Blueprint Reading

Blueprint Reading

MARK LORIER

WISTECH OPEN

Contents

ABOUT BLUEPRINT READING

Blueprint Reading is a practical guide designed to help students develop the essential skills needed to interpret and understand technical drawings, diagrams, and schematics. These visual documents are crucial across industries such as construction, manufacturing, engineering, and maintenance, guiding fabrication, construction, and equipment assembly processes. This book covers the basics of blueprint reading—including symbols, dimensions, and notations—while providing clear explanations, examples, and hands-on exercises to build a strong foundation. Whether you're pursuing a career in the trades, architecture, or engineering, the ability to extract key details from drawings and visualize complex designs is a fundamental skill for success in technical fields, and this resource serves as a valuable tool in mastering it.

USING THE DIGITAL TEXTBOOK

The following YouTube video provides a quick overview of how to navigate the digital version of the textbook:

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://wtcs.pressbooks.pub/blueprintreading/?p=4#oembed-1

ACKNOWLEDGMENT OF SUPPORT AND

DISCLAIMER

This book was developed through the WisTech Open project and supported by funds from the Wisconsin Technical College System (WTCS). However, this content does not necessarily represent the policy of WTCS, and you should not assume endorsement by the state or federal government. More information about the WisTech Open project can be found at wistechopen.org.

Preface

ABOUT THIS BOOK

Author

- Mark Lorier, Machining Instructor, Lakeshore College

Reviewers

Peer reviewers of the first edition of this textbook included faculty experts from Wisconsin technical colleges:

- Jonathan Edwardson, Welding/Metal Fabrication Instructor, Nicolet College
- Robert Leafblad, Welding Instructor, Northcentral Technical College
- Richard Pfister, Machinist/Tool & Die Apprenticeship Instructor, Lakeshore College
- Tom Roehl, Quality & Advanced Manufacturing Technology Instructor, Moraine Park Technical College
- Mark Rasmussen, Machine Tool Technics Instructor, Northwood Technical College
- Peter Rathsack, CNC Programming Instructor, Lakeshore College
- Matthew Roskpf, Electro-Mechanical Automation/Maintenance Instructor, Lakeshore College
- Ben Sorensen, CNC Technician Instructor, Northwood Technical College
- Joe Vydrzal, Mechanical Design Instructor, Chippewa Valley Technical College

WisTech Open Team

- Nic Ashman, MLIS, Graphics Editor
- Sophia Dramm, MA, Copy Editor
- Kelly Carpenter, MLIS, Librarian
- Jane Flesher, MST, Copy Editor
- Vince Mussehl, MLIS, Project Director
- Samantha Rubenzer, MA – Teaching & Education, Instructional Designer/Technologist

---

LICENSING/TERMS OF USE

SUGGESTED ATTRIBUTION STATEMENT

Unless otherwise noted, below is a suggested attribution statement for this resource:

- Lorier, M. (2025). Blueprint Reading by WisTech Open is licensed under CC BY 4.0

If you use this textbook as a bibliographic reference, then you should cite it as follows:

- Lorier, M. (2025). *Blueprint reading*. WisTech Open. https://wtcs.pressbooks.pub/bluepintreading/

REPORT ADOPTIONS OR CORRECTIONS

If you've adopted this book for a course or would like to suggest a correction, please let us know by filling out the form at wistechopen.org/

contact. Your feedback helps us improve and advocate for WisTech Open—we appreciate your support!

Introduction to Blueprint Reading

In technical college programs across the country, students are preparing for hands-on careers in fields like construction, manufacturing, architecture, HVAC, and engineering. No matter the trade, one skill consistently rises to the top: the ability to read and interpret blueprints. *Blueprint Reading* is designed to help students develop this essential skill through clear explanations, real-world examples, and interactive practice. Whether you're just beginning your program or advancing in your field, this book lays the groundwork for understanding the visual language that drives design, fabrication, and construction processes.

To understand the importance of blueprint reading, imagine Jordan, a student in a technical college welding program. During a summer internship at a metal fabrication shop, Jordan was asked to interpret a complex shop drawing for a custom stair railing. Thanks to blueprint reading skills developed in class, Jordan could confidently identify key dimensions, symbols, and notations. Not only did this knowledge help avoid costly fabrication errors, but it also impressed supervisors and opened the door to a full-time job offer after graduation. Jordan's experience is just one example of how these skills can create real opportunities in the workforce.

This textbook supports the hands-on learning approach at the heart of technical education. It bridges classroom knowledge with workplace demands, giving students the confidence to read and apply technical drawings in real-world settings. Whether you're aiming to build, repair, design, or install, this resource will help you visualize projects before they come to life—and ensure they're built right the first time.

1. Types of Drawings and Prints

1.1 INTRODUCTION

Learning Objectives

- Explain the difference between drawings and prints
- Describe engineering drawings

Terms

- Design phase
- Production phase
- Computer-aided drafting (CAD)
- Three-dimensional (3D) model
- 3D printers
- Computer Numerical Control (CNC) machines
- Detail drawings
- Print reading

Have you ever considered the steps that go into making the everyday products on which we rely? In manufacturing any part, a drawing is crucial; this can vary from simple objects like a paper clip, which might need just a handful of drawings, to complex products such as computers or cars that require thousands of drawings for their creation.

The details of a person's design are communicated through the use of drawings and prints. This chapter will introduce the manufacturing process and the use of drawings and prints within the manufacturing process.

1.2 THE MANUFACTURING PROCESS

The process of manufacturing a part or component can be condensed into two categories. The first process is the **design phase**, where a person's or a group's ideas are moved to drawings and prints. The second process is the **production phase**, during which these drawings and prints are used to produce the final product.

1.3 DESIGN PHASE

After identifying the need or intent to develop a product, a specialized design team is assembled. This team typically comprises designers and engineers responsible for generating the essential documentation to achieve the design objectives. Additionally, machinists and inspectors are included to ensure that the design can be produced efficiently. This critical stage, the design phase, involves creating all necessary documents required for product fabrication.

Most products consist of multiple components. For example, see Figure 1-1. Each of these components is separately designed and frequently manufactured in different locations, often by distinct companies both domestically and internationally. These components are then transported to a specified location for assembly.

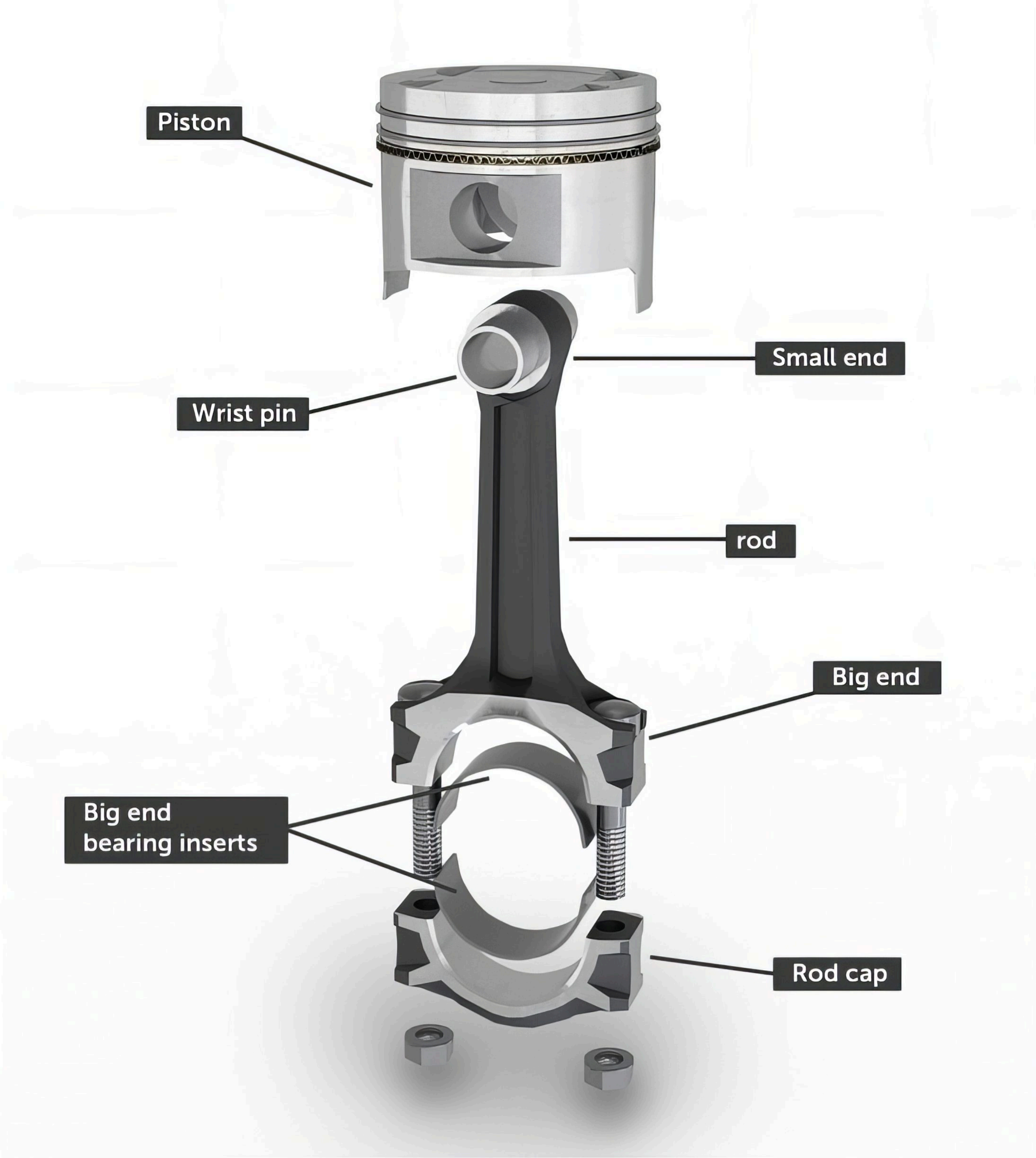


Figure 1-1. Several different components of a piston and connecting rod assembly.

1.4 THREE-DIMENSIONAL MODELS

Individual components are typically designed through **computer-aided drafting (CAD)**. A CAD system is a software program created to increase the speed, efficiency, accuracy, and modification of designs. CAD systems also allow the design to be created three-dimensionally. Before CAD systems were created, every design had to be drawn by hand.

A **three-dimensional (3D) model** is a digital representation of an object that allows the part to be viewed from multiple perspectives (see Figure 1-2). Having a 3D model of the design offers several advantages in creating and analyzing the design.

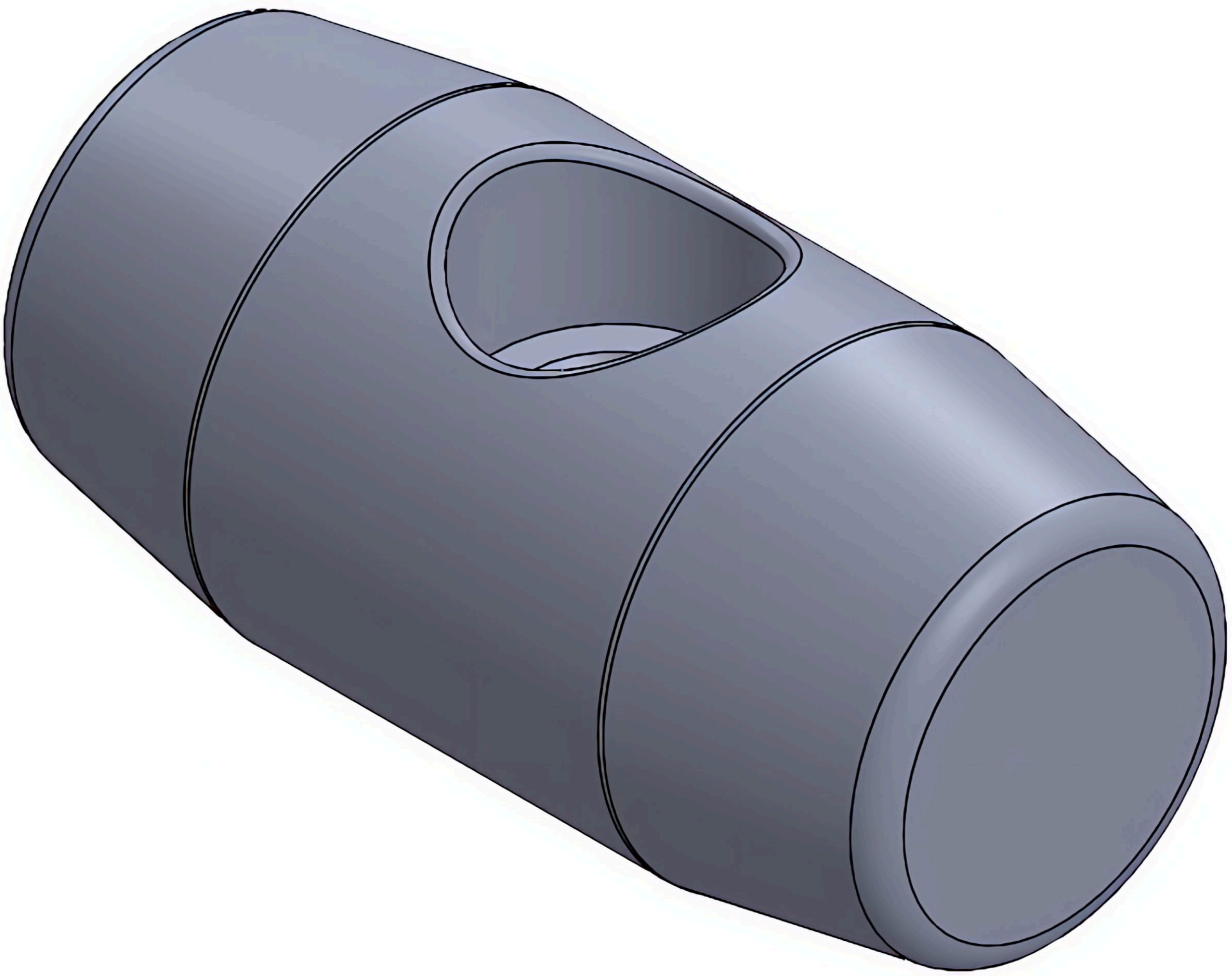
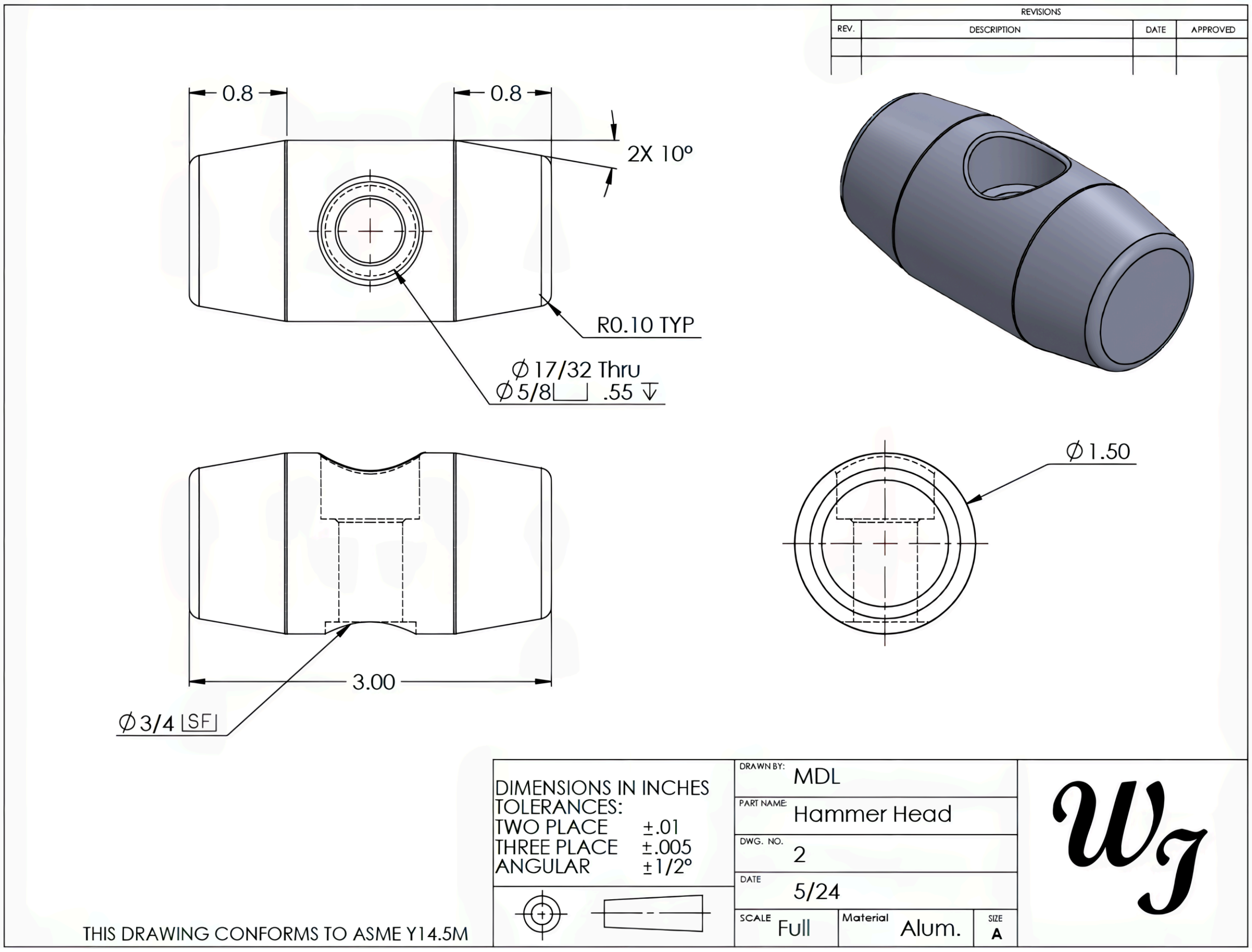


Figure 1-2. 3D model image of a hammerhead component.

A CAD system can quickly generate a detailed multi-view drawing from a 3D model, automatically transferring the model's details and dimensions to a two-dimensional (2D) drawing. See Figure 1-3. Any changes made to the 3D model can be automatically updated on the 2D drawing file.

- **Analyzing and Simulating.** A 3D model may be used with different software to create a calculated simulation of how the part or assembly will perform. Examples include simulating the manufacturing process or stress-testing a product when it is in service. These simulations allow for design adjustments prior to the production of parts, which saves time and material costs. Some examples would be a part's material may be used to predict how the part will react to stresses, or a 3D model of a part's mold can be used to simulate how the material will flow into the mold. Allowing design adjustments to be made before the parts are produced also saves time and material costs.

- **Verifying Assembly of Components.** A 3D assembly may be created from the individual component models to represent the final assembly, as shown in Figure 1-4. This can be used to verify how the final assembly will perform before an actual part is produced. This assembly can then be used to create 2D drawings of the final assembly.
- **Transferring the Model Directly to Production.** A 3D model can be used in software to create programs to be used by **3D printers** or by **Computer Numerical Control (CNC) machines** to produce a physical part. 3D printing, also referred to as additive manufacturing, is the building up of a three-dimensional object from a digital 3D model. This can be done in various processes where material (often plastic or a powdered metal) is deposited or joined together, typically layer by layer. A CNC machine is a computer-controlled device used to automate the operation of machine tools like lathes, mills, grinders, and routers. These machines follow programmed instructions to execute precise machining tasks, shaping materials such as metal, plastic, wood, and composites into desired forms.



DIMENSIONS IN INCHES
TOLERANCES:
TWO PLACE ±.01
THREE PLACE ±.005
ANGULAR ±1/2°

DRAWN BY: MDL
PART NAME: Hammer Head
DWG. NO. 2
DATE 5/24
SCALE Full
Material Alum.
SIZE A

Figure 1-3. 2D print created from the 3D model.

Figure 1-4. Assembly of 3D components.

1.5 DRAWINGS

The creation of detail and assembly drawings represents a crucial stage in the design process. **Detail drawings** are two-dimensional drawings that provide comprehensive information necessary to manufacture each part. These drawings not only provide dimensions that outline the shape and size of the part but also detail other important features. Additionally, drawings convey essential specifications such as material types, allowable size variations, and feature placements, ensuring all aspects are communicated clearly for production.

To help ensure part specifications are accurately conveyed through detailed drawings, companies adhere to drawing and manufacturing standards. These norms have been thoughtfully established by respected organizations such as the American Society of Mechanical Engineers

(ASME), the American National Standards Institute (ANSI), and the International Organization for Standardization (ISO).

It is imperative to have a comprehensive understanding of detail drawings to effectively interpret assembly drawings. Assembly drawings identify each component within a product and illustrate the manner in which these parts interconnect. Both detail and assembly drawings are created from the 3D models in CAD software.

1.6 PRINTS

The term “print” is often used to describe a physical copy of a part drawing, although in many industry settings, the words “print” and “drawing” are frequently used interchangeably. Understanding or interpreting the information presented in these drawings is known as **print reading**. Most physical paper prints are larger than a standard 8.5” x 11” sheet of paper and must be created using a plotter. However, many companies today are choosing to skip paper prints altogether, having them viewed by the reader electronically on larger computer screens, tablets, or laptops. This can save time not having to move paper copies through several departments or large production floors, as well as ensuring that the latest version of the print is being used by the reader. However the drawing is being viewed, print reading is an essential skill for anyone in the manufacturing field.

1.7 BLUEPRINT

The term “blueprint” has its roots in a time when these prints featured a blue background with white lines and symbols. Old blueprints were created by first sketching the design on tracing paper and inking the lines, as shown in Figure 1-5. The inked tracing paper was then placed on light-sensitive paper and exposed to ultraviolet light. This caused a chemical reaction, turning the exposed areas blue. This process required skilled draftsmen, resulting in the characteristic white lines on a blue background in blueprints.

Even today, many people continue to use the word "blueprint" when talking about drawings. These types of drawings were phased out when the printer, plotters, and copy machines became popular.

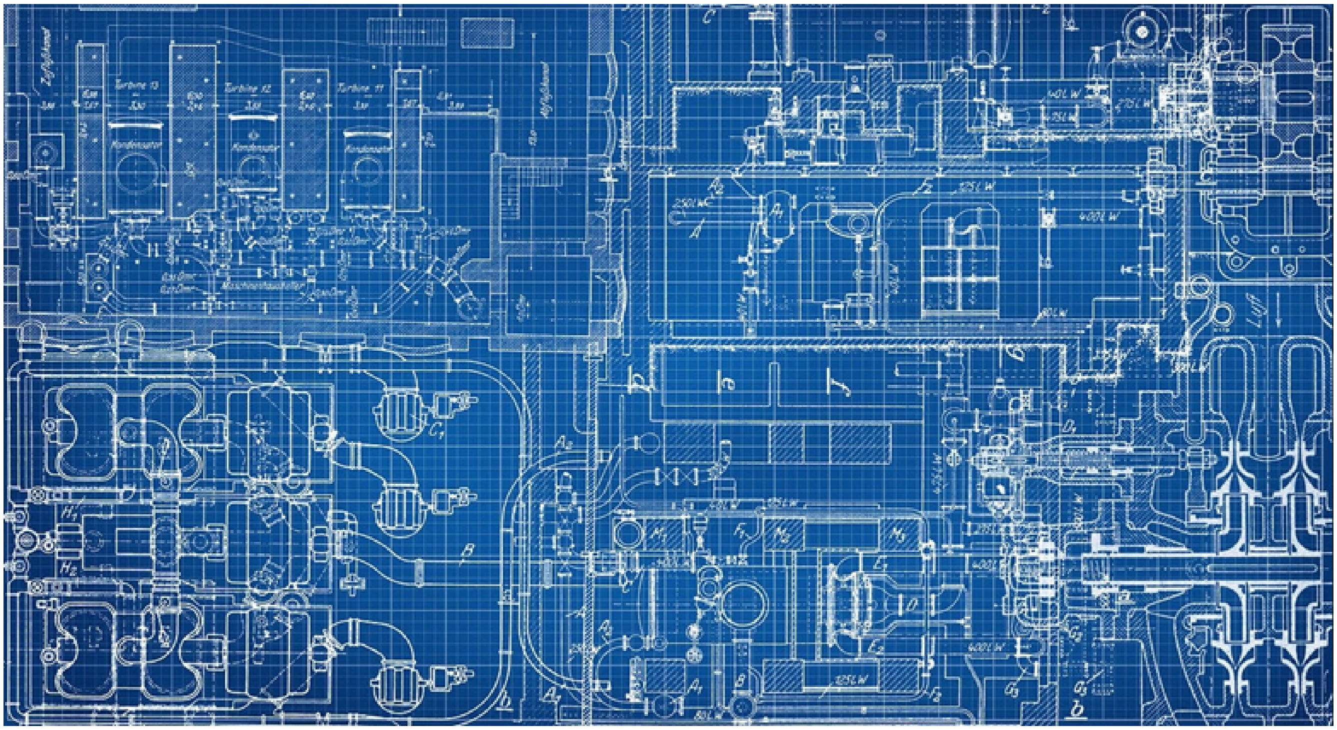


Figure 1-5. A traditional "blueprint" with white lines over a blue background.

LEARNING ACTIVITIES

Exercise 1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=5#h5p-2

References:

Barsamian, M. A., & Gizelbach, R. A. (2015). Unit 1 drawings and prints. In *Machine trades print reading* (7th ed.). Goodheart-Willcox.

Lorier, M. (2024). Simplified AI. [Large language model]. https://simplified.com/ai-writer

Images:

All images by Mark Lorier and licensed under CC BY-NC 4.0 except where otherwise noted.

Figure 1.1: "Drawing_of_a_connecting_rod_in_an_automobile_engine" by Abouttools is licensed under CC BY-SA 4.0

2. Views of an Object

2.1 INTRODUCTION

Learning Objectives

- Identify the primary views of an object
- Select the placement of views on a print
- Differentiate between pictorial and orthographic projection drawings
- Demonstrate the use of projection lines
- Determine appropriate dimensions of length, width, and height related to views
- Differentiate between first- and third-angle projection on prints
- Visualize the basic shapes of objects

Terms

- Multi-view drawing
- Isometric view
- Isometric drawing
- Pictorial drawing
- Orthographic projection
- Oblique drawings
- Perspective drawings

- Principal view
- Height
- Width
- Depth
- Third-angle projection
- First-angle projection

Multi-view drawings are a vital tool in print reading, enabling effective communication and understanding of complex objects. By presenting multiple views, such as the front, top, side, and isometric views, multi-view drawings provide a three-dimensional visualization of an object. They are a universal language in design and manufacturing, ensuring a shared understanding of design intent and specifications. Multi-view drawings also facilitate accurate measurements, precise manufacturing, assembly, quality control, and troubleshooting. Mastering the interpretation of multi-view drawings enhances print readers' understanding and contributes to successful projects in various industries. In this chapter, we will explore the significance of multi-view drawings in print reading, along with their aid in understanding and interpreting complex objects.

2.2 MULTI-VIEW DRAWINGS

Multi-view drawings are an essential component of print reading. They provide print readers with a comprehensive visual representation of an object from different perspectives, aiding in effective communication, accurate manufacturing, assembly, quality control, and maintenance processes. By mastering the interpretation of multi-view drawings, print readers can enhance their understanding of complex objects and contribute to successful projects in various industries. Figure 2-1 is an example of a multi-view drawing displaying the object as it would be viewed from the front, top, and side.

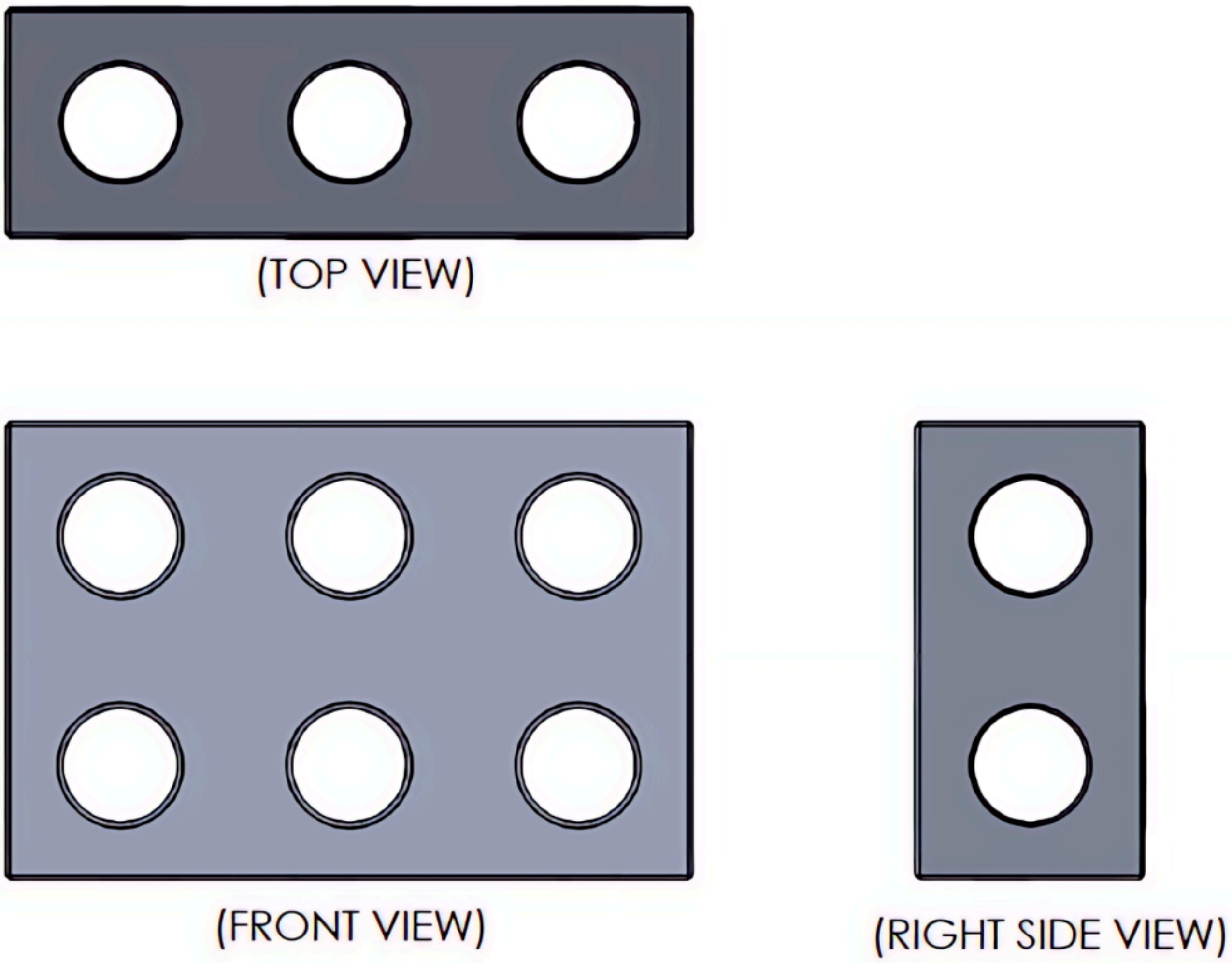


Figure 2-1. A multi-view example displaying three views of an object.

Visualization

Multi-view drawings provide print readers with a three-dimensional visualization of an object. By presenting multiple views, such as the front, top, and side views, print readers can grasp the shape, size, and proportions of the object accurately. An **isometric view** is a two-dimensional depiction of a three-dimensional object that presents the object's dimensions and angles from a particular perspective, thereby providing a sense of depth in its representation, as illustrated in Figure

2-2. These three-dimensional views will aid the reader in better understanding the part's shape and features. This visual representation allows for a better understanding of the object's design and aids in effective communication among designers, engineers, and manufacturers.

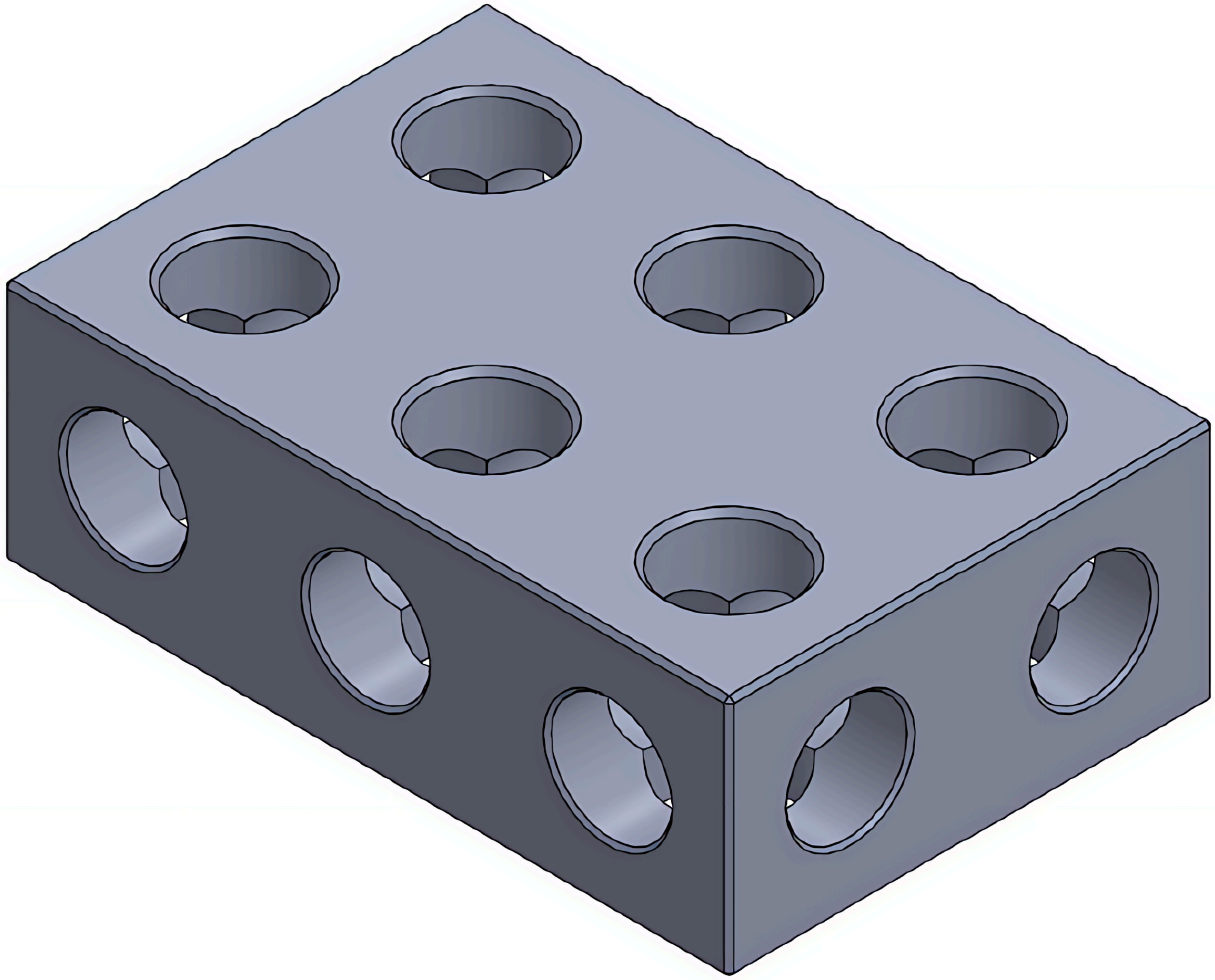


Figure 2-2. Isometric perspective offering a feeling of depth in the depiction of an object.

Design Communication

Multi-view drawings serve as a universal language in the design and manufacturing industries. They act as a bridge between different stakeholders, ensuring that everyone involved in the project understands the design intent and specifications. By providing a clear and precise representation of the object, multi-view drawings facilitate effective communication and minimize misunderstandings.

Dimensioning and Tolerancing

Accurate measurements and tolerances are critical in the manufacturing process. Multi-view drawings include dimensions and tolerances that specify the exact measurements and allowable variations or tolerance for each feature of the object. See Figure 2-3. Print readers can interpret these dimensions to ensure precise manufacturing and assembly processes, resulting in high-quality products.

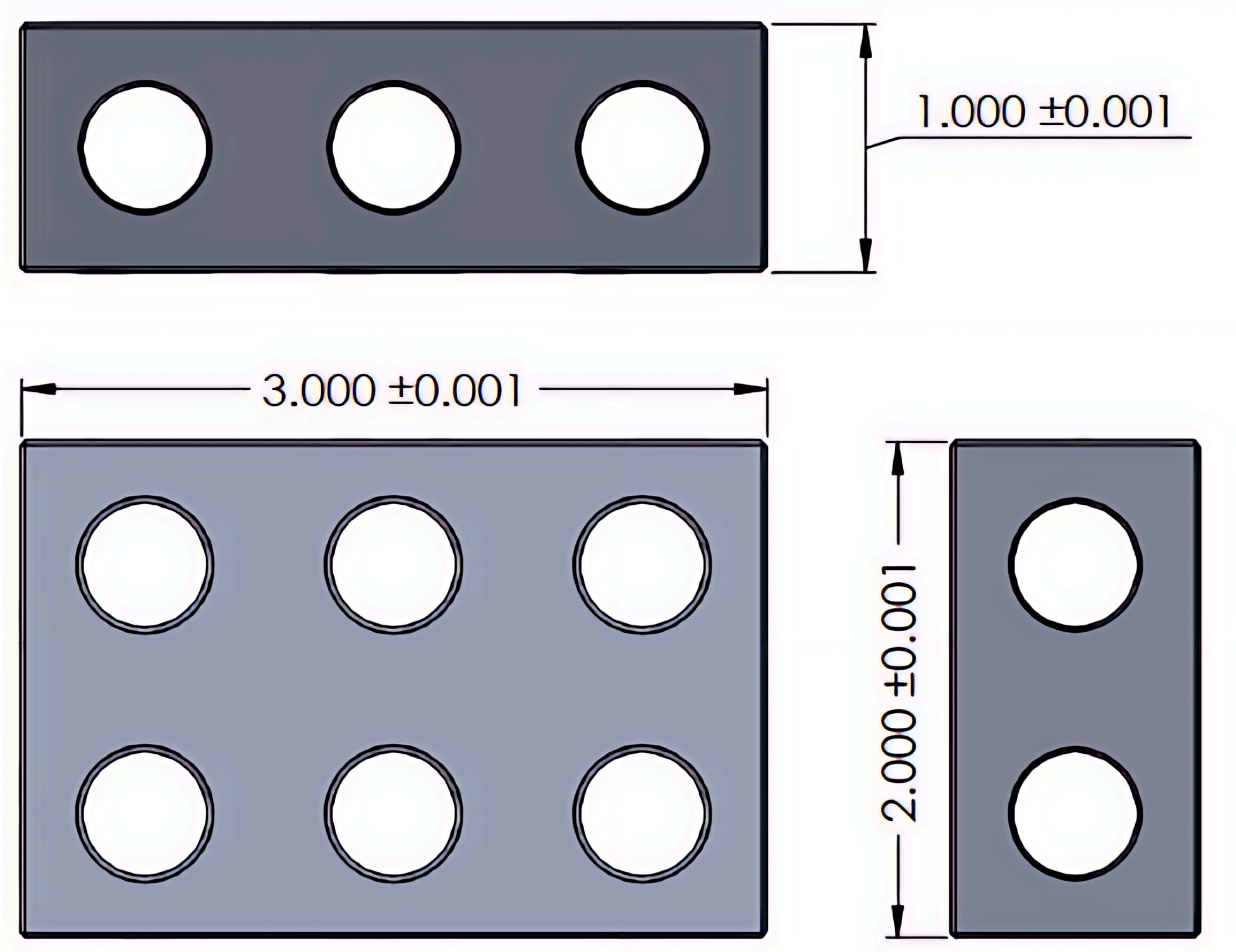
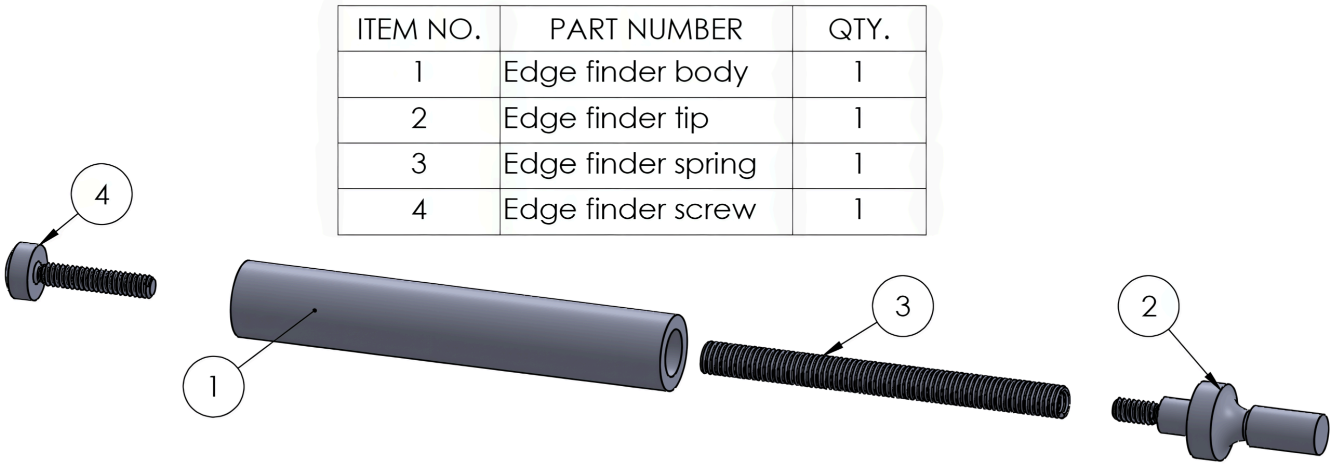


Figure 2-3. Dimensions identify part and feature sizes and allowable size variations.

Assembly and Disassembly

Understanding how different parts of an object fit together is essential for efficient assembly and disassembly processes. Multi-view drawings provide print readers with a comprehensive view of the object, allowing them to identify the relationships between components, their orientations, and the sequence of assembly or disassembly. This knowledge ensures smooth and error-free construction or maintenance procedures. In Figure 2-4, the components of an assembly are aligned in relation to how they are to assemble together, along with a list of the parts included in the drawing.



ITEM NO.	PART NUMBER	QTY.
1	Edge finder body	1
2	Edge finder tip	1
3	Edge finder spring	1
4	Edge finder screw	1

Figure 2-4. Assembly drawing with a list of included parts.

Quality Control

Multi-view drawings serve as a reference for quality control inspections. Print readers compare the actual object or part against the drawing to verify if it meets the specified dimensions, tolerances, and other requirements. This ensures that the final product aligns with the design intent and meets the desired quality standards. Oftentimes, the drawing is considered a contract between a customer and the manufacturer. Ensuring the part passes inspection is a crucial step in maintaining relationships between customers and the manufacturer.

Troubleshooting and Maintenance

Multi-view drawings are invaluable resources for troubleshooting and maintenance purposes. They provide a detailed representation of the object, enabling print readers to identify and understand the internal components, connections, and potential issues. This knowledge aids in efficient problem-solving and maintenance procedures.

Component drawings provide visual depictions of individual parts, as well as their assemblies, aiding technicians in understanding how components integrate and function within a bigger system. These diagrams guarantee accurate identification of components, detailing their dimensions, materials, and specifications—critical factors for ordering replacements that match correctly. In the process of diagnosing problems, component diagrams assist in locating the specific site and nature of issues by enabling technicians to compare the diagram with the actual part and spot any differences. Furthermore, they frequently include essential maintenance details like lubrication points, wear thresholds, and assembly guidelines to ensure that maintenance activities are performed properly. These drawings also serve as invaluable training resources for new technicians by providing a visual reference to the equipment. Ultimately, by offering a clear outline for repairs and upkeep, component drawings help minimize downtime and enhance safety by ensuring all work adheres to established specifications.

2.3 PICTORIAL DRAWINGS VS. ORTHOGRAPHIC PROJECTION DRAWINGS

Pictorial drawings aim to provide a visually realistic and intuitive representation of an object, emphasizing its appearance and overall shape, as shown in Figure 2-5.

An **orthographic projection** prioritizes accuracy and technical information, providing a detailed and precise representation of an object's dimensions and features. See Figure 2-6.

Pictorial drawings and orthographic projection drawings are two different approaches to representing three-dimensional objects on a two-dimensional surface. While both methods serve the purpose of visual communication, they differ in terms of the level of realism and the information they convey.

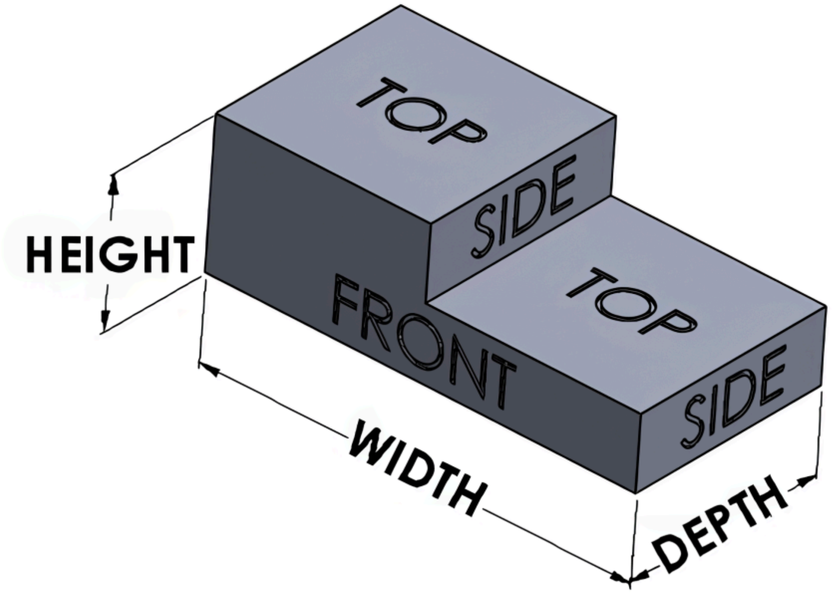


Figure 2-5. Pictorial drawing.

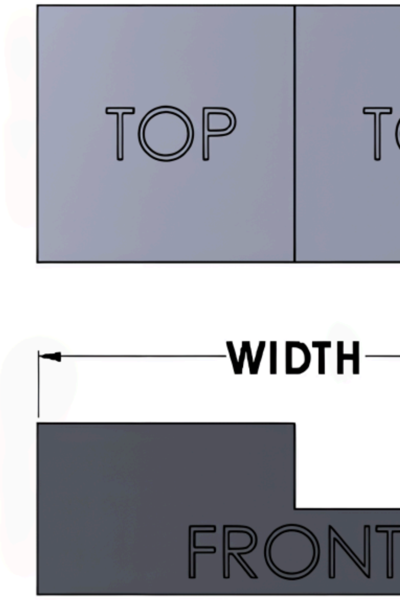


Figure 2-6. Orthographic proj

The views of the orthographic projection drawing will maintain alignment with each other. The front and top views will align vertically, and the front and right views will align horizontally. A 45° projection line can be used as an intersection point to connect projection lines between the top and right-side views, as shown in Figure 2-7.

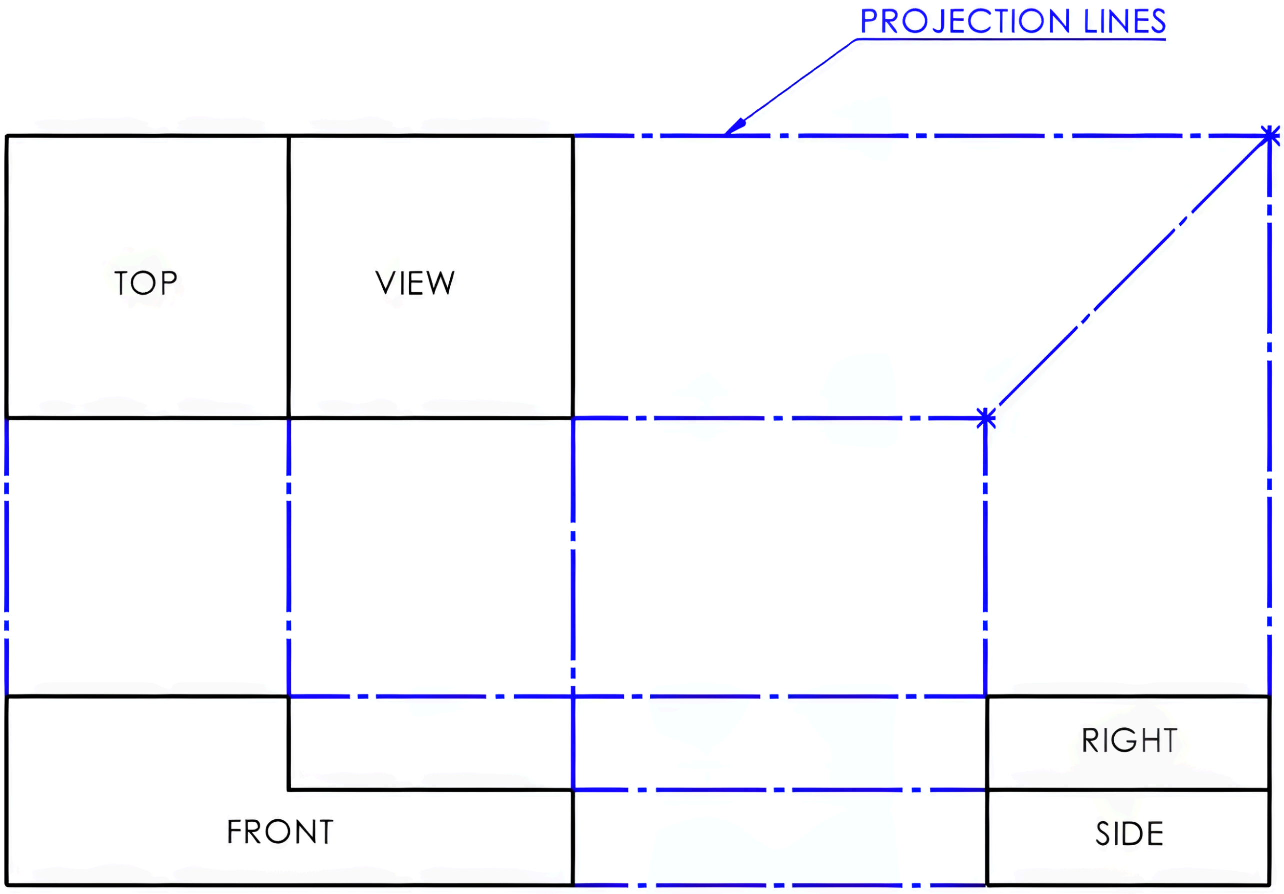


Figure 2-7. Blue projection lines illustrating the alignment relationship among the three views.

2.4 PICTORIAL DRAWINGS

Pictorial drawings, also known as pictorial views or pictorials, are drawings that aim to represent an object in a visually realistic and intuitive manner. These drawings provide a more artistic and subjective representation of the object, emphasizing its appearance and overall shape. Pictorial drawings are often used to convey the visual impact and aesthetic qualities of an object.

There are different types of pictorial drawings, including isometric, oblique, and perspective drawings. In **isometric drawings**, the object is represented by parallel lines and equal 30° angles, giving the illusion of depth and three-dimensionality. **Oblique drawings**, illustrated in Figure 2-8, show the object with one face parallel to the picture plane while the other faces are at an angle. **Perspective drawings** create the illusion of depth and distance by using vanishing points and converging lines, as shown in Figure 2-9.

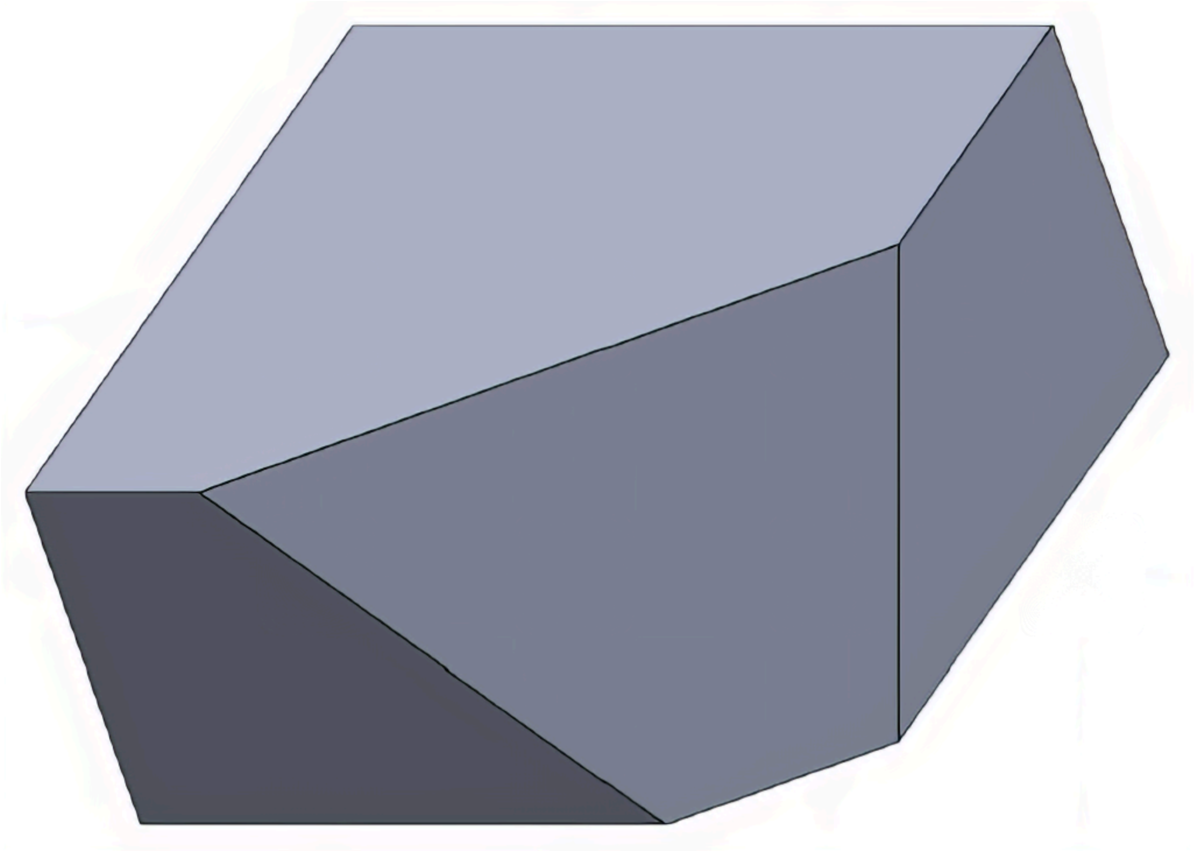


Figure 2-8. Oblique drawing.

Figure 2-9. Perspe

Pictorial drawings are useful for presenting an overall impression of an object, showcasing its design and providing a visual representation that is easily understood by non-technical individuals. However, they may not provide precise measurements or detailed information about the object's dimensions and features.

2.5 ORTHOGRAPHIC PROJECTION DRAWINGS

Orthographic projection drawings, often known as orthographic views or multi-view drawings, are a reliable and standardized method for depicting objects. This technique systematically captures an object by projecting it onto planes that are positioned at right angles (90°) to one another.

These detailed 2D representations often feature front, top, and side views of the object. Each view is drawn to a matching scale and aligned with others, ensuring precise measurements and dimensioning. By doing this, these drawings offer a clear representation of the object's shape, size, and the relationships between its various components.

To create such a drawing, an object is positioned so that each face aligns with one of up to six view planes. Once in place, projections are made onto each plane—resulting in separate front, top, and side views (and more if needed depending on complexity). Some parts, often round parts, may be fully represented in as little as two views (see Figure 2-10). Others may require all six views to fully represent the necessary features as shown in Figure 2-11. Each view shows how the object appears from specific perspectives: head-on from the front view; directly from above for the top view; and straight from either side for side views. The drawing will include the number of views to provide enough detail and information required to manufacture the part and prevent confusion.

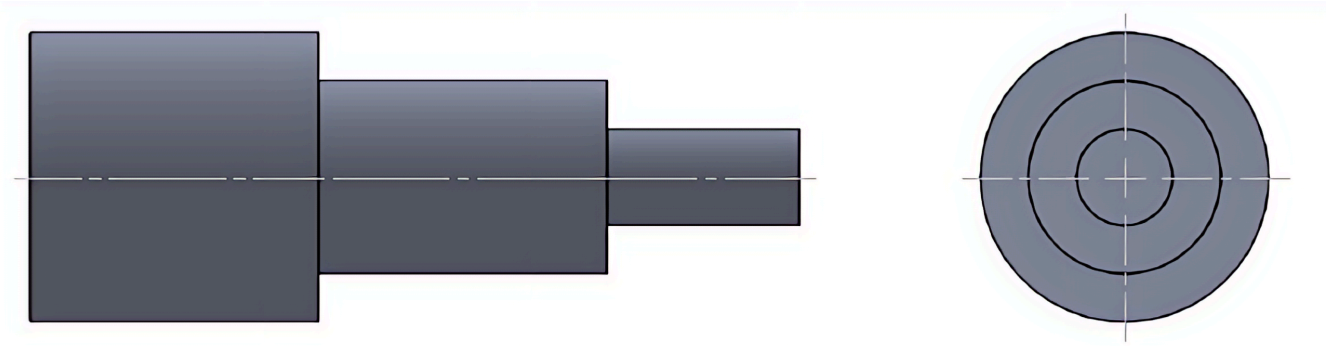


Figure 2-10. Two-view drawing used to fully identify a part.

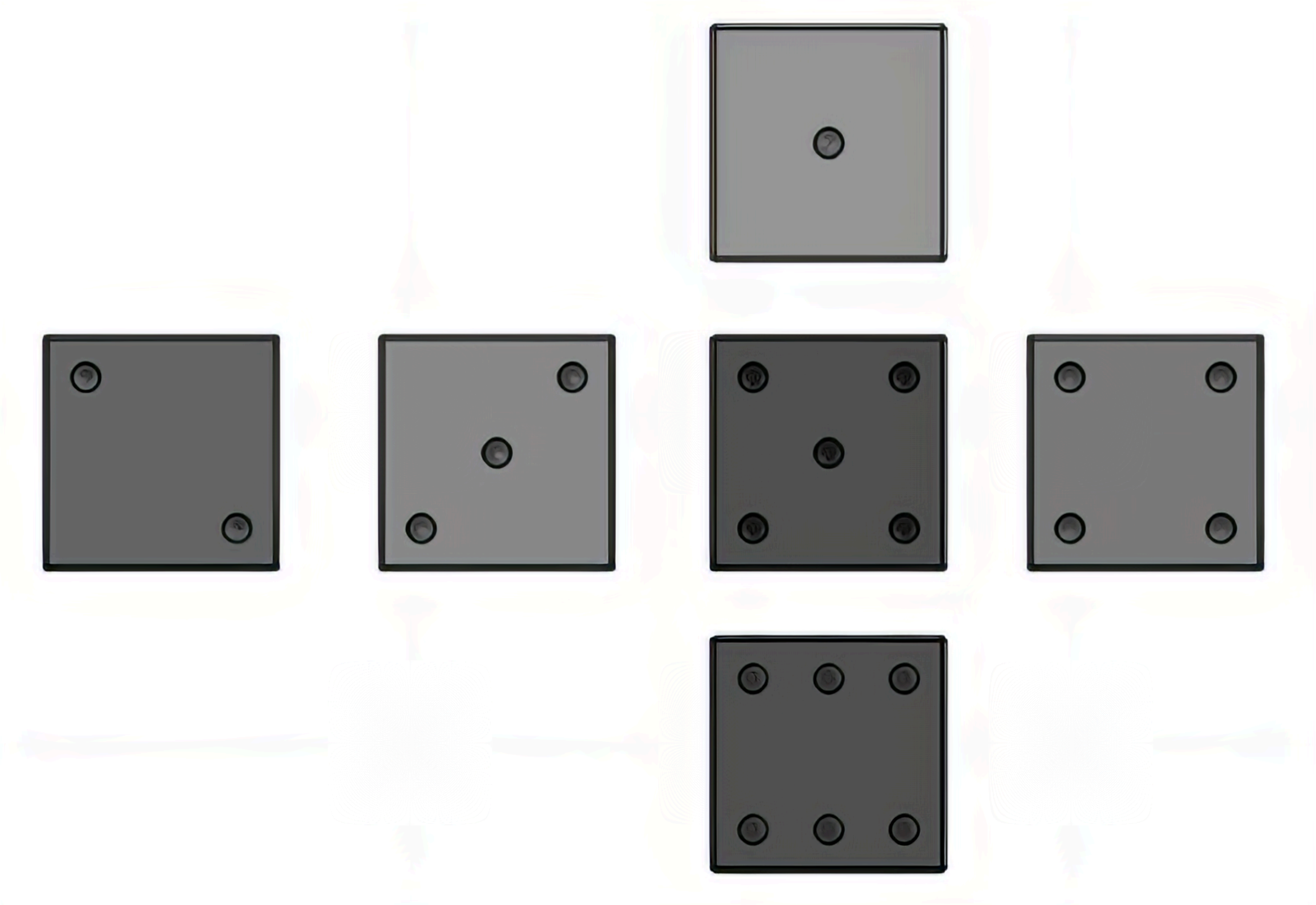


Figure 2-11. Six views used to identify all part features.

Orthographic projection drawings are invaluable for capturing every nuance of an object's dimensions and interrelationships while ensuring that all measurements remain accurate throughout the design process.

Summary

Unlike pictorial drawings, orthographic projection drawings prioritize accuracy and technical information over visual aesthetics. They are commonly used in engineering, architecture, and manufacturing industries for design, communication, manufacturing, assembly, and quality control purposes.

The main difference between pictorial drawings and orthographic projection drawings lies in their purpose and level of realism. Pictorial drawings aim to provide a visually realistic and intuitive representation of an object, emphasizing its appearance and overall shape. Orthographic projection drawings prioritize accuracy and technical information, providing a detailed and precise representation of an object's dimensions and features.

2.6 ORTHOGRAPHIC VIEW PLACEMENT

The relationship between the views can be compared to unfolding a cardboard box. Figure 2-12 illustrates a cardboard box as it is folded into the box shape with the top, front, and right sides identified with their size values.

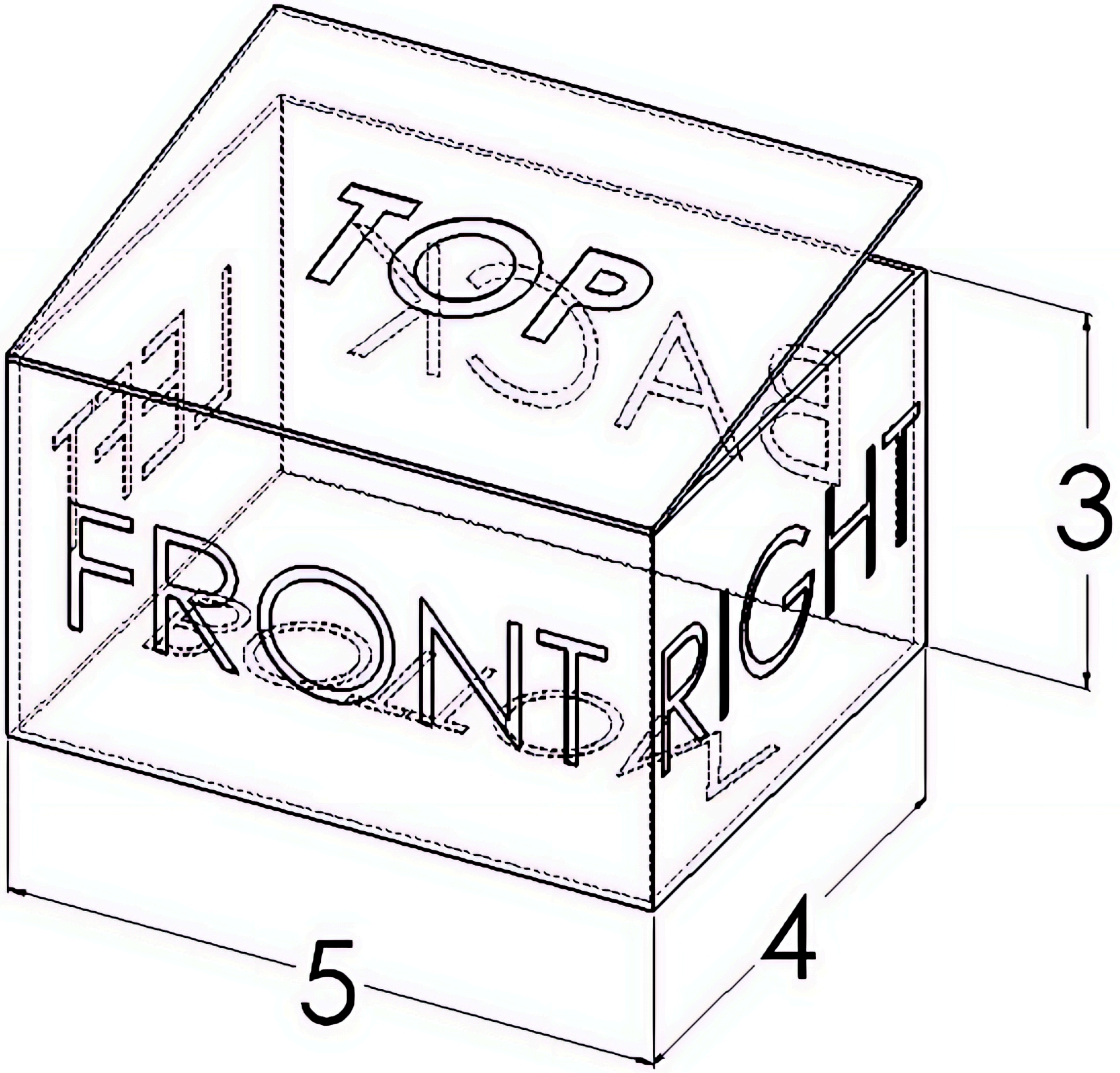


Figure 2-12. Cardboard box used to identify the relationship of the orthographic projection views.

As you unfold the box, you'll notice that the views are connected to each other, as shown Figure 2-13. When laid flat, the front surface serves as a central point: the top view is positioned above it, the right view goes on its right side, the bottom view beneath it, and the left view to its left. The back view unfolds further and connects directly to the left view.

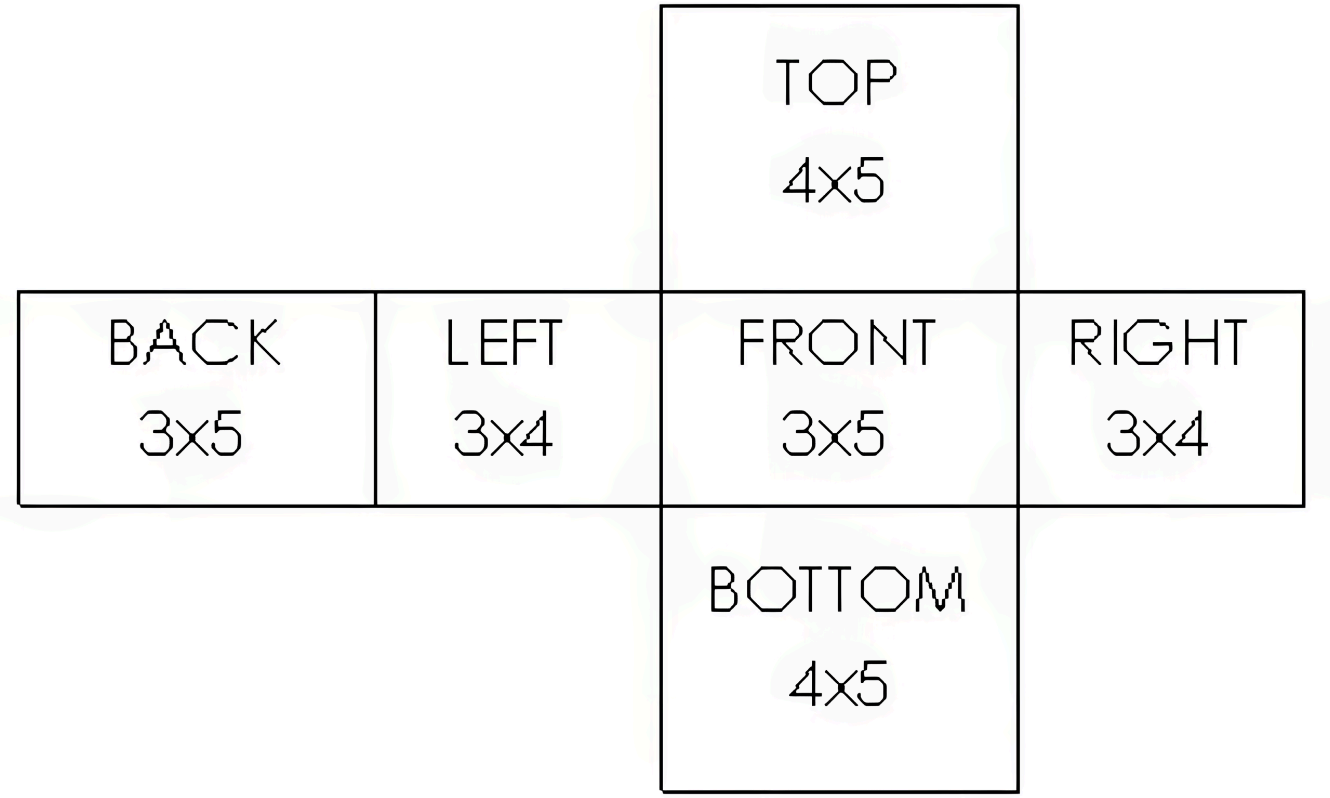


Figure 2-13. This figure illustrates the box from Figure 2-12 in its unfolded state. The labeled sides will correspond to the names of each orthographic view they depict.

In Figure 2-14, you will see that the views are separated but maintain an identical orientation as the unfolded box. These six views are all known as the six **principal views**. Most part drawings will not use all six views; only the views necessary to communicate the details of the part effectively will be used. A three-view drawing is often sufficient to fully describe a part with the front, top, and right-side views most commonly used. A two-view drawing is typically sufficient for a cylindrical component to communicate the shape and necessary details. Views other than the six principal views are known as auxiliary views. These additional views are discussed in Chapter 12.

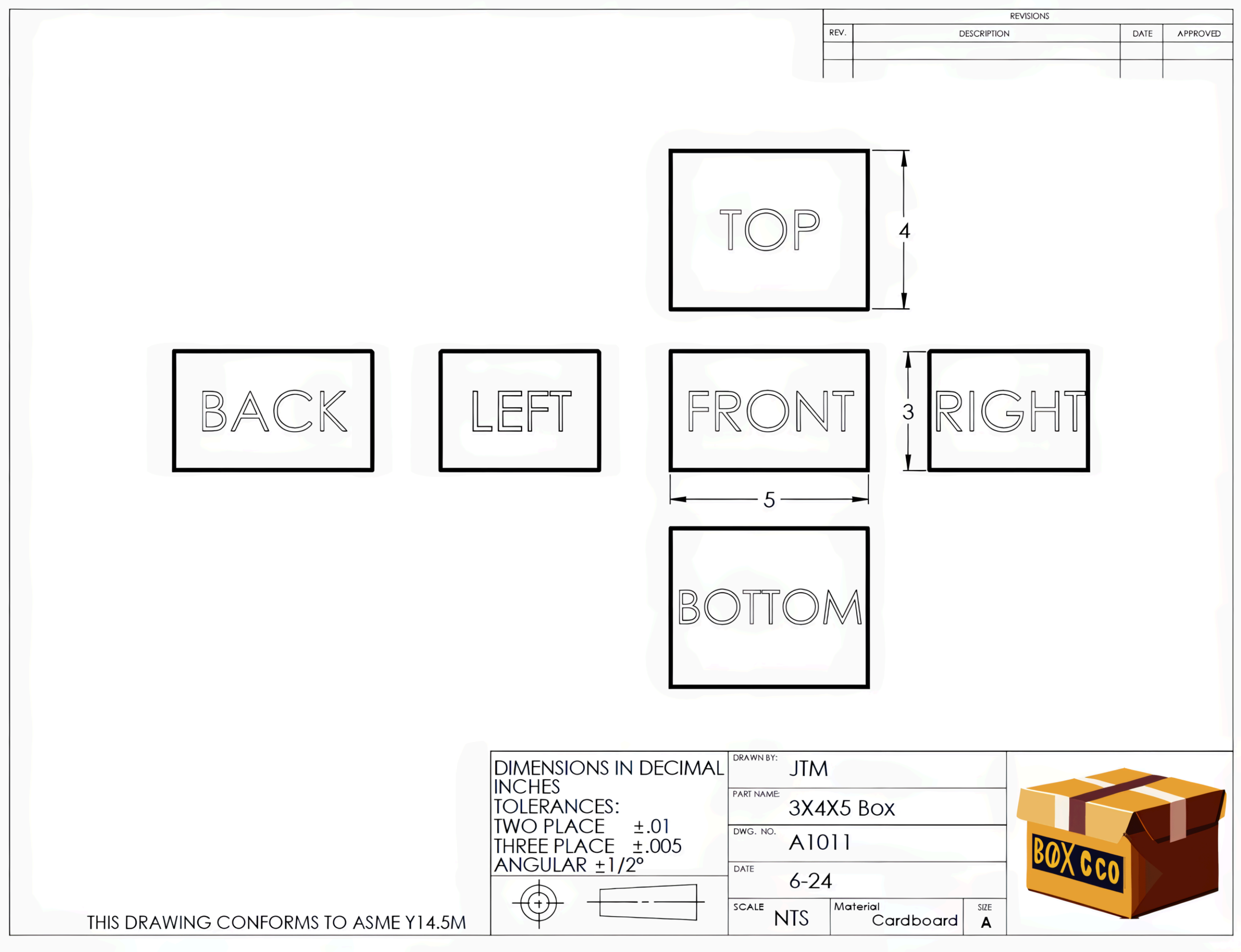


Figure 2-14. The six principal view placements as they would appear on a part drawing.

To apply this orthographic projection to a part drawing, the part can be viewed as if it is placed inside a box with the part views projected onto each surface. Figure 2-15 displays a part placed inside a clear box. Unfolding or rotating this box using the orthographic projection view placement will produce the six principal views of the part shown in Figure 2-16.

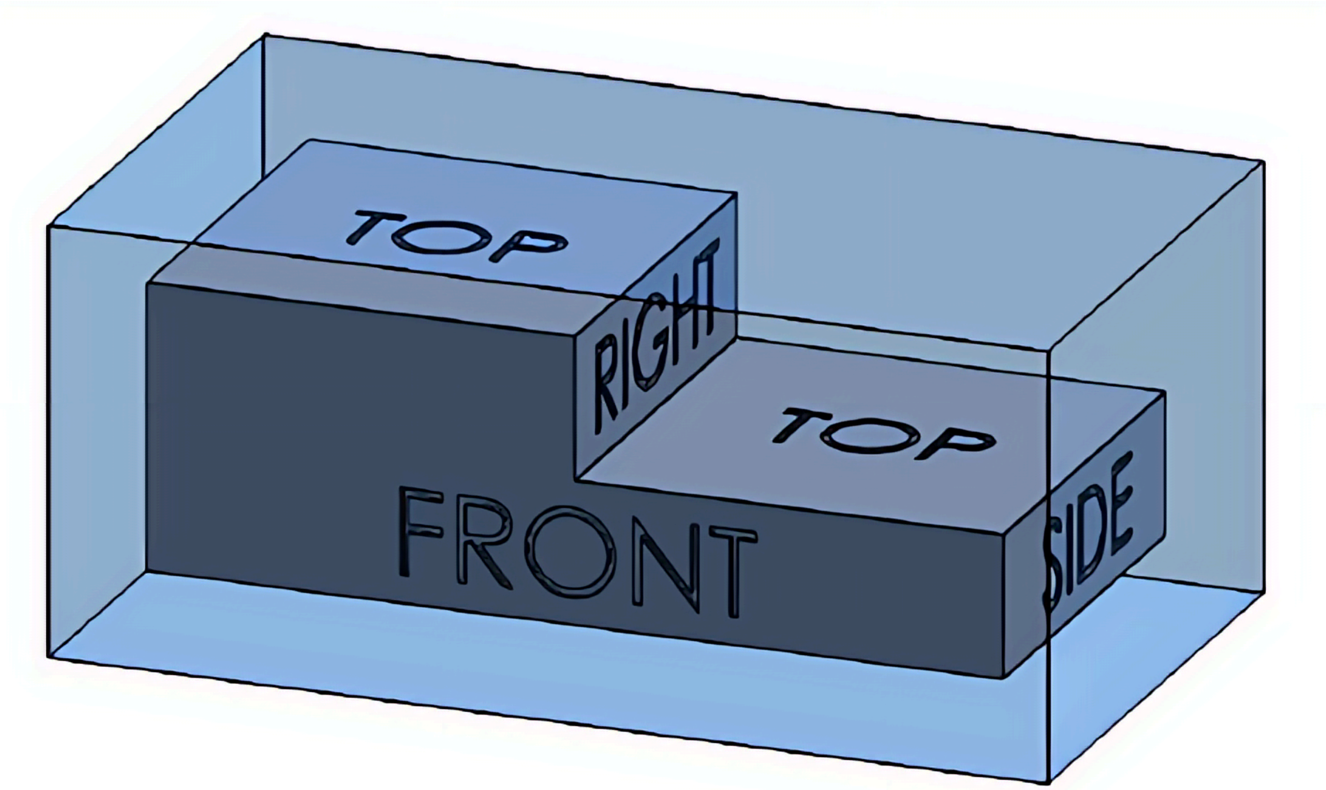


Figure 2-15. Part placed inside a clear box with views projected onto the six surfaces.

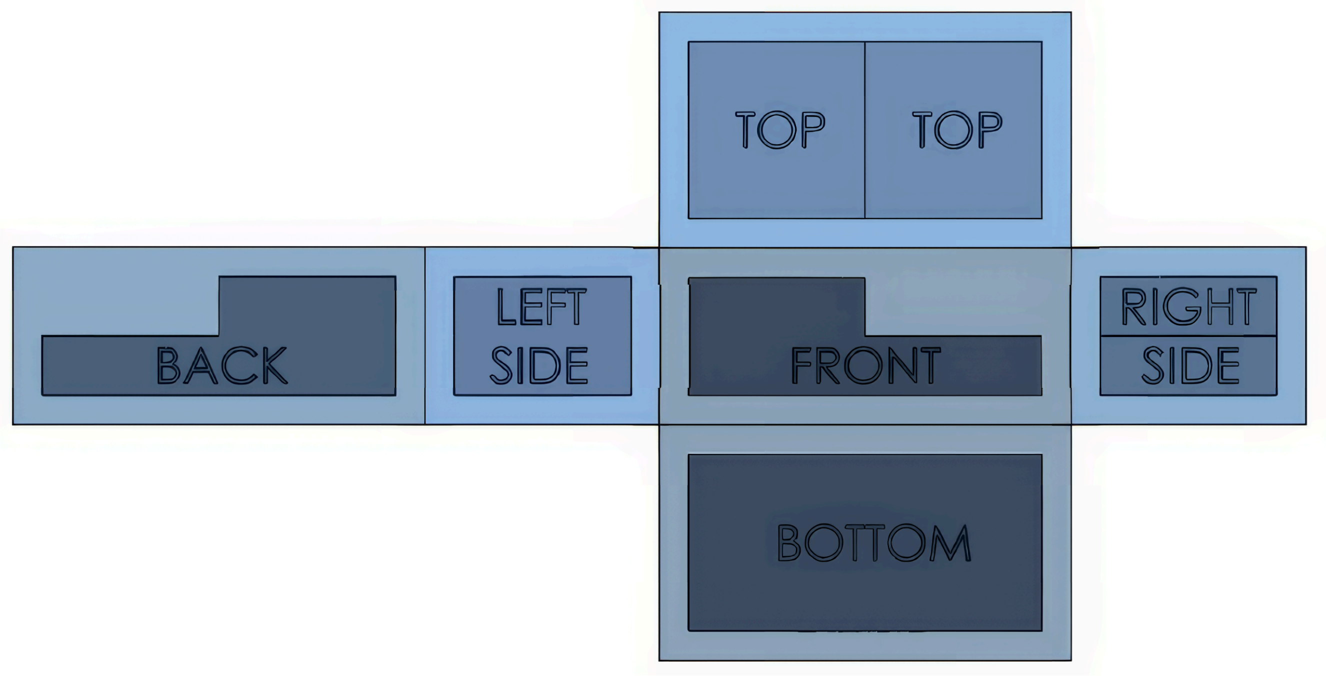


Figure 2-16. Six principal view placement of Figure 2-15 as they would appear on a part drawing.

2.7 NAMES OF THE VIEWS

The names of the views on an orthographic print are not as important as the relationship between each view. The front view will typically provide the best representation or profile of the part being drawn. An example of this would be if a car were to be drawn in orthographic projection. The side view of the car would likely be used as the front view on the print because it would show a more distinguishing profile than what the front of the car would show.

Observe the difference in how the views are arranged for the block in Figures 2-17 and 2-18. Each drawing provides a clear representation of the identical block. The relationship between the views is more important than the name of the view.

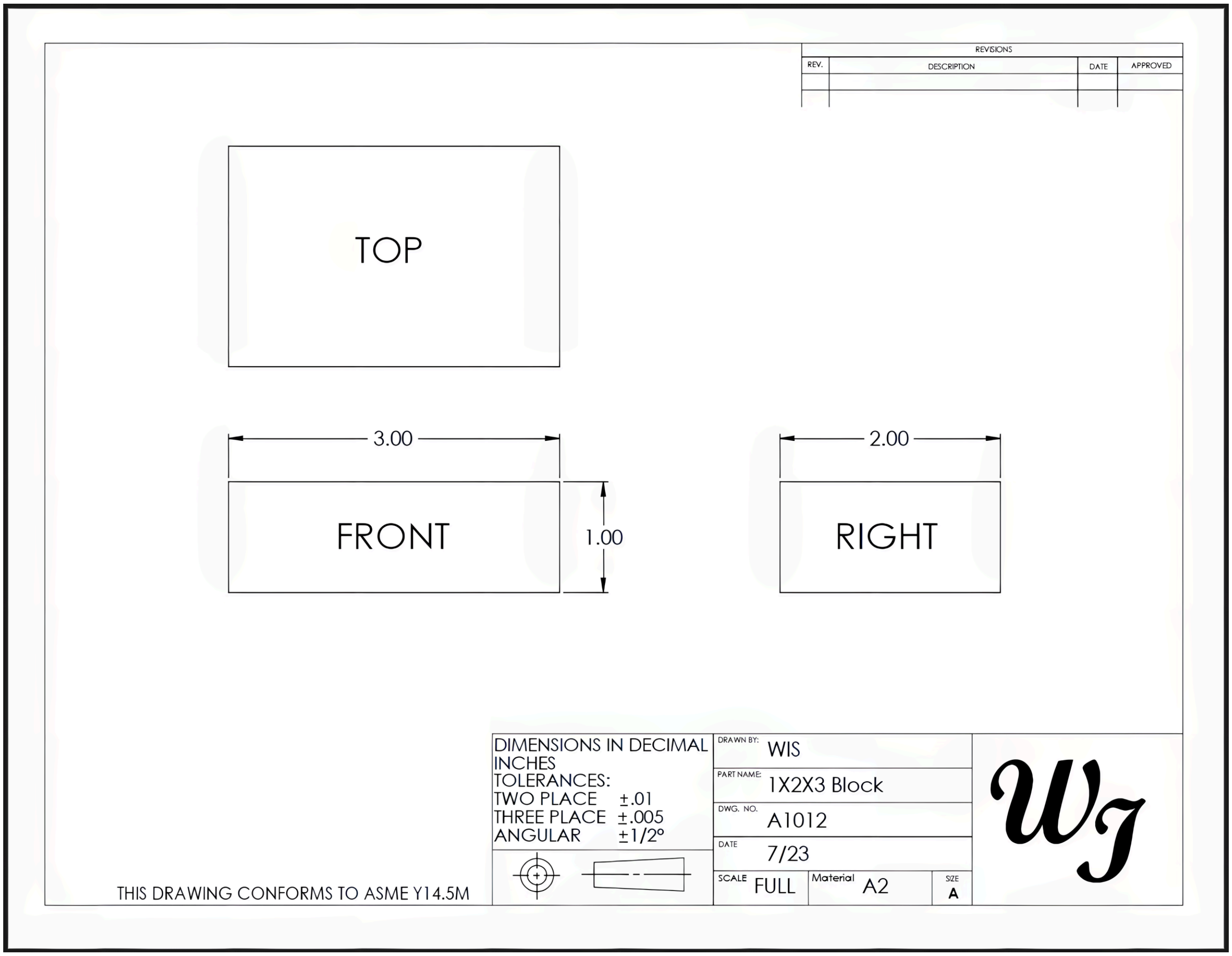


Figure 2-17. Rectangular block in a three-view drawing as compared to Figure 2-18 view orientation.

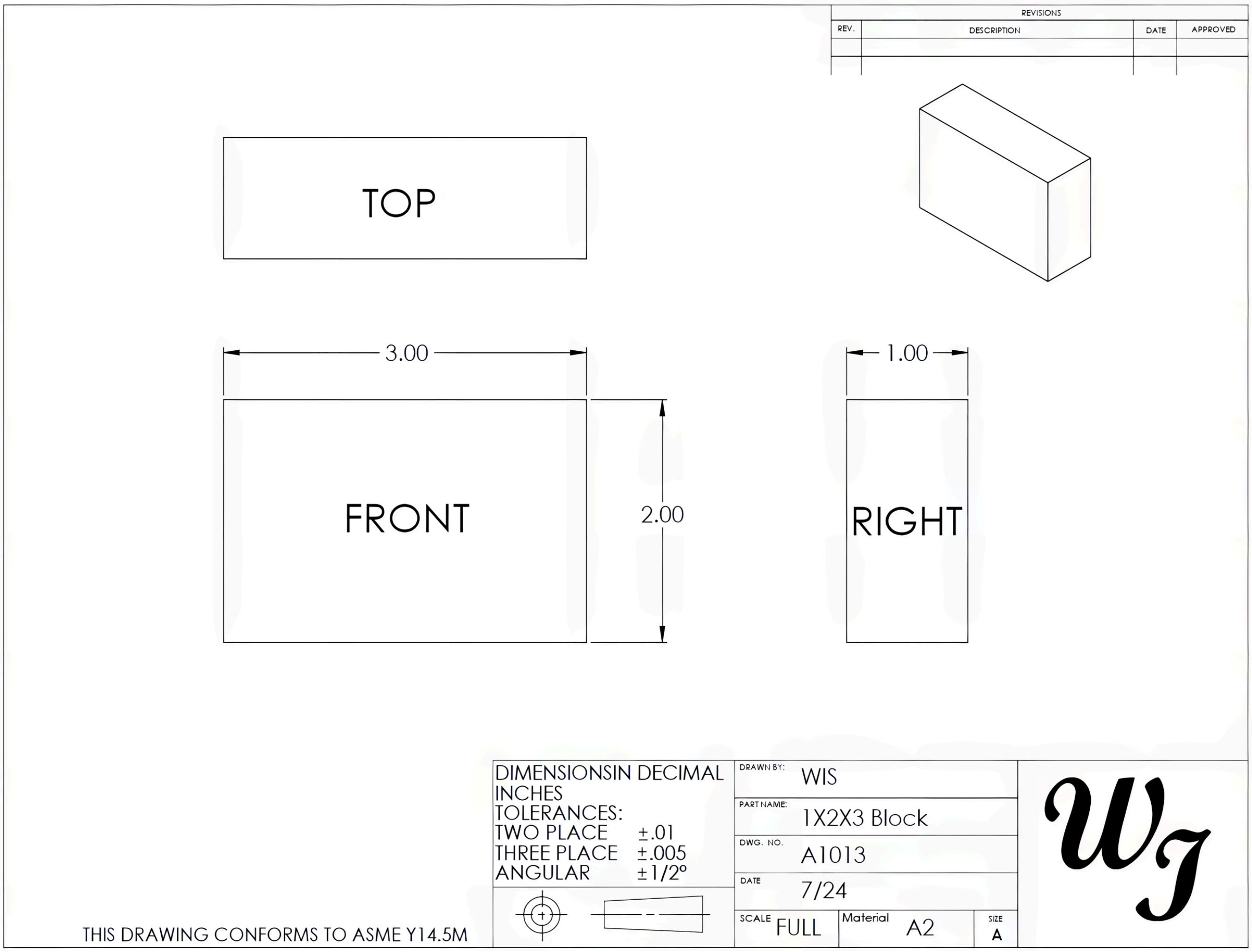


Figure 2-18. Rectangular block in a three-view drawing as compared to Figure 2-17 view orientation.

2.8 HEIGHT, WIDTH, AND DEPTH

The common terminology used when referring to an object's overall size is height, width, and depth. It is important to use these standard terms correctly when referring to a part's size (see Figure 2-19). Because the three-dimensional parts are projected to a two-dimensional view, only two of these three values may be observed from any one view; the third value would be found in one of the perpendicular views. An object may also be described using the terms "length" and "thickness." When these terms are used, the thickness will refer to the smallest outside value, and the length will be the longest value.

The **height** of the part is the distance between the top and bottom of the part. Because the height cannot be observed in the top view, the height measurement will only appear in the front, back, or side views.

Width, also called length, is the distance between the left and right sides of the part and can only be visible in the front, back, top, or bottom views. The width measurement cannot be observed in the side views.

The **depth**, sometimes referred to as the thickness, is from the front of the part to the back of the part. The depth measurement can only be seen in the top, bottom, or side views.

See Figure 2-19 below for an illustration of height, width, and depth.

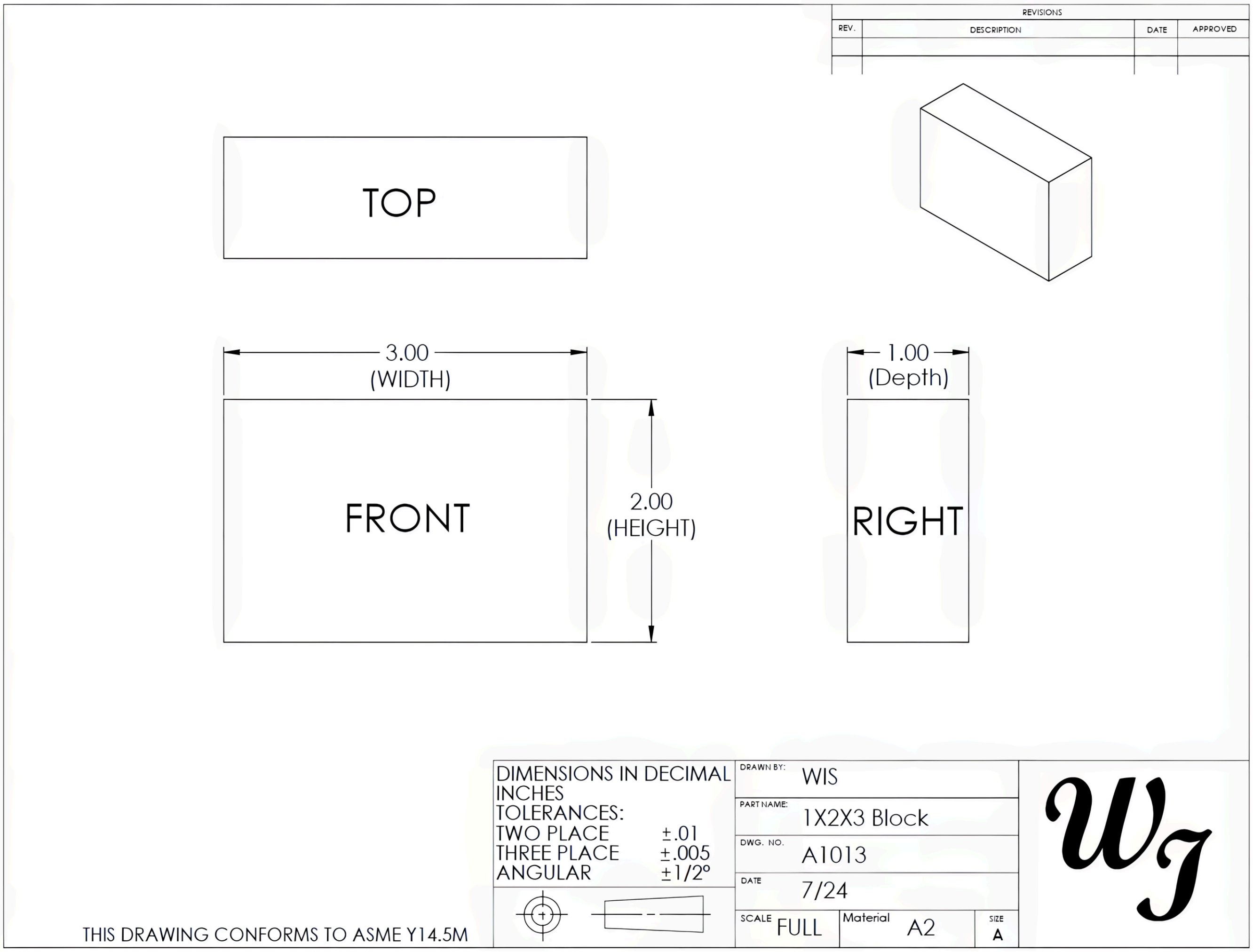


Figure 2-19. Part drawing identifying the placement of the height, width, and depth values.

View the supplementary YouTube video for further explanations and visual examples on how to distinguish between height, width, and depth measurements: Height, Length/Width, and Depth dimensions on blueprints

2.9 PROJECTION VIEWS

When vertical and horizontal viewing planes intersect each other, four separate quadrants are created: first, second, third, and fourth. See Figure 2-20. If an object is placed in the first quadrant, the views are projected back to the vertical plane and down to the horizontal plane. When an object is placed in the third quadrant, the views are projected forward to the vertical plane and up to the horizontal plane. This results in two distinct methods for projecting or positioning the two-dimensional representations of a three-dimensional object on a drawing sheet. An object placed in the second or fourth quadrant would create the same resulting views of the first and third and are, therefore, not used.

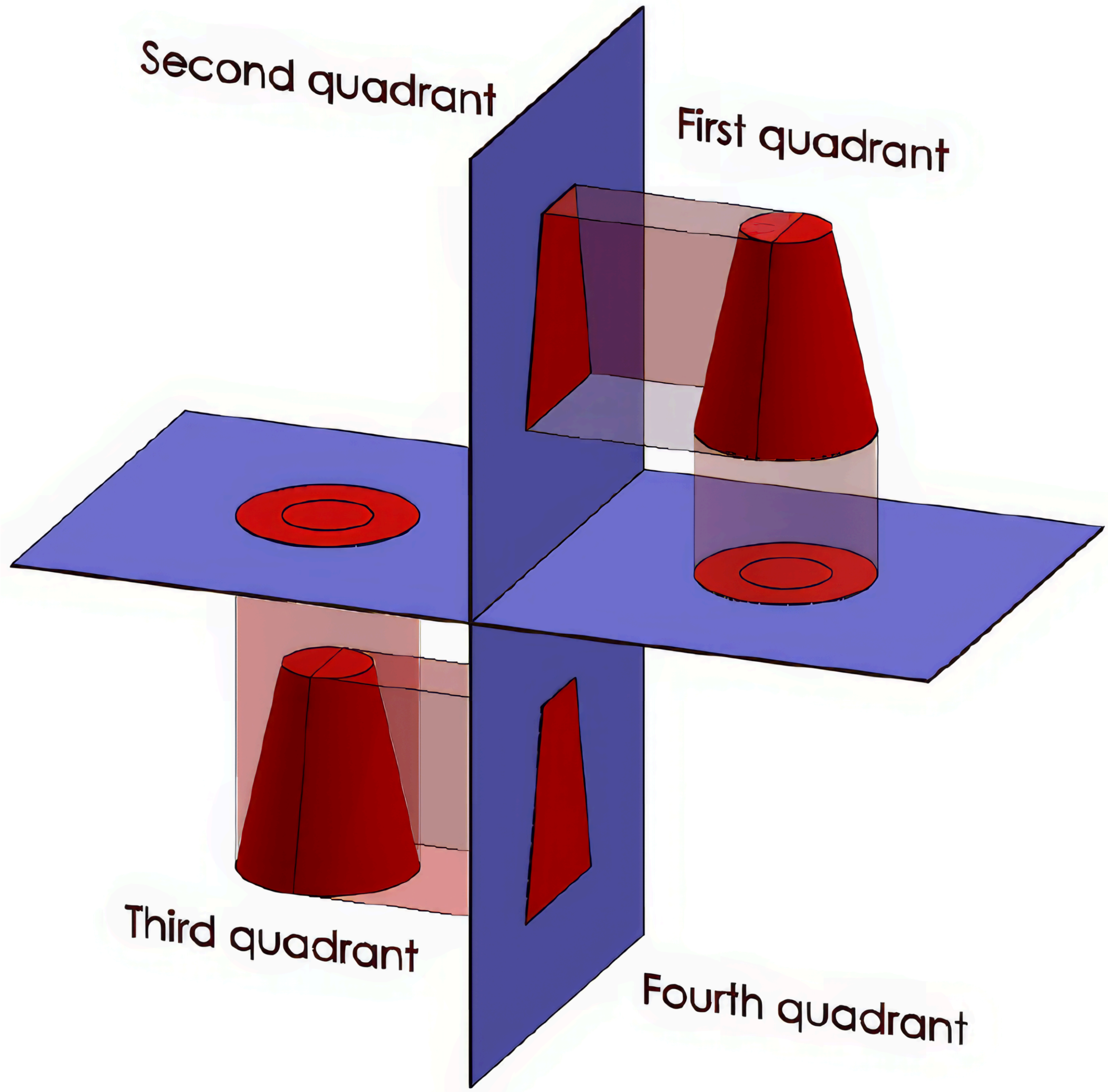


Figure 2-20. Viewing planes create four quadrants; both first and third views create separate projection techniques.

When the first quadrant projection style is used for a multiple-view drawing, it is known as **first-angle projection**. In Figure 2-21, you will see that as the viewing planes are laid flat, the resulting placement leaves the top view below the front view.

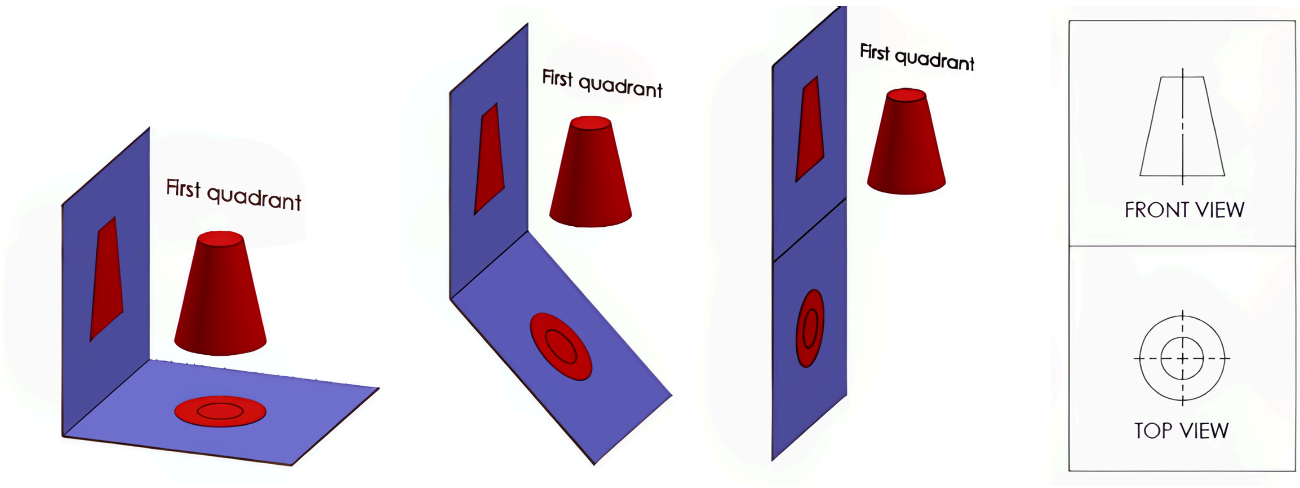


Figure 2-21. The first quadrant view results in the top view placed below the front view.

When the third quadrant projection style is used for a multiple-view drawing, it is known as **third-angle projection**. In Figure 2-22, as these views are laid flat, the top view is now placed above the front view.

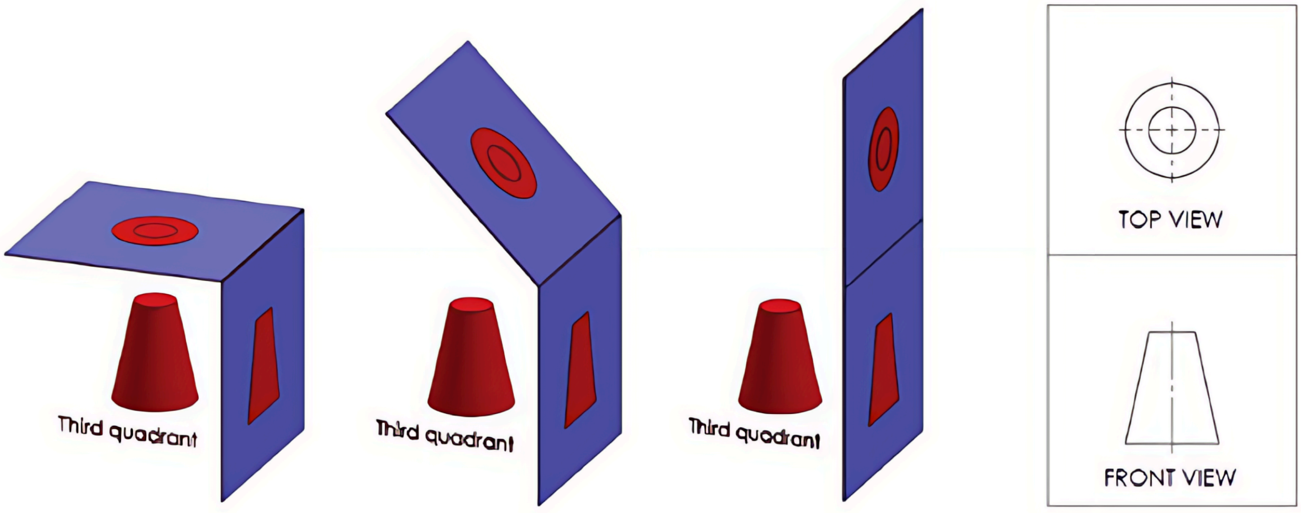


Figure 2-22. The third quadrant view results in the top view placed above the front view.

The two views of the cone-shaped object are placed on drawing sheets to identify which view style is being used. Either first-angle (Figure 2-23) or third-angle projection (Figure 2-24) will be identified by the placement of the two views of this or a similar object's views. The words "first angle" or “third angle” may be used in place of these views on a drawing sheet as well.

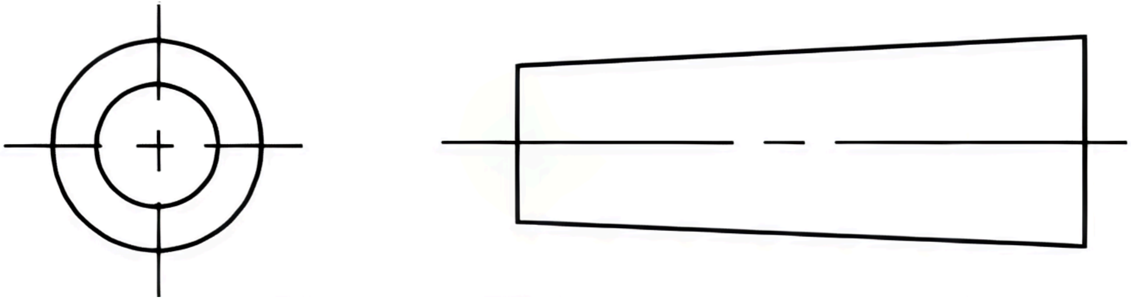


Figure 2-23. A commonly used third-angle projection symbol.

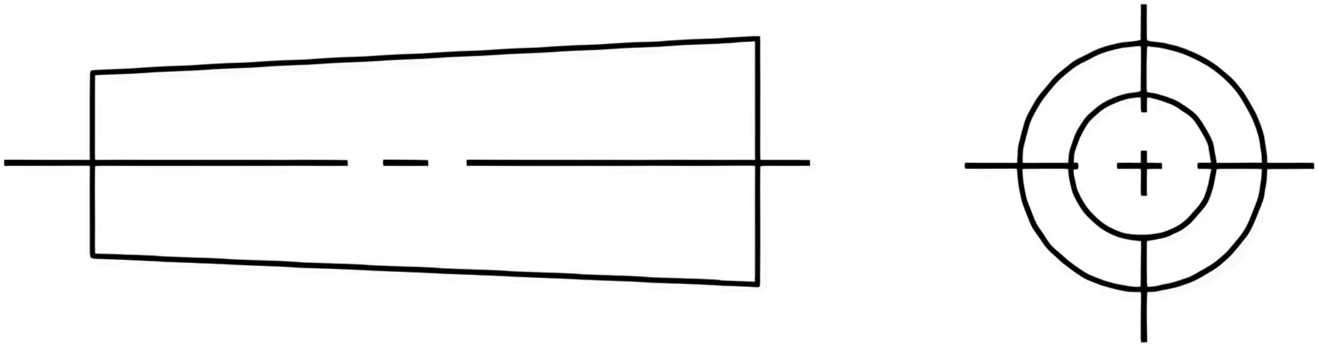


Figure 2-24. A commonly used first-angle projection symbol.

- Visit GD&T Basics’ First vs Third Angle – Orthographic Views for additional information on identifying first- and third-angle projection in orthographic projection drawings.

2.10 THIRD-ANGLE PROJECTION

The layouts of the views we've explored so far have been presented using third-angle projection. This method of creating multi-view drawings has been the standard in North America since the early 1900s. Third-angle projection was adopted because it offers a more intuitive way to arrange views within a drawing. Just like an unfolded box, you'll see the top view directly above the front view and the right view positioned to its right, making everything easier to understand at a glance, as shown in Figure 2-25. Observe the third-angle projection symbol in Figure 2-25 as well.

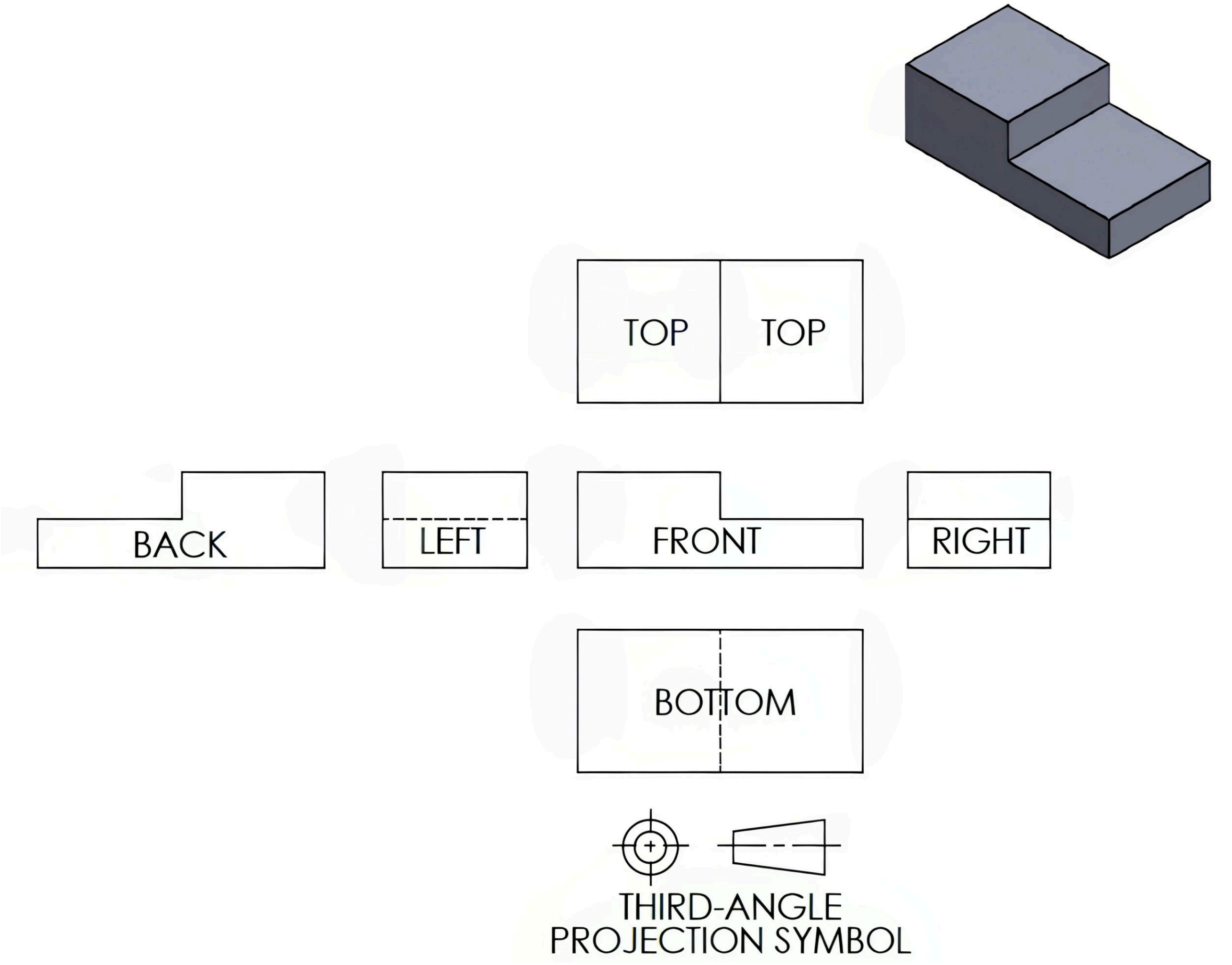


Figure 2-25. Third-angle projection.

2.11 FIRST-ANGLE PROJECTION

While the third-angle projection is the standard method used in the United States and Canada, a slightly different projection method is used in most other industrial nations. In first-angle projection, the views surrounding the front view are reversed when compared to third-angle projection. Observe Figure 2-26 where the top view is placed below the front view, and the right view is to the left of the front view.

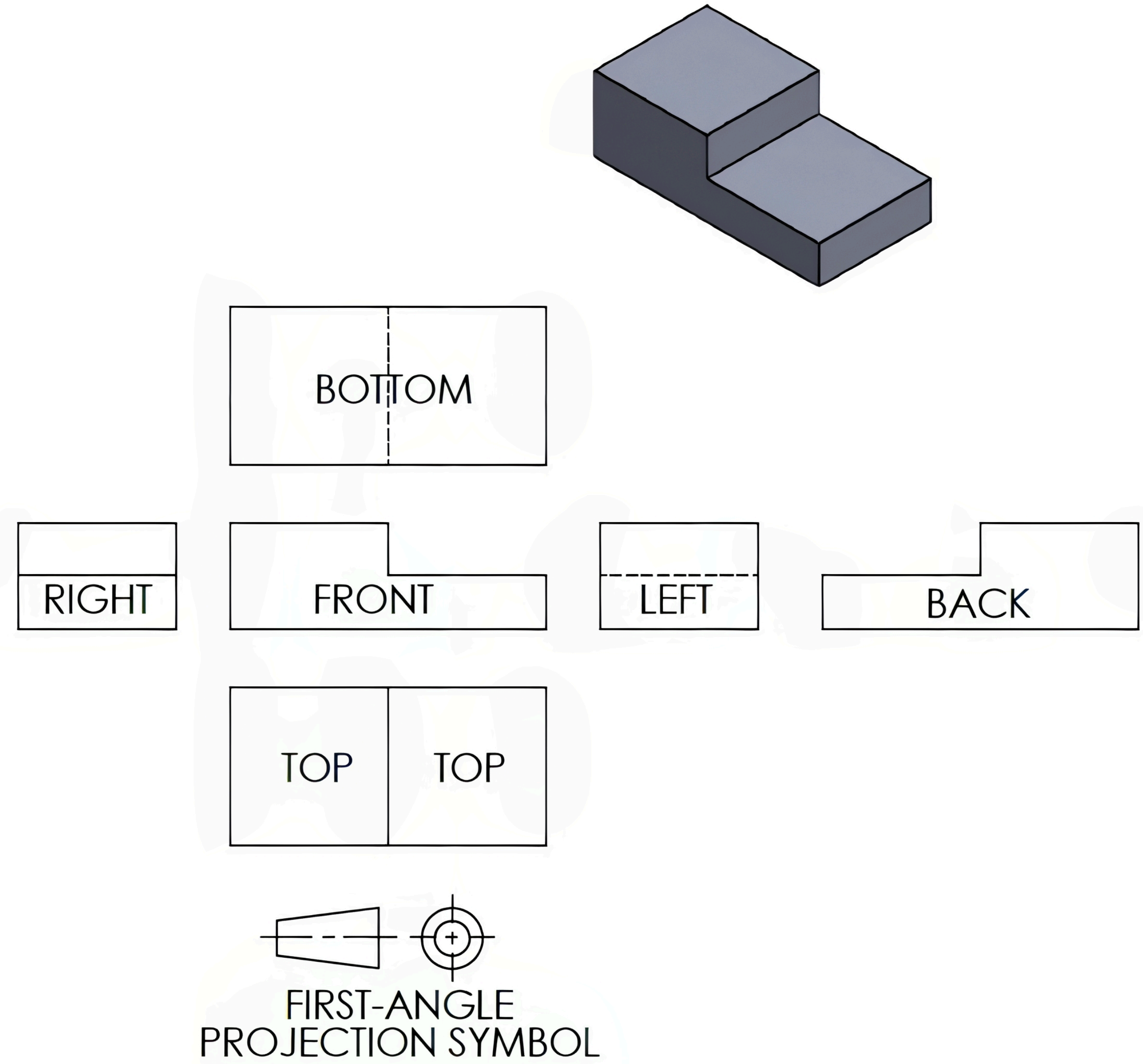


Figure 2-26. First-angle projection.

View Third Angle Projection Vs First Angle Projection YouTube video for an additional resource.

Take a moment to notice the differences in how views are arranged around the front view when comparing first-angle and third-angle projections. Pay close attention to the “cone” symbol as well—it follows the same orientation rules for both styles. Given that many industries operate on an international scale, it’s crucial to get into the habit of identifying which projection style is being used before diving into any drawings. Your awareness can prevent misunderstandings and ensure smooth communication across borders.

LEARNING ACTIVITIES

View Identification Quiz

Note: To enlarge this image, simply click on the plus sign icon in the upper right-hand corner of the print.

An interactive H5P element has been excluded from this version of the text. You can view it

online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=253#h5p-44

Surface Identification Quiz 1

Directions: Click and drag each letter in the gray boxes below to their corresponding open circle according to its surface identification.

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=253#h5p-43

Surface Identification Quiz 2

Directions: Click and drag each letter in the gray boxes below to their corresponding open circle according to its surface identification.

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=253#h5p-42

References:

Schultz, R. R., & Smith, L. (2011). *Unit 1: Dictionary of Terms. Standard Abbreviations. Alphabet of Lines.* In *Blueprint reading for machine trades* (7th ed.). Pearson.

Images:

Videos:

Skeen, C. (2013, October 3). *Height, Length/Width, and Depth dimensions on blueprints* [Video]. YouTube. All rights reserved. https://www.youtube.com/watch?v=dqS-atGfsH4

Maker Clan. (2014, September 27). *Third angle projection vs first angle projection 3D part 1* [Video]. YouTube. All rights reserved. https://www.youtube.com/watch?v=bk2E8P33Ztc

3. Types of Lines Found on Prints

3.1 INTRODUCTION

Learning Objectives

- Identify various lines used on prints
- Describe the purposes of the various types of lines used on prints
- Locate lines used on prints

Terms

- Alphabet of lines
- Visible line
- Hidden line
- Centerline
- Cutting-plane line
- Viewing-plane line
- Section line
- Dimension line
- Extension line
- Leader line
- Symmetry line
- Short break line
- Long break line

- Phantom line
- Chain line
- Precedence of lines

Each line on a print serves a unique purpose, conveying critical information with precision. The ASME, an independent organization, sets the standards that guide these engineered drawings. One of their guidelines is the **alphabet of lines**, an organized system in which every distinct line has its specific role.

It's truly essential to grasp the alphabet of lines if you want to accurately visualize and interpret a drawing. Lines come with different traits—some are solid or continuous without any breaks, while others appear as dashes with spaces in between. The width of these lines can also vary, being either thin or thick based on their purpose. Understanding these nuances is key to making sense of any detailed drawing.

3.2 ALPHABET OF LINES

Visible Line

Visible lines, often called object lines, are essential for highlighting every edge and intersection of an object in the view. These thick, continuous lines are designed to stand out clearly on any print. See Figure 3-1 for a representation of visible lines.

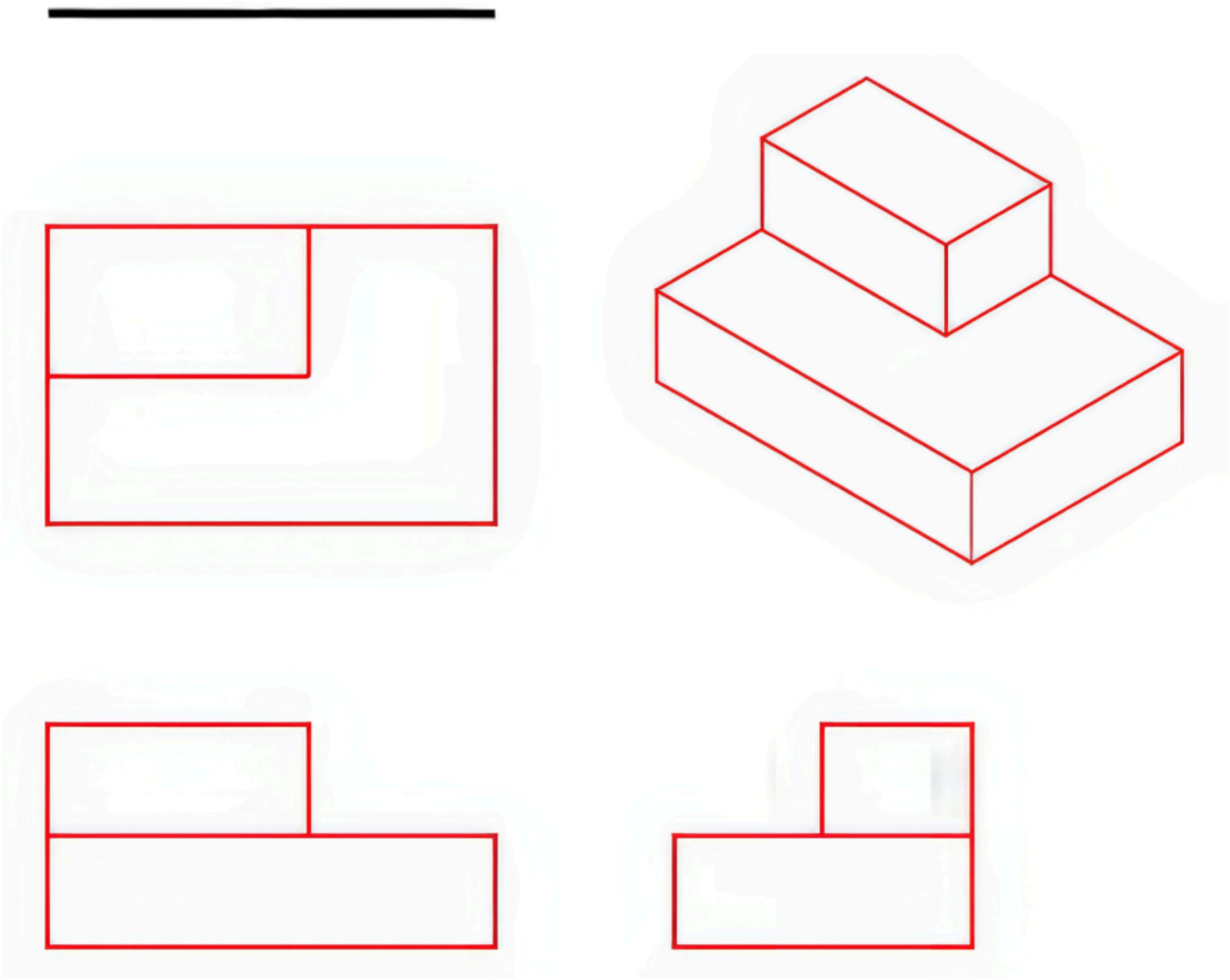


Figure 3-1. Visible lines represent all edges and surfaces of an object as seen in the view.

Hidden Line

Hidden lines represent the edges or surfaces not visible from the view. These lines have a series of evenly spaced thin dashes, as shown in Figure 3-2.

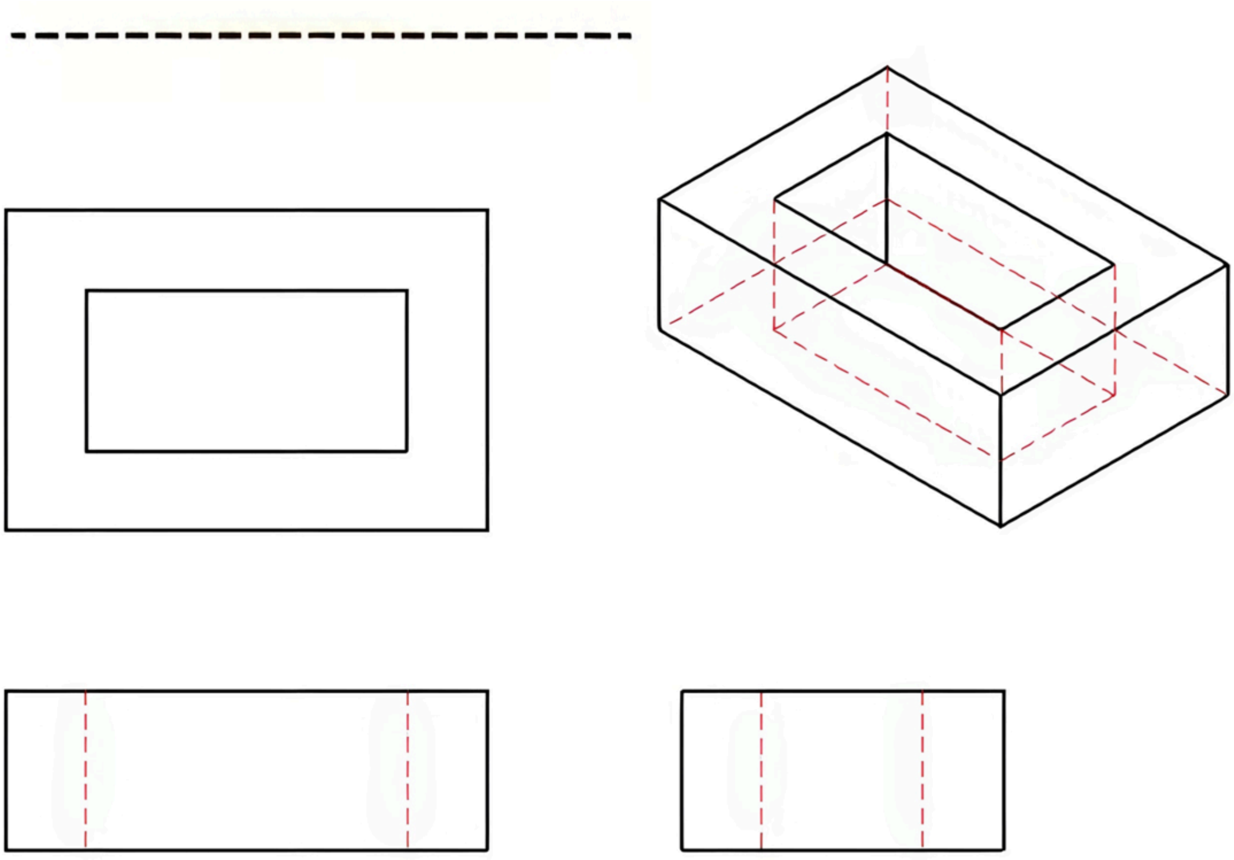


Figure 3-2. Hidden lines represent edges or surfaces not visible in the view.

Centerline

Centerlines are essential in identifying the exact center point of a hole or the axis of a part, and they can also mark the center of an arc or path of motion. As shown in Figure 3-3, these lines are drawn thin and consist of alternating long and short dashes.

