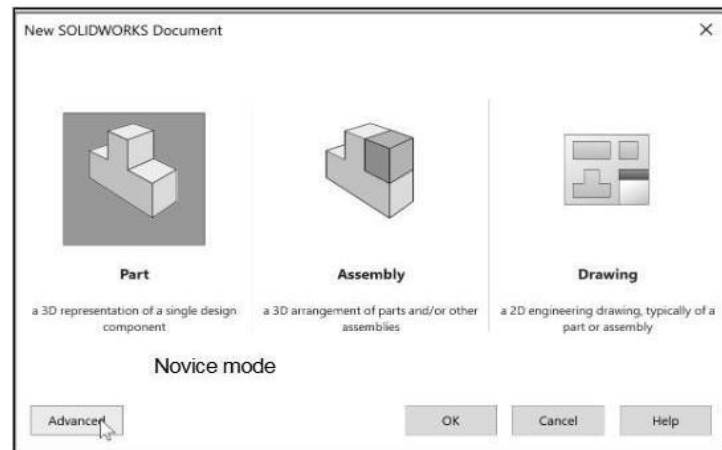
To close a SOLIDWORKS drop-down menu, press the Esc key. You can also click any other part of the SOLIDWORKS Graphics window or click another drop-down menu.



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.

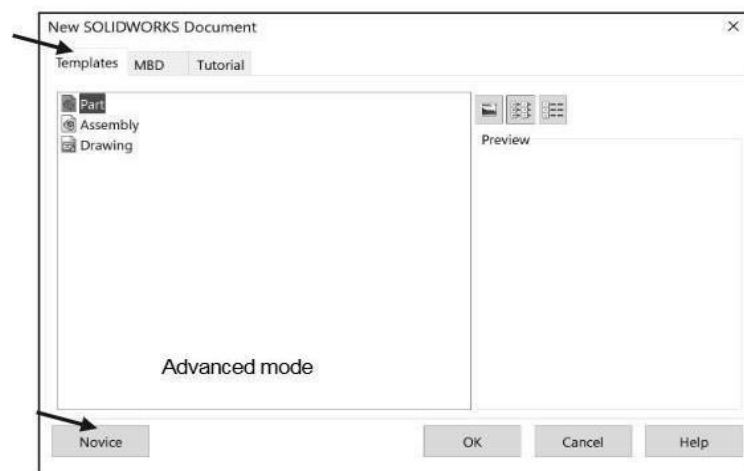
Create a new part.

- 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.

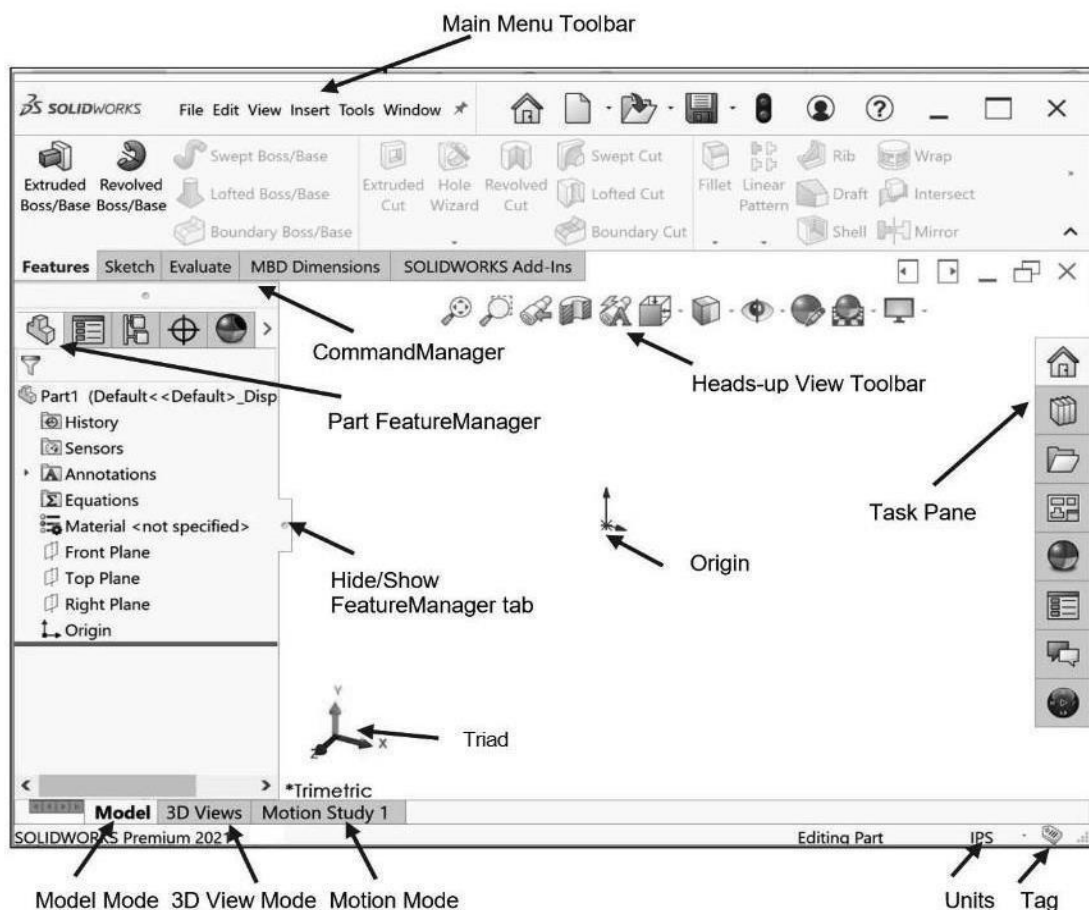
The default SOLIDWORKS installation contains three tabs in the New SOLIDWORKS Document dialog box: *Templates*, *MBD*, and *Tutorial*. The *Templates* tab corresponds to the default SOLIDWORKS templates. The *MBD* tab corresponds to the templates utilized in the SOLIDWORKS (Model Based Definition). The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

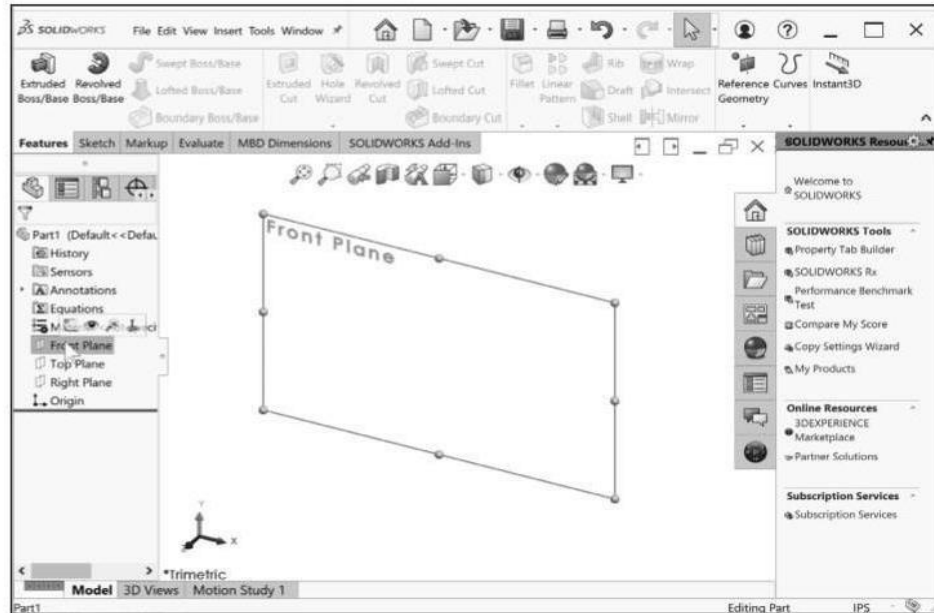
 Edit the document units directly from the Graphics window as illustrated.

 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.

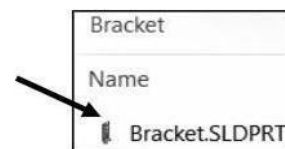
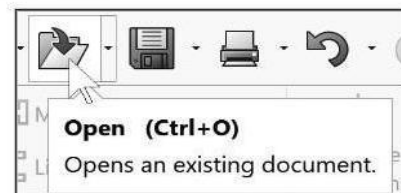


Download all model files from the SDC Publications website (<https://www.sdcpublications.com/downloads/978-1-63057-409-3>). Open the Bracket part. Review the features and sketches in the Bracket FeatureManager. Work directly from a local hard drive.

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

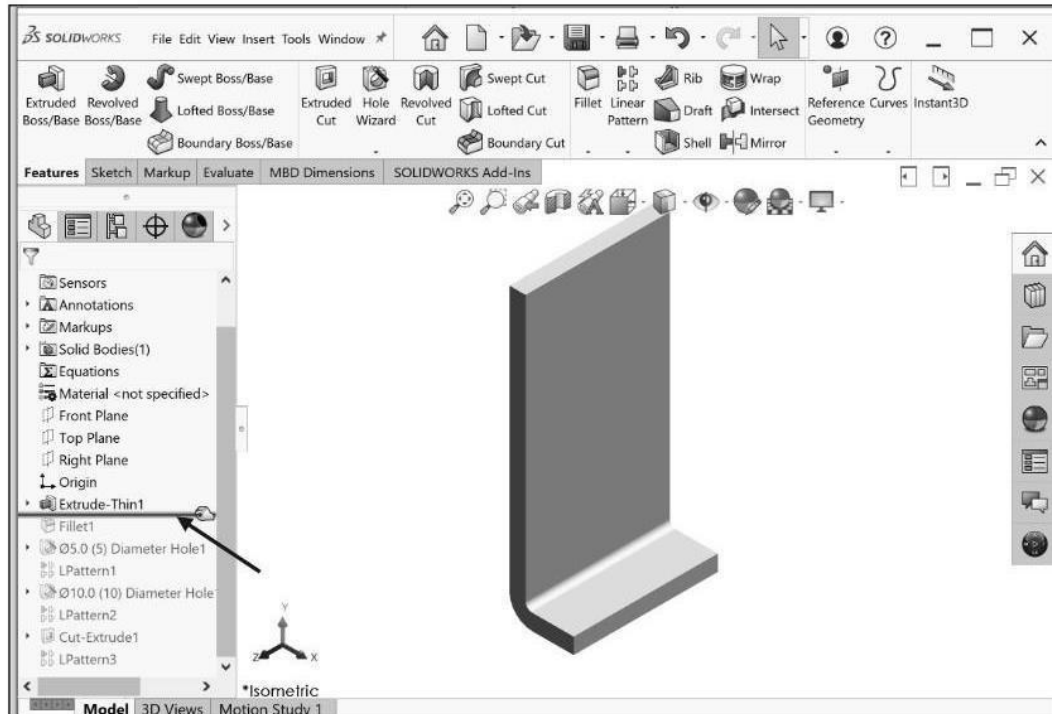
Download the SOLIDWORKS folder. Open an existing SOLIDWORKS part.

- 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.



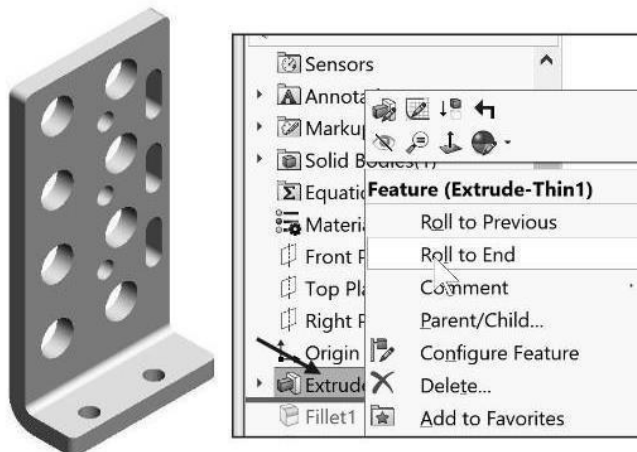
View the first feature in the Bracket Part.

- 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.

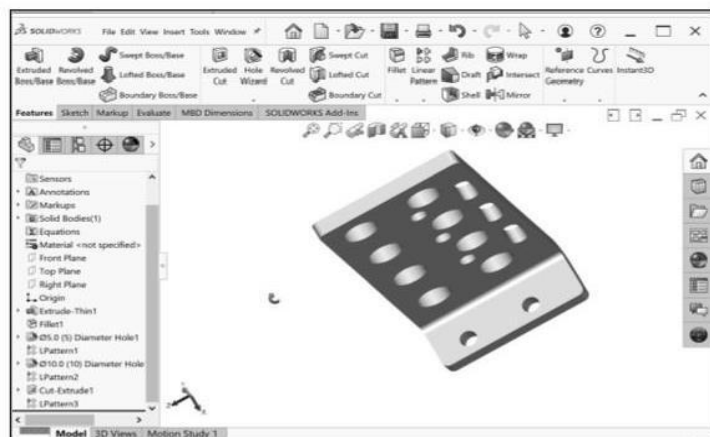
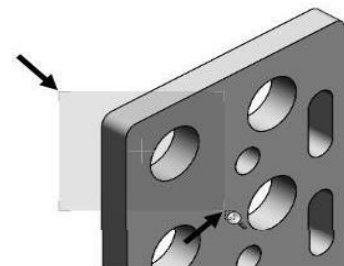
- 29)** Click the **Zoom to Area**  icon.

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.

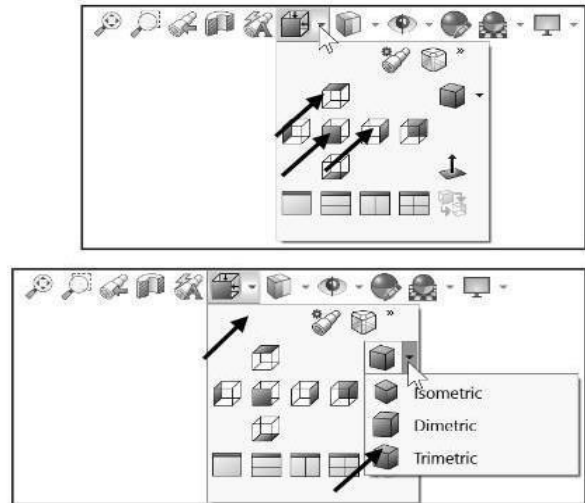


Display a few Standard Views.

- 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.

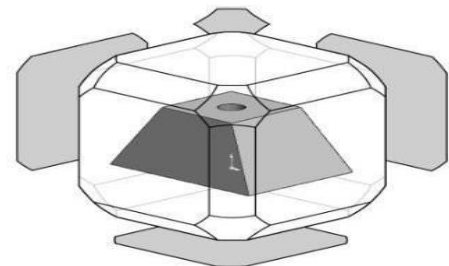


- **Section View** : Display a cutaway of a part or assembly, using one or more cross section planes.
- **Dynamic Annotation Views** : Only available with SOLIDWORKS MBD. Control how annotations are displayed when you rotate a model.



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).

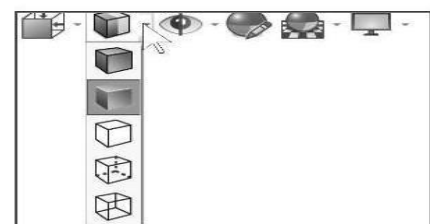
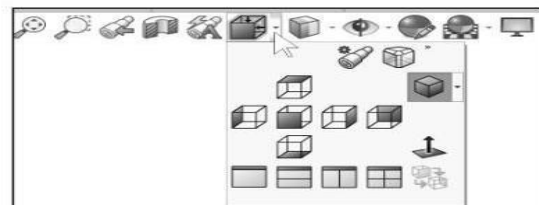


Click the View Selector icon in the Orientation dialog box to show or hide the in-context View Selector.

 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*.



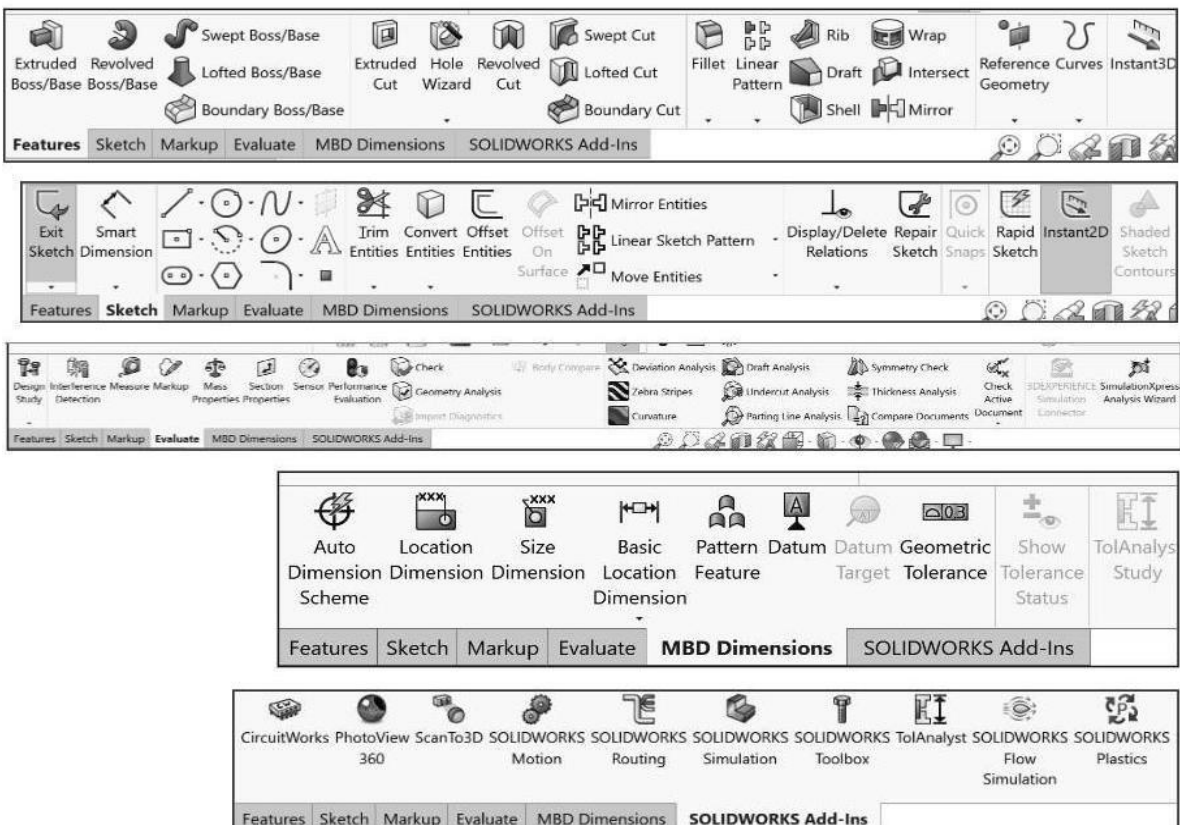
SOLIDWORKS CommandManager

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.

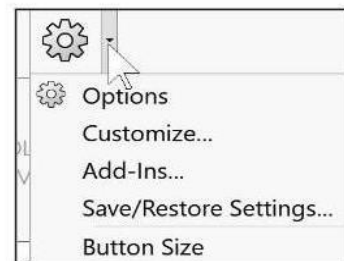


For commercial users, SOLIDWORKS Model Based Definition (MBD) and SOLIDWORKS CAM is a separate application. For education users, SOLIDWORKS MBD and SOLIDWORKS CAM is included in the SOLIDWORKS Education Edition.

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

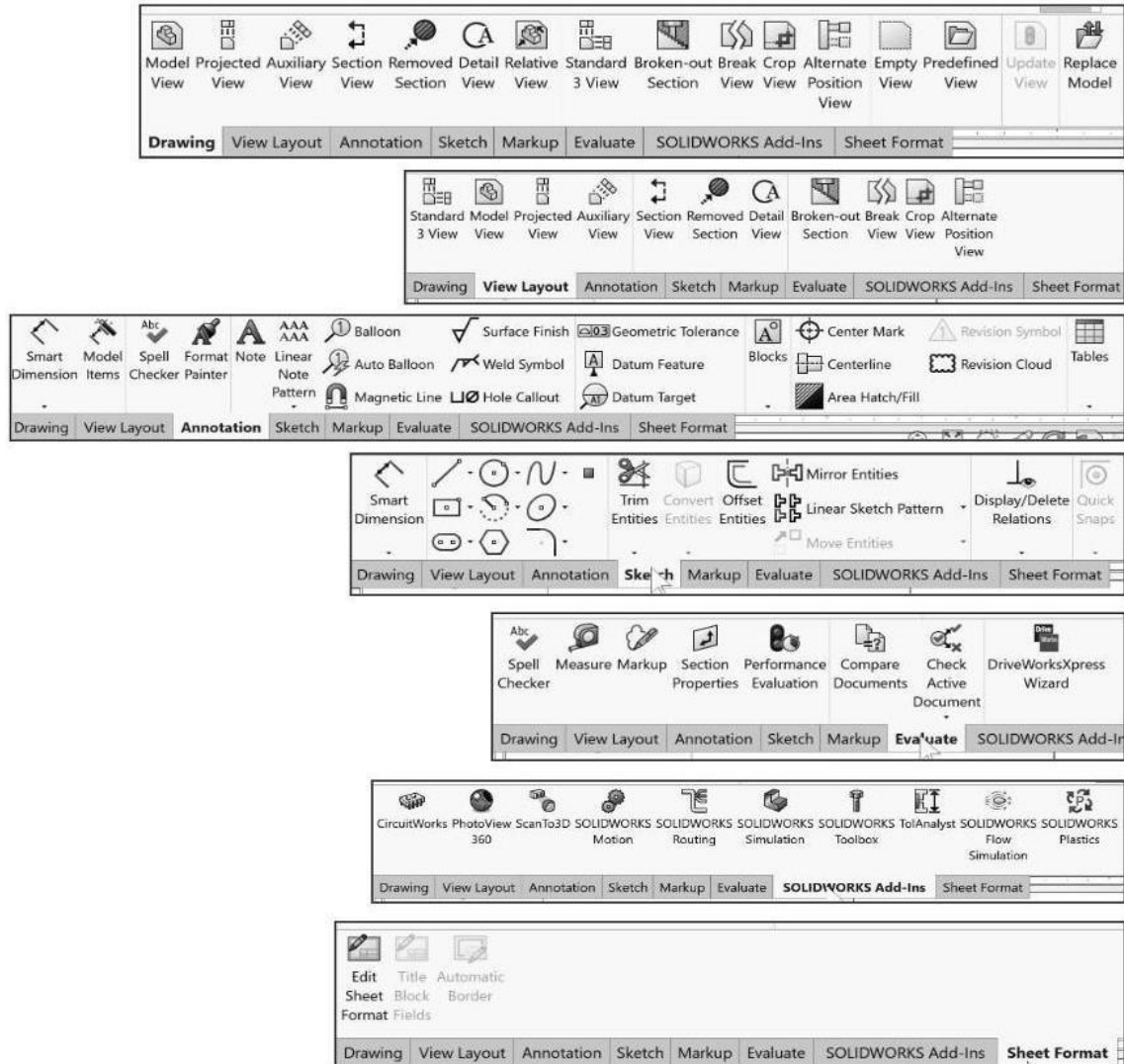


Set button size from the Toolbars tab of the Customize dialog box. To facilitate element selection on touch interfaces such as tablets, you can set up the larger Size buttons and text from the Options menu (Standard toolbar).

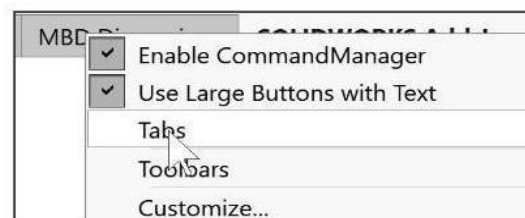


The SOLIDWORKS CommandManager is a Context-sensitive toolbar that automatically updates based on the toolbar you want to access. By default, it has toolbars embedded in it based on your active document type. The available tools are feature and document dependent.

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

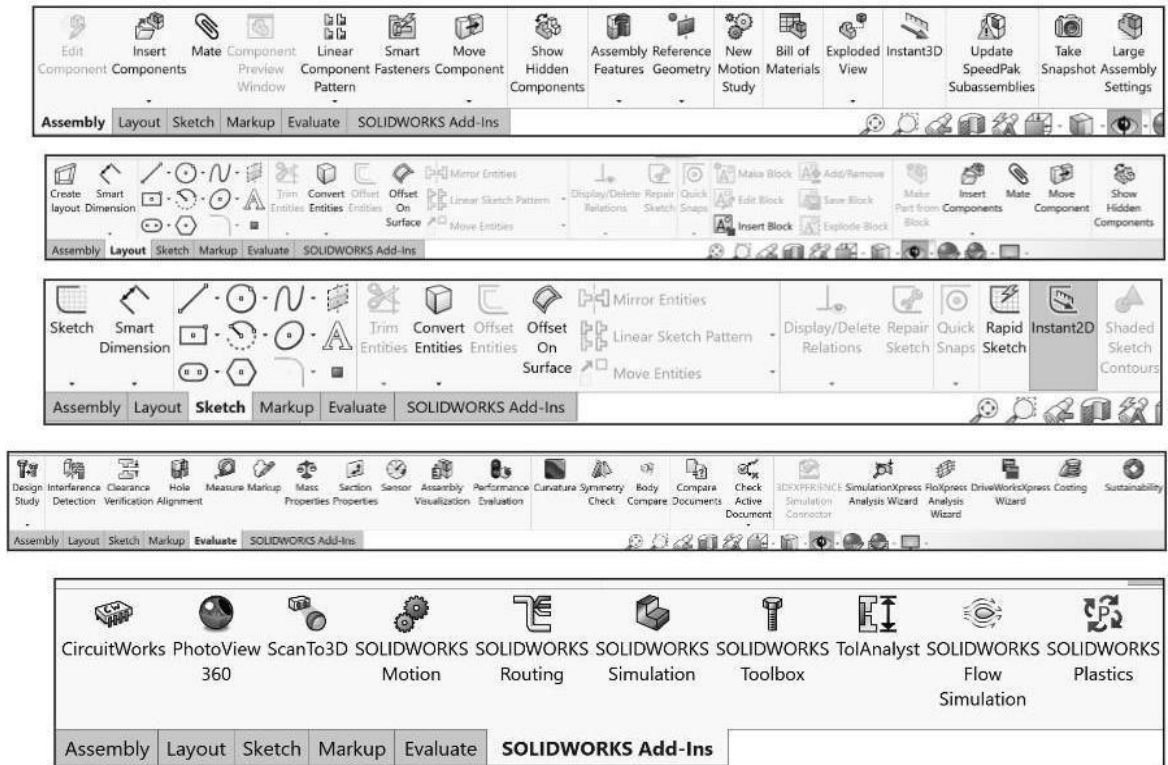


To add a custom tab, right-click on a tab and click Customize. You can also select to add a blank tab and populate it with custom tools from the Customize dialog box.



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.

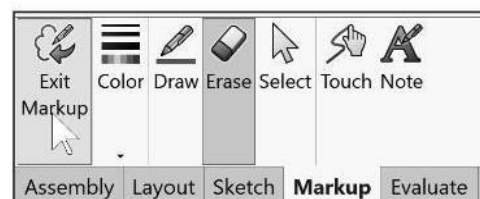
Below is an illustrated CommandManager for an **Assembly** document. Tabs will vary depending on system setup and Add-ins.



 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** and  **Touch** are **not** available for non-touch devices.



Design Library

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

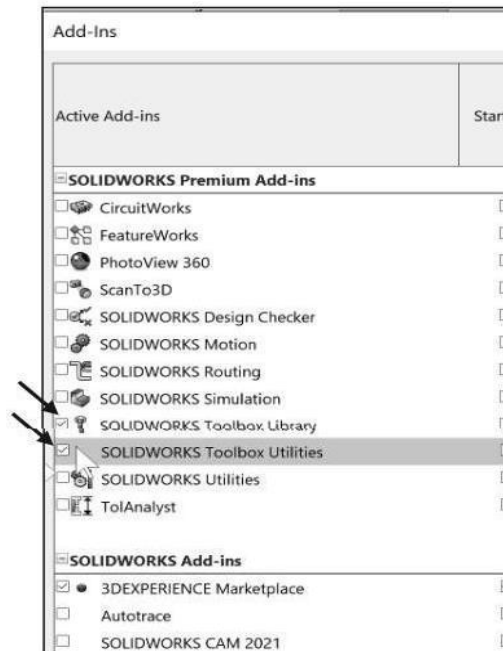
The Design Library tab contains default selections. Each default selection contains additional sub categories.

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.*

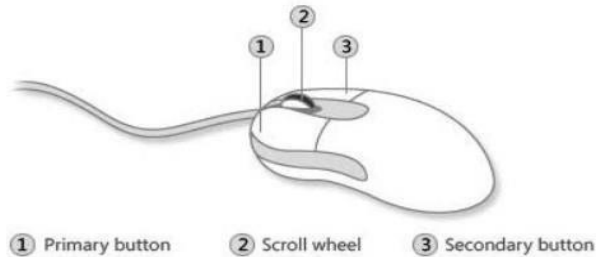
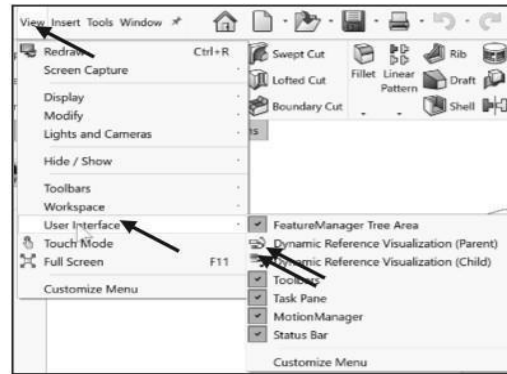


Use Dynamic reference visualization for a part, assembly and mates.

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).

Learning Outcome 2.2: Create 2D Drawing



Content:

- How to create a 2D drawing
- Opening a new SolidWorks part project
- Preparing a sketching plane
- Creating a sketch.
- Adding dimensions
- Trimming entities



Assessment Criteria

1. Be able to use SolidWorks 2D Sketch Tool.
2. Be able to create 2D drawing.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. A computer set
2. SolidWorks software
3. Drawing instructions



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Create 2D Drawing	• Information Sheet: 2.2-1 to 2.2-2 • Self-check: 2.2-2 • Answer Script: 2.2-2 • Jobb Sheet: 2.2 (a) to 2.2 (c)

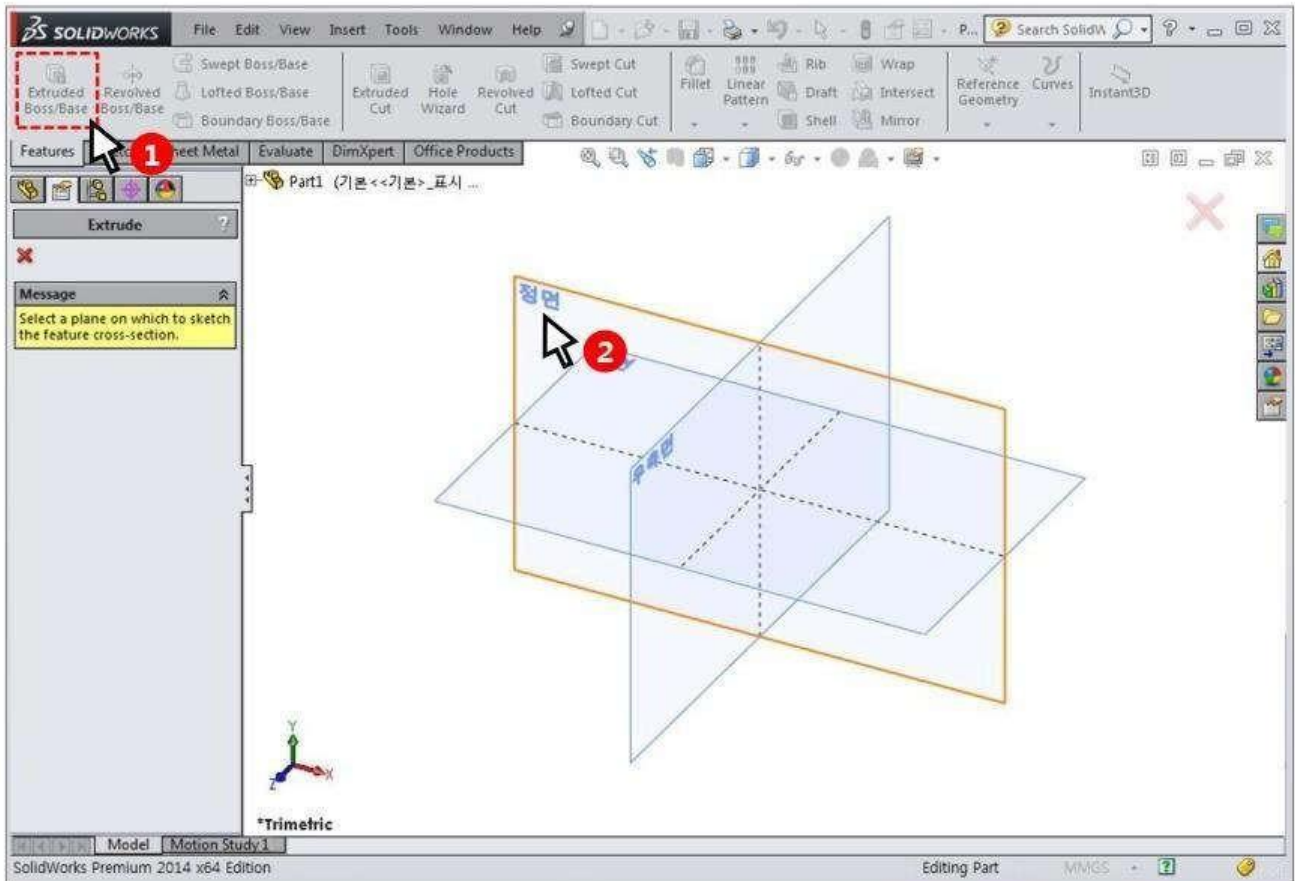


Information Sheet: 2.2-1

Learning Objective: Selection of Different Planes and Activating Sketch Tools.

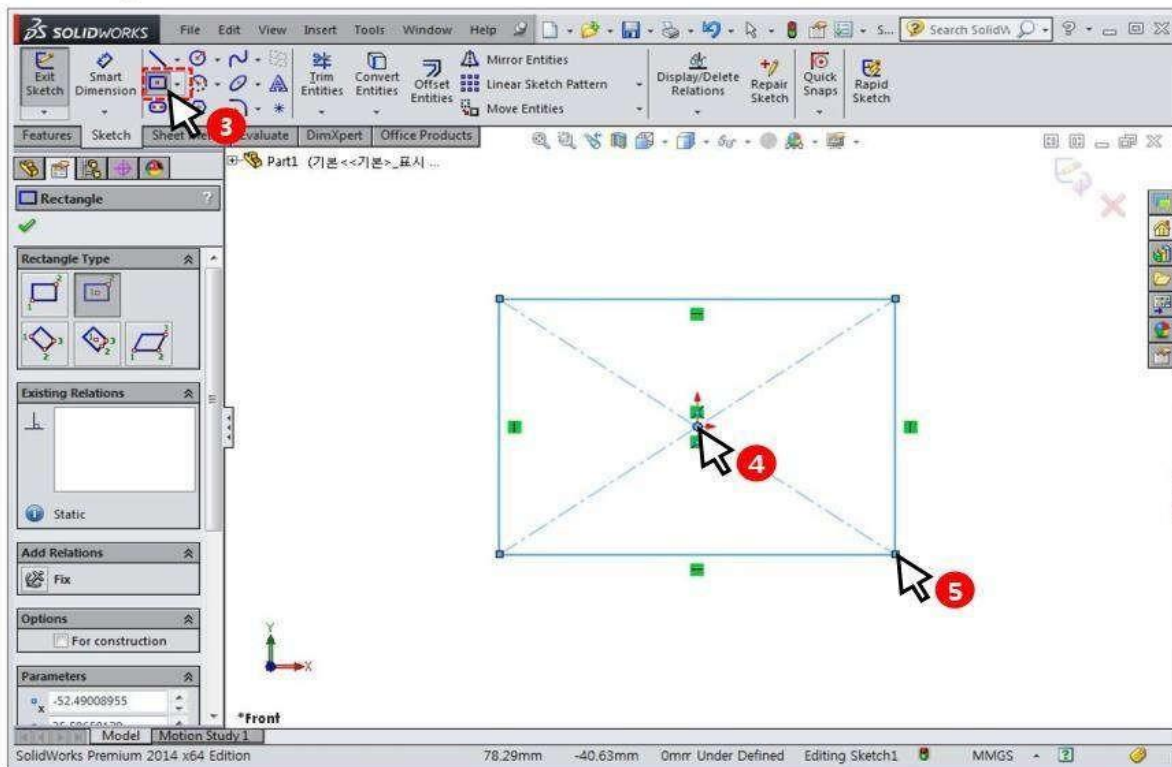
1.

Create a new sketch



2.

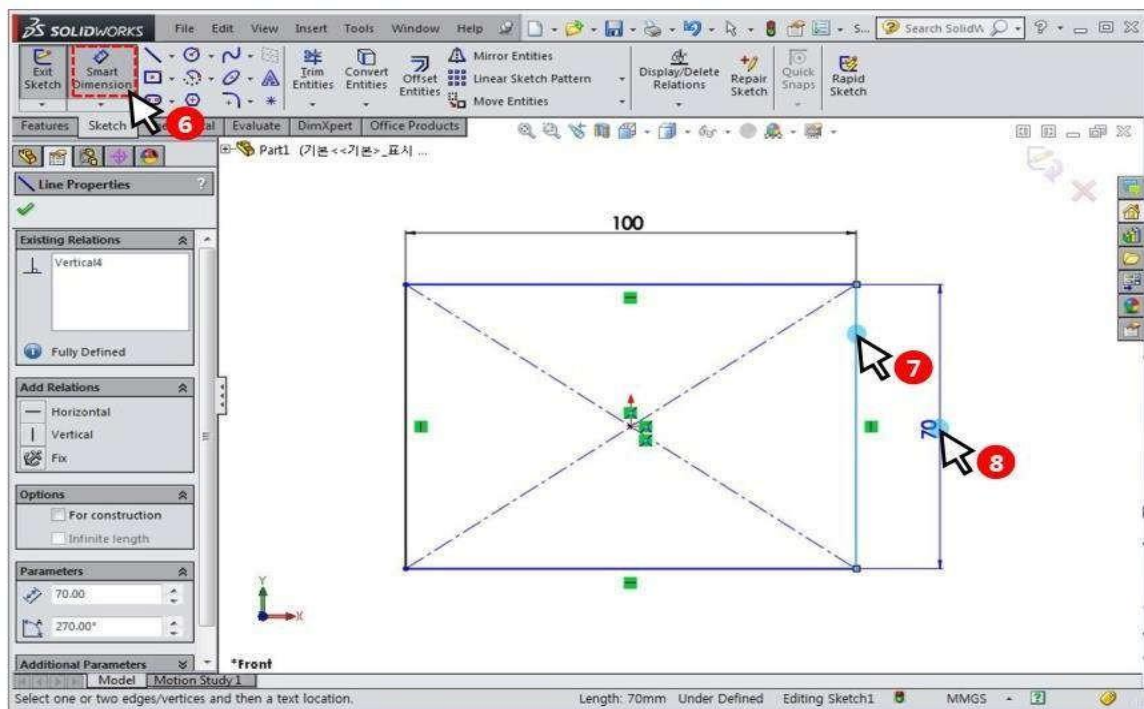
Rectangle



- Sketches a rectangle from the center.

3.

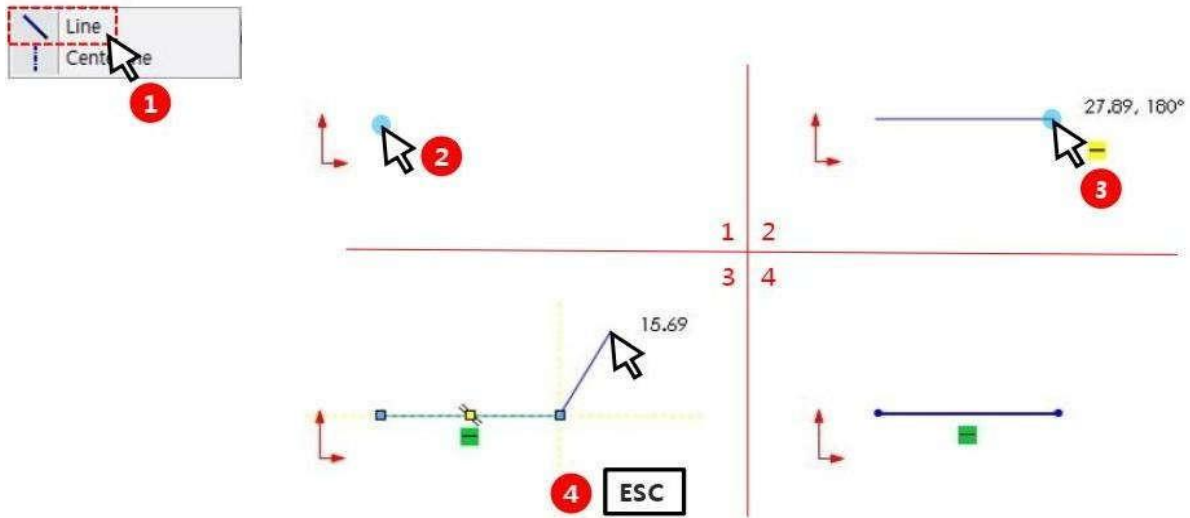
Smart Dimension



- Creates a dimension for one or more selected entities.

4.

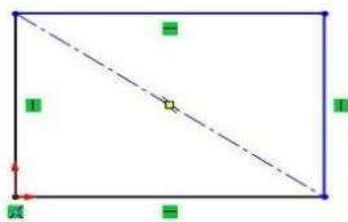
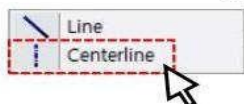
■ **Line** : Sketches a line.



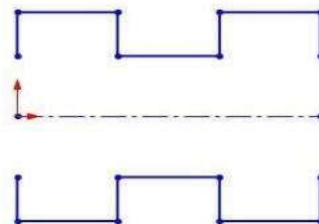
1. In an active 2D sketch, click Sketch tab > Line
2. In the graphics window, click to set the start point.
3. Do one of the following:
 - Continue clicking to create contiguous straight-line segments, or double-click to end the line.

5.

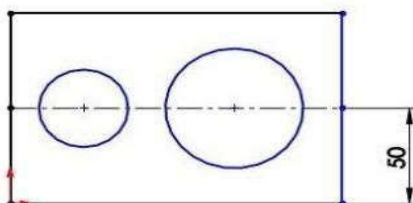
■ **Centerline** : Sketches a centerline. Use centerlines to create symmetrical sketch element, revolved features, or as construction geometry.



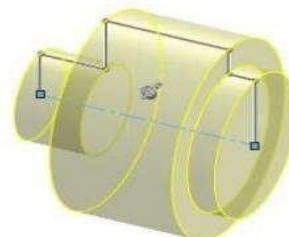
- Centerline to give constraints



- Centerline used for mirror entities



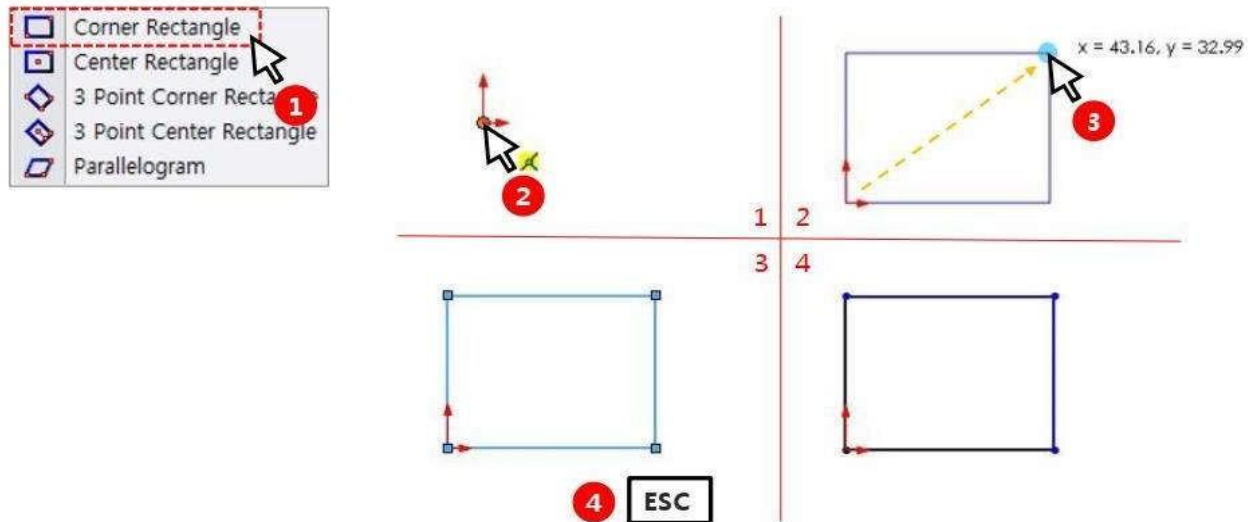
- Centerline to give dimensions



- Centerline for creating rotational features

6.

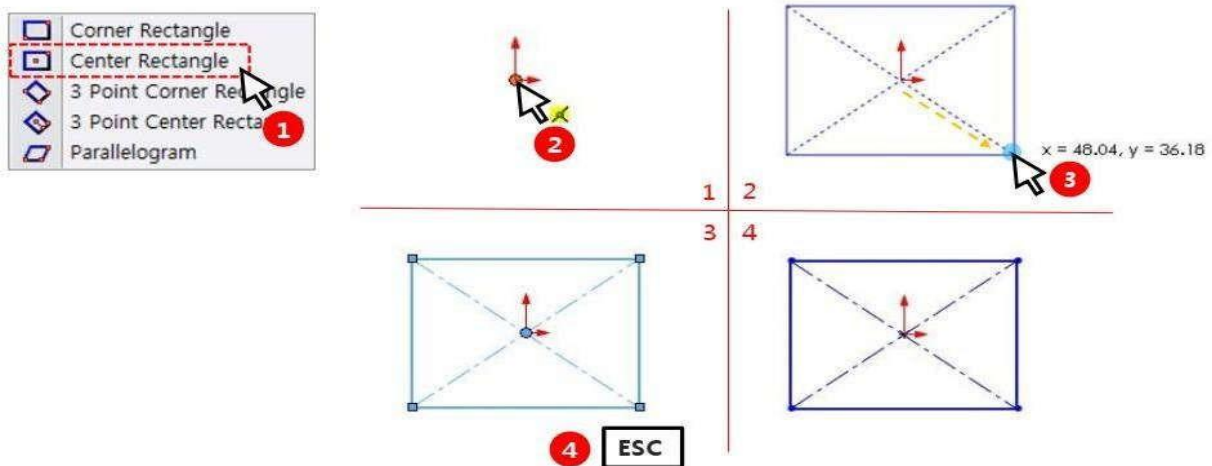
■ **Corner Rectangle** : Sketches a rectangle from the corner points.



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.

7.

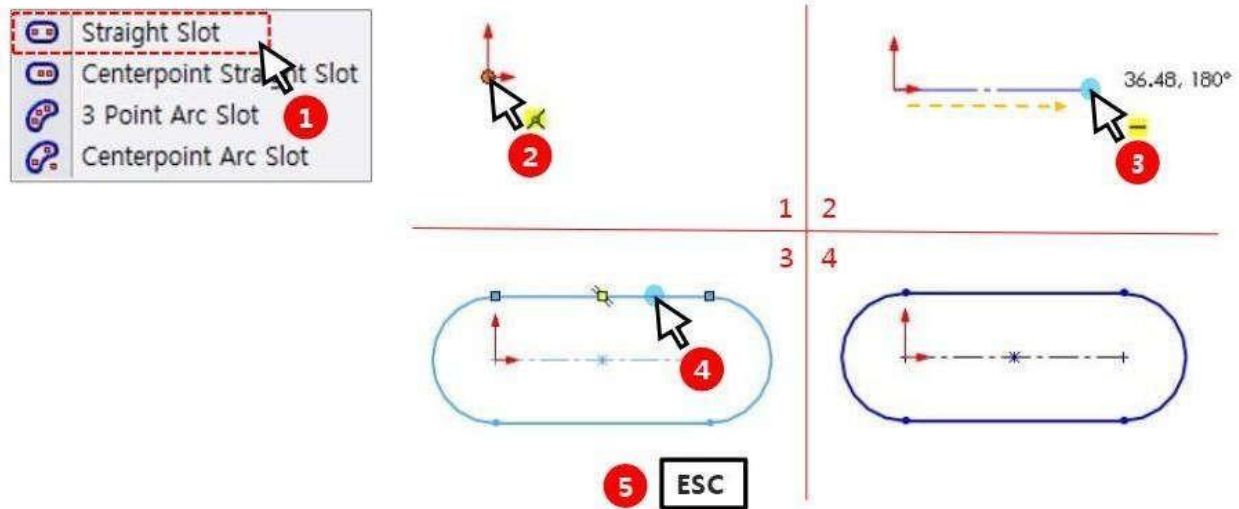
■ **Center Rectangle** : Sketches a rectangle from the center.



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.

8.

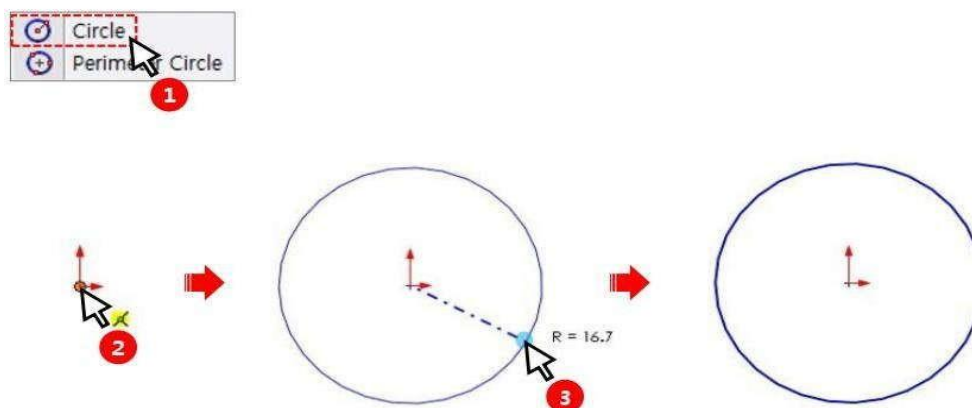
■ **Straight Slot** : Sketches a linear slot by defining the placement and distance of slot arc centers, and then the slot width.



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.

9.

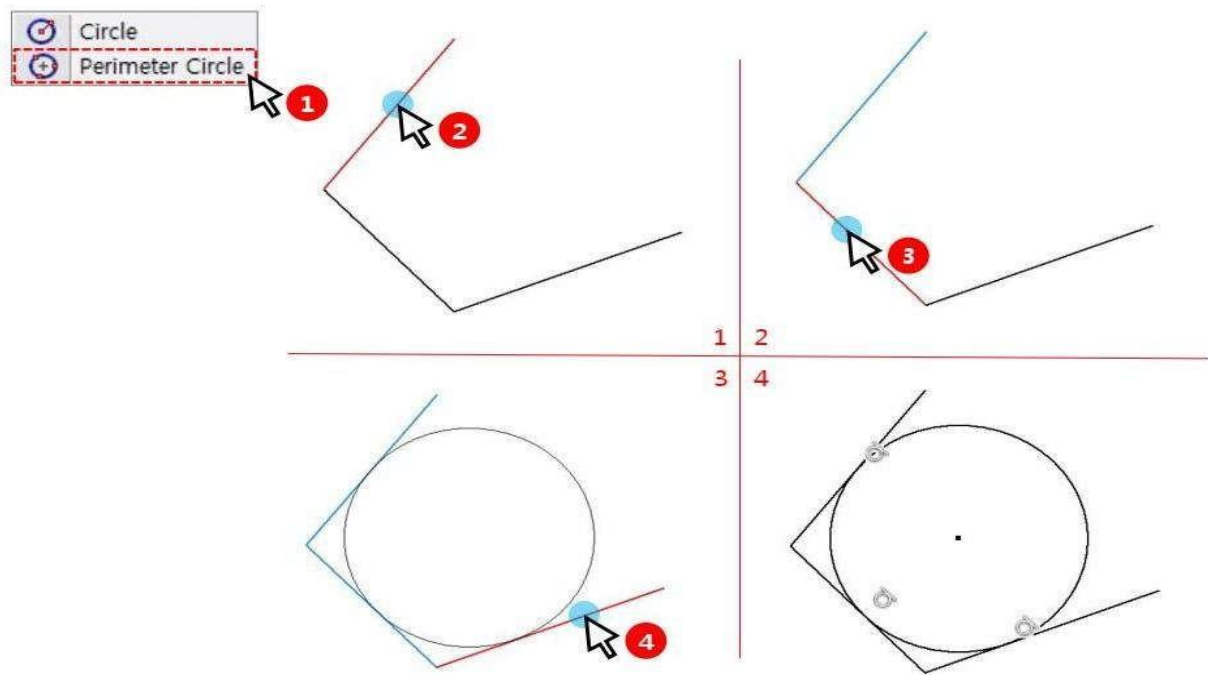
■ **Circle** : Sketches a circle from the center point and radius.



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,

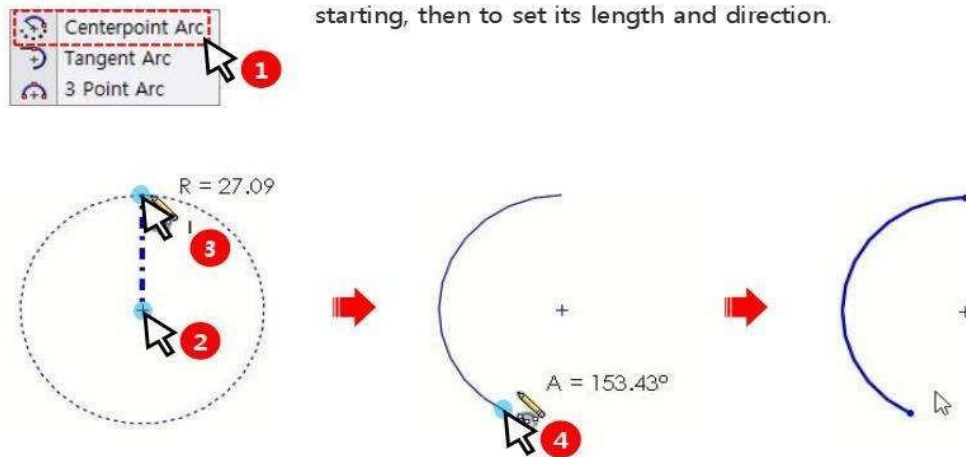
10.

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



11.

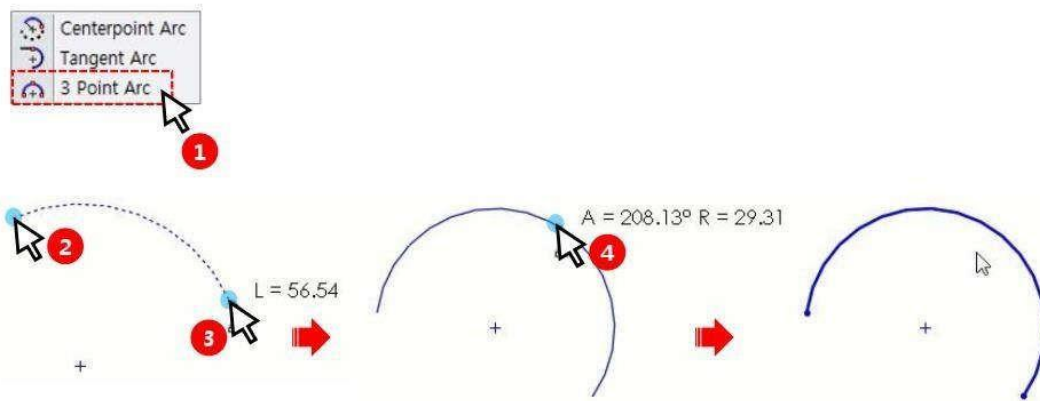
■ **Centerpoint Arc** : Sketches a center point arc. Set the center point. Drag to place the arc starting, then to set its length and direction.



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 .

12.

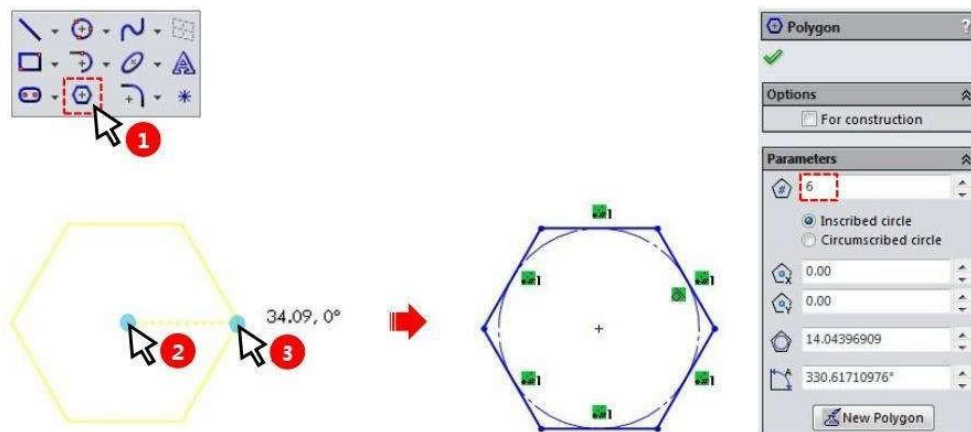
■ **3 Point Arc** : Sketches arcs by specifying three points (start, end, and midpoint).



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 .

13.

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



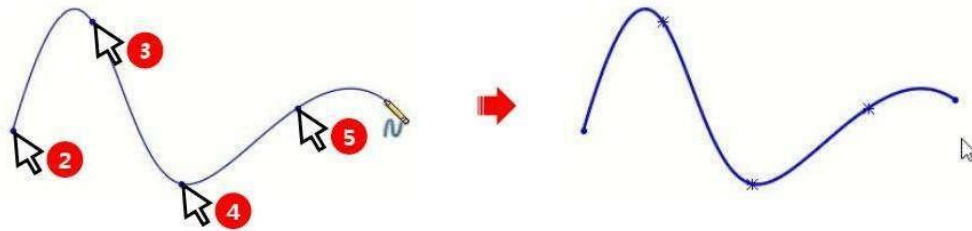
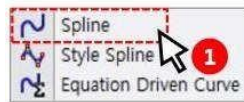
1. Click **Polygon** on the Sketch toolbar, or click **Tools > Sketch Entities > Polygon**.

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.

14.

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



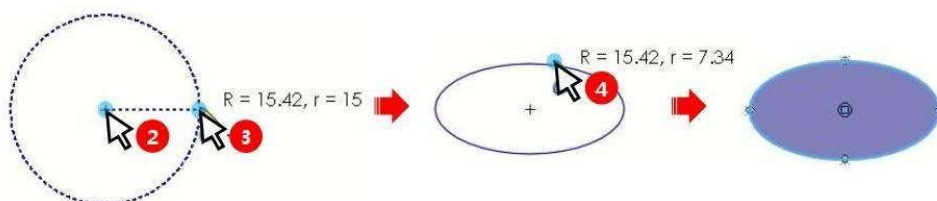
1. Click **Spline**  (Sketch toolbar) or **Tools > Sketch Entities > Spline**.

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.

15.

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



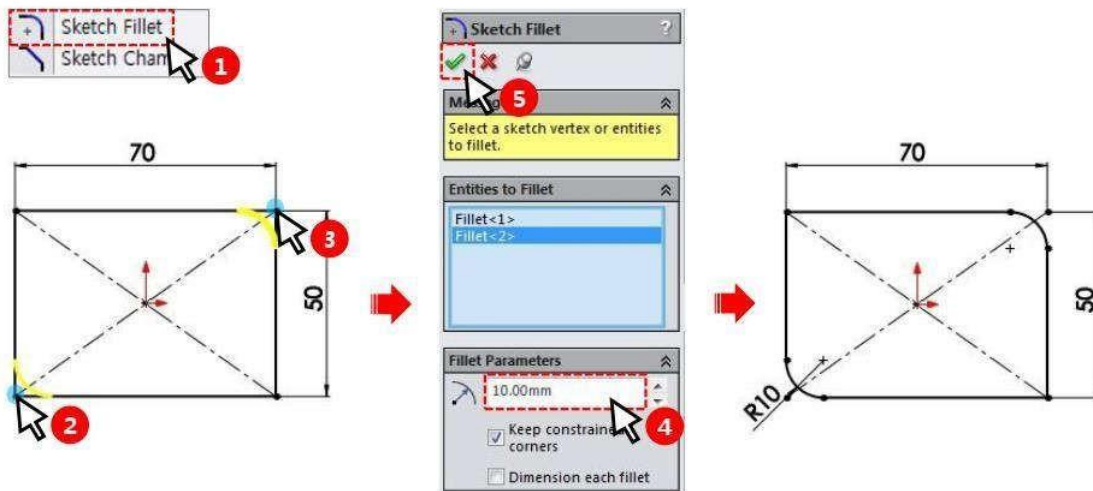
1. Click **Ellipse**  on the Sketch toolbar or **Tools > Sketch Entities > Ellipse**.

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.

16.

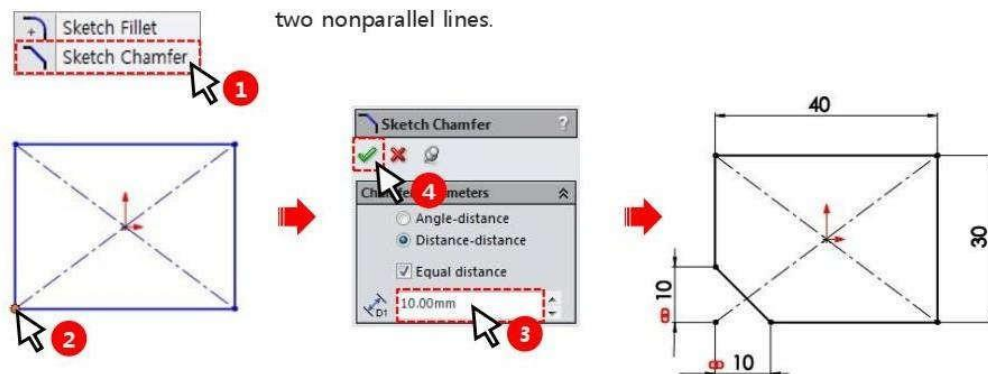
■ **Sketch Fillet** : Sketches rounded corners (vertices) at the intersection of two selected lines.



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.

17.

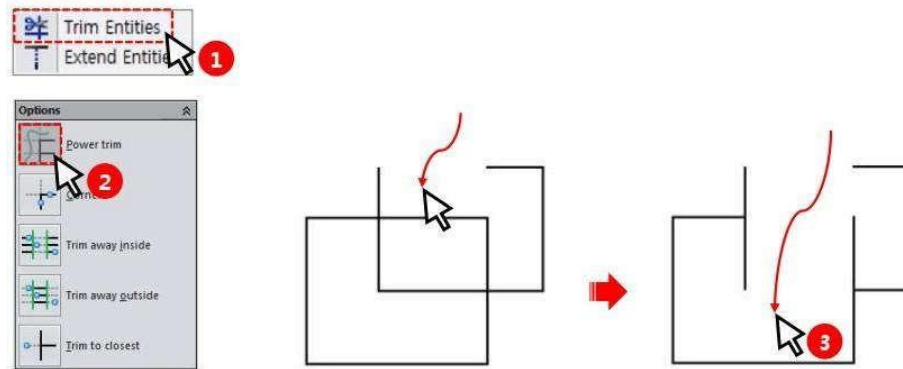
■ **Sketch Chamfer** : The Chamfer tool creates a beveled corner at the intersection of any two nonparallel lines.



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.

18.

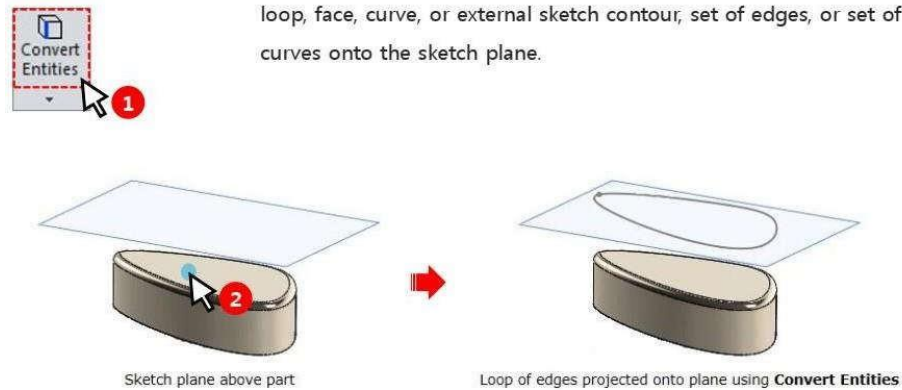
Trim Entities (Power trim)



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 .

19.

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.



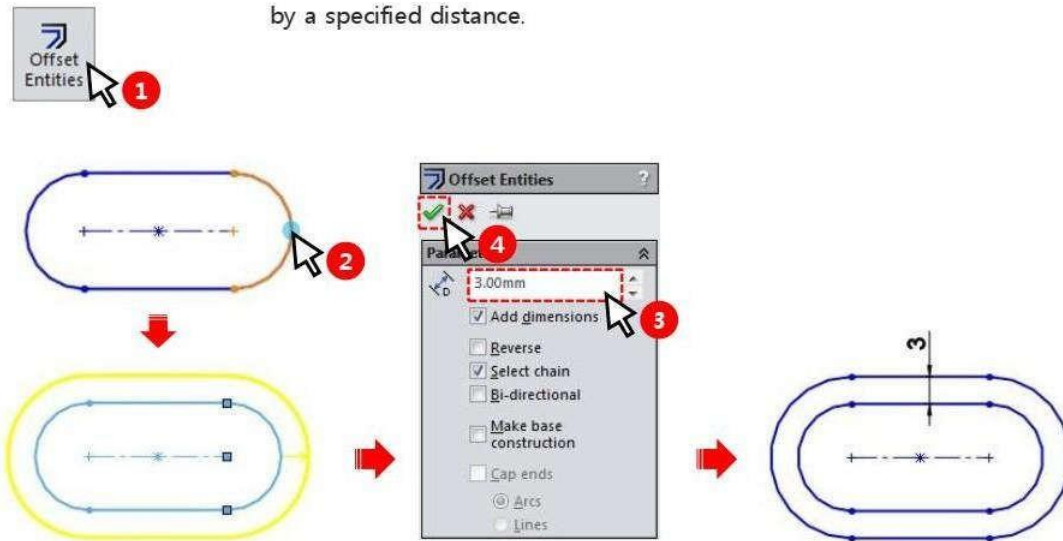
1. In an open sketch, click a model edge, loop, face, curve, external sketch contour, set of edges, or set of curves.

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 .

20.

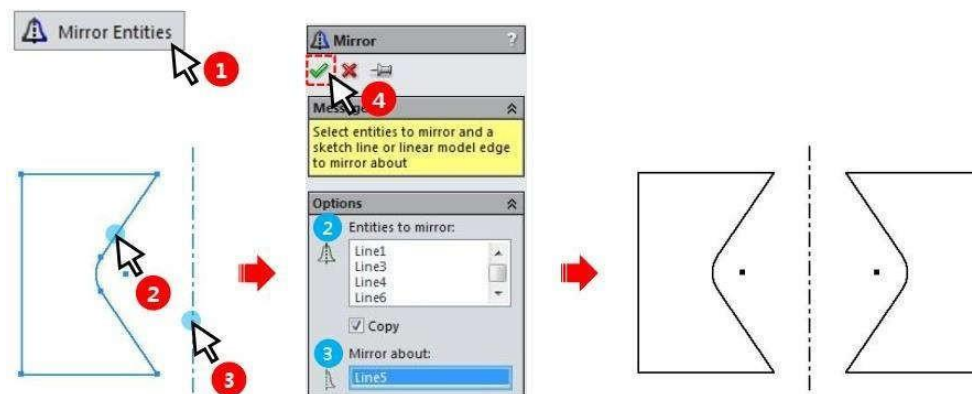
Offset Entities : Offset one or more sketch entities, selected model edges, or model faces by a specified distance.



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.

21.

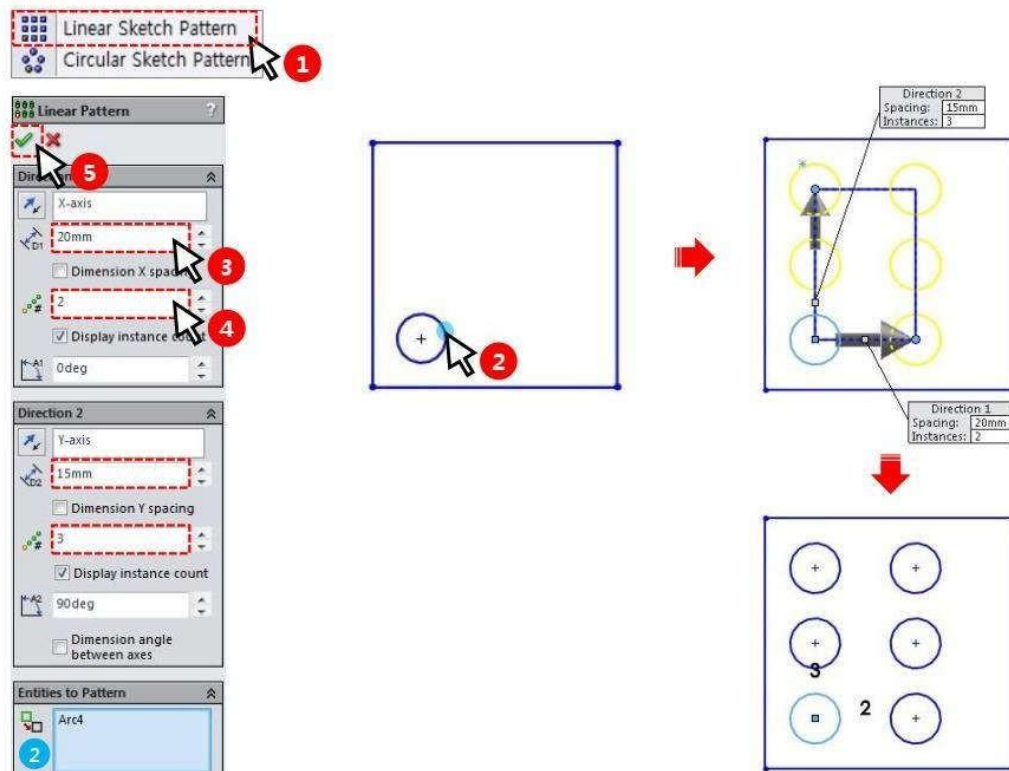
Mirror Entities



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.

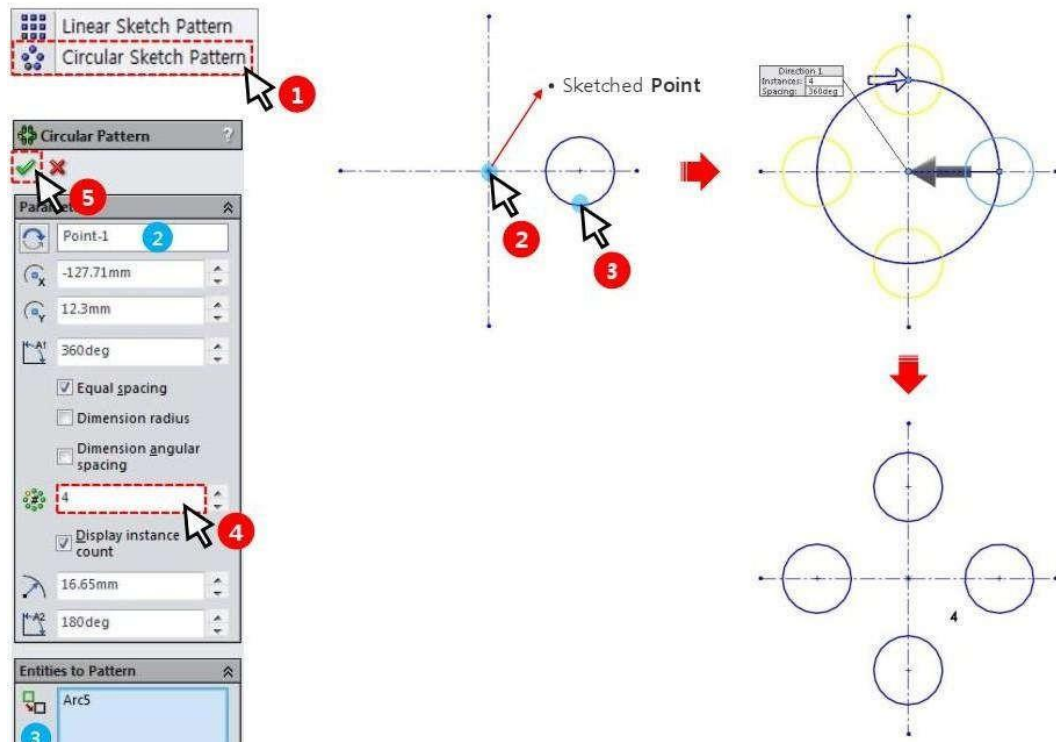
22.

■ Linear Sketch Pattern



23.

■ Circular Sketch Pattern



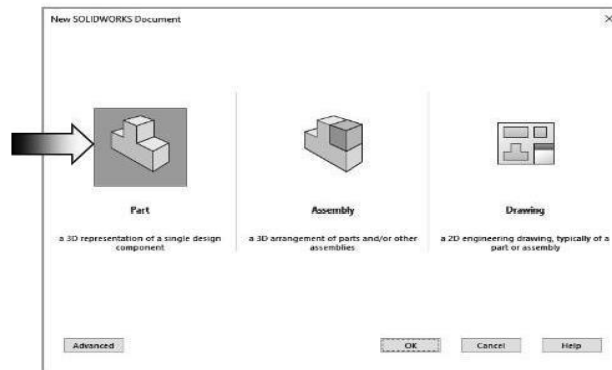


Information Sheet: 2.2-2

Learning Objective: 2D Drawing by Sketch Tool.

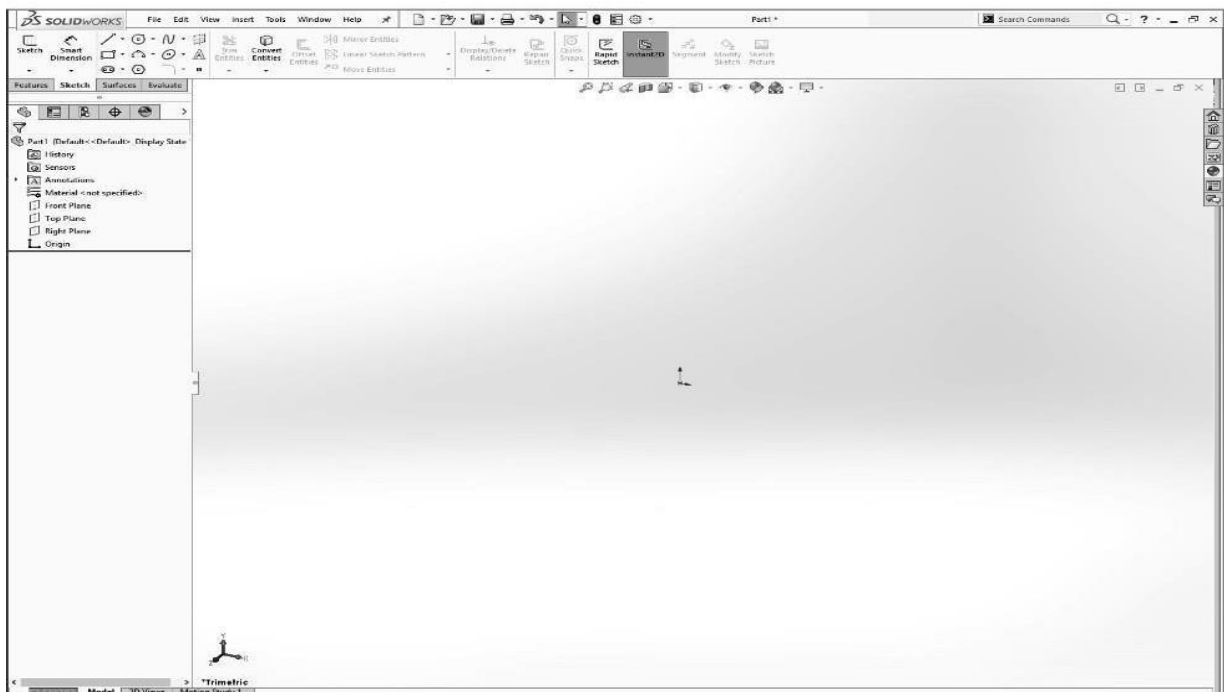
1. Starting a new Part:

From the **File** menu, select **New / Part**, or click the **New** icon.



Select the **Part** template from either the Templates or Tutorial folders.

Click **OK** ; a new part template is opened.



2. Starting a new Sketch:

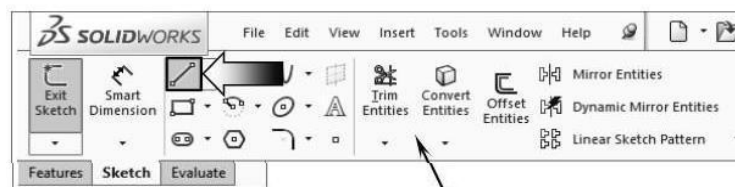
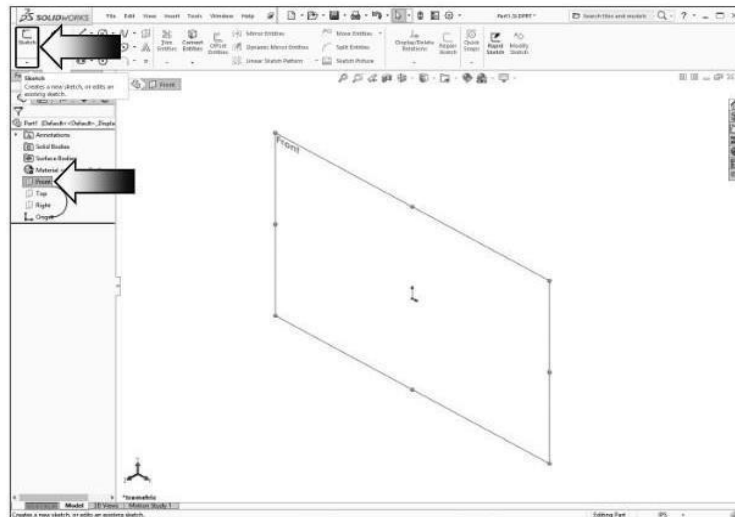
Select the **Front** plane from the Feature-Manager tree and click the **Sketch** drop-down arrow and select the **Sketch** command to start a new sketch.

A sketch is usually created first, relations and dimensions are added after, and then it gets extruded into a 3D feature.

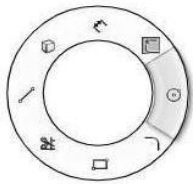
From the Command-Manager toolbar, select the **Line**



command.



