

Engineering Working Drawings Basics

Engineering graphics is an effective way of communicating technical ideas and it is an essential tool in engineering design where most of the design process is graphically based. Engineering graphics is used in the design process for visualization, communication, and documentation.

Graphics is a visual communications language that include images, text, and numeric information. Graphics communications using engineering drawings and models is a clear and precise language with definite rules that must be mastered in order to be successful in engineering design. Graphics communications are used in every phase of engineering design starting from concept illustration all the way to the manufacturing phase.

- An **engineering (or technical) drawing** is a graphical representation of a part, assembly, system, or structure and it can be produced using freehand, mechanical tools, or computer methods.
- **Working drawings** are the set of technical drawings used during the manufacturing phase of a product. They contain all the information needed to manufacture and assemble a product.

Codes and Standards

Codes and standards are made to organize and unify the engineering work.

Imagine; what if there was no standard for bolts?

- A code is a set of specifications for the analysis, design, manufacture, and construction of something.
- A standard is a set of specifications for parts, materials, or processes intended to achieve uniformity, efficiency and specific quality.

Examples of the organizations that establish standards and design codes: ISO , AISI, SAE, ASTM, ASME, ANSI, DIN.

There are many different standards related to technical drawings.

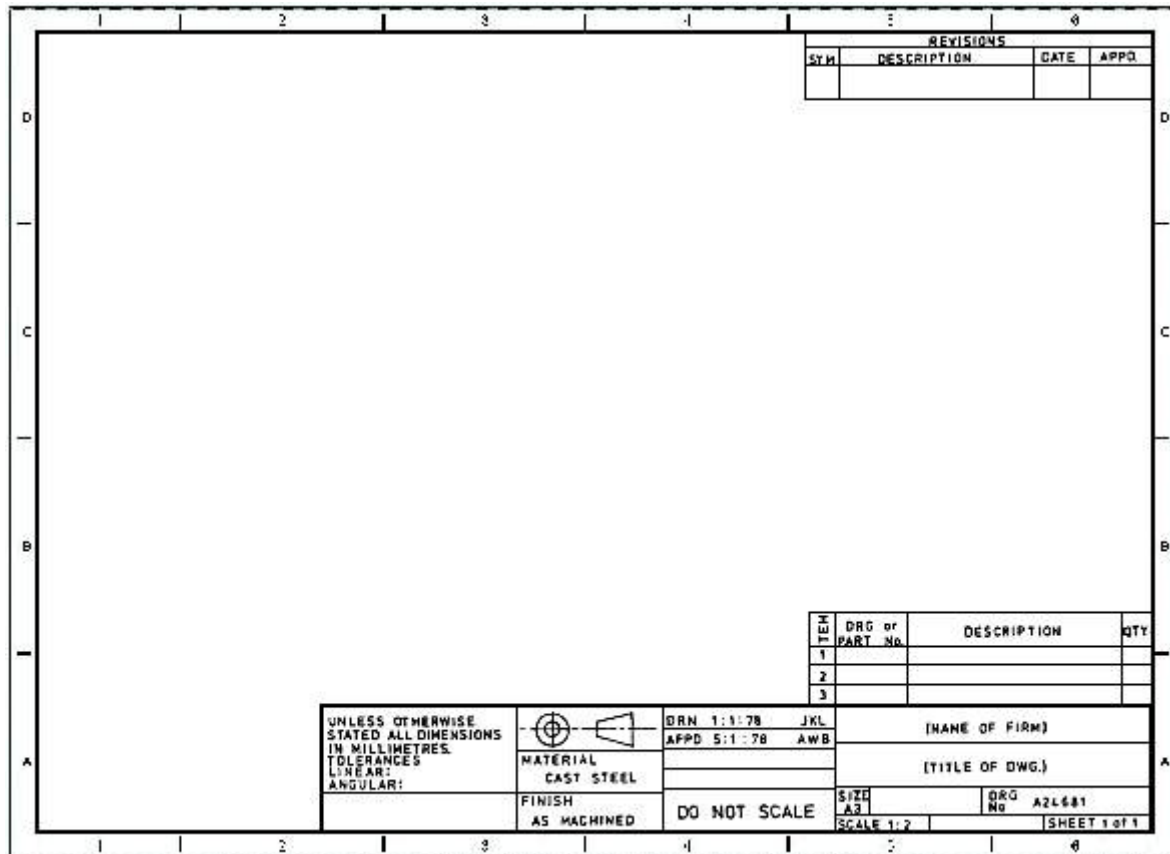
The ISO standards for technical drawings are found in a two volumes handbook:

- *ISO Standards Handbook: **Technical drawings***, Volume 1: Technical drawings in general
- *ISO Standards Handbook: **Technical drawings***, Volume 2: Mechanical engineering drawings; Construction drawings; Drawing equipment

Drawing Sheet Layout

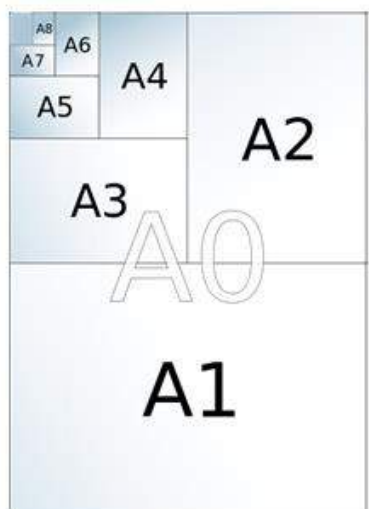
Standard layouts of drawing sheets are specified by the various standards organizations.

The figure shows the layout of a typical sheet, showing the drawing frame, a typical title block, parts list (*bill of materials*) and revision table.

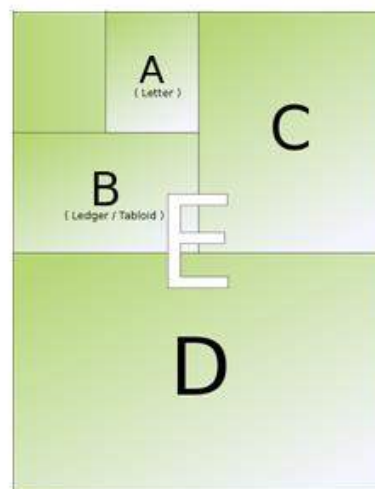


Sheet Frames It is standard practice for a drawing frame to be printed on each sheet, defining a margin around the outside of drawing area.

- The margins of the drawing frames are standardized for each size of paper.



ISO standard paper sizes (ISO 216)
A0 : 1189 mm x 841 mm



American standard paper sizes
E : 44 in x 34 in

- The standard sizes of drawing frames for ISO sized sheets, with no filing margins, are as shown in the table:

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

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

- the name of the company or organization
- the title of the drawing
- the drawing number, which is generally a unique filing identifier
- the scale
- the drawing size
- the angle of projection used, either first or third, generally shown symbolically
- the signature or initials of the draftsman, checker, approving officer, and issuing officer, with the respective dates
- the material of the part
- the revision number
- the sheet number for multi-sheet drawings
- other information as required (tolerances, surface finish, etc.)

In addition to the information above, for drawings produced using CAD software, it is highly recommended to have the following information in the title block:

- the name of the CAD software used and its version
- the name of the drawing file
- the name of the source part or assembly file
- the units of the dimensions (*if the drawing is mistakenly printed on a different paper size, the scale becomes meaningless*)

Revision Table, A revision table is normally located in the upper right of the drawing frame. All modifications to the drawing should be documented there.

Material or Parts List (Bill of Materials), If the drawing contains a number of parts, or if it is an assembly drawing, a tabulated parts list is added to the drawing. The bill of materials is usually placed at the bottom right of the drawing frame, just above the title block.

The bill of materials should give the following information:

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

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

Zoning, A drawing may be divided up into a grid using letters and numbers. When zoning is used it is located inside the drawing frame. Zoning is usually used for large size drawing sheets where it allows easy references to various parts of the drawing by referencing a coordinate such as B4.

Notes, Information other than pictorial views and dimensions necessary for completing a drawing is classified as “notes”. Notes are usually placed in the lower left corner of the drawing sheet and they are numbered consecutively downward.

Some examples of the notes used in working drawings are as follows:

- Dimension Notes ALL DIMENSIONS APPLY AFTER SURFACE TREATMENT.
- Heat Treating Notes HEAT TREAT IN ACCORDANCE WITH standard-XXX TO XXX-hardness
- Joining Method Notes (welding, brazing, etc.) FUSION WELD IN ACCORDANCE WITH standard-XXX
- Plating and Coating Notes CADMIUM PLATE IN ACCORDANCE WITH standard-XXX
- Surface Preparation Notes SURFACE TO BE CLEANED AND POLISHED FOR VACUUM USE.
- Testing and Inspection Notes RADIOGRAPHIC INSPECTION OF ALL WELDS IN ACCORDANCE WITH standard-XXX
- Threaded Inserts — Cleaning and Installation Notes TIGHTENING TORQUE OF THREADED FASTENERS SHOULD BE AS LISTED IN TORQUE TABLE
- Miscellaneous Notes PLACE IN BAG AFTER FINAL ELECTROPOLISHING.

