4. Principles of Dimensioning

Drawings for manufacturing, architecture, construction, and other fields are usually incomplete without dimensions. *Dimensions* are defined as numerical values expressed in appropriate units of measure and indicated on a drawing and in other documents. Dimension numerals are placed along with lines, symbols, and notes to define the size and geometric or construction characteristics of a feature, part, or construction process or building component. Dimension numerals in mechanical manufacturing drafting are given in decimal inches or in metric units of millimeters.

Mechanical drafting usually uses *unidirectional dimensioning,* in which all of the dimension numerals are placed horizontally and read from the bottom of the sheet. Architectural, structural, and civil drafting usually use *aligned dimensioning,* in which text for horizontal dimensions is placed horizontally and reads from the bottom of the sheet, text for vertical dimensions is placed vertically and reads from the right side of the sheet, and dimensions placed at an angle have the text placed at the same angle. Electronic drafting projects use either decimal inches or millimeters and use either unidirectional or aligned dimensioning.

Dimensions fall into two types, size dimensions and location dimensions. Size *dimensions* are used to provide the measurements for the size of features and objects. *Location dimensions* are used to locate features and objects. Notes are also placed on drawings. *Notes* are not normally part of the dimensions but they may contain dimensional information. The two types of notes are general notes and specific notes. *General notes* supply information that applies to the entire drawing. *Specific notes,* also known as local notes, apply to specific features on the drawing and may be connected to the feature with a leader line.

In addition, dimensions can be put into three groups relative to the function of a product: functional dimensions, non-functional dimensions, and auxiliary dimensions.

4.1. FUNCTIONAL DIMENSIONS

Functional dimensions directly affect the function or working of a product and may be of the size or location types. Functional dimensions should be based on the function of the component and they can also show the method of locating the component in its appropriate assembly, thus ensuring its correct working within the whole product.

A *datum* is a reference line on the drawing from which a component is dimensioned. In practice, a datum is any functional surface or axis used for manufacture, inspection, location, or assembly purposes. To ensure the required accuracy of measurement, the datum surfaces are machined to the required degree of finish.

The main function of the components in the assembly shown in Figure 4.1 is to support a shaft. Hence the functional location dimension, F_1, is between the hole centre and the datum mating face. The fixing holes must be positioned in relation to the shaft and the shoulder on the mating face of the table. The functional dimension F_2 is used for this purpose.
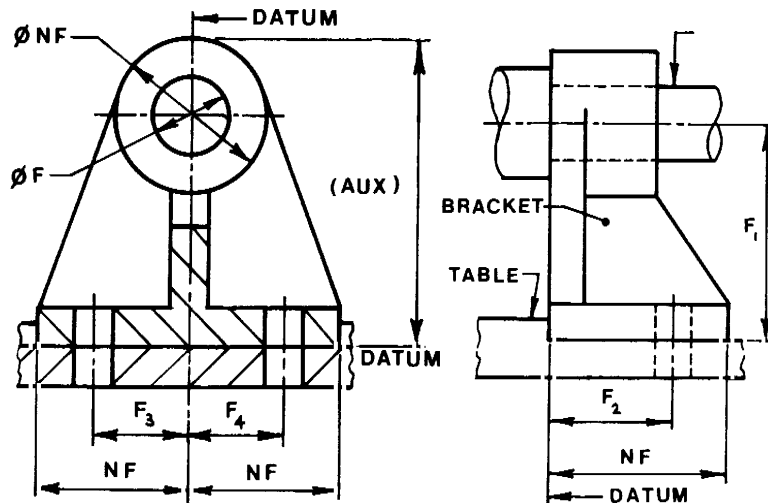
Figure 4.1 Functional, non-functional, and auxiliary dimensions

The same dimension F_2 will apply to the bracket and the table. Also, the centre lines of the holes must be positioned relative to the vertical datum using the functional dimensions F_3 and F_4. Functional dimensions are used for production and inspection purposes. They should always be toleranced.

