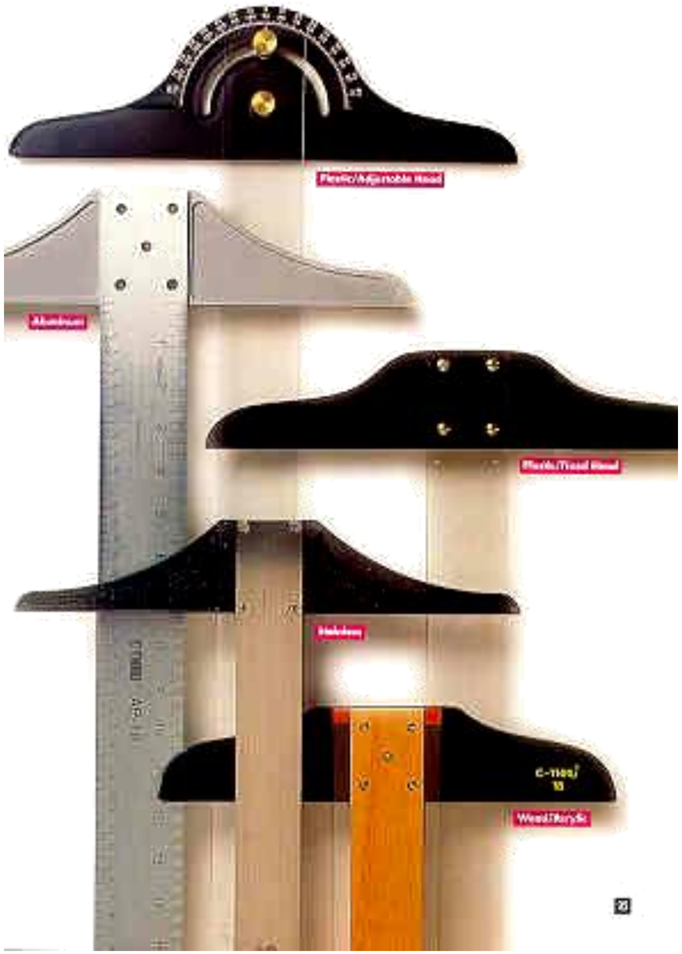
Engineering Drawing

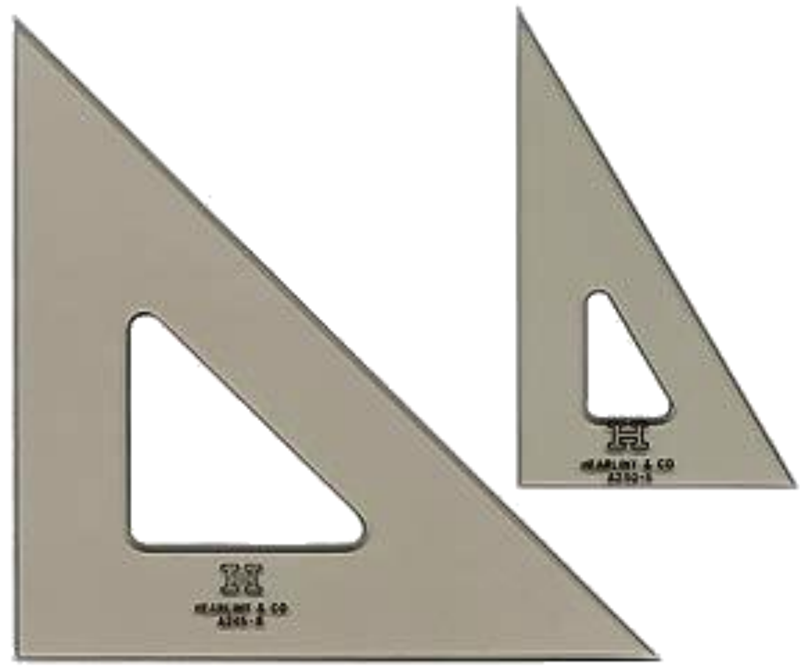
Traditional Drawing Tools

DRAWING TOOLS

DRAWING TOOLS



1. T-Square



2. Triangles

DRAWING TOOLS

3. Adhesive Tape

HB for thick line

2H for thin line

4. Pencils

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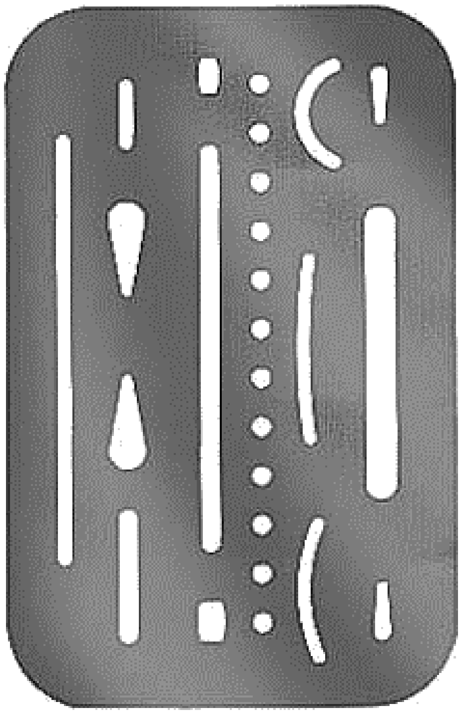
5. Sandpaper

6. Compass

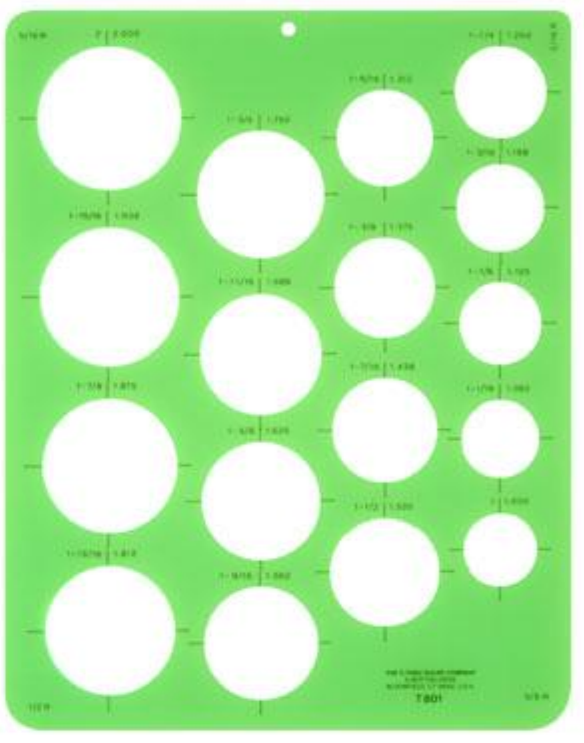
DRAWING TOOLS

7. Pencil Eraser



8. Erasing Shield

DRAWING TOOLS

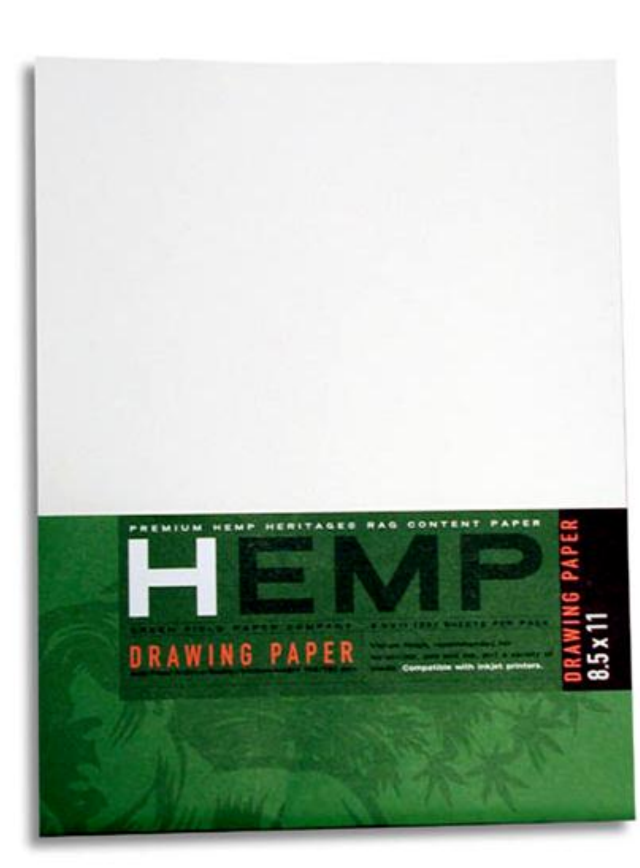


9. Circle Template

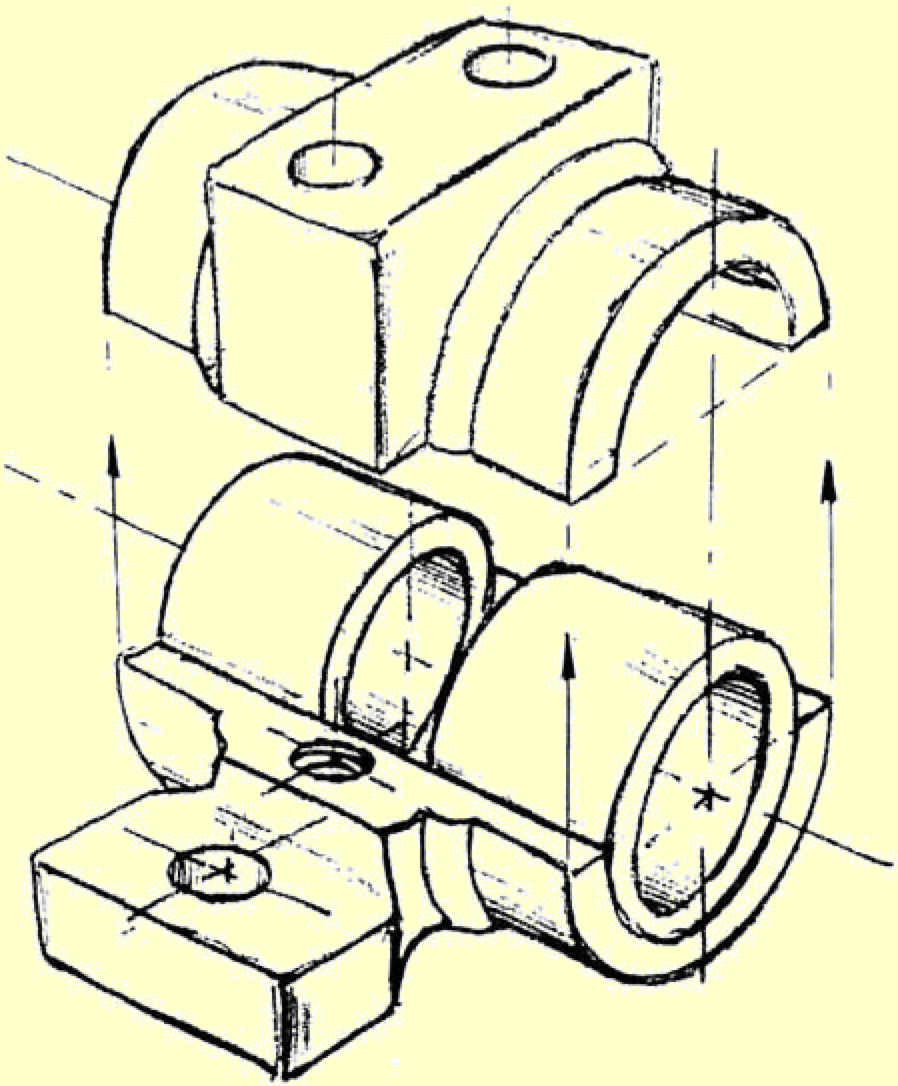
10. Tissue paper

DRAWING TOOLS

11. Sharpener



12. Clean paper



Freehand Sketching

Straight Line

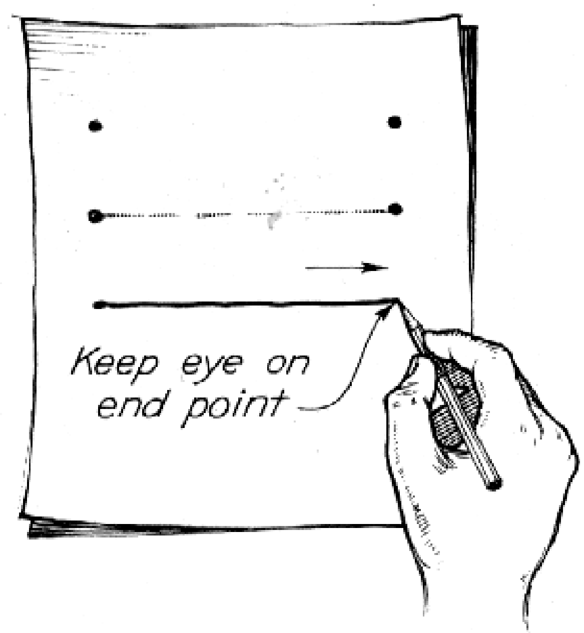
1. Hold the pencil naturally.

2. Spot the beginning and end points.

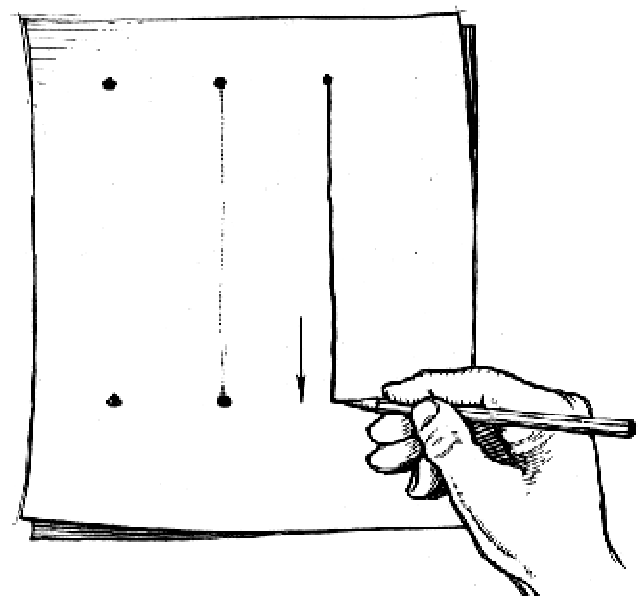
3. Swing the pencil back and forth between the points, barely touching the paper until the direction is clearly established.

4. Draw the line firmly with a free and easy wrist-and-arm motion

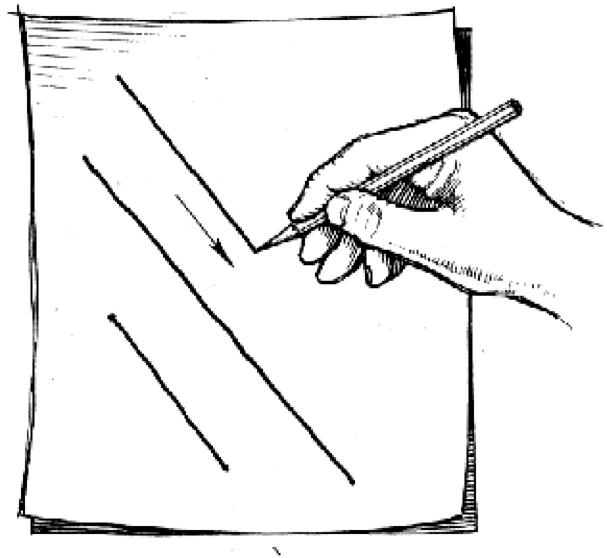
Horizontal line

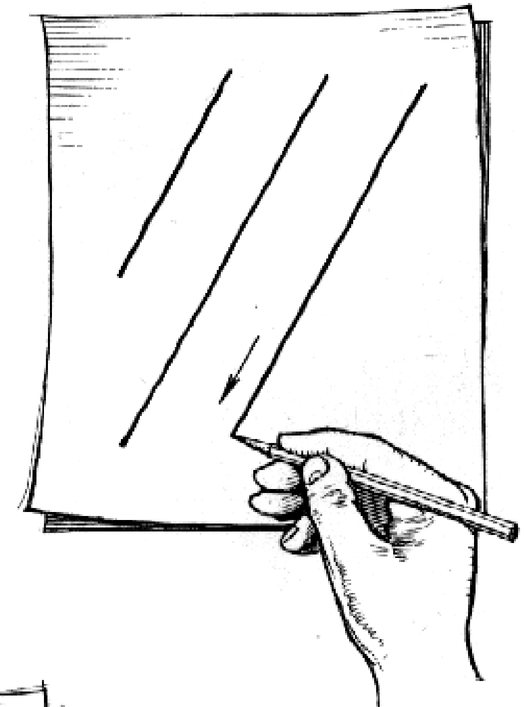


Vertical line

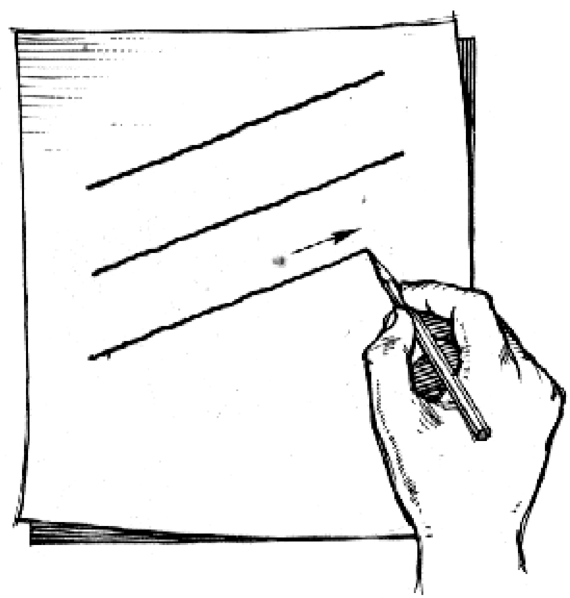


Nearly vertical inclined line





Nearly horizontal inclined line



Small Circle

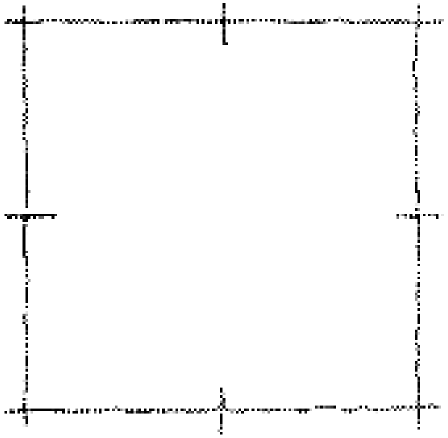
Method 1 : Starting with a square

1. Lightly sketching the square and marking the mid-points.

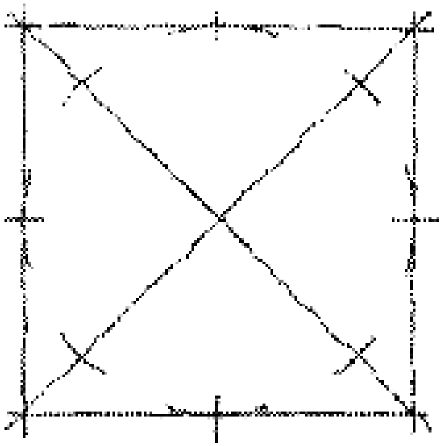
2. Draw light diagonals and mark the estimated radius.

3. Draw the circle through the eight points.

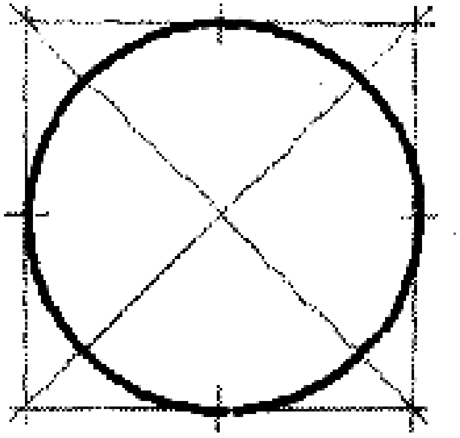
Step 1



Step 2



Step 3

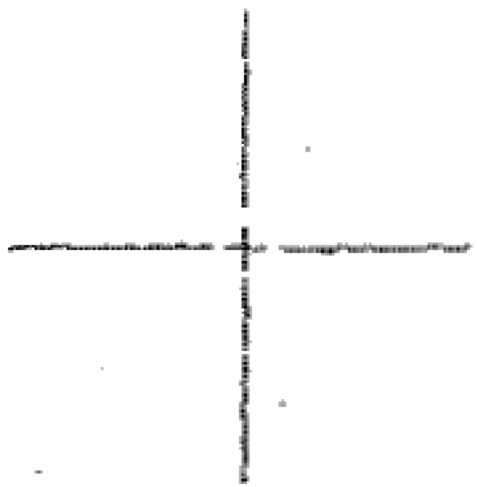


Small Circle

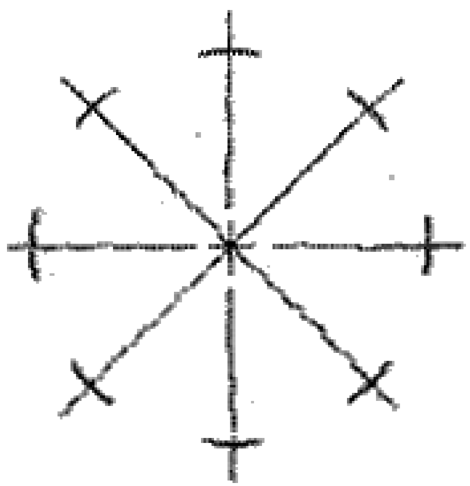
Method 2 : Starting with center line

1. Lightly draw a center line.
2. Add light radial lines and mark the estimated radius.
3. Sketch the full circle.

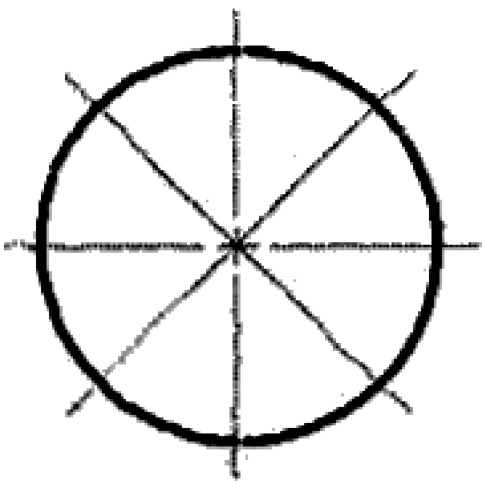
Step 1



Step 2

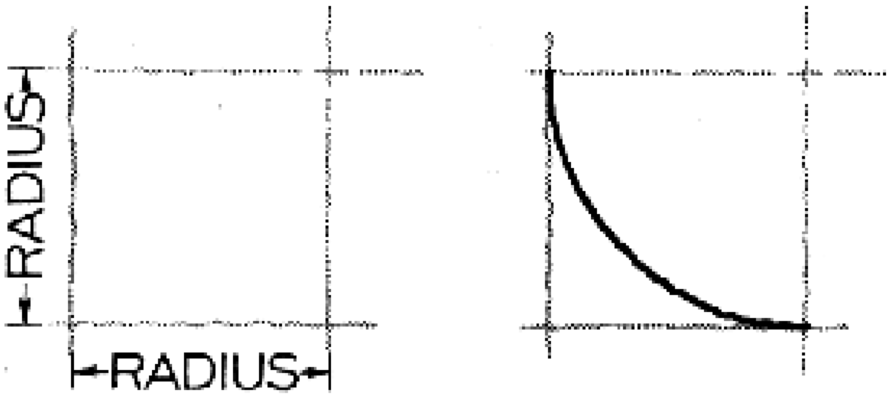


Step 3

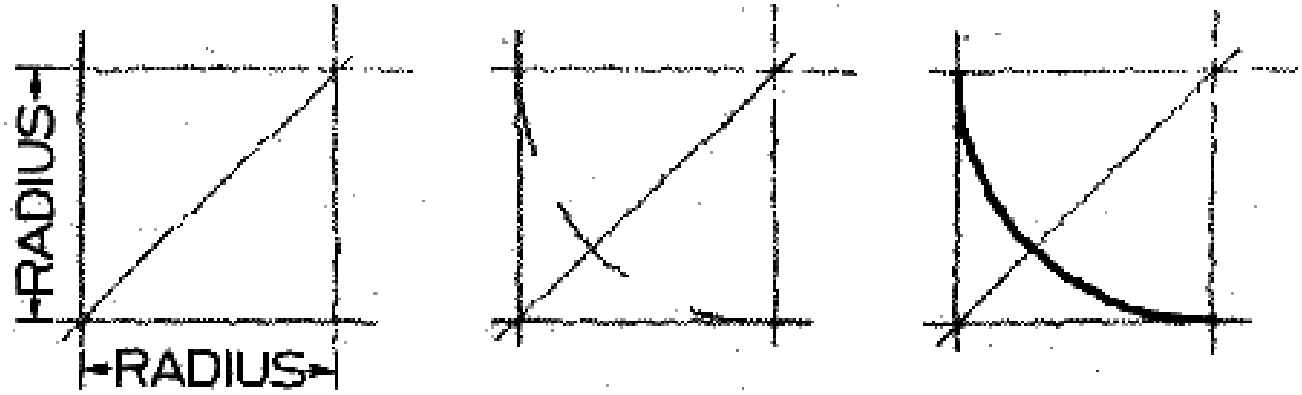


Arc

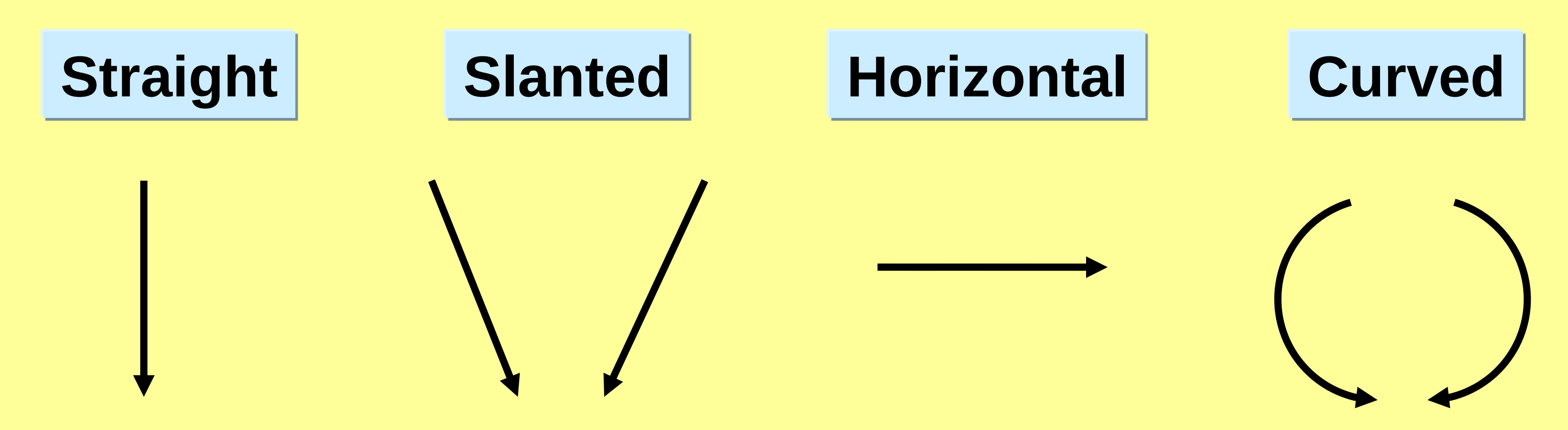
Method 1 : Starting with a square



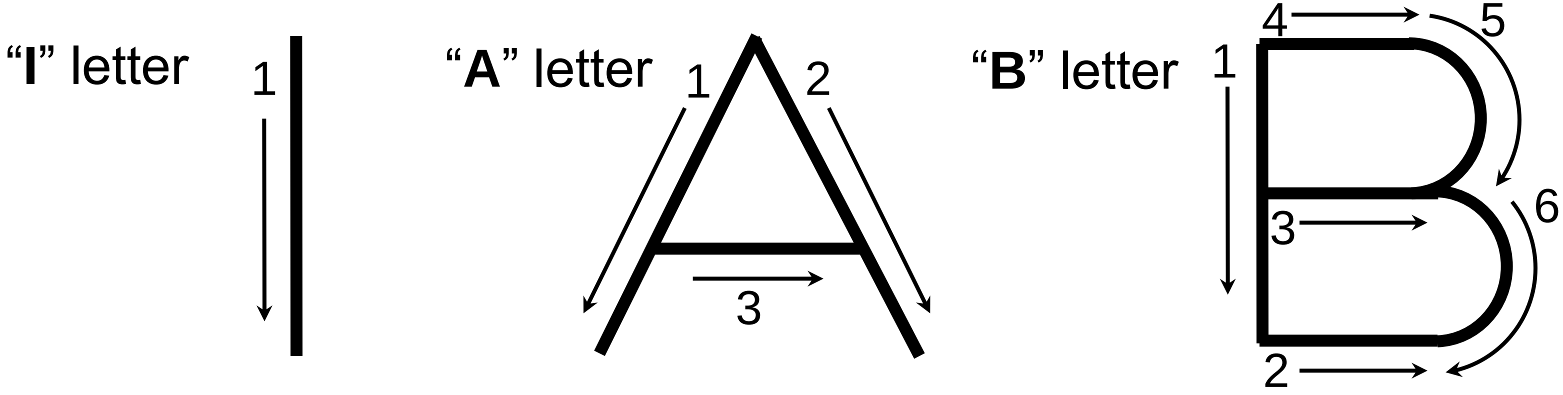
Method 2 : Starting with a center line



Basic Strokes



Examples : *Application of basic stroke*

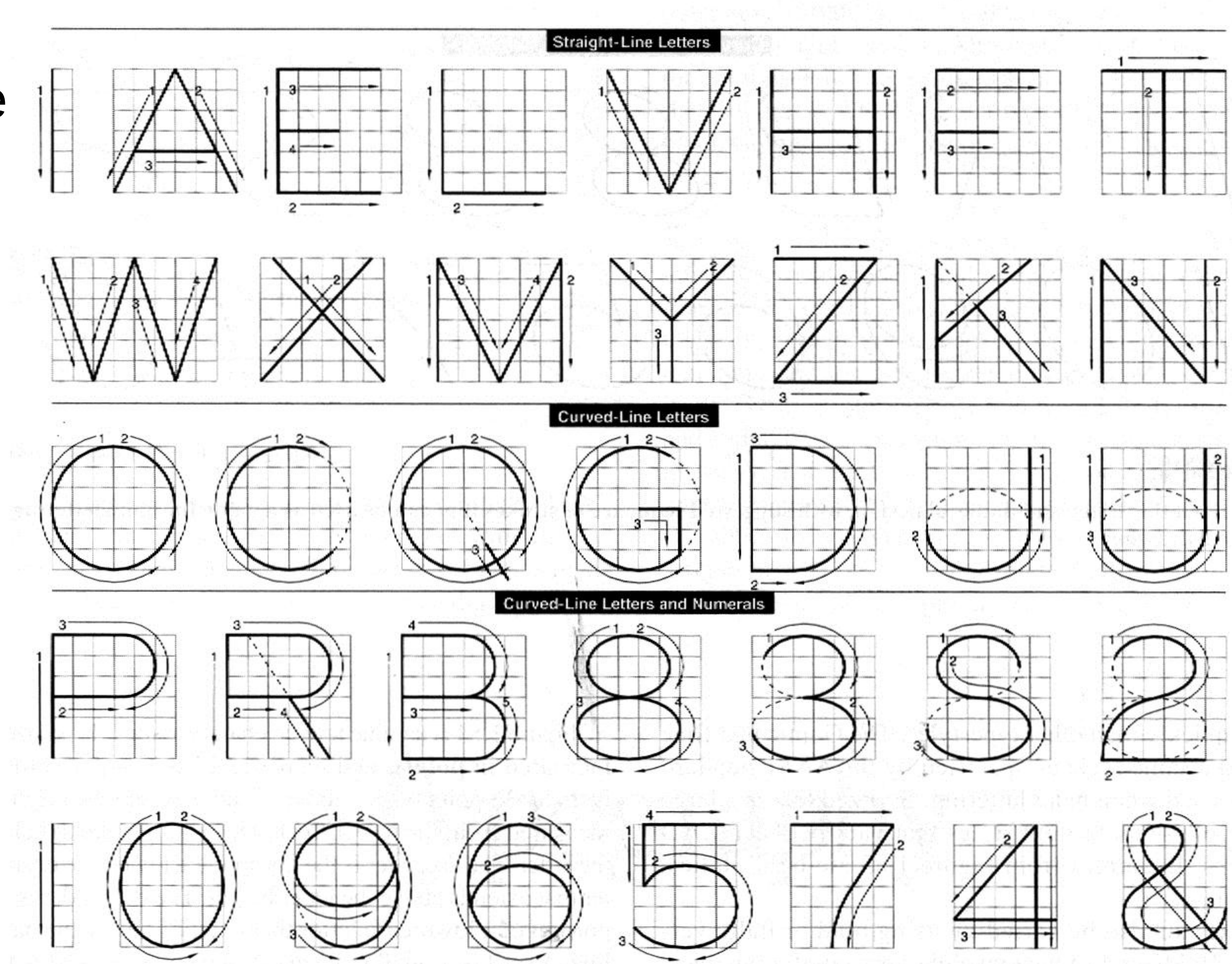


Upper-case letters & Numerals

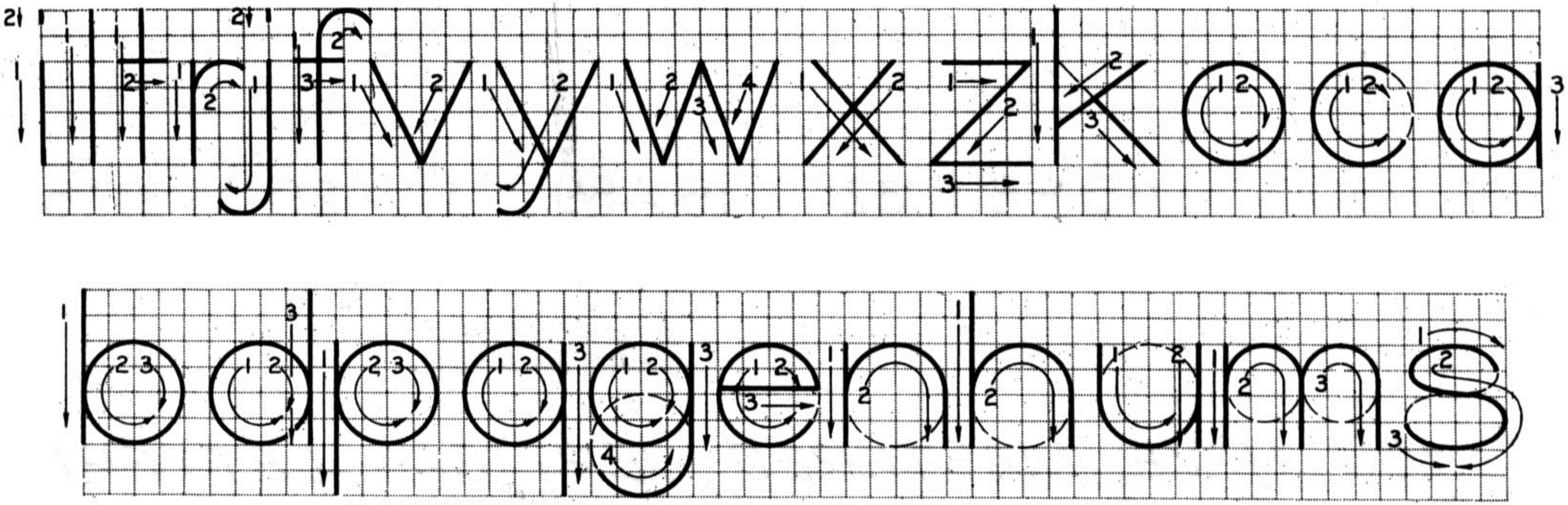
Straight line letters

Curved line letters

Curved line letters & Numerals



Lower-case letters



- The text’ s body height is about 2/3 the height of a capital letter.

Example : *Good and Poor Lettering*

GOOD

Not uniform in style.

Not uniform in height.

Not uniformly vertical or inclined.

Not uniform in thickness of stroke.

Area between letters not uniform.

Area between words not uniform.

Sentence Composition

- Leave the space between words equal to the space requires for writing a letter “O”.

Example

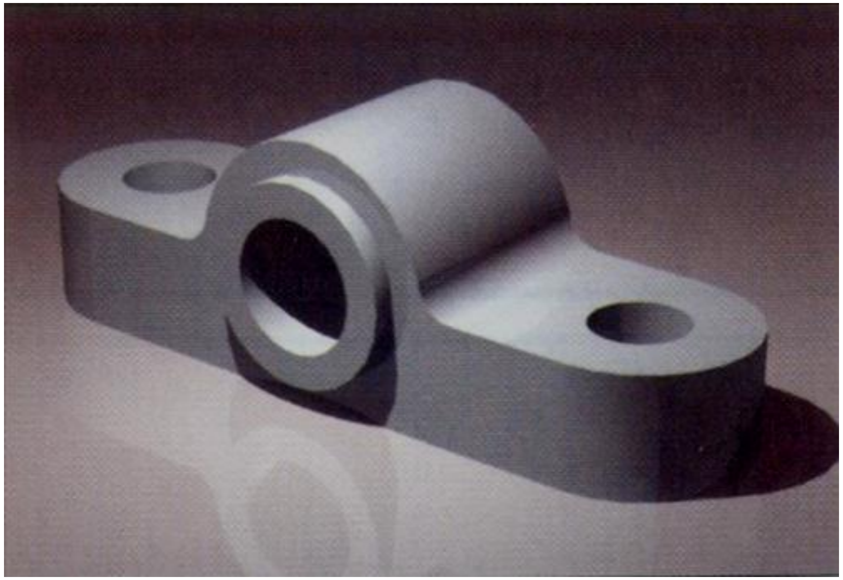
ALLODIMENSIONSOAREOIN
MILLIMETERSOUNLESS
OTHERWISEOSPECIFIED.

GRAPHICS LANGUAGE

Effectiveness of Graphics Language

1. Try to write a description of this object.

2. Test your written description by having someone attempt to make a sketch from your description.



You can easily understand that …

The word languages are inadequate for describing the ***size***, ***shape*** and ***features*** completely as well as concisely.

Composition of Graphic Language

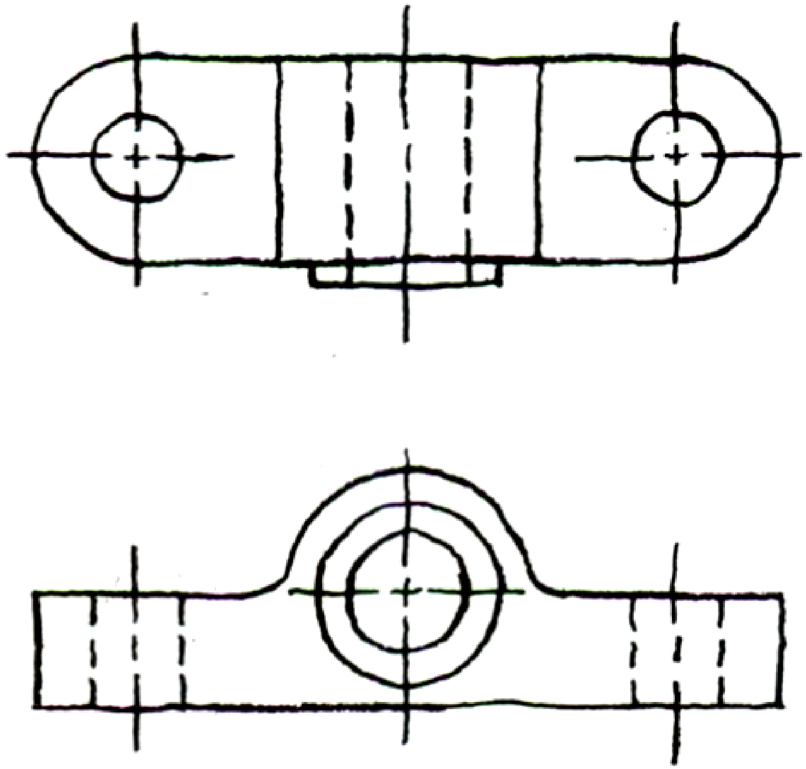
Graphic language use ***lines*** to represent the ***surfaces***, ***edges*** and ***contours*** of objects.

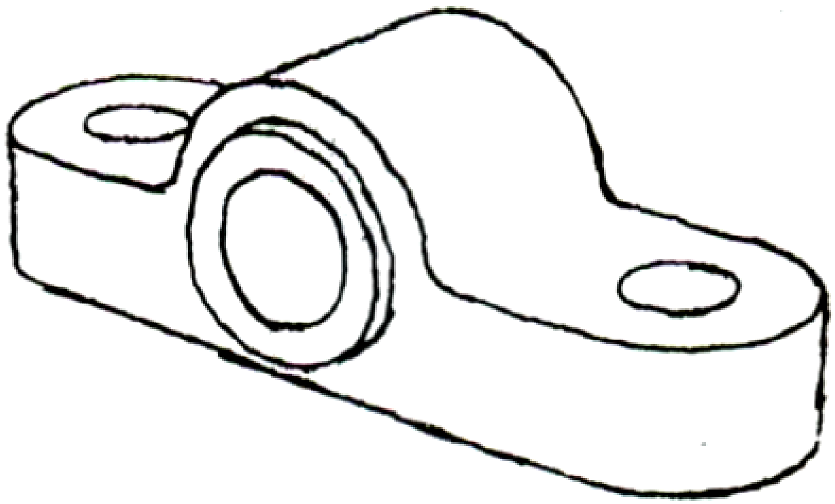
- The language is known as "***drawing***" or "***drafting***" .
- A drawing can be done using ***freehand***, ***instruments*** or ***computer*** methods.

Freehand drawing

The lines are sketched without using instruments other than pencils and erasers.

Example

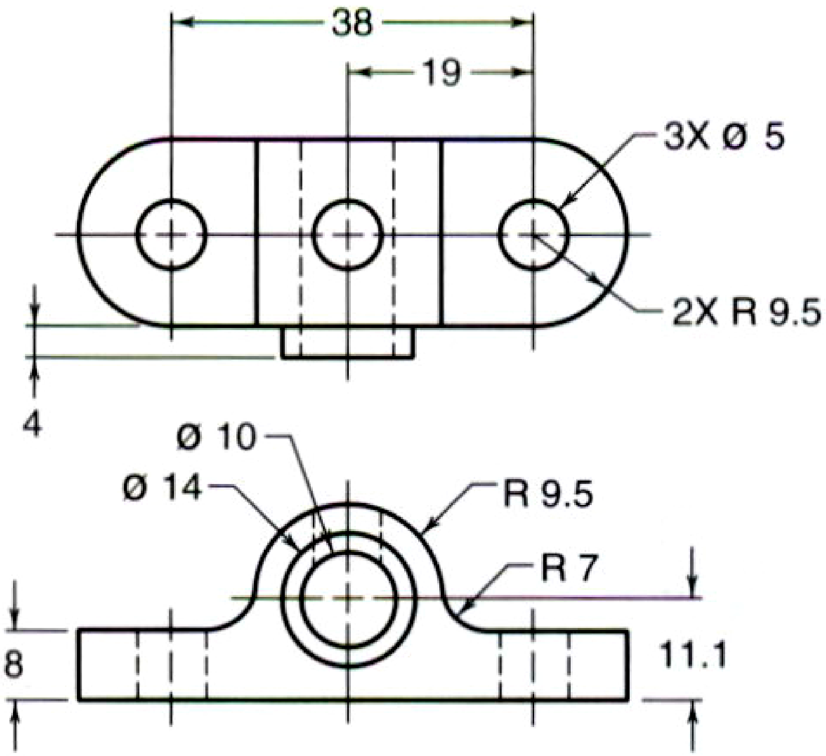




Instrument drawing

Instruments are used to draw straight lines, circles, and curves concisely and accurately. Thus, the drawings are usually made to scale.

