

The Fundamentals of Design Drafting

A Student's Guide

By Daryll Smith

INTRODUCTION

Welcome to the Fundamentals of Design Drafting. The content presented in the Fundamentals of Design Drafting text is written to assist students in learning and developing a core knowledge of design/drafting and skill-building procedures. It provides an industry perspective of the basic concepts and principles that are used in the design and drafting industry. The content in the text is intended to help students begin preparing for the American Design Drafting Association (ADDA) industry recognized certification exams.

A student's success in this course is directly related to his or her ability to understand how to proceed in traditional and non-traditional class settings. It is imperative for students to:

1. Read all material carefully. Reread the material several times for total understanding. **DO NOT SKIM.**
2. Understand the concept of an activity before you start the process of typing commands on the keyboard.
3. The information you will type is presented in numbered sequence. Follow the steps carefully, watching the screen as you proceed.
4. Read the concepts again if you are having difficulty understanding a particular item.
5. Repeat the steps of an exercise over and over to develop mastery. Mastery means you are able to complete an exercise without looking at the book, and understand why you performed that particular function.
6. Ask questions if you do not understand or if you are having difficulty with the key strokes.

The Fundamentals of Design Drafting resources are flexible and instructors should feel comfortable supplementing curriculum resources that they have found successful throughout the years.

We welcome your suggestions, and hope that you will become part of the collaborative effort in educating our future engineers and architects.

ABOUT THE AUTHOR

Daryll Smith is a Principal Tool Designer for Medtronic, Inc. in Tempe, Arizona. Medtronic is the world leader in medical technology providing lifelong solutions for people with chronic disease. Medtronic's advanced technology and products are used to treat over 5 million people each year with conditions such as diabetes, heart disease, neurological disorders, and vascular illnesses. Mr. Smith is also an Adjunct Instructor for Glendale Community College in the Computer Aided Drafting department. Mr. Smith has over 35 years of hands-on repair experience in the Automotive, Aviation, and Marine fields along with over 17 years of experience in the medical tool design field.

Raised in Phoenix, Arizona, Mr. Smith started his CAD career by starting his own small business in 1990, Design CAD Systems, and soliciting contractual work as a Design Detailer while also consulting with design and manufacturing companies on how to use the CAD technology. Mr. Smith continues to teach various classes at the college and business level on subjects ranging from Mechanical Blueprint Reading, Jig & Fixture Designing, Descriptive Geometry, to Printed Circuit Board Design.

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CHAPTER 1

Basic Drafting Equipment

Throughout time, technical drawers use various methods and tools to draw and communicate their designs. Drawings can be made using a wide variety of media and plotting tools. The following chapter you will learn of some of the basic tools used that have helped drafters historically and today communicate their ideas.

Media/Reproduction

Throughout time, vellum was used to communicate important information such as documents and manuscripts because of its high quality and durable life. Vellum is a thin translucent material traditionally made from animal skins. Today, most vellum is made from cotton and is known as paper vellum. It is commonly used where tracing is needed and can be purchased at art or drafting supply stores. Drawings using vellum are produced in low quantity due to its high cost.



An English deed hand-written on fine vellum with large pine resin seal - dated 1638

Drafters also use polyester film for its durable properties. This includes high strength, dimensional stability, clarity, tear resistance, and moisture resistance. The film can be easily machined, and its surface is treatable. Many refer to this material as Mylar. Mylar is the DuPont trademark for polyethylene terephthalate (polyester) film. Polyester film is used in many types of packaging applications ranging from food and drugs to industrial and consumer goods. Engineers and architects create plans and draw-

ings to be plotted onto sheets of polyester film. The plotted sheets then become legal documents from which copies or blueprints are made. These sheets are more durable and can withstand more handling than typical bond paper..

A drafter completed drawing represents a tremendous amount of time. This is why the reproduction process is important to understand. There are various ways to reproduce drawings including blueprinting, diazo, microfilm, photocopying, etc. Each has its pros and cons. For example, in blueprinting the original must be transparent. Blueprinting is one of the older reproduction processes and is frequently used today because of its low cost. Blueprinting uses various chemicals that produce white lines on a blue background. The blueprinting method allows an unlimited number of prints to be made without quality degradation. This ensures highly accurate dimension information on all copies. The process that a blueprint goes through provides durability to prevent fading, marking, and alteration.



Architectural Blueprint

The common blueprint in some cases was replaced by lower cost alternatives. One of these alternatives was diazo prints or white-prints. Diazo paper is a combination of two chemicals coated on paper to create blue lines with a white background. These prints are also called bluelines. Another alternative is photocopy printing. Photocopying print technology is basically the copy machine technology using toner on bond paper. The term “blueprint” has continued to be used informally to refer to any of the methods of media reproduction.

Drafting firms today invest in the technology of large format plotters and scanners. Technology has taken a great hold in drawing storage, retrieval and production. Drawings that are created with computer aided drafting can be digitally stored on a computer's hard drive or some other storage device. Pencil, blueprints, and other drawings can be scanned, stored, and printed using scanners and large format printers. These large format printers use ink and toner cartridges to print or plot the drawings. Toner is a powder used in laser printers and photocopiers. The toner forms text or images by melting toner particles onto paper, which allows them to fuse together. Cost of toner and ink vary depending on the toner cartridge size and the quality of the ink. Logically the more toner or ink needed, the higher the production cost of the print.



hp designjet 820 mfp

The technology found in today's scanners and large format printers have helped drafters become more proficient in communicating and storing their drawings. Printers and plotters provide drafters reproduction speed, accuracy, and repeatability. As technology continues to evolve and become more affordable, drafters will continue to take advantage of innovative reproduction methods.



KIP 3000

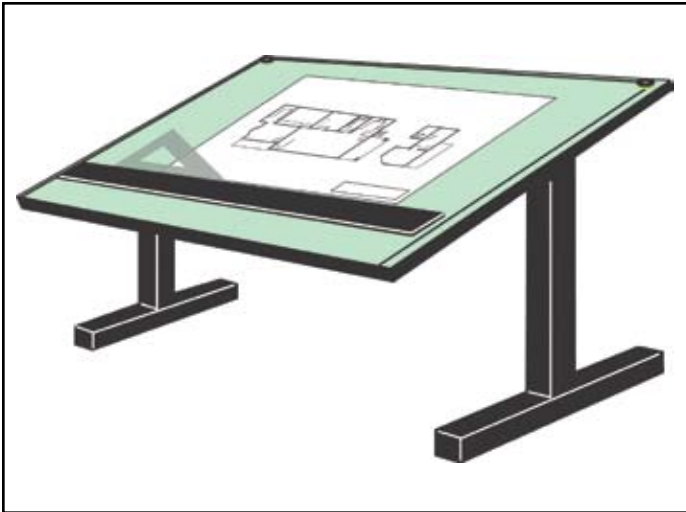
1. Operator panel with touch LCD display
2. Integrated copy and scan
3. Guides for engineering and architectural standards
4. Print Engine - 100% toner efficiency
5. Technical LCD - Provides technical information for system status
6. Media Capacity - One or two rolls, field upgradeable
7. Image Processing System - ensures high quality and fast processing
8. Integrated Stacking - front or rear output stacking for up to 200 prints
9. Small Footprint - fully integrated, space saving device

Drafting Tools

Drafters use a number of drafting tools that assist them in drawing their designs. This section will provide a brief overview of some of the well known board drafting tools that are being used today.

Drafting Board

From artists to technical drawers, drafting boards provide an area and surface for these professional to draw. Drafting boards come in many sizes and can be customized to fit the need or desire of the drafter. In general, a drafting table has a large rectangular surface that can accommodate the size of the paper being used. The table typically has a plastic covering which must be very smooth to avoid any imperfections during drawing. Drafting tables require a smooth straight edge for a T square or other rulers to be utilized. Although computer aided drafting has become popular, many graphic designers, artists, and architects still rely on paper and pencil/pen graphics that require the use of a drafting table.



Typical Drafting Table

T Square

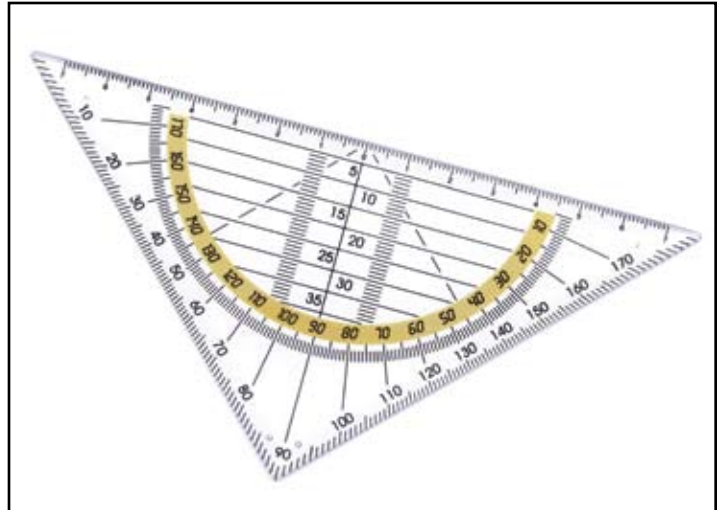
A T square tool is shaped like a letter T and has two parts, the head and the blade. The head is in a 90 degree alignment with the blade. When used with the drafting table, the 90 degree alignment allows drafters to draw horizontal parallel lines across their sheet. The T square is used in connection with other drafting tools (triangles, protractors, curves, etc) to help guide and draw accurate angles and shapes. An alternative to a T square, the straight edged or parallel rulers that are attached to the drafting board which slide up and down provide the same purpose as the T square.



Basic T Square

Triangles

This tool is triangle shaped and comes in both 45/45 and 30/60 degree angle patterns. With the T square, triangles are used to draw lines at 15, 30, 45, 60, 75, and 90 degree angles.



45 Degree Triangle

Protractors

A protractor can be a half or full circle that is divided into degrees. The protractor is used to measure angles from 0 to 360 degrees.

Compass

The compass tool is used to draw circles and arcs. The compass is comprised of two legs, one with a point, the other with lead/ink and the two are connected by some type of adjustable hinge. The compass point is set at the center mark of the desired circle or arc and the lead is then pivoted around that point. A divider tool looks similar to the compass, but the divider has two points and no lead/ink. The purpose of the divider is to measure the distance from one point to another.



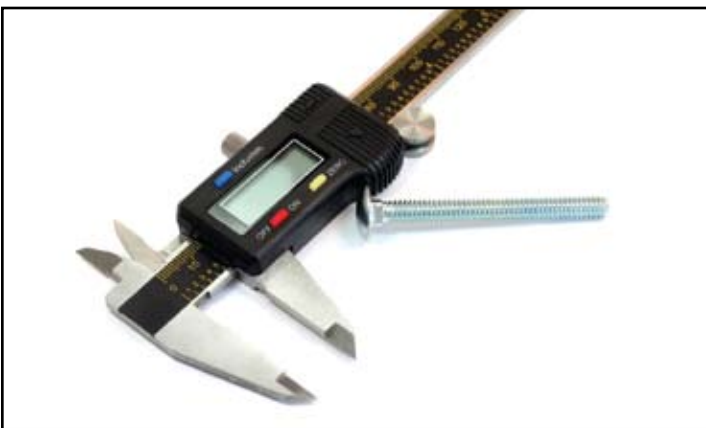
Protractor & Compass



French Curves

Caliper

A caliper is a device used to measure the distance between two opposing sides and can be adjusted to fit across the points to be measured. A caliper is frequently used to measure the dimensions of smaller objects. Calipers can accurately dimension internal and external components within 0.001 cm.



Digital Caliper

Curves

A curve is a thin plastic or metal tool that has a variety of curve shaped edges. The curve tool is used for irregular curves that can't be drawn free hand or with a compass. The most common curve is the French curve. There are templates other than the curve that drafters can use to produce various shapes.

Lettering Guide

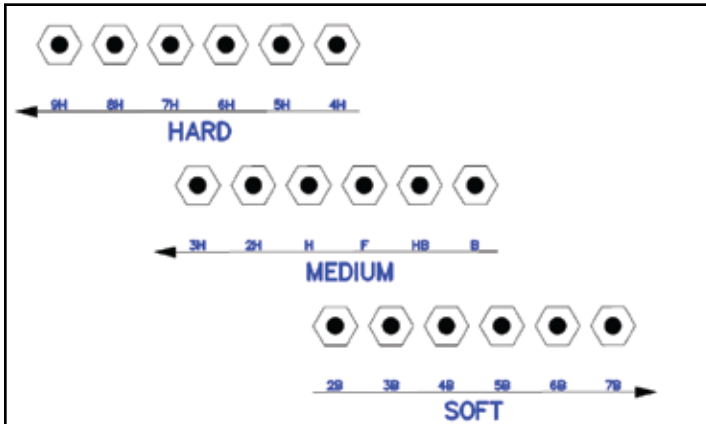
It is sometimes said that a good drawing can be ruined with bad lettering. A lettering guide is a tool that is used to properly space lines for lettering. Standards are set forth by the American National Standards Institute to provide guidelines for drafters to correctly letter. These standards lay a foundation for drafters to communicate details in their drawing. The lettering guide helps drafters' accuracy and efficiency comply with these lettering standards set forth.

Pencils & Pens

A drafting pencil is an important tool of the traditional drafter. Not any old pencil will do; drafting pencils are high quality and very sharp to draw precisely drawn lines. Drafting pencils come in a variety of pencil lead grades.

The grade of hardness of pencil lead is indicated on the pencil by a number-letter code. Leads range in hardness from very soft (7B) to very hard (9H). Soft lead pencils produce a dark line yet can be smudged easily; hard lead pencils do just the opposite. The grade of pencil lead required depends on the function to be performed. For example, if an architect is lettering or drawing construction lines where accuracy is important, a hard pencil such as 4H, 5H, or 6H lead is required. Softer, 4-6B pencils are used for larger quick sketches or figure studies.

Drafting pencils are manufactured as wooden pencils, lead holders, and mechanical pencils. Wooden pencils are created with wood and loaded with one of the various grades of lead. Lead holders allow the drafter to swap lead grades as needed and have any amount of lead exposed and sharpened to any size.



Pencil Grades

A mechanical drafting pencil is built to hold only one size of lead. Mechanical drafting pencils can support lead in diameter sizes of 0.3mm to 0.9mm and can come in the numerous lead grades.

When drawings are to be made with ink, drafters use technical pens. Civil drafters, for example, in fact some drafting fields such as civil require that the drawing is made of ink. One of the important reasons for using technical pens is because they reproduce well and can be filed on film. Technical pens also offer sharp lines without the hazard of lead smudges. An important reason for using technical pens is because they reproduce well and can be filed on film. Technical pens vary in line width to provide different line sizes and sharpness.

Technical drawing ink must be of high quality and be fast drying to ensure reduced smearing and efficient work flow. Ink should be semi flat black in color to avoid light reflection, and must not crack, chip, or peel from the drawing. Drafters must also determine if the ink should be waterproof or erasable for corrections.



Erasers & Shields

The use of erasers is important to drafters as they edit, erase, and remove unwanted lines or letters. Erasers come in all shapes and sizes but there are a few which are most widely used. The most commonly used pink eraser is found on the typical wood pencil. The pink eraser is good for general erasing purposes, but often smears and can damage the paper if pressed to hard. Plastic erasers are another type, and they are the most often used in technical drawing for plastic lead or ink on polyester film. Plastic erasers are usually white in color and come in block or pencil shapes.

Drafters also use other tools to assist in the erasing or removing of discarded marks or lines. One of these tools is called an erasing shield. An erasing shield does exactly what its name implies; it protects the rest of the drawing from being erased. Shields are usually thin sheets of plastic or metal that has holes or other shapes to guide the eraser.



Eraser & Shield

The shield is laid flat on the drawing with a specific hole or shape over the desired area to be removed, and then the eraser is applied. Another tool used is a dusting brush to easily remove left over eraser particles to reduce the possibility of smearing or smudges. For ink, drafters use different types of eradicating fluids to remove unwanted ink and film from the various drawing media.

CHAPTER 2

Basic Measurement Systems

A unit of measurement sets forth a standard to quantify a given physical property. For this purpose drafters must adhere to measurement standards to be able to communicate their ideas and drawings for others to comprehend. Common Systems of Measurement include the following:

- Length
- Area
- Volume
- Angle
- Mass/Weight
- Power
- Force
- Acceleration
- Temperature

Drafters use various scales to stay within the standards as set for in two different measurement systems English/Imperial System or Metric System. In the United States the English/Imperial system or U.S. customary units refers to the measurement standards used primarily in the U.S. The metric system is also known as the International System of Units, is published by the International Standard Organization (ISO), which is the measurement system that the majority of countries have adopted.

There have been efforts to move to the metric system but these customary units of measurement have a strong hold due to the size and growth of the U.S. in industry and the high financial costs to convert the existing infrastructure to metric. U.S. companies who do international business are more likely to use the metric system due to international standards. Because of this metric system is preferred in certain fields such as science, medicine and technology. It is critical for drafters to have an understanding of these two measurement systems due to their wide use.

Scales and Scaling

Utilizing a scale is an important part of being a drafter and making the appropriate use of measurements. Scales help a drafter draw a drawing “to scale”, which means that they can accurately portray a physical object, on paper or on screen, by enlarging or diminishing the scale of the object. Architectural and civil drawings are usually scaled much smaller, but some small mechanical drawings must be scaled larger to be illustrated on a standard paper or in a CAD drawing for precise measurements. The size of the scale used also depends on the amount of detail or the amount of information that a drafter needs to correctly describe the design.



Typical Scales Used by Drafters

Scales can either be open divided or fully divided. Open divided scales do not include the subdivided scale marks only the main units, such as inch and half inch. Fully divided scales are marked or subdivided to include the smaller units like one sixteenth (1/16”).

Depending on what is being drawn and why it is being designed, drafters utilize a number of different types of scales. Scales come in various shapes and sizes, the most commonly used scale is the triangular scale. Scales are based upon one of two measurement systems the U.S. customary system or the metric (ISO) measuring system. Scales that use the U.S. customary system use the inch as standard unit of measurement. Scales using the metric (ISO) system use the millimeter as the standard unit of measurement.

Inch Scales

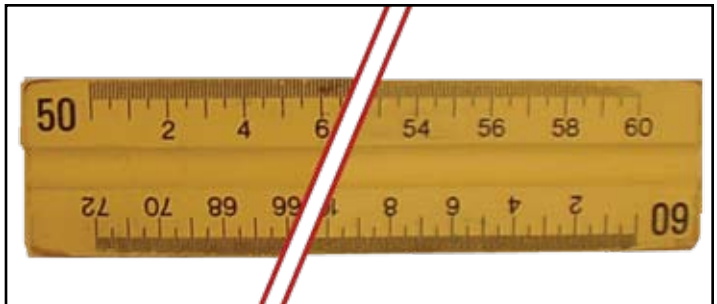
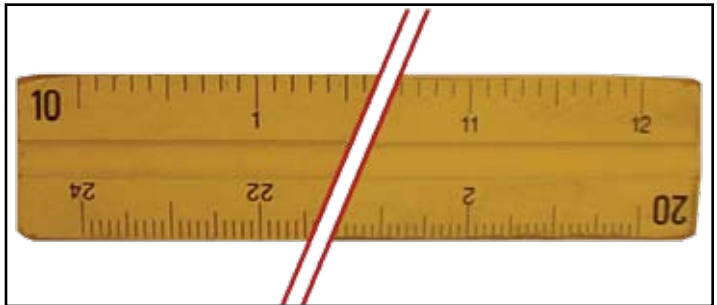
Typical inch scales are divided into feet, inches, and fractions of an inch. A foot is divided into 12 inches and an inch is divided into 16 equal parts or fractions of an inch. The inch scale is used to measure feet, inches and fractions of an inch. The types of scales that use the U.S. customary system include architects scale, mechanical engineers scale, and civil engineers scale.

Architects Scale

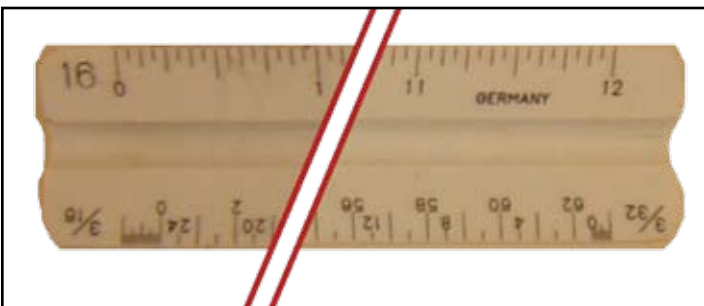
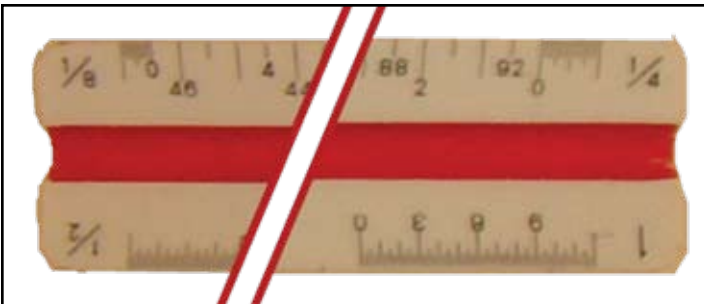
The architect scale is used to develop or read blueprints and designs for buildings and structural plans. On a triangular architect scale there are 11 different scales (3/32", 1/8", 3/16", 1/4", 3/8", 1/2", 3/4", 1", 1 1/2", 3", & 12"). Each of these 11 scales is equated to one foot (1'-0"). For example, when using the 1/4" = 1'-0" scale, every 1/4th of an inch is equal to one foot. It is important to note that scale does not equal size. Which means that a 1/4" scale converted in ratio form is equal to 1:48 size.

Civil Engineering Scale

The civil engineering scale is used to show the size and features of land in relation to buildings, roads, drainage and other civil infrastructure. The civil engineering scale uses inches that are divided into decimal parts. There can be 10, 20, 30, 40, 50, and 60 decimal parts per each inch. These decimal parts can represent any measurement such as an inch, a foot, a yard, or even a mile. Civil engineer's scales have numbers that run incrementally from left to right. The whole number to the left of the number line indicates the scale those numbers represent.



Civil Engineers Scales

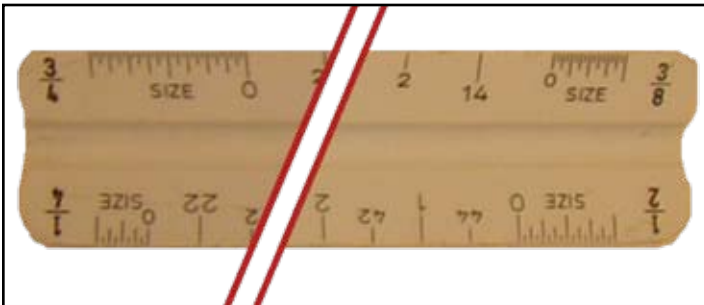
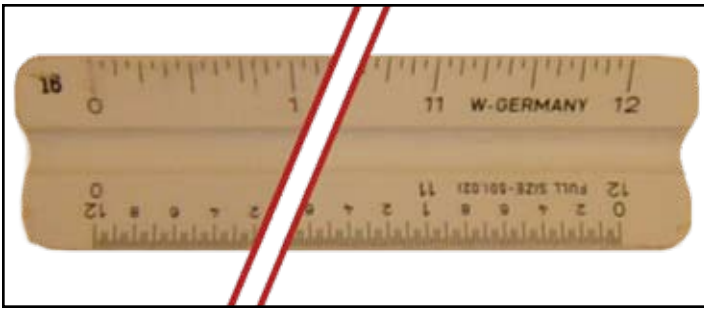


Architects Scales

It is important to look closely at the subdivisions labeled on the scale. Each side of the architect scale represents two scales both with numbers that run incrementally from left to right and from right to left. The number or fraction labeled at each end of the triangular scale tool indicates the scale those numbers represent.

Mechanical Engineering Scale

The mechanical engineer's scale is used mainly in the manufacturing sector doing mechanical drawings that use fractions and decimal inches. On the mechanical scale an inch can be divided into 16th's, 10th's, or 50th's. When each inch is divided into 16th's the scale is similar to the architect's scale; however, when it is divided into 10's or 50's units per inch it is much like the civil engineer's scale. Using the 50 units per inch in a scale brings a high level of accuracy, which is needed for small mechanical or electrical parts. Mechanical engineer's scales also have scales that run incrementally both from left to right and from right to left. Mechanical engineering scales subdivisions include 1/32", 1/16", 1/8", 1/4", 3/8", 3/4", 1/2", and 1" units.



Mechanical Engineer's Scales

Frequently used language for mechanical drafters for scaling include:

Full scale	1" to 1"	or	1:1
Half scale	1/2" to 1"	or	1:2
Quarter scale	1/4" to 1"	or	1:4
Twice (double) scale	2" to 1"	or	2:1

Metric Scale

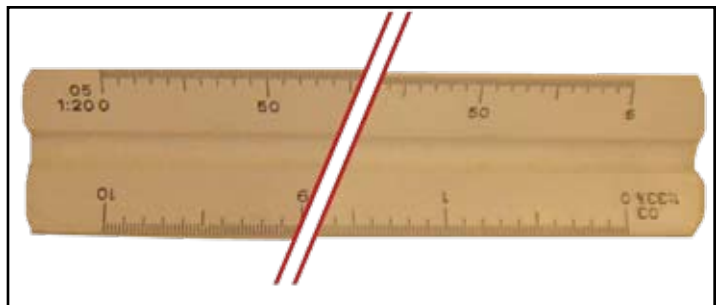
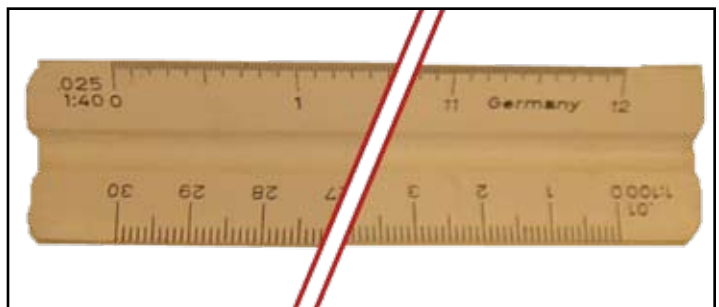
The metric scale or the international standard (ISO) used on International System of Units (ISO). The metric scale is in decimal form and based upon the meter. On the metric scale the units are based upon multiples of ten and there are no fractions. The primary unit of measure used in this system is the millimeter (mm). The metric units of linear measurement are identified below.

(ISO) Metric Symbols and Conversions

Millimeter (mm)	10 millimeters	= 1 centimeter
Centimeter (cm)	10 centimeters	= 1 decimeter
Decimeter (dc)	10 decimeters	= 1 meter
Meter (m)	10 meters	= 1 decameter
Decameter (dam)	10 decameter	= 1 hectometer
Hectometer (hm)	10 hectometer	= 1 kilometer
Kilometer (km)	1 kilometer	= 1000 meters

(Most commonly used are highlighted)

On the common triangle metric scale the following scales are paired together: 1:1 / 1:10, 1:2 / 1:20, 1:5 / 1:50, 1:100 / 1:200, 1:500 / 1:1000, 1:1250 / 1:2500. It is also important to be careful when measuring with the metric scale because different scales are paired on the same line but begin on opposite sides. A drawing that uses the metric scale and that is scaled using a ratio of 1:10, simply means that the real object is 10 times larger than the diagramed object. If the scale is 10:1 then the real object is 10 times smaller than the diagramed object.



Metric Scales

Comparing Units of Length

Unit	U.S.	ISO Equal
1 inch (in)		25.4 mm
1 foot (ft)	12 in	0.3048 m
1 yard (yd)	3 ft	0.9144 m
1 mile (mi)	5,280 ft	1.609344 km

Basic Measurement Quantities

Drafters must be well versed in all fundamental measurement systems that require mathematical applications. Today, most drafters use computer programs that computer the mathematical applications measures. However, each drafter must have a fundamental understanding of the mathematics and measurements behind the computer program. Located below are some of the various measurement systems that must be understood and a few basic examples of how an engineer or architectural drafter applies these systems.

Mass/Weight

Mass is the amount of matter an object has, and weight is the force generated by the gravitational attraction of the earth. These two terms differ, however in general use today, weight and mass are equivalent. The U.S. commonly uses the ounce, pound, and ton; the hundredweight is also used in agriculture and shipping purposes..

Units of Mass

Unit	U.S.	ISO (equal)
1 ounce (oz)		28.349523125 g
1 pound (lb)	16 oz	453.59237 g
1 hundredweight (cwt)	100 lbs	45.359237 kg
1 (short) ton	2000 lbs	907.184 74 kg

Volume

Identifying volume of a solid object is trying quantify how much space it occupies. An object must have some sort of three-dimensional space to hold volume. Units of volume can be measured in liquid and dry material. The generally used U.S. units of volume as well as liquid and solid units can be viewed below.

Volume in General

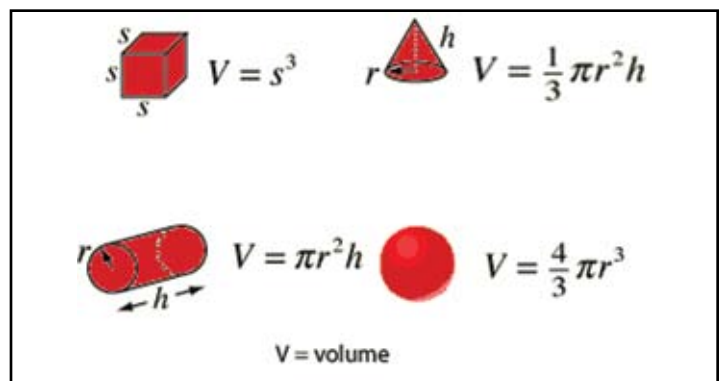
Unit	Divisions	ISO Equal
1 cubic inch (cu in)		16.387064 mL
1 cubic foot (cu ft)	1728 cu in	28.316846592 L
1 cubic yard (cu yd)	27 cu ft	764.55485798 L

Liquid volume

Unit	U.S.	ISO Equal
1 fluid ounce (fl oz)		29.57353 mL
1 cup (cp) or (c)	8 fl oz	236.5882 mL
1 (liquid) pint (pt)	32 fl oz	473.1765 mL
1 (liquid) quart (qt)	2 pt / 32 fl oz	946.3529 mL
1 gallon (gal)	4 qt /128 fl oz	3.785412 L
1 oil barrel (bbl)	42 gal	158.9873 L

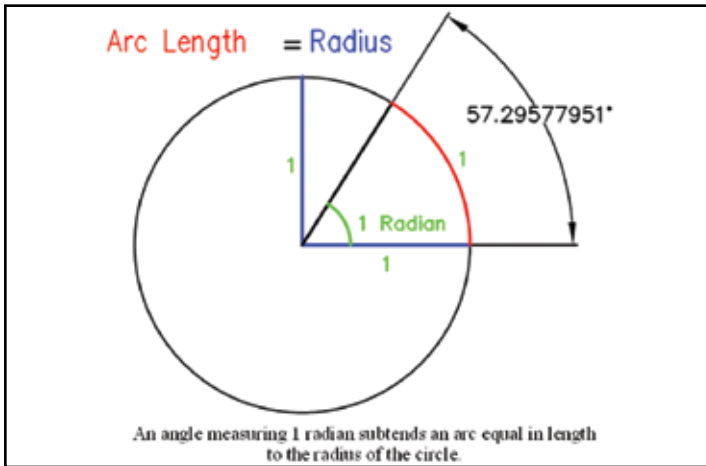
Dry volume

Unit	Divisions	ISO Equal
1 (dry) pint (pt)	33.60031 cu in	550.610 5 mL
1 (dry) quart (qt)	2 pt	1.101221 L
1 (dry) gallon	4 qt	4.404884 L
1 bushel (bu)	4 pk	35.23907 L



Degrees/Radians

Mass is the amount of matter an object has, and weight is the force generated by the gravitational attraction of the earth. These two terms differ, however in general use today, weight and mass are equivalent. The U.S. commonly uses the ounce, pound, and ton; the hundredweight is also used in agriculture and shipping purposes.



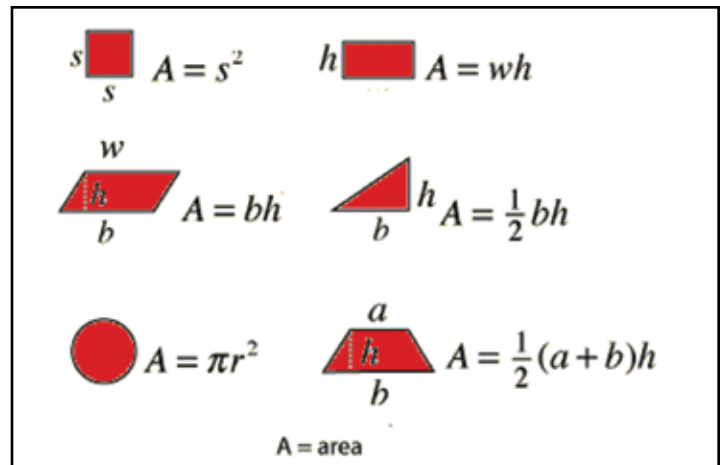
An angle measuring 1 radian subtends an arc equal in length to the radius of the circle.

Area

Area is a measurement of the space surrounded by an enclosed region. U.S. units of area are measured by squaring the inch, foot, yard, and the mile. Architectural drafters use the unit of square feet to describe the area specifications of a room in a building or a home. Civil drafters use the acre unit to communicate land or property size.

Units of Area

Unit	U.S.	ISO (Metric)
1 sq inch		645.16 mm ²
1 sq foot	144 sq in	0.09290304 m ²
1 sq yard	9 sq ft	0.83612736 m ²
1 sq mile	3,097,600 sq yd	2.58998811033km ²



Perimeter

The perimeter is the measurement around a given two-dimensional space. Perimeter is used when calculating the border of an object such as a yard or flowerbed when a fence or other border will be installed around the edges. Calculating the perimeter of an object is different than the area because area takes into consideration all of the surface area within the perimeter, not just the surrounding border of the object. Circumference is the perimeter term used when calculating the distance around a closed curve such as a circle.

Common Perimeter (P) Formulas

Shape	Formula
Square (Condition: a, b, c, d indicate the length of each side)	$P = a + b + c + d$
Triangle (Condition: a, b, c indicate the length of eachside)	$P = a + b + c$
Circle (Condition:r stands for the radius and $\pi = 3.14159265$)	$P = 2 \cdot \pi \cdot r$
Equilateral Polygons (Condition: n represents the number of sides; a indicates the length of each side)	$P = na$

Mathematical Review

Having a good understand of general mathematical applications is critical for a successful drafter. Drafters must work with fractions, decimals, percentages, and use the fundamental math formulas mentioned above. Engineers and architects will be trained in advanced mathematical problem solving to understand the math behind computer software programs that design their work. The purpose of this text is to review the basic math applications that all drafters should recognize.

Fractions

There are two components to a fraction, a numerator (top part of the fraction) and the denominator (bottom part of the fraction).

Fractions must be reduced to the lowest terms. To do this, divide the numerator and denominator by their greatest common factor or any common factor. Keep dividing until there are no more common factors.

Example: $4/8 = 2/4 = 1/2$

- **Adding & Subtracting Fractions**

Denominators must be the same to add and subtract fractions.

Example: $3/8 + 1/8 = 4/8 = 1/2$

(remember to reduce to the lowest terms)

Example: $3/8 - 1/8 = 2/8 = 1/4$

If the fractions have different denominators:

First find the least common denominator for both fractions.

Write the equivalent fractions using the new denominator.

Then add or subtract the fraction, and reduce if necessary.

$1/3 + 3/5 = ?$

The least common denominator is 15.

$1/3 + 3/5 = 5/15 + 9/15 = 14/15$

- **Multiplying Fractions**

Denominators do not have to be the same. Simply multiply the numerators together and then multiply the denominators together.

Example: $1/3 \times 3/4 = 3/12 = 1/4$ (again, remember to reduce to the lowest terms)

- **Dividing Fractions**

First, invert the second fraction upside down, and then multiply.

Example: $3/4 \div 2/3 = (3/4 \times 3/2) = 9/8$ or $1 \frac{1}{8}$

- **Working with Compound Fractions**

To add, subtract, multiply, or divide compound fractions, the numbers must first be converted to improper fractions. To do this, the whole number must be multiplied by the denominator of the fraction and then added to the numerator while keeping the same denominator.

Converting a compound fraction to an improper fraction:

$5 \frac{3}{4} = [(5 \times 4 + 3) / 4] = 23/4$

Example: $2 \frac{1}{2} \times 3 \frac{1}{3} = 5/2 \times 10/3 = 50/6$ or $8 \frac{1}{3}$

Decimals

- **Converting Common Fractions into Decimals**

Fractions can be written as decimals. To convert a fraction to a decimal, divide the numerator by the denominator.

Example: $12/7$ is 12 divided by 7.

Using a calculator, $12 \div 7$, equates to 1.714285.

- **Converting Decimals into Common Fractions**

Write the decimal and divide it by 1. Then multiply the nominator and denominator by a factor of 10 for a single digit after the decimal. Add an additional zero for each additional number so that there is a zero for each number after the decimal.

Example: convert 0.8 to a common fraction

Divide decimal by 1: 0.8/1

Multiply nominator and denominator by 10:

$$(.8 \times 10) / (1 \times 10) = 8/10$$

Reduce fraction: 4/5

Example: convert .375 to a common fraction

Divide decimal by 1: 0.375/1

Multiply nominator and denominator by 1000:

$$(.375 \times 1000) / (1 \times 1000) = 375/1000$$

Reduce fraction: 3/8

• **Decimal to Percentage**

To create a percentage from a decimal move the decimal point two places to the right.

Examples: .10 = 10%; .40 = 40%; .0525 = 5.25%

Reverse the process to create a decimal from a percentage.

Move the decimal point two places to the left.

Examples: 20% = .2; 62.5% = .625; 4.55% = .0455

• **Solving Percentage Problems**

To calculate the percentage of a number, first convert the percentage to a decimal then multiply the numbers.

Example: Calculate 250% of 50:

$$(250\% = 2.5) \quad 2.5 \times 50 = 125$$

Example: Calculate 12.5% of 200:

$$(12.5\% = .125) \quad .125 \times 200 = 25$$

To determine the value that represents the percentage of a number, first convert the percentage to a decimal then divide the numbers.

Example: 15% of what number is 630?

$$(15\% = .15) \quad 630/.15 = 4200$$

Decimal Values

1.714285



Decimal Values

1.714285

Tenth

Hundredth

Thousandth

Ten Thousandth

Hundred Thousandth

Millionth

Percentages

Percentages are similar to fractions and decimals since all three methods provide different ways to communicate the same thing: describing numbers that have been divided into parts. Percentages are representations of parts per hundred. This means that a percentage such as 50% really means that you are working with 50/100 or 1/2. It is important to remember that percentages do not always represent parts smaller than one hundred. For example, 200% is twice the number, or in scaling terms twice or double scale (2:1) of the object.

CHAPTER 3

Hardware and Software Components

One of the purposes of this text is to acquaint each student with the principles of drawing via a computer. To accomplish this, each user must first learn the basics of how to effectively operate a microcomputer before attempting to learn the mechanics of any drafting software.