4.2. NON-FUNCTIONAL DIMENSIONS

Non-functional dimensions, NF in Figure 4.1, are those dimensions which are used for production purposes but which do not directly affect the function or working of a product. Non-functional dimensions usually are not inspected.

4.3. AUXILIARY DIMENSIONS

Auxiliary dimensions, (AUX) in Figure 4.1, are given for information only. They are not used for production or inspection purposes, they should not be toleranced, and they should always be inserted in parentheses (brackets). Auxiliary dimensions are *redundant* dimensions, which provide useful information but do not govern acceptance of the product.

4.4. PRINCIPLES OF DIMENSIONING

Dimensions are normally expressed in millimeters. Dimensions of less than unity should be preceded by zero, e.g. 0.6 mm. Each dimension should appear only once — it should not be repeated on other views. Dimensions relative to a particular feature should be placed in one view, which shows the relevant features most clearly, rather than spread over several views. Principles of dimensioning are shown in Figure 4.2.
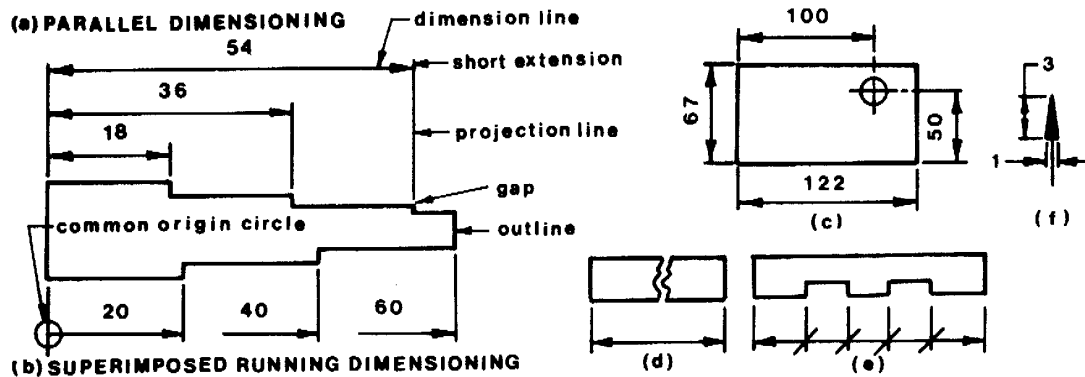
PDF created with FinePrint pdfFactory Pro trial version http://www.fineprint.com

Figure 4.2 Principles of dimensioning

4.5. AUTOMATIC DIMENSIONING

Some more powerful CADD systems allow dimensions to be added automatically to a drawing. In a drawing made by hand, there is no necessary or inherent correspondence between the dimensions of the drawn outline and the intended dimensions of the object, which it represents, until a scale has been specified. Moreover, the dimensional accuracy of the outline depends on the skill of the draughtsman. However, in a computer-generated drawing the actual coordinate values of points are entered explicitly and the image is generated *from* these. The machine can therefore calculate dimensions of length and angle from the coordinate information directly, and attach dimensional notations to the drawings for arrowheads, leaders, and the positioning and size of lettering. This is not, however, a trivial task, since it requires some judgment and skill on the part of the human draughtsman — which the machine must emulate — to decide where to place lettering and leaders so that these are clear and legible.

Computer drawings, unlike drawings made by hand, do not have to be constructed to some fixed scale. The actual dimensions of the required object are entered, and the drawing is displayed in a screen window at some arbitrary scale suited to the window size. The user can zoom in on selected parts of the drawing, in order to work on finer detail and to overcome the problem of the limited resolution of the screen. A fixed scale needs only be decided when the finished drawing is dispatched for plotting.