Example



Computer drawing

The drawings are usually made by commercial software such as AutoCAD, solid works etc.

Example

Architectural Graphics

Elements

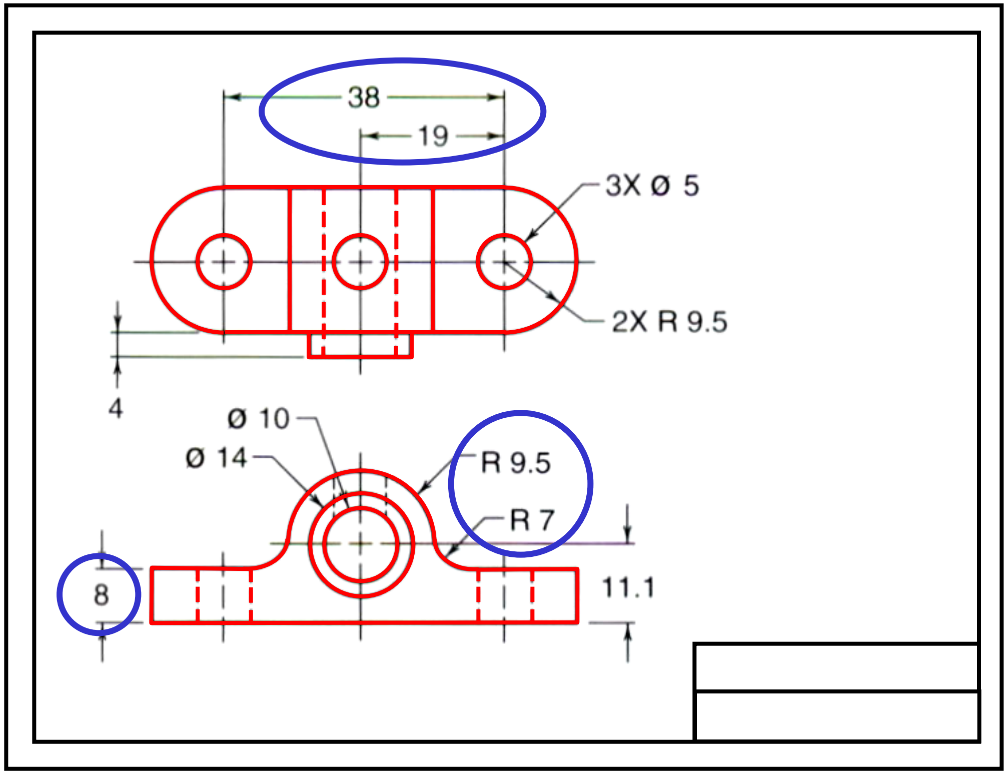
Drawing comprises of ***graphics language*** and ***word language***.

Graphics language

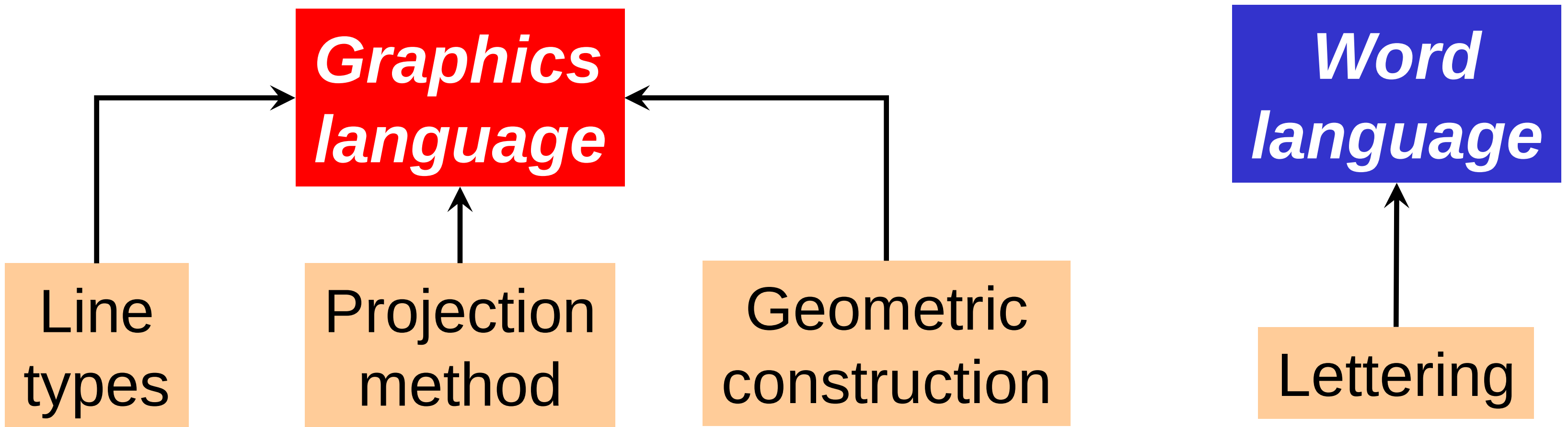
Describe a shape (mainly).

Word language

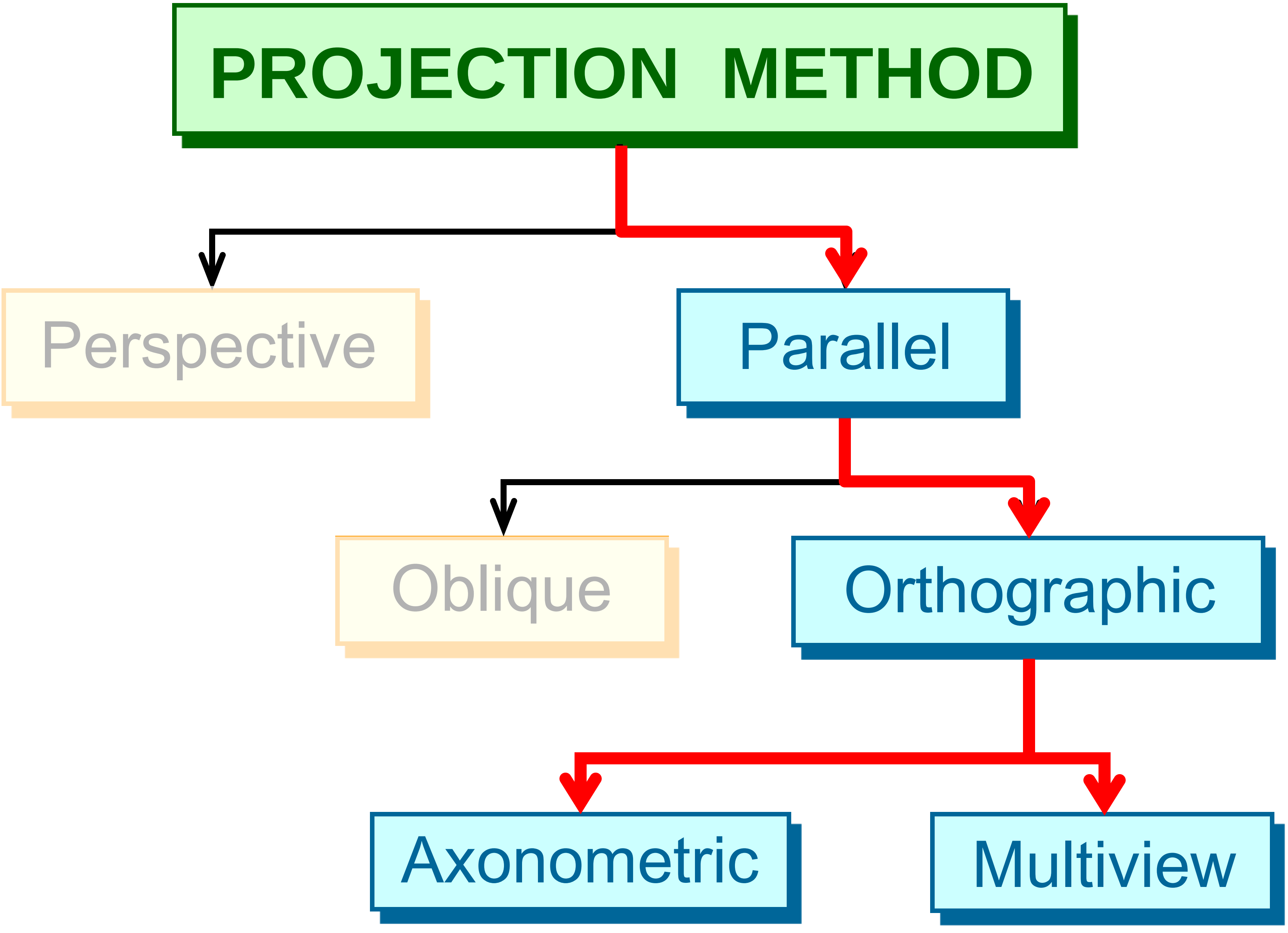
Describe size, location and specification of the object.



Basic Knowledge for Drafting



PROJECTION METHOD


PROJECTION METHOD
Perspective
Parallel
Oblique
Orthographic
Axonometric
Multiview

PROJECTION THEORY

- The projection theory is used to graphically represent 3-D objects on 2-D media (paper, computer screen).

- The projection theory is based on two variables:

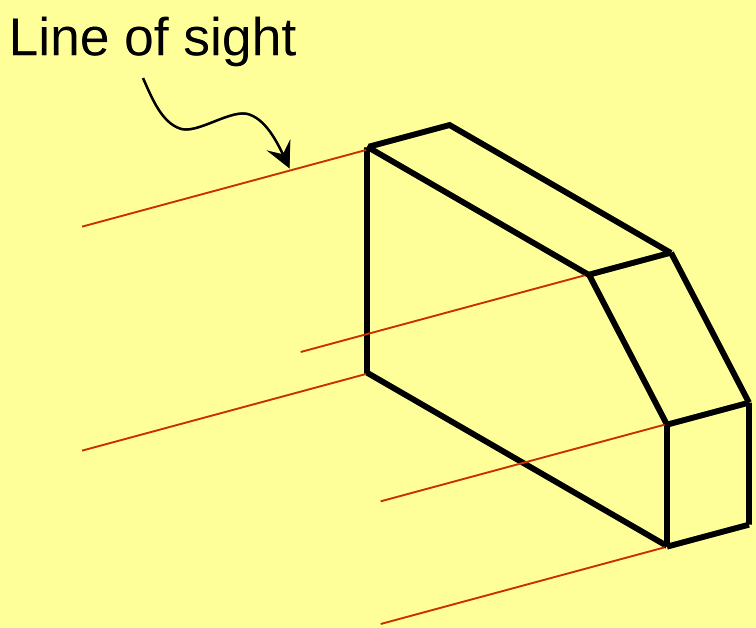
  1) Line of sight

  2) Plane of projection (image plane or picture plane)

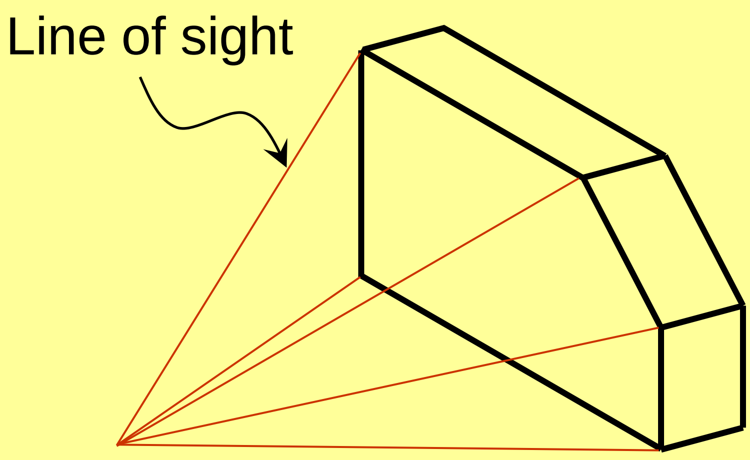
Line of sight is an imaginary ray of light between an observer's eye and an object.

- There are 2 types of LOS : parallel and converge

Parallel projection



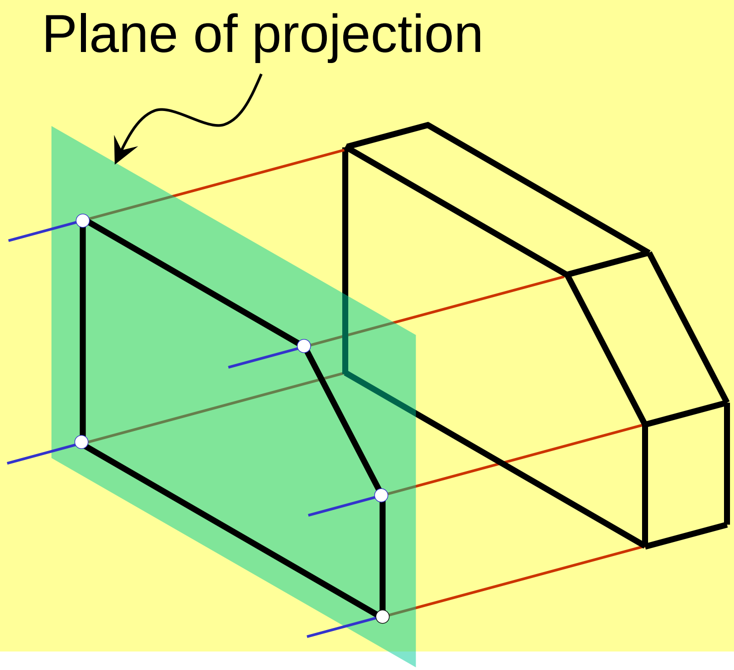
Perspective projection



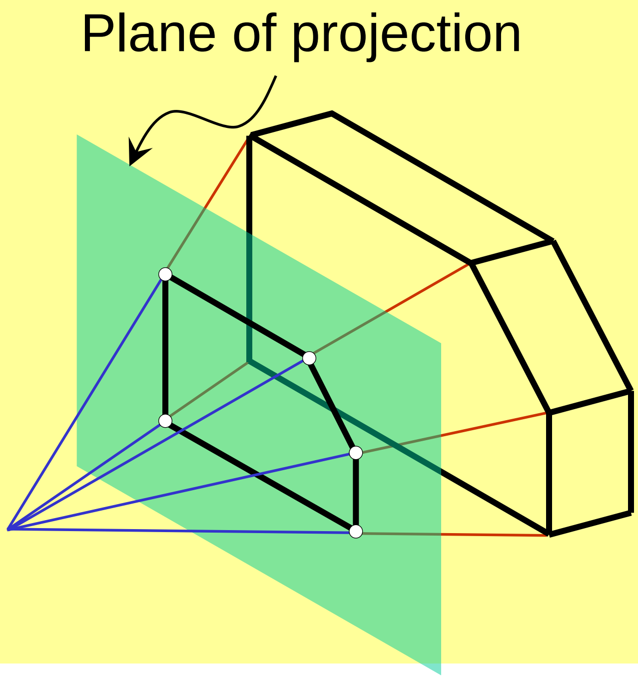
Plane of projection is an imaginary flat plane which the image is created.

- The image is produced by connecting the points where the LOS pierce the projection plane.

Parallel projection



Perspective projection



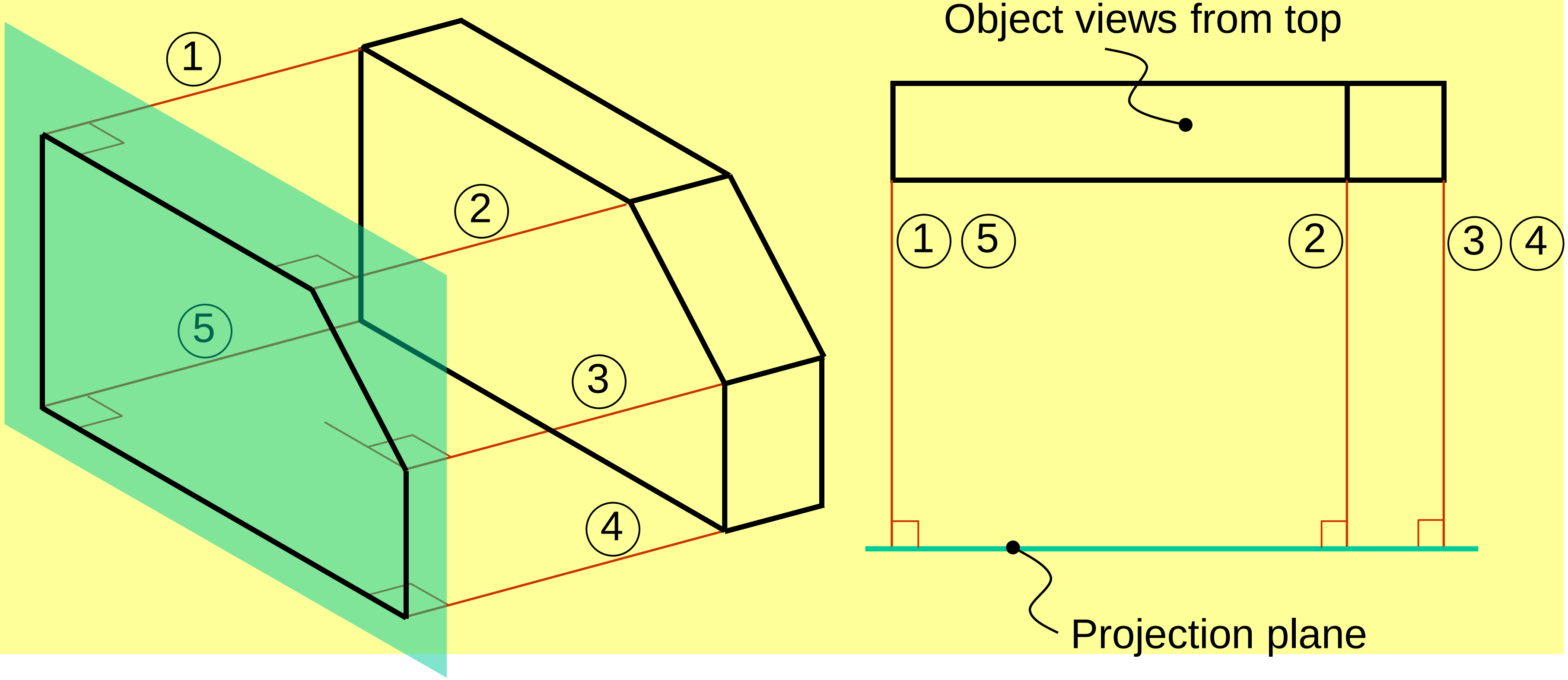
Disadvantage of Perspective Projection

1) It is difficult to create.
2) It does not reveal exact shape and size.

Orthographic Projection

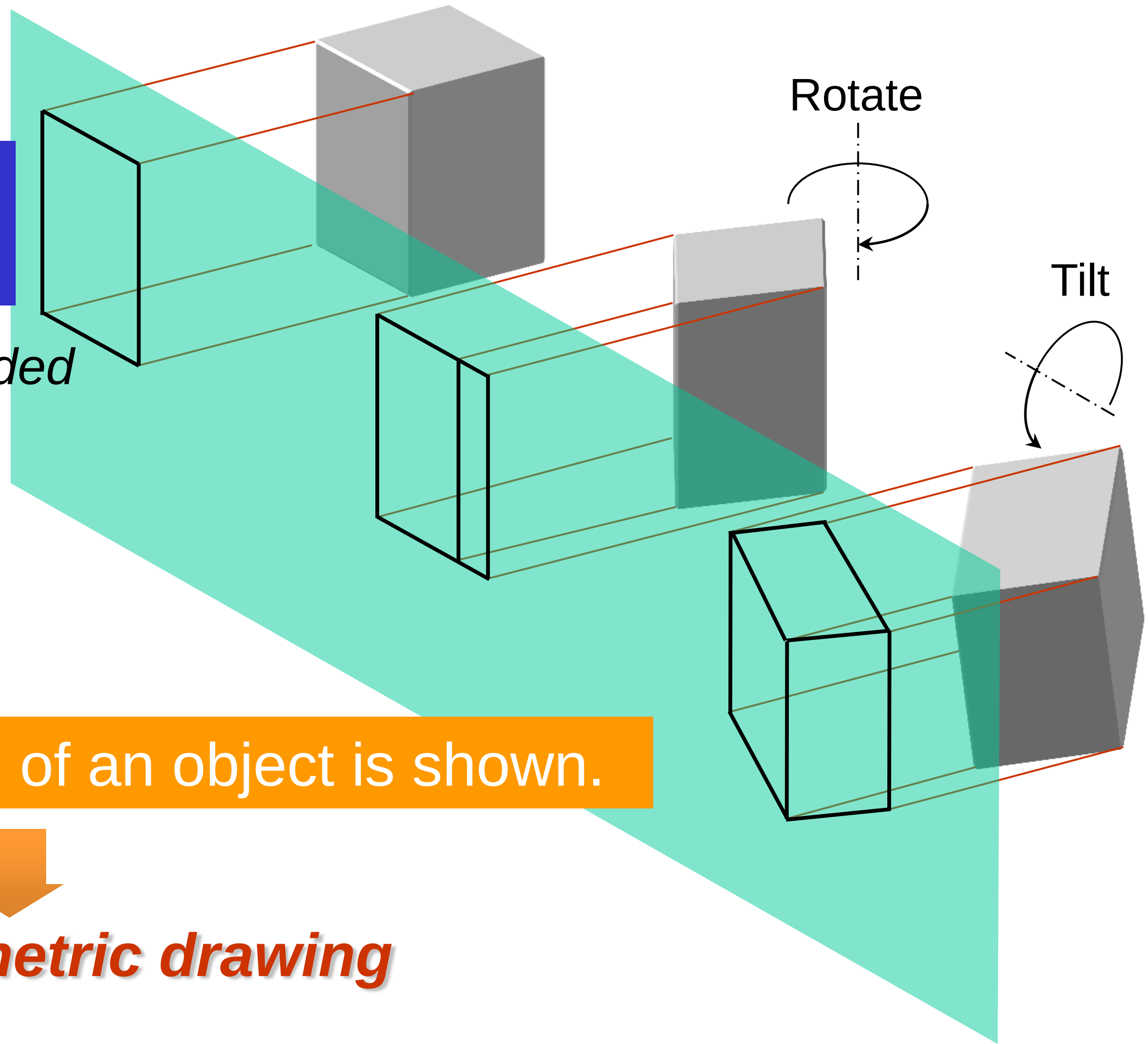
MEANING

Orthographic projection is a parallel projection technique in which the parallel lines of sight are ***perpendicular*** to the projection plane



ORTHOGRAPHIC VIEW

Orthographic view depends on relative position of the object to the line of sight.



Two dimensions of an object is shown.

More than one view is needed to represent the object.

Multiview drawing

Three dimensions of an object is shown.

Axonometric drawing

ORTHOGRAPHIC VIEW

NOTES

- Orthographic projection technique can produce either

  1. ***Multiview drawing***
     that each view show an object in two dimensions.

  2. ***Axonometric drawing***
     that show all three dimensions of an object in one view.

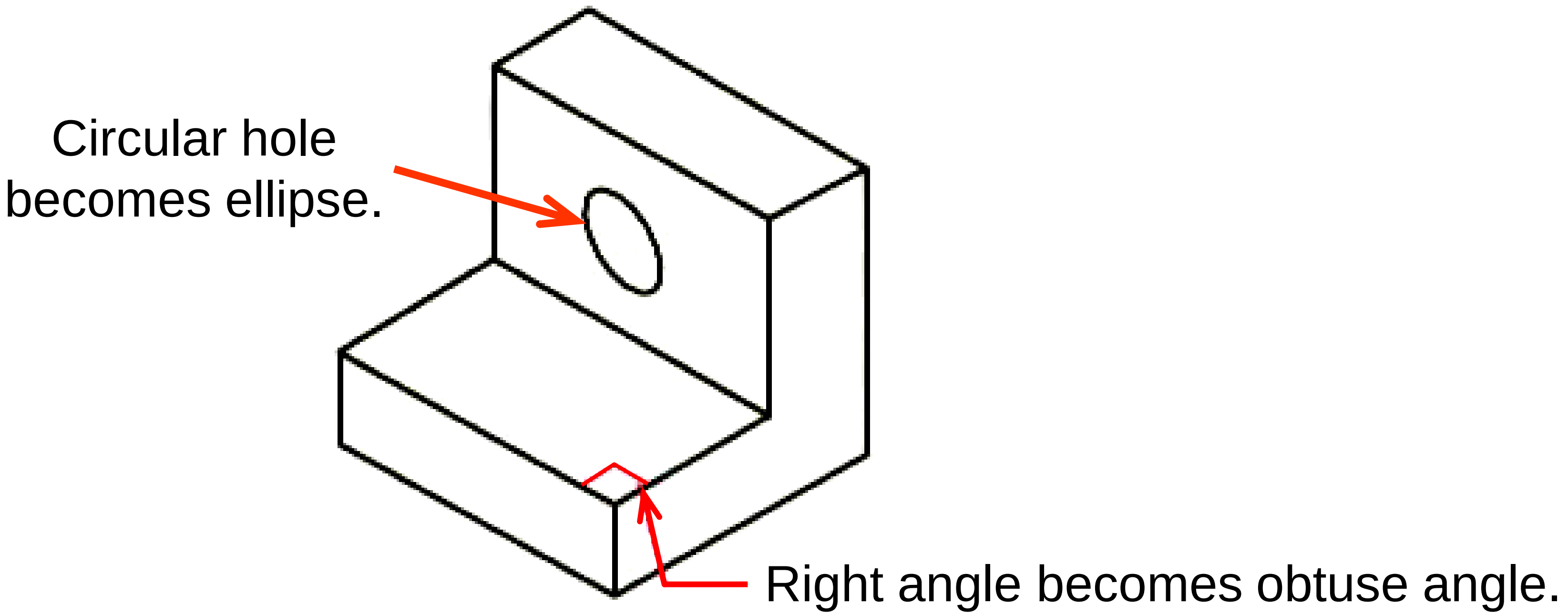
- Both drawing types are used in technical drawing for communication.

Axonometric (Isometric) Drawing

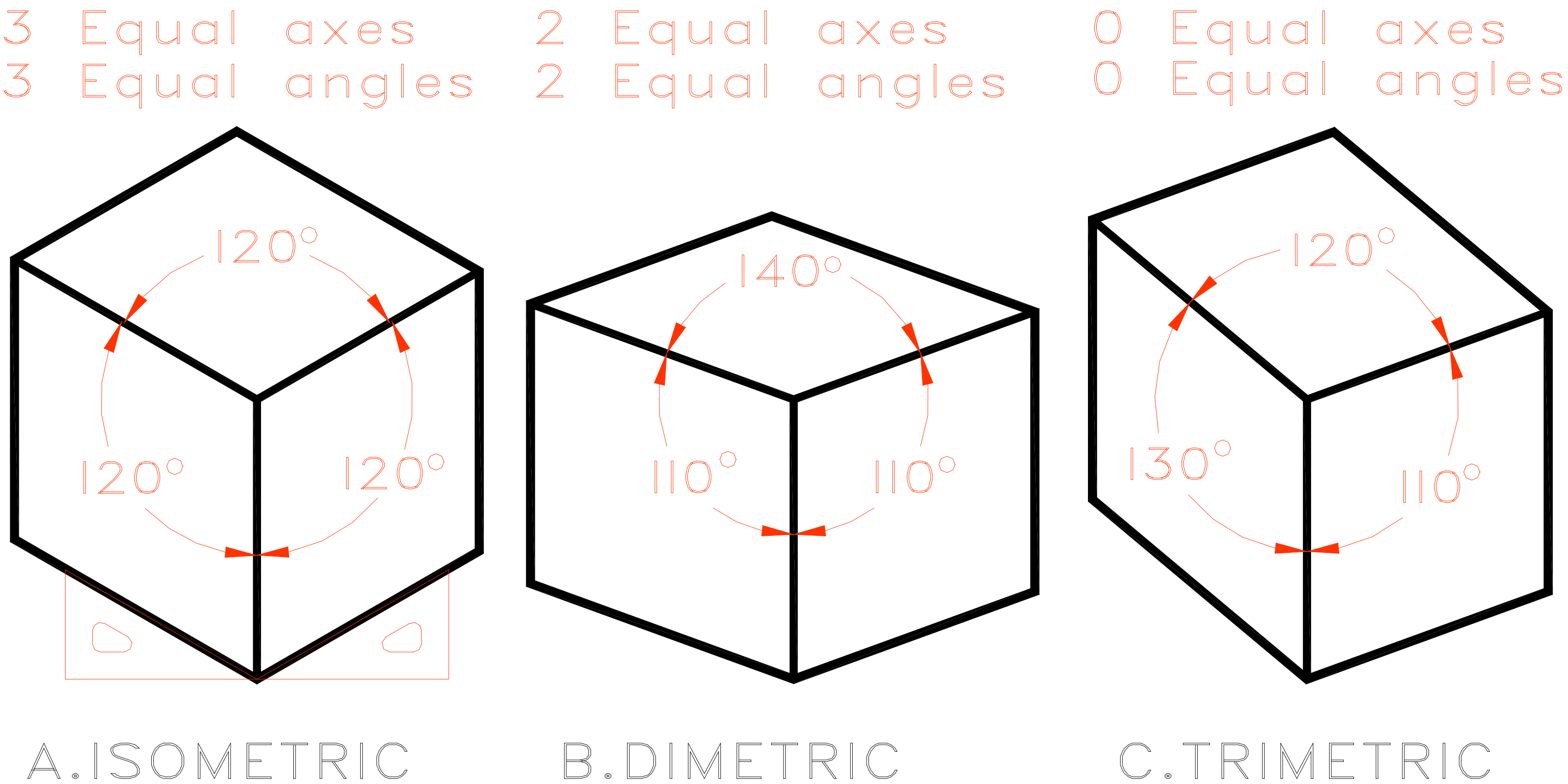
Advantage Easy to understand

Disadvantage Shape and angle distortion

Example Distortions of shape and size in isometric drawing



Types of Axonometrics

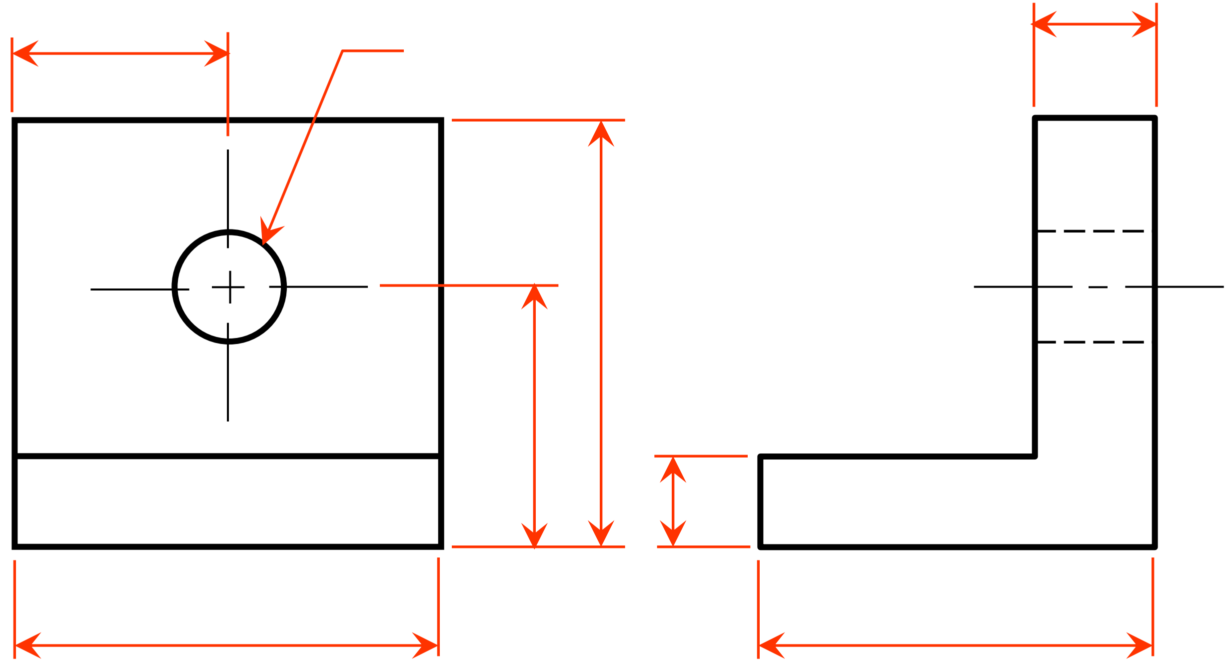


Multiview Drawing

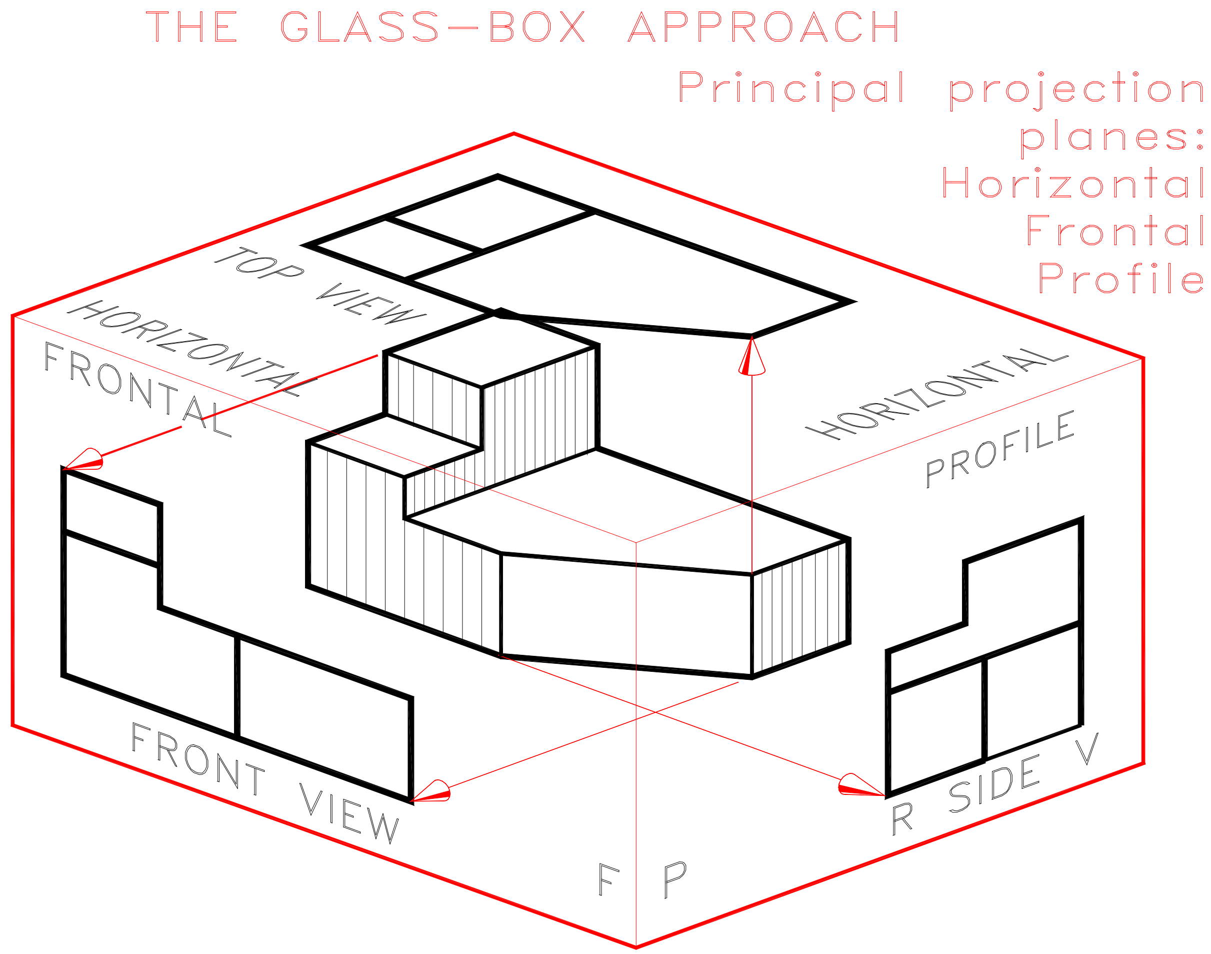
Advantage It represents accurate **shape and size.**

Disadvantage Require practice in writing and reading.

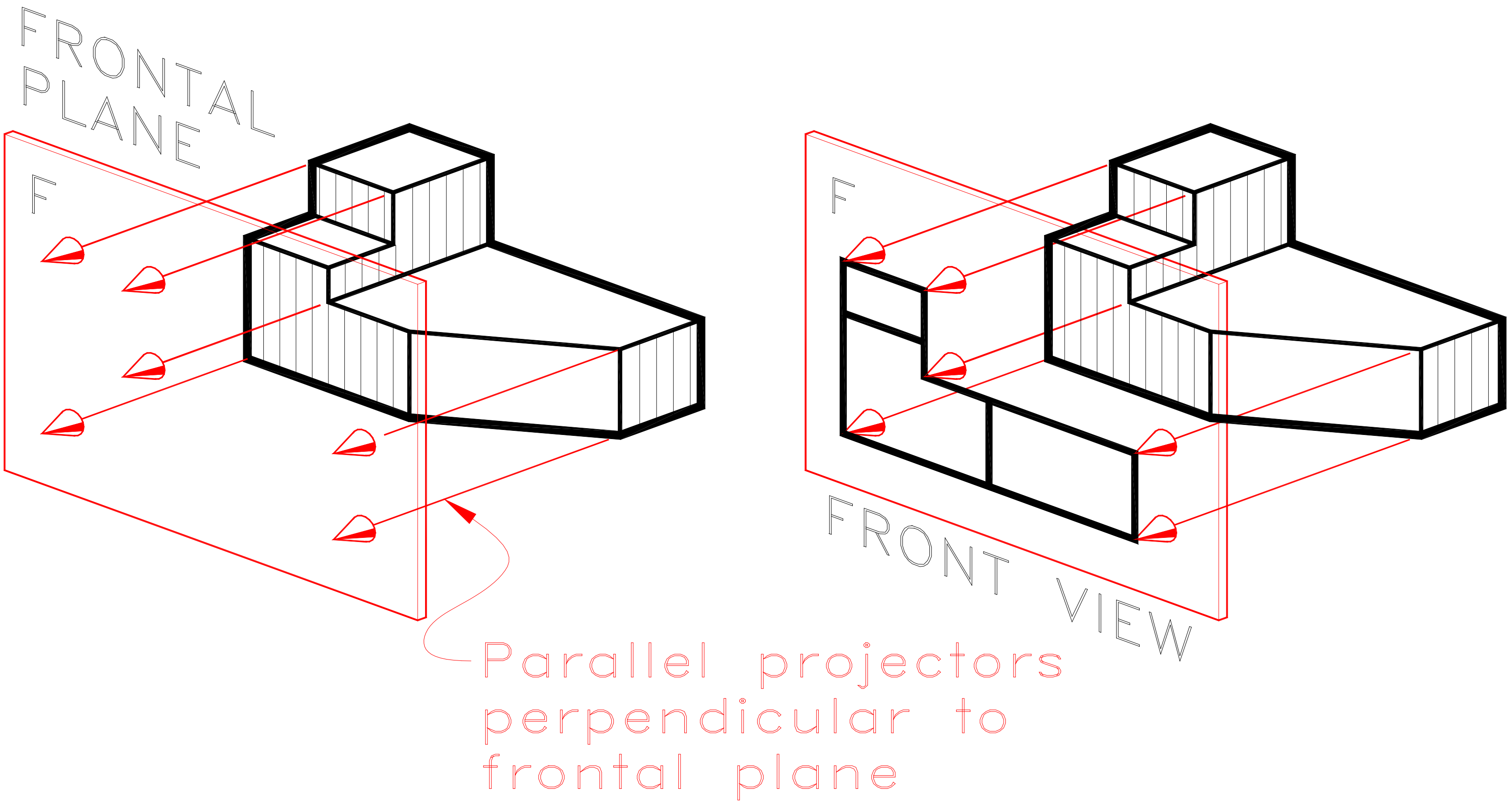
Example Multiviews drawing (2-view drawing)