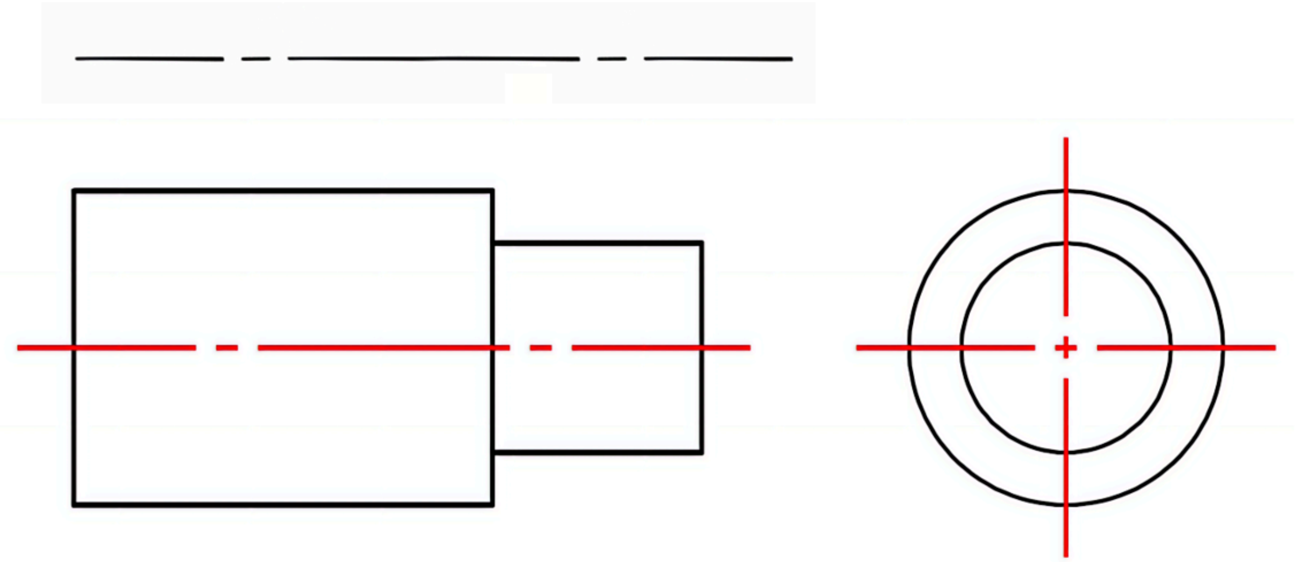


Figure 3-3. Centerlines indicate center location of symmetrical objects and the center locations of arcs, holes, and other circular features.

Cutting-plane Line

Cutting-plane lines will be drawn on the part to visually represent a "cut" through the material, known as a section view. The style may vary according to the specific standards used by the drafter. Cutting-plane lines are drawn thick, with arrows indicating the direction in which the "cut" section is being viewed. The letters near the arrows are in place to be correctly matched with the corresponding section view. See Figure 3-4 for an example of cutting-plane lines.

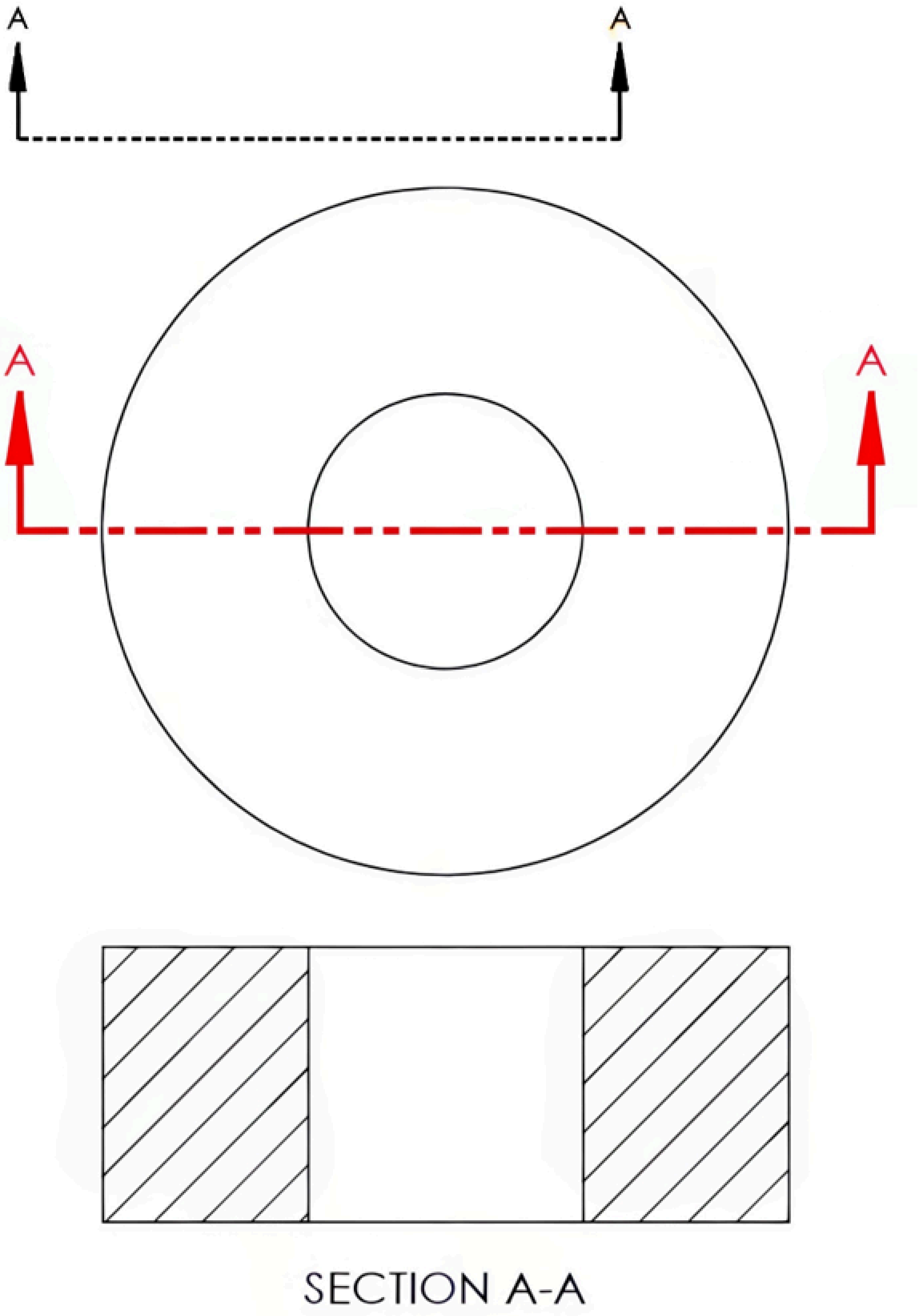


Figure 3-4. Cutting-plane lines indicate where a view has been "cut" through and the direction the resulting section is viewed.

Viewing-plane Line

The cutting-plane line and the **viewing-plane line** are essentially the same; what sets them apart is where they are positioned on the drawing. Unlike the cutting-plane line, which slices through the part, the viewing-plane line is placed near the part to offer a separate view of the object from a specific viewing direction, as depicted in Figure 3-5. When more than one cutting-plane or viewing-plane line is used on a drawing, the letters will be used to correspond with the correct view.

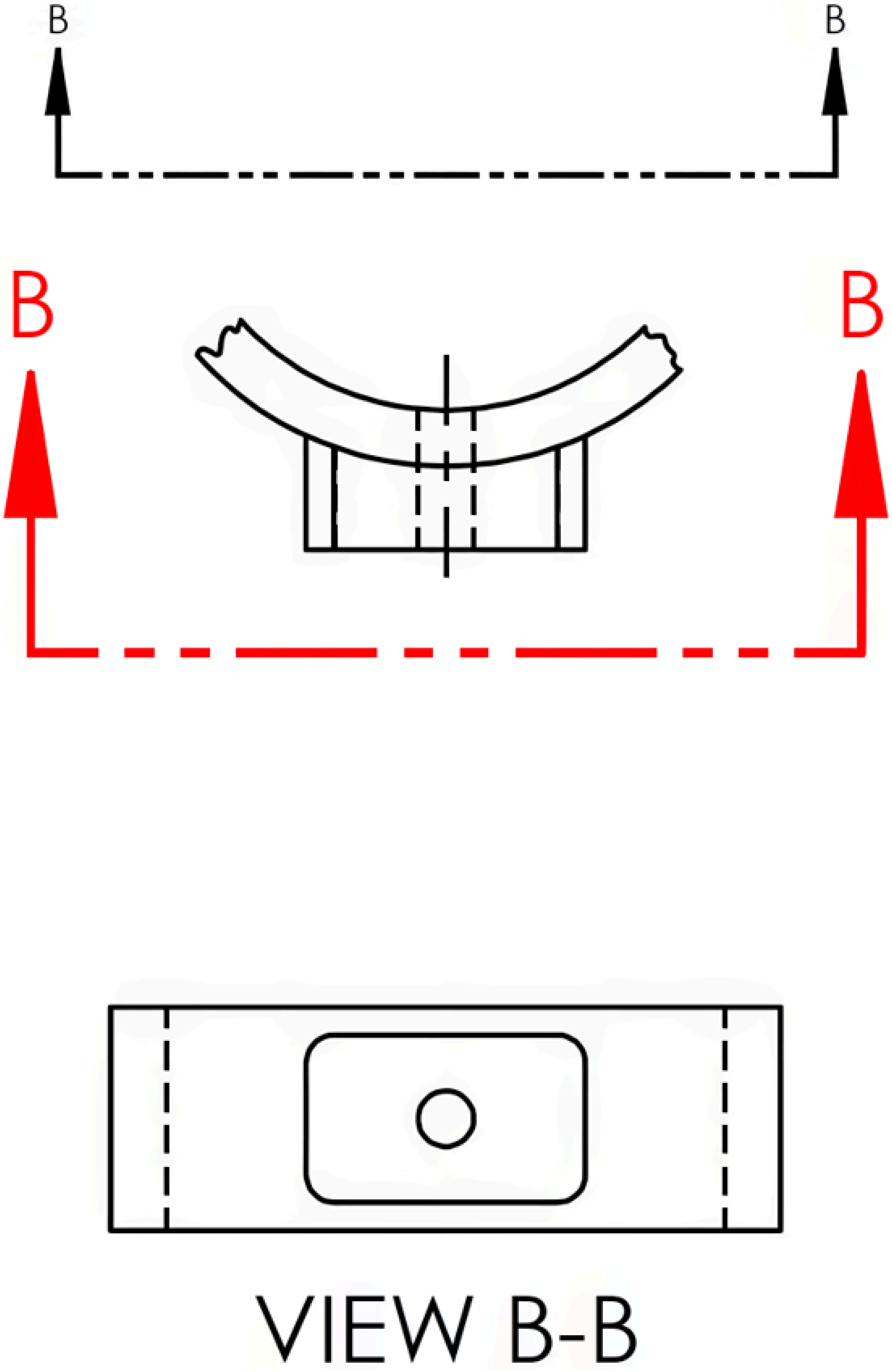


Figure 3-5. Viewing-plane lines indicate the viewing direction of a separated view.

Both cutting-plane and viewing-plane lines may use different line styles than shown above. Figure 3-6 displays an alternative version of the ANSI (American National Standards Institute) style on the left and the ISO (International Organizations for Standardization) style on the right.

Figure 3-6. Additional line styles may be used for both cutting-plane and viewing-plane lines.

Section Lines

A section view offers a glimpse into the inner workings of a part by showing you what's inside from one of the primary perspectives. The **section lines** indicate where the material has been "cut" through, unveiling these interior details; see Figure 3-7. While the diagonal pattern is the most frequently used style for these lines, there are many other styles available that can reflect different types of materials being sectioned. These alternative patterns can be substituted for the more commonly used diagonal style to better represent specific materials.

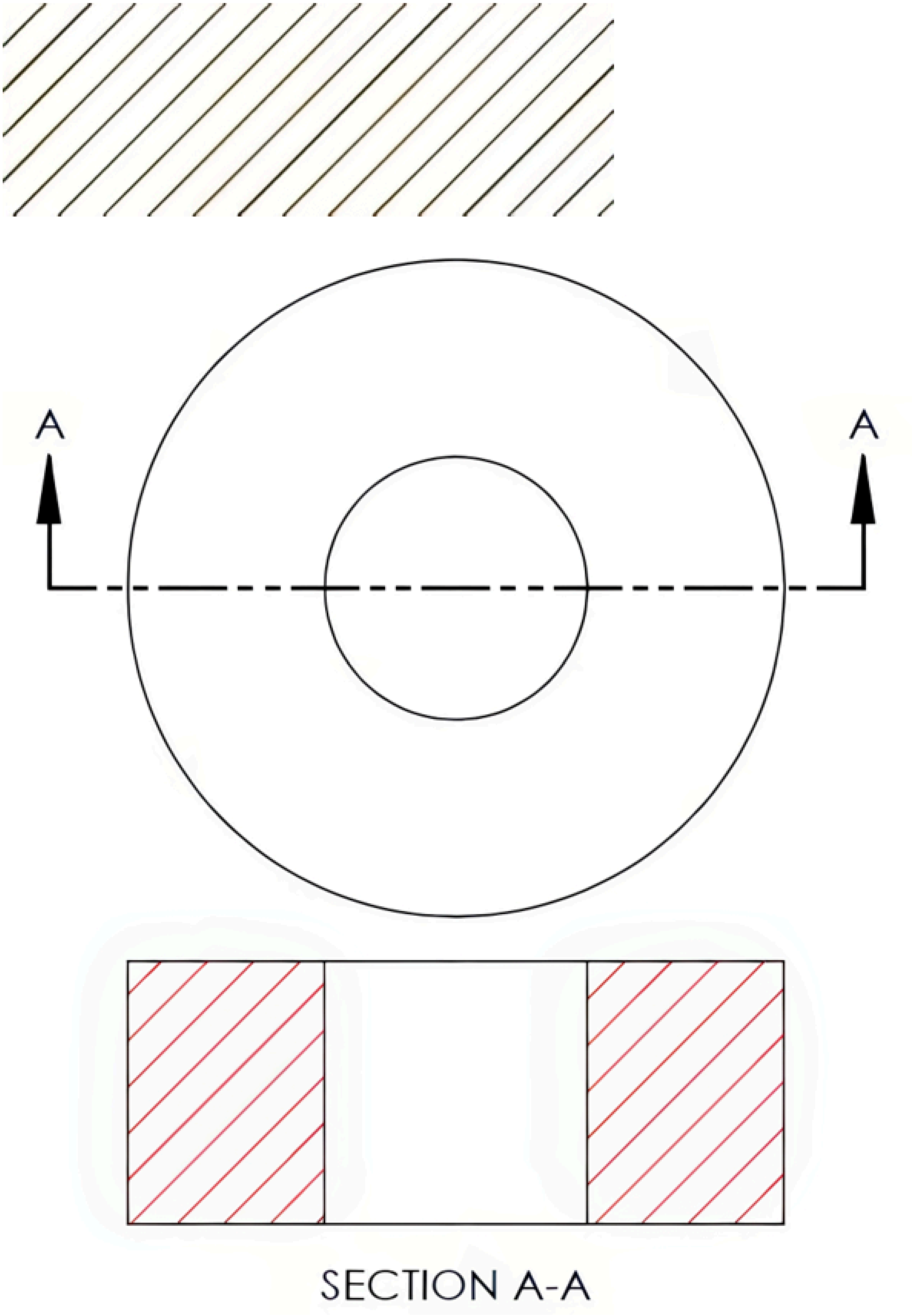


Figure 3-7. Section lines indicate where the material has been “cut” through.

Dimension Line

The **dimension line** spans the distance of the feature measurement and contains the numerical value, called a dimension. These lines will typically end with arrowheads, as shown in Figure 3-8.

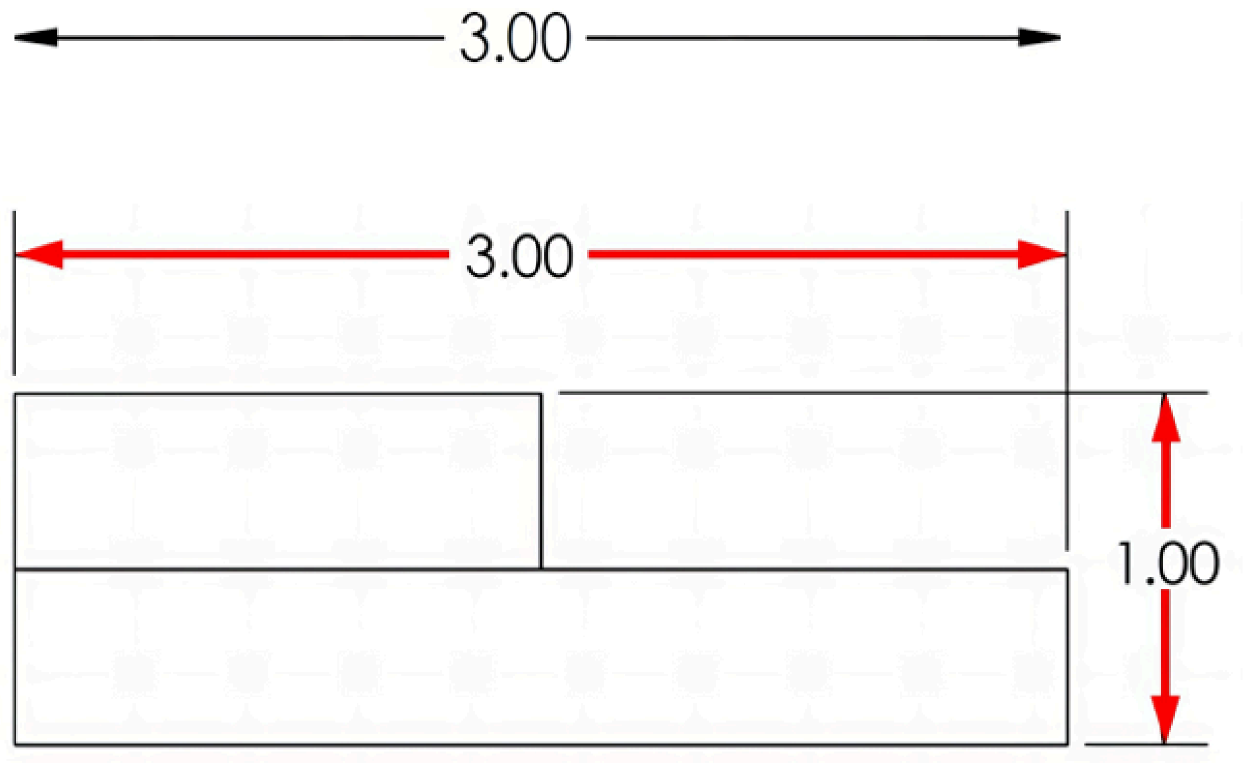


Figure 3-8. Dimension lines span the distance of a measured feature and contain the measurement value.

Extension Line

The **extension lines** will reach out from the ends of the dimension lines and extend to the exact point of measurement on the feature. However, they'll stop just short of actually touching the part or feature, as illustrated in Figure 3-9.

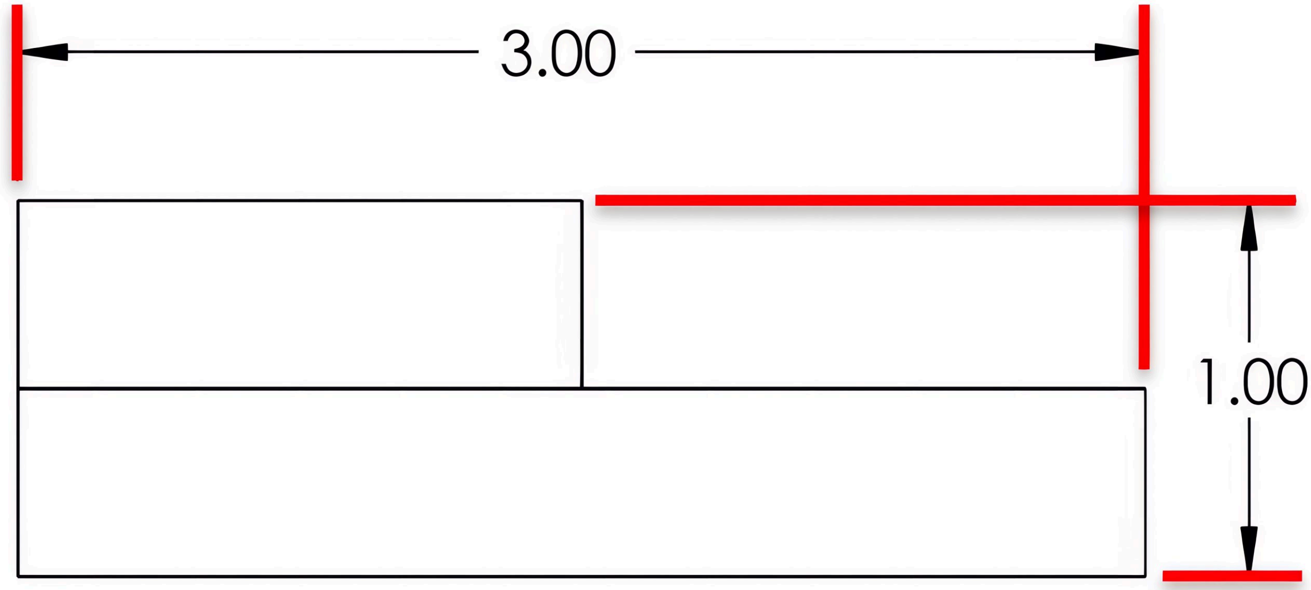


Figure 3-9. Extension lines reach from the measurement points or edges to the dimension lines.

Leader Line

The **leader line** is a thin slanted line, generally with an arrowhead pointing to a feature to provide information. Information associated with a leader line may include dimensions, notes, item or part numbers, or specific instructions for that feature. The leader may also end with a dot in place of the arrowhead. See Figure 3-10.

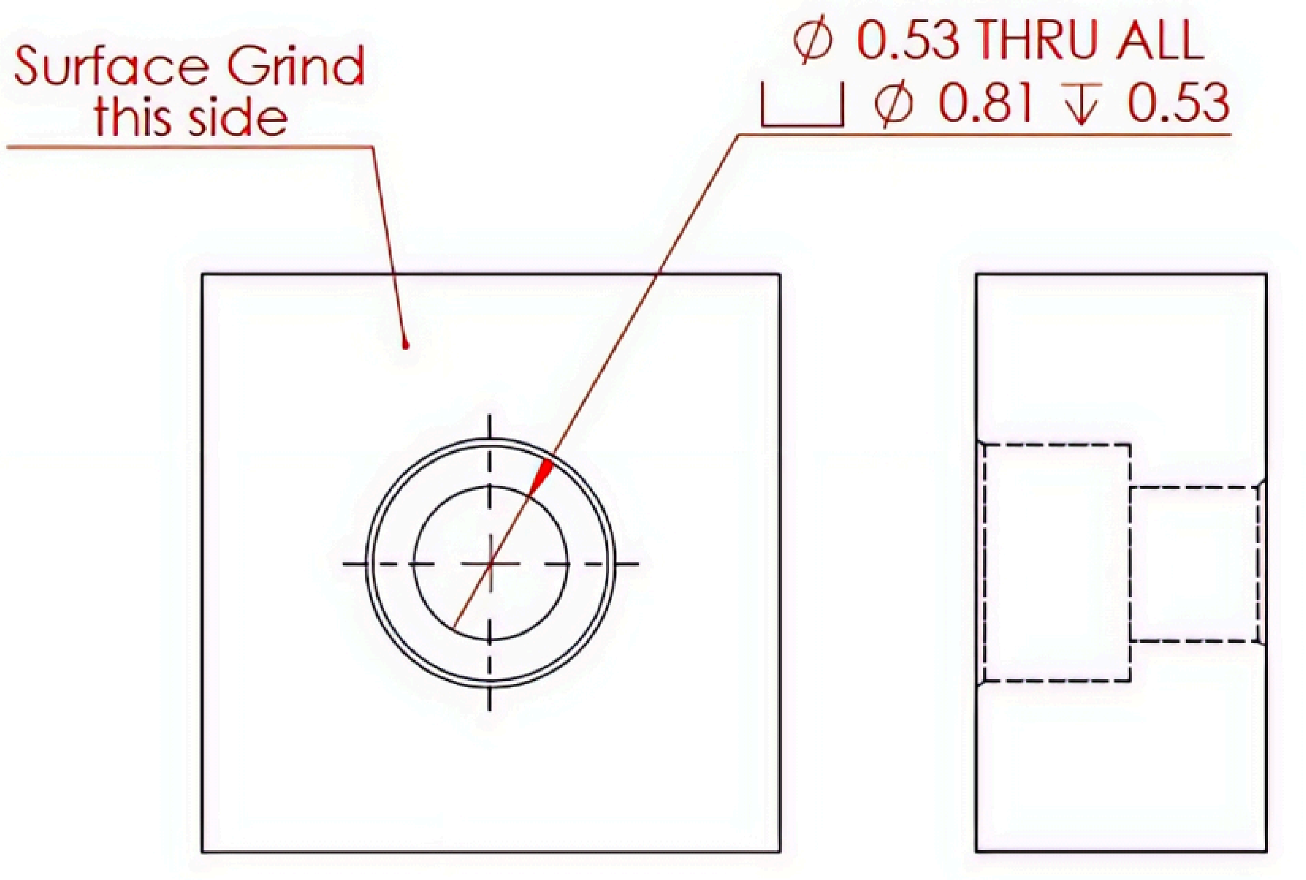


Figure 3-10. Leader lines point the reader to notes, item or part numbers, dimensions, or instructions.

Symmetry Line

A **symmetry line** will be used where only one half of a symmetrical part is drawn. This may be used on a part to save space on the drawing. A symmetry line is a centerline accompanied by two thicker parallel lines positioned at each end, perpendicular to the main line. Figure 3-11 shows an example of a symmetry line.

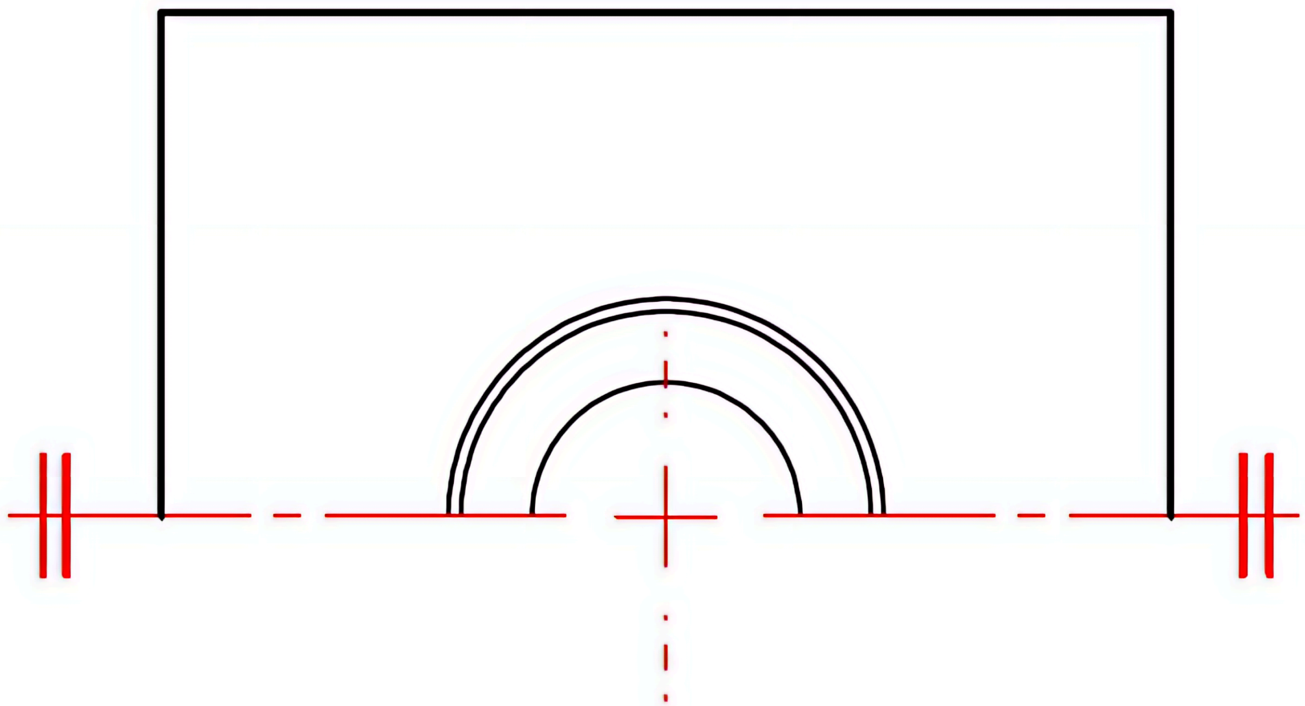


Figure 3-11. Symmetry line indicating the drawing is divided at its symmetrical edge.

Short Break Line

A **short break line** shows where a part has been broken away to conserve space when the full part does not need to be shown (see Figure 3-12). Short break lines will also be used in broken-out section views, removing material to expose internal part features, as show in Figure 3-13. These lines are drawn in a jagged freehand pattern.

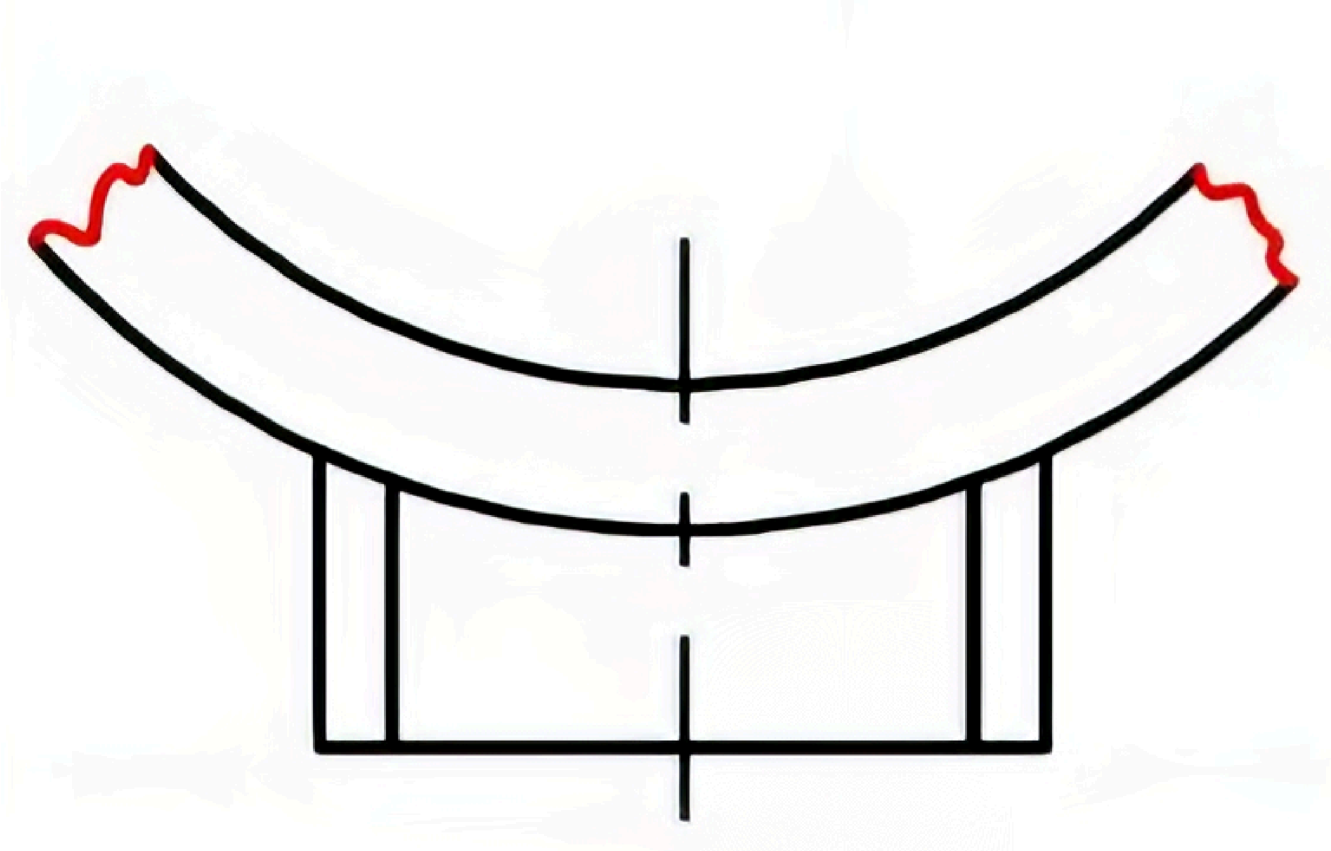


Figure 3-12. Short break line indicates a portion of the part has been broken from the whole part.

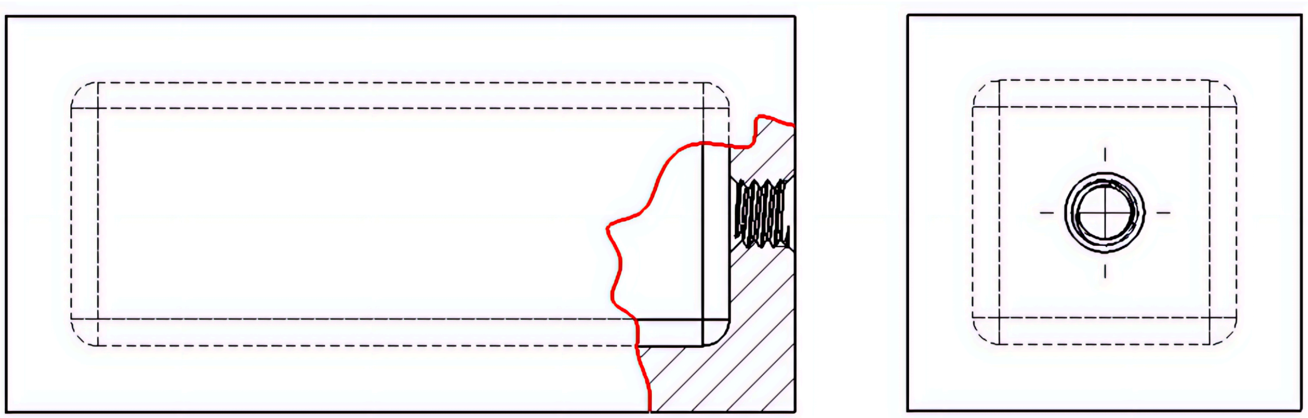


Figure 3-13. Short break line used to remove material exposing hidden features.

Long Break Line

A **long break line** will be used in pairs to conserve space by removing the central portion of a longer uniform part, allowing a larger drawing scale to be used. These lines are thin and straight with a zigzag shape, used in pairs, and parallel to each other. See Figure 3-14 for an illustration of a long break line. A long break line on a cylindrical part may use an "S"-shaped break line to identify it as cylindrical, as shown Figure 3-15.

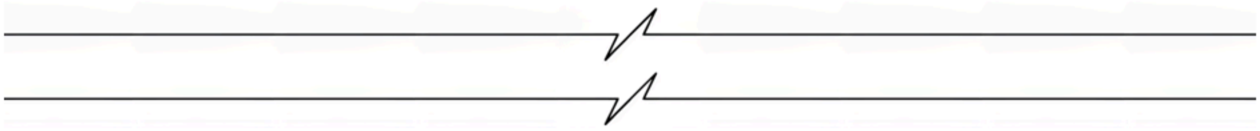


Figure 3-14. Long break line allows the removal of a portion of a longer part to shorten a drawing view.

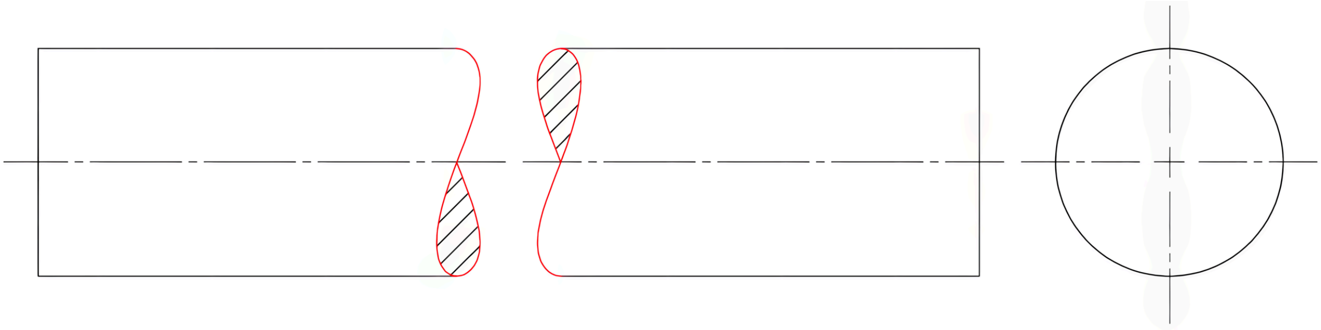


Figure 3-15. Cylindrical part using an "S"-shaped long break line.

Phantom Line

A **phantom line** is a thin line with a long line followed by two short dashes. A phantom line represents the outline of a part's alternate position or a mating part/datum plane. This line could also be used to replace repeated details such as threads or gear teeth; this application was more commonly used before CAD systems when prints were hand drawn. See Figure 3-16.

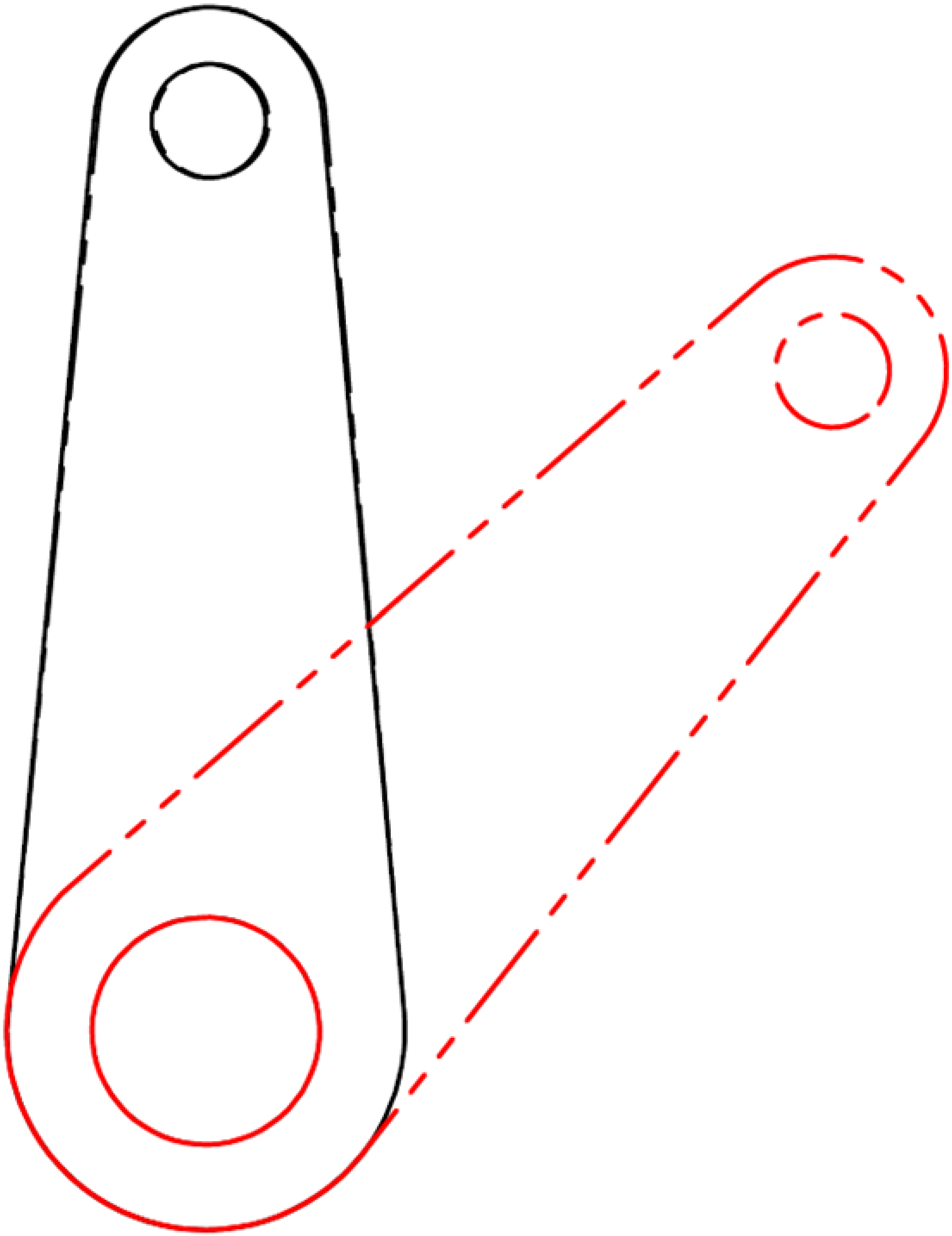


Figure 3-16a. Illustration of phantom lines to indicate a moving part's alternate position.

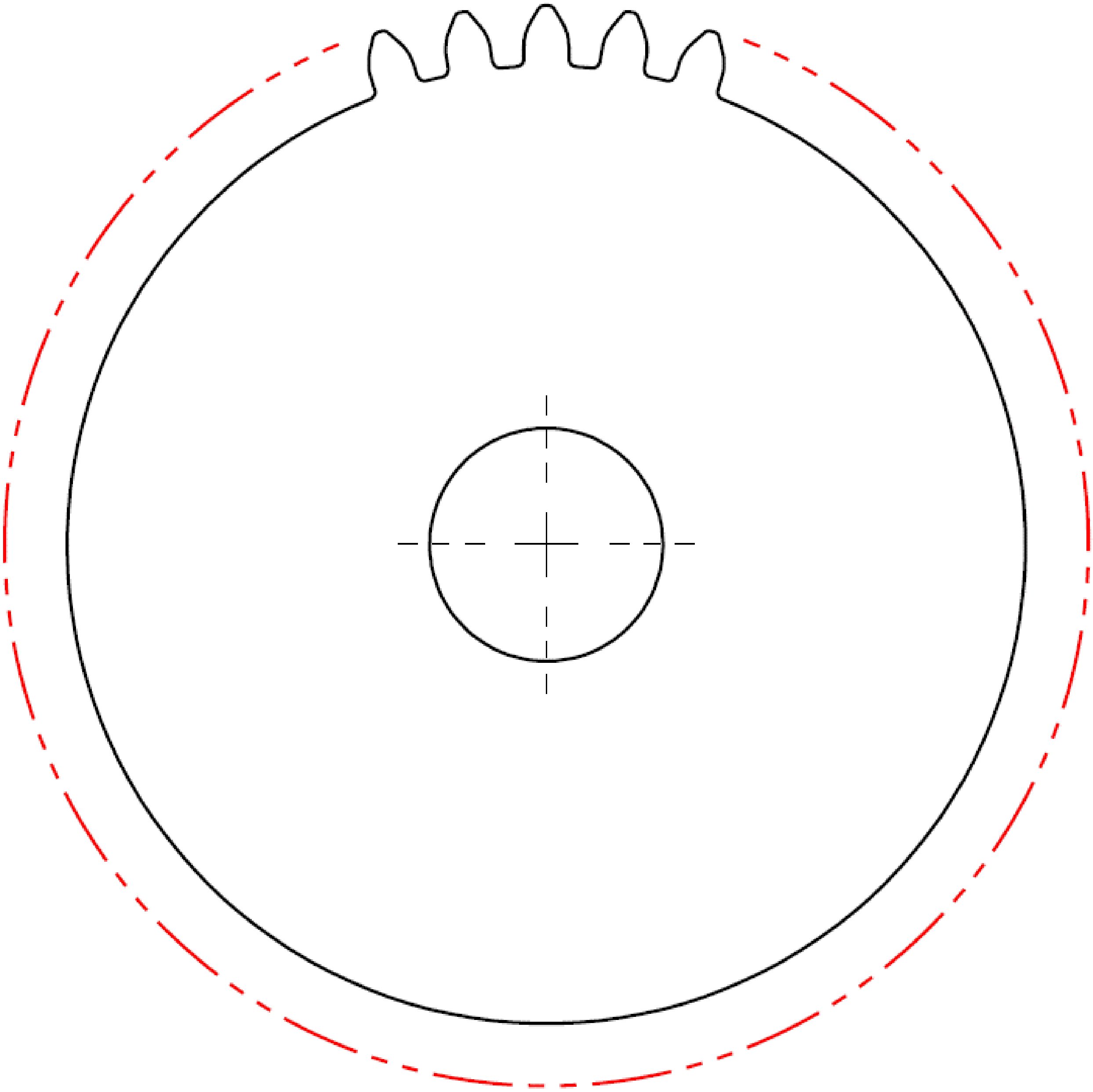


Figure 3-16b. Phantom lines are used to indicate a moving part's alternate position and replace repeated gear teeth.

Chain Line

A **chain line** is characterized by a series of alternating long and short lines, drawn thicker to differentiate from centerlines. This type of line highlights areas on a surface that require special attention or treatment. Additionally, chain lines can mark the projected tolerance zone in geometric dimensioning and tolerancing practices. See Figure 3-17 below for an example of a chain line.

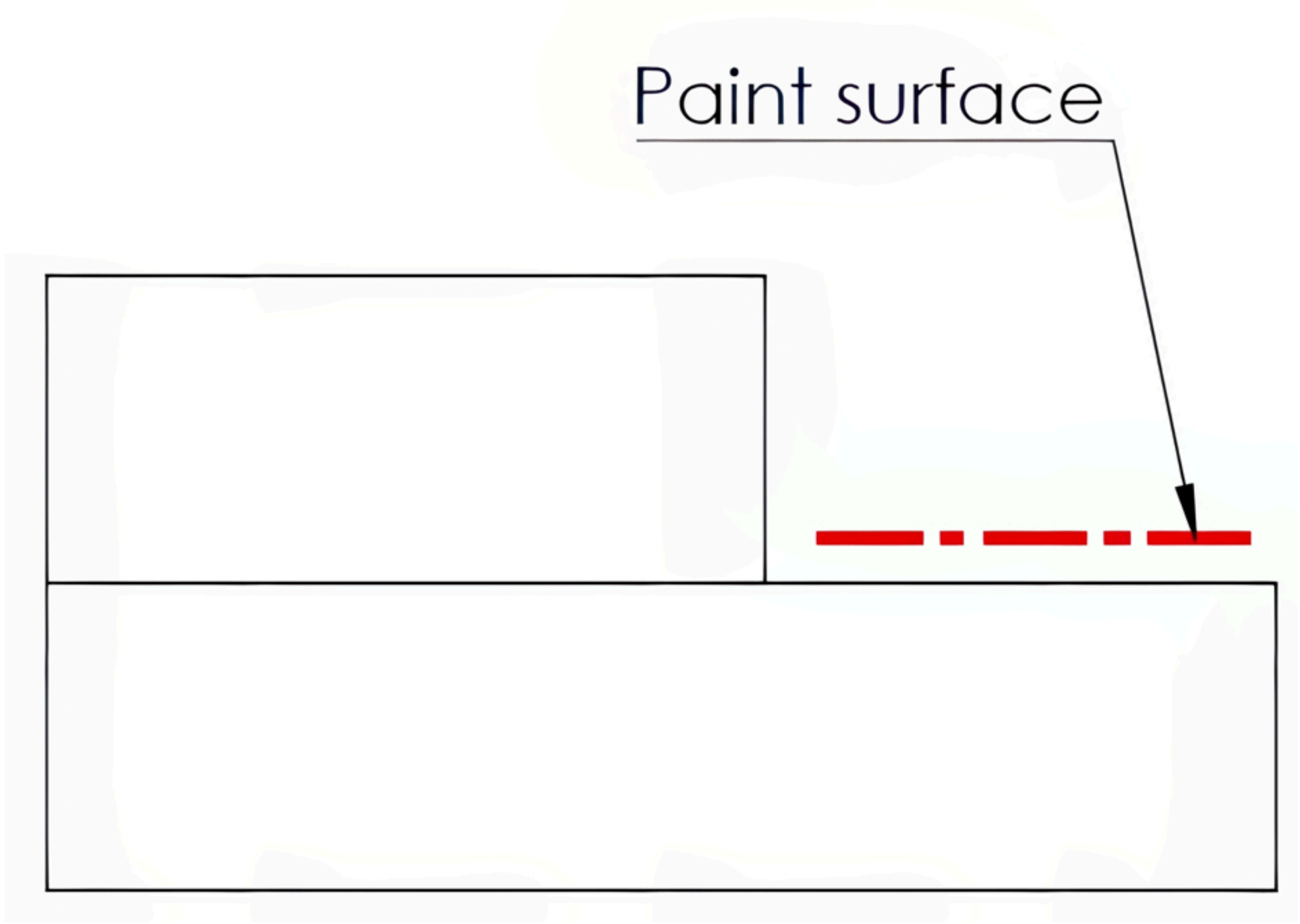


Figure 3-17. Chain line used to indicate special treatment for a portion of the part.

3.3 PRECEDENCE OF LINES

In numerous illustrations, it is common for the lines of a component to coincide in a particular drawing view. The guidelines that govern these situations are referred to as the **precedence of lines**, which dictate which line remains visible. Visible lines take priority over hidden and centerlines, while hidden lines supersede centerlines in visibility. See Figure 3-18 below.

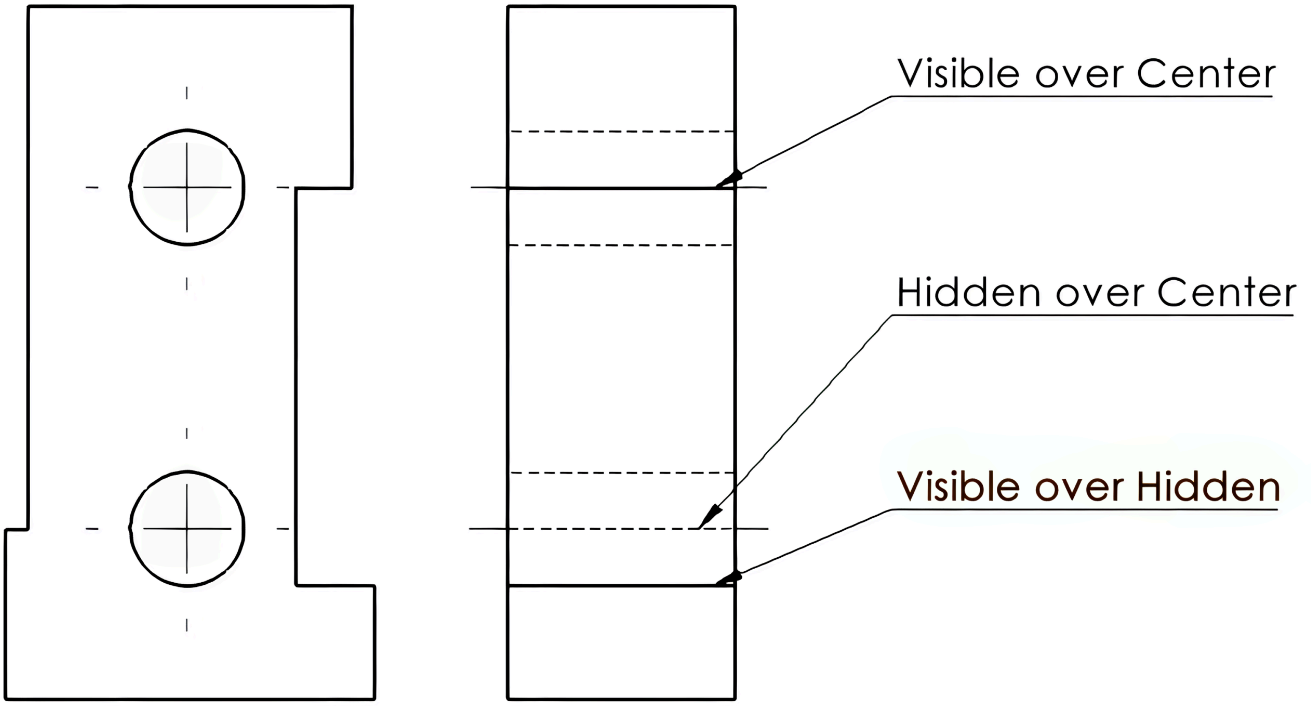
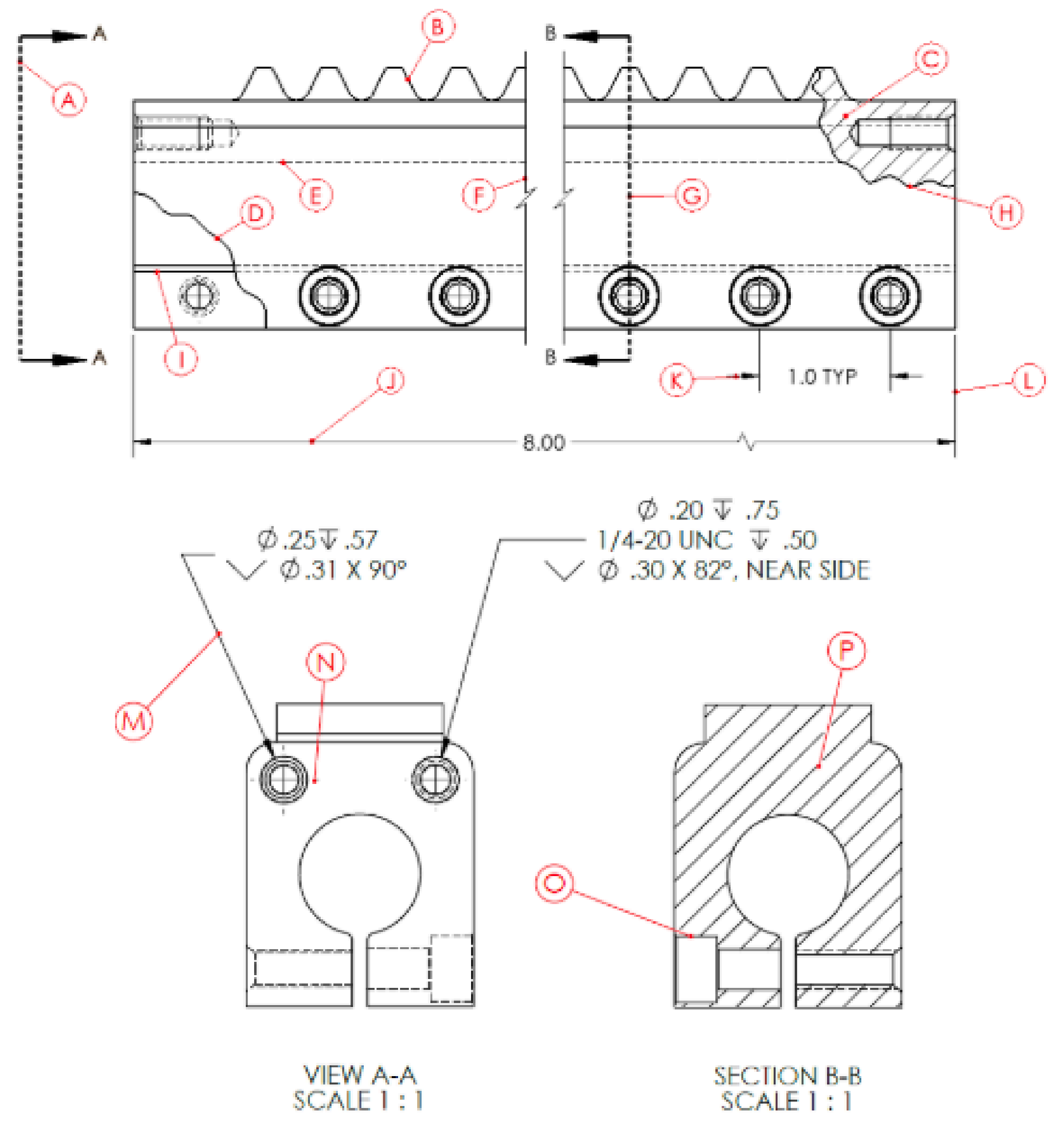


Figure 3-18. Precedence of lines dictates which line will remain visible when multiple lines coincide with each other.

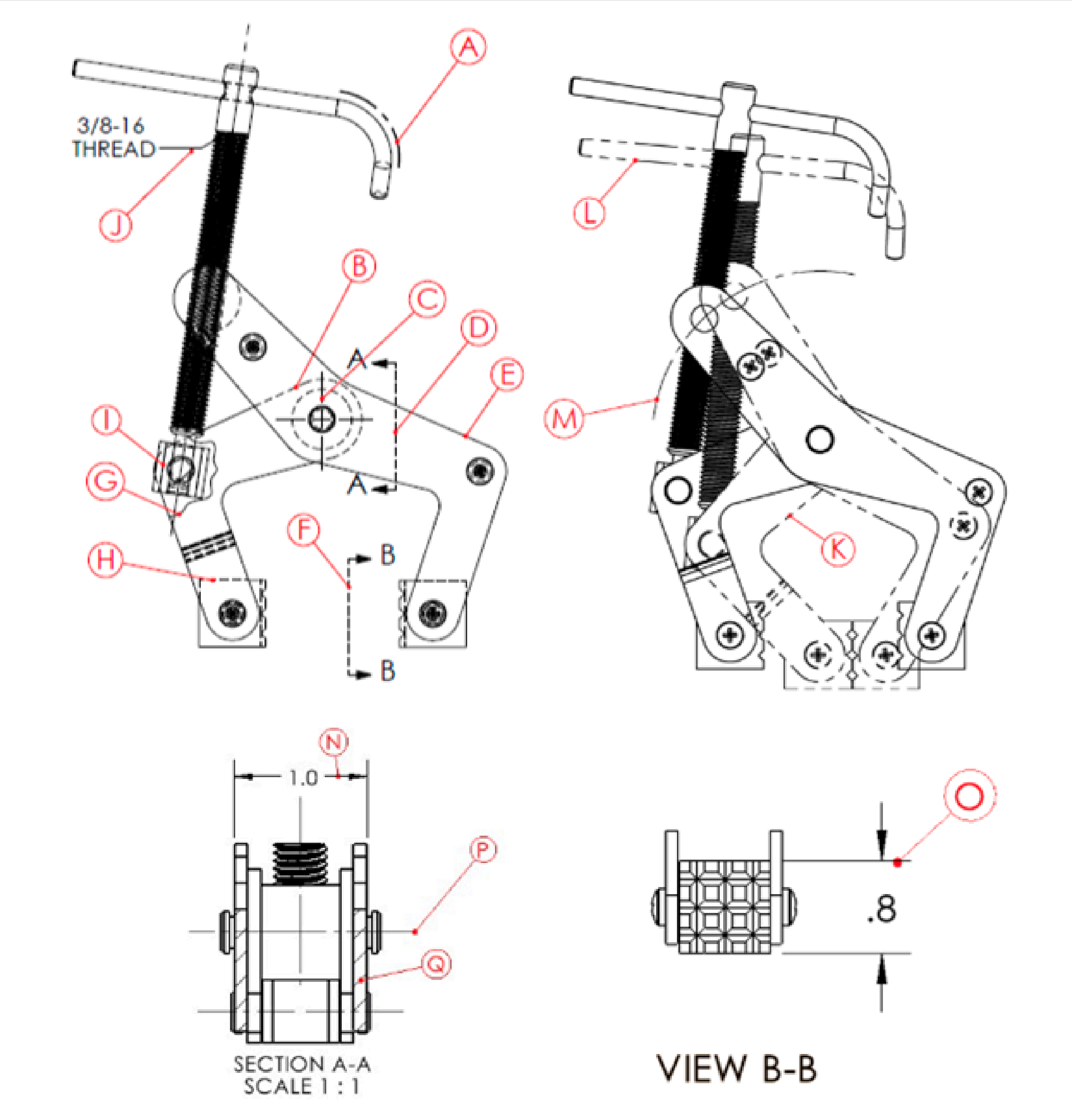
LEARNING ACTIVITIES

Line Quiz 1



An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=52#h5p-18

Line Quiz 2



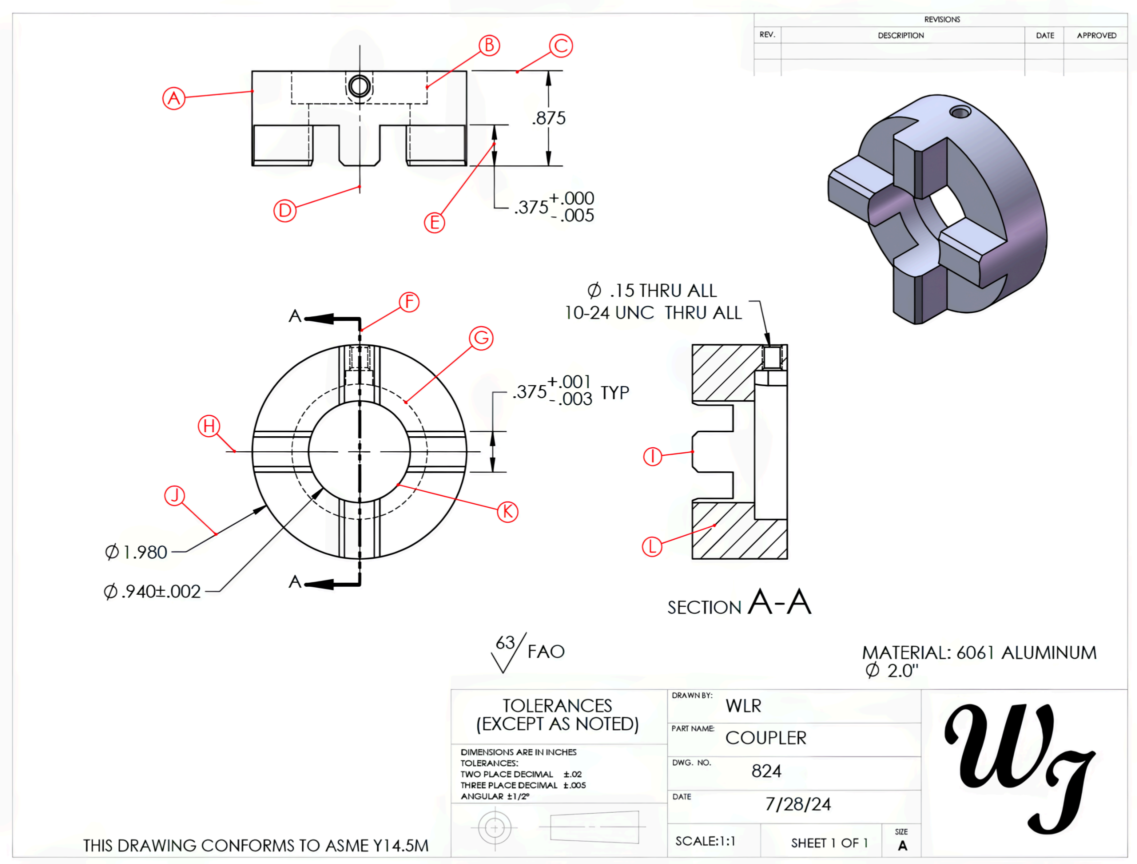
An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=52#h5p-19

Line Quiz 3

Line Quiz 3

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=52#h5p-20

Line Quiz 4



An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=52#h5p-21

References:

CAERT, Inc. (n.d.). *Alphabet of lines [PDF].* https://acrobat.adobe.com/link/review?uri=urn%3Aaaid%3Ascds%3AUS%3A62dfefd9-af0c-3968-9543-9e5062ac8e3a

Schultz, R. R., & Smith, L. (2011). Unit 1: Dictionary of terms. Standard abbreviations: Alphabet of lines. In *Blueprint reading for machine trades* (7th ed.). Pearson.

Images:

All images by Mark Lorier and licensed under CC BY-NC 4.0 except where otherwise noted.

4. Objects in Different Views

4.1 INTRODUCTION

Learning Objectives

- Identify correct views for objects
- Visualize how actual part features should appear based on print features
- Sketch drawings incorporating the alphabet of lines
- Sketch simple orthographic drawings
- Draw missing views of objects based on given views

Terms

- Isometric view
- Orthographic view
- Dimetric view
- Trimetric view

In this chapter, we will identify the correct views of objects by use of visible and hidden lines. We will also explore the art of sketching objects using both orthographic and isometric techniques. Sketching is a fundamental skill for designers, engineers, and hobbyists, as it allows them to quickly convey their ideas and designs on paper. By mastering both orthographic and isometric sketching, you will be able to accurately capture the dimensions and details of any object. Throughout this chapter, we will look into the principles of sketching, practice different techniques, and develop the ability to sketch in both styles, as well as give you a greater understanding of these views as you work with them.

4.2 VIEW IDENTIFICATION

The ability to visualize how an actual part should appear based on print features is crucial in print reading. This skill ensures that the interpretation of technical drawings translates accurately into physical components, reducing errors and improving efficiency in manufacturing. **Isometric views** provide a three-dimensional representation of an object. This view helps in understanding the overall shape and structure of the part. On the other hand, **orthographic views** offer a series of two-dimensional projections of the object, typically including the front, top, and side views. These projections are essential for detailing the exact dimensions and relationships between different features of the part. Mastering both isometric and orthographic views allows for a comprehensive understanding of blueprints, ensuring precise and effective communication of design intent.

Complete the exercises below to enhance your skill in imagining the physical forms of each section as you complete the missing views. Apply what you've learned about projection views, along with visible and hidden lines.

View Identification Exercise 1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=87#h5p-3

View Identification Exercise 2

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=87#h5p-4

4.3 SKETCHING ORTHOGRAPHIC VIEWS

Whether you're producing part prints for manufacturing or need to quickly convey the specifics of a three-dimensional item on a two-dimensional sheet, mastering orthographic drawings is essential for ensuring a clear grasp of the object's form, size, and characteristics. Enhancing this skill will not only improve your capability to communicate design concepts to others without relying on CAD software, but also improve your ability to visualize and perceive three-dimensional shapes from an orthographic perspective.

The following YouTube video will illustrate the isometric view of

Orthographic Sketch 1 transferred into a three-view orthographic sketch.

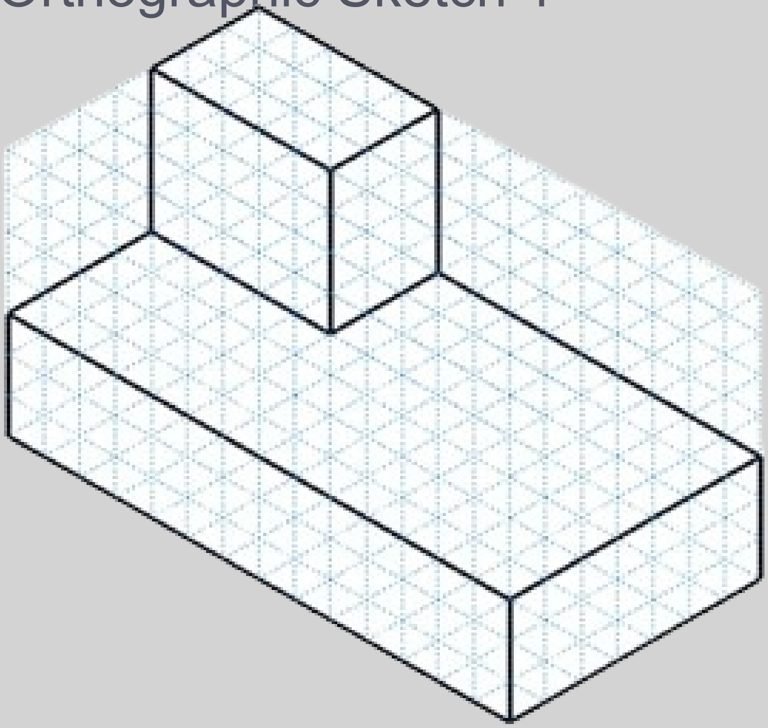
One or more interactive elements has been excluded from this version of the text. You can view them online here: https://wtcs.pressbooks.pub/blueprintreading/?p=87#oembed-1

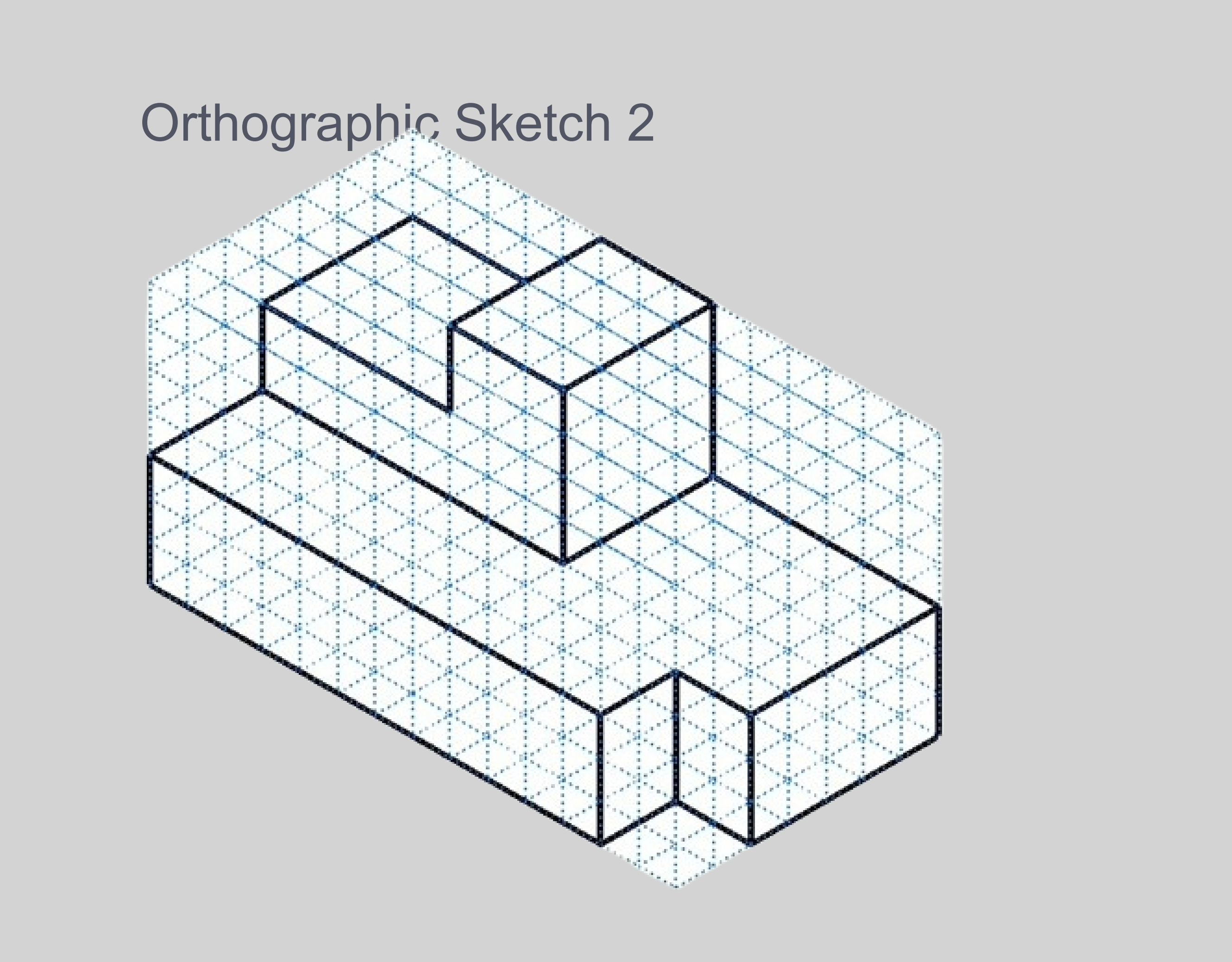
Orthographic Sketching Exercises

Use grid paper to complete the missing views to each orthographic sketch. Include any visible and hidden lines. (Solutions are provided in the Learning Activities—Solutions section.)

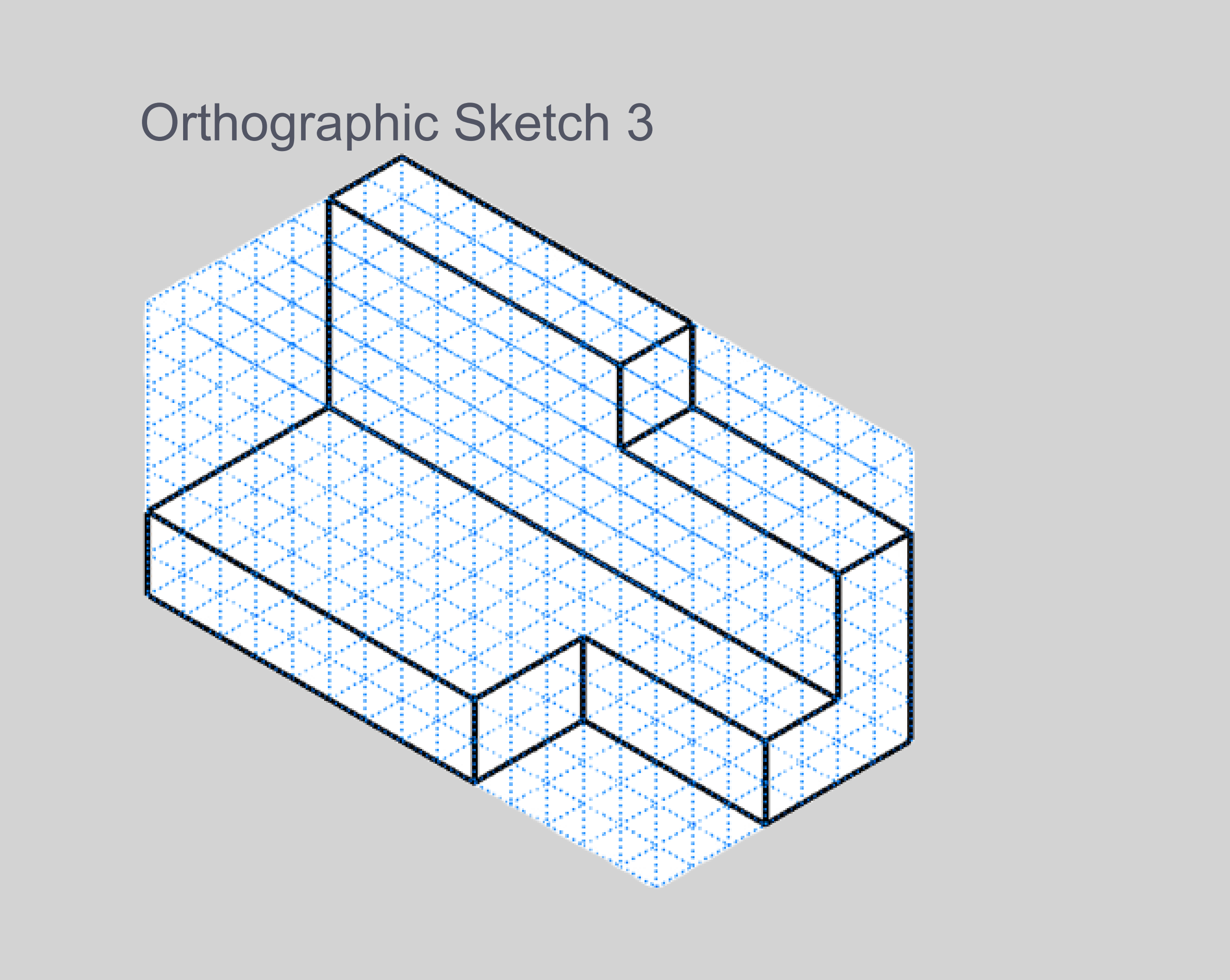
Orthographic Sketching Exercises

Orthographic Sketch 1

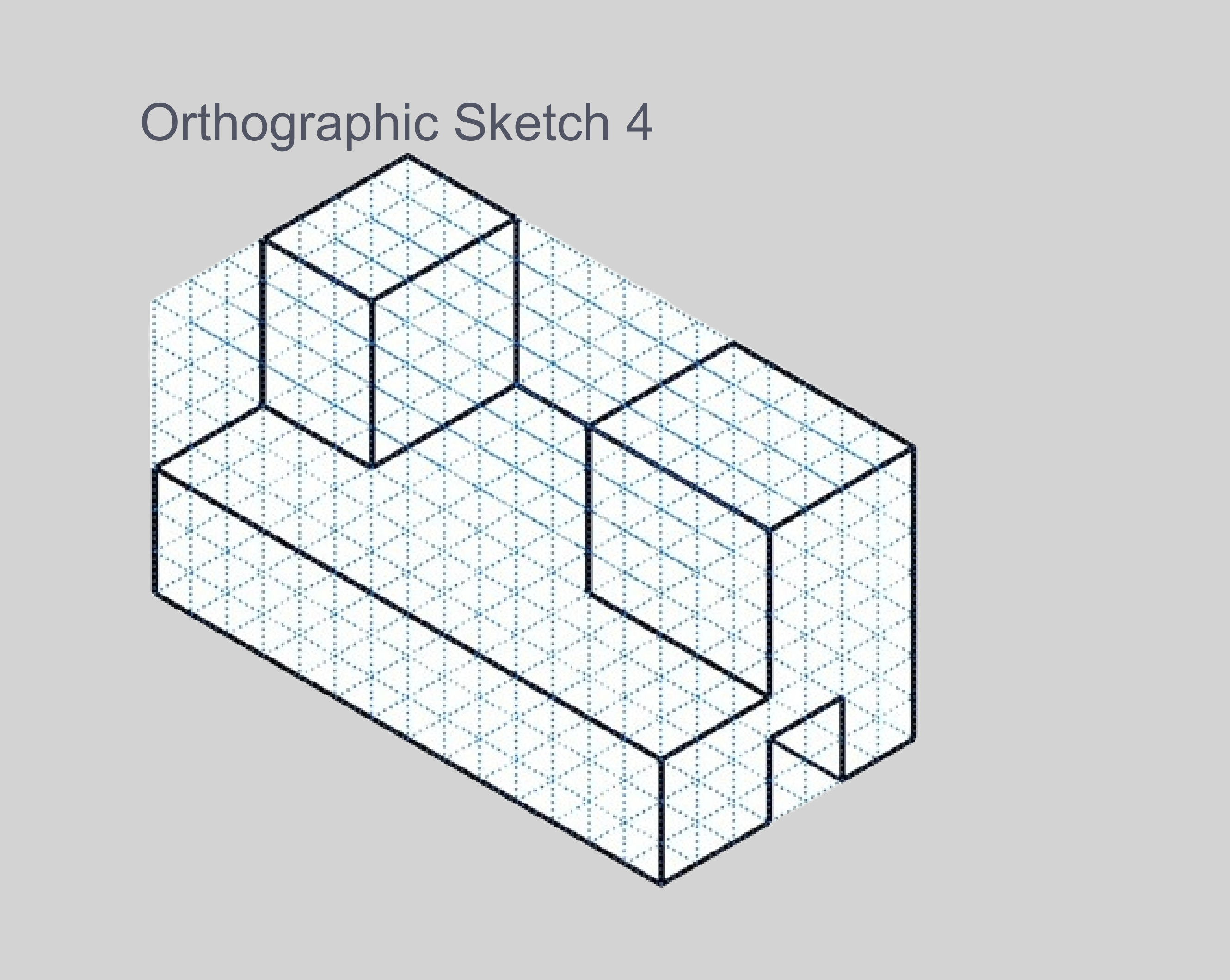




Orthographic Sketch 2

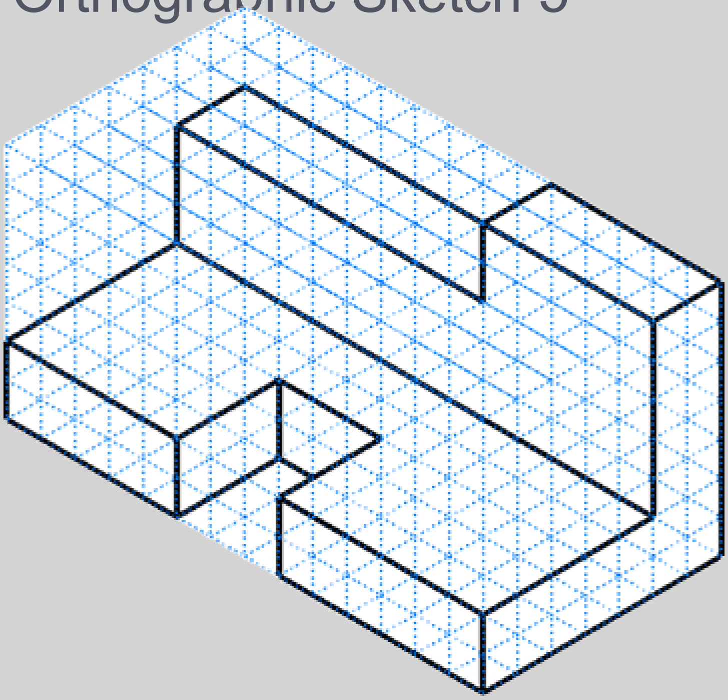


Orthographic Sketch 3

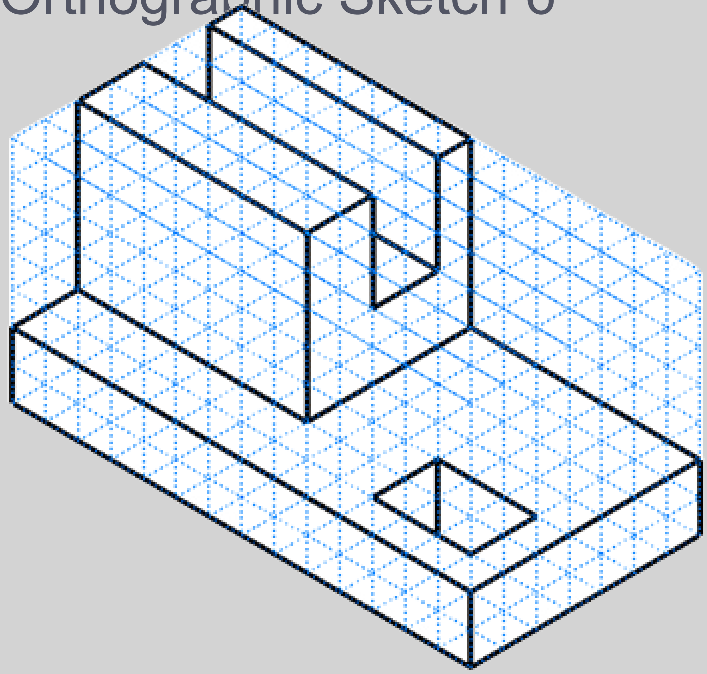


Orthographic Sketch 4

Orthographic Sketch 5



Orthographic Sketch 6



4.4 SKETCHING ISOMETRIC VIEWS

While an orthographic sketch can help communicate an object's details and dimensions, an isometric drawing provides a clear and comprehensive way to visualize a three-dimensional object on a two-dimensional surface. The ability to represent an object three-dimensionally by creating an isometric sketch from orthographic views will also enhance one's ability to visualize complex multi-view drawings quickly.

When it comes to pictorial views of objects, three styles are commonly used: isometric, dimetric, and trimetric views. An isometric, meaning "equal measure," drawing has width, depth, and height features that are all projected 120° from each other, as shown in Figure 4-1. A **dimetric** pictorial drawing will have two of these three angles drawn equal to each other, as shown in Figure 4-2. A **trimetric** drawing will have all three angles unequal to each other, as shown in Figure 4-3.

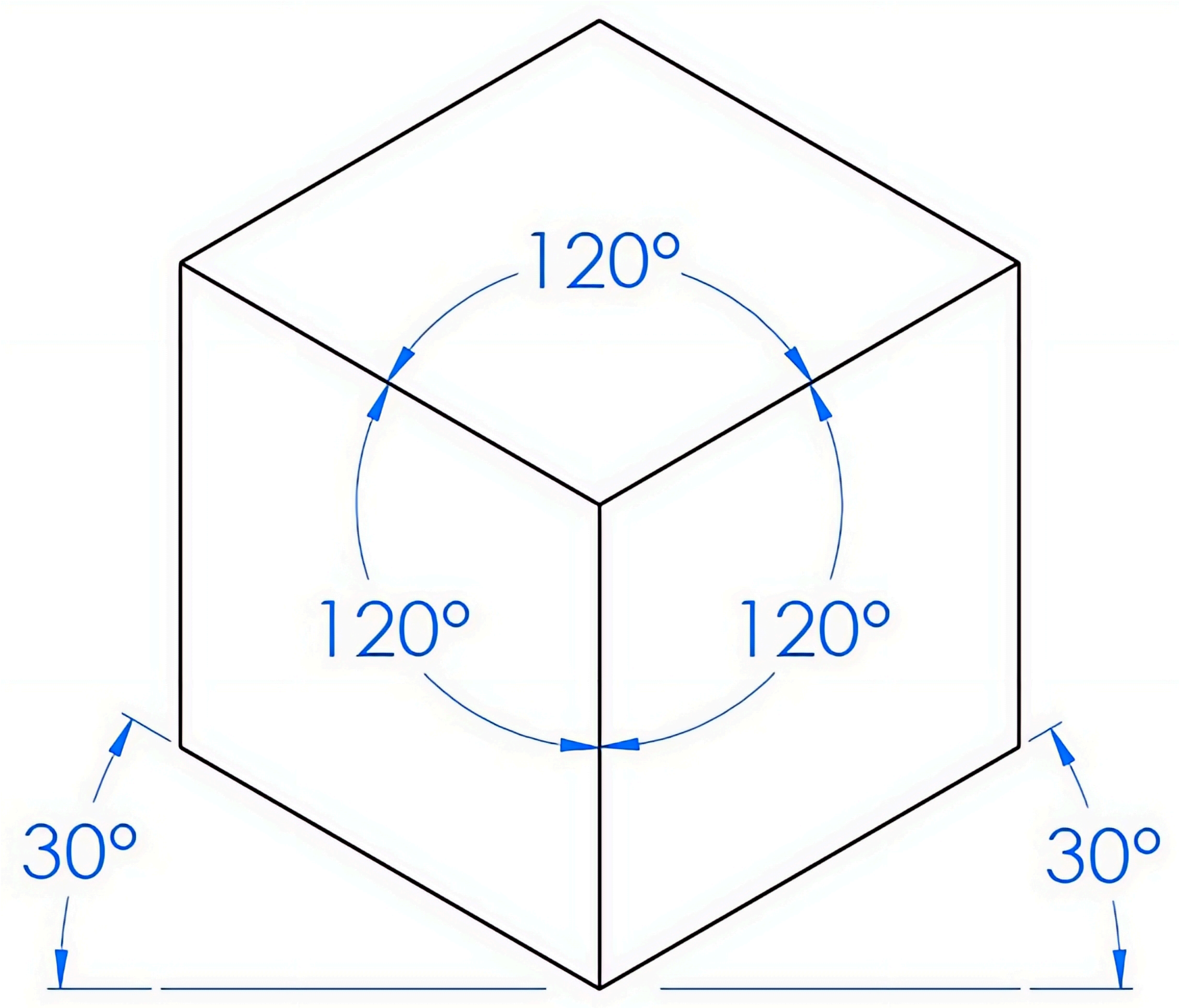


Figure 4-1. Isometric view has all three projections drawn at equal angles to each other.

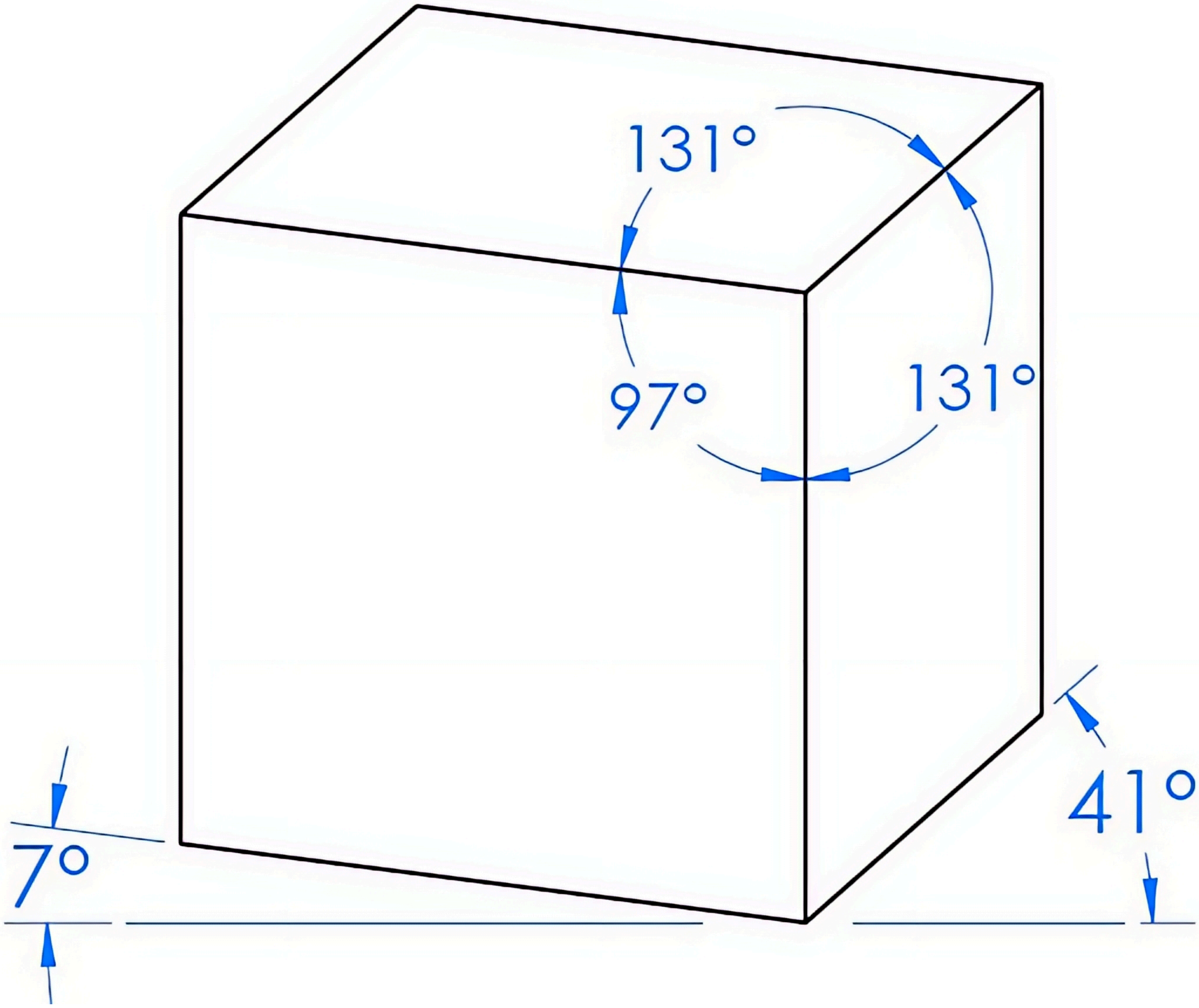


Figure 4-2. Dimetric view has two of the three projections drawn at equal angles.

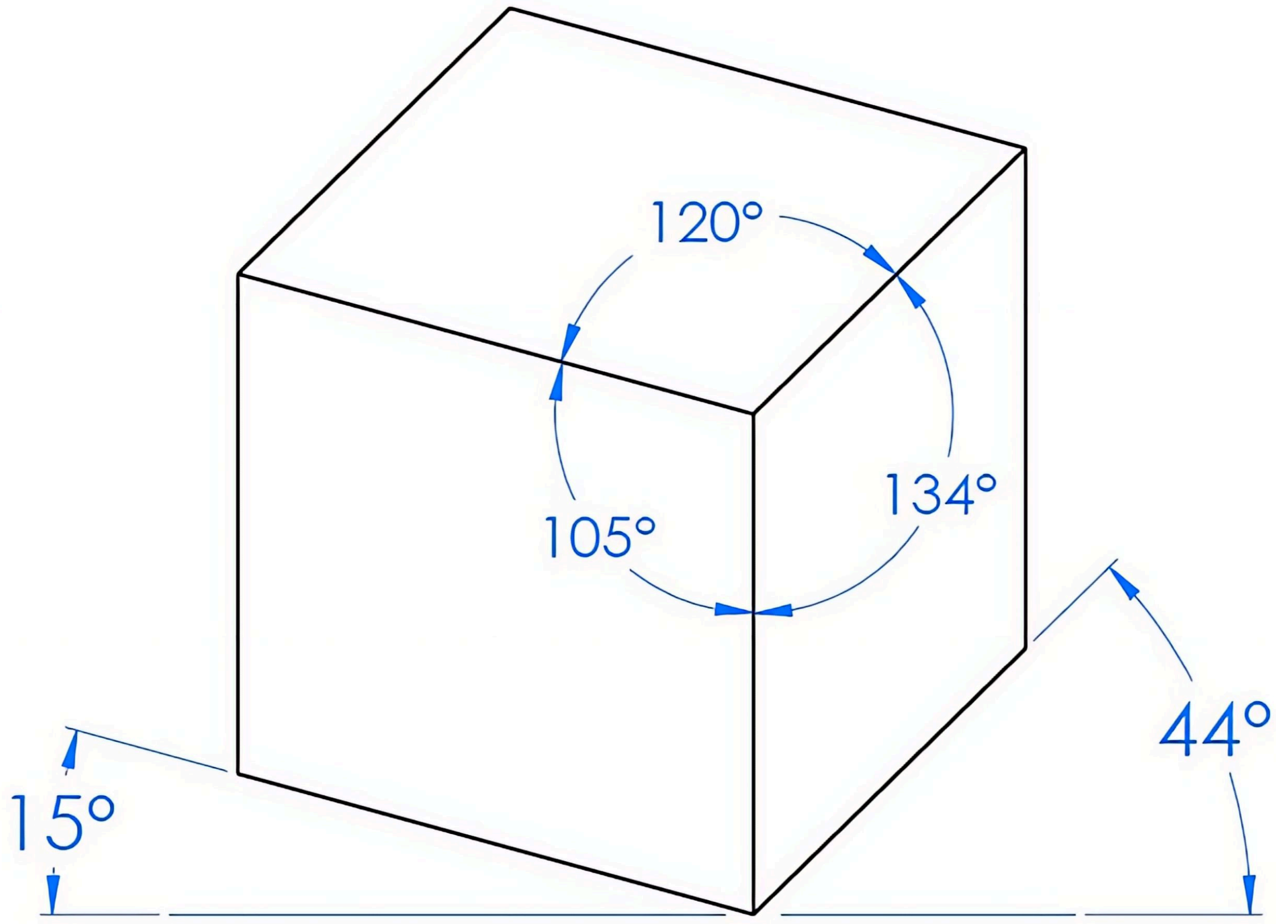


Figure 4-3. Trimetric view has the three projections drawn at unequal angles to each other.

While all three of these pictorial views are sometimes used in computer-aided drafting, the isometric view is most commonly used in sketching. Because the angles are projected equal to each other, isometric sketching can be easier than the other views.

To sketch an isometric drawing view, you can begin in the front corner with one vertical line and lines 30° from the horizontal to the right and the left. See Figure 4-4.

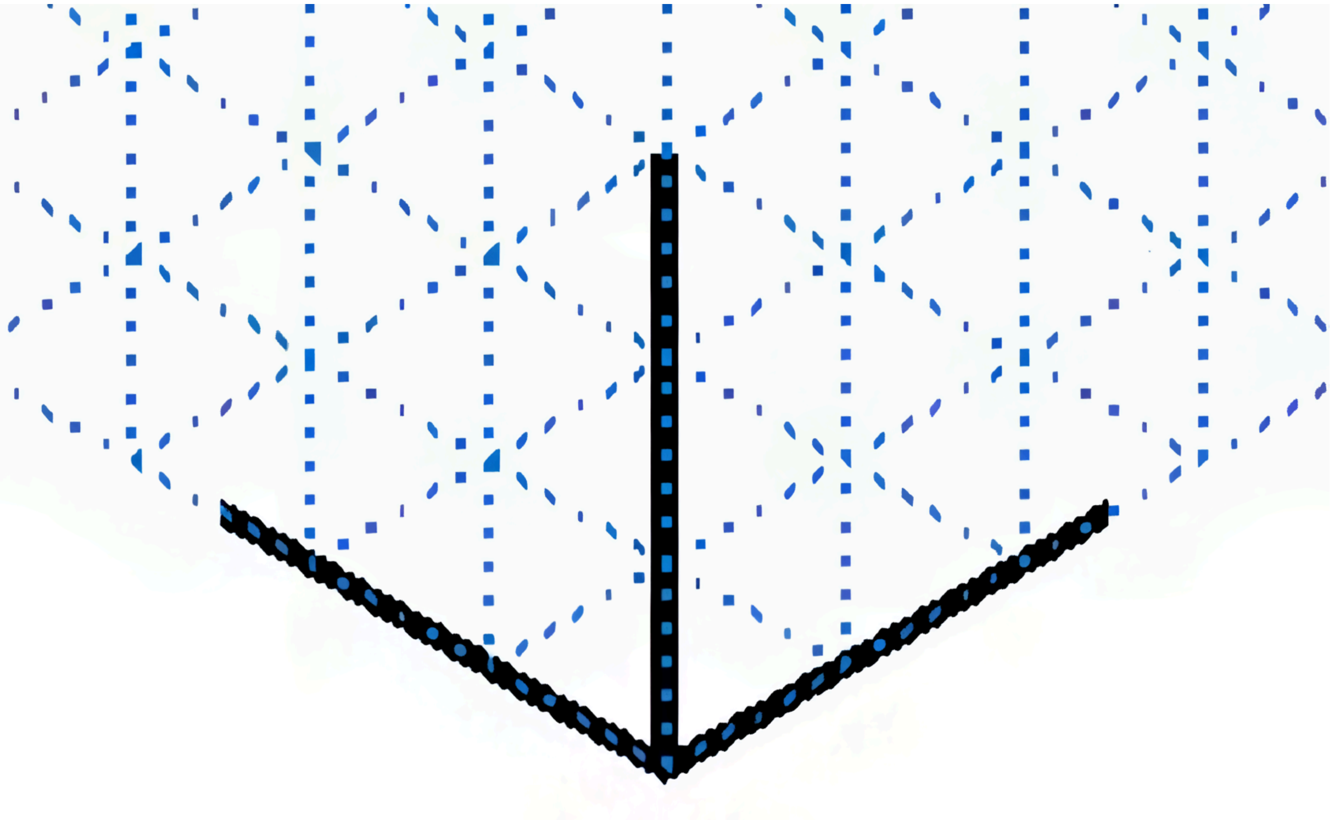


Figure 4-4. Isometric sketch reference lines.

As you sketch, all height lines will be parallel to the vertical line, all right-view horizontal lines will be parallel to the right 30° line, and all front-view horizontal lines will be parallel to the left 30° line. Hidden lines are not typically included in pictorial drawings.

The following YouTube video will illustrate the orthographic views of Isometric Sketch 1 transferred into an isometric sketch.

One or more interactive elements has been excluded from this version of the text. You can view them online here: https://wtcs.pressbooks.pub/blueprintreading/?p=87#oembed-2

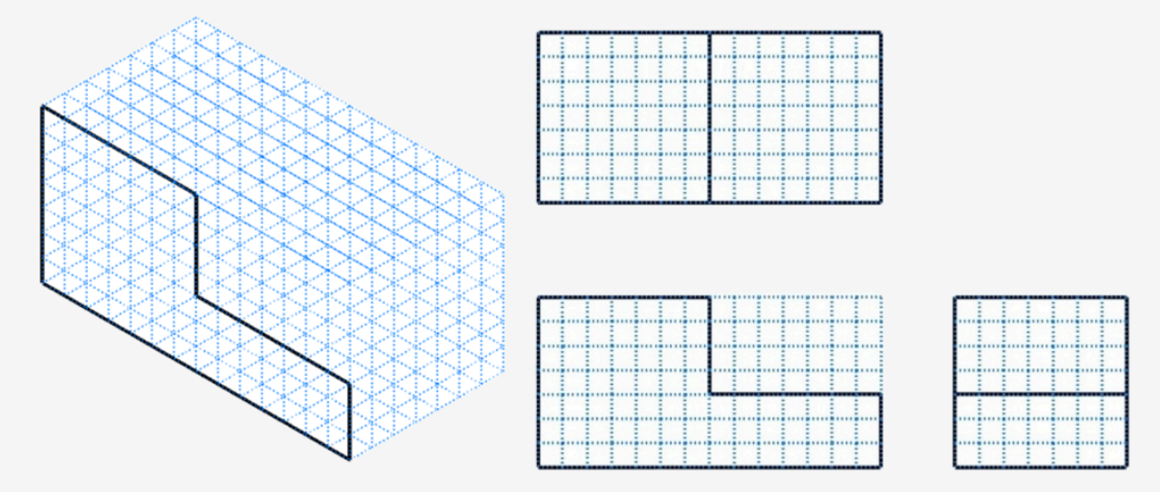
LEARNING ACTIVITIES

Isometric Sketching Exercise

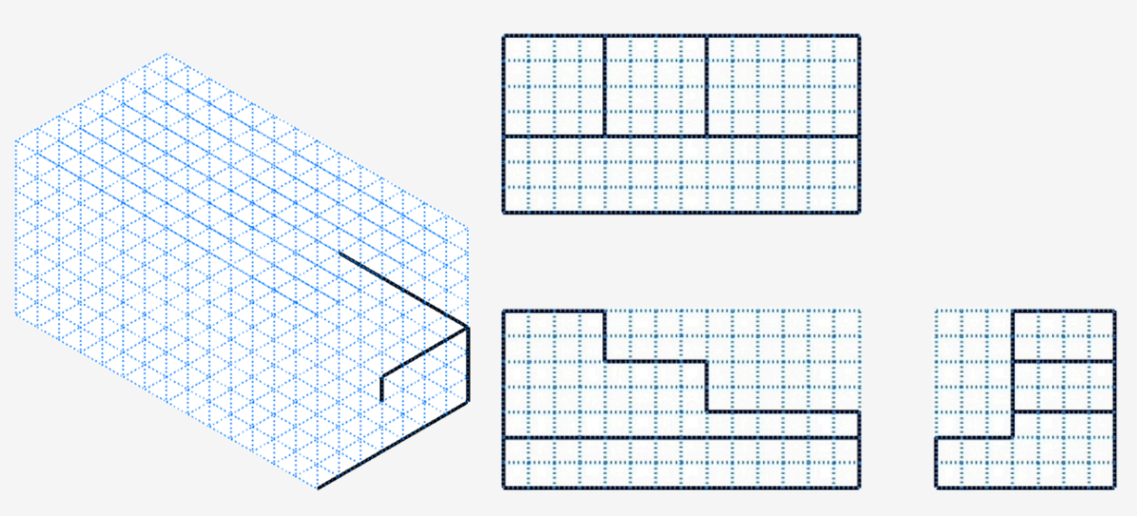
Using your own grid paper, complete the missing isometric views below, using grid lines as a guide. Some sketch lines have been entered to help you get started. Only the visible lines are required. (Solutions are provided in the Learning Activities—Solutions section.)

Isometric Sketching Exercises

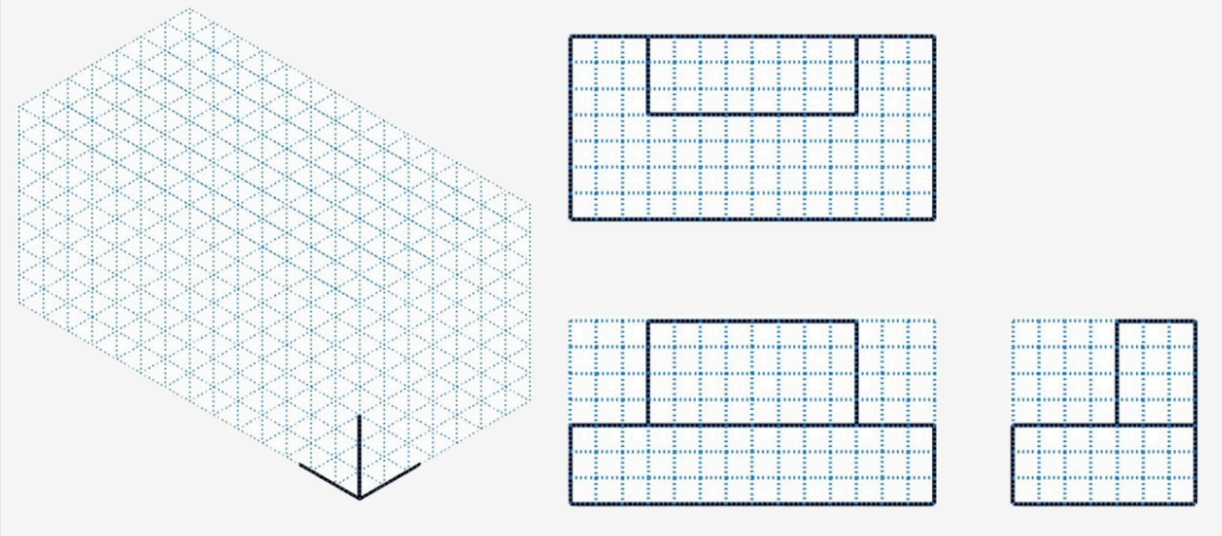
Isometric Sketch 1



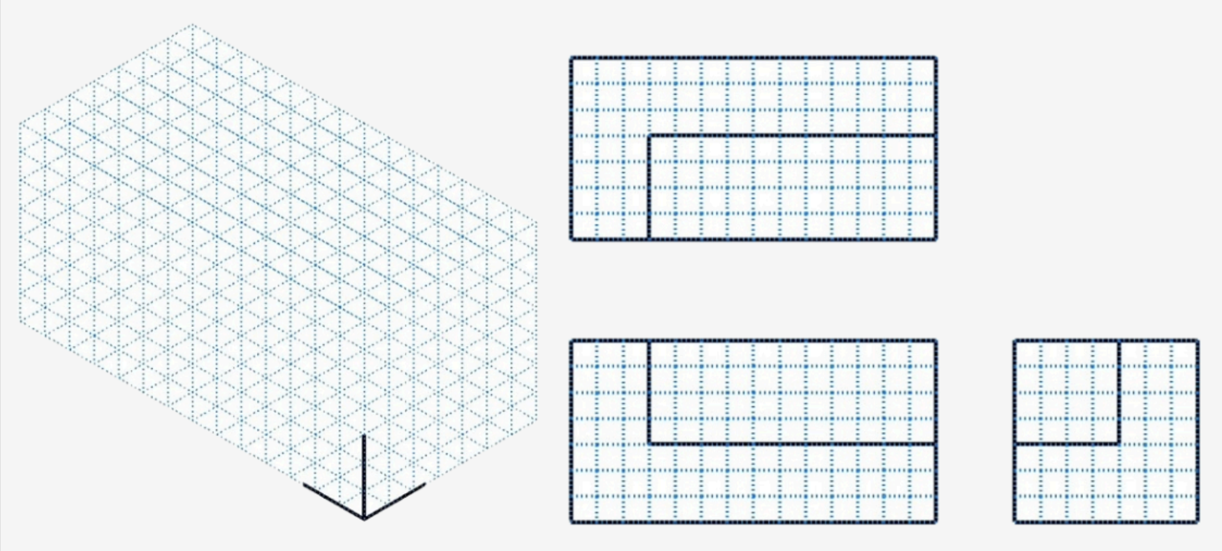
Isometric Sketch 2



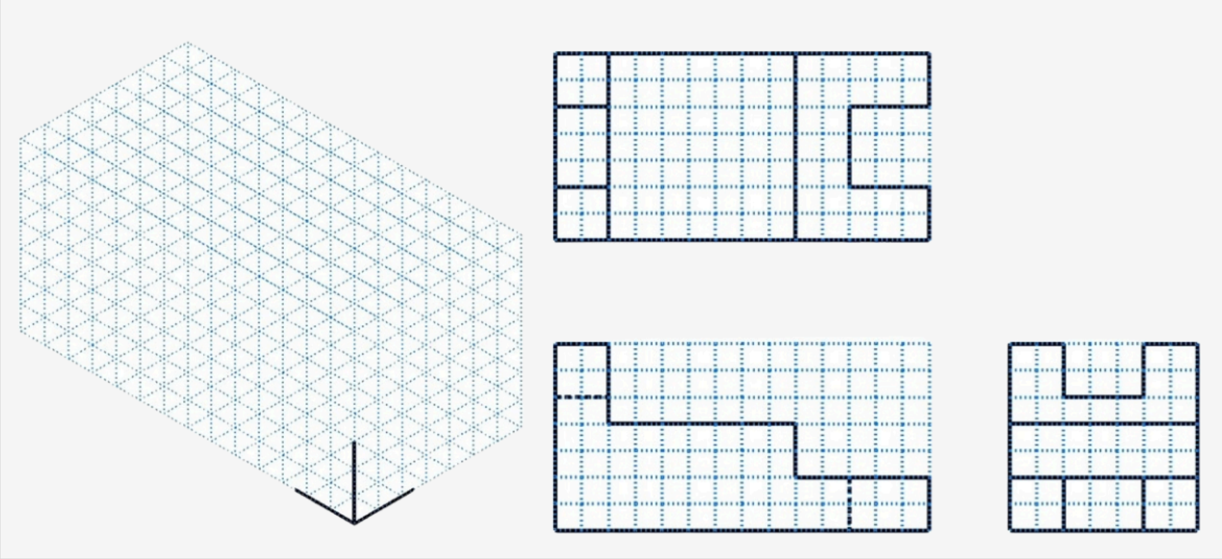
Isometric Sketch 3



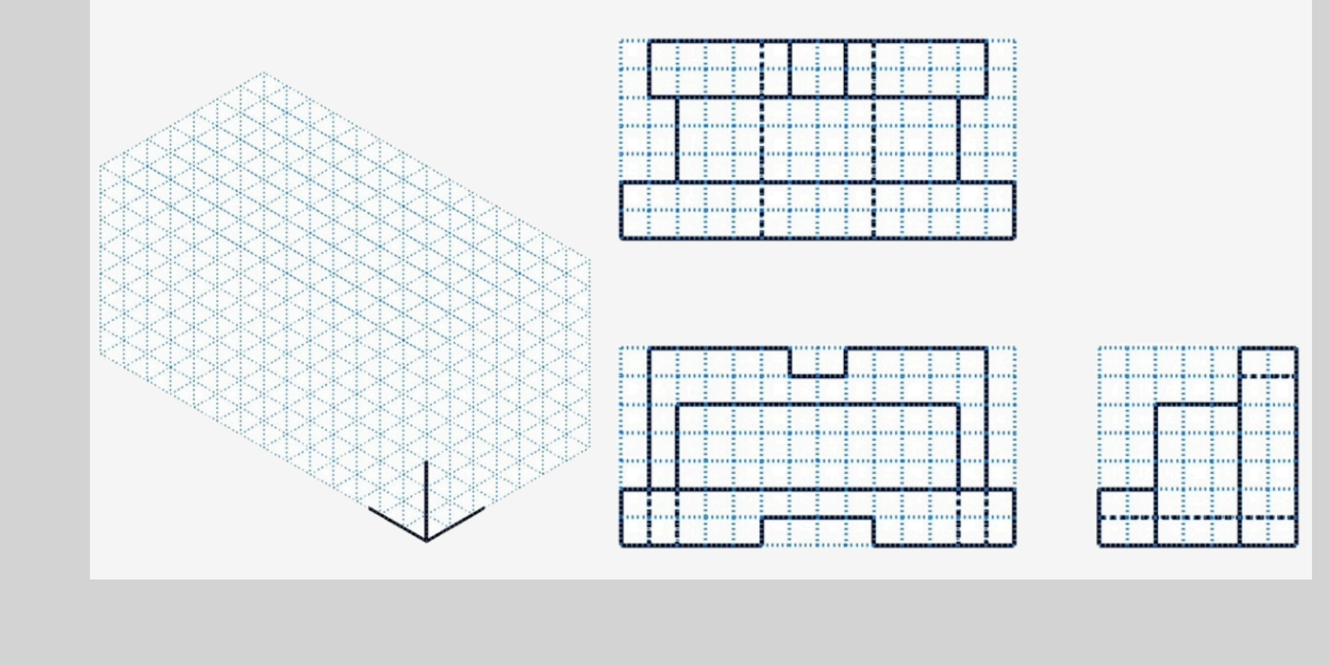
Isometric Sketch 4



Isometric Sketch 5



Isometric Sketch 6



References:

Schultz, R. L., & Smith, L. L. (2012). *Blueprint reading for machine trades* (7th ed.). Pearson.

Images:

All images by Mark Lorier and licensed under CC BY-NC 4.0 except where otherwise noted.

Videos:

LTC Machining. (2024, November 3). *Orthographic sketch exercise 1.* [Video]. YouTube. Used with permission. https://www.youtube.com/watch?v=jAH8C7fXjsE

LTC Machining. (2024, November 3). *Isometric sketch exercise 1.* [Video]. YouTube. Used with permission. https://www.youtube.com/watch?v=oIB2Zbc_MYs

5. Dimensioning Systems

5.1 INTRODUCTION

Learning Objectives

- Describe methods of dimensioning drawings
- Relate dimensioning systems to mathematical concepts
- Explain how baselines are used for locating a part for manufacture
- Explain how coordinate dimensioning is used in CNC machining
- Convert dimensions to other units
- Demonstrate the rules of dimensioning a part
- Calculate the dimensions on a print

Terms

- Dimension
- Tolerance
- Datum
- Datum dimensioning
- Arrowless dimensioning
- Chain dimensioning
- Broken-chain dimensioning

- Tabular dimensioning
- Cartesian coordinate
- Absolute zero
- Decimal inch dimension
- Fractional dimensioning
- Nominal size
- Metric dimensioning
- Dual dimensioning

A **dimension** refers to the specific measurements that define the size, shape, and location of features on a drawing, such as the size and location of holes on a part. These dimensions are essential for accurately interpreting and constructing the design as intended. One key aspect of reading and interpreting technical drawings is interpreting the dimensioning system. This involves assigning numerical values to various dimensions to ensure the final product meets the desired specifications. This chapter will explore the different methods of dimensioning features and the dimensioning techniques that are used.

5.2 DIMENSIONING TECHNIQUES

When we start to explore drawing dimensions, it is essential to grasp some fundamental dimensioning rules and best practices that drafters should follow. Having a basic understanding of these practices will help you gain a better understanding of the drafter's intentions and will assist you in locating the required sizes.

In any detailed drawing, all required dimensions that fully define the part must be present. The printed design should not need to be measured or "scaled." If you, as the print reader, find that a drawing has incomplete or unclear information, seek clarification from the draftsperson or others involved in the design. Assumptions should be avoided.

Additionally, when the drawings, lines, and features are visually aligned or appear perpendicular, they will be in line unless stated otherwise. Lastly, the positioning of a feature's dimensions should be placed on the view in which its profile is most clearly visible. Dimensioning to hidden lines is to be avoided when possible.

A print can be dimensioned in a variety of ways to control the functional sizes of the features and overall size. While all the design's size information should be present on the drawing, all size information must only be presented once to avoid confusion with the **tolerance** or allowable size variation of a dimension; this is further covered in Chapter 8. Because of this, some feature sizes must be calculated from other dimensions.

Datum Dimensioning

A **datum** represents a precise theoretical location, like the edge of a part or the center of a hole, that serves as a reference baseline for measurements. Datum locations are typically chosen by the drafter to best control the function of the part and often to control the relationship between mating parts within an assembly. This datum should provide a consistent reference for part sizes and feature locations. **Datum dimensioning** (also referred to as baseline dimensioning) provides the dimensions or distances from features back to the datum. Figure 5-1 uses datum dimensioning to locate each of the four steps on this part. Each step is referenced from the same left edge of the part using extension and dimension lines. Finding the lengths of the second, third, and fourth steps would each require calculating the difference between two dimensions.

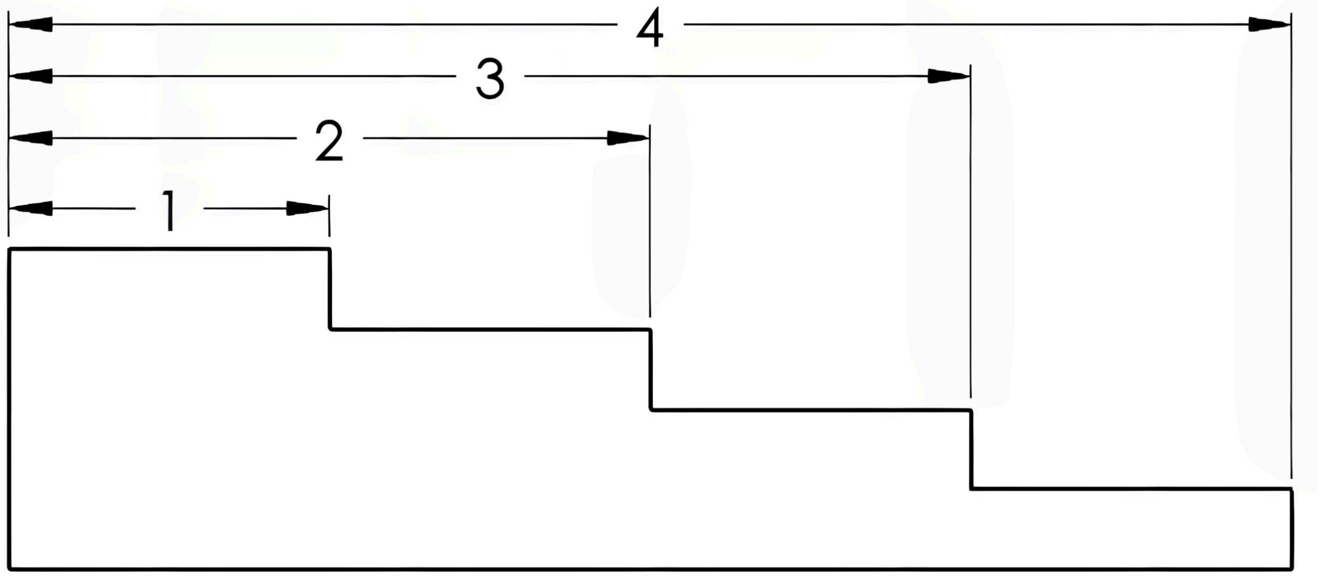


Figure 5-1. Datum dimensioning using dimension and extension lines.

Another style of datum dimensioning that leaves out the dimension lines is known as **arrowless dimensioning**. This style of dimensioning omits these extra lines to help minimize clutter on the print, as shown in Figure 5-2. This style uses only the extension line ending with the dimension presented perpendicular to the line. In arrowless dimensioning, the dimensions will maintain vertical or horizontal alignment with each other. Often these dimensions may be too close to maintain a straight extension line without the dimension values overlapping each other. In this case, the extension line will "bend" outside of the part drawing, as observed in Exercise 5.2-2 near the end of this chapter. Arrowless dimensioning is often used for parts to be manufactured using computer numerical control (CNC) machining processes due to its common reference location and cleaner layout, reducing the difficulty in identifying the correct dimension.

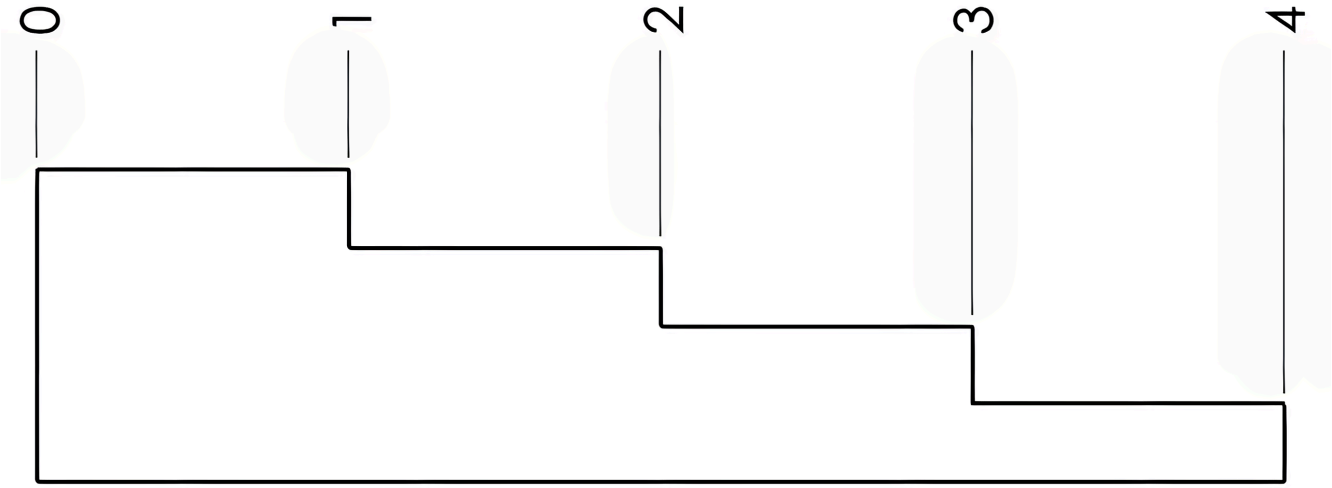


Figure 5-2. Arrowless dimensioning omits the dimension lines.

Chain Dimensioning

Chain dimensioning style involves dimensioning from one feature to another. Rather than locating from one consistent datum, the second feature is referenced from the first, the third referenced from the second, and continuing with any number of features being located from the previous one. Finding the lengths from the end to the second and third steps, as well as the overall length, requires adding the dimensions of each included step. See Figure 5-3 for an illustration of chain dimensioning**.**

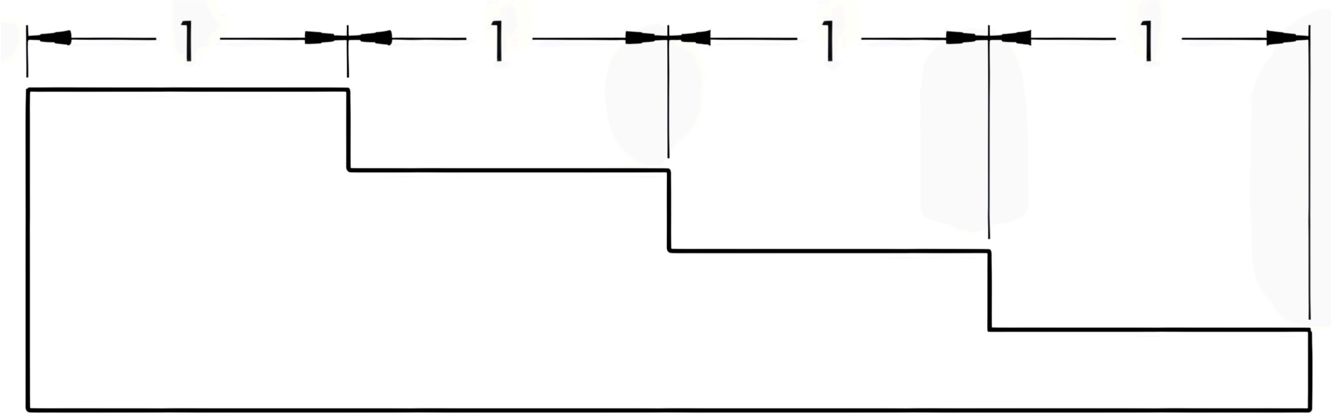


Figure 5-3. Chain dimensioning identifies the length of each step

Broken-chain Dimensioning

The **broken-chain dimensioning** method omits one chain but includes the overall length. The three steps are subtracted from the overall length to find the missing length, as shown in Figure 5-4.

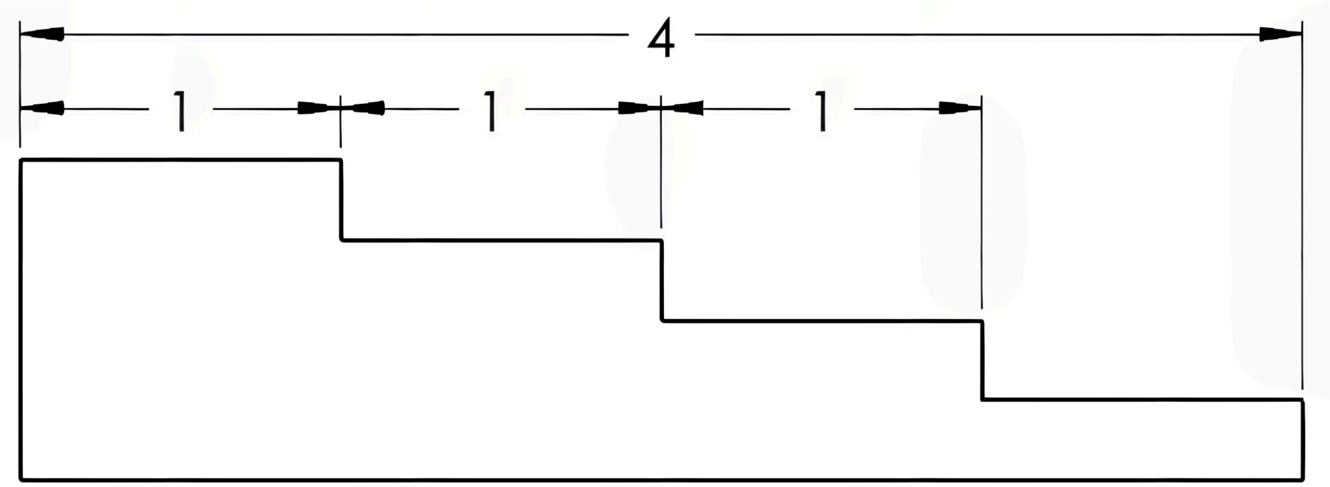
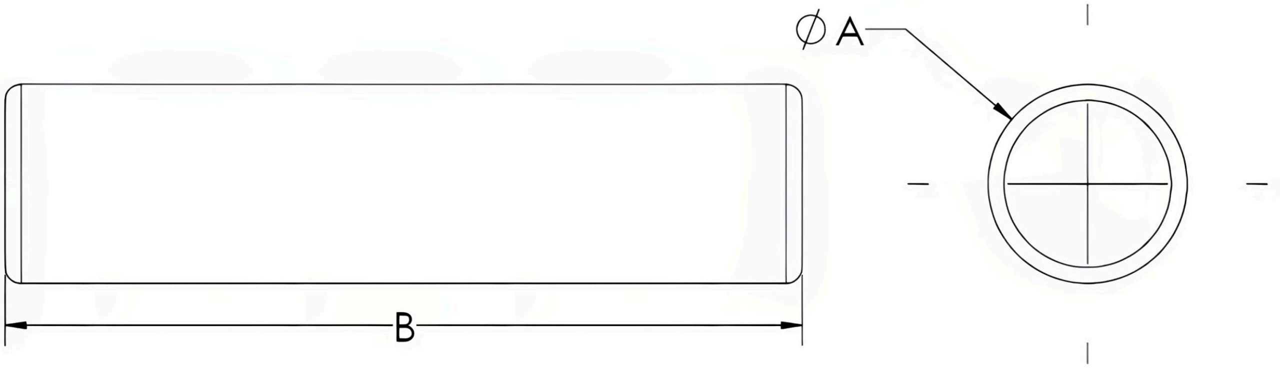


Figure 5-4. Broken-chain dimensioning includes the overall length and omits one of the step dimensions.

