Chapter 6: Dimensions and Annotations

In this chapter, you will learn to do the following:

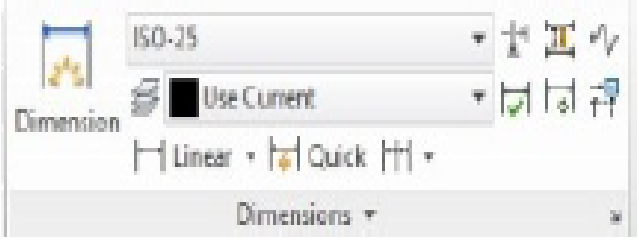
- Create Dimensions
- Create Dimension Style
- Add Leaders
- Add Dimensional Tolerances
- Add Geometric Tolerances
- Edit Dimensions

Dimensioning

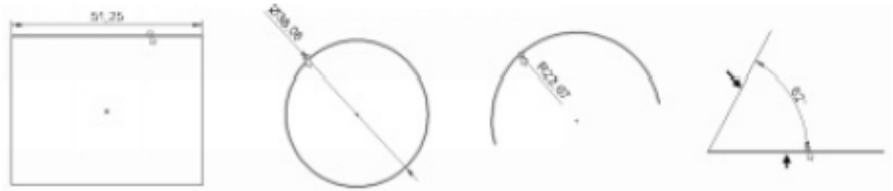
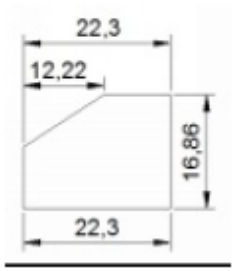
In previous chapters, you learned how to draw shapes of various objects and create drawings. However, while creating a drawing, you also need to provide the size information. You can provide the size information by adding dimensions to the drawings. In this chapter, you will learn how to create various types of dimensions. You will also learn about some standard ways and best practices of dimensioning.

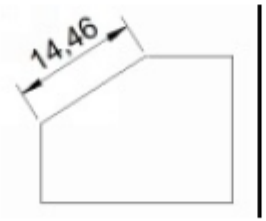
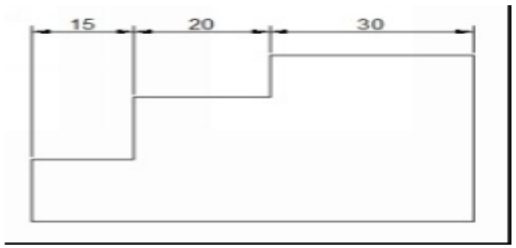
Creating Dimensions

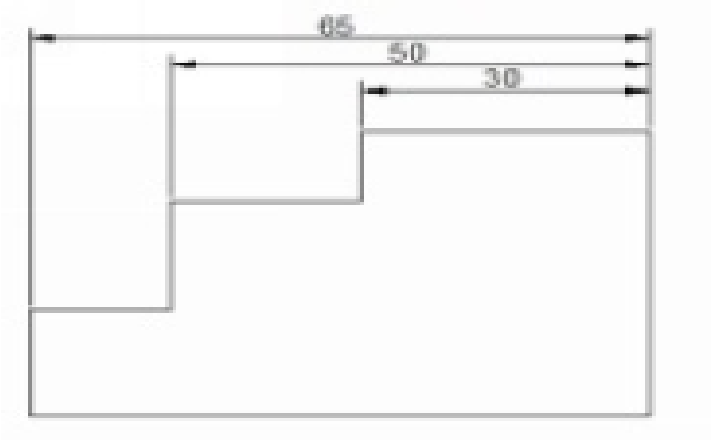
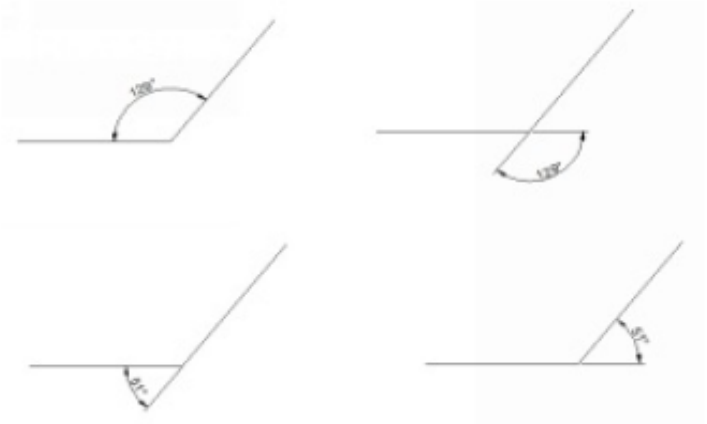
In AutoCAD, there are many tools available for creating dimensions. You can access these tools from the Ribbon, Command line, and Menu Bar.

