

Dimensioning

Objectives:

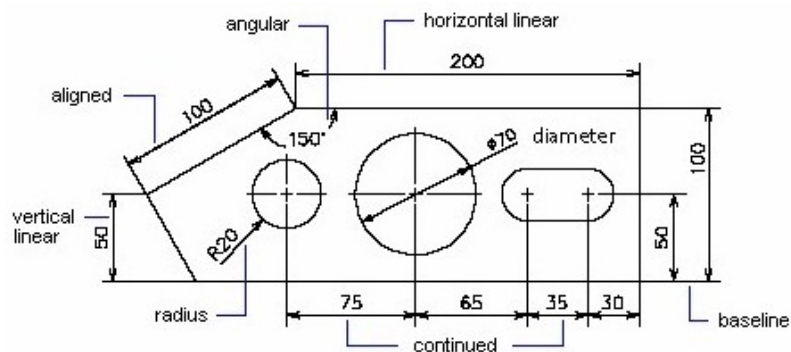
- To define dimensioning.
- To recognise the different types of dimensions.
- To define and create a dimension style.
- To recognise the dimension toolbar and the dimensioning commands.
- To create dimensions and add dimensions to a drawing.
- To edit a dimension.
- To recognise some special characters in dimensioning.

Subject Matters:

Dimensioning

Dimensioning is a process of adding dimensions and tolerances to a drawing. Dimensions show the geometric measurements of objects, the distances or angles between objects, or the co-ordinates of a feature.

Types of dimensions



1. Linear dimensions

- (i) **Horizontal linear dimension** – the dimension line is horizontal
- (ii) **Vertical linear dimension** – the dimension line is vertical
- (iii) **Aligned dimension** – the dimension line is parallel to the extension line origins.
- (iv) **Baseline dimension** – multiple dimensions measured from the same baseline.
- (v) **Continued dimensions** – multiple dimensions placed end to end.
- (vi) **Ordinate dimensions** – dimensions measured the perpendicular distance from an origin point (datum) to a dimensioned feature.

2. Radial dimensions

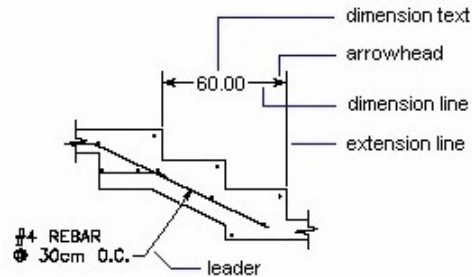
Dimensions that measure the radii and diameters of arcs and circles.

3. Angular dimensions

Dimensions that measure the angle between two lines or three points.

Parts of a dimension

Dimensions have several distinct elements: dimension text, dimension lines, arrowheads, and extension lines.



1. **Dimension text** is a text string that usually indicates the measurement value.
2. A **dimension line** indicates the direction and extent of a dimension. For angular dimensions, the dimension line is an arc.
3. **Arrowheads**, also called symbols of termination, are displayed at each end of the dimension line.
4. **Extension lines**, also called projection lines or witness lines, extend from the feature to the dimension line.

The Dimension toolbar

The **Dimension** toolbar can be called for display on screen for use in the same way as other toolbars.



This toolbar shows most of the dimensioning tools including:

- | | |
|--------------------|---------------------|
| Linear Dimension | Aligned Dimension |
| Ordinate Dimension | Radius Dimension |
| Diameter Dimension | Angular Dimension |
| Quick Dimension | Baseline Dimension |
| Continue Dimension | Quick Leader |
| Tolerance | Center Mark |
| Dimension Edit | Dimension Text Edit |
| Dimension Update | Dimension Style |

Dimension style

A dimension style is a group of settings (dimension variables) that control the appearance of the dimensions. These include:

1. Extension lines, dimension lines, arrowheads, centre marks or lines, and the offsets between them.
2. The positioning of the parts of the dimension in relation to one another and the orientation of the dimension text.
3. The content and appearance of the dimension text.

ISO-25 dimension style is the default style. By modifying the dimension style, all existing dimensions created previously with that dimension style can be updated.