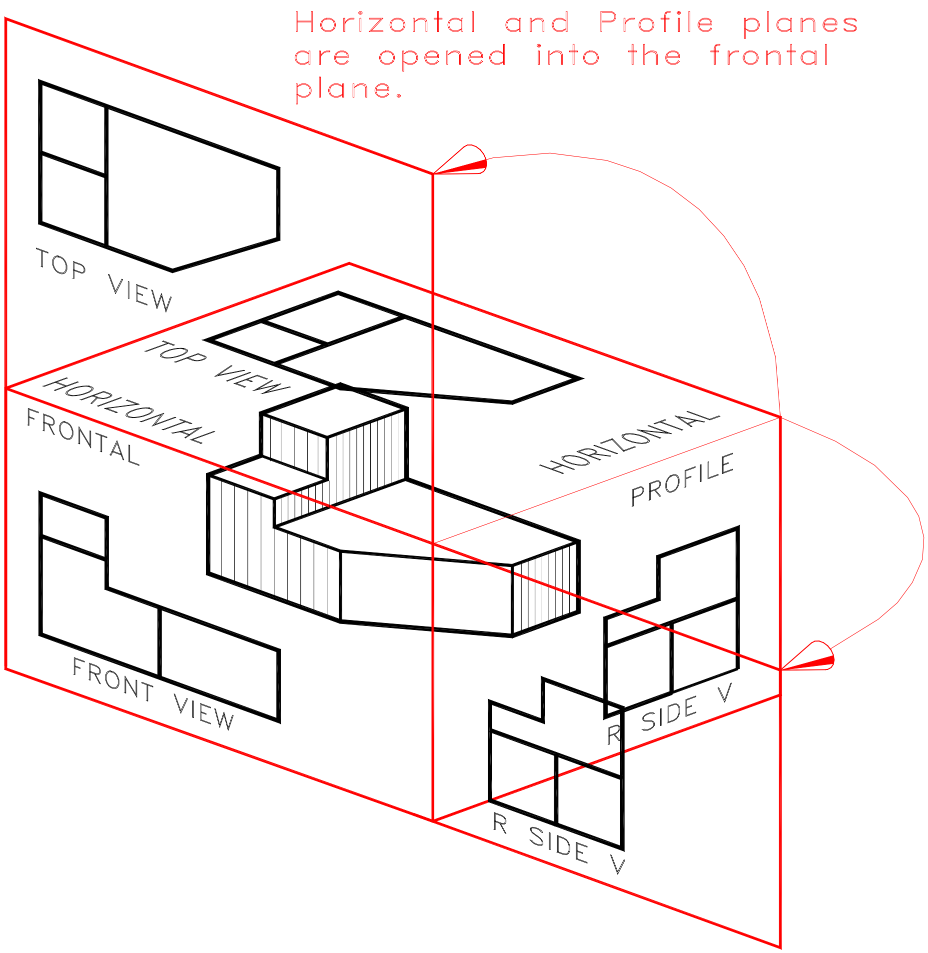
The Glass Box Approach



Orthographic Projection

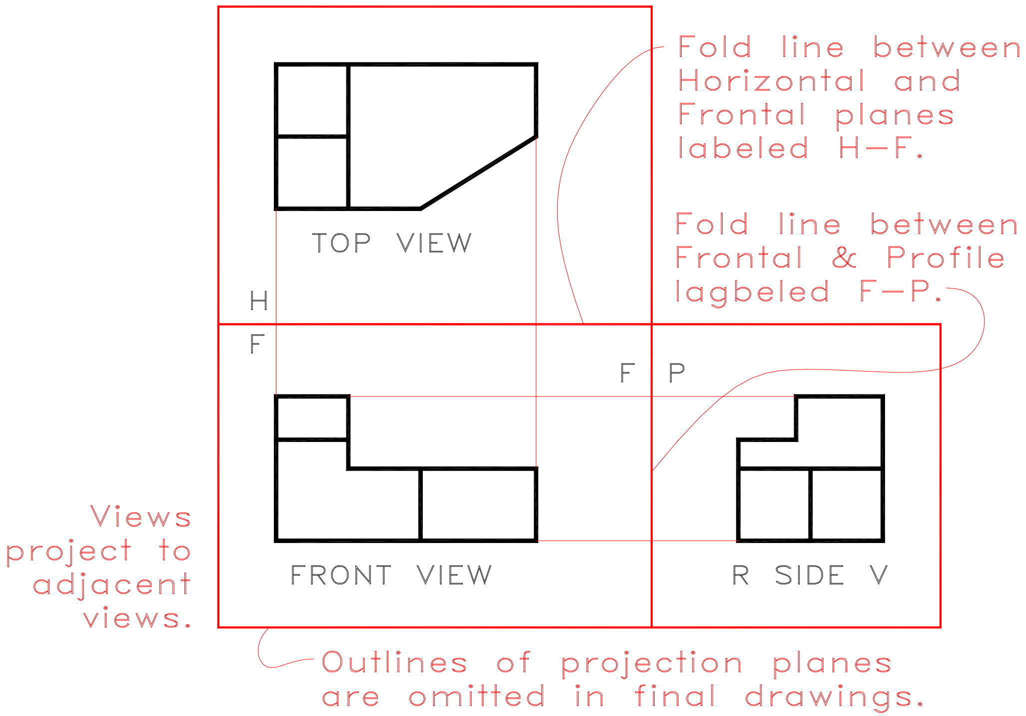


Opening the Box



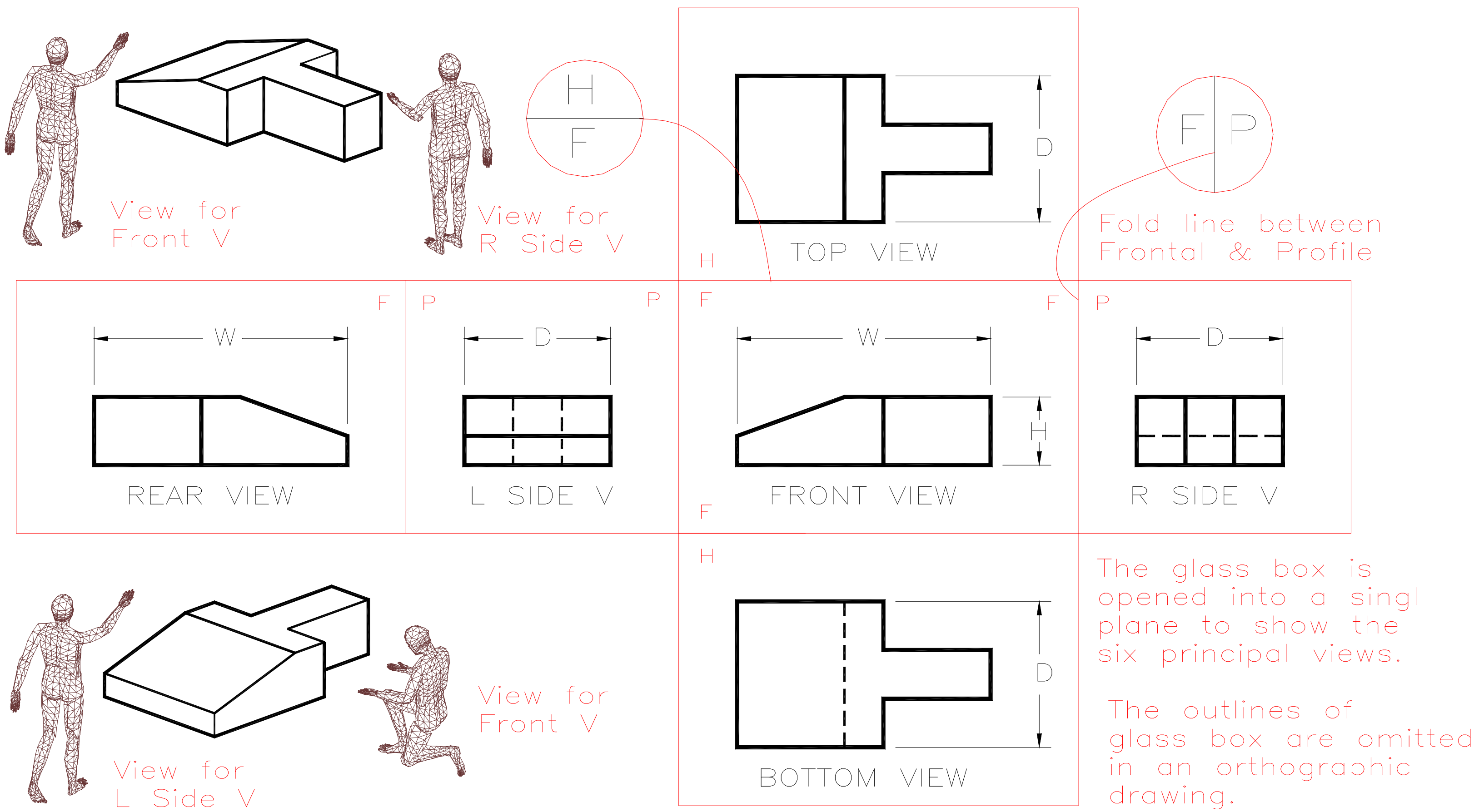
Final Views

The standard arrangement of
three orthographic views:
Top View above the Front View
R Side View right of the Front View

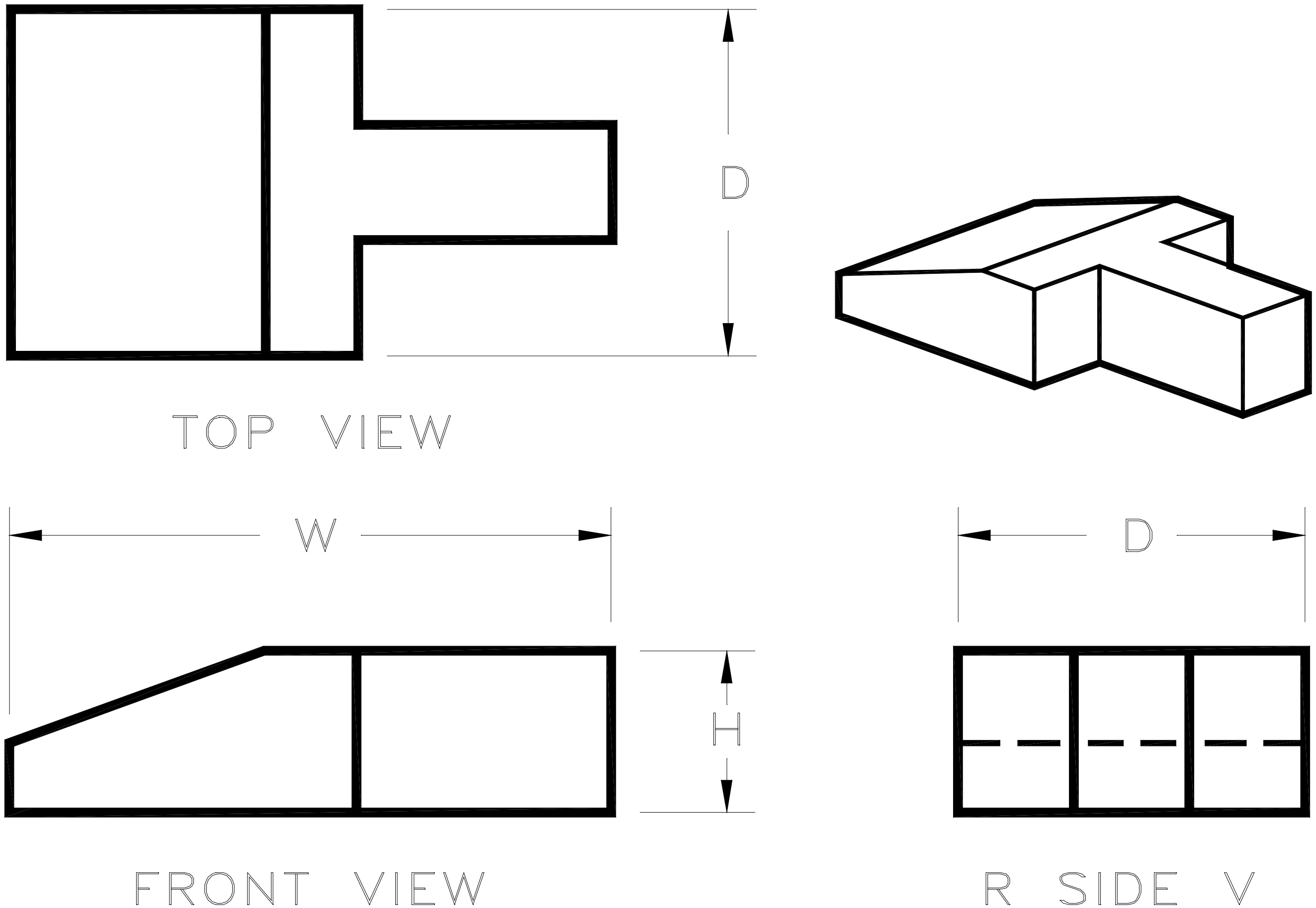


Six Orthographic Views

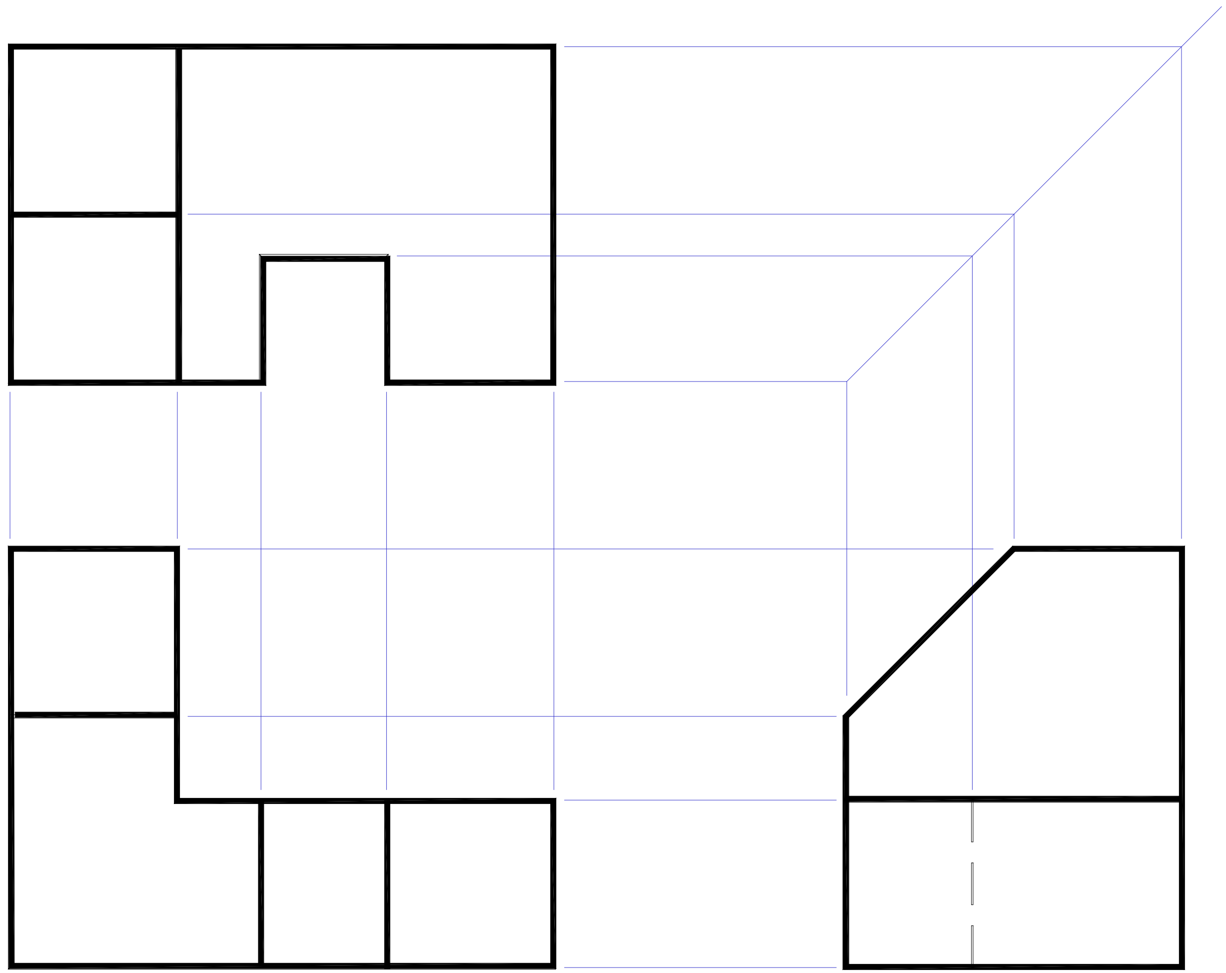
Laying Out All Six Views



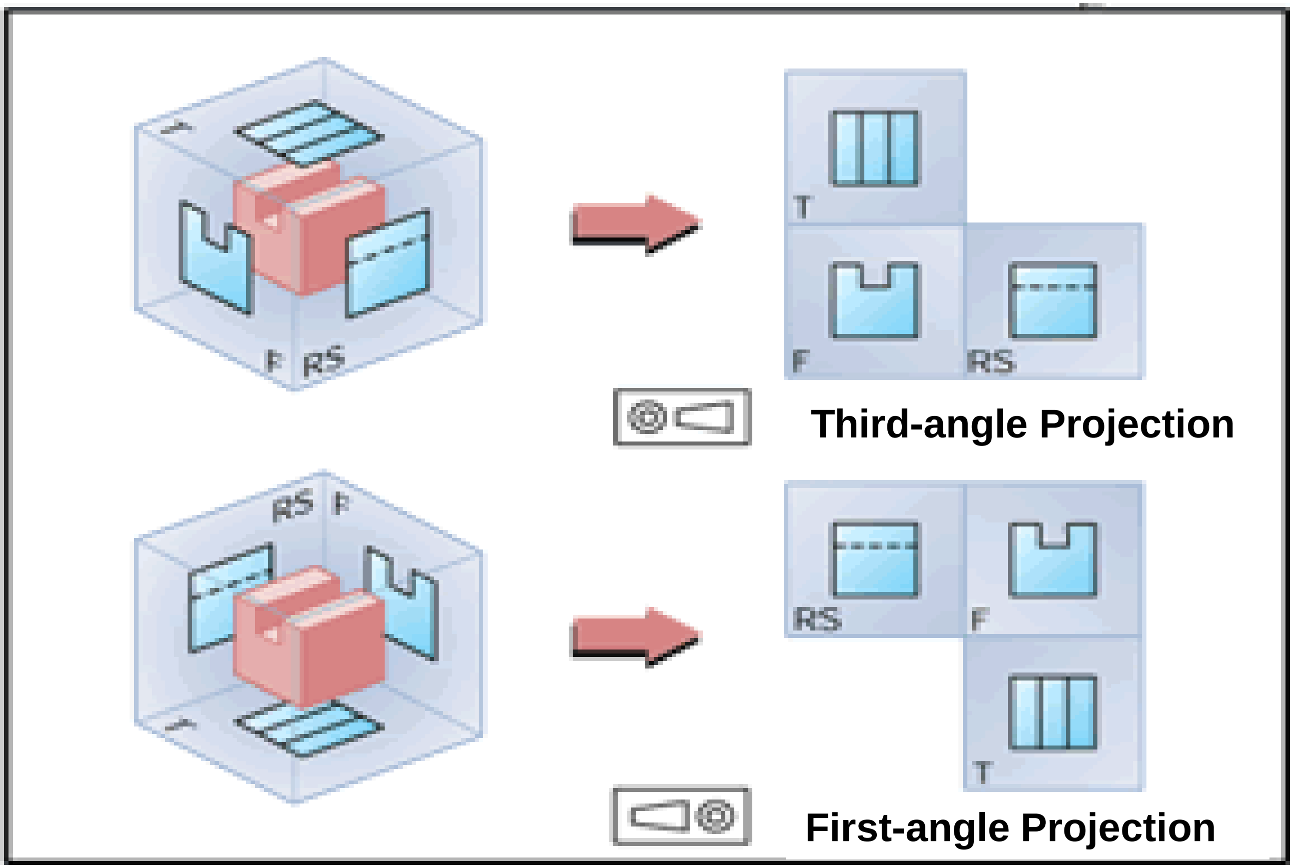
Three Primary Views



Construction of Views



First and Third Angle Projections



- First Angle – International
- Third Angle – U.S.

Basic Line Types

Types of Lines	Appearance	Name according to application
Continuous thick line		Visible line
Continuous thin line		Dimension line Extension line Leader line
Dash thick line		Hidden line
Chain thin line		Center line

Meaning of Lines

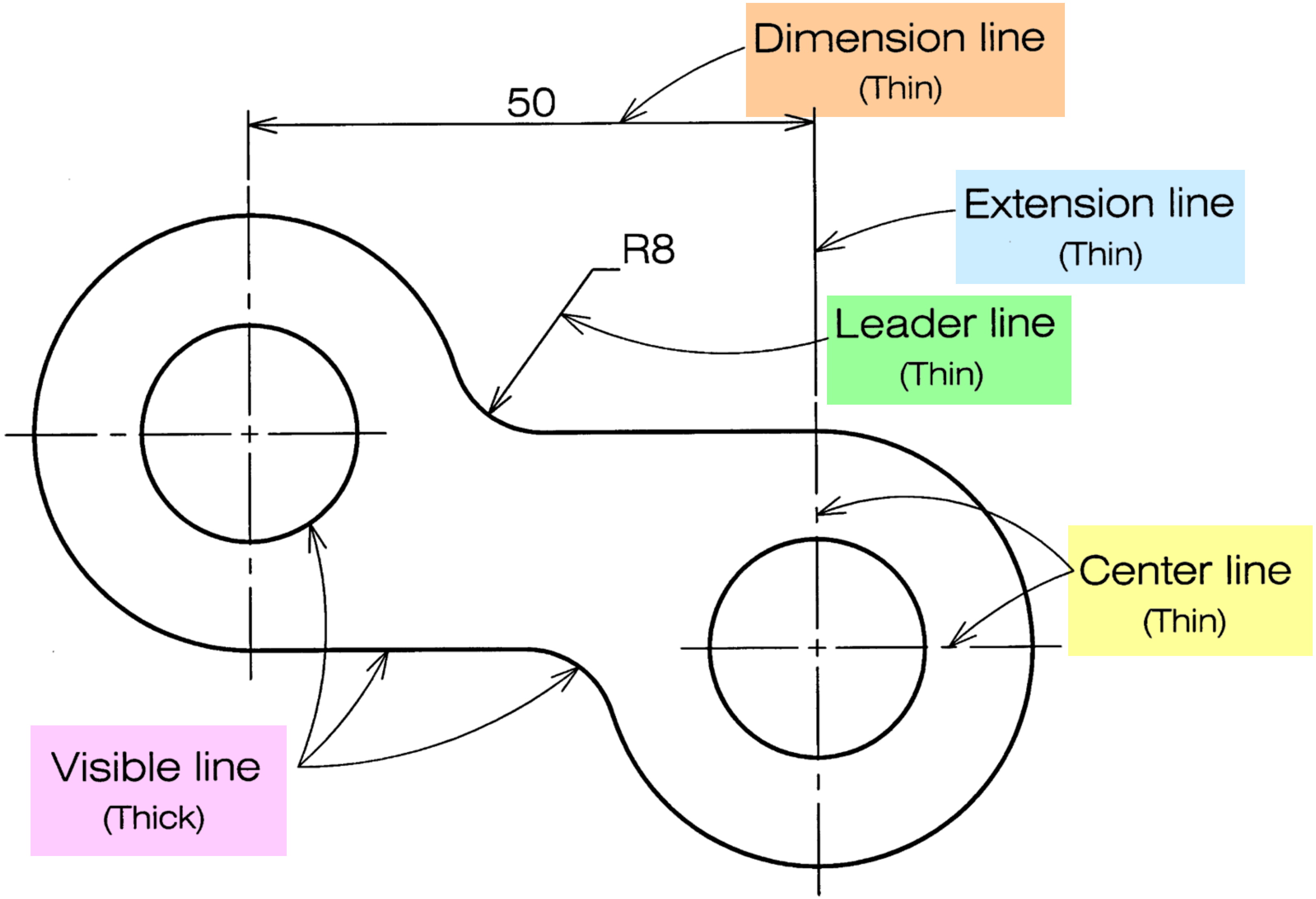
Visible lines represent features that can be seen in the current view

Hidden lines represent features that can not be seen in the current view

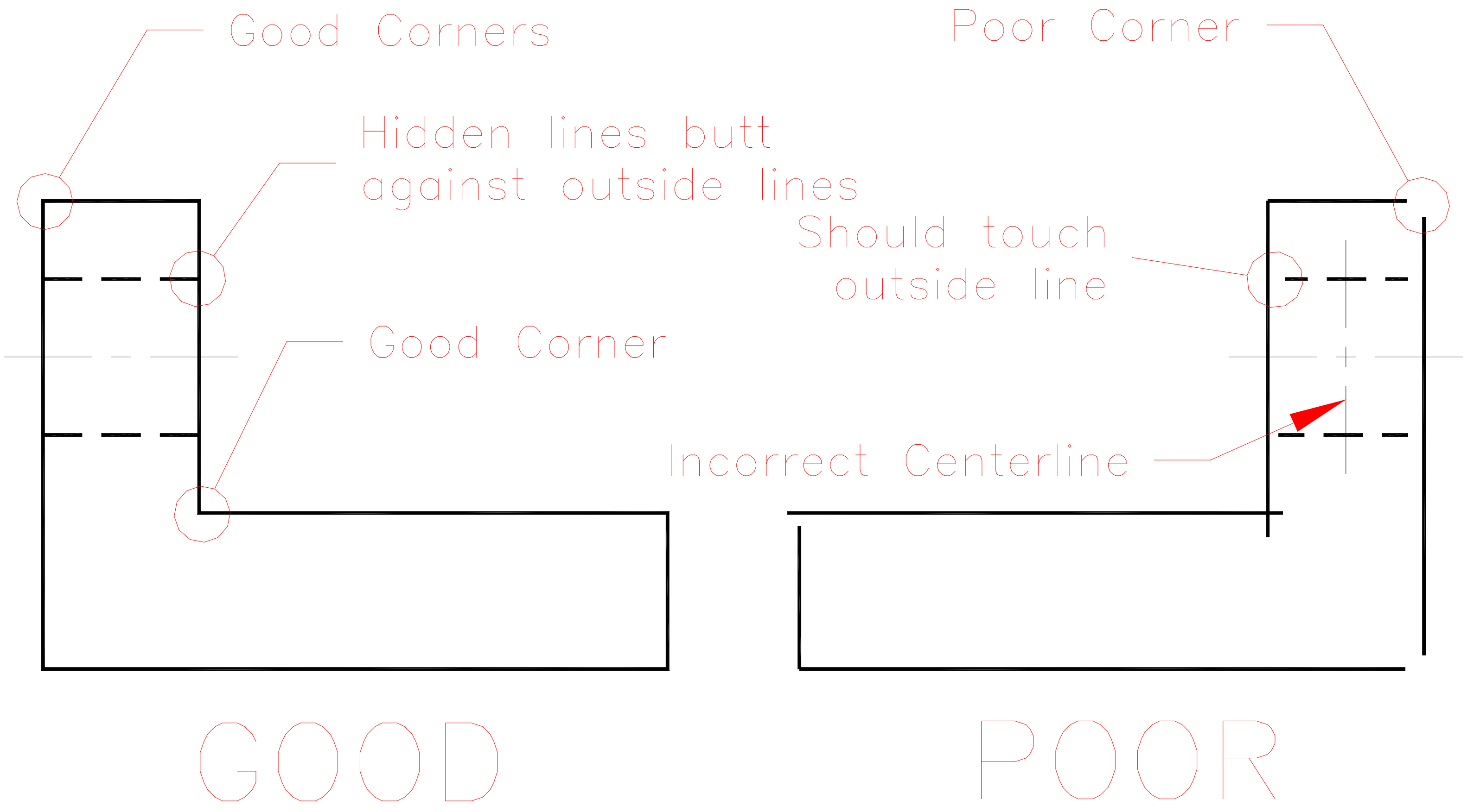
Center line represents symmetry, path of motion, centers of circles, axis of axisymmetrical parts

Dimension and Extension lines indicate the sizes and location of features on a drawing

Example : *Line conventions in engineering drawing*



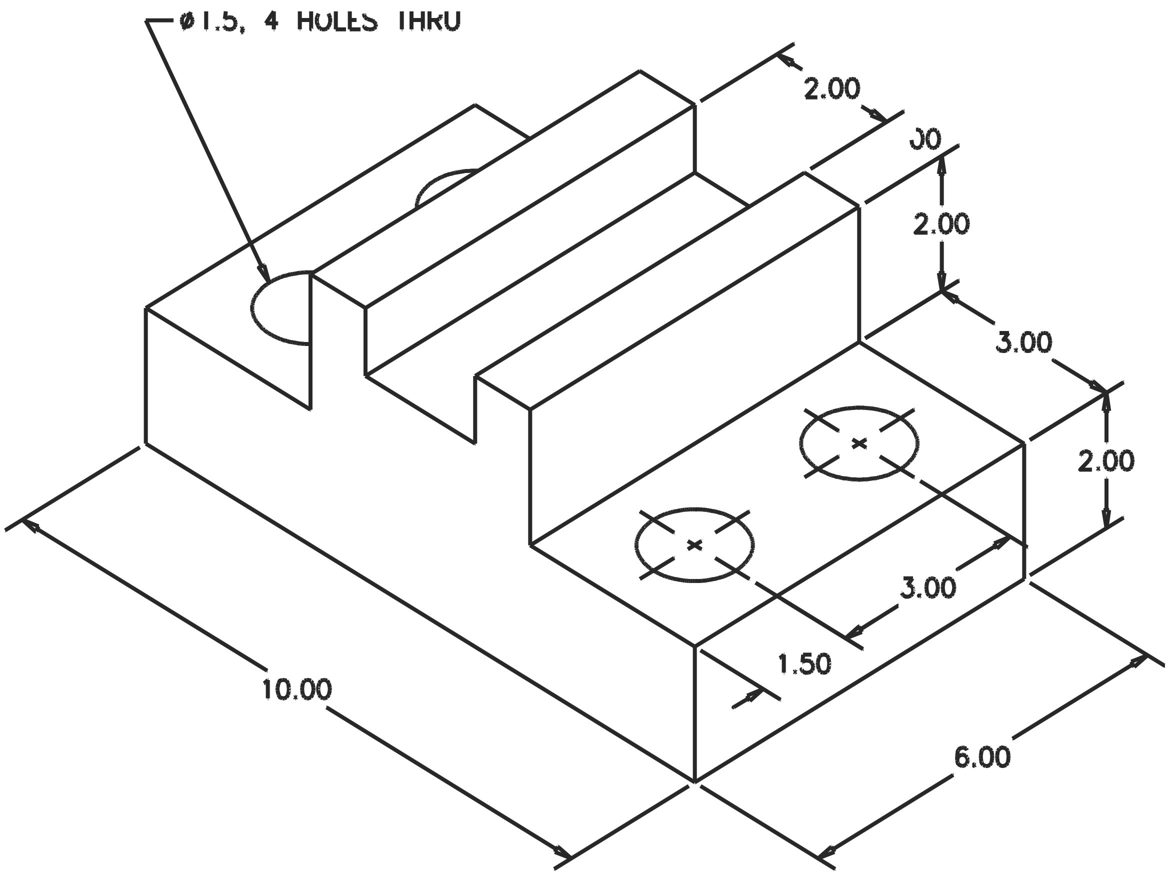
Good practice



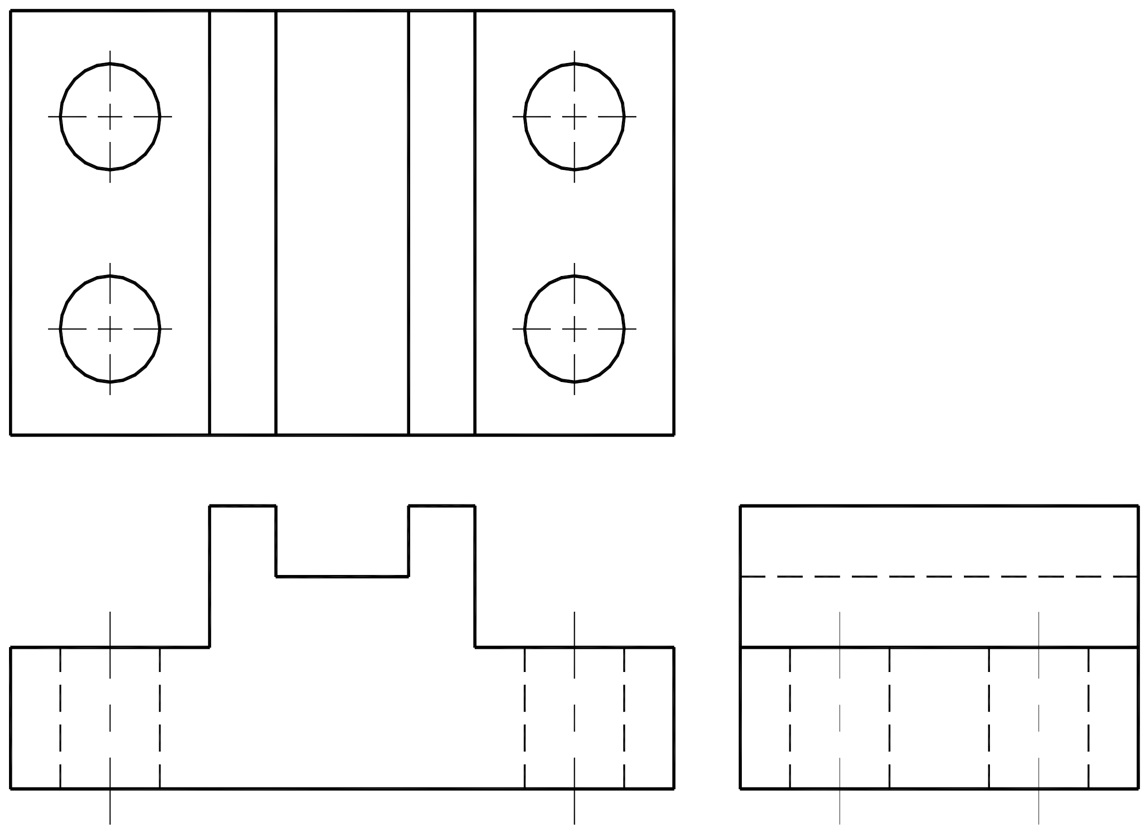
Exercise

- Complete three orthographic views of the object shown on the next slide.
- Include visible, hidden, and center lines where appropriate.
- You will be given 7 minutes.

Object for exercise



Solution




Drawing Standard

Introduction

Standards are set of rules that govern how technical drawings are represented.

- Drawing standards are used so that drawings convey the same meaning to everyone who reads them.

Standard Code

Country	Code	Full name
USA	ANSI	**A**merican **N**ational **S**tandard **I**nstitute
Japan	JIS	**J**apanese **I**ndustrial **S**tandard
UK	BS	**B**ritish **S**tandard
Australia	AS	**A**ustralian **S**tandard
Germany	DIN	**D**eutsches **I**nstitut für **N**ormung
	ISO	**I**nternational **S**tandards **O**rganization

Partial List of Drawing Standards

Code number	Contents
JIS Z 8311	***Sizes and Format of Drawings***
JIS Z 8312	***Line Conventions***
JIS Z 8313	***Lettering***
JIS Z 8314	***Scales***
JIS Z 8315	Projection methods
JIS Z 8316	Presentation of Views and Sections
JIS Z 8317	Dimensioning

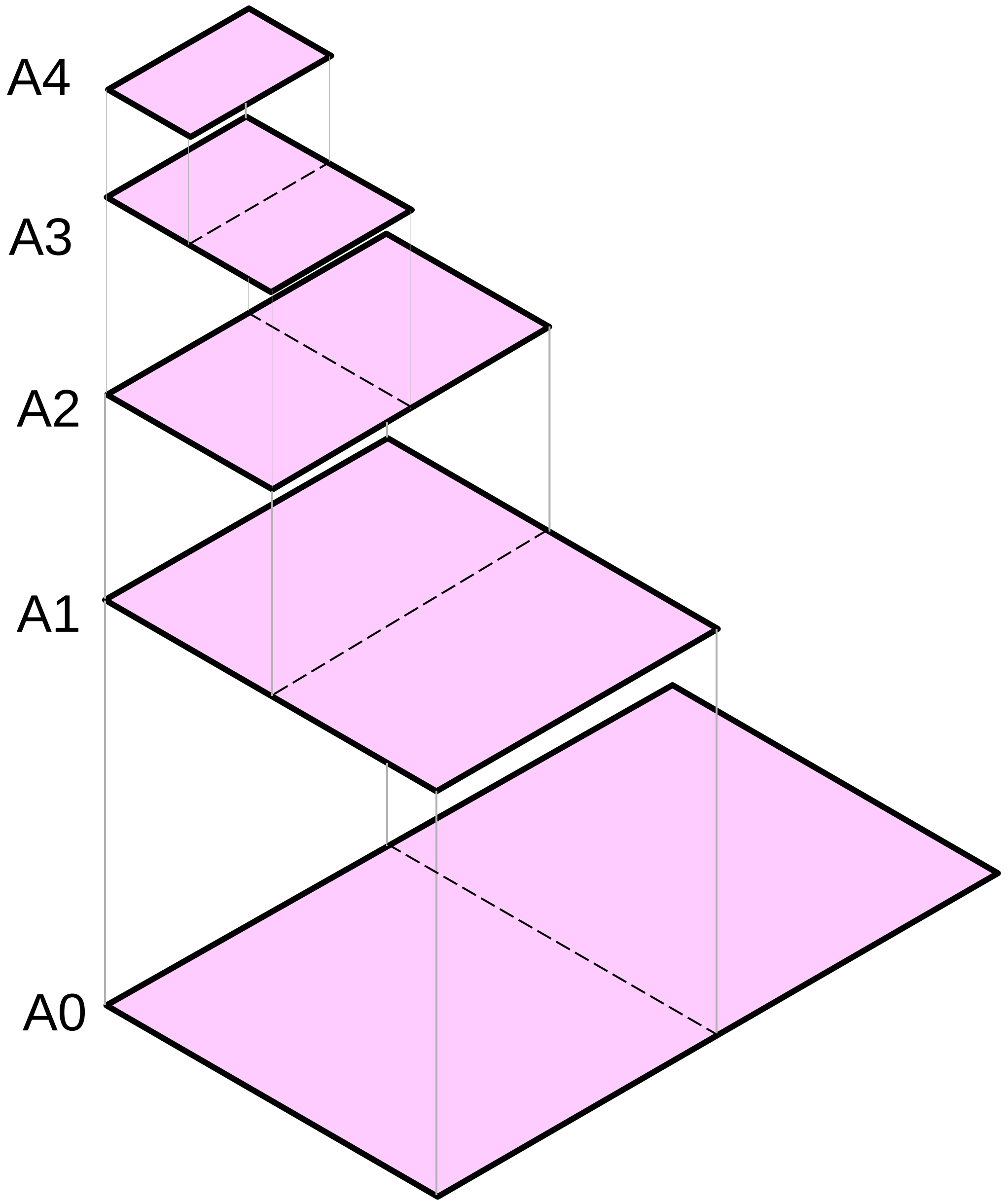
Drawing Sheet

- Trimmed paper of a size A0 ~ A4.

- Standard sheet size (**JIS**)

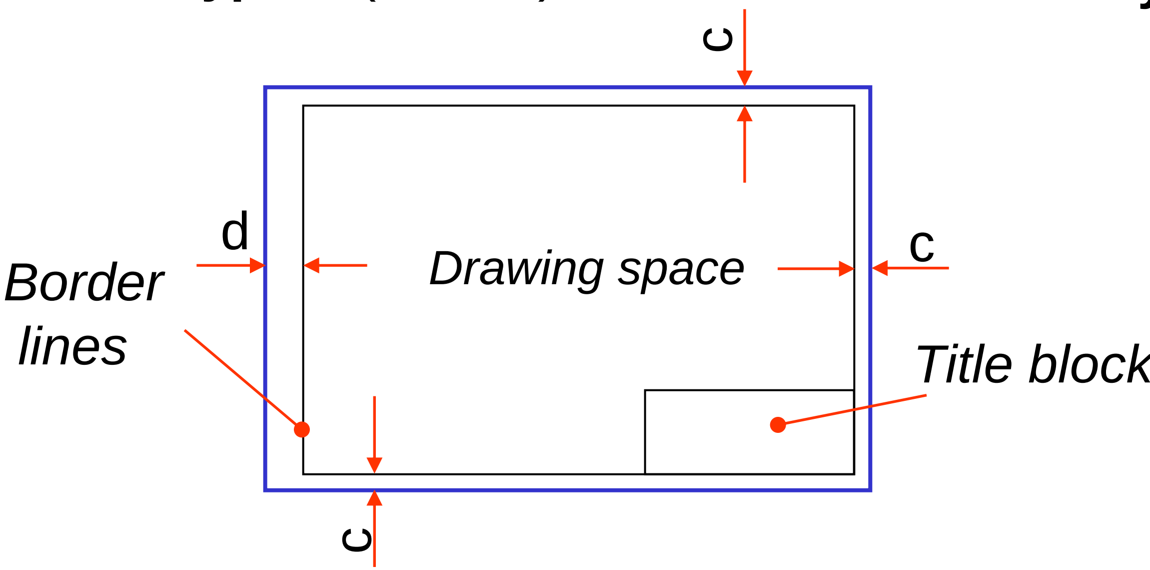
A4	210 x 297
A3	297 x 420
A2	420 x 594
A1	594 x 841
A0	841 x 1189

(Dimensions in millimeters)

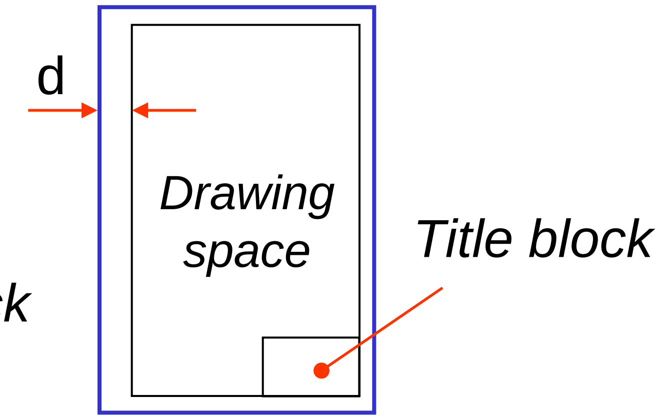


Orientation of drawing sheet

1. Type X (A0~A4)



2. Type Y (A4 only)

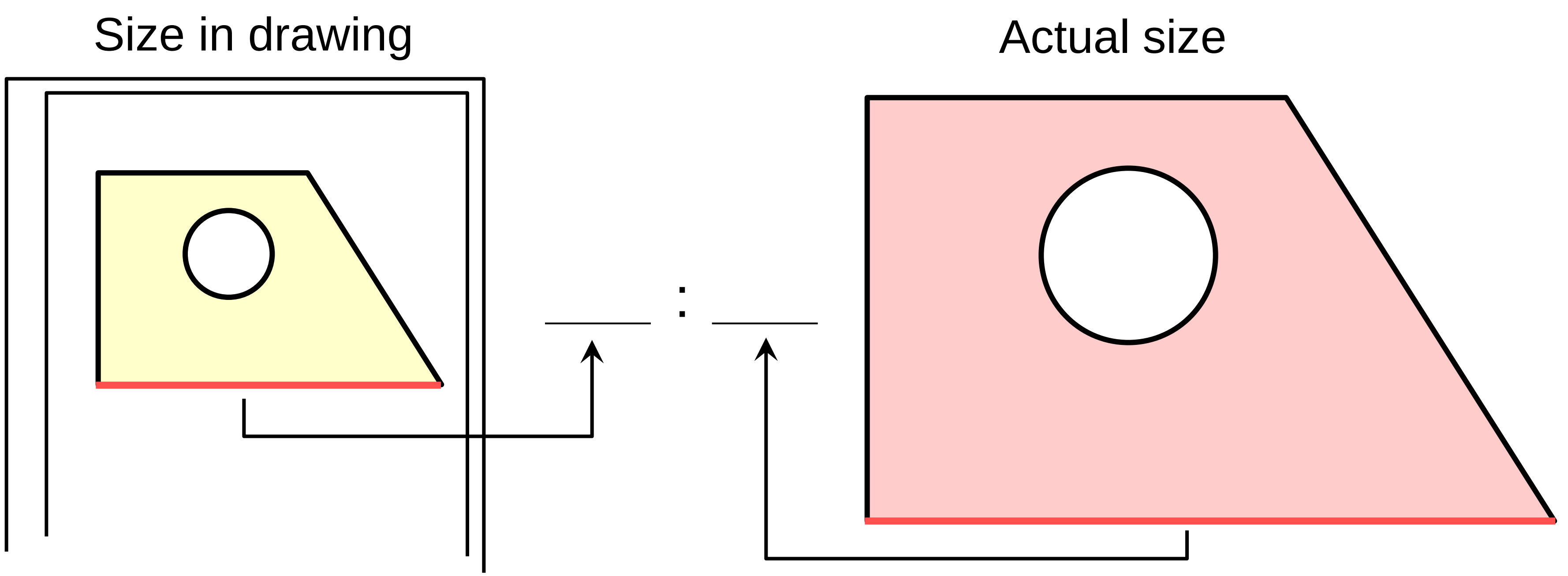


Sheet size	c (min)	d (min)
A4	10	25
A3	10	25
A2	10	25
A1	20	25
A0	20	25

Drawing Scales

Length, size

Scale is the ratio of the linear dimension of an element of an object shown in the drawing to the real linear dimension of the same element of the object.