4.6. DIMENSION PLACEMENT AND CONVENTIONS

Engineering drawings are usually drawn to scale, but it is still necessary to specify numerical dimensions for convenience and to ensure accuracy. In general, sufficient dimensions must be provided to define precisely the geometry of the part, but redundant dimensions should be avoided. Furthermore, dimensions on the drawing should reflect the way the part is made or the critical dimensions of the part. But suppose that the distance between the holes is critical. This might be necessary if the holes in the plate are supposed to align with another part. Then the dimensions on the middle drawing would clearly show that the distance between the holes is most important by omitting the dimension from the right hole to the right edge. The right drawing indicates that the distance from the left edge to each hole, not the distance between the holes, is critical. The engineer or designer must consider the optimal way to display dimensions to clearly show which dimensions are most important.

It is natural for a person reading the drawings to look for the dimensions of a feature in the view where the feature occurs in its most characteristic shape and where it is visible (as opposed to hidden). This is known as *contour dimensioning.* For example, the location and size of a hole should be dimensioned in the view

3

where the hole appears as a circle. Likewise, it is best to show the overall dimensions of the object in a view that is most descriptive of the object. If possible both the horizontal and vertical location of a feature should be dimensioned in the same view.

4.6.1. Dimensioning Concept

Dimensions show the geometric measurements of objects, the distances or angles between objects, X and Y coordinates of a feature. CADD software such as AutoCAD provides three basic types of dimensioning: linear, radial and angular. Linear dimensions include horizontal, vertical, aligned, rotated, ordinate, baseline and continued dimensions. A simple example of each dimension is shown in Figure 4.3.
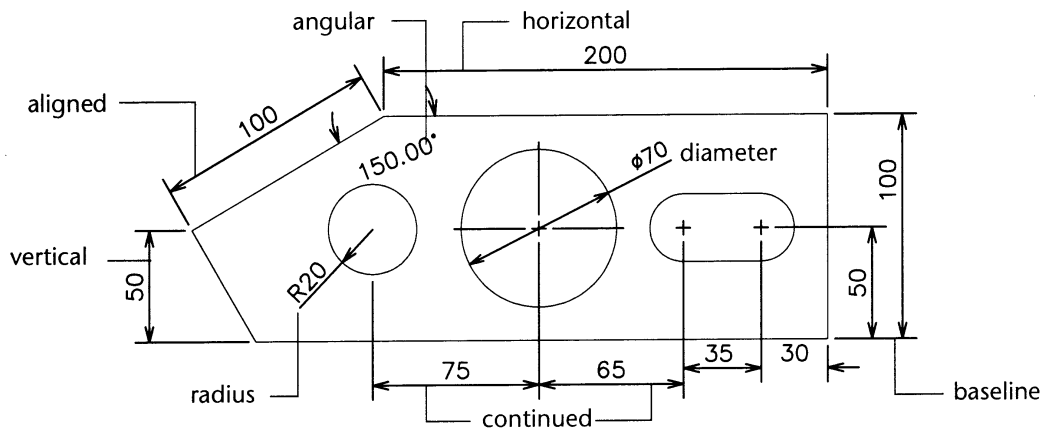

Figure 4.3. Dimensioning

4.6.2. Dimensioning System Variables

The dimensioning system variables control the appearance of dimensions. You may set these variables on the command line. Table 4.1 summarizes the most important dimensioning system variables and special characters are listed in Table 4.2.

Table 4.1. Dimensioning system variables

Variable	Value	Explanation
Dimasz	<2.5>	Controls the size of the dimension arrowhead
Dimgap	<1.5>	Sets the distance around the dimension text above its line
Dimexe	<2>	Determines how far to extend the extension line
Dimexo	<0>	Determines how far extension lines are offset from origin point
Dimdli	<10>	Sets the distance between adjacent dimension lines
Dimrnd	<0>	Rounding dimension text value

Table 4.2 Special characters

Diameter	%%C	Ø	%%C 50	Ø 50
Degree	%%D	°	50%%D	50 °
Plus/minus	%%P	±	50 %%P 2	50 ± 2

4

4.7. DIMENSION TOLERANCES

Tolerance is the total amount that a specific dimension is permitted to vary; it is the difference between the maximum and minimum limits for the dimension. Tolerances can be applied directly to dimensioned features, using limit dimensioning. The maximum and minimum sizes are specified as part of dimensions. In plus/Minus Dimension approach, the basic size is given, followed by a plus/minus sign and the tolerance value. Tolerances can be unilateral or bilateral. A unilateral tolerance varies in only one direction. A bilateral tolerance varies in both directions from the basic size.