Tabular Dimensioning

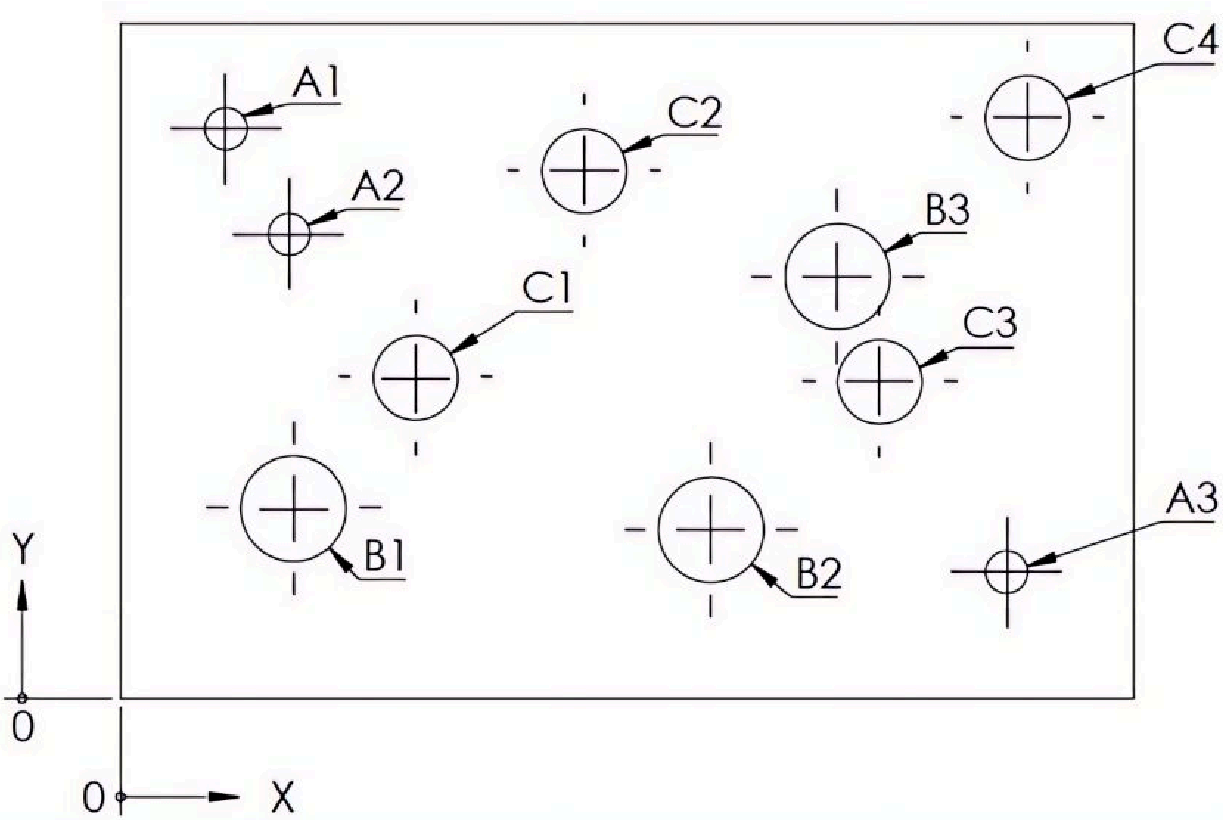
Tabular dimensioning is a system in which letters and/or numbers are used to identify a feature size or location, which is referenced to a table containing the coordinating size dimensions. This system of dimensioning is commonly used when the shape of a part is consistent, but the size dimension may vary among different parts. Some examples of this tabular style may be a bolt or a pin, as shown in Figure 5-5.

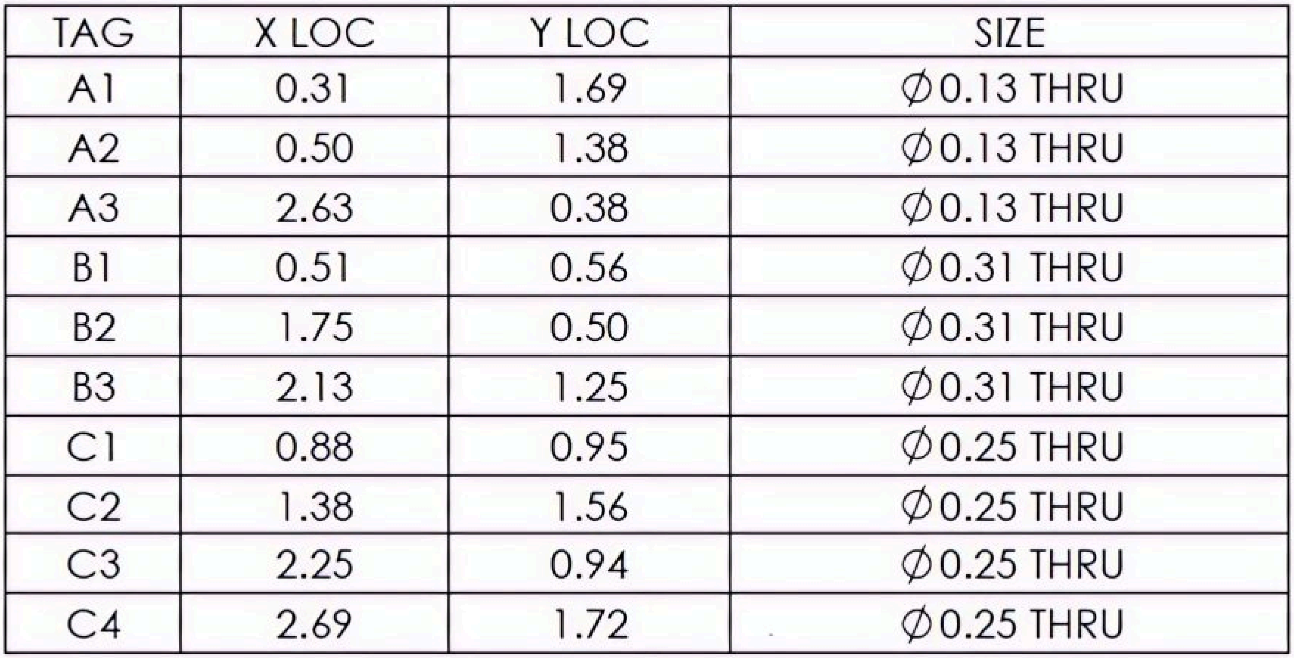


PART #	A	B
1025	.250	.75
1038	.375	1.00
1050	.500	1.25

Figure 5-5. Tabular dimensioning is used for parts similar in shape and of varying sizes.

Another opportunity for the use of tabular dimensioning may occur on a drawing containing a large number of similar features, such as a part containing multiple hole locations. Tabular dimensioning, also referred to as a hole table, in this case, will prevent having an excessive amount of dimension and extension lines, creating a difficult-to-read drawing. The table row will contain an identifying letter and/or number label to correspond with the label or “tag” on the part feature. In Figure 5-6, the hole positions are measured from a specified origin datum and given in X and Y coordinate locations in the table, along with the hole size.





TAG	X LOC	Y LOC	SIZE
A1	0.31	1.69	∅0.13 THRU
A2	0.50	1.38	∅0.13 THRU
A3	2.63	0.38	∅0.13 THRU
B1	0.51	0.56	∅0.31 THRU
B2	1.75	0.50	∅0.31 THRU
B3	2.13	1.25	∅0.31 THRU
C1	0.88	0.95	∅0.25 THRU
C2	1.38	1.56	∅0.25 THRU
C3	2.25	0.94	∅0.25 THRU
C4	2.69	1.72	∅0.25 THRU

Figure 5-6. Tabular dimensioning is used with coordinate locations to reduce the number of lines and information displayed within the part drawing.

The X and Y coordinate positions are known as a **Cartesian coordinate** system. A Cartesian coordinate system (also called a rectangular coordinate system) is an ordered pair of perpendicular lines (axes) using the same unit of length for each axis. The intersection where the axes meet is the origin (0, 0) for each line. In most Cartesian coordinate illustrations, as well as a typical milling machine application, the horizontal line is referred to as the X-axis, and the vertical, as shown two-dimensionally, is referred to as the Y-axis. All X values to the right of the origin are positive; the left values are negative. All Y values above the origin are positive, and values below the origin will be negative. Each coordinate location is identified by its X location first and then its Y location (X, Y). The four points in Figure 5-7 are marked and labeled with their coordinates: (2, 3) in green, (−3, 1) in red, (−1.5, −2.5) in blue, and the origin (0, 0) in purple. This dimensioning style is often used for drawings to be machined on CNC milling machines, with the same datum used on the drawing being used as the absolute zero location of the part in the machine and using the X and Y locations to create a program for manufacturing the part. **Absolute zero** location means that the location values will always be referenced from this location.

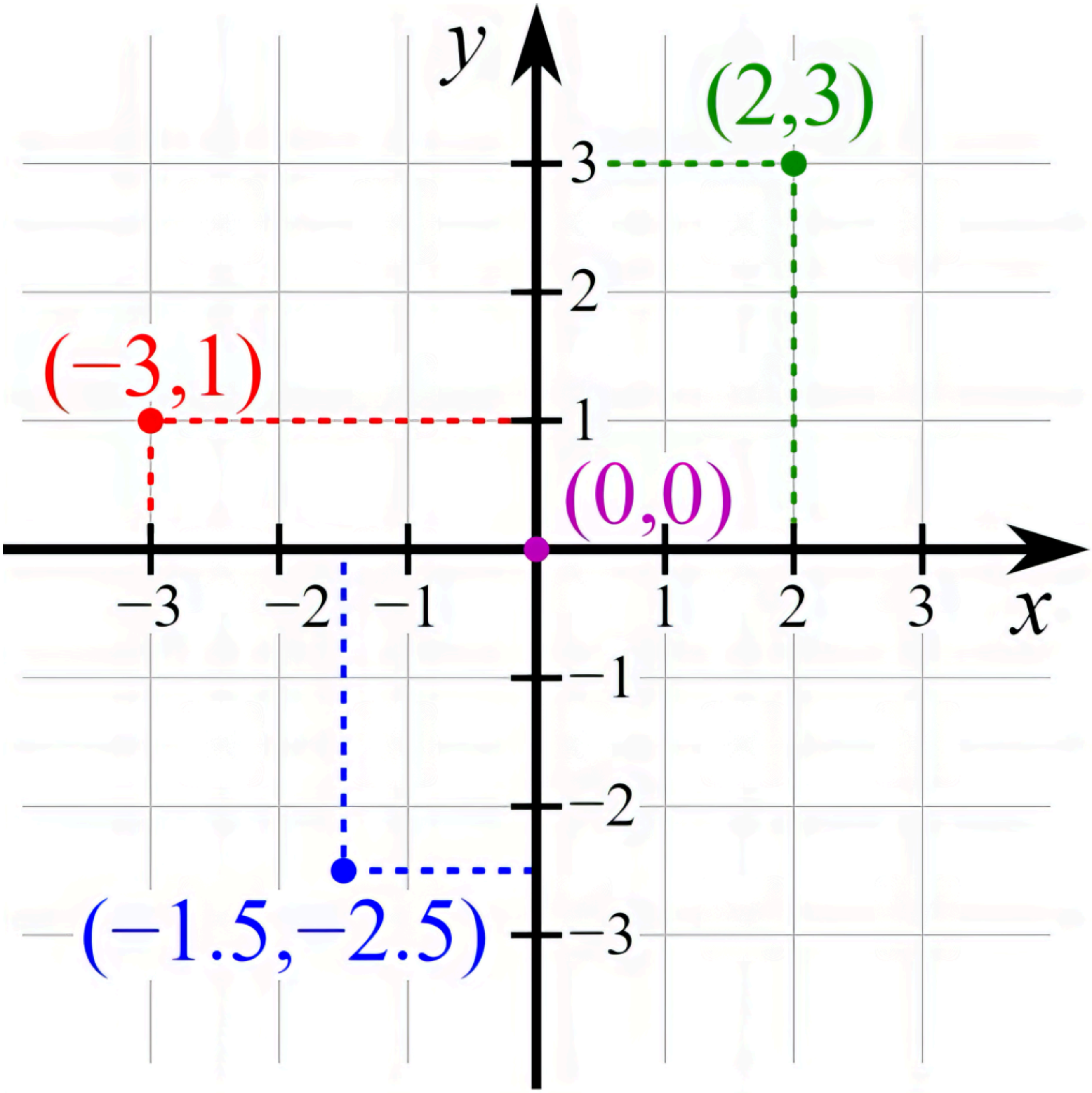


Figure 5-7. Illustration of a Cartesian coordinate plane. Four points are marked and labeled with their coordinates: (2, 3) in green, (−3, 1) in red, (−1.5, −2.5) in blue, and the origin (0, 0) in purple.

5.3 DIMENSION UNITS

There are two primary measurement units commonly used for dimensioning prints. Both imperial (inch) and metric units are frequently used for dimensioning purposes. Inch dimensions, also known as US customary units, are widely adopted across numerous industries and companies within the United States and Canada. In most other countries, metric values represent the standard measurements. Metric values will also appear on US drawings for products intended for international use.

Decimal inch dimensions are based on decimal inches. A decimal inch is a fraction or portion of an inch expressed in decimal form. These are the most commonly used dimension units in the United States. These units are expressed in tenths (.1 = 1/10), hundredths (.01 = 1/100), thousandths (.001 = 1/1000), and ten thousandths (.0001 = 1/10,000) of an inch. Decimal inch values allow for simplified calculations and an increased accuracy over fractional dimensions. The number of decimal places expressed on the print will determine the accuracy required; see Figure 5-8. This concept will be further explained in Chapter 8.

When discussing decimal inch values in a manufacturing setting, the tenths (.1) and hundredths (.01) values are not expressed as they would be in other settings. The values will be expressed in thousandths of an inch regardless of the number of decimal places used. For example, a value of .25 will be expressed as “two-hundred and fifty thousandths” and not “twenty-five hundredths.” When a fourth decimal place (ten-thousandths value) is expressed, the last number is added to the expression. A value of .0625 would be “sixty-two thousandths and five-ten-thousandths” or simplified to “sixty-two and five-tenths.”

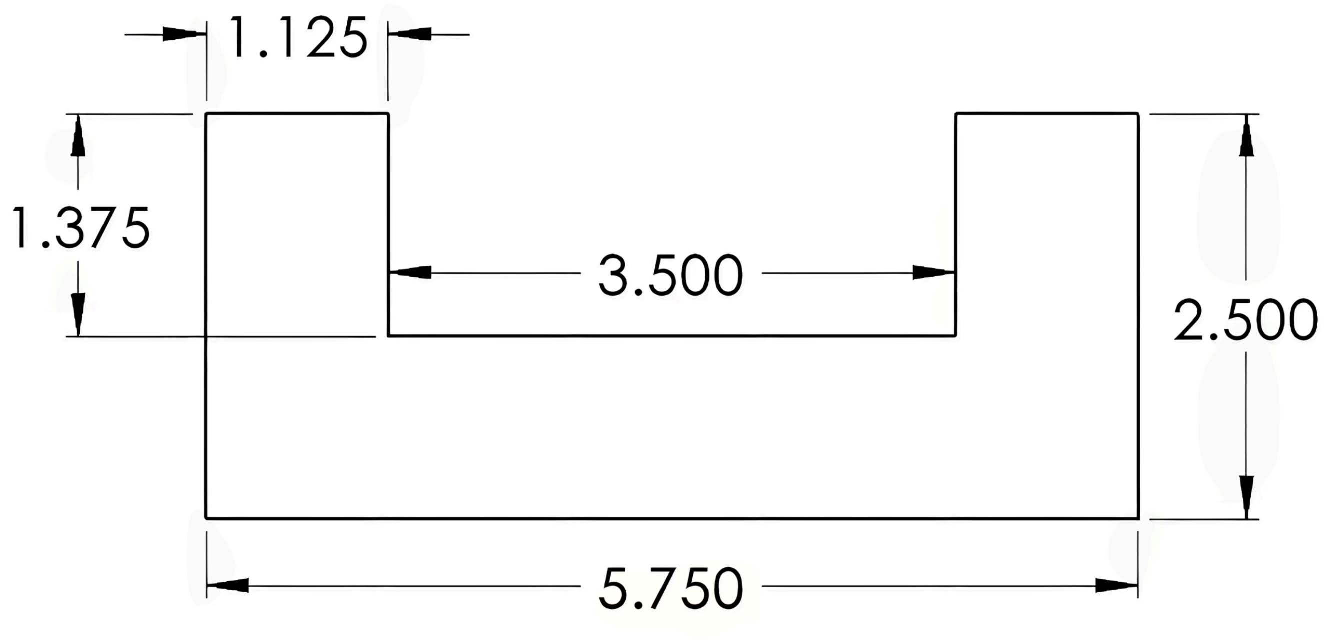


Figure 5-8. Decimal inch values are used for dimensional values.

Fractional inch dimensions are usually measured/represented in values of 1/2 (halves), 1/4 (quarters), 1/8 (eights), 1/16 (sixteenths), 1/32 (thirty-seconds), and 1/64 (sixty-fourths). This dimensioning style may be found on drawings with more generous tolerances such as a casting or a welded fabrication print. Fractional sizes are often used when displaying the material’s stock size or a feature’s nominal fraction size. A **nominal size** refers to a general dimension used for identification and communication purposes, but it does not necessarily reflect the actual physical measurement of the part or feature. One example would be a 1/2-13 bolt, with 1/2" as the nominal outside diameter. However, the actual diameter may vary from .4822" to .500". Another commonly used example is in dimensional lumber, where the finished measured size of a 2"x4" is actually 1 1/2" x 3 1/2". See Figure 5-9.

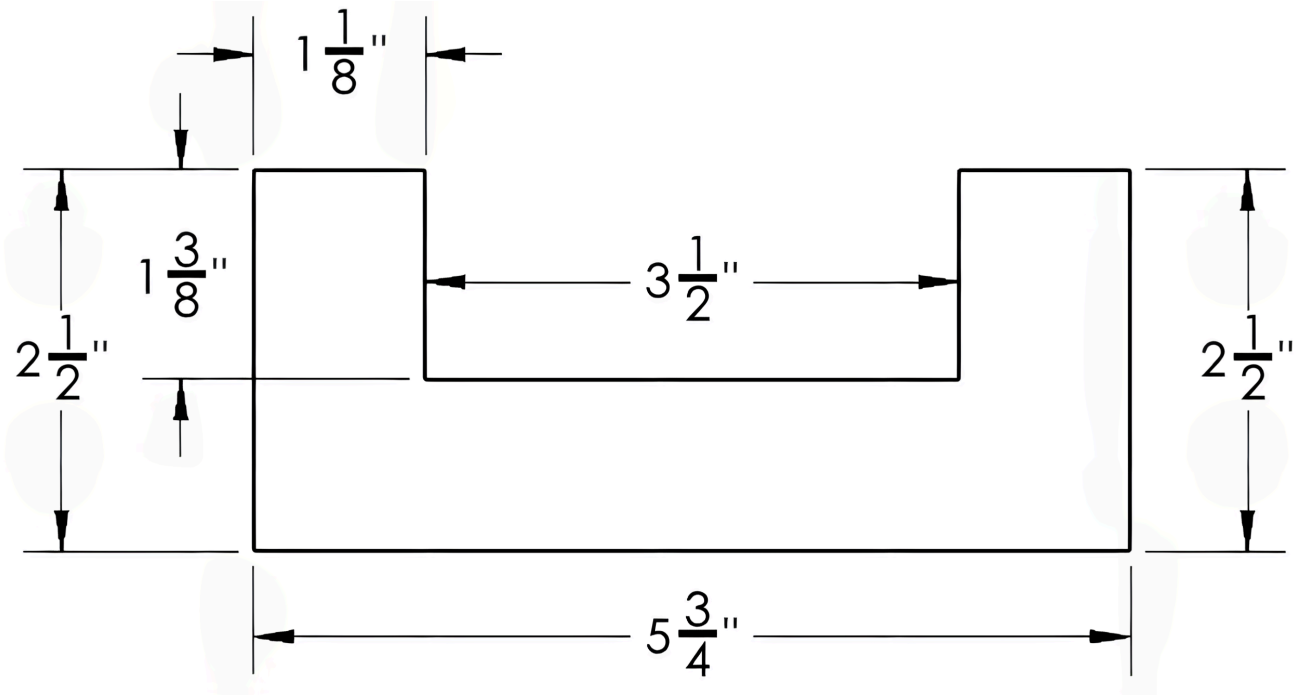


Figure 5-9. Fractional inch values are used for dimensional values.

Metric dimensioning is commonly used in many industrial prints, known as the International System of Units, abbreviated “SI” from its French translation. The meter is the base unit of the metric system, with the millimeter (mm) being the most common metric unit for drawings, as depicted in Figure 5-10. Depending on your environment, equipment, and measuring tools available, a drawing using metric units may require the reader to convert the units into decimal inch equivalents. This can be done by using the ratio of 1 inch equaling 25.4 millimeters.

To convert an inch value to metric, multiply the value by 25.4 (inch value x 25.4 = mm).

Example: *3.5 inches x 25.4 = 88.9 mm*

To convert from millimeters to its inch equivalent, divide by 25.4 (metric value / 25.4 = inch).

Example: *146.05 mm ÷ 25.4 = 5.750 inches*

The number .03937 may be used in place of 25.4, with the formulas reversed: inch value / .03937 = mm and a metric value x .03937 = inch.

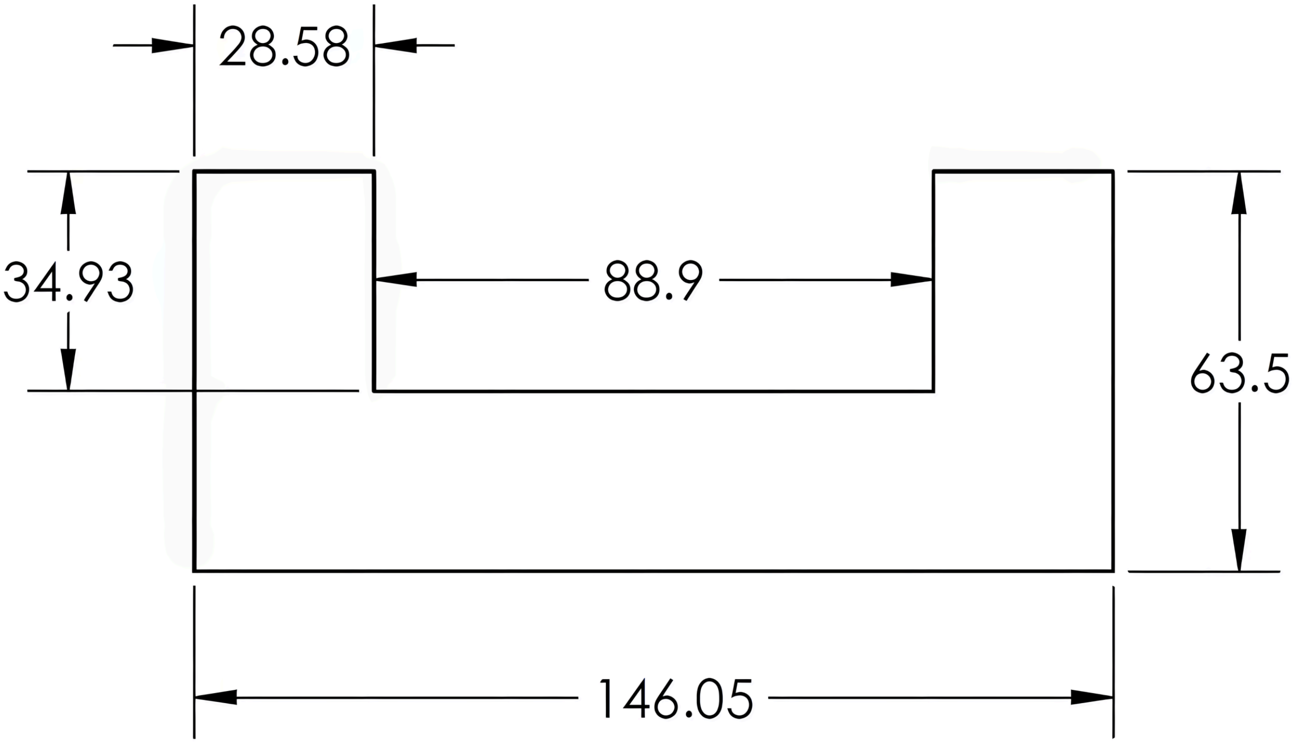


Figure 5-10. Metric millimeter values are used for dimensional values.

Dual dimensioning is a system that displays both inch and metric values together. In dual dimensioning, the original system used is displayed along with the alternate system shown in brackets [ ]. The manufacturing company's preferred unit measurement may not be the same as every print reader involved in the process of the final part. Dual dimensioning eliminates the need to calculate the conversion of each dimension. View Figure 5-11.

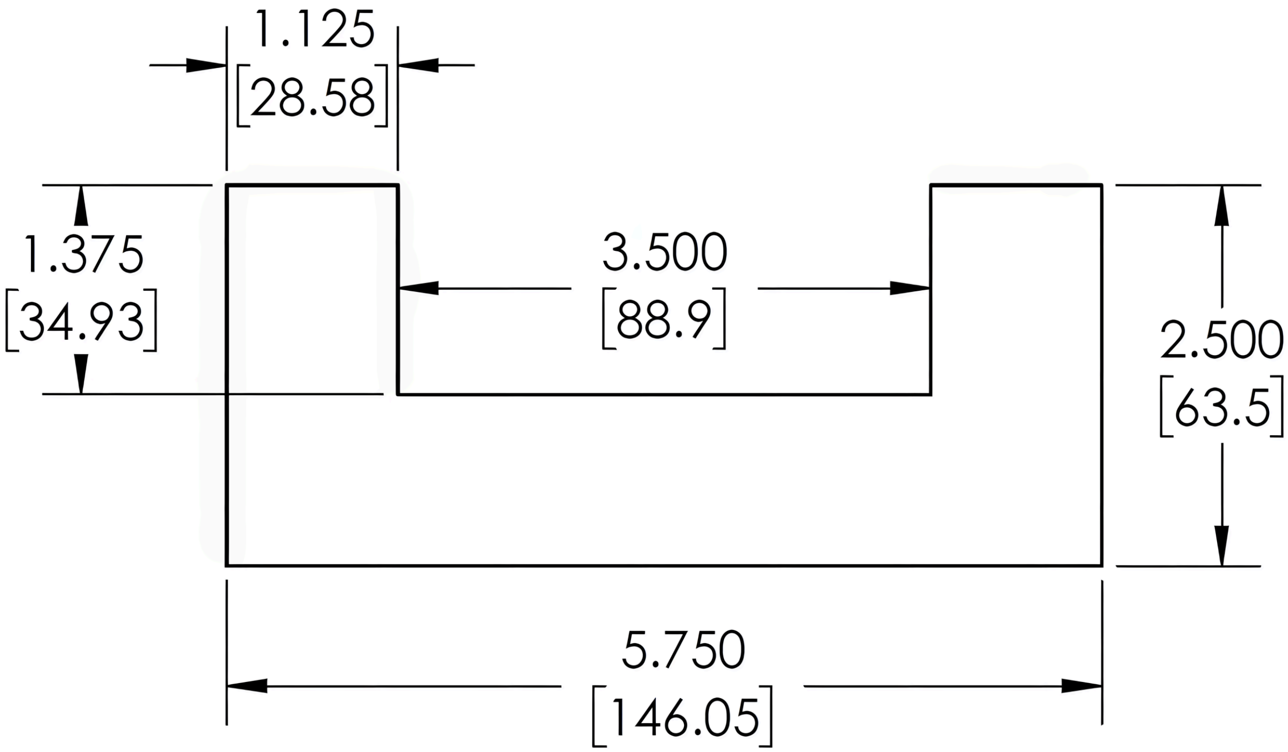


Figure 5-11. Dual dimensioning displays two units of measurement for each value.

LEARNING ACTIVITIES

Exercise 5.2-1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=154#h5p-5

Exercise 5.2-2

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=154#h5p-6

Exercise 5.3-1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=154#h5p-7

Exercise 5.3-2

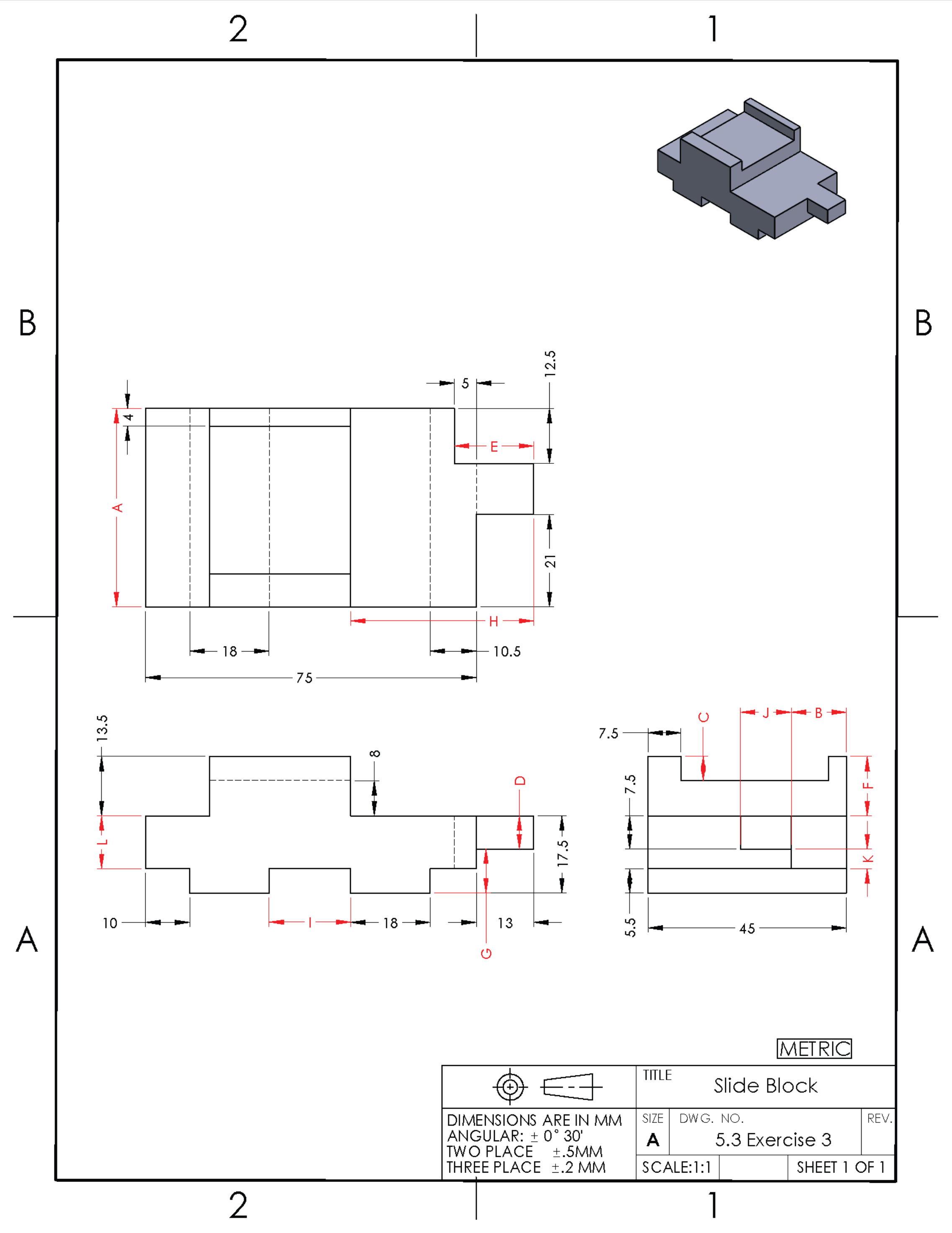
Directions: Click and drag each dimension to the corresponding letter in the print below.

An interactive H5P element has been excluded from this version of the text. You can view it

online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=154#h5p-9

Exercise 5.3-3

Directions: Click the image to open it in a new tab. Reference the image to answer the following questions.


2
1
B
B
A
A
12.5
5
4
A
E
21
H
18
10.5
75
13.5
8
D
L
17.5
10
I
18
13
G
7.5
C
J
B
7.5
F
K
5.5
45
METRIC
TITLE
Slide Block
DIMENSIONS ARE IN MM
ANGULAR: ± 0° 30'
TWO PLACE ±.5MM
THREE PLACE ±.2 MM
SIZE
A
DWG. NO.
5.3 Exercise 3
REV.
SCALE:1:1
SHEET 1 OF 1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=154#h5p-8

References:

Barsamian, M. A., & Gizelbach, R. (2022). *Machine trades print reading*. Goodheart-Willcox.

Schultz, R. L., & Smith, L. L. (2012). *Blueprint reading for machine trades* (7th ed.). Pearson.

Images:

All images by Mark Lorier and licensed under CC BY-NC 4.0 except where otherwise noted.

6. Print Dimensions

6.1 INTRODUCTION

Learning Objectives

- Determine overall part sizes for stock selection
- Apply mathematical concepts to interpret print dimensions
- Determine part feature locations
- Interpret angular measurements on prints
- Calculate missing dimensions
- Interpret part feature sizes
- Identify critical dimensions

Terms

- Radius
- Diameter
- Typical dimensions
- Angular dimensioning
- Degrees, Minutes, Seconds (DMS)
- Decimal degree
- Linear dimensions
- Included angle
- Bolt circle

- Concentric

Interpreting and calculating dimensions on prints is a vital skill in various industries. Interpreting and calculating dimensions for overall stock, missing sizes, radii, diameter, and angular features are common requirements in many drawings.

It requires a thorough understanding of the provided measurements and their implications for manufacturing or assembly processes. By mastering this skill, professionals can ensure the accurate production of components and the successful completion of projects.

6.2 SINGLE-VIEW PRINTS

It is important to note that certain drawings may not necessitate multiple views to convey all essential information regarding the component. For instance, flat parts, such as plates or gaskets, typically require only thickness specifications. The part in Figure 6-1 only requires a single view to communicate the necessary features of the part. The thickness of the part is described in the material box as .063 SS (stainless steel). Round components may only provide a profile view and may indicate their diameter values in the same view. The circular part in Figure 6-2 uses a single view to dimension all of the part's lengths and diameters.

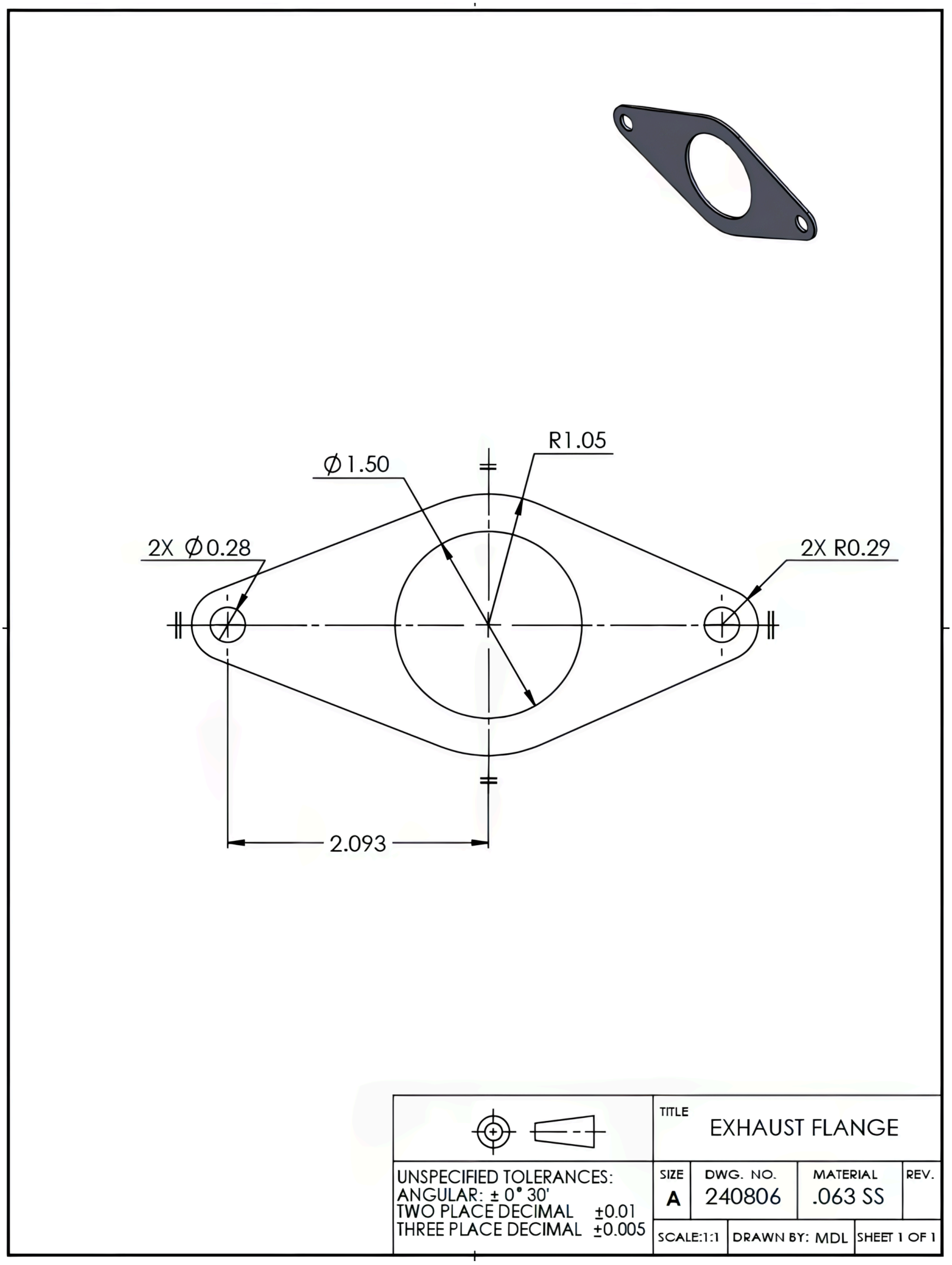

R1.05
∅1.50
2X ∅0.28
2X R0.29
2.093
TITLE
EXHAUST FLANGE
UNSPECIFIED TOLERANCES:
ANGULAR: ± 0° 30'
TWO PLACE DECIMAL ±0.01
THREE PLACE DECIMAL ±0.005
SIZE
A
DWG. NO.
240806
MATERIAL
.063 SS
REV.
SCALE:1:1
DRAWN BY: MDL
SHEET 1 OF 1

Figure 6-1. A single-view drawing is used to fully dimension a flat part with the thickness identified with the material description.

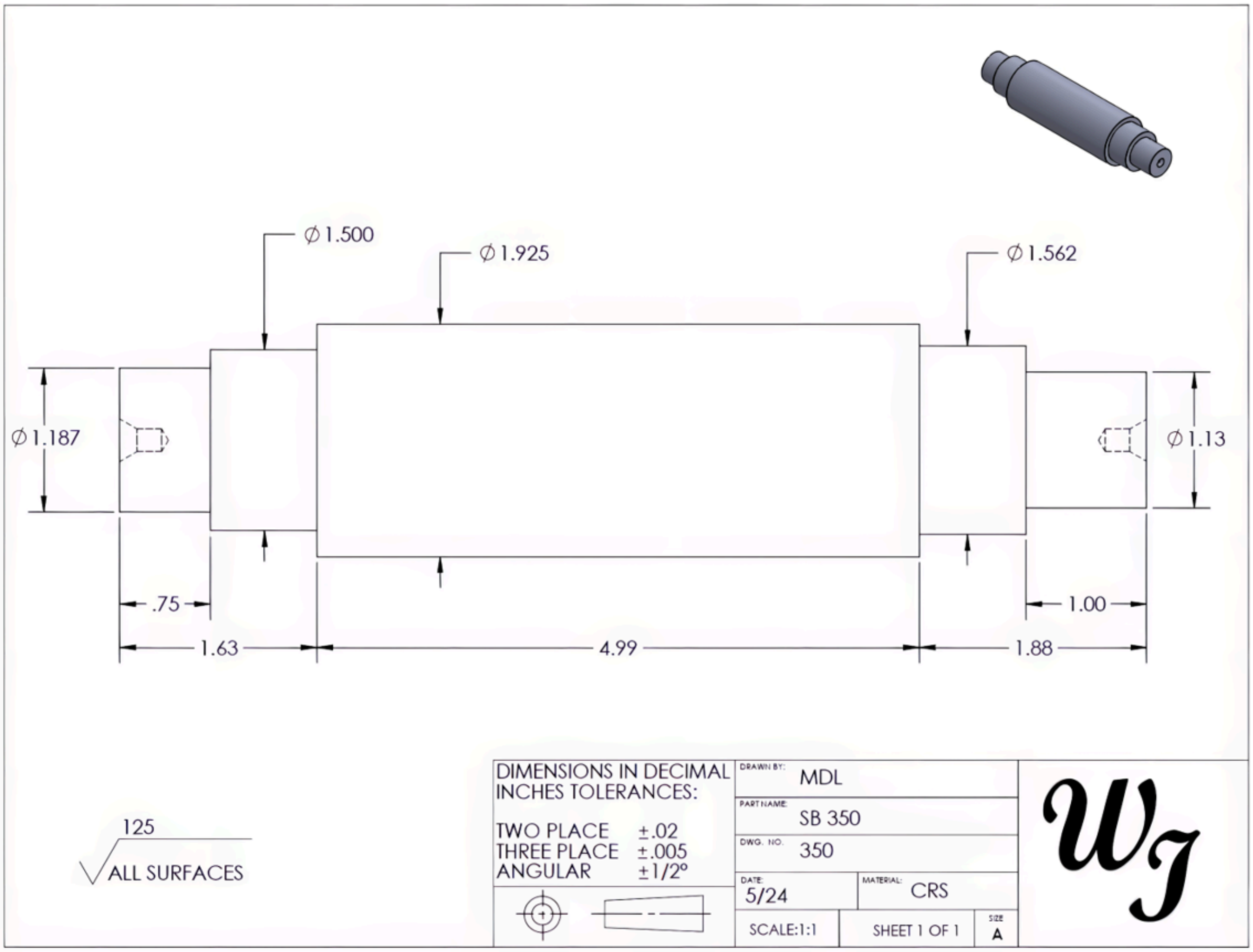


Figure 6-2. A circular object may only require a single-view drawing.

6.3 RADII AND DIAMETER CALCULATIONS

In many technical drawings featuring round or circular elements, you'll find that dimensions are expressed using radius or diameter values. A

radius is the distance from the center point to the outside of a circle or partial circle. A radius measurement is indicated by an "R" and the numerical value. A **diameter** is a straight-line distance across a circle that passes through the center of the circle. A diameter measurement uses the symbol "⌀" with its value. In earlier dimensioning standards, the abbreviation for diameter, "DIA," was used to identify a dimension as a diameter value. Though the abbreviation has been replaced, it may still be seen on many drawings.

It's common to use these radius and diameter values to determine additional dimensions. For instance, in the examples provided below, we will see how these values will be used to calculate wall thickness. To calculate for "X" in Figure 6-3, the small radius (0.25) is subtracted from the larger radius (0.50), giving a difference of 0.25 for the X value. When working with radii, you simply subtract the smaller radius from the larger one.

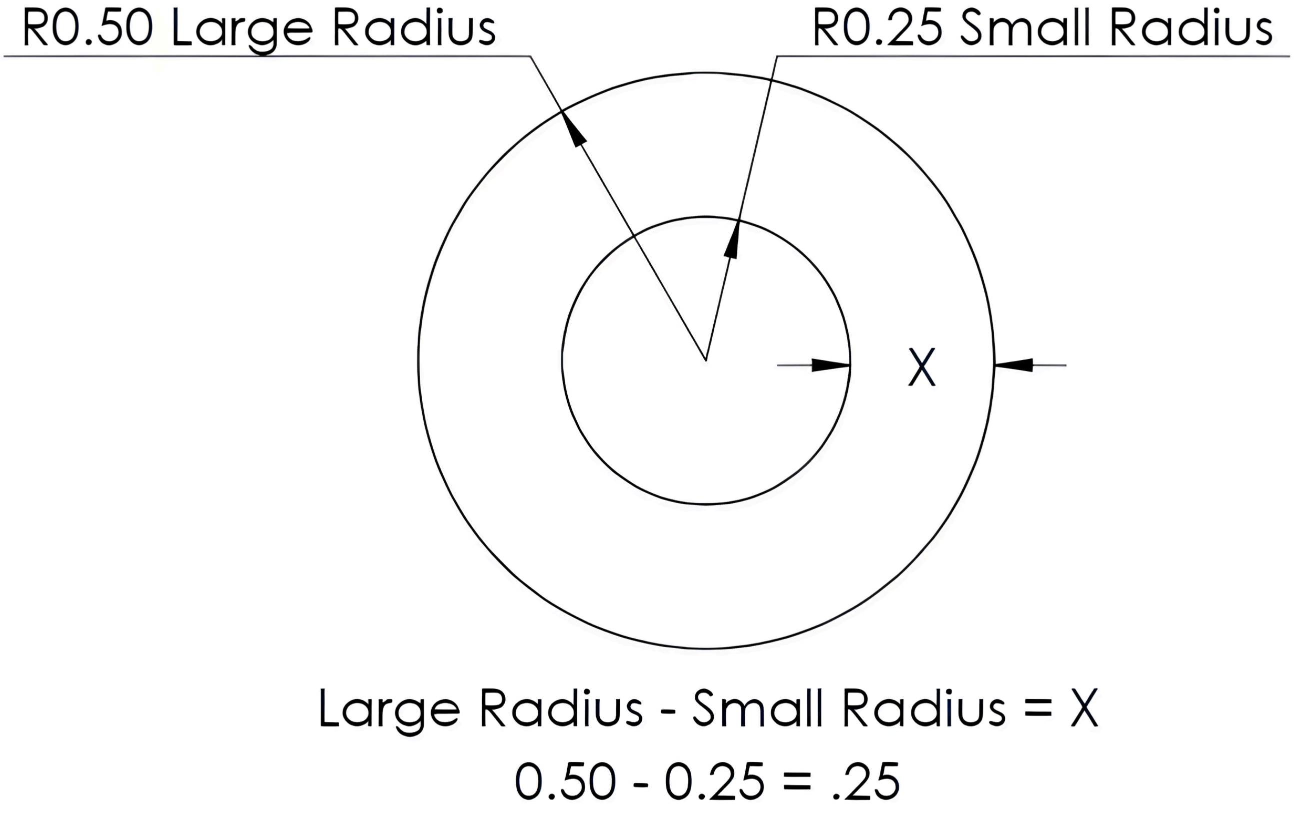


Figure 6-3. To calculate the value for X, given the large and small radii values, the smaller radius is subtracted from the larger radius.

On the other hand, if you're dealing with diameters, remember to divide your result by two after subtraction. To calculate for "X" in Figure 6-4, the small diameter (0.50) is subtracted from the larger diameter (1.00). The difference is then divided by 2, resulting in the .25 value for the X dimension.

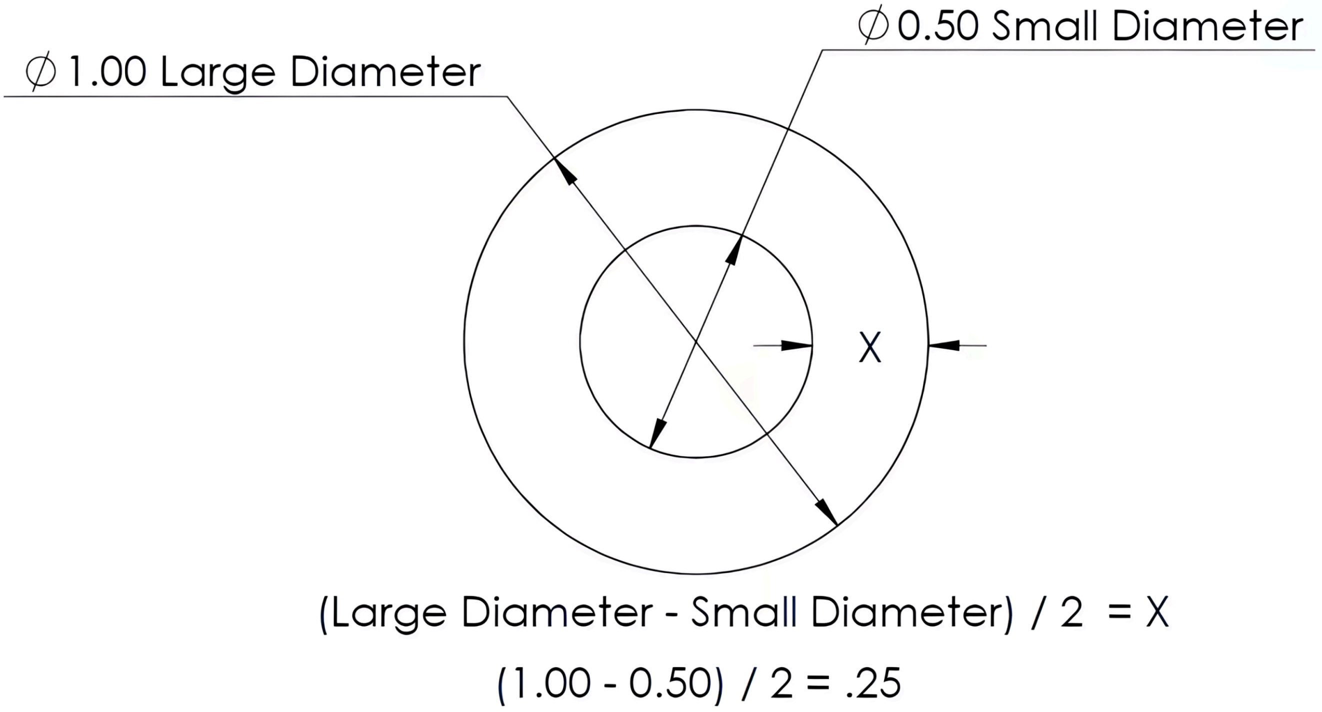


Figure 6-4. Given the large and small diameter values, the value for X is calculated by subtracting the smaller diameter from the larger diameter and then dividing it by 2 since a diameter is twice the size of the radial value needed here.

As you progress through this material, please pay close attention to which values you use for calculations. This will ensure accuracy in your work.

6.4 REPETITIVE FEATURES

Repetitive "X" Dimensions

When a particular feature appears multiple times on a drawing, its details may be provided just once. The frequency of that feature will then be indicated with the letter “X” to designate “times.” Figure 6-5 uses 2X to identify the two identical angles as both 60° and 5X to identify the five smaller holes as identical.

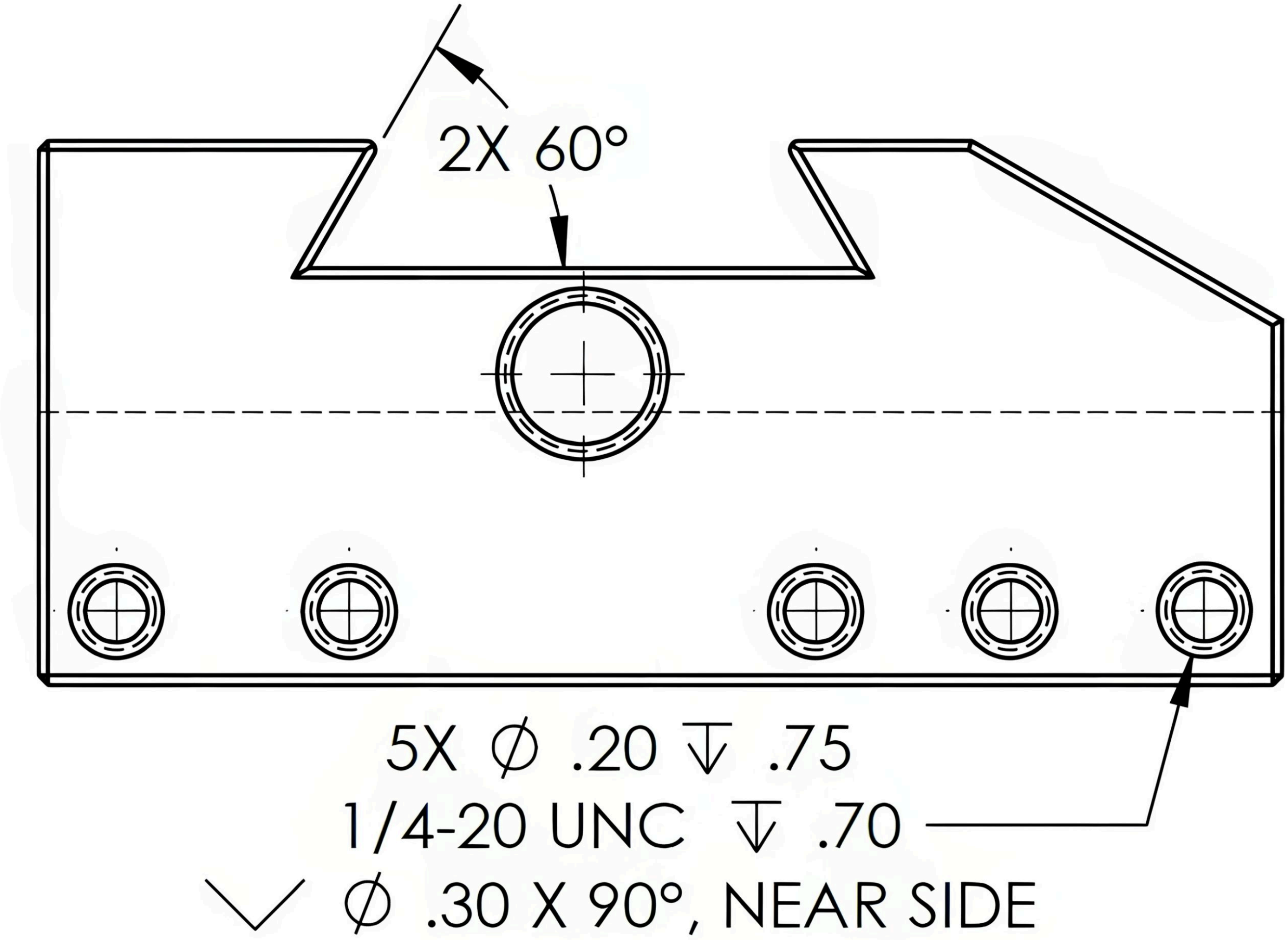


Figure 6-5. “X” used to identify the number of times a feature is repeated.

Typical Dimensions

Typical dimensions are repetitive features identified by the abbreviation “TYP” for typical. For a location dimension, TYP can appear after a dimension to apply the same value to remaining undimensioned locations, as shown in Figure 6-6. This abbreviation is also applied to features such as radii or chamfers; all undimensioned features of similar size are intended to use the same size value as the TYP dimension.

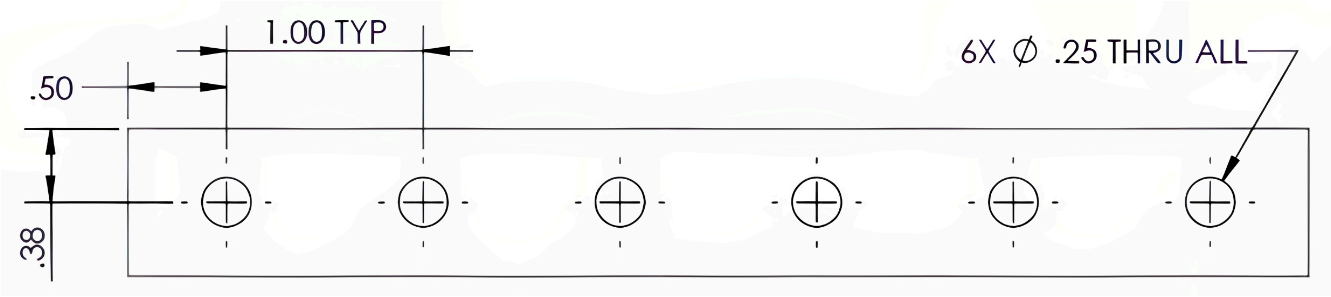


Figure 6-6. The abbreviation for typical (TYP) is used to avoid repetitive location dimensions.

Exercise 6.4-1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=186#h5p-10

Exercise 6.4-2

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=186#h5p-11

Exercise 6.4-3

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=186#h5p-12

Exercise 6.4-4

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=186#h5p-13

6.5 ANGULAR DIMENSIONING

Angular dimensioning is used to identify the angular size from a known reference. In most cases, the angle will be dimensioned from the vertical or horizontal surface or centerline. Angular dimensions will be expressed in degrees (°). A complete circle contains 360°. In an angular dimension, the dimension line will be curved with the curve's center point (vertex) matching the center point of the angle. When a degree dimension is further split into a smaller portion of a degree, this may be expressed in two different ways: in degrees, minutes, and seconds; or in decimal degrees.

Degrees, Minutes, and Seconds (DMS) is a system for measuring angles using degrees (°), minutes (‘), and seconds (“). In this system, a degree is divided into 60 minutes, and each minute is divided into 60 seconds. The format for expressing DMS is degrees° minutes’ seconds”. For example, the angle could be expressed as 40° 26' 30" or 40 degrees, 26 minutes, and 30 seconds. See Figure 6-7.

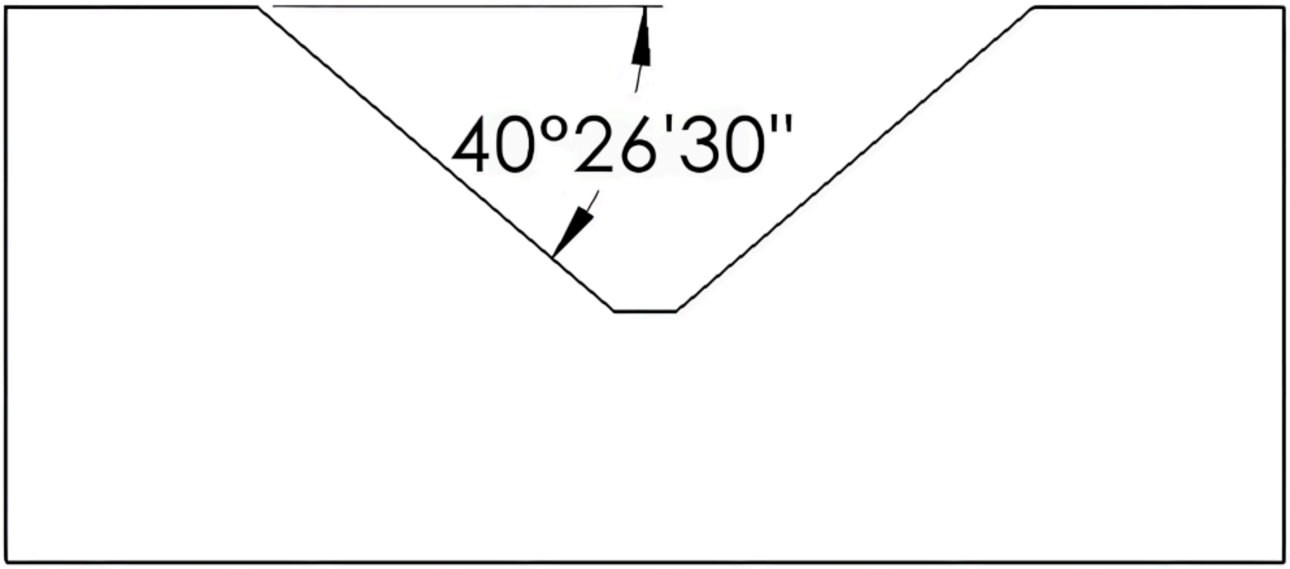


Figure 6-7. Degrees, minutes, and seconds are used to identify an angular dimension.

Decimal degrees is a system for measuring angles or coordinates using decimal fractions of a degree. The formula displays how the conversion is made. To convert minutes, you take the number of minutes and divide it by 60 (because there are 60 minutes in a degree). For seconds, you divide the number of seconds by 3,600 (60 seconds x 60 minutes = 3,600 seconds within a degree). After that, you simply add both results together to get your decimal value.

For example, let’s say we have an angle of 40° 26' 30". You would calculate it as follows: Take the minutes (26) and divide that by 60 to get approximately .433. Then, for the seconds (30), divide that by 3,600, resulting in about .0083.

40° + .433 + .0083

The resulting decimal degree angle is 40.4417°. View Figure 6-8.

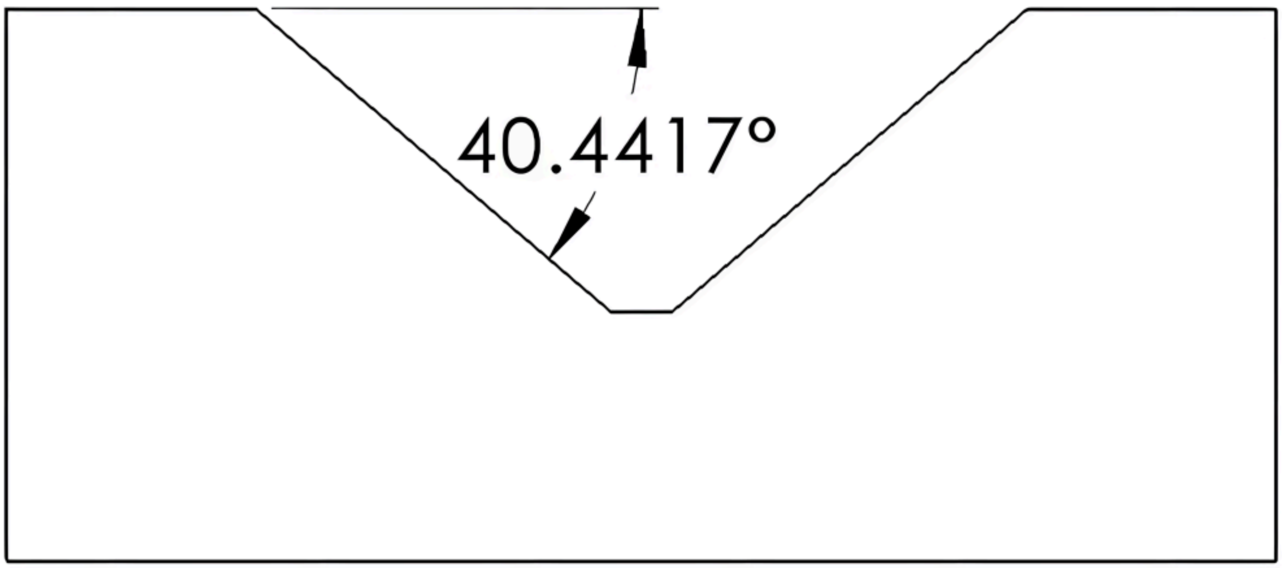


Figure 6-8. Decimal degrees are used to identify an angular dimension.

Both DMS and decimal degrees are used in industrial prints to accurately communicate angular dimensions. The choice between the two formats may depend on the designer's preference or a company's standard practice. DMS may have the advantages of being a more familiar format for many professionals and correlating the format of many angle measuring instruments. The decimal degree format has the advantage of using one singular continuous number, simplifying the process of performing angular calculations.

Linear dimensions indicate the size or distance between two points. These are often used to provide the endpoints of the angle rather than providing the angle value. This technique, which gives the coordinate locations of the angle endpoints, may be preferred by many designers because it can provide better control over the two endpoints of the angle. Figure 6-9 depicts linear dimensions.

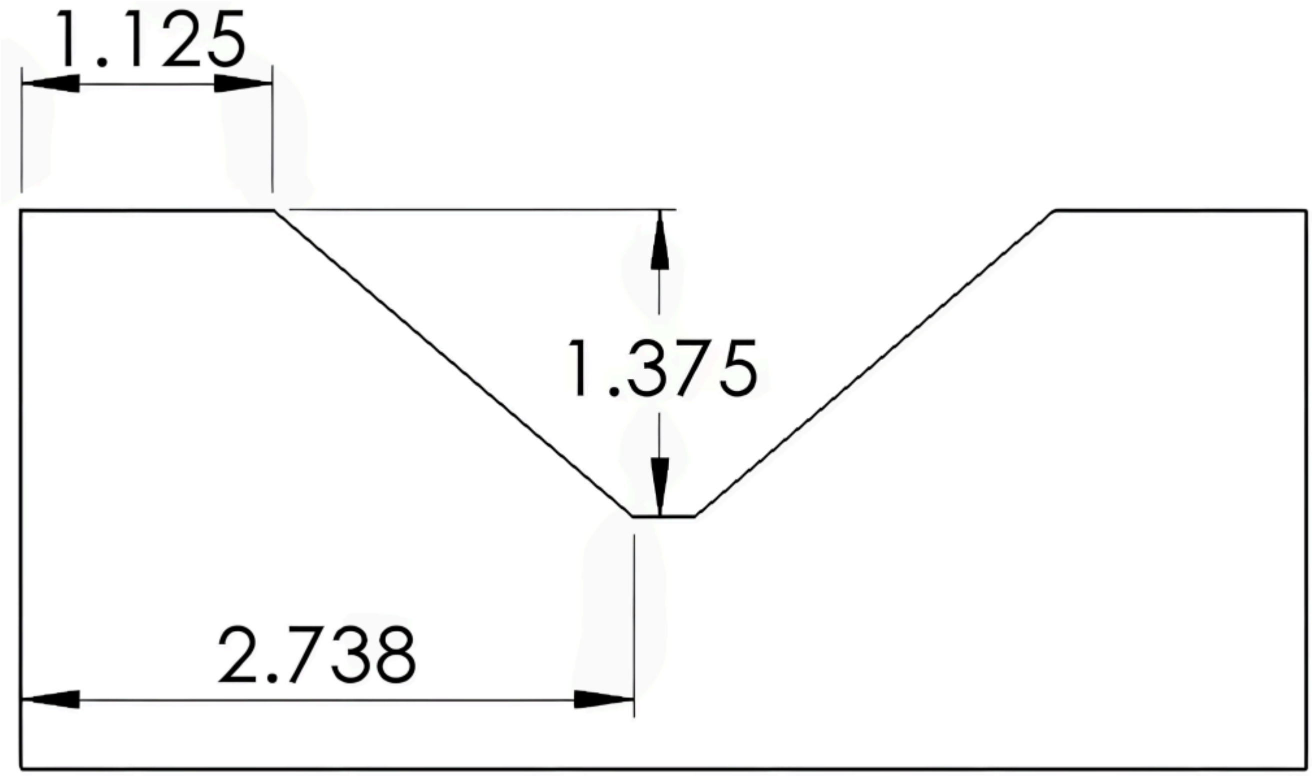


Figure 6-9. Linear dimensions are used to identify endpoints of an angular dimension.

When an angular dimension is assigned a size tolerance, the distance of the variation increases as the distance from the vertex increases; Figure 6-10 illustrates this concept. In this example, the allowable variation in angle is illustrated with the blue phantom lines. An angle dimension allowing a variance of ±2° could have an endpoint location variance of 0.442" over a distance of 6.00".

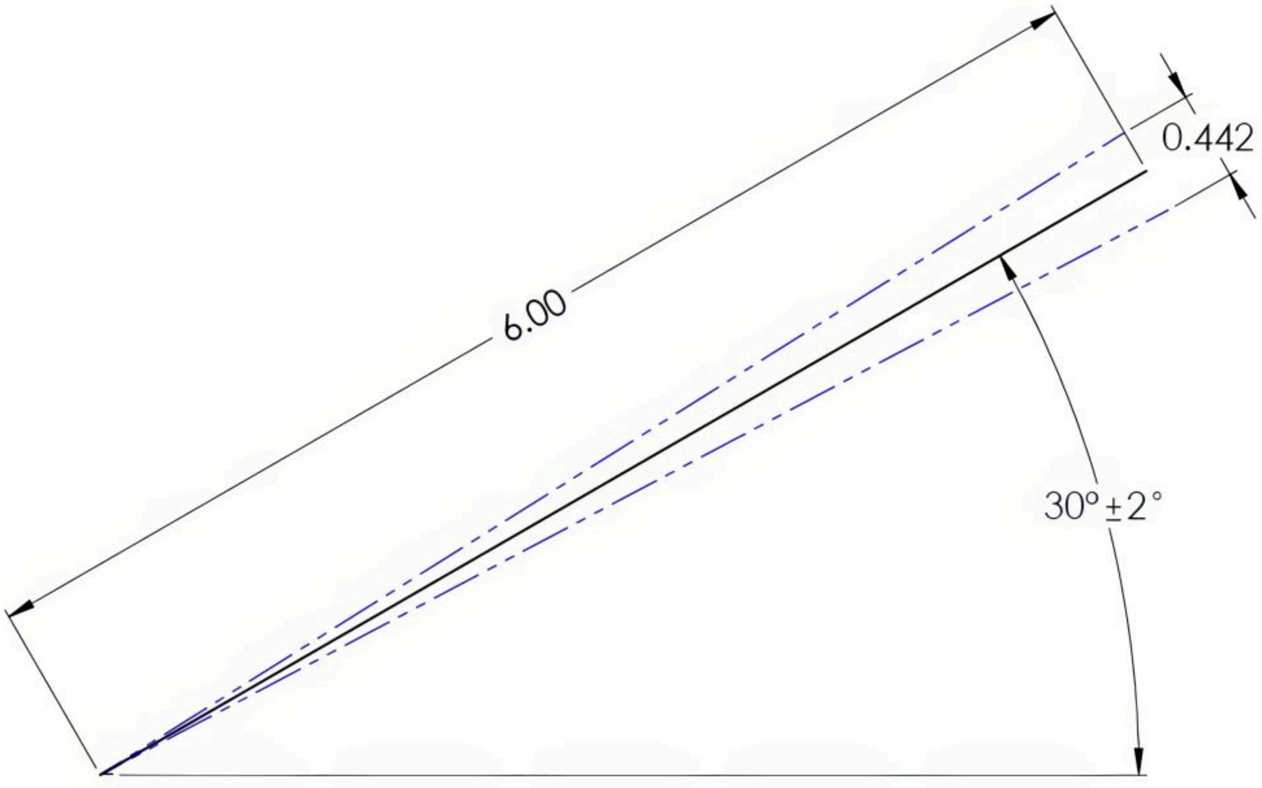


Figure 6-10. In an angular dimension tolerance, the distance variation increases as the distance from the vertex increases.

Angular Calculations

Whether angular or coordinate dimensioning is used for angles, this information can be used for calculations of lengths and missing angles in part drawings. Knowing some information about triangles, particularly right triangles, will aid in many angular calculations. All triangles will contain three angles with a sum equal to 180°. Triangles containing one angle of 90°, known as right triangles, will leave the sum of the two remaining angles equal to 90°. In Figure 6-11, for example, the angle of 40°, along with the linear dimensions, can be used to calculate the remaining dimensions.

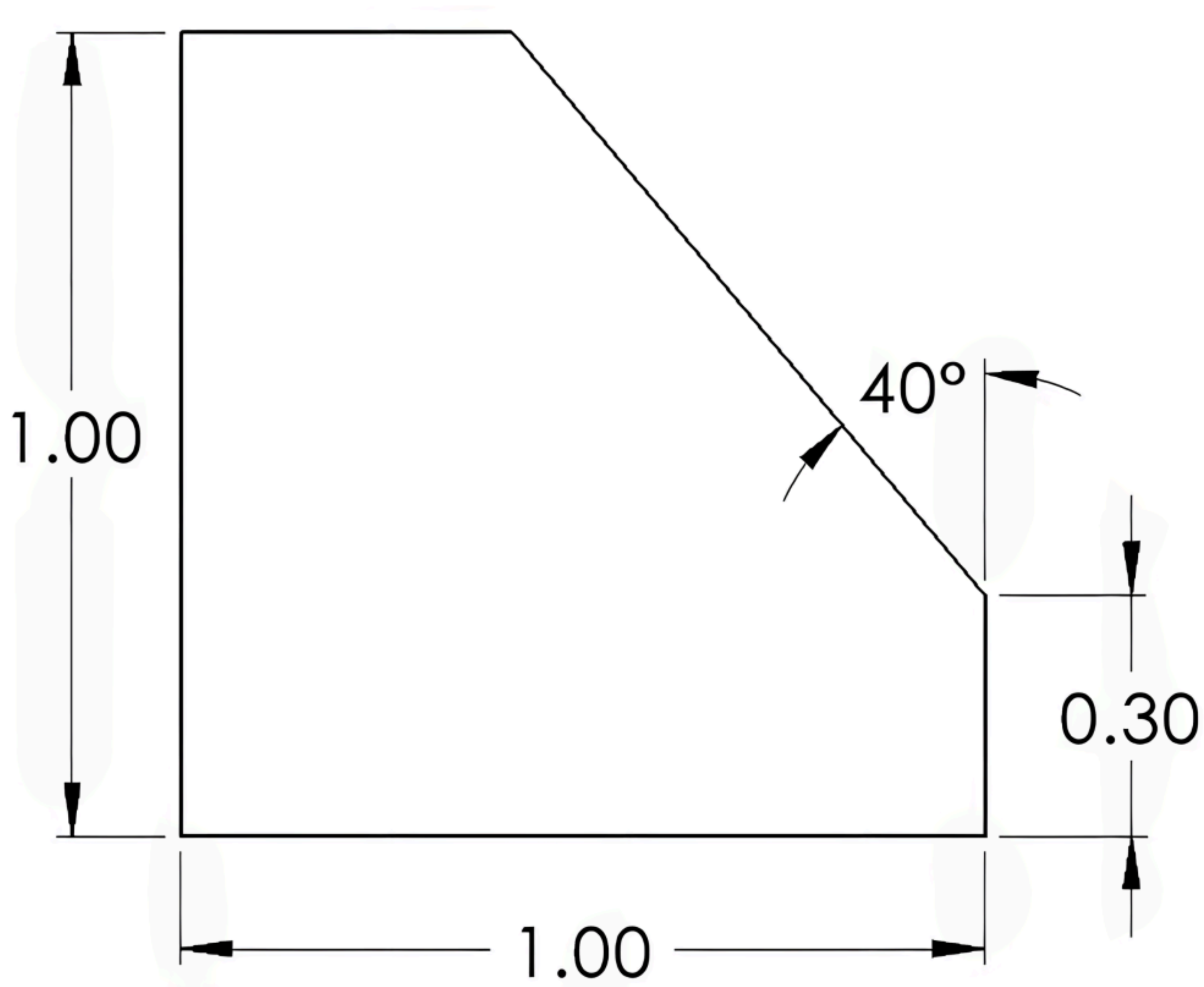


Figure 6-11. The angular dimension given can be used to calculate other dimensions.

In Figure 6-12, you can see how a right triangle can be used to identify the 50° angle. The triangle drawn in Figure 6-13 contains a 90° angle, identified by the square in the corner, leaving 90° between the remaining two angles. With 40° being dimensioned, 50° remains for the third angle. In the example, 0.70 is obtained by subtracting 0.30 from the height of

1.00. The values of 0.41 and 0.59 will require the use of trigonometric calculations involving right triangles to determine.

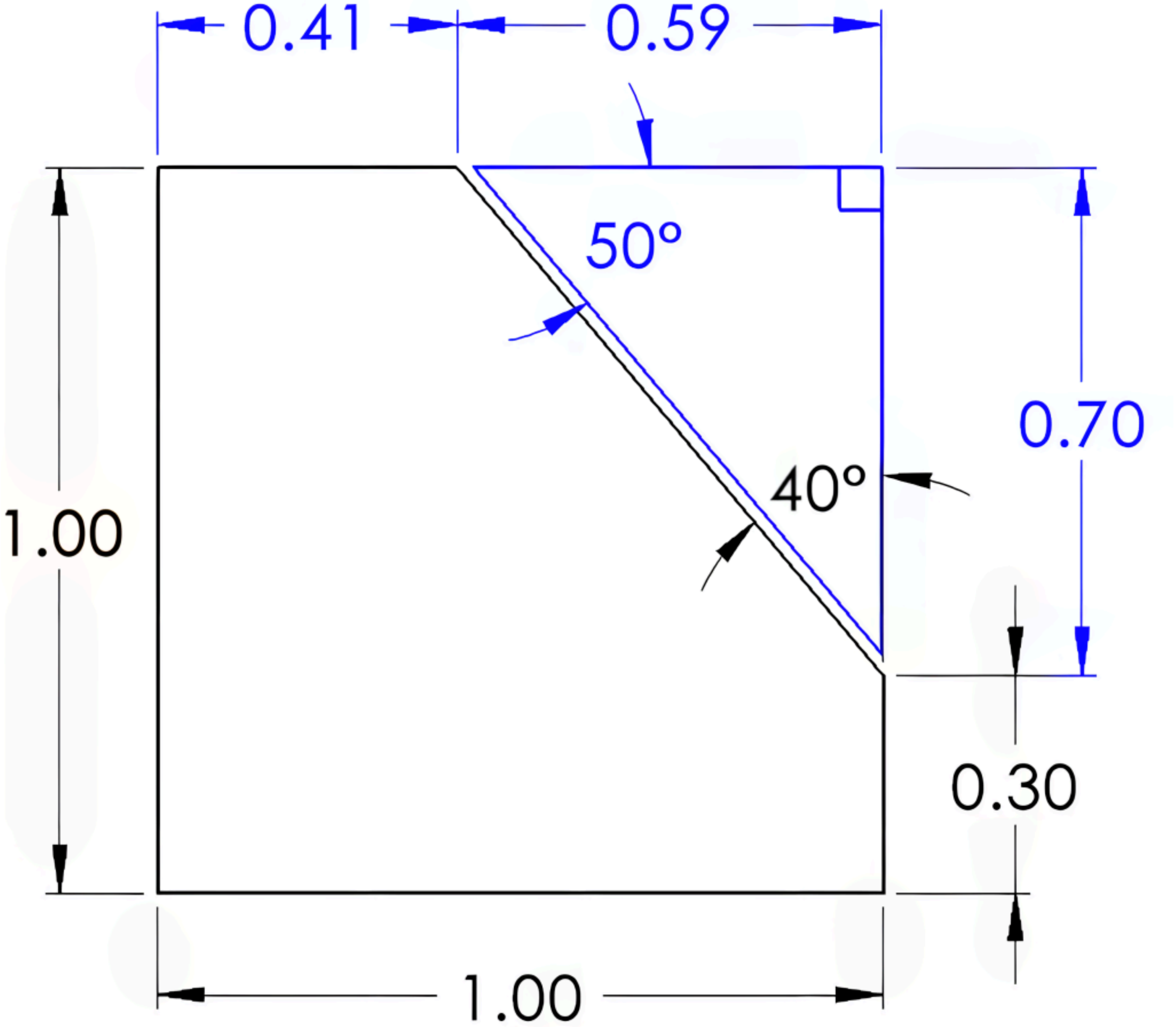


Figure 6-12. The information in Figure 6-12 can be used to calculate additional dimensions.

6.5 INCLUDED ANGLES

An **included angle** is the angle between two sides of a triangle or two lines that intersect at a point. It is the angle formed by the two sides or lines when they meet at a common vertex. The included angle will be less than 180°. Some common examples are the angles of drill points shown in Figure 6-13 and countersinks in Figure 6-14.

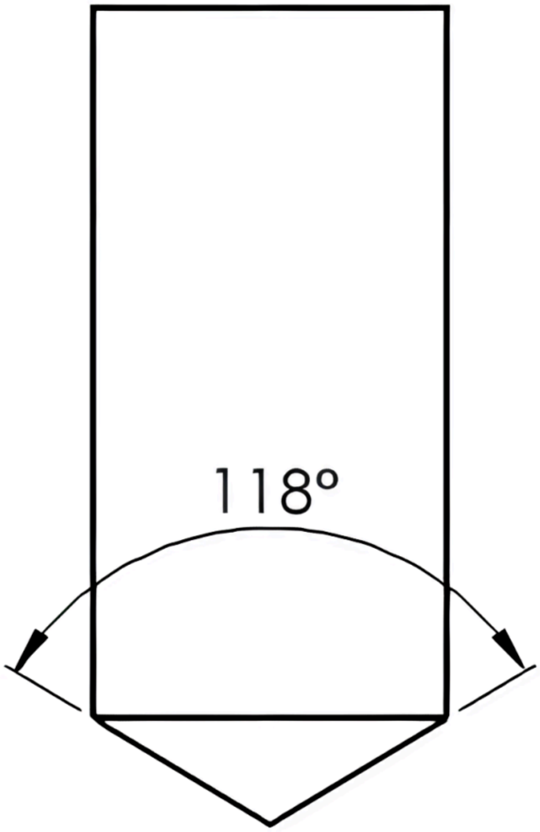


Figure 6-13. Drill point included angle.

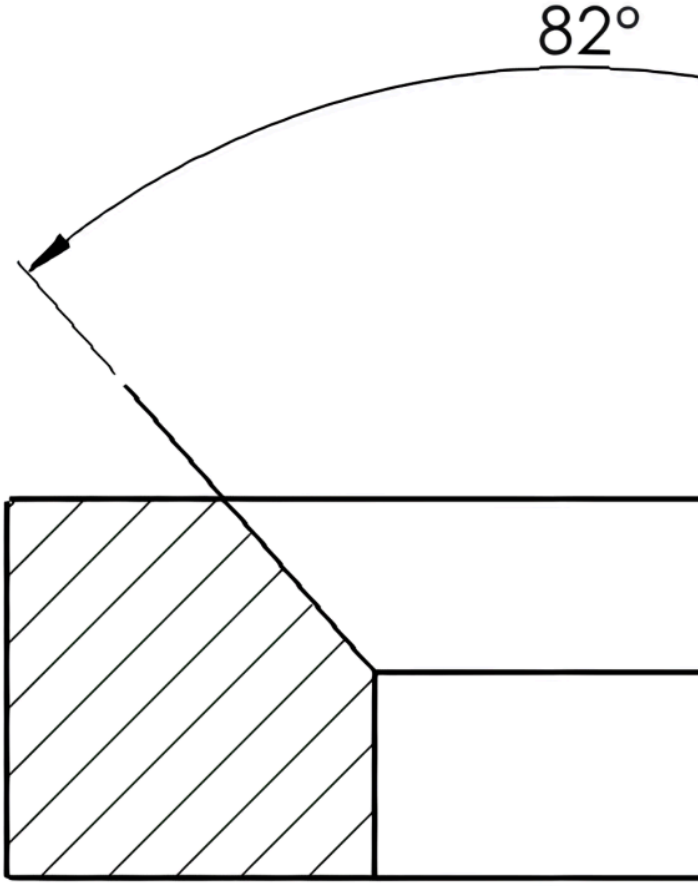


Figure 6-14. Countersink included angle.

In Figure 6-15 the included angles of 90° and 100° are calculated from the given angles of 45° and 50°. The centerline indicates the part is symmetrical, making the angle opposite the 45° also 45°. Extending these lines to where they meet will create a triangle. Subtracting the two 45° corners from 180° will leave the 90°. The 50° angle can be doubled to obtain the 100° angle. A straight line of 180° minus the 45° will leave 135°.

The 50° angle can be used to create a right triangle to determine the 40° angle.

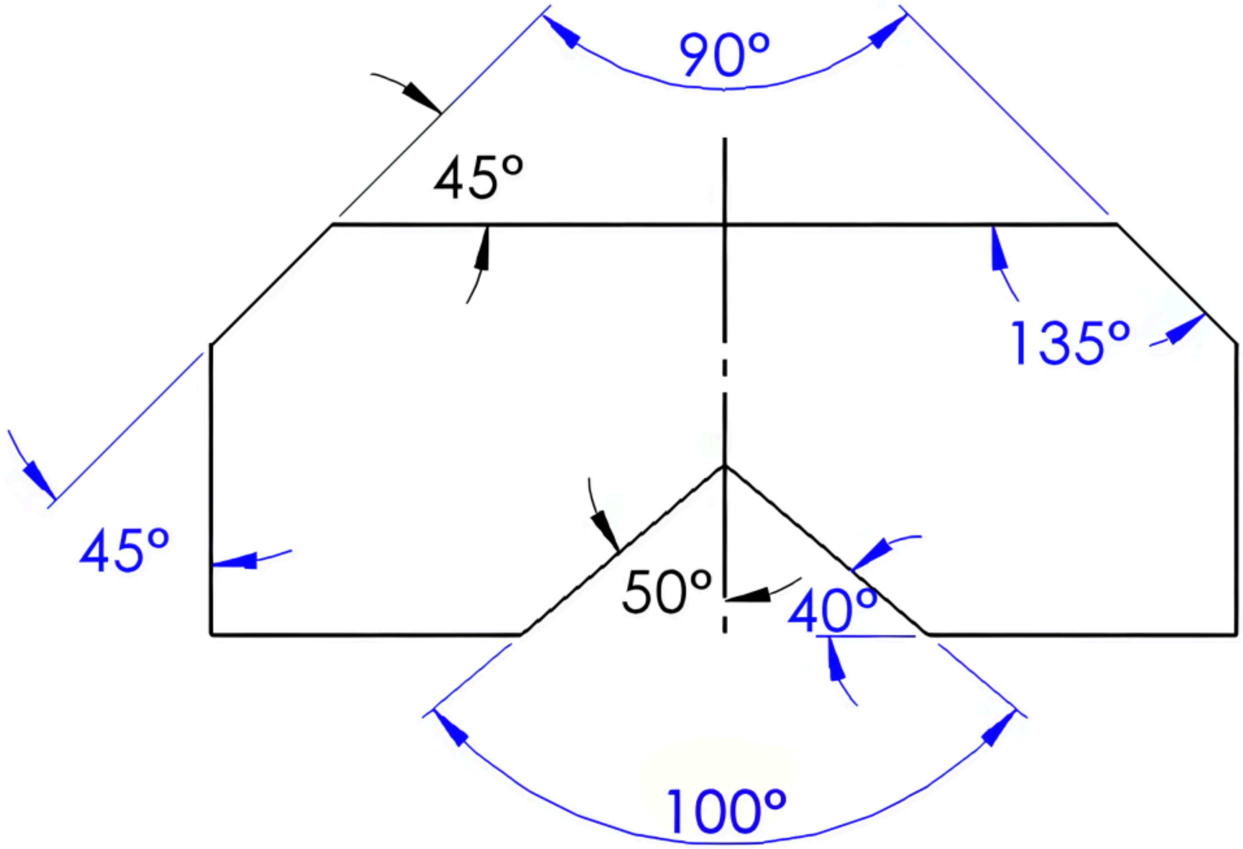


Figure 6-15. The angles in blue are calculated from the information in black.

6.6 BOLT CIRCLES

Bolt circles, commonly found in circular parts, are a series of clearance holes or threaded holes designed for fastening. These patterns consist of three or more holes arranged around a shared center point, making them **concentric**. The center line of this hole pattern represents the bolt circle itself, which is dimensioned using either a leader line with the bolt circle (BC) diameter following the hole information or a diameter dimension line.

When it comes to the placement of these holes, there are two common methods for dimensioning: their positions can be referenced using coordinate locations, or angular dimensions based on vertical or horizontal centerlines. The holes are often equally spaced, abbreviated “EQL SP,” with at least one hole aligned along the vertical or horizontal axis, as in Figure 6-16, or measured angularly from the vertical or horizontal axis, as in Figure 6-17. If you’re calculating angles between equally spaced holes, you can simply divide 360° by the total number of holes involved.

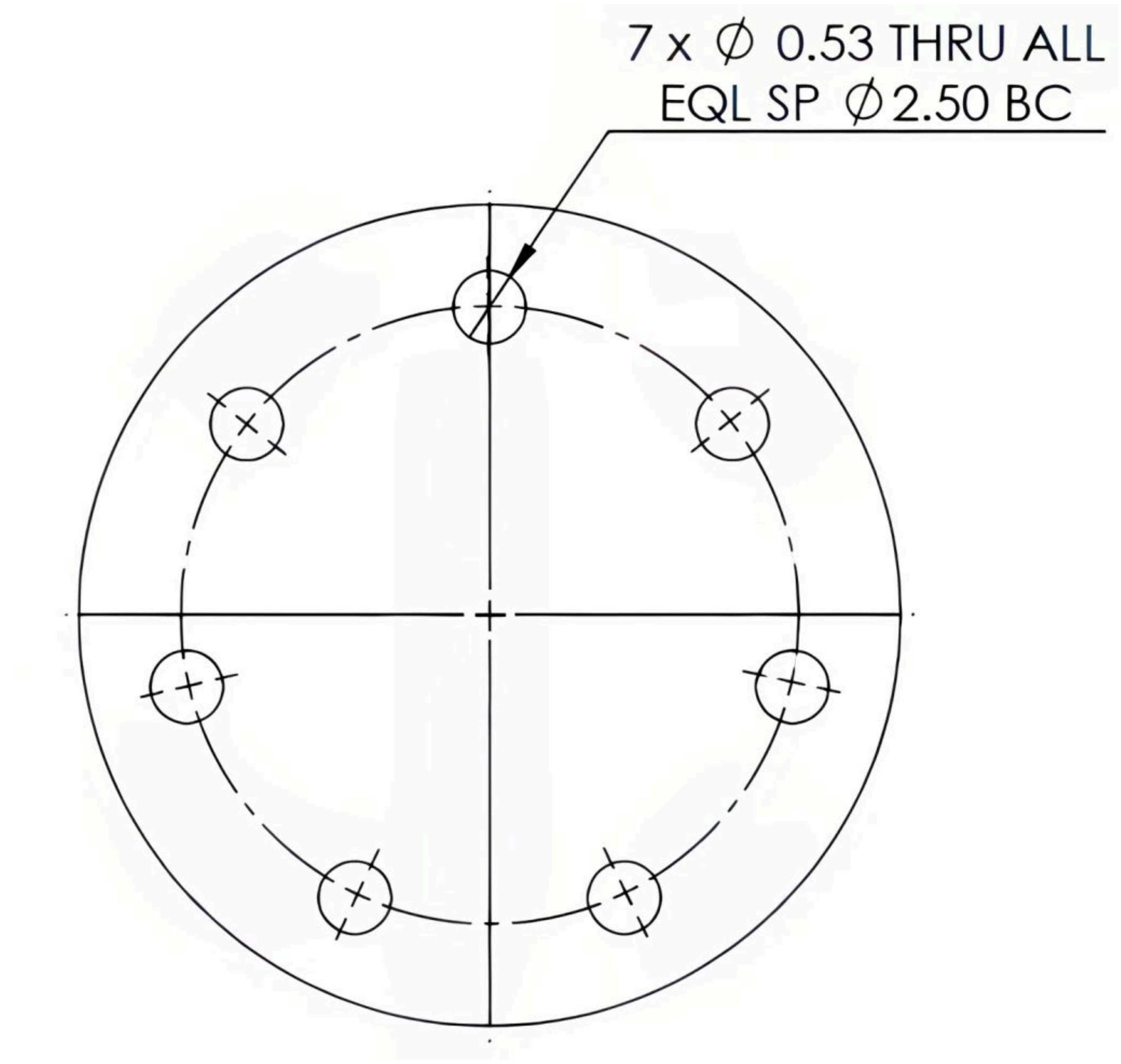


Figure 6-16. Bolt circle diameter callout with a leader. Bolt circle centerline diameter dimensioned.

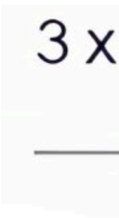


Figure 6
vertical

Coordinate dimensioning of the locations dimensioned from the vertical and horizontal centerlines is commonly used in milling machines for locating the X and Y axis locations. This style of bolt circle dimensioning can be most relevant to the reader but may create congestion in the drawing. See Figure 6-18. The coordinate locations may also be calculated using right triangle trigonometry functions, the known angle, and hypotenuse length (the bolt circle radius) as shown in Figure 6-19. Many CNC machines, programming software, and even digital readouts are capable of calculating these positions as well.

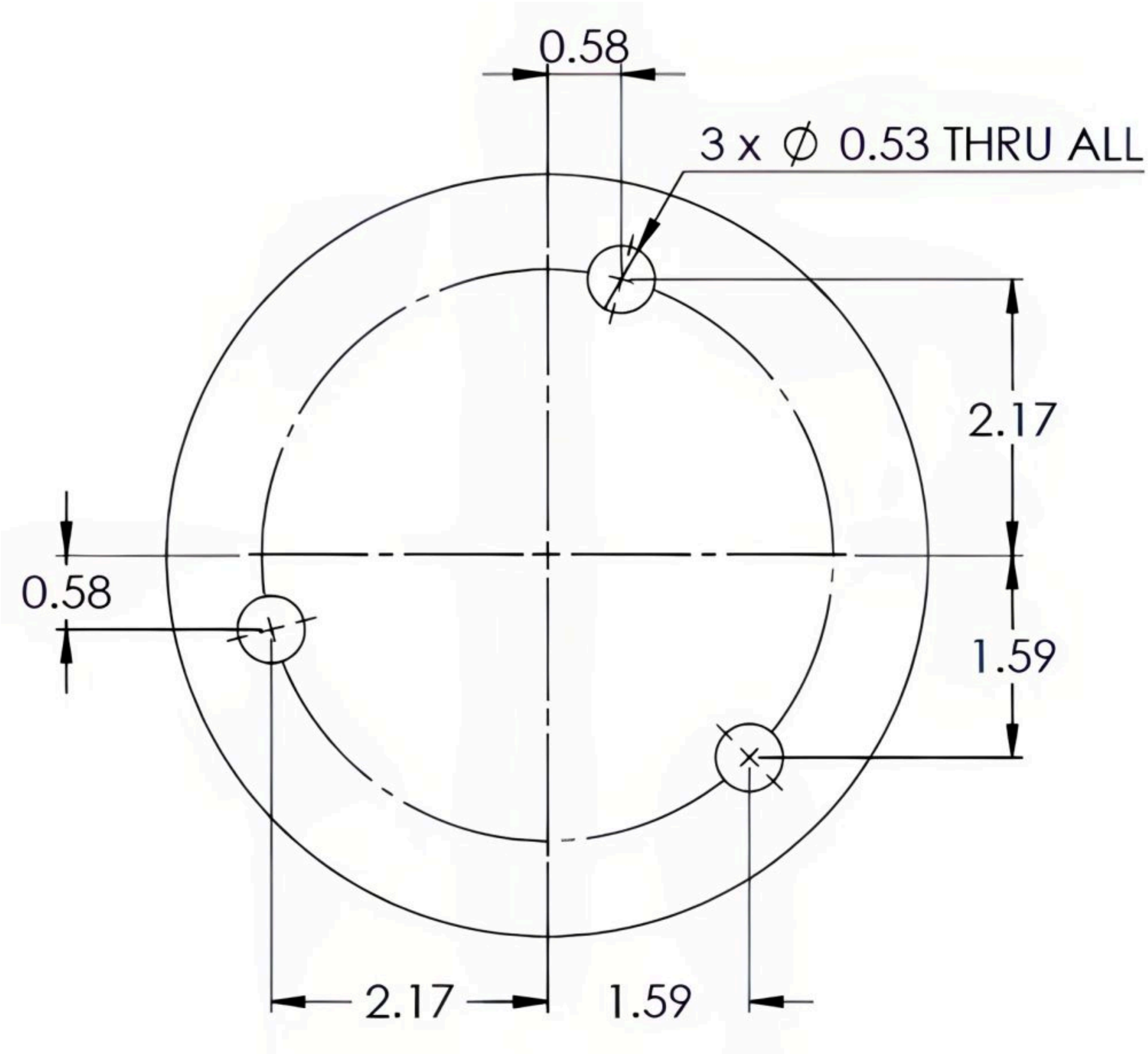


Figure 6-18. Bolt circle locations are dimensioned using coordinate positions.

Figur

LEARNING ACTIVITIES

Exercise 6.5-1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=186#h5p-14

Exercise 6.5-2

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=186#h5p-15

Exercise 6.6-1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=186#h5p-16

Exercise 6.6-2

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=186#h5p-17

References:

Barsamian, M. A., & Gizelbach, R. (2022). *Machine trades print reading.* Goodheart-Willcox.

Schultz, R. L., & Smith, L. L. (2012). *Blueprint reading for machine trades* (7th ed.). Pearson.

Images:

All images by Mark Lorier and licensed under CC BY-NC 4.0 except where otherwise noted.

7. Title Blocks

7.1 INTRODUCTION

Learning Objectives

- Describe the information found in a title block
- Identify other elements of information found on a print
- Identify the current drawing revision and the changes
- Apply tolerance information found in the title blocks
- Differentiate between specified and unspecified tolerances
- Define tolerance terminology

Terms

- Title block
- Drawing number
- Part number
- Scale
- Revision
- Zone

Understanding the title block of a print is essential for interpreting and understanding important information about the drawing. The title block is a standardized section located in the lower right corner of the print and contains crucial details about the drawing. It typically includes information such as the title of the drawing, the drawing number, the revision number, the scale of the drawing, the date of creation or revision, the material, and the name of the drafter or designer. The title block is a reference point for identifying and organizing drawings, ensuring accuracy, and tracking revisions. Familiarizing yourself with title block information will help ensure you understand all the necessary information accurately.

7.2 PRINT FORMATS

It's important to note that the layout of the title block and certain elements can vary among companies. Some organizations follow established drafting standards from respected bodies like the ANSI or ISO. In contrast, others may opt for a more tailored approach that suits their needs. Regardless of the chosen style, you can always expect that all drawings will include the essential information required to understand each part.

7.3 PRINT SIZES

The sizes of the prints are identified by standards as well. Whether in digital form or physical paper, size codes are identified by letters A through E for decimal-inch drawings or A4 through A0 for metric drawings. The print size will depend on the space necessary to communicate the information effectively. A small, simple part may fit on a small sheet, while a large or complex part or assembly may require more space. The initial size is a standard letter format measuring 8.5 x 11 inches, and from there, the dimensions will increase, with the previous length becoming the new width, as shown in Figure 7-1.

Inch	**Metric**
A: 8.5 x 11 inches	A4: 210 x 297 mm
B: 11 x 17 inches	A3: 297 x 420 mm
C: 17 x 22 inches	A2: 420 x 594 mm
D: 22 x 34 inches	A1: 594 x 841 mm
E: 34 x 44 inches	A0: 841 x 1189 mm

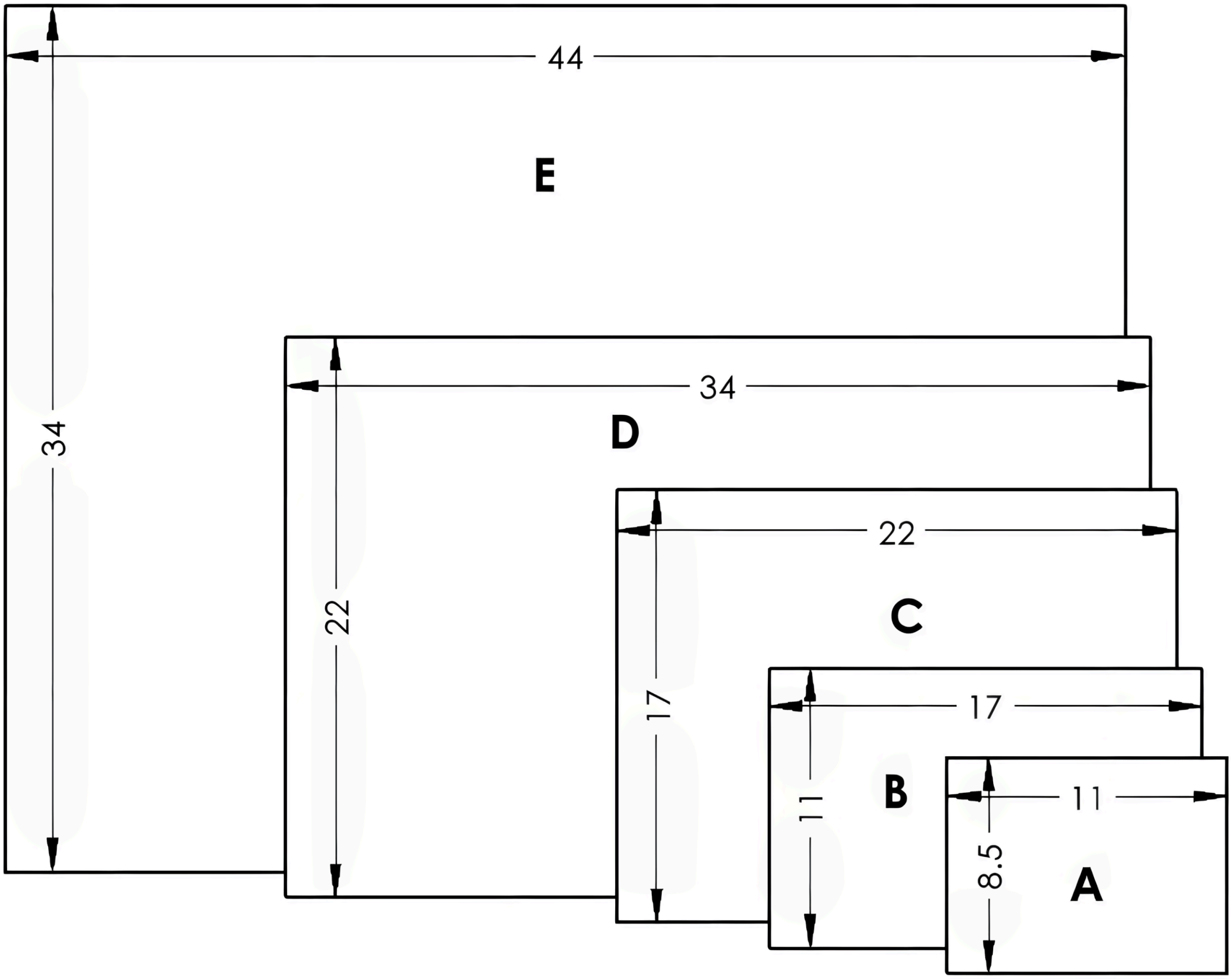


Figure 7-1. Standard sheet sizes used for prints.

7.4 TITLE BLOCK INFORMATION

The **title block** is the rectangular area in the lower right containing several information boxes. Some of the information includes the company name and/or logo, drawing and part number, designer, scale, and drawing dates. The boxes keep the project information organized and easily located. Figure 7-2 identifies common title block features with their definitions.

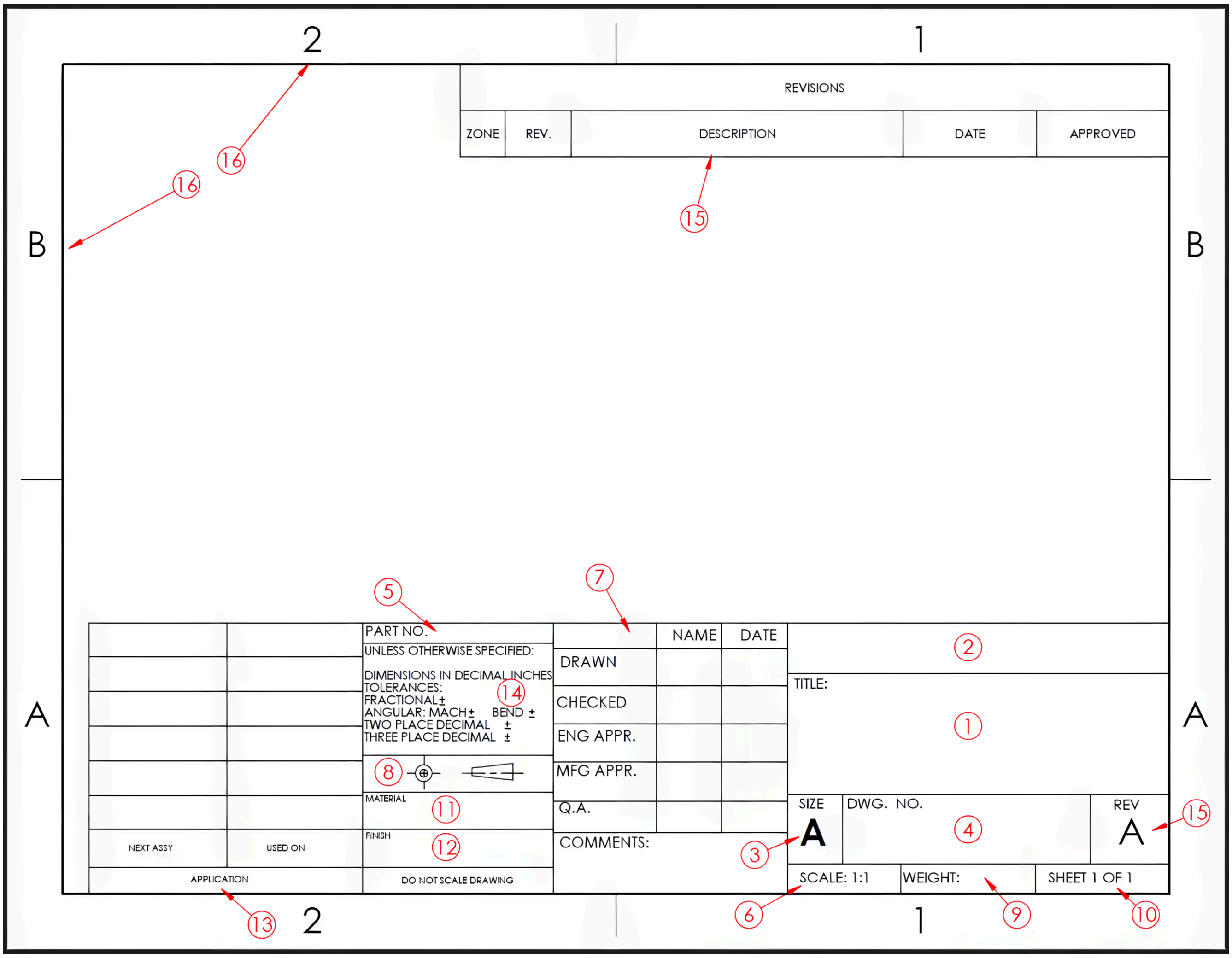
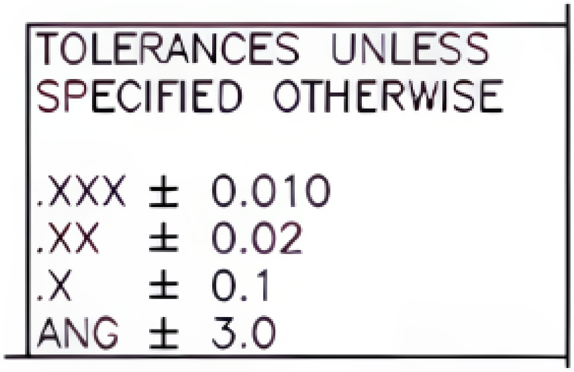


Figure 7-2. Print sheet with common title block features.

1. **Part Title** – The title is a descriptive name given to the drawing, typically the largest box in the title block.

2. **Company Information** – The name of the company responsible for the drawing will also appear in a larger box in the title block. Often, a logo for the company will be displayed as well.
3. **Size** – The size block displays the letter of the sheet size of the print.
4. **Drawing Number** – The **drawing number** is a numbering system the company uses to aid in filing and tracking the drawing.
5. **Part Number** – A **part number** may be used to identify a specific part, such as in an assembly parts list.
6. **Scale** – The **scale** displays the proportion of the drawing compared to the object. When the part is too large to fit on the sheet, the drawing may be reduced to ½, ¼, ⅛, etc., of the part size. When the part is smaller than the sheet, the drawing may also be increased proportionally. The ratio displayed in the scale box is arranged by the size of the drawing followed by the size of the part. This can be called the "DO" scale, where "D" is the drawing, and "O" is the object (DRAWING:OBJECT). A scale of 1:2 indicates that the drawing is "1" and the actual object is "2," or the drawing is half the size of the part. A scale of 2:1 indicates the drawing is twice the size of the part. When the part and drawing are the same size, the scale can be indicated as 1:1, or "Full" may be displayed in the scale box. The drawing size may be made to best fit the sheet size without being a nominal or basic proportion. The scale box will display "NTS" for "Not To Scale" if the drawing does not follow a basic proportional size.
7. **Drawn** – This contains the name or initials of the drafter and the date of the drawing, as well as any other individuals involved with the project.
8. **Projection** – Projection identifies whether the drawing is a first-angle or third-angle projection. This may be indicated with the projection cone or written out.
9. **Weight** – Weight may be used to indicate the weight of the finished part when required.
10. **Sheet** – The sheet box displays the current and total number of sheets in that drawing set.

11. **Material** – The material section identifies the material specifications required.
12. **Finish** – The finish box is used to display any part finish specifications. Finish specifications may also be used for some regions of the part, and they are indicated with leader lines for those areas.
13. **Application** – The application block is used to identify the assemblies on which the part is used.
14. **Tolerance** – The tolerance box displays the tolerance or amount of variation allowed on the dimension values. These tolerances apply to dimensions that do not have a tolerance specified with a particular dimension. These tolerances are applied by the type of dimension, fractional, angular, or the number of decimal places displayed on decimal inch or metric values. See Figure 7-3.



TOLERANCES UNLESS SPECIFIED OTHERWISE

.XXX ± 0.010
.XX ± 0.02
.X ± 0.1
ANG ± 3.0

Figure 7-3. The tolerance box displays the amount of tolerance allowed for dimensions not otherwise specified.

With the tolerance block used in Figure 7-3, a two-place dimension of 2.00 allows for a ±0.02 tolerance, indicating that any measurement from 1.98 to 2.02 is permissible. For a three-place dimension of 2.000, acceptable measurements must fall within the range of 1.990 to 2.010.

---

15. **Revision** – **Revisions** are changes made to the drawing from the original version. These revisions can be to correct errors, improve the design, or make modifications due to a change in the manufacturing process to reduce expense. The revision level or letter is indicated in the lower right title block. The changes that have been made will typically be located in the upper right of the sheet, with the matching revision letter often in a triangular shape indicated at the location of the change. You must be sure you are working with the latest revision

level to ensure the correct part is being produced. If the drawing is released as a new drawing with no revision, a dash can be placed in the revision box. Some companies may instead use the letter “A” in the revision box to release the initial drawing. Both are acceptable.

16. **Zones** – Sheet **zones** help locate a particular detail or view. They are beneficial when working with larger sheets.

LEARNING ACTIVITIES

Exercise 7.4-1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=304#h5p-22

Exercise 7.4-2

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=304#h5p-23

Exercise 7.4-3

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=304#h5p-24

References:

Barsamian, M. A., & Gizelbach, R. (2022). *Machine trades print reading*. Goodheart-Willcox.

Schultz, R. L., & Smith, L. L. (2012). *Blueprint reading for machine trades* (7th ed.). Pearson.

Images:

8. Tolerance on Dimensions

8.1 INTRODUCTION

Learning Objectives

- Define tolerance terminology
- Apply mathematical concepts to determine tolerances on dimensions
- Determine tolerances on dimensions
- Determine limits on dimensions
- Describe how tolerances affect mating parts

Terms

- Tolerance
- Bilateral tolerance
- Equal bilateral
- Unequal bilateral
- Unilateral tolerance
- Limits
- Single-limit dimensioning
- Specified tolerance
- Unspecified tolerance
- Material condition
- Interference fit

- Clearance fit
- Reference dimension

Dimension tolerances are critical in ensuring that products meet the required specifications and perform their intended functions. Each dimensioned value of a part's sizes and features will have some allowable variance in size. It is important to comprehensively understand dimension tolerances, their significance, and their application in various industries.

This chapter will explore dimension tolerances and their connection to a product's intended functionality and quality. It will also investigate various tolerances, including bilateral, unilateral, and limit tolerances, and their definitions and understanding in engineering drawings.

8.2 TOLERANCE

Tolerance is the allowable variation or deviation from a specified dimension, measurement, or other characteristic of a product or component. Because it is impossible for a part to be produced to an exact theoretical size, providing a tolerance for the size is necessary. It defines the acceptable range within which a part or feature can deviate from its intended design while maintaining its functionality and meeting the required specifications. Tolerances are crucial in engineering and manufacturing to ensure proper fit, assembly, and performance of parts and facilitate interchangeability and quality control. Tolerances are typically specified on engineering drawings or technical documentation and expressed as numerical values, limits, or a combination. Tolerances are vital in determining manufactured products' precision, accuracy, and interchangeability.

The amount of tolerance on any part will directly impact the manufacturing cost. A designer must be careful not to apply a closer tolerance value than is necessary for the part function, as this will unnecessarily increase the cost due to the extra time and often extra processes necessary to meet the closer sizes.

8.3 BILATERAL TOLERANCE

When a tolerance is allowed to deviate from the dimension in both the positive and negative directions, it is known as a **bilateral tolerance**. When the tolerance is equal in both directions, it is an **equal bilateral**, or symmetrical, tolerance. The example in Figure 8-1 has an equal bilateral tolerance of ±0.002. In this example, the measured size may fall within the range of 0.002 above the dimensioned size to or 0.002 below the dimensioned size. In the Figure 8-1 example of the 2.000 dimension, the allowable size may range from 1.998 to 2.002, giving a total tolerance range of 0.004.

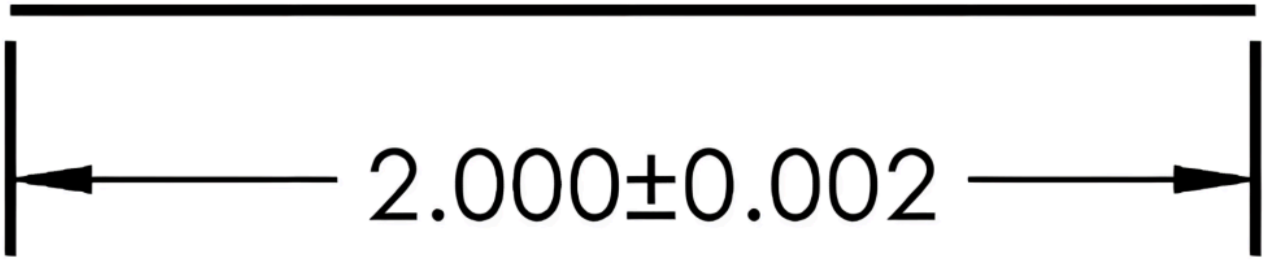


Figure 8-1. Equal bilateral tolerance – equal tolerance value in the positive and negative direction. In this example, the dimension value may range from +0.002 above the dimension size to -0.002 below the dimension size.

When a dimension's tolerance is bilateral, moving in both positive and negative directions but with different tolerance values, it is called an **unequal bilateral** tolerance. The tolerance in Figure 8-2 value may be anywhere from larger than the dimension by 0.001 to smaller by 0.002. In this 2.000 example, the size may range from 1.998 to 2.001 for a total tolerance range of 0.003.

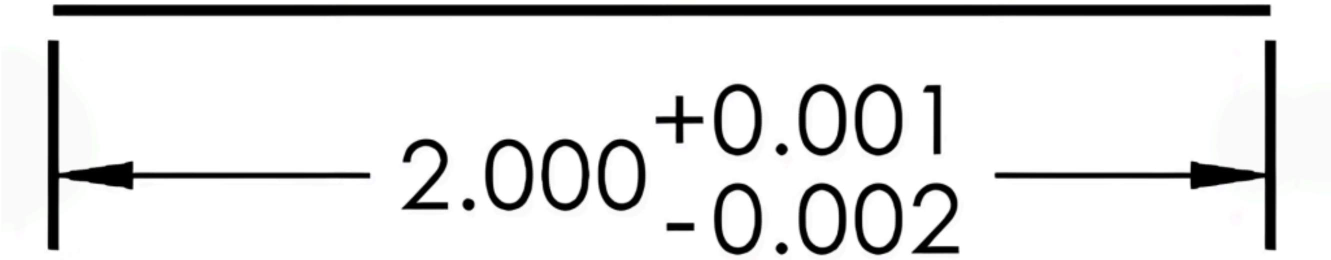


Figure 8-2. Unequal bilateral tolerance – value in both positive and negative directions of different values. In this example, the dimension value may range from +0.001 above the dimension size to -0.002 below the dimension size.

When the tolerance of a dimension moves only in one direction, having a zero value in either its positive or negative direction, this is called **unilateral tolerance**. In the example shown in Figure 8-3, the size tolerance may not move above the dimension value but can move below the dimension value by 0.005. In this 2.000 example, the allowable size may range between 1.995 and 2.000 for a total tolerance range of 0.005.

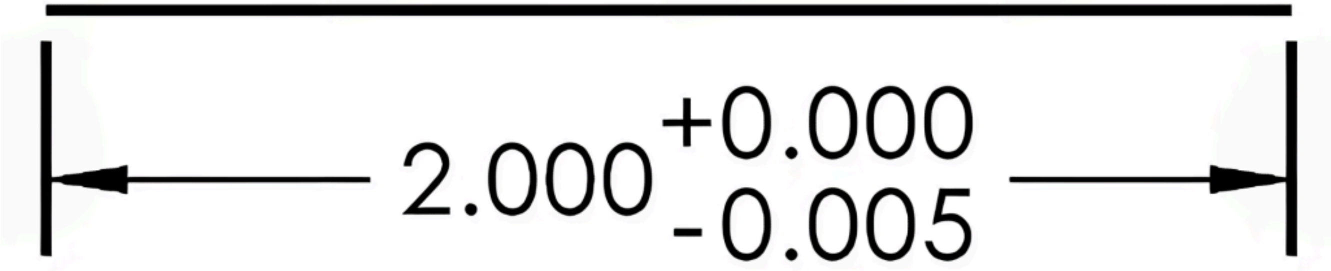


Figure 8-3. Unilateral tolerance – the tolerance value exists only in the positive or negative direction, with a zero value in the other direction. In this example, the dimension may not move above the dimension value in the positive direction, but may range to -0.005 below the dimensioned size.

8.4 LIMIT DIMENSIONING

Dimensional **limits** are the maximum and minimum allowable sizes of the dimension. The maximum and minimum permissible sizes will be displayed in place of plus or minus values. With limit dimensioning, the maximum or high limit will appear above the minimum or low limit value. Figure 8-4 displays a limit dimensioning example. In this example, 2.002 is the high limit, and 1.998 is the low limit, giving a total tolerance range of 0.004.

Figure 8-4. Limit dimension – the size for this dimension may range between 2.002 and 1.998.

Single-limit dimensioning will use only one limit value, the minimum or the maximum, unlike limit dimensioning, where a range of acceptable variation is displayed. Single-limit dimensioning specifies a single value representing a dimension’s minimum or maximum limit. The minimum limit is denoted by its abbreviation “MIN,” while the maximum limit is defined by “MAX.”

Single-limit dimensioning is used when there is a need to control only one side of the acceptable variation for a dimension. It may be employed when the other side of the tolerance range is not critical or when there is a specific requirement to limit the variation in one direction while allowing more flexibility in the other direction. A single-limit dimension is often used in depths of holes or threads and radii or chamfer values. In Figure 8-5, single-limit dimensions are used for minimum and maximum values on radii, chamfer, and thread length values.

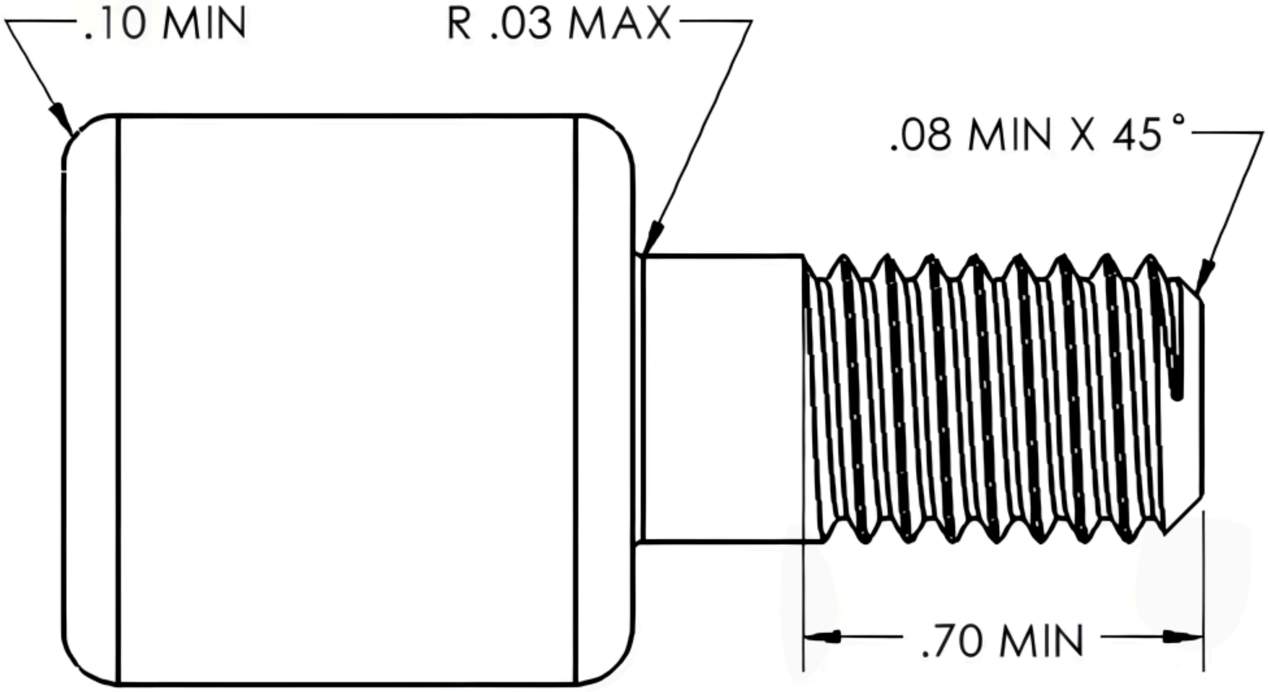
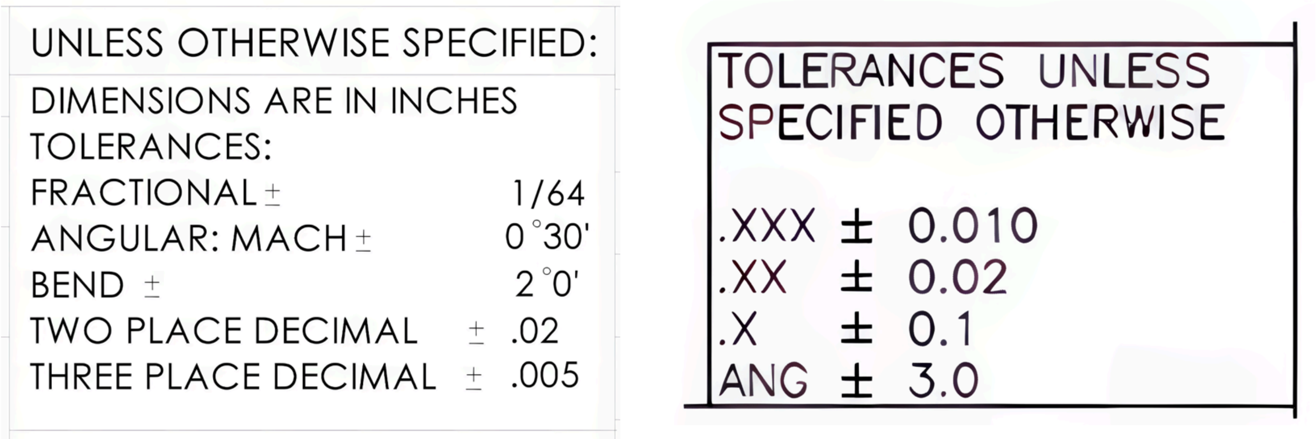


Figure 8-5. Single-limit dimensions are used to limit the dimensional size only in one direction.

8.5 UNSPECIFIED TOLERANCES

The examples above are known as specified tolerances. **Specified tolerances** appear with the specific dimensioned value and are directly applied to that value. Specified tolerances are not shared with any other dimension.

When a dimension does not have a specified tolerance, the tolerance value from the print's tolerance block will be applied, known as **unspecified tolerances**. In the print's tolerance block, you'll see the applicable tolerances outlined according to the type of dimension you're working with—whether it's fractional, angular, metric, or based on decimal places for either decimal inch or metric values. Figure 8-6 displays examples of tolerances applied to all dimensions without a specified tolerance.