CommandManager

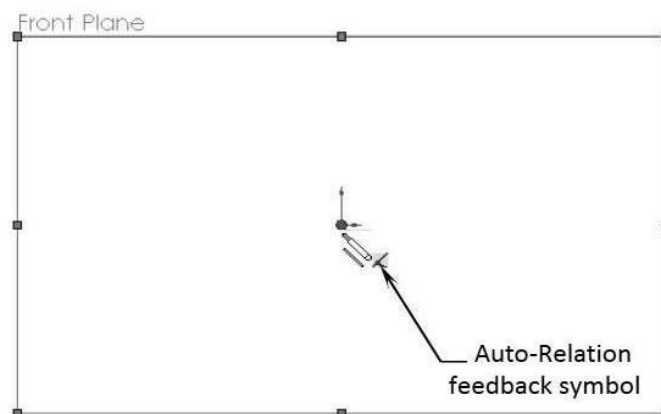


Mouse-Gesture

OPTION:

Right-Drag to display the Mouse-Gesture wheel and select the Line command from it. (See the Introduction section, page XVIII for details on customizing the Mouse Gesture.)

Hover the mouse cursor over the Origin point; a yellow feedback symbol appears to indicate a relation (Coincident) is going to be added automatically to the 1st endpoint of the line. This endpoint will be locked at the zero position.



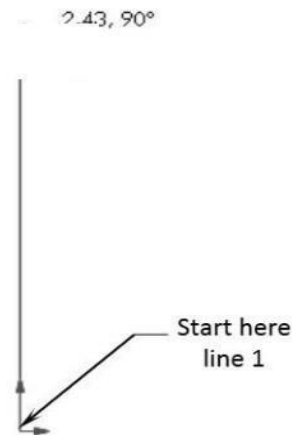
Note: There are 3 different ways to use the line tool: **Single line**, **Multiple (connecting) Lines**, and **Line-to-Arc** (transitioning from a line to a tangent arc).

The following steps are examples to demonstrate how to use all embedded functions of the Line command. We will delete the example and go back to the lesson on page 3-8.

A. To sketch a single line, use the **click + hold + drag** method:

With the Line tool already selected, click on the Origin point to start the line (press and the mouse button), drag the cursor upward, look for the **vertical relation symbol**, and release the mouse button.

Only 1 line is drawn; the mouse cursor is not connecting to the last line.



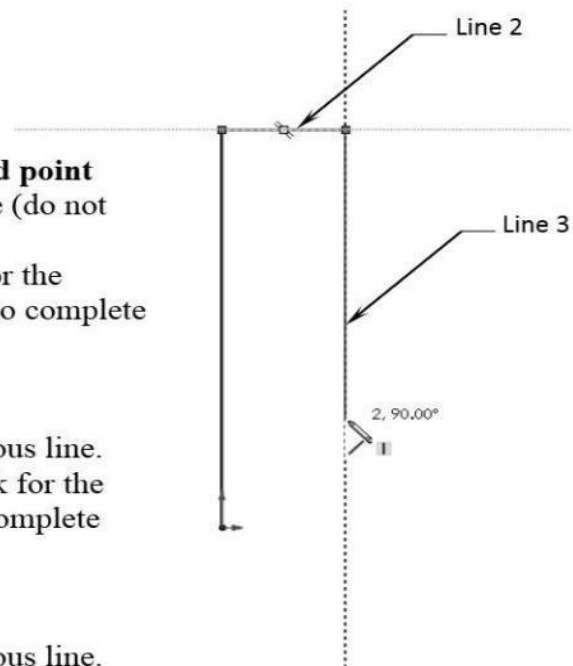
B. To sketch multiple lines, use the **click + release + click** method.

Position the mouse cursor at the **top end point** of the first line and click to start the line (do not hold the mouse button this time).

Move the cursor to the right and look for the **Horizontal relation symbol** and click to complete the line.

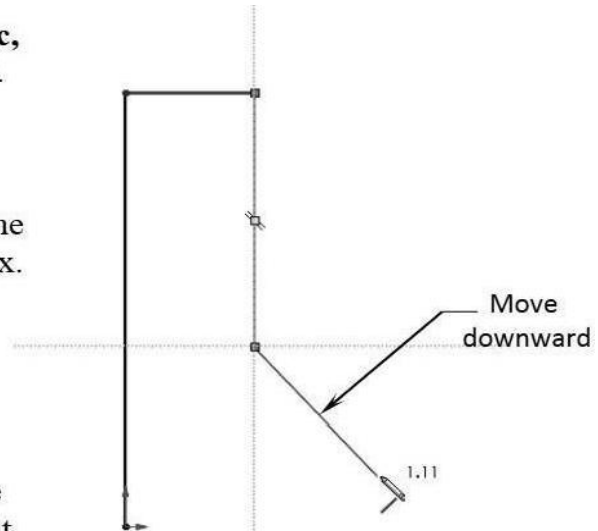
The next line is connecting to the previous line. Move the mouse cursor downward, look for the **vertical relation symbol** and click to complete the line.

The next line is connecting to the previous line.



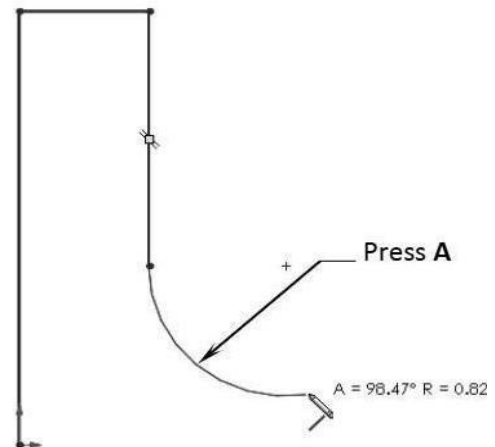
C. To change from a **Line** to a **Tangent Arc**, use the **click + release + press A** method.

Start a new line at the bottom of the 3rd line and drag the mouse cursor outward approx. as shown.



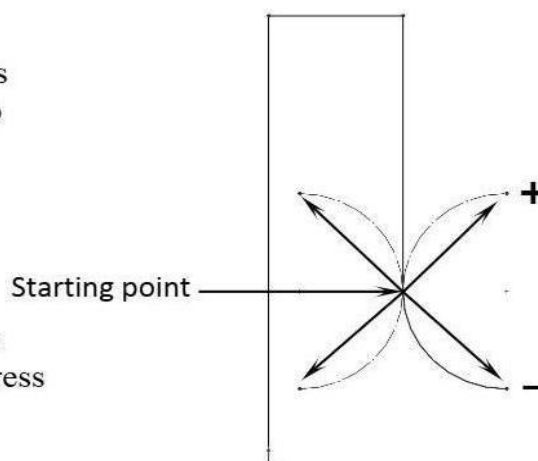
D. Press once on the **A** key and then move the cursor slightly to the left or to the right. The line is changed into a **Tangent Arc**.

Move the cursor back and forth, up and down to see how the Tangent Arc maintains its tangent relation with the line that it is connecting with.



Based on the starting point, the arc changes its direction from negative to positive when the cursor moves past its zero point.

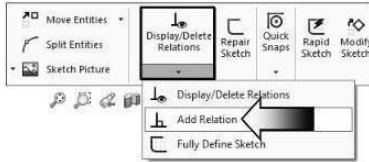
Move the mouse cursor towards the lower right side and click to complete the arc.



Delete all sketch entities before moving on to step number 4 (press Control+A then push Delete).

3. Adding Geometric Relations*:

Click **Add Relation**  under Display/Delete Relations - OR - select **Tools / Relations / Add**.

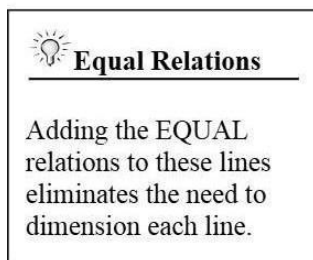


Select the **4 lines** shown below.

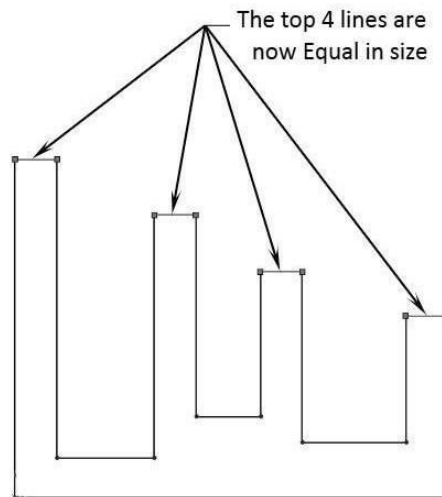
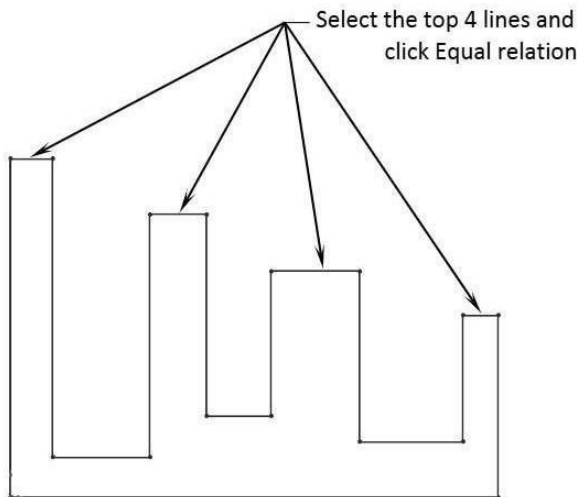
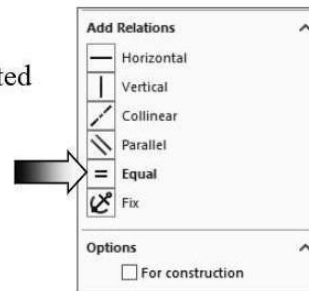
Click **Equal** from the Add Geometric Relation dialog box. This relation makes the length of the four selected lines equal.

* Geometric relations are one of the most powerful features in SOLIDWORKS. They are used in the sketch level to control the behaviors of the sketch entities when they are moved or rotated and to keep the associations between one another.

(When applying geometric relations between entities, one of them should be a 2D entity and the other can either be a 2D sketch entity, a model edge, a plane, an axis, or a curve, etc.)



Geometric relations can be created manually or automatically. The next few steps in this chapter will demonstrate how geometric relations are added manually.



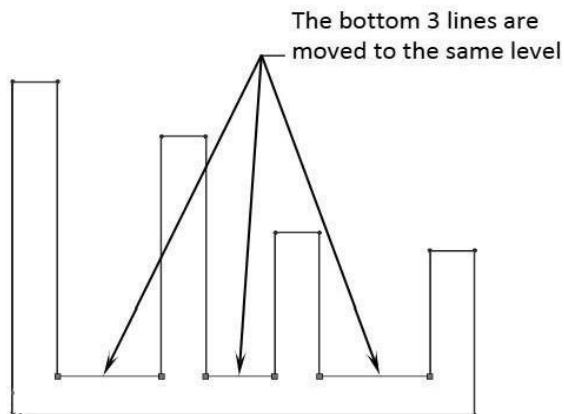
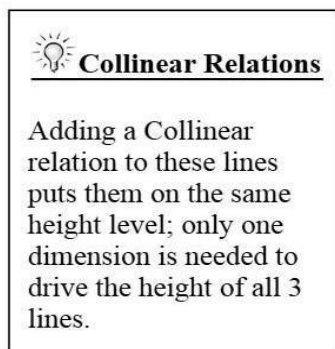
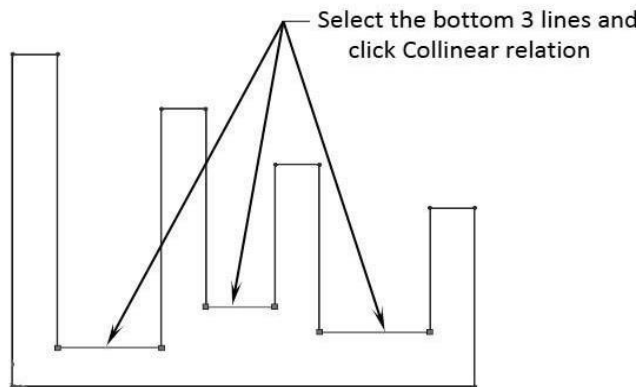
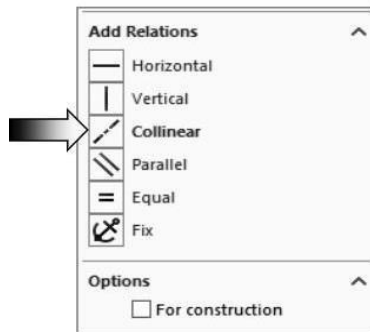
4. Adding a Collinear relation**:

Select the **Add Relation**  command again.

Select the 3 lines shown below.

Click **Collinear** from the Add Geometric Relations dialog box.

Click **OK**.

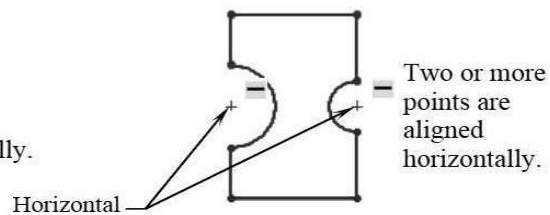
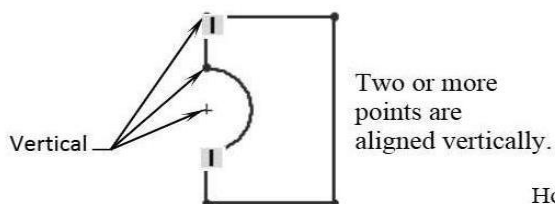
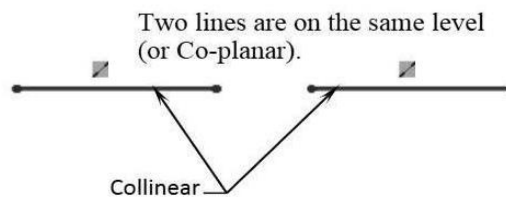
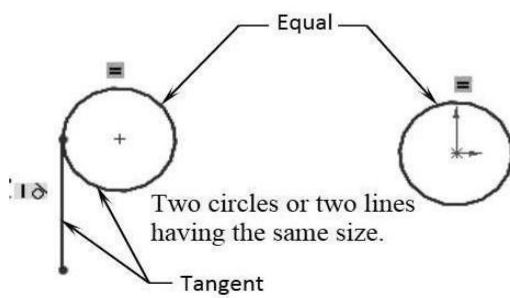
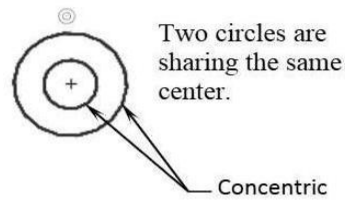
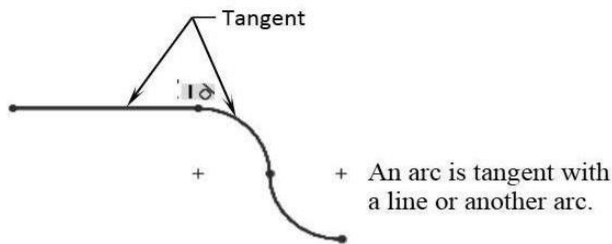
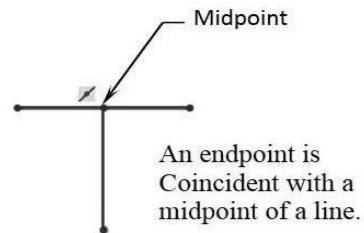
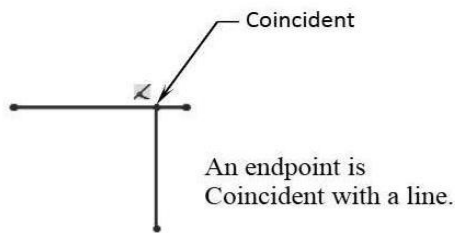


** Collinear relations can be used to constrain the geometry as follows:

Collinear between a line and another line(s) (2D and 2D).

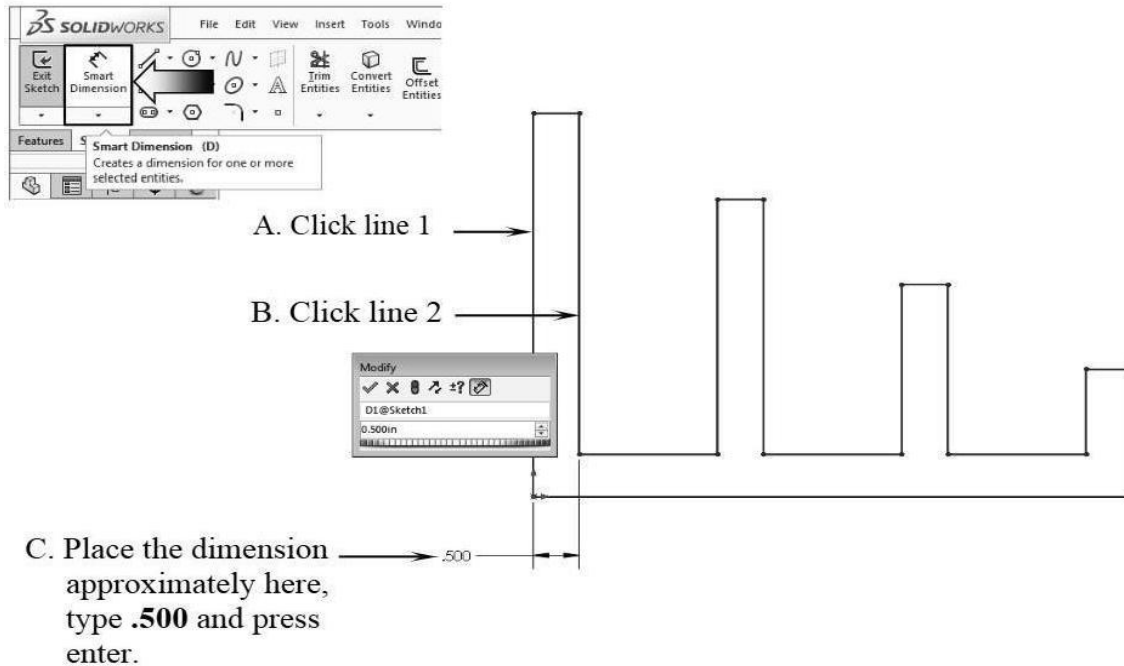
Collinear between a line(s) to a linear edge of a model (2D and 3D).

Geometric Relations Examples



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).

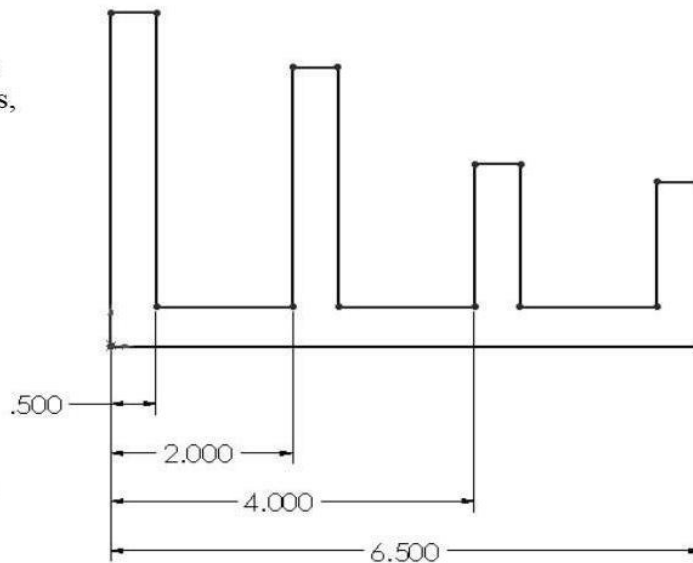


The Inch-Units is filled in automatically because it has been set previously to Inches, 3 decimal places.

Continue adding the horizontal dimensions as shown here.

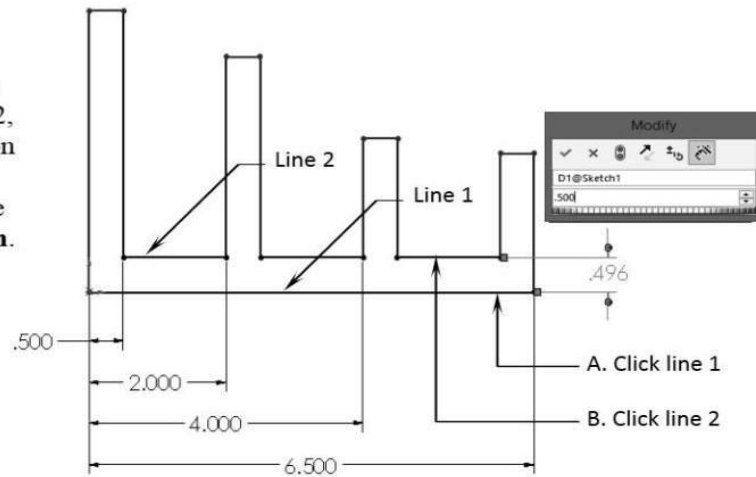
NOTE:

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

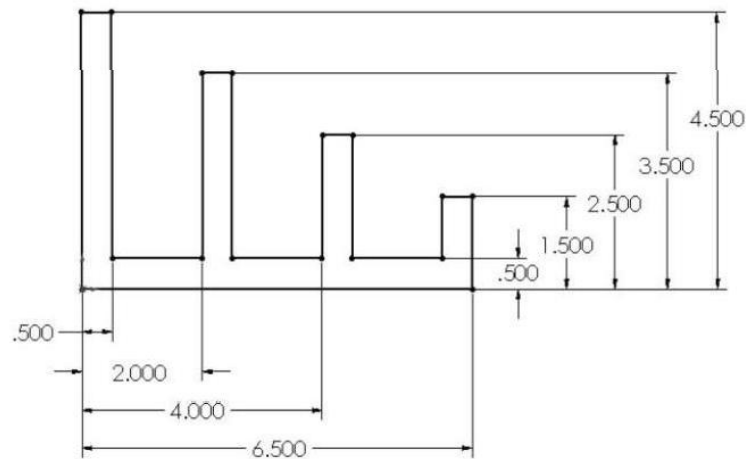


6. Adding the Vertical dimensions:

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.



F

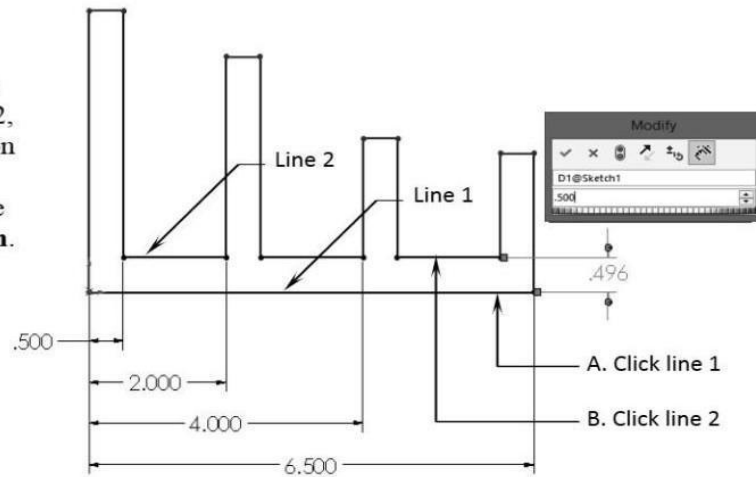
The Status of a Sketch:

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

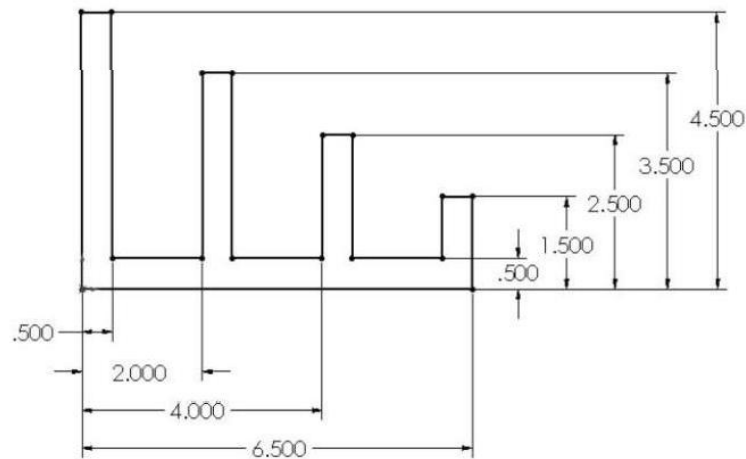
Fully Defined	=	Black	Fully Defined
Under Defined	=	Blue	Under Defined
Over Defined	=	Red	Over Defined

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.



F

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



Self-Check: 2.2-2

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?



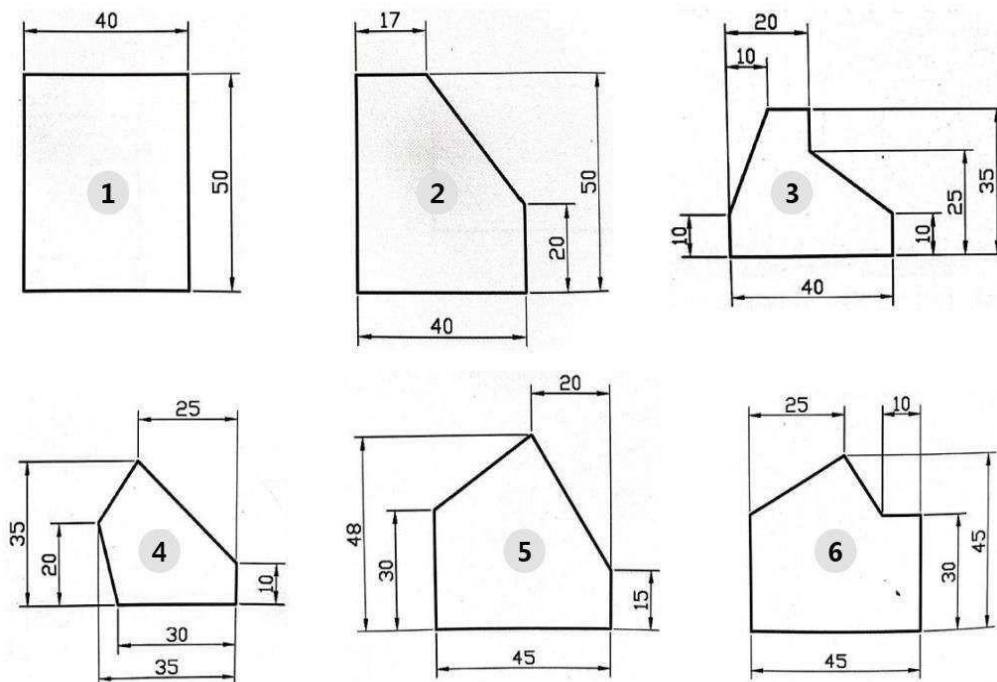
Answer Script: 2.2-2

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.

Job Sheet: 2.2 (a)

Job	Create 2D drawings of samples	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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.	

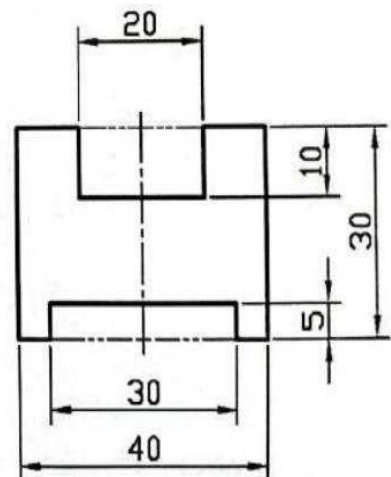
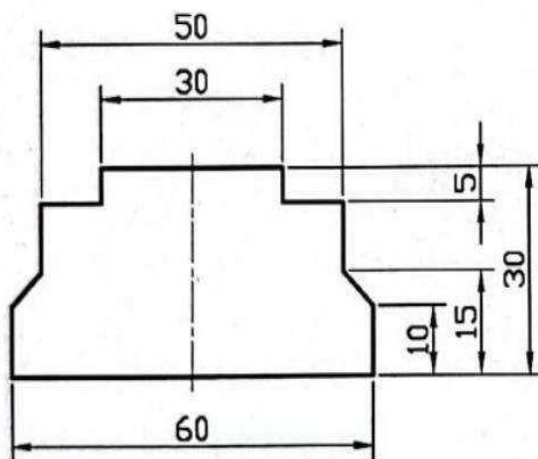
• Sample



Job Sheet: 2.2 (b)

Job	Create 2D drawings of samples	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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.	

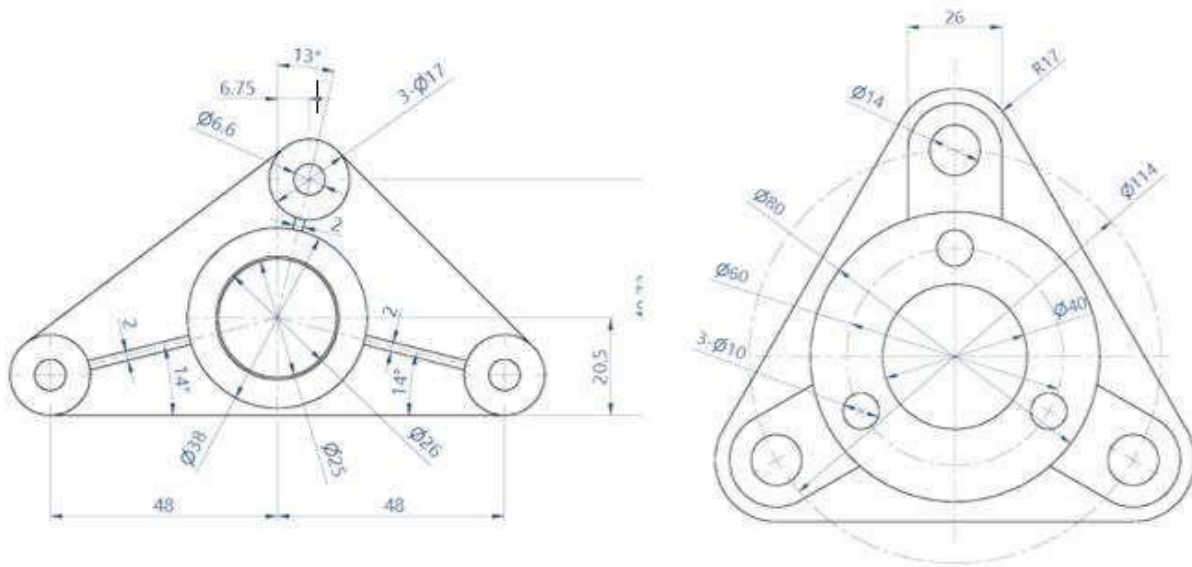
• **Sample:**



Job Sheet: 2.2 (c)

Job	Create 2D drawings of samples	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer set	1 Pc
	CAD software (SolidWorks)	1 Pc
Procedure	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



Learning Outcome 2.3: Develop 3D Model



Content:

- Conceptualization and planning
- Draw the base sketch
- Add dimensions and relations
- Create the first 3D



Assessment Criteria

1. Features tools are identified and selected for 3D model.
2. Be able to create 3D model.
3. Direct editing tools are identified and selected for 3D model.
4. Be able to create 3D model to 2D Sheet preparation.



Necessary material and Instrument

Trainees must provide the following equipment and supplies:

1. A computer set
2. SolidWorks software
3. Drawing instructions



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Develop 3D Model	• Information Sheet: 2.3-1 to 2.3-2 • Self-Check: 2.3-1 • Answer Script: 2.3-1 • Job Sheet: 2.3 (a) to 2.3 (j)



Information Sheet: 2.3-1

Learning Objective: Usage of Feature Tool for 3D Modeling.

3D Model: “3D” which known as “three-dimensional Model” basically 3D displays a picture in a form that appears to be physically present with a structure of design is necessary that it allows pictures which appears easy to the human eye. So, this can be also explained in a sense of items, which is display in a form that allow for different dimensions to be represented. Therefore, height, width and depth are also included in 3D dimension.

There are 3 methods for creating 3D solid model: -

- 1. Parametric Modelling:** a type of solid modelling that uses 2D sketches to create features and objects that are modifiable with a feature history tree. Features are located and manipulated using geometric and functional relationships with other sketches, features and objects.
- 2. Direct Modelling:** like parametric modelling, direct modelling uses sketches to create 3D features based on relationships with existing geometry. However, there is no history tree, and the original sketch is absorbed into the feature and future edits are made on the resulting geometry.
- 3. Surface Modelling:** Another type of modelling that relies on surfaces to generate geometry. Surface modelling can be both parametric (based on geometric and functional relationships) or freeform. Freeform surface modelling is used to manipulate the surface of the model similar to how someone would model with clay. Check out entries on Sculptress and Mesh Mixer to learn more about freeform surface modelling.

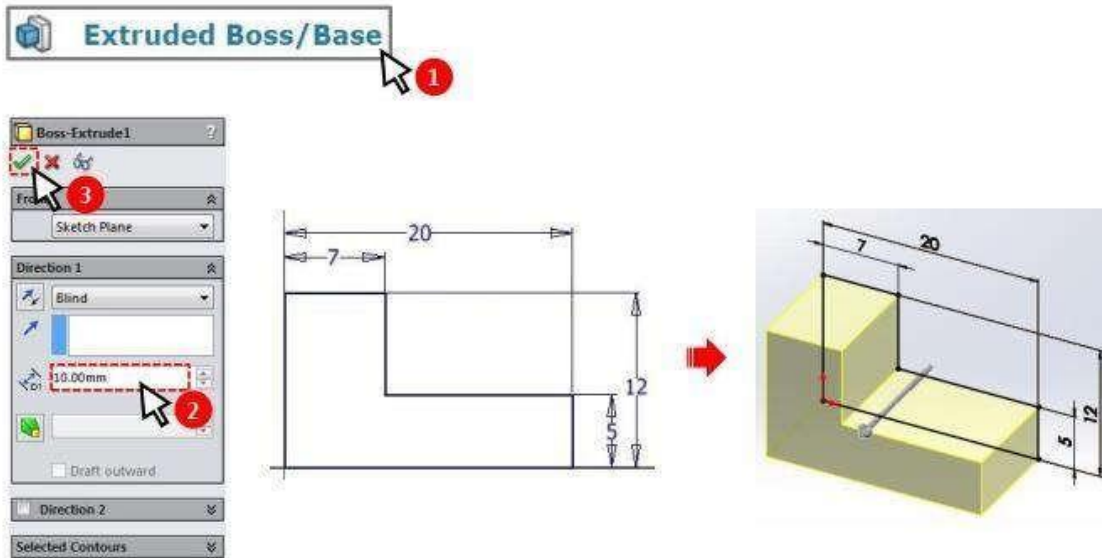
■ Features Toolbar

- 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		

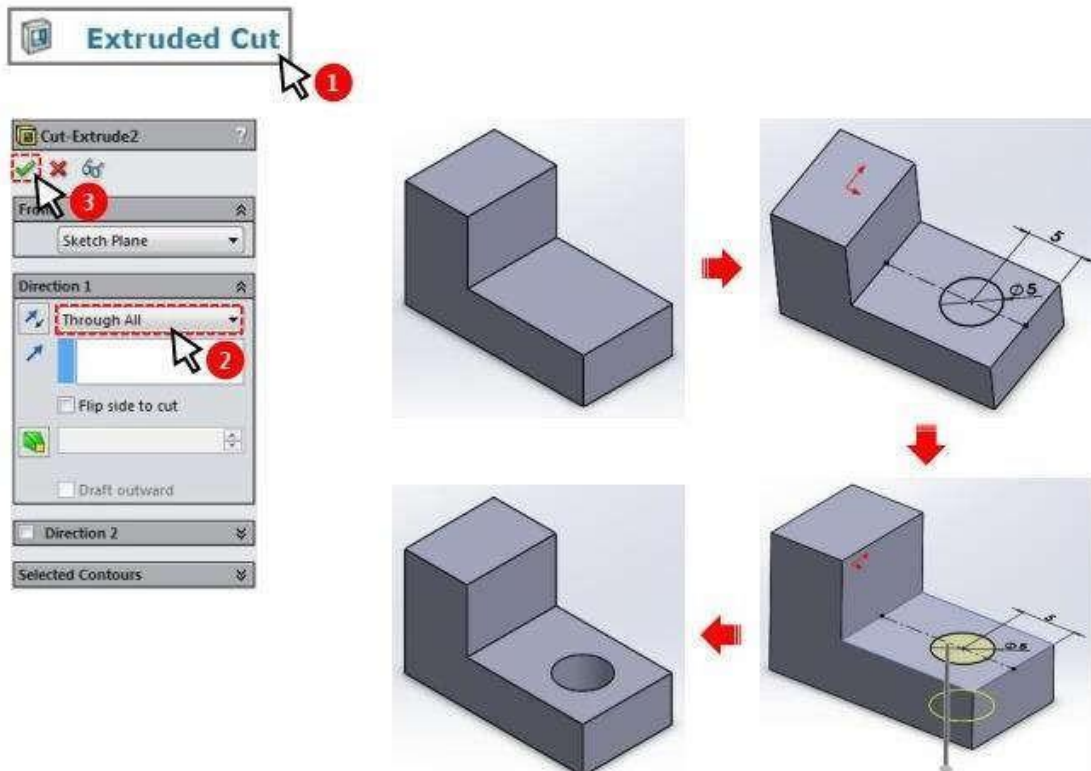
1.

■ Extruded Boss/Base



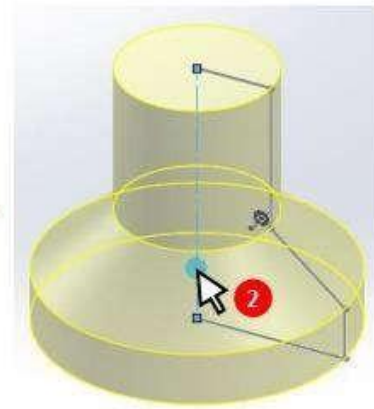
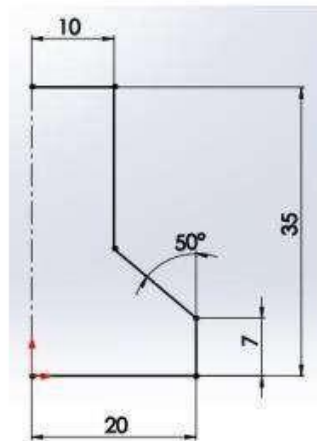
1. On the ribbon, click **Extruded Boss/Base**.
2. The profile to be extrude is automatically selected. (when there is only one profile)
3. Determine the distance and direction and complete the creation of the solid feature.

2.



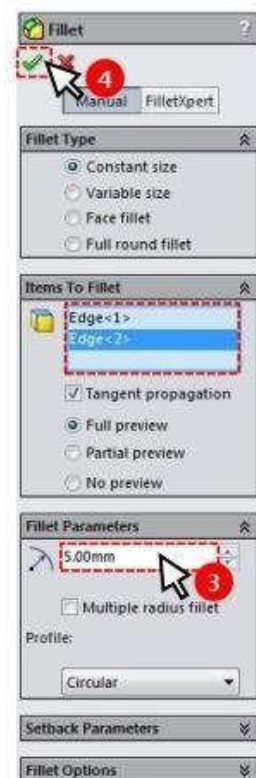
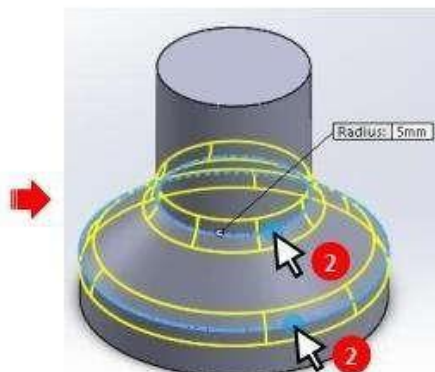
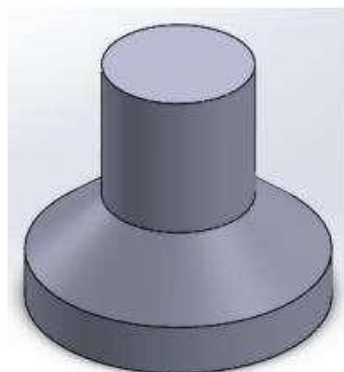
3.

■ Revolved Boss/Base



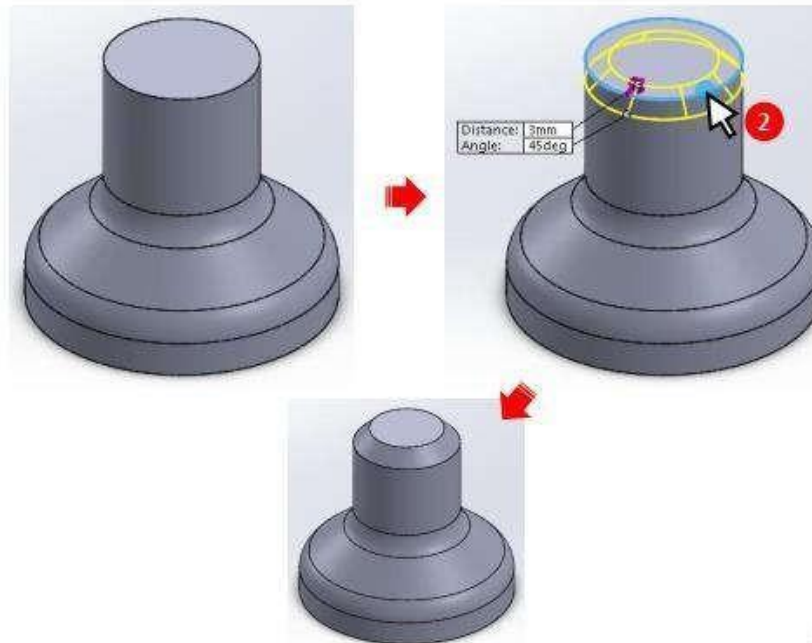
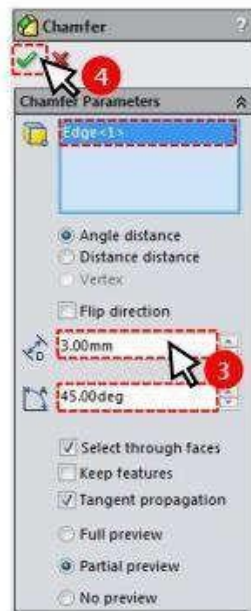
4.

■ Fillet



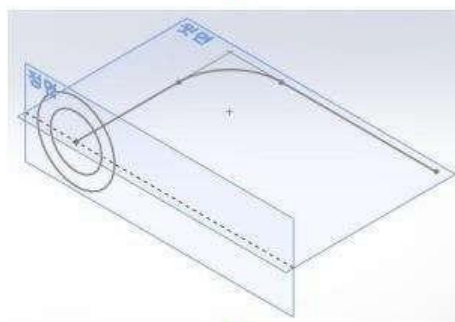
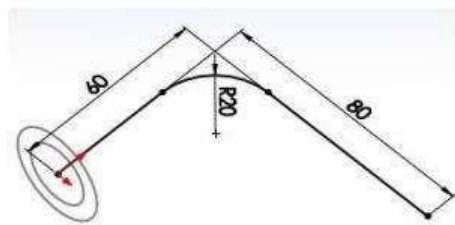
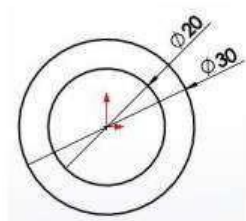
5.

■ Chamfer



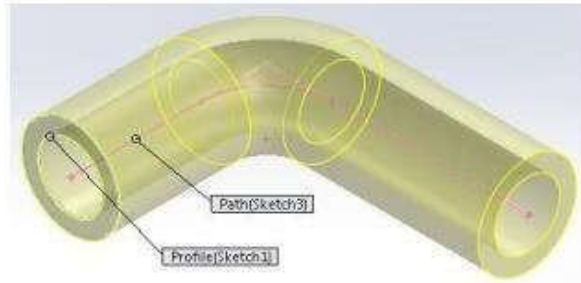
6.

■ Swept Boss/Base



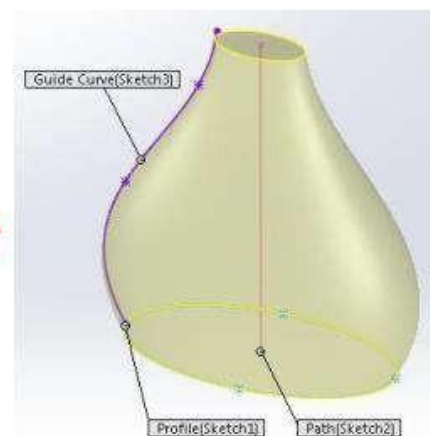
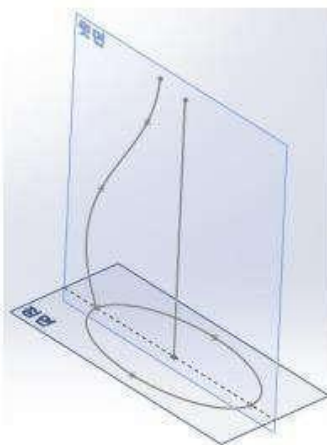
7.

■ Swept Boss/Base



8.

■ Lofted Boss/Base

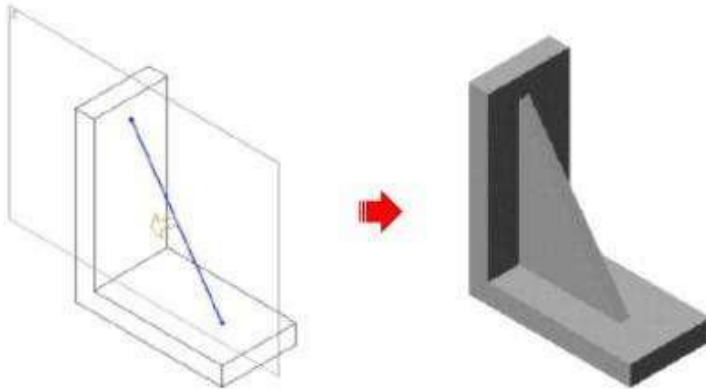


9.

■ 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.



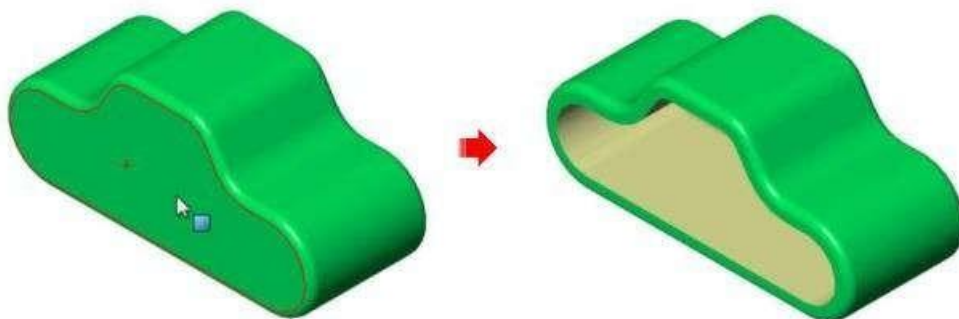
2. Click **Rib**  (Features toolbar) or **Insert > Features > Rib**.
3. Set the PropertyManager options.
4. Click **OK** .

10.

■ 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 .





Self-Check: 2.3-1

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?



Answer Script: 2.3-1

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.

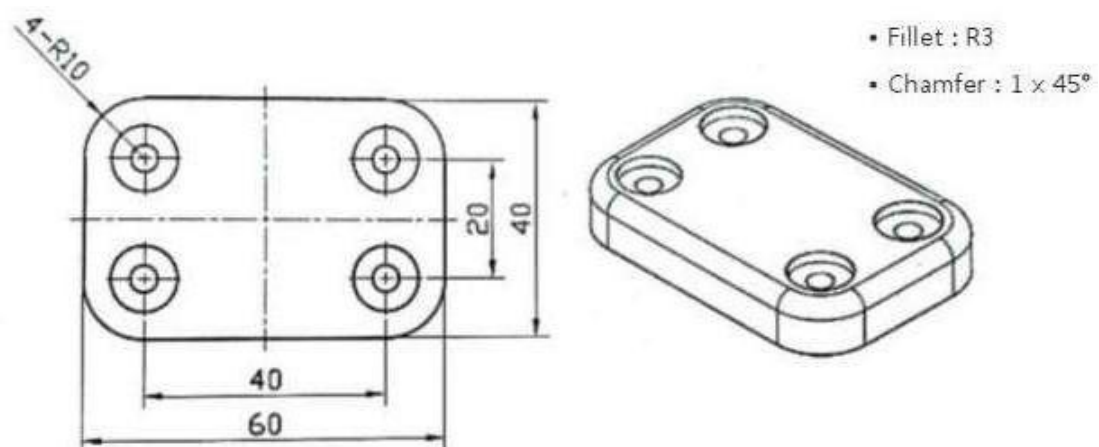
5. The primary purpose of the **Shell** feature is to hollow out a part, leaving a thin, uniform wall thickness.

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.

Job Sheet: 2.3 (a)

Job	Create 3D drawings of sample	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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 Tools for 3D drawings.	
	Follow trainers/instructors.	
	Save drawings in a folder.	

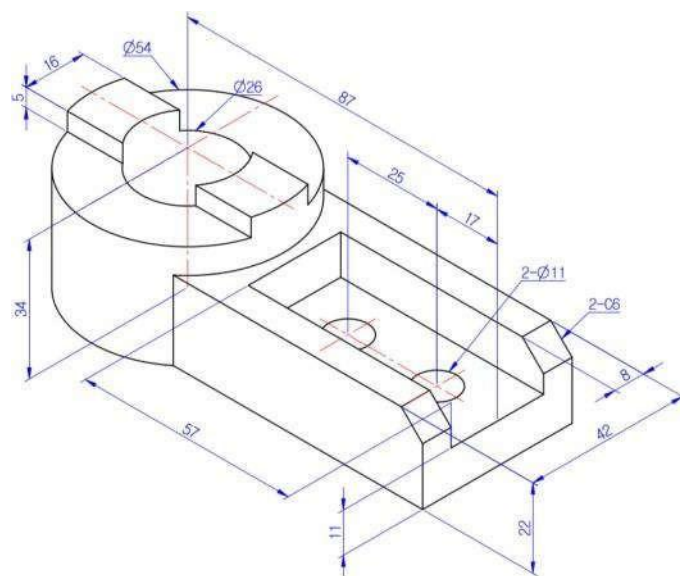
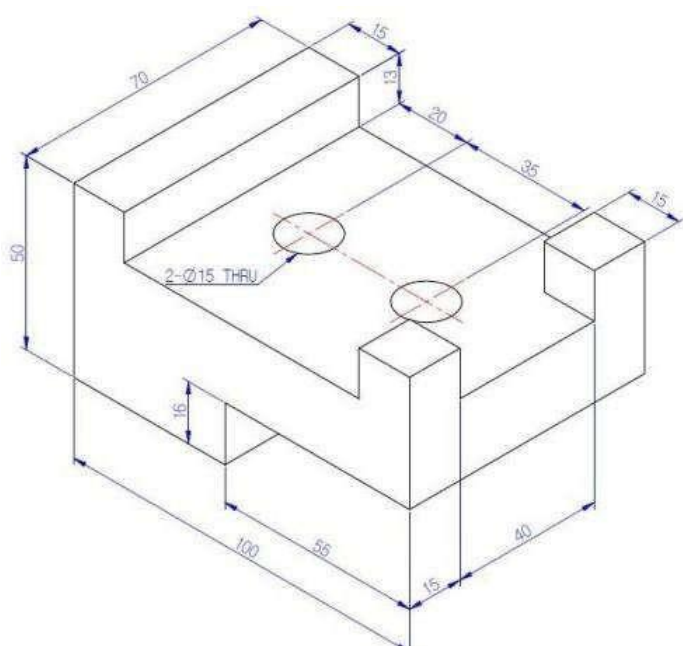
• Sample:



Job Sheet: 2.3 (b)

Job	Create 3D drawings of samples	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Set
	Computer set	1 Pc
	CAD software (SolidWorks)	1 Pc
Procedure	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 Tools for 3D drawings.	
	Follow trainers/instructors.	
	Save drawings in a folder.	

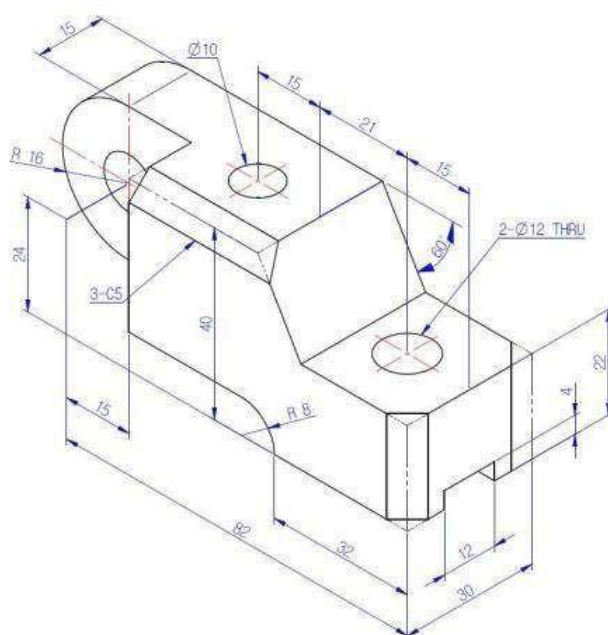
- **Samples:**



Job Sheet: 2.3 (c)

Job	Create 3D drawings of sample	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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 Tools for 3D drawings.	
	Follow trainers/instructors.	
	Save drawings in a folder.	

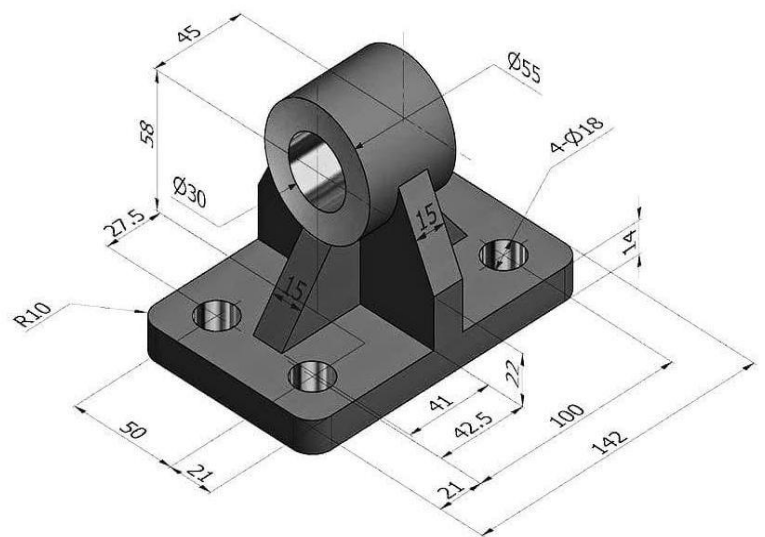
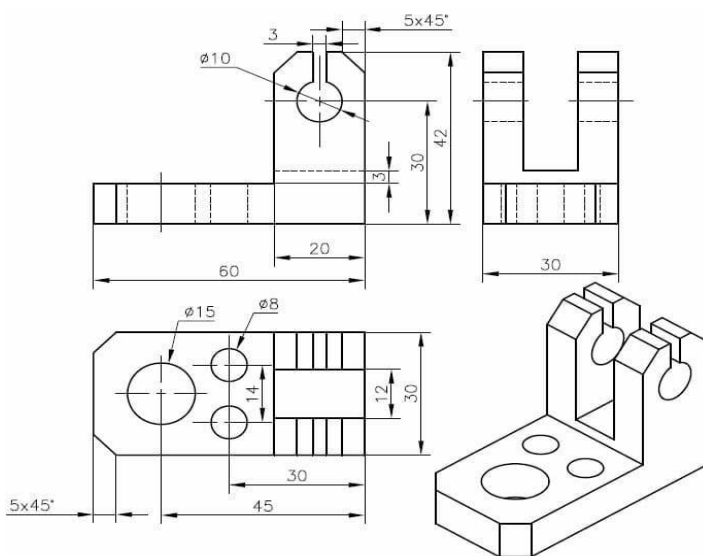
- **Samples:**



Job Sheet: 2.3 (d)

Job	Create 3D drawings of samples	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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 Tools for 3D drawings.	
	Follow trainers/instructors.	
	Save drawings in a folder.	

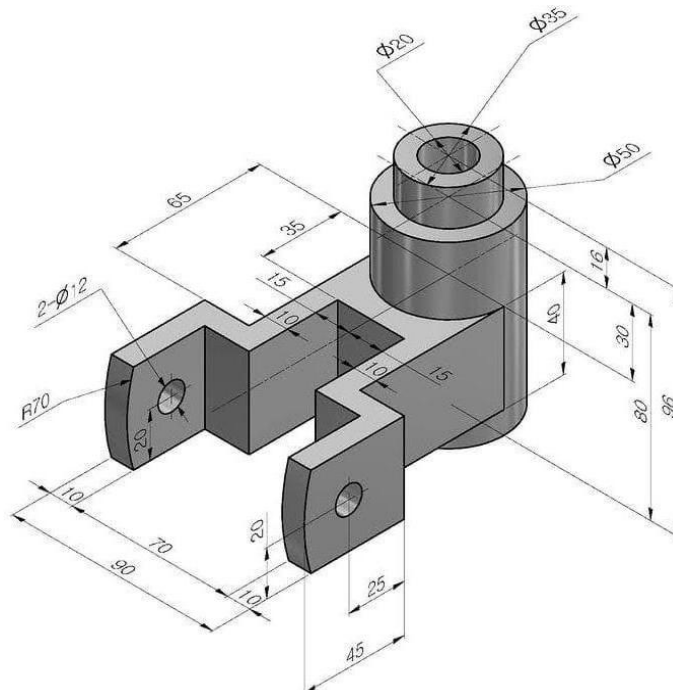
• Samples:



Job Sheet: 2.3 (e)

Job	Create 3D drawings of sample	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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 Tools for 3D drawings.	
	Follow trainers/instructors.	
	Save drawings in a folder.	

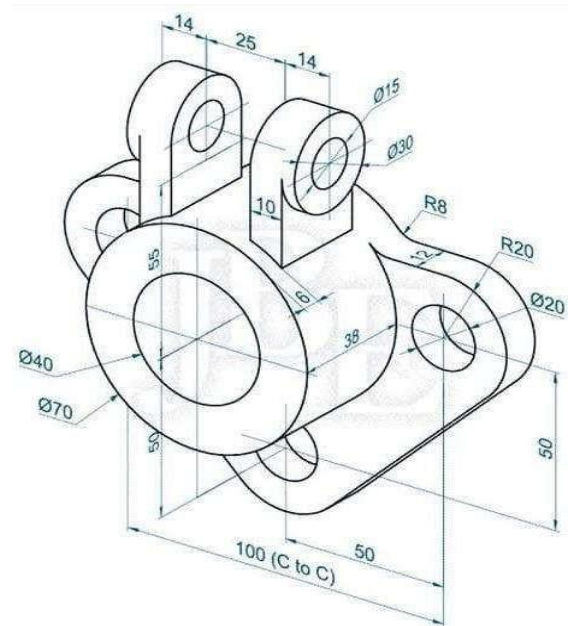
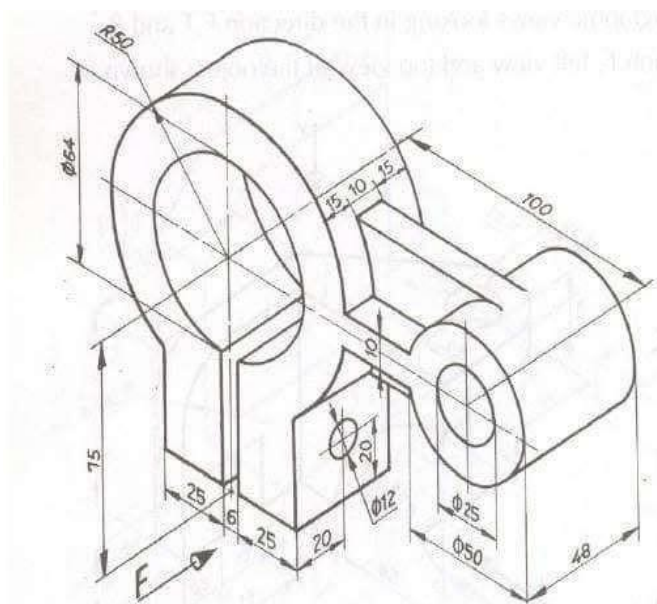
- **Sample:**



Job Sheet: 2.3 (f)

Job	Create 3D drawings of samples	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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 Tools for 3D drawings.	
	Follow trainers/instructors.	
	Save drawings in a folder.	

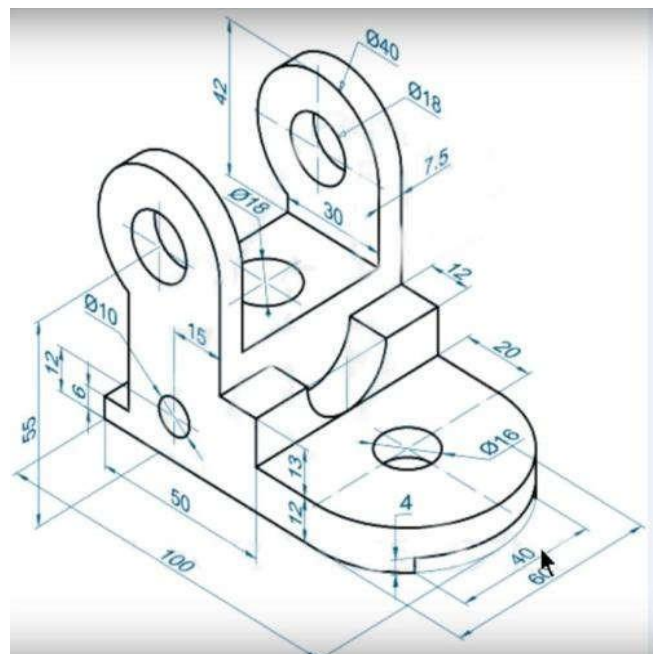
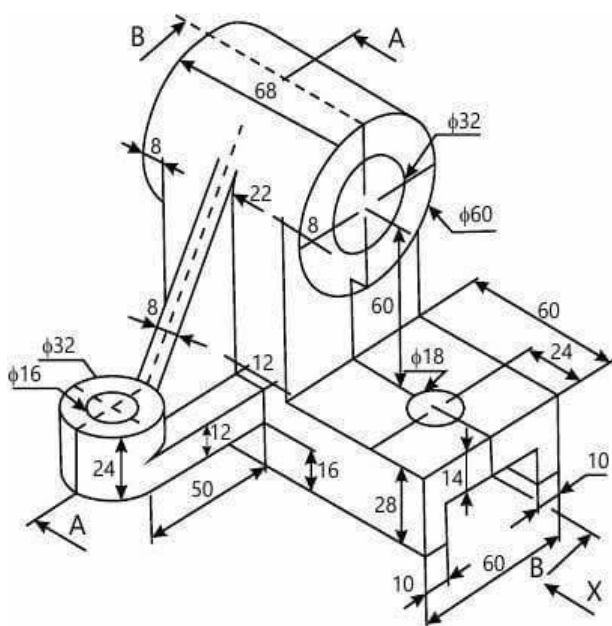
- Samples:**



Job Sheet: 2.3 (g)

Job	Create 3D drawings of samples	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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 Tools for 3D drawings.	
	Follow trainers/instructors.	
	Save drawings in a folder.	

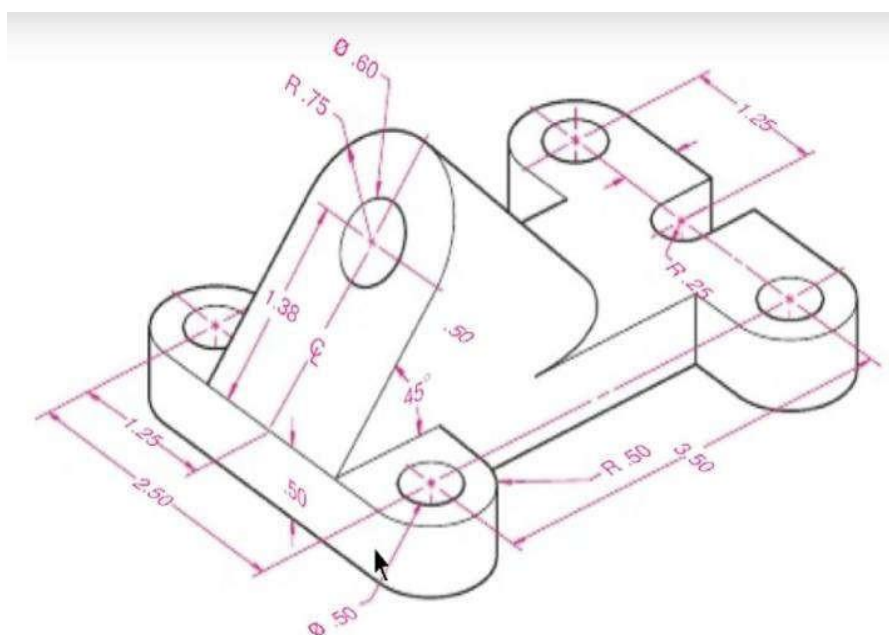
• Samples:



Job Sheet: 2.3 (h)

Job	Create 3D drawings of sample	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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 Tools for 3D drawings.	
	Follow trainers/instructors.	
	Save drawings in a folder.	

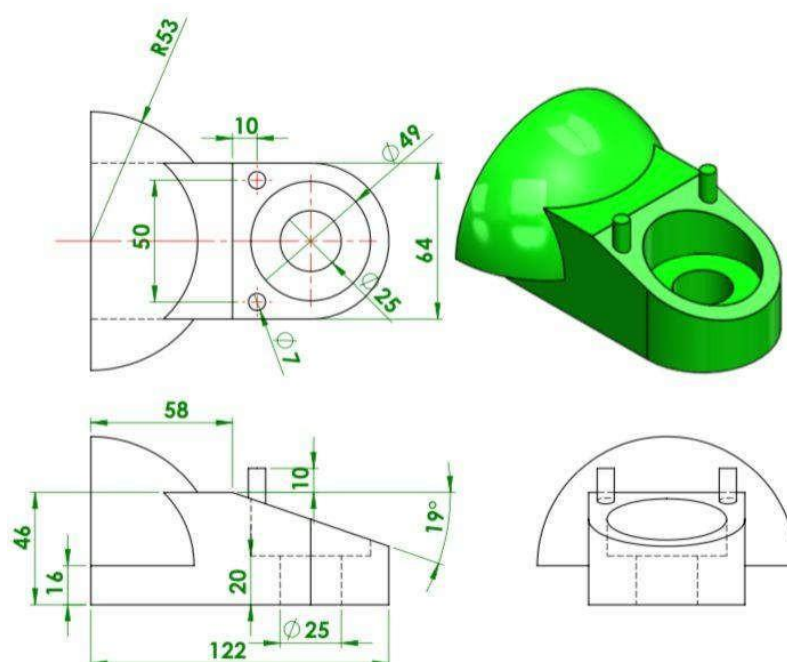
- **Sample:**



Job Sheet: 2.3 (i)

Job	Create 3D drawings of sample	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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 Tools for 3D drawings.	
	Follow trainers/instructors.	
	Save drawings in a folder.	

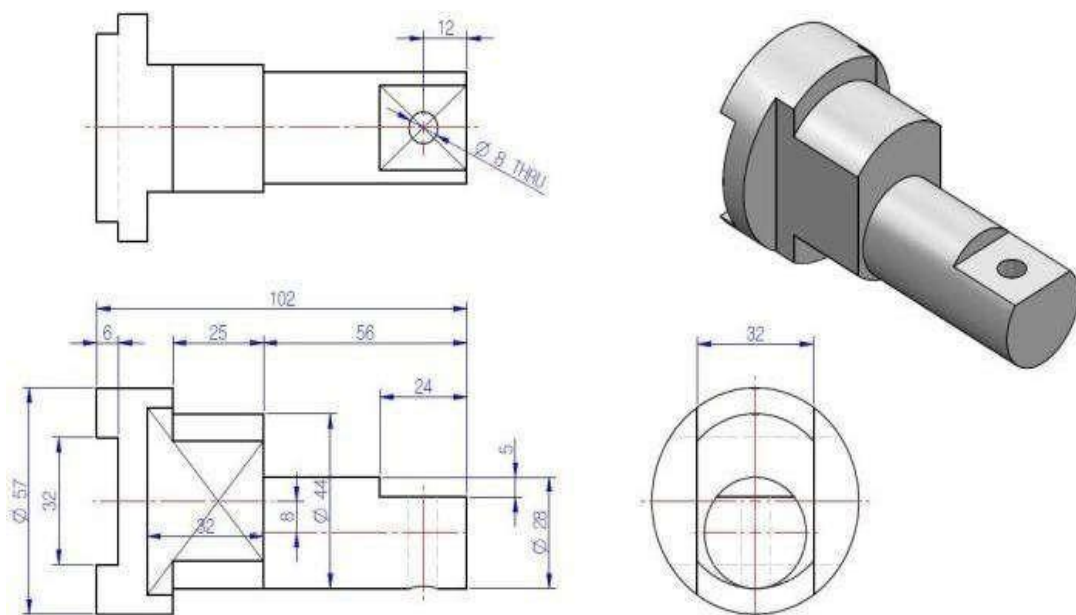
- **Sample:**



Job Sheet: 2.3 (j)

Job	Create 3D drawings of sample	
Materials/Resources	Pencil	1 Pc
	Drawings and/or sketches	1 Pc
	Scale	1 Pc
	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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 Tools for 3D drawings.	
	Follow trainers/instructors.	
	Save drawings in a folder.	

- Sample:**

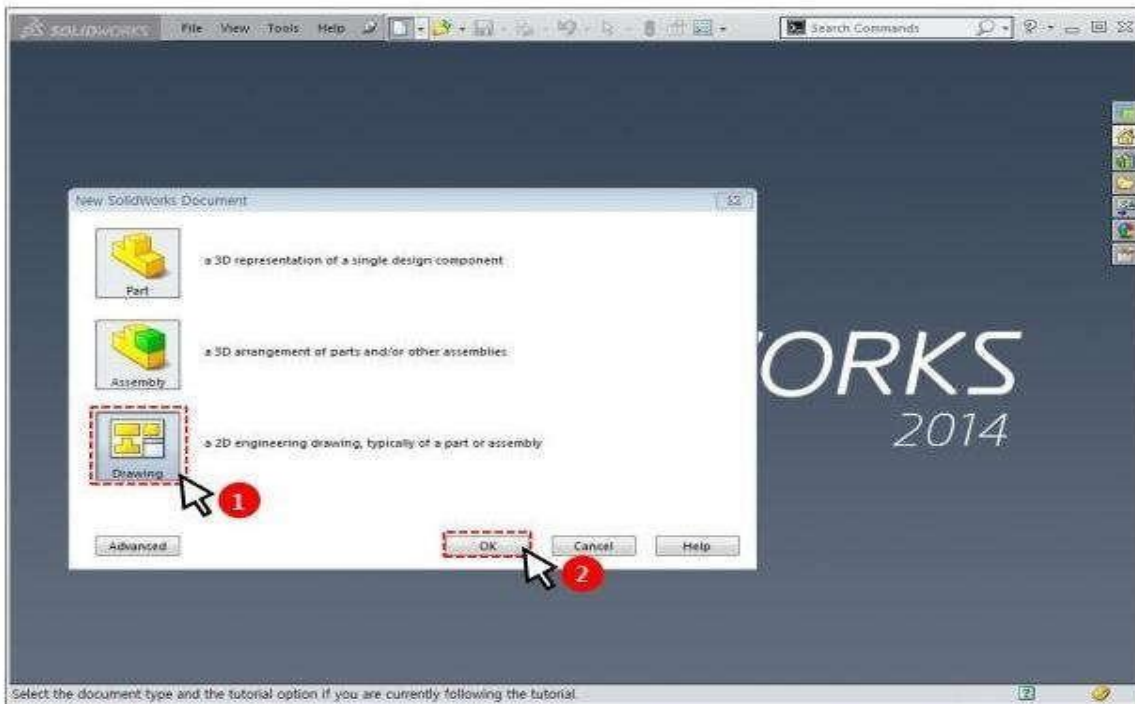




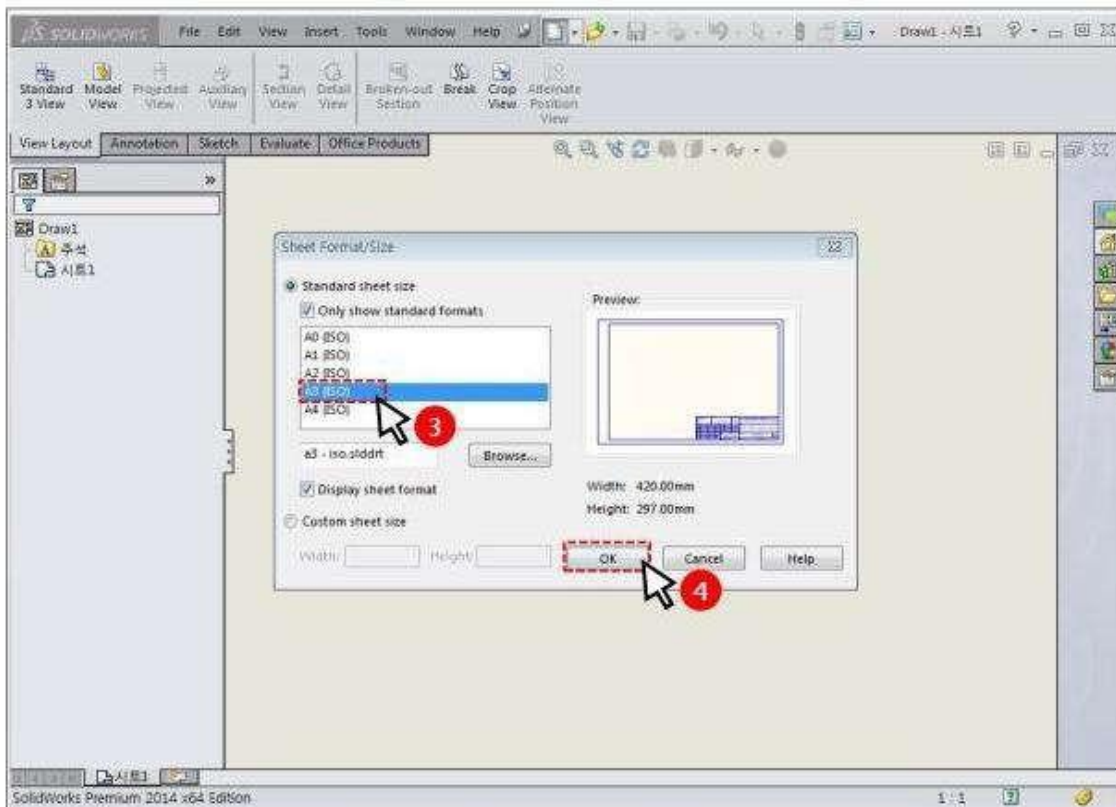
Information Sheet: 2.3-2

Learning Objective: Preparation of 2D Drawing Sheet from of 3D Part.

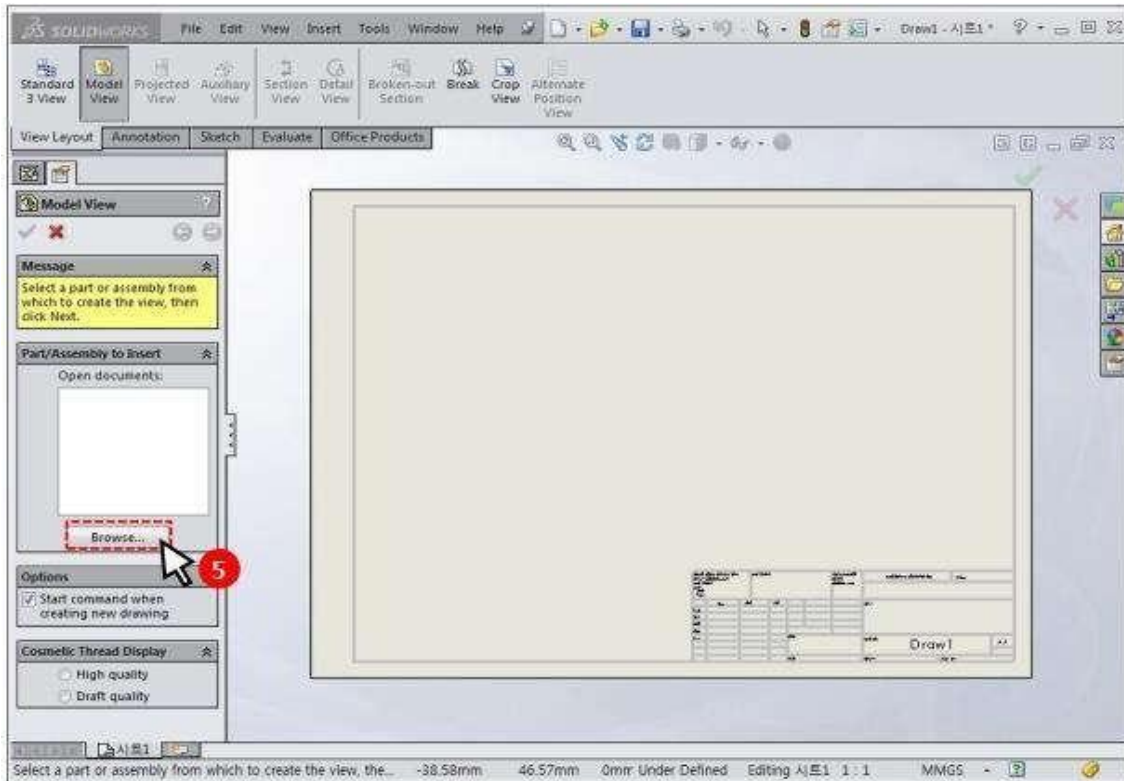
1. Select Drawing and press on Ok.



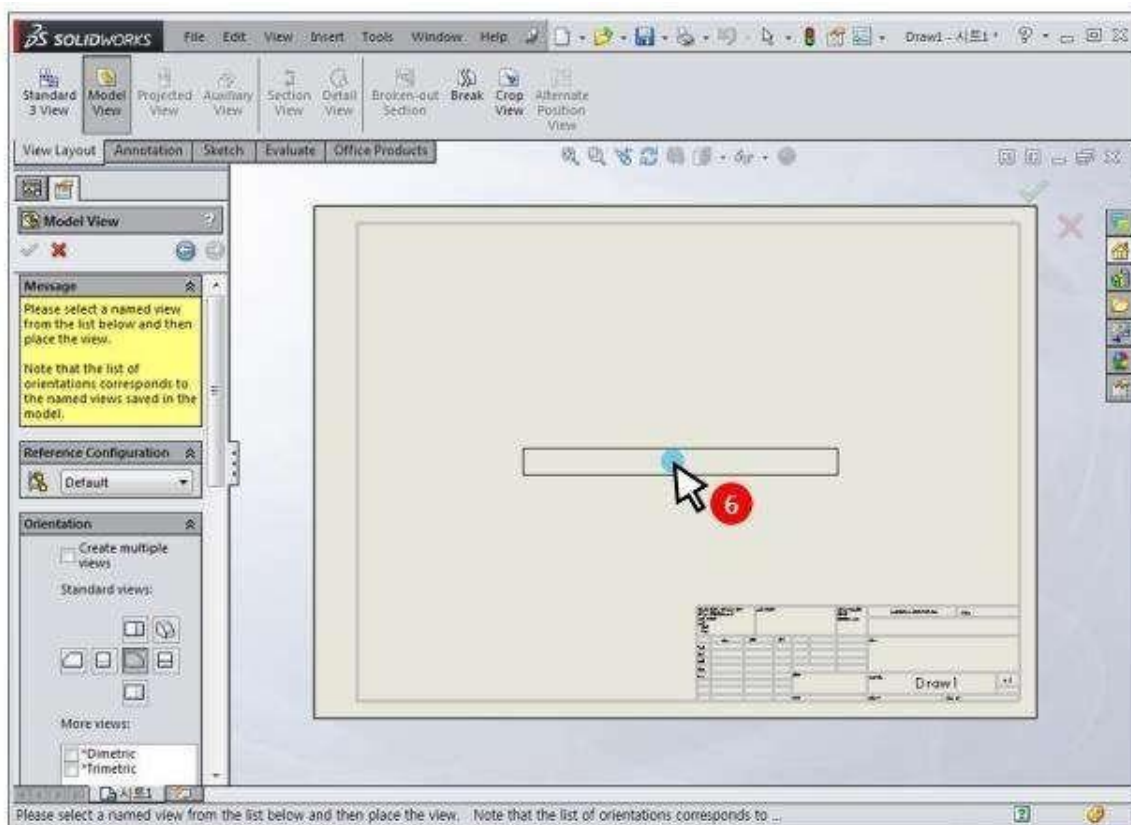
2. Select page size.



