INTRODUCTION

Engineering drawing is a means of communication. "Communication" usually connotes writing and speaking, because they are more commonly employed in the course of everyday life. Writing and speaking, however, are insufficient to communicate design ideas. For one thing, the many languages around the world tend to inhibit written and verbal communication. It is more effective to transmit and receive design intent through drawings, sketches, pictures, graphs, etc. Graphic communication is clear, precise and unambiguous, conveying the same meaning anywhere in the world. This is increasingly important as the world develops into a global economy.

Both hand sketching and computer-aided design (CAD) are the two main tools to formulate and convey design intent. While CAD is an important tool for the engineering designer, hand sketching is still extremely important. The ability to clearly communicate design ideas on a blackboard or a piece of paper is an essential skill for an engineer. In the early conceptual stages of the design process, CAD can actually interfere with the creative flow of ideas.

Four aspects of engineering graphics are discussed in this chapter. First, the creation of pictorial drawings to show the three-dimensional (3D) form of objects is presented. The multi-view convention of engineering drawing is discussed next, followed by a discussion of the necessary dimensions and specifications that engineering drawings need to contain in order to portray clear communication of design intent. Finally, a broad overview of CAD, including an introduction to SolidWorks 98® software, is presented. For more in-depth coverage of engineering drawing, consult a comprehensive textbook [1].
PICTORIAL DRAWINGS

A pictorial drawing is a 2D illustration of a 3D object. Many people have a difficult time mentally assembling the three standard engineering views (front, top and side) to "see" an object; pictorial drawings effectively convey its form, bringing objects to life. Because pictorials are so easy to visualize, they are often used for catalogs, maintenance manuals and assembly instructions.

Three different types of pictorials are commonly used:

- Isometric
- Oblique
- Perspective

A simple cube drawn in the three different types of pictorials is illustrated in Figure 10.1. The isometric pictorial is drawn with its three axes spaced 120° apart. The term isometric means "equal measurement," indicating that the sides are all scaled by the same factor relative to their true length. Parallel lines defining edges on the object are also parallel on the isometric drawing. Drawing paper with isometric axes—available in good office and drafting supply stores—greatly facilitates drawing an isometric pictorial. Also, most 3D CAD programs generate isometric views automatically.

Oblique pictorials are drawn with the front view shown to true scale in the x-y plane. Oblique lines, which represent the z-axis, are projected at some angle, usually 30°- 45°. Parallel lines defining edges on the object are also parallel on the oblique drawing.

A perspective drawing represents most realistically what is actually seen. Artists draw or paint using perspective style. While engineers sometimes represent their designs in this style, it is the most difficult of the three types of pictorials to master. In perspective drawing, there is no well-
defined coordinate system. Parallel lines converge to a vanishing point as they recede from the observer. The use of converging lines instead of parallel lines and the foreshortening of dimensions gives the drawing perspective, but makes it difficult to scale accurately.

**Isometric Drawings**

In an isometric pictorial, the three axes are equally spaced 120° from each other, as shown in Figure 10.2. The axes divide the paper into three zones that represent three views of the object. If the axes make a Y, as in Figure 10.2, the view is downward toward the object. The top view is drawn in between the top branches of the Y. The left-side view appears to the left of vertical axis, and the front view is displayed to the right of this axis.

![Isometric axes equally spaced at 120° angles divide the drawing into three regions: front, top and left-side views.](image)

The isometric pictorial in Figure 10.3 was drawn on isometric paper, which provides evenly spaced lines parallel to the isometric axes to facilitate hand sketching, either freehand or guided by a straight edge. Also, Figure 10.3 depicts the three orthogonal views of the same object, with corresponding planes numbered in circles. A good exercise is to re-create the isometric view from the three views given.