UNLESS OTHERWISE SPECIFIED:

DIMENSIONS ARE IN INCHES

TOLERANCES:

FRACTIONAL ±		1/64
ANGULAR: MACH ±		0°30'
BEND ±		2°0'
TWO PLACE DECIMAL	±	.02
THREE PLACE DECIMAL	±	.005

TOLERANCES UNLESS SPECIFIED OTHERWISE

.XXX	±	0.010
.XX	±	0.02
.X	±	0.1
ANG	±	3.0

Figure 8-6. Tolerances applied to unspecified dimensions.

8.6 MATERIAL CONDITION

Material condition refers to the maximum or least amount of material allowed within the tolerance of a feature size. These are referred to as Least Material Condition (LMC) and Maximum Material Condition (MMC). For an external feature, Least Material Condition will be the smallest size allowed, and the largest size allowed is the Maximum Material Condition. For an internal feature, the Least Material Condition will be the largest size allowed, and the smallest size allowed will be the Maximum Material Condition. To better understand this, consider what happens as material is removed from the surface. As material is removed from an outside surface, such as an outside diameter, the part size will get smaller.

Conversely, as material is removed from an inside surface, such as a hole, the hole size will become larger. When machining these features, you aim to be material-safe at the MMC dimension. No further material can be removed at the LMC dimension. In the Figure 8-7 example below, the MMC of the shaft is 1.001, and the LMC is 0.999. In Figure 8-8 below, the MMC of the hole would be 0.999, and the LMC of the hole would be 1.001.

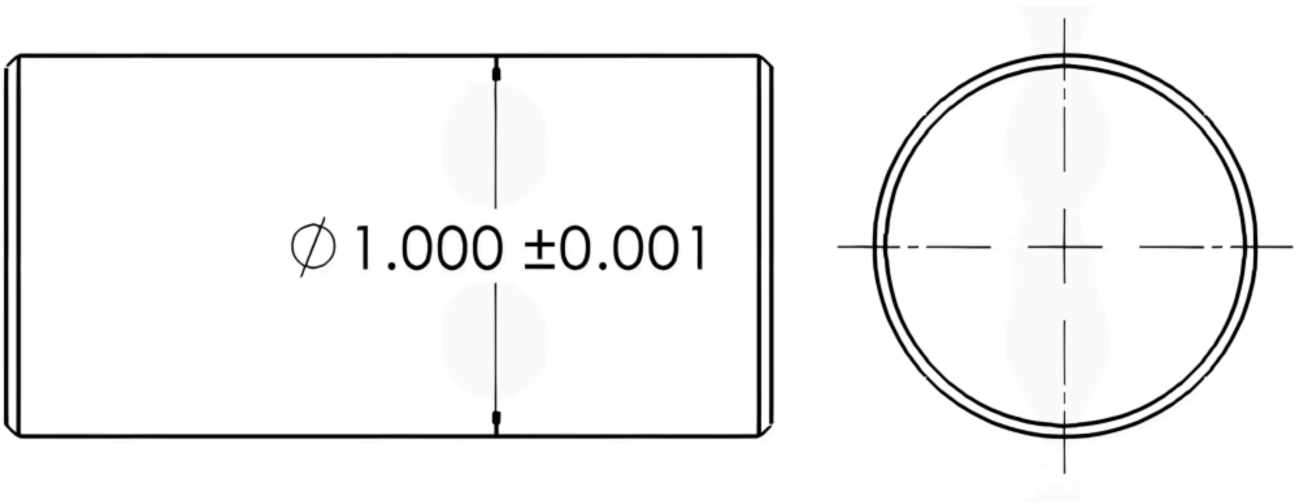


Figure 8-7. Maximum Material Condition (MMC) and Least Material Condition (LMC) for an external feature example.

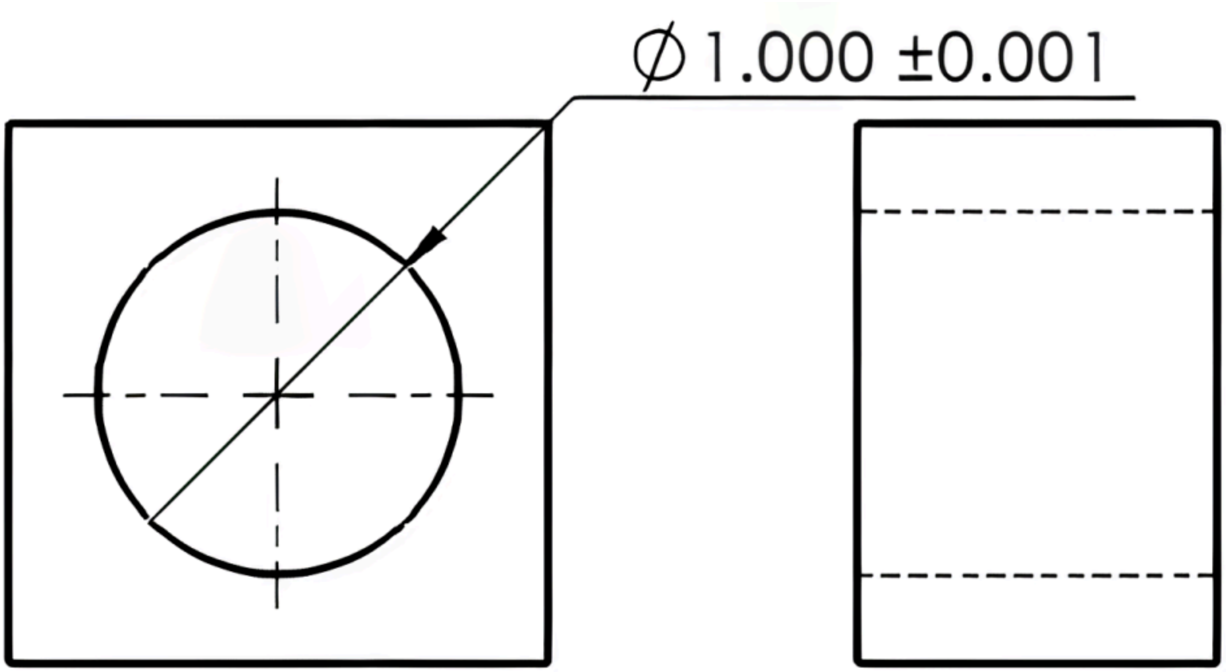


Figure 8-8. Maximum Material Condition (MMC) and Least Material Condition (LMC) for an internal feature example.

8.7 CLEARANCE AND INTERFERENCE

When creating a print, it's essential to consider the sizes and tolerances involved thoughtfully. An **interference fit** occurs when a part is larger than the part it is to be fit into; this is also referred to as a press fit. A **clearance fit** occurs when a part is smaller than the part it is to be fit into; this is also called a slip fit. Maintaining the desired design function of components requires careful attention to all acceptable dimensions. In the example shown in Figure 8-9, the sizes and tolerances for the hole component (shown in the middle of the figure) and Pin A must ensure they maintain the interference fit as intended. The dimensions and tolerances are compared in the Pin A interference fit at MMC and LMC.

On the other hand, if our goal is to achieve clearance or slip fit, we can consider Pin B. Its size and tolerance have been carefully chosen to guarantee adequate clearance, regardless of allowable size variations.

Standard fit sizes for both inch and metric values based on the basic part size and fit required have been established and are often referenced by the designer. These values are available in various reference books, such as the *Machinery Handbook*. The Figure 8-9 clearance fit example examines scenarios in which parts are created at MMC and LMC while maintaining their desired clearance fit.

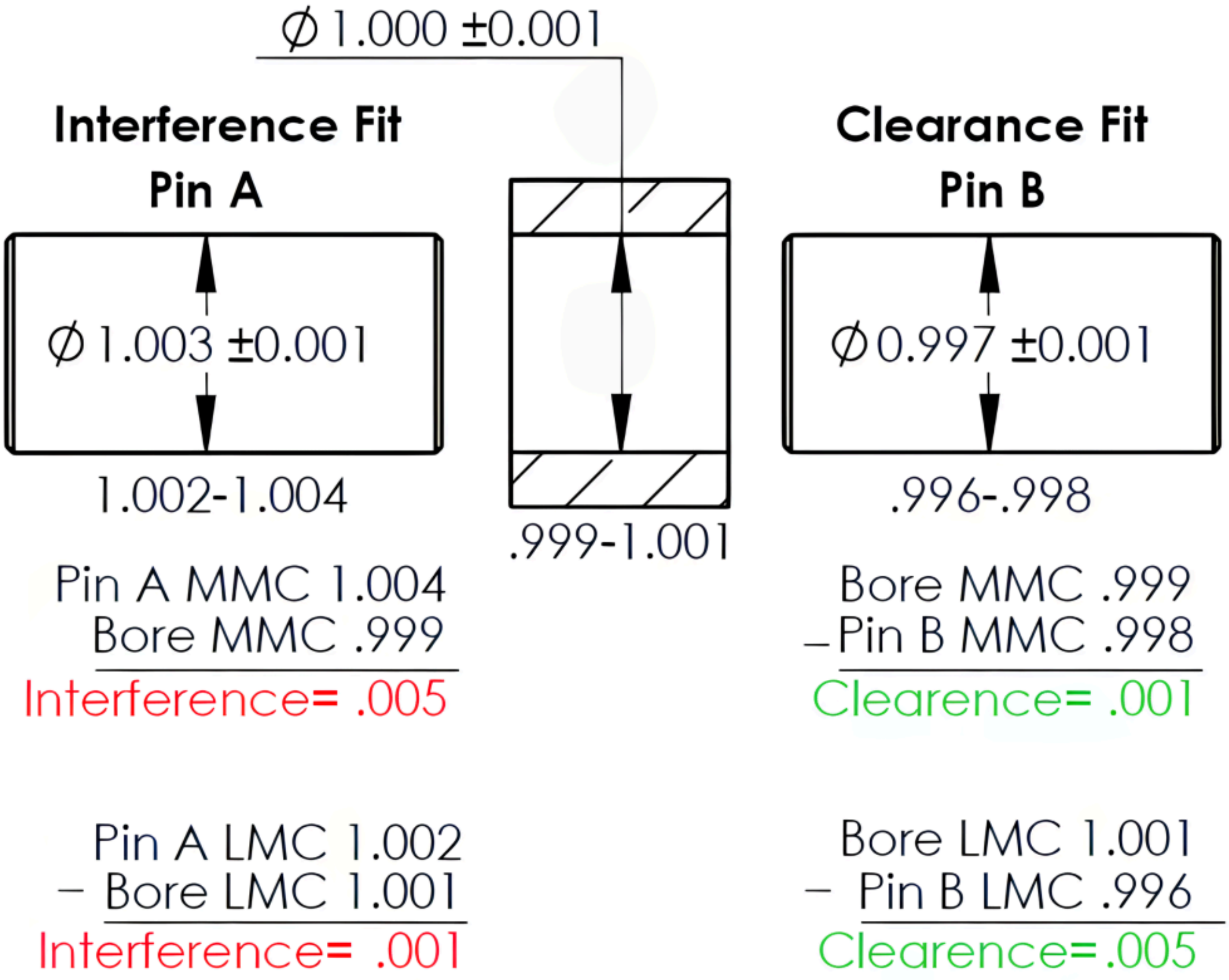


Figure 8-9. Examines the dimension and tolerance scenarios to maintain an interference fit and a clearance fit with the parts at their MMC and LMC sizes.

8.8 TOLERANCE ACCUMULATION

As each drawing dimension will have a tolerance assigned, it is essential not to have a feature dimensioned more than once, making the drawing over-dimensioned. Over-dimensioning is avoided because a feature may contain more than one tolerancing option. Because of this, many sizes must be calculated from other dimensions. When dimensions are calculated, the tolerance of each dimension used accumulates as well.

Even when dimensions are subtracted, the tolerances of each dimension used to obtain the feature size will accumulate.

An earlier chapter discussed the different methods of dimensioning used below. This chapter examines how these various methods can affect the tolerances of the four lengths and the overall length of the object.

Figure 8-10 uses datum-dimensioning, with each dimension referenced from the left edge. In this datum-dimensioned example, each measurement from the datum end maintains the ±0.01 tolerance. This example means the 1.00, 2.00, 3.00, and 4.00 all have a tolerance value of ±0.01. To obtain the value for length B, the 1.00 is subtracted from 2.00. Because the two dimensions used each have a tolerance of ±0.01, these combine for a ±0.02 tolerance on length B. The addition of these tolerances is illustrated in the equations below. If the 2.00 is at its MMC and the 1.00 at its LMC, length B will be 1.02. Conversely, if the 2.00 is at its LMC and the 1.00 is at MMC, then length B will equal 0.98. This concept will repeat for lengths C and D.

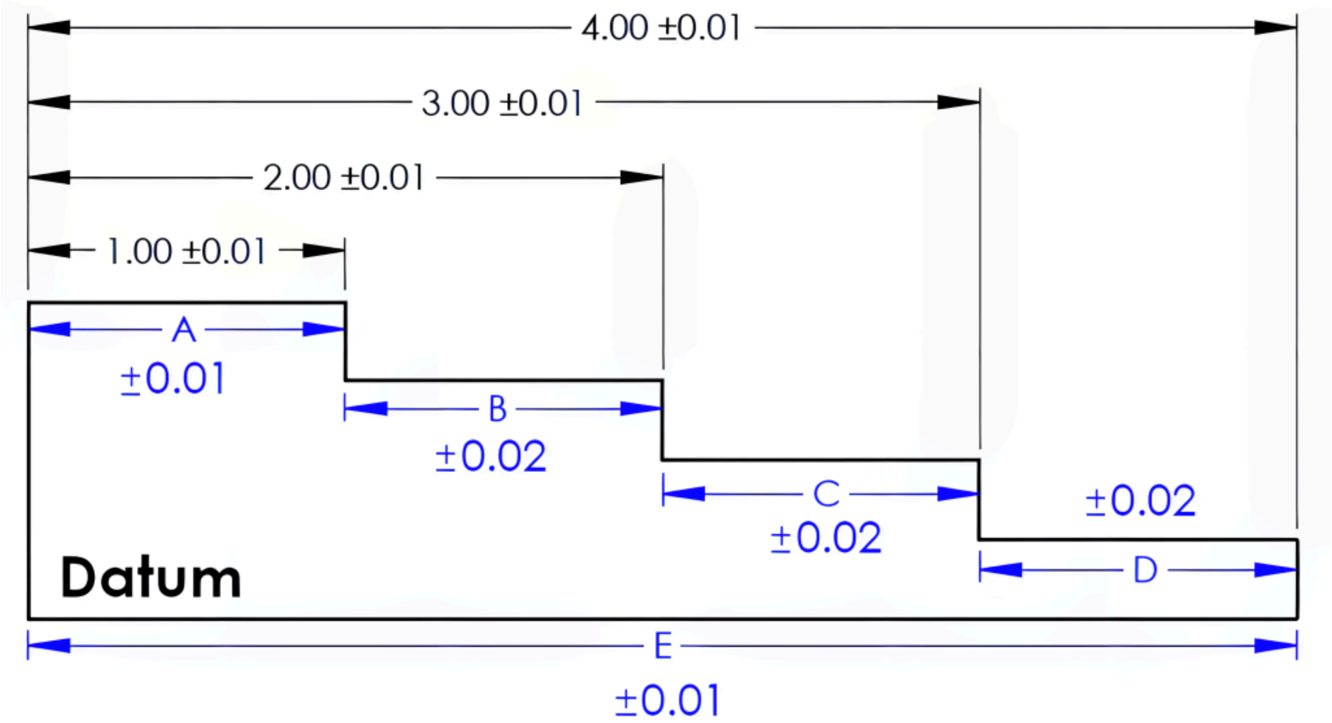


Figure 8-10. Datum-dimensioning effects on tolerance.

Figure 8-11 uses a chain-dimensioned example; the distance of each length is toleranced independently. The overall distance would need each dimension added together, making the overall length a tolerance of ±0.04. As illustrated in the equations below, adding the MMCs of each length results in the overall length having a +0.04 tolerance, and adding the LMCs of each length will result in an overall length of -0.04. Each combination of lengths added together would accumulate the ±0.01 tolerance as well.

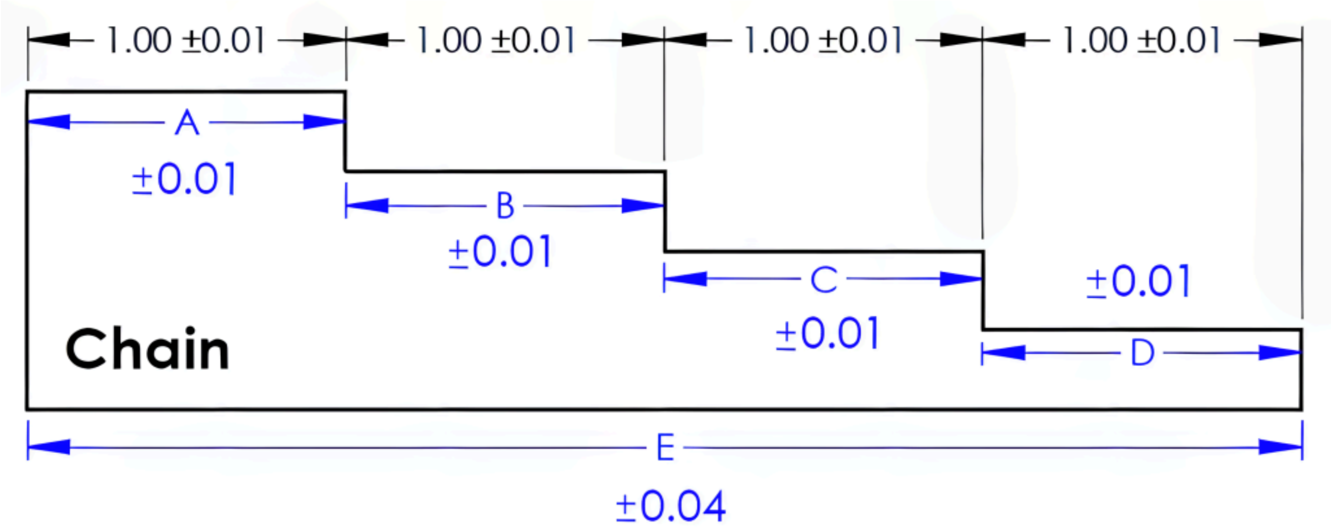


Figure 8-11. Chain-dimensioning effects on tolerance.

Figure 8-12 uses broken-chain dimensioning. In this example, length D is left undimensioned. To obtain the length of D, lengths A, B, and C are to be subtracted from the overall length E. The tolerance of each length will stack together in length D for a ±0.04 tolerance for this dimension. The formula below shows the LMC of lengths A, B, and C subtracted from the MMC of the overall length (E), resulting in a 1.04 maximum length for D. The calculation for the minimum length for D begins with the LMC for the overall length and then subtracts the maximum lengths of A, B, and C for a 0.96 minimum length for D.

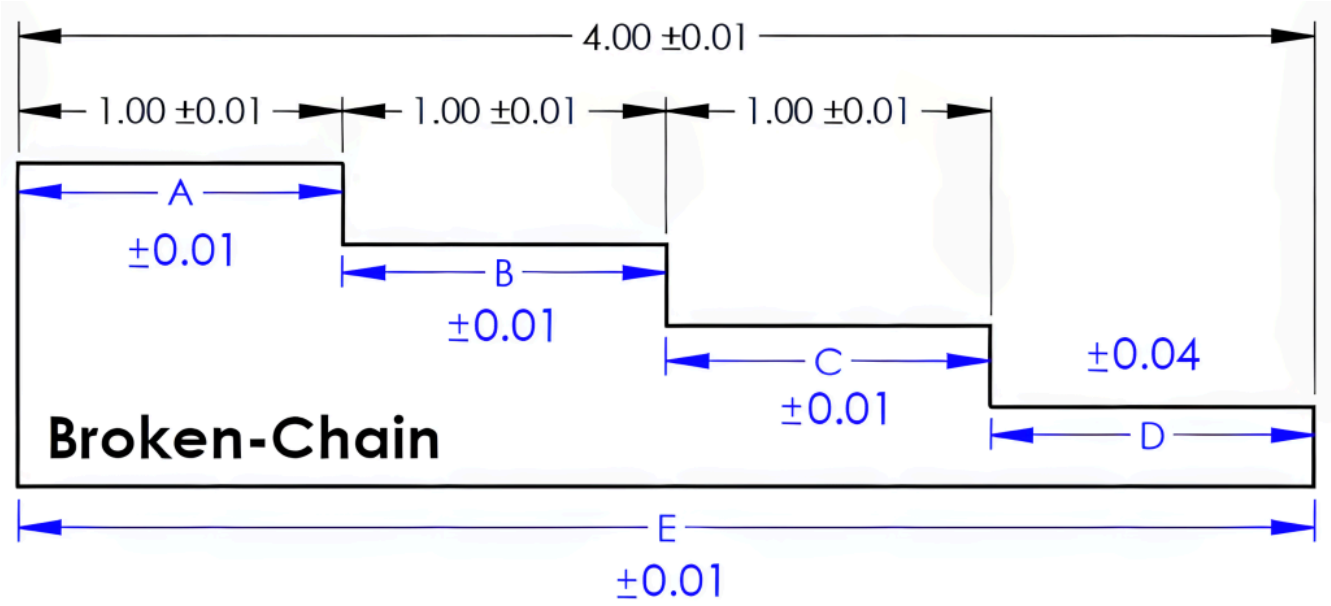


Figure 8-12. Broken-chain dimensioning effects on tolerance.

8.9 REFERENCE DIMENSION

In these scenarios above, notice that if all four lengths and the overall length dimensions were to be included, the tolerances would interfere with each other. If a drawing contains all these dimensions, it would create what is referred to as an over-dimensioned drawing. However, these missing dimensions may be included without carrying a tolerance. These non-tolerance dimensions are known as **reference dimensions**. The drafter will indicate the dimension as a reference by enclosing the value in parentheses, or the value will be followed by the abbreviation REF, as shown in Figure 8-13. Reference dimensions can be useful in a variety of ways, such as in identifying stock sizes.

Figure 8-13. Dimensions noted as reference will not include a tolerance value.

The reference dimension example in Figure 8-14 is included to help the reader identify the 0.85 dimension without calculating the value. However, the tolerance that would accumulate in calculating this value from the other dimensions will still be applied. Therefore, these reference dimensions should not be inspected, as they do not carry tolerance values.

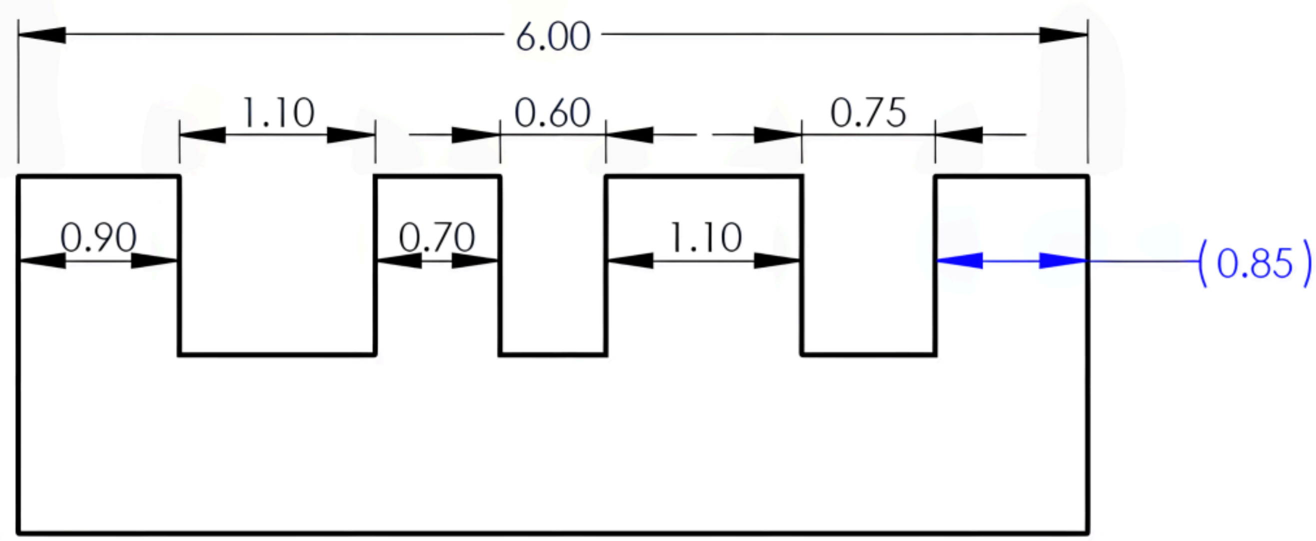


Figure 8-14. The reference dimension is included for the reader. No tolerance value is applied to reference dimensions.

LEARNING ACTIVITIES

Exercise 8.9-1 Tolerance

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=315#h5p-45

Exercise 8.9-2 Tolerance

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=315#h5p-46

Exercise 8.9-3 Tolerance

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=315#h5p-47

Exercise 8.9-4 Tolerance

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=315#h5p-48

Exercise 8.9-5 Tolerance

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=315#h5p-49

References:

Barsamian, M. A., & Gizelbach, R. (2022). *Machine trades print reading*. Goodheart-Willcox.

Schultz, R. L., & Smith, L. L. (2012). *Blueprint reading for machine trades* (7th ed.). Pearson.

TLDR Technologies. (2024). *Simplified AI*. [Large language model]. https://simplified.com/ai-writer

Images:

All images by Mark Lorier and licensed under CC BY-NC 4.0 except where otherwise noted.

9. Print Symbols and Notes

9.1 INTRODUCTION

Learning Objectives

- Interpret notes commonly found on prints
- Identify symbols found on prints
- Interpret surface texture symbols found on prints
- Interpret machining symbols found on prints
- Interpret common geometric symbols found on prints

Terms

- General note
- Local note
- Fillet
- Round
- Counterbore
- Spotface
- Countersink
- Chamfering
- Depth symbol
- Surface texture
- Surface roughness
- Surface texture symbol

- Waviness
- Lay
- Surface finish comparator
- Profilometer
- Datum
- Basic dimension

Surface finish symbols offer essential insights into the desired texture and quality of a component's surface. They outline key parameters like roughness, waviness, and lay patterns—guiding manufacturers toward achieving the ideal finish that balances functionality and aesthetics.

Geometric symbols are equally important in print reading. They clarify how parts should be shaped, oriented, or positioned relative to one another. These symbols outline critical dimensions pertaining to angles, radii, concentricity, and symmetry, among other features.

Prints often contain notes that provide supplementary information or instructions in addition to symbols. These notes may include material specifications, manufacturing processes, assembly instructions, safety warnings, or any other relevant details crucial for successfully interpreting the design.

9.2 DRAWING NOTES

In reviewing drawings, you might come across various notes that serve essential purposes. These notes will fall into one of two categories: a general note or a local note. **General notes** relate to the entire print, typically positioned above or near the title block. These may include vital details like tolerances, finishes, or material specifications as shown in Figure 9-1.

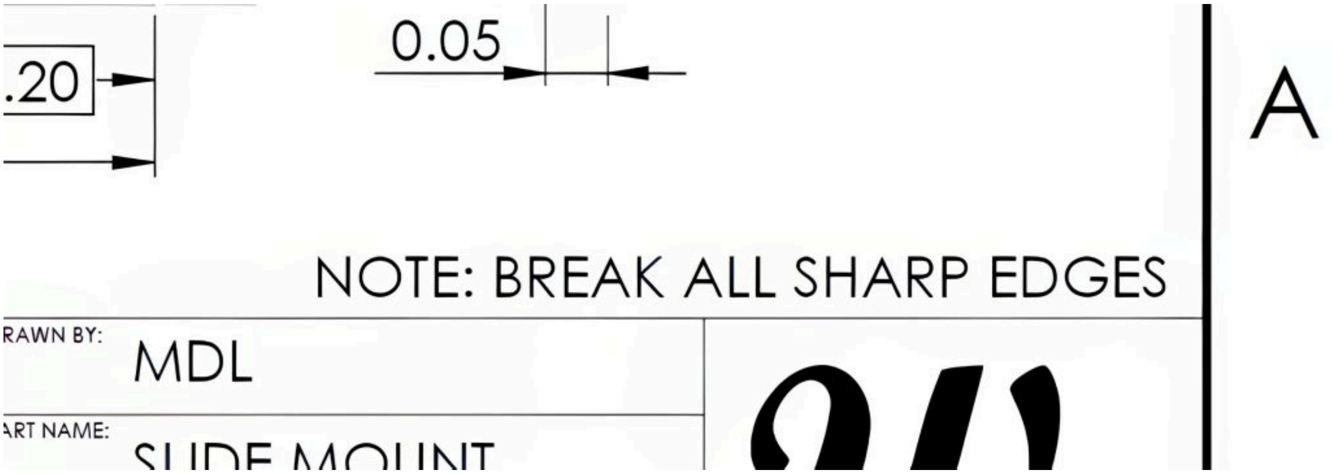


Figure 9.1. General note applies to the entire drawing and is often placed near the title block.

Local notes are more specific notes that address only particular drawing features. These local notes will typically use leaders to point directly to where they apply—for instance, providing information about hole sizes, a chamfer, radius, or special features, as depicted in Figure 9-2. Paying attention to both types of notes is essential as they contribute significantly to understanding the drawing's intent and requirements.

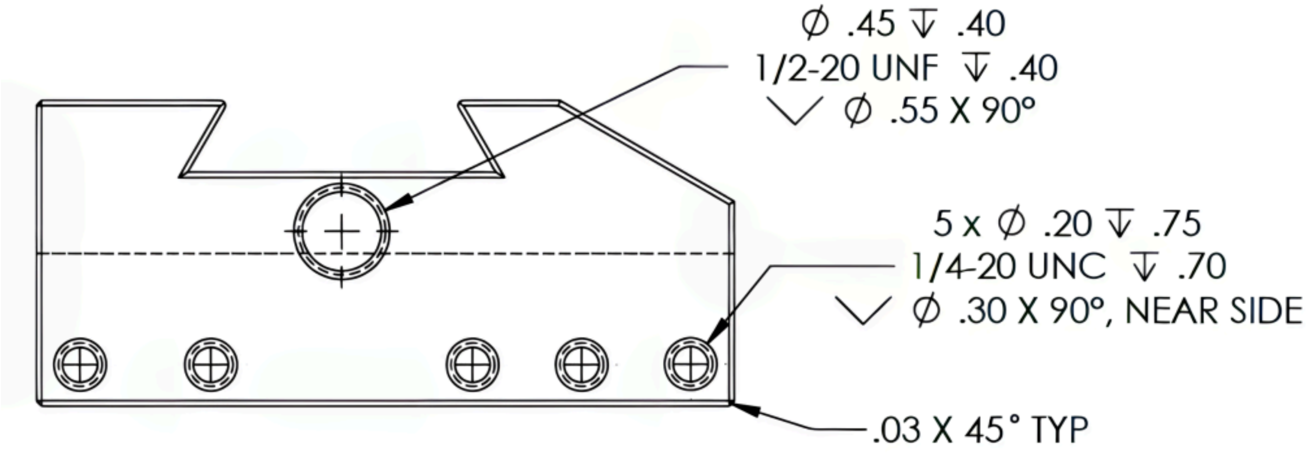


Figure 9-2. Local notes apply only to a specific area or feature.

9.3 DIMENSIONING SYMBOLS

As the use of prints becomes increasingly global, there's been a meaningful shift toward international standardization in drawings. This change often favors symbols over words and abbreviations for clarity and understanding. Below are some standard symbols commonly utilized in dimensioning systems with which you should become familiar.

The diameter symbol identifies a circular feature. See Figure 9-3. A diameter symbol is used with a value to apply to the circle's diameter size. Previous drawing standards have used the abbreviation for diameter, "DIA." Although this DIA abbreviation is no longer a current standard, this may still be observed in drawings.

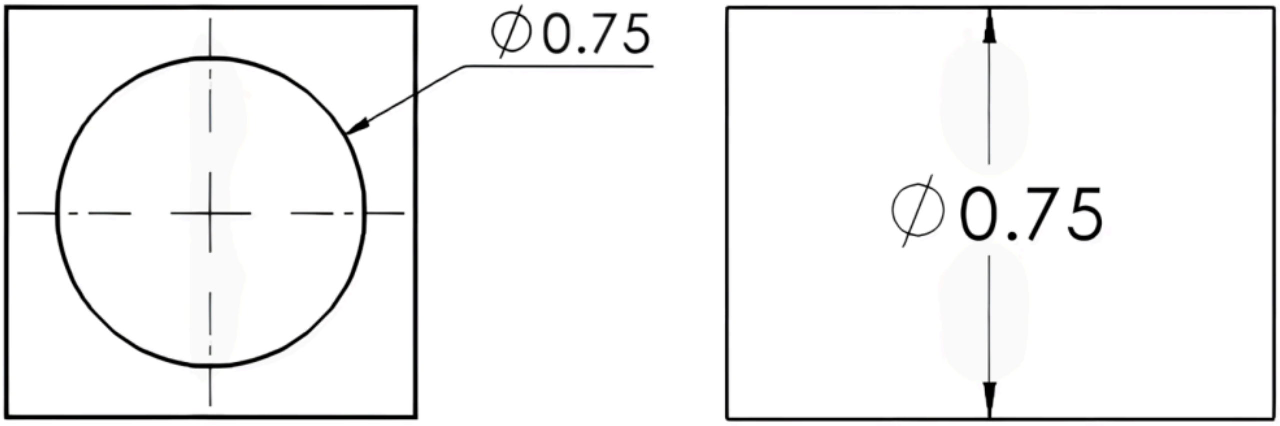


Figure 9-3. The diameter symbol is used to identify the hole size and an outside diameter size.

The "R" for radius or radii is used to identify a partial or complete circular feature. The "R" may be used instead of diameter when a circle is dimensioned from the center. A radius value is also used in partial circular features or when removing sharp edges of surface intersections. Radius values are also applied at intersections of surfaces to avoid sharp edges or corners. The internal radii are called **fillets**; external radii may be called **rounds**, as shown in Figure 9-4.

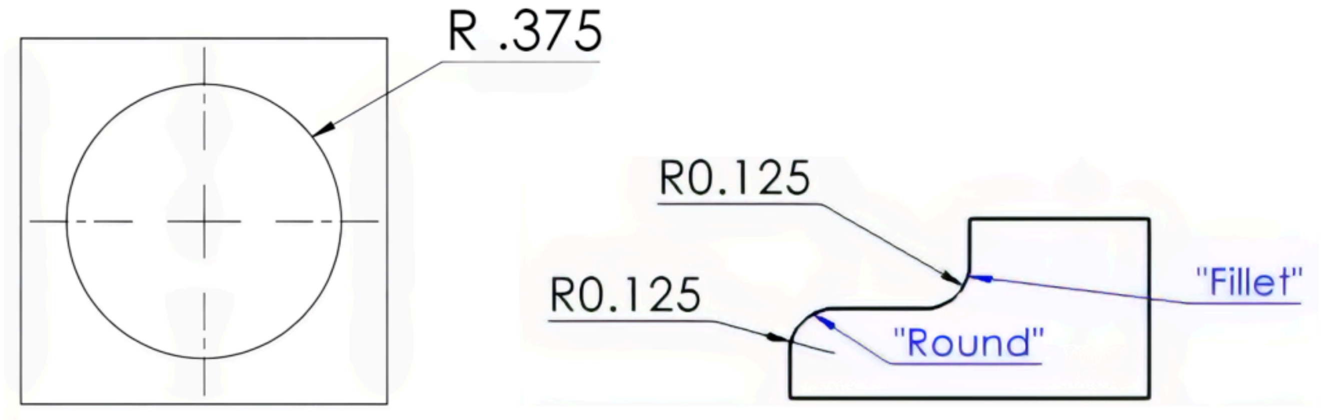


Figure 9-4. A fillet is the internal radius, while a round is an external radius. A radius value is identified by the letter "R."

The square symbol may be used to indicate a square feature with four equal sides and equal 90° angles. It may also eliminate the need for additional dimensioning. In Figure 9-5 this symbol identifies the features as square and eliminates the need to provide dimensions for the two sides because the sides of a square will be equal.

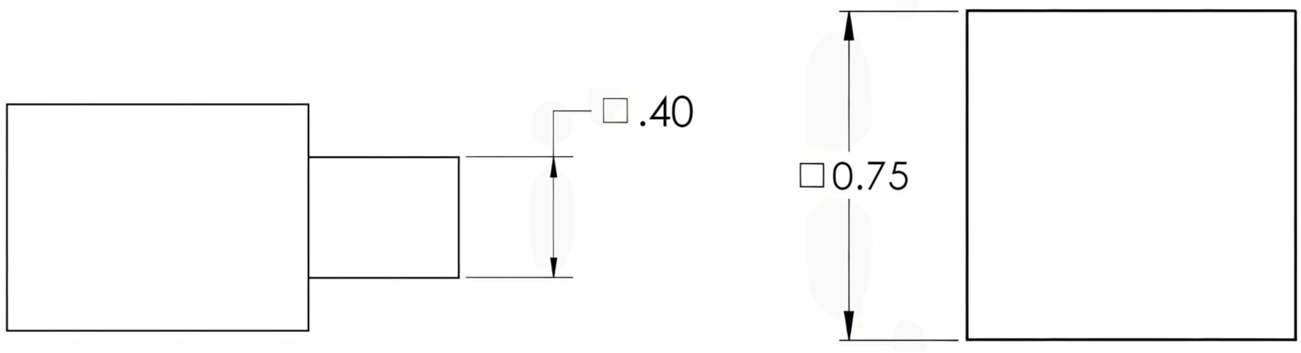


Figure 9-5. The square symbol is used to identify a feature as a square shape.

This squared "U" shaped symbol is used for a **counterbore**. A counterbore enlarges a hole at one end. A counterbore is commonly used to allow the head of a fastener to sit flush or below the surface. The abbreviations of "C'BORE" or "CBORE" for counterbore have been used in previous drawing standards and may still be observed on drawings. In Figure 9-6, the counterbore symbol is used to identify the features as being counterbores. In the Figure 9-6 examples, the first counterbore symbol is followed by the diameter and depth of the counterbore, and the second example only has the counterbore diameter following the symbol with the depth identified in the other view.

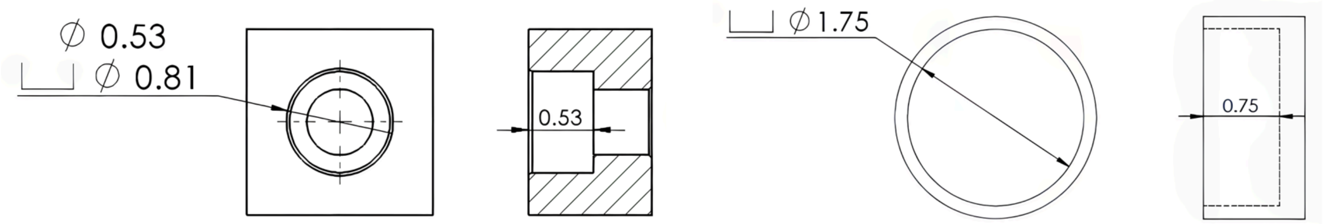


Figure 9-6. The counterbore symbol is used to identify the counterbore features.

With the addition of "SF," the counterbore symbol will indicate that a **spotface** operation is required. A spotface is an operation used to provide a smooth, flat, circular surface for a mating part or a fastener to contact the material around a hole. A spotface is often found on material that may not already be flat, such as a cast or molded part, where a fastener is placed near a fillet or raised surface, causing an interference. The depth of a spotface is typically .03 to .06 of an inch unless a specific depth of the spotface is noted or dimensioned. Previous drawing standards used the abbreviation "SFACE," which may still be found on drawings instead of the symbol. The example in Figure 9-7 uses the spotface symbol to describe the feature and the depth of the feature.

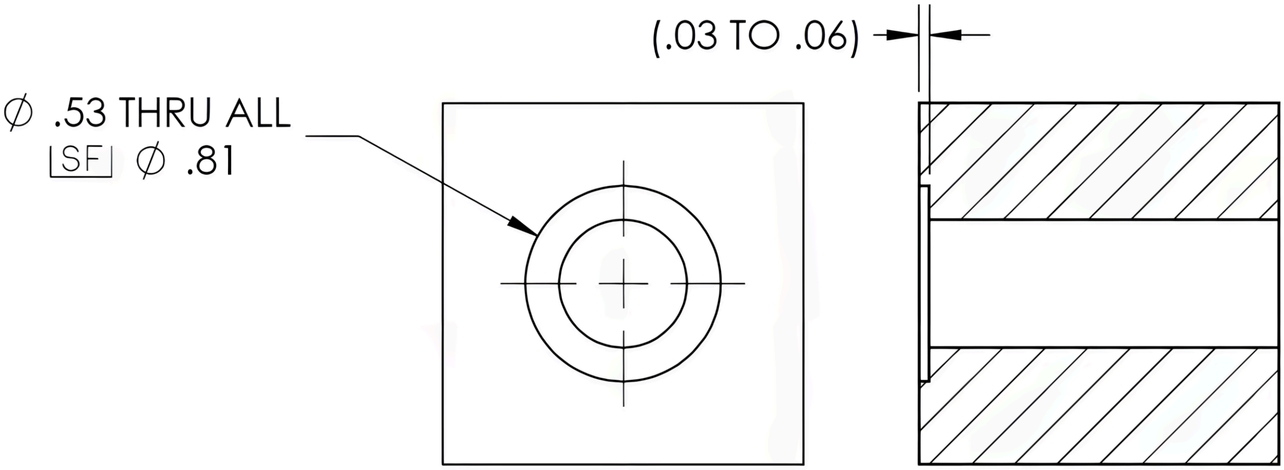


Figure 9-7. The spotface symbol is used to identify the hole feature.

Represented by a wide “V”-shaped symbol, a **countersink** is a cone-shaped recess created in the material at the end of a hole. Countersunk holes allow a flathead fastener to sit flush or below the surface. The abbreviation of “CSK” had been used in previous drawing standards and may still be seen on drawings. This countersink symbol is also used for **chamfering** or removing the sharp edges of the hole. The angle included for the countersunk holes will follow the symbol. These included angles are often either an 82° or a 90° angle. The included angle of a metric flathead fastener is 90°, while an inch flathead fastener will have an 82° angle. In the example of Figure 9-8, the countersink symbol is used to identify the feature shape with the size of the outside diameter of the countersink and the angle following the symbol.

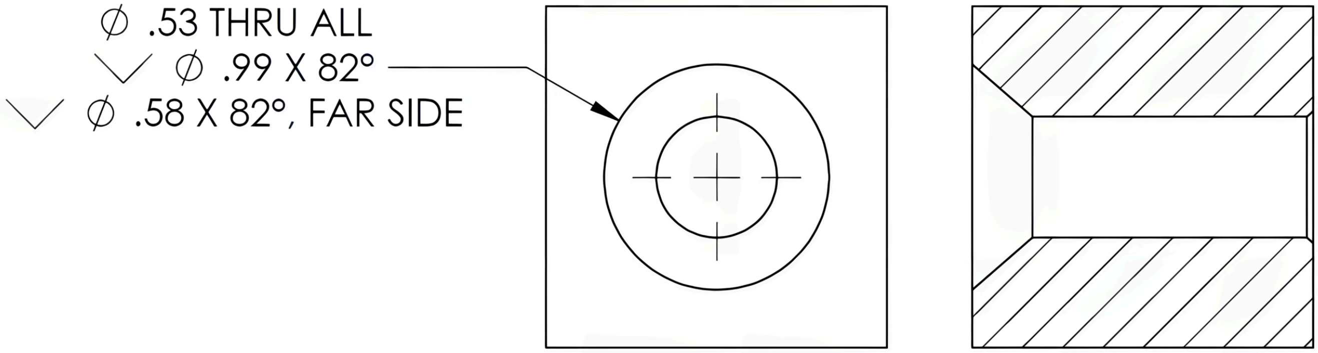


Figure 9-8. The countersink symbol is used to identify the hole feature.

The **depth symbol**, followed by a numerical value, is used in drawings to communicate the required depth for features such as holes, slots, or recesses. The symbol features an arrow pointing down with a horizontal line on the top. Previous drawing standards used the abbreviation “DP” in place of the depth symbol; this abbreviation may still be found in some drawings. Figure 9-9 features depth symbols followed by the depth value.

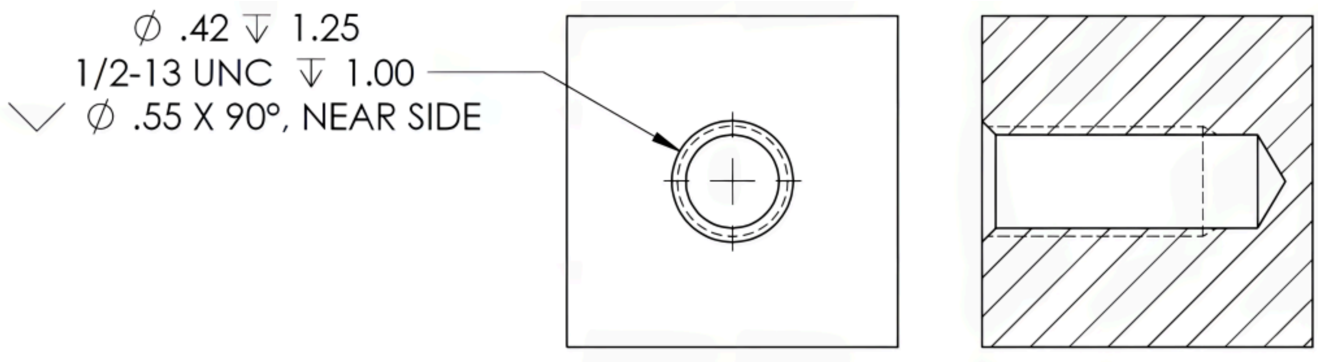
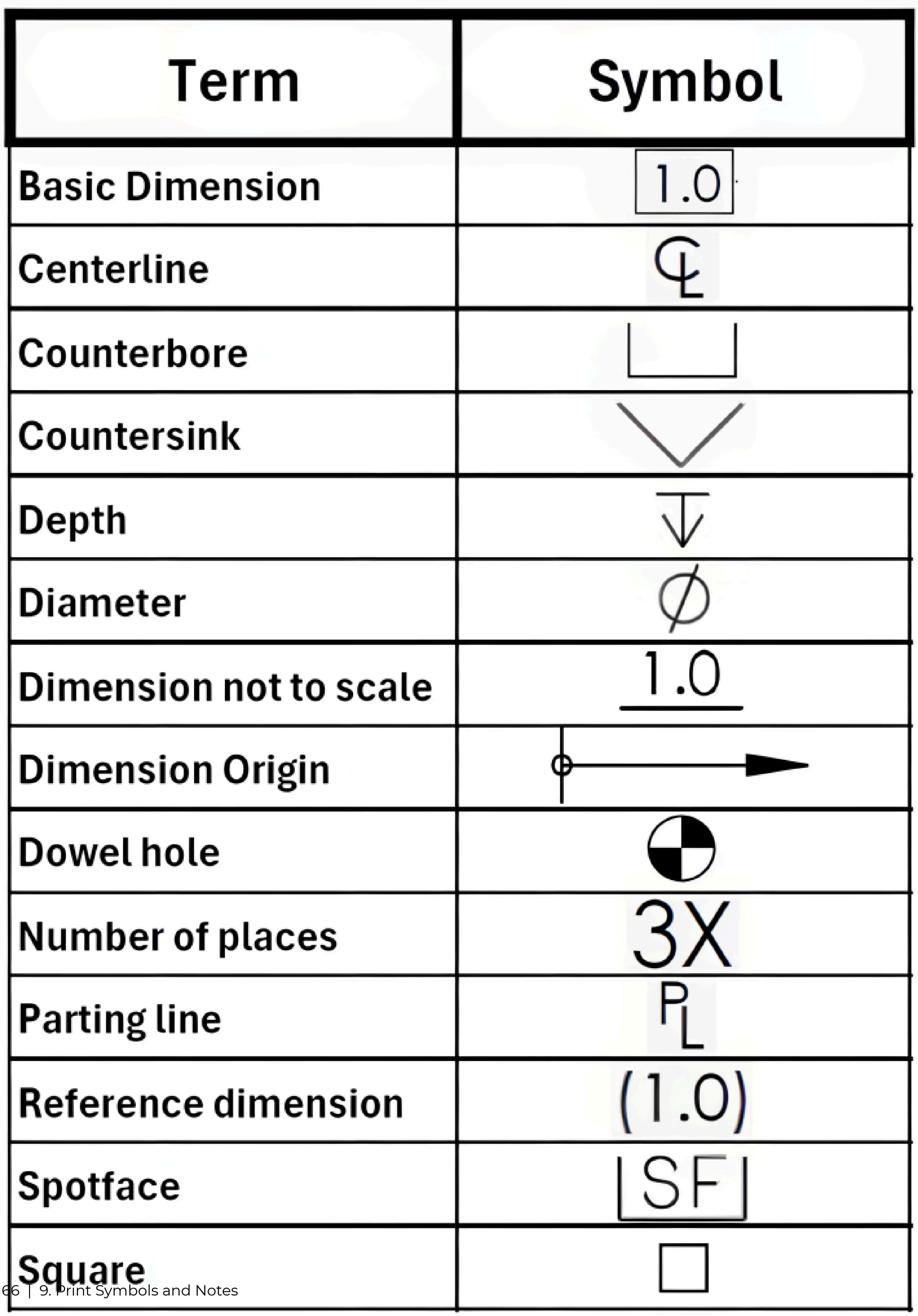


Figure 9-9. The depth symbol is used to identify a feature's depth.

Figure 9-10 below shows a list of these symbols and other symbols used in the ANSI and ISO engineering drawing standards.



Term	Symbol
Basic Dimension	$\boxed{1.0}$
Centerline	℄
Counterbore	⌴
Countersink	⌵
Depth	↧
Diameter	⌀
Dimension not to scale	$\underline{1.0}$
Dimension Origin	⊸──▶
Dowel hole	⊕
Number of places	3X
Parting line	⅊
Reference dimension	(1.0)
Spotface	⌴SF
Square	□

Figure 9-10. Drawing symbols commonly used in engineering drawings.

9.4 SURFACE TEXTURE

Surface texture refers to the comprehensive quality of a surface and is defined by its waviness, lay, and roughness characteristics. When collectively considering all aspects of waviness, lay, and roughness, the term “surface texture” is frequently used to prevent misunderstandings because machinists commonly use “surface finish” to denote surface roughness. Figure 9-11 illustrates waviness, lay, and roughness as they are applied to a part’s surface.

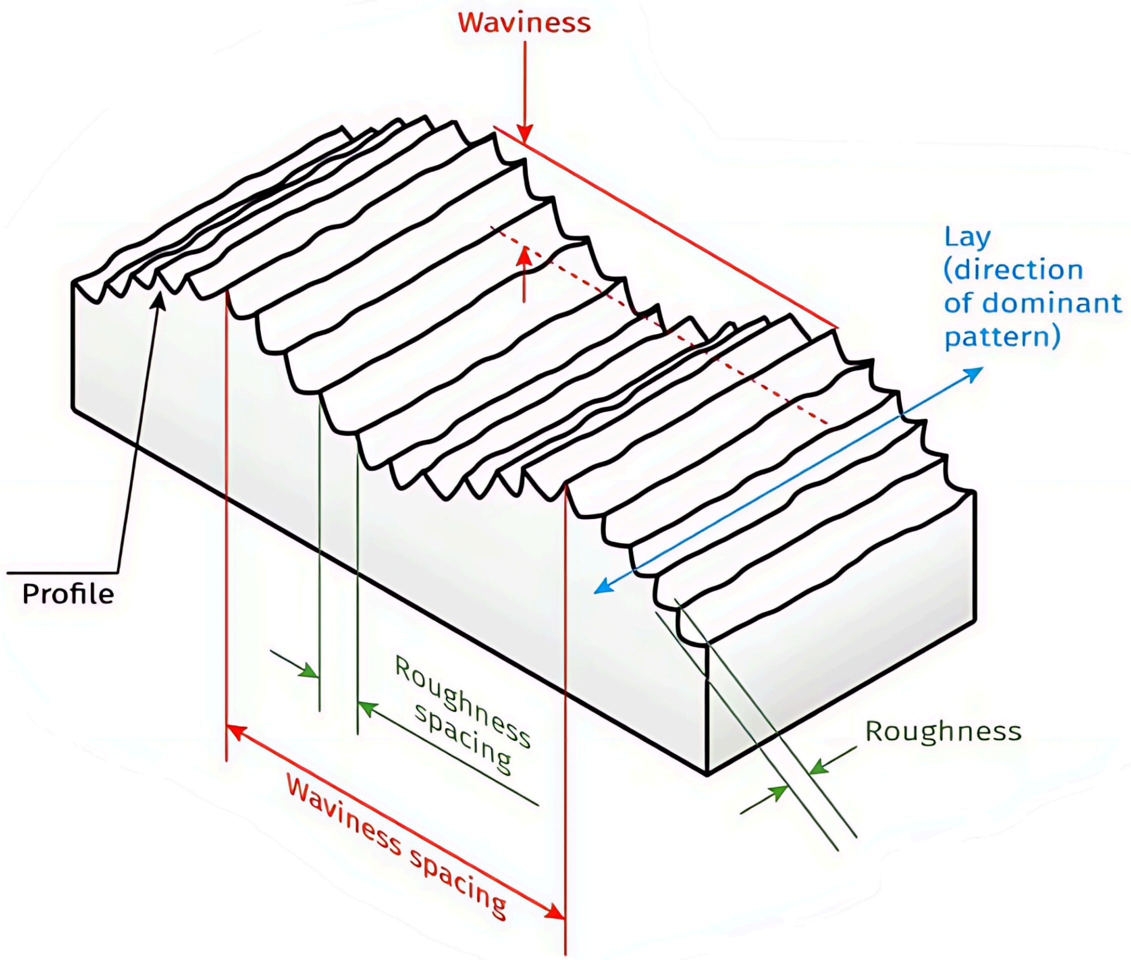


Figure 9-11. An illustration of lay, waviness, and roughness applied to a surface. Reused with permission.

Surface Roughness

Surface roughness refers to small irregularities in the surface geometry. It is often referred to as “surface finish” because it is the most commonly specified aspect of surface texture. This roughness value is measured as an average reading in microinches (millionths of an inch) or micrometers (millionths of a meter), meaning a reading of 63 would represent an actual value of .000063 inches, and a value of 1.6 would represent an actual value of .0000016 meters.

The basic **surface texture symbol** is a checkmark with the bottom point resting on the surface to be specified and the value above the point. Figure 9-12 displays basic surface finish symbols used to identify the required surface finish to apply to the features and surfaces of a part. A general note may be used to identify the finish of the entire part with the abbreviation “FAO” for Finish All Over.

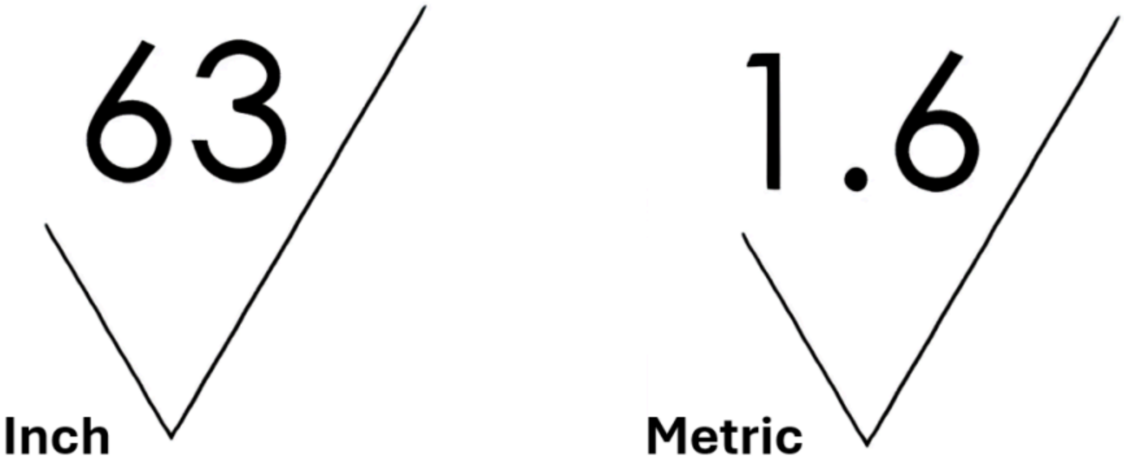


Figure 9-12. Basic surface texture symbol with inch and metric values.

With this open checkmark symbol, the designer does not specify the methods required to produce the desired surface finish. See the basic surface texture symbol in Figure 9-13.

Figure 9-13. Basic surface texture symbol.

Material Removal by Machining is Required

When a horizontal bar is added to the symbol, it indicates that the surface is to be machined to produce the finish required, as shown in Figure 9-14.

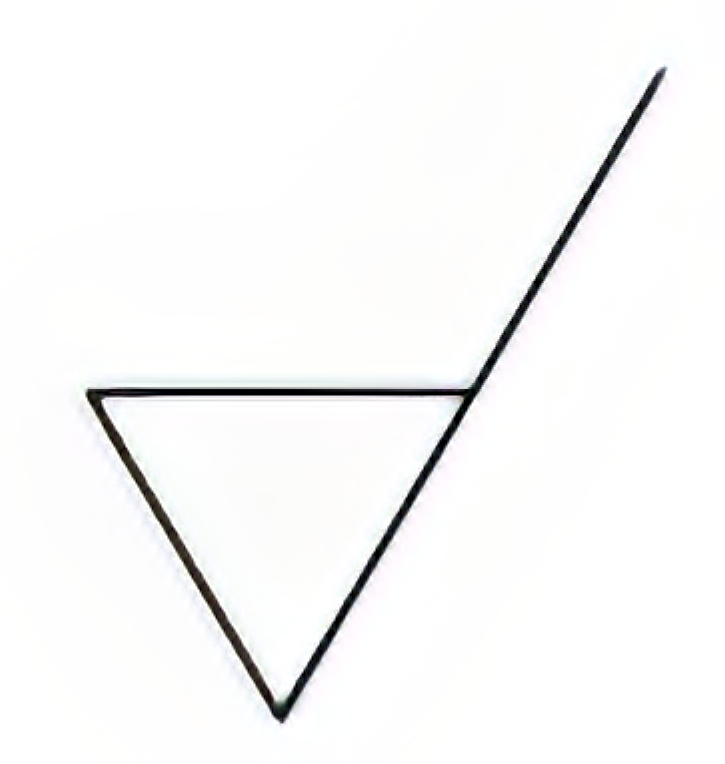


Figure 9-14. This symbol indicates machining is required to obtain the desired surface finish.

Material Removal Prohibited

With a circle placed in the check symbol, the required finish must be obtained without removing material. The finish must be produced by processes such as casting, die casting, or injection molding. View Figure 9-15.

Figure 9-15. This symbol indicates no material may be removed to obtain the desired surface finish.

To add additional information, including sampling length, waviness, and direction of the surface lay, a horizontal line is added to the check mark symbol. View Figure 9-16.

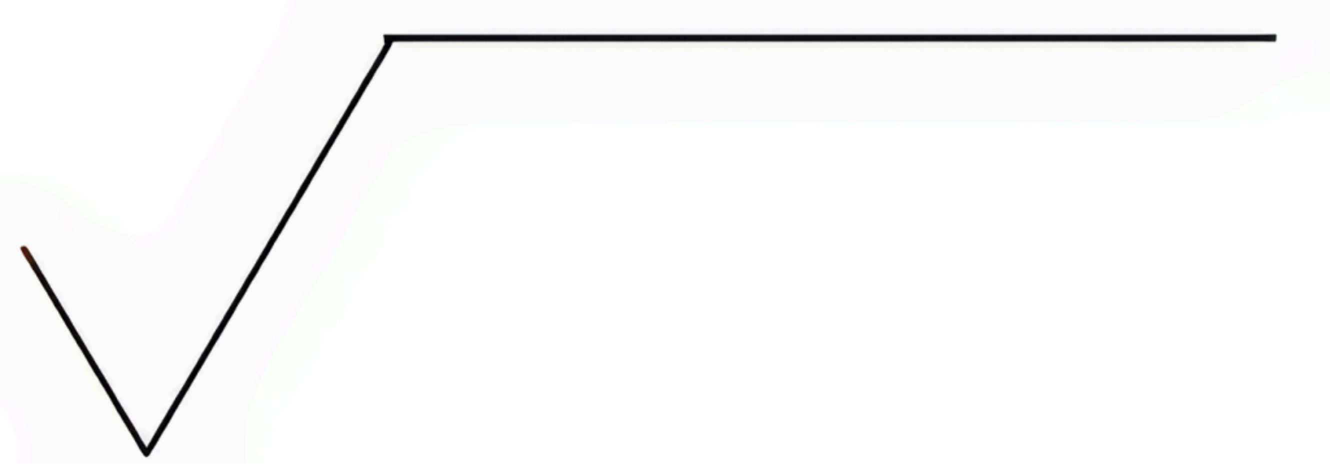


Figure 9-16. A surface texture symbol with a horizontal line is used to provide additional surface information.

As with many of the standard practices provided by the ANSI and ISO drawing standards, the surface texture rules have also evolved. Figure 9-17 illustrates a surface texture symbol with commonly used placement of surface texture information, although some variation may be observed between drawing standards. This example provides placement for maximum roughness average (Max Ra) and minimum roughness average (Min Ra), maximum waviness height and width, roughness sampling length and spacing, and lay symbol.

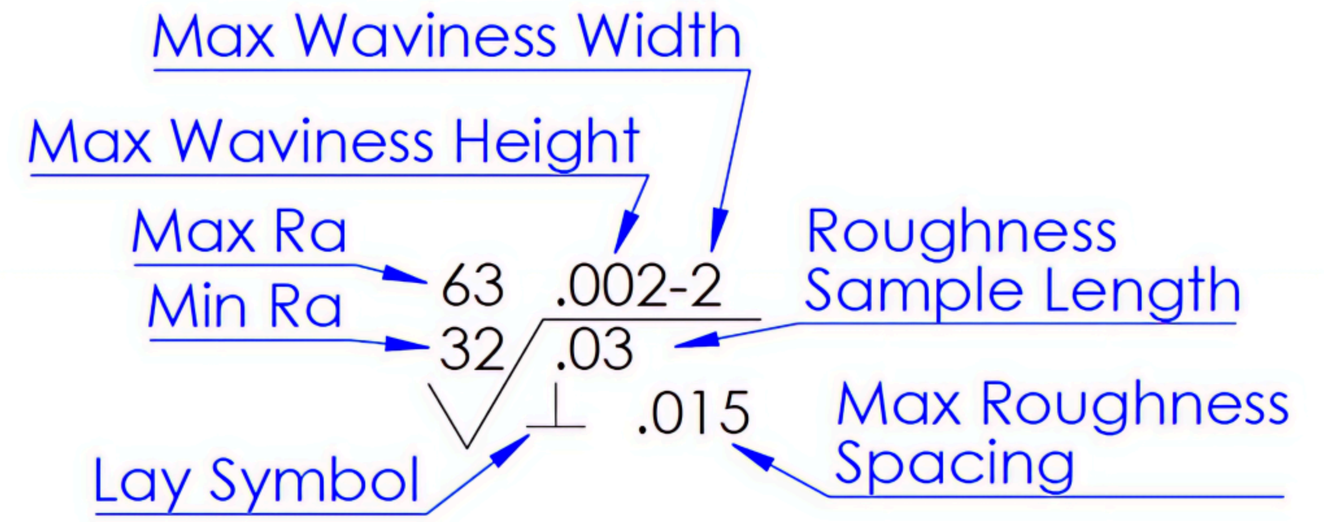


Figure 9-17. Illustration of placement for all surface texture information, which may be used with a surface texture symbol.

Waviness

Waviness refers to the irregularities or deviations from a desired smooth surface that occur over a longer distance than roughness. It represents the larger-scale variations in the surface texture.

Waviness is typically characterized by longer wavelength components, which can result in peaks and valleys on the surface. These deviations are often caused by factors such as machine vibrations, tool wear, or inherent material properties. Waviness height represents the vertical distance between the highest peak and lowest valley within a specific length or area. Waviness spacing refers to the distance between successive peaks or valleys. Waviness can be vital in certain components, such as bearing surfaces or sealing surfaces.

Lay

Lay is the dominant pattern of the surface and the orientation of that pattern. The manufacturing process generates the patterns and orientations of the patterns. These patterns can be parallel, perpendicular, circular, cross-hatched, radial, multi-directional, or isotropic (non-directional). Figure 9-18 displays the lay symbols that may be used.

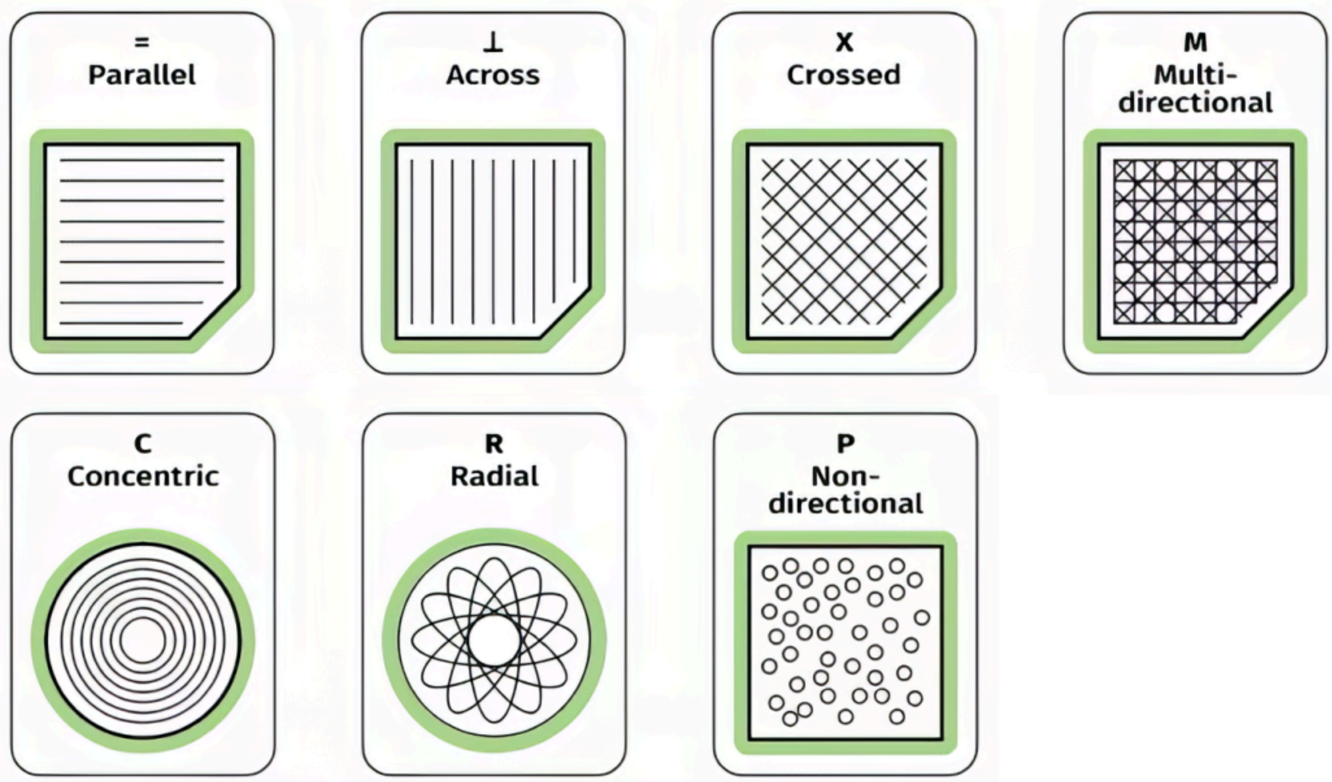
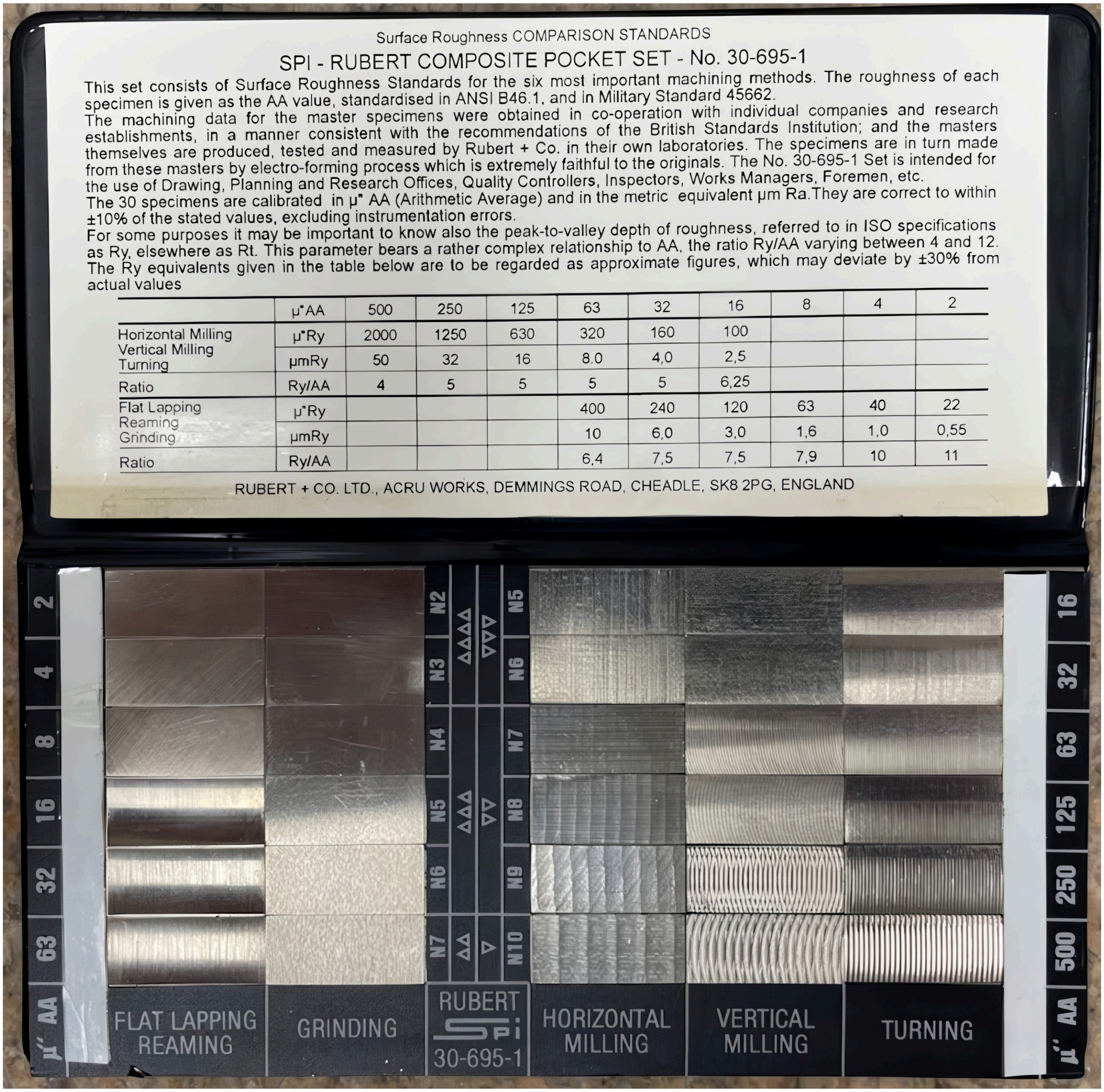


Figure 9-18. Lay symbols that may be used with a surface texture symbol. Reused with permission.

Measurement of Roughness

Surface roughness can be compared to a surface finish reference plate called a **surface roughness comparator**, as shown below in Figure 9-19. This can give a general measurement by best matching the surface roughnesses. Surface roughness is more accurately measured by an instrument known as a **profilometer**. The profilometer shown in Figure 9-20 uses a small stylus moving across the surface to measure the small peaks and valleys of the surface and provides an average of these measurements to provide the surface finish reading, which is illustrated in Figure 9-21. The profilometer generates a graph showing variations in surface height with changes in position.



Surface Roughness COMPARISON STANDARDS

SPI - RUBERT COMPOSITE POCKET SET - No. 30-695-1

This set consists of Surface Roughness Standards for the six most important machining methods. The roughness of each specimen is given as the AA value, standardised in ANSI B46.1, and in Military Standard 45662.
The machining data for the master specimens were obtained in co-operation with individual companies and research establishments, in a manner consistent with the recommendations of the British Standards Institution; and the masters themselves are produced, tested and measured by Rubert + Co. in their own laboratories. The specimens are in turn made from these masters by electro-forming process which is extremely faithful to the originals. The No. 30-695-1 Set is intended for the use of Drawing, Planning and Research Offices, Quality Controllers, Inspectors, Works Managers, Foremen, etc.
The 30 specimens are calibrated in μ" AA (Arithmetic Average) and in the metric equivalent μm Ra.They are correct to within ±10% of the stated values, excluding instrumentation errors.
For some purposes it may be important to know also the peak-to-valley depth of roughness, referred to in ISO specifications as Ry, elsewhere as Rt. This parameter bears a rather complex relationship to AA, the ratio Ry/AA varying between 4 and 12. The Ry equivalents given in the table below are to be regarded as approximate figures, which may deviate by ±30% from actual values

	μ"AA	500	250	125	63	32	16	8	4	2
Horizontal Milling Vertical Milling Turning	μ"Ry	2000	1250	630	320	160	100			
	μmRy	50	32	16	8.0	4,0	2,5			
Ratio	Ry/AA	4	5	5	5	5	6,25			
Flat Lapping Reaming Grinding	μ"Ry				400	240	120	63	40	22
	μmRy				10	6,0	3,0	1,6	1,0	0,55
Ratio	Ry/AA				6,4	7,5	7,5	7,9	10	11

RUBERT + CO. LTD., ACRU WORKS, DEMMINGS ROAD, CHEADLE, SK8 2PG, ENGLAND

Figure 9-19. Surface roughness comparator.

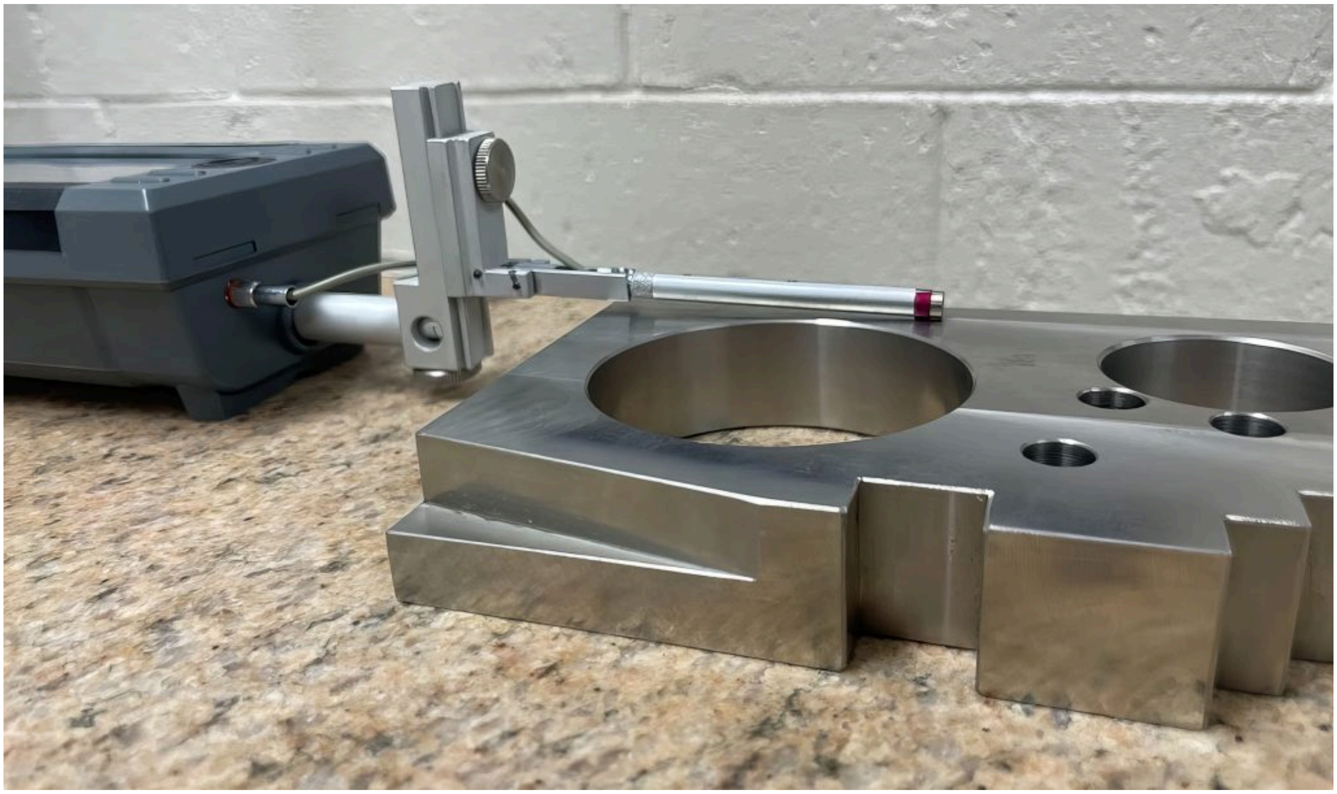


Figure 9-20. Surface roughness profilometer.

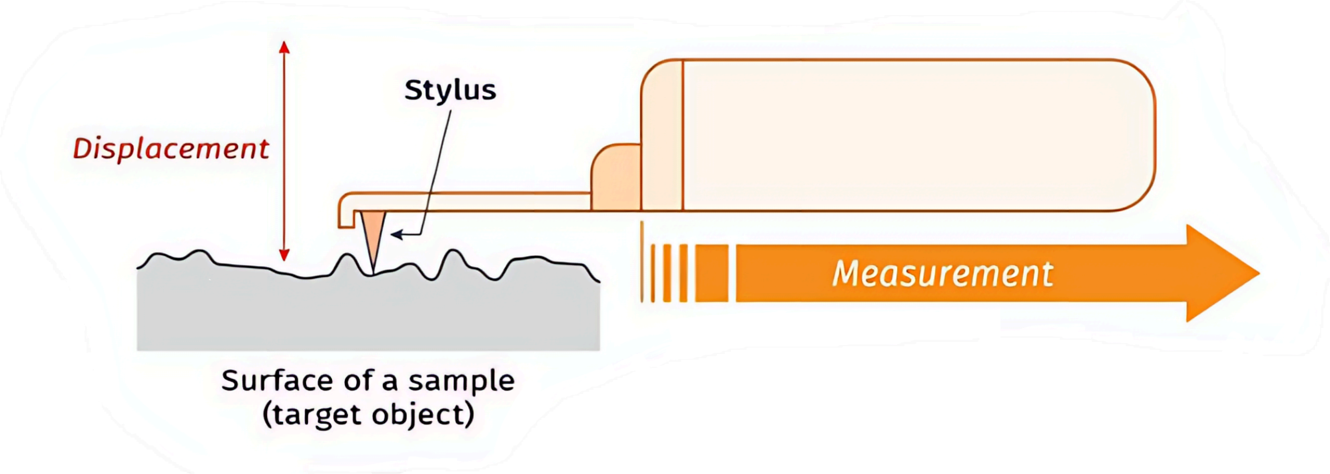


Figure 9-21. Illustration of a profilometer measurement. Reused with permission.

The graph in Figure 9-22 is known as the measured surface profile. It will show not only roughness but also any wave and flatness defects that may be present.

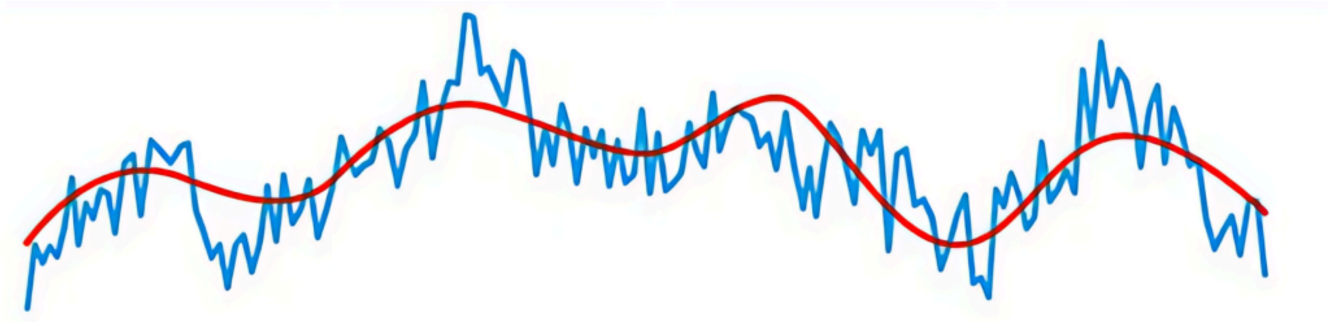


Figure 9-22. Illustration of roughness with waviness. Reused with permission.

We must remove the wave and flatness defects from the profile to examine only roughness. Without smoothing the profile, the mean line will represent surface height variation due to waviness and flatness defects. To measure roughness only, waviness is removed, and the profile line is straightened. This new, straighter line is known as the surface roughness profile, as depicted in Figure 9-23.

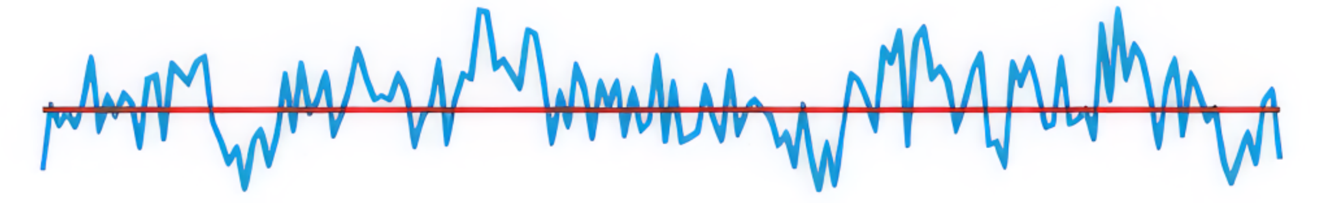


Figure 9-23. Illustration of roughness only. Reused with permission.

The roughness average (Ra) is the average of the measured height variations across the sampled length, as illustrated in Figure 9-24.

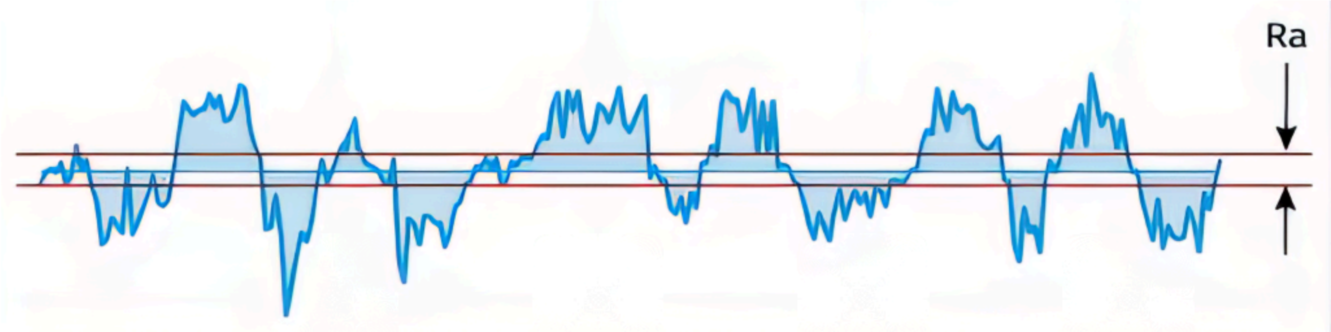


Figure 9-24. Illustration of roughness average. Reused with permission.

Surface Roughness Chart

The information in Figure 9-25 represents the typical roughness values resulting from various production processes. Some examples from the chart indicate that a drilling process will provide a surface finish between 250 and 63 roughness average (Ra), while a grinding process will provide a surface finish between 63 and 4 Ra.

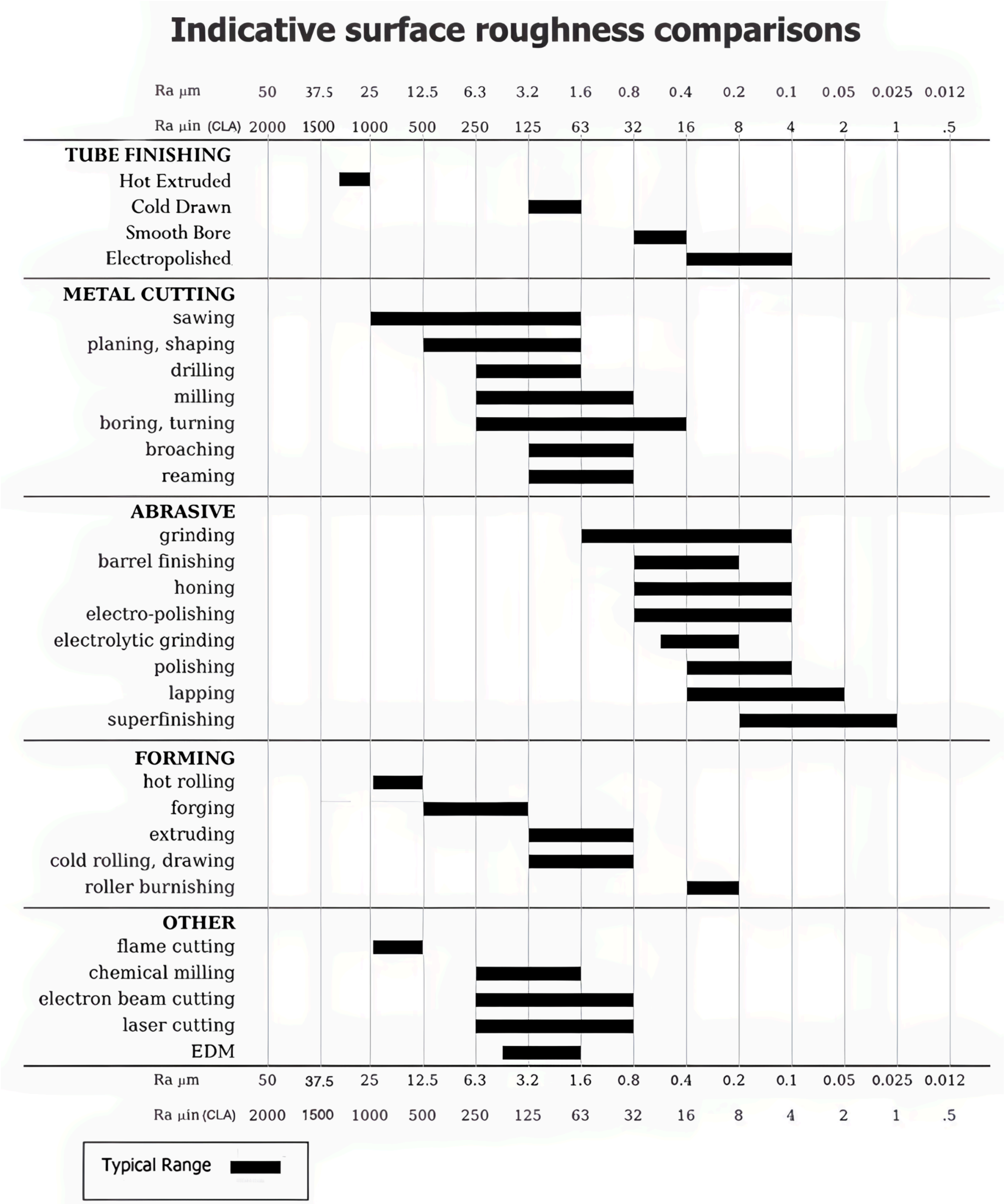


Figure 9-25. This chart displays the surface roughness range for common metal production processes. Reused with permission.

9.5 GEOMETRIC DIMENSIONING AND TOLERANCING (GD&T)

Feature Control Frame, Datum, and Basic Dimension

A feature control frame is used in geometric dimensioning and tolerancing to describe the conditions and tolerances of a geometric control on a part's feature. The feature control frame consists of four pieces of information, along with the datums and basic dimension information, referenced in Figure 9-26.

1. Symbol – The first box will contain the geometric symbol being identified.
2. Tolerance – The next box contains the tolerance and possibly a diameter symbol.
3. Modifiers – After the tolerance, a modifier may be used, such as a circled "M" or "L" for Maximum Material Condition and Least Material Condition.
4. Datum Reference – This may be used to reference a surface or feature.
5. **Datums** – Datums are features or surfaces that are used to establish a reference point or plane. The datum feature symbol is a square frame with the identifying letter inside, ending with a triangle extending from the surface.
6. **Basic Dimension** – A basic dimension is a theoretically exact value without a tolerance. The dimension is enclosed in a rectangular frame, indicating that it does not carry the tolerance defined by the title block but uses the positional tolerance identified by the feature control frame.

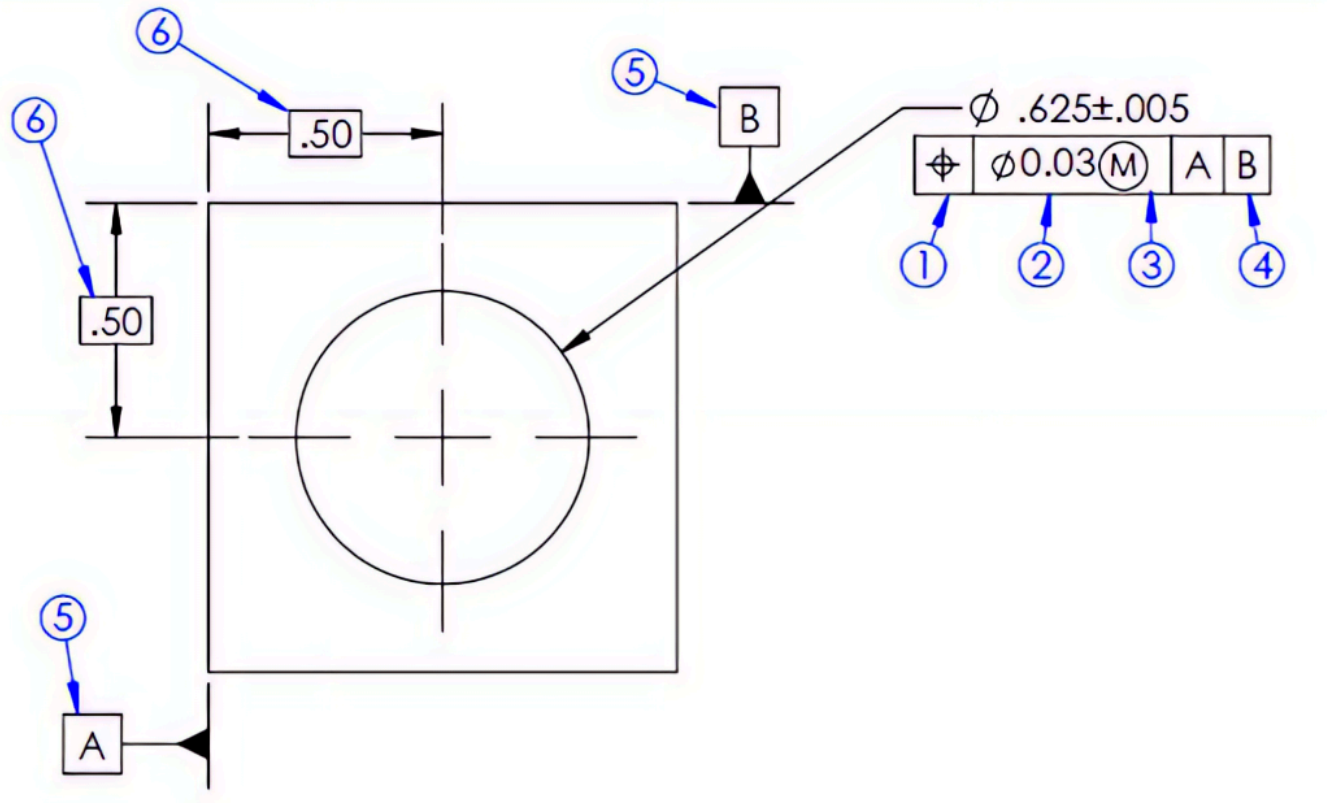
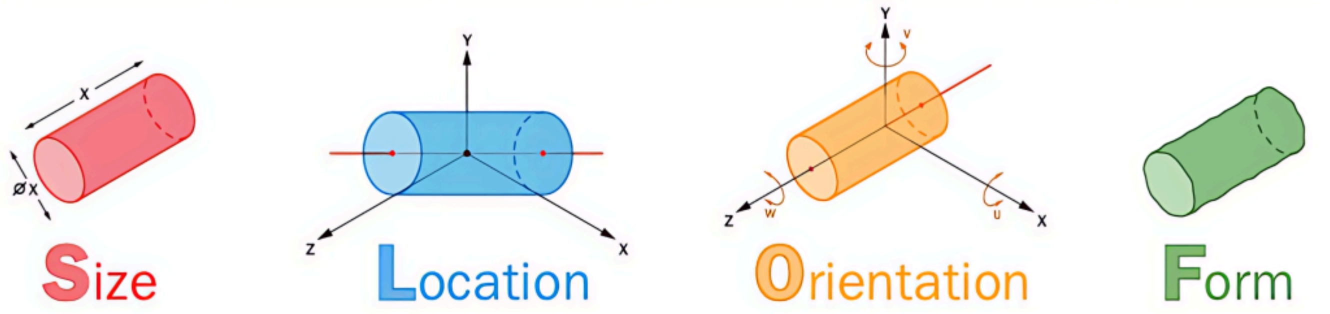


Figure 9-26. Feature control frame, datum, and basic dimension example.

You may be familiar with the term **geometric dimensioning and tolerancing**, or GD&T, which is a set of rules and symbols used to communicate the intent of a design, focusing on the function of the part. The use of GD&T may prevent an otherwise functional part from being rejected with traditional dimensions and tolerances. These geometric characteristics fall into four fundamental categories: size, location, orientation, and form. Study the symbols and descriptions below to relate the symbol with the characteristic it represents.



Reused with permission.

Form Tolerances

Form tolerance is a geometric tolerance that determines and controls the form or shape of the part or feature. Form tolerances do not reference datums.

Straightness

Surface straightness is a two-dimensional tolerance that controls the form of a line somewhere on the surface of the feature. Axis straightness is a tolerance that controls how much curve is allowed in the part's axis and depicted in Figure 9-27.

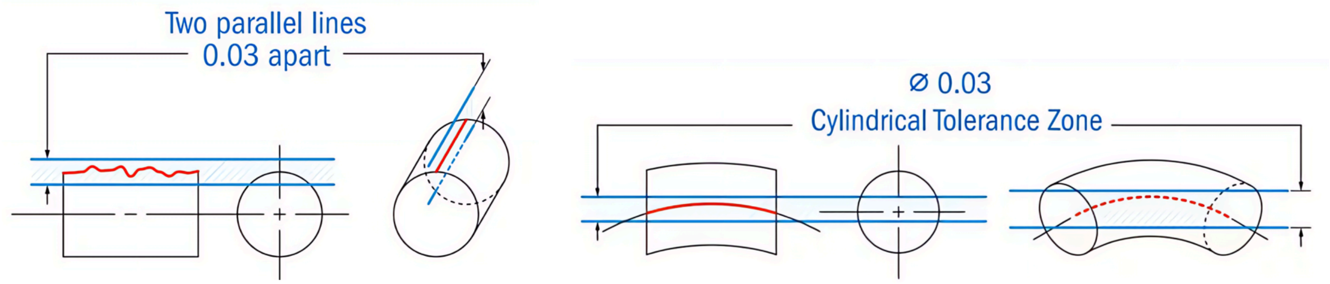
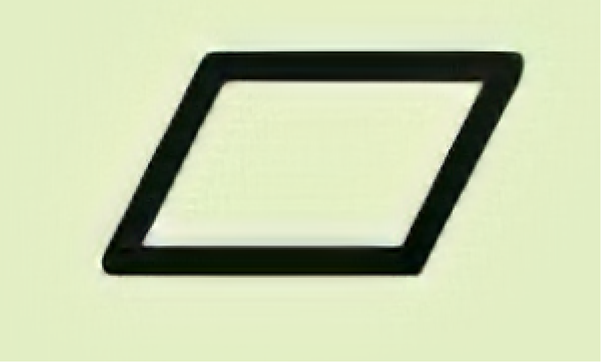


Figure 9-27. Straightness tolerance may be applied to a part's surface or the straightness of a part's axis. Reused with permission.

Flatness



The flatness is a three-dimensional tolerance that references two parallel planes (parallel to the surface that it is called out on) that define a zone where the entire reference surface must lie. Flatness tolerance will always be less than the dimensional tolerance associated with it. This is because the feature will be flat within its size tolerance, as shown in Figure 9-28.

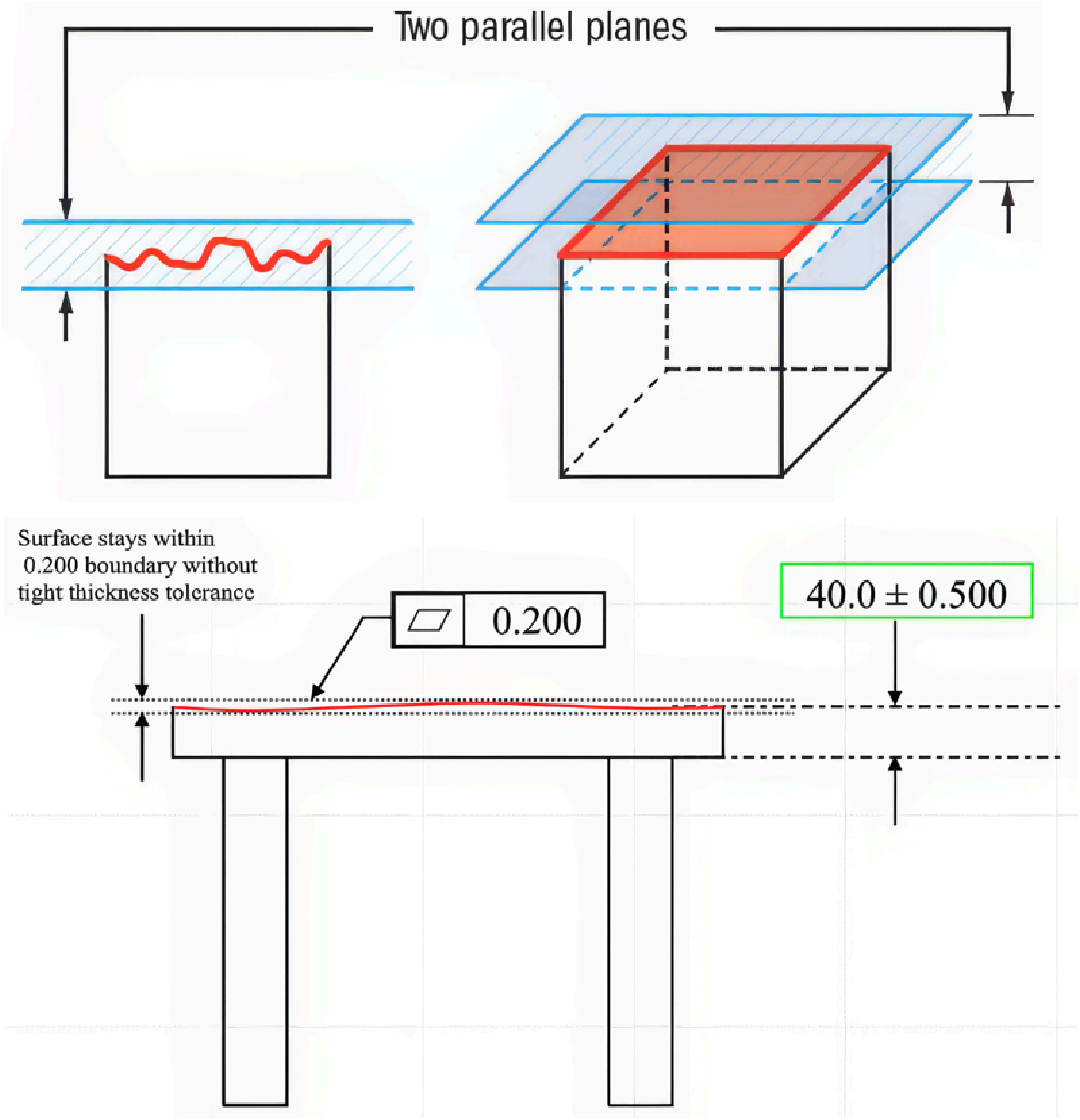


Figure 9-28. Flatness tolerance can control the flatness of a feature without requiring a tight size tolerance. Reused with permission.

Circularity

Sometimes called roundness, circularity is a two-dimensional tolerance that controls the overall form of a circle, ensuring it is not too oblong, square, or out of round. See Figure 9-29.

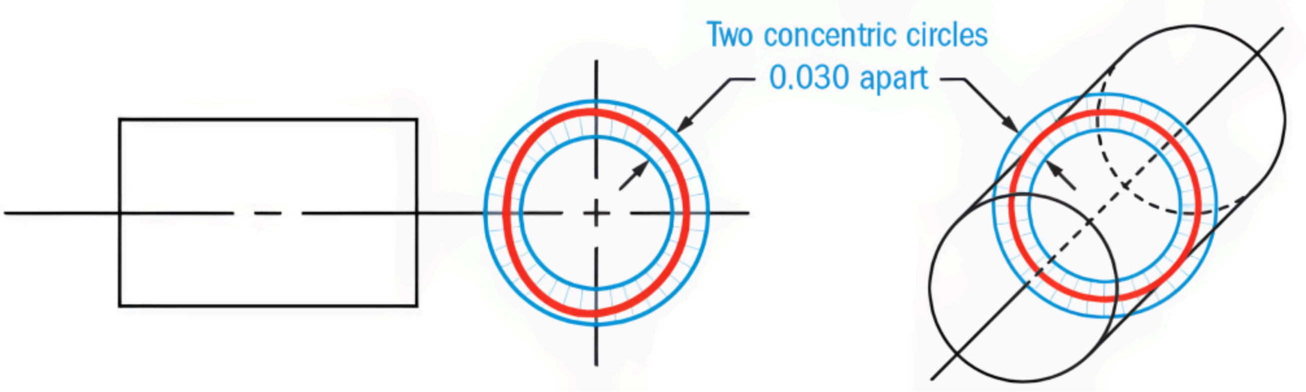


Figure 9-29. Circularity tolerance controls the form of a circle. Reused with permission.

Cylindricity

The cylindricity symbol is used to describe how close an object conforms to a true cylinder. Cylindricity is a three-dimensional tolerance that controls the overall form of a cylindrical feature to ensure that it is round enough and straight enough along its axis, as shown in Figure 9-30.

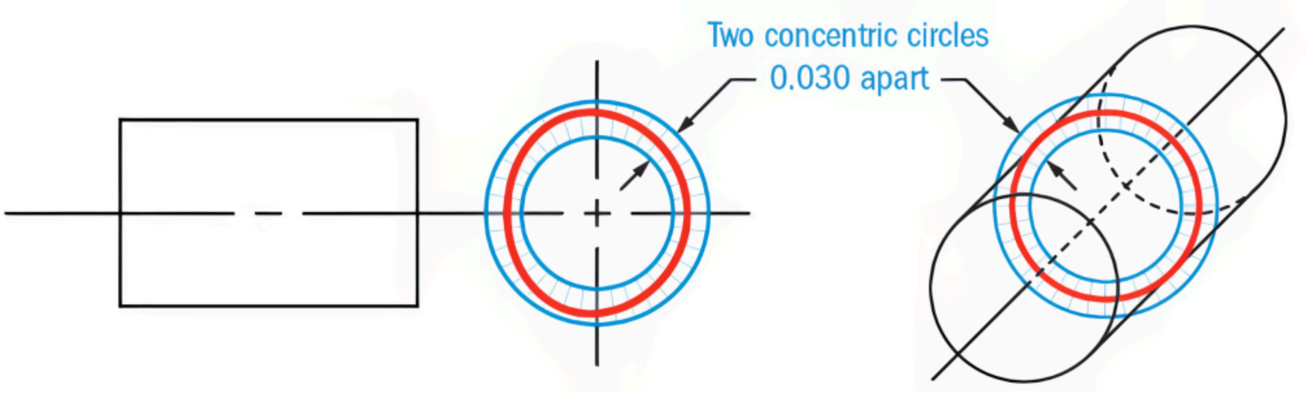


Figure 9-30. Cylindricity is used to control the form of a cylindrical shape. Reused with permission.

Profile Tolerances

Profile tolerance is a geometric, three-dimensional tolerance that defines a boundary around which the surface must stay within. Profile tolerances can use all of the four fundamental categories listed above.

Line Profile

Profile of a line describes a tolerance zone around any line in any feature, usually of a curved shape. The profile of a line is a two-dimensional tolerance range that can be applied to any linear tolerance. View Figure 9-31.

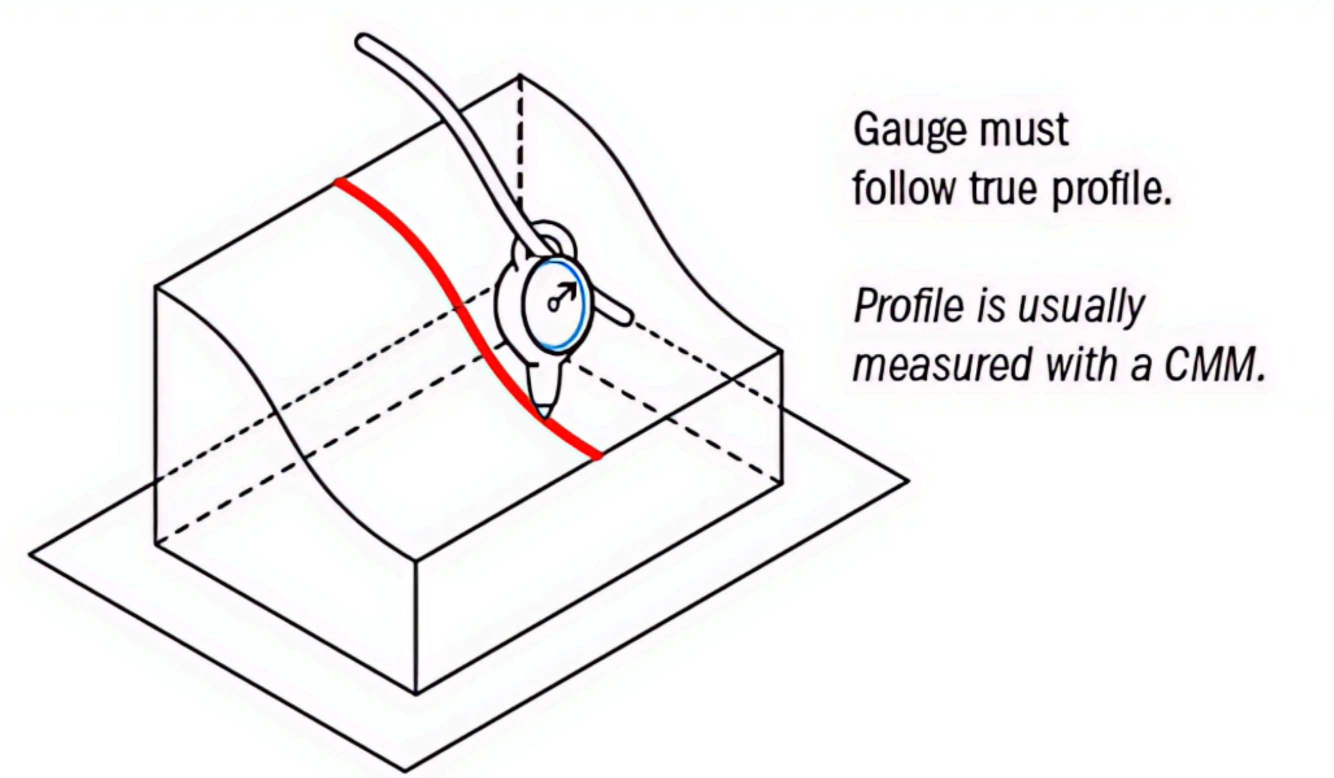
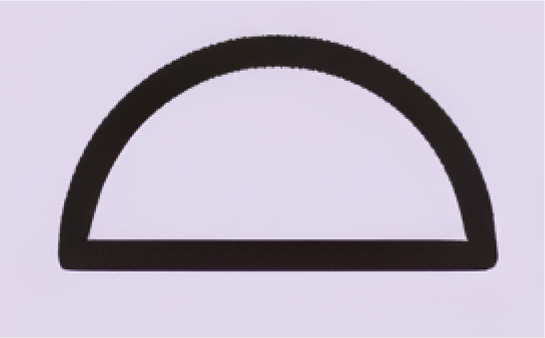


Figure 9-31. The profile of a line is a two-dimensional tolerance applied to a linear tolerance. Reused with permission.

Profile Surface



Profile of a surface describes a three-dimensional tolerance zone around a surface, which is usually an advanced curve or shape. Profile controls all the points along the surface within a tolerance range that directly mimics the designed profile. Any point on the surface would not be able to vary inside or outside by more than the surface profile tolerance, as shown in Figure 9-32.

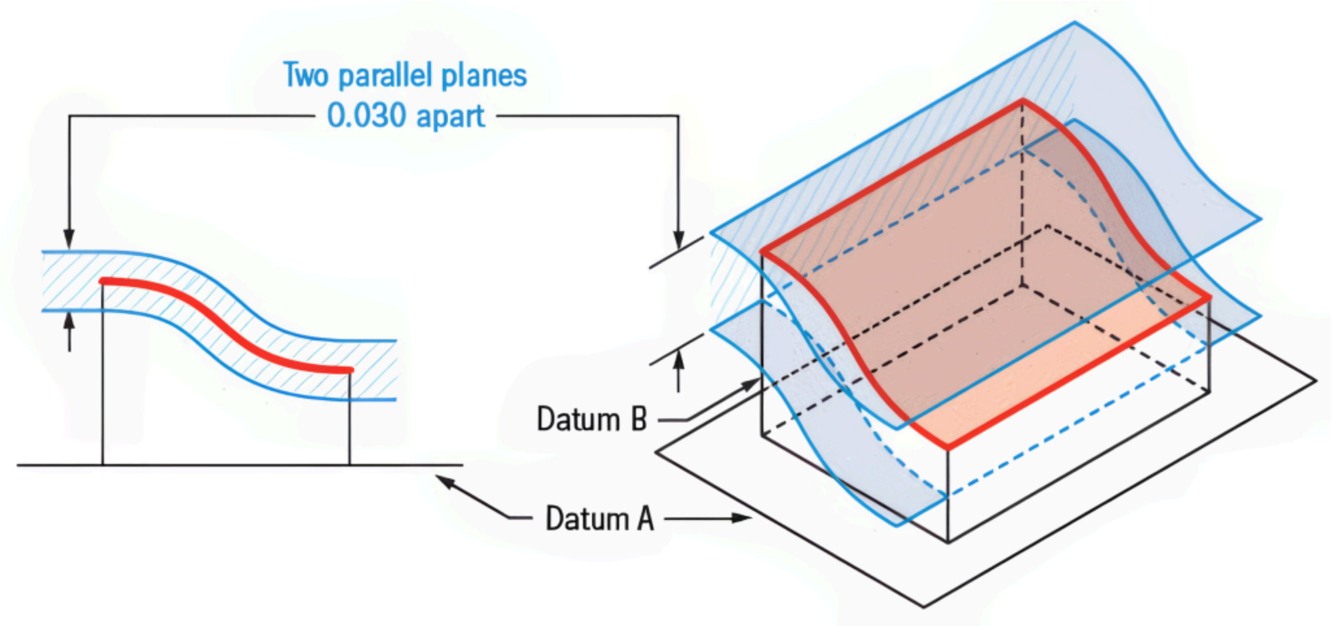


Figure 9-32. The profile of a surface applies to the three-dimensional tolerance zone of a surface. Reused with permission.

Orientation Tolerances

Angularity

Angularity is the symbol that describes the specific orientation of one feature (datum) to another at a referenced angle. See Figure 9-33.

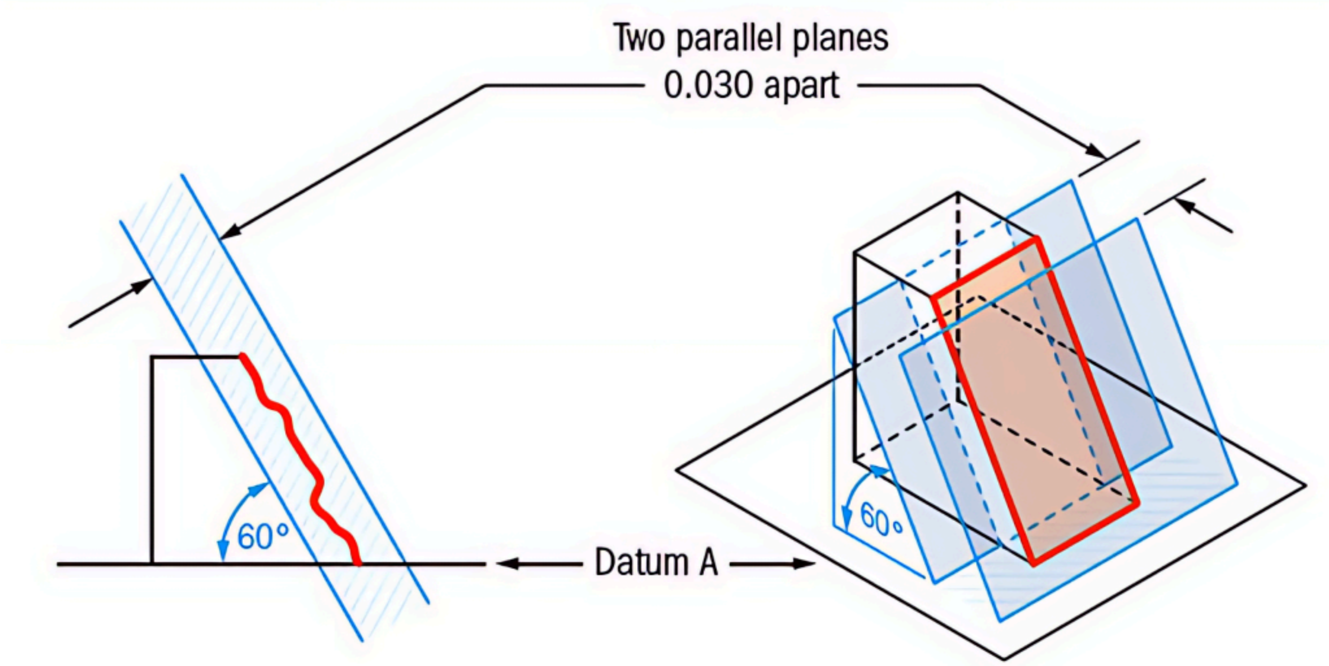
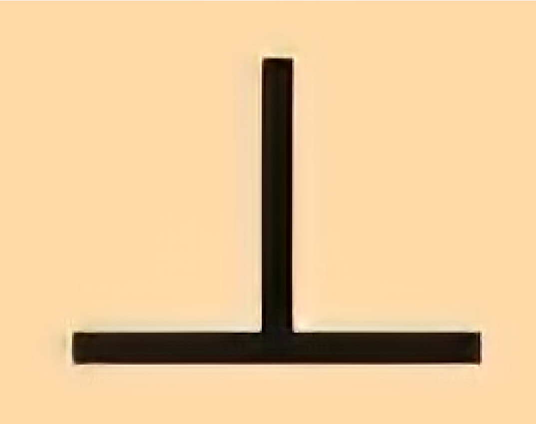


Figure 9-33. Angularity describes the orientation of angular features. Reused with permission.

Perpendicularity



Surface perpendicularity is a tolerance that controls perpendicularity between two 90° surfaces or features. Surface perpendicularity is controlled with two parallel planes acting as its tolerance zone. Axis perpendicularity is a tolerance that controls how perpendicular a specific axis is to a reference (datum). Axis perpendicularity is controlled by a cylinder around a theoretically perfectly parallel axis. See Figure 9-34.

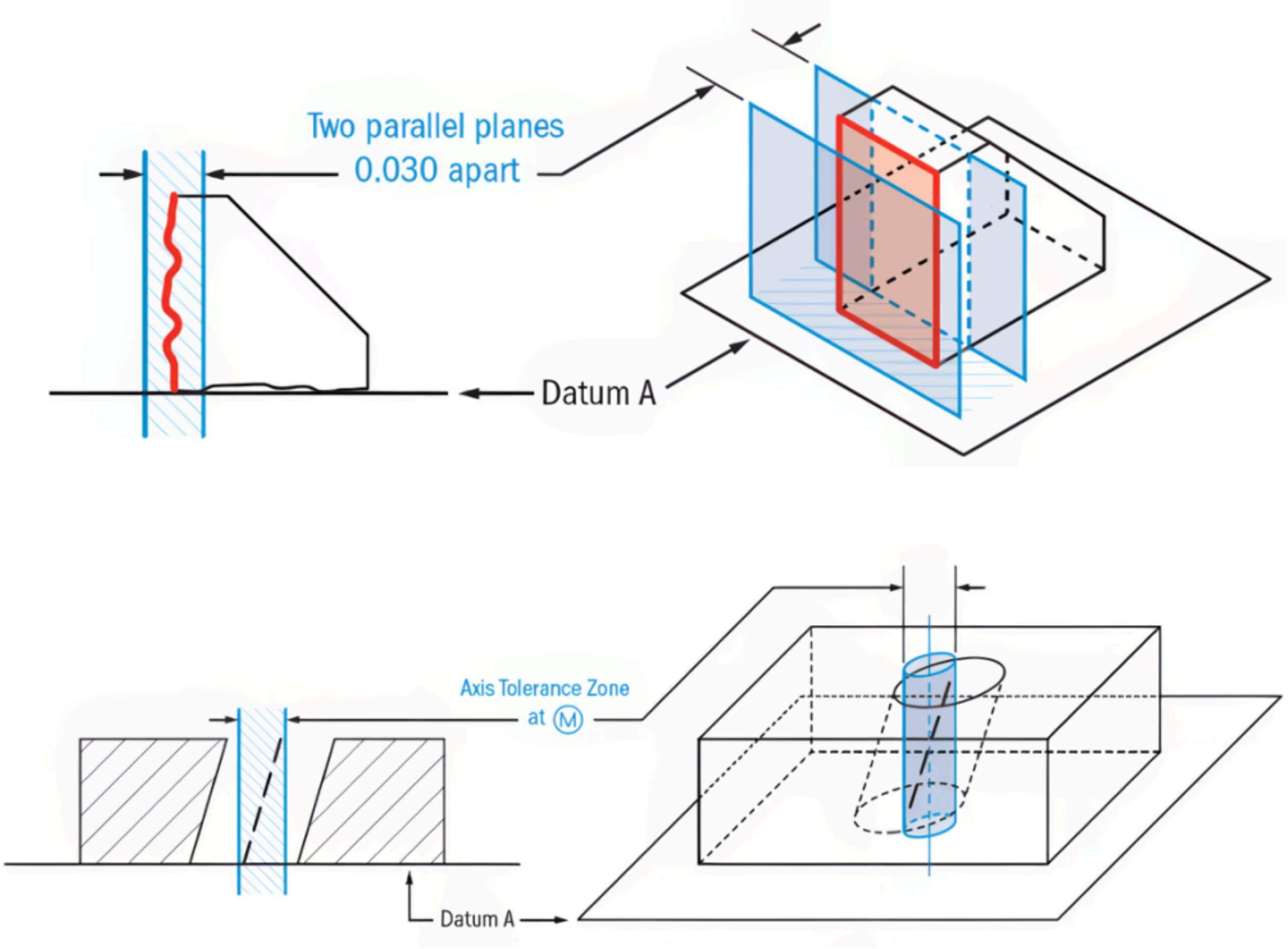


Figure 9-34. Perpendicularity tolerance controls the 90° angle between a feature and a reference surface. Reused with permission.

Parallelism

Surface parallelism is a tolerance that controls parallelism between two surfaces or features, one being a datum. The surface form is controlled similarly to flatness with two parallel planes acting as its tolerance zone, as illustrated in Figure 9-35.

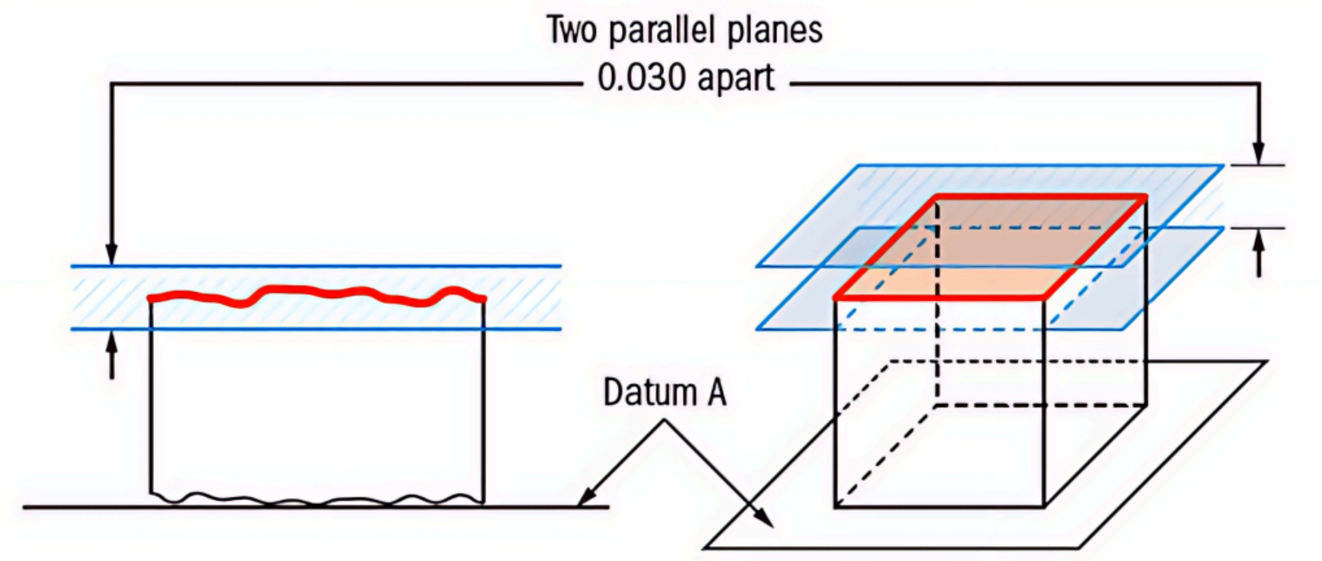


Figure 9-35. Parallelism tolerance controls the parallelism between two surfaces or features. Reused with permission.

Location Tolerances

Location tolerance is a geometric tolerance that determines the location (true position) of a feature in relation to a datum.

Position

The true position, or just position as the ASME standard calls it, is defined as the total permissible variation that a feature can have from its "true" position. See Figure 9-36.

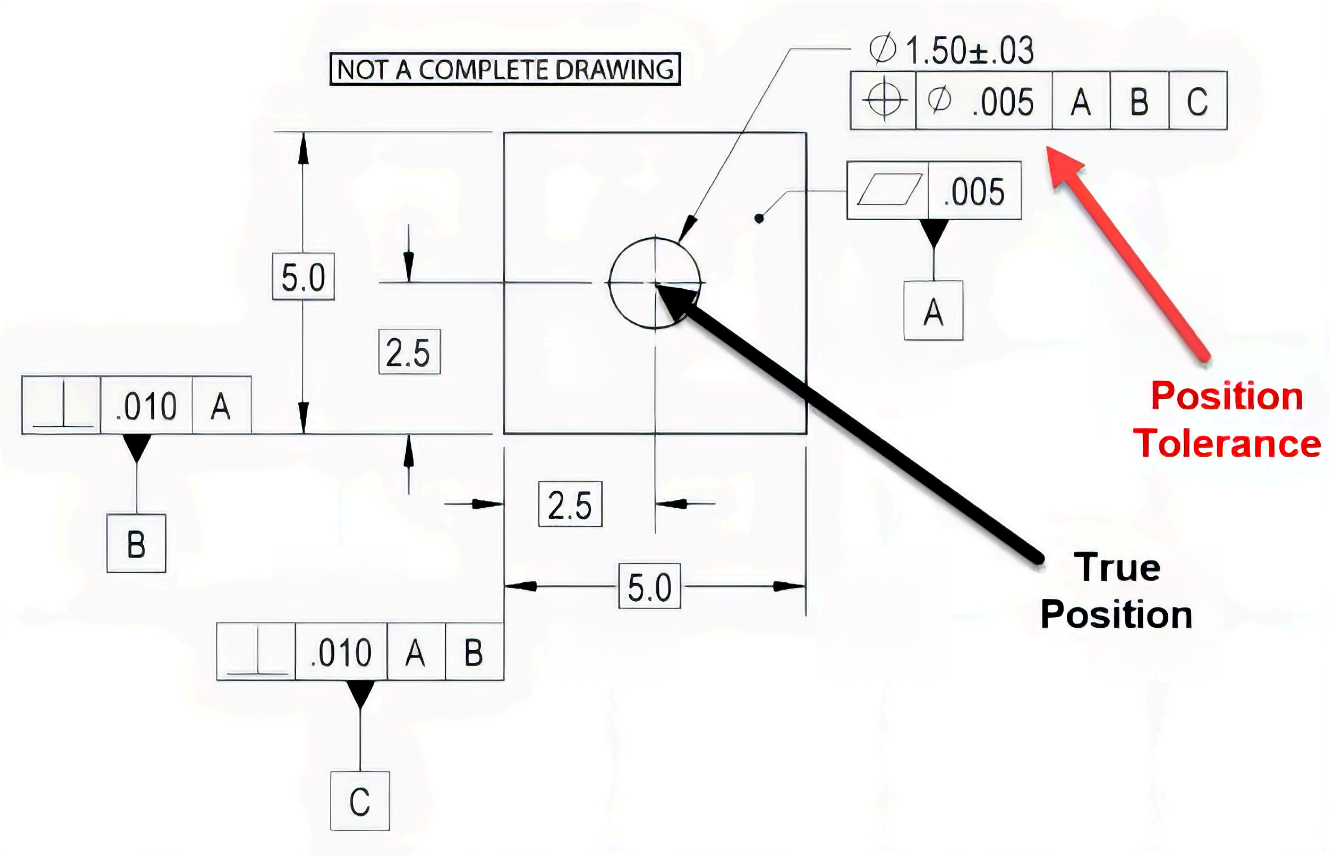


Figure 9-36. A position tolerance controls the location of a feature from a reference. Reused with permission.

