

DIMENSIONING OBJECTIVES

At the end of the unit you will be able to:

- Create linear dimensions with DIMLINEAR, DIMCONTINUE, DIMBASELINE, DIMALIGNED.
- Create radial dimensions with DIMDIAMETER and DIMRADIUS.
- Create Angular dimensions with DIMANGULAR.
- Create Ordinate dimensions with DIMORDINATE.
- Create and edit the dimensions.

INTRODUCTION

As you know, drawings created with CAD systems should be constructed with the same dimensions and units as the real-world objects they represent. The important of this practice is evident when you begin applying dimensions to the drawing geometry in AutoCAD. The features of the object that you specify for dimensioning are automatically measured, and those values are used for the dimensioning text. If the geometry has been drawn accurately, the dimensions will be created correctly. The correct use of AutoCAD dimension tools is the key to producing clear and concise measured drawings.

AutoCAD provides a whole range of dimensioning tools which can be used to quickly dimension any drawing without the need for measurement. Dimensioning in AutoCAD is automatic; lines, arrows and text are all taken care of by the dimension commands. AutoCAD dimensions are special blocks which can easily be edited or erased as necessary.

AutoCAD provides lots of control over the way dimensions look. Using a system similar to text styles, dimension styles allow you to design dimensions so that they look just the way you want them to.

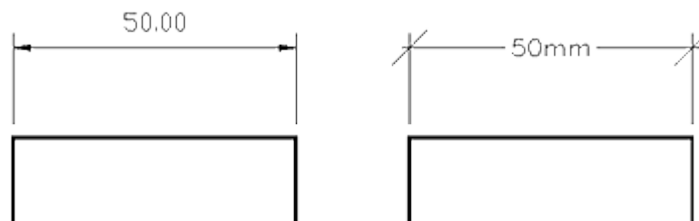


Figure 1

For example, *Figure 1* above shows two different dimension styles. The one on the left is the default style known as STANDARD. If you do not create a style of your own or modify the standard style, all dimensions will look like this. The dimension line has arrow heads and the dimension text is positioned above the line and is drawn using the current text style. The dimension on the right has been drawn using a new style. The

arrows have been changed to *obliques*, the vertical alignment of the text has been centered and the current text style has been changed.

AutoCAD divides dimensions into four main categories: **Linear**, **Radial**, **Ordinate** and **Angular**. When you create dimensions, AutoCAD automatically creates a new layer called "Defpoints". This is a special layer which cannot be deleted or renamed. AutoCAD uses this layer to store dimension information and you can effectively ignore it.

When working with dimensions it is very important that line origins are picked accurately so that the resulting measurement and text are correct. *Always* use an *Osnap* to pick dimension line origins.

SELECTING DIMENSION COMMANDS

Selecting and working with the dimension commands in AutoCAD R14 is much easier than in previous versions. All commands can be accessed from the keyboard and now most commands are also available from the Dimension *pull-down menu* and the *Dimension toolbar*. The *Dimension toolbar* is particularly useful because it places all the dimension commands a single mouse click away. Since the *Dimension toolbar* is not displayed by default you will need to enable it from the *Toolbars dialogue box*.

To display the Toolbar dialogue box, select

Pull-down menu : **View/Toolbars**
Command : **Toolbar**

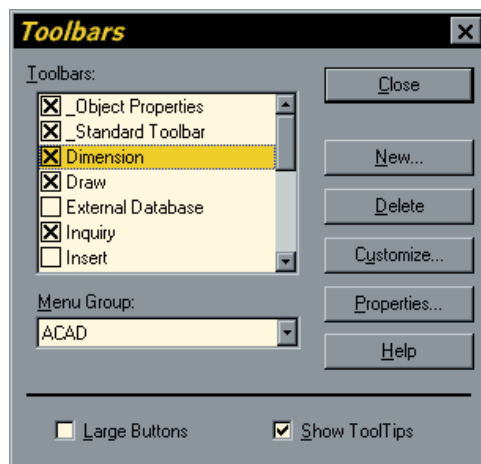


Figure 2: Toolbars Box

To display the *Dimension toolbar*, click in the checkbox against "Dimension" in the toolbar list.

The main components of a dimension are

1. Dimension line
2. Extension lines
3. Dimension text
(usually a numeric value)
4. Arrowheads or tick marks

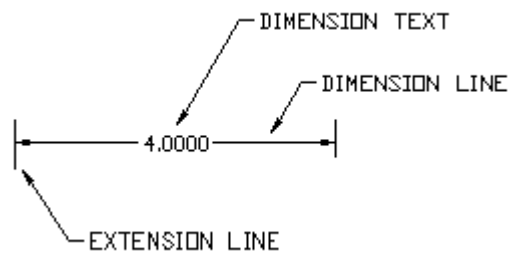


Figure 3


Dimensioning an object correctly requires a number of different dimensioning methods based on the object you are dimensioning. This includes:-

1. Linear methods for horizontal and vertical.
2. Aligned.
3. Angular method for angles.
4. Diameter and Radius methods for circles and arcs.
5. Ordinate method for datum-based.