![Isometric pictorial of a block with a step and tapered side.](image)
Circular features appear as ellipses in isometric drawings. Figure 10.4 illustrates the isometric pictorial of a right-circular cylinder of diameter D and height H. Note that the circular-top view appears as an ellipse that is tangent to the isometric axes at four points. The major axis of the ellipse is horizontal, and the minor axis is vertical. Figure 10.5 shows how circles in the top, left and front views appear as ellipses on an isometric pictorial.

![Isometric pictorial of a right-circular cylinder of diameter D and height H.](image)

Figure 10.4. Isometric pictorial of a right-circular cylinder of diameter D and height H.

![Circles appear as ellipses in an isometric drawing.](image)

Figure 10.5. Circles appear as ellipses in an isometric drawing.

**Oblique Drawings**

Oblique drawings are the easiest type of pictorial to draw by hand, but they are also the least realistic. Oblique and isometric pictorials are similar because both use parallel lines in constructing the three views. The difference between oblique and isometric pictorials lies in the definition of the axes. Oblique drawings use an x-, y- and z-coordinate system as shown in Figure 10.6. The three coordinate axes divide the sheet into three regions for drawing the front-, top- and right-side views. With the axes defined as shown in Figure 10.6, the object is viewed from above, looking from right to left. The z-axis, which is the receding axis in Figure 10.6, is drawn at a 45° angle relative to the x-axis; however, other angles such as 30° or 60° are often employed.
Figure 10.7 shows a simple oblique pictorial of a cube. The front view is drawn full size, which means that horizontal and vertical dimensions may be accurately scaled. The z-axis depth, however, is usually drawn at a lesser scale because drawing its true size creates an illusion of exaggerated depth. In this case, the receding z-axis is drawn at 30° to the horizontal. Figure 10.8 is a more complex oblique drawing of two connected blocks, along with the three orthogonal views of the same object, with corresponding planes numbered in circles. A scaling factor of ¾ is used for the depth for better realism.
Figure 10.8. Three-view drawing of a pair of connected blocks and an oblique pictorial of the same object.

Figure 10.9 illustrates how circles appear in oblique pictorials. Circular features in the front view appear as true circles, which facilitates hand drawing. A compass or circle template can be used to accurately depict the circular feature. However, circular features on the top or side planes would appear to be elliptical. Notice that only a portion of the circle formed where the hole intersects the back plane of the object is visible in Figure 10.9(b).

Figure 10.9. Oblique pictorials. (a) Right-circular cylinder (b) Rectangular block with circular hole.

**Perspective Drawing**

Prior to the Renaissance in the 15th Century, paintings looked flat and unrealistic. Then, artists discovered how to create drawings that represent on a 2D medium what the human eye (or a camera) sees. The role for perspective drawing in engineering is primarily to communicate to others how objects appear.
The major difference distinguishing perspective drawings from oblique and isometric pictorials is that in the latter, the lines defining edges (e.g., top and bottom) are parallel to the axes, whereas parallel lines converge in perspective drawing. In Figure 10.10, for example, the railroad tracks converge to a single point and the ties appear to get shorter, even though in reality the tracks are parallel and the ties are all the same lengths.

Figure 10.10. Parallel tracks converge and the ties appear to shorten as they recede in this photograph of railroad tracks.

Four terms are useful in describing perspective drawing:

1. **Picture plane**: the surface (i.e., the sheet of paper) of the pictorial. The edges of the paper represent the window through which you “see” the 3D object.

2. **Horizon line**: divides the sky and the land or the sea if outdoors. The horizon line is at the eye level and changes with elevation. In a room where the true horizon cannot be located because the walls block the view, assume a horizon line at about eye level.

3. **Viewing point and direction of view**: the location of eyes relative to the object. Objects can be viewed from left to right, right to left, downward, upward, etc. How an object appears in a drawing changes markedly depending on these parameters.

4. **Vanishing point**: the point at which parallel lines converge as they recede into the distance. The vanishing point is where the tracks appear to meet in Figure 10.10.

