Mechanics

LEVEL- III

Learning Guide #01

Unit of Competence: Perform Advanced Engineering Detail Drafting

Module Title: Performing Advanced Engineering Detail Drafting

Module Code: XXX

LG Code: XXX

TTLM Code: XXXX

LO 1: Determine drawing Requirements

Instruction Sheet: 1	Learning Guide #01

This learning guide is developed to provide you the necessary information regarding the following **content coverage** and topics:

- Checking and interpreting requirements and purpose of drawing
- Sourcing of required information from workshop manuals,
  - ✓ customer specifications,
  - ✓ product suppliers, and
  - ✓ Designers or similar.
- planning Scope of drawing

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, **upon completion of this Learning Guide, you will be able to**:

- Check and interpret drawing Requirements and purposes from work order.
- Source required information from workshop manuals, customer specifications, product suppliers, and designers or similar.
- Plan scope of drawing including layout, additional required information and resources.

Learning Instructions:

1. **Read the specific objectives of this Learning Guide**.
2. Follow the instructions described below 3 to 6.
3. Read the information written in the information “Sheet 1, Sheet 2, and 3 Sheet”
4. Accomplish the “Self-check 1, Self-check t 2, and Self-check 3” **in page -17, 21, and 25.**

Information Sheet-1	Checking and interpreting requirements and purpose of drawing

1. Introduction/ Overview of Drawing

A drawing is a graphic representation of an object, or a part of it, and is the result of creative thought by an engineer or technician. When one person sketches a rough map in giving direction to another, this is graphic communication. Graphic communication involves using visual materials to relate ideas. Drawings, photographs, slides, transparencies, and sketches are all forms of graphic communication.

A technical person can use the graphic language as powerful means of communication with others for conveying ideas on technical matters. However, for effective exchange of ideas with others, the engineer must have proficiency in (i) language, both written and oral, (ii) symbols associated with basic sciences and (iii) the graphic language. Engineering drawing is a suitable graphic language from which any trained person can visualize the required object. As an engineering drawing displays the exact picture of an object, it obviously conveys the same ideas to every trained eye. Hence, an engineer should posses good knowledge, not only in preparing a correct drawing but also to read the drawing correctly. This module is expected to meet these requirements. The study of machine drawing mainly involves learning to sketch machine parts and to make working and assembly drawings. This involves a study of those conventions in drawings that are widely adopted in engineering practice.

1.1. Classifications of Drawing

1.1.1. Machine Drawing

It is pertaining to machine parts or components. It is presented through a number of orthographic views, so that the size and shape of the component is fully understood. Part drawings and assembly drawings belong to this classification. An example of a machine drawing is given in Fig. 1.1.

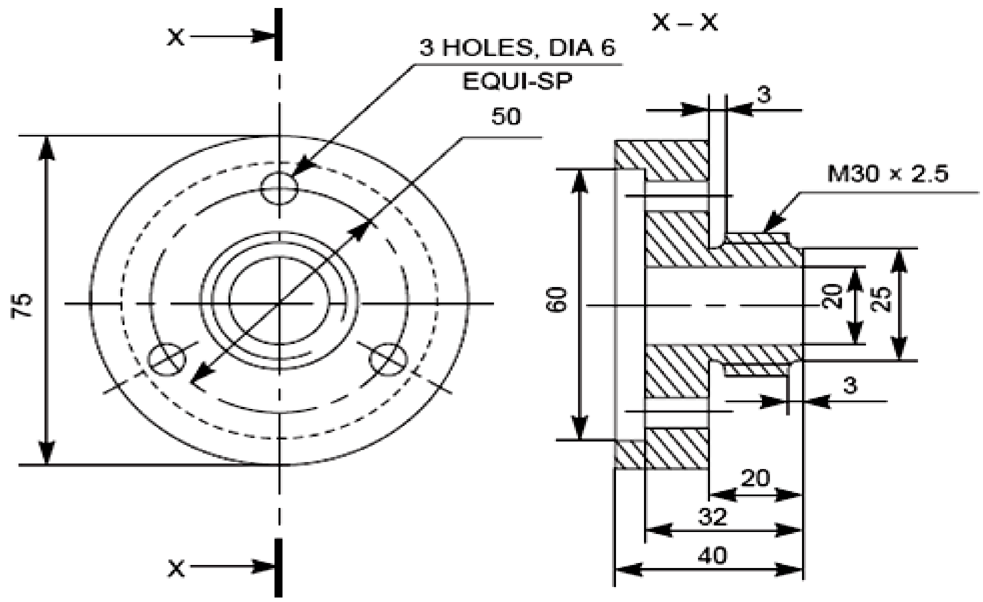


Figure 1.1 Machine Drawing

1.1.2. Production Drawing

A production drawing, also referred to as working drawing, should furnish all the dimensions, limits and special finishing processes such as heat treatment, honing, lapping, surface finish, etc., to guide the craftsman on the shop floor in producing the component. The title should also mention the material used for the product, number of parts required for the assembled unit, etc. Since a craftsman will ordinarily make one component at a time, it is advisable to prepare the production drawing of each component on a separate sheet. However, in some cases the drawings of related components may be given on the same sheet. Figure 1.2 represents an example of a production drawing.

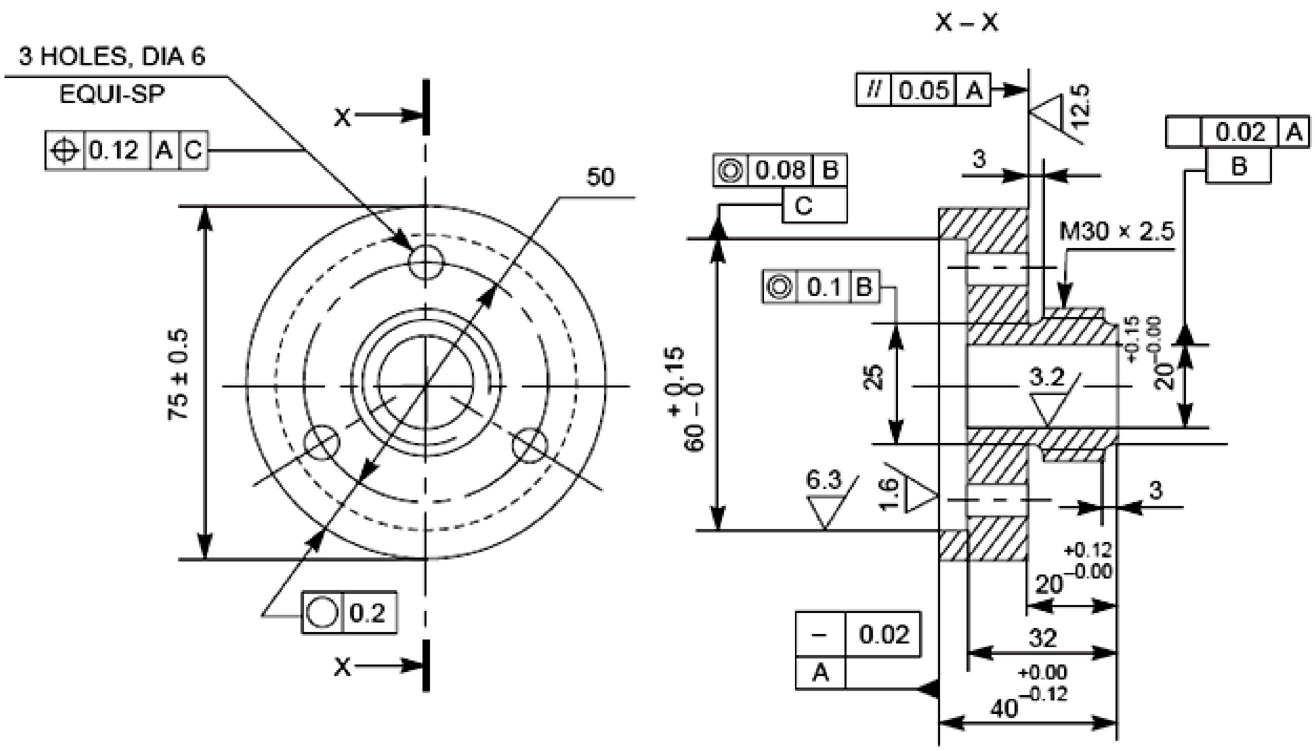


Figure 1.2. Production Drawing

1.1.3. Part Drawing

Component or part drawing is a detailed drawing of a component to facilitate its manufacture. All the principles of orthographic projection and the technique of graphic representation must be followed to communicate the details in a part drawing. A part drawing with production details are rightly called as a production drawing or working drawing.

1.1.4. Assembly Drawing

A drawing that shows the various parts of a machine in their correct working locations is an assembly drawing (Fig. 1.3). There are several types of such drawings.

1.1.4.1. Design Assembly Drawing

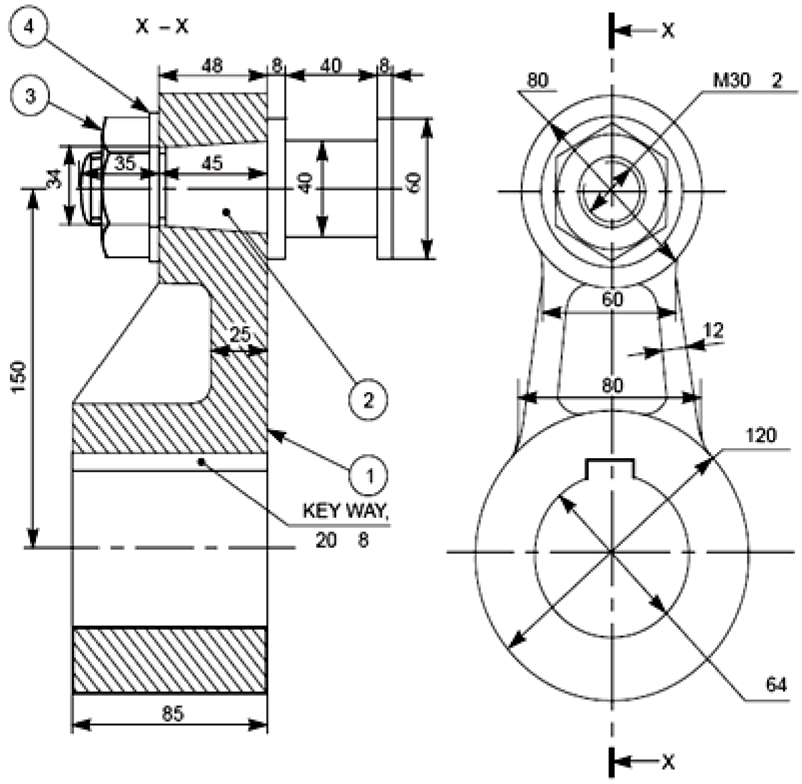
When a machine is designed, an assembly drawing or a design layout is first drawn to clearly visualize the performance, shape and clearances of various parts comprising the machine.

1.1.4.2. Detailed Assembly Drawing

It is usually made for simple machines, comprising of a relatively smaller number of simple parts. All the dimensions and information necessary for the construction of such parts and for the assembly of the parts are given directly on the assembly drawing. Separate views of specific parts in enlargements, showing the fitting of parts together, may also be drawn in addition to the regular assembly drawing.

1.1.4.3. Sub - Assembly Drawing

Many assemblies such as an automobile, lathe, etc., are assembled with many pre-assembled components as well as individual parts. These pre-assembled units are known as sub-assemblies. A sub-assembly drawing is an assembly drawing of a group of related parts, that form a part in a more complicated machine. Examples of such drawings are: lathe tail-stock, diesel engine fuel pump, carburetor, etc.



Parts List

Part No.	Name	Material	Qty
1	Crank	Forged Steel	1
2	Crank Pin	45C	1
3	Nut	MS	1
4	Washer	MS	1

Figure1.3. Assembly Drawing

1.1.4.4. Installation Assembly Drawing

On this drawing, the location and dimensions of few important parts and overall dimensions of the assembled unit are indicated. This drawing provides useful information for assembling the machine, as this drawing reveals all parts of a machine in their correct working position.

1.1.4.5. Assembly Drawing for Catalogue

Special assembly drawings are prepared for company catalogues. These drawings show only the pertinent details and dimensions that would interest the potential buyer. Figure 1.4 shows a typical catalogue drawing, showing the overall and principal dimensions.

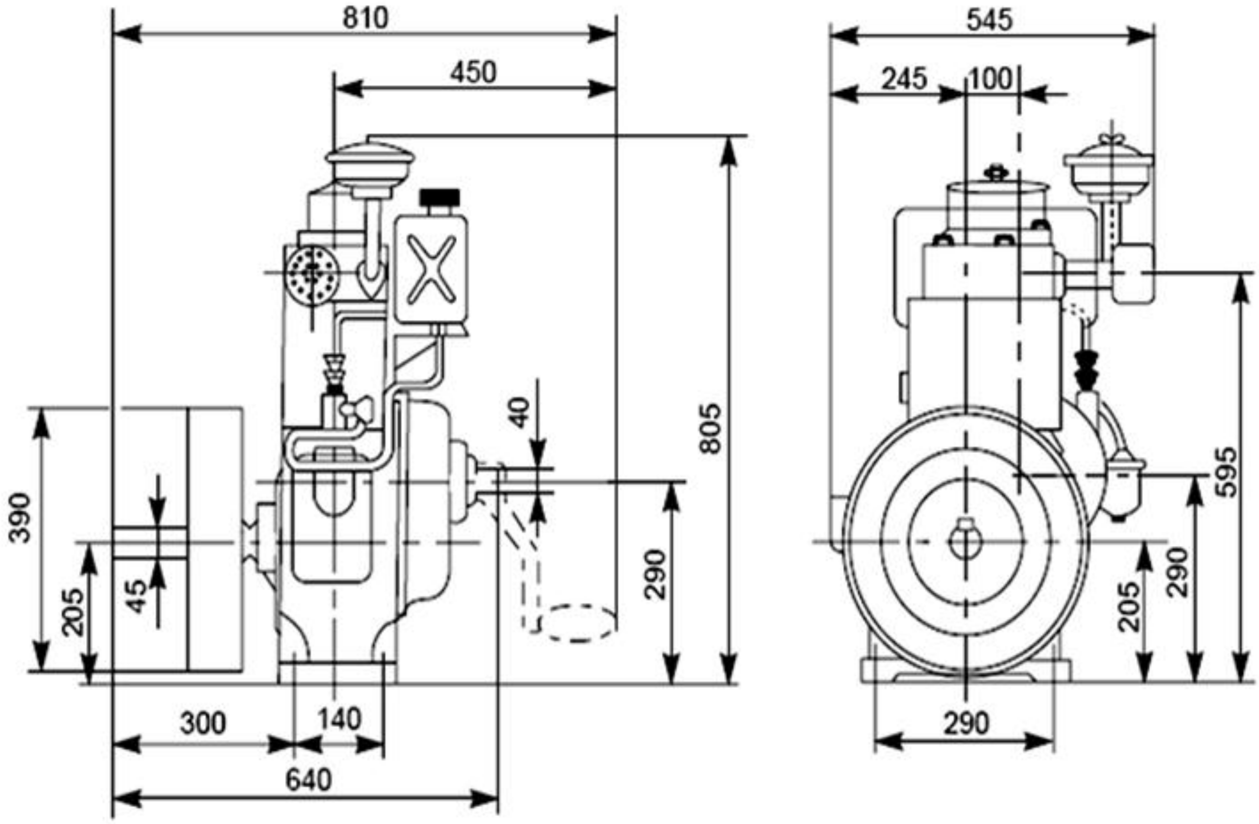
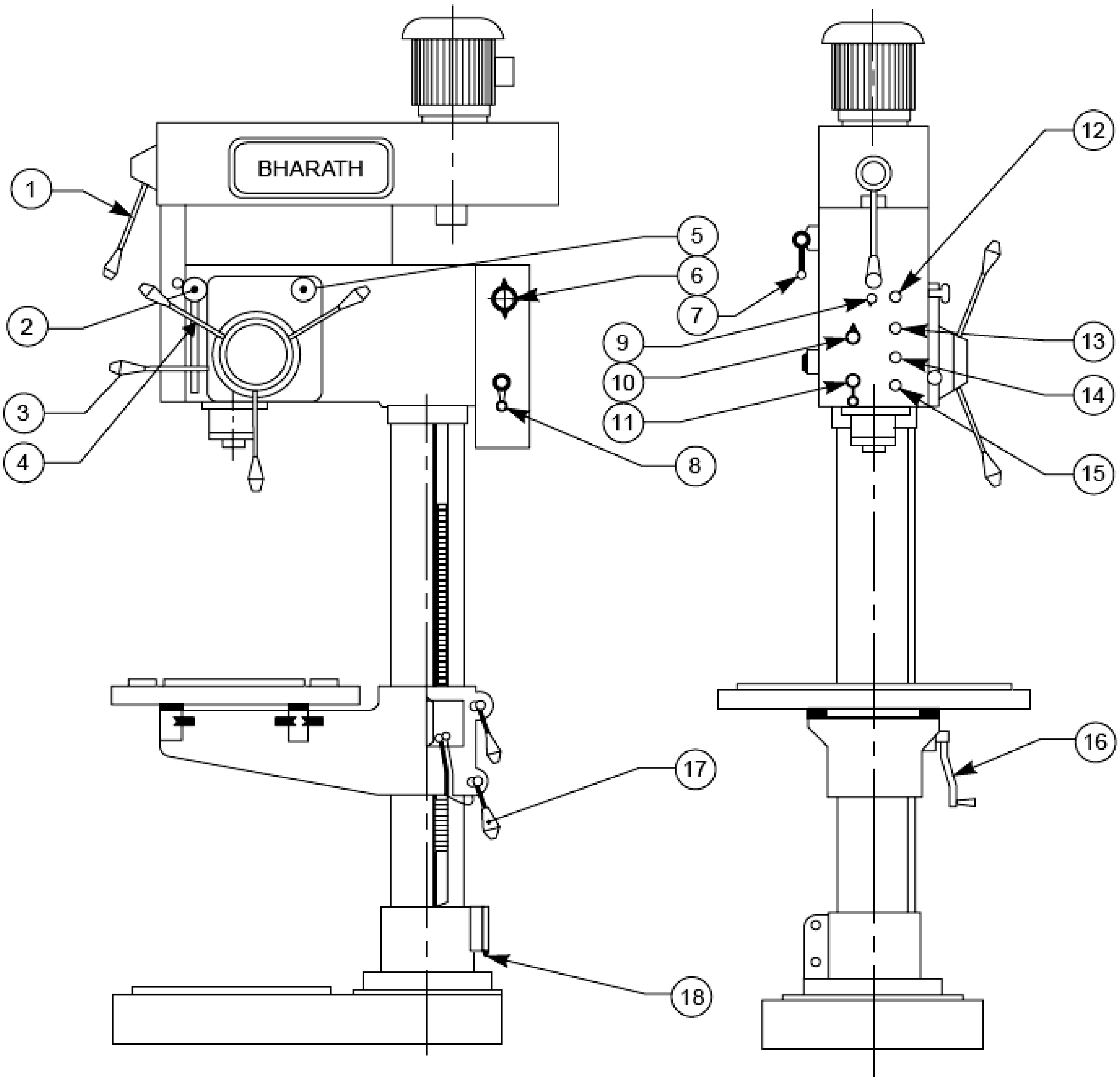


Figure1.4. Catalogue Drawing

1.1.4.6.Assembly Drawing for instruction manual

These drawings in the form of assembly drawings are to be used when a machine, shipped away in assembled condition, is knocked down in order to check all the parts before reassembly and installation elsewhere. These drawings have each component numbered on the job. Figure 1.5 shows a typical example of such a drawing.



Speed change lever (1)	Selector switch (10)
Depth adjusting knob (2)	Forward/reverse switch (11)
Mech. feed engagement lever (3)	Pilot lamp (12)
Hand feed lever (4)	Feed disengagement push button (13)
Feed change knob (5)	Start push button (14)
Switch for tapping (6)	Emergency stop (15)
Gear shifting lever (7)	Elevating handle (16)
Main switch (8)	Clamping handle (17)
Lamp switch (9)	Supply inlet (18)

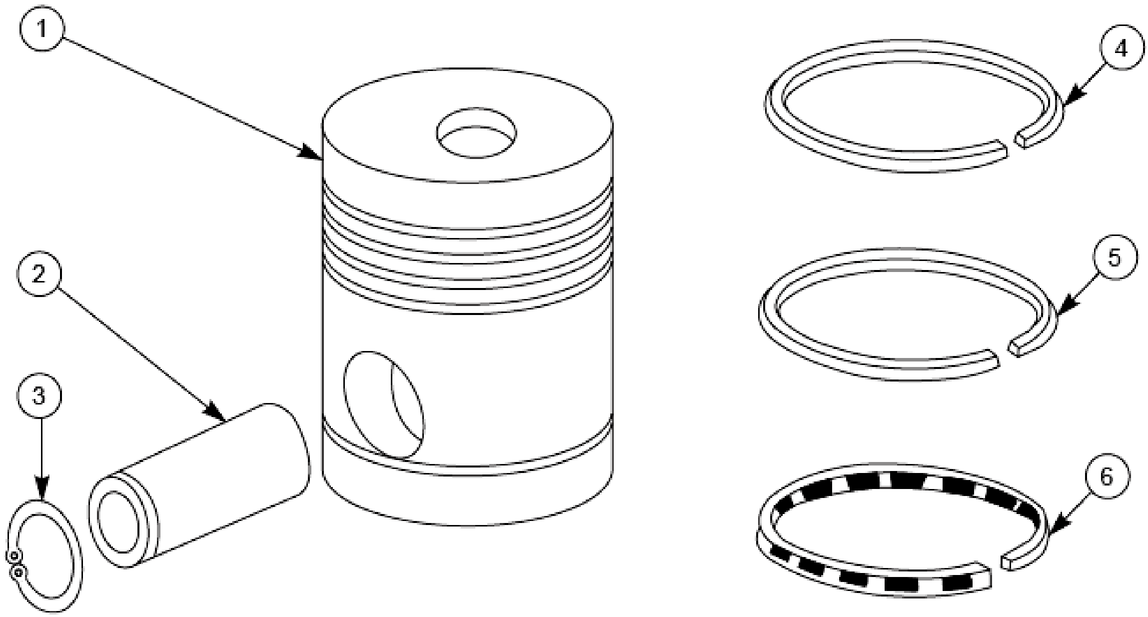
Fig. 1.5 Assembly drawing for instruction manuals

1.1.4.7. Exploded Assembly Drawing

In some cases, exploded pictorial views are supplied to meet instruction manual requirements. These drawings generally find a place in the parts list section of a company instruction manual. Figure 1.6 shows drawings of this type which may be easily understood even by those with less experience in the reading of drawings; because in these exploded views, the parts are positioned in the sequence of assembly, but separated from each other**.**

1.1.4.8. Machine Shop Drawing

Rough castings and forgings are sent to the machine shop for finishing operation (Fig. 1.8). Since the machinist is not interested in the dimensions and information of the previous stages, a machine shop drawing frequently gives only the information necessary for machining. Based on the same principle, one may have forge shop drawing, pattern shop drawing, sheet metal drawing, etc.



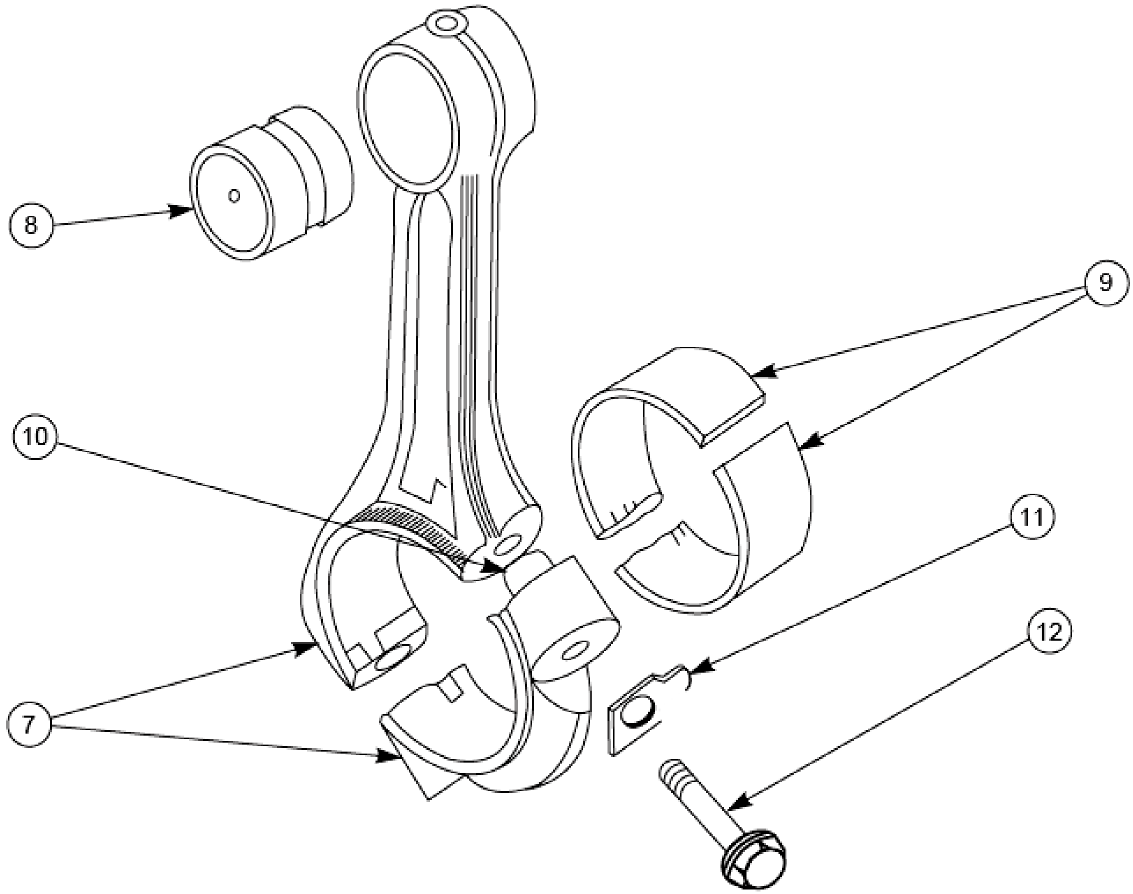


Fig. 1.6 Exploded assembly drawing

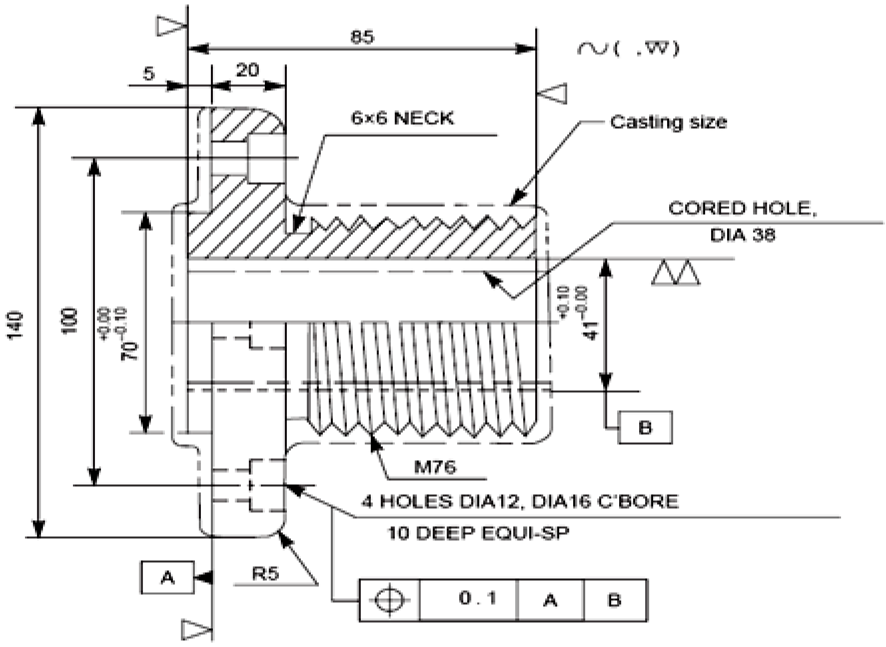


Figure 1.7 Machine Shop Drawing

1.1.5. Sectional Drawing

1.1.6. Sectional View

An engineering drawing has the function of showing shape and size of an object clearly and completely. In certain cases, however, the representation in views is not sufficient. Many objects have internal shapes which are so complicated in nature that it is virtually impossible to show their true shape without employing numerous confusing hidden lines.

View (**a**) (b) Representation of the intersection

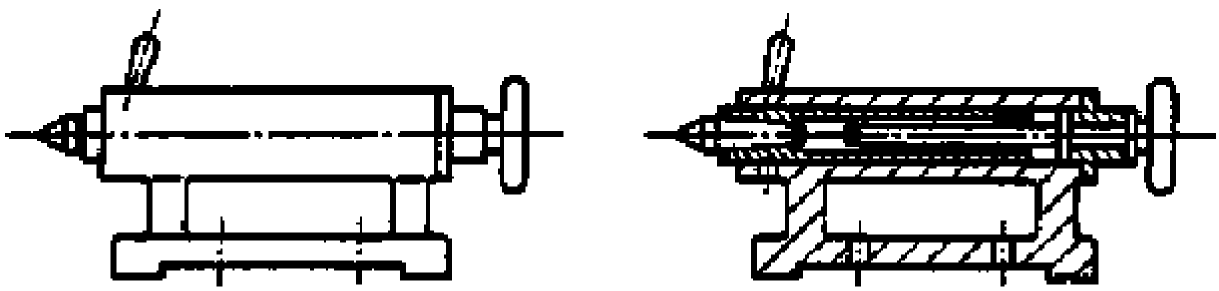
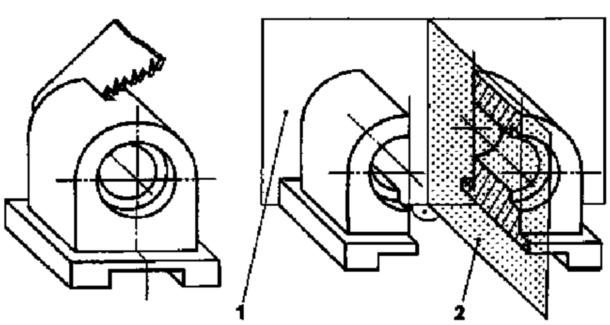


Figure 1.8. Sectional View

The solution of the problem is the use of one or more sectional views: One imagines the object cut by passing a cutting plane through it so that the internal shape is revealed. The purpose of a sectional view is to show the internal shape of an object. In this way, the dimensions of the internal shape can be applied to visible edges.

By passing a cutting plane (e.g. a saw or a knife) that portion of the object is removed which obstructs the view into the interior.



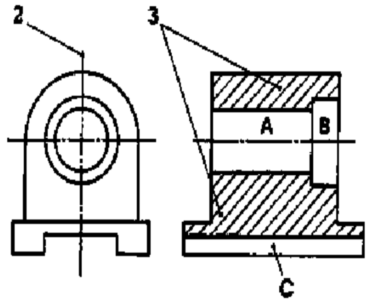


Figure1.9

(**1**) Drawing plane

(2) Cutting plane

(3)Shading

To produce a section view a cutting plane is passed through the part (figure a). The cutting plane is removed and the two halves are drawn apart (figure b) exposing the interior detail. A section view obtained by passing the cutting plane fully through the object is called a full section. In the front view the cutting plane appears as a line called a cutting-plane line. The arrows at the ends of the cutting-plane line indicate the direction of sight for the section view. To obtain the section view the right half of the front view is only imagined to be removed. The cross-hatched areas of the section view are those portions that are in actual contact with the cutting plane.

A correct front view and section view are shown in figure (a) and figure (b). All visible edges and contours behind the cutting plane should be shown. Section views are used primarily to replace hidden lines so as a rule hidden lines should be omitted in section views (figure d).

A section-lined area is always completely bounded by a visible outline, never by a hidden line as in figure (e).

Section lines in a section view must be parallel and at the same angle and direction (figure f).

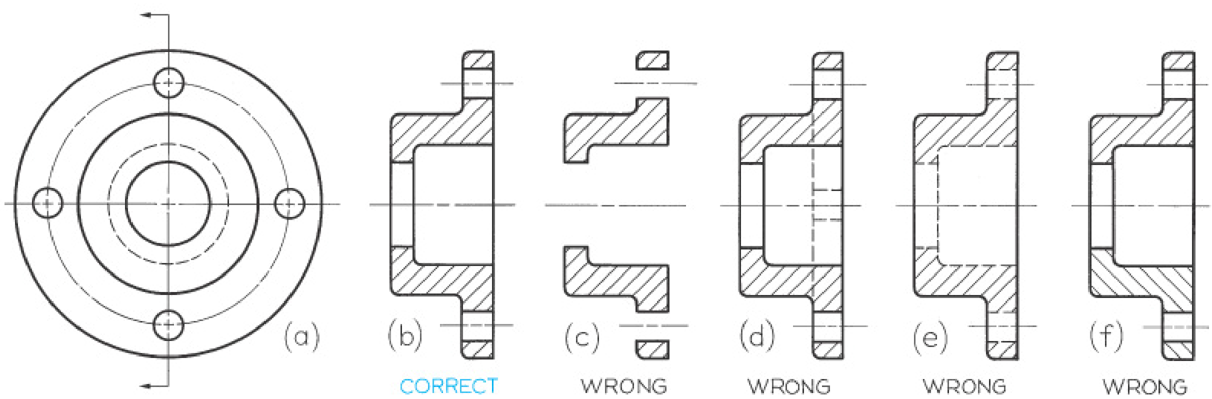


Figure: 1.10. (a) And (b) shows front view and section view

Shading with thin solid lines at an angle of 45 is the general hachure which is used irrespective of the material.

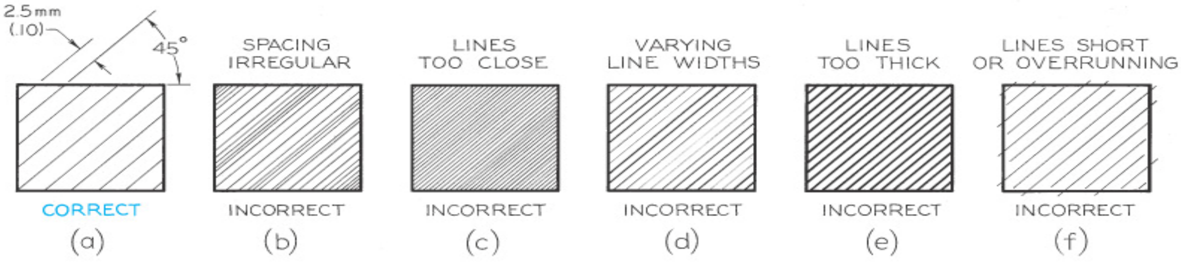


Figure1. 11. Shading with thin solid lines (Hatching)

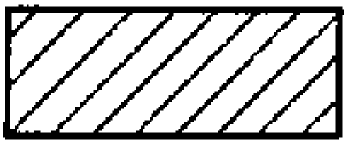
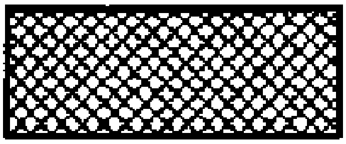
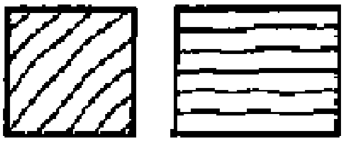
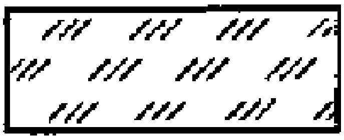
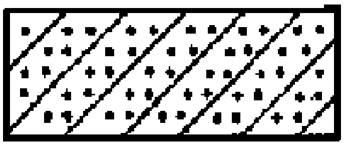
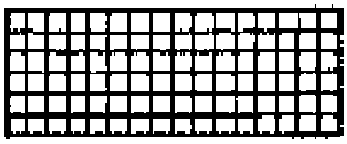
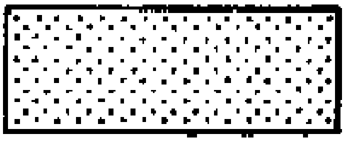
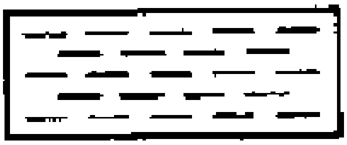
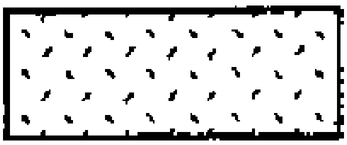
Material	Shading characterising the material
Metal	
Plastics, rubber, felt, leather, filler, material	
Wood (cross–grained wood, other wood)	
Glass and the like	
Reinforced concrete	
electrical windings	
Sintered materials	
Liquids	
Porcelain, marble, slate	

Table1.1. Shading Characterizing of material

1.1.6.1. Types of Sectional View

1 Drawing plane, 2 Cutting plane

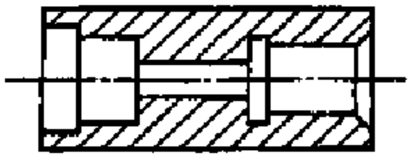
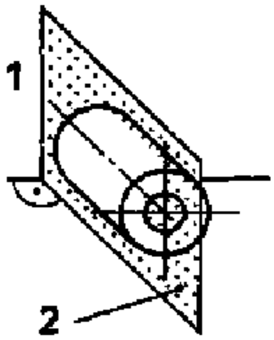
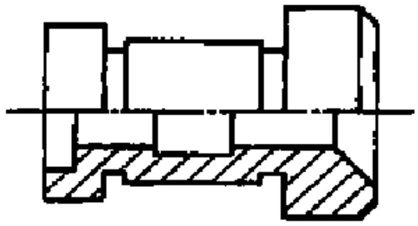
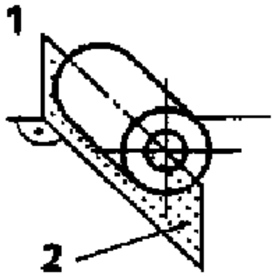
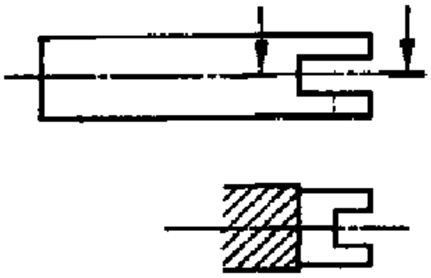
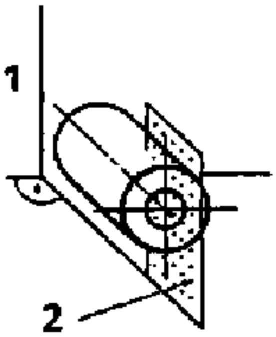
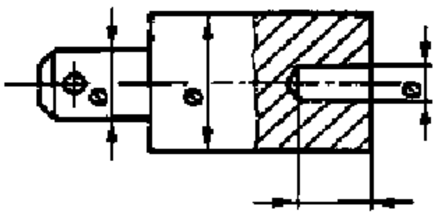
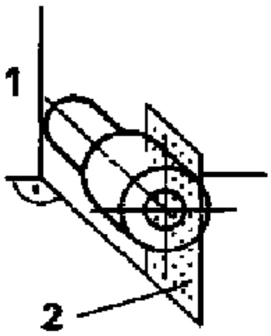
Type of section	Explanations	Applications	Principle
Full sectional view	Objects with intricate internal shapes. Symmetric or non–symmetric or prismatic or cylindrical.		
Half section	Objects with intricate internal and external shapes; usually symmetric, prismatic or cylindrical		
Partial section	An internal partial shape is to be made visible but a full section is not necessary. It would not show more. Cutting line indicated by arrows.		
Broken–out section	This section is used to show only a desired feature of the object. A full section is not necessary or not permitted (shafts, pins, rivets). The view must be shown completely because of other features of the object.		

Table1.2. Types of Sections

To obtain a sectional view, a cutting plane is assumed to be passed through the part (a). This cuts the part in two halves.

The cutting plane is then removed, and the two halves are drawn apart, exposing the interior construction. In (b), the direction of sight is towards the left half. The section view will be in the position of a right side view.

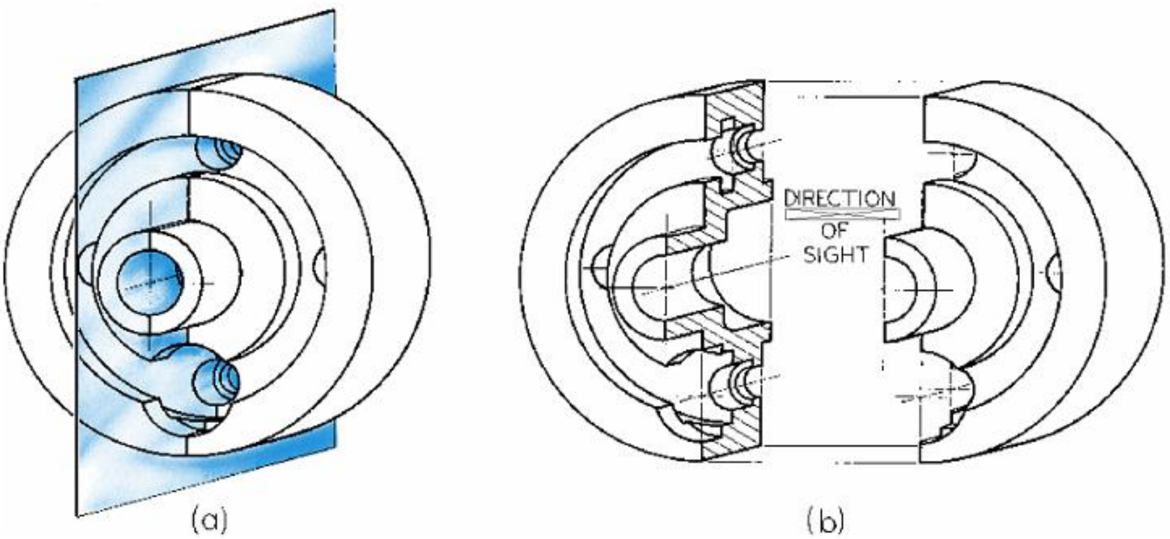


Figure1.12: Sectional View

Cutting Plane Line

- The cutting plane line is a thick dark line which uses one of the special patterns shown.

- The cutting plane line can be left out when it is obvious where it must lie from the appearance of the section itself

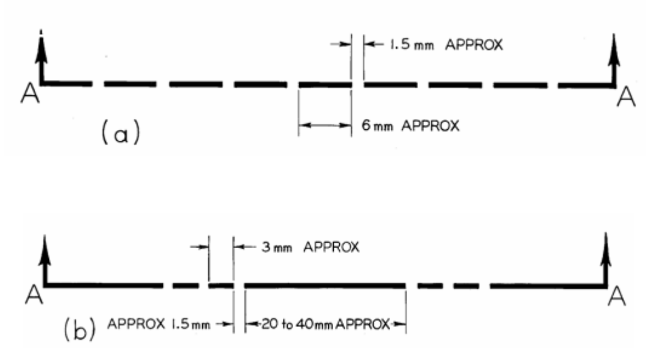


Figure 1.13: Cutting Plane Line

Cutting Plane in Sectional Views

- The cutting plane is indicated in a view adjacent to the sectional view. In this view, the cutting plane appears edgewise as a line, called the cutting plane line.

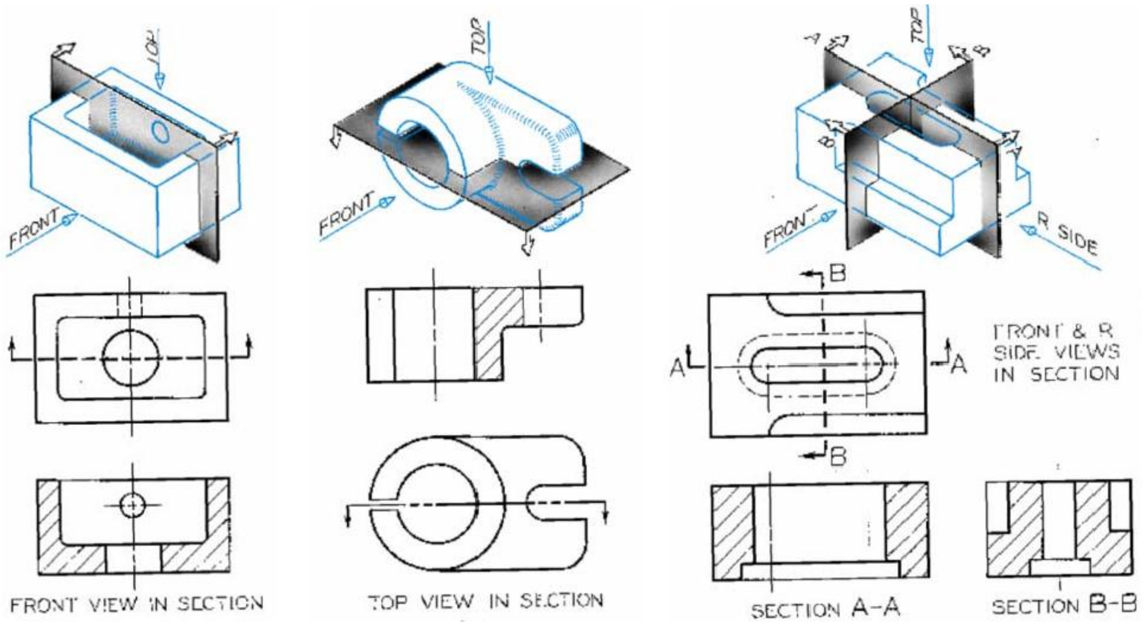


Figure 1.14: Cutting plane lines

Self-Check-1	Choice Test

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have 2 pts)

1. It is pertaining to machine parts or components and also presented through a number of orthographic views, so that the size and shape of the component is fully understood.

a. Assembly Drawing b. Working Drawing c. Machine Drawing d. all

2. ________ also referred to as working drawing, should furnish all the dimensions, limits and special finishing processes such as heat treatment, honing, lapping, surface finish, etc., to guide the craftsman on the shop floor in producing the component.

a. Sectional Drawing b. Production Drawing c. Assembly Drawing d. Machine Drawing

3. For effective exchange of ideas with others the engineer, must have proficiency in

a. Language, both written and oral.

b. Symbols associated with basic science

c. The graphic language

d. All of above

4. A drawing that shows the various parts of a machine in their correct working locations.

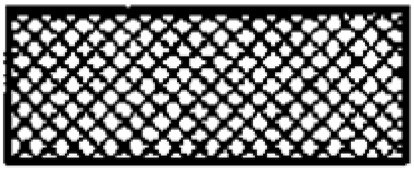
a. Detail Drawing

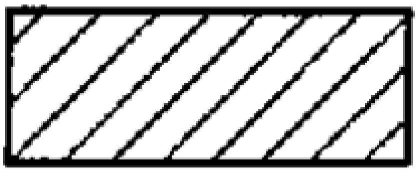
b. Assembly Drawing

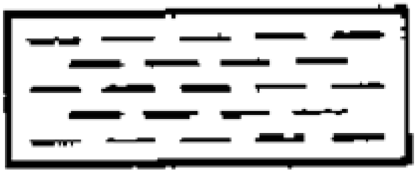
c. Sectional Drawing

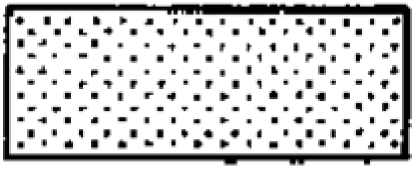
d. Production drawing

5. Which of shading Characterizing represents metal?

a. 

c. 

b. 

d. 

Note: **Satisfactory rating –5 points and above** **Unsatisfactory – below 5 points**

You can ask you teacher for the copy of the correct answers.

Name: ____________________________ Date: _________________

Answer Sheet

Score = ___________

Rating: ____________

Information Sheet-2	Sourcing of required information from workshop manuals, customer specifications,

2.1. Introduction to Blue Print Reading

In many ways, learning to read a drawing is the same as learning to read a language. Blueprint is the common name of the copies taken from an original drawing, usually drawn on a tracing paper. The copies may be obtained by way of reprographic processes.

For blueprint reading and understanding the drawing, one must have a thorough knowledge of the principles of drawing and orthographic projections. The knowledge of various manufacturing processes and the sequence of operations required to obtain the finished shape, intended by the designer, also helps in interpreting the drawings.

In this content, the examples chosen help providing guidelines to enable trainee to understand the shape and size of a component in the case of component drawings, and also its location, in the case of assembly drawings. While reading the drawings, the details such as shape, size, through dimensions, notes and material to be used, and additional notes to the workman on machining, surface finish, tolerances, etc., are to be noted carefully.

2.1.1. Examples

2.1.2. Rear Tool Post

Rear tool post is generally used on capstan lathes, mainly for parting-off operations. It is fixed on the cross-slide in the slots, provided at the rear side of the lathe. Study the drawing shown in Fig.1 and answer the following:

1. What is the overall size of the tool post?

— 102mm × 70mm × 62mm

2. How many bolts are provided for fixing the tool, and what is the size of each bolt?

—3, M10

3. What type of tool can be used with it?

— Parting tool

4. What is the maximum height of the tool holder?

— 25mm

5. How many screws are provided to locate the tool?

— 2

6. What is the purpose of the threaded hole marked ‘X’?

— For adjusting the tool height, by means of a screw

7. Explain the note—4 HOLES, M10.

—There are three tapped holes in the body to clamp the tool in position by screws

And the fourth tapped hole is at the bottom of the base. The size of the tap is 10mm.

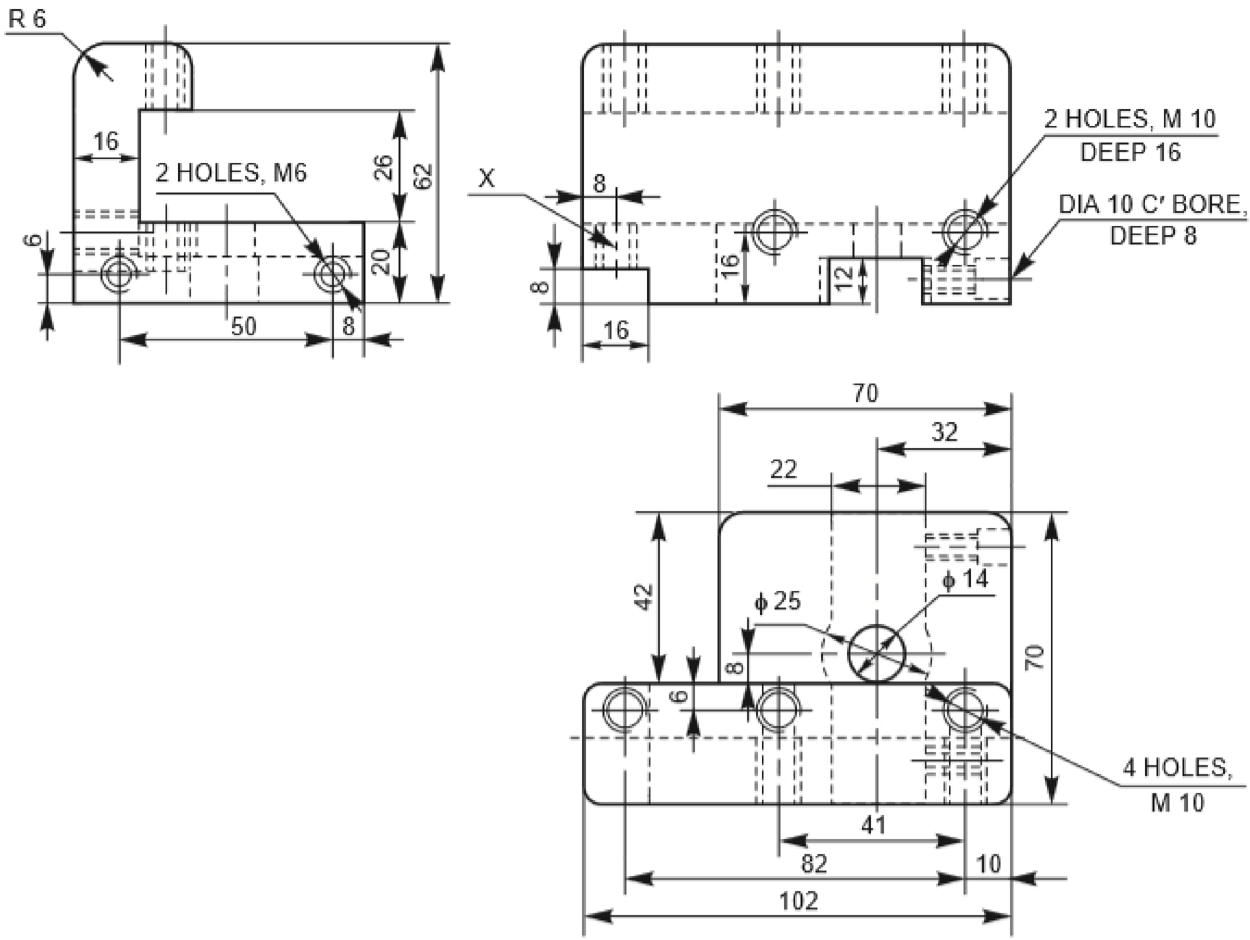


Figure1: Rear tool post

Self-Check-2	Multiple Choice

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have 3 pts)

1. __________ is the common name of the copies taken from an original drawing, usually drawn on a tracing paper. The copies may be obtained by way of reprographic processes.

   a. Drawing **b.** Blue print **c.** detail drawing d. all

2. For blueprint reading and understanding the drawing, one must have a thorough knowledge of.
   a. The principles of drawing
   b. Orthographic projections
   c. various manufacturing processes
   d. All of the above

***Note:* Satisfactory rating –3 points and above Unsatisfactory – below 3 points**

You can ask you teacher for the copy of the correct answers.

Name: ____________________________ Date: ________________

Answer Sheet

Score = ____________

Rating: ______________

Information Sheet-3	planning Scope of drawing

3.1. Introduction

The drawings prepared by any technical person must be clear, unmistakable in meaning and there should not be any scope for more than one interpretation, or else litigation may arise. In a number of dealings with contracts, the drawing is an official document and the success or failure of a structure depends on the clarity of details provided on the drawing. Thus, the drawings should not give any scope for misinterpretation even by accident. It would not have been possible to produce the machines/automobiles on a mass scale where a number of assemblies and sub-assemblies are involved, without clear, correct and accurate drawings. To achieve this, the technical person must gain a thorough knowledge of both the principles and conventional practice of drawing. If these are not achieved and or practiced, the drawings prepared by one may convey different meaning to others, causing unnecessary delays and expenses in production shops.