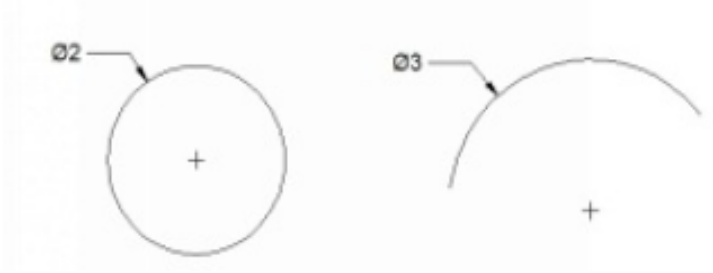
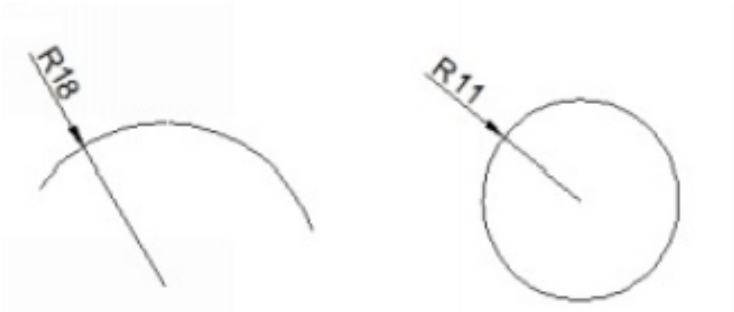
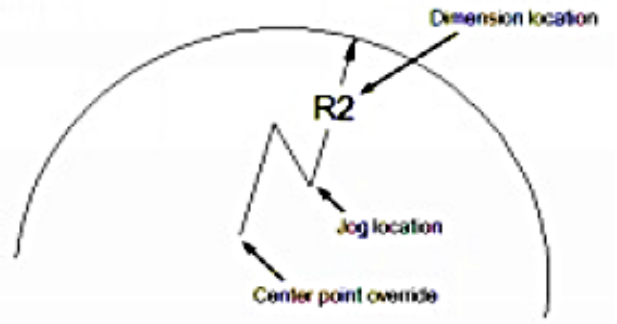


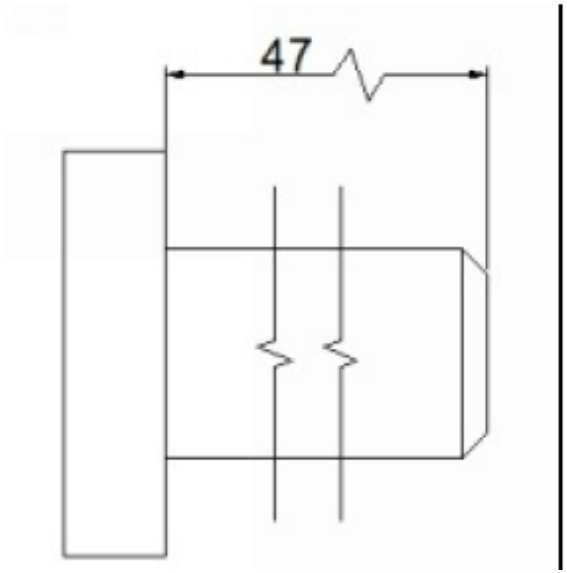
The following table gives you the functions of various dimensioning tools.