**One-Point Perspective**

An object can be represented using one-, two- or three-point perspective. In one-point perspective, the true width and height of an object in the front view are placed in the picture plane as illustrated in Figure 10.11. A horizontal construction line represents the horizon. The location of this line depends on the viewing point and the viewing direction. In Figure 10.11, the block is viewed straight on and from above. The elevation is taken into account by raising the horizon line. The vanishing point is centered on the horizon line because the view of the block is straight on. Construction lines are drawn from the vanishing point to the top corners of the block in the front view. The back edge on
the top view is drawn parallel to the front top edge to establish the depth of the block. Note that the back edge is much shorter in length than the front edge. The shortening of the lines on the recessed planes and the converging edges give the illusion of the third dimension.

Another example of one-point perspective is the drawing of a coffee table, as shown in Figure 10.12. The front of the table is drawn to scale in the picture plane. Coffee tables are low, so the viewpoint is chosen looking downward, but straight on toward the top surface of the table. The horizon line is drawn at eye level aligning the vanishing point with the center of the table. Light construction lines connect the corners of the table to the vanishing point. These construction lines define several triangles. The larger outer triangle is used to define the top surface, while the smaller triangles are used to draw the bottom edges of the legs visible under the table.

Two-Point Perspective

One-point perspective is useful when an object is viewed straight on so that its front view lies in the picture plane. However, if the object is rotated so that neither the front or side view is in the picture plane, as illustrated in Figure 10.13, a two-point perspective is required.
Chapter 10: Engineering Drawing

Consider a rectangular block that is viewed diagonally, so that only one vertical edge lies in the picture plane. Figure 10.14 shows the two-point perspective and its construction. First, an elevated horizon line is drawn because the viewpoint is above the block. Two vanishing points (VP-R and VP-L) are drawn; spacing these points at different distances reflects viewing the block from a slight angle. Then, a true-length vertical line is drawn (1), and its endpoints are connected to both VP-R and VP-L with construction lines. The two vertical surfaces can then be drawn (ABCD and ABEF). Finally, points D and F are connected to the appropriate vanishing points to define the top surface. Notice that in one-point perspective, two edges of the top cube face are parallel and two are not (Figure 10.11), while in two-point perspective, none of the top edges are parallel (Figure 10.14).

Figure 10.13. The direction from which an object is viewed controls the degree of perspective: (a) Straight on—use one-point perspective (b) Angle view—use two-point perspective.

Consider a rectangular block that is viewed diagonally, so that only one vertical edge lies in the picture plane. Figure 10.14 shows the two-point perspective and its construction. First, an elevated horizon line is drawn because the viewpoint is above the block. Two vanishing points (VP-R and VP-L) are drawn; spacing these points at different distances reflects viewing the block from a slight angle. Then, a true-length vertical line is drawn (1), and its endpoints are connected to both VP-R and VP-L with construction lines. The two vertical surfaces can then be drawn (ABCD and ABEF). Finally, points D and F are connected to the appropriate vanishing points to define the top surface. Notice that in one-point perspective, two edges of the top cube face are parallel and two are not (Figure 10.11), while in two-point perspective, none of the top edges are parallel (Figure 10.14).

Figure 10.14. Rectangular block shown in two-point perspective.

Figure 10.15 shows a two-point perspective drawing and the construction of the coffee table previously shown in one-point perspective. The front-most vertical edge is drawn true length, but the vertical lines which define the width and depth of the table are drawn “by eye” to obtain correct proportions for the table.
Three-point perspective is used when an object is very tall. Architects drawing a city view with skyscrapers use three-point perspective and taper the building as it extends into the sky. Engineers usually deal with smaller objects that can be represented in pictorials with either one- or two-point perspective. Therefore, three-point perspective is not described in this chapter, but Powell [2] provides an excellent in-depth description of perspective drawing.