TORQUE TABLE	
THREAD SIZE	TORQUE, IN-LBS
6-32 UNC	x.x
...	...
...	...

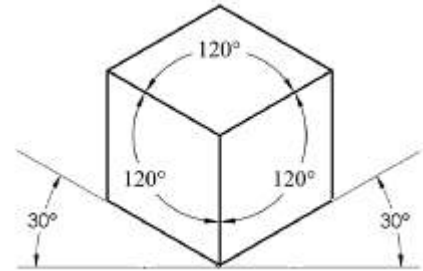
Three-Dimensional Views

Working drawings usually include a three-dimensional view (or views) of the part or assembly as needed (but it is not a must).

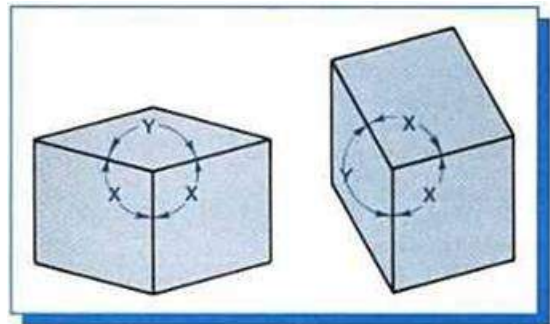
Axonometric Views

An axonometric view is a view in which an object appears to be rotated to show its all three dimensions. Axonometric views are classified according to how the axes are oriented into Isometric, Diametric and Trimetric.

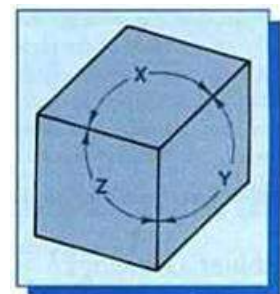
- **Isometric View** In isometric views the two edges of the view make 30° angles with the horizontal and that makes the three angles between the view axes to be equal to 120° .



- **Diametric View** In diametric views the angles may vary but two of the three angles between the view axes are equal.

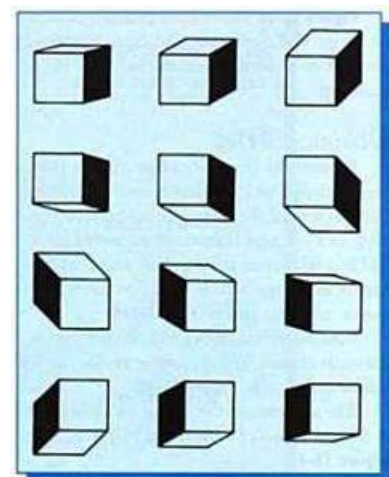


- **Trimetric View** In trimetric views all three angles between view axes are different.



Oblique Views

The front face of an oblique view is drawn to the exact shape and size while the other two faces are projected from this front face at any convenient angle (usually 45°) and in any direction. An oblique view is not a realistic kind of view that can be obtained by rotating the part using 3D CAD software, however it is convenient and easy to use for freehand sketching.

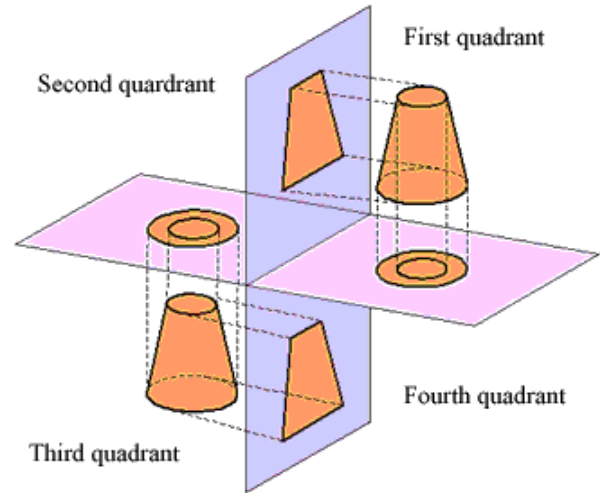


Orthographic Projections (two-dimensional views)

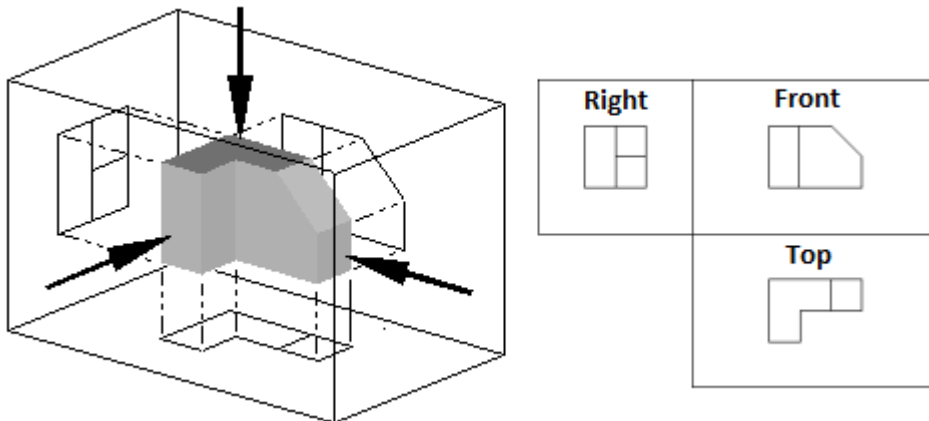
Orthographic projection (or orthogonal projection) is a means of representing a three-dimensional object in two dimensions.

A multi-view orthographic projection is an illustration technique in which up to six pictures of an object are produced (*usually three are sufficient in most cases or two in some cases*), with each projection plane being perpendicular to one of the coordinate axes of the object.

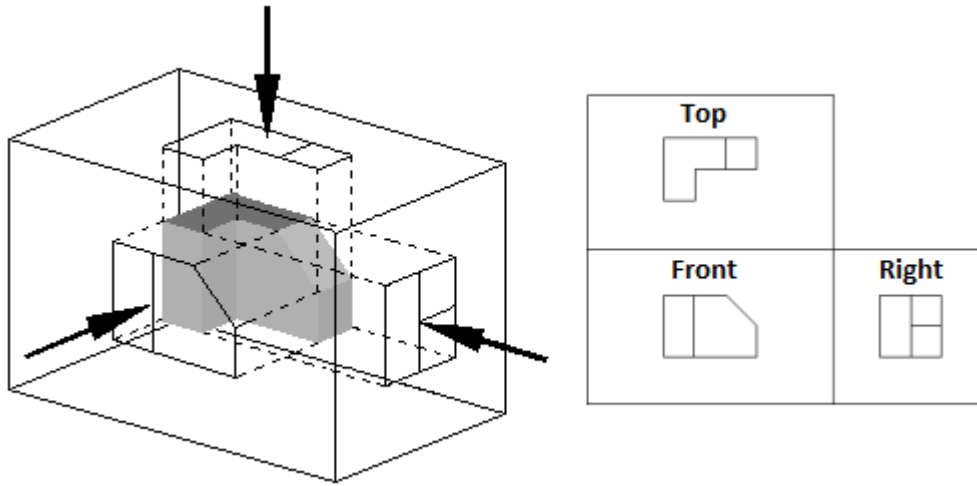
The views are positioned relative to each other according to either of two schemes: first-angle or third-angle projection.



- **First-Angle Projection:** In first-angle projection the object is conceptually located in quadrant I (*i.e., it floats above and before the viewing planes*), the planes are opaque, and each view is pushed through the object onto the plane furthest from it.



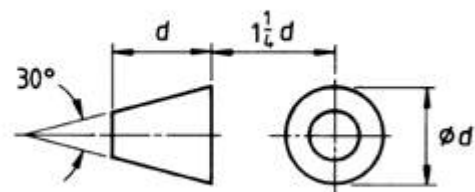
- **Third-Angle Projection:** In third-angle projection the object is conceptually located in quadrant III (*i.e., it is positioned below and behind the viewing planes*), the planes are transparent, and each view is pulled onto the plane closest to it.



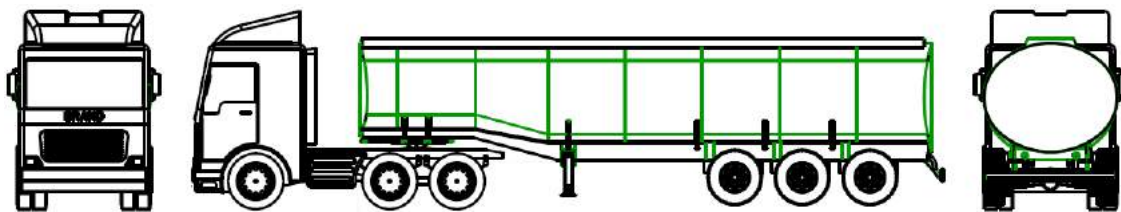
Projection Symbols

A standard projection symbol is used in drawings to identify the projection system of the orthographic views.