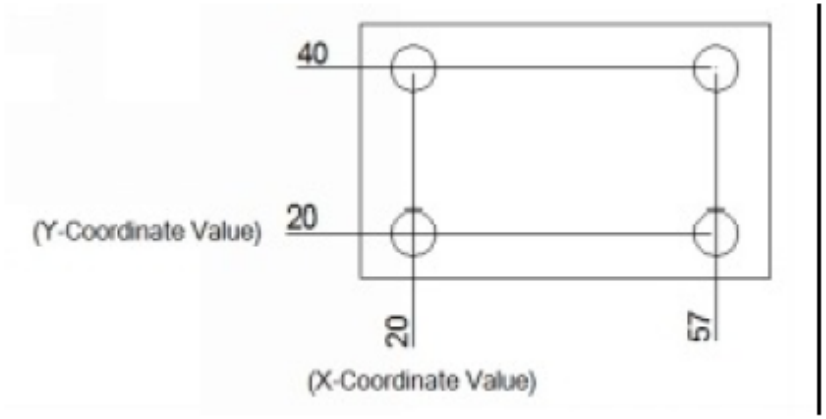
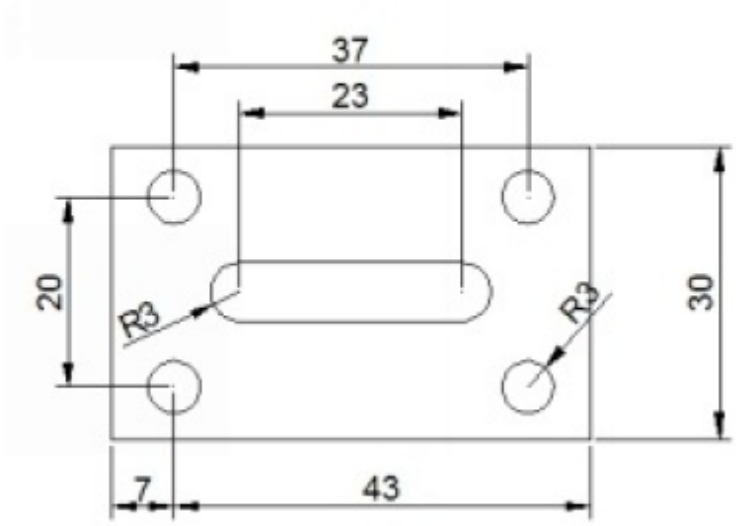
Tool	Shortcut	Function
Dimension 	DIM	This tool creates a dimension based on the selected geometry. 
Linear 	DLI	This tool creates horizontal and vertical dimensions. 

