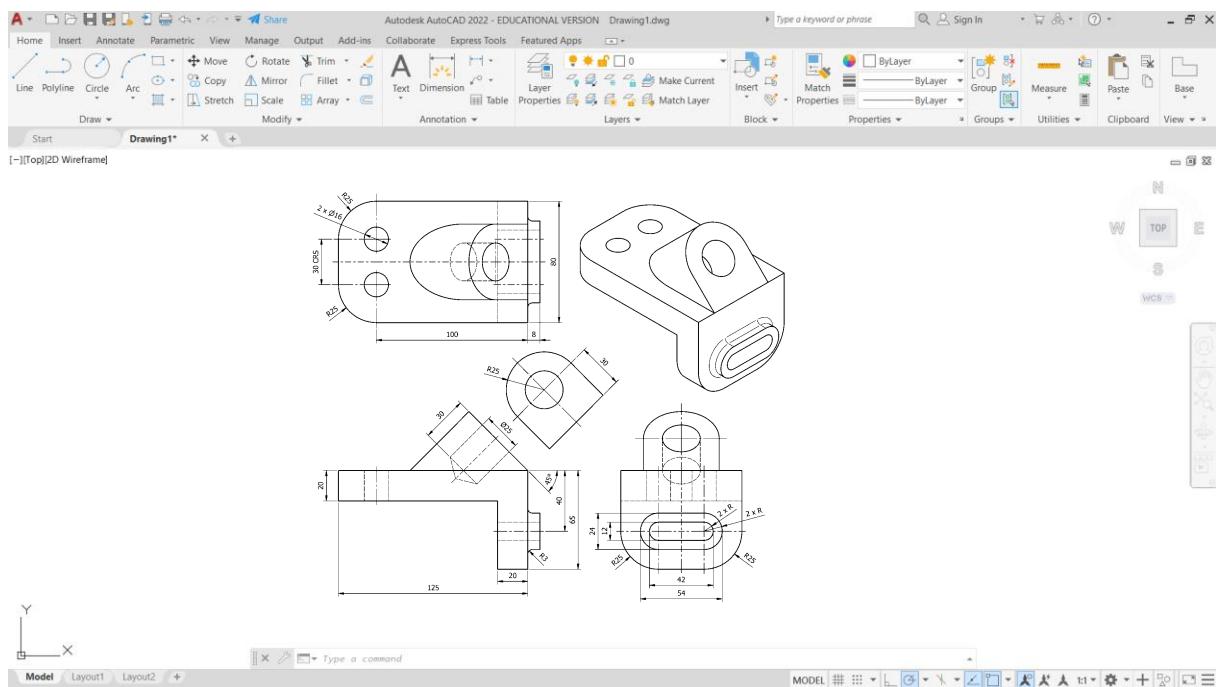




School of Mechanical & Aeronautical Engineering

2023/2024 Semester 1



Computer-Aided Drafting

ME1201

Singapore Polytechnic
School of Mechanical and Aeronautical Engineering

Memorandum

From : Module Co-ordinator of ME1201
To : All Students taking Computer-Aided Drafting, ME1201
Date : April 2023

1. The continuous assessments and weightages of this module are as follows:

Assessment	Weightage
CA1 (Test 1)	25%
CA2 (MCQ Quiz)	20%
CA3 (Assignment & Tutorial)	25%
CA4 (Test 2)	30%

2. The marks obtained from the above will be used to compute the final grade for this ME1201 module. There is no **Semestral Examination** for this module.
3. There will be no **CA1** and **CA4** make-up arranged for students who are absent with approved Leave of Absence (LOA). The following will apply:
 - a. **CA1:** Students absent with approved LOA will have **CA1**'s weightage redistributed to other components subjected to the 85% floor rule.
 - b. **CA4** is a non-redistributable component: This means that students who are absent will get **Zero** score even with approved LOA.
4. The approved LOA must be submitted to your lecturer concerned **immediately (within 24 hrs)** to ensure that the approved leave has been recorded. The co-ordinator will check on individual absentee. Failing to inform your lecturer on time will render yourself as absence without reason and the CAs mark will be recorded as **Zero**.
5. Students who are **absent without an approved LOA** from their CAs will get **Zero** score for the respective CAs.
6. Student will be given a **Pass (P)** or **Fail (F)** grade, i.e. no grade is awarded for this module when his attendance falls below 75%.
7. Students are expected to work diligently and complete the work during the allocated time. Besides academic performance, students are strongly urged to monitor the following:
 - a. Attendance
 - b. Punctuality in class.
 - c. Attire
 - d. Learning attitude and behaviour in class.
 - e. Originality of work, i.e. do not copy others' work.

- f. Discipline in class.
 - g. Good classroom management, eg. no littering and push back the chair at the end of the lesson.
 - h. Bring along the notes, tutorials and flash drive.
8. You are therefore urged to monitor your own performance closely and seek help from your lecturer early, should you face any problem.

MODULE OVERVIEW

1. Introduction

Computer-Aided Drafting (CAD) is a first year core module for the following courses:

Diploma in Aeronautical Engineering
Diploma in Mechanical Engineering
Diploma in Mechatronics and Robotics
Common Engineering Programme

2. Module Aims

The aim of this module is to train students with knowledge in the application of computer drafting software for the preparation of mechanical engineering drawings in accordance with ISO Standard recommendations. The module will provide students with knowledge in blue print reading, orthographic projection, sectioning and dimensioning of mechanical components which are key elements of engineering graphics communication. Students will also be developing their ability to use Computer-Aided Drafting and Design (CADD) software to create parametric solid models of mechanical components.

3. Module Contents

The topics within the module and their allocated lecture hours are listed below:

Topic Title	Hours
Engineering Drawing Practices	1
Computer-Aided Drafting using AutoCAD	4
Orthographic projection	10.5
Blue print reading	0.5
Introduction to Sectioning	6
Dimensioning	4
Parametric Solid modelling using Autodesk Inventor	13
Engineering Communication Assignment	5
Revision	8
Tests	8
Total	60

Note: The actual hours used for each topic may be adjusted to suit different teaching styles and student profiles.

4. Teaching Plan

The teaching plan of the module is outlined under “Contents & Lecture Schedule”.

5. Assessment

The assessment consists of coursework, **one** assignment and **two** tests.

CA1 (Test 1)	25%
CA2 (MCQ Quiz)	20%
CA3 (Assignment & Tutorial)	25%
CA4 (Test 2)	30%

6. Reference

Simmons, Colin H., et al., 2012. *Manual of Engineering Drawing*. 4th ed. Butterworth-Heinemann.

International Organization for Standardization, 2003. ISO 128. *Technical Drawings – General Principles of Presentation*. Switzerland: ISO.

International Organization for Standardization, 2004. ISO 129-1:2004. *Technical Drawings – Indication of Dimensions and Tolerances*. Switzerland: ISO.

Boundy, A.W., 2007. *Engineering Drawing*. 7th ed. McGraw-Hill Companies.

Autodesk Inc., 2019. *Autodesk Inventor Help 2020* [online]. Available from: <http://help.autodesk.com/view/INVNTOR/2020/ENU/>.

Autodesk Inc., 2019. *Autodesk Inventor Help 2020* [online]. Available from: <http://help.autodesk.com/view/INVNTOR/2020/ENU/>.

Shaun Bryant, 2019. *AutoCAD 2020 Essential Training*. LinkedIn Learning.

John Helfen, 2019. *Autodesk Inventor 2020 Essential Training*. LinkedIn Learning.

Autodesk Official Training Guide Essentials - Learning Autodesk Inventor 2010, Volume 1, 2009. Autodesk Inc.

Autodesk Official Training Guide Essentials - Learning Autodesk Inventor 2010, Volume 2, 2009. Autodesk Inc.

Shih, Randy H, 2019. *Learning Autodesk Inventor 2020: modeling, assembly and analysis*. SDC Publications.

7. Acknowledgement and Copyrights

For Unit 7, Parametric Solid Modelling using Autodesk Inventor, part of the materials and information presented are obtained from Autodesk Help – Inventor and Learning Autodesk Inventor 2010, Volume 1. Written permission had been obtained from Autodesk Inc for the use of their copyrighted materials in this unit.

The Autodesk copyrighted materials in Unit 7 of this module note shall only be used in the teaching and learning of Autodesk Inventor for the modules conducted in Singapore Polytechnic and thus shall not be reproduced in any form, by any method for other purposes without prior written permission.

Contents & Lecture Schedule for Term 1

Week	Topics	Tutorial
1	Unit 1: Engineering Drawing Practices Engineering drawing layout and practices. Principle of third-angle and first-angle orthographic projection. Presentation of holes. Specification ISO Metric screw threads.	Tutorial 1
2	Unit 2: Computer-Aided Drafting using AutoCAD Operation of AutoCAD. Create a general drawing template. Use appropriate AutoCAD commands to construct geometric figures of engineering component.	Tutorial 2
3	Unit 3: Orthographic Projection Prepare orthographic drawing of engineering component in third-angle projection.	Tutorial 3
4	Unit 3: Orthographic Projection Prepare orthographic drawing of engineering component in first-angle projection.	Tutorial 3
	Unit 4: Blue Print Reading (Self-Directed Learning) Recognise and apply drawing conventions, technical terms of engineering features.	Tutorial 4
5	Unit 5: Introduction to Sectioning Principles of sectioning. Conventional Practices of Sectioning. Types of sectional views. Use HATCH command to create sectional views.	Tutorial 5
6	Revision 1	Revision 1
7	CA1 (Test 1 conducted during MST week) Date and Venue to be provided later.	
8 - 9	Term Break	

Contents & Lecture Schedule for Term 2

Week	Topics	Tutorial
10	Unit 6: Dimensioning Dimensioning in accordance with ISO Standard recommendations. Setup Dimension style in AutoCAD. Place relevant ISO dimensions in engineering drawing.	Tutorial 6
11	Engineering Communication Assignment (Part of CA3) Assignment Brief. Measure Part to capture its features information. Make drawings views and dimensions of Part. Create parametric model of Part. Recap Unit 5 & Unit 6: Create sectional views. Place relevant ISO dimensions in engineering drawing.	E-Assignment
12	Unit 5: Introduction to Sectioning Create sectional views.	Tutorial 5
	Unit 6: Dimensioning Place relevant ISO dimensions in engineering drawing.	Tutorial 6
	Unit 7: Parametric Solid Modelling using Autodesk Inventor User Interface. File Types. Sketch tools for constructing features' profiles. Extrude, Revolve.	Tutorial 7
13	Unit 7: Parametric Solid Modelling using Autodesk Inventor Sketch tools for constructing features' profiles. Extrude, Revolve, Hole, Rib, Fillet, Chamfer, Pattern, Mirror, Work planes and Work axes for part design modelling.	Tutorial 7
14	Unit 7: Parametric Solid Modelling using Autodesk Inventor Sketch tools for constructing features' profiles. Extrude, Revolve, Hole, Rib, Fillet, Chamfer, Pattern, Mirror, Work planes and Work axes for part design modelling.	Tutorial 7
	CA2 (MCQ QUIZ)	
15	Break	
16	Unit 7: Parametric Solid Modelling using Autodesk Inventor Sketch tools for constructing features' profiles. Extrude, Revolve, Hole, Rib, Fillet, Chamfer, Pattern, Mirror, Work planes and Work axes for part design modelling.	Tutorial 7
17	Revision 2	Revision 2
18	Revision 2	
	CA4 (Test 2)	

UNIT 1 ENGINEERING DRAWING PRACTICES

Learning Objectives

By the end of this unit, students should be able to:

- Identify the typical layout of an engineering drawing
- Identify the various types of lines used for representing drawing information
- Identify the common abbreviations and symbols used in engineering drawing
- Describe the principle of third-angle orthographic projection.
- Describe the principle of first-angle orthographic projection.
- Distinguish between third-angle and first-angle orthographic projection.
- Identify various types of holes.

1.1 Introduction

Engineering drawing is the main method of communication between all persons concerned with design and manufacture of components, building and construction of works, and carrying out engineering projects required by management or professional engineering staff.

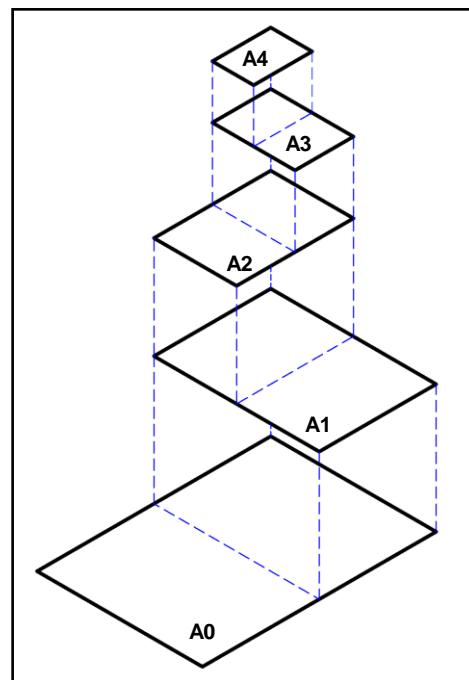
This unit presents the engineering drawing practices, which are relevant to mechanical drawing as well as providing other introductory information, which is often required for reference by students and engineering personnel. These are standardised practices to ensure the desired interpretation for efficient communication.

1.2 Drawing sheets

Engineering drawings are prepared on drawing sheets whose paper sizes are based on the ISO “A” series. The recommended sizes are as follows:

Sheet Size	Dimensions
A0	1189 X 841 mm ² (1m ²)
A1	841 X 594 mm ²
A2	594 X 420 mm ²
A3	420 X 297 mm ²
A4	297 X 210 mm ²

The sides of all sheets are in the ratio $1:\sqrt{2}$. All sizes in the range are exact halves of each preceding sheet.



1.3 Layout of drawing

Drawing sheets must include a border all round at a distance inward from the edge of the paper. A title block will be placed at the bottom right hand corner of the border that provides the following general information about a drawing:

- a) Title of the drawing
- b) Drawing number
- c) Scales of the drawing
- d) Date of the drawing
- e) Name of Draftsperson, checker.
- f) Projection symbol
- g) Sheet size

A sample of the drawing layout is shown in Fig 1.1.

1.4 Scales

It is often necessary to prepare an engineering drawing larger or smaller than the actual size of the component. In such cases, the drawing is prepared to a scale. The scale used should be indicated in or near the title block. The recommended scales for metric units are:

Type of Scales	Format	Interpretation
Full size	1 : 1	Same as the actual size of part
Enlargement	2 : 1	Twice full size
	5 : 1	Five times full size
	10 : 1	Ten times full size
Reduction	1 : 2	Half full size
	1 : 5	one-fifth full size
	1 : 10	one-tenth full size

1.5 Letters and numerals

When placing notes on a drawing, the following general rules should be observed:

- a) Upper case (capital) letters should be used.
- b) All characters in the same category (e.g. dimensions) should be of the same height.
- c) All notes should be placed in a horizontal position.

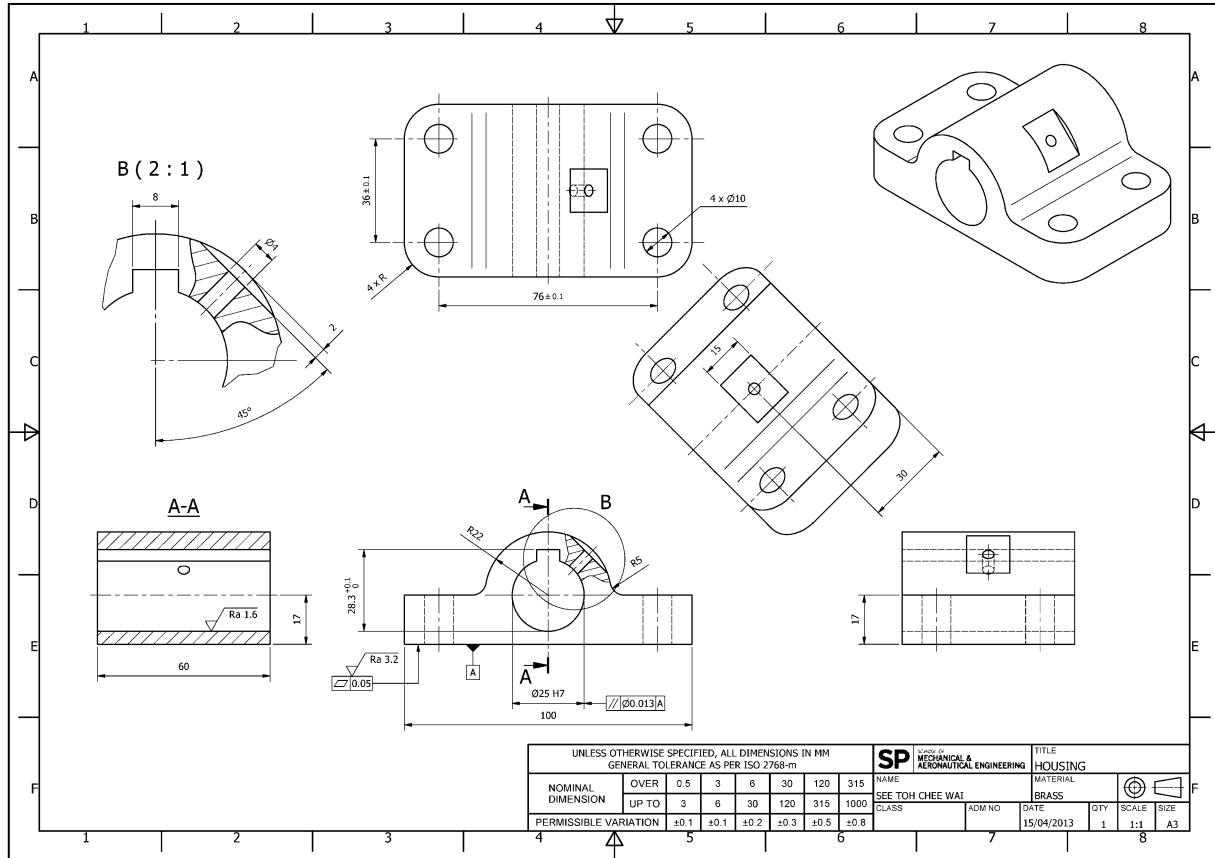


Figure 1.1 A sample of drawing layout

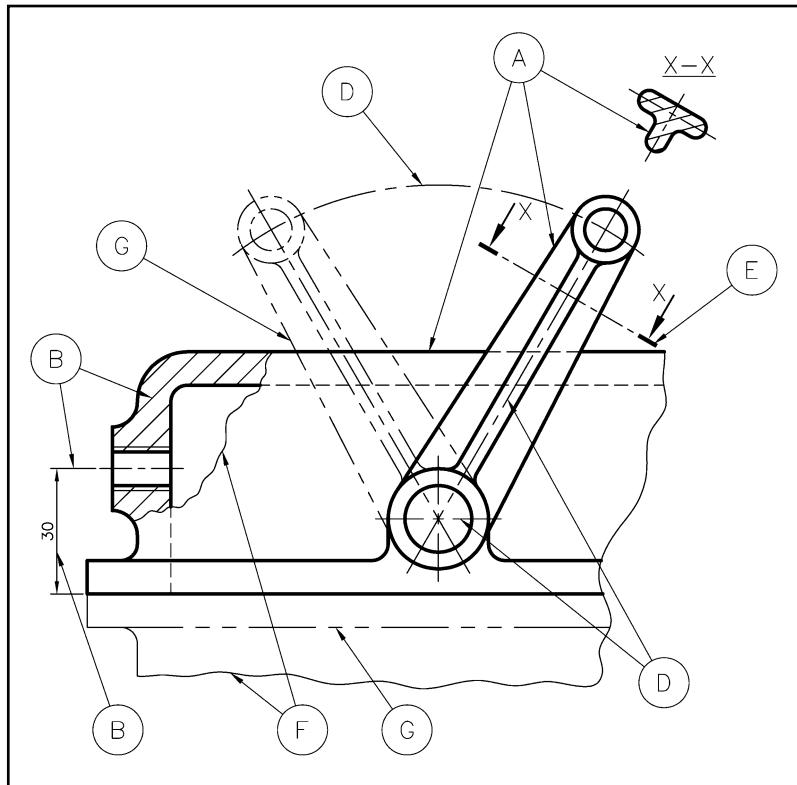


Figure 1.2 Types of lines used in engineering drawing.

1.6 Lines

Figure 1.2 presents a drawing that comprises of various types of lines. Each type of line is used to present specific information. Table 1.1 illustrates the various types of line that are used in engineering drawings. The line style and thickness are to be adhered when apply.

The line thickness should be uniform. Dashes should be of consistent length and spacing for dashed line and chain line.

Table 1.1 Applications of the various types of line.

	Types of Line	Application
A 	Thick continuous	Visible outlines and edges. (Thickness = 0.5mm)
B 	Thin Continuous	Dimensions and leader lines, projection lines, hatching, outlines of revolved sections. (Thickness = 0.3mm)
C 	Thin short dashes	Hidden outlines and edges. (Thickness = 0.3mm)
D 	Thin chain	Centre lines, lines of symmetry, pitch circles (Thickness = 0.3mm)
E 	Thin chain, thick at ends and at changes of direction	Cutting plane
F 	Thin continuous, irregular	Limits of partial view or sections. (Thickness = 0.3mm)
G 	Thin chain short double dashes	Outlines and edges of adjacent parts and extreme positions of movable parts. (Thickness = 0.3mm)

1.7 Abbreviations and symbols

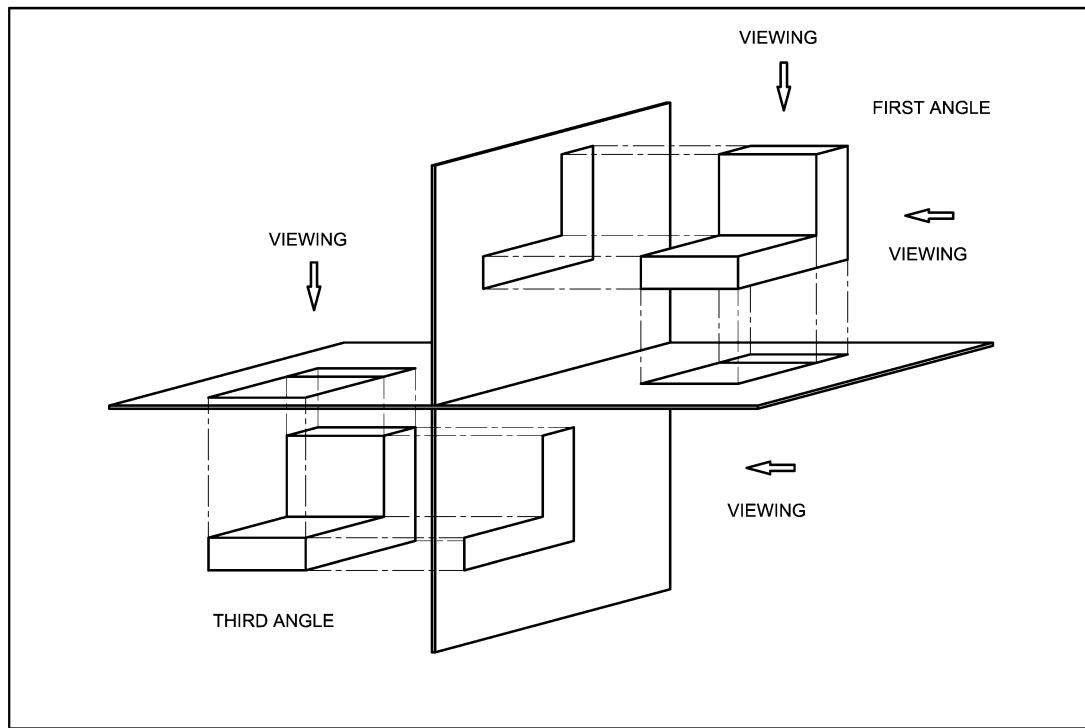
Common engineering terms and expressions are frequently replaced by abbreviations or symbols on drawings.

No.	Abbreviation or symbol	Term
1	AC	Across Corners
2	AF	Across Flats
3	ASSY	Assembly
4	CBORE	Counterbore
5	CH HD	Cheese head
6	CHAM	Chamfer
7	CL or 	Centrelines
8	CRS	Centres
9	CSK	Countersunk
10	CSK HD	Countersunk head
11	CYL	Cylinder
12	DIA (in a note)	Diameter
13	DRG	Drawing
14	EXT	External
15	FIG. (with full stop)	Figure
16	HEX	Hexagonal
17	HEX HD	Hexagonal head
18	INT	Internal
19	LG	Long
22	LH	Left hand
20	MATL	Material
21	MAX	Maximum
23	MIN	Minimum
24	NO. (with full stop)	Number
25	PCD	Pitch circle diameter
#26	R (preceding a dimension)	Radius
27	RD HD	Round head
28	RH	Right hand
29	SCR	Screw
30	SFACE	Spot face
31	SH	Sheet
#32	SR (preceding a dimension)	Spherical radius
#33	SØ (preceding a dimension)	Spherical diameter
34	SQ (in a note)	Square
35	STD	Standard
36	TOL	Tolerance
37	UCUT	Undercut
#38	Ø (preceding a dimension)	Diameter
#39	□ (preceding a dimension)	Square
40		Taper on diameter or width

1.8 Principle of Orthographic Projection

Orthographic projection is a method of projecting a number of separate two dimensional inter-related views which are mutually at right angles to each other.

Orthographic projection is based on two principal planes – one horizontal (HP) and one vertical (VP) – intersecting each other and forming right angles and quadrants as shown.



PRINCIPLES OF ORTHOGRAPHIC PROJECTION

There are two forms of orthographic projection:

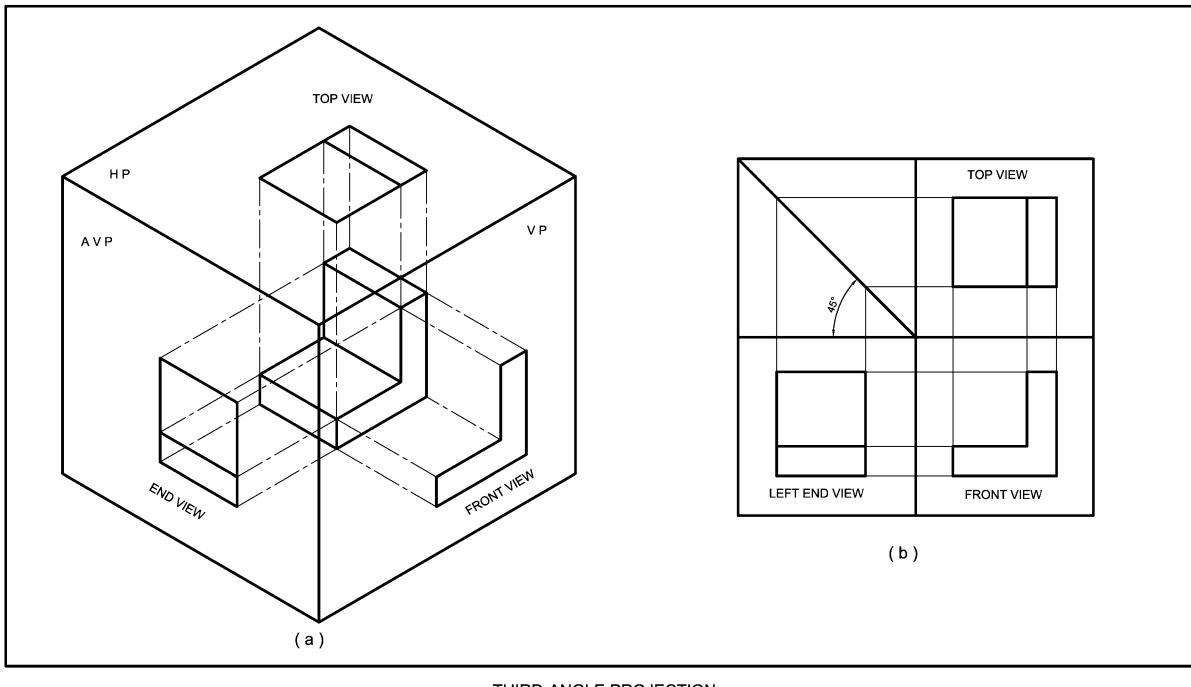
- Third-angle or “American”.
- First-angle or “European”.

1.9 Third-Angle Orthographic Projection

In third-angle projection, an object is positioned in the space of the third-angle quadrant between two principal planes. An additional plane, called the auxiliary vertical plane (AVP), is placed at 90° to the principal planes. The planes are imagined to be transparent and the projected views of the object are viewed through the plane as shown in figure (a).

- A view of the object projected by drawing parallel projecting lines from the object to the vertical principal plane (VP) is called a **front view** or **elevation**.
- A view similarly projected on to the horizontal principal plane (HP) is called a **top view** or **plan**.

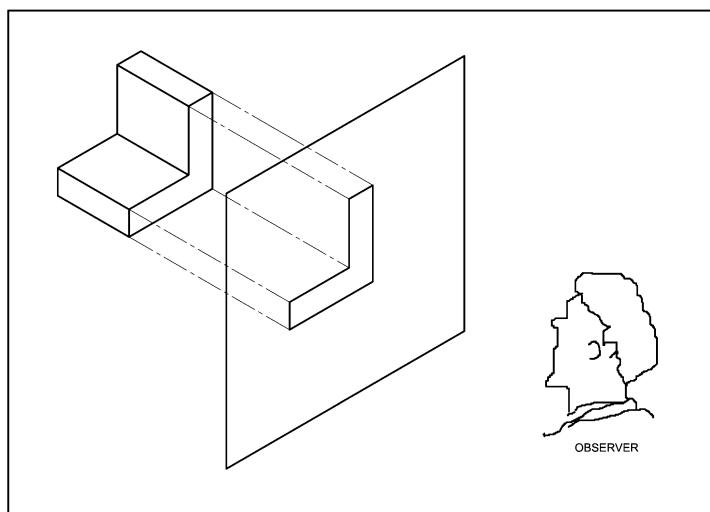
- A view projected on to the auxiliary vertical plane (AVP) is called an **end view** or **side view**.



THIRD-ANGLE PROJECTION

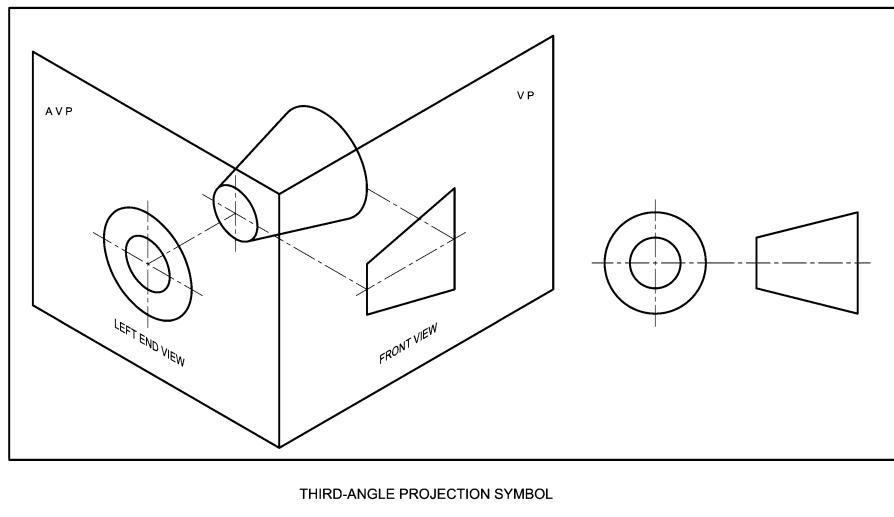
By means of projectors, all three planes of the ‘ glass box ‘ can be unfolded and the three views of the object can be shown simultaneously on drawing paper as shown in figure (b). The **end view** is projected horizontally and the **top view** is projected vertically from the **front view**.

In third-angle projection, the transparent projection plane always comes between the eyes of the observer and the object as shown.



PRINCIPLE OF THIRD-ANGLE PROJECTION

The symbol used to indicate third-angle projection is derived from views of a circular taper as shown. The symbol shows a front view and a left end view of the circular taper in third-angle projection.

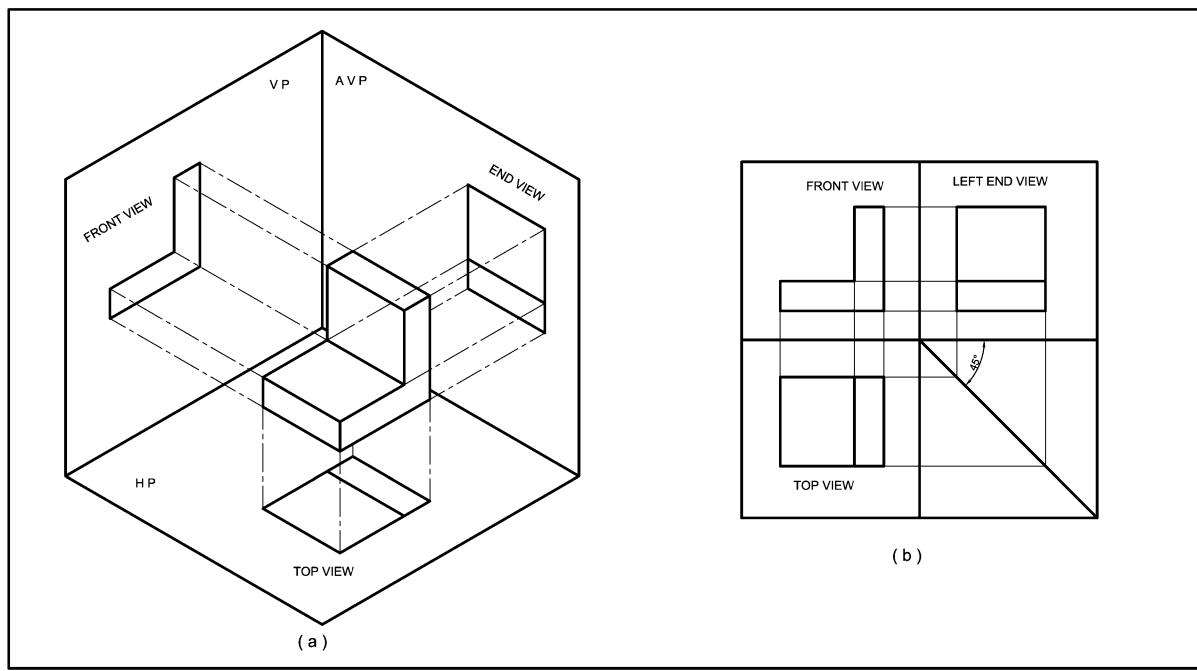


1.10 First-Angle Orthographic Projection

In first-angle projection, an object is positioned in the space of the first-angle quadrant between two principal planes. An additional plane, called the auxiliary vertical plane (AVP), is placed at 90° to the two principal planes.

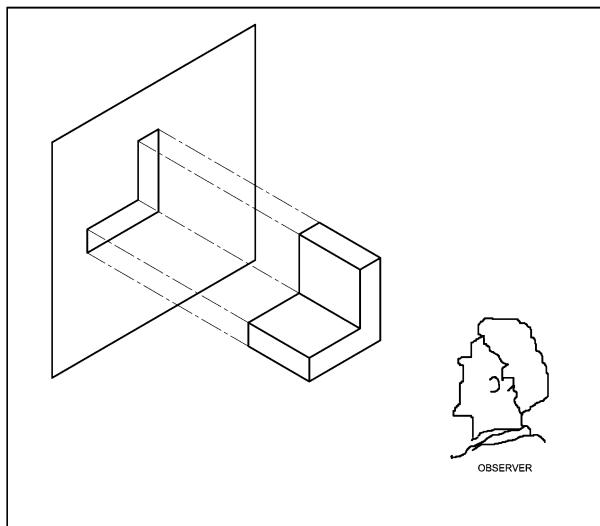
Similar to third-angle of projection, all the three planes can be unfolded and the three views of the object can be shown simultaneously on drawing paper as shown.

In both first-angle and third-angle projection, the three views are **identical** but the positioning of each is **different**.



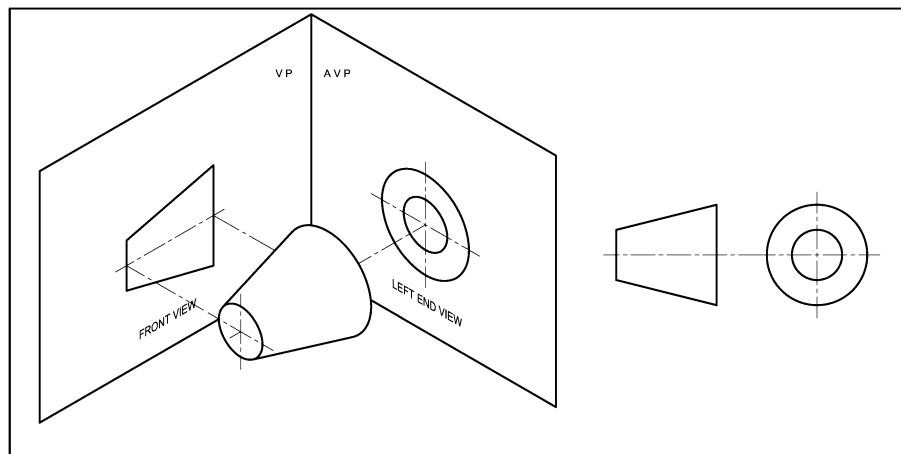
FIRST-ANGLE PROJECTION

In first-angle projection, the object always comes between the eyes of the observer and the projection plane as shown.



PRINCIPLE OF FIRST-ANGLE PROJECTION

The symbol used to indicate first-angle projection is derived as for third-angle projection but the views are positioned differently as shown.



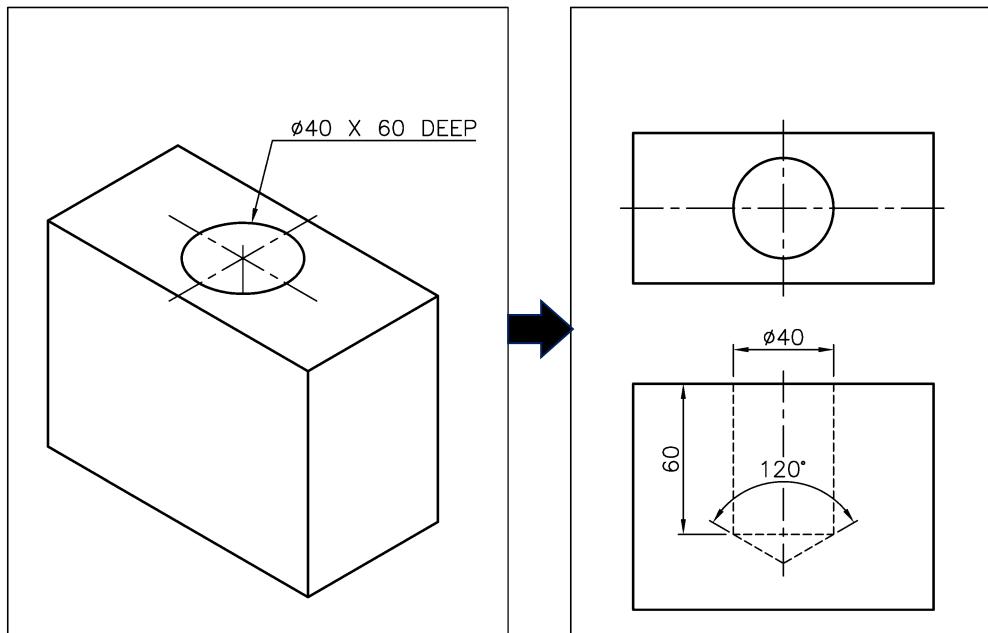
FIRST-ANGLE PROJECTION SYMBOL

1.11 Presentation of Holes in Orthographic Views

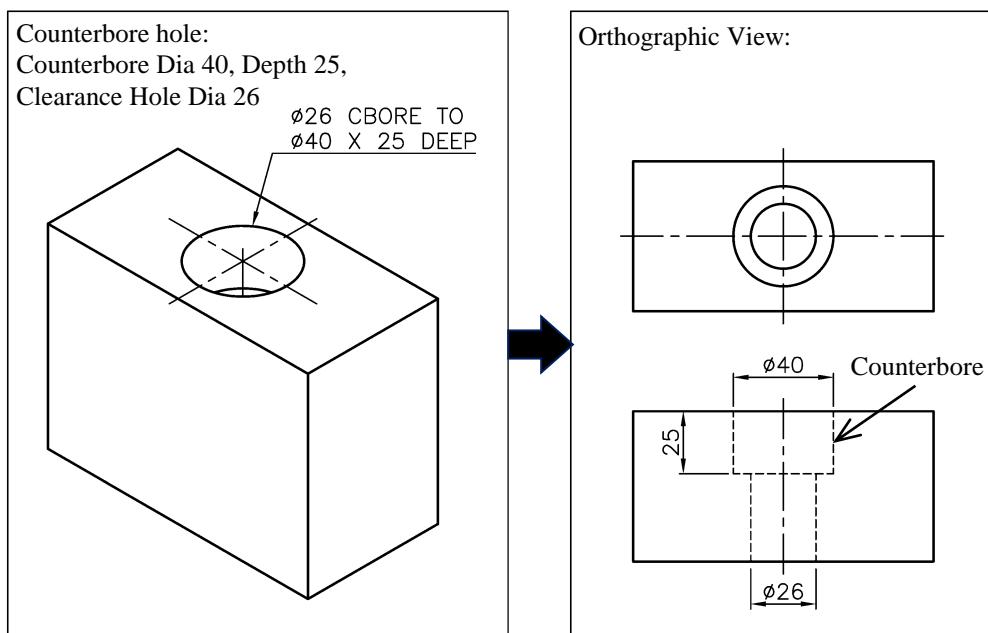
1.11.1 Blind Drilled-Hole

The blind end of the drilled-hole is represented by an angle of 120 degrees as shown.

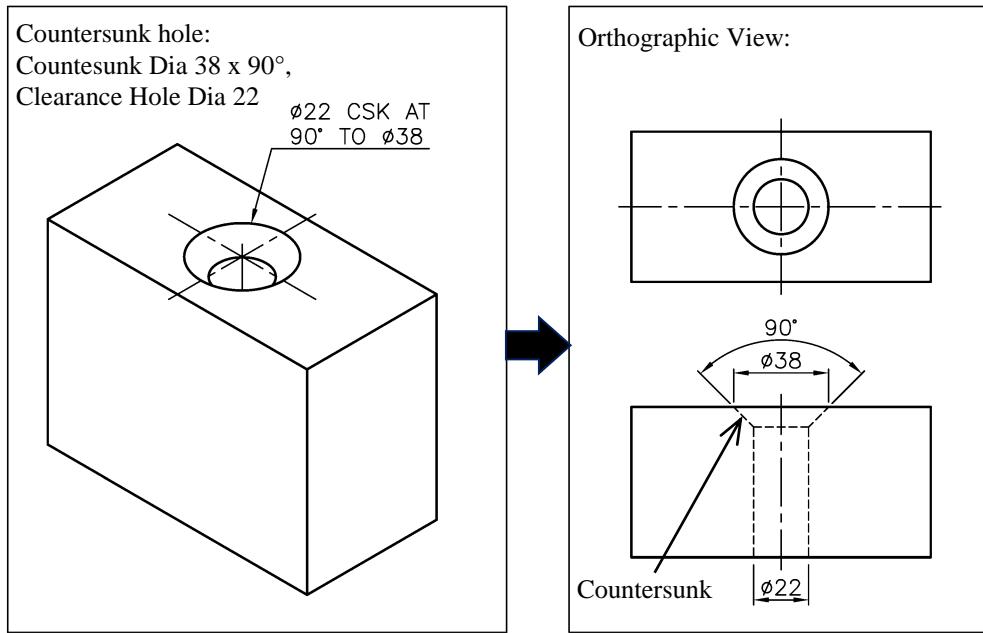
Note that the total depth of the hole does not include the tip end of the hole.



1.11.2 Counterbore Hole



1.11.3 Countersunk Hole

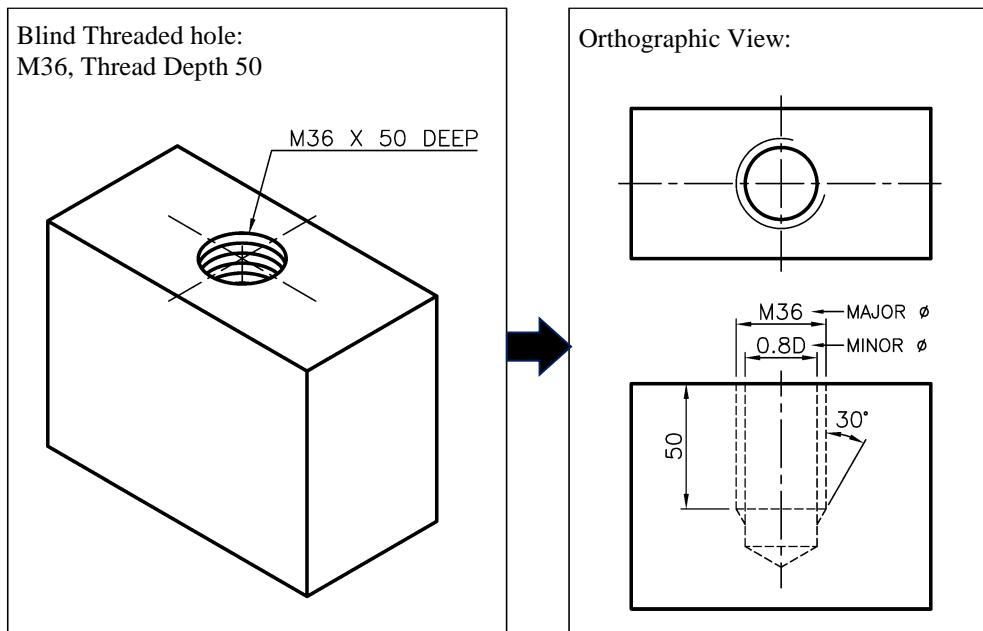


1.11.4 Blind Threaded Hole

The drawing of blind threaded hole is represented by the internal screw thread convention as shown in the figure.

The major diameter is drawn to the size of the thread, i.e. $D = M36 = \varnothing 36$. The minor diameter is drawn with a dimension of $0.8D$.

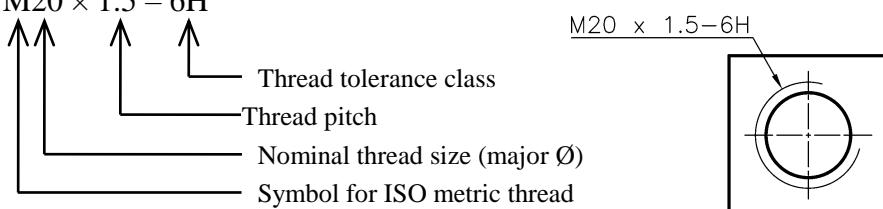
The blind hole will be deeper than the depth of the threaded hole for the necessary clearance during thread cutting.



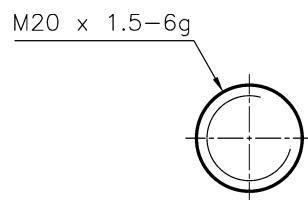
1.12 Specification of ISO Metric Screw Threads

On the engineering drawings, the nominal size (major diameter) of ISO metric screw threads is preceded by the alphabet **M**. The full specification of ISO metric screw threads is shown below:

- a. Internal thread: $M20 \times 1.5 - 6H$



- b. External thread: $M20 \times 1.5 - 6g$



Alternate forms of thread specifications can be:

1. **M20** : ISO metric thread, nominal size is 20 mm.
2. **M20 × 1.5** : ISO metric thread, nominal size is 20 mm, pitch of thread is 1.5 mm.

Tutorial 1

1. Write out in full the following abbreviations:

MAX :	INT :
RH :	ASSY :

2. Write out the abbreviation for the following engineering terms:

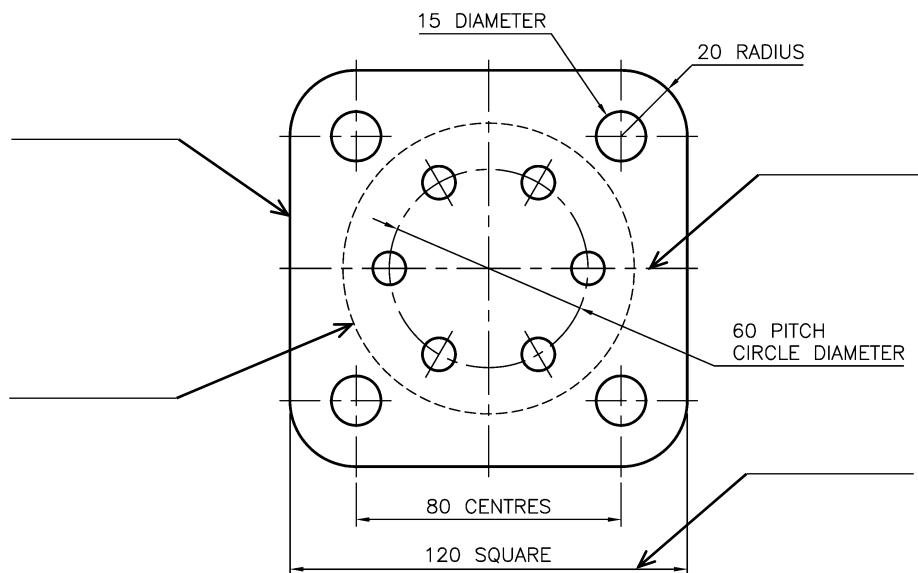
Figure :	Counterbore :
Centres :	Material :

3. Refer to the below figure,

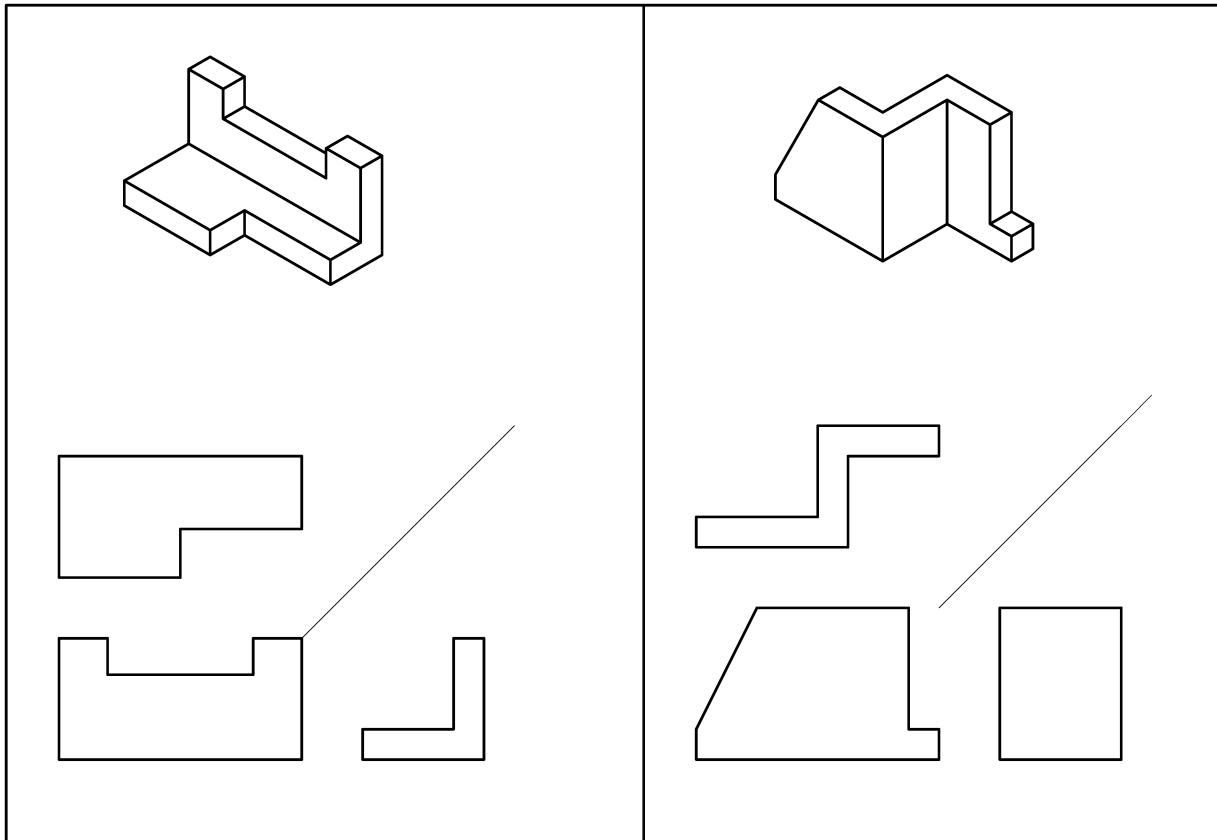
- a. Rewrite the dimensions using appropriate abbreviation and symbol:

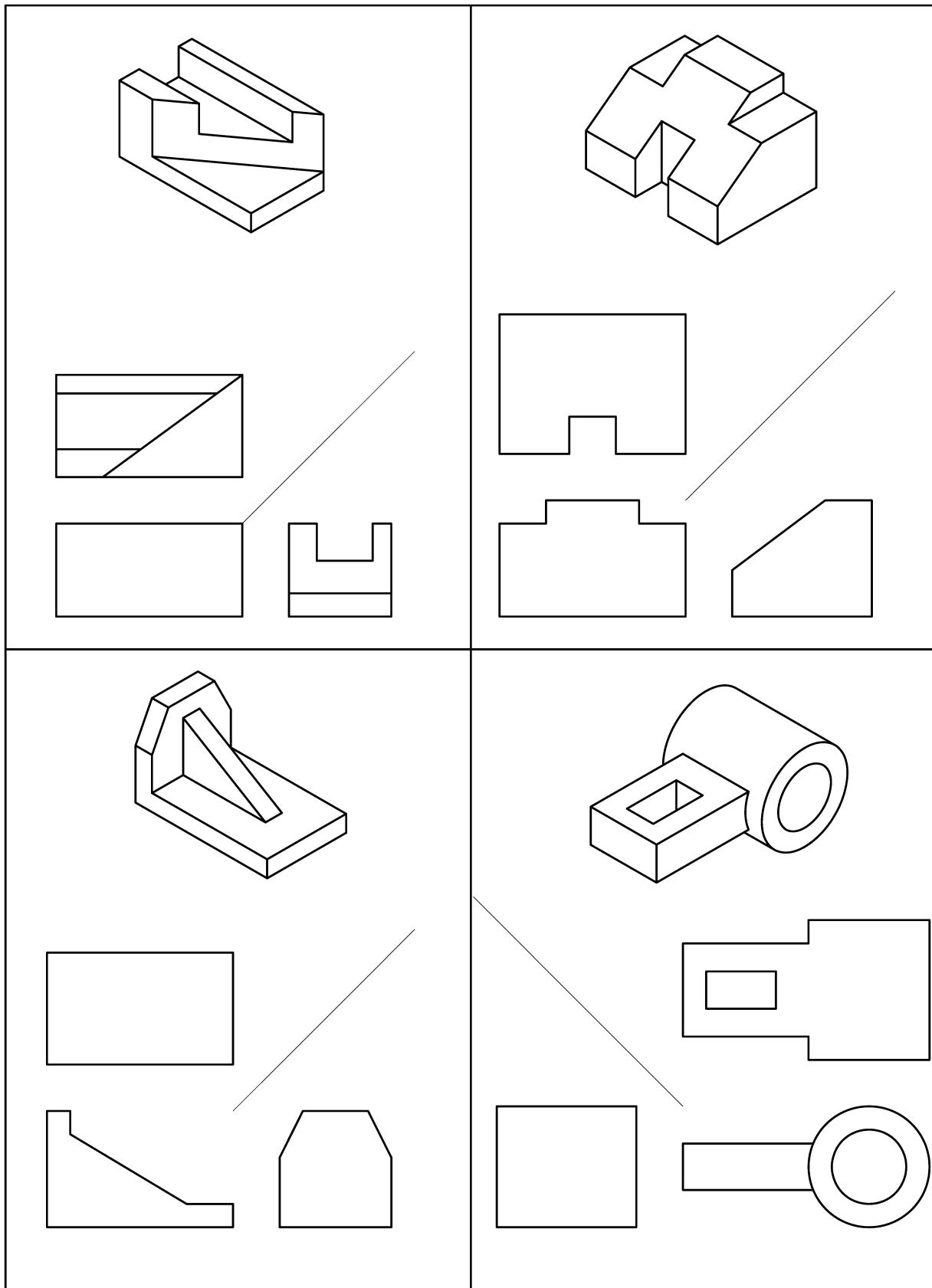
- i. 15 DIAMETER
- ii. 20 RADIUS
- iii. 80 CENTRES
- iv. 60 PITCH CIRCLE DIAMETER
- v. 120 SQUARE

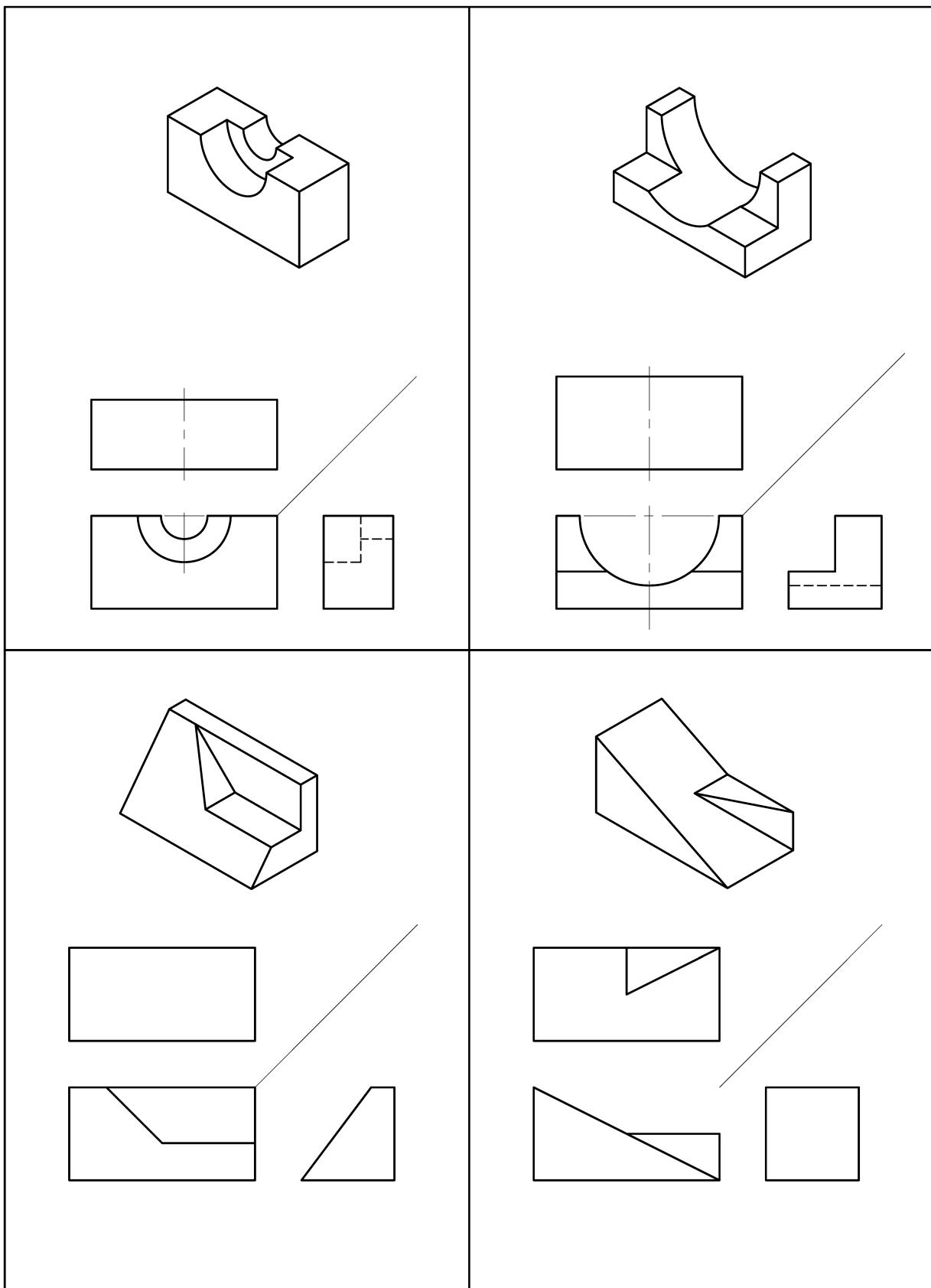
- b. Indicate the types of line used denoted by the leader.

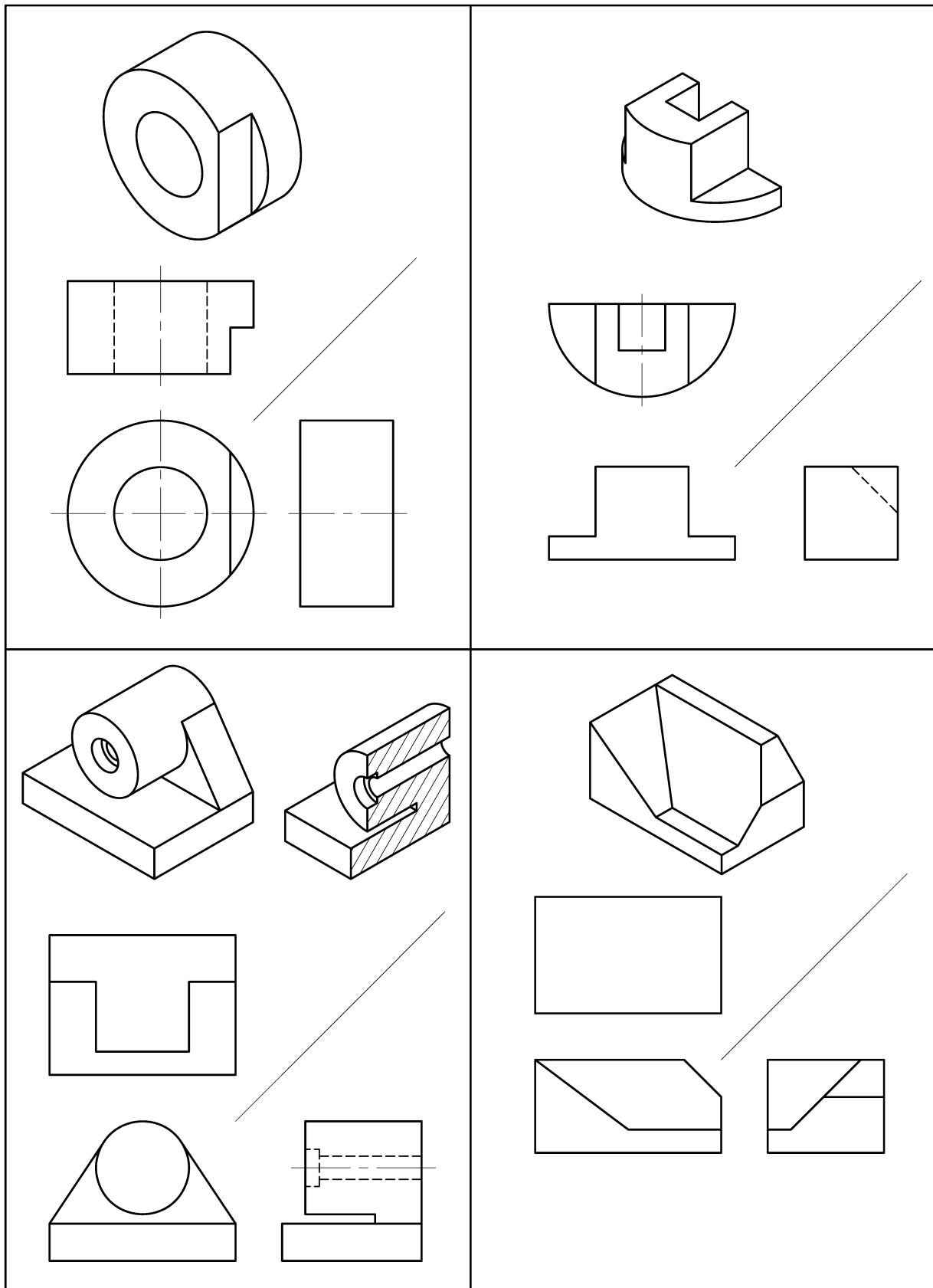


3. For the respective pictorial figure, add the missing lines of the appropriate line type on the corresponding orthographic views (in Third Angle projection) shown below. All projection lines must be shown using pencil as thin and light lines, outlines using blue/black pen and centre and hidden lines using red pen. Complete it on the worksheet given to you and submit.









BLANK

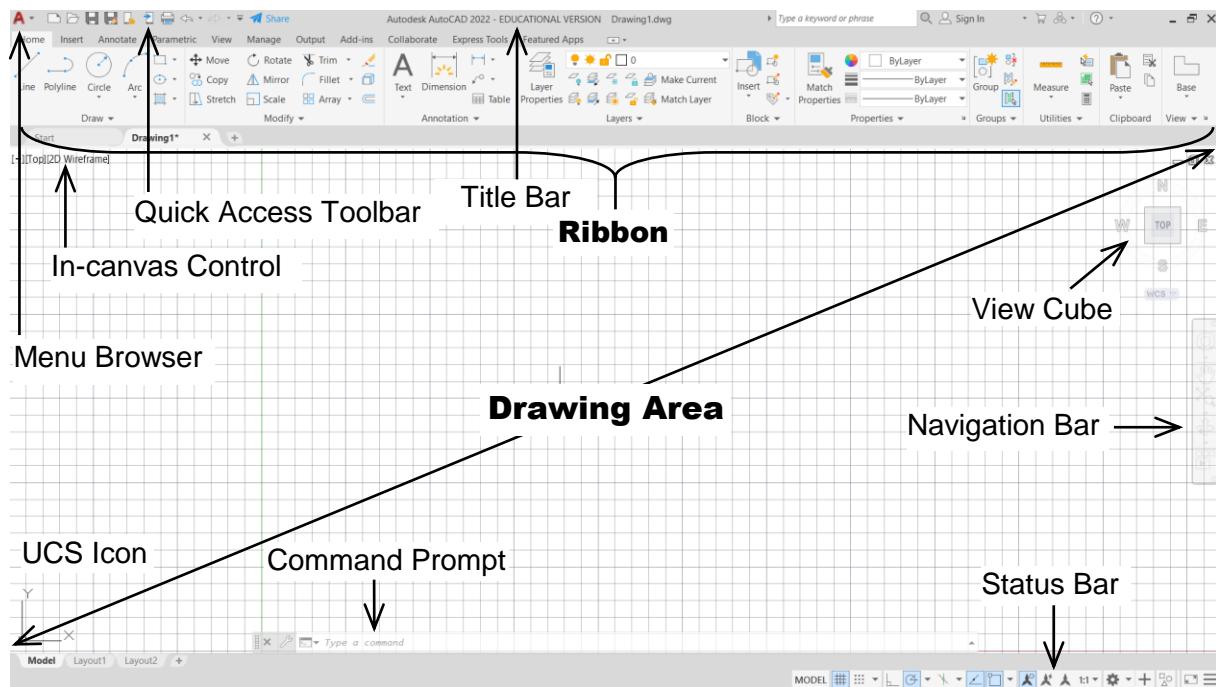
UNIT 2 COMPUTER-AIDED DRAFTING USING AUTOCAD

Learning Objectives

By the end of this unit, students should be able to:

- Recognise AutoCAD graphics window.
- Execute basic operation.
- Recognise the respective co-ordinates entry format for various co-ordinate systems.
- Select appropriate co-ordinate system to create geometries accurately.
- Organise geometric objects with appropriate colours, linetypes and linewidths using layer properties manager.
- Select objects efficiently using one or more of these means: cursor, window and crossing.
- Manage the drawing view using ZOOM and PAN via the mouse wheel.
- Construct geometric figures of engineering components with a range of AutoCAD commands.
- Choose precise geometrical locations on drawing objects using appropriate object snaps.
- Place notes on drawing with the TEXT command.

2.1 The AutoCAD Graphics Window



AutoCAD displays information in several areas:

Title bar

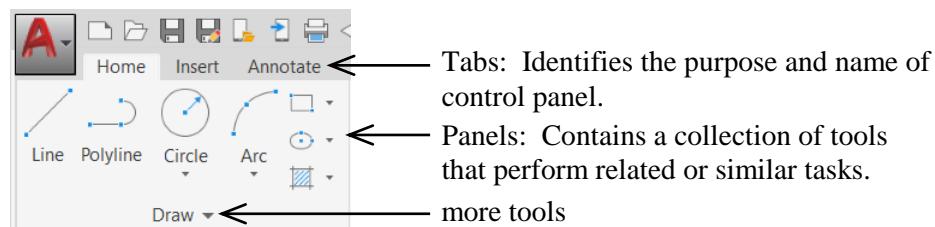
It is located across the top of the screen. It tells you the name of the software you are running (AutoCAD) and the name of the drawing that is currently opened.

Menu Browser

It contains a number of menus for categorising commands. A *left mouse click* on the menu browser will display the menu list for selecting the desired command.

Ribbon

The ribbon is a special tool palette that provides a single, compact placement of tools and controls that are relevant to the current workspace. It eliminates the need to display multiple toolbars, reducing clutter in the application and maximizing the area available for work using a single compact interface. It is composed of a series of panels, which are organised into tabs labelled by task.



Command Prompt

This is the traditional section for entering AutoCAD commands and where AutoCAD responds with prompts and messages. If you are unsure as to what AutoCAD expects of you, the prompt line is one place to look.

Status Bar

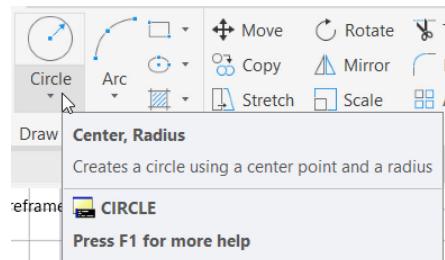
The status bar is located at the bottom right of the AutoCAD window. The status bar displays the cursor location, drawing tools, and tools that affect the drawing environment.

The status bar provides quick access to some of the most commonly used drawing tools, like toggle settings such as grid, snap, polar tracking, and object snap. Additional settings for some of these tools can be accessed by clicking their drop down arrows.



Tooltips

Tooltips are descriptive messages that are displayed near the cursor when it is hovered over a toolbar or panel button, or menu item. It provides a simple description about the command.



2.2 Entering commands

Commands can be entered in AutoCAD via the buttons located on the ribbon or toolbars, pull-down menus and the keyboard. The command input is then displayed either in the command prompt area or on the screen.

Some commands contain additional options given within the brackets []. To choose a different option, enter the letters capitalized in one of the options in the brackets. For example, when you pick the circle command, the following prompt is displayed:

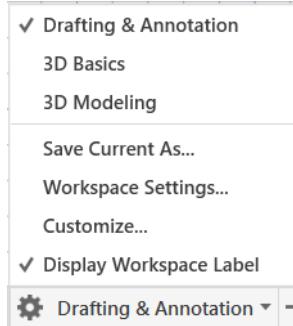
Specify center point for circle or [3P/2P/Ttr (tan, tan, radius)]:

You can specify the centre point or choose a different option such as the three-point option (3P) by keying **3p**.

2.3 Workspaces

Workspaces are sets of menus, toolbars, palettes, and ribbon control panels that are grouped and organised to work in a custom, task-oriented drawing environment. When a workspace is selected, only the menus, toolbars, and palettes that are relevant to a task are displayed. Workspaces that are already defined in the product are:

1. 2D Drafting & Annotation
2. 3D Basics
3. 3D Modeling



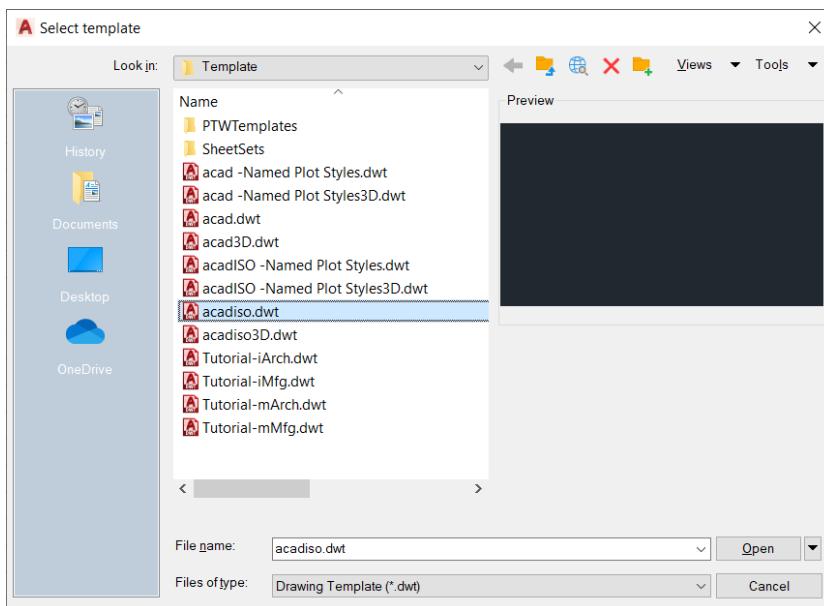
Workspaces can be switched via the Workspace Switching button on the status bar.



2.4 Create a new drawing

Command: New or QNew or 

The “Select Template” dialog box is displayed using the above command. Select a template file (“acadiso.dwt” for metric drawing).



2.5 LINE command

Ribbon access: ***Home tab > Draw panel > Line***

The **LINE** command allows user to draw straight line segments.

The input and command options in the LINE command are:

- first point - specify the starting point for the first line segment.
- next point - specify the endpoint for the line segment.
- Close - add a final line segment between the first and last point.
- Undo - undo the previous line segment during the Line command.

To exit the **LINE** command, just press **Enter** or **Esc**.

2.6 AutoCAD Coordinate Systems

Many commands in AutoCAD often require user to input a point. The point can be specified in one of two basic methods:

- Picking a point on the screen with the pointing device (mouse).
- Entering the coordinates of the point at the Command prompt (when it is requesting point entry) either in Absolute Cartesian coordinates, Relative Cartesian coordinates or Relative Polar coordinates format.

2.6.1 Absolute Cartesian Coordinates

In this format, the 2D points are defined by two distances along the x and y axes measured from the origin point (0,0). The point is specified by entering its X, Y coordinates separated by commas. The following illustrates this method of constructing line segments:

```
Command: _line
Specify first point : 50,50
Specify next point or [Undo] : 250,50
Specify next point or [Close Undo] : 250,200
Specify next point or [Close Undo] : 100,200
Specify next point or [Close Undo] : 100,100
Specify next point or [Close Undo] : 50,100
Specify next point or [Close Undo] : c (for Close)
```

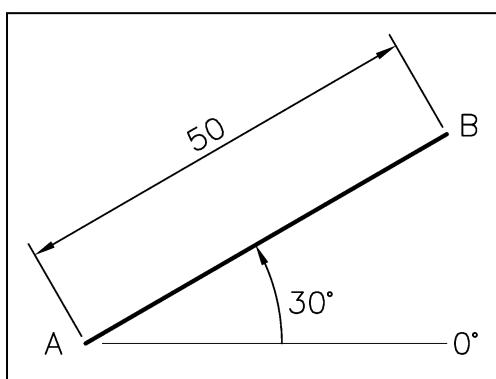
2.6.2 Relative Cartesian Coordinates

In this format, the 2D points are located in relation to the last specified position or point, rather than the origin. When specifying relative coordinates, the entry must be preceded the symbol “@”. The above example constructed by *Relative Cartesian coordinates* is as follow:

```
Command: _line
Specify first point : 300,50 (or pick any point on the screen)
Specify next point or [Undo] : @200,0
Specify next point or [Close Undo] : @0,150
Specify next point or [Close Undo] : @-150,0
Specify next point or [Close Undo] : @0,-100
Specify next point or [Close Undo] : @-50,0
Specify next point or [Close Undo] : c (for Close)
```

2.6.3 Relative Polar Coordinates

Relative Polar coordinates are used to specify a 2D point as a distance and angle from the last specified point. In AutoCAD, the angle is indicated by prefixing the angle with an angle bracket “<”, such as <20 for 20 degrees. Thus, the format for relative polar coordinates is “**@distance<angle**”. For example, to draw a line segment from A to B such that B is 50 unit from A, and at an angle of 30 degrees from the horizontal:



```
Command: _line
Specify first point : (specify point A)
Specify next point or [Undo] : @50<30
```

(Note: default positive angle direction is
counterclockwise where 0 degree
corresponds to the East.)