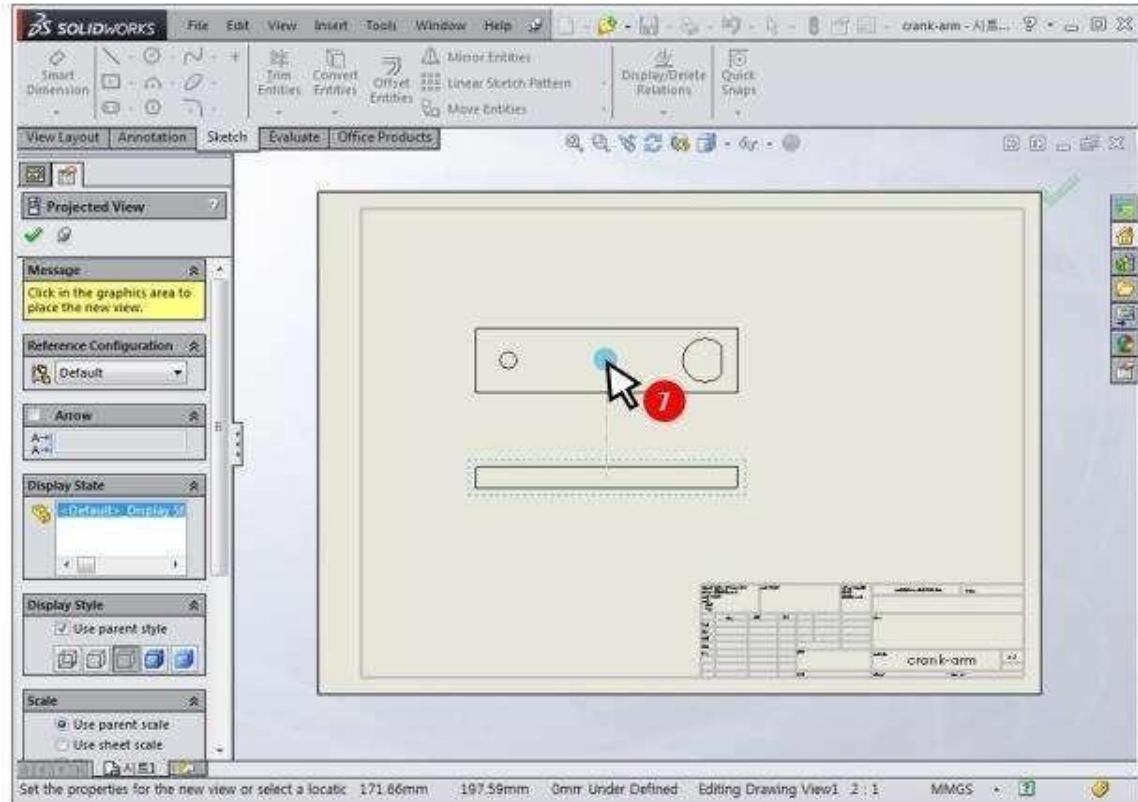
3. Browse file.



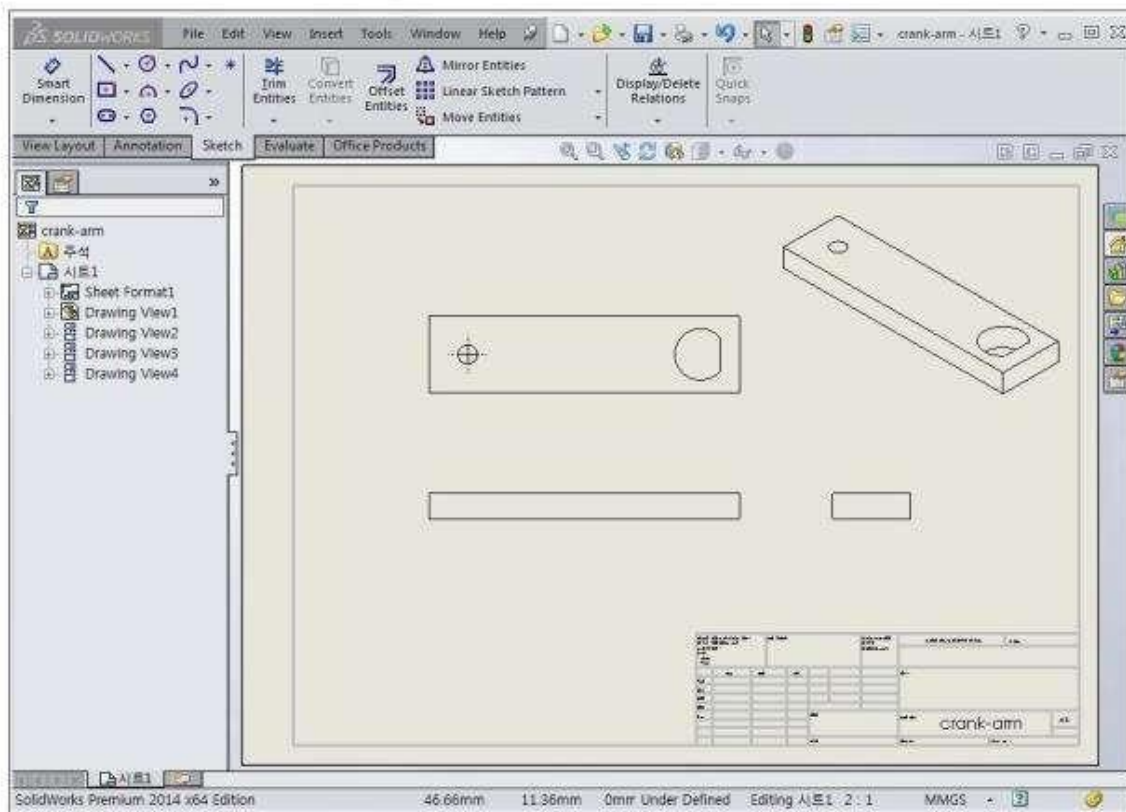
4. Import part file and make 2D orientation.



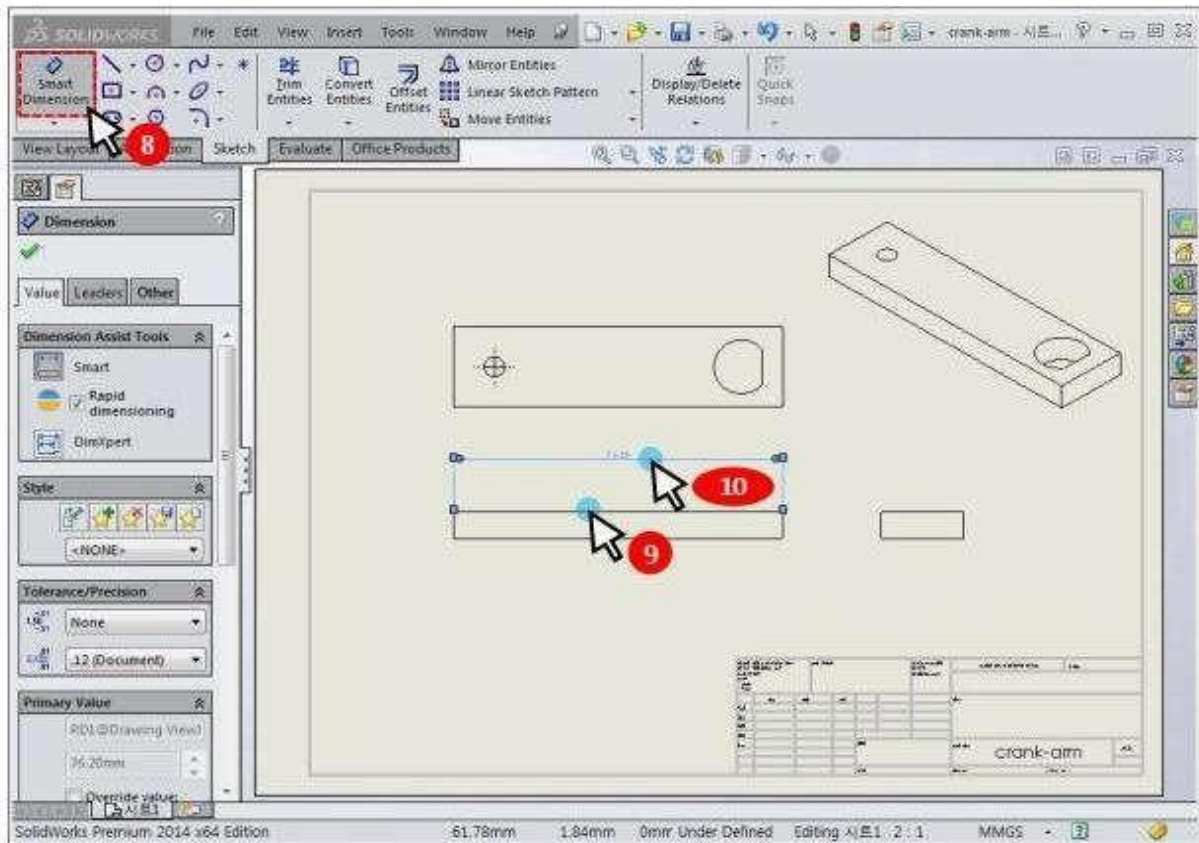
5. Drag mouse and create front view



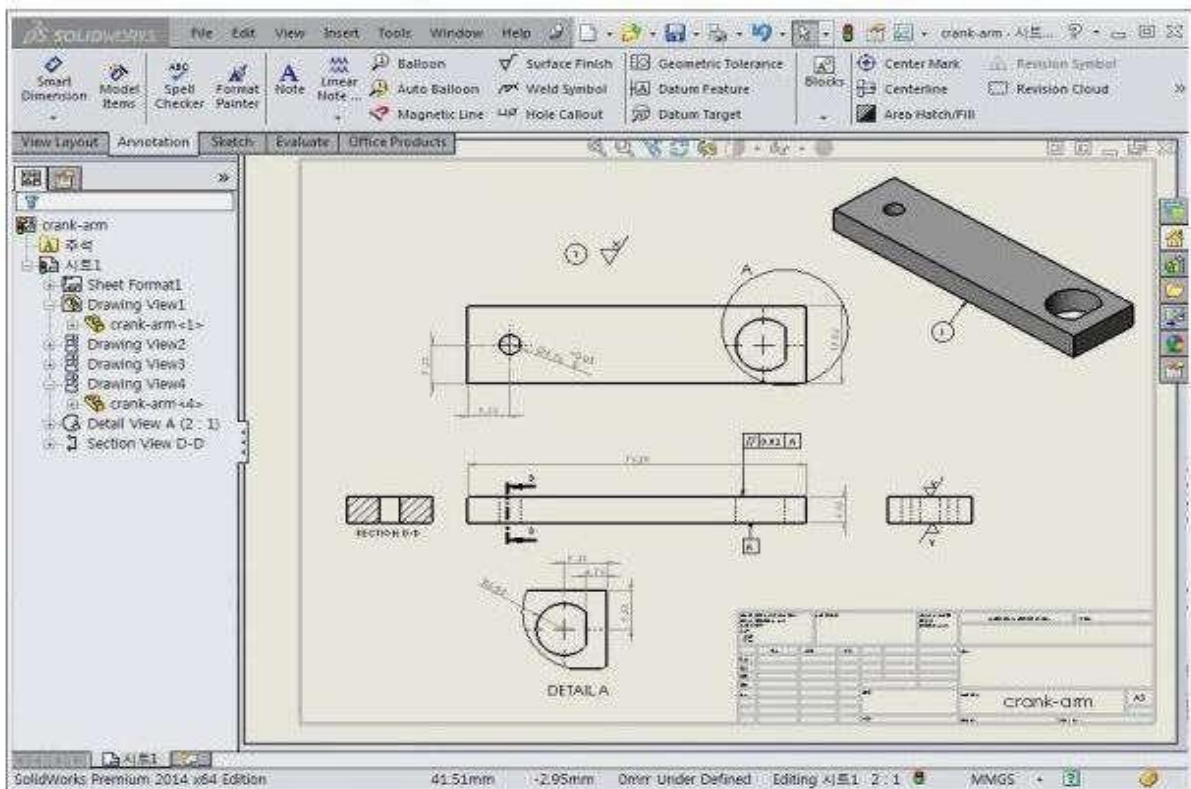
6. Create Isometric view.



7. Add dimentions.



8. Add detail view.



Learning Outcome 2.4: Create Mold-Based 3D Model



Content:

- Create mold based plastic product
- Understand draft angle, wall thickness
- Understand parting line, shut off surfaces, parting surface
- Creating mold core and cavity
- Assemble the mold



Assessment Criteria

1. Be able to understand dimensions and shape.
2. Be able to understand draft angle, wall thickness.



Necessary material and Instruments

Trainees must provide the following equipment and supplies:

1. A computer set
2. SolidWorks software
3. Product sample and drawing instructions



Activity/Work

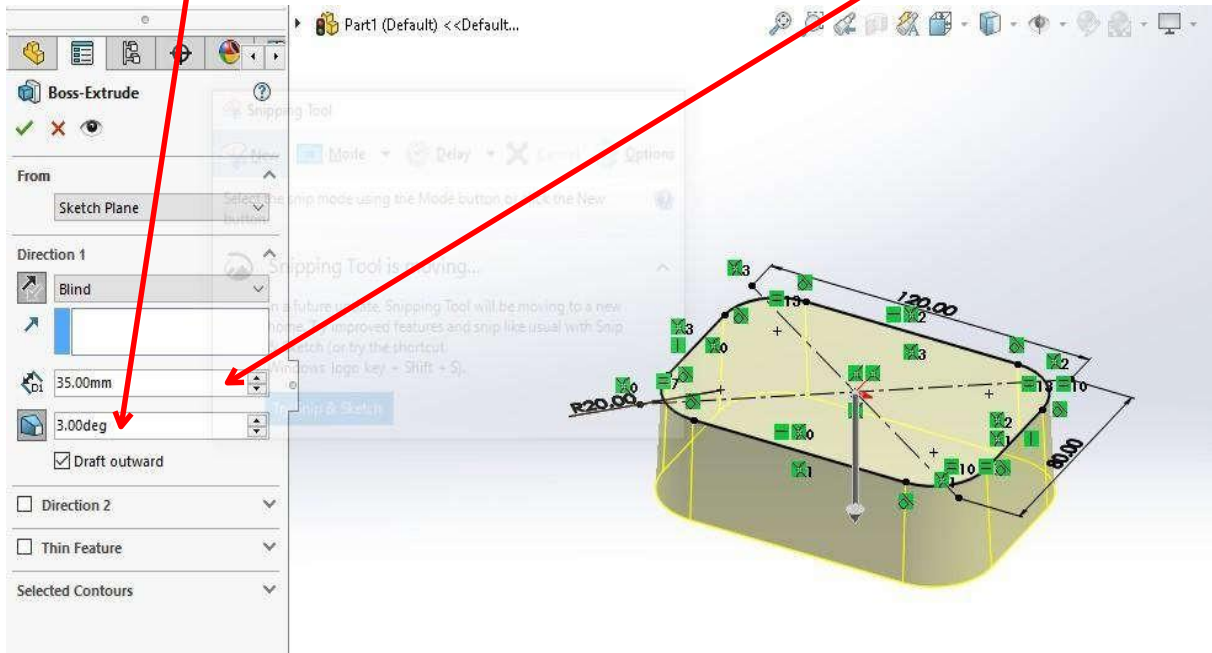
Activity/Work	Resource/Special Instruction/Reference
Create Mold-Based 3D Model	• Information Sheet: 2.4-1 to 2.4-2 • Self-Check: 2.4-1 to 2.4-2 • Answer Script: 2.4-1-2.4-2 • Job Sheet: 2.4 (a)



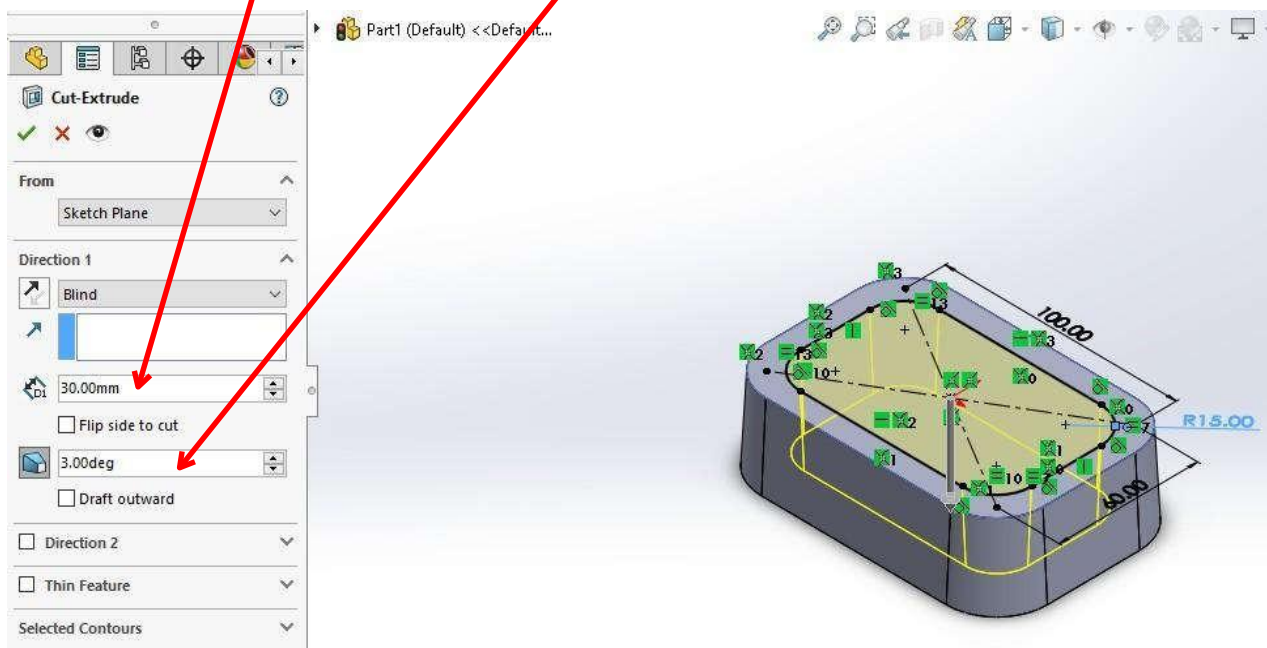
Information Sheet: 2.4-1

Learning Objective: Create a Mold Based Plastic Soap Case Product.

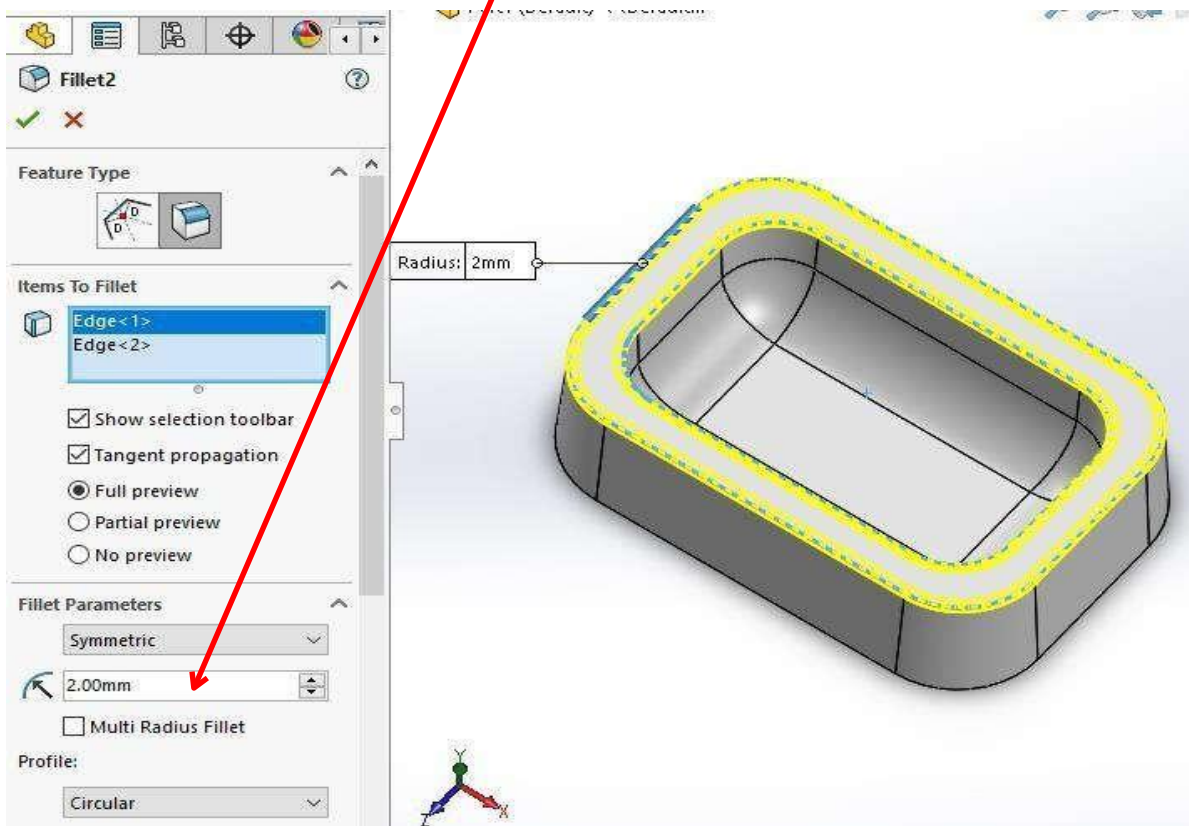
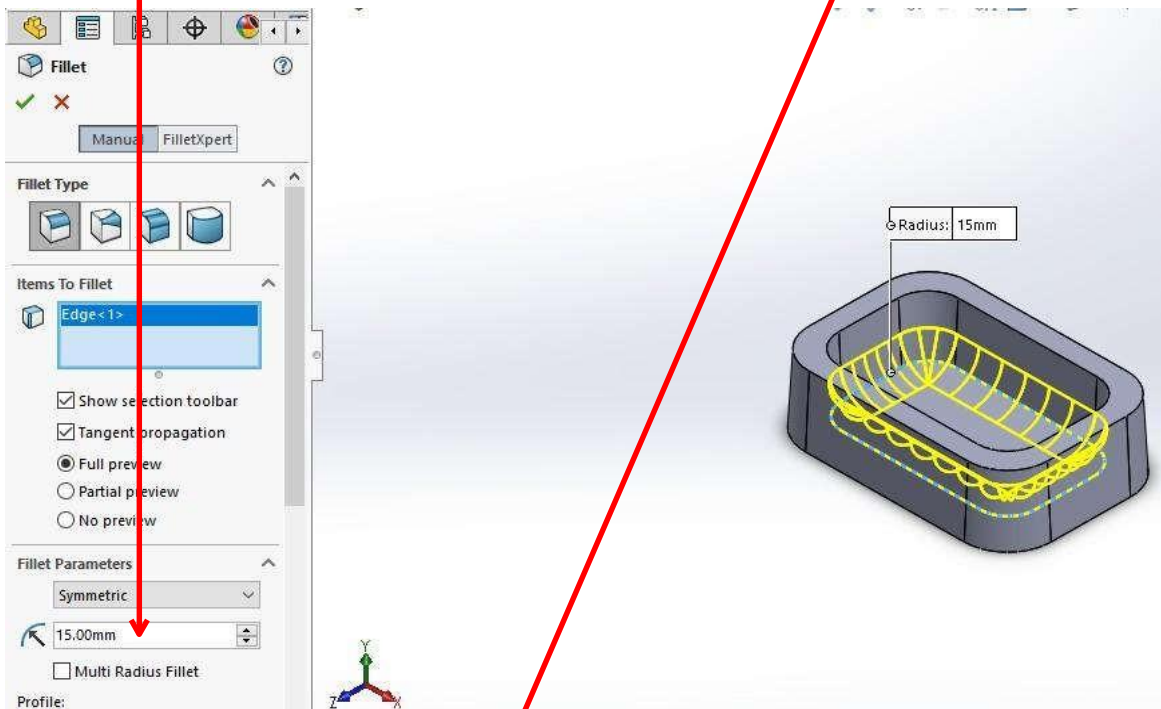
1. Draw a rectangle of 120*80mm and add fillet 20mm. Then **Extrude Boss/Base** it 35mm with draft angle **3 deg outward**.



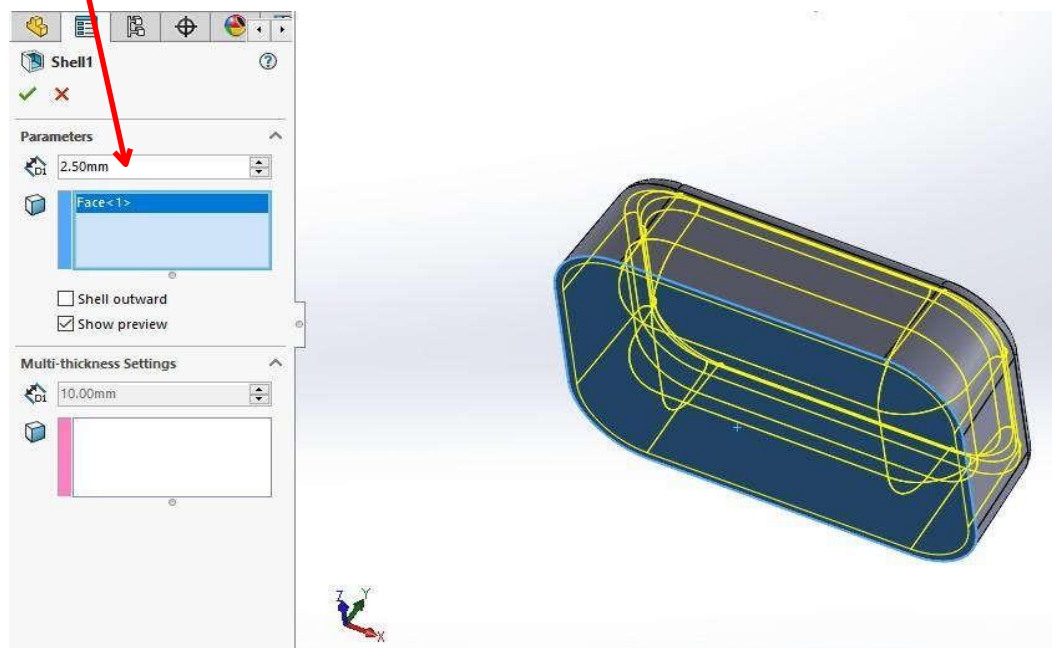
2. Make a rectangle of 100*60mm, and add fillet 15mm on the top face of the extruded part. Then do **Extrude Cut** with depth 30mm and **3 deg inward** draft angle.



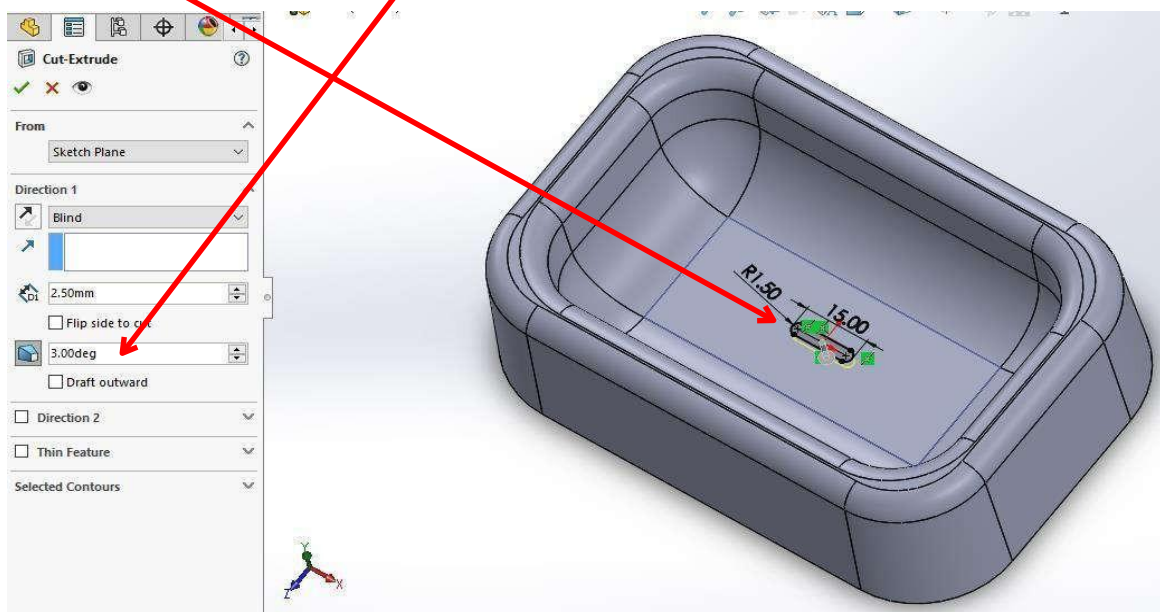
3. Add **15mm Fillet** in Extruded cut side of the part. Then Add **2mm Fillet** in the face of the part.



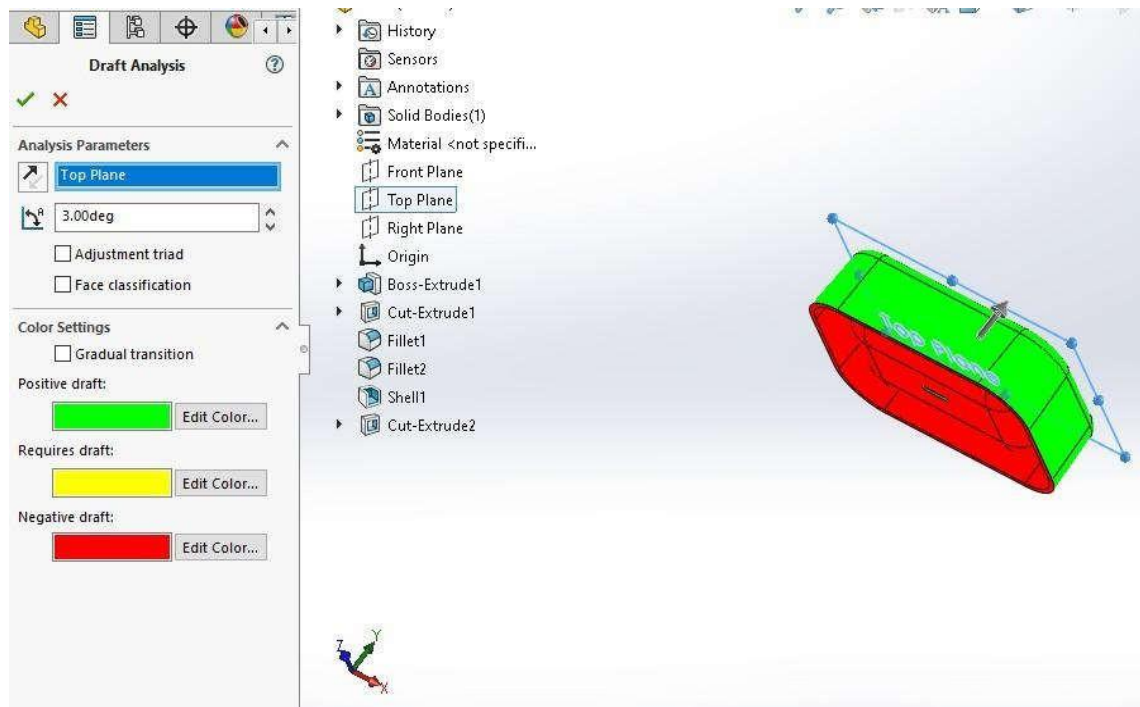
4. Add **2.5mm Shell** on the full part.



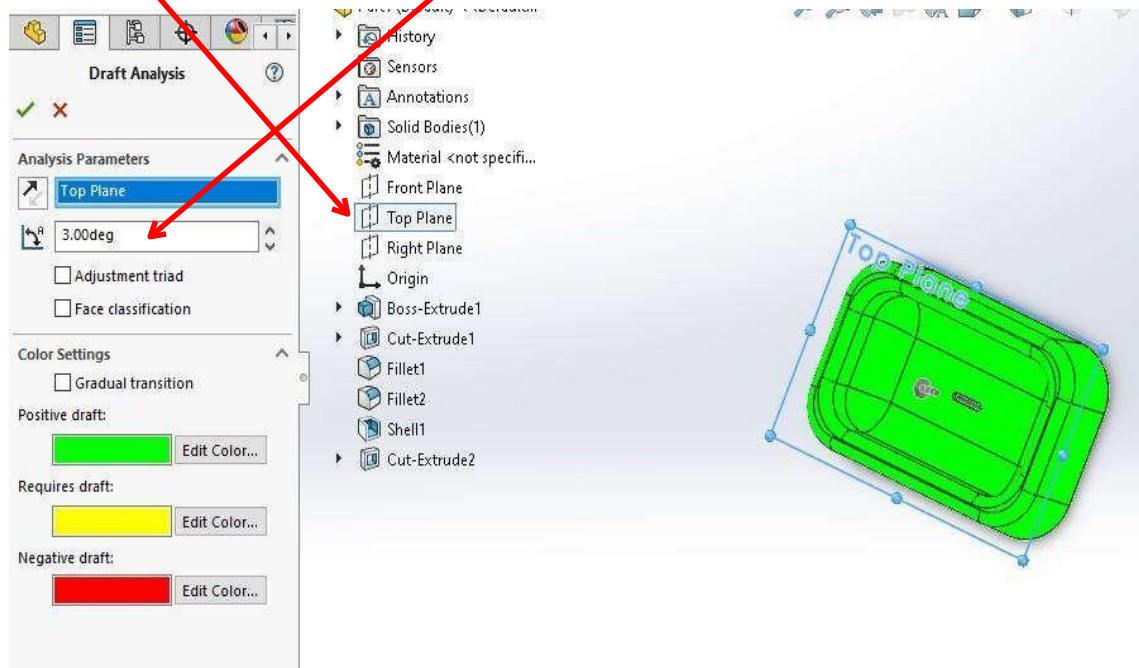
5. Cut a **15*3mm Slot** with **3 deg inward** draft angle in the center of the part.



6. Go to **Mold Tab** and use Draft Analysis.



7. Select **Top Plane** and **Draft Angle ≤ 3 Degree**

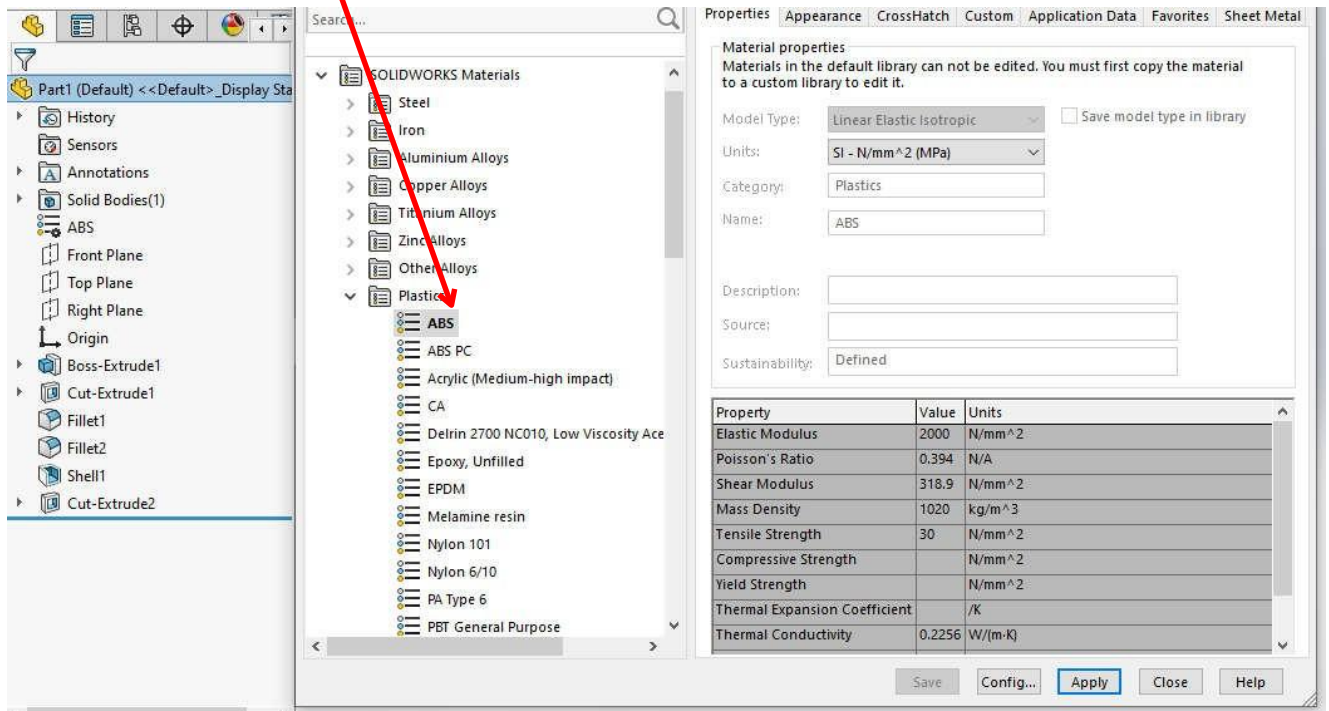




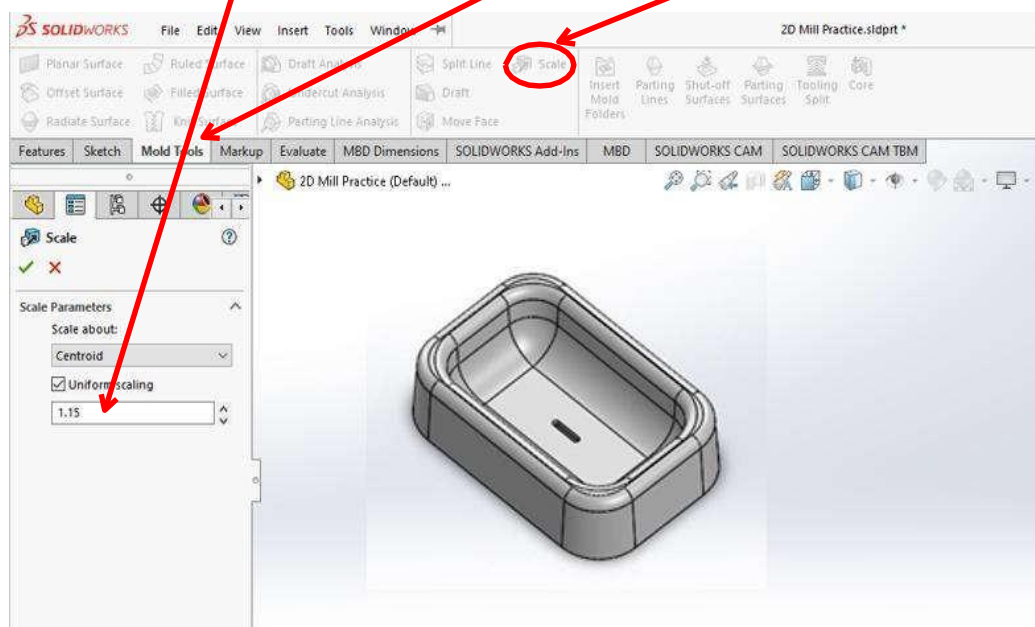
Information Sheet: 2.4-2

Learning Objective: Create mold Core and Cavity based on plastic Soap Case product.

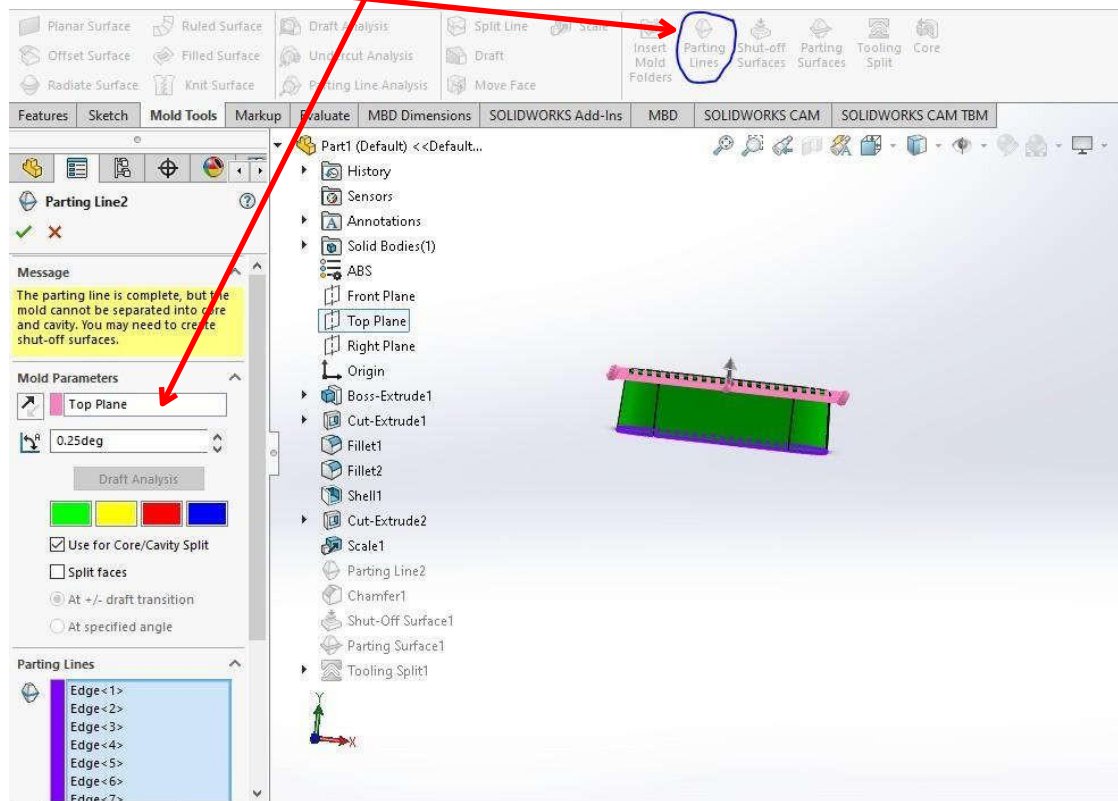
1. Add Material as **ABS**



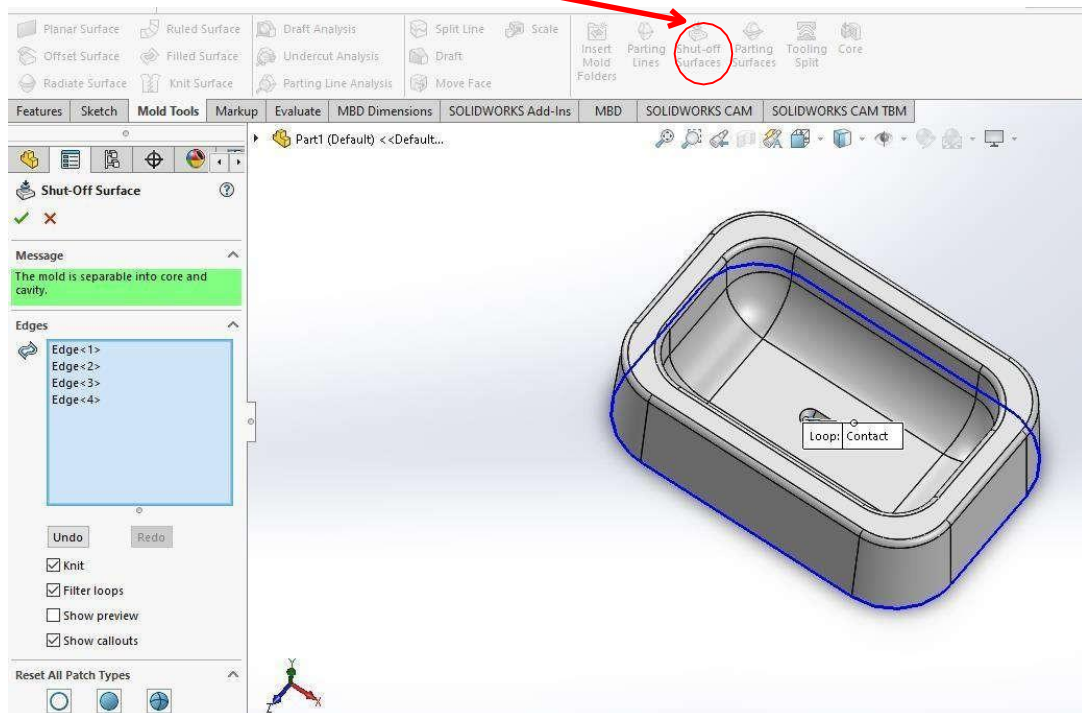
2. Add Shrinkage Allowance **1.15** for ABS from **Mold Tab** using **Scale** tool.



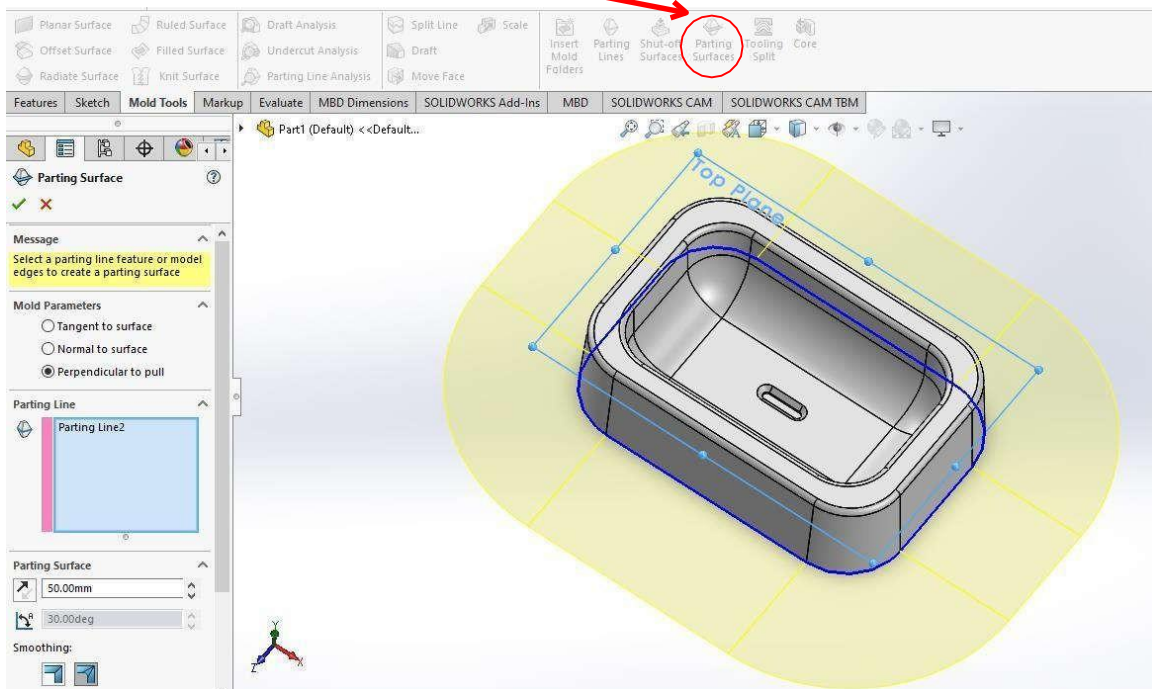
3. Add **Parting Line**. Select **Top Plane** as Mold Parameters



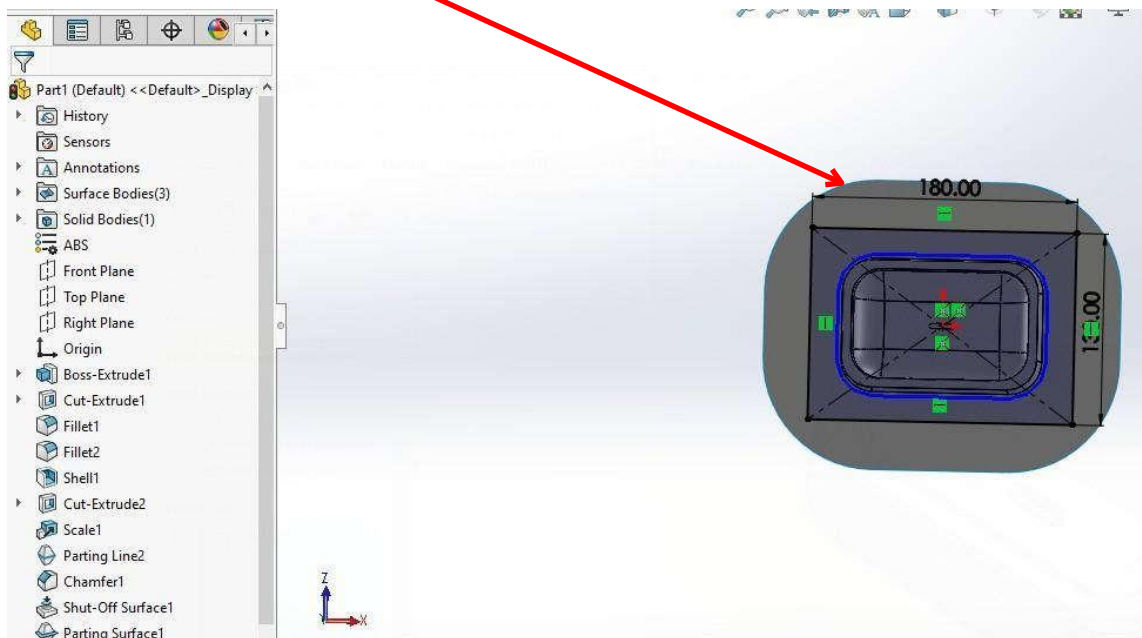
4. Use **Shut-Off Surfaces**.



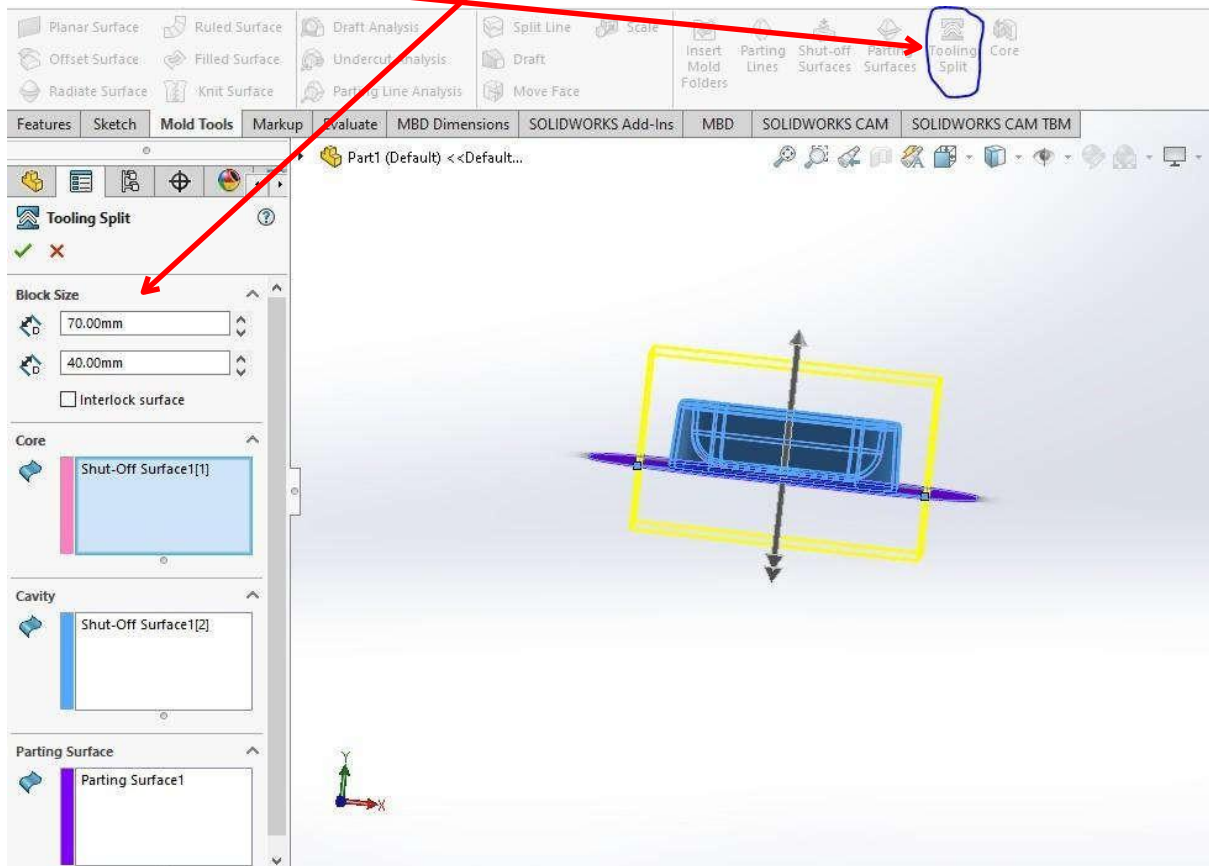
5. Use **Parting Surface**.



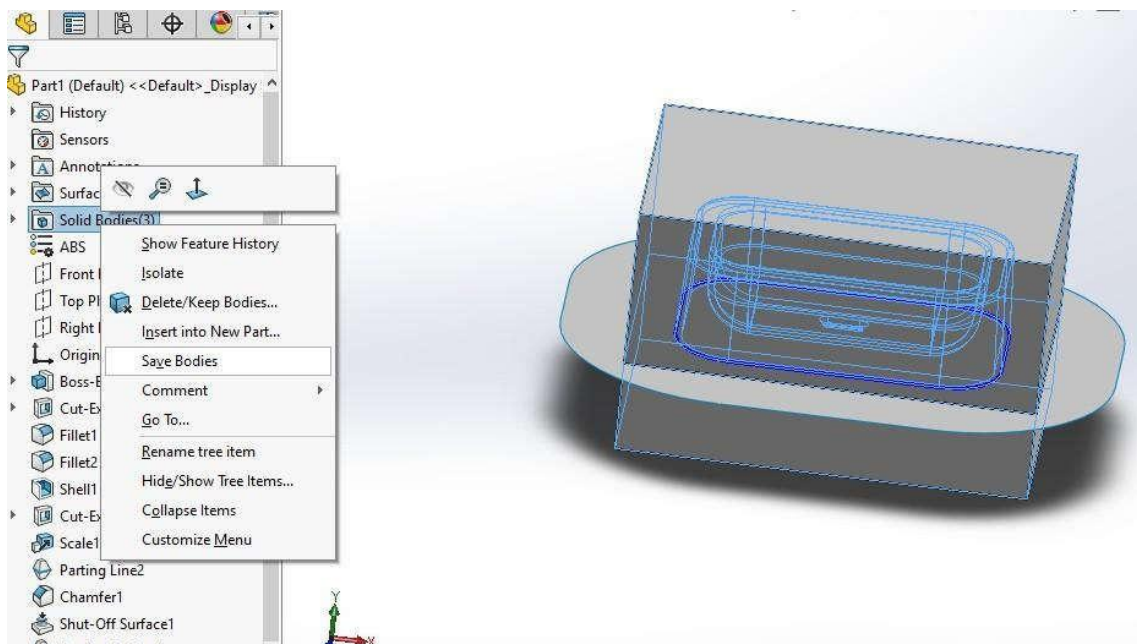
6. Sketch a rectangle of **180*130mm** on **Parting Surface**.



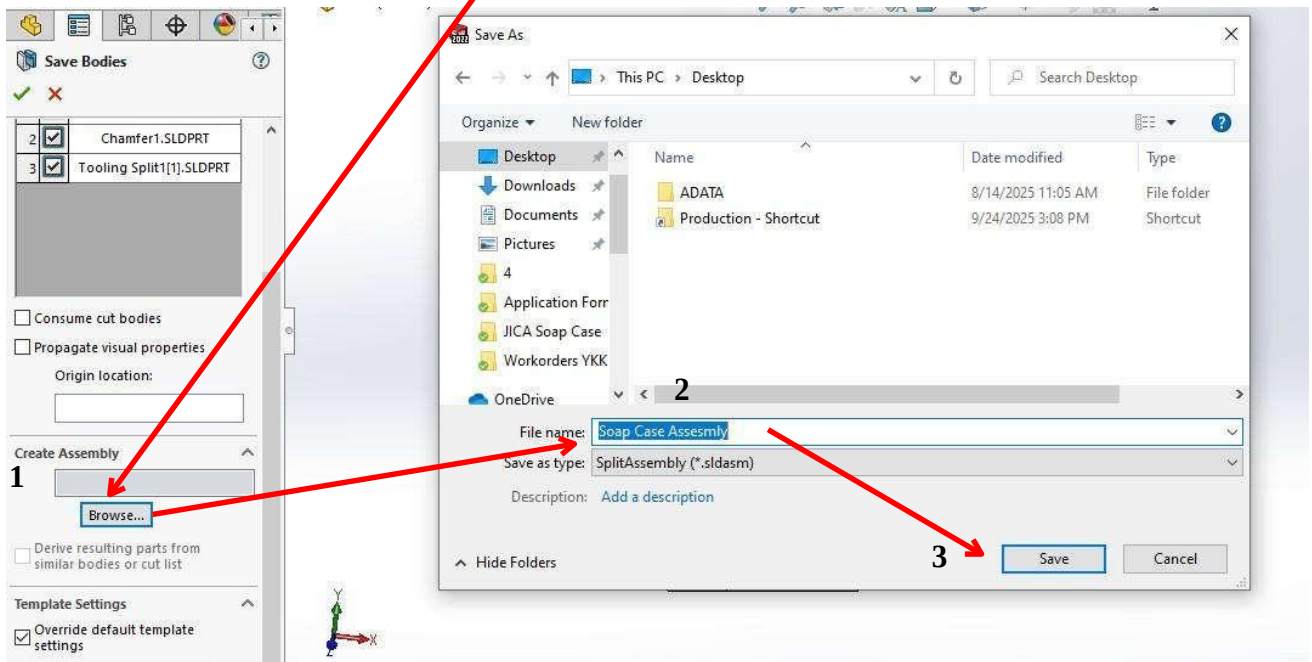
7. Add **Tooling Split.** Give **Block Size 70mm and 40mm**



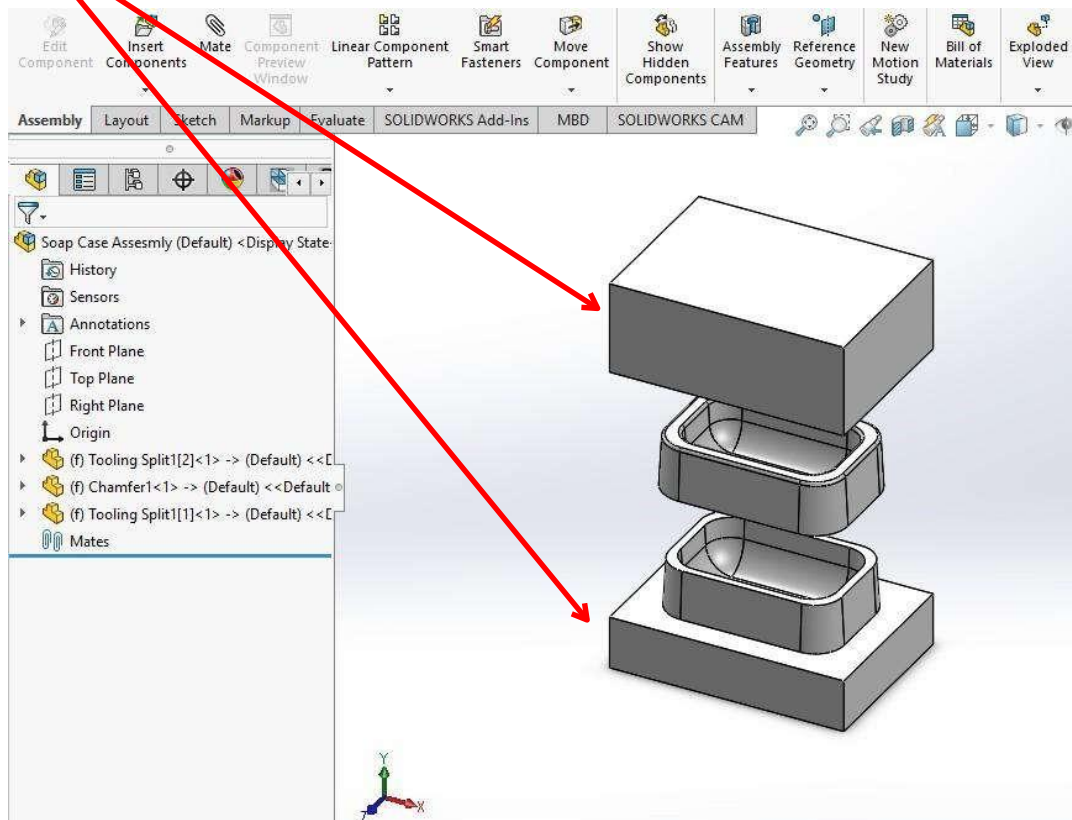
8. Save **Solid Bodies.**



9. Click Auto Assign and Save **Assembly**



10. **Core and Cavity** are Ready





Self-Check: 2.4-1 to 2.4-2

- What is the purpose of running a **Draft Analysis**?
- What is the function of a **Parting Line** in the mold-making process?
- What is created by using the **Shut-Off Surface** tool?
- What is the purpose of the **Parting Surface** tool?
- What tool is used to separate the part into the **Core** and **Cavity** blocks?
- What does the **Scale** tool do in the context of mold design?
- How does the **Cavity** feature differ from the **Tooling Split** feature?



Answer Script: 2.4-1 to 2.4-2

1. The purpose of a Draft Analysis is to check if a molded or cast part has the appropriate draft angles to be easily and cleanly removed from the mold.
2. In mold-making, a Parting Line is the line or edge on a molded part where the two halves of the mold (the core and cavity) meet. It is used to define where the mold will separate and is crucial for creating the parting surfaces needed for mold design.
3. The Shut-Off Surface tool creates surface patches to close up through-holes or openings in a part. This is a crucial step before splitting the mold, as it ensures there are no leaks or gaps where molten material could escape during the molding process.
4. The Parting Surface tool creates the surface that extends from the parting line and defines the boundary between the core and cavity sections of the mold.
5. The **Tooling Split** feature is the primary tool used to create the separate core and cavity blocks for a mold. It uses the previously defined parting and Shut-off surfaces to split a pre-created tooling block into the two mold halves.
6. In mold design, the Scale tool is used to enlarge the part to compensate for material shrinkage. Different materials shrink by different amounts when they cool, and the Scale tool ensures that the molded part will have the correct final dimensions.

7.

The **Cavity** feature subtracts a part body from a larger solid body, typically a mold base, leaving a cavity that matches the shape of the part.

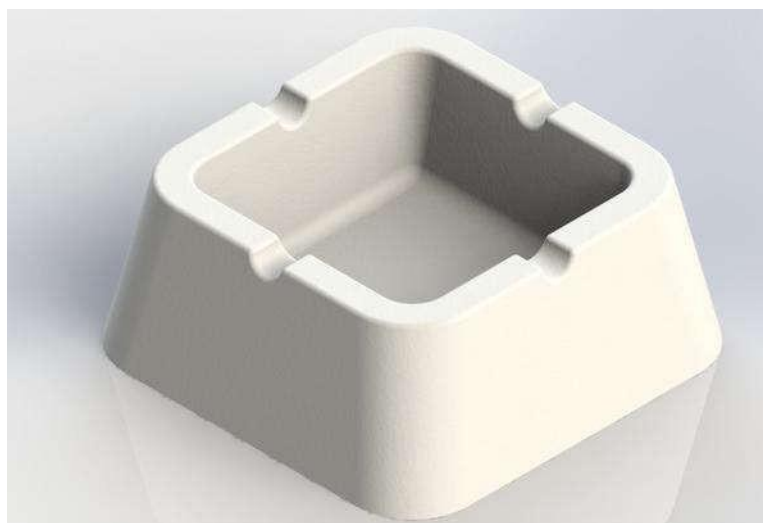
The **Tooling Split** feature is a more comprehensive tool specifically for mold design. It uses parting lines, shut-off surfaces, and a sketched outline to split a single tooling block into the final core and cavity blocks, and can also create interlock surfaces to improve mold alignment.

Job Sheet: 2.4 (a)

Job	Draw a Mold based plastic product and create Core & Cavity	
Materials/Resources	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
Procedure	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 Tools for 3D drawings.	
	Use Mold Tools.	
	Follow trainers/instructors.	
	Save Part files in a folder.	

- **Sample:**

(Assume necessary dimensions and relevant parameters)





Assessment Plan

Module -2	Written test	Performance test	Observation	Oral test	Portfolio	Third party report	Discussion	Project	Weekly/Fin al report	Remarks
Learning Outcome 1		√	√	√	√			√		
Learning Outcome 2		√	√	√	√			√		
Learning Outcome 3		√	√	√	√			√		
Learning Outcome 4		√	√	√	√			√		



Competency Review

Here is a competency review for perform CAD modeling.

Performance Criteria	Yes	No
Able to read and interpret different types of mechanical drawings (detail, assembly, weldment, etc.)?		
Correctly interprets general notes, specifications, draft angles, shrinkage allowances?		
Correctly dimensions different geometric features (holes, slots, radii, angles)?		
Correctly uses symbols for diameter (Ø), radius (R), countersink, counterbore, and depth?		
Can identify and define datums and their role in a tolerance scheme?		
Can generate 2D drawing?		
Can generate 3D model?		
Understand Mold features?		
Can generate Core and cavity?		

Module 3: Select Cutting Tools and Workpiece



Module Content

Module Description: This module describes the knowledge, skills and attitudes required of the operator to understand different types of cutting tools and work pieces. It includes selecting cutting tools for lathe operation, selecting cutting tools for milling operation. Students will learn how the properties of different materials influence the cutting process, tool wear, and part quality. The module covers the characteristics of common tool materials and workpiece materials.

Minimum Time: 20 Hours.



Learning Outcome:

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

- 3.1 Understand cutting tools
- 3.2 Select cutting tools for lathe operations
- 3.3 Select cutting tools for lathe operations
- 3.4 Select cutting tools according to appropriate consistency



Performance Criteria:

- 1. Geometry of cutting tools is identified.
- 2. Types of cutting tools are identified.
- 3. Cutting parameters are identified.
- 4. Cutting tools for lathe operation are identified and the use of tools is described.
- 5. Workpiece material is identified.
- 6. Cutting tools for lathe operation are selected for use.
- 7. Cutting tools for milling operation are identified and the use of tools is described.
- 8. Cutting tools for milling operation are selected for use.

Learning Outcome 3.1: Understand Cutting Tools



Content:

- Single-point tools
- Multi-point tools
- Cutting tool geometry
- Cutting tool materials and properties
- Tool life



Assessment Criteria

1. Be able to understand various types of cutting tools.
2. Be able to understand cutting conditions.
3. Be able to select specific tool for specific operation.
4. Appropriate cutting parameters are followed as per manufacturer chart
5. Be able to predict machining performance.
6. Be able to select different types of cutting tools and inserts.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. Tooling samples
2. Illustrated guides on tool geometry
3. Detailed class notes and manuals



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Understand Cutting Tools	• Information Sheet: 3.1-1 • Self-Check: 3.1-1 • Answer Script: 3.1-1 • Manufacturing Processes by Michael P. Groover



Information Sheet: 3.1-1

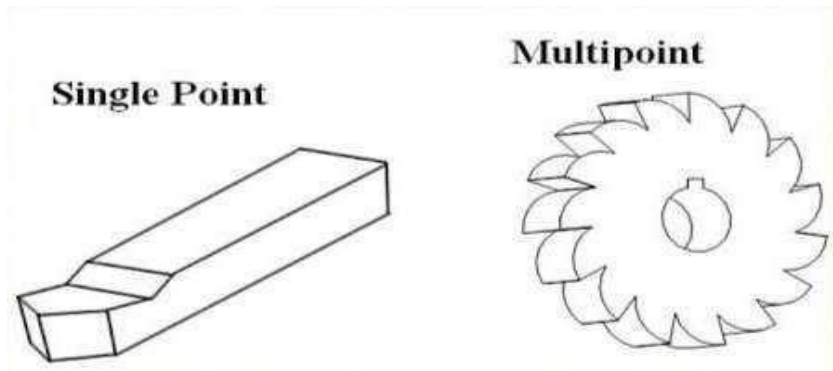
Learning Objective: Understand Cutting Tools.

Definition: It is any tool that is used to remove material from the work piece by means of shear deformation. They must be made of a material harder than the material which is to be cut, and the tool must be able to withstand the heat generated in the metal cutting process.

Different Types of Cutting Tools:

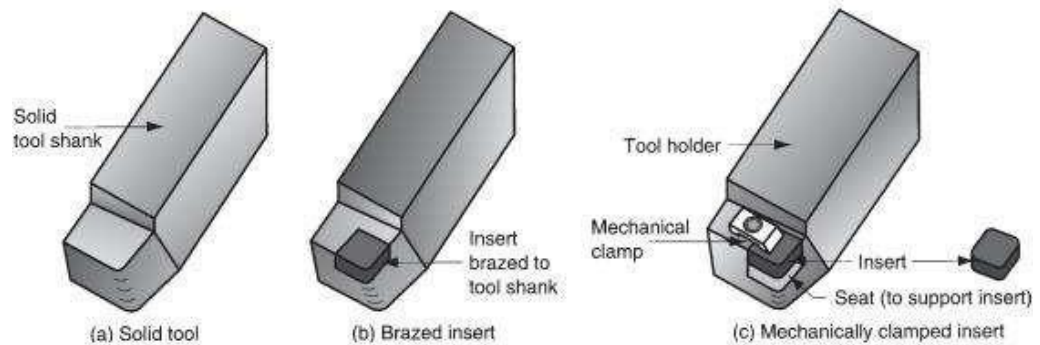
According to the number of cutting edge the cutting tools broadly classified into:

1. Single point cutting tool
2. Multi point cutting tool

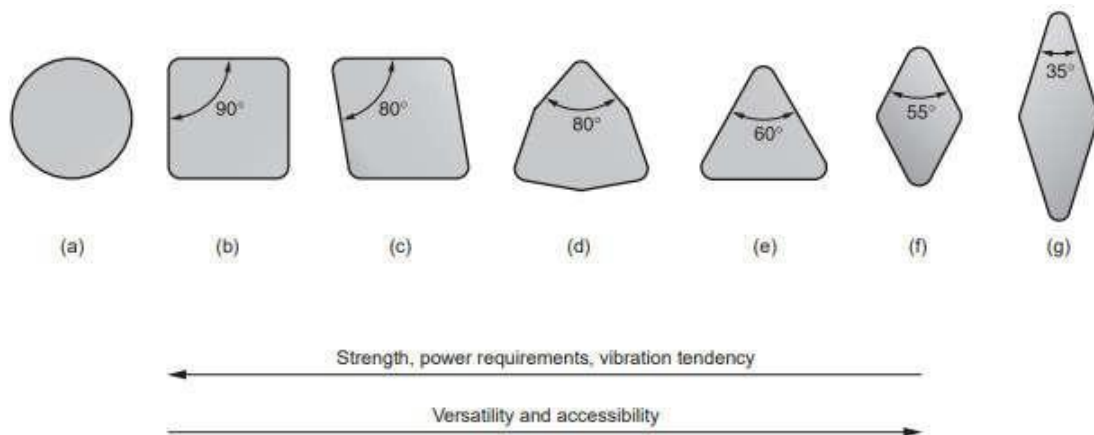


1. **Single Point Cutting Tool:** This type of tools has only one cutting edge to remove excess material from the workpiece. They may be left/ right-handed according to the side of cutting edge when viewing from the end point. Example: tools used in lathes, planers, shapers and boring machines
2. **Multi-Point Cutting Tool:** They have more than one cutting edge to remove excess material. It considered as a series of single point cutting edge on the periphery and during operation each cutting edge independently cut the workpiece. Example: Milling cutters, grinding wheels, drills, reamer, broaches

Ways of Holding and Presenting the Cutting Edge for a Single-Point Tool:



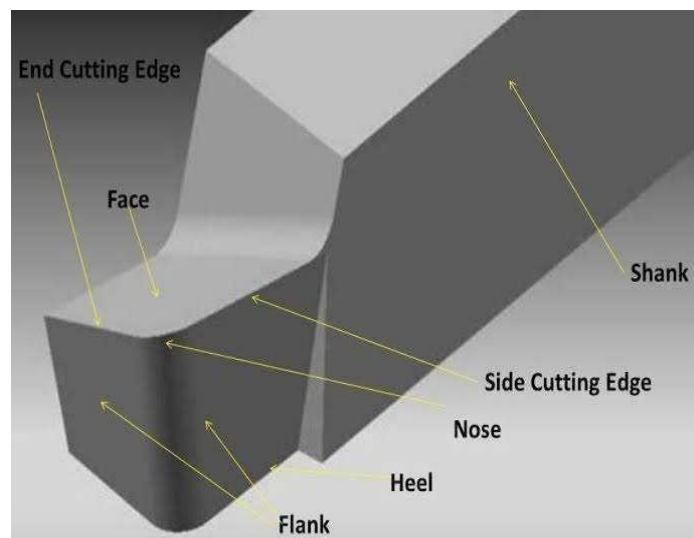
Types of Insert:



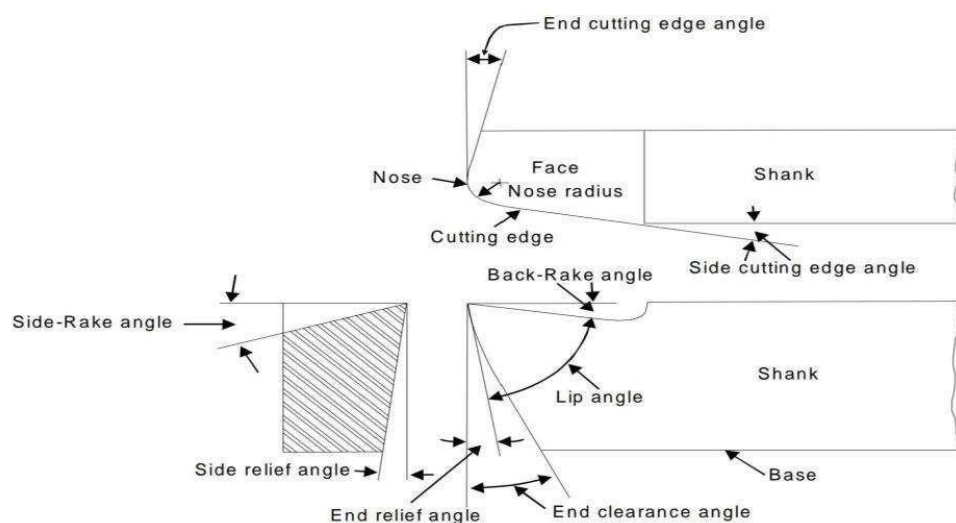
a) round, **b)** square, **c)** rhombus with two 80 point angles, **d)** hexagon with three 80 point angles, **e)** triangle (equilateral), **f)** rhombus with two 55 point angles, **g)** rhombus with two 35 point angles. Also shown are typical features of the geometry. Strength, power requirements, and tendency for vibration increase as we move to the left; whereas versatility and accessibility tend to be better with the geometries at the right.

Geometry of Cutting Tool:

- **Shank:** Main body of tool, it is part of tool which is gripped in tool holder.
- **Face:** Top surface of tool b/w shank and point of tool. Chips flow along this surface.
- **Point:** Wedge shaped portion where face and flank of tool meet.
- **Heel:** It is the intersection of the flank and the base of the tool.
- **Flank:** Portion tool which faces the work. It is surface adjacent to below the cutting edge when tool lies in a horizontal position.
- **Base:** It is the portion of the flank that lies opposite to the top face of the shank.
- **Nose:** The intersection point of major cutting edge and minor cutting edge is called nose.



- **Cutting Edge:** The edge on the tool which removes materials from the work piece is called cutting edges. It lies on the face of the tool. (i) End cutting edge (ii) Side cutting edge



1. Clearance angle: This is the angle between the machined surface and the underside of the tool called the flank face.

a. Side relief angle: It is the angle between the portion of the side flank immediately below the side cutting edge and a line perpendicular to the base of the tool and measured at the right angle to the end flank.

b. End relief angle: It is the angle between the portion of the end flank immediately below the end cutting edge and a line perpendicular to the base of the tool and measured at the right angle to the end flank.

2. Rake angle: It is the angle between the face of the tool called the rake face and the normal to the machining direction.

a. Back rake angle: It is the angle between the tool face and a line parallel to the base of the tool and measured in a plane perpendicular through the side cutting edge.

b. Side rake angle: It is the angle between the tool face and a line parallel to the base of the tool and measured in a plane perpendicular to the base and the side cutting edge.

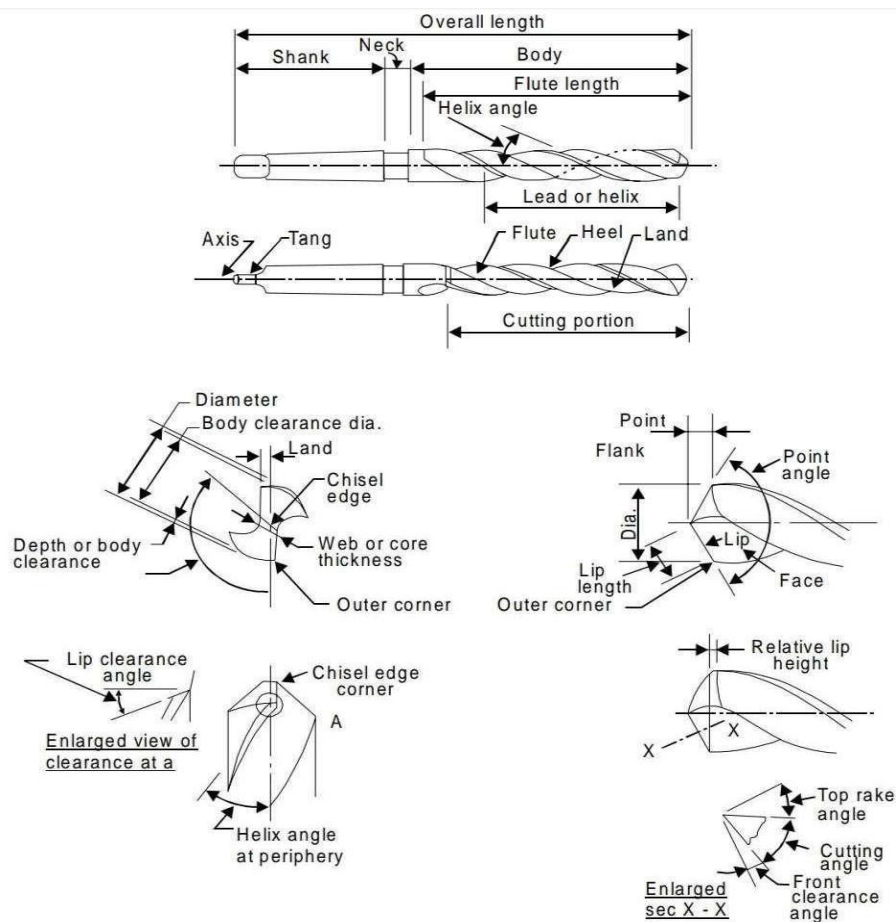
3. Side cutting edge angle: This angle also is known as the lead angle. This is the angle between the side cutting edge and side of the tool shank.