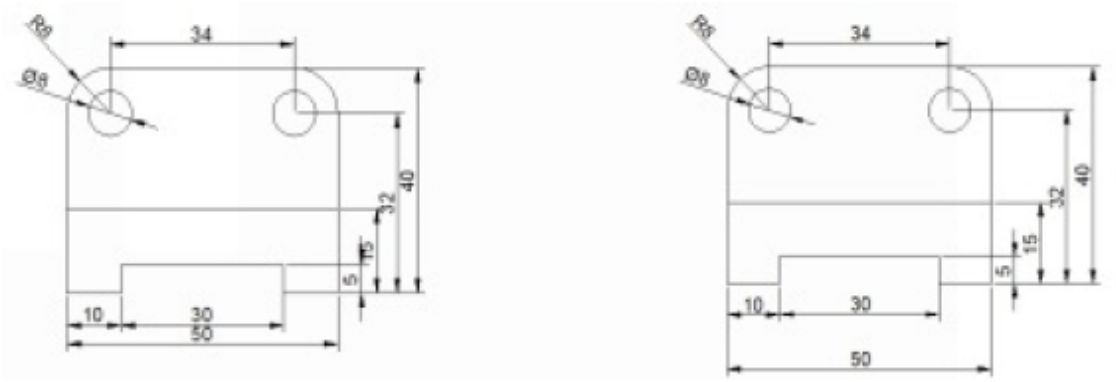
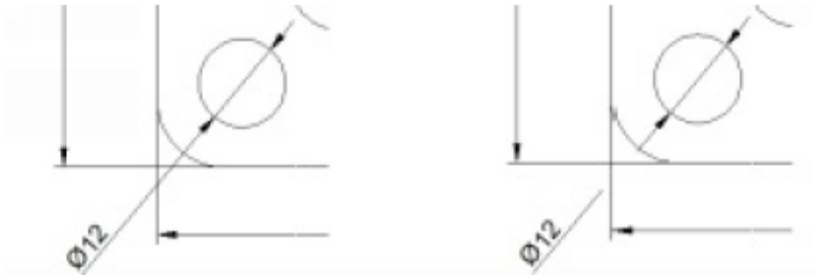
Aligned 	DAL	This tool creates a linear dimension parallel to the object. 
Arc Length 	DAR	It dimensions the total or partial length of an arc.  
Continue	DCO	It creates a linear dimension from the second extension line of the previous dimension.
		

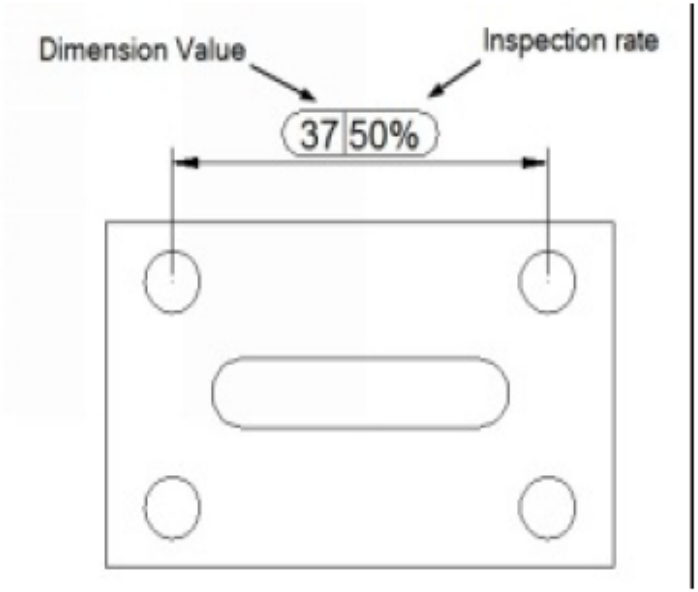
Baseline 	DBA	It creates dimensions by using the previously created dimension, as shown below. 
Angular 	DAN	It creates an angular dimension. 

Diameter 	DIA	It adds a diameter dimension to a circle or an arc. 
Radius 	DRA	It adds a radial dimension to an arc or circle. 
Jogged 	DJO	It creates jogged dimensions. A jogged dimension is created when it is not possible to show the center of an arc or circle. 

Dimension, Dimjogline	DJL	It creates a jogged linear dimension. 
Center Mark	DCE	It adds a center mark to a circle or an arc. The type of center mark will depend on the value of the DIMCEN variable. For a positive value, center marks are created and for a negative value, center lines are created.