Drawing Scales

- Designation of a scale consists of the word "SCALE" followed by the indication of its ratio, as follow

SCALE 1:1 for full size

SCALE X:1 for ***enlargement*** scales

SCALE 1:X for ***reduction*** scales

Course notes - Engineering Drawing

Contents:

1 The design process and the role of the design model.

1.1 The design process:

Almost everything around us has been created by, or is influenced by, engineers:

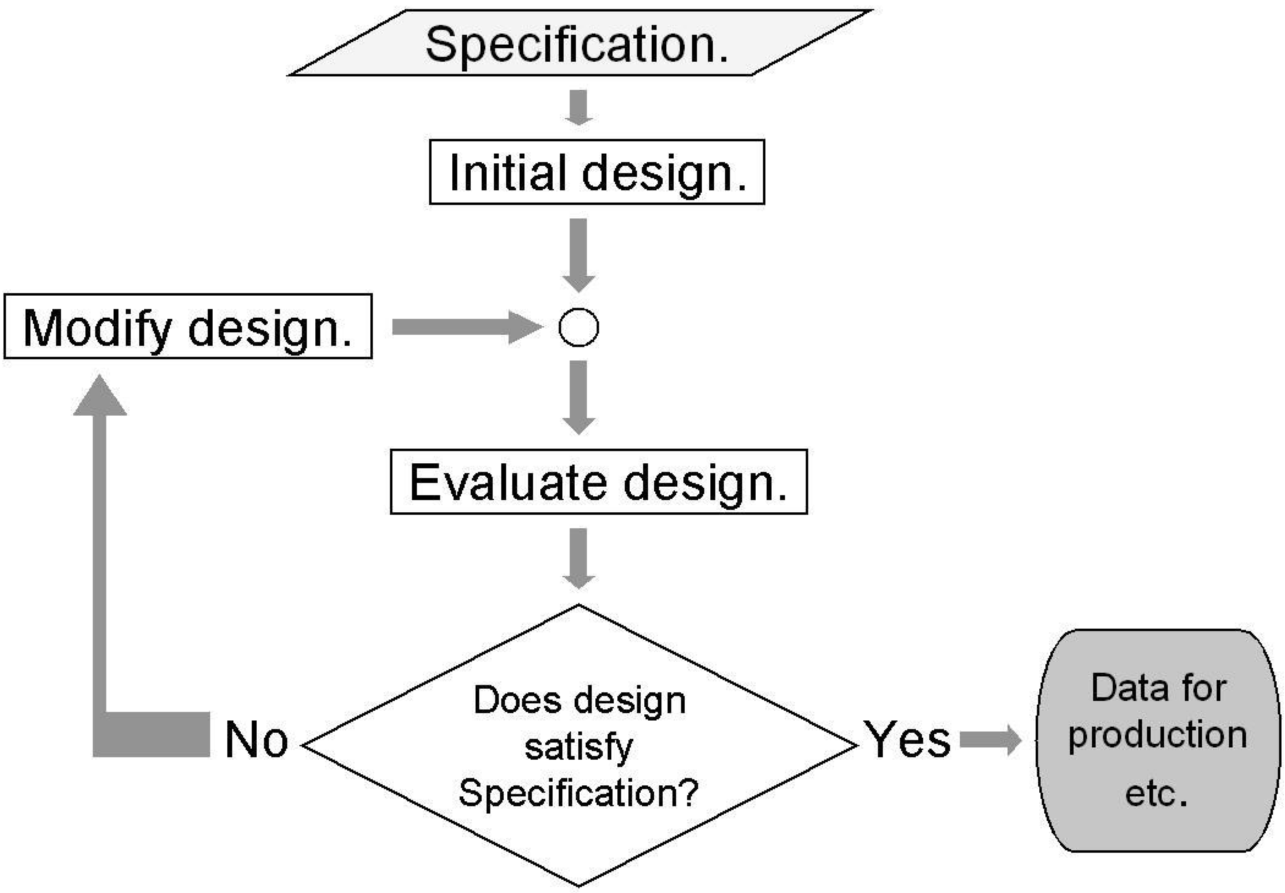
Buildings, vehicles, roads, railways, food growing and processing, books, medical care, recreation, etc.

All of these have either been conceived and created from scratch or have evolved from existing ideas. Either way, an engineering design process will have been followed, in one form or another. The Design as a generic tool module provides an interesting a comprehensive introduction to engineering and design, so a detailed discussion of the design process will not be included here.

In essence, designs progress from :

some statement of need
to..
identification or specification of problem
to..
search for solutions
and finally to...
development of solution to manufacture, test and use.

This sequence is usually iterative. It repeats until a satisfactory solution has evolved, as indicated in the flow diagram below.



1.2 The design model.

The concept of the designer working with a model of a design is fundamental to the design process.

The design model is a representation of the design. This model could be anything from a few ideas in the designers head, through to rough sketches and notes, calculations, sets of detailed formal engineering drawings, computer generated 3D representations, physical prototypes, etc.

The design model would be used by the designer to record and develop ideas and to provide a basis to evaluate the design.

Larger design projects are undertaken by more than one engineer. Design models are used to communicate and demonstrate ideas between all those concerned with the product design, development, manufacture and use.

A designer needs to have the skills to generate and work with this model in order to communicate ideas and develop a design.

1.3 Types of design model.

Designers use a variety of different models, depending on what property of the design is to be considered and for whom the information is destined.

Typically a designer may model:

- Function
- Structure
- Form
- Material properties, surface conditions

All of these areas probably encompass a large portion of the degree syllabus. Within this module we will concern ourselves primarily with form, i.e. the shape of parts or components and how they fit together.

2.1 Projections.

2.1.1 Orthographic projection.

We have discussed both the role of the design model in the design process and the importance of the representation of the form or shape in this role.

Now we will consider in detail the methods designers use to represent the form of their designs.

Back in the 18th century a French mathematician and engineer, Gaspard Monge (1746-1818), was involved with the design of military armory. He developed a system, using two planes of projection at right angles to each other, for graphical description of solid objects.

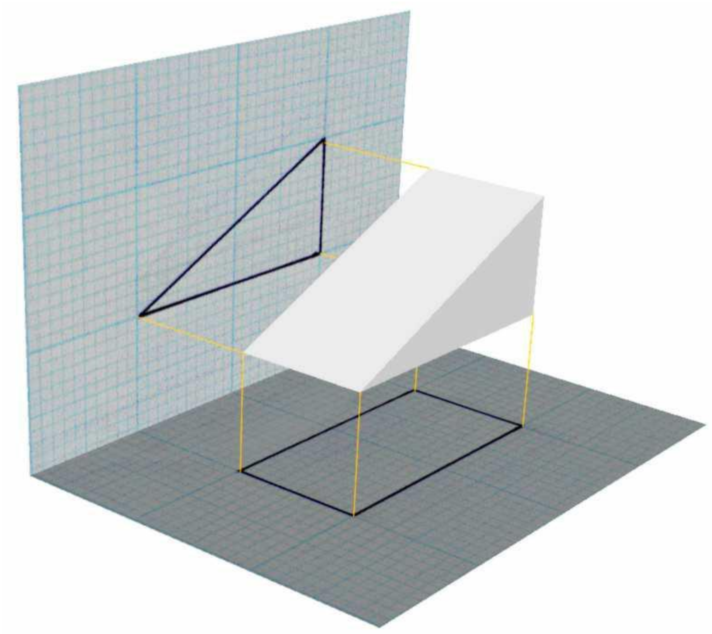


Figure 2.1a, two right angle planes of projection.

This system, which was, and still is, called Descriptive Geometry, provided a method of graphically describing objects accurately and unambiguously. It relied on the perpendicular projection of geometry from perpendicular planes.

Monge's Descriptive Geometry forms the basis of what is now called Orthographic Projection.

The word orthographic means to draw at right angles and is derived from the Greek words:

ORTHOS - straight, rectangular, upright
GRAPHOS - written, drawn

Orthographic projection is the graphical method used in modern engineering drawing. In order to interpret and communicate with engineering drawings a designer must have a sound understanding of it's use and a clear vision of how the various projections are created.

There are two predominant orthographic projections used today. They are based on Monge's original right angle planes and are shown fully in Figure 2.1b. They define four separate spaces, or quadrants. Each of these quadrants could contain the object to be represented. Traditionally however, only two are commonly used, the first and the third.

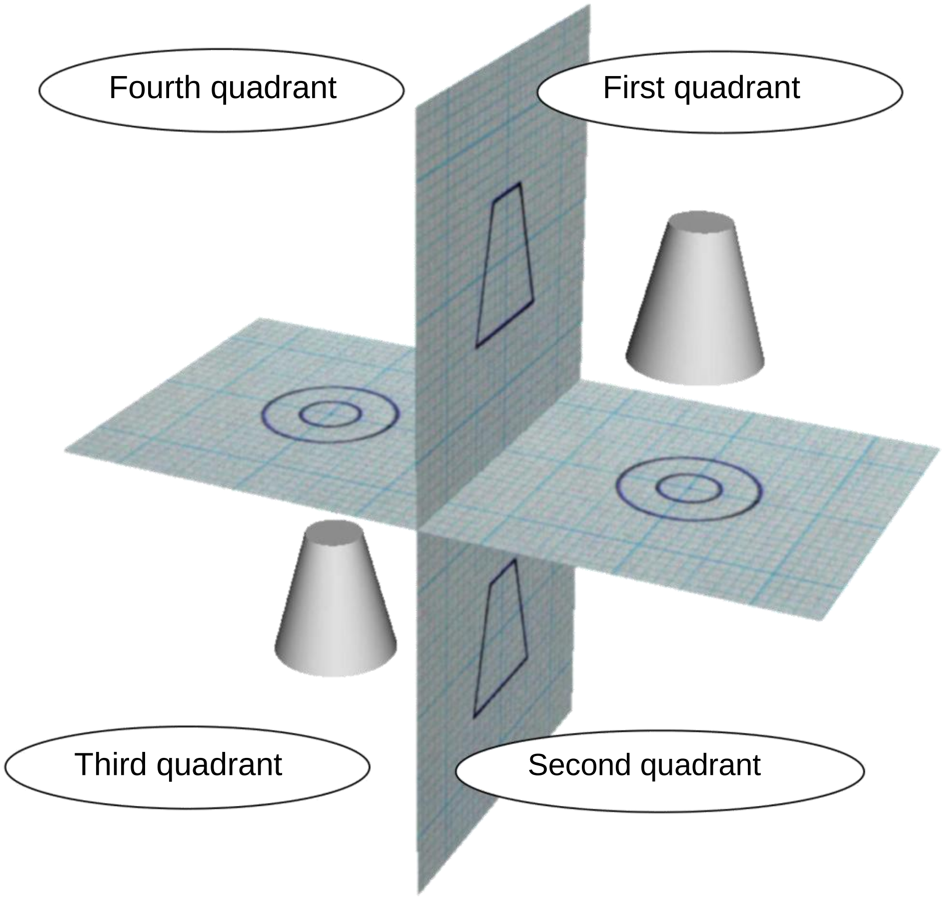


Figure 2.1b.

Projections created with the object placed in the first quadrant are said to be in First Angle projection, and likewise, projections created with the object placed in the third quadrant are said to be in Third Angle projection.

2.1.2 First angle projection.

Consider the first quadrant in Figure 2.1b. The resultant drawing of the cone would be obtained by flattening the two perpendicular projections planes, as shown in Figure 2.1c.

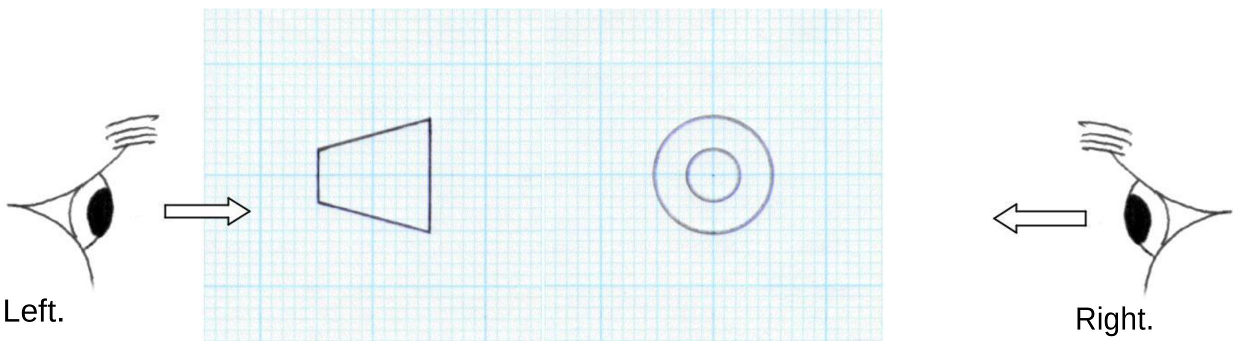


Figure 2.1c, First Angle.

For this example, you could say that the right hand side image is the plan or top elevation and the image to the left is the side elevation.

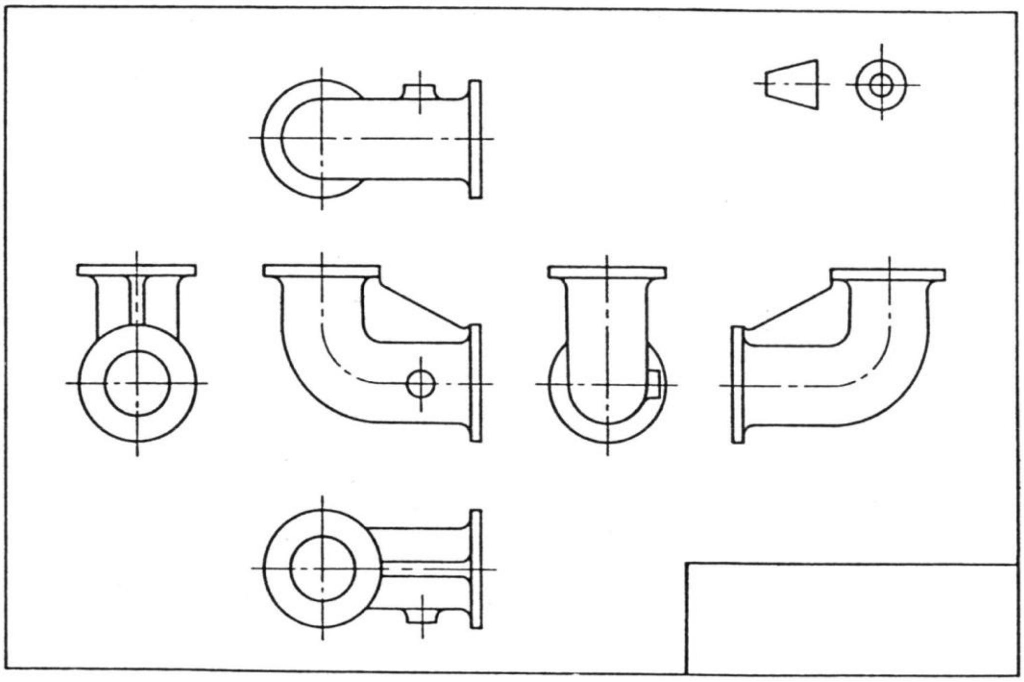
Whether you view the objects from the left or the right, the order in which the drawing views are arranged puts the image that you see after the object, object first then the image. This is always true for First Angle projection.

Put another way:

- Viewing from the left: The drawn image on the right is your view of the drawn object on the left.
- Viewing from the right: The drawn image on the left is your view of the drawn object on the right.

This can get confusing, particularly when also considering other drawings created using other projections. You may develop your own way of recognising First Angle projection. The author uses:

The OBJECT is FIRST for FIRST Angle projection.
or...
EYE > OBJECT > IMAGE
or...
You look through the object and place the image



An example of a component represented in a multiview drawing, in First Angle projection.

2.1.3 Third angle projection.

Consider the third quadrant in Figure 2.1b. The resultant drawing of the cone would be obtained by flattening the two perpendicular projections planes, as shown in Figure 2.1d.

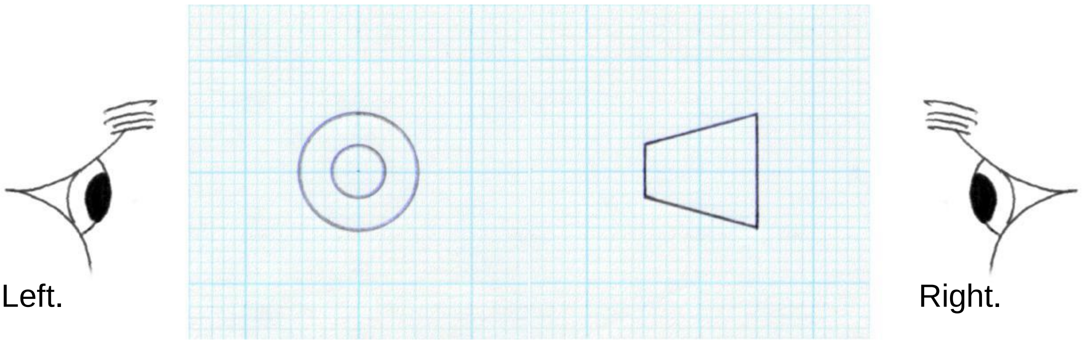


Figure 2.1d, Third Angle.

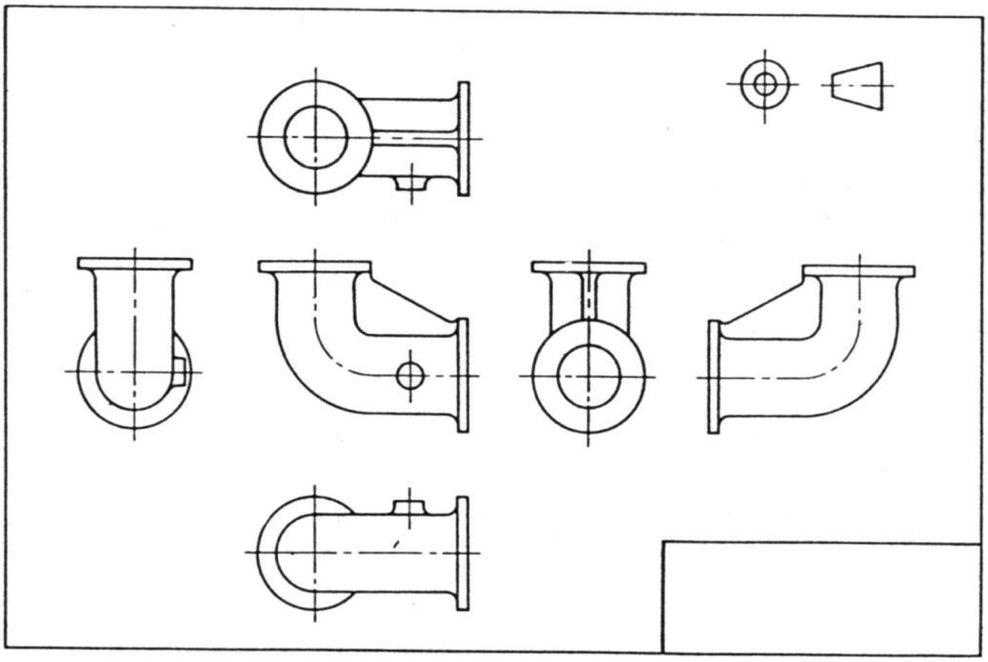
For this example of the cone, you would say that the left hand image is the plan or top elevation and the image to the right is the side elevation.

Whether you view the objects from the left or the right, the order in which the drawing views are arranged puts the image that you see before the object, image first then the object. This is always true for Third Angle projection.

Put another way:

- Viewing from the left: The drawn image on the left is your view of the drawn object on the right.
- Viewing from the right: The drawn image on the right is your view of the drawn object on the left.

Again, you may develop your own way of recognising Third Angle projection. Perhaps: EYE > IMAGE> OBJECT



The same component shown using Third Angle projection.

2.1.4 Orthographic projection symbols.

Both systems of projection, First and Third angle, are approved internationally and have equal status. The system used must be clearly indicated on every drawing, using the appropriate symbol shown in Figure 2.1e below.

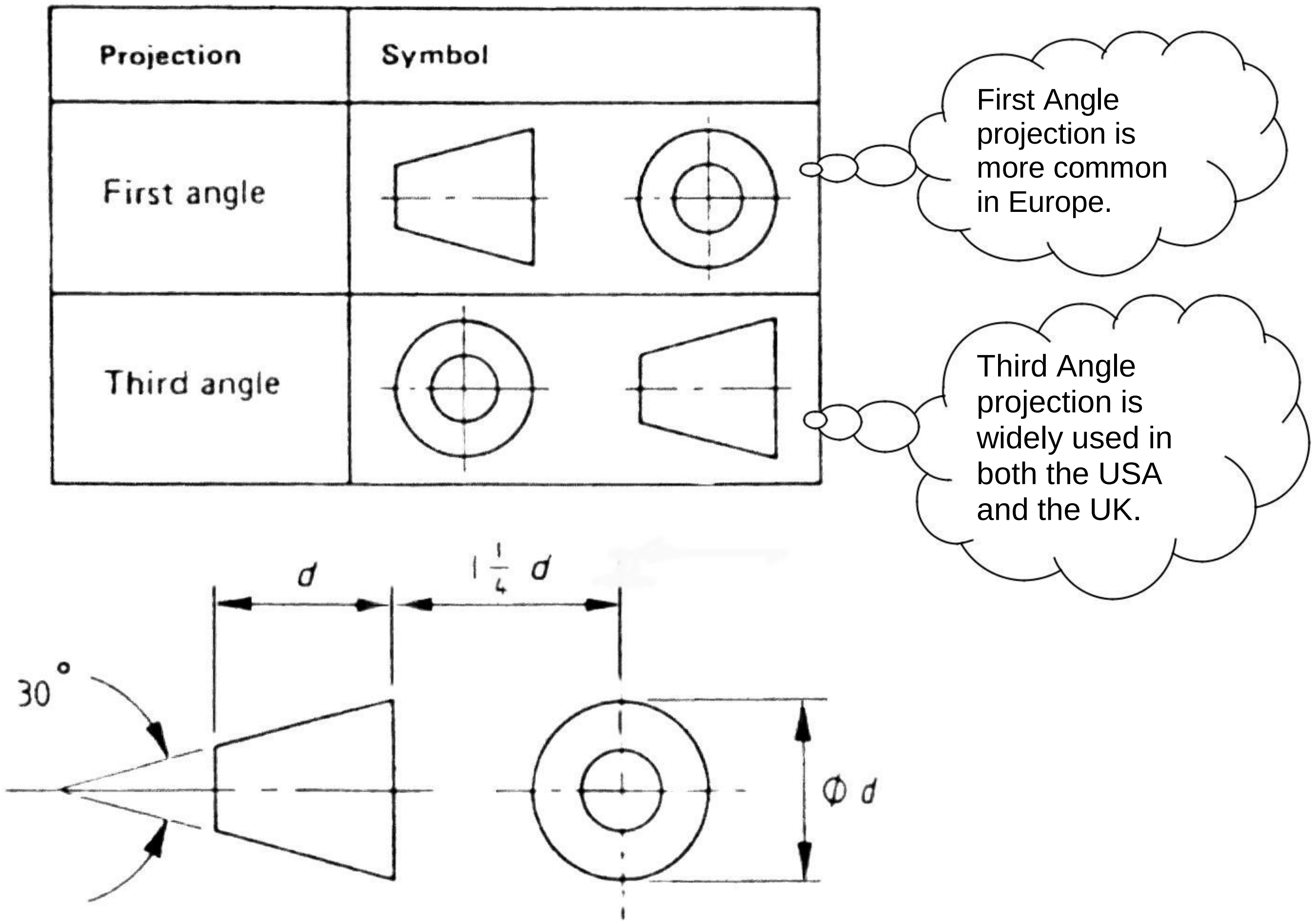
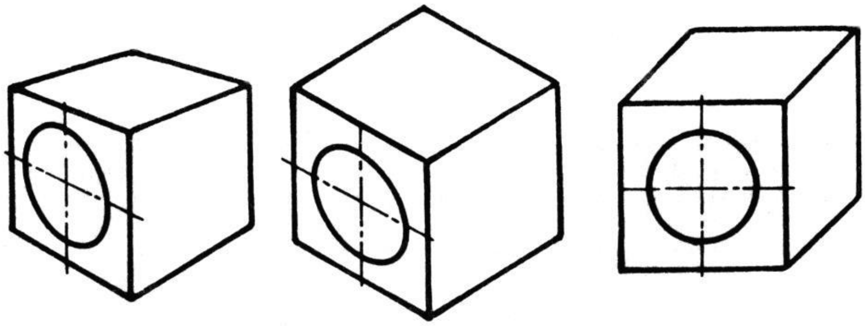


Figure 2.1e. Projection system symbols and recommended proportions.

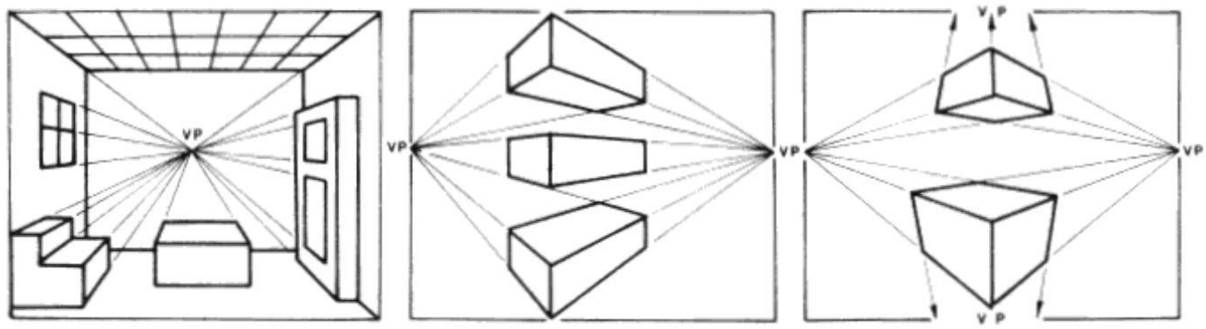
2.1.5 Pictorial Drawing.

Orthographic projection is used as an unambiguous and accurate way of providing information, primarily for manufacturing and detail design. This form of representation can however make it difficult to visualise objects. Pictorial views can be created to give a more three dimensional impression of the object. There are three types of pictorial projections commonly used, as shown below.

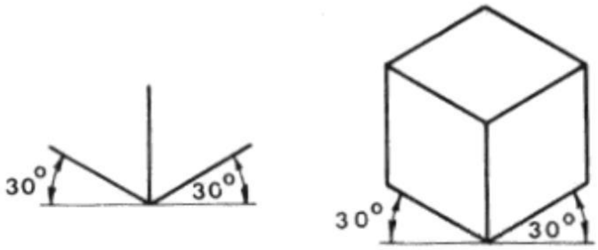


Perspective, isometric and oblique pictorial projections.

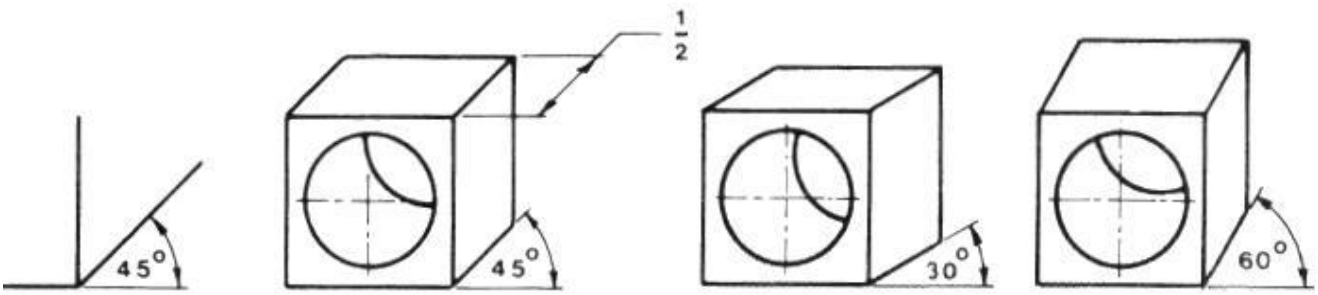
Perspective: Used more with freehand sketching.
Parallel lines appear to converge and meet at what is referred to as the vanishing point. You can have one, two or three vanishing points (VP).



Isometric: Receding lines drawn at 30º and are usually kept at true measured lengths.



Oblique: Front face sketched as a true shape. Starts with two axes, one horizontal, one vertical. The third axis is usually drawn at 45º and lengths are reduced by 50% of true lengths. Sometimes called 'cabinet' projection.



2.2 Creating orthographic projection drawings.

This is an introduction into how to create and interpret multi-view orthographic projection drawings.

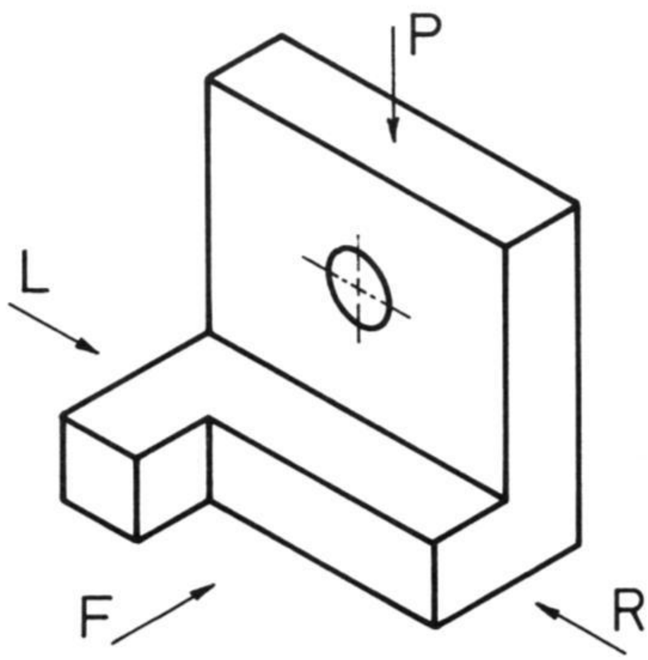
2.2.1 First angle projection.

The component:

Your drawing will, for this example consist of four views:

- Front F
- Left L
- Right R
- Plan (Top) P

Usual practice is to orient the component in a position that it is most likely to be found in.



Your aim is to create, from the front view, an orthographic projection drawing as shown below in Figure 2.2a. Note how the views are constructed in line with each other, allowing the features to be 'projected' between the views.

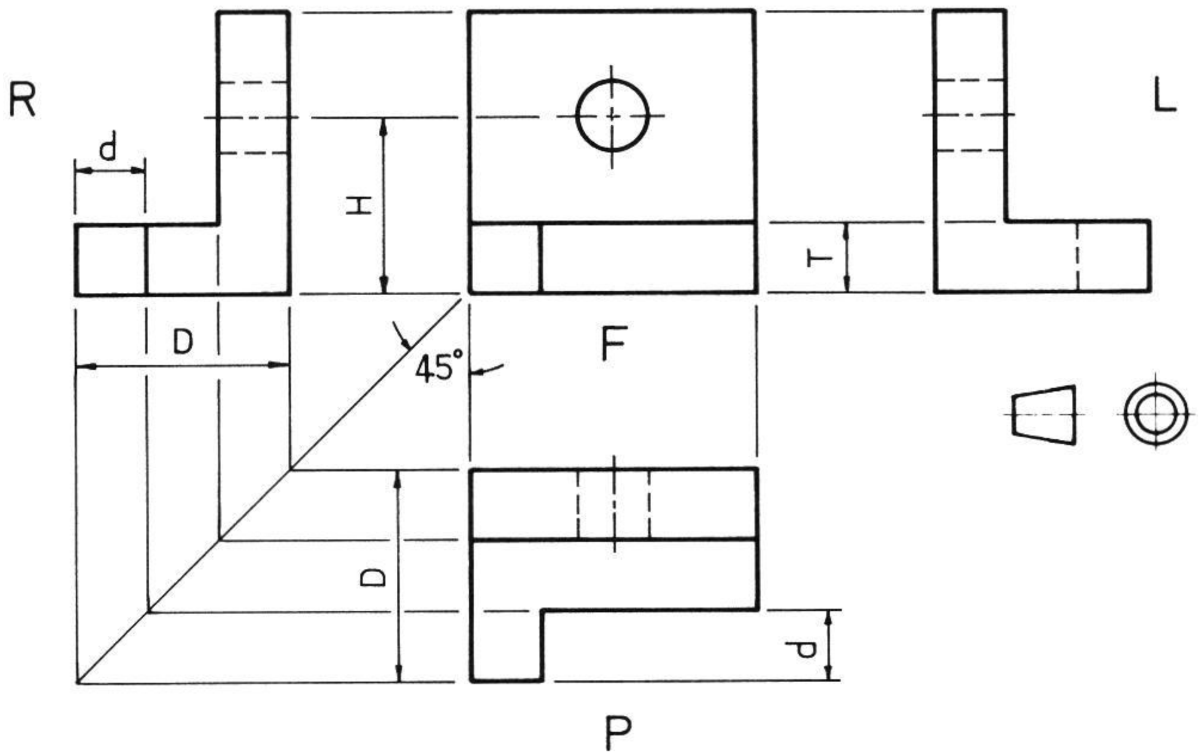
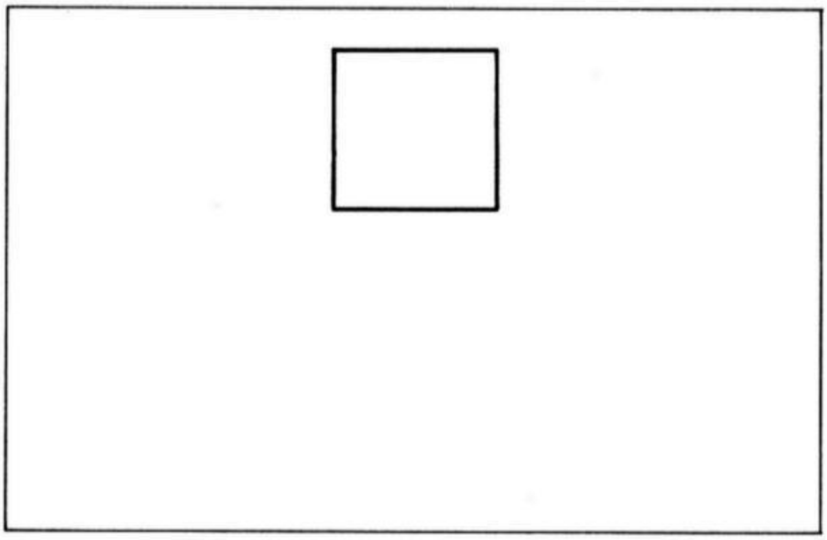


Figure 2.2a. A completed First angle projection drawing.

So, the stages are:

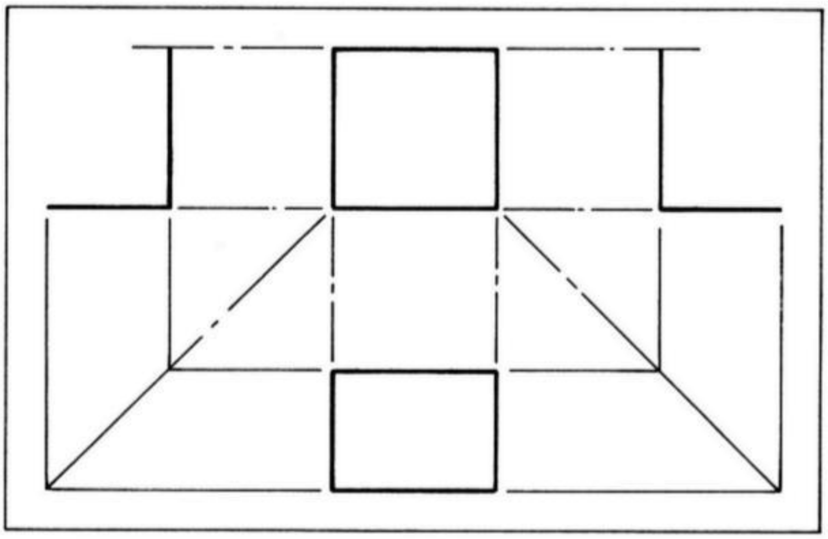
1) Choose which view direction or face will be used as the front view of the component.

2) Draw the outline of the front view, leaving room for the other views.



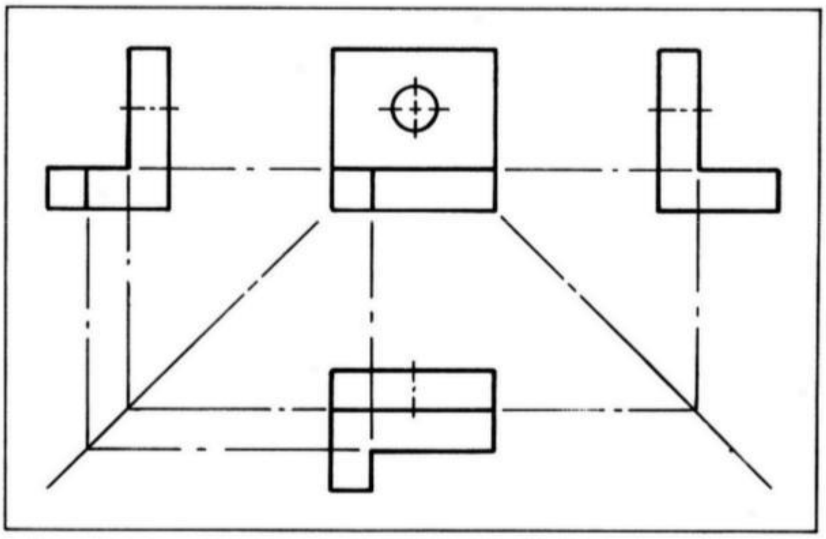
3) Draw feint construction lines out from the front view.

4) Start to draw the outlines of the other views, using sides you know the length of.



5) Complete the details of the views by adding any required hidden detail lines, other outlines and center lines.

(Refer to section 2.3 for line style conventions.)



With first angle projection the plan view is below the front view. If you had placed the plan view above the front view it would actually have to become the bottom or underside view!

2.2.2 Third angle projection.

The construction method used is the same. The difference between first and third angle projection when creating or reading really lies with the positions of the views. For the same component, an orthographic projection drawing with the same front, side and plan views would look like Figure 2.2b below.

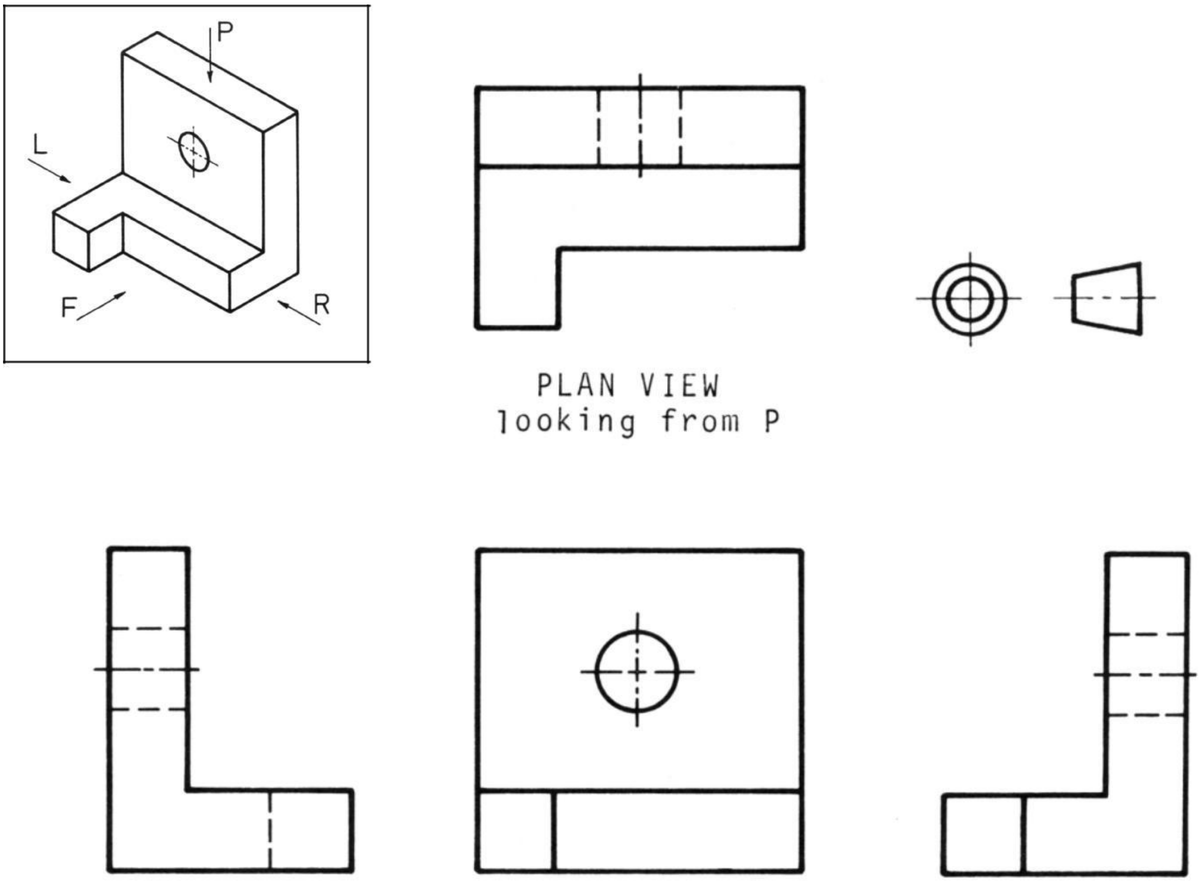


Figure 2.2b. Third angle projection.

Observe how, in third angle, the views give the image then the object. In other words, what you see then what you are looking at.

In first angle you are given the object then the image, or what you are looking at, then what you see.

2.3 Drawing conventions.

2.3.1 Introduction.

In order for anyone to be able to understand exactly what a drawing represents, sets of precise rules and conventions have to be followed, much like a language. These rules are usually referred to as Standards.

When a designer works with an engineerign drawing they must be familiar with the precise meaning of the various line styles, abbreviations, drawing simplifications and terminology as specifiied in the relevant standards. This section introduces you to some of the conventions defined in BS 8888.

Standards are developed both privately by companies and by internationally recognised institutions.

Two such international standards are:

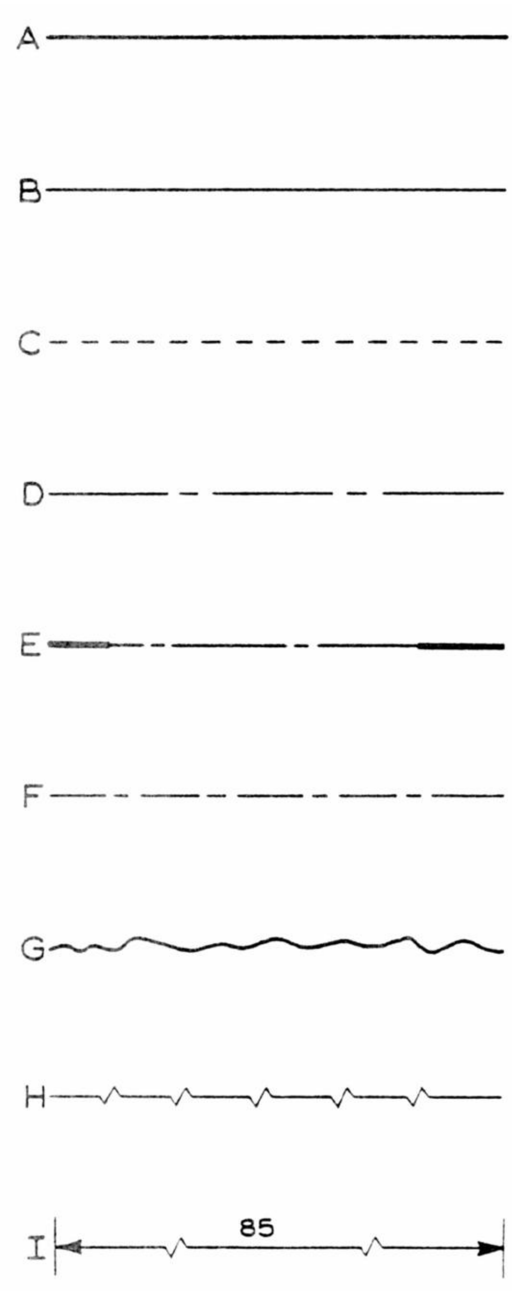
British Standard Institution: BS 8888 (Superceded BS 308)

American National Standards Institute: Y14 series

2.3.2 Line styles or types.

Each line on a drawing represents specific precise information regarding the components design.