**MULTI-VIEW DRAWING**

Multi-view engineering drawings are the standard format for communicating design details from the designer to the manufacturer. A multi-view drawing consists of as many 2D views of a part as are necessary to define it completely and unambiguously. A multi-view drawing is universal shorthand that contains all the information necessary to allow a part to be accurately and repeatedly fabricated. The ability to create and read engineering drawings is an important skill for design engineers. Even if the same person is designing and fabricating the part, which is often the case for student projects, an accurate engineering drawing is an invaluable tool for both design and manufacturing.

Consider a rectangular block with a slot, shown in an isometric pictorial in Figure 10.16. The arrows represent different directions of viewing the block, which isolate the front, side and top views. A three-view drawing of the block is shown in Figure 10.17. The single pictorial drawing of the block is now represented by three different 2D drawings representing the front, top and side views. These three drawings completely define the proportions of the block, its size and the location and size of the principle feature (the slot). Each view is accurately scaled.
The arrangement of the views is important because anyone reading the drawing will assume that this convention is being followed. A drawing that does not follow convention may be confusing. The front view is placed in the lower left-hand corner of the paper, the top view is directly above it, and the side view is directly to the right of the front view. This arrangement is called “orthographic projection,” allowing dimensions to be projected from one view to another. The width of the object is shared in both the top and front views, and the front and side views share the height dimensions of the block.

Figure 10.17 also illustrates another convention in engineering drawings; namely, solid lines are used to show lines that are visible in a view, and dashed lines represent edges that are hidden in that view. For example, all of the edges formed by intersections of planes are visible in the front and top views, so all the object lines are solid. However, in the side view, the line formed by the inside corner of the slot is hidden from view. Adding the dashed line to the side view clarifies the drawing.

Drawings differ from photographs by the use of dashed lines to indicate hidden features. Figure 10.18 shows the conventional meanings of different types of lines.
Next, consider a block with both a slot and a step, as shown in Figure 10.19. Figure 10.20 shows the three-view drawing for that part. The front view is drawn first, in the lower left-hand corner of the paper. Even with the step and slot, the outline of the block is shown as a rectangle in the front view. The lines defining the slot and step are added next, completing the front view. The top view is constructed next, based on the thin construction lines projecting upward, as well as the known dimensions of the depths of the block, slot and step. Finally, the side view is drawn, based on the information contained in the front and top views. In this case, projecting the top view to the right to intersect the 45° construction line, and then projecting downward helps to define the geometry of the side view. Notice the hidden line, which again shows the depth of the slot.

Figure 10.19. Isometric drawing of a block with a slot and step.

Figure 10.20. Three-view drawing of the block shown in Figure 10.19.
As a final example, consider the rectangular block with a step and a hole, shown in Figure 10.21. The corresponding three-view drawing is shown in Figure 10.22. The circular boundary outlining the step is evident only in the top view. Instead, the rightmost projection of the step shows in the front view as a single vertical line. Note the use of centerlines (long and short dashes) to define the location of the center of the hole in the top view. Centerlines are a form of shorthand which tell readers that the feature defined by the vertical hidden lines in the front and side views is a circular hole, as opposed to a square hole.

![Isometric pictorial of a block with a circular step and hole.](image)

![Three-view drawing of the block.](image)

**DIMENSIONING**

Multi-view drawings are drawn to a precise scale, so theoretically they contain adequate information about the size of parts. However, for a variety of reasons, an engineering drawing must also contain explicit numerical dimensions that precisely define its geometry.
Tolerances

No part can be manufactured exactly to specification, so designers dimension components so that they will function correctly when assembled into a system. Associated with dimensions are tolerances, the allowable variation in dimensions that still creates a functioning product. Tolerances can be stated explicitly; e.g., a dimension of $X = 2.500 \pm .001$ in. says that the part is acceptable as long as dimension $X$ lies between 2.499 and 2.501 in. It is important to realize that the tolerances are implicit, even if they are not explicitly stated. Table 10.1 gives commonly agreed-upon values for implicit tolerances based on the number of significant figures in the dimension value. CAD systems have the capability to change the decimal precision of dimensions to imply the correct tolerance.