4. End cutting edge angle: This is the angle between the end cutting edge and a line normal to the tool shank

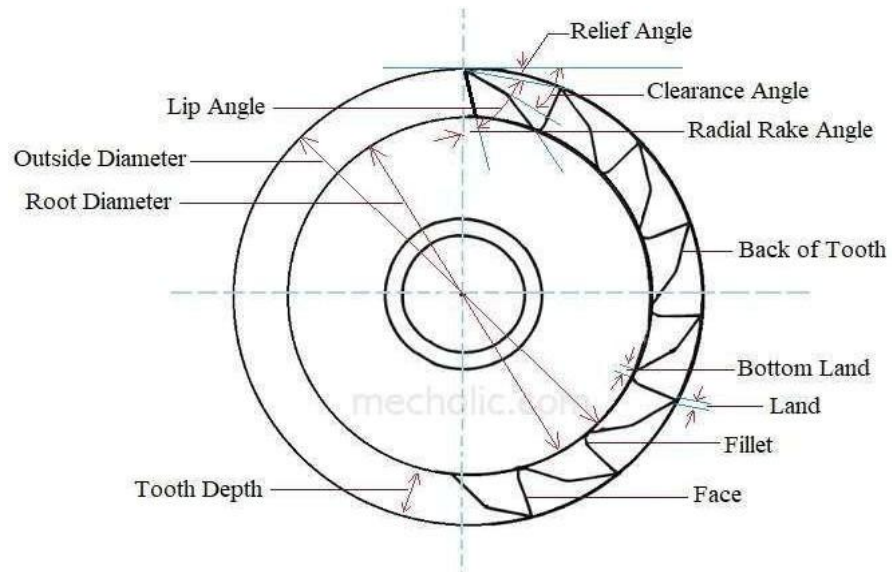
Tool Signature:

- The seven elements that comprise the signature of a single point cutting tool can be state in the following order:
- For example: Tool signature 0-7-6-8-15-16-0.8
 - Back rake angle (0°)
 - Side rake angle (7°)
 - End relief angle (6°)
 - Side relief angle (8°)
 - End cutting edge angle (15°)
 - Side cutting edge angle (16°)
 - Nose radius (0.8 mm)

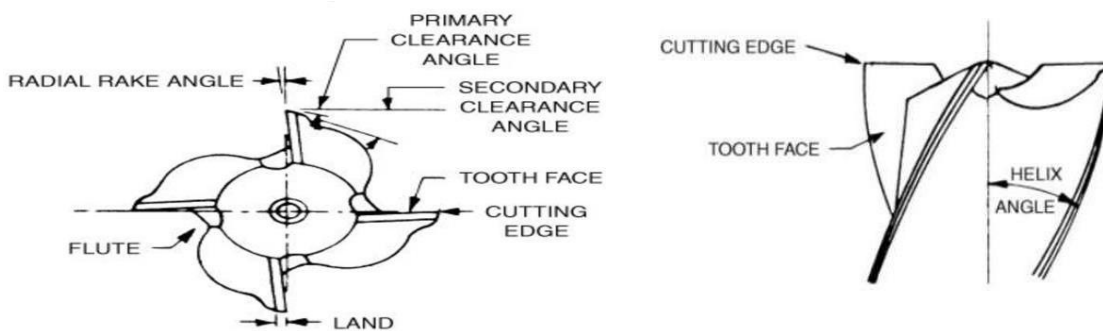
Drill Tool Geometry:



Plain Milling Cutter:



Nomenclature of End Mill:



Cutting Tool Material:

1. **Red hardness or Hot Hardness:** It is the ability of a material to retain its hardness at elevated temperatures.
2. **Wear resistance:** It enables the cutting tool to retain its shape and cutting efficiency
3. **Toughness:** It relates to the ability of a material to resist shock or impact loads associated with interrupted cuts.
4. **Chemical stability:** Tool material must have chemical stability, so that any undesirable reactions between tool material, and work material are avoided.
5. **Shock resistance:** Tool material must have high resistance against thermal and mechanical shocks, especially in intermittent cutting in which tool-work engages and dis-engages at regular intervals.
6. **Low friction:** Tool material must have low coefficient of friction. So that the heat generated will be lower, and tool life increases.

Typical hardness values (at room temperature) and transverse rupture strengths for various tool materials.

Material	Hardness	Transverse Rupture Strength	
		MPa	lb/in ²
Plain carbon steel	60 HRC	5200	750,000
High-speed steel	65 HRC	4100	600,000
Cast cobalt alloy	65 HRC	2250	325,000
Cemented carbide (WC)			
Low Co content	93 HRA, 1800 HK	1400	200,000
High Co content	90 HRA, 1700 HK	2400	350,000
Cermet (TiC)	2400 HK	1700	250,000
Alumina (Al ₂ O ₃)	2100 HK	400	60,000
Cubic boron nitride	5000 HK	700	100,000
Polycrystalline diamond	6000 HK	1000	150,000
Natural diamond	8000 HK	1500	215,000

Tool Life:

There are three common ways of expressing Tool Life:

1. As Time Period in minutes between two successive grindings.
2. In terms of Number of Components machined between two successive grindings. This mode is commonly used when the tool operates continuously, as in case of Automatic Machines.
3. In terms of the Volume of Material removed between two successive grindings. This mode of expression is commonly used when the tool is primarily used for heavy stock removal.

Factors affecting life of Cutting tool:

- a) Cutting speed
- b) Feed and depth of cut
- c) Tool material
- d) Work material
- e) Nature of cutting
- f) Use of cutting fluid



Self-Check: 3.1-1

1. What are two common materials used for cutting tools?
2. What is a rake angle, and what does it do?
3. What is the "nose radius" of a cutting tool, and how does it affect the cut?
4. What is tool life?
5. What are the two basic categories of cutting tools, based on the number of cutting edges?



Answer Script: 3.1-1

1. High-Speed Steel (HSS) and Carbide. Carbide is used for high-speed cutting of hard materials, while HSS is for general-purpose use.
2. The rake angle is the angle of the tool's face. It directs the flow of chips as they are cut from the workpiece. A positive rake angle creates a sharp edge for soft materials, while a negative rake angle creates a stronger edge for hard materials.
3. The nose radius is the rounded tip of the cutting edge. A larger nose radius creates a stronger tool point and can produce a better surface finish, while a smaller radius creates a sharper point for finer work.
4. Tool life is the duration a tool can be used for cutting before it needs to be replaced or resharpened due to wear.
5. Single-point tools (used for turning and boring) and multi-point tools (used for drilling and milling).

Learning Outcome 3.2: Select Cutting Tools for Lathe Operations



Content:

- Select the appropriate cutting tools for CNC lathe operations.
- Lathe machine operations



Assessment Criteria

1. Be able to understand part geometry.
2. Be able to select specific tools for specific operations.
3. Be able to predict machining performance.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. Tooling samples
2. Illustrated guides on tool geometry
3. Detailed class notes and manuals



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Select Cutting Tool for Lathe Operation	• Information Sheet: 3.2-1 to 3.2.-2 • Self-Check: 3.2-1 to 3.2-2 • Answer Script: 3.2-1 3.2-2 • Manufacturing Processes by Michael P. Groover



Information Sheet: 3.2-1

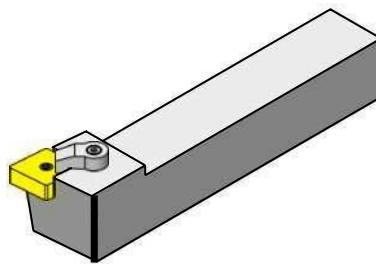
Learning Objective: Introduction to Cutting Tools for CNC Lathe Operations.

Lathe Tool Types:

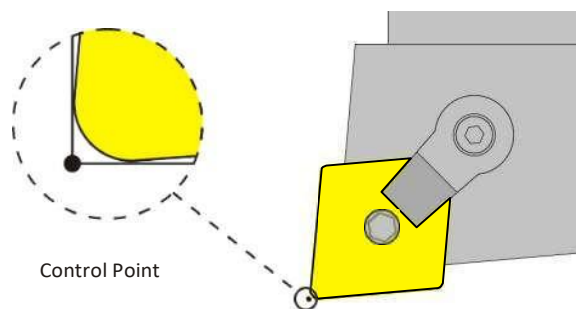
The following is a list of the most common lathe specific tool types. While these are the most common types you are likely to use, they represent only a small number of the tools that are available.

Face/Turn:

For facing and rough turning, use a more rigid tool such as a round, square, or 80-degree diamond. Finishing may require a more versatile tool, such as a 55- or 35-degree diamond. These provide more side and end cutting angle relief to reach and contour part details. Inserts must match the tool holder, and that means the right type, size, shape, and clamping feature(s).



Iso View

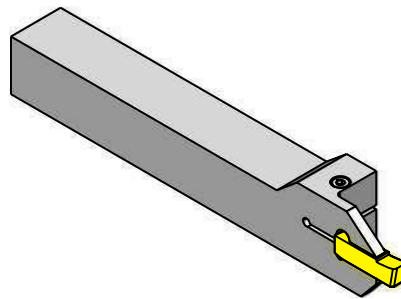


Top View

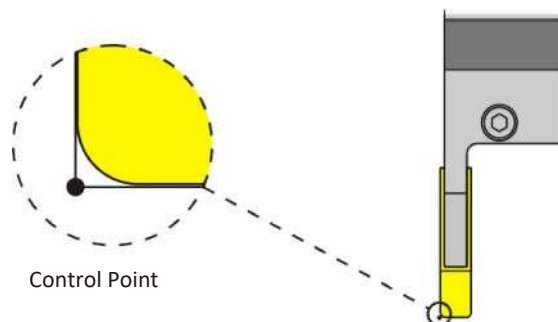
Groove:

Groove tools are classified in part by their width and corner radii. Though used mostly for making groove features, such as O ring, newer generations of these tools can be used for rough and finish contouring operations. While not the best choice for all roughing and finishing, they work well in areas where a diamond or other shape cannot easily fit.

Besides there being many types of groove tools, there are many types of holders, depending mostly on the cut direction for the tool. For example, there are groove holders for OD, ID, and Face grooves, and these are all available in Left Hand or Right-Hand models.



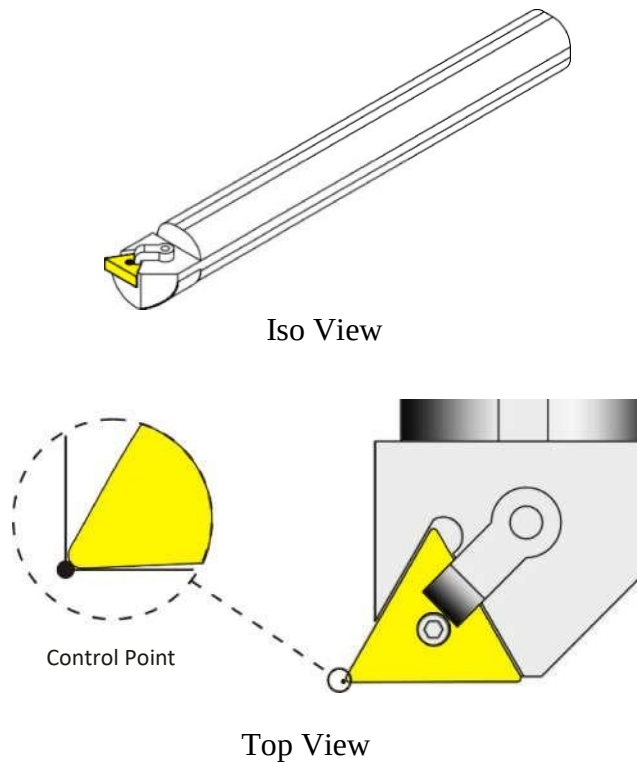
ISO View



Top View

Bore:

Precision holes are often finished with a boring tool. Boring bar tools are mounted parallel to the machine spindle. They require a hole in the part large enough to allow the bar to safely enter and exit the bore.

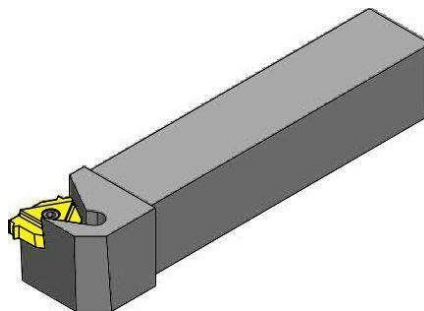


Thread:

Tapped holes at the center of part, up to about one inch diameter, can be made using a form or cutting tap, just like on a mill. Larger ID threads and all ID threads use a thread insert.

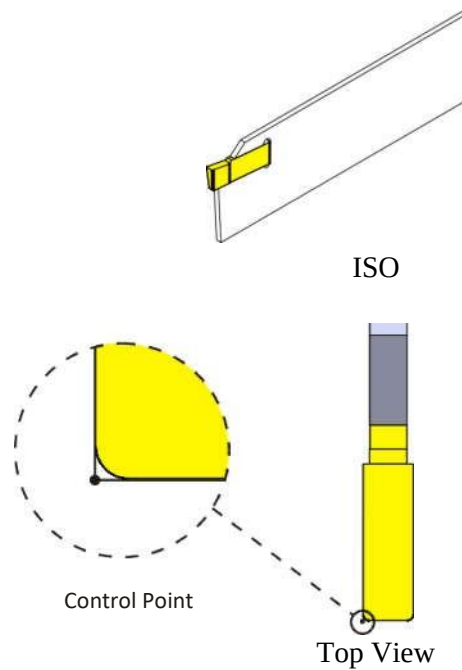
Thread tools are set to the tip of the thread point in Z and X. Z is set by touching off on the edge of the part, and then taking into account the distance from this edge to the tip of the thread point, a distance included in the insert documentation.

Often a thread gage is used to check threads, and the X offset for the thread tool adjusted to achieve the proper size and fit.



Cutoff:

Once the part is finished, it is usually parted, or cut off from the stock. A cutoff tool is a special kind of groove tool that is designed to take deeper cuts. Cutoff tools are classified in part by their width and maximum cutting depth. The blade shape of the cutoff tool allows it to cut deeper into the material than a groove tool. This shape does limit the side forces the tool can withstand.

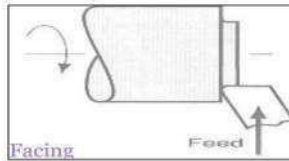




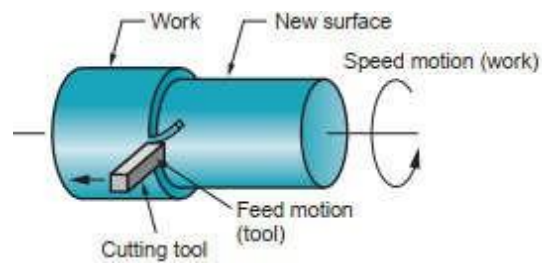
Information Sheet: 3.2-2

Learning Objective: Learning Lathe Machine Operations.

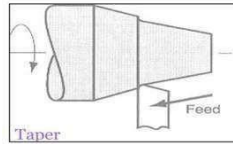
- **Facing:** The tool is fed radially into the rotating work on one end to create a flat surface on the end.



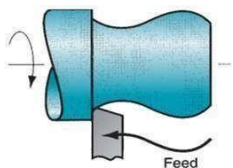
- **Straight Turning:** The work piece is turned straight when it is made to rotate about the lathe axis and the tool is fed parallel to the lathe axis.



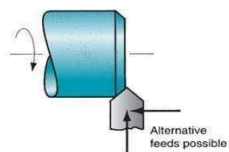
- **Taper Turning:** Instead of feeding the tool parallel to the axis of rotation of the work, the tool is fed at an angle, thus creating a tapered cylinder or conical shape.



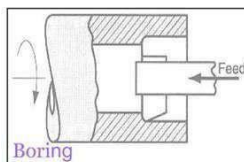
- **Contour Turning:** Instead of feeding the tool along a straight line parallel to the axis of rotation as in turning, the tool follows a contour that is other than straight, thus creating a contoured form in the turned part.



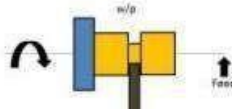
- **Chamfering:** The cutting edge of the tool is used to cut an angle on the corner of the cylinder, forming what is called a “chamfer.”



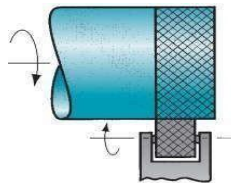
- **Boring:** A single-point tool is fed linearly, parallel to the axis of rotation, on the inside diameter of an existing hole in the part.



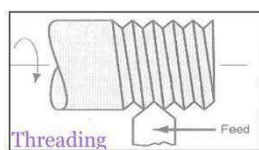
- **Parting/Cut Off:** Parting is a machining operation that results in a part cut-off at the end of the machining cycle. The process uses a tool with a specific shape to enter the workpiece perpendicular to the rotating axis and make a progressive cut while the workpiece rotates.



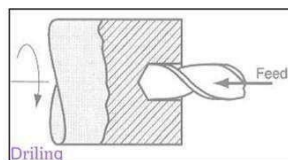
- **Knurling:** This is not a machining operation because it does not involve cutting of material. Instead, it is a metal forming operation used to produce a regular crosshatched pattern in the work surface.



- **Threading:** A pointed tool is fed linearly across the outside surface of the rotating work part in a direction parallel to the axis of rotation at a large effective feed rate, thus creating threads in the cylinder.



- **Drilling:** Drilling can be performed on a lathe by feeding the drill into the rotating work along its axis.





Self-Check: 3.2-1 to 3.2-2

1. What is a lathe machine?
2. What is the difference between turning and facing?
3. Which type of tool is used for reducing the external diameter of a workpiece?
 - a) Parting tool
 - b) Boring tool
 - c) Turning tool
 - d) Facing tool
4. What is the primary difference between a roughing tool and a finishing tool?
 - a) Their color
 - b) The direction of the feed
 - c) The nose radius and cutting-edge geometry
 - d) The tool holder used
5. Which tool material is generally preferred for high-speed machining of hardened steel?
 - a) High-Speed Steel (HSS)
 - b) Carbide
 - c) Aluminum
 - d) Plastic
6. What is the primary function of a boring tool?
 - a) Creating a flat end surface
 - b) Creating threads
 - c) Enlarging an existing hole
 - d) Cutting off a part from the main stock
7. What is the purpose of a chip breaker on a cutting tool?
 - a) To increase the cutting speed
 - b) To reduce friction and heat
 - c) To break chips into smaller, manageable pieces
 - d) To improve surface finish



Answer Script: 3.2-1 to 3.2-2

1. A lathe is a machine that rotates a workpiece against a cutting tool to shape or remove material.
2. Turning is the process of reducing the external diameter of a workpiece, while facing is the process of creating a flat surface on the end of a workpiece.
3. c) Turning tool
4. c) The nose radius and cutting-edge geometry
5. b) Carbide
6. c) Enlarging an existing hole
7. c) To break chips into smaller, manageable pieces

Learning Outcome 3.3: Select Cutting Tools for Milling Operations



Content:

- Select the appropriate cutting tools for CNC milling operations
- Milling machine operations



Assessment Criteria

1. Be able to understand part geometry.
2. Be able to select specific tools for specific operations.
3. Be able to predict machining performance.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. Tooling samples
2. Illustrated guides on tool geometry
3. Detailed class notes and manuals



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Select Cutting Tools for Milling Operations	• Information Sheet: 3.3-1 to 3.3-2 • Self-Check: 3.3-1 to 3.3-2 • Answer Script: 3.3-1 3.3-2 • Manufacturing Processes by Michael P. Groover



Information Sheet: 3.3-1

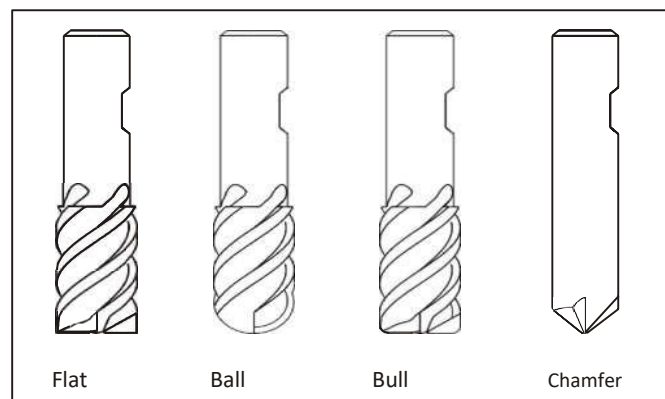
Learning Objective: Introduction to Cutting Tools for CNC Milling Operations.

A wide range of tool types and configurations are available for CNC milling machines. Discussing every type, variation and use is beyond the scope of this course. This chapter introduces the most commonly used tools for prototype and short run production machining. Any tool supply catalog will list many others.

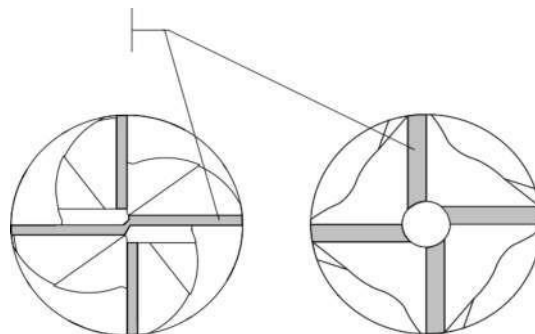
- End mills (Flat, Ball, Bull and Chamfer)
- Face mill
- Corner rounding tools
- Slot tools
- Center drill, Twist drill
- Tap, Reamer & Counterbore

End Mills:

Milling tools include flat, ball, bull nose and chamfer.

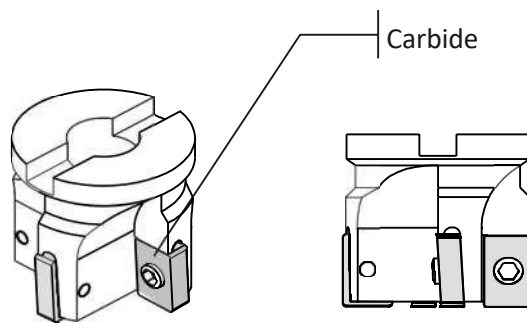


Flat nose mills are used for milling 2D contours and pockets. **Ball nose** mills are used for 3D milling. **Bull nose** end mills have a radius corner. They are used to create a fillet on the bottom of a wall. Because they are sturdier than an end mill, they are also sometimes used for roughing operations. **Chamfer** mills have an angled nose used to create a chamfer parts.



Face Mill:

A face mill has cutting inserts that are replaced when worn. They are rigid, may have up to eight or more cutting edges, and can remove material quickly. They are often used for the first machining operation to quickly create a flat finished face on the part.

**Corner Radius Tool:**

Corner radius (also called Corner Round) tools are used to place a fillet on the outside corner of a part.

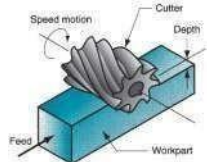




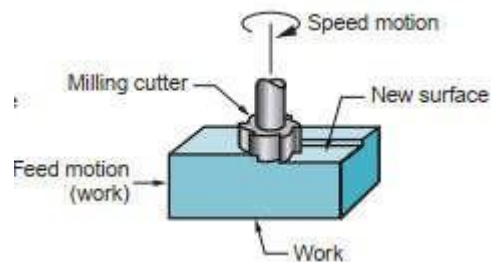
Information Sheet: 3.3-1

Learning Objective: Learning Milling Machine Operations.

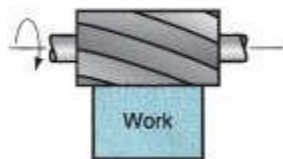
- ▣ **Plain/Peripheral Milling:** The axis of the tool is parallel to the surface being machined, and the operation is performed by cutting edges on the outside periphery of the cutter.



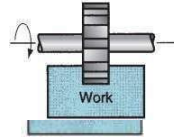
- ▣ **Face Milling:** In face milling, the axis of the cutter is perpendicular to the surface being milled, and machining is performed by cutting edges on both the end and outside periphery of.



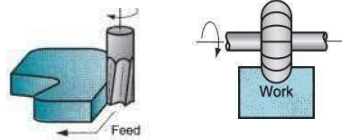
- **Slab Milling:** It is the basic form of peripheral milling in which the cutter width extends beyond the workpiece on both sides.



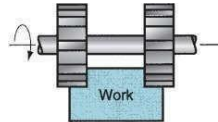
- **Slotting/Saw Milling:** In which the width of the cutter is less than the workpiece width, creating a slot in the work—when the cutter is very thin, this operation can be used to mill narrow slots or cut a work part in two, called saw milling.



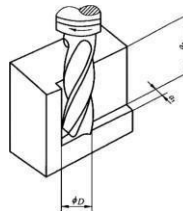
- **Form Milling:** In which the milling teeth have a special profile that determines the shape of the slot that is cut in the work. Form milling is therefore classified as a forming operation.



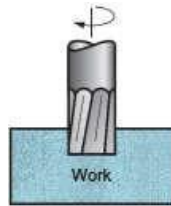
- **Straddle Slitting:** It is the same as side milling, only cutting takes place on both sides of the work.



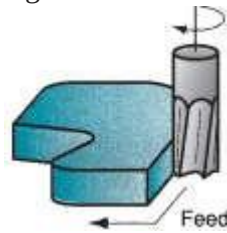
- **Side Milling:** The process of milling surfaces that are at right angles to the axis of rotation of the cutter with a side millingcutter.



End Milling: In which the cutter diameter is less than the work width, so a slot is cut into the part.



➤ **Profile Milling:** It is a form of end milling in which the outside periphery of a flat part is cut.





Self-Check: 3.3-1 to 3.3-2

1. What is a milling machine?
2. Name common milling operations.
3. What is up milling and down milling?
4. What are the differences between face milling and end milling?



Answer Script: 3.3-1 to 3.3-2

1. Answer: A machine tool used to remove material using a rotating multi-point cutting tool (milling cutter).
2.
 - a) Plain (slab) milling
 - b) Slot milling
 - c) Side milling
 - d) Form milling
 - e) Gear cutting
3.
 - Up milling (conventional): Cutter rotates opposite to feed.
 - Down milling (climb): Cutter rotates in same direction as feed.
4.
 - Face milling: Cutter axis perpendicular to the surface; used for flat surfaces.
 - End milling: Cutter axis parallel to surface; used for slots, pockets, and profiles.

Learning Outcome 3.4: Select Cutting Tools According to Appropriate Consistency



Content:

- Select the appropriate cutting tools for operations
- Understand part geometry
- Understand surface finish



Assessment Criteria

1. Be able to understand part geometry.
2. Be able to select specific tools for specific operations.
3. Be able to predict machining performance.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. Tooling samples
2. Illustrated guides on tool geometry
3. Detailed class notes and manuals



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Select Cutting Tools According to Appropriate Consistency	• Information Sheet: 3.4-1 • Job Sheet: 3.4(a)



Information Sheet: 3.4-1

Learning Objective: Selection of Cutting Tools According to Appropriate Consistency.

Selecting cutting tools based on appropriate consistency involves matching the characteristics of the cutting tool with the material being machined, as well as considering factors such as tool life, surface finish, and machining efficiency.

A. Here's how you might select cutting tools based on different consistencies of materials:

1. Soft Materials (e.g., Aluminum, Plastics): For soft materials, carbide cutting tools are often suitable due to their sharpness and ability to maintain cutting edges.

- High-speed steel (HSS) tools can also be effective for softer materials, especially if cost is a concern.
- Consider using high-rake angle cutting tools for efficient chip evacuation and reduced cutting forces.
- **Hard materials (e.g., hardened steel, titanium alloys):** For hard materials carbide cutting tools with specialized coatings like TiAlN (titanium aluminum nitride) or TiCN (titanium carbonitride) are preferred due to their high wear resistance.
- Consider using inserts or end mills specifically designed for hard milling applications.
- Use cutting parameters such as lower speeds and feeds to reduce heat generation and prolong tool life.

2. Brittle materials (e.g., ceramics, glass):

- Diamond-coated tools or PCD (polycrystalline diamond) tools are often used for machining brittle materials due to their exceptional hardness and abrasion resistance.
- Carefully control cutting forces and minimize vibrations to prevent chipping or fracturing of the workpiece.

3. Tough materials (e.g., stainless steel, Inconel):

- Use carbide cutting tools with tough substrates and coatings designed to withstand high temperatures and abrasion.
- Consider using coolant or cutting fluids to reduce heat buildup and improve chip evacuation.
- Use proper tool geometry to manage cutting forces and prevent tool wear.

4. Abrasive materials (e.g., composites, fiber-reinforced plastics):

- Specialized carbide or diamond-coated cutting tools are often used for machining abrasive materials to maintain sharp cutting edges and prevent premature wear.
- Consider using proper cutting speeds and feeds to prevent overheating and elimination of the workpiece.

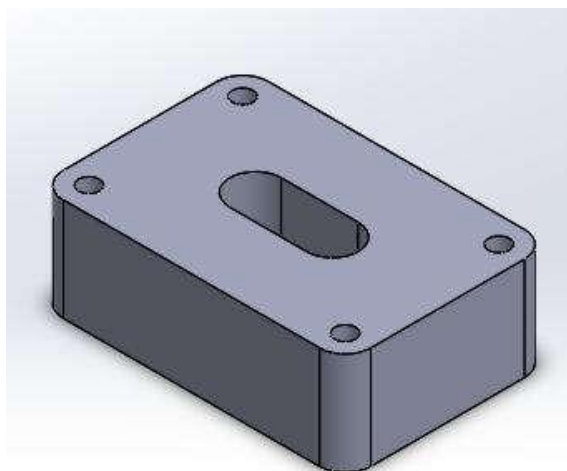
B. Here's a breakdown of suitable cutting tools for the materials you listed, Considering their consistency:

- 1. Mild Steel:** This is a relatively soft metal. You can use High-Speed Steel (HSS) tools for most applications. For high-production environments or heavy cuts, consider Tetanized HSS or Carbide tools.
- 2. Aluminum:** Similar to mild steel, HSS tools work well for aluminum. Carbide tools can be beneficial for high-speed machining or continuous cutting.
- 3. Alloy Steel:** The hardness of alloy steel varies depending on the specific alloy. HSS tools may be sufficient for softer alloy steels, but Carbide is generally recommended for most applications.
- 4. Bronze:** Bronze is softer than steel. HSS tools are suitable, but Carbide can offer advantages in terms of wear resistance and longer tool life.
- 5. Copper:** Another soft metal, copper can be effectively machined with HSS tools.
- 6. Stainless Steel:** Varies in hardness depending on the grade. HSS can work for softer grades, but Carbide is the preferred choice for most stainless-steel machining due to its higher wear resistance.
- 7. Tool Steel:** Extremely hard and wear-resistant. Carbide tools are necessary for machining tool steel.
- 8. General Plastics:** HSS or Carbon steel tools can be used for many plastics. However, specific plastic types may require specialized tool materials to avoid melting or excessive wear.
- 9. Teflon (PTFE):** Prone to melting at high temperatures. Specialized coated tools or cryogenic cooling techniques are often used to minimize heat generation during cutting.
- 10. Fiber:** The type of fiber (e.g., glass fiber, carbon fiber) influences tool selection. HSS or diamond-coated tools might be suitable depending on the specific fiber and its reinforcement material.
- 11. Nylon:** Relatively soft plastic. HSS or even Carbon steel tools can be effective.

Job Sheet: 3.4 (a)

Job	Select Cutting Tools According to Appropriate Sequence of Operation	
Materials/Resources	PPE	1 Set
	Cutting tools	As required
	Drawing/ specification sheet	1 Pc
	Paper and pen	1 Pc
Procedure	Collect required materials/resources.	
	Follow OSH.	
	Read job sheet and specification sheet.	
	Collect all tools & equipment.	
	Take paper and pen.	
	Note cutter name.	
	Clean working area.	
	Store all equipment safely.	

- **Sample:**





Assessment Plan

Module -3	Written test	Performance test	Observation	Oral test	Portfolio	Third party report	Discussion	Project	Weekly/Final report	Remarks
Learning Outcome 1	√		√	√						
Learning Outcome 2	√		√	√						
Learning Outcome 3	√		√	√						
Learning Outcome 4	√		√	√						



Competency Review

Here is a competency review for select cutting tool and work piece.

Performance Criteria	Yes	No
Able to select the appropriate workpiece material based on mechanical properties and intended application?		
Can identify and define the machinability of different materials?		
Can determine the correct cutting tool material (e.g., HSS, carbide, ceramic) based on workpiece hardness and cutting speed?		
Correctly determines cutting parameters (speed, feed rate, depth of cut) for a given tool-workpiece combination?		

Module 4: Perform CAM Programming



Module Content

Module Description: This module describes the knowledge, skills and attitudes required of the operator to perform CAM (Mastercam) programming. It specifically includes the tasks of installing CAM software (Mastercam), using CAM software (Mastercam) interface, carrying out 2D CAM programming, carrying out 3D CAM programming, loading and running program.

Minimum Time: 110 Hours.



Learning Outcome:

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

- 4.1 Install and activating CAM software (Mastercam)
- 4.2 Use CAM software interface
- 4.3 Carry out 2D CAM programming
- 4.4 Carry out 3D CAM programming
- 4.5 Load and run program



Performance Criteria:

- 1. CAM package is installed as per standard operating procedure.
- 2. System parameters are identified and selected according to job requirement.
- 3. CAM software interface is used as per job requirements.
- 4. CAD model is imported and oriented.
- 5. Reference point is established or originated based on job requirement.
- 6. Stock set-up is performed.
- 7. Cutting tools are identified and selected.
- 8. Sequential 2D and 3D toolpaths for milling operations are identified, generated and verified.
- 9. Sequential 2D toolpaths for lathe operations are identified, generated and verified.
- 10. G code and M code are generated.
- 11. Program is loaded using appropriate device.
- 12. Program is executed to produce work piece.

Learning Outcome 4.1: Install CAM Software



Content:

- Check system requirements
- Find CAM software (Mastercam) license key number
- Download and run the installer
- Navigate to the downloaded files and find the setup.exe



Assessment Criteria

1. Be able to check system requirements.
2. Be able to find CAM software (Mastercam) license key number.
3. Be able to download and run the installer.
4. Be able to navigate to the downloaded files and find the setup.exe.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. A computer set.
2. Must meet the minimum and recommended specifications:
 - a. Operating System: 64-bit Windows 10 or 11
 - b. Processor: Intel or AMD processor, 3.3 GHz or higher
 - c. RAM: Minimum 8GB, with 32GB or more recommended for larger assemblies and simulation
 - d. Storage: A Solid-State Drive (SSD) with at least 20GB of free disk space
 - e. Graphics Card: A certified graphics card (NVIDIA Quadro or AMD Radeon Pro)
3. Mastercam with license key number.



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Install CAM Software	• Information Sheet: 4.1-1



Information Sheet: 4.1-1

Learning Objective: Install and CAM Software (Mastercam).

CAM: CAM means Computer-Aided Manufacturing. It is an application technology that uses computer software and machinery to facilitate and automate manufacturing processes. CAM application is used to both design a product and programming manufacturing processes, specifically, CNC machining. CAM software uses the models and assemblies created in CAD software to generate toolpaths that drive machine tools to turn designs into physical parts.

Installation:

Step 1: Download the installation files.

- Browse to Mastercam.com. At the top right, click Login (myMastercam).
- Sign in to your account. If you need help creating an account or linking it to your assets, refer to this article on linking your account.
- Hover over the Downloads link at the top of the page and choose Mastercam or your desired software package.

Mastercam.com | English | Marcus Brown | Search

VERSIONS | LEARNING | COMMUNITIES | **DOWNLOADS** | FIND A CHANNEL PARTNER

Version:
Mastercam 2025

Mastercam Downloads

Full Installations

ITEM	POSTED	SIZE	DOWNLOAD
Mastercam 2025 Update 3 - Full Installation - Mastercam only - Web Download - Self-Extracting EXE	October 22, 2024	2.03 GB	
Mastercam 2025 - Full Installation - Mastercam only - Web Download - Self-Extracting EXE	July 1, 2024	2.03 GB	

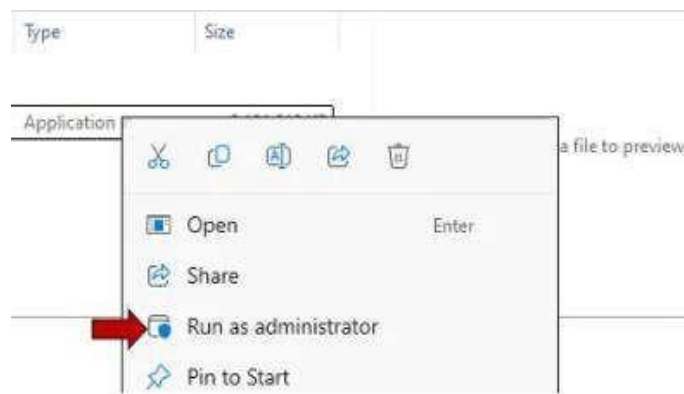
Updates

ITEM	POSTED	SIZE	DOWNLOAD
Mastercam 2025 Update 4 - Patch Installation - Self-extracting *.exe file	December 3, 2024	132.42 MB	
Mastercam 2025 Update 3 - Patch Installation - Self-extracting *.exe file	October 22, 2024	85.58 MB	

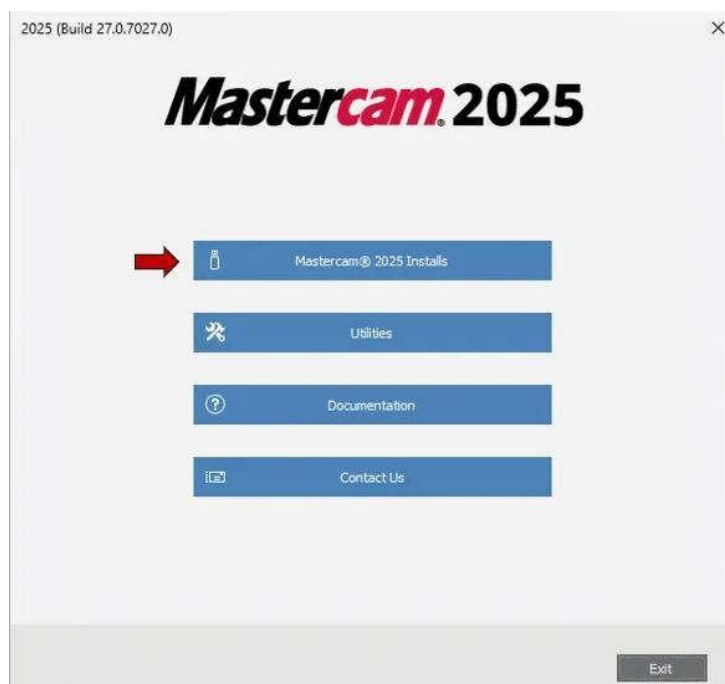
- d. Click the direct download link, or for more information select the link on the download version, then scroll down on that page and click Download. The file will be about 2 GB in size and will be named similar to mastercam2026-web.exe or mastercam2025-update3-web.exe.
- e. Once it is downloaded, run the file to begin the installation. Refer to these instructions to install Mastercam.

Step 2: Find the **mastercam.exe** on your computer.

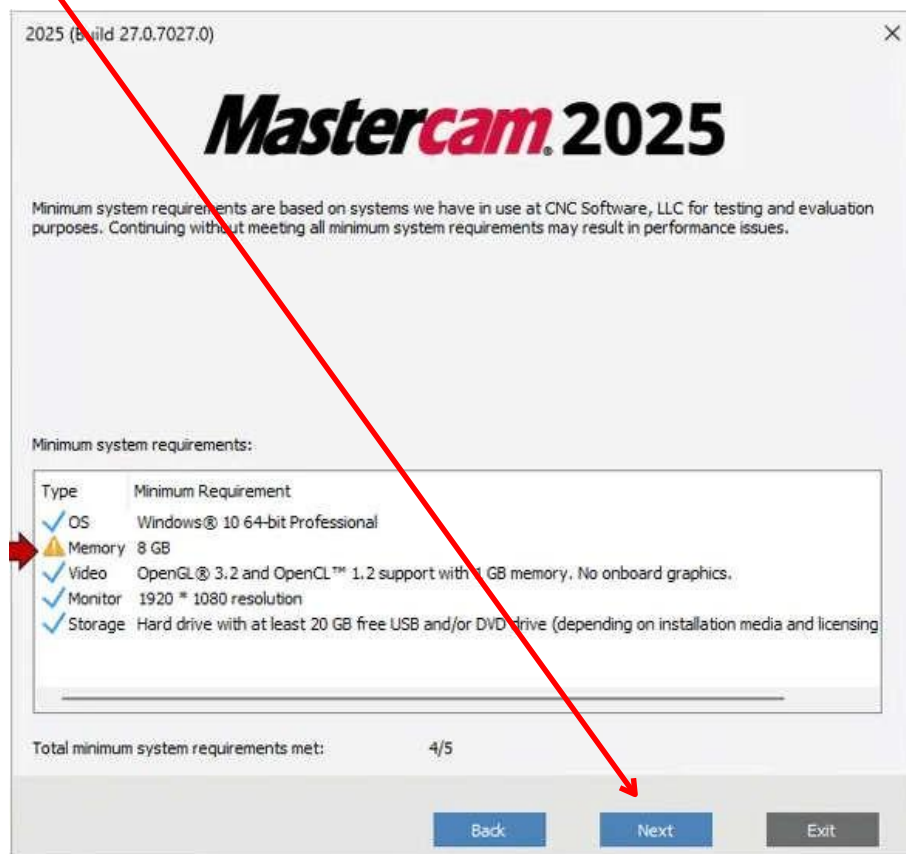
Step 3: Right click on the **mastercam.exe** file, you downloaded and select **Run as administrator**.



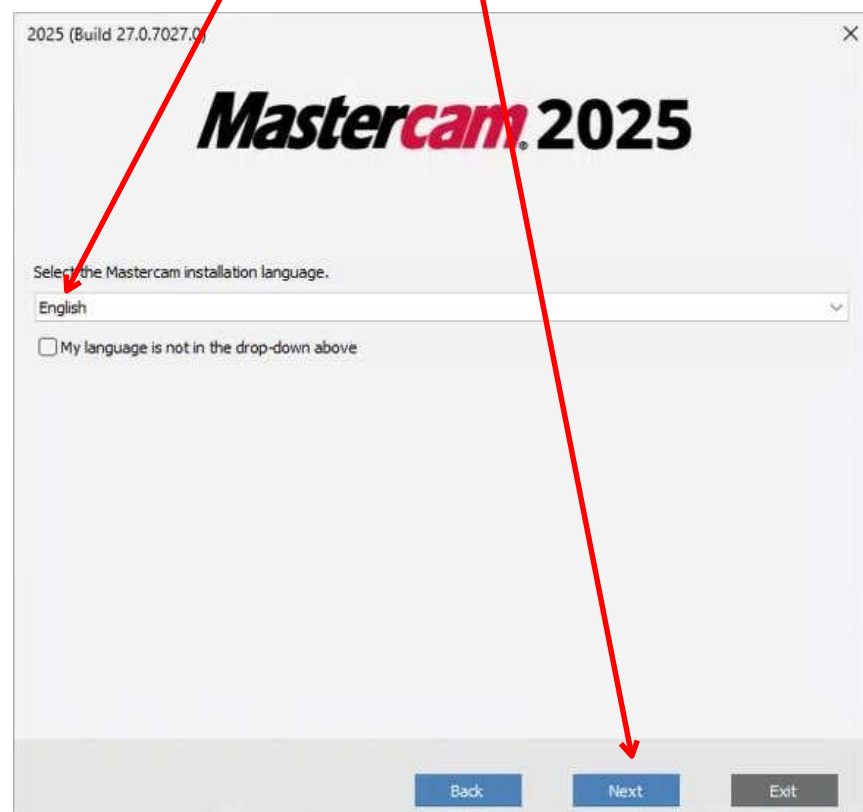
Step 4: Click on the **Mastercam <version> Installs** button.



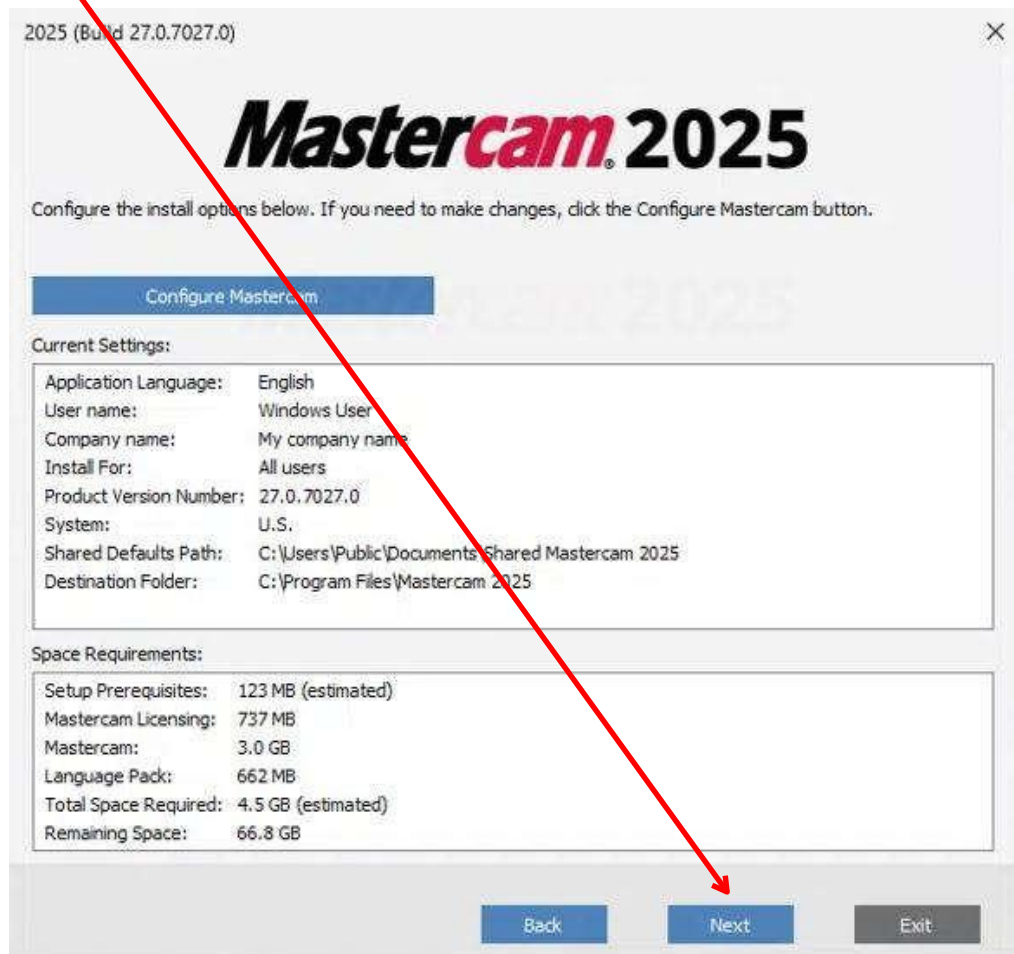
Step 5: Clicking on **Next** will proceed with the installation.



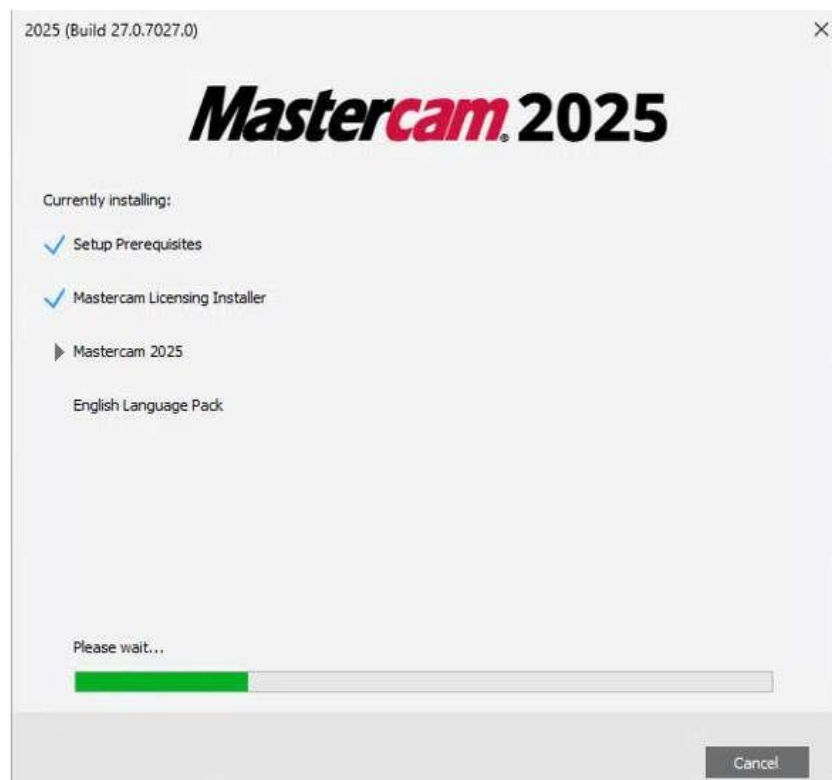
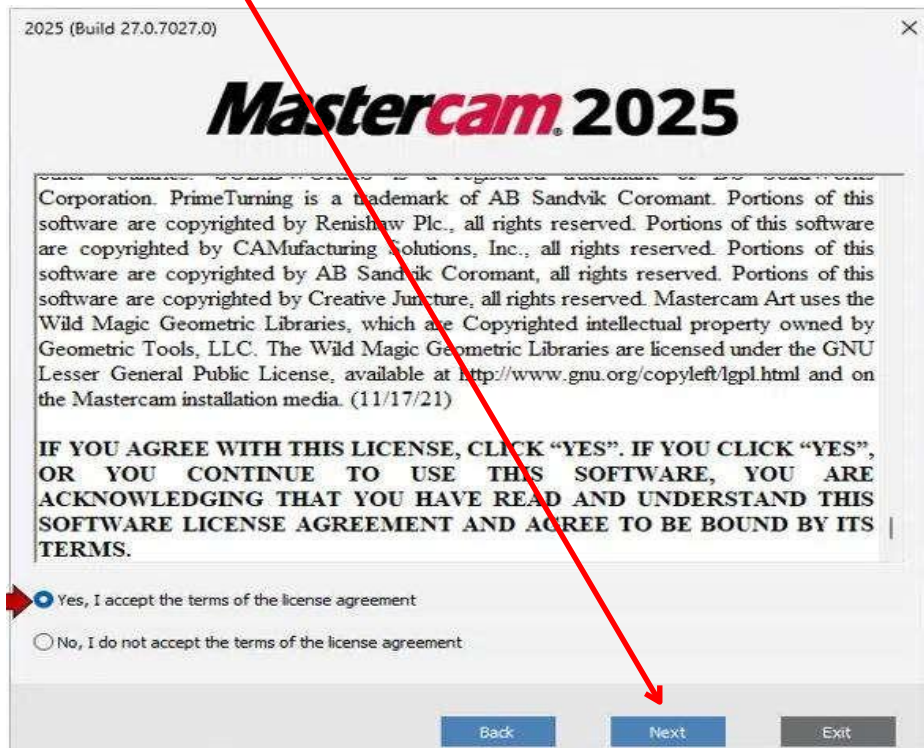
Step 6: Select the desired language **English** and click on **Next** to continue.



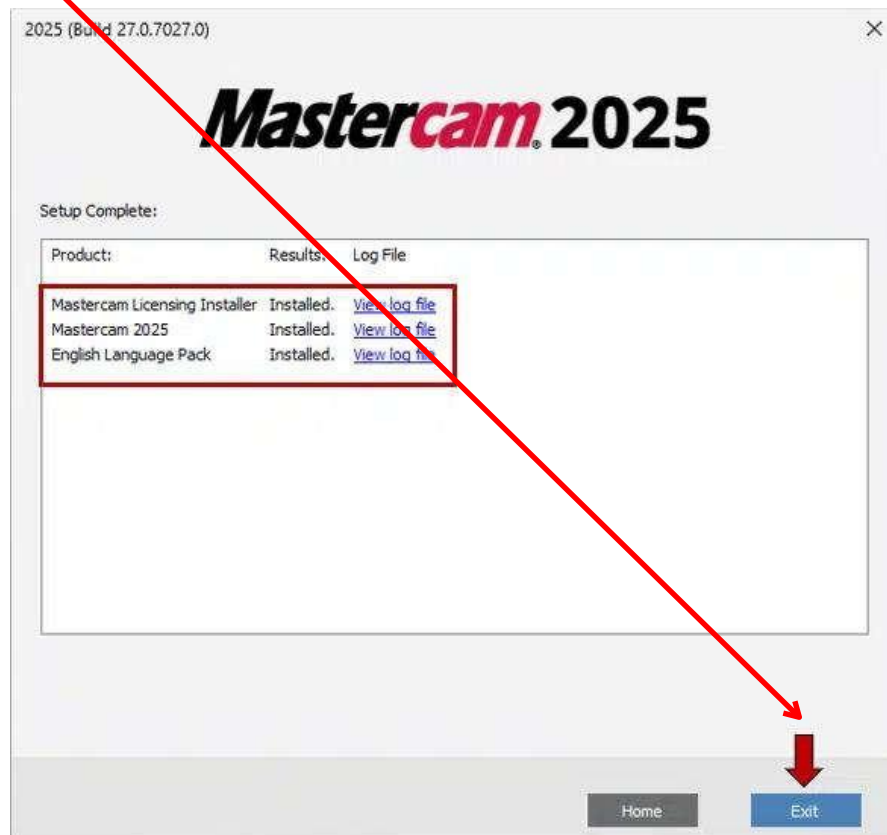
Step 7: Click on **Next** to continue.



Step 8: Select **Yes** and Click on **Next** to start the installation.

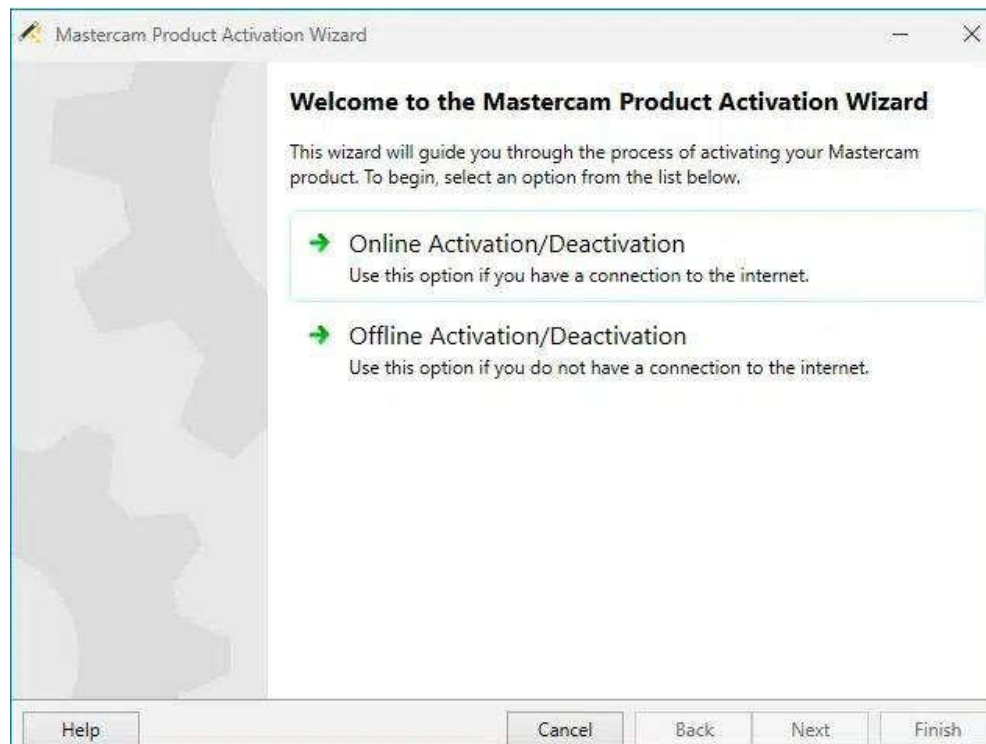
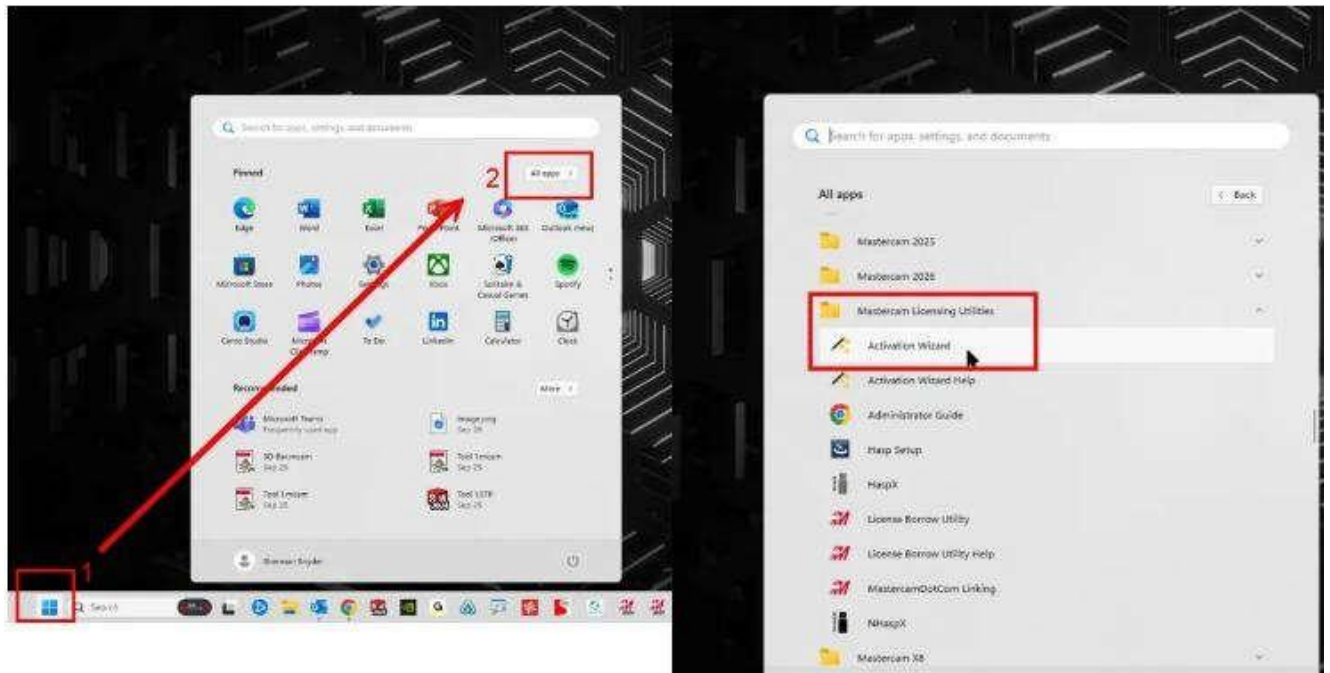


Step 9: Click on **Exit** to close the installer.



Step 10: Activation

To begin this process, you will need to find and launch the Mastercam Activation Wizard on the machine that Mastercam is installed on. The Activation Wizard can be found under your Start menu. Browse the **All-apps** list and expand the **Mastercam Licensing Utilities** folder.



Internet Access:

- Select **Online Activation/Deactivation**.
- Select **Activate a New License**.
- Accept the terms on the **Terms and Conditions** page, and then click **Next**.
- Enter in your **License number** and **Activation code** on the **Product Activation Code** page.
- Click **Next**.
- The Activation Wizard then checks your computer to see if it is connected to the Internet.
- The **Verify License Information** page displays the list of products that will be activated during this session. If the products are incorrect, please contact your local Mastercam Reseller.
- If the products listed are correct, select the checkbox to accept the terms, and click **Next**.
- The Activation Wizard begins the activation process. Upon successful activation, you may begin using Mastercam.

No Internet Access:

- Select **Offline Activation/Deactivation**.
- Select **Activate a New License**.
- Accept the terms on the **Terms and Conditions** page, and then click **Next**.
- Enter in your **License number** and **Activation code** on the **Product Activation Code** page.
- Click **Next**. On the **Offline Activation** page, select **Save**.
- The **Save As** dialog box displays.
- Save the license request file (.mcReq) to a network location, or a USB drive. Your Internet-connected computer must be able to access this file.
- On your Internet-connected computer, use a browser to go to **Activate.mastercam.com**.
- Select **Activate a license** and Select **Browse for your .mcReq file**. Navigate to the .mcReq file, and select **Open**.
- Select **Upload and Verify**.
- When your license is verified, your information, and list of products displays. Ensure that the information displayed is correct. If there are any issues, please contact your local Mastercam Reseller.
- Select **Download your .mcLic File**. Save the file to a network location or a USB drive.
- If you did not close the Activation Wizard, select **Next**. Then select **Finish Installing a New License**.
- If you did close the Activation Wizard, restart it, and select **Offline Activation/Deactivation, Finish Installing a New License**.
- On the **Install License** page, select **Open**.
- Navigate to your .mcLic file, and select **Open**.
- Select **Next**.
- Upon successful activation, you may begin using Mastercam.

Learning Outcome 4.2: Use CAM Software Interface



Content:

- CAM interface
- Home Tab
- Wireframe
- Solid
- Machine
- Plane
- Level



Assessment Criteria

1. Be able to use CAM interface.
2. Be able to work with different tabs.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. A computer set.
2. Mastercam software with license key.



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Use CAM Software (Mastercam) Interface	• Information Sheet: 4.2-1 • Self-Check: 4.2-1



Information Sheet: 4.2-1

Learning Objective: Use CAM Software (Mastercam) Interface

This section briefly touches on the elements that make up the Mastercam interface to give you a better understanding of how to use these elements to create and machine your parts.

1. Exploring Mastercam's interface

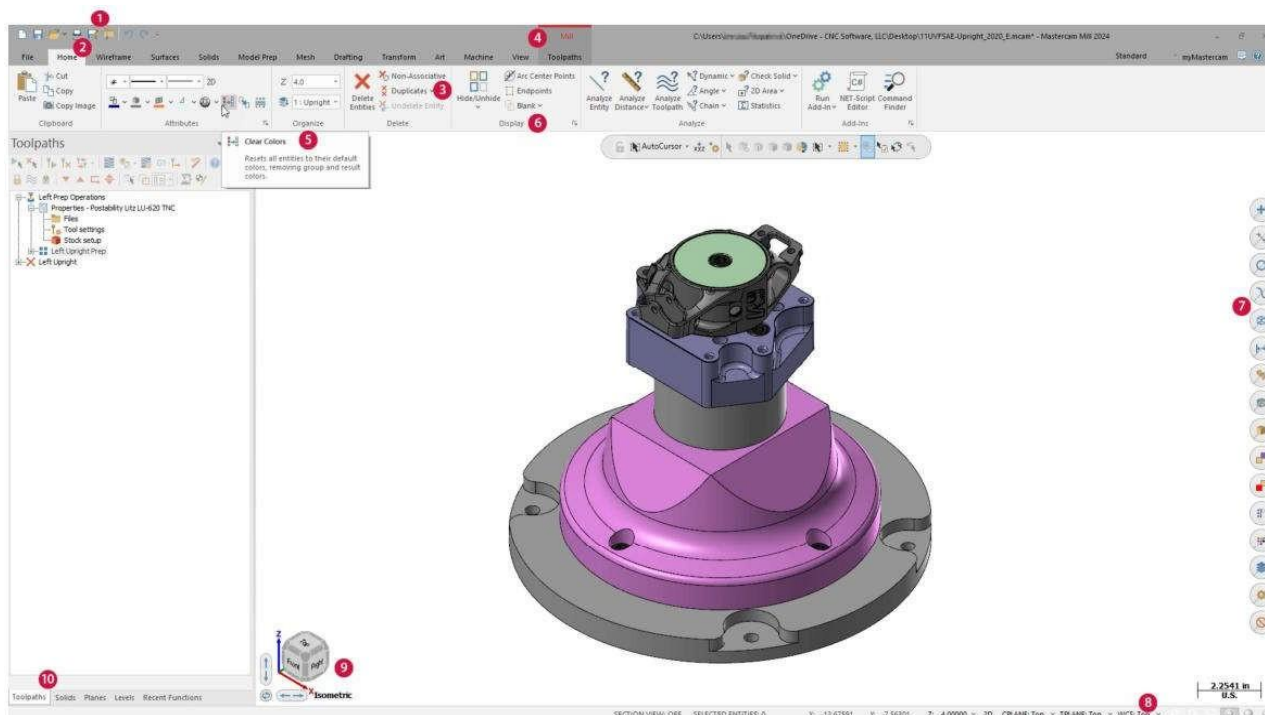
Double-click Mastercam's desktop icon.



Or,

Launch Mastercam from the Windows **Start** menu.

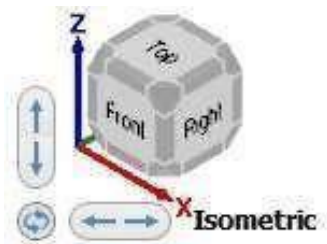
The following image displays the elements that make up the Mastercam interface. The corresponding sections provide brief descriptions of each element and its use.



1. **Quick Access Toolbar (QAT):** A customizable set of commonly used functions that are always available in the interface. The QAT can be located above or below the ribbon.
2. **Tab:** Groupings of related controls. Tabs are organized in a simple-to-complex workflow from left to right.
3. **Tab Group:** A region of the tab that contains a set of related controls.
4. **Contextual Tab:** A tab that displays when you make a specific selection in Mastercam. A contextual tab presents controls and commands relevant to your current activity.
5. **Tooltip:** A small window with descriptive text that displays when you hover over a command or control.
6. **Selection Bar:** A toolbar that combines Auto Cursor controls and general selection tools used to select entities in the graphics window. There are two selection modes, Standard Selection and Solid Selection, that are activated based on the function you are using.
7. **Quick Masks:** A group of controls that helps you to select *all* entities of a certain type, or to select *only* entities of certain type. Most Quick Mask controls are divided in half. Clicking the left or right side of the control toggles between selection modes.

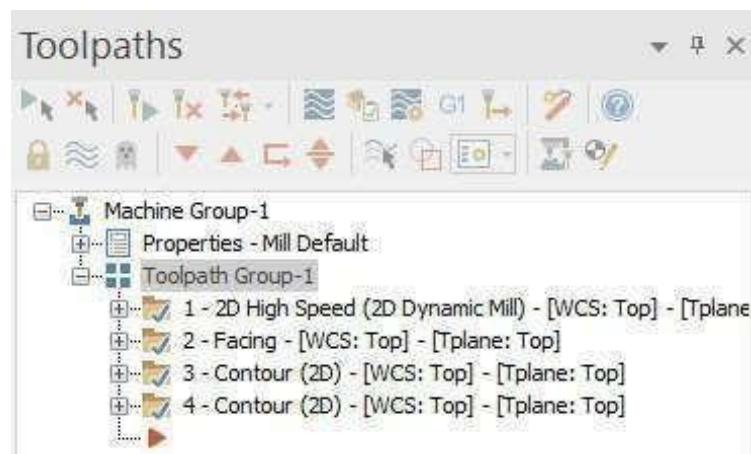
8. Status Bar: A bar across the bottom of the workspace that shows the coordinate position of the cursor and provides quick access to modify planes, construction mode, and Z depth. The right side of the Status Bar has wireframe, shading, and translucency controls that change the appearance of your part. The left side of the Status bar indicates the number of selected entities and the status of Section View.

9. G-View Cube: An augmented and more graphical way to show part and plane orientation. Click directly on the different faces and sides of the cube to control the view and on -screen orientation of your part. The Gview cube is relative to the WCS by default.



10. Managers: Managers include controls for toolpaths, solids, planes, levels, recent functions, and Art. The Managers can be rearranged to suit your workflow.

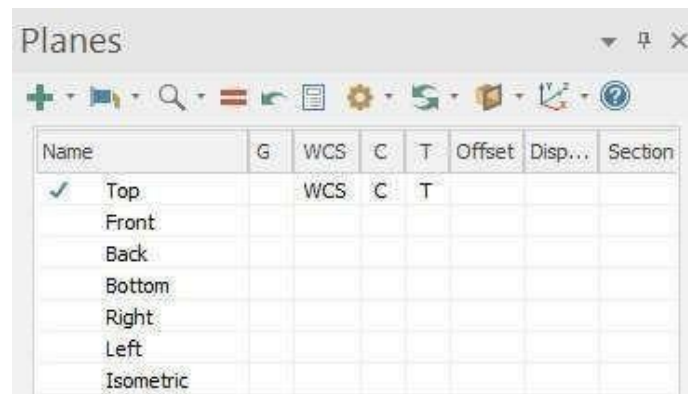
a) **Toolpaths Manager:** Lists the toolpath groups and machine types for the current file. Use the Toolpaths Manager to control your operations.



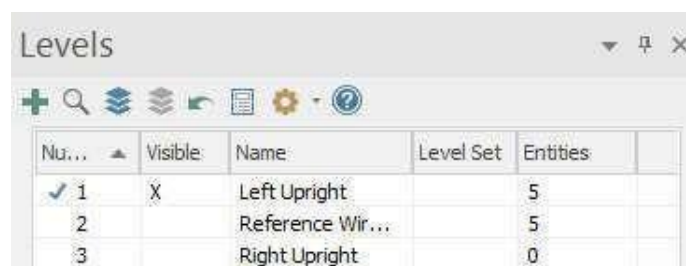
- b) **Solids Manager:** Lists each solid in the current file along with its operation history and associated toolpaths. Use the Solids Manager to edit solids and their operations.



- c) **Planes Manager:** Shows the planes in the current file. Use the Planes Manager to adjust planes and work offsets.



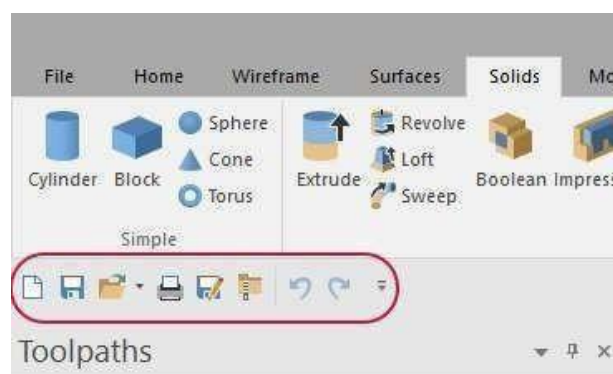
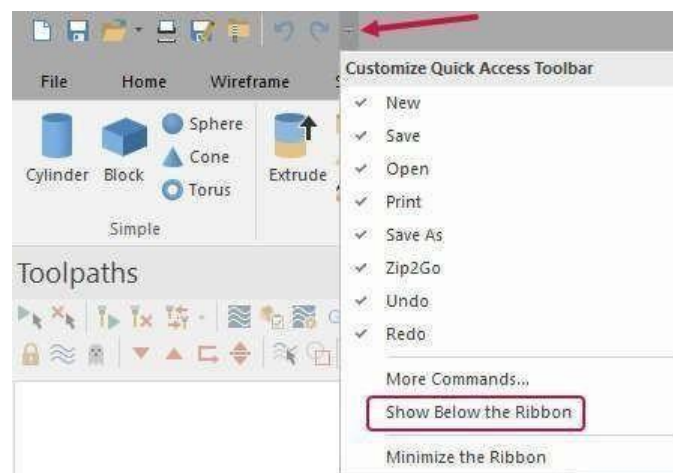
- d) **Levels Manager:** Shows the levels in the current file. Use the Levels Manager to control your levels.



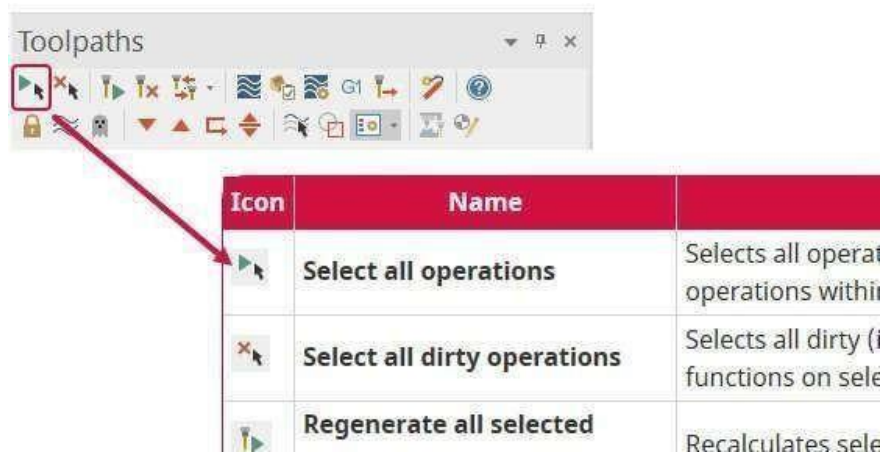
11. Exploring Mastercam's Backstage

Ico	Option	Icon	Optio
	OK and Create		Cance
	OK/Sav		Hel
	Appl		About

12. Customizing the Quick Access Toolbar: Click the **Customize Quick Access Toolbar** drop-down, and select **Show Below the Ribbon**.



13. Toolpaths:

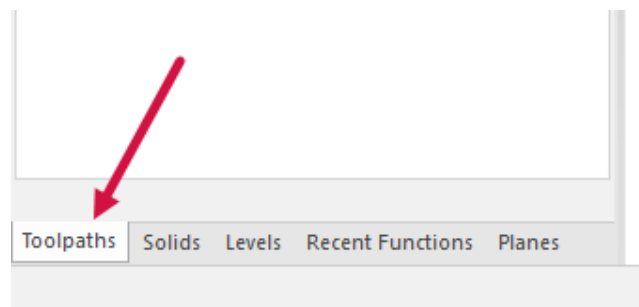


14. Moving and Docking Managers:

Mastercam has several movable, dockable managers that provide flexible access to frequently used functions including toolpaths, solids, planes, levels, and recent functions.

You can toggle the display of the managers, as well as move one or more managers to a new location.

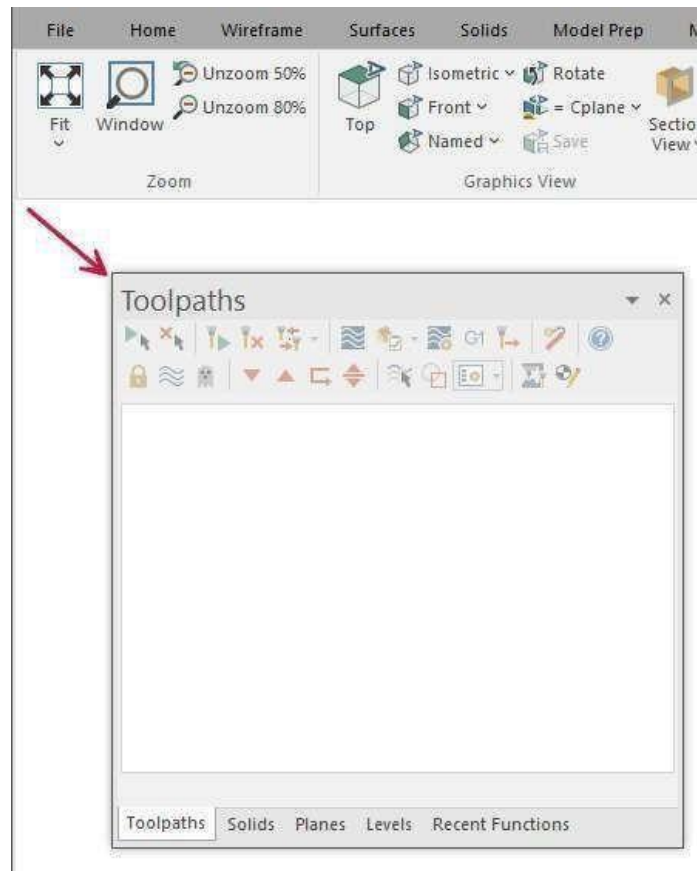
- Select **File, New** to start with a new part file. Select **Don't Save** if you are prompted to save changes to the current Mastercam part file.
- Click the **Toolpaths** tab to bring the Toolpaths Manager to the front.



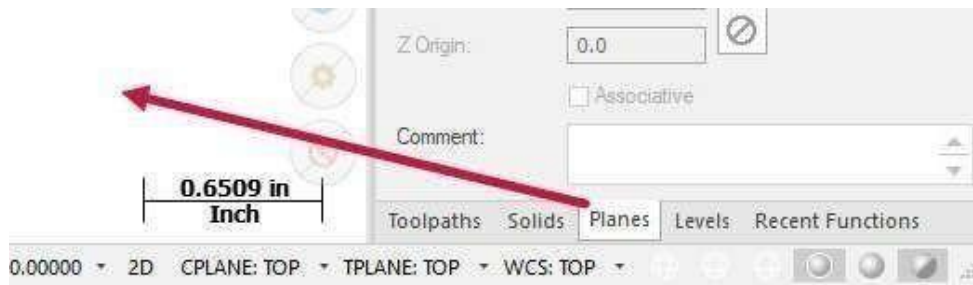
- Hide the Toolpaths Manager by clicking the **Close** button in its upper right corner.



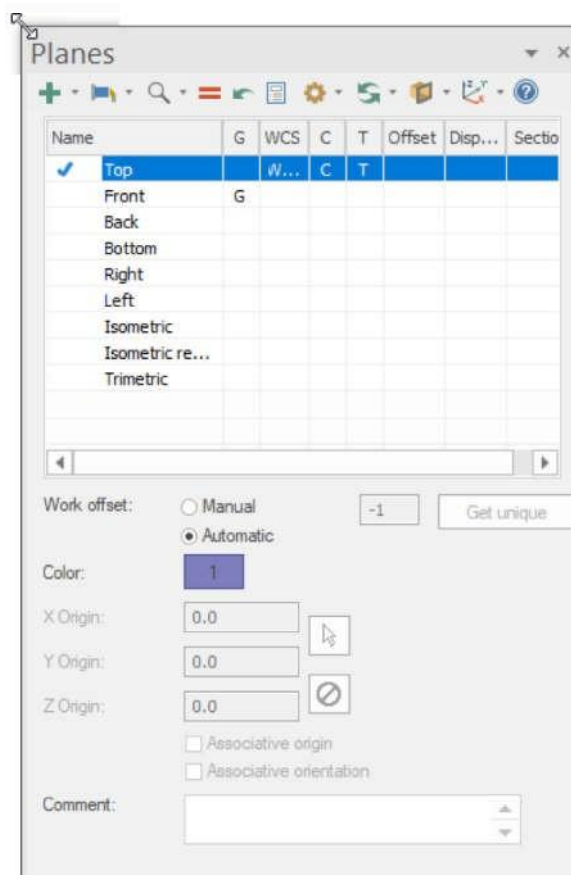
d) Select the title bar of the **Toolpaths Manager**, and drag the panel of managers into the graphics window.



15. Click the **Planes** tab, and drag and drop it anywhere on the workspace (except on a docking icon).



16. Resize the Planes Manager by clicking and dragging any of its edges.



- Double-click the Planes Manager's title bar to redock it on top of the stack of managers.
- Click the title bar of the stack of managers, and drag it to the left side of the graphics window, where it was docked at the beginning of this exercise.
- If you work with multiple monitors, you can position the managers onto the monitor that is not running Mastercam to free the entire graphics window for drawing.



Self-Check: 4.2-1

1. Name the main areas of the Mastercam user interface.
2. What is the function of the Toolpath Manager?
3. What is the function of the Operations Manager?
4. What are the key machining parameters in Mastercam?