Type: (thickness)	Example:	Application:
Continuous 0.7mm	A	Visible outlines
Continuous (thin) 0.3mm	B	Dimension lines
Short dashes 0.3mm	C	Hidden detail
Long chain 0.3mm	D	Center lines
Chain, thick at ends 0.7 – 0.3mm	E	Section cutting planes
Short chain 0.3mm	F	Developed views
Continuous wavy boundaries 0.3mm	G	Broken
Straight zigzag 0.3mm	H	Break lines
Straight lines with two short zigzags 0.3mm	I	Dimension lines



2.3.3 Lettering.

All characters on a drawing must be legible and consisitent, with consideration being given to the possibility of drawing reductions and poorer quality reproductions being made.

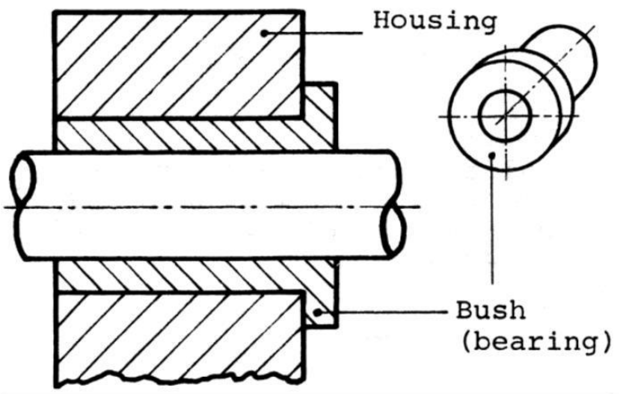
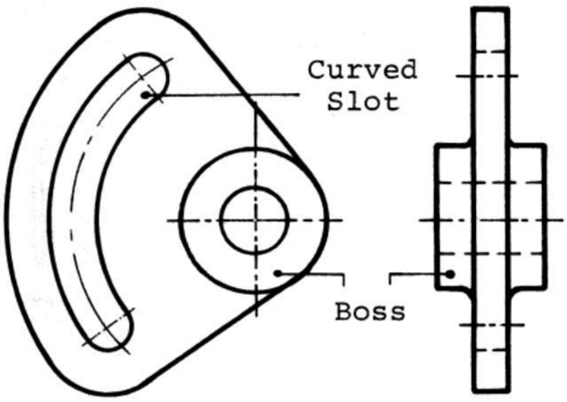
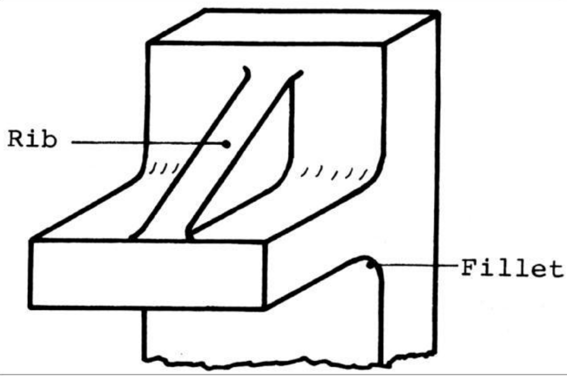
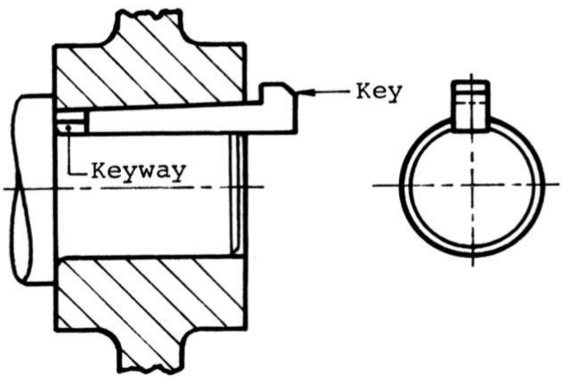
No particular style is required, but characters should all be consisitent on the same drawing. Capital letters are preferred to lower case ones.

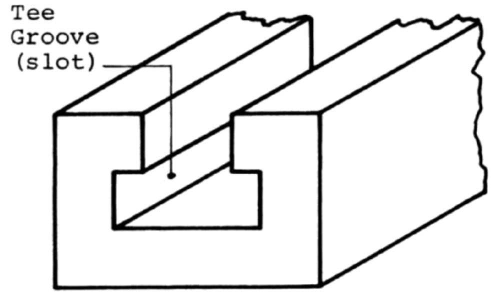
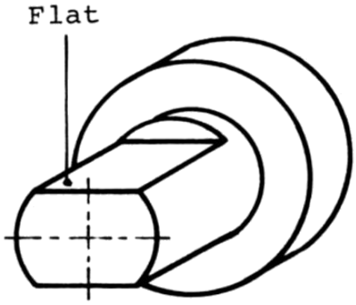
Size of lettering is given as a minimum height, relating to drawing size, as shown below:

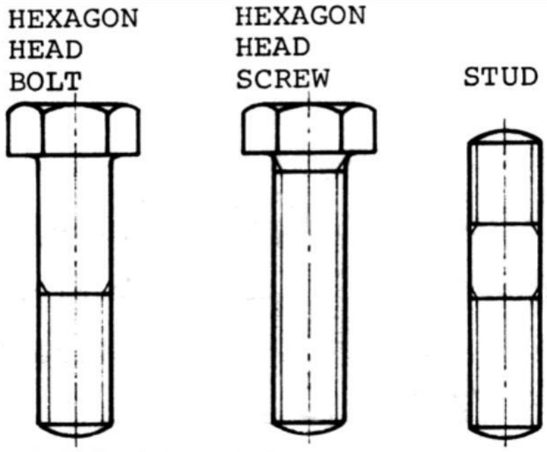
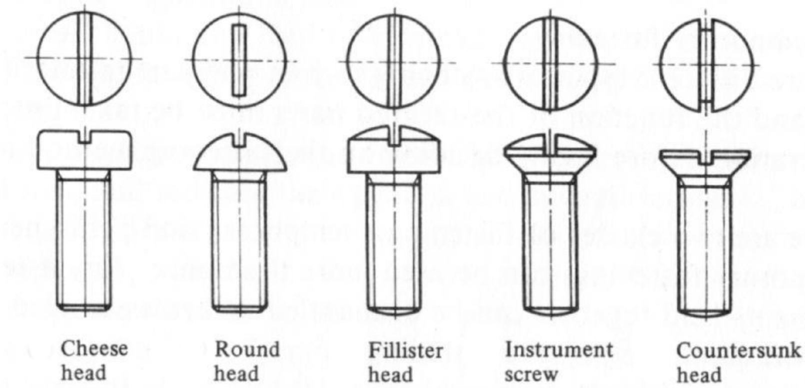
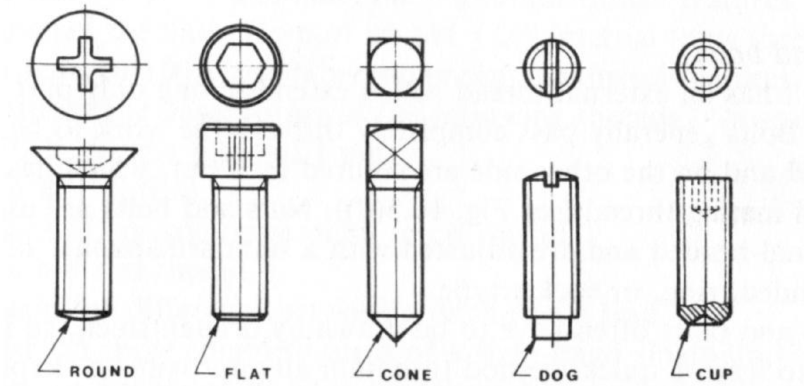
Application	Drawing sheet size	Min. character height (in mm)
Drawing numbers	A0, A1, A2 & A3	7
Titles, etc.	A4	5
Dimensions &	A0	3.5
Notes.	A1, A2, A3 & A4	2.5

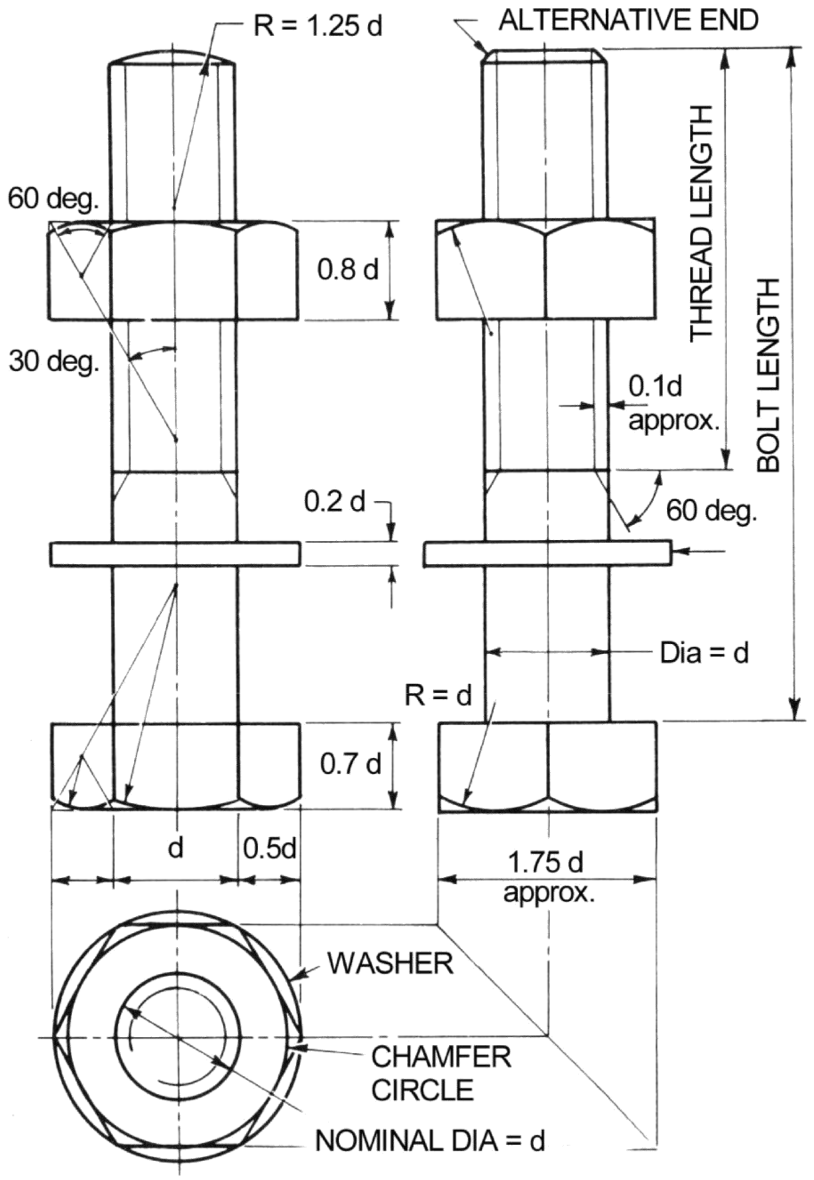
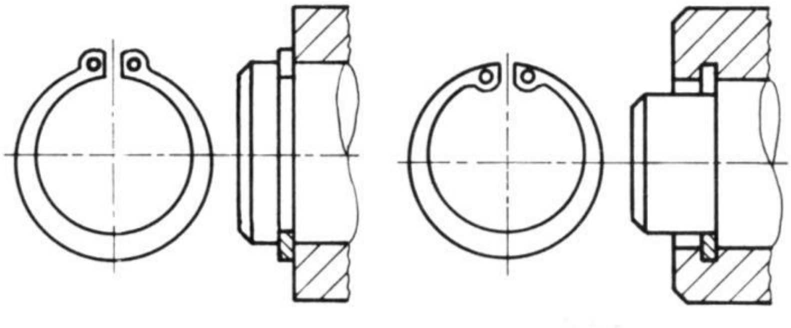
2.3.3 Terminology & representations of standard components.

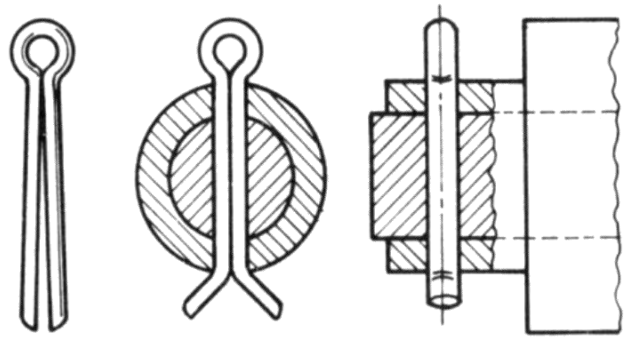
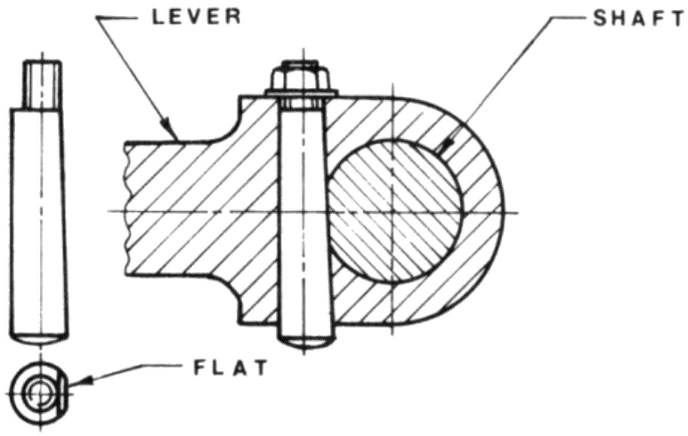
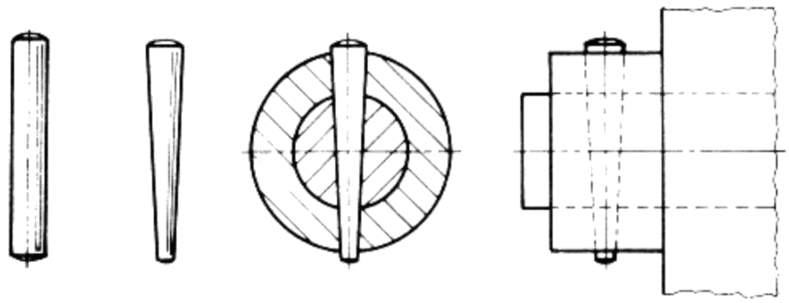
Here are some examples of commonly used engineering components and features of components.

<table>
<tr><td colspan="2">General:</td></tr>
<tr><td>Housing:
A component into which a 'male' mating part fits, sits or is 'housed'.</td><td rowspan="2">
</td></tr>
<tr><td>Bush/bearing:
A removable sleeve or liner. Known also as a simple or plane bearing.</td></tr>
<tr><td>Boss:
A cylindrical projection on surface of component.</td><td rowspan="2">
</td></tr>
<tr><td>Curved slot:
Elongated hole, whose centerline lies on an arc. Used usually on components requiring adjustment.</td></tr>
<tr><td>Rib:
A reinforcement, positioned to stiffen surfaces.</td><td rowspan="2">
</td></tr>
<tr><td>Fillet:
A radius or rounded portion suppressing a sharp internal corner.</td></tr>
<tr><td>Key:
A small block or wedge inserted between a shaft and a mating part (a hub). Used to prevent relative rotation of the two parts.</td><td rowspan="2">
</td></tr>
<tr><td>Key way:
A parallel sided slot or groove cut into a bore or a shaft, to 'house' a mating key.</td></tr>
</table>

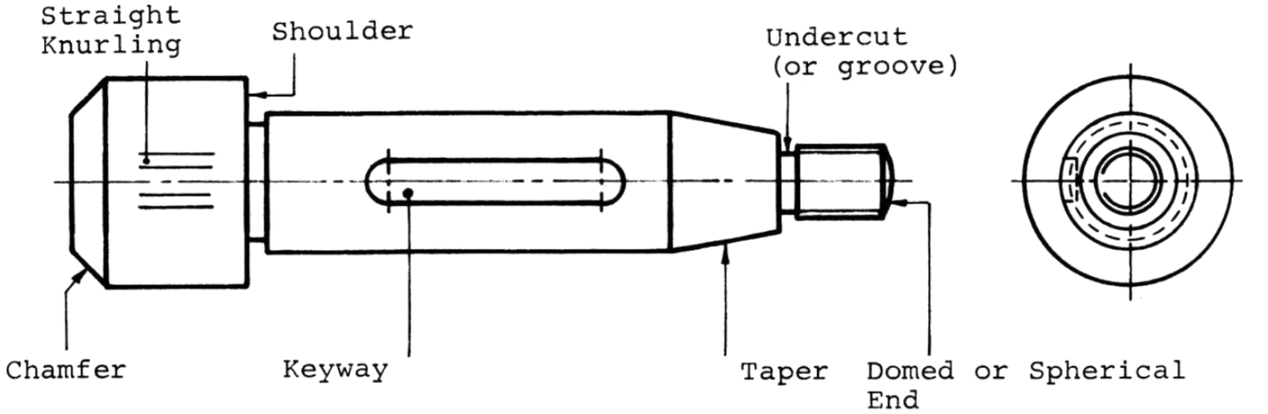
Tee Groove (slot): Machined to 'house' mating fixing bolts and prevent them from turning.	
Flat: A surface machined parallel to the shaft axis.	

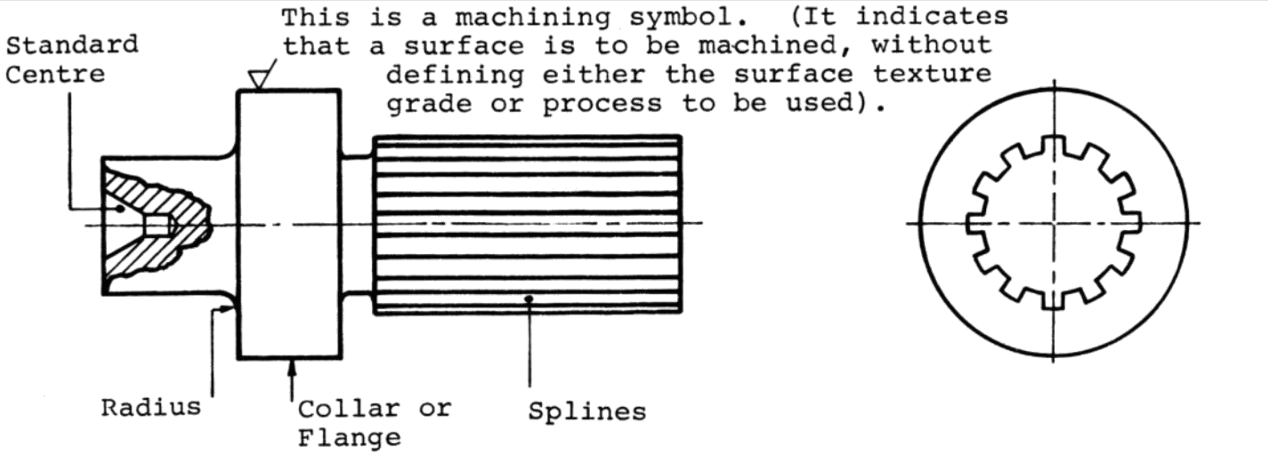
Fasteners:	
Bolts, screws & studs: Threaded fasteners. Bolts have a shank partially threaded, whereas screws are threaded along the entire length. For guidance on dimensioning, see next page.	 
The last three examples here are called set screws and are used to position or lock components.	

This diagram gives approximate dimensioning methods for drawing hexagon headed metric bolts, nuts and plane washers. (Manufacturers data sheets may give more accurate measurements.)	
Circlip:	
Internal & external.	

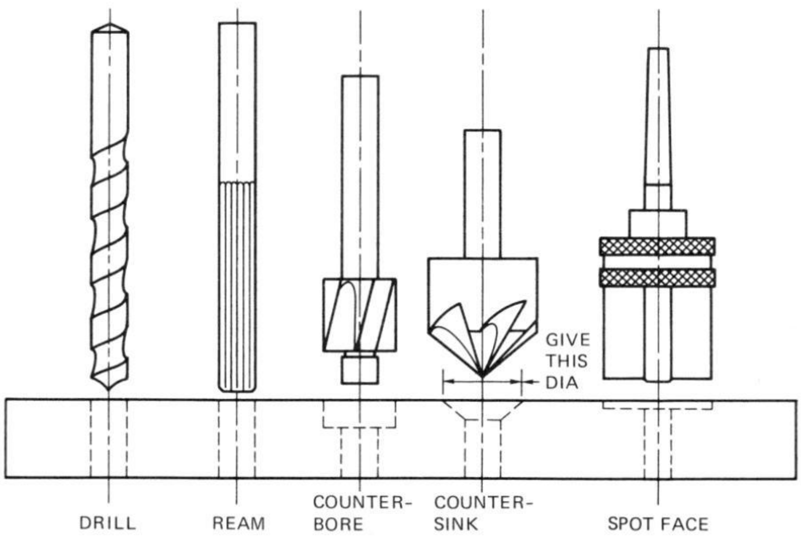
Pins:	
Split Cotter Pin: Used to lock components, prevent fasteners from coming 'un-fastened'. e.g. lock-nuts on suspension systems.	
Cotter Pin: Used to retain components, usually where loads are transmitted.	
Dowel Pin & Taper Pin: Provides location, alignment.	

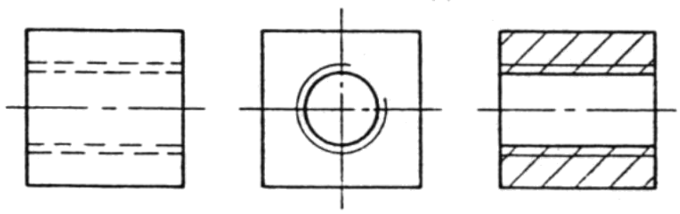
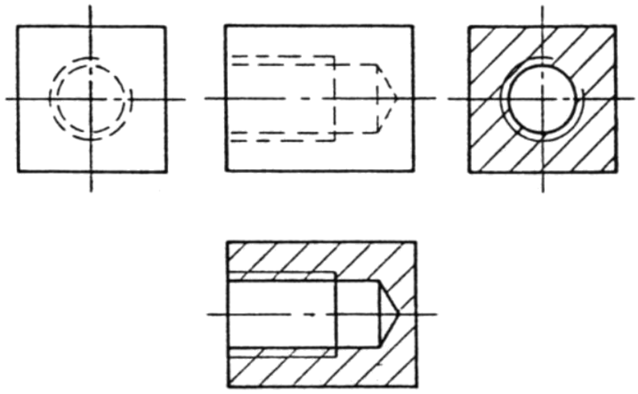
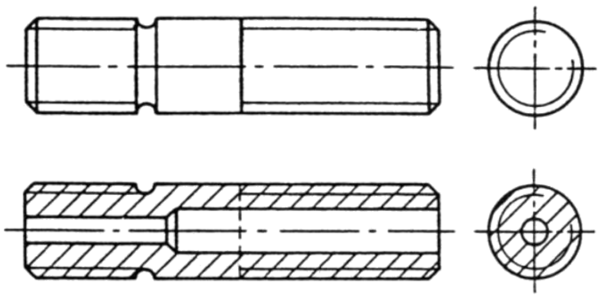
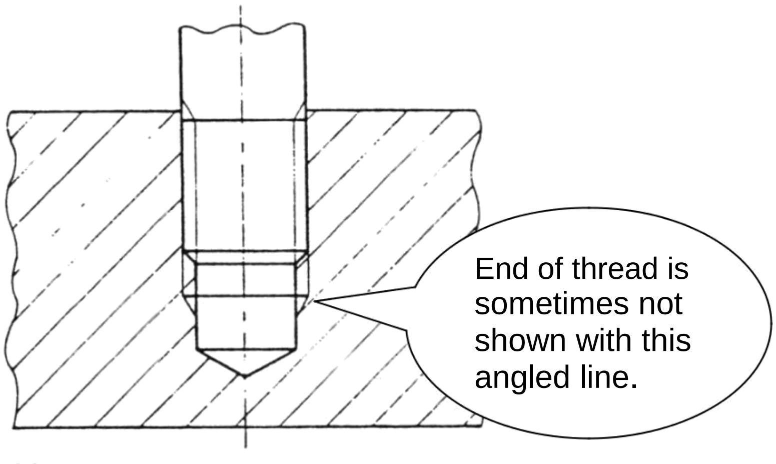
Features usually relating to components turned on a lathe:

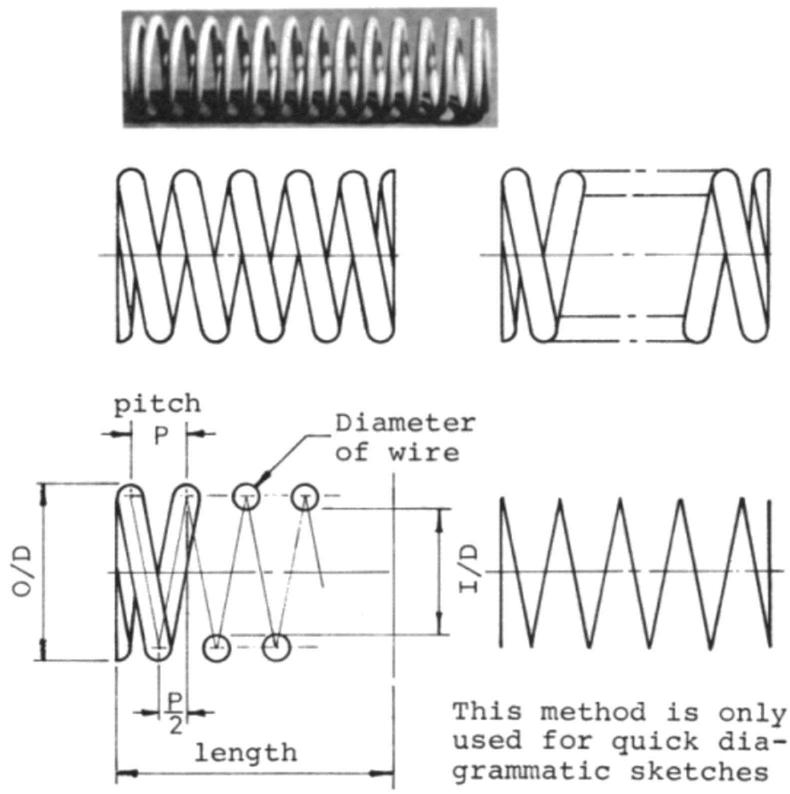
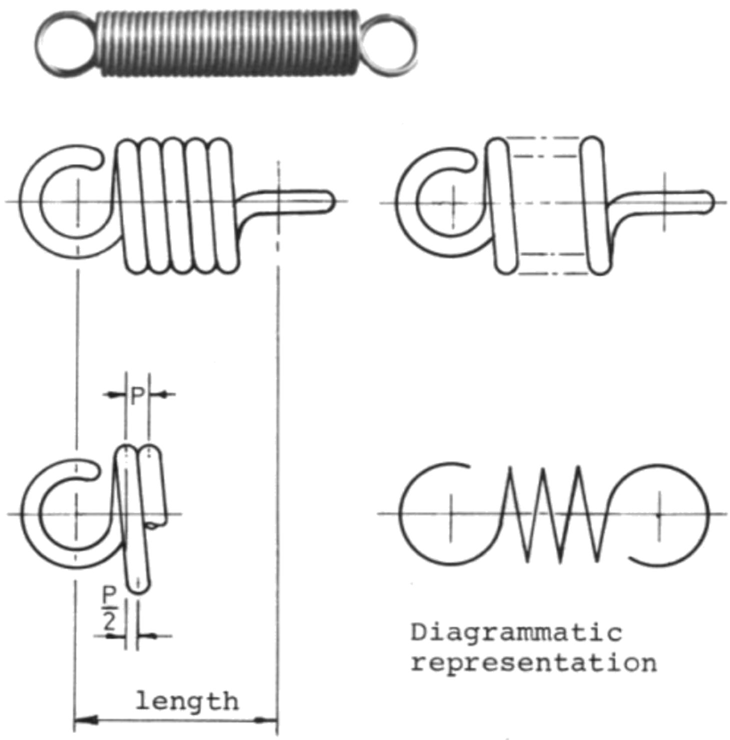


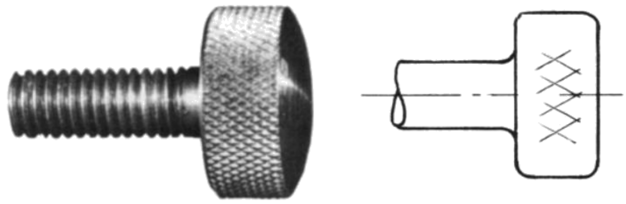
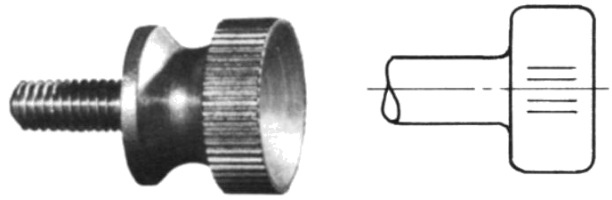


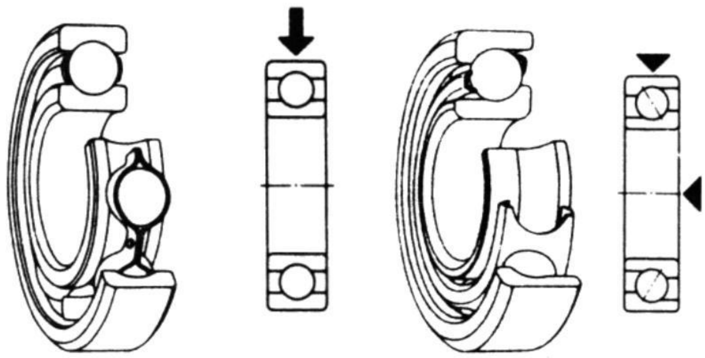
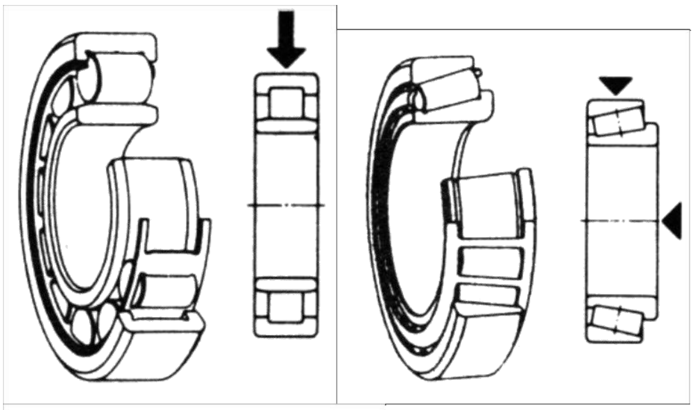
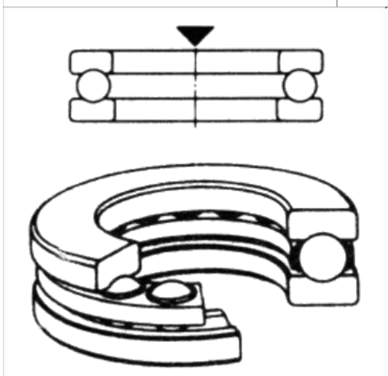
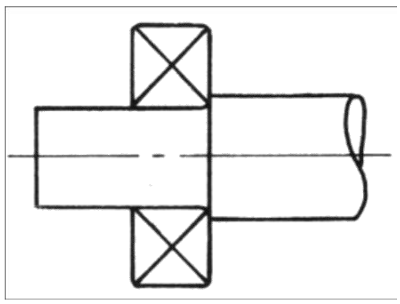
Holes:

Drilled: Loose tolerance, for pilot holes or clearance holes for fasteners.	
Reamed: Accurate finishing process after drilling or boring.	
Counterbore: Usually used to recess the head of a square shouldered fastener.	
Countersunk: Usually used to recess the head of a countersink screw.	
Spotface: Used to clean up and level the surrounding area, usually for a fastener or something such as a hydraulic fitting using a seal.	

Screw threads:	
Female thread, through: Usually drilled and tapped.	
Female thread, blind: Usually drilled and tapped.	
Male thread: Usually cut with a die, turned or rolled. Note use of undercut or groove and appearance of thread in sectioned view.	
Male & Female: e.g. a fastener in a tapped hole. Note here that the tapped hole is sectioned, the fastener is not.	

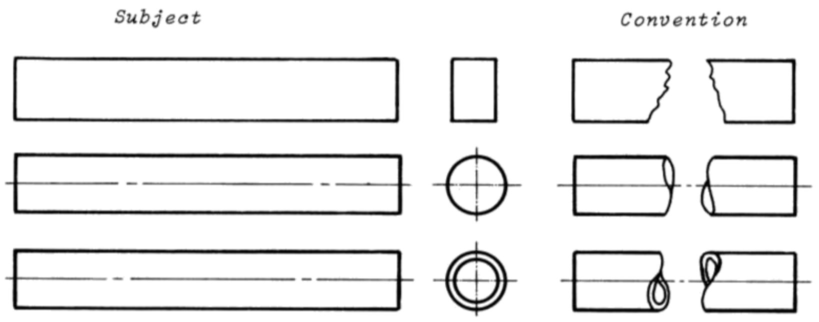
Springs: Compression:	
Tension:	

Knurling:	
Diamond.	
Straight.	

Bearings:	
Some examples of rolling element barings. Arrows indicate directions of load bearing. Deep groove (near). Angular contact (far).	
Roller (near). Taper roller (far).	
Thrust (near). Standard drawing representation of a bearing.	 

Long components:

Rectangular bar:

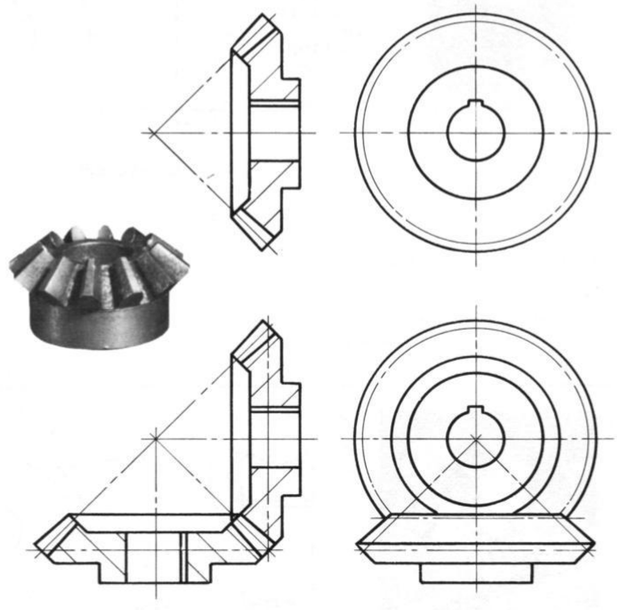


Round bar:

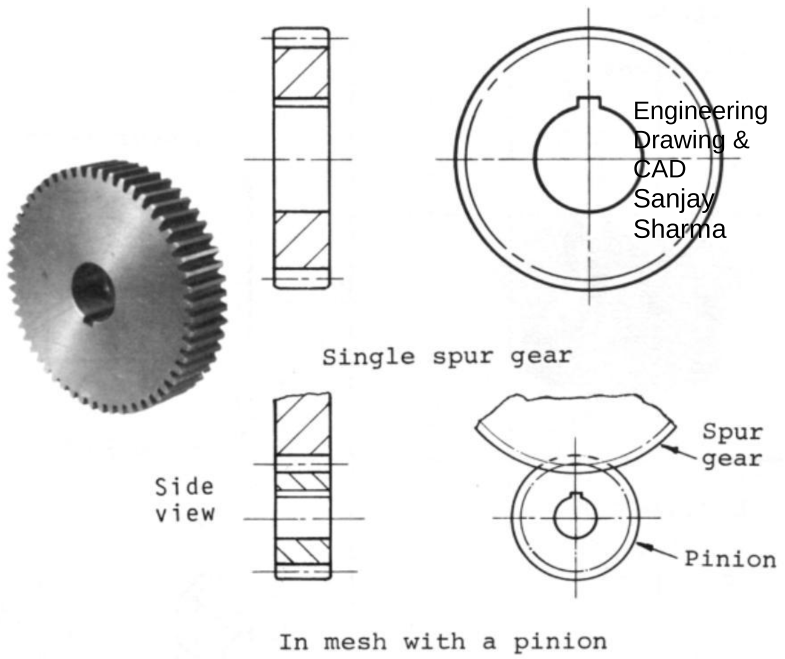
Round tube:

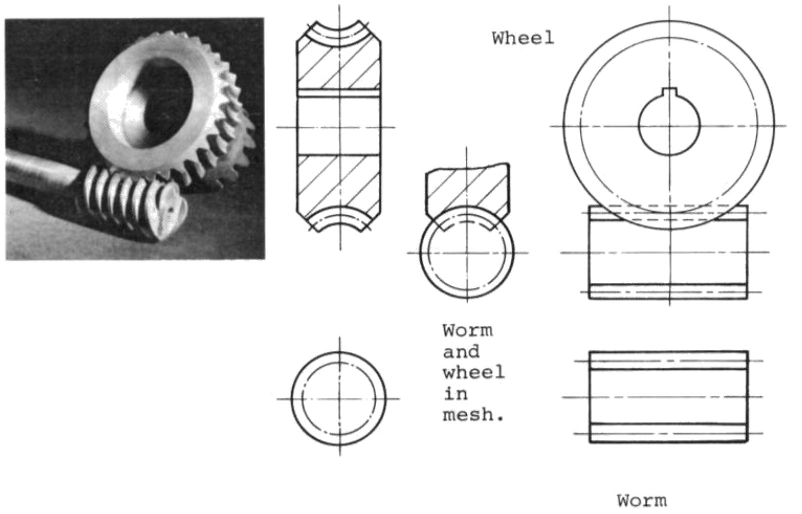
Gears:

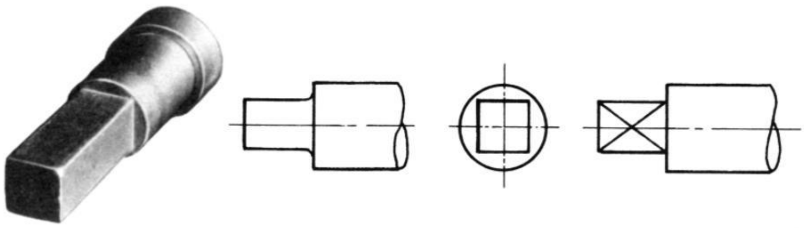
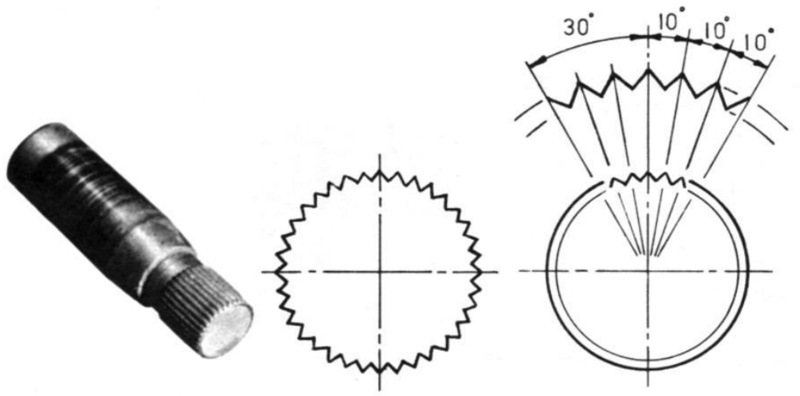
Bevel:

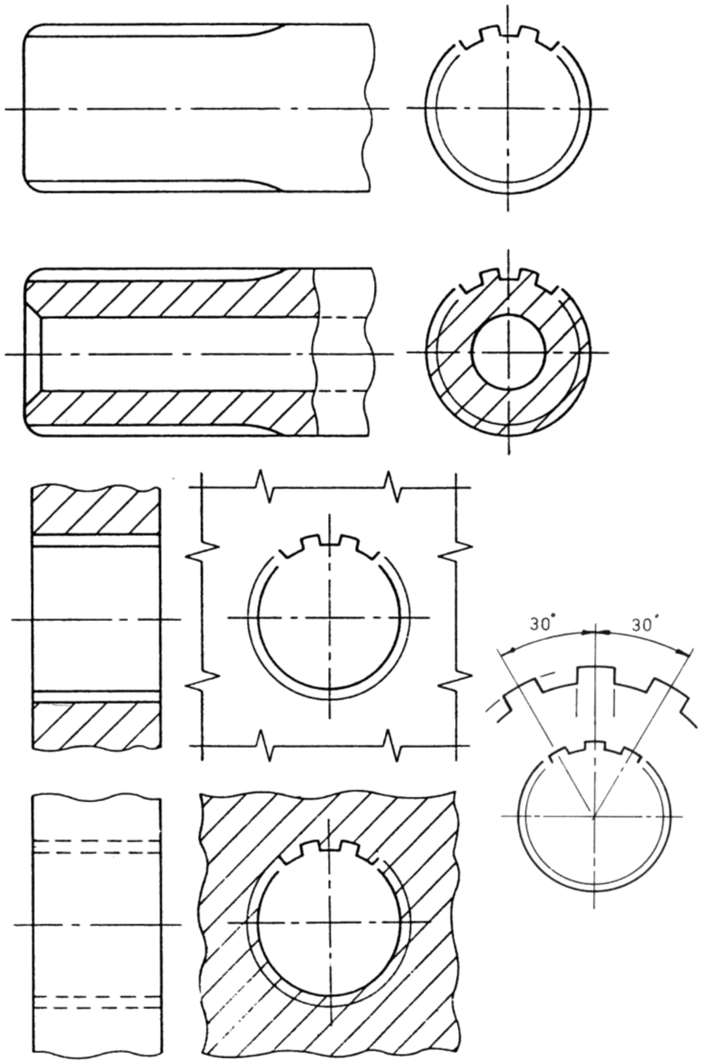


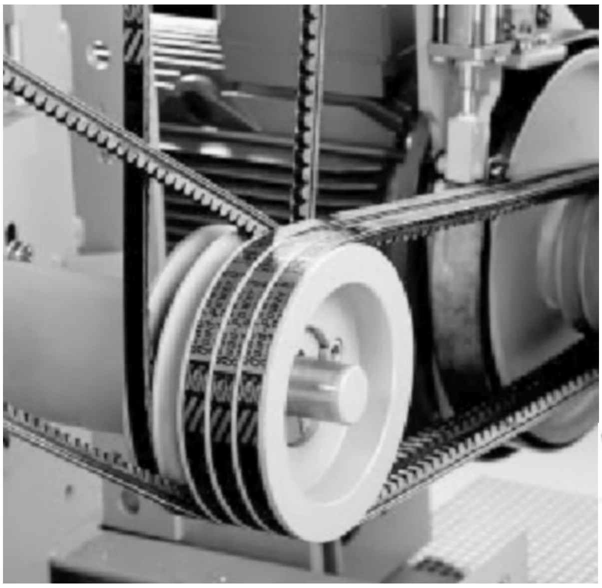
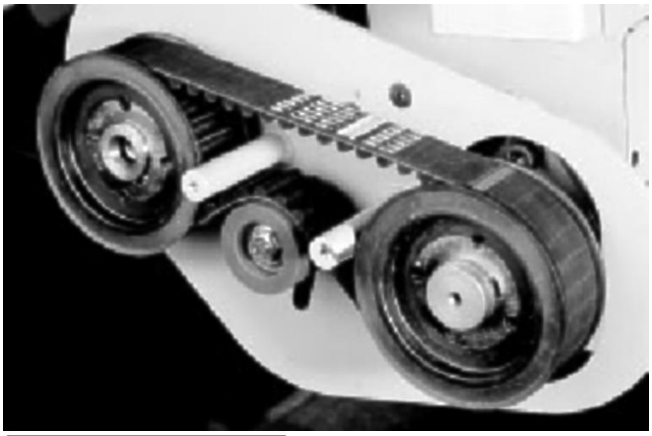
Spur:



Worm & wheel:	

Shaft ends:	
Square: Frequently used for hand driven adjustments with removable handles, such as those found on machine tools, etc.	
Serrations: Often used for push fit components such as plastic fans or pulleys, or levers such as motorcycle gear shifters.	