Components of a Microcomputer System

A microcomputer or personal computer (PC) is an electronic device used as a tool to help you perform a certain task. This tool is made up of several components (parts) that together become a microcomputer system. The three components of a microcomputer system are the hardware, the software, and the user.

Hardware

Hardware is the physical machinery component of a microcomputer system. The hardware you will use for this course consists of three parts:

Input There are two input devices, the keyboard and the mouse. The keyboard will be used to put information into the working part of the computer. When you begin to draw, the mouse will serve as the major input device.

CPU The central processing unit (CPU) is the brain of the computer. This is the place where the information you type is converted into a language the computer can understand. The information is manipulated in the CPU according to your instructions. This hardware unit also houses the primary storage area of the system. (A detailed discussion of storage will follow.)

Output The output component is the part of the system that is used to receive and display the information from the CPU. Two forms of output will be used during this course: the monitor and the plotter. The monitor is the screen that looks similar to a TV screen. The plotter (the drafting term used for printer) may be used to print a “hardcopy” of the information received from the CPU.

Software

Software is the set of instructions that tells the computer what to do. Software is divided into two main categories:

Operating System (Windows). An operating system contains a series of software programs (files) that: 1) supervise the operations of the Central Processing Unit. 2) control the input, output and storage functions of the microcomputer system, and 3) support the use of applications programs. Windows programs such as Windows 95, Windows 98, Windows 2000, Windows XP, and Windows Vista have replaced the older DOS system. This book was written to correspond with Windows XP and illustrations are based on this system.

Applications. Application software can be divided into two parts:

Program files. The five basic types of software applications programs are: word processing, spreadsheet, database, graphics and communications. You will be using A+CAD (a graphics program) to produce the required drawings for this course.

Data files. Data files are the documents you produce as a result of using the program files.

Windows. WINDOWS is a program that enables your computer to perform in a graphic-oriented environment. This environment is similar to the way a Macintosh functions and is typically called WYSIWYG--what you see is what you get. What you see on the window is what you will get on the printed copy.

User

The third component of a computer is the user, the person who is going to operate the computer. You are the most important component of the system. The hardware and software cannot run themselves. Nothing will happen unless a person inserts the disk containing the software program, turns on the computer, and instructs the computer to perform specific tasks.

Units of Measure

Ounces, pints, quarts, and gallons are used as units of measure for containers of liquid. Similarly, there are units of measure for the space consumed by the information used in a microcomputer system. Operating files, program files, and data files all consume space. Typically, the units of measure for computer systems are:

Bit A bit is the smallest unit of measure. Eight bits are used together in the form of electronic impulses to produce a simple letter or character (called a byte).

Byte One character space of information such as a letter, number, symbol, or space on the Spacebar. A byte is made up of eight bits.

Kb A Kb (Kilobyte) is equal to 1024 bytes. Most people working with computers round this figure to 1000 for ease of use. **K** stands for “Kilo” which means 1000.

Mb A Megabyte is equal to a million bytes of information.

Gb A Gigabyte is equal to a billion bytes of information.

Memory and Storage

The containers that the computer system uses for holding information are called storage areas. These storage areas are called primary and secondary storage.

Primary Storage. The primary storage area resides in the CPU and is called “memory.” The memory in the CPU is divided into two parts:

ROM. ROM stands for Read Only Memory. The program files that are stored in this part of the CPU are permanent files. They cannot be altered or deleted by the user. The files contained in ROM are similar to a human being’s central nervous system. Your heart beats and you breathe even when you are asleep. The program files in ROM run as needed without the user telling the computer system to run them.

RAM. RAM stands for Random Access Memory. The files that reside in this part of the CPU are there temporarily. This is the area of storage that contains the program and data files that you use and create. When the power is turned off, RAM is emptied. When the power is turned on, the program files must be loaded back into RAM in order to be used. Remember, this is temporary memory. Think of RAM as being similar to a human being’s conscious memory. When you are awake, you consciously carry out specific tasks; but when you fall asleep, those tasks are gone from your conscious memory. The amount of RAM contained in a CPU varies from computer to computer. The amount of RAM a computer requires is dependant upon the operating system and the applications (programs) that it will run. Software manufacturers usually publish “system requirements” for their software which should specify the operating system and hardware requirements needed for the application. As programs become more sophisticated, they require more temporary memory.

Secondary Storage. The secondary storage containers used by a microcomputer system are usually disk storage containers. System files, program files, and data files that need to be used at another time or that should not be forgotten are stored on a disk. This is similar to the process human beings use for writing on paper the things that should not be forgotten. The information on paper can be put back into the conscious memory by reading, or new information can be added to the conscious memory by reading.

Three types of secondary disk storage are:

Hard (fixed) disk. Most microcomputers have the capability of containing a hard or fixed disk drive within the CPU. A hard disk drive is not a part of the central processing unit, but it is often housed in that piece of hardware for safety and convenience purposes. Typically, a hard disk drive may contain two, three, or four hard disks. These disks are made of metal and are stationary, thus the terms “hard” and “fixed.” The user

does not insert a hard disk into a disk drive, because it is already in the drive. The controller instructs the computer system to retrieve and/or save information to or from the hard disk. Hard disks have exploded in storage capacities in the past few years. Storage capacities have multiplied a hundred fold in less than ten years. One hundred gigabyte drives are common as this is written.

USB Flash Drives USB flash drives are flash memory data storage devices integrated with a USB (universal serial bus) interface. They are typically small, lightweight, removable and rewritable. These are commonly referred to as a “ThumbDrive” or “JumpDrive”. USB Flash drives offer advantages over other portable storage devices, particularly the floppy disk. They are more compact, generally faster, hold more data, and are more reliable than floppy disks. These types of drives use the USB mass storage standard, supported natively by modern operating systems such as Windows, Mac OS X, Linux, and Unix.

Compact Disc. A Compact Disc or CD is an optical disc used to store digital data, originally developed for storing digital audio. Compact Disc technology was later adapted for use as a data storage device, known as a CD-ROM, and to include record-once and re-writable media (CD-R and CD-RW respectively)

Digital Versatile Disc. DVD (“Digital Versatile Disc” or “Digital Video Disc”) is an optical disc storage media format that can be used for data storage. DVDs resemble Compact Discs in that they have the exact appearance and both are optical storage media so similar that a DVD reader or writer can usually read CDs, but DVDs are encoded in a different format of much greater density, allowing a data storage capacity 8 times greater.

Keyboards and Special Keys

The location of certain keys on a keyboard may vary depending upon the style and the make of the microcomputer being used. It is important for you to become familiar with the keyboard on the microcomputer you are using before you start to type.

Cursor	The cursor is a blinking rectangle that appears on the monitor screen to indicate the position where the next message will display or where the next key that is typed will display.
Function Keys	These special, dark gray keys, numbered F1 through F10, may be positioned at the left side of the main keyboard, or at the top. The functions they perform will change depending on the software program being used.
Booster Keys	The Ctrl (control), Alt (alternate), and Shift keys are called booster keys because they boost other keys to perform a function they do not ordinarily perform. A booster key does not perform any function until it is used in conjunction with another key. You can hold the shift key down as long as you want to, but nothing is going to happen until you tap another key. The same is true for Ctrl and Alt. When a booster key is required, hold the booster key down, then tap the other key that is required for the special function, and then release both keys.
Enter or Return	The Enter key is similar to a Return key on an electric typewriter. It is used to advance the cursor to the beginning of the next line and/or to enter information into RAM. This is usually a large key on the right side of the alphabet keys and may be marked with “Enter,” “Return,” or an arrow pointing down and to the left.

Caps Lock	<p>The Caps Lock key is used to display all alphabet characters in capital letters (uppercase). Numbers, symbols, and punctuation marks are not affected by Caps Lock button. Some keyboards have a light indicating when the Caps Lock function is on. Since this key is a toggle, tapping Caps Lock will turn the function off.</p>	<p>You should have a basic understanding of the following terms before proceeding:</p>																						
ESC	<p>The Esc (Escape) key is your friend. It is used to back out of, or interrupt commands or functions you no longer wish to perform. Its location will depend on the keyboard you are using. It may be positioned at the upper left of the common keys or at the upper left on the numeric keypad.</p>	<table border="0"> <tr> <td>microcomputer system components</td> <td>hardware</td> </tr> <tr> <td>software</td> <td>input</td> </tr> <tr> <td>CPU</td> <td>output</td> </tr> <tr> <td>hardcopy</td> <td>DOS</td> </tr> <tr> <td>applications software</td> <td>program files</td> </tr> <tr> <td>data files</td> <td>byte</td> </tr> <tr> <td>Kb</td> <td>Mb</td> </tr> <tr> <td>primary storage</td> <td>ROM</td> </tr> <tr> <td>RAM</td> <td>secondary storage</td> </tr> <tr> <td>cursor</td> <td>function keys</td> </tr> <tr> <td>booster keys</td> <td></td> </tr> </table>	microcomputer system components	hardware	software	input	CPU	output	hardcopy	DOS	applications software	program files	data files	byte	Kb	Mb	primary storage	ROM	RAM	secondary storage	cursor	function keys	booster keys	
microcomputer system components	hardware																							
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data files	byte																							
Kb	Mb																							
primary storage	ROM																							
RAM	secondary storage																							
cursor	function keys																							
booster keys																								
Backspace	<p>The Backspace key is usually used to delete the character to the left of the cursor. This key is located above the Enter key on the top row of numbers. It may contain an arrow pointing to the left.</p>																							
Del	<p>The Del (delete) key is used to delete the character on which the cursor is positioned.</p>																							
Num Lock	<p>This key is a toggle that changes the action of the numeric keypad from cursor movement to numbers, and back again. The keyboard you are using may have a light that indicates when the Num Lock function is activated, thus allowing you to use the numeric keypad for entering numbers but not for moving the cursor. If you wish to deactivate this function, tap the Num Lock key.</p>																							
Cursor Movement Keys	<p>These keys are marked with arrows pointing up, down, left, and right. The keys marked Home, End, PgUp, and PgDn may also be used to move the cursor. You may have two sets of these keys on the keyboard you are using. You may use either set. Keep in mind that the Num Lock function must be deactivated for you to use the cursor movement keys on the numeric keypad.</p>																							

Learning to use Windows

Chapter Performance: When you have completed this chapter, you will be able to arrange and organize group windows on the desktop. This chapter is for students that are not familiar with the Windows XP interface, and the Explorer for basic folder and file management.

Windows is an application program that operates in a graphical environment that allows MS-DOS microcomputers to function in a manner similar to Macintosh microcomputers. A graphical users interface has a WYSIWYG environment, which means *what you see is what you get*. What is seen on the document screen is what is printed on paper.

A graphics-oriented environment is considered to be *user friendly*, and does not require the user to use special keystrokes and coding to cause the computer to function. Instead, options are selected from a variety of menus and boxes. Once you learn to use one Windows application, it will be easy to learn another because each one operates in a similar manner.

Windows may or may not be an operating system. Earlier versions of Windows were used in conjunction with MS-DOS and added several new dimensions to the disk operating system. Windows allows the user to run several applications at the same time. For example, spreadsheet, word processor, and database programs may co-exist simultaneously in the computer. Users may switch from one application to the other, efficiently combining information to create striking letters, reports, and presentations. New Windows applications, such as Windows Vista, Windows XP, Windows 2000, Windows 98 and Windows 95 can be stand-alone operational systems.

Starting the Computer and Windows XP

Section Performance: When you have completed this section, you will be able to start a computer and load Windows XP's program into memory.

Now that you have a basic understanding of a computer system, you will learn to start the computer and the Windows XP application program.

If you are using a hard disk system, follow the steps in the section under the heading, *Hard Disk Users*. If you are using a network system, follow the steps in the section under the heading, *Network Users*.

Hard Disk Users

Typically, hard disk computers have enough storage capacity for you to store the Windows XP program and the data you create using those programs. For the purposes of this course, however, you will learn to save your data to a USB flash drive. (USB thumb drive)

1. Turn on the power to the computer and the monitor (if necessary).
2. Sit back and relax while the computer goes through the booting (starting) process.
 - The computer performs its normal checking procedures and loads the WINDOWS XP files into memory. You may see some text appear and quickly disappear off the screen. When the booting process is complete, information will no longer disappear from the screen, and the disk drive lights will be turned off.
3. If your computer displays a menu with a number of choices, locate the choice for Windows XP and follow the screen directions to start the program.
 - Momentarily, your screen display should look similar to the illustration in Figure 2.1. The *Desktop* icons (pictures representing groups and/or



Figure 2.1

programs) that display on your screen may be different from the Desktop displayed in Figure 2.1. If you see Recycle Bin, anywhere on the Desktop of your screen, you have successfully started the Windows XP program.

4. If your computer displays

C:\> type **win** and tap **↵** to start the program.
(pertaining to Win 95)

5. If your computer starts the *Windows XP* program for you automatically, check your screen display to see if it looks similar to the illustration in Figure 2.1.
6. Continue to the section, *Defining the Windows XP Environment*.

Network Users

A network provides a way for many people to use the same application programs. Typically, these programs are stored in a large computer that may be in another room or even in another building from the one where your computer is located. A cabling system connects your computer and, many other computers, to the large computer. The network is a combination of software and hardware that enables you to access the programs and/or files that are stored on the large computer.

Each network system will have its own special procedures for gaining access to a program. We recommend that you have a written copy of the steps you need to take to log on to the network you are using.

1. Ask your instructor or lab manager for a copy of the log-on procedures that you need to follow.
 - Most network systems require the user to enter a password to gain access to the system. This password may be your initials, your social security number, or some other combination of letters and/or numbers that only you will use.
2. Follow the steps for logging on to the network system you are using.
3. If necessary, select the *Windows XP* menu option to select the Windows XP program.
 - If you have difficulty logging on, please ask for help. If you were successful, your screen display should look similar to the illustration in Figure 2.1. The Desktop icons that display on your screen during a network boot may be different from the Desktop displayed in Figure 2.1. If, however, you see the words, *Recycle Bin*, anywhere your screen, you have successfully started the *Windows XP* program.

Defining the Windows XP Environment

When you have completed this section, you will be able to identify the parts of a window.

The *Windows XP* environment includes many components. The components that you will need to know before using *A+CAD* are explained in this section.

This book was written in reference to Windows XP, however.

Identifying Parts of a Window

A *window* is a rectangular section on the screen that contains pictures and information that you use when performing tasks. A window is the area where you view an application or document. You may be able to see several windows on your screen at the same time.

Typically, when you start Windows XP, the *Desktop* displays shortcut icons, these are generally “start” icons for the most frequently used programs or folders. The *My Computer* icon activates a window that contains other smaller windows. Typically, the smaller windows represent a group of application programs and are, therefore, called *group windows*. Within each group window, you will find pictures with names below them. The pictures are called icons and are used to represent programs and/or documents.

You need to know what to call the various parts and/or contents of a window to enable you to efficiently use Windows XP and the A+CAD program. Figure 2.2 identifies the windows parts. Locate each window part in Figure 2.2 and study the definition in the following list before continuing.

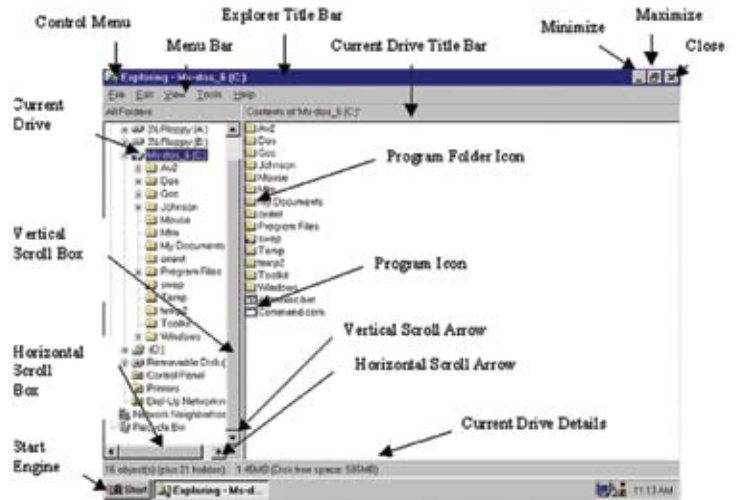


Figure 2.2

Parts of a Window (continued)

Icon A graphic representation (picture) for various items in Windows.

2. My Computer icon Contains icons to disk drives, printers and networks (hard

Application A computer program used to perform a task such as creating a drawing.

Control menu icon Contains menu items that help you organize and size the windows on your screen.

Desktop Main window containing My Computer and other frequently used icons and/or program applications.

Explorer File management tool within Windows XP (previously File Manager).

Group window A window containing a collection of program applications.

Maximize/Restore Used to enlarge and restore the active window to and from filling the entire screen.

Menu bar A list of items or commands.

Minimize/Restore Used to reduce and restore the active window to and from the taskbar.

Scroll arrow Used to move one line of information at a time into view within the window.

Scroll bar Used to move up, down, left, or right in a document or window.

Scroll box Used to move to a general location in a document or window.

Taskbar/(Restore)	Memory collection for current running programs. Click to restore.
Title bar	Displays the name of an application or document.
Window	A rectangular area that contains a software application or a document file.

Identifying the Desktop

The *desktop* is the screen background for Windows XP. It is similar to the surface of a desk. It is the part of the screen where objects are placed, similar to the placement of pieces of paper and pencils or pens on a desk.

Sometimes your computer screen desktop may display one open window, like having one piece of paper on your office desk.

Sometimes your computer screen desktop may display several open windows of various sizes, like having several pieces of paper of various sizes on your office desk. Also, just like your office desk, on rare occasions the desktop may be empty.

Identifying the Mouse Pointer

A mouse is a hardware device that, when moved across a desk surface or mouse pad, positions a pointer, usually in the form of an arrow, on the computer screen. It is the primary means used for selecting items (i.e. command buttons or menus) and for moving and sizing windows.

Typically, the mouse is held between the thumb and ring finger, allowing the index and middle fingers to operate the buttons on top of the mouse. A pointer on the screen moves in combination with the movement of the mouse on the pad or desk surface.

1. Locate the *mouse pointer arrow* on your screen by moving the mouse forward, backward, or from side-to-side on the mouse pad and watching the screen display.

2. Continue to move the mouse on the mouse pad and notice how the pointer moves proportionately on the screen. Repeat this step until you feel comfortable moving the mouse and locating the pointer on the screen.
 - Rotating the mouse in small circles on the mouse pad in any direction causes the screen pointer arrow to move in small circles on the screen.
3. Lift the mouse up off the pad and rotate the mouse in the air, noticing that the arrow pointer does not move on the screen.
4. Return the mouse to a new location on the pad and rotate once more, noticing that now the arrow pointer moves on the screen.
 - You can relocate the mouse on the pad without affecting the arrow on the screen by lifting the mouse off the pad before relocating it.

The mouse arrow pointer may be used to select a specific symbol, box, word, or picture on the window. Study the following list of terms and their meanings so you will understand what to do with the mouse when you encounter these terms in the instructions in this book.

Click	To quickly tap and release the left mouse button.
Double-click	Click (tap) the left mouse button twice in rapid succession.
Drag	Hold down the left mouse button while you slide the mouse on the pad.
Point	Slide the mouse (without holding down the button) until the tip of the pointer rests on the item of your choice.
Pick	A pick is a synonymous with click.

HELPFUL HINT: Mouse Pointer

You may notice that occasionally the mouse pointer disappears from the screen. This happens when it has not been used for a few minutes. A slight movement of the mouse will cause the pointer to reappear on the screen.

Working with Windows XP

Section Performance: When you have completed this section, you will be able to manipulate the Windows application to organize the desktop.

Selecting the Control Menu

Selecting the *Control-menu icon* at the left edge of the title bar displays a menu that contains commands that enable you to change the size of the active window, switch to a different window or program and exit the Windows program. Every window has a *Control-menu icon* in the title bar.

HELPFUL HINT: Active Windows

Only one window can be active at a time and only one icon within a window can be active at a time. The technique to activate a window is very easy; simply click anywhere within a window to activate it. Similarly, an icon can be made active by clicking one time anywhere on the icon within the active window.

This concept is important because once a window or icon is activated, any command that you select will be carried out on the active window or icon.

1. To do file management you must first go to the “**Start**” button located in the lower corner of your screen.
2. Pick **Start**.
 - A vertical menu pops up. (See figure 2.3)
3. Scroll up and pick **Programs** and then pick **Accessories**.
 - A second and then a third vertical menu appears. (See figure 2.4)
4. Pick **Windows Explorer**.
 - Your file manager dialog box appears. (See figure 2.5)

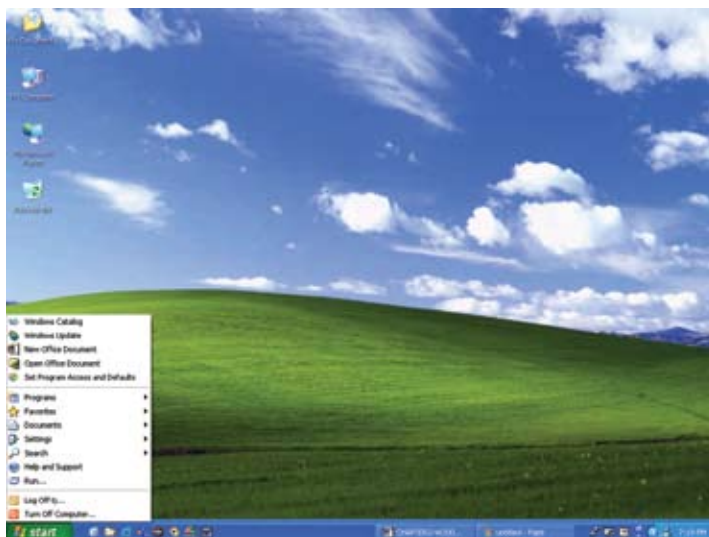


Figure 2.3



Figure 2.4

8. Study the list of commands on this menu.
 - These commands are used to help you organize the windows on the desktop. Notice that some items on the list appear in dark print while other items appear in light print. The dark items are the ones you can use now, but the light items are not available to you at the present time.
9. Again, point the arrow to the *Control-menu* icon at the left of the title bar and click the left mouse button once.
 - The pull-down menu disappears when the left mouse button is *moved* and *clicked* again.

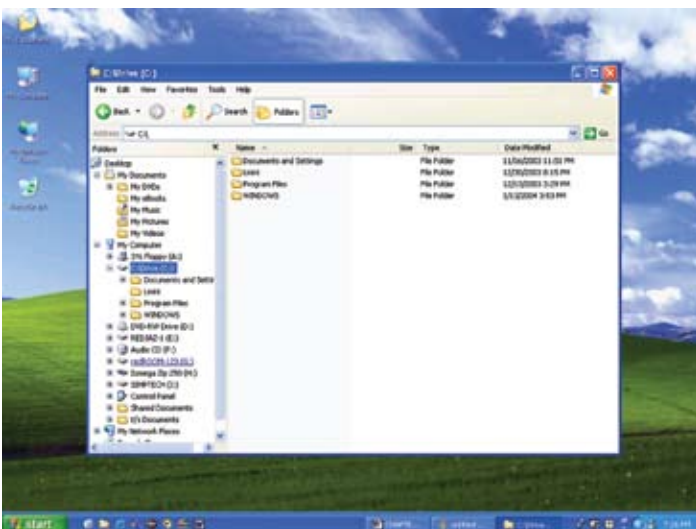


Figure 2.5

10. Hold down the **Alt** key on the keyboard and tap the **Spacebar** to open the Explorer Control-menu.
 - Windows provides several methods for selecting most items. Now you know how to select the Control-menu using the mouse and using the keyboard.
11. Move the mouse pointer so that the arrow displays near the center of the desktop and click.
 - Now you know how to close the menu by pointing to two different places on the desktop and clicking.
12. Point to the *Control-menu* icon and double click.
 - Double clicking on the Control-menu icon is a shortcut for selecting the *Close* command.

5. Locate the *title* bar near the top left of the Explorer window.
6. Point to the *Control-menu* icon by moving the mouse pointer to the icon until the tip of the pointer is on top of the icon.
 - Refer to Figure 2.2 for help in locating the Control-menu icon.
7. Click the left mouse button once to display the pull-down menu.

2. **Selecting Menus from the Menu Bar**

Windows XP uses the same menu names for common functions to make it easy for you to learn new Windows based applications. Learning to use the *Explorer* menus will make it easier to learn to use *A+ CAD menu*.

1. Start your computer and open the Windows XP program. If necessary, refer to the previous section on *Starting the Computer and Windows XP*.
2. If necessary, go to the windows *Explorer* and open the Explorer window.
3. Move your pointer to the drive letter for your USB flash drive.

Locate the menu bar directly below the title bar. The menu bar contains the *File*, *E**dit*, *V**iew*, *F**avorites*, *T**ools* and *H**elp* menus. You may select a menu in one of two ways:



pointing to the menu and clicking



holding down the *Alt* key and tapping the underlined letter of the desired menu name.

5. Move the pointer to *File* and *click* to display the pull-down menu.
6. Point to *E**dit* and *click*. Briefly read the pull-down menu.
7. Hold down the **Alt** key and tap **v**. Release the Alt key.
 - The *V**iew* pull-down menu appears on the screen.
8. Briefly read the selections. Tap the **Alt** key and the menu disappears.
9. Hold down the **Alt** and tap **h**. Release the Alt key to select the *H**elp* menu.




10. Briefly read the selections. Tap the **Alt** key and the menu disappears.
11. Select each menu by pointing to the command word and *clicking*.
12. Select each menu by holding down the **Alt** key and tapping the underlined letter.
13. Repeat the steps until you feel confident in selecting a menu using the mouse and the keyboard.

Now that you know how to select menus, you will want to know how to select a command from a menu. Let's experiment with the USB flash drive. (It will check the information on your personal diskette.)

1. Use one of the previous methods to select the *F**ile* menu, then **pick the drive letter assigned to the USB flash drive**
2. ***Point to Properties and click*** to select the command.
 - The "*Properties*" dialog box displays. This shows the amount of data stored on your USB flash drive. Do not make any changes or selections in the box at this time.
 - If we had just made selections within this dialog box an *OK* here would enact the changes.
1. ***Click the Cancel*** button to cancel the command.
 - No changes were made. If you "cancel" a command, no changes are made. You should "cancel" each of the experiments below.
4. Select the *F**ile* menu again.
5. Type **r** to select *P**roperties*, and click *Cancel* to cancel the command.
6. Select the *F**ile* menu one more time.

7. Tap ↓ until *Properties* is highlighted and tap **Enter** ↵ to select the command.
8. Click *Cancel* to cancel the command.

Now you know how to use three methods for selecting commands from a pull-down menu. Pull-down menu commands may be selected by

-  moving the pointer to the desired command and clicking
-  typing the underlined letter of the desired command
-  using the ↓ or ↑ arrow keys to highlight the command and tapping Enter ↵ to execute the command.

Using Minimize/Restore and Maximize/Restore

One of the advantages of working with Windows is that you can change the size of a window located on the desktop. If you are working with one window only, why not make it large enough to fill up the entire screen so you can see all of the information in the window? If you have several windows open on the desktop, but only need to work with one, why not make the others very small (the size of a button) and get them out of your way? You will learn to use the *minimize* button to change the size of a window to a taskbar box (icon), and you will learn to use the *maximize* button to change the size of a window to fill the desktop.

Minimize/Restore. The *Minimize* button is the first button at the right end of the *title bar*. It is used to shrink the active window to a symbol called a *taskbar box*. *Minimize* is commonly used when several applications are running at the same time and the user wishes to view an application window different from the one currently displayed on the screen. Minimizing the window does not close the program; the program is still running in the computer's memory and is physically present down on the *taskbar*. The *taskbar* is usually located at the bottom of the screen and is connected to the start *engine*. We will talk more about the *Start Engine* or the *Rev Engine* later.

Refer to Figure 2.2 for help in locating the Minimize button.

1. Point to the *Minimize* button at the right of the *Explorer* title bar and *click*.
 - The Explorer window disappears and is placed on the taskbar near the bottom of the desktop.
 - If the taskbar is not currently visible, move your mouse cursor to the extreme bottom of the screen and the taskbar will become visible. This display will list all the programs currently running and should include your reduced Explorer. (Your machine can run numerous programs at one time; the number depends upon the amount of memory available.)
2. Point anywhere on the *Explorer box*.
3. *Click* on the box to again display the Explorer *Control Menu* box on the screen. This is one example of **restoring** a window.
 - This is the same menu list you saw previously. *Minimize* the Explorer again. This time we will use the *restore* command to get back to the Explorer. The **Restore** command is used to restore the Explorer window.
4. Point to the *Explorer* box and *right-click*. Move your pointer to **Restore** the Explorer window to its previous size.
 - Notice that the box on the taskbar no longer displays near the bottom of the desktop, but **Restore** also restored the group windows that were open on the Explorer window.

Maximize/Restore. The *Maximize* button is used to enlarge the active window to the size of the entire work space on the desktop. Often an application is presented to the user on less than a full screen. Clicking on the *Maximize* button will enlarge the application window to fill the work space.

Refer to Figure 2.2 for help in locating the *Maximize* button.

1. Point to the *Maximize* button at the end of the *Explorer* title bar.
 - If you do not see the *Maximize* button at the end of the title bar, your window is already at its maximum size, and you should skip to step 3.
2. Click on the *Maximize* button to enlarge the window to fill the desktop.
3. Point to the *Minimize* button and click.
4. Double click on the *Explorer title* bar to **restore** the window to its previous size.
 - This is another way to maximize a window. Now hit the *minimize* button.
5. Click on the *Recycle Bin* window on your Desktop to make it the active window.
6. Point to the *Maximize* button on the *Recycle Bin* window title bar and click. Now go to the taskbar and click the *Explorer* box.
 - Now the *Explorer* window fills the desktop. Where is the *Recycle Bin*? It is behind the *Explorer* window fully maximized but underneath the *Explorer*. You now have windows stacked like a stack of paper on your desk.

Restore (General). The *Restore* button is used to reduce the active window to the previous size. The *Restore* button is used to reduce from maximized and enlarge from minimized (taskbar) and is the second button at the right end of the title bar. It is used to shrink the active window to a smaller size window. *Restore* is commonly used when several applications are running at the same time and the user wishes to view an application window different from the one currently displaying on the screen. Restoring the window does not close the program; it just reduces the size of the current window.

Sizing a Window

In addition to minimizing and maximizing a window, you can change a window's size to just about any size you want. If you are trying to compare the contents of one window with the contents of another, why not make them both small enough to display them side by side so you can see information in both windows at the same time?

You will learn to change the size of a window by *pointing* to the window border and *dragging* the border to the desired location.

1. If necessary, reactivate the windows **Explorer**.
2. If necessary, *restore* the *Explorer* window to a smaller sized window.
3. Point to the border at the right edge of the *Explorer* window until the pointer changes to a double-headed arrow (↔).
4. Hold down the left mouse button and *drag* the mouse to the left to bring the border in about one inch towards the center of the desktop, and release the mouse button.
 - The size of the window changes.
5. Point to the border at the bottom of the *Explorer* window until the pointer changes to a double-headed arrow (↔).
6. Hold down the left mouse button and *drag* the mouse upward on the pad to bring the border up about one-half inch, and release the mouse button.
7. Repeat this procedure to change the size of the *Recycle Bin* window.
8. Now try to drag the window borders to their original positions.

- It's okay if you are unable to make them exactly like they were previously.

You can change the size of a window in two directions at the same time. Positioning the double-headed arrow to one of the corners and dragging diagonally inward or outward enables you to change the size of two sides of a window simultaneously.

1. Point to the lower, right corner of the *Explorer* window until the pointer changes to the double-headed arrow, hold down the left mouse button, drag the border in and up towards the center of the window about one inch, and release the button.
2. Point to the lower, left corner of the *Recycle Bin* window until the pointer changes to the double-headed arrow, hold down the left mouse button, drag the border in and up towards the center of the window about one-half inch, and release the button.
3. Using the previous procedure, return the borders on the *Explorer* and *Recycle Bin* window to their approximate, original positions.

Arranging Windows

You can arrange the windows on your desktop by using commands available on the windows menu, or you can manually move your windows to a different location on the desktop.

Cascading. Cascading windows arranges open windows in an overlapping fashion on the Desktop so that the title bar from each window displays. Cascading is useful when you are working with a large number of open windows.

REMEMBER! You may substitute other windows if the window names in this textbook do not match the window names displayed on your screen but make sure that the windows are **open**.

1. Point to *Taskbar* at the bottom of the screen and *right-click*.

2. Move pointer up the menu to *cascade* and *left-click*.

- Momentarily, the windows display in a cascading fashion with the active window on top. The title bar for each window is visible.

Tiling. Tiling windows arranges open windows on the Desktop window like tiles on a floor or counter top. Tiling is useful when you want to see all program windows at the same time.

1. Point to *Taskbar* at the bottom of the screen and *right-click*.
2. Move pointer up the menu to tile. Here you have two possibilities. Tile vertically puts the windows side by side (vertically.) Tile horizontally puts the windows on top of the other,(stacks them horizontally.)
 - Momentarily, the windows display in a tiled fashion with the active window title bar displaying in a different color or a darker shade than the title bars on the other windows.

Moving. If you are not satisfied with Tiling or Cascading, you can move the windows to the location of your choice within the Desktop.

1. Move the mouse pointer to the word, *Recycle Bin*, in the title bar of the *Recycle Bin* window.
2. Hold down the left mouse button and *drag* the window so that it partially covers the *Explorer* window, and release the mouse button.
 - * The key to moving a window is to be sure the mouse pointer is pointing to the name on the title bar.
3. Use the same procedure to move the *Recycle Bin* window back to its previous location, remembering to point to the name in the title bar and drag.

4. Move the *Explorer* window to the center of the desktop by pointing to the title bar and dragging.
 5. Point to the *taskbar* at the bottom of the screen and *right-click*.
 6. Move pointer up the menu to *minimize all windows* and *left-click*. They are all back on the Taskbar now.
4. Watch the scroll box in the vertical scroll bar move as you click the up-pointing arrow in the vertical scroll box several times. (Be careful! Do NOT click the *Maximize* button. If you did, click the *Restore* button, and then repeat this step.)
 - The scroll box moves along the vertical scroll bar as you click the arrow. Pointing to the up-pointing arrow tells Windows XP that you want to see what is up above the window view.

Using the Scroll Bars

A scroll bar is a tool that allows the user to move the contents of a window up or down, left or right. This action is called *scrolling*. A window may have one, two, or no *scroll bars* displaying at a time. If a window has more information than can be displayed in the vertical direction of the window, a vertical *scroll bar* appears automatically. If a window has more information than can be displayed in the horizontal direction of the window, a horizontal *scroll bar* displays.

Refer to Figure 2.2 for the location of the vertical and horizontal scroll bars.

1. If necessary, *click* on the Explorer window to make it the active window.
2. Select *View*; verify that the **left** window selects the A: (A: drive), verify that either the large or small icons are checked and verify that the check mark displays in front of *Auto Arrange*. If it does not, select *Auto Arrange*.

If necessary, resize the window by moving the borders until the window is small enough for scroll bars to appear across the bottom and inside the right border.

3. Point to the down-pointing arrow on the vertical scroll bar and click the left mouse button several times to see the information in the window scroll up.

- * Pointing to the down-pointing arrow tells Windows XP that you want to see what is down below the window view.

5. Locate the scroll box in the horizontal scroll bar.
6. Click on the arrow at the opposite end of the scroll box in the horizontal scroll bar and watch the box move along the bar as the information in the window moves to the left or right.

You can scroll horizontally or vertically by dragging the scroll box along the bar.

1. Point to the scroll box in the vertical scroll bar, drag the box to the top or bottom of the bar, and release the button.
 - An outline of the box displays during the dragging process. When the mouse button is released, the actual scroll box replaces the outline.
2. Drag the scroll box in the horizontal scroll bar to one end of the bar.
 - Dragging to the left or right end of the horizontal scroll bar scrolls to the left or right edge of the window.
3. Practice using the scroll arrows and scroll bars until you can use them efficiently.
4. Double click on the Control-menu box at the left of the Explorer title bar. Again, this closes the Explorer window.
5. When you wish to quit, pick Start, (lower left corner of the screen.) Pick the Shut Down button from the fly-up menu. Make sure a click is made to “shut down windows”, and click the OK button to exit the Windows program.

6. If necessary, remove any floppy disks and turn off the power to the computer and the monitor.

Test your understanding of the concepts by completing the Checking Your Mastery items.

Checking Your Mastery

Circle T (True) or F (False) for each one of the following items. The correct answers may be found on the next page.

- T F 1. A window is a rectangular area that contains smaller windows and/or icons.
- T F 2. The desktop is the main surface in Windows XP that positions a pointer on the screen.
- T F 3. A picture that represents a program or a document is called an *icon*.
- T F 4. You can use the mouse pointer to select a command from a menu box or the menu bar.
- T F 5. You can exit the Windows XP program by selecting *Start Engine*, selecting *Shut Down* and selecting *Yes* to the Shut Down The Computer toggle.
- T F 6. The Minimize button is selected to help you minimize errors when changing the size of a window.
- T F 7. The Restore button is selected to restore the active window to its previous size.
- T F 8. Tiling windows arranges windows in an overlapping fashion enabling you to read the title bar of each window, but not allowing you to see the contents of each window.
- T F 9. The size of a window may be changed in two directions at once.
- T F 10. Scroll bars will not be active when all of the information in a window is in view.

Answers to Checking Your Answers

- | | |
|------------------|------------------|
| 1. T (p. 12) | 6. F (p. 20) |
| 2. T (p. 12, 13) | 7. T (p. 20, 21) |
| 3. T (p. 11, 12) | 8. F (p. 23) |
| 4. T (p. 14) | 9. T (p. 22) |
| 5. T (p. 25) | 10. T (p. 24) |

Folders

Objective: By the end of this exercise you will be understand the use of file folders.

Folders are like filing cabinet drawers. They are separate areas or containers that can hold many files. Folders can be made as a simple WINDOWS function, created through the Windows XP software. Regardless of how they are created, their function remains the same. CAD users not only place specific kinds of information within folders, but also many software's come divided into folders. It is crucial therefore, that a CAD user be able to create, remove, or move information from one folder to another.

The purpose of this book is to get you acquainted with folder characteristics through the use of Windows *graphical user interface* (GUI).

Let's get to the same point of departure. Remember, when starting a work session (or booting the machine), the machine should boot automatically into Windows XP. Our objective is to get to the Desktop screen. From there go through the **Start** engine, to **Programs**, to **Accessories** and then to *Windows Explorer*. We should now be ready to create some folders.

Creating Folders

Now, once in the Explorer, determine where you want the new *folder* to be made. You do this by selecting a drive letter in the left window (i.e., (A:), (C:), etc.). For our purposes here we will use the drive letter assigned to the USB flash drive.

1. Select the **USB flash drive** on which the folder will be created.
2. Pick ***File***.
 - Move to ***New***. Then move to ***Folder*** and *left-click*.
3. Automatically in the right side screen, a highlighted and blinking “**New Folder**” appears.
 - This blinking cursor is telling you that it’s ready to receive text.
4. Type in a name for your folder. (Type in your first name.)
5. After you’ve entered a name, **Pick your left mouse button** or tap the *Enter* key on your keyboard.
 - Note that your name now appears in the right-side Directory window. Also note that folders have no file extension; nothing appears after your first name. The shape of the folder icon is also different than the files icon. (If you have any files on your USB flash drive, compare the icons.)