Answer Script: 4.2-1

3.
 - Ribbon toolbar
 - Graphics window
 - Toolpath manager
 - Status bar
 - Operations manager
 - Property bar
4. It lists all toolpath operations and allows users to edit, regenerate, or post-process them into G-Code.
5. It controls toolpaths, stock setup, machine group, and post-processor settings.
6.
 - Spindle speed (RPM)
 - Feed rate (mm/min or inch/min)
 - Plunge rate
 - Depth of cut (axial & radial)
 - Step-over
 - Tool number and tooltype

Learning Outcome 4.3: Carry Out 2D CAM Programming



Content:

- Milling 2D CAM operations (Facing, Contour, Pocket etc.)
- Lathe 2D CAM operation (Facing, Turning, Drilling, Boring, Threading)
- Tool path generation
- Simulation
- Code generation



Assessment Criteria

5. Be able to analysis CAD file.
6. Beable to perform work holding and stock definition.
7. Be able to perform Machine Origin (work zero) setup.
8. Be able to select Tool.
9. Be able to perform simulation, verification, and G-code generation.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. A computer set
2. Mastercam software
3. Job sheet with proper drawing and instructions



Activity/Work

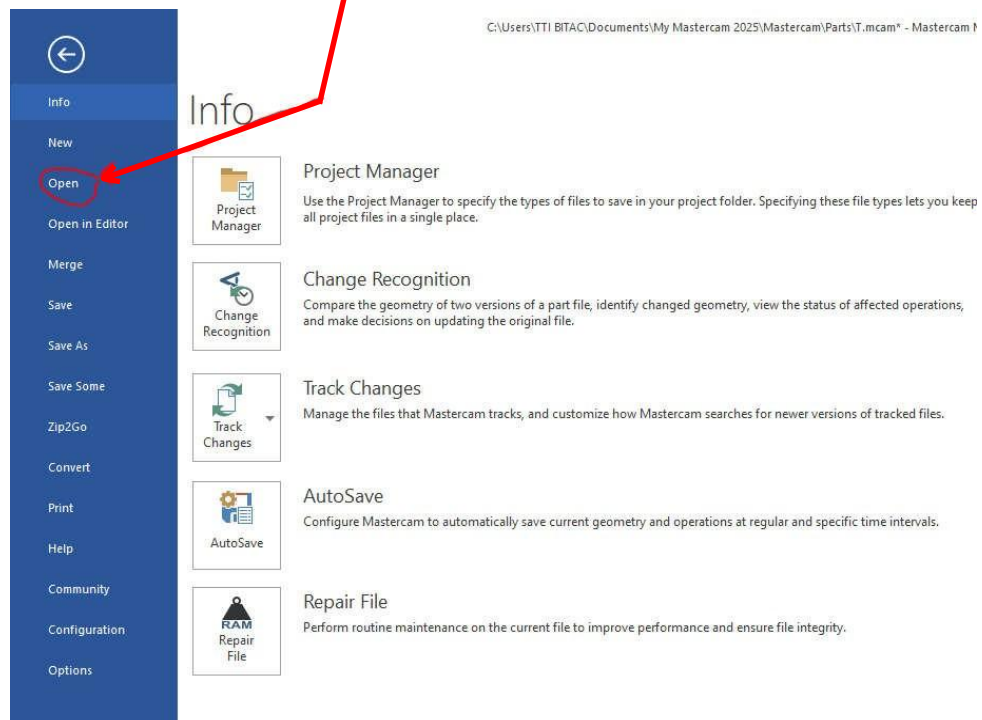
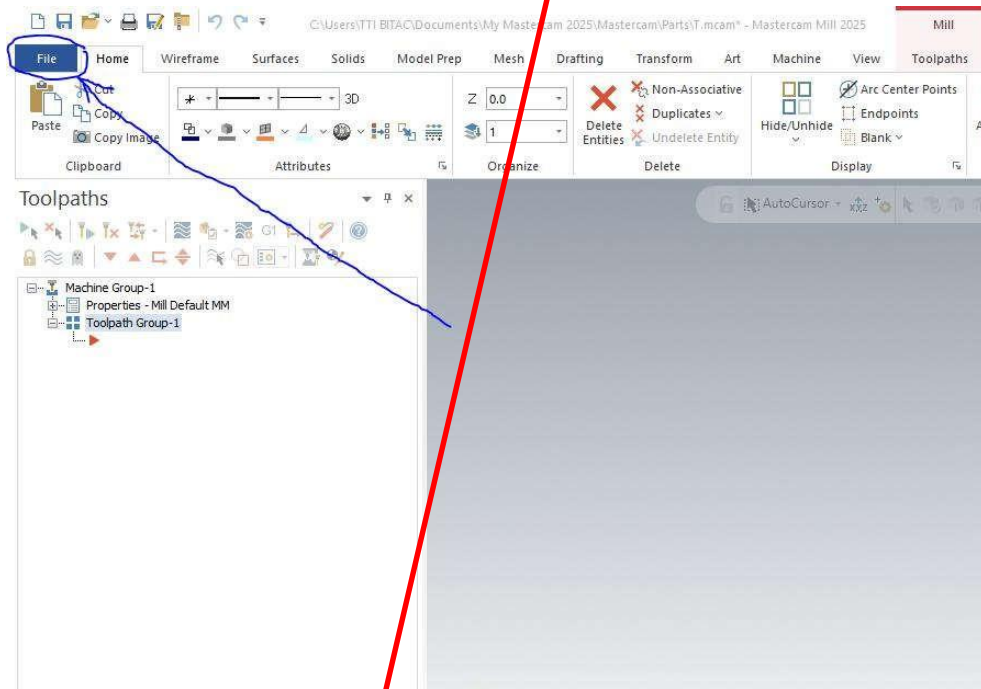
Activity/Work	Resource/Special Instruction/Reference
Carry Out 2D CAM Programming	• Information Sheet: 4.3-1 to 4.3-8 • Self-Check: 4.3-1 to 4.3-4 • Answer Script: 4.3-1 to 4.3-4 • Self-Check: 4.3-5 to 4.3-8 • Answer Script: 4.3-5 to 4.3-8 • Job Sheet: 4.3 (a) to 4.3 (b)



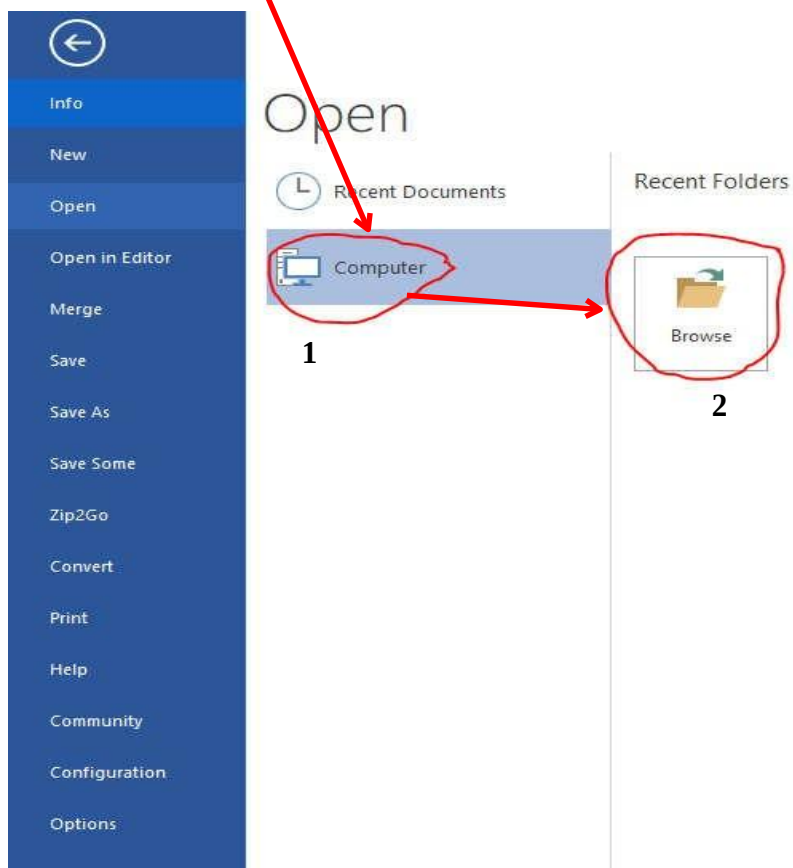
Information Sheet: 4.3-1

Learning Objective: Carry Out Milling 2D CAM Programming (Facing).

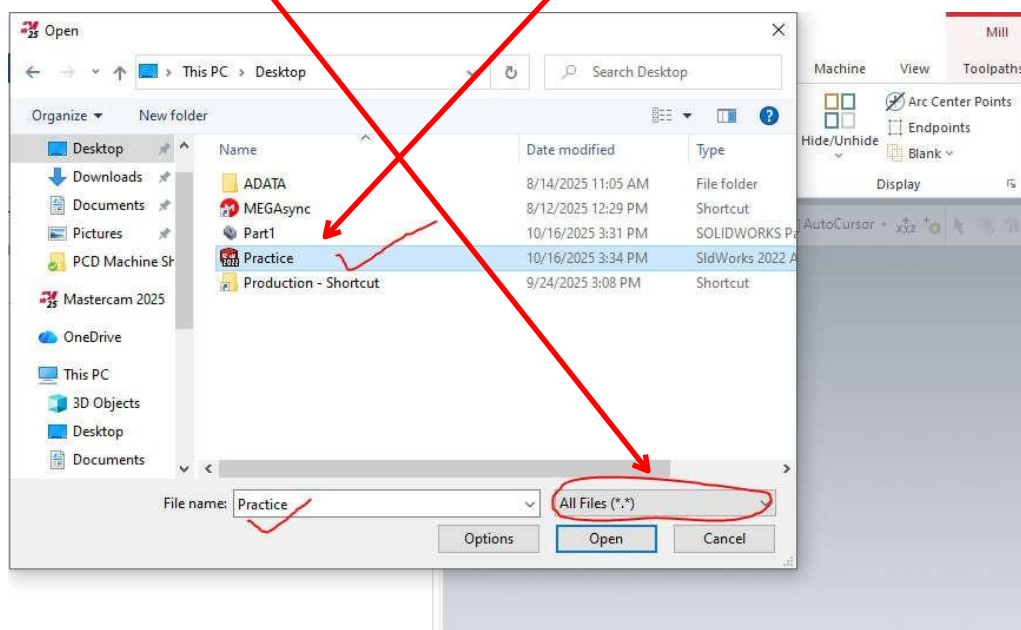
Step 1: Open Mastercam, go to **File** and click on **Open.**



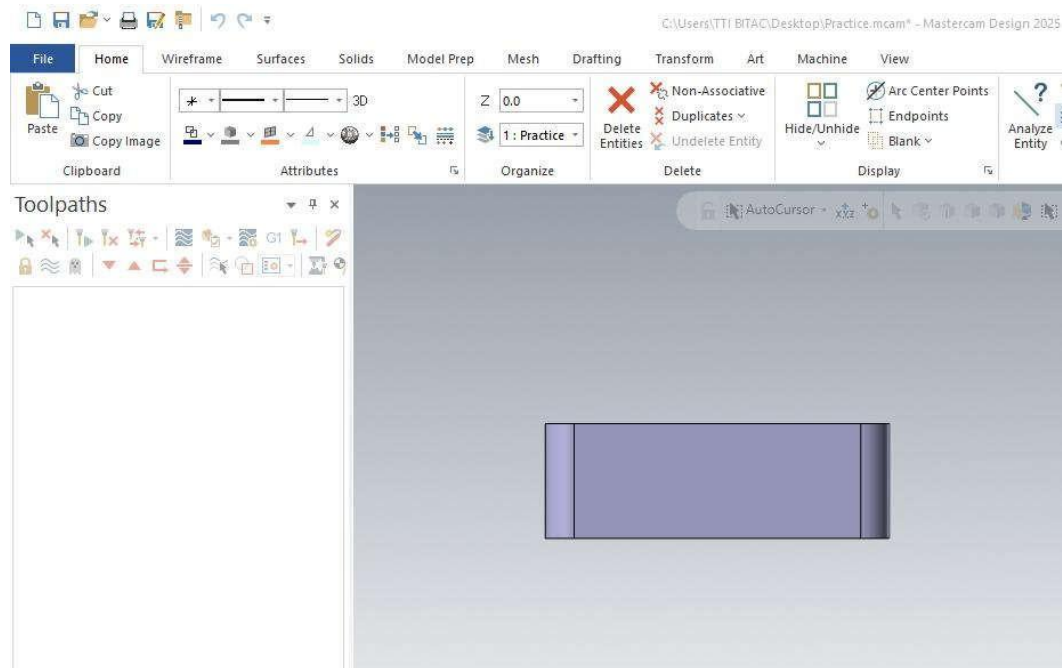
Step 2: Click on **Computer and Browse.**



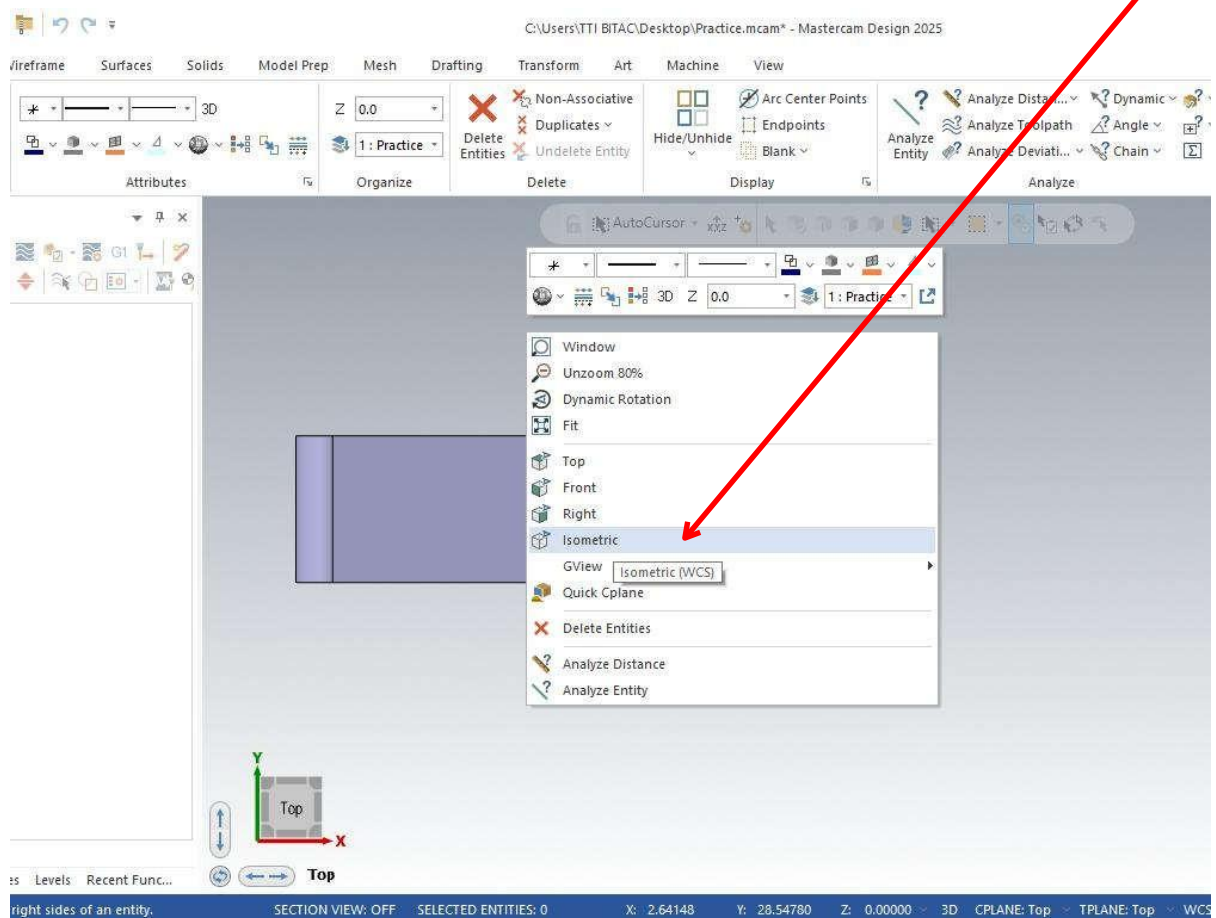
Step 3: Import Mastercam supported part file (.stepAP2014/ .stepAP203/ .igs/ *.x_t). In the file format select **All Files**



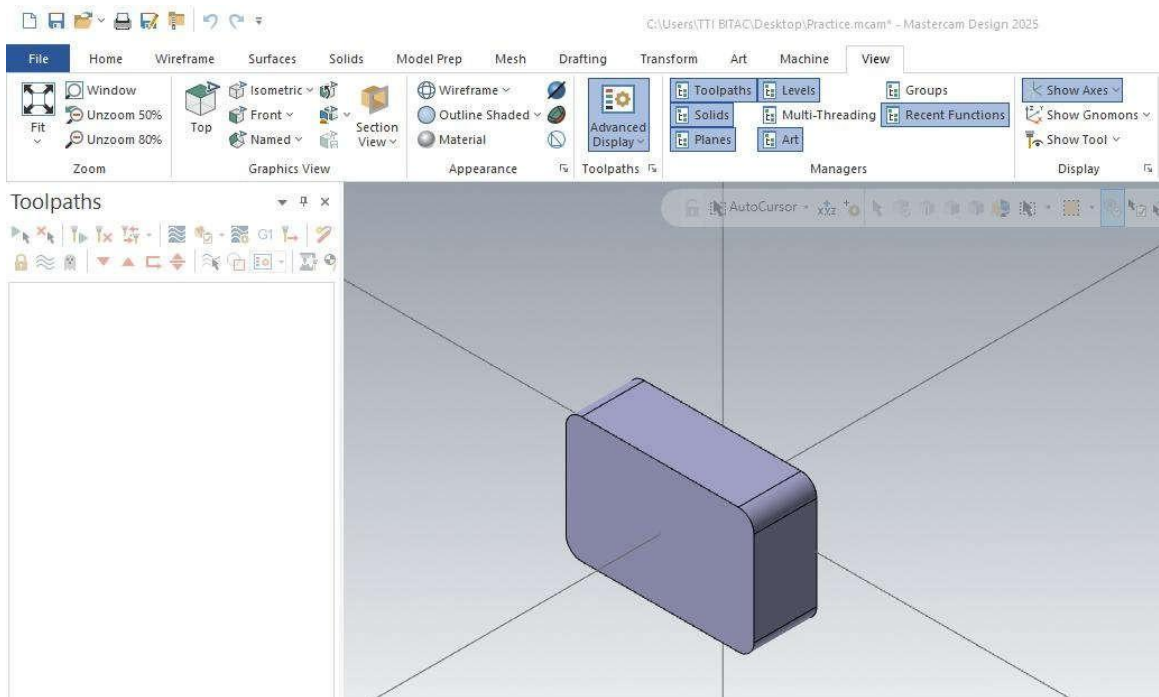
Step 4: Now the file **Practice** is in the Mastercam interface



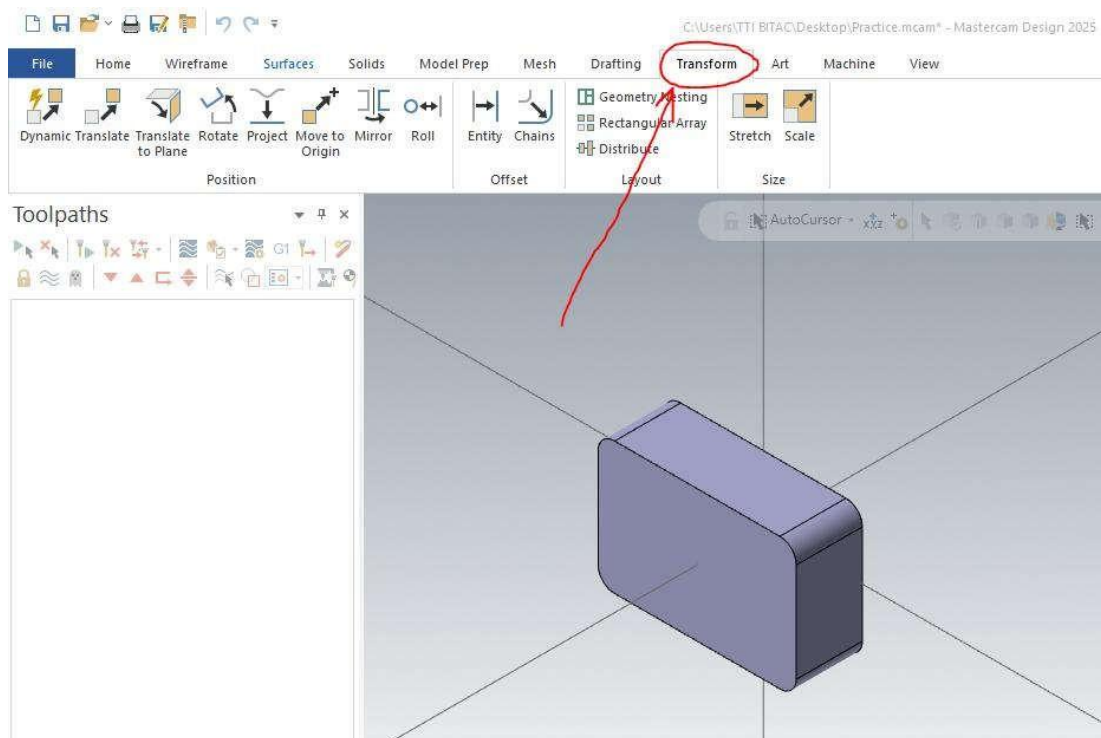
Step 5: Place mouse in the middle screen and click right button of the mouse. Then select **Isometric**.



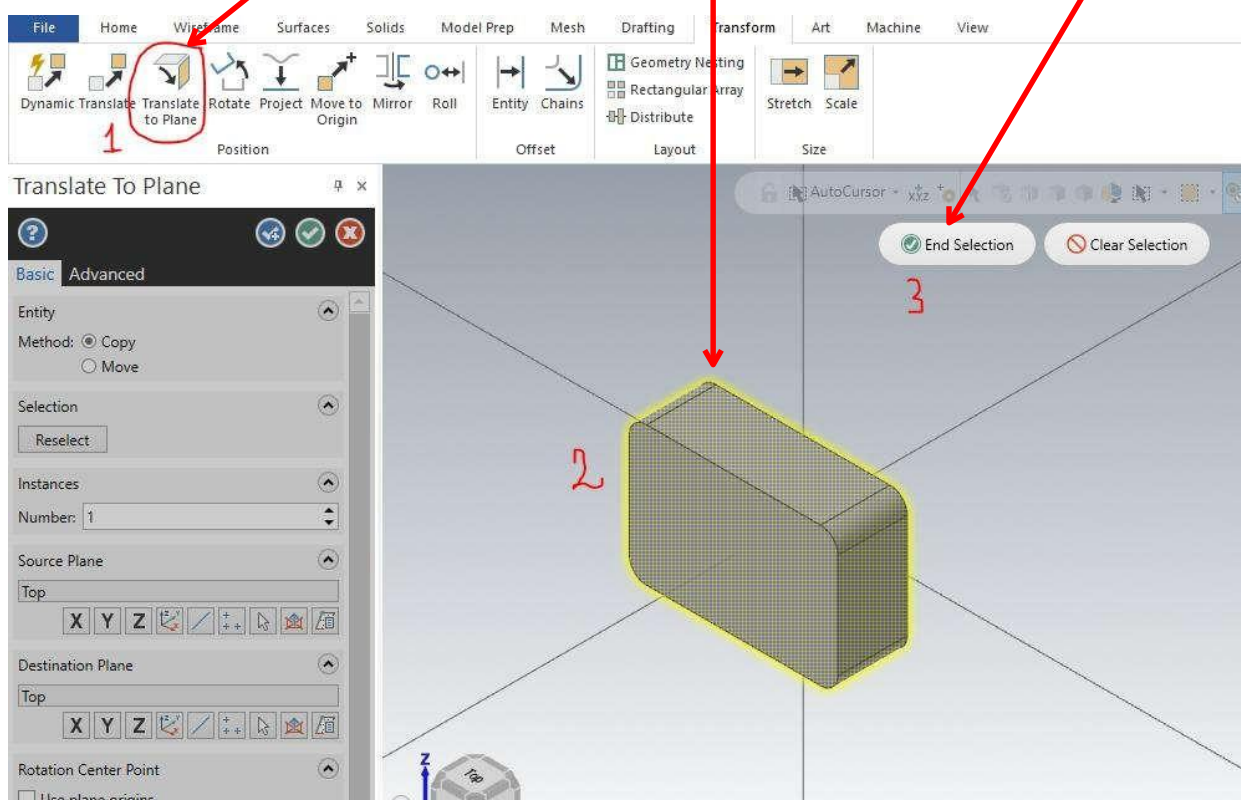
Step 6: In the Isometric (WCS) view the orientation of the job is seen clearly. Now we need to make proper orientation of the job for manufacturing in the machine.



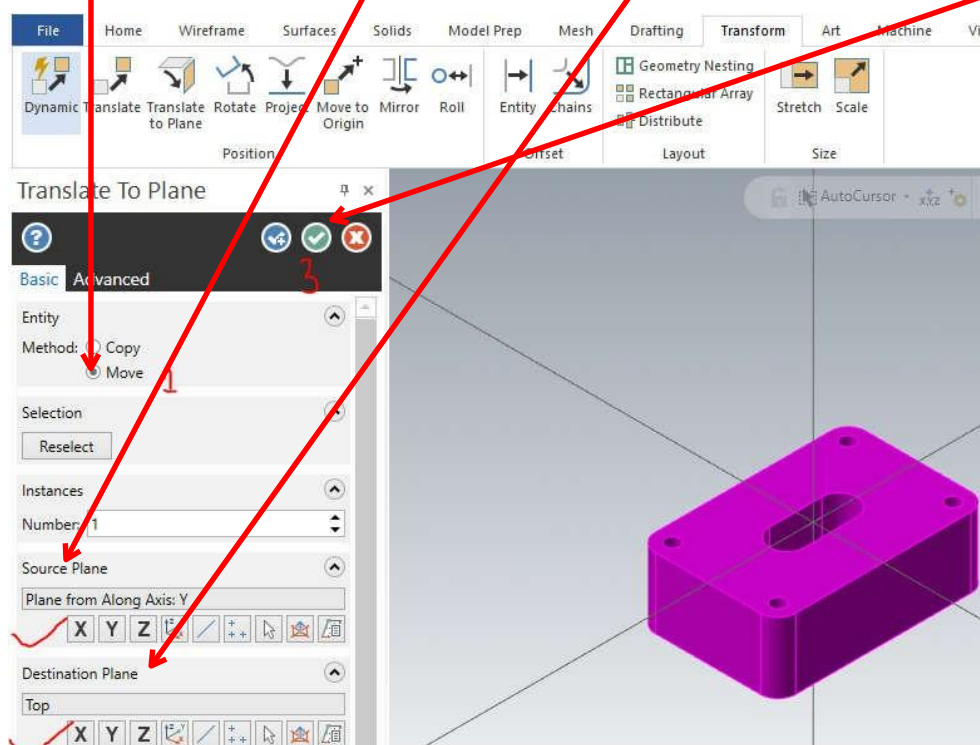
Step 7: Click on **Transform Tab.**



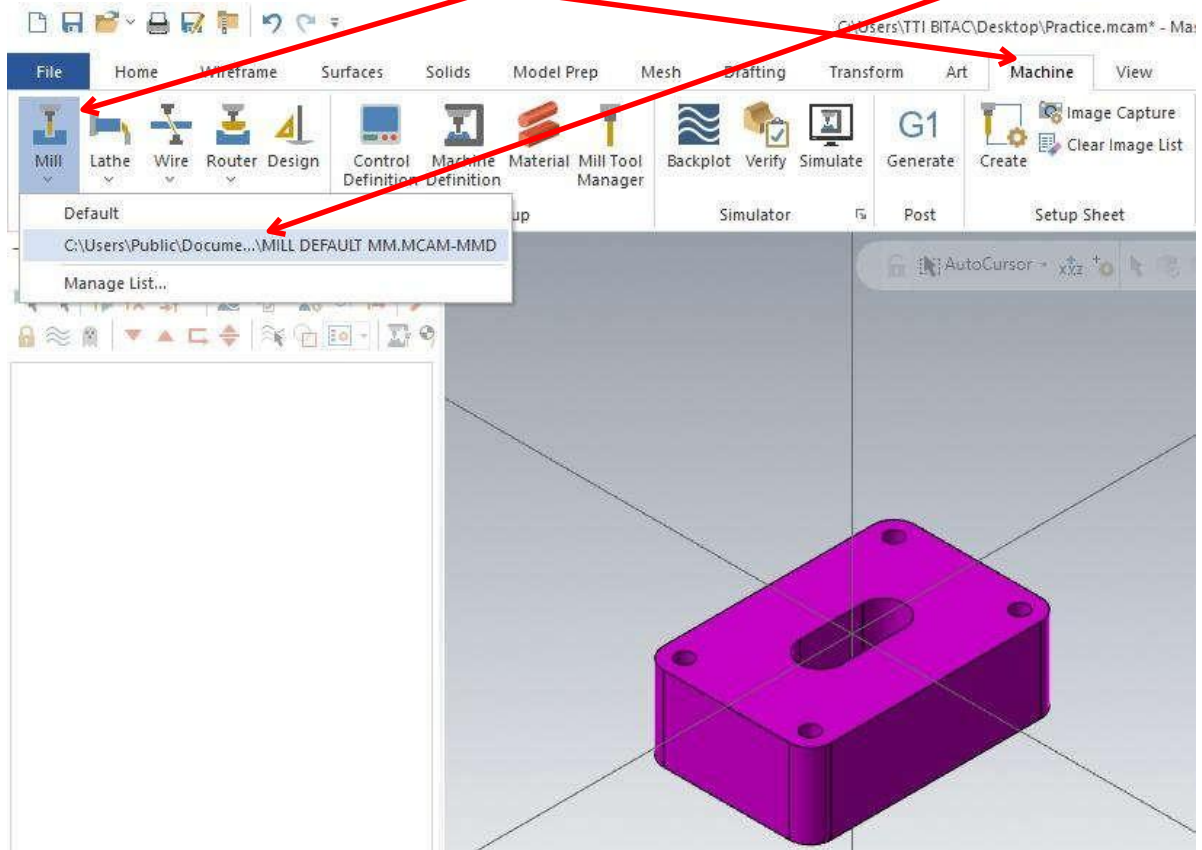
Step 8: Click **Translate to Plane** then select the **Full Body** and finally click the **End Selection**.



Step 9: Select **Move**, then choose **Source plane** and **Destination Plane**. Finally click on **Ok**.



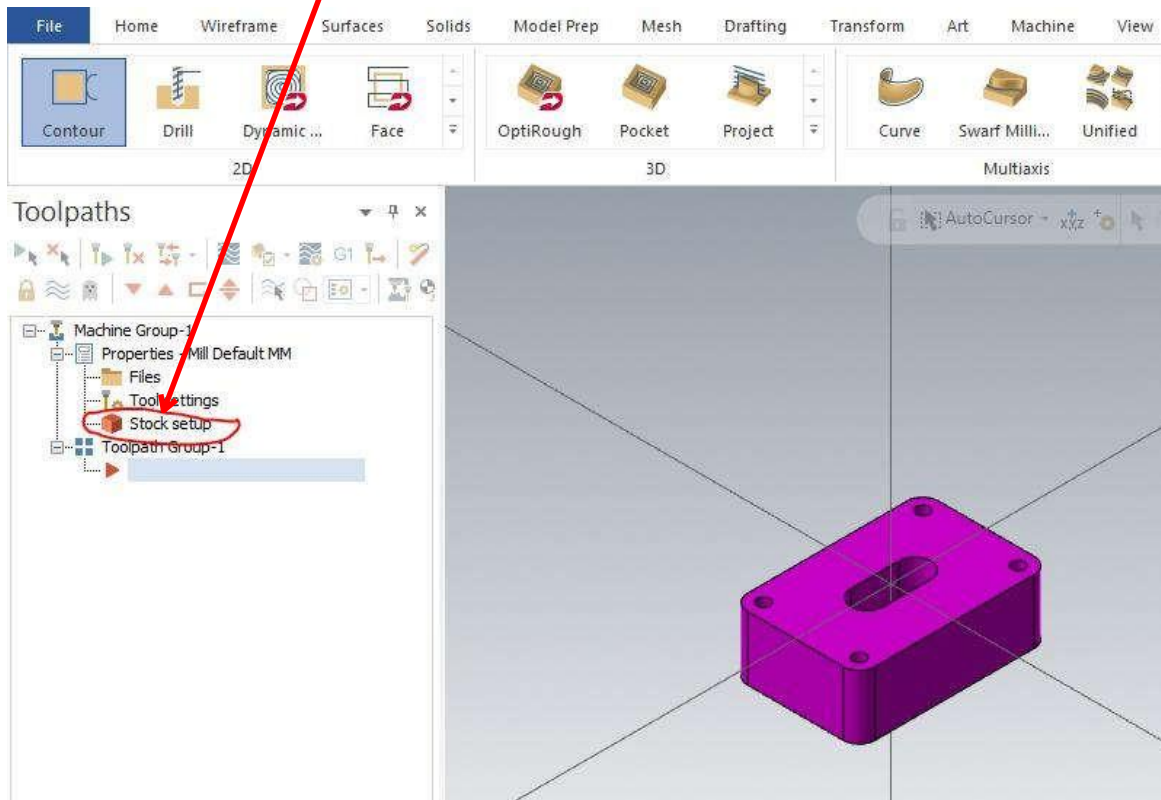
Step 10: Click on **Machine Tab**, click on **Mill**, then select **Mill DEFAULT MM.MCAM-MMD.**



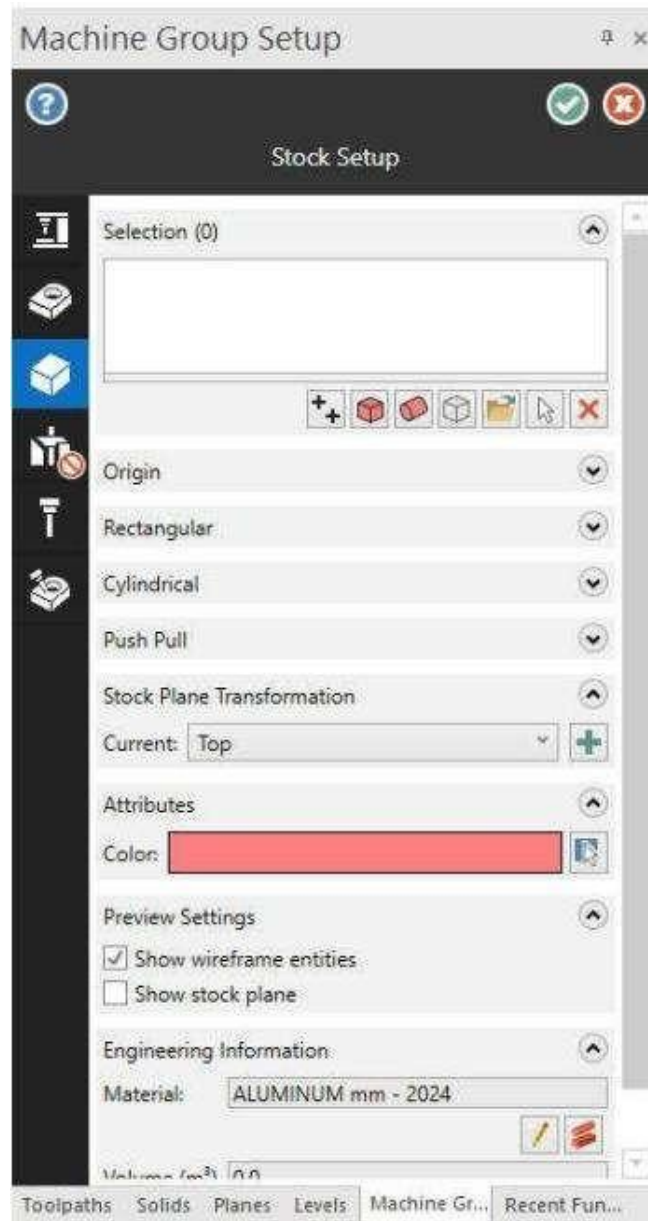
Step 11: Click on **Machine and Stock setup**. Go to the Toolpaths Manager. Machine Group-1 is present because you selected a Mill machine during the chapter on selection. Machine groups organize your cutting operations by machine.

a) **Expand** Properties - Mill Default.

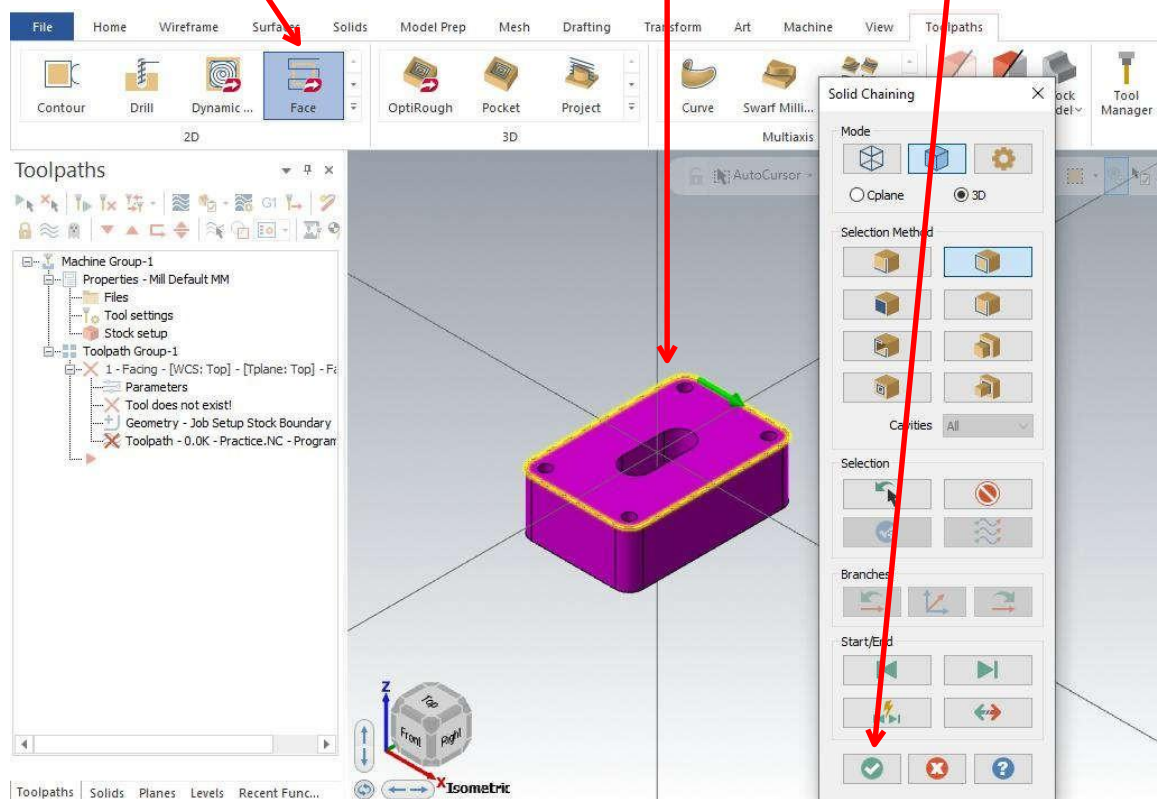
b) **Click** Stock setup



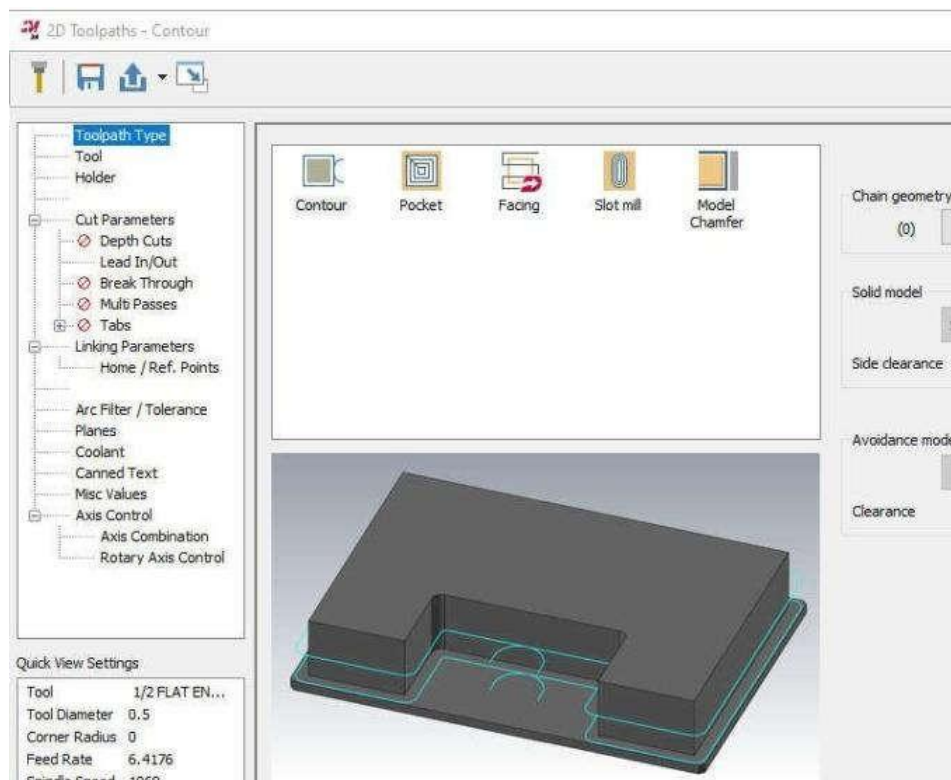
- c) Select machine files, tool and operations libraries, and operation defaults
- d) Define the final part (the workpiece) after all machining is complete
- e) Configure stock shape, origin, and wireframe



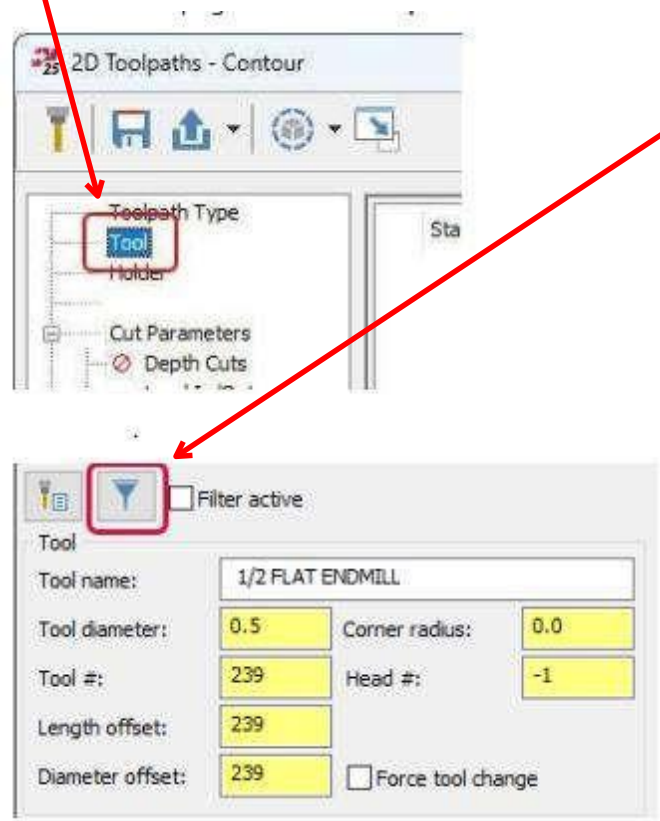
Step 12: Click on **2D Mill Face** operation and select **Chain**. Finally click on **Ok**.



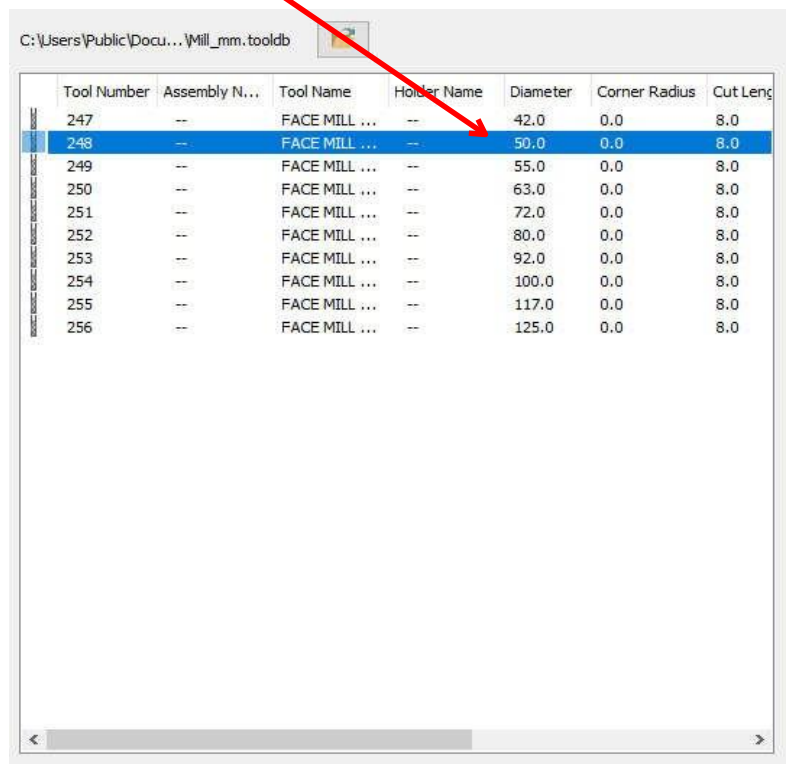
Step 13: This is where you program your toolpath.



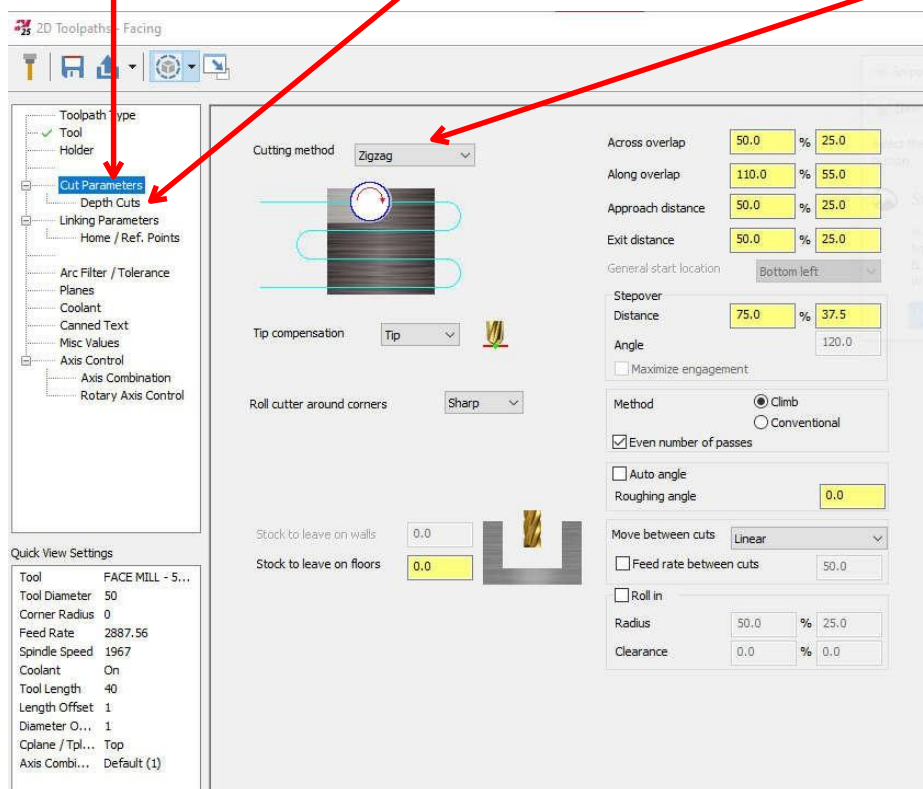
Step 14: Click on the **Tool** page in the **2D Toolpaths – Face** dialog box. Then click on **Filter.**



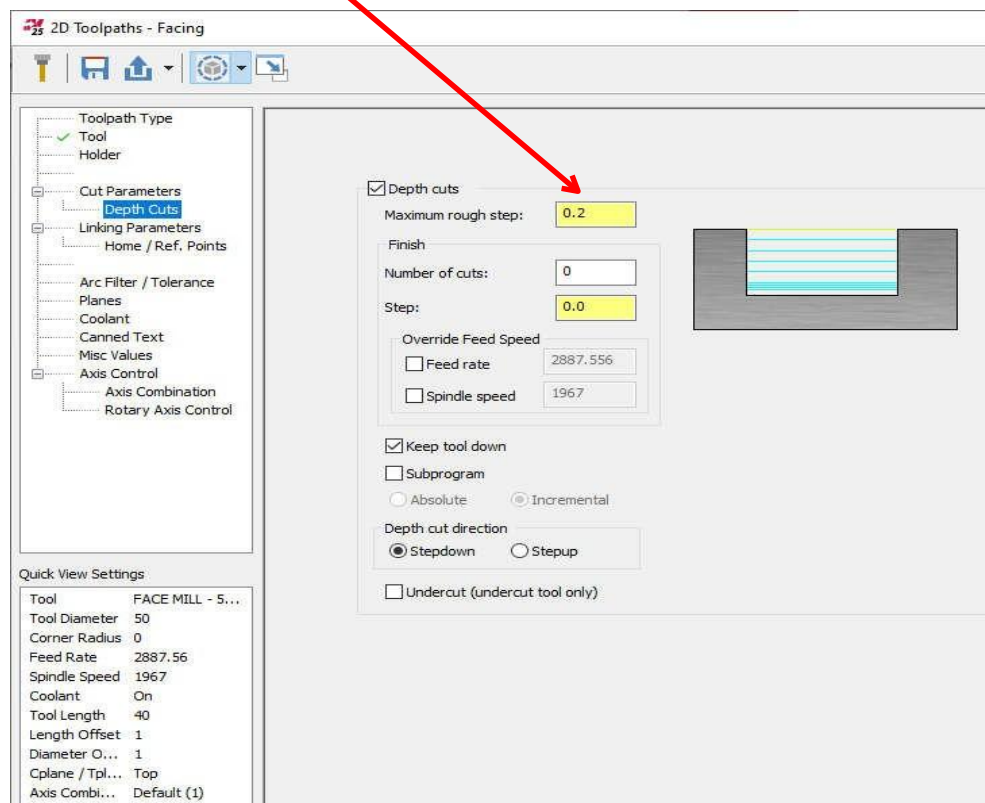
Step 15: Select Cutting Tool: **50mm Face Mill** from **Tool Library**.



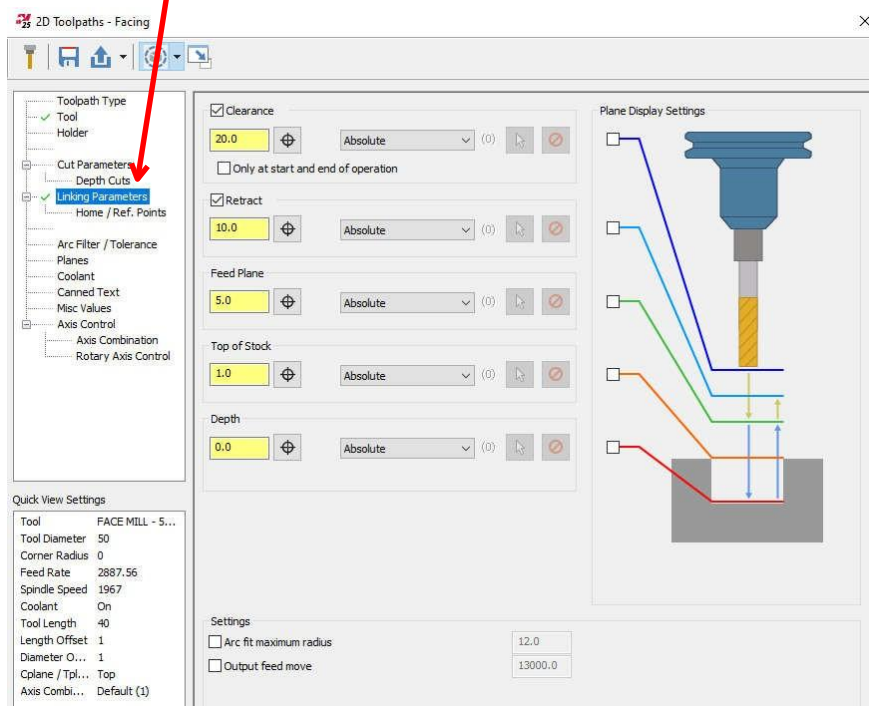
Step 16: Select **Cut Parameters** and **Depth of Cuts**. Choose Cutting method **Zigzag**



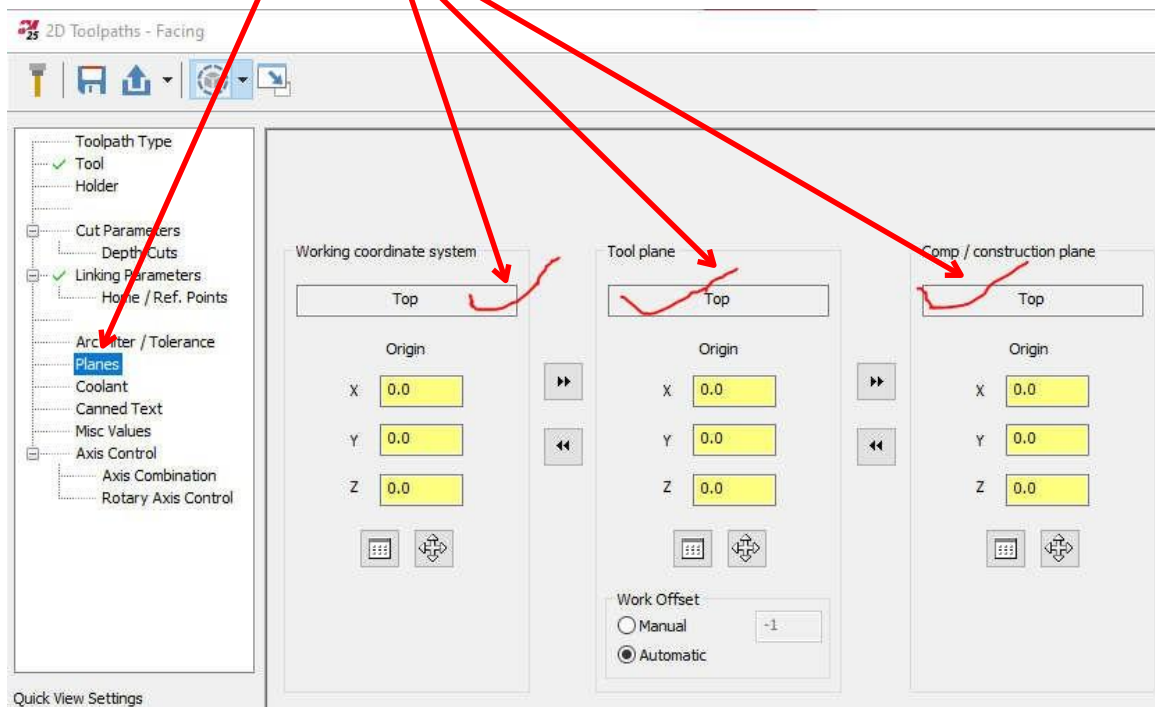
Step 17: Change **Max rough step to 0.2**. This is the maximum amount of material to remove with each rough cut.



Step 18: Select **Linking Parameters**. Enter **20.0** and select **Absolute** for **Clearance. Retract:** Enter **10.0** and select **Absolute. Feed Plane:** Enter **5.0** and select **Absolute. Top of Stock:** Enter **1.0** axis and select **Absolute** to measure from the origin 0,0,0. **Depth:** Enter **0** and select **Absolute** to set the final machining depth.



Step 19: Ensure all **Planes** are **Top**. Finally click on **Ok**.

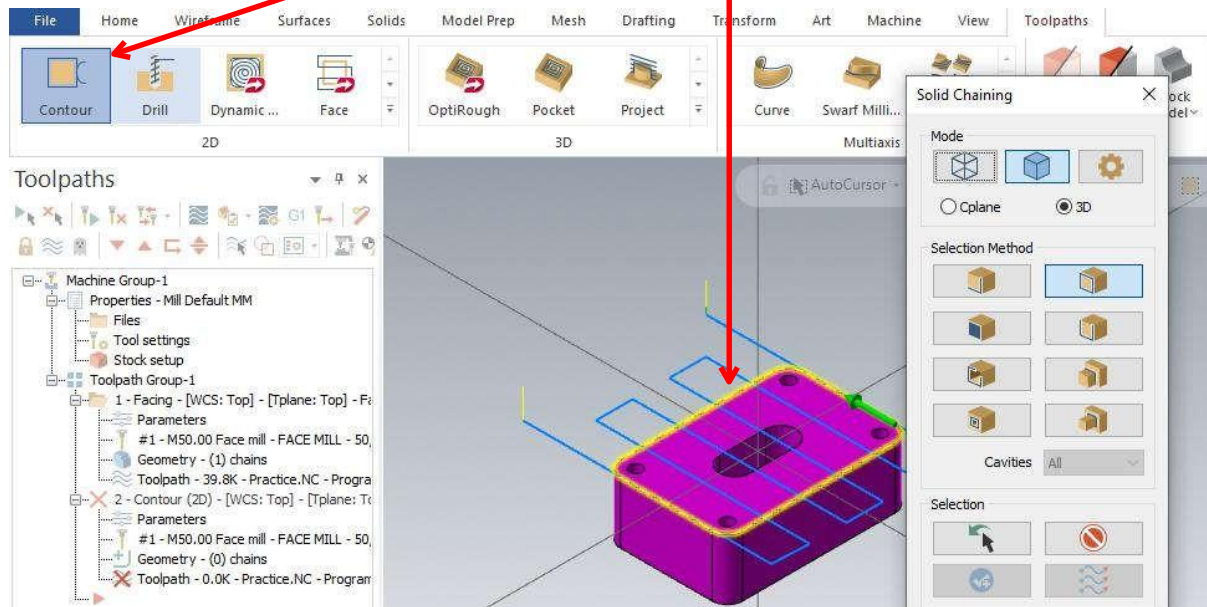




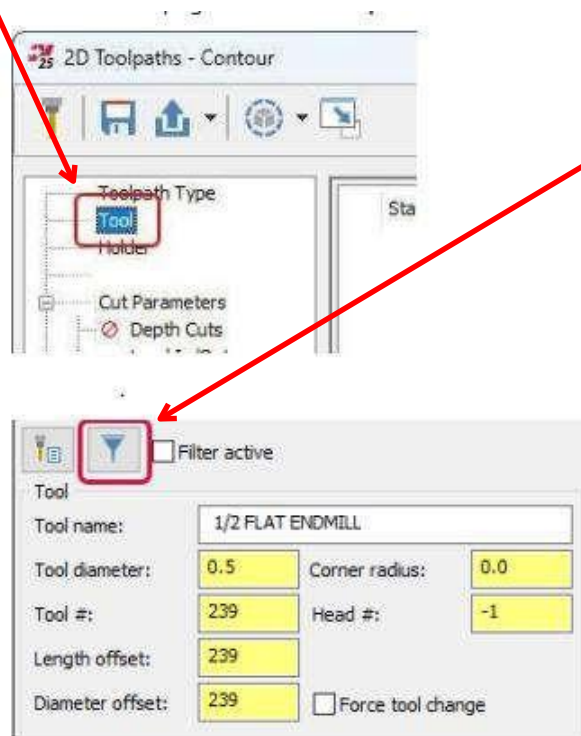
Information Sheet: 4.3-2

Learning Objective: Carry Out Milling 2D CAM Programming (Contour)

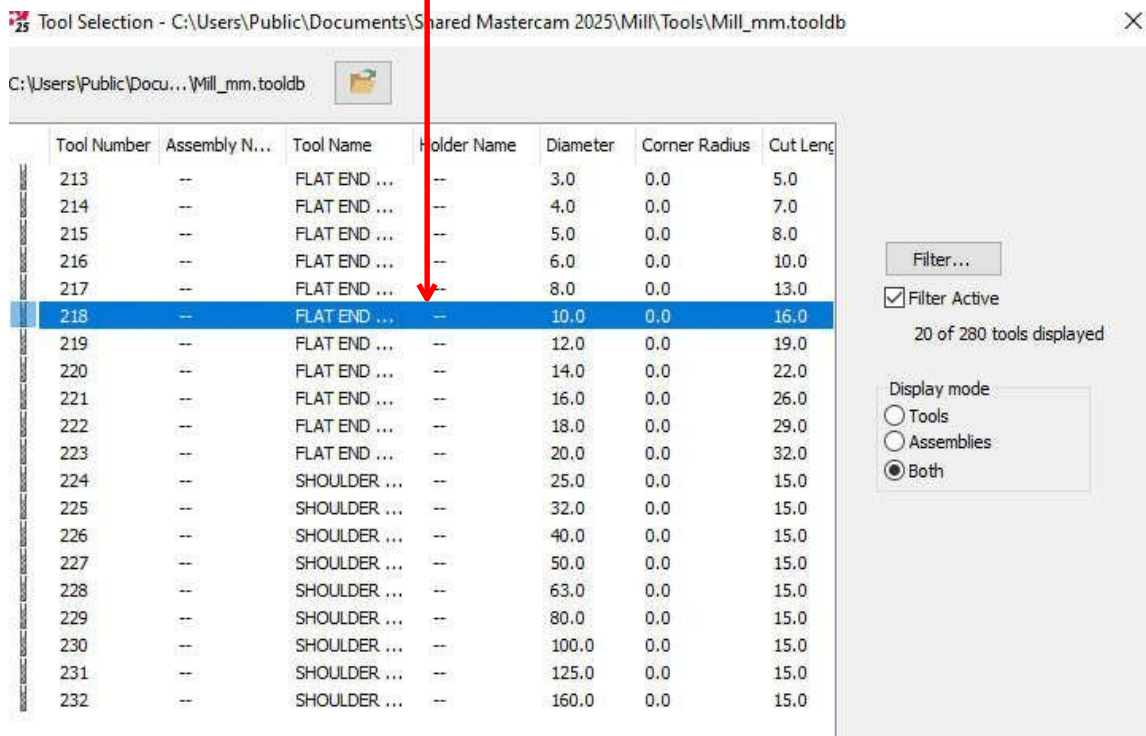
Step 1: Click on 2D Mill **Contour** operation and select **Chain**. Finally click on **Ok**.



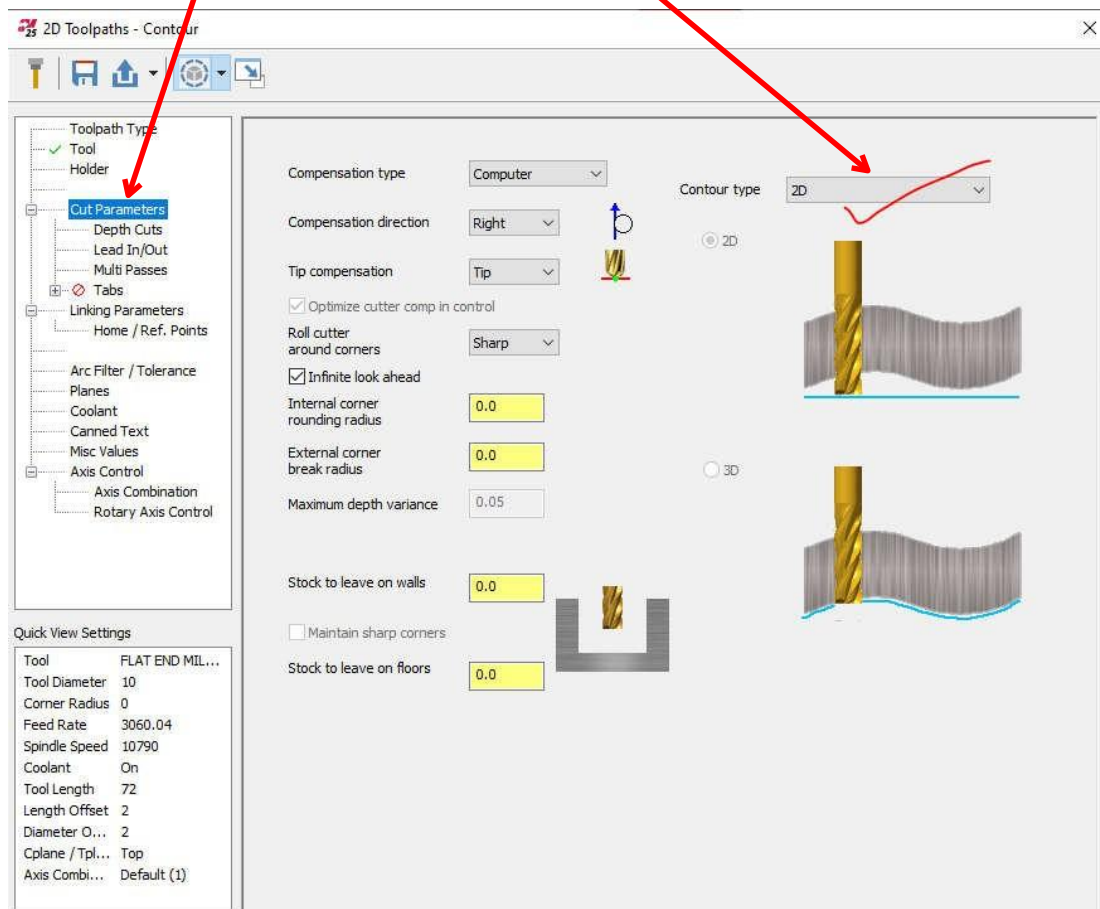
Step 2: Click on the **Tool** page in the 2D Toolpaths – Contour dialog box. Then click on **Filter**.



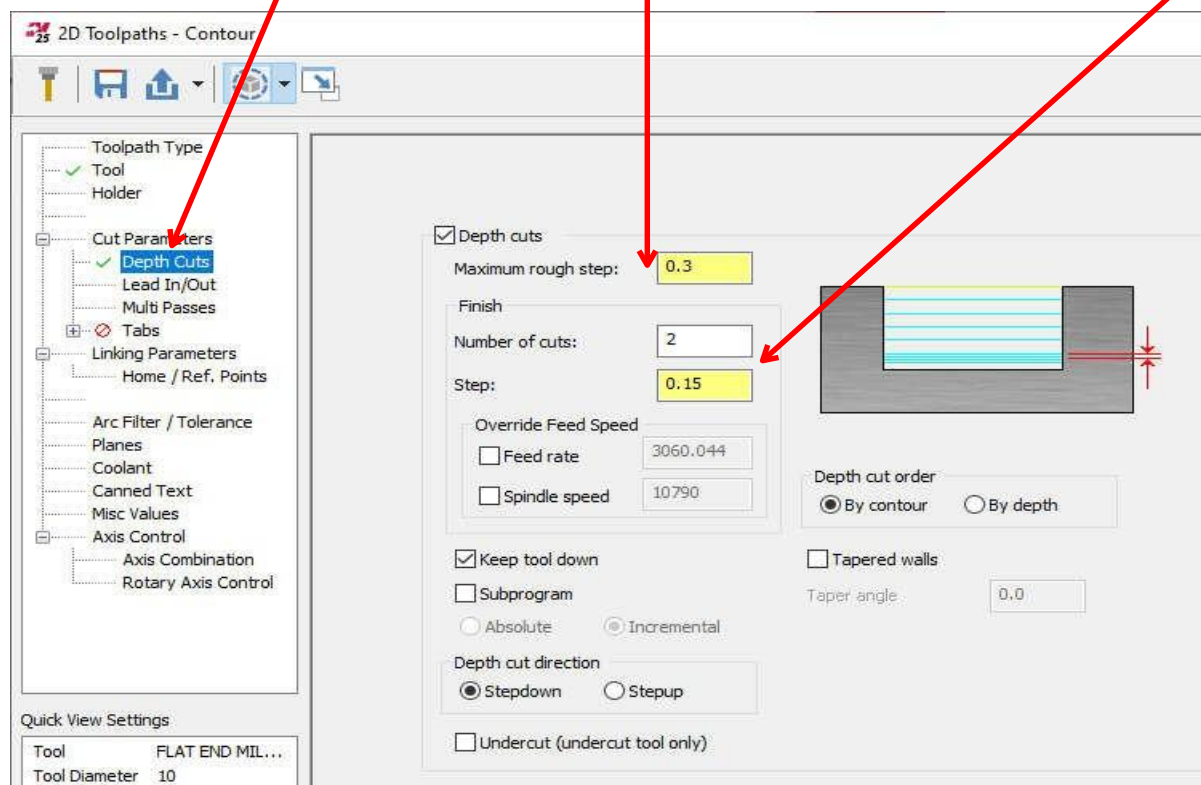
Step 3: Select Cutting Tool: 10mm Flat End Mill from Tool Library.



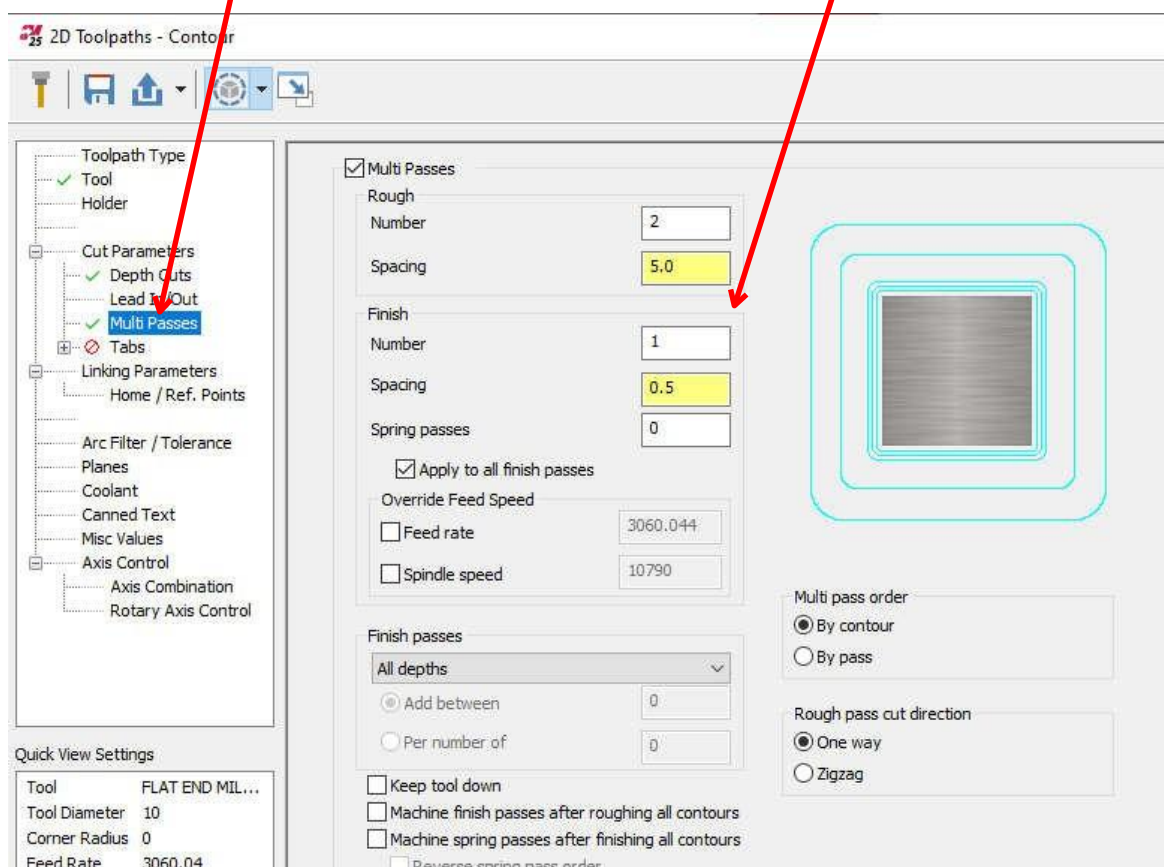
Step 4: Select Cut Parameters and Select Contour type: 2D.



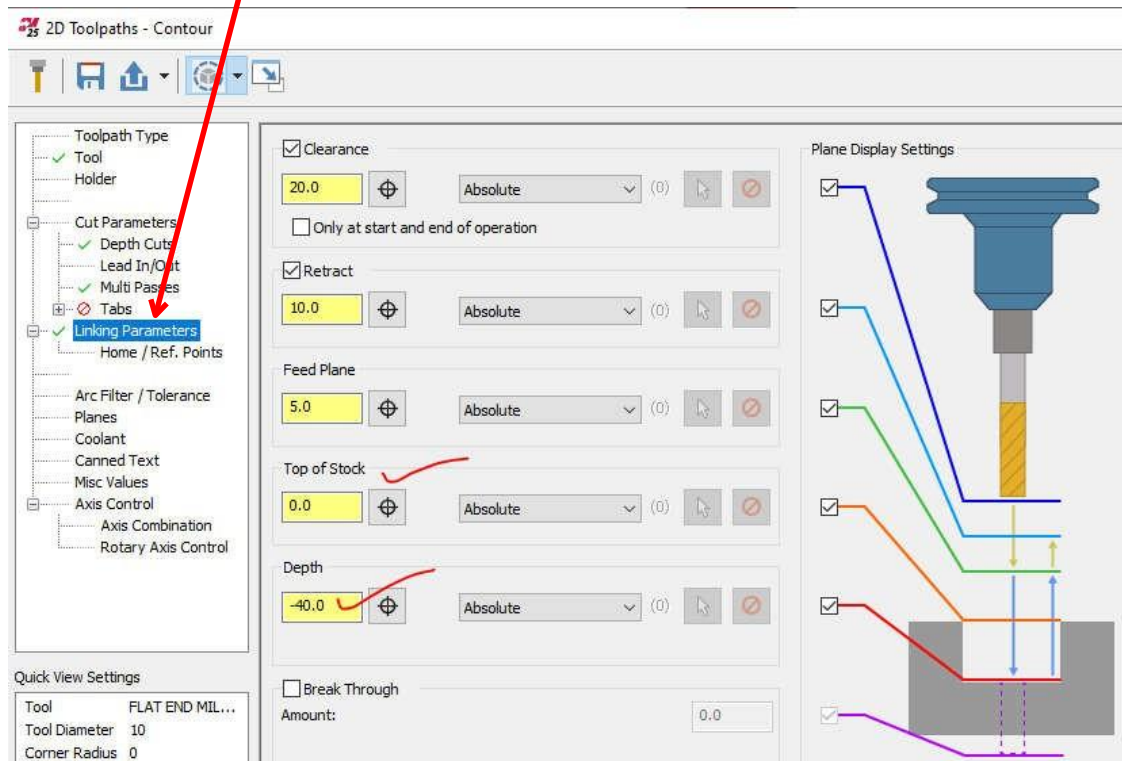
Step 5: Select **Depth of Cuts**. Change **Max rough step to 0.3** and No. Finish cuts **2** with Step **0.15**.



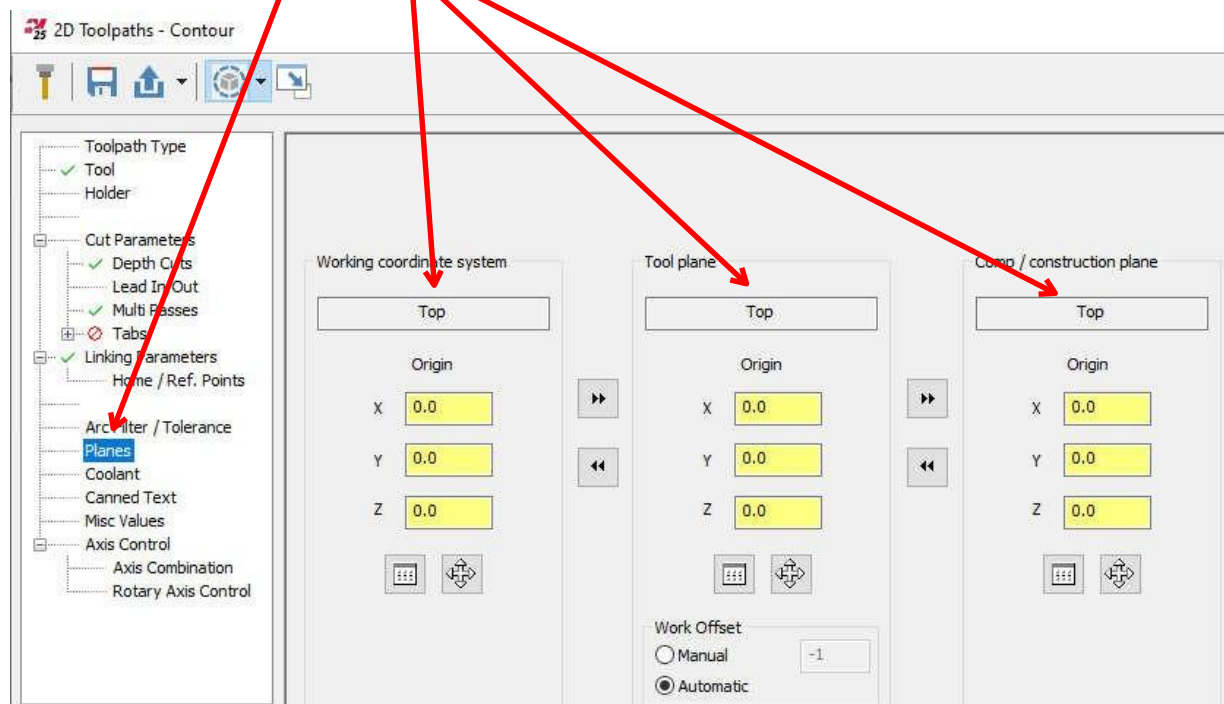
Step 6: Select **Multi Passes** and Change **Rough Number 2**, Spacing **5.0**; Finish Number **2**, Spacing **0.5**.



Step 7: Select **Linking Parameters**. Enter **20.0** and select **Absolute** for **Clearance**. **Retract:** Enter **10.0** and select **Absolute**. **Feed Plane:** Enter **5.0** and select **Absolute**. **Top of Stock:** Enter **0.0** and select **Absolute**. **Depth:** Enter **-40.0** and select **Absolute** to set the final machining depth.



Step 8: Ensure all **Planes** are **Top**. Finally click on **Ok**.