Splines: Usually used for transmitting rotational torque and allowing an axial 'sliding' movement. Examples can be found on automotive drive shafts. The figures opposite show splined shafts and housings in sectioned and non-sectioned views.	

Belt drives:	
V belt drives: Used for transmission of rotary power, good for space restricted applications. V-belts grip on the sides of the V. Often found on automotive engines to drive alternators and water pumps, or on pillar drills, and other industrial drives.	 
Timing or synchronous drives: Used for transmission of rotary power, as are v-belts, and, because of the toothed design (no slip) they are used for timed (synchronised) drives, where relative rotational positions have to be controlled. Some type of tensioning system is usually required. These drives are often found on camshaft drives on modern automotive engines, replacing chains.	 

2.3.5 Abbreviations of terms frequently used on drawings.

Abbreviations are used on drawings to save time and space. Most of these conform to BS 8888. They are the same singular or plural, full stops are only used where word may be confusing.

A/C	Across corners
A/F	Across flats
HEX HD	Hexagon head
ASSY	Assembly
CRS	Centers
CL	Center line
CHAM	Chamfer
CH HD	Cheese head
CSK	Countersunk
CBORE	Counterbore
CYL	Cylinder or cylindrical
DIA	Diameter (in a note)
Ø	Diameter (preceding a dimension)
R	Radius (preceding a dimension, capital only)
RAD	Radius (in a note)
DRG	Drawing
FIG.	Figure
LH	Left hand
LG	Long
MATL	Material
NO.	Number
PATT NO.	Pattern number
PCD	Pitch circle diameter
I/D	Inside diameter
O/D	Outside diameter
RH	Right hand
RD HD	Round head
SCR	Screwed
SPEC	Specification
SPHERE	Spherical
SFACE	Spotface
SQ	Square (in a note)
TYP	Typical or typically
THK	Thick

□	Square (preceding a dimension)
STD	Standard
UCUT	Undercut
M/CD	Machined
mm	Millimeter
NTS	Not to scale
RPM	Revolutions per minute
SWG	Standard wire gauge
TPI	Teeth per inch

2.4 Sections.

To show the inside details of a component it is imagined to be cut or sectioned along a plane, the cutting plane. Cutting planes are designated with capital letters, such as A-A in Figure 2.4a.

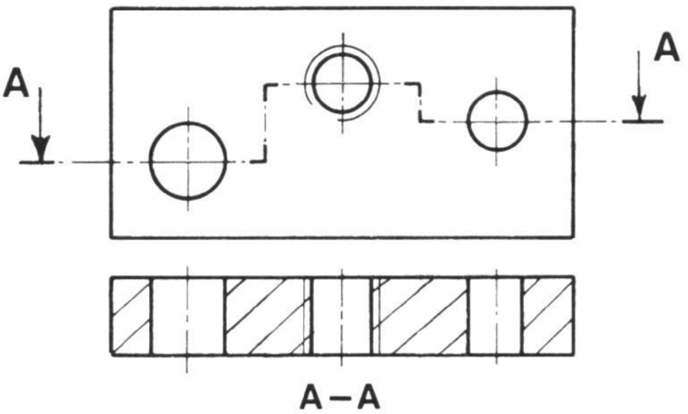
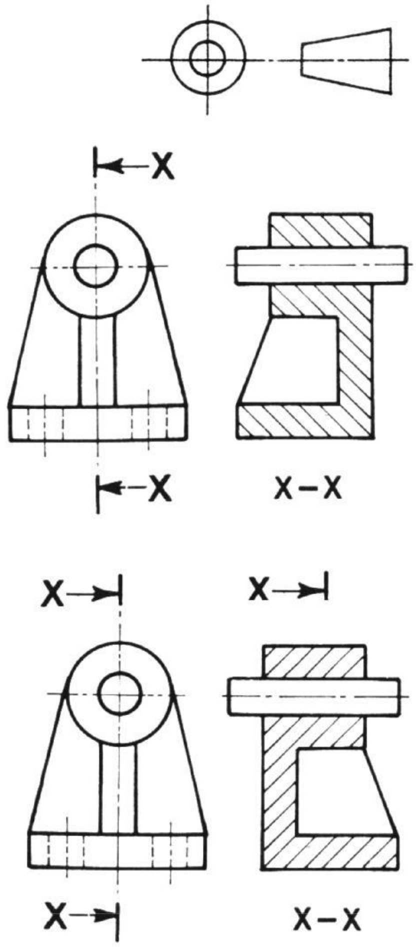


Figure 2.4a.

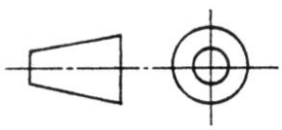


The side of the plane nearest the viewer is removed and the remaining details are shown as a sectional view, as demonstrated with section X-X in Figure 2.4b. The arrows indicate the direction to view the component when defining the sectioned view. Note that First or Third angle orthographic projection systems are still used and are indicated by use of the appropriate symbols.

Sectional views are produced to:

- clarify details
- show internal features clearly
- reduce number of hidden detail lines required
- aid dimensioning
- show cross-section shape
- clarify an assembly

Figure 2.4b.



Surfaces cut by the cutting plane are usually hatched at an appropriate angle, say 45° with a density of lines in proportion with the component.

Symmetrical parts can be shown in half sections. Part or 'broken out' sections can be used.

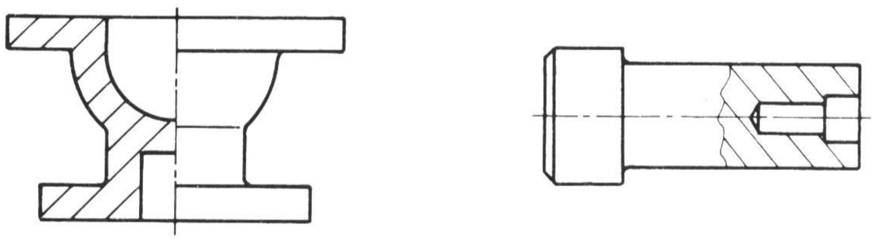


Figure 2.4c. Half section and a part or 'broken out' section.

Revolved sections are useful when clarifying local cross-section shapes as shown in Figure 2.4d.

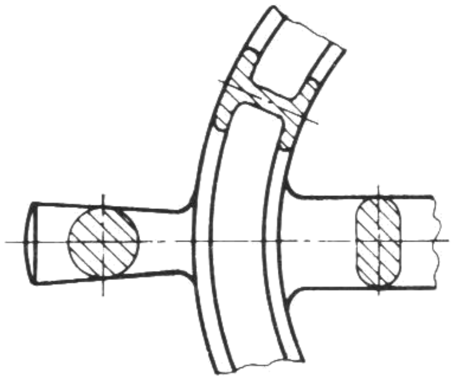


Figure 2.4d.

There are some exceptions to the general rules of sectioning:

- Webs, see Figure 2.4e.
- Shafts, rods, spindles, see Figure 2.4f.
- Bolts, nuts and thin washers.
- Rivets, dowels, pins and cotters.

These parts would not be shown as sections if their center lines lie on the cutting plane.

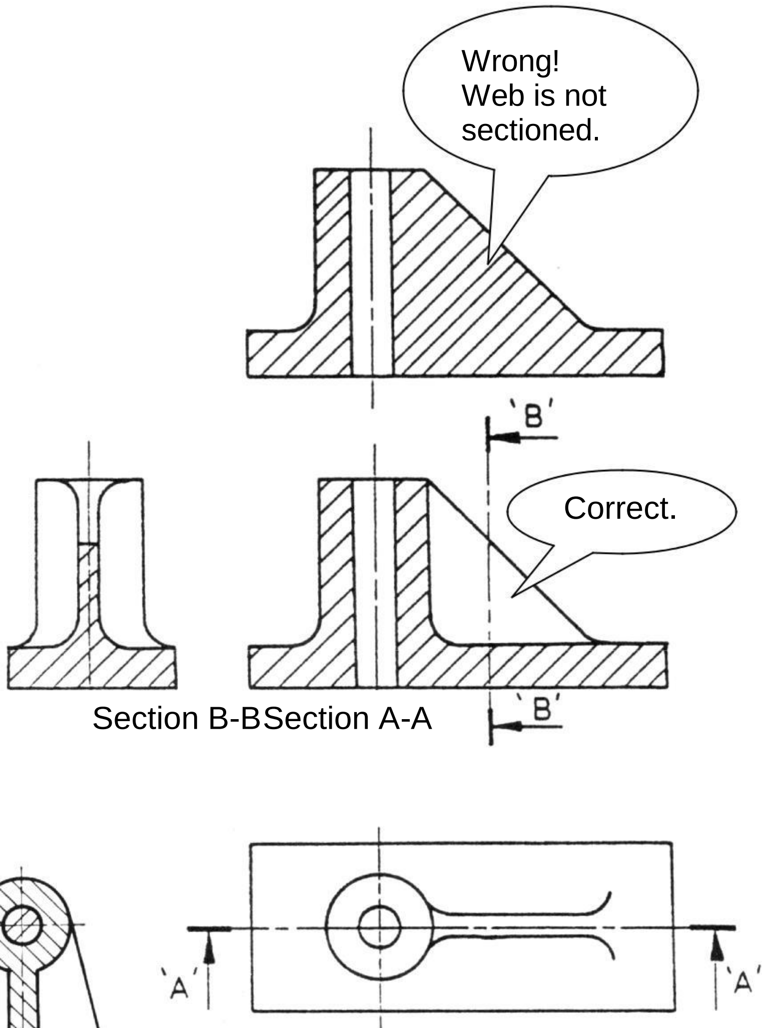
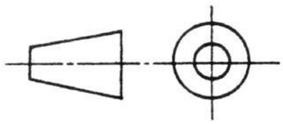


Figure 2.4e. Web sections.



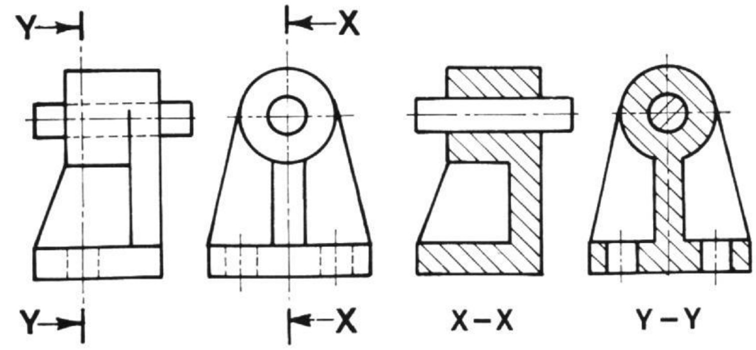
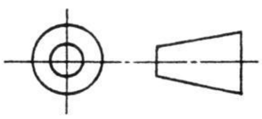
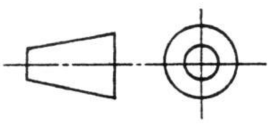


Figure 2.4f.

Section X-X



Section Y-Y



It may be appropriate to use Removed sections, for webs, beams or arms, as shown in Figure 2.4g below. Note the absence of viewing arrows.

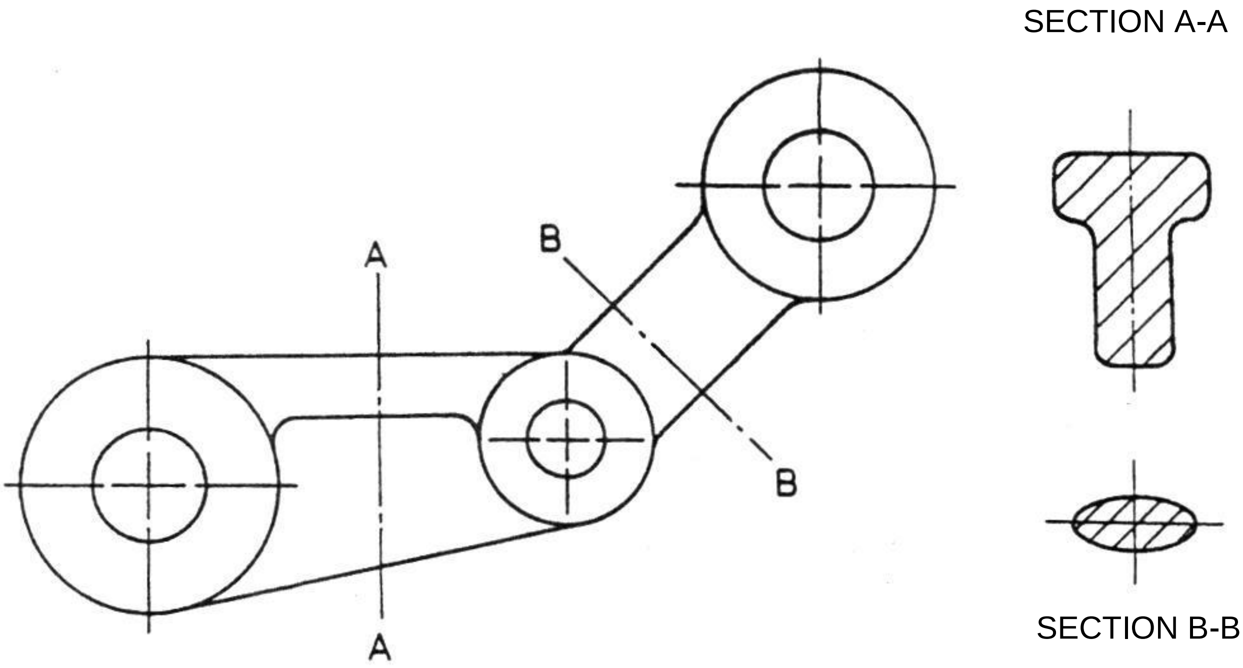


Figure 2.4g. Removed sections.

Assemblies can be greatly clarified using sections. See the example below in Figure xx.

Note:

- Revolved sections.
- Part sections.
- Different hatching directions and spacings.
- Un-sectioned components such as shafts, keys, nuts etc.

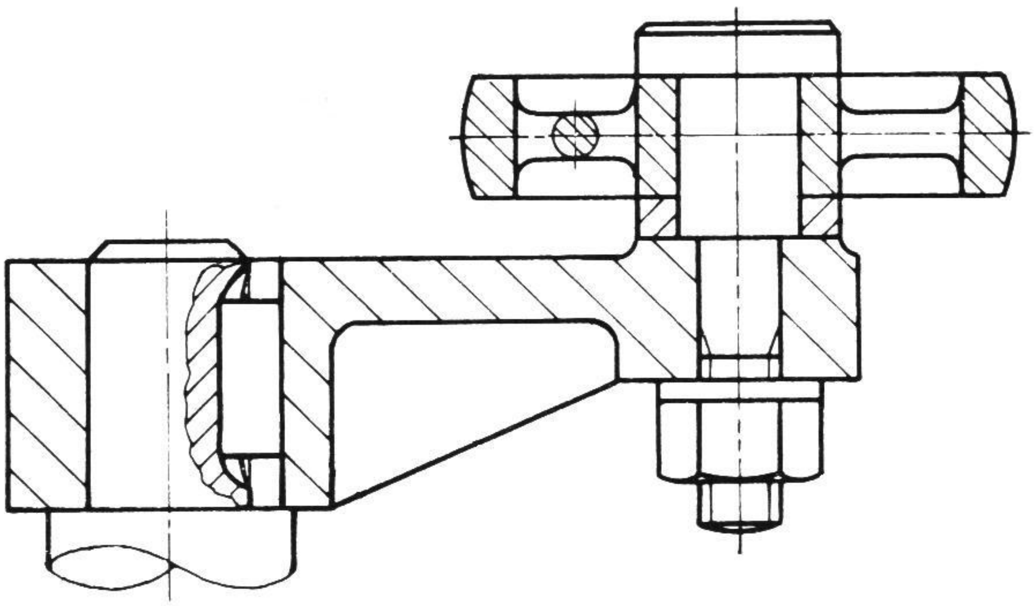


Figure 2.4h. An assembly drawing view, clarified using sections.

2.5 Dimensions.

A drawing from which a component is to be manufactured must communicate all of the required information by:

- describing the form or shape of the component, usually by using orthographic and sometimes pictorial views...
- giving actual dimensions of all features of the geometry...
- giving information about any special manufacturing processes and materials required.

The design engineer should have a good understanding of projection methods, dimensioning methods and the manufacturing methods to be used.

This section introduces some basic guidelines and examples to help explain the general rules of dimensioning, based on BS 8888.

2.5.1 General rules.

- Standards and conventions should be followed.
- Dimensions should be placed on drawings so that they may be easily read.
- The drawing must include the minimum number of dimensions required to accurately manufacture th edesign.
- A dimension should not be stated more than once, unless it aids communication.
- It should not be necessary for the operator manufacturing the component to have to calculate any dimensions.

2.5.2 Types of dimension.

Types of dimensioning can be broadly classified as:

- Size dimensions. Used to describe heights, widths, diameters, etc.
- Location dimensions. Used to place various features of a component relative to each other, such as a hole centre line to a reference surface.
- Mating dimensions. Used for parts that fit together requiring a certain degree of accuracy.

2.5.3 Dimensioning conventions.

2.5.3.1. General.

Observe the dimensioning features shown for the plate in Figure 2.5a below. Note:

- parallel dimensions, indicating the size of the plate
- edges A and B are being used as the reference edges
- minimum number of dimensions required are specified
- use of description of 'plate 3mm thick', so that no side view is required
- evenly spaced dimension lines

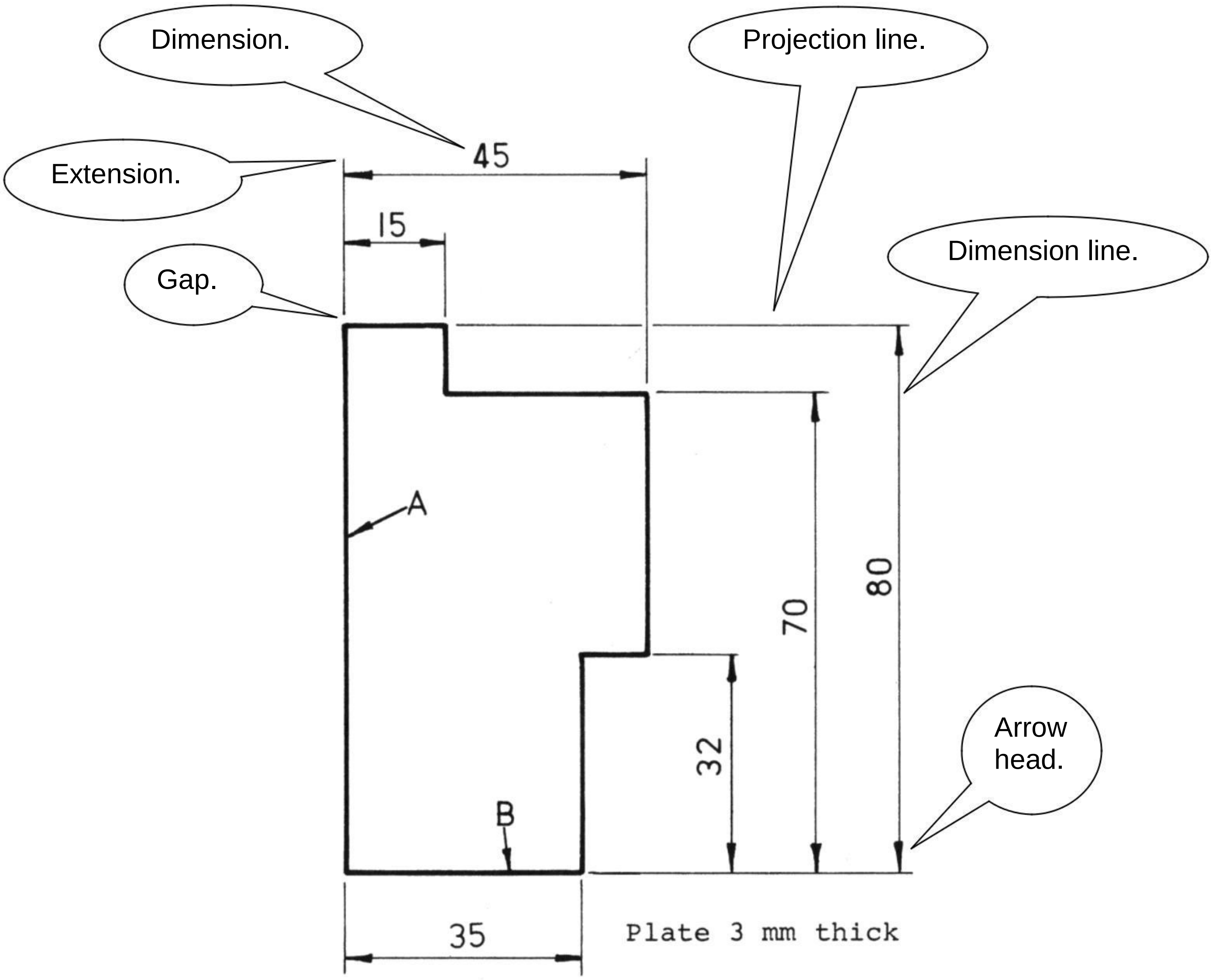
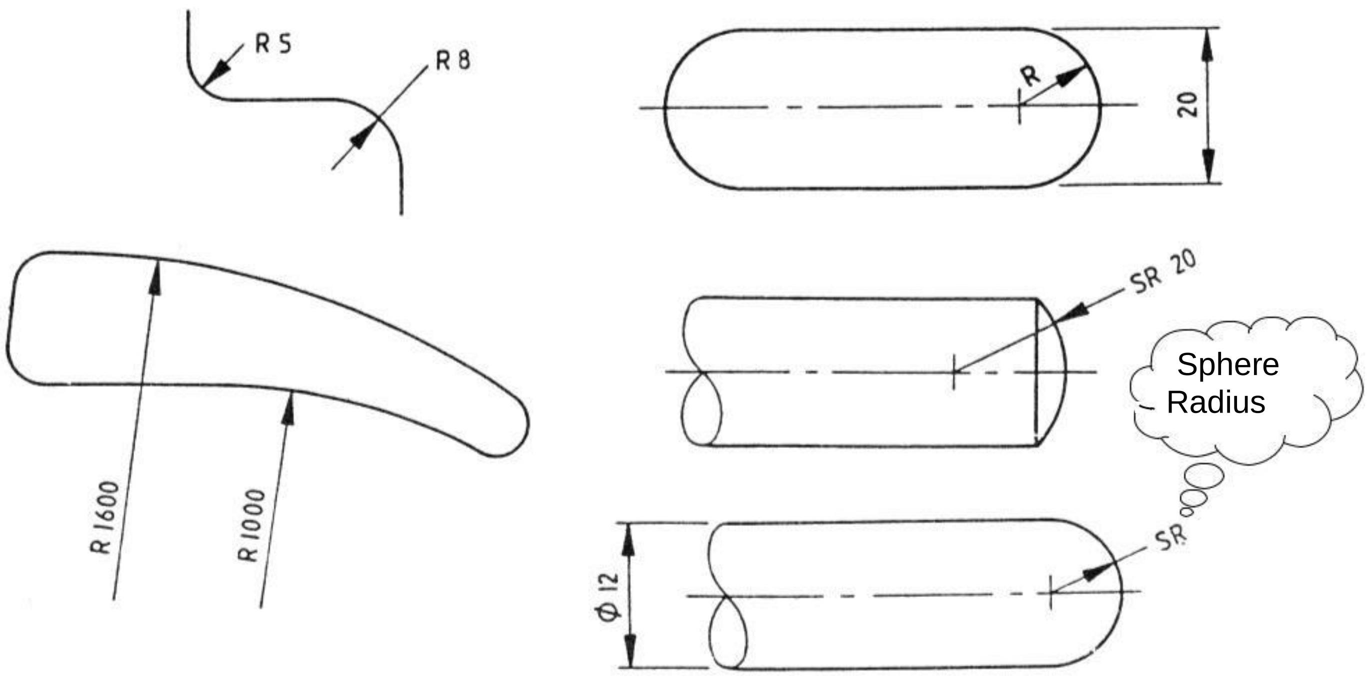


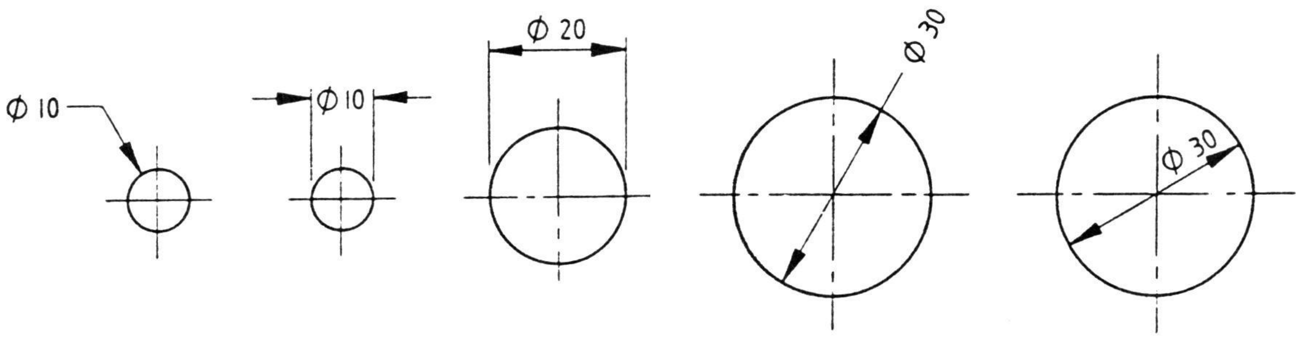
Figure 2.5a.

2.5.3.2 Radii:

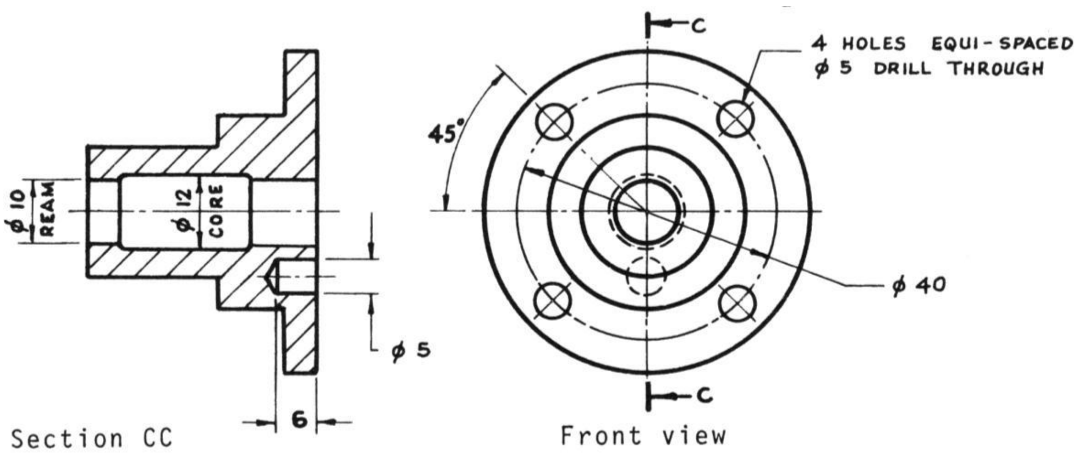


2.5.3.3 Circles:

Circles on engineering drawings are usually either spheres, holes or cylinders of some description. The dimension refers to the diameter, and the diameter symbol is ⌀.



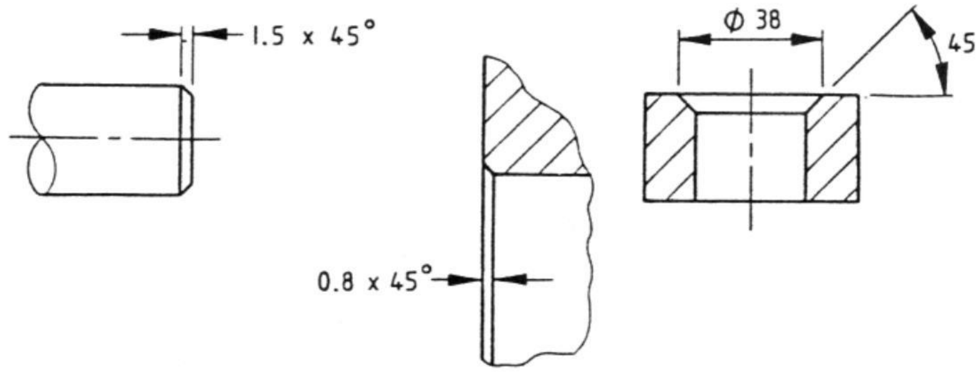
Holes equally spaced on a pitch circle can be dimensioned as shown below.



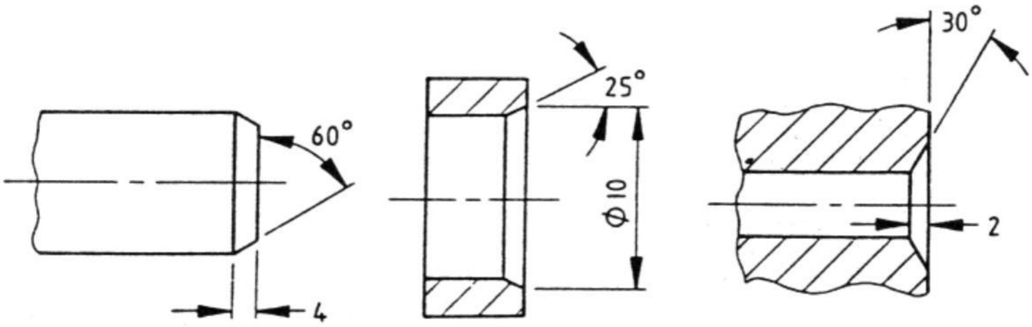
The ⌀40 dimension can also be refereed to as the PCD or Pitch Circle Diameter.

2.5.3.4 Chamfers, countersinks and counterbores:

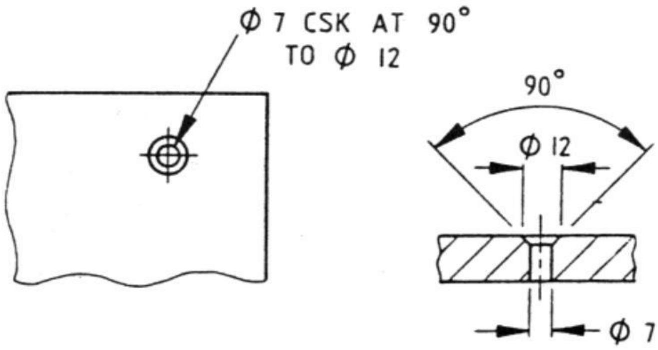
Chamfer at 45º:



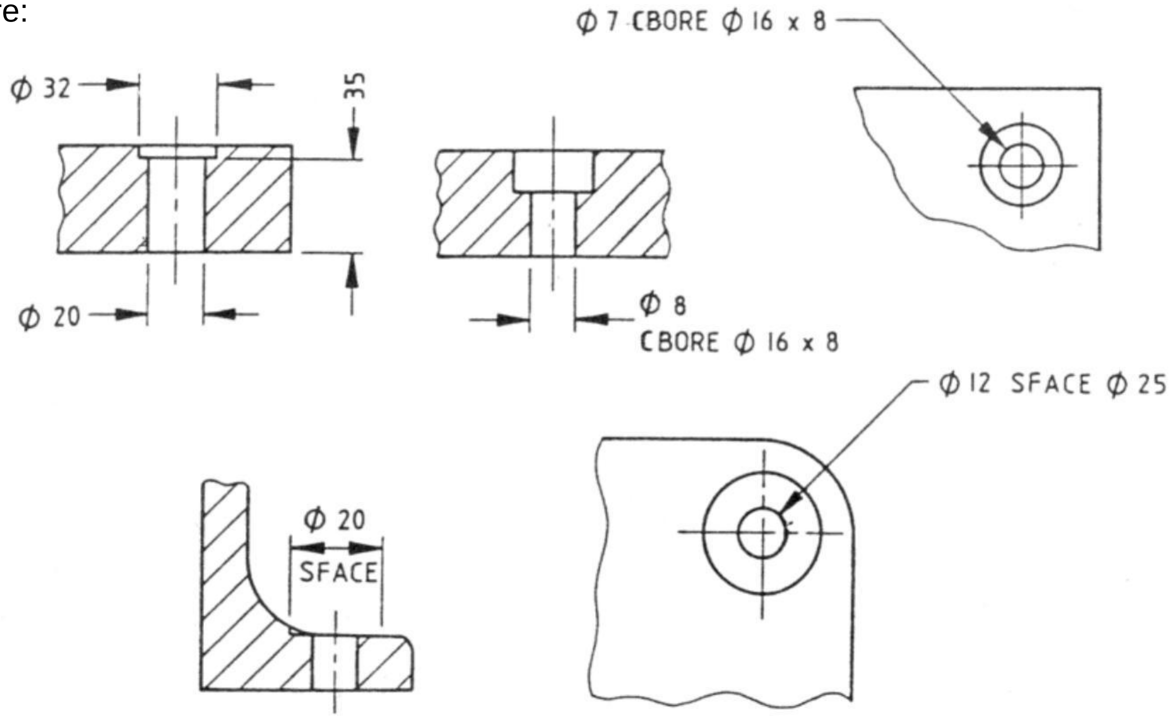
Chamfer at angles other than 45º:



Countersink:



Counterbore:



2.5.3.5 Location dimensions:

Due to the nature of manufacturing, actual finished dimensions of manufactured components are never perfect. This has to be considered when dimensioning features that require accurate location. Inorder to enable accyrate measurement, such a feature is usually dimensioned from a reliable reference such as a machined surface. This reference is refered to as a Datum.

1) 2) 3)

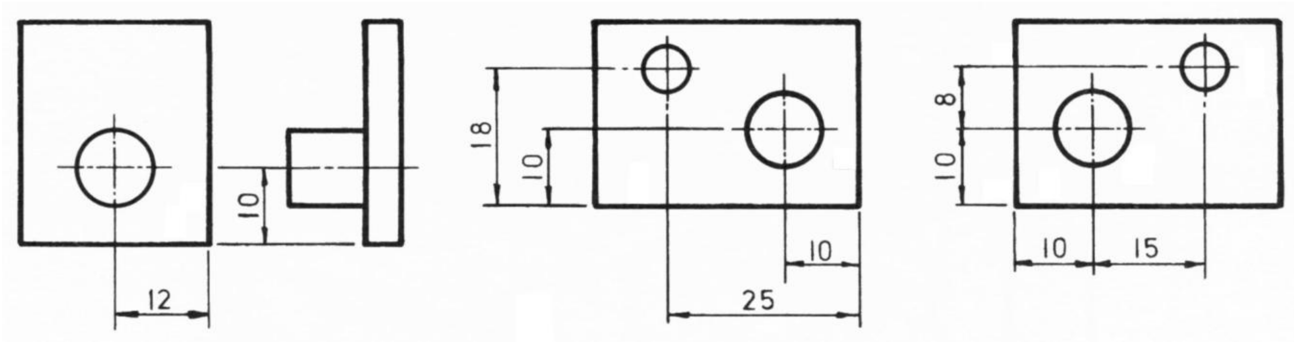
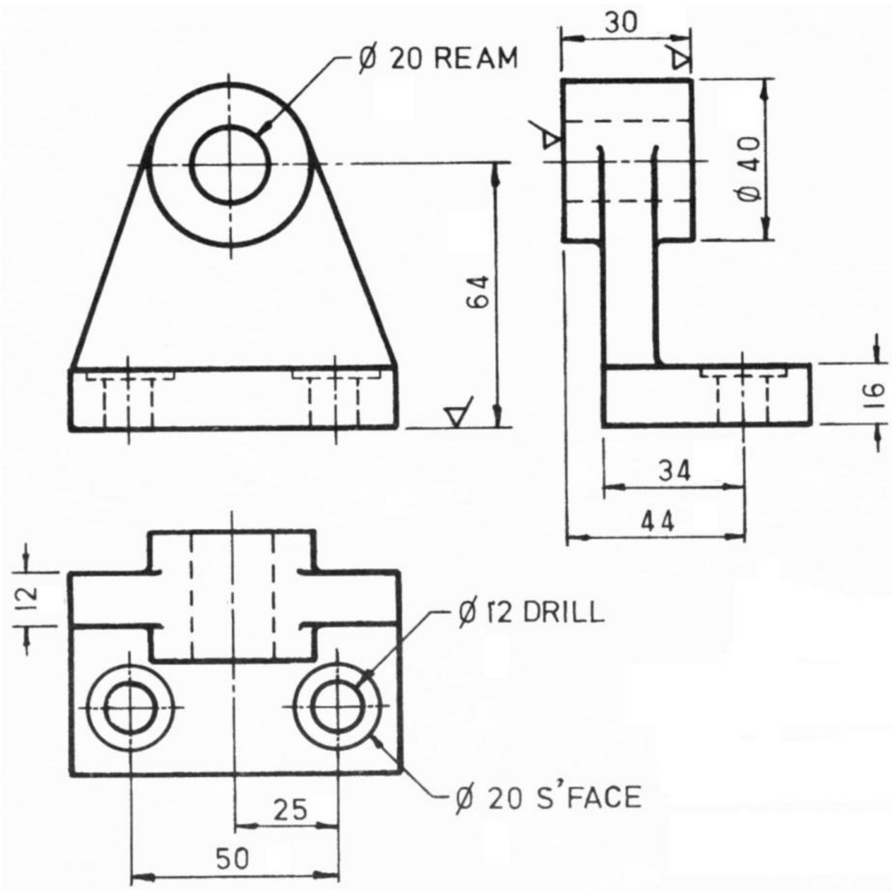


Figure 2.5b.

Figure 2.5b shows: 1) A spigot located from two reference edges.
2) Two holes located from two reference edges.
3) The large hole located from two reference edges and the small hole from the center of the large hole.

The simple bearing bracket casting below shows both size and location dimensions.

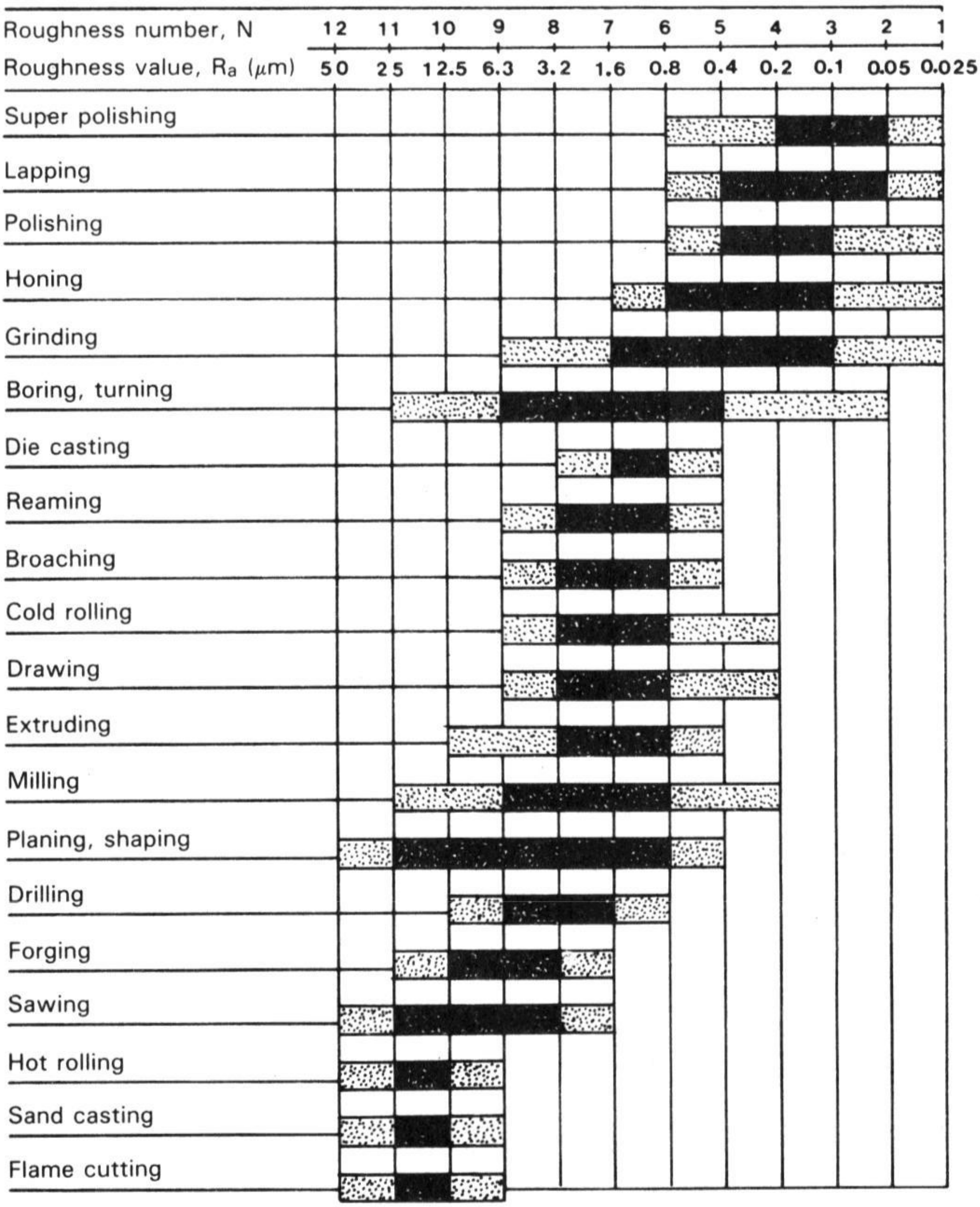


Note that machined surfaces are specified using this British Standard machining symbol:

2.5.3.5 Surface finish:

Surface textures resulting from manufacturing processes consist of many complex peaks and valleys varying in height and spacing. The Roughness value of a surface is a measure of this surface quality. The table below gives some nominal values of roughness resulting from various common manufacturing processes.

If a particular surface finish is required you give clear instructions on the drawing using the British Standard machining symbol.



2.6 Tolerances, limits and fits.

In order to ensure that assemblies function properly their component parts must fit together in a predictable way. As mentioned in section 2.5, no component can be manufactured to an exact size, so the designer has to decide on appropriate upper and lower limits for each dimension.