Projection	Symbol
First angle	
Third angle	

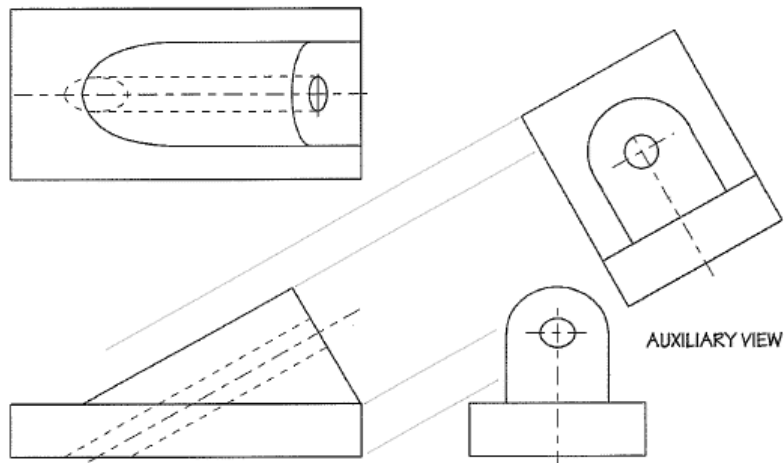


First or third angle views?



Auxiliary Views

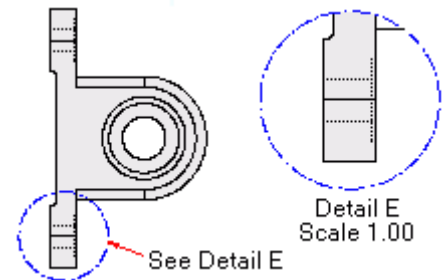
Auxiliary views utilize an additional projection plane other than the common planes in a multi-view. Since the projections of an object need to show the true shape and size of the object, the projection plane must be



parallel to the object surface. Therefore, any surface that is not in line with the three major axis needs its own projection plane to show the features correctly.

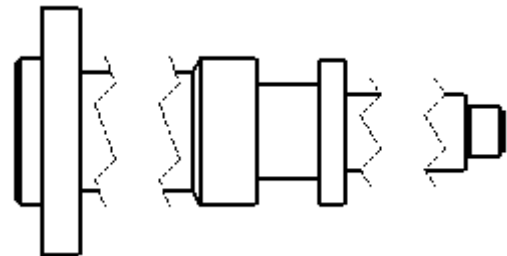
Detail Views

A detail view is a separate large-scale drawing view of a small section of another view. It is usually used to show features that are small relative to the size of the part or assembly being depicted by the drawing. A detail view is designated by a letter and its scale is indicated.



Broken Views

Broken views are usually used for parts having one of its dimensions significantly larger than the other dimension (*such as long shafts*) in order to make it possible to display the drawing view in a larger scale on a smaller size drawing sheet. In a broken view, portions of the view that do not contain any details are broken out (*removed*) and the remaining portions of the view are brought closer together.



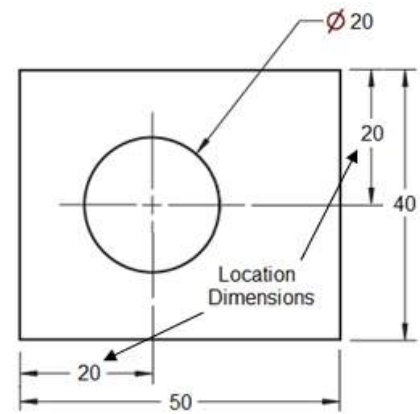
Dimensioning

Dimensioning is the process of defining the size, form and location of geometric features and components on an engineering drawing.

Before an object can be built, complete information about both the size and shape of the object must be available. The exact shape of the part or assembly is shown by the different views in the drawing sheet. Dimensions are added to the two-dimensional views (*not to the 3D view*) in the drawing sheet such that it will show all the size and location details of the part.

- In metric drawings, generally, dimensions are in millimeters. To avoid having to specify 'mm' after every dimension, a label such as '**all dimensions in mm**' or '**unless otherwise stated all dimensions are in mm**' is usually contained in the title block.
- If the dimension is less than one a leading zero should be used before the decimal point (e.g., **0.5**).
- Dimensions used in drawings can be categorized as:

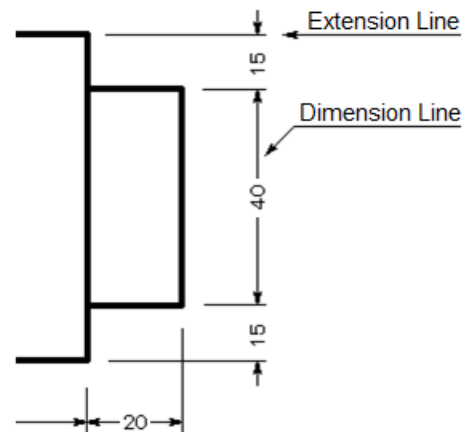
- **Size dimensions** - define size of features (*radius, diameter, length, width, thickness, etc.*).
- **Location dimensions** - define location of part features (*such as holes*).



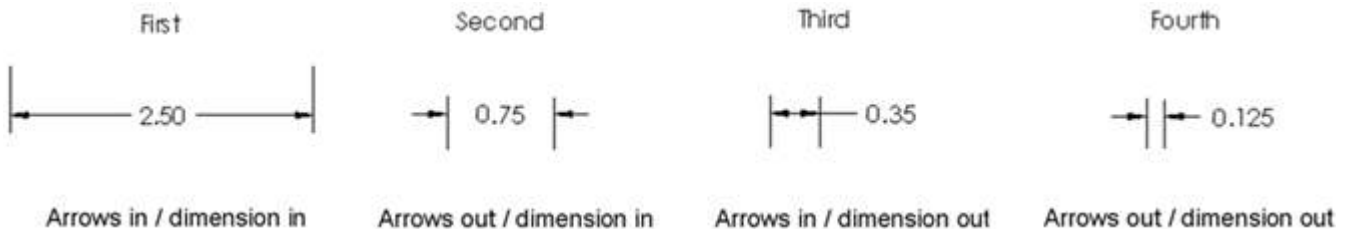
Dimension Lines and Extension Lines

Extension (or Projection) lines are used to indicate the extremities of a dimension. They are generally drawn up to 1 mm from the outline of the object.

Dimension lines are used to label a particular dimension. They have one or more arrowheads.



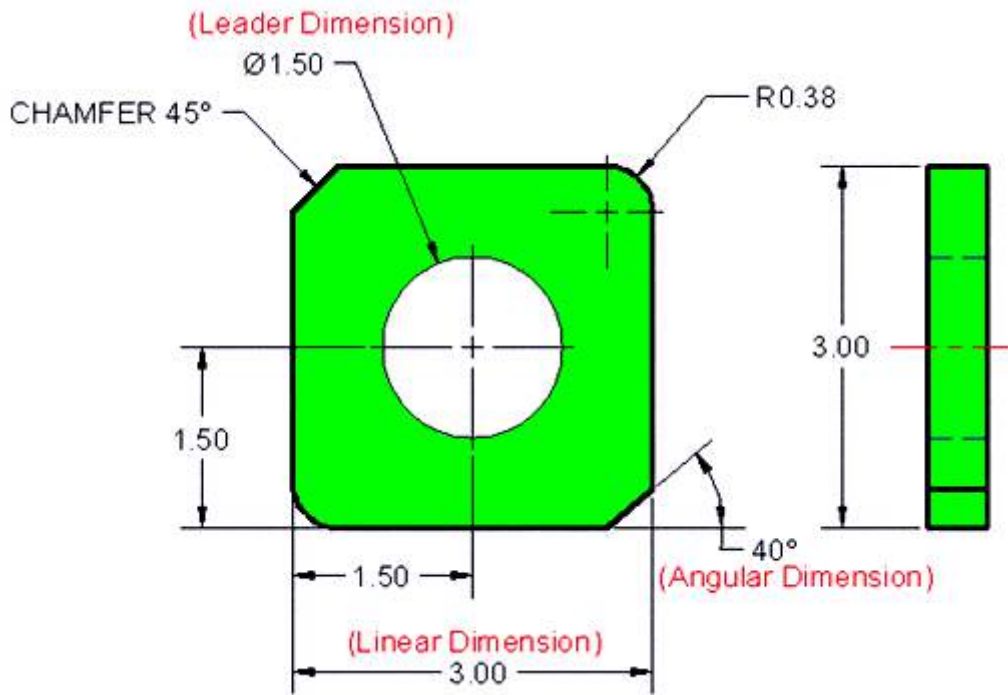
- Dimensions are usually placed between extension lines. But when there is no enough room to accommodate the dimension, either the dimension value or the dimension lines can be located the outside extension lines as illustrated.



Types of Dimensions

Dimensions may be divided into three different types; Linear dimensions, Angular dimensions, and Leader dimensions.