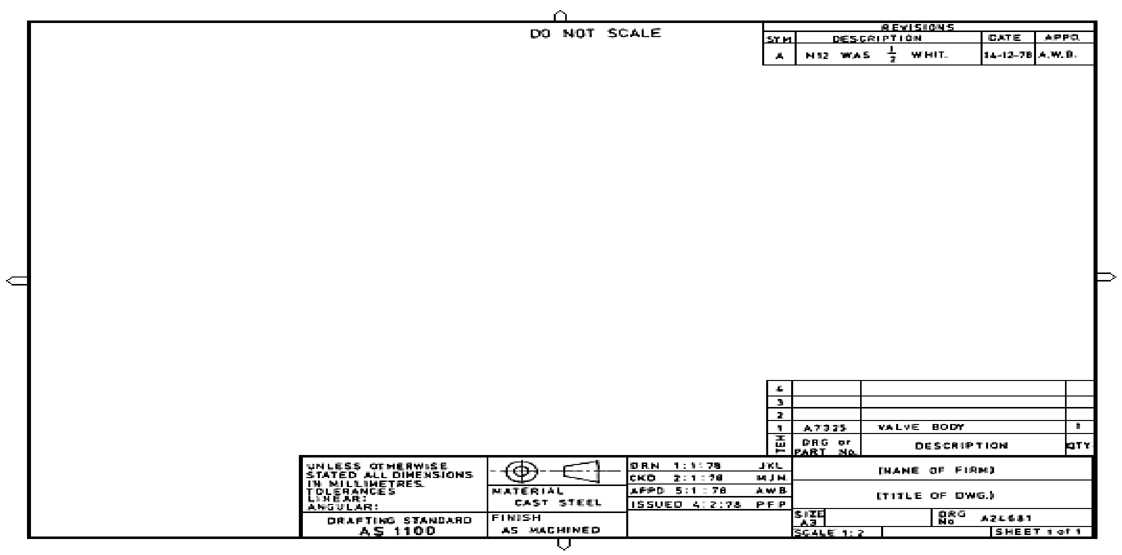
3.2. Items required for drawing

Drawing board
Drawing sheet (element of title block)
T- Square , Set Squares
Compass,
Divider
Scales
Protractor

French curves
Drawing pencils
Eraser
Drawing clip/pin/adhesive tape
Sharpener
Duster, etc

Drawing Sheet Layout

Standard layouts of drawing sheets are specified by the various standards organizations. This is the layout of a typical sheet, showing the drawing frame, the microfilm camera alignment marks, a typical title block, parts list and revision table:



Drawing Frames with No Filing Margin

Paper size	Border width(MM)		Dimensions of Drawing frame(MM)	
	Left &right	Top &bottom	Width	height
A0	28	20	1133	801
A1	20	14	801	566
A2	14	10	566	400
A3	10	7	4003	283

Title Block

The title block is normally placed in the bottom right of the drawing frame, and it should contain the following information:

- the name of the company or organization
- the title of the drawing
- the drawing number, which is generally a unique filing identifier
- the scale
- the angle of projection used, either first or third, generally shown symbolically
- the signature or initials of the draftsman, checker, approving officer, and issuing officer, with the respective dates
- Other information as required.

Material or Parts List

If the drawing contains a number of parts, or if it is an assembly drawing, a tabulated parts list is attached to the bottom right of the drawing frame, just above the title block.

The parts list should give the following information:

- the part number
- the part name
- the quantity required
- material specifications
- the drawing number of each individual part
- other applicable information

When the parts list is very large a separate drawing sheet may be used for the parts list alone.

Self-Check -3	Multiple Choice

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have 2.pts)

1. What is the scope a Drawing?
   a. Give information clear unmistakable in meaning
   b. Give clarity of details provided on the drawing
   c. Avoiding misinterpretation of drawing
   d. All of above
2. Which information is contained in Title block?
   a. the name of the company or organization
   b. the title of the drawing
   c. the drawing number,
   d. the scale
   e. the angle of projection used, either first or third
   f. All of above
3. Which information is contained in Part list?
   a. the part number
   b. the part name
   c. the quantity required
   d. material specifications
   e. the drawing number of each individual part
   f. All of the above

***Note:* Satisfactory rating –4 points and above** **Unsatisfactory – below 4 points**

You can ask you teacher for the copy of the correct answers.

Name: ___________________________ Date: ________________

Answer Sheet

Score = ___________
Rating: ____________

List of materials

1- BOOKS

2- Machine drawing, therd edition,DR.KL.Narayana, Dr. M.A. Veluswami

3- Text book of engineering drawing,K.VENKATA Reddy,second edition,BS.Pabilication

4. – KHURMI R S AND GUPTA J. K (1979). A Text Book of Machine Design. ISN 81-219-0501-x, Published by Scand and Company ltd

Mechanics

LEVEL III

Learning Guide#02

Unit of Competence: Perform Advanced Engineering Detail Drafting

Module Title: Performing Advanced Engineering Detail Drafting

Module Code: XXX

LG Code: XXX

TTLM Code: XXX

LO 2: Prepare assembly, lay-out and Detail drawing

Instruction Sheet: 2	Learning Guide #02

This learning guide is developed to provide you the necessary information regarding the following **content coverage** and topics:

- Determining drawing details and specifications
- Undertaking engineering calculations to determine all dimensions Including limits and fits ,geometric tolerance
- Inserting dimensions and geometric tolerances
- Using appropriate symbols for limits and fits, surface texture and geometric tolerances.
- Showing correct convention of parts based on ISO standard
- Producing drawing, auxiliary views, sections and assemblies in third angle projection.
- Producing all drawings in an acceptable ISO.
- Selecting Components, material and/or assemblies from manufacturing Catalogues.

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, **upon completion of this Learning Guide, you will be able to**:

- Determine drawing details and specifications
- Undertake engineering calculations to determine all dimensions Including limits and fits ,geometric tolerance
- Insert dimensions and geometric tolerances
- Use appropriate symbols for limits and fits, surface texture and geometric tolerances.
- Show correct convention of parts based on ISO standard
- Produce drawing, auxiliary views, sections and assemblies in third angle projection.
- Produce all drawings in an acceptable ISO.
- Select Components, material and/or assemblies from manufacturing Catalogues.

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below 3 to 6.
3. Read the information written in the information "Sheet 1, Sheet 2, Sheet 3 , Sheet 4 , sheet 5, sheet 6, sheet 7 and sheet 8"
4. Accomplish the "Self-check 1, Self-check 2, Self-check 3, Self-check 4, Sheet 5,Sheet 6, Sheet 7 and Sheet 8 " **in page -31, 60,70 and 82,89, 103 and 127** respectively
5. If you earned a satisfactory evaluation from the "Self-check" proceed to "Operation Sheet 1, Operation Sheet 2 and Operation Sheet 3, Operation 4, Operation 5 " **in page -132,133,134,135, 136,137 and 138.**
6. Do the "LAP test" **in page – 139** (if you are ready).

Information Sheet-2	Determining drawing details and specifications

2.1. Introduction to Production Drawing

A production drawing, also known as working drawing, supplies information and instructions for the manufacture or construction of machines or structures. A production drawing should provide all the dimensions, limits, special finishing processes, surface quality, etc.

The particulars of material, the number of components required for the assembly, etc., are given in the title block. The production drawing of a component should also indicate the sub-assembly or main assembly where it will be assembled. Since the working drawings may be sent to other companies to make or assemble the unit, the drawings should confirm with the standards followed in the country. For this reason, a production drawing becomes a legal document between the parties, in case of disputes in manufacturing. Working drawings may be classified into two groups: (i) detail or part drawings and (ii) assembly drawings.

2.2. Types of Production Drawing

2.2.1. Detail Drawing or Part Drawing

A detail or part drawing is nothing but a production or component drawing, furnishing complete information for the construction or manufacture of the part. This information may be classified as:

1. **Shape description** This refers to the selection of number of views to describe the shape of the part. The part may be drawn in either pictorial or orthographic projection; the latter being used more frequently. Sectional views, auxiliary views and enlarged detailed views may be added to the drawing in order to provide a clear image of the part.

2. **Size description** Size and location of the shape features are shown by proper dimensioning. The manufacturing process will influence the selection of some dimensions, such as datum feature, tolerances, etc.

3. **Specifications**: This includes special notes, material, heat treatment, finish, general tolerances and number required. All this information is mostly located near the title block.

4. Additional information such as drawing number, scale, method of projection, date, names of the parts, the daughter's name, etc., come under additional information which is included in the title block. Since the craftsman will ordinarily make one component at a time, it is advisable to prepare the production drawing of each component, regardless of its size, on a separate sheet. Figures1 and 2 show the detailed drawings of a template jig and gear.

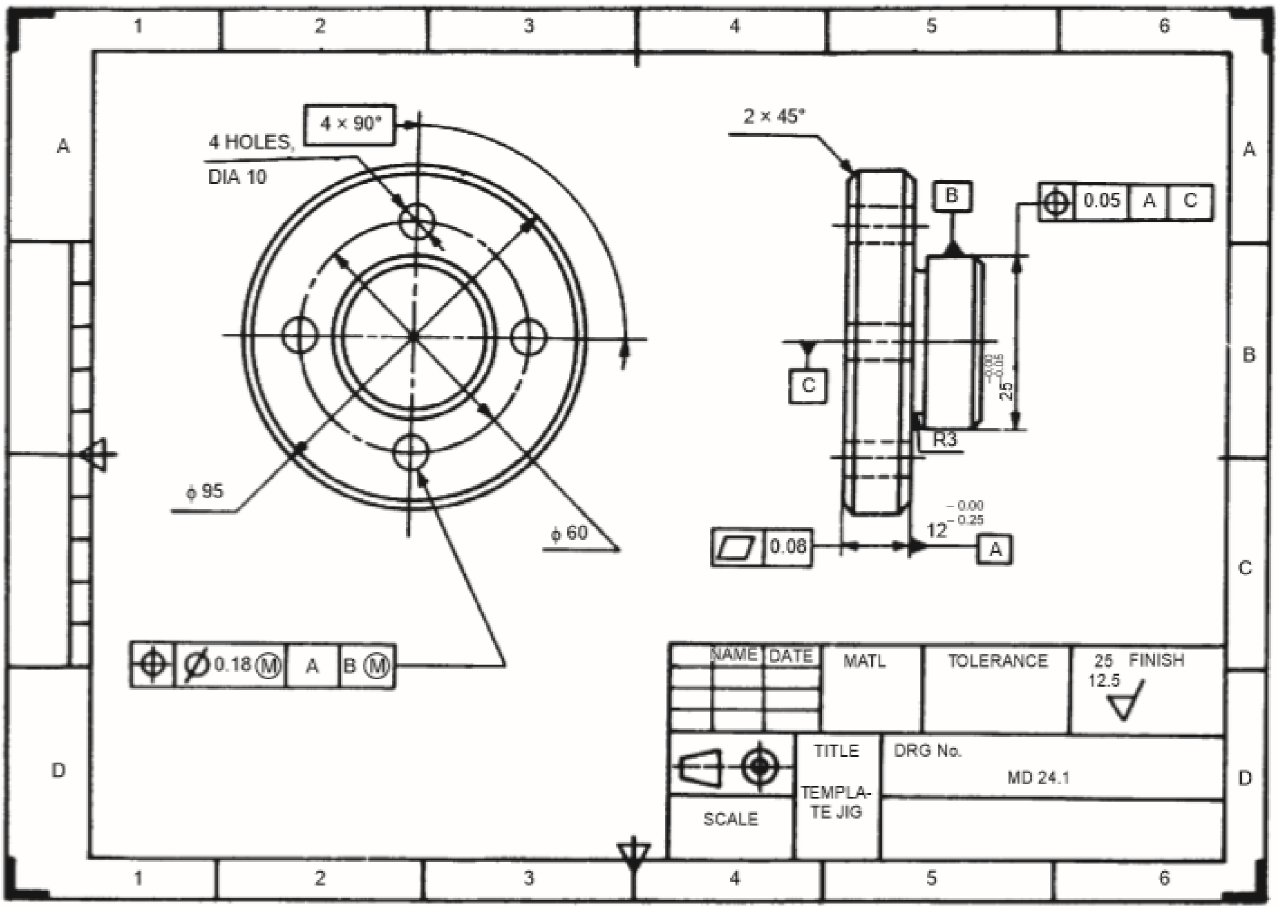


Figure 1.1: Detail drawing of template jig

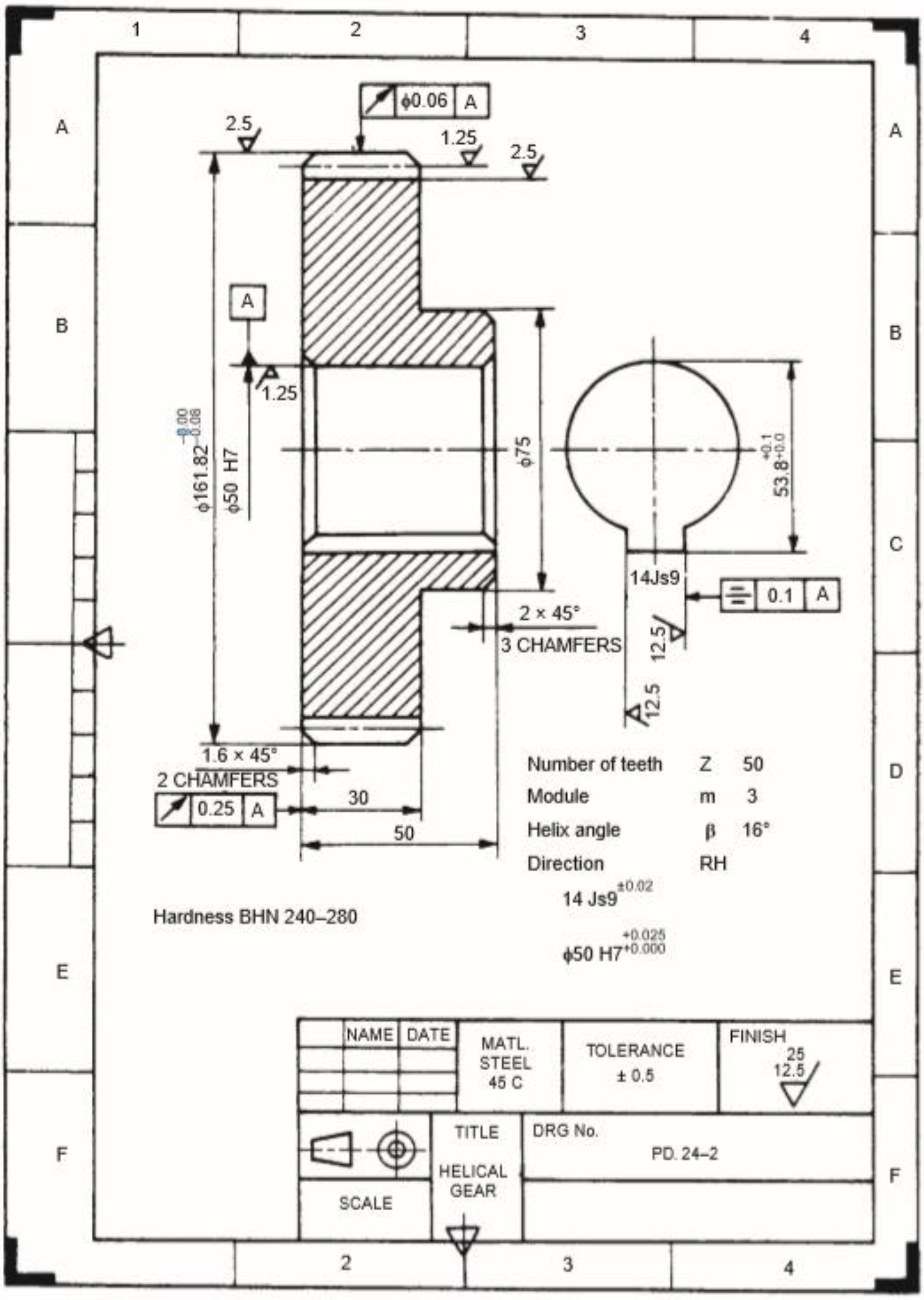


Figure 1.2: Detail Drawing of Gear

Self-Check -1	Multiple Choice

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have 2 pts)

1. ______ is nothing but a production or component drawing, furnishing complete information for the construction or manufacture of the part.
   a. detail or part drawing   b. Assembly drawing   c. Sub assembly drawing d. all
2. Which information included under detail or part drawing?
   a. Shape description   b. Size description   c. Specifications
   d. Additional Information such as drawing number, scale   e. All of the above
3. ____________ refers to the selection of number of views to describe the shape of the part.
   a. Shape description   b. Size description   c. Specifications   d. all

Note: **Satisfactory rating –4 points and above   Unsatisfactory – below 4 points**

You can ask you teacher for the copy of the correct answers.

Name: ___________________________   Date: ________________

Answer Sheet

Score = ___________

Rating: ____________

3.3. Introduction to Limits, geometric tolerance and Fits

The manufacture of interchangeable parts requires precision. Precision is the degree of accuracy to ensure the functioning of a part as intended. However, experience shows that it is impossible to make parts economically to the exact dimensions. This may be due to,

(i) Inaccuracies of machines and tools,

(ii) Inaccuracies in setting the work to the tool, and

(iii) Error in measurement, etc.

The workman, therefore, has to be given some allowable margin so that he can produce a part, the dimensions of which will lie between two acceptable limits, a maximum and a minimum. The system in which a variation is accepted is called the limit system and the allowable deviations are called tolerances. The relationships between the mating parts are called fits. The study of limits, tolerances and fits is a must for technologists involved in production. The same must be reflected on production drawing, for guiding the craftsman on the shop floor.

3.4. Limits System

Following are some of the terms used in the limit system

3.4.1. Tolerance

The permissible variation of a size is called tolerance. It is the difference between the maximum and minimum permissible limits of the given size. If the variation is provided on one side of the basic size, it is termed as unilateral tolerance. Similarly, if the variation is provided on both sides of the basic size, it is known as bilateral tolerance.

International tolerance grade (IT) A set of tolerances that varies according to the basic size and provides a uniform level of accuracy within the grade. For example, the dimension 50H8 for a close-running fit, the IT grade is indicated by the numeral 8. (The letter H indicates the tolerance is on the hole for the 50 mm dimension). In all, there are 18 IT grades – IT01, IT0, and IT1 to IT16. See Figures 16 and 17 for IT grades related to machining processes and the practical use of the IT grades.

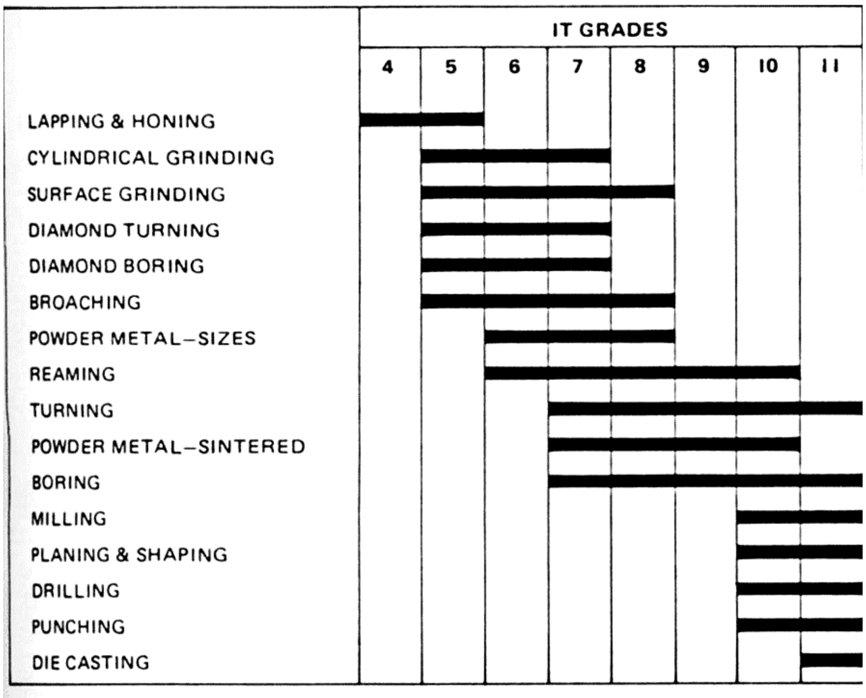


Figure 1. International Tolerance Grades Related to Machining Processes

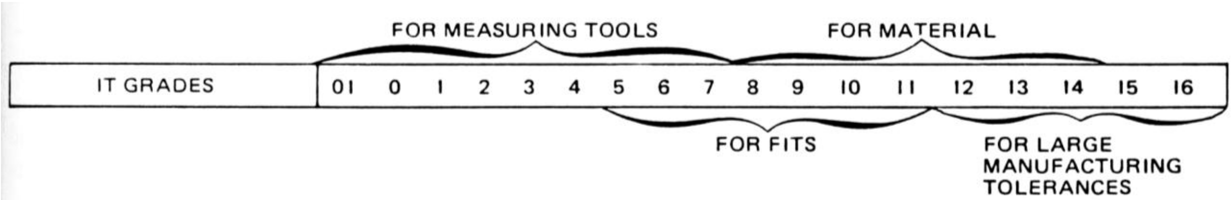


Figure 2. Practical Use of International Tolerance Grades

3.4.2. Limits

The two extreme permissible sizes between which the actual size is contained are called limits. The maximum size is called the upper limit and the minimum size is called the lower limit.

3.4.3. Deviation

It is the algebraic difference between a size (actual, maximum, etc.) and the corresponding basic size.

3.4.4. *Fundamental deviation*

The deviation closet to the basic size (This compares with the minimum allowance in the decimal-inch system.)

3.4.5. Upper Deviation

It is the algebraic difference between the maximum limit of the size and the corresponding

basic size.

Lower Deviation

It is the algebraic difference between the minimum limit of the size and the corresponding basic size.

3.4.6. Allowance

It is the dimensional difference between the maximum material limits of the mating parts, intentionally provided to obtain the desired class of fit. If the allowance is positive, it will result in minimum clearance between the mating parts and if the allowance is negative, it will result in maximum interference.

3.4.7. Basic Size

It is determined solely from design calculations. The theoretical size from which limits of size are derived by the application of allowances and tolerances. It is the size from which limits are determined for the size, shape, or location of the feature.

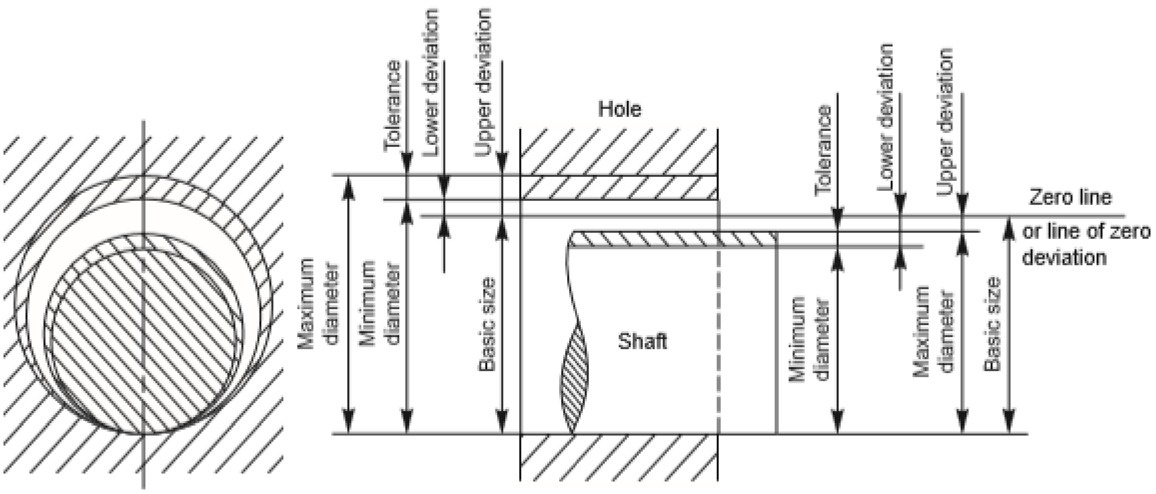


Figure 3. Diagram illustrating basic size deviations and tolerances

3.4.8. **Actual size*:*** the measured size of the finished part.

3.5. Tolerances

Great care and judgment must be exercised in deciding the tolerances which may be applied on various dimensions of a component. If tolerances are to be minimum, that is, if the accuracy requirements are severe, the cost of production increases. In fact, the actual specified tolerances dictate the method of manufacture. Hence, maximum possible tolerances must be recommended wherever possible.

3.5.1. Tolerance zone

The tolerances and its position in relation to basic size. It is established by a combination of the fundamental deviation indicated by a letter and the IT grade number. In the dimension 50H8, for the close-running fit, the H8 specifies the tolerance zone.

N.B: The following examples are taken from machine drawing book, written by DR.K.L. Narayana, Refers the tables 15.1, 15.2, 15.3, 15.4 and 15.6 to understand examples and try to see Appendix's.

Example 1 *Calculate the fundamental tolerance for a shaft of 100 mm and grade 7.*

The shaft size, 100 lies in the basic step, 80 to 120 mm and the geometrical mean is

$$D = \sqrt{80 \times 120} = 98 \text{ mm}$$

The tolerance unit, $i = 0.45\sqrt[3]{98} + 0.001 \times 98 = 2.172$ microns

For grade 7, as per the Table 15.1A, the value of tolerance is,

$$16i = 16 \times 2.172 = 35 \text{ microns}$$

Example 2 *Calculate the fundamental deviations for the shaft sizes given below :*

(a) 30 e8 (b) 50 g6 (c) 40 m6.

From Table 15.4, the deviations for shafts are obtained :

(*a*) The upper deviation *es* for the shaft *e*

$$= -11\, D^{0.41}$$

The value for $D = \sqrt{18 \times 30} = 23.24$ mm.

Hence, *es* = – 40 microns (tallies with the value in Table 15.2).

(*b*) The upper deviation *es* for the shaft *g*

$$= -2.5\, D^{0.34}$$

The value for $D = \sqrt{30 \times 50} = 38.73$ mm.

Hence, *es* = – 9 microns (tallies with the value in Table 15.2)

(*c*) The lower deviation *ei* for the shaft *m*

$$= +(\text{IT } 7 - \text{IT } 6)$$

From the Table 15.1, the size 40 is in the range 30 and 50 and hence the mean diameter *D*, is 38.73 mm

Tolerance unit $i = 0.45\sqrt[3]{D} + 0.001\,D$

$$= 1.58 \text{ microns}$$

The fundamental tolerance for grade 7, from the Table 15.1 is 16*i*, *i.e.*, 25 microns.

The fundamental tolerance for grade 6 is 10*i* or 16 microns.

Hence, *ei* = 25 (IT 7) – 16 (IT 6) = + 9 microns (tallies with the value in Table 15.2).

Example 3 *Calculate the fundamental deviations for the hole sizes given below :*

(a) 40 D9 (b) 65 F8.

From Table 15.4, the deviations for holes also can be obtained (article 15.3.2.2).

(*a*) The lower deviation EI for the hole *D* is given by

$$EI = + 16\, D^{0.44}, \text{ where } D = \sqrt{30 \times 50} = 38.73 \text{ mm}$$

Thus, EI = 80 microns (tallies with the value in Table 15.3).

(*b*) Lower deviation EI for the hole F

$$= + 5.5\, D^{0.41}, \text{ where } D = \sqrt{50 \times 80}$$

Hence, EI = 30 microns (tallies with the value in Table 15.3).

Example 4 *A journal bearing consists of a bronze bush of diameter 100 mm fitted into a housing and a steel shaft of 50 mm diameter, running in the bush, with oil as lubricant. Determine the working dimensions of (a) bore of the housing, (b) bush and (c) shaft. Calculate the maximum and minimum interference or clearance.*

Step 1: Select the nature of assembly or fit based on the function. Referring to Table 15.6, the fits to be employed are selected as below:

(*a*) for the bush and housing, H7/p6 (interference fit),

(*b*) for the shaft and bush, H7/f7 (normal running fit).

Step 2: Obtain the tolerances on the linear dimensions of the parts. From Table 15.1, the fundamental tolerances (IT) for different grades, based on the size are :

(*a*) for dia. 100 and grade 6 = 22 microns,

(*b*) for dia. 100 and grade 7 = 35 microns,

(*c*) for dia. 50 and grade 7 = 25 microns.

Step 3: Obtain the fundamental deviations based on the type of hole/shaft and thus the respective sizes. From Table 15.2,

(*a*) for a hole of type H (housing)

$$\text{lower deviation, EI} = 0.000$$

$$\text{upper deviation, ES} = \text{EI} + \text{IT} = 0.035 \text{ mm}$$

Hence, dimension of the housing bore = $100^{+0.035}_{+0.000}$.

(*b*) for a shaft of type *p* (bush),

$$\text{lower deviation, } ei = +\,0.037 \text{ (Table 15.2)}$$

$$\text{upper deviation, } es = ei + \text{IT} = 0.037 + 0.022 = 0.059 \text{ mm}$$

Hence, the outside size of the bush = $100^{+0.059}_{+0.037}$.

(*c*) for a hole of type H (bush),

$$\text{lower deviation, EI} = 0.000$$

$$\text{upper deviation, ES} = \text{EI} + \text{IT} = 0.025 \text{ mm}$$

Hence, the bore of the bush = $50^{+0.025}_{+0.000}$.

(*d*) for a shaft of type *f*,

$$\text{upper deviations, } es = -\,0.025 \text{ (Table 15.2)}$$

$$\text{lower deviation, } ei = es - \text{IT} = -\,0.025 - 0.025 = -\,0.05 \text{ mm}$$

Hence, shaft dimension is = $50^{-0.025}_{-0.050}$

Step 4: Calculate the interference/clearance

(*a*) between the bush and housing :

$$\text{Maximum interference} = 100.00 - 100.059 = -\,0.059 \text{ mm}$$

$$\text{Minimum interference} = 100.035 - 100.037 = -\,0.002 \text{ mm}$$

(*b*) between the bush and shaft :

$$\text{Maximum clearance} = 50.025 - 49.050 = +\,0.075 \text{ mm}$$

$$\text{Minimum clearance} = 50.000 - 49.075 = +\,0.025 \text{ mm}$$

Mating Parts

Mating parts are parts that fit together within a prescribed degree of accuracy (Figure 5). The upper piece is dimensioned with two measurements that indicate the upper and lower limits of the size. The notch is slightly larger, allowing the parts to be assembled with a clearance fit.

An example of mating cylindrical parts is shown in Figure 6. Part B of the figure illustrates the meaning of the tolerance dimensions. The size of the shaft can vary in diameter from 1.500” (its maximum size) to 1.498” (its minimum size). The difference between these limits on a single part is tolerance, 0.002” in this case. The dimensions of the hole in Part A are given with limits 0f 1.503” and 1.505”, for a tolerance of 0.002” (the difference between the limits as illustrated in Part B).

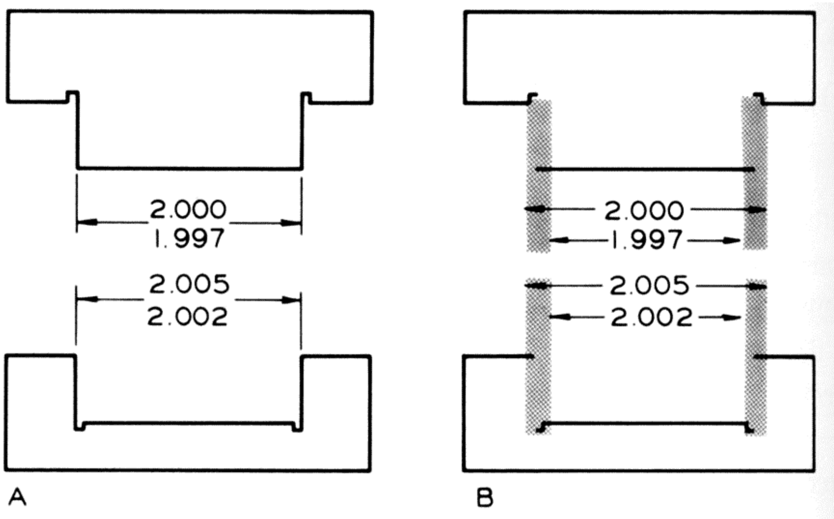


Figure 5**:** Mating Part

Each of these mating parts has a tolerance of 0.003” (variation in size). The allowance between the assembled parts (tightest fit) is 0.002”.

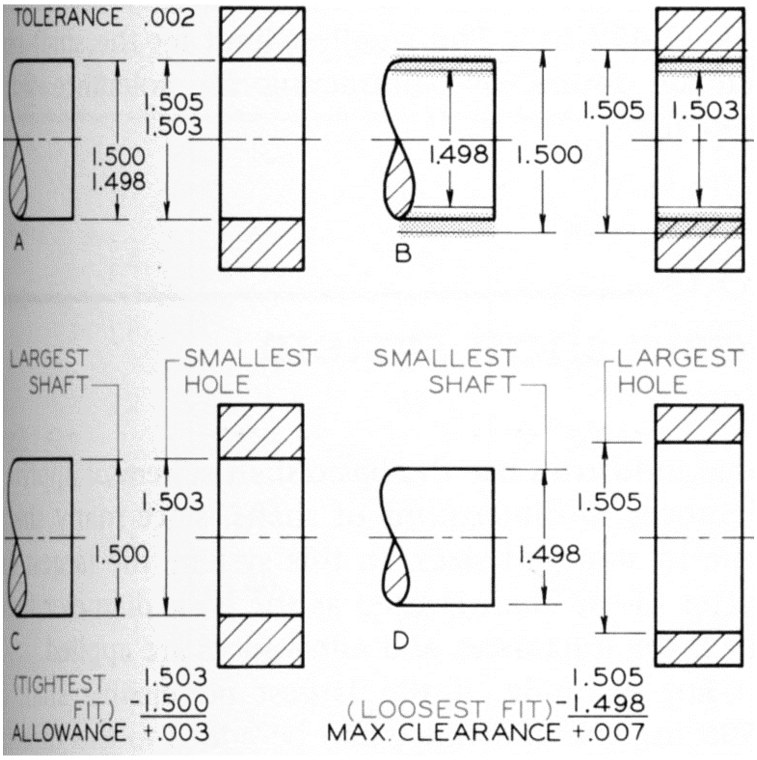


Figure 6. The allowance (tightest fit) between these assembled parts is +0.003”. The maximum clearance is 0.007”.

PREFERRED SIZES

The preferred basic sizes for computing tolerances are given in Table 1. Basic diameters should be selected from the first choice column since these are readily available stock sizes for round, square, and hexagonal products.

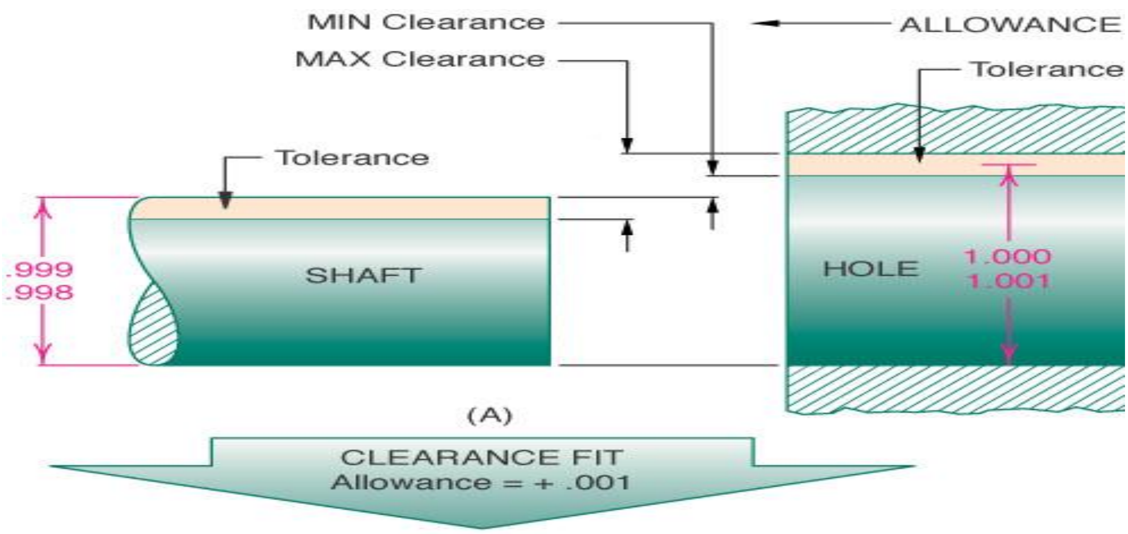
Basic Size, mm		Basic Size, mm		Basic Size, mm	
First Choice	Second Choice	First Choice	Second Choice	First Choice	Second Choice
1		10		100	
	1.1		11		110
1.2		12		120	
	1.4		14		140
1.6		16		160	
	1.8		18		180
2		20		200	
	2.2		22		220
2.5		25		250	
	2.8		28		280
3		30		300	
	3.5		35		350
4		40		400	
	4.5		45		450
5		50		500	
	5.5		55		550
6		60		600	
	7		70		700
8		80		800	
	9		90		900
				1000	

Table1. Preferred Sizes

FITS BETWEEN MATING PARTS

“Fit is the general term used to signify the range of tightness or looseness that may result from the application of a specific combination of allowances and tolerances in mating parts” [ANSI Y14.5M-1982 (R1988)]. There are four general types of fits between parts.

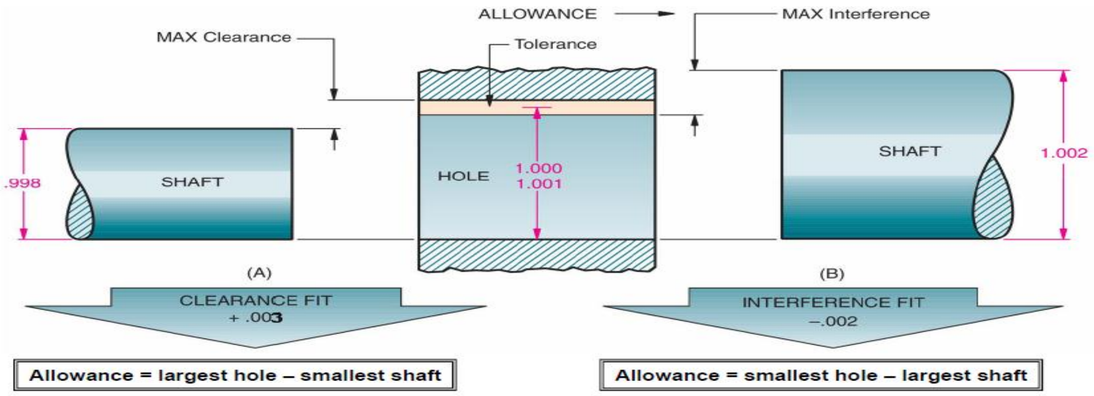
Clearance fit occurs when two tolerance mating parts will always leave a space or clearance when assembled. The largest that shaft (A) can be manufactured is .999 and the smallest the hole can be is 1.000. The shaft will always be smallest than the hole.



Allowance = smallest hole – largest shaft

Figure 7. Clearance fit

Interference fit occurs when two toleranced mating parts will always interfere when assembled. This fit type would be necessary to stretch the hole or shrink the shaft or to use force to press the shaft into the hole. For example this fit type can be used to fasten two parts together without the use of mechanical fasteners or adhesive.



Allowance = smallest hole – largest shaft

Figure 8. Interference fit

Transition fit occurs when two toleranced mating parts will sometimes be an interference fit and sometimes be a clearance fit when assembled.

Allowance = largest hole – smallest shaft

Allowance = smallest hole – largest shaft

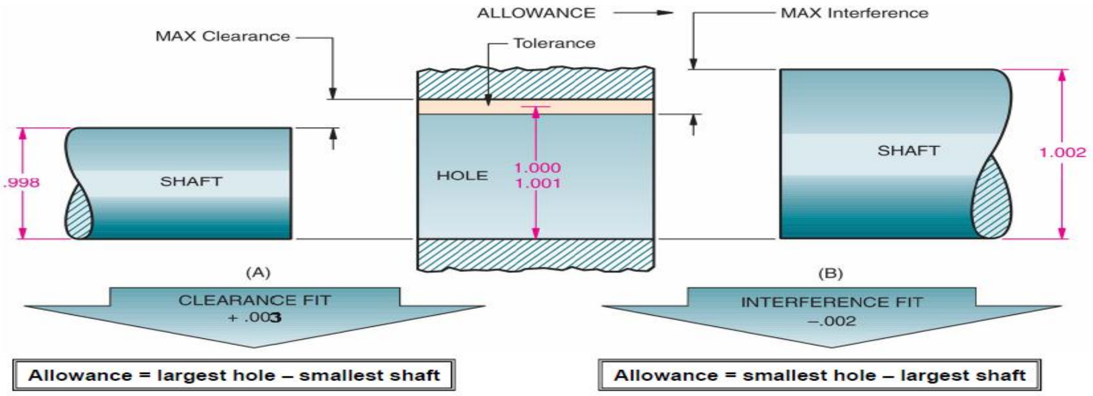


Figure 9. Transition Fit

Line fit In which limits of size are so specified that a clearance or surface contact may result when mating parts are assembled**.**

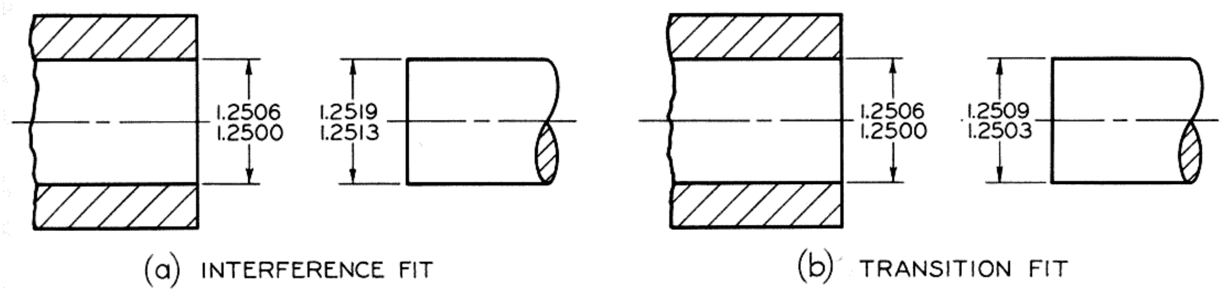


Figure 10: Fits between parts

BASIC HOLE SYSTEM

Standard reamers, broaches, and other standard tools are often used to produce holes, and standard plug gages are used to check the actual sizes. On the other hand, shafting can easily be machined to any size desired. Therefore, toleranced dimensions are commonly figured on the so-called *basic hole system*. In this system, *the minimum hole is taken as the basic size*, an allowance is assigned, and tolerances are applied on both sides of, and away from, this allowance.

In Figure 11(a) the minimum size of the hole, 0.500', is taken as the basic size. An allowance of 0.002" is decided on and subtracted from the basic hole size, giving the maximum shaft, 0.498". Tolerances of 0.002" and 0.003", respectively, are applied to the hole and shaft to obtain the maximum hole of 0.502" and the minimum shaft of 0.495". Thus, the minimum clearance between the parts becomes 0.500" – 0.498" = 0.002" (smallest hole minus largest shaft), and the maximum clearance is 0.502" – 0.495" = 0.007" (largest hole minus smallest shaft).

In the case of an interference fit, the maximum shaft size would be found by *adding the desired allowance* (maximum interference) to the basic hole size. In Figure 10(a) the basic size is 1.2500”. The maximum interference decided on was 0.0019”, which added to the basic size gives 1.2519”, the largest shaft size.

The basic hole size can be changed to the basic shaft size by subtracting the allowance for a clearance fit, or adding it for an interference fit. The result is the largest shaft size, which is the new basic size.

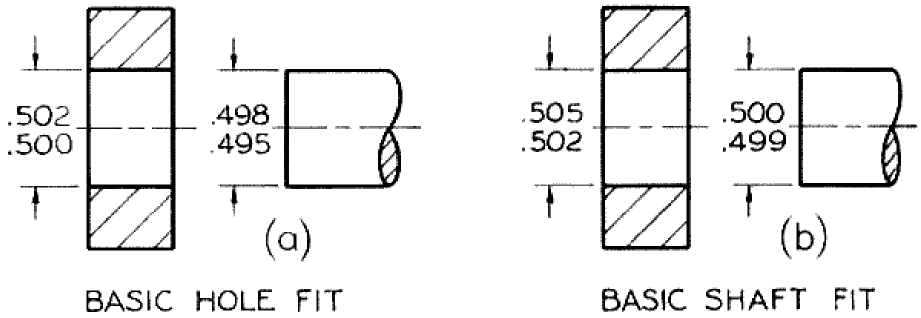


Figure 11. Basic Hole and Basic Shaft Systems

BASIC SHAFT SYSTEM

In some branches of industry, such as textile machinery manufacturing, in which use is made of a great deal of cold-finished shafting, the basic shaft system is often used. The system should be used only when there is a reason for it. For example, it is advantageous when several parts having different fits, but one nominal size, are required on a single shaft. In this system, the maximum shaft is taken as the basic size, an allowance for each mating part is assigned, and tolerances are applied on both sides of, and away from, this allowance.

In Figure11 (b) the maximum size of the shaft, 0.500”, is taken as the basic size. An allowance of 0.002” is decided on and added to the basic shaft size, giving the minimum hole, 0.502”. Tolerances of 0.003” and 0.001”, respectively, are applied to the hole and shaft to obtain the maximum hole, 0.505”, and the minimum shaft, 0.499”. Thus, the minimum clearance between the parts is 0.502” – 0.500” = 0.002” (smallest hole minus largest shaft), and the maximum clearance is 0.505” – 0.499” = 0.006” (largest hole minus smallest shaft).

In the case of an interference fit, the minimum hole size would be found by subtracting the desired allowance from the basic shaft size.

The basic shaft size may be changed to the basic hole size by adding the allowance for a clearance fit or by subtracting it for an interference fit. The result is the smallest hole size, which is the new basic size.

PREFERRED FITS

The symbols for either the hole-basis or shaft-basis preferred fits (clearance, transition, and interference) are given in Table 2. Fits should be selected from this table for mating parts were possible.

The values corresponding to the fits are found in Appendixes 7-10. Although second and third choice basic size diameters are possible, they must be calculated from the tables not included in this text. For the generally preferred hole-basis system, note that the ISO symbols ranges from H11/c11 (loose running) to H7/u6 (force fit.) For the shaft-basis system, the preferred symbols ranges from C11/h11 (loose fit) to U7/h6 (force fit.),

Assume that it is desired to use the symbols to specify the dimensions for a free-running fit (hole basis) for a proposed diameter of 48 mm. Since 48 mm is not listed as a preferred size in Table 1, the design is altered to use the acceptable 50 mm diameter. From the preferred fits description in Table 2, the free-running fit (hole basis) is H9/d9. To determine the upper and lower deviation limits of the hole as given in the preferred hole-basis table, Appendix 7, follow across from the basic size of 50 to H9 under "Free running." The limits for the hole are 50.000 and 50.062 mm. Then, the upper and lower limits of deviation for the shaft are found in the d9 column under "Free running." They are 49.920 and 49.858 mm, respectively. Limits for other fits are established in a similar manner.

The limits for the shaft basis dimensioning are determined similarly from the preferred shaft basis table in Appendix 9. See Figures 12 and 13 for acceptable methods of specifying tolerances by symbols and drawings. A single note for the mating parts (free-running fit, hole basis) would be ∅50 H9/d9, Figure 20.

Fit type	ISO Symbol: Hole Basis	ISO Symbol: Shaft[a] Basis	Description
Clearance Fits	H11/c11	C11/h11	***Loose running*** fit for wide commercial tolerances or allowances on external members.
Clearance Fits	H9/d9	D9/h9	***Free running*** fit not for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures.
Clearance Fits	H8/f7	F8/h7	***Close running*** fit for running on accurate machines and for accurate location at moderate speeds and journal pressures.
Clearance Fits	H7/g6	G7/h6	***Sliding*** fit not intended to run freely, but to move and turn freely and locate accurately.
Transition Fits	H7/h6	H7/h6	***Locational clearance*** fit provides snug fit for locating stationary parts; but can be freely assembled and disassembled.
Transition Fits	H7/k6	K7/h6	***Locational transition*** fit for accurate location, a compromise between clearance and interference.
Transition Fits	H7/n6	N7/h6	***Locational transition*** fit for more accurate location where greater interference is permissible.
Interference Fits	H7/p6	P7/h6	***Locational interference*** fit for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements.
Interference Fits	H7/s6	S7/h6	***Medium drive*** fit for ordinary steel parts or shrink fits on light sections, the tightest fit usable with cast iron.
Interference Fits	H7/u6	U7/h6	***Force*** fit suitable for parts which can be highly stressed or for shrink fits where the heavy pressing forces required are impractical.

↑ More clearance — More interference ↓

Table 2. Preferred Fits

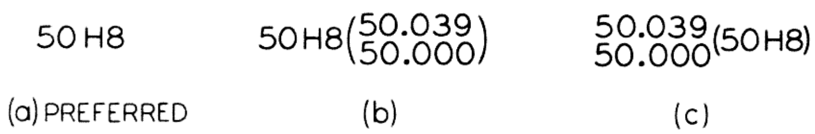


Figure12. Acceptable Methods of Giving Tolerance Symbols

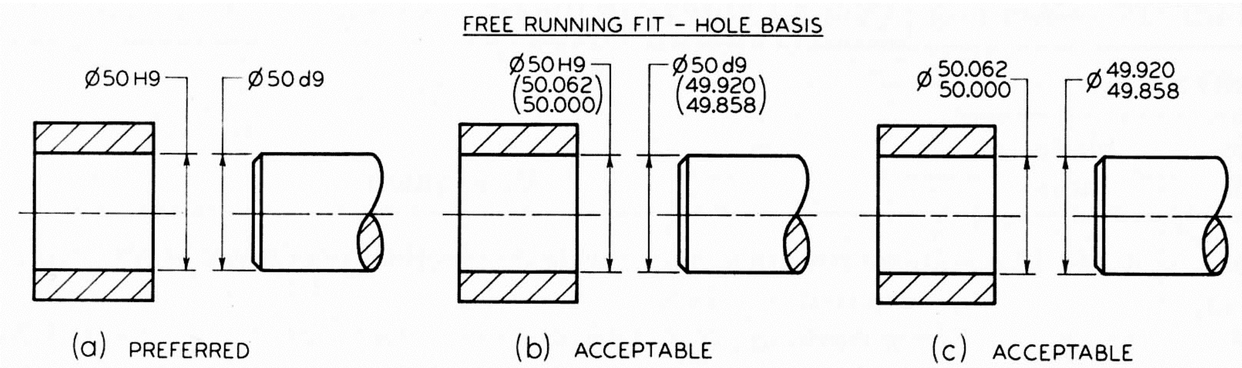


Figure13. Methods of Specifying Tolerances for Mating Parts

PREFERRED FITS-HOLE BASIS SYTEM Figure 14 illustrates the symbols used to show the combinations of fits that are possible when using the hole basis system. There is a clearance between the two parts at A, a transition fit at B, and an interference fit at C.

PREFERRED FITS-SHAFT BASIS SYSTEM Figure 15 illustrates the preferred fits based on the shaft basis system, where the largest shaft size is the basic diameter. Varying the size of the holes causes the variation in the fit between the parts. This results in a range of fits from a clearance fit of C11/h11 to an interference fit of U&/h6.

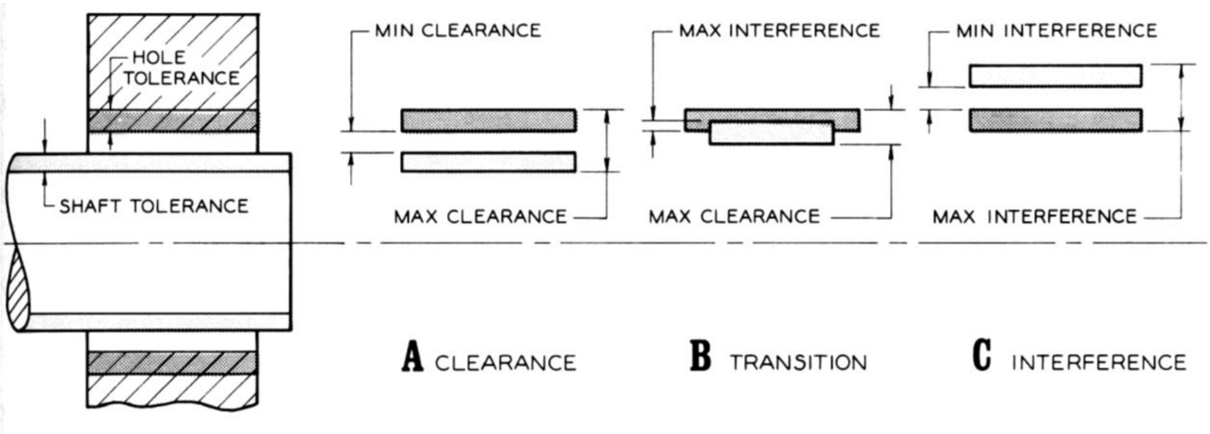


Figure 14. Types of fit (A) A clearance fit; (B) a transition fit where there can be interference or a clearance; and (C) an interference fit, where the parts must be forced together.

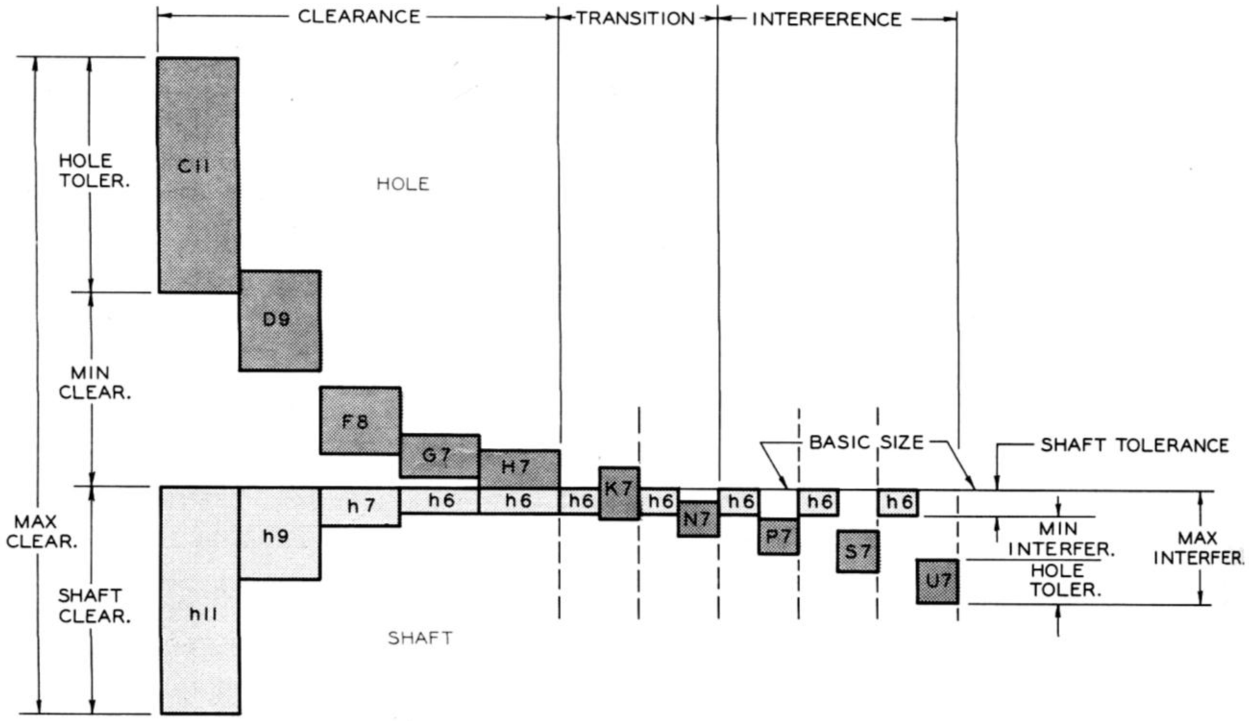


Figure15. The preferred fits for a shaft basis system. These fits correspond to those given in Table 2.