Creating a dimension style

A new dimension style is created by changing the variables of the current style (i.e. **ISO-25**). The variables are **Lines and Arrows, Text, Fit, Primary Units, Alternate Units, and Tolerances**.

Calling the **Dimension Style Manger** to create and edit dimension style:



Format menu: Dimension Style

Dimension menu: Style

Command line: `dimstyle`

In the **Dimension Style Manager**,

1. The dimension styles of the active drawing are listed in the **Styles** list.
2. A preview of the dimensions for the selected style is shown in the **Preview** window.
3. A description also appears to show how the selected dimension style differs from the current style settings.
4. A list of buttons (**Set Current, New, Modify, Override, Compare**) is used to create and manage dimension styles.

Procedure:

- (i) Enter **dimstyle** on the command window to call the **Dimension Style Manager** dialogue box on screen.
- (ii) Left-click the **New** button.
- (iii) To start with the current style for all dimensions, enter a **new style name** and click **Continue**.
- (iv) Change the dimension variables in the **Lines and Arrows** tab: *Dimensions Lines, Extension Lines, Arrowheads, Center Marks for Circles*.
- (v) Change the dimension variables in the **Text** tab: *Text Appearance, Text Placement, Text Alignment*.
- (vi) Change the dimension variables in the **Fit** tab: *Fit Options, Text Placement, Scale for Dimension Features, Fine Tuning*.
- (vii) Change the dimension variables in the **Primary Units** tab: *Linear Dimensions, Angular Dimensions*.
- (viii) Change the dimension variables in the **Alternate Units**: *Display alternate units* check box, *Alternate Units, Zero Suppression, Placement*.
- (ix) Change the dimension variables in the **Tolerances** tab: *Tolerance Format, Alternate Unit Tolerance*.
- (x) Click **OK** after changing the dimension variables and return to the **Dimension Style Manager**.
- (xi) Click **Set Current** button to make the new style the current style.
- (xii) Click **Close** to close the **Dimension Style Manager**.

Tips: It is not a good CAD practice to change the default dimension style (ISO-25).

Managing dimension styles

The **Dimension Style Manger** provides a list of buttons (**S**et **C**urrent, **N**ew, **M**odify, **O**uerride, **C**ompare) to manage dimension styles.

1. **Set Current:** Sets the style selected under **Styles** to current.
2. **New:** Displays the **Create New Dimension Style** dialogue box, in which new dimension styles can be defined.
3. **Modify:** Displays the **Modify Dimension Styles** dialog box, in which dimension styles can be modified. Dialog box options are identical to those in the **New Dimension Style** dialog box. **When a dimension style is modified, all dimensions in the drawing that use this style will be updated automatically to the new definition.**
4. **Override:** Displays the **Override Current Style** dialog box, in which temporary overrides to dimension styles can be set. Dialog box options are identical to those in the **New Dimension Style** dialog box. **An override can only be created for the current style. Existing dimensions in the drawing could be updated manually to the new override definition. Any overrides will be discarded when another style is selected and picked as current.**
5. **Compare:** Displays the **Compare Dimension Style** dialogue box, which compares the properties of two dimension styles or lists all the properties of one style.

Creating dimensions

Dimensions can be created in two ways:

Method 1

By specifying the extension line origins and the location of the dimension line.

Procedure:

- (i) Select a dimensioning icon (e.g. Linear Dimension) from the **Dimension** toolbar.
- (ii) Locate the first extension line origin (assist with a suitable object snap tool).
- (iii) Locate the second extension line origin (assist with a suitable object snap tool).
- (iv) Drag the generated dimension to a suitable location and left-click to end.

Method 2:

By selecting an object to dimension and specifying the location of the dimension line.

Procedure:

- (i) Select a dimensioning icon (e.g. Linear Dimension) from the **Dimension** toolbar.
- (ii) Right-click and the cross hair cursor changes to a pick box.
- (iii) Select the object to dimension.
- (iv) Drag the generated dimension to a suitable location and left-click to end.

Dimension commands

Linear Dimension



Dimension toolbar: 

Dimension menu: Linear

Command line: **dimlinear**

Function: It creates a linear dimension, e.g. horizontal dimension, vertical dimension.