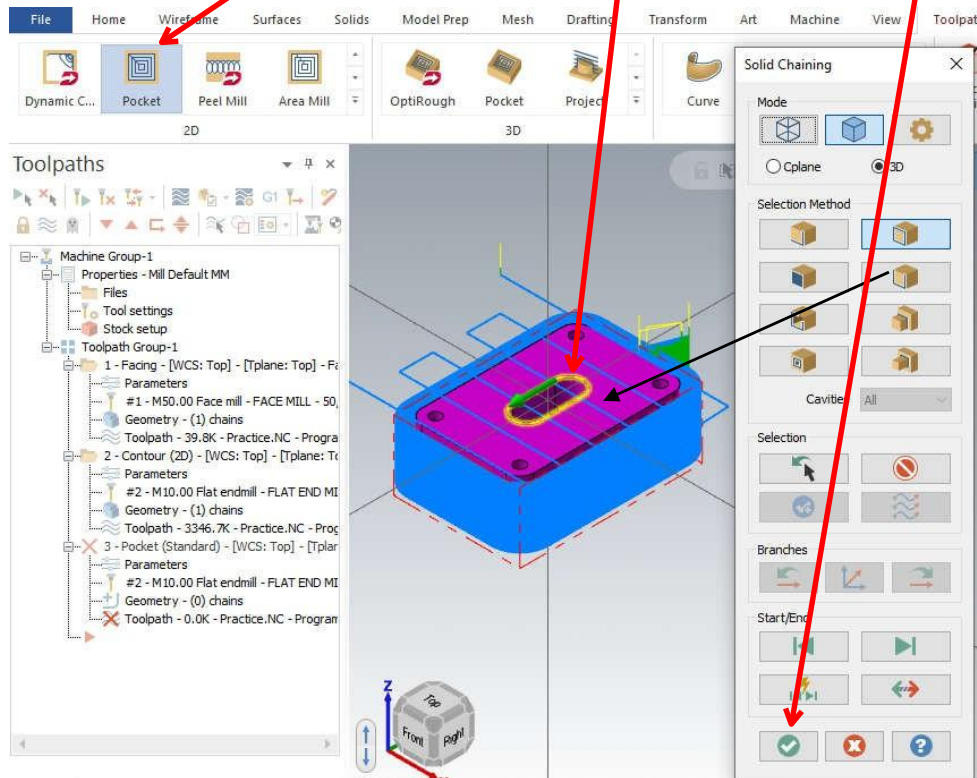




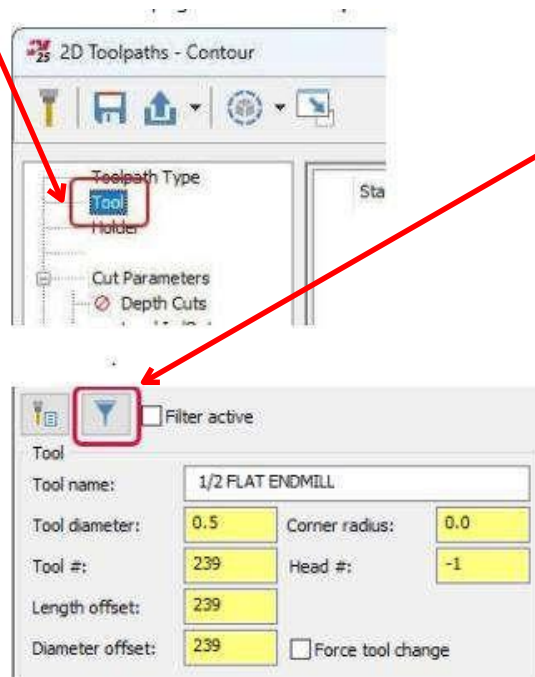
Information Sheet: 4.3-3

Learning Objective: Carry out milling 2D CAM programming (Pocket)

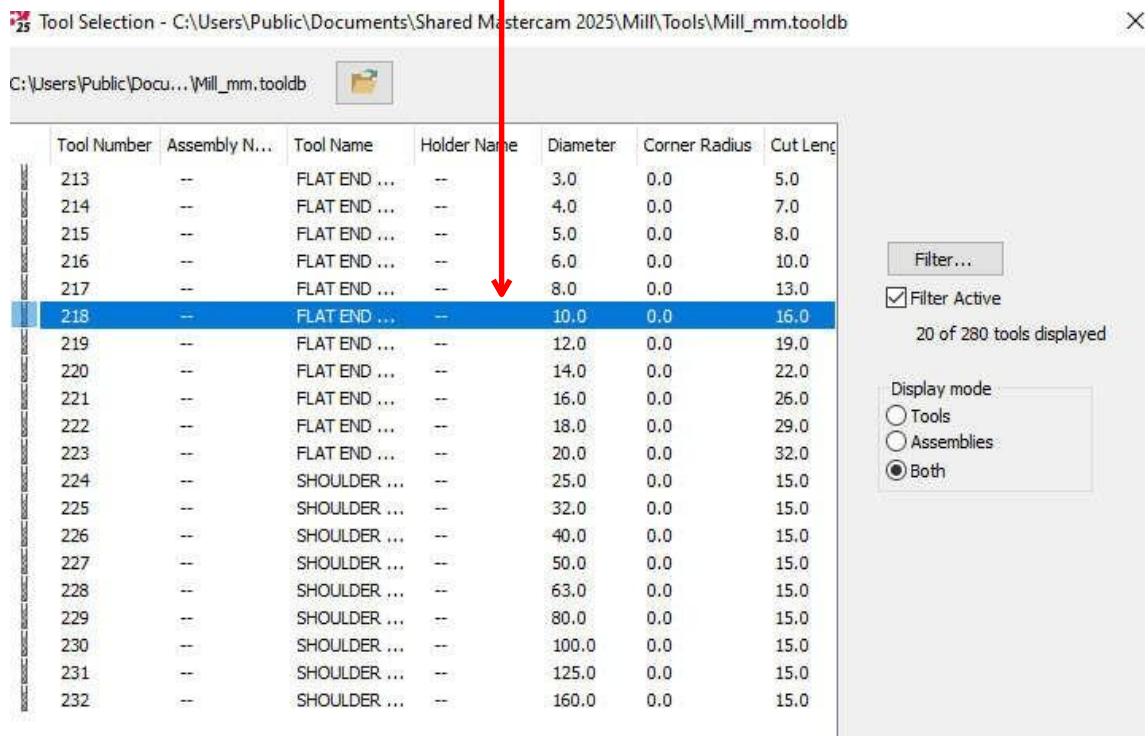
Step 1: Click on 2D Mill **Pocket** operation and select **Chain**. Finally click on **Ok.**



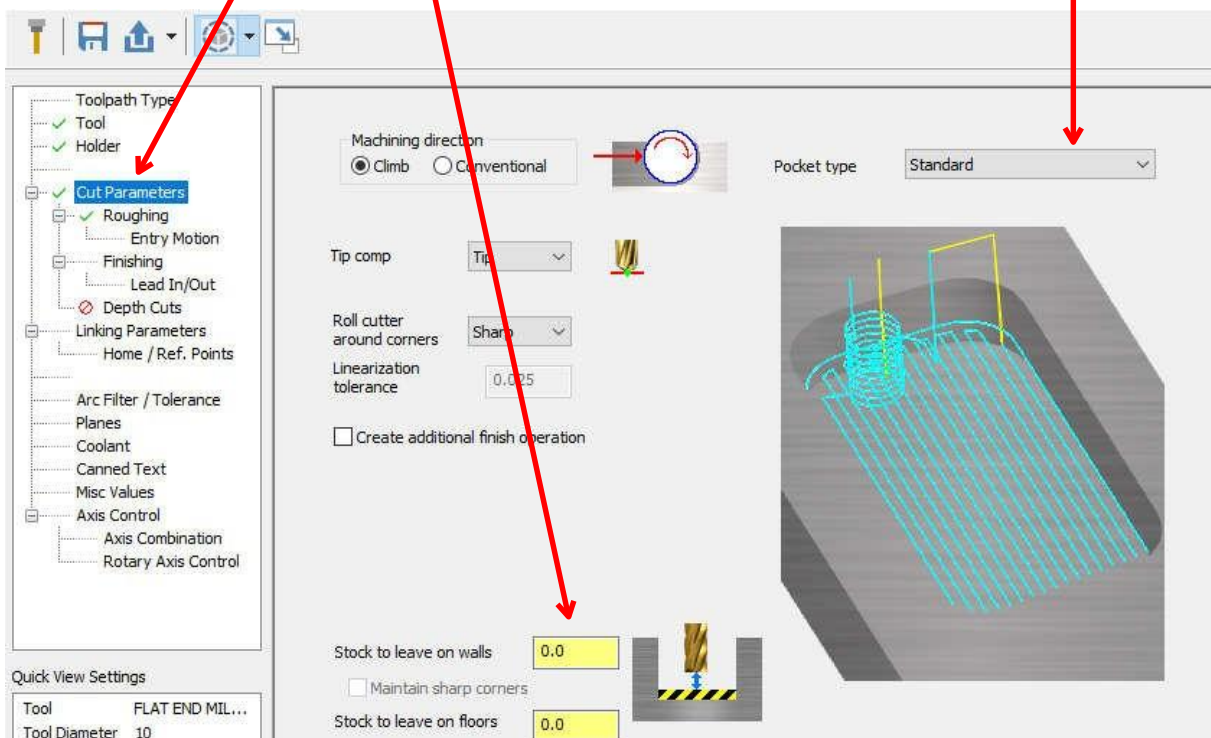
Step 2: Click on the **Tool** page in the 2D Toolpaths – Pocket dialog box. Then click on **Filter.**



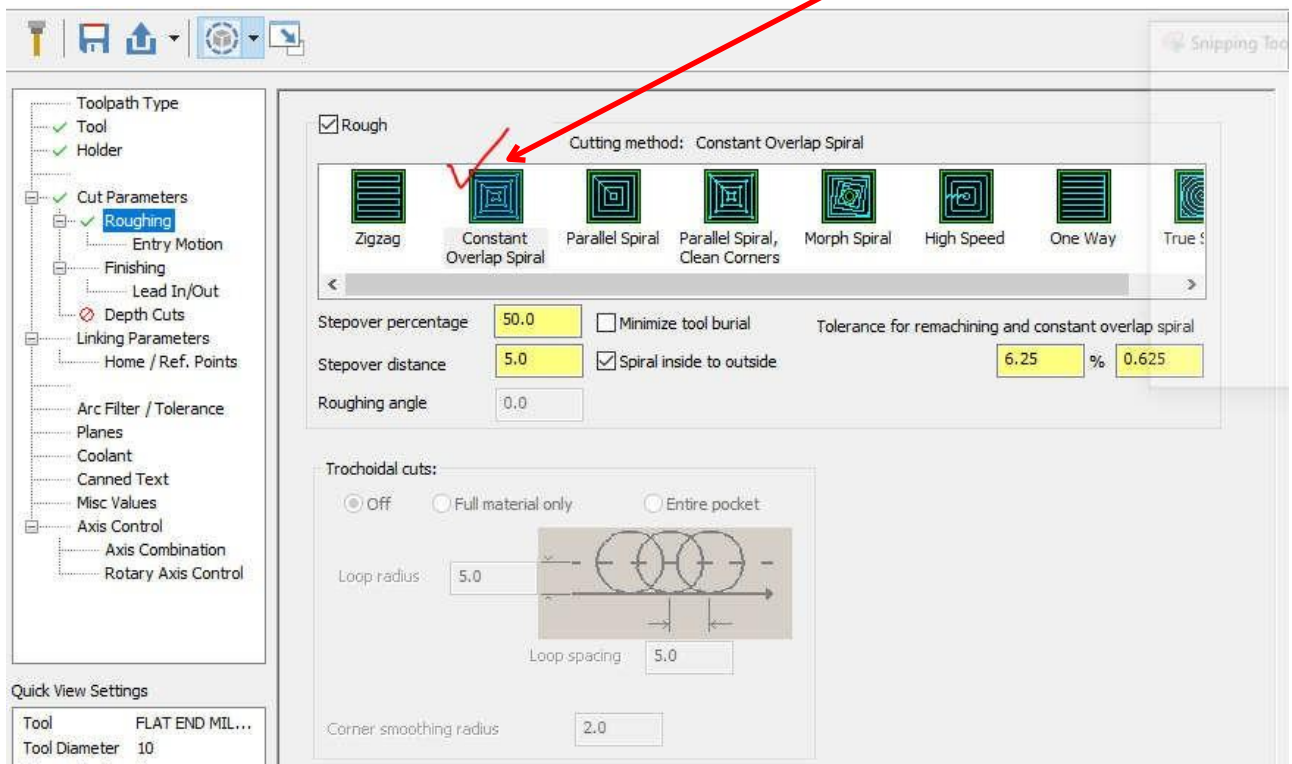
Step 3: Select Cutting Tool: 10mm Flat End Mill from Tool Library.



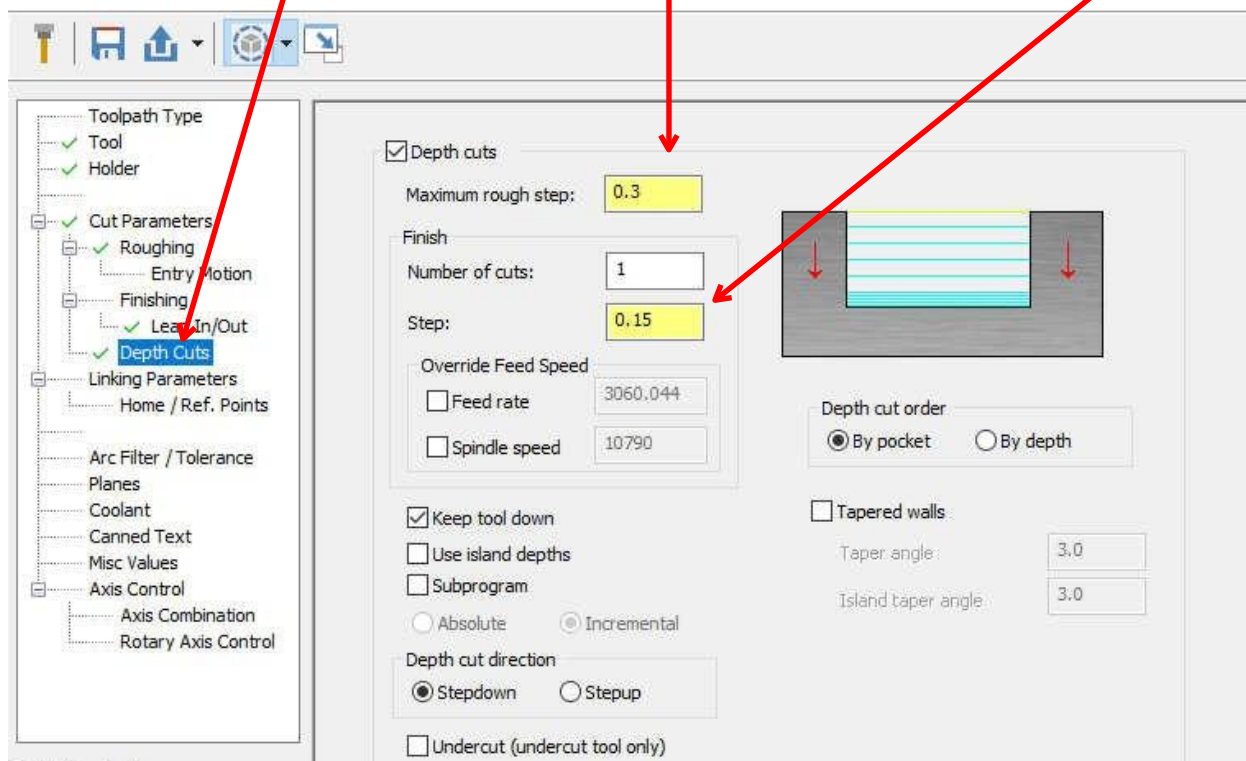
Step 4: Select Cut parameters. Choose Machining direction and Pocket type: **Standard**. Change Stock to leave on walls to 0.00.



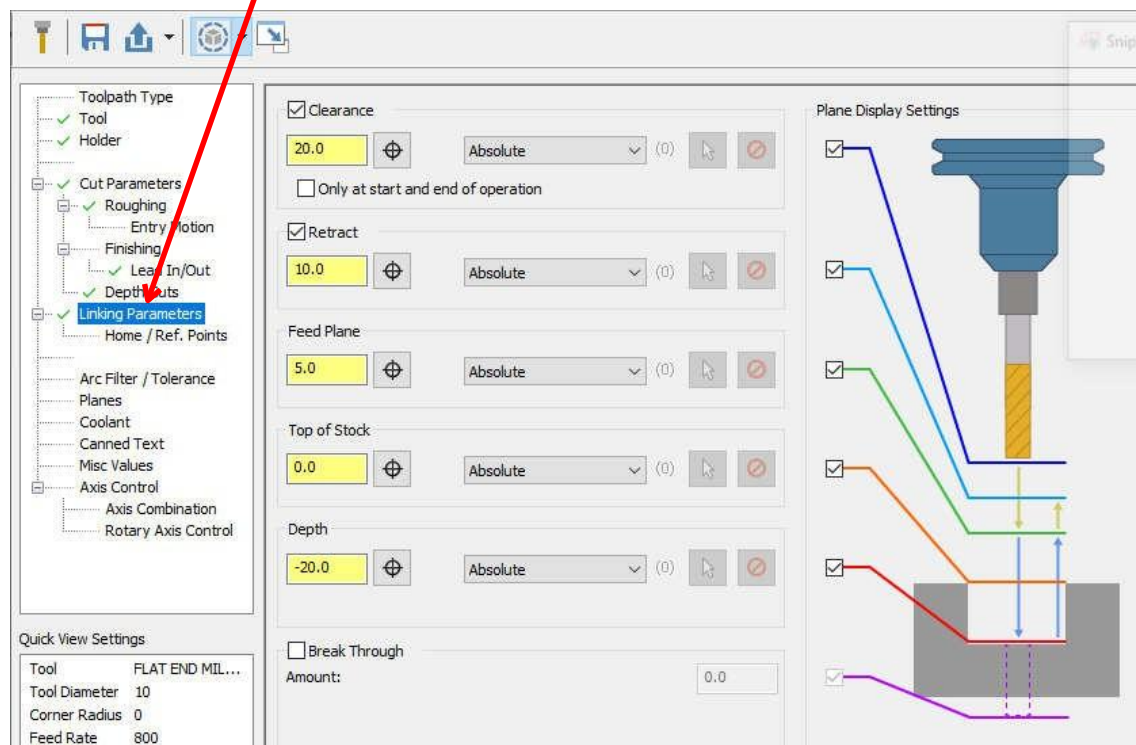
Step 5: Go to Roughing and select Cutting method: **Constant Overlap Spiral**.



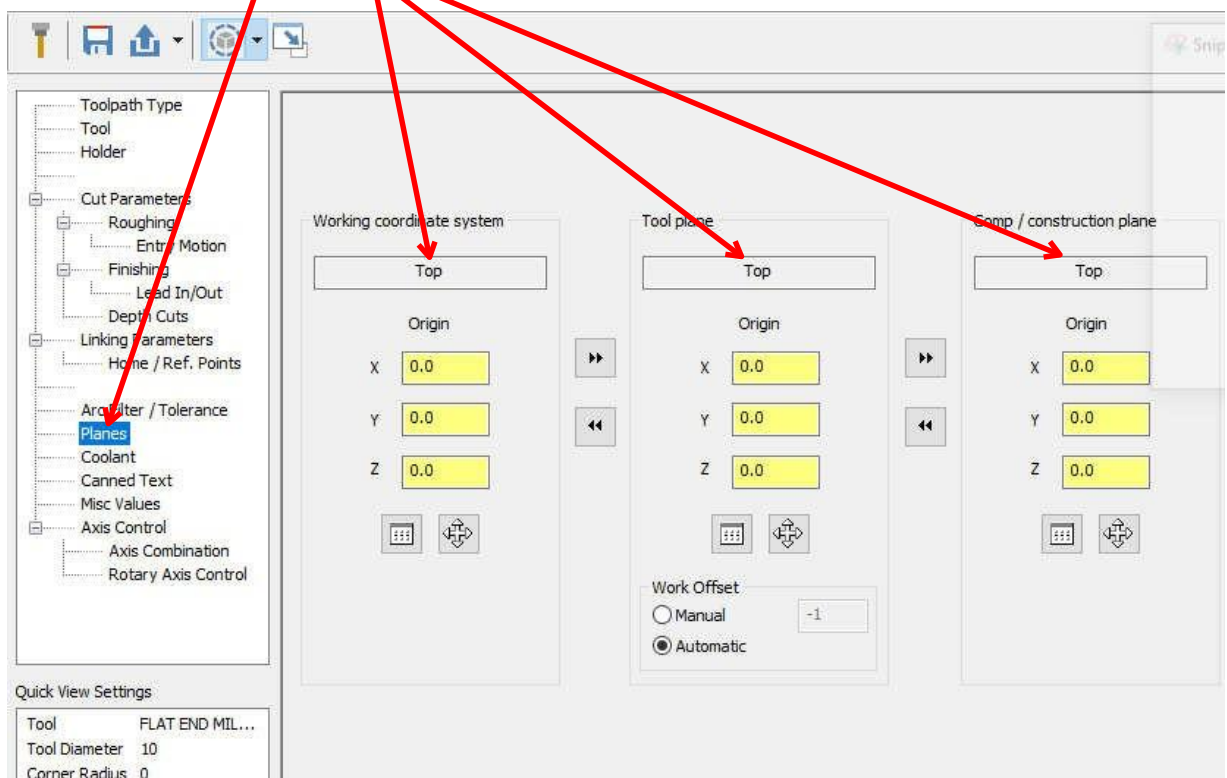
Step 6: Select **Depth of Cuts**. Change **Max rough step to 0.3** and No. Finish cuts **1** with Step **0.15**.



Step 7: Select **Linking Parameters**. Enter **20.0** and select **Absolute** for **Clearance**. **Retract:** Enter **10.0** and select **Absolute**. **Feed Plane:** Enter **5.0** and select **Absolute**. **Top of Stock:** Enter **0.0** and select **Absolute**. **Depth:** Enter **-20.0** and select **Absolute** to set the final machining depth.



Step 8: Ensure all **Planes** are **Top**. Finally click on **Ok**.

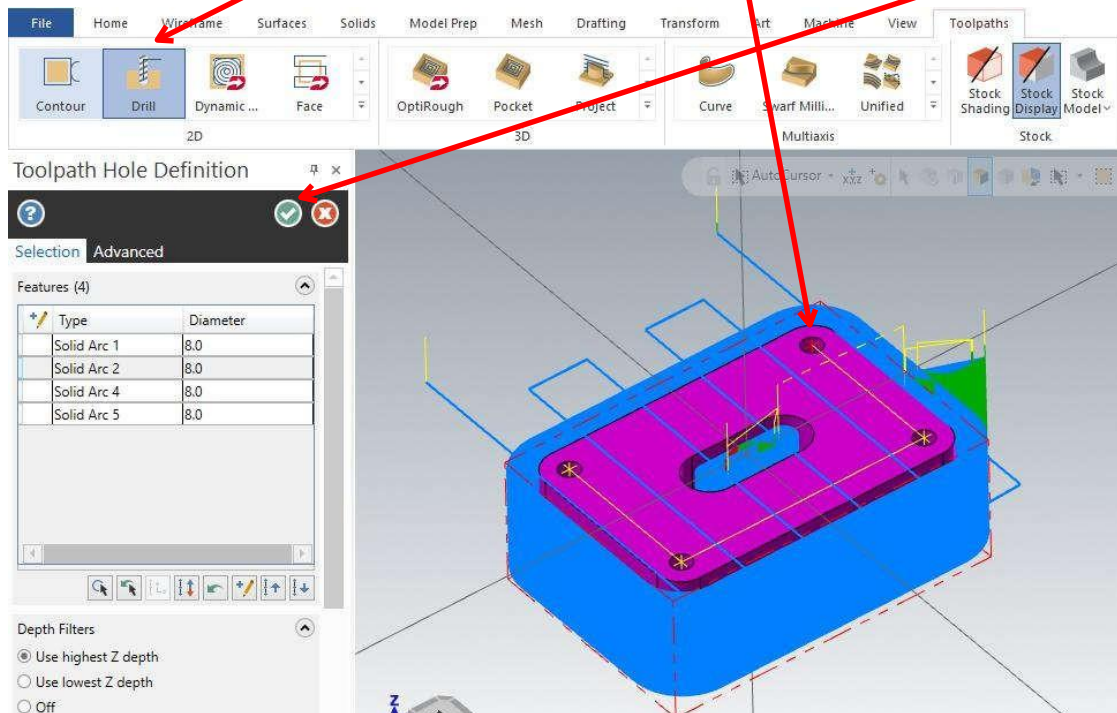




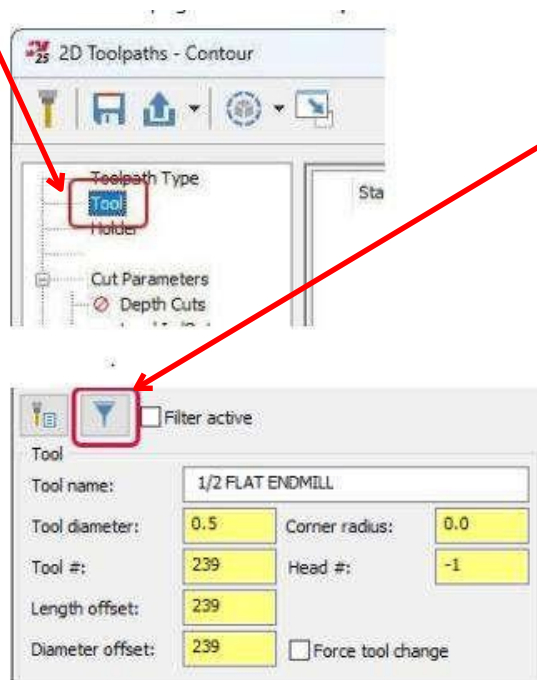
Information Sheet: 4.3-4

Learning Objective: Carry Out Milling 2D CAM Programming (Drilling).

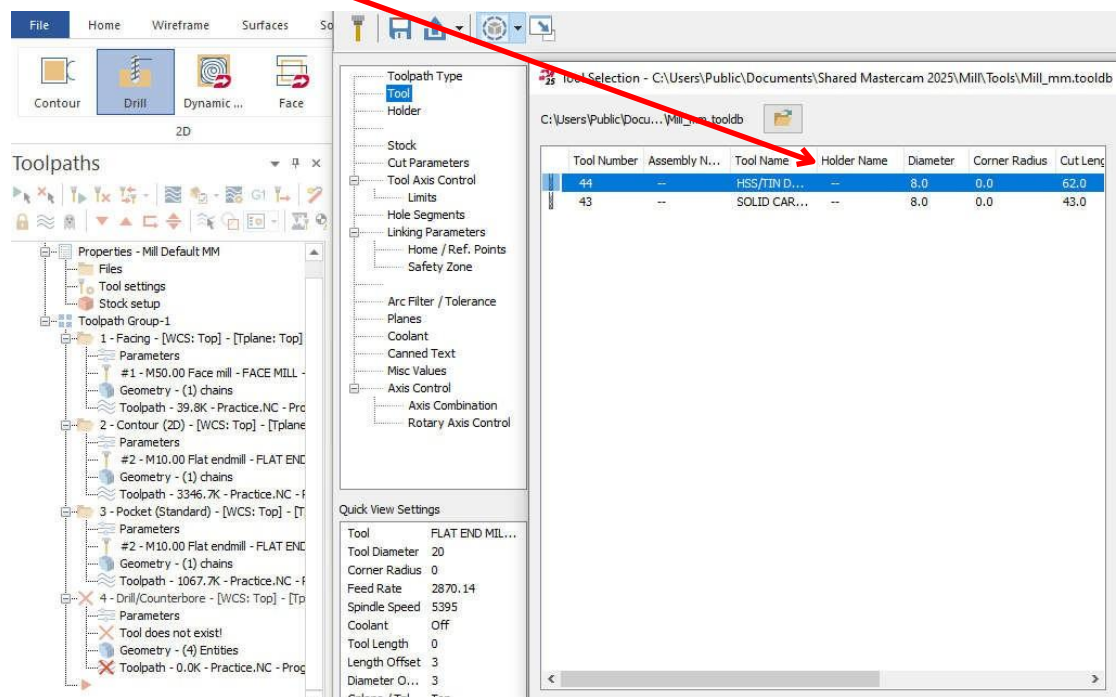
Step 1: Click on 2D Mill **Drill** operation and select **Drill Location**. Finally click on **Ok**.



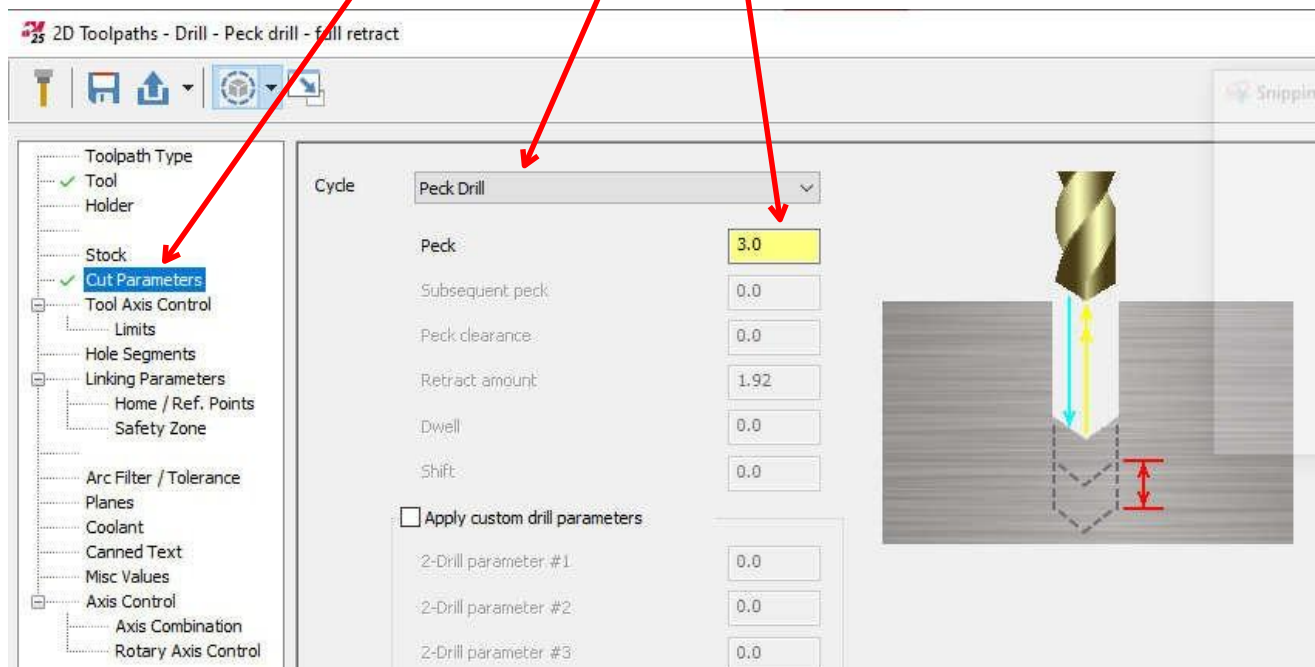
Step 2: Click on the **Tool** page in the **2D Toolpaths – Drill** dialog box. Then click on **Filter**.



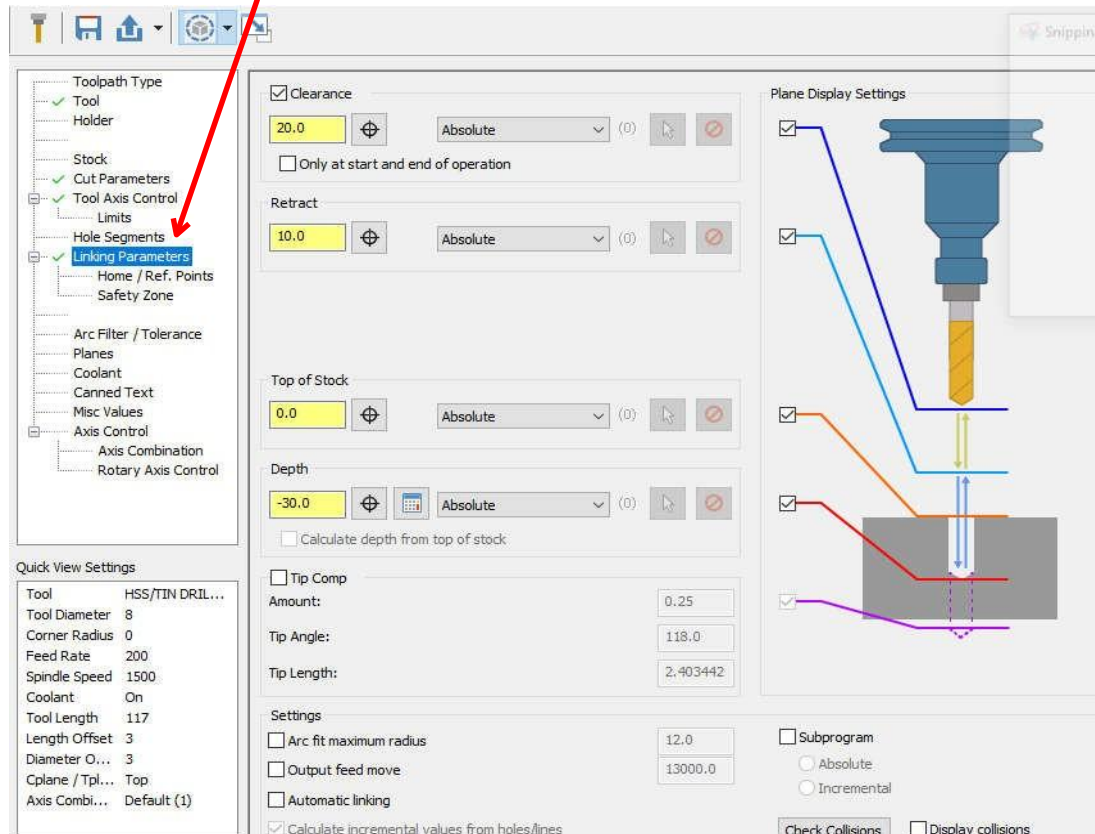
Step 3: Select **Drill: 8mm** from **Tool Library**.



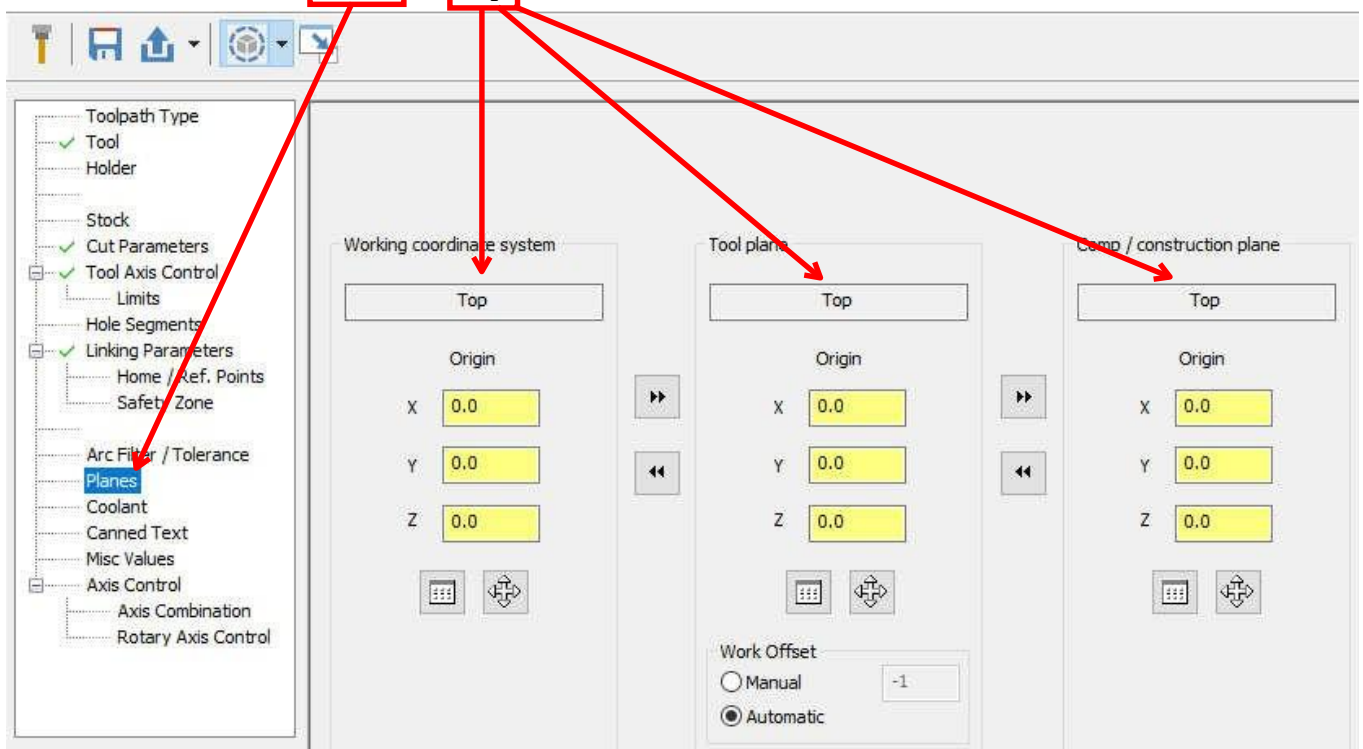
Step 4: Select **Cut Parameters**, **Peck Drill: 3mm**.



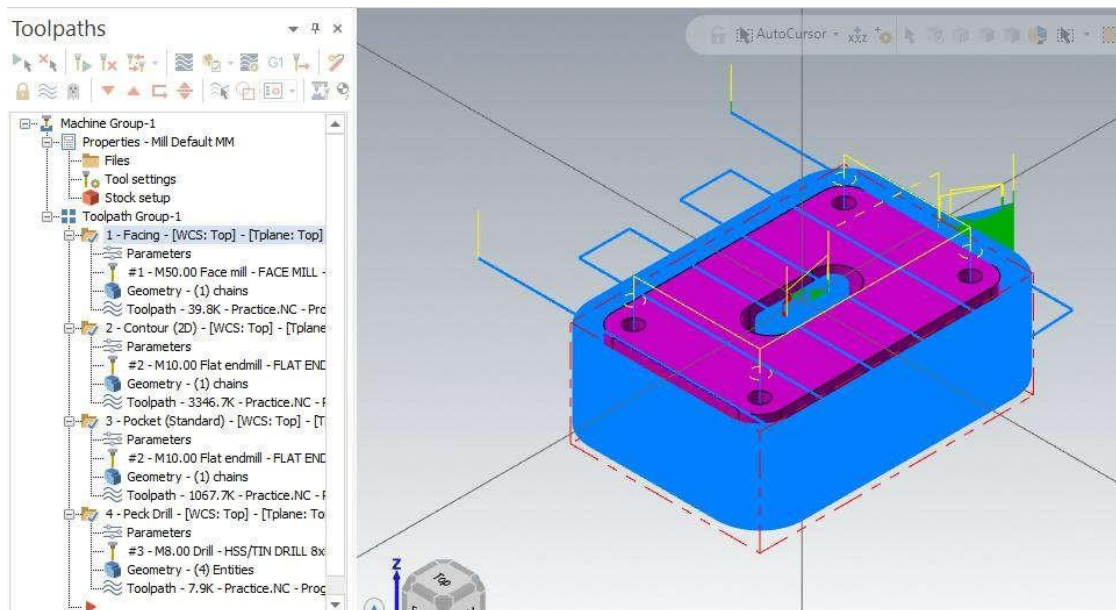
Step 5: Select Linking Parameters. Select **Linking Parameters**. Enter **20.0** and select **Absolute** for **Clearance**. **Retract**: Enter **10.0** and select **Absolute**. **Feed Plane**: Enter **5.0** and select **Absolute**. **Top of Stock**: Enter **0.0** and select **Absolute**. **Depth**: Enter **-30.0** and select **Absolute** to set the final machining depth.



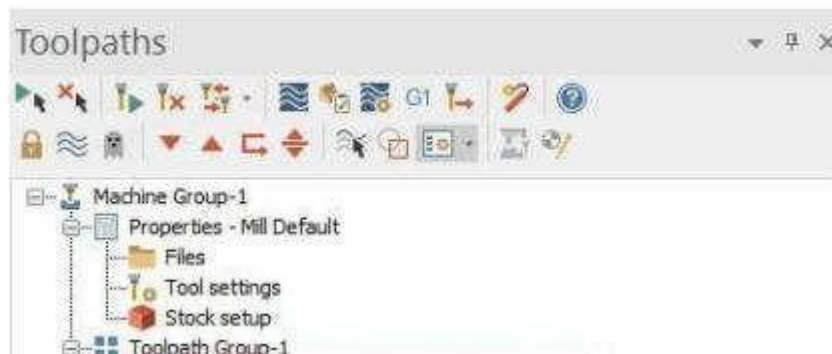
Step 6: Ensure all Planes are Top. Finally click on **Ok**.



Step 7: Tool path generation is done for Mill 2D operations (Facing, Contour, Pocket, Drilling).



Step 8: Backplotting and Verifying. In the **Toolpaths Manager**, check that your 2D Contour operation is selected.



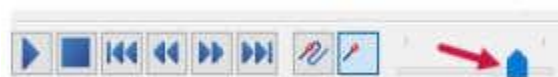
Step 9: Click Backplot selected operations. **Backplot dialog box:** Controls the backplot display and allows you to save geometry or settings. **Play bar:** Adjusts the speed of the simulation and allows you to pause, play, and set stops. **Tool and holder:** Simulate the selected tool and holder.



Step 10: Click the **Play** button or press [R].



Step 11: When the simulation has finished, change the simulation speed by adjusting the run speed slider.



Step 12: Click OK in the Backplot dialog box



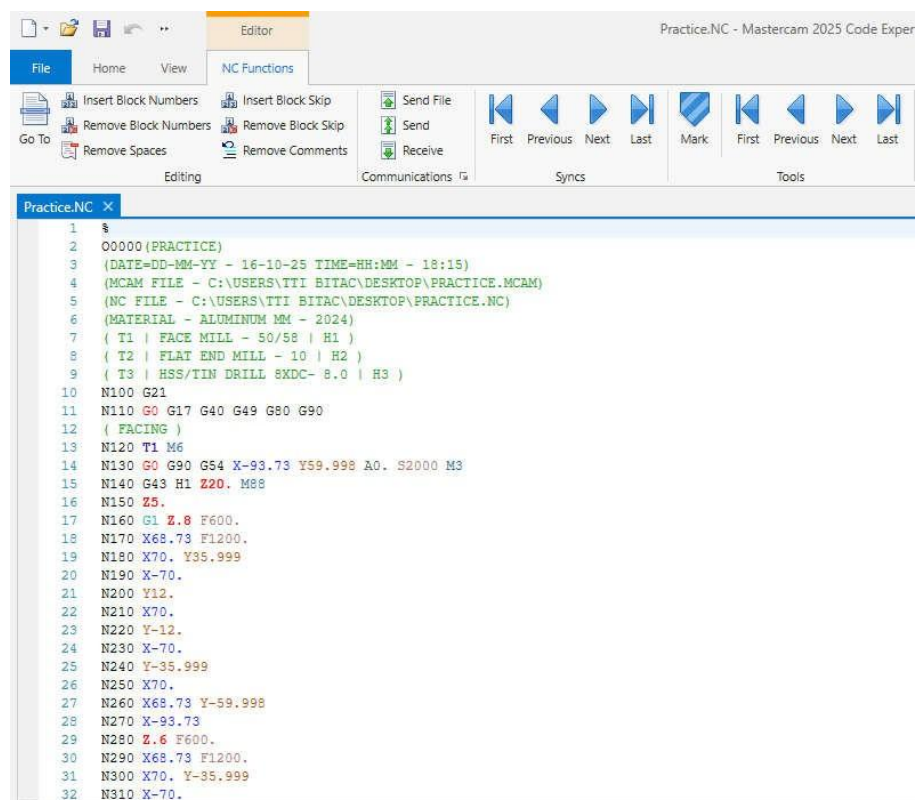
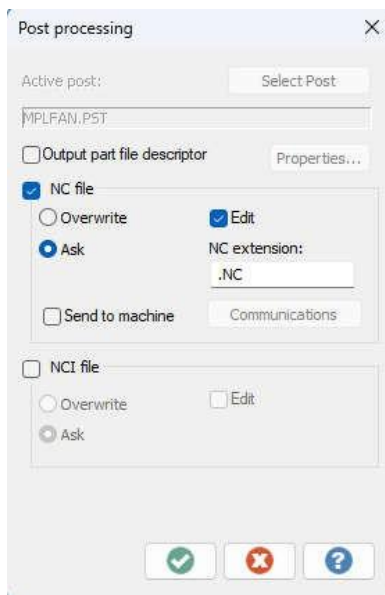
Step 13: Post Processing the file. **Click** Select all operations.



Step 14: Select the **Post selected operations** button in the **Toolpaths Manager**.



- Accept the **NC name**.
- Save the file.
- **Mastercam Code Expert** will be launched and the G code will display as shown.





Self-Check: 4.3-1 to 4.3-4

1. What is the primary function of the **Tool Path Manager**?
2. Which menu is used to select and define the **stock setup**?
3. In Mastercam, what does the term **WCS** stand for?
4. What is the main difference between a **Standard Pocket** and an **Open Pocket** toolpath?
5. What is the fundamental difference between a **Drill/Counterbore** cycle and a **Peck Drilling** cycle?



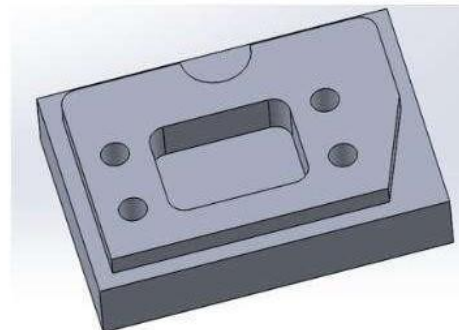
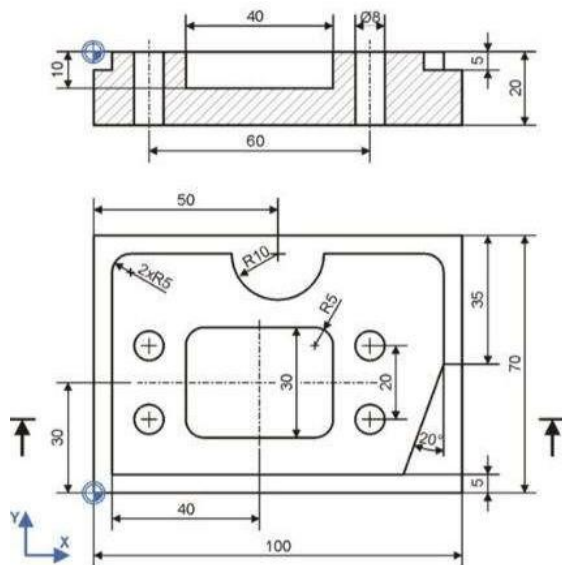
Answer Script: 4.3-1 to 4.3-4

1. To organize, view, modify, and regenerate all the toolpaths associated with the part.
2. The **Toolpaths tab** (specifically, the **Stock Setup** option).
3. **Work Coordinate System.**
4. A **Standard Pocket** requires the tool to plunge into the material, while an **Open Pocket** allows the tool to enter from the outside edge of the stock.
5. A **Drill/Counterbore** cycle plunges straight to depth in one motion (G81), while a **Peck Drilling** cycle (G83) plunges, retracts partially or fully to clear chips, and then repeats until the final depth is reached.

Job Sheet: 4.3 (a)

Job	Perform 2D Milling operations in Mastercam	
Materials/Resources	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
	CAM Software (Mastercam)	1 Pc
Procedure	Collect required materials/resources.	
	Open CAD software (SolidWorks)	
	Use Feature Tools for 3D drawings in SolidWorks.	
	Open CAM Software (Mastercam).	
	Set Origin of the job.	
	Perform Stock-setup.	
	Choose appropriate 2D Milling operations (Face, Contour, Pocket, Drill).	
	Set cutting Tools and parameters.	
	Generate Toolpaths.	
	Visualize simulation.	
	Generate NC codes	
	Follow trainers/instructors.	
	Save Part files in a folder.	

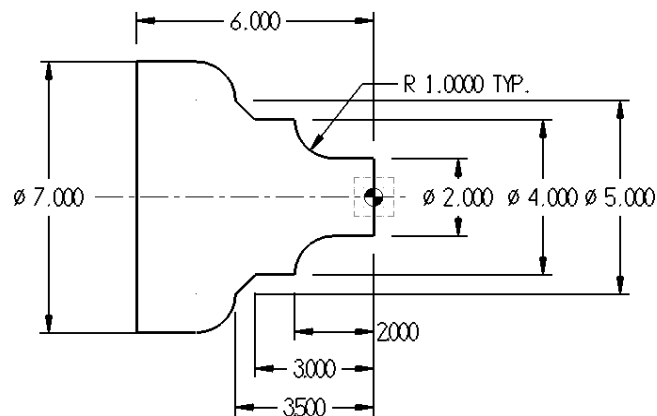
• Sample:





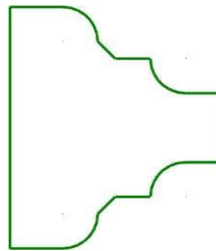
Information Sheet: 4.3-5

Learning Objective: Carry Out Lathe CAM Programming.

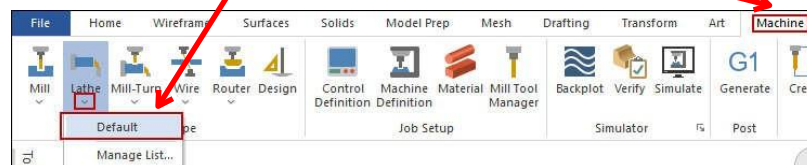


Step 1: Create and locate part **geometry**

In this step you will create the geometry using the following commands: rectangle, lineparallel, line endpoints, fillet and delete.

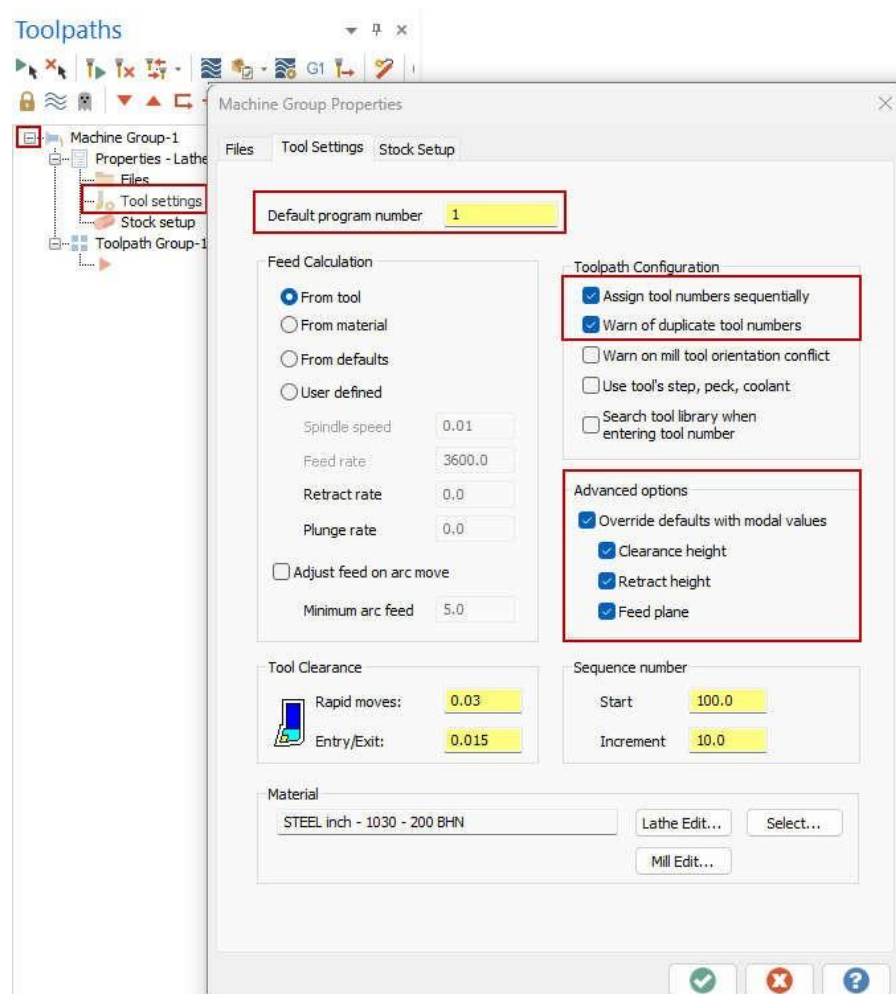


Step 2: From the **Machine** tab, **Select Lathe and Default**



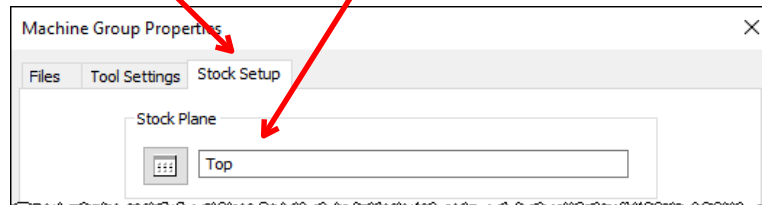
Step 3: Create the job setup

- Expand the Properties in the Toolpath Manager.
- Click the plus (+) sign if the Files, Tool Settings and Stock Setup are not visible as shown.
- Select the Tool settings and change the parameters as shown.
- Program # = 1.
- Feed Calculation set to From Tool.
- Enable Assign tool numbers sequentially.
- Enable Warn of duplicate tool numbers.
- Enable Advanced Options.



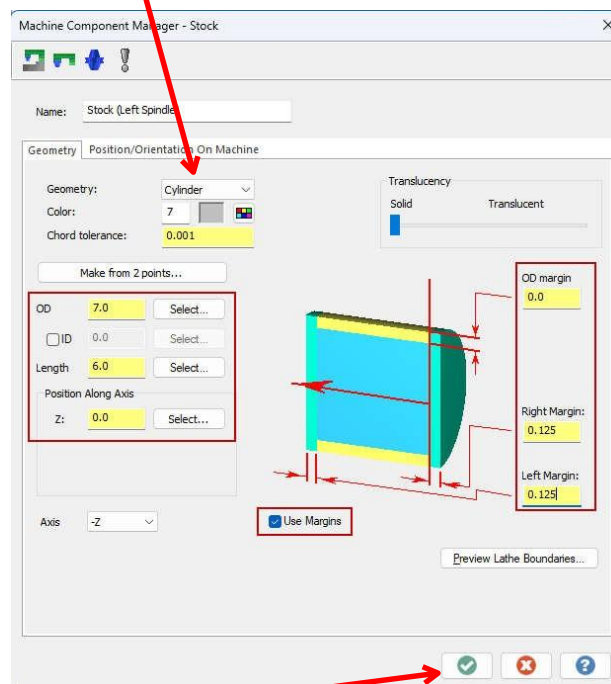
Step 4: Set the Stocksettings. Make sure that the **Stock Plane** is set to **Top**.

- Select the **Stock Setup** tab.
- Enable **Left Spindle**.
- Set the stock size by selecting **Properties**.



Step 5: Set the **Geometry to Cylinder**. Set the **OD = 7.0"**. **Length = 6.0"**. **Z = 0"**.

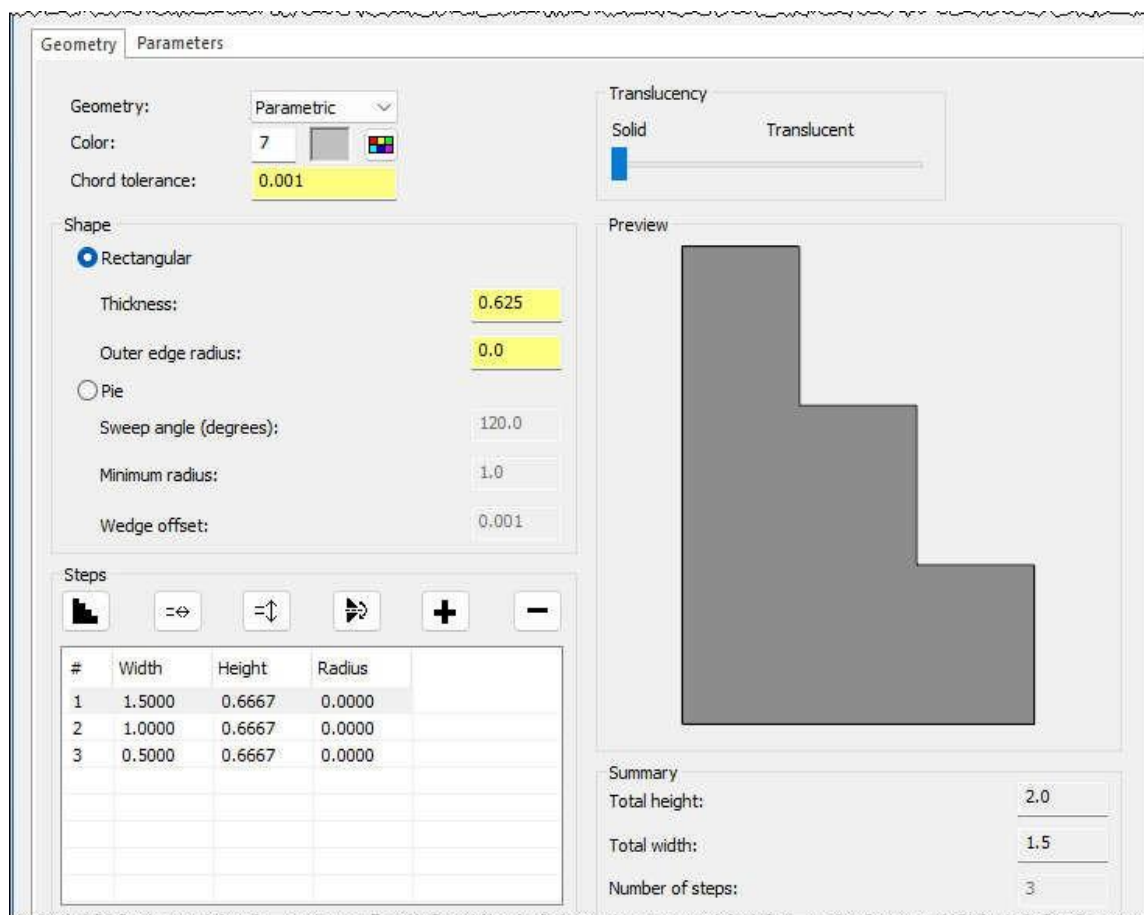
- Axis **-Z**.
- Enable **Use Margins**. **OD Margin = 0.0"**.
- **Right margin = 0.125"**.



Step 6: Click on **OK** button to exit the **Machine Component Manager - Stock** dialog box.

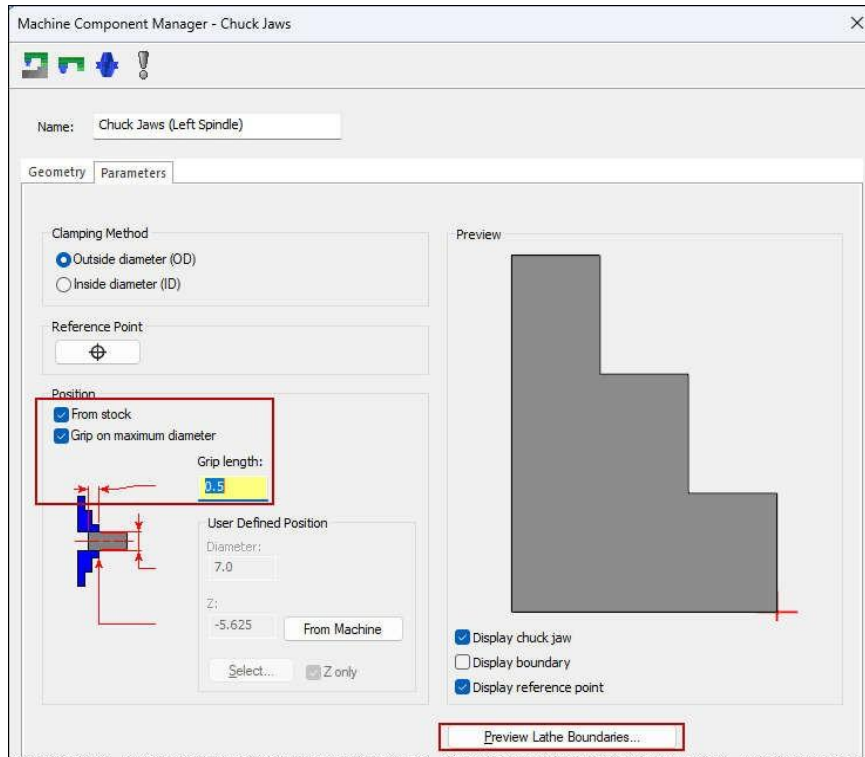
Step 7: Set the Chuck Jaw

- Enable **Left Spindle.**
- Set the chuck size and the location by selecting **Properties**
- In the **Geometry** dialog box, leave the default parameters for the chuck geometry as shown.



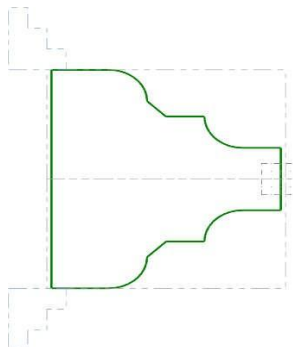
Step 8: In the **Parameters**, make sure **Clamping Method** is set to **Outside diameter (OD)**.

- In the **Position** enable **from stock**, **Grip on maximum diameter**.
- **Grip length** = **0.5"**.
- Select the **Preview Lathe Boundaries** button to check the stock and the chuck.

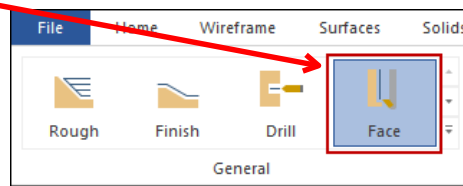


Step 9: The stock should look as shown.

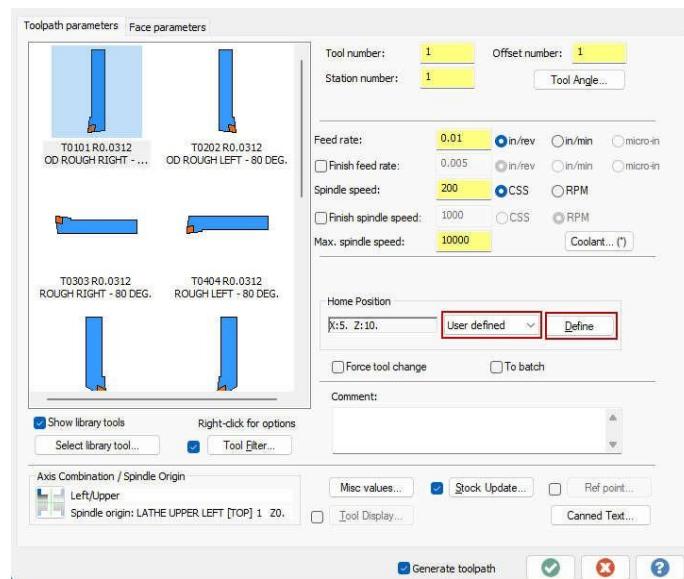
- Press **Enter** to continue.
- Click **OK** button to exit the **Machine Component Manager** dialog box
- Click **OK** button to exit the **Machine Group Properties** dialog box.



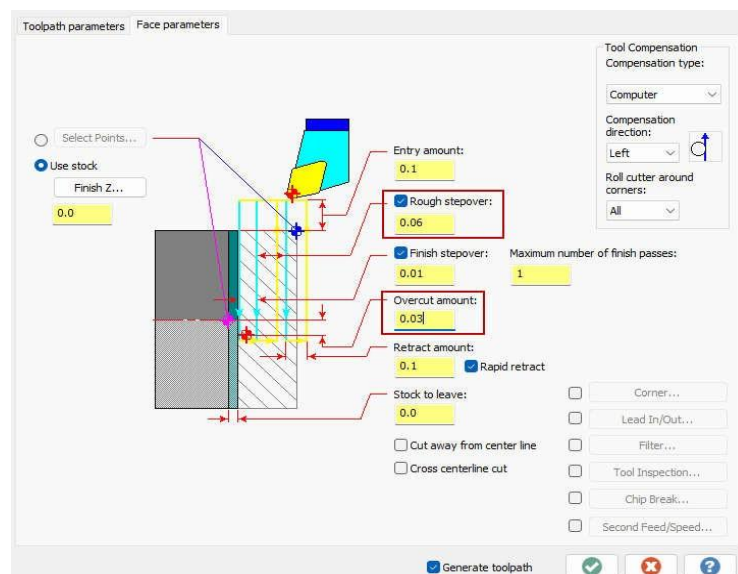
Step 10: Face toolpath selection



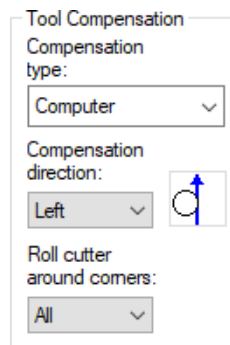
Step 11: Set the Toolpath parameters. Select the OD Rough Right 80 deg tool. Make sure the Tool/Offset Number is set to 1. Set the CoolantFlood On.



Step 12: Make sure that **Use stock** is enabled. **Finish Z = 0"**. **Entry amount = 0.1"** (sets the distance from the stock that the cutter starts to feed). **Rough stepover = 0.06**. **Finish stepover = 0.01"**. **Overcut amount = 0.03"**. **Retract amount = 0.1"**. **Stock to leave = 0.0"**.



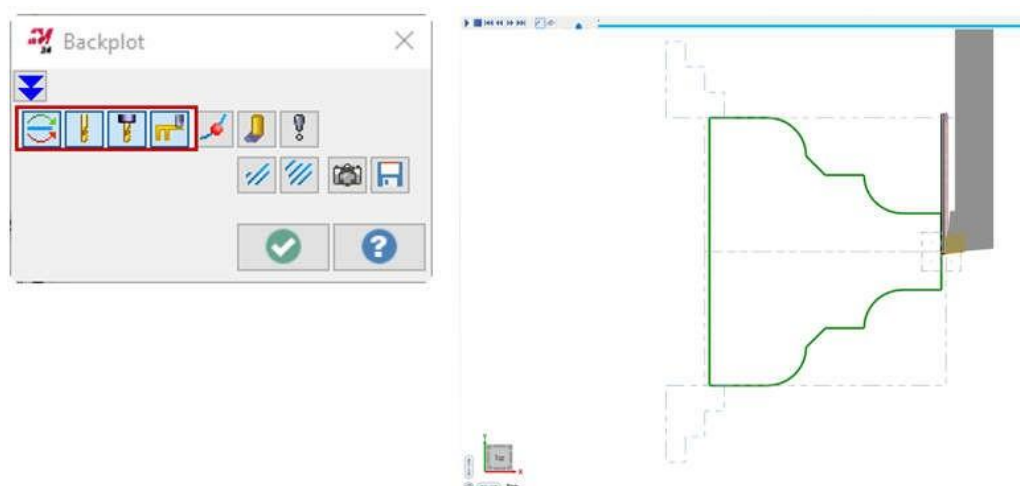
Step 13: Establish the cutter compensation type and direction. Cutter compensation set to **Computer** and the compensation direction set to **Left**.



Step 14: From **Toolpaths Manager**, select the **Backplot** selected operations button.



Step 15: To fit the workpiece onto the screen, right mouse clicks in the graphics window and select the **Fit** icon. You can step through the **Backplot** by using the **Step forward**  or **Step back**  buttons. You can adjust the speed of the backplot. . Select the **Play Simulation** button in the VCR bar to run Backplot. . Select the **OK** button to exit **Backplot** dialog box .



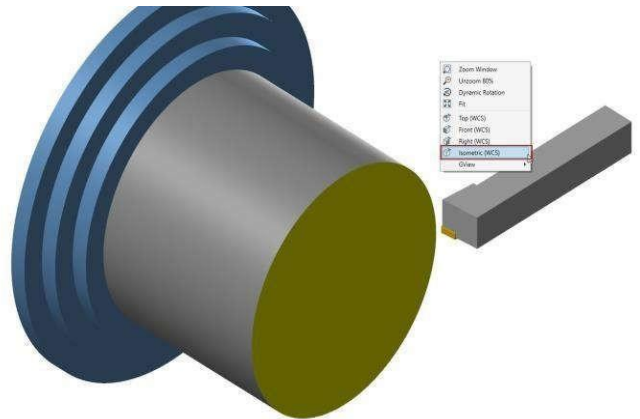
Step 16: To start **Verify**, from the **Toolpaths Manager**, select the **Verify selected operations** icon as shown.



- Select the **Play** button in the **VCR** bar to run



- To see the part from an **Isometric** view, right click in the graphics window and select the **Isometric** view.
- To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.



Step 17: From the General group, select the **Rough** icon as shown.

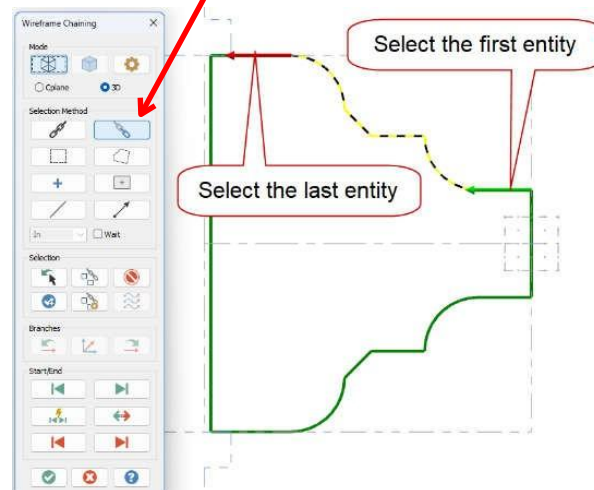




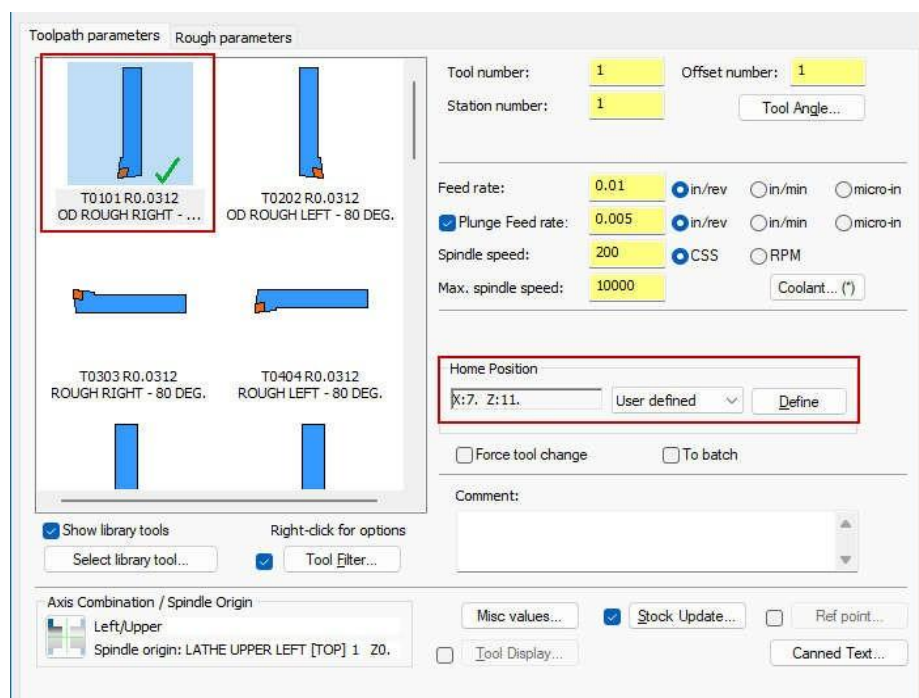
Information Sheet: 4.3-6

Learning Objective: Carry Out Lathe CAM Programming (**Rough Turning**).

Step 1: In the **Chaining** dialog box, the **Partial** button should be enabled as shown. Select the **First entry** of the chain and the **Last entity** of the chain as shown.

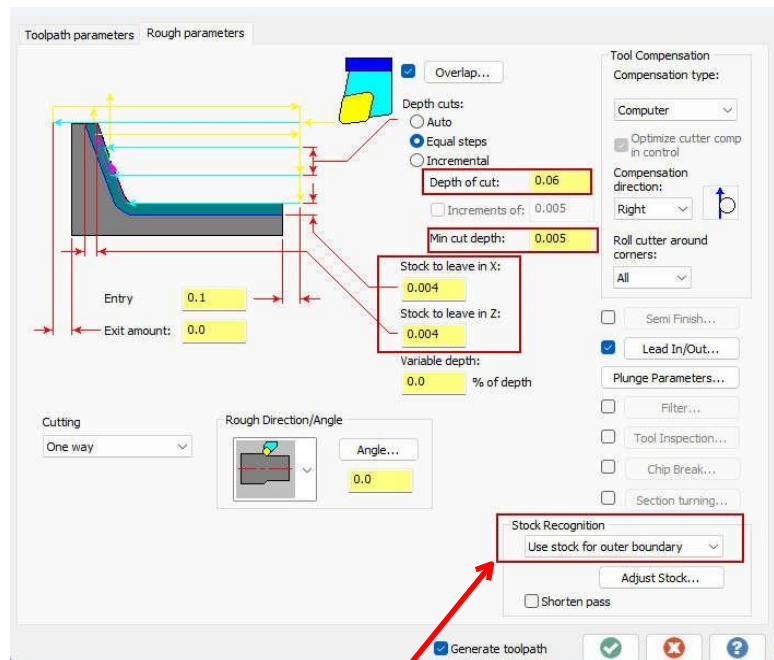


Step 2: Select the existing **OD Rough Right 80 deg** tool. Set the **Tool/Offset Number** to **1**. Set the **CoolantFlood On**. Set the **Home Position** to **User Defined X7; Z11**.

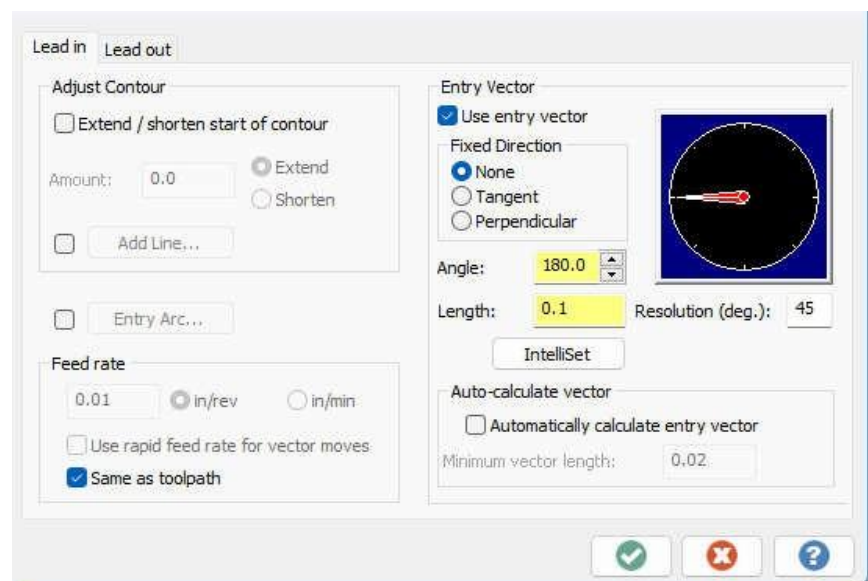


Step 3: Set the Rough parameters

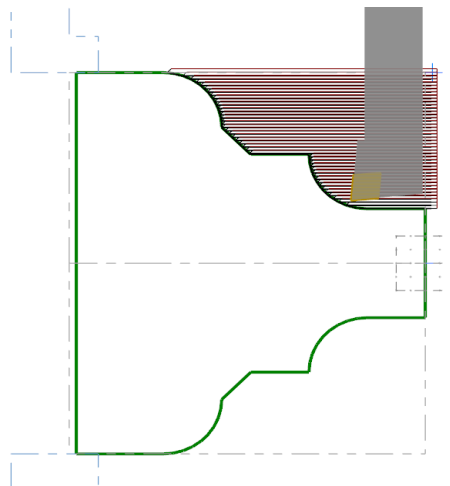
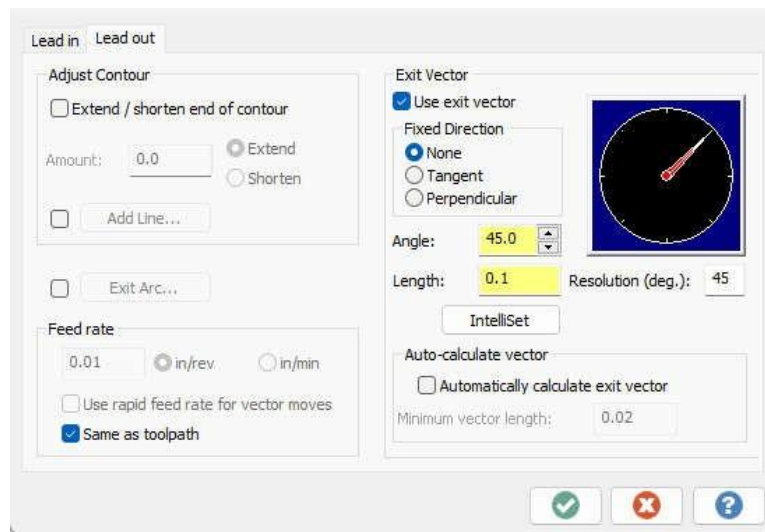
- Make sure that **Overlap** is enabled to overlap the previous cut before making the next cut.
- **Entry amount = 0.1"**.
- Enable **Equalsteps**.
- **Depth of cut = 0.06"** (sets the amount of stock that gets removed during each cutting pass).
- **Minimum cut depth = 0.05"** (determines the smallest cut that can be taken per pass). Stock to leave in X = 0.004".
- **Stock to leave in Z=0.004"**.
- **Cutting method** set to **One-way** and the **Rough Direction/Angle** = 0
- Set the **Cutter compensation** to **Computer** and to the **Right**.
- **Stock Recognition** set to **Use stock for outer boundary**.



- Select **Lead In/Out** and in the **Lead in** tab, leave **Use entry vector**.
- Angle = 180.0 degrees.
- **Length = 0.1**.



Step 4: Set in the **Lead out**, the **Exit Vector** to **Angle =45** and **Length = 0.1"**. Backplot the Roughing Toolpath.

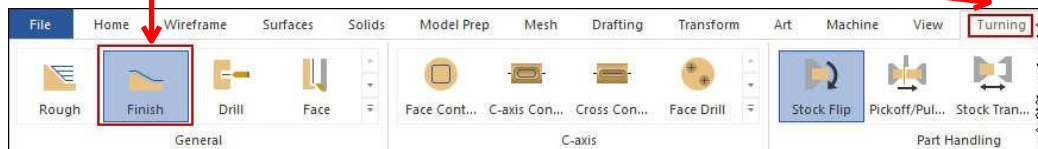




Information Sheet: 4.3-7

Learning Objective: Carry Out Lathe CAM Programming (**Finish Turning**).

Step 1: Select the **Finish** Toolpath **Turning**

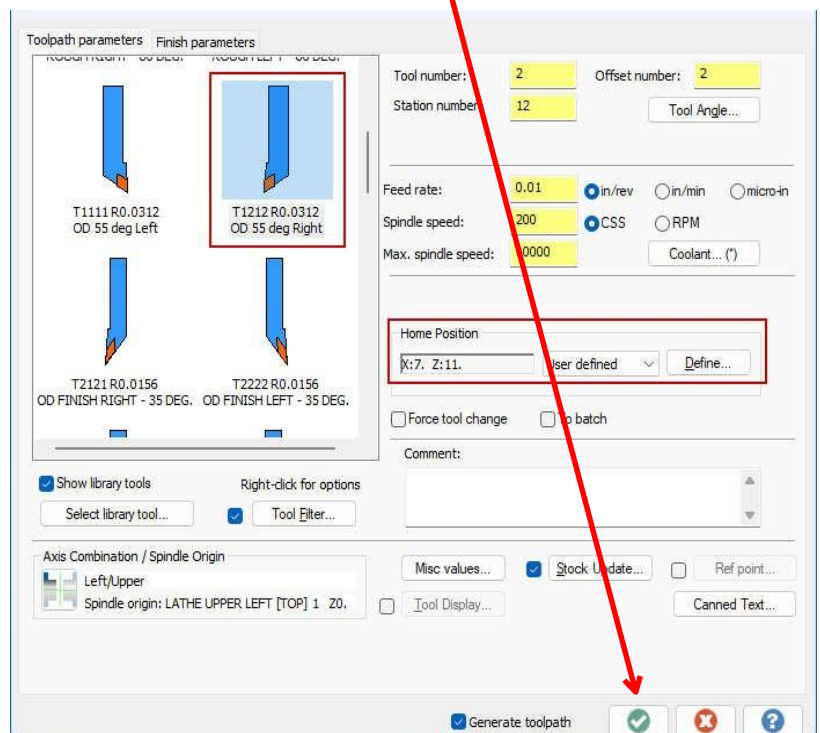


Step 2: In the **Wireframe Chaining** dialog box, click on the **Last** button. Mastercam will select the same chain as the one selected in the previous operation. Select the **OK** button to continue.