THE LINEAR DIMENSION COMMAND

As the name suggests the Linear dimension commands are used to dimension along straight lines. There are five linear dimension commands, namely: **DIMLINEAR**, **DIMCONTINUE**, **DIMBASELINE**, **DIMALIGNED** and **DIMROTATED**. The DIMLINEAR command is probably the most common dimension command you will use.

The Linear Dimension

Toolbar : 

Pull-down : **Dimension/Linear**

Keyboard : **DIMLINEAR**

You can use this command to generate horizontal and vertical dimensions. Creating a linear dimension is easy. All you have to do is start the command, specify the two points between which you want the dimension to be drawn and pick a point to fix the position of the dimension line. Consider the *Figure 4* whilst working through the following examples.

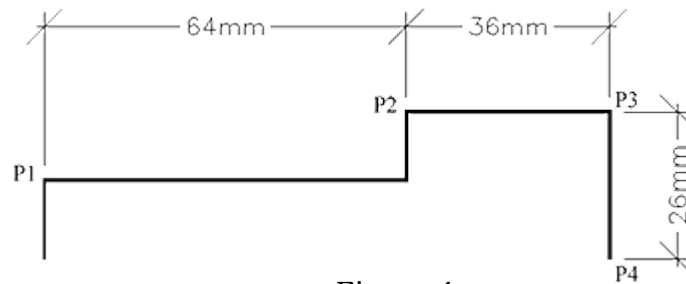


Figure 4

Command Sequence

Command: **DIMLINEAR**

First extension line origin or press ENTER to select: (*pick P1*)

Second extension line origin: (*pick P2*)

Dimension line location (Mtext/Text/Angle/Horizontal/Vertical/Rotated): (*pick a point to position the dimension line, you will see the dimension rubber banding*)

You may have noticed that the first prompt asks you to pick the first extension line origin or to press the ENTER key. Pressing the Enter/Return key results in the following prompt:

Select object to dimension:

AutoCAD allows you to dimension an object simply by picking it. Try this out. Draw a line or a circle and use this option rather than the two point option to see what happens.

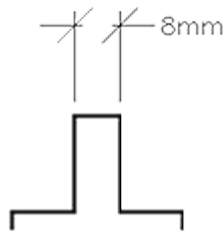


Figure 5

Dimensions will automatically adjust themselves to accommodate most situations. For example, the *Figure 5* shows what happens to a dimension if the gap between the two extension lines is too small for the dimension text.

The Continue Dimension

Toolbar



Pull-down

Dimension/Continue

Keyboard

DIMCONTINUE

You can use the Continue command to add a string of dimensions. In the *Figure 4* the "36mm" dimension has been continued from the "64mm" dimension.

Command Sequence

Command: **DIMCONTINUE**

Specify a second extension line origin or (Undo/<Select>): (pick **P3**)

Specify a second extension line origin or (Undo/<Select>): (pick another or \leftarrow to end)

There is no prompt for the first line origin, AutoCAD automatically selects the second line origin of the previous dimension to be the first of the new dimension.

There is also no prompt for the dimension line position, AutoCAD automatically matches up with the previous dimension.

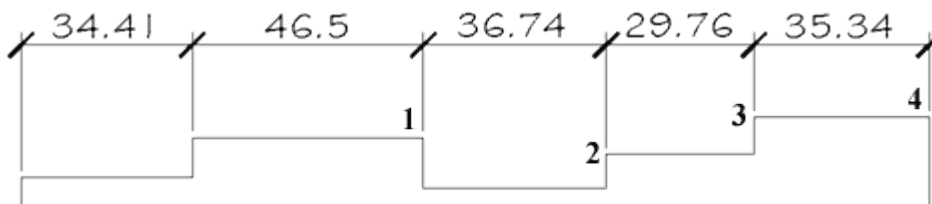


Figure 6

Using the Continue command you can very quickly generate a string of dimensions which align perfectly. In the example above (*Figure 6*), the "34.41" dimension was drawn with the DIMLINEAR command; all the other dimensions were drawn using the DIMCONTINUE command and simply picking the four points, one after the other. You can only continue a dimension in a single direction. To generate the "26mm" dimension in *Figure 6*, you will need to use the DIMLINEAR command and pick *P3* and *P4* or *Enter* at the first prompt and pick the line.

The Baseline Dimension

Toolbar 

Pull-down **Dimension/Baseline**

Keyboard **DIMBASELINE**

You can use this command to generate a series of dimensions from a single base point. You must already have created the first dimension in the sequence using a command such as DIMLINEAR. The DIMBASELINE command then creates further dimensions in a similar way to the DIMCONTINUE command. All the user has to do is pick points.