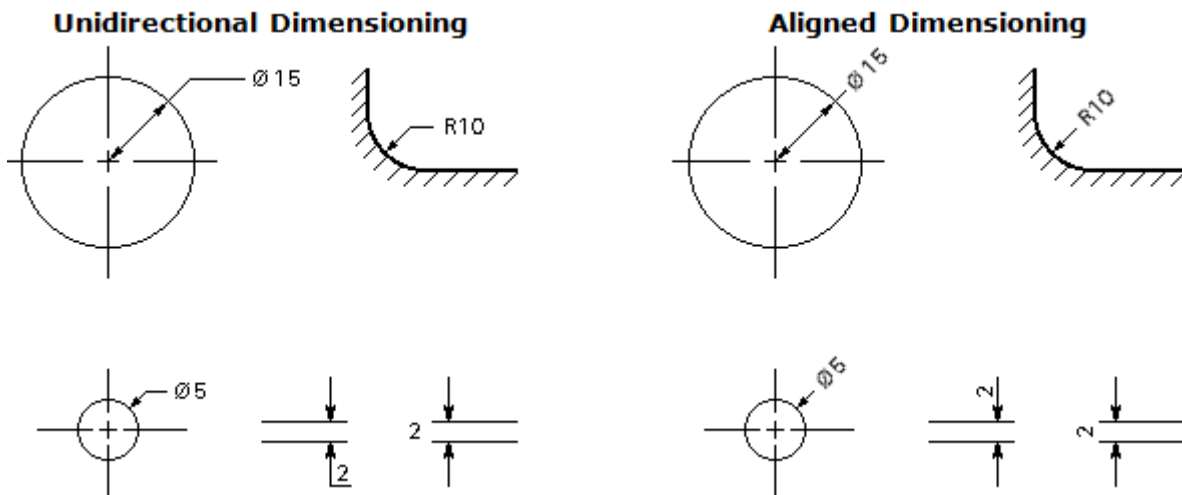
- **Linear Dimensions** - they are either horizontal or vertical to the dimensioning plane.
- **Angular Dimensions** - they are usually specified in decimal degrees (e.g., **27.5°**). Also they can be specified using degrees and minutes or degrees minutes and seconds (e.g., **27°30'** or **0°15'40"**).
- **Leader Dimensions** - they are usually used to specify a diameter or a radius where a leader line is used to point towards the feature being dimensioned.



Dimensioning Methods

Two methods of dimensioning are in common use.

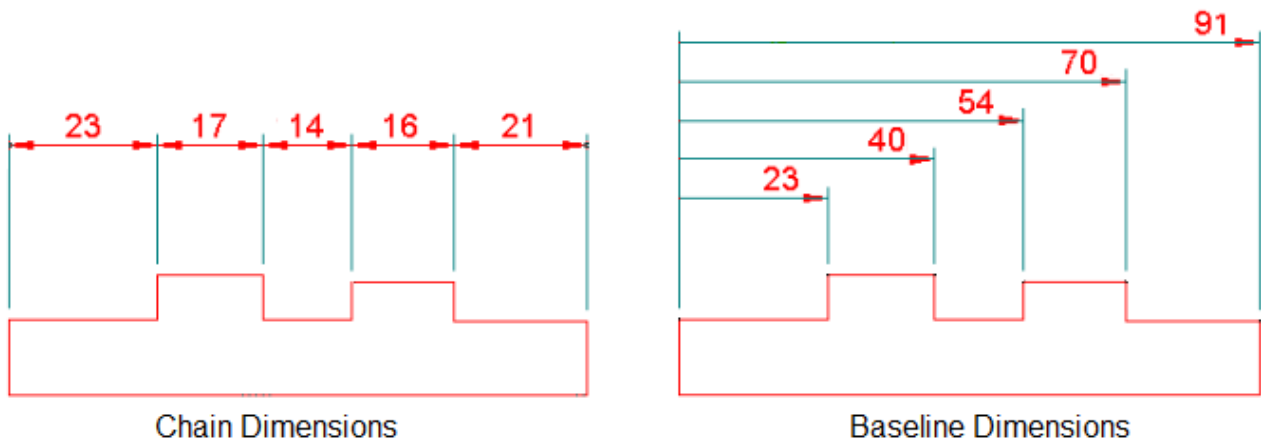
- **Unidirectional**, The dimensions are written horizontally.
- **Aligned**, The dimensions are written parallel to their dimension line. Aligned dimensions should always be readable from the bottom or the right of the drawing.



Chain Dimensioning and Baseline Dimensioning

Chain Dimensioning (dimensioning from feature to feature) is commonly used and easy to insert. Chains of dimension should only be used if the function of the object won't be affected by the accumulation of the tolerances.

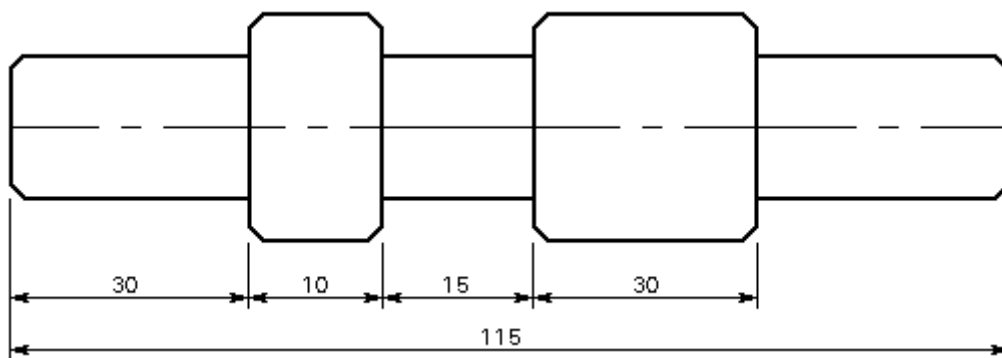
Baseline (or Ordinate) Dimensioning is used when the location of features must be controlled from a common reference point or plane. It is used to ensure that tolerances (or errors) in manufacturing do not add up.



Overall Dimensions

When several dimensions make up an overall length, the overall dimension can be shown outside these component dimensions.

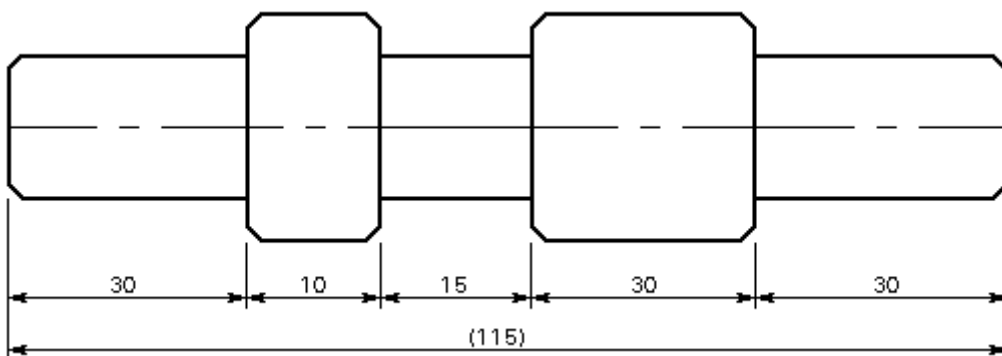
When specifying an overall dimension, one or more non-critical component dimensions must be omitted.



Auxiliary Dimensions (Reference Dimensions)

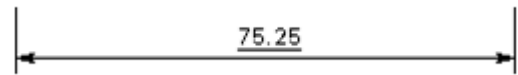
When all of the component dimensions must be specified, an overall length may still be specified as an auxiliary dimension (also called reference dimension).

Auxiliary dimensions are never toleranced, and are shown in brackets.



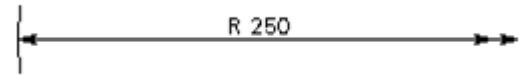
Dimensions not to Scale

Dimensions that are not to scale are underlined.



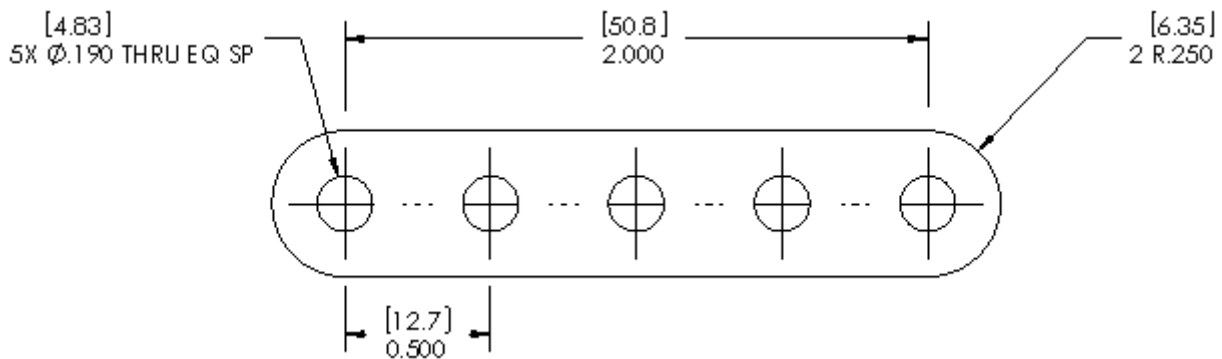
Dimensions not Complete

When a dimension line cannot be completely drawn to its normal termination point, the free end is terminated in a double arrowhead.