Now let’s create another folder using your last name. Repeat steps 2 through 6 again. You should have two folders on your USB flash drive: now.

Selecting/Opening a folder

Selecting a folder is very easy. Simply move your cursor to the folder you want activated, and click. Any menu command will now be enacted upon this *selected drive*.

Opening a folder is as easy. This requires moving your pointer to the folder you want activated and *double-clicking* on the name of the folder. This will *open* another window that displays the contents of the folder you just clicked on.

1. *Double-click* (open) the folder of your last name.
 - A new Directory window appears. Notice the top label shows the letter of the drive, and the name of the folder. For our exercise, the folder is called your LASTNAME.
2. Pick the USB flash drive at the top of the (left) Directory window.
 - The Directory window disappears, returning you to viewing the drive window.

Copying Information Between Folders

Objective: By the end of this exercise you will be able to copy or move files from one folder to another.

Because CAD operators use folders to categorize their work, it is essential that you learn how to copy or move information from one folder to another.

Select the desired drive

Return to the Explorer dialog box.

1. Activate (open) the folder we created earlier that we called your *first name*.
 - A two-part Directory window appears. The left side of the Directory window indicates the name of the folder on the drive. The right side displays the name of the files within the folder.

2. Create a few new text files in the folder. Move to the File menu, move to NEW, left-click on “new text document”. Do this a few times to create multiple files. You do not have to name the files for this exercise.

Select the desired files

1. Depress the control key on the keyboard while picking the files you want to copy with your mouse. (Depressing the control key after selecting the first file allows you to select more than one file at a time.) Select the text documents that we created in the previous step.
2. After selecting the files you want to copy, pick **E**dit from the Explorer menu bar and pick the **C**opy option.
 - The files that you just selected have just been sent into memory (onto the clipboard) to be retrieved upon the very next paste.
3. Now select the *last name* folder that we made earlier in the chapter.
4. Once the *last name* folder is selected, pick **E**dit from the Explorer menu bar and pick the **P**aste option. New copies of the previously selected files will now be made into the last name folder.
 - You will be able to see the names of the selected files as the machine duplicates them under the *last name* folder.

Deleting Folders

If you remember that a folder is like a file drawer within a cabinet then it is easy to understand that you may want to empty the folder (drawer) when you no longer have use for the files stored there, you should also remove the unwanted folders. Deleting unused or unwanted folders saves space on the disk or hard disk and makes finding usable information much easier.

In the *Windows XP* environment, deleting a folder is very quick and very easy. This convenience, however, can be highly detri-

mental if you make an error or do not understand that all files within the folder are deleted and therefore destroyed when a folder is deleted.

Objective: By the end of this exercise you should be able to successfully delete a folder from your disk.

In the preceding pages, we created several new folders on the USB flash drive:. Let’s experiment with the **Delete** Folder process by using your LASTNAME folder.

1. Go to the Explorer window.
2. Make sure your **USB flash drive** is active.
3. Pick your **USB flash drive** in the left window.
4. Select the name of the folder you wish to delete (Pick your last name folder in the left column.)
 - Notice that the files on the right side of the dialog box change as you select different directories on the left side. All files displayed on the right side will be deleted and destroyed when you delete the folder they are in.
5. Once you have selected the appropriate folder (your LASTNAME for this exercise).
6. Pick **F**ile, then **D**elete...
 - A horizontal dialog box should appear. After picking the folder for deleting in the left window the right window should display *contents* of the folder as the title states.
7. Pick **Y**ES
 - This is a Confirm Folder Delete dialog box. This is the machines way of warning you that if YES, or YES TO ALL is picked, the folder displayed will be removed - - - along with all files in it.

8. Pick **YES** or **YES TO ALL**

- Notice that all of the file names in the folder are entered to the right of the folder. As the file names appear and quickly disappear, the files are deleted.

After the machine has finished deleting all files within the selected folder, the names of the folders are also deleted. If needed, reselect your **USB flash drive** icon at the top left of the Explorer and check what information remains on your **USB flash drive**. Your **LASTNAME** folder should be gone - - - along with any files that were in it.

Summary Question

Are you comfortable with creating new folders and copying files into them? Since you cannot remove files or folders that do not exist, you may wish to review this section several times until you can create, copy information into, and delete folders without fear of deleting and therefore destroying files and folders you want to keep.

CHAPTER 4

Geometric Construction

The construction of geometric shapes is the foundation of drafting. Drawings created whether complex or simple are made up of some variety of geometric forms. It is critical that a drafter has a solid understanding of the geometric construction to create new drawings and modify existing ones. There are two basic types of geometric forms two-dimensional (2D) and three-dimensional (3D). 2D forms are created with lines, squares, polygons, circles, triangles, ellipses, and other free forms. 3D forms are also created using the same forms as two-dimensional, but differ in that 3D forms real world view of depth, height, and width. See figure to see the common geometric forms and terminology.

Important Note:

Included in this chapter are step by step instructions to for students to create basic geometric objects using the A+CAD technology. To open A+CAD and follow the step by step instructions pick the Start button at the very lower left corner of your screen, then choose Programs, next click on The CAD Academy, and then A+CAD 2008. Alternatively, double-click the A+CAD 2008 icon on the desktop to open the program.

Basic Shapes and Terms


Lines

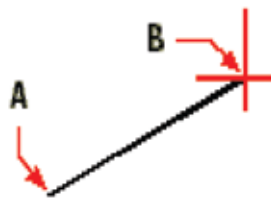
Straight lines: is the shortest distance between two points

Drawing lines

A line consists of two points: a start point and an endpoint. You can connect a series of lines, but each line segment is considered a separate line entity.

To draw a line

1. Do one of the following:
 - Choose Insert > Line.
 - On the Draw 2D toolbar, click the Line tool 
 - Type line and then press Enter.
2. Specify the start point.
3. Specify the endpoint.
4. In the prompt box, choose Done to complete the command.



Start point (A) and endpoint (B).


Curved lines: can be in the form of an arc, circle or any other irregular curve


There are many tools that are used to create curved lines. Two common ways other than an arc and a circle include freehand sketches and splines.

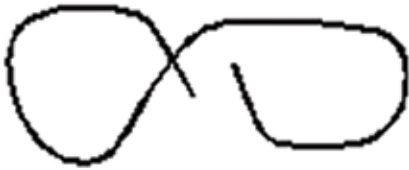
Creating freehand sketches

Before you begin creating a freehand sketch, you must set the length, or increment, of each segment. After you specify the length of the sketch segments, the crosshairs change to a Pencil tool. Your freehand sketch is not added to the drawing until you “write” the sketch into your drawing.

To create a freehand sketch:

1. Do one of the following:
 - Choose Insert > Freehand.
 - On the Draw 2D toolbar, click the Draw Freehand tool 
 - Type freehand and then press Enter

- Specify the length of the sketch segments.
- Click the mouse button to place the Pencil tool on the drawing to begin sketching 
- Move the pencil image to draw a temporary freehand sketch.
- Click the mouse button to lift the pencil up to stop sketching.
- In the prompt box, choose Write, Then Resume to write the temporary freehand sketch into the drawing.




A freehand sketch consists of individual line entities or a polyline.

Drawing splines

A spline is a smooth curve defined by a set of points. You can use splines to create sculptured shapes such as the cross section of a turbine blade or an airplane wing.

To draw a spline:

- Do one of the following:
 - Choose Insert > Spline.
 - On the Draw 2D toolbar, click the Spline tool  Type spline, and then press Enter.
- Specify the first point of the spline.
- Specify the second point of the spline.
- Specify as many more points as you want.
- When you have finished, press Enter.




Parallel lines: lines that maintain equal distance apart throughout the length

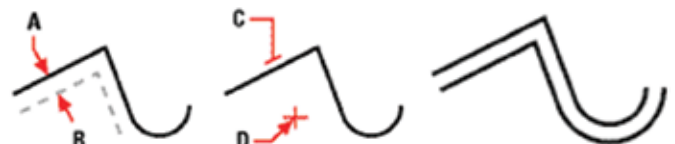
Making parallel copies

You can use the parallel feature to copy selected entities and align them parallel to the original entities at a specified distance. You can make parallel entities using arcs, circles, ellipses, elliptical arcs, lines, two-dimensional polylines, rays, and infinite lines.

Making parallel copies of curved entities creates larger or smaller curves, depending on which side of the original entity you place the copy. For example, placing a parallel copy of a circle outside the circle creates a larger concentric circle; positioning the copy inside the circle creates a smaller concentric circle.

To make a parallel copy by specifying the distance:

- Do one of the following:
 - Choose Modify > Parallel.
 - On the Modify toolbar, click the Parallel tool  Type parallel and then press Enter.
- Specify the distance by selecting two points or by entering a distance.
- Select the entity to copy.
- Specify on which side of the entity to place the parallel copy.
- Select another entity to copy, or press Enter to complete the command.



Result

To make a parallel copy, specify the distance between copies by entering a distance or selecting two points (A and B), select the entity to copy (C), and specify on which side to place the copy (D).

Intersecting lines: are lines that cross or meet at one and only one point

Intersection Snap tool

The Intersection Snap tool snaps to the actual intersection in three-dimensional space of any combination of entities. You can snap to the combination of an arc, circle, line, infinite line, polyline, ray, ellipse, elliptical arc, spline, hatch pattern, polygon mesh, or polyface mesh. You can also snap to an intersection point within a single entity, including a polyline or spline.

To set the Intersection Snap:

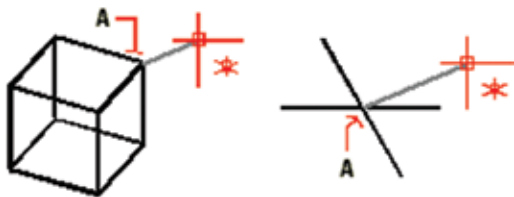
Do one of the following:

Choose Settings > Entity Snap > Intersection Snap.

On the Entity Snaps toolbar, click the Set Intersection Snap tool



Type intersection and then press Enter.



To snap to an intersection, select the intersection (A).

Point of intersection: a specific point at which two or more lines cross or meet

Perpendicular lines: where lines intersect at 90 degrees or a right angle

Perpendicular Snap tool

Use the Perpendicular Snap tool to snap to the perpendicular point of another entity. You can snap to an arc, circle, ellipse, line, polyline, infinite line, ray, spline, hatch pattern, or edge of a plane to form a perpendicular alignment with that entity or with an imaginary extension of that entity.

To set the Perpendicular Snap:

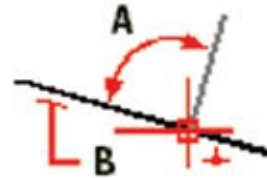
Do one of the following:

Choose Settings > Entity Snap > Perpendicular Snap.

On the Entity Snaps toolbar, click the Set Perpendicular Snap tool



Type perpendicular and then press Enter.



To form a perpendicular angle (A) to an entity (B), select anywhere on the entity.

Triangles

Equilateral triangle: all sides and angles are equal

Drawing polygons by side

The side polygon drawing method creates an equal-sided polygon defined by its center point and the distance to the midpoint of a side. You specify the number of sides, the center point, and the location of the midpoint of one side, which determines both the size and orientation of the polygon.

To draw a polygon by side:

1. Do one of the following:

Choose Insert > Polygon.

On the Draw 2D toolbar, click the Polygon, Center-Side tool



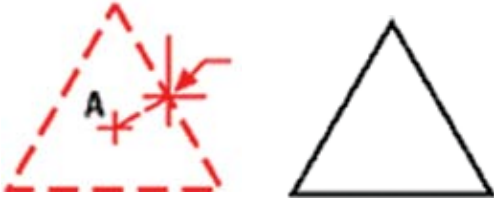
Type polygon, and then press Enter.

2. Type 3 to specify three sides for the polygon.

3. Specify the center of the polygon.

4. In the prompt box, choose Specify By Side, and then press Enter.

- Specify the midpoint of the side.



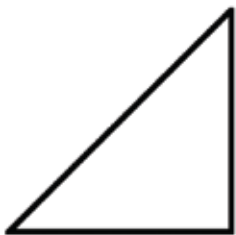
The center (A) and midpoint of one side (B).
Resulting polygon

Isosceles Triangle: one right angle and two equal sides

To draw an isosceles triangle:

- Choose Insert > Polyline
- Pick starting point: type: 0,0 (then press Enter)
- Press F8 to turn Ortho mode ON. This forces the crosshairs to move in vertical or horizontal directions only.
- Drag the mouse to the right (0 deg.) and type: 3 (then press Enter)
- Drag the mouse upward (90 deg) and type: 3 (then press Enter)
- Type: 0,0 (then Enter) to connect the third segment to the starting point.

Hit the Enter key to finish the command.



Resulting Isosceles Triangle

Right Triangle: has one angle of 90 degrees

- Hypotenuse: the opposite side of the 90 degree angle of the triangle
- Pythagoras' Theorem: a 3-4-5 right triangle where $a^2+b^2=c^2$

To draw a right triangle:

- Choose Insert > Polyline
- Pick starting point: type: 0,0 (then press Enter)
- Press F8 to turn Ortho mode ON. This forces the crosshairs to move in vertical or horizontal directions only.
- Drag the mouse to the right (0 deg.) and type: 3 (then press Enter)
- Drag the mouse upward (90 deg) and type: 4 (then press Enter)
- Type: 0,0 (then press Enter) to connect the third segment to the starting point.
- Press the Enter key to finish the command.



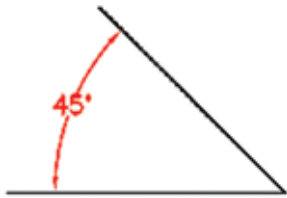
Resulting Right Triangle

Scalene triangle: one right angle with no equal sides or angles

1. Use the same steps as in the Right triangle.

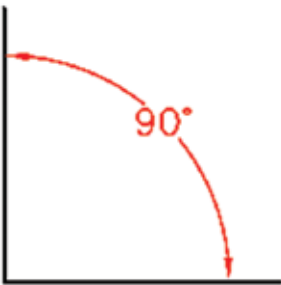
Angles

Acute angles: an angle that is smaller than 90 degrees



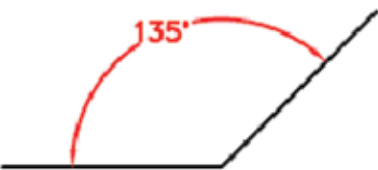
Acute Angle

Right angle: an angle that measures exactly 90 degrees



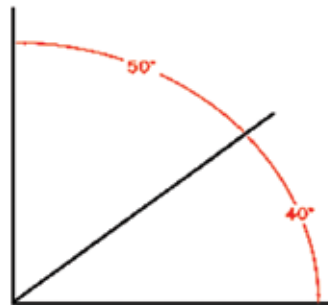
Right Angle

Obtuse angle: an angle that is larger than 90 degrees



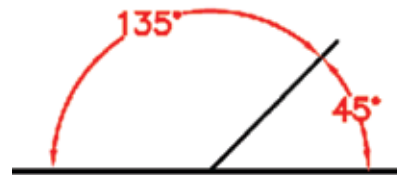
Obtuse Angle

Complementary angles: are angles that add up to 90 degrees or a right angle



Complementary Angles

Supplementary angles: are angles that add up to 180 degrees



Supplementary Angles

Creating Angles

Creating lines or polylines that are at specific angles can be achieved by using polar coordinates.


Using relative polar coordinates makes drawing a square tilted at a 45-degree angle a simple task. Polar coordinates base the location of a point on a distance and angle from either the origin (absolute coordinate) or from the previous point (relative coordinate).

To specify polar coordinates, type a distance and an angle, separated by the open angle bracket (<). For example, to use relative polar coordinates to specify a point 1 unit away from the previous point and at an angle of 45 degrees, type @1<45.

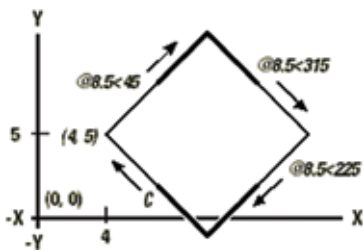
To draw the square tilted at a 45-degree angle:

1. Do one of the following:
 Choose Insert > Line, and then respond to the prompts as follows,

note: press the Enter key after each entry.

Start the Line command using the toolbar button 

2. Start of line: 4,5
3. Angle ? Length ? <End point>: @8.5<45
4. Angle ? Length ? Follow ? Undo ? <End point>: @8.5<315
5. Angle ? Length ? Follow ? Close ? Undo ? <End point>: @8.5<225
6. Angle ? Length ? Follow ? Close ? Undo ? <End point>: C
Drawing a tilted square using the relative polar coordinates method; enter C to close.









Resulting 45-degree tilted Square

Circles


Circle: a given set of points in a plane that are equal distant from a given point.

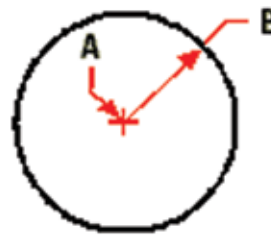
Drawing circles

The default method for drawing a circle is to specify a center point and radius. You can draw circles using any of the following methods:

- Center-Radius 
- Center-Diameter 
- Two points 
- Three points 
- Radius-Tangent-Tangent 
- Convert Arc to Circle 

To draw a circle by specifying its center and radius:

1. Do one of the following:
Choose Insert > Circle.
On the Draw 2D toolbar, click the Circle Center-Radius tool  Type circle and then press Enter.
2. Specify the center point.
3. Specify the radius of the circle.

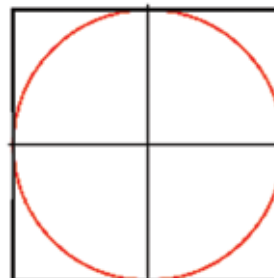


Center point (A) and radius (B).

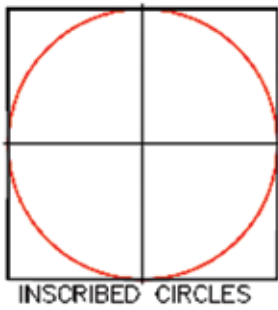
Concentric circles: are two or more circles that have a common center point



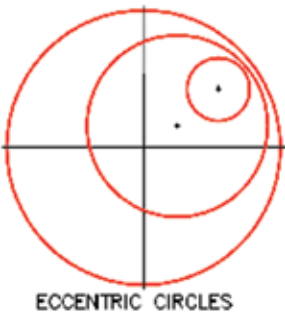
Inscribed circles: the largest circle that can be drawn inside a given figure



Circumscribed circles: a circle that passes through all the vertices of the figure



Eccentric circles: are two or more circles that do not have a common center point



Segment: the small portion of the circle that is separated by a chord

Chord: a line segment joining two points on a curve

Diameter: a straight line through the center of the circle connected to the circumference

Radius: the distance from the center of the circle to the edge or $\frac{1}{2}$ of the diameter

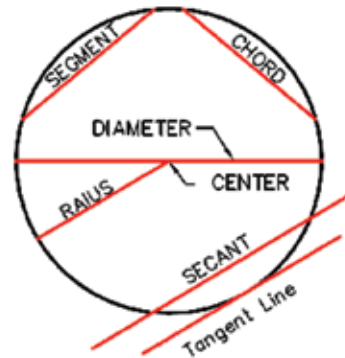
Circumference: the distance around the outside of the circle

Center: a point located in the exact center of the circle

Secant line: a line passing through two points of a curve

Tangent line: a line that touches a curve at one point, without cutting across it.

Vertex: a point where two or more points meet, also called vertices

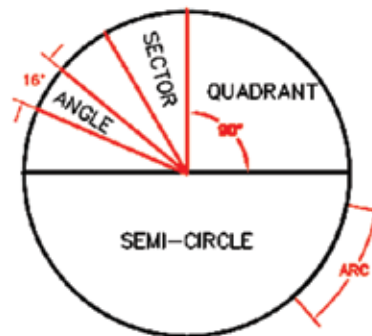


Sector: part of a circle or the area between two radii with connecting arc "slice of pie"

Arc: a part of the circumference of the circle

Quadrant: one quarter of a circle


Angle: two intersecting lines or line segments



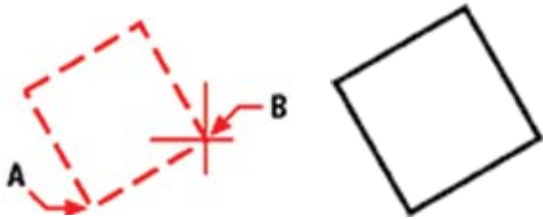
Regular polygons

Square: all sides are equal and internal angles are 90 degrees

To draw a square:

1. Do one of the following:
 Choose Insert > Rectangle.
 On the Draw 2D toolbar, click the Rectangle tool 
 Type rectangle, and then press Enter.
2. In the prompt box, choose Square.

- Specify one corner of the square.
- Specify the length of one side of the square by selecting its other end. The point you specify determines both the size and alignment of the square.




Endpoints of one side of the square (A and B). Resulting square

Rectangle: opposite sides are equal and internal angles are 90 degrees

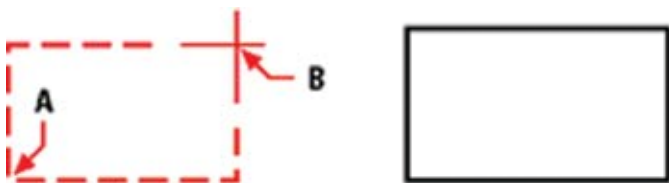
Drawing rectangles

With A+CAD, rectangles are closed polylines with four sides. You draw a rectangle by specifying its opposite corners. The rectangle is normally aligned parallel to the current snap and grid alignment, but you can use the Rotated option to align the rectangle to any angle.

To draw a rectangle:


- Do one of the following:
 Choose Insert > Rectangle.
 On the Draw 2D toolbar, click the Rectangle tool 
 Type rectangle, and then press Enter.


- Specify one corner of the rectangle.
- Specify the opposite corner of the rectangle.




Opposite corners (A and B).

Resulting rectangle.

You can edit each side of a rectangle individually using the Edit Polyline tool on the Modify toolbar. 

You can convert the sides into individual line entities using the Explode tool on the Modify toolbar. 

You can control whether wide rectangle lines are shown filled or as outlines using the Fill tool on the Settings toolbar. 

You can also use the Rectangle tool to draw a square. Instead of specifying the opposite corners, you specify the length of one side and the alignment of the square.

Rhombus: all sides are equal

Rhomboid: opposite sides are equal

Trapezoid: two sides are parallel

Trapezium: no sides are parallel

Pentagon: five equal sides and equal angles




Hexagon: six equal sides and equal angles

Heptagon: seven equal sides and equal angles

Octagon: eight equal sides and equal angles


Drawing polygons

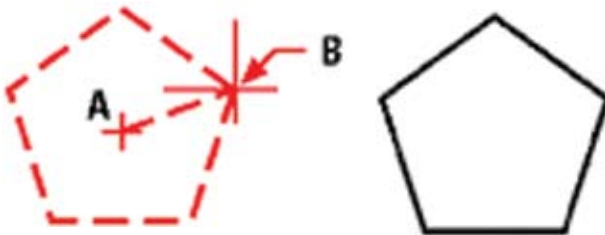
Polygons are closed polylines comprised of a minimum of three and a maximum of 1,024 equal-length sides. The default method for drawing a polygon is to specify the center of the polygon and the distance from the center to each vertex. You can draw a polygon using any of the following methods:

- Center - by vertex on the Draw 2D toolbar 
- Center - by side on the Draw 2D toolbar 
- Edge on the Draw 2D toolbar 

The vertex polygon drawing method creates an equal-sided polygon defined by its center point and the distance to its vertices. You specify the number of sides, the center point, and the location of one vertex, which determines both the size and orientation of the polygon.

To draw a polygon by vertex:

- Do one of the following:
Choose Insert > Polygon.
On the Draw 2D toolbar, click the Polygon, Center-Vertex tool 
Type polygon, and then press Enter.
- Type 5 (Pentagon) to specify five sides for the polygon.
- Specify the center of the polygon.
- Specify the vertex of the polygon.



The center (A) and vertex (B). Resulting polygon

3D Forms


Sphere: a three dimensional object shaped like a ball

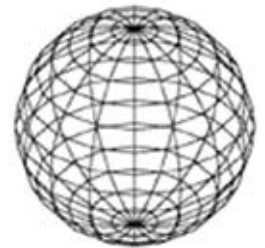
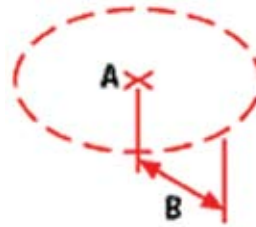
Creating spheres

You can create a sphere. The latitude lines of a sphere are always parallel to the xy plane of the current UCS; the central axis is always parallel to the z-axis. You position a sphere by specifying its center point. You determine the size of a sphere by specifying either its radius or its diameter.

NOTE The sphere is created as a three-dimensional ACIS solid.

To create a sphere:


- Do one of the following:
Choose Insert > 3D Entities > Sphere.
On the Solids toolbar, click the Sphere tool 
Type sphere and then press Enter.
- Specify the center of the sphere.
- Specify the radius or diameter.



Center (A) and radius (B) of the sphere Resulting Sphere

To create a sphere as a three-dimensional surface:

To create a sphere as a surface instead of as a solid, the procedure is similar.

- Do one of the following:
On the Draw 3D toolbar, click the Sphere tool 
Type a sphere and then press Enter.
- Specify the center of the sphere.
- Specify the radius or diameter.
- Specify the number of longitudinal sections that are perpendicular to the xy plane.

- Specify the number of latitudinal sections that are parallel to the xy plane.


Cube: contains six equal square plane faces, shaped like a dice

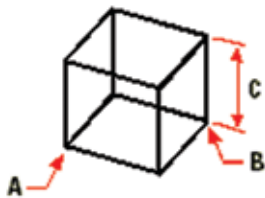
Creating boxes

You can create a rectangular box, or cube. A box consists of six rectangular surface planes. The base of the box is always parallel to the xy plane of the current UCS. You position the box by specifying either a corner or the center of the box. You determine the size of the box by either specifying a second corner and the height; defining the box to be a cube and then providing its length; or specifying the length, width, and height.

NOTE The box is created as a three-dimensional ACIS solid.

To create a box:


- Do one of the following:
 - Choose Insert > 3D Entities > Box.
 - On the Solids toolbar, click the Box tool  Type box and then press Enter.
- Specify the first corner of the base.
- Specify the opposite corner of the base.
- Specify the height.



First corner of the base (A), the opposite corner of the base (B), and the height (C).

To create a box as a three-dimensional surface:

To create a box as a surface instead of as a solid, the procedure is similar.

- Do one of the following:
 - On the Draw 3D toolbar, click the Box tool  Type a box and then press Enter.
- Specify the first corner of the base.
- Specify the opposite corner of the base.
- Specify the height.


Cylinder: has straight sides and circular ends of equal size, shaped like a tube

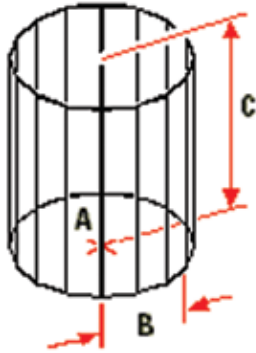
Creating cylinders

You can create a cylinder defined by a circular base. The base of a cylinder is always parallel to the xy plane of the current UCS; the height of a cylinder is always parallel to the z-axis. You position a cylinder by specifying the center of the base. You determine the size of a cylinder by specifying either the radius or diameter of the base and the height.

NOTE The cylinder is created as a three-dimensional ACIS solid.

To create a cylinder:

- Do one of the following:
 - Choose Insert > 3D Entities > Cylinder.
 - On the Solids toolbar, click the Cylinder tool  Type cylinder and then press Enter.
- Specify the center of the base of the cylinder.
- Specify the radius or diameter.
- Specify the height.




Center of the base (A), the radius of the base (B), and the height (C).

To create a cylinder as a three-dimensional surface:

To create a cylinder as a surface instead of as a solid, the procedure is similar.

1. Do one of the following:

On the Draw 3D toolbar, click the Cylinder tool  Type a cylinder and then press Enter.

2. Specify the center of the base of the cylinder.
3. Specify the radius or diameter.
4. Specify the height.


Pyramid: has triangular sides that slope to meet in a point

Creating pyramids

You can create a tetrahedron (three-sided pyramid) or a four-sided pyramid. The sides of the resulting pyramid can meet at a point (the apex) or can form a three- or four-edged top. The sides of a four-sided pyramid can also meet along a ridge defined by two points. The base of the pyramid is always parallel to the xy plane of the current UCS. You position the pyramid by specifying a corner of the base. You determine the size of the pyramid by specifying the base points and either the apex, the corners of the top surface, or the endpoints of the ridge.

NOTE The tetrahedron and pyramid are created as three-dimensional ACIS solids.

To create a tetrahedron:


1. Do one of the following:
Choose Insert > 3D Entities > Pyramid.
On the Solids toolbar, click the Pyramid tool  Type pyramid and then press Enter.
2. Specify the first point for the base of the pyramid.
3. Specify the second and third points.
4. In the prompt box, choose Tetrahedron.
5. Specify the apex of the tetrahedron.



The first point (A), second point (B), and third point (C) of the base, and the apex (D).

To create a pyramid as a three-dimensional surface:

To create a pyramid as a surface instead of as a solid, the procedure is similar.

1. Do one of the following:
On the Draw 3D toolbar, click the Pyramid tool  Type a pyramid and then press Enter.
2. Specify the points as described previously.


Cone: has a circular base that tapers to a point perpendicular to the base

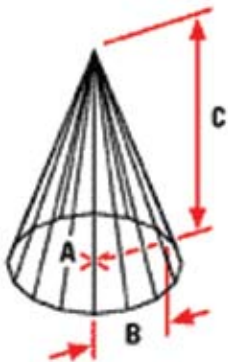
Creating cones

You can create a three-dimensional cone defined by a circular base and tapering to a point perpendicular to the base. The base of the cone is always parallel to the xy plane of the current UCS; the height of the cone is always parallel to the z-axis. You position the cone by specifying the center of the base. You determine the size of the cone by specifying either the radius or the diameter of the base and the height.

NOTE The cone is created as a three-dimensional ACIS solid.


To create a cone:

1. Do one of the following:
 - Choose Insert > 3D Entities > Cone.
 - On the Solids toolbar, click the Cone tool 
 - Type cone and then press Enter.
2. Specify the center of the base of the cone.
3. Specify the radius or diameter.
4. Specify the height.



Center of the base (A), radius of the base (B), and the height (C).

To create a cone as a three-dimensional surface:

1. Do one of the following:
 - On the Draw 3D toolbar, click the Cone tool 
 - Type a cone and then press Enter.
2. Specify the center of the base of the cone.
3. Specify the radius or diameter.
4. Specify the height.


Torus: three-dimensional donut or ring shaped entity

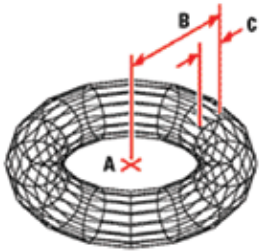
Creating Torus

You can create a three-dimensional donut or ring-shaped entity known as a torus. The diameter of a ring is always parallel to the xy plane of the current UCS. A torus is constructed by revolving a circle about a line drawn in the plane of the circle and parallel to the z-axis of the current UCS. You position a torus by specifying its center point. You determine the size of a torus by specifying its overall diameter or radius and the diameter or radius of the tube (the circle being revolved).

NOTE The torus is created as a three-dimensional ACIS solid.


To create a torus:

1. Do one of the following:
 - Choose Insert > 3D Entities > Torus.
 - On the Solids toolbar, click the Torus tool 
 - Type torus and then press Enter.
2. Specify the center of the whole torus.
3. Specify the radius or diameter of the whole torus.
4. Specify the radius or diameter of the body of the torus.



Center (A) and radius of the whole torus (B), and the radius of the body (C).

To create a torus as a three-dimensional surface:

- Do one of the following:
To create a torus as a surface instead of as a solid, the procedure is similar.
On the Draw 3D toolbar, click the Torus tool 
Type a torus and then press Enter.
- Specify the center of the whole torus.
- Specify the radius or diameter of the whole torus.
- Specify the radius or diameter of the body of the torus.
- Specify the number of longitudinal sections that are perpendicular to the xy plane.
- Specify the number of latitudinal sections that are parallel to the xy plane.


Applying Basic Geometric Construction

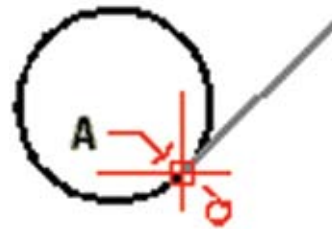
Tangent line:

Tangent Snap tool

Use the Tangent Snap tool to snap to the tangent point of another entity. You can snap to the point on an arc, ellipse, spline, or circle that, when connected to the previous point, forms a line tangent to that entity.

To set the Tangent Snap:


- Do one of the following:
Choose Settings > Entity Snap > Tangent Snap.
On the Entity Snaps toolbar, click the Set Tangent Snap tool 
Type tangent and then press Enter.

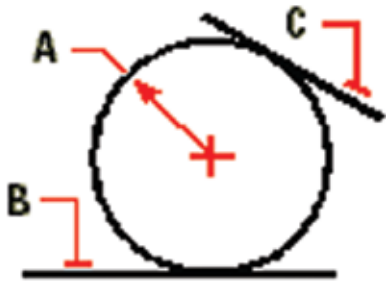


To snap to a tangent, select the entity near the tangent point (A).

Tangent arc: an arc that touches another curve or line at one point

To draw a circle tangent to existing entities:

- Do one of the following:
Choose Insert > Circle.
On the Draw 2D toolbar, click the Circle  Radius-Tangent tool
Go to step 3.
Type circle and then press Enter.
- In the prompt box, choose Radius-Tangent-Tangent.
- Specify the radius of the circle.
- Select the first entity to which to draw the circle tangent.
- Select the second entity to which to draw the circle tangent.



Radius of circle (A) and tangent lines (B) and (C). Similar process to create an arc tangent.

CHAPTER 5

Principles of Design & Documentation

In the drafting and design field, a person will encounter many opportunities to design various components. This can range from designing hardware such as fasteners and clamps, designing fixtures and jigs to support the manufacture of an item, design product transportation systems like conveyer belts that use small pallets to deliver product to various manufacturing steps, to designing tools used in a machine shop to fabricate complex shapes and features.

Design Process

The design process is an integration of sketching, creativity, analysis, and problem solving. With the use of CAD and the internet designers are better able to communicate their ideas and collaborate with their customer to produce a product faster, cheaper, and better than ever before. The power of our emerging technology is also changing the way we do business. Once, one company would produce all their own parts to form an assembly. Now we see parts being produced globally, and assembled globally, to further advance our lives and our comfort at a decreasing cost. Combining the design process with the globalization environment introduces new considerations for engineering and production.

- Federal and state regulations
- Environmental impacts
- Demographics
- Consumer trends
- Cultural trends
- Economic value

These changes also influence a designer's required knowledge base. Traditional training involved learning about such items as: design fundamentals, capabilities of tools and equipment, manufacturing processes, knowledge of materials, and general math. Today, a designer needs additional knowledge with the concepts of team functions, efficient manufacturing, data acquisition, packaging, safety and error-proofing, computer skills, and oral presentation skills.

The design process involves a sequence of stages including problem identification, forming preliminary ideas, conducting analysis, refinement, and implementation. **Stage one**, problem identification begins by identifying a need. What does the customer want? Where and how will this be used? Problem identification includes doing some research. Is a solution already available? Is it cost-effective to design something new? Perhaps an existing item can be modified to meet that need without having to “re-invent the wheel”. The **second stage** of the design process is to gather some preliminary ideas. Start by generating several options in the form of sketches. It may be necessary to work with a team to brainstorm a variety of solutions depending on the complexity of the need. It is also beneficial to gather the ideas and opinions of your customer.

The **third stage** of the design process involves analyzing your solution. This can be accomplished by creating a detailed design of the solution. One suggested analysis method includes creating a 3-D solid model of the solution which can have kinetics applied to its parts moving them through their range of motion to look for problems. Another method is to create a rapid prototype of the design from the 3-D solid model. This prototype could be made from liquid resin which is heated by a laser layer-by-layer to form a physical model of the design. This prototype is now a physical part that you can see, touch, and use in a design review with the customer.

The **fourth stage** is refinement of the design. Does the design meet the customer's need? This step includes re-evaluating the design if necessary to see if the design can be made more efficiently. Decisions are made to ensure the solution is designed for manufacturability. Does the design include parts off-the-shelf instead of machining new parts? It is less expensive to modify an existing item than it is to make a part completely from scratch.

The **final stage** of the design process is implementation. This includes the drafters complete working drawings of the final design; getting the customer's approval on the design; sending the working drawings out for manufacturing cost quotes; getting the parts manufactured per the design; writing a procedural manual (if necessary) explaining how to assemble, use, and maintain the parts; and documenting any design changes that may have occurred during manufacture.

Now that we have covered the design process, let's move into the mechanics of documenting a design. There are five major areas that will be covered: sketching; formats and the title block; drawing notes, drafting standards; and terminology and abbreviations.



Sketching

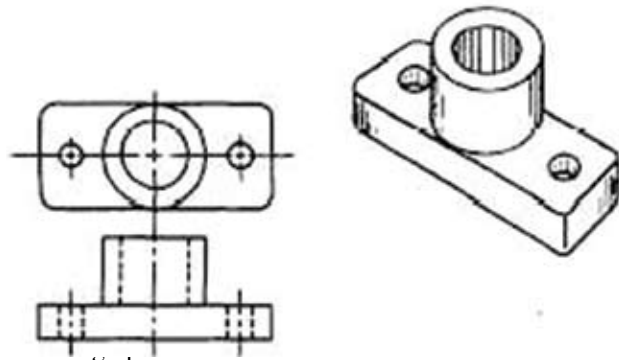
Sketches are an important communication tool. Sketching, or free-hand drawing with pencil and paper, is a way to visually express new ideas to solve design problems. Initial generation of ideas using a computer is uncommon because sketching can be done virtually anywhere: at your desk, in the lunch room, or during a conference meeting when a team is discussing ideas. Sketching also allows the drafter to generate many options very rapidly. These rough sketches are then refined by adding some basic dimensions and other information that will later be transferred into a computer aided design (CAD) drawing.

Sketching before drawing is essential. It has been documented that drawing before making a list of sketched alternatives will result in poorly conceived products. The reason behind this is simply based on customer expectations. A CAD drawing implies a more advanced state of planning which will intimidate the customer from making suggestions for change. The customer will be afraid to make suggestions because it appears that the designer has already invested too much time in the analysis stage of the design process to make any changes. This can result in a substandard design and poor product satisfaction during the implementation stage of design. Whereas, a sketch implies an early state of planning and design. This invites input and participation on the part of the customer. The resulting product is more likely to be appreciated and understood by the customer.

Sketching is a free-wheeling type of informal drawing that helps inspire the basic design of an object before advancing to the more exacting scaled drawings. The essence of sketching is simple: draw whatever pops into your mind, and revise as you go until you find a design you like best. With practice you will find sketching to be fun. Below are some sketches of a table that show the refinement steps used to achieve the final result.