Command Sequence

Command: **DIMBASELINE**

Specify a second extension line origin or (Undo/<Select>): *(pick next point)*

Specify a second extension line origin or (Undo/<Select>): *(pick another or Enter to end)*

Select base dimension: *(Enter again to end)*

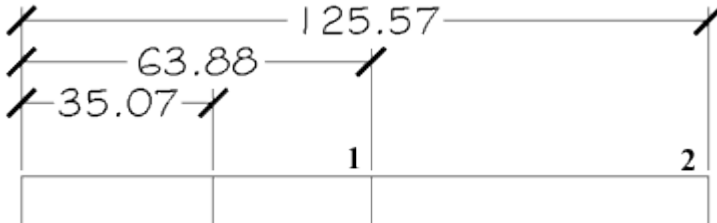


Figure 7

In the Figure 7, the "35.07" dimension was created using the DIMLINEAR command. The others were created using DIMBASELINE and picking points 1 and 2.

The Aligned Dimension

Toolbar



Pull-down **Dimension/Aligned**

Keyboard **DIMALIGNED**

You can use this command to generate aligned dimensions. These are dimensions along inclined lines, which cannot be dimensioned with the DIMLINEAR dimension command because that command will only give a measured dimension in either a horizontal or vertical direction. However, as you can see from the command sequence below, this command works in exactly the same way.

Command Sequence

Command: **DIMALIGNED**

First extension line origin or press ENTER to select: *(pick P1 at Figure 8)*

Second extension line origin: *(pick P2 at Figure 8)*

Dimension line location (Mtext/Text/Angle): *(pick a point)*

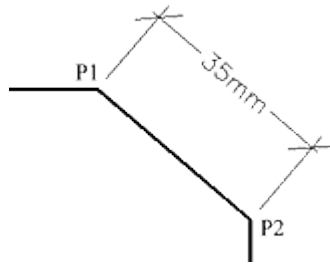


Figure 8

The DIMCONTINUE and DIMBASELINE commands can both be used in conjunction with DIMALIGNED dimensions.

THE RADIAL (CIRCLE / ARC) DIMENSION COMMAND

There are two main radial dimension commands, DIMDIAMETER and DIMRADIUS. Both commands result in a similar looking dimension so AutoCAD automatically inserts a "R" to indicate a radius and the dimension symbol to indicate a dimension. You can get AutoCAD to display the dimension symbol by including "%c" in any text string. For example, in order to draw the 40mm diameter text as it is shown in the illustration *Figure 9*, you would need to type "%c40mm". You can use this special character with any of the text commands.

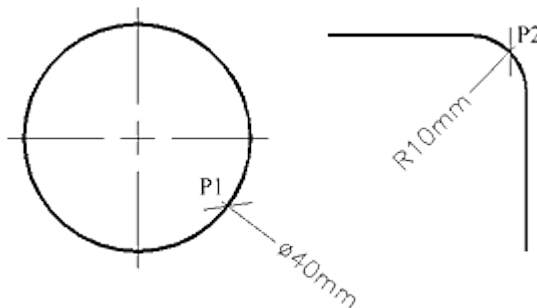


Figure 9

The Diameter and Radius commands are supplemented by the DIMCENTER command, which can be used to add a center mark to any circle or arc. The DIMDIAMETER and DIMRADIUS commands do not automatically draw a center mark.

By convention it is usual to dimension full circles using a diameter and arcs (partial circles) using radius.

The Diameter Dimension

Toolbar 
 Pull-down **Dimension/Diameter**
 Keyboard **DIMDIAMETER**

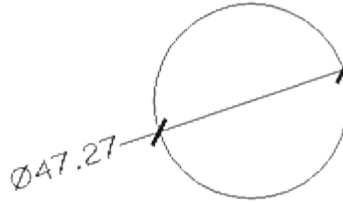


Figure 10

You can use the Diameter command to annotate a circle or an arc with a diameter dimension. To achieve this simply start the command, pick a point on the circumference of the circle, pick a second point to determine the length of the leader and then add the dimension text or *Return* to accept the default.

Command Sequence

Command: **DIMDIAMETER**

Select arc or circle: (*pick the circumference **P1** in Figure 9*)

Dimension line location (Mtext/Text/Angle): (*move the cursor until you are happy with the text position and then pick to complete the sequence*)

The Radius Dimension Command

Toolbar 
 Pull-down **Dimension/Radius**
 Keyboard **DIMRADIUS**

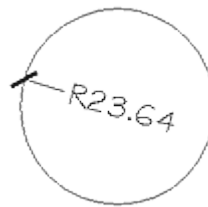


Figure 11

The Radius command is identical to the Diameter command except that the dimension measurement is a radius rather than a dimension and the resulting dimension text is prefixed with a "R" to indicate radius.

Command Sequence

Command: **DIMRADIUS**

Select arc or circle: (*pick the circumference **P2** in Figure 9*)

Dimension line location (Mtext/Text/Angle): (*move the cursor until you are happy with the text position and then pick to complete the sequence*)

Notice that in the *Figure 11* the radius dimension has been positioned inside the circle. Both diameter and radius dimensions can be positioned either inside or outside an arc or circle.

Practice with the Radial and Diameter commands until you understand how they work.