Concentricity

The ASME Y14.5 standard was last updated in 2018, and one of the substantial changes was the elimination of the concentricity GD&T control but still may be found on drawings. Previous versions of the standard recommended using runout (the amount a circular feature varies when rotated around an axis) or position controls instead of concentricity. The 2009 version of the Y14.5 GD&T standard defines concentricity as “the condition where the median points of all diametrically opposed elements of a surface of revolution (or the median points of correspondingly located elements of two or more radially disposed features) are congruent with a datum axis (or center point).” See Figure 9-37 below.

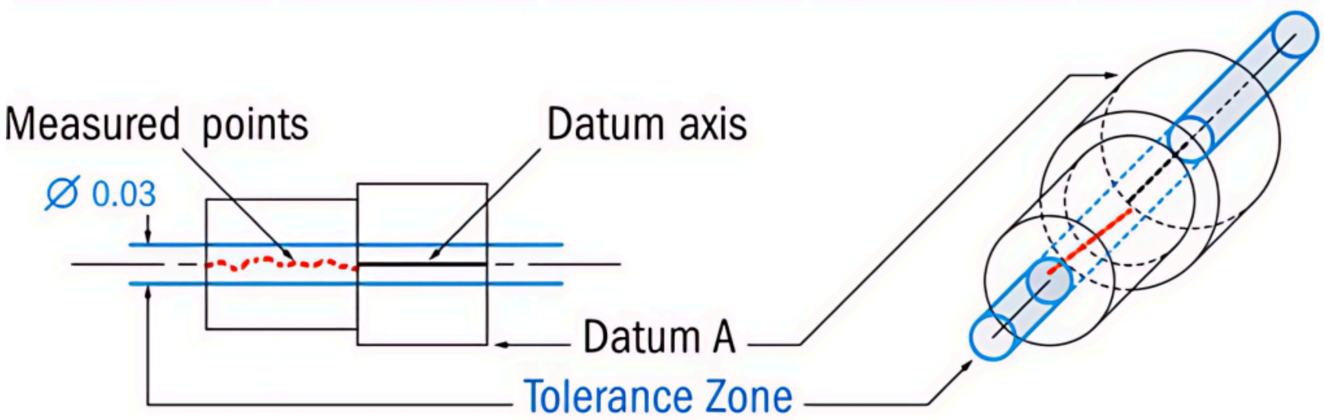


Figure 9-37. Concentricity tolerance controls the axis location of two cylindrical surfaces. Reused with permission.

Symmetry

GD&T symmetry is a three-dimensional tolerance that is used to ensure that two features on a part are uniform across a reference surface known as a datum plane, as shown in Figure 9-37. Along with concentricity, symmetry has also been removed from the latest 2018 ASME Y14.5 standard, although the tolerance may still be found on some drawings.

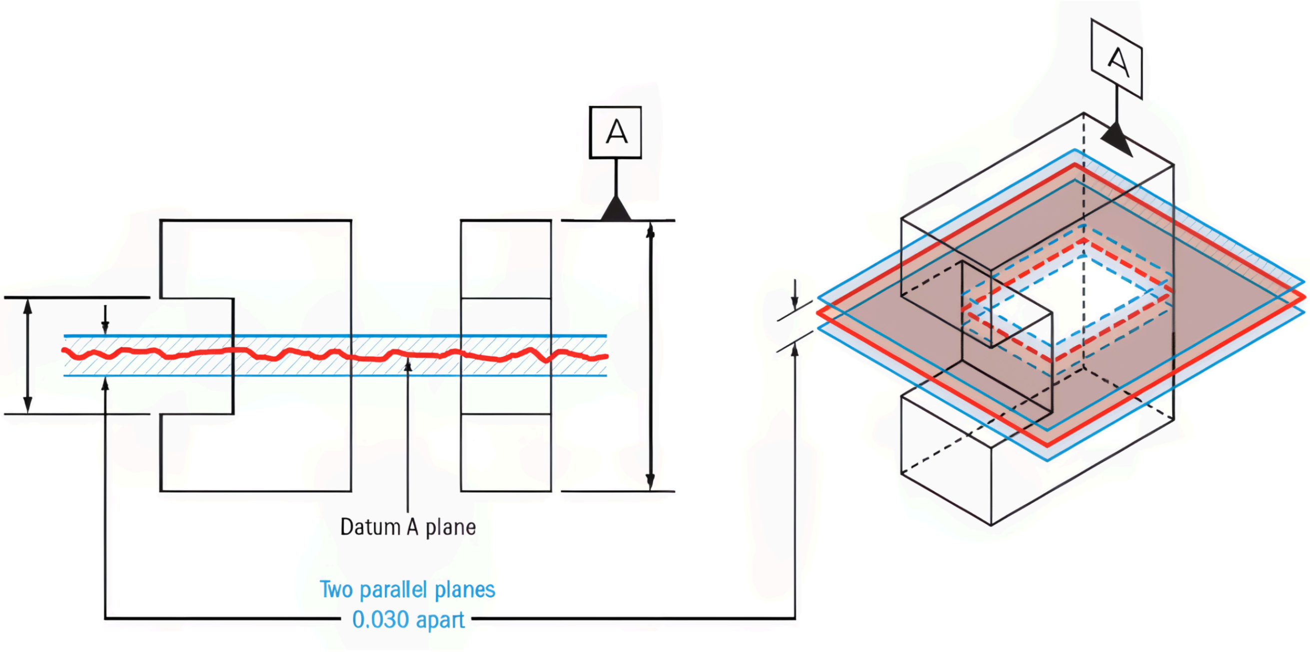


Figure 9-38. Symmetry is a three-dimensional tolerance to control two features of a part. Reused with permission.

Runout Tolerances

Runout tolerance is a geometric tolerance that specifies the amount a circular feature may vary when rotated.

Circular Runout

Runout is how much one given reference feature or features vary with respect to another datum when the part is rotated 360° around the datum axis. It is essentially a control of a circular feature and the amount of variation it has with the rotational axis. View Figure 9-39.

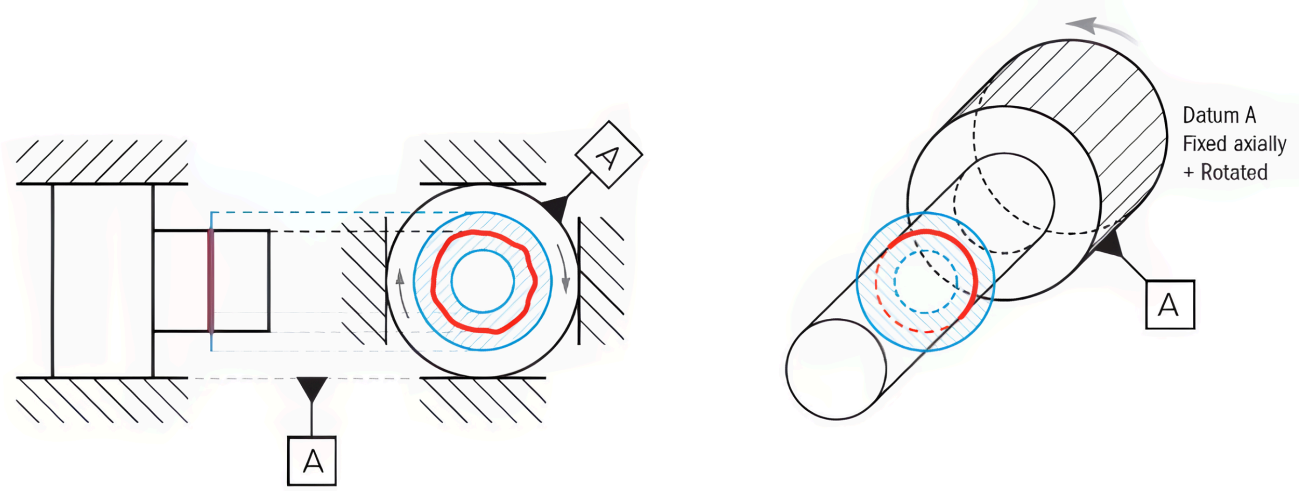
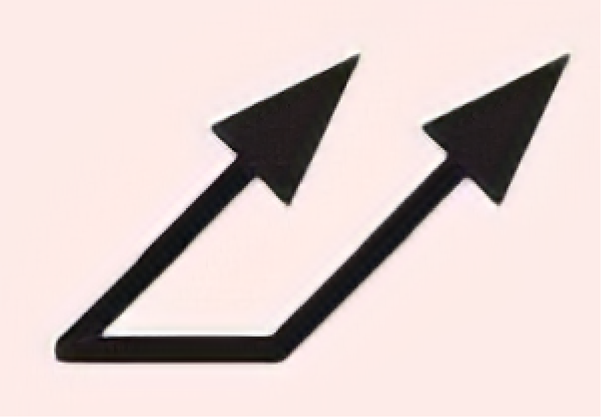


Figure 9-39. Circular runout tolerance controls the number of variations within a circular feature at a given location. Reused with permission.

Total Runout



Total runout is how much one entire feature or surface varies with respect to a datum when the part is rotated 360° around the datum axis. Total runout controls both the amount of variation in the surface as the part is rotated and the amount of variation in the axial dimension, illustrated in Figure 9-40.

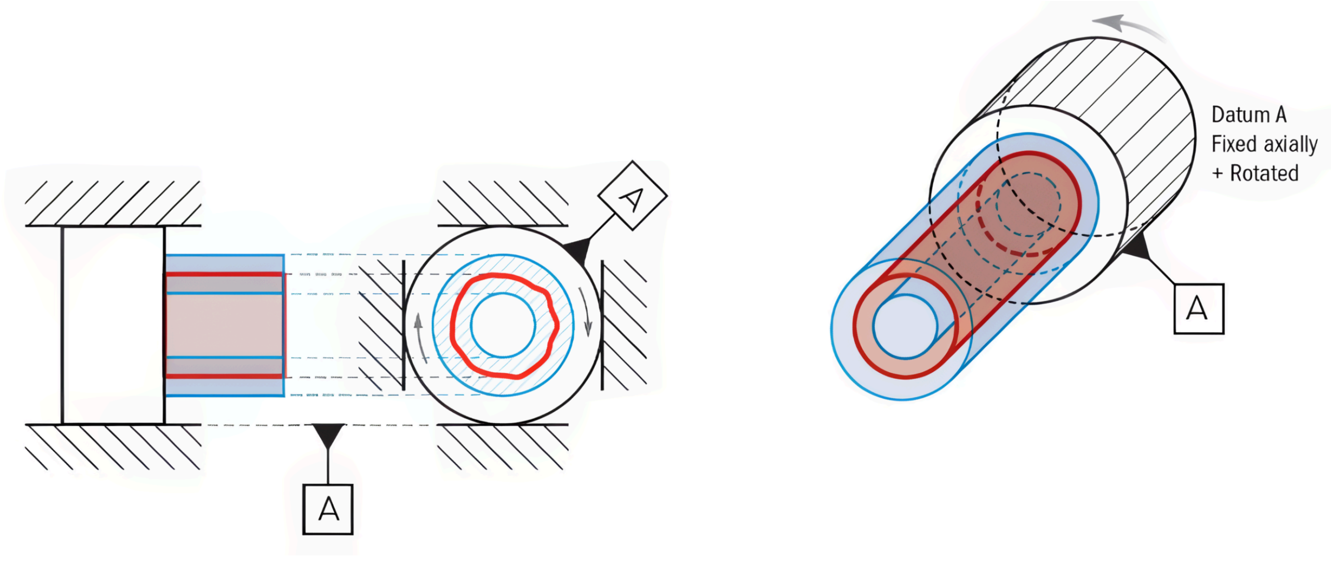


Figure 9-40. Total runout controls the variation in the rotation of the cylindrical surface. Reused with permission.

Material Condition Symbols

Earlier chapters introduced a feature's Maximum Material Condition (MMC) and Least Material Condition (LMC). These conditions are used in the feature control frame with a circled "M" for Maximum Material Condition and "L" for Least Material Condition. When these conditions are placed in the feature control frame, they are referred to as modifiers.

In the Figure 9-26 feature control frame example, the ⌀0.625 hole has a positional tolerance of ⌀0.03 when the hole is at its MMC of ⌀0.620. This means that when the hole is at its smallest allowable size (MMC), the center point position of the hole must be within a ⌀0.03 circle of the basic dimension location. As the hole size moves away from its MMC (becomes larger), the tolerance zone will also increase by the same amount. This is done because as the hole size increases, the position may deviate by the same amount, and the feature will still function. Figure 9-41 illustrates the ⌀0.03 center location zone with the hole size at ⌀0.620 (MMC) in red and the ⌀0.04 center location zone when the hole is at its largest allowable size of ⌀0.630 (LMC) in blue.

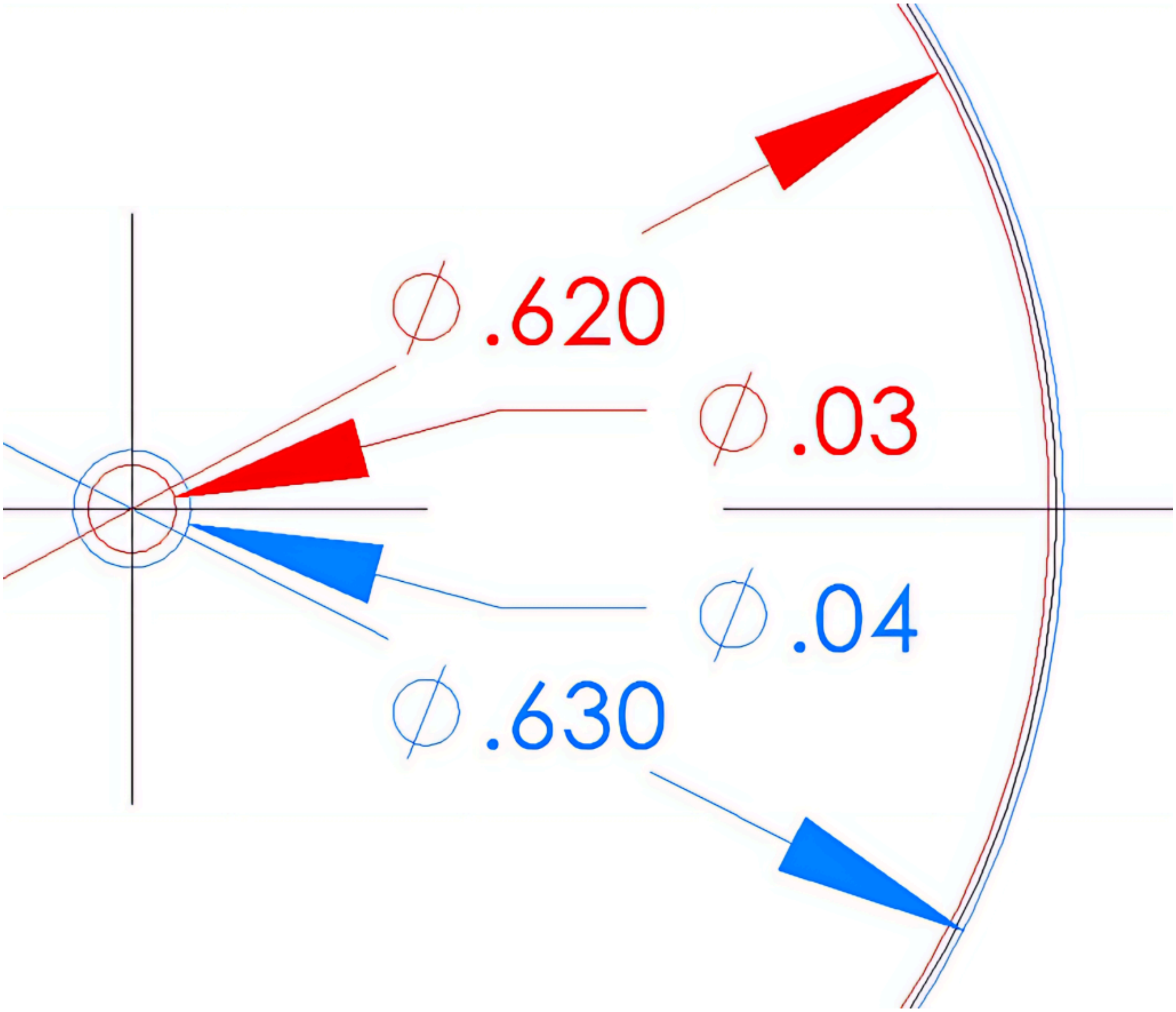


Figure 9-41. Positional tolerance illustration from Figure 9-26.

Projected Tolerance Zone

A third modifier that may be used is a circled “P” for the projected tolerance zone. In GD&T, the projected tolerance zone symbol indicates the tolerance zone of a feature that is to be assessed beyond the surface. In most cases, this symbol verifies that a fastener will still clear a mating part during assembly regardless of how thick the assembling part is and the orientation in which the fastener is assembled. See Figure 9-42.

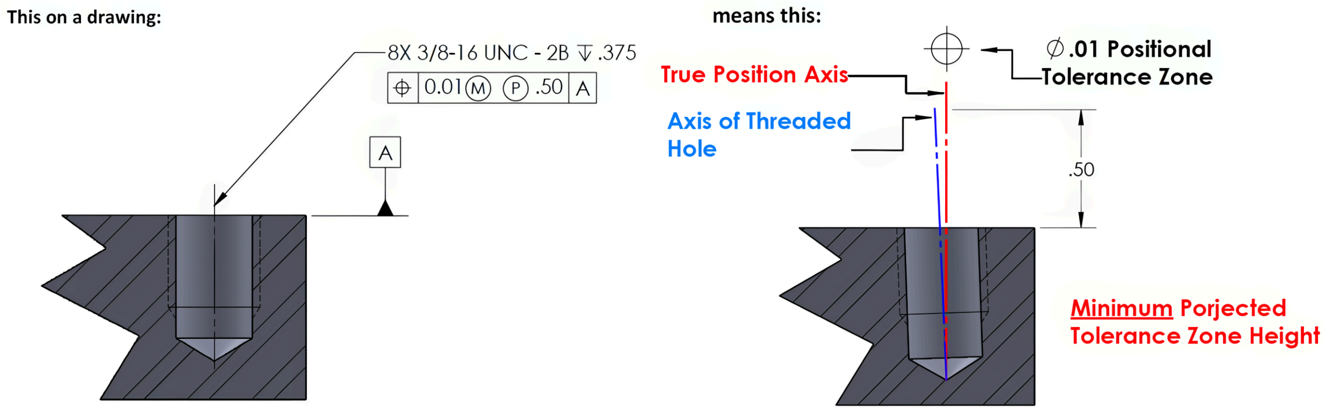


Figure 9-42. A projected tolerance zone is used to locate a positional tolerance above the part's surface.

For further information on Geometric Dimensioning and Tolerancing please visit: https://www.gdandtbasics.com/

LEARNING ACTIVITIES

Exercise 9.6-1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=345#h5p-25

Exercise 9.6-2

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=345#h5p-26

Exercise 9.6-3

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=345#h5p-27

Exercise 9.6-4

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=345#h5p-28

References:

Barsamian, M. A., & Gizelbach, R. (2022). *Machine trades print reading.* Goodheart-Willcox.

Schultz, R. L., & Smith, L. L. (2012). *Blueprint reading for machine trades* (7th ed.). Pearson.

Images:

All images by Mark Lorier and licensed under CC BY-NC 4.0 except where otherwise noted.

Figure 9-11 “Basics-of-Surface-Finish_components2-932×800” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-18 “Surface-Finish-Fig-6_Symbols_CloseUp2” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-21 “Profilometer-Fig1_Profilometer-Contact” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/what-is-a-profilometer/

Figure 9-22 This image is derivative of “Surface-Finish-Fig-23_surface-profile-mean-line2” by GDandTBasics.com and reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-23 This image is derivative of “Surface-Finish-Fig-23_surface-profile-mean-line2” by GDandTBasics.com and reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-24 “Surface-Finish-Fig-4_Ra2” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-25 “SurfaceFinishRoughnessByProcess” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-27 “Snag_59d7b2d-768×429” and “Snag_59e15b8-768×247” by GDandTBasics.com are reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-28 “Flatness-Tolerance-Zone” and “Flatness-Example1-Table-with-gdt” by GDandTBasics.com are reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-29 “Circularity-Tolerance-Zone” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-30 “cylindricityTolerance-Zone” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-31 “Profile-of-line-gauging” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-32 “Profile-of-surface-tolerance-zone” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-33 “Angularity-Tolerance-Zone” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-34 “04-Perpendicularity-surface-tolerance-zone” and “05-Perpendicularity-axis-tolerance-zone” by GDandTBasics.com are reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-35 “Parallelism-Tolerance-Zone” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-36 “Figure1-1024×740” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-37 “Snag_aa599d8-1024×349” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-38 “Symmetry-Tolerance-Zone” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-39 “runout-Tolerance-Zone” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-40 “01-Total-Runout-Tolerance-Zone1” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Figure 9-42 This image is derived from “Figure-3_Applying-Projected-Tolerance-Method-1_v2” by GDandTBasics.com is reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/basics-of-surface-finish/

Unnumbered images are all derived from “GD&T Basics Wall Chart PDF” by GDandTBasics.com are reused with permission. All rights reserved. Access for free at https://www.gdandtbasics.com/gdt-symbols-chart/

10. Machining Details

10.1 INTRODUCTION

Learning Objectives

- Define machining terminology
- Identify machining details
- Determine the location of machined details
- Determine the sizes of machined details
- Explain the machining processes used to achieve the detail in the part
- Calculate the distances between machined details
- Discuss the order of operations related to machined details
- Determine the order of operations based on print specifications
- Identify thread terms
- Identify thread series

Terms

- Neck
- Groove
- Slot
- T-slot

- Dovetail slot
- Key
- Keystock
- Woodruff key
- Keyseat
- Keyway
- Boss
- Pad
- Major diameter
- Minor diameter
- Pitch
- Lead
- Thread angle
- Pitch diameter

In this chapter, we will explore the various features frequently found on machined components that are essential for fastening or assembly. Understanding these commonly used elements is crucial not only for effective design but also for ensuring clarity in communication. We'll look closer at how these features are represented and dimensioned on drawings, as this knowledge plays a vital role in successfully interpreting detailed drawings.

10.2 NECKS

A **neck** is an important feature found on the outer surface of cylindrical components. It serves a vital purpose by creating a seamless transition between different diameters, helping to prevent any potential interference between connecting parts due to the radius of cutting tools.

Figure 10-1 illustrates how a neck can be used to allow proper assembly of components.

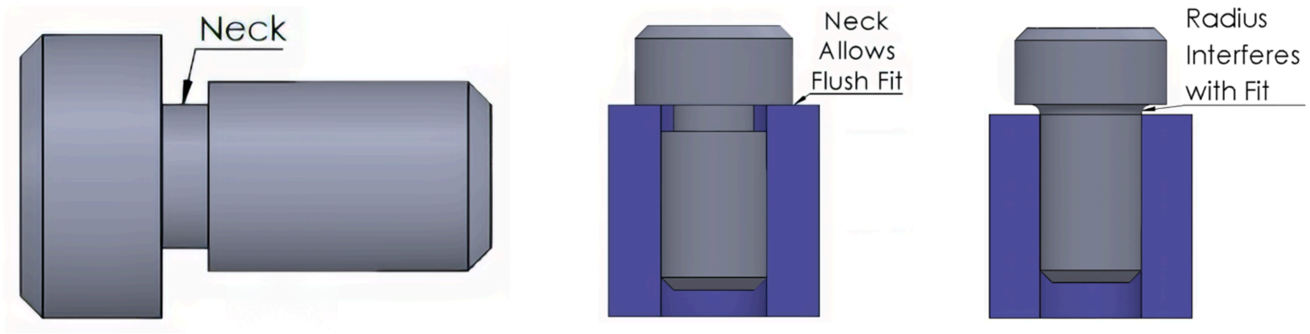


Figure 10-1. A neck eliminates interference in assembly.

Additionally, this neck—often called a relief—is also beneficial at the end of threading operations; it provides the necessary clearance for the threading tool and ensures that mating threads can be fully engaged without any obstruction, as shown in Figure 10-2.

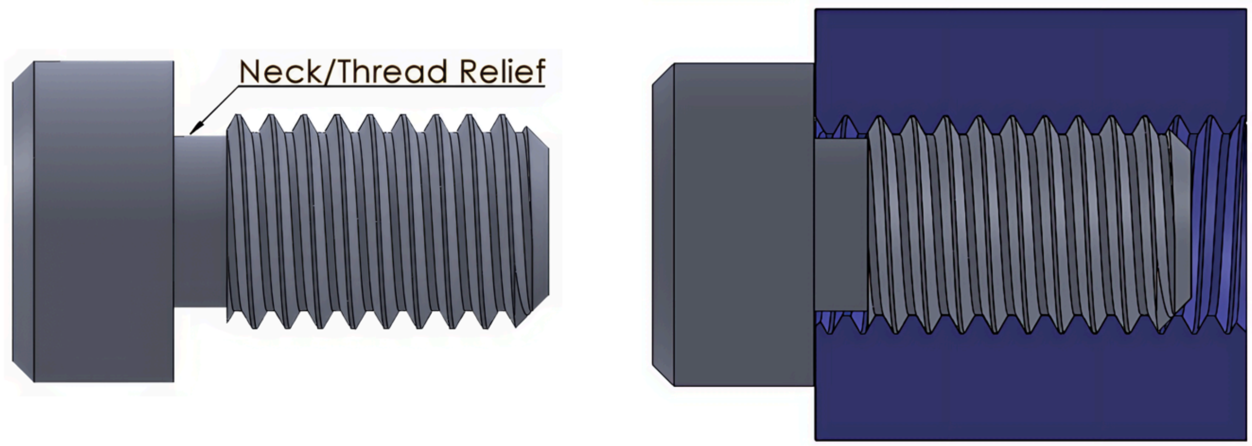


Figure 10-2. A neck allows threads to be fully engaged and can provide clearance during threading operations.

When it comes to dimensioning necks, there are two methods that may be used. A leader line with a note containing the neck width and radial depth information or dimension and extension lines with the measurement values can be used. See Figure 10-3. It's important to remember that sometimes precise neck sizes may not be critical enough to warrant specific dimensions. However, please keep in mind that if the neck is deeper than necessary, it could create a weak point within the component.

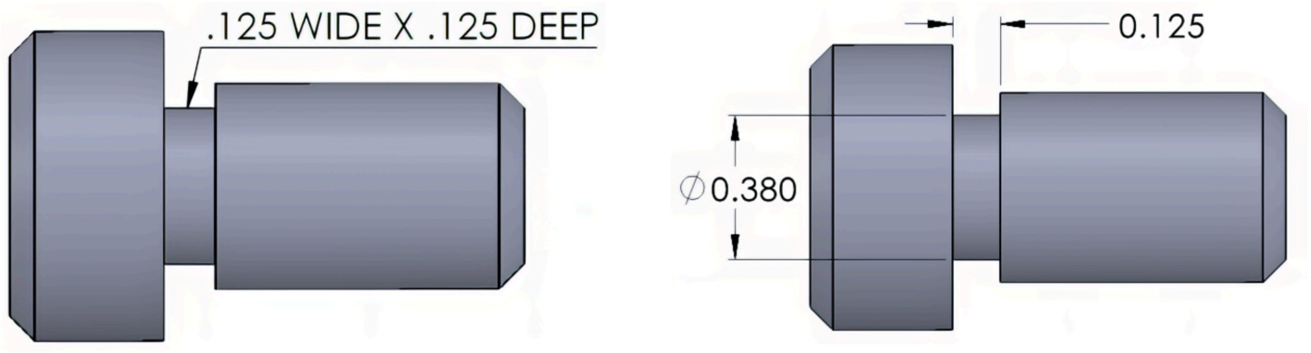


Figure 10-3. A neck may be dimensioned using a leader and note or dimension and extension lines.

Necks on threaded components may be dimensioned as shown in Figure 10-4, as the diameter needs to be only smaller than the minor diameter of the thread.

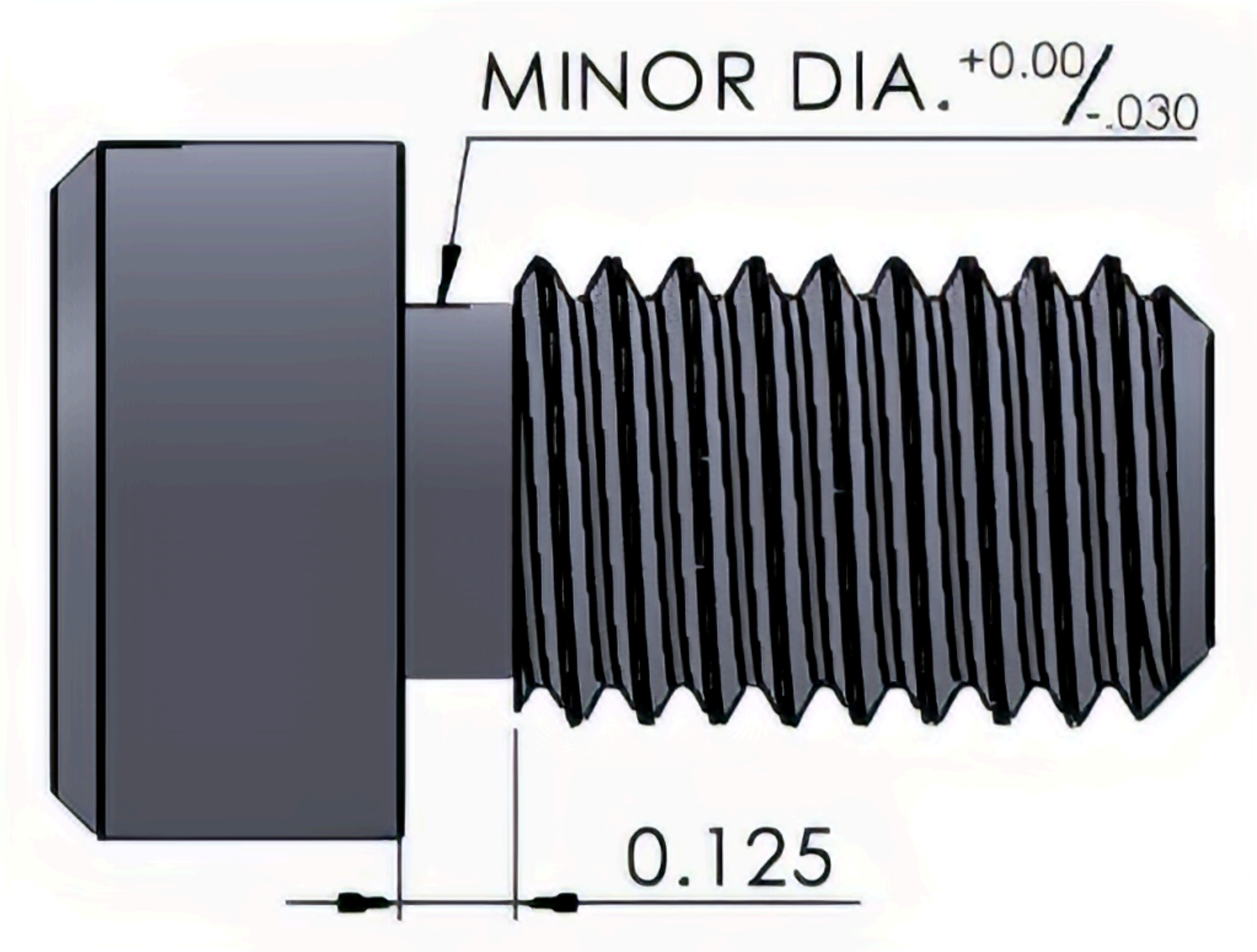


Figure 10-4. The neck used on threaded components is smaller than the minor diameter.

10.3 GROOVES

While necks refer to a recess on an outside diameter at the intersection of different diameters, a **groove** is a recess anywhere along an outside or inside diameter. Grooves are typically used for lubrication pathways, seals, and internal or external retaining rings. See Figure 10-5 for an illustration of grooves.

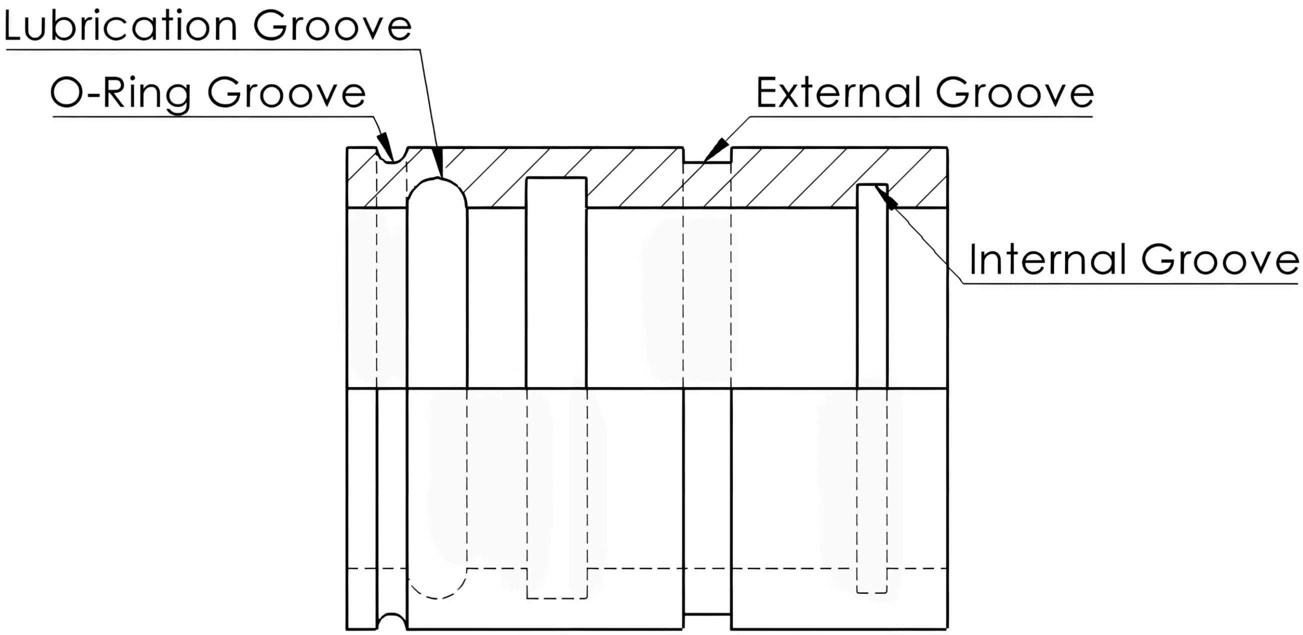


Figure 10-5. Grooves are recesses along an outside or inside diameter in a variety of shapes.

As with a neck, a groove may be dimensioned using a leader note or dimension and extension lines, as shown in Figure 10-6.

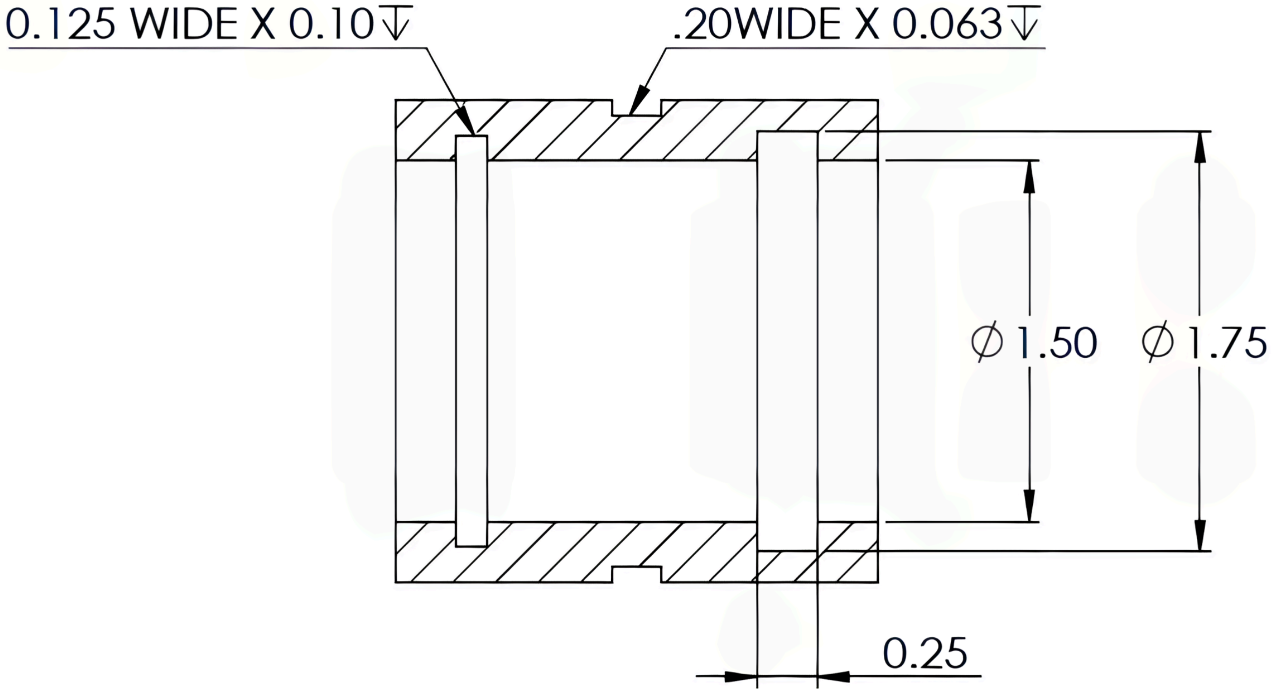


Figure 10-6. Grooves are recesses using different dimensioning styles.

Another common outside groove is the V-groove, typically used on pulleys for use with a “V”-shaped belt. A V-groove is commonly dimensioned with the V angle and the depth of the groove. See Figure 10-7.

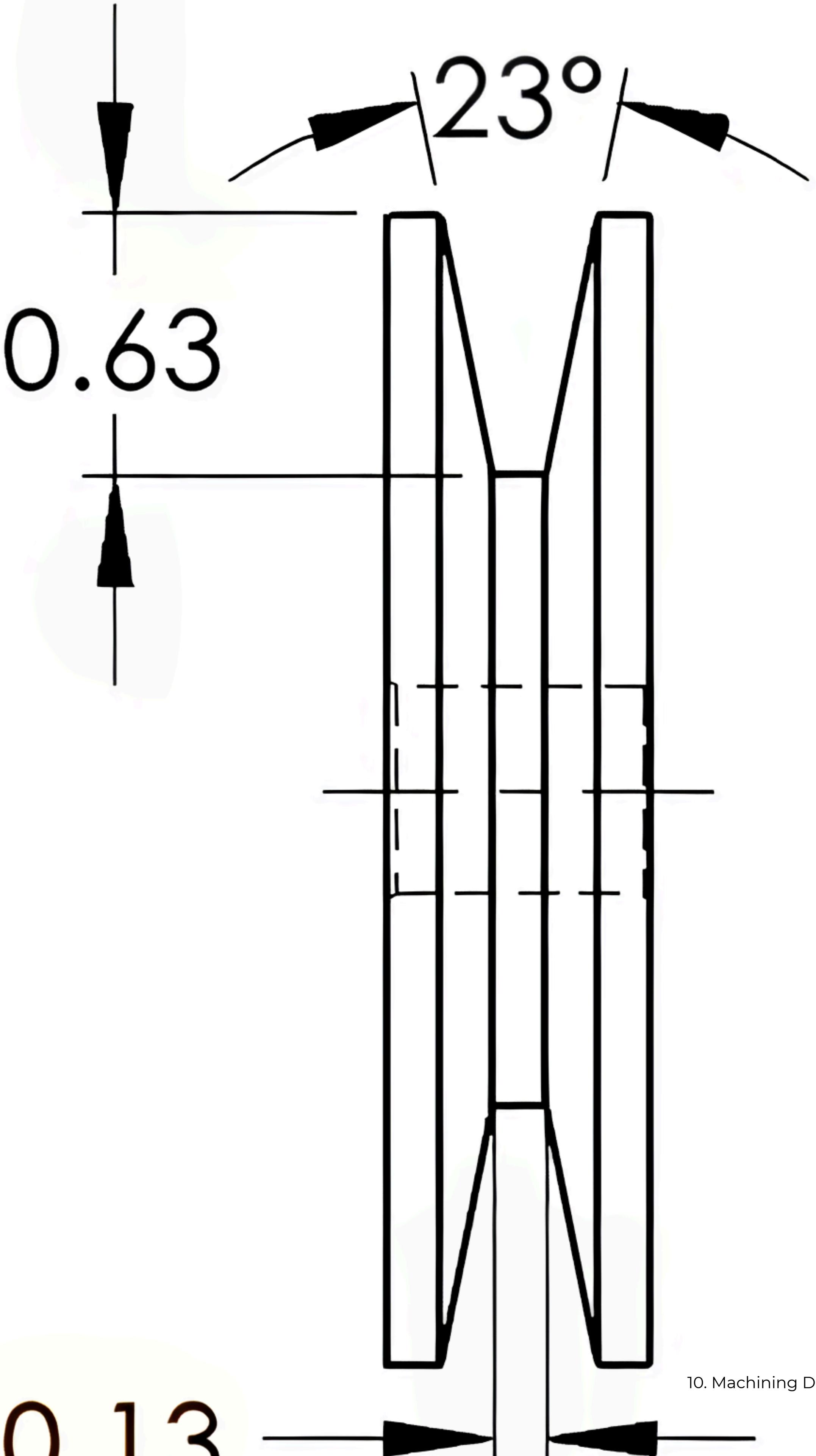

23°
0.63
0.13

Figure 10-7. Dimensioned V-groove pulley.

10.4 SLOTTED HOLES

Slots are features that often serve as a connection with another piece and are often used to provide adjustment. Slotted holes typically have a full radius on each end and may be dimensioned in various ways. Observe the three styles of dimensioning a slotted hole. The dimensions may locate the center point locations and width, center point locations and radius, or the outer dimensional sizes of the slot, as depicted in Figure 10-8.

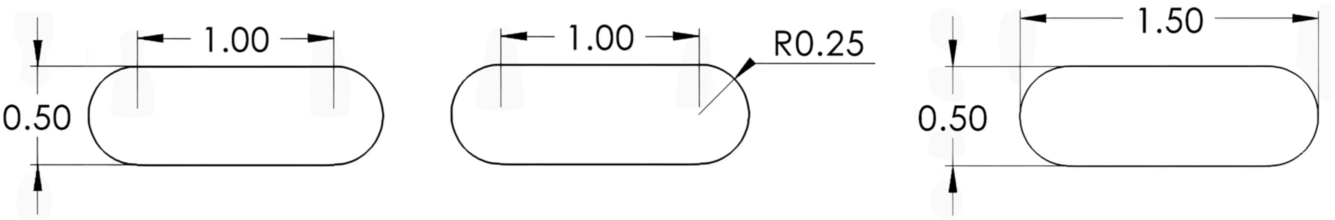


Figure 10-8. Slotted hole size and location dimensioning techniques.

10.5 T-SLOTS

T-slots play an important role in machining tables and various fixturing devices, as they enable secure clamping of workpieces with T-bolts. Recognized by their distinctive upside-down “T” shape, these slots offer the flexibility to adjust the placement of clamps according to specific needs. Figure 10-9 shows a T-slot.

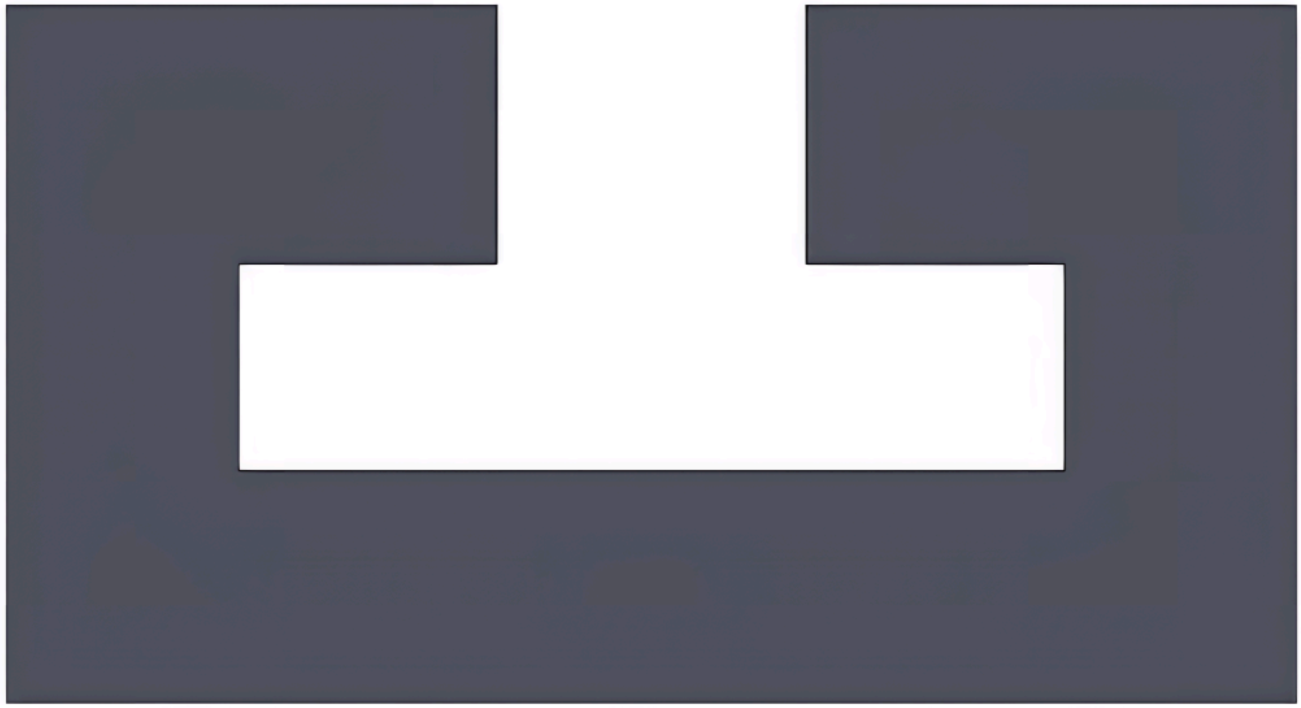


Figure 10-9. T-slots are used for clamping components

10.6 DOVETAIL SLOTS

Dovetail slots play a crucial role in machinery that requires precise and reliable movement between components. You can often find them on milling machine tables, cross-slides, lathe cross-slides, and compound rests. When creating T-slots and dovetail slots using a milling machine, two tools will typically be needed: one to remove the material for the width of the top opening and another to shape the undercut features. Dovetail slots are portrayed in Figure 10-10.

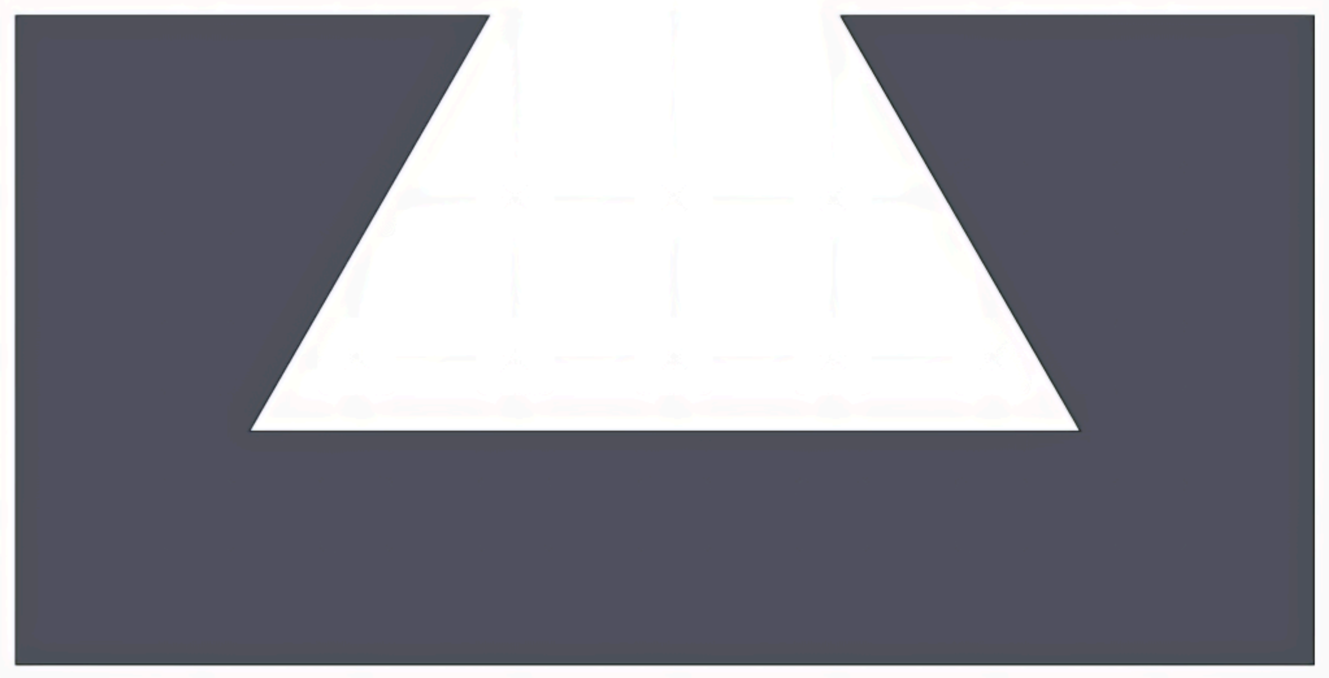


Figure 10-10. Dovetail slots are used for the linear movement of components..

10.7 KEYWAYS AND KEYSEATS

A **key** is an important fastener that helps connect a circular shaft and hub, ensuring they work together smoothly without rotating independently. You will often find these keys in various applications, such as gears, pulleys, hand-wheels, and cutting tools. They can come in different shapes, with square or rectangular options known as **keystock** being quite common. There's also the **Woodruff key**, which has a unique half-circular design, allowing its rounded part to fit securely into the shaft. The choice between a Woodruff key and a flat key depends on the specific application and requirements. Factors such as load capacity, space availability, and installation convenience are considered when selecting the key type.

The area on the shaft that is cut to receive the key is referred to as the **keyseat**. Keyseats are grooves cut along the shaft length to allow the key to sit in. These are cut with a milling machine using an endmill or special straight-fluted endmill designed for cutting keyways. Woodruff keyseat cutters have an upside-down "T" shape that creates a circular recess in the center of the shaft. Woodruff keys and keyseat cutters are identified by numbers representing their width in 1/32 of an inch and diameter in 1/8 of an inch. As an example, a #806 Woodruff key would have a width of 8/32 or 1/4 and a diameter of 6/8 or 3/4. In another example, a #1008 Woodruff key will have a width of 5/16 and a diameter of 1 inch. See Figure 10-11.

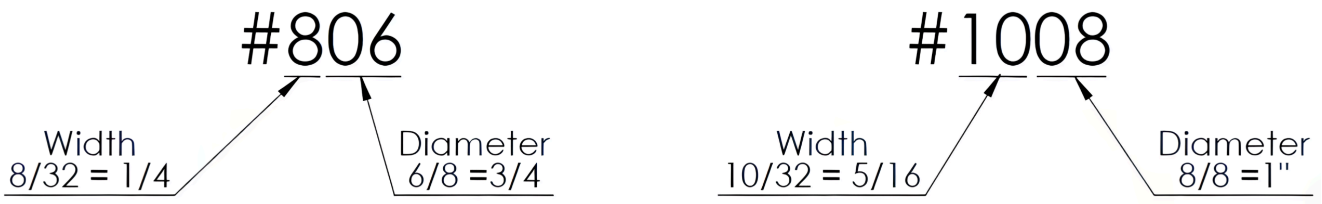


Figure 10-11. A Woodruff key is identified by the width in 32nds of and inch and diameter in 8ths of an inch.

The **keyway** is the groove along the internal length of the hub for the key to fit into. Keyways can be cut into the holes of the hub using a keyway broach set and press or by using a wire EDM (electrical discharge machining) machine, depending on the equipment available, as shown in Figure 10-12.

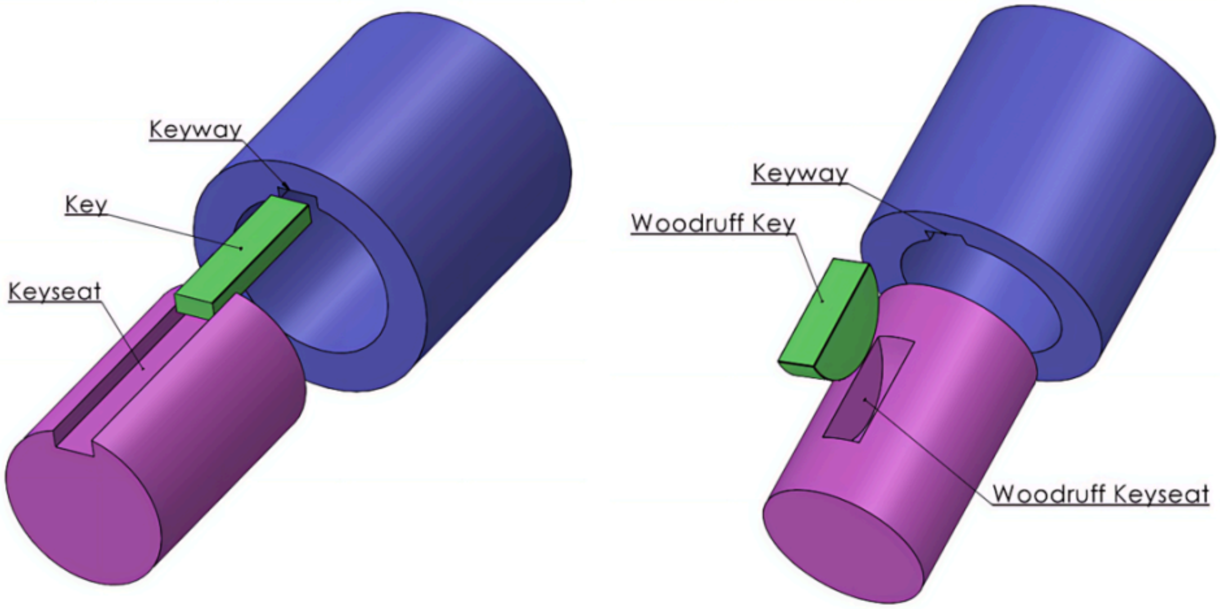


Figure 10-12. Straight key and Woodruff key assemblies.

Keyways and keyseats may be dimensioned by width and height or depth, or they may be noted with a leader line and local note. The standard dimensions for various key sizes can be referenced in texts such as the *Machinery's Handbook*. Figure 10-13 shows a keyway dimensioned from the keyway to the opposite diameter.

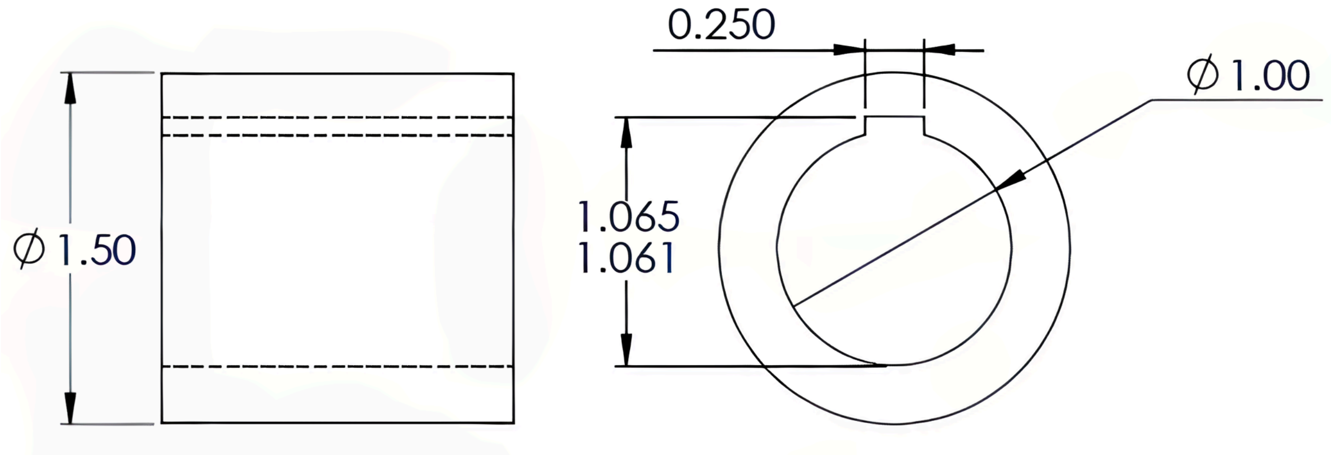


Figure 10-13. The keyway is dimensioned to the diameter opposite the keyway.

A keyseat is often dimensioned by the keyseat width and from the bottom of the keyseat to the opposite diameter, as depicted in Figure 10-14.

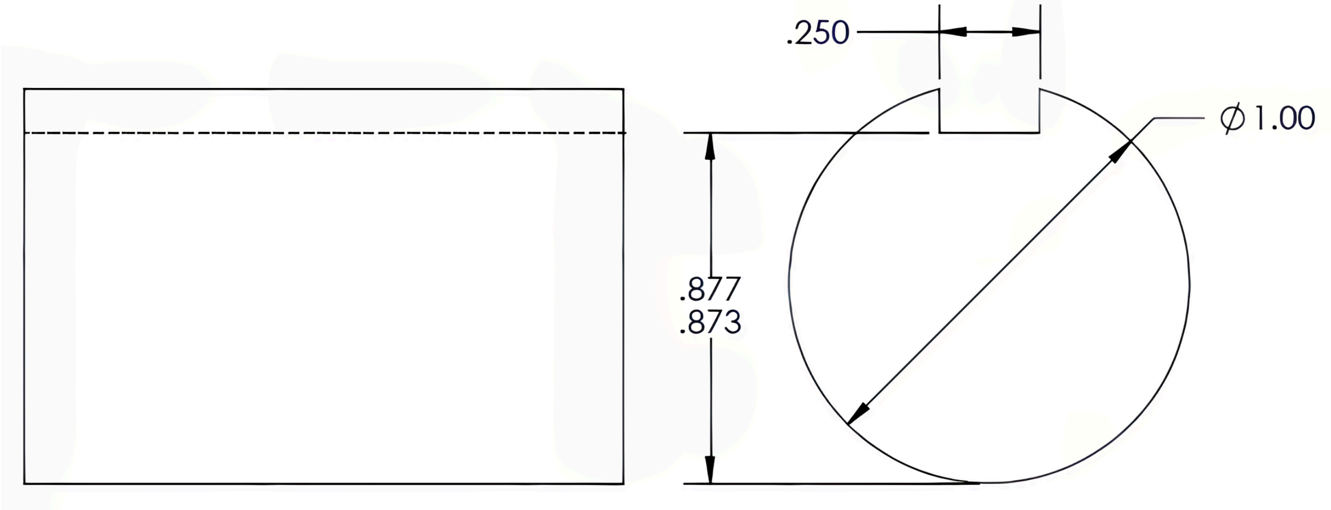


Figure 10-14. The keyseat is dimensioned by width and thickness from shaft diameter to keyseat.

10.8 BOSSES AND PADS

A boss and a pad are essential features in casting, designed to enhance the machining process that follows. A raised area that is circular in shape is referred to as a **boss**. A **pad** is a raised area of any other shape (but often oblong). These features are often used for a contact surface with a fastener or another mating part. This surface is often machined to obtain a flatter contact surface. View Figure 10-15 for an illustration of a boss and a pad.

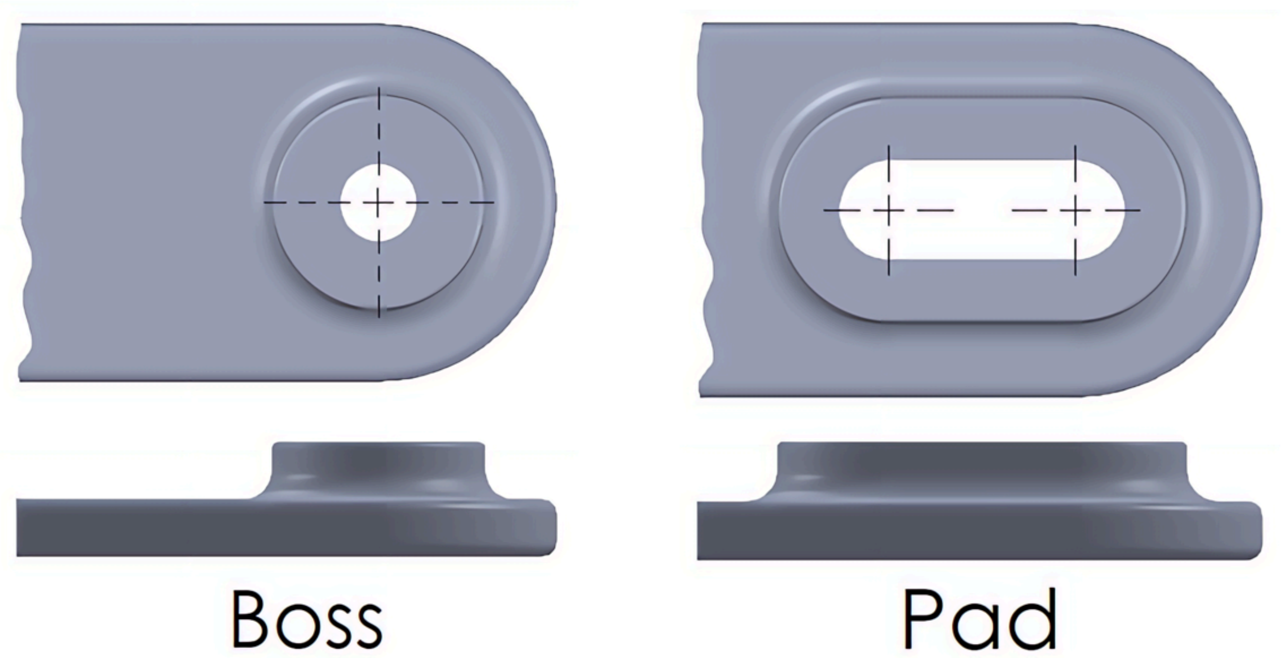


Figure 10-15. A boss is a raised circular feature; a pad is a non-circular raised feature.

10.9 THREAD TERMS

A screw thread is a helical ridge wrapping around a cylindrical shape. A helical shape (or helix) is a shape uniformly wrapping around a cylinder. An external thread is on the outside of a cylinder, such as the threads on a bolt, as shown in Figure 10-16. An internal thread is a thread on the cylinder of a hole, such as in a nut or tapped hole, as shown in Figure 10-17. The V-type thread design is the most prevalent type of thread used and plays a significant role in numerous industrial parts, primarily in fastening components together. Other thread shapes like the square thread are frequently applied in applications related to power transmission, such as driving axes in milling machines and lathes. The information in this chapter covers common thread terminology, representation, and specifications.

Major diameter is the largest diameter of the thread, measured across the diameter of the crest on an external thread and the diameter measurement across the root of an internal thread. Before cutting an external thread, the part's starting diameter will be the major diameter.

Minor diameter is the smallest diameter of the thread, measured across the diameter of the crest on an internal thread and the measurement across the root of an external thread.

Pitch is the distance between a point on one thread and the same point on the next thread. Metric will identify the major diameter followed by the pitch; for example, in M10 x 1.25, the pitch is 1.25 mm. An inch thread will identify the major diameter followed by the number of threads in a distance of one inch; for example, in 1/4-20, there are 20 threads within a 1-inch distance. Divide 1 by the number of threads per inch to obtain the pitch on an inch thread. In the 1/4-20 example, 1 divided by 20 will equal a pitch of .05 inch.

Lead is the amount the thread will advance in one revolution. Most threads will have a single start point; in which case the lead will match the pitch of the thread. To advance a thread more quickly, some threads will have multiple start points, known as multiple lead threads. A double lead thread will have two start points and a lead twice its pitch. An everyday example would be the cap of a plastic bottle, which will often have multiple start points.

Thread angle is the angle formed between the threads. The most common "V"-shaped thread will have an angle of 60°.

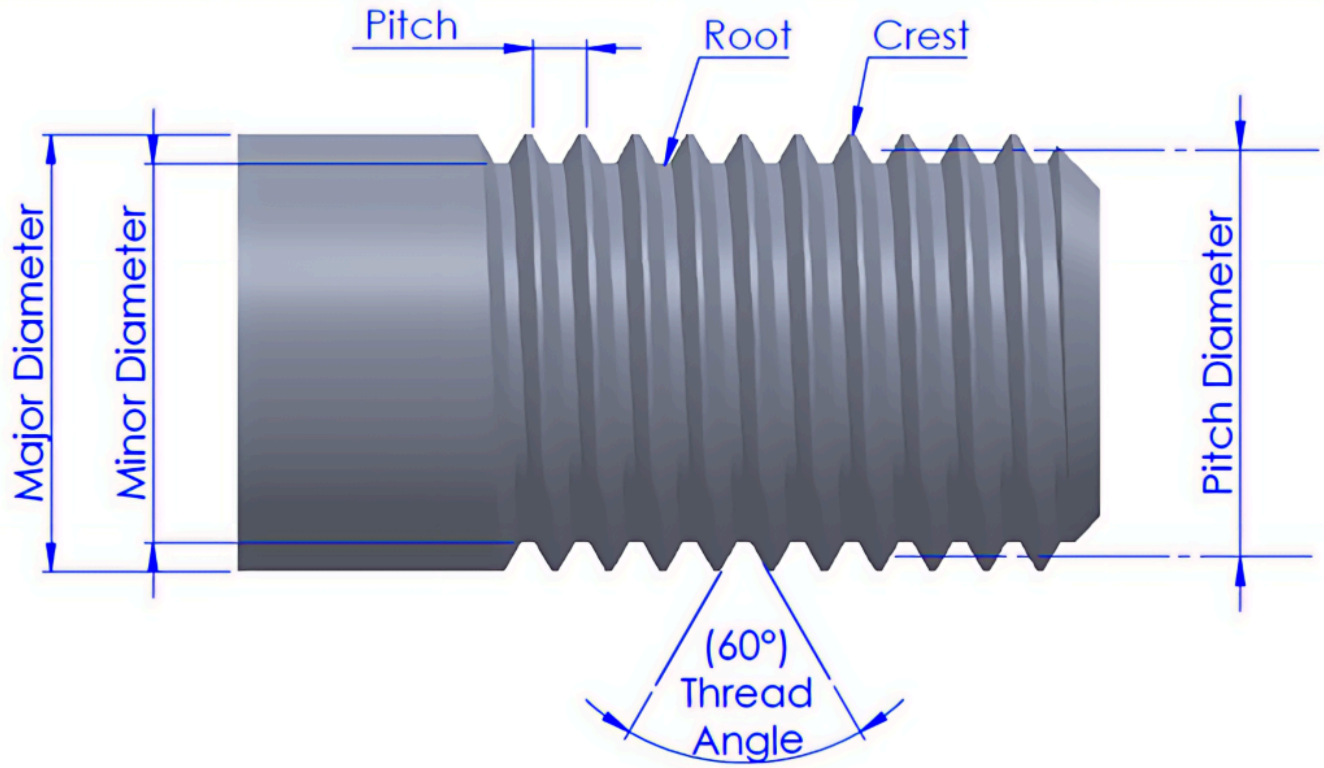


Figure 10-16. Thread terms are displayed on an external thread.

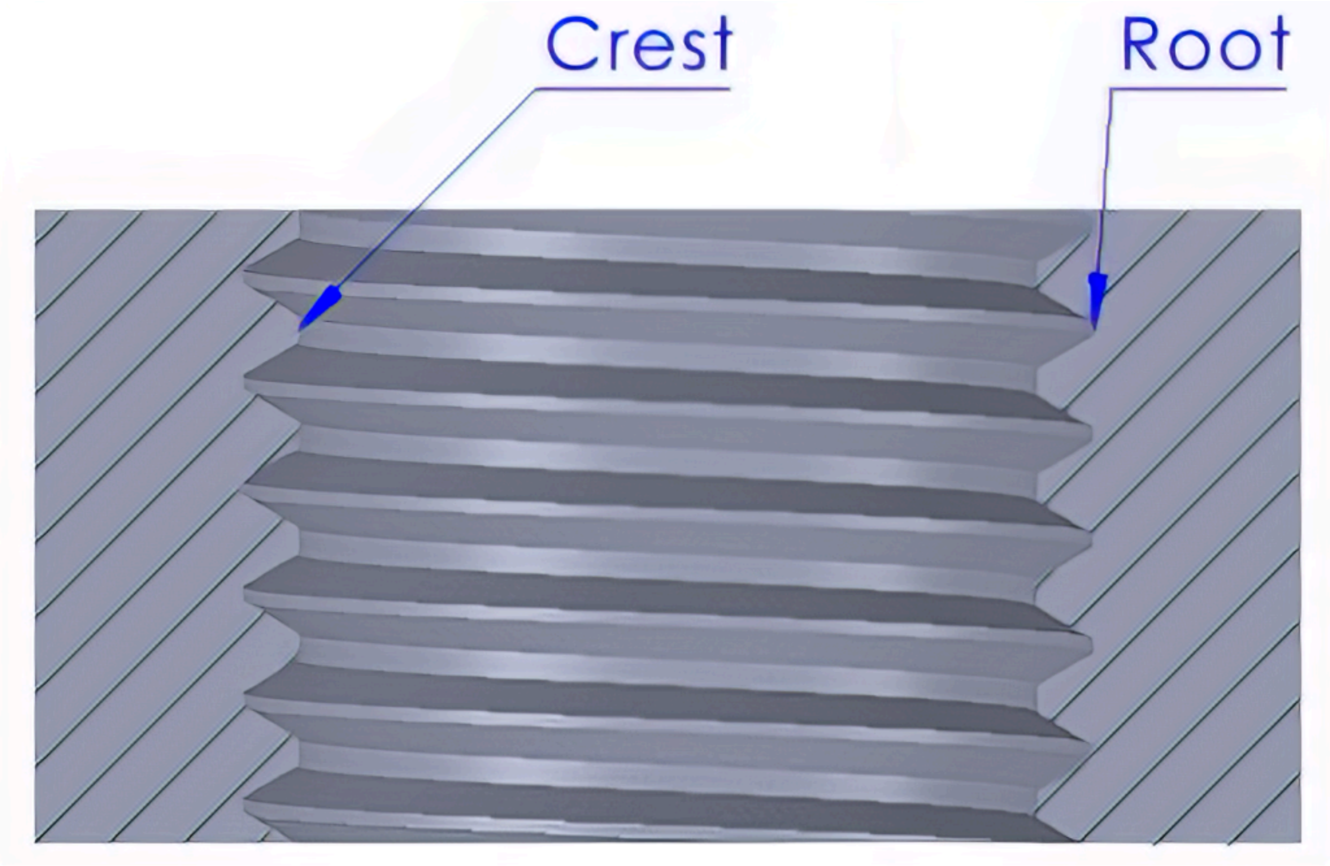


Figure 10-17. Internal thread crest and root.

Pitch diameter is where the thread groove's width matches the ridge's width. It is the critical value for obtaining the proper thread fit. See Figure 10-18.

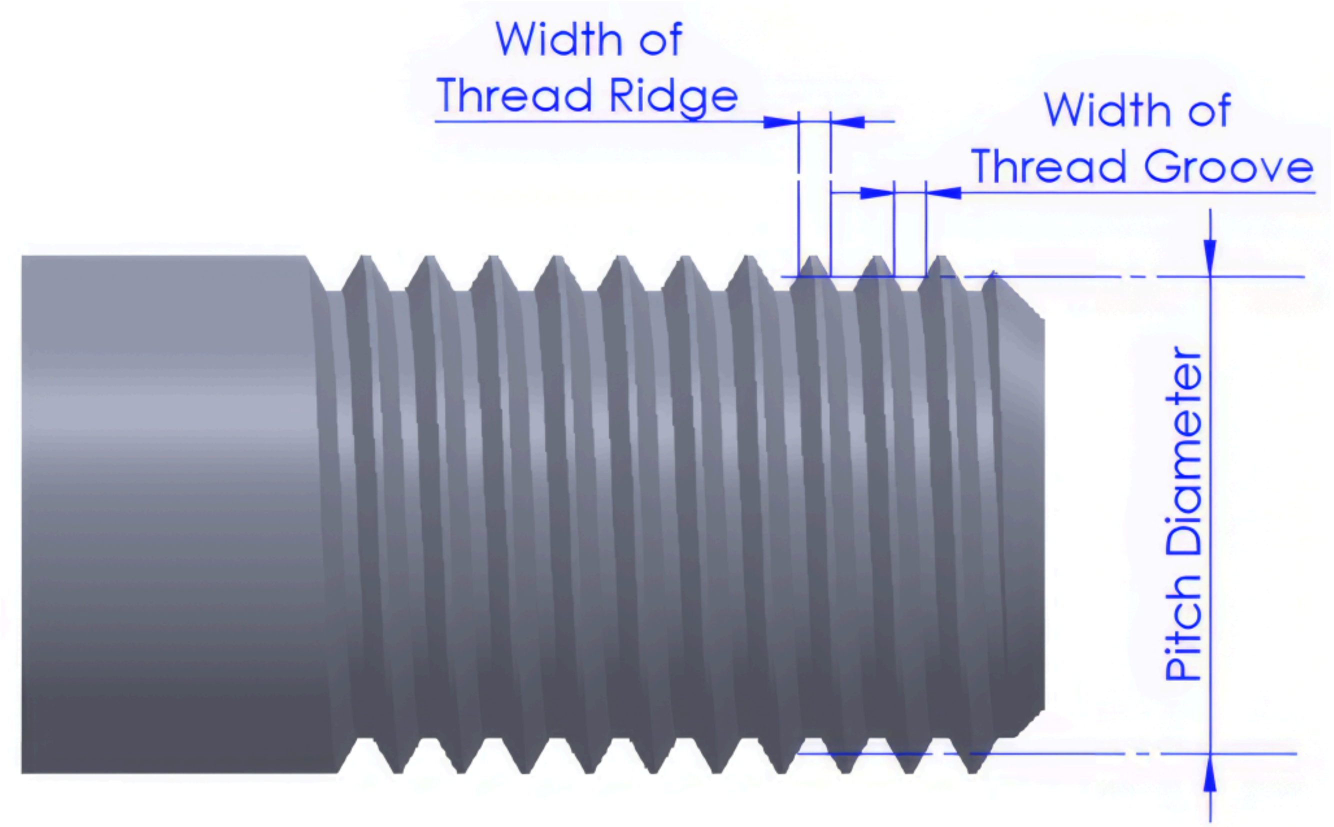
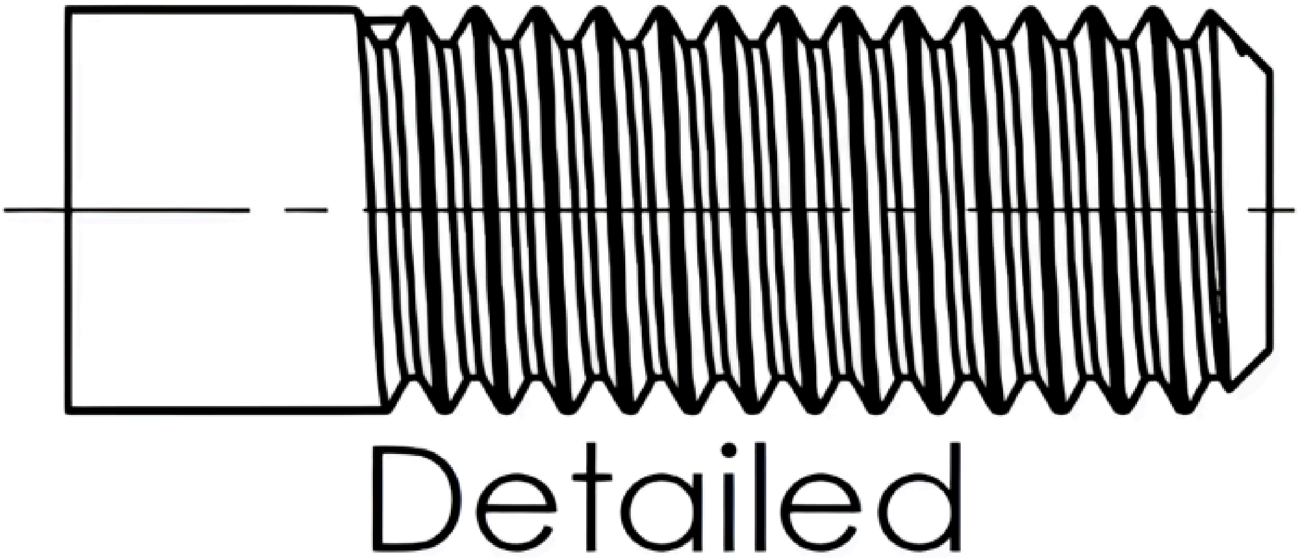
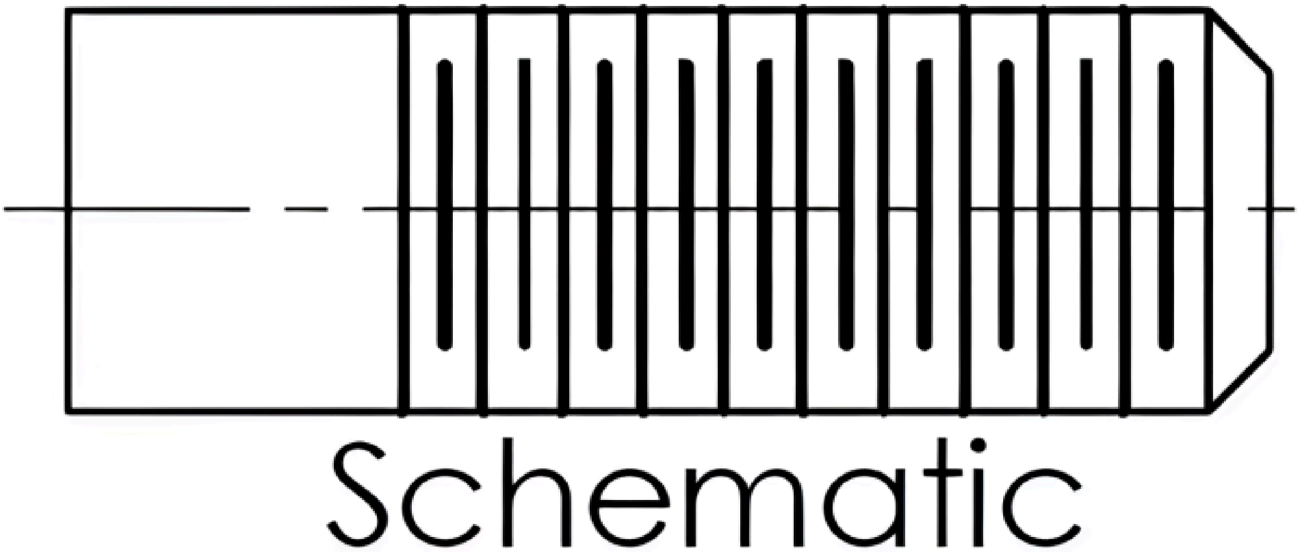


Figure 10-18. The pitch diameter is where the ridge and groove widths are equal.

10.10 THREAD REPRESENTATION

Threads may appear on a drawing in a few different ways. Threads may appear as detailed, schematic, or simplified. See these thread variations in Figure 10-19. While the schematic and simplified techniques were most common before the use of CAD systems, they will still be found on many drawings. The detailed thread representation provides a realistic view of the thread but can be difficult to view on smaller thread diameters. In this case, the schematic or simplified may be used to represent the thread.


Detailed


Schematic

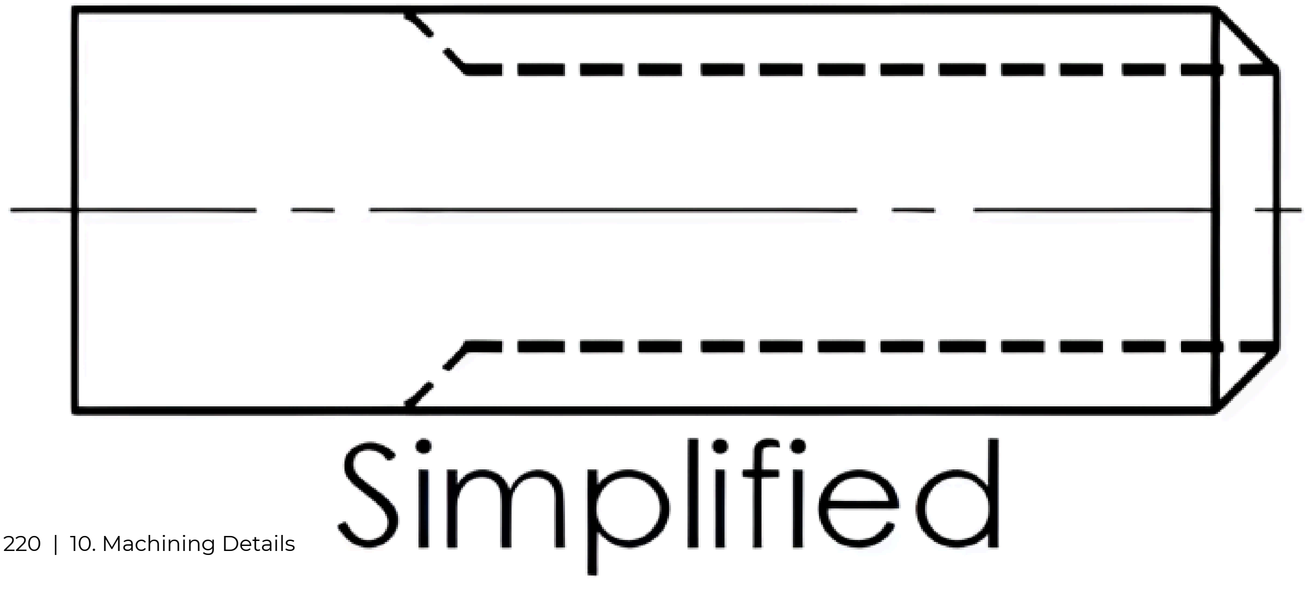

Simplified

Figure 10-19. Detailed, schematic, and simplified are used to represent threads.

10.11 THREAD SPECIFICATIONS

The details of the thread will be identified with a leader and note. The information will commonly include the major diameter, threads per inch, pitch for metric, thread series, and thread class. Additional information, such as thread direction and the number of starts on a multiple-start thread, may also be included, as shown in Figure 10-20.

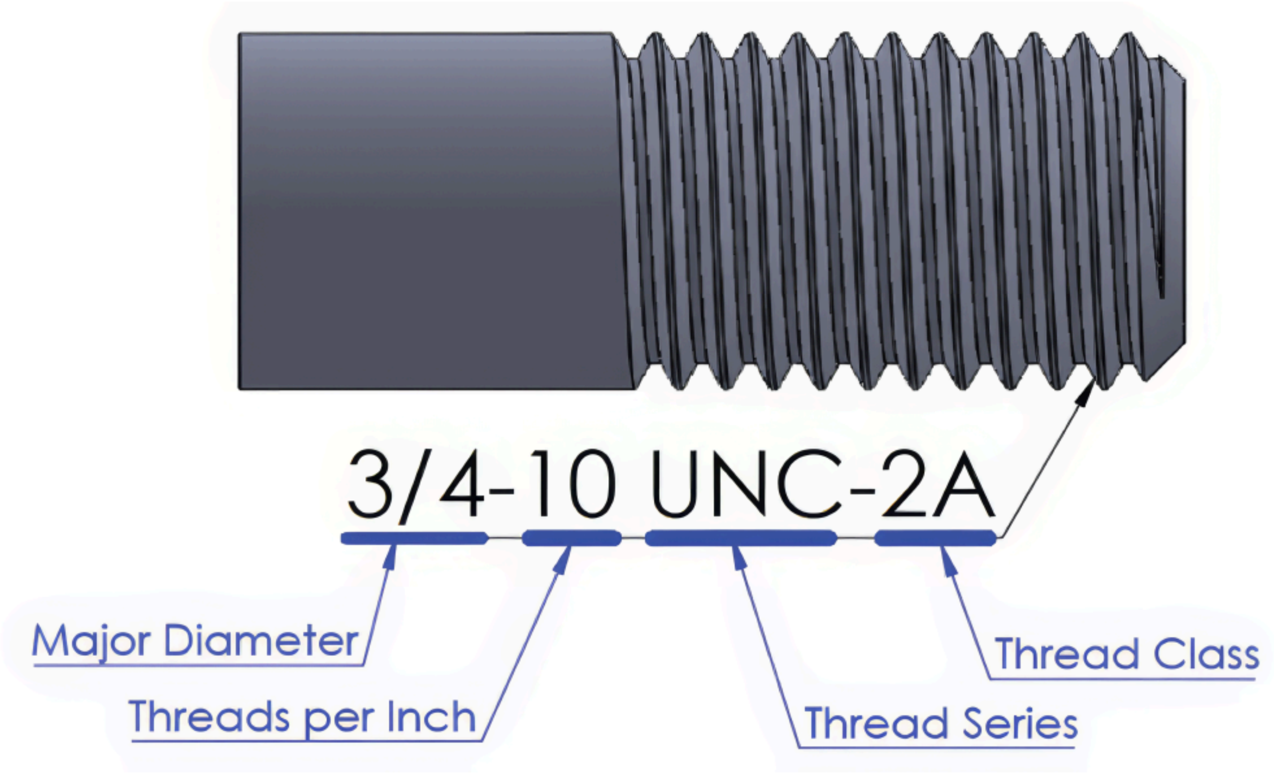


Figure 10-20. Inch thread specifications.

Major diameter is the nominal major diameter size. 1/4 inch and larger diameters will use fractional dimensions, and smaller diameters will use a number designation. The specific range for the major diameter size requirements can be referenced in the *Machinery's Handbook*.

Threads per inch is the number of threads within a 1-inch length of the thread.

Thread series refers to unified inch threads that will be in one of four thread series: course, fine, extra fine, and constant pitch.

- Unified course threads (UNC) are generally used, and are most commonly on nuts and bolts.
- Unified fine threads (UNF) will have a shorter pitch, which is useful where limited thread engagement is available, such as in thin material or where the thread is used for a fine adjustment application.
- Unified extra fine (UNEF) is less common than UNC and UNF series and used in applications where very little thread engagement is available.
- Unified constant-pitch (UN) is used in high-pressure applications and large-diameter threads.

Thread class refers to the fit tolerance of inch series threads and falls into three classes: 1A, 2A, or 3A for external threads; and 1B, 2B, or 3B for internal threads. These specific tolerance sizes are referenced in books such as the *Machinery's Handbook*.

- 1A and 1B are the loosest-fit thread and used in applications where frequent use is required, and a close tolerance is not required.
- 2A and 2B threads are used for general-purpose applications such as nuts and bolts. They are to be used where a thread classification is not included.
- 3A and 3B will have the tightest tolerance and used for more precise applications.

The details included in a metric series thread note will include metric series designation (M), major diameter, thread pitch, and thread tolerance class. See Figure 10-21.

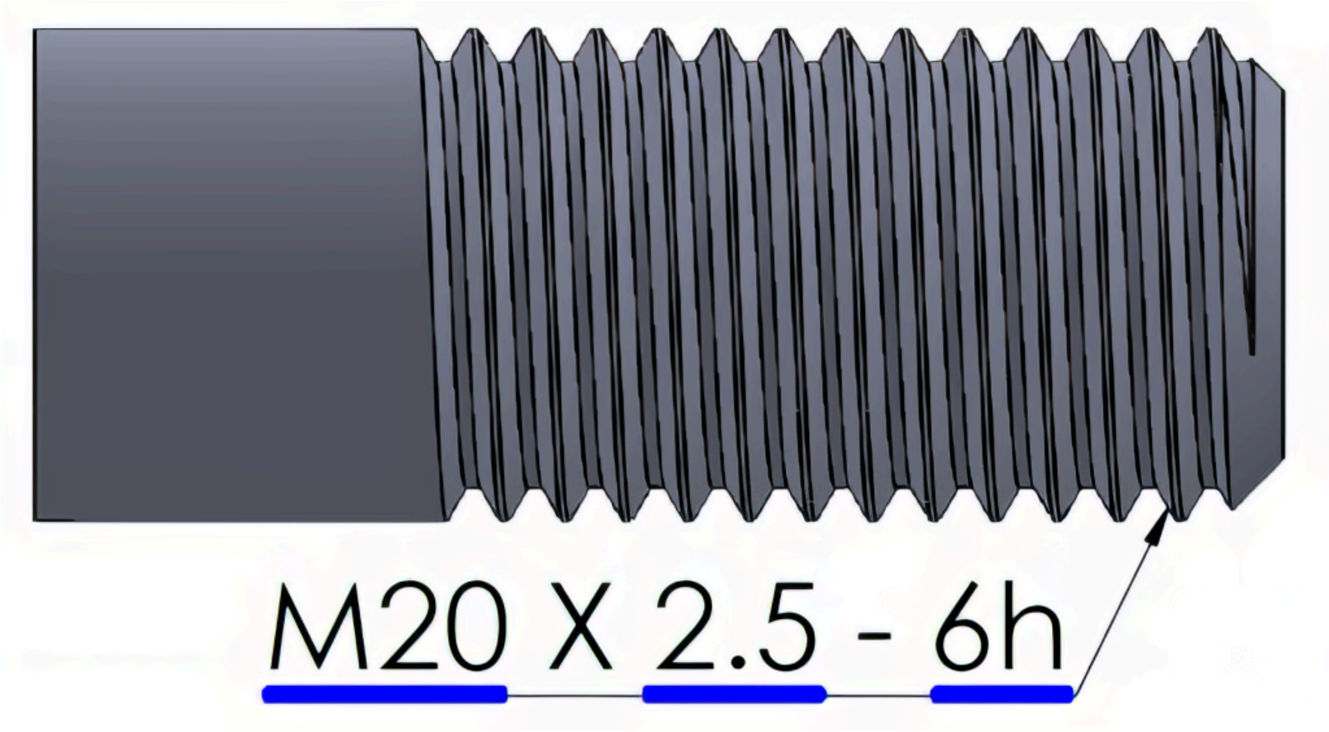


Figure 10-21. Metric thread specifications.

Metric series major diameter – This is the nominal major diameter size in millimeters. The specific range for the major diameter size requirements can be referenced in the *Machinery's Handbook*.

Thread pitch – This is the distance between threads in millimeters.

Thread tolerance class – Metric threads use numbers to indicate the amount of tolerance allowed: the smaller the number, the smaller the tolerance. Lowercase letters are used for external threads, and uppercase letters are used for internal threads. A 6H and 6g are comparable to an inch 2A and 2B and are applied when no tolerance is specified. The *Machinery's Handbook* and similar texts reference the specific tolerance sizes.

Additional information for a multiple-start thread, such as double-lead or triple-lead, may be included. Right-hand threads are presumed unless an LH is indicated to represent a left-hand thread.

LEARNING ACTIVITIES

Exercise 10.10-1

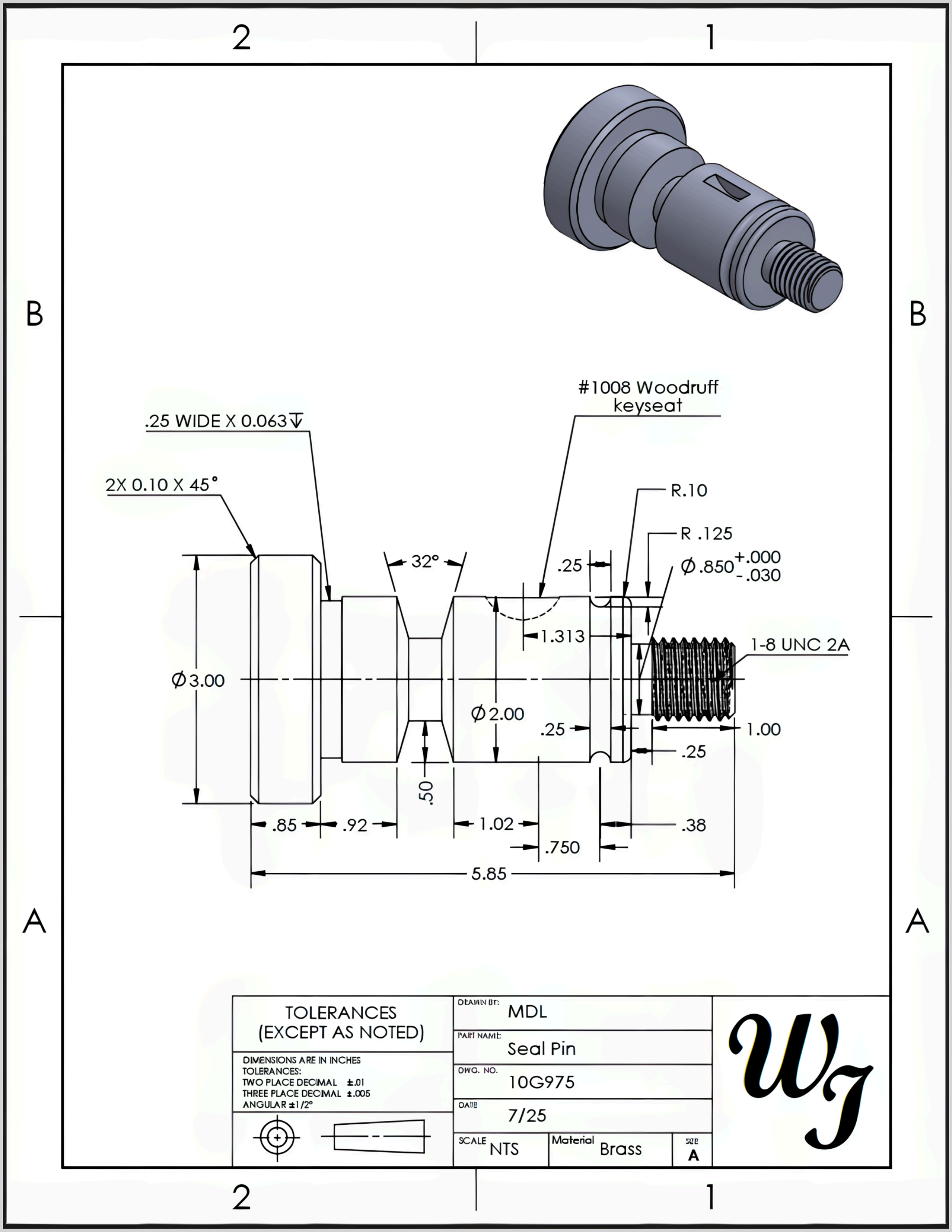
An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=424#h5p-29

Exercise 10.10-2

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=424#h5p-30

Exercise 10.10-3

Refer to the print below to respond to the missing dimensions in the activity beneath it.



TOLERANCES
(EXCEPT AS NOTED)

DIMENSIONS ARE IN INCHES
TOLERANCES:
TWO PLACE DECIMAL ±.01
THREE PLACE DECIMAL ±.005
ANGULAR ±1/2°

DRAWN BY: MDL
PART NAME: Seal Pin
DWG. NO. 10G975
DATE 7/25
SCALE NTS
Material Brass
SIZE A

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=424#h5p-31

References:

Barsamian, M.A., & Gizelbach, R. (2022). *Machine trades print reading*. Goodheart-Willcox.

Schultz, R. L., & Smith, L. L. (2012). *Blueprint reading for machine trades* (7th ed.). Pearson.

Images:

11. Section Views

11.1 INTRODUCTION

Learning Objectives

- Define section terminology
- Identify various section lines
- Interpret different types of sectional views

Terms

- Sectional view
- Full section
- Half-section
- Offset section
- Broken-out section
- Revolved section
- Removed section

Sectional views are a powerful tool that allows engineers, designers, and technicians to visualize the internal features of an object or component that cannot be seen in a regular orthographic view. By cutting or slicing through an object, a sectional view reveals the hidden details, such as internal structures, cavities, and complex geometries.

There are several different types of sectional views, including full sections, half-sections, offset sections, broken-out sections, revolved sections, and removed sections. These views are often an additional view placed somewhere on the drawing sheet but may also replace one of the primary views as well.

This chapter will cover the principles and techniques of interpreting sectional views, including the use of cutting planes and section lines. We will also discuss the importance of the dimensions and annotations of sectional views to convey critical information about the object's internal features.

11.2 SECTIONING PRINCIPLES

A sectional view represents the part of an object remaining after a portion has been imaginarily "cut" with a cutting-plane line, revealing the interior features. With the exception of some features such as thin ribs or spokes, the material that has cut through is represented with section lines. The diagonal and parallel section lines, known as the cast iron section line, are most commonly used for general purposes; however, the designer may use the material-specific sectioning style when called for. Some examples of the material-specific section styles are shown in Figure 11-1.

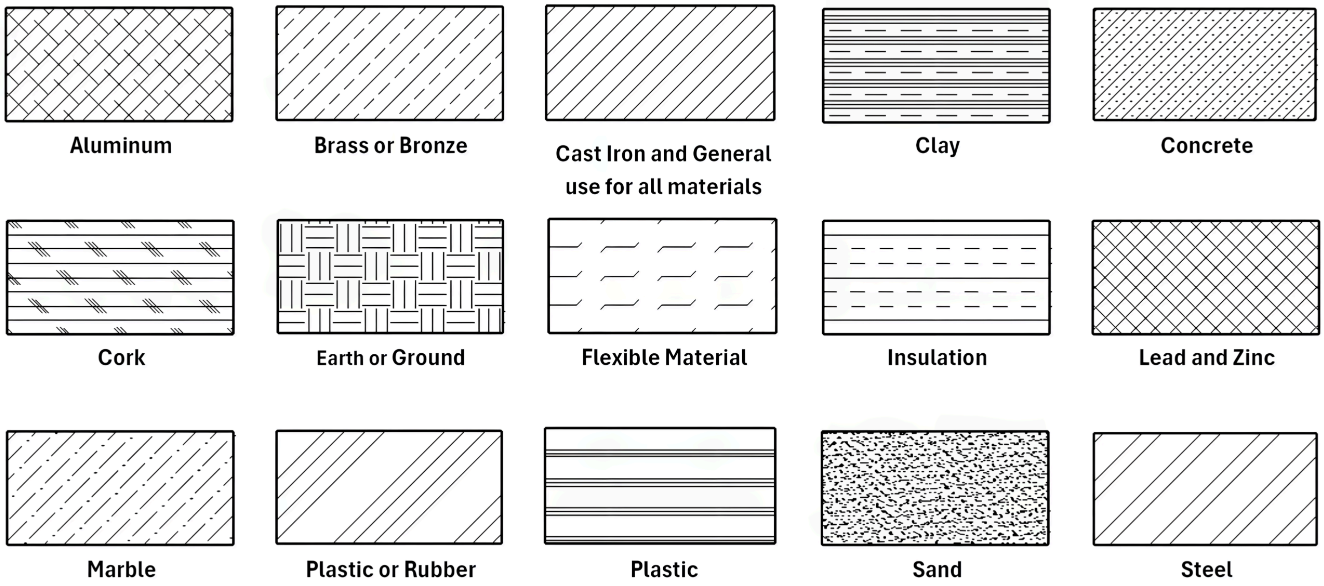
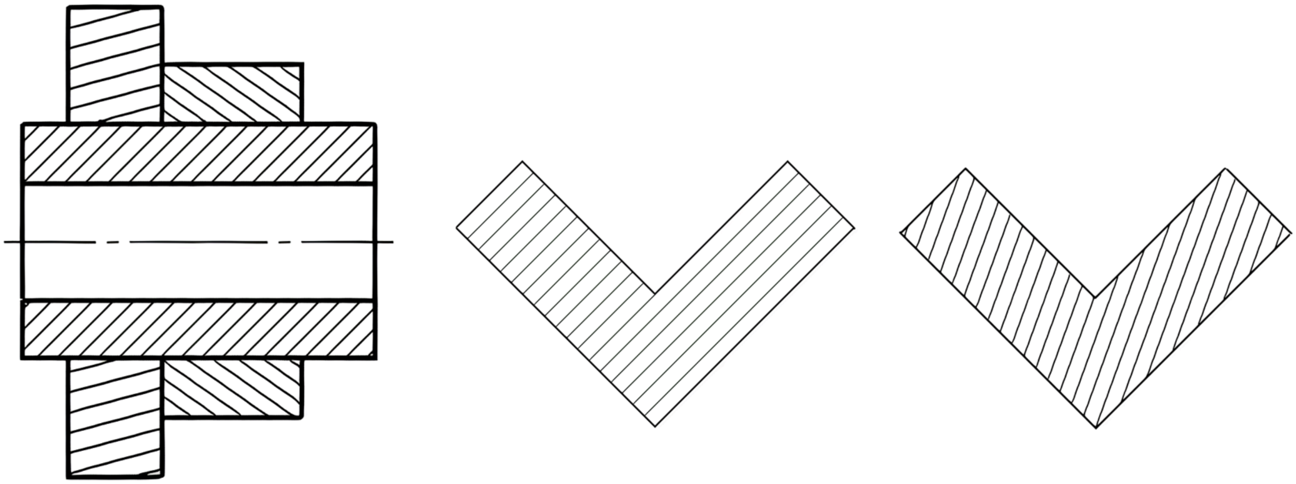


Figure 11-1. Material-specific section lines.

Line Orientation

A section view will reveal features hidden in other views, allowing these features to be drawn with visible lines rather than hidden lines. To avoid confusion, any remaining hidden features are not represented in the section view. The general section line pattern and angle may be adjusted for clarity to differentiate between separate components in an assembly and when the lines may run parallel to the parts' visible lines, as illustrated in Figure 11-2.



3-piece assembly **Incorrect** **Correct**

Figure 11-2. Section line pattern adjusted to avoid confusion.

Thin Features

When the cutting plane line passes through a thin web or rib feature, the section lines will not be included for that portion to avoid a false sense of thickness for this feature. See Figure 11-3.

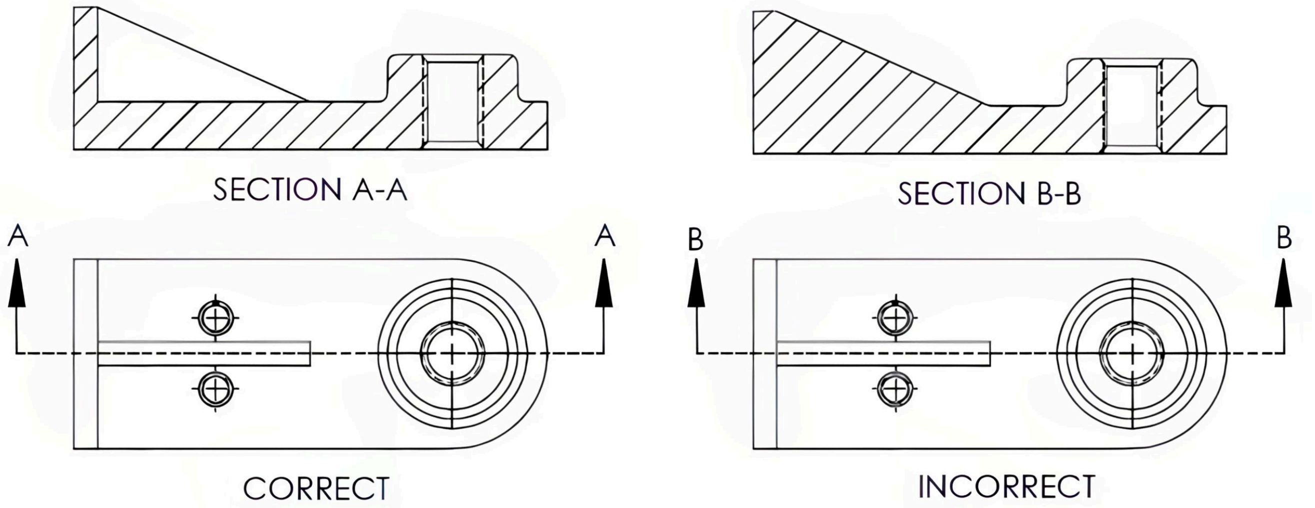


Figure 11-3. Thin features of a section view are not included in the section lines.

Rotated Features

When a sectioned object contains features such as holes or ribs that rotate around a common center point, these features may not be aligned with the cutting-plane line. When this occurs, the features may appear distorted because they are being seen at an angle to the section view and not perpendicular to it. Figure 11-4 shows the sectioned view as it would appear in the view; the rib on the left shows the actual shape, while the right appears distorted due to the perspective of the view. Traditional practice is to rotate these features in the section view to appear as their actual shape, as shown in Figure 11-5. Before CAD, this practice of rotating these features also made the drawings easier to create for the manual draftsperson. Today, with CAD-created drawings, this practice is no longer as prevalent as it once was.

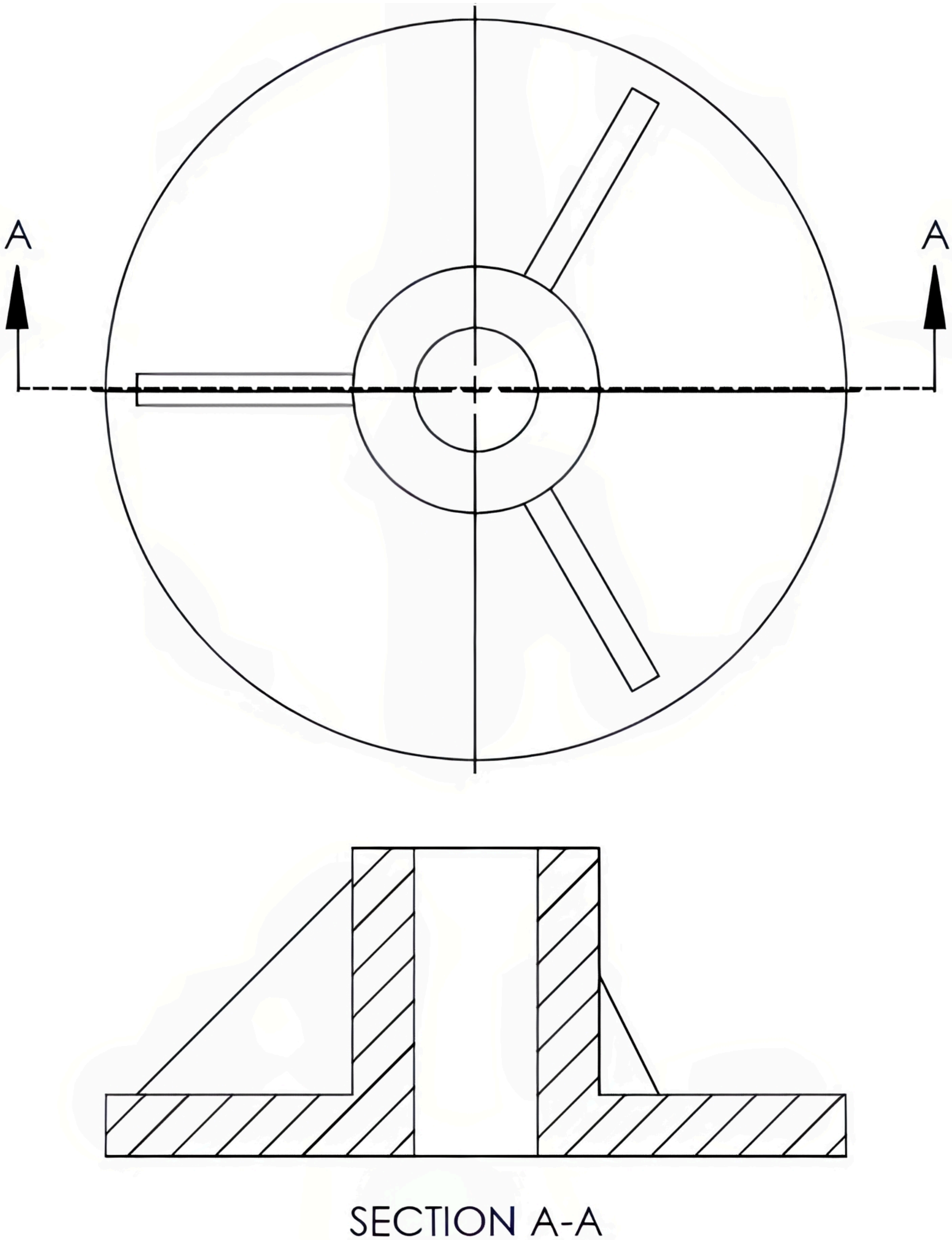


Figure 11-4. The rib on the right appears distorted, as seen from the sectioned view.

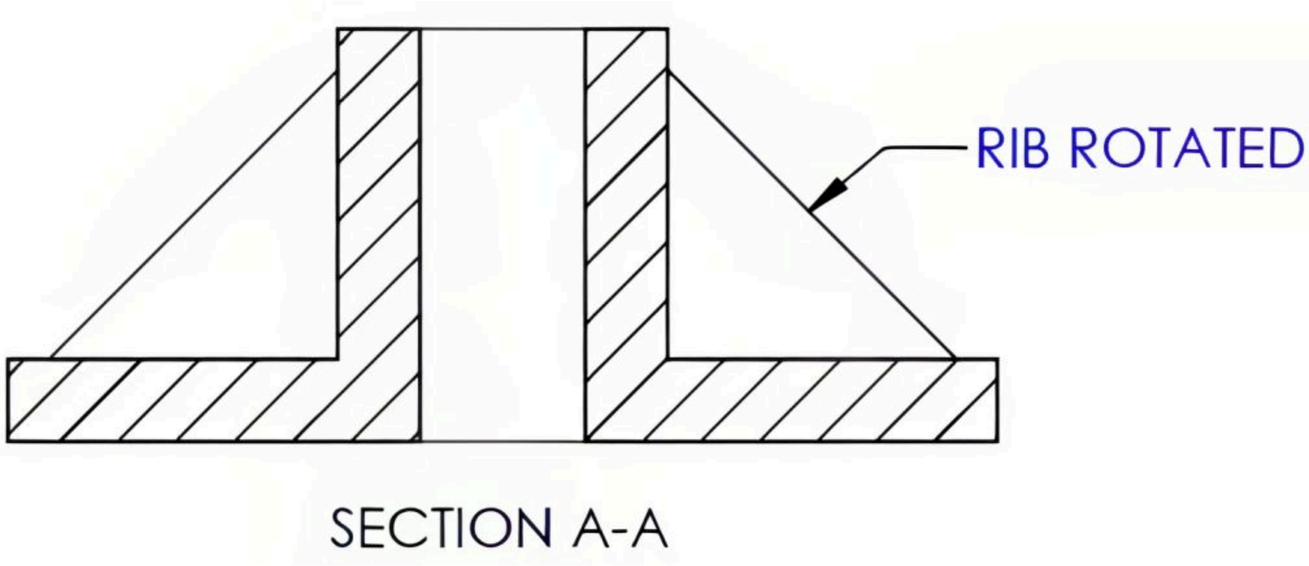


Figure 11-5. The rib on the right is rotated in the sectioned view, showing the actual shape of the rib.

11.3 FULL SECTION

A **full section** view is one in which the cutting-plane passes straight through the entire portion of the part where the cutting-plane line is placed as if the part were cut into two pieces. The cutting-plane line will be shown with the arrows indicating the direction in which the removed view is observed. The letters on the cutting-plane line will match the letters of the sectioned view to reduce confusion when multiple cutting-planes or viewing-planes are used on the drawing. If the sectioned view has a clear source, the identifying letters may be omitted.

Notice the hole in Section B-B is not represented with hidden lines as shown in Figure 11-6. Hidden lines are typically omitted when section views are used because the internal features will be viewed in the sectioned view.

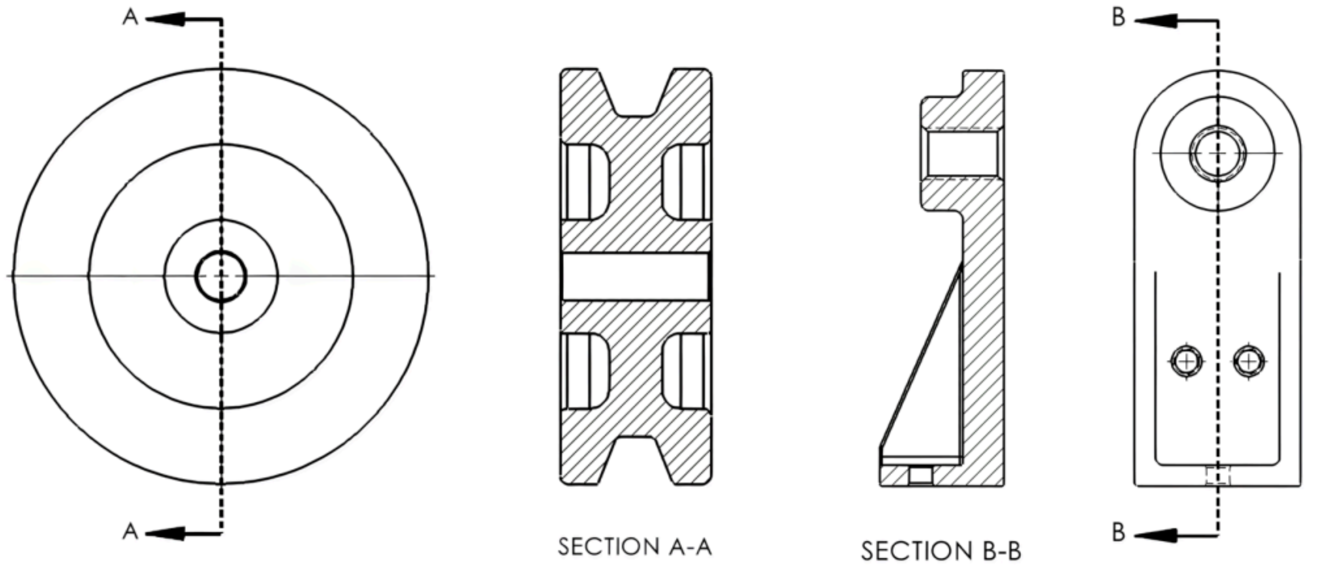


Figure 11-6. Full section view – part is cut in half, revealing the full section.

11.4 HALF-SECTION

A **half-section** view will show only half of the part sectioned. Rather than cutting straight through the part, the cutting-plane line is drawn to the center and turns 90° to remove one quarter of the part. Typically reserved for symmetrical parts, half-sections show half of the part's exterior and half of the part's interior in one view.

The half-section term is commonly misused for a full-section view where the part is cut in half. Remember that the full section shows the part fully sectioned, while the half-section shows half of the part sectioned. See Figure 11-7.

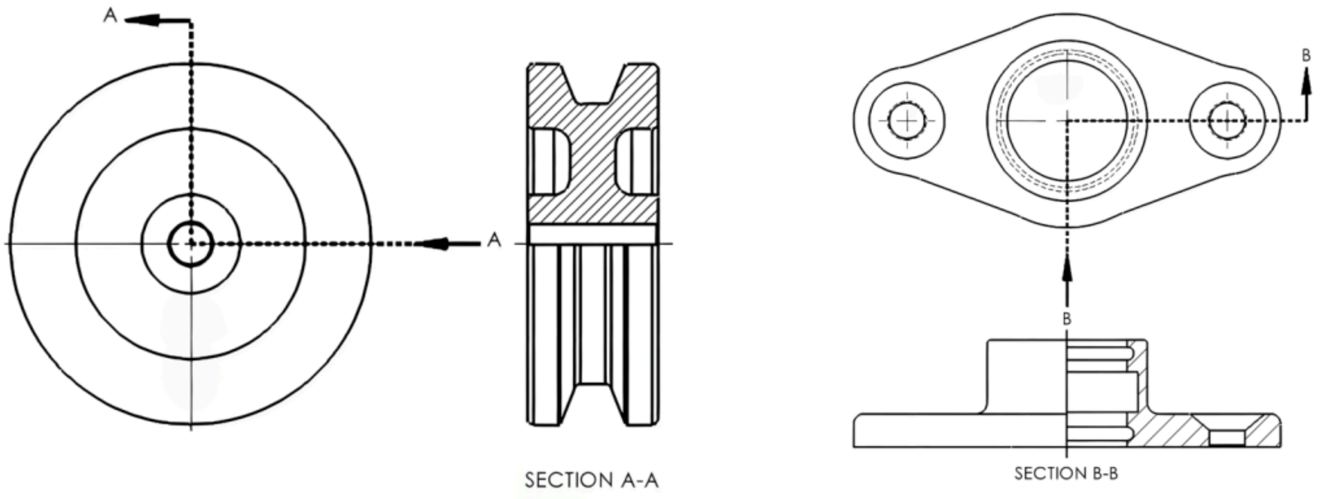


Figure 11-7. Half-section view – one-fourth of the part is removed, revealing one-half of the material.

11.5 OFFSET SECTION

An **offset section** is designed with a cutting-plane line that takes a slight turn or "offset" to capture features that aren't aligned in a straight path, which wouldn't be possible if only a traditional cutting-plane were used. While viewing the sectioned area, you will not notice these offsets; the section lines will still look as though the cut is straight, yet they include those offset features.

You'll also notice different styles of cutting-plane lines in the examples below: Section A-A follows ANSI standards, while Section B-B adheres to the ISO style. Despite these stylistic differences, both serve the same essential purpose and function, as shown in Figure 11-8.

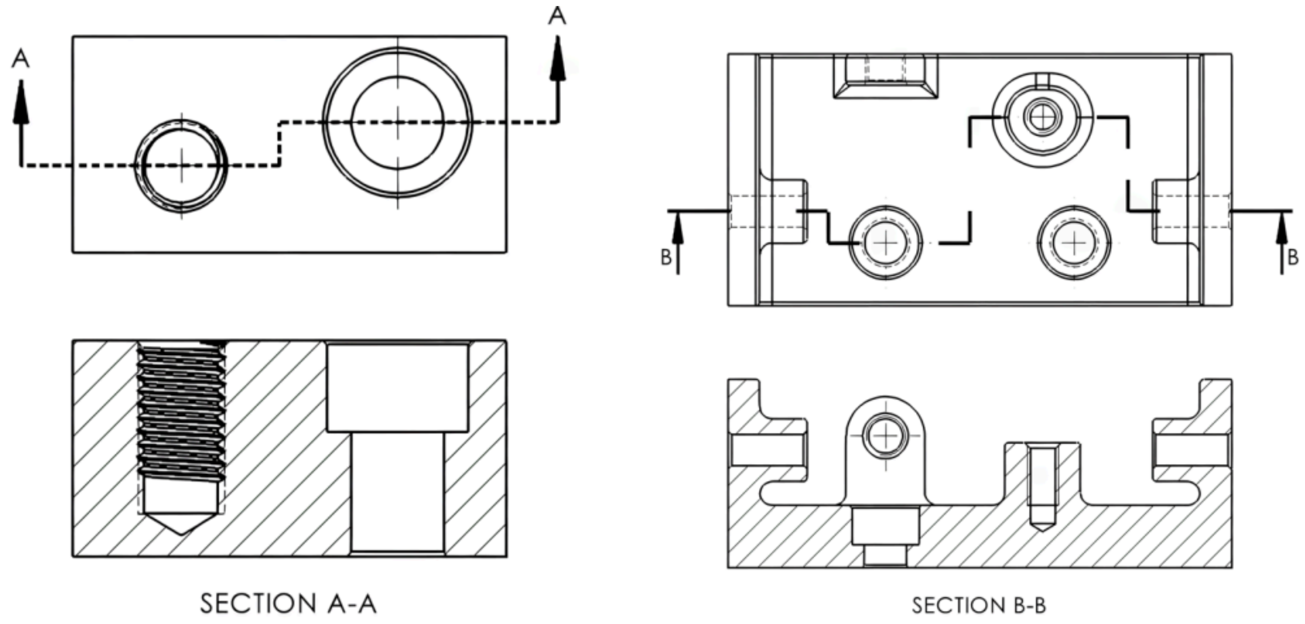


Figure 11-8. Offset section – moves the cutting line to view misaligned features. Shown with ANSI (right) and ISO (left) line styles.

11.6 BROKEN-OUT SECTION

A **broken-out section** is a sectional view located within an existing view, with the outline separated by a short break line. When a single detail requires a clearer view, the broken-out section removes a portion of the part's material, as if to take a "bite" out of the part, revealing the interior detail. View Figure 11-9.

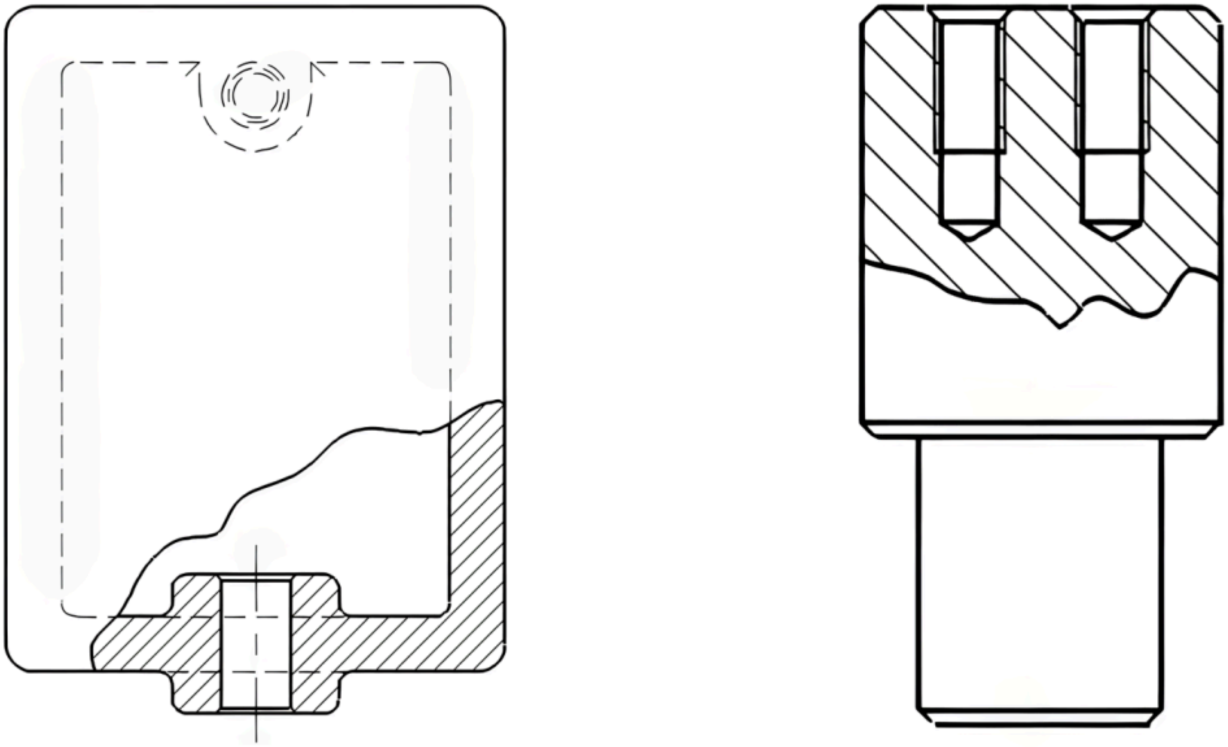


Figure 11-9. Broken-out section – removes a portion of the material to reveal hidden features.

11.7 REVOLVED SECTION

A **revolved section** is a cross-sectional view of a single portion of a part placed directly on one of the views. Rather than creating a separate view, a revolved section places the rotated portion within the same principal view. A centerline will be used to indicate the rotation of the view. Break lines may be used on each end of the section view, or the sectioned view may be placed directly on the part. Revolved sections are often used to display the material's shape for ribs, spokes, and stock shapes, as seen in Figure 11-10.