As tolerances become tighter, the manufacturing cost increases significantly. Therefore, designers use tolerances that are as loose as possible, providing the product will still function correctly.

<table>
<thead>
<tr>
<th>Fractional Precision</th>
<th>Implied Tolerance</th>
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<tbody>
<tr>
<td>$X.X$</td>
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<tr>
<td>$X.XX$</td>
<td>$\pm .01$ in.</td>
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<tr>
<td>$X.XXX$</td>
<td>$\pm .005$ in.</td>
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</tbody>
</table>

Dimensions

Dimensions are important from two points of view. The designer of a new component starts with a blank sheet of paper and assigns dimensions that optimize the design of the component. In redesigning an existing component, the designer begins with the drawing (or CAD model) of the part and modifies existing dimensions as appropriate to refine the design. Dimensions are also critical for a person using the drawing, as s/he may manufacture the part, assemble the product that incorporates the part or repair the product. Many people not involved in the design may use the single engineering drawing that defines the part; therefore, precise definition of the part is critical.

Dimensions have been added to the part shown in Figure 10.23 to illustrate several conventions about dimensioning. Note that units are not specified on each dimension value. Rather, they are specified in the drawing block, described in detail in Figure 10.24. Dimension lines and extension lines (the lines to which the arrowheads point) are usually fine lines, as opposed to the heavier lines that define the object. In a CAD drawing, line weights can be set to follow this convention.

Holes and other circular features are located from an edge of the part to the centerline of the hole, because the drill that is used to make the hole enters the part at the intersection of centerlines.
But note that the center of a hole, after it has been drilled, is an imaginary point in space. The symbol \( R \) denotes the radius of an arc. Complete circular features (e.g., holes or cylinders) are specified by their diameters and indicated by the symbol \( \phi \). Circular features are typically dimensioned in the view in which they appear circular (e.g., the top view on Figure 10.23).

![Dimensioning example showing centerlines, leaders, and radius and diameter specifications.](image)

While it is important to completely dimension a part, over-dimensioning (specifying the same dimension twice) should also be avoided. Because of the tolerances associated with actually producing any dimension, having multiple dimensions can lead to a confusing situation in which one dimension is satisfied but the other is not. However, it is good practice to give the overall dimensions of a part (length, width and depth), because it clearly tells the fabricator the size of material with which to begin. For example, the depth of the block shown in Figure 10.23 can be computed by multiplying the radius by two (1.20 \( \times \) 2 = 2.40). But to avoid confusion, this information has been added to the side view. Redundant dimensioning is avoided by adding the symbol REF, which indicates to the fabricator that this is a dimension to be used only for reference purposes. The specified radius of 1.20 in. is the critical dimension in this case. If not otherwise specified, Table 10.1 indicates that the tolerance for this dimension should be held to \( \pm 0.01 \) in. A parametrically-based CAD program, like SolidWorks, automatically tells the designer if a part is under-defined, fully-defined or over-defined.

Dimensions can be placed arbitrarily on a drawing, and most dimensions could appear in at least two different views. The general rule is to locate dimensions so as to keep the drawing clear and uncluttered; dimensions should not be located on the part itself. CAD packages, like SolidWorks, that support dimensioning also allow them to be easily moved and/or modified (e.g., typeface style or size, dimensional precision, explicit tolerances, etc.) after they have been added to a drawing.

**Drawing Blocks**

The drawing block serves an important function on an engineering drawing by specifying many details that are necessary to define the part. As illustrated in Figure 10.24, the drawing block is
located in the lower right-hand corner of the drawing, just inside the border. Some of the information conveyed in a typical drawing block is common to all of the drawings produced by a specific company, and some is unique to the individual drawing.

