

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.

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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.

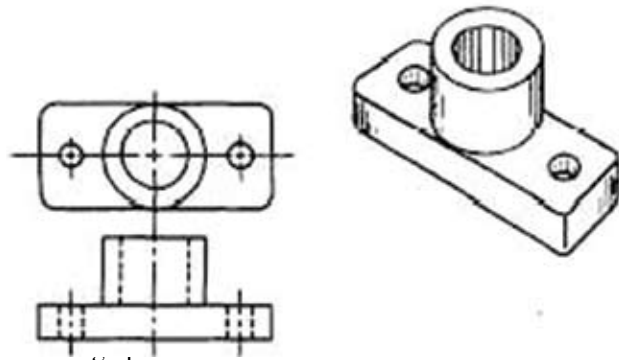
Sketching before drawing is essential. It has been documented that drawing before making a list of sketched alternatives will result in poorly conceived products. The reason behind this is simply based on customer expectations. A CAD drawing implies a more advanced state of planning which will intimidate 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 standard 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

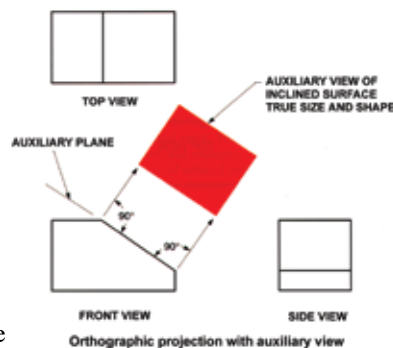
- Corner removed from end of cylindrical surface
- At an angle to face
- Used to facilitate assembly

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

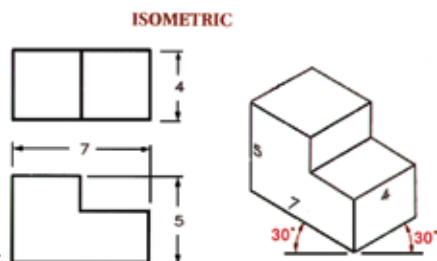
- Concave surface (interior radius)
- At intersection of two surfaces of an object

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



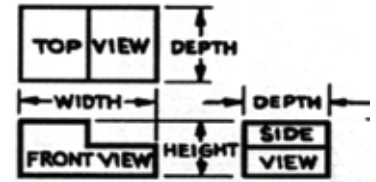
Jig

- 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

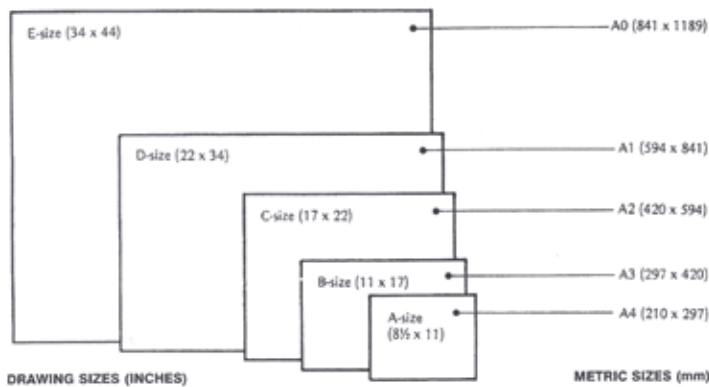
Drawings made for mechanical design are known as technical drawings. There are various elements of a technical drawing that have relative importance and relationship to each other. Every drawing should contain select information including the ***format size***, the ***title*** of the part, the assigned ***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

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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).