2.7 CIRCLE command

Ribbon access: **Home tab** > **Draw panel** > **Circle**

The **CIRCLE** command draws circles. Several options are available:

- Center, Radius - draws circle based on a centre point and radius.
- Center, Diameter - draws circle based on a centre point and diameter.
- 2P (2-Points) - draws circle based on two diameter points.
- 3P (3-Points) - draws circle based on three points on its circumference.
- Ttr (tan tan radius) - draws circle tangent to two existing objects at a specified radius.

2.8 Controlling the Drawing View

The **ZOOM** command allows user to specify which area of the drawing to display on the screen.

Ribbon access: **View tab** > **Navigate 2D panel**

Zooming does not change the absolute size of the drawing. It changes the size of the view within the graphic area. There are various **ZOOM** command options available such as Window, Previous, All, Scale, Center, etc. Refer to the command tooltips within AutoCAD for more details on each *Zoom* option.

The **PAN**  command moves around the view of the drawing without changing the view magnification.

The **MOUSE WHEEL** can **ZOOM** and **PAN** the drawing without invoking the **ZOOM** and **PAN** commands in the following ways:

- a. Roll the wheel button forward : Zoom In
- b. Roll the wheel button backward : Zoom Out
- c. Double-click the wheel button : Zoom Extents
- d. Hold down the wheel button and drag the mouse : Pan

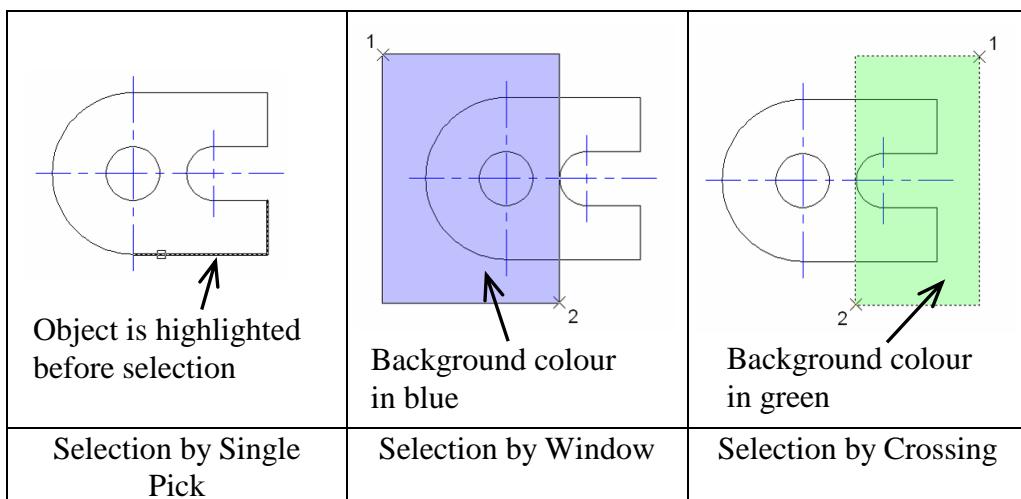
2.9 Objects Selection in Drawing

Objects in a drawing can be selected in two ways: Choose a command first, and then select objects, or select objects first, and then choose a command. Objects can be selected one at a time, multiples at the same time or combination of both.

For single object selection, a selection preview is available whereby the object is highlighted as the cursor rolls over it. This helps to identify the object before it is being selected.

For multiple objects selection, a rectangular selection area is specified. This rectangular selection area is defined by its two opposite corners. The background inside the area changes colour and becomes transparent. The direction of dragging the cursor from the first point to the opposite corner determines which objects are selected:

- **Window selection.** The cursor is dragged from left to right. Only those objects that enclosed completely by the rectangular area are selected. By default, the selection area is a **BLUE** background and its boundaries are solid lines.
- **Crossing selection.** The cursor is dragged from right to left. Those objects that enclosed or crossed by the rectangular area are selected. By default, the selection area is a **GREEN** background and its boundaries are dashed lines. This differentiates it from the Window selection.



The selected objects can be removed from the current selection set by holding down <SHIFT> and selecting them again with any of the above methods.

2.10 RECTANGLE command

Ribbon access: **Home tab > Draw panel > Rectangle**

By default, the **RECTANGLE** command draws rectangles by specifying two points that define the opposite corners of the rectangle. A number of other options are also available.

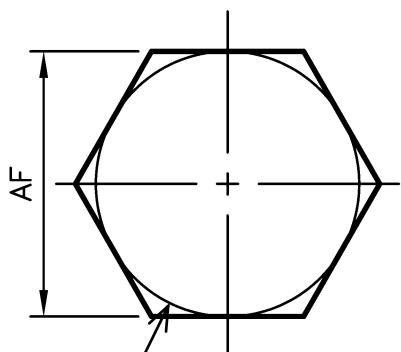
2.11 POLYGON command

Ribbon access: **Home tab > Draw panel > Polygon**

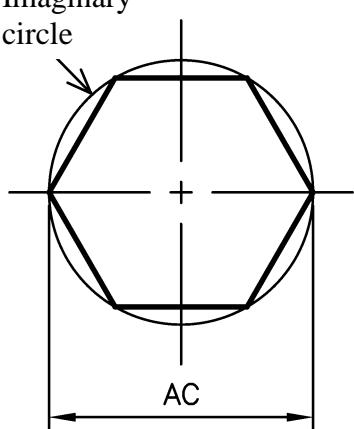
The **POLYGON** command is used to create regular polygons from 3 to 1024 equal sides. The polygons are drawn by any of the three methods:

- Edge - draws polygon by specifying the length of one edge. The length may be defined by picking two points.
- Inscribed - draws polygon by fitting it inside a circle of a specified radius.
- Circumscribed - draws polygon by fitting it outside a circle of a specified radius tangentially.

Using **POLYGON** command to create hexagon:



By Circumscribed option –
This option of the polygon command is used if the size of the hexagon is specified by the distance across the flats (AF) as shown. The radius of the circle is half the across flats' distance.

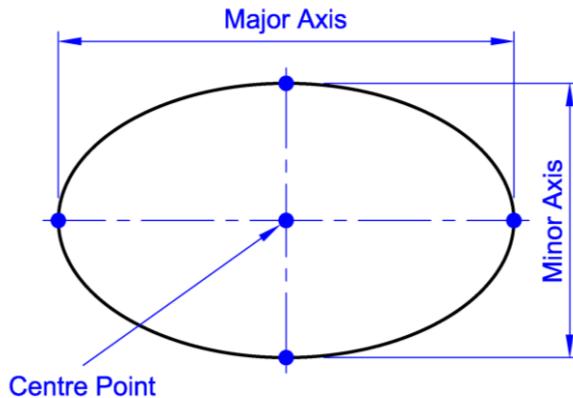


By Inscribed option –
This option of the polygon command is used if the size of the hexagon is specified by the distance across the corners (AC) as shown. The radius of the circle is half the across corners' distance.

2.12 ELLIPSE command

Ribbon access: **Home tab > Draw panel > Ellipse**

The **ELLIPSE** command creates ellipses and elliptical arcs. Ellipses are elongated circles drawn with two principal axes. The longer axis is known as the major axis and the shorter axis is known as the minor axis as shown in the figure below:



Default option:

Command: `_ellipse`

Specify axis endpoint of ellipse or

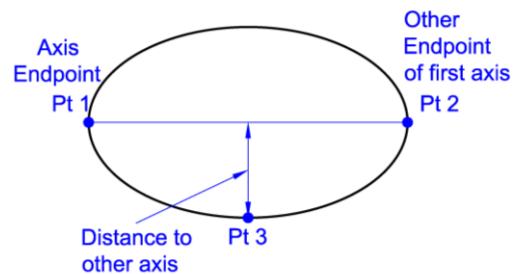
[Arc Center] : (pick Pt 1)

Specify other endpoint of axis : (pick Pt 2)

Specify distance to other axis or [Rotation] :

(enter distance from first axis to Pt 3) or

(pick Pt 3)



Center option:

Command: `_ellipse`

Specify axis endpoint of ellipse or

[Arc Center] : **c**

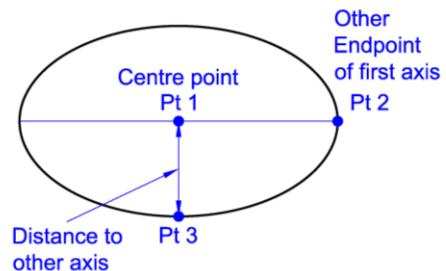
Specify center of ellipse : (pick Pt 1)

Specify other endpoint of axis : (pick Pt 2)

Specify distance to other axis or [Rotation] :

(enter distance from first axis to Pt 3) or

(pick Pt 3)



2.13 ERASE command

Ribbon access: **Home tab > Modify panel > Erase**

The **ERASE** command deletes objects from drawing.

2.14 UNDO command

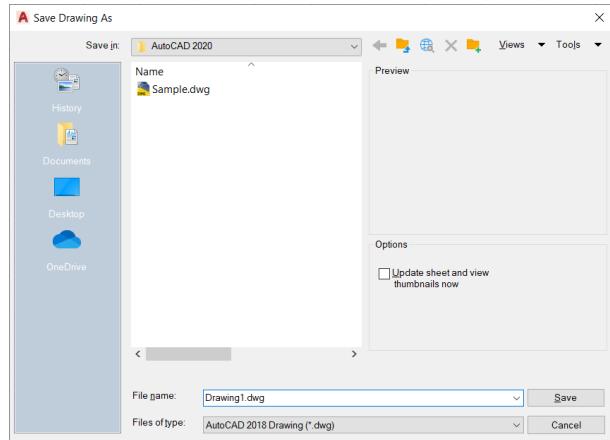
The **UNDO** command reverses the effect of commands.

2.15 REDO command

The *REDO* command reverses the effect of UNDO.

2.16 SAVE AS & SAVE commands

2.16.1 SAVE AS command



The *SAVE AS* command saves a copy of the current drawing under a *new file name*.

From the Menu Browser: select **File ➔ Save As**

The Save Drawing As dialog box appears.

Select the DRIVE and FOLDER location, and enter the new file name of drawing.

2.16.2 SAVE command

The *SAVE* command saves the drawing under the existing file name. However, when saving the drawing for the first time, the Save Drawing As dialog box will be displayed.

2.17 EXIT command

To exit from AutoCAD and return to Windows, use the *EXIT* command from the Menu Browser. If drawing has not been previously saved, the command will prompt whether to save changes, discard changes, or cancel. If drawing has already been saved, command will close AutoCAD.

2.18 TRIM command

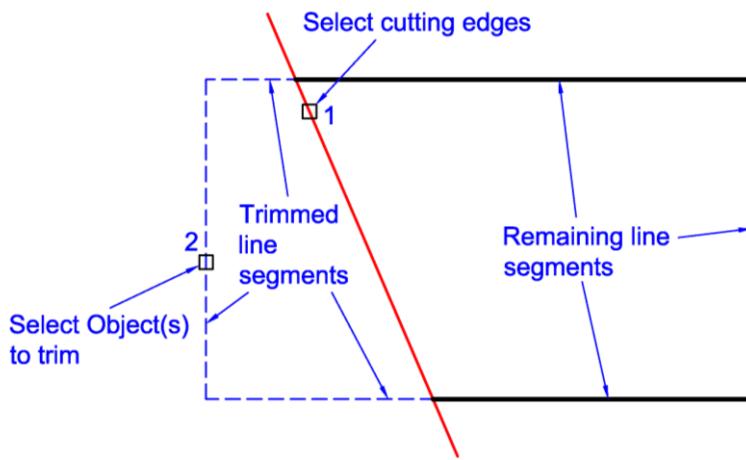
Ribbon access: **Home tab > Modify panel > Trim**

The *TRIM* command “cuts off” the portion of the object(s) to meet the edges of other objects. All objects are acting as cutting edges by default. Selected objects that cannot be trimmed will be deleted instead. Alternatively, object(s) may be selected as cutting edges prior to trimming.

The **TRIM** command has several options:

- cuTting edges - select one or more objects as boundary for the trim.
- Crossing - select objects to be trimmed that are within and crossing a rectangular area defined by two points.
- mOde - set the default trim mode either to Quick or Standard.
- eRase - erases objects during the trim process.
- Shift-select - extends the objects to the cutting edges instead of trimming.

Trim example (with Cutting Edge Selection):



Command: _trim

Current settings: Projection=UCS, Edge=None, Mode=Quick

Select object to trim or shift-select to extend or

[cuTting edges Crossing mOde Project eRase] : T

Current settings: Projection=UCS Edge=None

Select cutting edges ...

Select objects or <select all> : (pickbox 1)

Select objects : (press **Enter** to end cutting edges selection)

Select object to trim or shift-select to extend or

[cuTting edges Crossing mOde Project eRase] : (pickbox 2)

2.19 OFFSET command

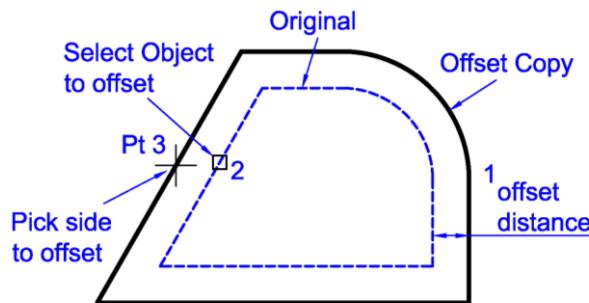
Ribbon access: **Home tab** > **Modify panel** > **Offset**

The **OFFSET** command makes parallel copies of the selected objects. It creates parallel lines, concentric circles, parallel curves. AutoCAD requires three pieces of information: (1) the offset distance, (2) the objects to offset, and (3) the side on which to place the offset copy.

The *OFFSET* command has several other options:

- **Through** - constructs the offset copy “through” a point.
- **Erase** - erases the source object after the offset copy.
- **Layer** - specifies the destination layer.
- **Multiple** - continue the offset with the current offset distance without reselect a source object.
- **Undo** - reverses the previous offset.

Offset example:



Command: `_offset`

Current settings: Erase source=No Layer=Source OFFSETGAPTYPE=0

Specify offset distance or [Through Erase Layer] <Through> : (Enter offset distance)

Select object to offset or [Exit Undo] <Exit> : (pickbox 2)

Specify point on side to offset or [Exit Multiple Undo] <Exit> : (pick Pt 3)

The offset distance can be specified by picking two points on the screen, or by entering the distance value. The “Select object to offset:” prompt repeats to allow the *OFFSET* to continue, but just one object at a time.

2.20 OBJECT SNAP

The *OBJECT SNAP* (or *OSNAP*) is a tool used to locate an exact geometric point on an object, such as the end point of a line or centre of a circle. Object snap can only be specified whenever a point is prompted. If object snap is used at the Command prompt, an error message will be displayed. To specify an object snap at a prompt for a point:

- Press `<SHIFT>` (or `<CTRL>`) key and right-click to display the Object Snap shortcut menu, **or**
- Right-click and choose an object snap from the Snap Overrides submenu, **or**
- Enter the name of an object snap.

AutoCAD uses a visual aid called AutoSnap™ that displays a marker and a tooltip when the cursor moves over an object snap location. This provides a clue of which object snaps are in effect. The object snap will stay in effect only for the prompted point.

Some of the useful object snaps are illustrated below:

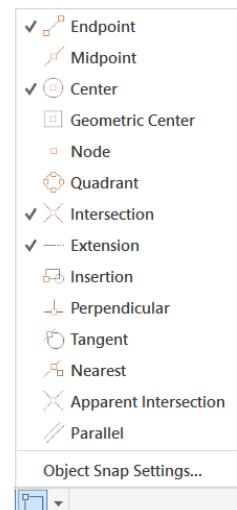
Icon	Description	Illustration	Icon	Description	Illustration
	ENDpoint Snaps to the closest endpoint of objects such as lines or arcs.			QUAdrant Snaps to the closest quadrant of an arc, circle, or ellipse (the 0°, 90°, 180°, 270° points).	
	MIDpoint Snaps to the midpoint of objects such as lines or arcs.			TANgent Snaps to the tangent of an arc, circle, ellipse or spline.	
	INTersection Snaps to the intersection (or extended intersection) point of objects.			PERpendicular Snaps to a point on an object that forms a normal or perpendicular alignment with another object.	
	CENter Snaps to the centre of an arc, circle, or ellipse.			Geometric CEnter Snaps to the centre of a polygon, closed polylines, or rectangle.	
	NEArest Snaps along and on to the object.				

2.21 Setting Running Object Snap

Running object snap is used to set those object snap modes that are used repeatedly. The required object snaps can be selected/unselected from the listing after clicking the drop down arrow next to the Object Snap button on the status bar.

The default running object snaps are Endpoint, Center, Intersection and Extension.

The running object snaps can be turned on and off by clicking the



2.22 Editing using Grips

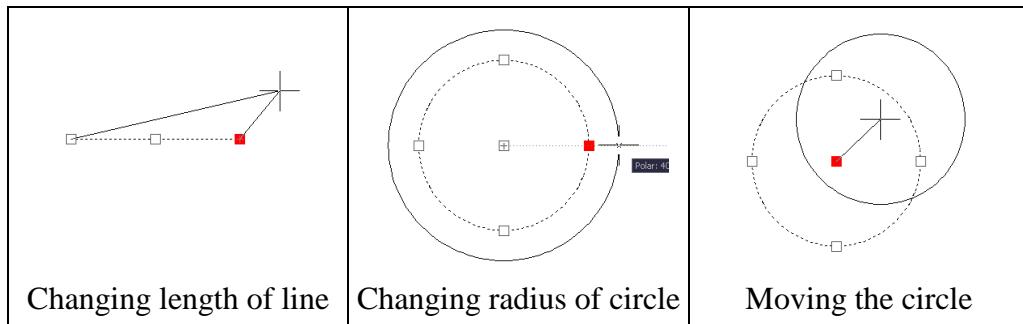
Grips allow direct editing of objects. Grips are small, coloured squares that are displayed at strategic points on the selected objects. These grips can be dragged to stretch, move, rotate, scale, or mirror objects quickly.

To use grips, pick one or more objects to be edited, with **NO command active**. AutoCAD will then highlight the selected objects by showing them in dashed lines, and display the grips. These objects can then be edited by manipulating their grips. To exit grip editing, press the <ESC> key.

Standard grips are assigned names and default colours as follows:

Grip Colour	Name	Meaning
Blue (Colour 150)	Cold	Grip is not selected.
Dark Red (Colour 12)	Hot	Grip is selected, and object can be edited.
Pink (Colour 11)	Hover	Cursor is positioned over grip.

When the end grip of a line or arc, or quad grip of a circle is selected, the object can be stretched or lengthened: Lines and arcs will be lengthened or shortened, while radius of circles will be changed. When the midpoint grip of a line or arc, or the centre of a circle is selected, the object can be moved.



2.23 LAYER command or LAYER PROPERTIES MANAGER

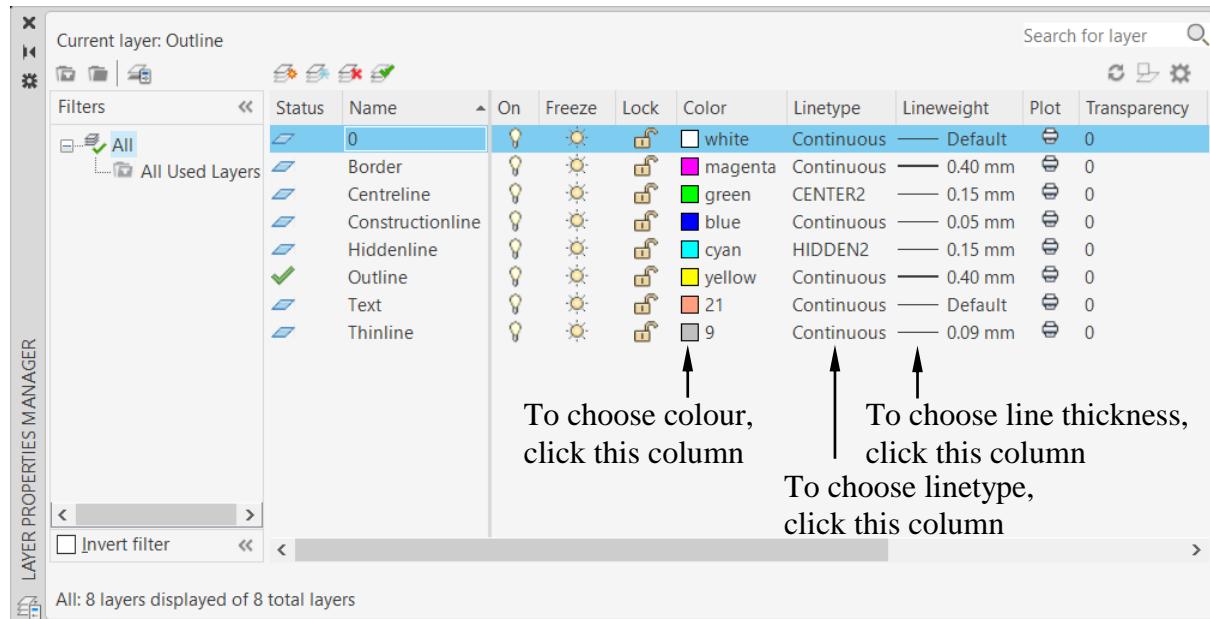
Ribbon access: **Home tab > Layers panel > Layer Properties**

Layers are used to group information by function and to enforce line type, colour and other standards. Layers are the equivalent of the overlays used in paper-based drafting. By creating layers, similar types of objects can be associated by assigning them to the same layer. For example, outlines, hidden lines, centre lines can be placed on separate layers. Layers can be used:

- to assign properties (colour, linetype, linewidth) to objects. Layers can be assigned a specific colour, linetype, linewidth so that all objects on these layers will assume these assigned properties.

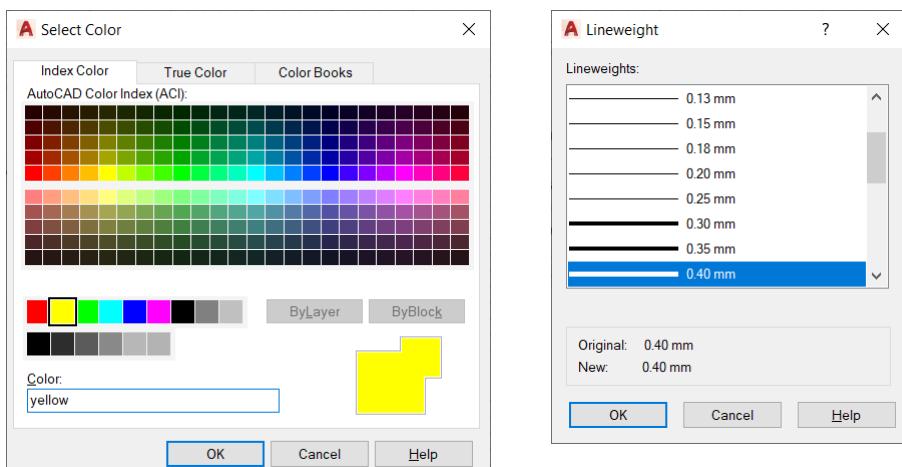
- to control the visibility of objects by turning the layers on/off, or freezing/thawing the layers.
- to control whether objects on a layer can be modified. Layers can be locked to prevent the objects on these layers from being modified.

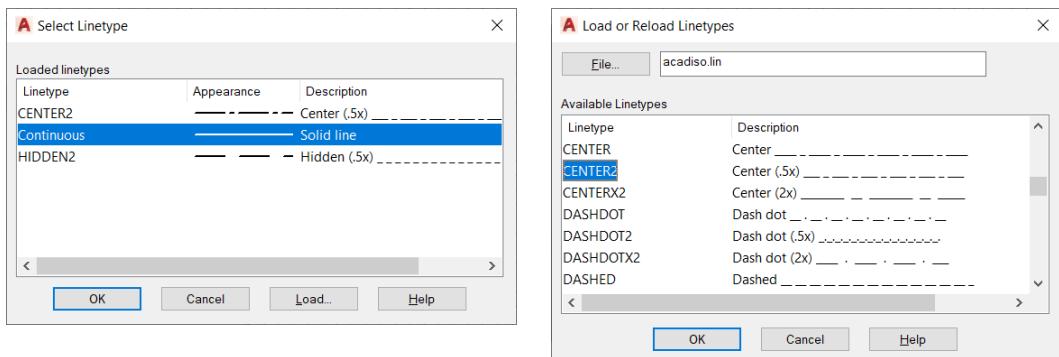
Layer properties manager is used to create layers and assign colour, linetype, linewidth to the layers for the drawing.



Below are the steps:

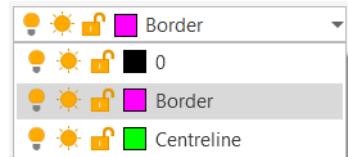
1. Click on to create a new layer in the Layer Properties Manager.
2. Enter the layer name, eg. Centreline.
3. Pick the colour under the “Color” column to select to the required colour.
4. Pick the ‘Continuous’ under the Linetype column to change to the desired linetype. If the linetype is not loaded, then click on ‘Load...’ to load in the desired linetype.
5. Select the required line thickness after picking the linewidth column.
6. Repeat step 3 to 6 for Outline, Hiddenline and Border.
7. Click **OK** once all layers with their respective colour, linetype and linewidth have been created.





Create a 10 mm border from the edge of the A3 drawing paper:

1. Click on the layer dropdown, select and select the Border to set Border as the current layer.
2. Click on the Rectangle icon in the Draw panel.



Command: `_rectang`

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width] : **10,10**
Specify other corner point or [Area/Dimensions/Rotation] : **410,287** (in command line)

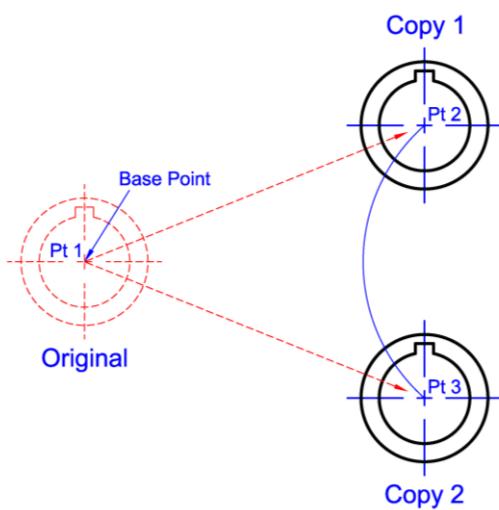
3. Set the layer to Outline as the current layer. This will allow any subsequent objects created will be associated to the Outline layer.

2.24 COPY command

Ribbon access: **Home tab > Modify panel > Copy**

The **COPY** command creates one or more copies of objects at a specified distance and direction from the originals. The **COPY** command repeats until the **ESC** key is pressed or the **Exit** option is chosen. AutoCAD requires three pieces of information to copy objects under the default *Displacement* Option: (1) the objects to be copied, (2) the point from which the copying takes place, and (3) the location to place the copies.

Copy example:



Command: `_copy`

Select objects : (pick one or more objects)

Select objects : (press **Enter** to end object selection)

Specify base point or [Displacement mOde] <Displacement> : (pick Pt 1)

Specify second point or [Array] <use first point as displacement> : (pick Pt 2)

Specify second point or [Array Exit Undo] <Exit> : (pick Pt 3)

Specify second point or [Array Exit Undo] <Exit> : (press **Enter** to exit)

The *COPY* command has following options:

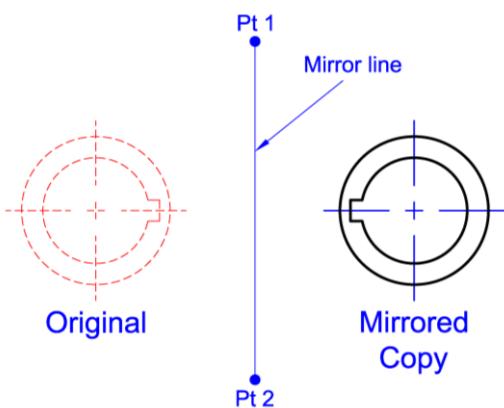
- **Displacement** - specifies a relative distance and direction using coordinates.
- **mOde** - controls Single or Multiple copying.
- **Array** - arranges a specified number of copies in a linear array.
- **Undo** - reverses the last copy action.

2.25 MIRROR command

Ribbon access: **Home tab** > **Modify panel** > **Mirror**

The *MIRROR* command creates mirrored copies about a specified axis. The original objects may be retained or deleted after mirrored. AutoCAD required three things to mirror objects: (1) the objects to be mirrored, (2) the mirror line, and (3) whether the source objects should be deleted.

Mirror example:



Command: `_mirror`

Select objects : (pick one or more objects)

Select objects : (press **Enter** to end object selection)

Specify first point of mirror line : (pick Pt 1)

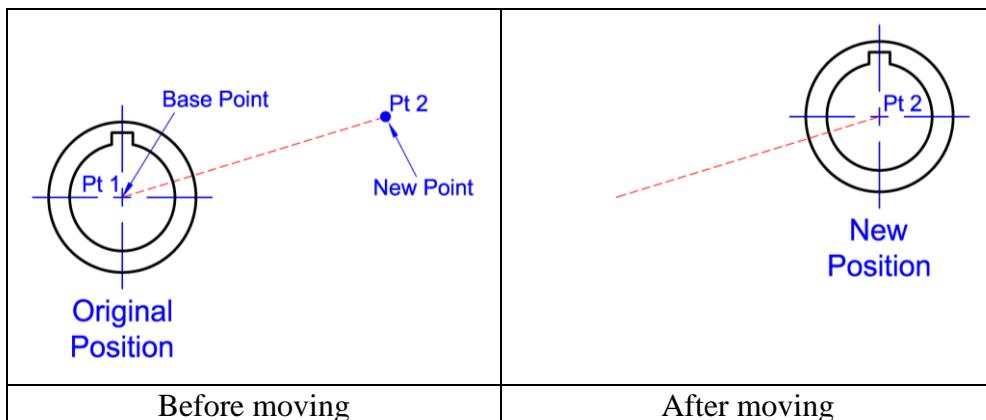
Specify second point of mirror line : (pick Pt 2)

Erase source objects? [Yes No] <N> : (enter **Y** or **N**)

2.26 MOVE command

Ribbon access: **Home tab > Modify panel > Move**

The **MOVE** command moves objects at a specified distance and direction from the originals. AutoCAD requires three things to move objects: (1) the objects to move, (2) the point from which the move takes place, and (3) the new location to place the objects. An example of using the **MOVE** command is shown below:



Command: `_move`

Select objects : (pick one or more objects)

Select objects : (press **Enter** to end object selection)

Specify base point or [Displacement] <Displacement> : (pick Pt 1)

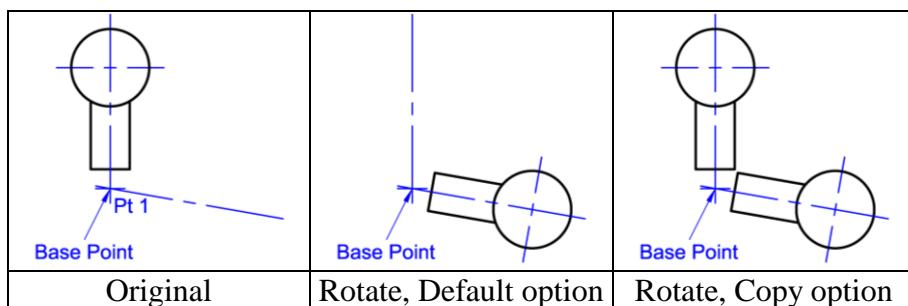
Specify second point or <use first point as displacement> : (pick Pt 2)

2.27 ROTATE command

Ribbon access: **Home tab > Modify panel > Rotate**

The **ROTATE** command rotates objects about a base point (centre of rotation). The **Copy** option will rotates a copy of the objects, leaving the original in place.

Rotate example:



Command: `_rotate`
Current positive angle in UCS : ANGDIR=counterclockwise ANGBASE=0
Select objects : (pick one or more objects)
Select objects : (press **Enter** to end object selection)
Specify base point : (pick Pt 1)
Specify rotation angle or [Copy Reference] <0> : (enter an angle value, such as **-100**)

To use the **Copy** option, on the following prompt:

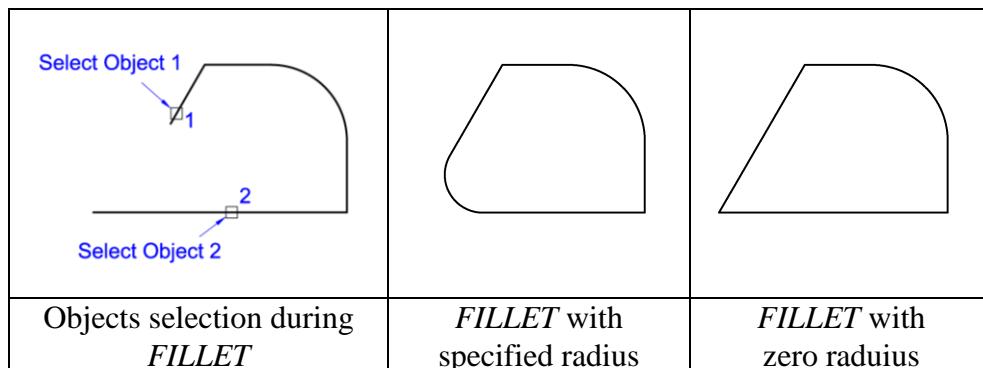
Specify rotation angle or [Copy Reference] <0> : (type **c** for “rotate copy”)

2.28 FILLET command

Ribbon access: **Home tab > Modify panel > Fillet/Chamfer**

The **FILLET** command connects two objects of lines, arcs, circles, etc with a tangential arc of specified radius or perfect intersection (zero radius). The two selected objects are trimmed or extended as necessary so that an arc fits between them. A zero radius fillet will connect the two selected objects with a perfect intersection.

Fillet example:



Command: `_fillet`
Current settings: Mode = TRIM, Radius = 0.0000
Select first object or [Undo Polyline Radius Trim Multiple] : (pickbox 1)
Select second object or shift-select to apply corner or [Radius] : (type **r** to set radius)
Specify fillet radius <0.0000> : (key in fillet radius, e.g. **10**)
Select second object or shift-select to apply corner or [Radius] : (pickbox 2)

Some useful options of **FILLET** command:

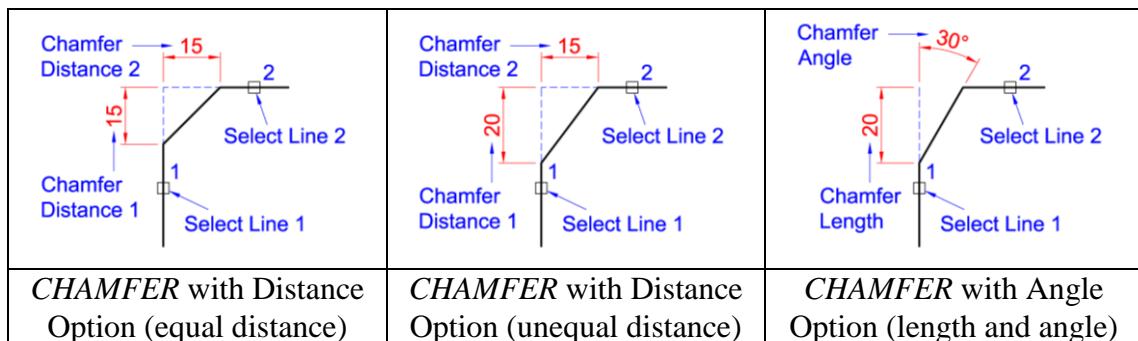
- **Undo** - undo last fillet operation.
- **Radius** - specifies the fillet radius.

- Trim - specifies whether the selected objects are trimmed or extended to endpoints of the fillet arc or left unchanged.
- Multiple - continues the *FILLET* command.
- Shift-select - changes fillet radius to zero and apply corner to selected objects.

2.29 CHAMFER command

Ribbon access: **Home tab > Modify panel > Fillet/Chamfer**

The *CHAMFER* command connects two non-parallel lines by trimming or extending them to join with an angled line. A zero distance chamfer will connect the two selected lines with a perfect intersection. The dimensions of the angled line can be specified by Distance or Angle options as illustrated.



Command: `_chamfer`

(Trim mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000

Select first line or [Undo Polyline Distance Angle Trim mETHOD Multiple] : (pickbox 1)

> For Distance Option:

Select first line or [Undo Polyline Distance Angle Trim mETHOD Multiple] : (type **d**)

Specify first chamfer distance <0.0000> : (key in chamfer distance 1, e.g. **15**)

Select second chamfer distance <15.0000> : (press **Enter** if same as the first)

> For Angle Option:

Select first line or [Undo Polyline Distance Angle Trim mETHOD Multiple] : (type **a**)

Specify chamfer length on the first line <0.0000> : (key in chamfer length, e.g. **20**)

Specify chamfer angle from the first line <0.0000> : (key in chamfer angle, e.g. **30**)

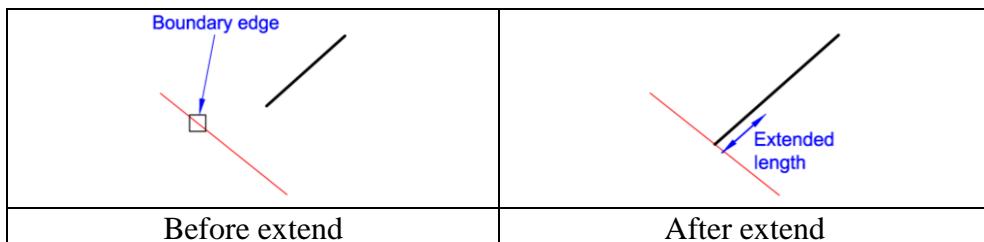
Select second line or shift-select to apply corner : (pickbox 2)

The Undo, Trim, Multiple and Shift-select options of *CHAMFER* command are similar in operation as the *FILLET* command.

2.30 EXTEND command

Ribbon access: **Home tab > Modify panel > Extend**

The **EXTEND** command lengthens objects to meet a boundary object. It is opposite to the **TRIM** command. It has similar command options as **TRIM**.



2.31 TEXT

Text conveys important information in your drawing. You use text for title blocks, to label parts of the drawing, to give specifications, or to make annotations.

AutoCAD provides various ways to create text. *Single line text* can be considered for placing short and simple line of text while *Multi-line text* can be used for placing paragraphs of formatted text.

2.31.1 Single Line Text

Ribbon access: **Home tab > Annotation panel > Single Line Text**

TEXT or **DTEXT** command creates one or more lines of text, ending each line by pressing **Enter**. Each text line is an independent object that can be relocated, reformatted or otherwise modified.

Command: **_dtext**

Current text style: "Standard" Text height: 0.0000

Specify start point of text or [Justify Style] : (specify the location of text)

Specify the height <2.500> : (enter text height)

Specify rotation angle of text <0> : (enter rotation angle)

Enter the text (the text and text cursor is being displayed on the screen), at the end of each line, press **ENTER**

AutoCAD
Single line text

Press **ENTER** on a blank line to exit command.

To edit line text

1. Double click on the text to be edited.
2. The text to be edited is highlighted on the screen and ready for editing.



3. After editing the text, press **ENTER** twice or click on the screen to exit.

2.31.2 Multi Line Text

Ribbon access: **Home tab > Annotation panel > Multiline Text**

MTEXT command creates one or more paragraphs of text in drawing and provides many formatting options through the In-Place Text Editor.

Command: **_mtext**

Current text style: “Standard” Text height: 2.5

Specify first corner : (specify the first corner of a text box)

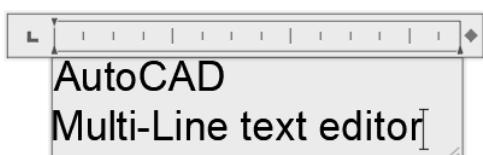
Specify opposite corner or [Height Justify ...] : (specify the other corner of text box)

The In-Place Text Window and Contextual Text Editor tab is available to enter and format text.

Contextual Text Editor Tab:



In-Place Text Window:



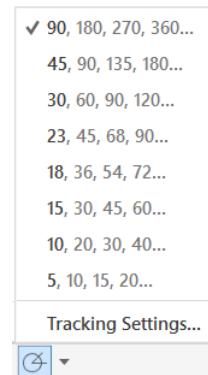
Upon completing the entry of text paragraphs, click anywhere outside the text window or on the “Close Text Editor”.

2.32 ORTHO and POLAR mode

ORTHO mode restricts cursor movement to horizontal and vertical to assist drawing at right angles while *POLAR* mode restricts cursor movement to specified angles or increments of specific angles.

The required polar tracking angle can be selected/unselected from a list of angles after clicking the drop down arrow next to the Polar Tracking button on the status bar. The default polar tracking angle is 90°.

Ortho and Polar mode cannot be turned on at the same time. Turning on Ortho will turn off Polar and vice-versa. The Ortho and Polar mode can be turned on and off by clicking the respective Ortho button and Polar Tracking button on the status bar. Alternatively, press <F8> and <F10> to toggle Ortho mode and Polar mode respectively



Other polar tracking angles not listed can be specified and set via the Drafting Settings dialog box (access via selecting Tracking Settings...).

2.33 POLAR ARRAY and RECTANGULAR ARRAY

Ribbon access: **Home tab** > **Modify panel** > **Polar Array / Rectangular Array**

The *ARRAYPOLAR* command will create copies of objects in a *circular pattern*. The *ARRAYRECT* command will create copies of objects in a *rectangular pattern*. The object in the array can be associative or independent. A preview of the resulting pattern is shown on the drawing area before accepting it.

Command: `_arraypolar`

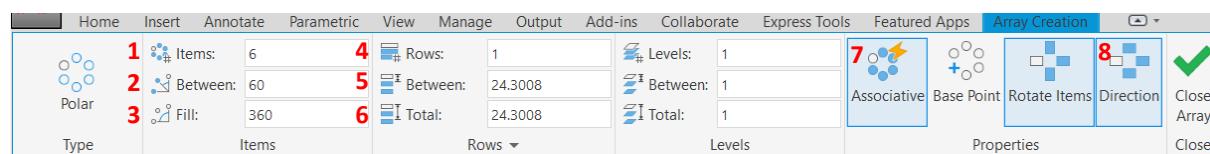
Select objects : (select the objects)

Select objects : (press **Enter** to end object selection)

Type = Polar Associate = Yes

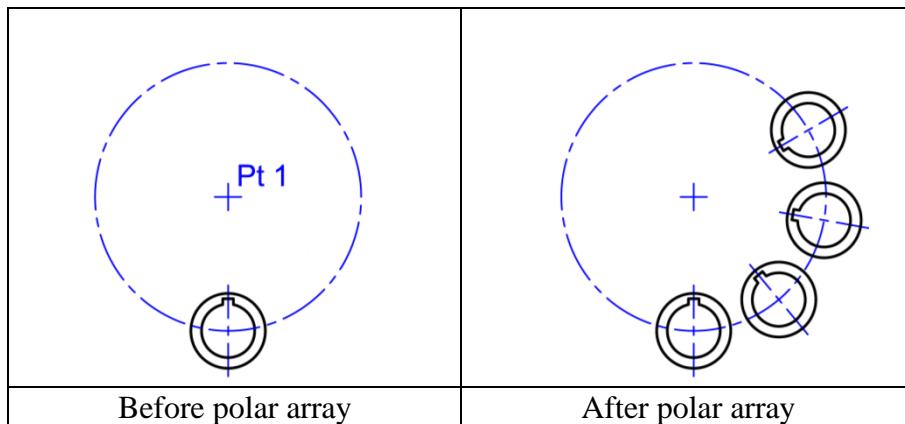
Specify center point of array or [Base point Axis of rotation] : (pick Pt 1)

The Array (Polar) Creation Ribbon Contextual Tab is displayed to specify the necessary information:



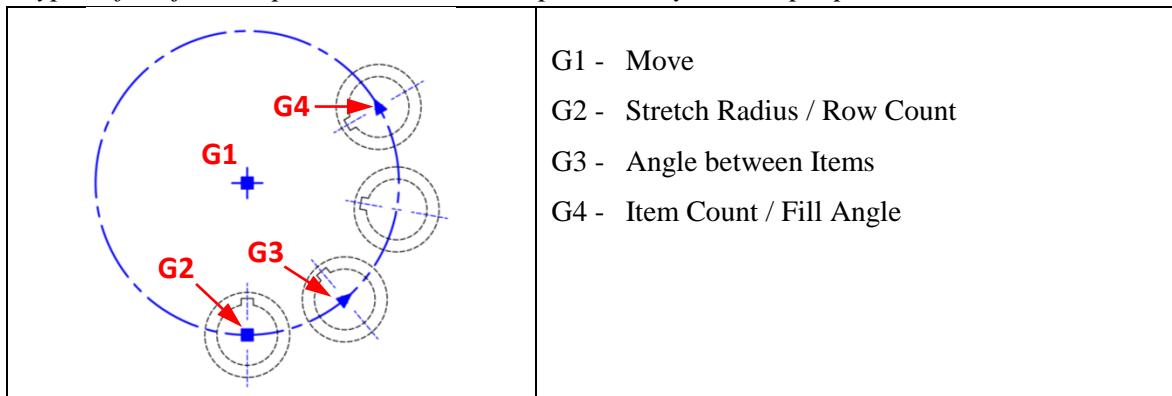
1. **Item Count:** Specify the number of items in the array.
2. **Angle between Items:** Specify the angle between items.
3. **Fill Angle:** Specify the angle between the first and last items in the array.

4. **Row Count:** Specify the number of rows in the array.
5. **Row Spacing:** Specify the distance between rows.
6. **Total Row Distance:** Specify the total distance between the first and last rows in the array.
7. **Associative:** Control whether an associative array is created.
8. **Direction:** Control the direction of creating an array, clockwise or counter-clockwise.



The associative polar array can be modified when it is selected, either via the *Array Ribbon Contextual Tab* or the object grips (blue boxes and arrows) of the array.

Types of Object Grips in the associative polar array and its purpose:



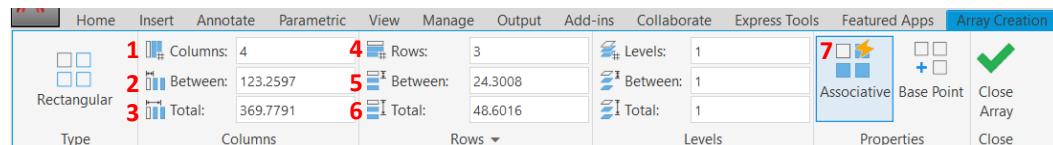
Command: `_arrayrect`

Select objects : (select the objects)

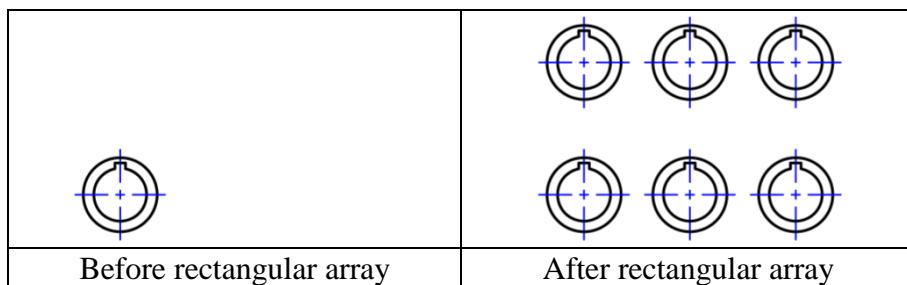
Select objects : (press **Enter** to end object selection)

Type = Rectangular Associative = Yes

The Array (Rectangular) Creation Ribbon Contextual Tab is displayed to specify the necessary information:

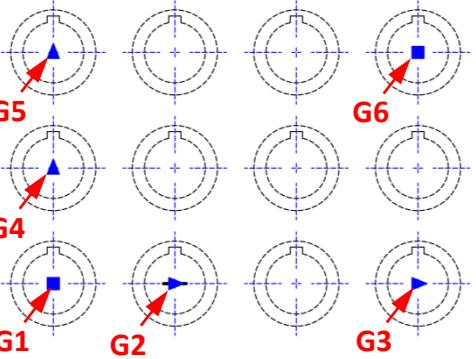


1. **Column Count:** Specify the number of columns in the array.
2. **Column Spacing:** Specify the distance between rows.
3. **Total Row Distance:** Specify the total distance between the first and last columns in the array.
4. **Row Count:** Specify the number of rows in the array.
5. **Row Spacing:** Specify the distance between rows.
6. **Total Row Distance:** Specify the total distance between the first and last rows in the array.
7. **Associative:** Control whether an associative array is created.



Likewise, the associative rectangular array can be modified when it is selected, either via the *Array Ribbon Contextual Tab* or the object grips (blue boxes and arrows) of the array.

Types of Object Grips in the associative rectangular array and its purpose:

	<p>G1 - Move</p> <p>G2 - Column Spacing</p> <p>G3 - Column Count / Total Column Spacing / Axis Angle</p> <p>G4 - Row Spacing</p> <p>G5 - Row Count / Total Row Spacing / Axis Angle</p> <p>G6 - Row and Column Count / Total Row and Column Spacing</p>
---	---

Other modifications that can be available to the associative array includes applying item overrides, editing source objects and replacing selected items. Refer to “Edit Associative Array” in AutoCAD help for details of such modifications.

2.34 CONSTRUCTION LINE command

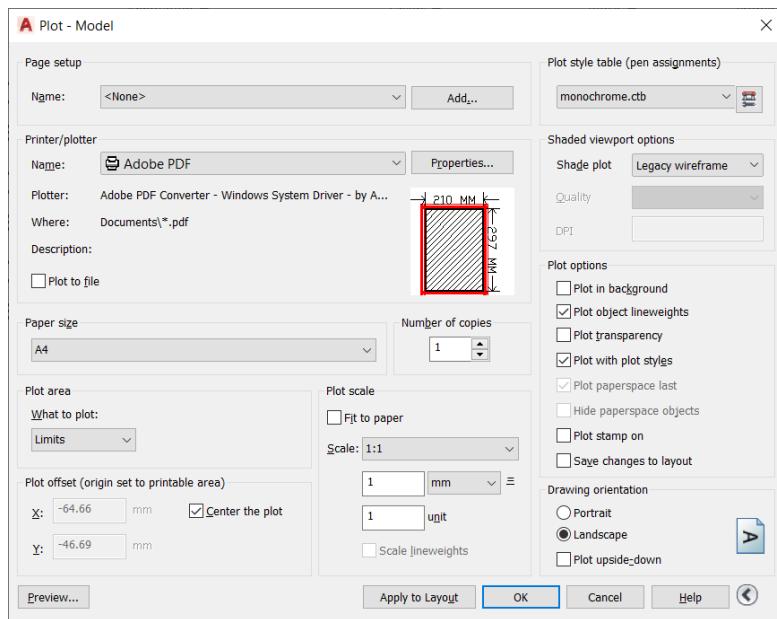
Ribbon access: **Home tab > Draw panel > Construction Line**

CONSTRUCTION LINE command creates an infinite line. It has the following options:

- **Hor** - creates horizontal construction line.
- **Ver** - creates vertical construction line.
- **Ang** - creates a construction line at a specified angle.
- **Bisect** - creates a construction line that divides the angle between two existing lines equally.
- **Offset** - creates a construction line parallel to an existing line at a specified distance.

2.35 PLOT command

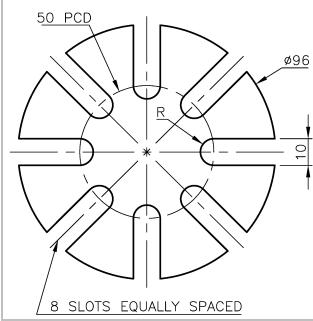
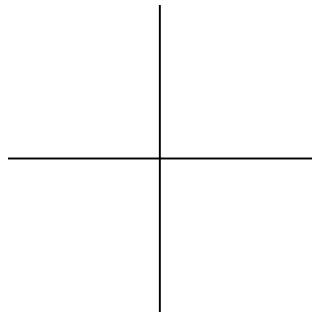
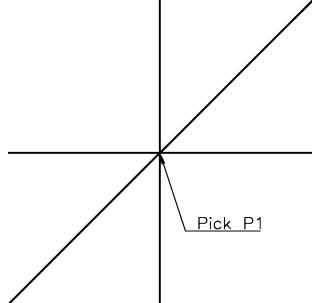
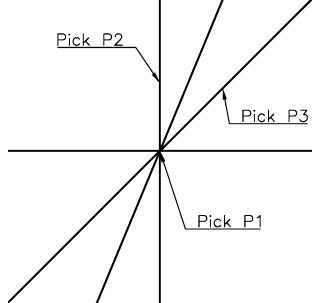
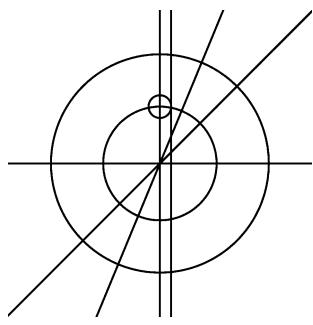
When the *PLOT* command is selected, a Plot Dialog box will be displayed for specifying the necessary setting required to print out the drawing on to the paper.



In the plot dialog box,

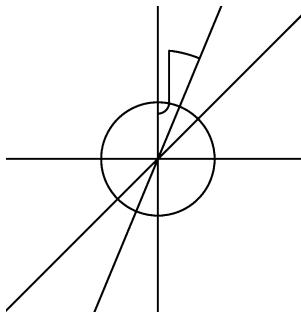
- a. select the plotter to use.
- b. select the paper size, drawing orientation.
- c. select the desired plot area - “Display” applies the active drawing view. “Limits” refer to the entire drawing area that is defined by the drawing limits. “Window” prints a rectangular area specified by coordinate locations like Zoom Window.
- d. select or specify the plot scale.
- e. select the plot style table - choose “monochrome” for black and white printing, “acad” for colour printing.
- f. plot preview (full preview) the final output. If output is not displayed to the requirement, repeat steps b, c, d and e.
- g. when it is ready to plot, choose OK.

2.36 Example

<p>Create the figure shown:</p> <p>STEP 1 Set outline layer to be the current layer.</p>	
<p>STEP 2 Create a vertical line and a horizontal line (any length).</p>	
<p>STEP 3 Select Construction Line icon. From the options, choose ANG by typing 'A' or 'a' from keyboard. Key-in the angle of 45 and osnap to intersection, pick P1.</p>	
<p>STEP 4 Select Construction Line icon. From the options, choose Bisect by typing 'B' or 'b'. Osnap to intersection, pick P1. Osnap to nearest, pick P2 & P3.</p>	
<p>STEP 5 Create the circles and offset the vertical line.</p>	

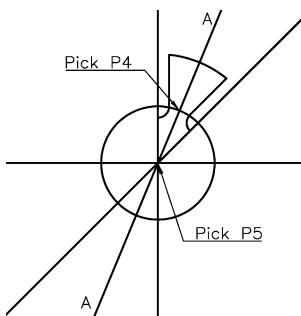
STEP 6

Trim the circles and straight line as shown.



STEP 7

Mirror the figure about the line AA. Use osnap intersection to pick P4 & P5 to define the first and second points of the mirror line.



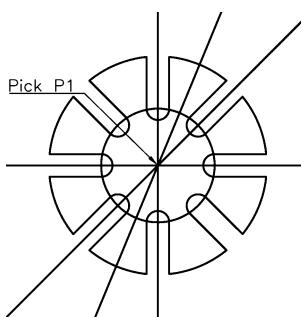
STEP 8

Array the figure about point P1, using **Polar Array**.

Centre point of the array : P1

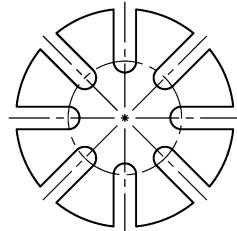
Number of items : 8

Angle to fill : 360



STEP 9

Change the vertical and horizontal lines into centrelines by changing the layer from outline layer into centreline layer. Tidy the centre lines as shown, i.e. centerline to be about 2 to 3 mm outside the concerned outline. Delete all unwanted lines.



STEP 10

Using **Rectangular Array**, obtain the figure shown by choosing a suitable row and column widths and angle.

Eg.

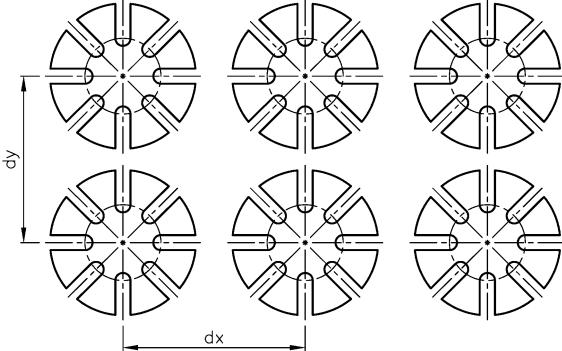
Rows: 2

Columns: 3

Row spacing: dy (positive or negative)

Column spacing : dx (positive or negative)

Angle of array : 0



2.37 TANGENCY

A tangent is a straight line which touches the circumference of a circle at one point only (point of tangency) as shown in Figure 2.1. A line drawn from the point of tangency to the centre of the circle is called a normal, and the tangent makes an angle of 90° with the normal.

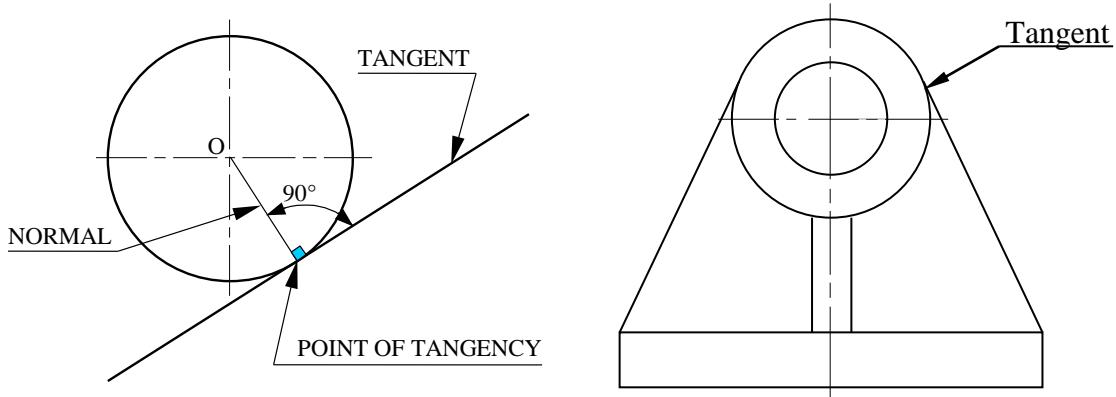


Figure 2.1

2.39 Geometrical Construction of Arcs

- 2.39.1 To draw an arc of given radius tangential to two given circles when the circles are outside the required arc: Figure 2.2.

1. Assume that the radii of the given circles are 20 mm and 25 mm, spaced 85 mm apart, and that the radius to touch them tangentially is 40 mm.
2. With centre A, draw an arc of radius equal to $20 + 40$ ($= 60$ mm).
3. With centre B, draw an arc of radius equal to $25 + 40$ ($= 65$ mm).
4. The above arcs intersect at point C. With a radius of 40 mm, draw an arc from point C as shown, and note that the points of tangency between the arcs lie along the lines joining the centre AC and BC.

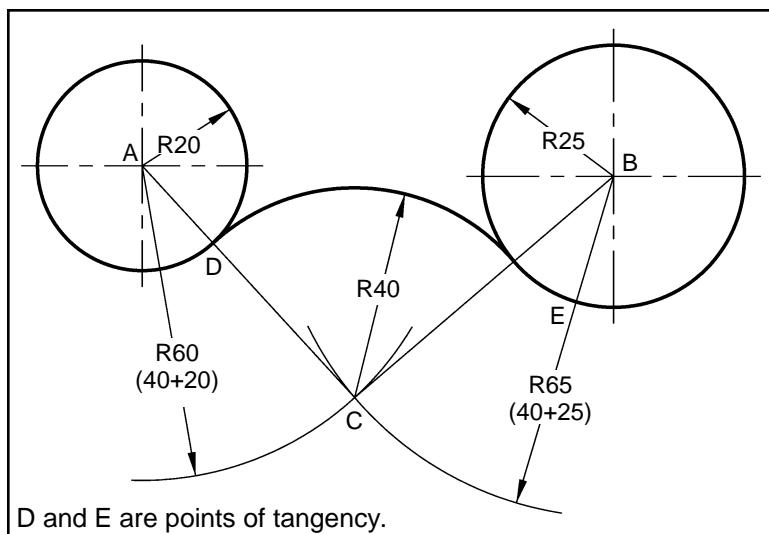


Figure 2.2

- 2.39.2 To construct an arc of given radius tangential to two given circles with the circles inside the required arc: Figure 2.3.

1. Assume that the radii of the given circles are 22 mm and 26 mm, spaced 86 mm apart and that the radius to touch them tangentially is 100 mm.
2. With centre A, draw an arc of radius equal to $100 - 22 (= 78 \text{ mm})$.
3. With centre B, draw an arc of radius equal to $100 - 26 (= 74 \text{ mm})$.
4. The above arcs intersect at point C. With a radius of 100 mm, draw an arc from point C, and note that in this case the points of tangency lie along line CA extended to D and along line CB extended to E.

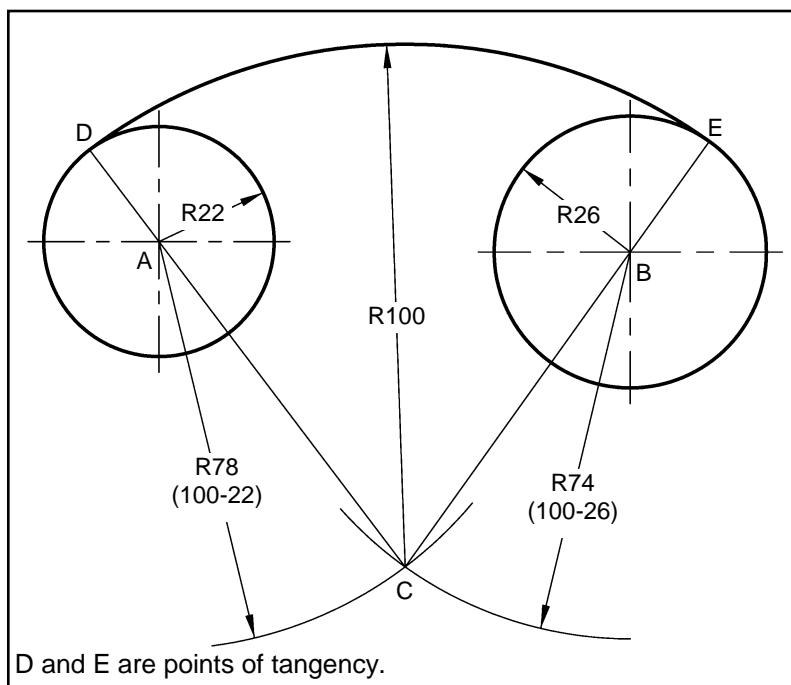
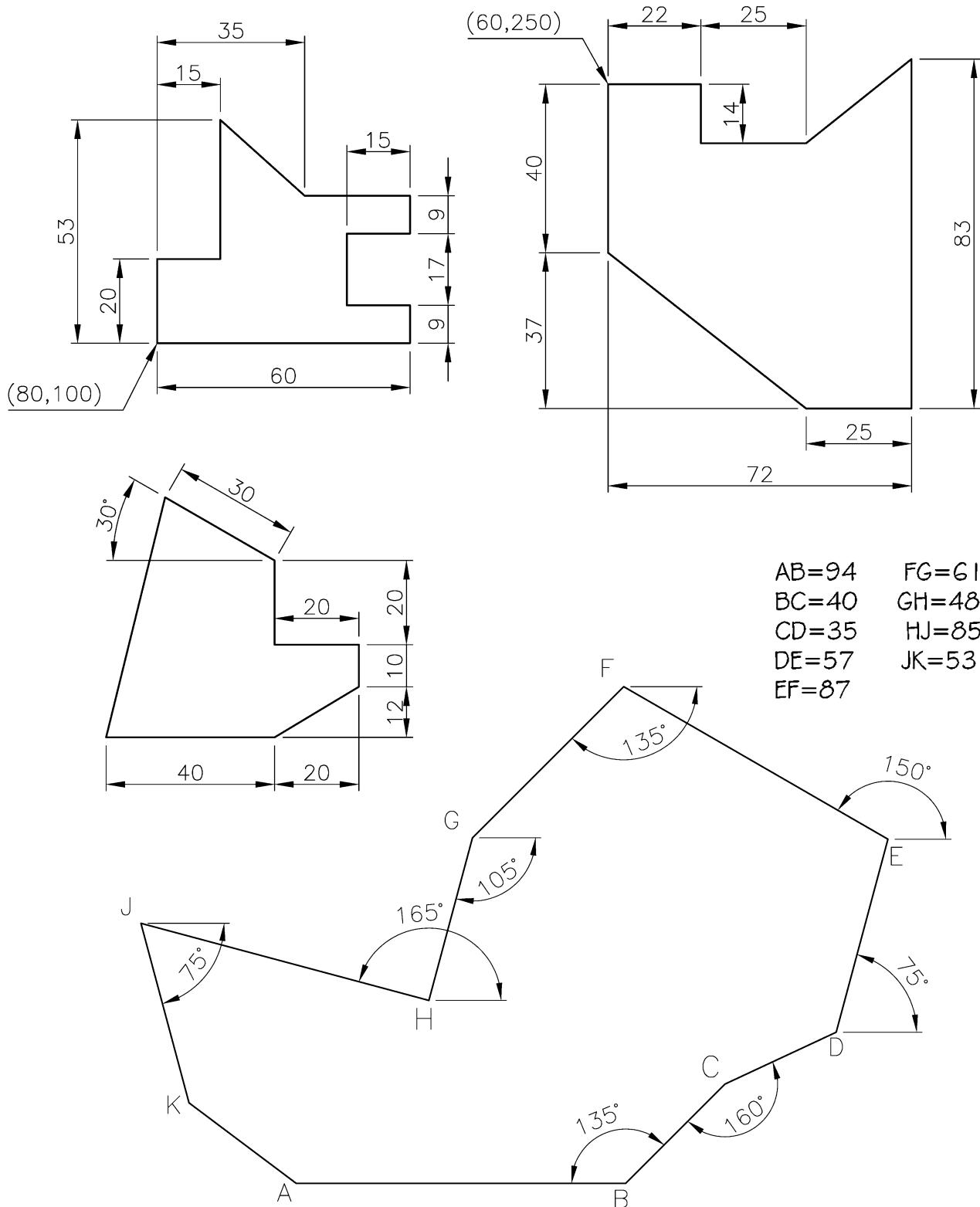


Figure 2.3

Tutorial 2

(Drawing files are to be setup with appropriate layers per para 2.23)

1. Construct the drawings as shown below using various entry methods for coordinates and distance. You may use starting coordinate for each of the drawing is indicated in the ().



2. Construct the following component figures using appropriate AutoCAD commands. All outlines and centre lines must be placed on the appropriate layers.

Note: Abbreviation used engineering drawing:

\emptyset (preceding a dimension) - Diameter
 R (preceding a dimension) - Radius

AF - Across Flats
AC - Across Corners

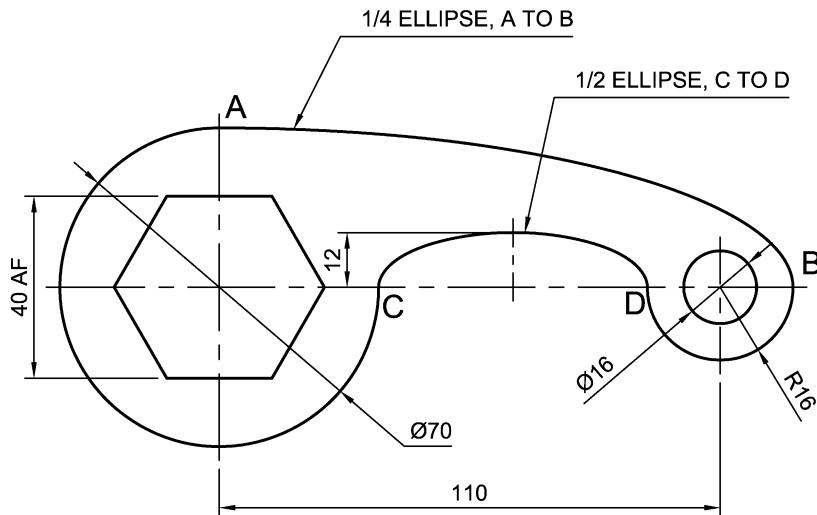


Figure 2-Q2a

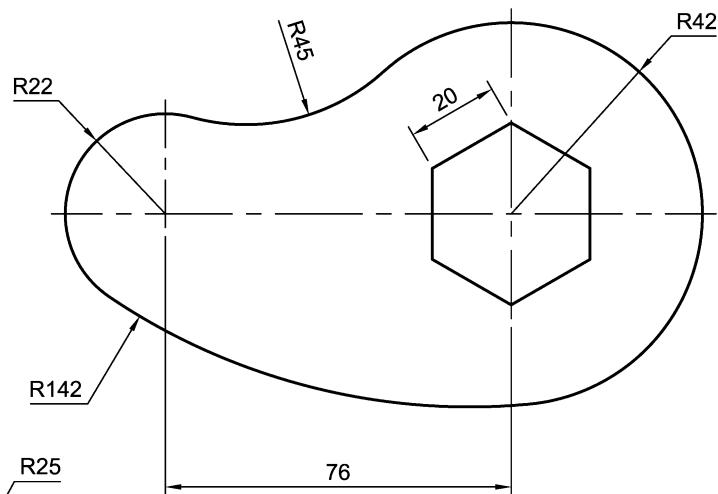


Figure 2-Q2b

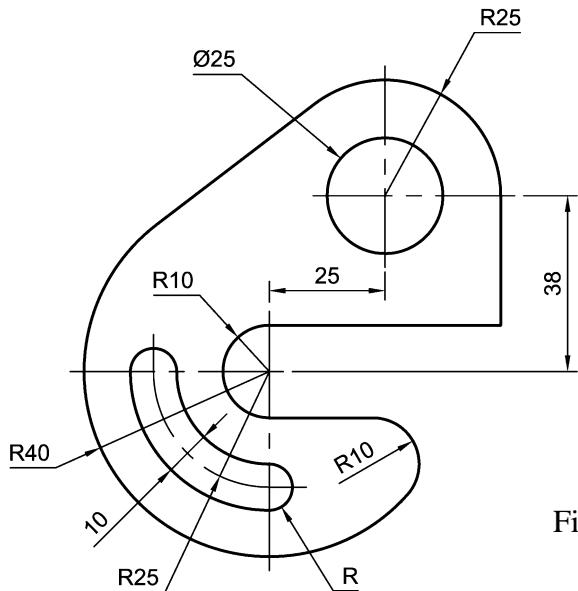


Figure 2-Q2c

3. Construct the following component figures using appropriate AutoCAD commands. All outlines and centre lines must be placed on the appropriate layers.

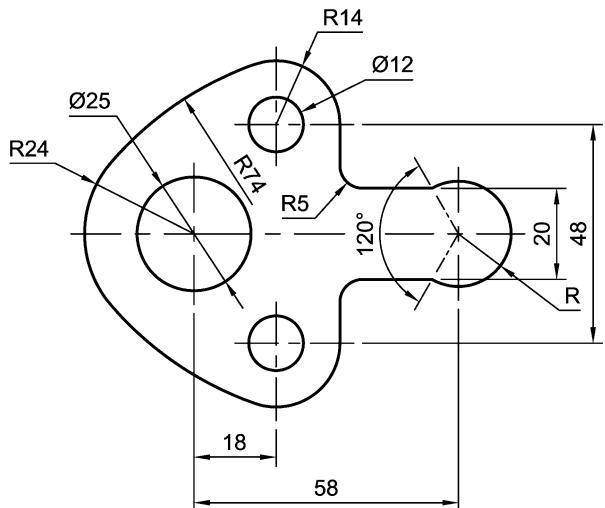


Figure 2-Q3a

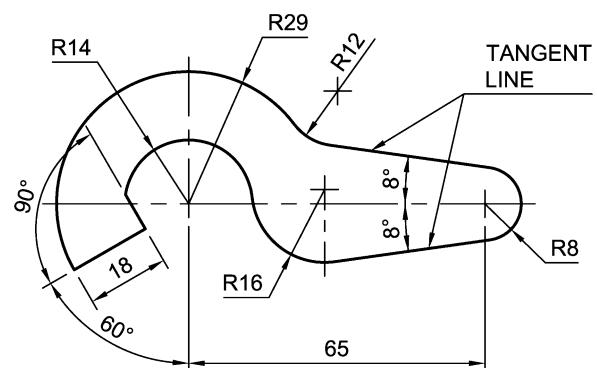


Figure 2-Q3b

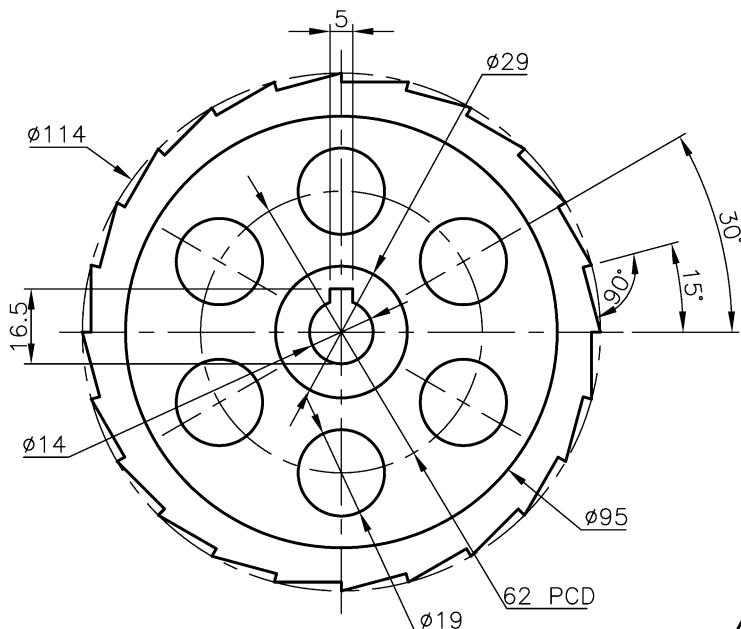


Figure 2-Q3c

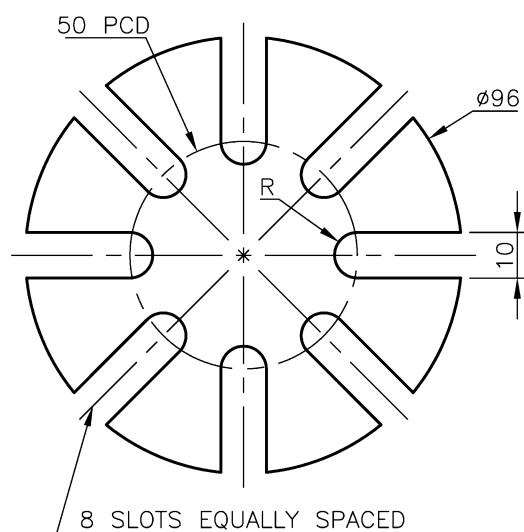


Figure 2-Q3d

4. Construct the following component figures using appropriate AutoCAD commands. All outlines and centre lines must be placed on the appropriate layers.