Command sequence:

Command: **DIMLINEAR**

Specify first extension line origin or <select object>: **(pick a point)**

Specify second extension line origin: **(pick a point)**

Specify dimension line location or

[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: **(drag the dimension to the desired location, left-click to end)**

Dimension text = (displayed dimension value)

Tips: Options [Mtext/Text/Angle/Horizontal/Vertical/Rotated] are available to specify the dimension text and its alignment. Enter the capitalised letter to select the required one.

Aligned Dimension



Dimension toolbar: 

Dimension menu: Aligned

Command line: **dimaligned**

Function: It creates an aligned dimension.

Command sequence:

Command: **DIMALIGNED**

Specify first extension line origin or <select object>: **↵**

Select object to dimension: **(drag the pick box to select an object)**

Specify dimension line location or

[Mtext/Text/Angle]: **(drag the dimension to the desired location, left-click to end)**

Dimension text = (displayed dimension value)

Tips: Dimension can also be created by specifying the first and second extension line origins.

Ordinate Dimension



Dimension toolbar: 

Dimension menu: Ordinate

Command line: **dimordinate**

Function: It creates ordinate point dimension and displays the X or Y ordinate of a feature along with a simple leader line, i.e. AutoCAD uses the current UCS to determine the measured X or Y ordinate and draws the leader line in a direction orthogonal to the axes.

Command sequence:

Command: **DIMORDINATE**

Specify feature location: **(pick a point)**

Specify leader endpoint or [Xdatum/Ydatum/Mtext/Text/Angle]: **(pick a leader endpoint)**

Dimension text = (displayed dimension value)

Tips: Also known as datum dimensions, displaying the absolute co-ordinate values.

Radius Dimension



Dimension toolbar: Radius

Dimension menu: Radius

Command line: **dimradius**

Function: It creates radial dimensions for circles and arc.

Command sequence:

Command: **DIMRADIUS**

Select arc or circle: **(drag the pick box to select an object)**

Dimension text = (displayed dimension value)

Specify dimension line location or [Mtext/Text/Angle]: **(drag the dimension to the desired location, left-click to end)**

Diameter Dimension



Dimension toolbar: Diameter

Dimension menu: Diameter

Command line: **dimdiameter**

Function: It creates diameter dimensions for circles and arcs.

Command sequence:

Command: **DIMDIAMETER**

Select arc or circle: **(drag the pick box to select an object)**

Dimension text = (displayed dimension value)

Specify dimension line location or [Mtext/Text/Angle]: **(drag the dimension to the desired location, left-click to end)**

Angular Dimension



Dimension toolbar: Angular

Dimension menu: Angular

Command line: **dimangular**

Function: It creates an angular dimension.

Command sequence:

For line:

Command: **DIMANGULAR**

Select arc, circle, line, or <specify vertex>: **(drag the pick box to select a line object)**

Select second line: **(drag the pick box to select another line object)**

Specify dimension arc line location or [Mtext/Text/Angle]: **(drag the dimension to the desired location, left-click to end)**

Dimension text = (displayed dimension value)

For arc:

Select arc, circle, line, or <specify vertex>: **(drag the pick box to select an arc)**

Specify dimension arc line location or [Mtext/Text/Angle]: **(drag the dimension to the desired location, left-click to end)**

Dimension text = (displayed dimension value)

For circle:

Select arc, circle, line, or <specify vertex>: **(drag the pick box to select a point on the circle)**

Specify second angle endpoint: **(drag the pick box to specify a second point on the circle)**

Specify dimension arc line location or [Mtext/Text/Angle]: **(drag the dimension to the desired location, left-click to end)**

Dimension text = (displayed dimension value)

Quick Dimension



Dimension toolbar: 
Dimension menu: Quick Dimension
Command line: `qdim`

Function: It quickly creates a series of dimension. The command is particularly useful for creating a series of baseline or continued dimensions, or for dimensioning a series of circles and arcs.

Command sequence:

Command: **QDIM**

Select geometry to dimension: **(drag the pick box to select object or use window to select a group of objects)**

Select geometry to dimension: **(right-click to end selection)**

Specify dimension line position, or

[Continuous/Staggered/Baseline/Ordinate/Radius/Diameter/datumPoint/Edit/seTtings] <Baseline>: **(drag the dimension to the desired location, left-click to end or specify an option)**

Tips: The option inside the angle bracket < > depends on the selected geometry.