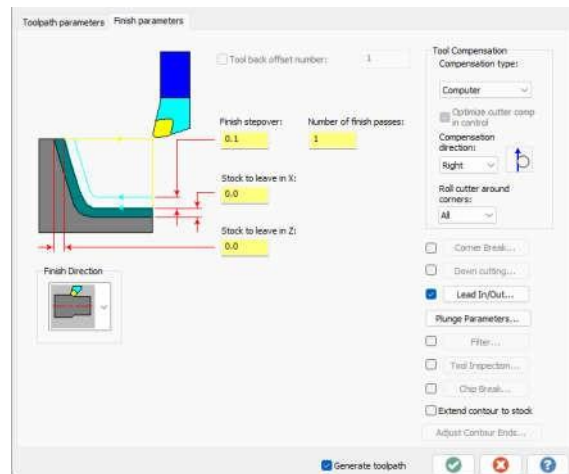
Step 3: Set the Toolpath parameters

- Select the **OD Finish Right 55 deg** tool.
- Make sure that the **Tool/Offset Number** is set to **2**.
- Set the **CoolantFlood On**.
- Set the **Home Position** to **User Defined X 7.0; Z11.0**.



Step 4: Set the Finish Parameters

- Stock to leave in **X** = **0.0"**.
- Stock to leave in **Z** = **0.0"**.
- **Finish Direction** set to **OD**.
- Set the **Tool compensation** to **Computer** and to the **Right**.



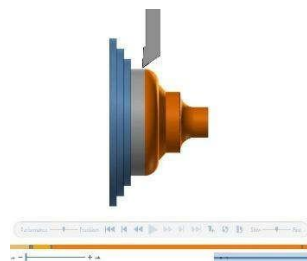
Step 5: Verify all toolpaths. Select the **Select All Operations** button in the **Toolpaths Manager**.



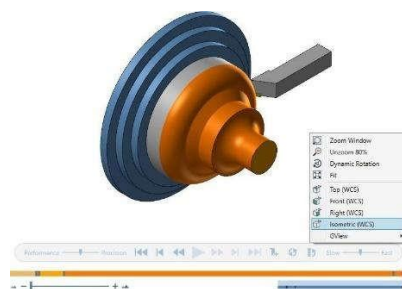
- Select the **Verify** button in the **Toolpaths Manager**



- Select the **Play** button to run Verify. The part should look as shown.



- To see the part from an **Isometric** view right click in the graphics window and select the **Isometric** view. Right mouse clicks in the graphics window and selects the **Top** view.





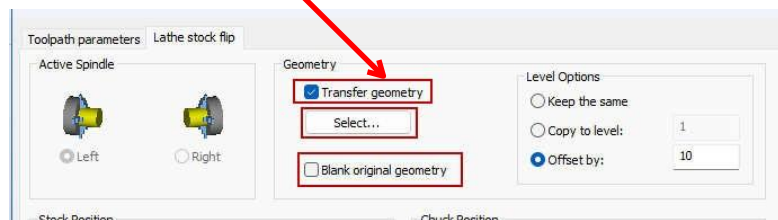
Information Sheet: 4.3-8

Learning Objective: Carry Out Lathe CAM Programming (**Stock Flip**).

Step 1: Flip the Part using The **Stock Flip** Toolpath. From the Part Handling group, select the Stock Flip icon as shown.

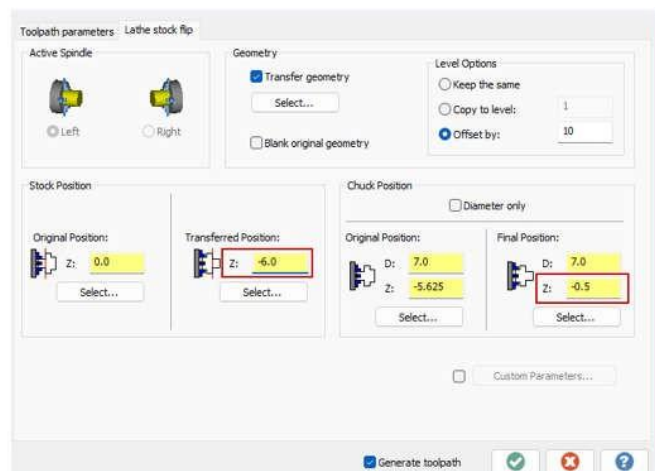
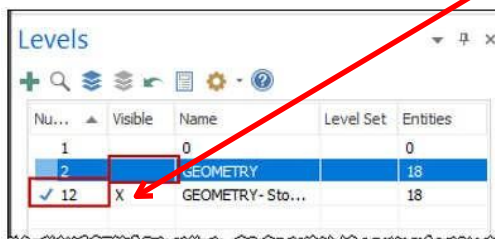


Step 2: Leave **Transfer geometry** enabled and click on the **Select** button and select the entire geometry (Ctrl+A). Disable **Blank original geometry** and in the **Level, options** enable **Offset by 10** as shown. The geometry to be machined will be moved to Level 12 since the current geometry is on level 2.

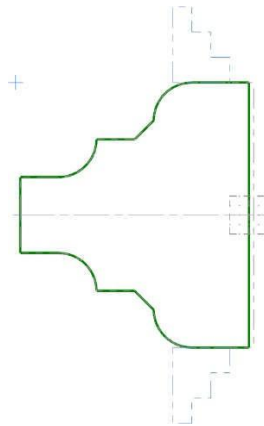


Step 3: Set the Stock Position. Original Position Z = 0.0". **Transferred Position Z = -6.0"**.

- Set the Chuck Position. Leave Original Position D = 7.0; Z = -5.625.
- **Final Position D = 7; Z = -0.5.**
- Select **OK** to transfer the stock.
- Click on the **Levels** tab and make only **Level 12** visible.

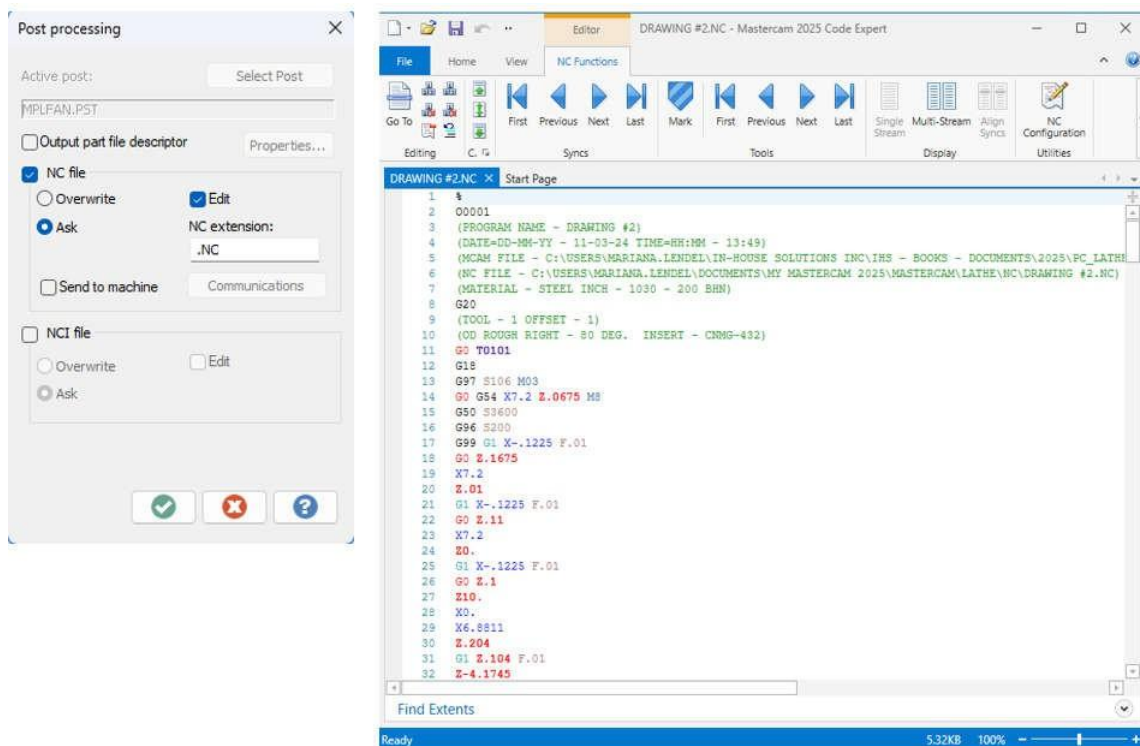


Step 4: The geometry should look as shown.



Step 5: Face the flipped side. Follow the same process mentioned in Information Sheet 4.3-5 at Step 10 to Step 17.

Step 6: Generate G codes. Accept the **NC name**. Save the file. **Mastercam Code Expert** will be launched and the G code will display as shown.





Self-Check: 4.3-5 to 4.3-8

1. How do you properly set up the stock in Mastercam before creating any toolpaths?
2. What is the difference between a constant surface speed (CSS) and a constant revolutions per minute (RPM) cutting method, and when is each used?
3. What is the difference between "Backplot" and "Verify" in Mastercam?
4. How can you use simulation to check for potential collisions or errors?
5. What is Lead in and Lead out?



Answer Script: 4.3-5 to 4.3-8

1. To properly set up the stock in Mastercam, you define the stock shape and dimensions in the "Toolpaths" -> "Stock Setup" menu. You can choose a shape like a cylinder for lathe work and input its dimensions, ensuring it accurately represents the raw material.

2. **Constant Revolutions Per Minute (RPM):** This means that as the tool moves across different diameters on the part, the actual surface speed (how fast the tool is moving over the material) changes.

Constant Surface Speed (CSS): With CSS, the machine automatically adjusts the RPM as the cutting tool moves to maintain a consistent surface speed. The RPM will increase as the tool moves toward the center of the part and decrease as it moves toward the outside diameter.

3. **Backplot:** This is a basic simulation that shows you a wireframe representation of the tool's centerline path.

Verify: It graphically displays what the finished part will look like after the toolpaths have been run.

4. **Lead-in:** It is designed to allow the tool to enter the material smoothly, avoiding abrupt changes in direction that can cause tool deflection, poor surface finish, or premature tool wear.

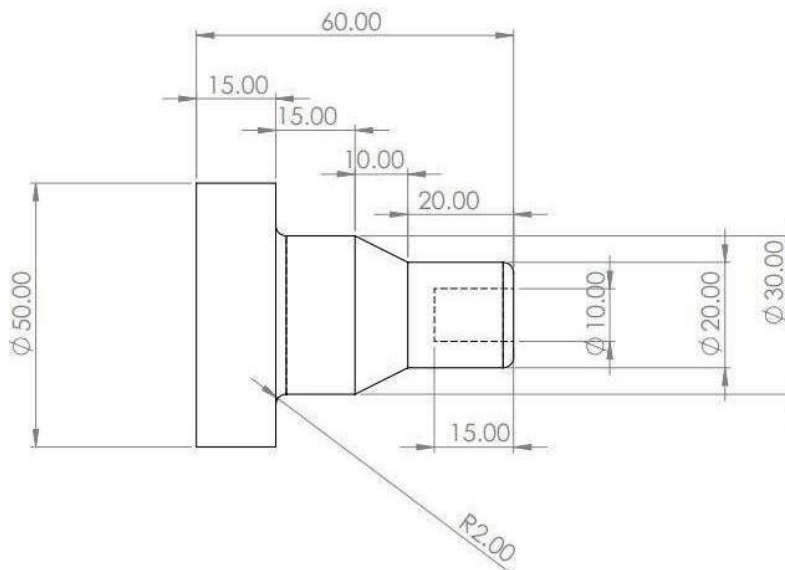
Lead-out: This is the tool's exit path after the cut is finished. It functions in the same way as the lead-in, allowing the tool to exit the cut smoothly and preventing dwell marks or blemishes on the finished part.

Job Sheet: 4.3 (b)

Job	Perform Lathe operations in Mastercam	
Materials/Resources	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
	CAM Software (Mastercam)	1 Pc
Procedure	Collect required materials/resources.	
	Open CAD software (SolidWorks)	
	Use Feature Tools for 3D drawings in SolidWorks.	
	Open CAM Software (Mastercam).	
	Set Origin of the job.	
	Perform Stock-setup.	
	Choose appropriate 2D Lathe operations (Face, Turning, Drilling).	
	Set cutting Tools and parameters.	
	Generate Toolpaths.	
	Visualize simulation.	
	Generate NC codes	
	Follow trainers/instructors.	
	Save Part files in a folder.	

• Sample:

(All dimensions are in Millimeter.)



Learning Outcome 4.4: Carry Out Mill 3D CAM Programming



Content:

- Milling 3D CAM operations (Pocket, Parallel, Contour, Scallop.)
- Tool path generation
- Simulation
- Code generation



Assessment Criteria

1. Be able to analysis CAD file.
2. Work holding and stock definition.
3. Machine origin (work zero) setup.
4. Tooling selection.
5. Simulation, verification, and G-code output.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. A computer set
2. Mastercam software
3. Job sheet with proper drawing and instructions



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Carry Out 2D CAM Programming	• Information Sheet: 4.4-1 to 4.4-2 • Self-Check: 4.4-1 to 4.4-2 • Answer Script: 4.4-1 to 4.4-2

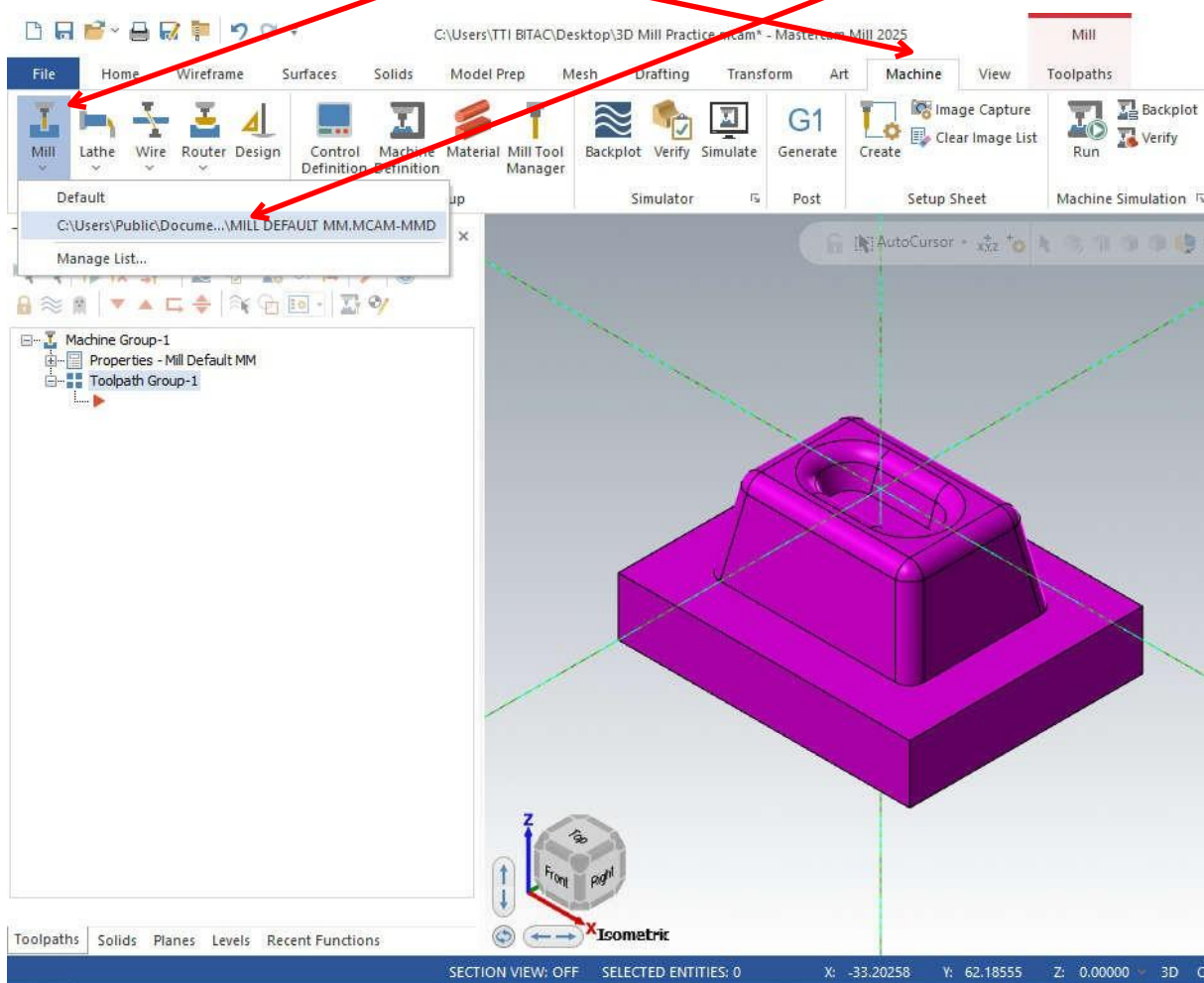


Information Sheet: 4.4-1

Learning Objective: Carry Out Milling 3D CAM Programming (Pocket).

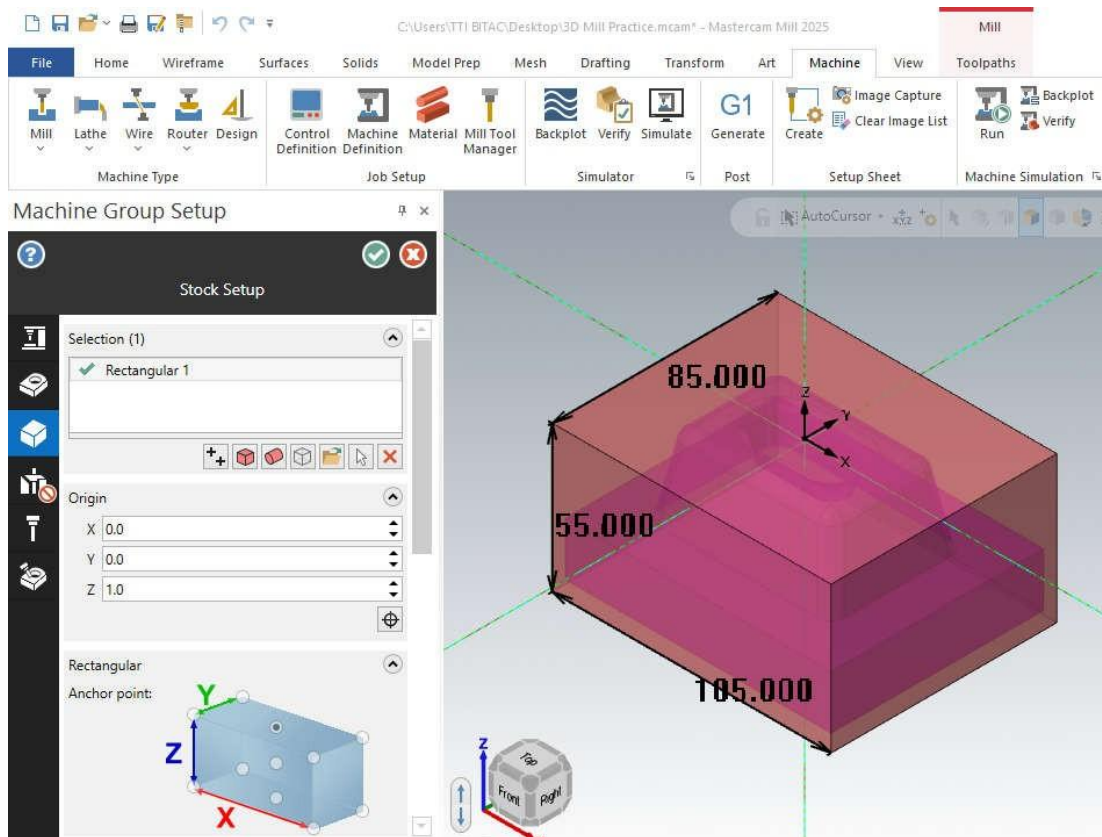
Step 1: Import the .stepAP214 file by following same methodology mentioned in the 2D Milling Information Sheet 4.3-1 (Step 1 to Step 9)

Step 2: Click on **Machine** Tab, click on **Mill**, then select **Mill DEFAULT MM.MCAM-MMD**.

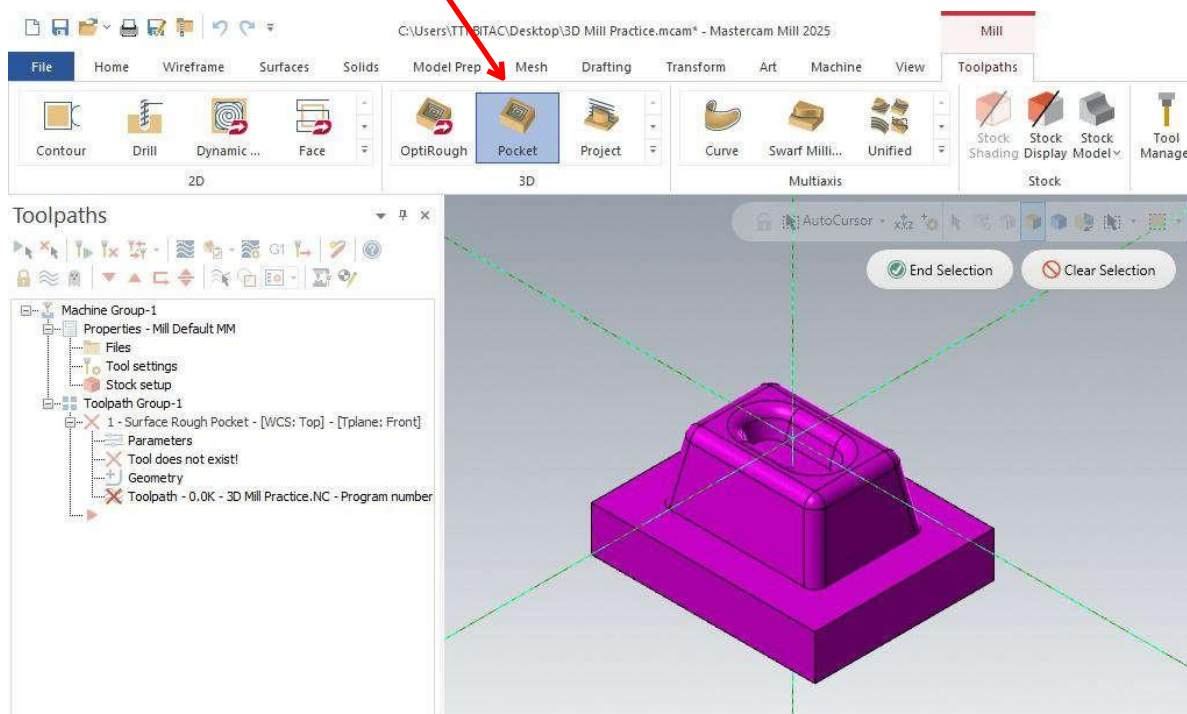


Step 3: Click on **Stock setup** and make stock size.

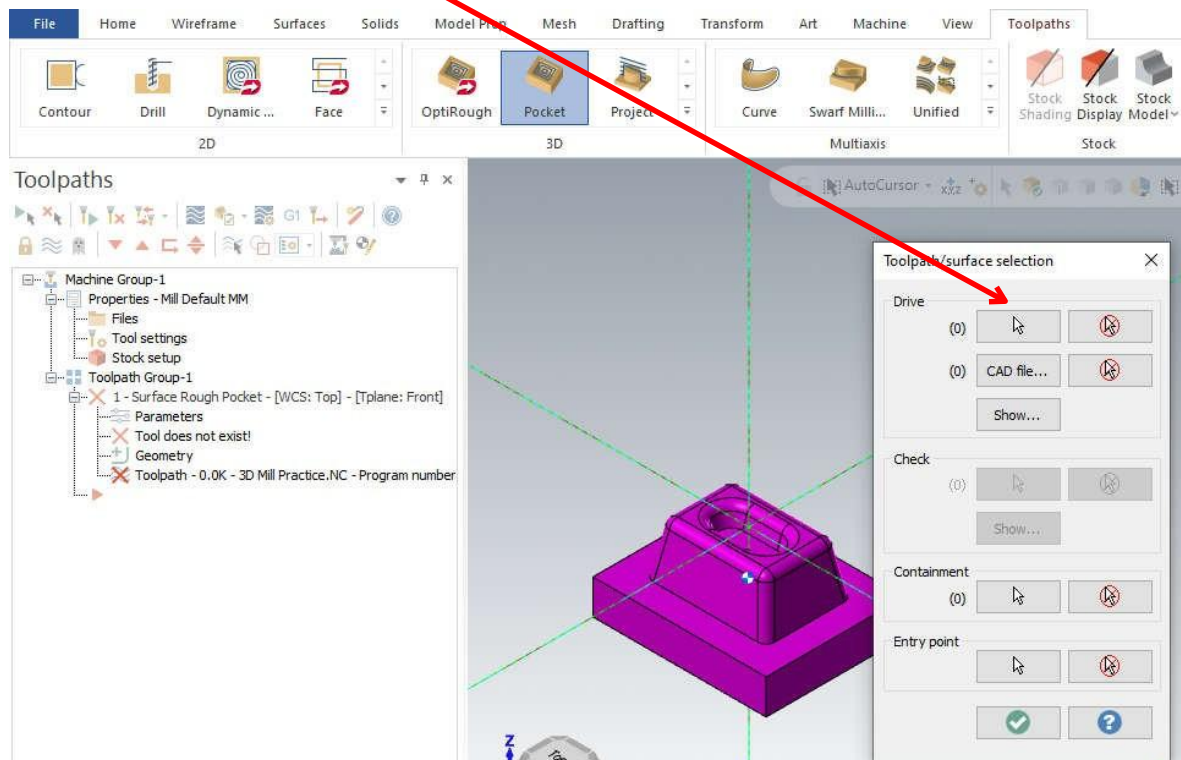
(The Stock setup process is same as the 2D Milling Toolpaths)



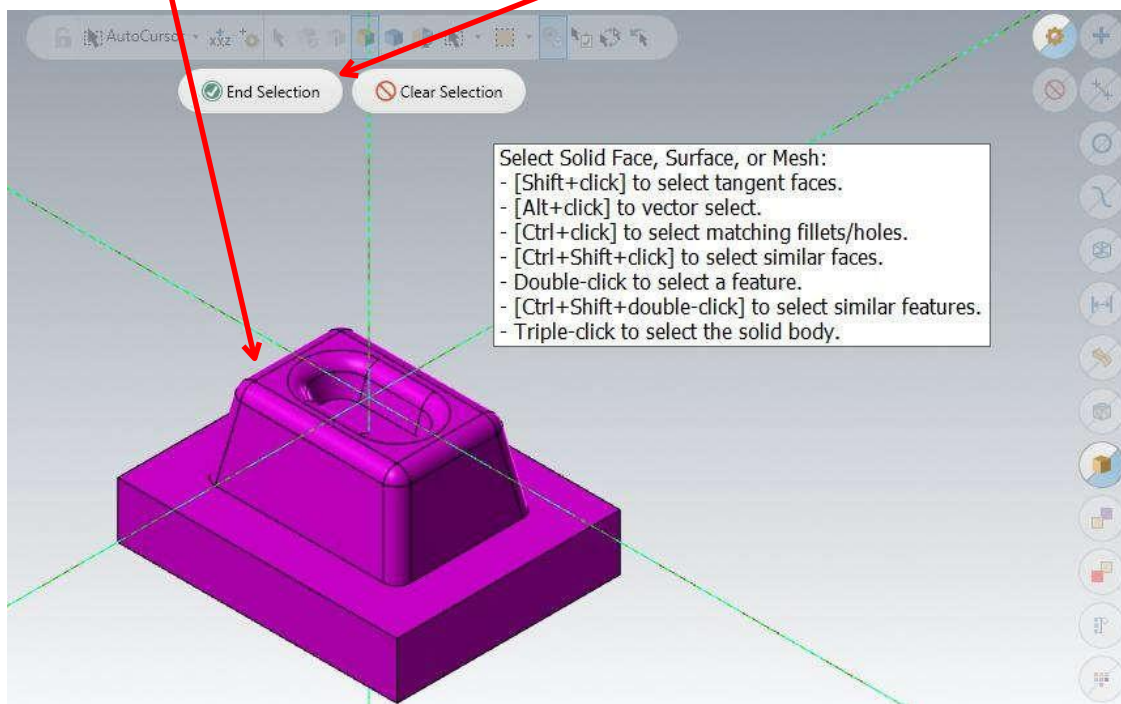
Step 4: Click on **3D Mill Pocket**.



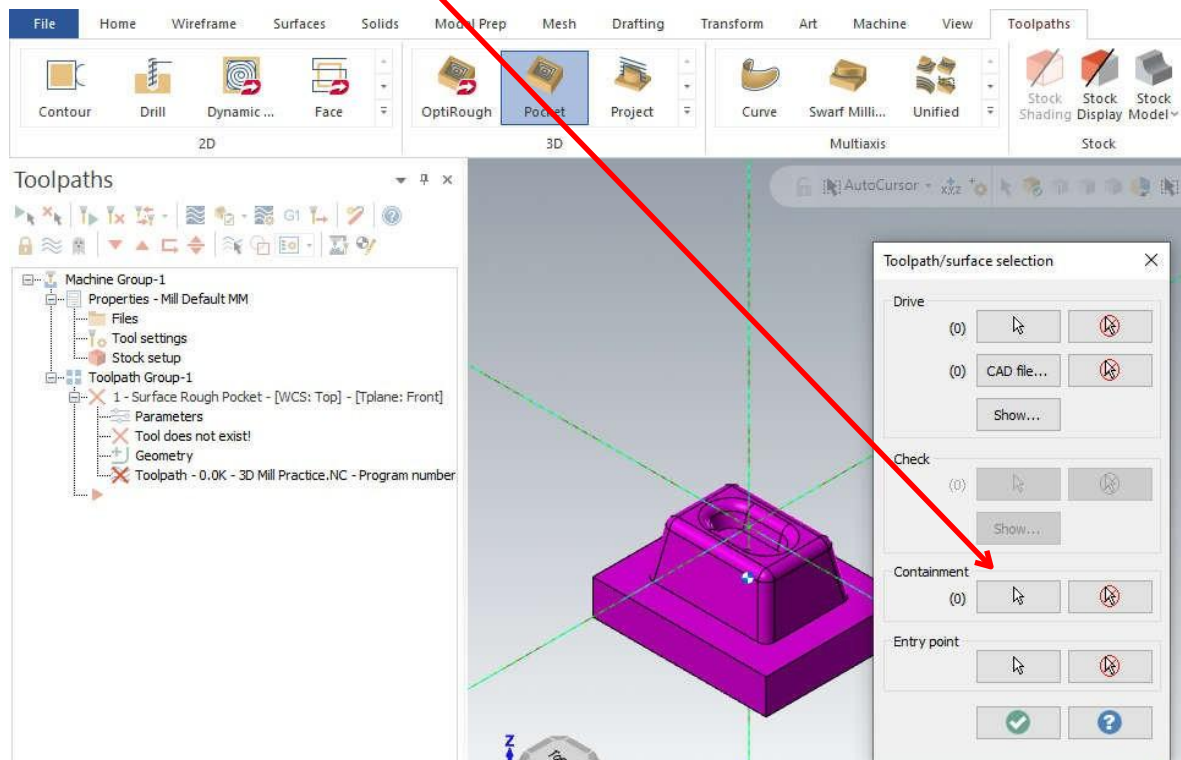
Step 5: Click on Drive.



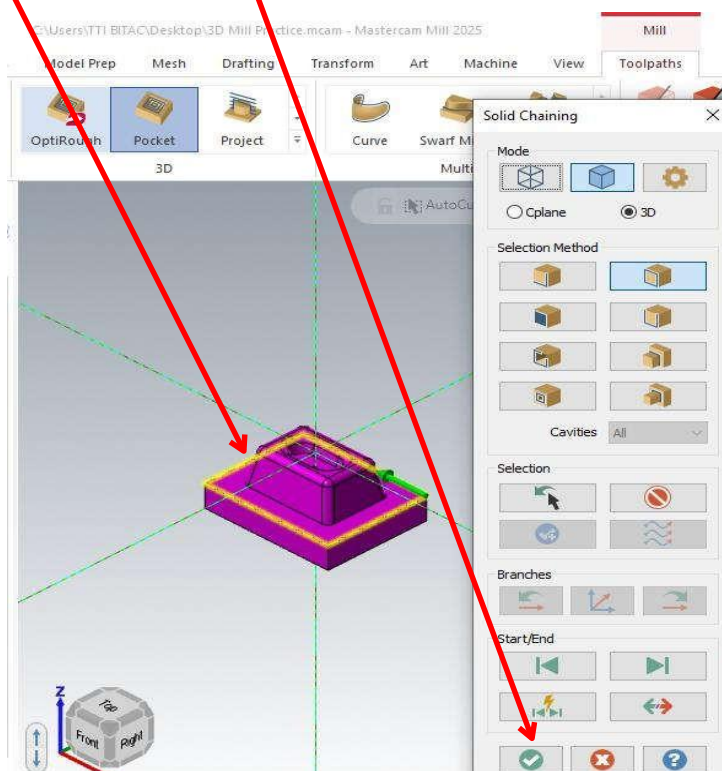
Step 6: Select Full Body and click on Ok and End Selection.



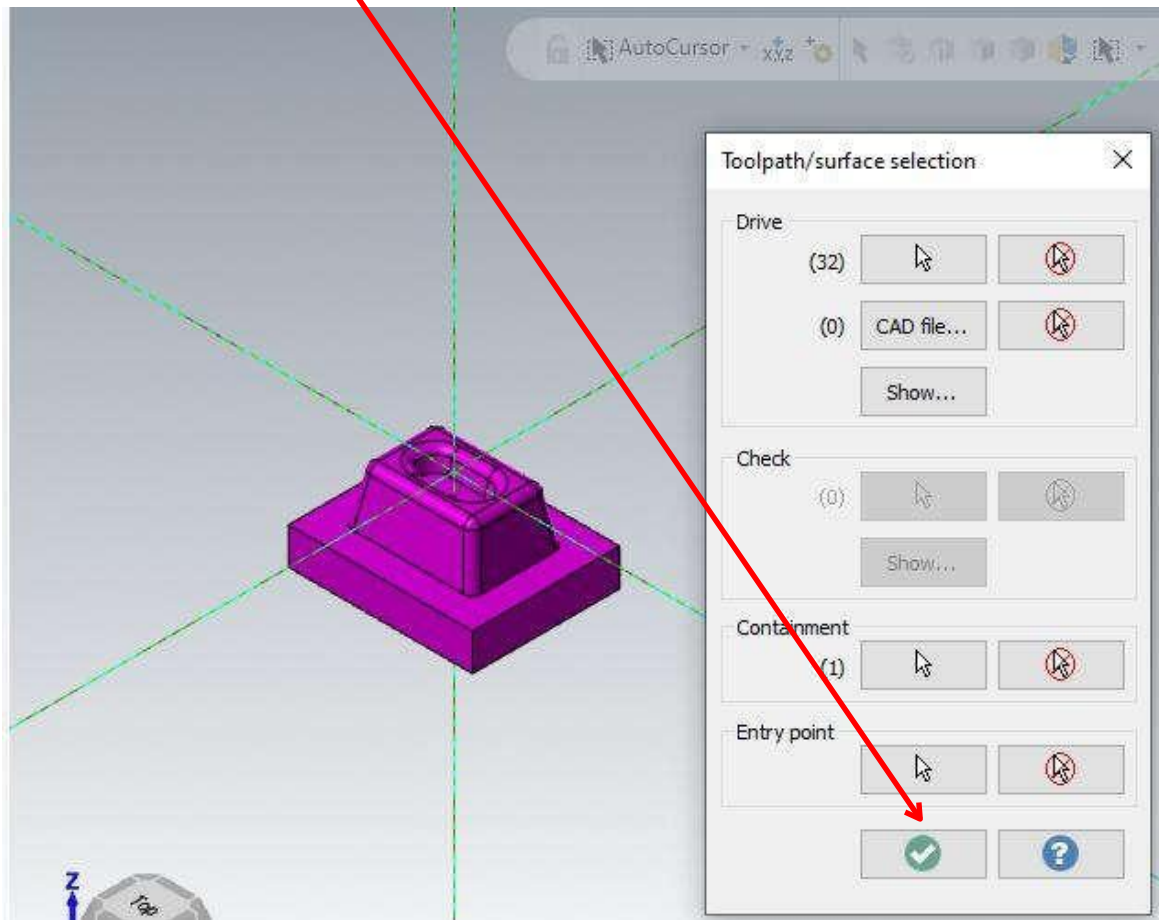
Step 7: Click on **Containment.**



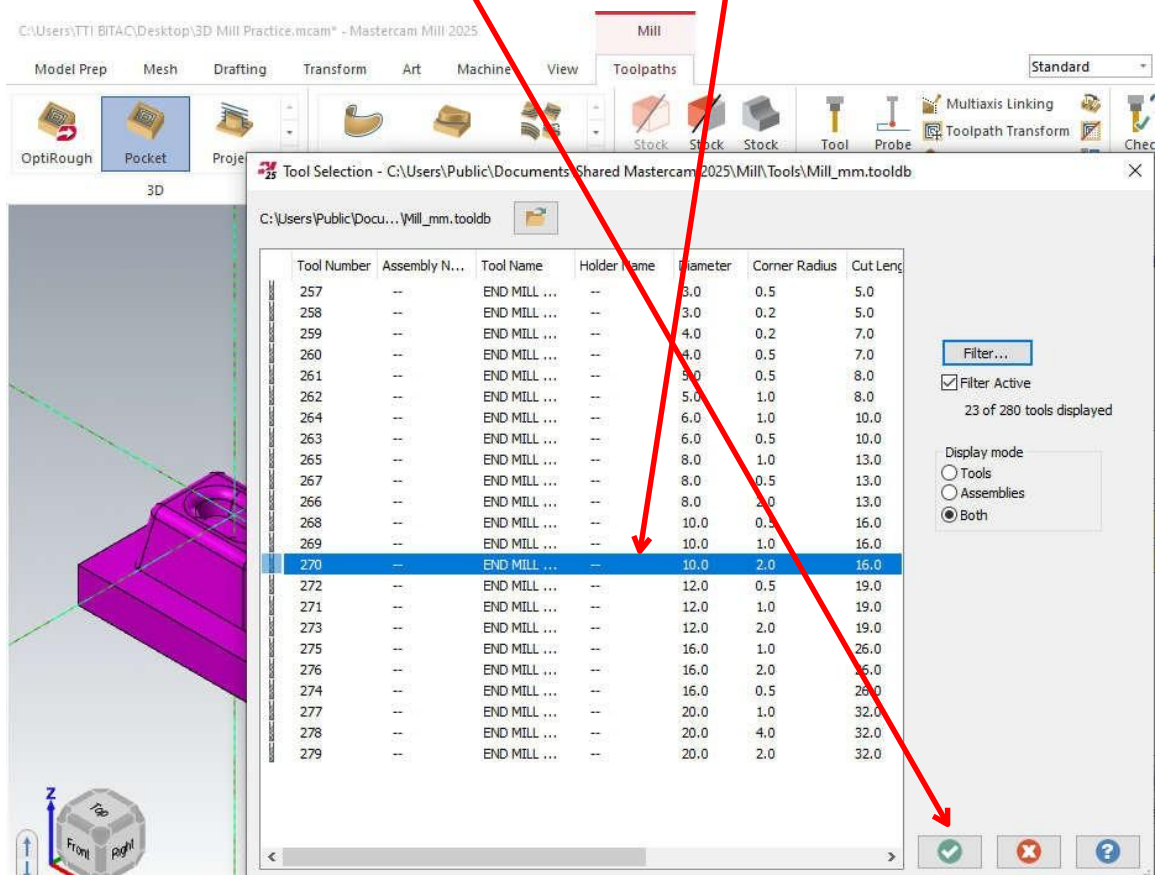
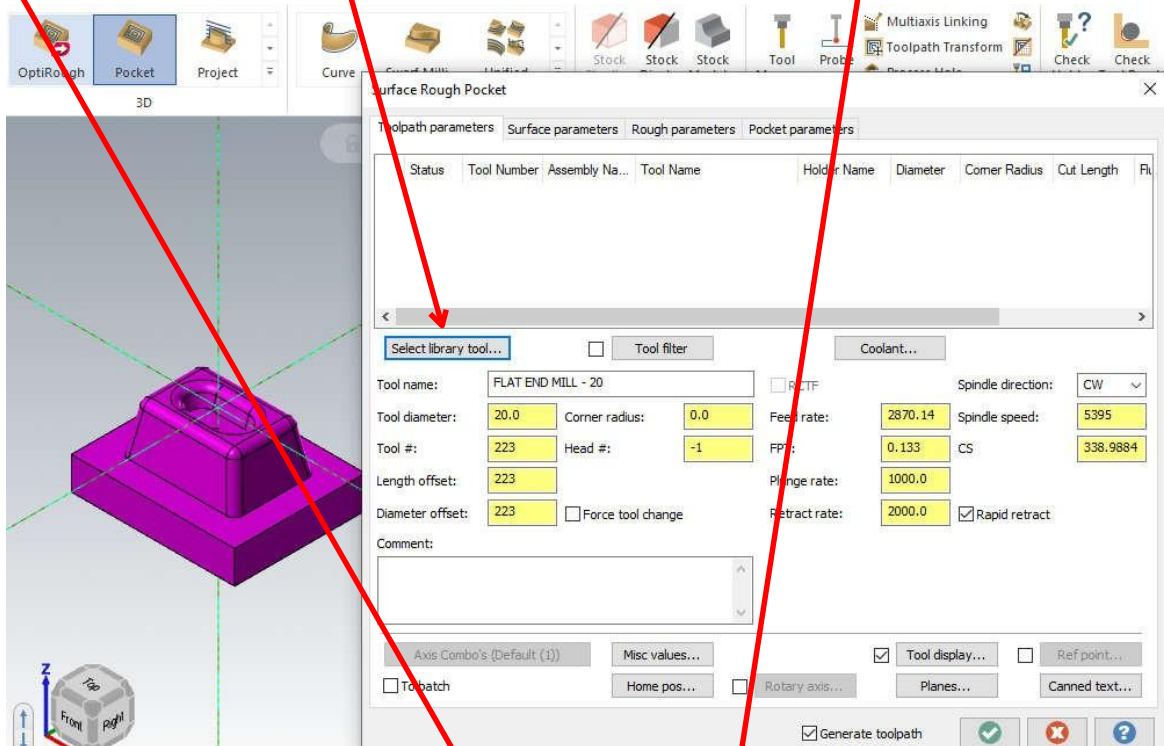
Step 8: Select **Chain and Click on **Ok**.**



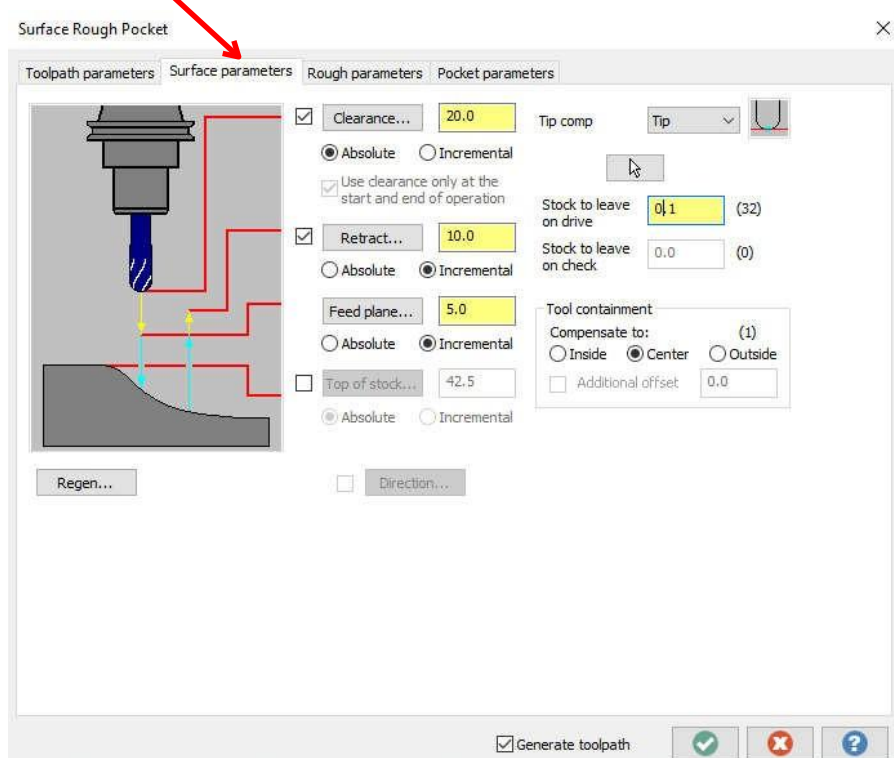
Step 9: Finally press **Ok.**



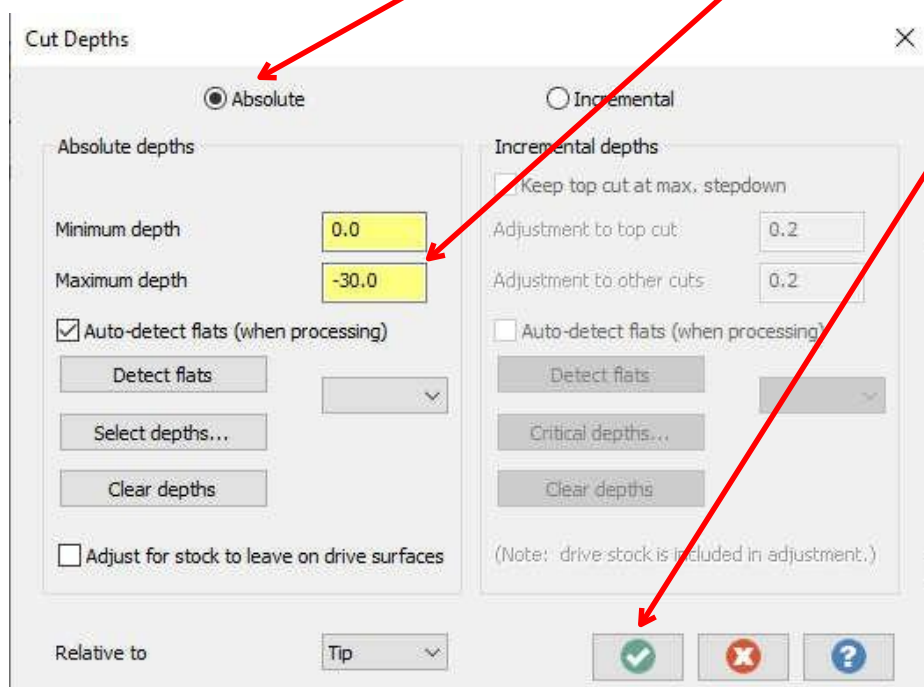
Step 10: Click on **Select library tool** and choose **10mm Bull Mill with corner radius 2** and press **Ok.**



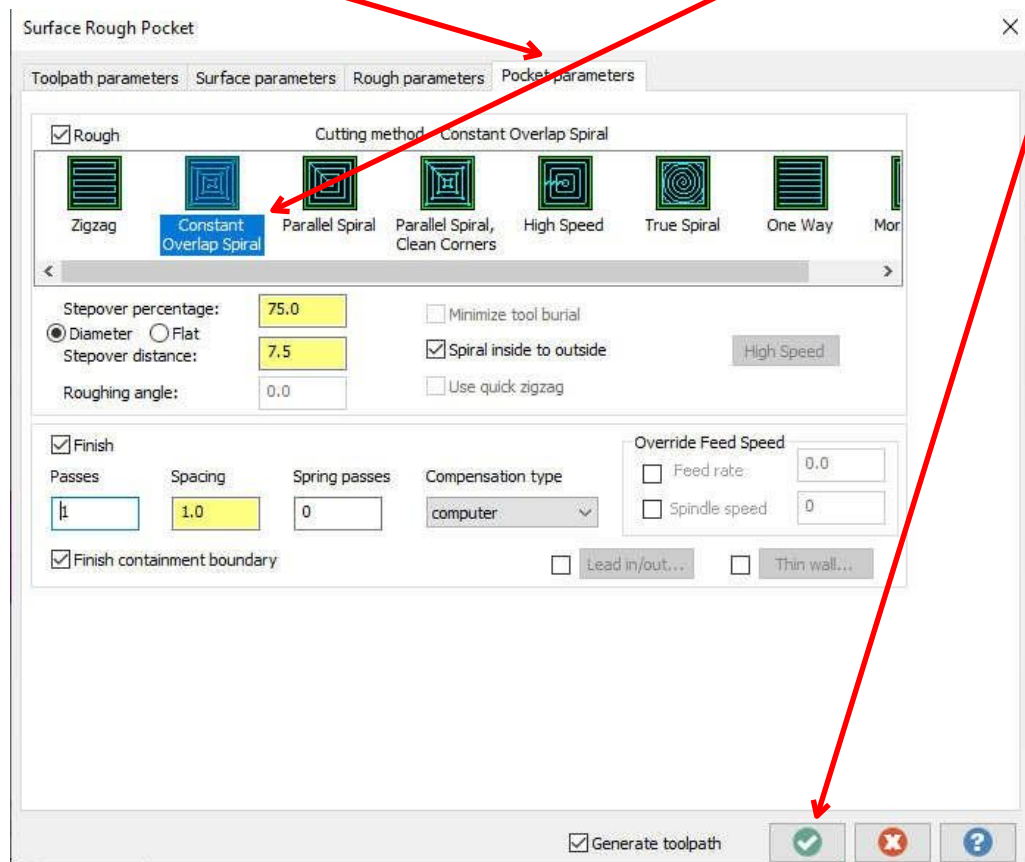
Step 11: Go to **Surface parameters**. Put values as per recommendation.



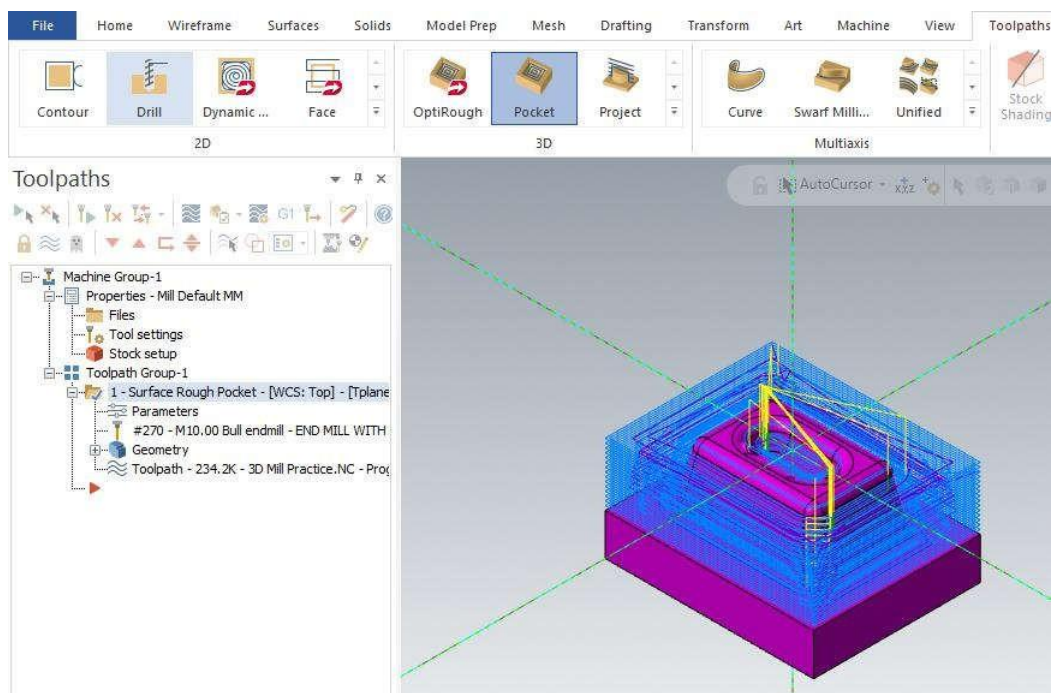
Step 12: Go to **Rough parameters**, select **Absolute** and set on **Cut Depths**. Finally press **Ok**.



Step 13: Go to **Pocket parameters** and choose **Constant Overlap Spiral**. Finally press **Ok**.



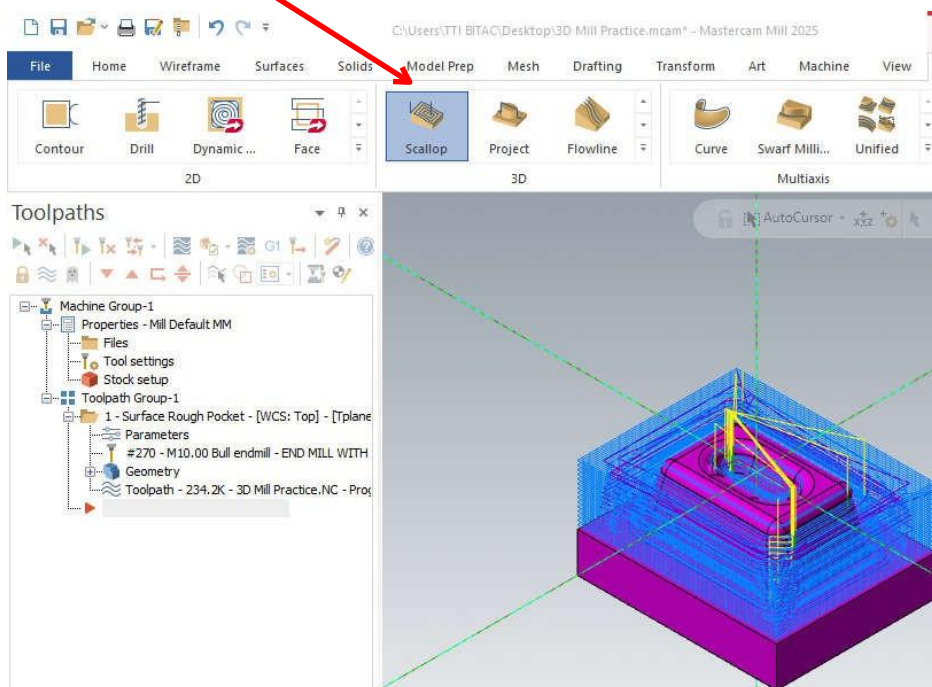
Step 14: Mill 3D Tool path is generated for Pocket operations.



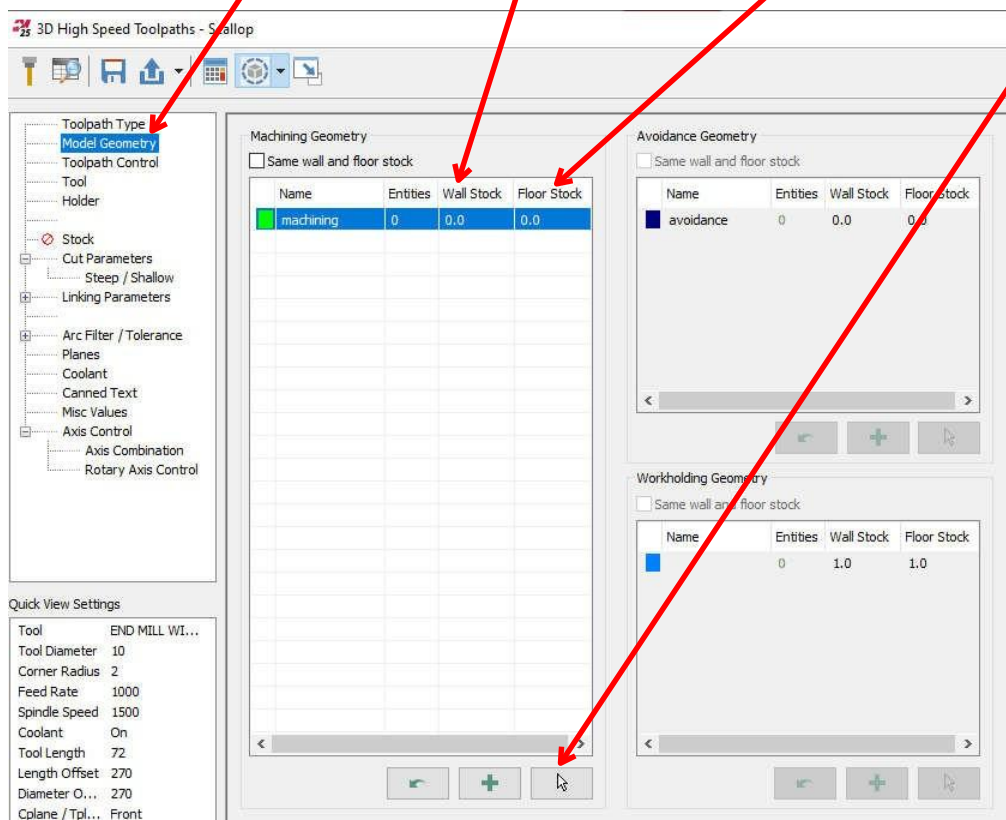


Information Sheet 4.4-2

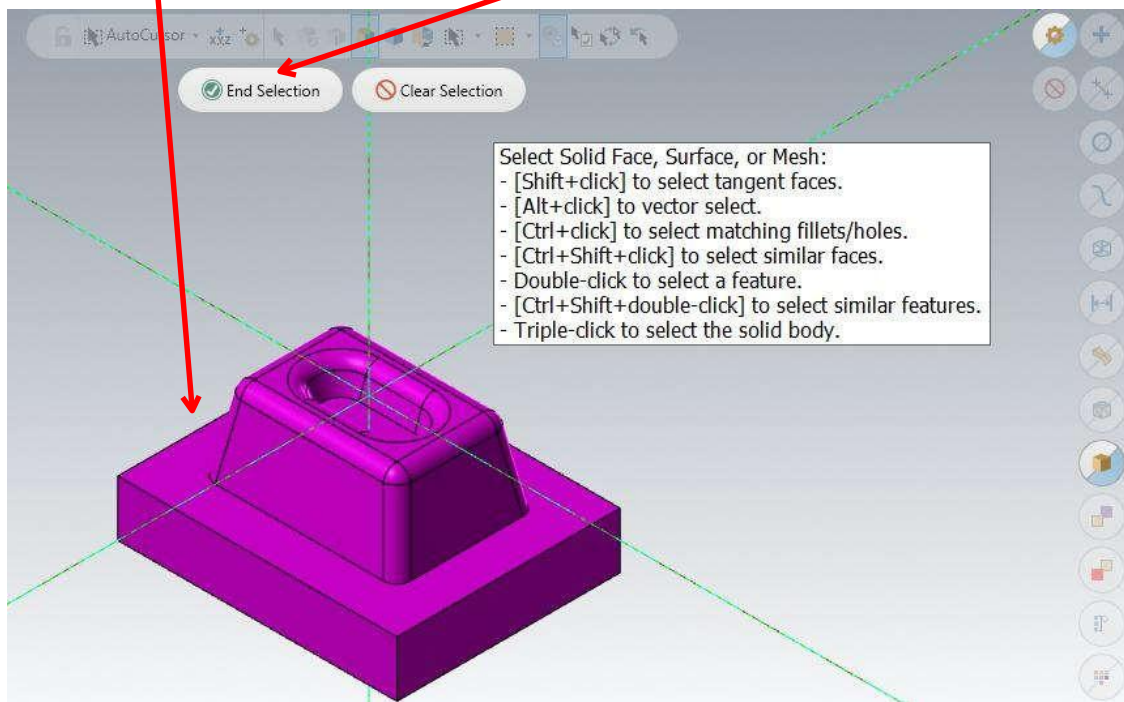
Step 1: Click on 3D Mill **Scallop** operation.



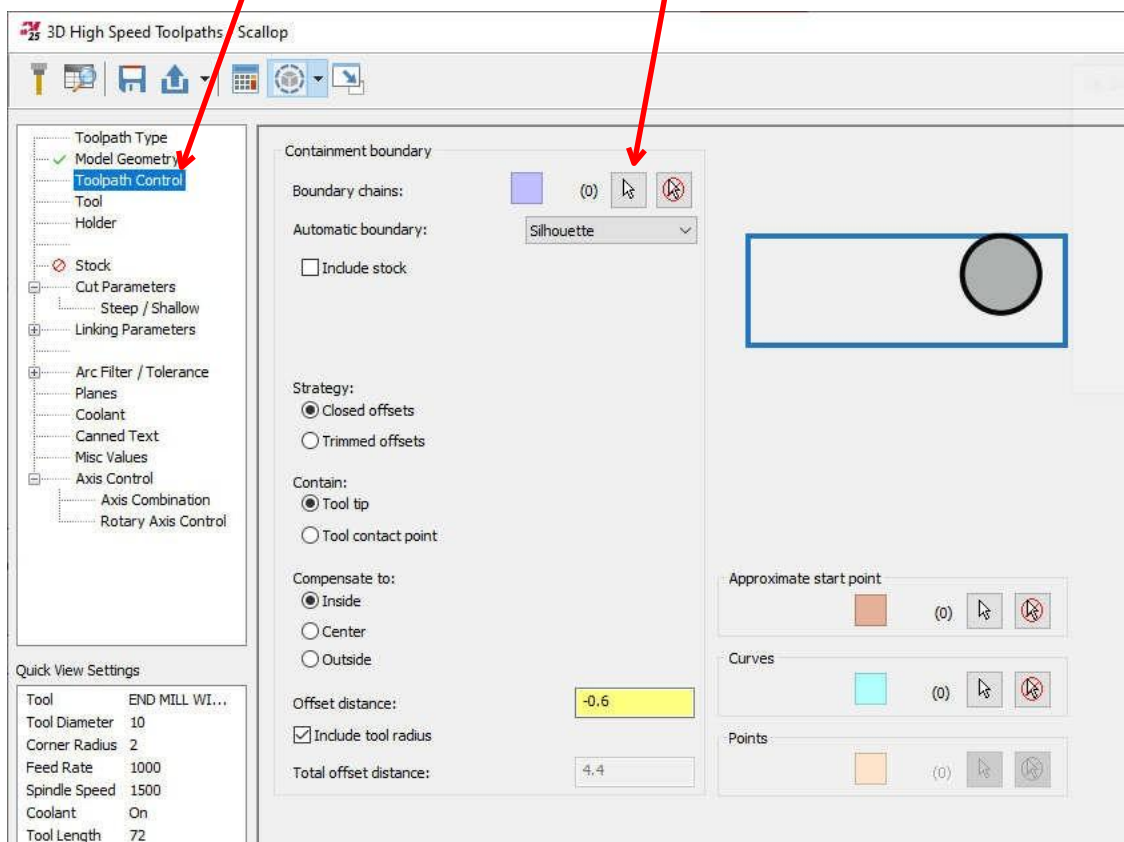
Step 2: Click on **Model Geometry**. Keep Wall Stock and Floor Stock 0.0. Click on **Containment**.



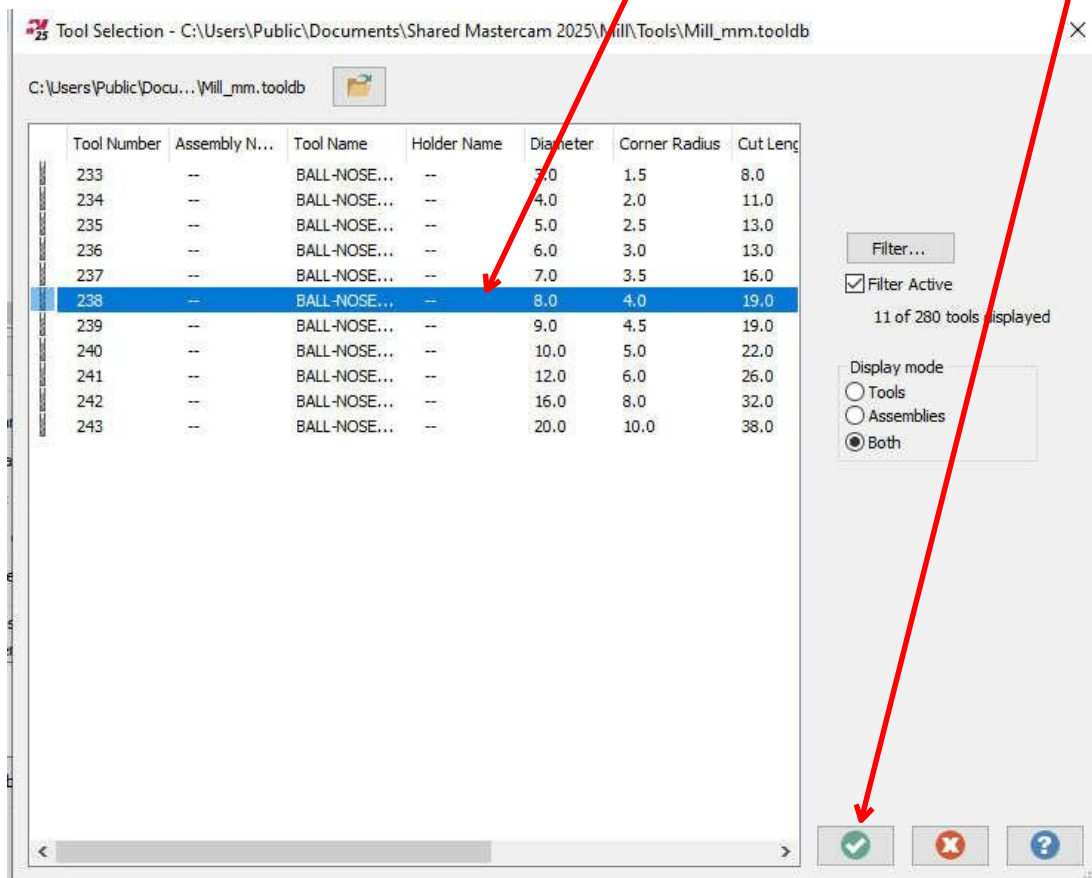
Step 3: Select **Full Body and click on **End Selection**.**



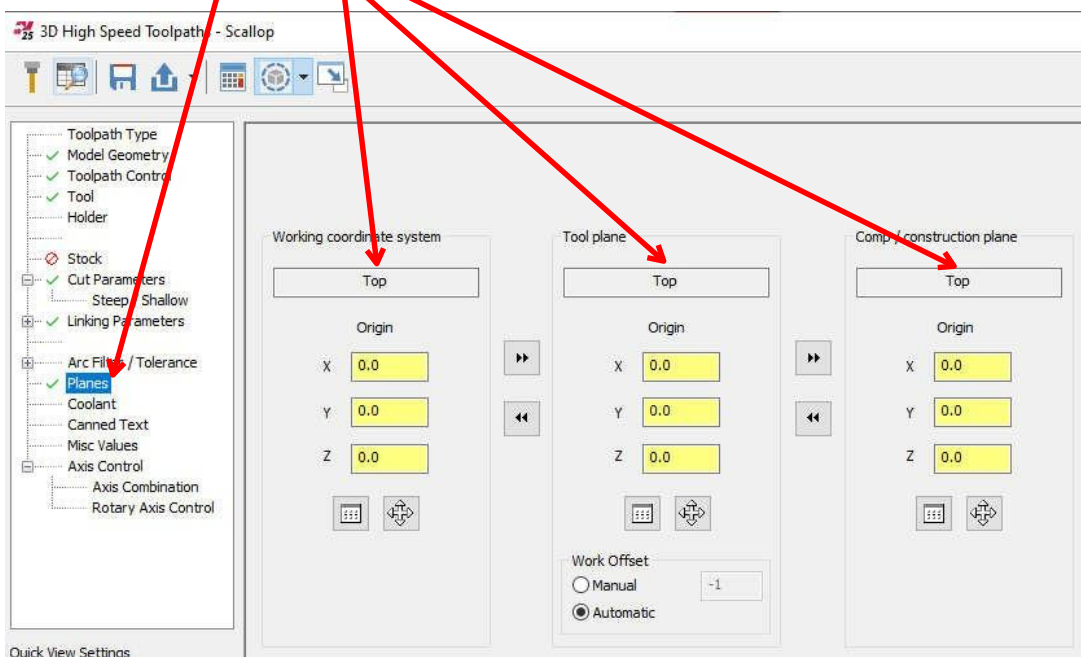
Step 4: Go to **Toolpath Control, and select **Boundary chains**.**



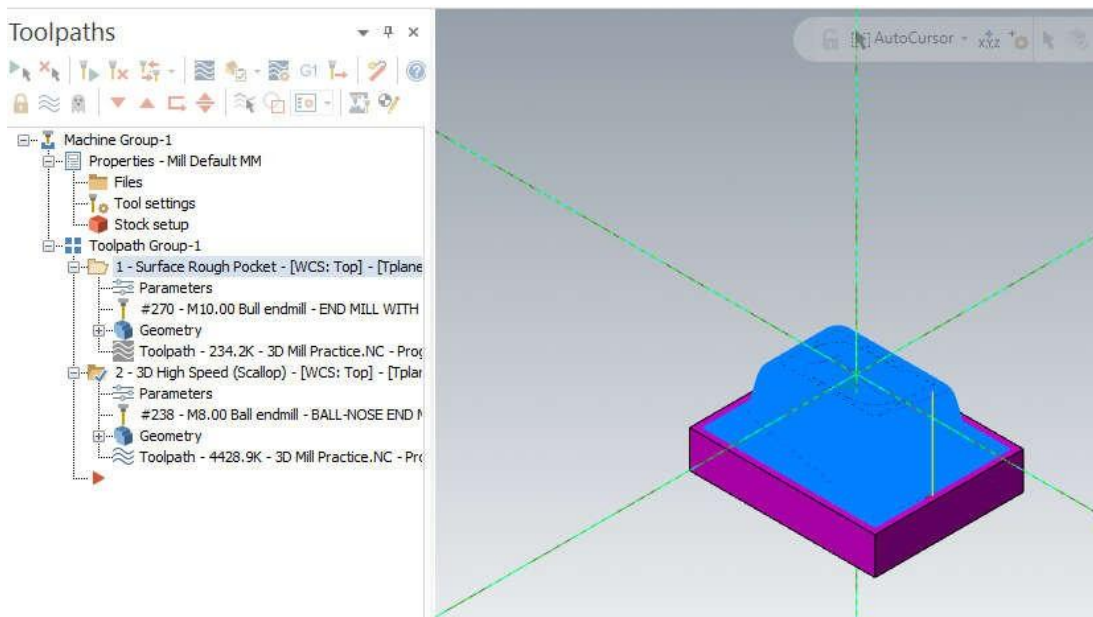
Step 5: Go to **Tool Library** and select **8mm Ball with corner radius 4.** Then click on **Ok.**



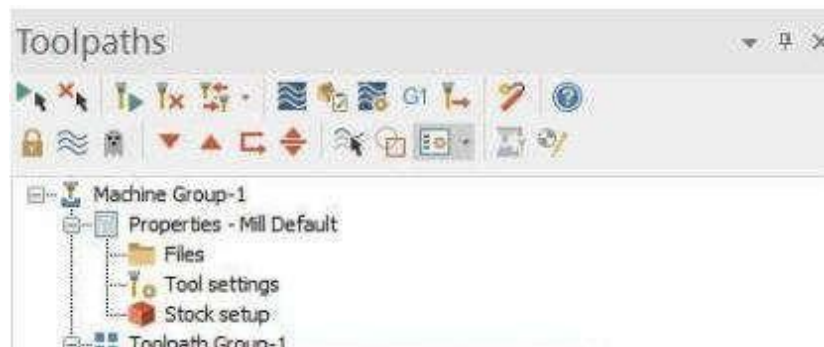
Step 6: Ensure all **Panes** are **Top.**



Step 7: Tool path generation is done for mill 3D operations (Pocket and Scallop).



Step 8: Backplotting and Verifying. In the **Toolpaths Manager**, check that your 2D Contour operation is selected.



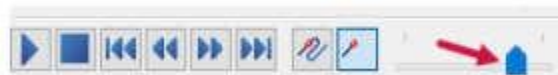
Step 9: Click Backplot selected operations. **Backplot dialog box:** Controls the backplot display and allows you to save geometry or settings. **Play bar:** Adjusts the speed of the simulation and allows you to pause, play, and set stops. **Tool and holder:** Simulate the selected tool and holder.



Step 10: Click the **Play** button or press [R].



Step 11: When the simulation has finished, change the simulation speed by adjusting the run speed slider.



Step 12: Click OK in the Backplot dialog box



Step 13: Post Processing the file. **Click** Select all operations.



Step 14: Select the **Post selected operations** button in the **Toolpaths Manager**.



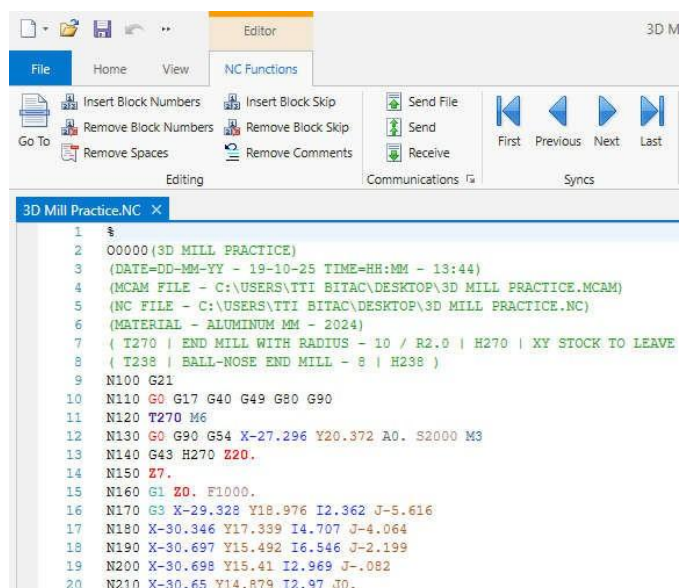
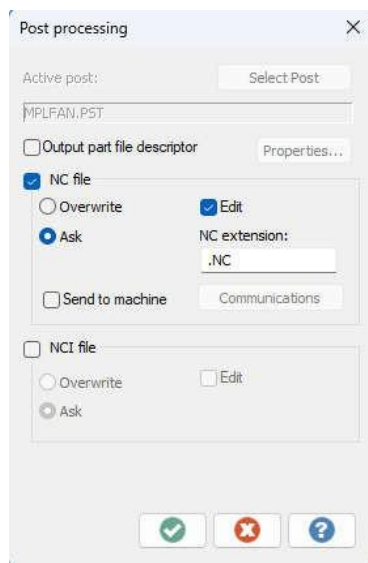
- Accept the **NC name**.
- Save the file.
- **Mastercam Code Expert** will be launched and the G code will display as shown.

Post processing the file

- Select the **Post selected operations** button in the **Toolpaths Manager**.



- Accept the **NC name**.
- Save the file.
- **Mastercam Code Expert** will be launched and the G code will display as shown.





Self-Check: 4.4.-1 to 4.4-2

1. What is the purpose of the "Stock to leave" parameter in a 3D Pocket toolpath?
2. Name two different methods a tool can use to enter the material when roughing with a pocket toolpath.
3. What type of tool is typically used for a 3D scallop finishing pass, and why?



Answer Sheet: 4.4.-1 to 4.4-2

1. In a roughing toolpath, leaving a small amount of material (**positive stock**) on the walls and floors ensures there is material left for a subsequent, more precise finishing pass.

This protects the finishing tool from excessive wear and allows for a better surface finish, since the finishing tool is only removing a small, consistent amount of material.

For a finishing toolpath, the stock should be set to zero to remove all remaining material up to the part's final dimensions.

2. **Two methods a tool can use to enter the material when roughing with a pocket toolpath:**

Ramping (Linear): The tool descends into the material at a shallow angle while moving linearly in a zigzag or zig-only pattern. This reduces stress on the tool compared to a straight vertical plunge.

Helical Interpolation: The tool spirals down into the material in a circular path. This is a very common and safe method, as it reduces the force on the tool's tip and ensures efficient chip evacuation.

3. A round ball mill is used because its rounded tip allows it to produce a consistent stepover (scallop height) regardless of the surface angle. This results in a uniform surface finish across complex, contoured, or organic shapes.

Learning Outcome 4.5: Load and Run Program



Content:

- Machine Set-up (Lathe, Milling)
- Parameter setting
- Software (SolidWorks, Mastercam)
- Tool and Work offset
- Dry Run (Simulation)



Assessment Criteria

1. Program is loaded using appropriate device.
2. Dry run/simulation is performed in machine as per standard operating procedure.
3. Program is executed to produce work piece.
4. Equipment is cleaned and maintained as per standard operating procedure.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. Workplace (simulated or actual)
2. Job specifications, drawings or work instructions
3. Learning manual/CBLM



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Load and Run Program	• Information Sheet: 4.5-1 to 4.5-3 • Job Sheet: 4.5 (a) to 4.5 (b) • https://www.youtube.com/watch?v=wHiwV6FBrY8&t=35s • https://www.youtube.com/watch?v=MkA6NC3Ex9U



Information Sheet: 4.5-1

Learning Objective: Load and Run Program in **HAAS Controller** CNC Milling

To load and run a program on a CNC Milling Machine, you will follow a general process of starting the machine, transferring your G-code program, setting up your work offsets and tools, and running the program in a safe, controlled manner. This guide covers the standard procedure, but always refer to your specific machine's manual for detailed safety protocols and instructions.

Pre-run safety and preparation:

- 1. Clear the work area.** Ensure there are no tools, rags, or other obstructions on the table or inside the machine's enclosure.
- 2. Verify air supply.** Check that the machine has a sufficient air supply, typically at least 70 PSI, for the tool changer to operate correctly.
- 3. Release the E-stop.** If the red Emergency Stop button is engaged, twist it clockwise to release it.

Step 1: Power up the machine.

- Press the green **POWER ON** button, then press the **RESET** button twice to clear any alarms. Press **POWER UP RESTART** to home all axes and clear the service-off message.
- **Release the E-stop button** by twisting it clockwise.
- Press the green **POWER ON** button, then press **RESET** twice to clear alarms.
- Press **POWER UP RESTART** to home all axes and clear the service message.
- If the mill has not run that day, run the spindle warm-up program (**O02020**) by pressing **LIST PROG**, navigating to the program, and pressing **CYCLE START**.

Step 2: Transfer the G-code program

- Use a USB flash drive to copy your G-code file ( or ) to the machine's memory.
- Insert your USB drive into the control panel's USB port.
- Press the **LIST PROG** button.
- On the screen, press the left arrow key to switch to the device list and highlight "USB Device".
- Use the arrow keys to find and select your program file.
- Press **F2** to copy the program from the USB drive to the machine's memory.
- A dialog box will appear. Select "Memory" and press **ENTER** twice.
- The program is now loaded in the machine's memory. Press the **MEM** button to activate it.

Step 3: Set work offsets (G54)

- **Clamp your material** in the vise on the machine table.
- Go to **HANDLE JOG** mode and use an edge finder or probe to find the X and Y edges of your part.
- On the control panel, press the **OFFSET** button and navigate to the **WORK** offsets tab.
- Jog the tool to the X-zero position on your part.
- Highlight the G54 **X-axis** value on the work offset page and press **PART ZERO SET**.
- Repeat for the **Y-axis**.
- For the **Z-axis**, jog the tool down until it just touches the top surface of your part. Highlight the G54 **Z-axis** value and press **PART ZERO SET**.

Step 4: Set tool length offsets

- Call the first tool for your program (e.g., in MDI, enter **T1 M6** , then press **CYCLE START**).
- Press **HANDLE JOG** and use the jog handle to bring the tool tip close to the top of the part.
- Place a piece of paper on the part surface and slowly jog the tool down in the Z-axis until it holds the paper.
- Navigate to the **TOOL** offset page.
- Highlight the **LENGTH GEOMETRY** value for the tool you are measuring and press **TOOL OFFSET MEASURE**.
- Repeat for each tool used in your program by calling the next tool and repeating the process.