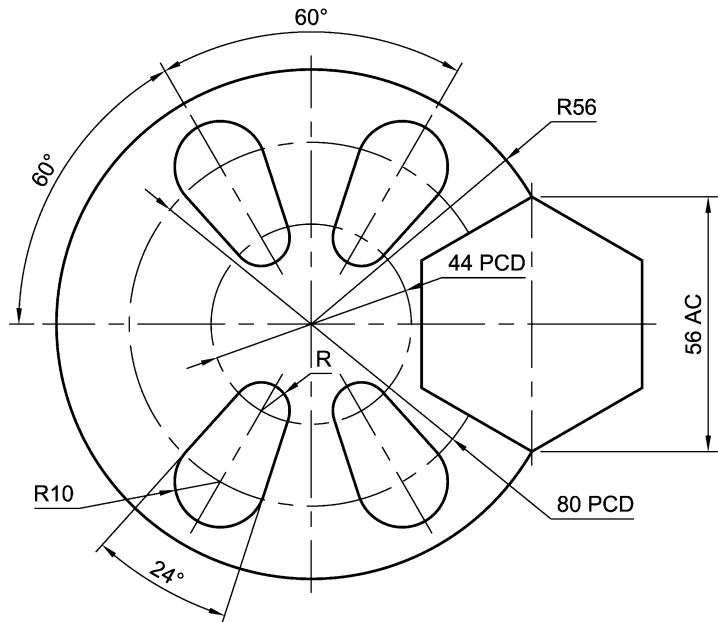


Figure 2-Q4a

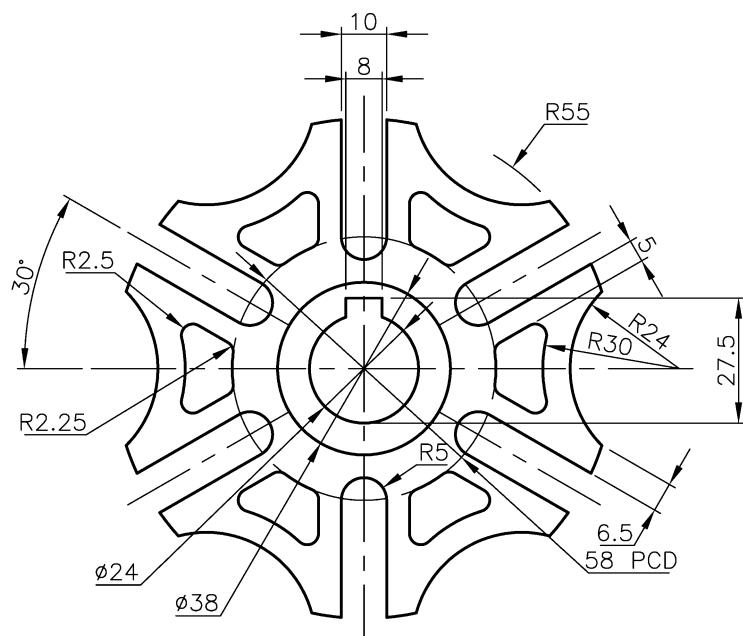


Figure 2-Q4b

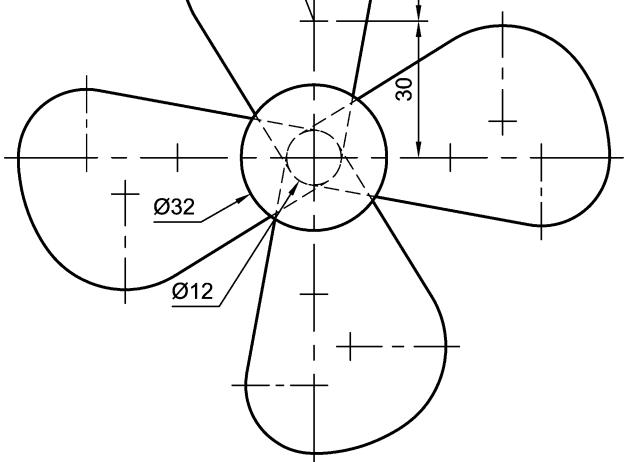


Figure 2-Q4c

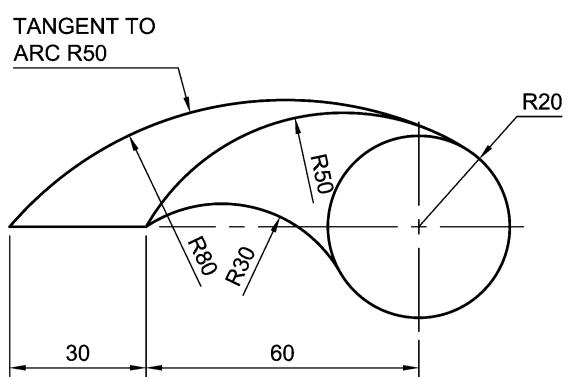


Figure 2-Q4d

UNIT 3 ORTHOGRAPHIC PROJECTION

Learning Objectives

By the end of this unit, students should be able to:

- Set up a drawing template for the subsequent new engineering drawing creation.
- Prepare orthographic drawing of engineering component using a range of AutoCAD commands.
- Communicate geometric information of engineering component through its orthographic drawing.
- Utilise the 45° mitre line.
- Draw various types of holes.

3.1 Creating a Drawing Template

STEP 1: Select  (New) from the quick access toolbar. Select “acadiso.dwt” as the drawing template and click Open.

STEP 2: Set the drawing limits. The drawing limits are:

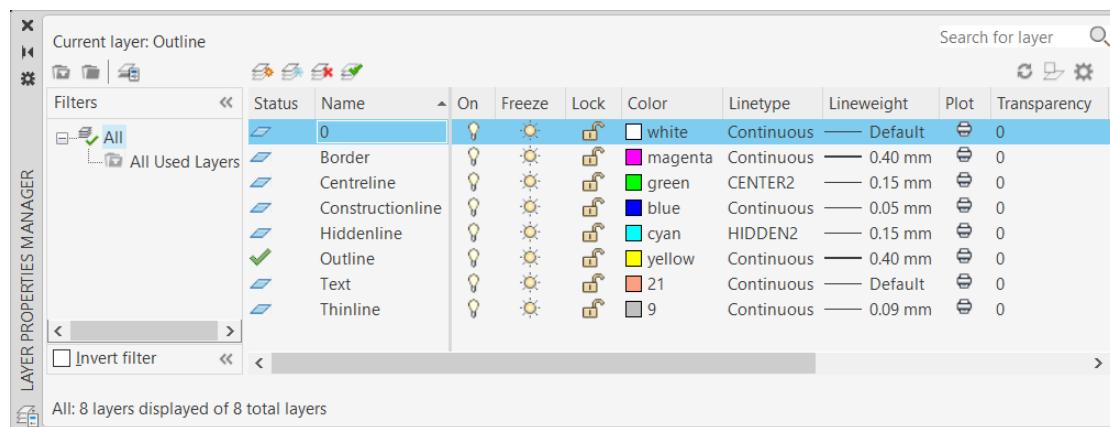
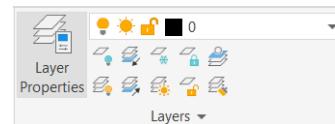
A4 size: 297 x 210. A3 size: 420 x 297. A2 size: 594 x 420.

Command: limits

Specify lower left corner or [ON/OFF] <0.0000,0.0000>: ↵

Specify upper right corner <420.0000,297.0000>: 420, 297 (for A3 size)

STEP 3: Setup the layers and the corresponding types of lines, colours and linewidth. Close the manager once the layers setting are done.

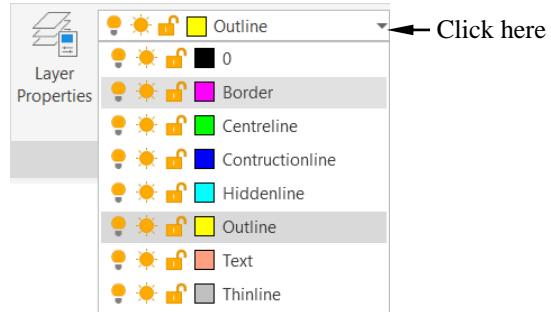


STEP 4: Click the down arrow within the layer control and select the Border layer to set it as current layer.

STEP 5: Draw a border of 10 mm, from the edges of the previously selected drawing limits, on the Border layer.

In the text layer, enter Name, Class and Admission Number at the bottom of the right hand corner with suitable text height (4 mm).

STEP 6: Save the drawing as *mytemplate.dwt*. This template will help to save time as user does not need to create layers again when starting a new drawing.



3.2 Dynamic Input

Dynamic Input is a command interface provided near to the cursor. When it is turn on (by left clicking the  on the status bar), it will display pointer coordinates, dimension input or dynamic prompts in a tooltip near to the cursor during drawing:

- **Pointer Input -**

When turn on, the coordinates of the crosshairs' location are displayed in the tooltip which can be edited. The default format for second and subsequent points is relative polar coordinates (except for *RECTANGLE* command which is relative cartesian). Thus, it is not required to enter the "@" sign for relative coordinates. For absolute coordinates, the entry is prefixed with the "#" sign. For example, entering "#10,100" for the second point prompt means that the absolute X and Y coordinate is 10 and 100 respectively.

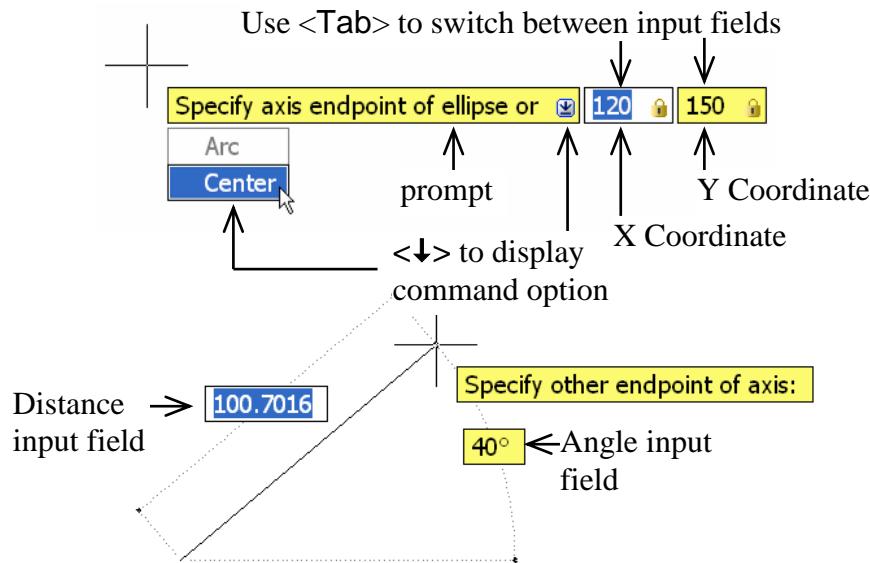
- **Dimension Input -**

When turn on, the distances and angle values are displayed in the tooltip when a command prompts for a seond point. This is available for *ARC*, *CIRCLE*, *ELLIPSE*, *LINE* and *PLINE* command.

- **Dynamic Prompts -**

When turn on, the commands and prompts are displayed in the tooltip. The input can be entered in the tooltip. The command options can be viewed and selected by pressing the DOWN ARROW (<↓>)key.

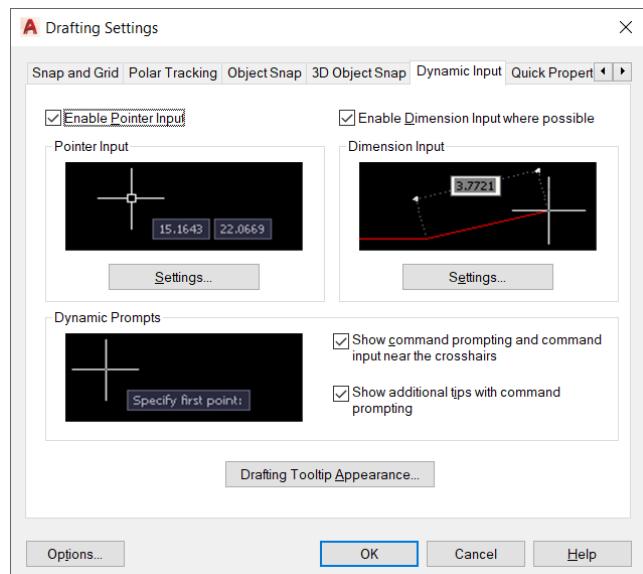
Dynamic Input as appeared on the drawing screen:



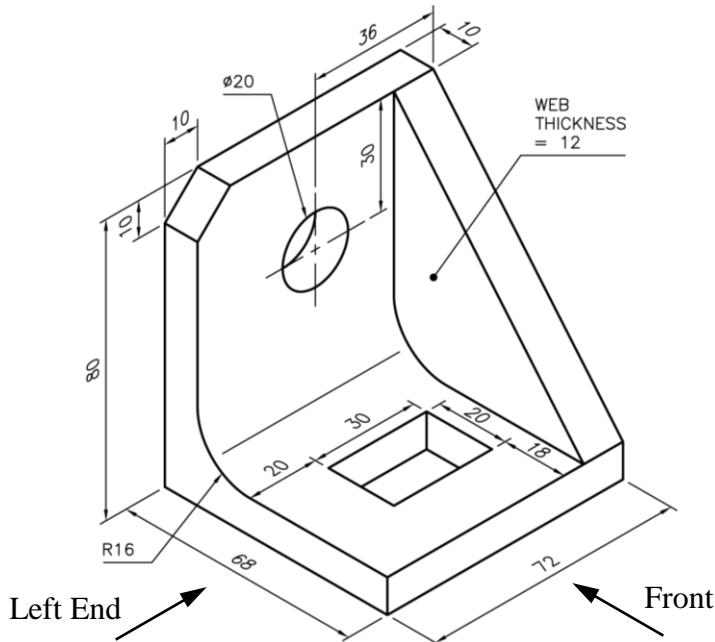
The settings of dynamic input may be changed via the Dynamic Input tab of the drafting settings.

To access:

Right click at the  (Dynamic Input Button) on the status bar, select Dynamic Input Settings...



3.3 Example of Third-Angle and First-Angle Orthographic Projection

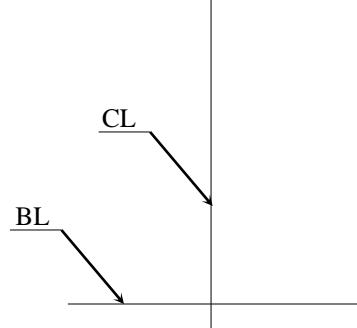


The steps in creating a three-view **third angle** orthographic drawing of the cast steel bracket are:

STEP 1

Draw the front view. Draw the centreline and base line.

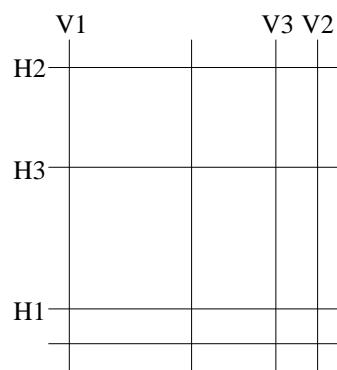
- Using Construction Line, draw a vertical centreline (CL) and a horizontal baseline (BL).



STEP 2

Draw vertical and horizontal guide lines.

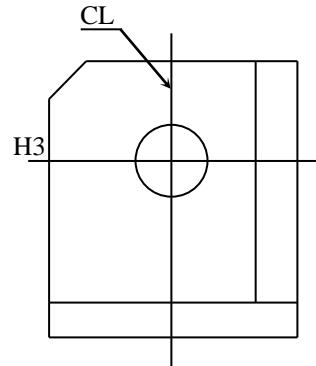
- Offset 36 to the left and right of CL to get V1 and V2 respectively.
- Offset 12 from V2 to get V3.
- Offset 10 & 80 above BL to get H1 & H2 resp.
- Offset 30 below H1 to get H3.



STEP 3

Draw circle and chamfer.

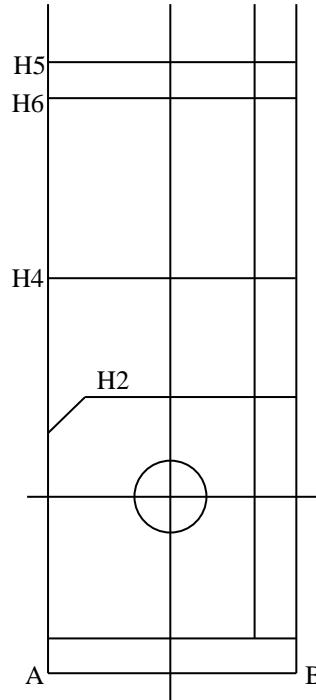
- Add circle, Ø20 with centre at the intersection of CL and H3.
- Chamfer the top left corner.



STEP 4

Project the plan. Project vertical guide lines.

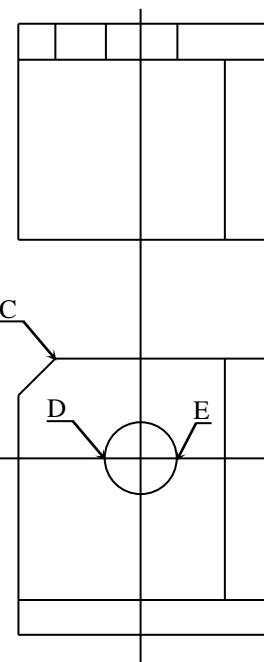
- Using Construction lines, draw vertical lines through points 'A' and 'B' on the front view.
- Offset, say, 50 above H2 to get H4.
- Offset 68 above H4 to get H5.
- Offset 10 below H5 to get H6.



STEP 5

Project guide lines for hole and chamfer to the Plan.

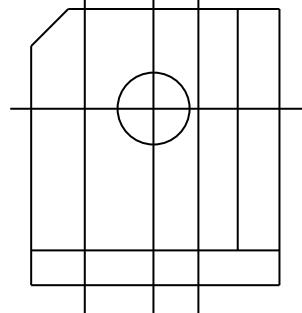
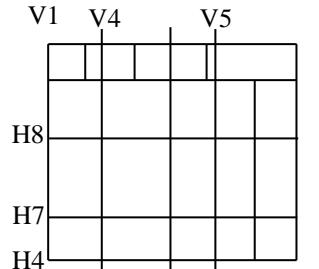
- Using Construction lines, draw vertical lines through points 'C', 'D' and 'E'.
- Trim to required shape.



STEP 6

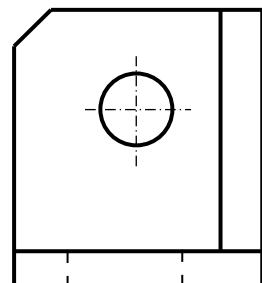
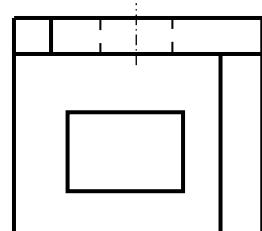
Create the slot in the plan.

- Offset 18 above H4 to get H7.
- Offset 20 above H7 to get H8.
- Offset 20 to the right of V1 to get V4.
- Offset 30 to the right of V4 to get V5.

**STEP 7**

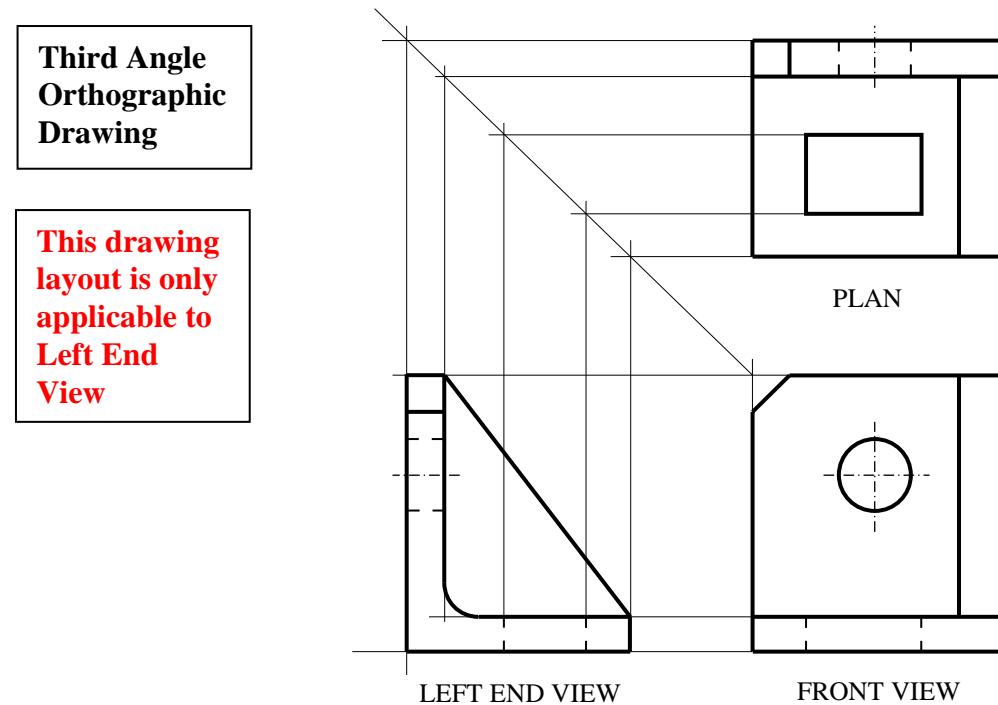
Change linetype for outlines, centrelines and hiddenlines.

- Trim to required shape.
- Edit centrelines as shown using hot grips with Ortho in the active mode.
- Note that centrelines to be 2 to 3 mm outside the concern outlines.
- Change all construction lines into the required linetypes as shown by transferring them to the respective layers.

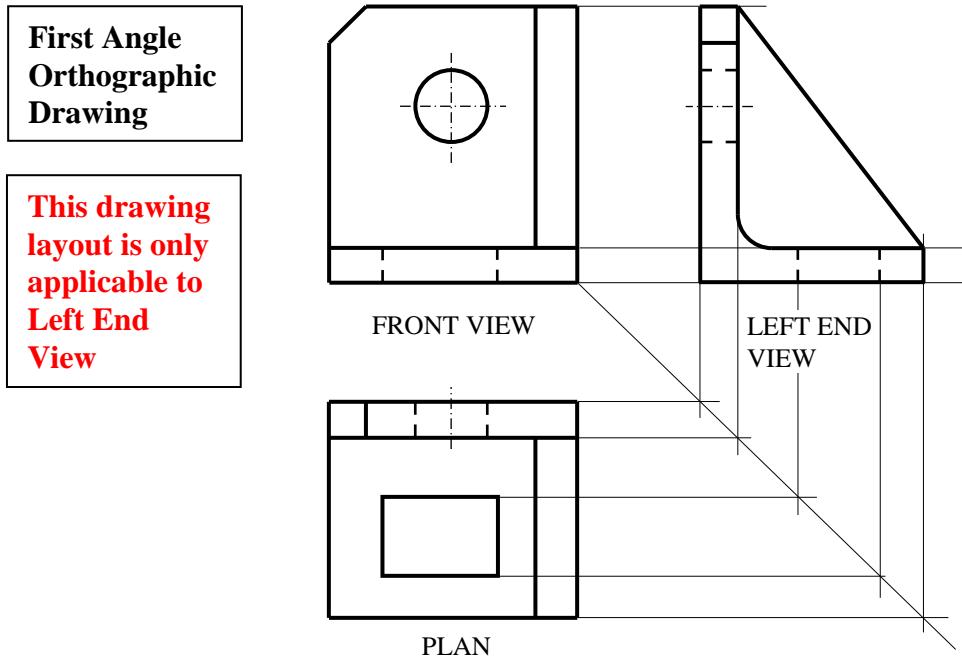


STEP 8

Project the end view. Proceed the same way for the end view.



The steps in creating a three-views **first angle** orthographic drawing of the cast steel bracket are similar to the above except that the views are placed differently as shown.



Tutorial 3

1. For the given component figures (3-Q1a to 3-Q1d), draw the following views in **THIRD-ANGLE** projection, using scale 1:1.

- A front view from A
- An end view from B
- A plan view

Do not dimension the drawings.

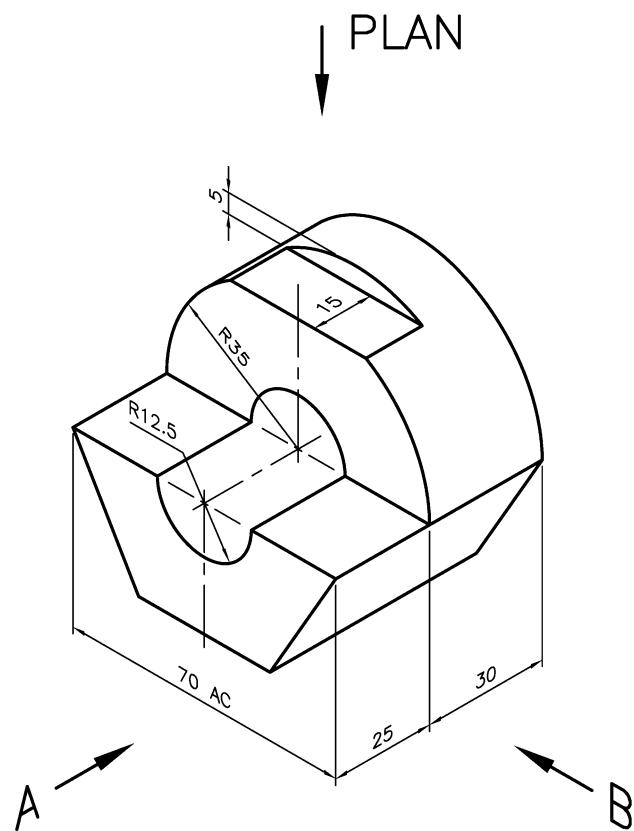


Figure 3-Q1a

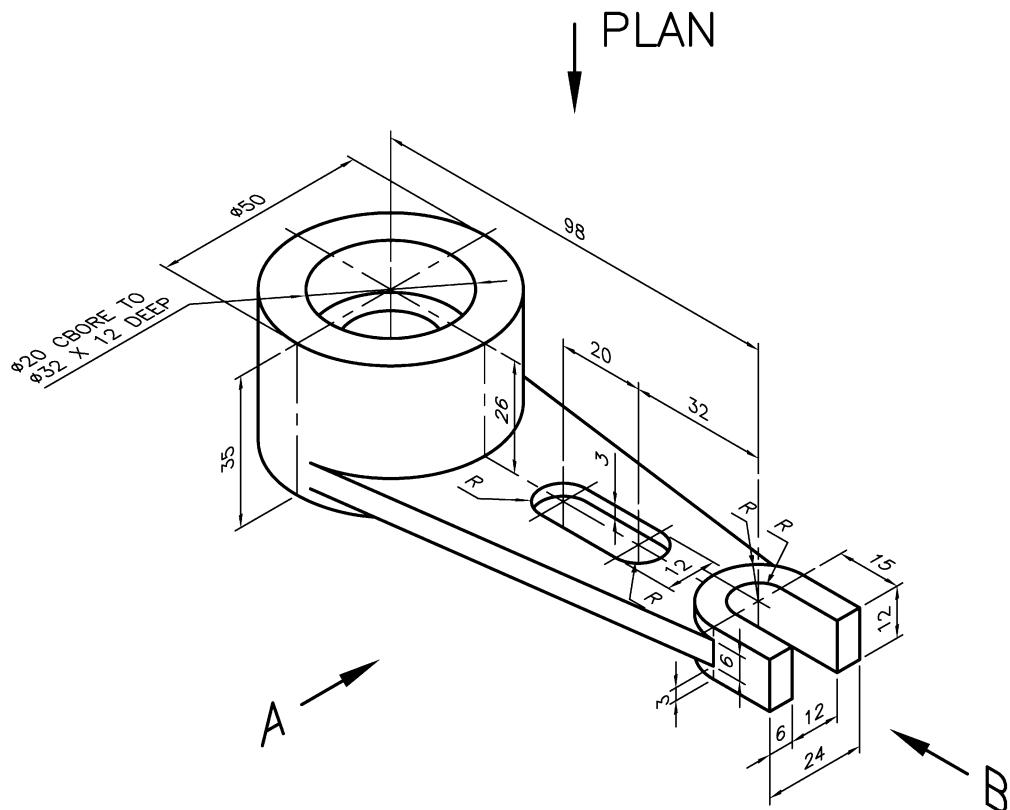


Figure 3-Q1b

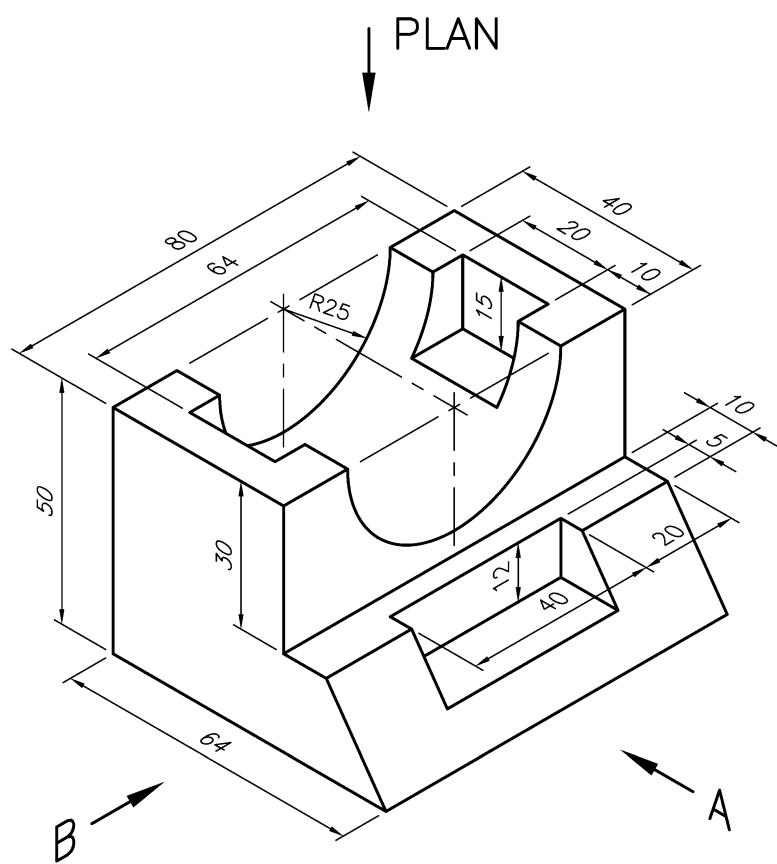


Figure 3-Q1c

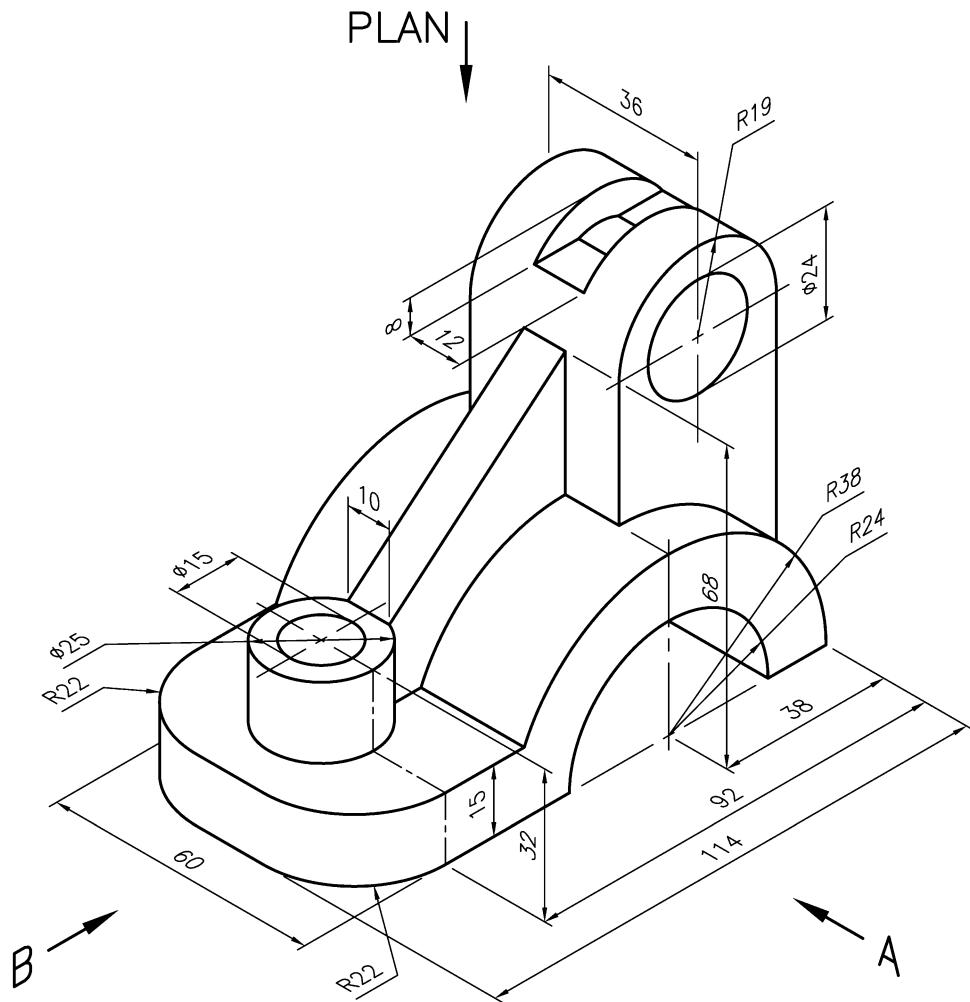


Figure 3-Q1d

2. For the given component figures (3-Q2a to 3-Q2d), draw the following views in **FIRST-ANGLE** projection, using scale 1:1.

- A front view from A
- An end view from B
- A plan view

Do not dimension the drawings.

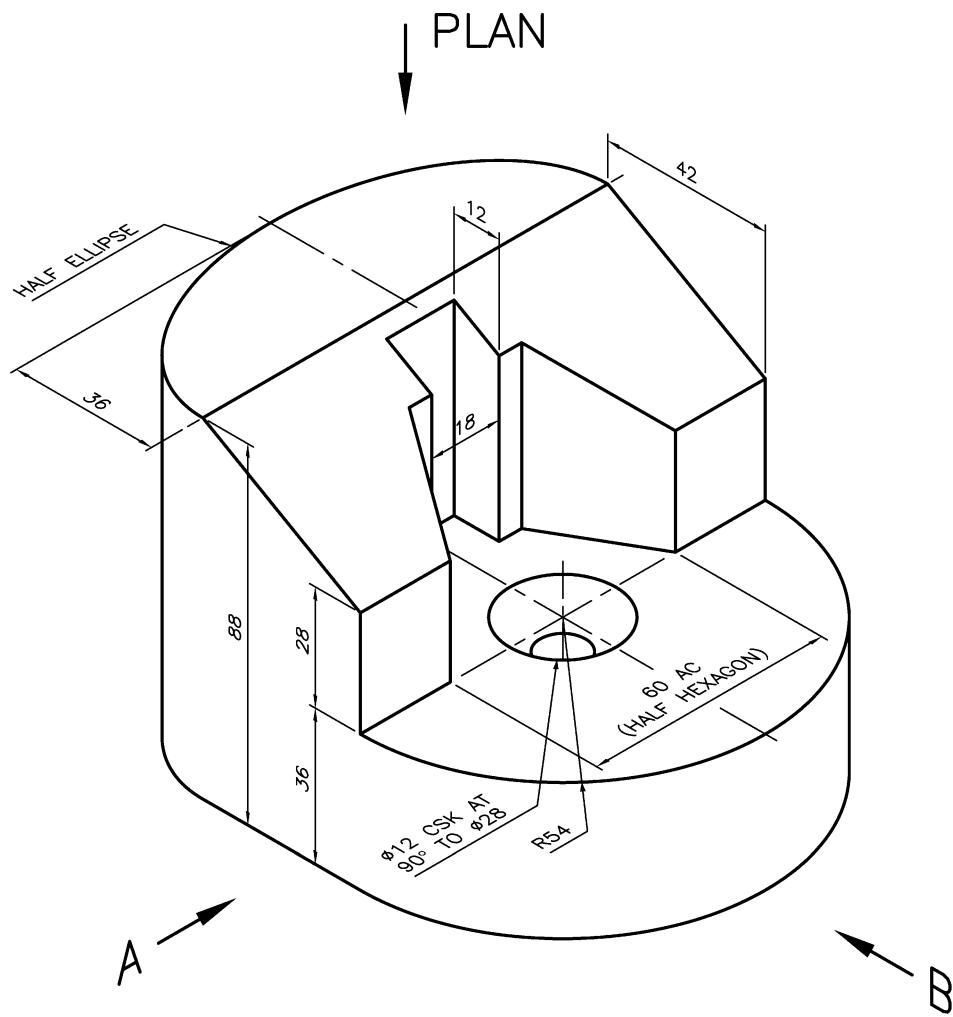


Figure 3-Q2a

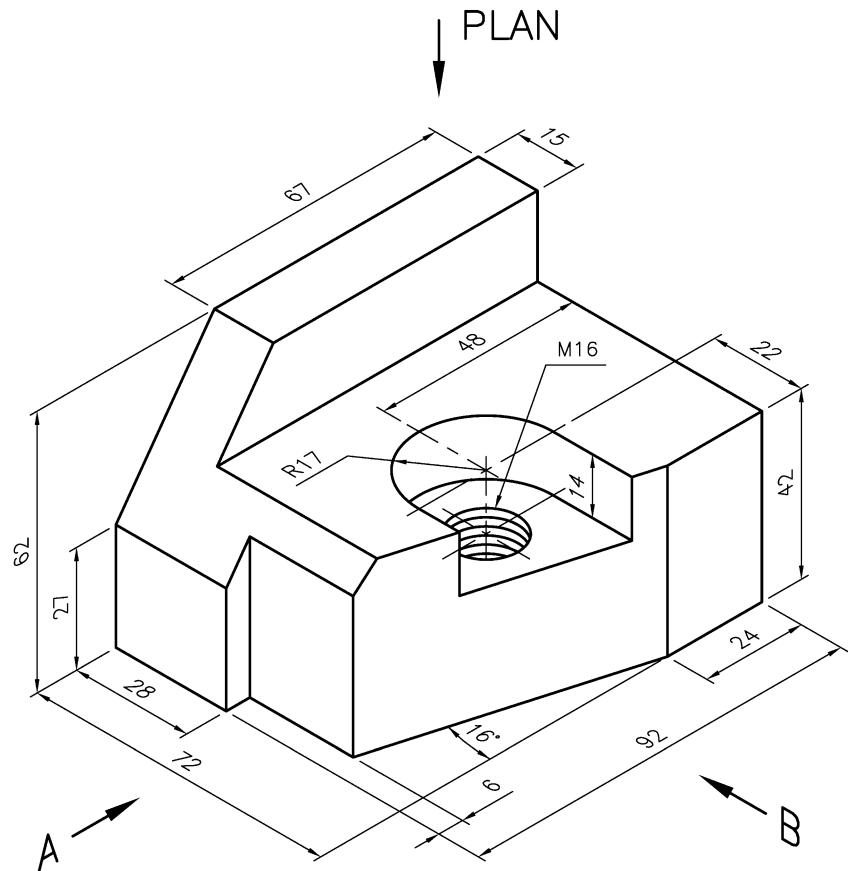


Figure 3-Q2b

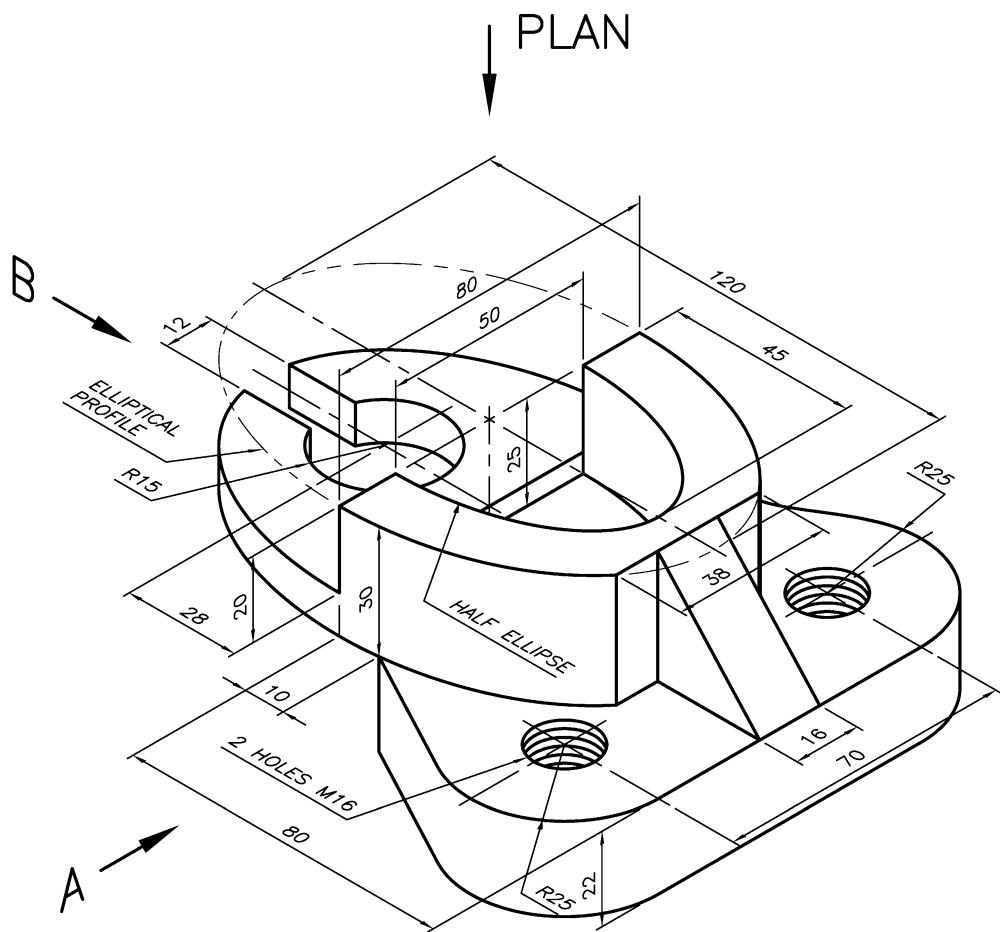


Figure 3-Q2c

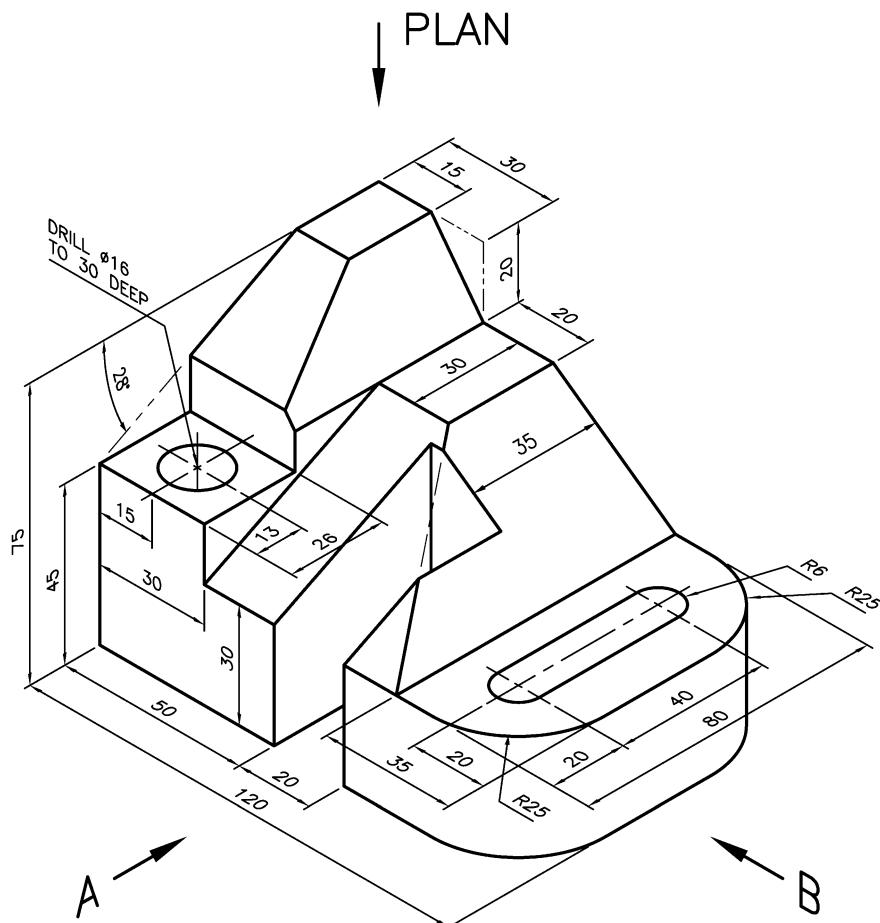
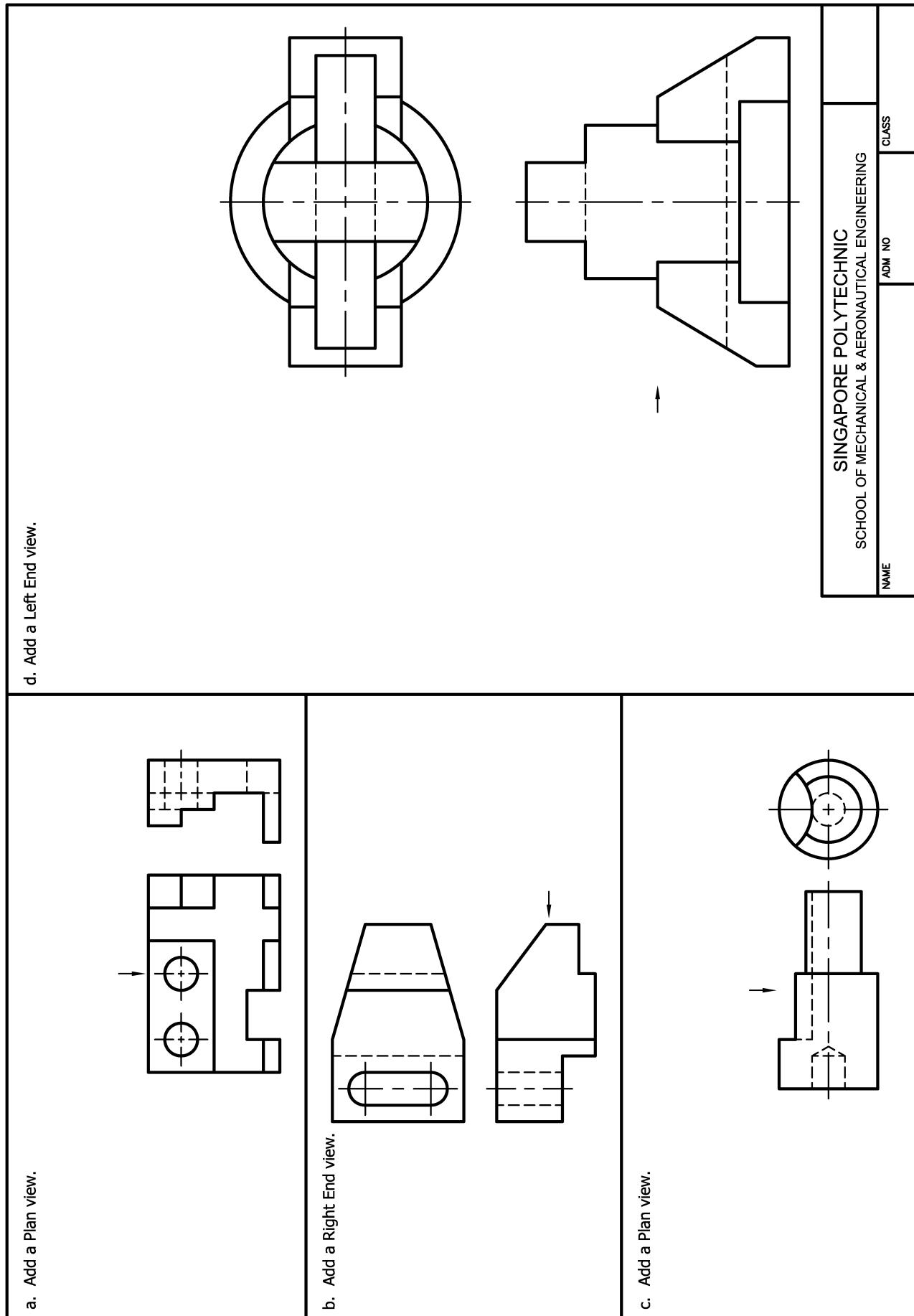
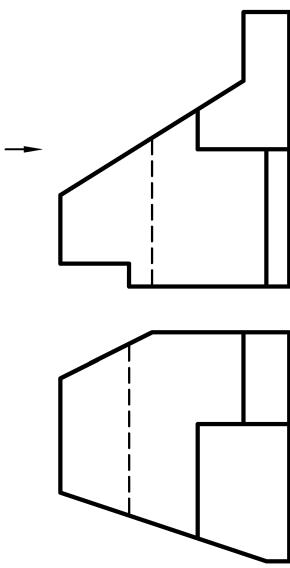


Figure 3-Q2d

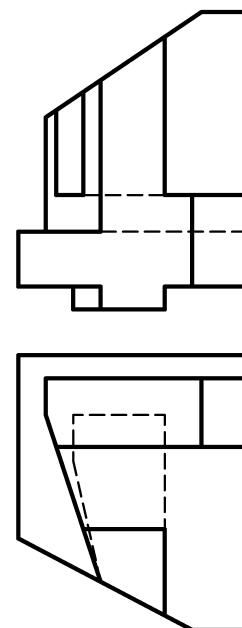
3. Refer to the given drawings next page, study the details on the two given views and add in the required view. Start with a New drawing file (Quick Access > New icon), select the template file from P:\MAE\ME1201\3 Orthographic Projection\tut3Q3.dwt.



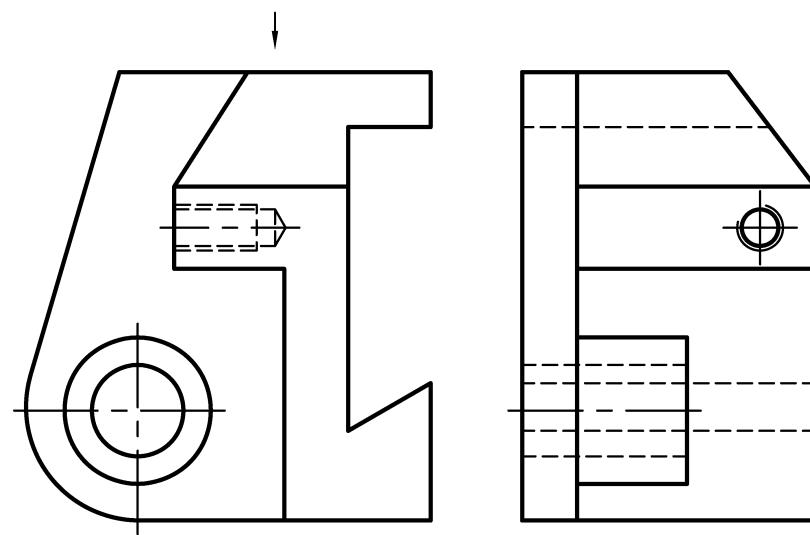
a. Add a Plan view.



b. Add a Plan view.



c. Add a Right End view.



SINGAPORE POLYTECHNIC	
SCHOOL OF MECHANICAL & AERONAUTICAL ENGINEERING	
NAME	ADM NO
CLASS	

BLANK

UNIT 4 BLUE PRINT READING

Learning Objectives

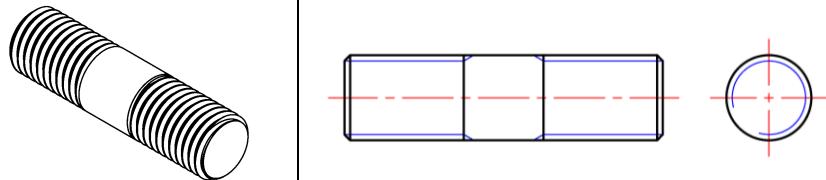
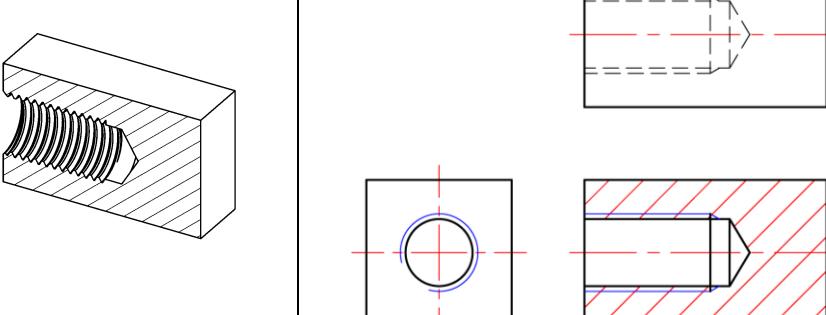
By the end of this unit, students should be able to:

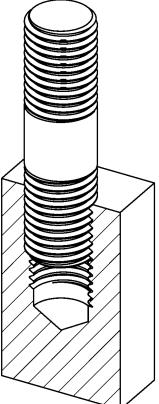
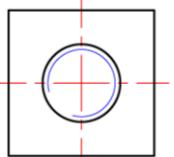
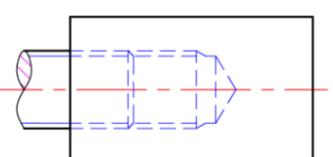
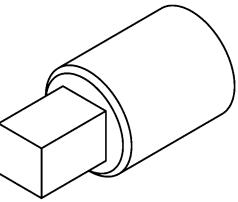
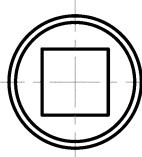
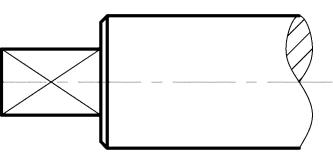
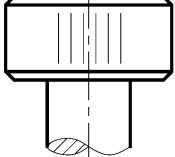
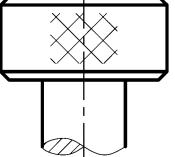
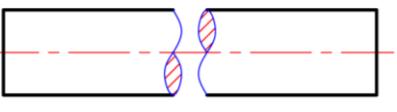
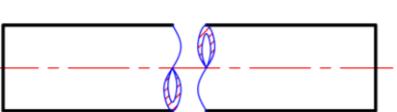
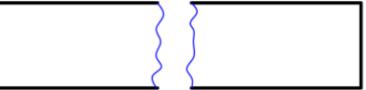
- Identify common engineering features and components through their conventional representations.
- Describe engineering components and features with appropriate technical terms.
- Differentiate between threaded fasteners and non-threaded locking devices.

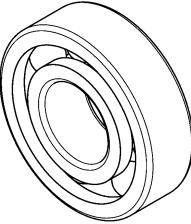
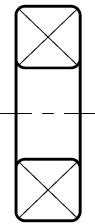
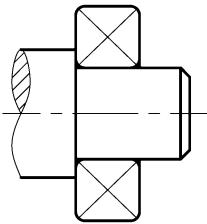
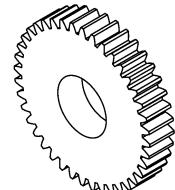
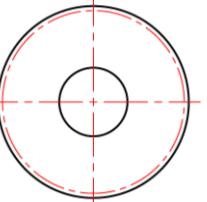
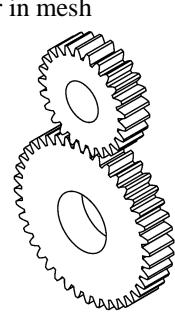
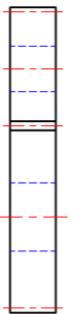
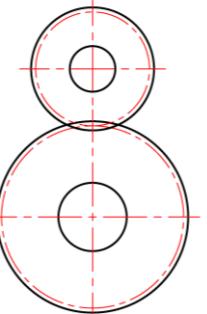
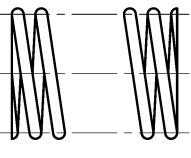
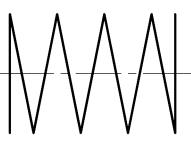
4.1 Conventional Representation of Common Features

There are many common engineering features in which it is difficult and tedious to draw in full. In order to save drafting time and space on drawings, these features are presented in a simple conventional form as shown in Table 4.1.

Table 4.1. Conventional Representation of Common Features

Engineering Feature	Convention
External Screw Thread	
Internal Screw Thread	

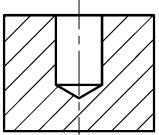
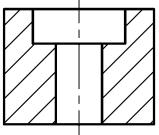
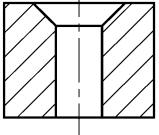
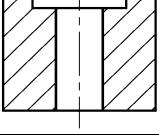
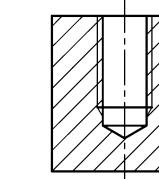
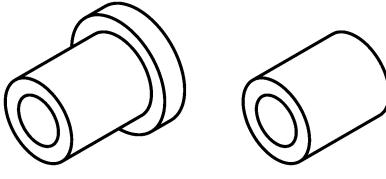
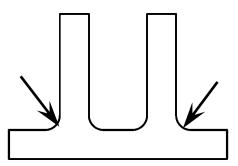
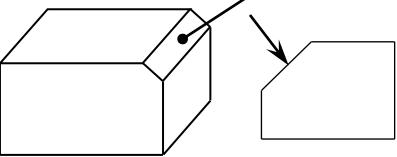
Engineering Feature	Convention
Screw Thread Assembly	  
Square on Shaft	  
Knurling	   <p style="text-align: center;">Straight Knurling Diamond Knurling</p>
Break in Long Section	Solid Circular Section  
	Hollow Circular Section  
	Other Section  

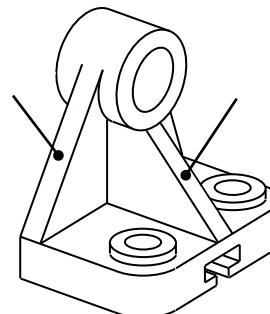
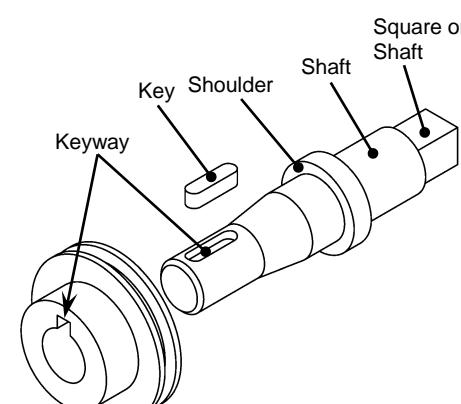
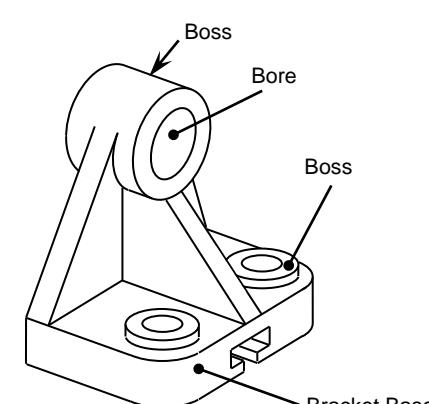
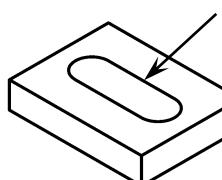
Engineering Feature	Convention
Bearing	   Bearing (when sectioned) Bearing mounted on shaft
Spur Gear	  
	  
Cylindrical Compression Spring	   Conventional Simplified

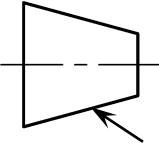
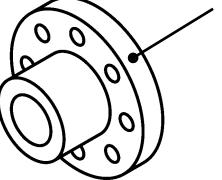
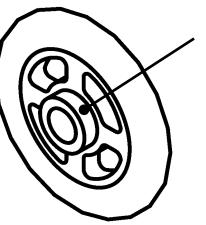
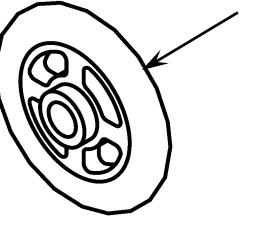
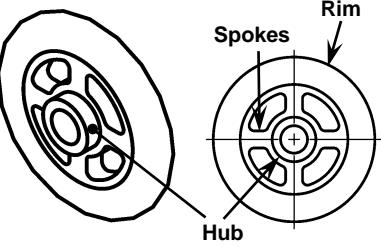
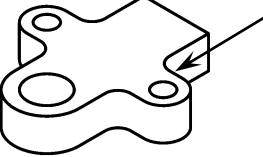
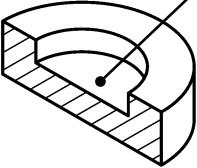
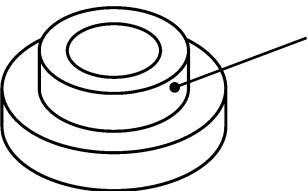
4.2 Technical Terms

Engineers and technicians need to know and understand the technical terms used to describe components and their features so as to be precise and concise in the communications of engineering information. A list of the technical terms are given in Table 4.2.

Table 4.2. Technical Terms

Technical Terms	Descriptions	Sketches (fill in missing items)
Blind-drilled hole	A hole which does not pass through a component.	
Counterbore hole	A hole, part of which is a larger diameter and flat bottom to conceal screw heads.	
Countersunk hole	A hole, part of which is conical to receive screw heads.	
Spot-faced surface	A flat circular surface concentric with a hole, used for seating screw heads etc.	
Blind Threaded hole	A blind-drilled hole which part of its depth is threaded for the purpose of receiving an external screw threads feature.	
Bush	A plain bearing supporting or guiding a shaft and can be easily replaced when worn out.	 Headed Bush Headless Bush
Fillet	Internal rounded corner of two lines or surfaces	
Chamfer	A surface produced by bevelling square edges.	

Rib or Web	A thin part used to support or strengthen heavier parts of a component.	
Shaft	A cylindrical rod upon which parts are fixed, used for transmission of motion.	
Shoulder	A sudden change in diameter of a shaft.	
Square on a shaft	A length of the shaft with a square cross section.	
Key	A piece of shaped metal which, is inserted in a shaft and a hub to prevent their relative movement.	
Keyway	A groove in a shaft or hub machined to accommodate a key.	
Bore	Cylindrical hole along a tube or a boss.	
Boss	Cylindrical protrusion to accommodate a hole.	
Bracket base	Bottom part of a projecting support, usually fixed to a flat surface.	
Slot	An elongated hole or groove.	

Taper	A gradual change in diameter of a component along its length.	
Flange	A projecting thin disc on pipes.	
Hub	The inner part of a wheel.	
Rim	The outer part of a wheel.	
Spokes	Rods radiating from the hub to the rim of a wheel.	
Lug	A projection used for fastening and adjusting purposes.	
Recess	A shallow hole to receive a spigot or a similar matching part.	
Spigot	A projection which fits into a corresponding recess and is used for precise-location purposes.	

4.3 Fasteners and Locking Devices

The most important types of fastenings for holding parts together in engineering are those with screw threads.

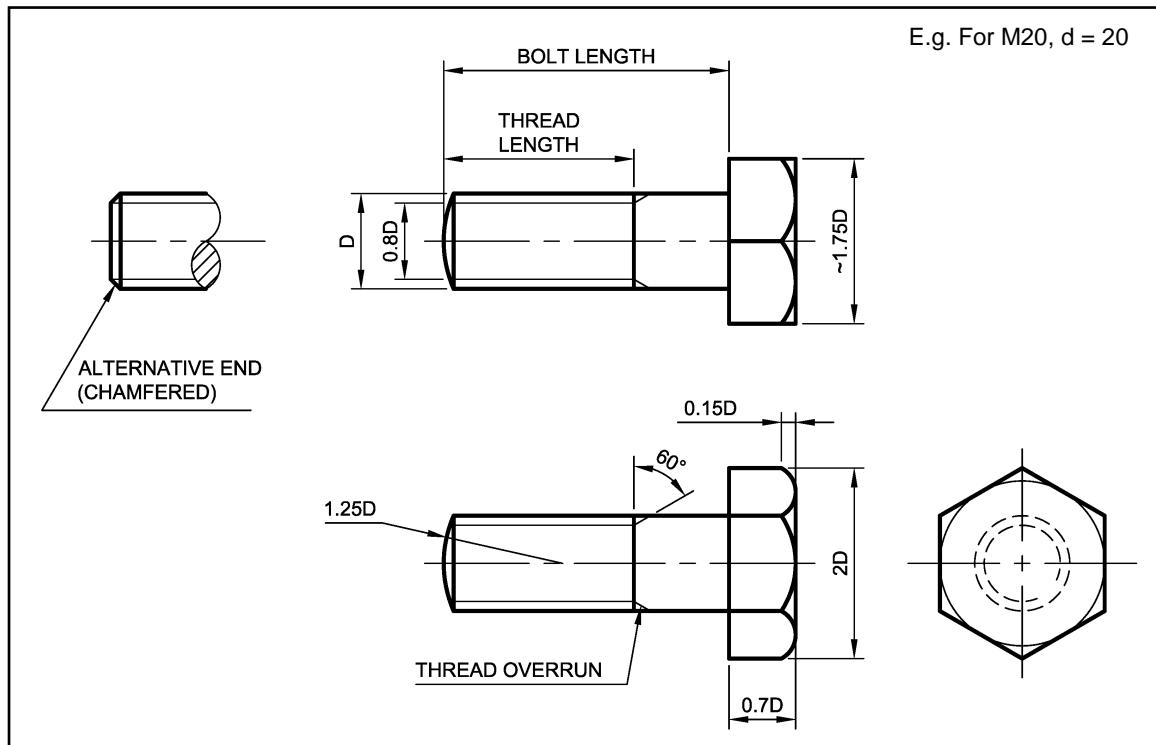
Screw threads are cut on the outside of shafts (e.g. screws) or on the inside of cylindrical holes (e.g. nuts) to form a series of helices.

4.3.1 Hexagonal Bolts and Nuts

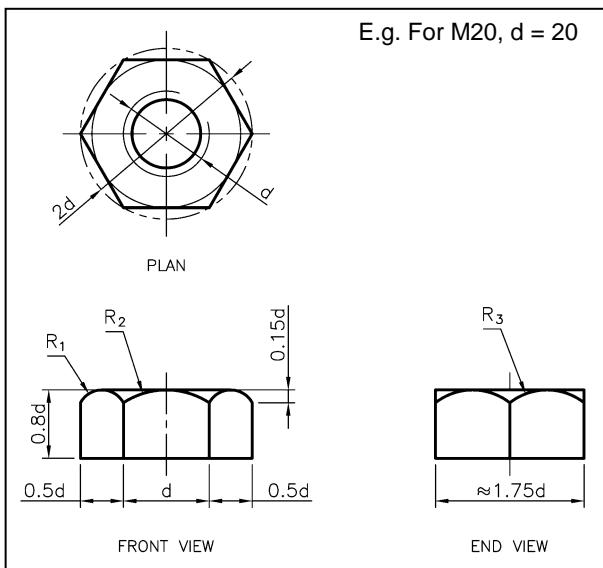
The most common form of nuts and bolts found in engineering applications are those with hexagonal shaped heads.

1. Hexagonal Bolt

A bolt having a hexagonal head is known as a hexagonal bolt. The size of the head has a definite relationship with the diameter of the bolt. Drawings below show the drawing proportions for hexagonal machine bolt.

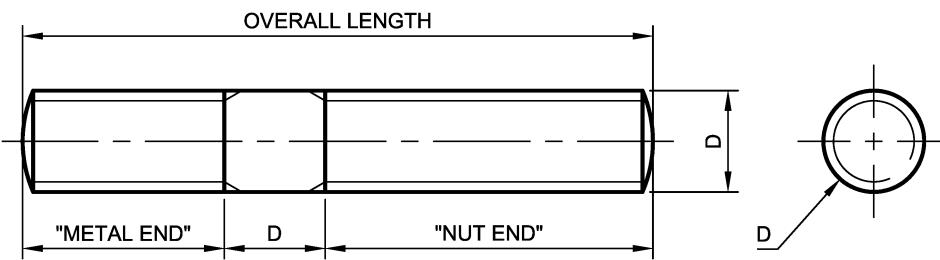


2. Hexagonal Nut



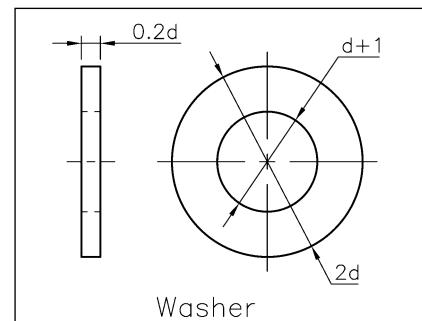
4.3.2 Stud

Studs are threaded on both ends, with an unthreaded shank in the middle, and are used for parts that must be removed frequently, like cylinder heads, covers, lids etc.



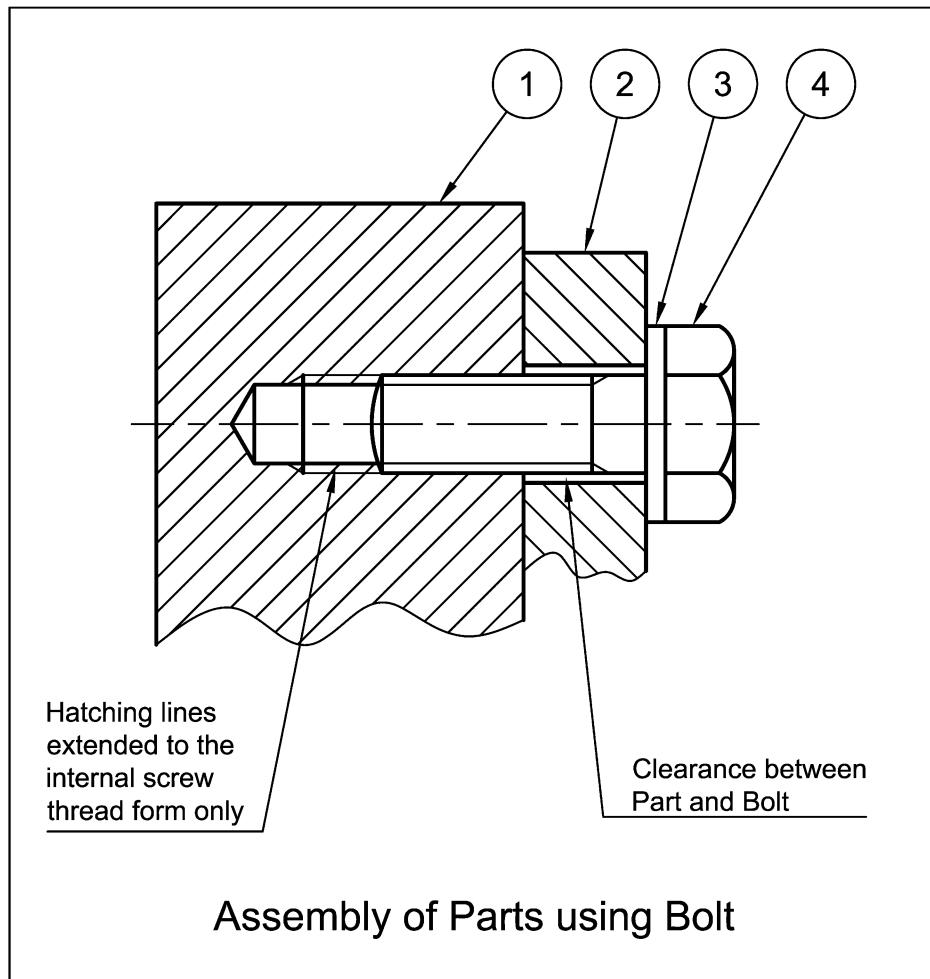
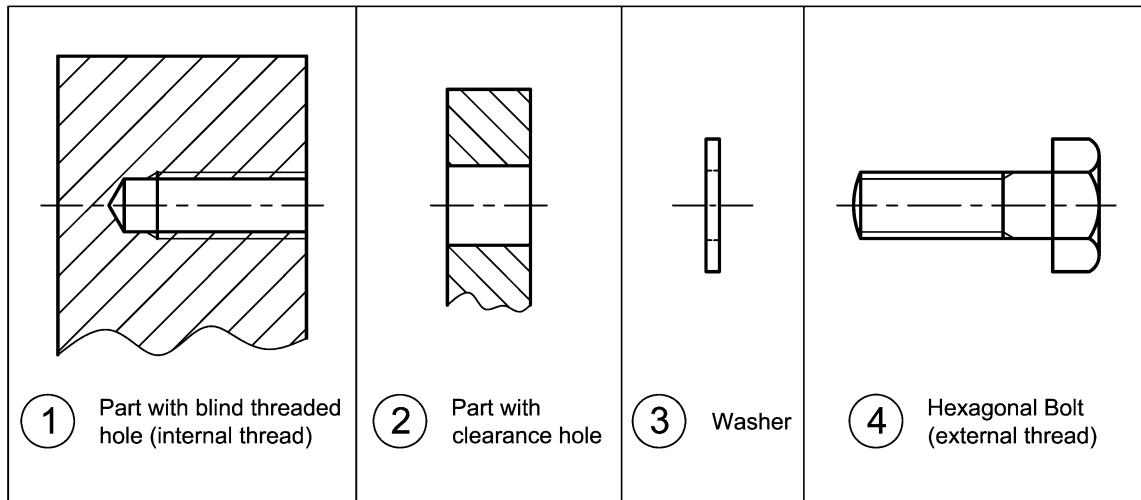
4.3.3 Plain Washer

Plain or Flat Washer is a flat ring that is placed below the nut or the head of a bolt to tighten the joint.

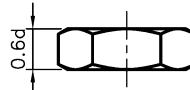
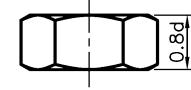
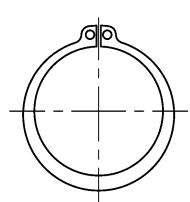
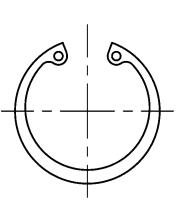
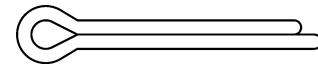
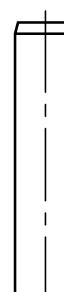
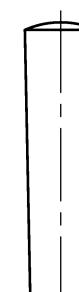


4.3.3 Assembly of Parts using Bolts

Drawings below show the assembly of two parts, one with a clearance hole and the other with a blind threaded hole, being assembled by the hexagonal bolt and washer.