The Center Mark Command

Toolbar 
 Pull-down **Dimension/Center Mark**
 Keyboard **DIMCENTER**

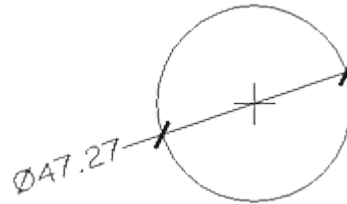


Figure 12

You can use the Center Mark command to annotate a circle or an arc with a cross at the center. The *Figure 12* shows a center mark added to a circle after a diameter has been drawn.

Command Sequence

Command: **DIMCENTER**

Select arc or circle: (*Pick the circumference of a circle or arc*)

A cross is drawn at the center point.

ANGULAR DIMENSIONS

There is only one command in this section and it is used to annotate angular measurements.

The Angular Dimension Command

Toolbar 
 Pull-down **Dimension/Angular**
 Keyboard **DIMANGULAR**

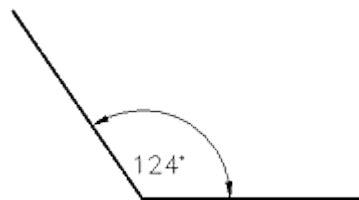


Figure 13

The Angular command is amazingly flexible and can be used to indicate an angle in almost any situation. Just like the other dimension commands, all parts of the process are rubber banded so you can see the results of your actions before you make the final pick.

Command Sequence

Command: **DIMANGULAR**

Select arc, circle, line, or press ENTER: (*pick a line*)

Second line: (*pick another line*)

Dimension arc line location (Mtext/Text/Angle): (*pick point*)

Move the cursor position until you are happy with the result. Notice that you can move the cursor to either side of the lines and the angular dimension will change accordingly.

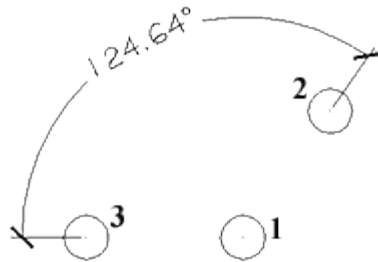


Figure 14

You may have noticed that at the first prompt you are given the option to press *ENTER*. If you use this option you will be prompted to pick the angle vertex and then the two angle endpoints. This is quite useful if the angle you need to dimension is not defined by physical lines on the drawing. The illustration on *Figure 14* shows the result of this option. The centre point of circle 1 was picked as the angle vertex and the centre points of circles 2 and 3 were picked for the two angle endpoints.

The degree character is automatically inserted for you, however, if you ever need to type it, you can do so by typing "%d". This is another of AutoCAD's special characters.

ORDINATE DIMENSIONS

Ordinate dimensions are not really dimensions at all in that they do not indicate a measurement. Rather they annotate known co-ordinate points. The DIMORDINATE command is used to indicate the X and Y ordinate values at any point.

The Ordinate Dimension Command

Toolbar 
 Pull-down **Dimension/Ordinate**
 Keyboard **DIMORDINATE**

The Ordinate command is used to annotate co-ordinate points with X or Y values. This may be useful for setting-out on site plans.

Command Sequence

Command: ***DIMORDINATE***

Select feature: *(pick the point to annotate)*

Leader endpoint (Xdatum/Ydatum/Mtext/Text): *(pick endpoint or use one of the options)*

By default a vertical leader will display the X ordinate and a horizontal one will display the Y ordinate. However, you can use the Xdatum and Ydatum options to override this default. In the Figure 15 , the building corner on the left has been annotated with X and Y ordinates using the default method.

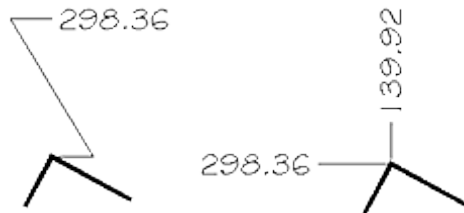


Figure 15

The one on the right has a Y ordinate, which has been forced to display in a vertical position using the Ydatum option. You could also use the Text or Mtext options to clearly describe the point you are annotating.

ANNOTATION WITH LEADERS

Ordinate dimensions are not really dimensions at all in that they do not indicate a measurement. Rather they annotate known co-ordinate points. The DIMORDINATE command is used to indicate the X and Y ordinate values at any point.

The Leader Command

Toolbar 
 Pull-down **Dimension/Leader**
 Keyboard **LEADER**

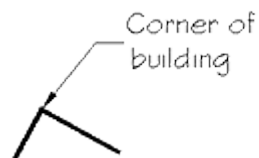


Figure 16

The Leader command can be used to annotate any point on a drawing. The command sequence below was used to draw the leader shown in the *Figure 16*.

Command Sequence

Command: **LEADER**

From point: *(pick the point to annotate)*

To point: *(pick vertex point)*

To point (Format/Annotation/Undo)<Annotation>: *(pick end point)*

To point (Format/Annotation/Undo)<Annotation>: ↵

Annotation (or press ENTER for options): Corner of ↵

MText: building ↵

MText: ↵ *(to end)*


Unlike other dimension commands the leader and annotation text are drawn as separate objects. So, if you need to move or edit the text, you can do so without affecting the leader line.

As you can see by the command line, there are a number of options with this command including "Format" options which include "Spline". Experiment with these options until you understand them.