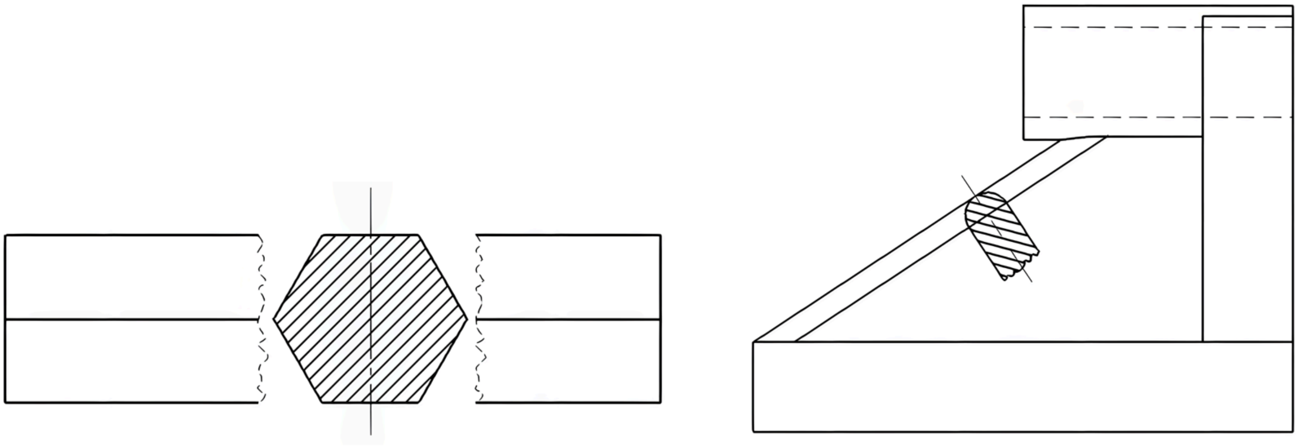
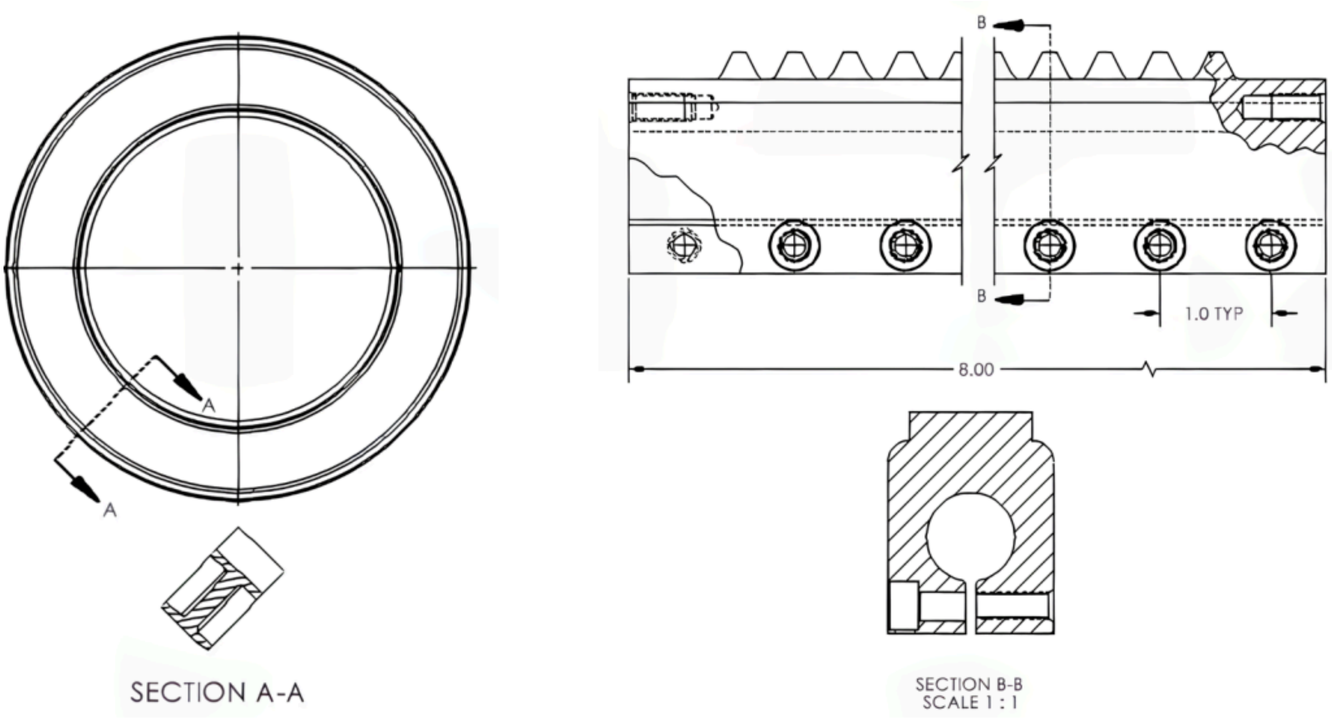


Figure 11-10. Revolved section – reveals a cross-sectional view within a principal view.

11.8 REMOVED SECTION

A **removed section** is similar to a revolved section but is removed from the view and placed elsewhere on the print sheet. Removed sections will not directly align with the part or cutting-plane lines, although the view will maintain the sight direction indicated by the arrows of the cutting-plane line. One benefit of the removed sections is that they enable the section to be positioned in an open space on the sheet, potentially decreasing the required sheet size. Another advantage is that the view may use a scale different from the drawing. Figure 11-11 shows a removed section.



SECTION A-A

SECTION B-B
SCALE 1 : 1

Figure 11-11. Removed section – removed from the drawing alignment, may be drawn using a scale different from the main view.

A removed section may also use a centerline extending from where the cross-sectional view is removed rather than using the cutting-plane line. A removed section view is only used where the removed section is a symmetrical shape. See Figure 11-12.

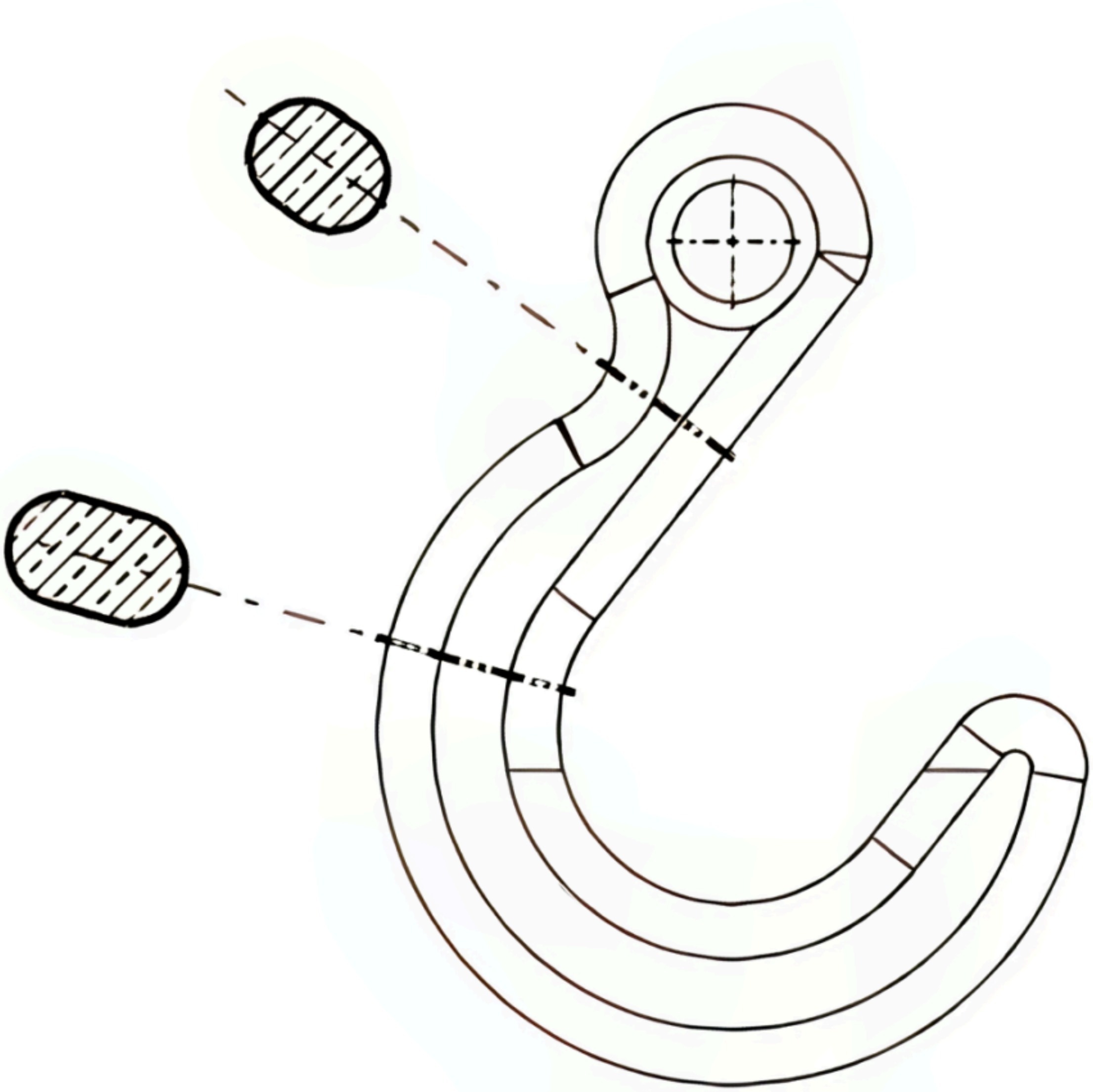


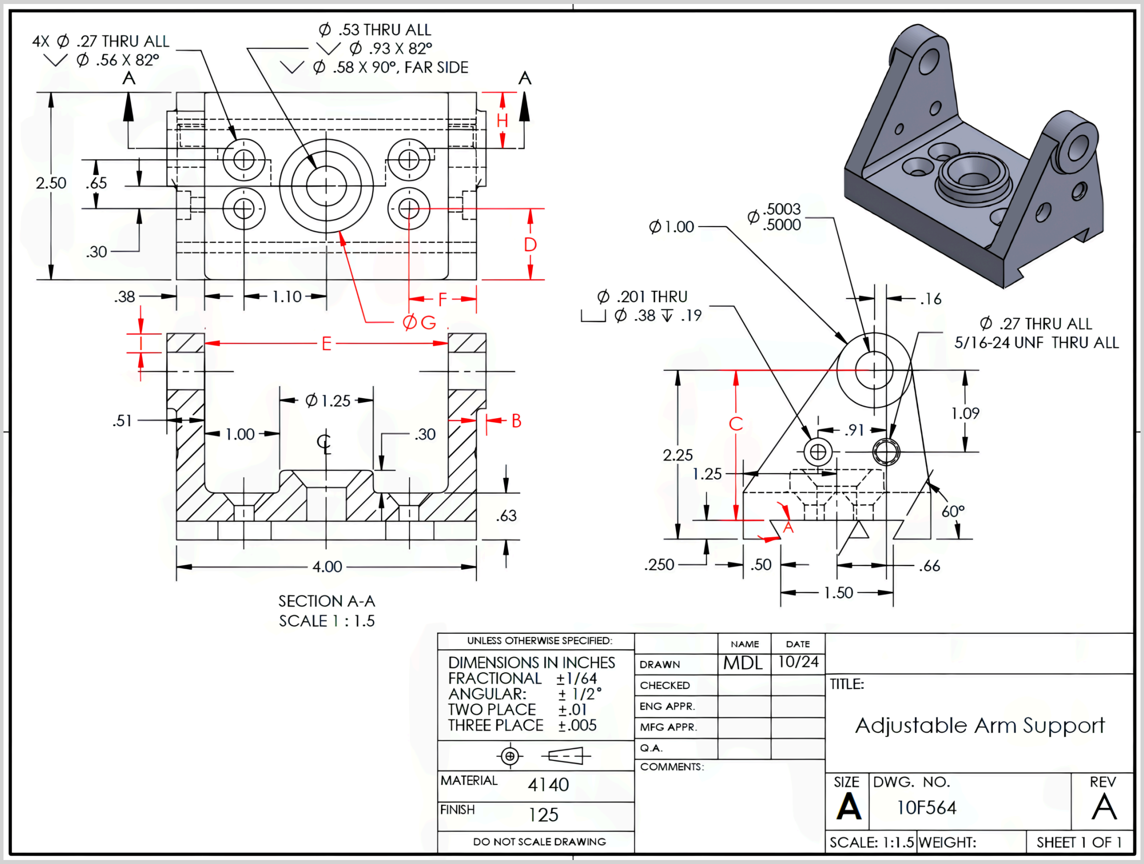
Figure 11-12 Removed section – projected from the symmetrical portion.

LEARNING ACTIVITIES

Exercise 11.8-1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=461#h5p-32

Exercise 11.8-2



An interactive H5P element has been excluded from this version of the text. You can view it

online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=461#h5p-33

References:

Barsamian, M. A., & Gizelbach, R. (2022). *Machine trades print reading.* Goodheart-Willcox.

Schultz, R. L., & Smith, L. L. (2012). *Blueprint reading for machine trades* (7th ed.). Pearson.

Images:

All images by Mark Lorier and licensed under CC BY-NC 4.0 except where otherwise noted.

12. Auxiliary Views

12.1 INTRODUCTION

Learning Objectives

- Identify auxiliary views
- Interpret details on auxiliary views
- Determine dimensions on auxiliary views

Terms

- Inclined plane
- Foreshortened view
- True shape
- Curved surface
- Auxiliary view
- Secondary auxiliary view
- Partially enlarged view

Typically, the six main views—front, back, top, bottom, right, and left—will cover all the essential information you need. However, there are instances where a print might need extra views to convey its features and dimensions fully. For example, when a part has an angled surface, the standard views will not show that surface's actual shape directly. In this case, including an additional view to represent the surface is helpful. This additional perspective can better represent the actual shape of that surface and accurately show where important features are located.

12.2 INCLINED PLANES

An **inclined plane** is any surface that isn't aligned parallel to the two-dimensional projection planes in an orthographic view. Because of this angle, these inclined planes don't appear in their actual shape when represented visually. When we project these surfaces onto the principal planes, the actual size and form of that plane appear smaller than they truly are; this phenomenon is known as a **foreshortened view**. When the view presents the part or feature in its actual shape, this is referred to as its **true shape**.

In Figure 12-1, notice the width of the inclined plane surface appears shorter when projected to the top and right views. Also, observe the visible line in the middle of the top and right-side views representing the intersection of the surfaces.

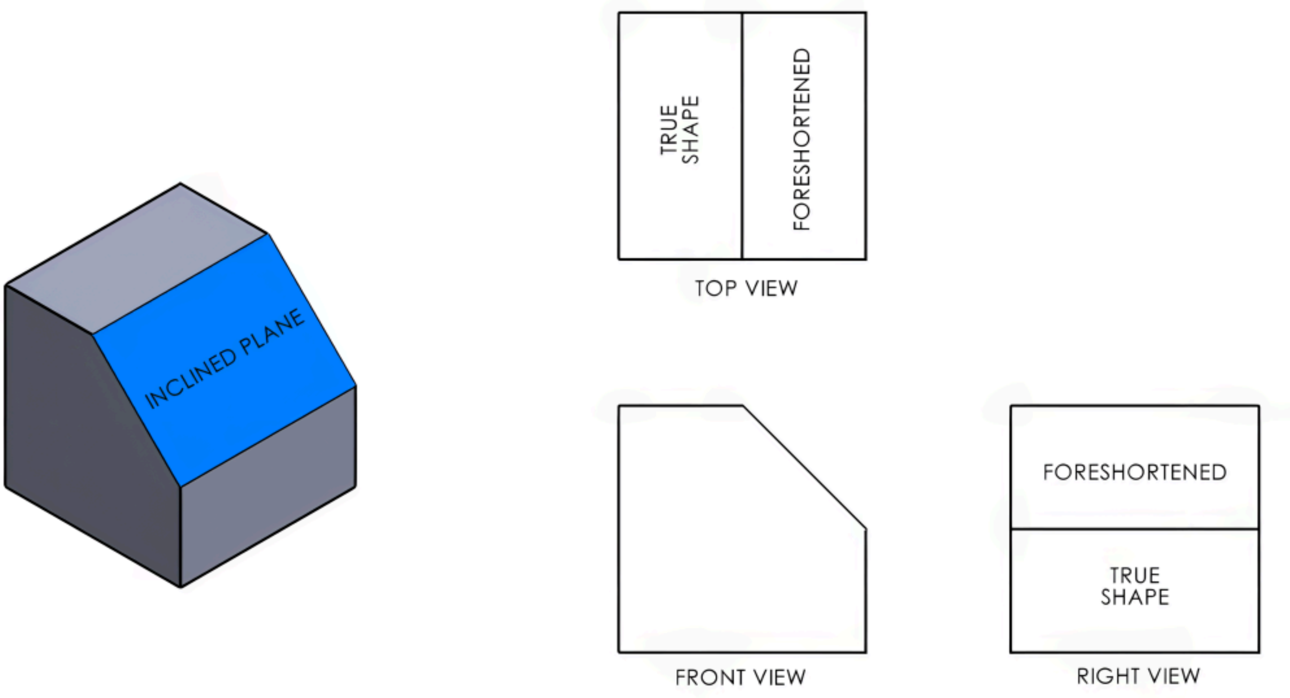


Figure 12-1. The inclined plane is shown as a foreshortened view.

12.3 CURVED SURFACE

Just as an inclined plane, or angled surface, will create a foreshortened view, a **curved surface** will create a foreshortened surface when projected to the orthographic views as well. In Figure 12-2, notice that the radius would not be represented in the top or right views. When a curved surface runs tangent to another surface, often a visible line will not be drawn to represent this transition.

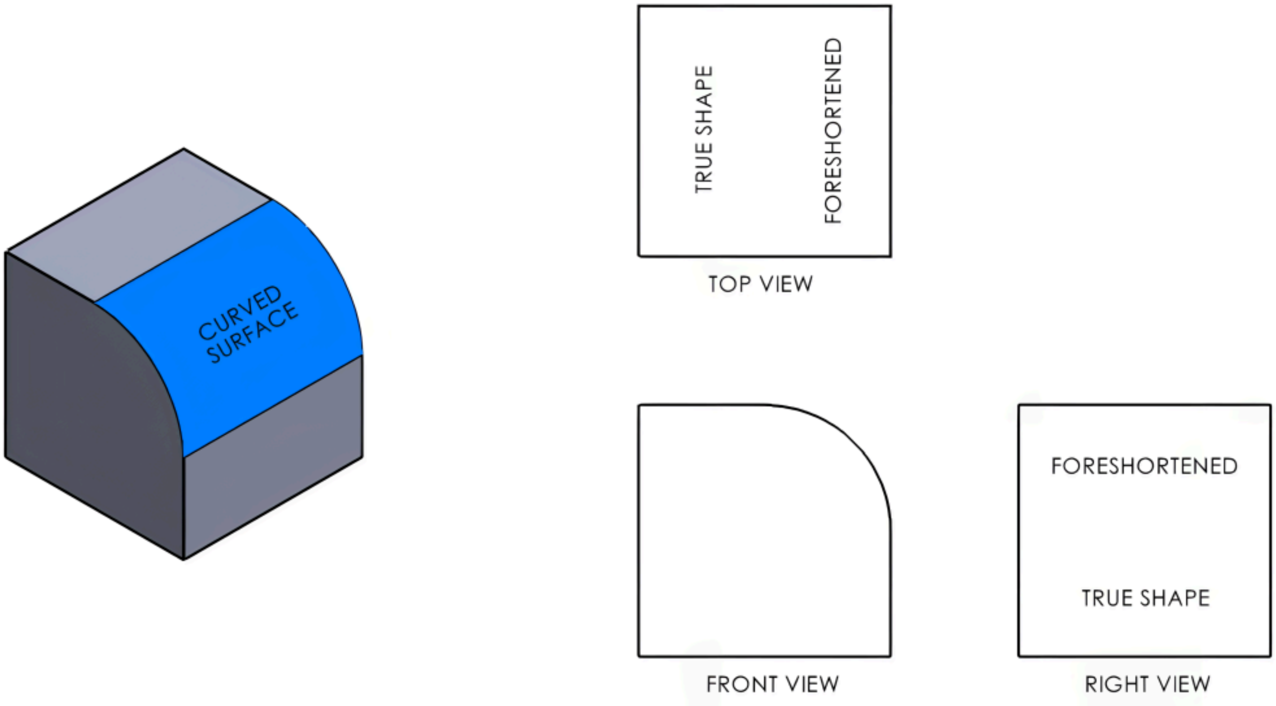


Figure 12-2. The curved surface is shown as a foreshortened view.

12.4 AUXILIARY VIEW

When the true shape of a part's surface or features cannot be shown in any of the six principal views, an auxiliary view is necessary. An **auxiliary view** is a view that is not aligned with the principal views used to present an inclined surface or feature in its true shape. An auxiliary view may be an additional view, or it may be used to replace a primary view.

In Figure 12-3, View A is an auxiliary view projected in line from the front view's inclined surface. This auxiliary view now displays the inclined plane in its true shape, allowing sizes and features to be properly dimensioned. Notice that the other two surfaces in View A are now foreshortened.

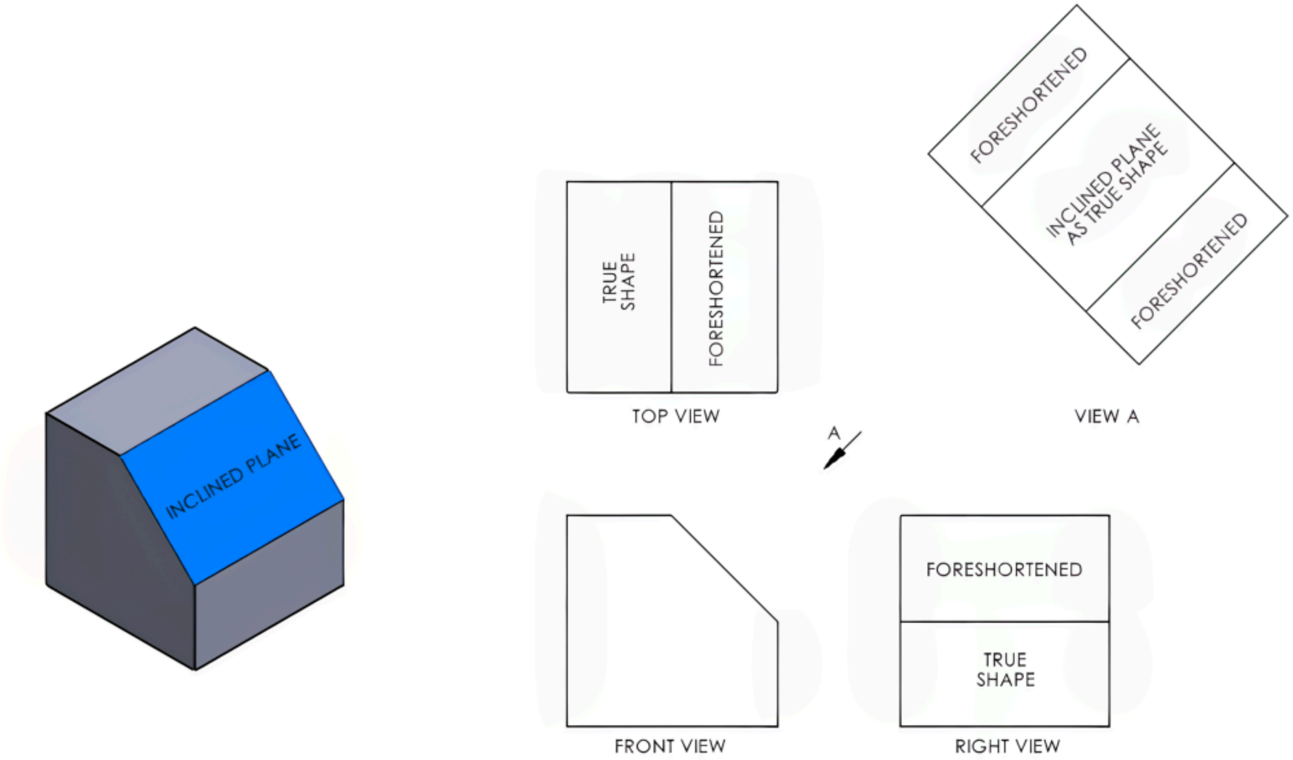


Figure 12-3. An auxiliary view provides a true shape view of the inclined plane.

An auxiliary view may also be projected from a feature such as a hole, as in View B, shown in Figure 12-4. In the View B example, this view is projected in line with the drilled hole.

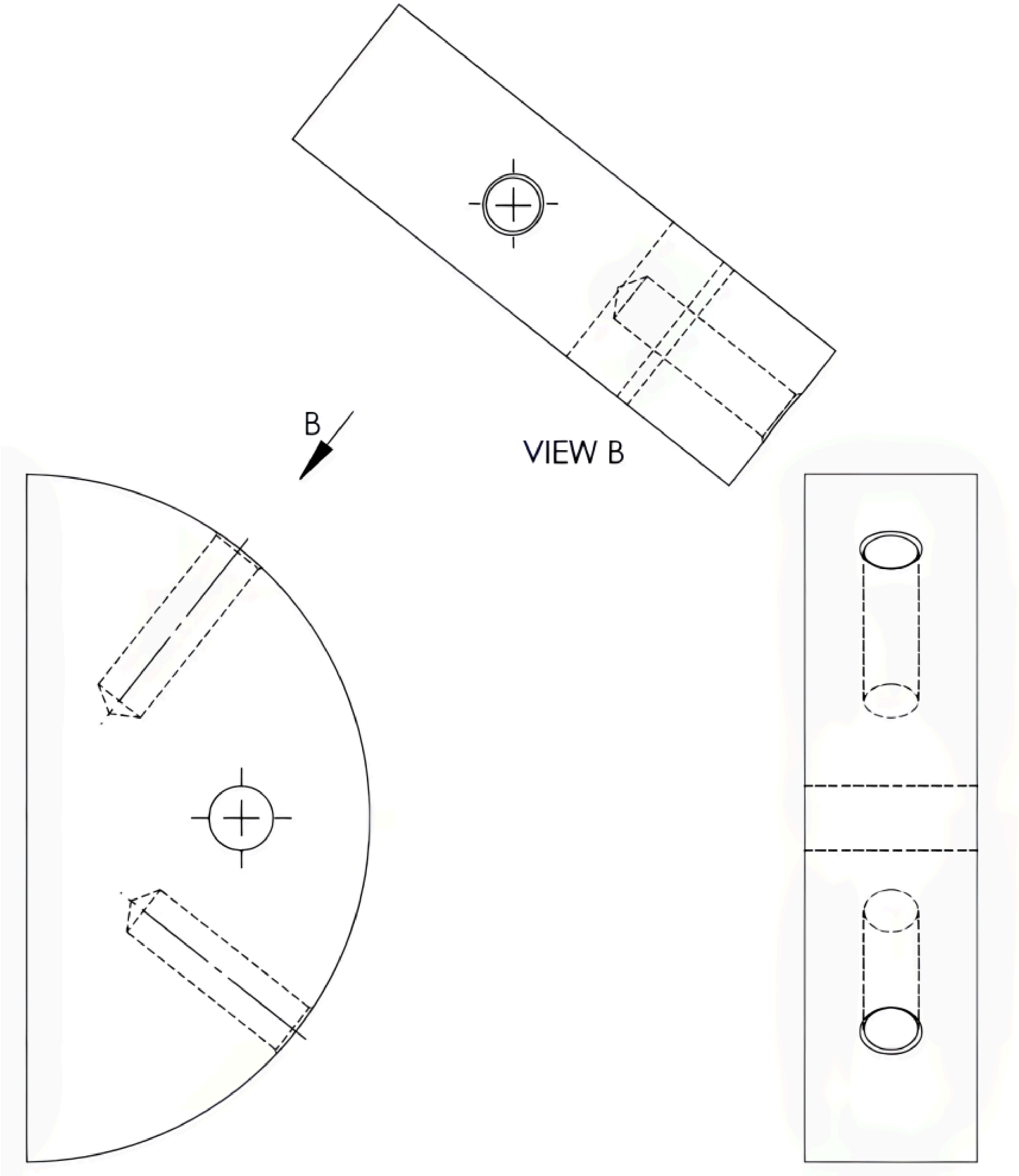


Figure 12-4. The auxiliary view is aligned with a feature from which it is projected.

12.5 SECONDARY AUXILIARY VIEWS

Surfaces that are at an angle to all of the principal planes are known as oblique surfaces. To show an oblique surface as a true shape, a secondary auxiliary view is necessary. A **secondary auxiliary view** is projected from a primary auxiliary view to properly display an oblique surface. A viewing-plane line and identifying view label may be omitted on a clearly represented auxiliary view. Figure 12-5 uses a secondary auxiliary view to view the oblique surface as a true shape.

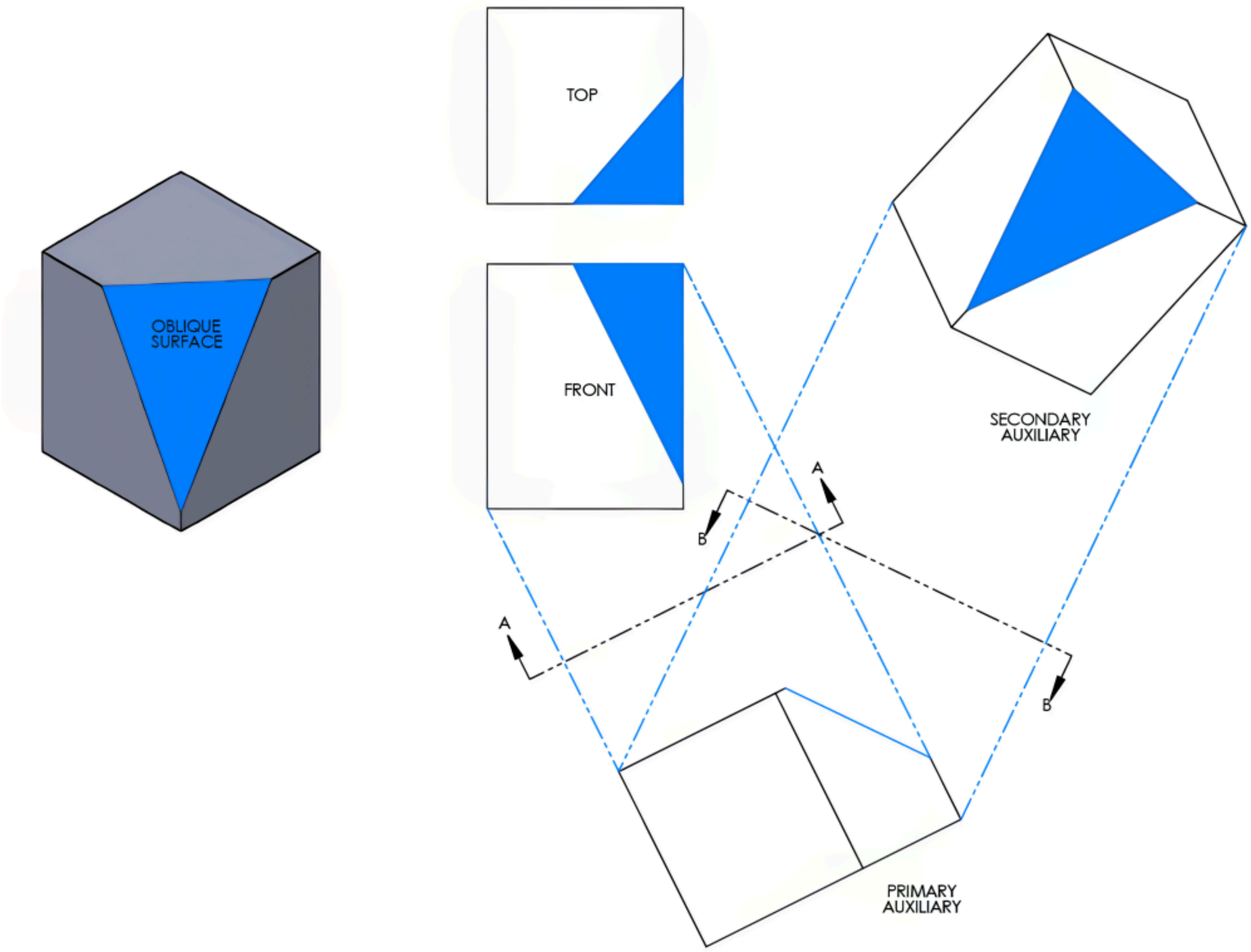


Figure 12-5. A secondary auxiliary view displays an oblique surface in its true shape form.

12.6 PARTIALLY ENLARGED VIEWS

When smaller details of a drawing are too small to easily be dimensioned, just that portion of the drawing may be increased to a larger scale. These **partially enlarged** or detailed views will be placed elsewhere on the print and identified with the view letter. They will also have their own increased scale located beneath the view. This allows for clearer dimensioning for these details without increasing the scale of the entire drawing. Figures 12-6 and 12-7 use a partially enlarged view to show the detailed portion on a larger scale. The location from which this view is removed will be identified on the main view with a letter and circular shape that is being enlarged.

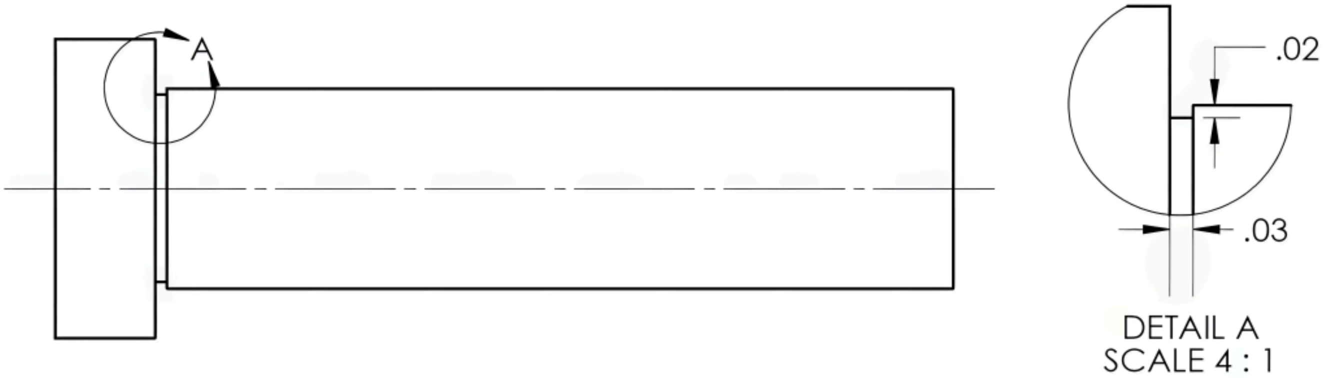


Figure 12-6. A partially enlarged view of a neck allows for a portion of the main view to be increased in scale for a clearer view.

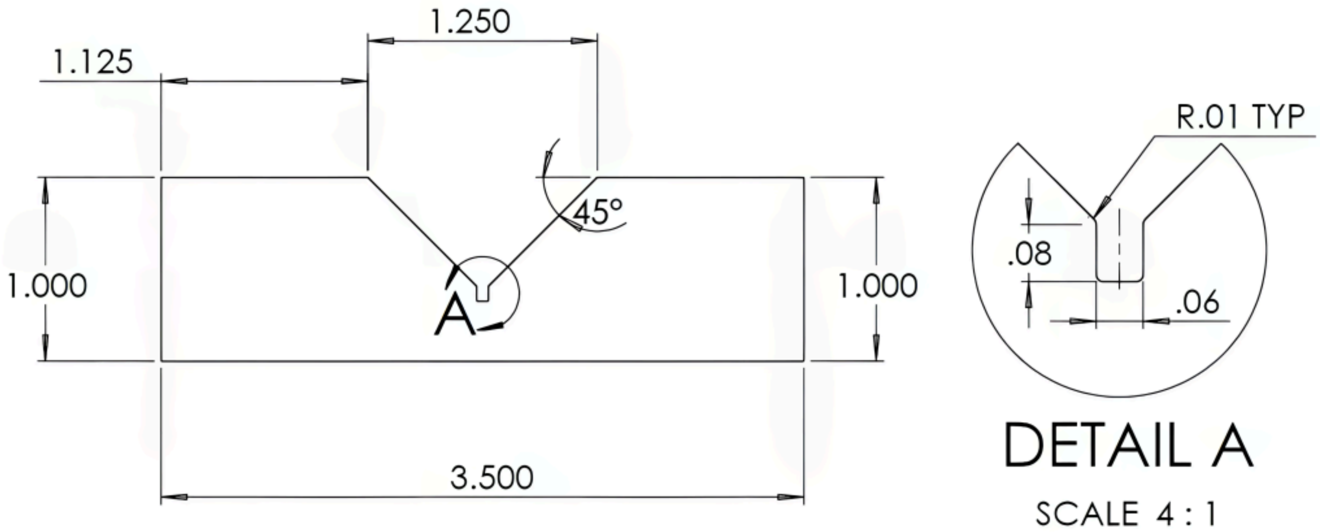


Figure 12-7. A partially enlarged view allows for a portion of the main view to be increased in scale for a clearer view.

LEARNING ACTIVITIES

Exercise 12.3-1 Surface Identification

Enter the letters that correspond to the correct spaces within the green circle of each slide.

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=480#h5p-34

Exercise 12.3-2 Surface Identification

Enter the letters from the isometric drawing below into the top of the correct balloons and indicate whether it is a true shape view with a “T,” an inclined surface with an “I,” or a curved surface with a “C” in the bottom of the balloon on the orthographic view.

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=480#h5p-35

Exercise 12.4-1 Auxiliary View

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=480#h5p-36

Exercise 12.4-2 Auxiliary View

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=480#h5p-37

Exercise 12.6-1 Auxiliary View

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=480#h5p-38

References:

Barsamian, M. A., & Gizelbach, R. (2022). *Machine trades print reading*. Goodheart-Willcox.

Schultz, R. L., & Smith, L. L. (2012). *Blueprint reading for machine trades* (7th ed.). Pearson.

Images:

13. Assembly Prints

13.1 INTRODUCTION

Learning Objectives

- Identify assembly drawings
- Identify parts lists
- Interpret parts lists
- Interpret sub-assembly drawings
- Interpret working assembly drawings

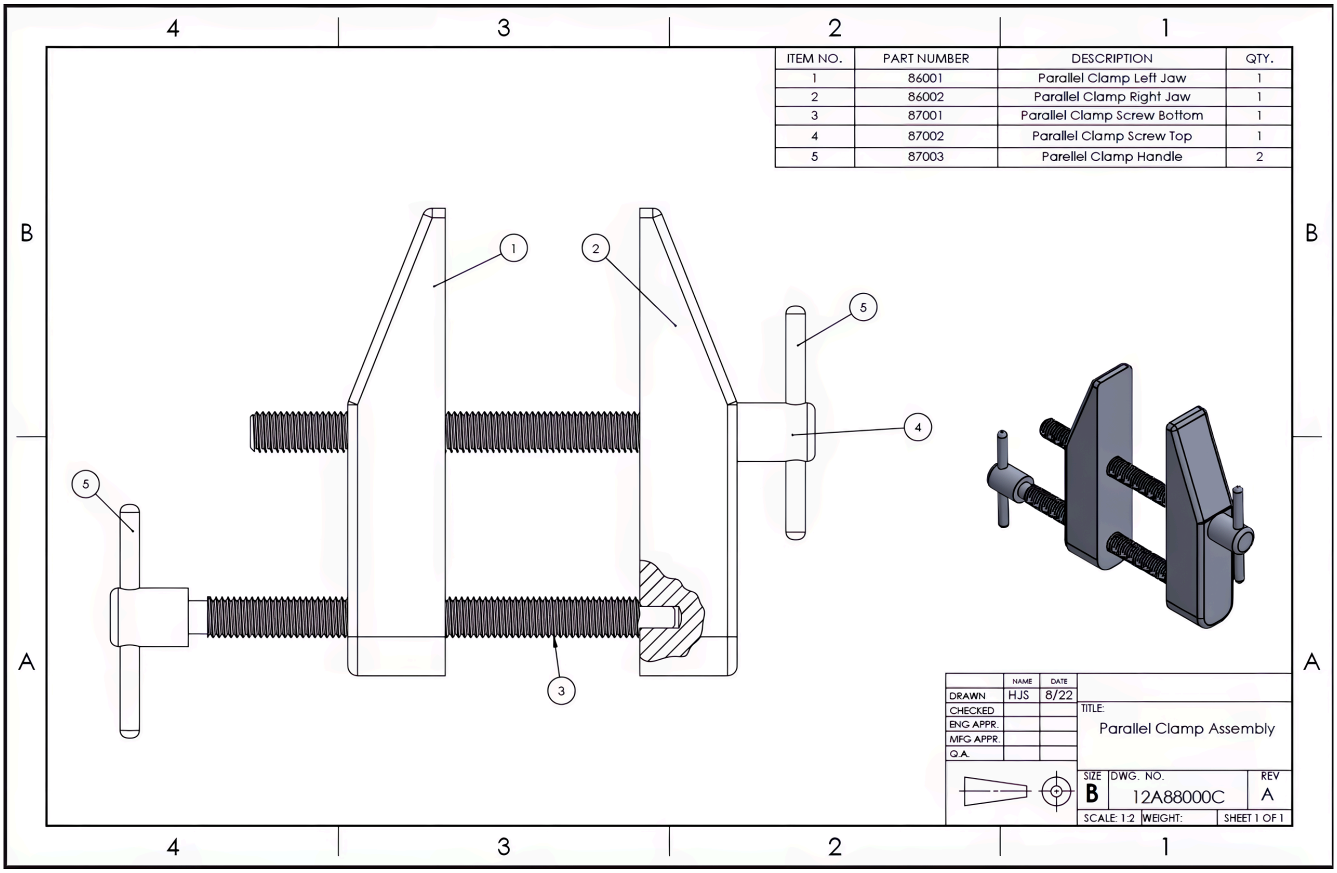
Terms

- Assembly drawing
- Exploded assembly drawing
- Subassembly drawings
- Parts list
- Balloons

The prints explored in the previous chapters have been detailed drawings, which are for one single part. The skills used to understand detailed drawings are also used to understand assembly drawings, which provide information on how the individual components come together to make up an assembly.

13.2 ASSEMBLY DRAWINGS

Assembly drawings depict how various components of a product, die, or other assembly come together to form a complete unit. These drawings include details of each individual component that comprises the assembly, along with their respective part numbers, dimensions, and specific notes pertaining to the assembly. To keep the drawings less congested, hidden lines are typically not included in assembly drawings unless they may aid in clarification in certain instances. Sectional views are commonly used in assembly to show a clearer view of certain features. Figure 13-1 shows an assembly drawing of a parallel clamp.



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	86001	Parallel Clamp Left Jaw	1
2	86002	Parallel Clamp Right Jaw	1
3	87001	Parallel Clamp Screw Bottom	1
4	87002	Parallel Clamp Screw Top	1
5	87003	Parellel Clamp Handle	2

	NAME	DATE
DRAWN	HJS	8/22
CHECKED		
ENG APPR.		
MFG APPR.		
Q.A.		

TITLE: Parallel Clamp Assembly

SIZE B

DWG. NO. 12A88000C

REV A

SCALE: 1:2

WEIGHT:

SHEET 1 OF 1

Figure 13-1. Assembly drawing.

13.3 EXPLODED ASSEMBLY DRAWINGS

An **exploded assembly drawing** illustrates the individual components of an assembly separated along an axis, providing a clear view of how the parts of the assembly fit together. The components are displayed in a way that they appear to be "exploded" away from their assembled position. This separation helps to visualize the relationship between the parts. View Figure 13-2.

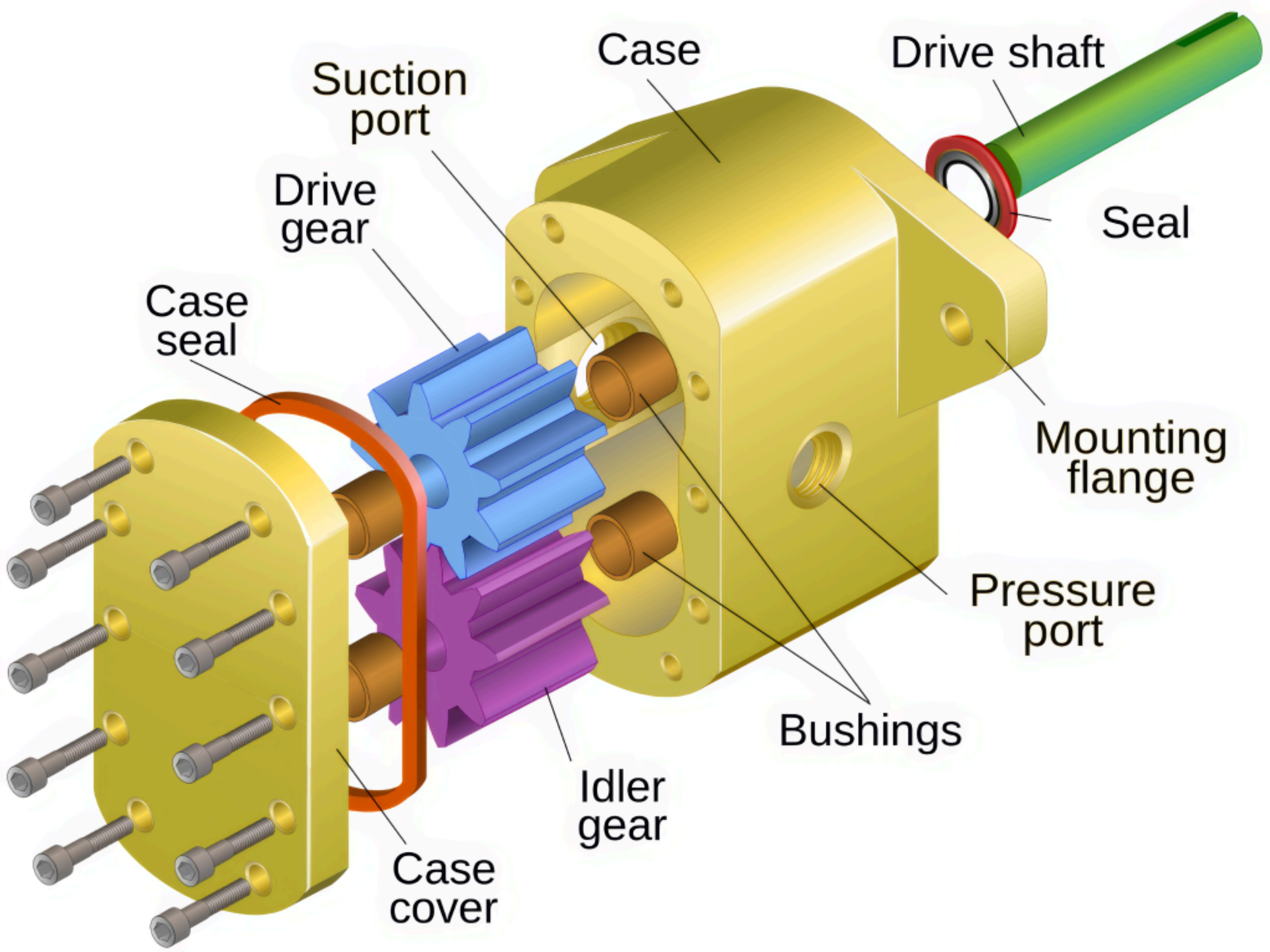


Figure 13-2. An exploded assembly drawing provides a view of the assembly components.

13.4 SUBASSEMBLY DRAWINGS

Subassembly drawings concentrate on a defined set of components within the larger assembly. They will illustrate how the subassembly is pieced together and its role in the complete assembly. These illustrations emphasize a certain segment, enabling a more thorough analysis of that specific area without the complexity of the entire assembly. Although the primary emphasis is on the subassembly, these drawings may also illustrate how individual parts fit together within the larger structure. Assembly drawings of complex machinery such as a CNC machine or an automobile may require thousands of subassembly drawings. The pictorial view in Figure 13-3 is a subassembly for a punch slide from a larger die set.

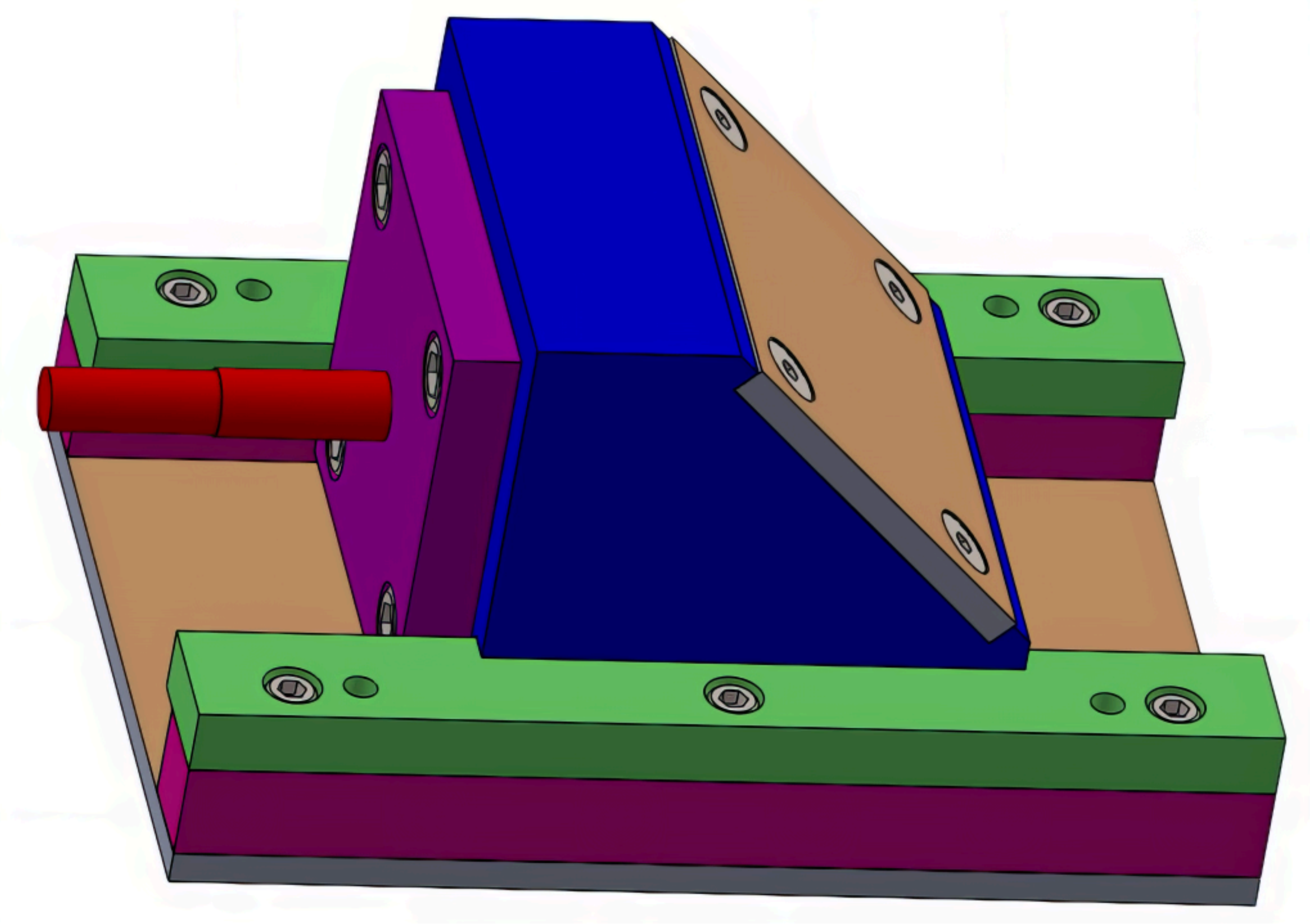
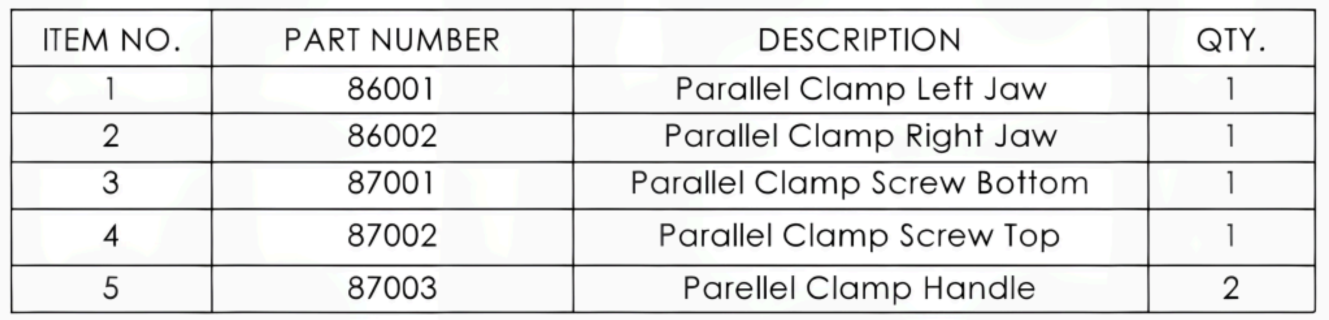


Figure 13-3. Subassembly drawing features a portion of a larger assembly.

13.5 PARTS LIST

An assembly drawing **parts list**, often referred to as a Bill of Materials (BOM) or parts schedule, is a comprehensive list that accompanies an assembly drawing. It details all the components, materials, and parts required to construct the assembly. See an example in Figure 13-4.



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	86001	Parallel Clamp Left Jaw	1
2	86002	Parallel Clamp Right Jaw	1
3	87001	Parallel Clamp Screw Bottom	1
4	87002	Parallel Clamp Screw Top	1
5	87003	Parellel Clamp Handle	2

Figure 13-4. An assembly parts list identifies the components of an assembly.

The parts list will include the part numbers, part names or descriptions, and the number of required parts. It may also include part drawing numbers, and the part's revision information will include supplier information for purchased components, such as fasteners. **Balloons** often identify the parts of the assembly with leader lines and numbers that match the parts list.

LEARNING ACTIVITIES

Exercise 13.5-1

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=501#h5p-39

Exercise 13.5-2

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=501#h5p-40

Exercise 13.5-3

An interactive H5P element has been excluded from this version of the text. You can view it online here:
https://wtcs.pressbooks.pub/blueprintreading/?p=501#h5p-41

References:

Barsamian, M. A., & Gizelbach, R. (2022). *Machine trades print reading.* Goodheart-Willcox.

Images:

All images by Mark Lorier and licensed under CC BY-NC 4.0 except where otherwise noted.

Figure 13-2 “Gear_pump_exploded” by Д.Ильин, Duk, cmglee is licensed under CC BY-SA 4.0

14. Print Specifications

14.1 INTRODUCTION

Learning Objectives

- Select material for layout
- Apply layout dye per procedural guidelines
- Select baselines for layout (select appropriate surfaces)
- Calculate dimensions
- Scribe lines for appropriate dimensions and detail
- Compare layout to print specifications

Terms

- Layout fluid
- Scriber
- Surface plate
- Height gauge
- Angle plate
- Surface gauge
- Protractor
- Sine bar
- Divider

This chapter will apply the skills from previous chapters to perform an accurate layout of feature locations obtained from the print specifications. A layout fluid will be applied to the material selected, with feature locations that are properly located to serve as a guide for machining operations.

14.2 LAYOUT FLUID

Layout fluid, also called layout dye, is applied to a part surface to provide a contrast for the layout lines. The fluid, typically a dark red or dark blue, is applied in a thin, even coat using an in-cap applicator or aerosol spray. The fluid will dry quickly once applied, and it should be completely dried before handling. Figure 14-1 shows an example of layout fluid.

Figure 14-1. Layout fluid applied to a part surface.

14.3 SCRIBING LINES

Lines are often created using scribers, along with a straight edge such as a combination square. A **scriber** is a sharp, fine-pointed tool used to create an accurate straight line on the laid-out surface. The scriber is best used by pulling or dragging along the straight edge. Pushing the scriber can cause the sharp point to catch on the material or straight edge, preventing a straight, even line. See a scriber in Figure 14-2.

Figure 14-2. A scriber is used to create an accurate line on a part of the surface.

A surface plate and height gauge, or surface gauge, may also be used to create straight scribed lines. A **surface plate** is a solid, flat plate used as the main horizontal reference plane for precision inspection and for layout to create straight parallel scribed lines. A **height gauge** is a precision measuring tool used to determine the height of an object. Using a height gauge on a surface plate has the advantage of setting the scribing edge to an accurate height without having to use another measuring tool. In Figure 14-3, a straight fine line is scribed by dragging the part along the scribe of a height gauge using a surface plate as a reference surface.

Figure 14-3. A scribe line was created using a surface plate and height gauge.

An **angle plate** is an L-shaped component with a precision right angle used for clamping work in a vertical position for inspection, machining, or layout. In Figure 14-4, an angle plate is used to secure the part in an accurate vertical position for layout with a height gauge.

Figure 14-4. An angle plate is used to secure the part for a vertical layout.

A **surface gauge** is a versatile tool made of a solid base and adjustable arm with sharp ends often used for scribing. The surface gauge arm can be accurately adjusted to a height using a secondary measuring tool such as a steel rule shown in Figure 14-5.

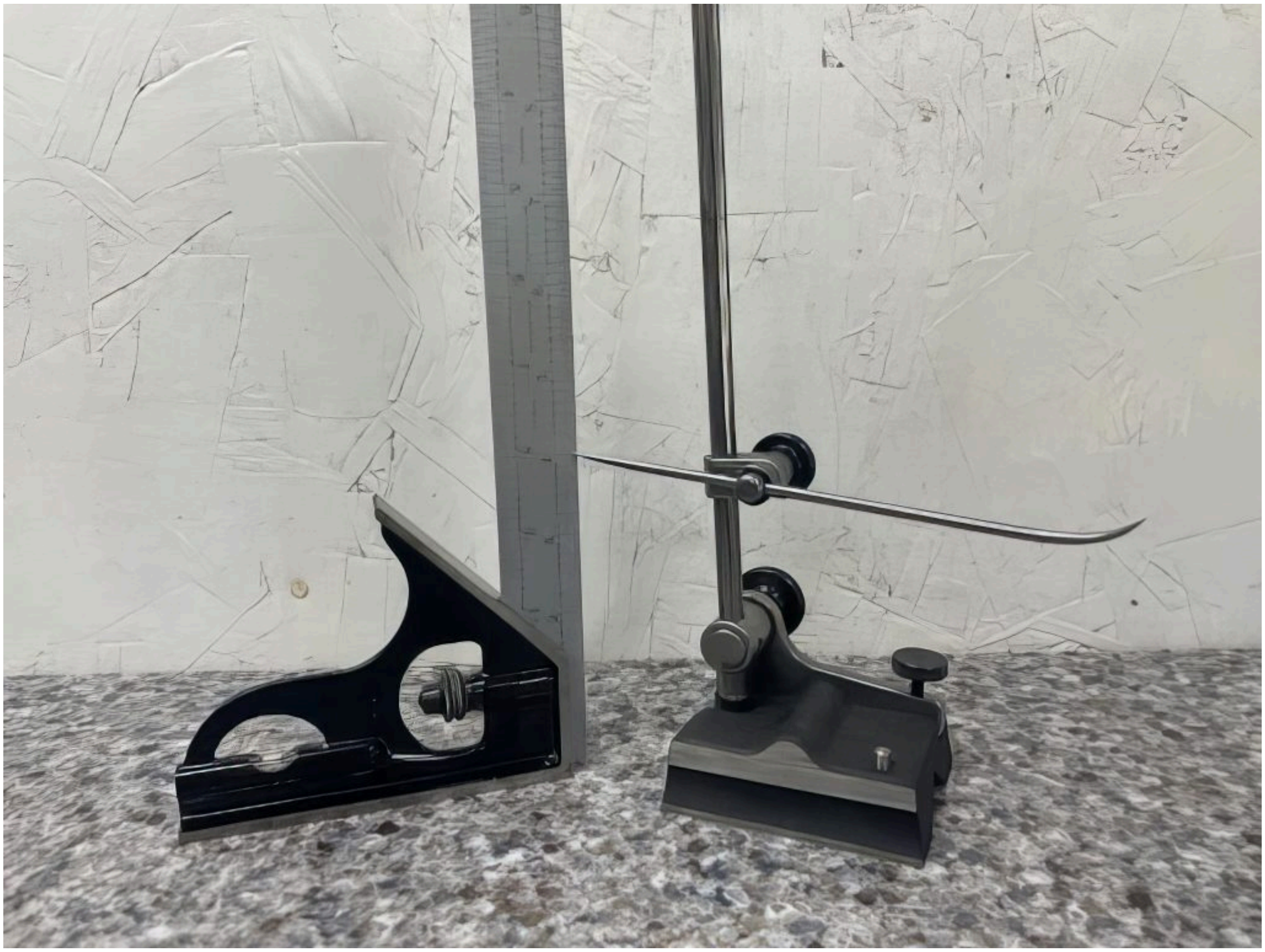


Figure 14-5. The surface gauge is set to a height measurement for scribing.

14.4 SCRIBING ANGLES

Angles can be scribed using a combination square protractor or a plain protractor, also called a plate **protractor**. A protractor can be used to both measure angles and locked at an angular position to the nearest degree to be used for angular scribing. In Figure 14-6, the protractor is locked to a specified angle for scribing. To accurately scribe the angle, the straight edge of the protractor's head is moved to the reference edge of the workpiece, and then the scriber is dragged along the blade edge.

Figure 14-6. A protractor is used to scribe an angular line on a part.

A sine bar may also be used for angular measurements. A **sine bar** is a tool used along with an accurate stack of blocks known as gauge blocks to create a precise angle. A sine bar creates the hypotenuse of a right triangle to set an accurate angle. To calculate the height of the stacked blocks needed, multiply the sine of the angle by the length of the sine bar. In Figure 14-7, a sine bar is set to an angle on a surface plate to scribe an accurate angular line with a height gauge.

Figure 14-7. The sine bar is used to scribe an angular line accurately.

14.5 SCRIBING ARCS

A **divider** is a scribe tool with two adjustable points for scribe circles, arcs, and radii. A divider is set to the radius of the desired arc, often using a ruler with one point placed on the 1-inch line and adjusting to the desired radius. See Figure 14-8.

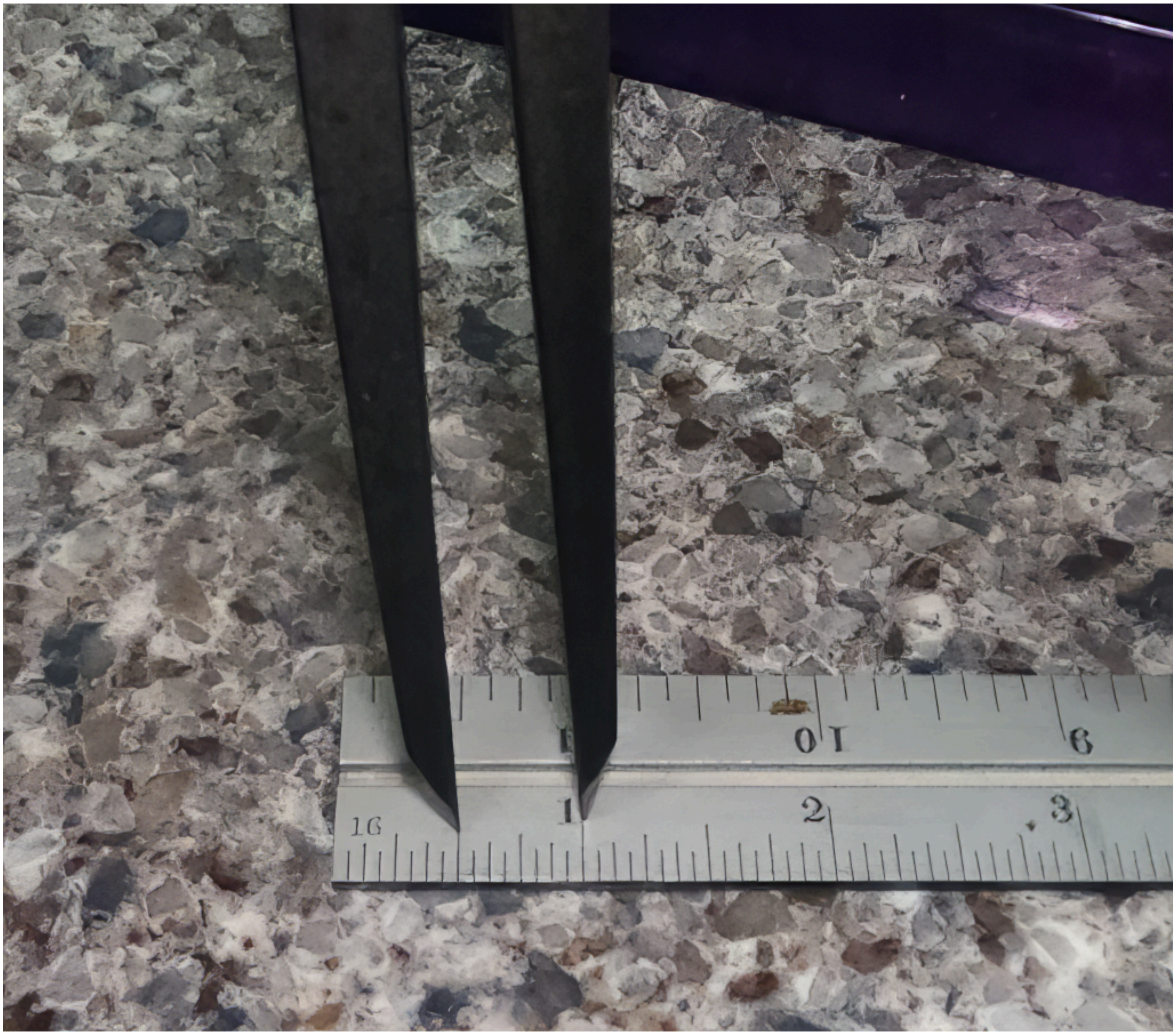


Figure 14-8. A ruler is used to set a divider to a desired measurement.

Some calipers will provide indentations on the back side to allow for a more accurate divider setting. View Figure 14-9.

Figure 14-9. A divider is set using indents on a caliper.

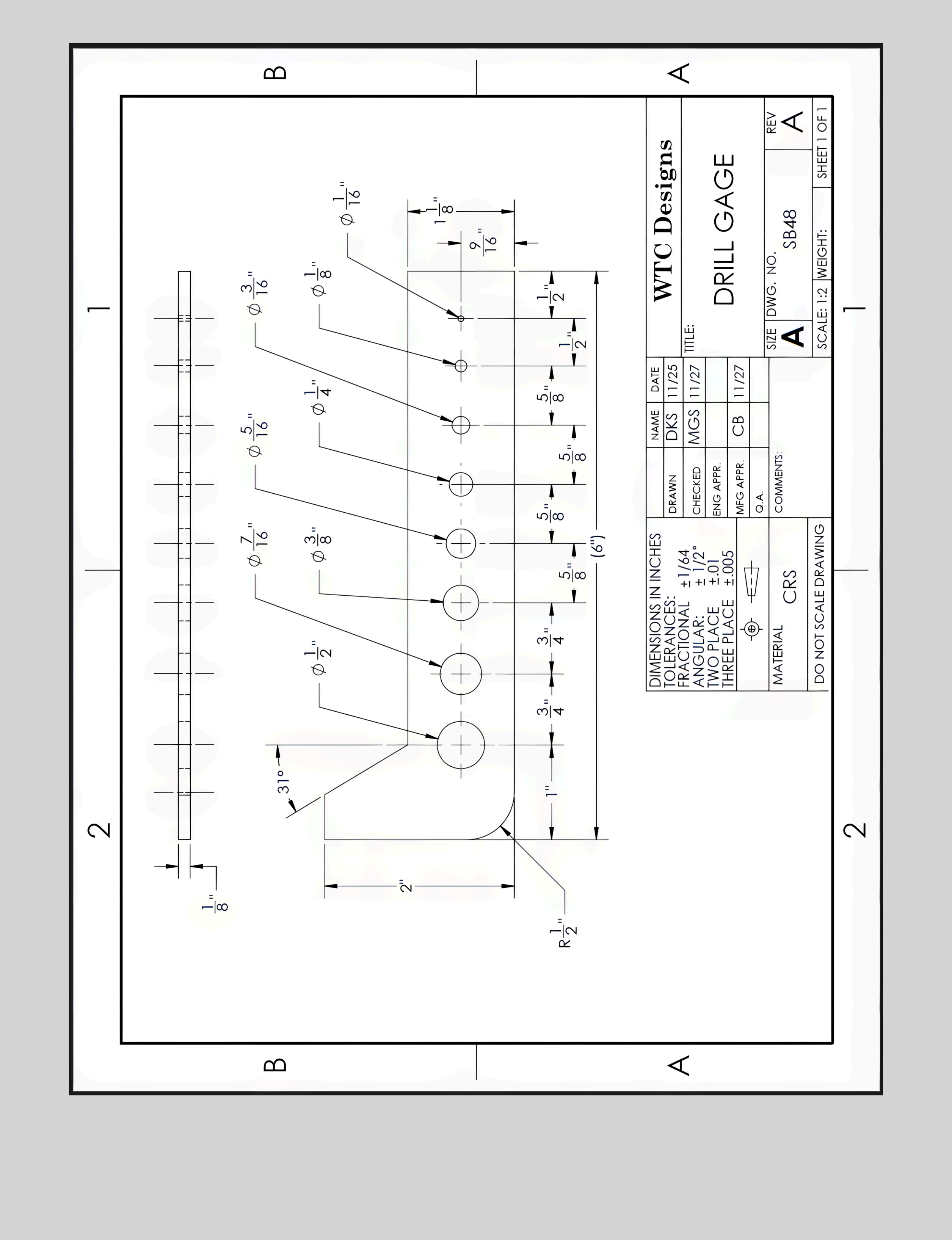
To scribe, one point of the divider is held in the arc's center point, and the other point is pulled to create the scribed arc, as illustrated in Figure 14-10.

Figure 14-10. A spring divider is used to create a scribed arc.

LEARNING ACTIVITIES

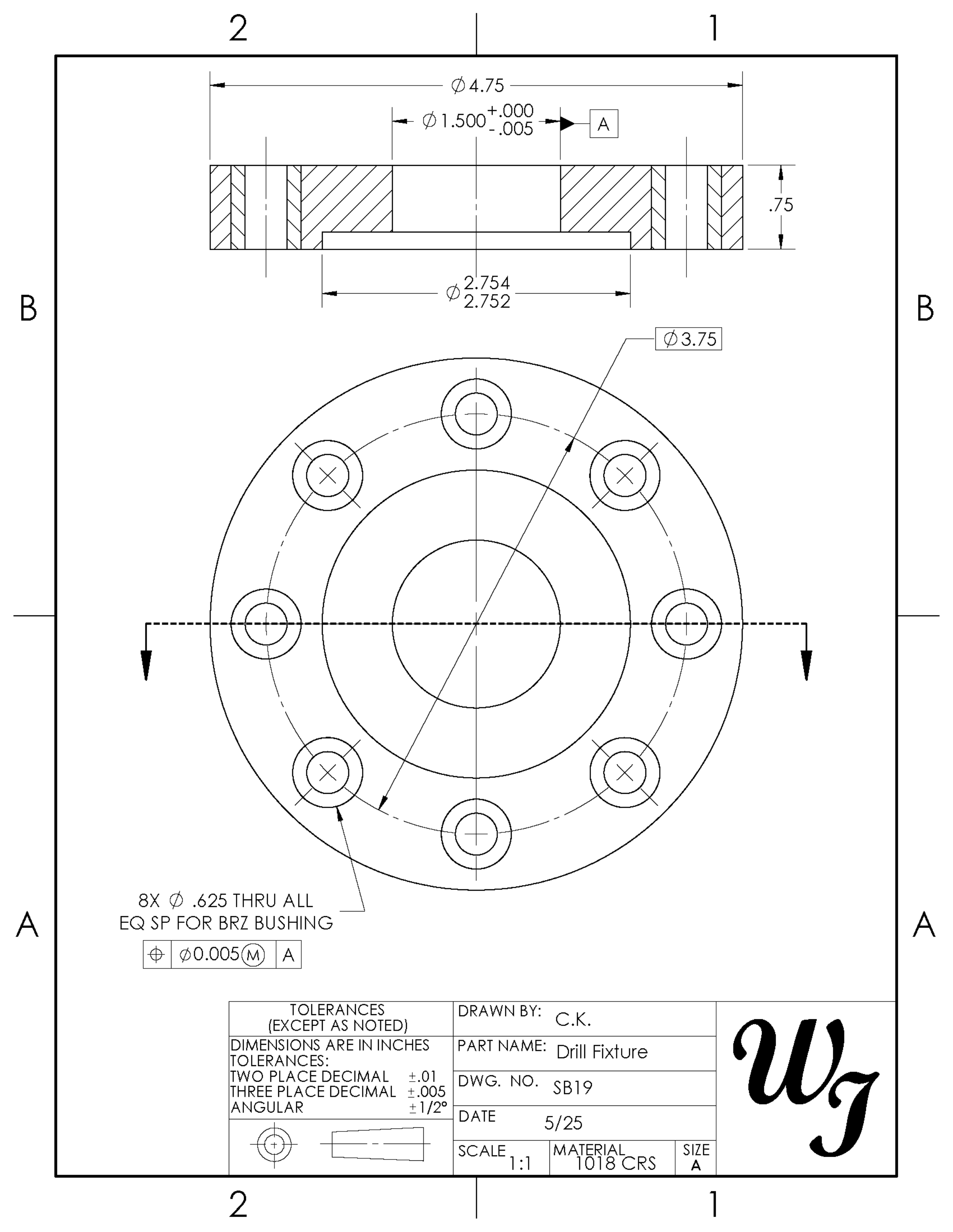
Exercise 14.5-1

Use 1/8" x 2" x 6" material to layout and scribe this workpiece according to the print.


WTC Designs
TITLE:
DRILL GAGE
SIZE A
DWG. NO. SB48
REV A
SCALE: 1:2
WEIGHT:
SHEET 1 OF 1
NAME
DATE
DRAWN DKS 11/25
CHECKED MGS 11/27
ENG APPR.
MFG APPR. CB 11/27
Q.A.
COMMENTS:
DIMENSIONS IN INCHES
TOLERANCES:
FRACTIONAL ±1/64
ANGULAR: ± 1/2°
TWO PLACE ±.01
THREE PLACE ±.005
MATERIAL CRS
DO NOT SCALE DRAWING
ϕ 1/16"
ϕ 1/8"
ϕ 3/16"
ϕ 1/4"
ϕ 5/16"
ϕ 3/8"
ϕ 7/16"
ϕ 1/2"
1 1/8"
9/16"
1/2"
1/2"
5/8"
5/8"
5/8"
5/8"
3/4"
3/4"
1"
(6")
31°
2"
R 1/2"
1/8"
A
B
1
2

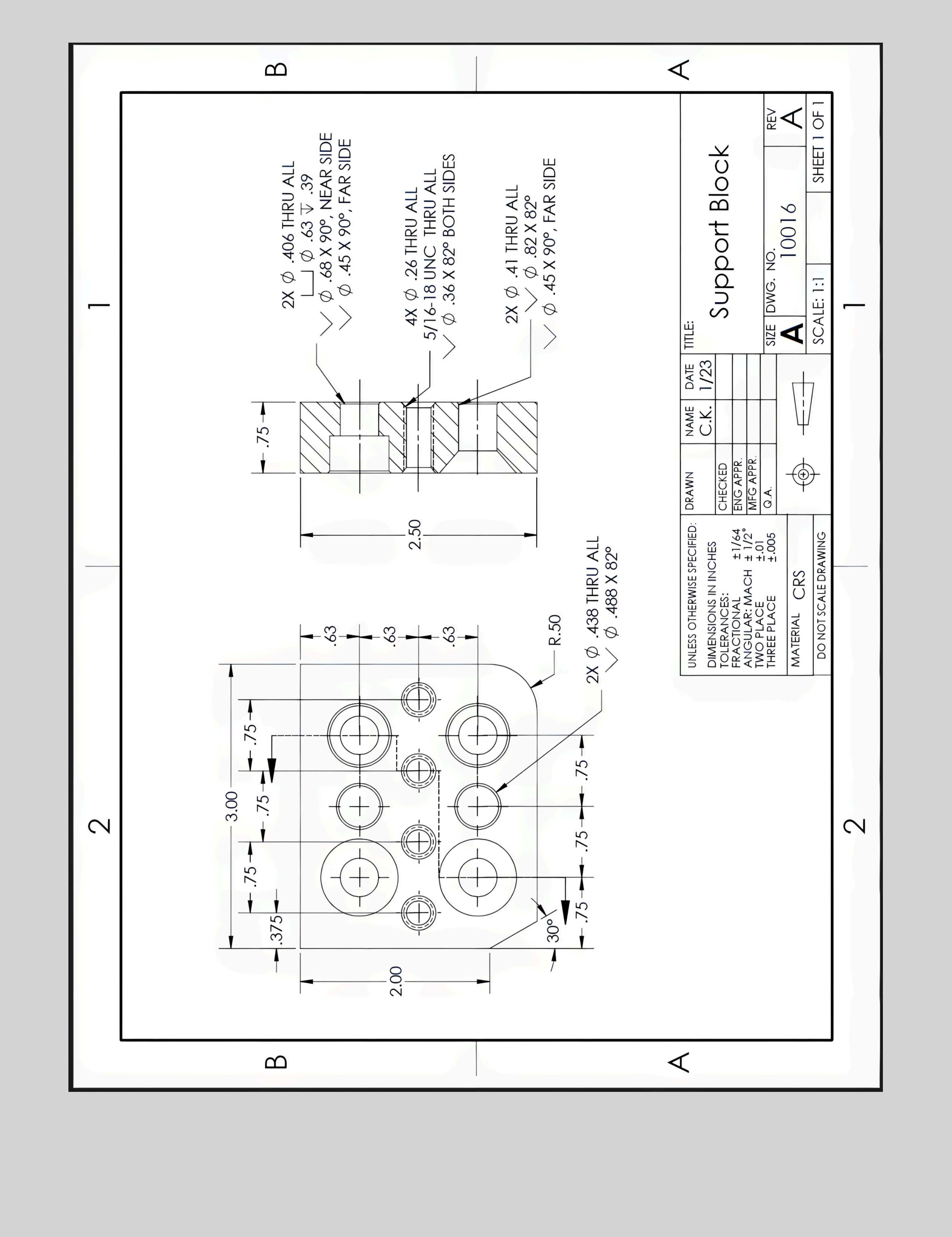
Exercise 14.5-2

Use .75" x 4.75" diameter material to scribe the circles for the 1.500", 2.754"/2.752", and 3.75"-diameter circles. Consult the *Machinery’s Handbook* for the “Chord Length for Given Number of Divisions” for the diameter of the bolt circle and the number of holes required to find the distance to set the divider to lay out the bolt hole locations, as shown in **Figure 14-10**.


2
1
B
A
∅4.75
∅1.500 +.000 -.005
A
.75
∅ 2.754 2.752
∅3.75
8X ∅ .625 THRU ALL
EQ SP FOR BRZ BUSHING
⌖ ∅0.005Ⓜ A
TOLERANCES (EXCEPT AS NOTED)
DIMENSIONS ARE IN INCHES
TOLERANCES:
TWO PLACE DECIMAL ±.01
THREE PLACE DECIMAL ±.005
ANGULAR ±1/2°
DRAWN BY: C.K.
PART NAME: Drill Fixture
DWG. NO. SB19
DATE 5/25
SCALE 1:1
MATERIAL 1018 CRS
SIZE A
WJ

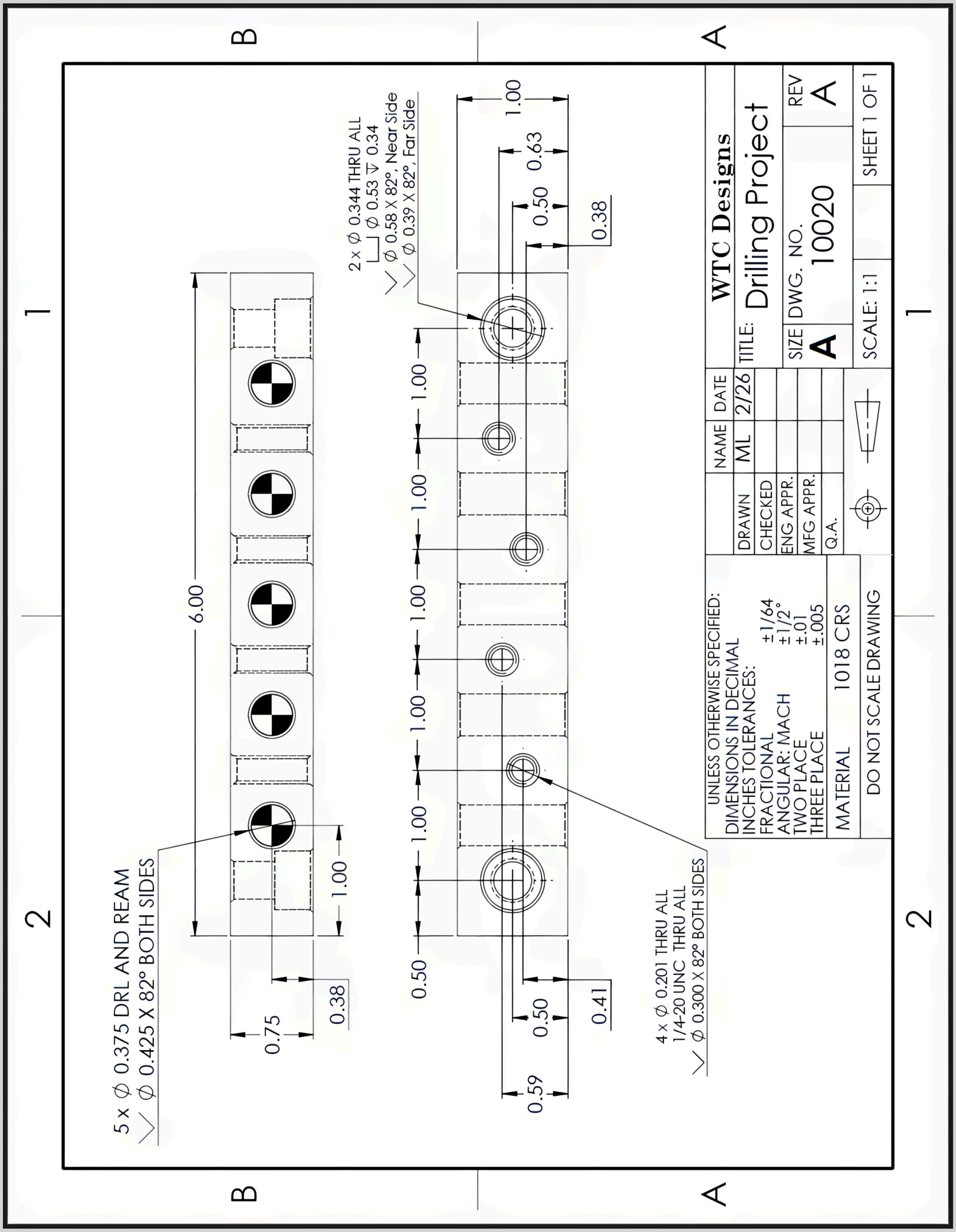
Exercise 14.5-3

Using .75" x 2.50" x 3.00" material, layout and scribe this workpiece according to the print.


2X ⌀ .406 THRU ALL
⌴ ⌀ .63 ↧ .39
⌵ ⌀ .68 X 90°, NEAR SIDE
⌵ ⌀ .45 X 90°, FAR SIDE
4X ⌀ .26 THRU ALL
5/16-18 UNC THRU ALL
⌵ ⌀ .36 X 82° BOTH SIDES
2X ⌀ .41 THRU ALL
⌵ ⌀ .82 X 82°
⌵ ⌀ .45 X 90°, FAR SIDE
.75
2.50
2X ⌀ .438 THRU ALL
⌵ ⌀ .488 X 82°
R.50
.63
.63
.63
3.00
.75
.75
.75
.375
.75
.75
.75
30°
2.00
A
B
1
2
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS IN INCHES
TOLERANCES:
FRACTIONAL ±1/64
ANGULAR: MACH ± 1/2°
TWO PLACE ±.01
THREE PLACE ±.005
MATERIAL CRS
DO NOT SCALE DRAWING
DRAWN
CHECKED
ENG APPR.
MFG APPR.
Q.A.
NAME C.K.
DATE 1/23
TITLE:
Support Block
SIZE A
DWG. NO. 10016
REV A
SCALE: 1:1
SHEET 1 OF 1

Exercise 14.5-4

Using .75" x 1.00" x 6.00" material, layout and scribe this workpiece according to the print.


5 x Ø 0.375 DRL AND REAM
⌵ Ø 0.425 X 82° BOTH SIDES
6.00
0.75
0.38
1.00
2 x Ø 0.344 THRU ALL
⌴ Ø 0.53 ↧ 0.34
⌵ Ø 0.58 X 82°, Near Side
⌵ Ø 0.39 X 82°, Far Side
1.00
0.63
0.50
0.38
0.50
1.00
1.00
1.00
1.00
1.00
0.59
0.50
0.41
4 x Ø 0.201 THRU ALL
1/4-20 UNC THRU ALL
⌵ Ø 0.300 X 82° BOTH SIDES
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS IN DECIMAL
INCHES TOLERANCES:
FRACTIONAL ±1/64
ANGULAR: MACH ±1/2°
TWO PLACE ±.01
THREE PLACE ±.005
MATERIAL 1018 CRS
DO NOT SCALE DRAWING
NAME
DATE
DRAWN ML 2/26
CHECKED
ENG APPR.
MFG APPR.
Q.A.
WTC Designs
TITLE: Drilling Project
SIZE A
DWG. NO. 10020
REV A
SCALE: 1:1
SHEET 1 OF 1
A
B
1
2

References:

Hoffman, P. (2020). Section 3: Unit 2. Layout. In *Precision machine engineering* (3rd ed.). Cengage Learning.

Images:

Printable Learning Activities by Chapter

CHAPTER 1: DRAWING V. PRINTS

Exercise 1

1. What two categories can the manufacturing process be condensed into?
   - a. Design phase and production phase
   - b. Sketching phase and assembly phase
   - c. Planning phase and testing phase
   - d. Prototype phase and final phase
2. What is the purpose of the design phase?
   - a. To generate essential documentation for product fabrication
   - b. To transport components to a specified location for assembly
   - c. To create three-dimensional models of the design
   - d. To analyze and simulate the performance of the design
3. What is the advantage of using a CAD system in the design process?
   - a. Speed and efficiency in creating designs

- b. Easy modification of designs
- c. Accuracy in creating designs
- d. All of the above

4. What can a 3D model be used for in the manufacturing process?

   - a. Creating detailed 2D drawings
   - b. Transferring the design directly to production
   - c. Analyzing and simulating the performance of the part or assembly
   - d. All of the above

5. What is the purpose of detailed drawings in the design process?

   - a. To provide comprehensive information necessary to manufacture each part
   - b. To outline the shape and size of the part
   - c. To convey essential specifications such as material types
   - d. All of the above

6. What is the origin of the term “blueprint”?

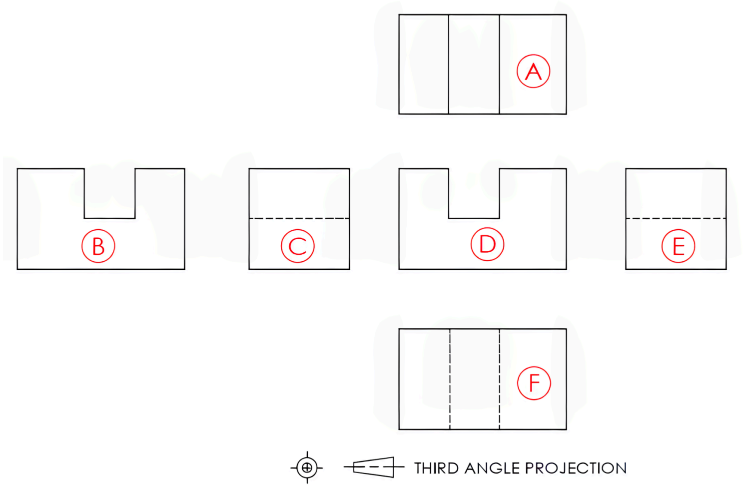
   - a. It refers to prints with a blue background and white lines
   - b. It originated from the process of sketching designs on tracing paper
   - c. It refers to the chemical reaction that turns exposed areas white

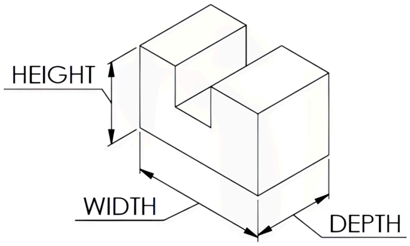
- ◦ d. All of the above

CHAPTER 2: VIEWS OF AN OBJECT

View Identification Quiz

Directions: Circle the letter corresponding to the Front, Top, Right, Left, Back, or Bottom, and circle two of the three basic dimensions of Height, Width, or Depth that may be from that view.





Circle one **Circle two**

Front View: A B C D E F – Height Width Depth

Top View: A B C D E F – Height Width Depth

Right View: A B C D E F – Height Width Depth

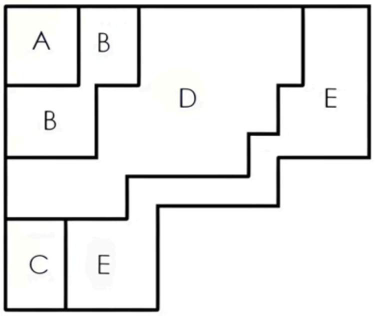
Bottom View: A B C D E F – Height Width Depth

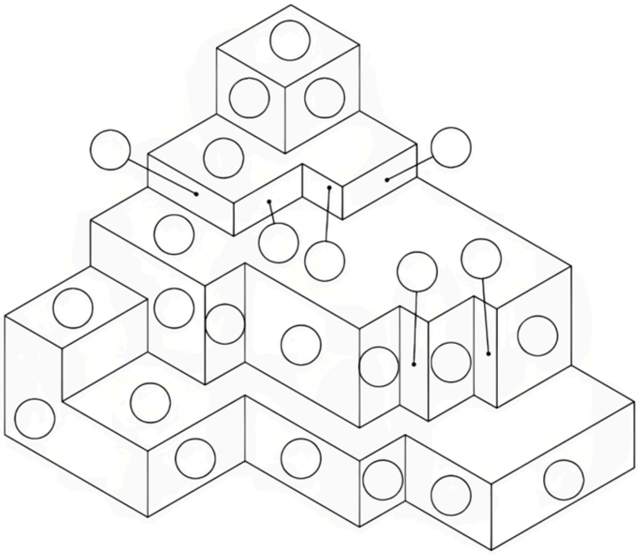
Left View: A B C D E F – Height Width Depth

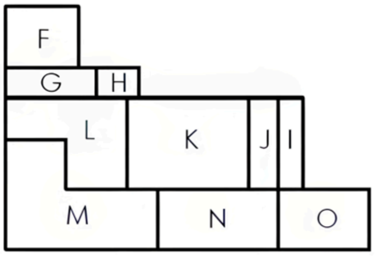
Back View: A B C D E F – Height Width Depth

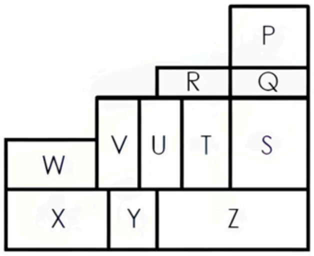
Surface Identification Quiz 1

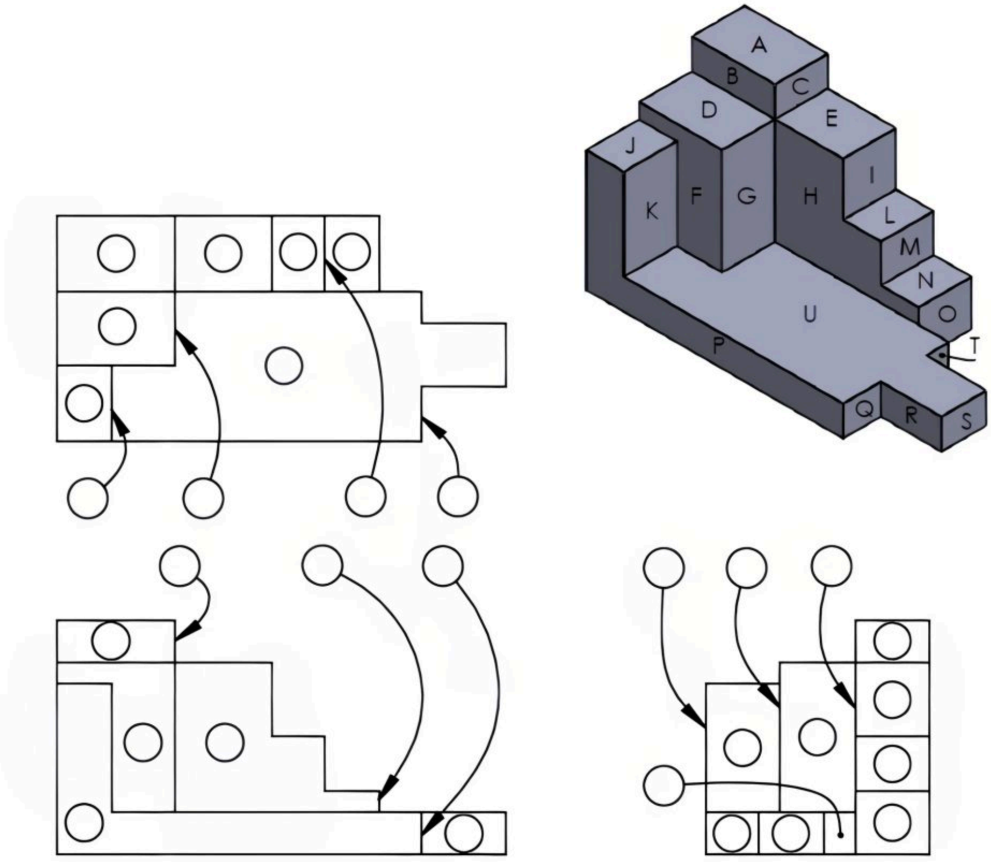
Directions: Enter the letters from the pictorial drawing into the correct balloons on the orthographic views.





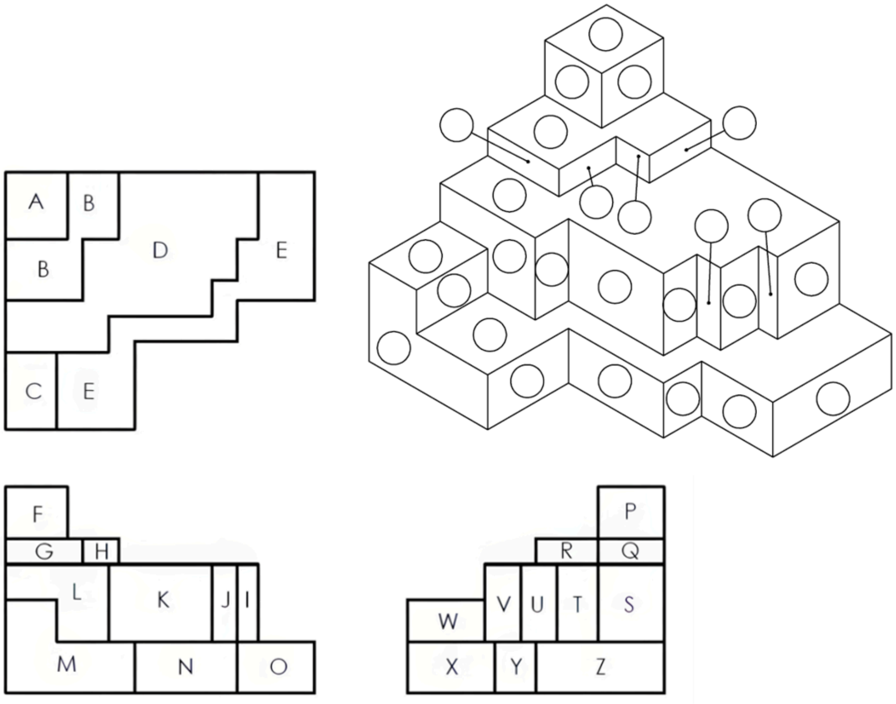






Surface Identification Quiz 2

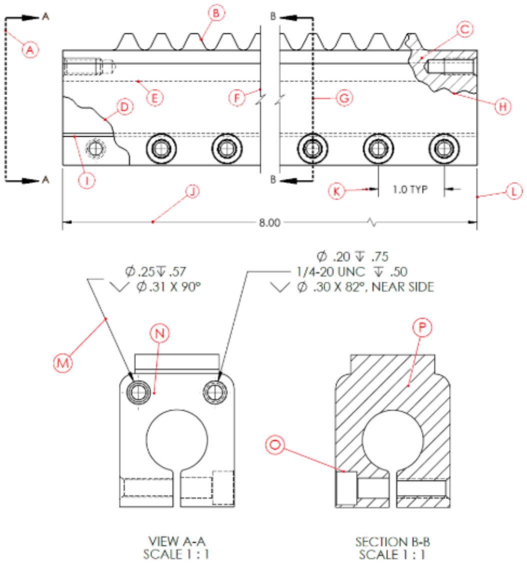
Directions: Enter the letters from the pictorial drawing into the correct balloons on the orthographic views.



CHAPTER 3: TYPES OF LINES

Line Quiz 1

Directions: Using the images below, identify the line type being used below. NOTE: Some lines will be identified more than once.



1. Centerline
2. Chain line
3. Cutting-plane line
4. Dimension line
5. Extension line
6. Hidden line
7. Leader
8. Long break line
9. Phantom line
10. Short break line
11. Section line
12. Symmetry line
13. Viewing-plane line
14. Visible line

A. ______________________ B. ______________________
C. ______________________

D. ______________________ E. ______________________
F. ______________________

G. ______________________ H. ______________________
I. ______________________

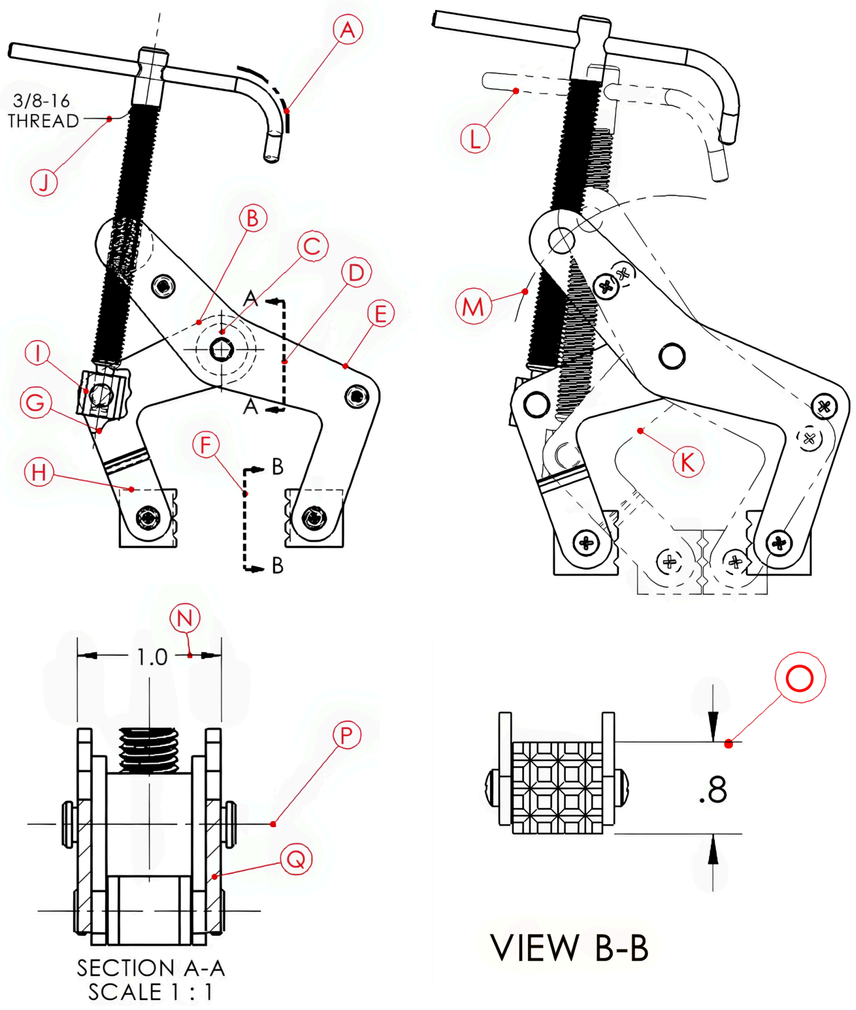
J. ______________________ K. ______________________
L. ______________________

M. ______________________ N. ______________________
O. ______________________

P. ______________________

Line Quiz 2

Directions: Using the drawing, identify the line type being used below. NOTE: Some lines will be identified more than once.



Centerline

Chain line

Cutting-plane line

Dimension line

Extension line

Hidden line

Leader

Long break line

Phantom line

Short break line

Section line

Symmetry line

Viewing-plane line

Visible line

A. ______________________ B. ______________________

C. ____________________

D. ______________________ E. ______________________

F. ____________________

G. ______________________ H. ______________________

I. ____________________

J. ______________________ K. ______________________

L. ____________________

M. ______________________ N. ______________________

O. ____________________

P. ______________________ Q. ______________________

Line Quiz 3

Directions: Enter the name of the line that best matches the descriptions.

1. Represents outside edges and intersections.

____

2. May be used in place of repeating features or to show an alternate position. ______________________________
3. Line that represents an imaginary cut to provide an alternate view. ______________________________
4. Points to a feature to provide information or a dimension ending with an arrowhead.
   ____
5. Indicates where an alternate view is taken from.
   ____
6. Represents the solid material that has been cut through. ______________________________
7. Indicates a surface or edge not visible in that view.
   ____
8. Ends with arrowheads reaching the distance of the feature to indicate the size.
   ____
9. Lines used in pairs that indicate a part has been shortened to conserve space on the print.
   ____
10. Shows the positioning of holes and the middle of symmetrical parts. ______________________________
11. Extends between the ends of the feature being measured to the dimension of the feature.
    ____
12. Indicates where a portion of the part has been broken away to show interior details more clearly or to show where portions of the part have been removed to conserve space. ______________________________
13. Highlights a portion of the part requiring special treatment or attention. ______________________________

Word Bank

Centerline
Chain line
Cutting-plane line

Dimension line
Extension line
Hidden line

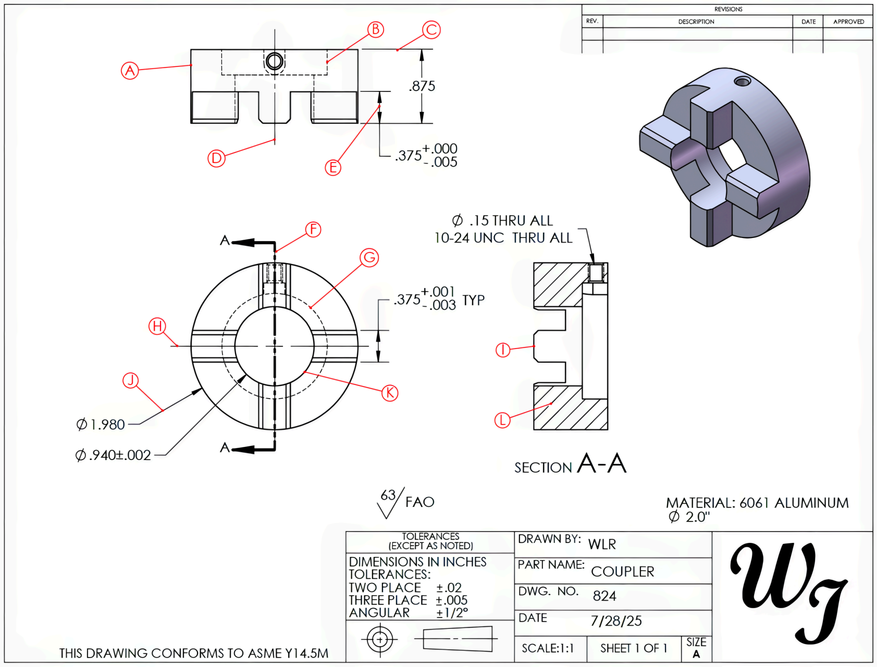
Leader
Long break line
Phantom line

Section line
Short break line
Viewing-plane line

Visible line

Line Quiz 4

Directions: Using the drawing below, identify the line type being used below. NOTE: Some lines may be identified more than once.



Centerline

Chain line

Cutting-plane line

Dimension line

Extension line

Hidden line

Leader

Long break line

Phantom line

Short break line

Section line

Symmetry line

Viewing-plane line

Visible line

A. ______________________ B. ______________________
C. ____________________

D. ______________________ E. ______________________
F. ____________________

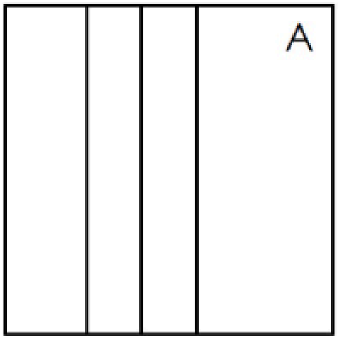
G. ______________________ H. ______________________
I. _____________________

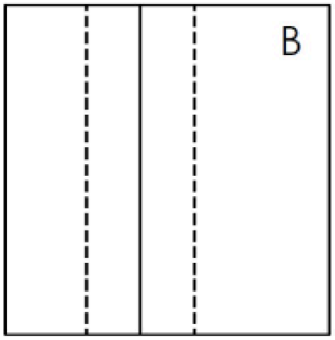
J. ______________________ K. ______________________
L. ____________________

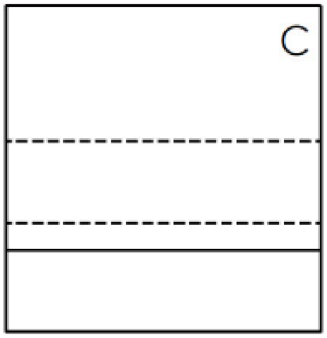
CHAPTER 4: OBJECTS IN DIFFERENT VIEWS

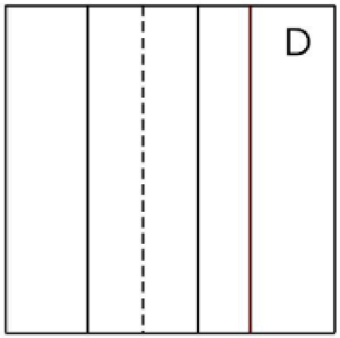
View Identification Exercise 1

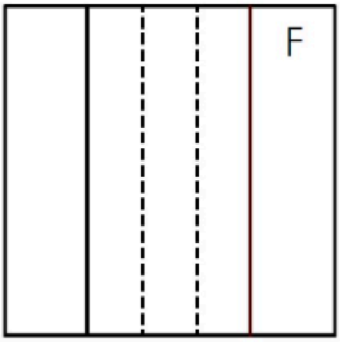
Enter the matching letter of the correct top and right-side views for each isometric view below.

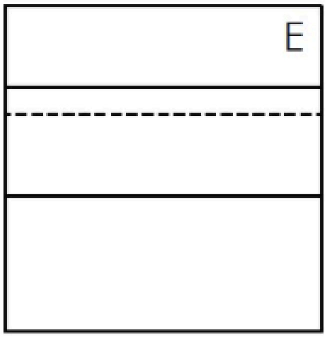

A


B

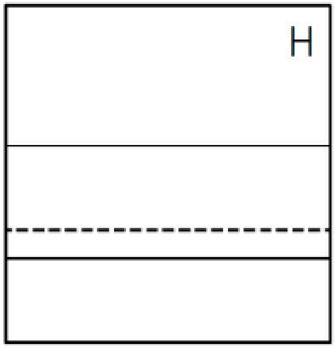

C

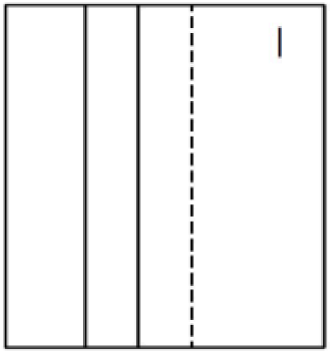

D

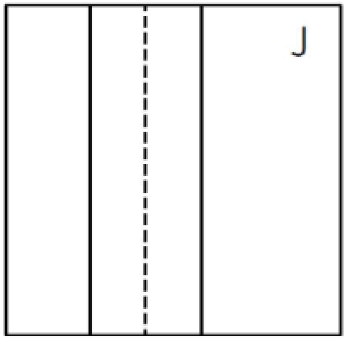

F

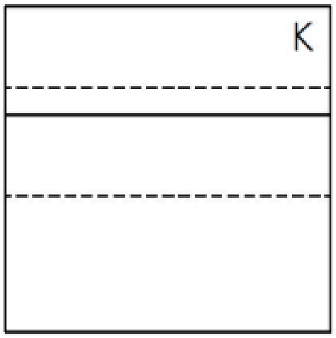

E


G


H


I

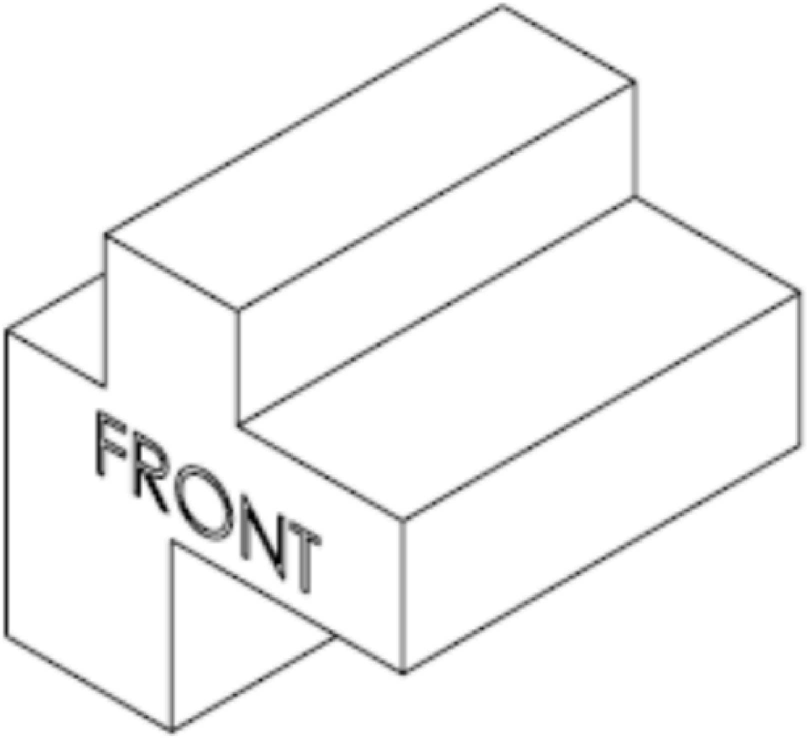

J


K


L

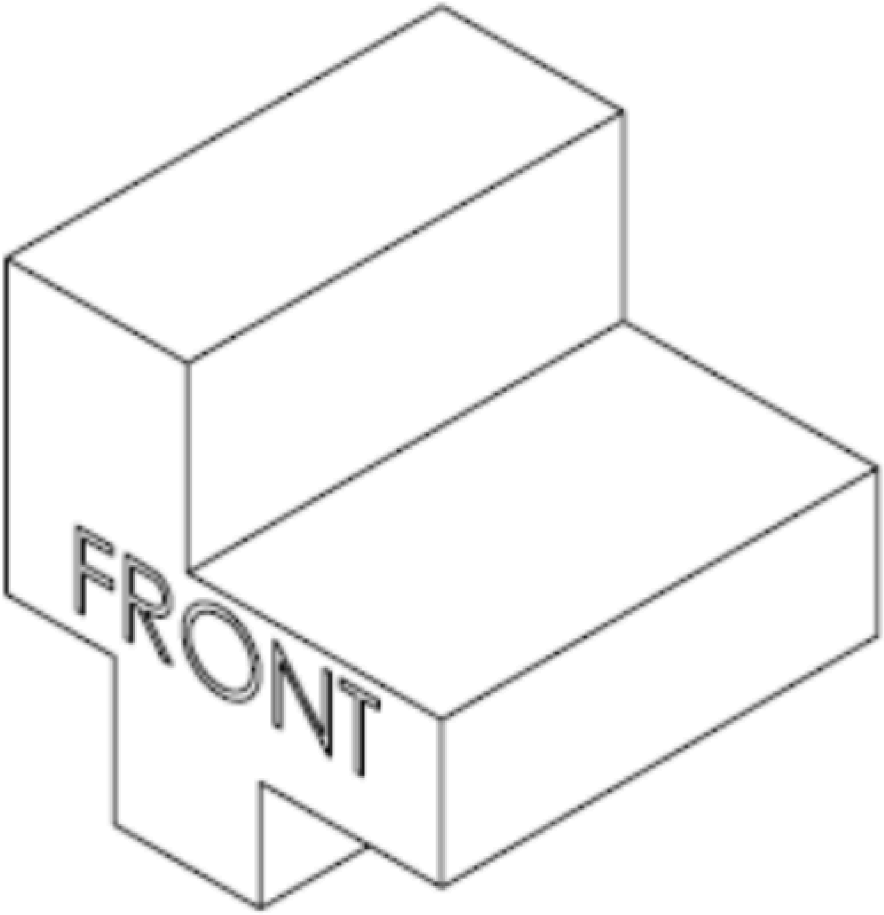
1. Top View: ________________

Right View: ______________



2. Top View: ___________

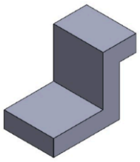
Right View: ___________

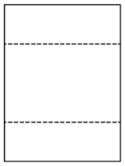


View Identification Exercise 2

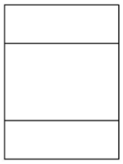
Select the correct orthographic views for each of the isometric views shown.