Information commonly shown in the drawing block includes:

1. Name of the company issuing the drawing.
2. Name of the part that the drawing defines.
3. Scale used in preparing the drawing.
4. Tolerances to be employed in manufacturing the part.
5. Date of the completion or the release of the drawing.
6. Material to be used in manufacturing the part.
8. Units of measurement to be used in manufacturing the part.
9. Initials of the individual preparing the drawing.
10. Initials of the individual checking the drawing.
11. A drawing number that uniquely identifies the drawing.

Figure 10.24. The Drawing block summarizes essential information about the part.

CAD programs usually have pre-defined drawing blocks in which the designer can store information common to most drawings and easily add the information specific to each individual drawing. The specific format is not as important as the information the drawing block conveys.

**COMPUTER-AIDED DESIGN (CAD)**

Historically, engineering drawings were done by hand and in pencil so that they could be changed. Because of the requirements for precision, this usually involved many hours of laborious work with drawing instruments (e.g., T-square, compass, triangles, etc.) on a specially designed drafting table. Drawings were difficult to alter, which discouraged iteration to optimize a design concept. Most drawings were 2D multi-view drawings; realistic images of the 3D part were usually left to photographs or technical illustrators.
One of the first uses of computers as an engineering tool was to simply automate the drafting process. Early CAD software programs (e.g., AutoCAD) had the capability to precisely create and connect drafting entities like lines, circles and arcs, first in a 2D space, and then later in a full 3D space. The 3D models of parts that could be created were restricted to these basic elements which defined the edges and intersections of planes and were referred to as wire frame models.

Recent advances in computer hardware and software allow a part to be represented as a full 3D solid model, which is a 3D digital representation of the part, instead of just its edges (Figure 10.26). This offers tremendous advantages to the designer, including the capability of performing complex finite element analysis to predict the stress and strain inside the part or the flow of heat through it. Another advantage that solid models offer is the capability to be automatically produced by computer-controlled machine tools, known as computer-aided manufacturing (CAM).

Solid modeling changes the way designers work. The designer starts with a sketch, manipulates the sketch to form features, then builds parts from those features. The 2D multi-view drawings, which still play a role, are created last to document the end result of the design process.

SolidWorks 98®

Created by engineers from Parametric Technology Corporation (producers of ProEngineer®) and Autodesk® (producers of AutoCAD®), SolidWorks is mechanical design-automation software
that operates under the Microsoft Windows® graphical user interface. It is extremely powerful, yet relatively simple to learn if the designer has a working familiarity with Windows. This section presents a broad overview of the SolidWorks software package. To learn to use the software, consult the Tutorial [3] or User’s Guide [4]. A novice user can create the part shown in Figure 10.26 in less than 40 minutes by following the Tutorial.

A SolidWorks 3D model consists of parts, assemblies and drawings (Figure 10.27). Any changes made to one view are automatically updated in the other views.

- A part is a completely defined single component, made from a single material. The creation of parts is the first step in the design process.
- An assembly is a combination of parts that are connected together as they would be in the physical system. Parts are mated together in an assembly according to kinematic constraints that define how parts can move relative to each other. This allows fully mobile 3D mechanisms to be simulated, as well as checking for how individual parts will fit together in the actual assembly.
- A drawing is a conventional 2D multi-view representation of a part or assembly. It typically contains the dimensions and other information (material, heat treatment, etc.) necessary to actually fabricate the part.

![Figure 10.27. SolidWorks parts, assembly and drawings.](image)

Other useful capabilities supported by SolidWorks include:

- Creating parts by stamping and then bending sheet metal (Figure 10.28).
- Creating molds from solid models that could be used, for example, to mass-produce plastic parts by injection molding.
- Creating photo-realistic renderings of models.
- Using a design table, which allows an easy way to design families of parts with similar features, but different dimensions (Figure 10.29).
- Modeling assemblies that are fabricated by welding individual parts together.
REFERENCES