Dual Dimensions

Working drawings are usually drawn with all metric or all US dimensions. Sometimes the object manufacturing requires both metric and US measuring system and in such case dual dimensioning may be necessary. The optional secondary units are normally displayed in brackets [].



Dimensioning Rules

Some of the most important rules for dimensioning are as follows:





- Dimension figures should never be crowded or in any way that make them difficult to read.
- Each feature should be dimensioned.
- Dimensions should be placed outside the part when possible.
- Dimensions should be evenly spaced and grouped.
- Dimensions should not be duplicated or the same information given in two different ways (*except when dual dimensioning is to be used*) and no dimensions should be given except those needed to produce or inspect the part.
- Dimensions should be placed in the views where the features being dimensioned are shown in true shape.
- Dimensioning to hidden lines should be avoided wherever possible (*section views may be used to show the shape and dimensions of internal features*).

- The longer dimensions should be placed outside all intermediate dimensions so that dimension lines will not cross extension lines.
- Dimension lines should not cross, if avoidable.
- Dimension lines and extension lines should not cross, if avoidable (*extension lines may cross each other*).
- Extension lines and object lines should not overlap.
- Detail dimensions should "line up" in chain fashion.
- Leaders should slope at 45°, or 30°, or 60° with horizontal but may be made at any convenient angle except vertical or horizontal.
- Center lines or marks should be used on all circles and arcs.
- In general, a circle is dimensioned by its diameter, an arc by its radius.
- Cylinders should be located by their center lines.
- Cylinders should be located in the circular views, if possible.
- A cylinder is usually dimensioned by giving both its diameter and length in the rectangular view when possible.

Line Styles and Conventions

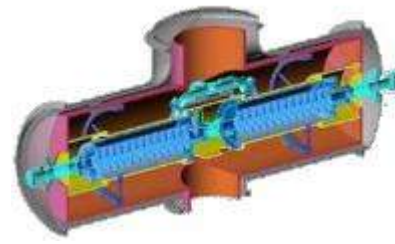
The use of different line styles and widths is important in technical drawings as they are used to indicate details and features of the drawing. Line styles make drawings easier to read where for example, solid lines used to show the object will stand out from dashed lines used to show hidden features.

The four main types of lines used in drawings are listed in the table:

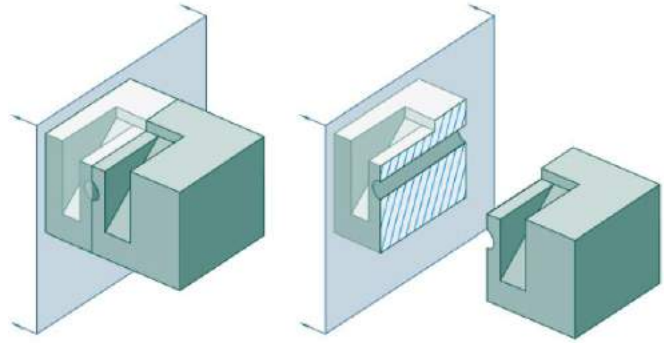
Line styles and conventions		
Thick continuous		VISIBLE LINES used on each view; includes arcs/circles/curves/title block and border
Thick dashes		HIDDEN LINES used on each view. Dashes start and end with contact to a visible or hidden lines
Thin continuous		THIN CONTINUOUS LINES used for dimensioning lines, projection lines, leaders, letterform used in title block.
Thin chain		CENTRE LINES, axis of solid forms, pitch lines (think roof line) Note: centre lines show symmetry

Section Views

Section views are used to reveal interior features of an object that are not easily represented using hidden lines in order to improve the visualization of parts or assemblies.

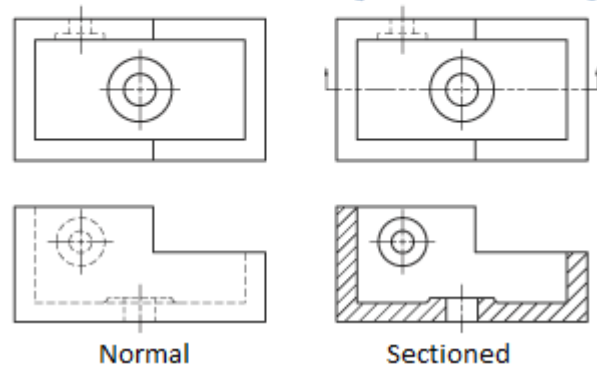
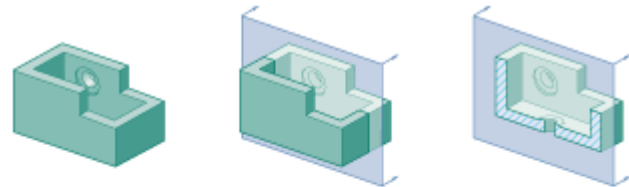


Traditional section views are based on the use of an imaginary cutting plane that cuts through the object to reveal interior features.



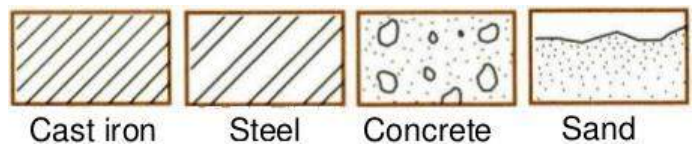
Section views are used in multi-view drawings in order to help in clarifying the drawings and to facilitate the dimensioning of drawings.

The figure shows a regular multi-view drawing and a sectioned multi-view drawing of the same part in the front view, the hidden features can be seen after sectioning.



- Hatching lines inclined at 45° are used to represent the new surfaces that are introduced by the section. The hatching lines spacing vary from 1.5mm for small sections to 3mm for large sections.

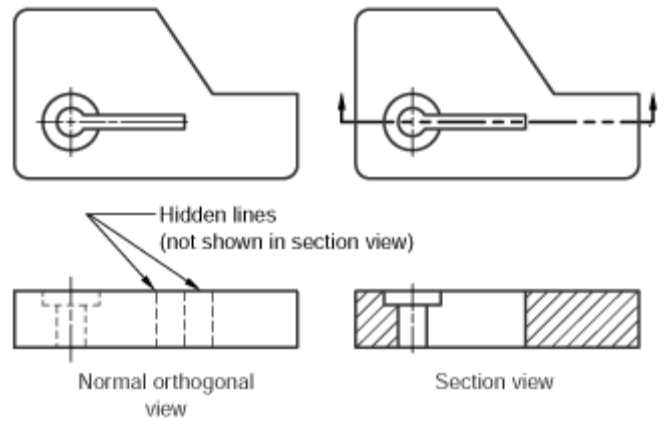
- In some cases, the hatching may be used to indicate the material where different standardized hatching patterns are used for identifying different materials.



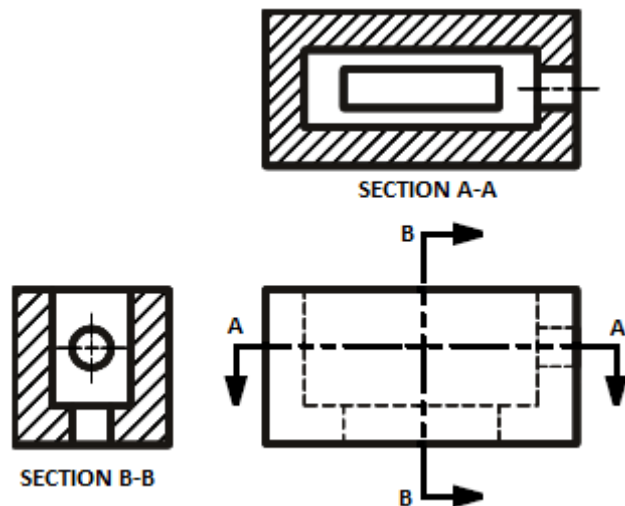
- The cutting plane is shown in orthogonal view as a cutting line (*a double dashed chain line is used*) with two arrows that point towards the portion of the object that will be kept. Sections are usually designated by uppercase letters.



- It is important to note that hidden lines should not be shown in section views, as seen in the figure.



- Multiple sections can be used in multi-view drawings when needed, as seen in the figure.

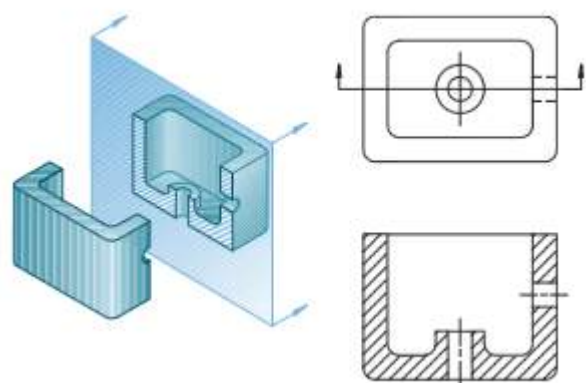


Section View Types

Different types of section views are used in technical drawing where the type of section to be used depends on the shape of the object to be sectioned and the internal features to be revealed by the section.