EDITING DIMENSIONS

The dimension edit commands, DIMEDIT and DIMTEDIT are used primarily to adjust the position of the text part of a dimension. This is usually only necessary if the drawing is quite complex and the dimension would read more clearly if it were in a different position.

The Dimension Text Edit Command

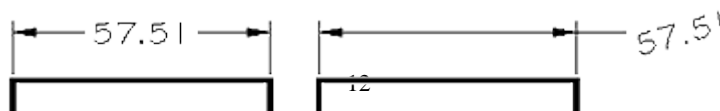
Toolbar 

Pull-down **Dimension/Align Text/** (Choose your options)

Keyboard **DIMTEDIT**

The Dimension Text Edit command is used to modify the text position of any single dimension. The command can be used to position the text dynamically (this is the default) or one of the options can be used for a specific type of movement. For example, the dimension shown on Figure 20 has been modified by dynamically moving the position of the text and then the text has been rotated using the Angle option.

Figure 20



Command Sequence

Command: **DIMTEDIT**

Select dimension: *(pick the dimension you want to edit)*

Enter text location (Left/Right/Home/Angle): *(pick a new position or use an option)*

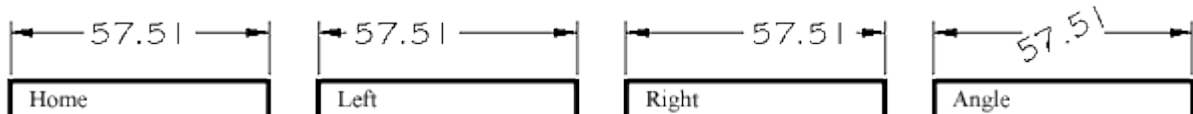


Figure 21

The results of the four available options are shown in the *Figure 21*.


The **Left** option moves the text to a left justified position within the dimension.

The **Right** option moves the text to a right justified position within the dimension.

The **Home** option returns the text to the home position after it has been modified.

The **Angle** option enables the text to be rotated about its center.

The Dimension Edit Command

Toolbar 

Pull-down **Dimension/Oblique** (other options are duplicated in DIMTEDIT so don't appear)

Keyboard **DIMEDIT**

The Dimension Edit command can be used to modify and change the text of any number of dimensions. The command could, for example, be used to add a standard prefix or suffix to a number of dimensions.

Command Sequence

Command: **DIMEDIT**

Dimension Edit (Home/New/Rotate/Oblique) <Home>: *(choose an option)*

Select objects: *(pick one or more dimensions)*

Select objects: *(pick more or ↵ end)*

The command sequence will vary depending upon which option has been chosen but the results of the various options are illustrated in Figure 22

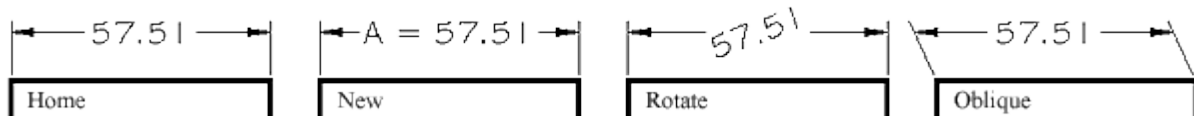


Figure 22

The **Home** option returns dimensions to their home position.

The **New** option displays the Multiline Text Editor. The changes you make to the text will be applied to all selected dimensions so it is important not to delete the "<>" marker from the text string. Deleting this marker will remove the values from all selected dimensions.

The **Rotate** option can be used to rotate dimension text about its center point. It works in exactly the same way as the Angle option of the DIMTEDIT command except that you can rotate any number of dimensions at once.

The **Oblique** option is used to set the dimension lines at an angle. This option can be very useful when you are dimensioning a drawing in *isometric projection* (see Figure 23). In this case the drawing has been dimensioned using the Aligned command and then the oblique angle modified to suit the dimension position. This usually means setting an angle of 30, 330 or 90 degrees depending upon the dimension orientation. If you are creating details in isometric projection make sure you are using the isometric snap/grid option for greater efficiency.

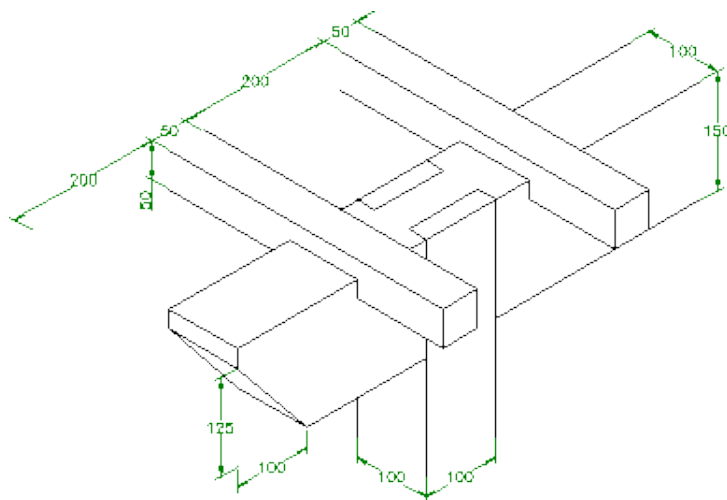


Figure 23

DIMENSION STYLES

Dimension styles are the main method used to control the way dimensions look. Using styles you can change the text font, the arrow head style, the relative position of the

text, the scale of dimensions and many other parameters. Styles are created using the DIMSTYLE command.