2.2. GEOMETRIC TOLERANCING

Geometric tolerances state the maximum allowable variations of a form or its position from the perfect geometry implied on a drawing. The term “geometric” refers to various forms, such as a plane, a cylinder, a cone, a square, a hexagon. Theoretically these are perfect forms, but, because it is impossible to produce perfect forms, it may be necessary to specify the amount of variation permitted. These tolerances specify either the diameter or the width of a tolerance within which a surface or the axis of a cylinder or a hole must be if the part is to meet the required accuracy for proper function and fit. When tolerances of form are not given on a drawing, it is customary to assume that, regardless of form variations, the part will fit and function satisfactorily.

Tolerances of form and position or location control such characteristics as straightness, flatness, parallelism, perpendicularity (squareness), concentricity, roundness, angular displacement, and so on.

Methods of indicating geometric tolerances by means of *geometric characteristic symbols*, as recommended by ANSI, rather than by traditional notes, are discussed and illustrated subsequently.

Characteristics to be toleranced		*Symbols*
Form of single features	Straightness	⏤
	Flatness	⏥
	Circularity (roundness)	○
	Cylindricity	⌭
	Profile of any line	⌒
	Profile of any surface	⌓
Orientation of related features	Parallelism	//
	Perpendicularity (squareness)	⊥
	Angularity	∠
Position of related features	Position	⌖
	Concentricity and coaxiality	◎
	Symmetry	⌯
	Run-out	↗

Table 3. Geometric Characteristics and Modifying Symbols

FORM TOLERANCES FOR SINGLE FEATURES

STRAIGHTNESS TOLERANCE A straightness tolerance specifies a tolerance zone within which an axis or all points of the considered element must lie, Figure 14. Straightness is a condition where an element of a surface or an axis is a straight line.

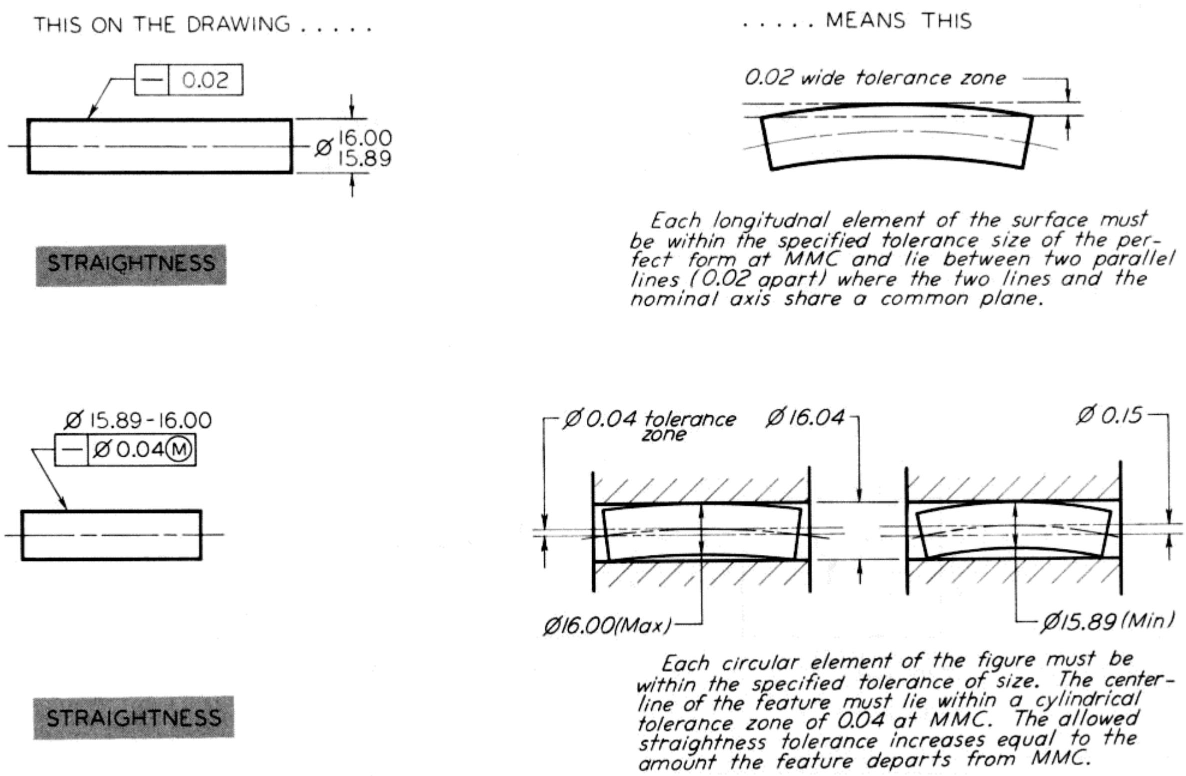


Figure 16. Specifying Flatness

ROUNDNESS (CIRCULARITY) TOLERANCE

A roundness tolerance specifies a tolerance zone bounded by two concentric circles within which each circular element of the surface must lie, Figure 38. Roundness is a condition of a surface of revolution where, for a cone or cylinder, all points of the surface intersected by any plane perpendicular to a common axis are equidistant from that axis. For a sphere, all points of the surface intersected by any plane passing through a common center are equidistant from that center.

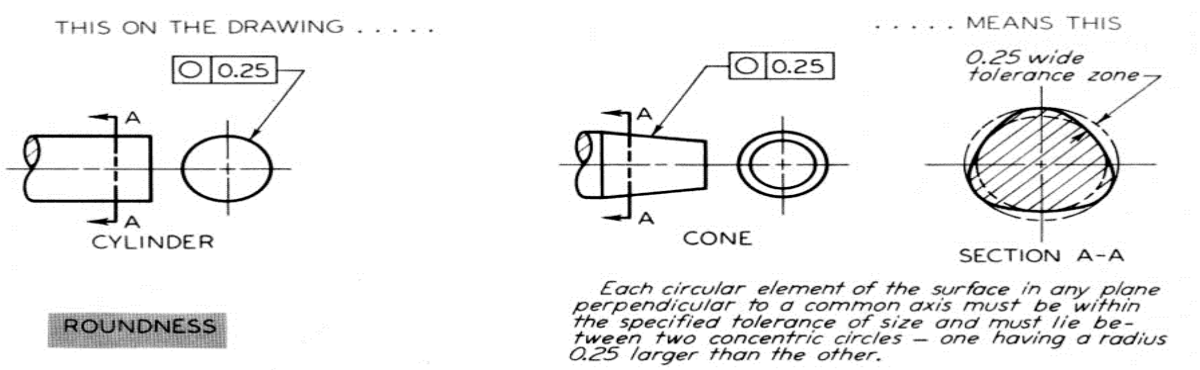


Figure 17. Specifying Roundness for a Cylinder or Cone

CYLINDRICITY TOLERANCE

A cylindricity tolerance specifies a tolerance zone bounded by two concentric cylinders within which the surface must lie, Figure 18. This tolerance applies to both circular and longitudinal elements of the entire surface. Cylindricity is the condition of a surface of revolution in which all points of the surface are equidistant from a common axis. When no tolerance of form is given, many possible shapes may exist within a tolerance zone, as illustrated in Figure 19

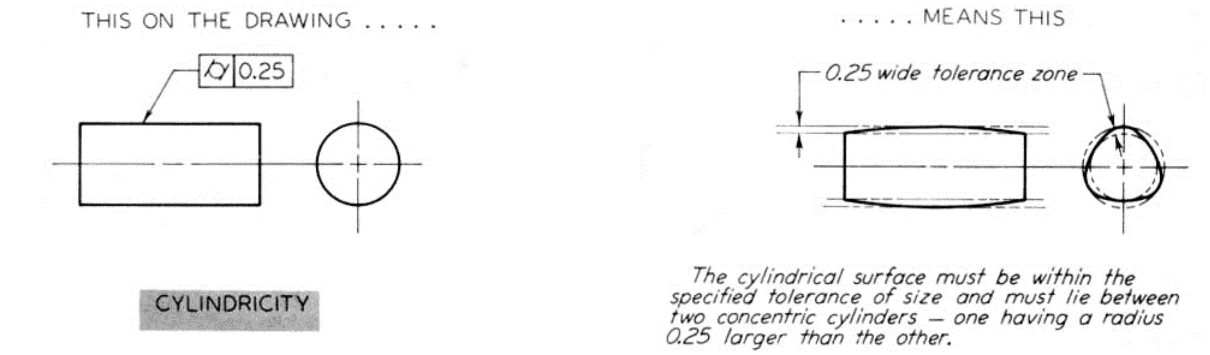


Figure18. Specifying Cylindricity

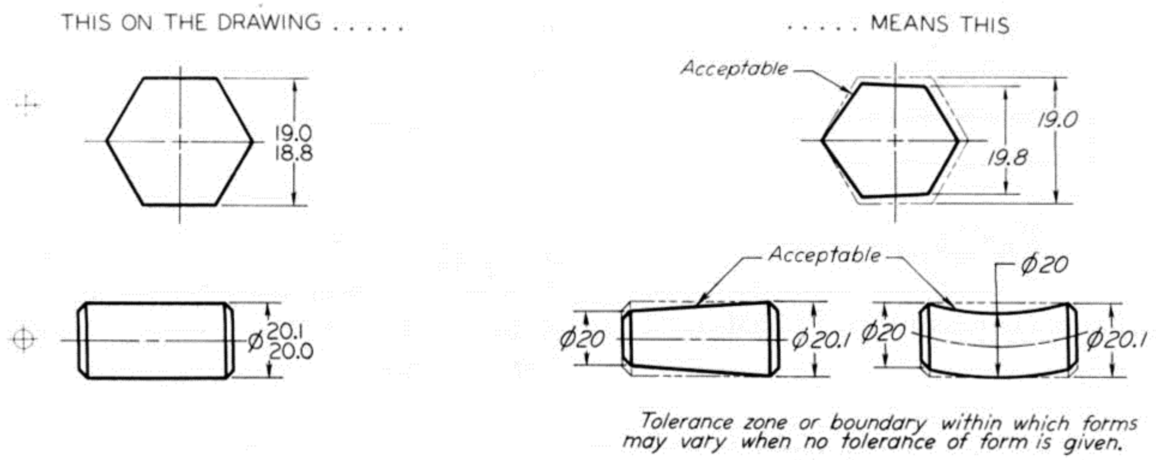


Figure 19. Acceptable Variations of Form – No Specified Tolerance of Form

PROFILE TOLERANCE

A profile tolerance specifies a uniform boundary or zone along the true profile within which all elements of the surface must lie, Figures 20. A profile is the outline of an object in a given plane (two-dimensional) figure. Profiles are formed by projecting a three-dimensional figure onto a plane or by taking cross sections through the figure with the resulting profile composed of such elements as straight lines, arcs, or other curved lines.

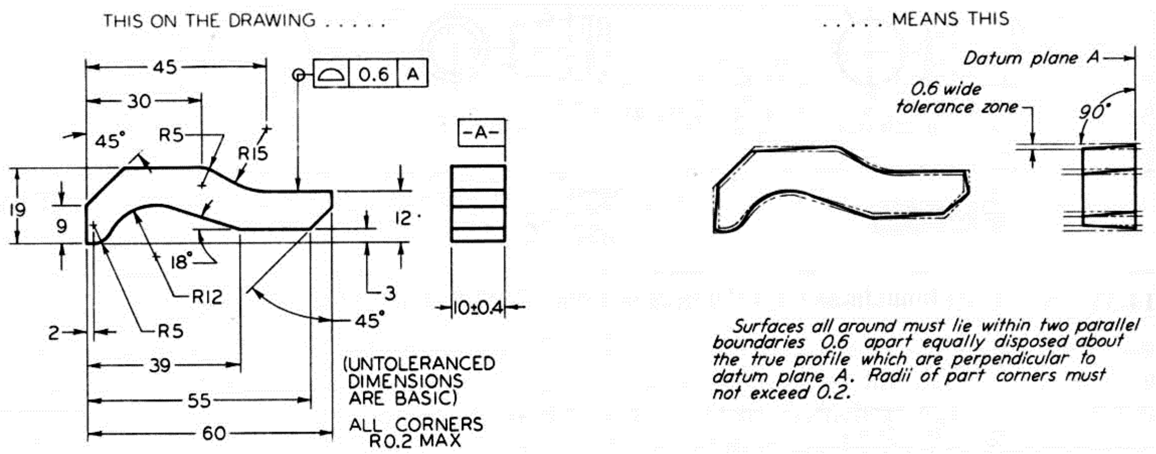


Figure 20. Specifying Profile of a Surface All Around

FORM TOLERANCES FOR RELATED FEATURES

ANGULARITY TOLERANCE An angularity tolerance specifies a tolerance zone defined by two parallel planes at the specified basic angle (other than 90°) from a datum plane or axis within which the surface or the axis of the feature must lie, Figure 21.

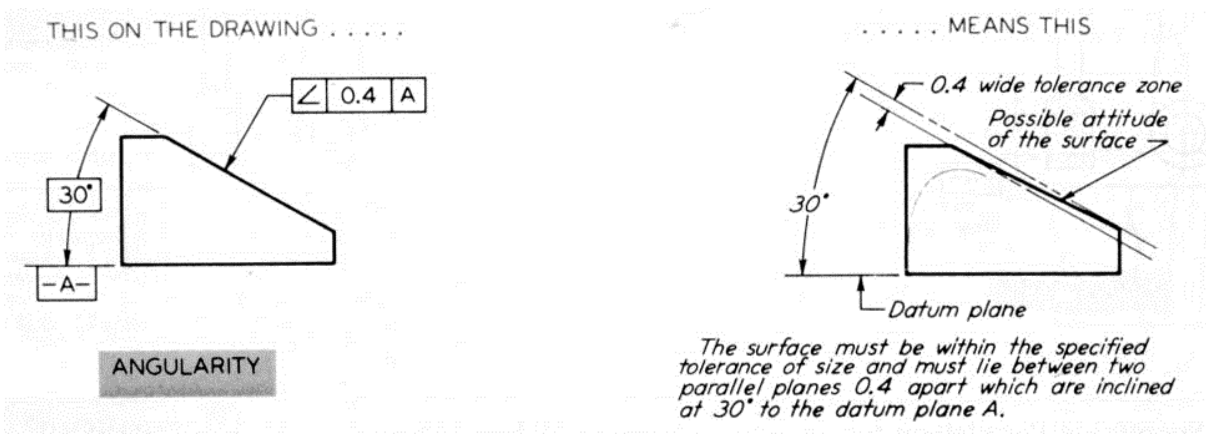


Figure 21. Specifying Angularity for a Plane Surface

PARALLELISM TOLERANCE

A parallelism tolerance specifies a tolerance zone defined by two parallel lines or lines parallel to datum plane or axis within which the surface or axis of the feature must lie, or the parallelism tolerance may specify a cylindrical tolerance zone parallel to a datum axis within which the axis of the feature must lie, Figures 22-24.

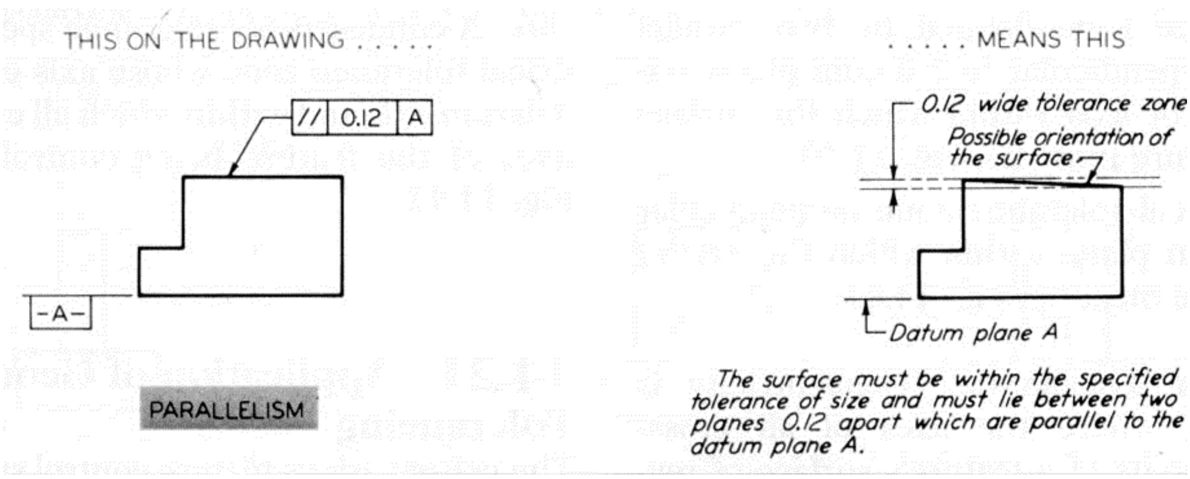


Figure 22. Specifying Parallelism for a Plane Surface

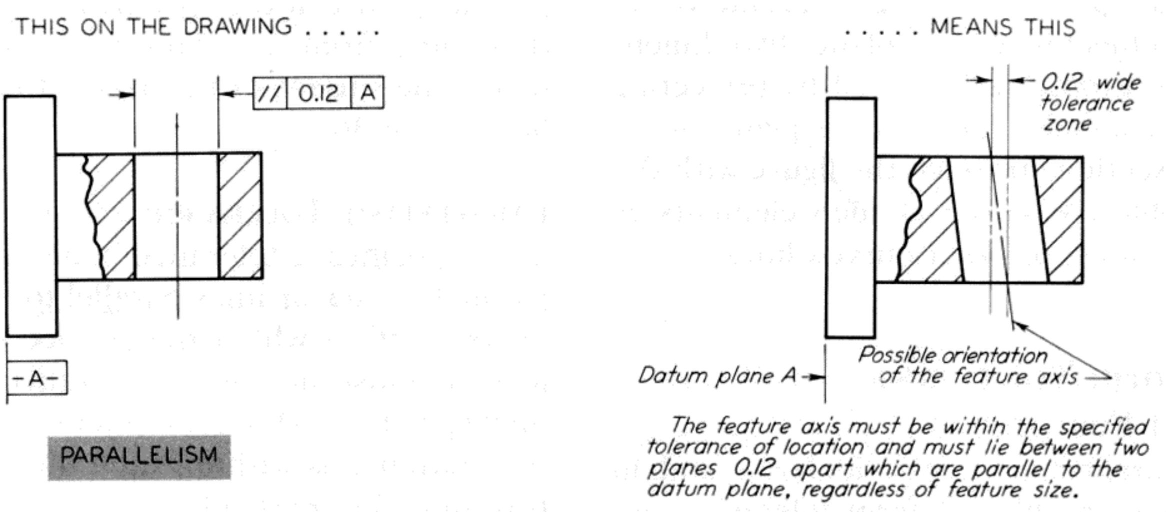


Figure 23. Specifying Parallelism for an Axis Feature RFS

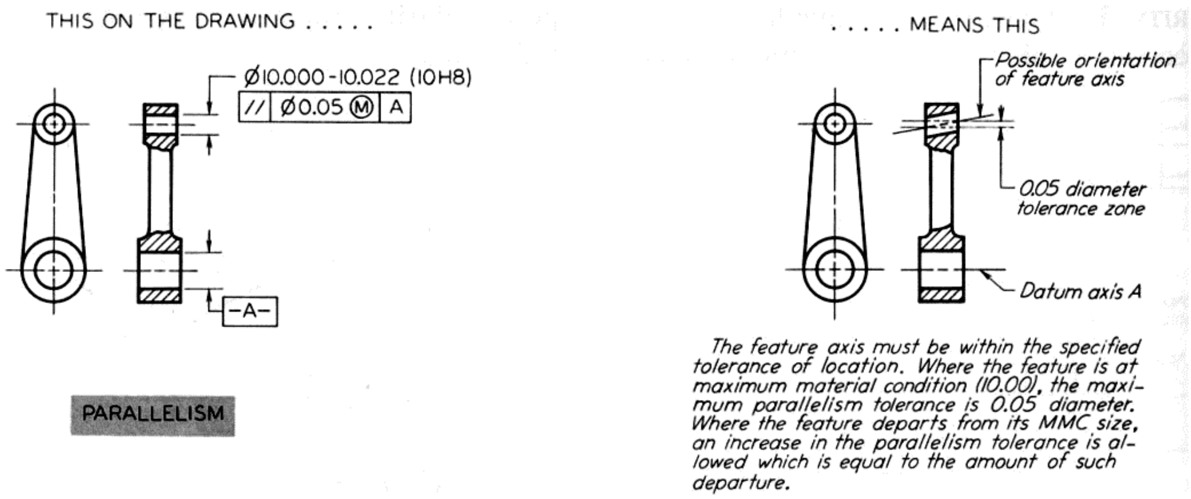


Figure 24. Specifying Parallelism for an Axis Feature at MMC

PERPENDICULARITY TOLERANCE

Perpendicularity is a condition of a surface, median plane, or axis at 90° to datum plane or axis. A perpendicularity tolerance specifies one of the following.

1. A tolerance zone defined by two parallel planes perpendicular to a datum plane, datum axis, or axis within which the surface of the feature must lie, Figure 25.
2. A cylindrical tolerance zone perpendicular to a datum plane within which the axis of the feature must lie, Figure 24.

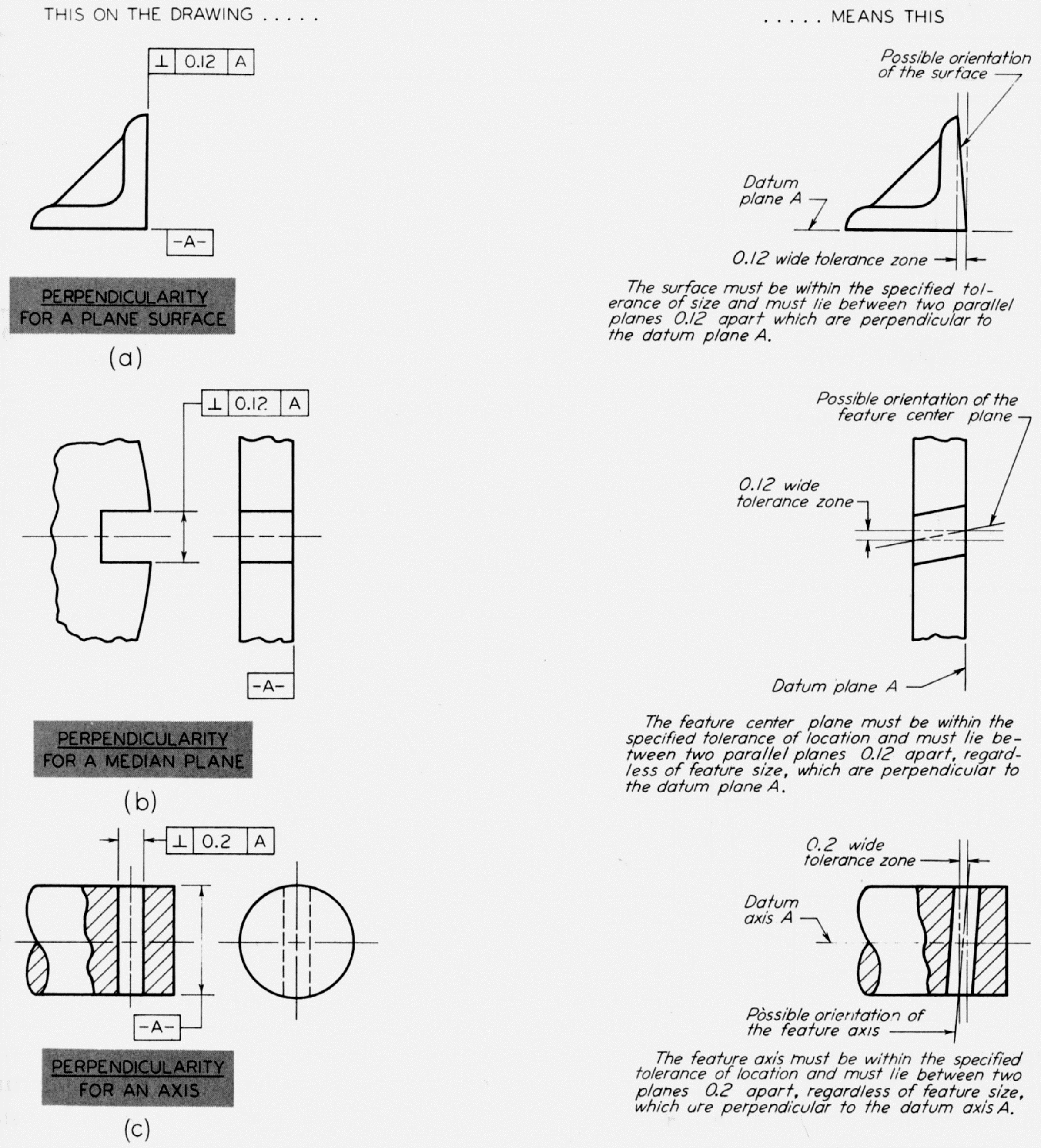


Figure 25. Specifying Perpendicularity

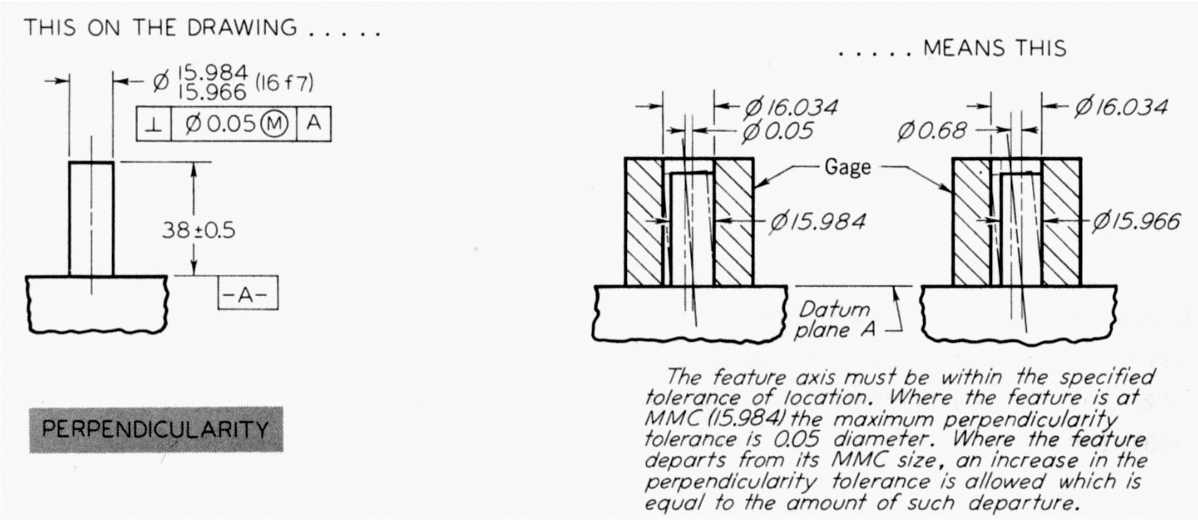


Figure 26 . Specifying Perpendicularity for an Axis, Pin, or Boss

CONCENTRICITY TOLERANCE Concentricity is a condition where the axes of all cross-sectional elements of a feature's surface of revolution are common to the axis of a datum feature. A concentricity tolerance specifies a cylindrical tolerance zone whose axis coincides with a datum axis and within which all cross-sectional axes of the feature being controlled must lie, Figure 25.

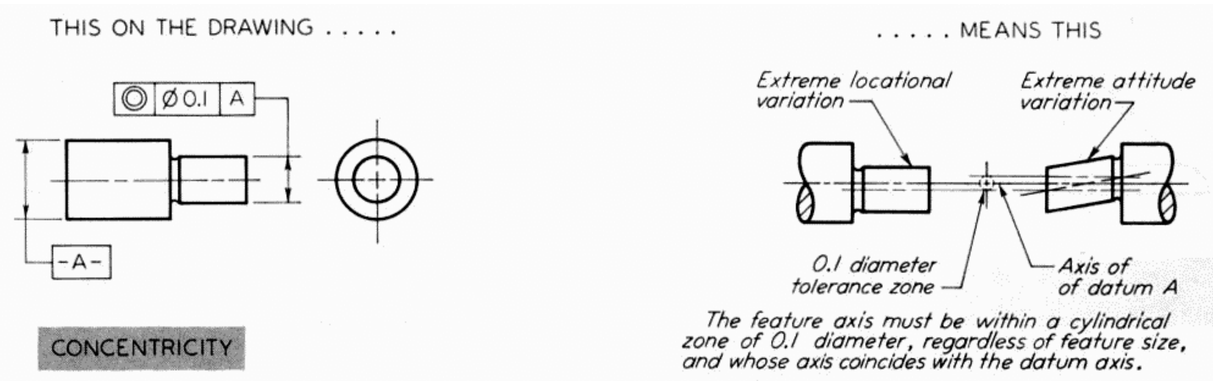
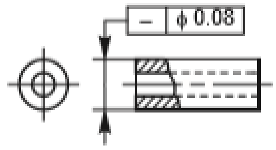
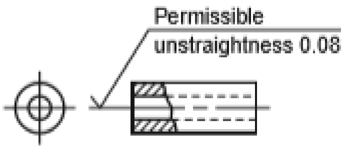
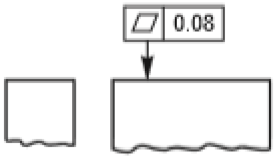
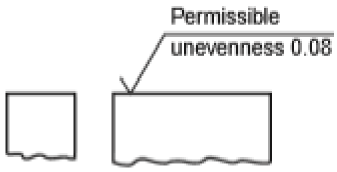
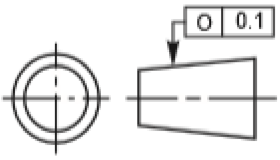
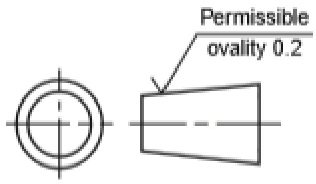
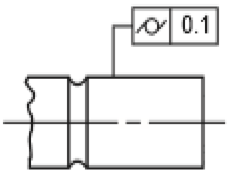
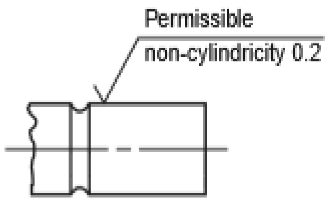
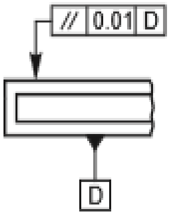
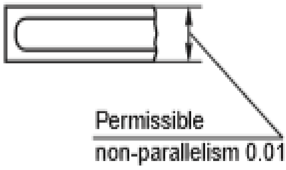
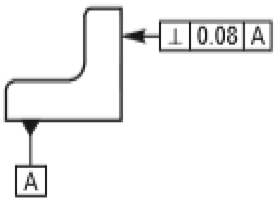
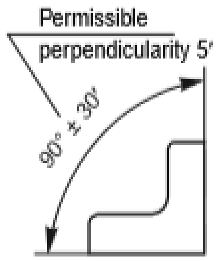


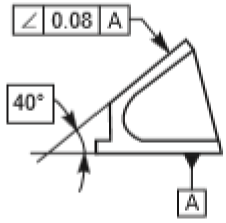
Figure 27 . **Specifying Concentricity**

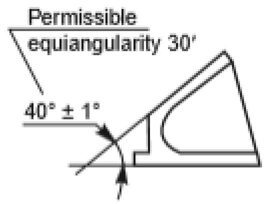
As per the standard	*As prevalent in industry*
1. Straightness tolerance	
	
2. Flatness tolerance	
	
3. Circularity tolerance	
	

(Contd.)

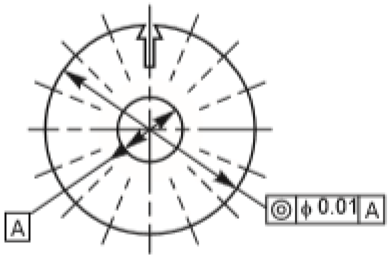
4. Cylindricity tolerance	
	
5. Parallelism tolerance	
	
6. Perpendicularity tolerance	
	

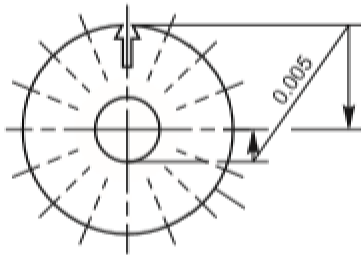
7. Angularity tolerance





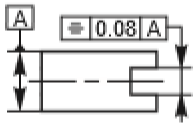
8. Concentricity and coaxiality tolerance

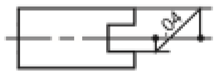




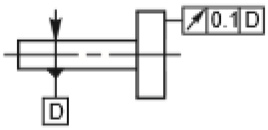
(Contd.)

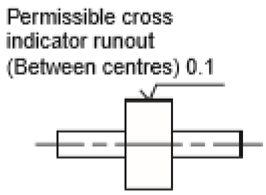
9. Symmetry tolerance



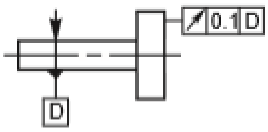


10. Radial run-out





11. Axial run-out



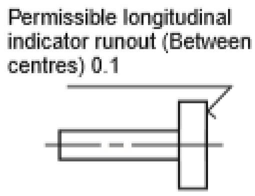


Table 3. Symbols representing the characteristics to be toleranced

APPLICATION OF GEOMETRIC TOLERANCING

The use of various feature control symbols in lieu of notes for position and form tolerance dimensions as abstracted from ANSI Y14.5M-1982 (R1988) is illustrated in Figure 26.

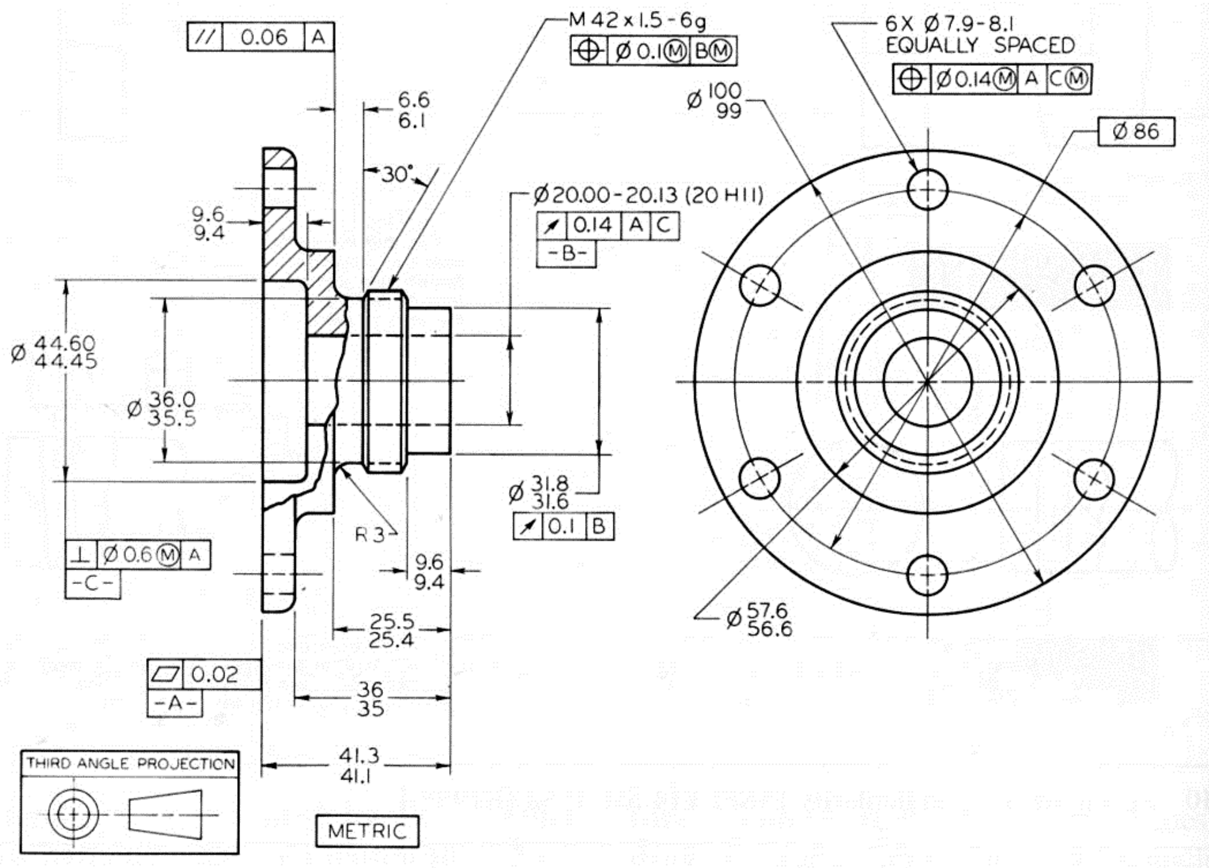


Figure 26. Application of Symbols to Position and Form Tolerance Dimensions

2.3. Introduction to Surface Roughness

It is not possible to achieve in practice, a geometrically ideal surface of a component and hence, production drawings of components must also contain information about the permissible surface conditions. Machine components which have undergone machining operation, when inspected under magnification, will have some minute irregularities. The actual surface condition will depend upon the finishing process adopted. The properties and performance of machine components are affected by the degree of roughness of the various surfaces. The higher the smoothness of the surface, the better is the fatigue strength and corrosion resistance. Friction between mating parts is also reduced due to better surface finish.

Surface Roughness

The geometrical characteristics of a surface include,

1. Macro-deviations,
2. Surface waviness, and
3. Micro-irregularities.

The surface roughness is evaluated by the height, Rt and mean roughness index Ra of the micro-irregularities. Following are the definitions of the terms indicated in Fig. 26

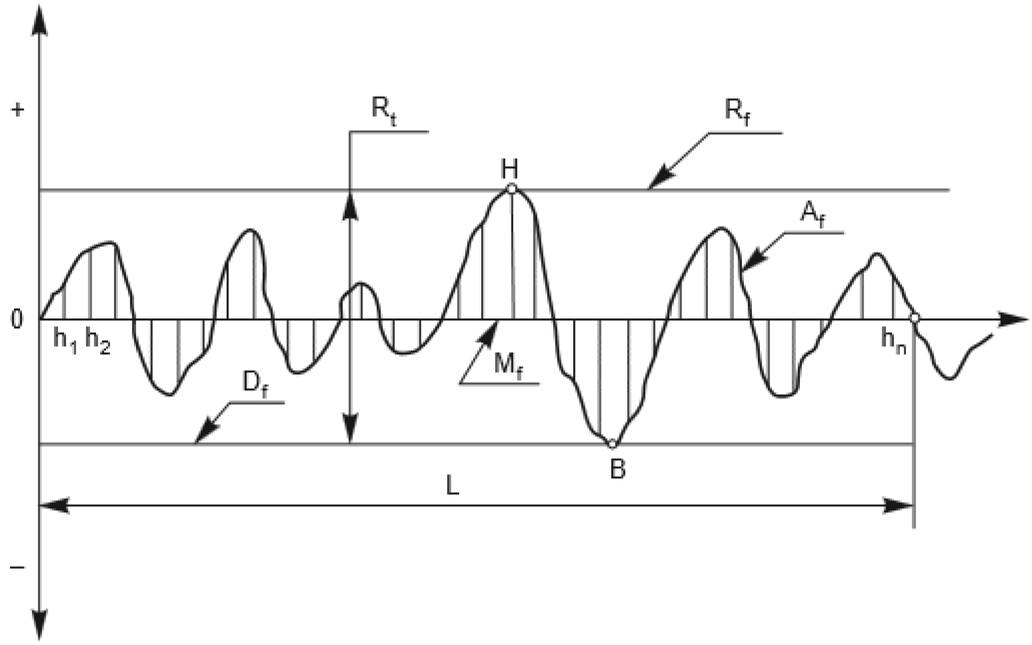


Figure 27 Surface roughness

Actual Profile, A_f

It is the profile of the actual surface obtained by finishing operation.

Reference Profile, R_f

It is the profile to which the irregularities of the surface are referred to. It passes through the highest point of the actual profile.

Datum Profile, D_f

It is the profile, parallel to the reference profile. It passes through the lowest point B of the actual profile,

Mean Profile, M_f

It is that profile, within the sampling length chosen (L), such that the sum of the material filled areas enclosed above it by the actual profile is equal to the sum of the material-void areas enclosed below it by the profile.

Peak to Valley Height, R_t

It is the distance from the datum profile to the reference profile

Mean Roughness Index, R_a

It is the arithmetic mean of the absolute values of the heights h_i between the actual and mean profiles. It is given by,

$$R_a = 1/L \int_{x=0}^{x=L} |h_i| \, dx \text{ , where } L \text{ is the sampling length}$$

The surface roughness may be measured, using any one of the following :

1. Straight edge
2. Surface gauge
3. Optical flat
4. Tool maker's microscope
5. Profilometer
6. Profilograph
7. Talysurf

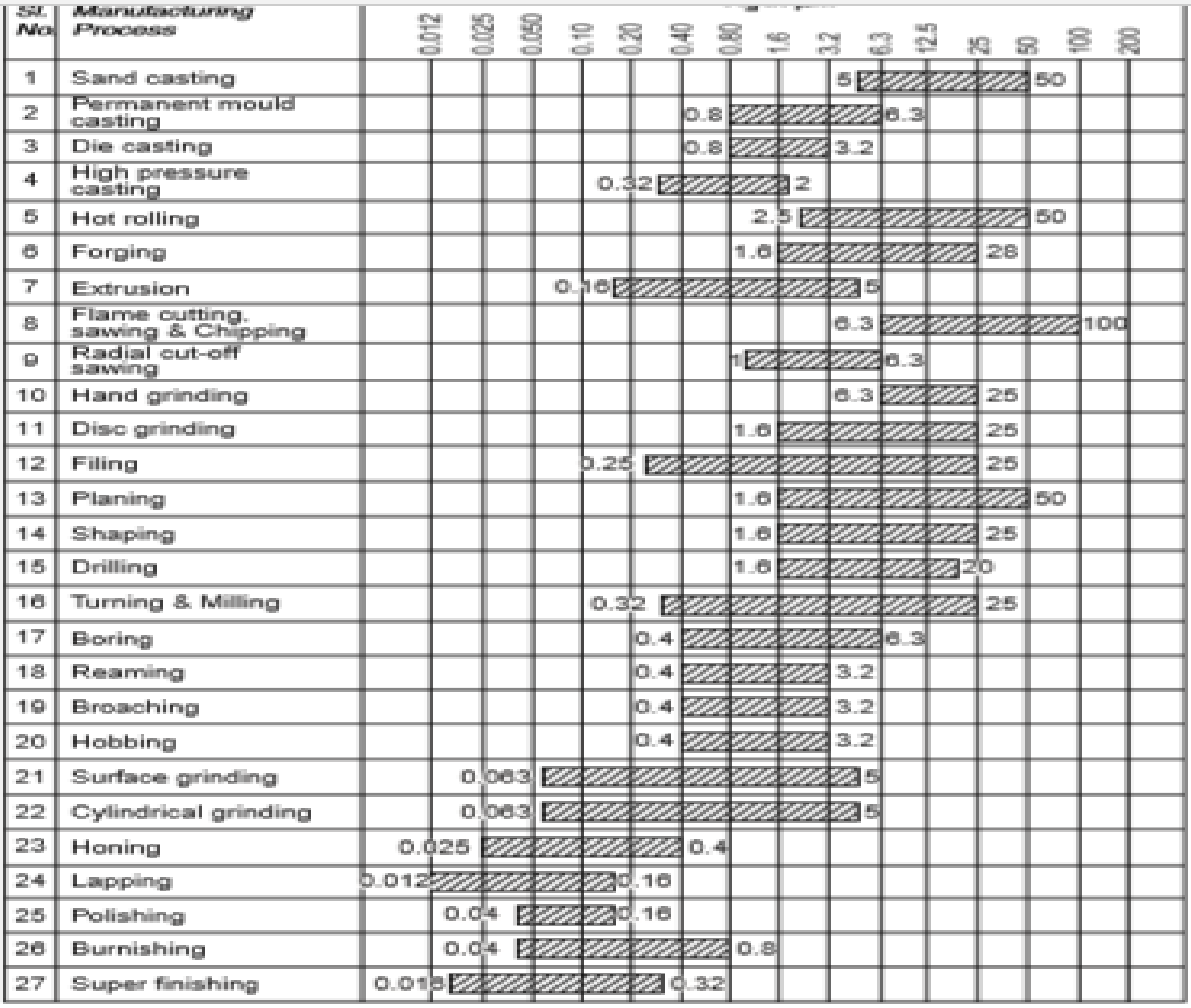


Figure 28. Surface roughness expected from various manufacturing processes

Self-Check -2	Written Test

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have 2.pts)

1. Why it is impossible to make parts economically to the exact dimensions.
   a. Inaccuracies of machines and tools,
   b. Inaccuracies in setting the work to the tool,
   c. Error in measurement
   d. All

2. . It is the difference between the maximum and minimum permissible limits of the given size.
   a. Tolerance b. Limit c. Deviation d. all
3. . The theoretical size from which limits of size are derived by the application of allowances and tolerances
   a. Upper deviation b. lower deviation c. basic size d. fits
4. Type of occurs when two toleranced mating parts will always interfere when assembled.
   a. Interference fit b. Clearance fit c. Line fit d. transition fit

***Note:* Satisfactory rating –4 points and above Unsatisfactory – below 4 points**

You can ask you teacher for the copy of the correct answers.

Name: ____________________________ Date: _________________

Answer Sheet

Score = ____________
Rating: ______________

Information Sheet-3	Inserting Dimensions and Tolerances for various components

3.1. Introduction to Dimensioning

A drawing of a component, in addition to providing complete shape description, must also furnish information regarding the size description. These are provided through the distances between the surfaces, location of holes, nature of surface finish, type of material, etc. The expression of these features on a drawing, using lines, symbols, figures and notes is called dimensioning.

Dimension is a numerical value expressed in appropriate units of measurement and indicated on drawings, using lines, symbols, notes, etc., so that all features are completely defined

1. As far as possible, dimensions should be placed outside the view.
2. Dimensions should be taken from visible outlines rather than from hidden lines.
3. Dimensioning to a centre line should be avoided except when the centre line passes through the centre of a hole.
4. Each feature should be dimensioned once only on a drawing.
5. Dimensions should be placed on the view or section that relates most clearly to the corresponding features.
6. Each drawing should use the same unit for all dimensions, but without showing the unit symbol.
7. No more dimensions than are necessary to define a part should be shown on a drawing.
8. No features of a part should be defined by more than one dimension in any one direction.

3.2. THREE RULES OF TOLERANCES

There are general rules of tolerance geometric features that should be followed in this type of dimensioning.

RULE 1 (INDIVISUAL FEATURE SIZE) When only a tolerance of size is specified on a part, the limits of size prescribed the amount of variation permitted in its geometric form. In Figure 1, you can see how the forms of the shaft and hole are permitted to vary within the tolerance of size indicated by the dimensions.

RULE 2 (TOLERANCES OF POSITION) When a tolerance of position is specified in a drawing. RFS, MMC, or LMC must be specified with respect to the tolerance, the datum, or both. You can see that the specification of symmetry of the part Figure 1 is based on a tolerance at RFS from a datum at RFS.

RULE 3 (ALL OTHER GEOMETRIC TOLERANCE) RFS applies for all other geometric tolerances for individual tolerances and datum references, if no modifying symbol is given in the feature control symbol. If a feature is to be at maximum material condition, MMC must be specified.

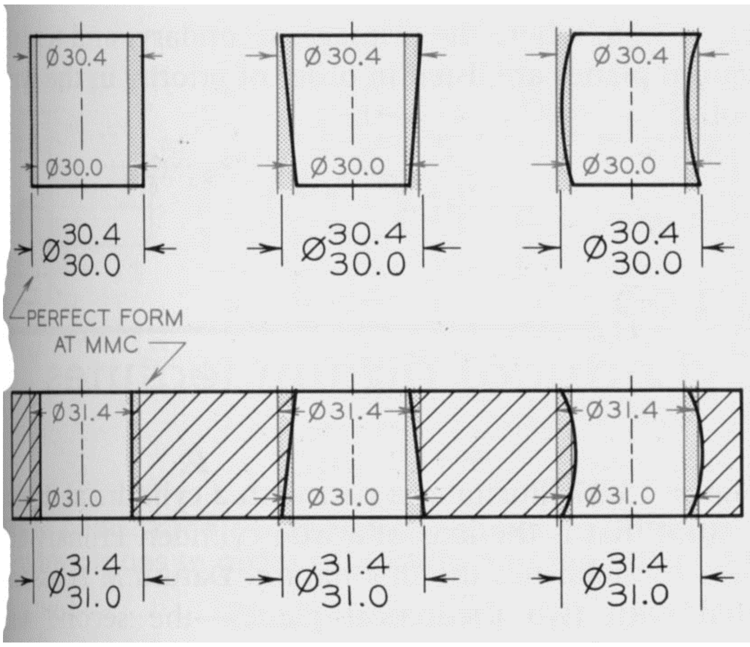


Figure 1. When only a tolerance of size is specified on a part, the limits prescribe the form of the part, as shown in these examples of shafts and holes with the same limits of tolerance.

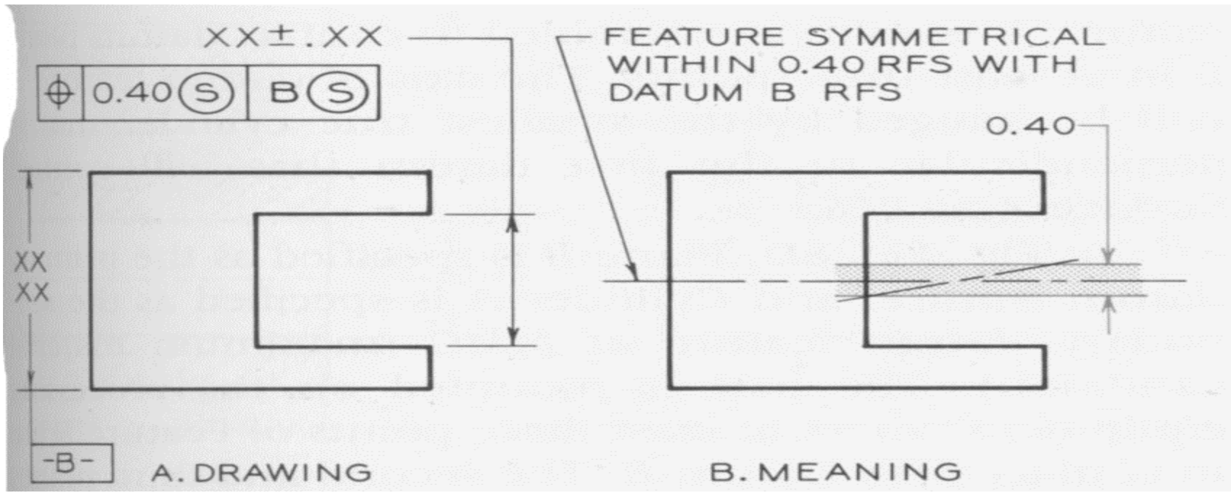


Figure 2. Tolerances of position should include a note of M, S, or L to indicate maximum material condition, regardless of feature size, or least material condition

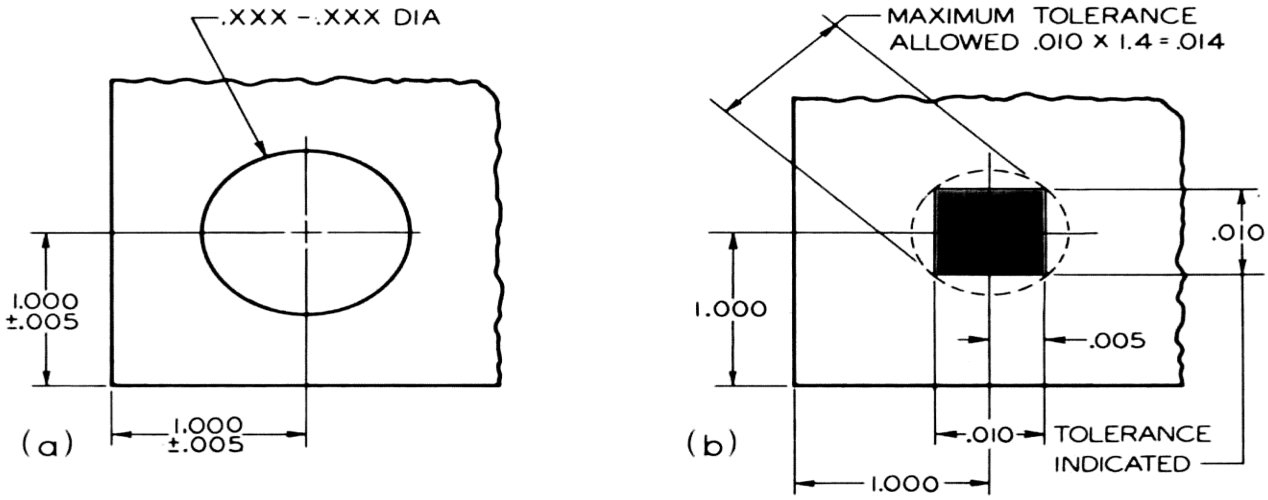


Figure 3. Tolerance zones

POSITIONAL TOLERANCES

Each dimension has a tolerance, either given directly or indicated on the completed drawing by a general note.

For example, Figure 3 (a) shows a hole located from two surfaces at right angles to each other. At shown at (b), the center may lie anywhere within a square tolerance zone, the sides of which are equal to the tolerances. Thus, the total variations along either diagonal of the square by the coordinate method of dimensioning will be 1.4 times greater than the indicated tolerance. Hence, a 0.014 diameter tolerance zone would increase the square tolerance zone area 57 percent without exceeding the tolerance permitted along the diagonal of the square tolerance zone.

Features located by toleranced angular and radial dimensions will have a wedge-shaped tolerance zone. See Figure 11.

If four holes are dimensioned with rectangular coordinates as in Figure 4 (a), acceptable patterns for the square tolerance zones for the holes are shown at (b) and (c). The locational tolerances are actually greater than indicated by the dimensions.
Feature control symbols are related to the feature by one of several methods illustrated in Figure 26. The following methods are preferred.

1. Adding the symbol to a note or dimension pertaining to the feature.
2. Running to a leader from the symbol to the feature.
3. Attaching the side, end, or corner of the symbol frame to an extension line from the feature.
4. Attaching the side or end of the symbol frame to the dimension line pertaining to the feature.

In Figure 4(a), hole A is selected as a datum, and the other three are located from it. The square tolerance zones for hole results from the tolerances on the two rectangular coordinate dimensions locating hole *a.*

The sizes of the tolerance zones for the other three holes result from the tolerances between the holes, while their locations will vary according to the actual location of the datum hole A. two of the many possible zone patterns are shown at (b) and (c).

Thus, with the dimensions shown at (a), it is difficult to say whether the resulting parts will actually fit the mating parts satisfactorily even though they conform to the tolerances shown on the drawing.

These disadvantages are overcame by giving exact theoretical locations by untoleranced dimensions and then specifying by a note how far actual positions may be displaced from these locations. This is called *true-position dimensioning*. It will be seen that the tolerance zone for each hole will be a circle, the size of the circle depending on the amount of variation permitted from "true position."