Sketches can be done in a couple of formats. The orthographic method creates 2-D views of your concept. This is usually the simplest format to start with when learning to sketch. Draw a front view and any other views necessary to describe your idea. The sketch does not need to be drawn at any particular scale. However, with practice and experience you will find that drawing additional features to the correct proportion of your general outline will aid in visualizing the true concept. The second, and most widely used method, is the isometric method which allows you to visualize your idea from multiple sides with only one drawing. With this method, create a 3-D view of your idea.

Which method you choose depends on your level of comfort and experience. It is more important to decide the method that best conveys your ideas in which you can present them to your customer as solutions to their needs.



Sketching Exercise:

Using pencil and paper, sketch some possible solutions to the following customer's request.

The customer would like a student designer to develop a device that will provide light as it is carried in one hand.

Requirements:

- 1) Small enough to carry in one hand

- 2) Able to be carried for extended periods of time and distance
- 3) Simple to operate and affordable to purchase

Student Designer: Interpret what the customer has asked for along with the stated requirements. Develop two possible solutions by sketching ideas on one or more pieces of paper. Use either sketching method. Submit the two sketches to your instructor for review.

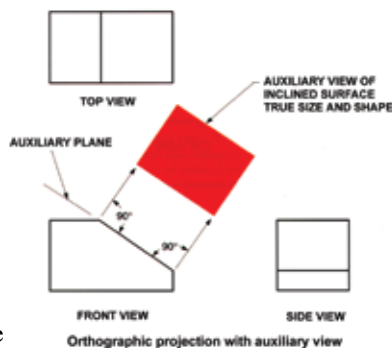
Terminology and Abbreviations

The following definitions represent the common terms and features designers will encounter during design and product manufacturing. The list of abbreviations that follow are those used as standard short-cuts in spelling out a word or phrase. Since there is an overabundance of terms and abbreviations known to exist, those listed here are most likely to be used by apprentices.

Terminology:

Angular dimension

- Measured dimensions represented by Degrees, Minutes, and Seconds



Auxiliary view

- Orthographic view projected off a plane showing true size and shape
- Shows features appearing on inclined surfaces
- Shows features appearing on oblique surfaces

Axis

- Central line about which parts are arranged
- Elements may or may not revolve around it

Baseline dimensioning

- System of dimensioning used to locate features of a part
- Uses a common set of datum's

Basic dimension

- Exact value of a feature

- Describes the following: Size and Location

Bilateral

- "Bi" means two
- Indicates that two sides are used. (E.g.: bilateral tolerances)

Blind hole

- Hole that does not go through all of the material
- Common in thick parts

Burr

- Jagged edge of metal produced by working the metal
- Common note of drawing is to "REMOVE ALL BURRS"

CAD drawing

- Drawing created by computer-aided drafting methods
- Electronically stored, revised, and retrieved

CAM

- Computer Aided Manufacturing
- Software used to convert CAD drawings into machine code for use in fabricating a part

Casting

- Metal object made by pouring molten metal into a mold. The molded, or cast, part is usually made from iron or aluminum

Chamfer

- Corner removed from end of cylindrical surface
- At an angle to face
- Used to facilitate assembly

Counterbore (v)

- Process to enlarge the end of a hole cylindrically
- Dimension specifications include diameter and depth of enlarged section

Countersink (w)

- Process to enlarge the end of a hole conically
- Dimension specifications include diameter and included angle

FUNDAMENTALS OF DRAFTING

Datum

- A reference point used to establish location of other features
- A datum can be: A point, a line, a surface, or a plane

Degree

- Unit of angular measurement

Depth (x)

- Used to show the distance of a feature that does not go through all of the material

Fastener

- Connector used to secure two or more parts together
- Examples: Bolts, Nuts, Screws, Rivets, etc.

Feature

- The term used to describe the form or shape of an object that has been added to a basic shape
- Examples: A notch, a hole, a tapped hole, an angle, a slot, etc.

Fillet

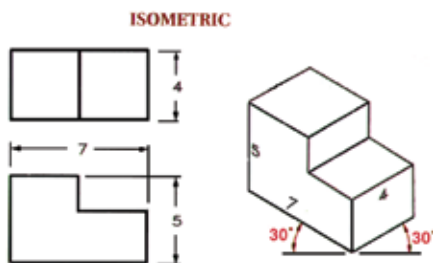
- Concave surface (interior radius)
- At intersection of two surfaces of an object

Fixture

- Device designed to position and hold a part in a machine tool
- Does not guide cutting tool
- References part to tool

Isometric drawing

- Three-dimensional pictorial drawing
- Horizontal surfaces on 30° axes from horizontal



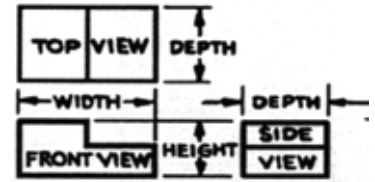
Jig

- Device used to hold a part to be machined
- Positions cutting tool

- Guides cutting tool

Orthographic projection

- Process of projecting essential views of three-dimensional object on a two-dimensional plane



Parallel

- Extending in the same direction (e.g.: two lines)
- Equidistant at any point
- Lines do not meet

Perpendicular

- Projecting at a right angle (90°) to a given line or plane

Radius (R)

- Distance from center of circle, or an arc, to its circumference
- Equal to one half of a diameter
- Radii is the plural form of radius

Reference dimension

- Used only for information purposes
- Does not receive standard print tolerances
- Should not be used for production or inspection

Scale

- Refers to relative size of drawing and size of part
- First number represents size of drawing
- Second number represents size of part
- Examples: 2:1 means the part is being shown twice as large as the full size part; 1:2 (half scale) means the part is being shown at one half of its full size.

Section

- Interior view of an object
- Drawn to expose features not otherwise visible

Symmetrical

- Same on each side
- Equal halves
- Centerline usually indicates symmetry

Tapped hole

- A feature applied to a part that includes the cutting (tapping) of threads

Tolerance

- Permissible variation from a given dimension
- Unilateral, bilateral

Abbreviations:

- ALUM = Aluminum
- BC or BS = Bolt Circle or Bolt Spacing
(Used interchangeably)
- CI = Cast Iron
- CRS = Cold Rolled Steel
- CBORE = Counterbore (obsolete)
- CSK = Countersink (obsolete)
- DIA = Diameter (obsolete)
- DP = Depth (obsolete)
- DR = Drill or Drill Rod
- ECN = Engineering Change Notice
- ECO = Engineering Change Order
- EQ = Equal or Equally
- FAO = Finish All Over
- FIL = Fillet (may also be a Round) (obsolete)
- GA = Gage (for sheet metal)
- HRS = Hot Rolled Steel
- IAW = In Accordance With
- ID = Inside Diameter
- LH = Left Hand
- MATL = Material
- MAX = Maximum
- MIN = Minimum
- OD = Outside Diameter
- R = Radius
- REF = Reference (obsolete)
- SECT = Section
- SP = Spaced
- SST = Stainless Steel
- STK = Stock
- STL = Steel
- THD = Thread
- THK = Thick or Thickness

- TOL = Tolerance
- TYP = Typical

Terminology and Abbreviations Exercise:

QUIZ:

1. Give the term which means “Software used to convert CAD drawings into machine code for use in fabricating a part.”
2. Give the term which means “Central line about which parts are arranged.”
3. Give the term which means “Connector used to secure two or more parts together.”
4. Give the term which means “A feature applied to a part that includes the cutting of threads.”
5. Write the abbreviation for the following items: Aluminum; Material; Radius; Stainless Steel; and Typical

Formats and the Title Block

Drawings made for mechanical design are known as technical drawings. There are various elements of a technical drawing that have relative importance and relationship to each other. Every drawing should contain select information including the *format*, the *size*, the *title* of the part, the assigned *part number*, the relative *scale* of the drawing, the *design* and *drafting approval* data, the *date* in which the drawing was produced, and *general notes*. The method of presenting and locating this information on a drawing varies from one industry to another, yet most methods derive

FUNDAMENTALS OF DRAFTING

from the American National Standards Institute and the American Society of Mechanical Engineers (ANSI/AMSE Y14.1-1995 DRAWING SHEET AND FORMAT STANDARDS) specification.

The format size and orientation chosen depends on the size and complexity of the object designed. This may further be influenced by the standards set forth by the customer, or your company. For instance, some companies prefer to use one dedicated format size for all drawings to keep them consistent from one design to the next.

There are ten format sizes available in which the last four special sizes are not often used. Of these ten sizes, the first four are the most commonly used formats (A– D).



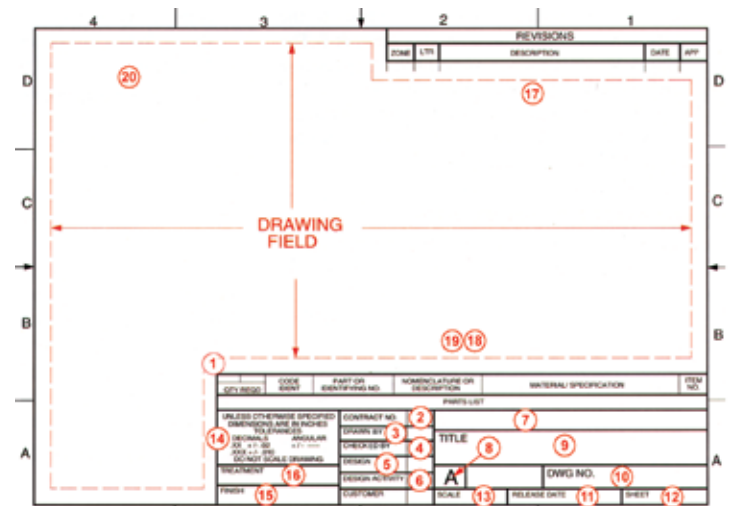
- A – Size = 8-1/2 x 11 (landscape or portrait orientation)
- B – Size = 11 x 17 (landscape or portrait orientation)
- C – Size = 17 x 22 (landscape orientation)
- D – Size = 22 x 34 (landscape orientation)
- E – Size = 34 x 44 (landscape orientation)
- F through J are available although not commonly used.

The components that make up the drawing format include:

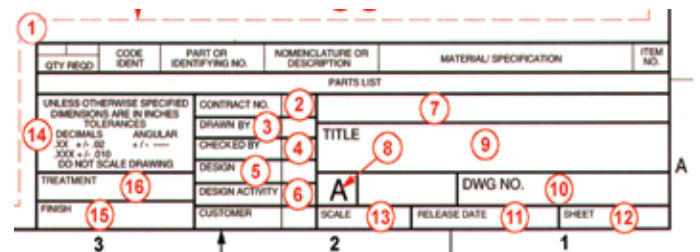
- Drawing field
- Title block location and contents
- Revision block location
- Parts listing or the Bill of Materials (BOM)
- Zoning

Every drawing has a **drawing field**. This is the main body of the drawing and is used to draw the object and all necessary views. This area will also include the related dimensions and notes to

complete the drawing. There is an imaginary border of approximately 1/2 inch (shown by the dashed lines) towards the inside to prevent the drawing information from overflowing past the bordered format.



Typical ANSI/AMSE Y14.1-1995 Drawing Format



Typical Drawing Title Block

1. **The Title Block** - is located in the lower-right area of the drawing format and includes areas for specific information. Most companies use preprinted sheets with the company's name, address, logo and borders as well as other pertinent information related to its particular business. Title block sizes and the type of information required will vary depending on the size of the company and the types of products it manufactures. Some companies require basic information in the block, while others, especially those under government contract, need to provide more data.
2. **CONTRACT NO. block** - The contract number is entered

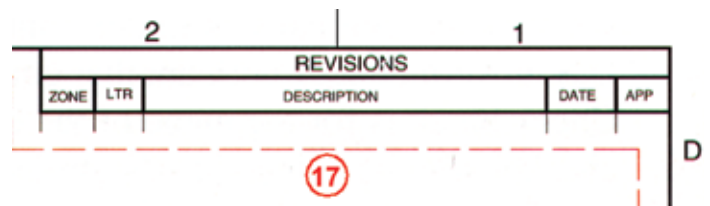
here only when appropriate or required.

3. **DRAWN BY block** - The drafter's name appears here in printed form. In addition, the day, month and year the drawing was completed is usually shown.
4. **CHECKED BY block** - This is the name (usually a signature) of the person who is assigned to check the drawing for accuracy and completeness.
5. **DESIGN block** - This is the name (usually a signature) of the responsible engineer.
6. **DESIGN ACTIVITY block** - This is the name (usually a signature) of the program manager or chief engineer.
7. **Company Name and Address block** - This block is for the company name, address, and trademark or logo.
8. **Format Size block** - This block contains a letter indicating the appropriate size of the drawing format.
9. **TITLE block** - This is where the drawing's name or title for the part or assembly is placed.
10. **DRAWING (DWG) NO. block** - The drawing's number is inserted here. Every company has some form of part-numbering system.
11. **RELEASE DATE block** - This block is for the date that the drawing is released for manufacturing. It does not change when the drawing is revised. It reflects the original information.
12. **SHEET block** - This block contains the number of sheets in a drawing set. (E.g.: 1 of 1; 1 of 2; 2 of 2; etc.)
13. **SCALE block** - This block indicates the scale of the part in the drawing. (E.g.: 1:1, FULL, 2:1, 4:1, 1:2; etc.)
14. **Standard Drawing Tolerance block** - This gives the

drawing's default tolerances for dimensions unless otherwise specified.

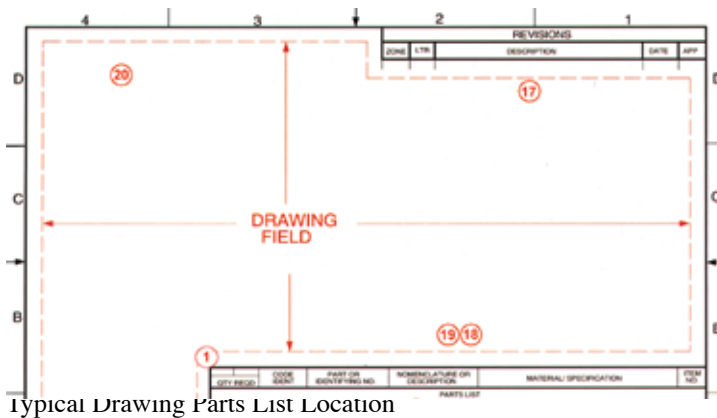
****As a side note**:** This information is usually established by the company based on their requirements or industries standards. This information is not usually changed from drawing to drawing, hence the title Standard, and instead any global tolerance changes should be noted in the general notes.

15. **FINISH block** - This block specifies the type of surface coating or process to be applied.
16. **TREATMENT block** - This block notes the type of material treatment to be applied. (E.g.: Heat Treat, Rockwell hardness, temper, etc.)



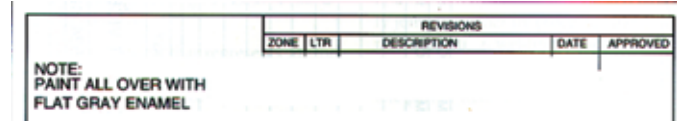
Typical Drawing Revision Block

17. **Revision area** - Most drawings require one or more revisions during their lifetime. Revisions can be in the form of a slight change in shape, dimensions, text, etc. due to changes in design, changes in a customer's requirements, or to correct errors in design or drafting. To perform an engineering change to a drawing, an authorized person must write a formal document stating all the things that are to be changed. This document is an **Engineering Change Notice (or ECN) or Engineering Change Order (ECO)**. To record engineering changes on drawings, a **revision block**, is established. According to ANSI specification, the revision block is located in the upper-right corner of the drawing and provides space for a zone location, revision letter, description of the change, date, and approval for the change. The ECN/ECO number is usually also noted.



Typical Drawing Parts List Location

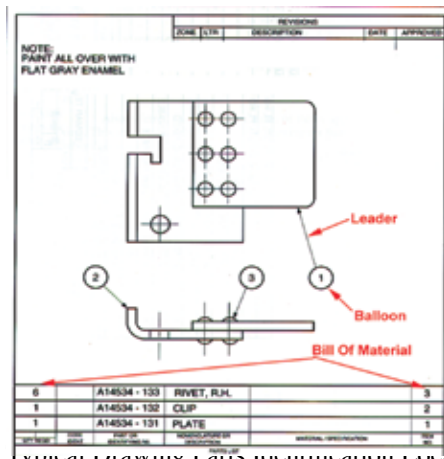
inch (1/2") text, however they can vary in size from .31 (5/16") to .75 inch (3/4") in diameter, depending on the size and scale of the assembly drawing.



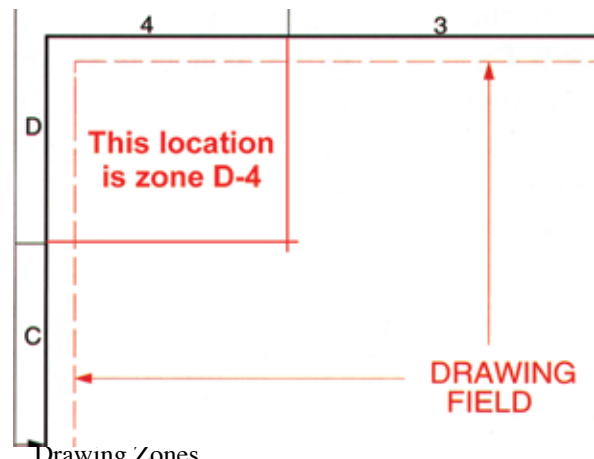
Typical General Notes Location

18. **Parts List** - The parts list is located in the lower-right corner of the drawing, directly above the title block. The list indicates the items, materials, and quantities required to complete the part or assembly. It is referred to as the *Bill of Materials (BOM)*. The parts are normally arranged with "Make" items listed first followed by "Buy" (or purchased) items. Items are usually listed in an ascending numerical order.

20. **General Notes** - Usually located in the upper left corner of the drawing field, or occasionally in the lower left corner depending on the company standards. These are used to indicate general manufacturing information. See the next subsection for more detail.



Typical Drawing Parts Identification Location (BOM)



Drawing Zones

19. **Parts Identification** - An item number is required for each part shown on an assembly. This ties the item to the parts list (18). The method for keying the two is through the use of balloon numbers, which consist of identification numbers placed inside circles called *Item Balloons*. Each item balloon is attached to a *Leader*. The leader is a thin line that ends in an arrowhead that points to and touches the related part in the drawing field. The leader is almost *never* shown as either a horizontal or a vertical line (always at an angle) to avoid confusion with other lines on the drawing. The item balloon should be placed horizontally or vertically aligned, and in numerical sequence when possible. Item balloons are usually .625 inch (5/8") in diameter with .50

Drawing Zones - The drawing field is divided into zones. These zones are marked by letters along the sides and numbers along the top and bottom. Zones are used to identify the location an item on a drawing and are usually specified in the Letter-Number call out. Zones can be used to identify the location of a drawing change, a displaced detail or section drawing, or to locate item balloons for parts listed in the BOM.

Formats and the Title Block Exercise:

QUIZ:

1. List the five components of a drawing format.
2. Identify the four most commonly used formats and their related sizes.
3. The Drawing Field is divided into sections called: _____
4. Where are the General Notes usually located in a drawing field?
5. Identify the four most commonly used formats and their related sizes.

NOTES(UNLESS OTHERWISE SPECIFIED):

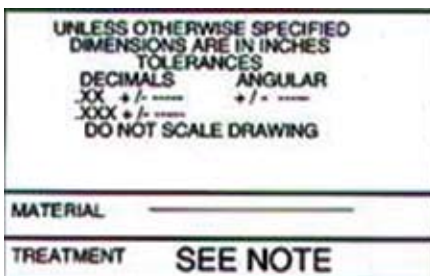
1. MATERIAL: ALUMINUM 6061 T4-T6.
2. FINISH: HARD CLEAR ANODIZE .0005 THICK. ALL DIMENSIONS APPLY AFTER ANODIZE.
3. BREAK ALL SHARP CORNERS AND EDGES.
4. ENGRAVE WHERE INDICATED:
"XXXXAXXX XXX"

General Notes

General notes supersede the information in the Title Block (that's why the UNLESS OTHERWISE SPECIFIED is noted in the block). General notes also carry the "UNLESS OTHERWISE SPECIFIED" note and since they apply to the whole drawing, they can only be superseded by the Part Detail note.

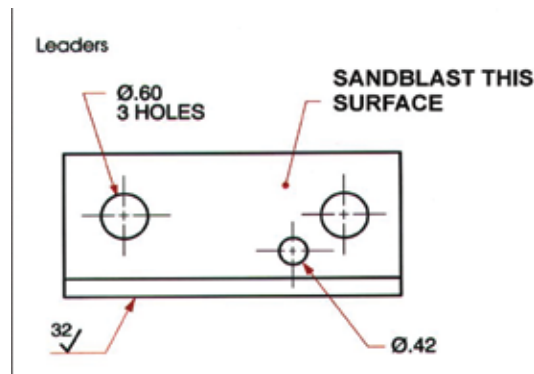
Drawing Notes

Drawings include a variety of notes and information for the customer and fabricator to interpret and use. This subsection will cover the location and precedence of these notes.



Title Block Notes

There are three main types of notes used on a drawing, listed in their order of importance, which are the Title Block notes, General notes, and the Part Detail notes. As mentioned in the above section, the Title Block contains an area where Standard Tolerances are noted and apply to the entire drawing unless otherwise specified. This area also contains additional information that applies to the entire drawing. That is, unless the General notes for the drawing have information that contradicts this information.



Part Detail notes have the highest priority in the chain of notes. A part detail note is used to describe a specific function that only applies to the highlighted feature of a part. This can be anything from a change in dimensional tolerance for a feature, a note about a change in surface roughness, or a note about a surface treatment. In summary, the precedence of drawing notes allows for information stated on a drawing to be superseded by virtue of where the information is noted.

Drawing Notes Exercise:

QUIZ:

1. Name the three types of notes used in their order of priority from lowest to highest.
2. Which type of note has the highest priority on a drawing?
3. Which type of note has the Standard Tolerances located within it?
4. Explain what “UNLESS OTHERWISE SPECIFIED” means.
5. List the three types of notes in their order of priority from lowest to highest.

Drafting Standards

Drafting standards are the “nuts & bolts” of making a good drawing. These include:

- Layer settings: Using good layering practices will making reading and plotting your drawing much easier.
- Line type settings: Following the “alphabet of lines” helps avoid confusion between elements of the drawing.
- Scaling and Scale Factors: This means drawing at full scale in model space and selecting the proper scale factor to apply to your paper space viewport.
- Text Styles and Dimensioning Element standards: Setting up and using industry approved text styles and dimensioning standards for your drawing.

Layer Settings

After opening a blank drawing session go to the Explore Layers icon to create the layers necessary for a technical drawing. The layer names listed here are just a few that can be added to be used in a drawing. The recommended layers are:

- 1 through 5 for Object Lines
- C or Center for Center Lines
- Cutting Plane
- D or Dimension for Dimension Lines
- H or Hidden for Hidden Lines
- P or Phantom for Phantom Lines
- VP or View Port
- X or Hatching for Cross-Hatch Lines

Layer Name	Color	Linetype
✓ 0	White	Continuous
1	Red	Continuous
2	Yellow	Continuous
3	Green	Continuous
4	Cyan	Continuous
5	Blue	Continuous
Center	Color 10	CENTER
Cutting Plane	Magenta	PHANTOM
Dimension	Color 11	Continuous
Hidden	Color 13	HIDDEN
Phantom	Color 14	PHANTOM
View Port	Color 8	DASHED
X-Hatching	Color 15	Continuous

There are a few additional layers that are often used in a drawing. They are F or Format to place your first sheet format on, F2 to place your second sheet format on (in a multi-sheet assembly drawing), and N or Notes to place any set-up notes or miscellaneous information that you don’t want plotted on your drawing. Next, assign colors to the layers. Color is used to easily distinguish the different layers based on their color and not their line type. Color selection can be a personal choice or can be mandated by the company or customer.

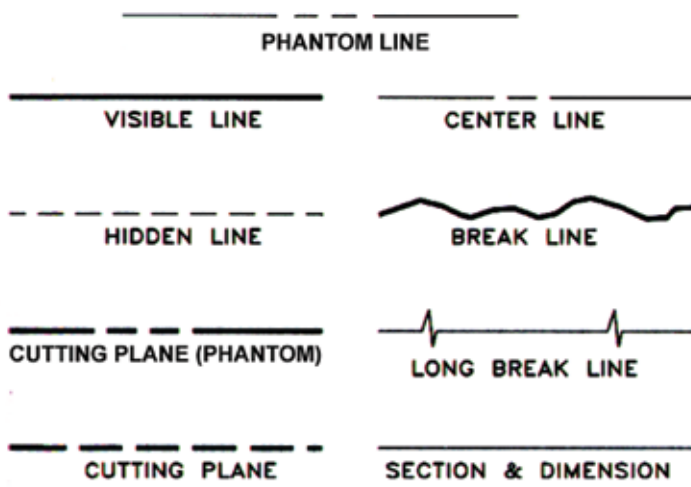
Line Type Settings

There are several line types used in drafting to describe a drawing. Each line type has a different thickness, or line weight, to help identify the different lines used in a drawing. This is referred to as the Alphabet of Lines. ANSI/ASME Y14.2-1992 LINE

CONVENTIONS AND LETTERING STANDARDS sets these requirements.

The Alphabet of Lines are to drafting what the alphabet is the English. In order for the drafter to communicate, the names and usage of each line type must be understood. The most common line type is the one showing the outline of the object. This is known as the object or visible line. Visible lines are drawn at a standard recommended 0.6 mm thick line weight. The line weight for thin lines is 0.3 mm. See below for a description of each line type, their appropriate line weight, and their common usage.

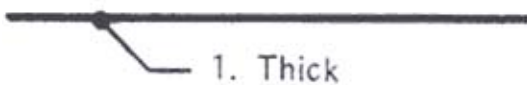
LINE WEIGHTS



Line Types used in a Technical Drawing

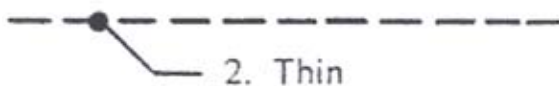
Visible line (Object line - #1)

- Represents visible edges
- Represents outline of the object being drawn



Hidden line (#2)

- Represents edges or outlines not visible in given view
- Represented by a series of short dashed lines



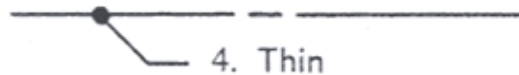
Section lines (Hatching - #3)

- Appear only on a sectional view
- Appears where surface has been cut
- Normally drawn diagonally



Centerline (#4)

- Indicates symmetry, center points, or axes
- Consists of alternating long and short dashes

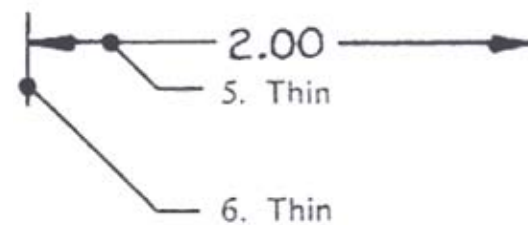


Dimension line (#5)

- Denotes the extent of the dimension
- Use with arrowheads

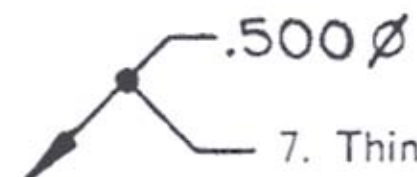
Extension line (#6)

- Extends surface or point away from view
- Used for the purpose of dimensioning



Leader (#7)

- Drawn diagonally
- Directs a dimension or note to an area
- Normally terminates with an arrow
- May terminate with a dot

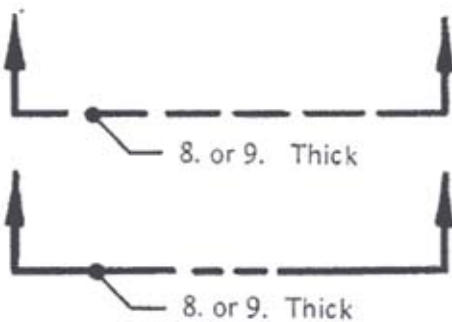


Cutting-plane line (#8 or #9)

- Shows where imaginary cutting takes place to create a sectional view
- May be a series of two short dashes between single long dashes or alternately a series of long dashes
- Arrowheads indicate the direction of sight

Viewing-plane line (#8 or #9)

- Used in conjunction with removed views to show where view would normally appear
- Arrowheads indicate direction of sight



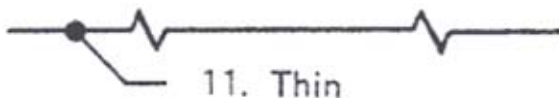
Short break line (#10)

- Terminates a view to conserve space and avoid congestion
- Separates internal and external features with broken-out sections



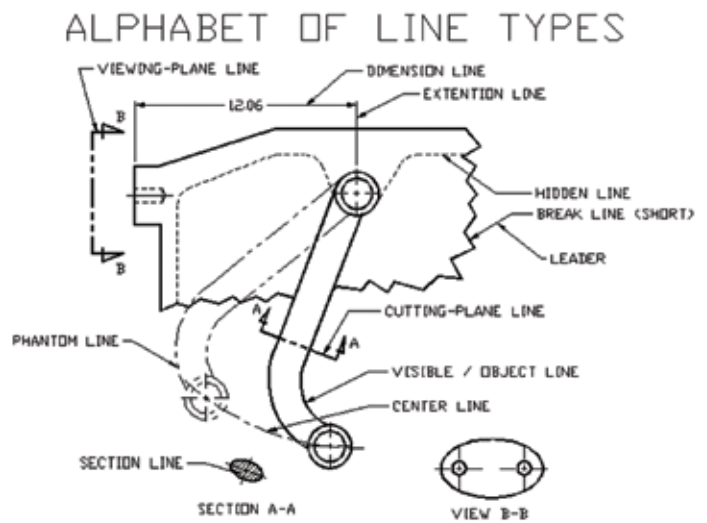
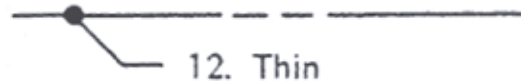
Long break line (#11)

- Allows removal of a long central portion of an object to shorten a view
- Normally used in pairs



Phantom line (#12)

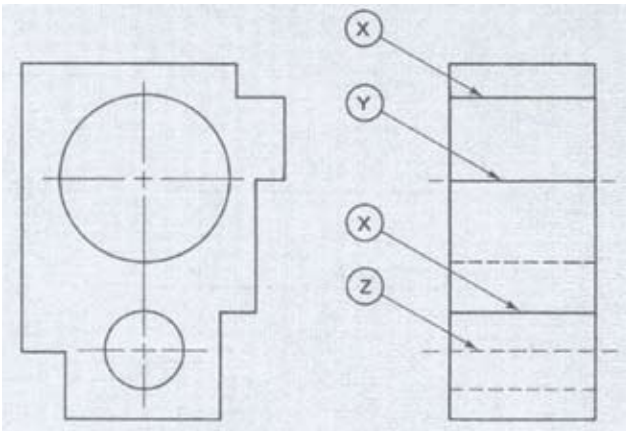
- Replaces repetitive detail - e.g.-coils, gear teeth, threads, etc.
- Drawn with two short dashes between single long dashes
- Lines are thinner than the cutting/viewing-plane line
- Represents the outline of an adjacent part
- Shows alternate position of a given part



Alphabet of Lines

It is not uncommon for line types to coincide or overlap in a drawing. When visible lines, hidden lines, or centerlines coincide in the same view, a preferred line is illustrated. That chosen line is determined by what is called the Precedence of Lines. This is the hierarchy of line importance when lines overlap. This list ranks the line type precedence from highest to lowest:

- Object or visible line
- Hidden line
- Cutting plane line
- Center line
- Break line
- Dimension and extension lines
- Sectioning lines



In the example you can see where lines coincide with one another from the front view to the side view. The “X” lines are object lines which overlap the hidden line of the circles (holes). The “Y” line is also an object line which overlaps the center line of the large circle. The “Z” line is a hidden line. Although the center line for the small circle (hole) is in front of the hidden line, the hidden line has precedence over the center line and is therefore shown instead of the center line.

Scaling and Scale Factors

To simplify applying scale factors, remember that CAD drawing environments include both model space and paper space. Use model space to draw your object at full scale (1:1). Use paper space to scale the drawing to fit within the drawing field (called a viewport in CAD) for the chosen format size.

After you have drawn your object full size in model space, you will have to decide which drawing format size to use. The size you use is dependent on the number of views and the level of detail you want to show. Once the format size is selected, you will enlarge or reduce the object size while in paper space to fit within the drawing field. This setting becomes the scale of the drawing (called viewport scale in CAD) and is used for scaling text, line types, and dimensions.

Viewport Scale	Text Height Scale
1:1	1:1
2:1	1:2 (.50)
4:1	1:4 (.25)
10:1	1:10 (.10)
1:2	2:1 (2.0)

1:4	4:1 (4.0)
1:10	10:1 (10.0)

Text height scaling is the inverse of the viewport scale selected for a drawing. For example: if your viewport scale is 1:1 then your text height scale will be 1:1. However, if your viewport scale is 2:1 then your text height scale will be 1:2 or half as large. See the accompanying table for some common scale factors.

Text Styles and Dimensioning Element Standards

These settings include 1) text height, 2) text style, 3) dimension settings, and 4) dimensioning symbols; and are necessary for effectively describing the basic elements used when detailing a drawing. The industry standard for mechanical / technical drawings is to use the following values when setting up your drawing. These values may vary from the ANSI/ASME Y14.5-M-1994 standard or from one company to the next.

- 1) Text height. These standards are at full scale (1:1). For that reason, if the viewport scale is not 1:1, adjust the text height according to the table above.

General Note heading text height:

- ¼” or .25; modified to 5/32” or .156 (rounded to .16) for A-Size drawings

Example: **NOTES (UNLESS OTHERWISE SPECIFIED):**

Body of General Note’s text height:

- 3/16” or .188 (rounded to .19); modified to 1/8” or (.125) for A-Size drawings

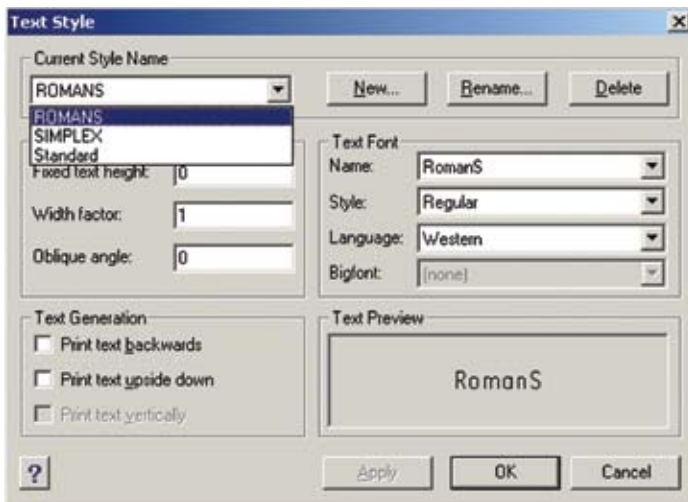
Example: 1. BREAK ALL SHARP CORNERS AND EDGES

General text height:

- 1/8” or .125; no modification for A-Size drawings
- Example: All dimensions, part notes, etc.)

- 2) Text Style. The CAD software default text style is called Standard, also known as Monotxt, and is acceptable although

very “boxy.” Some common alternate styles are RomanS and Simplex. To change the text style, type “style” at the command prompt for the Text Style dialog box to appear. Click on New and type RomanS in the accompanying dialog box. Next, select the black arrow in the Text Font field next to Name and scroll down to find the RomanS font. Select this font and your Text Style dialog box should look like the example shown. Repeat this procedure to add the Simplex style.



- Extension Line placement:
 - o Offset from Baseline: .380
 - o Offset from Origin: .0625 (also known as Extension Line Offset)
 - o Extend lines past: .125 (also known as Extension Line extension)
- Text Format:
 - o Text style: Standard, RomanS, or Simplex
 - o Text height: 1/8” or .125
- Units:
 - o Unit type: Linear
 - o Decimal places: 3
 - o Suppress leading zeros: On
 - o Suppress trailing zeros: Off
- Tolerances:
 - o Decimal places: 3
 - o Suppress leading zeros: On
 - o Suppress trailing zeros: Off
- Scale Factor:
 - o This value is dependent on the Viewport Scale.