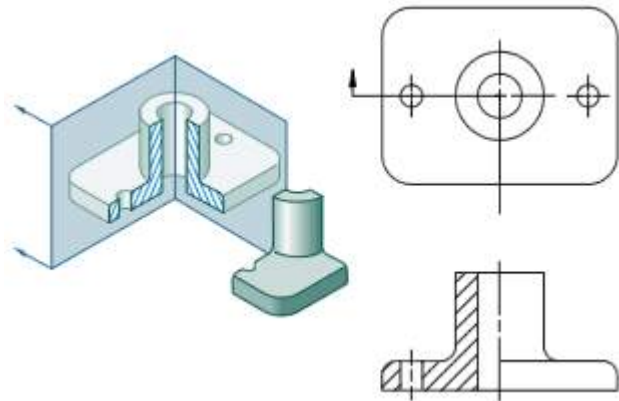
Full Section

A full section view is made by passing an imaginary cutting plane fully through an object. The figure shows an imaginary cutting plane passing fully through an object and half of it being removed. The section may cut the object in the middle or at any desirable location. In a multi-view drawing, a full section view is placed in the same position that an unsectioned view would normally occupy; that is, a front section view would replace the traditional front view.



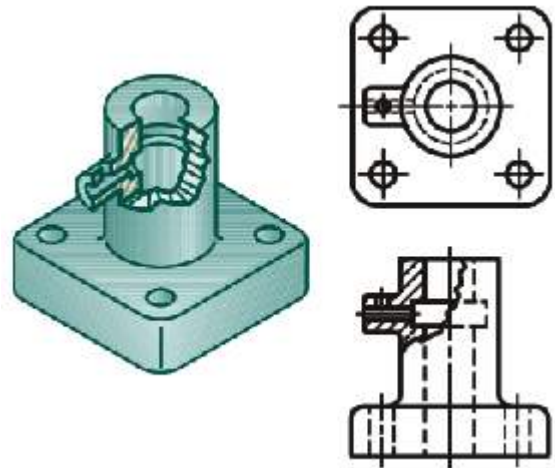
Half Section

A half section is made by passing two perpendicular imaginary cutting planes halfway through an object such that one quarter of the object is removed. Hidden lines are omitted on both halves of the section view. External features of the part are drawn on the unsectioned half of the view. A center line, not an object line, is used to separate the sectioned half from the unsectioned half of the view. Half section views are most often used on parts that are symmetrical, such as cylinders.



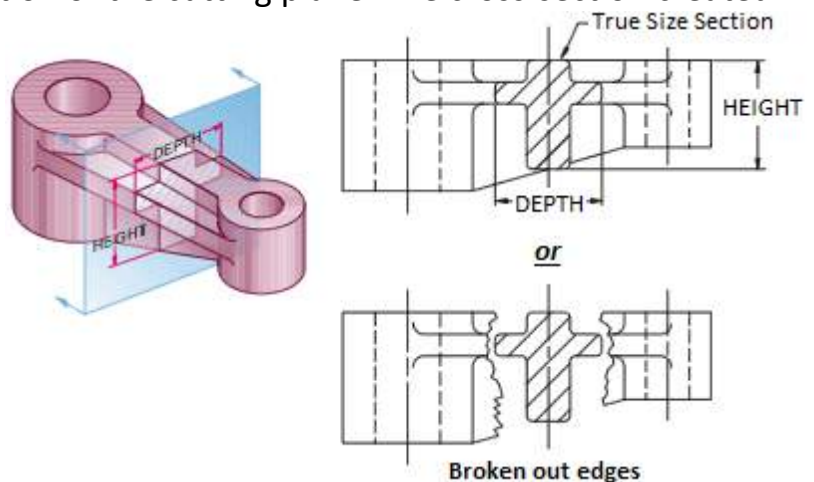
Broken-out Section

A broken-out section is used to show interior features of a part by breaking away some of the object. A broken-out section is used instead of a half or full section view to save time. A break line separates the sectioned from un-sectioned portion of the view. A jagged break line is drawn to represent the edge of the break. No cutting plane line is drawn for the broken-out section view. Unlike half sections, hidden lines are shown in the un-sectioned portion of the view.



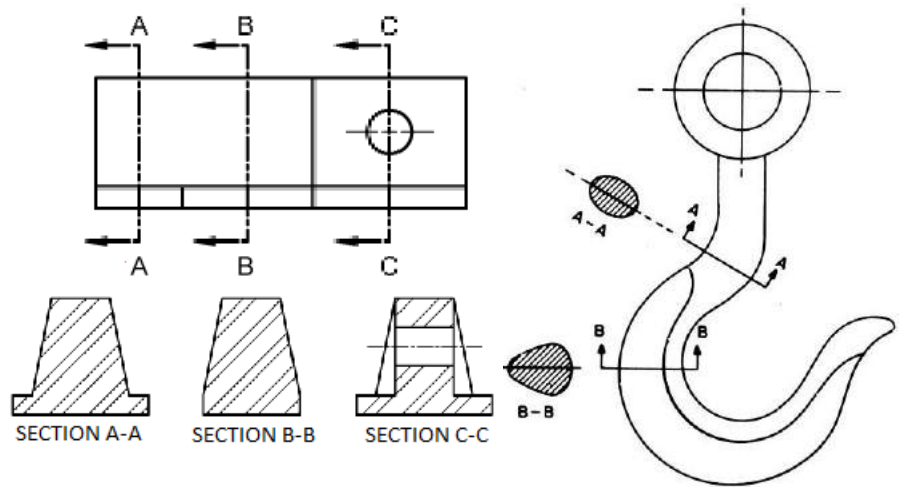
Revolved Section

A revolved section is made by revolving the cross-section view of the part about an axis of rotation placed at the location of the cutting plane. The cross-section created at the position that the cutting plane passed is revolved 90 degrees and drawn on the view. Visible lines adjacent to the revolved view can either be drawn or broken out using conventional breaks. No cutting plane lines are drawn for revolved sections because the section is shown in its actual location.



Removed Section

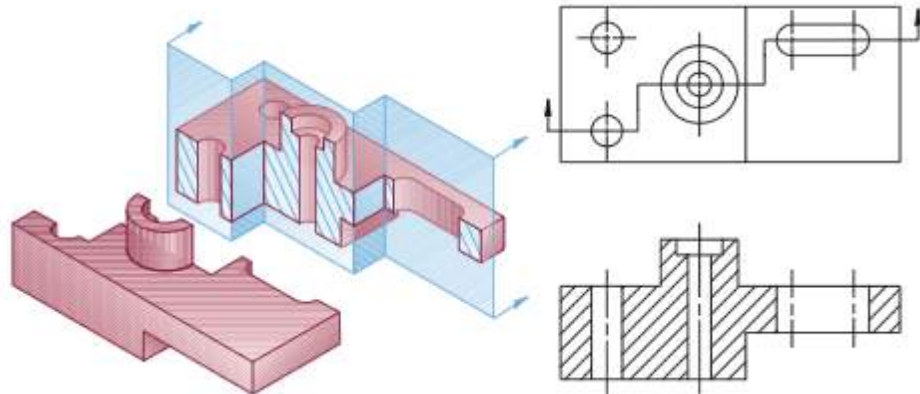
Removed sections are used to show the contours of complicated shapes such as wing and fuselage, blades for jet engines. Removed sections are made in a manner similar to revolved sections, by passing an imaginary cutting plane perpendicular to a part then revolving the cross section 90 degrees. However, the cross section is then drawn adjacent to the orthographic view, not on it. If a number of removed sections are done on a part, cutting plane lines may be drawn with labels to clarify the position from which each section is taken.



Offset Section

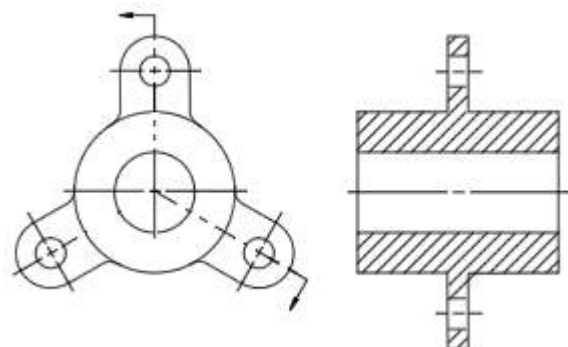
An offset section has its cutting plane bent at 90 degree angles to pass through important features. Offset sections are used for complex parts that have a number of important features that cannot be sectioned using a straight cutting plane.

It should be noted that the edges resulting from bending the cutting plane are not shown in the sectional view.