A true-position dimension denotes the theoretically exact position of a feature. The location of each feature such as a hole, slot, stud, and so on, is given by untoleranced basic dimensions identified by the enclosing frame or symbol. To prevent misunderstandings, true position should be established with respect to a datum.

In simple arrangements, the choice of a datum may be obvious and not require identification.

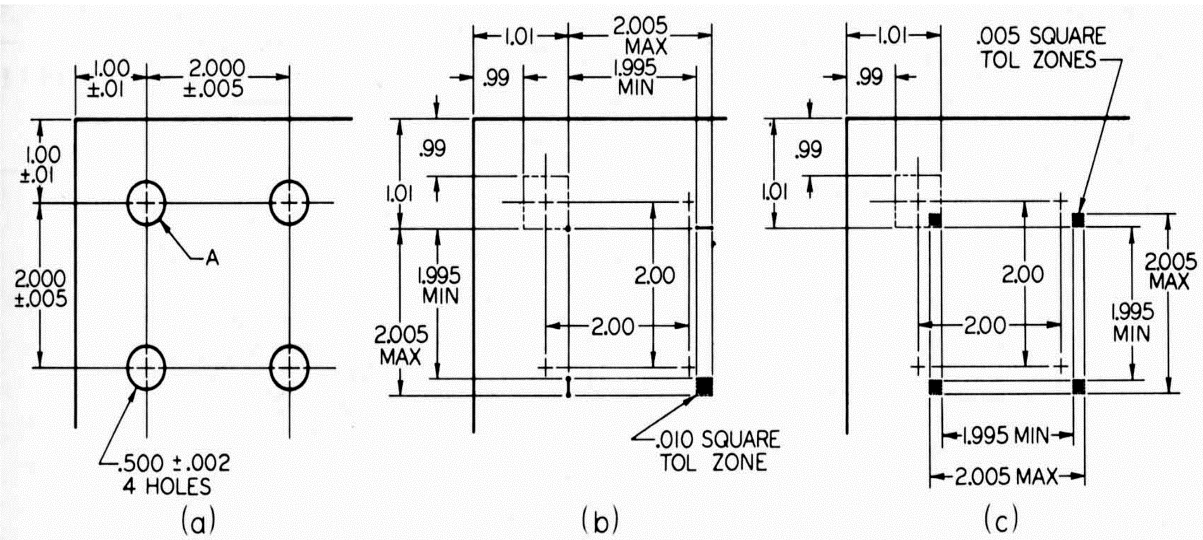


Figure 4. Tolerance zones

Positional tolerance is identified by a characteristic symbol directed to a feature, which establishes a circular tolerance zone, Figure 5.

Actually, the "circular tolerance zone" is a cylindrical tolerance zone (the diameter of which is equal to the positional tolerance and its length is equal to the length of the feature unless otherwise specified), and its axis must be within this cylinder, Figure 6.

The center line of the hole may coincide with the center line of the cylindrical tolerance zone (a), or it may be parallel to it but displaced so as to remain within the tolerance cylinder, (b), or it may be inclined while remaining within the tolerance cylinder, (c). In this last case, we see that the positional tolerance also defines the limits of squareness variation.

In terms of the cylindrical surface of the hole, the positional tolerance specification indicates that all elements on the hole surface must be on or outside a cylinder whose diameter is equal to the minimum diameter or the maximum diameter of the hole minus the positional tolerance (diameter, or twice the radius), with the center line of the cylinder located at true position, Figure 5.

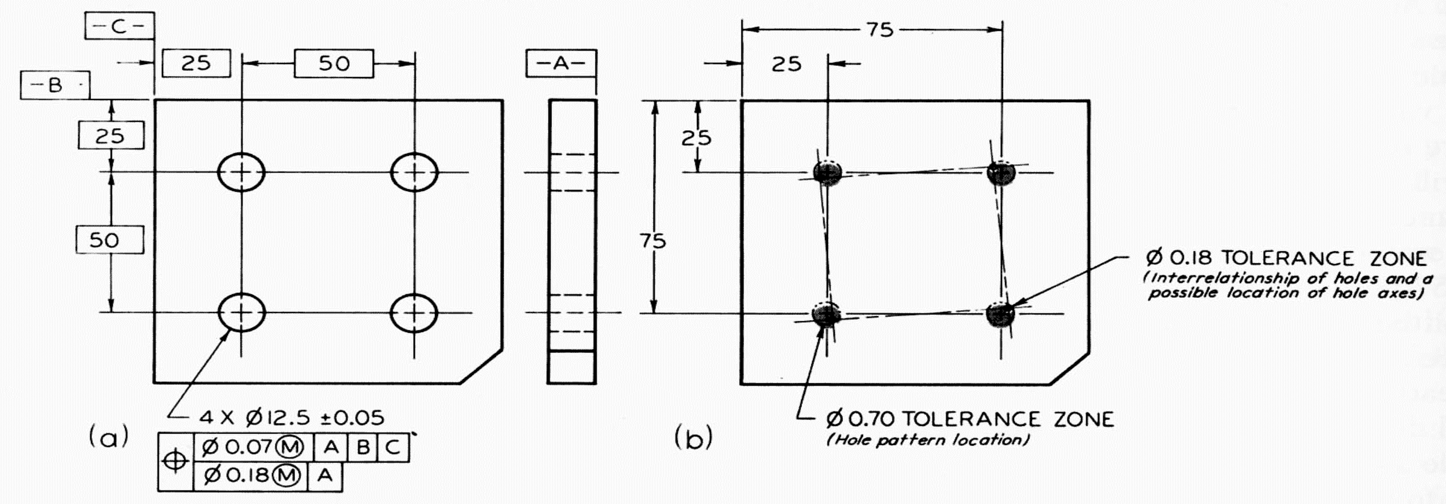


Figure 5. True-Position Dimensioning

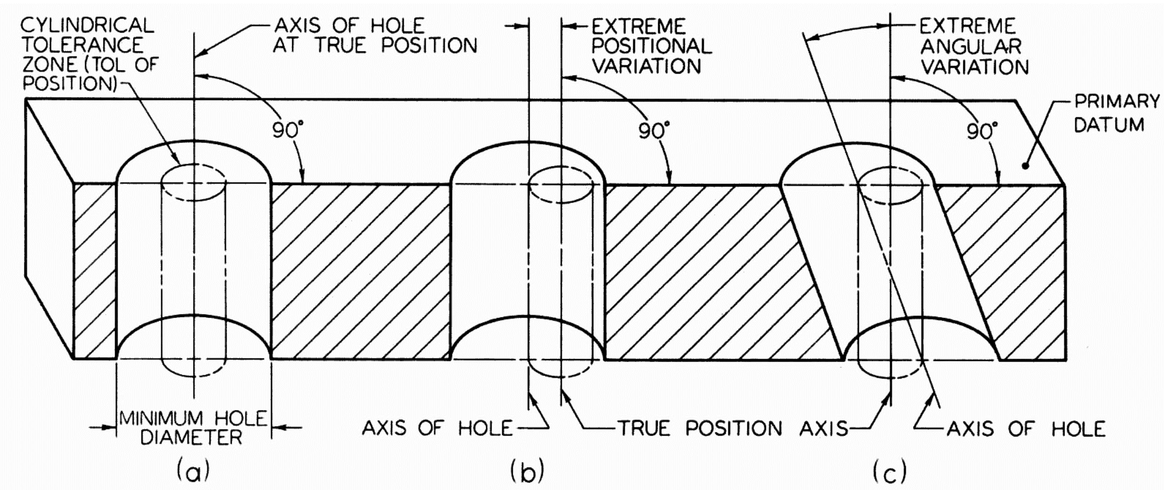


Figure 6. Cylindrical Tolerance Zone

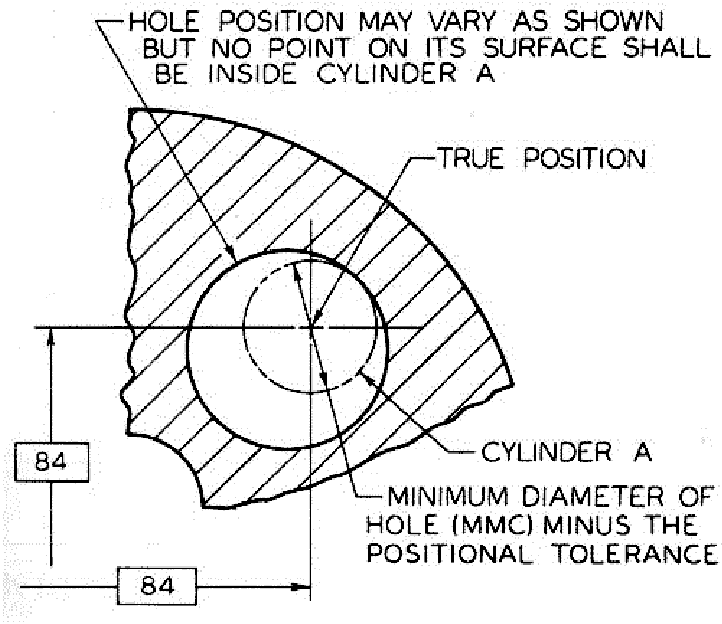


Figure7. True Position Interpretation

The use of basic un tolerance dimensions to locate features at true position avoids one of the chief difficulties in toleranced – the accumulation of tolerances even in a chain of dimensions. Figure 8.

While features, such as holes and bosses, may vary in any direction from the true-position axis, other features, such as slots, may vary on either side of a true-position plane, Figure 9.

Since un tolerance dimensions give the exact locations of the true positions, it is important to prevent the application of general tolerances to these. A note should be added to the drawing such as

GENERAL TOLERANCES DOES NOT APPLY TO BASIC

TRUE-POSITION DIMENSIONS.

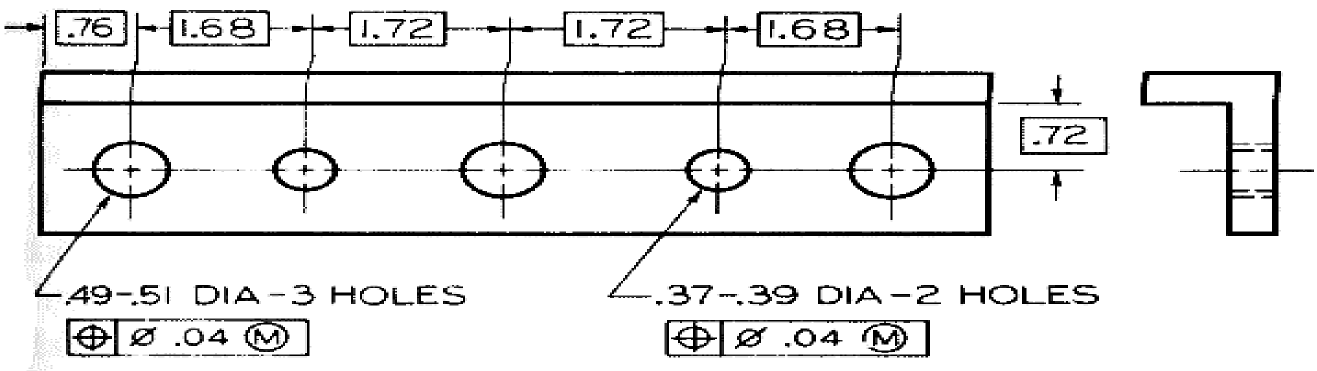


Figure 8. No Tolerance Accumulation

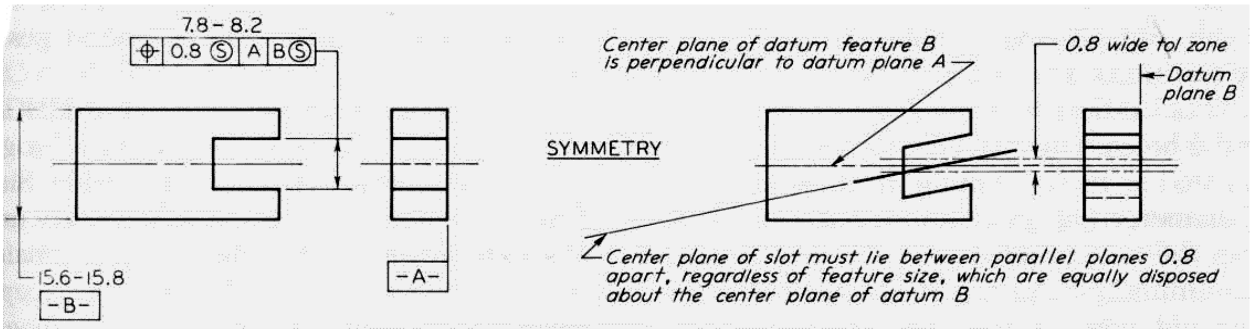


Figure 9. Positional Tolerance for Symmetry

MAXIMUM MATERIAL CONDITION

Maximum material condition, usually abbreviated to MMC, means that a feature of a finished product contains the maximum amount of material permitted by the tolerance size dimensions shown for that feature. Thus, we have MMC when holes, slots, or other internal features are at minimum size, or when shafts, pads, bosses, and other external features are at their maximum size. We have MMC for both mating parts when the largest shaft is in the smallest hole and there is the least clearance between the parts.

TOLERANCES OF ANGLES

Bilateral tolerances are traditionally been given on angles, as illustrated in Figure 10. Consequently, the wedge-shaped tolerance zone increases as the distance from vertex of the angle increases. Thus, the tolerance had to be figured after considering the total displacement at the point farthest from the vertex of the angle before a tolerance could be specified that would not exceed the allowable displacement. The use of angular tolerances may be avoided by using gages. Taper turning is often handled by machining to fit a gage or by fitting to the mating part.

If an angular surface is located by a linear and an angular dimension, Figure 11 (a), the surface must lie within a tolerance zone as shown at (b). The angular zone will be wider as the distance from the vertex increases. In order to avoid the accumulation of tolerances, that is, to decrease the tolerance zone, the basic angle tolerance method of (c) is recommended. The angle is indicated as basic with the proper symbol and no angular tolerance is specified. The tolerance zone is now defined by two parallel planes, resulting in improved angular control, (d).

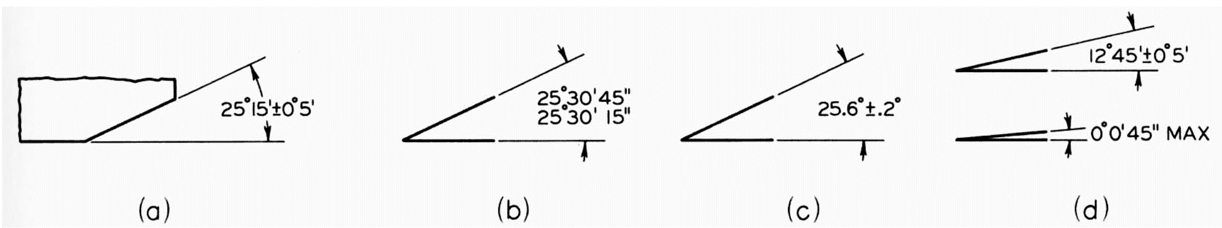


Figure10. Tolerances of Angles

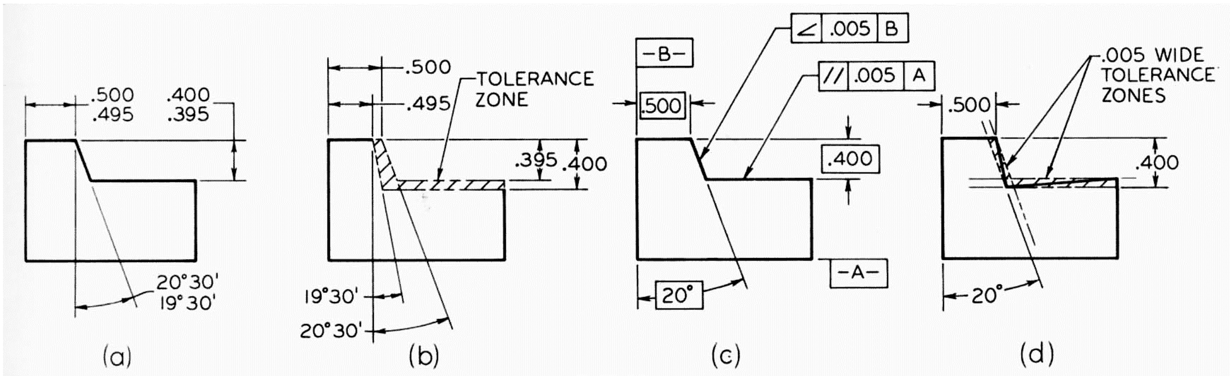


Figure11. Angular Tolerance Zones

Self-Check -3	Multiple choice

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have 2.pts)

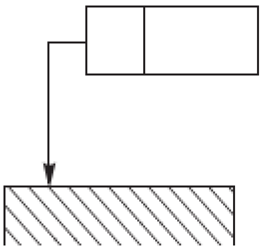
1. ________ means that a feature of a finished product contains the maximum amount of material permitted by the toleranced size dimensions shown for that feature

a. Tolerance angle   b. Maximum material condition (MMC)   c. both are answers

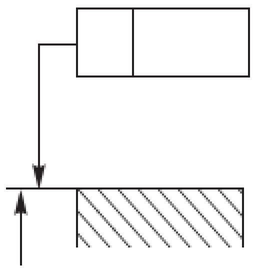
2. Which of the following methods used to relate feature control symbols to the feature?

a. Adding the symbol to a note or dimension pertaining to the feature.
b. Running to a leader from the symbol to the feature.
c. Attaching the side, end, or corner of the symbol frame to an extension line from the feature.
d. Attaching the side or end of the symbol frame to the dimension line pertaining to the feature.
e. All of above

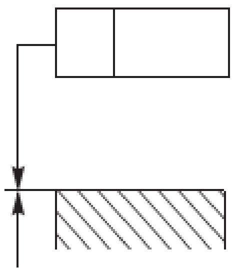
3. All figure correctly represent connection of tolerance frame with tolerance feature except



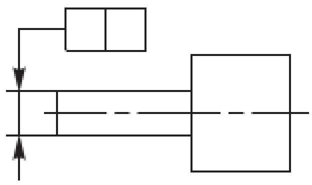
a



b



c.



d.

Note: **Satisfactory rating –4 points and above points** **Unsatisfactory – below 4 points**

You can ask you teacher for the copy of the correct answers.

Name: _________________________ Date: _______________

Answer Sheet

Score = ___________

Rating: ____________

Information Sheet-4	Appropriate Symbols for Limit and fits, geometric Tolerance

4.1. Symbols for limits and fits, surface texture and geometric tolerances

4.1.1. SYMBOLS FOR TOLERANCES OF POSITION AND FORM

Since traditional narrative notes for specifying tolerances of *position* (location) and *form* (shape) may be confusing or not clear, may require much of the space available on the drawing, and often may not be understood internationally, most multinational companies have adopted symbols for such specification. These ANSI symbols provide an accurate and concise means of specifying geometric characteristics and tolerances in a minimum of space, Table 1. Notes may supplement the symbols if the precise geometric requirements cannot be conveyed by the symbols. For construction details of the geometric tolerancing symbols, see Appendix 11.

Combinations of the various symbols and their meanings are given in Figure24. Applications of the symbols to a drawing are illustrated in Figure 26. application

The geometric characteristic symbols plus the supplementary symbols are further explained and illustrated with material adapted from ANSI Y14.5M-1982 (R1988), as follows.

Geometric characteristic symbols

	Type of Tolerance	Characteristic	Symbol
For individual features	Form	Straightness	—
		Flatness	▱
		Circularity (roundness)	○
		Cylindricity	/○/
For individual or related features	Profile	Profile of a line	⌒
		Profile of a surface	⌓
For related features	Orientation	Angularity	∠
		Perpendicularity	⊥
		Parallelism	//
	Location	Position	⌖
		Concentricity	◎
	Runout	Circular runout	↗ [a]
		Total runout	⌰ [a]

[a]Arrowhead(s) may be filled in.

Modifying symbols

Term	Symbol
At maximum material condition	Ⓜ
Regardless of feature size	Ⓢ
At least material condition	Ⓛ
Projected tolerance zone	Ⓟ
Diameter	∅
Spherical diameter	S∅
Radius	R
Spherical radius	SR
Reference	( )
Arc length	⌒

Table1. **Geometric Characteristics and Modifying Symbols**

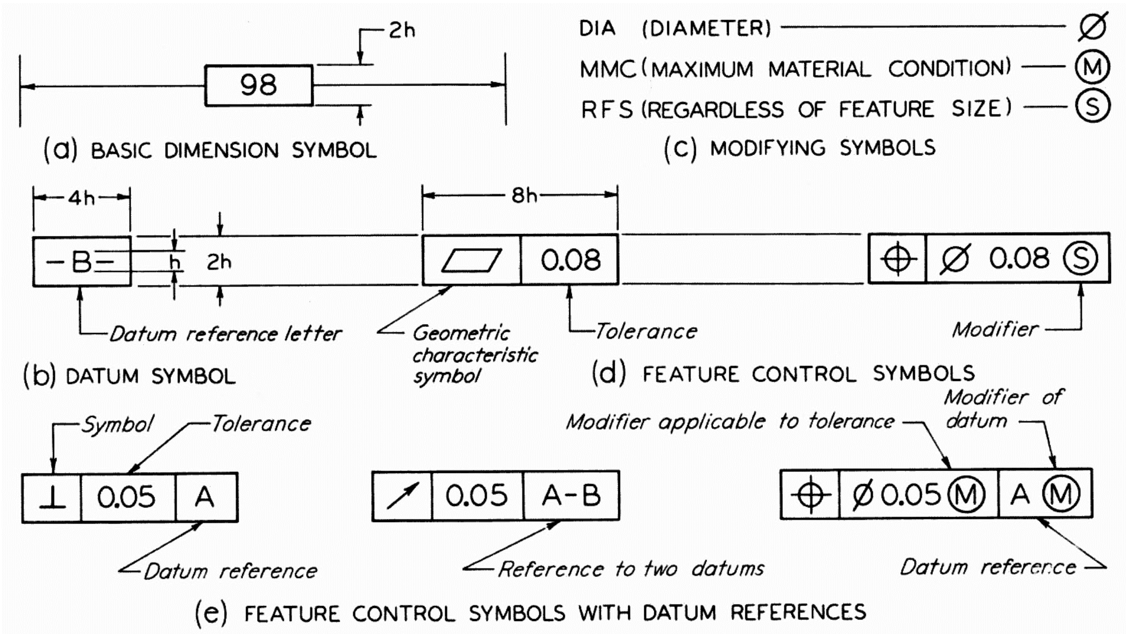


Figure 1. Use of Symbols for Tolerance of Position and Form

BASIC DIMENSION SYMBOL The basic dimension is identified by the enclosing frame symbol, Figure 1(a). The basic dimension (size) is the value used to describe the theoretically exact size, shape, or location of the feature. It is the basis from which permissible variations are established by tolerances on other dimensions in notes, or in feature control frames.

DATUM IDENTIFYING SYMBOL The datum-identifying symbol consists of a frame containing a reference letter preceded and followed by a dash, Figure1 (b). A point, line, plane, cylinder, or other geometric form assumed to be exact for purposes of computation may serve as a datum from which the location or geometric relationship of features of a part may be established.

SUPPLEMENTARY SYMBOLS The symbols for MMC (maximum material condition, i.e., minimum hole diameter, maximum shaft diameter) and RFS (regardless of feature size – the tolerance applies to any size of the feature within its size tolerance and or the actual size of a datum feature) are illustrated in Figure 1 (c). The abbreviations MMC and RFS are also used in notes. See Table 3.

The symbol for diameter is used instead of the abbreviation DIA to indicate a diameter, and it precedes the specified tolerance in a feature control symbol, Figure 1(d). This symbol for diameter instead of the abbreviation DIA may be used on a drawing, and it should precede the dimension. For narrative notes, the abbreviation DIA is preferred.

COMBINED SYMBOLS- Individual symbols, datum reference letters, needed tolerances, and so on may be combined in a single frame, Figure 1 (e).

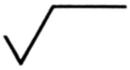
A position of form tolerance is given by a feature control symbol made up of a frame about the appropriate geometric characteristic symbol plus the allowable tolerance. A vertical line separates the symbol and the tolerance, Figure 1 (d). Where needed, the tolerance should be preceded by the symbol for diameter and followed by the symbol for MMC or RFS.

A tolerance of position or form related to a datum is so indicated in the feature control symbol by placing the datum reference letter following either the geometric characteristic symbol or the tolerance. Vertical lines separate the entries, and, where applicable, the datum reference letter entry includes the symbol for MMC or RFS. See Figure 1.

4.2. SURFACE ROUGHNESS, WAVINESS, AND LA

The modern demands of the automobile, the airplane, and other machines that can stand heavier loads and higher speeds with less friction and wear have increased the need for accurate control of surface quality by the designer regardless of the size of the feature. Simple finish marks are not adequate to specify surface finish on such parts.

Surface finish is intimately related to the functioning of a surface, and proper specification of finish of such surfaces as bearings and seals is necessary. Surface quality specifications should be used only where needed, since the cost of producing a finished surface becomes greater as the quality of the surface called for is increased. Generally, the ideal surface finish is the roughest one that will do the job satisfactorily.

Symbol	Meaning
(a) 	Basic Surface Texture Symbol. Surface may be produced by any method except when the bar or circle, (b) or (d), is specified.
(b) 	Material Removal By Machining Is Required. The horizontal bar indicates that material removal by machining is required to produce the surface and that material must be provided for that purpose.
(c) 3.5 	Material Removal Allowance. The number indicates the amount of stock to be removed by machining in millimeters (or inches). Tolerances may be added to the basic value shown or in a general note.
(d) 	Material Removal Prohibited. The circle in the vee indicates that the surface must be produced by processes such as casting, forging, hot finishing, cold finishing, die casting, powder metal-lurgy or injection molding without subsequent removal of material.
(e) 	Surface Texture Symbol. To be used when any surface characteristics are specified above the horizontal line or to the right of the symbol. Surface may be produced by any method except when the bar or circle, (b) or (d), is specified.

(f)

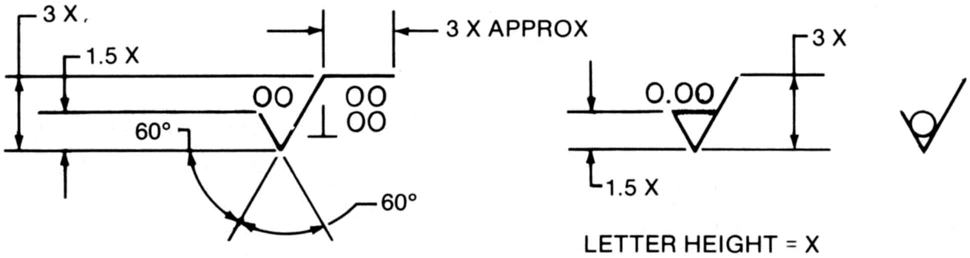


Figure 2. Surface Texture Symbols and Construction

The system of surface texture symbols recommended by ANSI for use on drawings, regardless of the system of the measurement used, is now broadly accepted by American industry. These symbols are used to define surface texture, roughness and lay. See Figure 2 for meaning and construction of these symbols. The basic surface texture symbol in Figure 2(a) indicates a finished or machined surface by any method just as does the general V symbol, Figure 4(a). Modifications to the basic surface texture symbol, Figure 2 (b) through (d), define restrictions on material removal for the finished surface. Where surface texture values other than roughness average (R_a) are specified, the symbol must be drawn with the horizontal extension as shown in (e). Construction details for the symbols are given in (f).

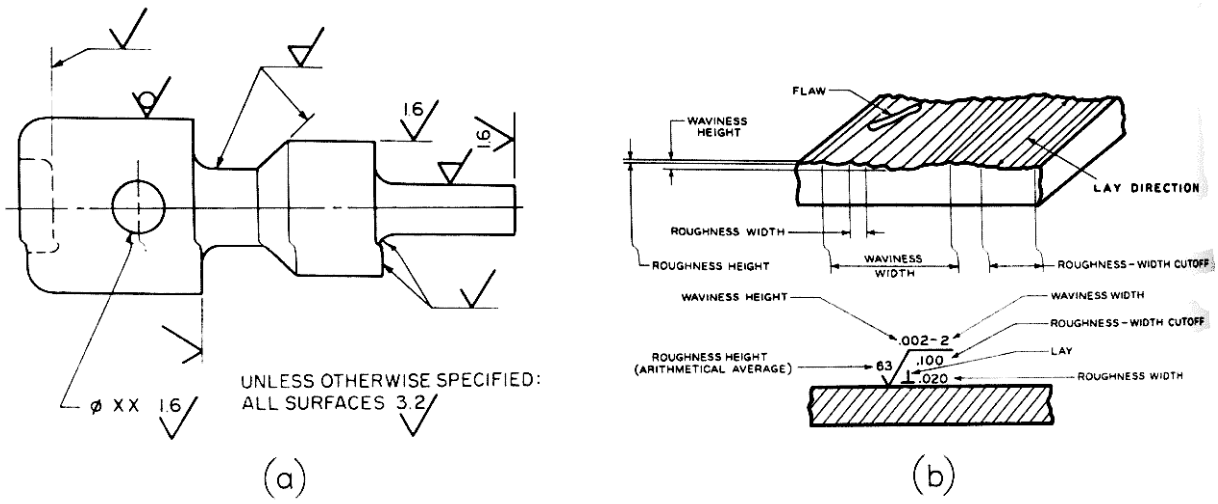


Figure 3. Application of Surface Texture Symbols and Surface Characteristics

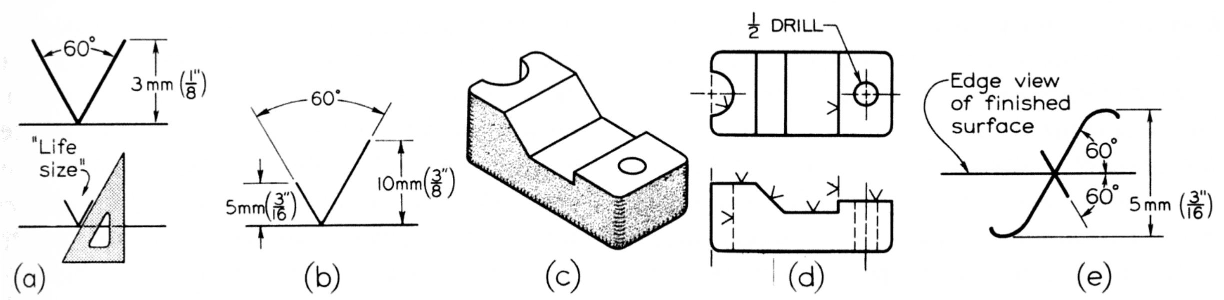


Figure 4. Finish Marks

Applications of the surface texture symbols are given in Figure3 (a). Note that the symbols read from the bottom and/or the right side of the drawing and that they are not drawn at any angle or upside down.

Measurements for roughness and waviness, unless otherwise specified, apply in the direction that gives the maximum reading, usually across the lay. See Figure 3 (b). The recommended roughness height values are given in Table 3.

Millimeters (mm)	Inches (in.)	Millimeters (mm)	Inches (in.)
0.08	.003	2.5	.1
0.25	.010	8.0	.3
0.80	.030	25.0	1.0

Table 2. Standard Roughness Sampling Length (Cutoff) Values

Recommended values are in color.

Micro-meters[a] (μm)	Micro-inches (μin.)	Micro-meters[a] (μm)	Micro-inches (μin.)
0.012	0.5	1.25	50
0.025	1	1.60	63
0.050	2	2.0	80
0.075	3	2.5	100
0.10	4	3.2	125
0.125	5	4.0	180
0.15	6	5.0	200
0.20	8	6.3	250
0.25	10	8.0	320
0.32	13	10.0	400
0.40	16	12.5	500
0.50	20	15	600
0.63	25	20	800
0.80	32	25	1000
1.00	40		

[a]Micrometers are the same as thousandths of a millimeter (1 μm = 0.001 mm).

Table 3. Preferred Series Roughness Average Values (R_a)

Millimeters (mm)	Inches (in.)	Millimeters (mm)	Inches (in.)
0.0005	.00002	0.025	.001
0.0008	.00003	0.05	.002
0.0012	.00005	0.08	.003
0.0020	.00008	0.12	.005
0.0025	.0001	0.20	.008
0.005	.0002	0.25	.010
0.008	.0003	0.38	.015
0.012	.0005	0.50	.020
0.020	.0008	0.80	.030

Table 4.

When it is necessary to indicate the roughness-width cutoff values, the standard values to be used are listed in Table 4. If no value is specified, the 0.80 value is assumedWhen maximum waviness height values are required, the recommended values to be used are as given in Table 5.

Millimeters (mm)	Inches (in.)	Millimeters (mm)	Inches (in.)
0.0005	.00002	0.025	.001
0.0008	.00003	0.05	.002
0.0012	.00005	0.08	.003
0.0020	.00008	0.12	.005
0.0025	.0001	0.20	.008
0.005	.0002	0.25	.010
0.008	.0003	0.38	.015
0.012	.0005	0.50	.020
0.020	.0008	0.80	.030

Table 5. Preferred Series Maximum Waviness Height Values

When it is desired to indicate lay, the lay symbols in Figure 5 are added to the surface texture symbol as per the examples given. Selected applications of the surface texture values to the symbol are given and explained in Figure 6..

A typical range of surface roughness values may be obtained from various production methods is shown in Figure 7. Preferred roughness-height values are shown at the top of the chart.

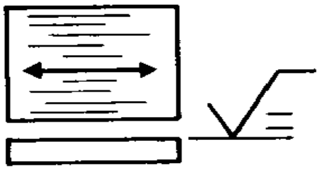
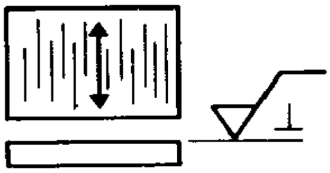
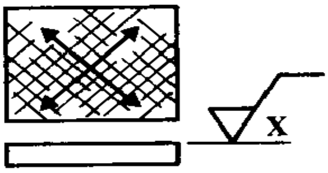
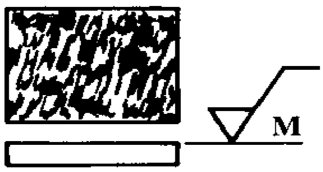
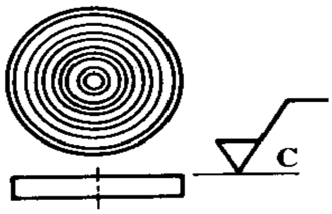
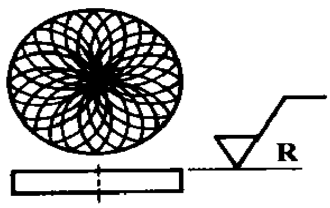
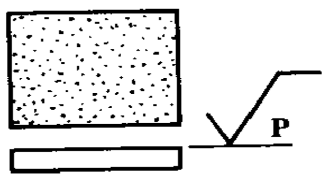
Lay Symbol	Meaning	Example Showing Direction of Tool Marks
=	Lay approximately parallel to the line representing the surface to which the symbol is applied.	
⊥	Lay approximately perpendicular to the line representing the surface to which the symbol is applied.	
X	Lay angular in both directions to line representing the surface to which the symbol is applied.	
M	Lay multidirectional.	
C	Lay approximately circular relative to the center of the surface to which the symbol is applied.	
R	Lay approximately radial relative to the center of the surface to which the symbol is applied.	
P	Lay particulate, non-directional, or protuberant.	

Figure 5. Lay Symbols

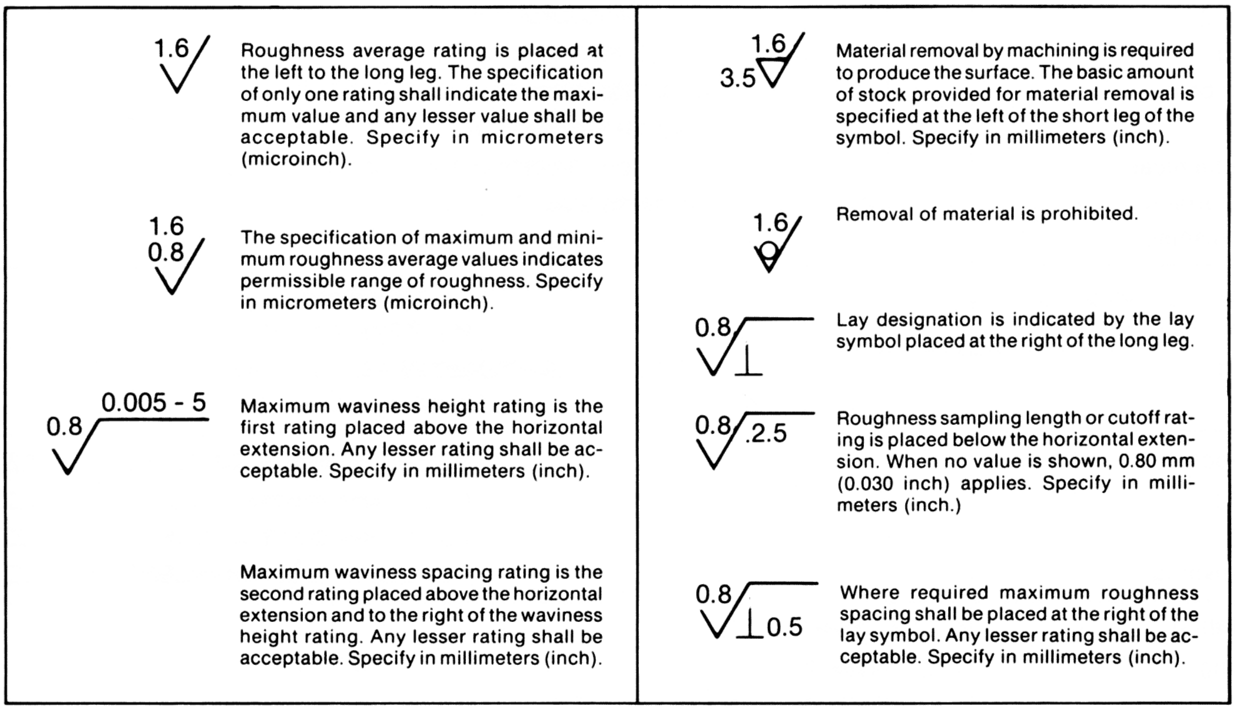


Figure 6.. Application of Surface Texture Values to Symbol

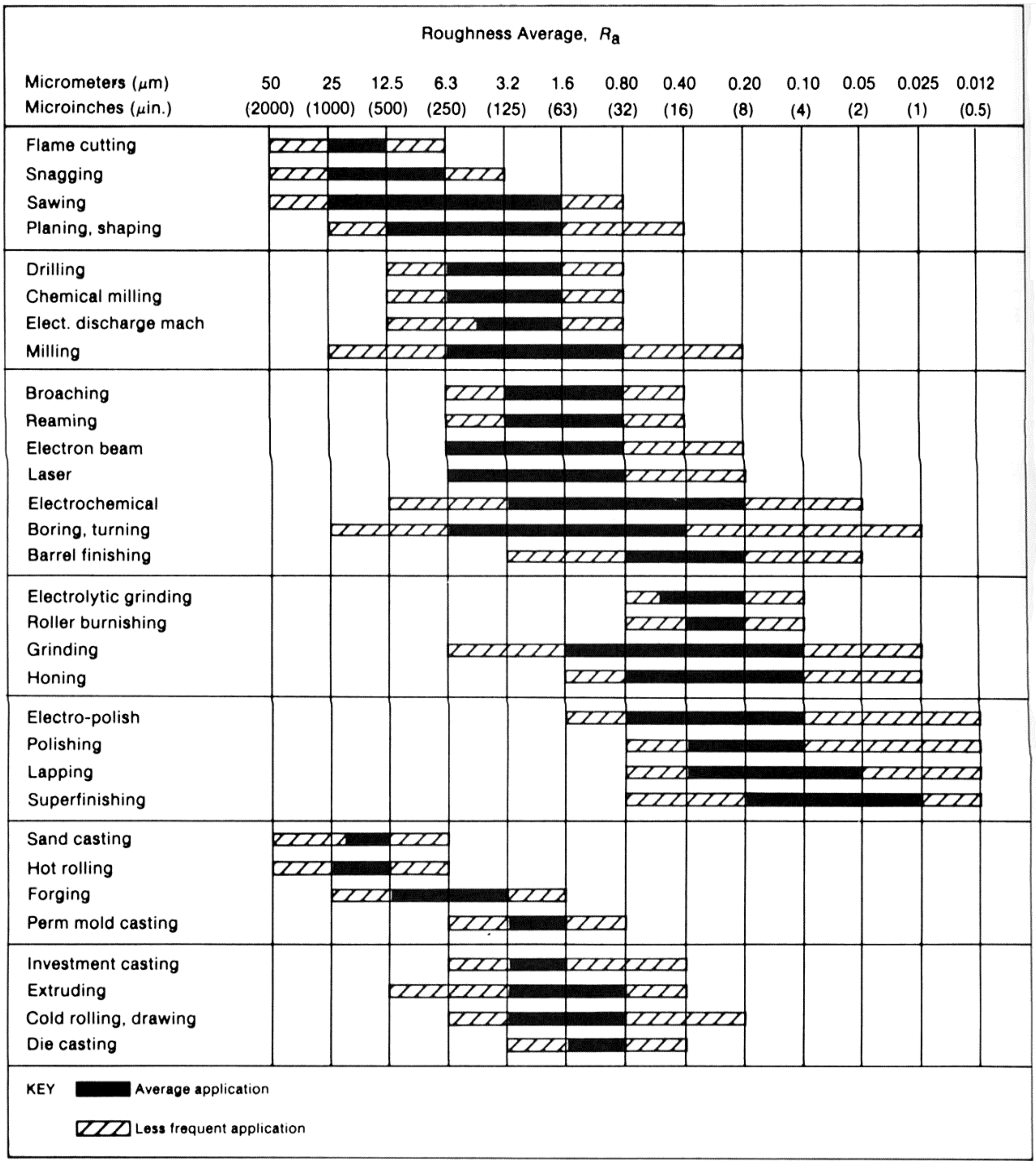
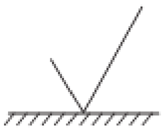


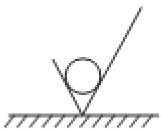
Figure 7. Surface Roughness Produced by Common Production Methods. The ranges shown are typical of the processes listed. Higher or lower values may be obtained under special conditions. ∅

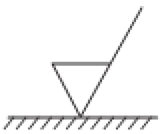
Self-Check -4	Matching Questions

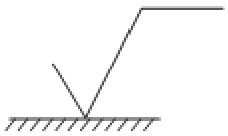
Directions: Match the symbols in Column A with their meaning under Column B (2 points each)

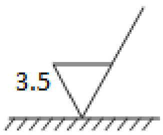
Column A | Column B

1.  A. Material removal prohibited

2.  B. Material removal by machining Is required

3.  C. Material removal Allowance,

4.  D. Surface machine

5. 

E. Special Surface Texture symbol

Note: **Satisfactory rating –above 5 points** **Unsatisfactory - below 5 points**

You can ask you teacher for the copy of the correct answers.

Answer Sheet

Score = ___________

Rating: ____________

Name: _________________________ Date: _______________

Information Sheet-5	Showing Correct Convention of parts based on ISO Standard

5.1. Conventional Representation

Certain draughting conventions are used to represent materials in section and machine elements in engineering drawings.

5.1.1. Materials

As a variety of materials are used for machine components in engineering applications, it is preferable to have different conventions of section lining to differentiate between various materials. The recommended conventions in use are shown in Fig 1.

5.1.2. **Machines**.

When the drawing of a component in its true projection involves a lot of time, its convention may be used to represent the actual component. Figure 2 shows typical examples of conventional representation of various machine components used in engineering drawing.

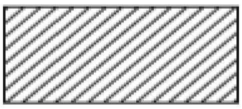
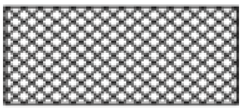
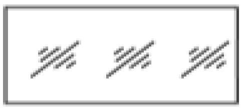
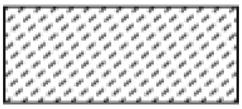
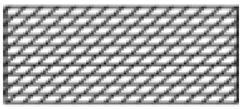
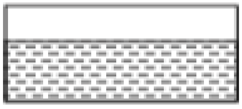
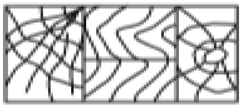
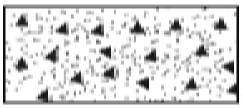
Type	Convention	Material
Metals		Steel, Cast Iron, Copper and its Alloys, Aluminium and its Alloys, etc.
		Lead, Zinc, Tin, White-metal, etc.
Glass		Glass
Packing and Insulating material		Porcelain, Stoneware, Marble, Slate, etc.
		Asbestos, Fibre, Felt, Synthetic resin products, Paper, Cork, Linoleum, Rubber, Leather, Wax, Insulating and Filling materials, etc.
Liquids		Water, Oil, Petrol, Kerosene, etc.
Wood		Wood, Plywood, etc.
Concrete		A mixture of Cement, Sand and Gravel

Figure:1 Conventional representation of materials

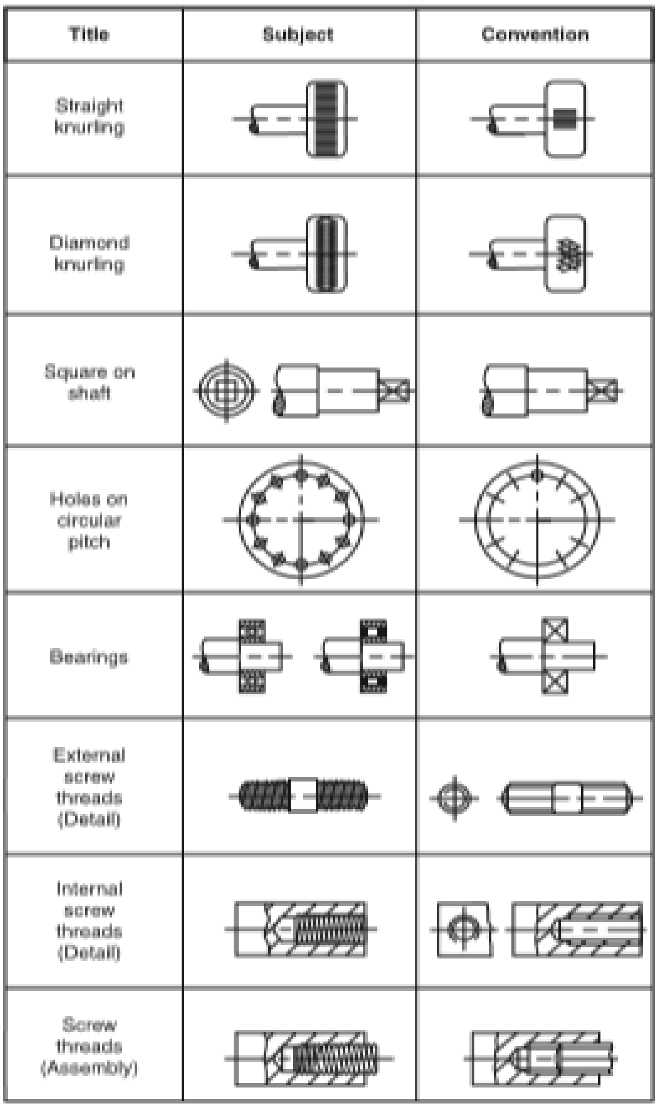
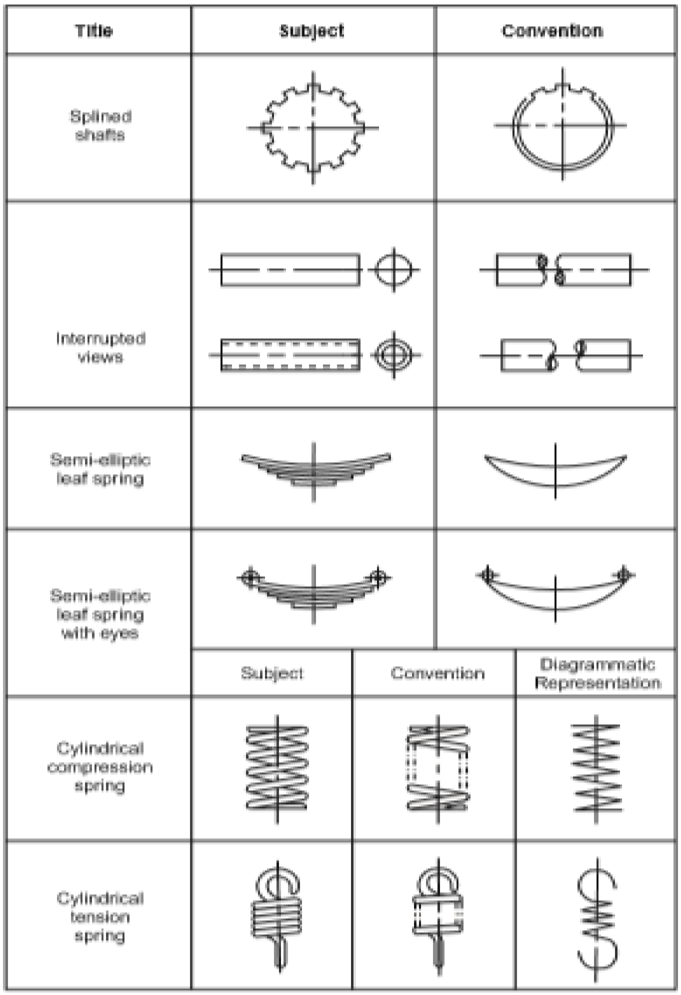


Figure. 2. Conventional representation of machine components (Contd .)



(b)

Figure 3 Conventional representation of machine components (Contd .)

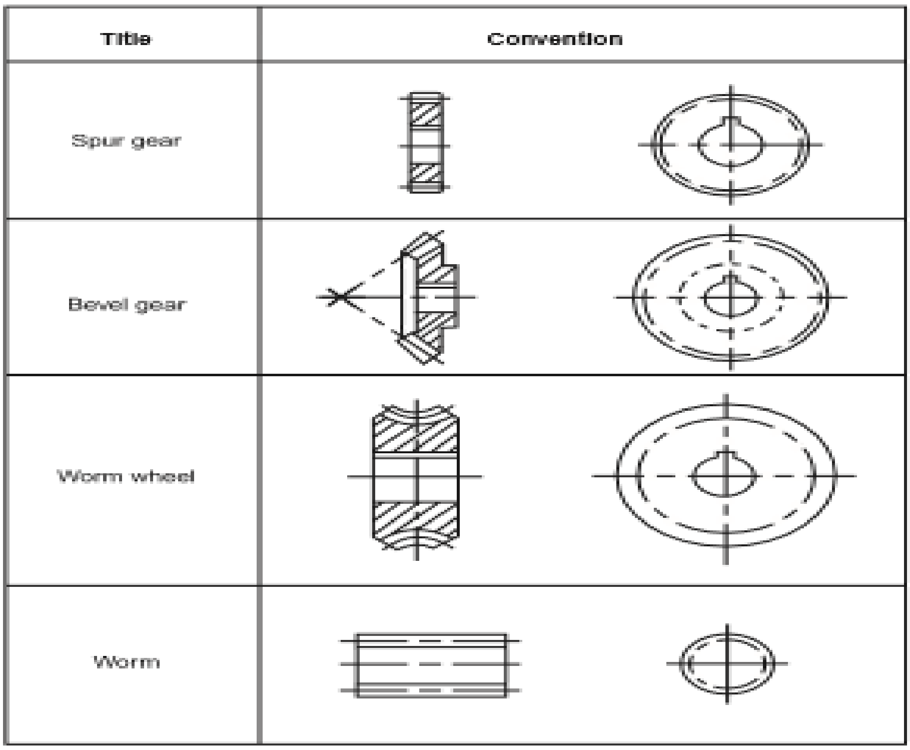
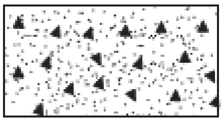


Figure 4 Conventional representations of machine components

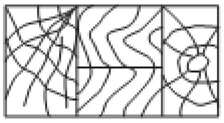
Self-Check -5	Multiple Choice

Test I: Multiple Choice Questions

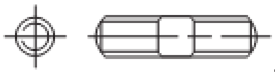
Directions: Choose the correct Answer (each question have 2. pts)

1.  For what material type this conventional representation is used?

   a. Metal b. Glass c. wood d. Concrete

2.  For what material type this conventional representation is used?

   a. Liquid b. concrete c. Wood d. Metal

3.  This conventional representation represents

   a. Bearing b. Internal screw threads c. External screw threads d. Bevel gear

***Note:* Satisfactory rating –4 points and above Unsatisfactory – below 4 points**

You can ask you teacher for the copy of the correct answers.

Name: ___________________________ Date: ________________

Answer Sheet

Score = ___________

Rating: ____________

.

6.1. Producing Section Drawings

6.1.2. Examples

Figure 1 shows the isometric view of a machine block and (i) the sectional view from the front, (ii) the view from above and (iii) the sectional view from the left.

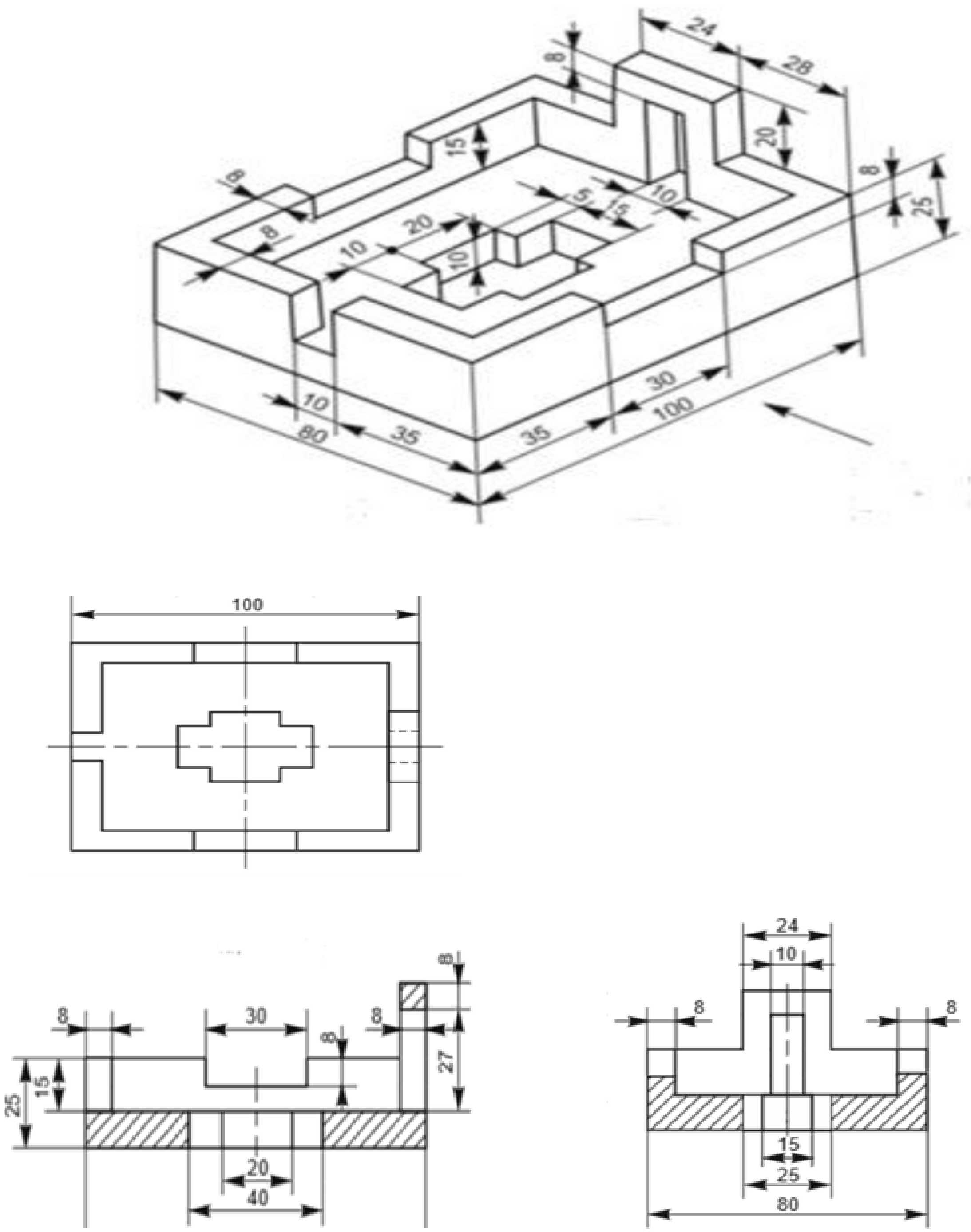


Figure 1 Machine Block

Figure 2 shows the isometric view of a shaft support. Sectional view from the front, the view from above and the view from the right are also shown in the figure.