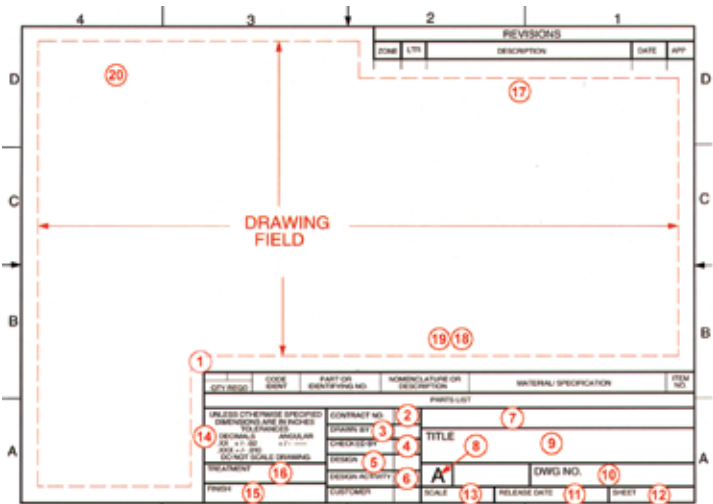
- A – Size = 8-1/2 x 11 (landscape or portrait orientation)
- B – Size = 11 x 17 (landscape or portrait orientation)
- C – Size = 17 x 22 (landscape orientation)
- D – Size = 22 x 34 (landscape orientation)
- E – Size = 34 x 44 (landscape orientation)
- F through J are available although not commonly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

The components that make up the drawing format include:

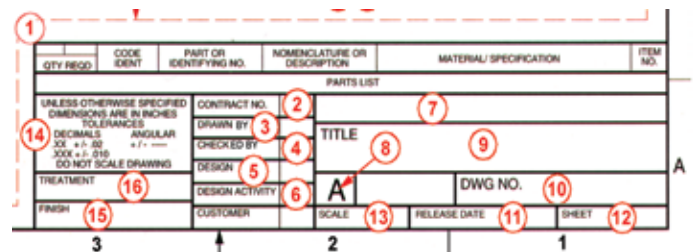
- Drawing field
- Title block location and contents
- Revision block location
- Parts listing or the Bill of Materials (BOM)
- Zoning

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

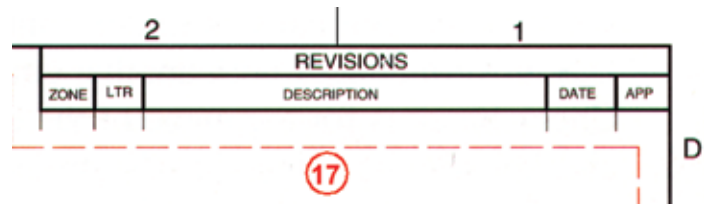
here only when appropriate or required.

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.

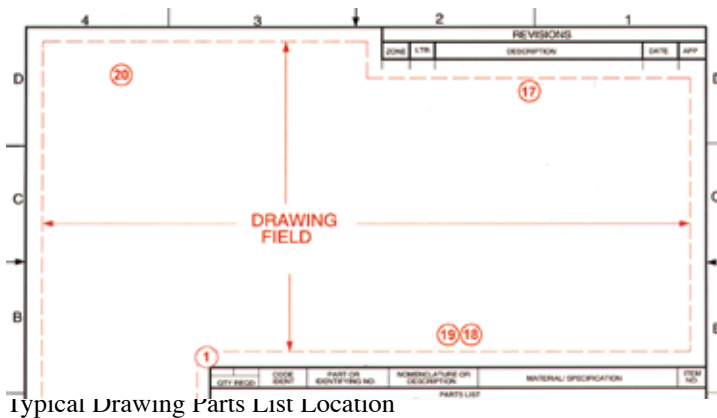
****As a side note**:** This information is usually established by the company based on their requirements or industries standards. This information is not usually changed from drawing to drawing, hence the title Standard, and instead any global tolerance changes should be noted in the general notes.

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.)

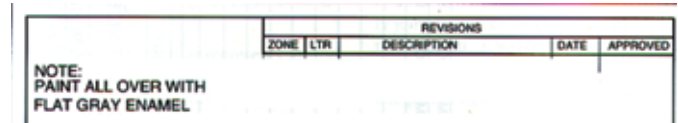


Typical Drawing Revision Block

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.