Aligned Section

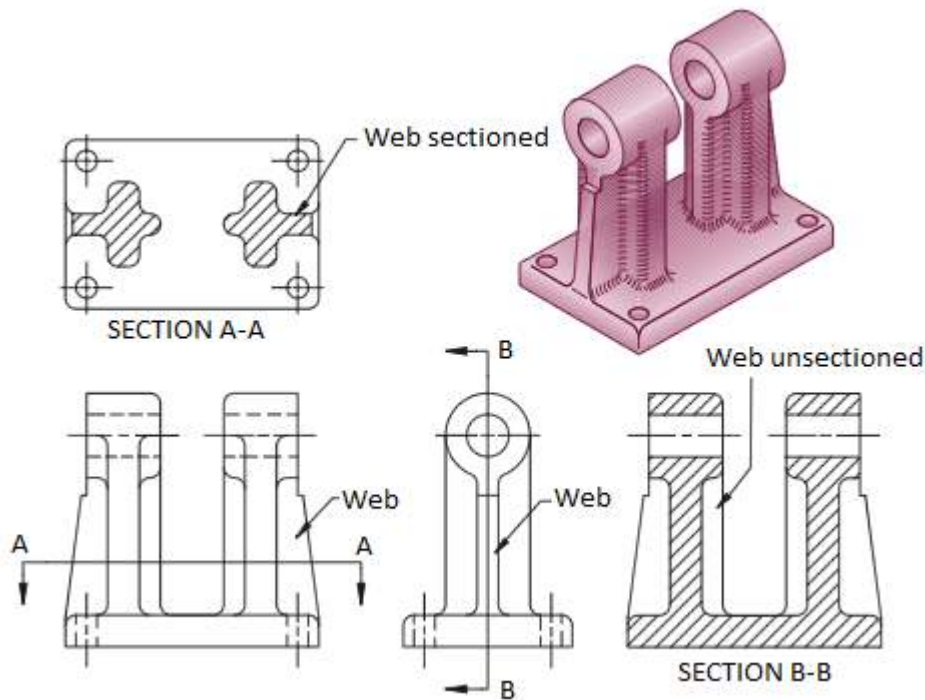
Aligned sections are special types of orthographic drawings used to revolve or align special features of parts to clarify or make them easier to represent in section. Two intersecting planes (the angle between them is more than 90 degrees) are used for sectioning. The sectioned surfaces resulting from the two cutting planes are aligned together along one plane. Normally the alignment is along a horizontal or vertical cutting plane.



Special Sectioning Conventions

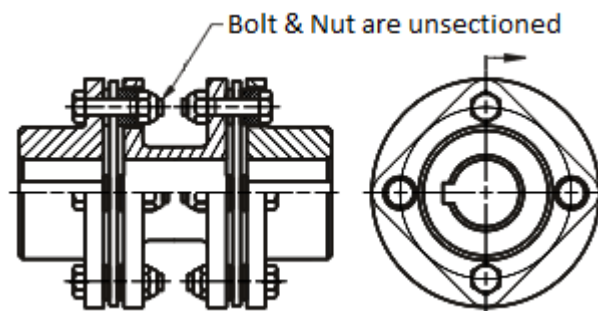
- **Ribs, Webs and Thin features**

When a section passes through a rib, web or a thin feature in general, hatching is not shown when the sectioning plane is parallel to the plane of the thin feature (*it will look as if the thin feature is un-sectioned*). However, when the sectioning plane is perpendicular to the plane of the thin feature, hatching is shown as usual.



- **Shafts, Bolts & Nuts, Keys, Pins, etc. in Assembly Drawings**

In assembly drawings, elements such as shafts, Bolts & Nuts, Keys, Pins, etc. are not sectioned even if the sectioning plane passes through these elements.

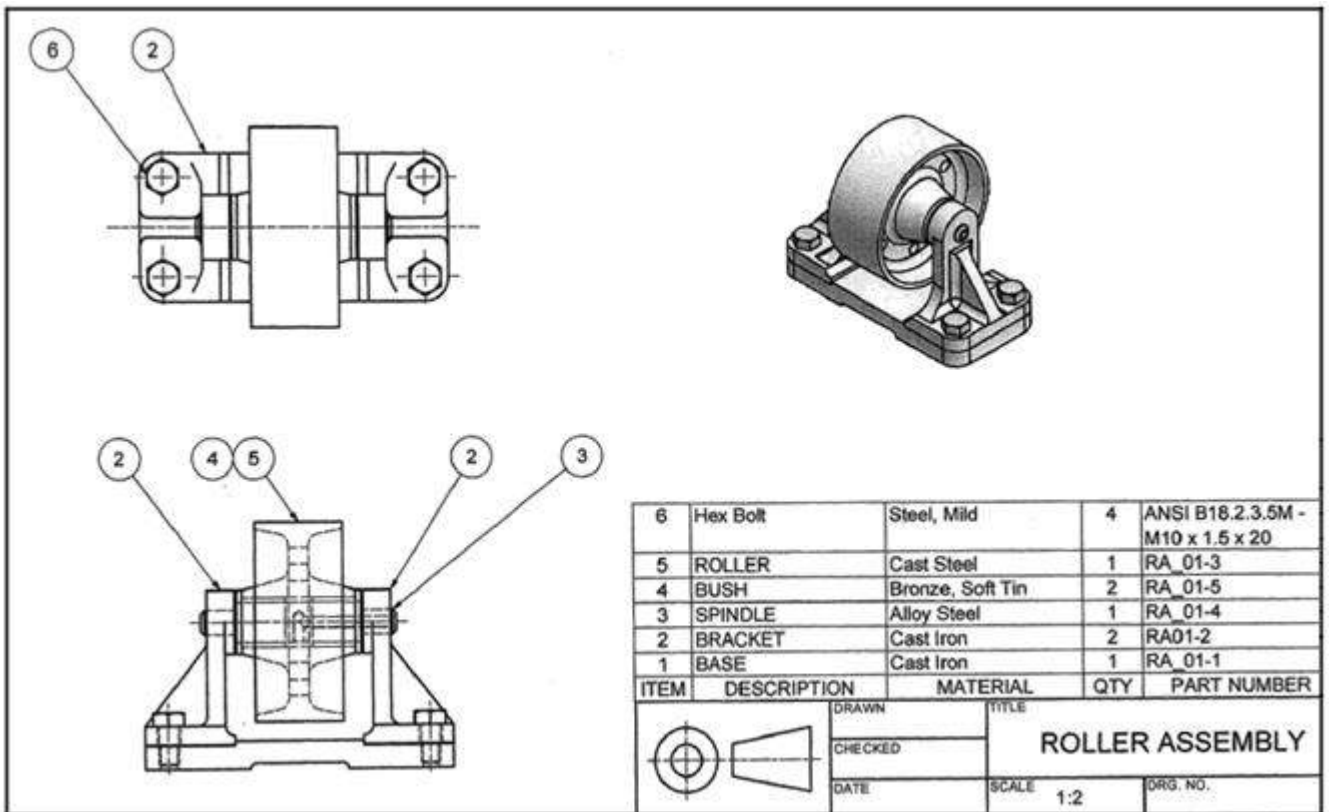


Assembly Drawings

Assembly Drawings demonstrate how a number of separate subassemblies, parts, standard components and specifications come together in a unified assembly.

Generally speaking, an assembly drawing is used to show fit and function, and verify how a product is put together. In order to fulfill its purpose, assembly

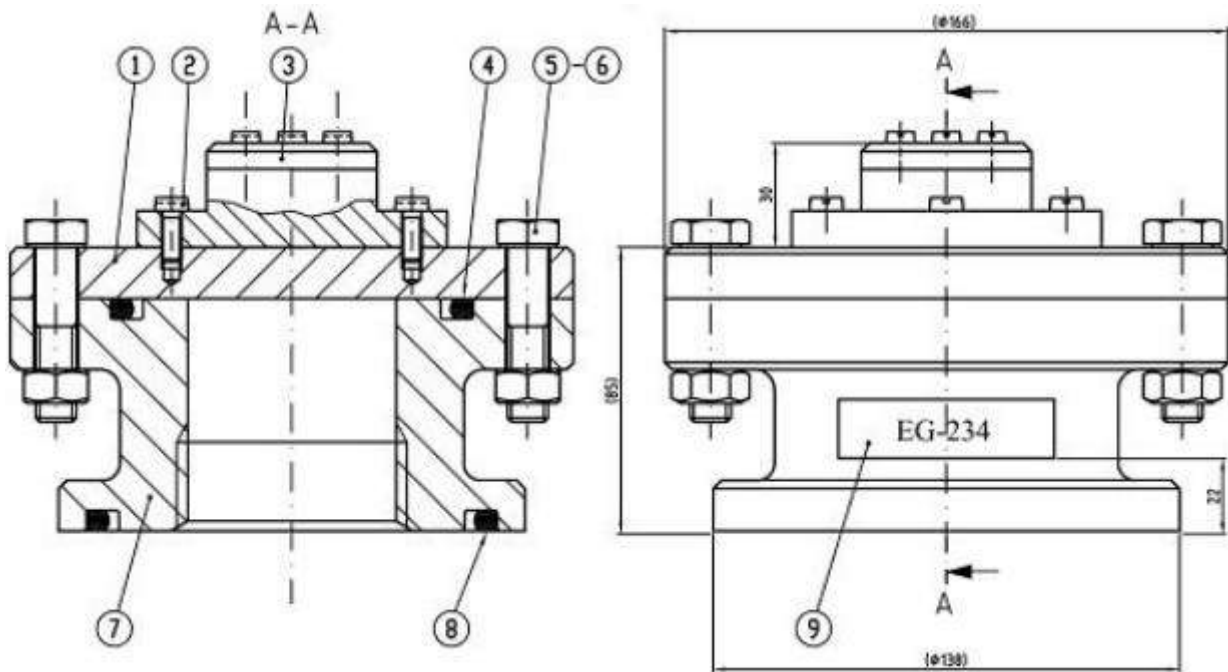
drawings must provide sufficient information to enable the assembly of a component.