Dimension styling is a relatively complex area of AutoCAD and the finer points are beyond the scope of this unit. However, the main points which will enable you to create clear, good looking styles are set out below.

The Dimension Style Command

Toolbar 
 Pull-down **Dimension/Style...**
 Keyboard **DDIM** short-cut **D**

The Dimension Style command can be used to change the appearance of dimensions. The best method is to create a new style before you start creating dimensions so that you can leave the STANDARD style as a default option. Having created a new style from STANDARD, then you can apply any modifications you generally require to the *parent* style and then more specific modifications to the *child* styles in order to create a style *family*.

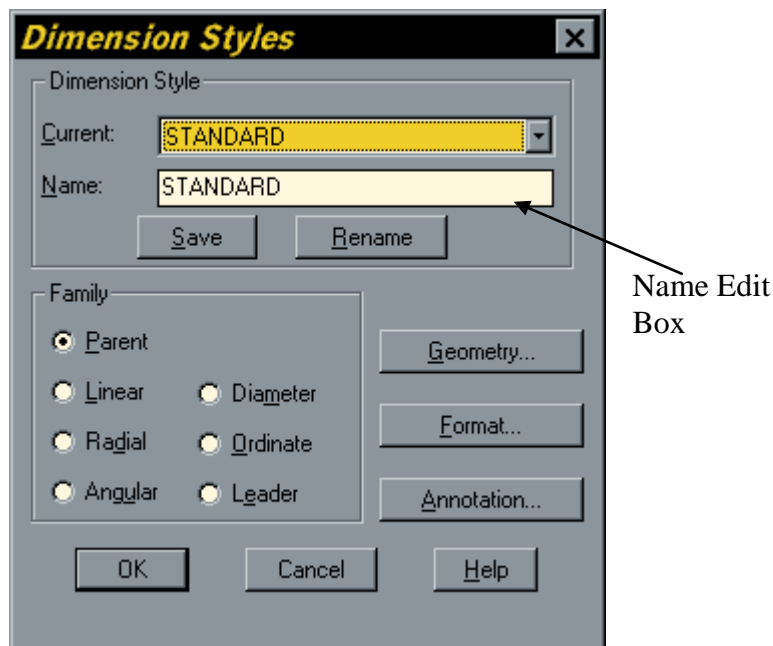


Figure 24

Dimension styles are created using the **Dimension Styles** dialogue box. The dialogue box is shown on *Figure 24*. As you can see from the dialogue box, a style is applied to a *family* of dimensions. By default, any style changes are made to the *parent*. Each style parent has six *child* styles. The child styles, Linear, Radial, Angular, Diameter,

Ordinate and Leader can be used to modify the parent style when that particular type of dimension is used.

For example, you may like to use a tick rather than an arrow head for your dimensions but this isn't really appropriate for a leader, so the Leader child style can be changed so that leaders will always be drawn with an arrow head whilst all other dimensions of the same style family are drawn using ticks.

Creating a New Style

To create a new dimension style, make sure the STANDARD style is the current style, click in the Name edit box and type the name of the new style you wish to create. Click the Save button. You will see a message in the lower left corner of the dialogue box which says "Created *name* from STANDARD" where name is the new style name which you typed. The new style is automatically set as the current style. You may rename the new style if you wish, simply by typing a new name in the Name edit box and clicking on the Rename button.

The new style which you have created is identical to the STANDARD style, so you must now modify your new style so that it can be used to create dimensions which conform to your own requirements. Style changes are made in three categories, **Geometry**, **Format** and **Annotation** (Figure 25). As you can see from the Dimension Styles dialogue box, each category is represented by a button which leads to a dialogue box which is used to modify the settings in that particular category.