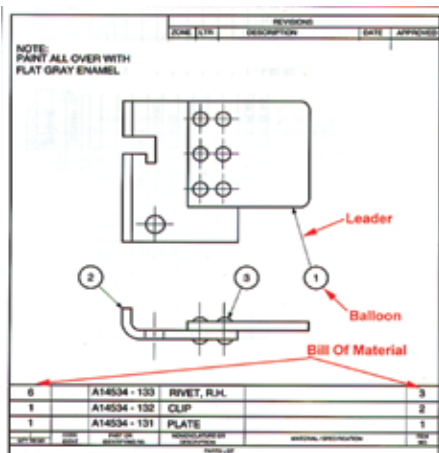
inch (1/2") text, however they can vary in size from .31 (5/16") to .75 inch (3/4") in diameter, depending on the size and scale of the assembly drawing.



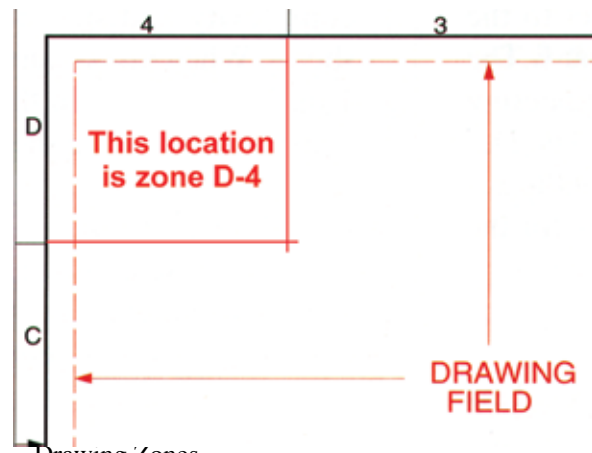
18. **Parts List** - The parts list is located in the lower-right corner of the drawing, directly above the title block. The list indicates the items, materials, and quantities required to complete the part or assembly. It is referred to as the *Bill of Materials (BOM)*. The parts are normally arranged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

Typical General Notes Location

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.



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:

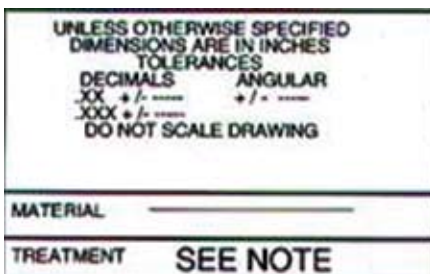
QUIZ:

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.

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

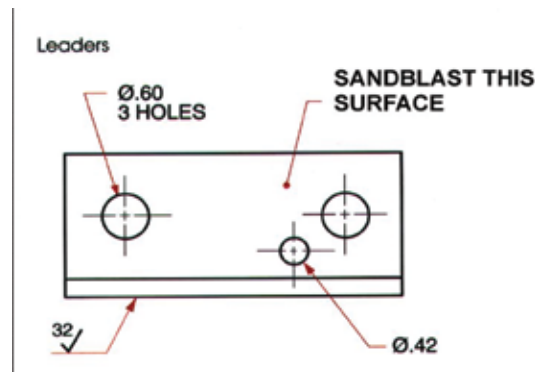
There are three main types of notes used on a drawing, listed in their order of importance, which are the Title Block notes, General notes, and the Part Detail notes. As mentioned in the above section, the Title Block contains an area where Standard Tolerances are noted and apply to the entire drawing 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.

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:
"XXXXXXX 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.



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.

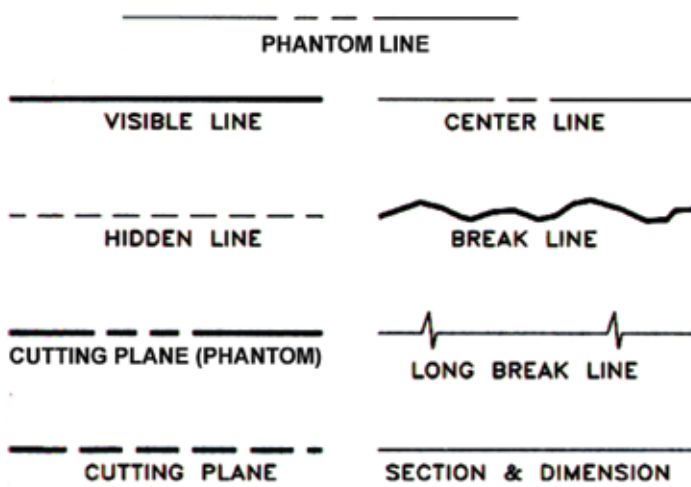
Line Type Settings

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.