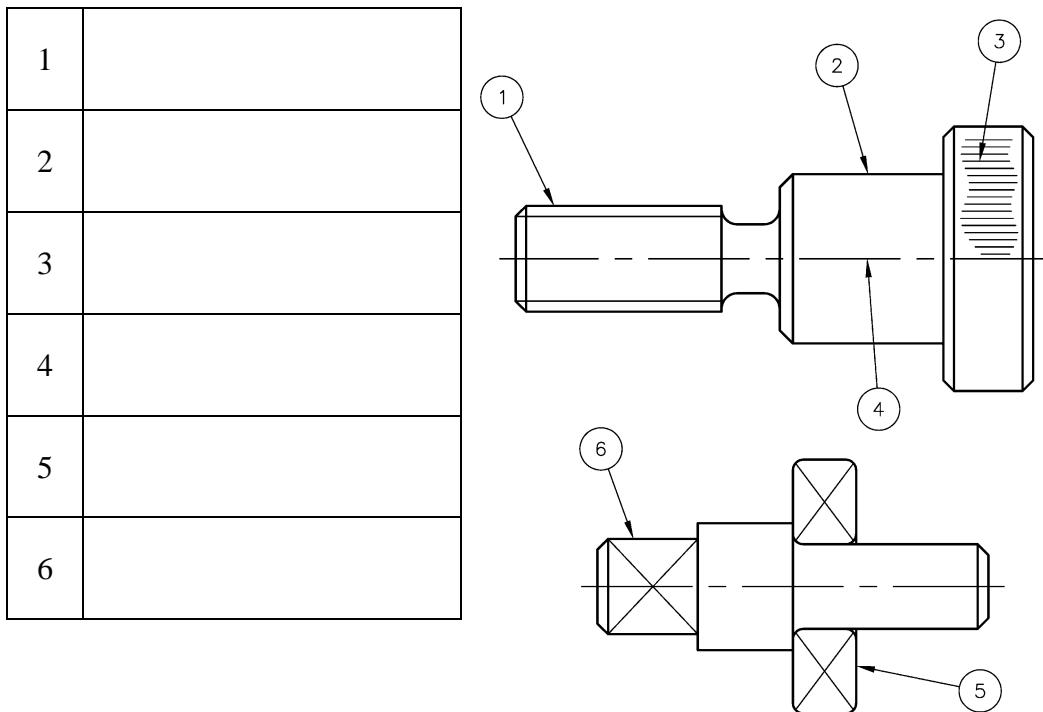


4.4 Other Fasteners and Locking Devices

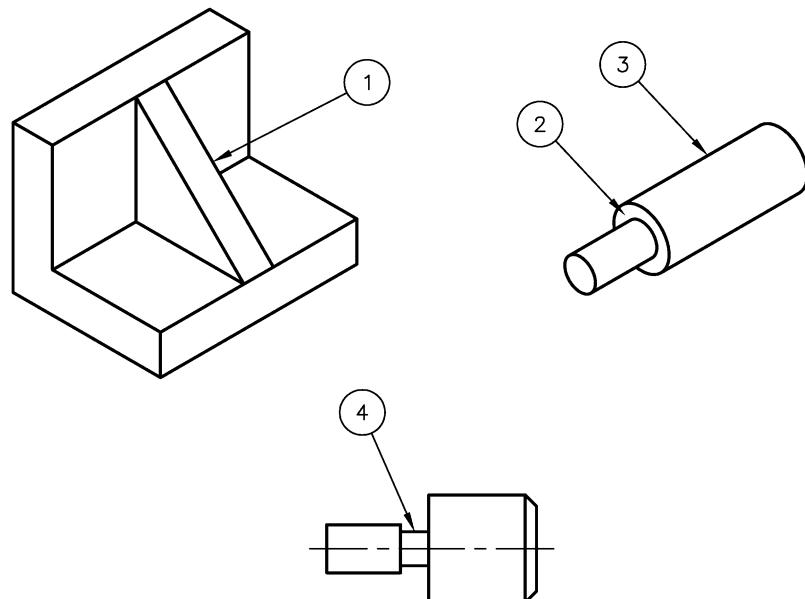
1	Lock Nut	This is a thin nut. It is used below a normal nut to act as a frictional locking device.	 Lock Nut	 Standard Hex Nut
2	Circlips	Circlips provide axial location between a shaft and its bearing. A locating groove is machined and the circlip is sprung into position. Circlip may be internal or external.	 External Circlip	 Internal Circlip
3	Split Pin	A split pin is a locking device when driven through the hole of a bolt or a shaft and its ends are split and open up.		
4	Pins	<p>Dowel Pin – A headless cylindrical pin used for precise-location purpose.</p> <p>Taper Pin – Conical with a slight taper. It is usually used to attach cotter, wheels, etc, to shafts.</p>	 Dowel pin	 Taper pin

Tutorial 4

1. Name the features indicated by the numbers as shown below.



2. Write down the names of the features/technical terms indicated by the numbers as shown below:



1	2	3	4

3. Identify the numerical value of the dimension letters given in figures Q3a and Q3b:

A _____
 B _____
 C _____
 D _____
 E _____
 F _____
 G _____

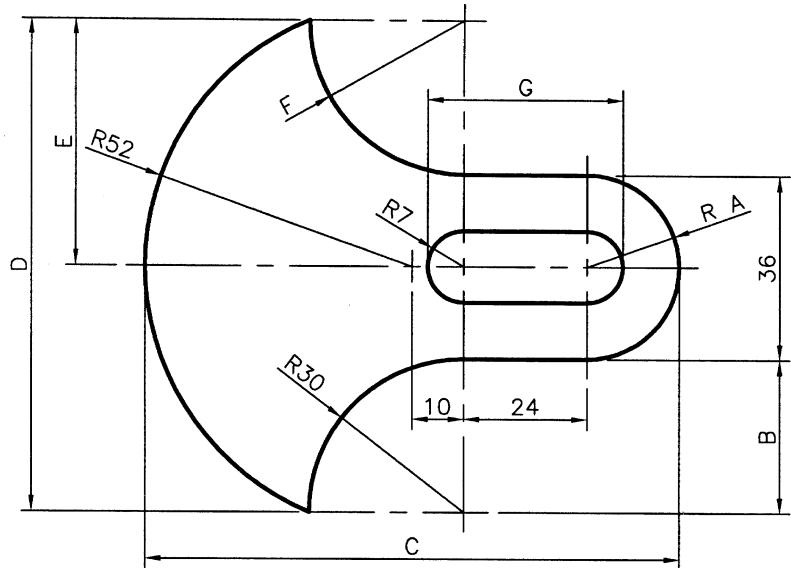


Figure Q3a

H _____
 J _____
 K _____
 L _____
 M _____
 N _____

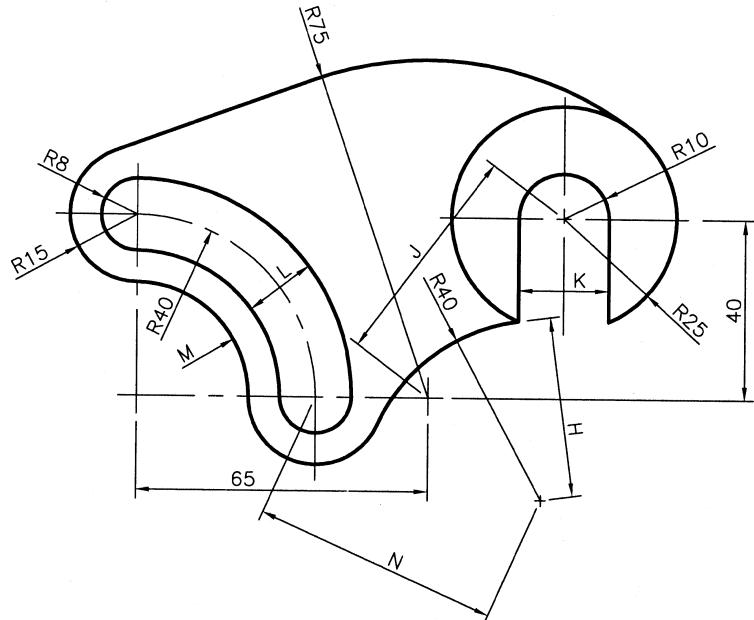


Figure Q3b

4. Figure Q4 shows the sectional view of a gearbox.

- i) Identify the parts and features associated with the following letters:

A _____
 B _____
 C _____

H _____
 I _____
 J _____

D _____

K _____

E _____

L _____

F _____

M _____

G _____

- ii) Identify the types of line and corresponding application associated with the following letters:

a _____

e _____

b _____

f _____

c _____

g _____

d _____

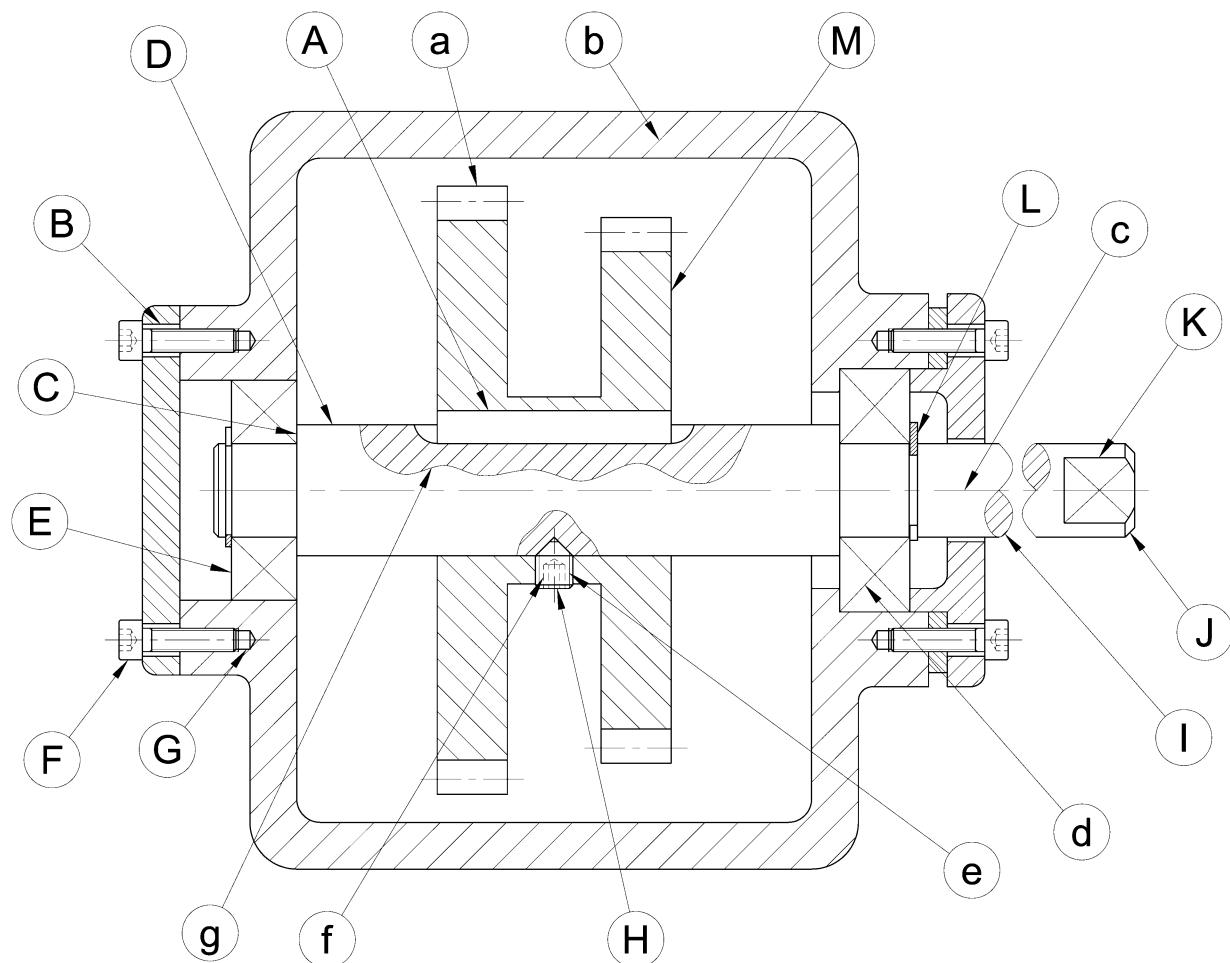


Figure Q4. Sectional View of a Gear Box

BLANK

UNIT 5 INTRODUCTION TO SECTIONING

Learning Objectives

By the end of this unit, students should be able to:

- Explain the principles of sectioning.
- Recognise the conventional practices of sectioning.
- Utilise the principle of sectioning and its conventional practices when preparing orthographic drawing containing sectional views.
- Identify the types of sectional views.
- Show sectioning lines on the cut areas of a sectional view with the HATCH command.

5.1 Principles of Sectioning

For a multi-views drawing obtained by orthographic projection, the internal features of a part are illustrated by thin short dashes lines. When there are a lot of internal features within a part, the drawing may be difficult to interpret as there will be too many thin short dashes lines present in the drawing views. Sectional view which is a cutaway view of the part is thus used to show the required internal features of such part so as to improve the clarity of the drawing.

The functions of sectional views are

1. to reveal the internal features of interest of a part in the drawing.
2. to reduce amount of hidden details.
3. to facilitate dimensioning of internal features.
4. to show the shape of cross section.
5. to show the relative positions of parts within an assembly drawing.

A sectional view is a view obtained by imaginarily removing a portion of the part nearest to the observer after being cut by a cutting plane as shown in Fig 5.1a and b. As a result, the outlines of the internal features become visible in the sectional view as shown in Fig 5.1c.

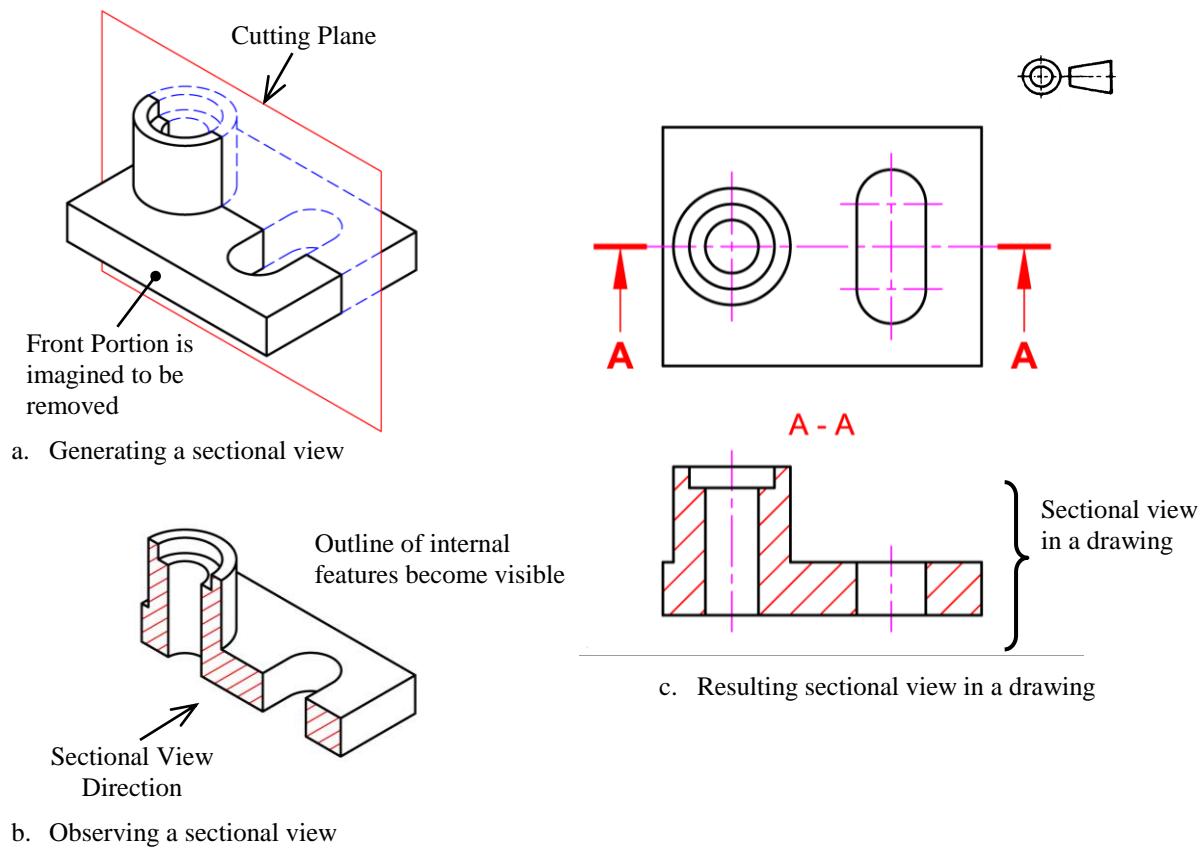


Figure 5.1. Principles of sectioning.

5.2 Conventional Practices of Sectioning

5.2.1 Cutting Plane Symbol

The cutting plane is denoted by a **thin chain line with thick ends** as shown in Fig 5.2. The cutting plane is drawn on one of the non-sectional views within the drawing. It is to indicate the location where the sectional view of the part is taken (see Fig 5.1c). In addition, arrow is placed at each of the thick ends of the cutting plane to indicate the direction of viewing for the sectional view. The cutting plane is then identified by letters placed near the tail of the arrow.

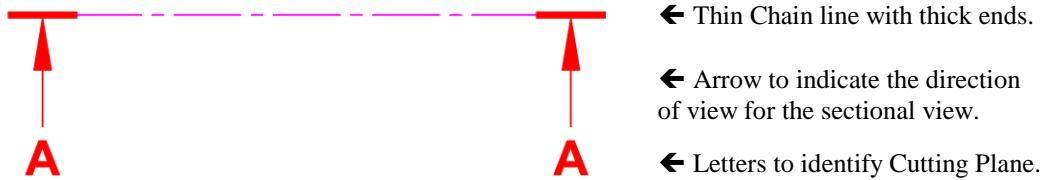


Figure 5.2. Cutting plane symbol.

5.2.2 Hatching Lines or Section Lines

On the drawing, hatching lines or section lines are used to distinguish those surfaces within a part that have been sectioned by the cutting plane. The hatching lines are drawn in **thin continuous lines** that are **spaced uniformly** at 45° or 135° as shown in Fig 5.3a. However, when the hatching lines are parallel or perpendicular to the outlines of the section, other angles such as 0° or 90° , 30° or 150° , 60° or 120° would then be used (see Fig 5.3b & 5.3c). The same angle of hatching lines shall apply to all the sectioned surfaces within the same part.

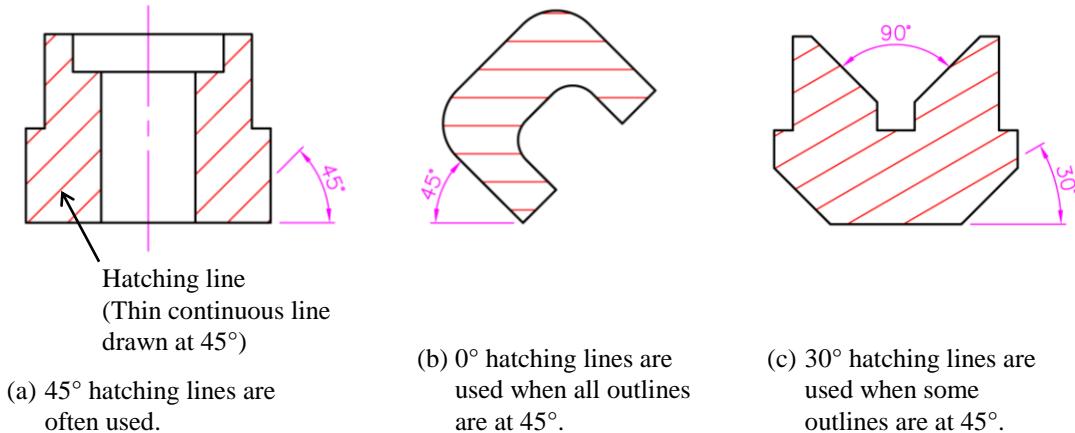


Figure 5.3. Correct application of hatching lines.

As a rule of thumb, no thick continuous lines should be drawn across the hatching lines. This is because thick continuous lines represent visible edges and these cannot exist on surfaces that have been cut by a cutting plane.

When several adjacent parts of a machine are being sectioned, each part will be distinguished from the other by varying the angle of hatching lines between 45° and 135° , or/and changing the spacing of hatching lines (see Fig 5.4).

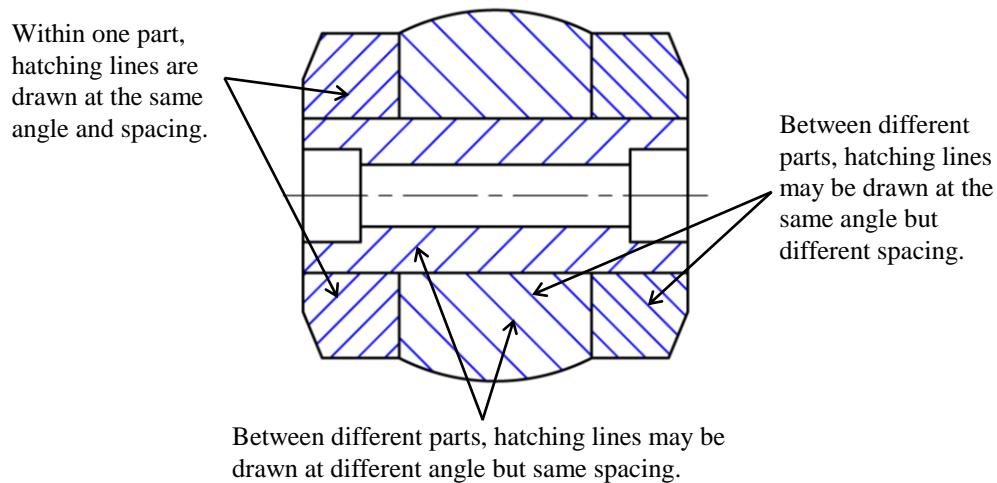


Figure 5.4. Correct application of hatching lines in an assembly.

5.2.3 Visible Outlines behind the Cutting Plane

All visible outlines behind the cutting plane must be included in the sectional view (see Fig 5.5).

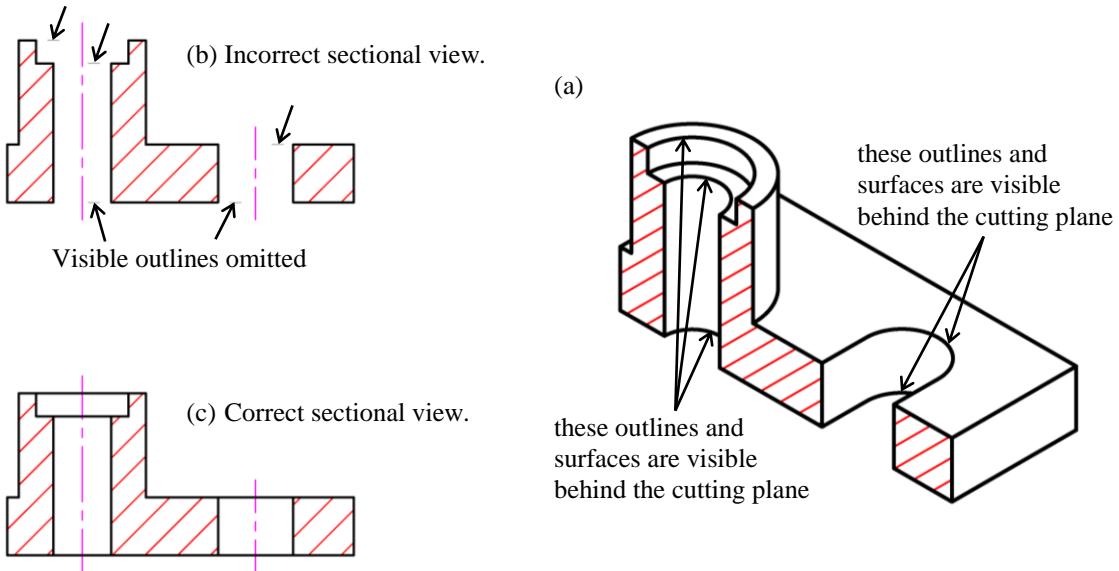


Figure 5.5. Visible outlines behind cutting plane.

5.2.4 Hidden Details in Sectional View

Hidden details in the sectional view are usually omitted.

5.2.5 Identifying the Sectional View

The sectional view is to be identified by letters that corresponds to its cutting plane, e.g. A-A (see Fig 5.1 and Fig 5.6).

5.3 Web in Sectional View

A web is a thin feature within the part for strengthening purpose. When drawing the sectional view, the web area will not be sectioned if the cutting plane is applied parallel to the web's larger area (see section A-A in Fig 5.6). The outlines of the web are to be included and drawn in thick continuous line. In this way, the web can be identified easily in the sectional view. At the same time, misleading interpretation such as the web's thickness is avoided. However, the web area will be sectioned when the cutting plane is applied perpendicular to the web's larger area (see section B-B in Fig 5.6).

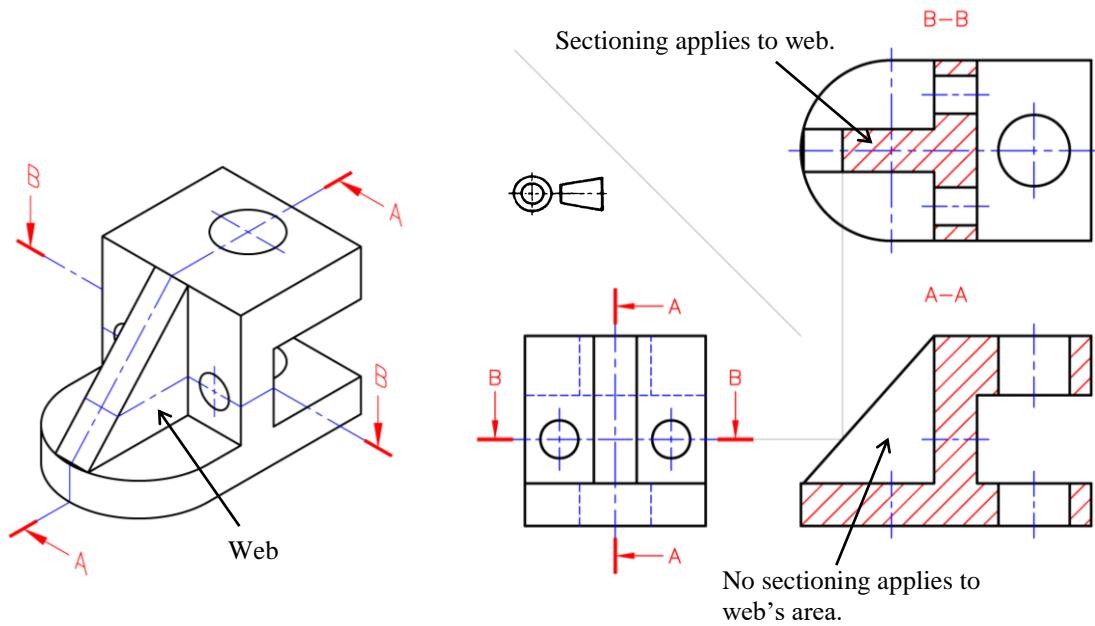


Figure 5.6. Application of sectioning to web.

5.4 Parts/Features not Sectioned

The following parts and features will not be shown in section when the cutting plane passes longitudinally through their axes:

- Shafts, axles and spindles.
- Screws, bolts, studs, nuts and washers.
- Dowel pins, taper pins, split pins, rivets and keys.
- Balls and rollers in bearings as well as bearing convention.
- Spokes of wheels and similar parts.
- Webs and ribs.

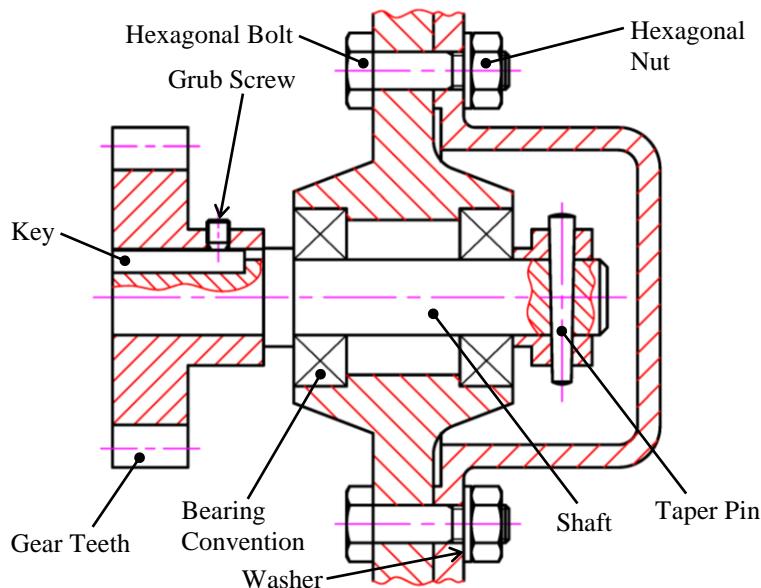


Figure 5.7. Parts/Features not sectioned in assembly.

The above parts and features are not sectioned because they have no internal details and they are easily recognised based on their outside views. However, if the cutting plane passes perpendicularly through their axes, the parts and features will have to be shown in section.

5.5 Types of Sectional Views

5.5.1 Full Sectional View

A **full** sectional view (Fig 5.8) is derived when a flat cutting plane passes through the part completely at the selected location.

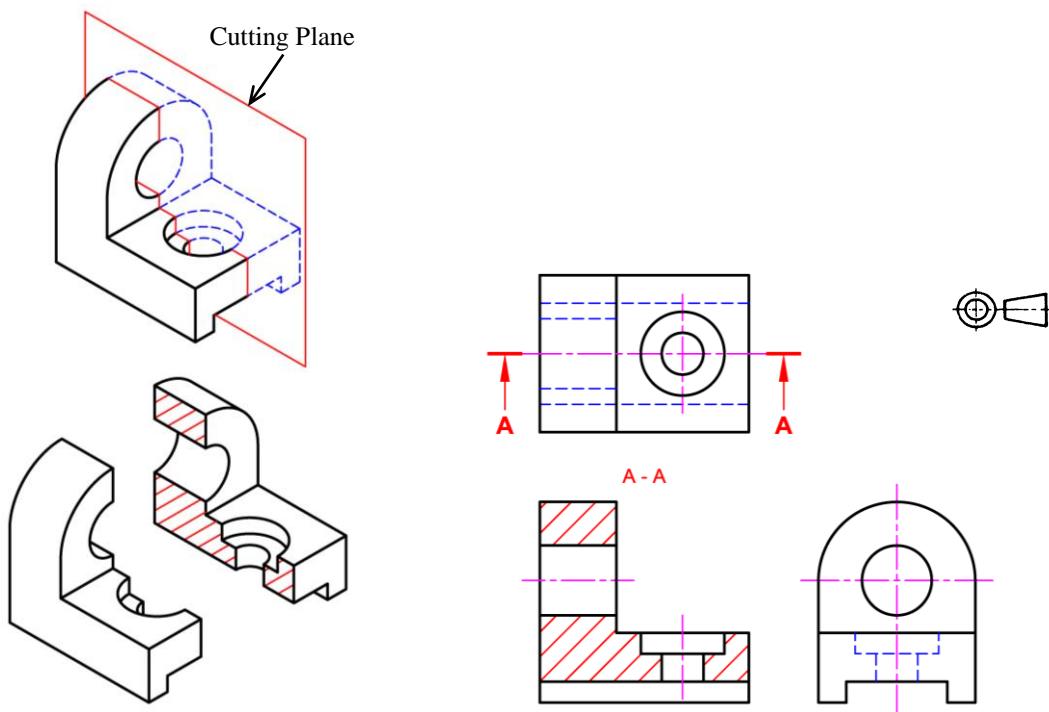


Figure 5.8. Full sectional view.

5.5.2 Offset Sectional View

In an **offset** section (Fig 5.9), the cutting plane is offset to include internal features that are not in a straight line. When showing the cutting plane in the drawing, thick dashes are used at the ends of offset to indicate change of direction of the cutting plane. Also, the offset edge of the cutting plane is not shown in the sectional view.

5.5.3 Half Sectional View

Half sectional view (Fig 5.10) is used on symmetrical parts such that half of the view is drawn in section and the other half as an outside view. This sectional view is derived by two perpendicular cutting planes that passed through the parts halfway at the planes of symmetry. The two halves of a half sectional view are to be separated by a centre line.

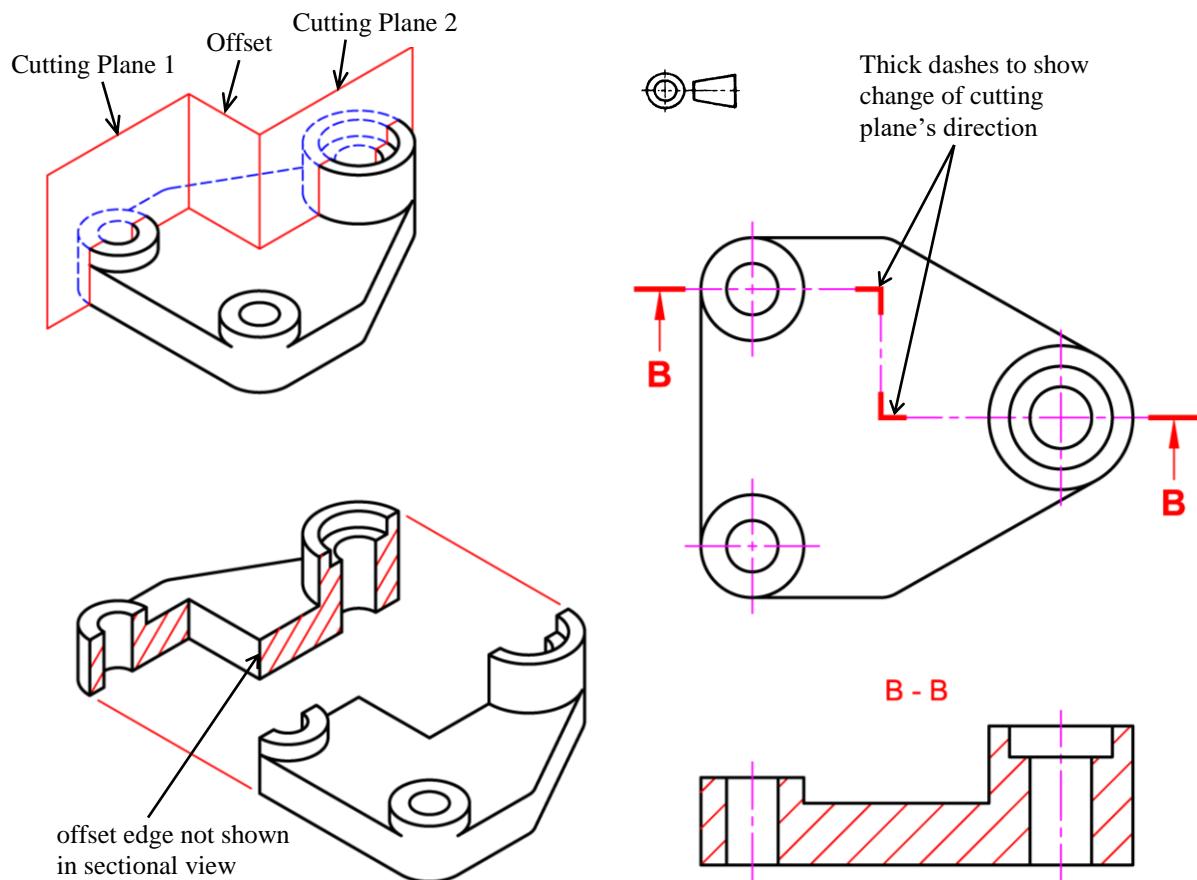


Figure 5.9. Offset sectional view.

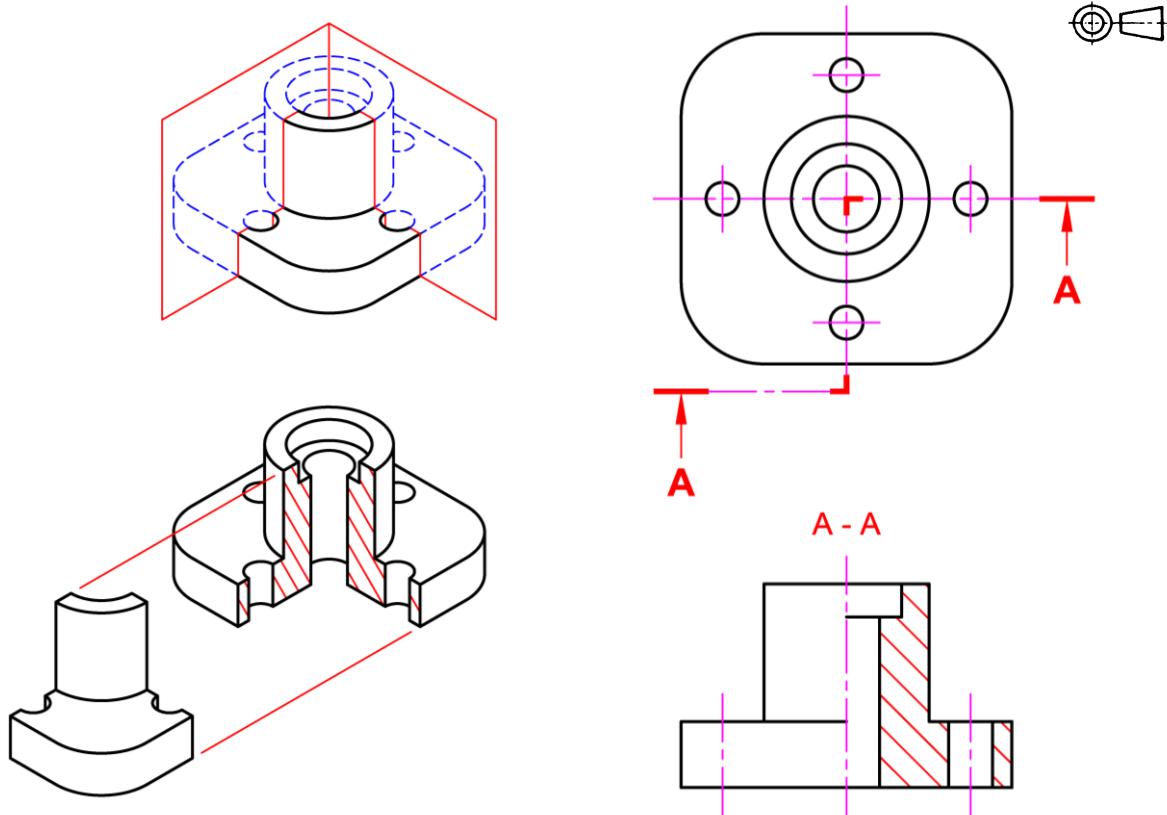


Figure 5.10. Half sectional view.

5.5.4 Aligned Sectional View

Aligned sectional view is used on parts where there are features present along the radial lines as shown in Fig 5.11. The cutting plane will pass through a principal centre line and one or more radial centre lines. The sectional view will then be drawn with features on the radial centre line rotated and aligned to the principal centre line. Thus, these features are shown at their true distances from the centre of the part.

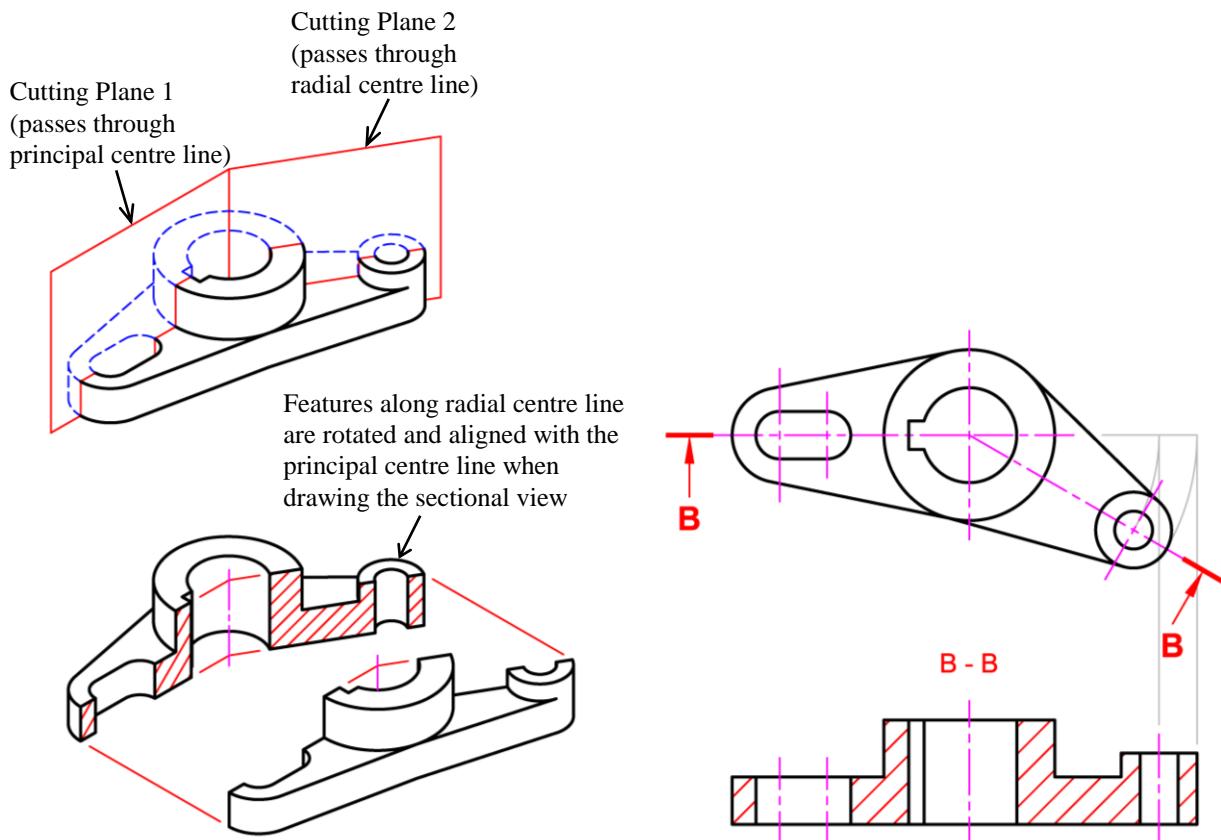


Figure 5.11. Aligned sectional view.

5.5.5 Revolved Section

Revolved section is used to show the shape of the local cross-section of a particular feature within the part. The “cross-section” is obtained at the cutting plane (see Fig 4.12a) and revolved 90° into the plane of the paper in the relevant view, i.e. perpendicular to the cutting. The outline of the revolved section is drawn directly on the relevant view (see Fig 4.12b) with **thin** continuous line.

5.5.6 Removed Section

Removed section is similar to revolved section except it is removed and drawn outside the relevant view (see Fig 5.12c). The outline of the removed section is drawn with **thick** continuous line. The removed section can be placed adjacent to the relevant view connected by a thin chain line.

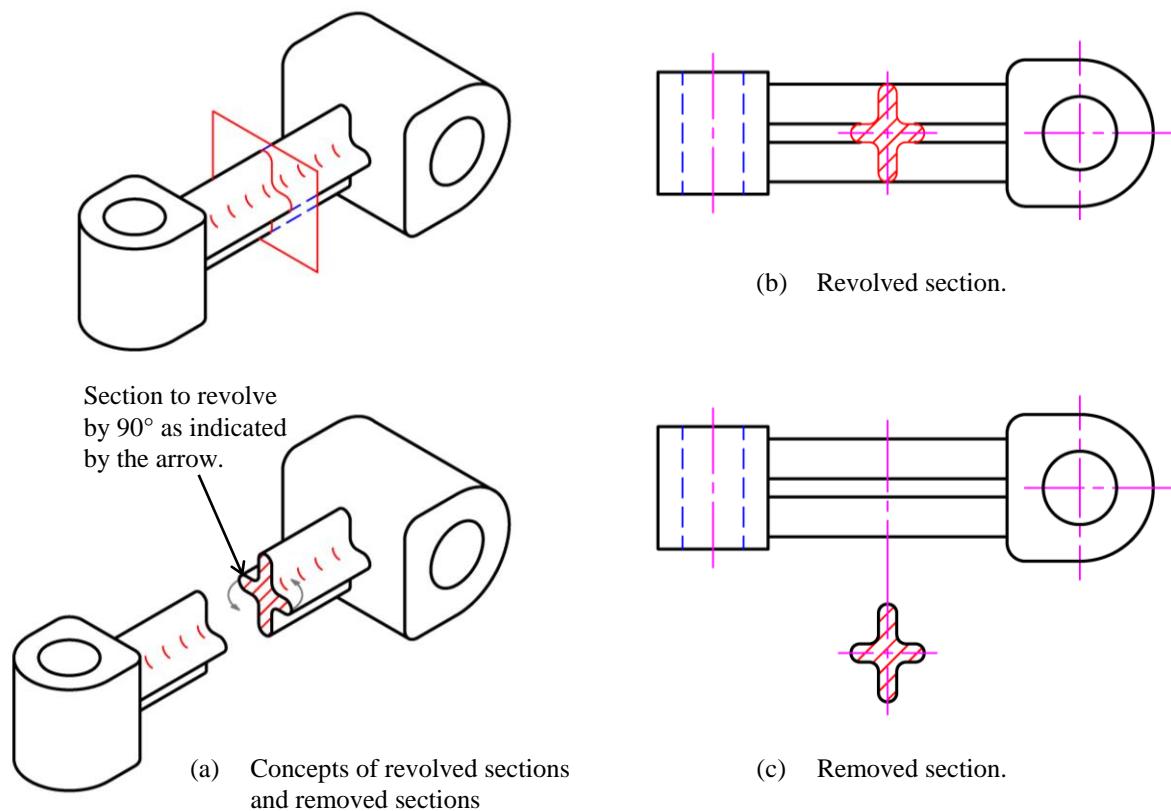


Figure 5.12. Revolved section and removed section.

5.5.7 Local Section or Part Section

A local section or part section (Fig 5.13) is used to show specific internal features on a principal view while preserving the necessary outside features. The desired region is broken out adequately to reveal the internal feature of interest. The outline of the break is drawn in thin irregular line.

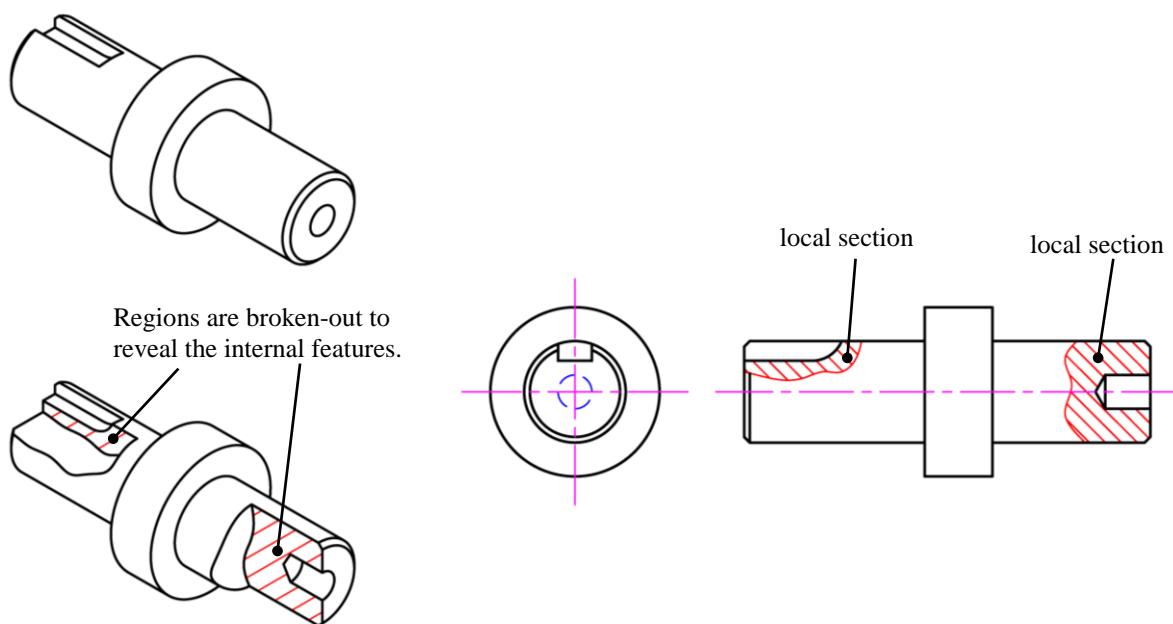
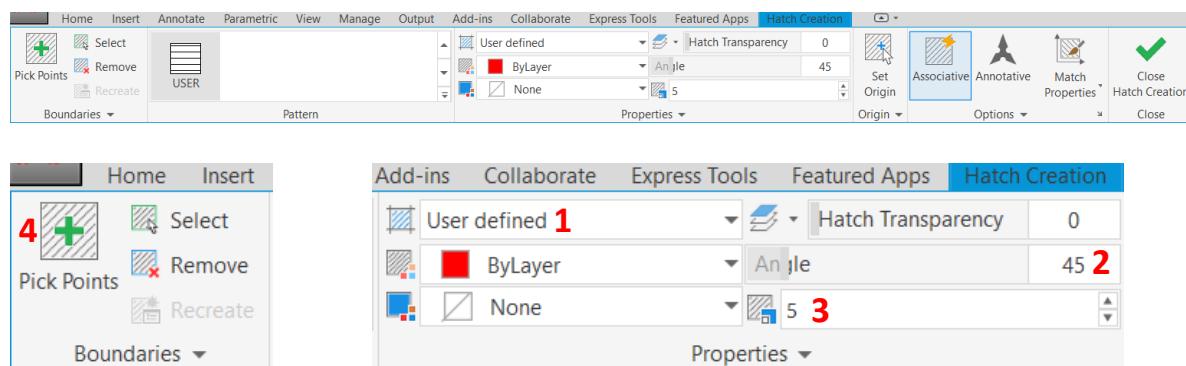


Figure 5.13. Local section or part section.

5.6 HATCH command

Ribbon access: **Home tab > Draw panel > Hatch**

The **HATCH** command is used to place pattern in closed areas. When drawing the sectional views, it can be used to fill the cut areas with hatching lines. The *Hatch Creation Ribbon Contextual Tab* is displayed when the **HATCH** command is selected.



The main steps to place a hatch pattern in a drawing through the *Hatch Creation Tab* are:

1. **Hatch Type:** Select “User defined” hatch. Other types of hatch available include Solid, Gradient and Pattern.
2. **Hatch Angle:** Specify the angle for “User defined” hatch. Angles to use are usually 45° or 135° .
3. **Hatch Spacing:** Specify the spacing for “User defined” hatch. Spacing to be 3mm or 5mm depending on the size of the area to be hatched.
4. **Pick Points:** Select the required areas to fill with the hatch pattern. For a preview of hatching, place the cursor within the required area.

5.6.1 Selecting Hatch Area

The simplest way to select areas for filling with a hatch pattern is to pick a point within the area to be hatched. This is done by clicking the “**Pick Points**” option in the “*Hatch Creation Tab*” box follow by picking a point within the hatched area when the command prompt displays as follows:

```
Command: _hatch
Pick internal point or [Select objects Undo seTtings] : (pick a point within the
hatch area.)
```

When a point is picked, AutoCAD will search for the bounding area, highlight it and fill it with the selected pattern. The prompt will repeat to allow more than one area to be selected at

the same time. Upon completing the selection of areas, press <Enter> to return to the dialog box.

Some of the selected hatch areas may be removed using “**Remove Boundary Object**” option in the Boundaries panel. The following prompt will appear when this option is selected:

Select objects or [Add boundaries] :

At the prompt, pick an object belonging to the boundaries of the areas to be removed.

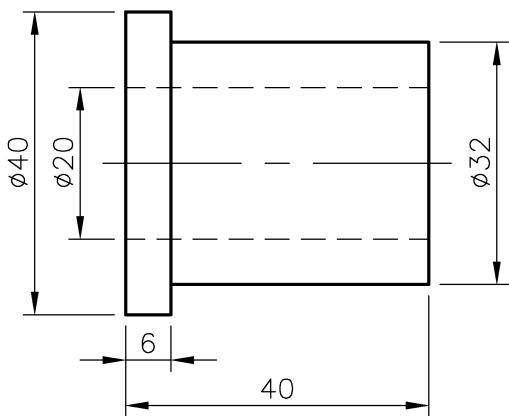
5.6.2 Modifying the Hatch

The properties of the hatch pattern may be modified by left-clicking the associative hatch object.

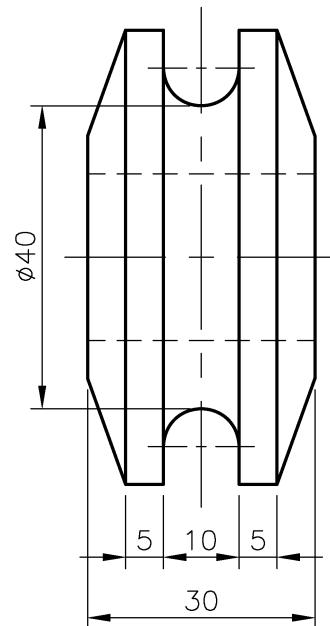
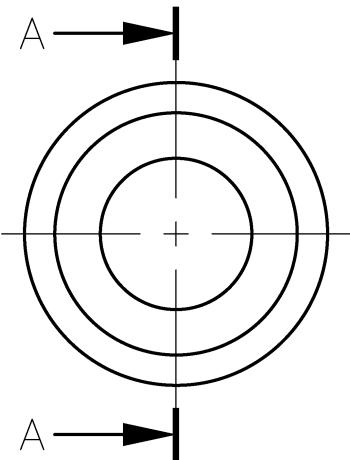
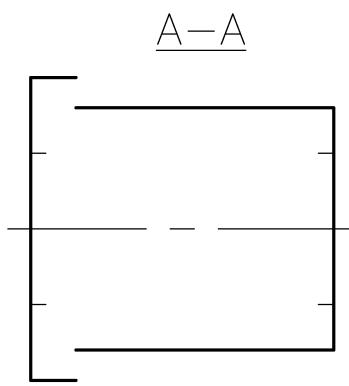
A *Hatch Editor Ribbon Contextual Tab* that is similar to the *Hatch Creation Ribbon Contextual Tab* will be displayed. Changes that can be made are hatch type, angle, spacing, hatch origin, and hatch areas (add or remove).

Tutorial 5

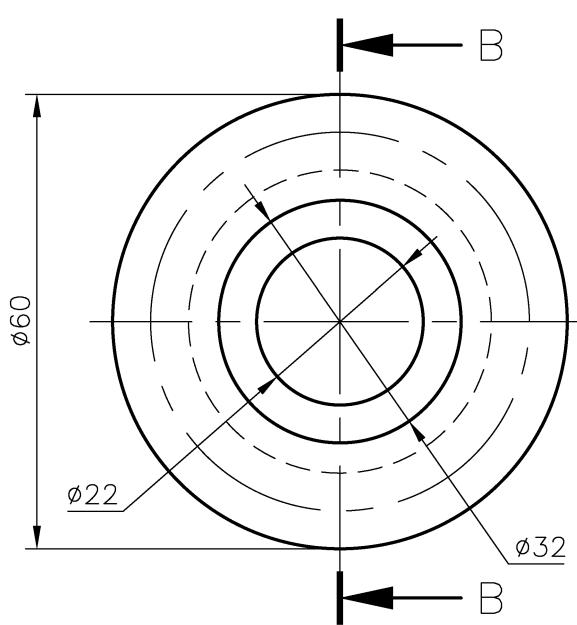
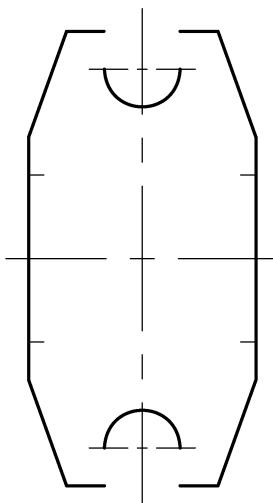
1. Complete the section view for the given components:

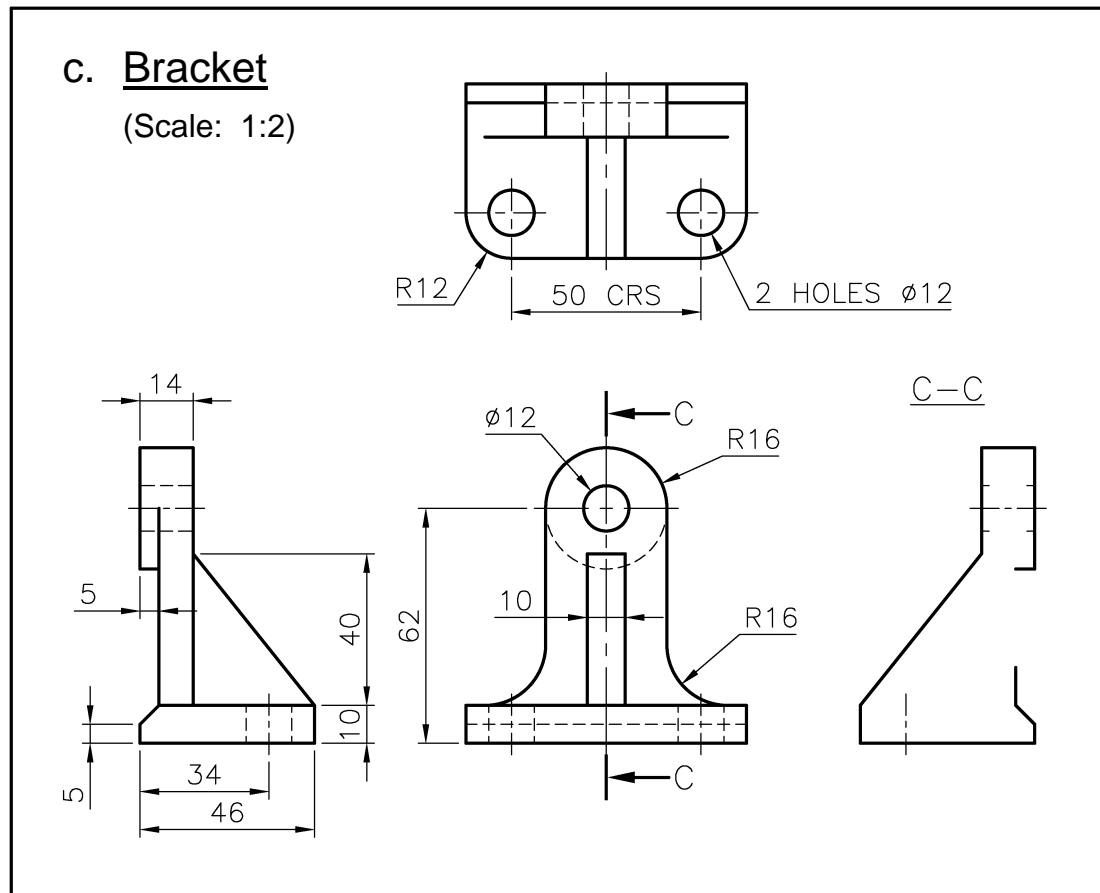
**a. Bush**

(Scale: 1:1)

**b. Pulley**

(Scale: 1:1)

**B-B**

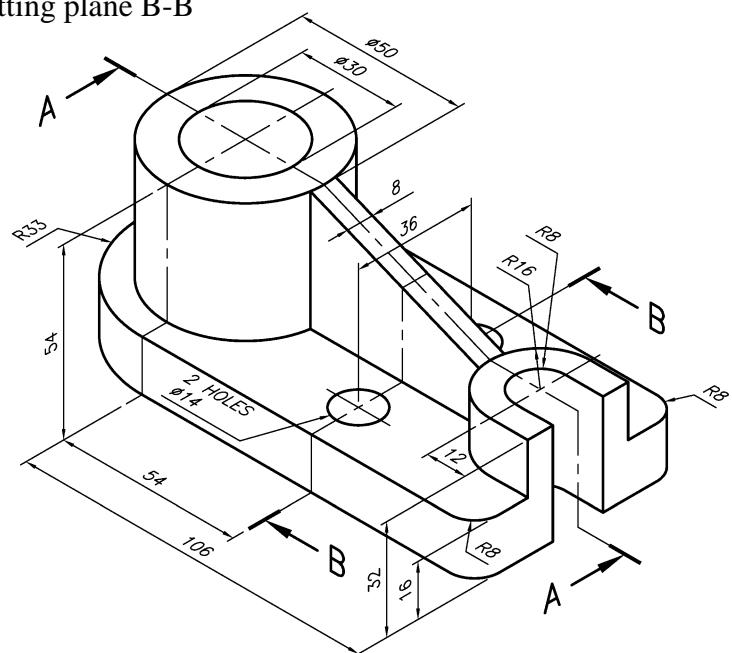


Note: For Q2 to Q5, complete the questions using the partially drawn views given in the template file, P:\MAE\ME1201\5 Sectioning\tut5.dwt

2. Draw/Complete the following views of the given figure in 3rd angle projection, using scale 1:1:

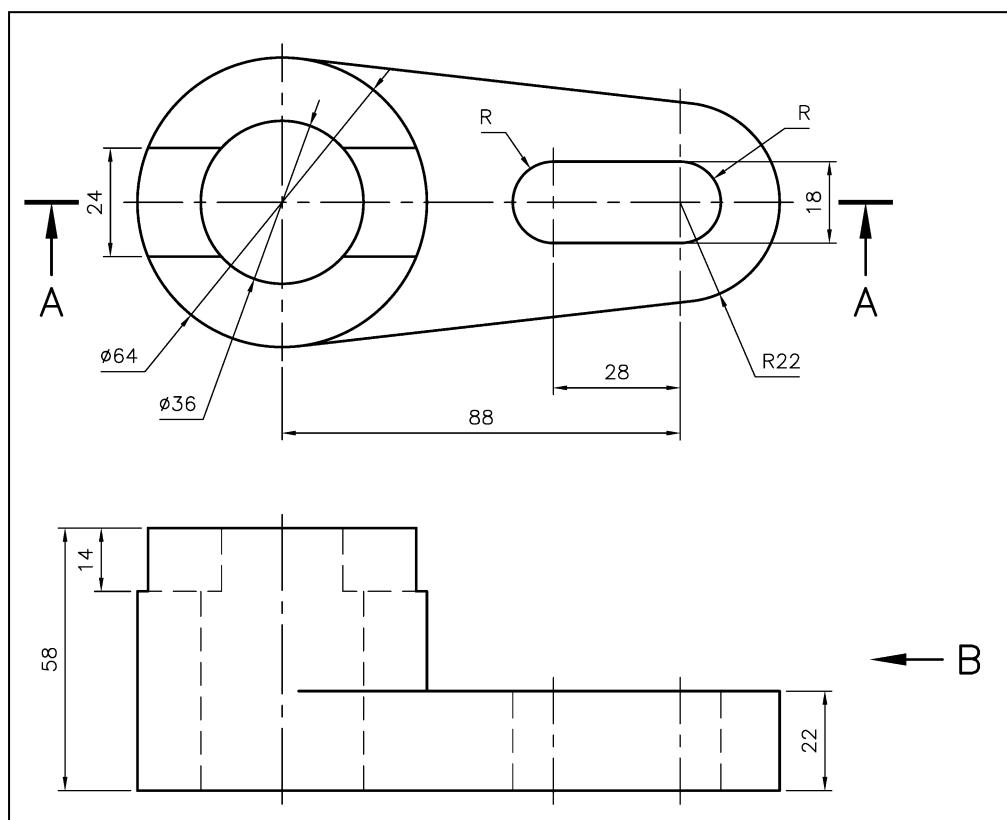
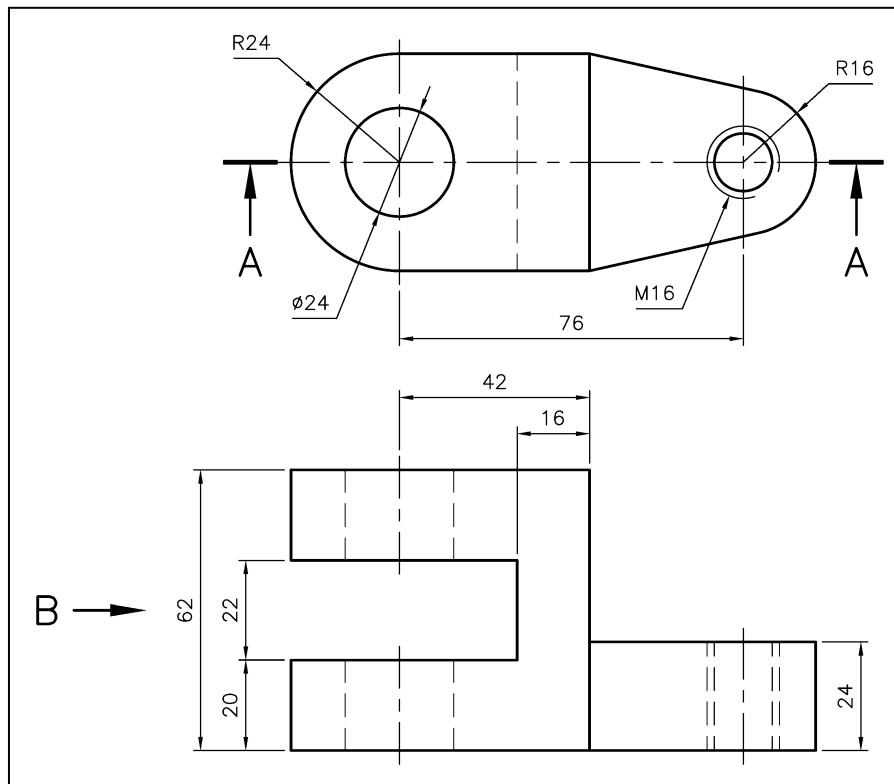
- a sectional front view due to cutting plane A-A
- a sectional end view due to cutting plane B-B
- a plan view

PLAN ↓



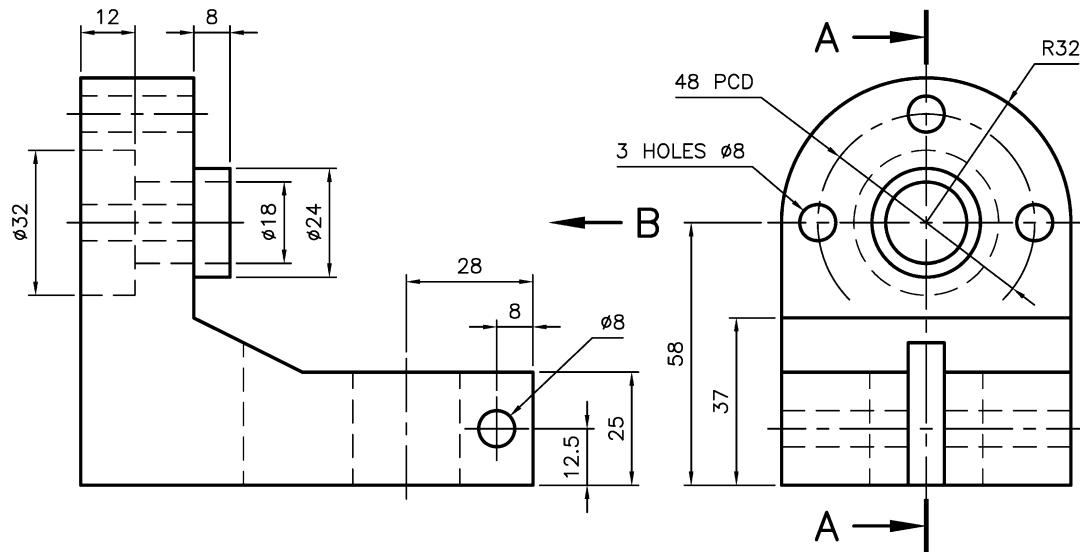
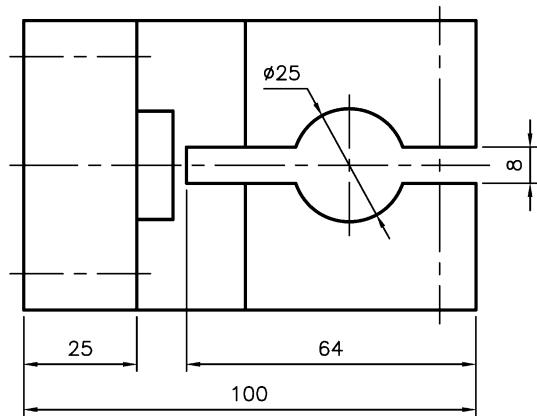
3. Draw/Complete the following views of the two given figures in 3rd angle projection, using scale 1:1:

- a sectional front view due to cutting plane A-A
- an end view from B
- the given plan view



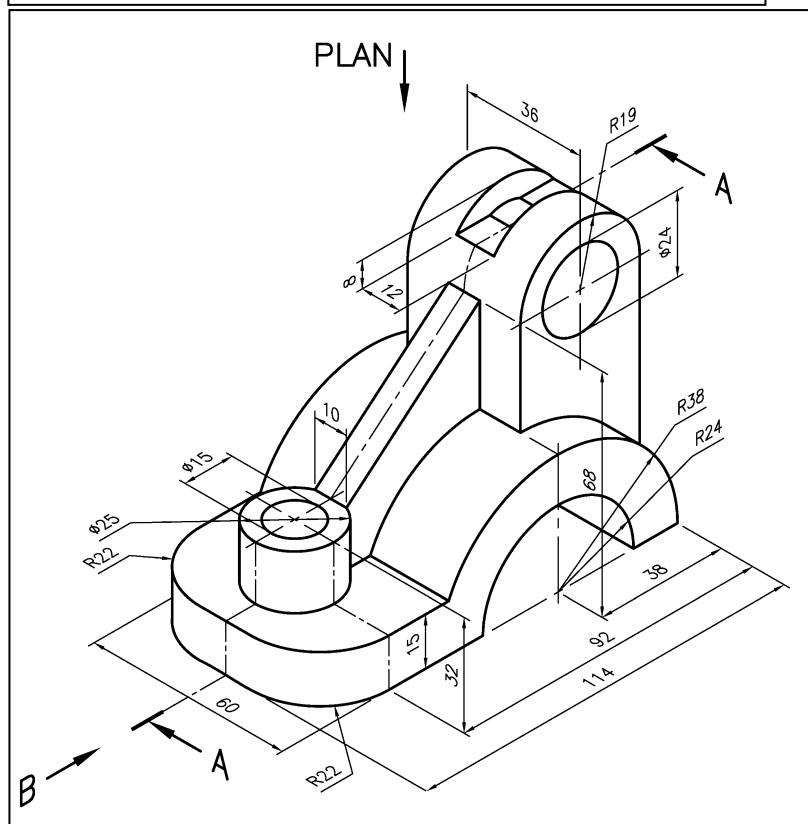
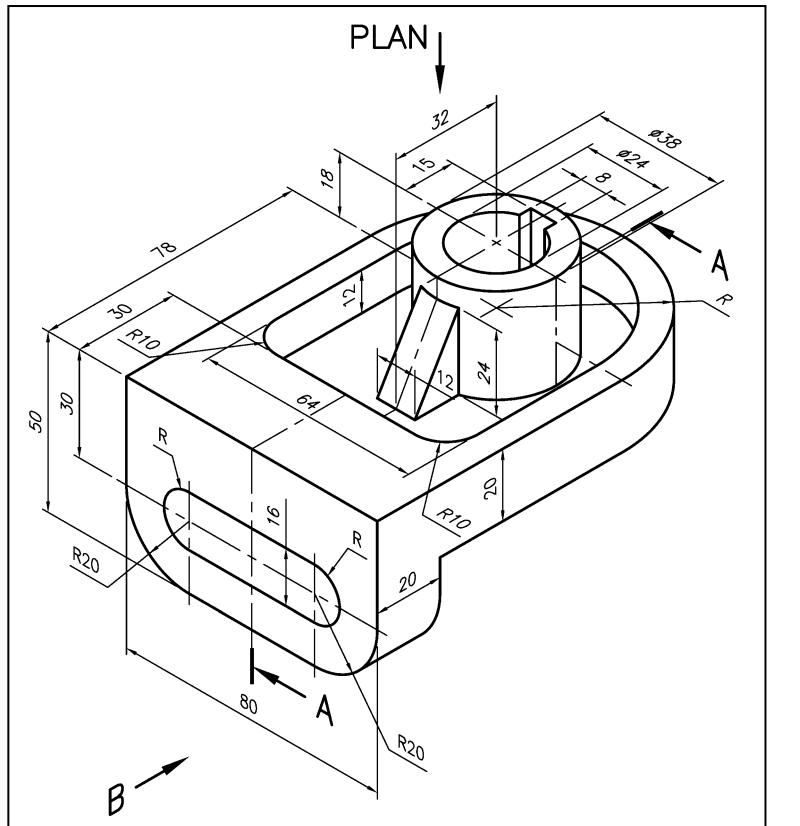
4. Draw/Complete the following views of the given figures in 3rd angle projection, using scale 1:1:

- a. a sectional front view due to cutting plane A-A
- b. the given end view from B
- c. the plan view with the required hidden details



5. Draw/Complete the following views of the two given figures in 3rd angle projection, using scale 1:1:

- a sectional front view due to cutting plane A-A
- an end view from B
- the plan view



UNIT 6 DIMENSIONING

Learning Objectives

By the end of this unit, students should be able to:

- Explain the practices of dimensioning in accordance to ISO Standard recommendations.
- Explain the basic types of dimensioning: Linear, angular, diameter, radius, and leader.
- Set up dimension style with appropriate settings using the dimension style manger.
- Compose dimensions on drawing in accordance with ISO Standard recommendations with various AutoCAD dimensioning commands.
- Modify the appropriate dimensions variables to meet the required standard.
- Communicate the feature sizes of engineering component through the dimensions specified in its engineering drawing.

6.1 Practices of Dimensioning

An engineering drawing conveys information in two ways:

- a. by pictorial or orthographic views of the object.
- b. by instructions in the form of given sizes or dimensions and notes specifying the manufacturing processes and materials.

There are two types of dimensions:

- a. **Size dimensions** which define the size and shape of the design feature.
- b. **Location dimensions** which specify the relative positions of design feature.

The essential elements of dimension are illustrated in Fig 6.1.

Note 1 Dimensional value: Numerical value that defines the size, shape, location of a design feature. It is placed parallel to their dimension line and preferably near the middle, above and clear of the dimension line. It may also be placed to the right or left of the middle to avoid congestion. It must not be crossed or separated by any line.

Note 2 Auxiliary dimension: This dimensional value is for information only and shall be enclosed in parentheses and shall never be tolerated.

Note 3 Dimension line: A thin continuous line (straight or curved) draws between two extension lines, or between a feature and an extension line, or between two features indicating the extent of a dimension graphically. It is placed outside the part wherever possible and spaced adequately away from the outlines. The longer

dimension lines are placed further than the shorter ones so that they will not cross by extension lines wherever possible.

- Note 4 Extension line: A thin continuous line connecting the feature(s) to be dimensioned and the ends of the corresponding dimension line.
- Note 5 Terminator: Closed and filled arrowhead serves as the ends of a dimension line and leader line.
- Note 6 Leader line: A thin continuous line connecting a feature to a dimension, note or symbol (e.g. surface texture symbol). It shall be drawn at an angle.
- Note 7 A permissible gap (approx. 8x the line width) between the feature and the beginning of extension line.
- Note 8 Extension line extending beyond the respective dimension line (approx. 8x the line width)
- Note 9 Dimensional value is placed so that it may be read from the bottom or
- Note 10 Dimensional value is placed so that it may be read from the right hand side of the drawing.
- Note 11 Diameter symbol: A symbol (\emptyset) that precedes a numerical value to indicate the diameter of a circular profile.
- Note 12 Radius symbol: A symbol (R) that precedes a numerical value to indicate the radius of an arc, round or fillet.

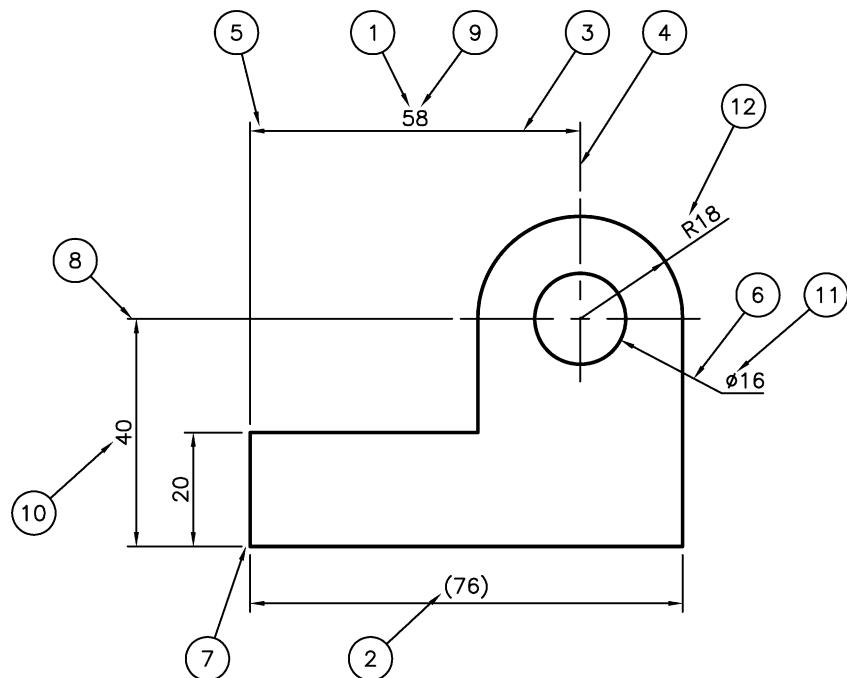


Figure 6.1. Essential Elements of Dimensions.

Other elements of dimensioning are:

1. Dimension lines need not be shown in full for the following:
 - Dimensions of diameters are indicated (Fig 6.2a).
 - Feature(s) in half sectional view (Fig 6.2a).
 - Feature(s) about the axis of symmetry in a half view (Fig 6.2b).
2. Extension lines may be interrupted if their continuation is recognisable (Fig 6.2c¹).
3. When the feature is interrupted, the corresponding dimension line shall still be uninterrupted (Fig 6.2c²).
4. Dimensional values can be positioned to adapt to different situations:
 - Where there is limited space above the dimension line, the dimensional value can be placed above the extension of the dimension line beyond one of the terminators, or above a leader line terminating on the corresponding dimension line without terminator (Fig 6.2c³).
 - Where space does not allow placement parallel to the dimension line, the dimensional value can be placed above a horizontal extension of a dimension line (Fig 6.2c⁴).