Baseline Dimension



Dimension toolbar: 
Dimension menu: Baseline
Command line: `dimbaseline`

Function: It creates a series of continuous dimensions, such as linear, angular or ordinate dimension, that measured from the baseline of the previous or selected dimension. Before creating a **baseline dimension**, specify a linear, angular or ordinate dimension as a base dimension.

Command sequence:

Command: **DIMBASELINE**

Select base dimension: **(drag the pick box to select a base dimension)**
Specify a second extension line origin or [Undo/Select] <Select>: **(drag the pick box to specify a point)**

Dimension text = (displayed dimension value)

Specify a second extension line origin or [Undo/Select] <Select>: **(drag the pick box to specify a point)**

Dimension text = (displayed dimension value)

Specify a second extension line origin or [Undo/Select] <Select>: **(drag the pick box to specify a point)**

Dimension text = (displayed dimension value)

Specify a second extension line origin or [Undo/Select] <Select>: **(right-click to invoke a menu and select Enter)**

Select base dimension: **(right-click to quit)**

Tips: If a dimension was created in the current session, AutoCAD skips the first prompt (Select base dimension:) and uses the dimension object that was last created in the session as the baseline.

Continue Dimension



Dimension menu: Continue

Command line: **dimcontinue**

Function: It creates a series of dimensions, such as linear, angular or ordinate dimension, that continues from the second extension line of the previous or selected dimension. Before creating a continue dimension, specify a linear, angular or ordinate dimension as a base dimension.

Command sequence:

Command: **DIMCONTINUE**

Select continued dimension: **(drag the pick box to select a continued dimension)**

Specify a second extension line origin or [Undo/Select] <Select>: **(drag the pick box to specify a point)**

Dimension text = (displayed dimension value)

Specify a second extension line origin or [Undo/Select] <Select>: **(drag the pick box to specify a point)**

Dimension text = (displayed dimension value)

Specify a second extension line origin or [Undo/Select] <Select>: **(right-click to invoke a menu and select Enter)**

Select continued dimension: **(right-click to quit)**

Tips: If a dimension was created in the current session, AutoCAD skips the first prompt (Select continued dimension:) and uses the dimension object that was last created in the session as the base for the continue dimension.

Quick Leader



Dimension menu: Leader

Command line: **qleader**

Function: It connects some annotation to a feature in a drawing. If the leader is at an angle greater than 15° from the horizontal, a hook line is used to connect the annotation to the leader.

Command sequence:

Command: **QLEADER**

Specify first leader point, or [Settings] <Settings>: **(specify the leader arrowhead point)**

Specify next point: **(specify the leader end point)**

Specify next point: **(right-click to skip)**

Specify text width <0>: **(right-click or enter a value)**

Enter first line of annotation text <Mtext>: **(enter the required text)**

Enter next line of annotation text: **(enter the required text or right-click to quit)**

Tips: The leader command is customised by the **Leader Settings** dialogue box (choose the default <Settings> when prompted for the first leader point). The number of leader points and the annotation type can be set to suit the drawing needs.

Tolerance



Dimension menu: Tolerance

Command line: tolerance

Function: It creates geometric tolerances. AutoCAD adds geometric tolerances to a drawing in feature control frames. The frames are divided into compartments that contain the geometric characteristic symbols followed by one or more tolerance values. Where applicable, the tolerance is preceded by the diameter symbol and followed by datums and symbols for their material conditions.

Command sequence:

Command: **TOLERANCE**

[The Geometric Tolerance dialogue box is on screen. Fill the dialogue box as required, e.g.