Tolerance dimensions are typically used on mechanical drawings for manufacturing. The tolerance method options are explained in the following list (Figure 4.4):

Symmetrical : This option is used to draw dimension text displaying an equal bilateral tolerance.
Unequal bilateral: This option is a tolerance for which the variation is permitted unequally in both directions from the specified (nominal) dimension.
Unilateral : This option is a tolerance for which the variation is permitted in only one direction for the specified (nominal) dimension.
Limits : Those are referred to as plus/minus tolerance. You may supply to calculate a maximum and minimum value.
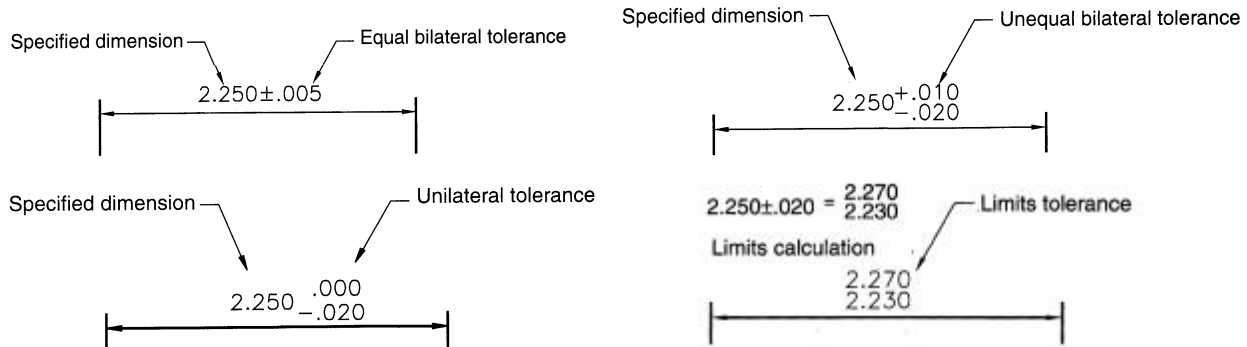

Figure 4.4. Dimension tolerances

Applying Dimension Tolerances in CADD

The Annotation dialog box from AutoCAD shown in Figure 4.5 contains the settings for the dimensional tolerances for both primary and secondary units. Pick to down arrow to see the other options in the Method list as shown in Figure 4.5.
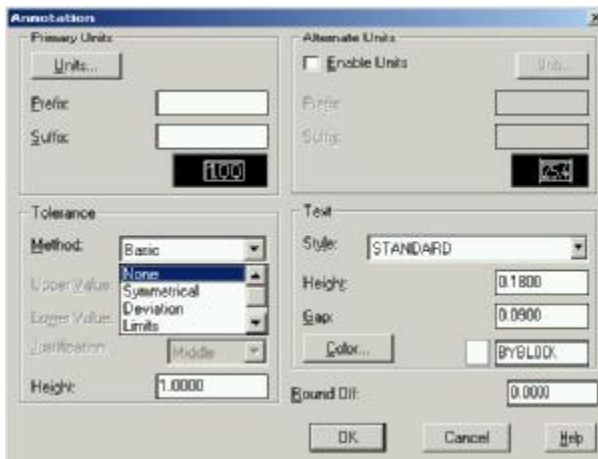

Figure 4.5. Applying Dimension Tolerances in AutoCAD

5

4.8. GEOMETRIC DIMENSIONING & TOLERANCING

Geometric dimensioning and tolerancing (GD&T) frames are a special language to describe manufacturing tolerances on part features that can not be specified as just a ± tolerance on a dimension.