Look on the next page for screen-shots of the above elements:

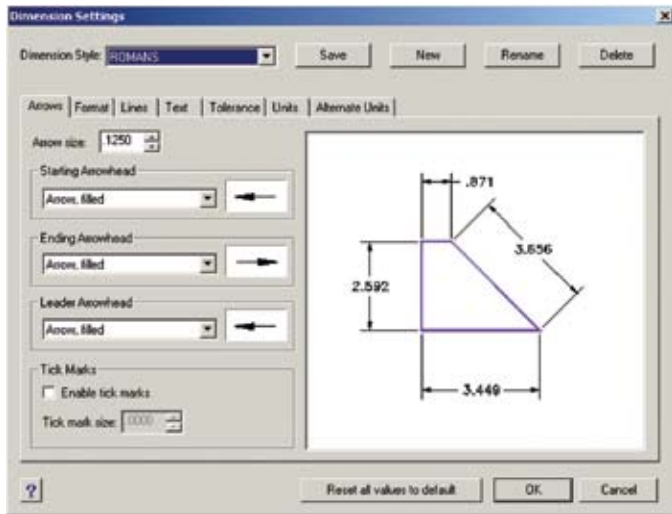
3) Dimension settings. The components that make up the basic dimensioning elements include:

- Dimension Format
- Arrowhead size
- Extension Line placement
- Text format and height
- Units
- Tolerances / Zero suppression
- Scale Factor

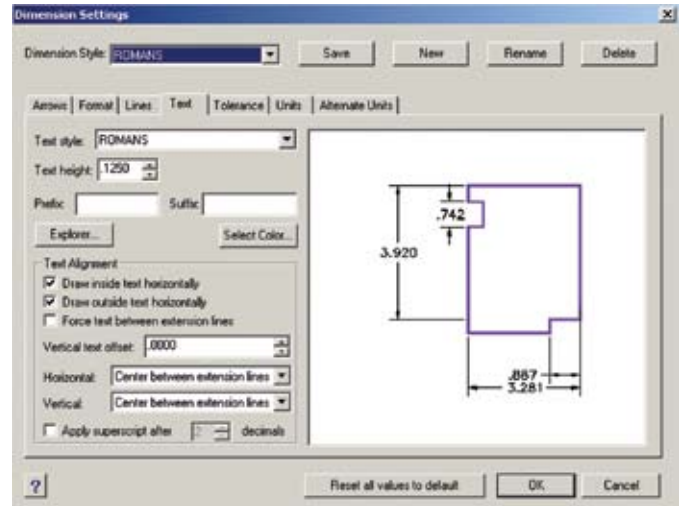
These components are set up by the drafter using the CAD software dimensioning style settings dialog box. Here are some of the more common values for each of these elements:

- Dimension Format:
 - o Fit Method: Best Fit (whenever possible)
 - o Text Placement: Beside the dimension line
 - o Distance around dimension text: .090
- Arrowhead size: 1/8” or .125

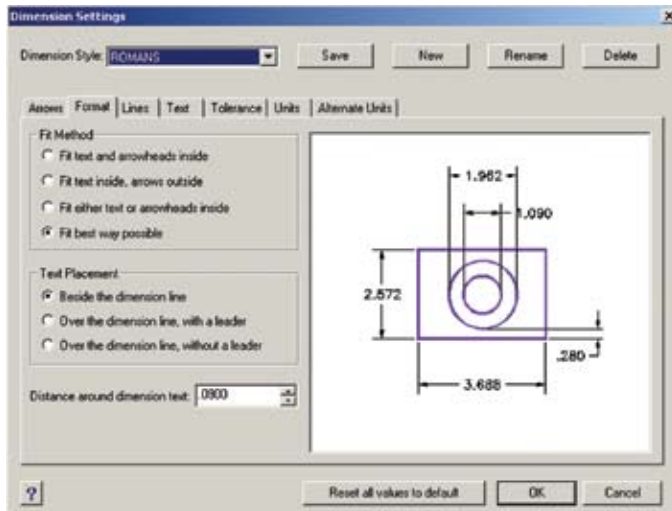
Arrowheads:



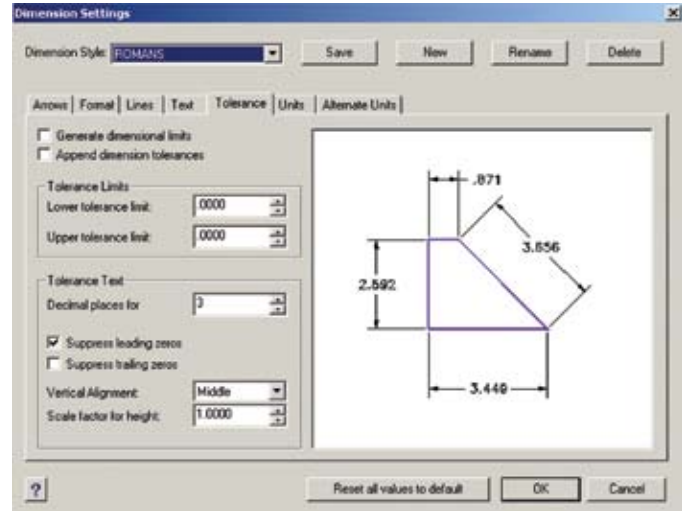
Text:



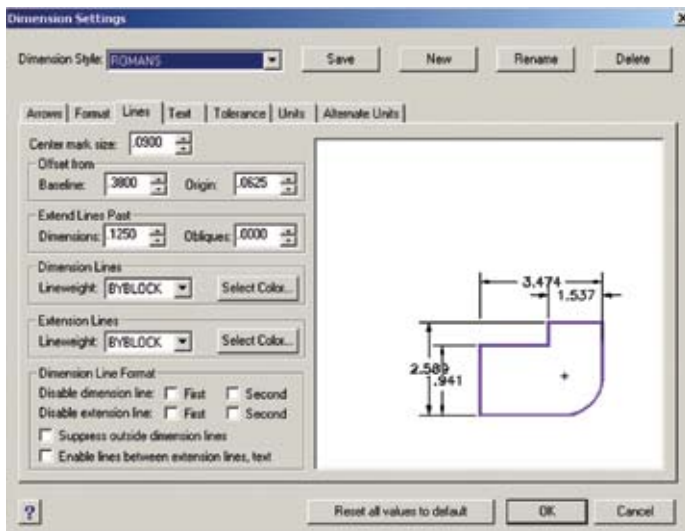
Dimension Format:



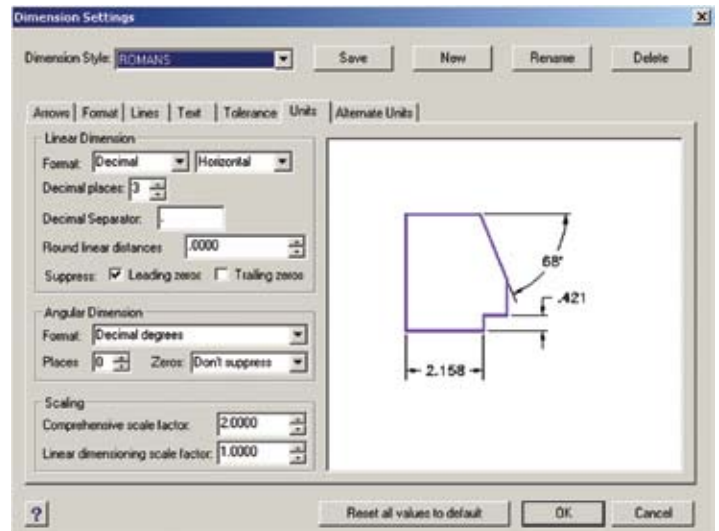
Tolerances:



Lines:

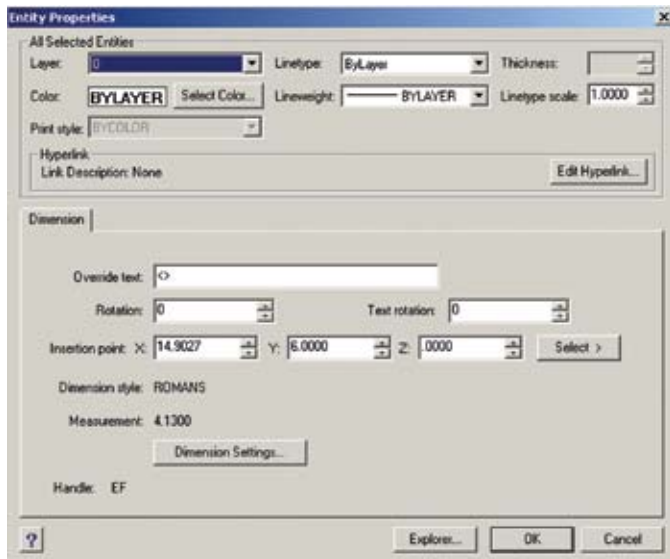


Units:



4) Dimensioning Symbols. Dimensioning symbols are used as a prefix or suffix to a given dimension. These symbols, adopted in the ANSI/ASME Y14.5-1994 standards, are used to replace obsolete abbreviations. Below you will see the symbol and its representation (the obsolete abbreviations are in parentheses).

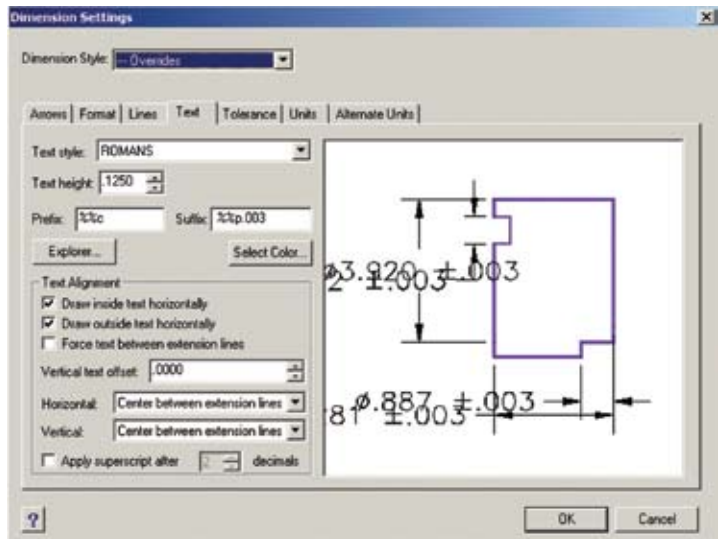
- ° = Degrees
- ± = C
- Ø = Diameter (DIA)
- v = Counterbore (CBORE)
- w = Countersink (CSINK)
- x = Depth (DP)



To apply these symbols to a displayed dimension you will need to highlight the desired dimension, right-click the mouse button to display a new dialog box from which you should select the Properties option. This will open the Properties dialog box. The first three noted symbols (degrees, plus/minus, and diameter) are the easiest to apply to the dimension. They are achieved by typing the percent symbol on your keyboard twice followed by the appropriate letter:

- %%d = Degrees
- %%p = Plus/Minus
- %%c = Diameter

There are two methods that you can use to append these symbols either before or after the dimension. The first method is to simply click in the “Override text” field ahead of the <> marks. As an example, let’s apply the diameter (Ø) symbol as a prefix and the plus/minus (±) symbol with a value of .003 to the selected dimension text. The entry in the “Override Text” row would be: %%c<>%%p.003 (notice there is no space after the > and before the %). This will display the new dimension value as: Ø 4.130±.003. The other method is more time consuming although more technically correct. This involves the same procedure to get to the Entity Properties dialog box; however, instead of just overriding the text, you will select the Dimension Settings button.



This will open a new dialog box titled Dimension Settings. Select the Text tab and enter the appropriate values in the Prefix and Suffix boxes. Press enter or select OK and your dimension text will yield the same result.. The last three symbols (counterbore, countersink, and depth) are created using a new text style called GDT. It’s best to load this style first before making changes to your dimensions. You can add this text style by selecting the Explore Layers icon and selecting Text Styles under the Element column. Select the New Item icon and type GDT. Next, highlight the Font Name, defaulted as “txt.shx” and select “gdt” from the pull-down list or type the name in the box. Once you have this accomplished you can proceed with editing your dimension text.

To form these three GDT symbols use the following values:

- v = Lowercase v
- w = Lowercase w
- x = Lowercase x

NOTE: It is very important to use the lowercase letters otherwise the symbols will not appear.

To edit the dimension text using these symbols, use the same steps outlined above for the technically correct method (select Dimension Setting button) and enter the appropriate letter in the Prefix and/or Suffix box. You can also add additional information or mix values to achieve the desired results. For example, prefix: w%%c Suffix: %%p x.500 would yield the following: w Ø4.130±.003 x.500. Here are some additional examples:

- v Ø.500 x.188
 - o Meaning: Counterbore a .500 diameter hole to a depth of .188
- w Ø.375 X 82°
 - o Meaning: Apply a countersink, with an included angle of 82°, to a major diameter of .375
- 1.2508±.0005
 - o Meaning: Apply a tolerance of plus .0005, and minus .0005 to the dimension

Drafting Standards Exercise:

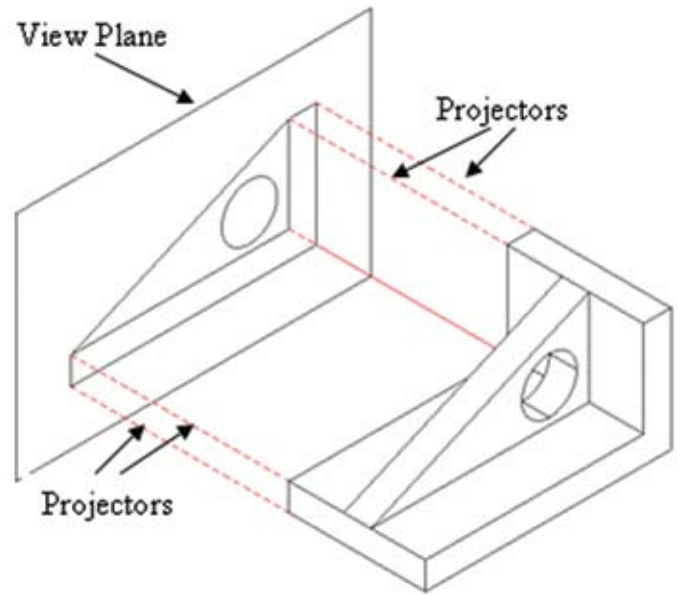
QUIZ:

1. Drafting standards are considered the “_____” of making a good drawing.
2. What is the term used to describe the collection of line types used in drafting?
3. Of the many line types which is the most common?
4. If an Object Line and a Center Line coincide in the same view which line type should be shown?
5. If your Viewport Scale is 4:1 (four times larger than full size), what scale would be applied to the Text Height to achieve the desired text height on the plotted drawing?
6. What is the publication that sets Line Conventions and Lettering Standards?
7. What is the industry standard text height for general text?
8. Dimensioning symbols can be appended as a _____ or _____ to a given dimension.
9. What is the “override text” you would enter to display the Degrees, Plus/Minus, and Diameter symbols?
10. What text style is used to create the counterbore, countersink, and depth symbols?

CHAPTER 6

Principles of Projection

Orthographic projection is the process of describing essential views of three-dimensional objects onto a two-dimensional plane. This section will demonstrate how to draw the necessary views that correctly identify all pertinent details of a part. There are times when one view will adequately describe a part; and there will be other times when five, six, or even seven views will be necessary to describe all the features of a particular part. The simplicity or complexity of the design will determine the number of necessary views and whether or not auxiliary and sectional views are required.



Types of Planar Geometric Projections

Perspective	Parallel		
One-Point			
Two-Point	Oblique		Orthogonal
Three-Point			
	Cavalier	Multiview	Axonometric
	Cabinet		
		Third Views	Isometric
		Auxiliary Views	Dimetric
		Sectional Views	Trimetric

Planar Geometric Projections

Planar geometric projections are a specific category of projections which maintain straight lines when mapping an image onto a viewing surface. The category differs from non-planar projections in the type of viewing surface and projectors used for displaying a three-dimensional object. In a planar geometric projection, straight lines form the projectors map the objects image to a planar viewing surface called the view plane. In contrast, a non-planar projection uses curved projectors to map an image onto a curved surface.

We will first begin discussion of **parallel projection**, one of two classes of planar geographic projection. With parallel projection the placement of the center of projection is at infinity, which means that edges parallel on the real object are also parallel in the projection. This allows for the least amount of distortion of features within the object. There are two different types of parallel projection, *oblique* and *orthogonal* which will be discussed below.

Orthographic Projection

The predominant method of producing technical drawings is through the principles of **orthographic projection**. The term *orthographic* is a contraction of two Greek words - *orthos*, which means “straight, correct, at right angles to,” and *graphikos*, which means “to write or to describe by drawing lines.” In orthographic projection, different views of an object are arranged on a drawing to convey information. Features, surfaces, lines, and edges are projected at straight or right angles from one view to another. The orthographic drawing is used to provide accurate measurements and details to produce the object. It is not used as a realistic view of the object. An *isometric projection* provides that visual three-dimensional representation of the object.

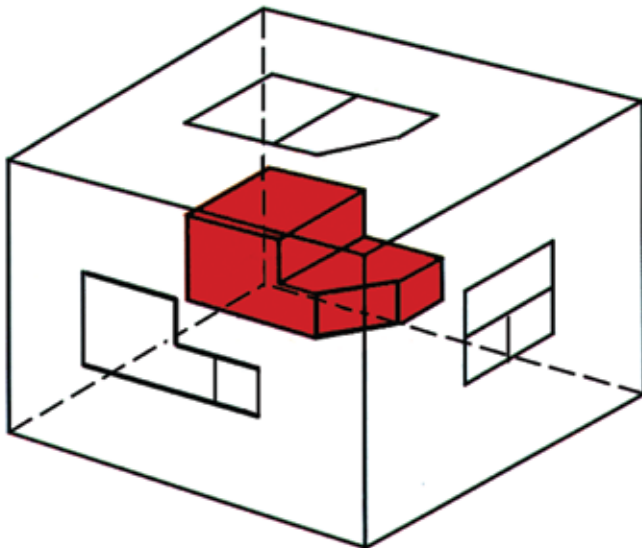
There are six **principle views** that can be used to describe an object. These are described as the Front, Top, Right Side, Left Side, Rear (or Back), and Bottom views. (There are other views that can be incorporated into a drawing, such as auxiliary views

and section views; however those will be discussed later.) These principle views show an objects true shape and size. You should use the minimum number of views that are necessary to properly dimension and describe your object. In other words, views shown on a drawing should add some value to your drawing and should have one or more dimensions associated with them. An object that is cylindrical or symmetrical about an axis needs at least two orthographic views drawn to completely describe it. Objects with more complex shapes need three or more views drawn.

There are two types of orthographic projection used in industry: **Third-angle projection** and **First-angle projection**.

The first, **Third-angle projection**, is used primarily in the United States and Canada. This projection type is sometimes described as “unfolding the box.” To understand this concept, visualize the transparent box illustrated below with the object to be drawn placed inside the box so that its axes are parallel to the axes of the box. The projections on the sides of the box are the views one sees by looking straight at the object through each side.

Object to be drawn in a transparent box



If each view is drawn as seen on the side of the box and the box is unfolded and laid flat, as shown below, the result is a six-view orthographic projection of the object. Any view of the object could be regarded as the principle view, and the box could be folded out from any view. The box shown at the right is unfolded from the front view, which is the most common practice.

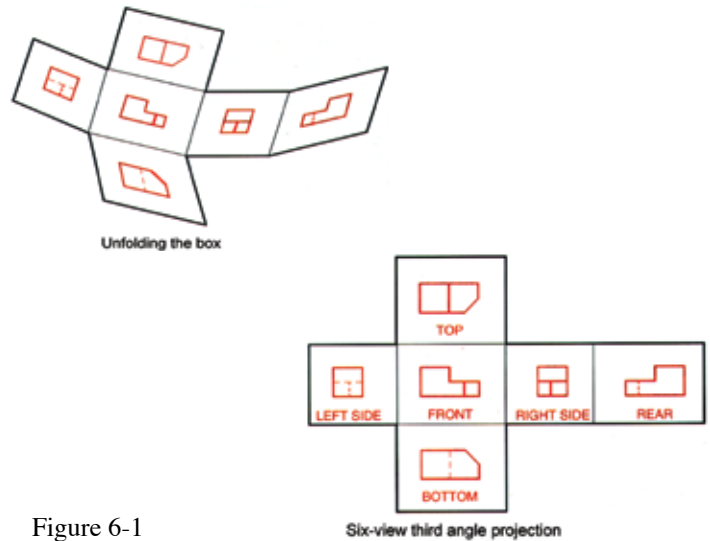


Figure 6-1

The second type, **First-angle projection**, is used by multi-national companies (those doing business in both the United States and abroad) and is the preferred method in most European and Eastern countries. Although the two projection methods may look the same at first glance, there are some subtle differences in how the same six views are projected to create the proper placement of the views. The process for projecting the views in this system is to envision your object in the same six-sided box although instead of “pulling” the Top view to the top surface of the transparent box we “push” it down to the bottom surface of the box. This is repeated with the other views as well. Look at Figure 6-2 and you will see the subtle differences in the views. The Top view in the Third-angle system becomes the Bottom view in the First-angle system as does the Left become the Right. The naming convention for the six principle views remains the same; it is their location, or placement, that changes.

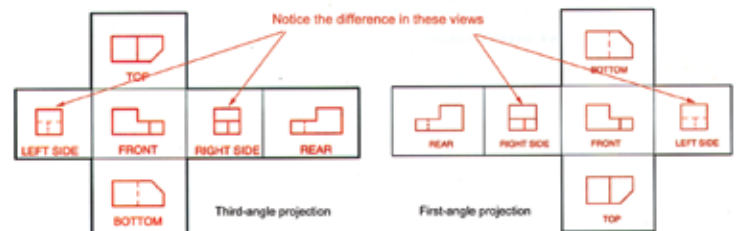


Figure: 6-2

To denote which projection system is used for a drawing, a projection symbol is typically located in the Title Block region of the drawing format. The symbols used are reflected in Figure 6-3.

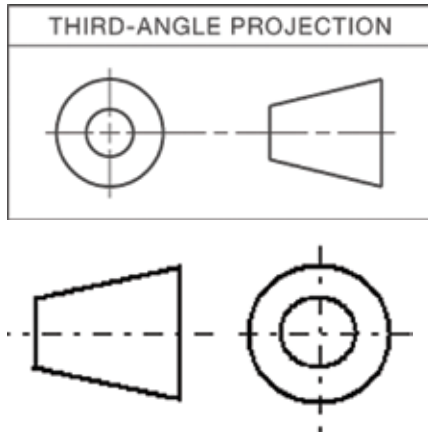


Figure 6-3

Orthographic Projections Exercise:

QUIZ:

1. List the six Principle Views.
2. Name the two projection systems.
3. Which projection system is described as “unfolding the box”?
4. Which projection system is used by multi-national companies?
5. Draw (or sketch) and label the principle views used in the projection system that is primarily used in the United States and Canada.

Single and Multiview Drawings

When deciding which views should be included on a drawing, keep this basic question in mind: “*Can the part be easily manufactured without confusion from the drawing?*” Select the least number of views that correctly detail the part.

The first type of drawing we will cover is the Single-View drawing. As the name implies, the drawing will contain only one view yet convey all the necessary dimensions to manufacture the item. This is used when the object is flat and has no other features on its sides that need to be dimensioned. Look at Figure 6-4 for an example of a thin Gasket that can be detailed as a single view drawing.

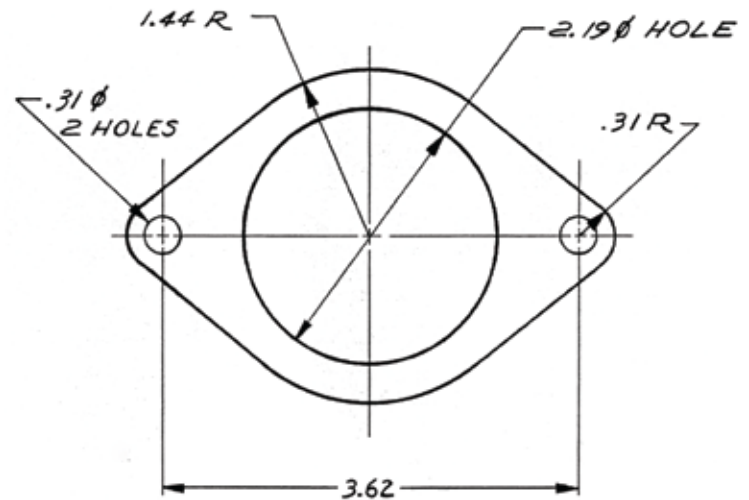


Figure 6-4

The thickness of the Gasket is given in the drawing as a general note: in this case, a .06 thick gasket stock. This general note precludes having to show a side view for that information. Cylindrical objects such as a shaft can also be detailed in a single-view when there are no other features except its length and diameter that need to be expressed.

When a single-view is not enough to detail an object, then the next selection could be a Two-View drawing. Select this type of drawing when two adjacent views will provide the necessary information to detail the part. Adding a third view would be an unnecessary duplication of information. Objects circular about an axis of symmetry are best described with two-view drawings. Figure 6-5 shows an example of two adjacent views that will adequately describe a hub when the dimensions are applied.

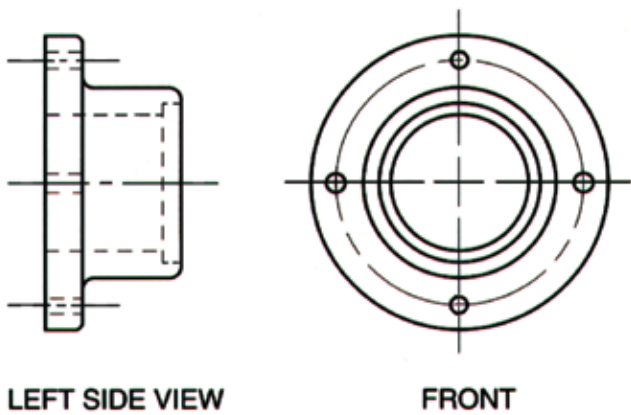


Figure 6-5

Sometimes a part is more complicated than the two shown above and it will require three or more views to properly describe and dimension the design features. This is when a three (or more) view drawing is necessary. With this type of drawing, the selection of the front principle view will determine the layout of the remaining views. When selecting the front view, consider the amount of detail provided on that view. Usually this view offers the most information that can be dimensioned in true shape and size. Here are some other guidelines for selecting the proper views:

- Select the view that will show the most detail as your **FRONT VIEW**.
- When selecting from these similar views, choose the one with the fewest hidden lines.
- Use a **RIGHT** or **LEFT VIEW** to show the profile features.
- Select the **TOP** or **BOTTOM VIEW** to show additional features.
- Use additional **VIEWS** as necessary to avoid dimensioning to hidden features.

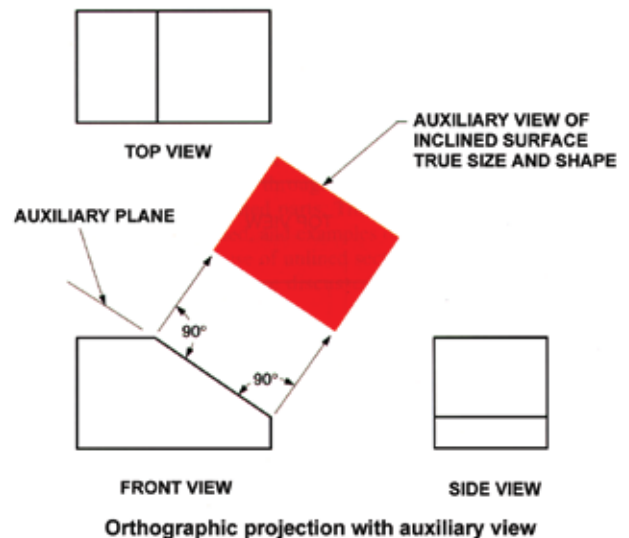
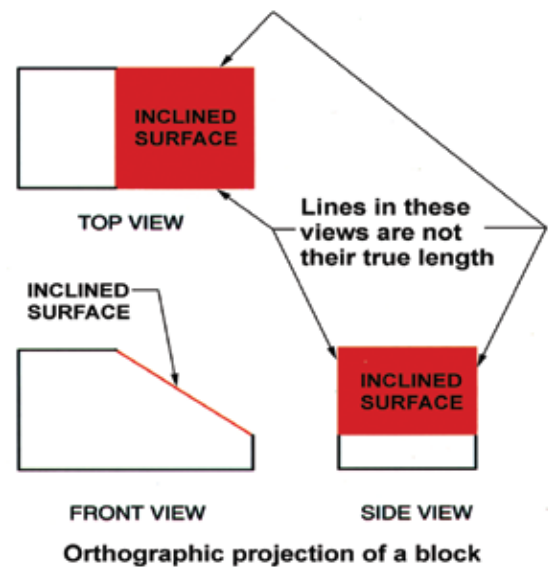
Generally, the three adjacent views are comprised of the Front, Top, & Right. More views are used to capture hidden features and make them visible for the application of dimensions.

Special Drawings

Special Drawings are defined as those that require views other than the six principle views. When an object has features that can not be shown in their true shape and size with one or more of the six principle views, or if there are features which are buried inside an object that can not be dimensioned without dimensioning to hidden features or causing excessive clutter, then special views such as auxiliary, section, and detail views are added to a drawing.

Auxiliary Views

The purpose of an auxiliary view is to clearly show the true shape and true size of an object that has an inclined surface. The auxiliary view is drawn as if it were hinged to the surface to which it is perpendicular and rotated up 90 degrees.



There are three basic types of auxiliary views—the primary, secondary, and partial auxiliary views. Each view has a particular function when it comes to detailing a part on a drawing. The primary auxiliary view is taken directly from any of the six principle views and there can be more than one primary auxiliary view associated with a part as long as it comes from a different principle view of the part. See Figure 6-6.

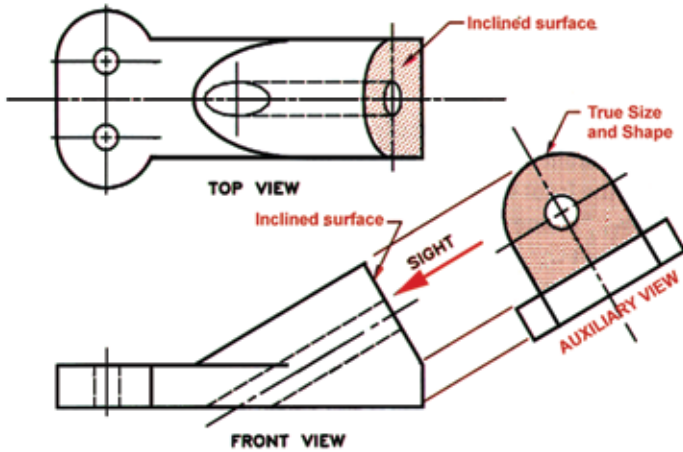


Figure 6-6

The secondary auxiliary view is a supplementary view taken directly from the primary auxiliary view when more detail is necessary to project the true shape and size of a complex inclined surface. See Figure 6-7.

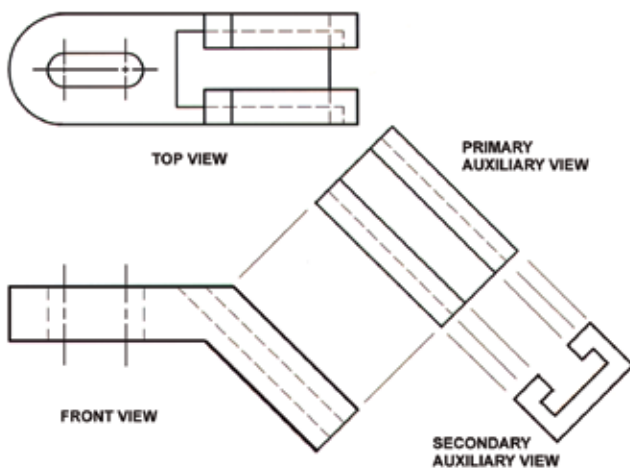


Figure 6-7

The last of the three basic auxiliary views is the partial auxiliary view. This view is characterized as a supplementary view that is used to avoid crowding of views. Break lines are used to reduce the size of a view to control crowding. See Figure 6-8 for an example of a partial auxiliary view. Without the use of break lines, this drawing would need to be drawn much larger in order to be adequately dimensioned.

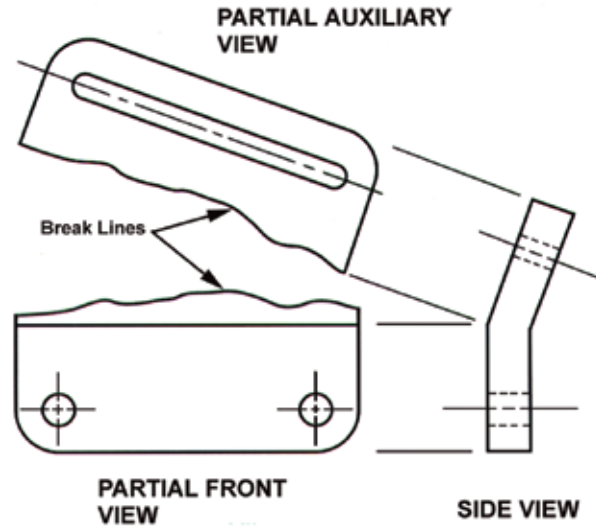


Figure 6-8

Sectional Views

The primary purpose of a **sectional view** in a drawing is to show the internal form and detail of a feature when it has an internal shape that cannot be accurately conveyed in one of the principle views. The sectional view represents a cutaway of an area that is used to apply detailed information about a feature. This cutaway usually eliminates the need for hidden lines, and shows the important outlines of the feature for the application of dimensions.

A sectional view is obtained by passing an imaginary **cutting plane** through an object and presenting the object as though the area cut off by this plane is removed. See Figure 6-9. There are several common types of sectional views: Broken or Partial, Full, Half, Offset, Removed, and Revolved. For the sake of simplicity, we will only cover the use of the broken, full, and removed views.

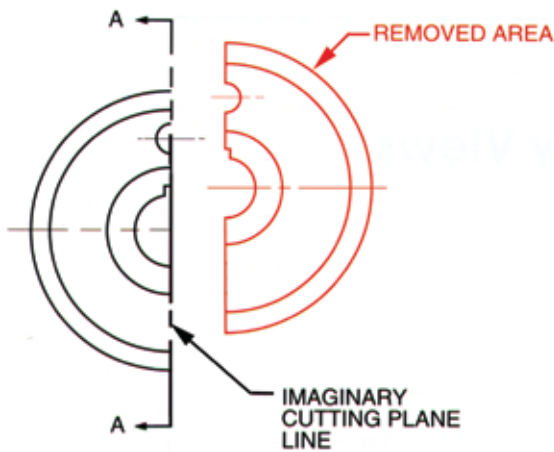


Figure 6-9

A **broken sectional view** is characterized by a partial area being highlighted with a Break Line across, or around, the area to be sectioned and detailed. See Figure 6-10. With the area to be detailed defined, the once hidden lines become object lines and a hatch pattern can be applied. The hatch pattern indicates the part of the object that has been sectioned.

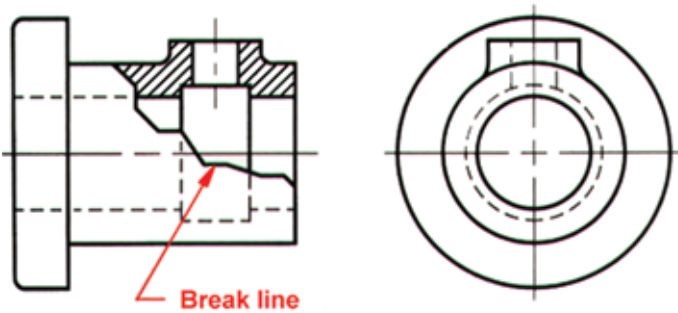


Figure 6-10

This particular view type is probably the most commonly used sectional view. It is very easy to place, doesn't require a separate view location, and adds clarity to a drawing to avoid mistakes when manufacturing the part.

The **full sectional view** is defined by a cutting plane line that passes through an object and exposes the full side of that object. The cutting plane line is drawn and labeled in an adjacent view from the section view. A label is used to define the sectional view, in the event that one or more sectional views exist for one part, and usually starts with the letter "A". The labels are placed near the ends of the section arrows. The full sectional view is then

labeled with the corresponding letter, such as SECTION A-A or SECT A-A. See Figure 6-11

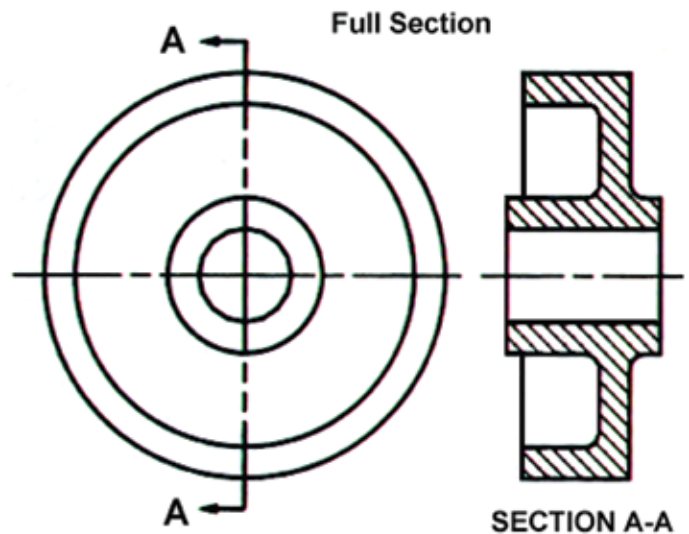


Figure 6-11

The last sectional view type is the **removed section**. This is similar to a full sectional view except that the section is moved to a location that is not adjacent to the view containing the cutting plane. A removed section can be placed anywhere on a drawing and does not have to remain at the same scale as the view with the cutting plane line. This allows for greater detailing of the section when necessary and allows the removed sectional view to be placed in an uncluttered location. The removed section is labeled like the regular full section; and if the scale of the view is different than the scale of the original view, the new scale must be noted. See Figure 6-12, and SECT A-A in Figure 6-13.

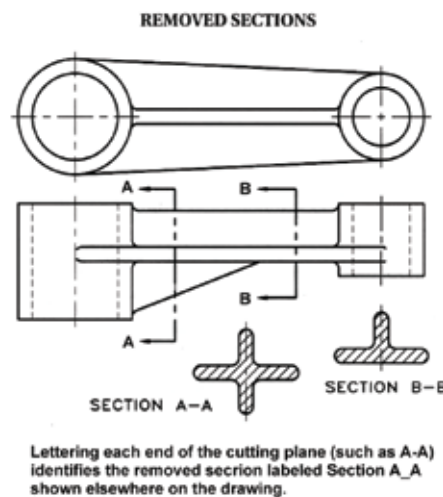


Figure 6-12

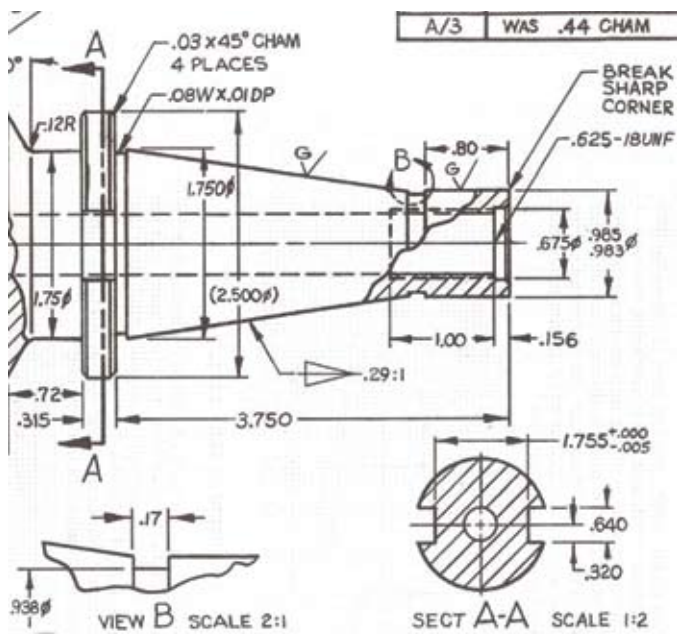


Figure 6-13

Applying sections with a CAD program requires simple knowledge of using the program's **hatch** command. This can also be referred to as *cross-hatching* or a *boundary hatch*. In your CAD program, create sections using a boundary hatch and the icon looks like this:



There are several different styles of hatch patterns to choose from as shown by Figure 6-14 below. While there are an overwhelming number of patterns to choose from, the most commonly used pattern is the ANSI 31, which is shown in Figure 6-14(a). This is the general pattern represented in most sectional views.

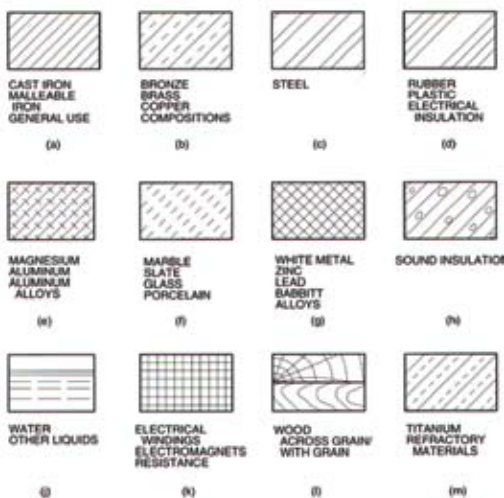
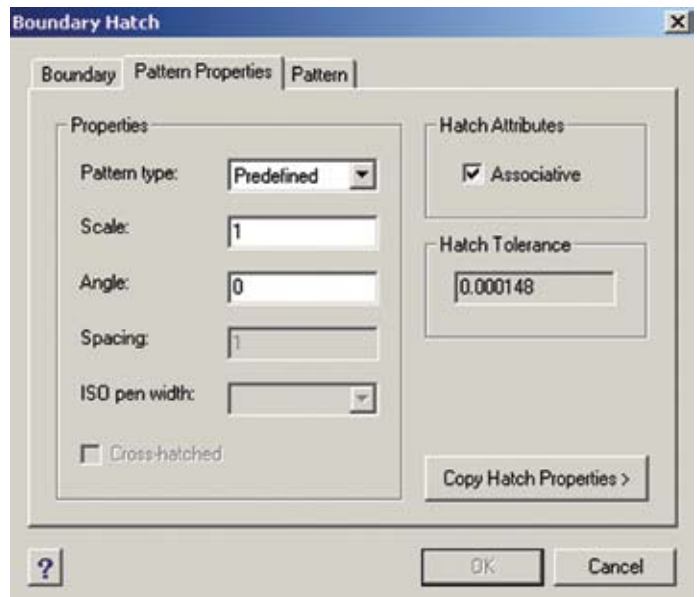


Figure 6-14

When creating a sectional view of a single object it is customary to have all the section lines going in the same direction. However, when creating a sectional view of a multi-part assembly it is customary to switch the orientation of the lines to avoid confusion of where one part ends and the next part begins. To make changes to a hatch pattern, open the Boundary Hatch dialog box and select Pattern Properties. Modify the Scale to affect the spacing of the lines within the boundary of the section. Adjust the Angle to change the orientation of the hatch pattern and thereby changing the appearance within the section. Depending on the size of the area in which you are applying a hatch pattern, it may be necessary to change the Scale value from the default of 1 to another value to achieve the desired effect. The scale is somewhat of a personal choice and depends on the amount of detail that you are trying to emphasize. For instance, a low scale value of .25 will bunch lines together and will be as distracting as having a high scale value of 10 thus having too few lines to properly highlight the view.