Step 5: Run the program safely

- Switch to **MEM** (Memory) mode.
- Turn on **SINGLE BLOCK** to run the program one line at a time for verification.
- Lower the **FEED RATE OVERRIDE** to a low percentage (e.g., 25%).
- Close the machine's doors, which must be latched for the program to run in Memory mode.
- Press **CYCLE START**. The machine will execute the first block and stop.
- Continue to press **CYCLE START** for each block. Use **FEED HOLD** to pause if you see any problems.
- Once you are confident in your setup, you can turn off **SINGLE BLOCK** and increase the feed rate override.
- **Monitor the machining process** closely for any issues and be ready to press the **E-stop** in an emergency.



Information Sheet: 4.5-2

Learning Objective: Load and Run Program in **Fanuc Controller** CNC Milling Machine.

Pre-run safety and preparation:

1. **Clear the work area.** Ensure there are no tools, rags, or other obstructions on the table or inside the machine's enclosure.
2. **Verify air supply.** Check that the machine has a sufficient air supply, typically at least 70 PSI, for the tool changer to operate correctly.
3. **Release the E-stop.** If the red Emergency Stop button is engaged, twist it clockwise to release it.

Step 1: Power up and prepare the Mill

- **Switch on the main power:** Turn on the main power switch, usually located on the electrical cabinet.
- **Press the Power ON button:** Press the green **POWER ON** button on the controlpanel.
- **Release the Emergency Stop button:** If the red Emergency Stop button is pushed in, twist it clockwise to release it.
- **Perform Zero Return (Homing):**
 - Press the **ZERO RETURN** button to enter this mode.
 - Press the plus (+) buttons for each axis (X+, Y+, Z+) to send the machine to its home or reference position. You will see an indicator light illuminate for each axis when it is homed.

Step 2: Transfer the G-code program from a USB drive

- **Insert the USB drive:** Plug your USB drive, containing the G-code file (ending in `.NC` or `.TXT`), into the port on the control panel.
- **Enter Edit mode:** Press the **EDIT** mode button on the control panel.
- **Navigate to the Program screen:** Press the **PROG** button.
- **Initiate file transfer:**
 - Press the **OPRT** softkey below the screen.
 - Use the arrow softkeys to find and press the **DEVICE CHANGE** softkey.
 - Select "USB Memory" from the list of devices.

Step 3: Copy the Program

- Press the **FILE INPUT** softkey, then the **SELECT PROGRAM** softkey, and choose your file.
- Press the **EXEC** softkey to copy the program from the USB to the machine's internal memory.

Step 4: Set work and tool offsets

This process tells the machine the location of your part zero and the lengths of your tools.

Set the work offset (G54)

- In **JOG** mode, manually move a tool to the X, Y, and Z zero positions on your part.
- Press the **OFFSET SETTING** button and navigate to the **WORK** tab.
- With your cursor on the G54 X-value, type `X0` and press the **INPUT** softkey.
- Repeat for `Y0` and `Z0`. The machine will now use the current machine position for G54.

Step 5: Set the tool length offsets (H)

- Mount the first tool in the spindle using the tool change command (T# M6) in **MDI** mode.
- Go to **JOG** mode and move the tool to the Z-zero position on your part, using a piece of paper or a probe for accuracy.
- Press the **OFFSET SETTING** button and navigate to the **TOOL** tab.
- Highlight the **LENGTH GEOMETRY** value for your tool number.
- Type **Z** and press the **MEASURE** softkey. The machine will calculate and input the length offset.
- Repeat for all tools used in the program.

Step 6: Run the program safely

➤ Select the program:

- Press the **MEM** (Memory) mode button.
- Press the **PROG** button and use the arrow keys to select your program from the list.

➤ Enable safety features:

- Press the **SINGLE BLOCK** button. This will run the program one line at a time for safety.
- Set the **FEED RATE OVERRIDE** to a low percentage (around 25%) to start slowly.

➤ Start the cycle:

- Press the **CYCLE START** button. The machine will execute the first block of code.

➤ Monitor the cut:

- For each subsequent block, press **CYCLE START**.
- Keep your hand on the **FEED HOLD** button to pause the program immediately if anything goes wrong.

➤ **Remove safety features gradually:** Once you have verified the initial cuts, you can turn off **SINGLE BLOCK** and increase the feed rate override as confidence grows.

➤ **Complete the process:** After the program finishes, turn off the spindle and coolant, clean up, and measure your finished part.



Information Sheet: 4.5-3

Learning Objective: Load and Run Program in **Haas Controller** CNC Lathe Machine.

To load and run a program on a CNC Lathe Machine, you will follow a general process of starting the machine, transferring your G-code program, setting up your work offsets and tools, and running the program in a safe, controlled manner. This guide covers the standard procedure, but always refer to your specific machine's manual for detailed safety protocols and instructions.

Pre-run safety and preparation:

1. **Clear the work area.** Ensure there are no tools, rags, or other obstructions on the table or inside the machine's enclosure.
2. **Verify air supply.** Check that the machine has a sufficient air supply, typically at least 70 PSI, for the tool changer to operate correctly.
3. **Release the E-stop.** If the red Emergency Stop button is engaged, twist it clockwise to release it.

Step 1: Power up and prepare the Lathe

- **Release the E-stop:** Twist the largered Emergency Stop button clockwise to release it.
- **Press Power On:** Push the green **POWER ON** button and allow the control to bootup.
- **Reset:** Press the **RESET** button twice to clear any alarms.
- **Power Up Restart:** Press the **POWER UP RESTART** button. This will home the machine's axes to their reference points.

Step 2: Load the G-code program

- Transfer your G-code file (typically a `.nc` or `.txt` file) to the lathe's internal memory via a USB drive.
- **Insert the USB drive:** Plug your USB flash drive into the control panel's USB port.
- **Open the device manager:** Press the **LIST PROG** button. The device manager screen will appear, showing available memory devices as tabs.
- **Navigate to the USB device:** Use the left or right arrow keys to select the "USB Device" tab.
- **Copy the program:**
 - Use the arrow keys to find your program file.
 - Press the **F2** button to copy the file to the machine's internal memory.
- **Select the program:** In the main program list, use the arrows to highlight your newly copied program and press **SELECT PROG** to prepare it for running.

Step 3: Set the work offset (G54)

- **Switch to the work offset screen:** Press the **OFFSET** button and use the tab arrow keys to navigate to the **WORK** offsets screen.
- **Find the G54 Z-value:** Highlight the Z-axis value for G54. It should already be zeroed based on the tool touch-off you performed earlier.
- **Use Z Face Measure:** If needed, you can use a tool to touch the part face and press **Z FACE MEASURE** to reset the G54 Z-value to zero.

Step 4: Set tool offsets

- **Load the first tool:** Enter `T1 M6` in **MDI** mode and press **CYCLE START**.
- **Set the Z-offset:**
 - In **HANDLE JOG** mode, jog the tool tip to touch the face of the part.
 - Press the **OFFSET** button and navigate to the **TOOL GEOMETRY** page.
 - Highlight the **Z GEOMETRY** value for tool 1.
 - Press **Z FACE MEASURE**. The control will populate the Z geometry offset.

➤ **Set the X-offset:**

- Take a light cut on the diameter of your stock.
- Stop the spindle and carefully measure the diameter of the cut.
- In the **TOOL GEOMETRY** page, highlight the **X GEOMETRY** value for tool 1.
- Press **X DIAMETER MEASURE**.
- Enter the measured diameter value and press **ENTER**.

Step 5: Run the program safely

➤ **Change to MEM mode:** Press the **MEM** button on the control panel.

➤ **Enable single block:** Press the **SINGLE BLOCK** button to execute one line of code at a time. This is critical for proving out a new program.

➤ **Set the feed rate override:** Turn the **FEED RATE OVERRIDE** dial to a low percentage (e.g., 25%).

➤ **Start the program:**

- Ensure the doors are closed and latched.
- Press **CYCLE START**. The first block will run and the machine will stop.

➤ **Monitor the process:**

- Continue to press **CYCLE START** for each block.
- Be ready to press **FEED HOLD** or the **E-stop** if you notice any problems.

➤ **Increase speed:** After verifying the initial cuts are correct, you can turn off **SINGLE BLOCK** and increase the feed rate.

➤ **Complete the process:** Once the program is finished, the machine will stop. Perform a final inspection of your part.

Job Sheet: 4.5 (a)

Job	Load and run program for pocket milling operation in CNC milling machine	
Materials/Resources	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
	CAM Software (Mastercam)	1 Pc
	CNC Milling Machine	1 Nos.
	Mild steel	1 Pc (100x80x55mm)
Personal protective equipment (PPE)	Apron	1 Pc
	Hand gloves	1 Pc
Procedure	Trainee will collect required materials/resources.	
	Set Origin of the job.	
	Perform Stock-setup.	
	Load program	
	Dry Run program	
	Check all data according to program	
	If error take corrective measure	
	Run the Program	
	Follow trainers/instructors.	
	Stop program, clean the job, Clean the machine	
	Measure the job with drawing/sample.	

Job Sheet: 4.5 (b)

Job	Load and run program for facing & turning operations in CNC lathe machine	
Materials/Resources	Sample drawing	1 Pc
	Computer	1 Set
	CAD software (SolidWorks)	1 Pc
	CAM Software (Mastercam)	1 Pc
	CNC Milling Machine	1 Nos.
	Mild steel	1 Pc (Dia 40mm, Length: 100mm)
Personal protective equipment (PPE)	Apron	1 Pc
	Hand gloves	1 Pc
Procedure	Collect required materials/resources.	
	Set Origin of the job.	
	Perform Stock-setup.	
	Load program	
	Dry Run program	
	Check all data according to program	
	If error take corrective measure	
	Run the Program	
	Follow trainers/instructors.	
	Stop program, clean the job, Clean the machine	
	Measure the job with drawing/sample.	



Assessment Plan

Module -4	Written test	Performance test	Observation	Oral test	Portfolio	Third party report	Discussion	Project	Weekly/Fin al report	Remarks
Learning Outcome 1	√	√	√	√				√		
Learning Outcome 2	√	√	√	√				√		
Learning Outcome 3	√	√	√	√				√		
Learning Outcome 4	√	√	√	√				√		
Learning Outcome 5	√	√		√				√		



Competency Review

Here is a competency review of perform CAM programming..

Performance Criteria	Yes	No
Able to select the appropriate workpiece material based on mechanical properties and intended application?		
Can identify and define the machinability of different materials?		
Can identify and recognize the Mastercam Interface?		
Can identify and define the different machining processes of a Part?		
Can determine the correct cutting tool material (e.g., HSS, carbide, ceramic) based on workpiece hardness and cutting speed?		
Correctly determines cutting parameters (speed, feed rate, depth of cut) for a given tool-workpiece combination?		
Can create appropriate Tool Path Group and generate G&M Codes?		
Can load program using appropriate device?		
Can perform simulation in machine as per standard operating procedure?		

Module 5: Carry Out CNC Milling Machine Operations



Module Content

Module Description: This module covers the skills, knowledge and attitudes required to carry out CNC milling machine operations. It specifically includes setting-up CNC milling machine, downloading and inputting program, performing CNC milling machine operations and, checking and measuring work piece.

Minimum Time: 20 Hours.



Learning Outcome:

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

- 5.1 Prepare CNC milling machine
- 5.2 Download and transfer program
- 5.3 Operate CNC milling machine
- 5.4 Check and measure work piece



Performance Criteria:

1. Main switch and emergency switch are turned on and voltage stabilizer are checked.
2. Lubricant, air & hydraulic pressure and coolant are checked as per manufacturer's specification.
3. Machine zero point is checked.
4. CNC machine controller interface is identified and described.
5. Cutting tools are set according to required sequence of operation.
6. Clamping devices are set and tightened according to standard operating procedures.
7. Work piece is mounted and centered on clamping device to required level of accuracy as per workplace procedures.
8. Part zero setting is performed.
9. Tool zero setting is performed for each tool.
10. Program is downloaded and transferred to the machine using appropriate device.
11. Program is simulated to determine the correctness of the toolpath and other work parameters.
12. CNC milling operations are performed to produce component as per program.
13. Corrective measures are performed, if necessary.
14. Work piece is checked and measured against specification using appropriate methods and measuring tools.
15. Defective work pieces are marked, recorded and reported for proper action.

Learning Outcome 5.1: Prepare CNC Milling Machine



Content:

- Coolant
- Machine position zero
- Tools set up
- CNC milling machine
- Cutting Tools
- Machining processes



Assessment Criteria

1. Be able to check oil and coolant as per manufacturer's specification.
2. Be able to check air and hydraulic pressure as per manufacturer's specification.
3. Be able to set machine zero.
4. Be able to set cutting tools.
5. Be able to hold work piece.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. Workplace
2. Personal protective equipment (PPE)
3. CNC milling machine
4. Cutting tools



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Prepare CNC Milling Machine	• Information sheet: 5.1-1 to 5.1-2 • Self-Check: 5.1-1 to 5.1-2 • Answer script: 5.1-1 to 5.1-2



Information Sheet: 5.1-1

Learning Objective: Introduction to CNC Milling Machine (VMC)

Meaning of CNC: CNC means Computer Numerical Control. CNC machine is controlled by a computer, instead of manually operating it. People work on conventional Lathe, Shaping, Drilling machines. etc. They manually move the workpiece, axes, cutting tools. In case of CNC, they write a program and feed it into the memory of machine (machine will have micro controller, memory just like personal computer). The programming language is termed as G coding. The table (on which workpiece is mounted) movement, cutting tool movement, Spindle speed, works of auxiliary (like tool change, coolant on/off, door opening/closing, clamping/unclamping workpiece etc.) will be coded in the program. So, this program gives commands to all motors, controller, accessories of machine to perform the necessary operations and does the machining work as designed by the programmer. Programing commands makes it very easy to perform complex operations.

Advantages of CNC systems:

1. Advantages of CNC systems.
2. High speed, accuracy, efficiency, and repeatability
3. Reduction in machining times and good tool life monitoring.
4. Easily produces complex shaped workpieces.

CNC Milling: CNC milling, the most common form of computer numerical control (CNC) machining, performs the functions of drilling, routing, surfacing, gear cutting, boring, slotting, reaming facing etc. CNC mills are categorized according to their number of axis and are traditionally programmed using a set of codes that represent specific functions. CNC milling machines can be updated by improving the software used to drive the machines. Axis is labelled as x and y for horizontal movement, and z for vertical movement. A standard manual light-duty mill is typically assumed to have four axes:

1. Table X
2. Table Y
3. Table Z
4. Milling Head Z

A. Hyundai Wia CNC Milling Machine (VMC):



Specifications

Units of Measure : ○ inch ● mm

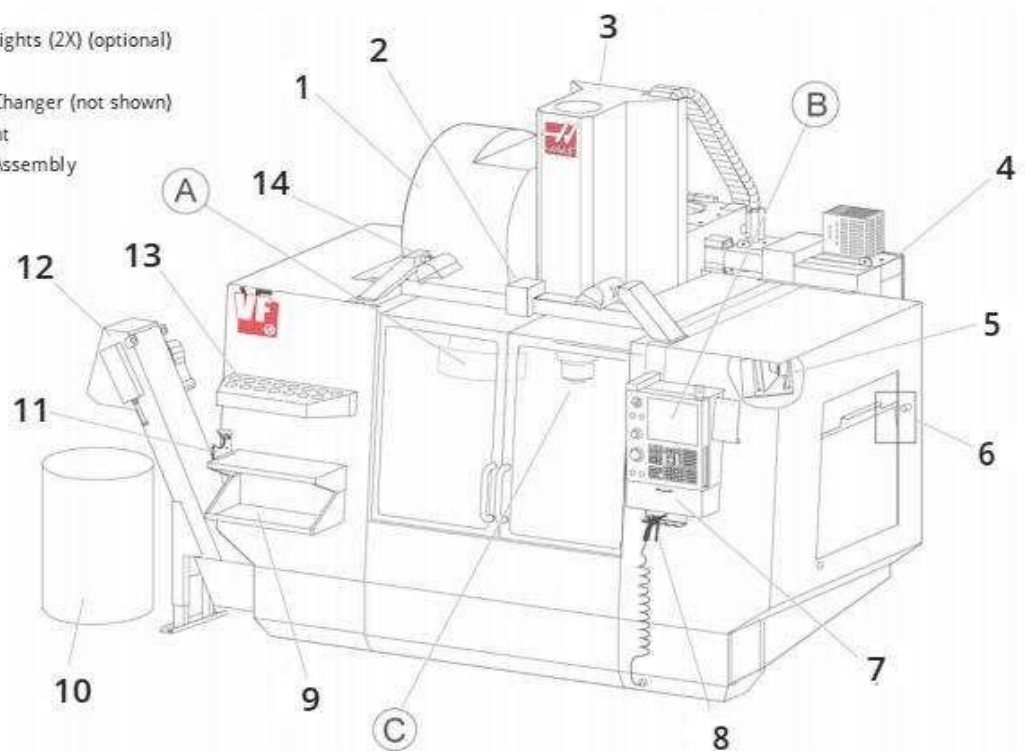
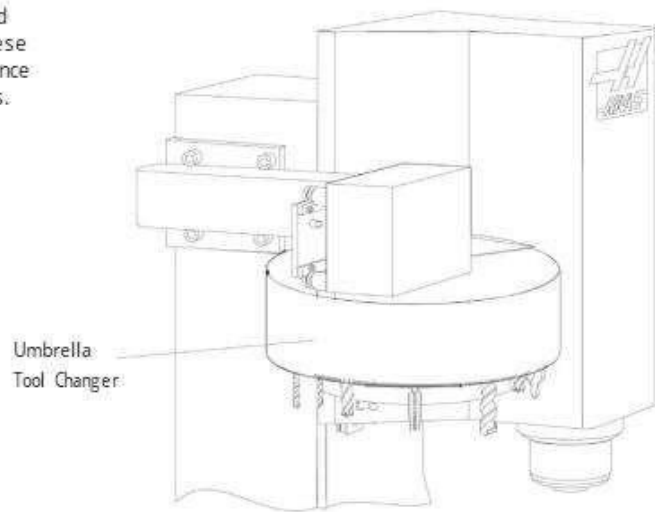
Description	Unit	i-CUT4000	i-CUT4500
Table Size	mm	650×400	800×450
Max. Load Capacity	kg	300	300
Spindle Driving Method	-	Direct	Direct
Spindle Taper	-	BBT30	BBT30
Spindle RPM	r/min	12,000	12,000
Spindle Power	kW	13/3.7	13/3.7
Spindle Torque	N.m	82.7/23.5	82.7/23.5
Travel(X/Y/Z)	mm	550/400/300	700/450/300
Rapid Feed Rate(X/Y/Z)	m/min	56/56/56	56/56/56
Slide Type	-	LM	LM
No. of Tools	EA	21	21
Tool Change Time (Chip To Chip)	sec	1.8	1.8
CNC	-	H/W FANUC i	H/W FANUC i

B. HAAS CNC Milling Machine (VMC):

The following figures show some of the standard and optional features of your Haas Vertical Mill. Note that these figures are representative only; your machine's appearance may vary depending on the model and installed options.

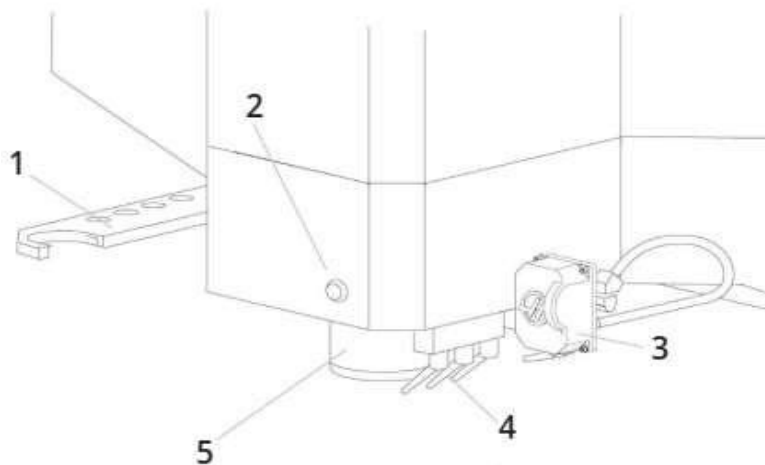
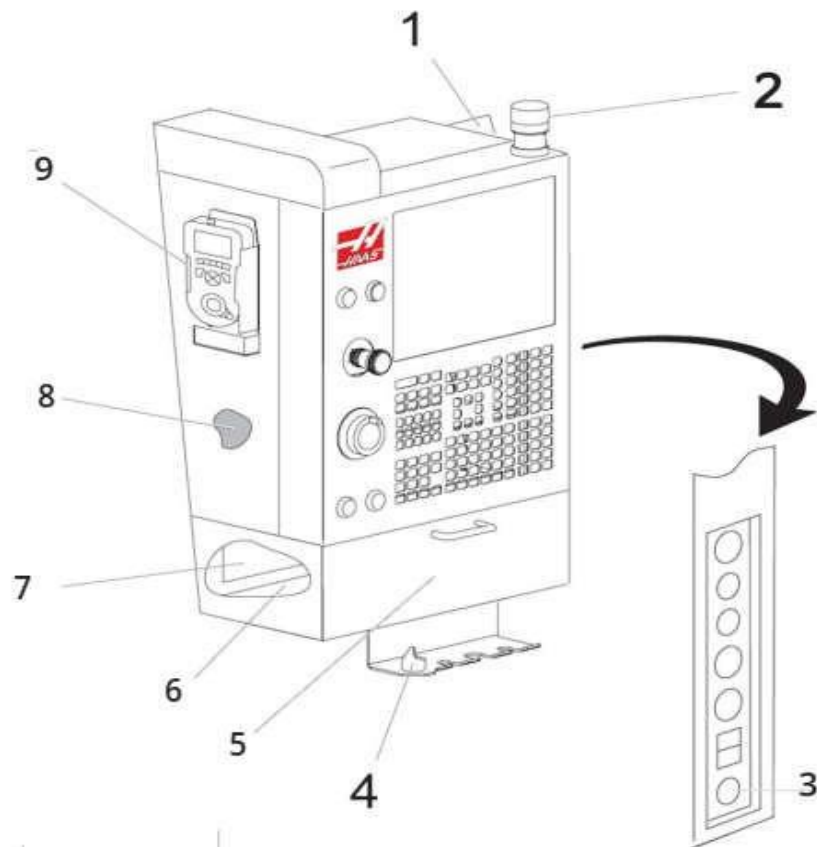
1. Side Mount Tool Changer (optional)
2. Auto Door (optional)
3. Spindle Assembly
4. Electrical Control Box
5. Work Light (2X)
6. Window Controls
7. Storage Tray
8. Air Gun
9. Front Work Table
10. Chip Container
11. Tool Holding Vise
12. Chip Conveyor (optional)
13. Tool Tray
14. High Intensity Lights (2X) (optional)

- A. Umbrella Tool Changer (not shown)
B. Control Pendant
C. Spindle Head Assembly



Control Pendant

1. Clipboard
2. Work Beacon
3. Hold To Run (where equipped)
4. Vise Handle Holder
5. Storage Pull Down Access Door
6. Tool Tray
7. G- and M-code Reference List
8. Operator's Manual and Assembly Data (stored inside)
9. Remote Jog Handle



Spindle Head Assembly

1. SMTC Double Arm (if equipped)
2. Tool Release Button
3. Programmable Coolant (optional)
4. Coolant Nozzles
5. Spindle

MILL - CONTROL PENDANT

Control Pendant Overview

The control pendant is the main interface to your Haas machine. This is where you program and run your CNC machining projects. This control pendant orientation section describes the different pendant sections:

- Pendant front panel
- Pendant right side, top, and bottom
- Keyboard
- Function/ Cursor Keys
- Display / Mode Keys
- Numeric / Alpha Keys
- Jog / Override Keys



PENDANT FRONT PANEL

Front Panel Controls

NAME	IMAGE	FUNCTION
POWER ON	I	Powers the machine on.
POWER OFF	O	Powers the machine off.
EMERGENCY STOP		Press to stop all axis motion, disable servos, stop the spindle and tool changer, and turn off the coolant pump.
HANDLE JOG		This is used to jog axes (select in HANDLE JOG Mode). Also used to scroll through program code or menu items while editing.
CYCLE START		Starts a program. This button is also used to start a program simulation in graphics mode.
FEED HOLD		Stops all axis motion during a program. The spindle continues to run. Press CYCLE START to cancel.



Information Sheet: 5.1-2

Learning Objective: Prepare CNC Milling Machine (VMC)

Tools and equipment required for CNC machine.

1. Micrometer
2. Dial Indicator
3. Adjustable bench
4. Vernier caliper
5. Collect Chuck
6. Part Catcher

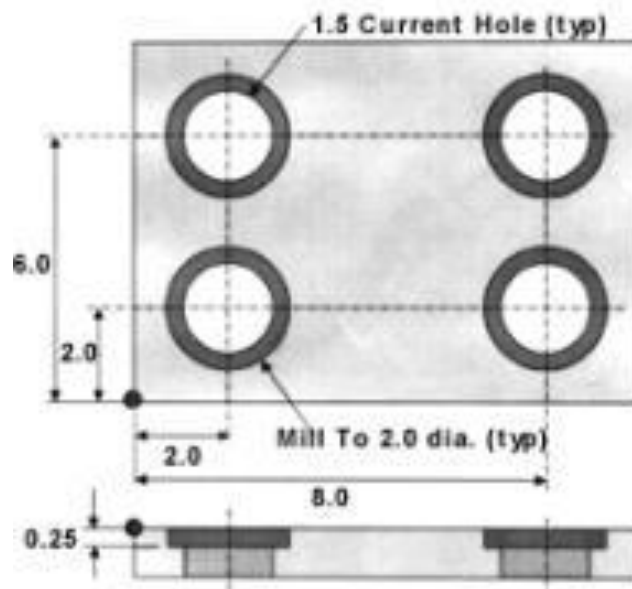
Step 1: Check oil, coolant, air and hydraulic in accordance with manufacturer's specification.

1. Check the hydraulic pressure.
2. Check the hydraulic fluids to make sure they're at the right operating level.
3. Check to make sure the chuck pressure is at the right operating pressure and grease chuck according to manufacturer's recommendation.
4. Make sure the way lube level is at the right operating level and replenish if needed.
5. If your CNC machine has a cooling system, make sure the cooling unit level is at the right operating level.
6. Clean the chips out of the chip pan.
7. Clean off the window of the door and the light so you can see inside your machine.
8. Wipe down any stainless-steel way covers and lubricate them with hydraulic oil, so they move smoothly.
9. Check and grease the chain on the chip conveyor.
10. Check and clean the filters on the coolant tank.

Step 2: Set machine zero point to the required position

Newer Fanuc and Fanuc-compatible controls have a command word to specify a temporary shift of the program zero point: the G52. Here's how it works:

For the workpiece shown in the Figure, the command G52 X2.0 Y2.0 will shift the program zero point to the center of the lower left hole (a great point of reference for programmed coordinates in the absolute mode). The command G52 X8.0 Y2.0 will shift the program zero point to the lower right hole. Though this application does not require it, you can also shift the Z axis program zero point. Note that the values specified in the G52 command are always specified from the original program zero point (not from one shifted position to another). At the end, remember to shift the program zero point back to its original position with the command G52 X0 Y0. Here's an example in Fanuc format. First, the subprogram is written using the center of a hole as the program zero point for all positioning movements. Note that for other kinds of workpiece attributes (rectangular pockets, slots, and so on), you can choose any point relative to the attribute as the temporary program zero point. This makes programming the machining of the attribute very simple.



Step 3: Set cutting tools according to required sequence of operations.

It is generally more efficient to use a combination of different toolpaths and tools to achieve a detailed model rather than assuming that a small tool with a smaller stepover is the only way. Often, a larger tool can achieve better finish results. In end milling, the cutter generally rotates on an axis vertical to the workpiece. Cutting teeth are located on both the end face of the cutter and the periphery of the cutter body.

1. A ball nose end mill, also known as a spherical end mill or ball end mill, has a semi sphere at the tool end. Ball nose end mills are used on workpieces with complex surfaces. Choosing flat end mill vs. a ball end mill will determine the characteristics of the tooling marks (or lack thereof) on your model. Most jobs will benefit from strategic use of multiple size and shape tools for milling different features.
2. End Mills are often used for roughing and 2D cutting and V-Bit and Ball Nose cutters are often used for finishing operations. There are up-cut, down-cut, compression cut end mills with varying numbers of flutes. End mills are intended to cut horizontally. Up-cut, down-cut and compression cut determine the way the chips (cut material) are ejected and the smoothness of the surface. With an up-cut end mill, the chips will be ejected upward, and the bottom of the material will be smooth. The down-cut end mill is the reverse by pushing the chips downward and the top of the material is smooth. The compression end mill creates a smooth surface on top and bottom, which is perfect for pre-laminated woods. End mills come in a variety of shapes. The most common are flat end mills and ball end mills. Flat end mills will cut flat areas with no scallops. However, they leave a terrace-like scallop on non-flat surfaces.
3. Ball end mills will leave smaller scallops for the same stepover value on sloped surfaces, but they will also leave scallops on flat areas. Models can be tooled with a combination of flat and ball end mills. If only one tool will be used for all surfaces a ball end geometry will give a more consistent overall feel and smooth result.
4. Flat end mills can be Centre Cutting and Non-center Cutting: Centre cutting square endmills are essential for plunge milling. Non-center cutting mills are used only for sidemilling.

Step 4: Set work holding and clamping devices and tightened according to standard operating procedures.

Work holding refers to any device that is used to secure a workpiece against the forces of machining. The most basic work holding device is a simple clamp, but work holding can also involve complex fixtures that are custom-built for parts. Other common work holding devices include vices and chucks, as well as indexers or rotary tables that can change the parts position while it is held, so the machine can reach various features. In most machining applications, work holding also locates the part. On a machining centre, for example, a vice or fixture may also provide the precise position and orientation where the machining program expects to find the workpiece.



Step 5: Workpiece zero point

This is the reference point for the specific workpiece being machined. It's usually established based on the design requirements of the part and the machining program. The workpiece zero point serves as the origin for all machining coordinates used in the CNC program. It's set relative to the machine zero point, often through the process of workpiece setup and alignment.

Step 6: Coordinate system

Once the workpiece zero point is established, a coordinate system is defined relative to it. Common coordinate systems include Cartesian coordinates (X, Y, Z) for three axis milling machines and additional axes for multi-axis machines. The CNC program specifies tool paths and machining operations using coordinates within this system.

Step 7: Requirement To Adjust Override Key

Here are some common requirements for adjusting the override key on a CNC machine:

1. Authorization and Training

- Operators typically need proper authorization and training to adjust override settings on CNC machines.
- Training ensures that operators understand the implications of changing override settings and how it can affect the machining process and part quality.

2. Safety considerations

- Operators should be aware of safety considerations when adjusting override settings.
- Changing parameters such as feed rate or spindle speed can impact the cutting forces and tool wear, potentially affecting operator safety and machine integrity.

3. Machine mode

- Depending on the CNC machine's configuration and control system, override adjustments may be accessible in different machine modes, such as manual mode, auto mode, or MDI (Manual Data Input) mode.
- Operators should ensure they are in the appropriate mode before attempting to adjust override settings.

4. Control interface

- CNC machines feature control interfaces where operators can access override functions.
- The override key may be labeled as "Feed Override," "Spindle Override," or something similar, depending on the machine's manufacturer and control system.

5. Limitations and guidelines

- CNC machines often have limitations on how much operators can adjust override settings.
- Manufacturers typically provide guidelines on the permissible range of override adjustments to prevent excessive strain on the machine and tooling.

6. Real-time monitoring

- Some CNC machines offer real-time monitoring of machining parameters, allowing operators to observe the effects of override adjustments on the machining process.
- Operators can use this feedback to make informed decisions and optimize machining operations.

7. Documentation and recordkeeping

- It's essential to maintain documentation of override adjustments and recordkeeping for quality control purposes.
- Keeping track of override settings helps identify trends, troubleshoot issues, and maintain consistency in machining processes.

Step 8: Hand Jog Rate

The hand jog rate on a CNC milling machine refers to the speed at which the operator can manually move the machine's axes (X, Y, and Z) using the jog controls. This feature allows for:

- Precise positioning of the cutting tool during setup and alignment.
- Inspection of the workpiece and tool during the machining process.
- Manual intervention in case of minor adjustments or troubleshooting.



- The hand jog rate is typically adjustable through dedicated controls on the machine's panel or jog handle. These controls can be:
 - Physical knobs or dials: These allow the operator to manually adjust the jog speed in a stepless manner.
 - Selection buttons: The machine might offer predefined jog speed options through buttons, allowing the operator to choose between different speeds.
 - Software interface: Some advanced CNC machines might allow adjusting the jog rate through the software interface.

Step 9: ATC Function

Here are some key aspects of the ATC function:

a) Tool storage: CNC machining centers equipped with an ATC typically have a tool magazine or carousel that stores multiple cutting tools. The tool storage capacity can vary depending on the machine model and configuration, ranging from a few tools to dozens or even hundreds of tools.

b) Tool selection: The CNC control system manages the selection of cutting tools from the tool magazine based on the programmed machining instructions. During the machining process, the control system determines which tool is required for the upcoming operation and commands the ATC to retrieve the appropriate tool from the tool magazine.

c) Tool change process: When it's time to change tools, the CNC machine pauses the machining operation and moves the spindle to the designated tool change position. The ATC then engages with the selected tool in the tool magazine and exchanges it with the currently installed tool in the spindle. After the tool change is complete, the CNC machine resumes the machining process with the newly installed tool.

d) Tool length measurement: Some CNC machines incorporate tool length measurement probes or sensors to ensure accurate tool positioning and tool length compensation. Before installing a new tool, the machine may automatically measure the length of the tool to compensate for any variations and maintain machining accuracy.

e) Tool change time: The efficiency of the ATC function is often measured by the tool change time, which refers to the time it takes for the machine to complete a tool change cycle. Faster tool change times contribute to higher productivity and throughput in CNC machining operations.

f) Tool management and monitoring: CNC machines with ATC capabilities often include features for tool management and monitoring. Operators can use the CNC control interface to view the status of tools in the magazine, track tool life, and receive alerts for tool wear or tool breakage. Overall, the Automatic Tool Changer function significantly enhances the capabilities and efficiency of CNC machining centers by automating the tool change process. It enables the machine to perform a wide range of machining operations with minimal downtime and operator intervention, making it an indispensable feature in modern manufacturing environments.



Self-Check: 5.1.-1 to 5.1-2

1. Write the name main components of CNC machining center.
2. How to control hand jog rate?
3. Write key aspects of the ATC function?



Answer Sheet: 5.1.-1 to 5.1-2

1.
 - a) Machine on-off switch
 - b) Emergency switch
 - c) Machine reference switch
 - d) Override controller
 - e) Hand jog rate
 - f) Work holding device
 - g) Tool holder
 - h) Clamping kit
 - i) Vice
 - j) Clamping set
 - k) Parallel bars
2. The hand jog rate is typically adjustable through dedicated controls on the machine's panel or jog handle. These controls can be:
 - Physical knobs or dials: These allow the operator to manually adjust the jog speed in a stepless manner.
 - Selection buttons: The machine might offer predefined jog speed options through buttons, allowing the operator to choose between different speeds.
 - Software interface: Some advanced CNC machines might allow adjusting the jog rate through the software interface.
3.
 - Tool Storage
 - Tool Selection
 - Tool Change Process
 - Tool Length Measurement

Learning Outcome 5.2: Download and Transfer Program



Content:

- Machine set-up (Milling)
- Parameter setting
- Software (SolidWorks, Mastercam)
- Tool and work offset
- Dry run (Simulation)



Assessment Criteria

1. Program is loaded using appropriate device.
2. Dry run/simulation is performed in machine as per standard operating procedure.
3. Program is executed to produce work piece.
4. Equipment is cleaned and maintained as per standard operating procedure.



Necessary Materials and Instruments

Trainees must provide the following equipment and supplies:

1. Workplace (simulated or actual)
2. Job specifications, drawings or work instructions
3. Learning manual/CBLM
4. USB device



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Load and Run Program	• Information Sheet: 5.2-1 to 5.2-2



Information Sheet: 5.2-1

Learning Objective: Download and Transfer Program in HAAS Controller CNC Milling Machine

Step 1: Transfer the G-code program

- Use a USB flash drive to copy your G-code file (`.nc` or `.txt`) to the machine's memory.
- Insert your USB drive into the control panel's USB port.
- Press the **LIST PROG** button.
- On the screen, press the left arrow key to switch to the device list and highlight "USB Device".
- Use the arrow keys to find and select your program file.
- Press **F2** to copy the program from the USB drive to the machine's memory.
- A dialog box will appear. Select "Memory" and press **ENTER** twice.
- The program is now loaded in the machine's memory. Press the **MEM** button to activate it.

Step 2: Copy the Program

- Press the **FILE INPUT** softkey, then the **SELECT PROGRAM** softkey, and choose your file.
- Press the **EXEC** softkey to copy the program from the USB to the machine's internal memory.

Step 3: Run the program safely

- Switch to **MEM** (Memory) mode.
- Turn on **SINGLE BLOCK** to run the program one line at a time for verification.
- Lower the **FEED RATE OVERRIDE** to a low percentage (e.g., 25%).
- Close the machine's doors, which must be latched for the program to run in Memory mode.
- Press **CYCLE START**. The machine will execute the first block and stop.
- Continue to press **CYCLE START** for each block. Use **FEED HOLD** to pause if you see any problems.
- Once you are confident in your setup, you can turn off **SINGLE BLOCK** and increase the feed rate override.
- **Monitor the machining process** closely for any issues and be ready to press the **E-stop** in an

emergency.

Step 4: Start the cycle:

1. Press the **CYCLE START** button. The machine will execute the first block of code.
 2. For each subsequent block, press **CYCLE START**.
 3. Keep your hand on the **FEED HOLD** button to pause the program immediately if anything goes wrong.
- **Remove safety features gradually:** Once you have verified the initial cuts, you can turn off **SINGLE BLOCK** and increase the feed rate override as confidence grows.
- **Complete the process:** After the program finishes, turn off the spindle and coolant, clean up, and measure your finished part.



Information Sheet: 5.2-2

Learning Objective: Download and Transfer Program in **Fanuc Controller** CNC Milling machine.

Step 1: Transfer the G-code program from a USB drive.

➤ **Insert the USB drive:** Plug your USB drive, containing the G-code file (ending in **.NC** or **.TXT**), into the port on the control panel.

➤ **Enter Edit mode:** Press the **EDIT** mode button on the control panel.

➤ **Navigate to the Program screen:** Press the **PROG** button.

➤ **Initiate file transfer:**

4. Press the **OPRT** softkey below the screen.

5. Use the arrow softkeys to find and press the **DEVICE CHANGE** softkey.

6. Select "USB Memory" from the list of devices.

Step 2: Copy the Program

➤ Press the **FILE INPUT** softkey, then the **SELECT PROGRAM** softkey, and choose your file.

➤ Press the **EXEC** softkey to copy the program from the USB to the machine's internal memory.

Step 3: Run the program safely

➤ Select the program:

- Press the **MEM** (Memory) mode button.
- Press the **PROG** button and use the arrow keys to select your program from the list.

➤ Enable safety features:

- Press the **SINGLE BLOCK** button. This will run the program one line at a time for safety.
- Set the **FEED RATE OVERRIDE** to a low percentage (around 25%) to start slowly.

➤ Start the cycle:

- Press the **CYCLE START** button. The machine will execute the first block of code.

➤ Monitor the cut:

- For each subsequent block, press **CYCLE START**.
- Keep your hand on the **FEED HOLD** button to pause the program immediately if anything goes wrong.

➤ **Remove safety features gradually:** Once you have verified the initial cuts, you can turn off **SINGLE BLOCK** and increase the feed rate override as confidence grows.

➤ **Complete the process:** After the program finishes, turn off the spindle and coolant, clean up, and measure your finished part.

Learning Outcome 5.3: Operate CNC Milling Machine



Content:

- Machine set-up (Milling) & parameter setting
- Tool and work offset
- Edit program and correct/adjust tool parameters as required
- Mount or set work piece in accordance with standard operating procedures
- Dry run (Simulation) & perform CNC milling operations to produce component as programmed



Assessment Criteria

1. Be able to mount work piece or set in accordance with standard operating procedures.
2. Be able to load program using appropriate device.
3. Be able to do dry run/simulation in machine as per standard operating procedure.
4. Be able to edit program and parameters as required.
5. Be able to perform CNC milling operations to produce component as programmed.
6. Equipment is cleaned and maintained as per standard operating procedure.



Necessary Materials and Instruments

Trainees must provide the following equipment and supplies:

1. Workplace (simulated or actual)
2. Job specifications, drawings or work instructions
3. Learning manual/CBLM
4. USB device



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Load and Run Program	• Information Sheet: 5.3-1 to 5.3-2



Information Sheet: 5.3-1

Learning Objective: Operate CNC Milling (Fanuc) Machine.

Step 1: Machine Preparation and Power-On

- **Check the Machine:** Ensure the machine is in good condition, checking for sufficient lubricating oil, adequate cutting fluid, and proper air compressor function.
- **Power On:**
 - Switch on the main power disconnect (usually on the side or rear cabinet).
 - Press the **ON** button on the control panel to start the CNC system.
 - Unlock the **Emergency Stop** button (pull it out) to enable the servo systems.
- **Perform Reference Return:** This establishes the machine's coordinate system.
 - Select the **ZERO RETURN** mode.
 - Press the desired axis buttons (e.g., +X, +Y, +Z) to move all axes to their home/reference points. A light will confirm each axis has reached its zero position.
- **Machine Warm-up (Optional but Recommended):** Run a warm-up program to bring the spindle and machine components to a stable operating temperature.

Step 2: Job Setup

- **Workpiece Clamping:**
 - Clean the workpiece to remove any burrs, oil, or chips.
 - Secure the workpiece onto the machine table using a vise, clamps, or a suitable work holding device, ensuring it is stable and securely clamped.
- **Tool Installation:** Install the required cutting tools into the spindle or tool changer magazine as specified by the program plan.
- **Program Input:**
 - Select the **EDIT** or **PROGRAM** mode.
 - Input the G-code program into the control system memory, either by typing it manually, via a USB drive, or network transfer.
 - Verify the program on the screen to prevent errors.
- **Set Tool and Work Offsets:** This step defines the relationship between the tool, the workpiece, and the machine's coordinate system.

- In **MDI** or **JOG** mode, manually move the tool tip to a known reference point on the workpiece surface (e.g., the corner or top face).
- Navigate to the **OFFSET** or **WORK** page on the control panel.
- Input the determined offset values for the workpiece origin (e.g., G54 X, Y, Z) and the length/diameter of each tool used in the program.

Step 3: Program Execution

- **Select Program:** Switch to the **AUTO** or **MEM** mode and select the correct program number for execution.
- **Safety Checks:**
 - Confirm the tool path and machining operations are correct.
 - Before running with a workpiece, perform a "dry run" with no material to ensure no collisions occur (air cutting).
 - Ensure all guards are in place and you are wearing appropriate personal protective equipment (PPE).
- **Start Machining:**
 - Ensure the feed rate and spindle speed override switches are set appropriately (often at a reduced rate for the first run).
 - Press the **CYCLE START** button to begin the program execution.
- **Monitor the Process:**
 - Continuously monitor the operation, including the worktable position, tool behavior, and machine status display.
 - Use the **FEED HOLD** button to pause the feed movement if any abnormality is observed or manual measurements are needed.
 - Adjust feed rate and spindle speed overrides as needed during the cut.

Step 4: Post-Processing and Shutdown

- **Completion:** The machine will stop automatically when the program finishes.
- **Cleanup:** Clean the machine and workpiece, removing all chips and cutting fluid.
- **Shutdown:** Follow the specified shutdown procedure for the machine model, which typically involves stopping the spindle, turning off the control power, and finally the main power disconnects.
- **Maintenance:** Perform any necessary post-operation maintenance as specified by the manufacturer's guidelines.



Information Sheet: 5.3-2

Learning Objective: Operate CNC Milling (HAAS) Machine.

Step 1: Machine Preparation and Power-On

- **Safety First:** Put on appropriate safety gear, especially safety glasses.
- **Check Machine Status:** Ensure the work area is clear, and coolant/oil levels are sufficient.
- **Power On Main Breaker:** Turn on the main electrical disconnect switch, usually at the back of the machine.
- **Power On Control:** Press the green **POWER ON** button on the control panel.
- **Reset E-Stop:** Rotate the red **EMERGENCY STOP** button clockwise to release it.
- **Clear Alarms:** Press the **RESET** button to clear any initial startup alarms.
- **Power Up/Restart:** Close the machine doors and press the **POWER UP/RESTART** button. The machine will automatically move all axes to their home (machine zero) positions and establish its coordinate system.
- **Warm-up Spindle (Recommended):** If the machine hasn't run that day, run a dedicated spindle warm-up program (typically found in **LIST PROG**) to bring the spindle to a stable operating temperature.

Step 2: Job Setup

- **Mount Workpiece:** Secure the material (workpiece) in the vise or fixture on the machine table.
- **Load Tools:** Install the required cutting tools into the tool holders, and then load the holders into the tool changer carousel in the correct sequence as defined in your program's tool list.
- **Set Tool Length Offsets (TLO):**
 - Jog the spindle (using the **HANDLE JOG** mode) to position the tip of the first tool close to the top of the workpiece.
 - Go to the **OFFSET** page and select the **TOOL** tab.
 - Touch off the tool to the workpiece surface (using a piece of paper, a gauge block, or an automatic tool setter).
 - Press the **TOOL LENGTH MEASURE** button to record the Z-axis length offset for that tool. Repeat for all subsequent tools.

➤ **Set Work Coordinate System (WCS):**

- Using a pointer or an edge finder (or a probe, if available), jog the spindle to locate the X and Y zero points on your workpiece.
- Go to the **OFFSET** page and select the **WORK** tab (e.g., G54).
- With the cursor on the desired X-axis offset field, press **PART ZERO SET**. Repeat for the Y-axis.

Step 3: Program Execution

➤ **Load the Program:**

- Press the **LIST PROG** button.
- Navigate to your program (either from memory or a USB drive) and select it.

➤ **Verify the Program:**

- Run the program in **GRAPHICS** mode first to check the tool path for potential errors without any machine movement.
- Perform a "dry run" with the tool positioned a couple of inches above the part, using **SINGLE BLOCK** and a reduced **RAPID** override (e.g., 25%) to safely verify the motion and collision avoidance.

➤ **Run the Job:**

- Switch to **MEM** (Memory) or **AUTO** mode.
 - Ensure all safety doors are closed.
 - Set the feed rate and spindle speed overrides to a low percentage initially.
 - Press the green **CYCLE START** button to begin the automated cutting process.
- **Monitor:** Keep a close eye on the process and listen for unusual sounds. Be ready to press **FEED HOLD** or **EMERGENCY STOP** if necessary.

Step 4: Cleanup and Shutdown

- **Finish Program:** The machine will stop when the program is complete.
- **Cleanup:** Turn on the chip auger and coolant to clean the interior of the machine. Remove the finished part and clean the vise and table.
- **Power Off**

Learning Outcome 5.4: Check and Measure Work Piece



Content:

- Visual check
- Dimensional verification
- Material properties



Assessment Criteria

1. Be able to Work piece is checked and measured against specification using appropriate methods and measuring tools.
2. Defective work pieces are marked, recorded and reported for proper action



Necessary Materials and Instruments

Trainees must provide the following equipment and supplies:

1. Workplace (simulated or actual)
2. Job specifications, drawings or work instructions
3. Tools and equipment



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Check and Measure Work Piece	• Information Sheet: 5.4-1 to 5.4-3 • Job Sheet: 5.4 (a) to 5.4 (b)



Information Sheet: 5.4-1

Learning Objective: Check and Measure Work piece.

A comprehensive quality control check after CNC milling involves inspecting the workpiece for dimensional accuracy, surface finish, and cosmetic defects. The required contents will vary based on the part's complexity, tolerances, and intended function.

1. Visual inspection and cleaning

Before measuring, the part should be cleaned and inspected for obvious defects.

- **Cleaning:** Remove all chips, coolant, and burrs. Residual debris can cause false readings during measurement.
- **Visual check:** Inspect the workpiece for any visible flaws, such as scratches, discoloration, dents, or burrs.

2. Dimensional and geometric inspection

This step ensures all features are within the specified tolerances by comparing the finished part to the engineering blueprint or Computer-Aided Design (CAD) file.

Common tools for dimensional inspection

- **Calipers (Digital and Vernier):** Versatile and easy to use for measuring lengths, outer and inner diameters, and depths. Digital calipers are especially useful for their high readability.
- **Micrometers:** More accurate than calipers for measuring precise external and internal dimensions, as well as depths.
- **Height gauges:** Used on a granite surface plate to accurately measure and scribe vertical dimensions.
- **Coordinate Measuring Machine (CMM):** A highly accurate tool for measuring complex geometries by using a probe to capture 3D data points. CMMs can be programmed to automate the inspection process for repeatability.
- **3D Scanners:** Non-contact devices that use lasers or structured light to create a comprehensive digital map of the part's geometry. This is ideal for complex, irregular, or delicate parts.
- **Dial Indicators:** Used to measure small linear distances and check for runout (wobble) on rotating features, or for squareness and flatness on a surface plate.
- **Pin and plug gauges:** These "go/no-go" gauges quickly check if a hole's diameter is within the specified tolerance.
- **Feeler gauges:** A series of thin blades used to measure the width of narrow gaps and clearances.

3. Features to measure

- **Holes and slots:** Check for correct diameter, depth, and placement.
- **Radii and angles:** Verify that all corners and angles conform to the design.
- **Flatness and parallelism:** Use a height gauge and surface plate to confirm surfaces are flat and parallel to each other.

2. Surface finish measurement

- For parts requiring a specific surface texture, a roughness test is performed to quantify the quality of the machined surface.
- **Surface Roughness Testers (Profilometers):** This tool traces a stylus across the surface to measure the micro-level variations in height and calculate a roughness value, such as Ra (arithmetic average roughness).

3. Documentation and data

Proper documentation is crucial for process control and future reference.

- **Inspection logs:** Record all measurement results, whether they are within tolerance, and any corrective actions taken.
- **Corrective actions:** Note any adjustments made to the CNC program or machine setup to address dimensional deviations.
- **Traceability:** Document all inspections to maintain a comprehensive quality record for each manufactured part or batch.

4. What to do with the findings

- **If the part is within spec:** Finalize inspection and proceed with the next step, such as post-processing or shipping.
- **If the part is out of spec:** Stop the process, analyze the problem, and perform a root cause analysis to prevent the issue from recurring. This may involve adjusting the machining parameters, recalibrating the machine, or replacing worn tooling.

Job Sheet: 5.4 (a)

Job	Set-up CNC Milling machine and cycle start	
Materials/Resources	Computer	1 Set
	CNC Milling Machine	1 Nos.
	Dial indicator	1 Nos.
	Adjustable wrench	1 Pc
Personal protective equipment (PPE)	Apron	1 Pc
	Hand gloves	1 Pc
Procedure	Collect required materials/resources.	
	Wear PPE.	
	Check the coolant and oil Set Origin of the job.	
	Perform Stock-setup.	
	Turn On main switch.	
	Release emergency switch.	
	Load program.	
	Dry Run program.	
	Check all data according to program.	
	If error take corrective measure.	
	Cycle start.	
	Follow trainers/instructors.	

Job Sheet: 5.4 (b)

Job	Check and measure work piece	
Materials/Resources	Computer	1 Set
	Micrometer	
	Protractor	1 Nos.
	Combination square set	1 Pc
Personal protective equipment (PPE)	Apron	1 Pc
	Hand gloves	1 Pc
Procedure	Trainee will collect required materials/resources.	
	Wear PPE.	
	Stop cycle start.	
	Open the door.	
	Remove the job.	
	Clean the job.	
	Check all data dimensions according to drawing/sample.	
	Follow trainers/instructors.	



Assessment Plan

Module -5	Written test	Performance test	Observation	Oral test	Portfolio	Third party report	Discussion	Project	Weekly/Financial report	Remarks
Learning Outcome 1	√	√	√	√				√		
Learning Outcome 2	√	√	√	√				√		
Learning Outcome 3	√	√	√	√				√		
Learning Outcome 4	√	√	√	√				√		



Competency Review

Here is a competency review for carry out CNC milling machine operations.

Performance Criteria	Yes	No
Able to select the appropriate workpiece material based on mechanical properties and intended application?		
Can identify and define the machinability of different materials?		
Can identify and recognize the Mastercam Interface?		
Can identify and define the different machining processes of a Part?		
Can determine the correct cutting tool material (e.g., HSS, carbide, ceramic) based on workpiece hardness and cutting speed?		
Correctly determines cutting parameters (speed, feed rate, depth of cut) for a given tool-workpiece combination?		
Can create appropriate Tool Path Group and generate G&M Codes?		
Can load program using appropriate device?		
Can perform simulation in machine as per standard operating procedure?		

Module 6: Carry Out CNC Lathe Machine Operations



Module Content

Module Description: This module covers the skills, knowledge and attitudes required to carry out CNC lathe machine operations. It specifically includes setting-up CNC milling machine, downloading and inputting program, performing CNC lathe machine operations and, checking and measuring work piece.

Minimum Time: 20 Hours.



Learning Outcome:

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

- 1.1 Prepare CNC lathe machine
- 1.2 Download and transfer program
- 1.3 Operate CNC lathe machine
- 1.4 Check and measure work piece



Performance Criteria:

1. Main switch and emergency switch are turned on and voltage stabilizer are checked.
2. Lubricant, air & hydraulic pressure and coolant are checked as per manufacturer's specification.
3. Machine zero point is checked.
4. **CNC machine controller** interface is identified and described.
5. **Cutting tools** are set according to required sequence of operation.
6. **Clamping devices** are set and tightened according to standard operating procedures.
7. Work piece is mounted and centered on clamping device to required level of accuracy as per workplace procedures.
8. Part zero setting is performed.
9. Tool zero setting is performed for each tool.
10. Program is downloaded and transferred to the machine using appropriate device.
11. Program is simulated to determine the correctness of the toolpath and other work parameters.
12. Work piece is checked and measured against specification using appropriate methods and measuring tools.

Learning Outcome 6.1: Prepare CNC Lathe Machine



Content:

- Coolant
- Machine position zero
- Tools set up
- CNC lathe machine
- Cutting tools
- Machining processes



Assessment Criteria

1. Be able to check oil and coolant as per manufacturer's specification.
2. Be able to check air and hydraulic pressure as per manufacturer's specification.
3. Be able to set machine zero.
4. Be able to set cutting tools.
5. Be able to hold workpiece.



Necessary Material and Instruments

Trainees must provide the following equipment and supplies:

1. Workplace
2. Personal protective equipment (PPE)
3. CNC lathe machine
4. Cutting tools



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Prepare CNC Lathe Machine	• Information sheet: 6.1-1 to 6.1-2 • Self-Check: 6.1-1 to 6.1-2 • Answer script: 6.1-1 to 6.1-2



Information Sheet: 6.1-1

Learning Objective: Introduction to CNC Lathe Machine.

CNC Lathe: A CNC lathe is a computer-controlled machine tool that rotates a workpiece while a cutting tool shapes it. The functions of a CNC lathe include turning, facing, drilling, boring, and threading to produce precise, symmetrical cylindrical parts. These machines are also known as CNC turning machines or turning centers.

Number of axes

The number of axes determines the machine's capability and complexity.

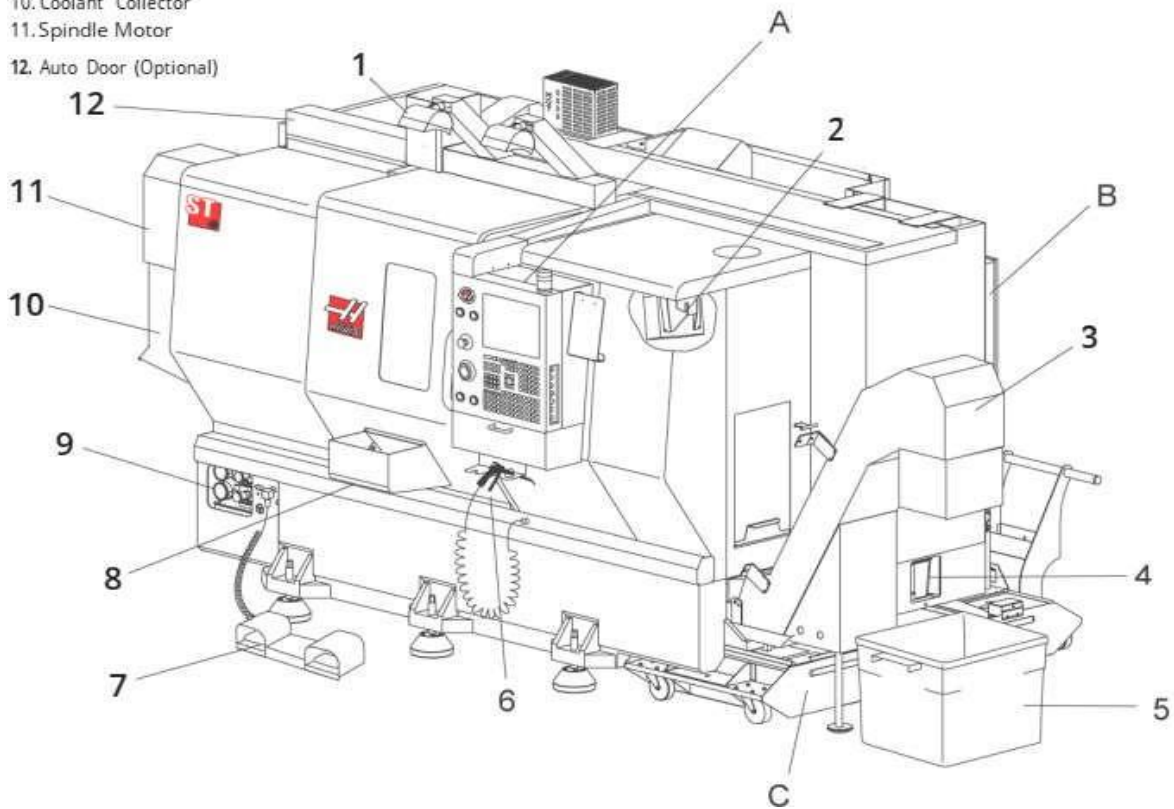
- **2-axis:** The most basic CNC lathe with **X** (radial) and **Z** (axial) movement. It can perform standard turning, facing, and drilling at the center of the part.
- **3-axis:** Adds a **C-axis** to the 2-axis machine, which enables the spindle to be rotated and positioned precisely. This allows for more complex milling and off-center drilling with live tooling.
- **4-axis:** Adds a **Y-axis** (perpendicular to X and Z) for off-center milling and drilling operations on a turned part. A 4-axis lathe may also add a second tool turret instead of a Y-axis.
- **5-axis or more:** Advanced lathes that add a second turret or a sub-spindle for machining both ends of the part in a single setup, drastically reducing cycle time.

HAAS CNC Lathe:

Lathe - Features (Front View)

These figures show some of the standard and optional features of your Haas lathe. Some of the features shown are highlighted in their appropriate sections. Note that these figures are representative only; your machine's appearance may vary depending on the model and installed options.

1. 2X High Intensity Lights (Optional)	A. Control Pendant
2. Work Light (2X)	B. Lubrication Panel Assembly
3. Chip Conveyor (Optional)	C. Coolant Tank
4. Oil Drain Container	
5. Chip Container	
6. Air Gun	
7. Foot Pedal	
8. Parts Catcher (Optional)	
9. Hydraulic Power Unit (HPU)	
10. Coolant Collector	
11. Spindle Motor	
12. Auto Door (Optional)	



Control Pendant Overview

The control pendant is the main interface to your Haas machine. This is where you program and run your CNC machining projects. This control pendant orientation section describes the different pendant sections:

- Pendant front panel
- Pendant right side, top, and bottom
- Keyboard
- Function/ Cursor Keys
- Display / Mode Keys
- Numeric / Alpha Keys
- Jog / Override Keys



Front Panel Controls

NAME	IMAGE	FUNCTION
POWER ON		Powers the machine on.
POWER OFF		Powers the machine off.
EMERGENCY STOP		Press to stop all axis motion, disable servos, stop the spindle and tool changer, and turn off the coolant pump.
HANDLE JOG		This is used to jog axes (select in HANDLE JOG Mode). Also used to scroll through program code or menu items while editing.
CYCLE START		Starts a program. This button is also used to start a program simulation in graphics mode.
FEED HOLD		Stops all axis motion during a program. The spindle continues to run. Press CYCLE START to cancel.

Keyboard

Keyboard keys are grouped into these functional areas:

1. Function
2. Cursor
3. Display
4. Mode
5. Numeric
6. Alpha
7. Jog
8. Overrides





Information Sheet: 6.1-2

Learning Objective: Prepare CNC Lathe Milling Machine (VMC)

Tools and equipment required for CNC machine.

1. Micrometer
2. Dial Indicator
3. Adjustable bench
4. Vernier caliper
5. Collect Chuck
6. Part Catcher

Step 1: Check oil, coolant, air and hydraulic in accordance with manufacturer's specification.

- Check the hydraulic pressure.
- Check the hydraulic fluids to make sure they're at the right operating level.
- Check to make sure the chuck pressure is at the right operating pressure and grease chuck according to manufacturer's recommendation.
- Make sure the way lube level is at the right operating level and replenish if needed.
- If your CNC machine has a cooling system, make sure the cooling unit level is at the right operating level.
- Clean the chips out of the chip pan.
- Clean off the window of the door and the light so you can see inside your machine.
- Wipe down any stainless-steel way covers and lubricate them with hydraulic oil, so they move smoothly.
- Check and grease the chain on the chip conveyor.
- Check and clean the filters on the coolant tank.

Step 2: Set machine zero point to the required position

The machine zero point can be set by three methods—by the operator, manually by a programmed absolute zero shift, or by work coordinates, to suit the holding fixture or the part to be machined. **MANUAL SETTING** - The operator can use the MCU controls to locate the spindle over the desired part zero and then set the X and Y coordinate registers on the console to zero.

Step 3: Set cutting tools according to required sequence of operations.

- **Determinetool order:** Arrange the operations in a logical sequence. A typical order is:
- **Install tools:** Securely fasten each tool holder and insert into the turret station according to your planned sequence. For example, if your program calls for a roughing tool (T1) first, followed by a finishing tool (T2), load them into turret stations #1 and #2, respectively.
- **Select a reference tool:** Choose a tool (often the facing or roughing tool) to serve as your reference for the Z-axis. Call up the tool in MDI mode (e.g., enter **T101** and press **[TURRET FWD]**).
- **Jog to the workpiece:** In **[HANDLE JOG]** mode, carefully move the reference tool toward the face of the workpiece until it lightly touches.
- **Measure the face:** With the tool touching the face, press the **[OFFSET]** key to navigate to the **Work Offset** page.
- **Set Z-zero:** Highlight the **Z** axis for the desired G-code offset (G54 is common) and press **[Z FACE MEASURE]**. The control will automatically record the machine's current Z-position in the offset table.

Step 4: Set tool offsets (X and Z)

Next, you will set the individual offsets for each cutting tool to account for variations in their length and tip position.

Method 1: Using the automatic tool presetter. This is the fastest and most accurate method if your Haas machine is equipped with one.

- **Call the tool:** In MDI mode, command a tool change (e.g., T101) to bring the first tool into position.
- **Activate the presetter cycle:** Follow the prompts on the Haas control for the Visual Quick Code (VQC) template for tool setting.
- **Start the cycle:** The tool will automatically be touched off on the presetter for both the X and Z axes, and the geometry offset values will be stored.
- **Repeat for each tool:** Repeat this process for every tool in the turret.

Method 2: Manual "touch-off" method. This method is used if your machine does not have an automatic presetter.

- **Make a test cut:** Use your first tool to take a small cut on the outside diameter (OD) and face of the workpiece.
- **Measure the diameter:** Move the tool away from the part in the X-axis and stop the spindle. Measure the diameter of the turned area using a micrometer or calipers.
- **Enter the X offset:** On the Haas control, navigate to the **Tool Offset** page.
 - Highlight the X offset for the current tool.
 - Press **[X DIAMETER MEASURE]**.
 - Type the diameter you just measured and press **[ENTER]**.
- **Enter the Z offset:** With the same tool, make a light skim cut on the face of the part.
- Highlight the Z offset for the current tool.
- Press **[Z FACE MEASURE]** to record the Z offset.
- **Repeat for all tools:** Repeat the touch-off process for each tool in the turret. The work offset (G54) has already been set, so each subsequent tool will be measured relative to the first. For a drill, you only need to set the Z offset, and then press **[X DIAMETER MEASURE] [F2]** to set the X value to zero.

Step 5: Coordinate system

Once the workpiece zero point is established, a coordinate system is defined relative to it. Common coordinate systems include Cartesian coordinates (X, Y, Z) for three axis milling machines and additional axes for multi-axis machines. The CNC program specifies tool paths and machining operations using coordinates within this system.

Step 6: Requirement To Adjust Override Key

Here are some common requirements for adjusting the override key on a CNC machine:

1. Safety considerations

- Operators should be aware of safety considerations when adjusting override settings.
- Changing parameters such as feed rate or spindle speed can impact the cutting forces and tool wear, potentially affecting operator safety and machine integrity.

2. Machine mode

- Depending on the CNC machine's configuration and control system, override adjustments may be accessible in different machine modes, such as manual mode, auto mode, or MDI (Manual Data Input) mode.
- Operators should ensure they are in the appropriate mode before attempting to adjust override settings.

3. Control interface

- CNC machines feature control interfaces where operators can access override functions.
- The override key may be labeled as "Feed Override," "Spindle Override," or something similar, depending on the machine's manufacturer and control system.

4. Limitations and guidelines

- CNC machines often have limitations on how much operators can adjust override settings.
- Manufacturers typically provide guidelines on the permissible range of override adjustments to prevent excessive strain on the machine and tooling.

5. Real-time monitoring

- Some CNC machines offer real-time monitoring of machining parameters, allowing operators to observe the effects of override adjustments on the machining process.
- Operators can use this feedback to make informed decisions and optimize machining operations.

Step 7: Hand Jog rate

The hand jog rate on a CNC milling machine refers to the speed at which the operator can manually move the machine's axes (X, and Z) using the jog controls. This feature allows for:

- Precise positioning of the cutting tool during setup and alignment.
- Inspection of the workpiece and tool during the machining process.
- Manual intervention in case of minor adjustments or troubleshooting.
- The hand jog rate is typically adjustable through dedicated controls on the machine's panel or jog handle. These controls can be:
 - Physical knobs or dials: These allow the operator to manually adjust the jog speed in a stepless manner.
 - Selection buttons: The machine might offer predefined jog speed options through buttons, allowing the operator to choose between different speeds.
 - Software interface: Some advanced CNC machines might allow adjusting the jog rate through the software interface.
- Here are some key points to remember about hand jog rate:
 - Safety: Always exercise caution when using the hand jog function. Ensure the work area is clear and never reach into the machine while it's in motion.
 - Start slow: It's recommended to begin with the lowest jog rate when initially positioning the tool or workpiece. Gradually increase the speed as needed while maintaining control and awareness of your surroundings.



Self-Check: 6.1-1 to 6.1-2

1. Write the name main components of CNC turning center.
2. Why CNC Lathe Machine is more appropriate than traditional Lathe machine?



Answer Sheet: 6.1-1 to 6.1-2

1.

- a) Machine Control Unit (MCU)
- b) Control Panel
- c) Drive System
- d) Feedback System

2. A CNC (Computer Numerical Control) lathe machine is more appropriate than a traditional manual lathe machine due to significant advancements in automation, precision, and efficiency. The primary advantages of a CNC lathe in modern manufacturing include

- a) High Tolerances
- b) Reduced Human Error
- c) Faster Production Speeds
- d) Multi-axis Machining

Learning Outcome 6.2: Download and Transfer Program



Content:

- Machine Set-up (Lathe)
- Parameter setting
- Software (SolidWorks, Mastercam)
- Tool and Work offset
- Dry Run (Simulation)



Assessment Criteria

1. Program is loaded using appropriate device.
2. Dry run/simulation is performed in machine as per standard operating procedure.
3. Program is executed to produce work piece.
4. Equipment is cleaned and maintained as per standard operating procedure.



Necessary Materials and Instruments

Trainees must provide the following equipment and supplies:

1. Workplace (simulated or actual)
2. Job specifications, drawings or work instructions
3. Learning manual/CBLM
4. USB device



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Load and Run Program	• Information Sheet: 6.2-1 • https://www.youtube.com/watch?v=t4NMdwQ5TSs



Information Sheet: 6.2-1

Learning Objective: Download and Transfer Program in HAAS Controller **CNC Lathe Machine**

Step 1: Transfer the G-code program

- Use a USB flash drive to copy your G-code file (`.nc` or `.txt`) to the machine's memory.
- Insert your USB drive into the control panel's USB port.
- Press the **LIST PROG** button.
- On the screen, press the left arrow key to switch to the device list and highlight "USB Device".
- Use the arrow keys to find and select your program file.
- Press **F2** to copy the program from the USB drive to the machine's memory.
- A dialog box will appear. Select "Memory" and press **ENTER** twice.
- The program is now loaded in the machine's memory. Press the **MEM** button to activate it.

Step 2: Load the G-code program

- Transfer your G-code file (typically a `.nc` or `.txt` file) to the lathe's internal memory via a USB drive.
- **Insert the USB drive:** Plug your USB flash drive into the control panel's USB port.
- **Open the device manager:** Press the **LIST PROG** button. The device manager screen will appear, showing available memory devices as tabs.
- **Navigate to the USB device:** Use the left or right arrow keys to select the "USB Device" tab.
- **Copy the program:**
 - Use the arrow keys to find your program file.
 - Press the **F2** button to copy the file to the machine's internal memory.
- **Select the program:** In the main program list, use the arrows to highlight your newly copied program and press **SELECT PROG** to prepare it for running.

Step 3: Run the program safely

- Switch to **MEM** (Memory) mode.
- Turn on **SINGLE BLOCK** to run the program one line at a time for verification.
- Lower the **FEED RATE OVERRIDE** to a low percentage (e.g., 25%).
- Close the machine's doors, which must be latched for the program to run in Memory mode.
- Press **CYCLE START**. The machine will execute the first block and stop.
- Continue to press **CYCLE START** for each block. Use **FEED HOLD** to pause if you see any problems.
- Once you are confident in your setup, you can turn off **SINGLE BLOCK** and increase the feed rate override.
- **Monitor the machining process** closely for any issues and be ready to press the **E-stop** in an emergency.

Learning Outcome 6.3: Operate CNC Lathe Machine



Content:

- Machine set-up (Lathe) & parameter setting
- Tool and work offset
- Edit program and correct/adjust tool parameters as required
- Mount or set work piece in accordance with standard operating procedures
- Dry run (Simulation) & perform CNC milling operations to produce component as programmed



Assessment Criteria

1. Be able to mount work piece or set in accordance with standard operating procedures.
2. Be able to load program using appropriate device.
3. Be able to do dry run/simulation in machine as per standard operating procedure.
4. Be able to edit program and parameters as required.
5. Be able to perform CNC milling operations to produce component as programmed.
6. Equipment is cleaned and maintained as per standard operating procedure.



Necessary Materials and Instruments

Trainees must provide the following equipment and supplies:

1. Workplace (simulated or actual)
2. Job specifications, drawings or work instructions
3. Learning manual/CBLM
4. USB device



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Operate CNC Lathe Machine	• Information Sheet: 6.3-1



Information Sheet: 6.3-1

Learning Objective: Operate CNC Lathe (HAAS) Machine.

Step 1: Machine Start-Up

- **Main Breaker On:** Locate the main power breaker on the back of the machine and switch it to the **ON** position.
- **Control Power On:** Press the green **POWER ON** button on the control panel.
- **Release E-Stop:** The screen will prompt you with basic instructions. Turn the red Emergency Stop button clockwise to release it.
- **Clear Alarms:** Press the **RESET** button to clear any startup alarms. You may need to press it twice.
- **Perform Power Up/Restart:** Close the machine door, ensure the work area is clear, and press the **POWER UP/RESTART** button. The machine will automatically move its axes to the home (zero return) position to establish its coordinate system reference points.

Step 2: Machine Setup

- Press the **MDI/DNC** button to enter Manual Data Input mode.
- Type "T" followed by the tool number (e.g., T1) and press **ATC FWD** or **ATC REV** to index the turret to the required tool position.
- Load the physical tool into the turret, ensuring it is securely fastened.
- **Mount Workpiece:**
 - Open the machine door.
 - Use the foot pedal (usually red) to open the hydraulic chuck jaws.
 - Insert the raw material and release the foot pedal to clamp it securely.
- **Set Offsets (Tool and Work):**
 - Press the **HANDLE JOG** button and select the desired jog increment (e.g., 0.001", 0.0001").
 - Use the hand jog wheel and axis buttons (X, Z) to carefully bring the tip of the tool to the workpiece surface.
 - Press the **OFFSET** button to access the offset pages.

- For the Z-axis work offset, jog the tool to the face of the part and press **PART ZERO SET** to set the G54 Z-value.
- For tool length/diameter offsets, use a tool setter or jog to known positions and use the **TOOL OFFSET MEASURE** function to record the values in the geometry or wear pages.

Step 3: Program Input and Verification

- Press the **LIST PROGRAM** button.
- Select the source (e.g., USB, machine memory) and highlight the program you wish to run.
- Press **SELECT PROGRAM** to make it the active program.
- **Verify the Program:**
- Enable safety features like **SINGLE BLOCK** (pauses after each line of code) and reduce the **RAPID** override to a low percentage (e.g., 5% or 25%).
- Run a "dry run" by pressing **CYCLE START** with the tool slightly away from the part to confirm the tool path in the graphics display and check for potential collisions.

Step 4: Machining and Shutdown

- Ensure the safety door is closed.
- Switch to **MEMORY** (Auto) mode.
- Deactivate Single Block (if desired for continuous running) and increase the rapid override to a suitable level.
- Press **CYCLE START** to begin the automated cutting process.
- Monitor the process closely and use **FEED HOLD** if needed.
- **Post-Operation and Shutdown:**
- After the program finishes (often marked by an M30 code), the machine will stop.
- Open the door, clean chips and coolant from the part and machine.
- Remove the finished part.
- To shut down, press **POWER OFF** on the control panel, and then turn off the main breaker at the back of the machine.

Learning Outcome 6.4: Check and Measure Work Piece



Content:

- Visual check
- Dimensional verification
- Material properties



Assessment Criteria

1. Be able to Work piece is checked and measured against specification using appropriate methods and measuring tools.
2. Defective work pieces are marked, recorded and reported for proper action



Necessary Materials and Instruments

Trainees must provide the following equipment and supplies:

1. Workplace (simulated or actual)
2. Job specifications, drawings or work instructions
3. Tools and equipment



Activity/Work

Activity/Work	Resource/Special Instruction/Reference
Check and Measure Work Piece	• Information Sheet: 6.4-1 • Job Sheet: 6.4 (a) to 6.4 (b)



Information Sheet: 6.4-1

Learning Objective: Check and Measure Work piece.

A comprehensive quality control check after CNC lathe involves inspecting the workpiece for dimensional accuracy, surface finish, and cosmetic defects. The required contents will vary based on the part's complexity, tolerances, and intended function.

1. Visual inspection and cleaning

Before measuring, the part should be cleaned and inspected for obvious defects.

- **Cleaning:** Remove all chips, coolant, and burrs. Residual debris can cause false readings during measurement.
- **Visual check:** Inspect the workpiece for any visible flaws, such as scratches, discoloration, dents, or burrs.

2. Dimensional and geometric inspection

This step ensures all features are within the specified tolerances by comparing the finished part to the engineering blueprint or Computer-Aided Design (CAD) file.

Common tools for dimensional inspection

- **Calipers (Digital and Vernier):** Versatile and easy to use for measuring lengths, outer and inner diameters, and depths. Digital calipers are especially useful for their high readability.
- **Micrometers:** More accurate than calipers for measuring precise external and internal dimensions, as well as depths.
- **Coordinate Measuring Machine (CMM):** A highly accurate tool for measuring complex geometries by using a probe to capture 3D data points. CMMs can be programmed to automate the inspection process for repeatability.
- **3D Scanners:** Non-contact devices that use lasers or structured light to create a comprehensive digital map of the part's geometry. This is ideal for complex, irregular, or delicate parts.
- **Dial Indicators:** Used to measure small linear distances and check for runout (wobble) on rotating features, or for squareness and flatness on a surface plate.
- **Pin and plug gauges:** These "go/no-go" gauges quickly check if a hole's diameter is within the specified tolerance.

3. Surface finish measurement

- For parts requiring a specific surface texture, a roughness test is performed to quantify the quality of the machined surface.
- **Surface Roughness Testers (Profilometers):** This tool traces a stylus across the surface to measure the micro-level variations in height and calculate a roughness value, such as Ra (arithmetic average roughness).

4. Documentation and data

Proper documentation is crucial for process control and future reference.

- **Inspection logs:** Record all measurement results, whether they are within tolerance, and any corrective actions taken.
- **Corrective actions:** Note any adjustments made to the CNC program or machine setup to address dimensional deviations.
- **Traceability:** Document all inspections to maintain a comprehensive quality record for each manufactured part or batch.

5. What to do with the findings

- **If the part is within spec:** Finalize inspection and proceed with the next step, such as post-processing or shipping.
- **If the part is out of spec:** Stop the process, analyze the problem, and perform a root cause analysis to prevent the issue from recurring. This may involve adjusting the machining parameters, recalibrating the machine, or replacing worn tooling.

Job Sheet: 6.4 (a)

Job	Set-up CNC Lathe machine and cycle start	
	Computer	1 Set
	CNC Lathe machine	1 Nos.
	Dial indicator	1 Nos.
	Adjustable wrench	1 Pc
Personal protective equipment (PPE)	Apron	1 Pc
	Hand gloves	1 Pc
Procedure	Collect required materials/resources.	
	Wear PPE.	
	Check the coolant and oil Set Origin of the job.	
	Perform Stock-setup.	
	Turn On main switch.	
	Release emergency switch.	
	Load program.	
	Dry Run program	
	Check all data according to program	
	If error take corrective measure	
	Cycle start.	
	Follow trainers/instructors.	

Job Sheet: 6.4 (b)

Job	Check and measure work piece	
	Computer	1 Set
	Micrometer	1 Nos.
	protractor	1 Nos.
	Combination square set	1 Pc
Personal protective equipment (PPE)	Apron	1 Pc
	Hand gloves	1 Pc
Procedure	Collect required materials/resources.	
	Wear PPE.	
	Stop cycle start.	
	Open the door.	
	Remove the job.	
	Clean the job.	
	Check all data dimensions according to drawing/sample.	
	Follow trainers/instructors.	



Assessment Plan

Module -6	Written test	Performance test	Observation	Oral test	Portfolio	Third party report	Discussion	Project	Weekly/Fin al report	Remarks
Learning Outcome 1	√	√	√	√				√		
Learning Outcome 2	√	√	√	√				√		
Learning Outcome 3	√	√	√	√				√		
Learning Outcome 4	√	√	√	√				√		



Competency Review

Here is a competency review for carry out CNC lathe machine operations.

Performance Criteria	Yes	No
Able to select the appropriate workpiece material based on mechanical properties and intended application?		
Can identify and define the machinability of different materials?		
Can identify and recognize the Mastercam Interface?		
Can identify and define the different machining processes of a Part?		
Can determine the correct cutting tool material (e.g., HSS, carbide, ceramic) based on workpiece hardness and cutting speed?		
Correctly determines cutting parameters (speed, feed rate, depth of cut) for a given tool-workpiece combination?		
Can create appropriate Tool Path Group and generate G&M Codes?		
Can load program using appropriate device?		
Can perform simulation in machine as per standard operating procedure?		