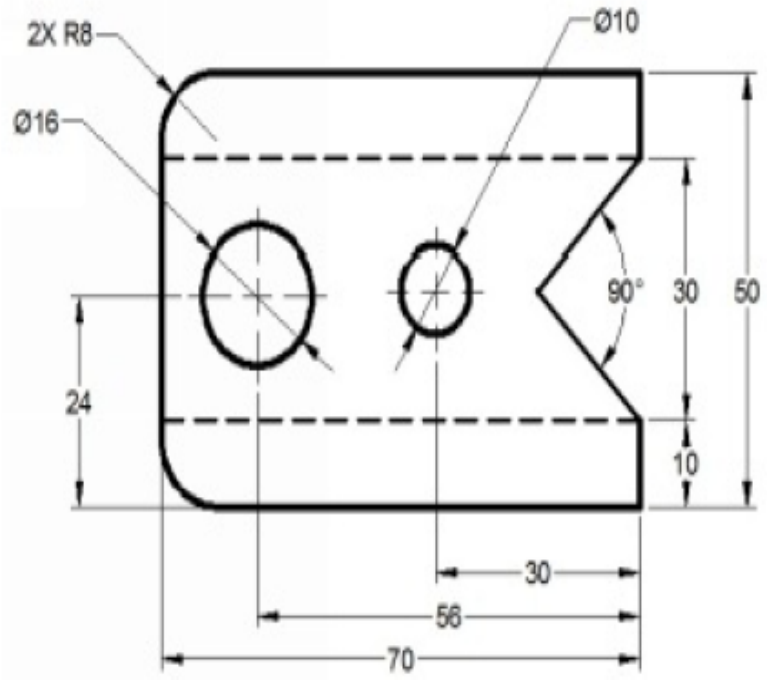
Ordinate 	DOR	It creates ordinate dimensions based on the current position of the User Coordinate System (UCS). 
Quick Dimension 	QDIM	It dimensions one or more objects at the same time. 

Adjust Space 	DIMSPACE	It is used to adjust the space between dimensions. 
Break 	DIMBREAK	It adds breaks to a dimension, extension, and leader lines. 

Inspect 	DIMINSPECT	It creates an inspection dimension. The inspection dimension describes how frequently the dimension should be checked during inspection process to ensure the quality of the component. 

Example:

In this example, you will create the drawing as shown in figure and add dimensions to it.



Create four new layers with the following

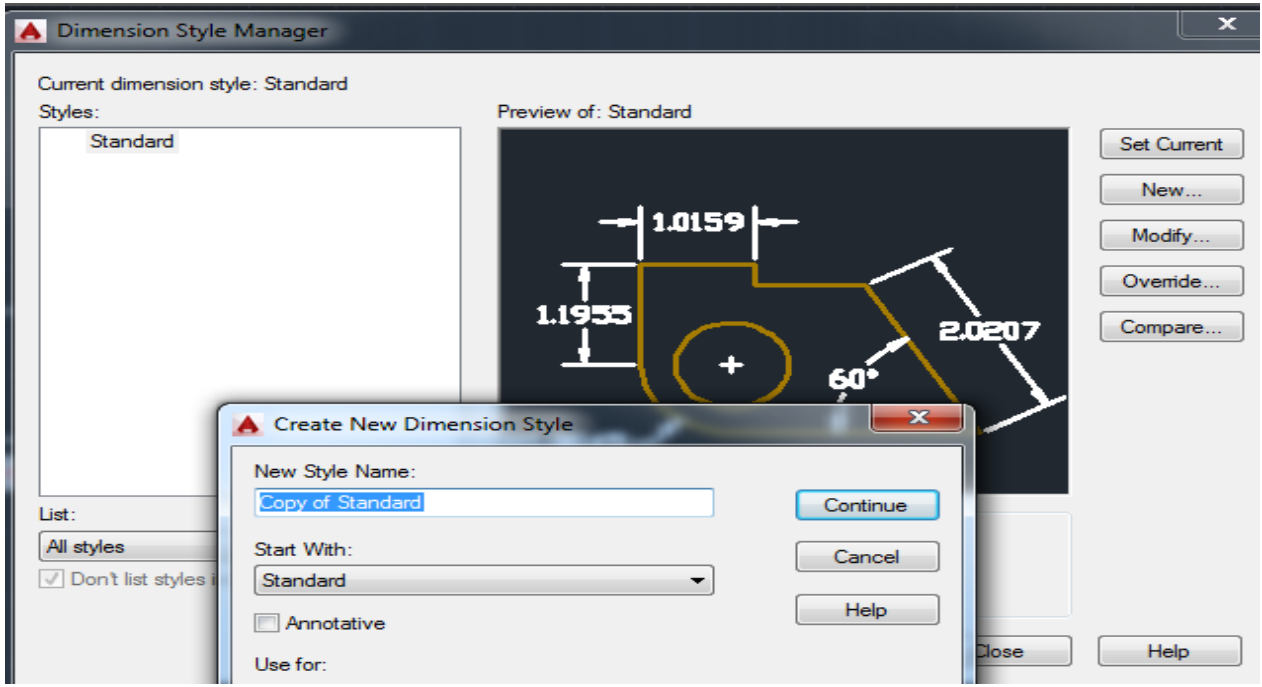
Layer	Lineweight	Linetype
Construction	0.00 mm	Continuous
Object	0.50 mm	Continuous
Hidden	0.30 mm	HIDDEN2
Dimensions	Default	Continuous