Detail Views

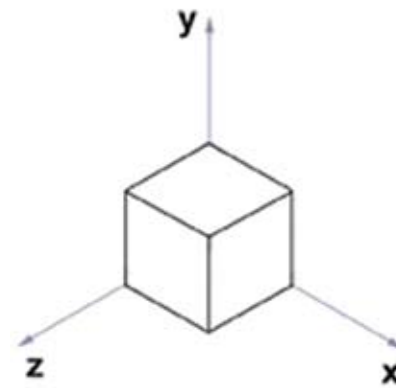
When a view is cluttered with a lot of lines and part details it is sometimes preferable to add a special view that can be relocated and scaled up to provide the necessary dimensional information. Detail views are used for such a purpose. Use detail views when a feature is too small to be properly detailed or dimensioned at the drawing's current scale. Detail views are generally twice the size / scale as the view from which it is taken although they can be scaled to any size that will define the area for proper dimen-

sioning. For example, if the current scale is 1:1, the scale for the detail view may be 2:1 or larger. See Figure 4c-8 and note the label and scale for VIEW B. Like the removed sectional view, detail views are labeled with a letter and their scale. It is customary to label the view with a consecutive letter following a sectional view label. The boundary for a detail view is highlighted using a cutting plane line type in the form of a circle or ellipse near the area that the view is extracted. It is also labeled with one of two methods: with a leader line note and a description, “SEE VIEW B”; or with the letter of the view placed near the boundary and arrows added as depicted in Figure 4c-8.

Axonometric Projections

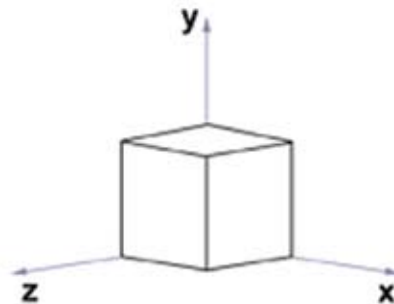
Axonometric projection is another type of orthographic projection that shows parallel projection. Axonometric projection is used to create a view showing all three dimensions of an object in one single picture. Axonometric projection is a pictorial form of orthographic projection. Pictorials illustrate the object viewed from a skew direction in order to reveal all three directions (axes) of space in one image. An axonometric pictorial drawing of an object is created by rotating the object on an axis relative to a projection. Axonometric drawings often appear distorted because they ignore the foreshortening effects of perspective. Foreshortening refers to the way objects appear to get smaller in both height and depth as they withdraw into the distance. There are three types of Axonometric projections: isometric, dimetric, and trimetric and they are classified by the angles between their axis lines. Axonometric projection is used for patent office records, furniture details, and structural designs. It is good for showing the 3-D nature of the object; however its lack of foreshortening creates a distorted appearance.

- **Isometric projection** is the most frequently used type of axonometric projection. All three axis lines are at 120 degree angles which means the object’s dimensions are kept in proportion. These drawings show all three views: top, front, and side. This view is particularly useful for displaying and communicating conceptual design features. The isometric cube is angled so that all of its surfaces have the same angle as the plane of projection. As a result, the length of each of the edges shown in the projection is somewhat shorter than the actual length of the edge on the object itself.



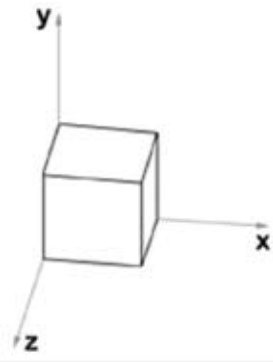
Isometric Figure

- **Dimetric projection** also shows all three views; however, two of the three axis lines are at equal angles with the plane of projection. The third axis line is therefore larger or smaller than the other two. The dimetric pictorials direction of viewing is such that two of the three faces appear equally foreshortened. The angles of the two equal front faces are determined by the direction of the view.



Dimetric Figure

- **Trimetric projection** is a drawing where none of the angles of the axis lines are equal or kept in proportion. Trimetric projections are considered more realistic than isometric projections since most things we see are in a trimetric view. Since there are no restrictions on the orientation of the views, you can adjust the object to emphasize one or two dimensions over the others.

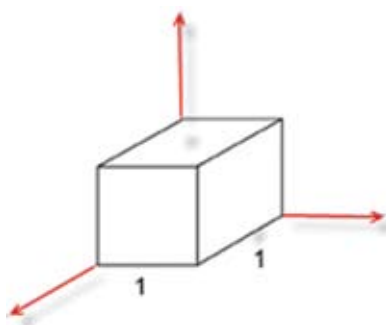


Trimetric Figure

Oblique Drawings

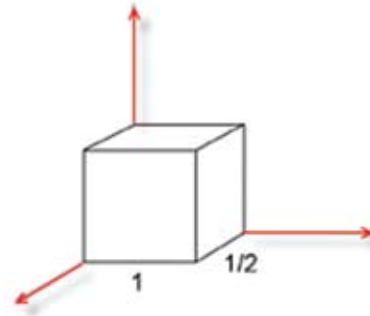
An oblique is the other type of parallel projection. The purpose of an oblique drawing is to show the true size of the front view while showing all three views of an object. In an oblique projection, the receding surfaces are drawn on an angle to give it a pictorial appearance. Oblique views are useful since they combine the advantages of both multiview and axonometric projections. One face (the front face) of the object is drawn without distortion. The remaining two views show some of the object's 3-D appearance. Because of its simplicity, oblique projection is used exclusively for pictorial purposes rather than as formal, working drawings. There are two types of oblique drawings, the cavalier and cabinet.

- Cavalier drawings are those in which all lines are made at their true length including the receding lines to maintain the true measurement of the object. They are drawn with the front face parallel to the picture plane and the receding lines drawn at 30, 45, or 60 degree angles.



Cavalier Figure

- Cabinet drawings have the receding lines shortened by one-half their true length. This helps compensate for distortion and to approximate more closely what the human eye would see. Cabinet drawings are the most used form of oblique drawings because it reduces the distortion of the appearance.

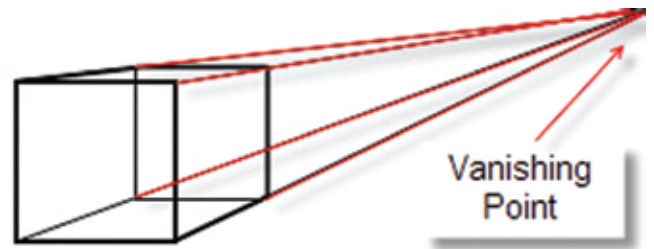


Cabinet Figure

Perspective Projection

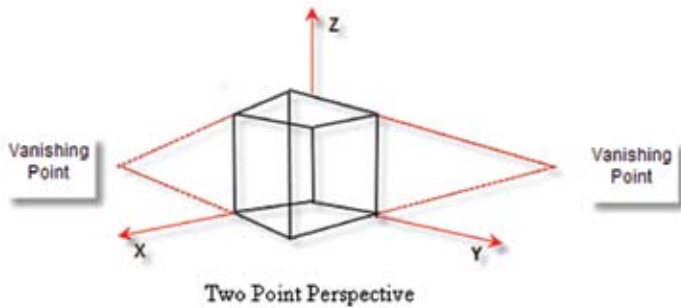
The second type of planar geometric projection is perspective projection. A perspective drawing approximates how a person would view an object with their own eyes. For example, if a drafter was portraying a hallway of a building with a perspective view, the drawing would have lines receding to a vanishing point. In other words, objects appear to get smaller and closer together demonstrating the illusion of depth. Perspective drawings are used for advertising and presentations of architectural, industrial, and engineering design. These views are also good for providing a realistic 3-D form of an object. There are three types of perspective views: one-point, two-point, and three-point perspectives.

- **One-point** views direct all of the lines to one vanishing point. One-point perspective would be like looking down a long straight road and seeing the road vanish in the distance.

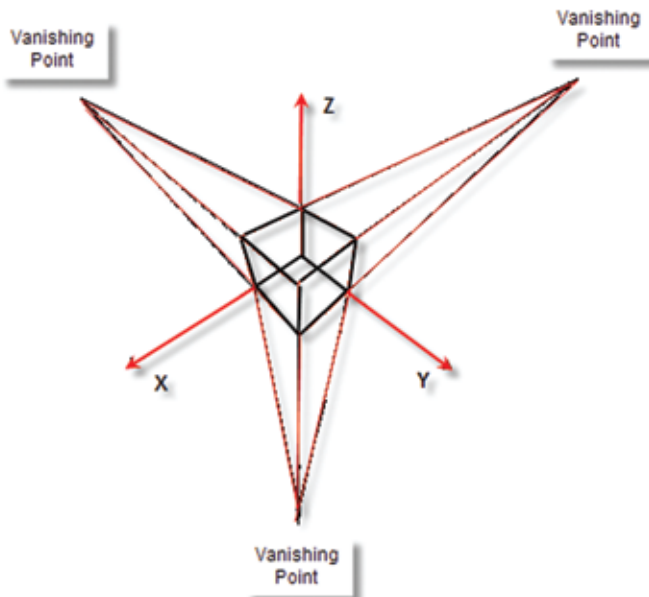


One Point Perspective

- **Two-point** views direct all lines to two different vanishing points in opposite directions. A two-point view can be described as a person viewing a corner of a building while being aligned straight with it. The two vanishing points will be viewed opposite of each other, one to the right and one to the left.



- **Three-point** views direct all lines to three different vanishing points. With a three-point perspective a person would view the same corner of the building as explained in the two-point view, where two of the vanishing points are on the sides with a third vanishing point is projected vertically above the horizon.



Special Drawings Exercise:

QUIZ:

1. The purpose of a special drawing is to allow the features of an object to be shown at their true _____ and _____.
2. An auxiliary view is used when an object has an (2 words) _____.
3. Name the three basic types of auxiliary views.
4. Briefly explain the primary purpose of a sectional view.
5. Name the three sectional view types highlighted in this module.
6. Which sectional view type is probably the most common?
7. What is the ANSI number for the most commonly used hatch pattern?
8. Which sectional view type allows the view to be relocated somewhere else on a drawing?
9. Name the special view that can be relocated on a drawing and does not require section lines.
10. How is a cutting plane line labeled when applied to a sectional view?

CHAPTER 7

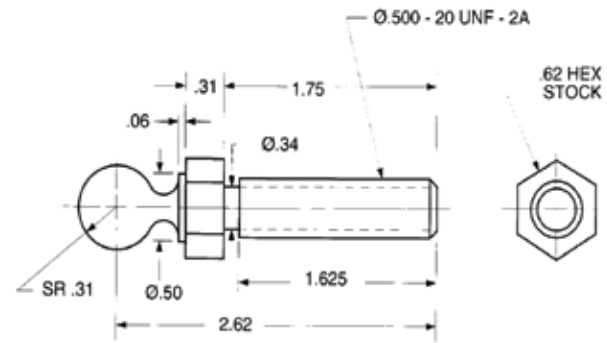
Drawing Types and Descriptions

To recap, you have learned to create an object and about the different ways in which features can be highlighted on a drawing. From the simplest form to the most complex, features can be properly displayed to avoid dimensioning to hidden lines and to show their true shape and size. You will now learn about the different drawing types and their descriptions.

Design and Manufacturing Documents

The first design drawing type is the Detail drawing. A Detail drawing represents all the necessary information to fabricate an item, including the shape description, size description, and specifications of the part. A Detail drawing normally encompasses one part although several Detail drawings can be combined on one drawing format if it is large enough. As an example, four B-Size Detail drawings can be combined on one D-Size drawing format; and although eight A-size drawing may fit on a D-size format, it is common practice to limit the number of details per D-size sheet to four. As a side note, *standard parts (purchased parts) do not require a detail drawing...unless altered or modified.*

When detailing a part with limited features, it is practical to use a smaller format size and create a *Mono-detail* drawing. A Mono-detail drawing is comprised of only one part to which all the necessary dimensions and notes are applied. See Figure 7-1.

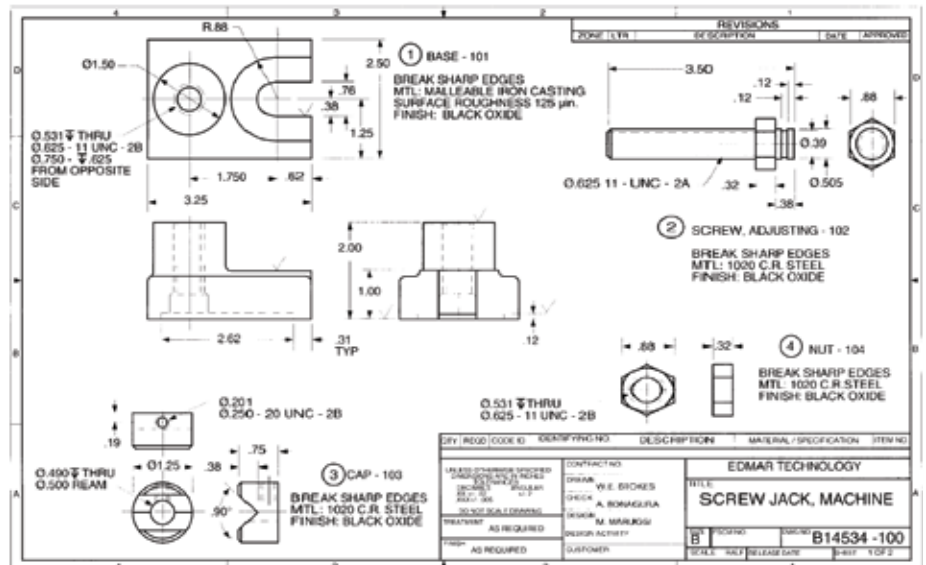


- NOTES:
 1. REMOVE ALL BURRS AND SHARP EDGES
 2. ALL MACHINED SURFACES HAVE A SURFACE ROUGHNESS OF $\sqrt{63}$ μin .
 3. CADMIUM PLATE TO BE PER MIL-C-5840

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES		CONTRACT NO.		EDMAR TECHNOLOGY Pittsford, New York 14534	
DECIMALS XX \pm .01	ANGULAR +/- .01	DRAWN J. BROWNE	92-6-4	TITLE SCREW, ADJUSTING	
XXX \pm .005	DO NOT SCALE DRAWING	CHECK R. BLACK	92-7-2	MATERIAL 1020 CRS	
		DESIGN S. GREENE	92-7-9	SIZE A	ITEM NO. A14534 - 110
TREATMENT CADMIUM PLATE	DESIGN ACTIVITY T. WHITE	92-7-9	SCALE	RELEASE DATE 92-10-3	SHEET 1 OF 1

Figure 7-1

When detailing the components related to an assembly it is common practice to detail several items on one drawing. This is called a Multi-detail drawing. The Multi-detail drawing is similar to the Mono-detail in that each item is individually detailed; however, with the use of a larger format size, multiple items can be detailed on a drawing. See Figure 7-2.



Occasionally when a company makes a detail drawing of a component or an assembly, they will offer the item for sale. When a customer buys that product it might be necessary for the new customer to have some general information about the part. That is when a Specification Control drawing is created. A Specification Control drawing only gives general information such as overall sizes, location and description of mounting holes, general features, and so on. The Specification Control drawing keeps enough of the manufacturing information secret to prevent a customer or competitor from stealing their intellectual property. See Figure 7-3 for an example of a Specification Control drawing for a Variable Transformer.

Next is the *Patent* drawing. Patent drawings have a flair all of their own. Even with today's CAD programs, it is not uncommon to see patents with hand-drawn sketches or even CAD drawings that appear to be from the early days of manual drafting. Patent drawings are used to document the intellectual property of a designer to prevent others from laying claim to their invention. See Figure 7-4 for an example of a Patent drawing for an Electric Light Fixture that was invented and filed around 1962. Note that it took over 3 years before it was officially awarded the patent number 3,198,462.

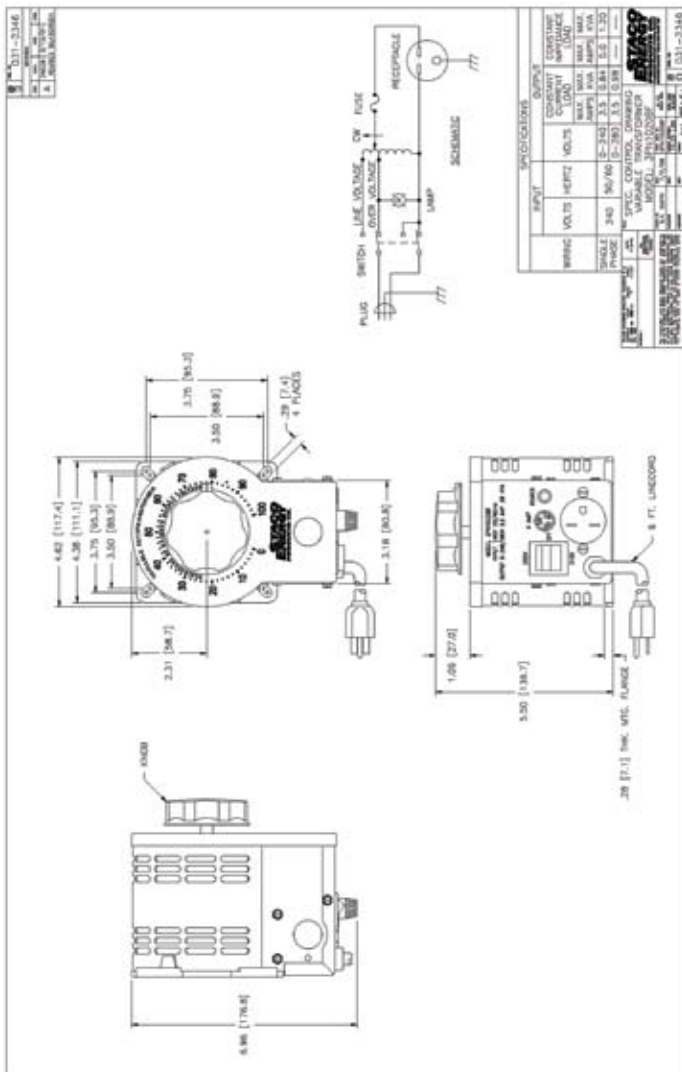


Figure 7-3

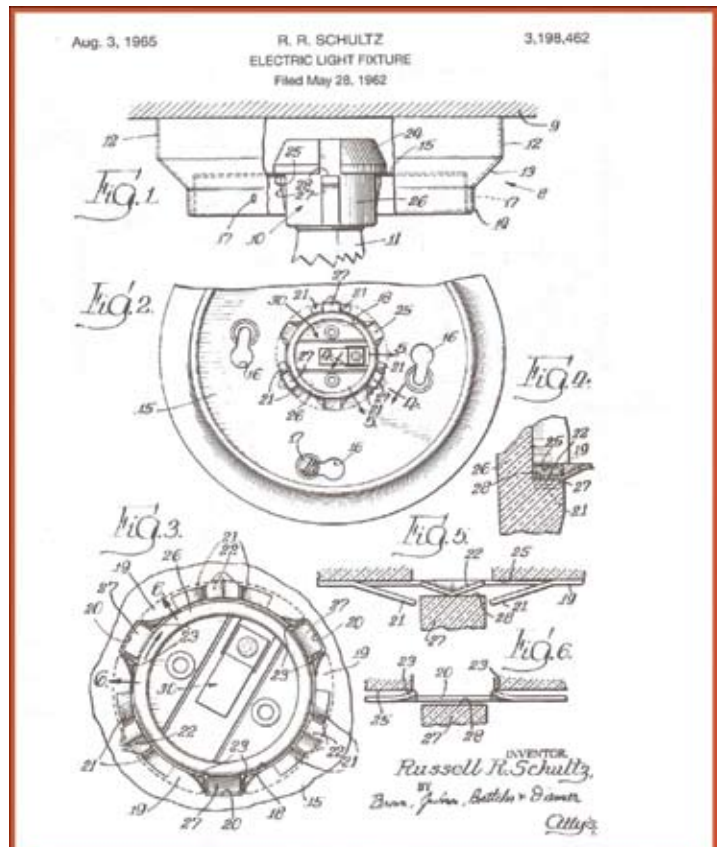


Figure 7-4

Another classification of drawing is the *Make-from* drawing. When a purchased part, such as the Eye Bolt in Figure 7-5, needs to be altered from its original size to a new size a Make-from drawing is needed to reflect the change. This can be a separate drawing (Mono-detail) or incorporated in to a larger drawing such as in Figure 7-2.

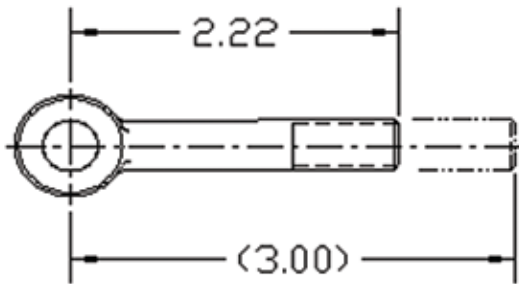


Figure 7-5

The description for the Make-from drawing would include the original purchase item description along with the word ALTER to denote a modified part. The example for Figure 7-5 would be: EYE BOLT – ALTER. Because the purchased item is no longer its original size, you will need to assign it a new drawing number (Mono-detail) or a new part number if its part of a larger drawing. Additional information for the item would include the original supplier's name, part number, and the original size. The “as purchased” features would be shown using phantom lines to represent the part prior to the alteration. The “as purchased” dimensions are shown as reference dimensions while the new features are dimensioned per drafting standards.

Another use for Make-from drawings is to detail a casting or forged part that is machined to its final shape with all included features. A casting (see Figure 7-6) generally is a metal part that is molded to the rough size and configuration of the finished part. The Make-from drawing would then reference the casting part number and detail the features that needed to be added to produce a finished part.



Figure 7-6

The automotive master brake cylinder shown above will need to have holes drilled and tapped on the side of the casting to attach the brake lines along with machining the nose-end for the internal components and mounting holes. All these post-casting features would be detailed on a Make-from drawing.

Design and Manufacturing Documents Exercise:

QUIZ:

1. Name the drawing type that is comprised of only one part.
2. Name the drawing type in which several details are included on one drawing.
3. Name the drawing type that gives general information, like the location for mounting holes, for a part or assembly.
4. Briefly describe the use of a Patent drawing.
5. Briefly describe how a Make-from drawing is used to detail a metal casting.

Assembly (Working) Drawings

This next section addresses what kind of drawing is used to detail a collection of Detail drawings that make up an assembly. An Assembly drawing (sometimes referred to as a Working drawing) is a drawing that represents the assembled relationship of two or more parts, or a combination of detail parts that are joined to form a subassembly or a complete unit. The levels of an Assembly drawing are dependent on the complexity of the assembly being detailed.

Assembly drawings can be comprised of a single sheet in which all the supporting detail drawings are individual drawings on that sheet. Or, they can be a combination of the assembly detail on one sheet and all supporting detail drawings incorporated into a multi-sheet drawing package. When setting up an Assembly drawing package (whether it is a single sheet or multi-sheet) it is common practice to show the assembled components on the first sheet of the package while the individual details are shown on subsequent sheets. See Figure 7-7.

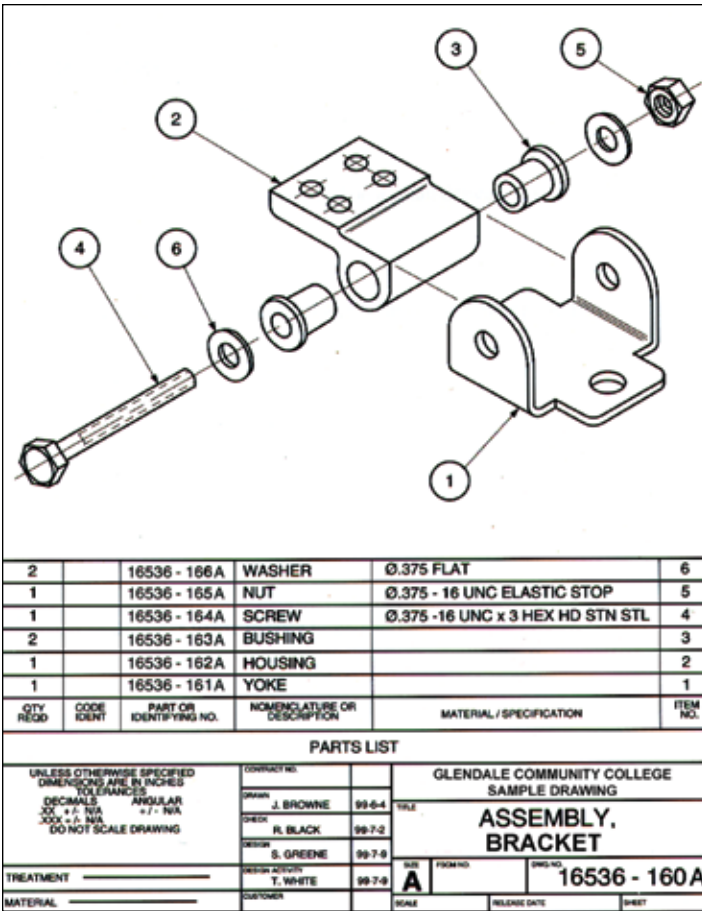


Figure 7-7

The Assembly drawing sheet typically includes the drawing format, title block information, a Bill of Materials (BOM), and the assembly. The assembly can be depicted in several ways and the best method is often dependent on its complexity. A common practice is to show the assembly in an exploded or expanded state. The exploded assembly graphically describes the assembly/disassembly relationship of the separate components. This allows for a better understanding by the manufacturer and assembler on how the parts relate to one another. This detail method is a little more time consuming than standard orthographic projections although it can be worth the extra time if it avoids confusion and ensures a properly assembled item.

After the assembly has been placed on the drawing and the method to depict the various parts has been applied, it is time to identify each component with an Item Balloon that will reference that item to the corresponding item in the BOM. The Item Balloon is attached to the part by means of a leader line with a

circle at the end and corresponding number inside. The circle, or Balloon, is usually 5/8" (0.625) in diameter and the text height of the number is 1/4" (0.25). The leader line can terminate with an arrowhead touching the edge of the item or with a “~” when placed on the surface of the item. Since aesthetics is a sign of a well conceived drawing, it is common practice to arrange the Item Balloons in a row or along an imaginary line. This presents a professional looking drawing.

Should an exploded assembly not be necessary on a drawing because of its simplicity, the use of orthographic projections is required. When this projection method is used, it may become necessary to add Broken sections where needed to highlight hidden or recessed items. See Figure 7-8 for an example of an orthographic assembly with Broken sections.

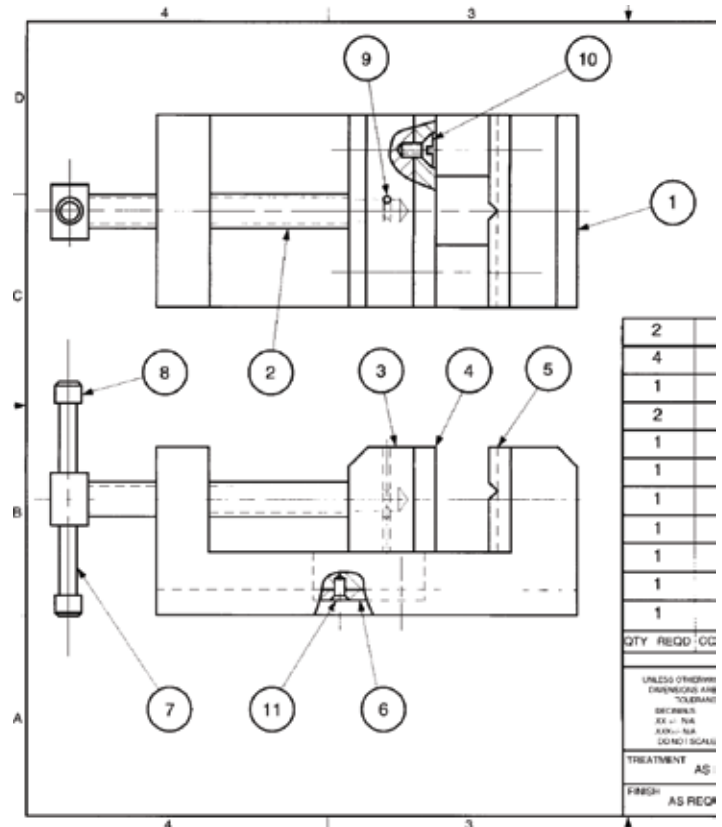


Figure 7-8

Occasionally, a Full Sectional Assembly drawing is used to illustrate the location of separate items within an assembly. This is similar to the exploded assembly view except the parts are shown in section with different hatch patterns applied to depict the different components. See Figure 7-9.

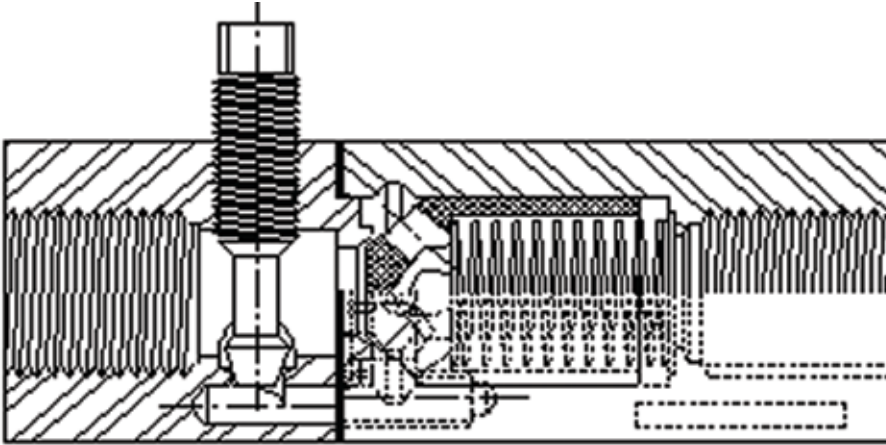


Figure 7-9

Purchased Parts

Purchased parts are those that are manufactured and available from a supplier. When designing a new part, it is best to take a moment to research whether or not a similar item already exists. It is more economical to purchase some items than to have them made from scratch. Where do you begin your research? With the vast amount of information available on the internet and the various search engines at your finger tips, it makes perfect sense to start with this resource. Should you find the exact item or something very similar, purchase that item instead of fabricating a new part. The purchased part already has all the design work and drawings along with any statistical and safety reports that may be required. This will save you a lot of time and money. Should you not find exactly what you are looking for, remember that a purchased part can usually be altered to perform your needed task. Aside from having to document your alteration with a drawing, you will still be saving money on the project.

Purchased parts are shown in the Bill of Materials (BOM) on an Assembly drawing. It is customary to include them on the list after the manufactured items are listed and to group them by supplier whenever possible. This helps the person who will be buying these components make sure that the right part gets ordered from the right supplier.

CHAPTER 8

Dimensioning Systems

When producing a drawing, first draw the features of the object then add the dimensions to describe the features of that object. Dimensions are placed on a drawing along with lines, symbols, and text, to define the geometric characteristics of the object. The purpose of dimensions is to provide information regarding the exact size, shape and location of each feature of the object so that it can be manufactured and inspected. These dimensions are not those values a drafter uses for drawing the object. This section covers the rules, nomenclature, features, and the systems of dimensioning along with an explanation on tolerances. The rules stated here are derived from the American National Standards Institute, ANSI Standards for dimensioning.

Rules of Dimensioning & Nomenclature

Whether the drawing is a very simple configuration, like a washer (two circles of different diameters), or a complex configuration, like that of a jet engine’s fuel valve body which has many holes and features that interconnect and mate to other components, the drawing must be dimensioned so that it can be manufactured. Remember that the values used for drawing the object are not necessarily the same ones used to dimension the object. The process used for dimensioning is based on fundamental rules adopted by ANSI Standards for dimensioning. By following these rules you should be able to fully dimension an object and have it ready for manufacture and inspection.

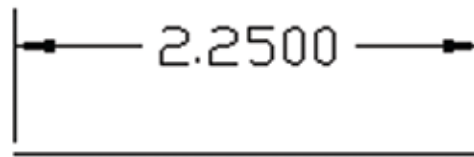
1. Sufficient dimensions must be shown so that sizes, shapes, and contours of an object can be determined without performing calculations or assuming distances.
2. Dimensions must be clearly stated.
3. Dimensions must be selected to minimize the accumulation of tolerances.

4. Each dimension is identified only once.
No duplicate dimensions.
5. Avoid placing dimensions to hidden lines.

The terminology of the components that make up a dimension were discussed with the other Alphabet of Lines. Here again is the definition of each dimensional component as they relate to linear and ordinate dimensioning.

Dimension line:

- Line with arrowheads at each end
- Shows direction and extent of a dimension
- Numerals indicate the number of units of a measurement



Extension Line:

- Aligned with feature and extends beyond object
- Normally shown perpendicular to dimension lines

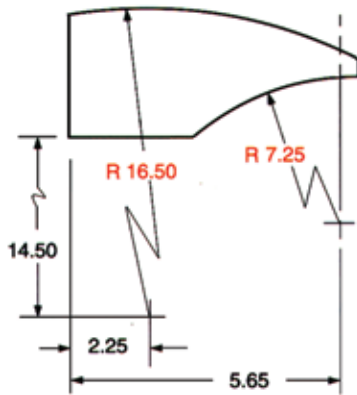
Leader:

- Line used to direct a dimension, note, or symbol to the intended location on a drawing
- Line is usually angled
- Ends with arrow, dot, or wave(~)



Dimensions not to scale:

- Marking are used to show when a dimension is not shown at actual size or location
- Use foreshorten dimension line symbol to indicate size

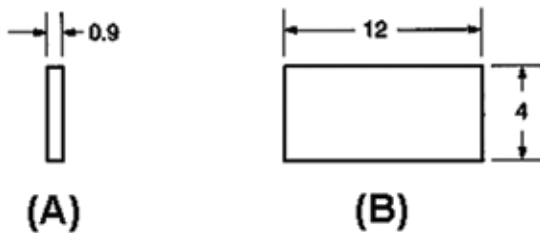


Dimensioning of Features

All objects require two types of dimensions to correctly describe its features: size and location.

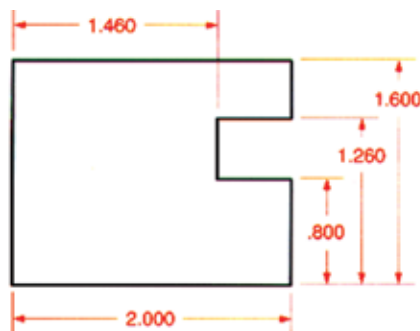
Size:

- Describes the general external features by their length, width, and thickness.
- Describes internal features such as a hole diameter, a slot width, etc.



Location:

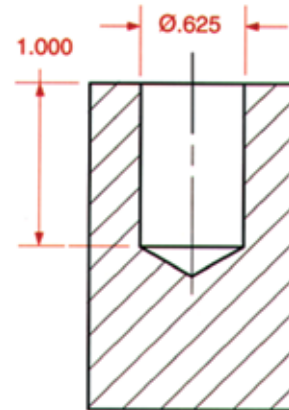
- Describes the location of internal and external features such as a notch, groove, hole, etc.



Some common drafting features include:

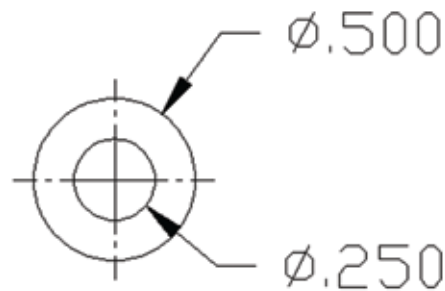
Drill Holes

- One of the most commonly dimensioned features on a drawing. Can be “through” (usually noted as THRU) or “blind”. A blind hole is one which does not go completely through the material and is dimensioned to the depth (x) of the full diameter.



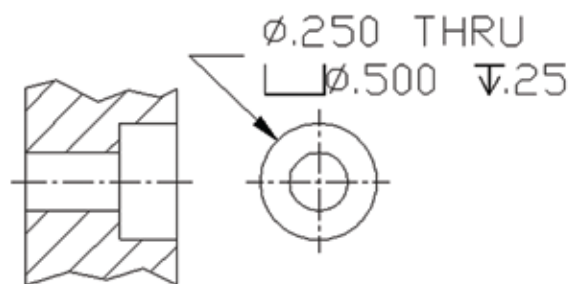
Diameter

- The length of a straight line passing through the center of a circle (hole) and ending at the circumference on each end.
- The symbol (Ø) is preceded by the value.



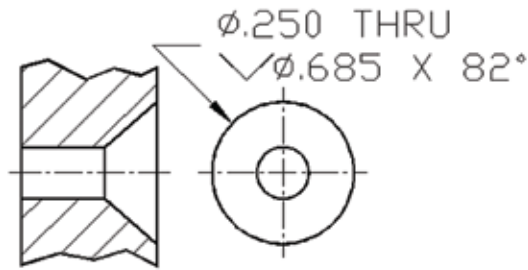
Counterbore (v)

- Larger hole to a specified depth
- Used for clearance of below-the-surface hardware
- A Spotface is similar to a counterbore but without depth

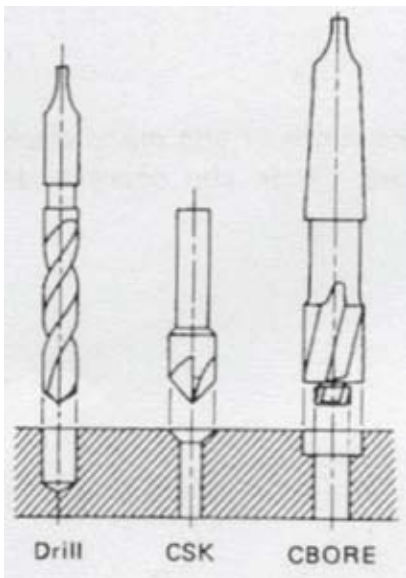


Countersink (w)

- Have angled sides
- Used to obtain a flush or below-the-surface mounting area



Tooling used to make these typical features include Drill bits (twist drill), Counterbore bits, and Countersink bits for standard hardware.