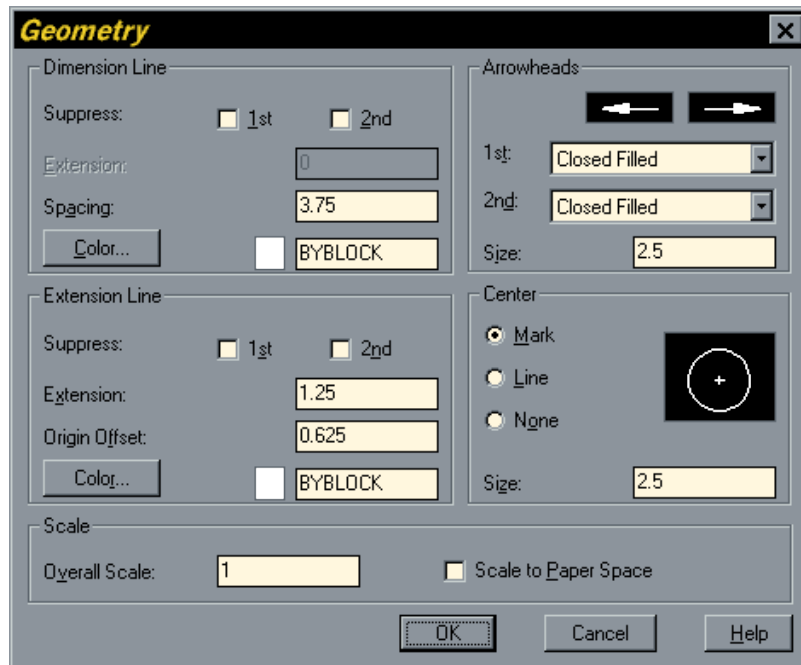


Figure 25

Setting the Arrow Head Type

The style of arrow heads is set using the Geometry dialogue box, illustrated in Figure 26. As you can see, the STANDARD style has Closed Filled arrow heads as a default. To change the arrow head style for a new dimension style, make sure the style is current and that the "Parent" radio button is selected (this assumes you are not modifying a child style), click on the "Geometry..." button and select a new arrow head type from the "1st" drop-down list. Once selected the new arrow type is illustrated in the dialogue box. If you require different arrow heads at each end of your dimensions you can set the other type using the "2nd" drop-down list. Click on "OK" to return to the Dimension Styles dialogue box.

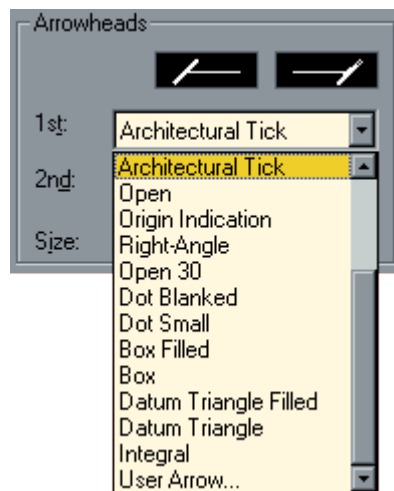


Figure 26

Dimension Scale

When you are working with drawings, which will be plotted, at different scales, you will need some way of changing the scale of the dimension lines relative to your drawing so that they always appear the same size, irrespective of plotting scale. You can achieve this by using the Scale variable. This option is also available from the Geometry dialogue box. The default value is set to 1.0. The larger the value the larger the dimension will appear.

For example, a value of 2.0 would double the text height and the arrow size. To change the scale of dimensions, simply type the required scale in the "Overall Scale" edit box. Try changing the scale factor and check the results. The scaling applies to individual styles, so you could create different styles with different dimension scales to be used for different plotting scales.

Note that changing the scale of dimensions does not affect the dimension value, this is always calculated in drawing units.

Setting the Text Location

To change the text location click on the **"Format..."** button in the *Dimension Styles* dialogue box. The Format dialogue box is shown in *Figure 27*. By default the *horizontal justification* is set to "Centered" and the *vertical justification* to "Above". This means that the dimension text will appear centered above a horizontal dimension line and centered left of a vertical dimension line. To have the text centered within the dimension line, click on the down arrow of the "Vertical Justification" pull-down list to reveal the options and click on "Centered". The illustration changes to reflect your choice. Click on "OK" to return to the Dimension Styles dialogue box. You can see the result of this action by looking at the illustration below. Experiment with the Horizontal Justification and Vertical Justification options to see what results they give.

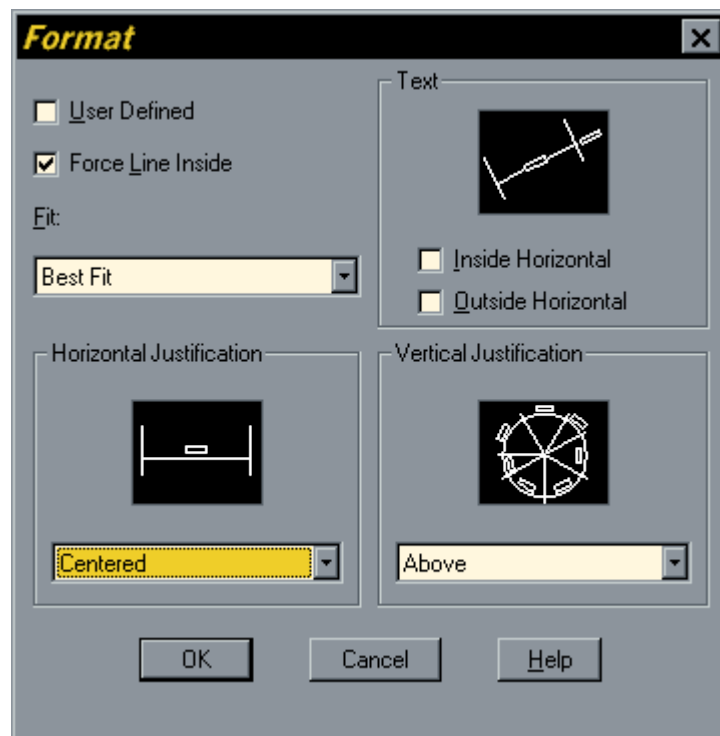


Figure 27

You can also use Text the option in this dialogue box to change the text orientation in aligned dimensions. By default all dimension text is aligned with the dimension. This option allows you to force text to appear horizontal, irrespective of the orientation of the dimension. You have independent control over dimension text which appears inside and outside of the dimension lines.