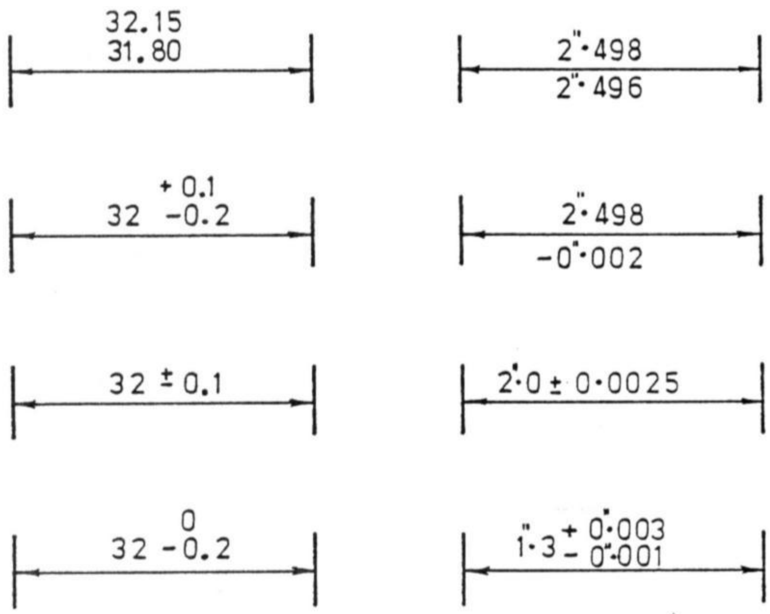
> Accurately toleranced dimensioned features usually take much more time to manufacture correctly and therefore can increase production costs significantly. Good engineering practise finds the optimum balance between required accuracy for the function of the component and minimum cost of manufacture.

2.6.1 Dimension tolerances.

If a dimension is specified, in millimeters, as 10 ± 0.02, the part will be acceptable if the dimension is manufactured to an actual size between 9.98 and 10.02 mm. Below are some examples of ways of defining such limits for a linear dimension.

Maximum and minimum limits of size:

Nominal size with limits of tolerance:



To give you a feel for the magnitude of decimal values in mm, consider these facts:

The thickness of the paper this page is printed on is approximately 0.100 mm.

Average human hair thickness is approximately 0.070 mm.

The human eye cannot resolve a gap between two points smaller than about 0.020mm, at a 20cm distance.

If you raise the temperature of a 100mm long block of steel by 10ºC it will increase in length by approximately 0.020mm.

2.6.2 General tolerancing.

General tolerance notes apply tolerances to all unspecified dimensions on a drawing. They can save time and help to make a drawing less cluttered. Examples are shown below.

TOLERANCE EXCEPT WHERE OTHERWISE STATED ± 0.5

TOLERANCES EXCEPT WHERE OTHERWISE STATED

SIZE		TOLERANCE
—	UP TO X	± 0.1
OVER X	UP TO XX	± 0.25
OVER XX	UP TO XXX	± 1
OVER XXX	—	± 2
	ON ANGLES	± 0.5°

TOLERANCE ON CAST THICKNESSES ± 1 %

FOR TOLERANCES ON FORGING DIMENSIONS SEE BS 4114

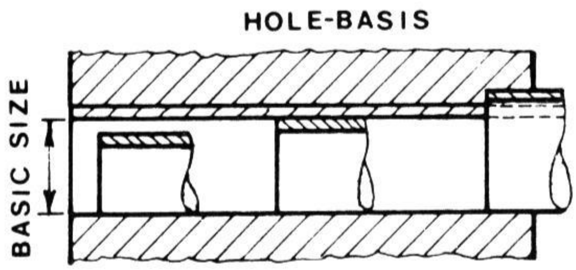
Some examples of general tolerance notes.

2.6.3 Limits and fits for shafts and holes.

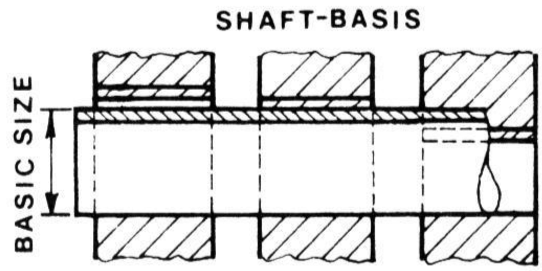
2.6.3.1 Basic size and shaft/hole tolerancing systems.

The basic size or nominal size is the size of shaft or hole that the designer specifies before applying the limits to it. There are two systems used for specifying shaft/hole tolerances:

Basic hole system: Starts with the basic hole size and adjusts shaft size to fit.



Basic shaft system: Starts with the basic shaft size and adjusts hole size to fit.



Because holes are usually made with standard tools such as drills and reamers, etc, the basic hole system tends to be preferred and will therefore be used here

2.6.3.2 Fit.

The fit represents the tightness or looseness resulting from the application of tolerances to mating parts, e.g. shafts and holes. Fits are generally classified as one of the following:

Clearance fit: Assemble/disassemble by hand.
Creates running & sliding assemblies, ranging from loose low cost, to free-running high temperature change applications and accurate minimal play locations.

Transition fit: Assembly usually requires press tooling or mechanical assistance of some kind.
Creates close accuracy with little or no interference.

Interference fit: Parts need to be forced or shrunk fitted together.
Creates permanent assemblies that retain and locate themselves.

2.6.3.3 ISO limits and fits.

Fits have been standradised and can be taken directly from those tabulated in the BS 4500 standard, 'ISO limits and fits.'

The BS 4500 standard refers to tolerance symbols made up with a letter followed by a number. The BS Data Sheet BS 4500A, as shown on the following two pages, shows a range of fits derived, using the hole basis, from the following tolerances:

Holes: H11 H9 H8 H7

Shafts: c11 d10 e9 f7 g6 k6 n6 p6 s6

Remember:

- Capital letters always refer to holes, lower case always refer to shafts.
- The greater the number the greater or wider the tolerances.

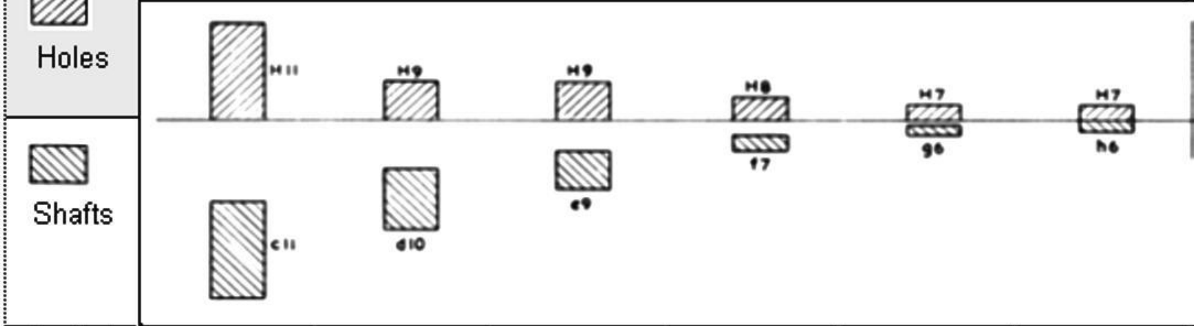
The selection of a pair of these tolerances will give you the fit. The number of possible combinations is huge. BS 4500 helps to standardise this and offers a range of fits suitable for most engineering applications.

Examine an extract from the BS 4500 data sheet on page 4 & 5 and you will observe the general class of fit specified on the top row. A more detailed description of the fit is given on the bottom row.

See the table in section 2.6.4 for guidance on the selection of types of fit.

Selected ISO Fits - Hole Basis. Extract from BS 4500, data Sheet 4500A.

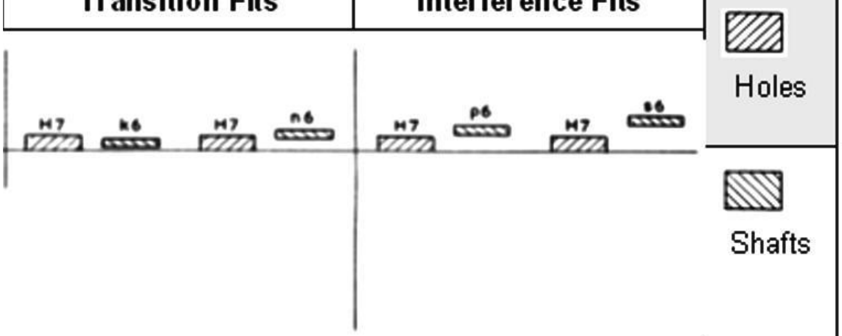
Clearance Fits



Nominal Sizes		Tolerance		Tolerance		Tolerance		Tolerance		Tolerance		Tolerance	
Over	To	H11	c11	H9	d10	H9	e9	H8	f7	H7	g6	H7	h6
mm	mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm
-	3	60 0	-60 -120	25 0	-20 -60	25 0	-14 -39	14 0	-6 -16	10 0	-2 -8	10 0	-6 0
3	6	75 0	-70 -145	30 0	-30 -78	30 0	-20 -50	18 0	-10 -22	12 0	-4 -12	12 0	-8 0
6	10	90 0	-80 -170	36 0	-40 -98	36 0	-25 -61	22 0	-13 -28	15 0	-5 -14	15 0	-9 0
10	18	110 0	-95 -205	43 0	-50 -120	43 0	-32 -75	27 0	-16 -34	18 0	-6 -17	18 0	-11 0
18	30	130 0	-110 -240	52 0	-65 -149	52 0	-40 -92	33 0	-20 -41	21 0	-7 -20	21 0	-13 0
30	40	160 0	-120 -280	62	-80	62	-50	39	-25	25	-9	25	-16
40	50	160 0	-130 -290	0	-180	0	-112	0	-50	0	-25	0	0
50	65	190 0	-140 -330	74	-100	74	-60	46	-30	30	-10	30	-19
65	80	190 0	-150 -340	0	-220	0	-134	0	-60	0	-29	0	0
80	100	220 0	-170 -390	87	-120	87	-72	54	-36	35	-12	35	-22
100	120	220 0	-180 -400	0	-260	0	-159	0	-71	0	-34	0	0
120	140	250 0	-200 -450	100	-145	100	-84	63	-43	40	-14	40	-25
140	160	250 0	-210 -460										
160	180	250 0	-230 -480	0	-305	0	-185	0	-83	0	-39	0	0
180	200	290 0	-240 -530	115	-170	115	-100	72	-50	46	-15	46	-29
200	225	290 0	-260 -550										
225	250	290 0	-280 -570	0	-355	0	-215	0	-96	0	-44	0	0
250	280	320 0	-300 -620	130	-190	130	-110	81	-56	52	-17	52	-32
280	315	320 0	-330 -650	0	-400	0	-240	0	-108	0	-49	0	0
315	355	360 0	-360 -720	140	-210	140	-125	89	-62	57	-18	57	-36
355	400	360 0	-400 -760	0	-440	0	-265	0	-119	0	-54	0	0
400	450	400 0	-440 -840	155	-230	155	-135	97	-68	63	-20	63	-40
450	500	400 0	-480 -880	0	-480	0	-290	0	-131	0	-60	0	0
		Slack Fit		Loose Fit		Easy Fit		Normal Fit		Close Fit		Slide Fit	

Selected ISO Fits - Hole Basis. Extract from BS 4500, data Sheet4500A.

Transition Fits				Interference Fits					



Tolerance		Tolerance		Tolerance		Tolerance			
H7	k6	H7	r6	H7	p6	H7	s6	Over	To
0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	0.001 mm	mm	mm
10 0	6 0	10 0	10 4	10 0	12 6	10 0	20 14	-	3
12 0	9 1	12 0	16 8	12 0	20 12	12 0	27 19	3	6
15 0	10 1	15 0	19 10	15 0	24 15	15 0	32 23	6	10
18 0	12 1	18 0	23 12	18 0	29 18	18 0	39 28	10	18
21 0	15 2	21 0	28 15	21 0	35 22	21 0	48 35	18	30
25	18	25	33	25	42	25	59	30	40
0	2	0	17	0	26	0	7	40	50
30	21	30	39	30	51	30 0	72 53	50	65
0	2	0	20	0	32	30 0	78 59	65	80
35	25	35	45	35	59	35 0	93 91	80	100
0	3	0	23	0	37	35 0	101 79	100	120
40	28	40	52	40	68	40 0	117 92	120	140
						40 0	125 100	140	160
0	3	0	27	0	43	40 0	133 108	160	180
46	33	46	60	46	79	46 0	151 122	180	200
						46 0	159 130	200	225
0	4	0	31	0	50	46 0	169 140	225	250
52	36	52	66	52	88	52 0	190 158	250	280
0	4	0	34	0	56	52 0	202 170	280	315
57	40	57	73	57	98	57 0	226 190	315	355
0	4	0	37	0	62	57 0	244 208	355	400
63	45	63	80	63	108	63 0	272 232	400	450
0	5	0	40	0	68	63 0	292 252	450	500
Push Fit		Drive Fit		Press Fit		Force Fit			

2.6.3.4 ISO limits and fits, determining working limits.

Consider an example of a shaft and a housing used in a linkage:

Type of fit:	'Normal' clearance fit.
Basic or Nominal size:	∅40mm

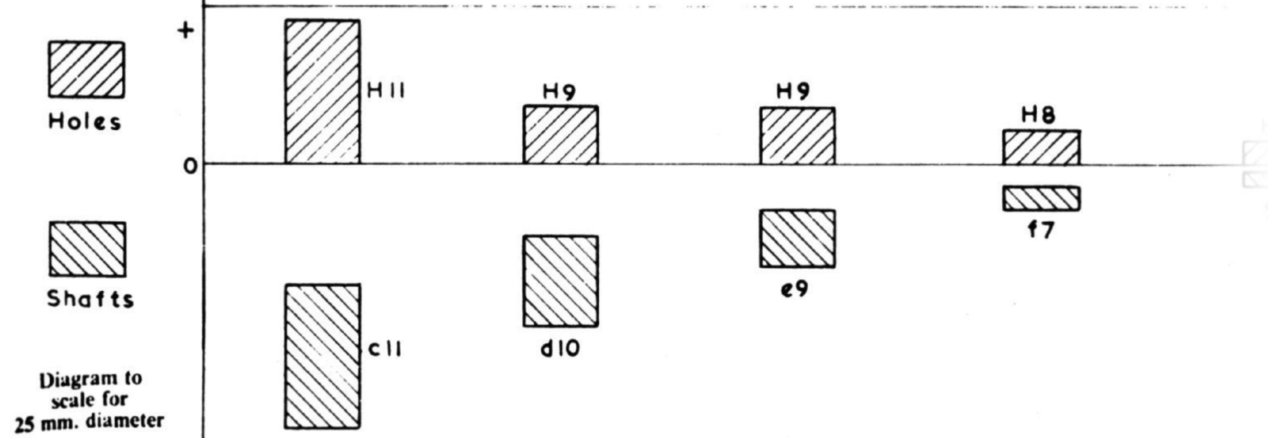
We will determine the actual working limits, the range of allowable sizes, for the shaft and the hole in the housing.

Look along the bottom of the ISO Fits Data Sheet 4500A and locate 'Normal Fit'. We will use this pair of columns to extract our tolerances.

The tolerances indicated are:	1st column	H8	for the hole	(upper case H)
	2nd column	f7	for the shaft	(lower case f)

The actual tolerances depend upon the basic, or nominal, diameter as well as the class of fit. So, locate 40mm in the left hand Nominal Sizes column. Either the 30 - 40 or 40 - 50 range is acceptable in this case. Read across and note the tolerance values for the hole and the shaft, as shown below.

Clearance fits



Nominal sizes		Tolerance		Tolerance		Tolerance		Tolerance		Tol...
Over	To	H11	c11	H9	d10	H9	e9	H8	f7	H7
mm	mm	0·001 mm	0·001 mm	0·001 mm	0·001 mm	0·001 mm	0·001 mm	0·001 mm	0·001 mm	0·001 mm
—	3	+ 60 0	− 60 − 120	+ 25 0	− 20 − 60	+ 25 0	− 14 − 39	+ 14 0	− 6 − 16	+ 10 0
3	6	+ 75 0	− 70 − 145	+ 30 0	− 30 − 78	+ 30 0	− 20 − 50	+ 18 0	− 10 − 22	+ 12 0
6	10	+ 90 0	− 80 − 170	+ 36 0	− 40 − 98	+ 36 0	− 25 − 61	+ 22 0	− 13 − 28	+ 15 0
10	18	+ 110 0	− 95 − 205	+ 43 0	− 50 − 120	+ 43 0	− 32 − 75	+ 27 0	− 16 − 34	+ 18 0
18	30	+ 130 0	− 110 − 240	+ 52 0	− 65 − 149	+ 52 0	− 40 − 92	+ 33 0	− 20 − 41	+ 21 0
30	40	+ 160 0	− 120 − 280	+ 62 0	− 80 − 180	+ 62 0	− 50 − 112	+ 39 0	− 25 − 50	+ 25 0
40	50	+ 160 0	− 130 − 290							
50	65	+ 190 0	− 140 − 330	+ 74	− 100	+ 74	− 60			
400	450	0	[illegible]							
450	500	+ 400 0	− 480 − 880	+ 155 0	− 230 − 480	+ 155 0	− 135 − 290	0	− 131	0
		Slack fit		Loose fit		Easy fit		Normal fit		Cl...

For the hole diameter we have a tolerance of: +0.039mm -0.000mm

For the shaft diameter we have a tolerance of: -0.025mm -0.050mm

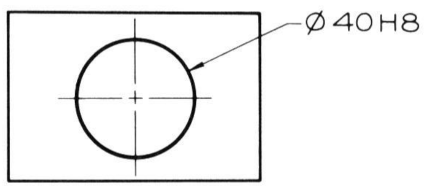
These tolerance values are simply added to the nominal size to obtain the actual allowable sizes.

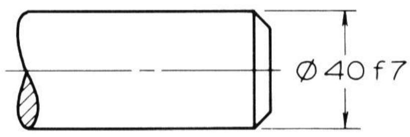
Note that this is a clearance fit. As long as the hole and shaft are manufactured within the specified tolerances the hole will always be either slightly oversize or spot on the nominal size and the shaft will always be slightly undersize. This ensures that there will always be a free clearance fit.

These tolerances may be expressed on a drawing in several ways:

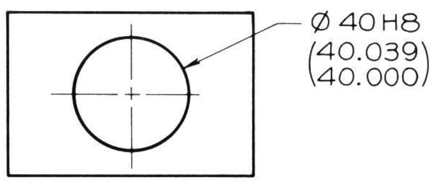
1) Simply as the nominal size with the tolerance class.

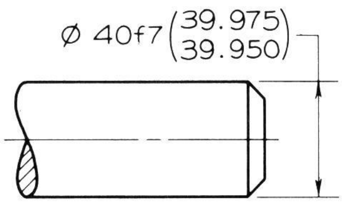
This is not always preferred as the machine operator has to calculate the working limits.



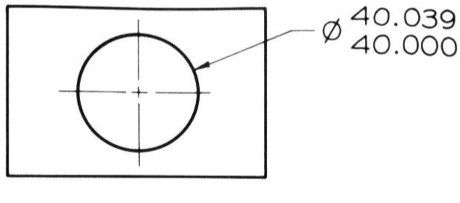


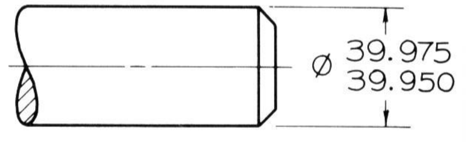
2) The nominal size with the tolerance class as above with the calculated working limits included.





3) The calculated working limits only.





2.6.4 Tabulated guide to types of ISO limits and fits.

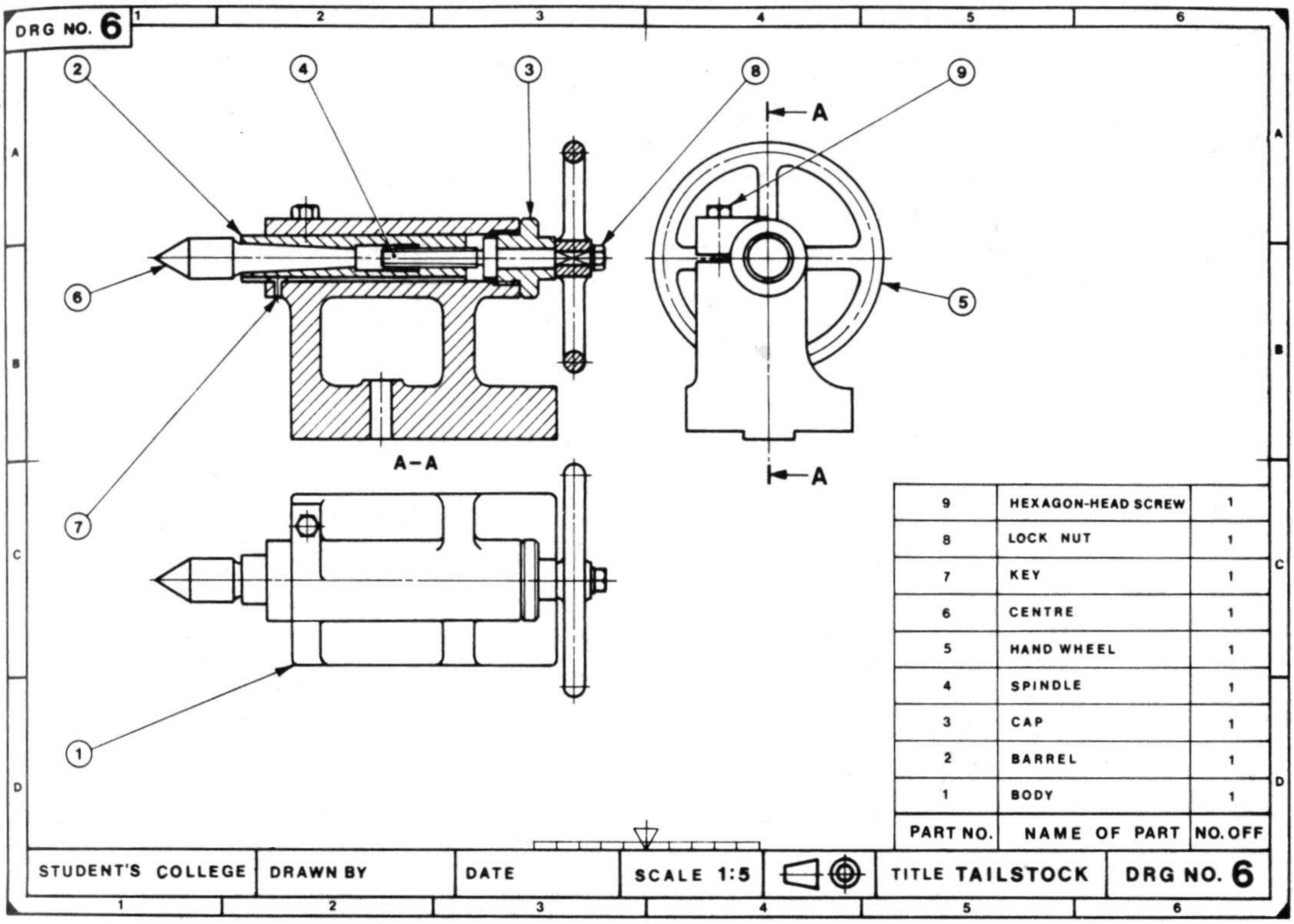
	ISO Symbol		
	Hole Basis	Shaft Basis	Description
Clearance fits (↓ More clearance)	H11/c11	C11/h11	**Loose running fit** for wide commercial tolerances or allowances on external members
	H9/d9	D9/h9	**Free running fit** not for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures
	H8/f7	F8/h7	**Close running fit** for running on accurate machines and for accurate location at moderate speeds and journal pressures
	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
	H7/k6	K7/h6	**Locational transisition fit** for accurate location; a compromise between clearance and interference
	H7/n6	N7/h6	**Locational transition fit** for more accurate location where greater interference is permissible
Interference fits (More interference ↓)	H7/p6*	P7/h6	**Locational interference fit** for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements
	H7/s6	S7/h6	**Medium drive fit** for ordinary steel parts or shrink fits on light sections; the tightest fit usable with cast iron.
	H7/u6	U7/h6	**Force fit** suitable for parts that can be highly stressed or for shrink fits where the heavy pressing forces required are impractical

2.7 Assembly drawings.

Assembly drawings can be used to:

- Name, identify, describe and quantify all of the components making up the assembly.
- Clearly show how all of the components fit together.
- Indicate all of the required fasteners.
- Record any special assembly instructions.
- Record any other relevant information.

Here is an example:



Note the use of sections, item numbers neatly layed out and the parts list.

2.8 Drawing checklist.

It is easy to accidentally omit various items when creating engineering detail drawings. Before passing on your work it is recommended that you work through the checklist below for each drawing:

The general drawing:

1 Do projections conform to the relevant conventions, usually 1st or 3rd angle?
2 Have you used the minimum number of views necessary to accurately show the information required?
3 Are the views laid out in appropriate positions relative to the size of paper?
4Has the title box been completed, particularly: Drawn by
Name of component
Date
Projection (1st or 3rd angle)
Paper size
Scale
5 If required, has the material been specified?

The geometry details:

7 Check to make sure that there are sufficient dimensions to manufacture the component. Check that positions and sizes of any features, such as holes, are clearly dimensioned.
8 No dimension should appear more than once on the drawing, do any?
9 Have the dimensions been laid out in consistent and clear positions, so that they are easy to read.
10 Have all of the dimension lines been constructed with correct extension lines and gaps?
11 Are the arrow heads all in the same style and the same size?
12 Have dimensions relating to a particular feature, such as a hole, been grouped together on one view, if possible?
13 Have appropriate line styles and line weights been used?
14 Have any surface finish requirements been specified?
15 Have any explicit tolerance requirements been specified?
16 Have any required center lines, break lines, etc. been used?
17 Have any required general notes been added, such as additional general tolerances, finish specifications or specification of special manufacturing processes?
18 If sections have been used do they conform to drawing conventions?

3 3D CAD & Solid Modelling.

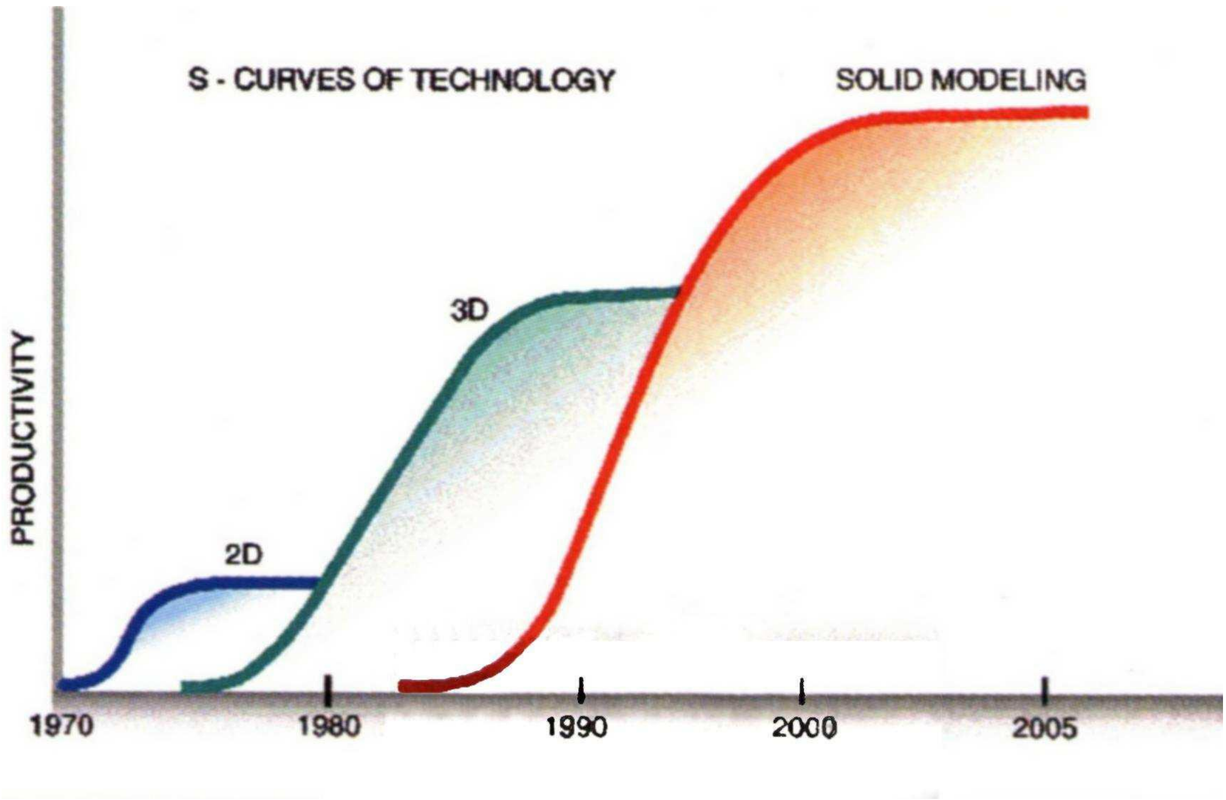
3.1 CAD technology.

Computer Aided Draughting or Design offers several methods of representing the design model:

2D	Lines and text, similar to conventional drawing board.
3D	Vertices (corners or points in space), edges, surfaces in x, y and z.
Solid modelling	Solid geometry, fully defined three dimensional solid shapes, with free-form curved faces, material and mass properties.

Different methods suit different design circumstances. This section will introduce you to the most significant and expanding technology, Solid Modelling.

The graph below gives a very crude indication of the productivity of companies developing CAD software, through time.



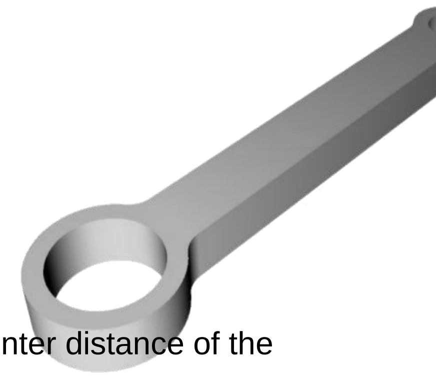
All of the acronyms below may be used in the context of mechanical computer aided engineering:

CAD	Computer Aided Design/Draughting
MCAD	Mechanical Computer Aided Design
CAE	Computer Aided Engineering

3.2 What you can do with solid modelling.

3.2.1 Representing your design.

Part modelling:



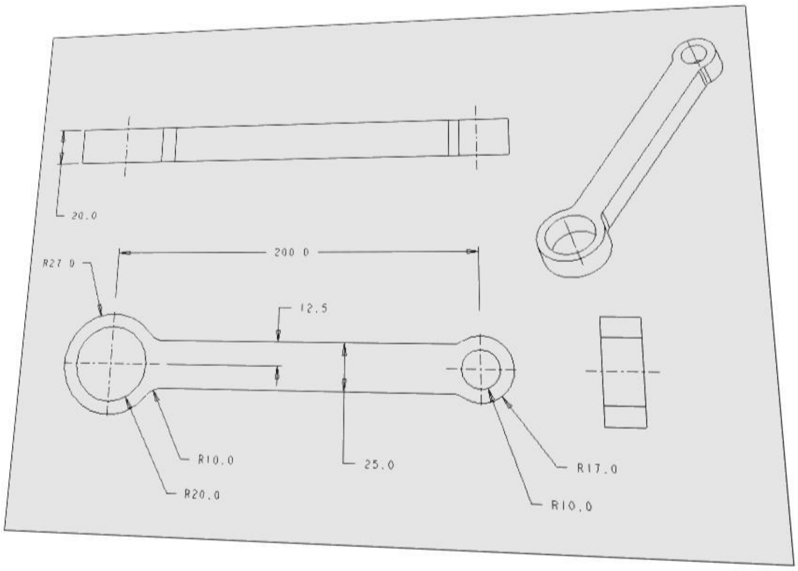
You can create 3D solid part models of your designs, such as this conrod. The dimensions that define the model are related to each other and can be changed and controlled. So, if you change one dimension, others will change with it. Software that allows this is refereed to as parametric. For example, change the center distance of the bores of this conrod and the whole model will stretch out.

You can also assign material properties, analyse mass properties, control the colour and texture of the appearance, create photo realistic images with lighting, shadows and perspective.

Orthographic drawing:

From the 3D model you can also create a detailed orthographic projection drawing.

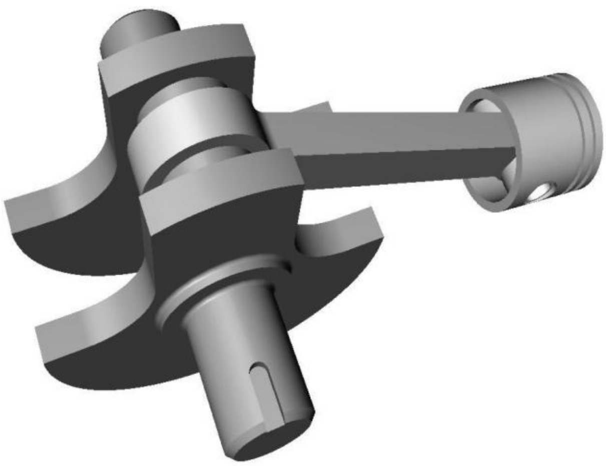
You can easily modify the design. Because the solid part file and the drawing file are connected, or associated with each other, a change in one will appear in the other. Change a dimension in the solid part and the same dimension will be updated in the drawing.



Most market leading solid modelling software offers this associativity and is usually referred to as 3D parametric associative solid modelling software.

Assembly modelling:

Solid model parts can be assembled. The assembly files can enjoy the same associativity as do part and drawing files. The conrod above has been assembled here with a crank shaft and a piston.

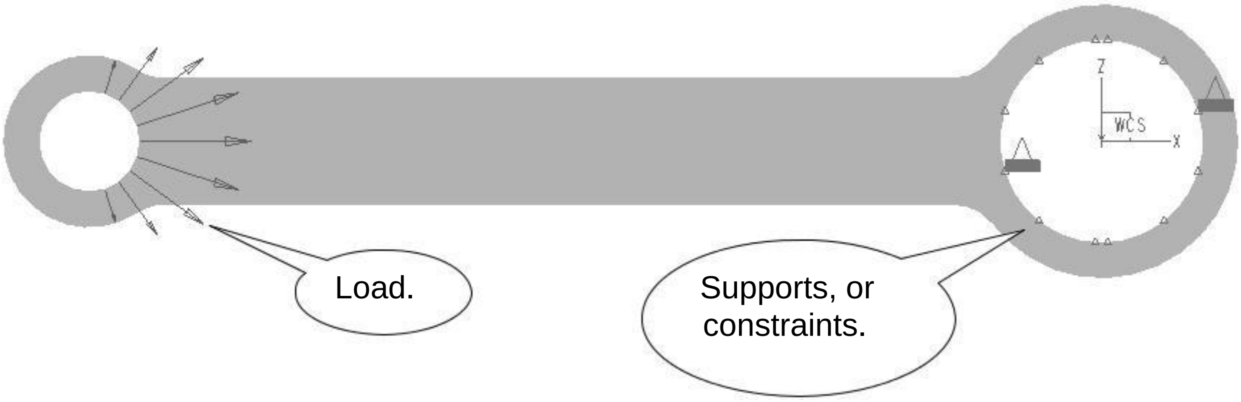


3.2.2 Analysing your design.

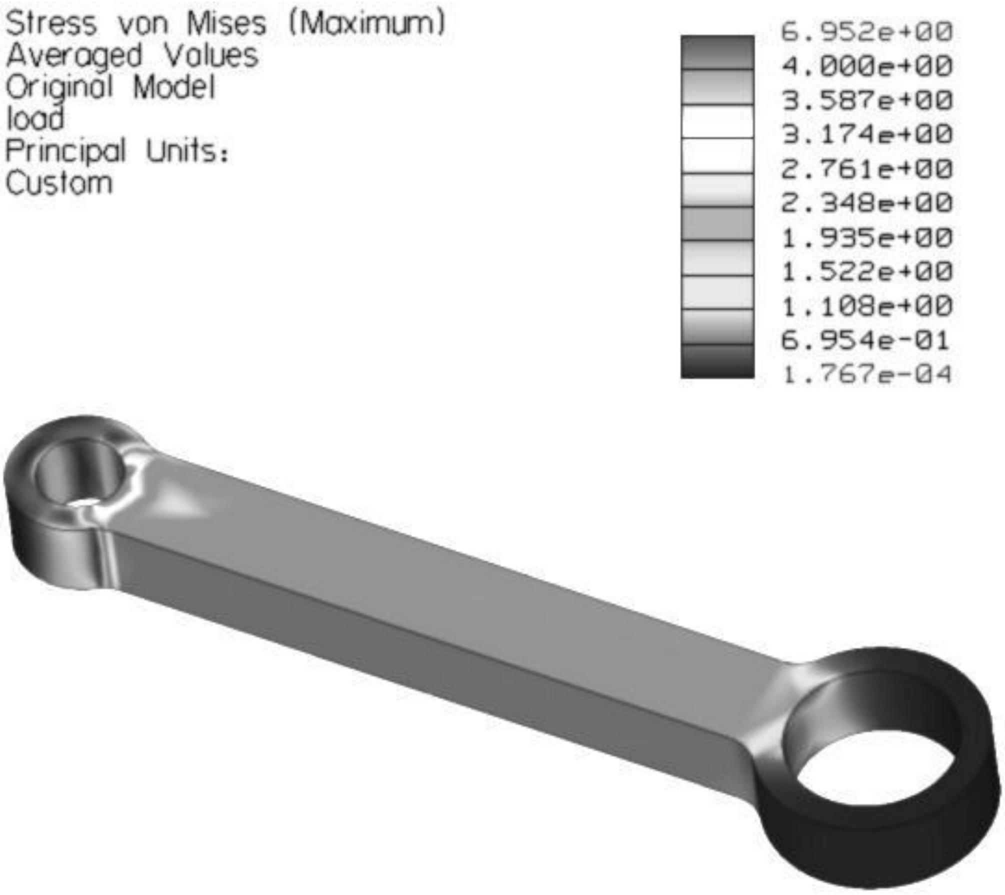
Having created a 3D solid model of a component, the geometry can then be used to predict how it may behave in real life.

For example:

To predict how high the stresses may be and how much the conrod may deflect under load, CAD software can be used to apply loads and supports and then analyse the structural behaviour of the model.



You, as the design engineer, can use the analysis results to help you decide whether the design is acceptable or requires modification. You may decide for the conrod, that the stresses are too high around the small end and modify the design accordingly. You run the analysis again, continuing the process until the predicted stress values are acceptable.



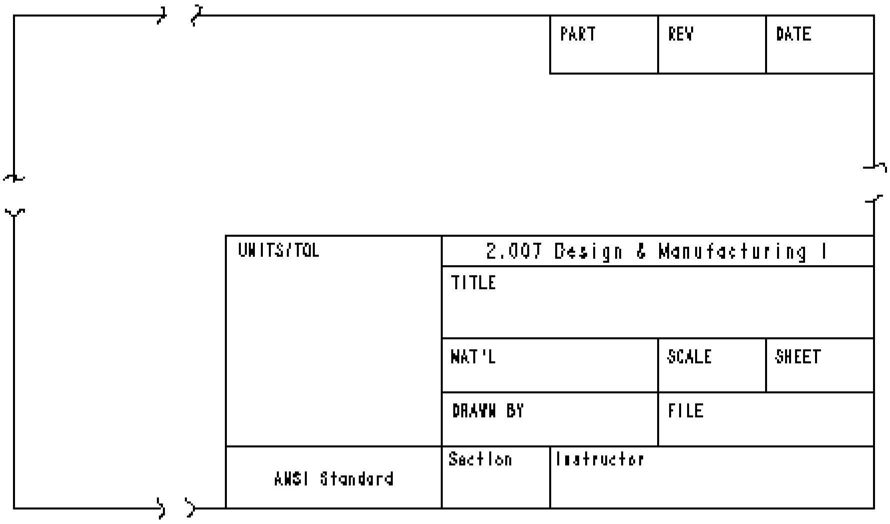
3.2.3 Visualise your design.

As time passes more and more 3D CAD software packages allow you to create high quality photorealistic images of your designs. By setting up an environment, with surrounding walls, a floor and a ceiling, lights, surface textures, etc. you can capture impressive images that cast shadows and reflections, giving a much more realistic impression of what your design may look like once manufactured. These facilities provide very powerful tools for developing, communicating and selling design ideas.

Most consumer product designs are modelled using 3D CAD software and then photo rendered as part of the product development process. Most public building designs now are also treated in the same way.

Engineering Drawings

The Blank Engineering Drawing Form



This blank template can be printed off the 2.007 locker on Server. You can use this form to make your own production drawings for class. This will get you used to the look and feel of an engineering drawing.

To print this file from Server:

```
server% add 2.007


server% lpr -P<printer name> /mit/2.007/blankdrawing.ps
```

Engineering Drawing and Sketching

Drawing Handout Index

Introduction

One of the best ways to communicate one's ideas is through some form of picture or drawing. This is especially true for the engineer. The purpose of this guide is to give you the basics of engineering sketching and drawing.

We will treat "sketching" and "drawing" as one. "Sketching" generally means freehand drawing. "Drawing" usually means using drawing instruments, from compasses to computers to bring precision to the drawings.

This is just an introduction. Don't worry about understanding every detail right now - just get a general feel for the language of graphics.

We hope you like the object in figure 1, because you'll be seeing a lot of it. Before we get started on any technical drawings, let's get a good look at this strange block from several angles.

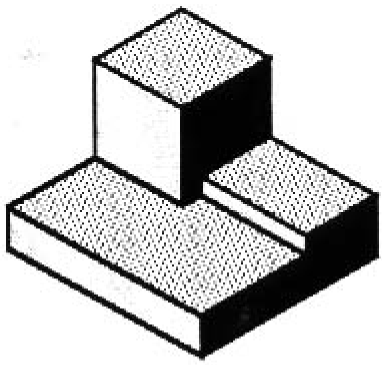


Figure 1 A machined block

Isometric Drawing

The representation of the object in figure 2 is called an isometric drawing. This is one of a family of three-dimensional views called pictorial drawings. In an isometric drawing, the object's vertical lines are drawn vertically, and the horizontal lines in the width and depth planes are shown at 30 degrees to the horizontal. When drawn under these guidelines, the lines parallel to these three axes are at their true (scale) lengths. Lines that are not parallel to these axes will not be of their true length.

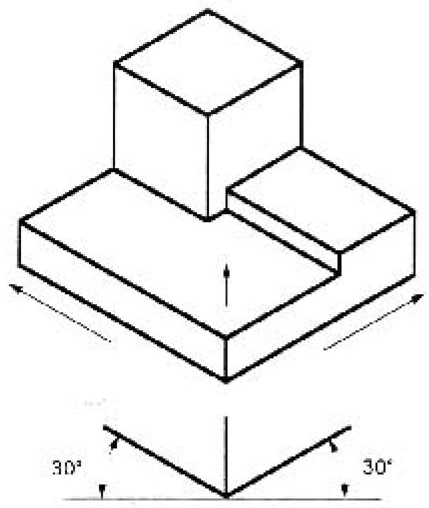


Figure 2 An isometric drawing

Any engineering drawing should show everything: a complete understanding of the object should be possible from the drawing. If the isometric drawing can show all details and all dimensions on one drawing, it is ideal.

One can pack a great deal of information into an isometric drawing. Look, for instance, at the instructions for a home woodworker in figure 3, taken from the Popular Mechanics magazine. Everything the designer needs to convey to the craftsperson is in this one isometric drawing.

However, if the object in figure 2 had a hole on the back side, it would not be visible using a single isometric drawing. In order to get a more complete view of the object, an orthographic projection may be used.

Orthographic or Multiview Drawing

Imagine that you have an object suspended by transparent threads inside a glass box, as in figure 4.