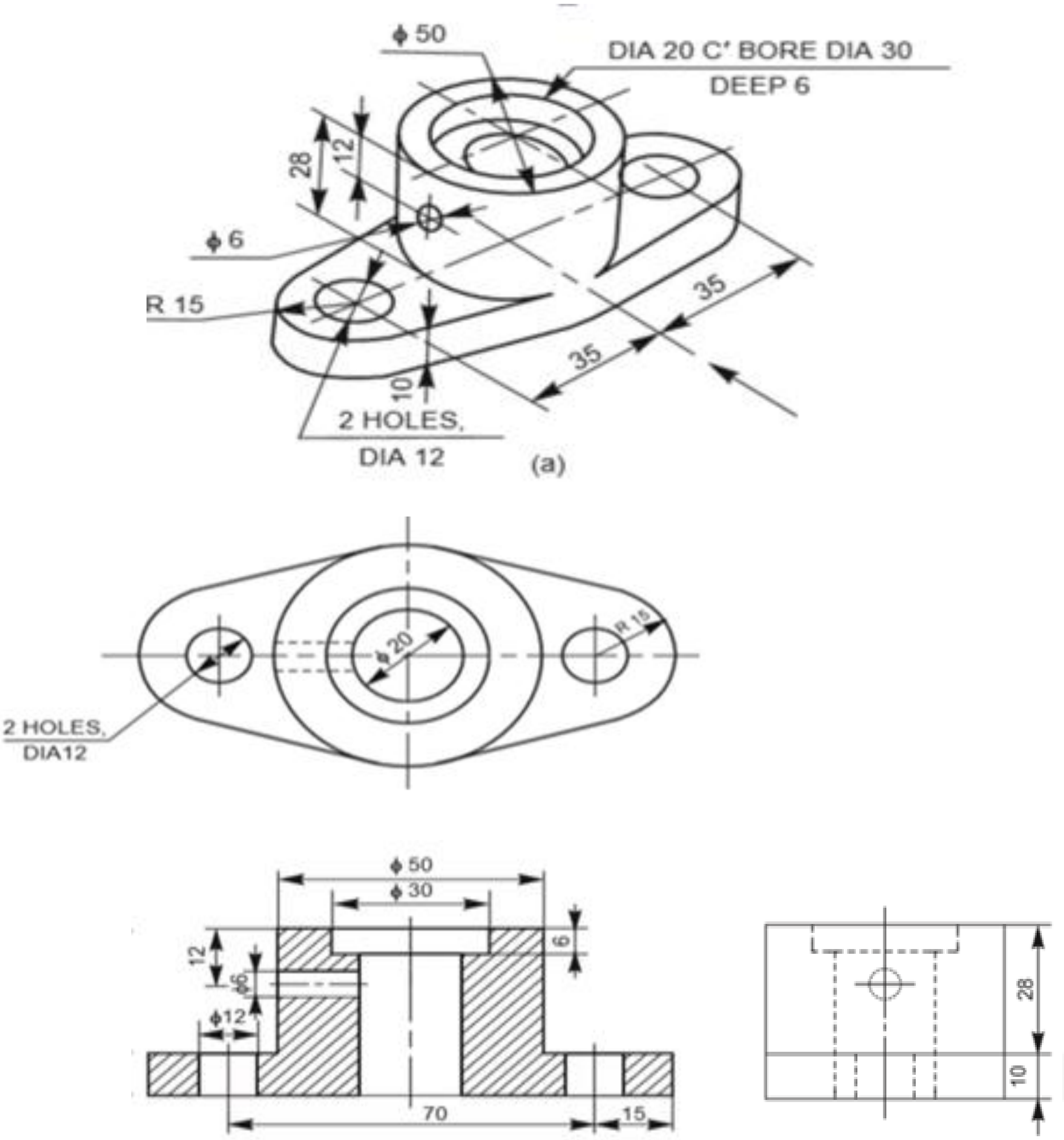


Figure 2. Shaft support

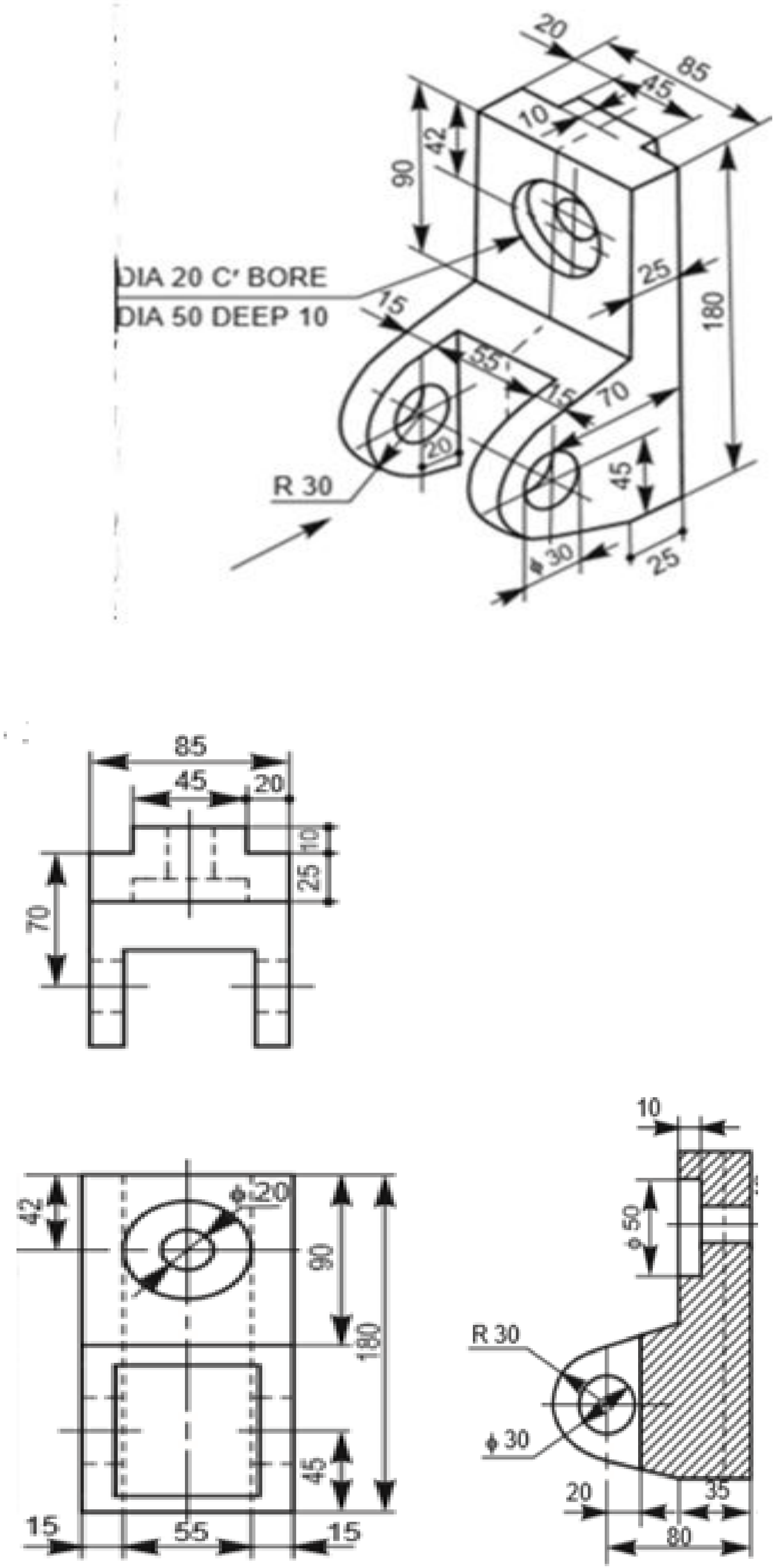


Figure 3. Sliding Block

6.1.3. Auxiliary Section

Auxiliary sections may be used to supplement the principal views used in orthographic projections. A sectional view projected on an auxiliary plane, inclined to the principal planes of projection, shows the cross-sectional shapes of features such as arms, ribs and so on. In Figure 4, auxiliary cutting plane X-X is used to obtain the auxiliary section X-X.

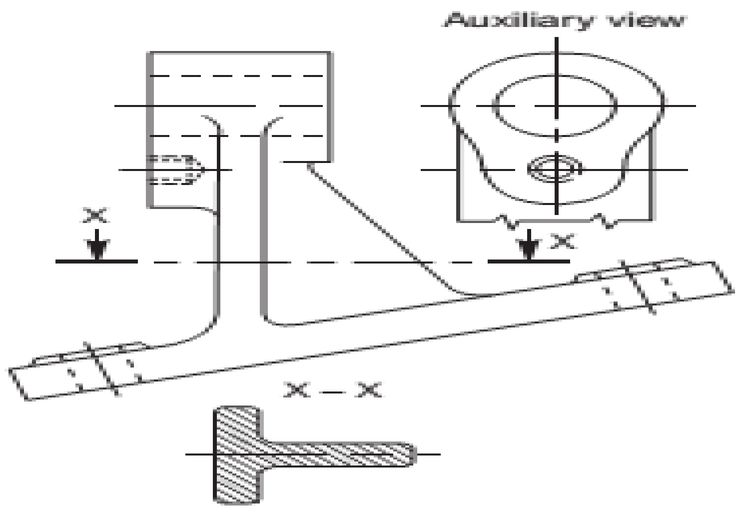
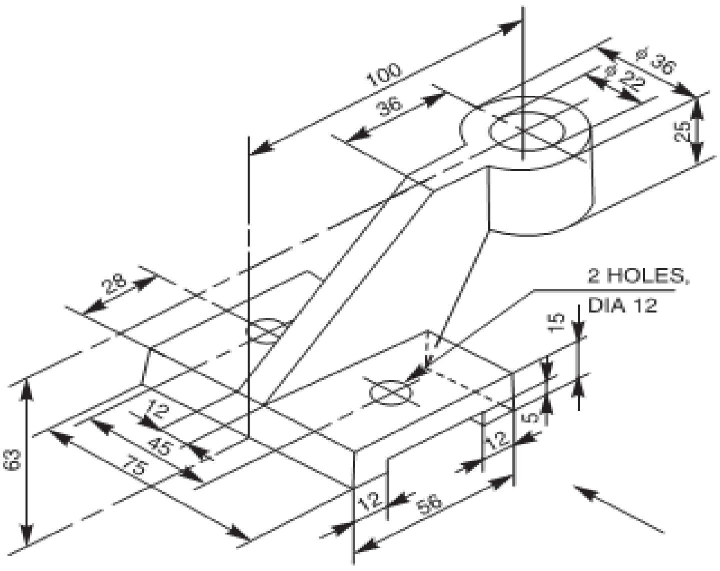


Figure 4. Auxiliary Section



(a)

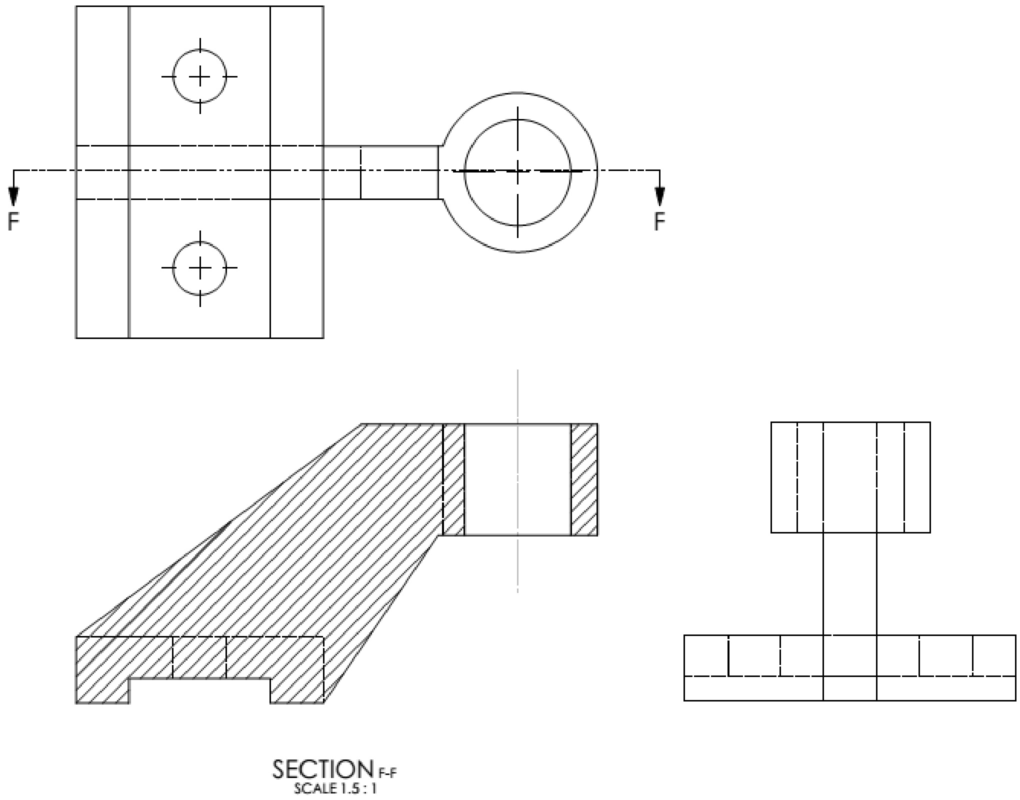


Figure 5. Section view

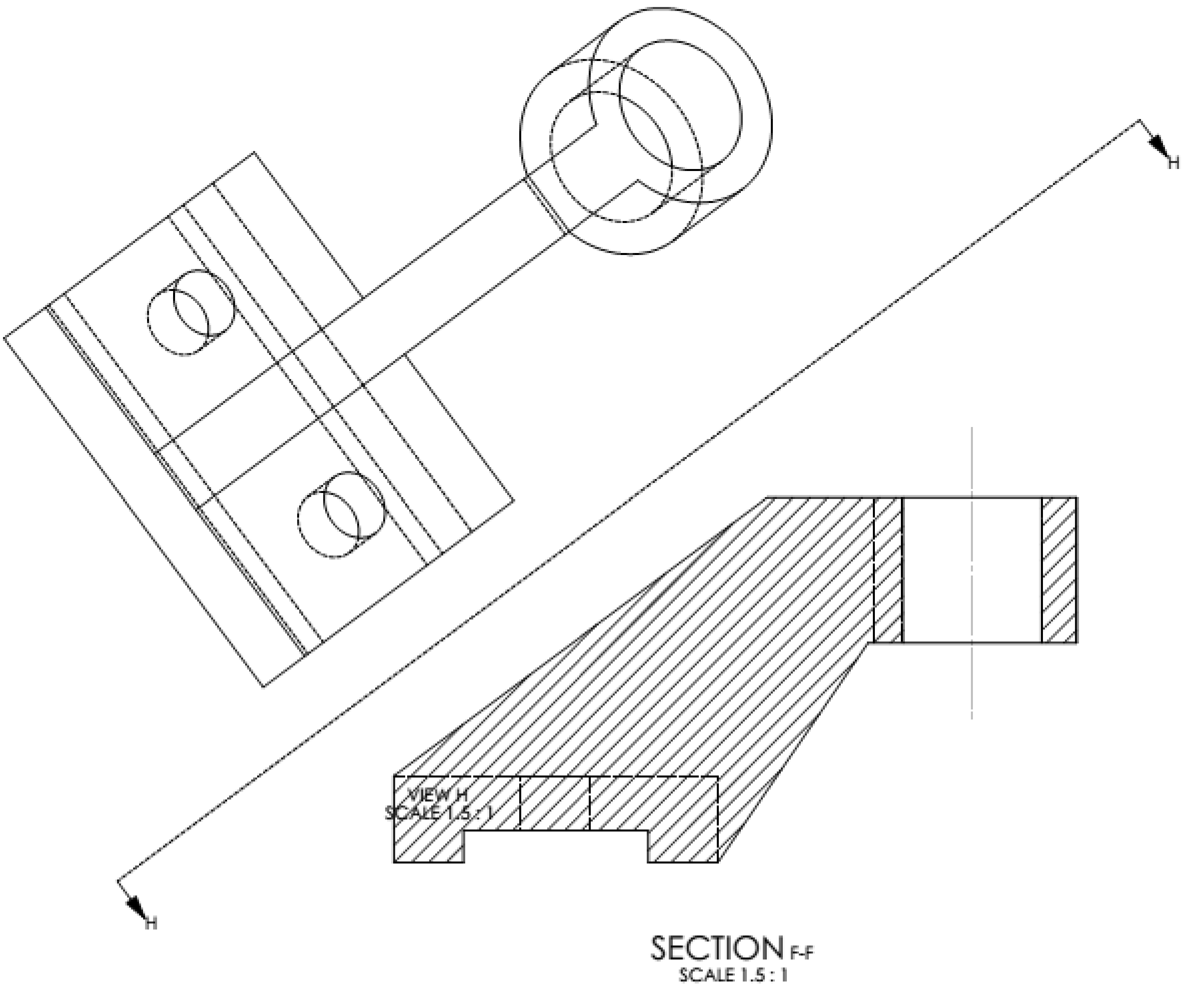


Figure 6. Auxiliary section view

6.2. Assembly Drawings

A machine is an assembly of various links or parts. It is necessary to understand the relation between the various parts of the unit for the purpose of design and production.

An assembly drawing is one which represents various parts of a machine in their working position. These drawings are classified as design assembly drawings, working assembly drawings, sub-assembly drawings, installation assembly drawings, etc. An assembly drawing made at the design stage while developing a machine is known as design assembly drawing. It is made to a larger scale so that the required changes or modifications may be thought of by the designer, keeping in view both the functional requirement and aesthetic appearance. Working assembly drawings are normally made for simple machines, comprising small number of parts. Each part is completely dimensioned to facilitate easy fabrication. A sub-assembly drawing is an assembly drawing of a group of related parts which form a part of a complicated machine. Thus, a number of such sub-assembly drawings are needed to make a complete unit. An installation assembly drawing reveals the relation between different units of a machine, giving location and dimensions of few important parts.

The final assembly drawings are prepared from design assembly drawings or from the working drawings (component drawings). The class-room exercises are designed to train the trainee to master fundamentals of machine drawing, such as principles of drawing, orthographic projections, etc. In addition, the trainee will understand the relation between the different parts of the components and working principles of the assembled unit. The following steps may be made use of to make an assembly drawing from component drawings:

1. Understand the purpose, principle of operation and field of application of the given machine. This will help in understanding the functional requirements of individual parts and their location.

2. Examine thoroughly, the external and internal features of the individual parts.

3. Choose a proper scale for the assembly drawing.

4. Estimate the overall dimensions of the views of the assembly drawing and make the outline blocks for each of the required view, leaving enough space between them, for indicating dimensions and adding required notes.

5. Draw the axes of symmetry for all the views of the assembly drawing.

6. Begin with the view from the front, by drawing first, the main parts of the machine and then adding the rest of the parts, in the sequence of assembly.

7. Project the other required views from the view from the front and complete the views.

8. Mark the location and overall dimensions and add the part numbers on the drawing.

9. Prepare the parts list.

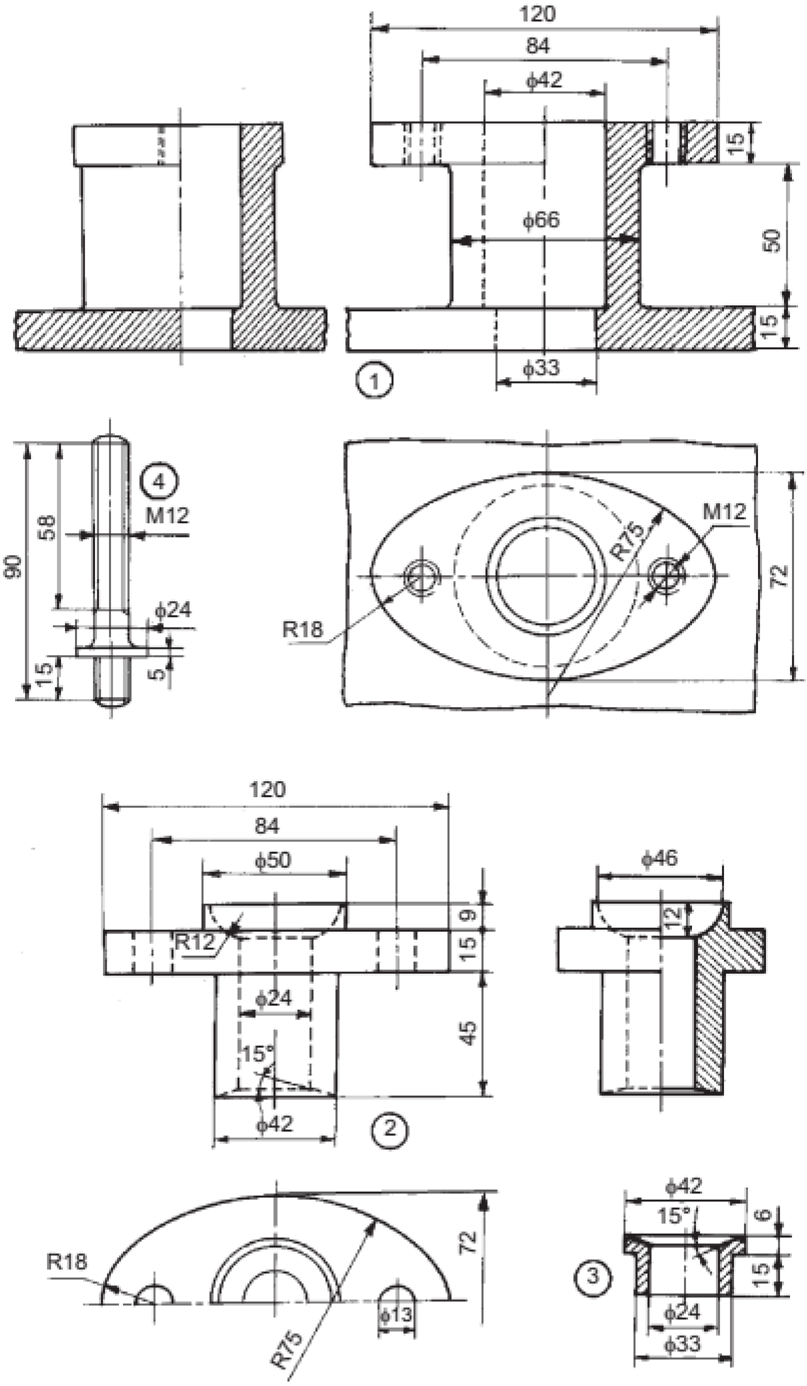
10. Add the title block.

NOTE: It is not advisable to complete one view before commencing the other. The better Method is to develop all the required views simultaneously.

Examples

Stuffing Box

It is used to prevent loss of fluid such as steam, between sliding or turning parts of machine elements. In a steam engine, when the piston rod reciprocates through the cylinder cover; stuffing box provided in the cylinder cover, prevents leakage of steam from the cylinder. Figure 8 shows the various parts of a stuffing box. At the base of stuffing box body 1, a bush 3 is placed such that the bevelled edge of the bush is at the inner side of the body. Gland 2 is placed at the other end of the body and is connected to the main body by means of studs 4 and nuts 5. The space between the reciprocating rod and the bush and the gland is packed with a packing material such as mineral fibers, leather, rubber or cork.



Parts list

Part No.	Name	Matl	Qty
1	Body	CI	1
2	Gland	Brass	1
3	Bush	Brass	1
4	Stud	MS	2
5	Nut, M12	MS	2

Figure 7: Stuffing Box

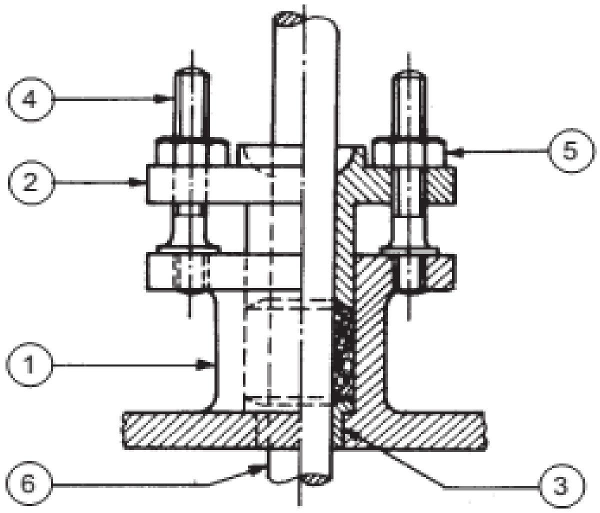


Figure 8. Stuffing Box Assembly

Example 2

A machine vice is a work holding device, used in machines such as drilling, milling, etc. A Swiveling type machine vice permits swiveling about its vertical axis, so that the work may be clamped at any angular position required in the machining operation. T-bolts (not shown) are used through the base plate, to fix the vice to the machine table. Figure 10 shows the details of a swivel machine vice. It consists of the swivel body 1 which is fixed to the base plate 3 by two bolts 6. The heads of the bolts are so shaped, that they can slide freely in the circular T-slot of the base plate. The graduations marked in degrees on the flange of the base plate, facilitate setting of the swivel body at any desired angle. The swivel body has a fixed jaw at one end. The movable jaw 2 is mounted on the swivel body by the screw 4. After the screw is inserted fully, it is held in position by a nut and pin to prevent its axial motion. Thus, when the screw is turned, the movable jaw slides on the swivel body guide ways. Steel jaw plates 5 are fitted to jaws by machine screws.

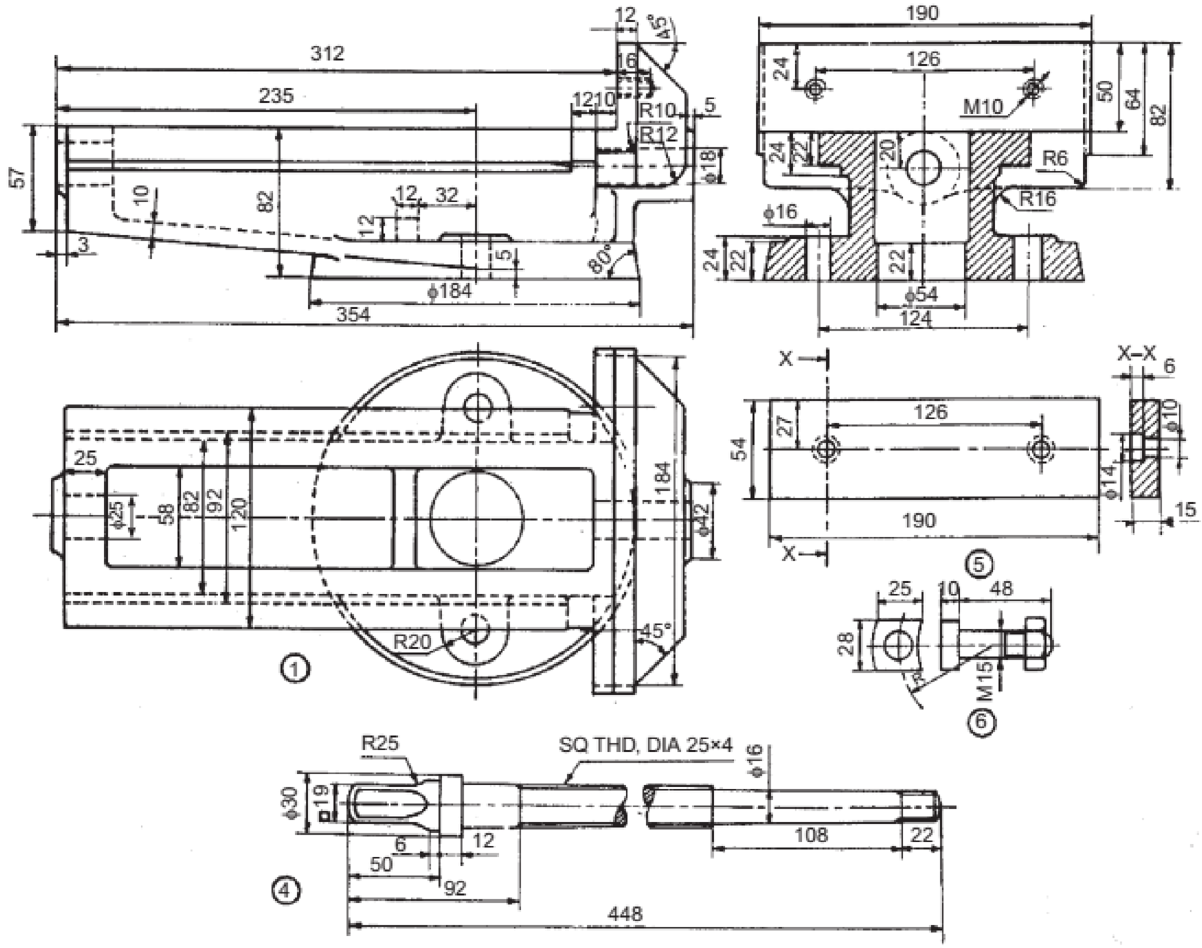


Figure 9: Machine Vice

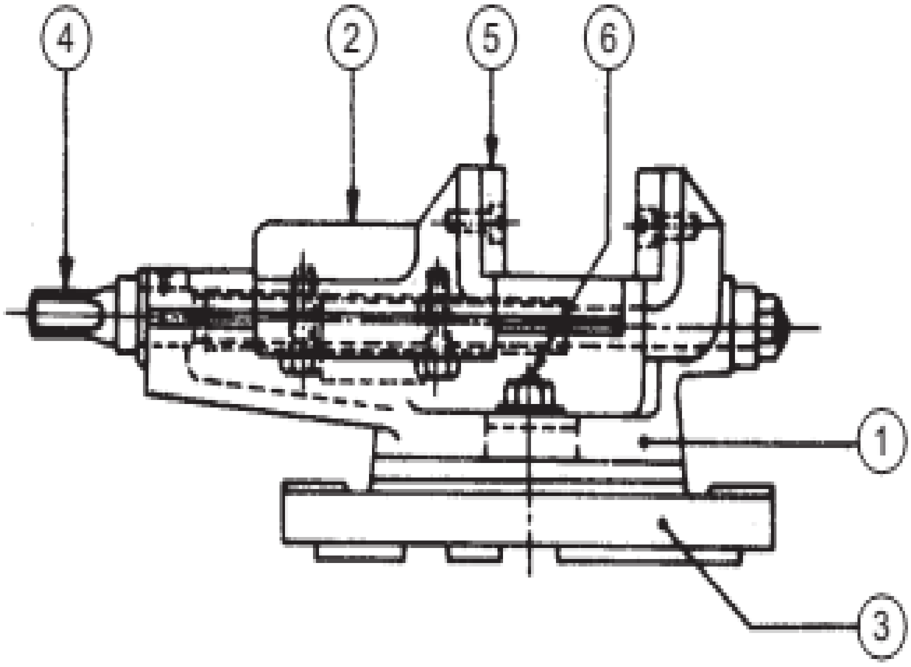


Figure 10. Machine vice Assembly

Self-Check -6	Multiple Choice

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have 3.pts)

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. What is the use Auxiliary view?
   a. To show true size of inclined surfaces
   b. To show true shape of inclined surface
   c. a & b
2. __________ is one which represents various parts of a machine in their working Position.
   a. Assembly Drawing b. Auxiliary Drawing C. Section Drawing D. All

Note: **Satisfactory rating –3 points and above Unsatisfactory – below 3 points**

You can ask you teacher for the copy of the correct answers.

Name: _________________________ Date: _______________

Answer Sheet

Score = ___________
Rating: ____________

.

7.1. ISO standard Drawing

7.1.1. Types of Lines

When you are preparing drawings, you will use different types of lines to convey information. Line characteristics (Figure 1), such as widths, breaks in the line, and zigzags, have definite meanings.

The widths of the various lines on a drawing are very important in interpreting the drawing. An ISO standard specifies that three widths of line should be used: thin, medium, and thick. As a general rule, on ink drawings, these three line widths are proportioned 1:2:4, respectively. However, the actual width of each type of line should be governed by the size and type of drawing.

The width of lines in format features (that is, title blocks and revision blocks) should be a minimum of 0.015 inch (thin lines) and 0.030 inch (thick lines). To provide contrasting divisions between elements of the format, use thick lines for borderlines, outline of principal blocks, and main divisions of blocks. Use thin lines for minor divisions of title and revision blocks and bill of materials. Use medium line widths for letters and numbers.

You cannot control the width of lines drawn with a pencil as well as the width of lines drawn with pen and ink. However, pencil lines should be opaque and of uniform width throughout their length. Cutting plane and viewing plane lines should be the thickest lines on the drawing. Lines used for outlines and other visible lines should be differentiated from hidden,

Extension, dimension, or center lines.

LINE STANDARDS			
Name	**Convention**	**Description and Application**	**Example**
Center Lines		Thin lines made up of long and short dashes alternately spaced and consistent in length. Used to indicate symmetry about an axis and location of centers.	
Visible Lines		Heavy unbroken lines Used to indicate visible edges of an object	
Hidden Lines		Medium lines with short evenly spaced dashes Used to indicate concealed edges	
Extension Lines		Thin unbroken lines Used to indicate extent of dimensions	
Dimension Lines		Thin lines terminated with arrow heads at each end Used to indicate distance measured	
Leader		Thin line terminated with arrowhead or dot at one end Used to indicate a part, dimension or other reference	
Break (Long)		Thin, solid ruled lines with freehand zigzags Used to reduce size of drawing required to delineate object and reduce detail	
Break (Short)		Thick, solid free hand lines Used to indicate a short break	
Phantom or Datum Line		Medium series of one long dash and two short dases evenly spaced ending with long dash Used to indicate alternate position of parts, repeated detail or to indicate a datum plane	
Stitch Line		Medium line of short dases evenly spaced and labeled Used to indicate stitching or sewing	
Cutting or Viewing Plane Viewing Plane Optional		Thick solid lines with arrowhead to indicate direction in which section or plane is viewed or taken	
Cutting Plane for Complex or Offset Views		Thick short dashes Used to show offset with arrowheads to show direction viewed	

Figure 1. Line characteristics and Conventions

Construction Lines

Usually the first lines that you will draw are construction lines. Use these same lines to lay out your drafting sheet; you will also use them to lay out the rest of your drawing. Line weight for construction lines is not important since they will not appear on your finished drawing. Construction lines should be heavy enough to see, but light enough to erase easily; use a 4H to 6H pencil with a sharp, conical point. With the exception of light lettering guidelines, you must erase or darken all construction lines before a drawing is reproduced.

Center Lines

Use center lines (Figure 2) to indicate the center of a circle, arc, or any symmetrical object. Compose center lines with long and short dashes, alternately and evenly spaced, with a long dash at each end. Extend center lines at least 1/4 inch outside the object. At intersecting points, draw center lines as short dashes. You may draw a very short center line as a single dash if there is no possibility of confusing it with other lines. You can also use center lines to indicate the travel of a moving center.

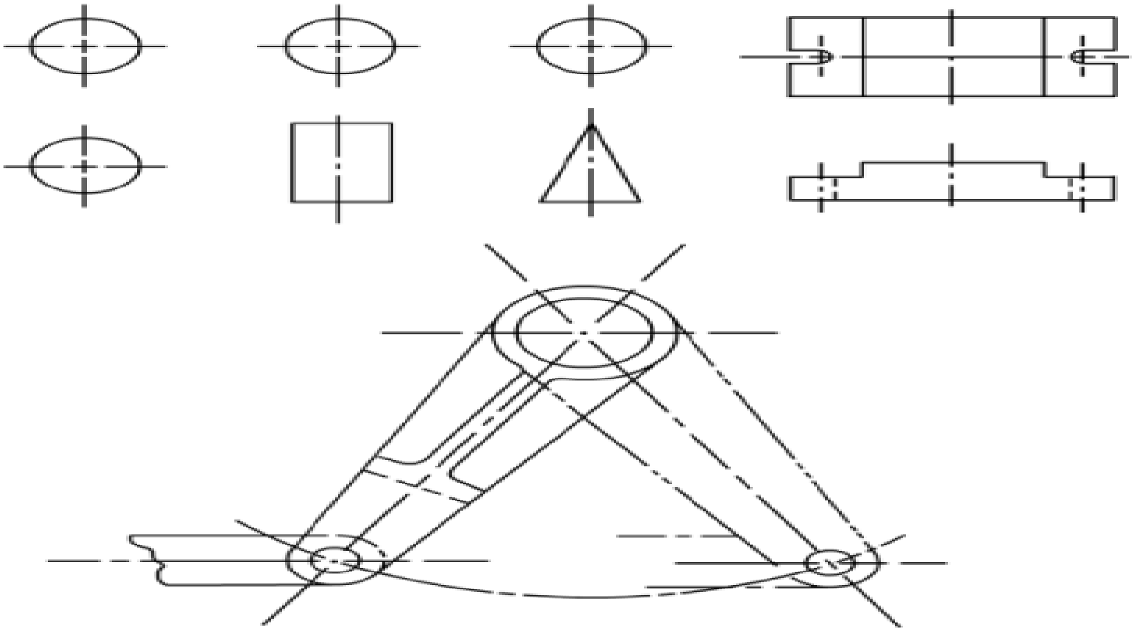


Figure 2. Use of Center line

Visible Lines

Draw the visible edge lines (Figure 3) of the View as solid, thick lines. The visible edge lines include not only the outlines of the view, but Lines defining edges that are visible within the View.

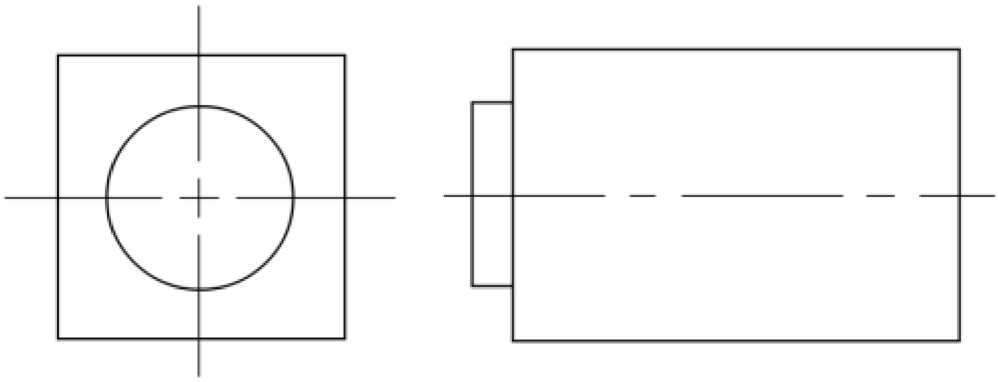


Figure 3. Use of visible edge line

Hidden Lines

Draw hidden edge lines (Figure 4) with short dashes and use them to show hidden features of an object. Begin a hidden line with a dash in contact with the line from which it starts, except when it is the continuation of an unbroken line.

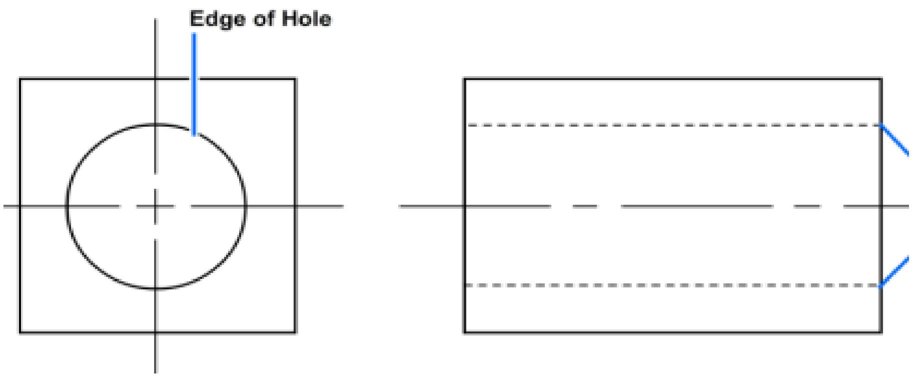


Figure 4. Hidden edge line

To prevent confusion in the interpretation of hidden edge lines, you must apply certain standard techniques in drawing these lines. A hidden edge line that is supposed to join a visible or another hidden line must actually contact the line, as shown in the upper views of Figure 5; the lower views show the incorrect procedure.

An intersection between a hidden edge line and a visible edge line is illustrated in Figure 6. Obviously, on the object itself the hidden edge line must be below the visible edge line. Indicate this face by drawing the hidden edge line as shown in the left view of Figure 6. If you drew it as indicated in the right view, the hidden edge line would appear to be above, rather than beneath, the visible edge line.

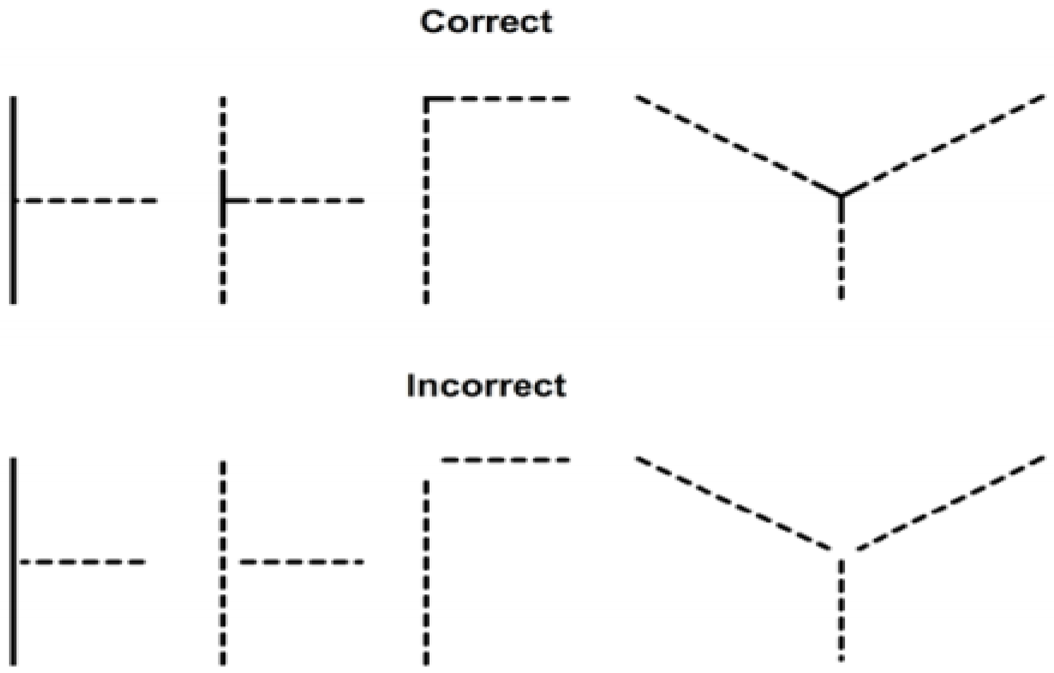


Figure 5 Correct and incorrect procedures for drawing adjoining lines.

Figure 6 Correct and incorrect procedures for drawing a hidden edge line that intersects a visible edge line.

Extension Line

Use extension lines (Figure 7) to extend dimensions beyond the outline of a view so that they can be read easily. Start these thin, unbroken lines about 1/16 inch from the outline of the object and extend them about 1/8 inch beyond the outermost dimension line. Draw extension lines parallel to each other and perpendicular to the distance you are showing. In unusual cases, you may draw the extension lines at other angles as long as their meaning is clear. As far as practical, avoid drawing extension lines directly to the outline of an object. When extension lines must cross each other, break them as shown in Figure 8.

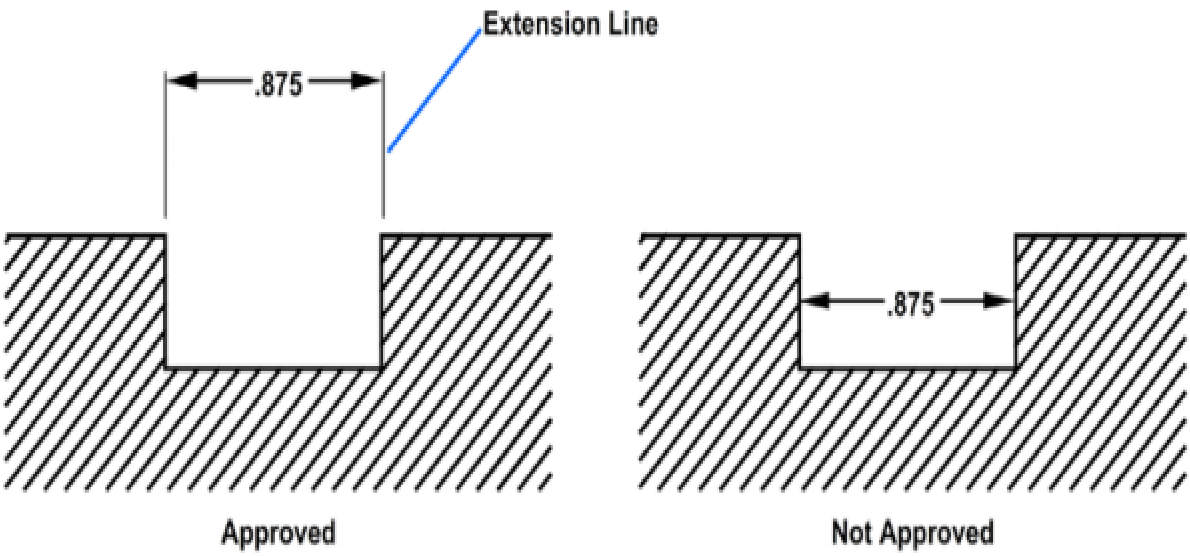


Figure 7. Use of extension lines.

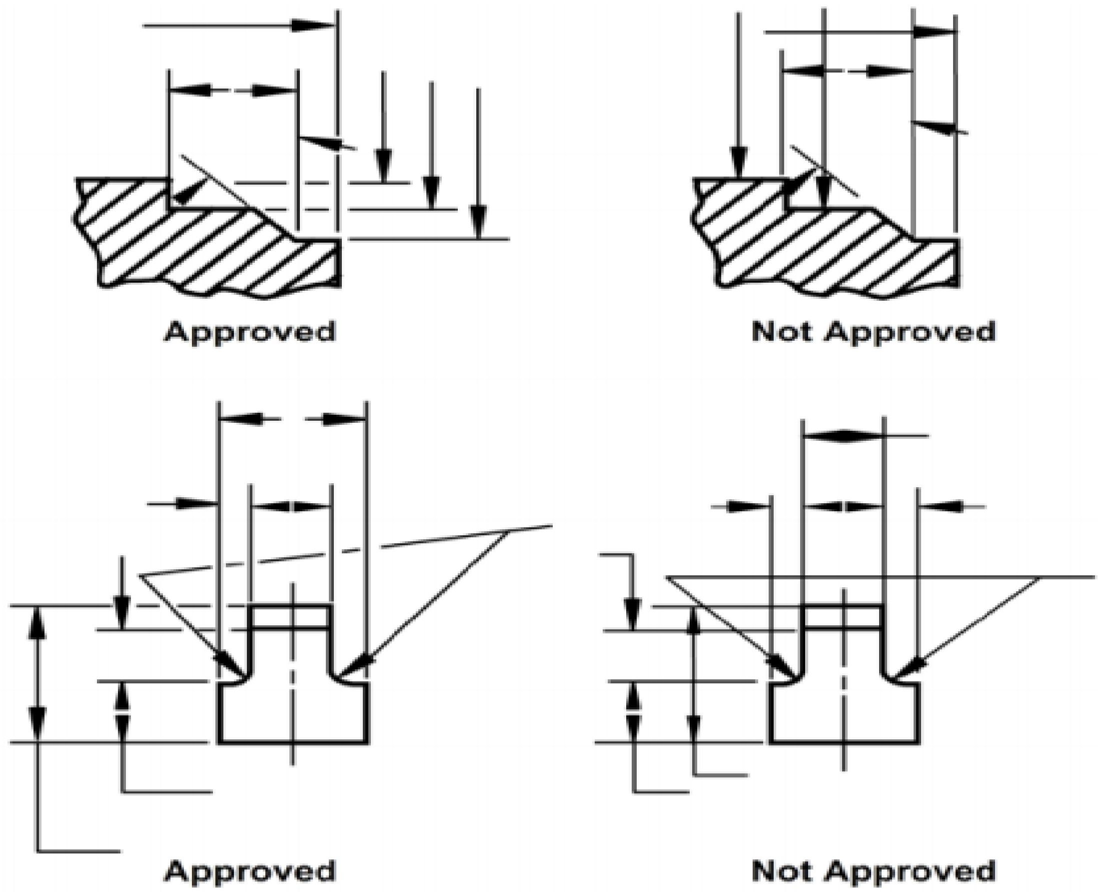


Figure 8. Breaking extension lines and leaders at points of intersections.

Dimensions

Insert a dimension line, terminating at either end in a long, pointed arrowhead (Figure 9), between each pair of extension lines. You will draw a dimension line as a thin line with a break to provide a space for the dimension numerals (except in architectural and structural drafting). Occasionally, when you need to indicate the radius of an arc, you will draw an arrow only the end of the line that touches the arc. The other end, without an arrow, terminates at the point used as the center in drawing the arc. The arrowhead on a dimension or leader line is an important detail of a drawing. If you draw these arrowheads sloppily and varied in size, your drawing will not look finished and professional. The size of the arrowhead used on a drawing may vary with the size of the drawing, but all arrowheads on a single drawing should be the same size, except occasionally when space is very restricted. The arrowheads you will use on Navy drawings are usually solid, or filled in, and are between 1/8 and 1/4 inch long, with the length about three times the spread.

Figure 9. Method of drawing an arrowhead

With a little practice, you can learn to make good arrowheads freehand. Referring to Figure 9, first define the length of the arrowhead with a short stroke as shown at A. Then draw the sides of the arrowhead as indicated at B and C. Finally, fill in the area enclosed by the lines, as shown at D.

Leaders

Use leaders to connect numbers, references, or notes to the appropriate surfaces or lines on the drawing (Figure 10). From any suitable portion of the reference, note, or number, draw a short line parallel to the lettering. From this line, draw the remainder of the leader at an angle (dog leg) to an arrowhead or dot. In this way, the leader will not be confused with other lines of the drawing. If the reference is to a line, always terminate the leader at this line with an arrowhead. However, a reference to a surface terminates with a dot within the outline of that surface.

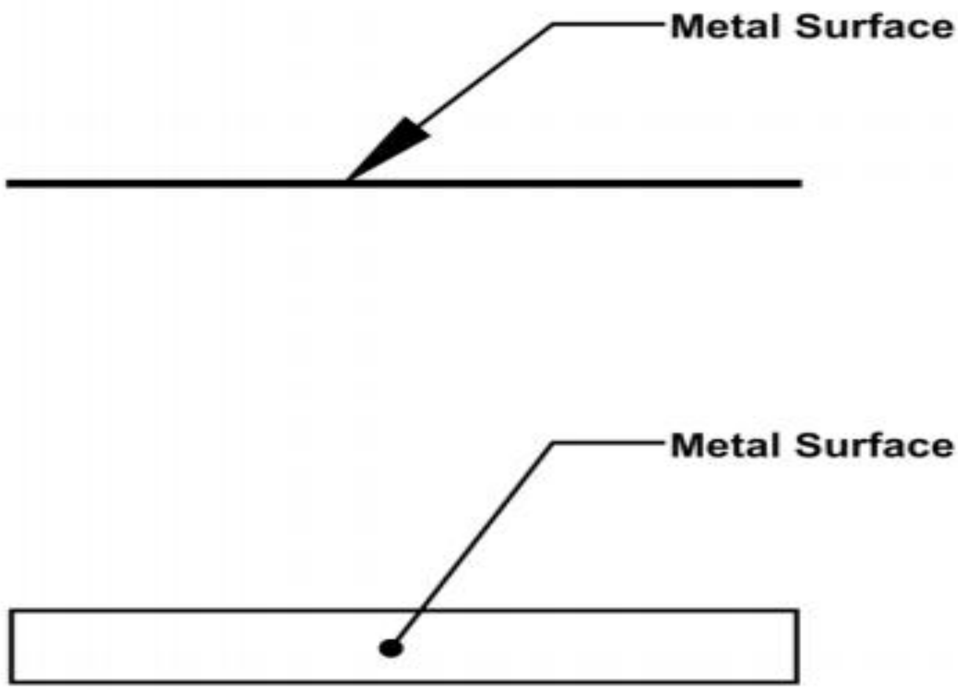


Figure 10 — Use of a leader

Phantom Lines

You will use phantom lines most frequently to indicate a moving part's alternate position, as shown in the left-hand view of Figure 11. Draw the part in one position in full lines and in the alternate position in phantom lines. You will also use phantom lines to indicate a break when the nature of the object makes the use of the conventional type of break unfeasible. The right hand view of Figure 11 shows an example of using of phantom lines. Datum

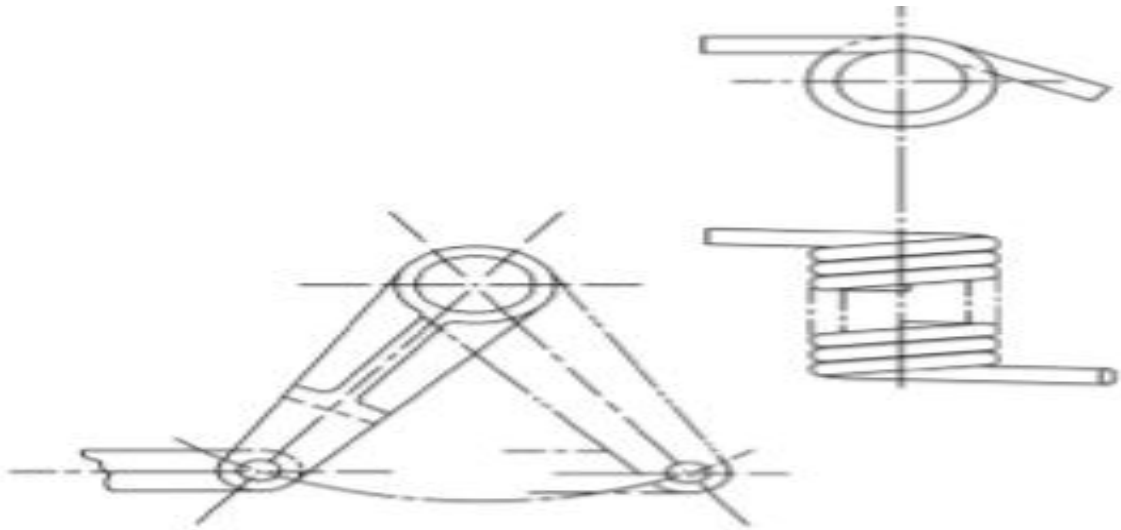


Figure 11 — Use of phantom lines.