The important features of assembly drawings are as follows:

- They must have a number of views to show how parts fit together.
- They usually have section views to show how parts fit and to eliminate hidden details.
- They typically have dimensions to indicate range of motion and/or overall size of assembly for reference purposes.
- They must have leader lines and balloons to identify Individual components.
- They must have parts list (or *BOM – Bill of Materials*) that is related to balloon numbers on drawing.
- Also, they may require multiple 3D-views (*in different orientations*) on a separate page for very large assemblies.

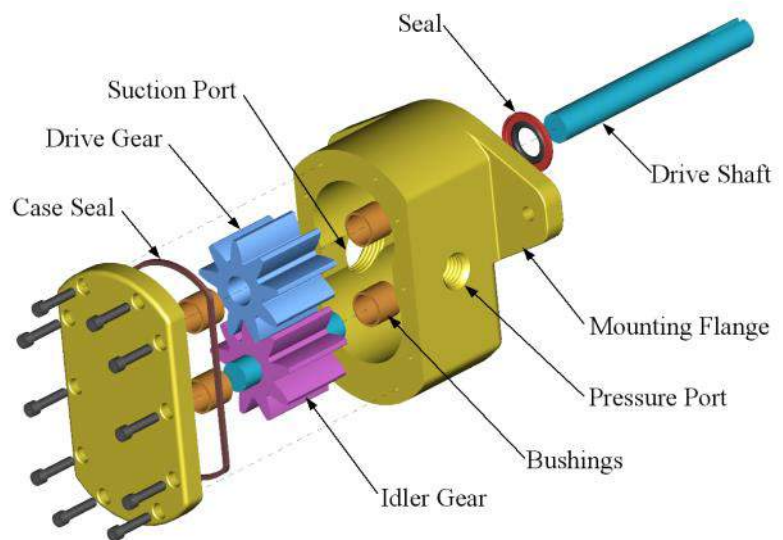
In the assembly drawing shown below notice how dimensions are used to give an overall representation of size (*dimensions are not included on individual components or standard components*). Detailed drawings with full dimensions are prepared for the individual parts. Also, it should be noted that, in the section view hatching oriented in opposite directions is used to make it easier to distinguish separate parts that are mated together.



Exploded Views

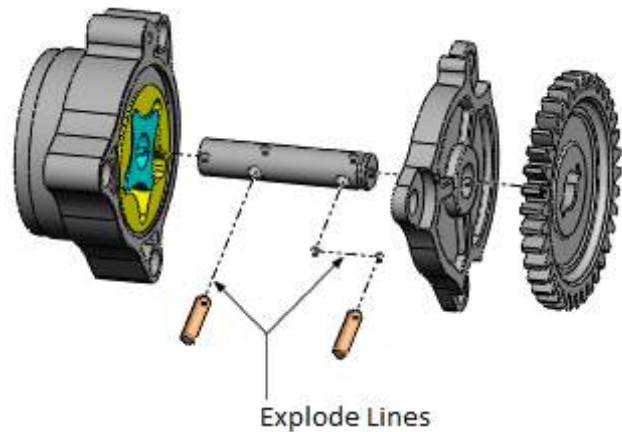
Exploded views are typically used in assembly drawings in order to show the relationship or order of assembly of the various parts. An exploded view shows the components of an object slightly separated by distance, or suspended in surrounding space in the case of a three-dimensional exploded diagram.

- In mechanical systems, usually the component closest to the center is assembled first, or is the main part in which the other parts get assembled. Exploded views can also help to represent disassembly of parts, where the parts on the outside normally get removed first.



- Explode lines are typically used in exploded views to help in visualizing the location of each component in the assembly. Explode lines connect each component in the exploded view to the position where this component fits in the assembly. The explode lines might be thought of as the path each component should move along in order to be placed in the assembly.

The use of explode lines becomes important for assemblies consisting of many different parts where parts have to be moved in different directions in order to make it possible to visualize all the parts of the assembly.



Numbering of Working Drawings

Any mechanical system or device generally consists of several levels of subassemblies. Each subassembly consists of several parts or lower-level subassemblies that are put together first then the different subassemblies are put together to make the main assembly.

In general, the working drawings for any mechanical system or device will include:

- Assembly drawing for the main assembly (*it will include several 3D and 2D views, section views, explodes views, etc. as needed and it can consist of several sheets*).
- Assembly drawings for each of the subassemblies.
- Detail drawings for each of the individual parts (*it should be noted that detail drawings are not needed for standard components such as bolts, nuts, keys, bearings, etc.*)
- It is extremely important to use a numbering system that assigns a unique identification number for each of the drawings. The numbering system should reference the various parent-child relationships between the main assembly and the various levels of subassemblies down to the parts level.
- One of the convenient methods for assigning numbers for drawings uses numbering system consisting of different levels where the number of levels being used depends on the number of subassembly levels. The general format of the numbering system is as follows:

XXXXX.AA.BB.CC.DD....

where,

XXXXX is the main assembly identification number

AA.BB.CC.DD.... are the identification numbers corresponding to the different levels of subassemblies down to the individual components level. The number of digits in each level can be two or more depending on the number of components at that level.

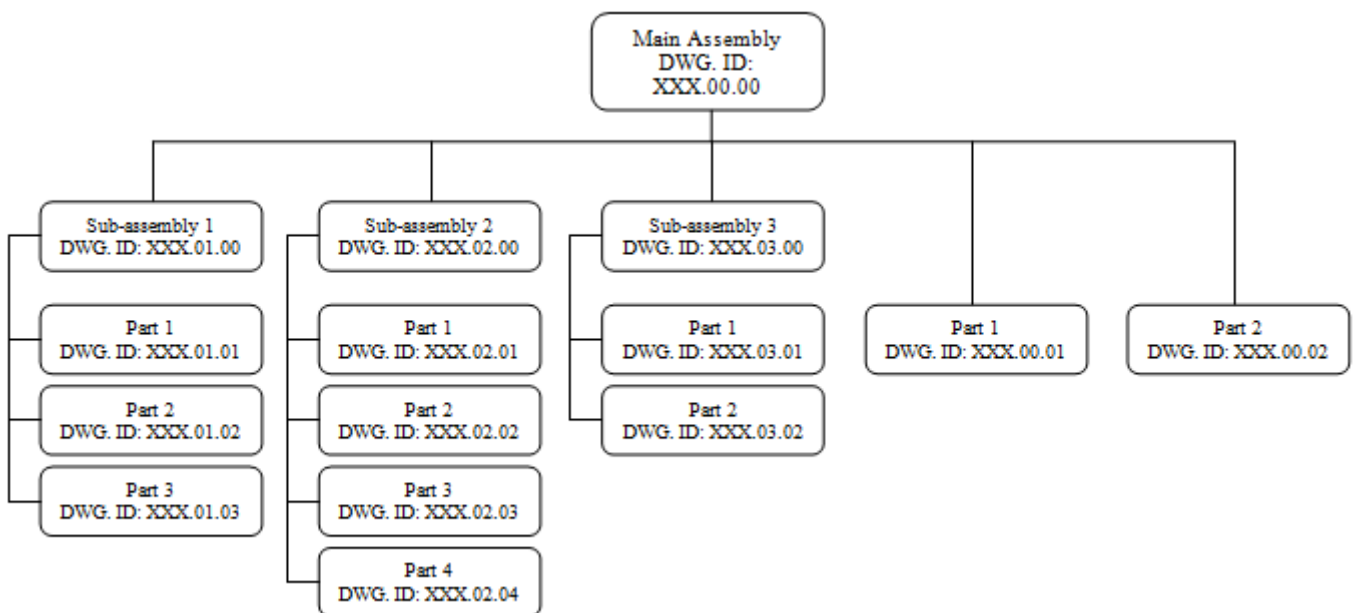
Example: Consider a main assembly that consists of three subassemblies and two main parts that fit directly in the main assembly. Each of the subassemblies consists of several parts as shown in the tree chart given below. Assign an appropriate number for each of the drawings.

Solution:

Here we have two levels below the assembly level (*subassemblies then parts*) therefore the numbering may take the general format: *XXX.AA.BB*

- Note that we consider the longest path down to the individual parts level in order to determine the number of levels in the numbering system.

The numbering of the drawings will be as shown in the chart:



where XXX represents the identification number for the main assembly.