The illustration on the Figure 28 shows a dimension with vertical justification set to "Above" (left) and to "Centered" (right).

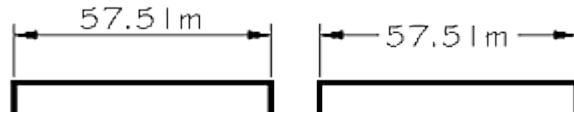


Figure 28

Setting Text Style and Units

Text style and units are both set using the Annotation dialogue box, illustrated in Figure 29. To set a text style to a dimension you must first have created the style using the Text Style command (**Format/Text Style...** from the *pull-down menu*). To assign the text style to a dimension style, click on the "Annotation..." button in the *Dimension Styles* dialogue box, click on the drop-down list in the "Text" area of the Annotation dialogue and select the required text style from the list. Click on "OK" to return to the *Dimension Styles* dialogue.

AutoCAD gives you the option to automatically include a unit prefix or suffix with the dimension text. For example, you could set the dimension style in such a way that it created dimensions with "m" to indicate meters after each dimension text. Most usually, dimensions are drawn without units displayed but with a note on the drawing indicating the units used, such as "All dimensions in meters". However, you may have a drawing where different units are being used for different elements of the drawing.

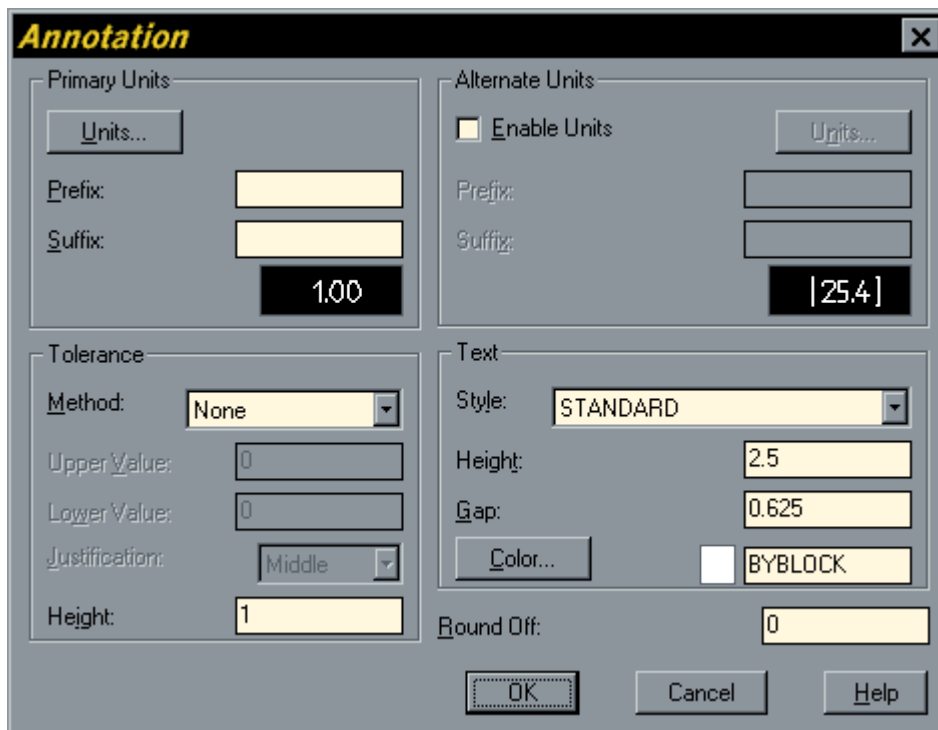



Figure 29

In such a case it is a good idea to include units to avoid confusion. Remember that the main idea behind dimensioning is to give the maximum amount of information in the clearest and most concise way. To add units to a dimension style, click on the "Annotation..." button in the Dimension Styles dialogue box and enter the required unit character(s) in the "Prefix" and/or "Suffix" edit boxes of the "Primary Units" area of the dialogue box. For example, if you wanted to display meters, you would type "m" in the "Suffix" edit box.

THE DIMENSION UPDATE COMMAND

Toolbar 
Pull-down **Dimension/Update**
Keyboard **DIM** ←**UPDATE**

The Dimension Update command is used to apply the current dimension style to existing dimensions. You can use this command to change the style of a dimension. Unlike text styles, dimension styles do not automatically update when the style is changed. The UPDATE command must be used to force dimensions to appear in the current text style.

Command Sequence

Command: **DIM**

Dim: **UPDATE**

Select objects: *(pick dimension to update)*

Select objects: *(pick more dimensions or ← to end)*

Dim: *(press the escape key, **Esc** to return to the command prompt)*