Datum Lines Use a datum line to indicate a line or plane of reference, such as the plane from which an elevation is measured. Datum lines consist of one long dash and two short dashes equally spaced. Datum lines differ from phantom lines only in the way they are used.

Viewing or Cutting Plane Lines Use viewing plane lines to indicate the plane or planes from which a surface or several surfaces are viewed. Cutting plane lines indicate a plane or planes in which a sectional view is taken. Section views give a clearer view of interior or hidden features of an object that cannot be clearly observed in conventional outside views. Obtain a section view by cutting away part of an object to show the shape and construction at the cutting plane. Notice the cutting plane line AA in Figure 12, view A; it shows where the imaginary cut has been made. The single view in Figure 12, view B, helps you to visualize the cutting plane. The arrows point in the direction in which you are to look at the sectional view. In Figure 12, view C, a front view shows how the object looks when cut it in half. The orthographic section view of Figure 12, view D should be used on the drawing instead of the confusing front view in Figure 12, view A. Notice how much easier it is to read and understand.

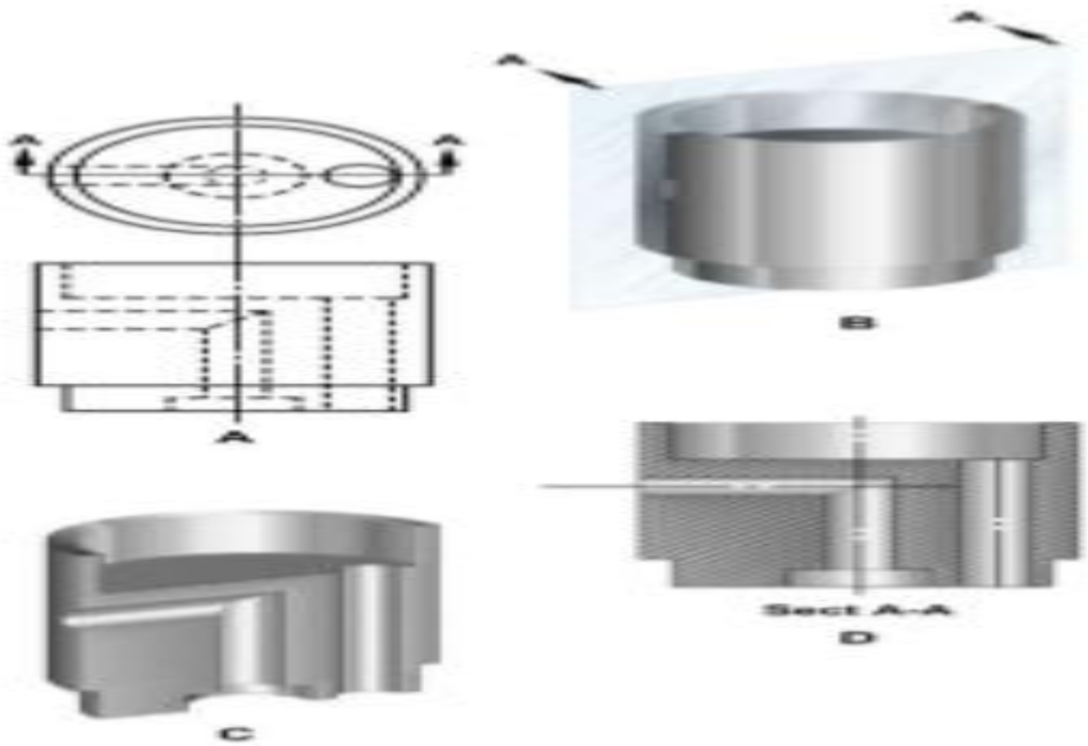


Figure 12 Action of Cutting plane

Note that hidden lines behind the plane of projection are omitted in the sectional view. These lines are omitted by general custom, because the elimination of hidden lines is the basic reason for making a sectional view. However, lines that would be visible behind the plane projection must be included in the section view.

Cutting plane lines, together with arrows and letters, make up the cutting plane indications. Placing arrows at the end of the cutting plane lines indicates the direction to view the sections. The cutting plane may be a single continuous plane, or it may be offset if the detail can be shown to better advantage. On simple views, indicate the cutting plane as shown in Figure 12, view A. On large, complex views or when the cutting planes are offset, indicate them as shown in Figure 13.

Identify all cutting plane indications with reference letters placed at the arrowhead points. When a change in direction of the cutting plane is not clear, you should place reference letters at each change of direction. When more than one sectional view appears on a drawing, alphabetically letter the cutting plane indications.

Include the letters that are part of the cutting plane indication as part of the title; for example, section A-A, section B-B, if the single alphabet is exhausted, multiples of letters may be used. You may abbreviate the word section, if desired. Place the title directly under the section drawing.

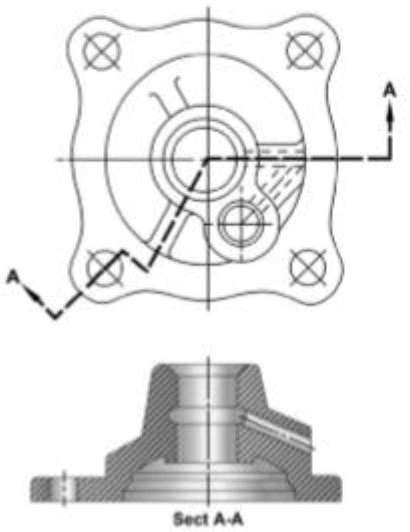


Figure 13. Use of an offset section

Section Lines

Sometimes you can best convey the technical information in a drawing by a view that represents the object as it would look if part of it were cut away. A view of this kind is called a section. The upper view of Figure 14 shows a plan view of a pipe sleeve. The lower view is a section, showing the pipe sleeve as it would look, viewed from one side, if you cut it exactly in half vertically. The surface of the imaginary cut is crosshatched with lines called section lines. According to the section lining shall be composed of uniformly spaced lines at an angle of 45 degrees to the baseline of the section. On adjacent parts, the lines shall be drawn in opposite directions. On a third part, adjacent to two other parts, the section lining shall be drawn at an angle of 30 to 60 degrees. You can use the cross-hatching shown in Figure 14 on any drawing of parts made of only one material (like machine parts, for example, which are generally made of metal). The cross-hatching is the symbol for metals and may be used for a section drawing of any type of material.

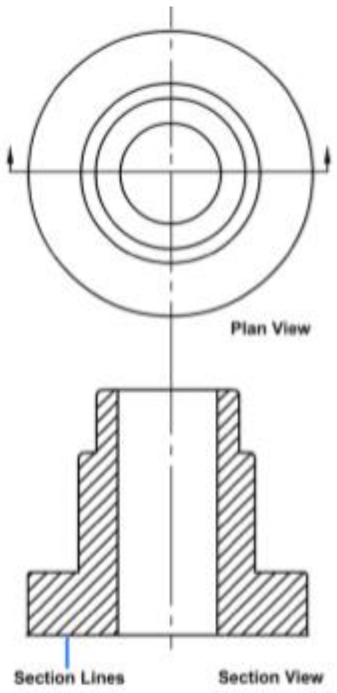


Figure 14. Drawing of a plan view and a full section

7.1.2. Order of Priority of coinciding

When two or more lines of different types coincide, the following order of priority should be observed:

- Visible outlines and edges
- Hidden outlines and edges
- Cutting planes
- Centre lines and lines of symmetry
- Centroidal lines
- Projection lines

7.2. Application of lines

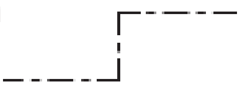
Line	*Description*	*General Applications*
A	Continuous thick	A1 Visible outlines
B	Continuous thin (straight or curved)	B1 Imaginary lines of intersection B2 Dimension lines B3 Projection lines B4 Leader lines B5 Hatching lines B6 Outlines of revolved sections in place B7 Short centre lines
C D 	Continuous thin, free-hand Continuous thin (straight) with zigzags	C1 Limits of partial or interrupted views and sections, if the limit is not a chain thin D1 Line (see Fig. 2.5)
E	Dashed thick	E1 Hidden outlines
G	Chain thin	G1 Centre lines G2 Lines of symmetry G3 Trajectories
H 	Chain thin, thick at ends and changes of direction	H1 Cutting planes
J	Chain thick	J1 Indication of lines or surfaces to which a special requirement applies
K	Chain thin, double-dashed	K1 Outlines of adjacent parts K2 Alternative and extreme positions of movable parts K3 Centroidal lines

Table 2. Application of Line

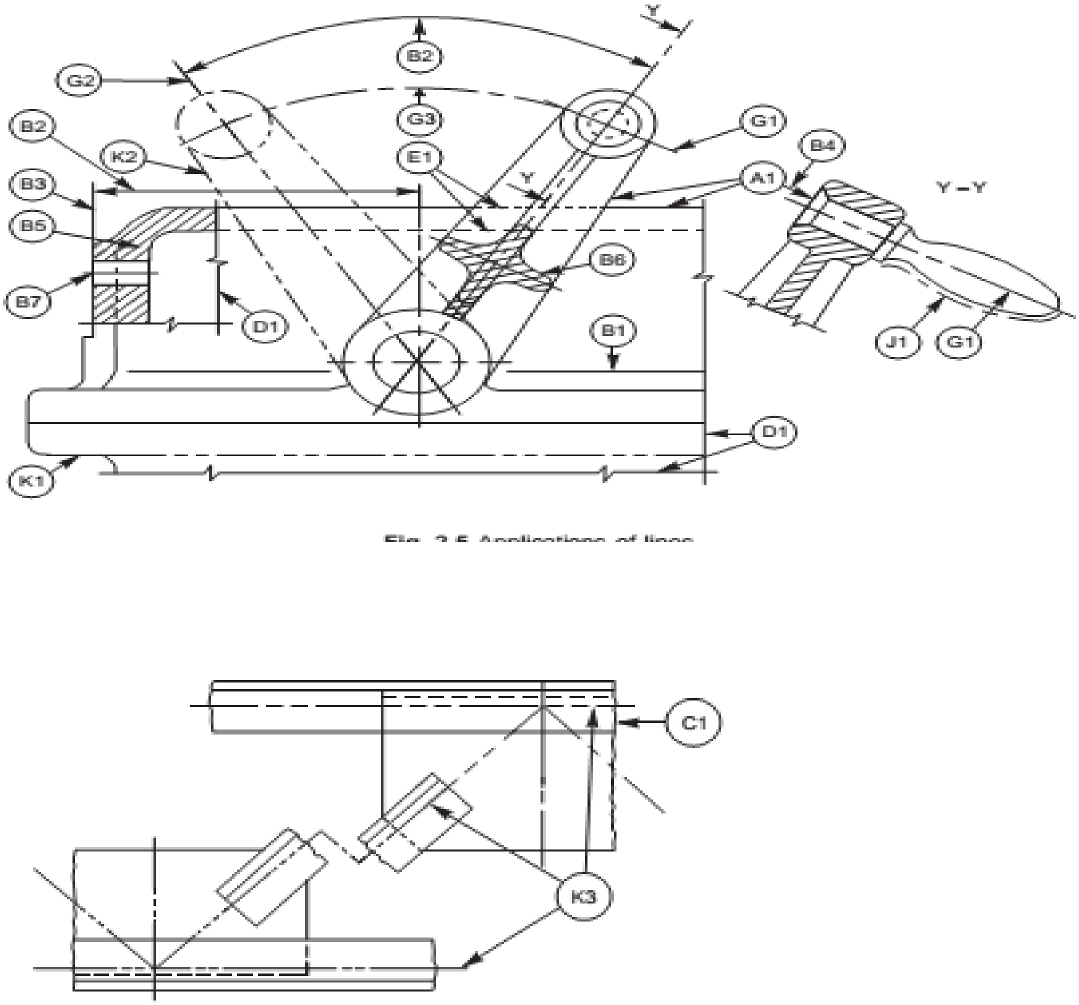


Figure 15. Application of Lines

Drawing Template and Title block

Drawing Templates

Drawing templates are a key component of the drawing system. They specify styles and layers available for such items as lines and hatched regions, in addition to specifying default properties for the current drawing elements.

Included are several predefined drawing templates conforming to ANSI English, ANSI Metric, and ISO drawing border standards. However, these templates contain only a minimum of defined named styles. If desired, you can specify an existing template to be used as a default each time a new drawing is created, or you can define a custom drawing environment as a new template.

Standard layouts of drawing sheets are specified by the various standards organizations.

This is the layout of a typical sheet, showing the drawing frame, the microfilm camera alignment marks, a typical title block, parts list and revision table:

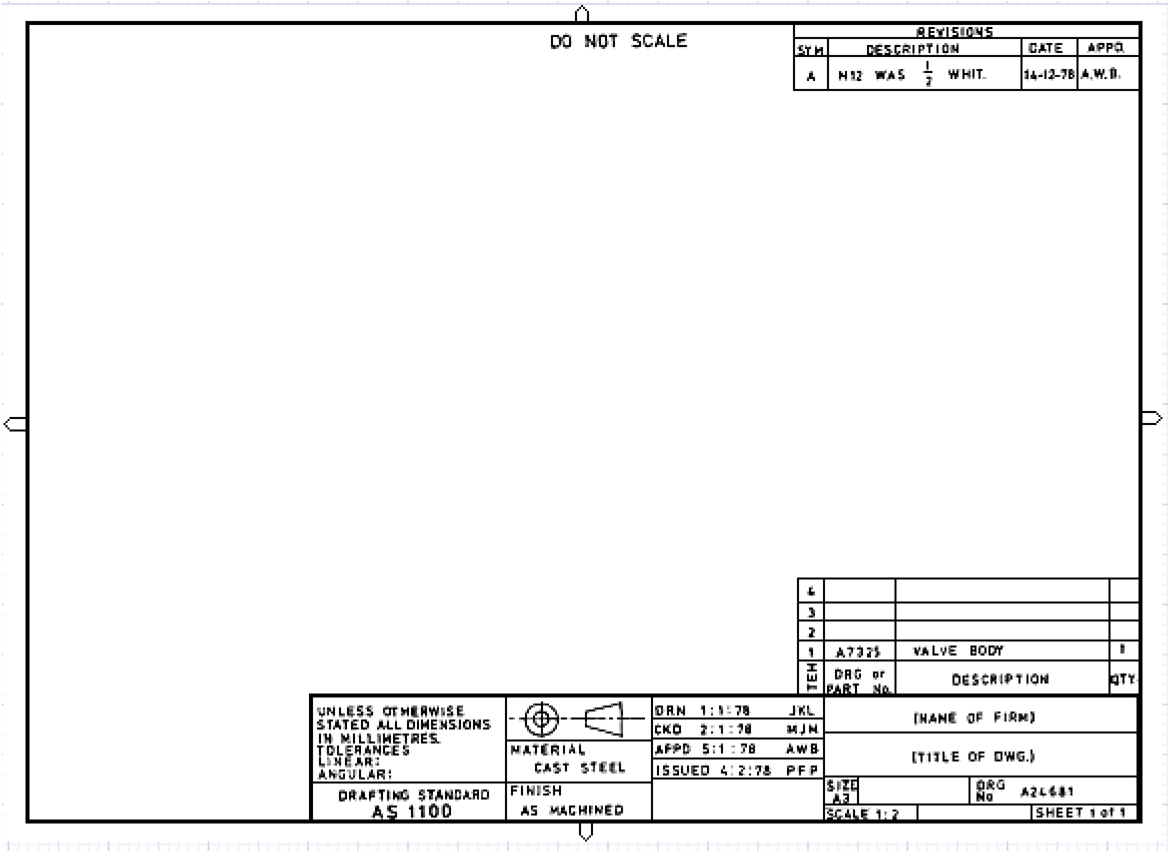


Figure 16. Title Block

ISO Standard Paper Sizes

The ISO 216 paper sizing system is used in most of the world, replacing traditional paper sheet sizes such as the 8.5 by 11 inch size familiar in the U.S

In brief, international paper sizes are in three series, designated A*n*, B*n*, and C*n*. Increasing the number *n* by 1 halves the area of the sheet, so that, for example, an A5 sheet is an A4 sheet cut in half. The basic sheet A0 has an area of 1 square meter, so an A4 sheet (the standard size for business letters) has an area of 1/16 square meter. The ratio between the height and width of a sheet is always the square root of 2 (about 1.414).

The area of a B*n* sheet is the area of the A*n* sheet multiplied by the square root of 2, so a B5 sheet, for example, is intermediate in size between an A4 and an A5 sheet. The C*n* size, intended mostly for envelopes, has an area equal to the fourth root of 2 (about 1.189) times the area of the A*n* sheet, which means that an A*n* sheet fits nicely, unfolded, in a C*n* envelope.

The dimensions of the sheets are computed from these formulas:

Format	Width [m]	Height [m]
A*n*	$2^{-1/4-n/2}$	$2^{1/4-n/2}$
B*n*	$2^{-n/2}$	$2^{1/2-n/2}$
C*n*	$2^{-1/8-n/2}$	$2^{3/8-n/2}$

With round off, the dimensions (in millimeters) are as follows:

A Series Formats		B Series Formats		C Series Formats	
4A0	1682 × 2378	–	–	–	–
2A0	1189 × 1682	–	–	–	–
A0	841 × 1189	B0	1000 × 1414	C0	917 × 1297
A1	594 × 841	B1	707 × 1000	C1	648 × 917
A2	420 × 594	B2	500 × 707	C2	458 × 648
A3	297 × 420	B3	353 × 500	C3	324 × 458
A4	**210 × 297**	B4	250 × 353	C4	229 × 324
A5	148 × 210	B5	176 × 250	C5	162 × 229
A6	105 × 148	B6	125 × 176	C6	114 × 162
A7	74 × 105	B7	88 × 125	C7	81 × 114
A8	52 × 74	B8	62 × 88	C8	57 × 81
A9	37 × 52	B9	44 × 62	C9	40 × 57
A10	26 × 37	B10	31 × 44	C10	28 × 40

Note: To convert the dimensions to inches, divide by 25.4. Thus an A4 sheet measures 8.27 by 11.69 inches, making it a little taller and narrower than an 8.5 by 11 inch sheet.

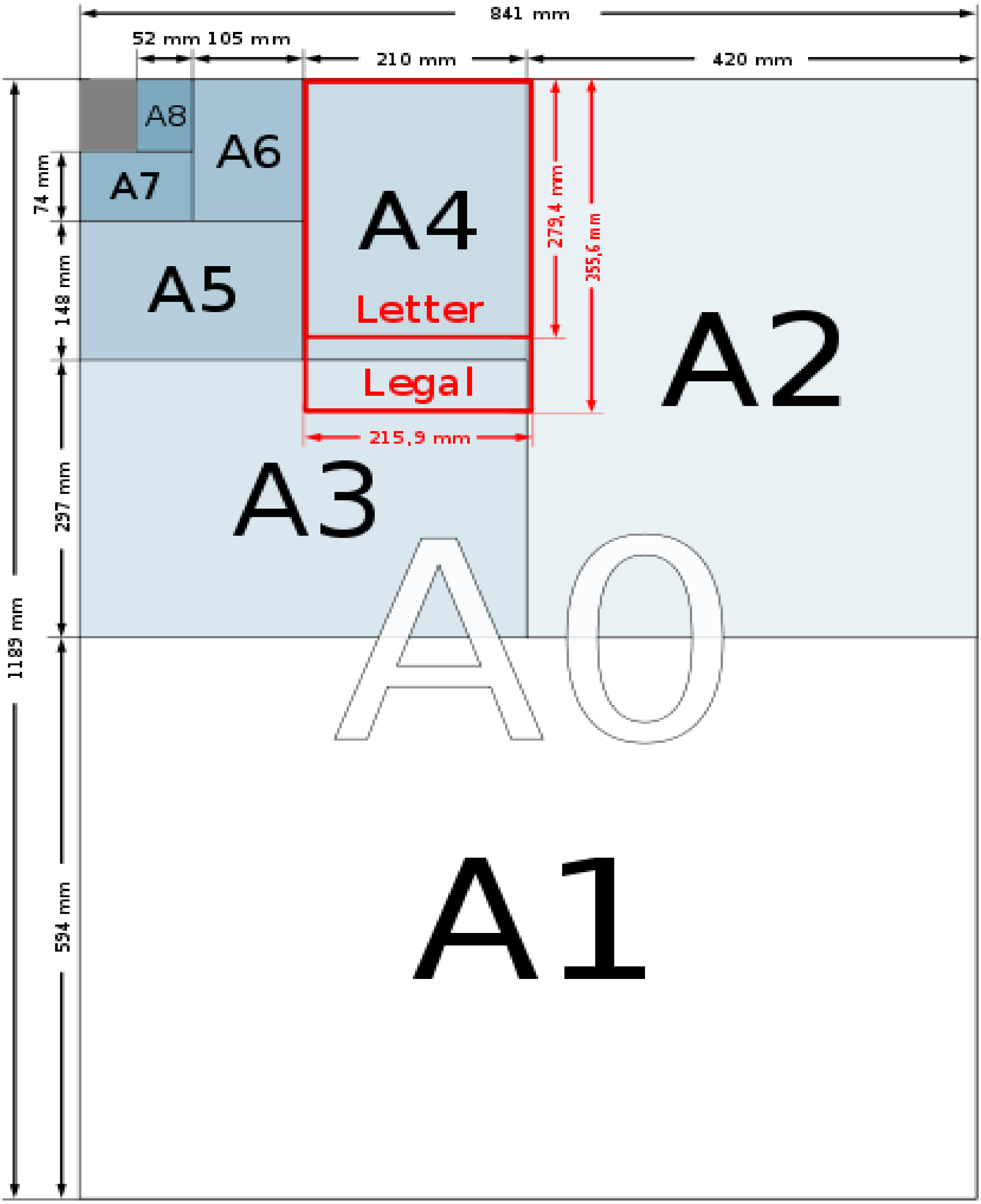


Figure 17. ISO "A series" used in most of the world

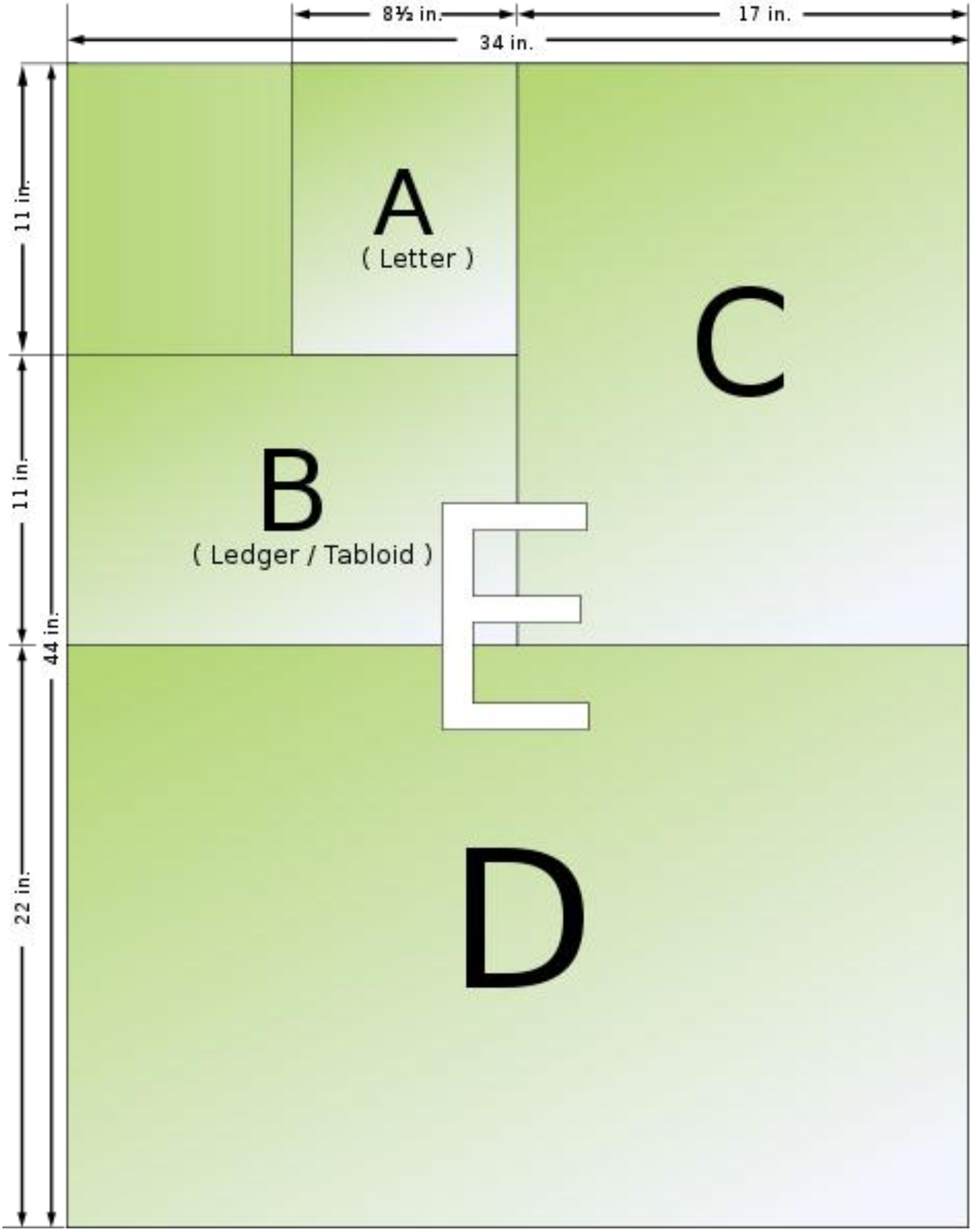


Figure 18 .North American paper sizes

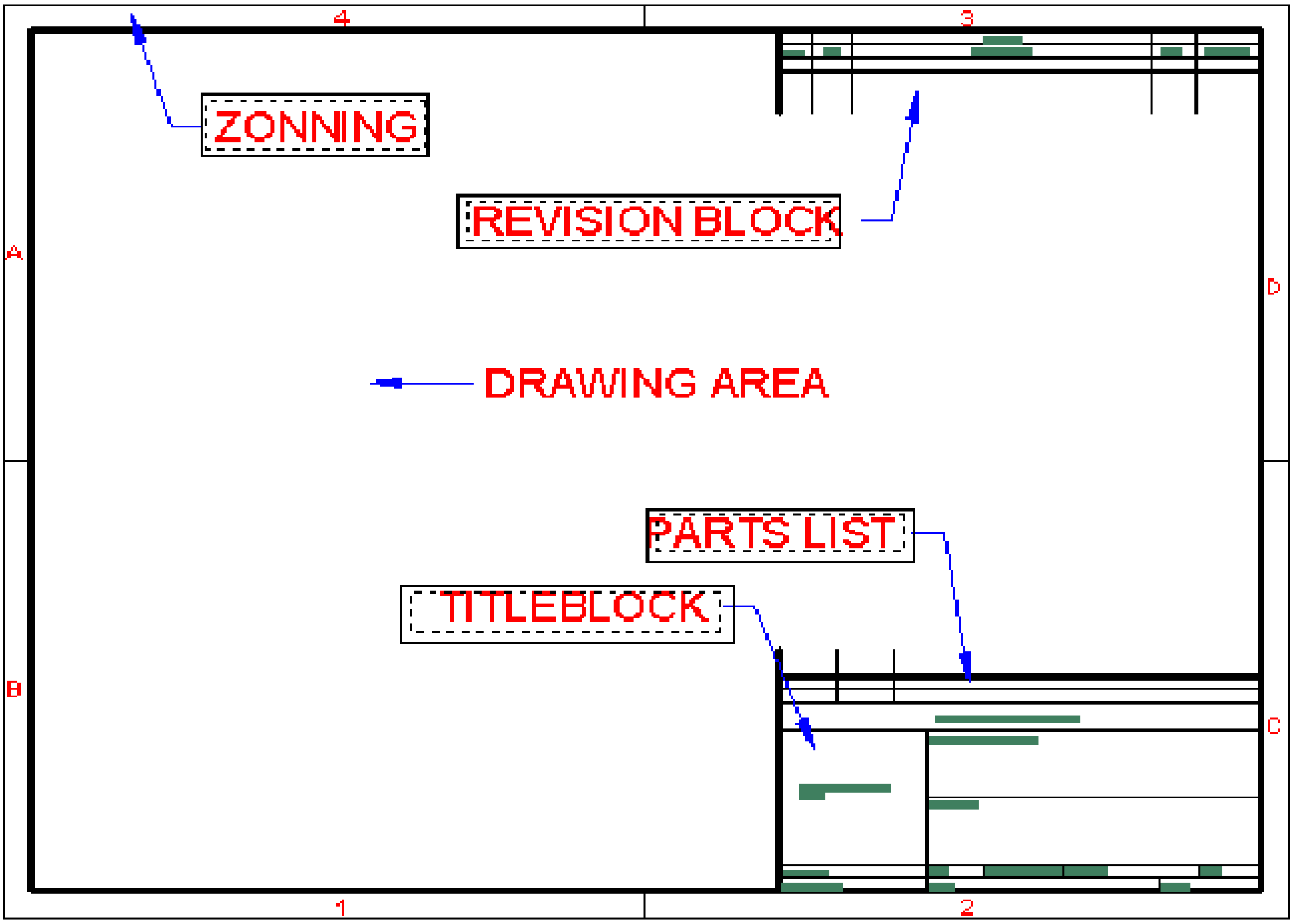


Figure19. Above is an example of a standard sheet with notation of its parts.

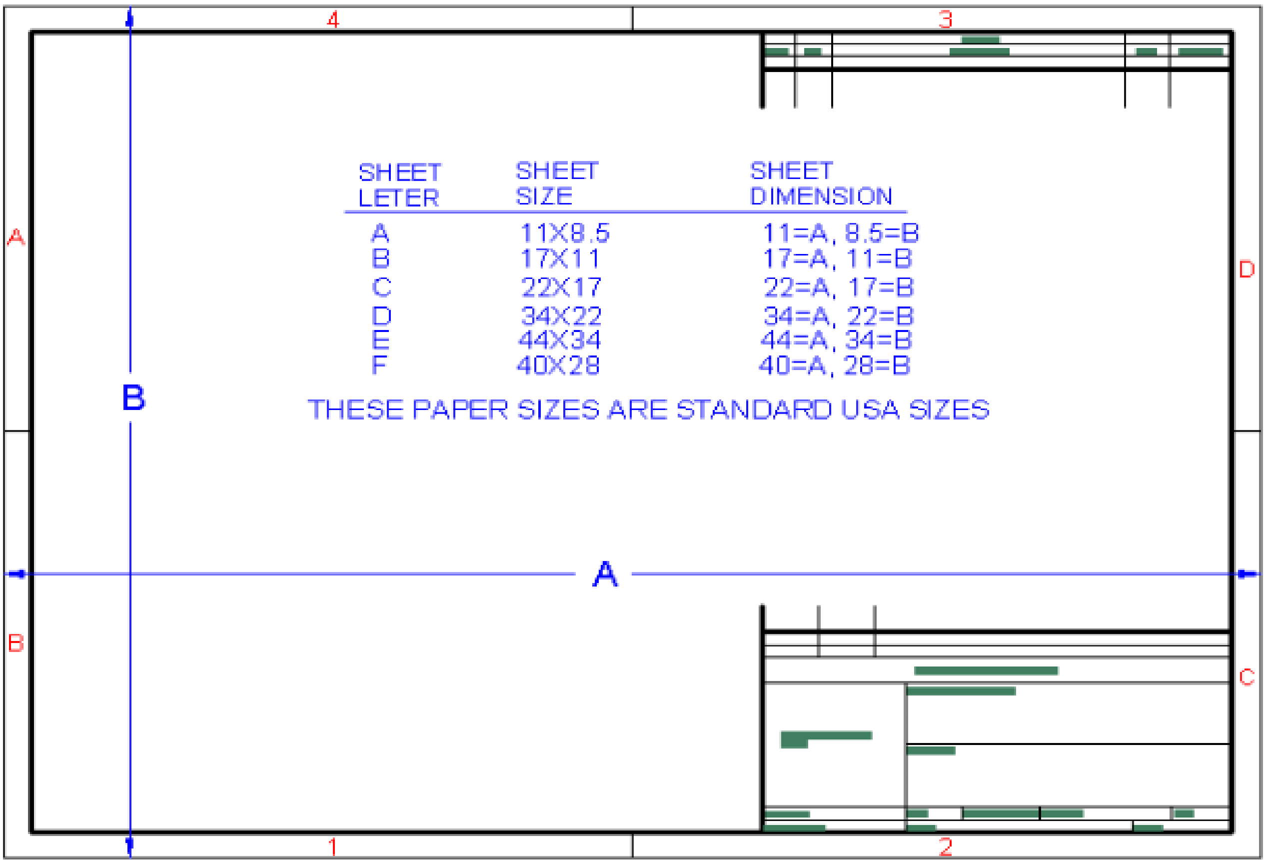
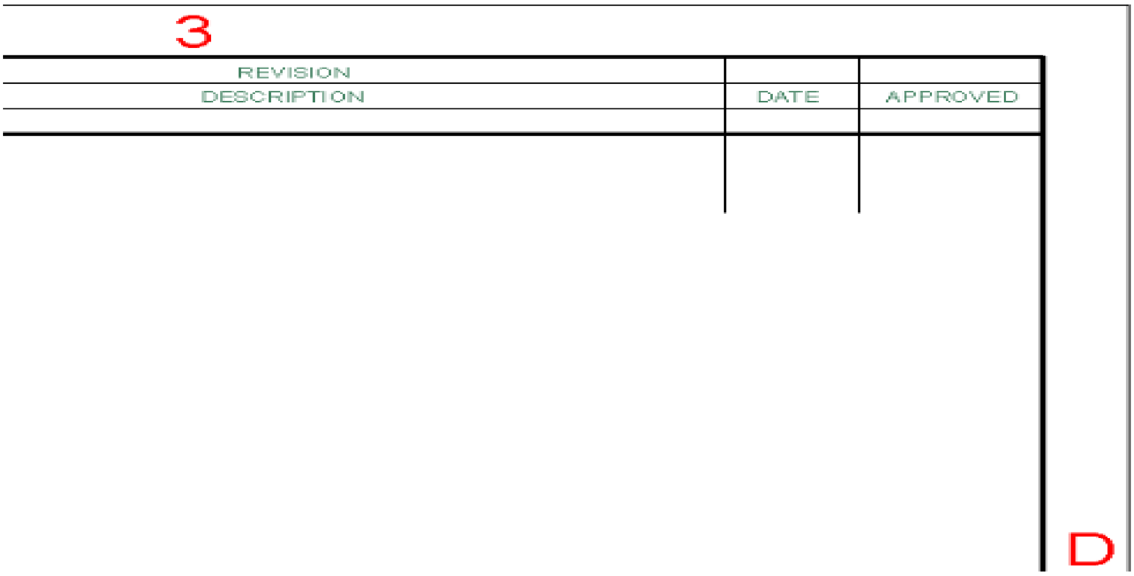
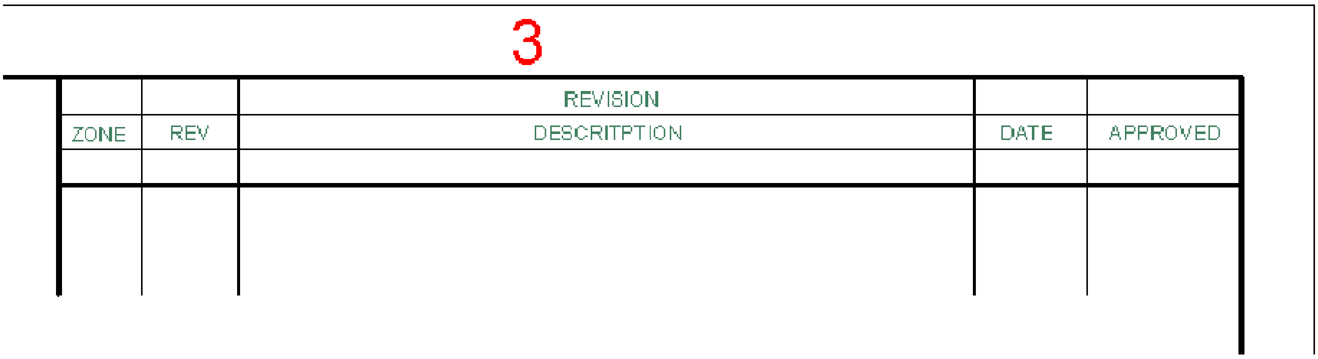


Figure 20 . Above is an example of a standard sheet with notation on its dimension

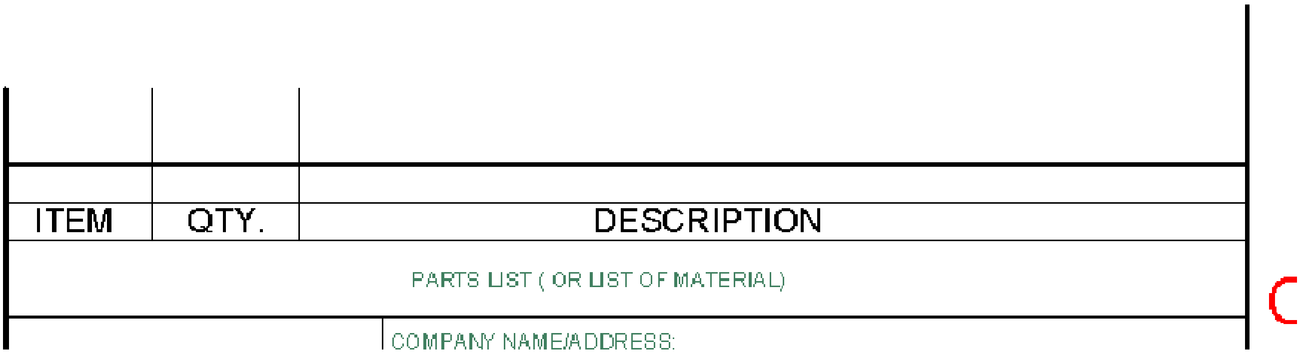


The process of **zoning** in a drawing is referring to finding the location of a specific part, usually used in an assembly drawing where there are many parts. This process is kind like reading a map. A drawing may be divided up into a grid using letters and numbers. When zoning is used it is located inside the drawing frame.

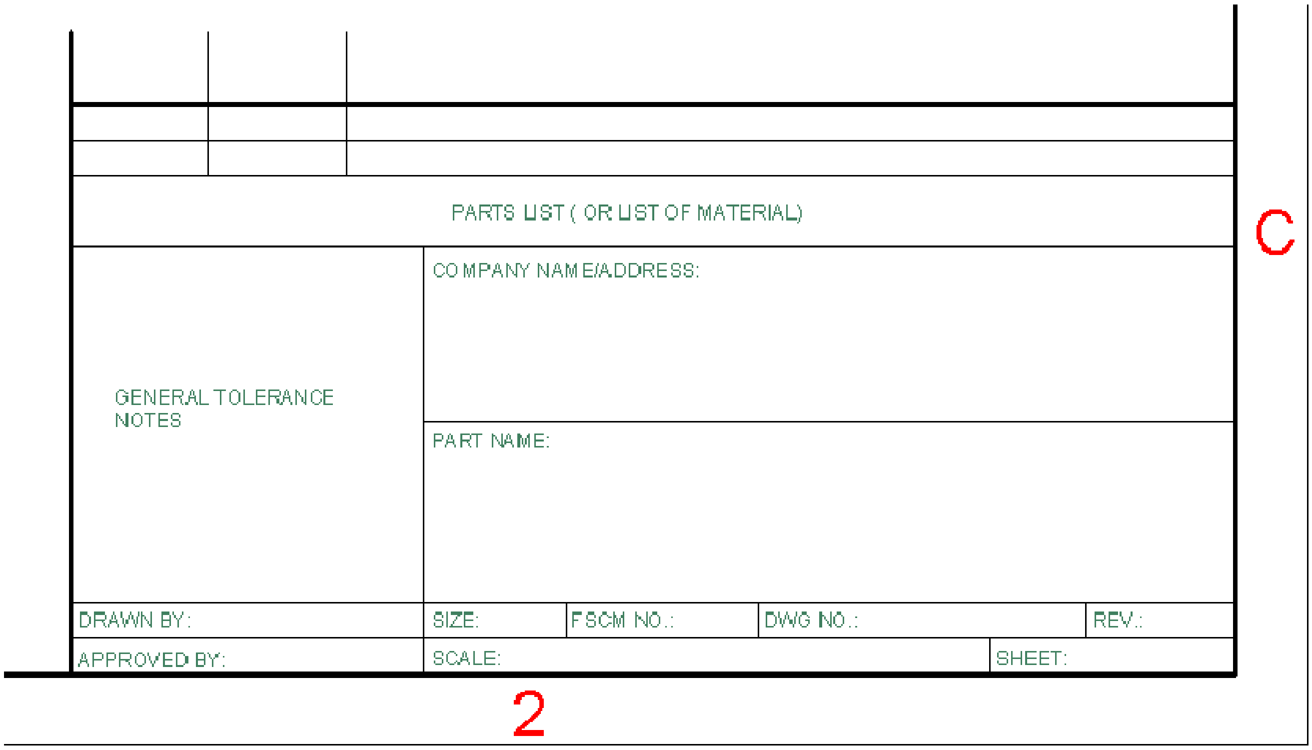
Zoning allows easy references to various parts of the drawing by referencing a coordinate such as **C7**.



The **revision block** or sometimes called the change block. This is record of changes made to the original drawing. Drawing revision are made to improve the design, reduce cost, clarify instruction, change dimension, correct errors, etc. A typical revision block may contain: Zone location, Revision number or letter, Description of change, The date the revision was made, The person approving the revision. All modifications to the drawing should be documented here.



A **Material list** or **Parts list** or sometimes called a bill of materials. This list is used primarily on assembly type drawings that show more than a single part. Some of the more common Materials list May contain such things as the: Item number, The quantity of items required, Description of the part, And other information.



Some important information that may be contained in the **title block** are: Company name, Part name, Who the drawing was drawn by, who the drawing was approved by, scale of the drawing, Drawing number, what is the revision number, how many sheet's it took to create this drawing

Title Blocks

The Title block is a boxed area containing general information about the part in the drawing. The main purpose of the title block is that it contains important text information about the part such as company name, drawing number, part number and other pertinent information. Different companies may have some what different formats for their title blocks, but most of the time the title block is located in the lower right corner of the drawing sheet.

Drawing Title Blocks

Standards

BS ISO 7200 Technical Drawings- Title Blocks identifies the title block requirements to be used on engineering drawings.... The drawing sheet size should be in accordance with "BS EN ISO 5457 TD- Sizes and layout of drawing sheets".

Notes

A title block is the form on which the actual drawing is a section. The title block includes the border and the various sections for providing quality, administrative and technical information. The importance of the title block cannot be minimized as it includes all the information which enables the drawing to be interpreted, identified and archived.

The title should include sufficient information to identify the type of drawing e.g. general arrangement, or detail. It should also clearly describe in a precise way what the drawing portrays

The notes below relate to the title boxes included on in the title block to convey the necessary information. The standard drawing sizes and layouts are described elsewhere.

The basic requirements for a title block located at the bottom right hand corner of a drawing are

1. The registration or ID number
2. The drawing title
3. The Legal Owner of the Drawing

These items should be written in a rectangle which is at the most 170mm wide.

The tile block should also include boxes for the legal signatures of the originator and other persons involved production of the drawing to the required quality.

In other forms of title block , the title block contains the following information:

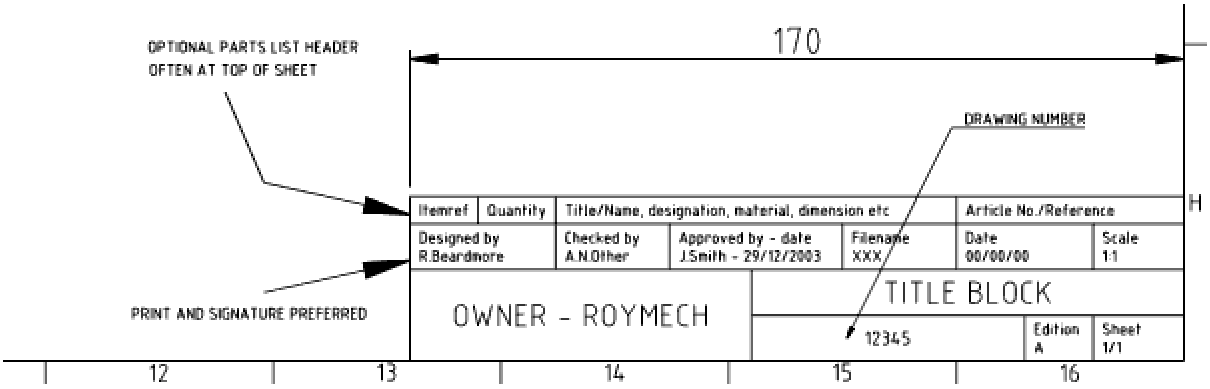
- the name of the company or organization
- the title of the drawing
- the drawing number, which is generally a unique filing identifier
- the scale
- the angle of projection used, either first or third, generally shown symbolically
- the signature or initials of the draftsman, checker, approving officer, and issuing officer, with the respective dates
- other information as required

The drawing should also include a symbol identifying the projection. The main scale and the linear dimension units if other than "mm".

Mechanical drawings should list the standards use for: indicating the surface texture: welds: general tolerances and geometric tolerances, as notes referring directly the the relevant standards or a general note referring to the BS 8888. (BS 8888 lists all of the relevant standards.) BS 8888 should really only be referenced if the drawing is in full accordance.

The drawing title block should indicate the date of the first revision. In separate boxes to the title block the current revision with an outline description of the revision should be indicated. On completion of each drawing revision an additional revision box should be completed thus providing a detailed history of the drawing

Typical Title Box



Typical Revision Box

Sheet Frames

It is standard practice for a drawing frame to be printed on each sheet, defining a margin around the outside of drawing area.

Drawing frames are standardized for each size of paper as per the following table:

Drawing Frames with No Filing Margin

Paper Size	Border Width (mm)		Dimensions of Drawing Frame (mm)	
	Left & Right	Top & Bottom	Width	Height
A0	28	20	1133	801
A1	20	14	801	566
A2	14	10	566	400
A3	10	7	4003	283
A4	7	5	283	200

Self-Check -7	Multiple Choice

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have 2.pts)

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

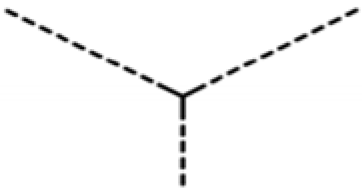
1. When center line, Projection line, Hidden outlines and cutting planes, are coincide, which line takes the Priority?

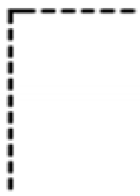
   a. Visible Line b. Center line c. Projection line d. Hidden line

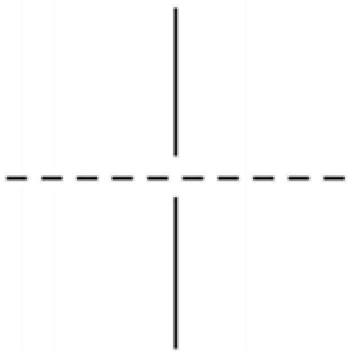
2. ______ type of line most frequently used to indicate a moving part's alternate position

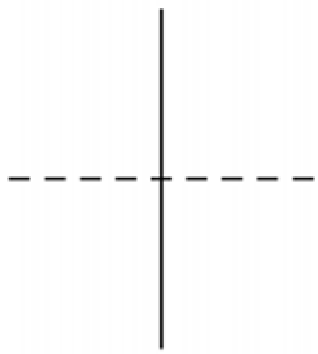
   a. Center line b. Phantom line c. visible line d. Hidden line

3. . All figure shows correct intersection between a hidden edge line and a visible edge line except

a.


b.


c.


d.


***Note:* Satisfactory rating 4 points and above** **Unsatisfactory – below 4 point**

You can ask you teacher for the copy of the correct answers.

Answer Sheet

Score = ___________
Rating: ____________

Name: _________________________ Date: _______________

Information Sheet-8	Selecting Components, material from manufacturing Catalogues

8.1. Introduction to Material Selection

Manufacturing Catalogues are a complete list of mechanical components, materials and items, typically arranged in alphabetical or symmetric order.

Design of an engineering component involves three interrelated problem.

- Selecting a material,
- Specifying a shape
- Choosing a manufacturing process

Getting this selection right the first time by selecting the optimal combination your design has enormous benefits to any engineering-based business. It leads to lower product costs, faster time-to-market, a reduction in the number of in-service failures and, sometimes, significant advantages relative to your competition.

When we talk about choosing materials for a component, we take into account many different factors. These factors can be broken down into the following areas.

Material Properties

The expected level of performance from the material

Material Cost and Availability

Material must be priced appropriately (not cheap but right) Material must be available (better to have multiple sources)

Processing

Must consider how to make the part, for example:

Casting

Machining

Welding

Environment

The effect that the service environment has on the part

The effect the part has on the environment

The effect that processing has on the environment

Now clearly these issues are inter-linked in some fashion. For example, cost is a direct result of how difficult a material is to obtain and to machine. And the effect of the environment on the material is clearly related to the material properties.

So if we really want to use a novel or unusual material, the choice must be made early in the design process. Then we can do the detailed design work using the correct material properties.

Consider the example of wooden airplanes and metal-framed airplanes. If we were to design an airplane of either material we will have to make the choice early. The end designs are quite different. So, the material choice can radically alter the final design. But the possibility also exists that it may not. After all what is the real difference between a 1045 and a 1035 carbon steel?

Kinds of Materials (What kind of materials can I use?)

Metals (Iron, Aluminum Copper, Magnesium)

Composites

Ceramics (Glass, Semi-conductors, structural ceramics (SiN, SiC), Refractory Composites

Polymers Rubber, Plastics, Liquids and Gases

Metal properties tend to be well understood and metals are somewhat forgiving materials. We can make small mistakes (sometimes big ones) and get away with a poor design as a result of metal's forgiving nature. We see ceramics and composites all around us, but they tend to be used in special applications because of fabrication costs. This however, is changing. Plastics are among the most common modern material choices. In large volume production, plastics are inexpensive. In small volume productions, plastics can be an extremely expensive choice due to high tooling costs.

Material Properties

We are most concerned with characteristics such as

Mechanical Properties

Strength (Yield Strength, Ultimate tensile Strength, Shear strength, Ductility, Hardness , density etc.)

You can get good information on particular materials from Standards handbooks, such as the ASM's Books on Metals. You can obtain information on gases and liquids from CRC's Handbooks. And the best place to get information on plastics and composites is from the manufacturer.

The choice of a material is frequently the result of several compromises. For example, the technical appraisal of an alloy will generally be a compromise between corrosion resistance and several other properties such as strength and weld ability. And the final selection may come down to a compromise between technical and economic factors. In identifying a material, approach the task in three stages:

List the material requirements for the design. Use the list of characteristics given above to help you in defining ALL the critical requirements. Rank the requirements in importance to the design's success.*

Select and evaluate candidate materials. By researching the various handbooks and resources, attempt to rank your candidate materials as to how well they meet the requirements. Use a decision table to identify the best choices**.**

Choose the most economical material. Research material costs and production costs based upon your anticipated production run. Choose the least expensive of your best choice candidate materials.

The four basic Steps Selecting Appropriate Materials

1) **Translation**: express design requirements as constraints and objectives

2) **Screening:** eliminate materials that cannot do the job

3) **Ranking:** find materials that best do the job

4) **Supporting Info:** Select, then verify with any supporting materials handbooks, expert systems, web, etc.

Self-Check -8	Multiple Choice

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have **2.pts**)

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. Which problems are encountered in design of an engineering component?

   a. Selecting a material, b. specifying a shape,

   C. choosing a manufacturing process d. All of above

2. __________ are a complete list of mechanical components, materials and items, typically arranged in alphabetical or symmetric order.

   a. Manufacturing Catalogue b. BOM (bill of material) c. manufacturing Process d.all

3. What are basic Steps in Selecting Appropriate Materials?

   a. Translation b. screening c. ranking d. supporting info e. all

***Note:* Satisfactory rating – 4 points Unsatisfactory - below 4 points**

You can ask you teacher for the copy of the correct answers.

Answer Sheet

Score = ___________

Rating: ____________

Name: _________________________ Date: _______________

Operation Sheet 1	Calculate Limits and Fits

1. Calculate the maximum and minimum limits for both the shaft and hole in the following; using the tables in Appendix for tolerances and name the type of fit obtained:

   (a) 45H8/d7 (b) 180H7/n6 (c) 120H7/s6

   (d) 40G7/h6 (e) 35 C11/h10

2. The dimensions of a shaft and a hole are given below:
   Shaft, Basic size = 60mm and given as 60 – 0.020
   Hole, Basic size = 60mm and given as 60 – 0.005
   Find out:
   (a) Tolerance of shaft
   (b) Tolerance of hole
   (c) Maximum allowance
   (d) Minimum allowance
   (e) Type of fit

3. A 30mm diameter hole is made on a turret lathe to the limits, 30.035 and 30.00. The following two grades of shafts are used to fit in the hole:

   (a) Φ 29.955mm and 29.925mm,
   (b) φ 30.055mm and 30.050mm.

   Calculate the maximum tolerance, clearance and indicate the type of fit in each case by a sketch.

1. By means of neat sketches and explanatory notes, interpret the meaning of the geometrical tolerances shown in Figure 1.

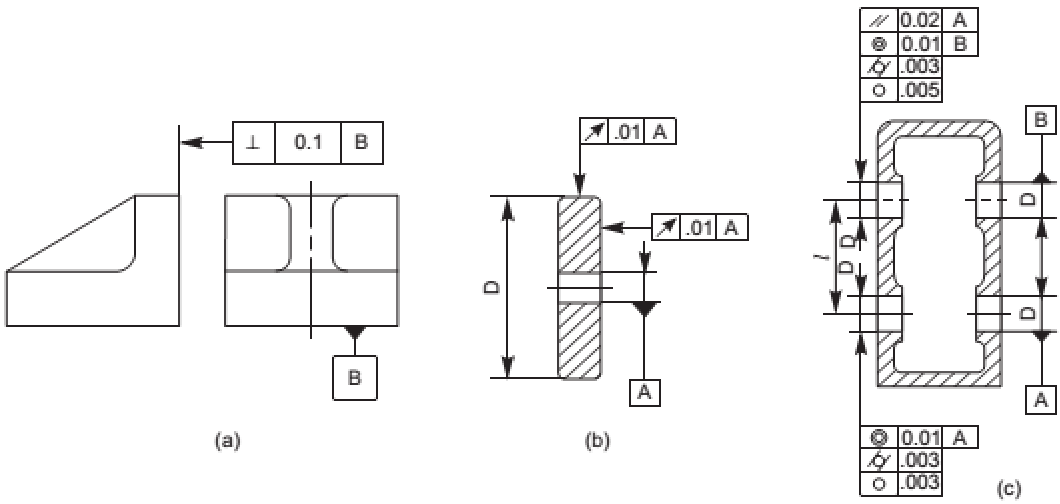


Figure 1

2. Explain the meaning of the geometrical tolerances indicated in microns, for the machine tool Components shown in Figure 2.