4.8.1. Datum

Datum feature frames and datum targets indicate key reference surfaces and points. They are normally applied to surfaces used to mount the part for machining, to locate the part in an assembly, or to mount the part on a coordinate measurement machine (CMM) for inspection. The datum symbol below placed on a drawing labels a surface as surface "A" that will be referred to in GD&T feature control frames (Figure 4.6a)

| A | | ¡ | $\varnothing$ 0.5 | A |

   (a)   (b)
Figure 4.6. Geometric tolerance frames

GD&T feature control frames contain three parts (Figure 4.6b). The first piece of information in a feature control symbol is always a geometric characteristic symbol. The next block contains tolerance values and optional modifying symbols. The third block contains datum references that this feature control frame refers to. This will depend on the geometric symbol in the first block. Table 4.3 summarizes the geometric characteristic symbols.

Tablo 4.3. Geometric characteristic symbols for GD&T

Symbol	Name	Symbol	Name
—	Straightness	◎	Concentricity
▱	Flatness	⟠	Symmetry
○	Circularity	≡	Positional tolerance
⌭	Cylindricity	⌒	Profile of a line
//	Parallelism	⌓	Profile of a surface
⊥	Perpendicularity	↗	Circular runout
∠	Angularity	⌰	Total runout

4.8.2. Modifying Symbols

In some cases, the following modifying symbols will be added to the tolerance field of the feature control frame. The symbol $\varnothing$ for diameter always is placed in front of the value, if used. The symbol (M) is used to state that a tolerance applies at the maximum material condition (MMC), the symbol (L) is used for the least material condition (LMC) and the symbol (S) is used for regardless of feature size (RFS).

An (M) symbol in a feature control frame as shown in Figure 4.6 means if the feature is at MMC, this is the tolerance in its position. If the symbol (M) were replaced by the symbol (S) in the feature control frame

in Figure 4.7, it would mean that the tolerance on the location of this hole is always 0.1 mm, regardless of the size of the hole. Of the three condition, this is the strictest.
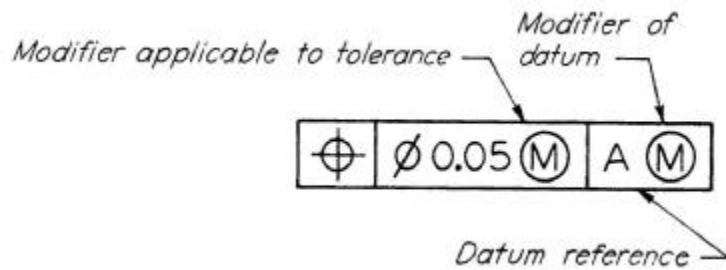

Figure 4.7. Maximum materials condition

4.8.3. Basic Dimensions

Basic dimensions are the theoretical sizes from which limits of size are derived by the application of allowances and tolerances. It is the size form which limits are determined for the size, shape or location of a feature. The basic dimension symbol is identified by the enclosing frame symbol (Figure 4.8).
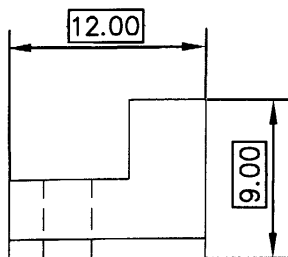

Figure 4.8. Basic dimensions

4.8.4. Geometric Tolerance Application in CADD

CADD software such as AutoCAD has built in dialog boxes which allow you to create feature control frames for geometric dimensioning and tolerancing. You can create a feature control frame by picking the Tolerance icon from AutoCAD's Dimensioning toolbar. When you do so the dialog box shown in Figure 4.9 will appear on your screen.
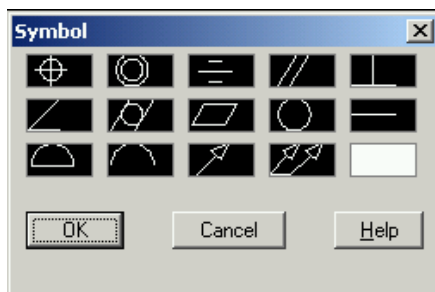