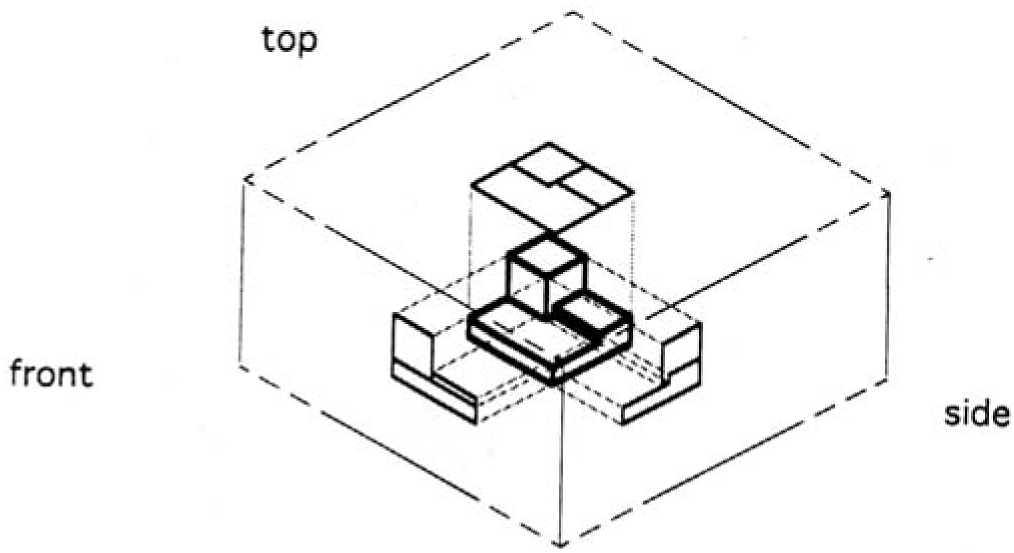


Figure 4 The block suspended in a glass box.

Then draw the object on each of three faces as seen from that direction. Unfold the box (figure 5) and you have the three views. We call this an "orthographic" or "multiview" drawing.

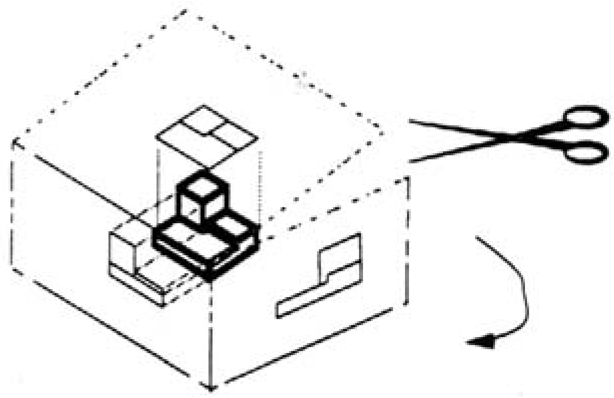


Figure 5 The creation of an orthographic multiview drawing

Figure 6 shows how the three views appear on a piece of paper after unfolding the box.

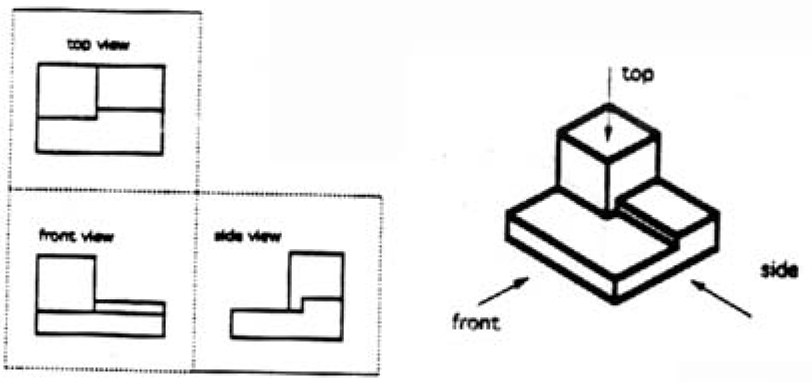


Figure 6 A multiview drawing and its explanation

Which views should one choose for a multiview drawing? The views that reveal every detail about the object. Three views are not always necessary; we need only as many views as are required to describe the object fully. For example, some objects need only two views, while others need four. The circular object in figure 7 requires only two views.

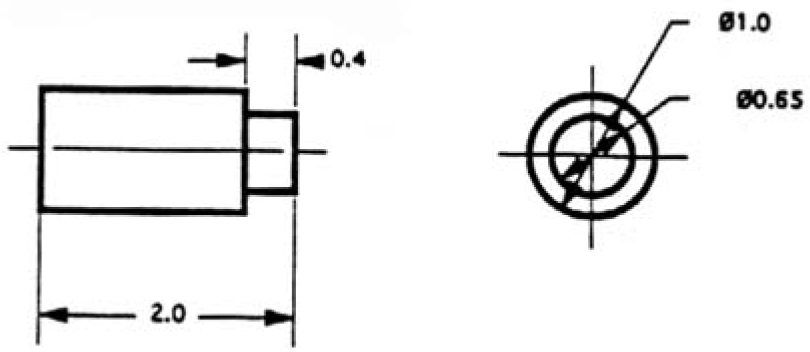


Figure 7 An object needing only two orthogonal views

Dimensioning

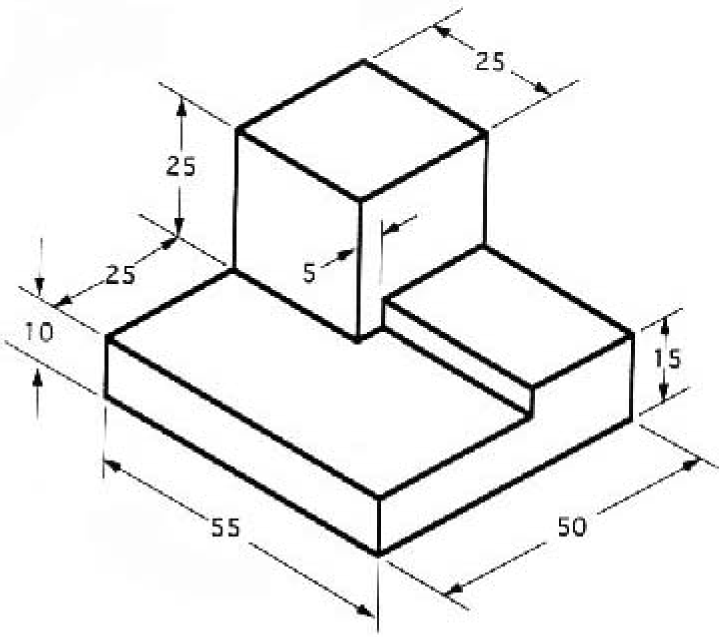


Figure 8 An isometric view with dimensions

We have "dimensioned" the object in the isometric drawing in figure 8. As a general guideline to dimensioning, try to think that you would make an object and dimension it in the most useful way. Put in exactly as many dimensions as are necessary for the craftsperson to make it -no more, no less. Do not put in redundant dimensions. Not only will these clutter the drawing, but if "tolerances" or accuracy levels have been included, the redundant dimensions often lead to conflicts when the tolerance allowances can be added in different ways.

Repeatedly measuring from one point to another will lead to inaccuracies. It is often better to measure from one end to various points. This gives the dimensions a reference standard. It is helpful to choose the placement of the dimension in the order in which a machinist would create the part. This convention may take some experience. It is covered later (figures 49-52).

Sectioning

There are many times when the interior details of an object cannot be seen from the outside (figure 9).

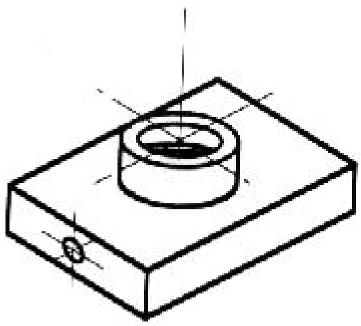


Figure 9 An isometric drawing that does not show all details

We can get around this by pretending to cut the object on a plane and showing the "sectional view". The sectional view is applicable to objects like engine blocks, where the interior details are intricate and would be very difficult to understand through the use of "hidden" lines (hidden lines are, by convention, dotted) on an orthographic or isometric drawing.

Imagine slicing the object in the middle (figure 10)

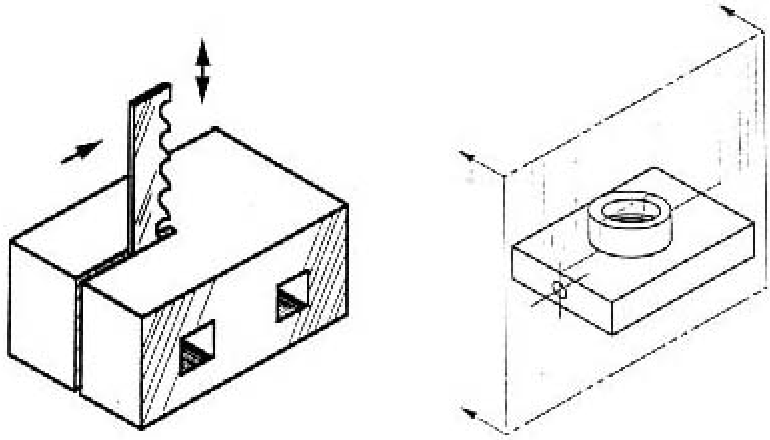


Figure 10 "Sectioning" an object

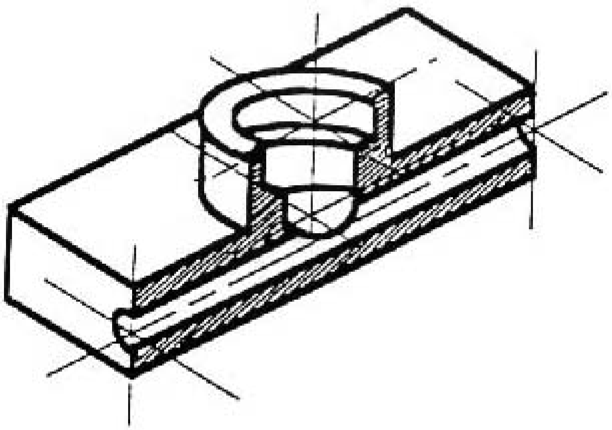


Figure 11 Sectioning the object in figure 9

Take away the front half (figure 11) and what you have is a full section view (figure 12).

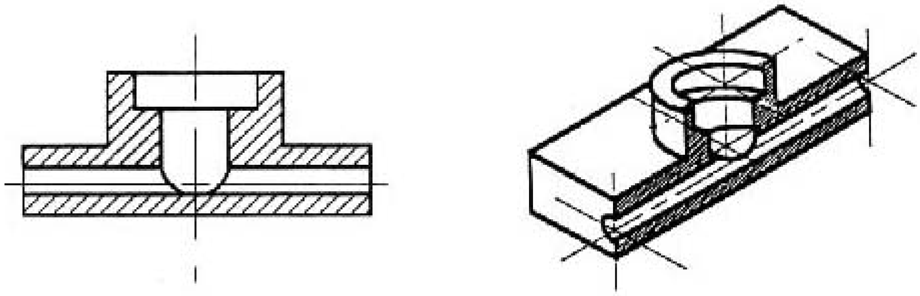


Figure 12 Sectioned isometric and orthogonal views

The cross-section looks like figure 12 when it is viewed from straight ahead.

Drawing Tools

To prepare a drawing, one can use manual drafting instruments (figure 13) or computer-aided drafting or design, or CAD. The basic drawing standards and conventions are the same regardless of what design tool you use to make the drawings. In learning drafting, we will approach it from the perspective of manual drafting. If the drawing is made without either instruments or CAD, it is called a freehand sketch.

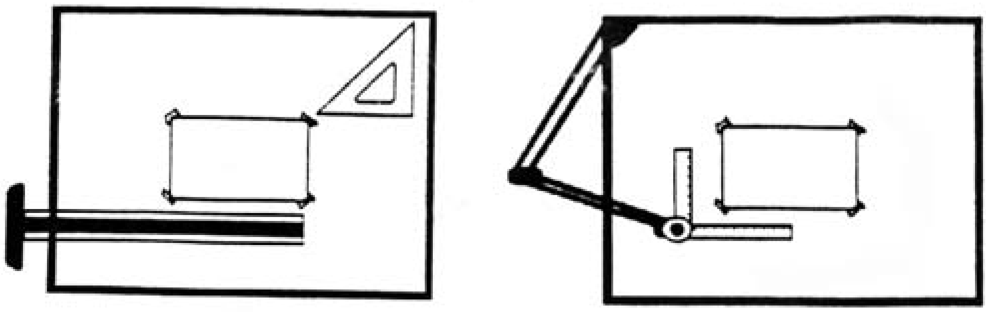


Figure 13 Drawing Tools

"Assembly" Drawings

An isometric view of an "assembled" pillow-block bearing system is shown in figure 14. It corresponds closely to what you actually see when viewing the object from a particular angle. We cannot tell what the inside of the part looks like from this view.

We can also show isometric views of the pillow-block being taken apart or "disassembled" (figure 15). This allows you to see the inner components of the bearing system. Isometric drawings can show overall arrangement clearly, but not the details and the dimensions.

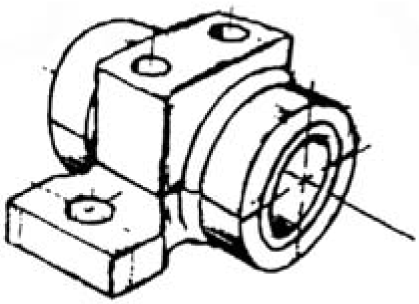


Figure 14 Pillow-block
(Freehand sketch)

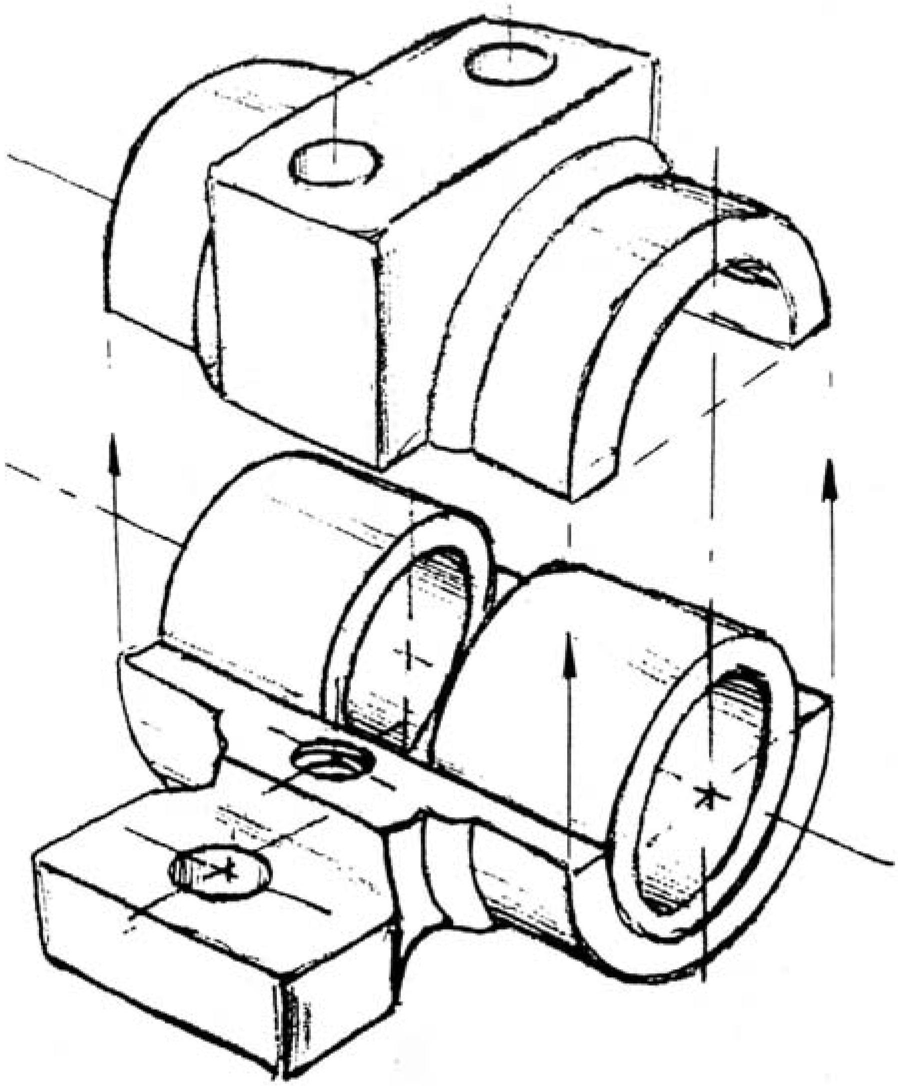


Figure 15 Disassembled pillow-block

Cross-Sectional Views

A cross-sectional view portrays a cut-away portion of the object and is another way to show hidden components in a device.

Imagine a plane that cuts vertically through the center of the pillow block as shown in figure 16. Then imagine removing the material from the front of this plane, as shown in figure 17.

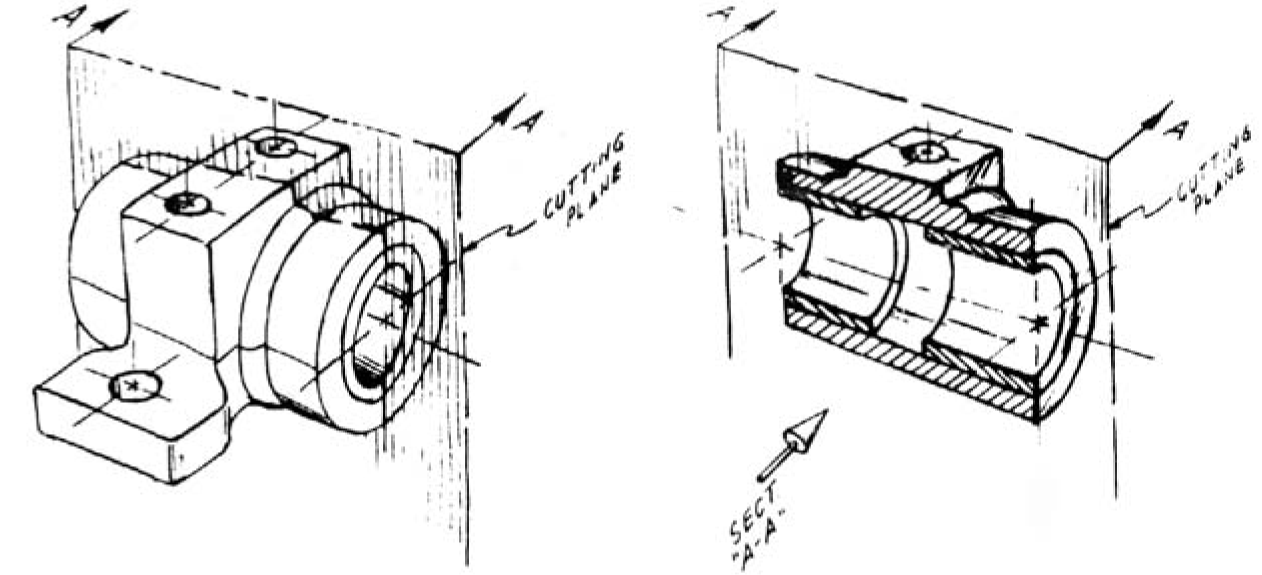


Figure 16&17 Pillow block

This is how the remaining rear section would look. Diagonal lines (cross-hatches) show regions where materials have been cut by the cutting plane.

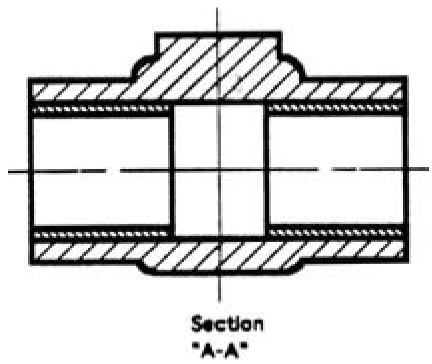


Figure 18 Section "A-A"

This cross-sectional view (section A-A, figure 18), one that is orthogonal to the viewing direction, shows the relationships of lengths and diameters better. These drawings are easier to make than isometric drawings. Seasoned engineers can interpret orthogonal drawings without needing an isometric drawing, but this takes a bit of practice.

The top "outside" view of the bearing is shown in figure 19. It is an orthogonal (perpendicular) projection. Notice the direction of the arrows for the "A-A" cutting plane.

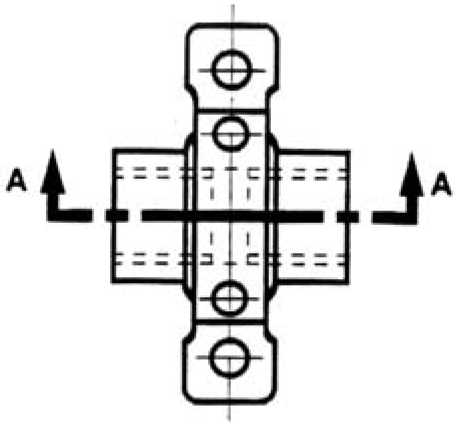


Figure 19 The top "outside" view of the bearing

Half-Sections

A half-section is a view of an object showing one-half of the view in section, as in figure 20a and b.

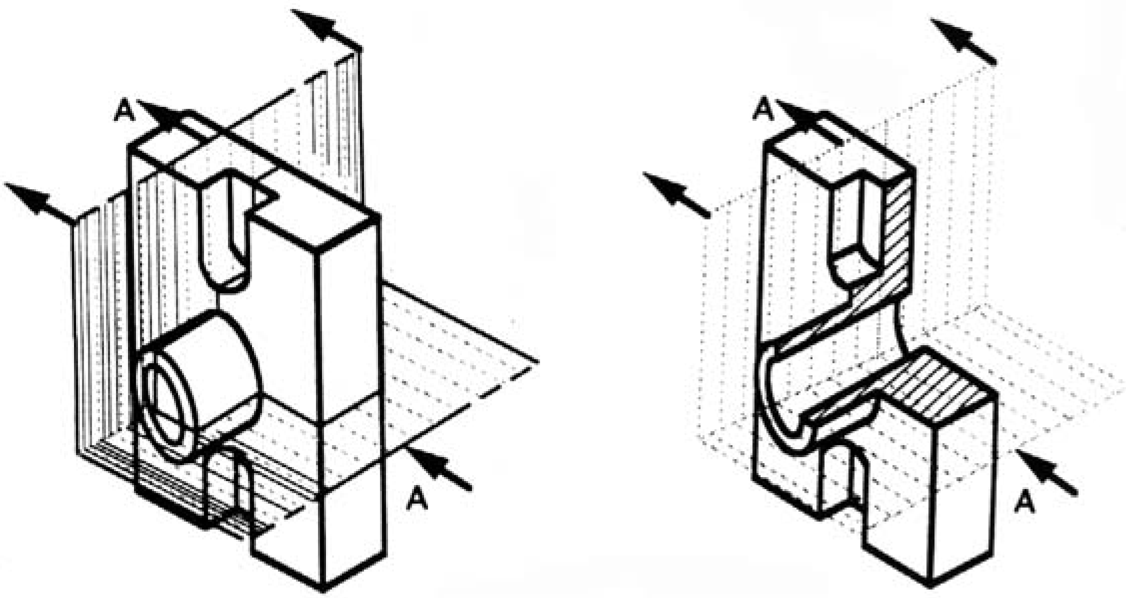


Figure 20a Full and sectioned isometric views

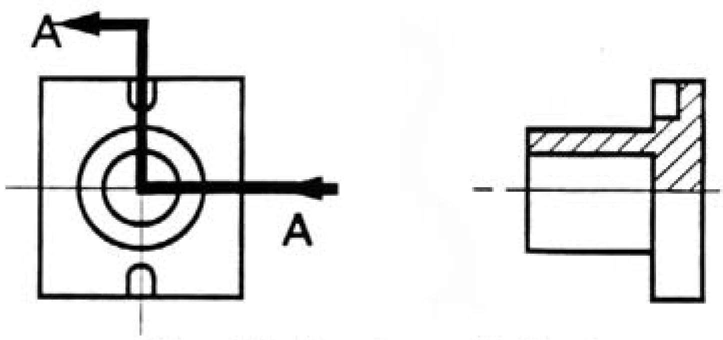


Figure 20b Front view and half section

The diagonal lines on the section drawing are used to indicate the area that has been theoretically cut. These lines are called *section lining* or *cross-hatching*. The lines are thin and are usually drawn at a 45-degree angle to the major outline of the object. The spacing between lines should be uniform.

A second, rarer, use of cross-hatching is to indicate the material of the object. One form of cross-hatching may be used for cast iron, another for bronze, and so forth. More usually, the type of material is indicated elsewhere on the drawing, making the use of different types of cross-hatching unnecessary.

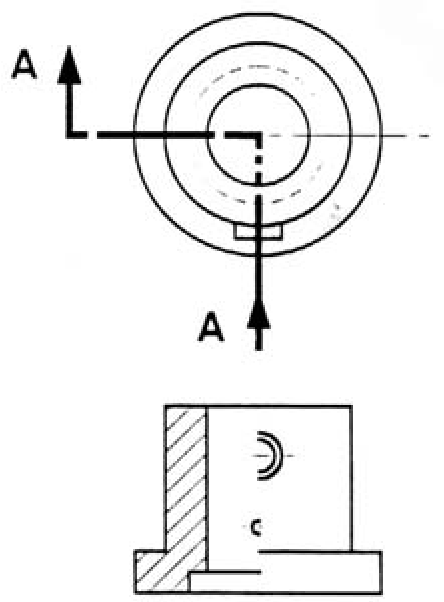


Figure 21 Half section without hidden lines

Usually hidden (dotted) lines are not used on the cross-section unless they are needed for dimensioning purposes. Also, some hidden lines on the non-sectioned part of the drawings are not needed (figure 12) since they become redundant information and may clutter the drawing.

Sectioning Objects with Holes, Ribs, Etc.

The cross-section on the right of figure 22 is technically correct. However, the convention in a drawing is to show the view on the left as the preferred method for sectioning this type of object.

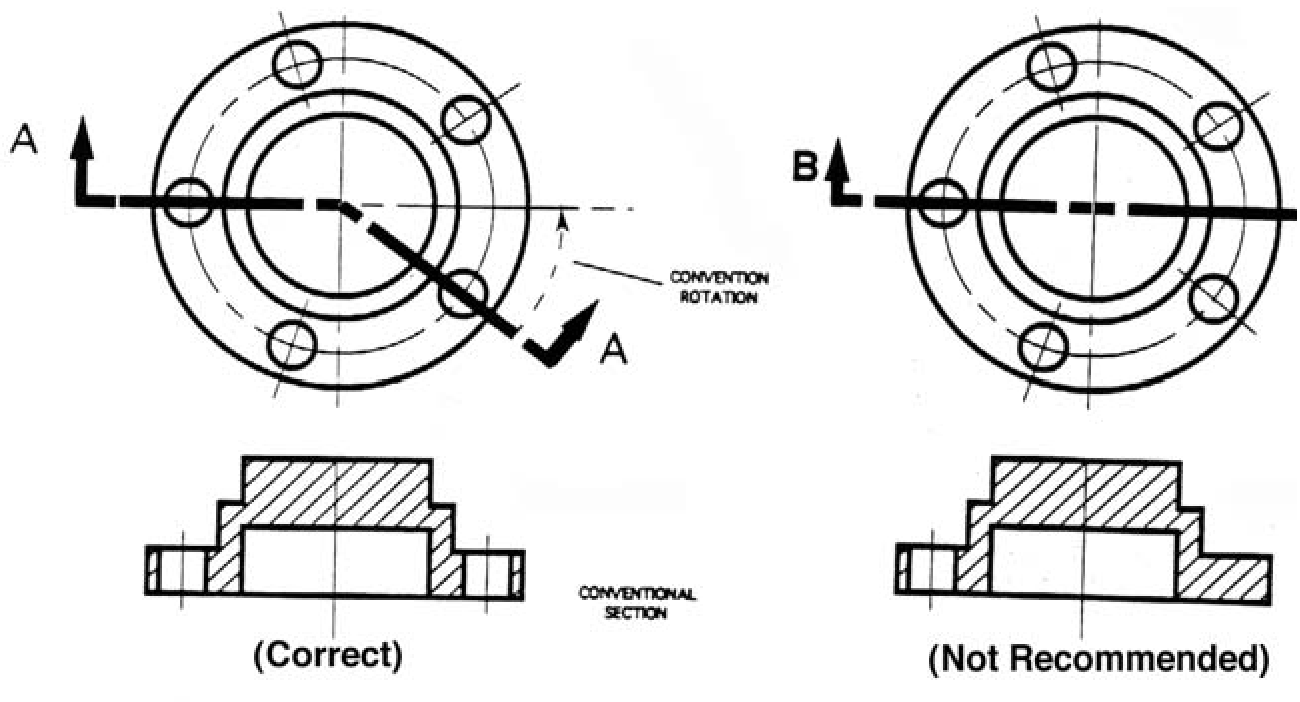


Figure 22 Cross section

Dimensioning

The purpose of dimensioning is to provide a clear and complete description of an object. A complete set of dimensions will permit only one interpretation needed to construct the part. Dimensioning should follow these guidelines.

1. Accuracy: correct values must be given.
2. Clearness: dimensions must be placed in appropriate positions.
3. Completeness: nothing must be left out, and nothing duplicated.
4. Readability: the appropriate line quality must be used for legibility.

The Basics: Definitions and Dimensions

The **dimension line** is a thin line, broken in the middle to allow the placement of the dimension value, with arrowheads at each end (figure 23).

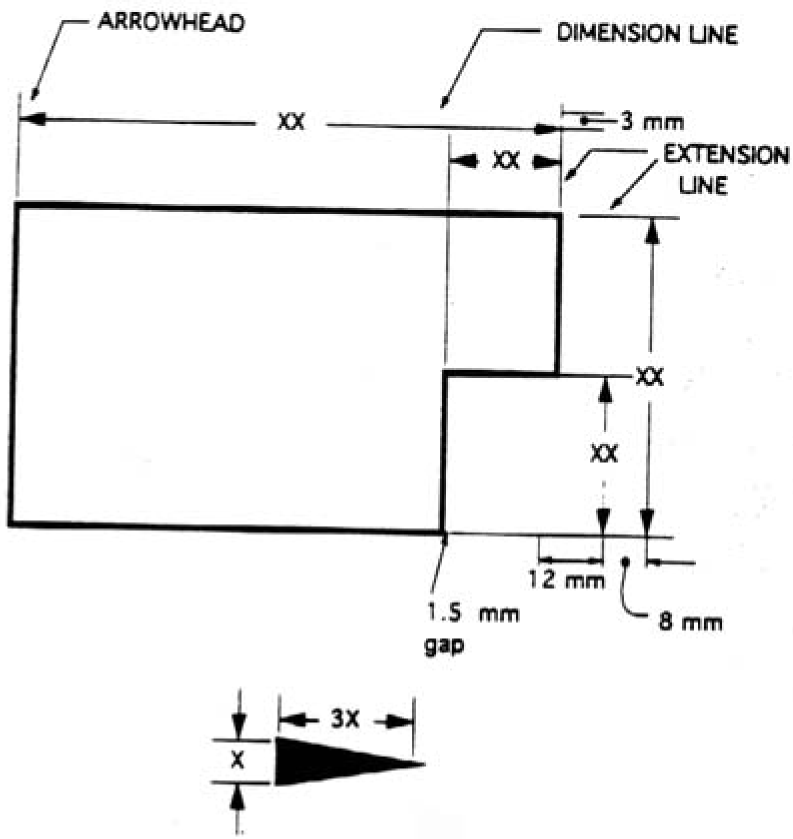


Figure 23 Dimensioning drawing

An **arrowhead** is approximately 3 mm long and 1 mm wide. That is, the length is roughly three times the width. An **extension line** extends a line on the object to the dimension line. The first dimension line should be approximately 12 mm (0.6 in) from the object. Extension lines begin 1.5 mm from the object and extend 3 mm from the last dimension line.

A leader is a thin line used to connect a dimension with a particular area (figure 24).

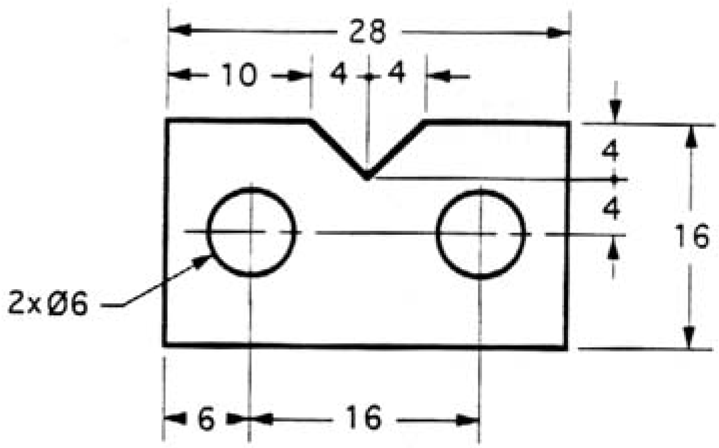


Figure 24 Example drawing with a leader

A leader may also be used to indicate a note or comment about a specific area. When there is limited space, a heavy black dot may be substituted for the arrows, as in figure 23. Also in this drawing, two holes are identical, allowing the "2x" notation to be used and the dimension to point to only one of the circles.

Where To Put Dimensions

The dimensions should be placed on the face that describes the feature most clearly. Examples of appropriate and inappropriate placing of dimensions are shown in figure 25.

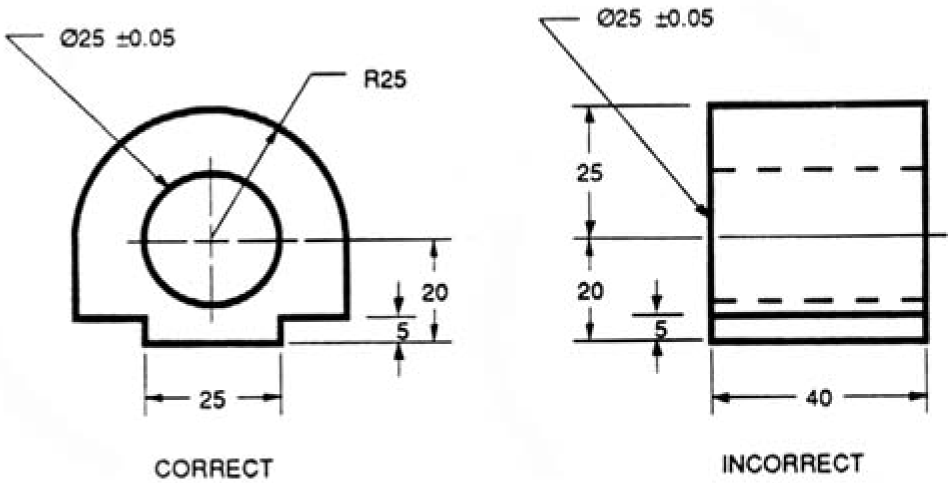


Figure 25 Example of appropriate and inappropriate dimensioning

In order to get the feel of what dimensioning is all about, we can start with a simple rectangular block. With this simple object, only three dimensions are needed to describe it completely (figure 26). There is little choice on where to put its dimensions.

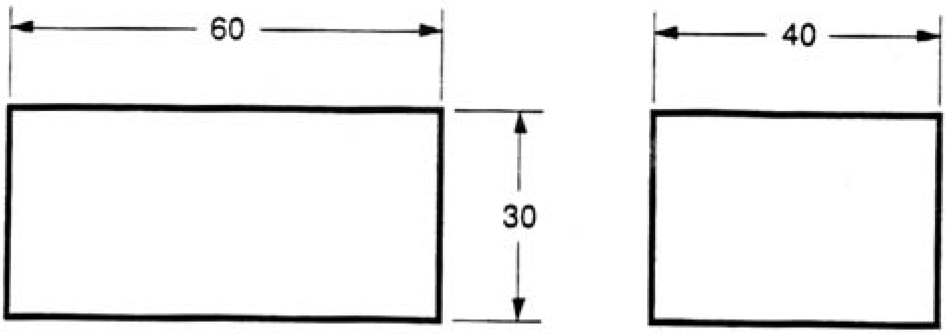


Figure 26 Simple object

We have to make some choices when we dimension a block with a notch or cutout (figure 27). It is usually best to dimension from a common line or surface. This can be called the datum line of surface. This eliminates the addition of measurement or machining inaccuracies that would come from "chain" or "series" dimensioning. Notice how the dimensions originate on the datum surfaces. We chose one datum surface in figure 27, and another in figure 28. As long as we are consistent, it makes no difference. (We are just showing the top view).

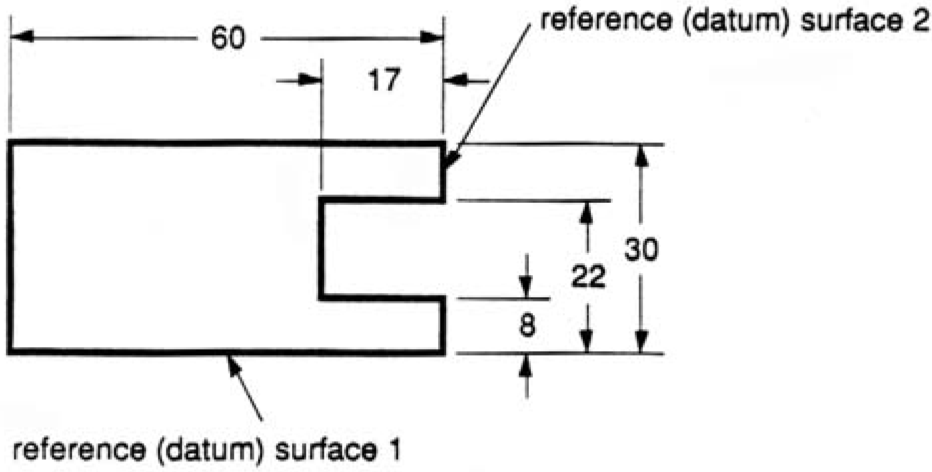


Figure 27 Surface datum example

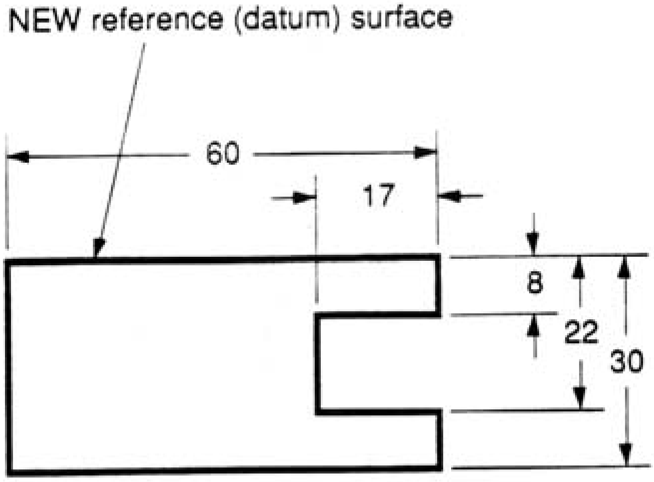


Figure 28 Surface datum example

In figure 29 we have shown a hole that we have chosen to dimension on the left side of the object. The Ø stands for "diameter".

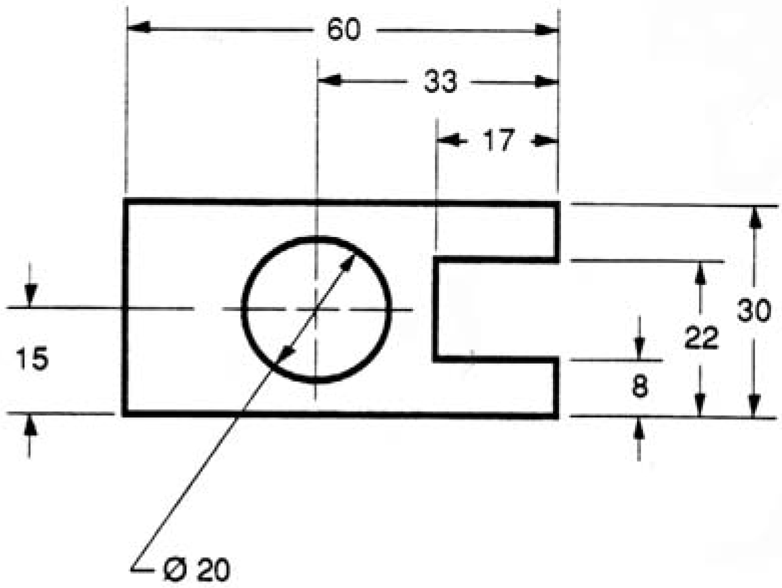


Figure 29 Example of a dimensioned hole

When the left side of the block is "radiuses" as in figure 30, we break our rule that we should not duplicate dimensions. The total length is known because the radius of the curve on the left side is given. Then, for clarity, we add the overall length of 60 and we note that it is a reference (REF) dimension. This means that it is not really required.

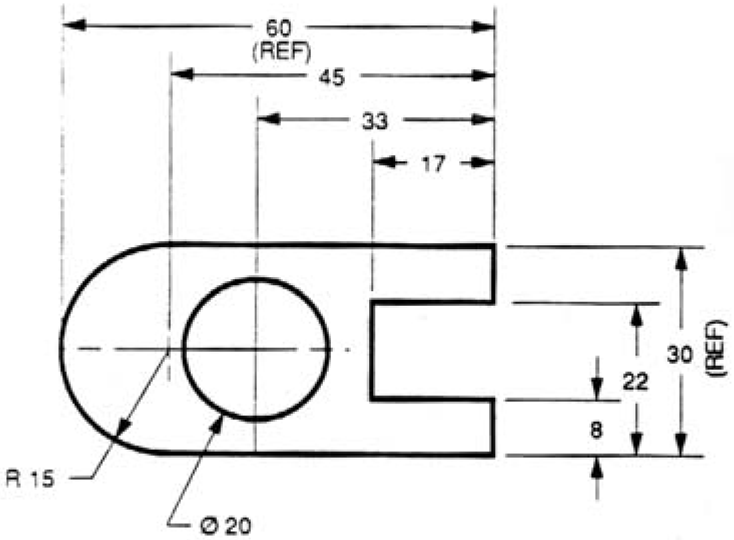


Figure 30 Example of a directly dimensioned hole

Somewhere on the paper, usually the bottom, there should be placed information on what measuring system is being used (e.g. inches and millimeters) and also the scale of the drawing.

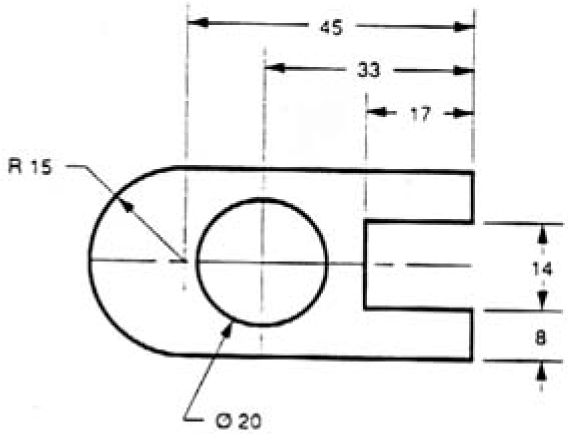


Figure 31 Example of a directly dimensioned hole

This drawing is symmetric about the horizontal centerline. Centerlines (chain-dotted) are used for symmetric objects, and also for the center of circles and holes. We can dimension directly to the centerline, as in figure 31. In some cases this method can be clearer than just dimensioning between surfaces.