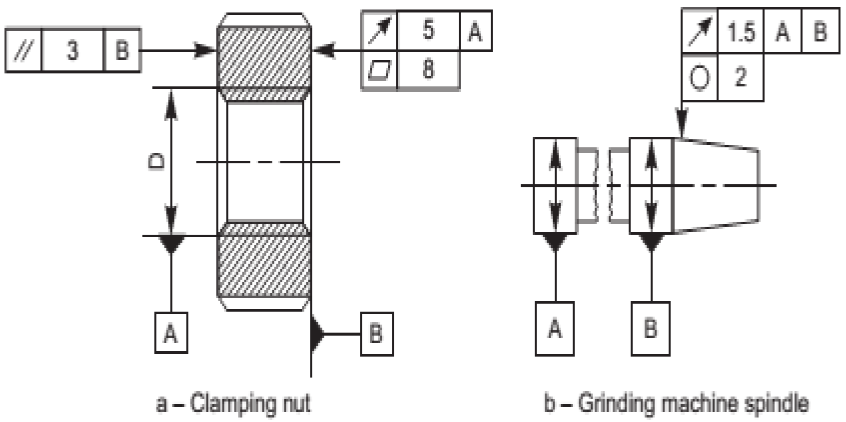


Figure 2

1. Identify (i) Functional, (ii) Non-functional and (iii) Auxiliary dimensions in Figure below.

Dimension Methods

Step 1. As far as possible, dimensions should be placed outside the view.
Step 2.. Dimensions should be taken from visible outlines
Step 3. Dimensioning to a centre line should be avoided except when the centre line passes through the centre of a hole.
Step 4 Each feature should be dimensioned once only on a drawing.
Step 5. Dimensions should be placed on the view or section that relates most clearly to the corresponding features.
Step 6. Each drawing should use the same unit for all dimensions, but without showing the unit symbol.

Step 7. No more dimensions than are necessary to define a part should be shown on a drawing.

Step 9. No features of a part should be defined by more than one dimension in any one direction.

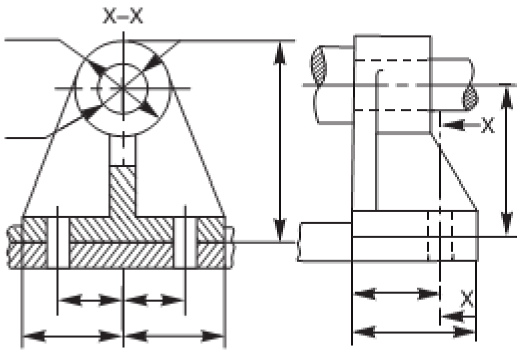


Figure 3.1

2. The drawings in Figure. 2 are not dimensioned properly. Correct them according to standards.

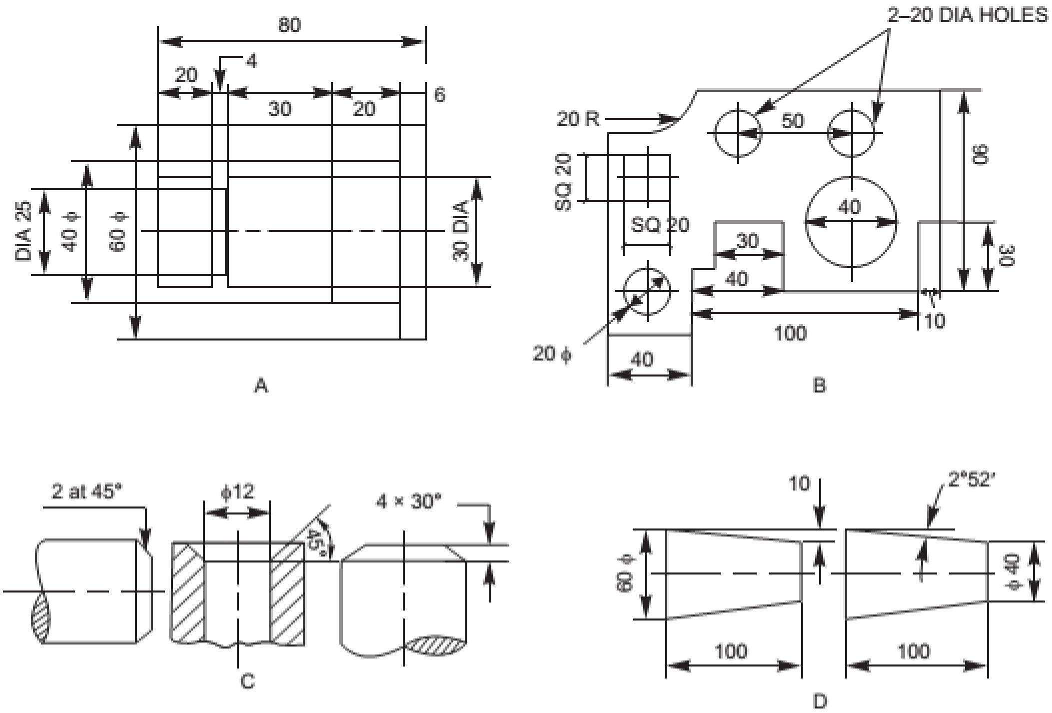

80
20
4
30
20
6
DIA 25
40 ϕ
60 ϕ
30 DIA
A
2–20 DIA HOLES
20 R
50
90
SQ 20
SQ 20
40
30
40
30
10
100
20 ϕ
40
B
2 at 45°
ϕ12
45°
4 × 30°
C
10
2°52′
60 ϕ
ϕ 40
100
100
D

Figure 3.2

Operation Sheet 4	Produce Sectional and Auxiliary section view In Third angle projection

For isometric drawing shown in figure 4.1. Draw

(i) the sectional view from the front,
(ii) the view from above
(iii) the sectional view from the left.
(iv) The auxiliary section view in third angle projection.

Steps to draw auxiliary view

Step 1. Construct an axis parallel to front View

Step 2. Make an auxiliary plane having the same gap with axis

Step 3. Project the primary auxiliary view from the inclined surface of an object

Step 4. Project the complete auxiliary of an object by projecting non parallel surfaces of an object including hidden surfaces

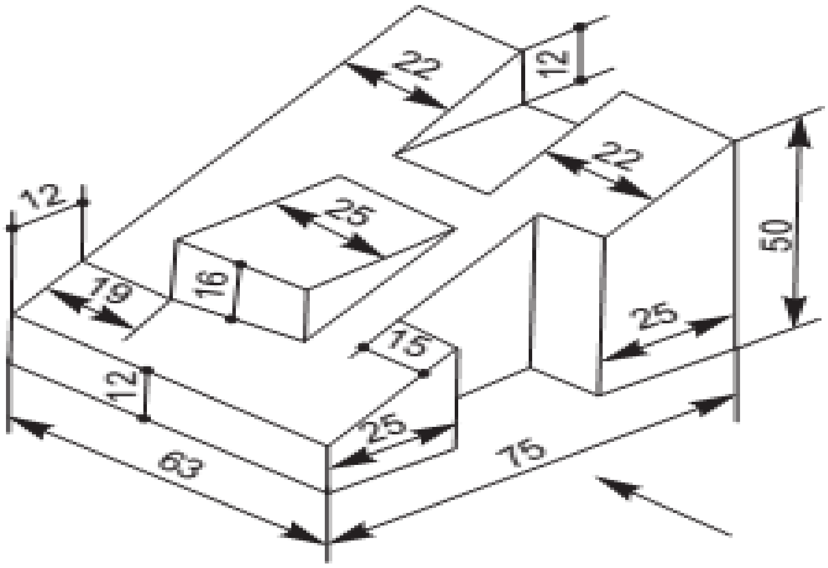


Figure 4.1

Operation Sheet 5	Produce Assembly Drawing

Assemble the components of the C-clamp shown in Figure. and draw, (i) sectional view from the front and (ii) view from the right.

Step 1. Study functional requirements of each component and their inter relationship

Step 2 Study carefully the views of each component in the detail drawing and decide the relative location of each part for the proper functioning of the machine.

Step 3 Decide the mating dimensions between two components which are required to be Assembled.

Step 4 Prepare free-hand sketch of the main view or an important view

Step 5 Select a suitable scale for the entire assembly drawing.

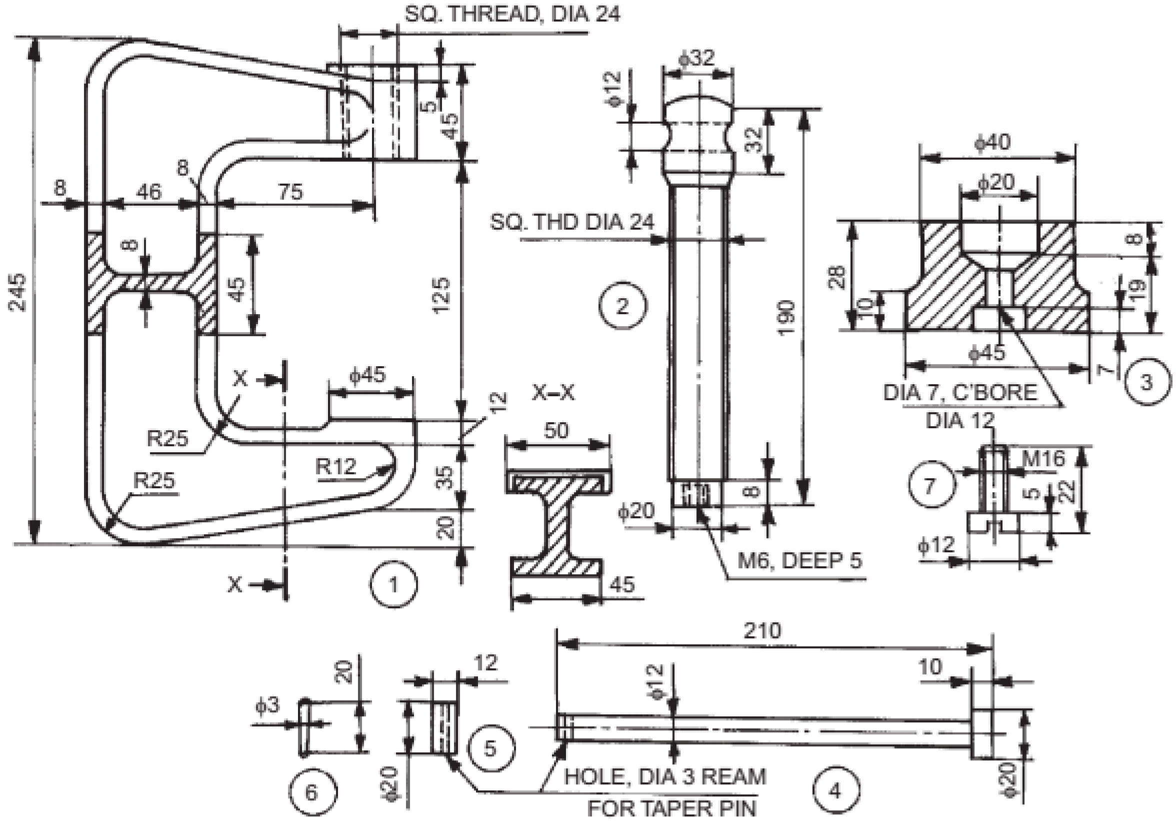
Step 6 Lay out the views of the assembly drawing so that it becomes easier to understand.

Step 7. Prepare the bill of materials.

Step 8. Show overall dimensions.

Step 9. Draw the section-lines according to the convention

Step 10. Show required fits and tolerances between the two mating components



Parts list

Sl. No.	Name	Matl.	Qty.
1	Frame	CI	1
2	Screw	MS	1
3	Pad	MS	1
4	Tommy bar	MS	1
5	Collar	MS	1
6	Pin	MS	1
7	Cap screw	MS	1

Figure 5.1

LAP Test 1	Practical Demonstration

Practical Demonstration Test

Name: ______________________________ Date: ________________

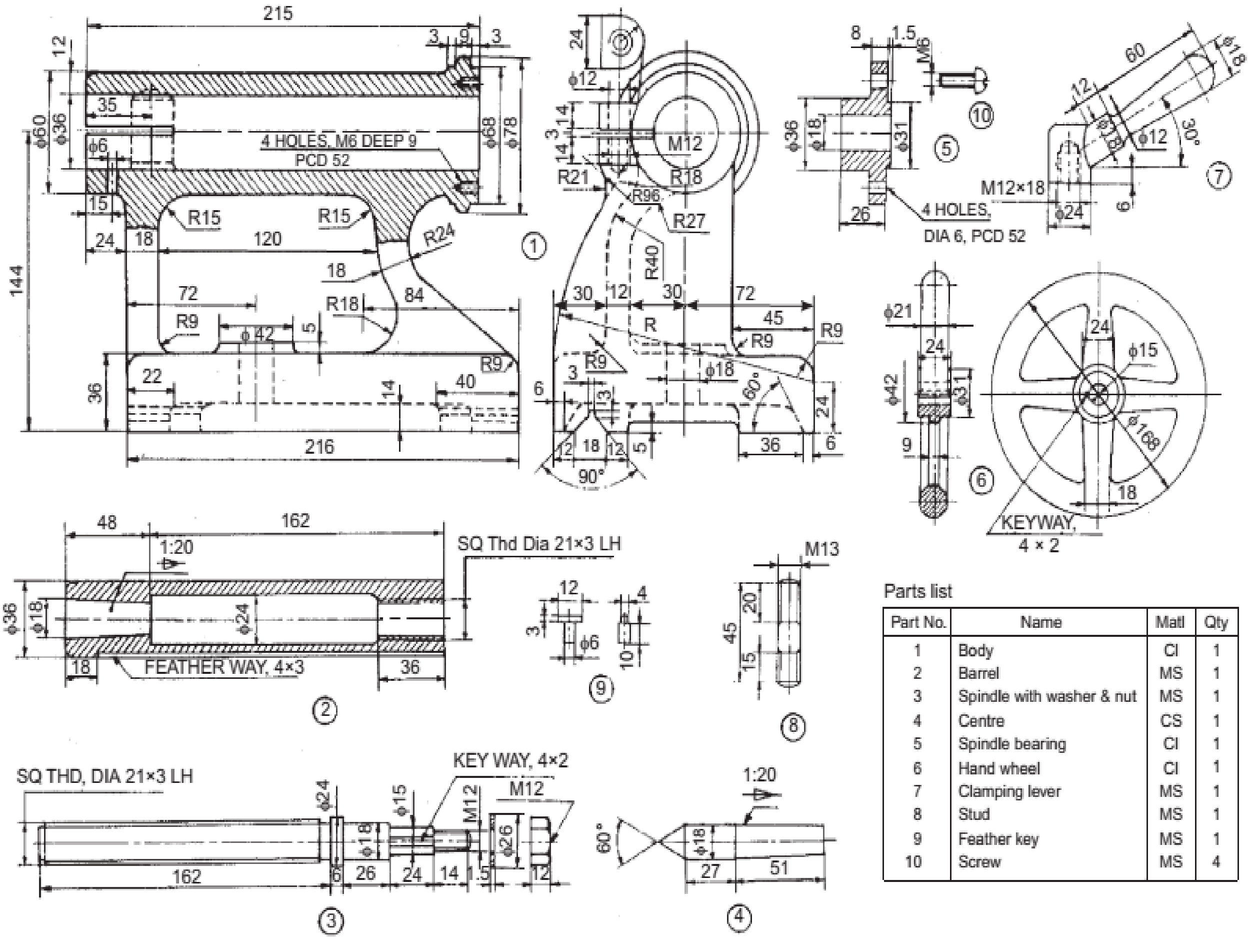
Time started: _________________________ Time finished: ________________

Instructions: Given necessary templates, tools and materials you are required to perform the following tasks within 8-12 hours.

Task 1: Assemble the components of Lathe machine tail-stock

Task 2: Draw the Sectional View and Auxiliary view

Task 3 Insert appropriate dimensions



Parts list

Part No.	Name	Matl	Qty
1	Body	CI	1
2	Barrel	MS	1
3	Spindle with washer & nut	MS	1
4	Centre	CS	1
5	Spindle bearing	CI	1
6	Hand wheel	CI	1
7	Clamping lever	MS	1
8	Stud	MS	1
9	Feather key	MS	1
10	Screw	MS	4

Figure 6: Lathe Tail Stock

List of Reference Materials

1. BOOKS

2. Machine drawing, therd edition,DR.KL.Narayana, Dr. M.A. Veluswami

3 Text book of engineering drawing,K.VENKATA Reddy,second edition,BS.Pabilication

4. – KHURMI R S AND GUPTA J. K (1979). A Text Book of Machine Design. ISN 81-219-0501-x, Published by Scand and Company ltd

Mechanics

LEVEL III

Learning Guide#03

Unit of Competence: Perform Advanced Engineering Detail Drafting

Module Title: Performing Advanced Engineering Detail Drafting

Module code: XXX

LG Code: XXX

TTLM Code: XXX

LO 3: Quality Assure Drawing

Instruction Sheet-3	Learning Guide #03

This learning guide is developed to provide you the necessary information regarding the following **content coverage** and topics:

- Checking drawings.
- Checking assembly/fabrication drawings.
- Issuing, filing and storing drawings

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, **upon completion of this Learning Guide, you will be able to**:

- Check drawings in Conformance with the specification.
- Check assembly/fabrication drawings.
- Issue, file and store drawings

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below 3 to 6.
3. Read the information written in the information "Sheet 1, Sheet 2, and Sheet 3".
4. Accomplish the "Self-check 1, Self-check t 2,and Self-check 3 " **in page -146, 149, and 154** respectively.

Information Sheet 1	Checking drawings

3.1. Introduction to Checking Drawings

The following list should be used as a basis for checking drawing:

Clarity - is the drawing clear in its intent and has sufficient information been shown.

Accuracy - is the drawing to scale, Are dimensions, levels are correct?

If any product information is used, is it the latest information?

Available from the latest catalogue?

Consistency - Is the drawing with other drawings in the same set

Drafting - Have the requirements of the drawing been met?

Check each component to ensure compliance with the specifications

Quality Control is the process of checking the accuracy of calculations and consistency of the drawings, detecting and correcting design omissions and errors prior to finalizing design plans and verifying the specifications for the load-carrying members are adequate for the service and strength loads. to document the Quality Control process.

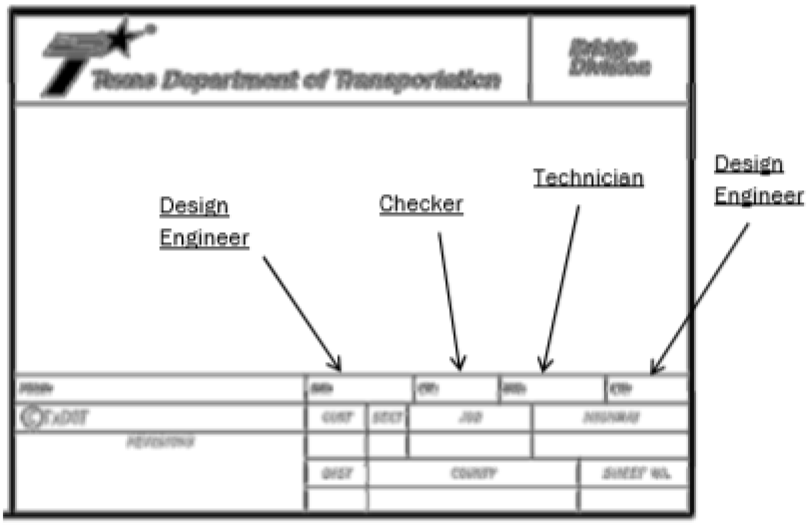


Figure 1: Detail Title Block

Quality Assurance is the process of reviewing the quality control process for use and effectiveness at preventing mistakes and ensuring compliance.

Self-Check -1	Multiple Choice

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have **2.pts**)
Directions: Answer all the questions listed below. Use the Answer sheet provided in the

1. Among the following lists which one is used as a basis for checking drawing?
   a. Clarity b. Accuracy c. Consistency d. Drafting e. all
2. ___________ is the process of reviewing the quality control process for use and Effectiveness at preventing mistakes and ensuring compliance.
   a. Quality Assurance b. Quality control c.TQM d. all
3. _____________ is the process of checking the accuracy of calculations and Consistency?
   a. TQM b. Quality control c. Quality Assurance d. All

***Note:* Satisfactory rating –3 points and above Unsatisfactory – below 3 points**

You can ask you teacher for the copy of the correct answers.

Score = ___________

Rating: _____________

Name: _____________________________ Date: _________________

Answer Sheet

Information Sheet 2	Checking assembly/ fabrication drawing

2.1. Introduction to Assembly Drawing

A drawing which displays the parts of a machine or a machine unit assembled in their relative working positions is known as assembly drawing. The assembly drawing would be such that it should satisfy: (i) Manufacturing requirements (ii) Operational requirements (iii) Maintenance requirements

2.2. Norms to be observed in preparing assembly drawings

(i) **Selection of views**: The main or important view which is usually in section should show all the individual parts and their relative locations. Additional views are shown only when they add necessary information.

(ii) **Sectioning:** The parts should be sectioned according to the requirements (i.e. half-section or partial section) to show important assembly details. Code of the BIS (SP: 46-1988) for general engineering drawings must be observed

(iii) **Dotted lines**: The dotted lines should be omit from the assembly drawing when a proper section is taken. If the view of a part is drawn by the half-section, then in un section portion of the view, the dotted lines may be drawn to clarify details of the part.

(iv) **Dimensions:** The overall dimensions and centre-to centre distances showing the relationship of parts to the machine as a whole, are sometimes shown.

(V) **Detailed dimensions** are given on working assembly drawings when the detailed drawings are not prepared.

Bill of materials: Each part of the machine is identified on assembly drawing by the leader line and number, which are used in the detail drawing and in the bill of material. The height of the number may be approximately 5 mm and encircled by 9 mm diameter. Leader lines are drawn radially touching the respective parts. The bill of materials also shows the following:

(a) Number of parts

(b) Material of parts

(c) Standard norm to Standard component

d) Scale

(e) Method of projection

(f) Shop processes

(g) Name of the company

(h) Designed by, drawn by and checked by

Sequences of preparing the assembly drawing

1. Study functional requirements of each component and their inter relationship. Learn the actual working of a machine.
2. Study carefully the views of each component in the detail drawing and decide the relative location of each part for the proper functioning of the machine.
3. Decide the mating dimensions between two components which are required to be assembled.
4. Prepare free-hand sketch of the main view or an important view
5. Select a suitable scale for the entire assembly drawing.
6. Lay out the views of the assembly drawing so that it becomes easier to understand.
7. Prepare the bill of materials.

   Label each component by the leader-line and number it.

8. Show overall dimensions.
9. Draw the section-lines according to the convention
10. Show required fits and tolerances between the two mating components.

Design for Assembly steps

1. Minimize part count
2. parts with self-locating features
3. parts with self-fastening features
4. Minimize reorientation of parts during assembly
5. parts for recovery, handling, & insertion
6. Emphasize 'Top-Down' assemblies
7. Standardize parts…minimum use of fasteners.
8. base part to locate other components
9. Design for component symmetry for insertion

Self-Check -2	Multiple Choice

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have **2.pts**)

Directions: Answer all the questions listed below. Use the Answer sheet provided in the

1. The assembly drawing would be satisfying _____?

   a. Manufacturing requirements b. Operational requirements c. Maintenance Requirements d. All

2. Which of Norms to be observed in preparing assembly drawings

   a**.** Sectioning b**.** Dimensions c. Bill of materials d. Detailed dimensions e. All

3. Which of the following are steps of Design for Assembly?

   a. Minimize part count b. Parts with self-locating features c. Emphasize 'Top-Down' assemblies d. Minimum use of fasteners e. All

Note: **Satisfactory rating –4 points and above Unsatisfactory – below 4 points**

You can ask you teacher for the copy of the correct answers.

Score = ___________
Rating: ____________

Name: _________________________ Date: _______________

Answer Sheet

Information Sheet 3	Issuing, filing and storing drawings

3.1. INTRODUCTION

By itself, any single paper item or book would seem easy to store and simple to preserve. However, most collections present challenges based simply on their size and the number of items they contain. When combined with considerations about storage space, storage methods, and shelving, the challenges of storing one item among many become complex.

Storage and handling methods have a direct impact on the useful life of drawings and the accessibility of information. Damage to drawings can be avoided by preventing overcrowded, careless, or haphazard storage conditions. Chemically unstable and improperly fitting shelving and storage enclosures accelerate the deterioration of drawing they are intended to protect. Normal use causes wear, but inexpert and rough handling can quickly lead to extensive damage to drawings requiring expensive repair or replacement. The longevity of drawings can be extended significantly by putting into practice the guidelines discussed here.

Photocopying or Scanning Bound Volumes drawing are often unnecessarily damaged during copying photocopy machines and flatbed scanners encourage pressing the binding flat in order to get a good image. Overhear scanners are better for public use because they allow a book page to be copied with the drawing open less than 180 degrees.

Copying or scanning of drawings is done by staff members (if the materials are particularly fragile). If materials are stable and an overhead scanner is available, researchers can be trained to make their own copies. Digital cameras can also be used with the proper policies in place. For guidance in using digital cameras in special collections and archives, see "Capture and Release": Digital Cameras in the Reading Room at http://www.oclc.org/research/publications/ library/2010/2010-05.pdf.

Issuing Drawings

Issuing is the process of moving drawing from storage or rooms to drawing production. The correct quantity of drawing must be issued to meet estimated guest demand. This process must be carefully controlled to minimize product misuse.

Importance of effective issuing should there be some relationship between the quantity of drawing and the quantity of drawing removed from storage areas?

1.1. Filing and handling Approved drawing/ blueprints

The method of folding prints depends upon the type and size of the filling cabinet and the loc ation of the identifying marks on the prints. It is best to place identifying marks at the point of prints when you file them vertically (upright), and at the bottom right corner when you file them flat. In some cases, construction prints are stored in rolls.

Blueprints are valuable permanent records. However, if you expect to keep them as permanent records, you must handle them with care. Here are a few simple rules that will help.

- Keep prints out of strong sunlight; they fade.
- Do not allow prints to become wet or smudged with oil or grease. Those substances seldom dry out completely, and the prints can become unreadable.
- Do not make pencil or crayon notations on a print without proper authority. If you are instructed to mark a print, use a proper colored pencil and make the markings a permanent part of the print. Yellow is a good color to use on a print with a blue background (blueprint).
- Keep prints stored in their proper place. If you receive prints that are not properly folded, refold them correctly.

With this article, we will review various types and methods of Blueprint Storage and Large Document Storage. We'll provide points for you to consider before making a purchase of any Blueprint Storage Box, Blueprint Storage Cabinet, Blueprint Storage Bags, or Blueprint Storage Rack. We understand that storing your Blueprints safely and securely is important, and that you want to have them organized for quick and easy reference and retrieval. When seeking Blueprint/hard copy of completed drawing Storage Solutions, you must first start by considering how many sheets and plan sets you have that you want to store. And, then consider how your collection will grow in the future. Will you have some storage solutions that offer quick retrieval but less protection? For example; a Blueprint Mobile Stand (also known as a Blueprint Stands, Blueprint Racks, and Hanging Stands) that is located near the people that use the plans the most. Basic starting points to consider when seeking to organize your Blueprint Storage would be; do you want flat storage, rolled storage, permanent storage, or locked storage. Rolling up drawings takes time, but it does allow for them to be stood up on end beside a workers desk. Corrugated Upright Roll Files and Wire Upright Roll Files offer quick access to rolled drawings right beside workers desk.
Something else you should consider is the investment in blueprints you or your company has. Some prints from CAD files that were printed on a CAD Plotter can be reproduced (printed again and again) easily, even if you have a disaster such as a fire. As long as the drawing data is backed up by your computer network administrator, you should be fine. But, what if your drawings have notes, or have been hand-drawn, or what if they were the only available original drawings. Your drawings may have many hours invested in them and as a result make the documents become very valuable and not easily replaced. Many Universities and Plants have the problem of having only originals that are available and once these documents are gone, they are lost forever. In these cases, we recommend a secured Plan Room with fire suppression (possibly not of the water type since water would ruin your documents. We recommend learn more about fire suppression systems which is outside the scope of this document), and we recommend Steel Flat Files that secure the drawings and add a level of protection. Some Blueprint Cabinets even offer fire protection as one of their features. You would need to read more about what each cabinet has to offer and how these levels of protection work.

In summary, we have touched on various points of which you should consider before you make a purchase of a Blueprint Storage Solution. We would even recommend possibly getting a meeting together within your company to discuss what each user would want.

STORING

There are several blueprint storage options to organize maps, plan drawings and other large format documents. Drafting Steals offers an entire line of storage options. Use Safco & Mayline Cabinets to safely store a vast number of maps, blue prints and other papers flat in minimal space. At the job site organize and quickly retrieve plans off racks and remove out of tubes and upright files. Protection of plans during transportation is important; we have blue print bags and document protectors to help protect valuable drawings. We have organized storage option in categories to make it easier to navigate. If you run into a problem, can't figure out what you need, don't hesitate to email or call us.

2.3. Storing Documents

As a document storage company, Ardington Archives have built up over 20 years experience managing our client's archive and storage requirements. We work closely with our clients to ensure best working practices and therefore provide guidance on storing documents prior to depositing at our facilities. Below are the 10 top tips for storing documents on site:

1. **Store documents in a stable environment.** Paper will deteriorate if kept in places which are too humid, too hot, too light and which allow uncontrolled access to pests (e.g. insects) and pollutants. In practice, typically bad places in which documents are often stored include basements (too humid and a danger of pests) or lofts (too hot in summer, too light and a danger of pests).
2. **Ensure documents are stored securely against theft.** Three key elements are involved in protecting your documents against theft:
    1. The physical security of the building: use good quality locks and metal doors, put bars on any windows, maintain an up to date alarm system, regularly check the building for any weaknesses or possible points of unauthorized entry.
    2. Controlling who is allowed into the document store: specify who is authorized to enter it and on what basis, monitor compliance with this, and do not allow visitors to have unsupervised access to the store.
3. **Maintaining confidentiality** regarding the location, identity and content of the document store – or, to put it simply, do not draw attention to the fact that particular documents are kept in a particular place.

- **Keep documents away from the risk of fire.** Storing documents in any place which is (a) liable to become very hot at certain times, or (b) exposed to direct sunlight, carries with it a risk of fire. Lofts with large plastic skylights are particularly dangerous in this regard.
- **Do not allow food to be consumed or stored in the same space as your documents.** Food and documents don't mix. The presence of even tiny traces of food will attract, sustain and promote the proliferation of pests such as insects and rodents which are more than capable of doing serious damage to your records.
- **Catalogue the contents of each archive box.** Cataloguing the contents of each box in which you are storing documents is undeniably a tedious task. However, having in

your possession a proper index to the contents of your boxes allows you to find particular documents quickly and will save you from the tedious task of opening up box after box to find that one file or document you just know is 'in there somewhere'.

- **Create a searchable storage space.** If you simply stack 100 boxes in a room without troubling to sort them into different locations or categories, you are likely to experience trouble finding individual boxes in the future. The best approach is to label boxes individually and catalogue the contents on a searchable index.
- Label **boxes properly.** Even if you have properly catalogued each box, it is still important to ensure that each has on it a clear, prominent and sufficiently descriptive label, which identifies the box within your catalogue.
- Allow **sufficient access space in the document store.** Space is always at a premium in any company, so it is always tempting to cram as much in as possible. However, while all 'dead space' should be filled, it is important to remember to leave enough space in the right places so that the boxes can be accessed without unnecessary loss of time and without compromising applicable regulations (particularly those pertaining to health and safety and fire safety).
- Use **strong boxes.** Flimsy boxes tend to be cheap, which is certainly a point in their favors, however they also have a tendency to collapse, tear or burst either when being moved or over a long period of storage. The consequences of this can include loss or damage to documents. More typically, they sag and collapse when stored in stacks over a long period, with the result that the contents have to be re boxed in sturdier boxes – a lab our and materials expense that could have been avoided.
- **Do not over fill boxes.** When filling up an archive box with documents and files, it is always tempting to try to jam as much in as possible and then shove the lid on. The result is usually an unsightly bulge at the top. This can be ignored of course – up to the point when you try to stack the boxes. You then find that the stacks are horribly unstable because the surfaces of the boxes on which each box stands are not flat. It is also important not to under fill boxes – unless they are of particularly strong build – since the unfilled (and therefore unsupported) top section of each box may collapse under the weight of the boxes stacked above it.
- CATALOG this article provides guidelines on managing architectural drawings, to help non-specialist archivists who have responsibility for these 'non-traditional' archives. The article gives advice on appraisal, sorting and cataloguing of drawings, based on the experience of a number of specialists working in dedicated architectural archives in the UK and abroad. Useful cataloguing terms are provided and defined, including expressions used to describe the purpose of specific kinds of drawing. The article is illustrated by three case studies, demonstrating appropriate levels of cataloguing for different types of drawing. Reference is also made to British and international cataloguing standards. Future collecting is considered too, specifically the collecting of architectural records in electronic formats.

Self-Check -3	Multiple Choice

Test I: Multiple Choice Questions

Directions: Choose the correct Answer (each question have **2.pts**)

Directions: Answer all the questions listed below. Use the Answer sheet provided in the

1. Which one is not included in the methods of storing the drawing files?
    a. The drawing must stored by their type
    b. The drawing must stored by their size
    c. The drawing must stored marked
    d. None
2. ___________ is the process of moving drawing from storage or rooms to drawing production.

    a. Issuing b. Catalog c. documenting d. All
3. Which of the following rules for keeping drawing?
    a. Do not allow prints to become wet or smudged with oil or grease
    b. Keep prints out of strong sunlight; they fade
    c. Do not make pencil or crayon notations on a print without proper authority
    d. Keep prints stored in their proper place
    e. All

***Note:* Satisfactory rating –4 points and above Unsatisfactory – below 4 points**

You can ask you teacher for the copy of the correct answers.

Name: ___________________________ Date: ________________

Answer Sheet

Score = ____________

Rating: _____________

List of Reference Materials

BOOKS

Machine drawing, therd edition,DR.KL.Narayana, Dr. M.A. Veluswami

Text book of engineering drawing,K.VENKATA Reddy,second edition,BS.Pabilication

KHURMI R S AND GUPTA J. K (1979). A Text Book of Machine Design. ISN 81-219-0501-x, Published by Scand and Company ltd

APPENDIX 1

Running and Sliding Fits[a]—American National Standard

RC 1 *Close sliding fits* are intended for the accurate location of parts which must assemble without perceptible play.

RC 2 *Sliding fits* are intended for accurate location, but with greater maximum clearance than class RC 1. Parts made to this fit move and turn easily but are not intended to run freely, and in the larger sizes may seize with small temperature changes.

RC 3 *Precision running fits* are about the closest fits which can be expected to run freely, and are intended for precision work at slow speeds and light journal pressures, but are not suitable where appreciable temperature differences are likely to be encountered.

RC 4 *Close running fits* are intended chiefly for running fits on accurate machinery with moderate surface speeds and journal pressures, where accurate location and minimum play are desired.

Basic hole system. Limits are in thousandths of an inch. See §14.8.
Limits for hole and shaft are applied algebraically to the basic size to obtain the limits of size for the parts.
Data in **boldface** are in accordance with ABC agreements.
Symbols H5, g5, etc., are hole and shaft designations used in ABC System.

Nominal Size Range, inches	Class RC 1			Class RC 2			Class RC 3			Class RC 4		
	Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits	
Over To		Hole H5	Shaft g4		Hole H6	Shaft g5		Hole H7	Shaft f6		Hole H8	Shaft f7
0 – 0.12	0.1 0.45	+0.2 −0	−0.1 −0.25	0.1 0.55	+0.25 −0	−0.1 −0.3	0.3 0.95	+0.4 −0	−0.3 −0.55	0.3 1.3	+0.6 −0	−0.3 −0.7
0.12– 0.24	0.15 0.5	+0.2 −0	−0.15 −0.3	0.15 0.65	+0.3 −0	−0.15 −0.35	0.4 1.12	+0.5 −0	−0.4 −0.7	0.4 1.6	+0.7 −0	−0.4 −0.9
0.24– 0.40	0.2 0.6	+0.25 −0	−0.2 −0.35	0.2 0.85	+0.4 −0	−0.2 −0.45	0.5 1.5	+0.6 −0	−0.5 −0.9	0.5 2.0	+0.9 −0	−0.5 −1.1
0.40– 0.71	0.25 0.75	+0.3 −0	−0.25 −0.45	0.25 0.95	+0.4 −0	−0.25 −0.55	0.6 1.7	+0.7 −0	−0.6 −1.0	0.6 2.3	+1.0 −0	−0.6 −1.3
0.71– 1.19	0.3 0.95	+0.4 −0	−0.3 −0.55	0.3 1.2	+0.5 −0	−0.3 −0.7	0.8 2.1	+0.8 −0	−0.8 −1.3	0.8 2.8	+1.2 −0	−0.8 −1.6
1.19– 1.97	0.4 1.1	+0.4 −0	−0.4 −0.7	0.4 1.4	+0.6 −0	−0.4 −0.8	1.0 2.6	+1.0 −0	−1.0 −1.6	1.0 3.6	+1.6 −0	−1.0 −2.0
1.97– 3.15	0.4 1.2	+0.5 −0	−0.4 −0.7	0.4 1.6	+0.7 −0	−0.4 −0.9	1.2 3.1	+1.2 −0	−1.2 −1.9	1.2 4.2	+1.8 −0	−1.2 −2.4
3.15– 4.73	0.5 1.5	+0.6 −0	−0.5 −0.9	0.5 2.0	+0.9 −0	−0.5 −1.1	1.4 3.7	+1.4 −0	−1.4 −2.3	1.4 5.0	+2.2 −0	−1.4 −2.8
4.73– 7.09	0.6 1.8	+0.7 −0	−0.6 −1.1	0.6 2.3	+1.0 −0	−0.6 −1.3	1.6 4.2	+1.6 −0	−1.6 −2.6	1.6 5.7	+2.5 −0	−1.6 −3.2
7.09– 9.85	0.6 2.0	+0.8 −0	−0.6 −1.2	0.6 2.6	+1.2 −0	−0.6 −1.4	2.0 5.0	+1.8 −0	−2.0 −3.2	2.0 6.6	+2.8 −0	−2.0 −3.8
9.85–12.41	0.8 2.3	+0.9 −0	−0.8 −1.4	0.8 2.9	+1.2 −0	−0.8 −1.7	2.5 5.7	+2.0 −0	−2.5 −3.7	2.5 7.5	+3.0 −0	−2.5 −4.5
12.41–15.75	1.0 2.7	+1.0 −0	−1.0 −1.7	1.0 3.4	+1.4 −0	−1.0 −2.0	3.0 6.6	+2.2 −0	−3.0 −4.4	3.0 8.7	+3.5 −0	−3.0 −5.2

[a]From ANSI B4.1—1967 (R1987). For larger diameters, see the standard.

Running and Sliding Fits[a]—American National Standard (continued)

RC 5, RC 6 — *Medium running fits* are intended for higher running speeds, or heavy journal pressures, or both.

RC 7 — *Free running fits* are intended for use where accuracy is not essential, or where large temperature variations are likely to be encountered, or under both these conditions.

RC 8, RC 9 — *Loose running fits* are intended for use where wide commercial tolerances may be necessary, together with an allowance, on the external member.

Nominal Size Range, inches	Class RC 5			Class RC 6			Class RC 7			Class RC 8			Class RC 9		
	Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits	
Over To		Hole H8	Shaft e7		Hole H9	Shaft e8		Hole H9	Shaft d8		Hole H10	Shaft c9		Hole H11	Shaft
0 - 0.12	0.6 1.6	+0.6 −0	−0.6 −1.0	0.6 2.2	+1.0 −0	−0.6 −1.2	1.0 2.6	+1.0 −0	−1.0 −1.6	2.5 5.1	+1.6 −0	−2.5 −3.5	4.0 8.1	+2.5 −0	−4.0 −5.6
0.12- 0.24	0.8 2.0	+0.7 −0	−0.8 −1.3	0.8 2.7	+1.2 −0	−0.8 −1.5	1.2 3.1	+1.2 −0	−1.2 −1.9	2.8 5.8	+1.8 −0	−2.8 −4.0	4.5 9.0	+3.0 −0	−4.5 −6.0
0.24- 0.40	1.0 2.5	+0.9 −0	−1.0 −1.6	1.0 3.3	+1.4 −0	−1.0 −1.9	1.6 3.9	+1.4 −0	−1.6 −2.5	3.0 6.6	+2.2 −0	−3.0 −4.4	5.0 10.7	+3.5 −0	−5.0 −7.2
0.40- 0.71	1.2 2.9	+1.0 −0	−1.2 −1.9	1.2 3.8	+1.6 −0	−1.2 −2.2	2.0 4.6	+1.6 −0	−2.0 −3.0	3.5 7.9	+2.8 −0	−3.5 −5.1	6.0 12.8	+4.0 −0	−6.0 −8.8
0.71- 1.19	1.6 3.6	+1.2 −0	−1.6 −2.4	1.6 4.8	+2.0 −0	−1.6 −2.8	2.5 5.7	+2.0 −0	−2.5 −3.7	4.5 10.0	+3.5 −0	−4.5 −6.5	7.0 15.5	+5.0 −0	−7.0 −10.5
1.19- 1.97	2.0 4.6	+1.6 −0	−2.0 −3.0	2.0 6.1	+2.5 −0	−2.0 −3.6	3.0 7.1	+2.5 −0	−3.0 −4.6	5.0 11.5	+4.0 −0	−5.0 −7.5	8.0 18.0	+6.0 −0	−8.0 −12.0
1.97- 3.15	2.5 5.5	+1.8 −0	−2.5 −3.7	2.5 7.3	+3.0 −0	−2.5 −4.3	4.0 8.8	+3.0 −0	−4.0 −5.8	6.0 13.5	+4.5 −0	−6.0 −9.0	9.0 20.5	+7.0 −0	−9.0 −13.5
3.15- 4.73	3.0 6.6	+2.2 −0	−3.0 −4.4	3.0 8.7	+3.5 −0	−3.0 −5.2	5.0 10.7	+3.5 −0	−5.0 −7.2	7.0 15.5	+5.0 −0	−7.0 −10.5	10.0 24.0	+9.0 −0	−10.0 −15.0
4.73- 7.09	3.5 7.6	+2.5 −0	−3.5 −5.1	3.5 10.0	+4.0 −0	−3.5 −6.0	6.0 12.5	+4.0 −0	−6.0 −8.5	8.0 18.0	+6.0 −0	−8.0 −12.0	12.0 28.0	+10.0 −0	−12.0 −18.0
7.09- 9.85	4.0 8.6	+2.8 −0	−4.0 −5.8	4.0 11.3	+4.5 −0	−4.0 −6.8	7.0 14.3	+4.5 −0	−7.0 −9.8	10.0 21.5	+7.0 −0	−10.0 −14.5	15.0 34.0	+12.0 −0	−15.0 −22.0
9.85-12.41	5.0 10.0	+3.0 −0	−5.0 −7.0	5.0 13.0	+5.0 −0	−5.0 −8.0	8.0 16.0	+5.0 −0	−8.0 −11.0	12.0 25.0	+8.0 −0	−12.0 −17.0	18.0 38.0	+12.0 −0	−18.0 −26.0
12.41-15.75	6.0 11.7	+3.5 −0	−6.0 −8.2	6.0 15.5	+6.0 −0	−6.0 −9.5	10.0 19.5	+6.0 −0	−10.0 13.5	14.0 29.0	+9.0 −0	−14.0 −20.0	22.0 45.0	+14.0 −0	−22.0 −31.0

[a]From ANSI B4.1—1967 (R1987). For larger diameters, see the standard.

APPENDIX 2

Clearance Locational Fits[a]—American National Standard

LC *Locational clearance fits* are intended for parts which are normally stationary, but which can be freely assembled or disassembled. They run from snug fits for parts requiring accuracy of location, through the medium clearance fits for parts such as spigots, to the looser fastener fits where freedom of assembly is of prime importance.

Basic hole system. Limits are in thousandths of an inch. See §14.8.
Limits for hole and shaft are applied algebraically to the basic size to obtain the limits of size for the parts.
Data in **boldface** are in accordance with ABC agreements.
Symbols H6, h5, etc., are hole and shaft designations used in ABC System.

Nominal Size Range, inches	Class LC 1			Class LC 2			Class LC 3			Class LC 4			Class LC 5		
	Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits	
Over To		Hole H6	Shaft h5		Hole H7	Shaft h6		Hole H8	Shaft h7		Hole H10	Shaft h9		Hole H7	Shaft g6
0 - 0.12	0 0.45	+0.25 −0	+0 −0.2	0 0.65	+0.4 −0	+0 −0.25	0 1	+0.6 −0	+0 −0.4	0 2.6	+1.6 −0	+0 −1.0	0.1 0.75	+0.4 −0	−0.1 −0.35
0.12- 0.24	0 0.5	+0.3 −0	+0 −0.2	0 0.8	+0.5 −0	+0 −0.3	0 1.2	+0.7 −0	+0 −0.5	0 3.0	+1.8 −0	+0 −1.2	0.15 0.95	+0.5 −0	−0.15 −0.45
0.24- 0.40	0 0.65	+0.4 −0	+0 −0.25	0 1.0	+0.6 −0	+0 −0.4	0 1.5	+0.9 −0	+0 −0.6	0 3.6	+2.2 −0	+0 −1.4	0.2 1.2	+0.6 −0	−0.2 −0.6
0.40- 0.71	0 0.7	+0.4 −0	+0 −0.3	0 1.1	+0.7 −0	+0 −0.4	0 1.7	+1.0 −0	+0 −0.7	0 4.4	+2.8 −0	+0 −1.6	0.25 1.35	+0.7 −0	−0.25 −0.65
0.71- 1.19	0 0.9	+0.5 −0	+0 −0.4	0 1.3	+0.8 −0	+0 −0.5	0 2	+1.2 −0	+0 −0.8	0 5.5	+3.5 −0	+0 −2.0	0.3 1.6	+0.8 −0	−0.3 −0.8
1.19- 1.97	0 1.0	+0.6 −0	+0 −0.4	0 1.6	+1.0 −0	+0 −0.6	0 2.6	+1.6 −0	+0 −1	0 6.5	+4.0 −0	+0 −2.5	0.4 2.0	+1.0 −0	−0.4 −1.0
1.97- 3.15	0 1.2	+0.7 −0	+0 −0.5	0 1.9	+1.2 −0	+0 −0.7	0 3	+1.8 −0	+0 −1.2	0 7.5	+4.5 −0	+0 −3	0.4 2.3	+1.2 −0	−0.4 −1.1
3.15- 4.73	0 1.5	+0.9 −0	+0 −0.6	0 2.3	+1.4 −0	+0 −0.9	0 3.6	+2.2 −0	+0 −1.4	0 8.5	+5.0 −0	+0 −3.5	0.5 2.8	+1.4 −0	−0.5 −1.4
4.73- 7.09	0 1.7	+1.0 −0	+0 −0.7	0 2.6	+1.6 −0	+0 −1.0	0 4.1	+2.5 −0	+0 −1.6	0 10	+6.0 −0	+0 −4	0.6 3.2	+1.6 −0	−0.6 −1.6
7.09- 9.85	0 2.0	+1.2 −0	+0 −0.8	0 3.0	+1.8 −0	+0 −1.2	0 4.6	+2.8 −0	+0 −1.8	0 11.5	+7.0 −0	+0 −4.5	0.6 3.6	+1.8 −0	−0.6 −1.8
9.85-12.41	0 2.1	+1.2 −0	+0 −0.9	0 3.2	+2.0 −0	+0 −1.2	0 5	+3.0 −0	+0 −2.0	0 13	+8.0 −0	+0 −5	0.7 3.9	+2.0 −0	−0.7 −1.9
12.41-15.75	0 2.4	+1.4 −0	+0 −1.0	0 3.6	+2.2 −0	+0 −1.4	0 5.7	+3.5 −0	+0 −2.2	0 15	+9.0 −0	+0 −6	0.7 4.3	+2.2 −0	−0.7 −2.1

Clearance Locational Fits[a]—American National Standard (continued)

Nominal Size Range, inches	Class LC 6			Class LC 7			Class LC 8			Class LC 9			Class LC 10			Class LC 11		
	Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits	
Over To		Hole H9	Shaft f8		Hole H10	Shaft e9		Hole H10	Shaft d9		Hole H11	Shaft c10		Hole H12	Shaft		Hole H13	Shaft
0 - 0.12	0.3 1.9	+1.0 −0	−0.3 −0.9	0.6 3.2	+1.6 −0	− 0.6 − 1.6	1.0 3.6	+1.6 −0	− 1.0 − 2.0	2.5 6.6	+ 2.5 − 0	− 2.5 − 4.1	4 12	+ 4 − 0	− 4 − 8	5 17	+ 6 − 0	− 5 −11
0.12– 0.24	0.4 2.3	+1.2 −0	−0.4 −1.1	0.8 3.8	+1.8 −0	− 0.8 − 2.0	1.2 4.2	+1.8 −0	− 1.2 − 2.4	2.8 7.6	+ 3.0 − 0	− 2.8 − 4.6	4.5 14.5	+ 5 − 0	− 4.5 − 9.5	6 20	+ 7 − 0	− 6 −13
0.24– 0.40	0.5 2.8	+1.4 −0	−0.5 −1.4	1.0 4.6	+2.2 −0	− 1.0 − 2.4	1.6 5.2	+2.2 −0	− 1.6 − 3.0	3.0 8.7	+ 3.5 − 0	− 3.0 − 5.2	5 17	+ 6 − 0	− 5 −11	7 25	+ 9 − 0	− 7 −16
0.40– 0.71	0.6 3.2	+1.6 −0	−0.6 −1.6	1.2 5.6	+2.8 −0	− 1.2 − 2.8	2.0 6.4	+2.8 −0	− 2.0 − 3.6	3.5 10.3	+ 4.0 − 0	− 3.5 − 6.3	6 20	+ 7 − 0	− 6 −13	8 28	+10 − 0	− 8 −18
0.71– 1.19	0.8 4.0	+2.0 −0	−0.8 −2.0	1.6 7.1	+3.5 −0	− 1.6 − 3.6	2.5 8.0	+3.5 −0	− 2.5 − 4.5	4.5 13.0	+ 5.0 − 0	− 4.5 − 8.0	7 23	+ 8 − 0	− 7 −15	10 34	+12 − 0	−10 −22
1.19– 1.97	1.0 5.1	+2.5 −0	−1.0 −2.6	2.0 8.5	+4.0 −0	− 2.0 − 4.5	3.0 9.5	+4.0 −0	− 3.0 − 5.5	5 15	+ 6 − 0	− 5 − 9	8 28	+10 − 0	− 8 −18	12 44	+16 − 0	−12 −28
1.97– 3.15	1.2 6.0	+3.0 −0	−1.2 −3.0	2.5 10.0	+4.5 −0	− 2.5 − 5.5	4.0 11.5	+4.5 −0	− 4.0 − 7.0	6 17.5	+ 7 − 0	− 6 −10.5	10 34	+12 − 0	−10 −22	14 50	+18 − 0	−14 −32
3.15– 4.73	1.4 7.1	+3.5 −0	−1.4 −3.6	3.0 11.5	+5.0 −0	− 3.0 − 6.5	5.0 13.5	+5.0 −0	− 5.0 − 8.5	7 21	+ 9 − 0	− 7 −12	11 39	+14 − 0	−11 −25	16 60	+22 − 0	−16 −38
4.73– 7.09	1.6 8.1	+4.0 −0	−1.6 −4.1	3.5 13.5	+6.0 −0	− 3.5 − 7.5	6 16	+6 −0	− 6 −10	8 24	+10 − 0	− 8 −14	12 44	+16 − 0	−12 −28	18 68	+25 − 0	−18 −43
7.09– 9.85	2.0 9.3	+4.5 −0	−2.0 −4.8	4.0 15.5	+7.0 −0	− 4.0 − 8.5	7 18.5	+7 −0	− 7 −11.5	10 29	+12 − 0	−10 −17	16 52	+18 − 0	−16 −34	22 78	+28 − 0	−22 −50
9.85–12.41	2.2 10.2	+5.0 −0	−2.2 −5.2	4.5 17.5	+8.0 −0	− 4.5 − 9.5	7 20	+8 −0	− 7 −12	12 32	+12 − 0	−12 −20	20 60	+20 − 0	−20 −40	28 88	+30 − 0	−28 −58
12.41–15.75	2.5 12.0	+6.0 −0	−2.5 −6.0	5.0 20.0	+9.0 −0	− 5 −11	8 23	+9 −0	− 8 −14	14 37	+14 − 0	−14 −23	22 66	+22 − 0	−22 −44	30 100	+35 − 0	−30 −65

[a]From ANSI B4.1—1967 (R1987). For larger diameters, see the standard.

APPENDIX 3

Transition Locational Fits[a]—American National Standard

LT *Transition fits* are a compromise between clearance and interference fits, for application where accuracy of location is important, but either a small amount of clearance or interference is permissible.

Basic hole system. Limits are in thousandths of an inch. See §14.8.
Limits for hole and shaft are applied algebraically to the basic size to obtain the limits of size for the mating parts.
Data in **boldface** are in accordance with ABC agreements.
"Fit" represents the maximum interference (minus values) and the maximum clearance (plus values).
Symbols H7, js6, etc., are hole and shaft designations used in ABC System.

