An Orthographic Projection drawing is a representation of the separate views of an object on a two-dimensional surface. It reveals the width, depth, and height of an object. There are six basic views of a of an orthographic multiview drawing: front, top, right side, back, left side, and bottom. The front, top, and right side are the three most common views drawn. Describing the shape of an object with a multiview drawing should always be done with the minimum number of views. The first view chosen to be drawn is the front view because it best shows the front of the object. The front view reveals the most unique and distinguishable shape feature of the object. All other views are drawn from the front view.
Above is an example of the different views of an object using orthographic projection.

In the image above you are shown the different views of a residential home using orthographic projection. The different views are called elevations in residential architecture.
Most all shapes in the world have three basics dimensions: Height, Width, and Depth. Hence the term 3 Dimensional or 3D.

The use of linetypes on a drawing are used to describe the various features of an object to the person reading the print. A line is the most fundamental and perhaps the most important, single entity on a technical drawing. Lines help to illustrate and describe the shape of objects that will later become real parts. The various lines used in a drawing are called "The Alphabet of Lines."
Object Line

**OBJECT LINE**

Object lines describe the visible surface or edge of an object. All visible surfaces of an object are connected using an object line.

Cutting Plane Line

**CUTTING-PLANE LINE**

Cutting-plane lines are used to identify where a section view is taken. A cut is made through the object where the cutting plane line passes through.

Hidden Line

**HIDDEN LINE**

Hidden lines represent an invisible edge on an object. Hidden lines show surfaces and edges that are hidden from the viewer's viewing plane.

Leader Line

**SPECIFIC NOTE**

Leader lines are used to point to a place on an object. Leader lines are usually used to point notes to specific location on an object. They can also be used to point to diameter and fillets.

Phantom Line

**PHANTOM LINE**

Phantom lines show alternate position of an object (also used for a cutting plane line). For example, a door handle when in a stationary position the door is closed, but when the door handle is in the alternate position, the door is open.

Section Line

**SECTION LINES**

Section Lines are used to show where the cutting-plane line has cut through material. Section lines are usually drawn at 45 degrees. There are different types of section lines, depending on the type of material being cut through.
**Dimension and Extension Line**

- **Dimension Line**: Indicates the length of the dimension.
- **Extension Line**: Shows the extent of a dimension.

**Center Line**

- **Center Lines**: Centerlines show the center of circles and radius.

**Use Of Lines Example 2**

- **Phantom Line**
- **Leader Line**
- **Center Line**
- **Hidden Lines**
- **Cutting Plane or Hatch Lines**
- **Object Lines**

**Use Of Lines Example 1**

- **Phantom Line**
- **Leader Line**
- **Hidden Lines**
- **Dimension Line**
- **Extension Line**
- **Hatch Lines**
- **Center Line**
- **Object Lines**

**Dimensions and Notes**

**Dimensioning**: is the process of defining the size, form and location of geometric features and components on an engineering drawing. Two general types of dimensions are used in drawings, size dimensions such as the size of holes and size of features such as widths and thickness, and location dimensions such as the location of holes. The purpose of a drawing is to convey to the workman the form, construction, and size of an object in order to manufacture or assemble it. Without dimensions, an object cannot be produced accurately.

There are two basic type of notes used in a technical drawing—"Local Notes" and "General Notes"—local notes pertain to a specific area of an object. General notes pertain to the drawing as a whole. Local notes are pointed to a specific feature with a leader attached. General notes can be located anywhere in the drawing. Some companies have standards as to where the general notes are located. Most general notes are located in the upper left corner of the drawing.
**Anatomy Of A Dimension**

A dimension consists of the following items: dimension arrow, dimension line, dimension value, extension line.

**Types Of Dimensions**

A location dimension locates holes or other part features. A size dimension gives a dimension of a radius, diameter, length, width, or thickness.

**Dimension Systems**

There are basically three types of dimensioning systems used in creating prints; they are as follows:

**U.S. System:**
ASME standards for the U.S. dimensioning use the decimal inch values. When the decimal inch system is used, a zero is not used to the left of the decimal point for values less than one inch. The same number of decimal places should be used for dimensions and tolerancing.

**Metric Dimensioning:**
ASME standards for the use of metric dimensioning require all the dimensions to be expressed in millimeters (mm). The mm is not needed on each dimension, but it is used when a dimension is used in a notation. Zeros precede decimal point when the value is less than one millimeters.

**Dual Dimensioning:**
Working drawings are usually drawn with all U.S. or all metric dimensions. Sometimes the object manufactured using both U.S. and metric measuring systems. Dual dimensioning may be necessary. When the object is to be manufactured is both measuring systems, a combination of both U.S. and metric dimensions are present.
**Leader Dimension**

A leader dimension is a leader line used to point toward a diameter or radius.

**Diameter Dimension**

A diameter dimension is a dimension that specifies a diameter of a circle.

**Radius Dimension**

A radius dimension is used to specify a radius.