Creating a Dimension Style

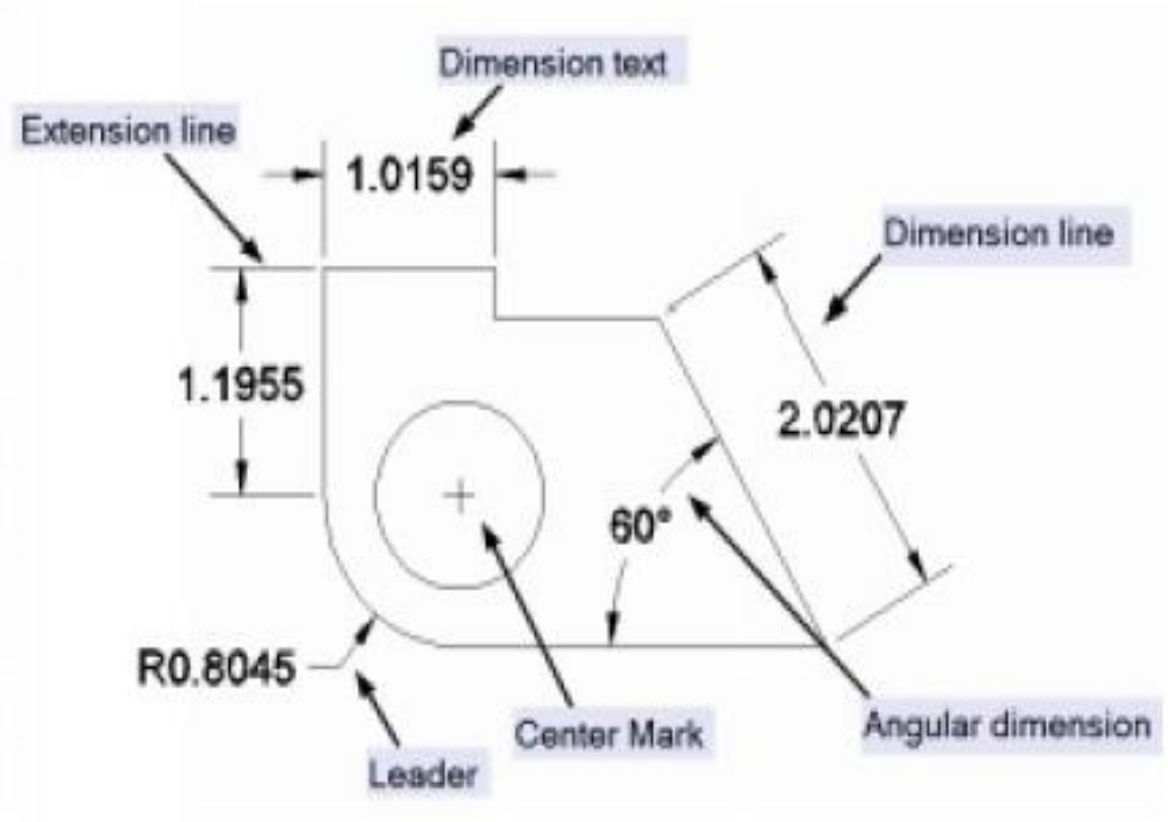
The appearance of the dimensions depends on the dimension style that you use.