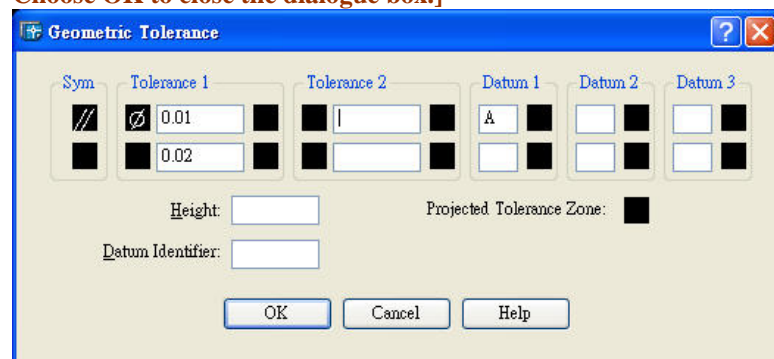
Select a geometric tolerance symbol (//) under Sym.

Under Tolerance 1, click the first black box to insert a diameter symbol (not required in this example).

In the Text box, enter the first tolerance value (0.1).

Under Datum 1, enter the datum reference letter (A).

Choose OK to close the dialogue box.]



Enter tolerance location: (place the frame to the desired position and left-click to quit)

Tips: To create a geometric tolerance with a leader, use the **QLEADER** command and set the **Annotation type** to **Tolerance** in the **Leader Settings** dialogue box.

To create linear tolerance, enter the values in the two text boxes under **Tolerance 1**.

Center Mark



Dimension menu: Center Mark

Command line: dimcenter

Function: Depending on the dimension style setting, it creates the centre mark or the centre lines of circles and arcs.

Command sequence:

Command: **DIMCENTER**

Select arc or circle: (drag the pick box to select an arc or circle)

Editing Dimensions

Dimensions are drawing objects that can be edited with modify tools. AutoCAD also provides tools to change the annotation, and the appearance and justification of the dimension text.

Dimension Edit



Function: It affects dimension text and extension lines on one or more dimension objects. The Home, New, and Rotate options affect dimension text. The Oblique option controls the angle of extension lines.

Home It moves the dimension text back to its default position.

New It changes the dimension text using the **Multiline Text Editor**.

Rotate It rotates the dimension text.

Oblique It adjusts the oblique angle of the extension lines for linear dimensions.

Command sequence:

Command: **DIMEDIT**

Enter type of dimension editing [Home/New/Rotate/Oblique] <Home>:
(enter an option, e.g. R)

Specify angle for dimension text: (enter a value)

Select objects: (drag the pick box to select a dimension)

Select objects: 1 found

Select objects: (right-click to quit)

Dimension Text Edit



Function: It moves and rotates dimension text and updates the location of the dimension text as it is dragged.

Left It left-justifies the dimension text along the dimension line.

Right It right-justifies the dimension text along the dimension line.

Center It centers the dimension text on the dimension line.

Home It moves the dimension back to its default position.

Angle It changes the angle of the dimension text.

Command sequence:

Command: **DIMTEDIT**

Select dimension: (drag the pick box to select a dimension)

Specify new location for dimension text or [Left/Right/Center/Home /Angle]: (move the dimension text to a new position and left-click or enter an option)

Dimension associativity It defines the relationship between geometric objects and the dimensions that give their distance and angles. AutoCAD provides three types of associativity between geometric objects and dimensions:

Associative dimensions	Dimensions automatically adjust their locations, orientations, and measurement values when the geometric objects associated with them are modified.
Nonassociative dimensions	Dimensions do not change when the geometric objects they measure are modified.
Exploded dimensions	Dimensions that contain a collection of separate objects rather than a single dimension object.

By default, associative dimension is **on**. Although non-associative dimensions can be created, it is recommended to maintaining all features in a dimension as a single object.

Special characters

Some text or figures in dimensioning or measurement require symbols to be included either before or after the text. Such symbols are entered using a pair of percent signs (%%). They can also be accessed from the **Symbol** menu in the **Multiline Text Editor**. Examples of these special characters are listed below:

%%o Toggles overscore mode on/off

%%u Toggles underscore mode on/off

%%d Draws degree symbol (°)

%%p Draws plus/minus tolerance symbol (\pm)

%%c Draws diameter dimensioning symbol (ϕ)

Hands on practice

Referring to your Engineering Drawing textbook, “Engineering Drawing with CAD Applications by O. Ostrowsky (Edward Arnold)”, draw the figures in Question 26-28 shown on Page 85.

Create a new dimension style, name it as draw_style and make it current. Use this style and add dimensions to the drawings in the exact pattern as shown.