1. Select the letter of the correct **Front** view

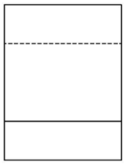




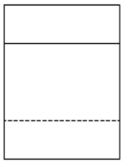
A



B

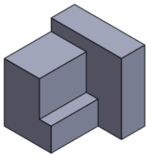


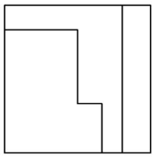
C



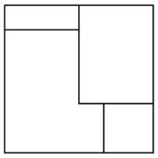
D

2. Select the letter of the correct **Front** view

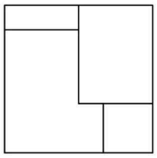




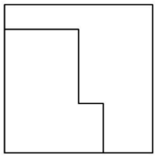
A



B

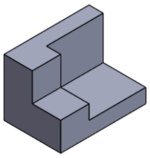


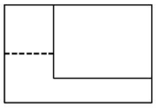
C



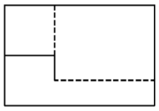
D

3. Select the letter for the correct **Right** view

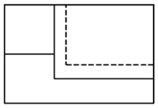




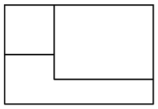
A



B

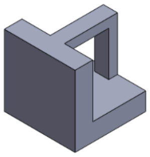


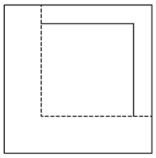
C



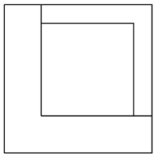
D

4. Select the letter for the correct **Right** view

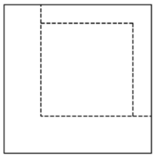




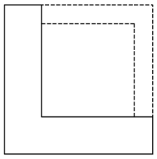
A



B

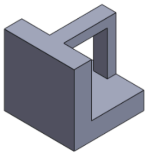


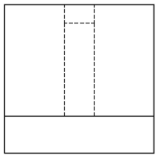
C



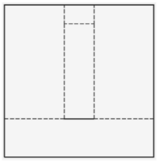
D

5. Select the letter for the correct **Front** view

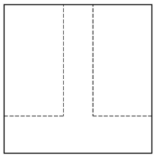




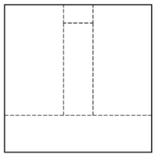
A



B

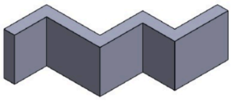


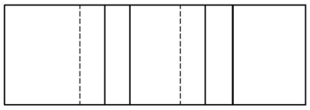
C



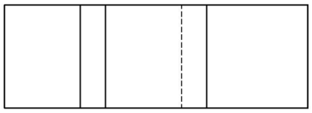
D

6. Select the letter for the correct **Right** view

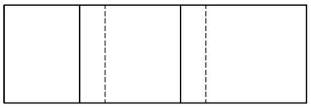




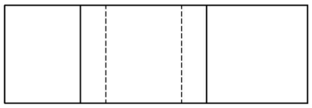
A



B

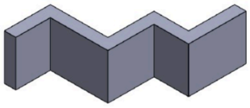


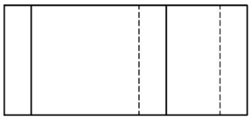
C



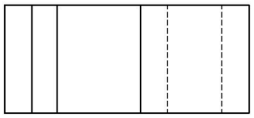
D

7. Select the letter for the correct **Front** view

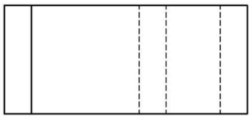




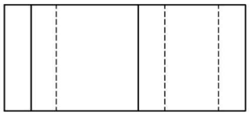
A



B

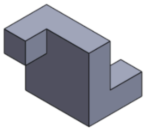


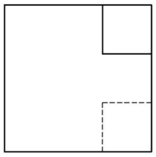
C



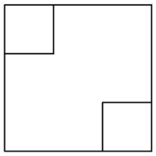
D

8. Select the letter for the correct **Front** view

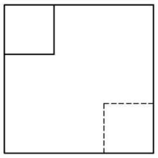




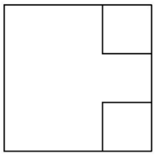
A



B

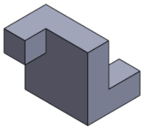


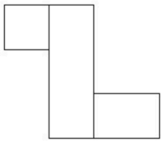
C



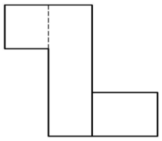
D

9. Select the letter for the correct **Right** view

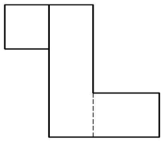




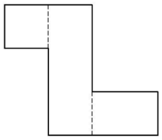
A



B

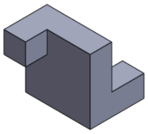


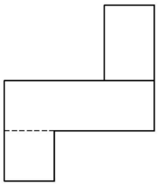
C



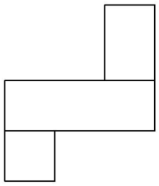
D

10. Select the letter for the correct **Top** view

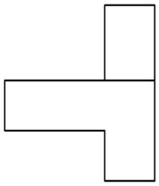




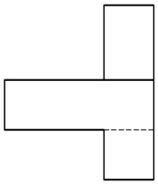
A



B

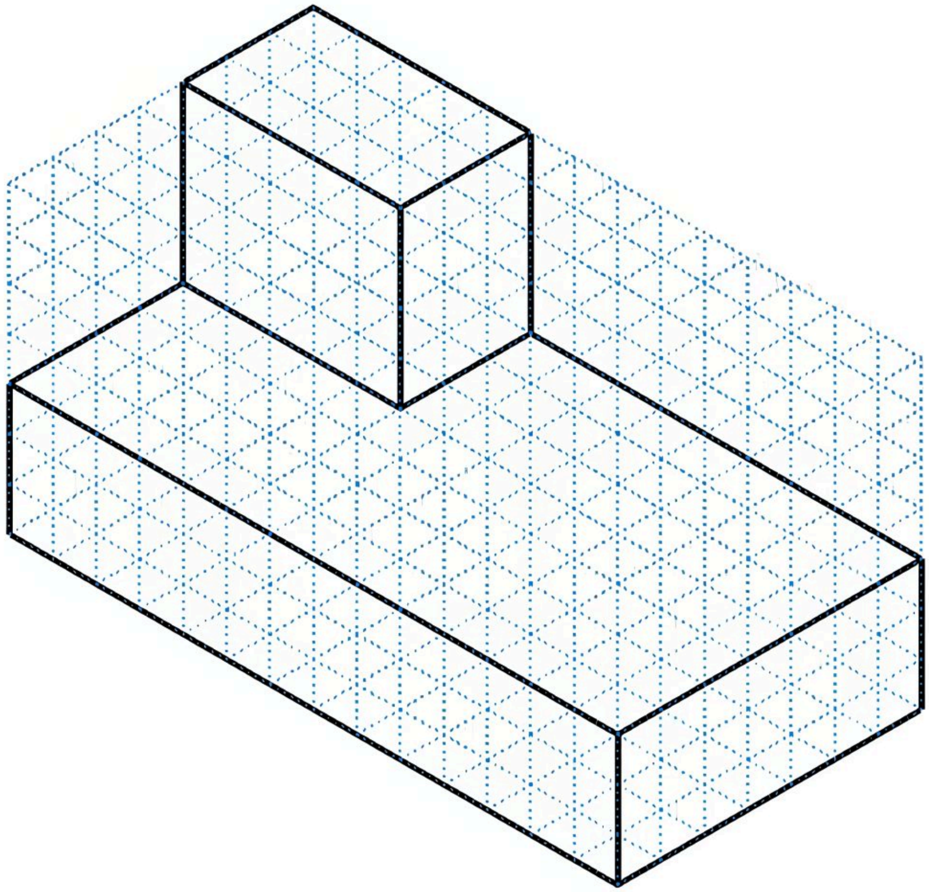


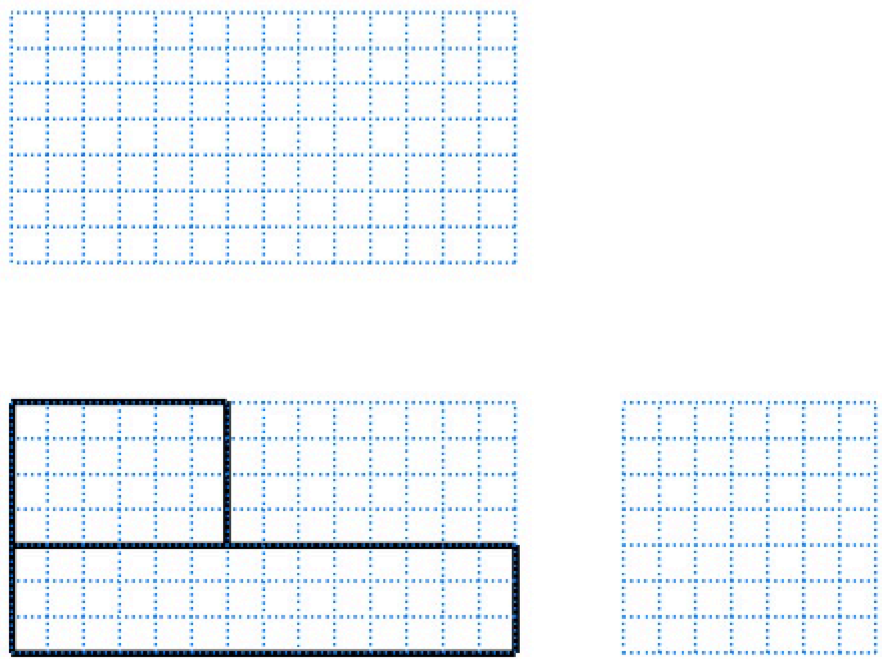
C



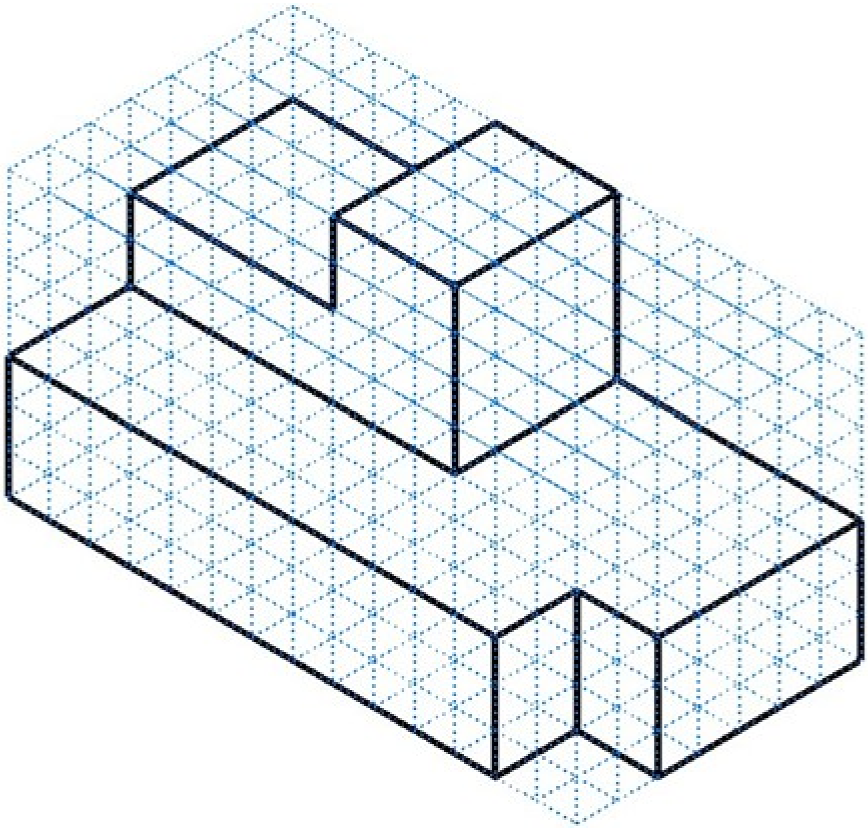
D

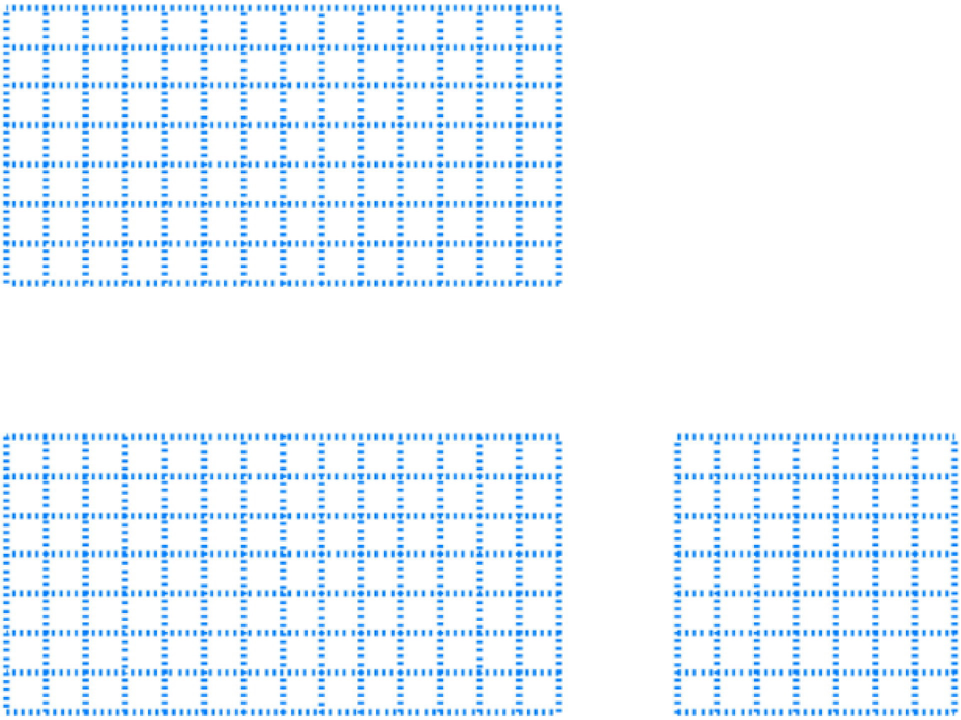
Orthographic Sketch 1



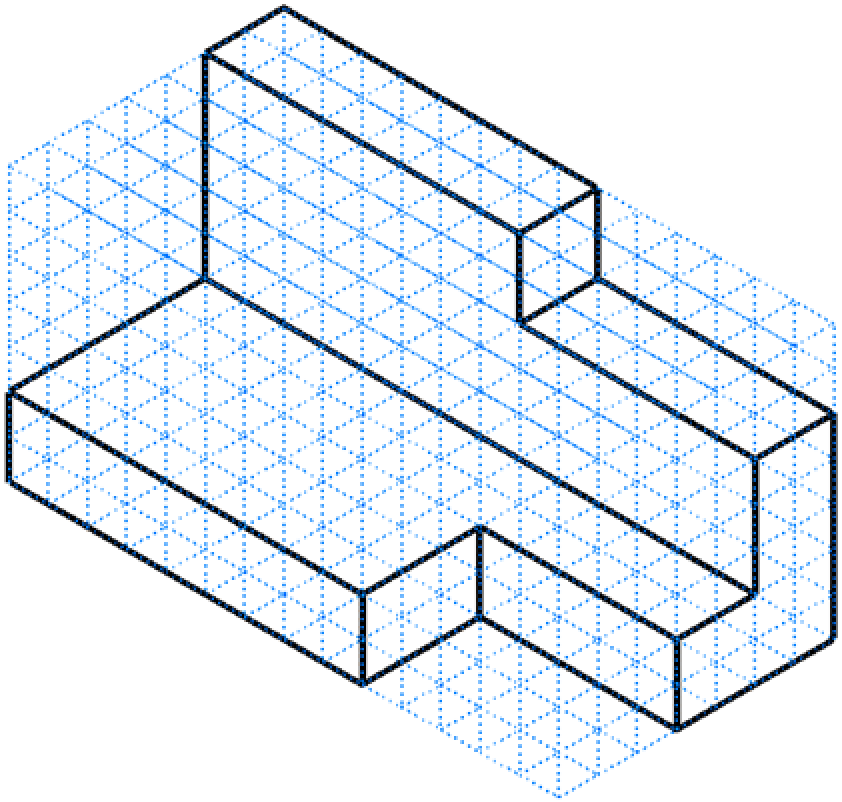


Orthographic Sketch 2

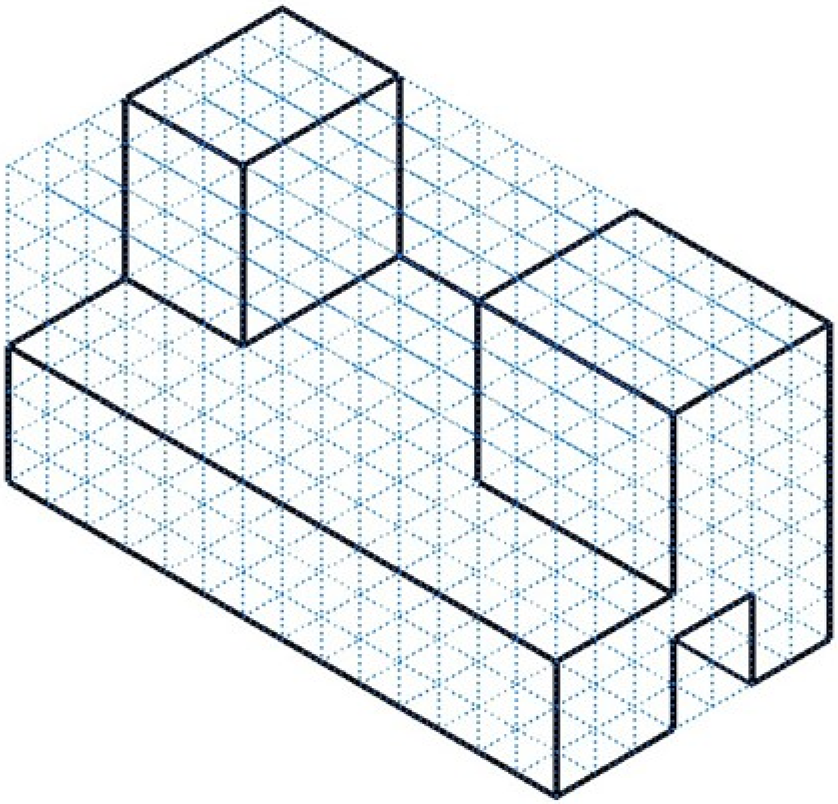


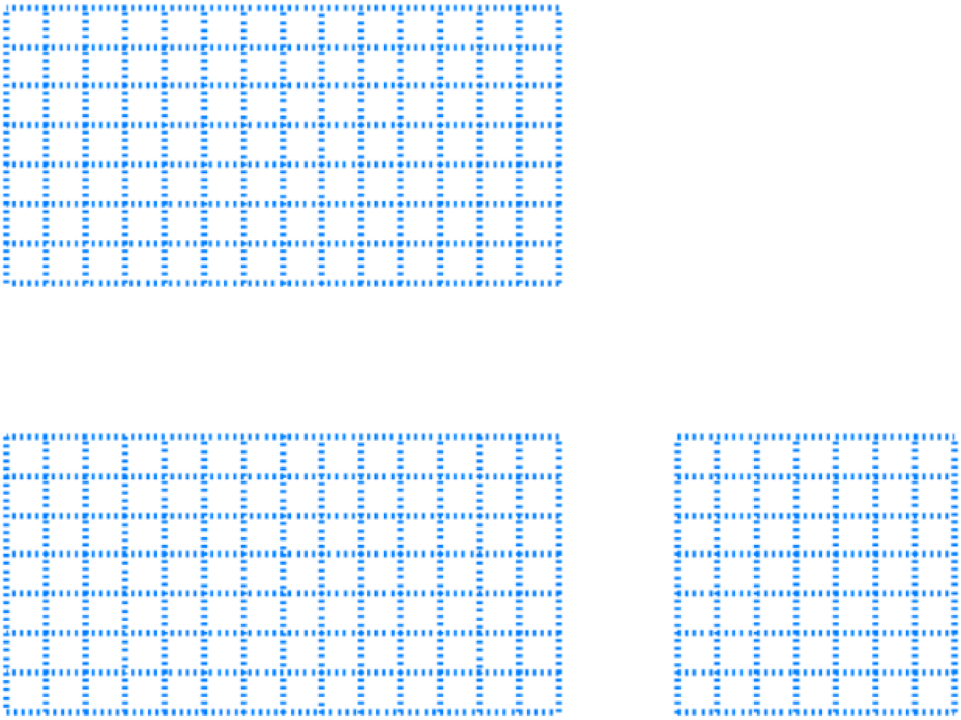


Orthographic Sketch 3

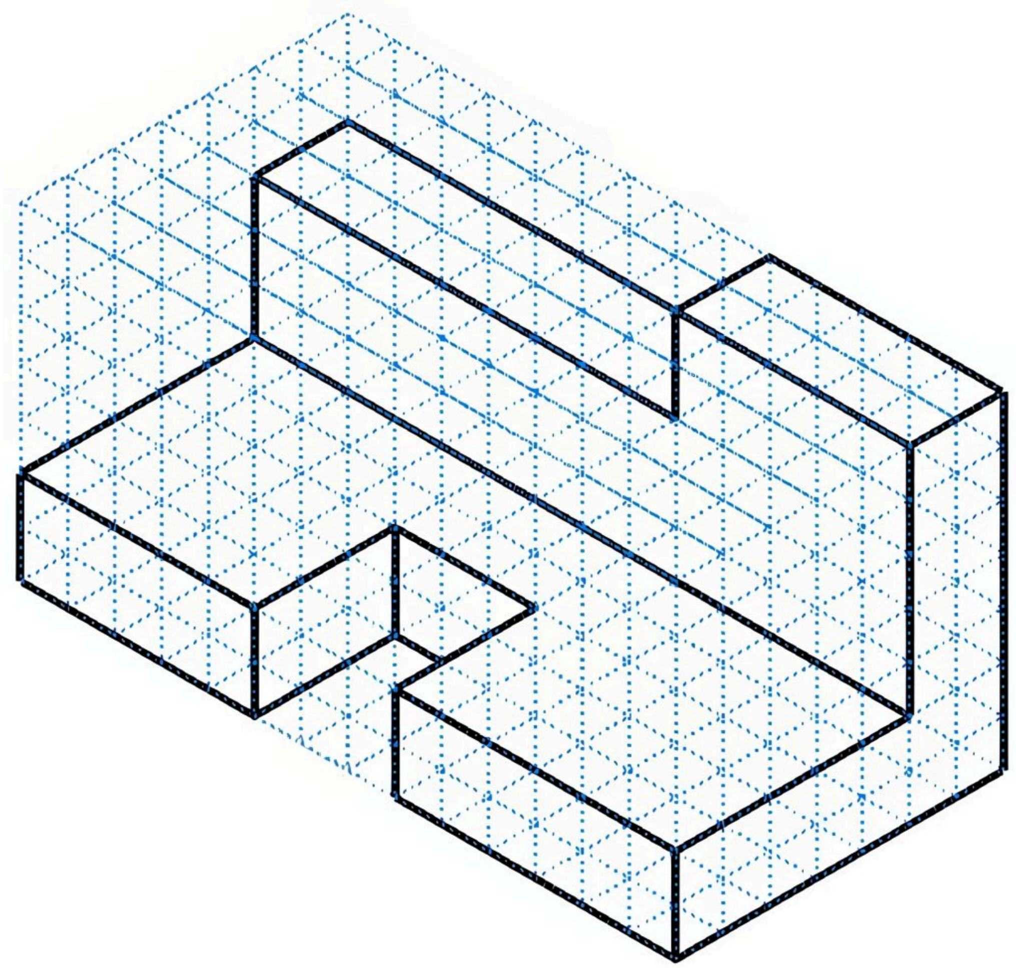


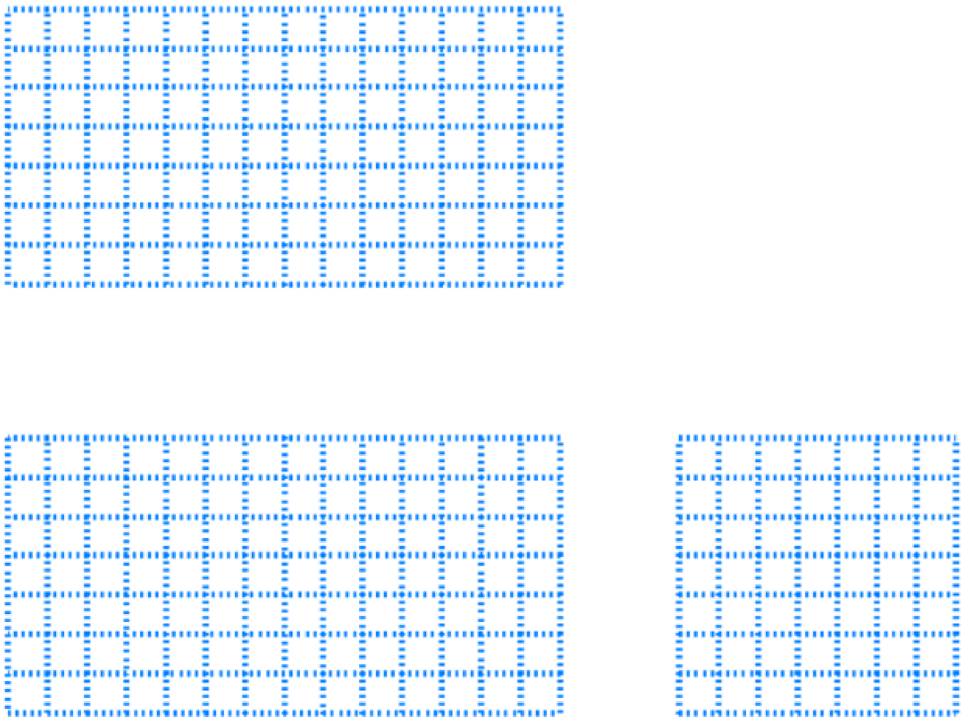
Orthographic Sketch 4



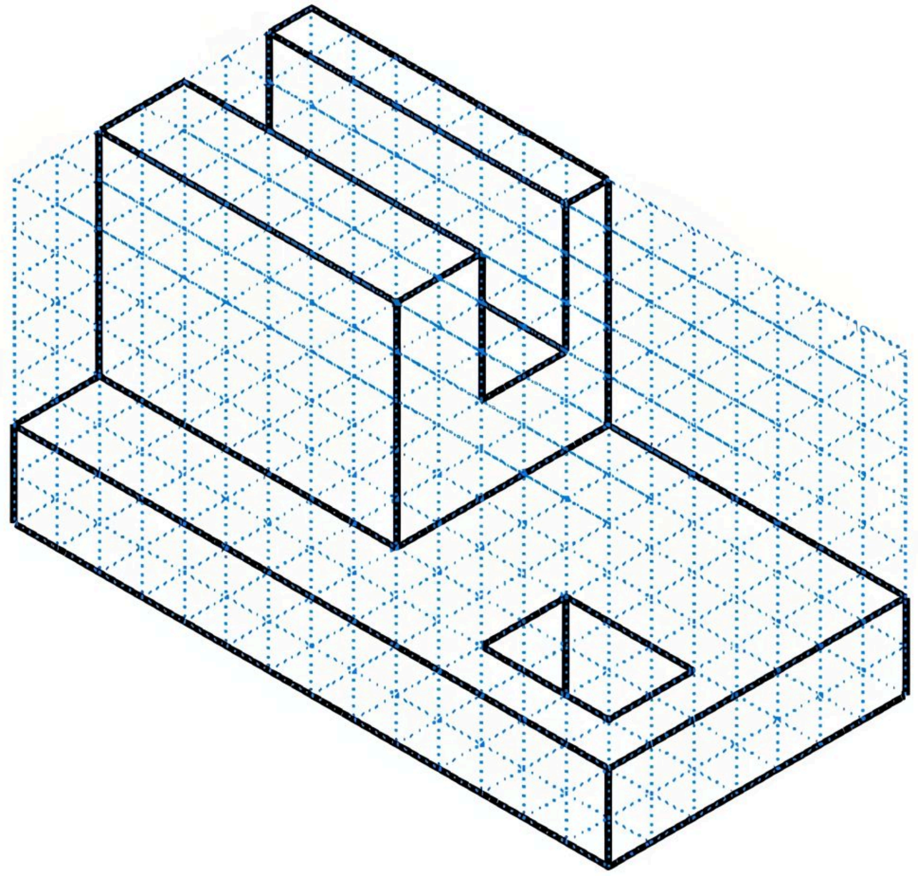


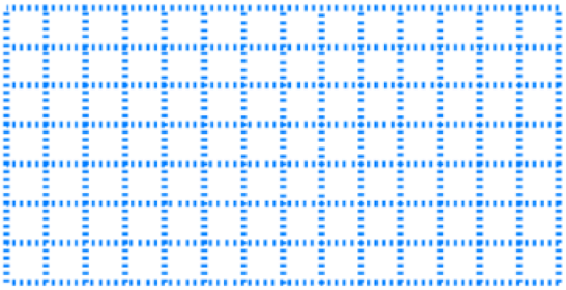
Orthographic Sketch 5



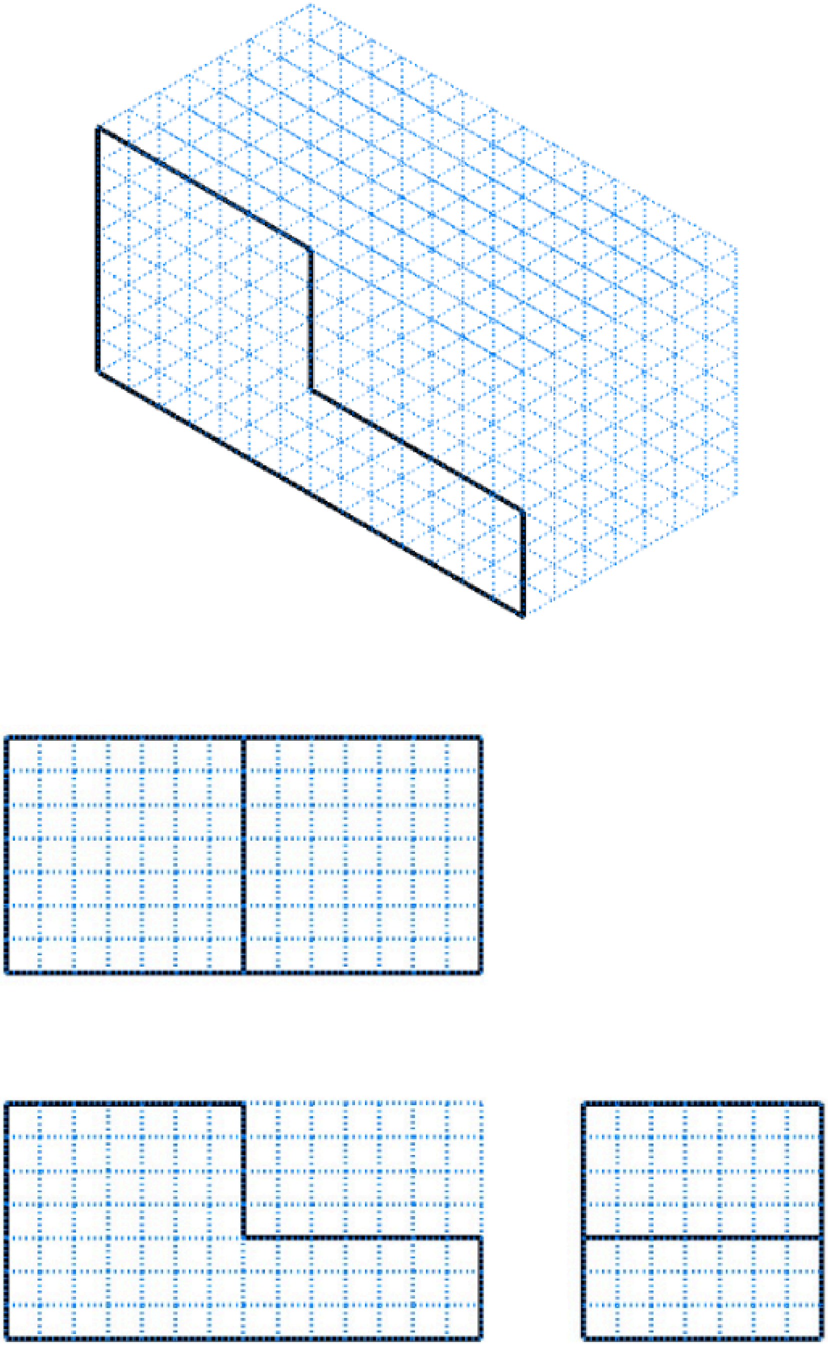


Orthographic Sketch 6

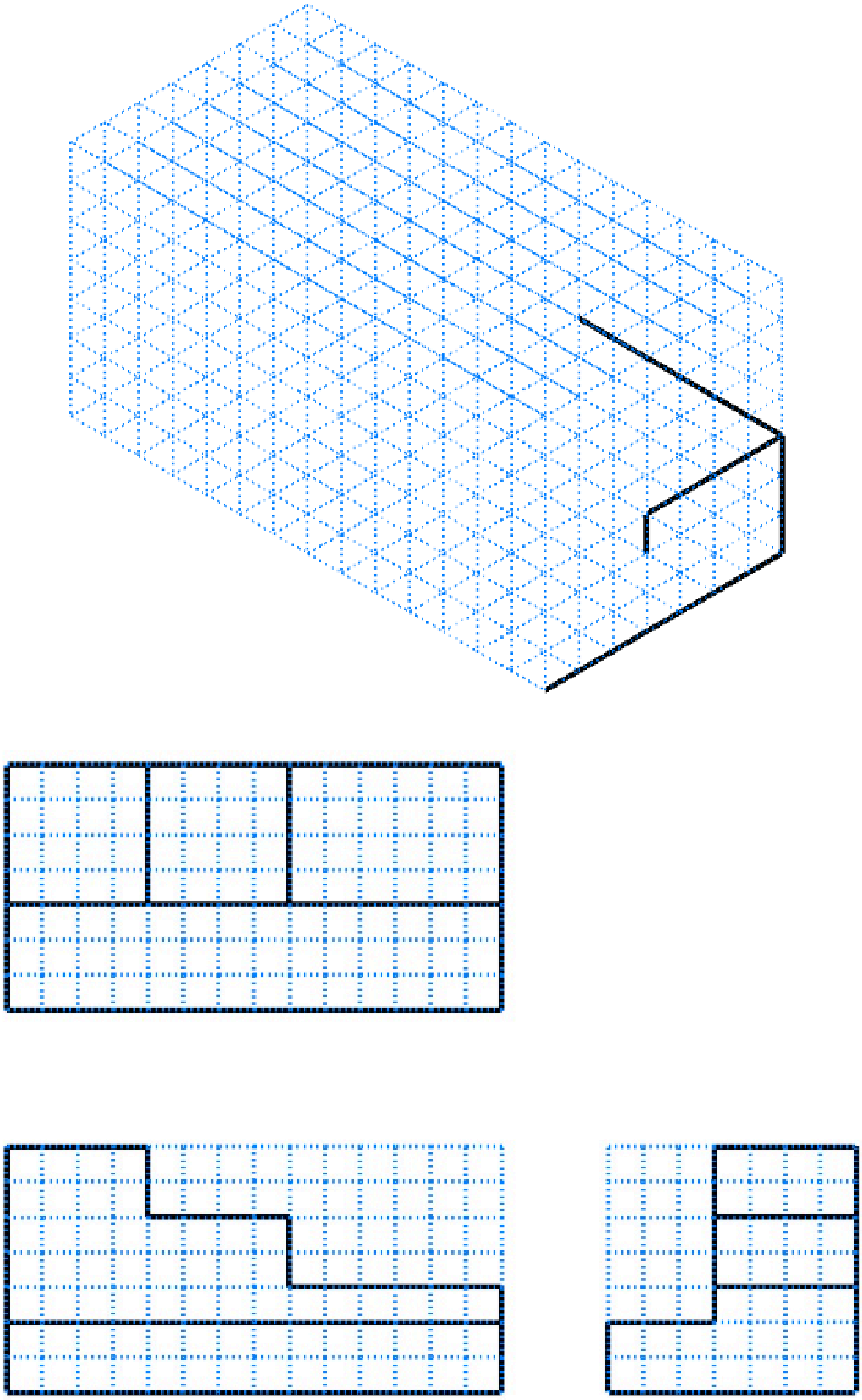




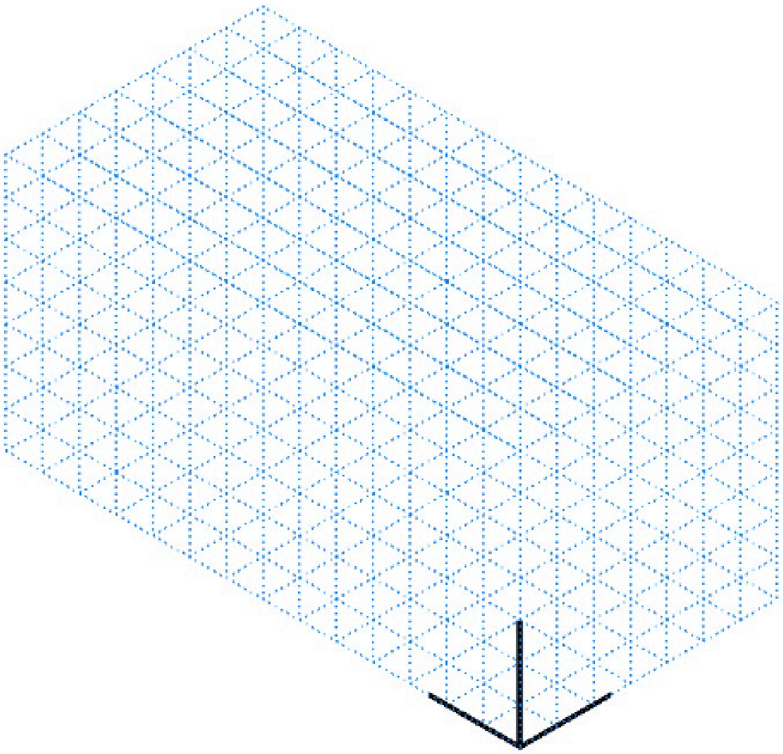
Isometric Sketch 1

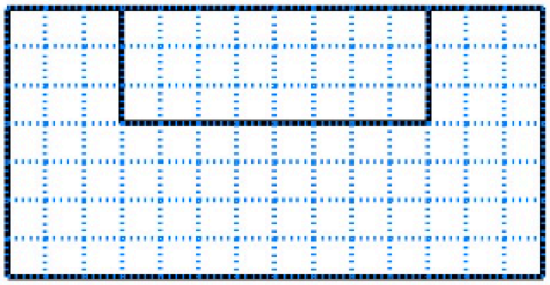


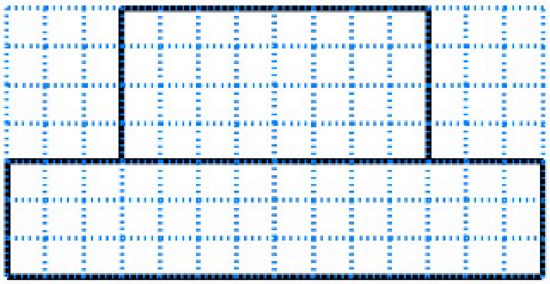
Isometric Sketch 2

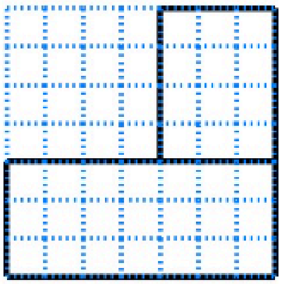


Isometric Sketch 3

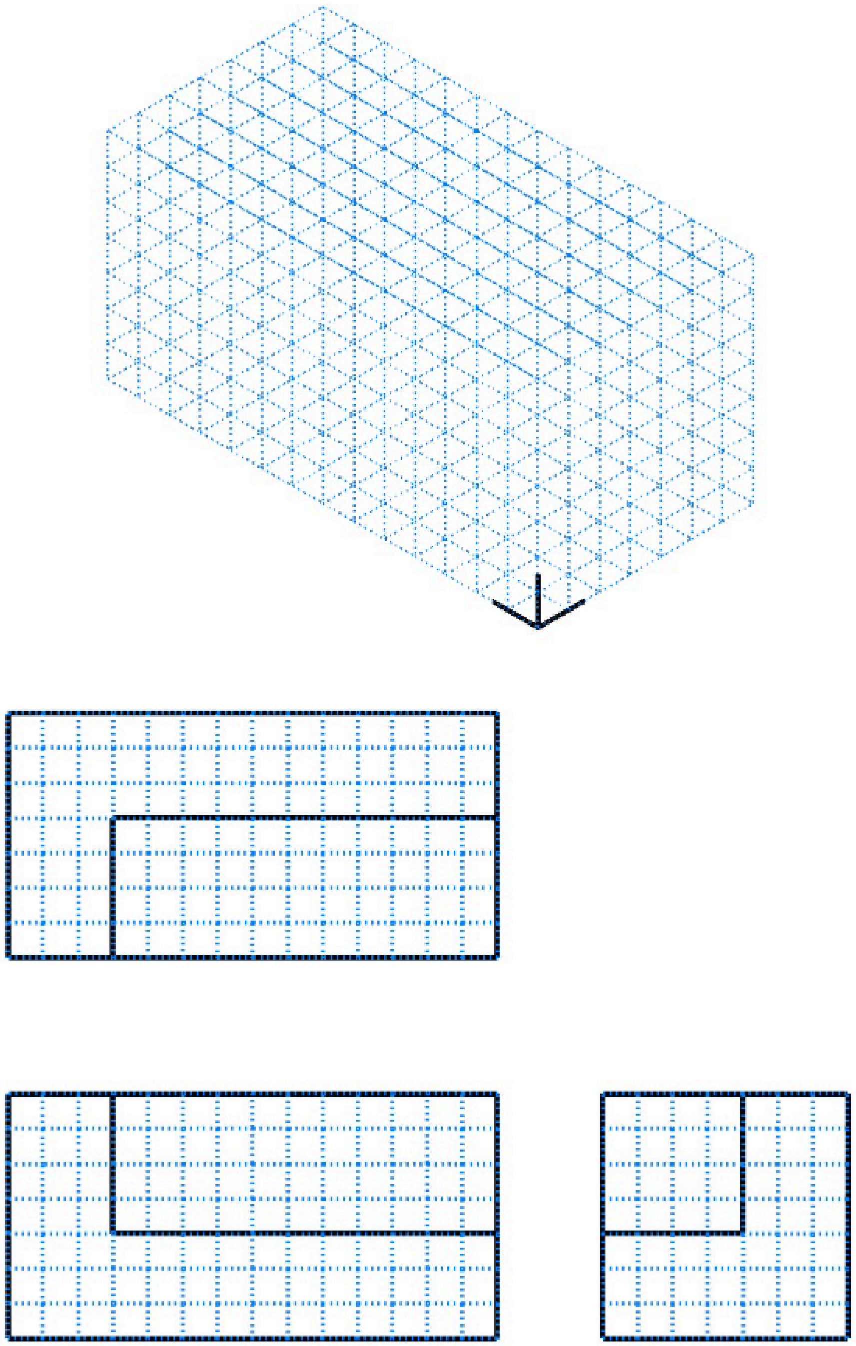




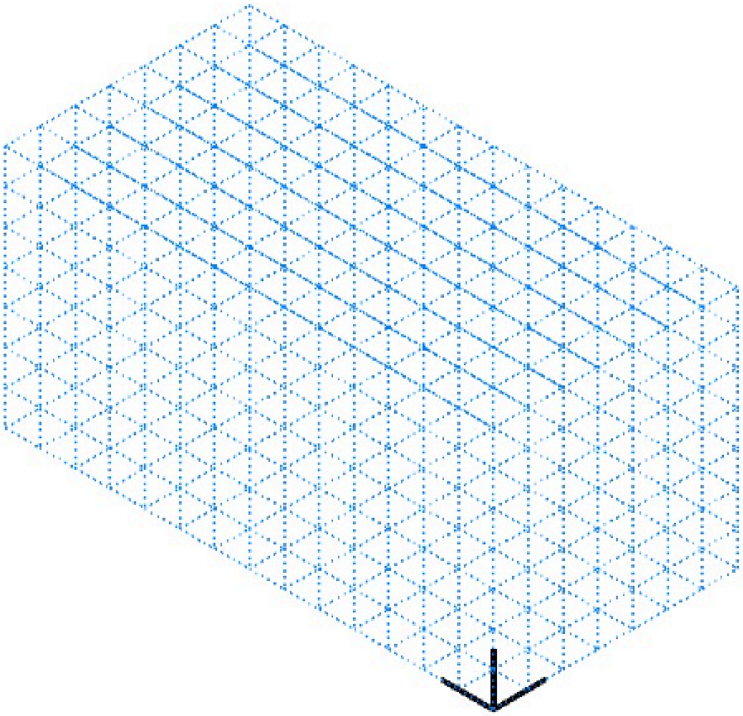


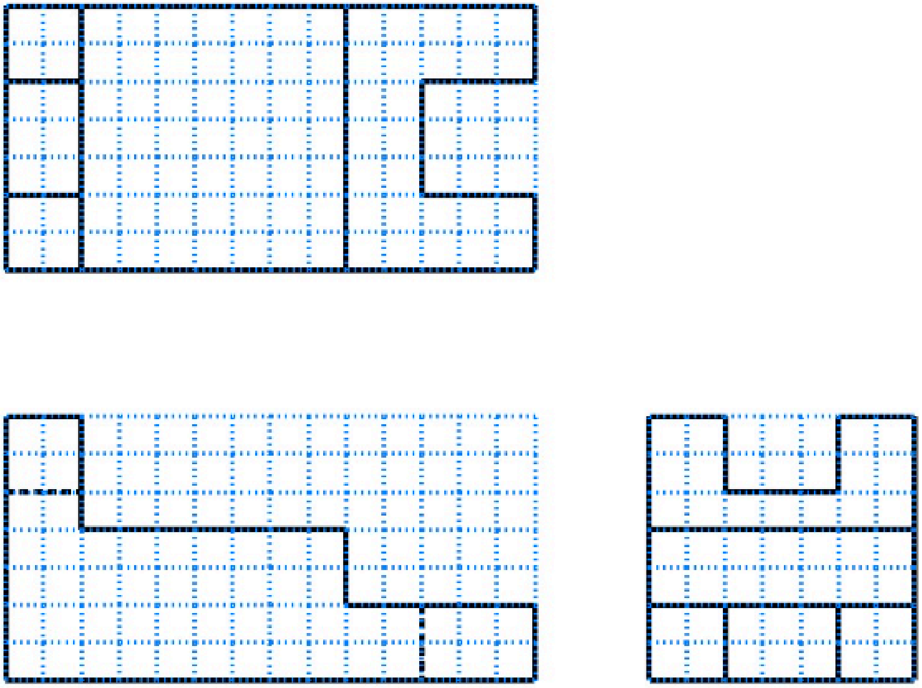


Isometric Sketch 4

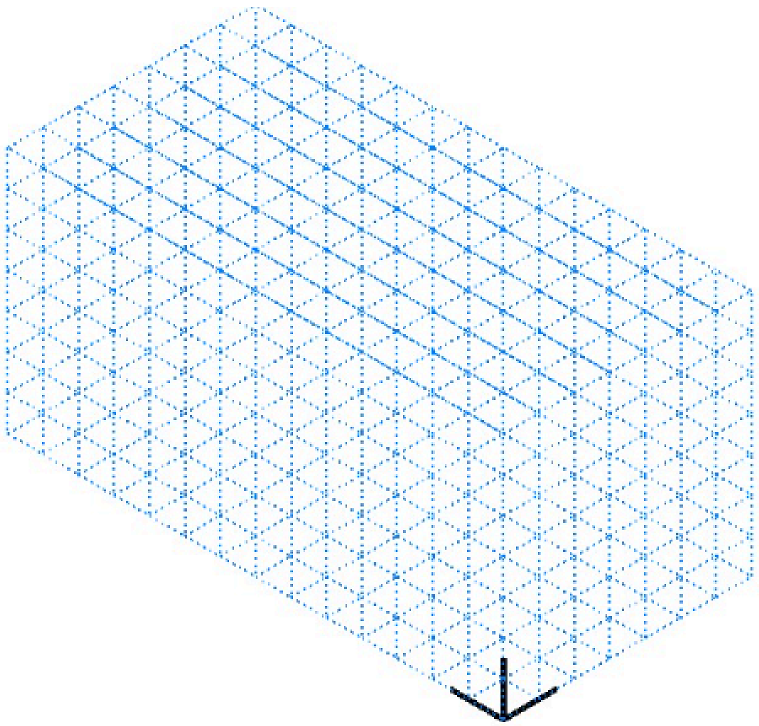


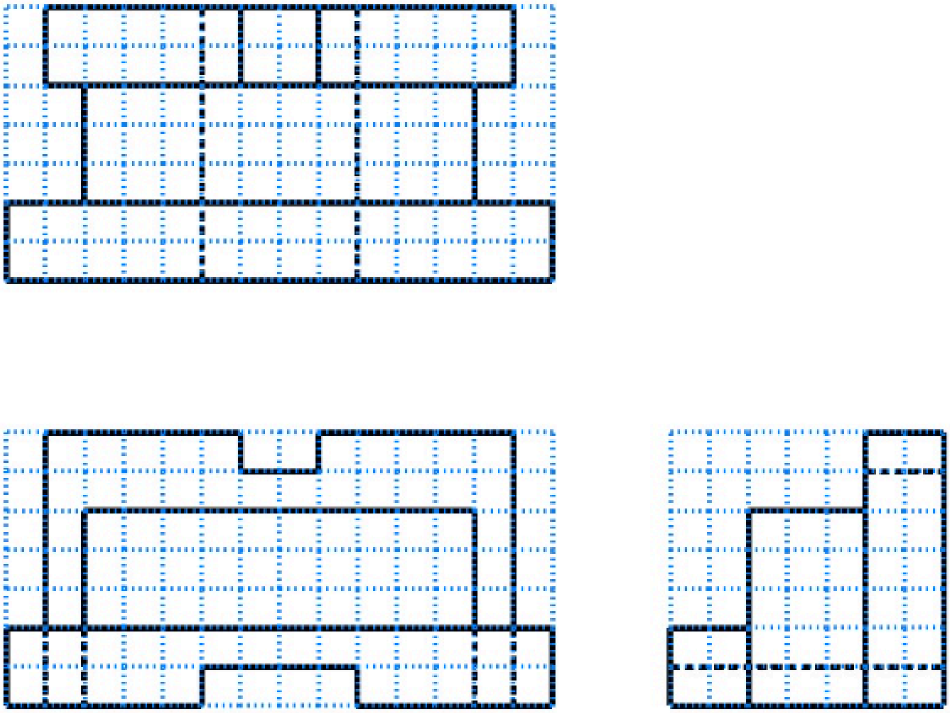
Isometric Sketch 5





Isometric Sketch 6

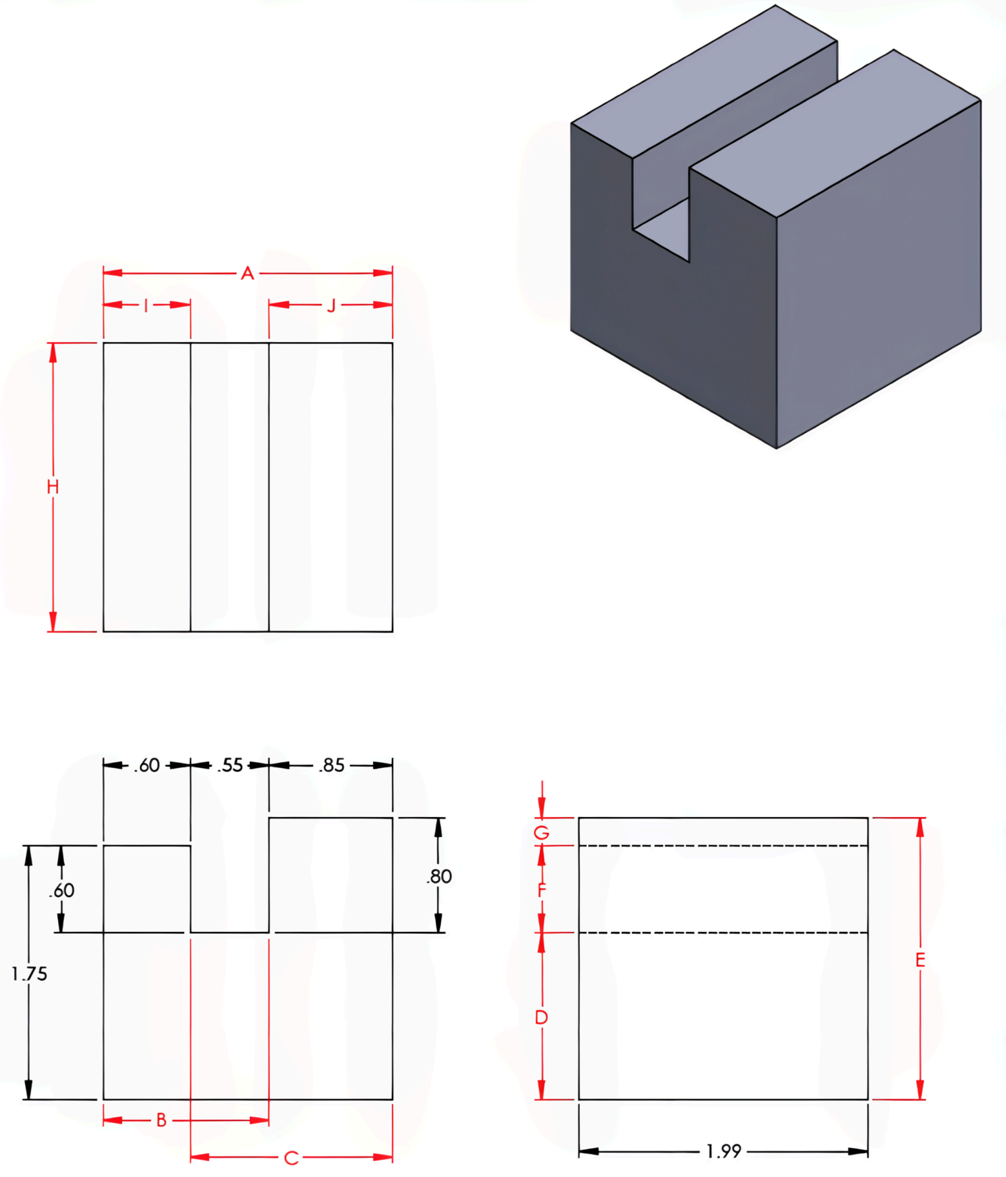




CHAPTER 5: DIMENSIONING SYSTEMS

Exercise 5.2-1

Directions: Calculate the dimensions for letters A through J using the support block below.



DIMENSIONS IN INCHES. TOLERANCES:
TWO PLACE DECIMAL ±.02
THREE PLACE DECIMAL ±.005

MATERIAL: Aluminium

DO NOT SCALE DRAWING

Support Block

SIZE A

DWG. NO. 5.2 Exercise 1

REV.

SCALE:1:1

A. ______________________________ E.

______________________________ I. ______________________________

B. ______________________________ F.

______________________________ J. ______________________________

C. ______________________________ G.

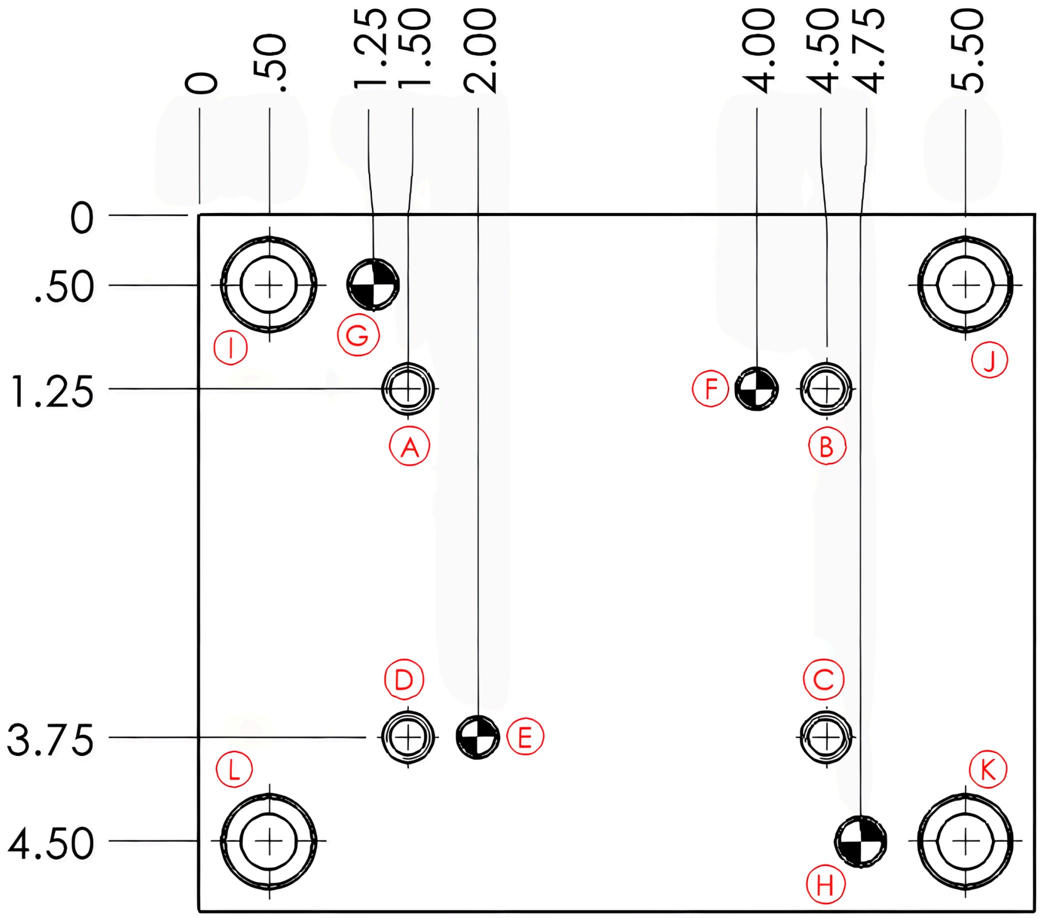
____

D. ______________________________ H.

____

Exercise 5.2-2

Directions: When programming or producing the part, the datum will be the X and Y absolute zero point. Identify the X and Y locations for letters A through L.



A. X ___________ Y ___________ E. X ___________ Y ___________

I. X ___________ Y ___________

B. X ___________ Y ___________ F. X ___________ Y ___________

J. X ___________ Y ___________

C. X ___________ Y ___________ G. X ___________ Y ___________

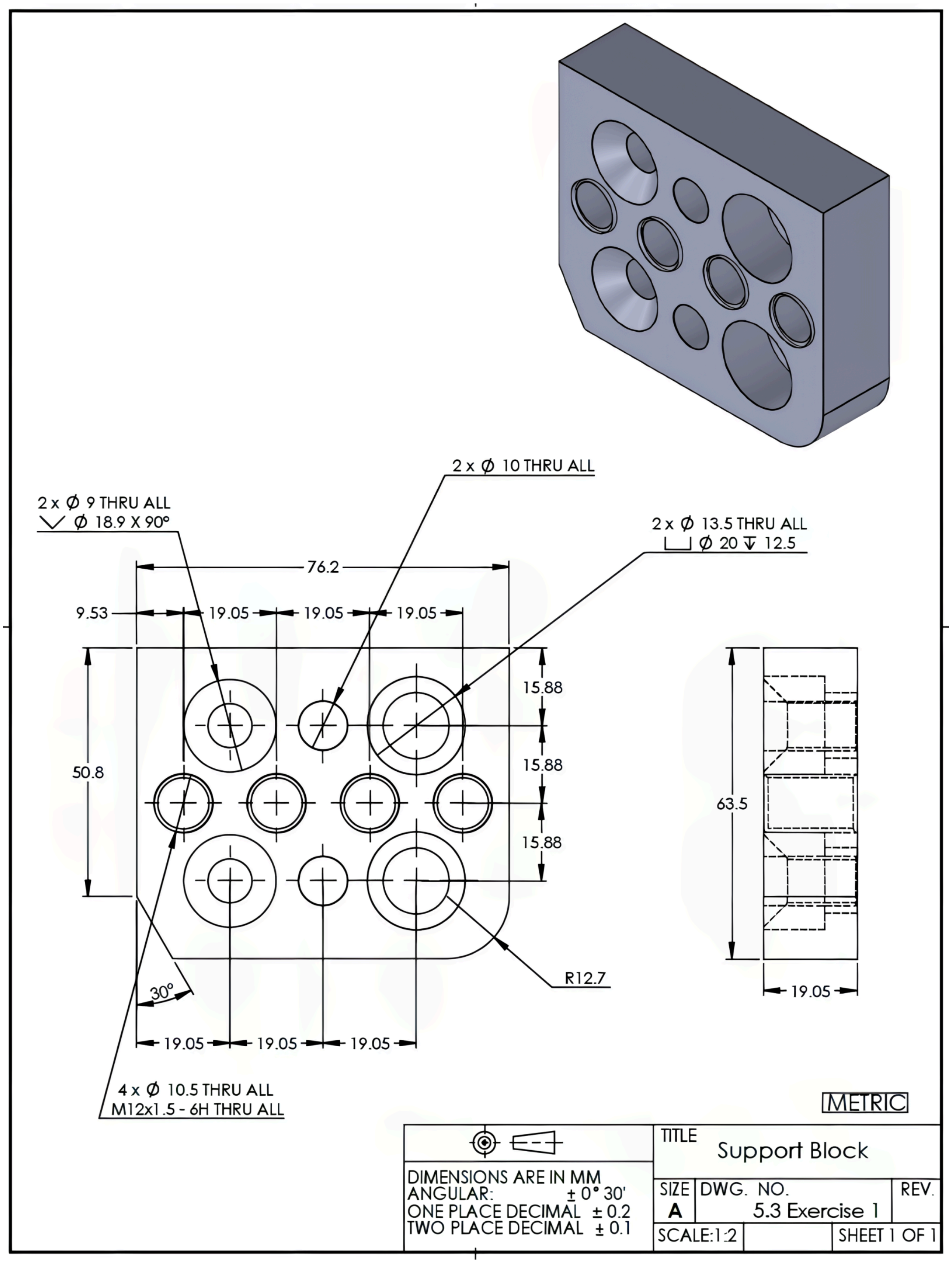
K. X ___________ Y ___________

D. X ___________ Y ___________ H. X ___________ Y ___________

L. X ___________ Y ___________

Exercise 5.3-1

Directions: Calculate the decimal inch equivalents for the metric dimensions below, and round answers to three decimal places.


2 x Ø 10 THRU ALL
2 x Ø 9 THRU ALL
⌵ Ø 18.9 X 90°
2 x Ø 13.5 THRU ALL
⌴ Ø 20 ↧ 12.5
76.2
9.53
19.05
19.05
19.05
15.88
15.88
15.88
50.8
63.5
30°
R12.7
19.05
19.05
19.05
19.05
19.05
4 x Ø 10.5 THRU ALL
M12x1.5 - 6H THRU ALL
METRIC
TITLE
Support Block
DIMENSIONS ARE IN MM
ANGULAR: ± 0° 30'
ONE PLACE DECIMAL ± 0.2
TWO PLACE DECIMAL ± 0.1
SIZE
A
DWG. NO.
5.3 Exercise 1
REV.
SCALE:1:2
SHEET 1 OF 1

A. 9.525) ______________________ F. 63.50)
______________________ K. 15.88) ______________________

B. 12.70) ______________________ G. 76.20)
______________________ L. 10.02) ______________________

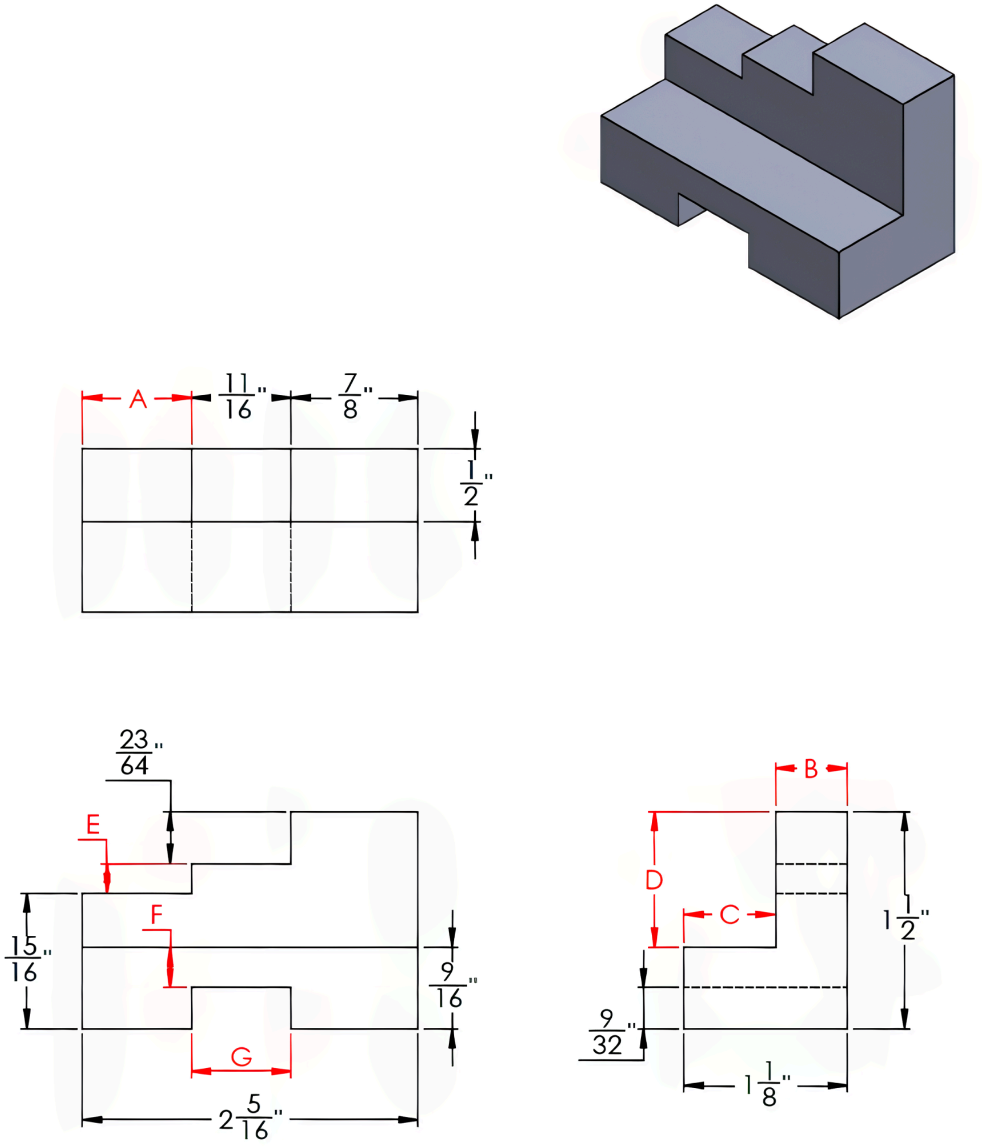
C. 15.875) ______________________ H. 6.53)
______________________ M. 17.15) ______________________

D. 19.05) ______________________ I. 9.21)
______________________ N. 11.35) ______________________

E. 50.80) ______________________ J. 10.08)
______________________ O. 12.38) ______________________

Exercise 5.3-2

Directions: Determine the missing fractional dimensions for the adjustment bracket. Refer to the drawing.



TITLE
Adjustment Bracket

DIM. IN FRACTIONAL INCH
ANGULAR: ± 0° 30'
FRACTIONS ±1/64

SIZE
A

DWG. NO.
5.3 Exercise 2

REV.

SCALE:1:1

SHEET 1 OF 1

A. ______________________________

____

D.

G. ______________________________

B. ______________________________

____

E.

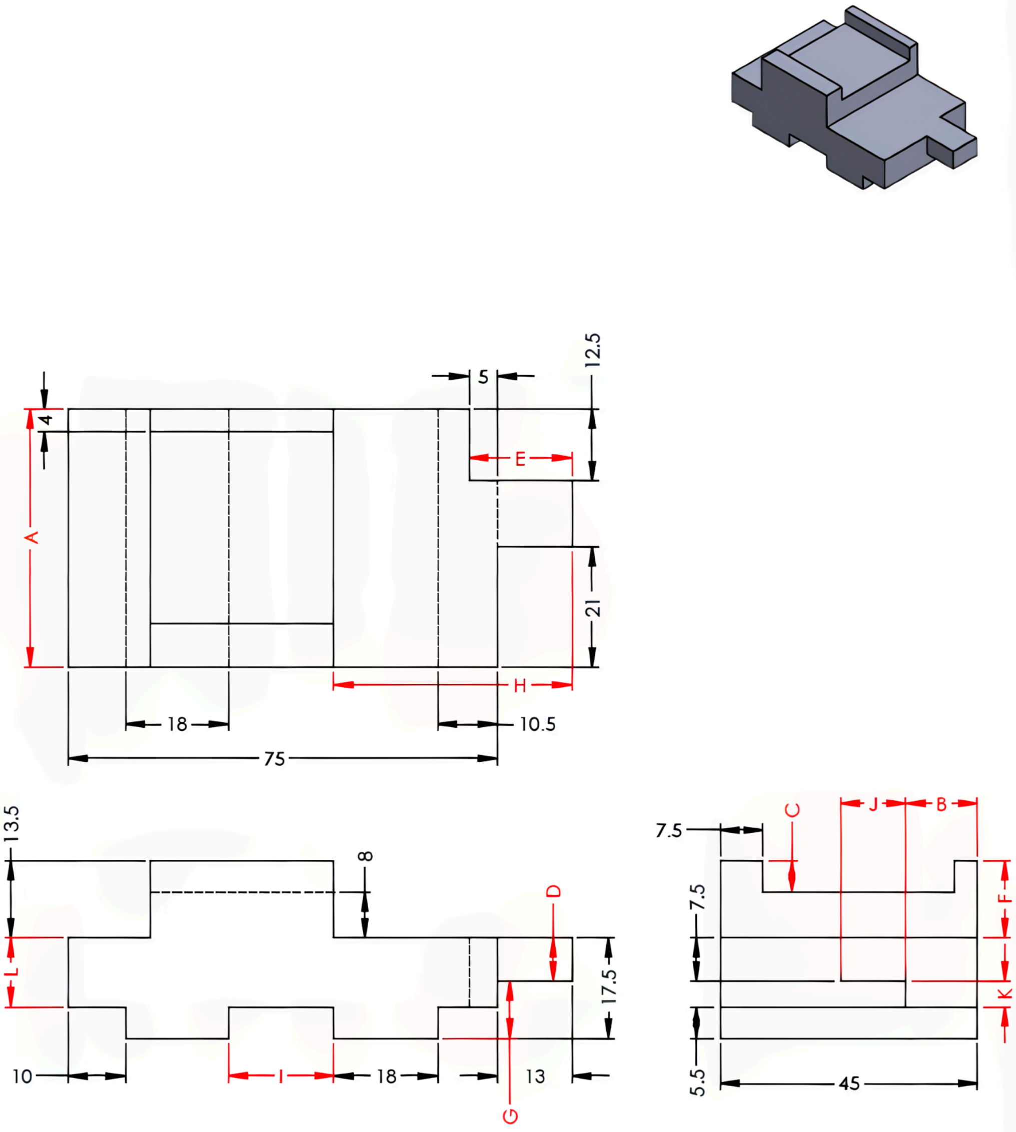
C. ______________________________

____

F.

Exercise 5.3-3

Directions: Determine the dimensions for the slide block.



METRIC

	TITLE Slide Block
DIMENSIONS ARE IN MM ANGULAR: ± 0° 30' TWO PLACE ±.5MM THREE PLACE ±.2 MM	SIZE A DWG. NO. 5.3 Exercise 3 REV. SCALE:1:1 SHEET 1 OF 1

A. ______________________________ E.
______________________________ I. ______________________________

B. ______________________________ F.
______________________________ J. ______________________________

C. ______________________________ G.
______________________________ K. ______________________________

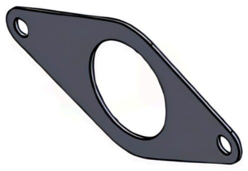
D. ______________________________ H.
______________________________ L. ______________________________

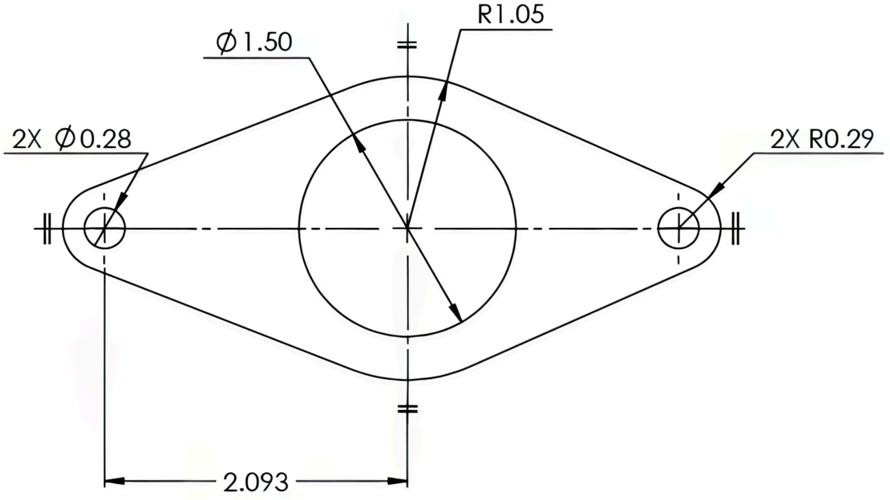
1. For the slot labeled “I” on the front view, why does the right hidden line not appear on the top view? ______________________________
2. Which system of measurement is being used (decimal, fractional, metric)? ______________________________
3. Which angle of projection is being used? ______________________________

CHAPTER 6: PRINT DIMENSIONS

Exercise 6.4-1

Directions: Answer the following questions about the exhaust flange print provided.





TITLE EXHAUST FLANGE

UNSPECIFIED TOLERANCES:
ANGULAR: ± 0° 30'
TWO PLACE DECIMAL ±0.01
THREE PLACE DECIMAL ±0.005

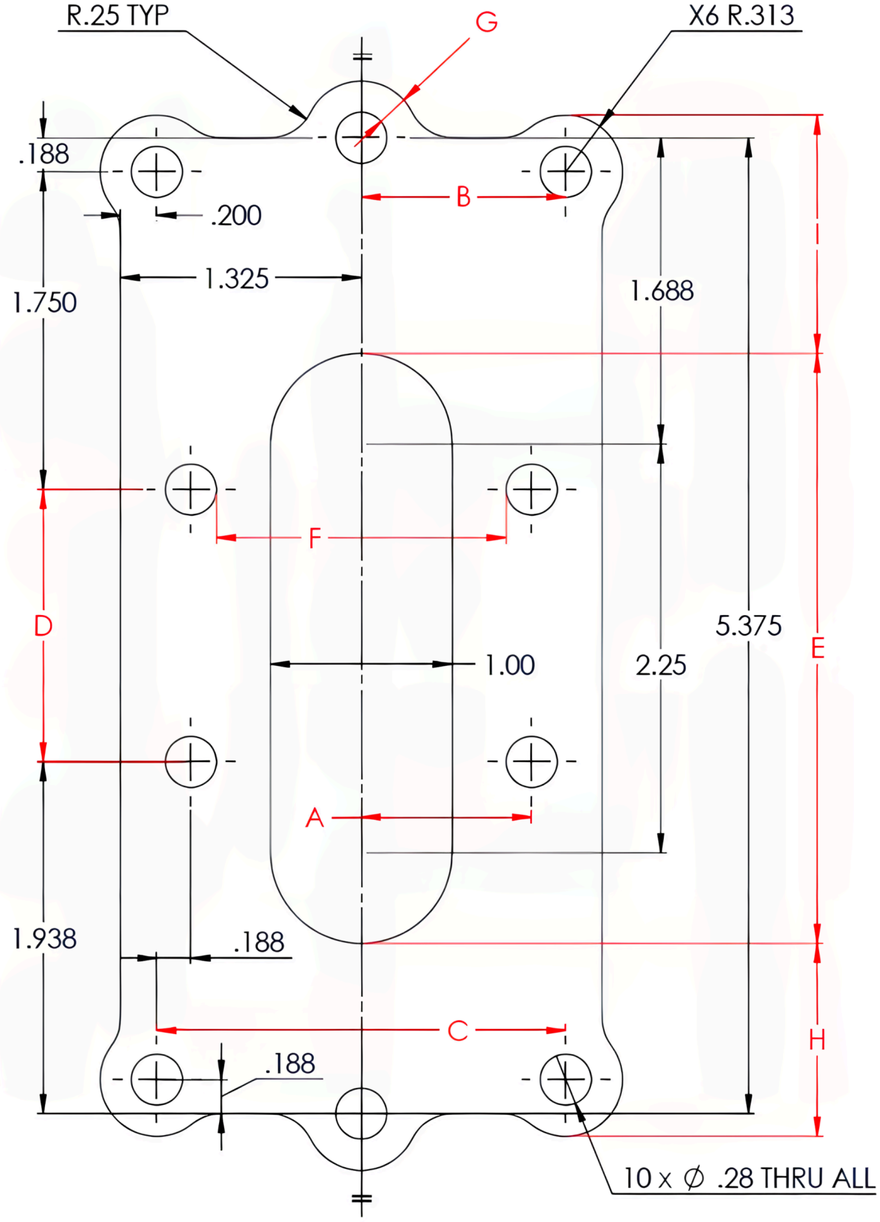
SIZE	DWG. NO.	MATERIAL	REV.
A	240806	.063 SS	

SCALE:1:1 | DRAWN BY: MDL | SHEET 1 OF 1

1. Is the flange symmetrical along the vertical or horizontal line? ________________________
2. Calculate the overall height of the flange. ________________________
3. Calculate the overall width of the flange. ________________________
4. What is the thickness of the flange? ________________________
5. What is the radius of the large hole? ________________________
6. Calculate the distance or wall thickness between the center hole and the top or bottom of the flange. ________________________
7. Calculate the wall thickness between one of the small holes and the outside edge. ________________________
8. Calculate the distance between the inside edges of the small holes. ________________________
9. Without regard to tolerance, calculate the distance between the edge of the large hole and the edge of the small hole. ________________________
10. Which dimensioning unit is being used? ________________________

Exercise 6.4-2

Directions: Calculate the following dimensions for the riser plate print provided. Represent all answers in three decimal places.



	TITLE Riser Plate		
DIMENSIONS IN MM ANGULAR: ± 0° 30' ONE PLACE ±0.5 TWO PLACE ±0.25	SIZE A	DWG. NO. 240807	MATERIAL .5 ALUM.
	SCALE:1.25:1	DRAWN BY: MDL	

A. ____________________________ E. ____________________________ I. ____________________________

B. ____________________________ F. ____________________________

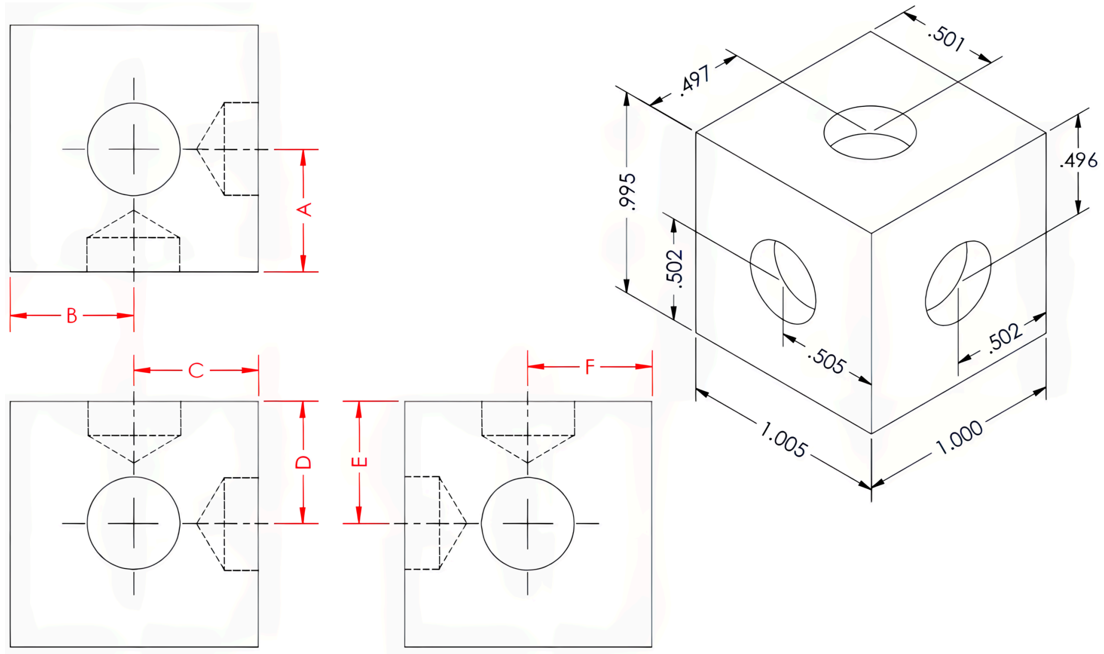
C. ____________________________ G. ____________________________

D. ____________________________ H. ____________________________

1. What is the overall height of the riser plate (top to bottom)? _______________________
2. Without regard to tolerance, what is the largest width of the riser plate (left to right)?____
3. What prevents the part from being symmetrical along its horizontal centerline? ____

Exercise 6.4-3

Directions: Use the dimensioned isometric view to complete the dimensions on the orthographic view.



1. Height: ______________________________ 4. A)

______________________________ 7. D)

____

2. Width: ______________________________ 5. B)

______________________________ 8. E)

____

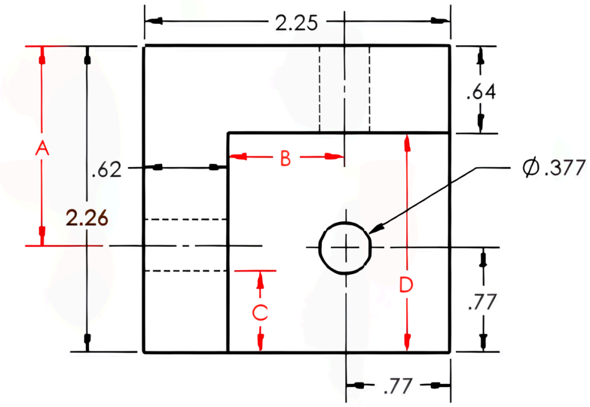
3. Depth: ______________________________ 6. C)

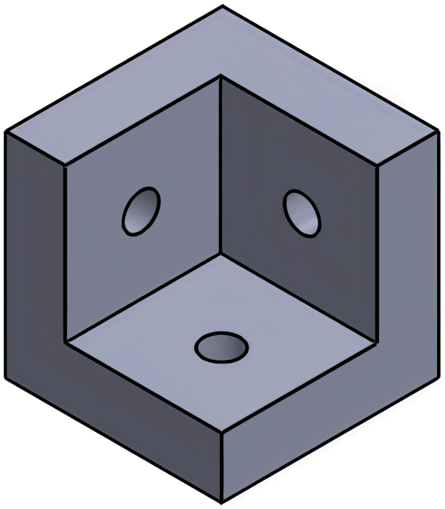
______________________________ 9. F)

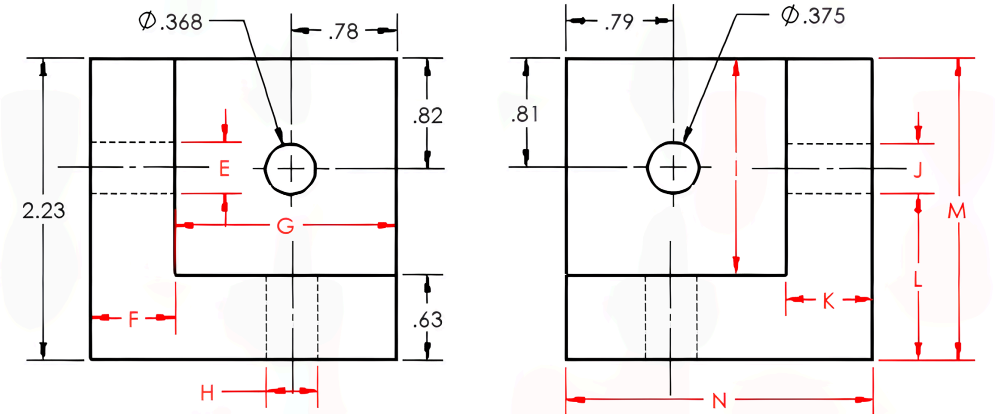
____

Exercise 6.4-4

Directions: Find the dimensions for the letters below using the multiple views.







1. A) ______________________ 6. F)

______________________ 11. K)

____

2. B) ______________________________ 7. G)

______________________________ 12. L)

____

3. C) ______________________________ 8. H)

______________________________ 13. M)

____

4. D) ______________________________ 9. I)

______________________________ 14. N)

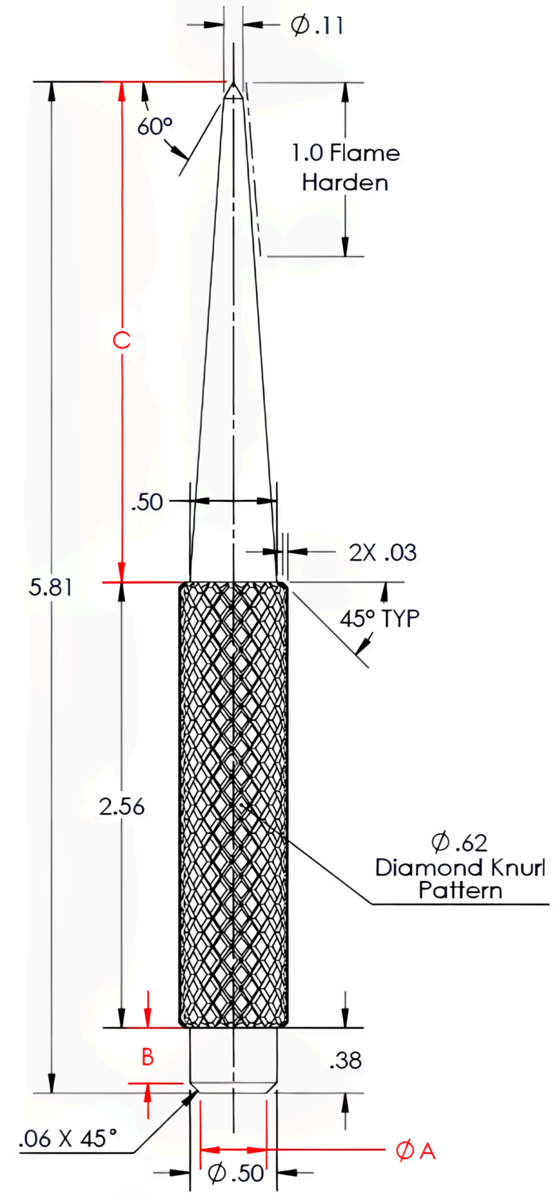
____

5. E) ______________________________ 10. J)

____

Exercise 6.5-1

Directions: Answer the questions referencing the prick punch print provided.

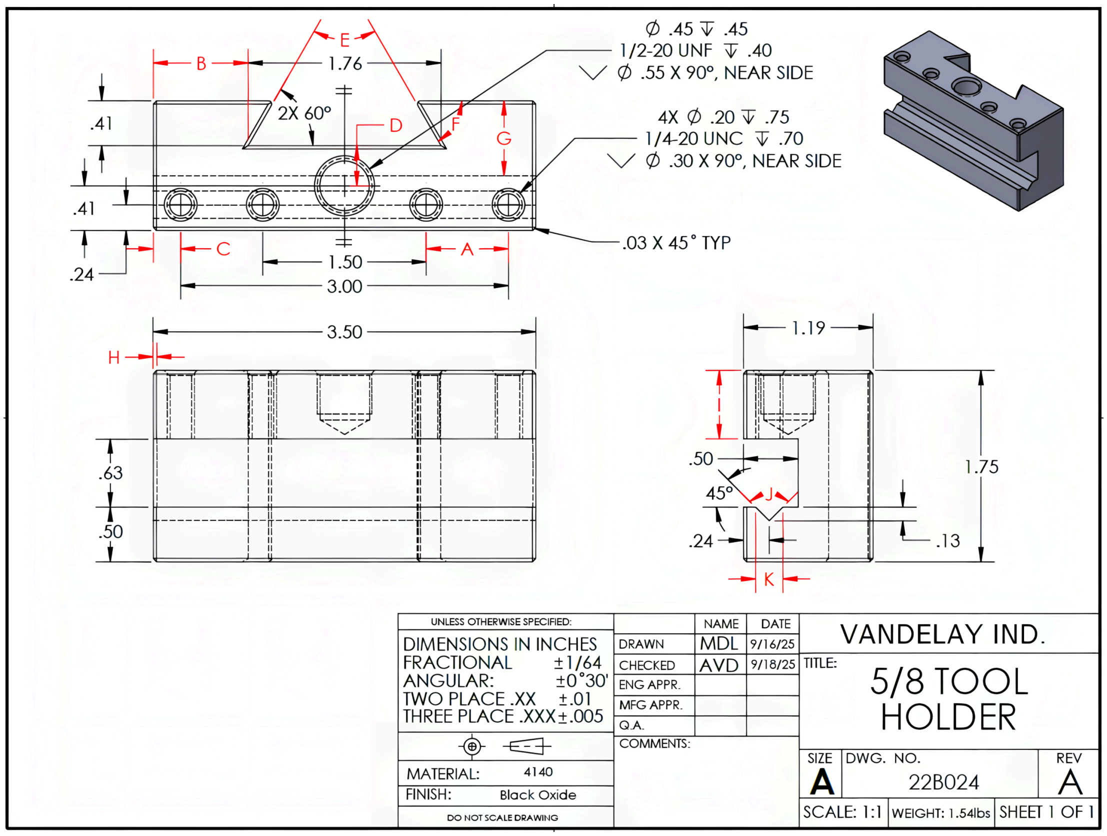


TOLERANCES (EXCEPT AS NOTED)	DRAWN BY:	JTS
DIMENSIONS IN INCHES: ONE PLACE DECIMAL ±.05 TWO PLACE DECIMAL ±.01 THREE PLACE DECIMAL ±.005 ANGULAR ±1/2°	PART NAME:	Prick Punch
	DWG. NO.	12A321
	DATE	8/25
	SCALE NTS	MATERIAL 1018 CRS SIZE A

1. What is the angle included in the punch tip? ________________________
2. What is the length of the knurled portion, NOT including the chamfers? ________________________
3. Which type of line is being used to identify the portion to be flame-hardened? ________________________
4. How much change in diameter occurs in the tapered portion? ________________________
5. What is the diameter A dimension? ________________________
6. What is the length of dimension B? ________________________
7. What is the length of dimension C? ________________________
8. How many .03 x 45° chamfers are on the part? ________________________

Exercise 6.5-2

Directions: Find the dimensions for the letters below using the multiple views.



1. 1. A) ______________________________

6. F)

____

11. K)

____

2. 2. B) ______________________________

7. G)

____

3. 3. C) ______________________________

8. H)

____

4. 4. D) ______________________________

9. I)

____

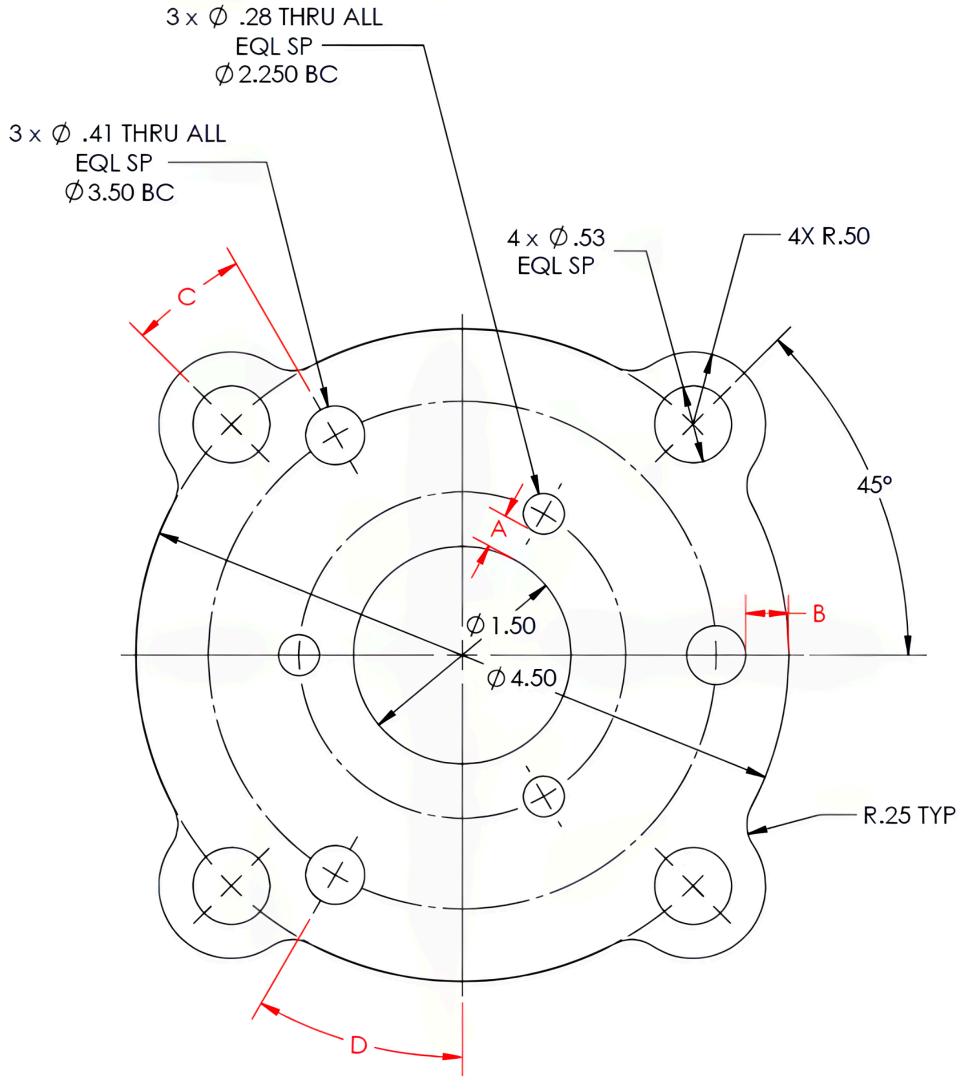
5. 5. E) ______________________________

10. J)

____

Exercise 6.6-1

Directions: Answer the following questions about the pump seal print below.



TOLERANCES
(EXCEPT AS NOTED)

DIMENSIONS IN INCHES
TOLERANCES:
2 PLACE DECIMAL ±.01
3 PLACE DECIMAL ±.005
ANGULAR ±1/2°

DRAWN BY: MDL

PART NAME: PUMP SEAL

DWG. NO. 240808

DATE 8/12/25

SCALE FULL

MATERIAL .06 COPPER

SIZE **A**

1. Is the seal symmetrical along the vertical or horizontal centerline? ________________________
2. How thick is the seal? ________________________
3. How many .25 radii are on the seal? ________________________
4. Are the three bolt circles concentric? ________________________
5. How many holes are on the part? ________________________
6. What is the angular spacing between the ⌀.28 holes? ________________________
7. What is the angular spacing between the ⌀.53 holes? ________________________
8. What is the wall thickness between the ⌀.53 hole and the nearest outside edge? ________________________

Enter the dimensions for the letters:

9. A) ___________

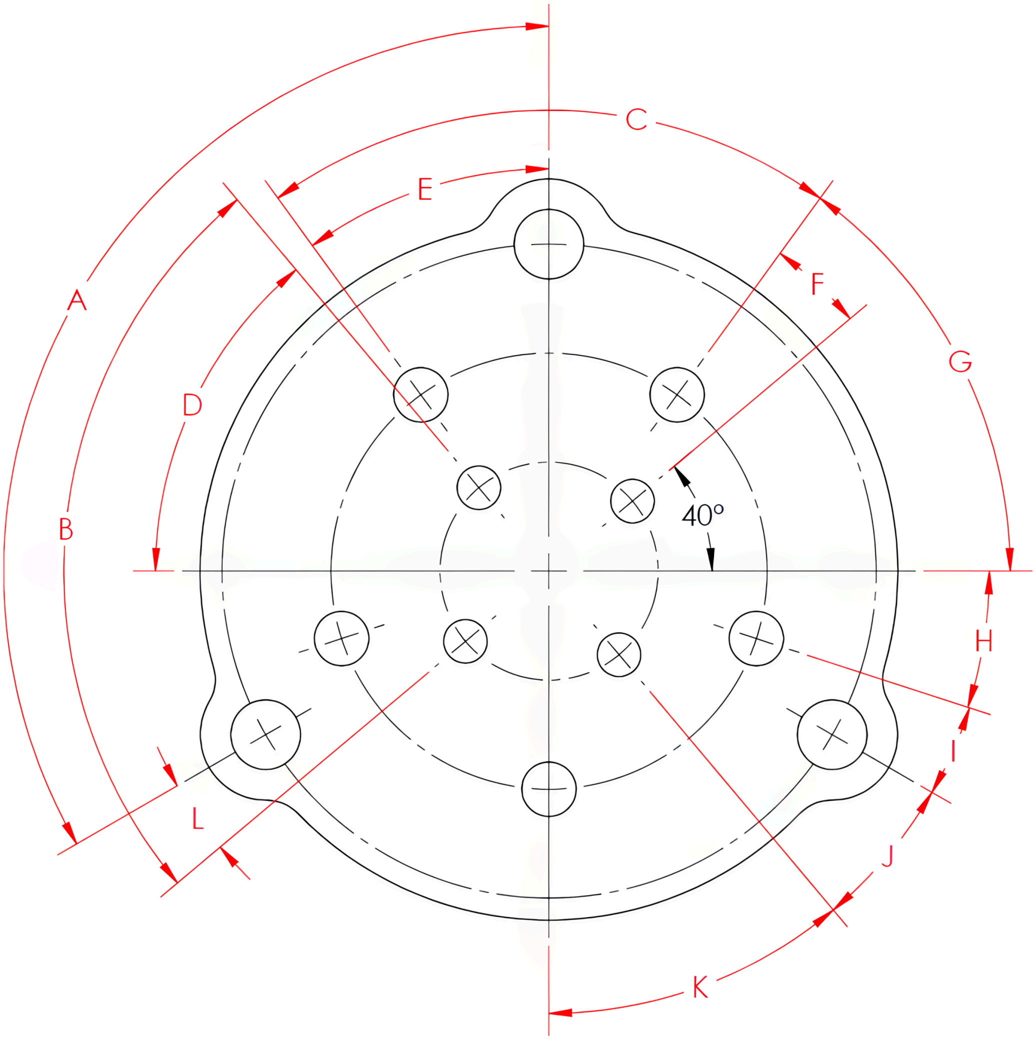
10. B) ___________

11. C) ___________

12. D) ___________

Exercise 6.6-2

Directions: Calculate the angular dimensions for the letters. Recall the angle between equally spaced bolt circles is 360° divided by the number of holes. A bolt-hole location will be located on or dimensioned from a vertical or horizontal line.



A. ______________________________ E.

______________________________ I. ______________________________

B. ______________________________ F.

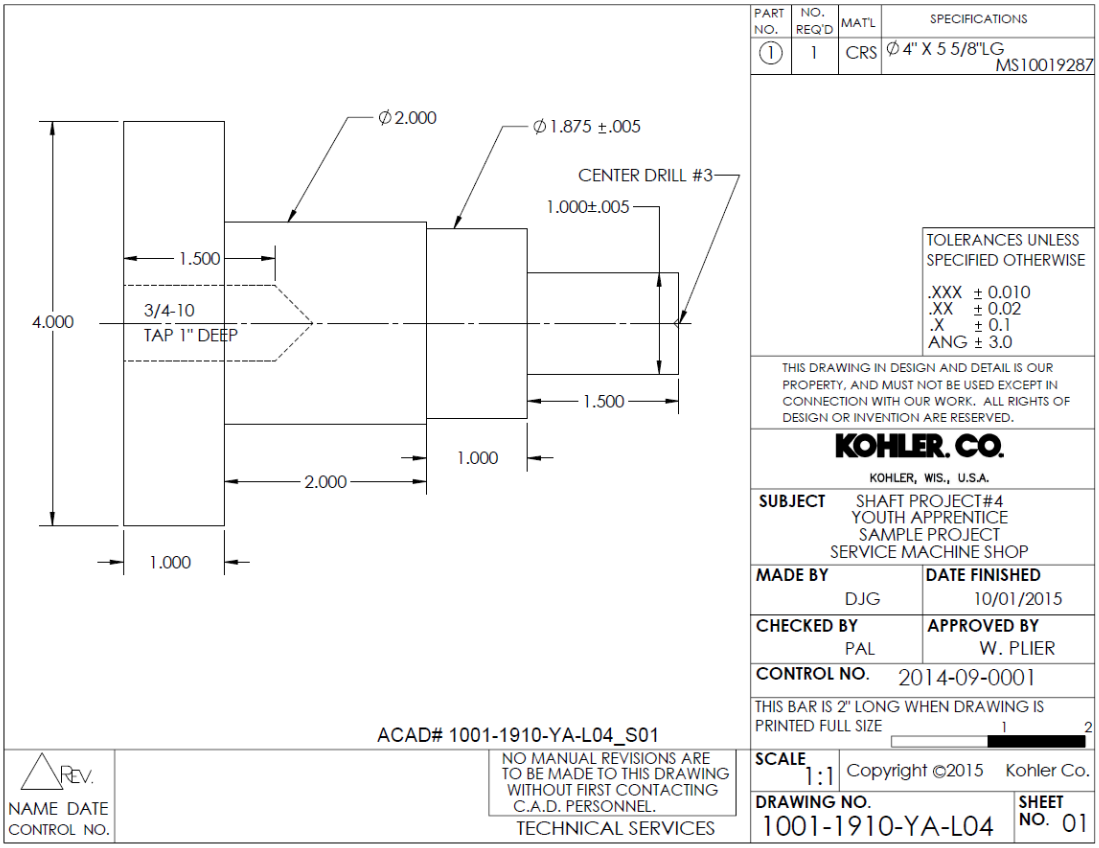
______________________________ J. ______________________________

C. ______________________________ G.
______________________________ K. ______________________________
D. ______________________________ H.
______________________________ L. ______________________________

CHAPTER 7: TITLE BLOCKS

Exercise 7.4-1

Directions: Refer to the shaft project to answer the following questions.



PART NO.	NO. REQ'D	MAT'L	SPECIFICATIONS
①	1	CRS	∅ 4" X 5 5/8"LG MS10019287

TOLERANCES UNLESS SPECIFIED OTHERWISE
.XXX ± 0.010
.XX ± 0.02
.X ± 0.1
ANG ± 3.0

THIS DRAWING IN DESIGN AND DETAIL IS OUR PROPERTY, AND MUST NOT BE USED EXCEPT IN CONNECTION WITH OUR WORK. ALL RIGHTS OF DESIGN OR INVENTION ARE RESERVED.

KOHLER. CO.
KOHLER, WIS., U.S.A.

SUBJECT SHAFT PROJECT#4
YOUTH APPRENTICE
SAMPLE PROJECT
SERVICE MACHINE SHOP

MADE BY DJG
DATE FINISHED 10/01/2015
CHECKED BY PAL
APPROVED BY W. PLIER
CONTROL NO. 2014-09-0001

THIS BAR IS 2" LONG WHEN DRAWING IS PRINTED FULL SIZE 1 2

ACAD# 1001-1910-YA-L04_S01

REV.
NAME DATE
CONTROL NO.

NO MANUAL REVISIONS ARE TO BE MADE TO THIS DRAWING WITHOUT FIRST CONTACTING C.A.D. PERSONNEL.
TECHNICAL SERVICES

SCALE 1:1 Copyright ©2015 Kohler Co.
DRAWING NO. 1001-1910-YA-L04
SHEET NO. 01

1. What is the drawing number for the print?

____

2. By whom was the drawing checked?

____

3. What is the name of the company responsible for the drawing? ____________________________

4. What is the scale of the print?

____

5. Describe in words what the scale of this drawing means. ____________________________

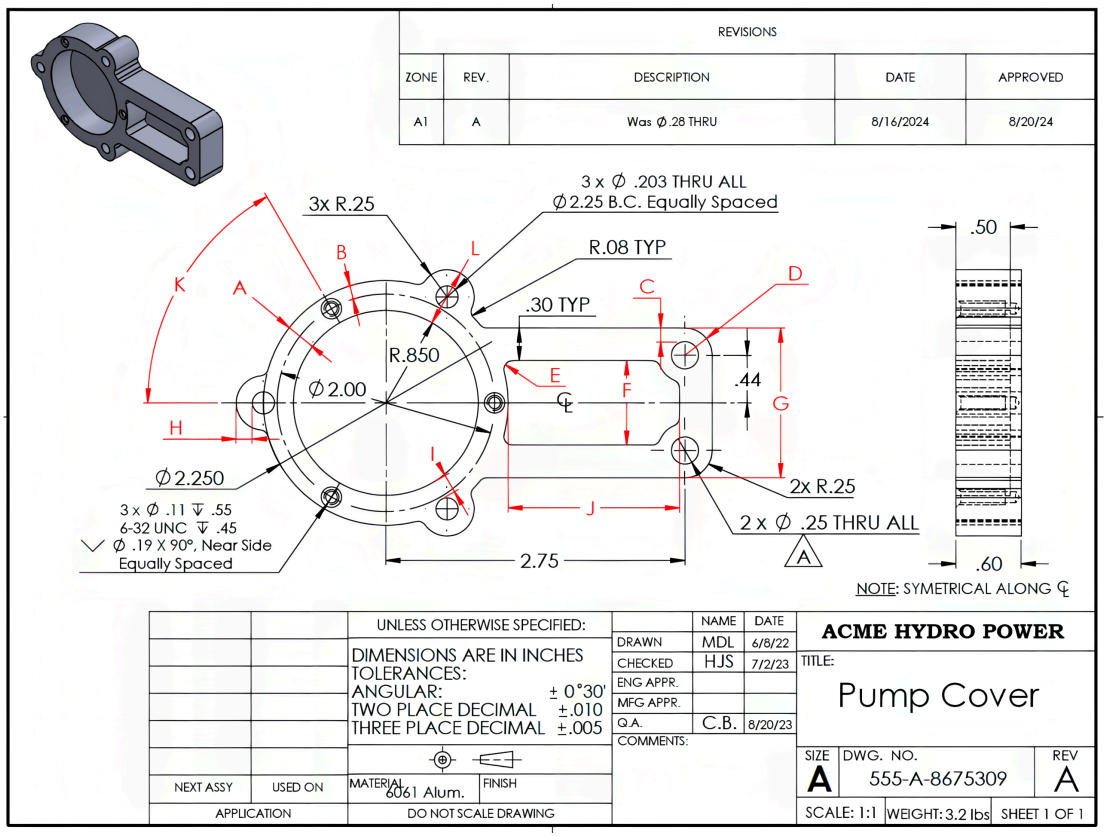
6. What material is required for this part?

____

7. What style of dimensioning is being used for the project? __________________________
8. What is the tolerance for two-place decimal dimensions? __________________________
9. What is the size tolerance of the ⌀2.000 dimension? __________________________
10. What is the size tolerance of the ⌀1.875 dimension? __________________________
11. How many pieces are required? __________________________
12. What is the size of the stock from which the project is made? __________________________

Exercise 7.4-2

Directions: Refer to the pump cover below to answer Questions 1 through 5 and identify the dimensions for letters A through L.



REVISIONS

ZONE	REV.	DESCRIPTION	DATE	APPROVED
A1	A	Was ∅ .28 THRU	8/16/2024	8/20/24

UNLESS OTHERWISE SPECIFIED:

DIMENSIONS ARE IN INCHES
TOLERANCES:
ANGULAR: ± 0°30'
TWO PLACE DECIMAL ±.010
THREE PLACE DECIMAL ±.005

	NAME	DATE
DRAWN	MDL	6/8/22
CHECKED	HJS	7/2/23
ENG APPR.		
MFG APPR.		
Q.A.	C.B.	8/20/23

COMMENTS:

NEXT ASSY
USED ON
APPLICATION
MATERIAL 6061 Alum.
FINISH
DO NOT SCALE DRAWING

ACME HYDRO POWER

TITLE: Pump Cover

SIZE A
DWG. NO. 555-A-8675309
REV A
SCALE: 1:1
WEIGHT: 3.2 lbs
SHEET 1 OF 1

1. What is the drawing number for the print?

____

2. What is the sheet size letter of the drawing?

____

3. What material is required for the pump cover?

____

4. What revision was made to the pump cover?

____

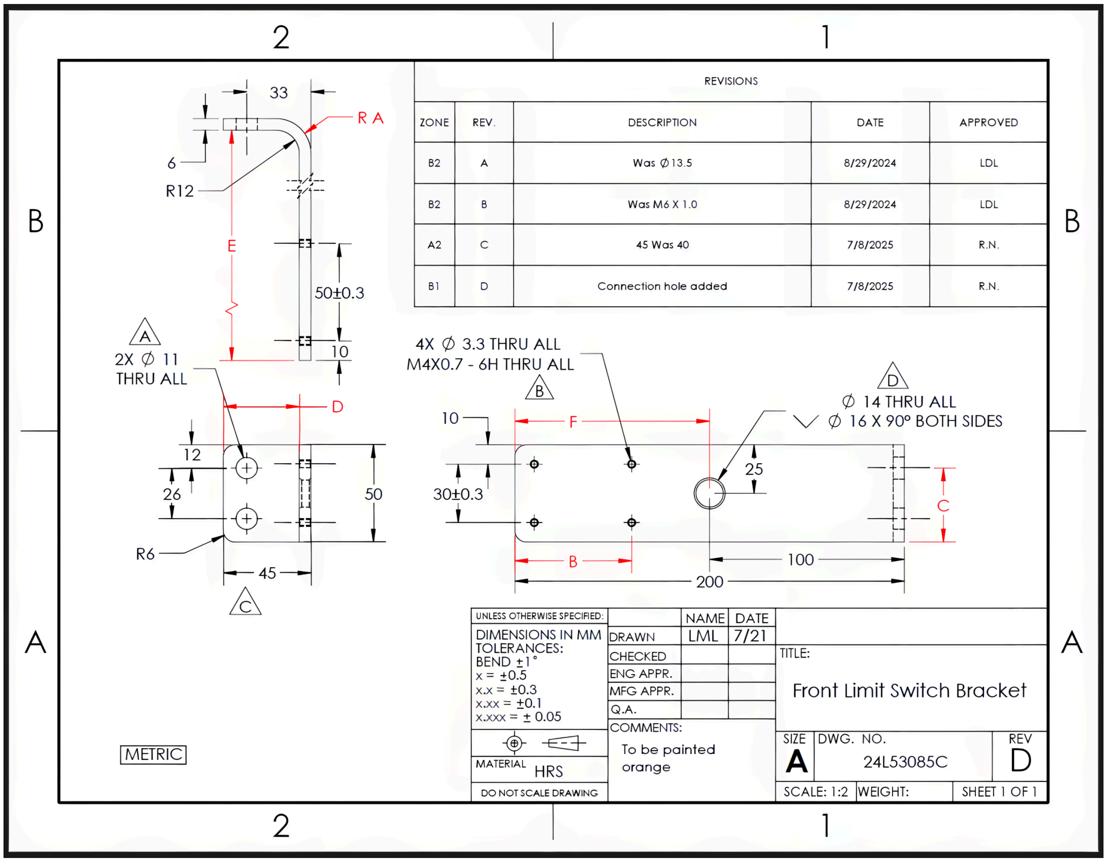
5. What is the overall length along the centerline?

____

A. ______________________________ E.
______________________________ I. ______________________________
B. ______________________________ F.
______________________________ J. ______________________________
C. ______________________________ G.
______________________________ K. ______________________________
D. ______________________________ H.
______________________________ L. ______________________________

Exercise 7.4-3

Directions: Answer the following questions for the front limit switch bracket drawing.



REVISIONS				
ZONE	REV.	DESCRIPTION	DATE	APPROVED
B2	A	Was ∅13.5	8/29/2024	LDL
B2	B	Was M6 X 1.0	8/29/2024	LDL
A2	C	45 Was 40	7/8/2025	R.N.
B1	D	Connection hole added	7/8/2025	R.N.

UNLESS OTHERWISE SPECIFIED:
DIMENSIONS IN MM
TOLERANCES:
BEND ±1°
x = ±0.5
x.x = ±0.3
x.xx = ±0.1
x.xxx = ± 0.05
MATERIAL HRS
DO NOT SCALE DRAWING

	NAME	DATE
DRAWN	LML	7/21
CHECKED		
ENG APPR.		
MFG APPR.		
Q.A.		

COMMENTS:
To be painted orange

TITLE:
Front Limit Switch Bracket

SIZE A
DWG. NO. 24L53085C
REV D
SCALE: 1:2
WEIGHT:
SHEET 1 OF 1

1. How many holes are on the bracket?

____

2. What system of measurement is used?

____

3. What size is the largest hole?

____

4. What size is the smallest hole?

____

5. How long is the bracket? ______________________

6. What is the material thickness?

____

7. What is the material thickness in thousandths of an inch? ________________________
8. How many revisions were made to the bracket? ________________________
9. What was revision C? ________________________
10. In which zone is revision C? ________________________
11. What type of break line is used? ________________________
12. What revision level is the drawing? ________________________

Calculate the following dimensions along with their accumulated tolerances.

A. ______________________________________

B. ______________________________________

C. ______________________________________

D. ______________________________________

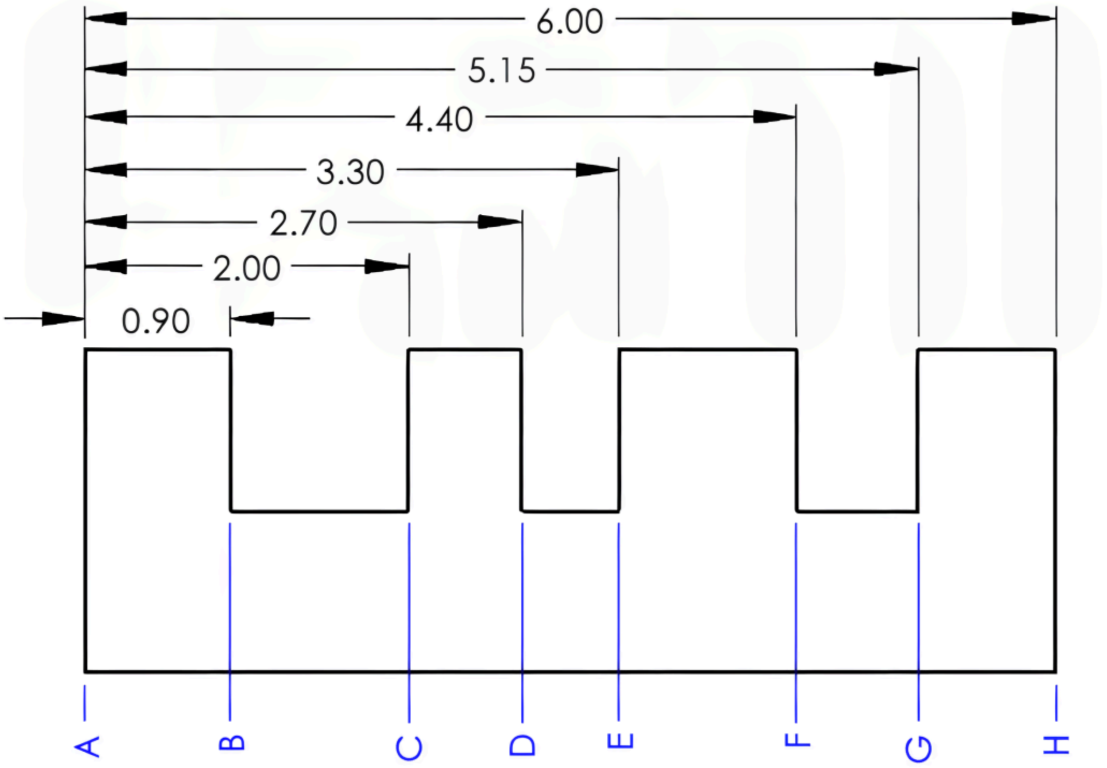
E. ______________________________________

F. ______________________________________

CHAPTER 8: TOLERANCE ON DIMENSIONS

Exercise 8.9-1

Directions: Calculate the distance and tolerance accumulations for the distances between the letters below.



XX = ±0.01

A to B = ________ ± ________

A to D = ________ ± _________

A to H = ________ ± _________

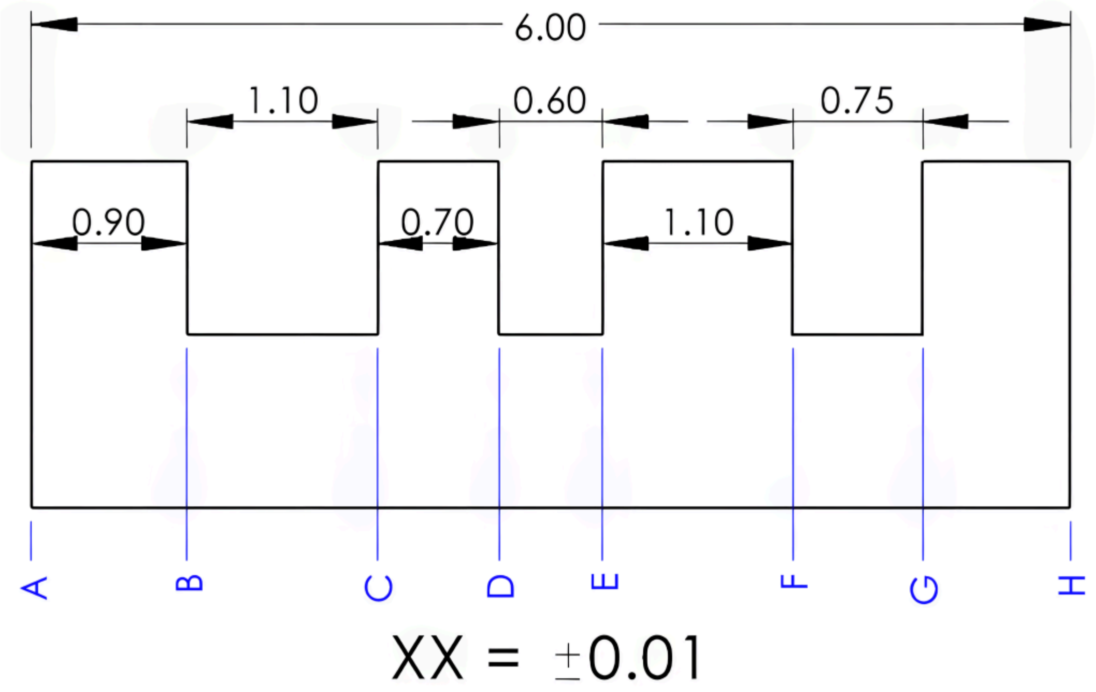
B to C = ________ ± _________

D to H = ________ ± _________

G to H = ________ ± _________

Exercise 8.9-2

Directions: Calculate the distance and tolerance accumulations for the distances between the letters below.



A to B = ________ ± ________

A to D = _______ ± ________

A to H = _______ ± ________

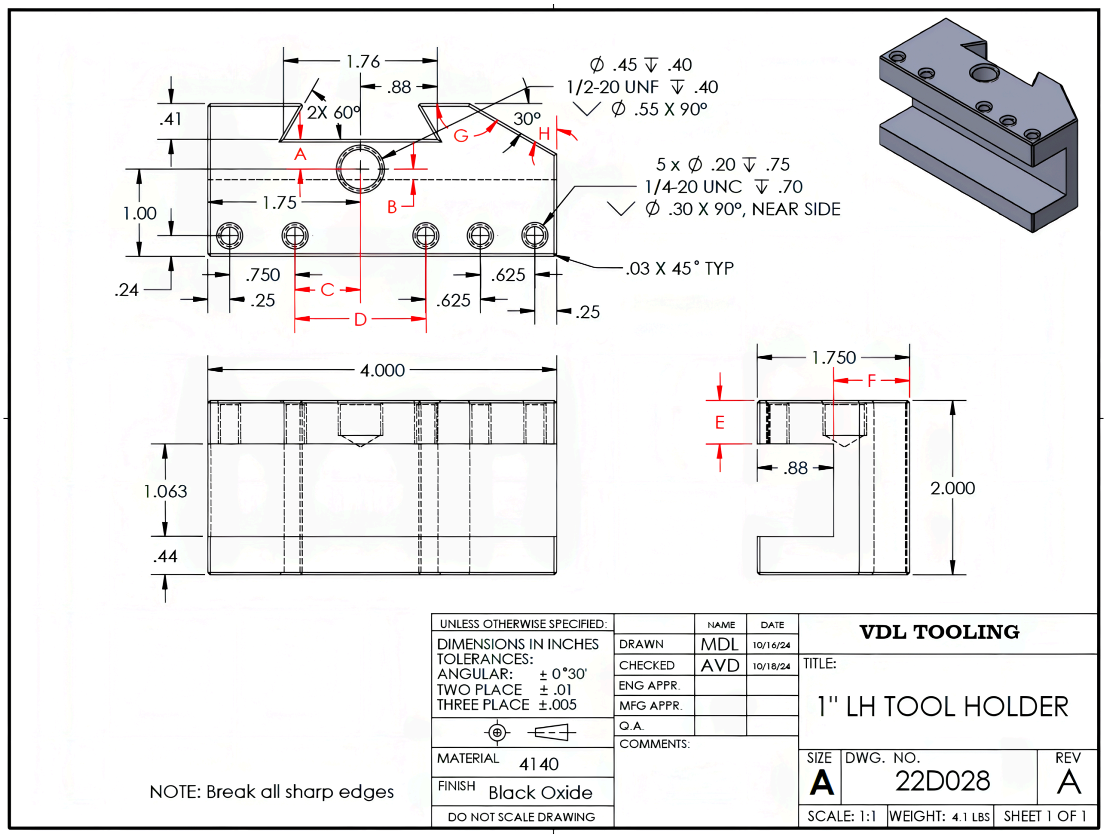
B to C = _______ ± ________

D to H = _______ ± ________

G to H = _______ ± ________

Exercise 8.9-3

Directions: Find the dimensions for the letters and the accumulated tolerances.



1. A) __________

2. How much tolerance accumulates for dimension A?

±__________

3. B) __________

4. How much tolerance accumulates for dimension B?

±__________

5. C) __________

6. How much tolerance accumulates for dimension C?

±__________

7. D) __________

8. How much tolerance accumulates for dimension D?

±__________

9. E) _________
10. How much tolerance accumulates for dimension E?

±_________

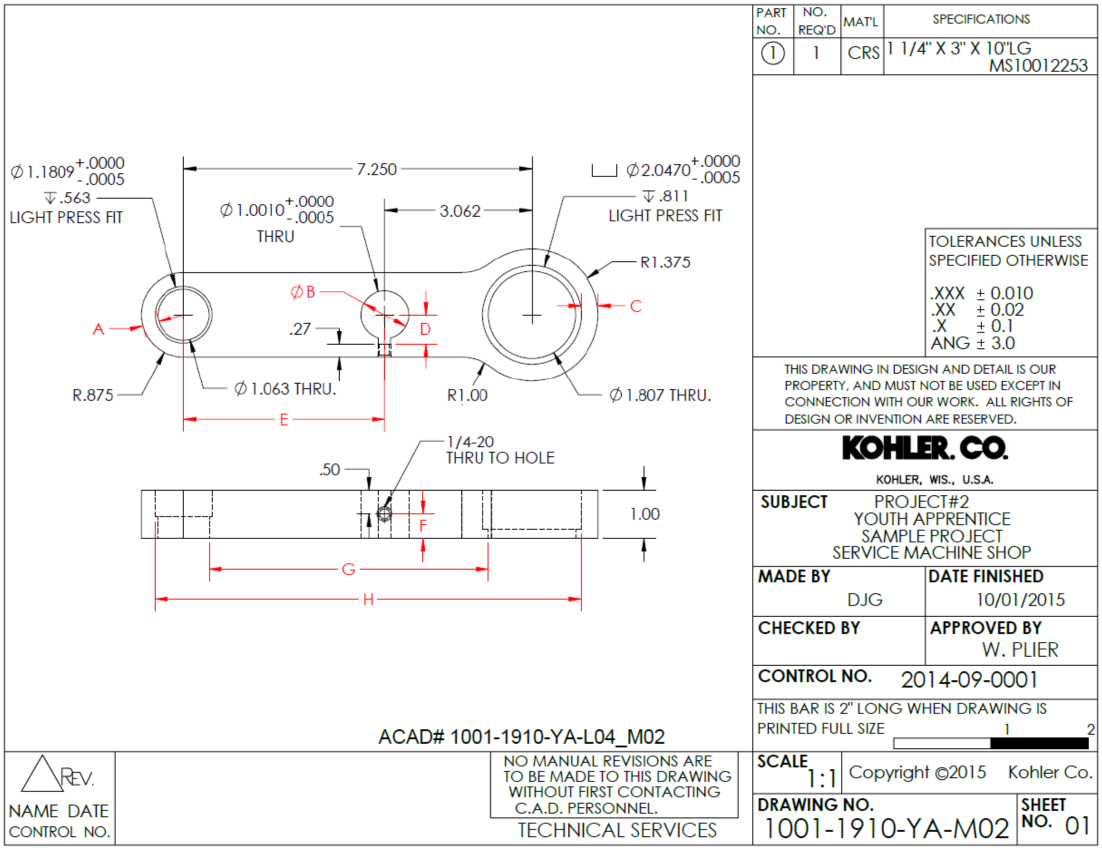
11. F) _________
12. How much tolerance accumulates for dimension F?

±_________

13. G) _________°
14. H) _________°

Exercise 8.9-4

Directions: Answer Questions 1 through 6 for the drawing.



1. How much tolerance applies to the thickness of the part? ______________________________
2. Is the thickness tolerance limit, bilateral or unilateral tolerancing? ______________________________
3. What is the MMC of the 1.1809 diameter?

   ____
4. What is the LMC of the 2.047 diameter?

   ____
5. Are the 1.1809 and 2.047 diameters limit, bilateral or unilateral tolerancing? ______________________________
6. What is the maximum overall length of the part,

including tolerances? ________________________

Use all tolerance values to find the maximum and minimum values for the matching letters.

A. MAX ______________________________________ MIN ______________________________________

B. MAX ______________________________________ MIN ______________________________________

C. MAX ______________________________________ MIN ______________________________________

D. MAX ______________________________________ MIN ______________________________________

E. MAX ______________________________________ MIN ______________________________________

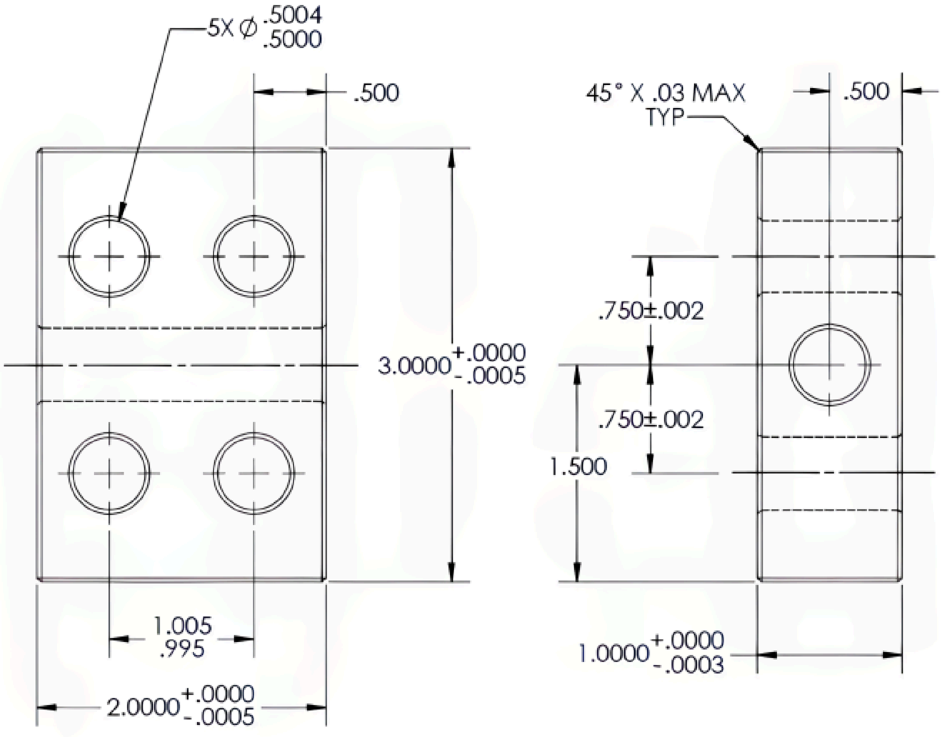
F. MAX ______________________________________ MIN ______________________________________

G. MAX ______________________________________ MIN ______________________________________

H. MAX ______________________________________ MIN ______________________________________

Exercise 8.9-5

Directions: Answer Questions 1 through 10 for the drawing.



TOLERANCES (EXCEPT AS NOTED)	DRAWN BY: GTA		
DIMENSIONS IN INCHES TWO PLACE DECIMAL ±.01 THREE PLACE DECIMAL ±.005 ANGULAR ±1/2°	PART NAME: Inspection Block		
	DWG. NO. 102030		
	DATE 12/15/25		
	SCALE 1:1	MATERIAL A2	SIZE A

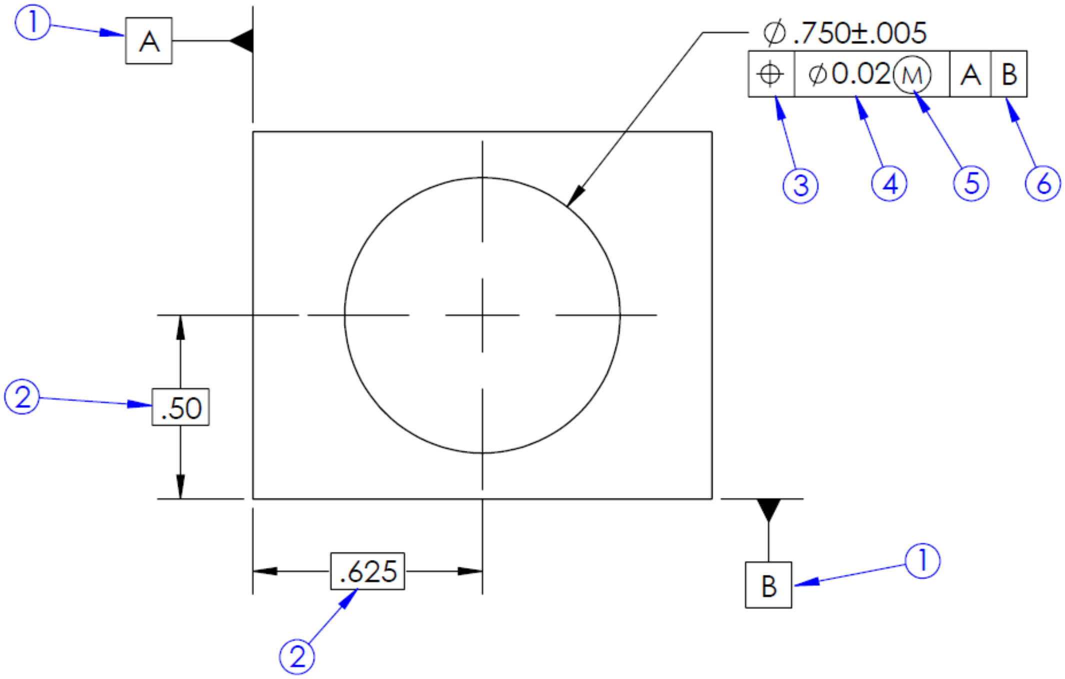
1. Does the drawing contain any single-limit dimensions? ___________________________
2. What type of tolerance is applied to the 5 holes? ________________________________
3. What type of tolerance is applied to the .750 dimensions? _________________________
4. What type of tolerance is applied to the hole spacing on the front view? _____________
5. What type of tolerance is applied to the 2.0000 dimension? ________________________
6. Express the hole size dimension as an equal bilateral dimension. ___________________
7. Calculate the minimum dimension between the centers of the top and bottom holes. ______________________
8. Calculate the maximum dimension between the centers of the top and bottom holes. ______________________
9. Calculate the minimum distance between the top of the block and the center of the top row of holes. _____________________
10. Calculate the maximum distance between the top of the block and the center of the top row of holes. _____________________

CHAPTER 9: PRINT SYMBOLS AND

NOTES

Exercise 9.6-1

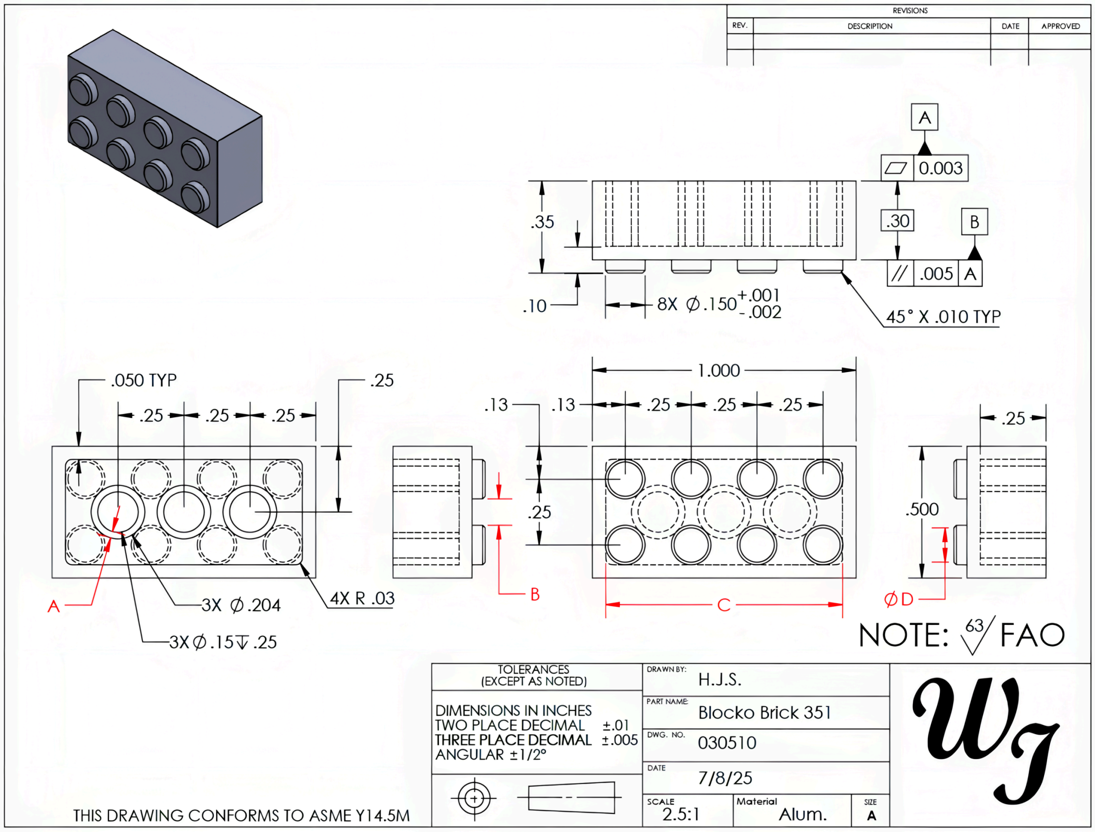
Directions: Match the correct term to the identifying number.



1. ________________ GD&T Symbol
2. ________________ Tolerance
3. ________________ Modifier
4. ________________ Datum
5. ________________ Datum Reference
6. ________________ Basic Dimension

Exercise 9.6-2

Directions: Answer the questions below referencing this block print.