You can create a new dimension style using the **Dimension Style Manager** dialog. In this dialog, you can specify various settings related to appearance and behaviour of dimensions.

The **Dimension Style Manager** dialog



The basic nomenclature of dimensions is given below.

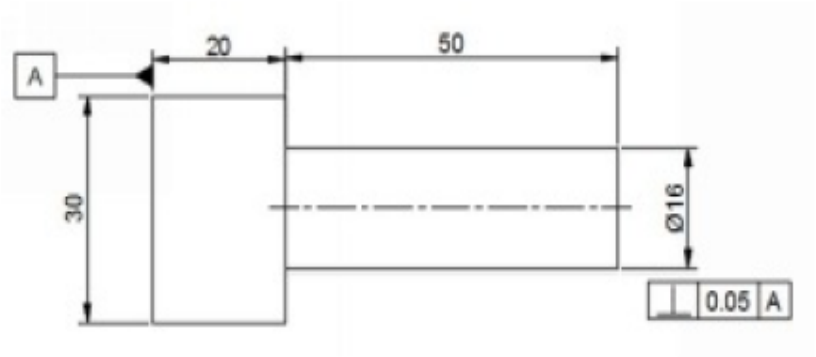


Geometric Dimensioning and Tolerancing

Earlier, you have learned how to apply tolerances to the size (dimensions) of a component.However, the dimensional tolerances are not sufficient for manufacturing a component.You must give tolerance values to its shape, orientation and position as well.

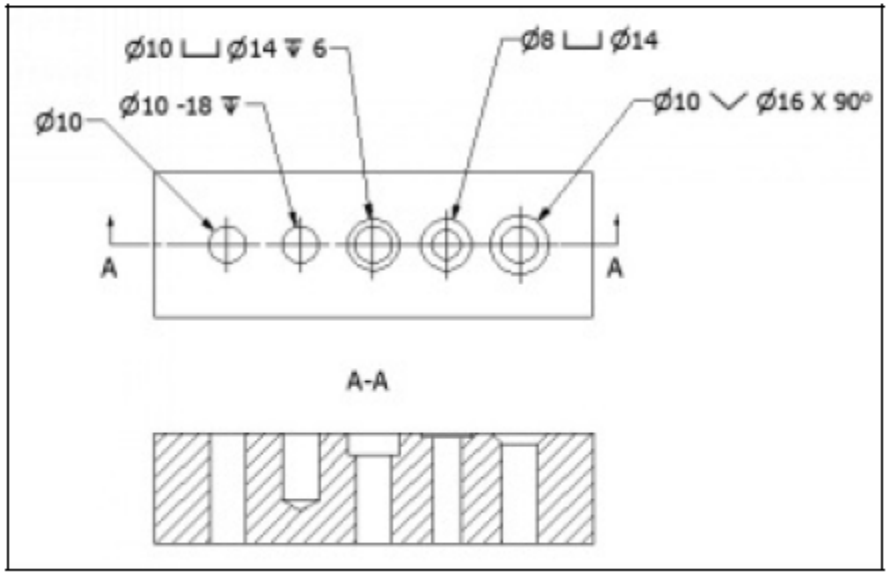
Example 1:

In this example, you will apply geometric tolerances to the drawing shown below.