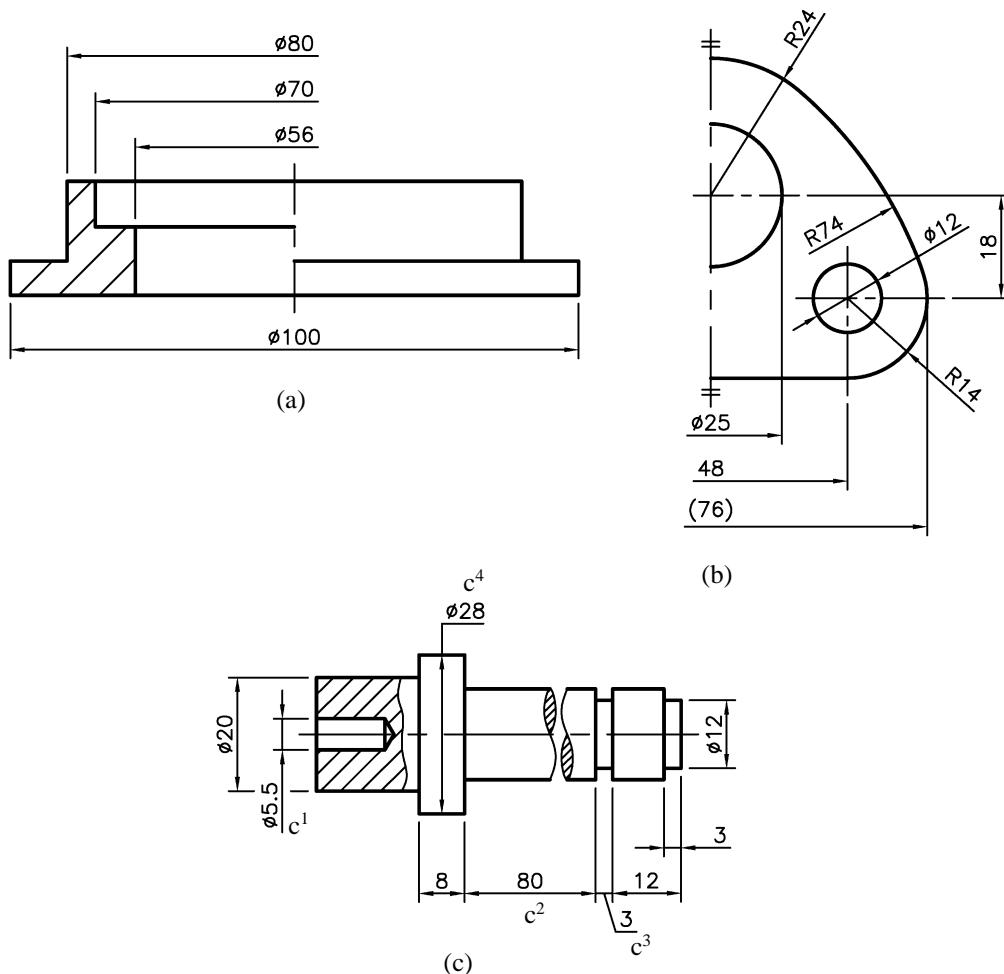


Figure 6.2. Other Elements of Dimensions.

5. Intersecting projection lines of feature outlines shall extend beyond their point of intersection (approx. 8x the line width). The extension lines will apply at the point of intersection of the projection lines.

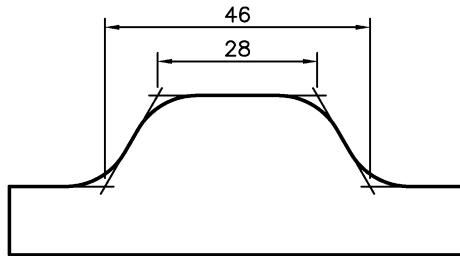


Figure 6.3. Intersecting Projection Lines.

6.2 Diameters and Radii Dimension

Diameter dimensions can be denoted on drawings as below and the ϕ symbol shall precede the dimensional value. When a diameter is illustrated by one arrowhead, the dimension line shall exceed the centre.

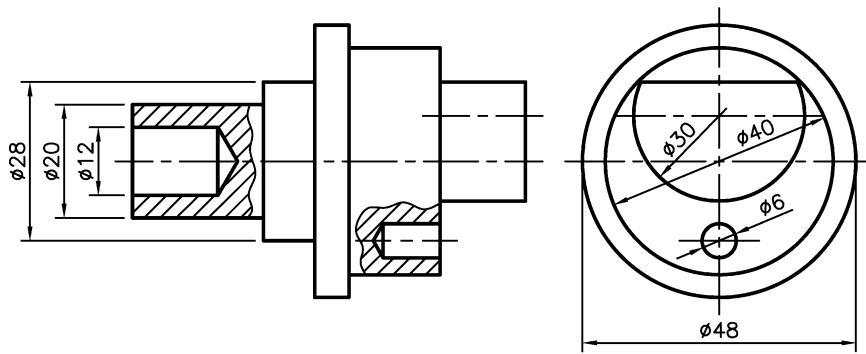


Figure 6.4. Diameter Dimensioning.

Radii dimensions are denoted on drawings as below and the letter R symbol shall precede the dimensional value. The dimension line shall be drawn from or in line with the centre of the arc and terminated with an arrowhead touching the arc, inside or outside the feature outline. Where the centre of a radius falls outside the available space, the dimension line shall be either broken (locating arc's centre is not required) or interrupted perpendicular (locating arc's centre is required).

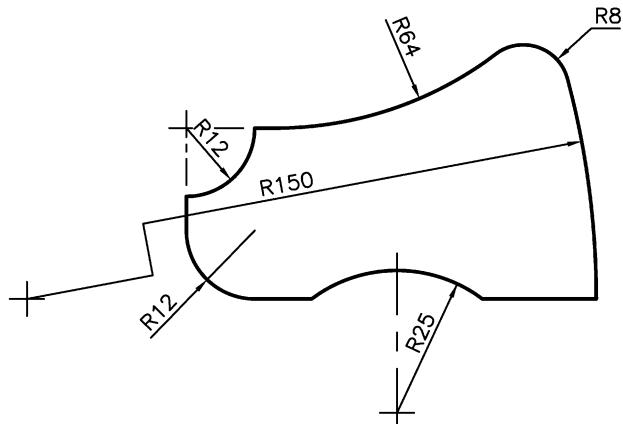


Figure 6.5. Radii Dimensioning.

6.3 Equally Spaced and Repeated Features

Features having the same spacing and uniformly arranged can have the spacing dimensioning simplified as below. Repeated linear and angular spacings can be indicated with the no. of spacings and their dimensional value or angle separated by “ \times ”. The overall dimension of the spacings shall be indicated as an auxiliary dimension.

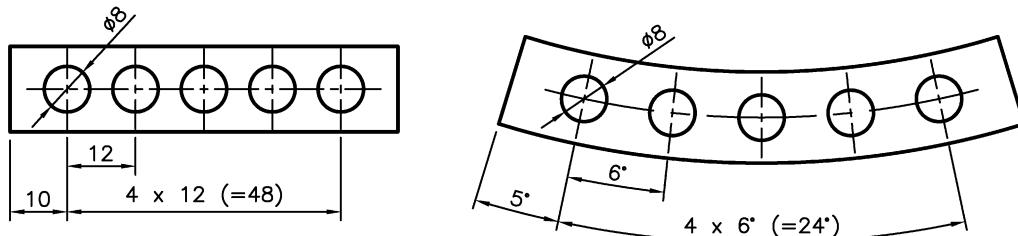


Figure 6.6. Equally Spaced Features.

Features having the same dimensional value may be dimensioned with the no. of features and their dimensional value separated by “ \times ” (Fig 6.7, [A]). However, if these repeated features are shown unambiguously, the dimension may also be indicated once.

The angular spacings of features may be omitted where they are self-evident (Fig 6.7, [B]).

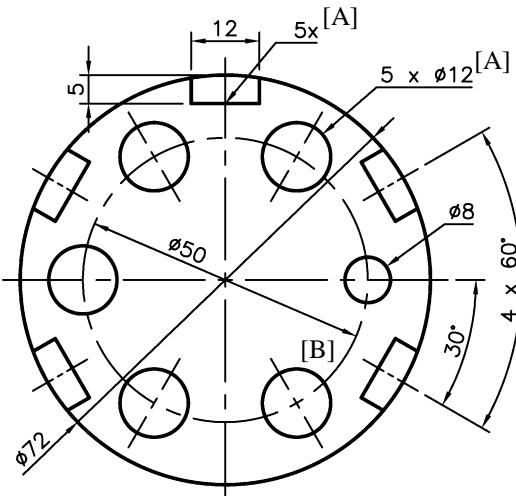


Figure 6.7. Repeated Features.

6.4 Arrangement of Dimensions

6.4.1 Chain dimensioning

A chain of single dimensions is arranged in a row. It should be used only where the possible accumulation of tolerances does not impinge on the functional requirements of the part.

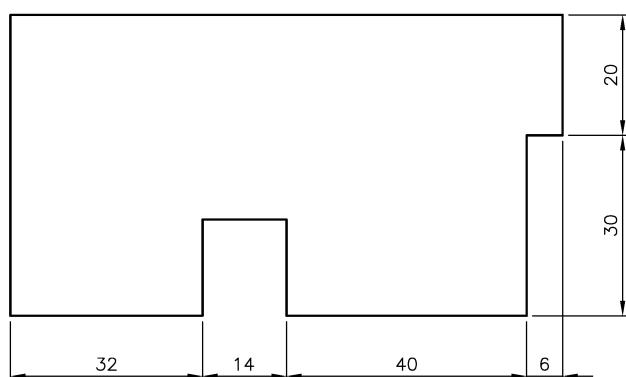


Figure 6.8. Chain Dimensioning.

6.4.2 Parallel dimensioning

Parallel dimensioning is the placement of a number of dimension lines parallel to one another and spaced out.

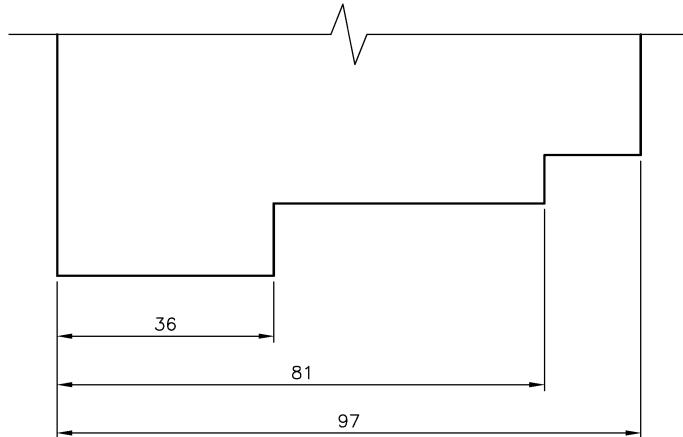


Figure 6.9. Parallel Dimensioning.

6.4.3 Combined dimensioning

Chain dimensioning and parallel dimensioning may be combined on a drawing whenever necessary.

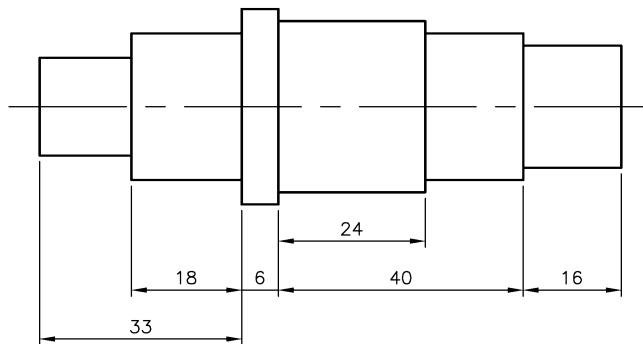
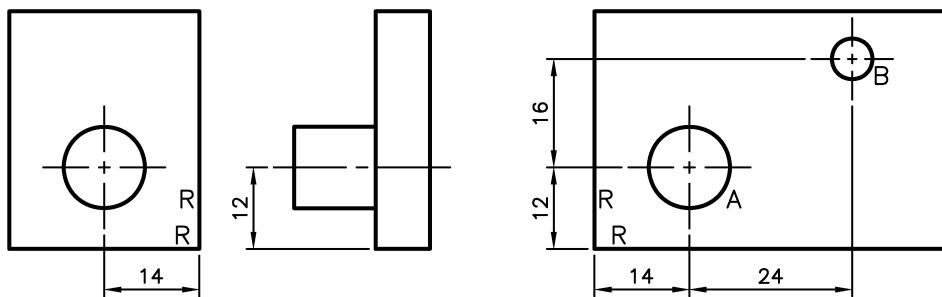


Figure 6.10. Combined Dimensioning.

6.5 Size and Location Dimensions

Size dimensions are used to describe heights, widths, thickness, diameters, radii, etc., and where appropriate, the shapes of the component. Location dimensions are necessary to locate the various features of a component relative to each other, to a reference surface or a centre line, etc.

Figure 6.11 shows how to dimension a component when location is necessary. The features can be located from a machined surface or a centre line. Such a surface or line is known as a DATUM.



(a) Spigot located from two reference edges (R)

(b) Hole A is located from two reference edges (R). Hole B is located from hole A.

Figure 6.11. Size and Location Dimensions.

6.6 Tolerances in Dimensioning

In manufacturing, it is impossible to produce components to an exact size. Thus, it is necessary to indicate the range of variation that is permitted within the parts. This range of variation is known as tolerance. It is the difference between the maximum and minimum limits of the specified dimension.

Tolerances should be specified in situations where the dimension is critical to the proper functioning or interchangeability of a component.

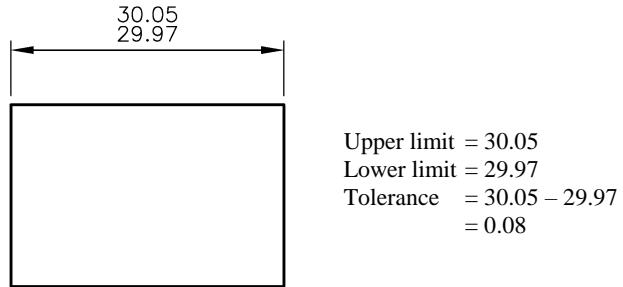
6.6.1 Specifications of tolerances

The various methods of expressing tolerances within the engineering drawing are as follows:

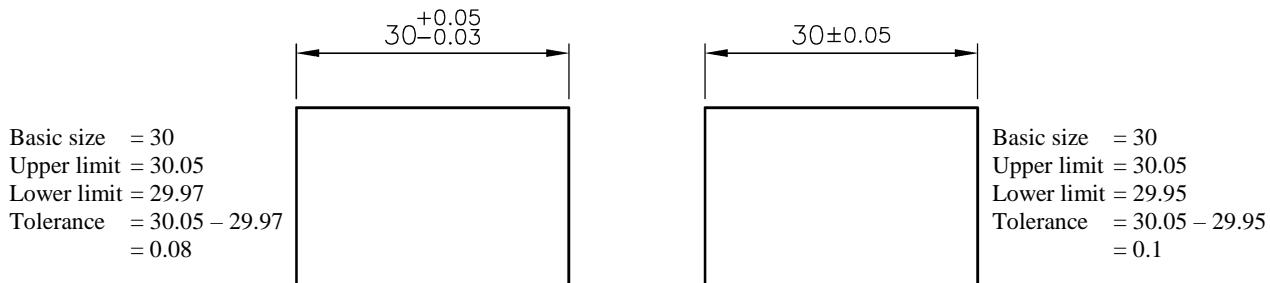
- a. General tolerances. A table note in the title block may be used to specify the tolerance for those dimensions which have no specific tolerances. An example of such is:

UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS IN MM GENERAL TOLERANCE AS PER ISO2768-m							
NOMINAL DIMENSION	OVER	0.5	3	6	30	120	315
	UP TO	3	6	30	120	315	1000
PERMISSIBLE VARIATION	± 0.1	± 0.1	± 0.2	± 0.3	± 0.5	± 0.8	

- b. Limits of size. The maximum and minimum limits of size are specified as shown in Fig 6.12a. The maximum value is placed above the minimum value.
- c. Bilateral tolerance. The dimension is specified by its basic size (theoretical size with no variation) followed by the upper (+) and lower variation (-) as shown in Fig 6.12b. The upper variation is placed above the lower variation.
- d. Unilateral tolerance. The dimension is specified by its basic size followed by either the upper or lower variation against the basic size as shown in Fig 6.12c.



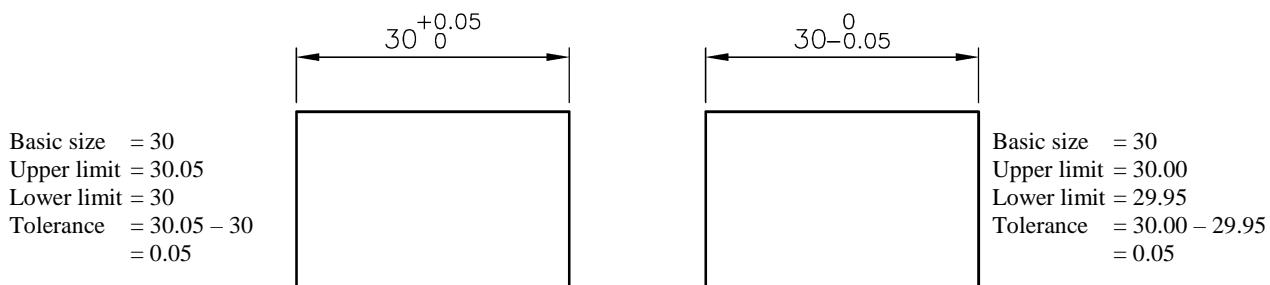
(a) Tolerance by limit of size.



(i) Unequal upper and lower variation

(ii) Equal variation

(b) Bilateral tolerance.



(c) Unilateral tolerance.

Figure 6.12. Methods of Specifying Tolerance for Dimensions.

6.7 Dimensioning Cylinder

Dimensions of cylinder should be placed in the most appropriate view to ensure clarity. Preferably, the diameter and length of cylinder will be placed in the longitudinal view as shown in Fig. 6.13. Dimensioning the diameter of cylinder in its circular view is not recommended unless clarity is improved. The size of cylinder should never be dimensioned by its radius.

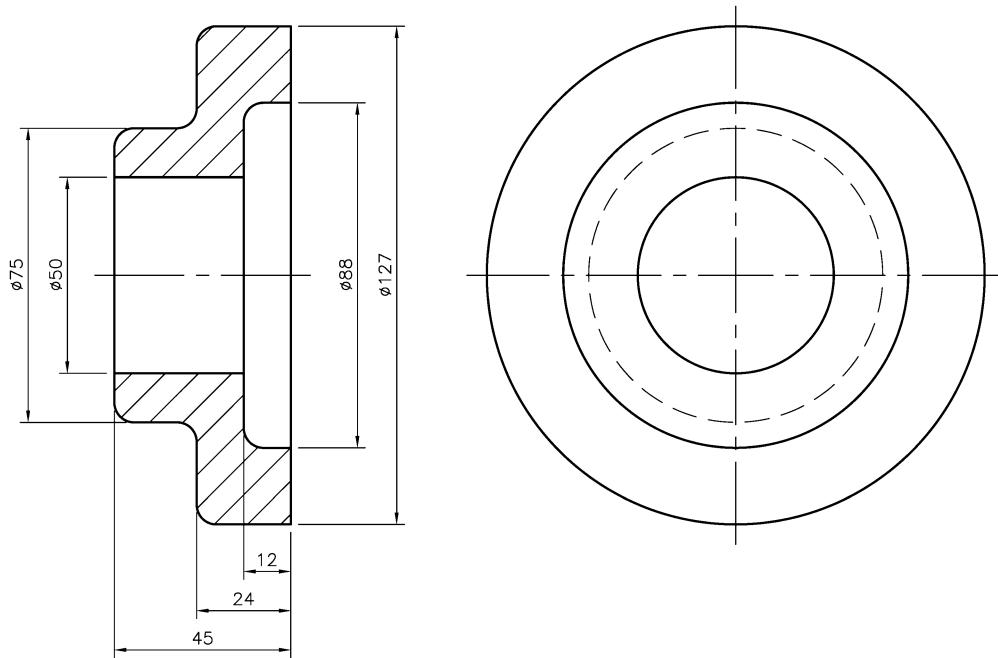
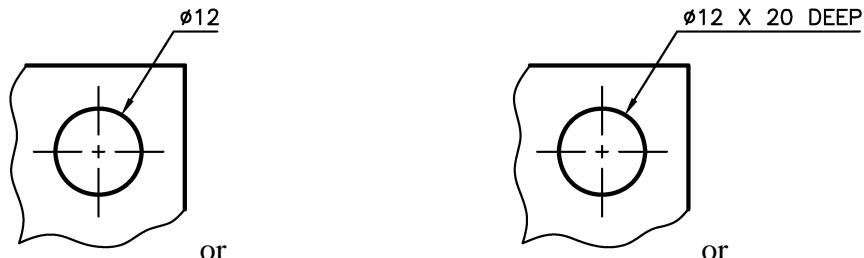


Figure 6.13. Dimensioning Cylinder.

6.8 Dimensioning Holes

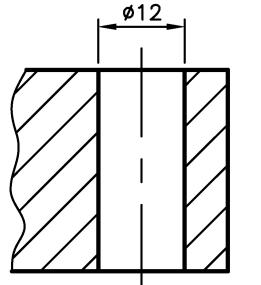
Figure 6.14 shows the dimensioning of various types of holes either as dimensions (**preferred**) or as notes. The leader must be in line with the centre of the circle.

As note:

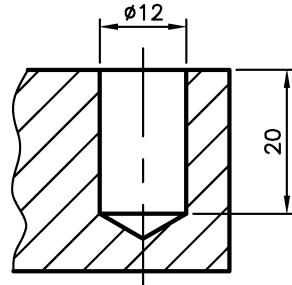


As dimensions:

(**Preferred**)

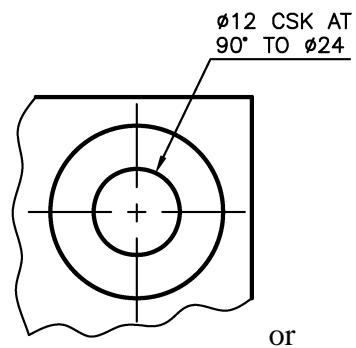
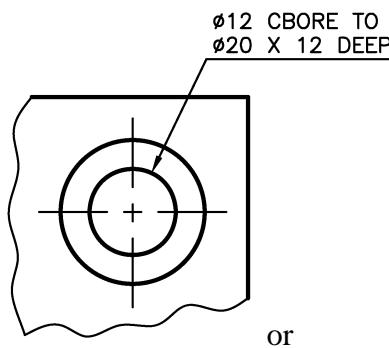


(a) Through hole



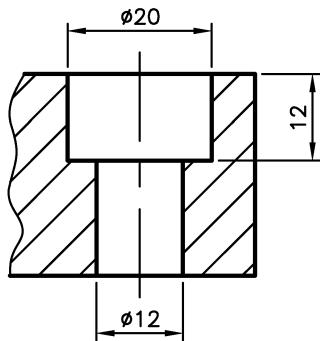
(b) Blind drilled hole

As note:

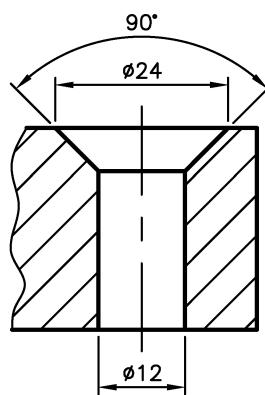


As dimensions:

(Preferred)



(c) Counterbore hole

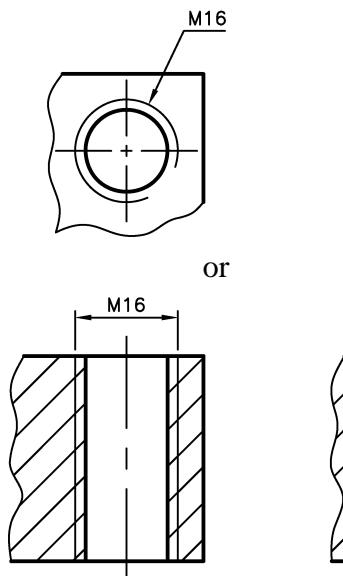


(d) Countersunk hole

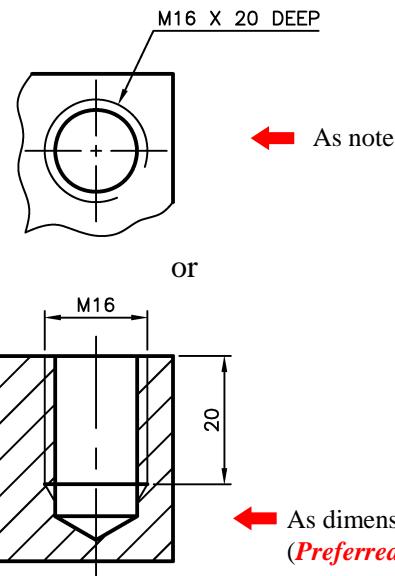
Figure 6.14. Dimensioning Holes.

6.9 Dimensioning Threaded Features

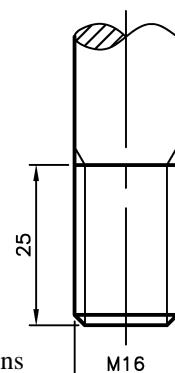
Thread features may be called out as shown in Fig 6.15.



(a) Dimensioning threaded holes



As note
(Preferred)



(b) Dimensioning threaded shaft

Figure 6.15 Dimensioning Threaded Features.

6.10 Dimensioning Chamfers

Chamfers may be dimensioned by one of the methods shown in Fig 6.16.

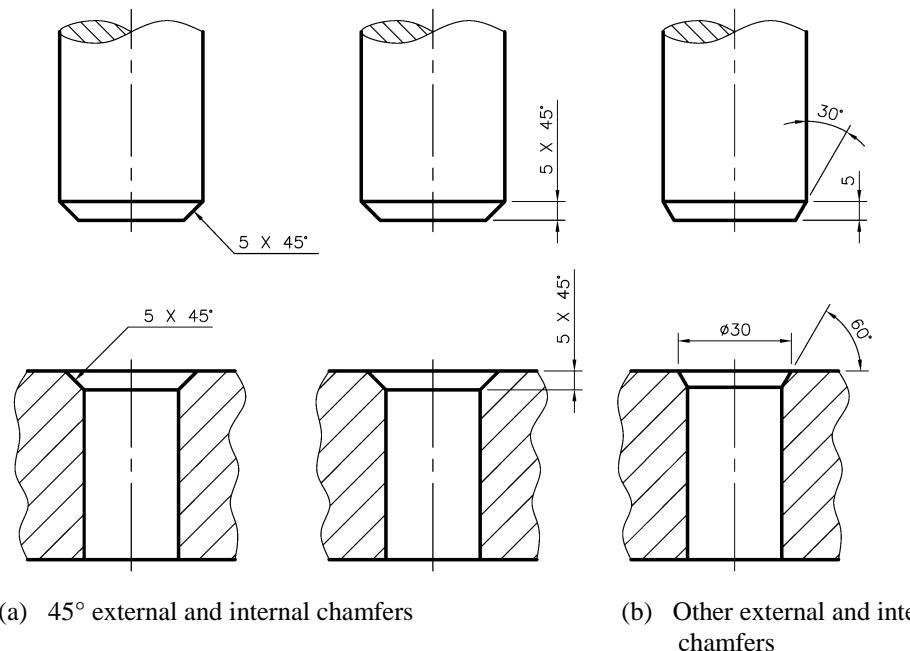


Figure 6.16. Dimensioning Chamfers.

6.11 Guidelines for Dimensioning

The essential principle of dimensioning is clarity. The ability to create good dimensioning depends on:

1. Choice of dimensions.
2. Placement of dimensions.
3. Good knowledge of manufacturing processes used.

Some guidelines of dimensioning are given below:

1. Each feature of a part is dimensioned only once. Dimensions should not be repeated, nor should the same information be given in two different ways.
2. Dimension the features in the view where the feature's shape is best shown.
3. Dimension features in the view where they are seen in true shape and size.
4. Dimensioning of hidden features should be avoided whenever possible. Sectional views should be used to reveal the hidden features to facilitate its dimensioning.
5. Circular features shall be located by dimensioning the centrelines.
6. Use diameter dimensions for circles ($> 180^\circ$). Use radial dimensions for arcs ($< 180^\circ$).
7. Dimension in one view as much as possible.

8. Leader lines used to dimension circles or arcs are to be radial and sloped at any convenient angle except vertical or horizontal.
9. Cylinders should preferably be dimensioned in their longitudinal view with diameters.
10. Diameters, Radii, Squares should be specified with appropriate symbol preceding the dimensional value.
11. No line of the drawing should be used as a dimension line or coincide with a dimension line.

6.12 Types of Dimensioning in AutoCAD

AutoCAD provides three basic types of dimensioning: linear, radial and angular as shown in Fig 6.17.

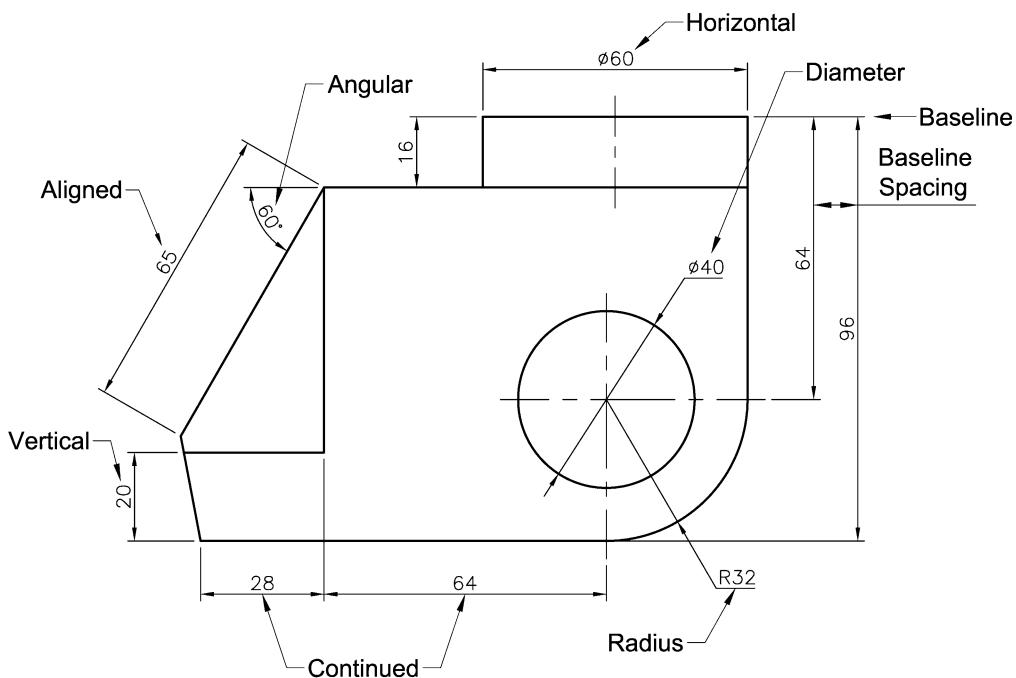


Figure 6.17. Types of Dimensioning available in AutoCAD.

6.13 Dimensioning Commands

Basic dimensioning commands can be accessed from the Ribbon:

Home tab > Annotation panel.

Icon	Description
	Linear Creates a linear dimension with a horizontal, vertical or rotated dimension line.

	Aligned Creates a linear dimension that is aligned to the origin points of the extension lines.
	Angular Creates an angular dimensions between selected objects or three points.
	Radius Creates a radius dimension for a circle or an arc.
	Diameter Creates a diameter dimension for a circle or an arc.
	Dimension Creates multiple types of dimensions: Linear, Aligned, Angular, Radius, Diameter, etc.
	Dimension Style Setups and controls the appearance of dimensions.

Other less common dimensioning commands can be accessed from the Ribbon:

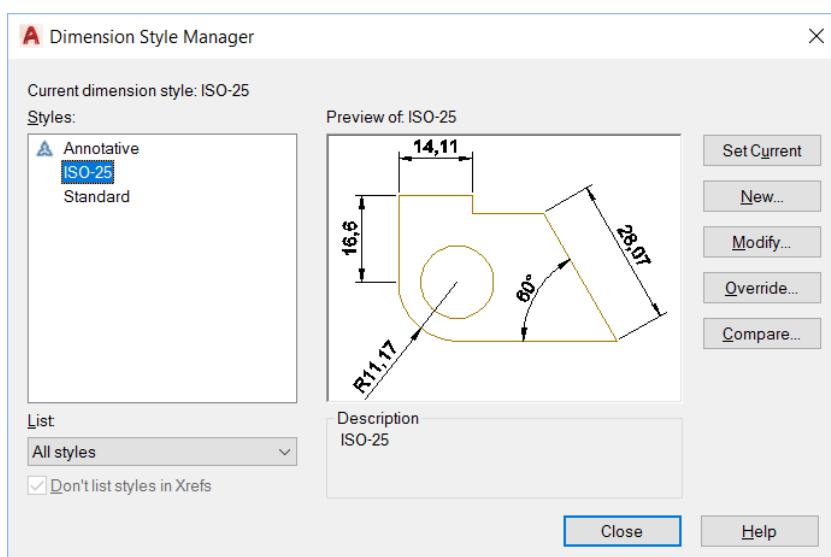
Annotate tab > Dimensions panel.

6.14 Creating Dimension Style

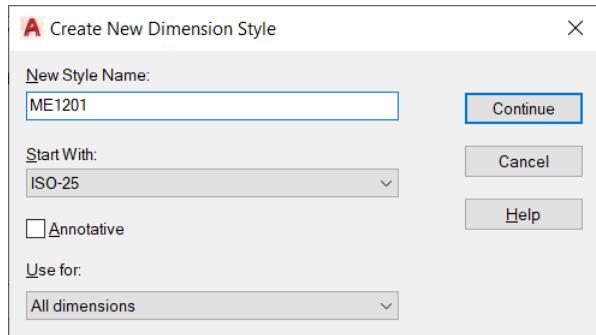
Dimension style is used to collate the necessary dimension settings so as to control the appearance of dimensions according to the adopted drawing standards.

To create a Dimension Style,

1. Click the Dimension Style icon  to bring out the Dimension Style Manager. Located under Ribbon: **Home tab > Annotation panel > more**

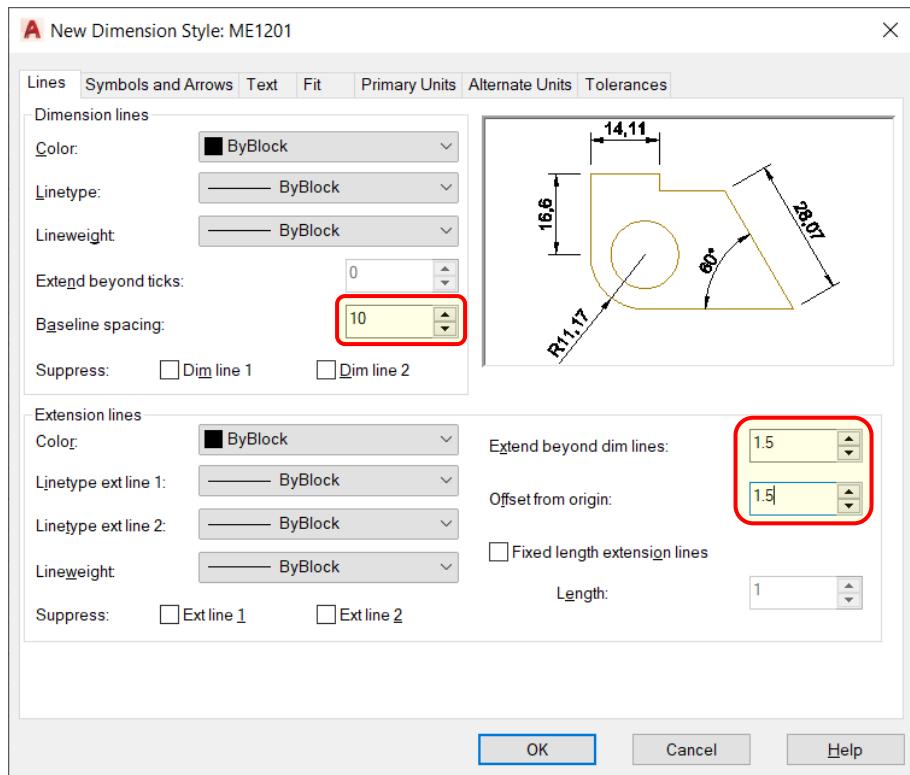


2. In the Dimension Style Manager dialog box, click “New”. The “Create New Dimension Style” dialog box is displayed. Enter the name of the new dimension style in the “New Style Name” field. Select the dimension style to be adopted from by the new style from the “Start With:” dropdown. Click “Continue” to proceed to configure the style.

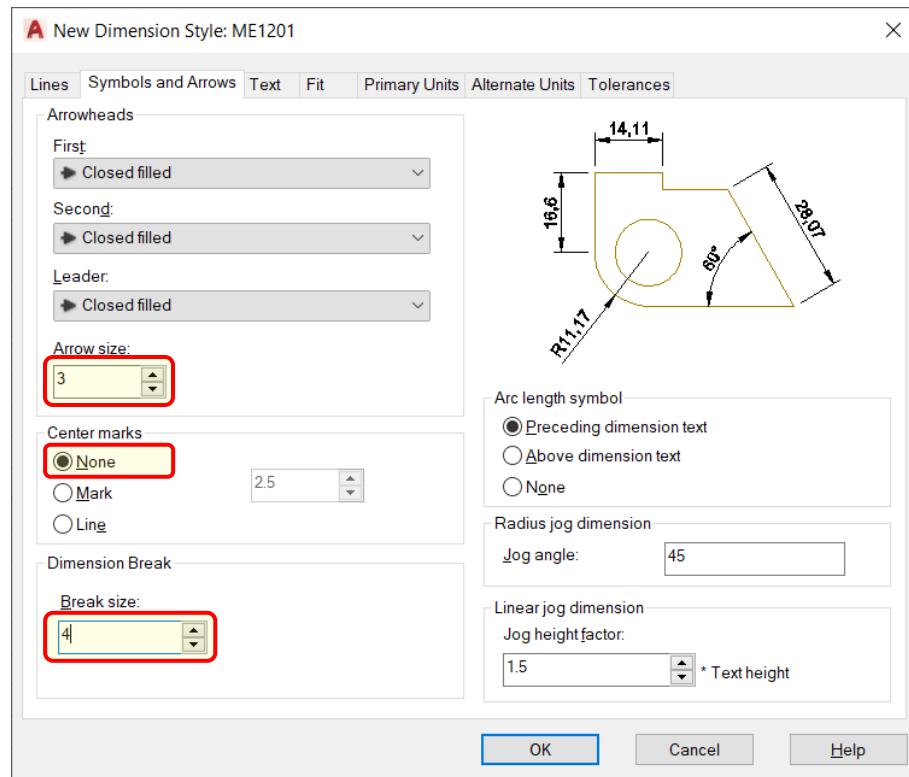


3. The “New Dimension Style” dialog box is displayed:

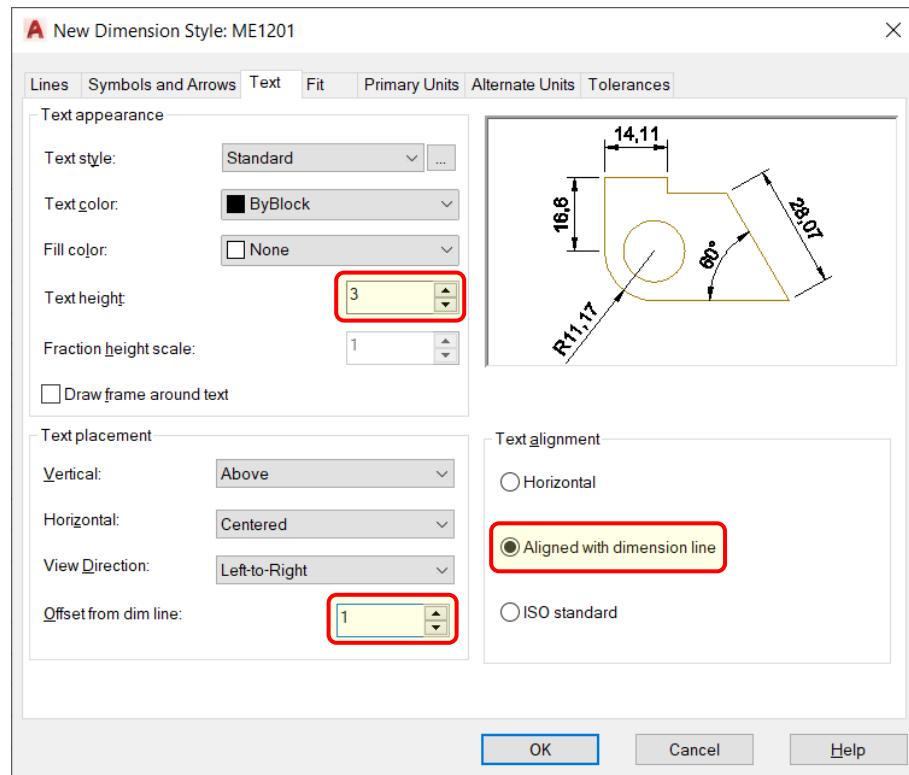
- a. Set the properties of dimension lines and extension lines in the “Lines” tab.



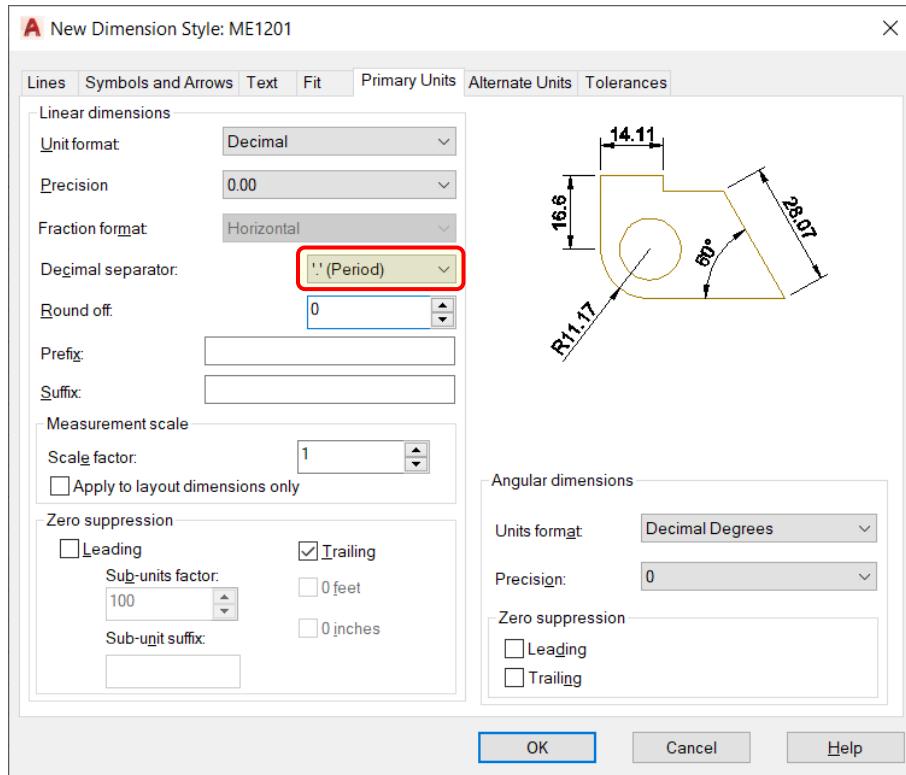
- b. Set the properties of arrowheads, center marks, break size in the “Symbols and Arrows” tab.



- c. Set the appearance, placement and alignment of text in the “Text” tab.



- d. Set the properties of dimension units in the “Primary Units” tab.



- e. Click the “OK” button to accept the settings and exit. Double-click on the selected dimension style in the Dimension Style Manager to set it as current dimension style.

6.15 Special Characters

Special characters can be added to the text by including control information (a pair of percent signs) in the text string. The most common symbols and their codes are listed in the following table:

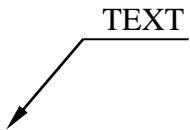
Code	<code>%%c</code>	<code>%%d</code>	<code>%%p</code>	<code>%%</code>
Symbol	\emptyset	$^\circ$	\pm	$\%$

Many standard symbols can be added in the In-Place text editor by clicking on the symbol icon “@” or right-click and locate the symbol section.

6.16 Leader Text

Leader and Leader text can be created using the *QLeader*. Type “*ql*” or “*le*” to invoke the command. Use the Leader Settings dialog box to customise the command so that it prompts for the number of leader points and the annotation type suited to the drawing needs

To create the below leader text:

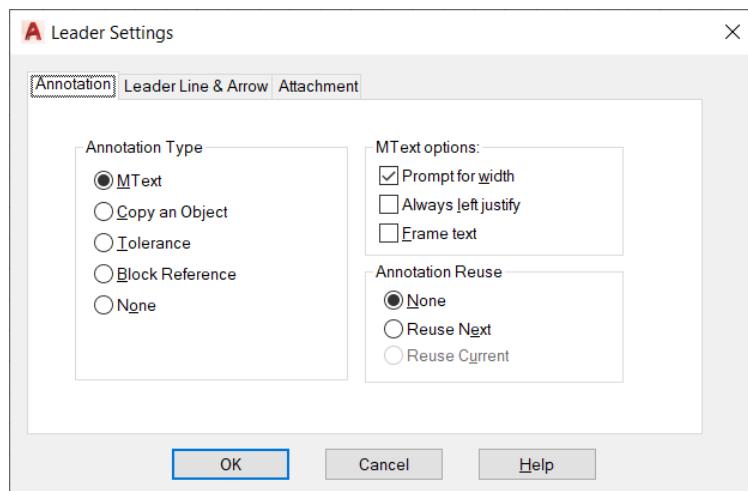


1. Type “ql” or “le” to begin.
2. Hit Enter to show the Leader Settings dialog box.

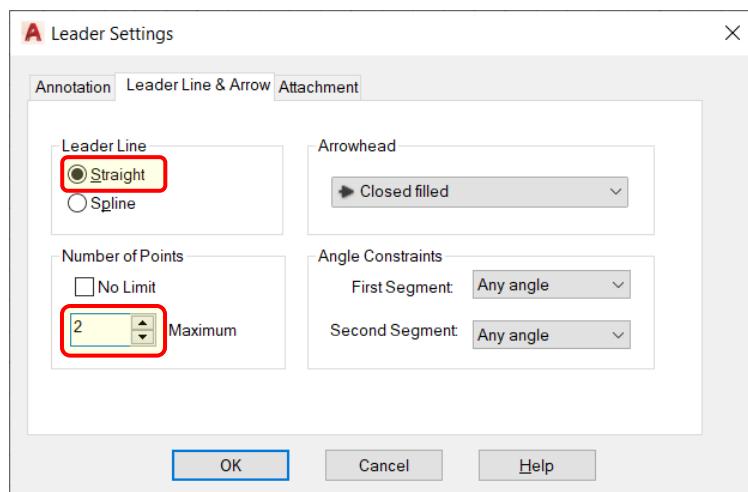
Command: QLEADER

Specify first leader point, or [Settings] <Settings>: (press *Enter*)

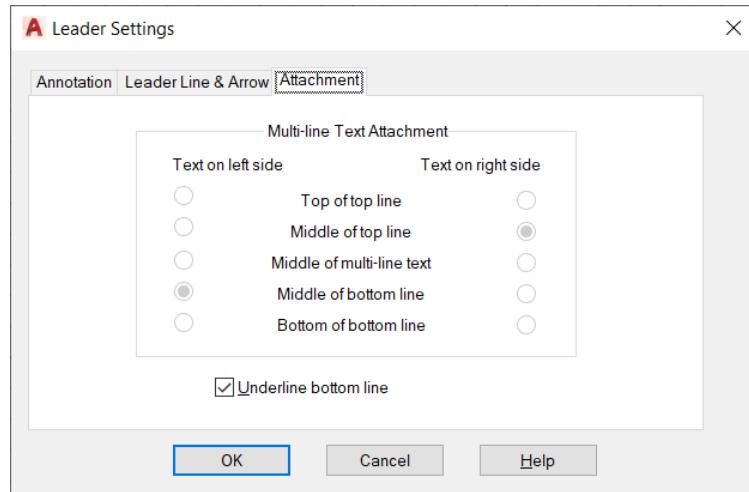
- a. Annotation Tab: Select the Annotation Type, i.e. MText for Leader with Text or None for Leader without Text.



- b. Leader Line & Arrow Tab: Leader Line is “Straight” by default. Number of Points set to 2 Maximum for 2 leader points.



- c. Attachment Tab: Check “Underline bottom line” to have the text above the leader line.



Command: QLEADER (continue)

Specify first leader point, or [Settings] <Settings>: pick leader's start point

Specify next point: pick leader's end point

Specify text width <0>: (press *Enter*)

Enter first line of annotation text <Mext>: (Type in the first line of leader text)

Enter next line of annotation text: (Type in the next line of leader text or Enter to end)

Tutorial 6

1. Open the file, **T6Q1.dwg** from P:\MAE\ME1201\6 Dimensioning. Create a Dimension Style as suggested by para 6.14. Reproduce a fully dimensioned drawing for the below parts.

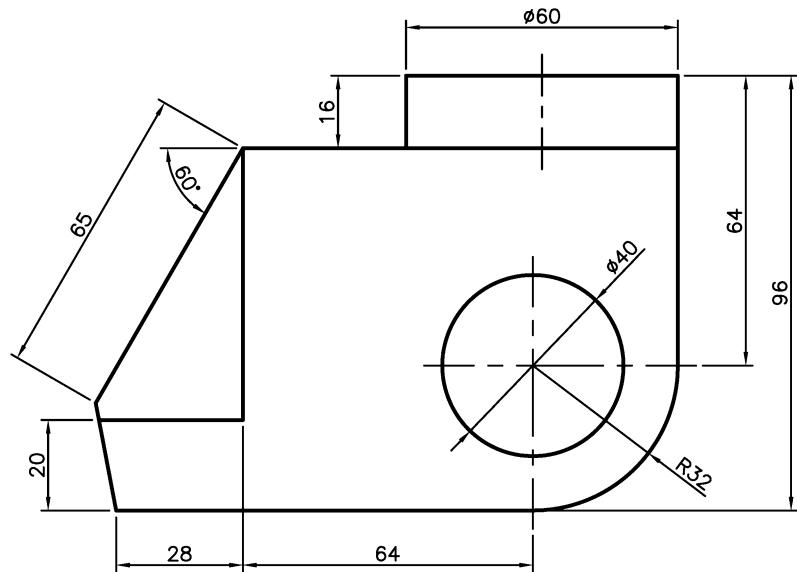


Figure 6-Q1a

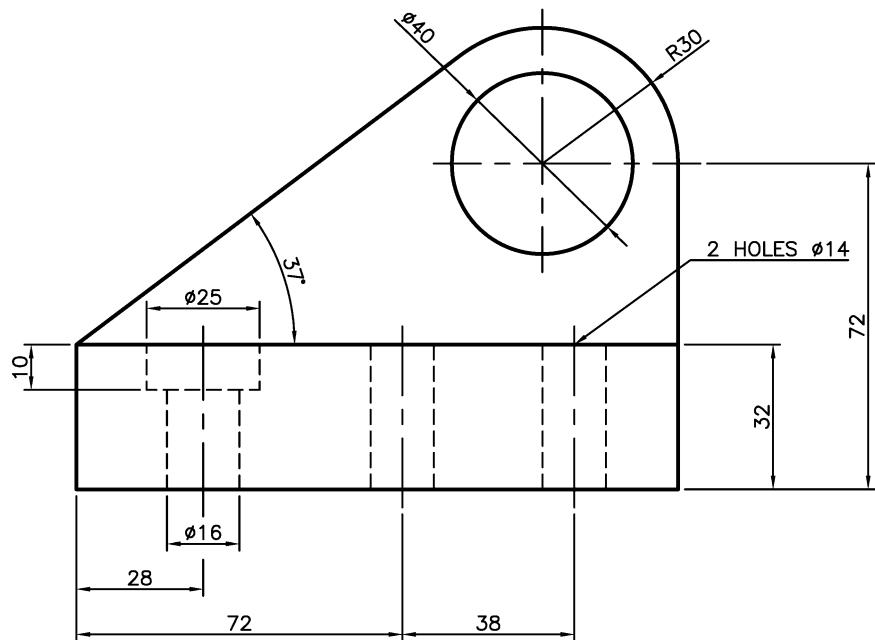
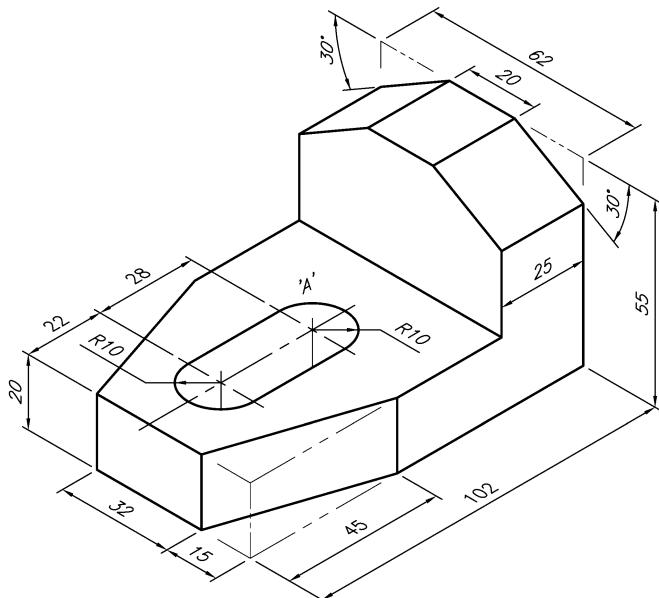


Figure 6-Q1b

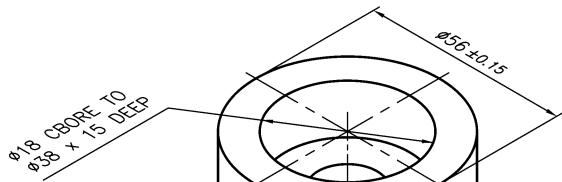
2. Relevant views for the below component figures were prepared. Start from a New file, select the respective template files, **T6Q2a.dwt**, **T6Q2b.dwt** and **T6Q2c.dwt** from **P:\MAE\ME1201\6 Dimensioning**, insert all the necessary dimensions on to their respective views to communicate their feature sizes. Also include relevant cutting plane for the given sectional view. Create a Dimension Style as suggested by para 6.14 for the dimensioning.



Note:

Slot 'A' shall be dimensioned by its length and width.

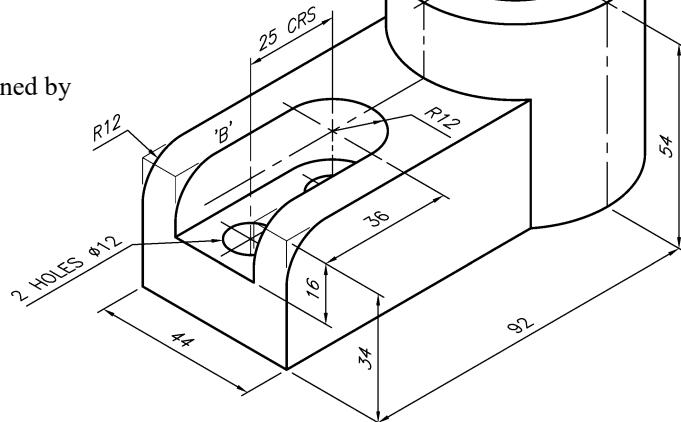
Figure 6-Q2a



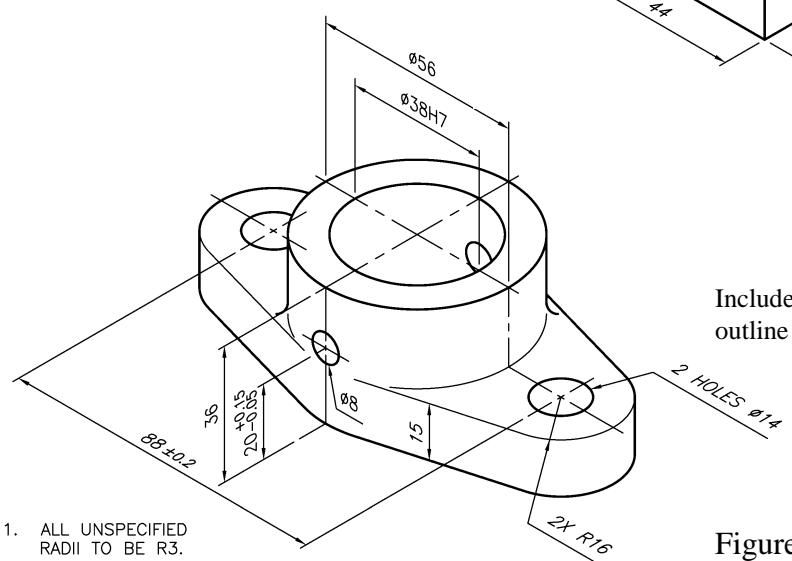
Note:

Open slot 'B' shall be dimensioned by its length and width.

Figure 6-Q2b



Include a local section to show the outline of one Ø8 hole.



1. ALL UNSPECIFIED RADII TO BE R3.

NOTES:

Figure 6-Q2c

3. Relevant views for the below component, Support Frame were prepared. Start from a New file, select the template files, **T6Q3.dwt** from P:\MAE\ME1201\6 Dimensioning:

- a. Update the Front View to a Sectional Front View due to cutting plane A-A, including relevant cutting plane.
- b. Create a Dimension Style as suggested by para 6.14 for the dimensioning.
- c. Insert all the necessary dimensions on to their respective views to communicate their feature sizes.

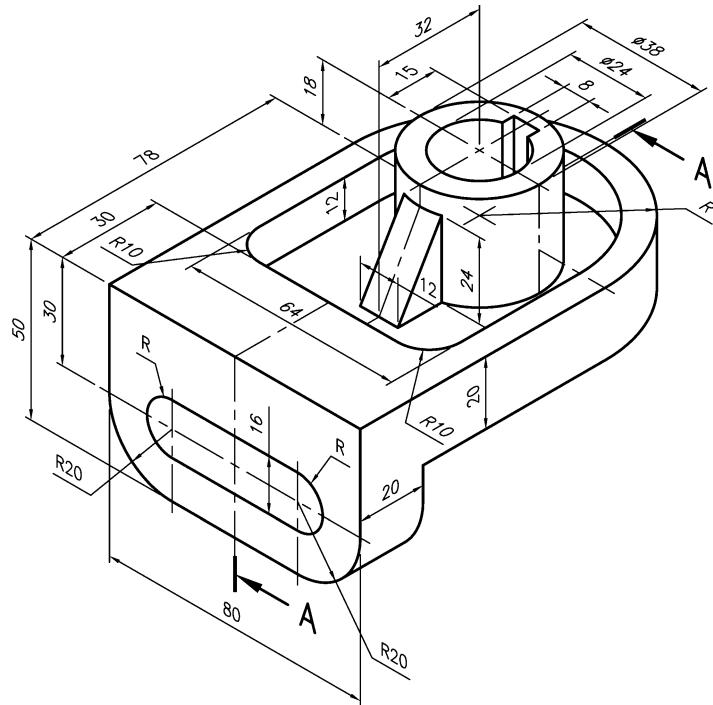


Figure 6-Q3

BLANK

UNIT 7 PARAMETRIC SOLID MODELLING USING AUTODESK INVENTOR

Learning Objectives

By the end of this unit, students should be able to:

- Explain the concept of feature based, parametric solid modelling.
- Recognise the user interface of Autodesk Inventor.
- Recognise the file types used in Autodesk Inventor: (ipt; iam; idw; ipn...).
- Plan the process of part model creation.
- Create fully constrained 2D sketch geometry of a solid feature using a range of sketch and editing tools.
- Create solid features using Extrude, Revolve, Hole, Rib, Fillet and Chamfer tools.
- Select appropriate default work plane and feature's surface to facilitate the creation of solid features.
- Create work planes, axes and points to support necessary part model creation.
- Create duplicate solid features using Pattern and Mirror tools.

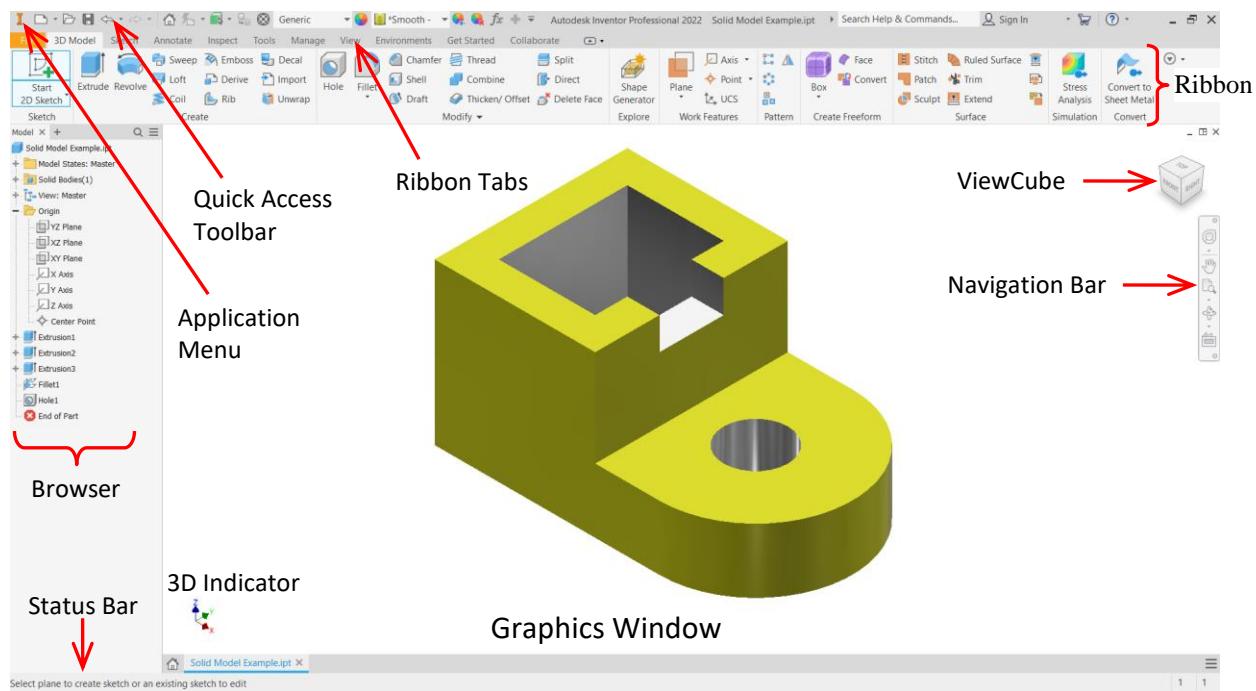
7.1 Basic Concept of Parametric Solid Modelling

A part model is an unambiguous representation of the geometry and topology of a part. It can thus provide physical data, such as volume, mass, centre of gravity, moments of inertia, etc about the part. Parametric solid modelling involves the use of parameters to construct a collection of features or solid bodies that define a part. These parameters include driving dimensions and geometric relationships. The parameters control the size and shape of a model. When a parameter is changed, the model updates to reflect the changes.

Features can be classified into two main types, namely sketched features and placed features. A sketched feature requires a 2-D sketch that is then transformed into a feature in one of four main ways: Extrude, Revolve, Sweep and Loft. Placed features are applied directly to the model and do not require a sketch. Fillet, Chamfer, Draft and Shell are examples of placed features.

7.2 User Interface of Autodesk Inventor

The Autodesk Inventor User Interface in the part modelling environment is shown below. In the part modelling environment, 3D part models are being created and edited. The User Interface is context-sensitive based on the environment being used.



7.2.1 Ribbon

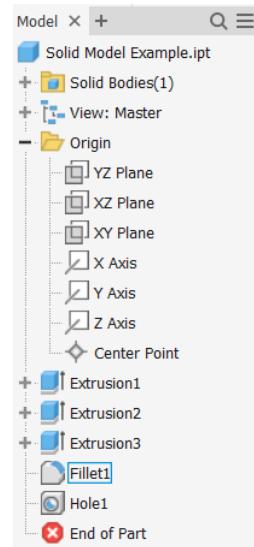
The ribbon is the primary interface for accessing the required tools to complete various tasks. It is context-sensitive and will present the relevant tools based on the current context of the task, such as part modelling, assembly modelling, drawing, etc.

7.2.2 Browser

The browser is also context-sensitive display information specific to the respective environment, such as Part Modelling, Assembly Modelling, Drawing, etc.

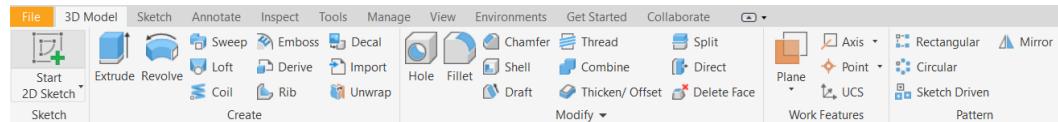
For part modelling, the browser displays all features that are used to create the part model. The features are listed in the order in which they are created. The browser also displays the Origin folder at the top of the list which contains the three default planes (XY, YZ, and XZ), three default axes (X, Y, and Z) and centre point (origin).

The default planes can be used to create 2D sketches.



7.2.3 3D Model Tab

In part modelling environment, the 3D Model tab is displayed when creating and editing part models. Tools available in the 3D Model tab are used to create parametric features on the part.



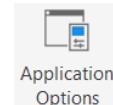
7.2.4 Sketch Tab

The Sketch tab in the modelling environments contains tools used to create 2D parametric sketches, dimensions, and constraints.



7.3 Inventor Application Options Settings

Ribbon access: *Tools tab > Options panel > Application Options*



Autodesk Inventor work environment can be customised to a set of user preferences via the various settings in the Application Options:

General tab: Sets the behavioural options of Autodesk Inventor operations.

Sketch tab: Sets the preferences for sketching.

Part tab: Sets the defaults for new part creation.

Drawing tab: Sets the options for working with drawings.

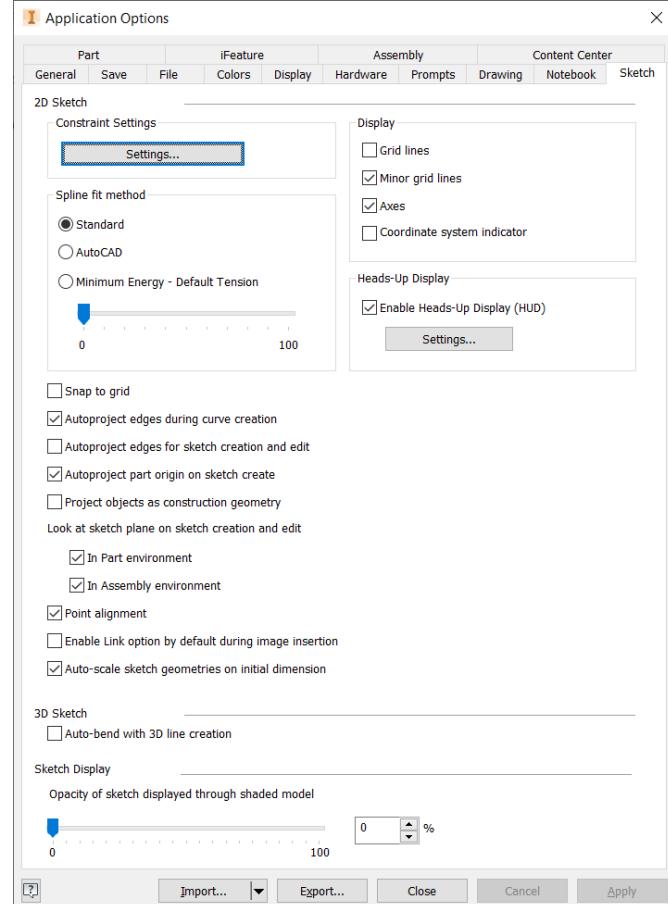
Assembly tab: Sets the preferences for working with assemblies.

Content Center tab: Sets the preferences for using Content Center.

Files tab: Sets the location of files that Autodesk Inventor uses for various functions.

Colors tab: Sets the colour scheme and background appearance of graphics window.

Display tab: Sets the model display parameters.

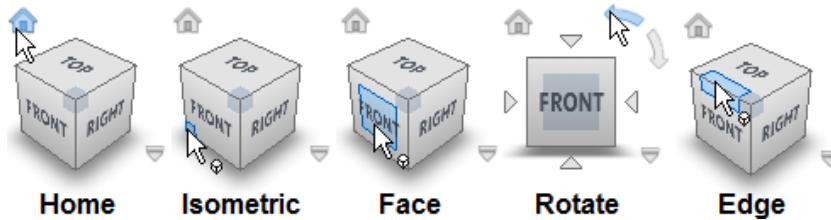


7.4 Viewing and Navigation Tools

All aspects of the Part and assembly models can be viewed by navigating around in 3D space. The viewing and navigation tools enable such viewing to be performed quickly.

7.4.1 ViewCube

The ViewCube tool is an interface that can be used to switch between standard and isometric views of the model, roll the current view, or change to the Home view of the model:



Home View: Place the cursor over the “Home” icon in the ViewCube to return to the default home view.

Isometric: Orientate to the isometric view by clicking one of the corresponding corners on the ViewCube.

Face: Orientate to one of the six standard orthogonal views of a model: Top, Bottom, Front, Back, Left and Right by clicking the corresponding Face on the ViewCube.

Roll a Face View: When a model is displayed to one of the face views, two roll arrows are displayed near the ViewCube. This view can be rotated 90° clockwise or counter-clockwise about the centre of the view using the roll arrows.

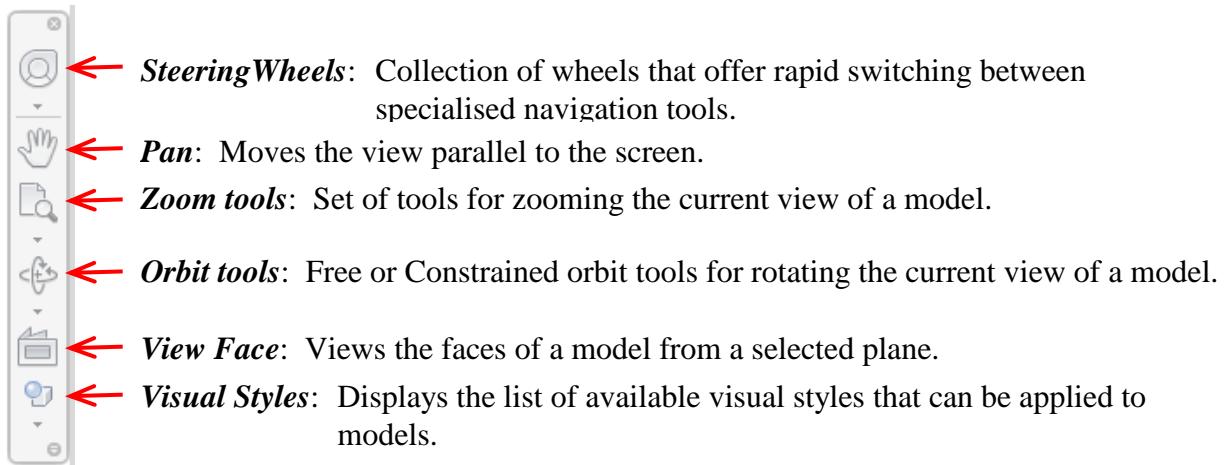
Switch to Adjacent Face: Switch to one of the adjacent face views using the four orthogonal triangles displayed near the ViewCube.

Edge: Orientate the viewpoint between two adjacent orthogonal views by clicking their shared edge on the ViewCube.

Free Orbit: Orbit the model by clicking the ViewCube, hold down the left mouse button, and drag it along the desired orbiting direction.

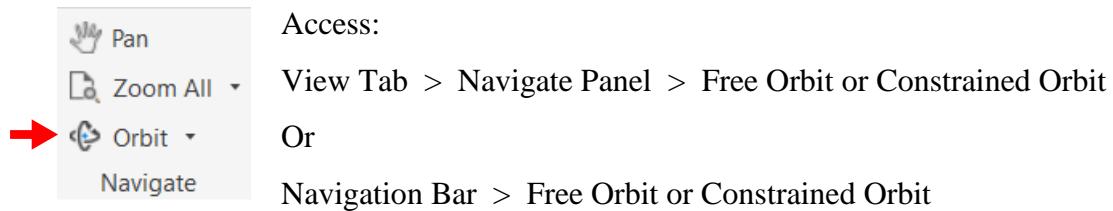
7.4.2 Navigation Bar

The navigation bar is displayed by default at the upper right in the graphics window. Viewing, Navigation and Visual Styles (upon adding) tools can be accessed from the navigation bar.



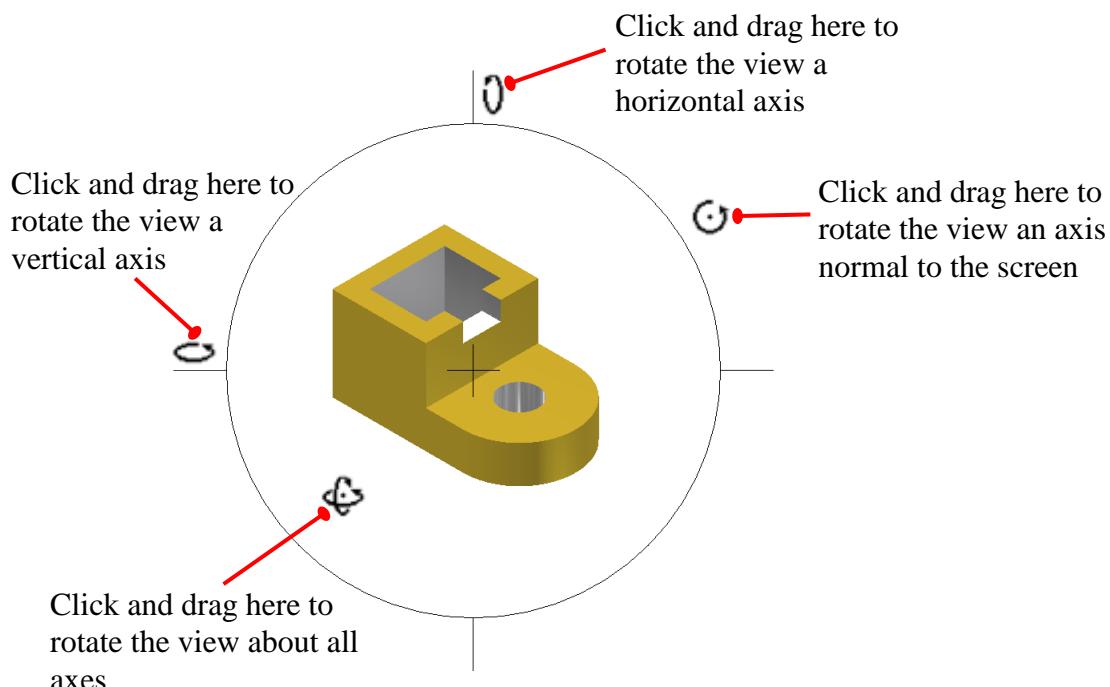
7.4.3 Orbit Tools

There are two options to rotate the views of models and assemblies: Free Orbit or Constrained Orbit.



Free Orbit:

The Free Orbit tool rotates the model freely in screen space. The view is rotated by clicking and dragging the cursor. Depending on the location of the cursor around the “circle”, the appearance of the cursor will change and the view can be rotated accordingly as follows:



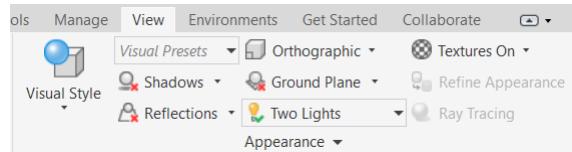
The Free Orbit tool can also be invoked with “F4” function key or mouse wheel (Shift + click and drag the mouse wheel).

Constrained Orbit:

The Constrained Orbit tool rotates the model around its vertical axis in a manner similar to the rotation of a turntable.

7.5 Appearance Options

The appearance of the 3D model can be changed via the **View Tab > Appearance Panel:**



Visual Style: Set the appearance of model faces and edges in one of the ten styles: Realistic, Shaded, Shaded with Edges, Shaded with Hidden Edges, Wireframe, Wireframe with Hidden Edges, Wireframe with Visible Edges Only, Monochrome, Watercolor and Illustration.

Shadows: Display model with various shadow options: Ground, Object, Ambient or All Shadows.

Reflections: Display model with Reflections.

Lights: Select or set the different lighting options.

Orthographic: Set the viewpoint to be Orthographic or Perspective.

Color: Sets the colour of current part. The colour setting is also available in the Quick Access toolbar.

7.6 Inventor File Types

Autodesk Inventor uses different file types for part model, assembly model, drawing, etc:



Project files (*.ipj) are XML files that contain search paths to locations of all the files in the project. The search paths are used to find the files in a project.



Part files (*.ipt) are used to maintain individual parts.



Assembly files (*.iam) are used to assemble multiple part files to represent an assembly. The assembly file contains references to all of its part files.



Presentation files (*.ipn) are used to create exploded views of the assembly.
Presentation file is associated to its assembly.



Drawing files (*.idw) are used to create 2D documentation, i.e. views, dimensions and annotations of parts and assemblies. Drawing file is associated to its parts and assemblies.



Inventor drawing files can also be stored in the standard DWG format. This format of 2D drawings can be opened and saved in AutoCAD.