**Angular Dimension**

An angular dimension is used to specify the amount of degrees between two lines that are parallel or colinear to each other.

**Linear Dimension**

A linear dimension is a dimension that is either horizontal or a vertical to the dimensioning plane.

**Hole Dimension**

Hole dimensions are used to denote drilled hole information by use a leader line pointing toward center of the hole.

**Notes**

Notes on a drawing provide added data and information not found in the views but are needed for the part to be completed. Notes should be read first before studying the views of the parts because they may advise you of certain requirements regarding the part. There are two kinds of notes: "local notes" and "general notes." Local notes are directed toward a specific location and connected with a leader. General notes apply to the drawing as a whole.

Notes:

1. FILETS .125 UNLESS OTHERWISE SPECIFIED
2. DEBUR ALL SHARP EDGES
Titleblocks

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

Sheet Sizes

<table>
<thead>
<tr>
<th>SHEET LETTER</th>
<th>SHEET SIZE</th>
<th>SHEET DIMENSION</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>11x17</td>
<td>11=A, 8 ½=B</td>
</tr>
<tr>
<td>B</td>
<td>17x11</td>
<td>17=A, 11=B</td>
</tr>
<tr>
<td>C</td>
<td>22x17</td>
<td>22=A, 17=B</td>
</tr>
<tr>
<td>D</td>
<td>34x22</td>
<td>34=A, 22=B</td>
</tr>
<tr>
<td>E</td>
<td>44x34</td>
<td>44=A, 34=B</td>
</tr>
<tr>
<td>F</td>
<td>40x28</td>
<td>40=A, 28=B</td>
</tr>
</tbody>
</table>

These paper sizes are standard USA sizes.

There are five basic sheet sizes used to create a drawing: A size, B size, C size, D size, E size.

Tolerances

Often when a part is being manufactured, it is least likely that we will get an exact size required in the design. We will get an accurate but not precise dimensions. For example, the diameter of a shaft in the design 2.00", but when the shaft is manufactured, the diameter may not be 2.00", it could be 2.02" or 1.98". If multiple shafts where to be manufactured at the same diameter (2.00"), we will have a series of dimensions that ranges from 1.99" to 2.02". These ranges are called tolerancing. Most of the time tolerances are determined by the way the parts are produced, e.g. a laser cutter will have better precision than a hole puncher. Depending on the application, some applications may require a higher precision and some may not. Higher precision parts would mean tighter tolerances, which cost more. Lower precision parts would mean less precision tolerances, which cost less. There are three basic types of tolerances: Bilateral and unilateral tolerances (Tolerance that can go both ways (Unilateral) or one way (Bilateral), Plus or minus tolerance (A tolerance that can go the same amount in the plus direction or the minus direction), Limits tolerance (Tolerance that specifies a maximum dimensional limit and minimum dimensional limit).
**Multiview Drawings**

A Multiview Drawing is a drawing with more than one view drawn. An object may need to have one or more views drawn of it to fully illustrate what the object actually looks like, to the reader of the print/drawing. For example a flat object may only need one view drawn of it to fully illustrate it, the thickness of the object can be a note placed somewhere on the drawing.

**One View Drawing**

Object consist of one view. Thickness note of 25. Know other views needed.

**Two View Drawing**

Object consist of two views. One orthographic view and one sectional view. Know other views needed.
Sectional Views

Sometimes it is difficult to show how the inside of an object looks like without using numerous hidden lines. To avoid this problem, a sectional view is used. The sectional view assumes that a part of the object has been cut by an imaginary cutting plane and a section removed to reveal its interior. A cutting plane line is used to show where the cut has been taken, it is terminated by arrows that show the viewer's direction of sight. There are four basic types of section.

Auxiliary Views

Some objects have slanted surfaces which cannot be clearly shown by means of conventional views. To accurately present the true shape of these surfaces, special views known as auxiliary views are needed. An auxiliary view is simply one that shows the outline of a slanted surface as it appears to the observer when looked directly at that surface. An auxiliary view is drawn parallel to the slanted surface.
Welding Drawings

Welding drawings are prints that describe several pieces that are welded together to make the finished part. This print shows where and what type of welds are to be made. Welding symbols are used to give welder information. Usually a materials list with all parts is included to make the final object.

Sample Drawing

Here you are shown a sample weld drawing of a bracket with included weld symbols.

Sheetmetal Drawings

Many objects, such as cardboard and metal boxes, cake pans, mail boxes, HVAC ducts, roof gutters, are made from flat sheet materials that, when folded, formed, or rolled, will take the shape of an object. Since a definite shape and size are desired, a regular orthographic drawing of the object is made first, then a development drawing is made to show the complete surface or surfaces laid out in a flat plane. Sheetmetal or surface development drawings are sometimes referred to as pattern drawings.
Dimensioning Flat Pattern Sheetmetal

Sheetmetal is usually dimensioned using the undirectional method of dimensioning. Dimension are read from top to bottom and from left to right. Under the undirectional method of dimensioning two types of dimensioning styles may be used, they are the linear dimensioning and datum dimensioning styles. The linear dimension style, dimensions are displayed horizontally and vertically on the object. In the datum dimensioning style all dimensions are taken from a 0 reference point and may be used by a CNC machine to drill and cut holes into the sheetmetal.