Figure 4.9. Tolerance symbol menu

To create a feature control frame all you need to do is double click on the symbol you want to use. When you have selected the symbol a new dialog box shown in Figure 4.10a will appear on your screen. You may also add diameter symbols, modifiers, datum references, or datum identifiers.
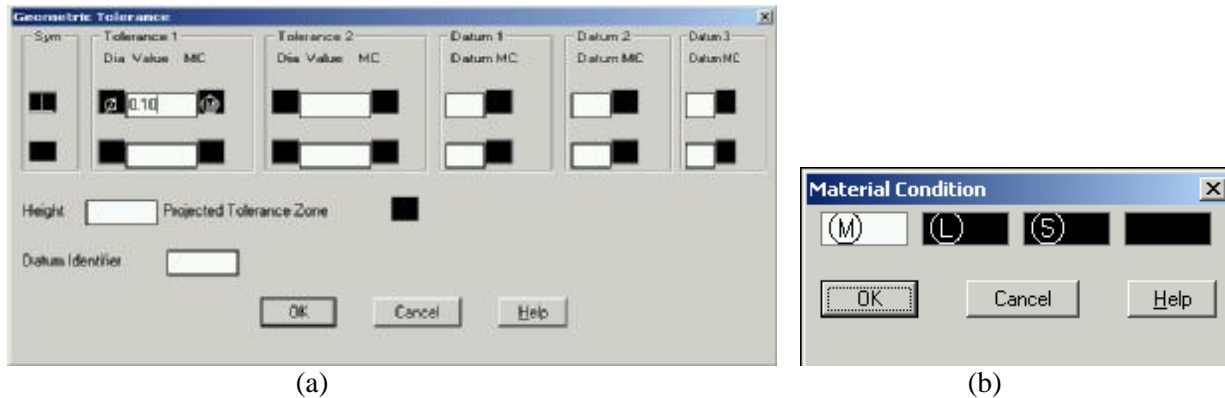
7

Figure 4.10. Geometric tolerancing and material dialog box

The diameter symbol shown in the dialog box was added just by picking in the empty bow below the heading **Dia**. The next area is a text entry box for the tolerance value. To add modifier symbol pick in the next empty box to the right. The material condition dialog box shown in Figure 4.10b pops up on the screen. Datum references can be created just as quickly by picking in the appropriate box and typing in the letter you want to use.

The some CADD software contains a syntax checker to check for the proper syntax of the GD&T symbol construction. This checks for proper use of datum references for geometric characteristic symbols that require them, and proper use of modifying symbols. GD&T control frames will be placed on drawings that are made of the part. They will also remain with the part for other downstream uses, such as manufacturing and inspection.

Questions
1. What dimension type that defines a feature must be available to allow for the creation of a rotational pattern?
2. Define associative dimensions
3. Explain the advantages of having associative dimensions.
4. How do you change the dimension variables on existing dimensions in a drawing?
5. Define the term dimension?
6. Explain the difference between size and location dimensions
7. What are linear dimensions?
8. How is chain dimensioning done?

References

1. D.S. Kelley, **Pro/Engineer Instructor**, McGraw-Hill, 2001
2. G.R.Bertoline, et.al., **Technical Graphics Communication**, WCB McGraw-Hill, 1997
3. J.Rooney, P.Steadman, **Principles of Computer-aided Design**, UCL Press, 1997
4. D.A. Madsen, T.M.Shumaker, J.L. Turpin, **Fundamentals of Drawing Technology,** Delmar Publ., 1994.
5. F.E. Giesecke, et.al., **Engineering Graphics**, Prentice Hall, 2000.
6. F.E. Giesecke, et.al., **Modern Graphics Communication**, Prentice Hall, 2001.
7. O. Ostrowsky, **Engineering Drawing with CAD Applications**, ELBS Pub., 1993.
8. M.A. Parker, **Drawing Standards for Computer-aided Engineering**, City and Guilds/Macmillan, 1995.
9. M.H. Lawry, **I-DEAS Student Guide**, McGraw Hill Higher Education Publ., 2001.