7.7 Shortcut Keys

Autodesk Inventor has many predefined shortcut keys and command aliases. The following is a list of some of the more commonly used shortcuts and command aliases predefined in Autodesk Inventor 2012:

S/N	Key	Result
1	Esc	Quit a command.
2	Delete	Delete selected objects.
3	F1	Help for the active command or dialog box.
4	F2 + LMC*	Pan the graphics window.
5	F3 + LMC*	Zoom In/Out in the graphics window.
6	F4 + LMC*	Free Orbit.
7	F5	Previous View.
8	F6	Home View.
9	F7	Slice Graphics in Sketch environment.
10	F8	Show all constraints in Sketch environment.
11	F9	Hide all constraints in Sketch environment.

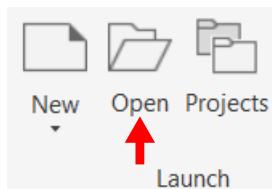
S/N	Key	Result
12	A	Create a centre point arc in a sketch environment.
13	B	Add a balloon to a drawing.
14	C	Add an assembly constraint in an assembly environment.
15	D	Add a dimension to a sketch or drawing.
16	E	Extrude a profile.
17	F	Add Fillet feature.
18	H	Add a Hole feature.
19	L	Create a line or arc in a sketch environment.
20	P	Place a component in an assembly environment.
21	R	Revolve a profile.
22	S	Create a 2D sketch on a face or plane.

LMC*: Left Mouse Click.

The **MOUSE WHEEL** can also perform **ZOOM**, **PAN** and Orbit in the following ways:

- a. Roll the wheel button forward : Zoom Out
- b. Roll the wheel button backward : Zoom In
- c. Double-click the wheel button : Zoom All
- d. Hold down the wheel button and drag the mouse : Pan
- e. Hold down SHIFT + wheel button : Orbit

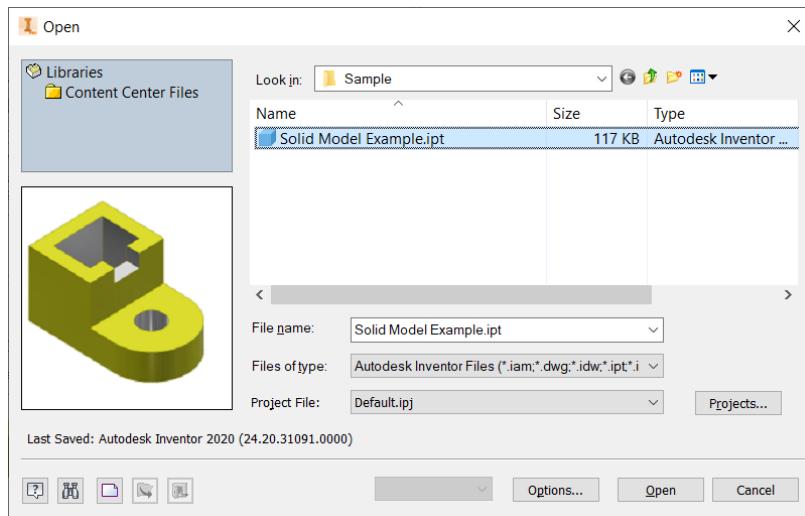
7.8 Open Files



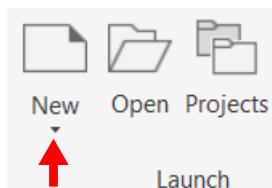
Access: ***Get Started Tab > Launch Panel > Open***
Quick Access Toolbar > Open

Shortcut: ***Ctrl + O***

The **Open Files** tool opens selected Autodesk Inventor files. The Open dialog box will appear for selecting the Autodesk Inventor files to open. The default “Look in” folder is defined in the respective Project File. It is not recommended to open files outside the locations defined in the current project file as part, drawing and assembly relationships may not be resolved or resolved incorrectly.



7.9 New Files

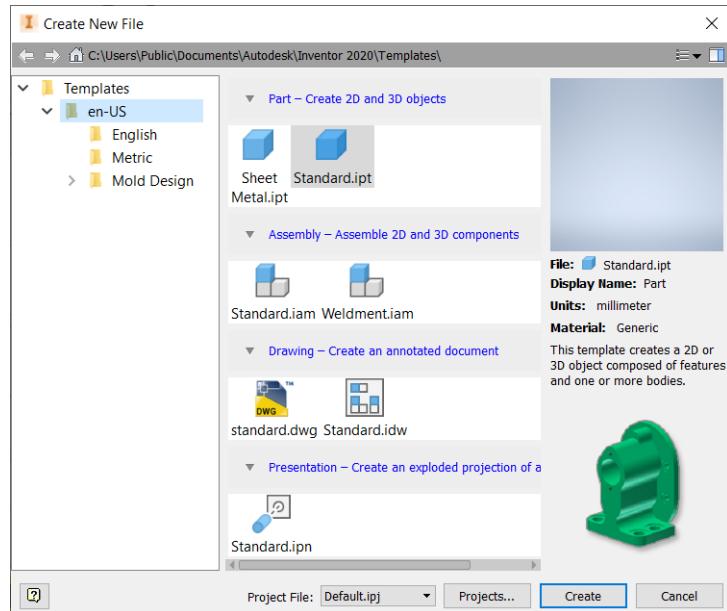


Access: ***Get Started Tab > Launch Panel > New***
Quick Access Toolbar > New

Shortcut: ***Ctrl + N***

The **New Files** tool creates a new Autodesk Inventor file. The New File dialog box will appear for selecting a template for the type of new Autodesk Inventor file to be created: Part, Assembly, Presentation, Sheet Metal or Drawing.

The Template has these folders: en-US, English, and Metric. The en-US folder presents template files based on the selection during Configure Default Templates, while the English and Metric folder presents template files in their respective units, i.e. inches or millimetre.



7.10 Parametric Part Modelling

Parametric part model is controlled and driven by geometric relationships and dimensional values. It is created progressively from a combination of 2D sketched features and placed features. Each feature is based upon the previous feature. When a dimensional value of a feature is changed, the part model will be adjusted according to that value and existing geometric constraints.

7.10.1 Base Feature

The first feature that is created is typically a sketch feature which is also referred to as the base feature. It is created from a 2D sketch on a sketching plane. Examples are extruded and revolved features. All subsequent features either add material to or remove material from the part model.

7.10.2 Sketched Features

Sketch features are features that add or remove material and are generated from a 2D closed loop sketch. Once the sketch is used by a feature, it is considered consumed by the feature and nested below that feature unless it is being shared.

7.10.3 Placed Features

Placed features have internally defined shape for adding or removing material. Such features only require their locations and sizes to be defined. Holes, fillets and chamfers are examples of placed features.

7.11 Design Intent for Parametric Part Modelling

Design intent is about capturing intelligence in the part model using dimensional and geometric relationships that define the fit and function of the part. Capturing design intent is a process in which the design intent is matched with a feature or capability that makes it possible to create the design in the most efficient way while enabling maximum flexibility in making changes whenever required.

The following guidelines for capturing design intent can be considered when starting a new part design:

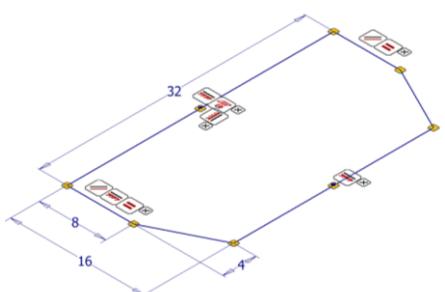
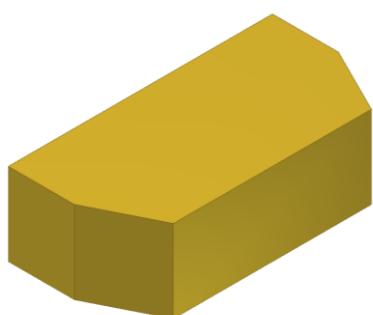
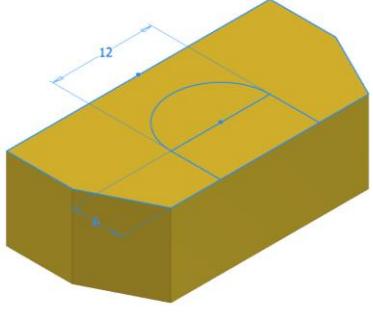
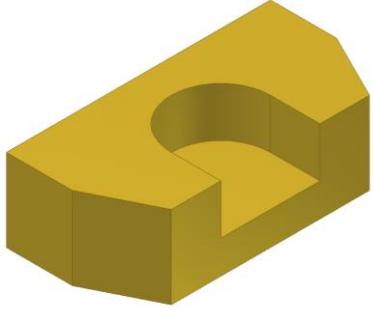
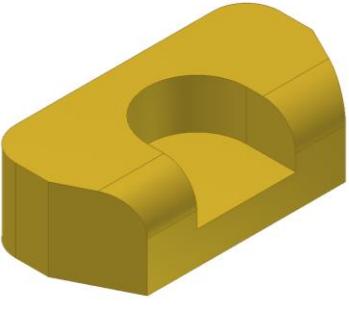
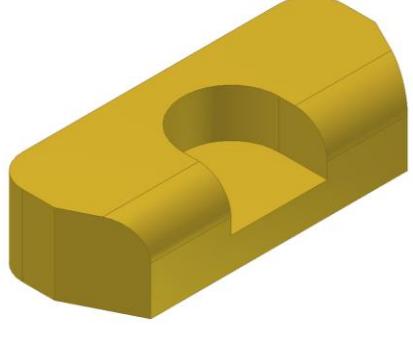
1. Identify geometric relationships. For example, a feature's length may be directly related to its width, or the width or length of another feature.
2. Identify areas of the design that may be prone to change as a result of design problems or revisions.
3. Identify areas of symmetry or areas where features are duplicated or patterned.

Once the potential ways to capture the design intent of a part had been identified, the intent can then be matched with a specific Autodesk Inventor tool or capability.

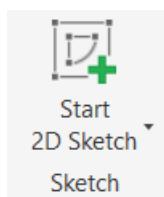
7.12 Part Design Workflow

The steps representing the workflow for creating the part models are as follows:

1. Determine the basic building blocks of the part model.
2. Use one of the part templates provided to create a new part.
3. Select a relevant workplane for the base sketch if the default sketch (on XY workplane) is not appropriate.
4. Create the profile of the base feature, capturing the design intents by applying geometric constraints and dimensions.
5. Use sketched features such as Extrude and Revolve to create the base feature.
6. Create additional sketched and placed features as required to complete the part model.

<p>1. Initial sketch is created. Geometric constraints and dimensions are applied.</p> 	<p>2. Base feature is created:</p> 
<p>3. Secondary sketch is added:</p> 	<p>4. Secondary feature is created from secondary sketch:</p> 
<p>5. Fillets (Placed feature) are added:</p> 	<p>6. Length is changed in initial sketch, causing part to update:</p> 

7.13 2D Sketch

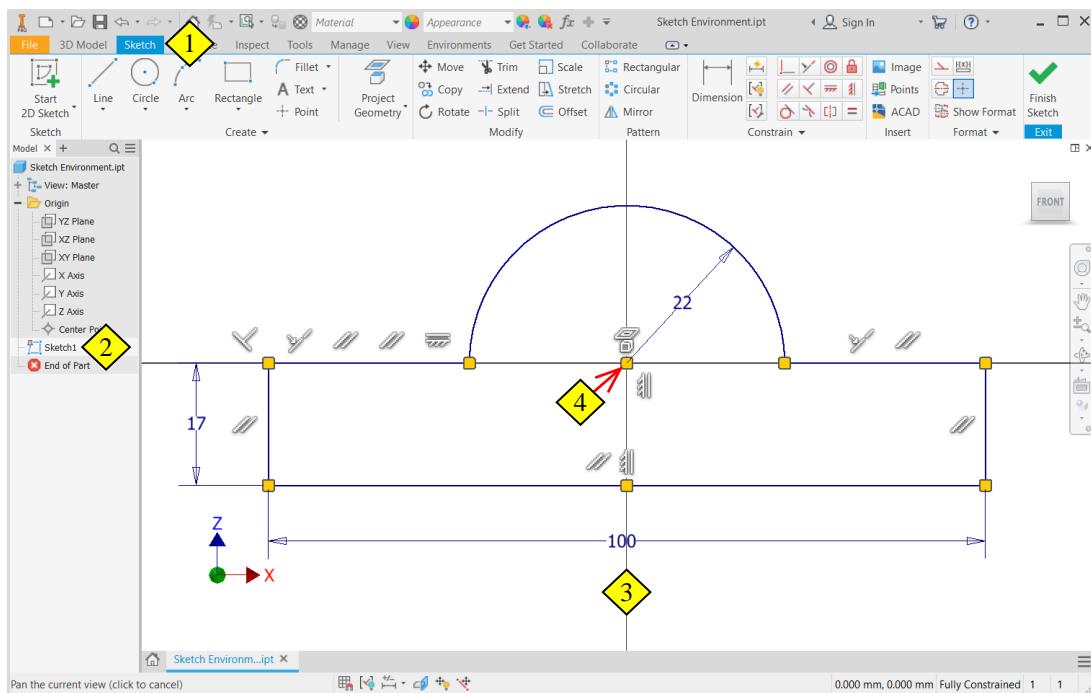


Access: **3D Model Tab > Sketch Panel > Create 2D Sketch**

Shortcut: **S**

The **2D Sketch** tool creates a 2D sketch on a planar face or work plane on a part, or on a work plane on an assembly. It also enables existing sketch to be edited when selected. The sketch environment is activated as shown with the following:

1. The Sketch tab is displayed when the sketch environment is active. It contains all the tools for creating, manipulating and controlling sketch geometry.
2. The active sketch is highlighted in the browser.
3. The X axis and Y axis are displayed by default in the graphics window.
4. The part origin is projected on to the active sketch by default.



An active sketch must be available to begin drawing a profile or path sketch. A sketch is a plane on which 2D objects are sketched. By default, the first sketch in a new part is automatically created on the XY plane.



Access: **Sketch Tab > Exit Panel > Finish Sketch** or

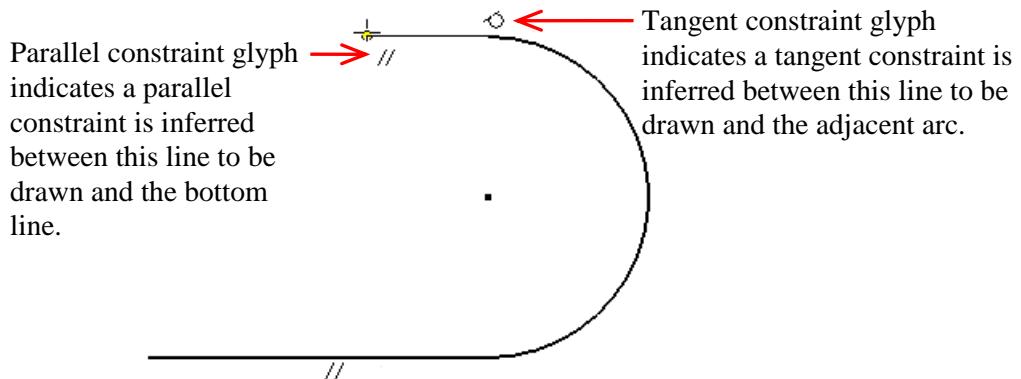
Shortcut: **S** (within the Sketch environment)

7.13.1 Constraints in Parametric Sketches

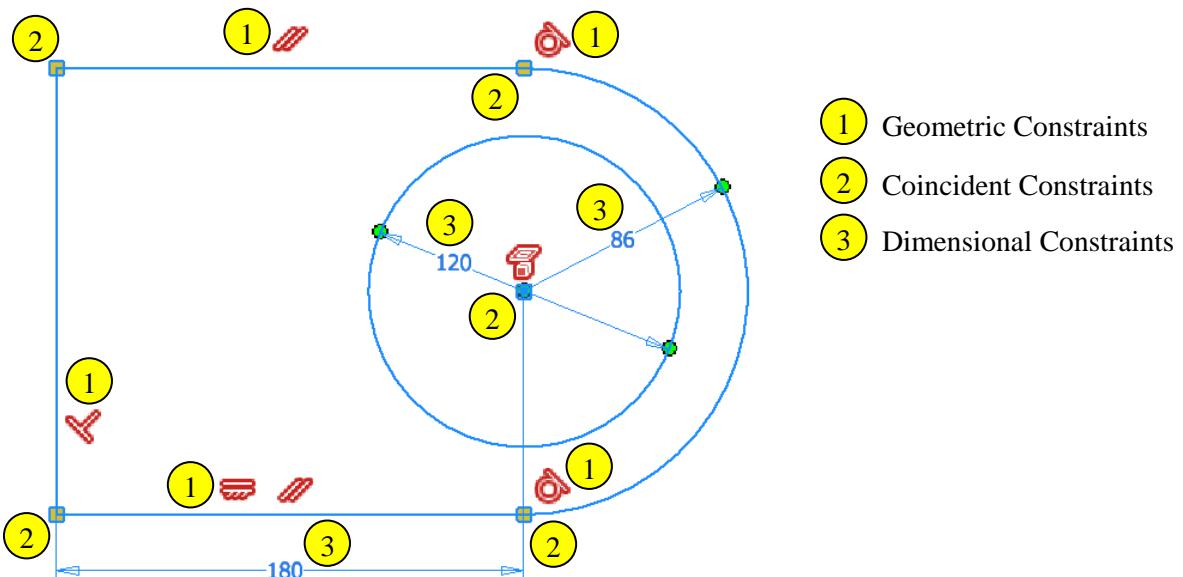
Parametric sketches are the base of each parametric part created in Autodesk Inventor. When creating a sketch, intelligence is added to the part to capture the design intent.

A parametric sketch consists of 2D geometry on which constraints are applied to control the size and potential behaviour of the 2D geometry. There are two different types of constraints: geometric constraints and dimensional constraints. Some geometric constraints are applied automatically (inferred) as geometries are created. In the below illustration, the symbols next to the geometry are known as "glyphs" and represent 2D constraints being inferred on which can be applied to the related geometries. Glyphs are displayed during sketching while a

sketch tool is active. The use of 2D constraints is one way in which design intent is automatically captured as sketch geometries are created.



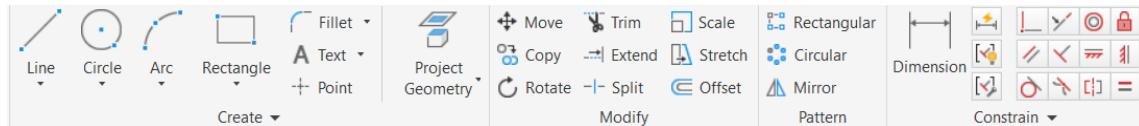
Dimensional constraints are added to each element of the sketch for which a dimension is needed to control the size of this element. Both geometric constraints and dimensional constraints applied to sketch geometry are as shown:

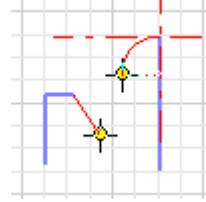


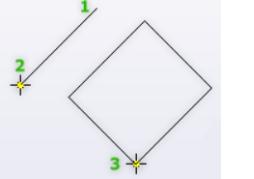
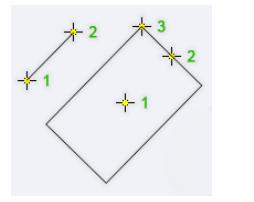
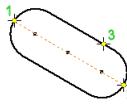
When applying constraints to a sketch, each constraint removes degrees of freedom from the geometry. By removing degrees of freedom, it will limit the direction or amount a given part of the sketch can be moved or resized. When a sketch has all degrees of freedom removed, it is considered to be fully constrained. While it is not necessary to fully constrain a sketch before creating 3D features, it is recommended. A fully constrained sketch is predictable in the manner in which it can change, and reduces the number of errors as changes are made to the parametric part. Once the sketch is fully constrained, the profile will be a single colour. Inventor uses colour differences and numerical feedback to identify fully constrained as opposed to under constrained geometry. The colours can be used to identify the elements that still require constraints. At the bottom right of the Inventor window, the application will indicate whether the sketch is fully constrained or the required number of dimensions to be fully constrained (see above figure). Colours used to show constraint conditions vary depending on the colour configuration for Inventor.

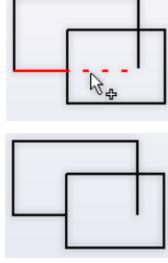
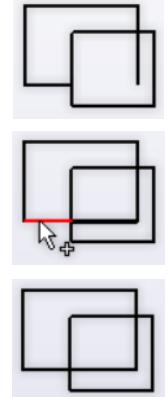
7.13.2 Essential Sketch Tools

Autodesk Inventor contains similar tools as AutoCAD for creating 2D sketch geometry organised in the Create, Modify and Pattern panels:



Tool	Icon	Function
Line		<p>Creates line segments and tangent or perpendicular arcs to geometry. To create a tangent or perpendicular arc, click and hold the end of a line or arc, then drag to preview the arc, and release the mouse button to end the arc.</p> <p>Line segments and arcs are individual curves whose endpoints are joined by a coincident constraint.</p> 
Circle		Creates a circle using a centre point and radius.
		Creates a circle tangent to three geometric elements.
Arc		Creates an arc defined by two endpoints and a point on the arc. The first two points set the ends of the arc and the third point sets the arc direction and radius.
		Creates an arc defined by its centre point and two endpoints. The first point sets the centre point, the second specifies the radius and start point, and the third point completes the arc.
		Creates an arc from the endpoint of an existing curve. The first point (on the endpoint of the curve) sets the tangent endpoint. The second point sets the end of the tangent arc.

Tool	Icon	Function
Rectangle	 (Two Point)	Creates a rectangle using two points for the diagonal corners to set the length and width. 
	 (Three Point)	Creates a rectangle using three points to set the length, direction and then the adjacent length. 
	 (Two Point Center)	Creates a rectangle using two points to define centre, width and length. 
	 (Three Point Center)	Creates a rectangle using three points to define centre, direction and then adjacent side. 
Slot	 (Center-Center)	Creates a linear slot defined by placement and distance of slot arc centres, and by slot width. 
	 (Overall)	Creates a linear slot defined by orientation, length, and width. 
	 (Center Point)	Creates a liner slot defined by a centre point, location of slot arc centres, and by slot width. 
	 (Three Point Arc)	Creates an arc slot defined by a three point centre arc and slot width. 
	 (Center Point Arc)	Creates an arc slot defined by a centre point, two point centre arc, and slot width. 
Ellipse		Creates an ellipse using a centre point, a major axis and a minor axis. The first point sets the centre point, the second sets the direction and length of the first axis, and the third point can be any point on the ellipse.
Polygon		Creates an inscribed or circumscribed sketched polygon with up to 120 edges.

Tool	Icon	Function
Move		<p>Moves selected sketch geometry from one point to another as specified, with an option to copy.</p> <p>Move results may be impacted by constraints shared between the selected geometry and unselected geometry. Options are available to set overrides for dimensional and geometric constraints.</p>
Copy		Copies selected sketch geometry and places one or more instances in the sketch.
Rotate		<p>Rotates selected sketch geometry or a copy of it relative to a specified centre point.</p> <p>Rotate results may be impacted by constraints shared between the selected geometry and unselected geometry. Options are available to set overrides for dimensional and geometric constraints.</p>
Trim		<p>Trims a curve to the nearest intersecting curves or selected boundary geometry. A coincident constraint is created between the endpoint of the trimmed curve and the boundary curves.</p> <p>Curves can be trimmed with or without defining boundary geometry. Hold down the CTRL key to select the boundary geometry and release when complete.</p> <p>Pause the cursor over the curve to preview the trim.</p> <p>Hold down the Shift key to switch temporarily to the Extend operation.</p> 
Extend		<p>Extends a curve to the nearest intersecting curve selected boundary geometry. A coincident constraint is created between the endpoint of the extended curve and the boundary curves.</p> <p>Curves can be extended with or without defining boundary geometry. Hold down the CTRL key to select the boundary geometry and release when complete.</p> <p>Pause the cursor over the curve to preview the extension.</p> <p>Hold down the Shift key to switch temporarily to the Trim operation.</p> 

Tool	Icon	Function
Split	—+—	Splits a selected curve to the nearest intersecting curves. Pause the cursor over a curve to preview the split.
Offset	🕒	Duplicates selected sketch geometry and dynamically offsets it from the original. The offset distance can be dimensioned. Loop Select and Constrain Offset are the default settings. Right-click to change.
Rectangular Pattern	□-□	Creates a rectangular array of selected sketch geometry with the specified number rows and columns.
Circular Pattern	○○○	Creates a circular array of selected sketch geometry with the specified number of elements and angular spacing.
Mirror	△△	Creates a mirrored copy of selected sketch geometry about an axis. A symmetry constraint will be applied to the mirrored geometries.

7.13.3 Geometric Constraints

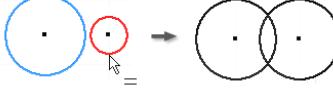


Access: **Sketch Tab > Constrain Panel > ...**

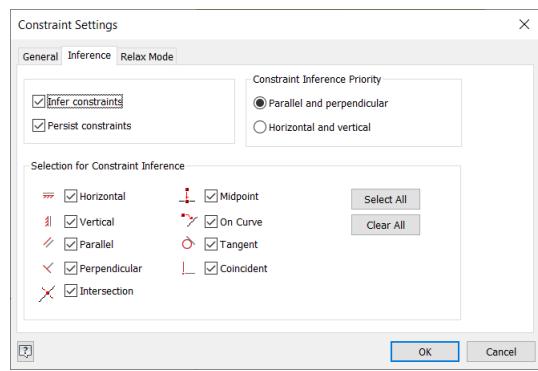
There are 12 types of geometric constraints that can be used to constrain a sketch. Each type of constraint can be applied to certain types of geometry and in certain situations. Some constraints such as perpendicular are relational constraints and must be applied to two elements in the sketch. A relational constraint defines a geometric relationship between two objects. Other constraints such as vertical can be applied to a single object or two points.

Constraint Types	Function	Illustration
Fix	Causes points or curves to be constrained to a fixed location relative to the sketch coordinate system. Projected (reference) geometry cannot be fixed. <i>(Note: Do not use this constraint in ME1201)</i>	

Constraint Types	Function	Illustration
Coincident	Causes two points to be constrained together or one point to a curve. When this constraint is applied to the centre points of two circles, arcs, or ellipses, the result is the same as a concentric constraint. A sketch point is created when a midpoint is constrained with a coincident constraint.	
Collinear	Causes two lines or ellipse axes to lie along the same line.	
Concentric	Causes two arcs, circles, or ellipses to be constrained to the same centre point. The result is the same as a coincident constraint applied to the centre points of the curves.	
Parallel	Causes two or more lines or ellipse axes to be constrained parallel to one another.	
Perpendicular	Causes selected curves or ellipse axes to lie at right angles to one another.	
Horizontal	Causes lines, ellipse axes, or pairs of points to lie parallel to the X axis of the sketch coordinate system. A sketch point is automatically created on a midpoint when it is constrained.	
Vertical	Causes lines, ellipse axes, or pairs of points to lie parallel to the Y axis of the sketch coordinate system. A sketch point is automatically created on a midpoint when it is constrained.	
Tangent	Causes two curves to be tangent to one another, even if they do not physically share a point (in a 2D sketch). Tangency is commonly used to constrain an arc to a line.	
Smooth (G2)	Create a curvature continuous (G2) condition between a spline and another curve, such as a line, arc, or spline.	

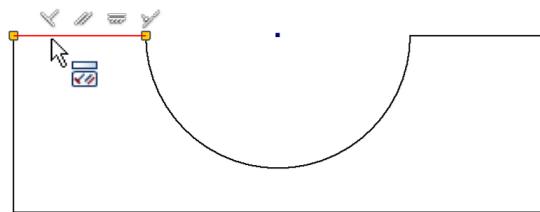
Constraint Types	Function	Illustration
 Symmetric	Causes lines and arcs to become aligned symmetrically about a selected line. Symmetry constraints are added to the selected geometry.	
 Equal	Resizes selected arcs and circles to the same radius, or selected lines to the same length.	

By default, as sketch geometries are created, valid geometric constraints can be applied to these geometries automatically (inferred). The automatic inference of constraints is dependent on the Constraint Settings option (Sketch Tab > Constraint Panel > Constraint Settings .

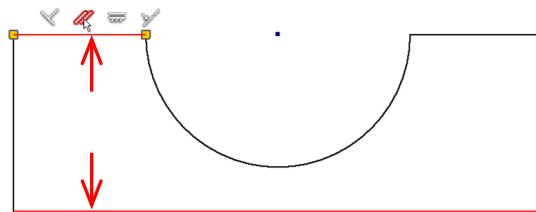


- Infer Constraints. When selected, constraints are inferred to the geometry created with the active command.
- Persist Constraints. Available only if Infer Constraints is selected. When Persist Constraints is selected, constraints are created automatically after finishing a sketch. If Infer Constraints is selected and Persist Constraint is deselected, constraints are not created automatically after finishing a sketch but the Inference position is correct.

The Show Constraints tool (Sketch Panel)  or **F8** is used to display constraint information for the selected sketch geometry: Place the cursor over geometry to preview the constraints. Select the geometry to display the constraints. Place the cursor over a constraint symbol to highlight its associated sketch elements.



Previewing geometry's constraints



Highlighting sketch elements related to the constraint symbol

7.13.4 Dimensional Constraints

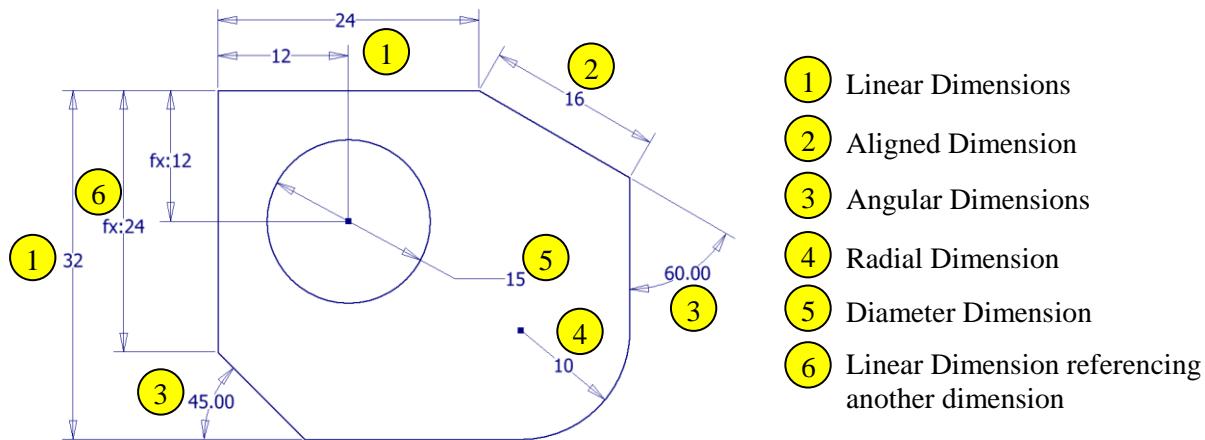


Access: *Sketch Tab > Constrain Panel > Dimension*

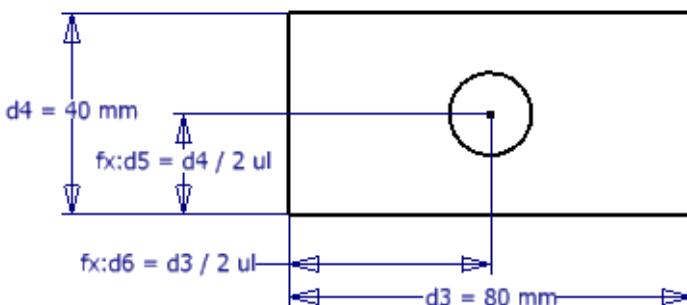
Shortcut: **D** (within the Sketch environment)

While geometric constraints stabilize the sketch and make it predictable, parametric dimensions set the size, angle, or position of the geometry according to the design intent when placed on the sketch geometry.

The General Dimension tool is used to place dimensions on the sketch. Linear, Aligned, Angular, Radial, and Diameter dimensions can be added using this single tool. This tool places the appropriate type of dimension based on the geometry selected. The shortcut menu provides additional options for placing the dimension.



The value of a dimension can be related to an existing dimension by selecting the dimension in the graphics window. A dimension that references another dimension has fx: preceding its value. Mathematical expression can also be incorporated when defining a dimension.



7.13.5 Guidelines for 2D Sketching, Geometric Constraining and Dimensioning Sketches

The following are some guidelines to consider during 2D sketching:

1. Keep the sketch simple: Do not fillet the corners of a sketch if a fillet can be applied to the edges of the finished 3D feature and achieve the same effect. Complex sketch geometry can be difficult to manage as designs evolve.
2. Repeat simple shapes to build more complex shapes.
3. Draw the profile sketch roughly to size and shape.
4. Use 2D constraints to stabilize sketch shape before setting size.
5. Use closed loops for profiles.

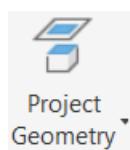
The following are some guidelines to consider when applying geometric constraints to the sketch geometry:

1. **Determine sketch dependencies:** During the sketch creation process, determine how sketch elements relate to each other and apply the appropriate sketch constraints.
2. **Analyse automatically applied constraints:** As the sketch is created, some constraints are automatically applied. After the sketch is completed, determine whether any degrees of freedom remain on the sketch. If required, delete the automatically applied constraints and apply the required constraints to remove the degrees of freedom.
3. **Use only needed constraints:** When constraining the sketch geometry, the design intent and the degrees of freedom remaining on the sketch should be taken into account. It is not necessary to fully constrain sketch geometry in order to create 3D features. In some situations, the sketch geometry may require to be under constrained.
4. **Stabilise shape before size:** The sketch should be constrained first to control how the geometry can change before placing dimensions on the sketch elements. By stabilising the geometry with constraints, the effect of the dimensions on the sketch geometry can be predicted.
5. **Identify sketch elements that might change size:** When constraining sketches, take into account features that may change as the design evolves. When sketch features that may change are identified, leave those features under constrained. When a feature is left unconstrained, the feature can change as the design evolves (adaptive feature).

The following are some guidelines to consider when adding dimensions to the sketch:

1. Use geometric constraints whenever possible. E.g. place a perpendicular constraint instead of adding an angle dimension of 90 degrees.
2. Place large dimensions before small ones.
3. Include relationships between dimensions. E.g. if two dimensions are supposed to be the same value, reference one dimension to the other so that if the first dimension changes, the other dimension changes as well.
4. Consider both dimensional and geometric constraints to meet the overall design intent.

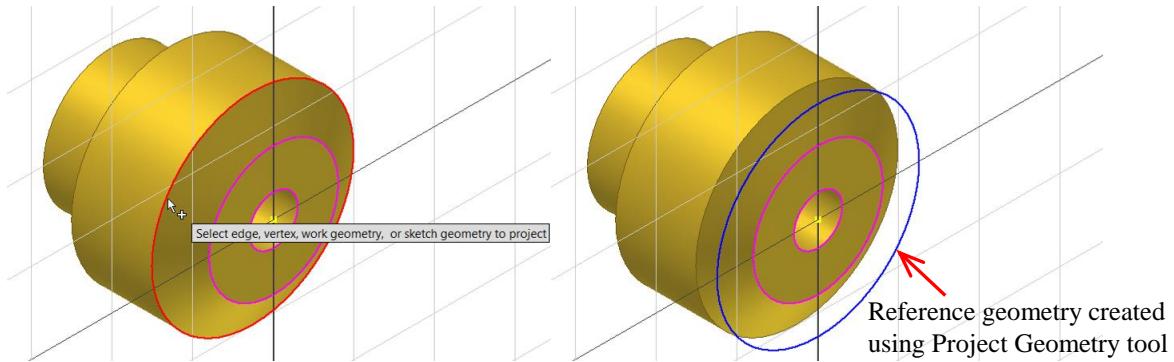
7.13.6 Project Geometry



Access: *Sketch Tab > Create Panel > Project Geometry*

The **Project Geometry** tool projects existing part vertices, edges, work features, loops, curves, and sketch geometry onto the existing sketch as reference geometry. As the geometry is selected, it is projected onto the current sketch plane as reference geometry and is always associative to the original source geometry. This means that if the source geometry changes, the reference geometry also changes. Following are some key attributes for projecting part edges:

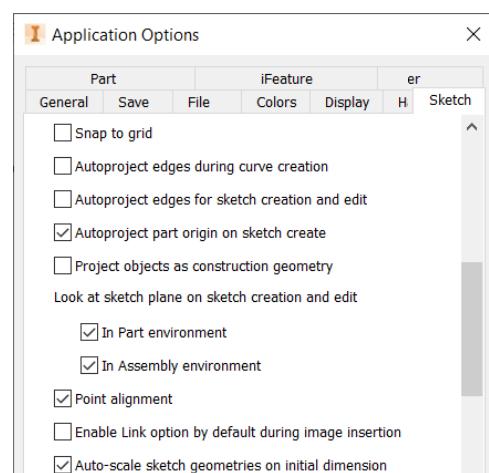
- ✓ Can be used as the basis for dimensions to new sketch geometry.
- ✓ Can be used to apply relational constraints to new sketch geometry.
- ✓ Cannot be dimensioned.
- ✓ Cannot be trimmed.
- ✓ Can be mirrored.
- ✓ Cannot be drawn; can only be created by using Project Geometry tool or by selecting the Auto-project Edges option.



Reference geometry can be projected automatically onto the sketch plane by selecting options from the Sketch tab in Application Options:

- ✓ **Autoproject edges during curve creation:** Geometry is projected automatically to the current sketch plane by “rubbing” the mouse cursor over any existing geometry to be projected while sketching.
- ✓ **Autoproject edges for sketch creation and edit:** When creating a new sketch, edges of the selected planar face are projected automatically on to the new sketch plane as reference geometry.

(Note: Such settings are not recommended as it will result in unnecessary projected geometries.)

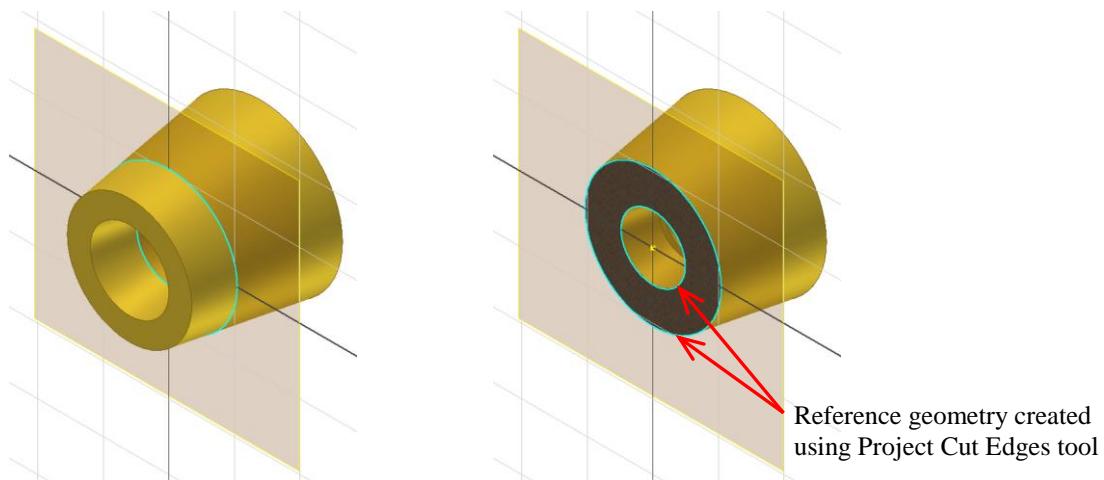


7.13.7 Project Cut Edges

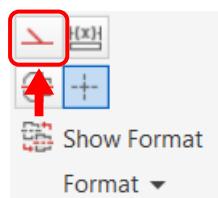


Access: *Sketch Tab > Draw Panel > Project Cut Edges*
(from Project Geometry's Drop down)

The **Project Cut Edges** tool projects part edges that intersect the active sketch plane into the sketch.

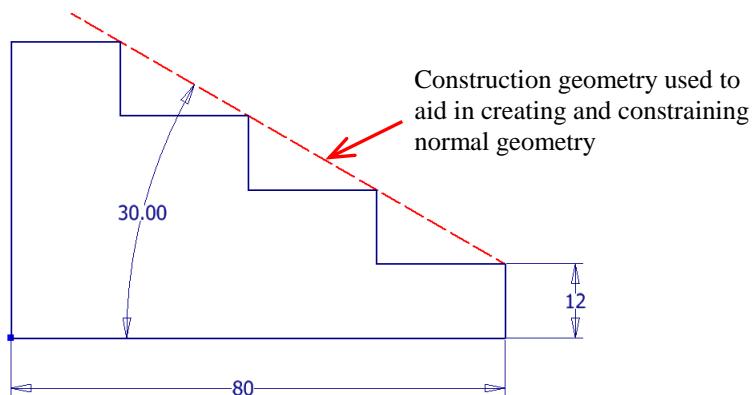


7.13.8 Construction Geometry

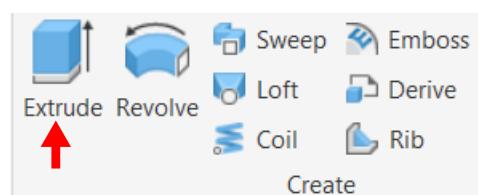


Access: **Sketch Tab > Format Panel > Construction**

The **Construction** tool changes selected sketch geometry to construction geometry, or creates new geometry as sketch construction geometry. Construction geometry is used to aid in creating and constraining normal geometry. Construction geometry is used when additional geometry is needed to constrain a sketch but not needed to participate in defining the profile for the feature.



7.14 Extrude

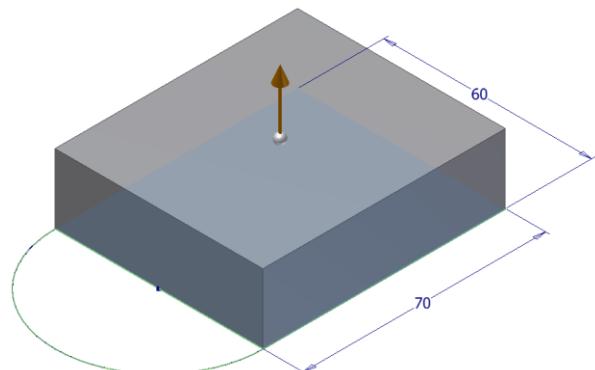
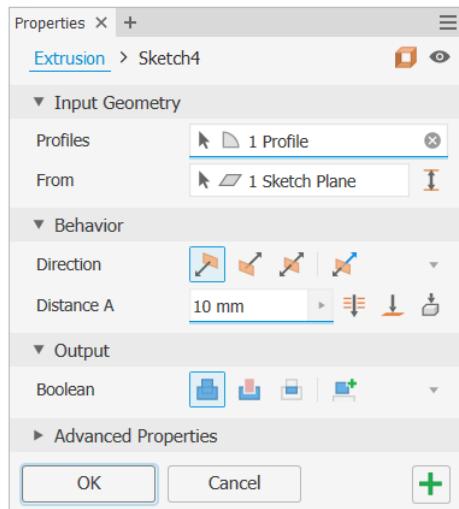


Access: **3D Model Tab > Create Panel > Extrude**

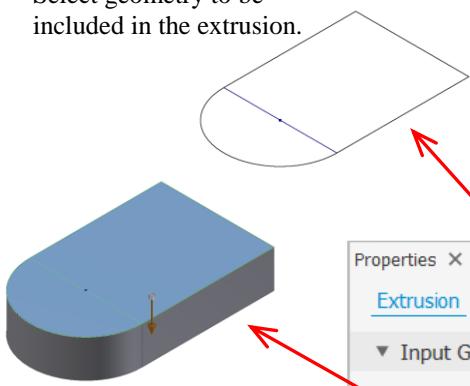
Shortcut: **E**

The **Extrude** tool creates a feature or body by adding depth to a sketch profile. The Extrude property panel will appear to choose the input geometry, behavior parameters and Boolean operation.

Extrude Property Panel:

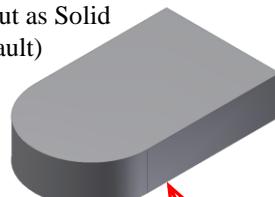


Select geometry to be included in the extrusion.

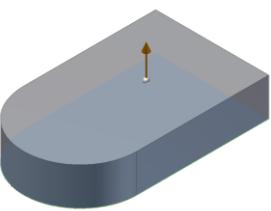
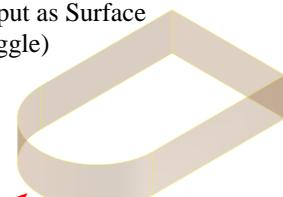


Extrude in Direction 2

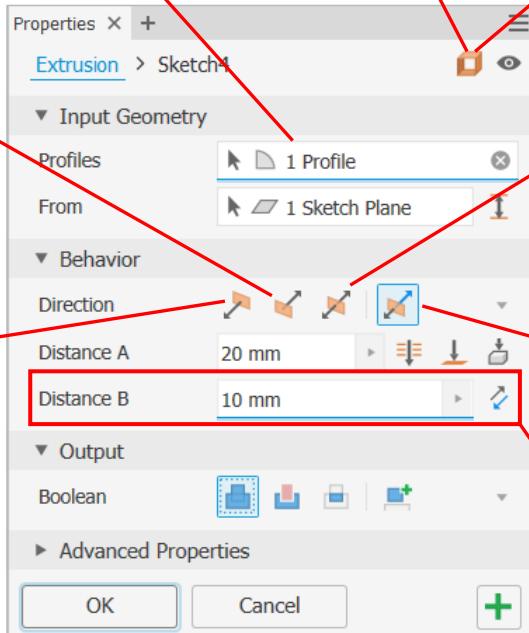
Output as Solid (Default)



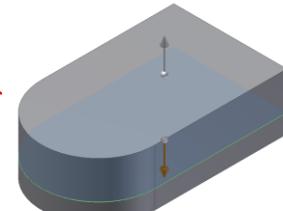
Output as Surface (Toggle)



Extrude in Direction 1

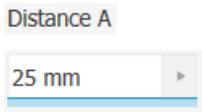
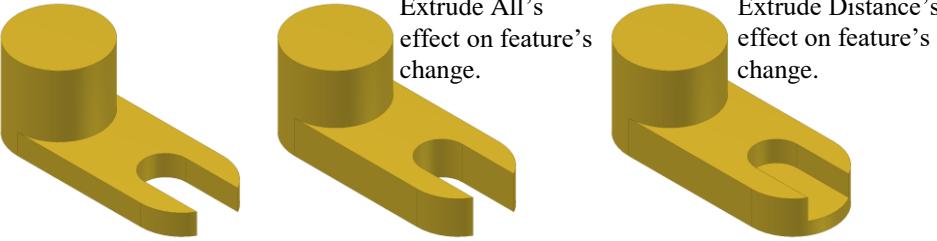
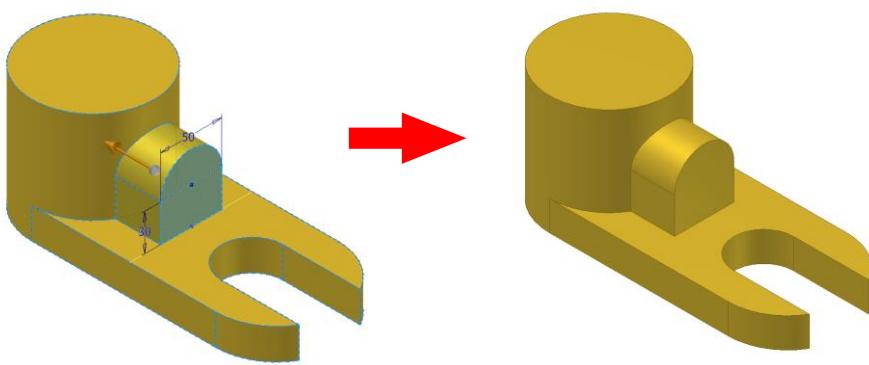
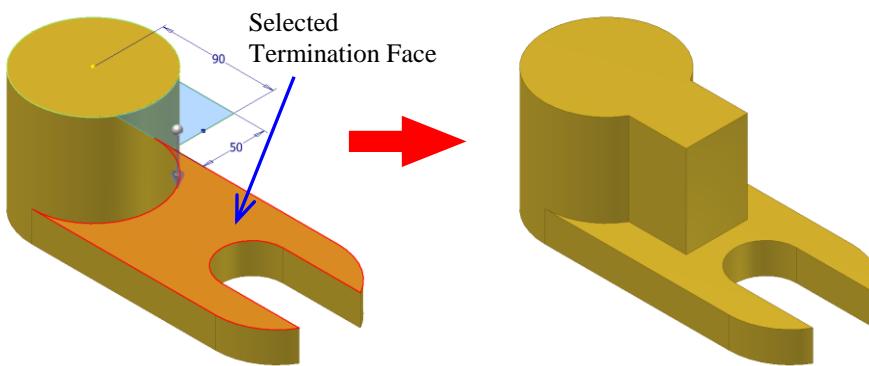


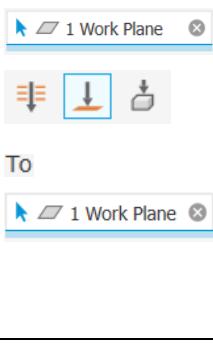
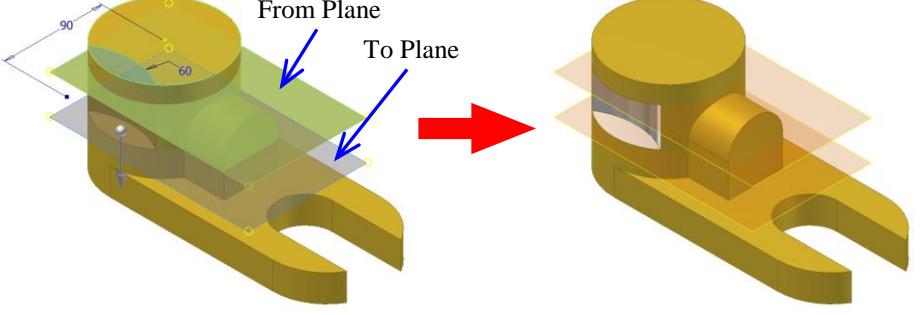
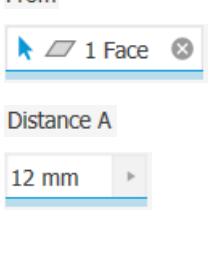
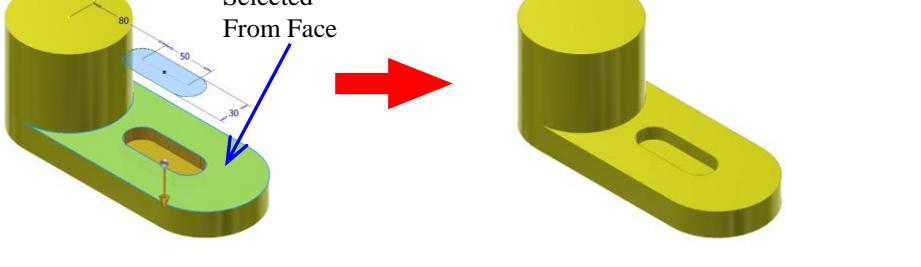
Extrude Symmetric



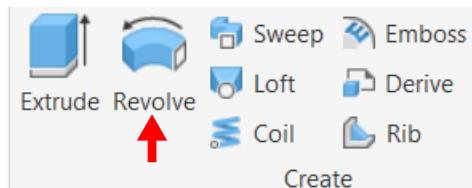
Only when Asymmetric is selected

The various options in the Behavior section to specify the starts and ends of extruded features are as follows:

Distance:  25 mm	Extrudes the profile based on the distance keyed-in.
All: 	Extrudes the profile through all features of the part. This is maintained even when the features changes.  Extrude All's effect on feature's change. Extrude Distance's effect on feature's change.
To Next: 	Extrudes the profile to the next possible face or plane. Profile must fully enclose in the area to which it is projecting.  A diagram showing a profile being extruded from a base feature to the next available face, resulting in a stepped extrusion.
To: 	Extrudes the profile to terminate on the selected face, plane or point. The default option “Select to terminate feature by extending the face” will terminate the feature on the extended face should the selected termination face does not completely enclose the extrusion profile.  A diagram showing a profile being extruded to a specific termination face, which is highlighted in orange. The angle between the profile and the termination face is labeled 90°, and the angle between the profile and the base feature is labeled 50°.

From/To: 	Extrudes the profile from one selected face or plane to another selected face or plane. 
From: 	Extrudes the profile from the selected face or plane based on the distance entered. 

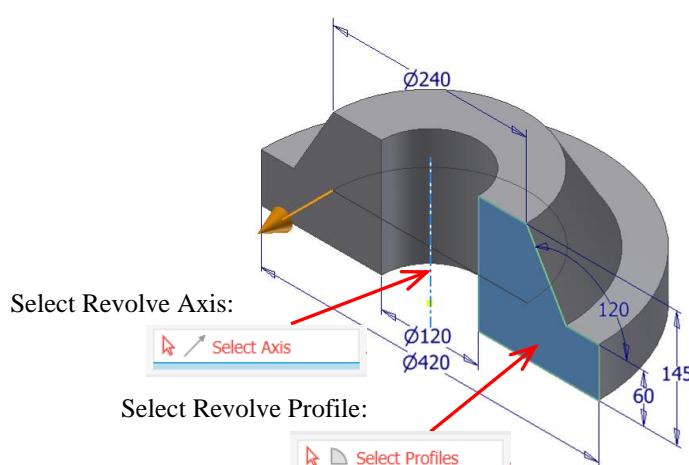
7.15 Revolve



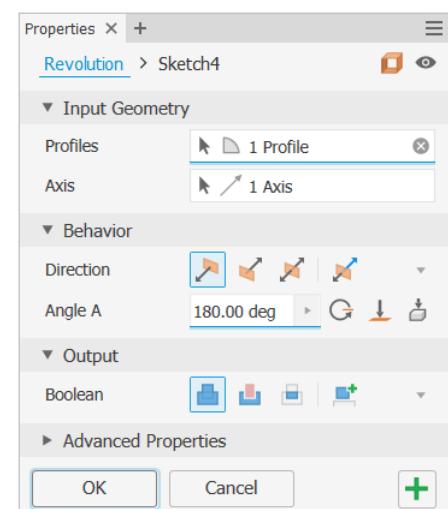
Access: **3D Model Tab > Create Panel > Revolve**

Shortcut: **R**

The **Revolve** tool creates a feature or body by revolving the selected sketch profiles about an axis. The Revolve property panel will appear to choose the input geometry, behavior parameters and Boolean operation.



Revolve Property Panel:



In the Input Geometry, select the revolve profile and axis. A line segment, a centreline, a work axis or a construction line can be used as the axis for the revolve feature. If the sketch contains a centreline, it will be selected automatically as the axis.

The option of **To** and **To Next** in the Behavior section for the **Revolve** tool is similar to the **Extrude** tool. The **Angle/Full** are specific to revolve features.

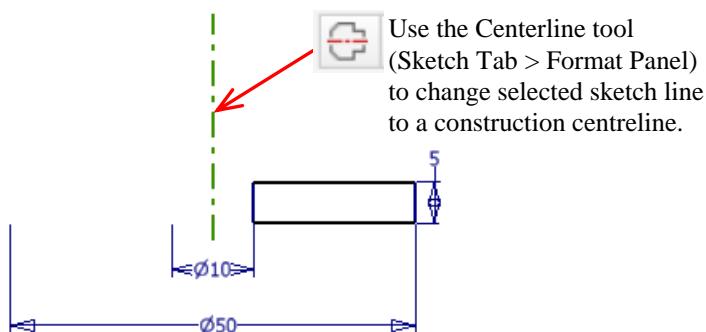
Angle: Revolves the profile at a specified angle around an axis.



Full: Revolves the profile a complete revolution around an axis.

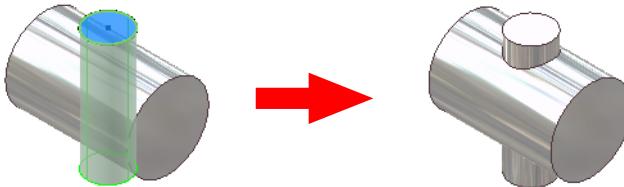
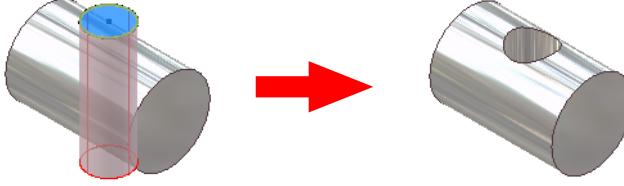
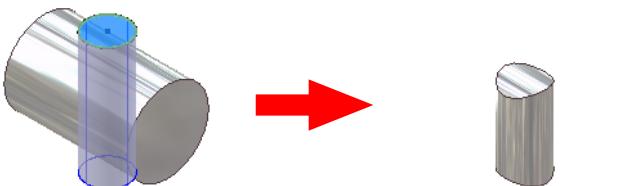


When creating a revolved feature, its revolve axis is **strongly recommended** to be defined as Centreline linetype. Dimensions added between the centreline and other sketch geometry will be treated as diameter dimensions.



7.16 Type of Boolean Operations

When creating sketched and placed features, the operation options are selectable to control the effect of the current feature on existing features. These operations are not available for the first feature of the part. Extrude, Revolve, Loft, Sweep, and Coil are features that contain the relationship options as below.

 Join	Adds the volume created by the extruded feature to another feature or body. 
 Cut	Removes the volume created by the extruded feature from another feature or body. 
 Intersect	Creates a feature from the shared volume of the extruded feature and another feature. Material not included in the shared volume is deleted. 
 New Solid	Creates a new solid body. This is the default selection if the extrusion is the first solid feature in a part file. Select to create a new body in a part file with existing solid bodies. Each body is an independent collection of features separate from other bodies. A body can share features with other bodies.

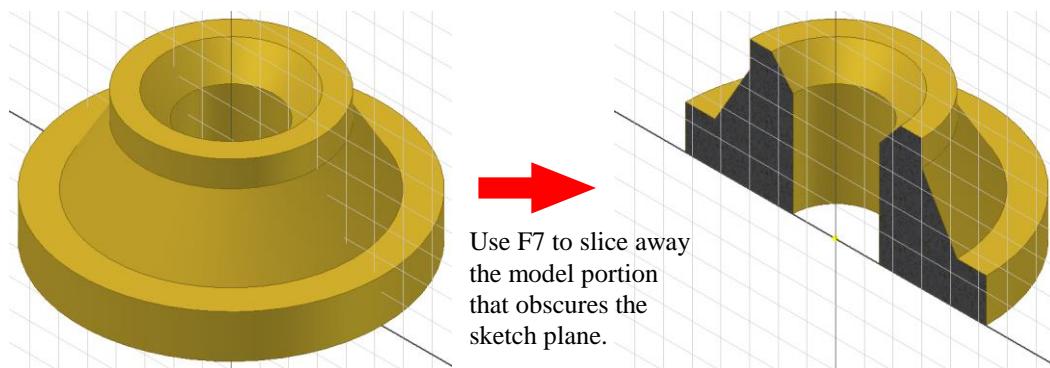
7.17 Slice Graphics



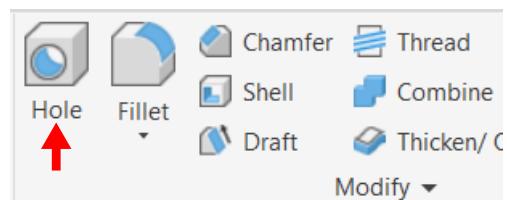
Access: **View Tab > Visibility Panel > Slice Graphics**

Shortcut: **F7**

When the sketch plane to sketch on is obscured by geometry, the Slice Graphics option (on the context menu or F7) can be used to slice away the portion of the model temporarily that obscures the sketch plane.



7.18 Hole

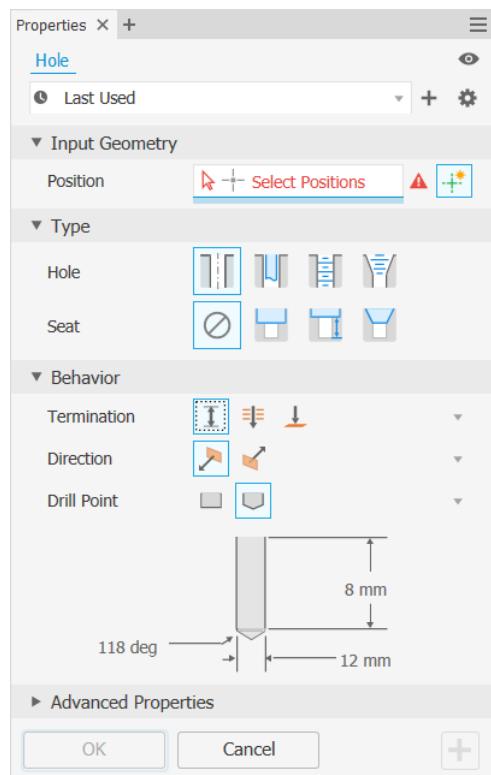


Access: **3D Model Tab > Modify Panel > Hole**

Shortcut: H

The **Hole** tool creates different type of holes based on sketch points or other geometric selections in a single property panel, rather than having to manually edit or create geometry. Types of hole include Simple, Counterbore, Spotface, Countersink hole, Clearance hole, Tapped hole and Taper Tapped hole. Although Hole features are considered to be placed features, sketch geometry will be used to locate the centre points for the holes.

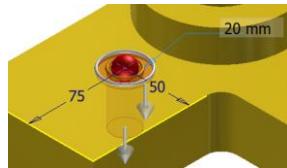
Hole Property Panel:



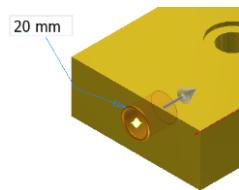
7.18.1 Types of Input Geometry for Hole

Linear:

Choose Planar Face or WorkPlane then linear edge(s) to place dimensions.

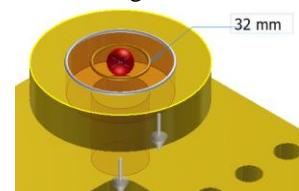


Work Point:

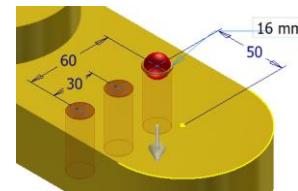


Concentric:

Choose Planar Face or WorkPlane then circular edge or Face as concentric reference.

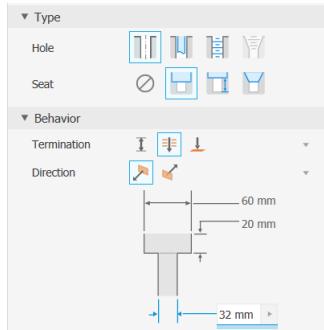


By Sketch:

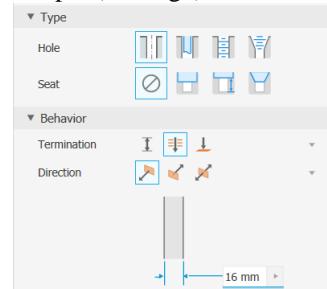


7.18.2 Types of Hole

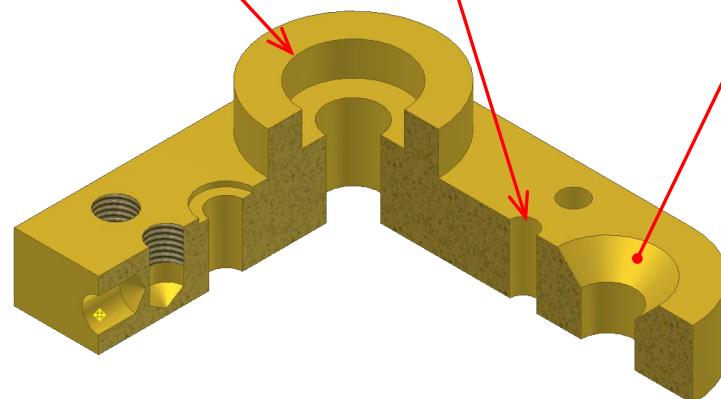
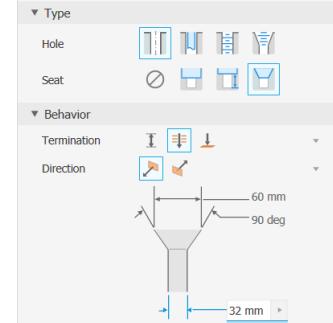
Counterbore

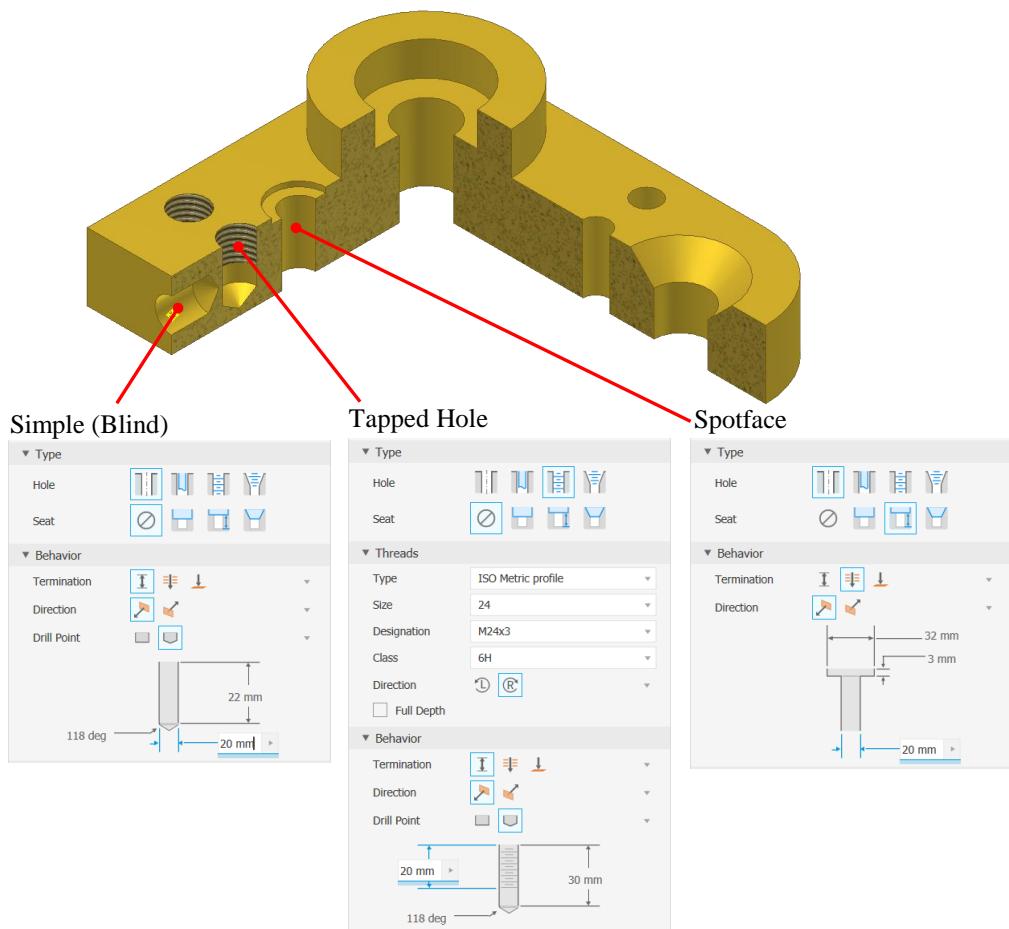


Simple (Through)

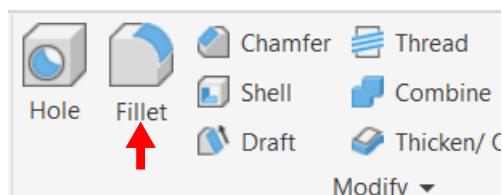


Countersink





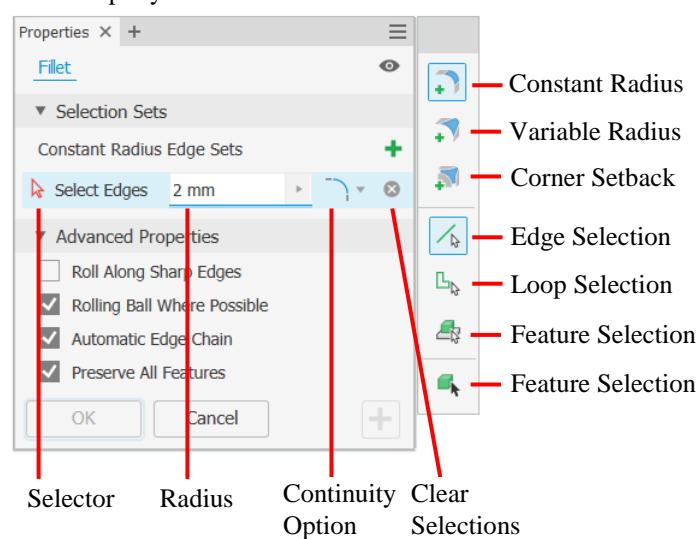
7.19 Fillet



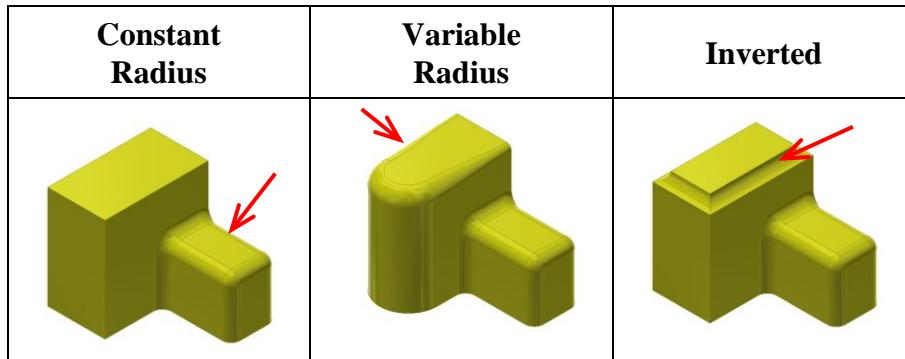
Access: **3D Model Tab > Modify Panel > Fillet**

Shortcut: F

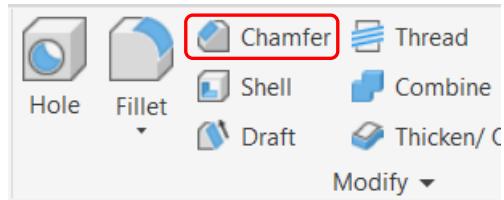
Fillet Property Panel:



The **Fillet** tool adds fillets or rounds to one or more edges on the part model. Constant-radius and variable-radius fillets, fillets of different sizes, and fillets of different continuity (Tangent, Smooth G2 or Inverted) can be created in a single operation.



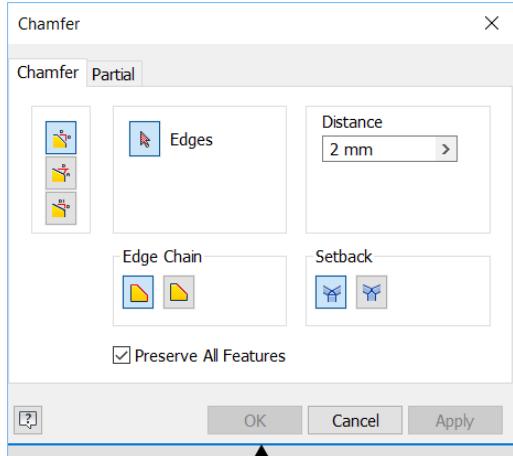
7.20 Chamfer



Access: **3D Model Tab > Modify Panel > Chamfer**

Shortcut: (Ctrl + Shift + K)

Chamfer Dialog Box:

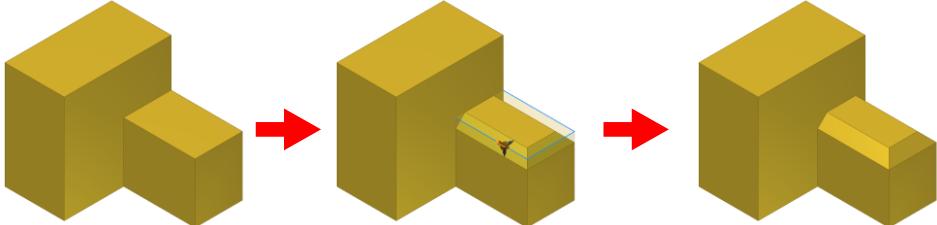
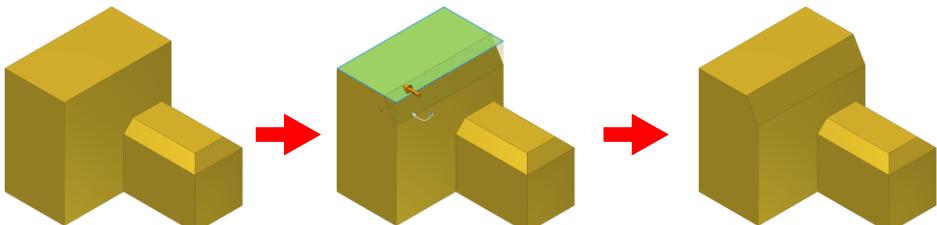
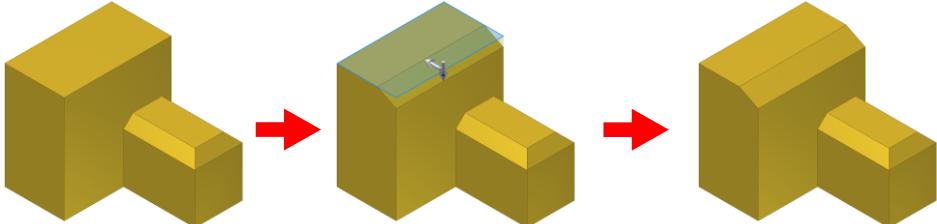


Chamfer Mini-toolbar:

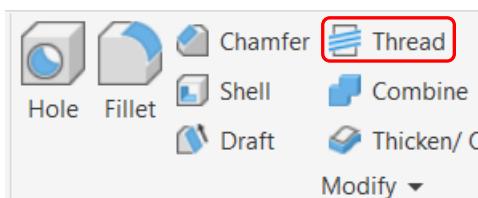


Click the help icon  in the Chamfer dialog box for details of various selection options.

The **Chamfer** tool applies bevels to component edges by:

Chamfer Options	Description
 Distance	<p>Specify a distance for the chamfer. The distance is applied to both sides of the selected edge, resulting in a 45° chamfer.</p> 
 Distance and Angle	<p>Select a face adjacent to the edge to be chamfered. The angle is measured from this face. Select the edge(s) to be chamfered. The edge(s) selected must be adjacent to the selected face. Specify a distance and angle for the chamfer. The distance is measured from the selected edge along the selected face. The angle is measured from the selected face.</p> 
 Two Distances	<p>Select the edge to be chamfered. For this method, only one edge can be chamfered at a time. Specify the first and second distance of the chamfer.</p> 

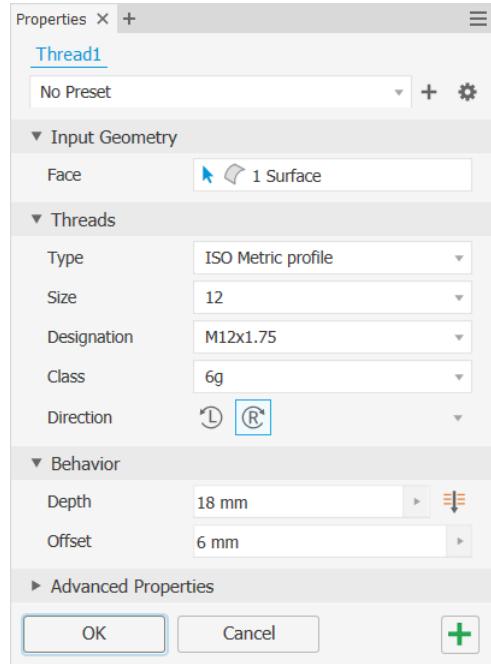
7.21 Thread



Access: **3D Model Tab > Modify Panel > Thread**

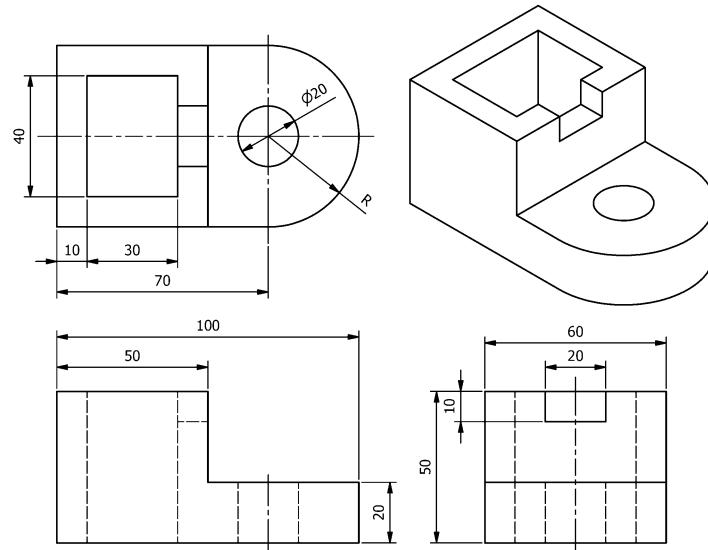
The **Thread** tool creates threads on **shafts**. It is *NOT applicable to threaded hole*.