Nominal Size Range, inches	Class LT 1			Class LT 2			Class LT 3			Class LT 4			Class LT 5			Class LT 6		
	Fit	Standard Limits		Fit	Standard Limits		Fit	Standard Limits		Fit	Standard Limits		Fit	Standard Limits		Fit	Standard Limits	
Over To		Hole H7	Shaft js6		Hole H8	Shaft js7		Hole H7	Shaft k6		Hole H8	Shaft k7		Hole H7	Shaft n6		Hole H7	Shaft n7
0 - 0.12	−0.10 +0.50	+0.4 −0	+0.10 −0.10	−0.2 +0.8	+0.6 −0	+0.2 −0.2							−0.5 +0.15	+0.4 −0	+0.5 +0.25	−0.65 +0.15	+0.4 −0	+0.65 +0.25
0.12- 0.24	−0.15 +0.65	+0.5 −0	+0.15 −0.15	−0.25 +0.95	+0.7 −0	+0.25 −0.25							−0.6 +0.2	+0.5 −0	+0.6 +0.3	−0.8 +0.2	+0.5 −0	+0.8 +0.3
0.24- 0.40	−0.2 +0.8	+0.6 −0	+0.2 −0.2	−0.3 +1.2	+0.9 −0	+0.3 −0.3	−0.5 +0.5	+0.6 −0	+0.5 +0.1	−0.7 +0.8	+0.9 −0	+0.7 +0.1	−0.8 +0.2	+0.6 −0	+0.8 +0.4	−1.0 +0.2	+0.6 −0	+1.0 +0.4
0.40- 0.71	−0.2 +0.9	+0.7 −0	+0.2 −0.2	−0.35 +1.35	+1.0 −0	+0.35 −0.35	−0.5 +0.6	+0.7 −0	+0.5 +0.1	−0.8 +0.9	+1.0 −0	+0.8 +0.1	−0.9 +0.2	+0.7 −0	+0.9 +0.5	−1.2 +0.2	+0.7 −0	+1.2 +0.5
0.71- 1.19	−0.25 +1.05	+0.8 −0	+0.25 −0.25	−0.4 +1.6	+1.2 −0	+0.4 −0.4	−0.6 +0.7	+0.8 −0	+0.6 +0.1	−0.9 +1.1	+1.2 −0	+0.9 +0.1	−1.1 +0.2	+0.8 −0	+1.1 +0.6	−1.4 +0.2	+0.8 −0	+1.4 +0.6
1.19- 1.97	−0.3 +1.3	+1.0 −0	+0.3 −0.3	−0.5 +2.1	+1.6 −0	+0.5 −0.5	−0.7 +0.9	+1.0 −0	+0.7 +0.1	−1.1 +1.5	+1.6 −0	+1.1 +0.1	−1.3 +0.3	+1.0 −0	+1.3 +0.7	−1.7 +0.3	+1.0 −0	+1.7 +0.7
1.97- 3.15	−0.3 +1.5	+1.2 −0	+0.3 −0.3	−0.6 +2.4	+1.8 −0	+0.6 −0.6	−0.8 +1.1	+1.2 −0	+0.8 +0.1	−1.3 +1.7	+1.8 −0	+1.3 +0.1	−1.5 +0.4	+1.2 −0	+1.5 +0.8	−2.0 +0.4	+1.2 −0	+2.0 +0.8
3.15- 4.73	−0.4 +1.8	+1.4 −0	+0.4 −0.4	−0.7 +2.9	+2.2 −0	+0.7 −0.7	−1.0 +1.3	+1.4 −0	+1.0 +0.1	−1.5 +2.1	+2.2 −0	+1.5 +0.1	−1.9 +0.4	+1.4 −0	+1.9 +1.0	−2.4 +0.4	+1.4 −0	+2.4 +1.0
4.73- 7.09	−0.5 +2.1	+1.6 −0	+0.5 −0.5	−0.8 +3.3	+2.5 −0	+0.8 −0.8	−1.1 +1.5	+1.6 −0	+1.1 +0.1	−1.7 +2.4	+2.5 −0	+1.7 +0.1	−2.2 +0.4	+1.6 −0	+2.2 +1.2	−2.8 +0.4	+1.6 −0	+2.8 +1.2
7.09- 9.85	−0.6 +2.4	+1.8 −0	+0.6 −0.6	−0.9 +3.7	+2.8 −0	+0.9 −0.9	−1.4 +1.6	+1.8 −0	+1.4 +0.2	−2.0 +2.6	+2.8 −0	+2.0 +0.2	−2.6 +0.4	+1.8 −0	+2.6 +1.4	−3.2 +0.4	+1.8 −0	+3.2 +1.4
9.85-12.41	−0.6 +2.6	+2.0 −0	+0.6 −0.6	−1.0 +4.0	+3.0 −0	+1.0 −1.0	−1.4 +1.8	+2.0 −0	+1.4 +0.2	−2.2 +2.8	+3.0 −0	+2.2 +0.2	−2.6 +0.6	+2.0 −0	+2.6 +1.4	−3.4 +0.6	+2.0 −0	+3.4 +1.4
12.41-15.75	−0.7 +2.9	+2.2 −0	+0.7 −0.7	−1.0 +4.5	+3.5 −0	+1.0 −1.0	−1.6 +2.0	+2.2 −0	+1.6 +0.2	−2.4 +3.3	+3.5 −0	+2.4 +0.2	−3.0 +0.6	+2.2 −0	+3.0 +1.6	−3.8 +0.6	+2.2 −0	+3.8 +1.6

APPENDIX 4

Interference Locational Fits[a]—American National Standard

LN *Locational interference fits* are used where accuracy of location is of prime importance, and for parts requiring rigidity and alignment with no special requirements for bore pressure. Such fits are not intended for parts designed to transmit frictional loads from one part to another by virtue of the tightness of fit, as these conditions are covered by force fits.

Basic hole system. Limits are in thousandths of an inch. See §14.8.
Limits for hole and shaft are applied algebraically to the basic size to obtain the limits of size for the parts.
Data in **boldface** are in accordance with ABC agreements.
Symbols H7, p6, etc., are hole and shaft designations used in ABC System.

Nominal Size Range, inches	Class LN 1			Class LN 2			Class LN 3		
	Limits of Interference	Standard Limits		Limits of Interference	Standard Limits		Limits of Interference	Standard Limits	
Over To		Hole H6	Shaft n5		Hole H7	Shaft p6		Hole H7	Shaft r6
0 - 0.12	0 0.45	+0.25 −0	+0.45 +0.25	0 0.65	+0.4 −0	+0.65 +0.4	0.1 0.75	+0.4 −0	+0.75 +0.5
0.12– 0.24	0 0.5	+0.3 −0	+0.5 +0.3	0 0.8	+0.5 −0	+0.8 +0.5	0.1 0.9	+0.5 0	+0.9 +0.6
0.24– 0.40	0 0.65	+0.4 −0	+0.65 +0.4	0 1.0	+0.6 −0	+1.0 +0.6	0.2 1.2	+0.6 −0	+1.2 +0.8
0.40– 0.71	0 0.8	+0.4 −0	+0.8 +0.4	0 1.1	+0.7 −0	+1.1 +0.7	0.3 1.4	+0.7 −0	+1.4 +1.0
0.71– 1.19	0 1.0	+0.5 −0	+1.0 +0.5	0 1.3	+0.8 −0	+1.3 +0.8	0.4 1.7	+0.8 −0	+1.7 +1.2
1.19– 1.97	0 1.1	+0.6 −0	+1.1 +0.6	0 1.6	+1.0 −0	+1.6 +1.0	0.4 2.0	+1.0 −0	+2.0 +1.4
1.97– 3.15	0.1 1.3	+0.7 −0	+1.3 +0.7	0.2 2.1	+1.2 −0	+2.1 +1.4	0.4 2.3	+1.2 −0	+2.3 +1.6
3.15– 4.73	0.1 1.6	+0.9 −0	+1.6 +1.0	0.2 2.5	+1.4 −0	+2.5 +1.6	0.6 2.9	+1.4 −0	+2.9 +2.0
4.73– 7.09	0.2 1.9	+1.0 −0	+1.9 +1.2	0.2 2.8	+1.6 −0	+2.8 +1.8	0.9 3.5	+1.6 −0	+3.5 +2.5
7.09– 9.85	0.2 2.2	+1.2 −0	+2.2 +1.4	0.2 3.2	+1.8 −0	+3.2 +2.0	1.2 4.2	+1.8 −0	+4.2 +3.0
9.85–12.41	0.2 2.3	+1.2 −0	+2.3 +1.4	0.2 3.4	+2.0 −0	+3.4 +2.2	1.5 4.7	+2.0 −0	+4.7 +3.5

[a]From ANSI B4.1—1967 (R1987). For larger diameters, see the standard.

Force and Shrink Fits[a]—American National Standard

FN 1 *Light drive fits* are those requiring light assembly pressures, and produce more or less permanent assemblies. They are suitable for thin sections or long fits, or in cast-iron external members.

FN 2 *Medium drive fits* are suitable for ordinary steel parts, or for shrink fits on light sections. They are about the tightest fits that can be used with high-grade cast-iron external members.

FN 3 *Heavy drive fits* are suitable for heavier steel parts or for shrink fits in medium sections.

FN 4, FN 5 *Force fits* are suitable for parts which can be highly stressed, or for shrink fits where the heavy pressing forces required are impractical.

Basic hole system. Limits are in thousandths of an inch. See §14.8.

Limits for hole and shaft are applied algebraically to the basic size to obtain the limits of size for the parts.
Data in **boldface** are in accordance with ABC agreements.
Symbols H7, s6, etc., are hole and shaft designations used in ABC System.

Nominal Size Range, inches	Class FN 1			Class FN 2			Class FN 3			Class FN 4			Class FN 5		
	Limits of Interference	Standard Limits		Limits of Interference	Standard Limits		Limits of Interference	Standard Limits		Limits of Interference	Standard Limits		Limits of Interference	Standard Limits	
Over To		Hole H6	Shaft		Hole H7	Shaft s6		Hole H7	Shaft t6		Hole H7	Shaft u6		Hole H8	Shaft x7
0 – 0.12	0.05 0.5	**+0.25 −0**	+0.5 +0.3	**0.2 0.85**	**+0.4 −0**	**+0.85 +0.6**				**0.3 0.95**	**+0.4 −0**	**+ 0.95 + 0.7**	**0.3 1.3**	**+0.6 −0**	**+ 1.3 + 0.9**
0.12– 0.24	0.1 0.6	**+0.3 −0**	+0.6 +0.4	**0.2 1.0**	**+0.5 −0**	**+1.0 +0.7**				**0.4 1.2**	**+0.5 −0**	**+ 1.2 + 0.9**	**0.5 1.7**	**+0.7 −0**	**+ 1.7 + 1.2**
0.24– 0.40	0.1 0.75	**+0.4 −0**	+0.75 +0.5	**0.4 1.4**	**+0.6 −0**	**+1.4 +1.0**				**0.6 1.6**	**+0.6 −0**	**+ 1.6 + 1.2**	**0.5 2.0**	**+0.9 −0**	**+ 2.0 + 1.4**
0.40– 0.56	0.1 0.8	**+0.4 −0**	+0.8 +0.5	**0.5 1.6**	**+0.7 −0**	**+1.6 +1.2**				**0.7 1.8**	**+0.7 −0**	**+ 1.8 + 1.4**	**0.6 2.3**	**+1.0 −0**	**+ 2.3 + 1.6**
0.56– 0.71	0.2 0.9	**+0.4 −0**	+0.9 +0.6	**0.5 1.6**	**+0.7 −0**	**+1.6 +1.2**				**0.7 1.8**	**+0.7 −0**	**+ 1.8 + 1.4**	**0.8 2.5**	**+1.0 −0**	**+ 2.5 + 1.8**
0.71– 0.95	0.2 1.1	**+0.5 −0**	+1.1 +0.7	**0.6 1.9**	**+0.8 −0**	**+1.9 +1.4**				**0.8 2.1**	**+0.8 −0**	**+ 2.1 + 1.6**	**1.0 3.0**	**+1.2 −0**	**+ 3.0 + 2.2**
0.95– 1.19	0.3 1.2	**+0.5 −0**	+1.2 +0.8	**0.6 1.9**	**+0.8 −0**	**+1.9 +1.4**	**0.8 2.1**	**+0.8 −0**	**+ 2.1 + 1.6**	**1.0 2.3**	**+0.8 −0**	**+ 2.3 + 1.8**	**1.3 3.3**	**+1.2 −0**	**+ 3.3 + 2.5**
1.19– 1.58	0.3 1.3	**+0.6 −0**	+1.3 +0.9	**0.8 2.4**	**+1.0 −0**	**+2.4 +1.8**	**1.0 2.6**	**+1.0 −0**	**+ 2.6 + 2.0**	**1.5 3.1**	**+1.0 −0**	**+ 3.1 + 2.5**	**1.4 4.0**	**+1.6 −0**	**+ 4.0 + 3.0**

[a]ANSI B4.1–1967 (R1987).

APPENDIX 5

Force and Shrink Fits[a]—American National Standard (continued)

Nominal Size Range, inches	Class FN 1			Class FN 2			Class FN 3			Class FN 4			Class FN 5		
	Limits of Interference	Standard Limits		Limits of Interference	Standard Limits		Limits of Interference	Standard Limits		Limits of Interference	Standard Limits		Limits of Interference	Standard Limits	
Over To		Hole H6	Shaft		Hole H7	Shaft s6		Hole H7	Shaft t6		Hole H7	Shaft u6		Hole H8	Shaft x7
1.58– 1.97	0.4 1.4	+0.6 −0	+1.4 +1.0	0.8 2.4	+1.0 −0	+2.4 +1.8	1.2 2.8	+1.0 −0	+ 2.8 + 2.2	1.8 3.4	+1.0 −0	+ 3.4 + 2.8	2.4 5.0	+1.6 −0	+ 5.0 + 4.0
1.97– 2.56	0.6 1.8	+0.7 −0	+1.8 +1.3	0.8 2.7	+1.2 −0	+2.7 +2.0	1.3 3.2	+1.2 −0	+ 3.2 + 2.5	2.3 4.2	+1.2 −0	+ 4.2 + 3.5	3.2 6.2	+1.8 −0	+ 6.2 + 5.0
2.56– 3.15	0.7 1.9	+0.7 −0	+1.9 +1.4	1.0 2.9	+1.2 −0	+2.9 +2.2	1.8 3.7	+1.2 −0	+ 3.7 + 3.0	2.8 4.7	+1.2 −0	+ 4.7 + 4.0	4.2 7.2	+1.8 −0	+ 7.2 + 6.0
3.15– 3.94	0.9 2.4	+0.9 −0	+2.4 +1.8	1.4 3.7	+1.4 −0	+3.7 +2.8	2.1 4.4	+1.4 −0	+ 4.4 + 3.5	3.6 5.9	+1.4 −0	+ 5.9 + 5.0	4.8 8.4	+2.2 −0	+ 8.4 + 7.0
3.94– 4.73	1.1 2.6	+0.9 −0	+2.6 +2.0	1.6 3.9	+1.4 −0	+3.9 +3.0	2.6 4.9	+1.4 −0	+ 4.9 + 4.0	4.6 6.9	+1.4 −0	+ 6.9 + 6.0	5.8 9.4	+2.2 − 0	+ 9.4 + 8.0
4.73– 5.52	1.2 2.9	+1.0 −0	+2.9 +2.2	1.9 4.5	+1.6 −0	+4.5 +3.5	3.4 6.0	+1.6 −0	+ 6.0 + 5.0	5.4 8.0	+1.6 −0	+ 8.0 + 7.0	7.5 11.6	+2.5 −0	+11.6 +10.0
5.52– 6.30	1.5 3.2	+1.0 −0	+3.2 +2.5	2.4 5.0	+1.6 −0	+5.0 +4.0	3.4 6.0	+1.6 −0	+ 6.0 + 5.0	5.4 8.0	+1.6 −0	+ 8.0 + 7.0	9.5 13.6	+2.5 −0	+13.6 +12.0
6.30– 7.09	1.8 3.5	+1.0 −0	+3.5 +2.8	2.9 5.5	+1.6 −0	+5.5 +4.5	4.4 7.0	+1.6 −0	+ 7.0 + 6.0	6.4 9.0	+1.6 −0	+ 9.0 + 8.0	9.5 13.6	+2.5 −0	+13.6 +12.0
7.09– 7.88	1.8 3.8	+1.2 −0	+3.8 +3.0	3.2 6.2	+1.8 −0	+6.2 +5.0	5.2 8.2	+1.8 −0	+ 8.2 + 7.0	7.2 10.2	+1.8 −0	+10.2 + 9.0	11.2 15.8	+2.8 −0	+15.8 +14.0
7.88– 8.86	2.3 4.3	+1.2 −0	+4.3 +3.5	3.2 6.2	+1.8 −0	+6.2 +5.0	5.2 8.2	+1.8 −0	+ 8.2 + 7.0	8.2 11.2	+1.8 −0	+11.2 +10.0	13.2 17.8	+2.8 −0	+17.8 +16.0
8.86– 9.85	2.3 4.3	+1.2 −0	+4.3 +3.5	4.2 7.2	+1.8 −0	+7.2 +6.0	6.2 9.2	+1.8 −0	+ 9.2 + 8.0	10.2 13.2	+1.8 −0	+13.2 +12.0	13.2 17.8	+2.8 −0	+17.8 +16.0
9.85–11.03	2.8 4.9	+1.2 −0	+4.9 +4.0	4.0 7.2	+2.0 −0	+7.2 +6.0	7.0 10.2	+2.0 −0	+10.2 + 9.0	10.0 13.2	+2.0 −0	+13.2 +12.0	15.0 20.0	+3.0 −0	+20.0 +18.0
11.03–12.41	2.8 4.9	+1.2 −0	+4.9 +4.0	5.0 8.2	+2.0 −0	+8.2 +7.0	7.0 10.2	+2.0 −0	+10.2 + 9.0	12.0 15.2	+2.0 −0	+15.2 +14.0	17.0 22.0	+3.0 −0	+22.0 +20.0
12.41–13.98	3.1 5.5	+1.4 −0	+5.5 +4.5	5.8 9.4	+2.2 −0	+9.4 +8.0	7.8 11.4	+2.2 −0	+11.4 +10.0	13.8 17.4	+2.2 −0	+17.4 +16.0	18.5 24.2	+3.5 +0	+24.2 +22.0

[a] From ANSI B4.1—1967 (R1987). For larger diameters, see the standard.

APPENDIX 6

International Tolerance Grades[a]

Dimensions are in millimeters.

Basic sizes		Tolerance grades[b]																	
Over	Up to and Including	IT01	IT0	IT1	IT2	IT3	IT4	IT5	IT6	IT7	IT8	IT9	IT10	IT11	IT12	IT13	IT14	IT15	IT16
0	3	0.0003	0.0005	0.0008	0.0012	0.002	0.003	0.004	0.006	0.010	0.014	0.025	0.040	0.060	0.100	0.140	0.250	0.400	0.600
3	6	0.0004	0.0006	0.001	0.0015	0.0025	0.004	0.005	0.008	0.012	0.018	0.030	0.048	0.075	0.120	0.180	0.300	0.480	0.750
6	10	0.0004	0.0006	0.001	0.0015	0.0025	0.004	0.006	0.009	0.015	0.022	0.036	0.058	0.090	0.150	0.220	0.360	0.580	0.900
10	18	0.0005	0.0008	0.0012	0.002	0.003	0.005	0.008	0.011	0.018	0.027	0.043	0.070	0.110	0.180	0.270	0.430	0.700	1.100
18	30	0.0006	0.001	0.0015	0.0025	0.004	0.006	0.009	0.013	0.021	0.033	0.052	0.084	0.130	0.210	0.330	0.520	0.840	1.300
30	50	0.0006	0.001	0.0015	0.0025	0.004	0.007	0.011	0.016	0.025	0.039	0.062	0.100	0.160	0.250	0.390	0.620	1.000	1.600
50	80	0.0008	0.0012	0.002	0.003	0.005	0.008	0.013	0.019	0.030	0.046	0.074	0.120	0.190	0.300	0.460	0.740	1.200	1.900
80	120	0.001	0.0015	0.0025	0.004	0.006	0.010	0.015	0.022	0.035	0.054	0.087	0.140	0.220	0.350	0.540	0.870	1.400	2.200
120	180	0.0012	0.002	0.0035	0.005	0.008	0.012	0.018	0.025	0.040	0.063	0.100	0.160	0.250	0.400	0.630	1.000	1.600	2.500
180	250	0.002	0.003	0.0045	0.007	0.010	0.014	0.020	0.029	0.046	0.072	0.115	0.185	0.290	0.460	0.720	1.150	1.850	2.900
250	315	0.0025	0.004	0.006	0.008	0.012	0.016	0.023	0.032	0.052	0.081	0.130	0.210	0.320	0.520	0.810	1.300	2.100	3.200
315	400	0.003	0.005	0.007	0.009	0.013	0.018	0.025	0.036	0.057	0.089	0.140	0.230	0.360	0.570	0.890	1.400	2.300	3.600
400	500	0.004	0.006	0.008	0.010	0.015	0.020	0.027	0.040	0.063	0.097	0.155	0.250	0.400	0.630	0.970	1.550	2.500	4.000
500	630	0.0045	0.006	0.009	0.011	0.016	0.022	0.030	0.044	0.070	0.110	0.175	0.280	0.440	0.700	1.100	1.750	2.800	4.400
630	800	0.005	0.007	0.010	0.013	0.018	0.025	0.035	0.050	0.080	0.125	0.200	0.320	0.500	0.800	1.250	2.000	3.200	5.000
800	1000	0.0055	0.008	0.011	0.015	0.021	0.029	0.040	0.056	0.090	0.140	0.230	0.360	0.560	0.900	1.400	2.300	3.600	5.600
1000	1250	0.0065	0.009	0.013	0.018	0.024	0.034	0.046	0.066	0.105	0.165	0.260	0.420	0.660	1.050	1.650	2.600	4.200	6.600
1250	1600	0.008	0.011	0.015	0.021	0.029	0.040	0.054	0.078	0.125	0.195	0.310	0.500	0.780	1.250	1.950	3.100	5.000	7.800
1600	2000	0.009	0.013	0.018	0.025	0.035	0.048	0.065	0.092	0.150	0.230	0.370	0.600	0.920	1.500	2.300	3.700	6.000	9.200
2000	2500	0.011	0.015	0.022	0.030	0.041	0.057	0.077	0.110	0.175	0.280	0.440	0.700	1.100	1.750	2.800	4.400	7.000	11.000
2500	3150	0.013	0.018	0.026	0.036	0.050	0.069	0.093	0.135	0.210	0.330	0.540	0.860	1.350	2.100	3.300	5.400	8.600	13.500

[a]From ANSI B4.2–1978 (R1984).

[b]IT Values for tolerance grades larger than IT16 can be calculated by using the formulas: IT17 = IT × 10, IT18 = IT13 × 10, etc.

APPENDIX 7

Preferred Metric Hole Basis Clearance Fits[a]—American National Standard

Dimensions are in millimeters.

Basic Size		Loose Running Hole H11	Loose Running Shaft c11	Loose Running Fit	Free Running Hole H9	Free Running Shaft d9	Free Running Fit	Close Running Hole H8	Close Running f7	Close Running Fit	Sliding Hole H7	Sliding Shaft g6	Sliding Fit	Locational Clearance Hole H7	Locational Clearance Shaft h6	Locational Clearance Fit
1	Max	1.060	0.940	0.180	1.025	0.980	0.070	1.014	0.994	0.030	1.010	0.998	0.018	1.010	1.000	0.016
	Min	1.060	0.880	0.060	1.000	0.955	0.020	1.000	0.984	0.006	1.000	0.992	0.002	1.000	0.994	0.000
1.2	Max	1.260	1.140	0.180	1.225	1.180	0.070	1.214	1.194	0.030	1.210	1.198	0.018	1.210	1.200	0.016
	Min	1.200	1.080	0.060	1.200	1.155	0.020	1.200	1.184	0.036	1.200	1.192	0.002	1.200	1.194	0.000
1.6	Max	1.660	1.540	0.180	1.625	1.580	0.070	1.614	1.594	0.030	1.610	1.598	0.018	1.610	1.600	0.016
	Min	1.600	1.480	0.060	1.600	1.555	0.020	1.600	1.584	0.006	1.600	1.592	0.002	1.600	1.594	0.000
2	Max	2.060	1.940	0.180	2.025	1.980	0.070	2.014	1.994	0.030	2.010	1.998	0.018	2.010	2.000	0.016
	Min	2.000	1.880	0.060	2.000	1.955	0.020	2.000	1.984	0.006	2.000	1.992	0.002	2.000	1.994	0.000
2.5	Max	2.560	2.440	0.180	2.525	2.480	0.070	2.514	2.494	0.030	2.510	2.498	0.018	2.510	2.500	0.016
	Min	2.500	2.380	0.060	2.500	2.455	0.020	2.500	2.484	0.006	2.500	2.492	0.002	2.500	2.494	0.000
3	Max	3.060	2.940	0.180	3.025	2.980	0.070	3.014	2.994	0.030	3.010	2.998	0.018	3.010	3.000	0.016
	Min	3.000	2.880	0.060	3.000	2.955	0.020	3.000	2.984	0.006	3.000	2.992	0.002	3.000	2.994	0.000
4	Max	4.075	3.930	0.220	4.030	3.970	0.090	4.018	3.990	0.040	4.012	3.996	0.024	4.012	4.000	0.020
	Min	4.000	3.855	0.070	4.000	3.940	0.030	4.000	3.978	0.010	4.000	3.988	0.004	4.000	3.992	0.000
5	Max	5.075	4.930	0.220	5.030	4.970	0.090	5.018	4.990	0.040	5.012	4.996	0.024	5.012	5.000	0.020
	Min	5.000	4.855	0.070	5.000	4.940	0.030	5.000	4.978	0.010	5.000	4.988	0.004	5.000	4.992	0.000
6	Max	6.075	5.930	0.220	6.030	5.970	0.090	6.018	5.990	0.040	6.012	5.996	0.024	6.012	6.000	0.020
	Min	6.000	5.855	0.070	6.000	5.940	0.030	6.000	5.978	0.010	6.000	5.988	0.004	6.000	5.992	0.000
8	Max	8.090	7.920	0.260	8.036	7.960	0.112	8.022	7.987	0.050	8.015	7.995	0.029	8.015	8.000	0.024
	Min	8.000	7.830	0.080	8.000	7.924	0.040	8.000	7.972	0.013	8.000	7.986	0.005	8.000	7.991	0.000
10	Max	10.090	9.920	0.260	10.036	9.960	0.112	10.022	9.987	0.050	10.015	9.995	0.029	10.015	10.000	0.024
	Min	10.000	9.830	0.080	10.000	9.924	0.040	10.000	9.972	0.013	10.000	9.986	0.005	10.000	9.991	0.000
12	Max	12.110	11.905	0.315	12.043	11.950	0.136	12.027	11.984	0.061	12.018	11.994	0.035	12.018	12.000	0.029
	Min	12.000	11.795	0.095	12.000	11.907	0.050	12.000	11.966	0.016	12.000	11.983	0.006	12.000	11.989	0.000
16	Max	16.110	15.905	0.315	16.043	15.950	0.136	16.027	15.984	0.061	16.018	15.994	0.035	16.018	16.000	0.029
	Min	16.000	15.795	0.095	16.000	15.907	0.050	16.000	15.966	0.016	16.000	15.983	0.006	16.000	15.989	0.000
20	Max	20.130	19.890	0.370	20.052	19.935	0.169	20.033	19.980	0.074	20.021	19.993	0.041	20.021	20.000	0.034
	Min	20.000	19.760	0.110	20.000	19.883	0.065	20.000	19.959	0.020	20.000	19.980	0.007	20.000	19.987	0.000
25	Max	25.130	24.890	0.370	25.052	24.935	0.169	25.033	24.980	0.074	25.021	24.993	0.041	25.021	25.000	0.034
	Min	25.000	24.760	0.110	25.000	24.883	0.065	25.000	24.959	0.020	25.000	24.980	0.007	25.000	24.987	0.000
30	Max	30.130	29.890	0.370	30.052	29.935	0.169	30.033	29.980	0.074	30.021	29.993	0.041	30.021	30.000	0.034
	Min	30.000	29.760	0.110	30.000	29.883	0.065	30.000	29.959	0.020	30.000	29.980	0.007	30.000	29.987	0.000

[a]From ANSI B4.2—1978 (R1984). For description of preferred fits, see Table 14.2.

APPENDIX 7

Preferred Metric Hole Basis Clearance Fits[a]— American National Standard (continued)

Dimensions are in millimeters.

Basic Size		Loose Running Hole H11	Loose Running Shaft c11	Loose Running Fit	Free Running Hole H9	Free Running Shaft d9	Free Running Fit	Close Running Hole H8	Close Running Shaft f7	Close Running Fit	Sliding Hole H7	Sliding Shaft g6	Sliding Fit	Locational Clearance Hole H7	Locational Clearance Shaft h6	Locational Clearance Fit
40	Max	40.160	39.880	0.440	40.062	39.920	0.204	40.039	39.975	0.089	40.025	39.991	0.050	40.025	40.000	0.041
	Min	40.000	39.720	0.120	40.000	39.858	0.080	40.000	39.950	0.025	40.000	39.975	0.009	40.000	39.984	0.000
50	Max	50.160	49.870	0.450	50.062	49.920	0.204	50.039	49.975	0.089	50.025	49.991	0.050	50.025	50.000	0.041
	Min	50.000	49.710	0.130	50.000	49.858	0.080	50.000	49.950	0.025	50.000	49.975	0.009	50.000	49.984	0.000
60	Max	60.190	59.860	0.520	60.074	59.900	0.248	60.046	59.970	0.106	60.030	59.990	0.059	60.030	60.000	0.049
	Min	60.000	59.670	0.140	60.000	59.826	0.100	60.000	59.940	0.030	60.000	59.971	0.010	60.000	59.981	0.000
80	Max	80.190	79.950	0.530	80.074	79.900	0.248	80.046	79.970	0.106	80.030	79.990	0.059	80.030	80.000	0.049
	Min	80.000	79.660	0.150	80.000	79.826	0.100	80.000	79.940	0.030	80.000	79.971	0.010	80.000	79.981	0.000
100	Max	100.220	99.830	0.610	100.087	99.880	0.294	100.054	99.964	0.125	100.035	99.988	0.069	100.035	100.000	0.057
	Min	100.000	99.610	0.170	100.000	99.793	0.120	100.000	99.929	0.036	100.000	99.966	0.012	100.000	99.978	0.000
120	Max	120.220	119.820	0.620	120.087	119.880	0.294	120.054	119.964	0.125	120.035	119.988	0.069	120.035	120.000	0.057
	Min	120.000	119.600	0.180	120.000	119.793	0.120	120.000	119.929	0.036	120.000	119.966	0.012	120.000	119.978	0.000
160	Max	160.250	159.790	0.710	160.100	159.855	0.345	160.063	159.957	0.146	160.000	159.986	0.079	160.040	160.000	0.065
	Min	160.000	159.540	0.210	160.000	159.755	0.145	160.000	159.917	0.043	160.000	159.961	0.014	160.000	159.975	0.000
200	Max	200.290	199.760	0.820	200.115	199.830	0.400	200.072	199.950	0.168	200.046	199.985	0.090	200.046	200.000	0.075
	Min	200.000	199.470	0.240	200.000	199.715	0.170	200.000	199.904	0.050	200.000	199.956	0.015	200.000	199.971	0.000
250	Max	250.290	249.720	0.860	250.115	249.830	0.400	250.072	249.950	0.168	250.046	249.985	0.090	250.046	250.000	0.075
	Min	250.000	249.430	0.280	250.000	249.715	0.170	250.000	249.904	0.050	250.000	249.956	0.015	250.000	249.971	0.000
300	Max	300.320	299.670	0.970	300.130	299.810	0.450	300.081	299.944	0.189	300.052	299.983	0.101	300.052	300.000	0.084
	Min	300.000	299.350	0.330	300.000	299.680	0.190	300.000	299.892	0.056	300.000	299.951	0.017	300.000	299.968	0.000
400	Max	400.360	399.600	1.120	400.140	399.790	0.490	400.089	399.938	0.208	400.057	399.982	0.111	400.057	400.000	0.093
	Min	400.000	399.240	0.400	400.000	399.650	0.210	400.000	399.881	0.062	400.000	399.946	0.018	400.000	399.964	0.000
500	Max	500.400	499.520	1.280	500.155	499.770	0.540	500.097	499.932	0.228	500.063	499.980	0.123	500.063	500.000	0.103
	Min	500.000	499.120	0.480	500.000	499.615	0.230	500.000	499.869	0.068	500.000	499.940	0.020	500.000	499.960	0.000

[a]From ANSI B4.2—1978 (R1984). For description of preferred fits, see Table 14.2.

APPENDIX 8

Preferred Metric Hole Basis Transition and Interference Fits[a]—American National Standard

Dimensions are in millimeters.

Basic Size		Locational Transn. Hole H7	Shaft k6	Fit	Locational Transn. Hole H7	Shaft n6	Fit	Locational Interf. Hole H7	Shaft p6	Fit	Medium Drive Hole H7	Shaft s6	Fit	Force Hole H7	Shaft u6	Fit
1	Max	1.010	1.006	0.010	1.010	1.010	0.006	1.010	1.012	0.004	1.010	1.020	−0.004	1.010	1.024	−0.008
	Min	1.000	1.000	−0.006	1.000	1.004	−0.010	1.000	1.006	−0.012	1.000	1.014	−0.020	1.000	1.018	−0.024
1.2	Max	1.210	1.206	0.010	1.210	1.210	0.006	1.210	1.212	0.004	1.210	1.220	−0.004	1.210	1.224	−0.008
	Min	1.200	1.200	−0.006	1.200	1.204	−0.010	1.200	1.206	−0.012	1.200	1.214	−0.020	1.200	1.218	−0.024
1.6	Max	1.610	1.606	0.010	1.610	1.610	0.006	1.610	1.612	0.004	1.610	1.620	−0.004	1.610	1.624	−0.008
	Min	1.600	1.600	−0.006	1.600	1.604	−0.010	1.600	1.606	−0.012	1.600	1.614	−0.020	1.600	1.618	−0.024
2	Max	2.010	2.006	0.010	2.010	2.010	0.006	2.010	2.012	0.004	2.010	2.020	−0.004	2.010	2.024	−0.008
	Min	2.000	2.000	−0.006	2.000	2.004	−0.010	2.000	2.006	−0.012	2.000	2.014	−0.020	2.000	2.018	−0.024
2.5	Max	2.510	2.506	0.010	2.510	2.510	0.006	2.510	2.512	0.004	2.510	2.520	−0.004	2.510	2.524	−0.008
	Min	2.500	2.500	−0.006	2.500	2.504	−0.010	2.500	2.506	−0.012	2.500	2.514	−0.020	2.500	2.518	−0.024
3	Max	3.010	3.006	0.010	3.010	3.010	0.006	3.010	3.012	0.004	3.010	3.020	−0.004	3.010	3.024	−0.008
	Min	3.000	3.000	−0.006	3.000	3.004	−0.010	3.000	3.006	−0.012	3.000	3.014	−0.020	3.000	3.018	−0.024
4	Max	4.012	4.009	0.011	4.012	4.016	0.004	4.012	4.020	0.000	4.012	4.027	−0.007	4.012	4.031	−0.011
	Min	4.000	4.001	−0.009	4.000	4.008	−0.016	4.000	4.012	−0.020	4.000	4.019	−0.027	4.000	4.023	−0.031
5	Max	5.012	5.009	0.011	5.012	5.016	0.004	5.012	5.020	0.000	5.012	5.027	−0.007	5.012	5.031	−0.011
	Min	5.000	5.001	−0.009	5.000	5.008	−0.016	5.000	5.012	−0.020	5.000	5.019	−0.027	5.000	5.023	−0.031
6	Max	6.012	6.009	0.011	6.012	6.016	0.004	6.012	6.020	0.000	6.012	6.027	−0.007	6.012	6.031	−0.011
	Min	6.000	6.001	−0.009	6.000	6.008	−0.016	6.000	6.012	−0.020	6.000	6.019	−0.027	6.000	6.023	−0.031
8	Max	8.015	8.010	0.014	8.015	8.019	0.005	8.015	8.024	0.000	8.015	8.032	−0.008	8.015	8.037	−0.013
	Min	8.000	8.001	−0.010	8.000	8.010	−0.019	8.000	8.015	−0.024	8.000	8.023	−0.032	8.000	8.028	−0.037
10	Max	10.015	10.010	0.014	10.015	10.019	0.005	10.015	10.024	0.000	10.015	10.032	−0.008	10.015	10.037	−0.013
	Min	10.000	10.001	−0.010	10.000	10.010	−0.019	10.000	10.015	−0.024	10.000	10.023	−0.032	10.000	10.028	−0.037
12	Max	12.018	12.012	0.017	12.018	12.023	0.006	12.018	12.029	0.000	12.018	12.039	−0.010	12.018	12.044	−0.015
	Min	12.000	12.001	−0.012	12.000	12.012	−0.023	12.000	12.018	−0.029	12.000	12.028	−0.039	12.000	12.033	−0.044
16	Max	16.018	16.012	0.017	16.018	16.023	0.006	16.018	16.029	0.000	16.018	16.039	−0.010	16.018	16.044	−0.015
	Min	16.000	16.001	−0.012	16.000	16.012	−0.023	16.000	16.018	−0.029	16.000	16.028	−0.039	16.000	16.033	−0.044
20	Max	20.081	20.021	0.019	20.021	20.028	0.006	20.021	20.035	−0.001	20.021	20.048	−0.014	20.021	20.054	−0.020
	Min	20.000	20.002	−0.015	20.000	20.015	−0.028	20.000	20.022	−0.035	20.000	20.035	−0.048	20.000	20.041	−0.054
25	Max	25.021	25.015	0.019	25.021	25.028	0.006	25.021	25.035	−0.001	25.021	25.048	−0.014	25.021	25.061	−0.027
	Min	25.000	25.002	−0.015	25.000	25.015	−0.028	25.000	25.022	−0.035	25.000	25.035	−0.048	25.000	25.048	−0.061
30	Max	30.021	30.015	0.019	30.021	30.028	0.006	30.021	30.035	−0.001	30.021	30.048	−0.014	30.021	30.061	−0.027
	Min	30.000	30.002	−0.015	30.000	30.015	−0.028	30.000	30.022	−0.035	30.000	30.035	−0.048	30.000	30.048	−0.061

[a]From ANSI B4.2—1978. For description of preferred fits, see Table 14.2.

APPENDIX 9

Preferred Metric Shaft Basis Clearance Fits[a]— American National Standard (continued)

Dimensions are in millimeters.

Basic Size		Loose Running Hole C11	Loose Running Shaft h11	Loose Running Fit	Free Running Hole D9	Free Running Shaft h9	Free Running Fit	Close Running Hole F8	Close Running Shaft h7	Close Running Fit	Sliding Hole G7	Sliding Shaft h6	Sliding Fit	Locational Clearance Hole H7	Locational Clearance Shaft h6	Locational Clearance Fit
40	Max	40.280	40.000	0.440	40.142	40.000	0.204	40.064	40.000	0.089	40.034	40.000	0.050	40.025	40.000	0.041
	Min	40.120	39.840	0.120	40.080	39.938	0.080	40.025	39.975	0.025	40.009	39.984	0.009	40.000	39.984	0.000
50	Max	50.290	50.000	0.450	50.142	50.000	0.204	50.064	50.000	0.089	50.034	50.000	0.050	50.025	50.000	0.041
	Min	50.130	49.840	0.130	50.080	49.938	0.080	50.025	49.975	0.025	50.009	49.984	0.009	50.000	49.984	0.000
60	Max	60.330	60.000	0.520	60.174	60.000	0.248	60.076	60.000	0.106	60.040	60.000	0.059	60.030	60.000	0.049
	Min	60.140	59.810	0.140	60.100	59.926	0.100	60.030	59.970	0.030	60.010	59.981	0.010	60.000	59.981	0.000
80	Max	80.340	80.000	0.530	80.174	80.000	0.248	80.076	80.000	0.106	80.040	80.000	0.059	80.030	80.000	0.049
	Min	80.150	79.810	0.150	80.100	79.926	0.100	80.030	79.970	0.030	80.010	79.981	0.010	80.000	79.981	0.000
100	Max	100.390	100.000	0.610	100.207	100.000	0.294	100.090	100.000	0.125	100.047	100.000	0.069	100.035	100.000	0.057
	Min	100.170	99.780	0.170	100.120	99.913	0.120	100.036	99.965	0.036	100.012	99.978	0.012	100.000	99.978	0.000
120	Max	120.400	120.000	0.620	120.207	120.000	0.294	120.090	120.000	0.125	120.047	120.000	0.069	120.035	120.000	0.057
	Min	120.180	119.780	0.180	120.120	119.913	0.120	120.036	119.965	0.036	120.012	119.978	0.012	120.000	119.978	0.000
160	Max	160.460	160.000	0.710	160.245	160.000	0.345	160.106	160.000	0.146	160.054	160.000	0.079	160.040	160.000	0.065
	Min	160.210	159.750	0.210	160.145	159.900	0.145	160.043	159.960	0.043	160.014	159.975	0.014	160.000	159.975	0.000
200	Max	200.530	200.000	0.820	200.285	200.000	0.400	200.122	200.000	0.168	200.061	200.000	0.090	200.046	200.000	0.075
	Min	200.240	199.710	0.240	200.170	199.885	0.170	200.050	199.954	0.050	200.015	199.971	0.015	200.000	199.971	0.000
250	Max	250.570	250.000	0.860	250.285	250.000	0.400	250.122	250.000	0.168	250.061	250.000	0.090	250.046	250.000	0.075
	Min	250.280	249.710	0.280	250.170	249.885	0.170	250.050	249.954	0.050	250.015	249.971	0.015	250.000	249.971	0.000
300	Max	300.650	300.000	0.970	300.320	300.000	0.450	300.137	300.000	0.189	300.069	300.000	0.101	300.052	300.000	0.084
	Min	300.330	299.680	0.330	300.190	299.870	0.190	300.056	299.948	0.056	300.017	299.968	0.017	300.000	299.968	0.000
400	Max	400.760	400.000	1.120	400.350	400.000	0.490	400.151	400.000	0.208	400.075	400.000	0.111	400.057	400.000	0.093
	Min	400.400	399.640	0.400	400.210	399.860	0.210	400.062	399.943	0.062	400.018	399.964	0.018	400.000	399.964	0.000
500	Max	500.880	500.000	1.280	500.385	500.000	0.540	500.165	500.000	0.228	500.083	500.000	0.123	500.063	500.000	0.103
	Min	500.480	499.600	0.480	500.230	499.845	0.230	500.068	499.937	0.068	500.020	499.960	5.020	500.000	499.960	0.000

[a]From ANSI B.42-1978. For description of preferred fits, see Table 14.2.

Preferred Metric Shaft Basis Transition and Interference Fits[a]— American National Standard

Dimensions are in millimeters.

Basic Size		Locational Transn. Hole K7	Shaft h6	Fit	Locational Transn. Hole N7	Shaft h6	Fit	Locational Interf. Hole P7	Shaft h6	Fit	Medium Drive Hole S7	Shaft h6	Fit	Force Hole U7	Shaft h6	Fit
1	Max	1.000	1.000	0.006	0.996	1.000	0.002	0.994	1.000	0.000	0.986	1.000	−0.008	0.982	1.000	−0.012
	Min	0.990	0.994	−0.010	0.986	0.994	−0.014	0.984	0.994	−0.016	0.976	0.994	−0.024	0.972	0.994	−0.028
1.2	Max	1.200	1.200	0.006	1.196	1.200	0.002	1.194	1.200	0.000	1.186	1.200	−0.008	1.182	1.200	−0.012
	Min	1.190	1.194	−0.010	1.186	1.194	−0.014	1.184	1.194	−0.016	1.176	1.194	−0.024	1.172	1.194	−0.028
1.6	Max	1.600	1.600	0.006	1.596	1.600	0.002	1.594	1.600	0.000	1.586	1.600	−0.008	1.582	1.600	−0.012
	Min	1.590	1.594	−0.010	1.586	1.594	−0.014	1.584	1.594	−0.016	1.576	1.594	−0.024	1.572	1.594	−0.028
2	Max	2.000	2.000	0.006	1.996	2.000	0.002	1.994	2.000	0.000	1.986	2.000	−0.008	1.982	2.000	−0.012
	Min	1.990	1.994	−0.010	1.986	1.994	−0.014	1.984	1.994	−0.016	1.976	1.994	−0.024	1.972	1.994	−0.028
2.5	Max	2.500	2.500	0.006	2.496	2.500	0.002	2.494	2.500	0.000	2.486	2.500	−0.008	2.482	2.500	−0.012
	Min	2.490	2.494	−0.010	2.486	2.494	−0.014	2.484	2.494	−0.016	2.476	2.494	−0.024	2.472	2.494	−0.028
3	Max	3.000	3.000	0.006	2.996	3.000	0.002	2.994	3.000	0.000	2.986	3.000	−0.008	2.982	3.000	−0.012
	Min	2.990	2.994	−0.010	2.986	2.994	−0.014	2.984	2.994	−0.016	2.976	2.994	−0.024	2.972	2.994	−0.028
4	Max	4.003	4.000	0.011	3.996	4.000	0.004	3.992	4.000	0.000	3.985	4.000	−0.007	3.981	4.000	−0.011
	Min	3.991	3.992	−0.009	3.984	3.992	−0.016	3.980	3.992	−0.020	3.973	3.992	−0.027	3.969	3.992	−0.031
5	Max	5.003	5.000	0.011	4.996	5.000	0.004	4.992	5.000	0.000	4.985	5.000	−0.007	4.981	5.000	−0.011
	Min	4.991	4.992	−0.009	4.984	4.992	−0.016	4.980	4.992	−0.020	4.973	4.992	−0.027	4.969	4.992	−0.031
6	Max	6.003	6.000	0.011	5.996	6.000	0.004	5.992	6.000	0.000	5.985	6.000	−0.007	5.981	6.000	−0.011
	Min	5.991	5.992	−0.009	5.984	5.992	−0.016	5.980	5.992	−0.020	5.973	5.992	−0.027	5.969	5.992	−0.031
8	Max	8.005	8.000	0.014	7.996	8.000	0.005	7.991	8.000	0.000	7.983	8.000	−0.008	7.978	8.000	−0.013
	Min	7.990	7.991	−0.010	7.981	7.991	−0.019	7.976	7.991	−0.024	7.968	7.991	−0.032	7.963	7.991	−0.037
10	Max	10.005	10.000	0.014	9.996	10.000	0.005	9.991	10.000	0.000	9.983	10.000	−0.008	9.978	10.000	−0.013
	Min	9.990	9.991	−0.010	9.981	9.991	−0.019	9.976	9.991	−0.024	9.968	9.991	−0.032	9.963	9.991	−0.037
12	Max	12.006	12.000	0.017	11.995	12.000	0.006	11.989	12.000	0.000	11.979	12.000	−0.010	11.974	12.000	−0.015
	Min	11.988	11.989	−0.012	11.977	11.989	−0.023	11.971	11.989	−0.029	11.961	11.989	−0.039	11.956	11.989	−0.044
16	Max	16.006	16.000	0.017	15.995	16.000	0.006	15.989	16.000	0.000	15.979	16.000	−0.010	15.974	16.000	−0.015
	Min	15.988	15.989	−0.012	15.977	15.989	−0.023	15.971	15.989	−0.029	15.961	15.989	−0.039	15.956	15.989	−0.044
20	Max	20.006	20.000	0.019	19.993	20.000	0.006	19.986	20.000	−0.001	19.973	20.000	−0.014	19.967	20.000	−0.020
	Min	19.985	19.987	−0.015	19.972	19.987	−0.028	19.965	19.987	−0.035	19.952	19.987	−0.048	19.946	19.987	−0.054
25	Max	25.006	25.000	0.019	24.993	25.000	0.006	24.986	25.000	−0.001	24.973	25.000	−0.014	24.960	25.000	−0.027
	Min	24.985	24.987	−0.015	24.972	24.987	−0.028	24.965	24.987	−0.035	24.952	24.987	−0.048	24.939	24.987	−0.061
30	Max	30.006	30.000	0.019	29.993	30.000	0.006	29.986	30.000	−0.001	29.973	30.000	−0.014	29.960	30.000	−0.027
	Min	29.985	29.987	−0.015	29.972	29.987	−0.028	29.965	29.987	−0.035	29.952	29.987	−0.048	29.939	29.987	−0.061

[a]From ANSI B4.2—1978 (R1984). For description of preferred fits, see Table 14.2.

APPENDIX 10

Preferred Metric Basis Transition and Interference Fits[a]— American National Standard (continued)

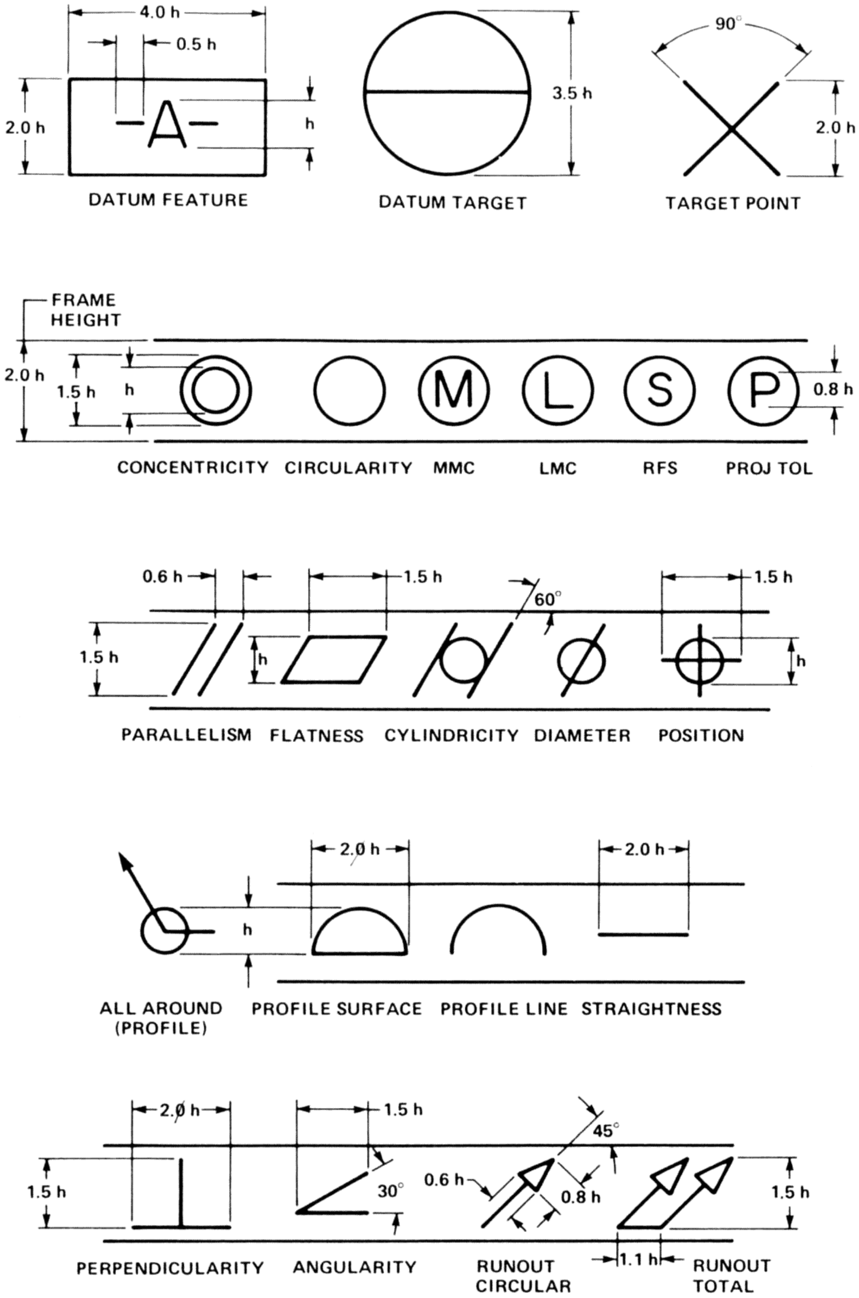
Dimensions are in millimeters.

Basic Size		Locational Transn.			Locational Transn.			Locational Interf.			Medium Drive			Force		
		Hole K7	Shaft h6	Fit	Hole N7	Shaft h6	Fit	Hole P7	Shaft h6	Fit	Hole S7	Shaft h6	Fit	Hole U7	Shaft h6	Fit
40	Max	40.007	40.000	0.023	39.992	40.000	0.008	39.983	40.000	−0.001	39.966	40.000	−0.018	39.949	40.000	−0.035
	Min	39.982	39.984	−0.018	39.967	39.984	−0.033	39.958	39.984	−0.042	39.941	39.984	−0.059	39.924	39.984	−0.076
50	Max	50.007	50.000	0.023	49.992	50.000	0.008	49.983	50.000	−0.001	49.966	50.000	−0.018	49.939	50.000	−0.045
	Min	49.982	49.984	−0.018	49.967	49.984	−0.033	49.958	49.984	−0.042	49.941	49.984	−0.059	49.914	49.984	−0.086
60	Max	60.009	60.000	0.028	59.991	60.000	0.010	59.979	60.000	−0.002	59.958	60.000	−0.023	59.924	60.000	−0.057
	Min	59.979	59.981	−0.021	59.961	59.981	−0.039	59.949	59.981	−0.051	59.928	59.981	−0.072	59.894	59.981	−0.106
80	Max	80.009	80.000	0.028	79.991	80.000	0.010	79.979	80.000	−0.002	79.952	80.000	−0.029	79.909	80.000	−0.072
	Min	79.979	79.981	−0.021	79.961	79.981	−0.039	79.949	79.981	−0.051	79.922	79.981	−0.078	79.879	79.981	−0.121
100	Max	100.010	100.000	0.032	99.990	100.000	0.012	99.976	100.000	−0.002	99.942	100.000	−0.036	99.889	100.000	−0.089
	Min	99.975	99.978	−0.025	99.955	99.978	−0.045	99.941	99.978	−0.059	99.907	99.978	−0.093	99.854	99.978	−0.146
120	Max	120.010	120.000	0.032	119.990	120.000	0.012	119.976	120.000	−0.002	119.934	120.000	−0.044	119.869	120.000	−0.109
	Min	119.975	119.978	−0.025	119.955	199.978	−0.045	119.941	119.978	−0.059	119.899	119.978	−0.101	119.834	119.978	−0.166
160	Max	160.012	160.000	0.037	159.988	160.000	0.013	159.972	160.000	−0.003	159.915	160.000	−0.060	159.825	160.000	−0.150
	Min	159.972	159.975	−0.028	159.948	159.975	−0.052	159.932	159.975	−0.068	159.875	159.975	−0.125	159.785	159.975	−0.215
200	Max	200.013	200.000	0.042	199.986	200.000	0.015	199.967	200.000	−0.004	199.895	200.000	−0.076	199.781	200.000	−0.190
	Min	199.967	199.971	−0.033	199.940	199.971	−0.060	199.921	199.971	−0.079	199.849	199.971	−0.151	199.735	199.971	−0.265
250	Max	250.013	250.000	0.042	249.986	250.000	0.015	249.967	250.000	−0.004	249.877	250.000	−0.094	249.733	250.000	−0.238
	Min	249.967	249.971	−0.033	249.940	249.971	−0.060	249.921	249.971	−0.079	249.831	249.971	−0.169	249.687	249.971	−0.313
300	Max	300.016	300.000	0.048	299.986	300.000	0.018	299.964	300.000	−0.004	299.850	300.000	−0.118	299.670	300.000	−0.298
	Min	299.964	299.968	−0.036	299.934	299.968	−0.066	299.912	299.968	−0.088	299.798	299.968	−0.202	299.618	299.968	−0.382
400	Max	400.017	400.000	0.053	399.984	400.000	0.020	399.959	400.000	−0.005	399.813	400.000	−0.151	399.586	400.000	−0.378
	Min	399.960	399.964	−0.040	399.927	399.964	−0.073	399.902	399.964	−0.098	399.756	399.964	−0.244	399.529	399.964	−0.471
500	Max	500.018	500.000	0.058	499.983	500.000	0.023	499.955	500.000	−0.005	499.771	500.000	−0.189	499.483	500.000	−0.477
	Min	499.955	499.960	−0.045	499.920	499.960	−0.080	499.892	499.960	−0.108	499.708	499.960	−0.292	499.420	499.960	−0.580

[a]From ANSI B4.2—1978 (R1984). For description of preferred fits, see Table 14.2.

APPENDIX 11

Form and Proportion of Geometric Tolerancing Symbols[a]



Revised By

Name: Mr. Kidu G/Cherkos

Qualification: MSc. Degree in Manufacturing Engineering

FTI, Addis-Ababa, Ethiopia

E-mail: kidmech2000@gmail.com

Phone: +251913830830