Fillets, rounds, & radii

- Fillets are rounded internal corners
- Rounds are rounded external corners
- Radii (plural of radius) can be either internal or external

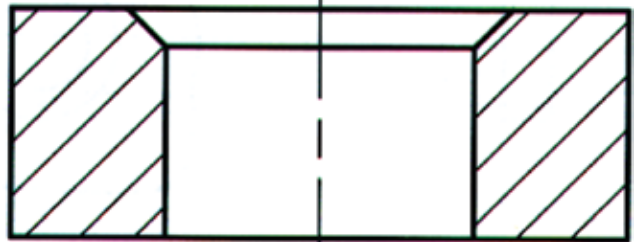
Slots

- An elongated, oblong, shaped feature
- Usually an internal feature with full radius ends



Chamfers

- A machined angle to break a sharp corner
- Can be internal or external
- Angle is normally specified



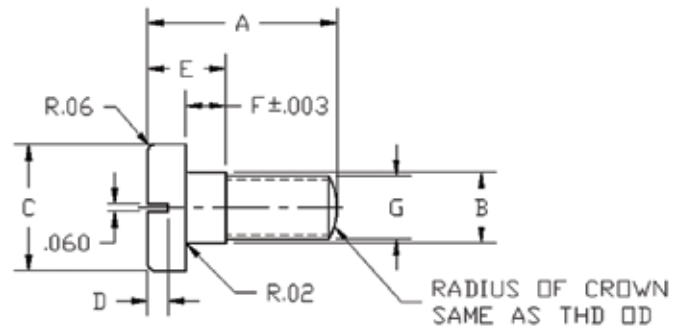
Dimensioning of Features Exercise:

QUIZ:

1. Name the feature that is one of the most commonly dimensioned features on a drawing.
2. What term describes the diameter of a hole in regards to other general internal features?
3. Name the feature that also represents a spotface.
4. Name the feature that has an elongated shape.
5. Draw the symbol which describes the diameter of a hole and write the special character code that generates this symbol.

Dimensioning Systems

There are various dimensioning systems used to properly describe an object for manufacture. The system used may be defined by a company’s drafting standards or by the best practices used in the industry. There is no restriction against starting with a particular system and including dimensions from another system. For example, if an object is dimensioned using the linear unidirectional system, it is appropriate to include dimensions based on the tabular system if it will better provide the information needed. Likewise, if an object is dimensioned with the arrowless system, it is common practice to include unidirectional dimensions to control the size or location of a feature to avoid unnecessary tolerance stack-up and provide fitting the feature with its mating part.



DETAIL	A	B	C	D	E	F	G=THD
1	.62	.38	.62	.06	.25	.135	5/16-18
2	1.00	.44	.75	.12	.38	.197	3/8-16
3	1.50	.56	1.00	.16	.62	.323	1/2-13

Figure 8-2

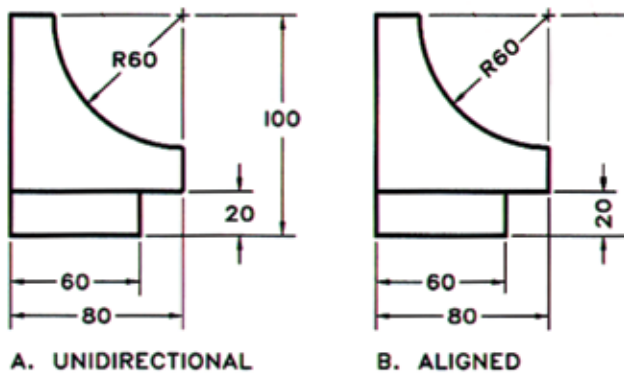


Figure 8-1

Unidirectional: (most common) Figure 8-1A

- Implies that the dimension text is placed in one direction, usually horizontally, and parallel to the bottom edge of the drawing.

Aligned: (rotated) Figure 8-1B

- Dimensions are aligned to the drawings bottom edge for horizontal dimensions and rotated, parallel to the right side, for vertical dimensions (never to the left side). This system is commonly used for architectural drawings and rarely for mechanical drawings.

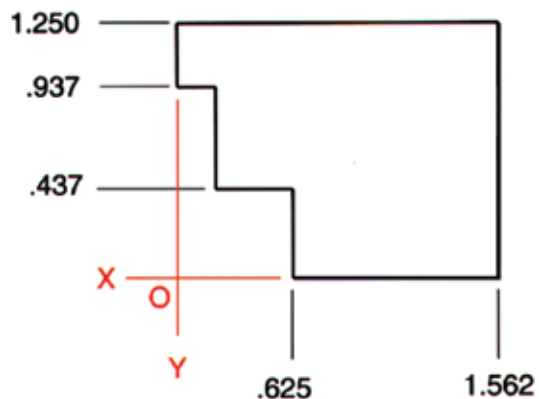
Tabular: Figure 8-2

- Used when a feature can be identified by several dimensions
- Used to identify multiple similar features (such as holes) in one drawing view

Arrowless (Ordinate):

This system is similar to Baseline dimensioning for linear dimensioning except the overlapping extension lines at the Base (Datum) have been eliminated and the dimension values have been placed closer to the object. This system is very handy from a manufacturing point-of-view; however it can lead to a stack-up of tolerances if not used with an occasional linear dimension for specific features.

- Feature and tolerance dependent
- Dimensions extend from a “datum” point



Datum illustration

Dimensioning Systems Exercise:

QUIZ:

1. Name the dimensioning system that is most commonly used.
2. Name the dimensioning system rarely used for mechanical drawings.
3. Explain when the Tabular dimensioning system might be used.
4. What system is similar to baseline dimensioning that does not use arrowheads?
5. Explain the benefit of incorporating two or more dimensioning systems together on a drawing.

Tolerances

All parts must be produced to some measurable acceptable size so when they are assembled they will fit and operate properly. The most proficient machinist operating the most accurate equipment must be allowed some variation in dimensions when producing a part. This allowable variance is called a tolerance. The concept of tolerances was developed to ensure that a part made at one manufacturing facility can be used with one produced at another facility. Technical drawings provide an appropriate variation with each dimension so parts will fit, function as required, and still be economical to manufacture. The saying ‘tolerance = money’ implies that the tighter the tolerance, the higher the price to manufacture the object. Like dimensioning, proper use of tolerances is a vast subject and will be covered only briefly to provide a general understanding.

Here are some general terms and definitions associated with tolerances.

Tolerance - The total amount by which a dimension is permitted to vary. The tolerance is the difference between the upper (maximum) and lower (minimum) limits of a dimension.

Limits of size (Figure 8-3) - These are limits or limit dimensions that describe the extent within which variations of geometric form and size are allowed. They are the specified maximum and minimum sizes of a dimension, as shown below.

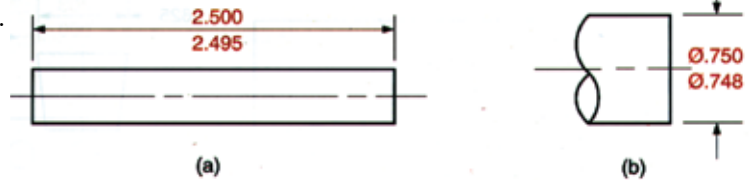


Figure 8-3

Unilateral tolerance (Figure 8-4) - A tolerance in which variation is permitted in one direction from the specified dimension. Examples of this type of tolerance are shown below.

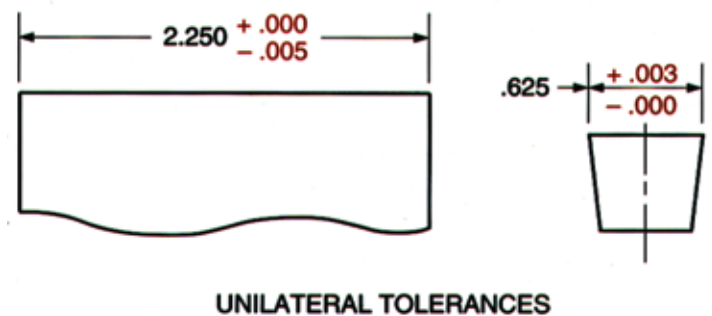


Figure 8-4

Bilateral tolerance (Figure 8-5) - A tolerance in which variance is permitted in two directions from the specified dimension, as illustrated below. If the upper and lower variance is the same value, this is a Symmetrical tolerance.

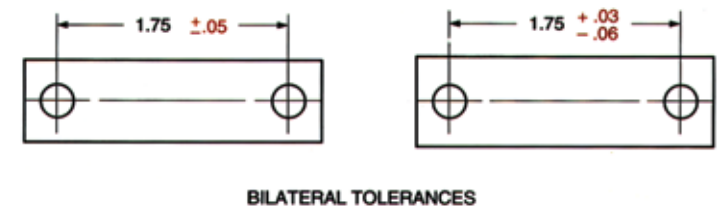


Figure 8-5

Minimum and Maximum tolerance (Figure 8-6) - Sometimes only one dimensional limit is needed. When this is the case the abbreviation MIN or MAX appears immediately following the dimension. This situation often occurs on such part features as the depth of holes, the length of threads, corner radii, chamfers, etc. Single limits are used where the feature that is dimensioned can exceed the dimension in one direction only without concern.

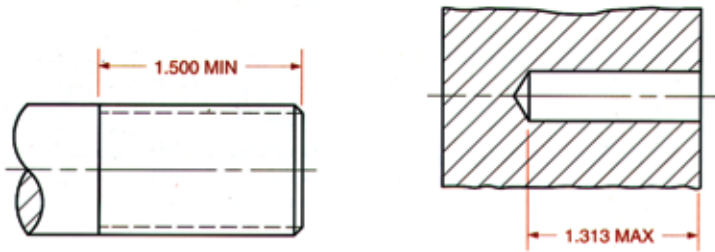


Figure 8-6

Mating Parts - Parts that will be assembled together, such as a shaft in a hole, a screw in a tapped hole, or a key in a slot.

Fit - The looseness or tightness of two mating parts when assembled.

Allowance - The fit between two mating parts when they are at their closest size. IE: When a shaft is at its largest diameter and its mating hole is at its smallest diameter.

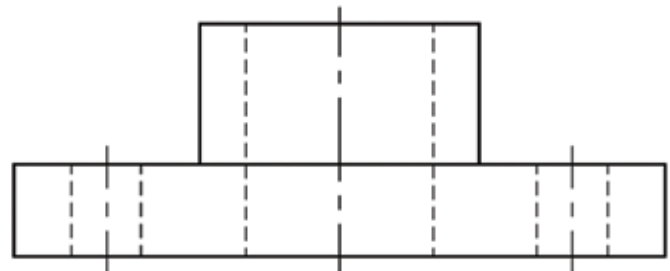
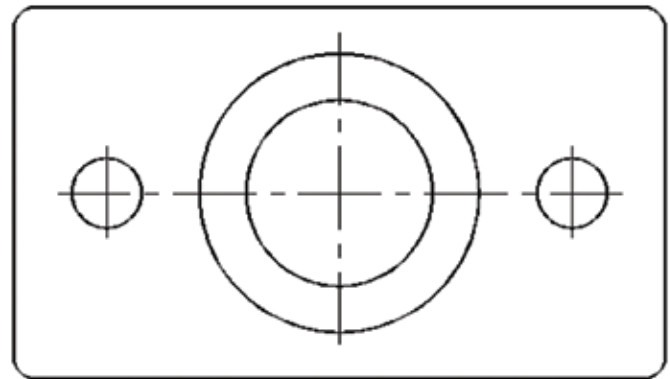
Interference fit - This occurs when one feature is larger than the feature in which it is supposed to mate, causing the two parts to be pressed or forced together. Also known as a Press fit.

Clearance fit - A type of fit that provides clearance, or space, between mating parts at all size extremes. Also known as a Slip fit.
Dimensioning Systems Exercises:

- Using an A-Size format, draw the following object and apply any of the different dimensioning systems to describe the object.

- Outside shape: 3.500 X 2.000
- Small diameter holes: $\text{Ø}0.375 \text{ X}2$; centered on the width, 1.250 to the left of the $\text{Ø}1.000$ hole, and spaced 2.500 on center.
- Large inside diameter: $\text{Ø}1.000 \text{ .}003$; centered on outside shape.
- Large outside diameter: $\text{Ø}1.500$
- Flange thickness: .500
- Overall height: 1.250
- Corner radii: .12 four places

- Dimension with new values for the same part that is scaled up 1.25 percent.
- Dimension with new values for the same part that is scaled down .80 percent.



CHAPTER 9

Basic Mechanical Hardware

In the world of manufacturing and mechanical design, hardware is the main component of almost any item we design. Take the cellular phone for instance. Without the threaded fasteners (screws), washers, and spring, the outside covers would not stay together and the ear piece would not stay open when flipped up. You would be carrying around a bunch of loose electrical components that would be subjected to damage. Because of its importance, you need to be familiar with the common types of hardware used in manufacturing.

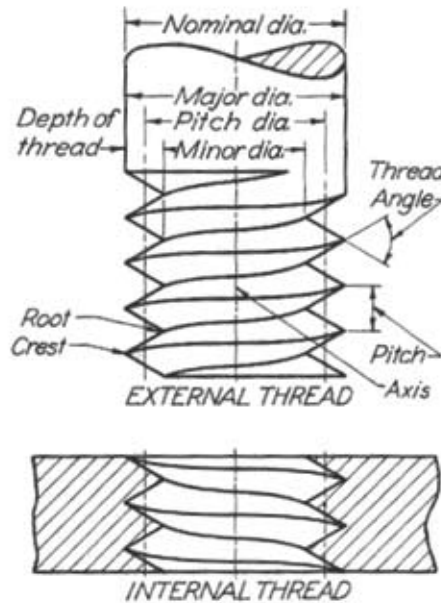
The term hardware encompasses all the locking, holding, and positioning items used to build an assembly. This ranges from screws, bolts, washers, nuts, dowels, and springs which are the major categories of hardware.

Threads and Fasteners

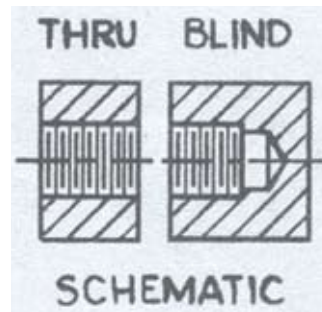
Threaded fasteners are frequently used in the assembly of a majority of items from toys to aircrafts and everything in between. They are perfect for securing parts together while providing a strong connection between those parts. Here you will learn to describe how to represent threads associated with fastening, interpret a thread note, and describe some common fastener types.

First is to identify the ridges that make up the holding part of a fastener. These are called the *thread form* and there are several different series of thread forms. The most commonly used series is called the **Unified Thread Form Series**. It is a compromise between the American National and British Whitworth forms and is mechanically interchangeable with the American National form. When detailing threads on a drawing, there are three different ways to represent them: detailed, schematic, and simplified, though only one is commonly used for drafting purposes.

Detailed – Provides the complete detail of the thread. It is not commonly used for it is time consuming to draw.



Schematic – Not commonly used except for a general depiction of threads on a Specification Control drawing.



Simplified – Most commonly used.

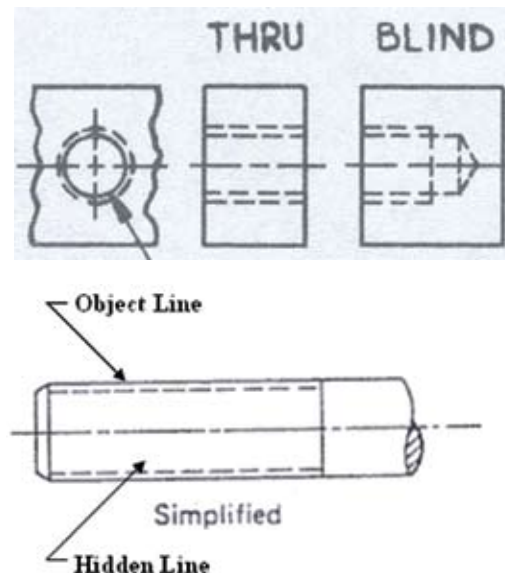


Figure 9-1

In the simplified method, threads are drawn using the major and minor diameter values as identified on a thread chart. For a shaft, such as a bolt, the major diameter is drawn with an Object Line while the minor diameter, also known as the tap drill size, is drawn using Hidden Lines. The opposite is true for a hole. A tapped hole will use a Hidden Line for the major and an Object line for the minor diameters. Refer to the drawings above for the correct depiction when detailing the side view of threads.

In addition to detailing threads on a drawing, a drafter must be able to read and identify a thread note. The *thread note* is the call-out on a drawing that identifies the type of thread to be used on a part. There are English and Metric threads available; however, for this subject we will reference English threads only. By understanding the basic components of the English thread note you can apply that knowledge to similar notes for metric threads.

Thread Note (See Figure 9-1)

Interpret the Thread Note: .250-20 UNC-2B
(also written as: ¼-20 UNC-2B)

Size: (The first value)

- Denotes the major thread diameter.
 - o In this example: Ø0.250 inches
- Expressed in either decimal or fractional form.
- Drawn as a hidden line for holes or as an object line for shafts.

Threads Per Inch (TPI): (The second value)

- The number of threads per inch of length or depth.
 - o In this example: .250-20 indicates there will be 20 threads per inch;
 - o While .250-28 indicates there will be 28 threads per inch.
- Value correlates to the Series depending on size.

Form & Series: (The third value)

- The standard Form here is Unified National (UN).
- There are three common Series:
 - o Coarse (C)
 - o Fine (F)

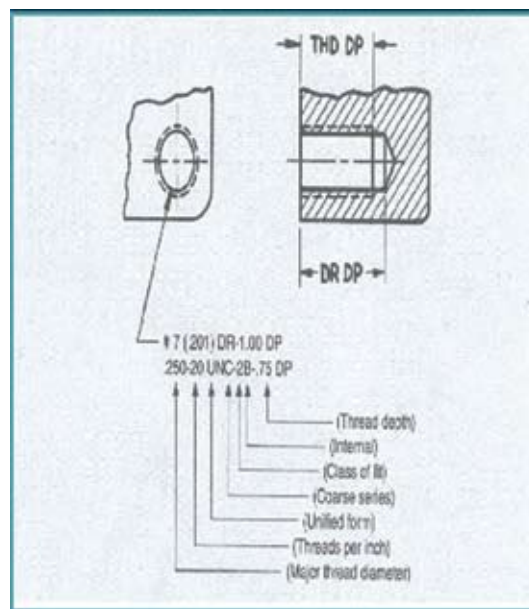
- o Extra Fine (EF)
- o Examples: UNC, UNF, or UNEF

Class of Fit: (The fourth value)

- Class 1 allows the greatest clearance between mating threads.
- Class 2 is the standard for most applications.
- Class 3 allows the closest fit between mating threads.
 - o Used for critical applications, like aviation

External / Internal: (The fifth value)

- A = External (a threaded shaft such as a bolt)
- B = Internal (a tapped hole)



For the Thread Note in Figure 9-1 there is some additional information regarding the tapped hole. (NOTE: This thread call-out does not follow current drafting standards and is for general information only) The first line of the note reads: #7 (.201) DR – 1.00 DP. This means to use a numbered drill, #7, which is .201 in diameter and drill to a depth of 1.00. The second part of the note reads: .250-20 UNC-2B - .75 DP. This means to tap the drilled hole for a ¼-20 thread to a depth of .75.

Thread notes are linked to specific fastener types when used for manufacturing the product. Threaded fasteners come in a variety of different types to accommodate a variety of different applications. Fasteners allow an assembly to be quickly put together and disassembled when necessary; unlike other methods, such as welding.

Three of the most common fastener types are bolts, screws, and nuts. They come in various shapes and types. The head configuration on bolts and screws, for example, vary because their shape and drive-style are dependent on their application. Below are some general characteristics of these common fastener types.

Bolts:

- Major thread diameter generally ¼” diameter or larger.
- Bolt Head Styles:
 - Allen Head (internal hex design)
 - Hex (External shape)
 - Square
- Bolt Head Types:
 - Button Head
 - Flat Head: (Used in a countersink hole or finishing washer)
 - Socket Head



Screws:

- Major thread diameter generally smaller than ¼” diameter.
- Come in various shapes and types.
 - Cap Screws, Wood Screws, Machine Screws, Sheet Metal Screws, Self Drilling Screws, etc.
- Screw Head Styles:
 - Allen Head (internal hex design)
 - Hex (External shape)
 - Slotted
 - Phillips-Head
 - Torx
- Screw Head Types:
 - Button Head
 - Flat Head
 - Oval Head
 - Pan Head
 - Socket Head



Nuts:

- Come in various shapes and types.
 - Hex, Nylon Insert Lock, Jam, Wing, Acorn, Square, etc.



Threads and Fasteners Exercise:

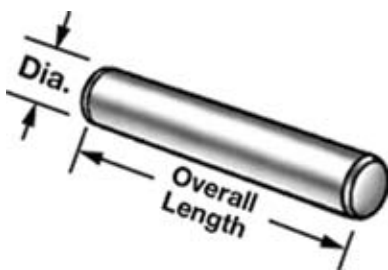
QUIZ:

1. What is the main type of hardware that holds a majority of items together?
2. List the three different types of thread representations.
3. In the thread note: ½-11 UNC-2A, what does the ½ represent?
4. In the thread note: ½-11 UNC-2A, what does the 11 represent?
5. In the thread note: ½-11 UNC-2A, what does the “A” represent?

Other Basic Hardware

There are other types of common hardware used to assemble jigs, fixtures, machines, and other manufactured products. Due to the vast amount of items identified as hardware, we will narrow our discussion to just a few: dowels, springs, and washers.

Dowels: Dowels are made of precisely ground metal rod that is sectioned into short lengths and used to obtain accurate alignment of two mating parts. They are normally used in pairs. One dowel is used to locate while the other prevents rotation. Dowels are usually used in conjunction with threaded fasteners. The dowels provide the alignment while the fastener provides the force to hold the parts together.

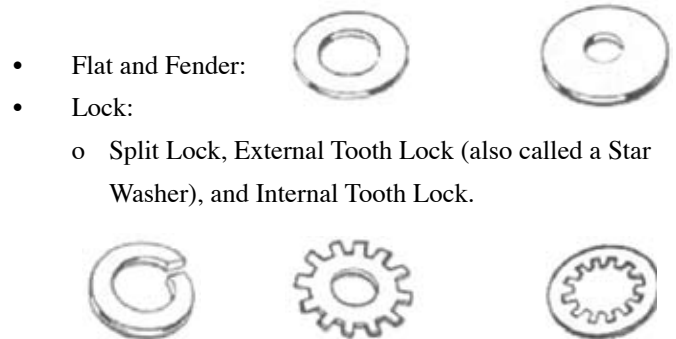


Springs: There are three major types of springs and they each serve a particular function. The first type is a Compression Spring. It is made by a coil of wire formed around a shaft called a mandrel and has a generous amount of space between each coil. The compression spring is used to support an item and will compress under a load. Examples include the spring used in a ball point pen and in the mattress of a bed. The second type is the Extension Spring. It is also made by coiling wire around a mandrel except instead of space between each coil the wire is wound next to each other and loops are formed on each ends. The extension spring is used to pull things together and will extend under a load. Some examples would be the springs used in some garage door assemblies, carburetors, and around the perimeter of trampolines. The third type is the Torsion Spring. This spring is formed by coiling wire around a mandrel and leaving the ends long. Torsion springs can be wound to either pull the ends together or push them apart. A clothespin is a common example of the use for a torsion spring as is the little torsion spring that opens the lid of a CD Player. On a larger scale, before gas struts, the hood of an automobile was once held open by torsion springs. See Figure 9-2.



Figure 9-2

Washers: These come in various shapes, types, and sizes. They are made of various materials such as metal, plastic, rubber, or leather. When placed beneath a nut or fastener, washers can distribute pressure over a larger surface area. They can also help prevent leakage such as with a garden hose, or to prevent a fastener from loosening.



CHAPTER 10

The Welding Process

The task of welding is usually performed by a Welder. This person is usually certified in the various disciplines of welding and requires a significant amount of training. For drafting purposes, here is an overview of the subject of welding.

Welding is a form of fastening materials of the same type together. Welding permanently attaches two similar metals together. This differs from the hardware fastening types discussed above which are semi-permanent, meaning they can be easily removed with the proper tools and can be applied to dissimilar materials. Welding occurs by heating the material to a high enough temperature that it begins to soften or melt. This is done in a localized area to bond the two metals together into one piece. When the process is properly done, the welded joint will be as strong, or stronger, than the original metal.

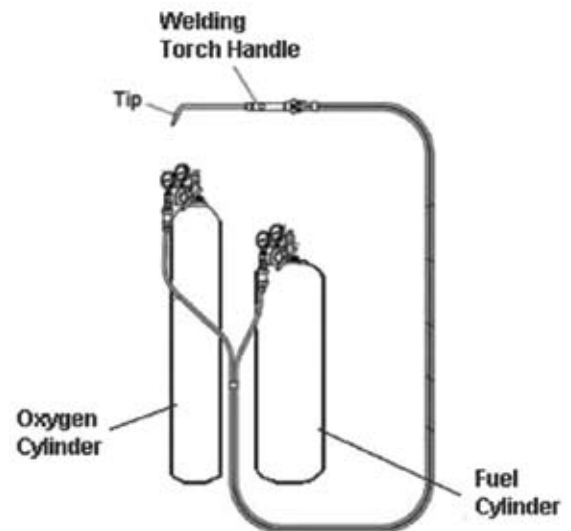


There are various methods and types of welding that are used in each of the major drafting disciplines that will be covered in this module. Welding, also called weldments when two or more items are connected together, is used in the architectural and civil fields in the form of connecting rebar together for the foundation of a building, connecting I-beams that form the skeleton of a building, or connecting piping that is used to transfer liquids or hazardous gases above or below ground. The mechanical field uses welding to form the chassis used for automobiles, the framework for electrical equipment, as well as the airframe for small aircraft to attach the various links that form a very rigid and safe structure.

Welding Processes

Welding is accomplished by using one of several processes. Some processes are more common than others. Oxyacetylene, arc, and spot welding are the three most commonly used methods for the process of welding.

Oxyacetylene is the name used to describe oxygen gas welding. Two cylinders (bottles) of pressurized gas are combined with separate hoses attached to a mixing device that also functions as the handle used by the welder. One of the cylinders contains oxygen and the other can be a burning fuel such as natural gas, propane, or propylene. When these fuels are mixed in the correct proportions and ignited, it causes a flame that is hot enough to melt softer metals. Figure 10-1



Arc welding is a general term describing several methods of welding that use electricity to cause an arc hot enough to melt metal. This method uses a work cable that is temporarily clamped to one end of the metal pieces being welded while the welder holds the other end known as the electrode holder. With electricity going to the electrode holder and a welding rod in place, when brought close to the metal, the electricity jumps the gap causing a bright arc. Special welding hoods are used to protect your eyes from the intense brightness. Figure 10-2



Figure 10-2

Spot welding is similar to arc welding in that electricity is applied to the two metals. This method uses a temporary application of high electrical current between two electrodes. When the metals are placed between the electrodes, and the electricity is turned on, the path of the current passes through the metals melting them together at that one spot. This type of welding is used mainly for thin materials such as sheet metal. Most metal car bodies are assembled by robotic spot welders. Figure 10-3

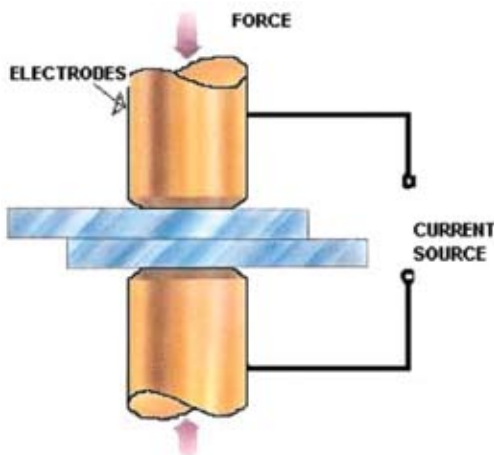


Figure 10-3

Welding Symbols

Welding symbols are established by the American Welding Society (AWS) to allow a uniform standard in the drafting and placement of symbols. The document, known as ANSI/AWS A2.4-93 (Standard Symbols for Welding, Brazing, and Nondestructive Examination) details how the welding symbol is drawn and identifies each of the various line types, symbols, call-outs, and weld type descriptions.

This basic weld symbol is made up of a leader line with an arrow, a reference line, and a tail which may or may not be used depending on the type of call-out. See Figure 10-4. The reference line is the horizontal line that will indicate the weld type symbol and all the dimensional values associated with the correct size and location of the weld. The tail of the weld symbol is used to identify a specification, a process (such as OAW for oxyacetylene welding, or RSW for resistance spot welding), or other type of reference. The tail is omitted when no reference is used.

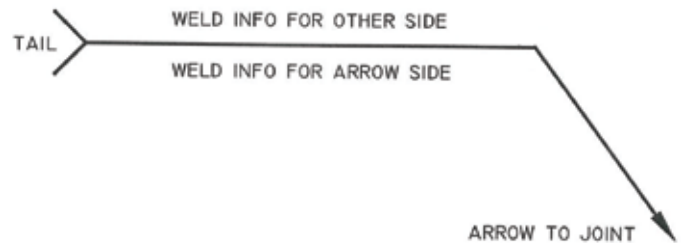


Figure 10-4

The weldment surfaces are identified by the side that the weld symbol arrow points to; this is called the Arrow Side while the opposite surfaces are identified as the Other Side. Figure 10-5

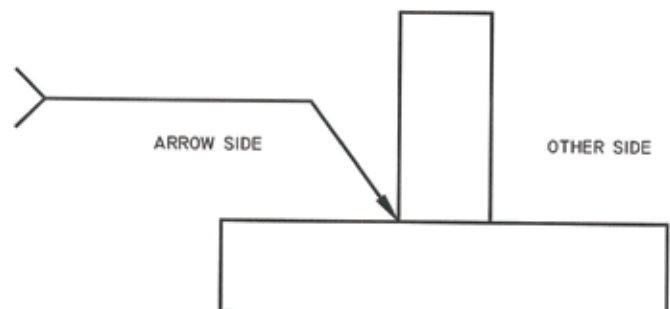


Figure 10-5

When a weld type is to be applied to the arrow side the information and proper symbol is attached to the lower side of the reference line. However, if the weld type will be applied to the other side then the information and proper weld symbol are shown above the reference line. Additionally, if a weld type applies to both sides of the joint then the information and proper symbols are shown above and below the reference line.

There are two commonly used supplementary symbols that can be attached to the weld symbol. They are the Field Weld flag and Weld All Around circle. The Field Weld flag denotes that the pieces of the weldment are to be set in place at the job site before they are welded together. The Weld All Around circle is used to denote when a continuous bead of weld is to be applied where the end meets up with the beginning, such as when welding a pipe seam or mounting a post to a flat plate. Figure 8-10

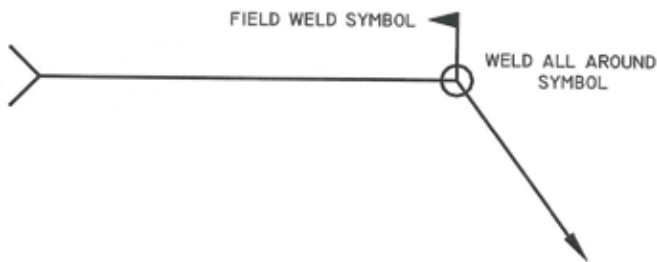


Figure 10-6

Welding Types and Their Symbols

The following information will identify the various weld types and their correlated weld symbol. The weld type symbol is normally the next item placed on the weld symbol. The weld type symbol identifies the form of the weld being applied to the surfaces. Through the use of dimensional values, you can specify the size of the weld leg length, the distance of separation (root opening), and the angle of the surfaces being welded. (Note: The following figures show a graphical representation of the weldment types and not the penetration of the weld metal.)

Fillet Welds: A fillet weld (pronounced “FILL-it”) is used to fasten two pieces of metal together that have been assembled to form a T-joint, L-joint (corner joint), or lap joint (one piece

placed on top of another piece). The fillet weld is roughly triangular in cross-sectional shape. When both legs of the fillet weld are the same length, the size is given once, as shown in Figure 10-7, 10-8, and 10-9. When the leg lengths are different in size, the vertical dimension is shown ahead of the horizontal dimension as shown in Figure 10-10. See Figures 10-11 and 10-12 for examples of a Corner joint and a Lap joint.

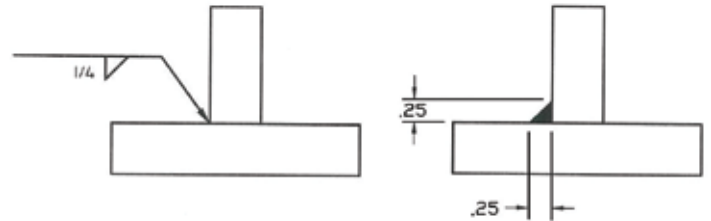


Figure 10-7: Fillet weld (T-joint) with equal legs on arrow side.

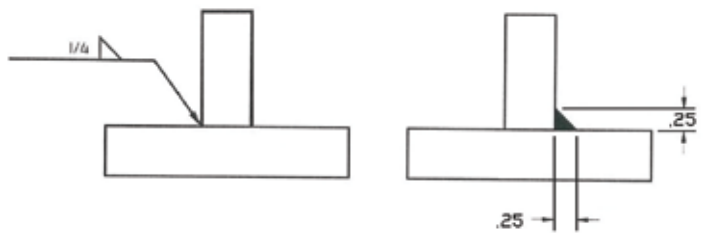


Figure 10-8: Fillet weld (T-joint) with equal legs on other side.

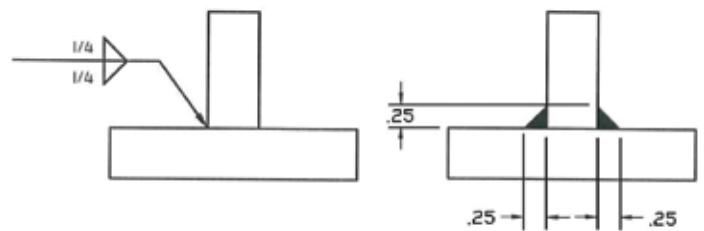


Figure 10-9: Fillet weld (T-joint) with equal legs on both sides.

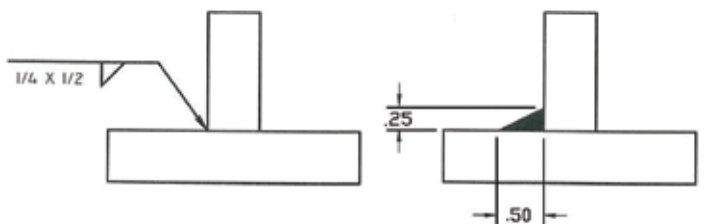


Figure 10-10: Fillet weld (T-joint) with different leg lengths on arrow side.

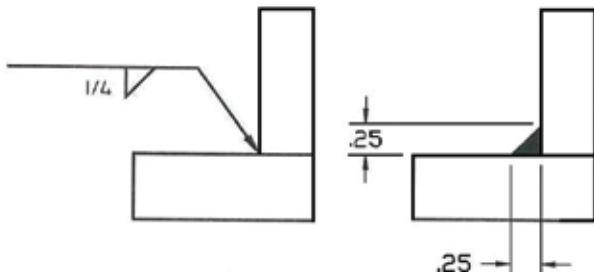


Figure 10-11: Fillet weld (Corner joint) with equal legs on arrow side.

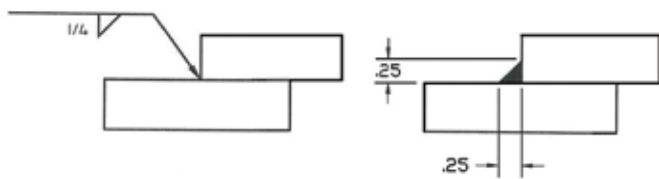


Figure 10-12: Fillet weld (Lap joint) with equal legs on arrow side.

Groove Welds: Groove welds can take on several different configurations; from the plain Square Groove weld (butt joint), Bevel Groove, V-Groove, to the U-Groove and J-Groove welds. Although some look very similar, they each have different roles in regards to strength and penetration of the weld. Take the Square Groove weld. This weld type is used to make edge-to-edge joints where the two pieces have a tight joint or a slight separation. When a separation is involved, this distance is known as the root opening. It is referenced inside the square groove symbol. See Figure 10-13.

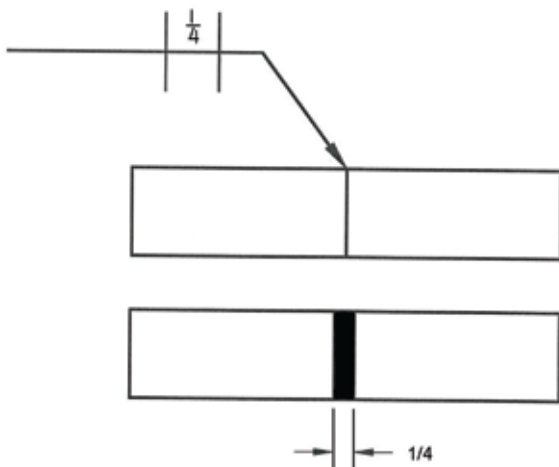


Figure 10-13: Square Groove weld showing root opening.

Next is the Bevel Groove weld. This weld type is created when one piece is square and the other piece has a chamfered edge that spans the whole thickness of the piece. The angle of the chamfer is called-out on the weld symbol and the perpendicular line of the bevel groove symbol is always drawn on the left side, regardless of the orientation of the weld itself, as shown in Figure 10-14.

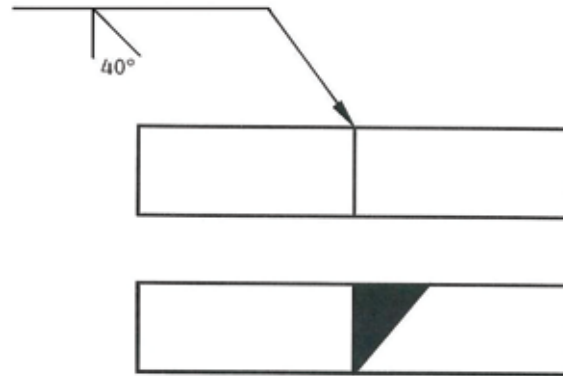


Figure 10-14: Bevel Groove weld.

Another Groove-type weld is the V-Groove weld. This weld type is similar to the Bevel Groove weld except that both edges are chamfered and that the angle given in the weld symbol is the included angle for the opening to be welded. See Figure 10-15a for an example. If the depth of the V is not full thickness, or half thickness in the case of a V on both sides, the depth is given on the left of the weld symbol. See Figure 10-15b.

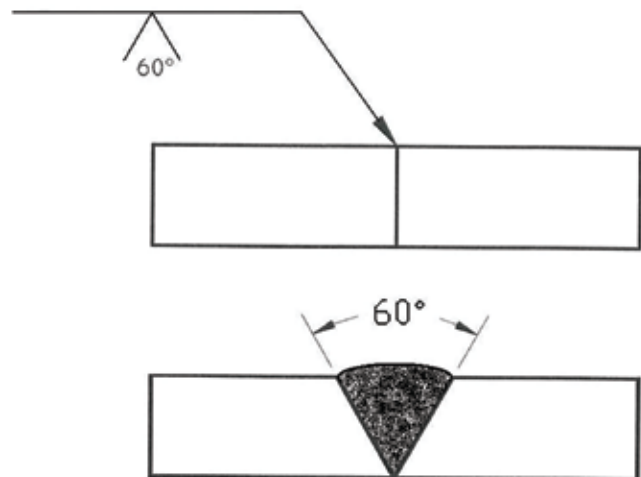


Figure 10-15a: V-Groove weld on arrow side and full thickness.