Thread Property Panel:

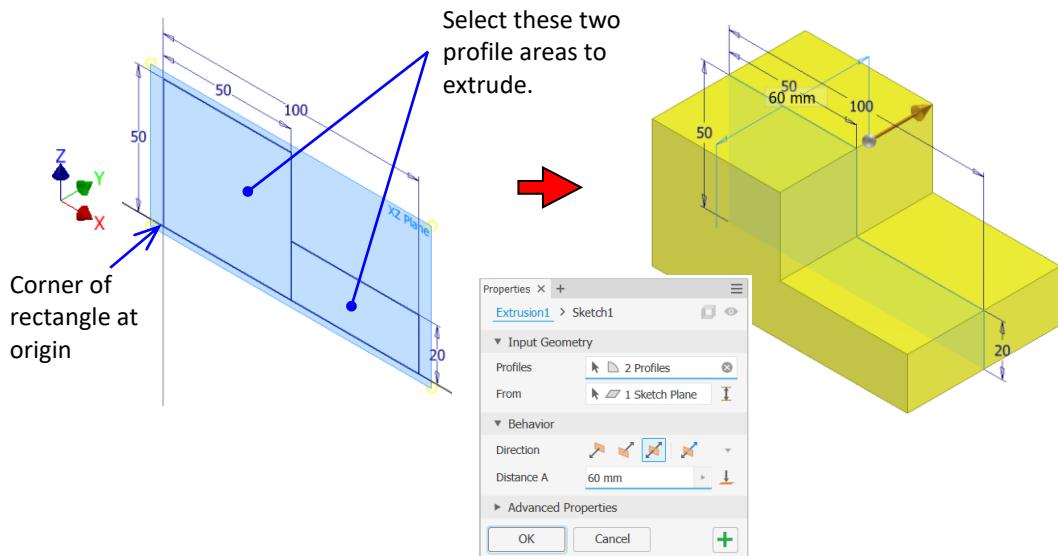


7.22 Work Example I

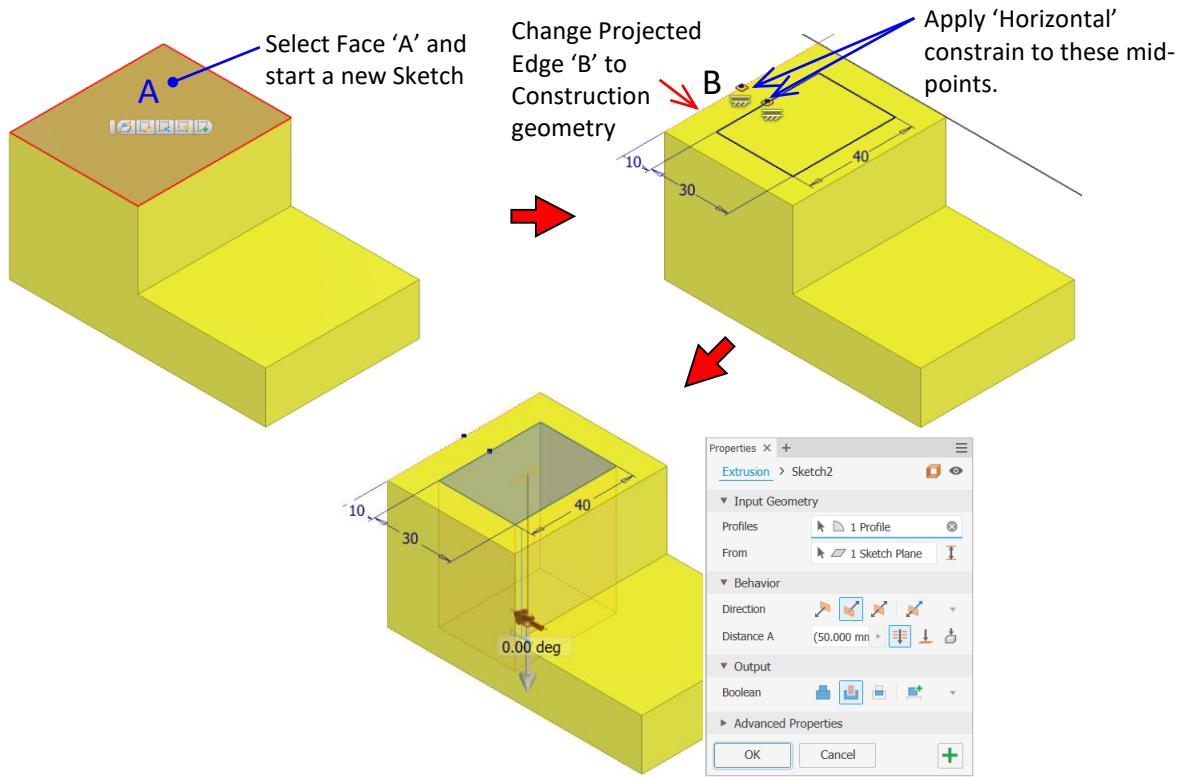
- Create a part model for the below figure using the given dimension (design intent):



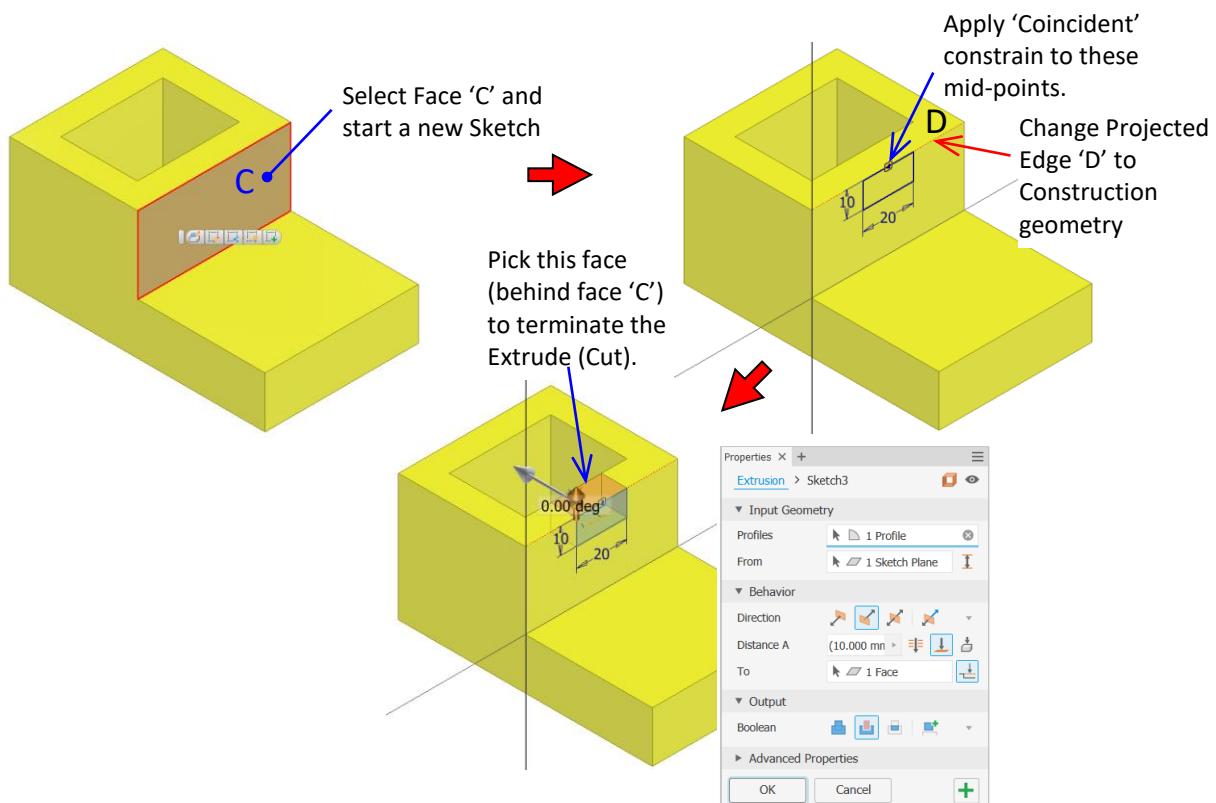
Step 1: Start a new sketch on XZ Plane. Use **Rectangle** tool (*Two Point*), draw two rectangles, dimension and constrain as below. Use **Extrude** tool to extrude the sketch (Direction = Symmetric; Distance A = 60).



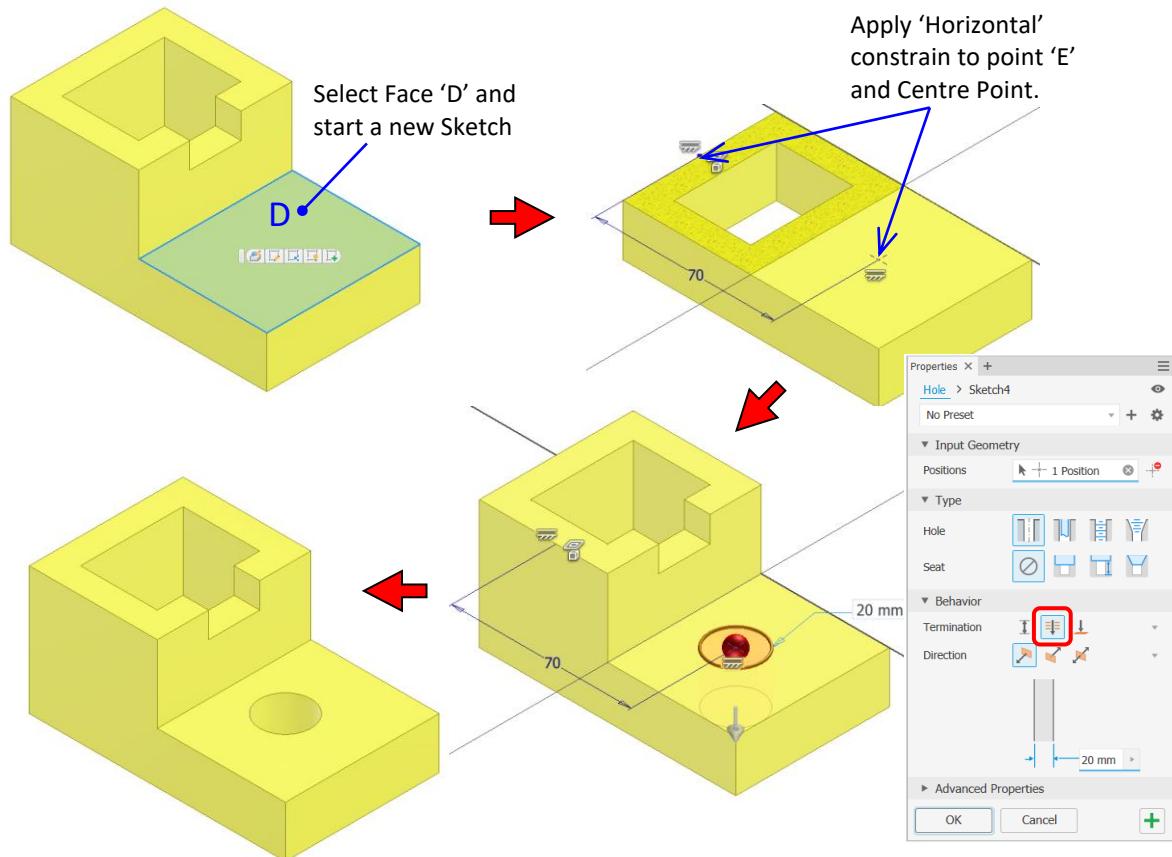
Step 2: Select Face 'A' and start a new sketch. Select **Project Geometry** tool → Select edge 'B' → Change the projected edge 'B' to Construction geometry. Use **Rectangle** tool (*Two Point*), draw a 40 x 30 rectangle profile, dimension and constrain as shown. Use **Extrude** tool to extrude the sketch (Distance A = 'Through All' along -ve Z; Boolean = 'Cut').



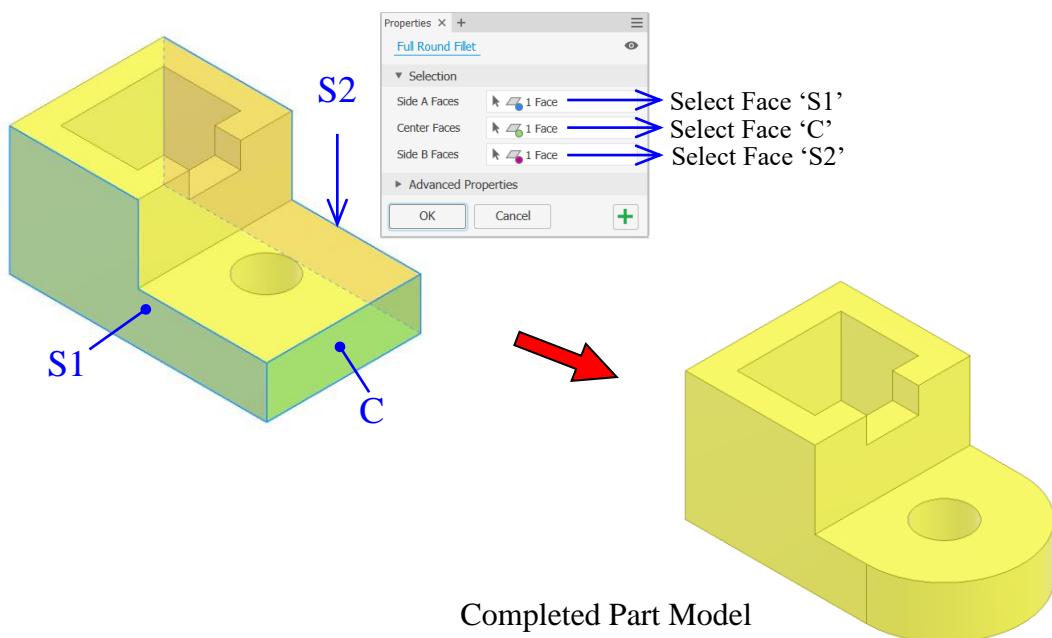
Step 3: Select Face 'C' and start a new sketch. Select **Project Geometry** tool → Select edge 'D' → Change the projected edge 'D' to Construction geometry. Use **Rectangle** tool (*Two Point*), draw a 20 x 10 rectangle profile, dimension and constrain as shown. Use **Extrude** tool to extrude the sketch (Distance A = 'To' along -ve X; Boolean = 'Cut').



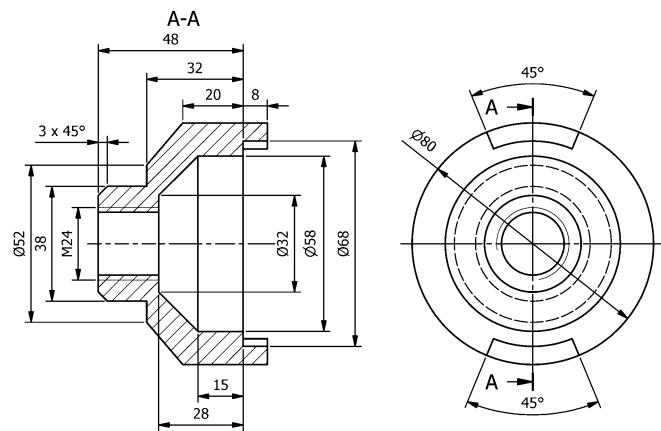
Step 4: Select Face 'D' and start a new sketch. Use **Point** tool, place a point 'E' on Face D. Press **F7 (Slice Graphics)**, dimension and constrain as shown. Use **Hole** tool to create the Ø20 hole, (Position = Point 'E' will be automatically selected; Termination = 'Through All').



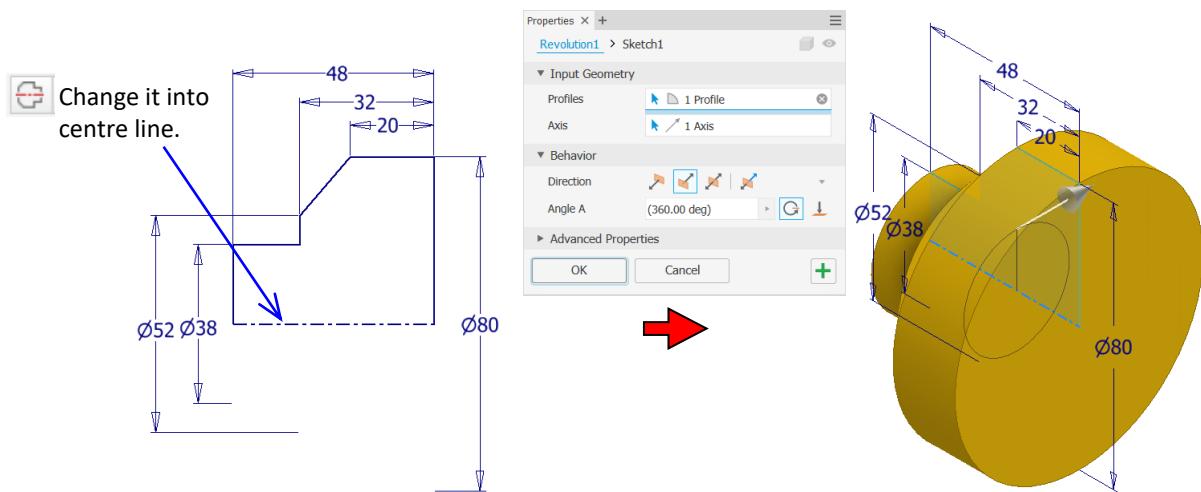
Step 5: Use **Fillet** tool (*Full Round Fillet*) to make a full round as shown.



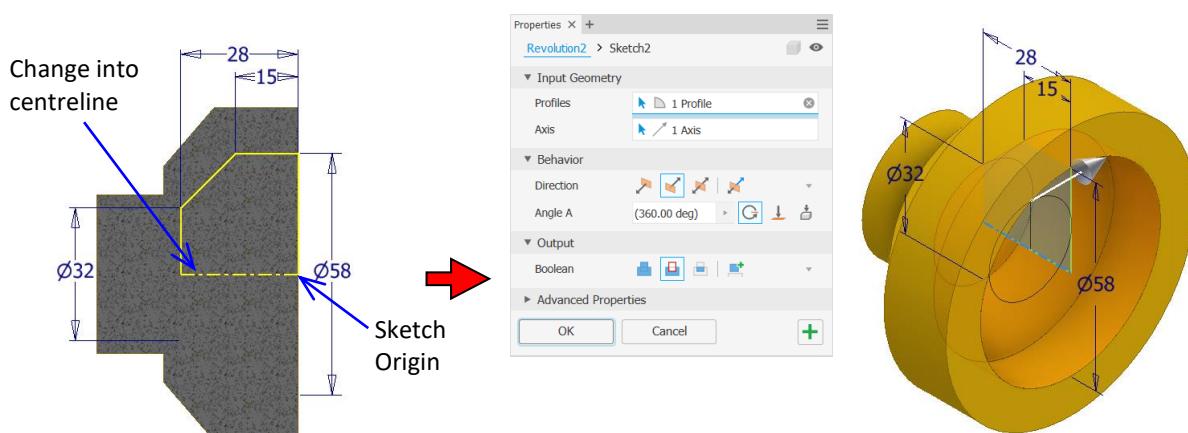
2. Create a part model for the below figure using the given dimension (design intent):



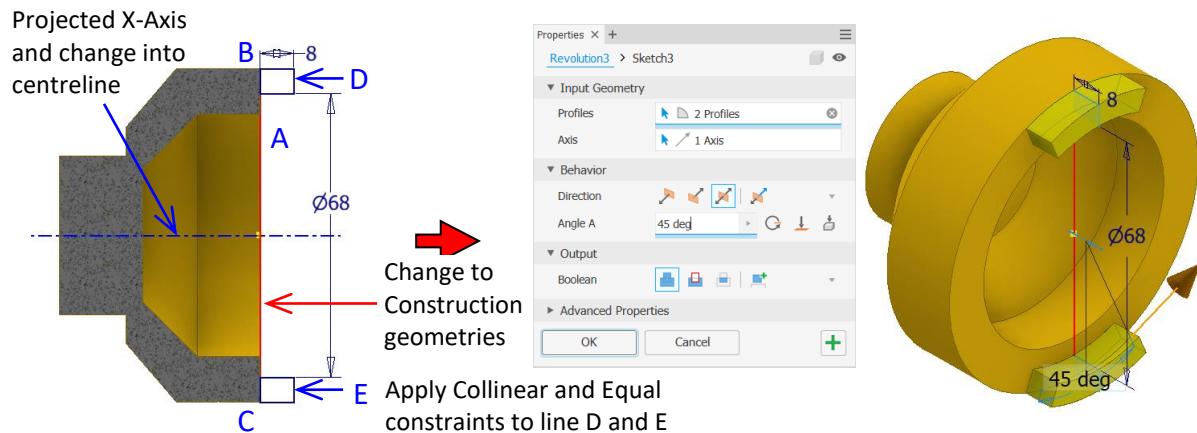
Step 1: Start a new sketch on XZ Plane. Create the profile with dimensions as below. Inferred Constrains should meet design intent. Use **Revolve** tool to revolve the sketch (Angle A = ‘Full’).



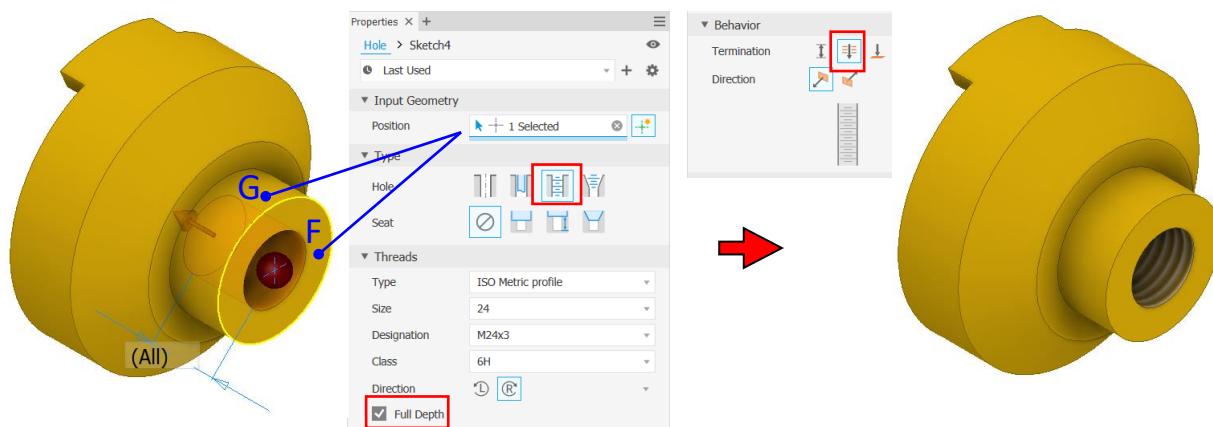
Step 2: Start another new sketch on XZ Plane. Create the profile with dimensions as below. Inferred Constrains should meet design intent. Use **Revolve** tool to revolve the sketch (Angle A = ‘Full’; Boolean = ‘Cut’).



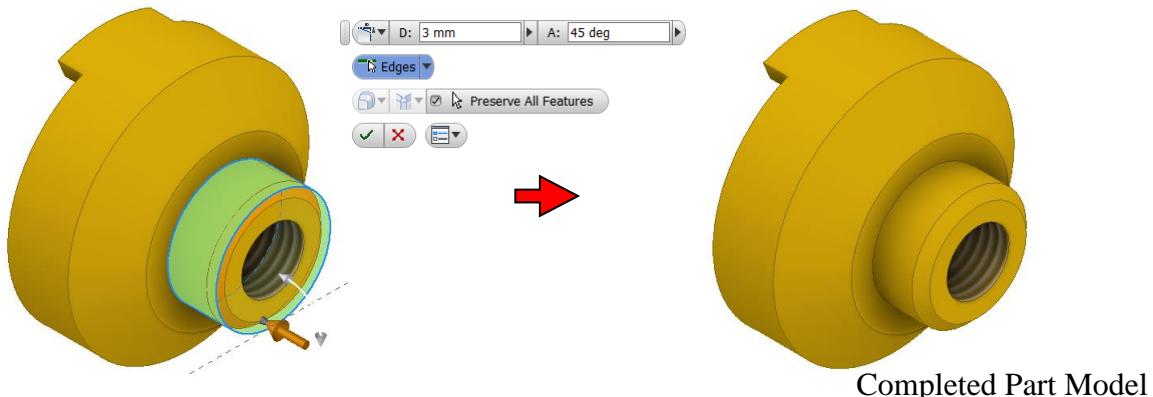
Step 3: Start another new sketch on XZ Plane. Select Project Geometry tool → Select edge ‘A’ → Change the projected edge ‘A’ to Construction geometry. Use **Rectangle** tool (*Two Point*) → Draw two rectangle profiles from point B and C as shown → Apply Collinear and Equal constraints to line ‘D’ and ‘E’ → Dimension as shown. Use **Revolve** tool to revolve the sketch (Direction = ‘Symmetric’; Angle A = 45; Boolean = ‘Join’).



Step 4: Use **Hole** tool to create the M24 tapped hole, (Position = ‘Face F’ then ‘Face G’; Hole = ‘Tapped Hole’, Thread Type = ‘ISO Metric profile’, Size = ‘M24’, Designation = ‘M24x3’; Termination = ‘Through All’).



Step 5: Use **Chamfer** tool to apply a 3 x 45° chamfer to the selected edge (Option = ‘Distance and Angle’).



7.23 Work Features

All part will contain a default set of work planes, work axes, and a centre point (see Fig 7.1). These default work features are placed in the Origin folder of the Part browser. These default work features are used to define the initial orientation of the part design, create new sketches, provide option to terminate feature, and create new work features.

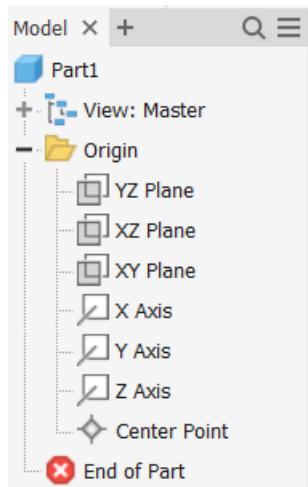
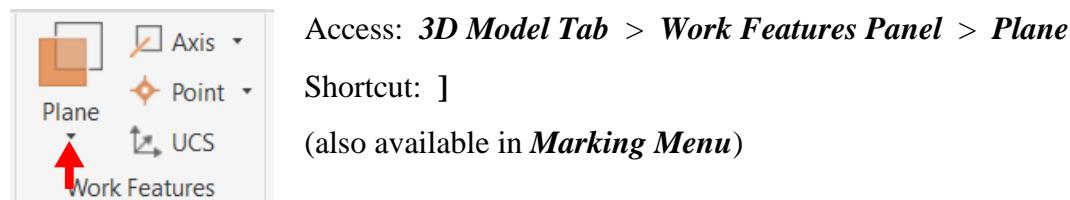


Fig 7.1 Default work features in a part model tree.

Parametric work features, i.e. work plane, work axis and work points that are based on faces, edges, and vertices of the part will need to be created during part design when existing part geometry does not exist for the subsequent creation of part features. Some situations are:

- Define a sketch for new features whereby a part face is not available.
- Establish an intermediate position that is required to define other work features.
- Provide an axis or point of rotation for revolved features and patterns.
- Provide a feature termination plane off the part when there are no existing part faces available for the termination.

7.23.1 Work Plane



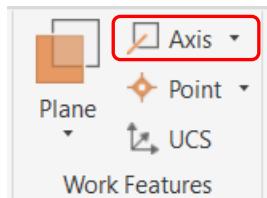
Work plane is required when

- A part face is not available to select as a sketch plane for creating new feature.
- A part face is not available to terminate the new feature to be created.
- An intermediate position is required to define other work planes (e.g. at an angle to a face and at an offset distance).

Work plane option:

Option	Selection	Result
 Plane	Appropriate vertices, edges, or faces to define a work plane	Create a work plane through the selected objects.
 Offset from Plane	A planar face or work plane.	Create a work plane parallel to the selected face/plane at the specified offset distance.
 Parallel to Plane Through Point	A planar face or work plane and a point or work point in either order.	A work plane parallel to the selected face/plane and through the selected point.
 Midplane Between Two Planes	Two planar faces or planes.	A work plane at the mid-plane of the two selected faces/planes.
 Midplane of Torus	A torus.	A work plane through the centre or mid-plane of the torus.
 Angle to Plane Around Edge	A planar face or work plane and any parallel edge, line or work axis.	Create a work plane at the specified angle from the selected face/plane about the edge/line/axis.
 Three Points	Any three points (endpoints, intersections, midpoints, centre points or work points).	A work plane through these selected points.
 Two Coplanar Edges	Two coplanar work axes, edges or lines.	A work plane through the two selected axes, edges or lines.
 Tangent to Surface Through Edge	A curve face and a linear edge, line or work axis in either order.	A work plane through the selected edge, line, or axis and tangent to the selected curve face.
 Tangent to Surface Through Point	A curve face and an endpoint, midpoint, centre point or work point.	A work plane through the selected point and tangent to the selected curve face.
 Tangent to Surface and Parallel to Plane	A curve face and a planar face or work plane in either order	A work plane that is parallel to the selected face/plane and tangent to the selected curve face.
 Normal to Axis Through Point	A linear edge, line or work axis and a point in either order	A work plane that is perpendicular to the selected edge/line/axis and pass through the selected point.
 Normal to Curve at Point	Nonlinear edge or sketch curve (arc, circle, ellipse, or spline) and a vertex, edge midpoint, sketch point or work point on the curve.	A work plane that is normal to the selected curve and pass through the selected point.

7.23.2 Work Axis



Access: **3D Model Tab > Work Features Panel > Axis**

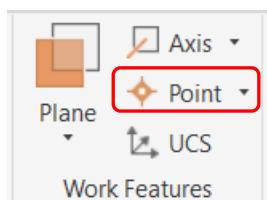
Work axis can be created for the following:

- As axis of revolution for circular pattern.
- Constrain parts in an assembly using assembly constraints.
- Basis for new work features.
- Representation of centre lines on sketches.

Work axis option:

Option	Selection	Result
Axis	Appropriate edges, lines planes or points to define a work axis.	Create a work axis through the selected objects.
On Line or Edge	A linear edge or sketch line.	A work axis collinear with the selected edge or sketch line.
Parallel to Line Through Point	A linear edge or sketch line and an endpoint, midpoint, sketch point or work point.	A work axis parallel to the selected edge/line and through the selected point.
Through Two Points	Two endpoints, intersections, midpoints, sketch points or work points.	A work axis through the two selected points.
Intersection of Two Planes	Two non-parallel work planes or planar faces.	A work axis coincident with the intersection of the two selected planes/faces.
Normal to Plane Through Point	A planar face or work plane and a point.	A work axis perpendicular to the selected face/plane and through the point.
Through Center of Circular or Elliptical Shape	A circular edge, elliptical edge or fillet edge.	A work axis coincident with the circular, elliptical or fillet axis.
Through Revolved Face or Feature	A revolved face or feature.	A work axis coincident with the face or feature axis.

7.23.3 Work Point



Access: **3D Model Tab > Work Features Panel > Point**

Work point can be created for the following:

- Basis for new work features such as work axis and work plane.
- Place a hole feature (using the On Point option in the Hole tool).
- Place 3D lines on the work points.

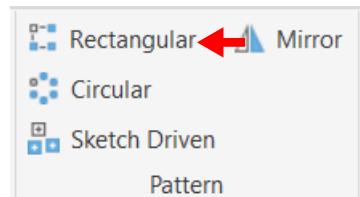
Work point option:

Option	Selection	Result
 Point	Appropriate vertices, edge and faces to define a work point.	Create a work point through the selected objects.
 Grounded Point	A work point, midpoint, or vertex.	A work point that is grounded. Denoted by a pushpin cursor symbol.
 On Vertex, Sketch Point, or Midpoint	A 2D or 3D sketch point, vertex, endpoint or midpoint of a line or linear edge.	A work point on the selected point.
 Intersection of Three Planes	Three intersection work planes or planar faces.	A work point on the intersection point of the three selected planes/faces.
 Intersection of Two Lines	Two lines including linear edges, 2D or 3D sketch lines, and work axes.	A work point on the intersection point of the two selected edges/lines/axes.
 Intersection of Plane/Surface and Line	A planar face or work plane and a work axis, linear edge or line.	A work point on the intersection point of the two selected geometries.
 Center Point of Loop of Edges	An edge of a closed loop of edges.	A work point on the centre point of the selected loop of edges.
 Center Point of Torus	A torus.	A work point on the centre point of the selected torus.
 Center Point of Sphere	A sphere.	A work point on the centre point of the selected sphere.

7.24 Pattern

Pattern tools are used to duplicate existing geometry according to the specified parameters. Occurrences (Duplicates) of the original features are created in either circular or rectangular pattern. The occurrences are associative to the original feature, i.e. changes in the original feature are automatically updated to the pattern occurrences.

7.24.1 Rectangular Pattern

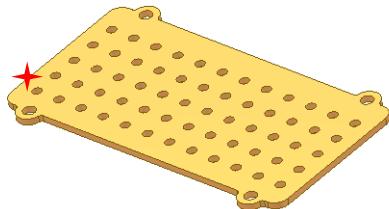


Access: **3D Model Tab > Pattern Panel > Rectangular**

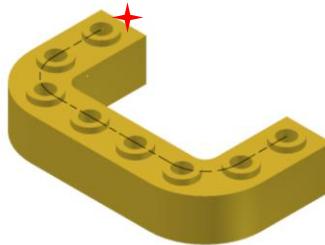
Shortcut: **Ctrl + Shift + R**

Duplicates one or more features or bodies and arranges the resulting occurrences in a rectangular array or along a path in one or two directions, with options to control the occurrences spacing.

Rectangular pattern:

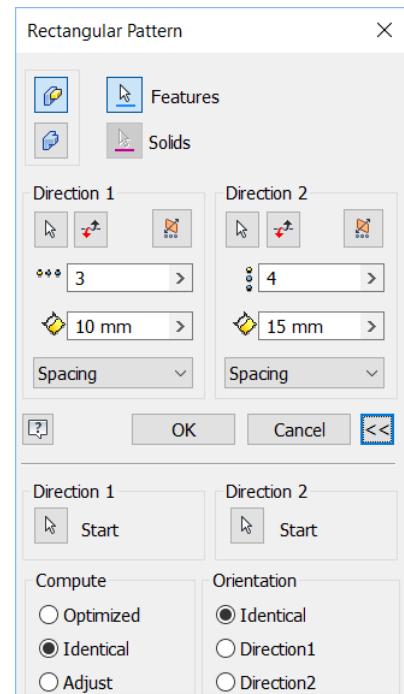


Pattern along a path:

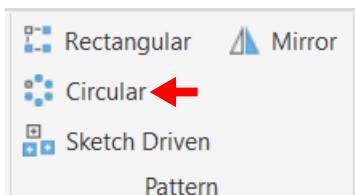


Click the help icon  in the *Rectangular Pattern* dialog box for details of each selection option.

Rectangular Pattern Dialog Box:



7.24.2 Circular Pattern

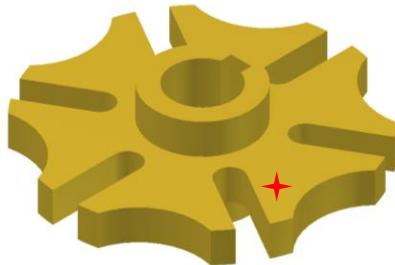


Access: **3D Model Tab > Pattern Panel > Circular**

Shortcut: **Ctrl + Shift + O**

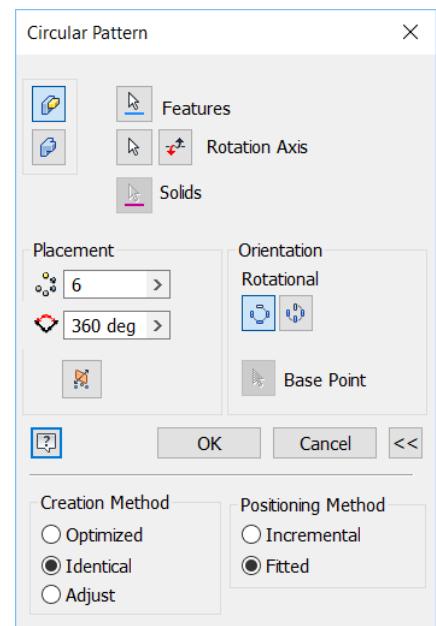
Duplicates one or more features or bodies and arranges the resulting occurrences in a circular array, with options to specify the placement of the occurrences.

Circular pattern:



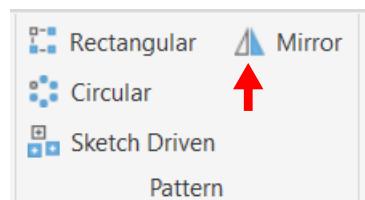
Click the help icon  in the *Circular Pattern* dialog box for details of each selection option.

Circular Pattern Dialog Box:



7.25 Mirror

Parts may contain features that are mirror images of one another. For such mirror features (or the part itself) about a plane of symmetry, mirror tool will be ideal for creating them instead of building up from beginning.



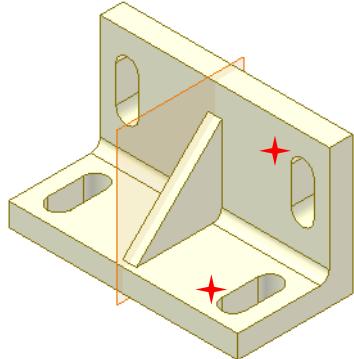
Access: **3D Model Tab > Pattern Panel > Mirror**

Shortcut: **Ctrl + Shift + M**

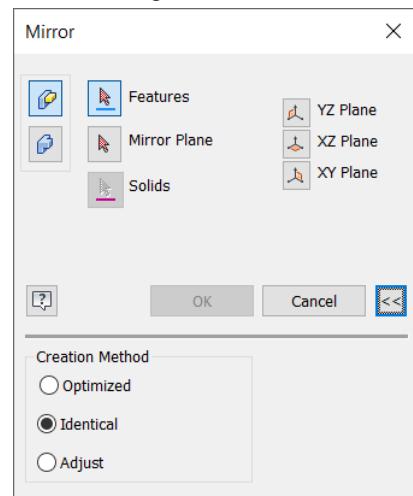
The symmetry plane can be any of the following:

- An existing face on the part.
- Any one of the origin work planes.
- A new defined work plane.

Mirroring:



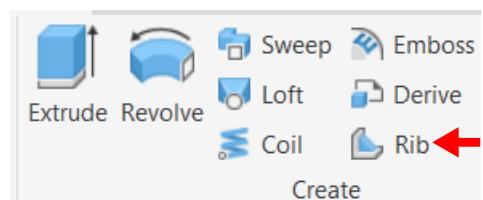
Mirror Dialog Box:



Click the help icon in the *Mirror* dialog box for details of each selection option.

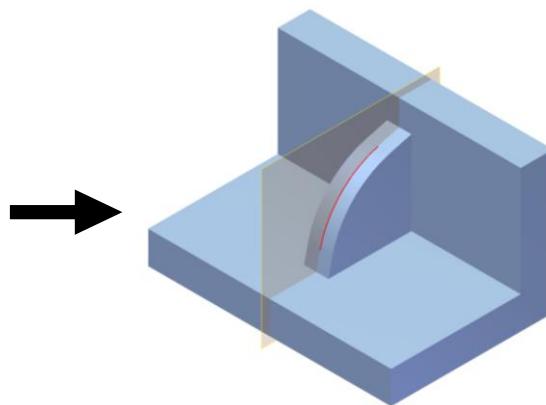
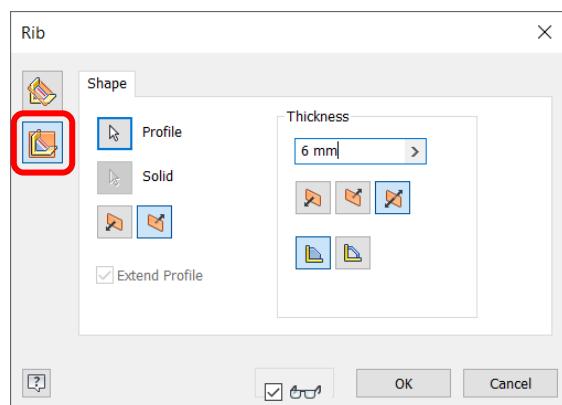
7.26 Rib

Ribs and webs are thin-walled support features added to the part for increased rigidity. Rib tool creates such features from an open or closed, unconsumed sketch profile.



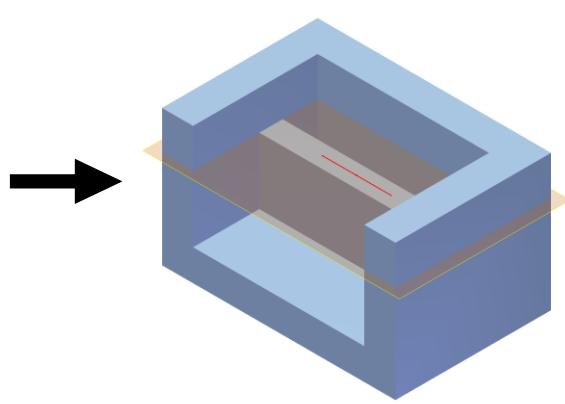
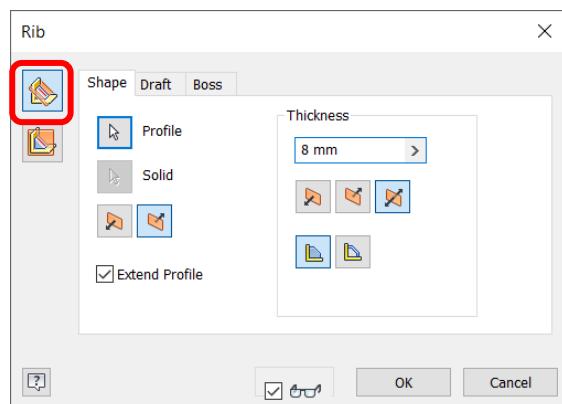
Access: **3D Model Tab > Create Panel > Rib**

Rib (Parallel to Sketch Plane) Dialog Box:



Selecting “Parallel to Sketch Plane” will create a rib feature with its rib area parallel to the sketch plane; extruding the sketch geometry parallel to the sketch plane.

Rib (Normal to Sketch Plane) Dialog Box:

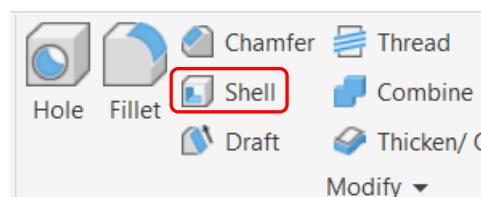


Selecting “Normal to Sketch Plane” will create a rib feature with its rib area perpendicular to the sketch plane; extruding the sketch geometry perpendicular to the sketch plane. Rib Draft and Boss feature may also be included to such rib feature too.

Click the help icon  in the *Rib* dialog box for details of each selection option as well as the Draft and Boss tab.

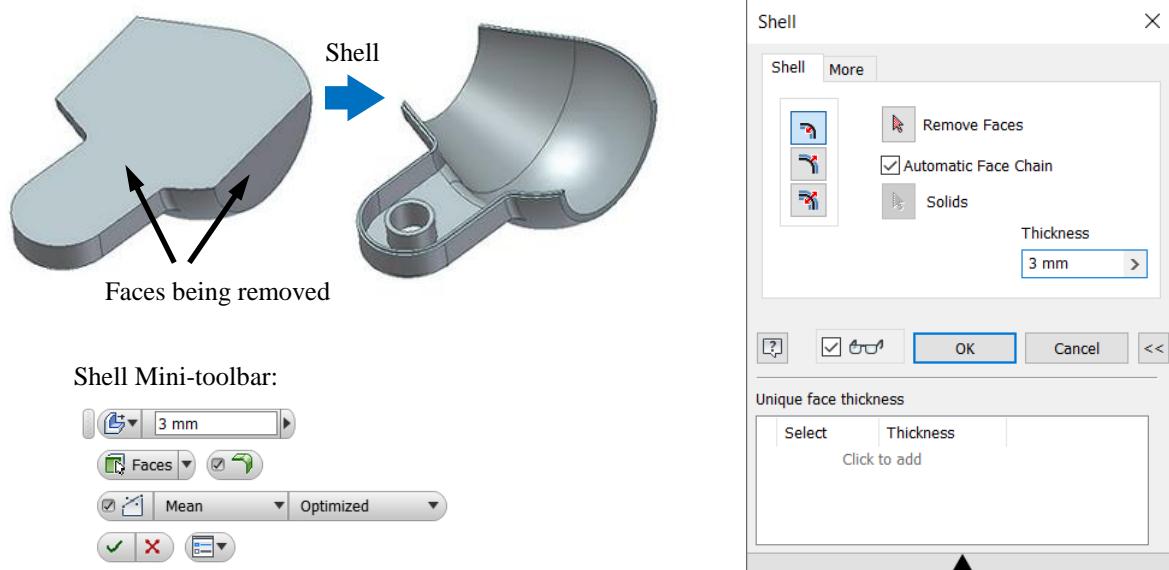
7.27 Shell

The Shell tool creates thin-walled part by removing material from the interior of a solid geometry and adding the specified wall thickness to the faces of the part. Each face of the part can have different wall thickness. One or more faces can also be removed from the shell feature; otherwise a hollow part is created.



Access: **3D Model Tab > Modify Panel > Shell**

The following illustration shows the part before and after adding a shell feature.



Click the help icon in the *Shell* dialog box for details of each selection option as well as the More tab.

7.28 Face Draft

Draft is a taper that is applied to one or more part faces. Besides applying draft during Extrusion or Sweep by specifying a taper angle, draft can be added to an existing feature or individual faces using the Face Draft tool. When face draft is applied to faces, the relationship between the pull direction and the fixed edge, face, or plane determines the result of the operation.

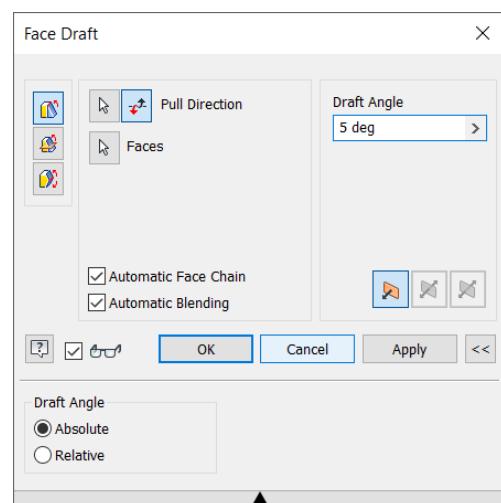


Access: **3D Model Tab > Modify Panel > Draft**
Shortcut: D

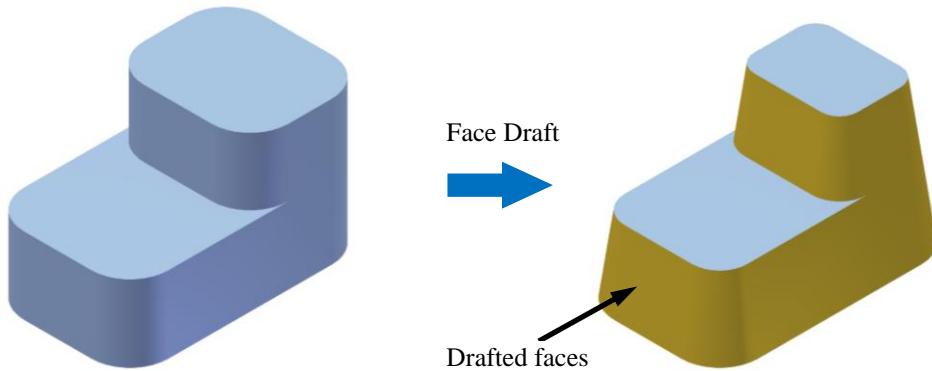
Face Draft Mini-toolbar:



Face Draft Dialog Box:



The following illustration shows the part before and after applying the Face Draft.

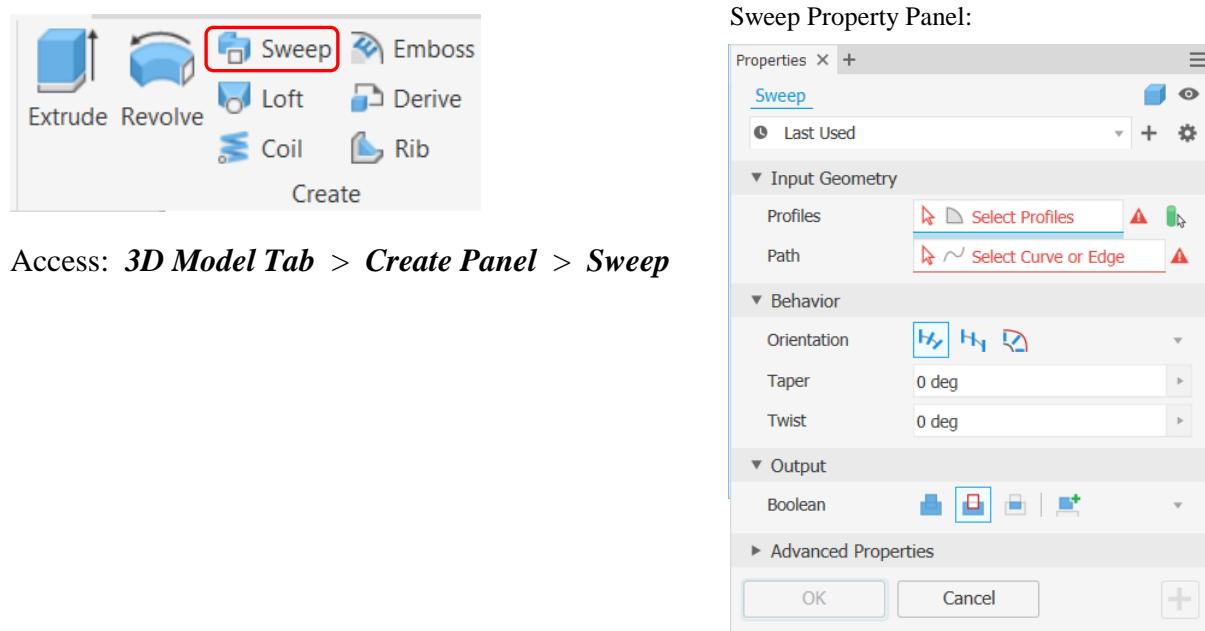


Click the help icon  in the *Face Draft* dialog box for details of each selection option.

Combining the additional parametric part modelling tools introduced in this unit with the part modelling tools acquired in Computer-Aided Drafting module, the 3D part modelling capabilities have been extended to cover most part design requirements. Please refer to the Autodesk Inventor help to learn those tools that are not mentioned to take the 3D part modelling capabilities even further.

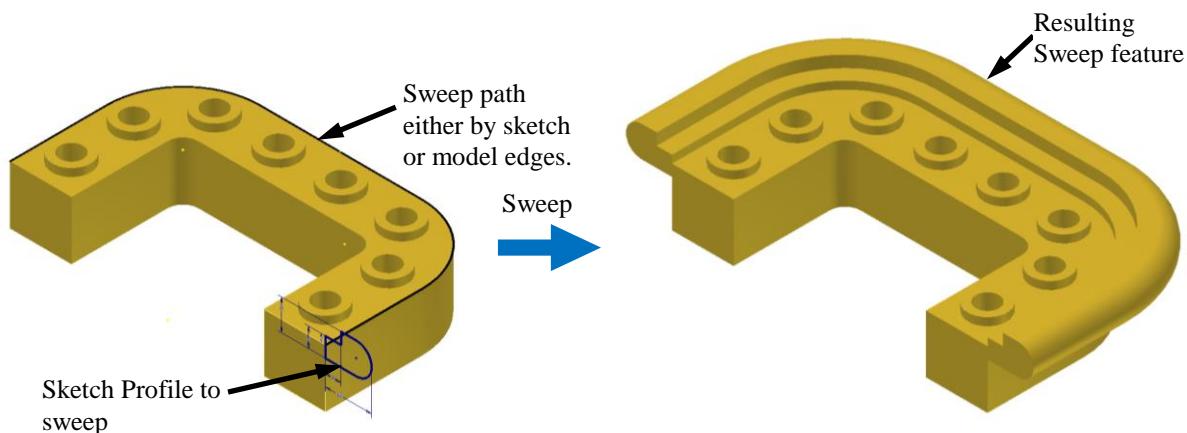
7.29 Sweep

Sweep tool creates either a solid or surface feature by sweeping the relevant sketch profiles along a selected path which can be an open or closed loop. Thus, a sweep feature would require an unconsumed sketch for the profile, and an unconsumed sketch or an existing model edge for the path. The sketch plane of the profile is typically perpendicular to the path at its start point. The start point of the path is often projected into profile sketch to provide a reference point.



Access: **3D Model Tab > Create Panel > Sweep**

The following illustration shows the part before and after executing the Sweep tool.



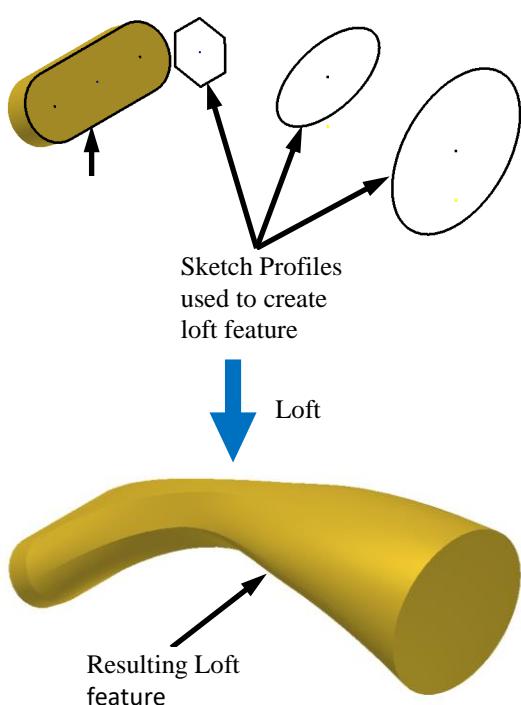
7.30 Loft

The Loft tool creates solid or surface features by blending multiple profiles, called sections, and transitioning them into smooth shapes between profiles or part faces. The loft sections can be 2D or 3D sketch profiles, face loops or points. The loft shape can be further refined by rails, or a centreline and point mapping, to control the shape and prevent twisting.

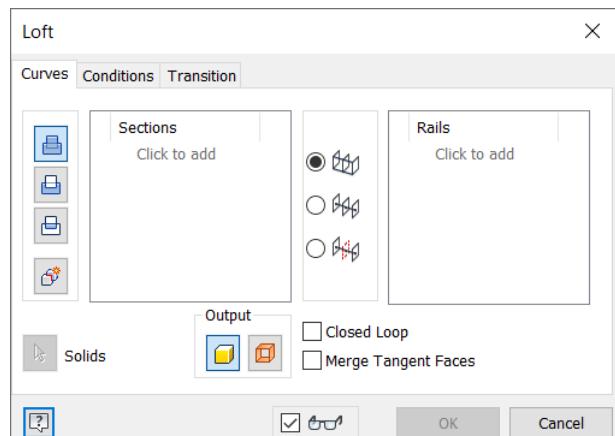


Access: **3D Model Tab > Create Panel > Loft**

The following illustration shows the part before and after executing the Loft tool.



Loft Dialog Box:



Click the help icon in the *Loft* dialog box for details of each selection option as well as the Conditions and Transition tabs.

7.31 Part Modelling Practices

The following are the part modelling practices to be adopted for this module:

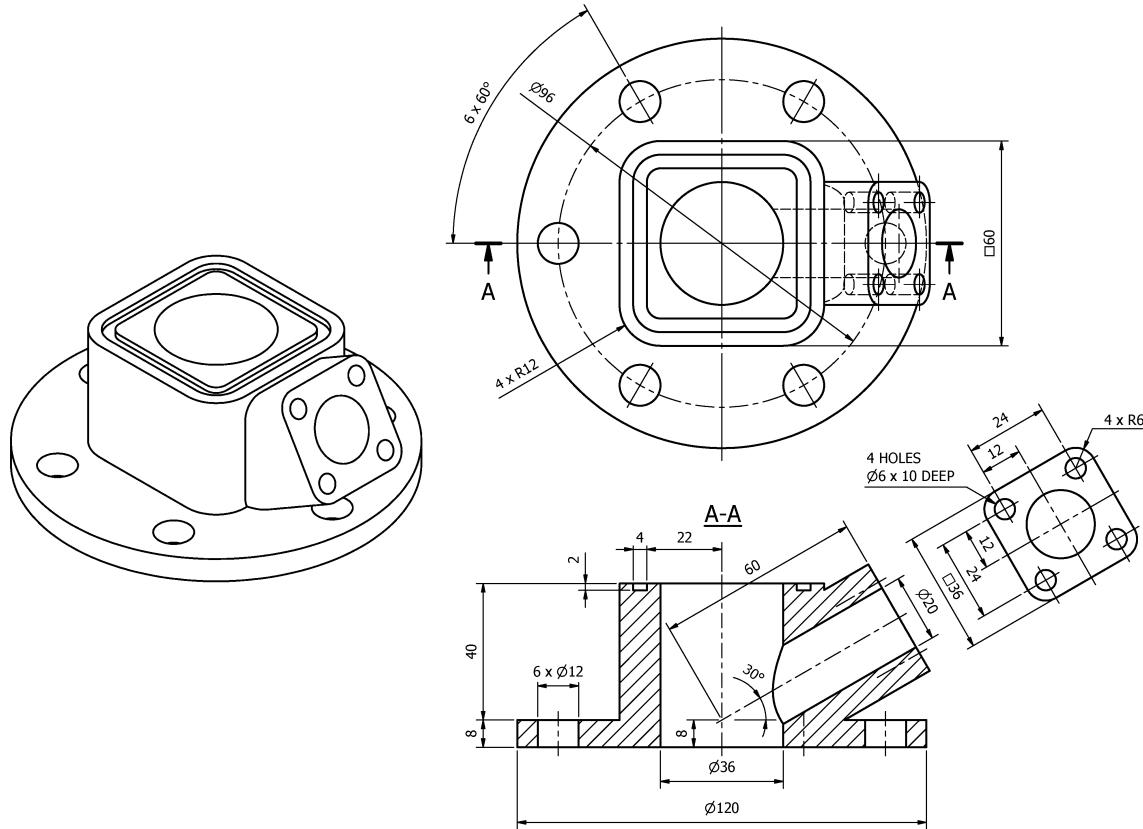
1. Refer to para 7.13.5, page 7-21 for guidelines on 2D Sketching, Geometric Constraining and Dimensioning Sketches.
 2. Sketch profiles/outlines shall be fully 3. Keep sketch simple.
 4. Dimensional constraints and Feature parameter's **must** follow the given dimensions scheme, (no calculation required).
 5. **Additional** dimensional constraints shall **NOT** be used for the purpose of fully constrained the sketch. Use appropriate geometric constraints instead.
 6. Do not fillets/chamfers on sketch, draw it with as sharp edges. Create the fillets and chamfers as placed features using 3D fillets and chamfers.
 7. Do not pattern sketch geometries. Use feature-pattern to create repeated patterns (circular/rectangular) of features.
 8. Recommendations for 2D Sketch (Tools > Application Options > Sketch):
 - a. Keep "Autoproject part origin on sketch create".
 - b. Keep "Point alignment".
- Recommendations for 2D Sketch:

<input type="checkbox"/> Snap to grid
<input type="checkbox"/> Autoproject edges during curve creation
<input type="checkbox"/> Autoproject edges for sketch creation and edit
<input checked="" type="checkbox"/> Autoproject part origin on sketch create
<input type="checkbox"/> Project objects as construction geometry
Look at sketch plane on sketch creation and edit
<input type="checkbox"/> In Part environment
<input type="checkbox"/> In Assembly environment
<input checked="" type="checkbox"/> Point alignment
<input type="checkbox"/> Enable Link option by default during image insertion
<input checked="" type="checkbox"/> Auto-scale sketch geometries on initial dimension
9. Project **only** the relevant geometries to the active sketch.
 10. Project Edges rather than Project Surfaces as individual edges cannot be deleted independently for those edges of projected surface.
 11. The part model's history tree shall **NOT** contain any **unconsumed** sketches.
 12. Create features in a "**non-patching**" way, i.e. incorrectly created feature are to be edited/updated/re-created instead of correcting it in a patch-up way.
 13. Appropriate Extents/Termination, i.e. All, To, To Next, Between are to be used with priority over Distance input.
 14. User-defined work features, i.e. work planes, work axes, work points shall be minimise. This requires good planning and study of the part information.
 15. "Holes" features shall be created via Hole tool and not by Extrude-cut or Revolve-cut.

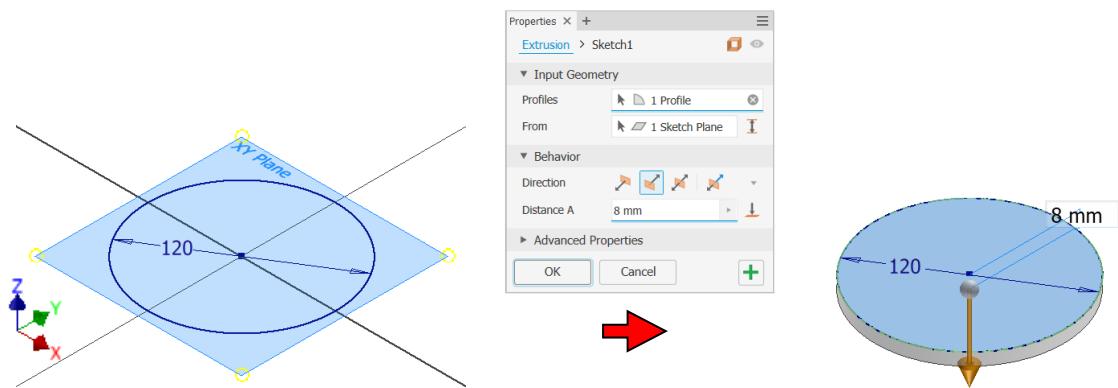
7.32 Work Example II

These guided examples will illustrate the application of the part modelling tools introduced in this unit.

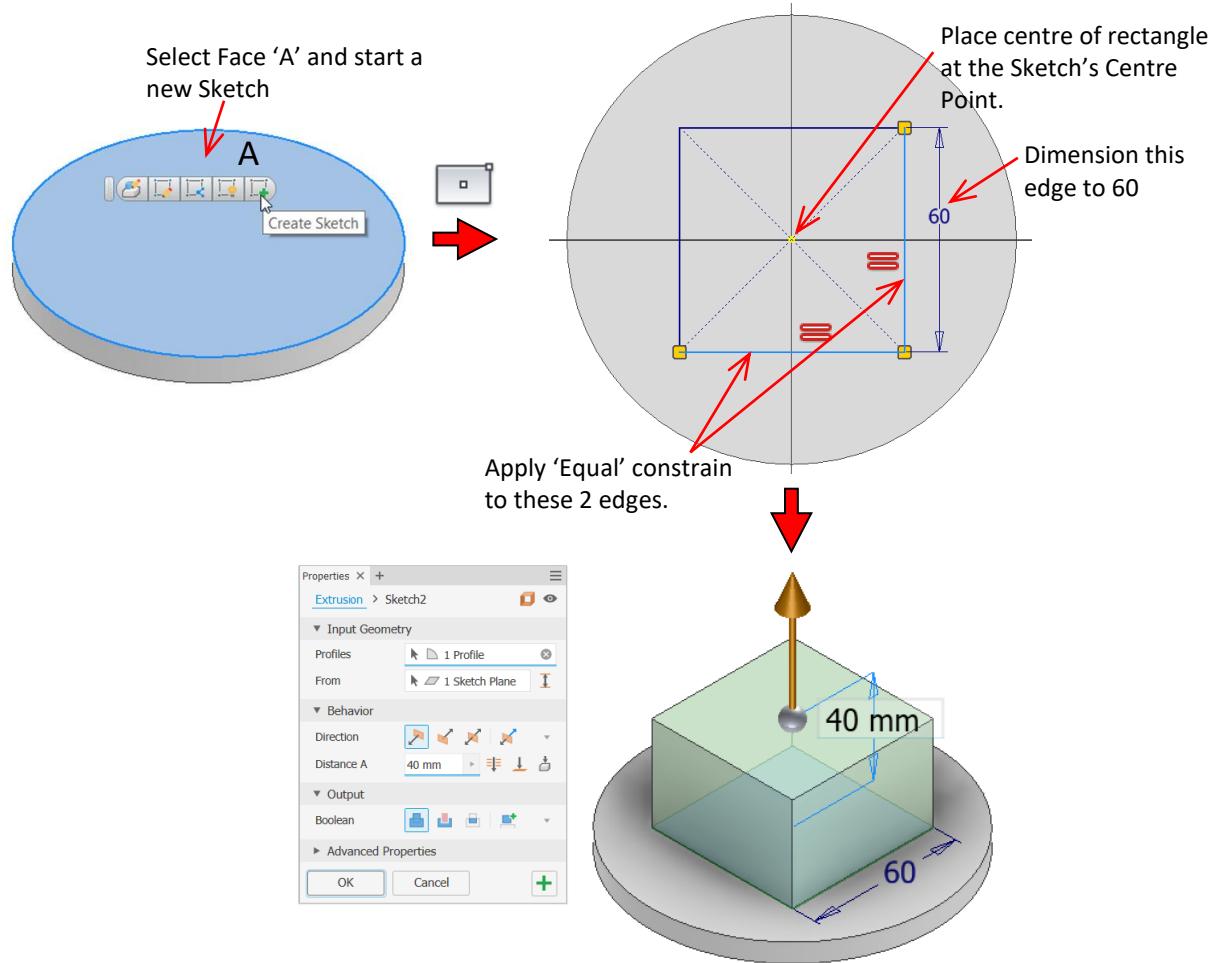
7.32.1 Part Model A



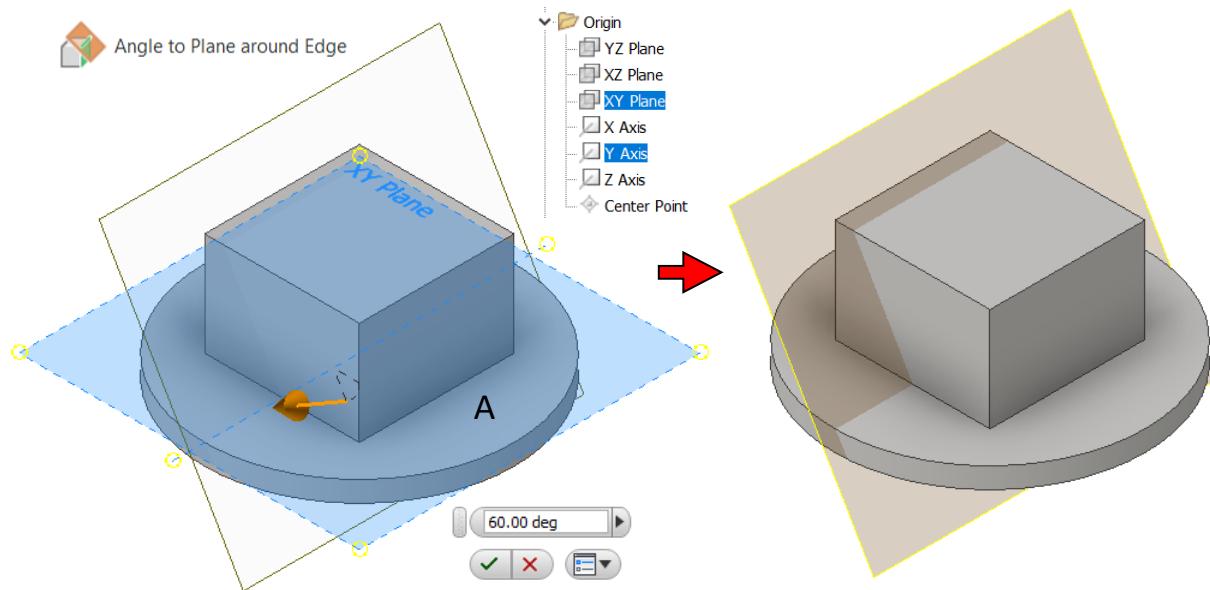
Step 1: Start a new sketch on XY Plane. Draw a circle sketch (centre at origin, Dia = 120). Use **Extrude** tool to extrude the sketch (Distance A = 8 along -ve Z).



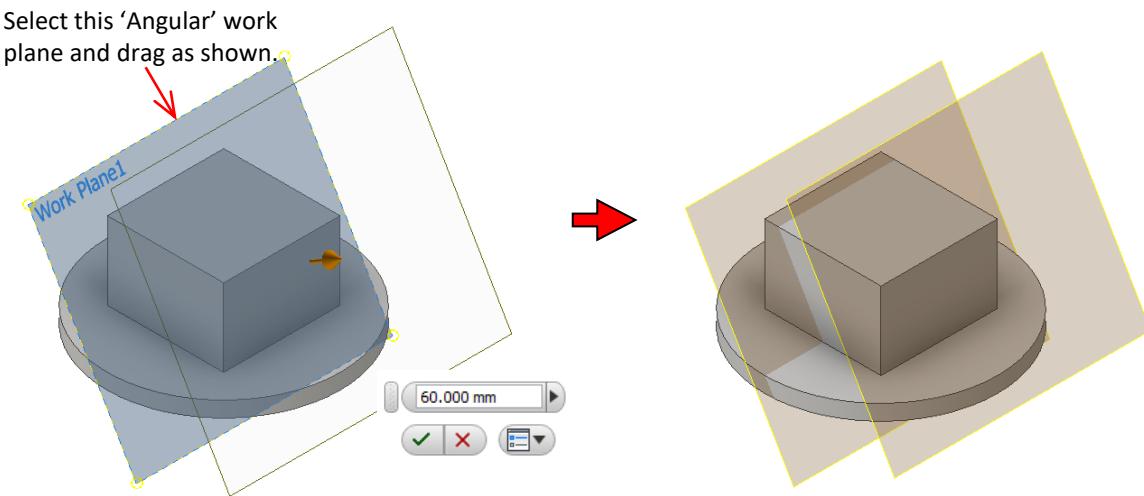
Step 2: Select the Face 'A' and start a new sketch. Draw a 60mm square profile using the **Rectangle** tool (*Two Point Center*) and locate its centre to the sketch's centre point. Extrude this profile (Distance A = 40 along +ve Z).



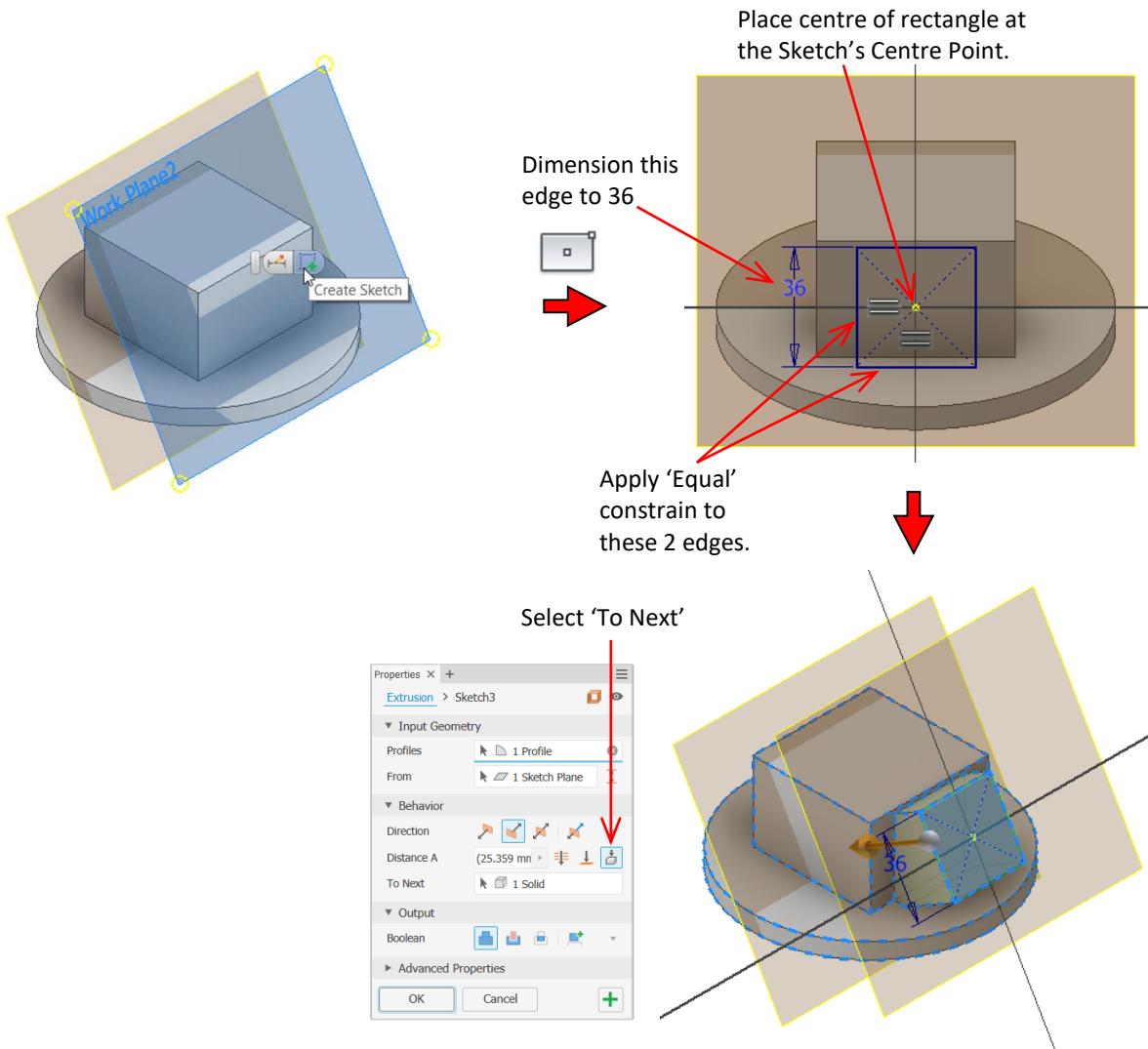
Step 3: Create a 60° Angular work plane as shown: Pick the **Work Plane** tool (*Angle to Plane around Edge*) → Select the XY Plane (or previous Face A) and Y Axis from the browser tree → Enter the angle value = 60.



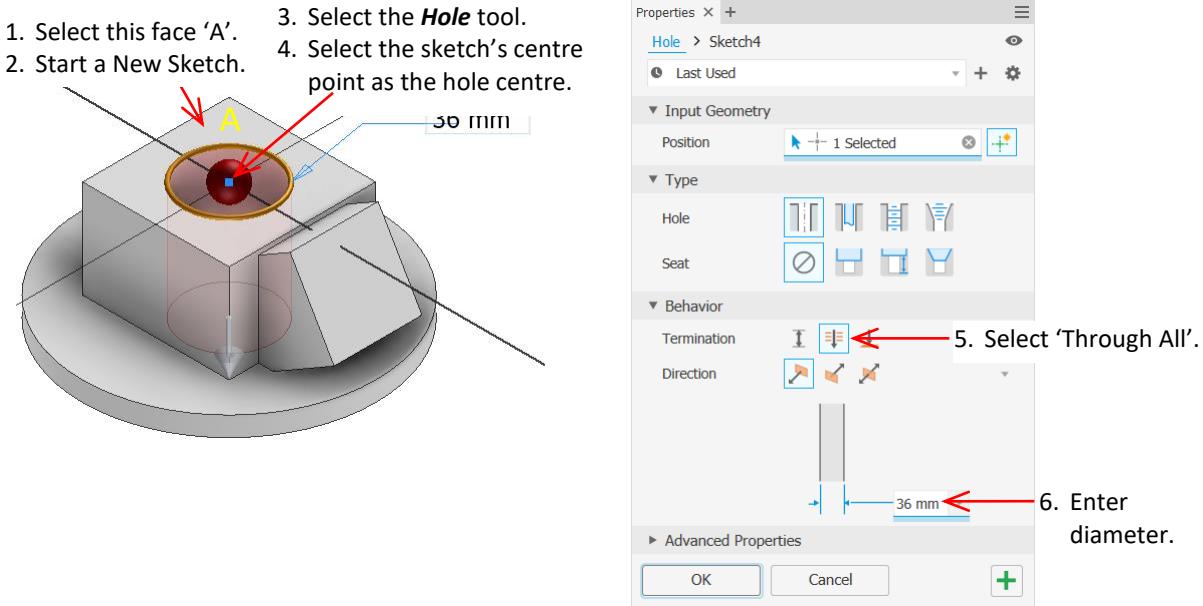
Step 4: Create an ‘Offset’ work plane as shown: Pick the **Work Plane** tool (*Offset from Plane*) → Select the ‘Angular’ work plane created in step 3 → Drag outward → Enter the offset value = 60.