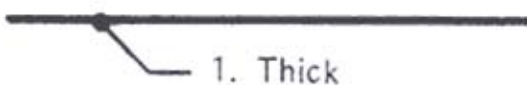
LINE WEIGHTS



Line Types used in a Technical Drawing

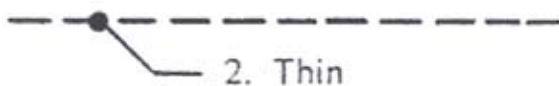
Visible line (Object line - #1)

- 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



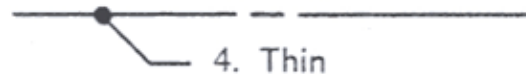
Section lines (Hatching - #3)

- 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

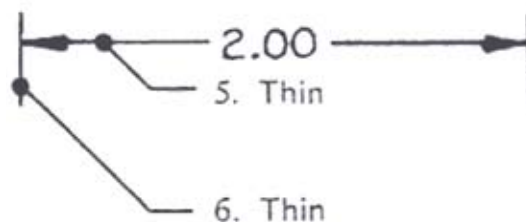


Dimension line (#5)

- Denotes the extent of the dimension
- Use with arrowheads

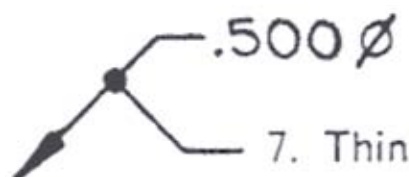
Extension line (#6)

- 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

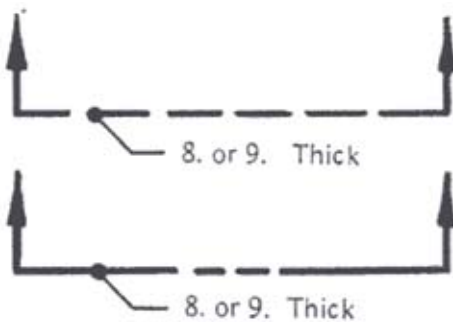


Cutting-plane line (#8 or #9)

- 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

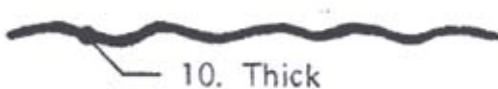
Viewing-plane line (#8 or #9)

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



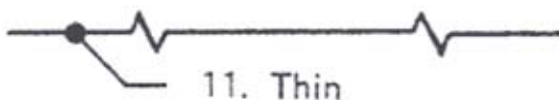
Short break line (#10)

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



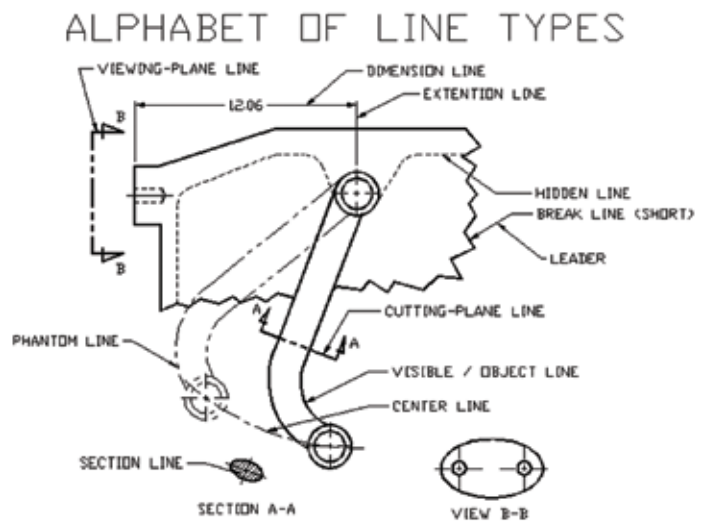
Long break line (#11)

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



Phantom line (#12)