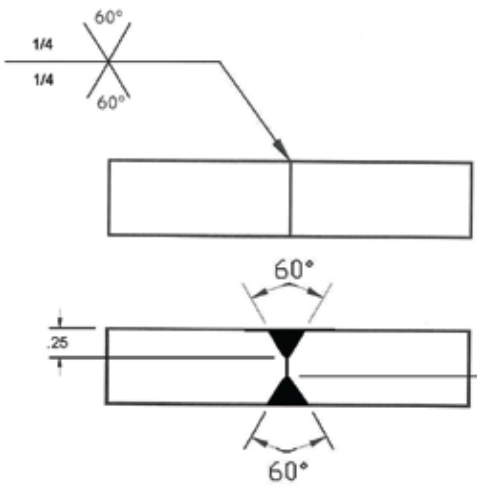


Figure 10-15b: V-Groove weld on both sides and partial thickness.

The next two Groove-type welds are similar in that they both use a concave treatment to the edge, or edges, of the pieces to be welded. The J-Groove weld has one piece with a square edge and the other with a concaved edge. As with the bevel groove weld, the perpendicular line is always drawn on the left side, and the arrow points to the piece that receives the edge treatment as shown in Figure 10-16. The U-Groove weld is the same except that both edges receive the concave treatment as shown in Figure 10-17.

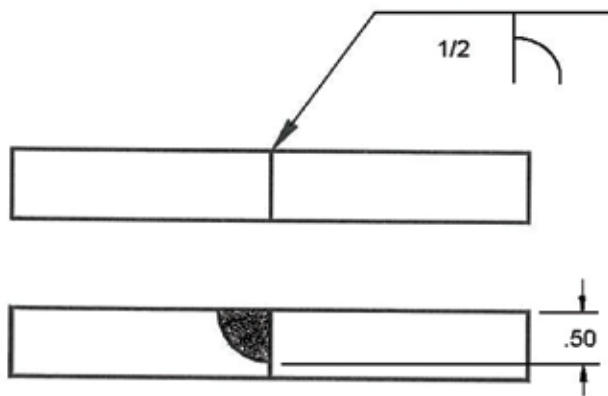


Figure 10-16: J-Groove weld on arrow side.

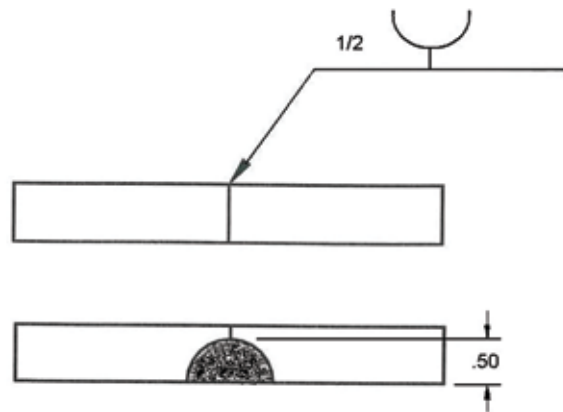


Figure 10-17: U-Groove weld on other side.

Resistance Welding: This type of welding involves passing a momentary electrical current between two electrodes in which the base materials are clamped. The two common weld types related to this method are Spot welding and Seam welding.

Spot welding is the most commonly used type of welding when thin materials need to be permanently bonded together. Sheet metal parts such as auto body components and air handling duct work are the usual recipient of spot welding. The process letters of RSW (resistance spot welding) are placed behind the weld symbol tail to identify this weld type. See Figure 10-18.

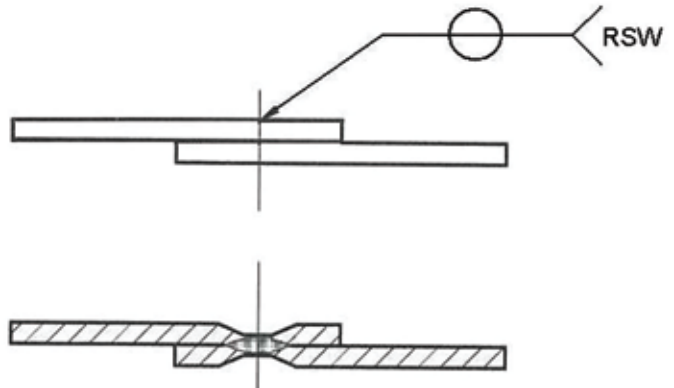


Figure 10-18: Spot weld (RSW) on both sides.

Seam welding is similar to spot welding except that the spots overlap each other which make a continuous weld seam. This method of welding is used when high shear strength is necessary or when connecting two edges together for a leak-proof seam. The process letters of RSEW (resistance seam welding) are placed behind the weld symbol tail to identify this weld type. See Figure 10-19 for an example.

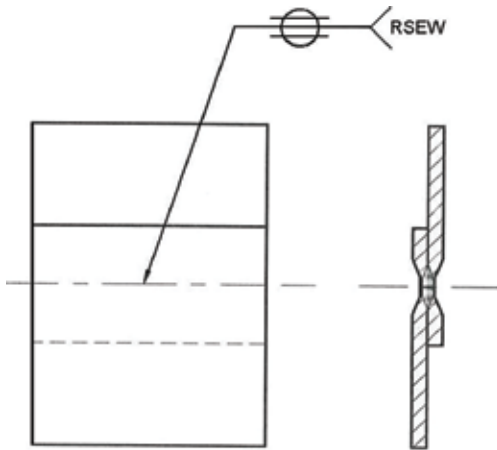


Figure 10-19: Seam weld (RSEW) on both sides.

Two other forms of weld types are Plug welds and Slot welds. These two weld types are very similar to one another. They both apply when joining overlapping metal pieces (similar to the principle of spot welding) except the material is thicker and therefore the pieces must be welded together. Plug welds use a series of holes in one of the pieces to allow welding of the two metals together. Slot welds use large slots in one of the pieces which allows for more weld material to be applied which usually yields a stronger bond for higher shear strength. See Figures 10-20 and 10-21 for examples of these two weld types.

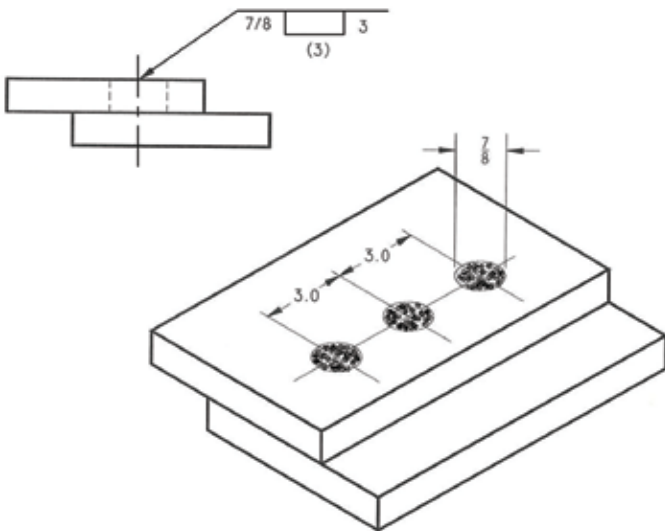


Figure 10-20: Plug weld.

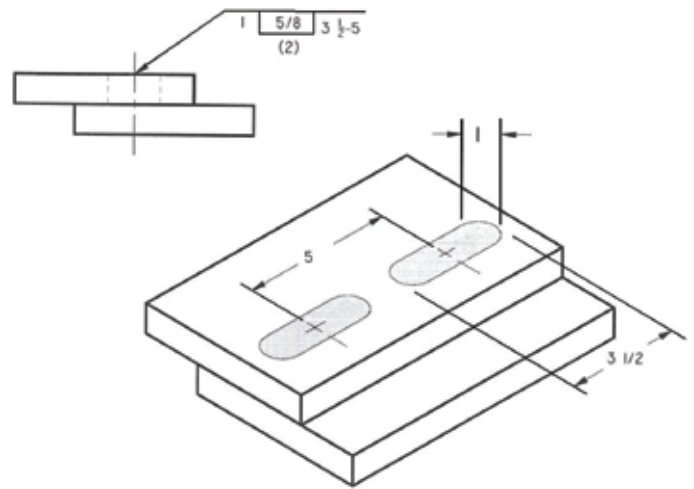


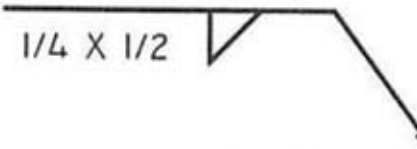
Figure 10-21: Slot weld.

Welding Exercise:

QUIZ:

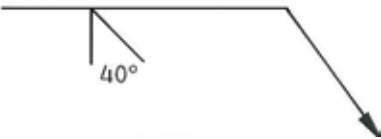
1. How strong is a properly welded Joint?
2. List the three most commonly used welding processes.
3. For oxygen gas welding, one cylinder is oxygen and the other is what kind of cylinder?
4. Why is a special hood worn when arc welding?
5. Name the parts that the metal is placed in between (current flows through them) when spot welding.
6. Welding symbols are standardized by what document?
7. Name the three main components of a weld symbol.
8. Which part of the weld symbol can be omitted if no reference is used?
9. Name the two sides that are used to identify the weldment surfaces in relationship to the weld symbol.
10. Name the two supplementary symbols explained in this module.

11. Name the three types of Fillet Weld joints.

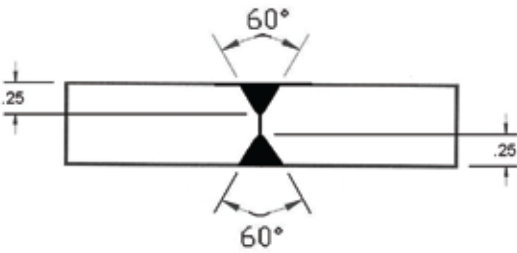


12. Explain this weld symbols type, weld side, and weld size.

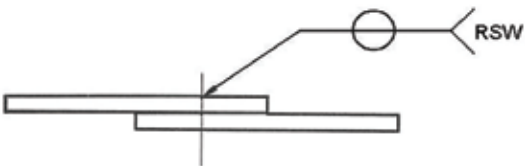
13. Explain this weld symbols type, weld side, and weld size.



14. Draw the weld symbol for this weld type.



15. Identify this weld type and its process letters.



CHAPTER 11

Project: Chuck Jaw

In this module, the mechanical drafter will understand the procedure to design a Chuck Jaw that will be used to shape a cylindrical component called a Hub. You will be guided through the various phases that you are likely to encounter when designing a machine tool such as a Chuck Jaw. You will also develop a better understanding of the design process from the beginning (the need for a tool) to the end (the completed tool ready to produce the Hub).

Important Note: for step by step instructions on how to create the Chuck Jaw project using A+CAD see chapters 13 – 17 of the text “Inside A+CAD 2008”.

First, you need to know what the function of a Chuck Jaw is before you design one for the Hub you will create. A Chuck Jaw is a component of a holding device called a Chuck (Figures 11-1a and 11-2). A chuck is a component of a machine called a Lathe (Figure 11-1); and a lathe is a machine that is used to cut cylindrically shaped material, such as a shaft or round bar, by turning the material in a rotary motion like that of a driveshaft of a car or the blades on a fan.

Figure 11-1: Lathe - Courtesy Micro-Mark tools

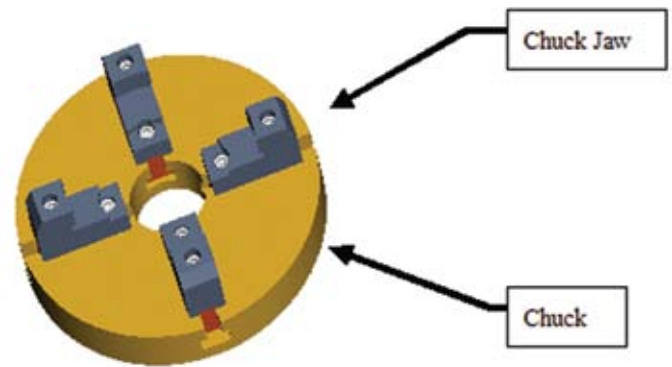


Figure 11-2: Chuck assembly

The lathe will produce the necessary features of the Hub. To do this, the lathe uses a tool post that holds a cutting tool that will be advanced into the spinning material to cut the features of the Hub. In some cases, special jaws need to be made to hold a part while it is being machined by the lathe. The design for your Chuck Jaw is one of these special cases.

In other words, to produce the Hub, the mechanical drafter will have to design the special Chuck Jaw first. The initial step is to determine which features from the Hub will be used on the Chuck Jaw to support the part while it is being machined. From this knowledge you should be able to design the Chuck Jaw and produce a detailed drawing from which the part will be fabricated.

The design process begins with a detail drawing for the Hub. (This completed drawing is shown in figures 11-3, 11-4, and 11-5.) You need to request from your machine shop to have a set of “soft” jaws manufactured to hold the material for the Hub while the additional features are added. Soft jaw is the term used to describe jaws that are specifically made to hold a specific part during a machining operation. They are usually made from a soft material, such as Aluminum or Steel, as opposed to the extremely hard material of standard jaws. Soft jaws are used to prevent surface damage by the clamping force needed to hold the spinning material in place during the cutting operation.

Your objective is to design and detail this set of soft jaws that will mount on the 4-jaw chuck shown in Figure 11-2 that will hold the

Hub shown in Figure 11-4 during the machining operation. To begin, identify the surface that the jaws will clamp onto and the surface that the jaws will rest upon.

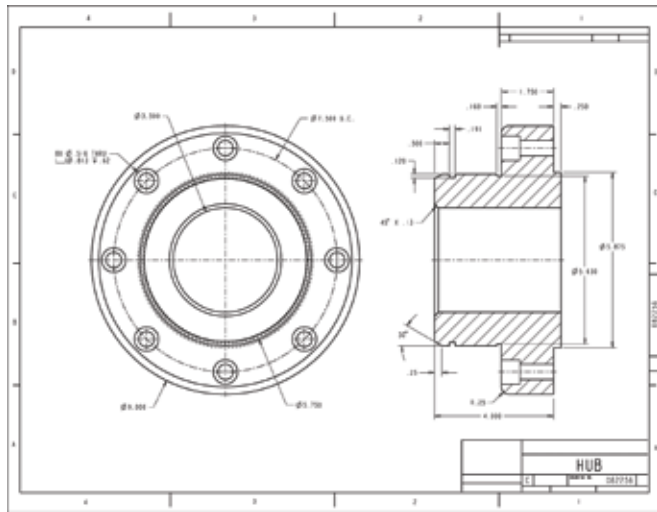


Figure 11-3: Hub detail drawing



Figure 11-4: Hub (Solid model isometric view)

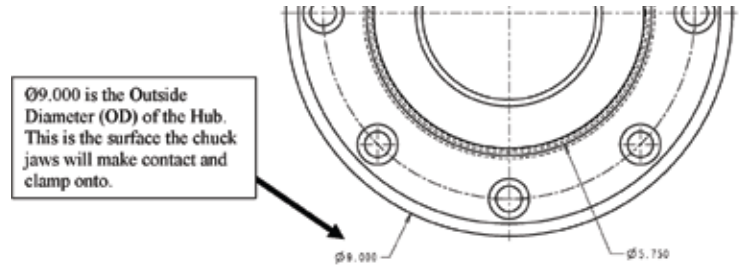


Figure 11.5: Top view of Hub

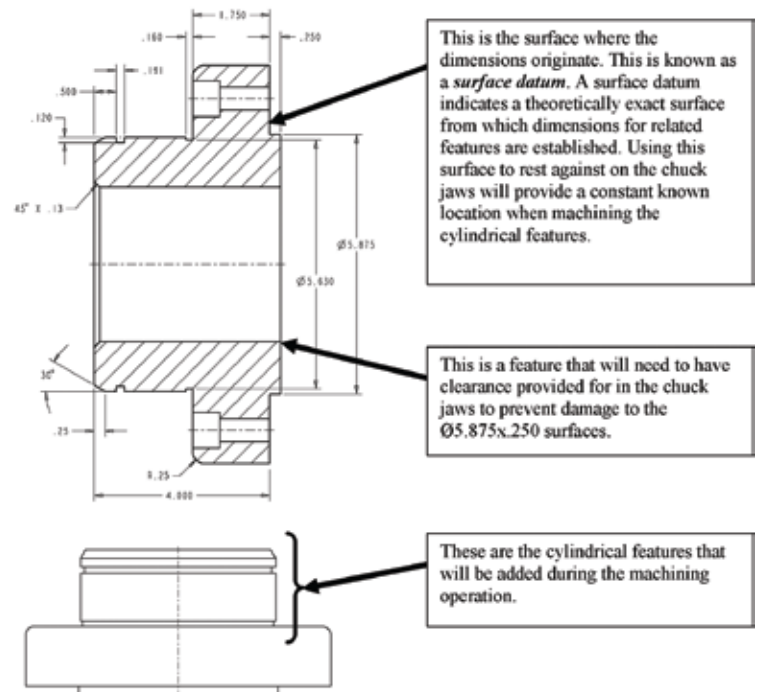


Figure 11-6: Side views of Hub

Now that the clamping and resting surfaces have been identified (Figures 11-5 & 11-6), it is time to explore some other design requirements that need to be established before designing the chuck jaw. The next element is to figure out how the chuck jaw will mount to the chuck (Figure 11-7).

The 4-Jaw Chuck as shown in Figure 11-7 is a simple representation shown for educational purposes only. This chuck is designed to have replaceable jaws which are illustrated by the four keys (shown in orange) that have two threaded holes for mounting bolts and one dowel for locating the jaws (Figure 11-8a & b). This particular chuck key has two tapped holes sized for 7/16-14 UNC bolts and one Ø.500 dowel used to locate the chuck jaw in an exact location in relation to the key.



Figure 11-7: Sample of a 4-Jaw Chuck

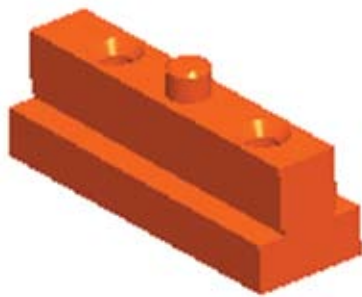


Figure 11-8a: Sample Chuck Key

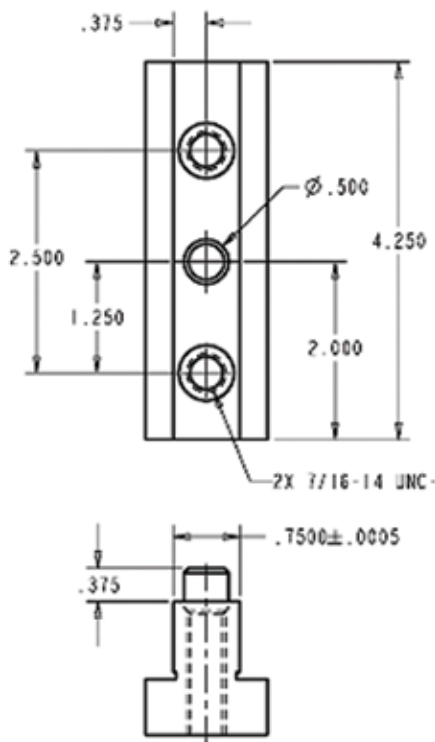


Figure 11-8b: Sample Chuck Key Detail

Notice the dimensions that explain the location of the two 7/16-14 UNC-2B tapped holes in relationship to the $\text{Ø}.500$ dowel. The dowel is the main feature that locates the two holes in regards to the Chuck Key. (Remember, a dowel is precise in size so a standard tolerance does not apply.) You will need to reference all of these dimensions in your design of the Chuck Jaw.

There is one more dimension that you will need to know before you begin the layout. That dimension is the height of the Chuck Key which extends above the surface of the Chuck (Figure 11-9). This is necessary to insure enough clearance for the jaws to move in and out on the keys when the jaws are tightening against the Hub.

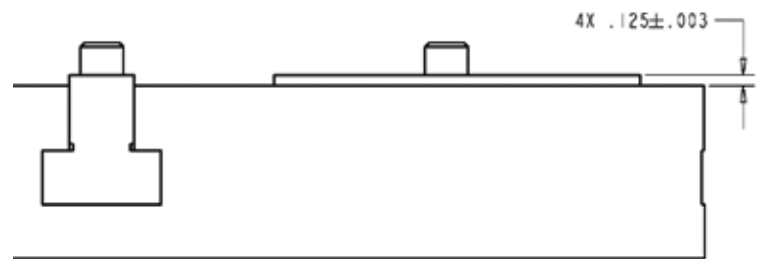


Figure 11-9: Chuck Key height extending above Chuck

Layout of the Chuck Jaw Drawing

To begin this drawing, start with some basic drafting and drawing setup information. This is going to be a Mechanical Format drawing using the English decimal units as opposed to the ISO metric units. Because you are using the English units, you will be using the orthographic projections for the Third Angle Projection scheme.

NOTE: **Process Notes** will be added to alert you of time-saving tips and other information that will aid you in producing a well-defined drawing.

Step 1: Start a new drawing using a C-Size (22 x 17) Mechanical Format (ANSI-C) layout.

Step 2: Set up Layers for your Object lines, Center lines, Hidden lines, Phantom lines, Cutting Plane lines, Dimension lines, and Notes. While in the Layers Explorer dialog box, set the layer colors and line weights. There is a 2 to 1 aspect ratio for line weight of the Object and Cutting plane lines to the supporting lines (all the other line types). For your drawing, set a line weight of .60 mm for the Object and Cutting Plane lines and use a line weight of .30 mm for the remaining supporting line types.

Process Note: Use a Text Style (Font) of RomanS for all notes and dimensions unless otherwise instructed.

Step 3: Set your Layer Control to the Note layer and add the following information in the Title Block: the name of the drawing: CHUCK JAW; a Drawing Number (such as 082257); the Standard Tolerances of: .XX = ±.01, .XXX = ±.005, .XXXX = ±.0005, and a = ±30'; note that you are using THIRD ANGLE PROJECTION and show the proper symbol indicating this projection scheme; and add your name as the DRAFTER. (The CHECKER will be the person who reviews your drawing)

Process Note: Text height of information in the Title Block will vary. The drawing name should be .25 (1/4") high while the remaining text will be between .07 and .16 in height. Adjust as necessary to make the information fit yet readable when printed.

Step 4: While still on the Note layer, add the following information in the upper left corner for your General Notes:

NOTES: (UNLESS OTHERWISE SPECIFIED)

1. MATERIAL: ALUMINUM 6061 T4 – T6
2. BREAK ALL SHARP CORNERS AND EDGES.
3. FINISH: 63 R.M.S. DO NOT ANODIZE.
4. MARK PART NUMBER ON FRONT SIDE.

The heading, NOTES, has a text height of .25 (1/4") while the body of the notes has a text height of .19 (3/16").

Process Note: Before you start designing your project, set up your Dimension Settings at this point so that they are ready when you to detail your part.

Step 5: Use .125 (1/8") for arrow size, extension line extension (extend lines past), and text height. Use .062 for extension line outset (offset from origin). Set your initial Tolerance Text Decimal places and the Linear Dimension Decimal places to 3 (default is 4). Finally, under the Units tab, set the zero suppression for Linear Dimensions to suppress Leading zeros, and set the suppression to "both zeros" for Angular Dimensions. (Remember that the Comprehensive scale factor (default of 1.0000) may need to be adjusted to produce the desired text height on the printed copy should you use a Paper Space scale of anything other than 1:1.)

Process Note: As stated earlier, you will design one "soft" Chuck Jaw that will fit on the illustrated 4-Jaw Chuck shown in Figure 11-7. Here is some further information about the size of the Chuck Jaw (referred to as "part" in the following steps) and some required features that need to be designed into the part:

- The part should be overall 4.25 long by 2.63 tall by 1.75 wide;
- It will need two clearance holes for mounting bolts and a precision hole in the bottom on which to locate with the Ø.500 dowel on the Key;
- It will need a shelf for clearance of the Ø5.875 X .250 flange on the Hub as well as a corner relief on the primary shelf for the Ø9.000;
- Review Figures 11-5 and 11-6, along with their related text, for supplementary information.

Step 6: Start designing. Make sure you are in Model Space and remember to always draw at full size. Set your Layer Control to one of your Object line layers. Draw a horizontal construction line followed by a vertical construction line to the left and intersecting with the horizontal line. See Figure 8-10.

Process Note: Construction lines allow for a quick representation of the part.



Step 7: Use the Offset command to generate the “box” shape of the part. Offset the horizontal line by 2.63 (up) and the vertical line by 4.25 (to the right). See Figure 11-11. Follow the same steps to create a Top View. See Figure 11-12. Remember to keep all views aligned with each other.

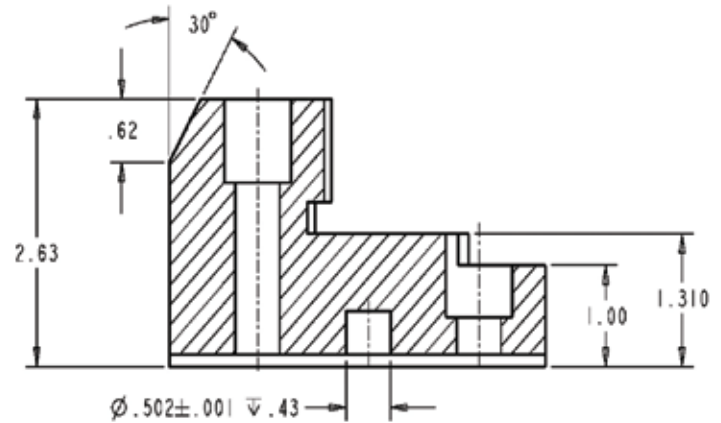


Figure 11-13a: Sectional view of front

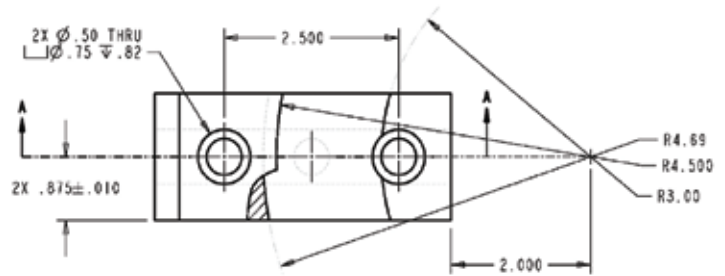
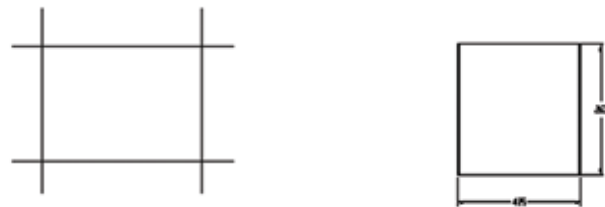
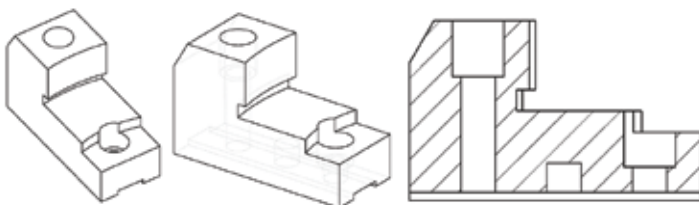


Figure 11-13b



Process Note: Refer to the following images to see how the finished part will look.



Step 8: Draw the orthographic projection of the part. See Figures 11-13a through 11-13c for the dimensions for which you can construct the part.

Step 9: Once the three basic views are constructed, draw the bottom view. Refer to Figure 11-8b for helpful dimensions on detailing the bottom view. Note the $.7500 \pm .0005$ dimension for the Key’s width and the slot feature in Figure 11-13c with the $.7515$ dimension. This slot feature will aid in locating the part on the Key. See Figure 11-13d for the dimension values that locate the hole for the $\varnothing .500$ dowel on the Key and the starting location for one of the through holes for the fastening hardware.

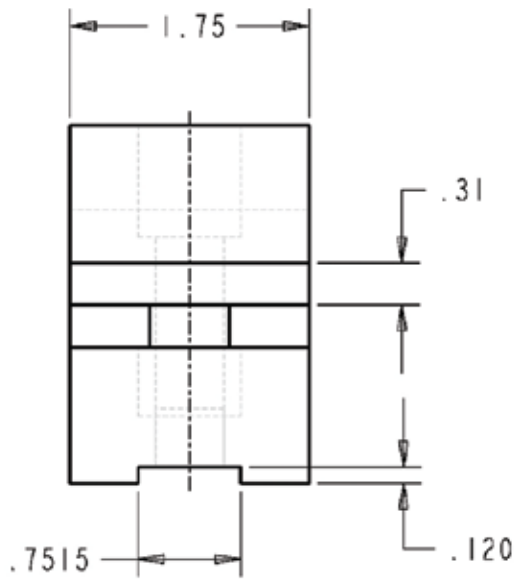


Figure 11-13c

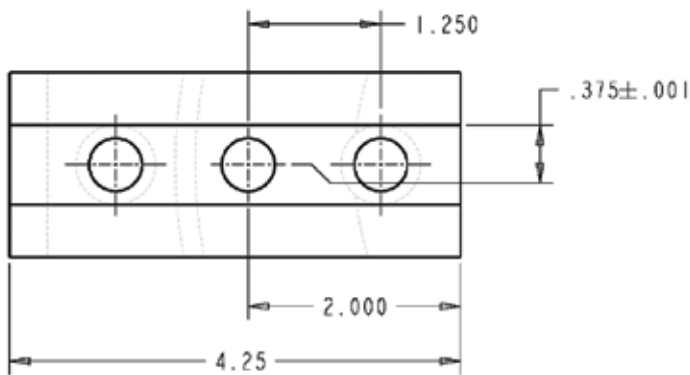


Figure 11-13d

Process Note: The dimension values and their respective applied tolerances may not be entirely correct. Case in point, the 1.310 value in the Front view will not accurately locate the surface for the Surface Datum of the Hub to rest on. A better choice would be to dimension from the top of the key slot to that surface and apply a tolerance of $\pm .003$ to this feature. The new dimension would be $1.190 \pm .003$.

Step 10: Check your four views for proper alignment. Make sure that all lines are transferred from one view to the next. Once you have done so, remember to change the layer values for all lines so that hidden lines are hidden lines and the lines that represent the center of the holes are center lines.

Step 11: Continue applying the remaining drawing details to each view. Observe that in all Figures of 11-13 the proper use of hidden lines, center lines, a Full Sectional view, and a Broken Section view. The Top view has the Broken Section view to highlight the radius value of 4.69 for the corner relief feature. Without doing so would have left a hidden line feature that could not be dimensioned. The Full Sectional view in the Front view may not be necessary however it does add clarity to the features. Add the Cutting Plane line in the Top view and note the section name on the Front view.

Step 12: Arrange the views to be about 2.38 apart initially. Once you start applying dimensions you may find this distance needs adjustment. (The Right Side view may end up about 3.25 away from the Front view.)

Step 13: Apply the dimensions including the appropriate tolerances where needed.

Process Note: You should be able to show the model at 1:1 in Paper Space and therefore you can dimension the model with a Dimension Setting Comprehensive Scale Factor of 1.

Step 14: Switch to Paper Space and arrange the drawings to clearly show all views. Refer to Figure 11-14 for a sample representation of the part layout with the dimensions applied. The general notes and title block detail have been omitted.

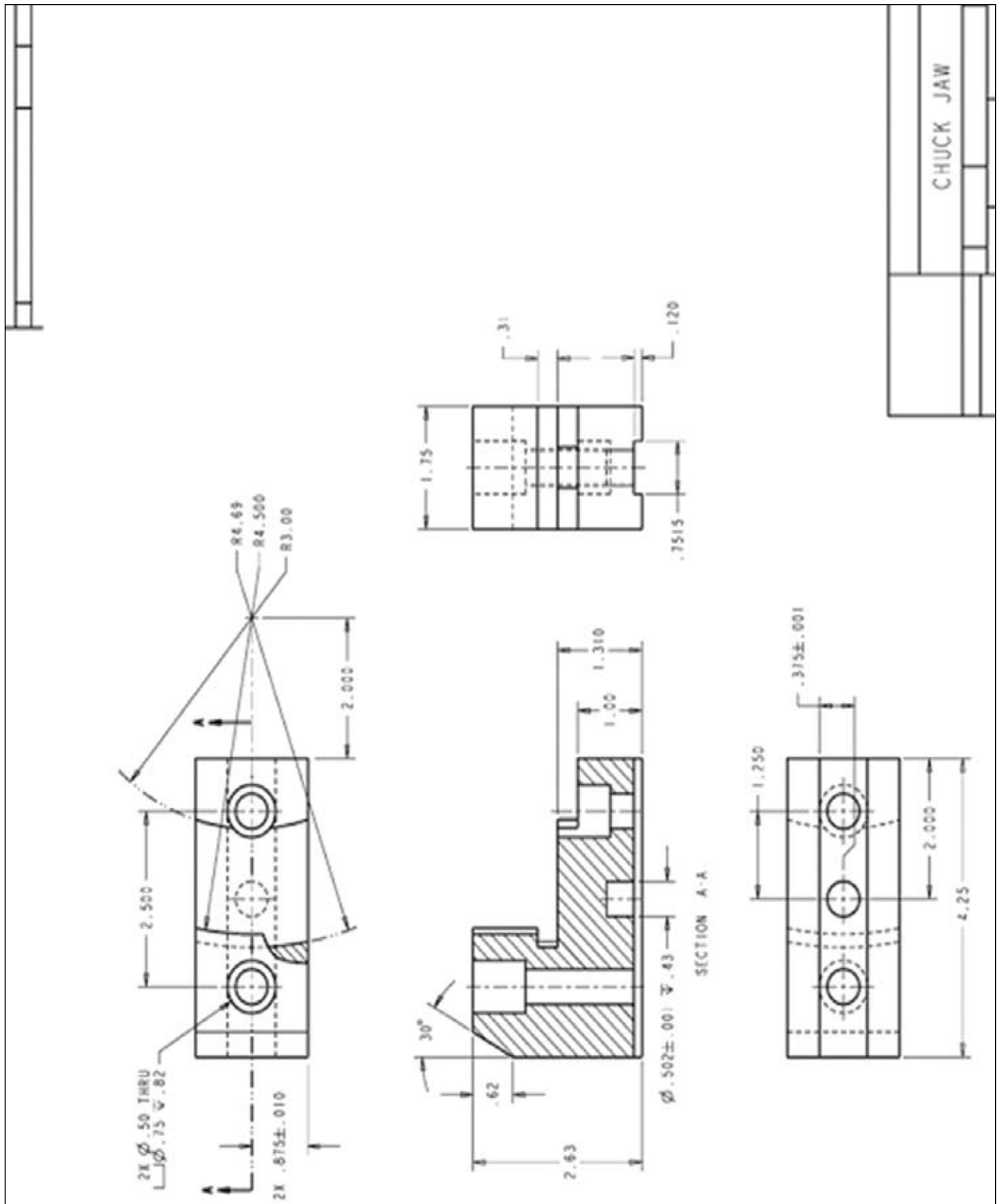


Figure 11-14: Chuck Jaw

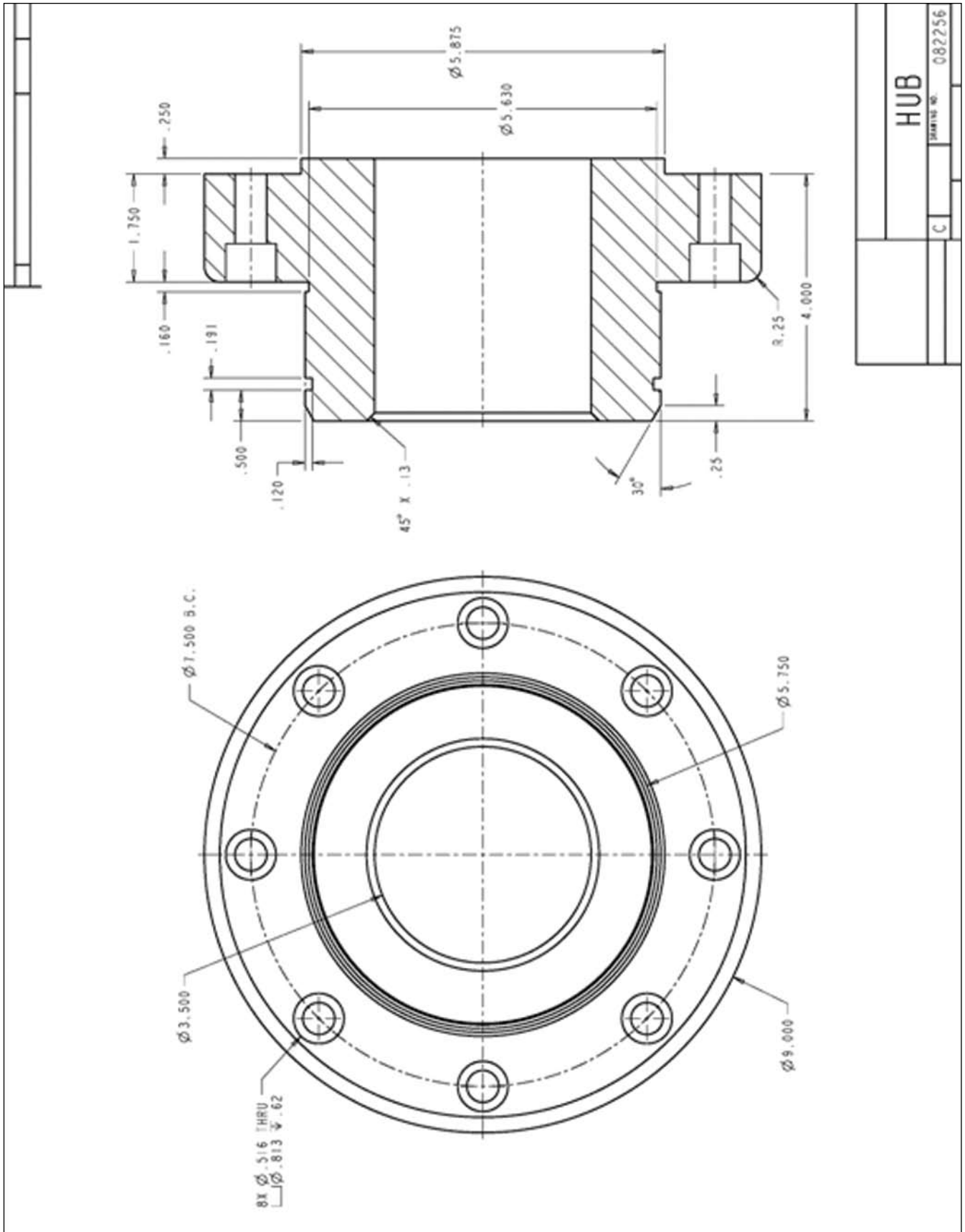


Figure 11-15: Hub