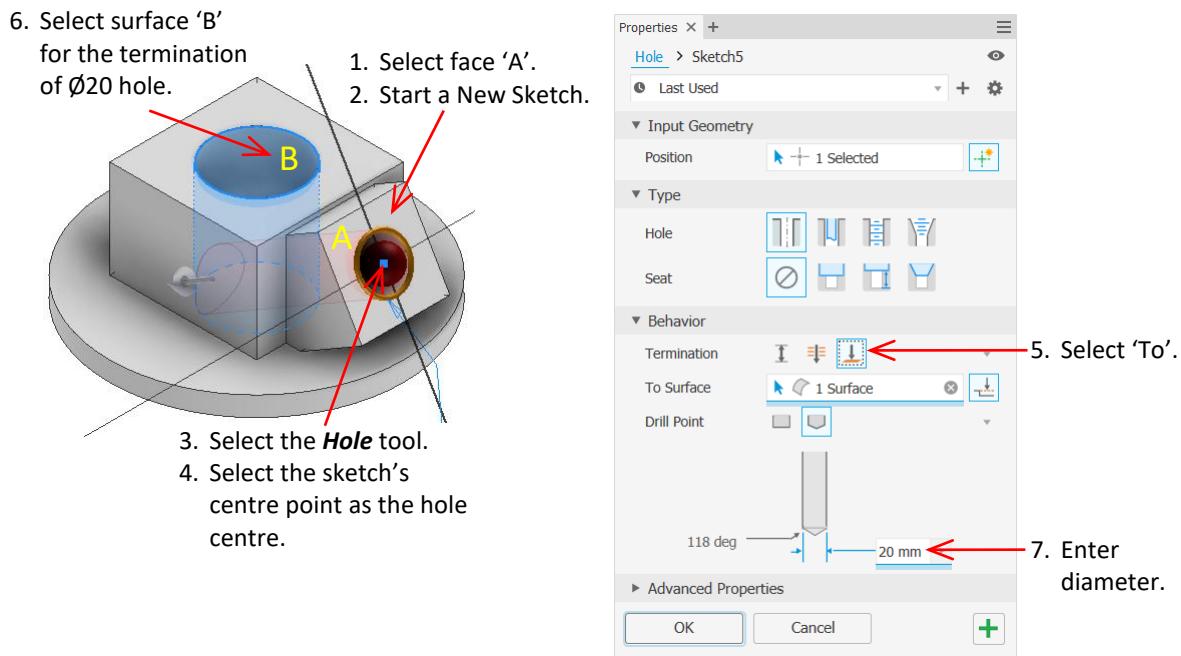
Step 5: Select the ‘Offset’ work plane created in step 4 and start a new sketch. Draw a 36mm square profile using the **Rectangle** tool (*Two Point Center*) and locate its centre to the sketch’s centre point. Extrude this profile (Distance A = ‘To Next’).



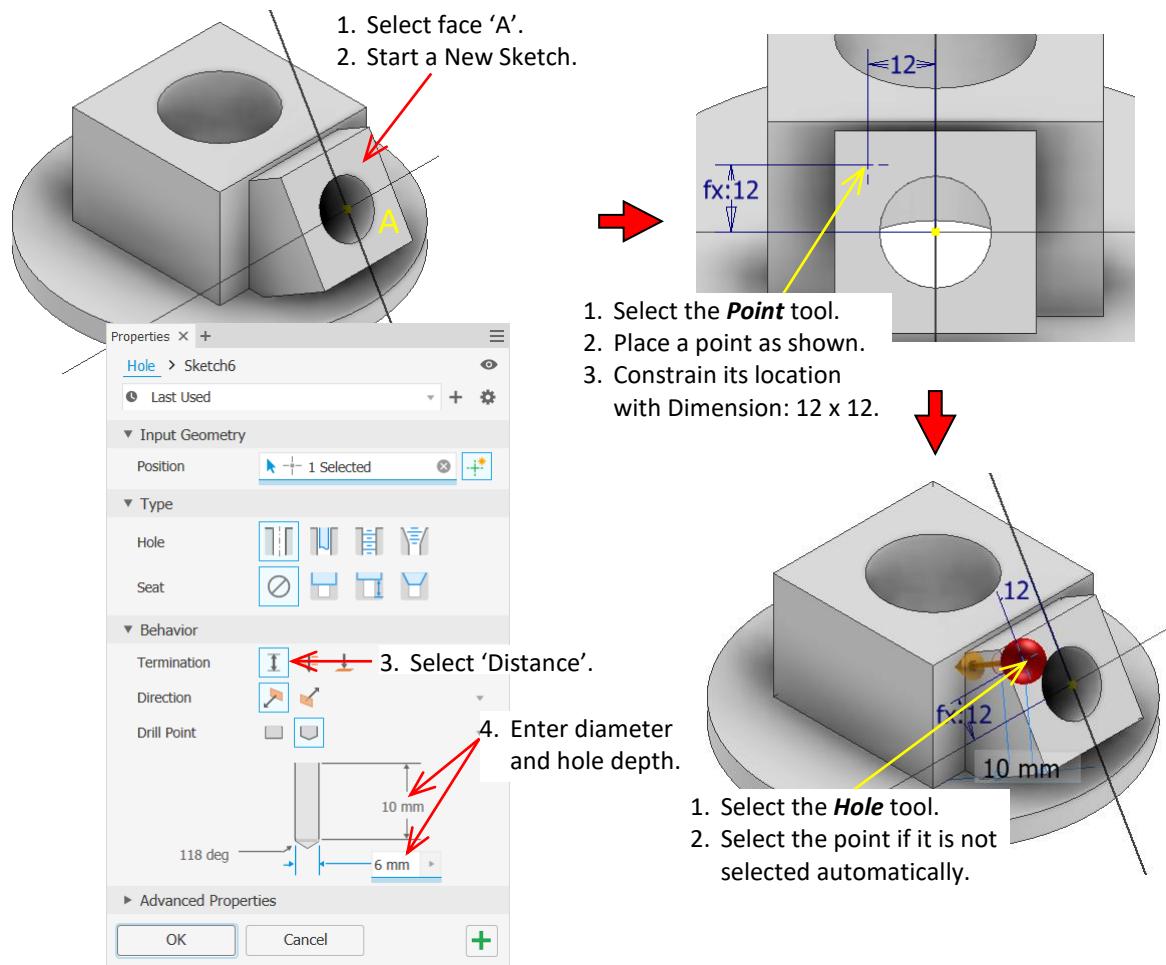
Step 6: Select Face ‘A’ and start a new sketch. Use the **Hole** tool to create a Ø36 hole at the sketch’s centre point (Termination = ‘Through All’).



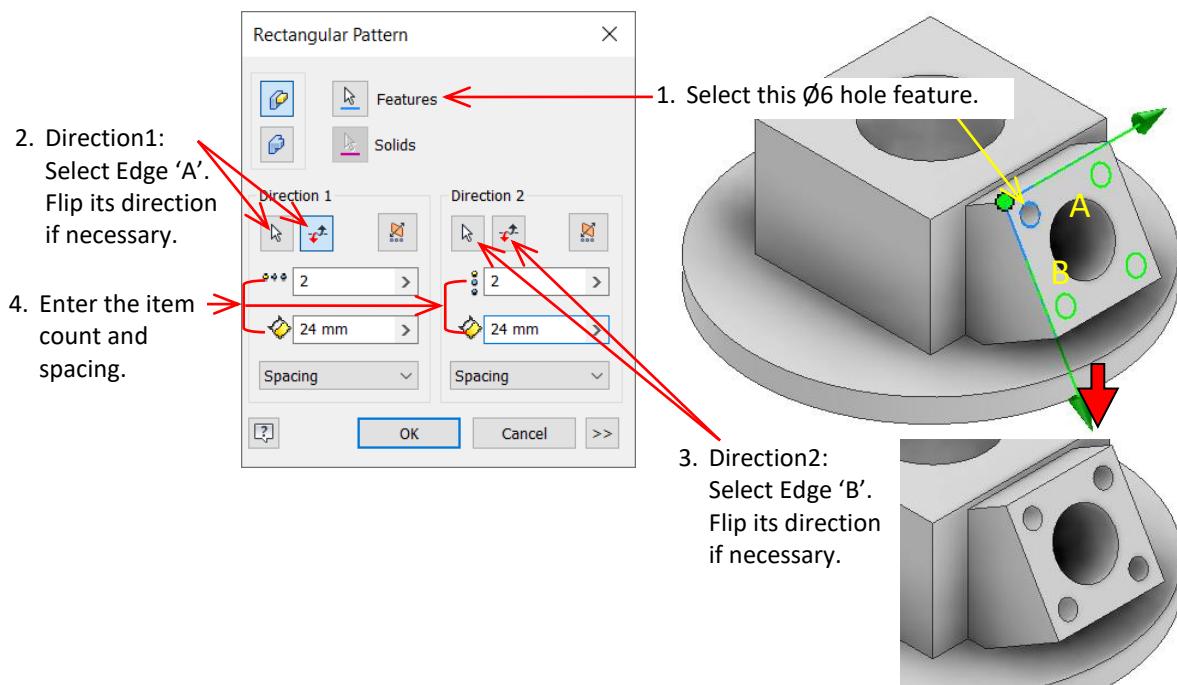
Step 7: Select the face ‘A’ and start a new sketch. Use the **Hole** tool to create a Ø20 hole at the sketch’s centre point (Termination = ‘To’, select surface ‘B’ as the termination surface).



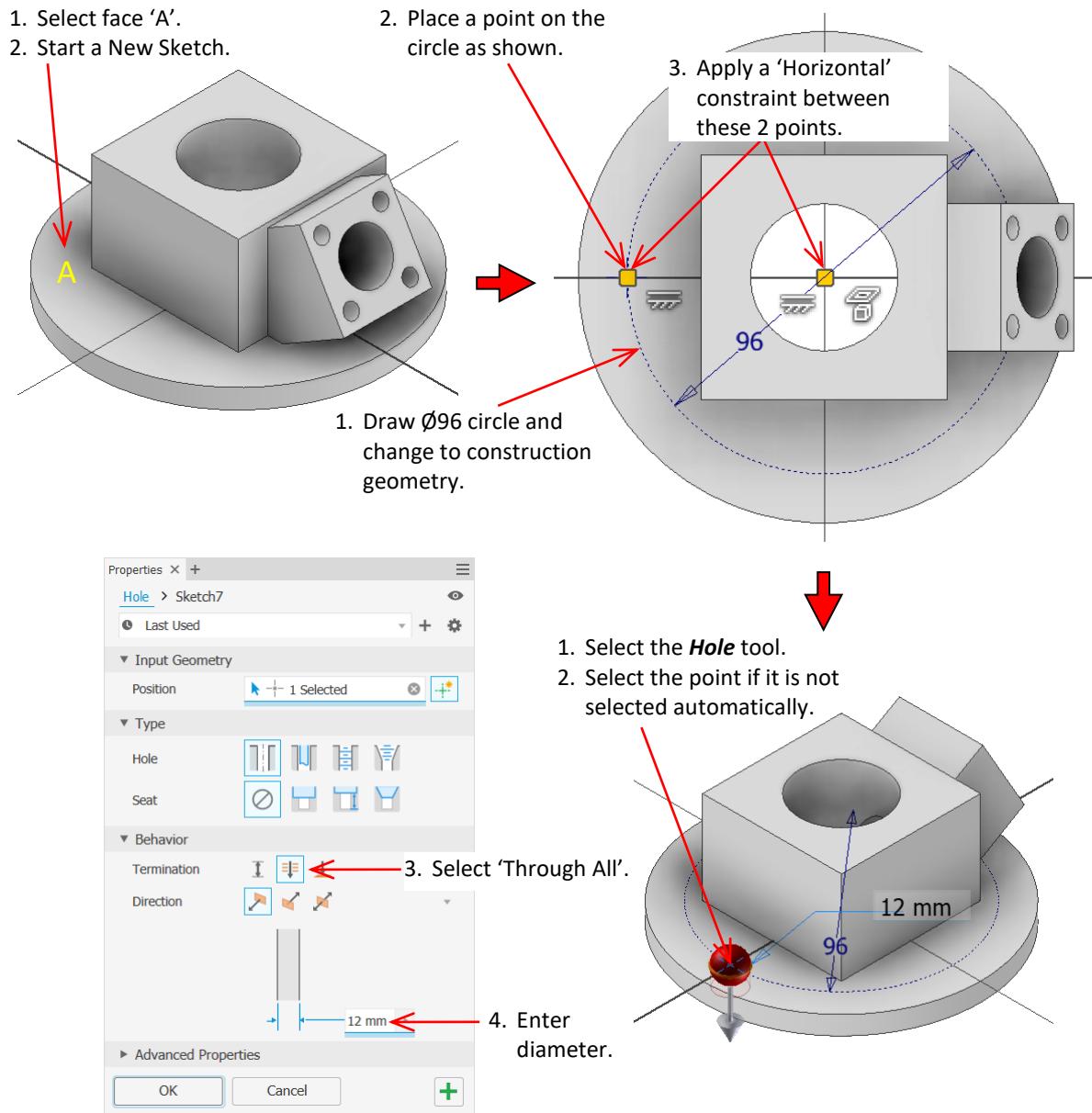
Step 8: Select the face ‘A’ and start a new sketch. Use the **Point** tool to place a centre point as shown. Constrain its location with Dimension, 12 x 12. Use the **Hole** tool to create a Ø6 x 10 deep hole at the sketch point (Termination = ‘Distance’, Diameter = 6, Depth = 10).



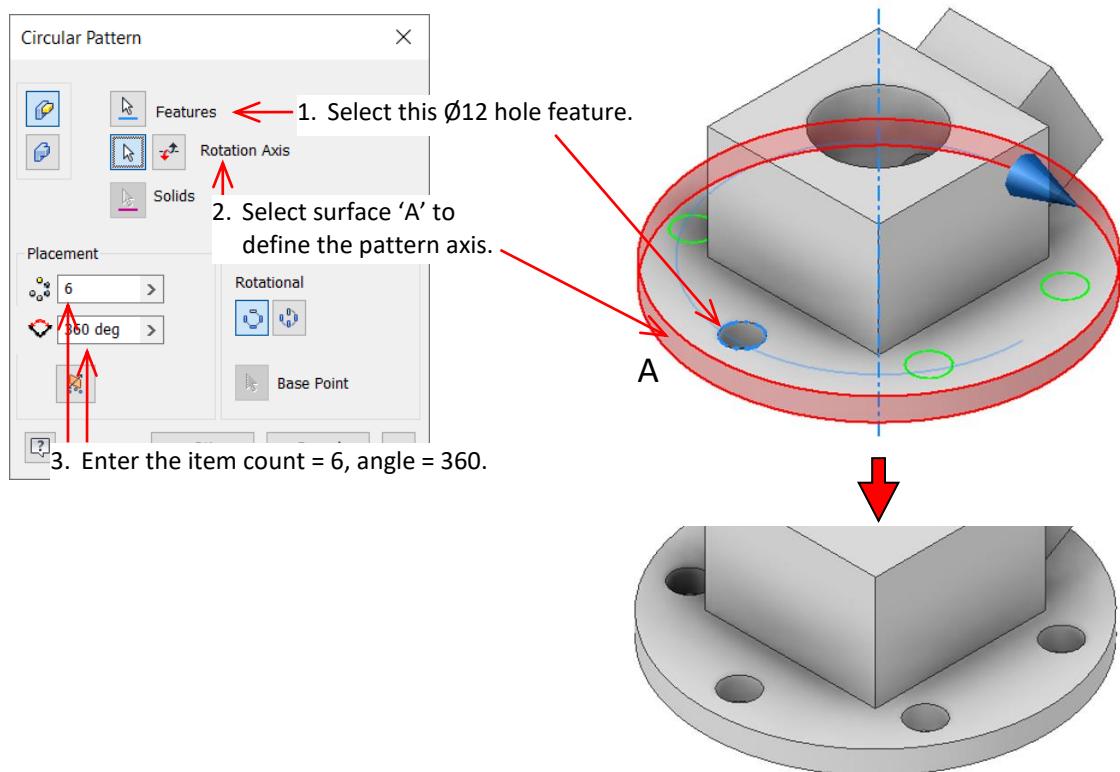
Step 9: Create a 2 x 2 rectangular pattern of the Ø6 x 10 deep hole: Select the **Rectangular Pattern** → Select the Ø6 hole feature → Select edge 'A' for pattern direction 1 (Flip its direction if required) → Select edge 'B' for pattern direction 2 (Flip its direction if required) → Enter the item count (2) and spacing (24) for both directions → Click 'OK'.



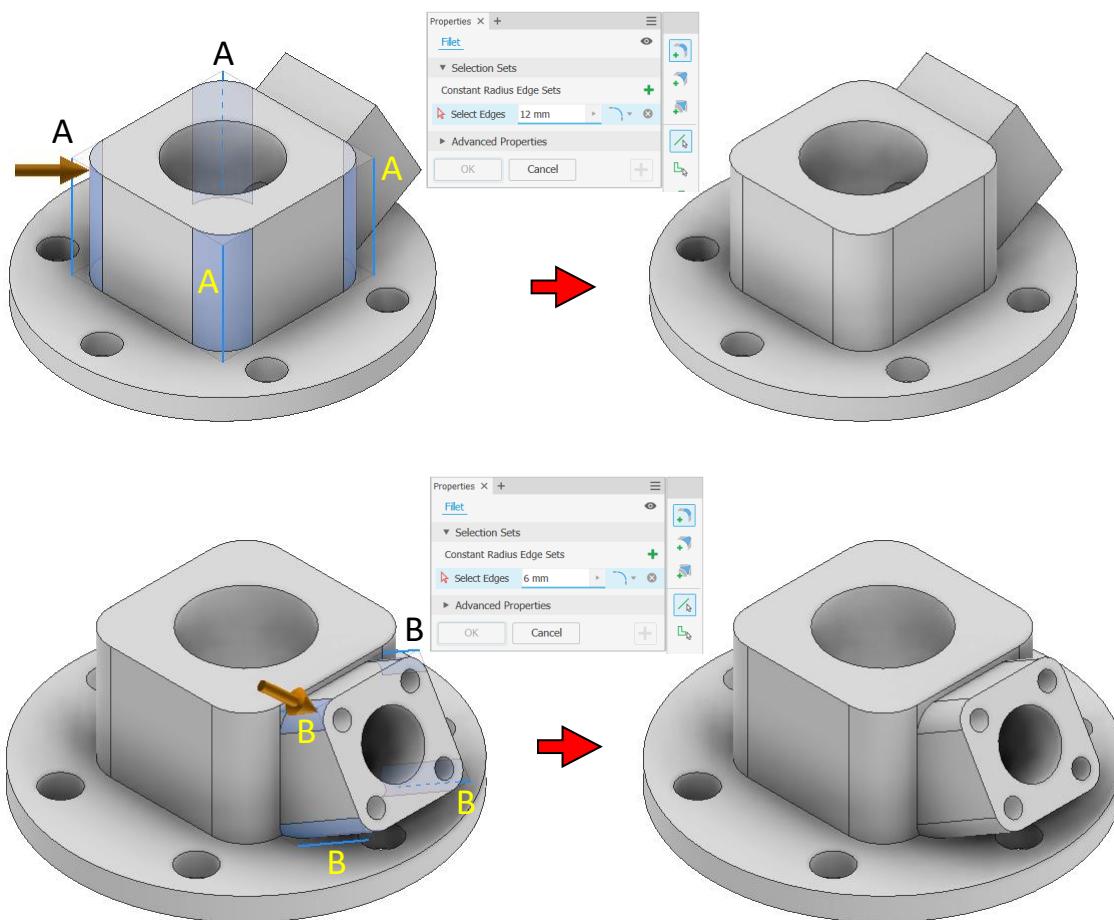
Step 10: Select the face ‘A’ and start a new sketch. Draw a circle sketch (centre at origin, Dia = 96) and change it to a construction geometry. Use the **Point** tool to place a centre point as shown. Constrain its location with ‘Horizontal’ constraint to the sketch’s centre point. Use the **Hole** tool to create a Ø12 at the sketch point (Termination = ‘Through All’, Diameter = 12).



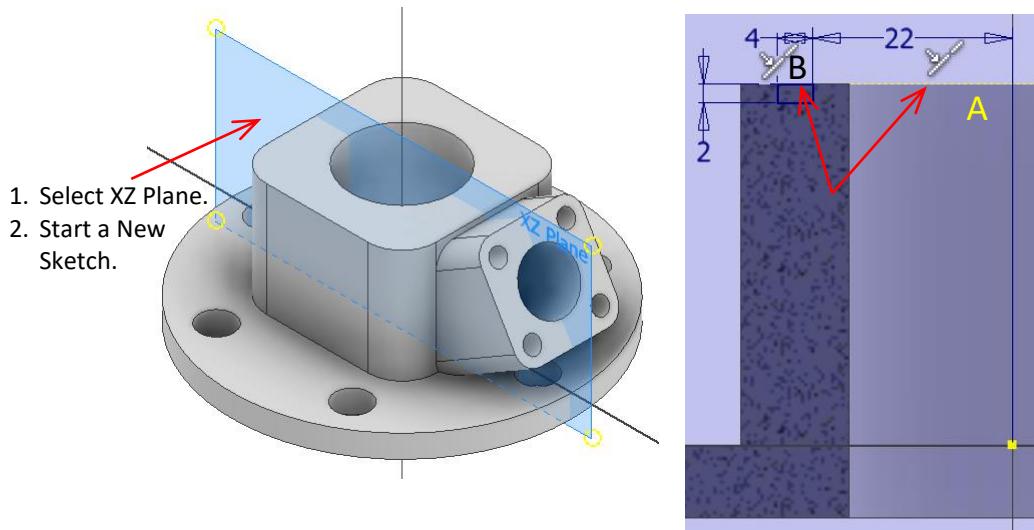
Step 11: Create a circular pattern of the Ø12 hole: Select the **Circular Pattern** → Select the Ø12 hole feature → Select surface ‘A’ for rotation axis → Enter the item count (6) and angle (360) → Click ‘OK’.



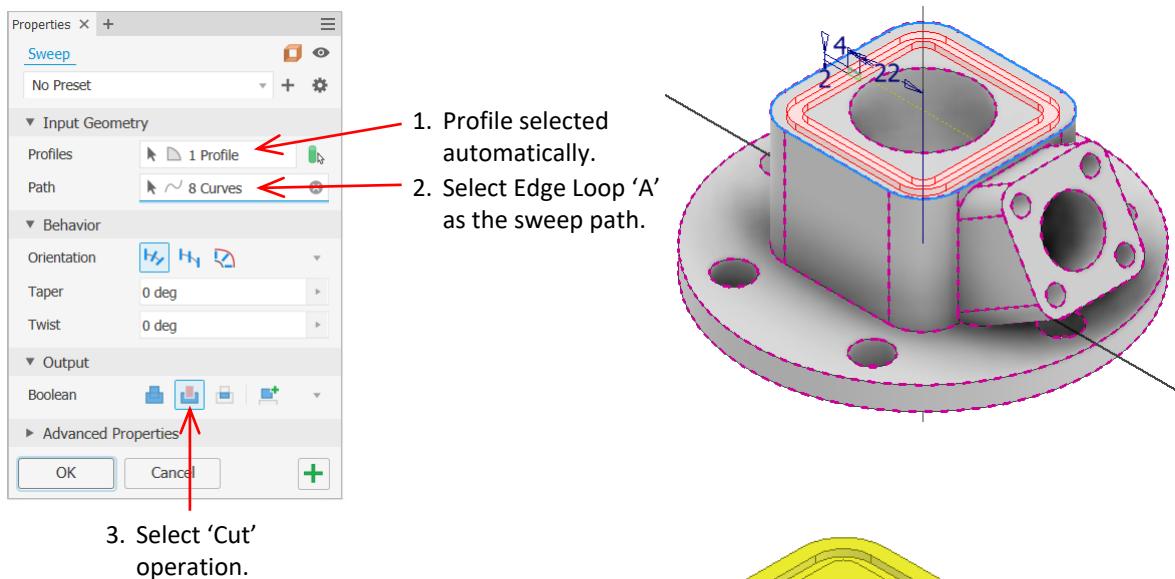
Step 12: Use the **Fillet** tool: Round the 4 edges 'A' with radius 12. Round the 4 edges 'B' with radius 6.



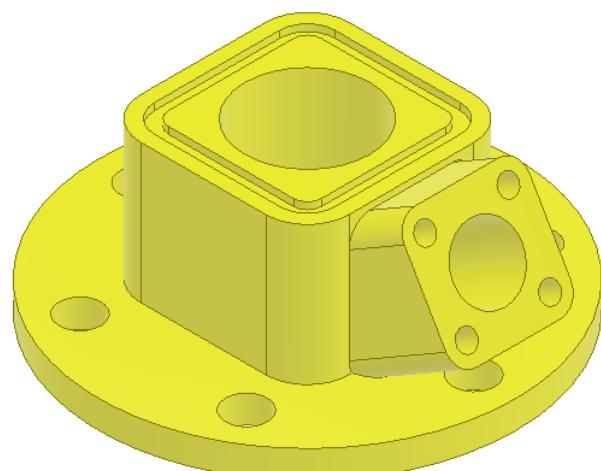
Step 13: Start a new sketch on XZ Plane: Select **Project Geometry** tool → Select edge ‘A’ → Change edge ‘A’ to a construction geometry. Select **Rectangle** tool (*Two Point*) → Draw a 4 x 2 rectangle as shown → Apply Collinear constraint to edge ‘B’ and edge ‘A’.



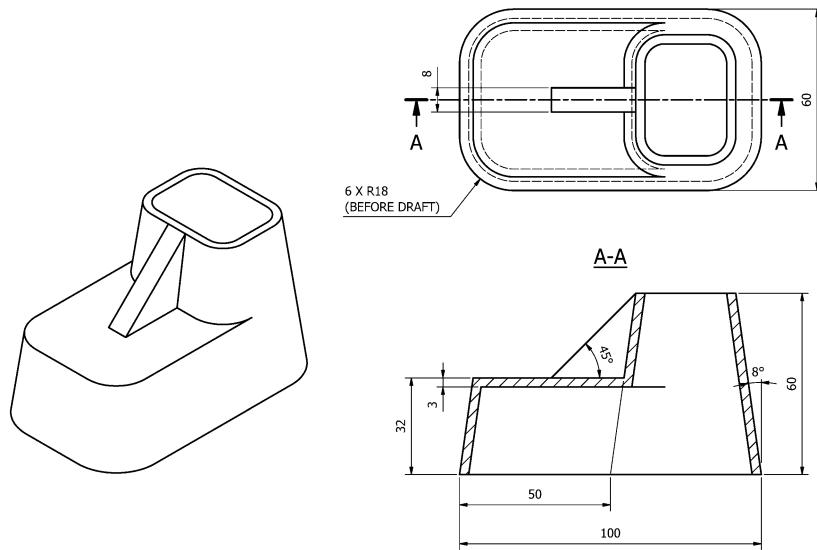
Step 14: Select **Sweep** tool → Select edge loop ‘A’ as the sweep path → Select ‘Cut’ operation → Click ‘OK’.



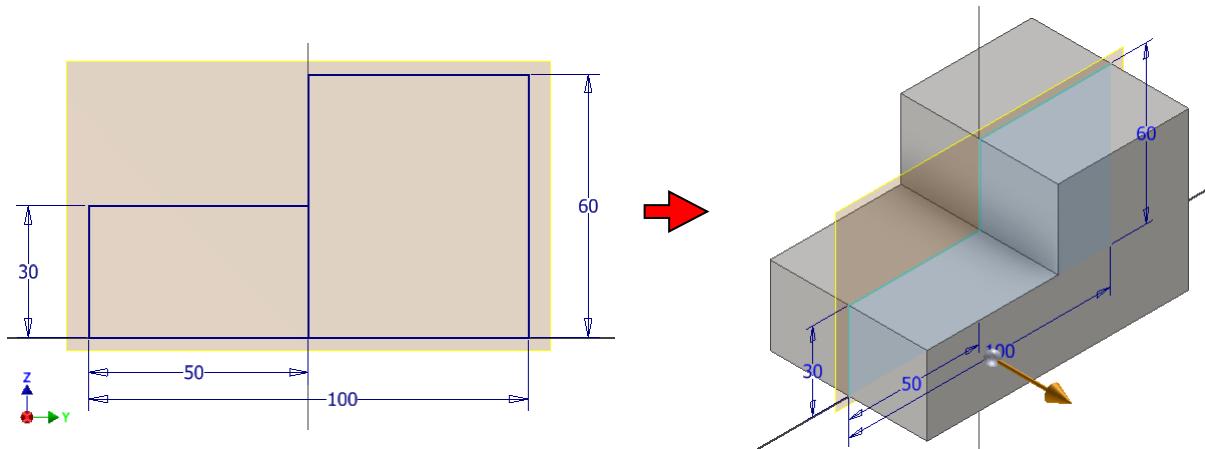
Completed Solid Model:



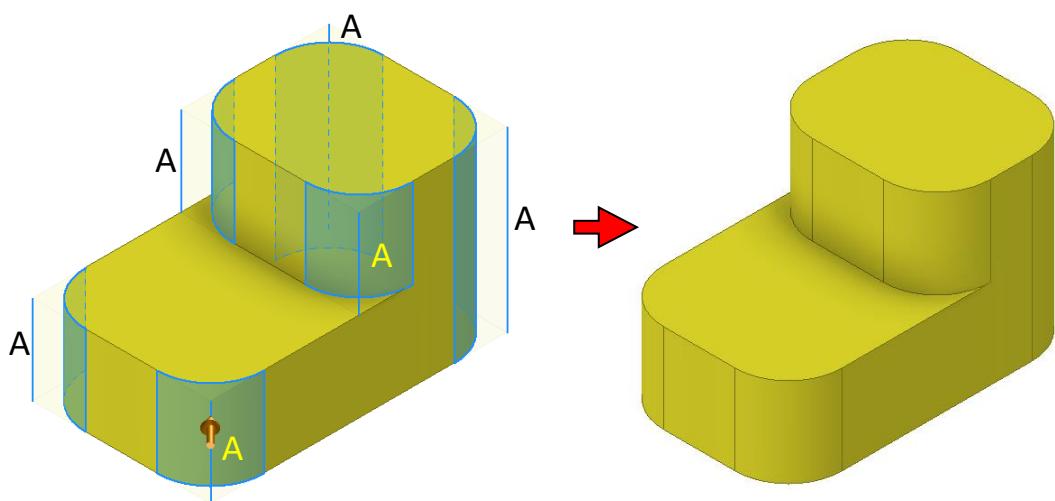
7.32.2 Part Model B



Step 1: Start a new sketch on YZ Plane: Select **Rectangle** tool (*Two Point*) → Draw two rectangle profiles (one corner point to be at the sketch centre point) → Dimension as shown → Select **Extrude** tool → Select the two rectangle profiles → (Direction = ‘Symmetric’; Distance A = 60).



Step 2: Use the **Fillet** tool: Round the 8 edges ‘A’ with radius 18.

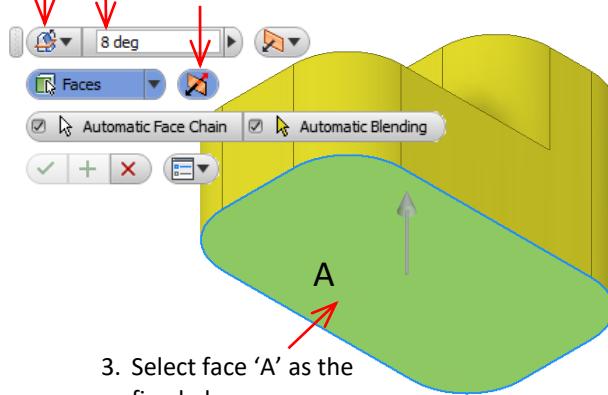


Step 3: Select the **Draft** tool → Enter the draft angle value = 8 → Select Fixed Plane → Select face ‘A’ → Flip the Pull Direction → Select chain faces ‘B’ to draft → Click ‘OK’.

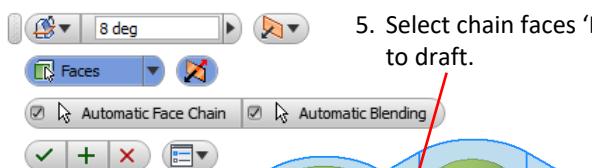
2. Select Fixed Plane from this list.

1. Enter draft angle value.

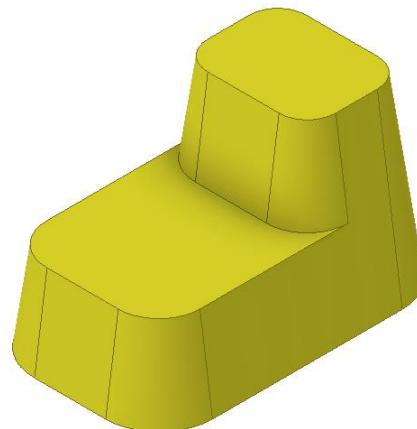
4. Click to flip pull direction.



3. Select face ‘A’ as the fixed plane.



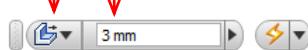
5. Select chain faces ‘B’ to draft.



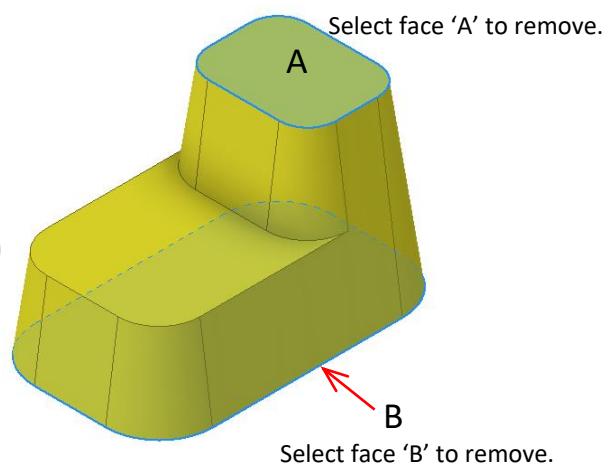
Step 4: Select the **Shell** tool → Select Inside → Select face ‘A’ and ‘B’ to remove faces → Enter Thickness = 3 → Click ‘OK’.

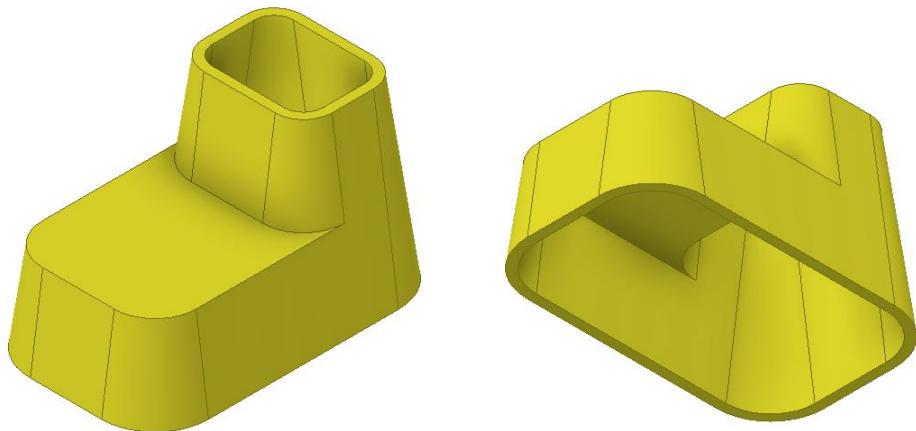
1. Select Inside.

3. Enter Thickness value =3.

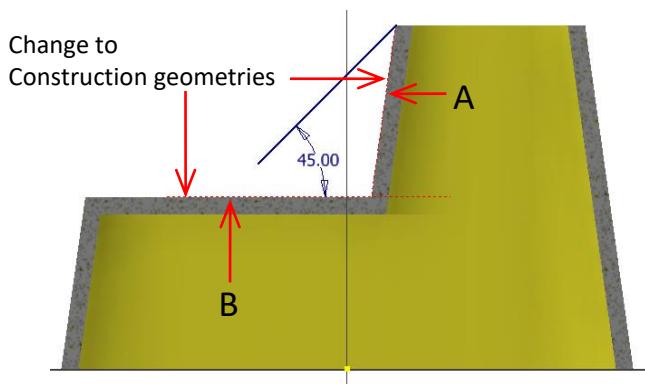


2. Select face ‘A’ and ‘B’ to remove faces.

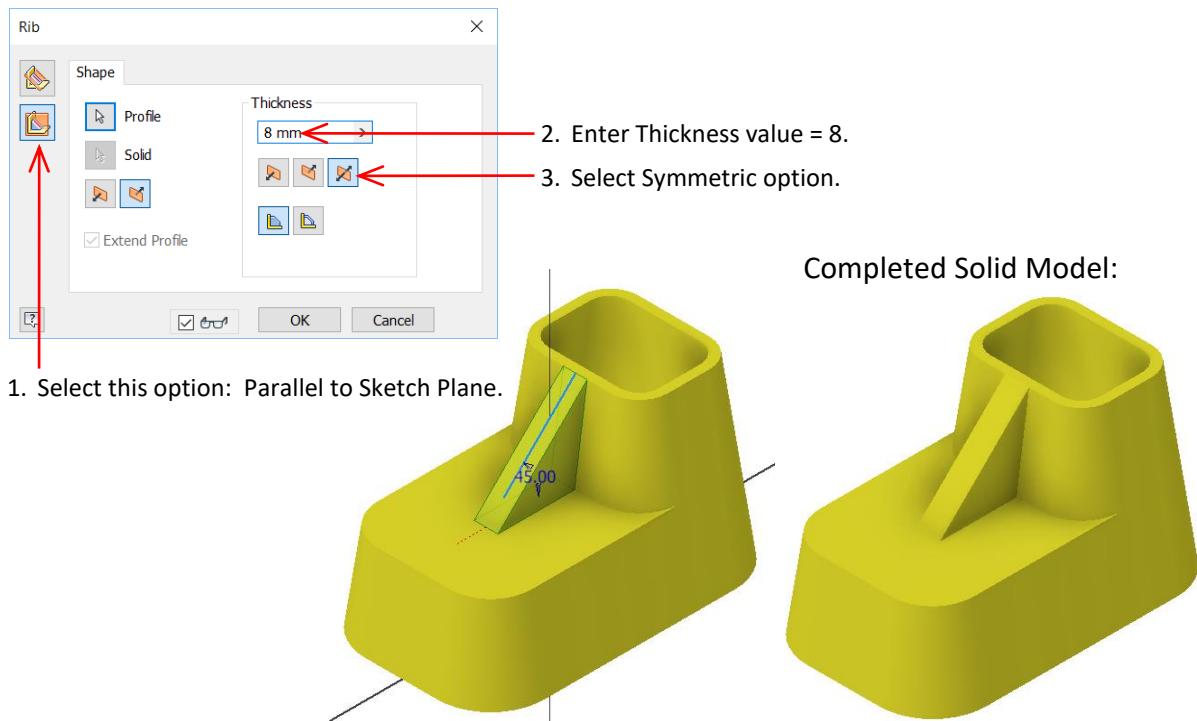




Step 5: Start a new sketch on YZ Plane: Select **Project Geometry** tool → Select edge ‘A’ and ‘B’ → Change the projected edges ‘A’ and ‘B’ to Construction geometries → Select **Line** tool → Draw a line and constrain as shown.



Step 6: Select **Rib** tool → Select Parallel to Sketch Plane → Enter Thickness value = 8 → Select Symmetric option → Click ‘OK’.



Tutorial 7

- Create part model for the below figures. All sketches are to be fully constrained. Features should be created such that they can be modified at a later stage.

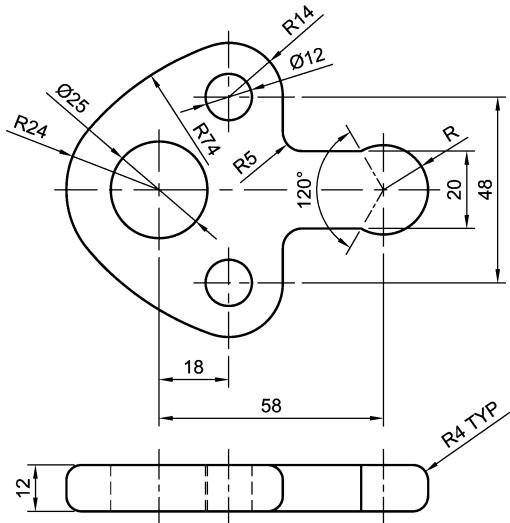


Figure 7-Q1a

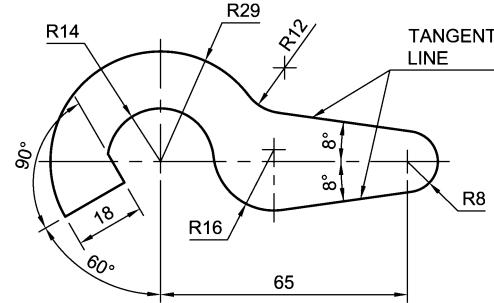


Figure 7-Q1b

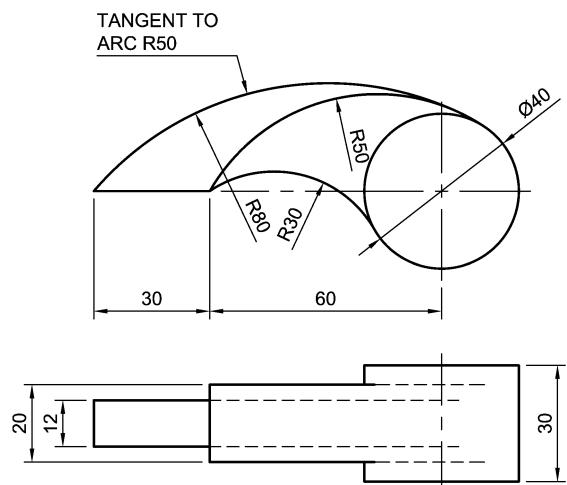


Figure 7-Q1c

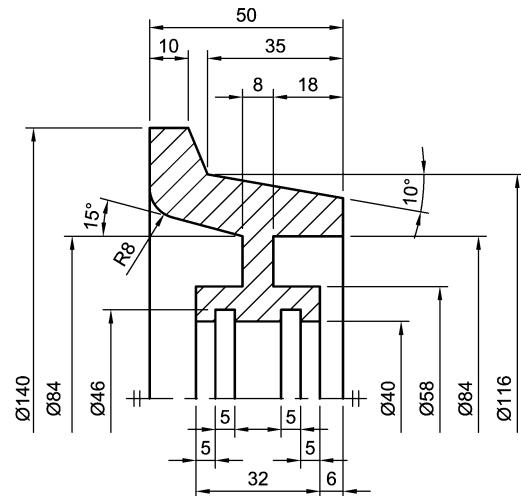


Figure 7-Q1d

2. Create part model for the below figures. All sketches are to be fully constrained.
Features should be created such that they can be modified at a later stage.

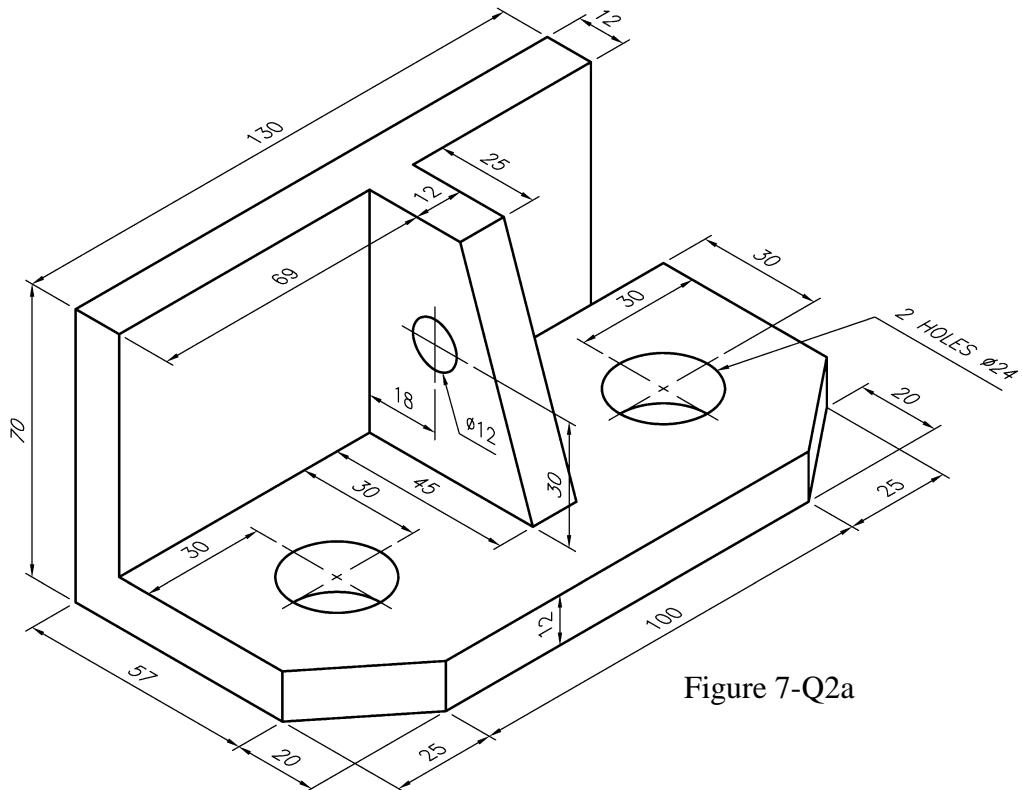


Figure 7-Q2a

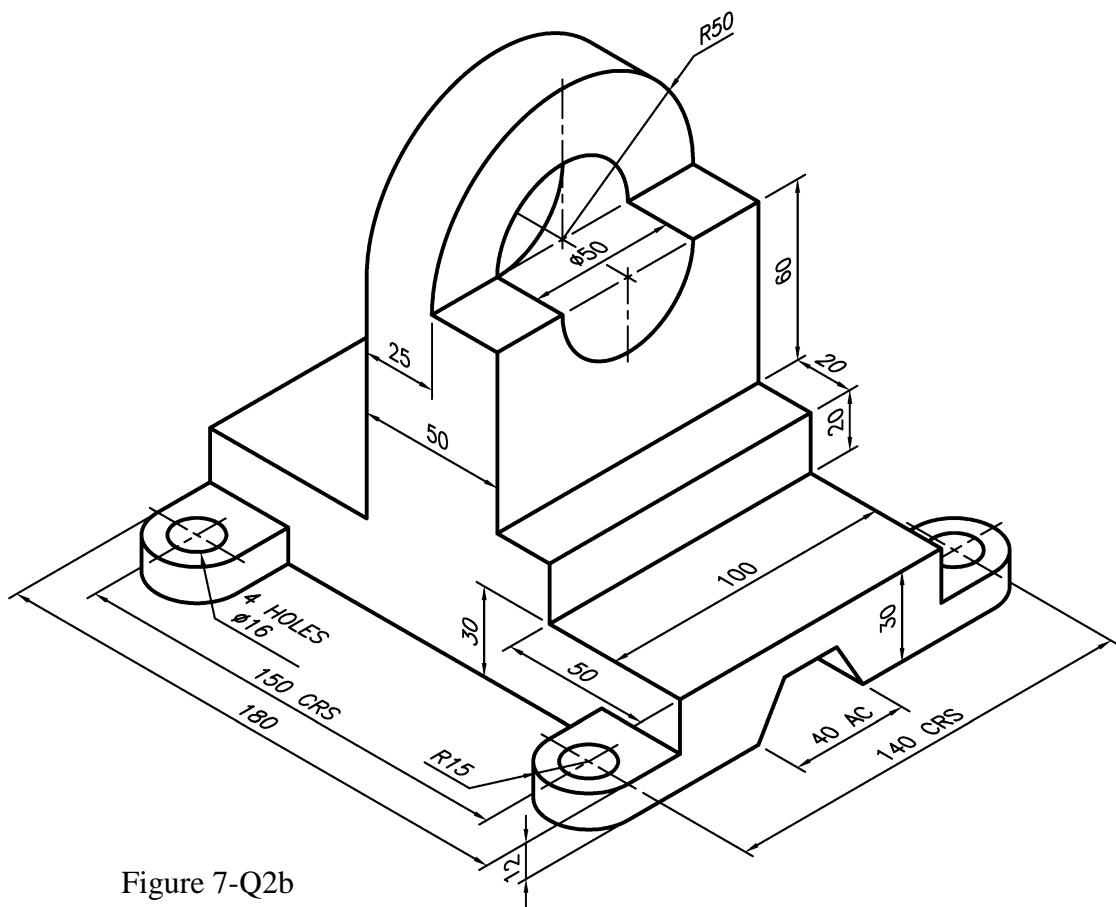


Figure 7-Q2b

3. Create part model for the below figures. All sketches are to be fully constrained. Features should be created such that they can be modified at a later stage.

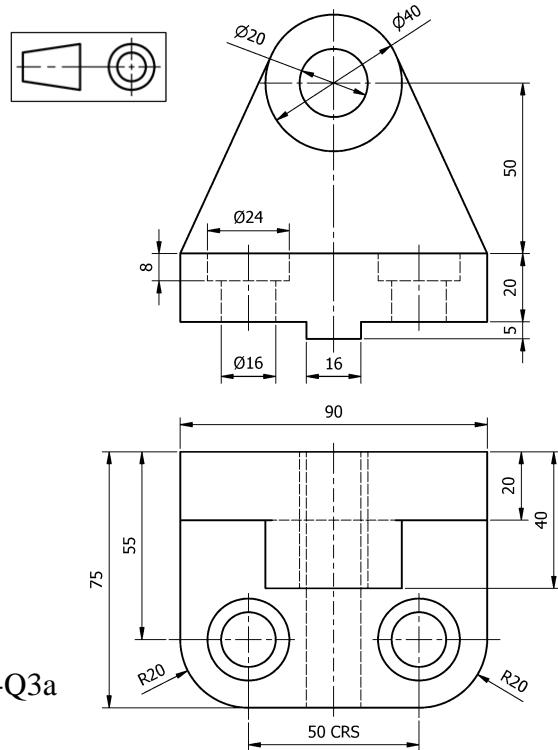


Figure 7-Q3a

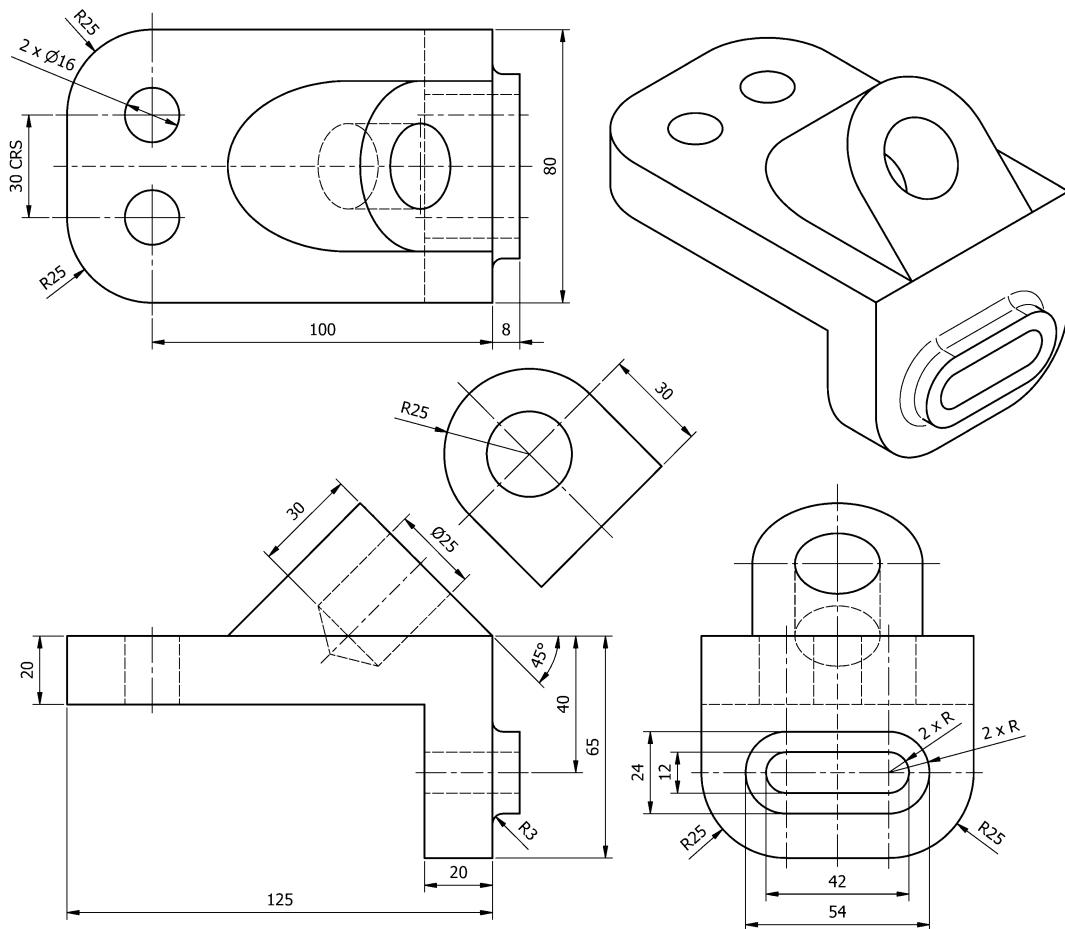


Figure 7-Q3b

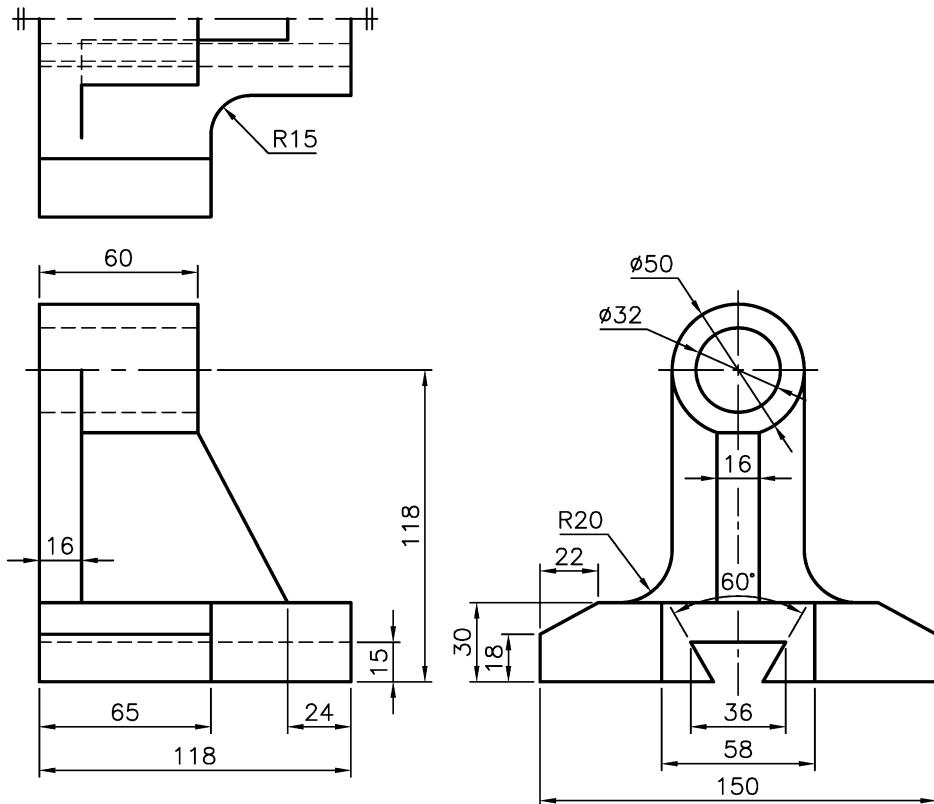


Figure 7-Q3c

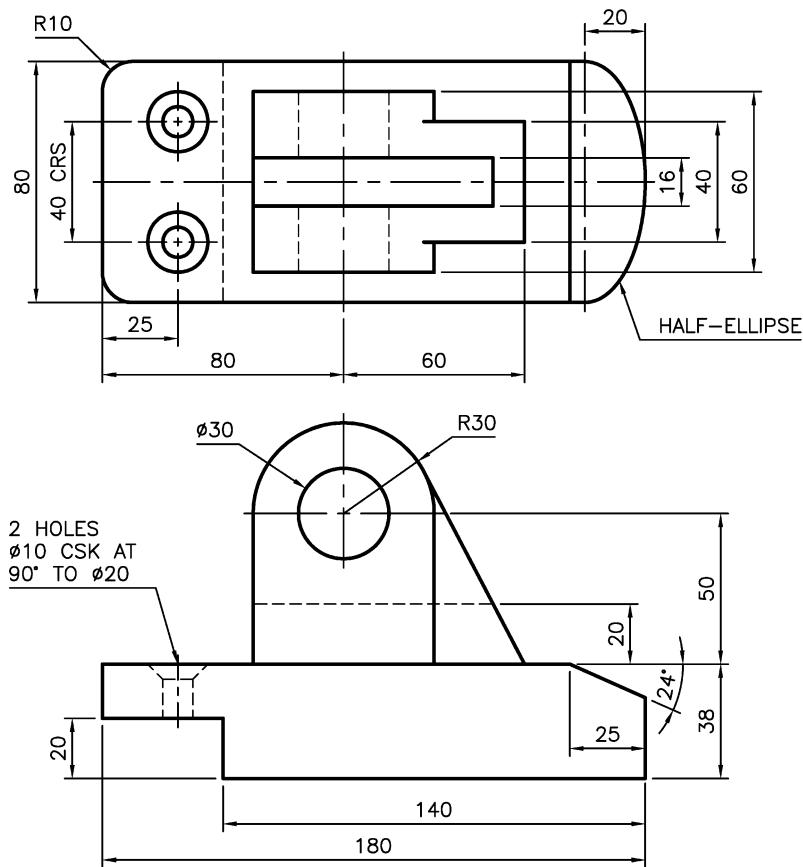


Figure 7-Q3d

4. Create solid model for the below parts. All sketches are to be fully constrained.
Features should be created such that they can be modified at a later stage.

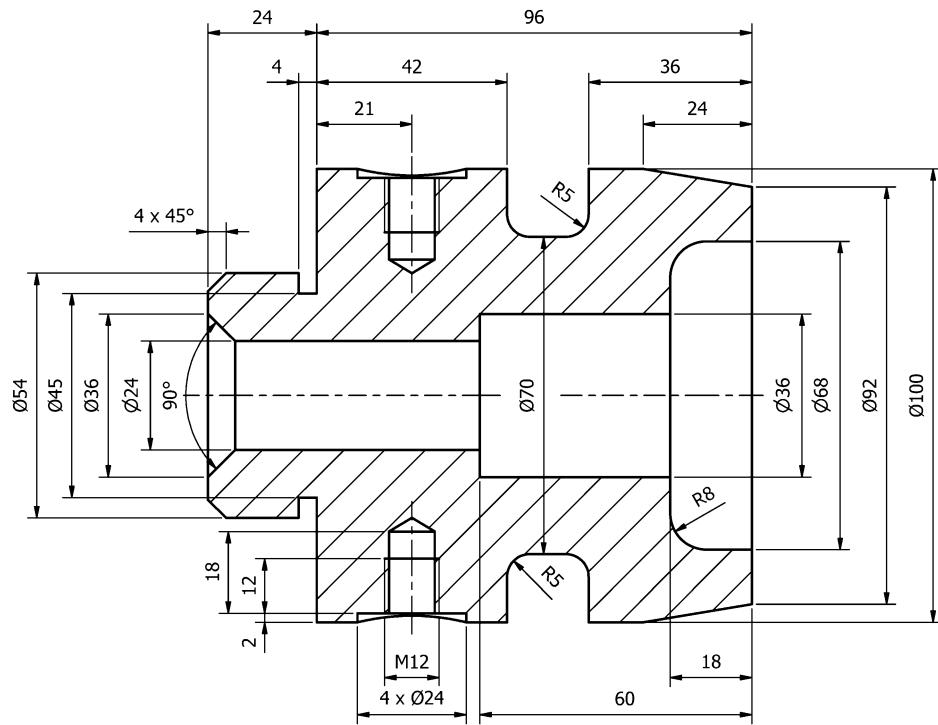


Figure 7-Q4a. Part 1 - Spindle

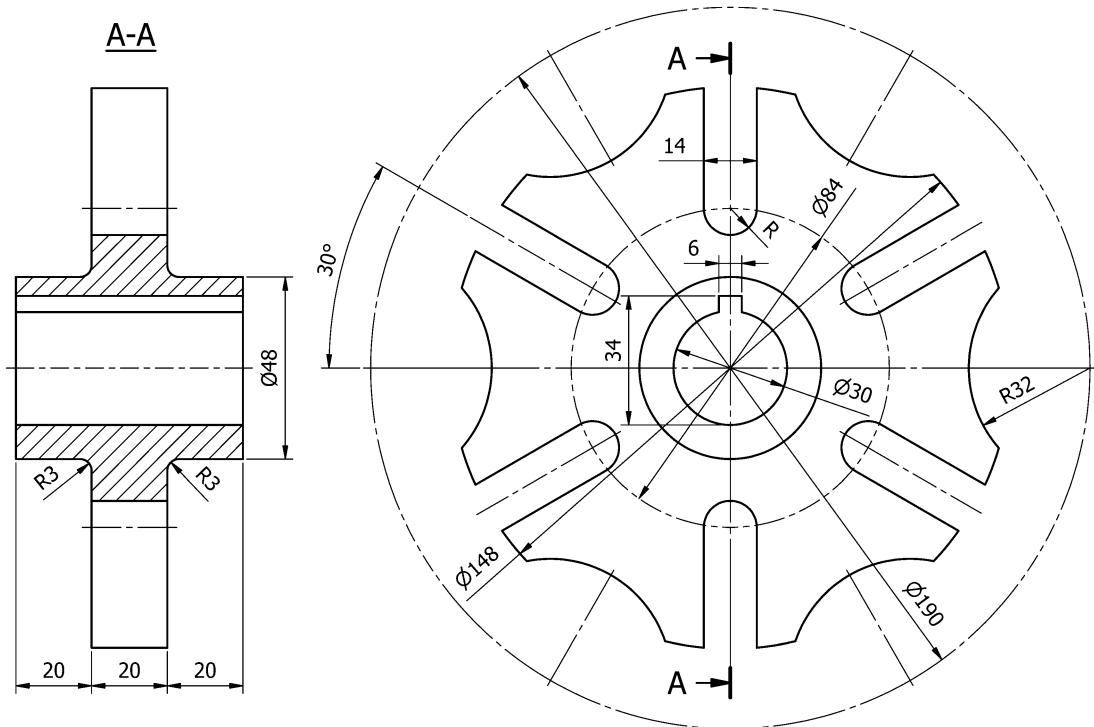
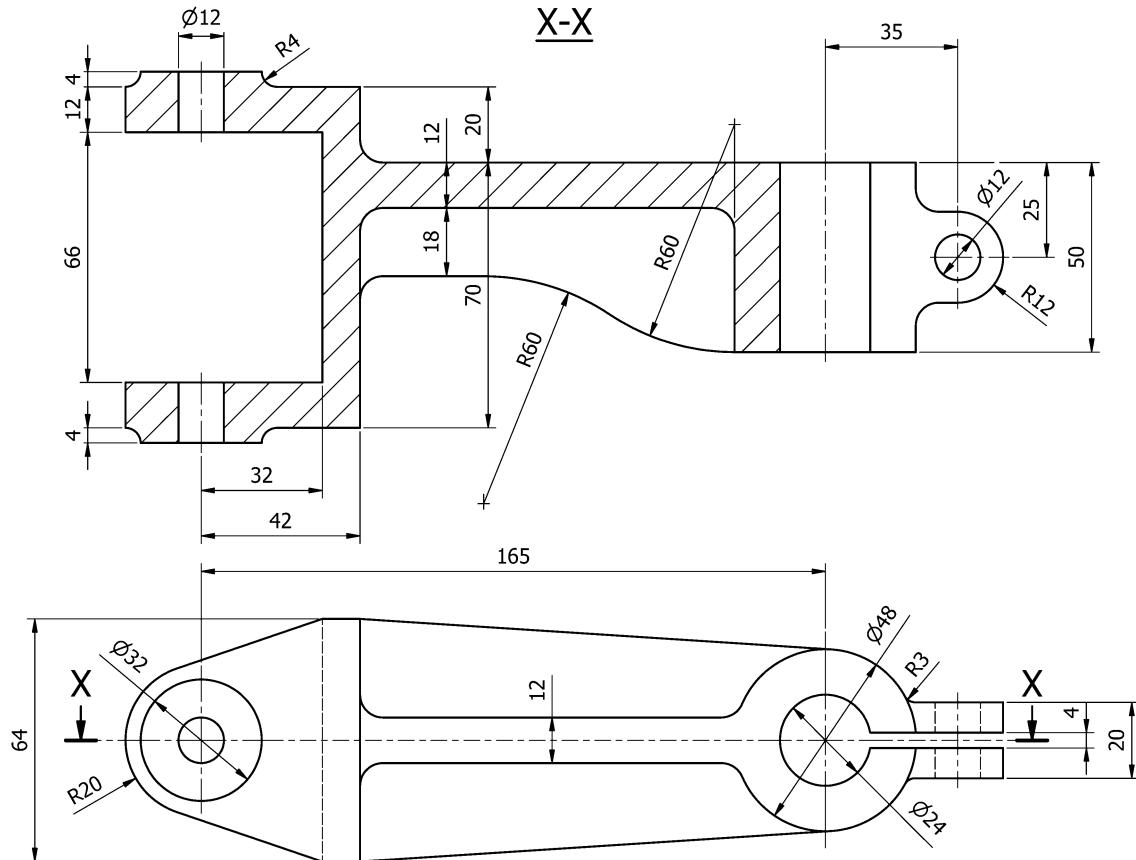


Figure 7-Q4b. Part 2 - Geneva Wheel



ALL UNSPECIFIED RADII TO BE R6.

Figure 7-Q4c. Part 3 – Roller Support

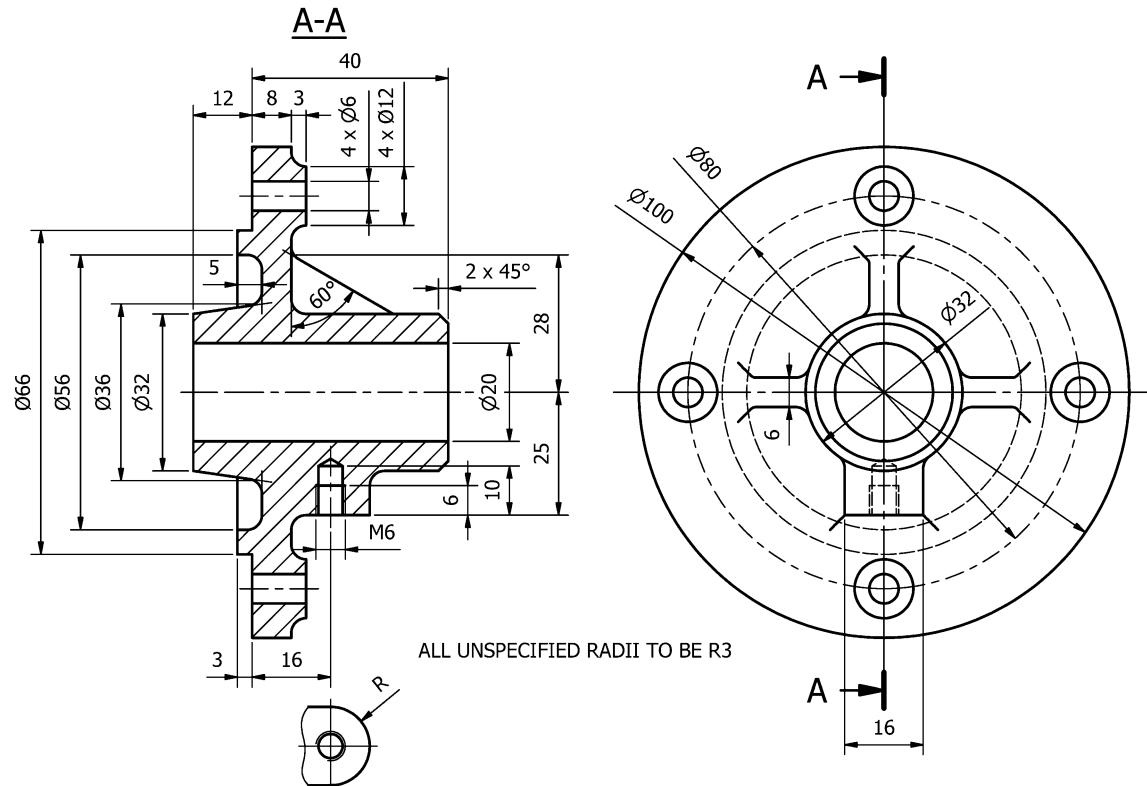


Figure 7-Q4d. Part 4 - Coupling

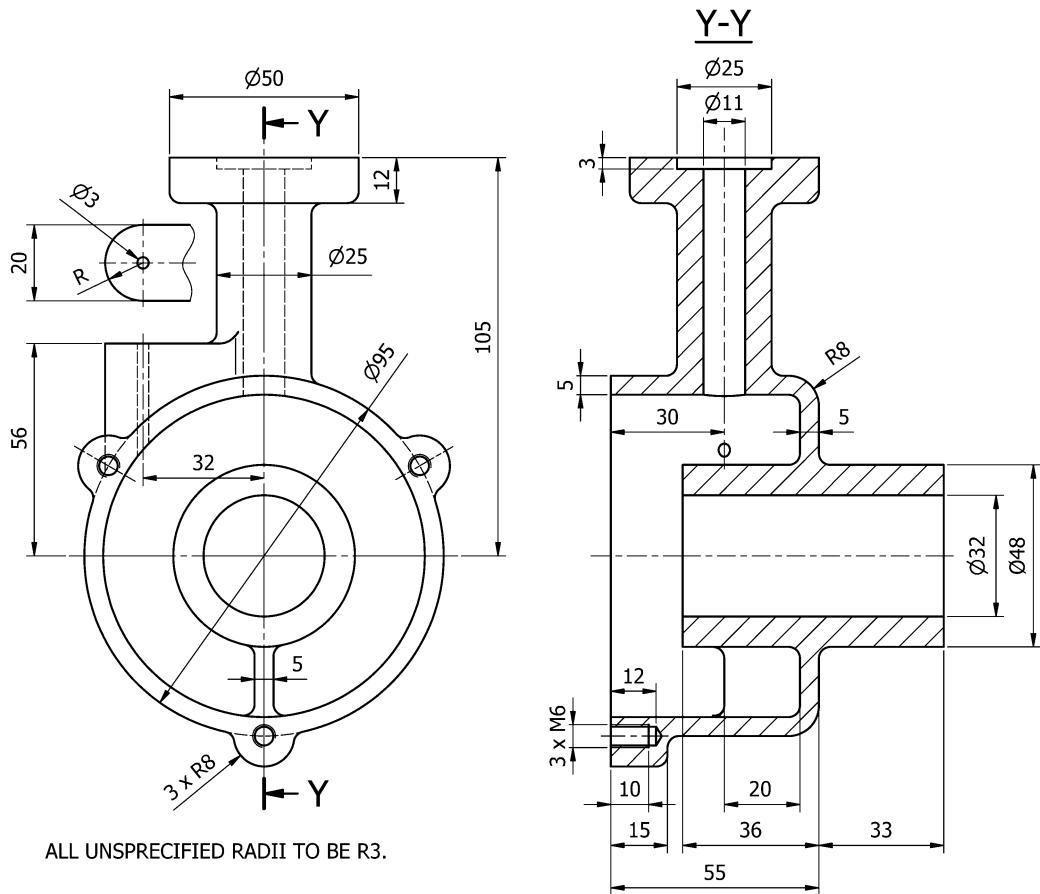


Figure 7-Q4e. Part 5 – Gear Housing

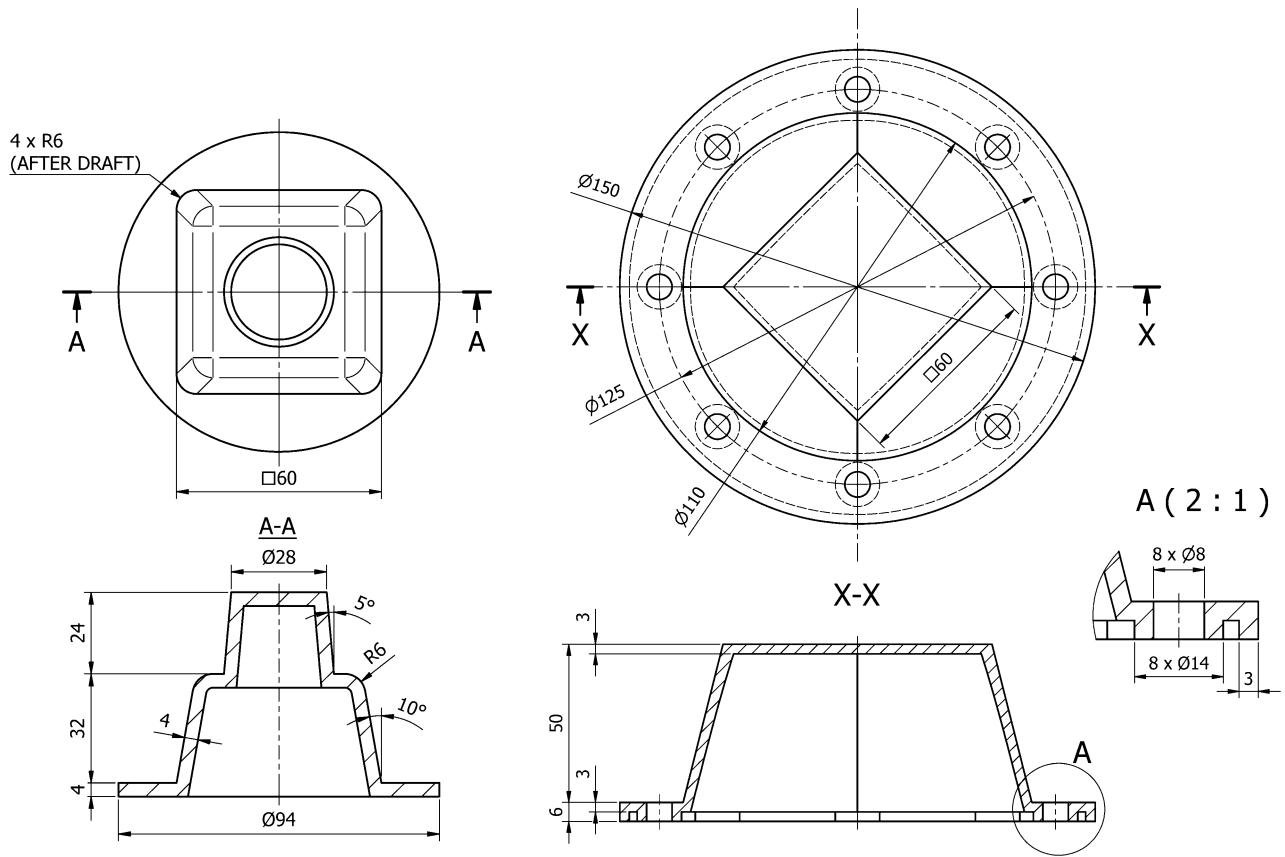


Figure 7-Q4f. Part 6 – Cover

Figure 7-Q4g. Part 7 – End Cap