Working and Detail Drawings

A working drawing provides specific information and instructions needed to manufacture or construct products. Working drawings are used in all phases of manufacturing. There are to basic categories of working drawings, detail and assembly drawing. All important information must be contained on a working drawing or detail drawing. The person that needs to read these drawings, must not be forced to guess about any detail to produce the product or design. These drawings must be complete and correct with all dimension and notation added.

Working Drawings

A working drawing usually include fully dimensioned multiview drawings of the assembled product, plus detail drawing of each part and component.

Detail Drawings

A product may consist of a single part or contain hundreds or thousands of parts. Each part of an entire assembly will need to have a detailed drawing created to fully manufacture it.
Above is an example of an assembly drawing.

There are basically two ways of representing threads on a print: the schematic method and the simplified method.

The above image is an example of a thread callout on a technical drawing, with important information required to machine the hole.
Scales and Measurements

There are basically two types of scales used on prints:

Engineer Scale "decimal inch".
When drawings are created some typical scales are: 1/4, 1/2, Full (1=1) for mechanical drawings. For example a 1/4 scale drawing means the object is drawn a quarter inch its actual size on the sheet.

Metric Scale "millimeters": Drawings are drawn in millimeters instead of decimal inch. The conversion factor from millimeters to decimal inch is 25.4 divided by the decimal inch.

For example the scale of a drawing as noted on drawing sheet 1=2 (half) this means if this object were to exist in the real world, that 1" on the object would equal 2" on the drawing sheet. If you wanted to enlarge an object on the drawing sheet you would just reverse the values 2" on the drawing would equal 1" on the object.

The scale of a drawing is usually noted in the title block or below the view of the object that differs in scale to that given in the title block.

Rulers are used for linear measurement. The most common ruler in a machine shop are the bench and steel rule. Both edges of both sides are graduated. The rulers measure in "" (1.00), 1/4"" (.25), 1/8"" (.125), and 1/16"" (.0625) increments.

Linear Measurement and Conversion

Linear measure refers to measuring the straight line distance between two points. There are two measuring systems used, they are decimal inch and metric systems. Your shop work may require you to change back and forth between feet, and millimeters.

Converting feet to inches:
7 ft = 7 x 12 = 84 in
27.35 ft = 27.35 x 12 = 328.2 in
1 in = 1/12 ft = 0.0833 ft

Converting Inches to feet:
109 in = 109 / 12 = 9 ft. 1 in.
219.1/4 in = 219.1/4 / 12 = 18 ft. 3.1/4 in.
20.7 in = 20.7 / 12 = 1.717 ft

Converting Inches to Millimeters:
16 in. = 16 x 25.4 = 406.4 = 406 mm
93.375 in. = 93.375 x 25.4 = 2371.725 = 2371.725 mm
1/16 in = 1/16 x 25.4 = 1.5875 = 1.5875 mm

Converting Millimeters to inches:
1789 mm = 1789 / 25.5 = 70.4 in
93.32 mm = 93.32 / 25.4 = 3.69 in
360.5 mm = 360 / 25.4 = 14.19 in

Converting fractions to decimals:
21/100 = 0.21 / 3/100 = .021
divide the top "numerator" by the bottom "denominator"

Circles

A circle is a plane figure generated around a centerpoint. All circles contain 360 degrees. The diameter is the distance from circumference to circumference through the center.
An angle is the intersection of two lines going into different directions. Angles are measured in degrees, minutes, and seconds. There are 60 minutes in one degree and 60 seconds in one minute.

A triangle is a three-sided polygon with three interior angles. The sum of the three angles of a triangle is always 180 degrees.

A straight line is the shortest distance between two points. It is commonly referred to as a line.

Here are some standard abbreviations that you might see in a drawing. There are others but these are the most common. Try to commit them to memory. Abbreviations are used in mechanical drawings instead of writing out extensive long one or two line notes. The three reasons that abbreviations are used on a drawing is because they conserve space, promote consistency, and are easily recognizable. A symbol is a conventional representation of a quantity or unit. A symbol is preferred over the use of abbreviations when an option exists.

Abbreviations:
- COUNTERBORE
- X PLACES X OR BY
- Q CENTERLINE
- MATL MATERIAL
- BC BOLT CIRCLE
- TYP TYPICAL
- DEGREES
- Ø DIAMETER SYMBOL
- RADIUS
- THK THICKNESS
- PLCS PLACES
- CBORE COUNTERBORE
- CSK COUNTERSINK
- SP SPOTFACE
- CDRL CDRL