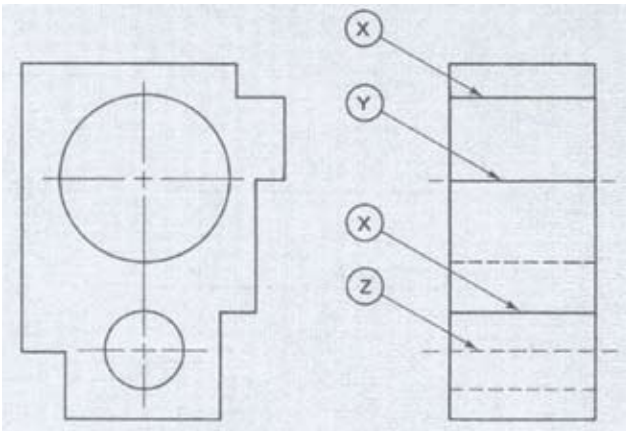
- Replaces repetitive detail - e.g.-coils, gear teeth, threads, etc.
- Drawn with two short dashes between single long dashes
- Lines are thinner than the cutting/viewing-plane line
- Represents the outline of an adjacent part
- Shows alternate position of a given part



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

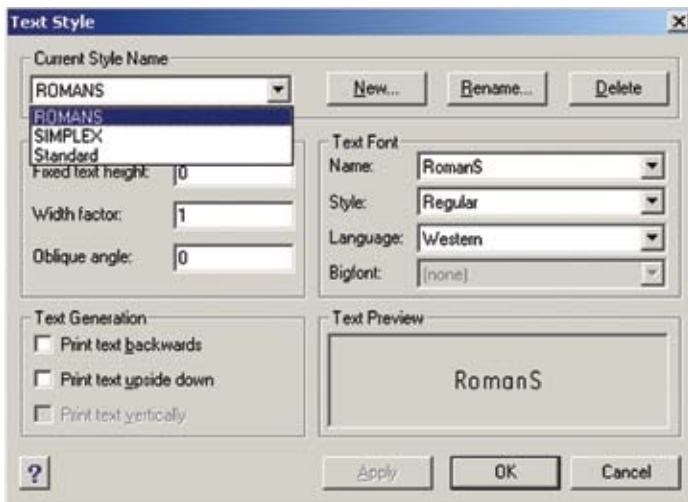
Example: 1. BREAK ALL SHARP CORNERS AND EDGES

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:

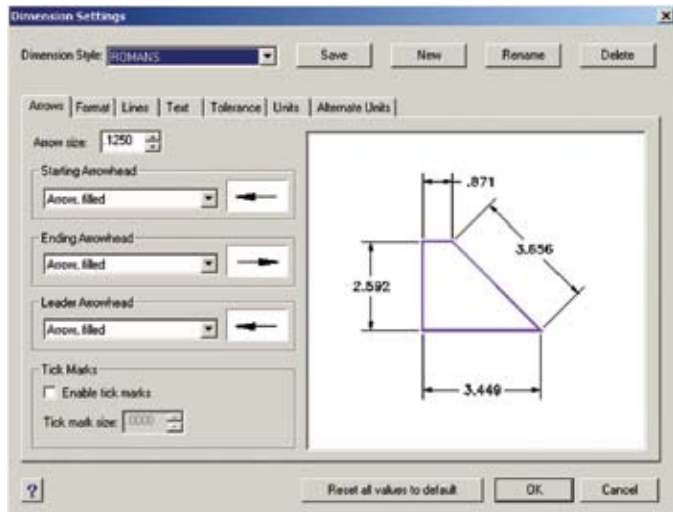
3) Dimension settings. The components that make up the basic dimensioning elements include:

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

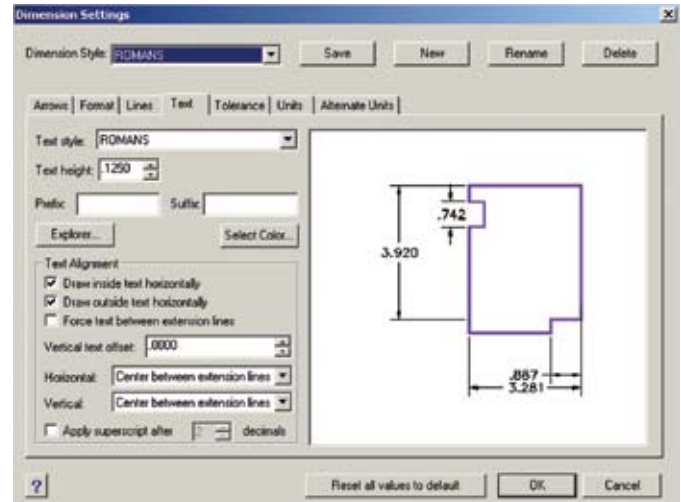
These components are set up by the drafter using the CAD software dimensioning style settings dialog box. Here are some of the more common values for each of these elements:

- 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

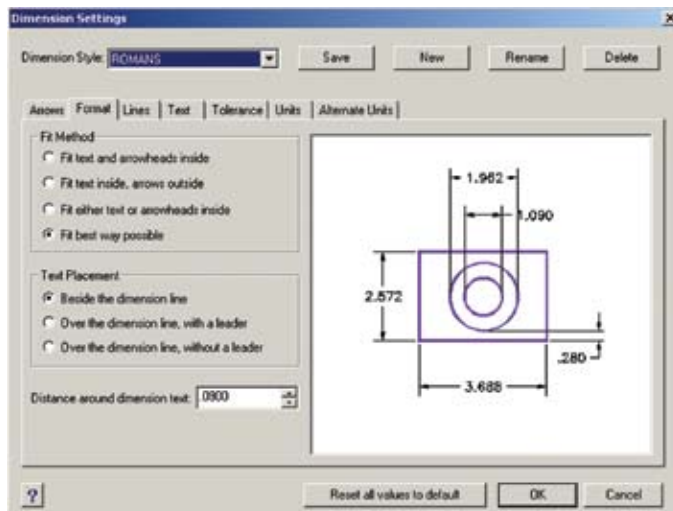
Arrowheads:



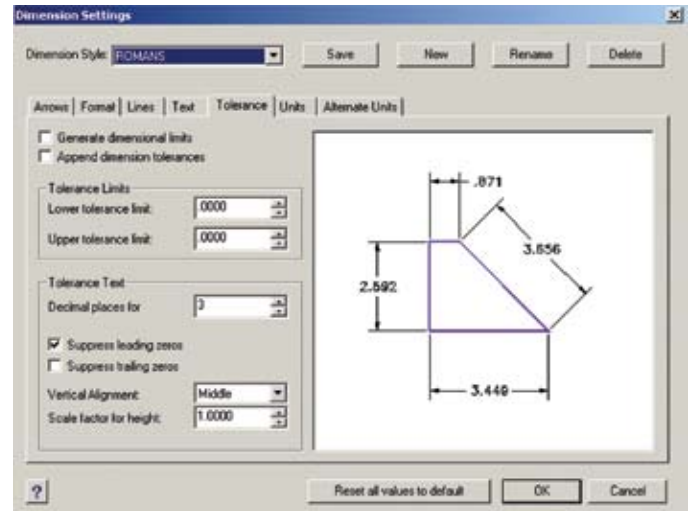
Text:



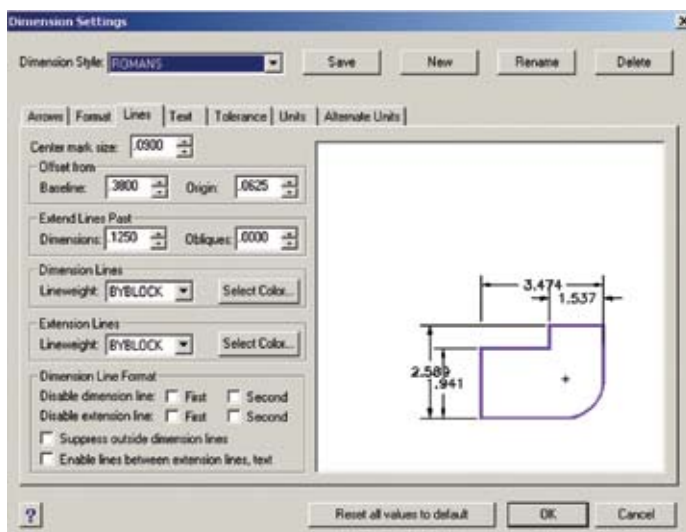
Dimension Format:



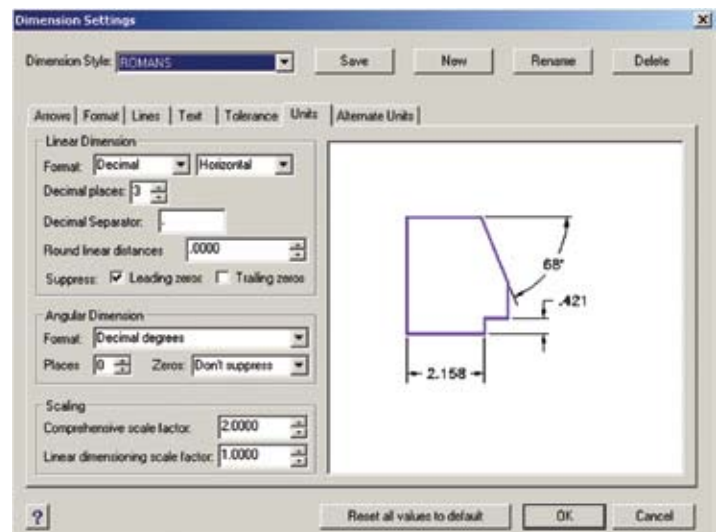
Tolerances:



Lines:

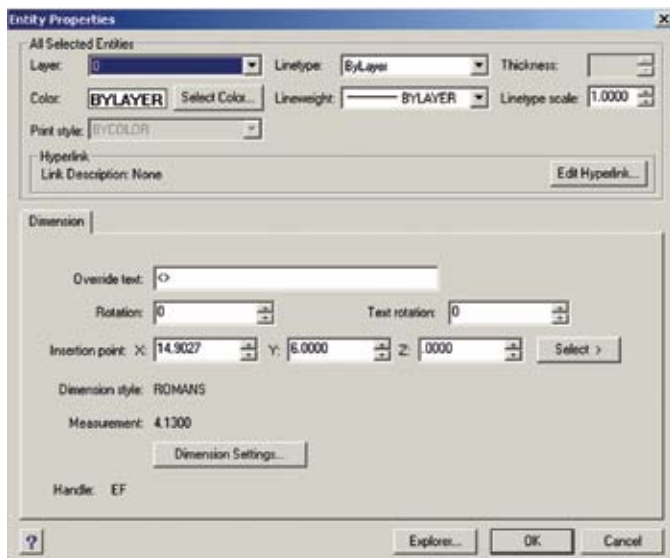


Units:



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).

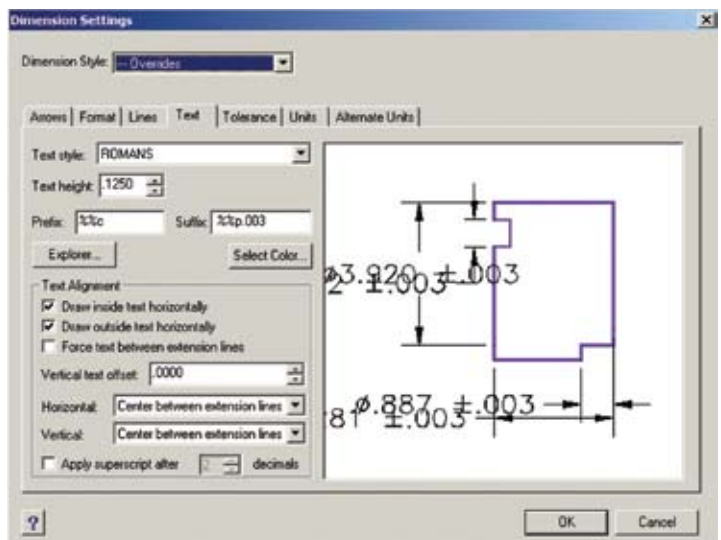
- ° = Degrees
- ± = C
- Ø = Diameter (DIA)
- v = Counterbore (CBORE)
- w = Countersink (CSINK)
- x = Depth (DP)



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?