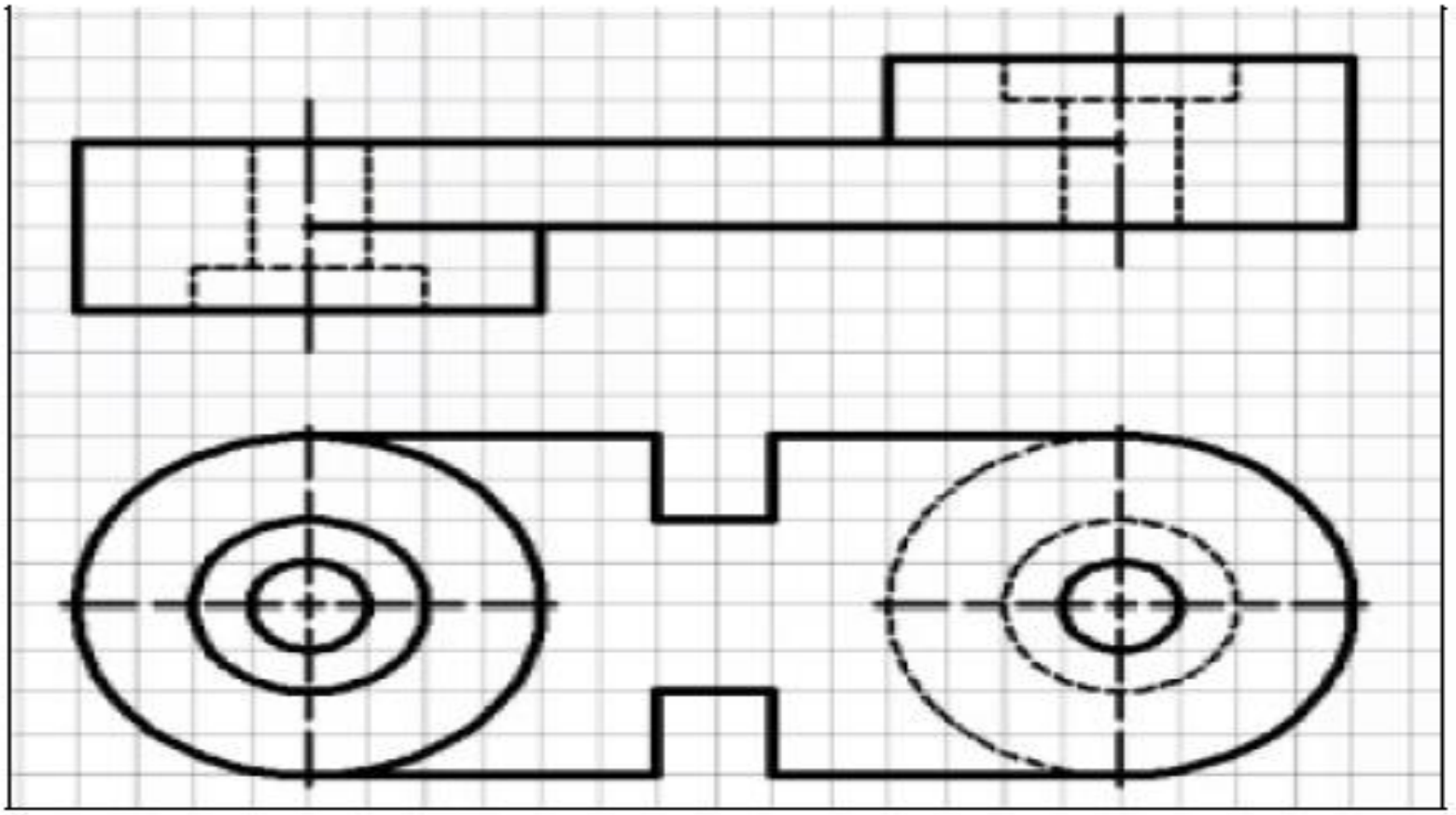
Exercise 1

Create the drawing shown below and create hole callouts for different types of holes. Assume missing dimensions.



Exercise 2

Create the following drawings and apply dimensions and annotations. The Grid Spacing X= 10 and Grid Spacing Y=10.



Exercise 3

Create the drawing shown below. The Grid spacing is 10 mm. After creating the drawing, apply dimensional tolerances to it. The tolerance specifications are given below.

Method: Limits

Precision: 0.00

Upper Value: 0.05

Lower Value: 0.05