1. How many geometric tolerances are identified?

   ____

2. How many separate datums are identified?

   ____

3. What surface finish is required? ______________
4. What is the surface finish value in decimal form?

____

5. Is the surface finish note a general or local note? ________________
6. What does FAO abbreviate? ________________
7. What is the geometric characteristic symbol on the datum A surface? ________________
8. What is the geometric characteristic symbol on the datum B surface? ________________
9. What is the value of the basic dimension? ________________
10. What is the scale of the drawing? ________________

Calculate the following dimensions along with their accumulated tolerances.

A. MAX____________MIN_____________

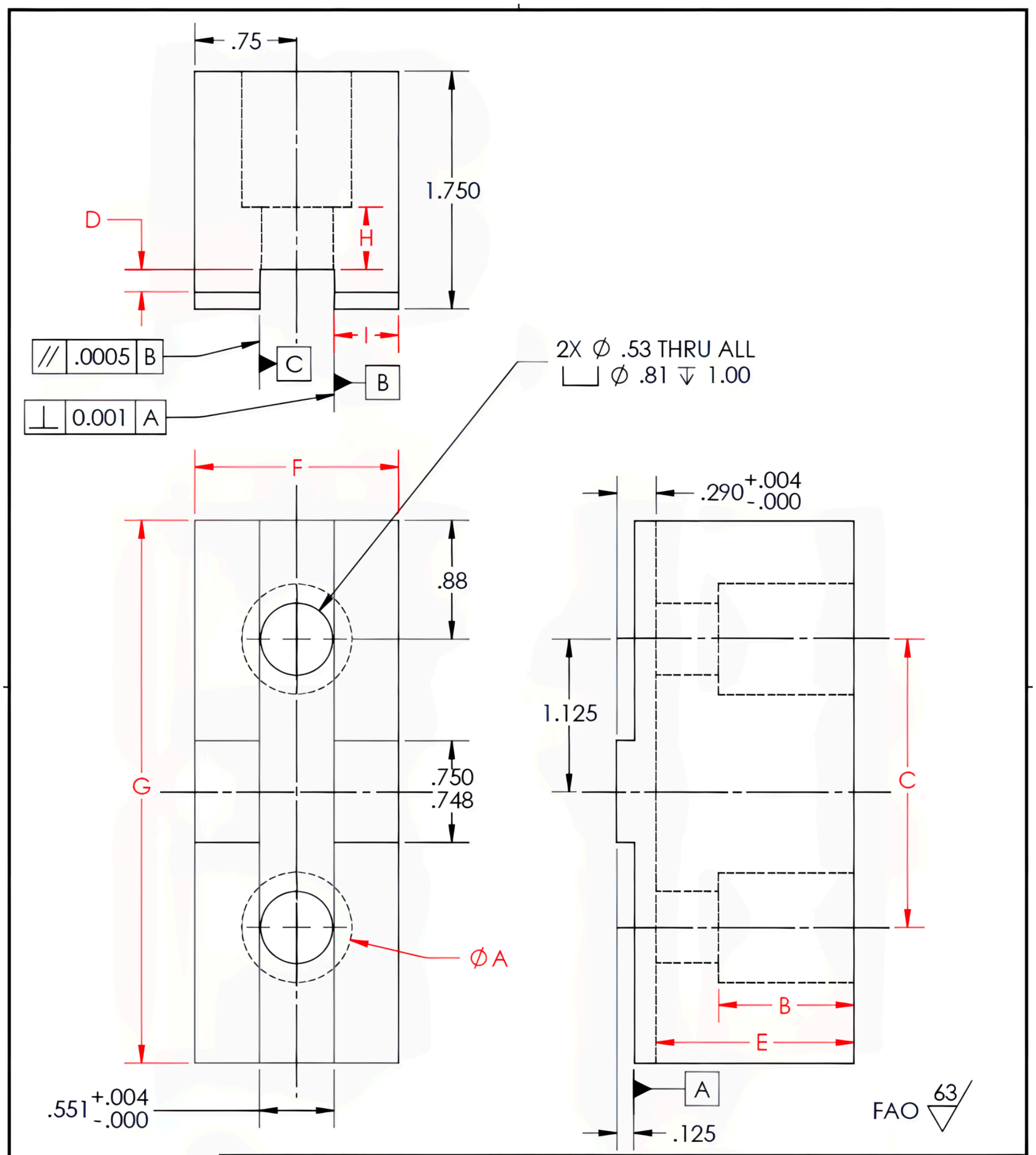
B. MAX____________MIN_____________

C. MAX____________MIN_____________

D. MAX____________MIN_____________

Exercise 9.6-3

Directions: Answer the following questions for the soft jaw drawing.



TOLERANCES (EXCEPT AS NOTED)	DRAWN BY:	LDS		
DIMENSIONS IN INCHES TWO PLACE DECIMAL ±.01 THREE PLACE DECIMAL ±.005 ANGULAR ±1/2°	PART NAME:	Lathe Soft Jaw		
	DWG. NO.	24L53013C		
	DATE	12/25		
	SCALE FULL	MATERIAL 1018 HRS	SIZE **A**	

3 PIECES REQD

1. How many datum features are identified? ________________
2. Is machining required to obtain the surface finish? ________________
3. What is the MMC of the ⌀.53 holes? ________________
4. How many feature control frames are being used? ________________
5. Which dimension is in limit style? ________________
6. Which dimension(s) is unilateral? ________________
7. What is the surface finish value in decimal form? ________________
8. What size fasteners are intended for the .53 diameter holes? ________________
9. How many parts are required? ________________
10. Are the GD&T symbols tolerances of size, location, orientation, or form? ________________
11. What is the LMC of the .551 slot? ________________

Calculate the following dimensions, along with their accumulated tolerances.

A. ____________

B. ____________

C. ____________

D. ____________

E. MAX____________MIN____________

F. MAX____________MIN_____________

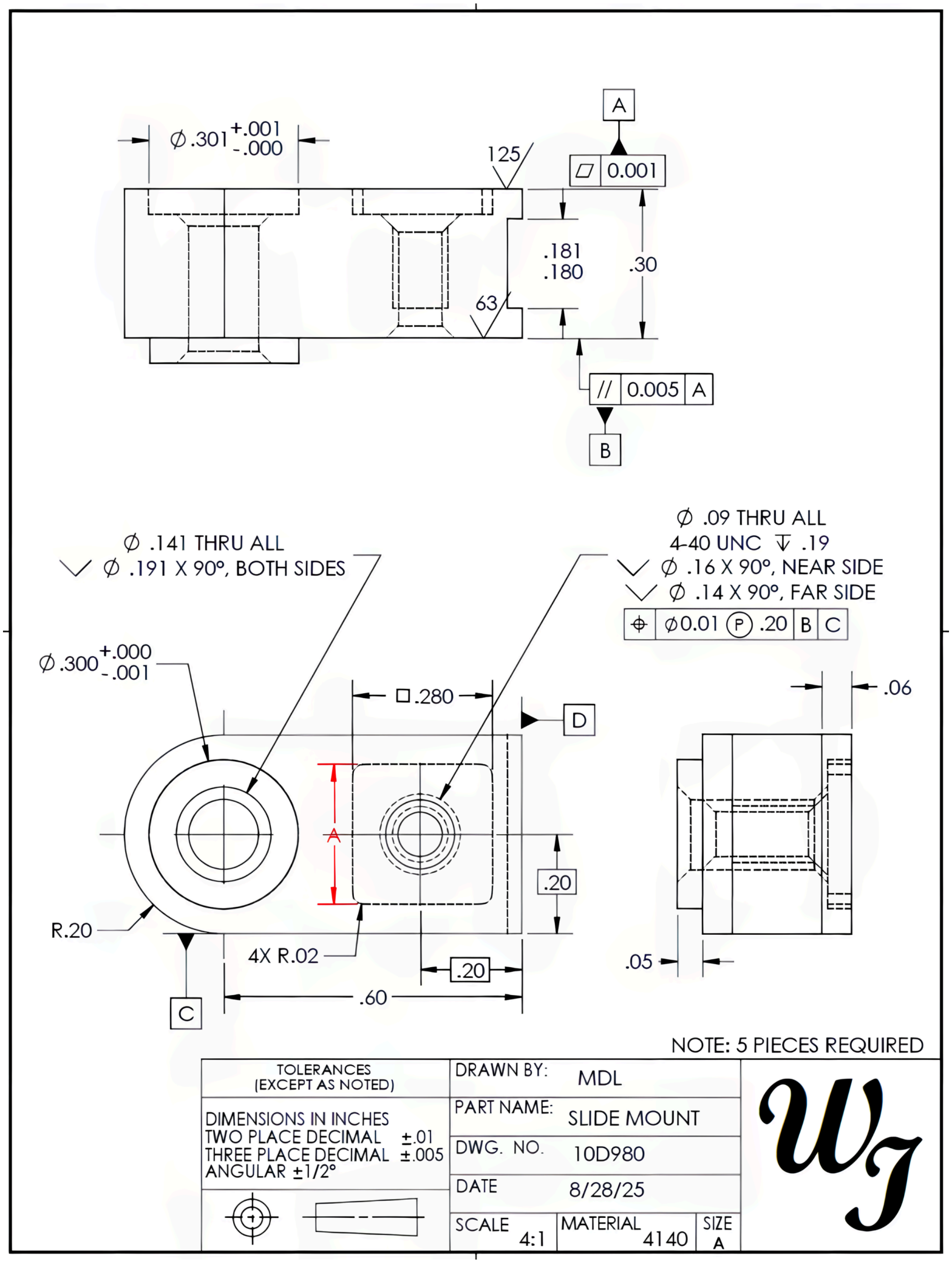
G. MAX____________MIN_____________

H. MAX____________MIN_____________

I. MAX____________MIN_____________

Exercise 9.6-4

Directions: Answer the questions below referencing this slide block print.



1. What value is identified as a basic dimension?

____

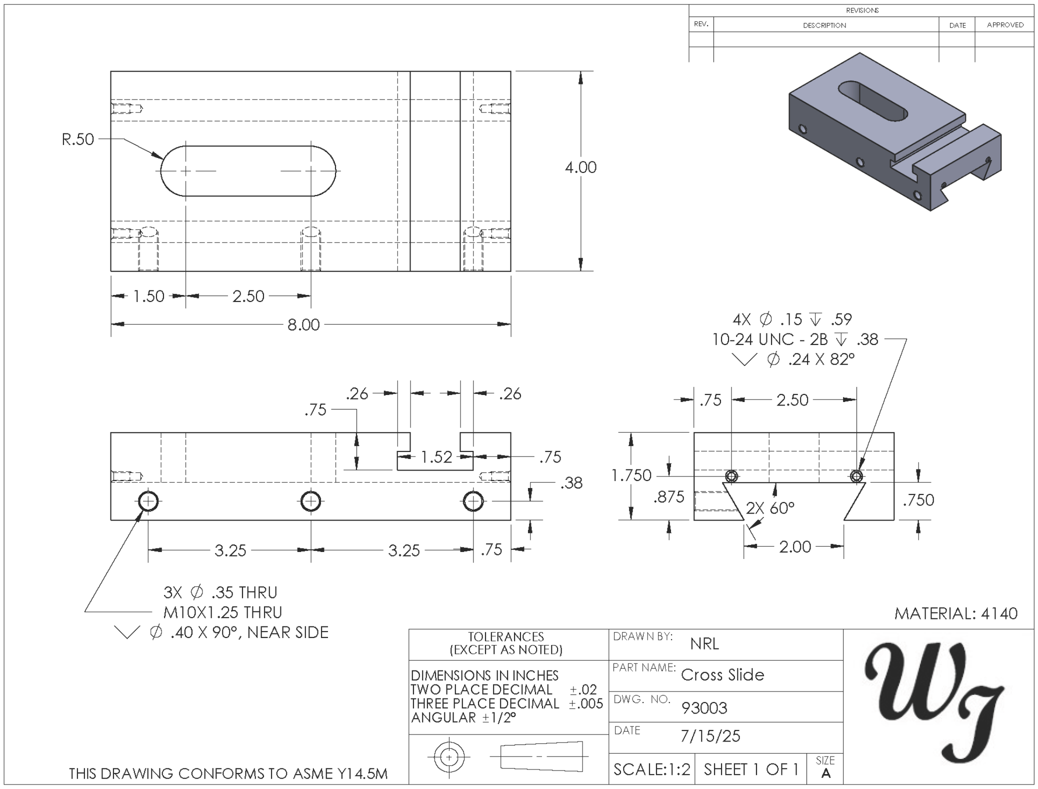
2. How many geometric tolerances are identified? ________________
3. How many separate datums are identified? ________________
4. How many pieces are required? ________________
5. Is the number of pieces required note a general or local note? ________________
6. How many countersink dimensions are identified? ________________
7. What surface finish values are identified? ________________
8. What are the surface finish values in decimal form? ________________
9. What is the geometric characteristic symbol on the datum A surface? ________________
10. At what height is the positional tolerance to be measured? ________________
11. If two pieces were to fit together at the .30 diameters, what is the minimum amount of clearance? ________________
12. What is the maximum amount of clearance between the .300 and .301 diameters? ________________
13. What type of tolerance is the 0.181/0.180 value? ________________
14. What type of tolerance does the ⌀0.300 value have? ________________
15. What is the depth of the tapped hole? ________________
16. What is the value of dimension “A”? ________________
17. What is the scale of the

drawing?______________________________

CHAPTER 10: MACHINING DETAILS

Exercise 10.10-1

Directions: Answer the questions below referencing the cross slide print.



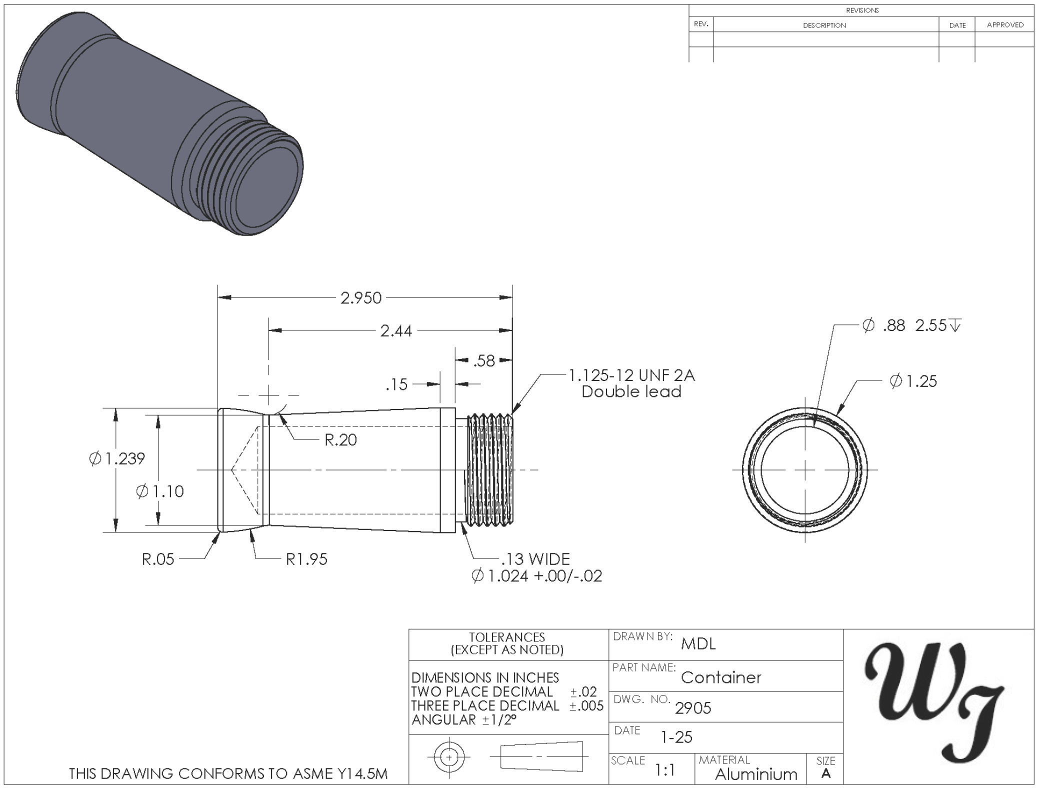
1. Not including tolerance, what is the depth or the T-slot? ________________
2. Not including tolerance, what is the width across the top of the T-slot? ________________
3. What is the minimum width across the top of the T-slot? ________________
4. What is the maximum depth of the dovetail slot? ________________
5. What is the minimum width across the narrow end of the dovetail slot? ________________
6. What is the maximum width of the slotted hole?

____

7. What is the maximum overall length of the slotted hole? ________________
8. How many threads per inch on the 4 threaded holes? ________________
9. What is the pitch of the 4 threaded holes? (round to three decimal places) ________________
10. What does the abbreviation UNC stand for? ________________
11. What class of fit is designated for the 4 threaded holes? ________________
12. What is the nominal major diameter of the 3 threaded holes in mm? ________________
13. What is the nominal major diameter of the 3 threaded holes in decimal inches? (round to two decimal places) ________________
14. Which dimensioning style is used for the spacing of the 3 threaded holes on the front view, datum, chain, or broken-chain? ________________

Exercise 10.10-2

Directions: Answer the questions below referencing the container print.



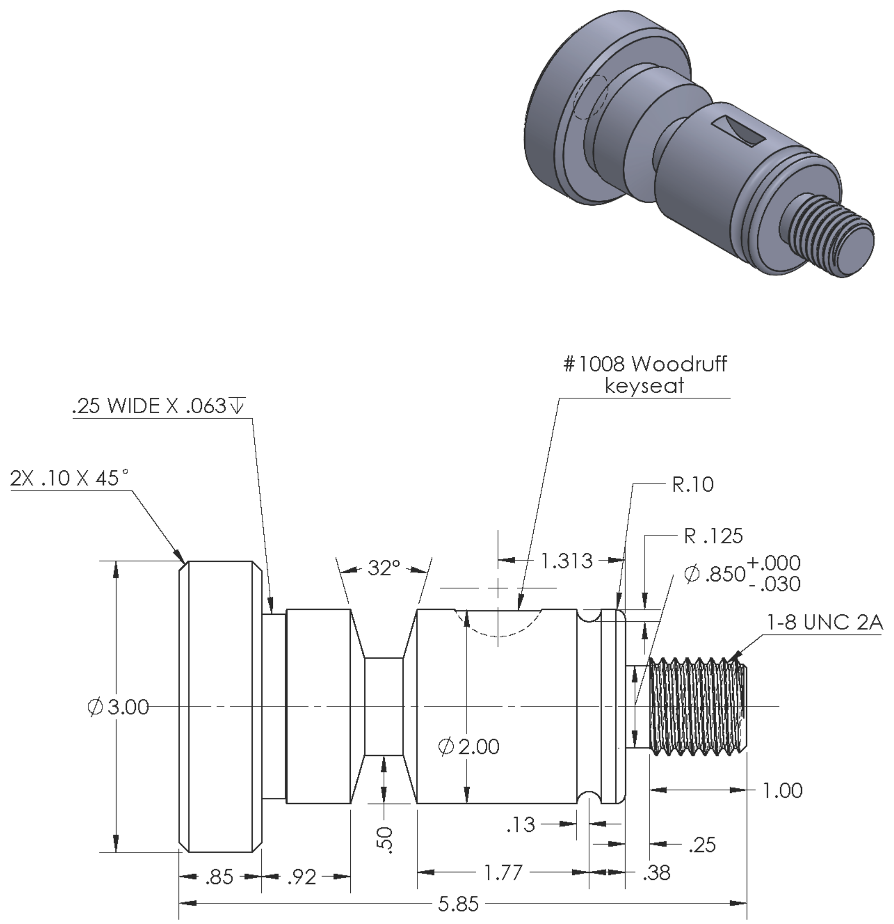
1. What is the minimum depth of the ⌀ .88 hole? ________________
2. What is the maximum diameter of the largest outside diameter of the container? ________________
3. What is the width of the neck, not including tolerance? ________________
4. What is the minimum diameter of the neck? ________________
5. What is the maximum diameter of the neck? ________________
6. What is the major diameter of the thread?

____

7. How many of the threads will there be in a length of ½ inch? ________________
8. To the nearest three-place decimal, what is the pitch of the thread? ________________
9. Is the thread a single, double, or triple start thread? ________________
10. To the nearest three-place decimal, what is the lead of the thread?________________

Exercise 10.10-3

Directions: Answer the questions below referencing the seal pin print and illustration.



TOLERANCES
(EXCEPT AS NOTED)

DIMENSIONS IN INCHES
TWO PLACE DECIMAL ±.01
THREE PLACE DECIMAL ±.005
ANGULAR ±1/2°

DRAWN BY: MDL

PART NAME: Seal Pin

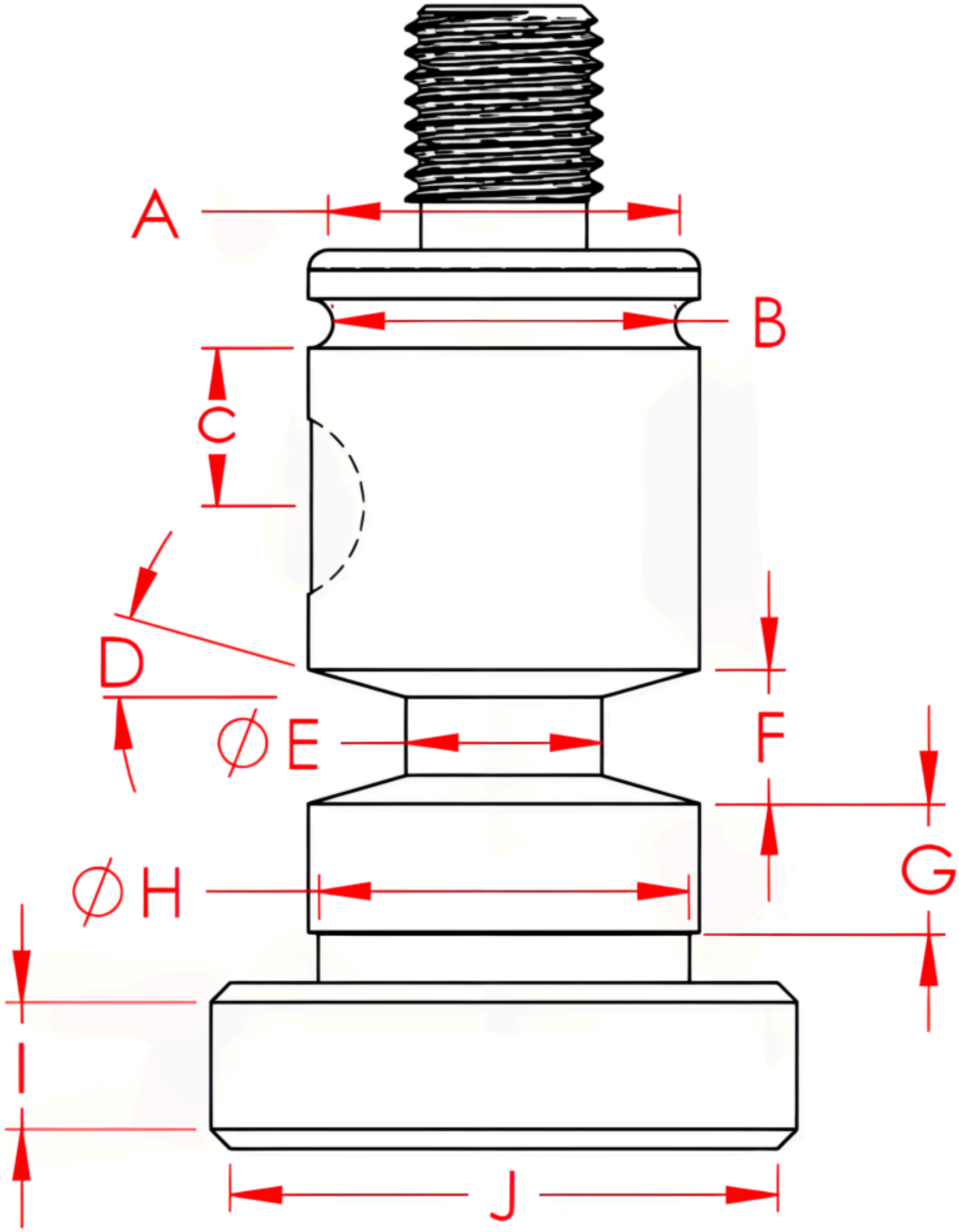
DWG. NO. 10G975

DATE 7/25

SCALE NTS

MATERIAL Brass

SIZE A



1. What is the MMC of the ⌀2.00 dimension?

____

2. What is the value of dimension A, not including

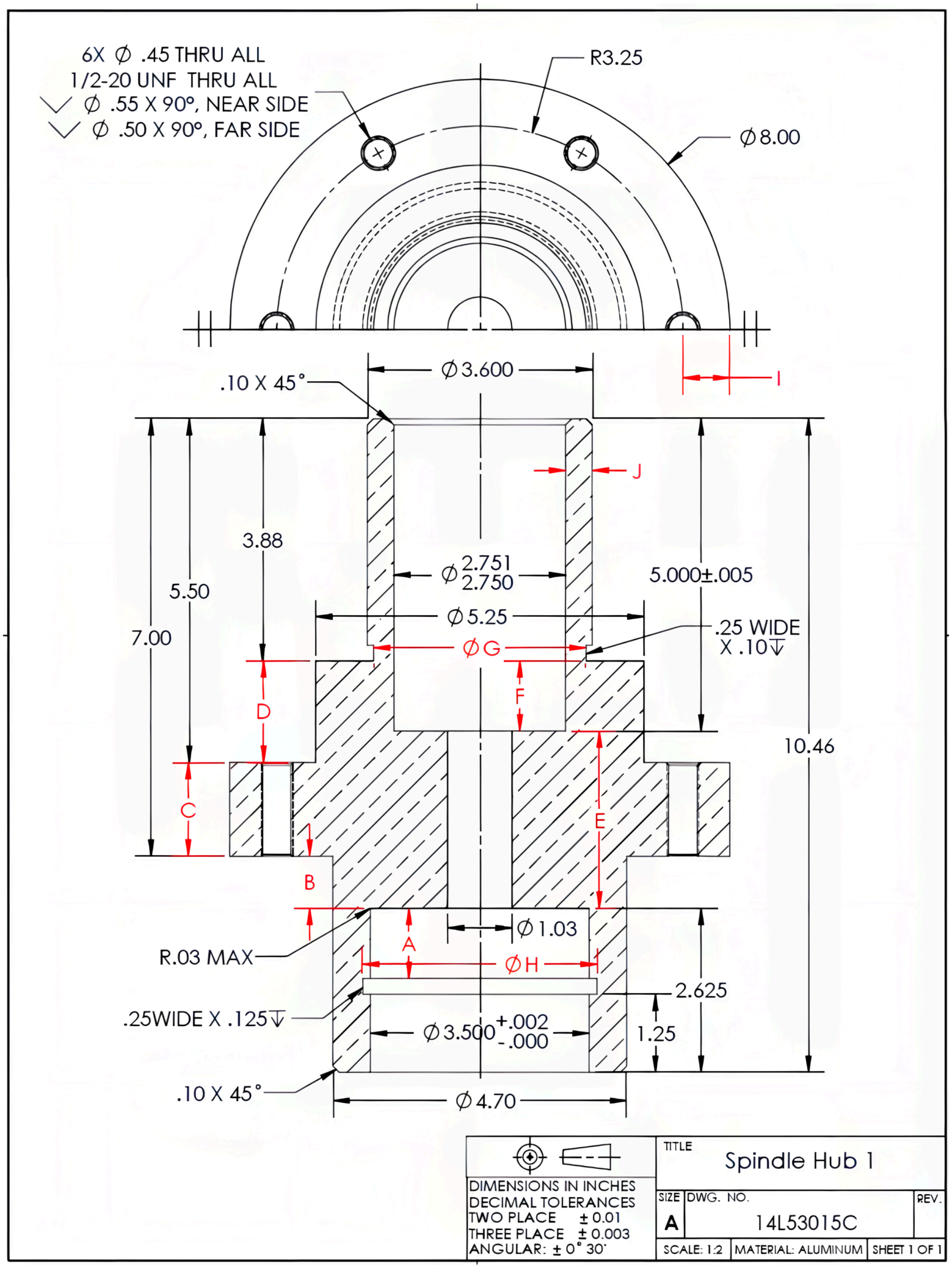
tolerance? ________________

3. What is the value of dimension B, not including tolerance? ________________
4. What is the minimum value of dimension B? ________________
5. What is the value of dimension C, not including tolerance? ________________
6. What is the diameter of the Woodruff keyseat? ________________
7. What is the width of the Woodruff keyseat? ________________
8. What is the angle of dimension D? ________________
9. What is the diameter of dimension E, not including tolerance? ________________
10. Using the linear length dimensions, what is the value of dimension F, not including tolerance? ________________
11. What is the tolerance accumulation of dimension F? ________________
12. What is the value of dimension G, not including tolerance? ________________
13. What is the diameter of dimension H? ________________
14. What is the maximum diameter of H? ________________
15. What is the value of dimension I, not including tolerance? ________________
16. What is the value of dimension J, not including tolerance? ________________
17. What is the maximum value of J? ________________

CHAPTER 11 – SECTION VIEWS

Exercise 11.8-1

Directions: Answer the following questions for the spindle hub drawing.



DIMENSIONS IN INCHES
DECIMAL TOLERANCES
TWO PLACE ± 0.01
THREE PLACE ± 0.003
ANGULAR: ± 0° 30'

TITLE
Spindle Hub 1

SIZE A
DWG. NO. 14L53015C
REV.

SCALE: 1:2
MATERIAL: ALUMINUM
SHEET 1 OF 1

1. What do the parallel lines on the bottom of the top view indicate? ________________
2. What type of sectional view is used? ______________________________________
3. Is the drawing half the size or twice the size of the actual part? _________________
4. Do the section lines match the style of the material used? ________________
5. Which dimension is in limit style? ____
6. Which dimension is unilateral? ____
7. Which dimension is equal bilateral? ______________________________________
8. Which dimension is an example of a single limit? ____________________________

Calculate the following dimensions along with their accumulated tolerances.

A = _______ ± _______

B = ______ ± _______

C = ______ ± _______

D = ______ ± _______

E = ______ ± _______

F = ______ ± _______

G = ______ ± _______

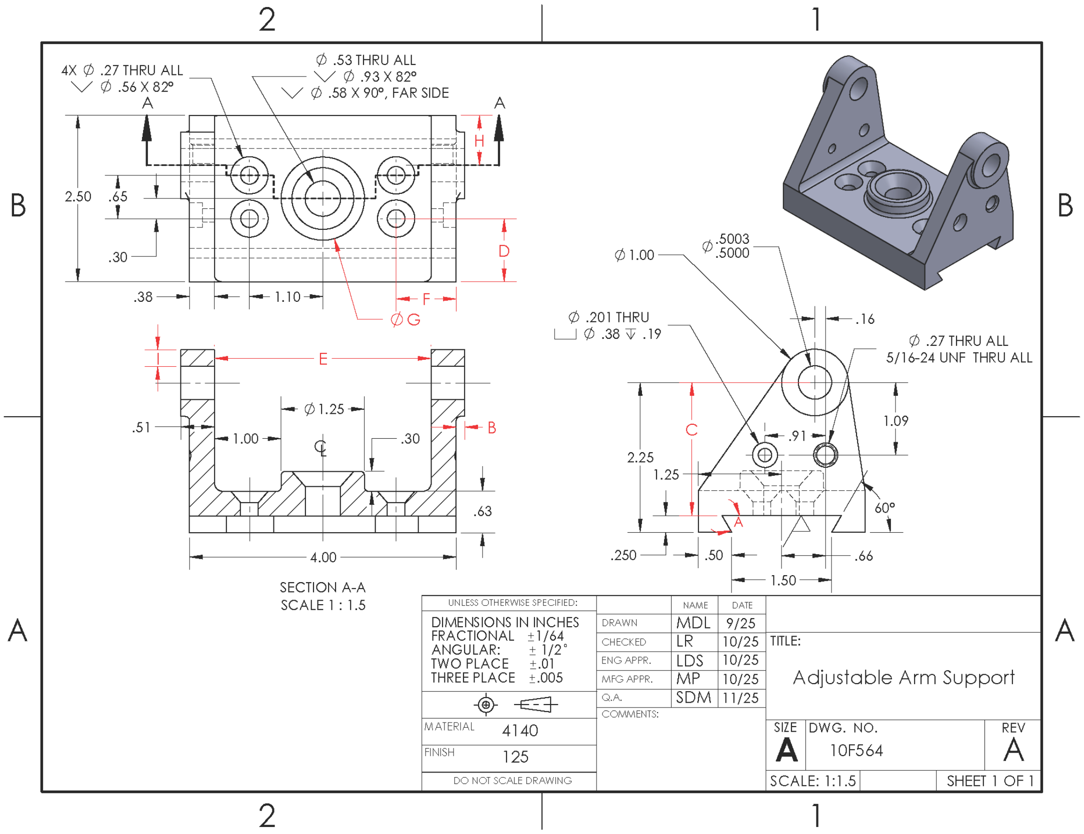
H. MAX ___________ MIN ___________

I. MAX ____________ MIN ____________

J. MAX ____________ MIN ____________

Exercise 11.8-2

Directions: Using the adjustable arm support print, answer Questions 1 through 7 and find the dimensions for letters A through I.



1. What type of sectional view is used? ______________________________________
2. What is the sheet size letter of the drawing? _________________________________
3. Which view is replaced with the sectional view? ______________________________
4. What is the counterbore depth? ____
5. What is the LMC of the .5000 hole? _______________________________________
6. What is the overall height of the support? ___________________________________
7. Does the surface finish symbol indicate that material removal is required? ___________

A = ________

B = ________

C = ________

D = ________

E = ________

F = ________

G = ________

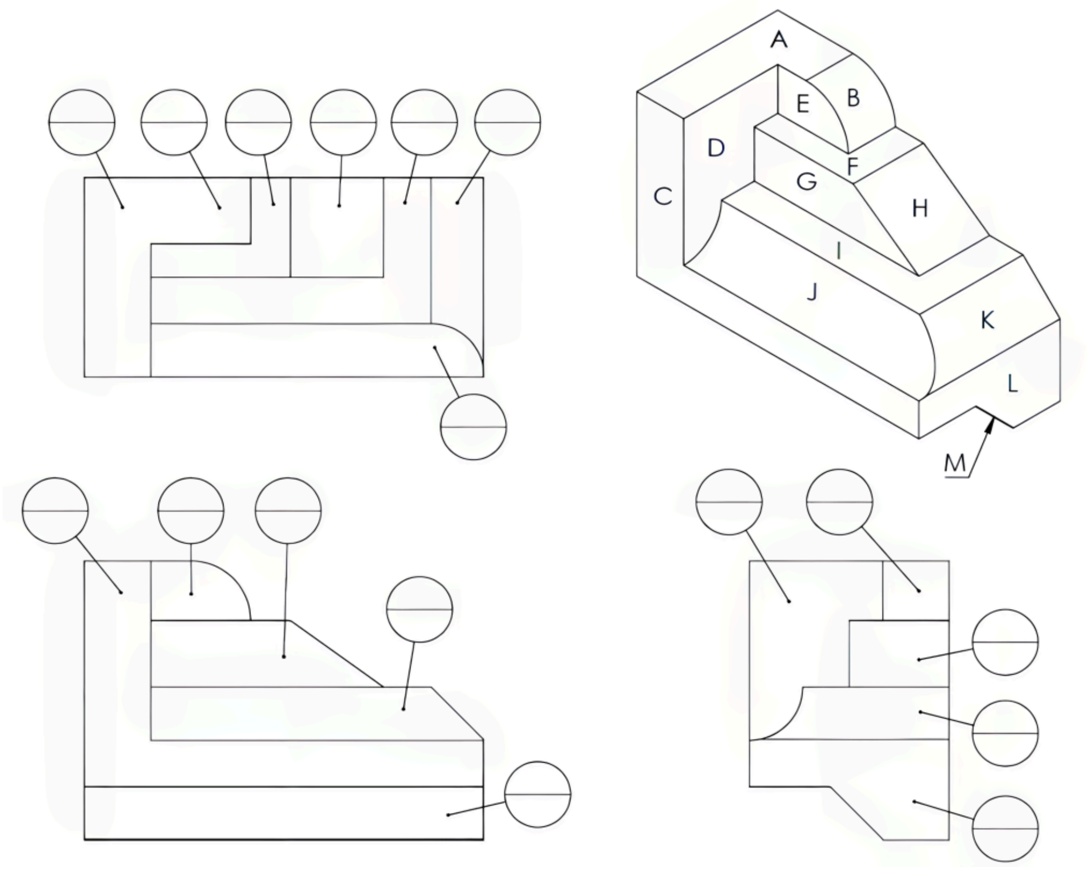
H = ________

I. MAX____________MIN____________

CHAPTER 12 – AUXILIARY VIEWS

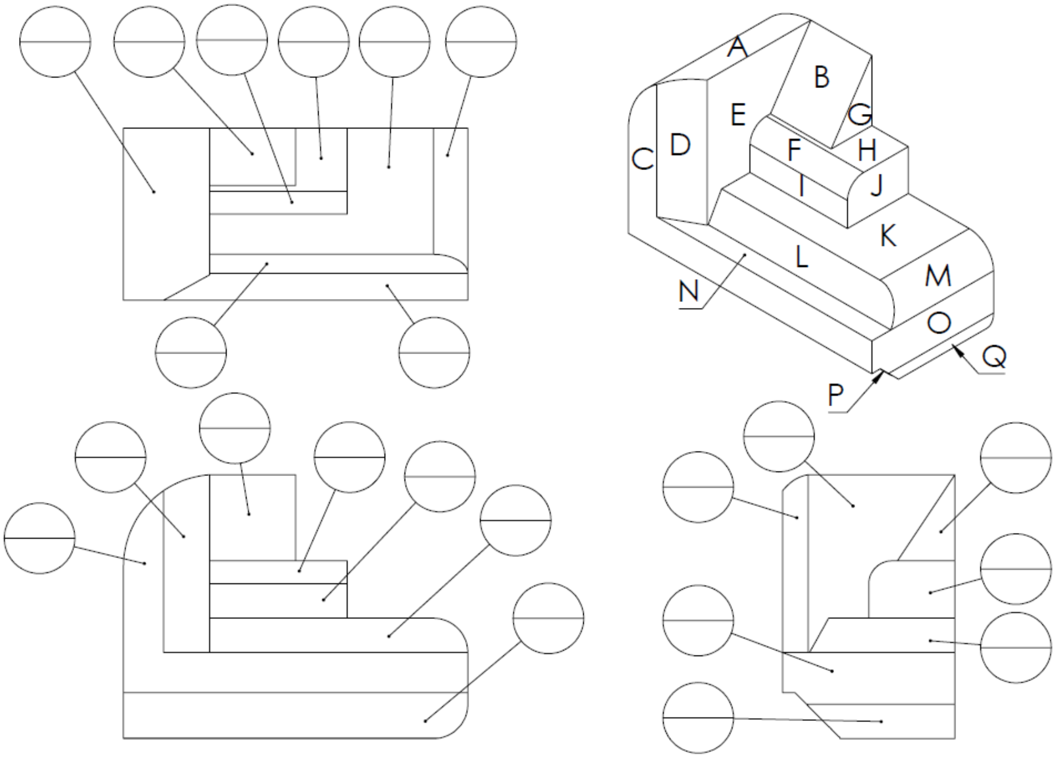
Exercise 12.3-1

Directions: Enter the letters from the isometric drawing below into the top of the correct balloons and indicate whether it is a true shape view with a "T," an inclined surface with an "I," or a curved surface with a "C" in the bottom of the balloon on the orthographic view.



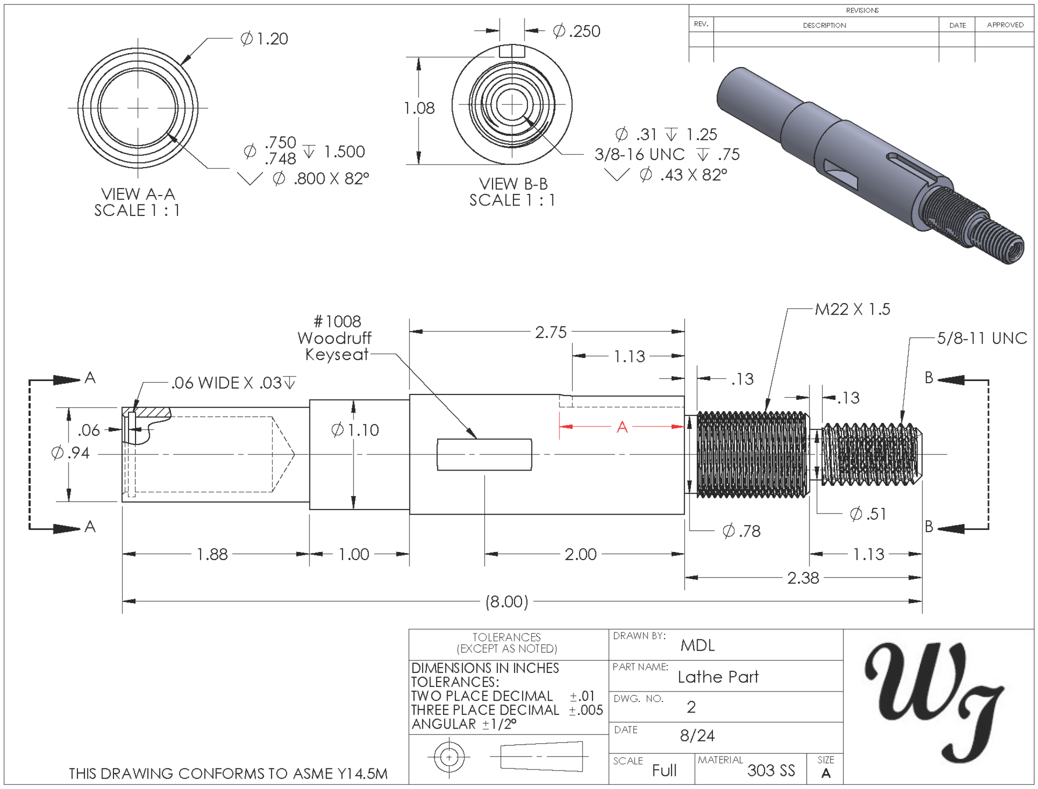
Exercise 12.3-2

Directions: Enter the letters from the isometric drawing below into the top of the correct balloons and indicate whether it is a true shape view with a “T,” an inclined surface with an “I,” or a curved surface with a “C” in the bottom of the balloon on the orthographic view.



Exercise 12.4-1

Directions: Answer the following questions for the lathe part drawing.



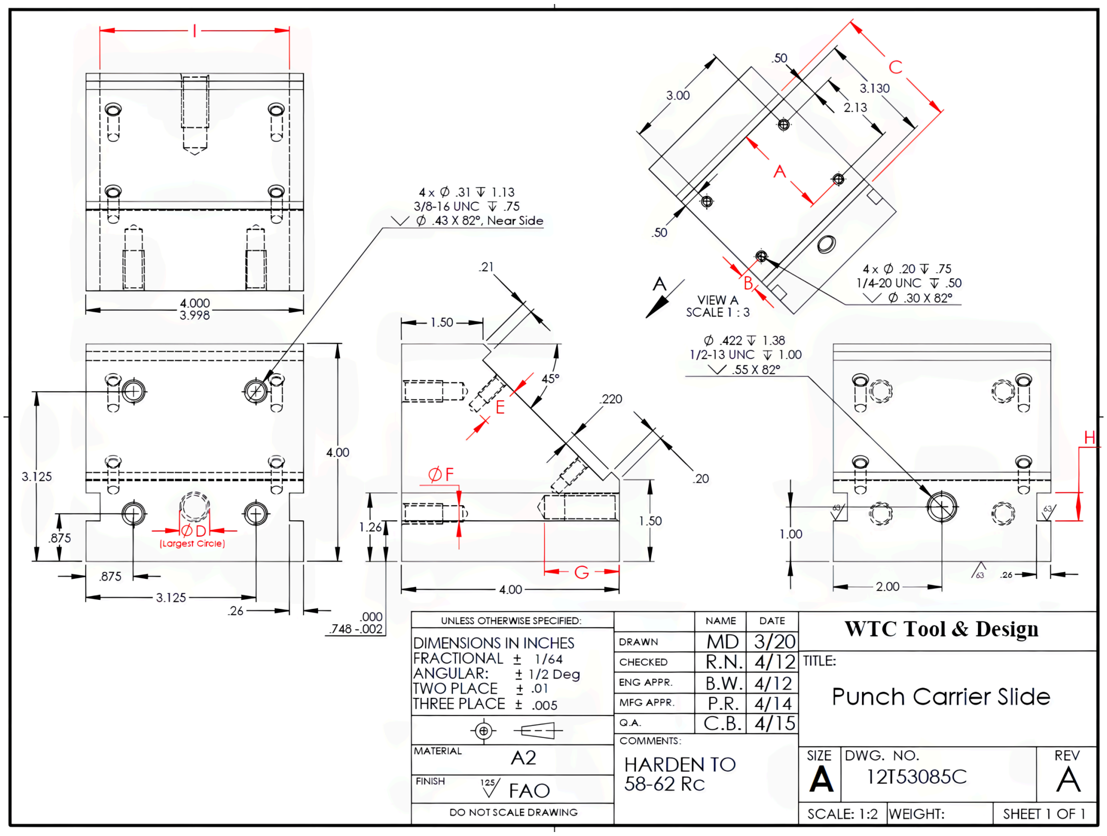
1. What type of view replaces the primary views on the right and left? ________________
2. What is the nominal major diameter of the internal thread? ________________
3. What is the scale of the drawing? ________________
4. Not including tolerance, what is the depth of the straight keyway from the outside diameter? ________________
5. Which type of view is used to dimension the internal

groove? ________________

6. Which dimension is a reference dimension? ________________
7. What is the pitch of the metric thread? ________________
8. What are the threads per inch of the ⅝ thread? ________________
9. Rounded to three decimal places, what is the pitch of the ⅝ thread? ________________
10. Rounded to four decimal places, what is the pitch of the internal thread? ________________
11. What is the fit classification of the internal thread? ________________
12. What is the largest size diameter allowed for the small neck diameter? ________________
13. Including tolerance, what is the maximum diameter of the internal groove? ________________
14. Not including tolerance, what is the length of dimension A? ________________
15. What is the maximum overall length of the part? ________________

Exercise 12.4-2

Directions: Answer the following questions for the punch carrier slide drawing.



1. How many threads per inch is the largest threaded hole? ________________
2. What is the pitch of the ¼-20 threaded hole? (two decimal places) ________________
3. What is the scale of the auxiliary view?

____

4. Which primary view is the auxiliary view projected from? ________________
5. Which dimension is in limit style?
6. What is the MMC of the .748 dimension?

____

7. Which view shows the dimensional locations of the

¼-20 holes? ________________

8. What is the maximum depth of the .220 dimension? ________________
9. What is the smoothest surface finish required on the slide? ________________
10. Not including hidden surfaces, how many of the views contain a foreshortened surface? ________________

Calculate the following dimensions along with their accumulated tolerances.

A. ___________ ± ___________

B. ___________ ± ___________

C. ___________ ± ___________

D. ___________ ± ___________

E. ___________ ± ___________

F. ___________ ± ___________

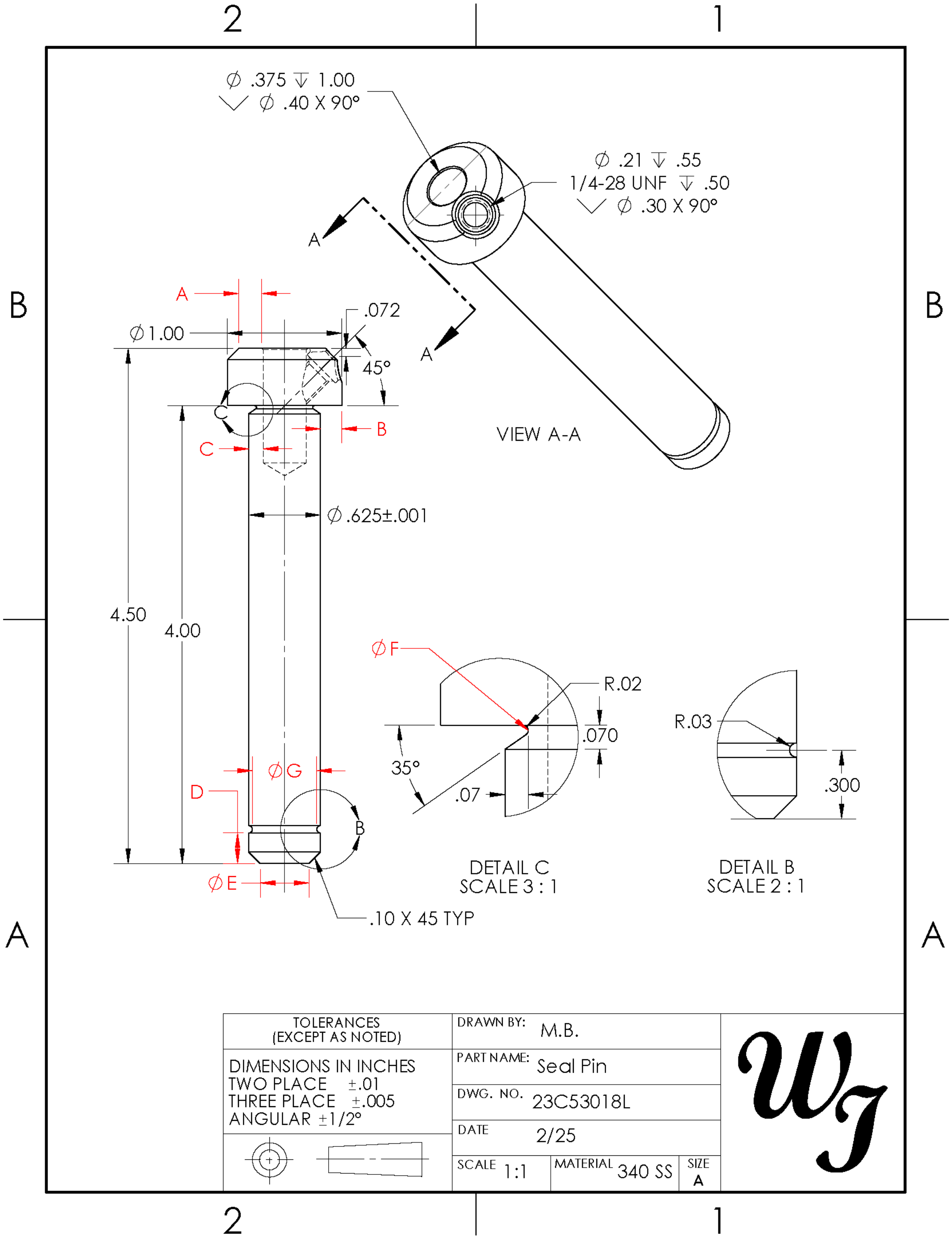
G. ___________ ± ___________

H. MAX ___________ MIN ___________

I. MAX ___________ MIN ___________

Exercise 12.6-1

Directions: Answer the following questions for the seal pin drawing.



TOLERANCES (EXCEPT AS NOTED)

DIMENSIONS IN INCHES
TWO PLACE ±.01
THREE PLACE ±.005
ANGULAR ±1/2°

DRAWN BY: M.B.
PART NAME: Seal Pin
DWG. NO. 23C53018L
DATE 2/25
SCALE 1:1
MATERIAL 340 SS
SIZE A

1. What type of material is used for the seal pin?

____

2. What type of view are details B and C? ________________
3. What is the scale of detail C? ________________
4. What surface or feature is the auxiliary view projected from? ________________
5. Is the ⌀.625 tolerance a unilateral, equal bilateral, or limit style? ________________

A. ____________

B. ____________ ± ____________

C. ____________ ± ____________

D. ____________ ± ____________

E. ____________ ± ____________

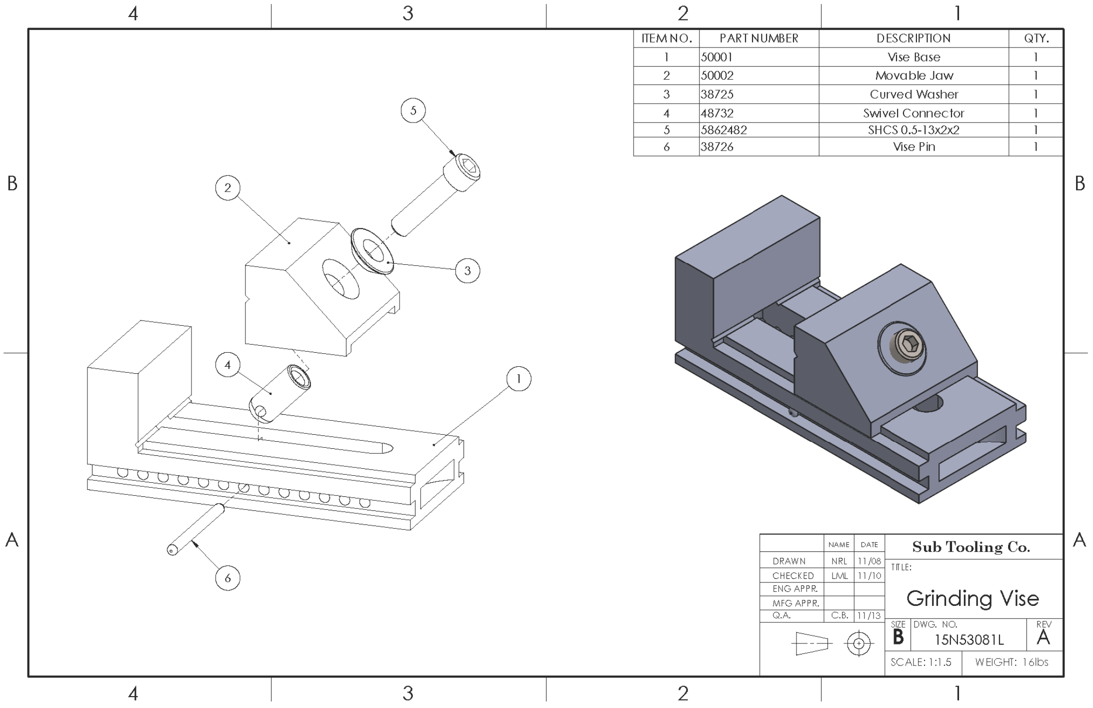
F. ____________ ± ____________

G. ____________ ± ____________

CHAPTER 13 – ASSEMBLY PRINTS

Exercise 13.5-1

Directions: Refer to the grinding vise drawing to answer the following questions.



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	50001	Vise Base	1
2	50002	Movable Jaw	1
3	38725	Curved Washer	1
4	48732	Swivel Connector	1
5	5862482	SHCS 0.5-13x2x2	1
6	38726	Vise Pin	1

	NAME	DATE
DRAWN	NRL	11/08
CHECKED	LML	11/10
ENG APPR.		
MFG APPR.		
Q.A.	C.B.	11/13

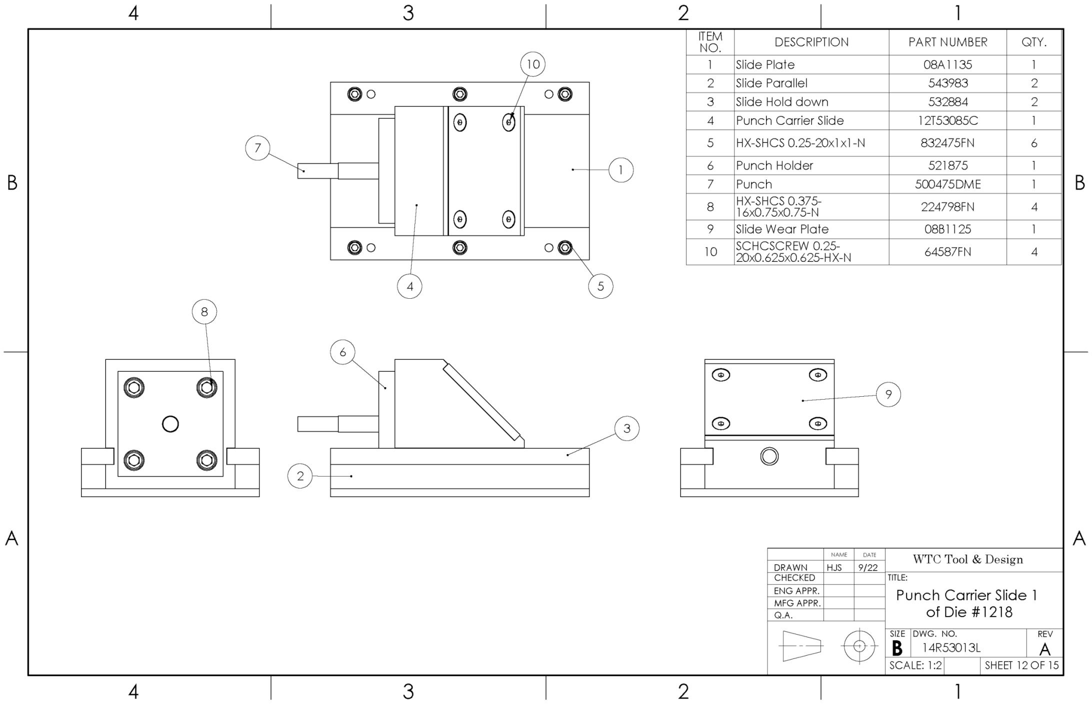
Sub Tooling Co.
TITLE:
Grinding Vise
SIZE B
DWG. NO. 15N53081L
REV A
SCALE: 1:1.5
WEIGHT: 16lbs

1. What sheet size is used for this drawing?
   ____
2. How many parts are used in the vise assembly?
   ____
3. What is the description of item 5? ________________
4. What type of assembly view is used to identify the parts? ________________
5. How are items 4 and 5 assembled? ________________
6. What is the scale of the assembly drawing?
   ____
7. What is the part number of item number 6?
   ____

8. What is the revision level of the drawing?________________

Exercise 13.5-2

Directions: Refer to the punch carrier slide 1 drawing to answer the following questions.



ITEM NO.	DESCRIPTION	PART NUMBER	QTY.
1	Slide Plate	08A1135	1
2	Slide Parallel	543983	2
3	Slide Hold down	532884	2
4	Punch Carrier Slide	12T53085C	1
5	HX-SHCS 0.25-20x1x1-N	832475FN	6
6	Punch Holder	521875	1
7	Punch	500475DME	1
8	HX-SHCS 0.375-16x0.75x0.75-N	224798FN	4
9	Slide Wear Plate	08B1125	1
10	SCHCSCREW 0.25-20x0.625x0.625-HX-N	64587FN	4

	NAME	DATE
DRAWN	HJS	9/22
CHECKED		
ENG APPR.		
MFG APPR.		
Q.A.		

WTC Tool & Design
TITLE:
Punch Carrier Slide 1
of Die #1218
SIZE B
DWG. NO. 14R53013L
REV A
SCALE: 1:2
SHEET 12 OF 15

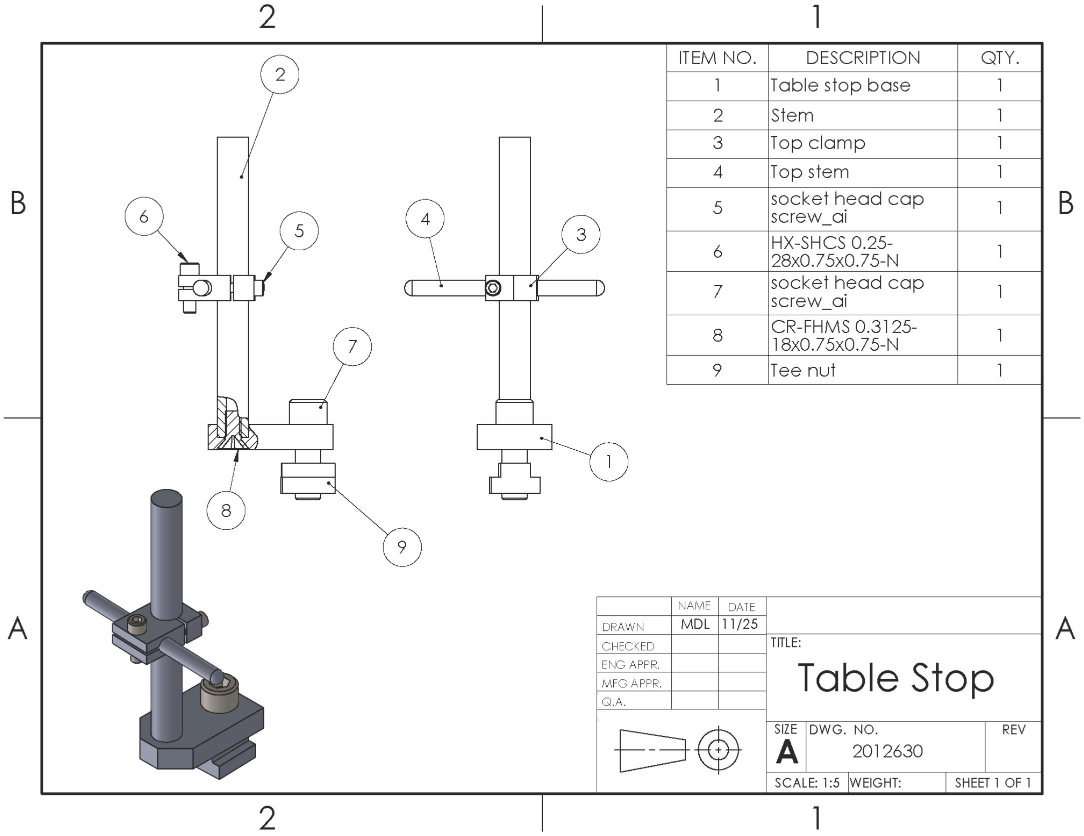
1. What are the dimensions of the sheet used for this drawing? ________________
2. How many total parts are used in the assembly?

____

3. What is the description of item 9? ________________
4. Does the assembly use a multi-view or pictorial view? ________________
5. How many threaded fasteners are used? ________________
6. What is the scale of the assembly drawing? ________________
7. What is the part number of item number 10? ________________
8. Is the assembly half the size or twice the size on the drawing? ________________

Exercise 13.5-3

Directions: Refer to the table stop drawing to answer the following questions.



ITEM NO.	DESCRIPTION	QTY.
1	Table stop base	1
2	Stem	1
3	Top clamp	1
4	Top stem	1
5	socket head cap screw_ai	1
6	HX-SHCS 0.25-28x0.75x0.75-N	1
7	socket head cap screw_ai	1
8	CR-FHMS 0.3125-18x0.75x0.75-N	1
9	Tee nut	1

1. What sheet size is used for this drawing? ________________
2. How many items are used in the table stop assembly? ________________
3. What is the description of item 3? ________________
4. What type of section view is used? ________________
5. What is the scale of the assembly drawing? ________________
6. What are the dimensions of the sheet used for this drawing? ________________
7. What is the description of item 9? ________________

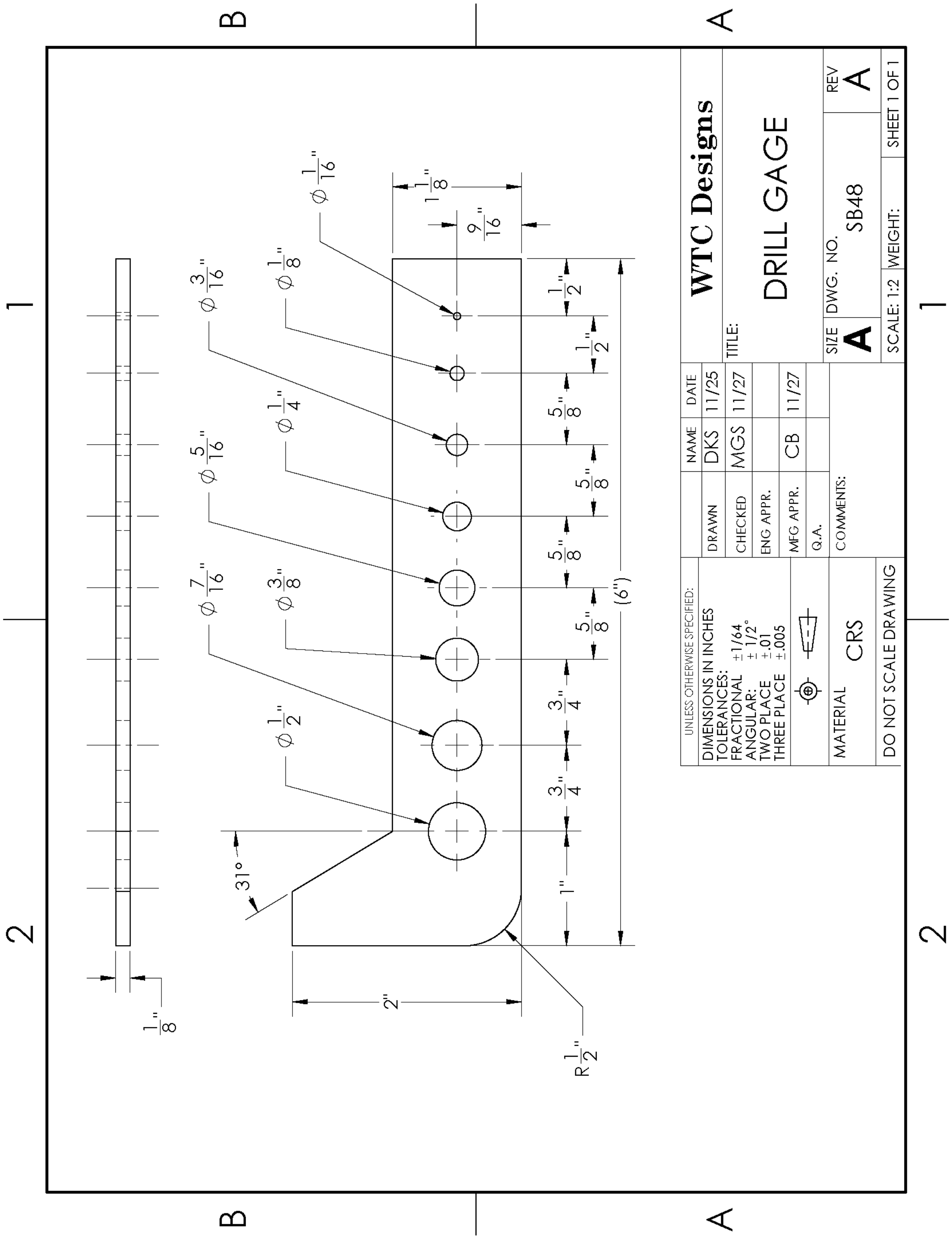
8. How many threaded fasteners are used?

____

CHAPTER 14 – PRINT SPECIFICATIONS

Exercise 14.5-1

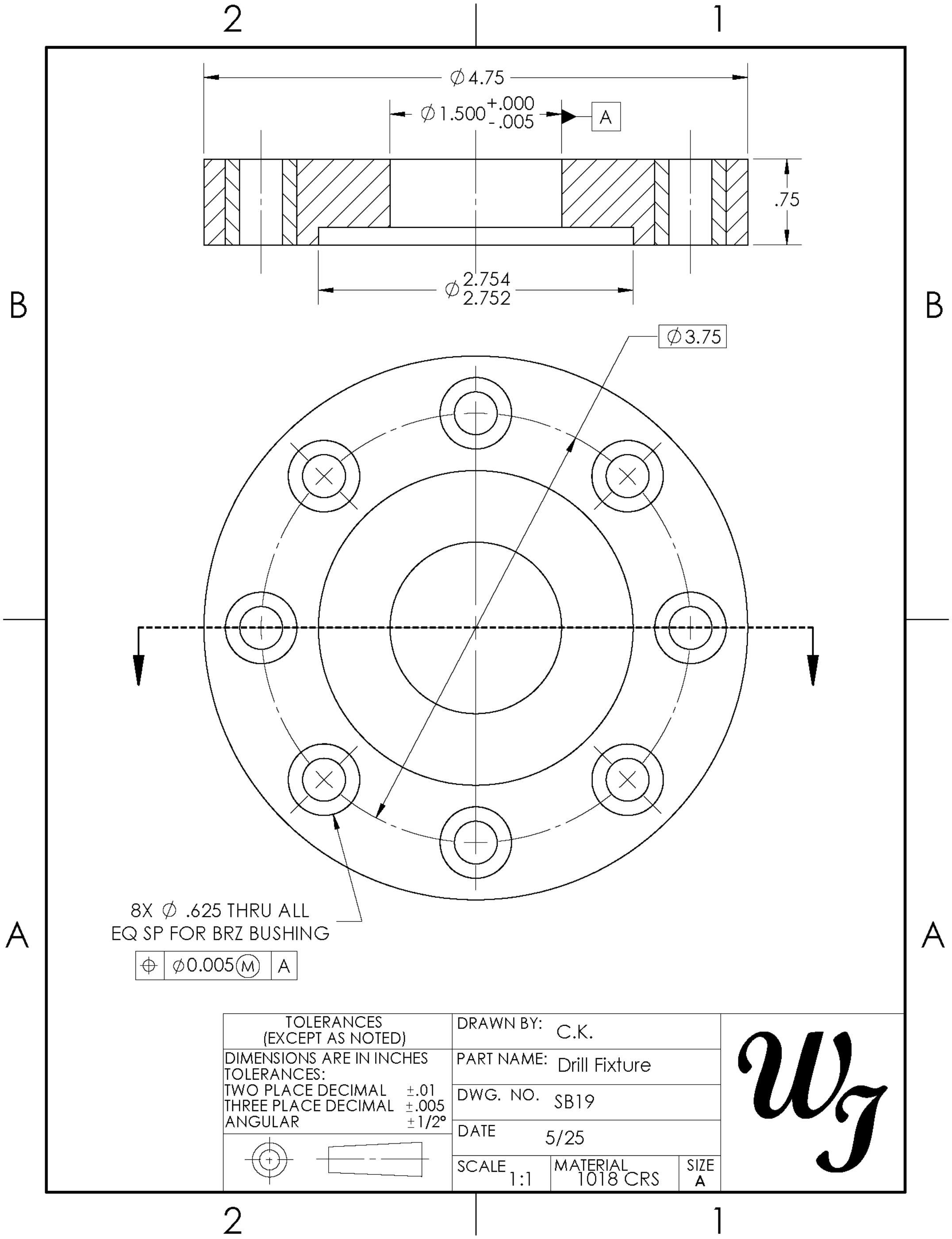
Directions: Layout and scribe this workpiece according to the print.


WTC Designs
TITLE:
DRILL GAGE
SIZE
A
DWG. NO.
SB48
REV
A
SCALE: 1:2
WEIGHT:
SHEET 1 OF 1
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS IN INCHES
TOLERANCES:
FRACTIONAL ±1/64
ANGULAR: ±1/2°
TWO PLACE ±.01
THREE PLACE ±.005
MATERIAL
CRS
DO NOT SCALE DRAWING
NAME
DATE
DRAWN
DKS
11/25
CHECKED
MGS
11/27
ENG APPR.
MFG APPR.
CB
11/27
Q.A.
COMMENTS:
⌀1/16"
⌀1/8"
⌀3/16"
⌀1/4"
⌀5/16"
⌀3/8"
⌀7/16"
⌀1/2"
1 1/8"
9/16"
1/2"
1/2"
5/8"
5/8"
5/8"
5/8"
3/4"
3/4"
1"
(6")
31°
2"
R1/2"
1/8"
A
B
1
2

Exercise 14.5-2

Directions: Scribe the circles for the 1.500, 2.754/2.752, and 3.75-diameter circles. Consult the *Machinery's Handbook* for the "Chord Length for Given Number of Divisions" for the diameter of the bolt circle and the number of holes required to find the distance to set the divider to lay out the bolt hole locations, as shown in Figure 14-10.

Figure 14-10. A spring divider is used to create a scribed arc

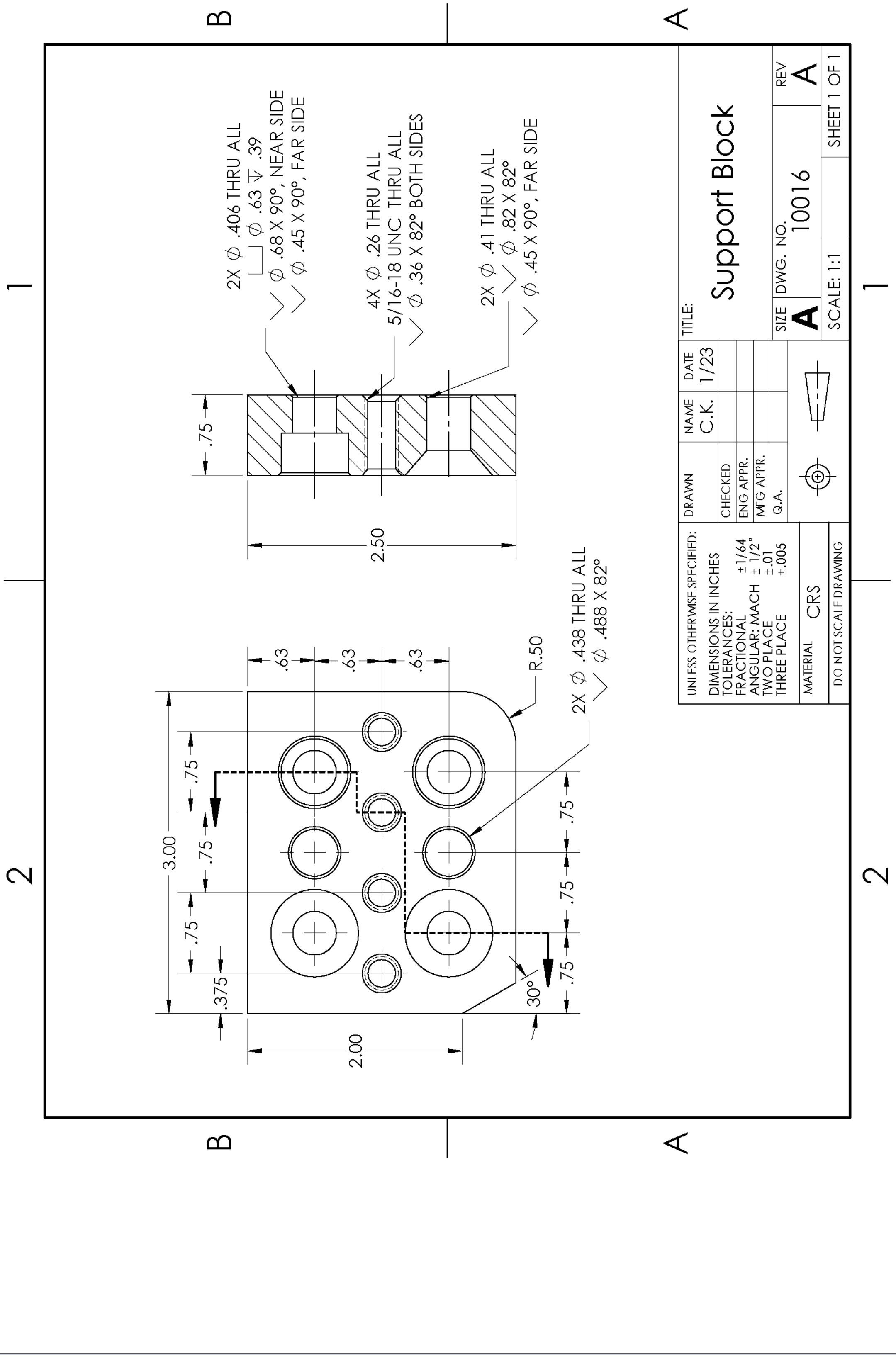


TOLERANCES
(EXCEPT AS NOTED)
DIMENSIONS ARE IN INCHES
TOLERANCES:
TWO PLACE DECIMAL ±.01
THREE PLACE DECIMAL ±.005
ANGULAR ±1/2°

DRAWN BY: C.K.
PART NAME: Drill Fixture
DWG. NO. SB19
DATE 5/25
SCALE 1:1
MATERIAL 1018 CRS
SIZE A

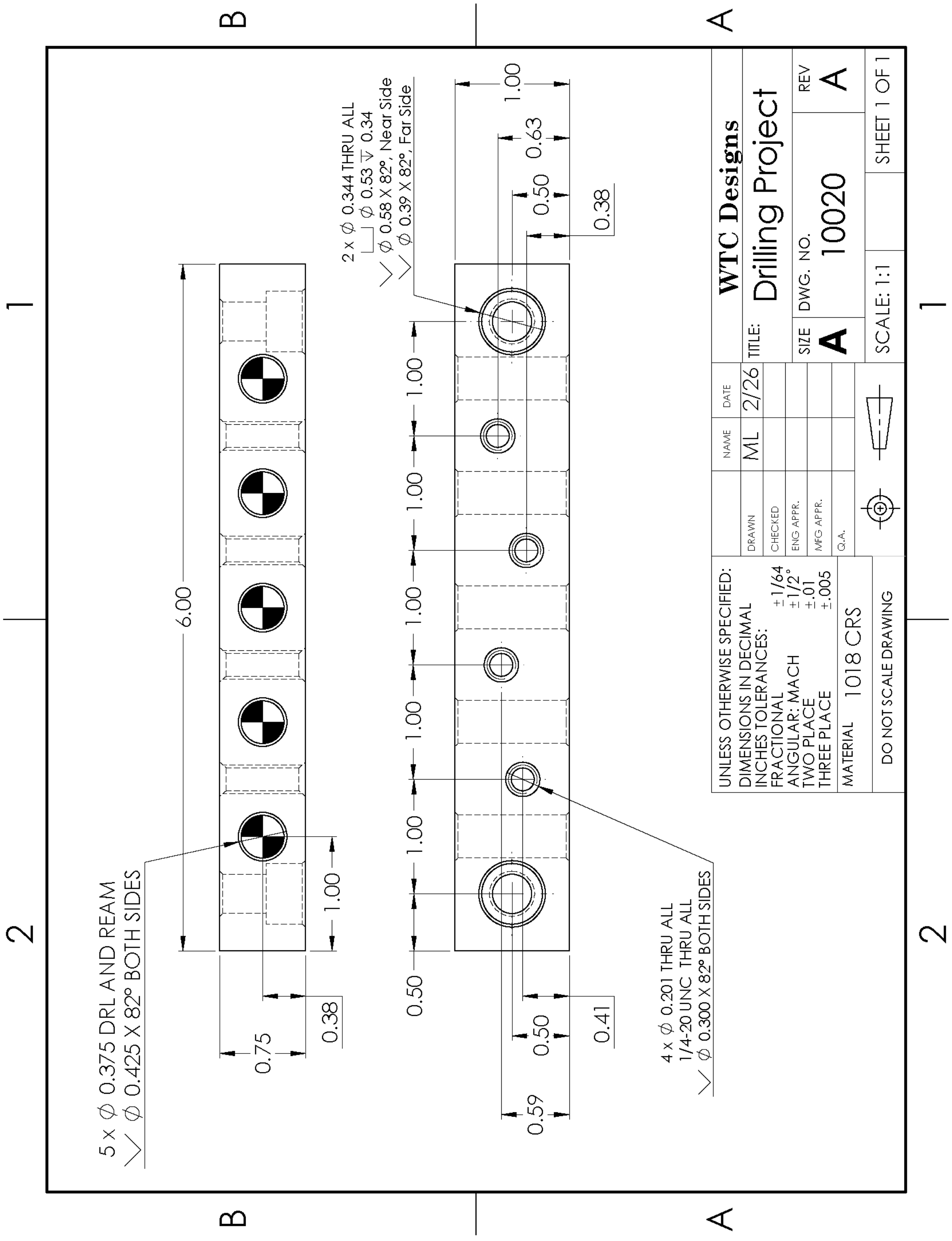
Exercise 14.5-3

Directions: Layout and scribe this workpiece according to the print.


2X ⌀ .406 THRU ALL
⌴ ⌀ .63 ↧ .39
⌵ ⌀ .68 X 90°, NEAR SIDE
⌵ ⌀ .45 X 90°, FAR SIDE
4X ⌀ .26 THRU ALL
5/16-18 UNC THRU ALL
⌵ ⌀ .36 X 82° BOTH SIDES
2X ⌀ .41 THRU ALL
⌵ ⌀ .82 X 82°
⌵ ⌀ .45 X 90°, FAR SIDE
2X ⌀ .438 THRU ALL
⌵ ⌀ .488 X 82°
R.50
.75
2.50
3.00
2.00
.63
.63
.63
.75
.75
.75
.375
.75
.75
.75
30°
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS IN INCHES
TOLERANCES:
FRACTIONAL ±1/64
ANGULAR: MACH ±1/2°
TWO PLACE ±.01
THREE PLACE ±.005
MATERIAL CRS
DO NOT SCALE DRAWING
DRAWN
CHECKED
ENG APPR.
MFG APPR.
Q.A.
NAME C.K.
DATE 1/23
TITLE:
Support Block
SIZE A
DWG. NO. 10016
REV A
SCALE: 1:1
SHEET 1 OF 1
A
B
1
2

Exercise 14.5-4

Directions: Layout and scribe this workpiece according to the print.


5 x ⌀ 0.375 DRL AND REAM
⌵ ⌀ 0.425 X 82° BOTH SIDES
6.00
1.00
0.75
0.38
2 x ⌀ 0.344 THRU ALL
⌴ ⌀ 0.53 ↧ 0.34
⌵ ⌀ 0.58 X 82°, Near Side
⌵ ⌀ 0.39 X 82°, Far Side
1.00
0.63
0.50
0.38
1.00
1.00
1.00
1.00
1.00
0.50
0.50
0.41
0.59
4 x ⌀ 0.201 THRU ALL
1/4-20 UNC THRU ALL
⌵ ⌀ 0.300 X 82° BOTH SIDES
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS IN DECIMAL
INCHES TOLERANCES:
FRACTIONAL ±1/64
ANGULAR: MACH ±1/2°
TWO PLACE ±.01
THREE PLACE ±.005
MATERIAL 1018 CRS
DO NOT SCALE DRAWING
NAME
DATE
DRAWN ML 2/26
CHECKED
ENG APPR.
MFG APPR.
Q.A.
WTC Designs
TITLE: Drilling Project
SIZE A
DWG. NO. 10020
REV A
SCALE: 1:1
SHEET 1 OF 1
A
B
1
2

Images:

All images by Mark Lorier and licensed under CC BY-NC 4.0 except where otherwise noted.

Learning Activities—Solutions

CHAPTER 1: DRAWING VS. PRINTS

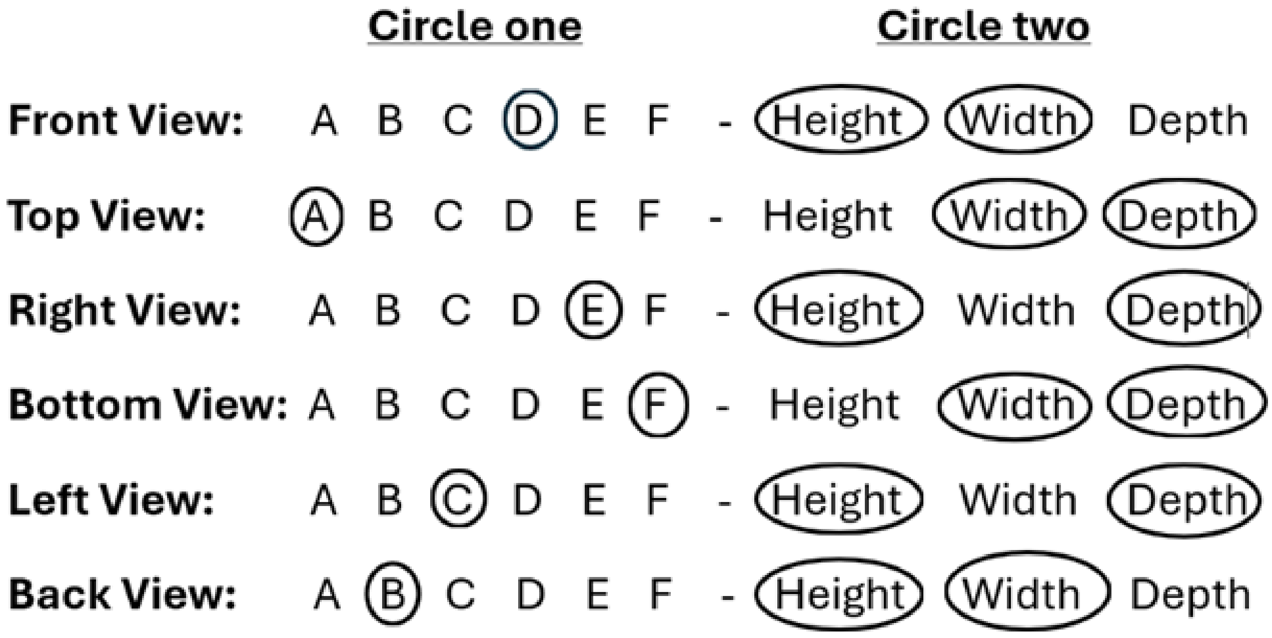
Exercise 1

1. a) Design phase and production phase
2. a) To generate essential documentation for product fabrication
3. d) All of the above
4. d) All of the above
5. d) All of the above
6. a) It refers to prints with a blue background and white lines

---

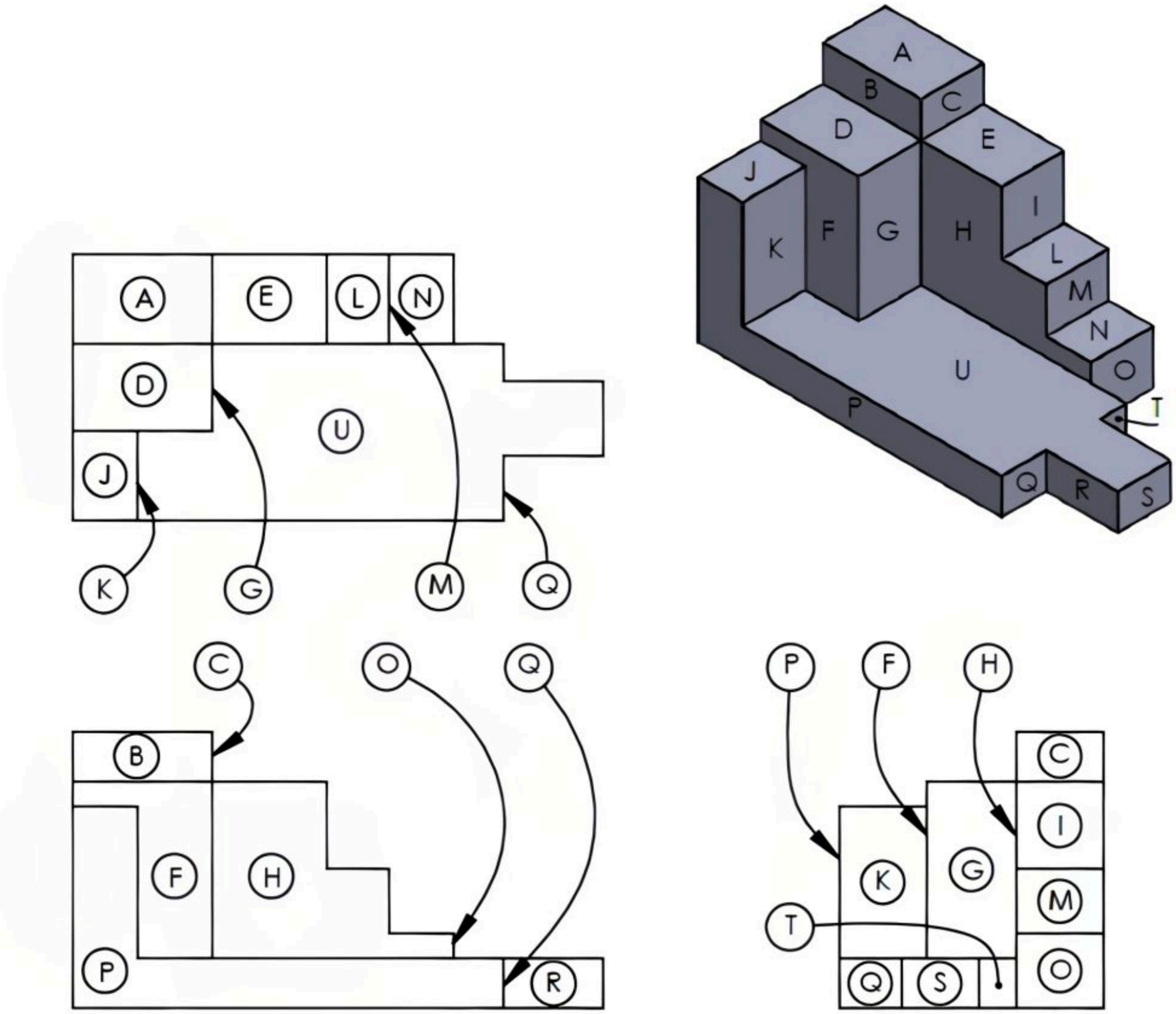
CHAPTER 2: VIEWS OF AN OBJECT

View Identification Quiz

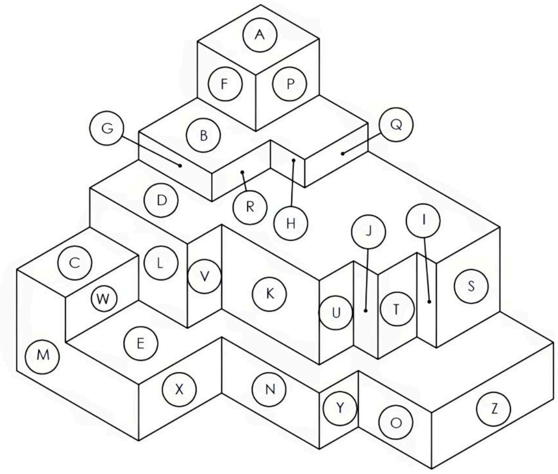


	Circle one	**Circle two**
Front View:	A B C [✓ D] E F	- [✓ Height] [✓ Width] Depth
Top View:	[✓ A] B C D E F	- Height [✓ Width] [✓ Depth]
Right View:	A B C D [✓ E] F	- [✓ Height] Width [✓ Depth]
Bottom View:	A B C D E [✓ F]	- Height [✓ Width] [✓ Depth]
Left View:	A B [✓ C] D E F	- [✓ Height] Width [✓ Depth]
Back View:	A [✓ B] C D E F	- [✓ Height] [✓ Width] Depth

Surface Identification Quiz 1



Surface Identification Quiz 2



CHAPTER 3: TYPES OF LINES

Line Quiz 1

A. Viewing-plane line
- B. Visible line
- C. Section line
- D. Short-break line
- E. Hidden line
- F. Long-break line
- G. Cutting-plane line
- H. Short-break line
- I. Visible line
- J. Dimension line
- K. Dimension line
- L. Extension line
- M. Leader

N. Center line
O. Visible line
P. Section line

Line Quiz 2

A. Chain line
B. Hidden line
C. Centerline
D. Cutting-plane line
E. Visible line
F. Viewing-plane line
G. Short-break line
H. Hidden line
I. Visible line
J. Leader
K. Phantom line
L. Phantom line
M. Centerline
N. Dimension line
O. Extension line
P. Centerline
Q. Section line

Line Quiz 3

1. Visible line
2. Phantom line
3. Cutting-plane line
4. Leader
5. Viewing-plane line
6. Section line
7. Hidden line
8. Dimension line

9. Long break line
10. Centerline
11. Extension line
12. Short-break line
13. Chain line

Line Quiz 4

A. Visible line
B. Hidden line
C. Extension line
D. Centerline
E. Dimension line
F. Cutting-plane line
G. Hidden line
H. Centerline
I. Visible line
J. Leader
K. Visible Line
L. Section line

CHAPTER 4: OBJECTS IN DIFFERENT VIEWS

View Identification Exercise 1

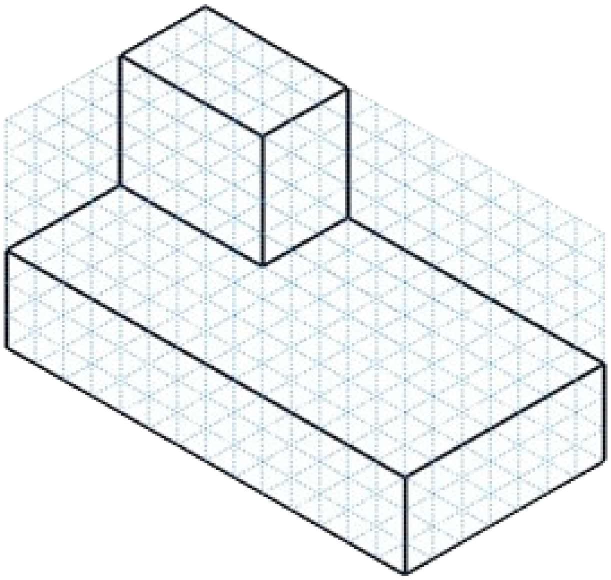
1. Top view: J; Right view: E
2. Top view: B; Right view: H

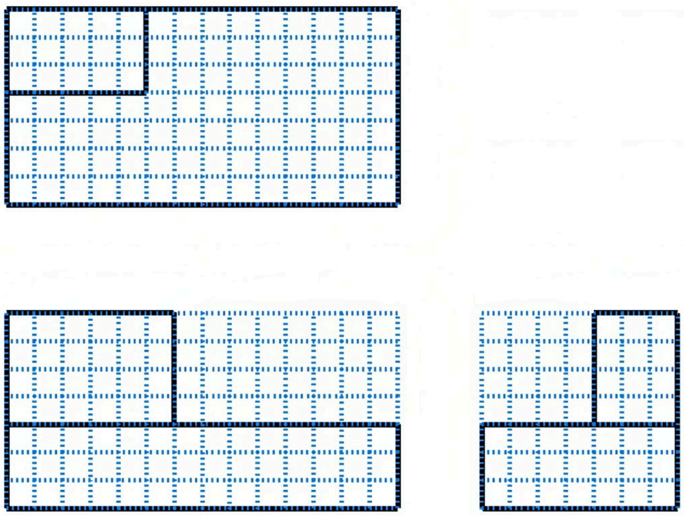
View Identification Exercise 2

1. C
2. D
3. D
4. B
5. D
6. C
7. A
8. C
9. C
10. A

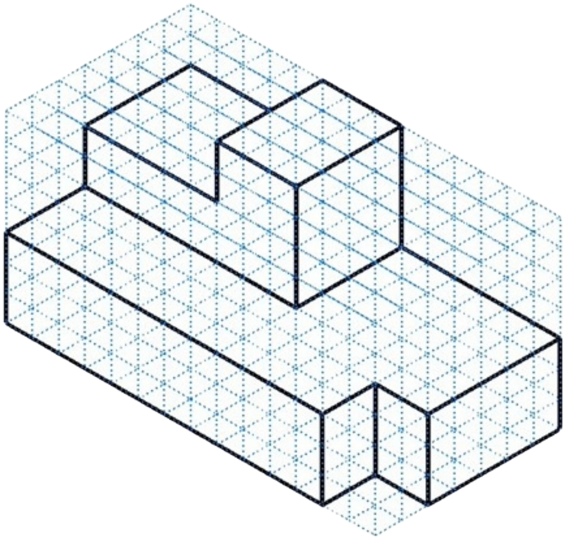
Orthographic Sketching Exercise Solutions

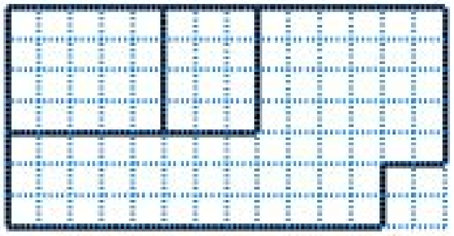
Orthographic Sketch 1

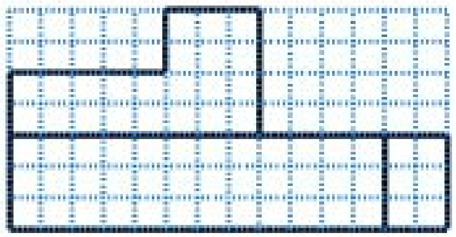


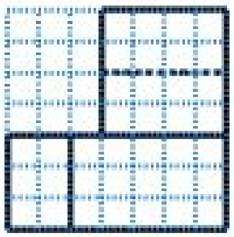


Orthographic Sketch 2

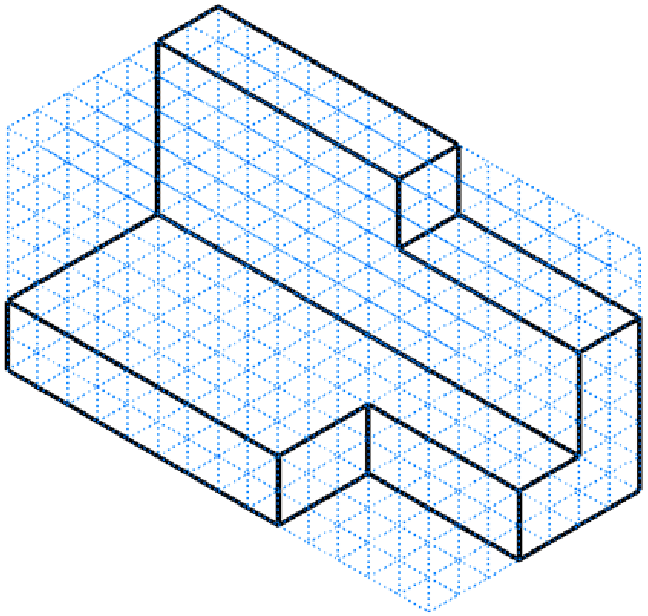


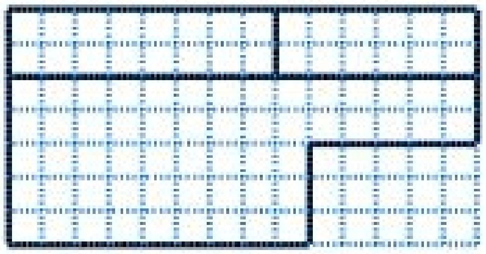


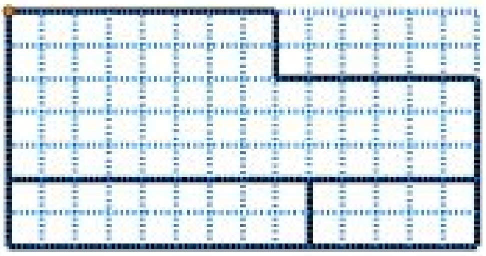


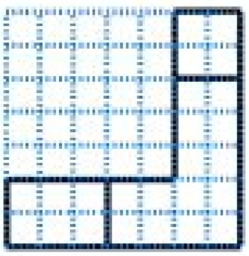


Orthographic Sketch 3

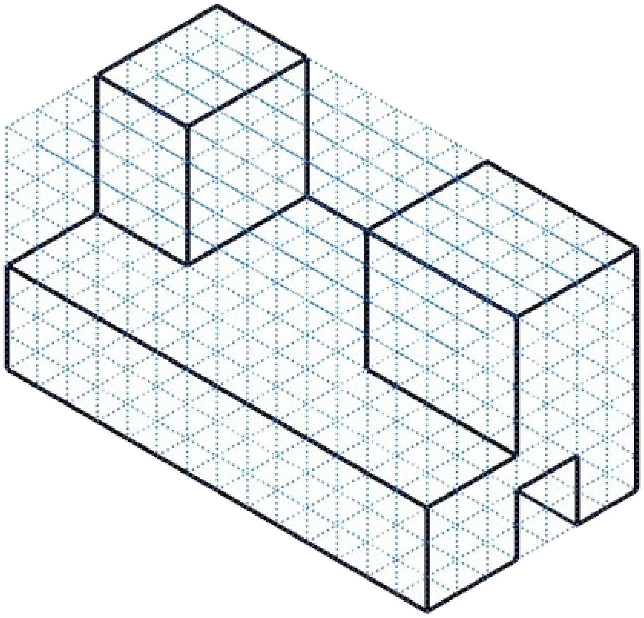


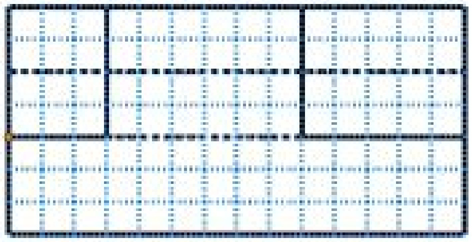


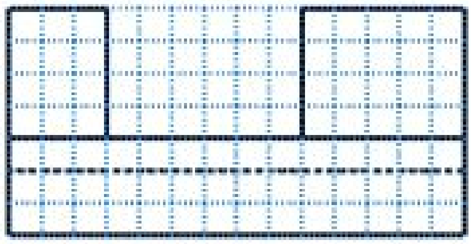


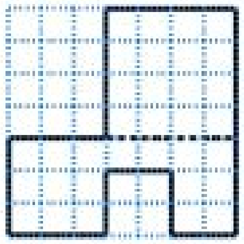


Orthographic Sketch 4

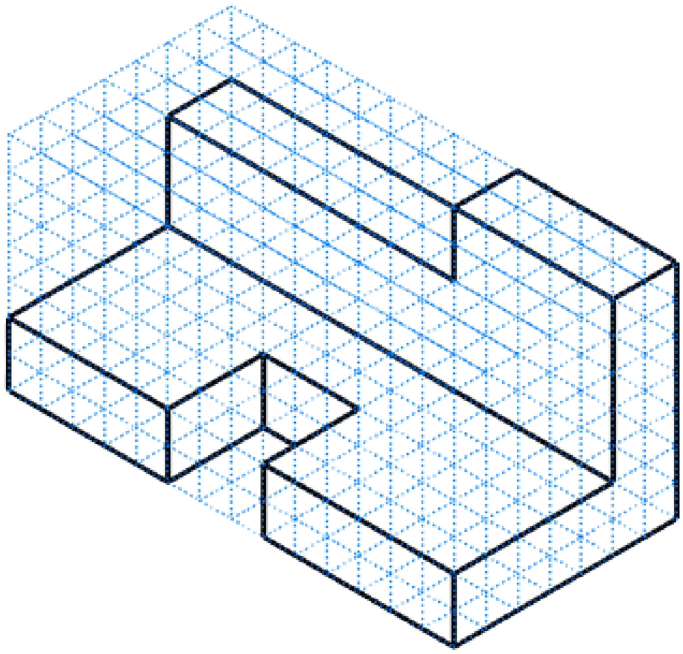


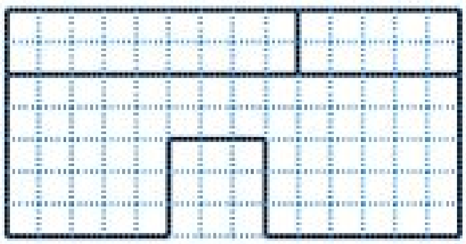


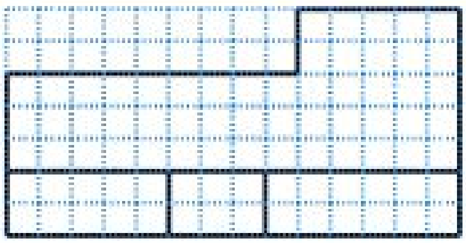


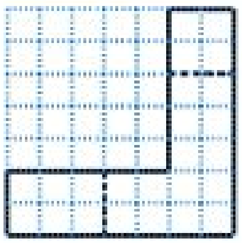


Orthographic Sketch 5

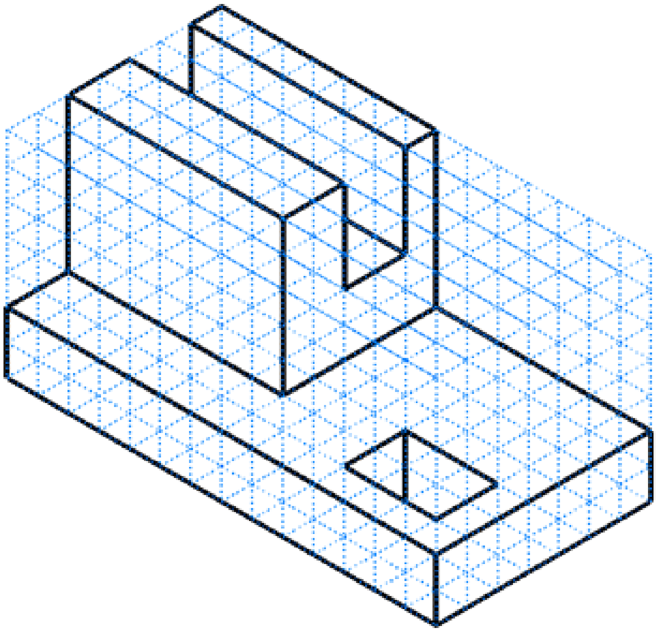


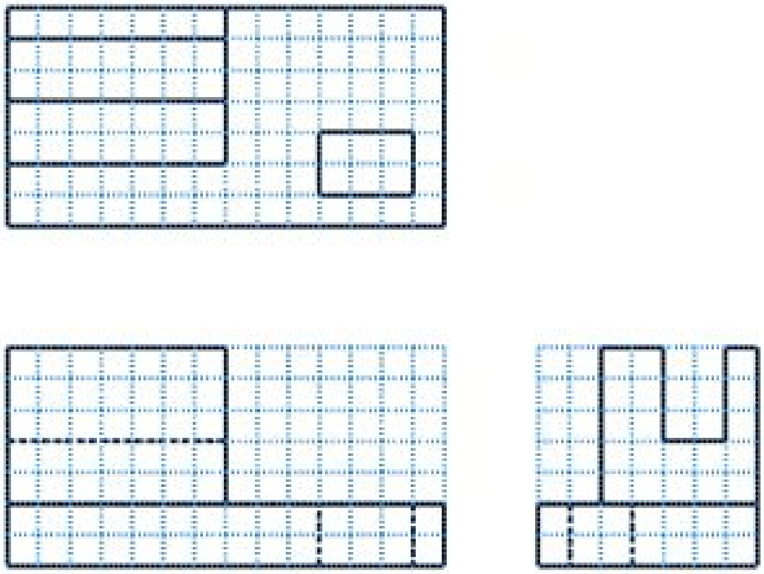






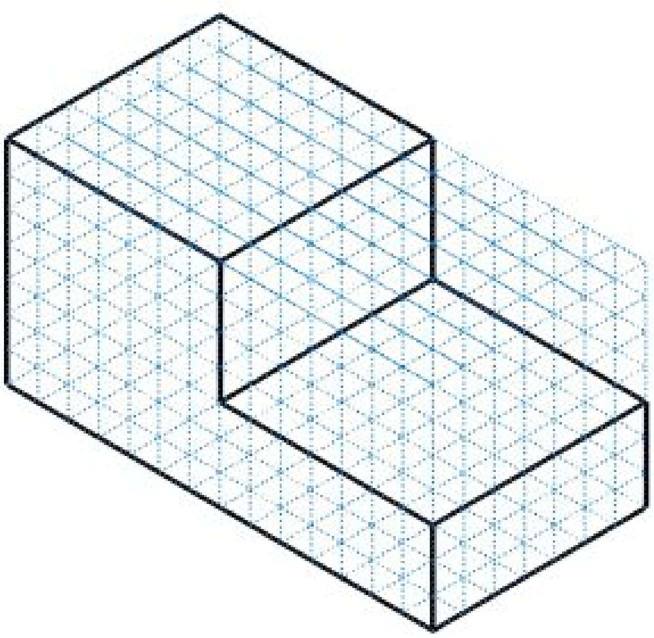
Orthographic Sketch 6



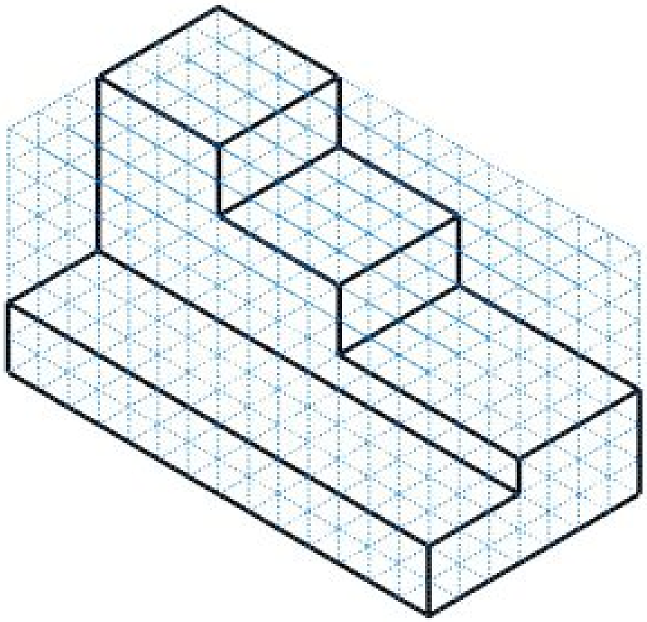


Isometric Sketching Exercise Solutions

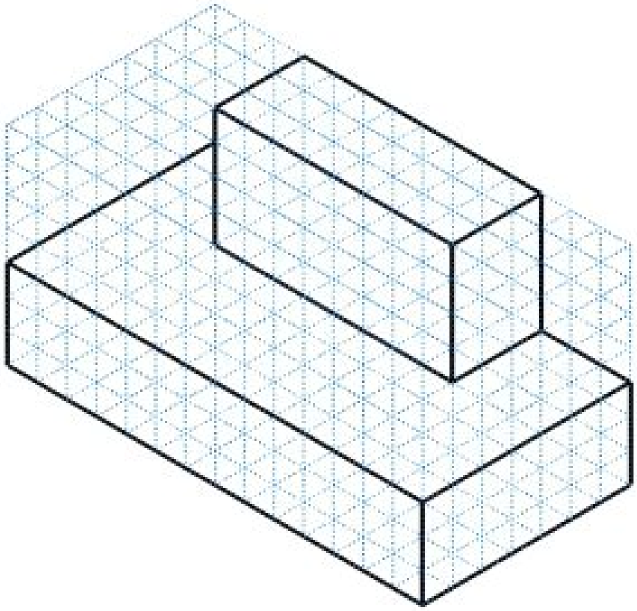
Isometric Sketch 1



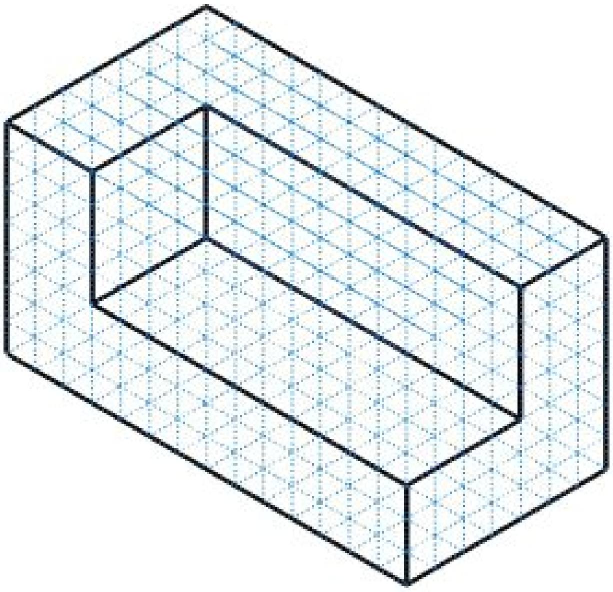
Isometric Sketch 2



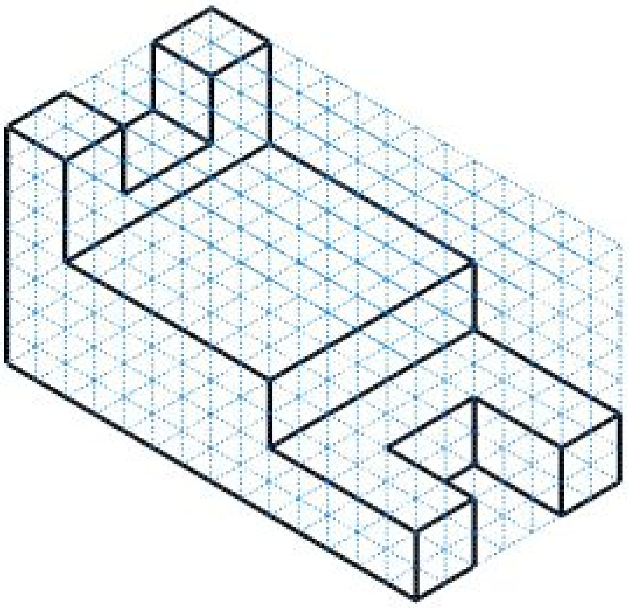
Isometric Sketch 3



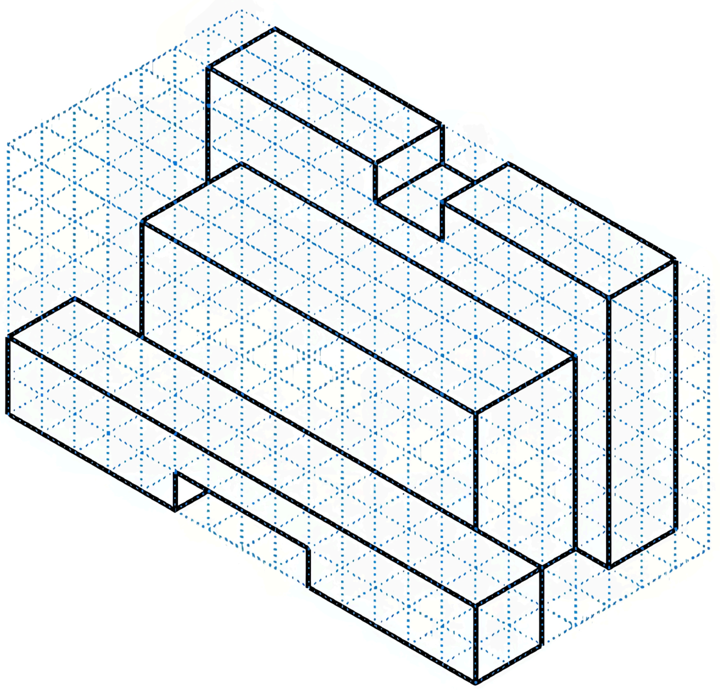
Isometric Sketch 4



Isometric Sketch 5



Isometric Sketch 6



CHAPTER 5: DIMENSIONING SYSTEMS

Exercise 5.2-1

A. 2.00
B. 1.15
C. 1.40
D. 1.15
E. 1.95
F. 0.60
G. 0.20
H. 1.99
I. 0.60
J. 0.85

Exercise 5.2-2

A. X 1.50 Y -1.25
B. X 4.50 Y -1.25
C. X 4.50 Y -3.75
D. X 1.50 Y -3.75
E. X 2.00 Y -3.75
F. X 4.00 Y -1.25
G. X 1.25 Y -0.50
H. X 4.75 Y -4.50
I. X 0.50 Y -0.50
J. X 5.50 Y -0.50
K. X 5.50 Y -4.50
L. X 0.50 Y -4.50

Exercise 5.3-1

A. 9.525) .375

B. 12.70) .500
C. 15.875) .625
D. 19.05) .750
E. 50.80) 2.000
F. 63.50) 2.500
G. 76.20) 3.000
H. 6.53) .257
I. 9.21) .363
J. 10.08) .397
K. 15.88) .625
L. 10.02) .394
M. 17.15) .675
N. 11.35) .447
O. 12.38) .487

Exercise 5.3-2

A. 3/4
B. 1/2
C. 5/8
D. 15/16
E. 13/64
F. 9/32
G. 11/16

Exercise 5.3-3

A. 45.0
B. 12.5
C. 5.5
D. 7.5
E. 18.0
F. 13.5
G. 10.0

H. 41.5
I. 18.5
J. 11.5
K. 4.5
L. 12.0

1. Precedence of lines
2. Metric
3. Third-angle projection

CHAPTER 6: PRINT DIMENSIONS

Exercise 6.4-1

1. Both
2. 2.10
3. 4.766
4. .063
5. .75
6. .30
7. .15
8. 3.906
9. 1.203
10. Decimal inch

Exercise 6.4-2

A. 0.937
B. 1.125
C. 2.250

D. 1.499
E. 3.250
F. 1.594
G. 0.173
H. 1.062
I. 1.313

1. 6.001
2. 2.876
3. 1.00 slot location

Exercise 6.4-3

1. Height: 0.995
2. Width: 1.005
3. Depth: 1.000
4. A) 0.498
5. B) 0.500
6. C) 0.504
7. D) 0.496
8. E) 0.493
9. F) 0.503

Exercise 6.4-4

1. A) 1.47
2. B) 0.85
3. C) 0.603
4. D) 1.62
5. E) 0.375
6. F) 0.62
7. G) 1.63
8. H) 0.377
9. I) 1.60

10. J) 0.368
11. K) 0.64
12. L) 1.226
13. M) 2.23
14. N) 2.26

Exercise 6.5-1

1. 60°
2. 2.50
3. Chain
4. .39
5. .38
6. .32
7. 2.87
8. 2

Exercise 6.5-2

1. A) 0.75
2. B) 0.87
3. C) 0.25
4. D) 0.37
5. E)60°
6. F) 60°
7. G) 0.69
8. H) 0.03
9. I) 0.62
10. J) 90°
11. K) 0.26

Exercise 6.6-1

1. Horizontal

2. .06
3. 8
4. Yes
5. 11
6. 120 degrees
7. 90 degrees
8. .235

Enter the dimensions for the letters:

1. A) .235
2. B) .295
3. C) 15 degrees
4. D) 30 degrees

Exercise 6.6-2

A. 120
B. 90
C. 72
D. 50
E. 36
F. 14
G. 54
H. 18
I. 12
J. 20
K. 40
L. 10

CHAPTER 7: TITLE BLOCKS

Exercise 7.4-1

1. 11001-1910-YA-L04
2. PAL
3. KOHLER CO.
4. 1:1
5. FULL
6. CRS
7. CHAIN
8. ±0.02
9. ±0.010
10. ±.005
11. 1
12. ⌀4” X 5 ⅝” LG

Exercise 7.4-2

1. 555-A-8675309
2. A
3. 6061 Alum.
4. Was ⌀.28 thru
5. 4.375

A. .275
B. .125
C. .125
D. .25
E. 0.08
F. .78
G. 1.38
H. .1485
I. .1735
J. 1.575
K. 60 degrees
L. .525

Exercise 7.4-3

1. 7
2. Metric
3. 14.0
4. 3.3
5. 200.0
6. 6.0
7. 0.236
8. 4
9. 45.0 was 40.0
10. A2
11. Long Break
12. D

A. 18.0
B. 60
C. 38.0
D. 39.0
E. 194
F. 100

CHAPTER 8: TOLERANCE AND DIMENSIONS

Exercise 8.9-1 Tolerance

A to B = 0.90 ± .01	**A to H = 6.00 ± .01**	**D to H = 3.30 ± .02**
A to D = 2.70 ± .01	B to C = 1.10 ± .02	G to H = 0.85 ± .02

Exercise 8.9-2 Tolerance

A to B = 0.90 ± .01	**A to H = 6.00 ± .01**	**D to H = 3.30 ± .04**
A to D = 2.70 ± .03	B to C = 1.10 ± .01	G to H = 0.85 ± .07

Exercise 8.9-3 Tolerance

1. A) .34
2. ±.025
3. B) .12
4. ±.020
5. C) .75
6. ±.025
7. D) 1.50
8. ±.040
9. E) .497
10. ±.02
11. F) .87
12. ±.015
13. G) 150°
14. H) 60°

Exercise 8.9-4 Tolerance

1. ±0.02
2. Bilateral
3. 1.1804
4. 2.0470
5. Unilateral
6. 9.530

A. **MAX** 0.885-(1.053/2) = 0.3585 **MIN** .865-(1.073/2) = .3285

B. **MAX** 1.0010 **MIN** 1.0005

C. **MAX** 1.385-(2.0465/2) = .3618 **MIN** 1.365-(2.0465/2) = .3415

D. **MAX** .885-.25 = .635 **MIN** .865-.29 = .575

E. **MAX** 7.260-3.052 = 4.208 **MIN** 7.240-3.072 = 4.168

F. **MAX** 1.02-.48 = .54 **MIN** .98-.52 = .46

G. **MAX** 7.260-91.797/2)-(1.053/2) = 5.835 **MIN** 7.240-(1.817/2)-1.073/2) = 5.795

H. **MAX** 7.260+(2.0470/2)+(1.1809/2) = 8.8740 **MIN** 7.240+(2.0465/2)+1.1804/2) = 8.8535

Exercise 8.9-5 Tolerance

1. Yes (.03)
2. Limit
3. Equal bilateral
4. Limit
5. Unilateral
6. 5002 +/- .0002
7. 1.496
8. 1.504
9. 2.9995-1.505-.752= .7425
10. 3.0000-1.495-.748= .7570

CHAPTER 9: PRINT SYMBOLS AND NOTES

Exercise 9.6-1

1. Datum
2. Basic Dimension
3. GD&T Symbol
4. Tolerance
5. Modifier
6. Datum Reference

Exercise 9.6-2

1. 2
2. 2
3. 63
4. .000063
5. General
6. Finish All Over
7. Flatness
8. Parallelism
9. .30
10. 2.5:1

A. **MAX** (.209-.14)/2 = .0345 **MIN** (.199-.16)/2 = .0195
  B. **MAX** .26-.148 = .112 **MIN** .24-.151 = .089
  C. **MAX** 1.005-.045-.045 = .915 **MIN** .995-.055-.055 = .885
  D. **MAX** .151 **MIN** .148

Exercise 9.6-3

1. 3
2. Yes
3. .52
4. 2
5. .748/.750
6. .551 & .292
7. .000063”
8. 1/2”
9. 3
10. Orientation
11. .555

A. .81
B. 1.00
C. 2.25
D. 0.165
E. **MAX** 1.755-.290 = 1.465 **MIN** 1.745-.294 = 1.451
F. **MAX** 0.76×2 = 1.52 **MIN** .74×2 = 1.48
G. **MAX** (1.130+.89)2 = 4.04 **MIN** (1.120+.87)2 = 3.98
H. **MAX** 1.755-.99-.290 = .475 **MIN** 1.745-1.01-.294 = .441
I. **MAX** 0.76-(.551/2) = .4845 **MIN** 0.74-(.555/2) = .462

Exercise 9.6-4

1. .20
2. 3
3. 4
4. 5
5. General
6. 4
7. 125, 63
8. .000125 and .000063

9. Flatness
10. .20
11. .001
12. .003
13. Limit
14. Unequal bilateral
15. .19
16. .280
17. 4:1

CHAPTER 10: MACHINING DETAILS

Exercise 10.10-1

1. .75
2. 1.52-.26-.26 = 1.0
3. 1.50-.28-.28 = .94
4. .755
5. 1.98
6. .52+.52 = 1.04
7. 2.52+.52+.52 = 3.56
8. 24
9. 1/24 = .042
10. Unified course
11. 2B
12. 10
13. 10/25.4 = .39
14. 1/24 = .042
15. Chain

Exercise 10.10-2

1. 2.53
2. 1.27
3. .13
4. 1.004
5. 1.024
6. 1.125
7. 6
8. 1/12 = .083
9. Double
10. (1/12)2 = .167

Exercise 10.10-3

1. 2.01
2. 2.00-(.1 x2) = 1.80
3. 2.00-(.125 x 2) = 1.750
4. 1.99-(.13×2) = 1.730
5. 1.313-.38-.13 = .803
6. 1
7. 5/16 or .3125
8. 16°
9. 2.00-.50-.50 = 1.00
10. 5.85-.85-.92-1.77-.38-.25-1.00 = 0.68
11. ±.07
12. .92-.25 = 0.67
13. 2.00-.063-.063 =1.874
14. 2.01-.058-.058 = 1.894
15. .85-.10-.10 = .650
16. 3.00-.01-.01 = 2.80
17. 3.01–09-.09 = 2.83

CHAPTER 11: SECTION VIEWS

Exercise 11.8-1

1. Symmetry
2. Full Section
3. Half
4. No
5. ⌀2.751 / 2.750
6. ⌀3.500
7. 5.000
8. R .03 MAX

A. 1.125 ± .023
B. .835 ± .023
C. 1.5 ± .02
D. 1.62 ± .020
E. 2.835 ± .018
F. 1.12 ± .015
G. 3.40 ± .023
H. **MAX** 3.758 **MIN** 3.744
I. **MAX** .765 **MIN** .735
J. **MAX** 0.4265 **MIN** 0.4230

Exercise 11.8-2

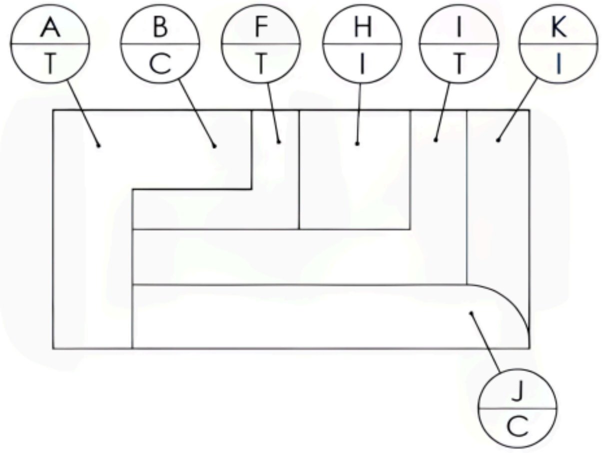
1. Offset
2. A
3. Front
4. .19
5. .5003

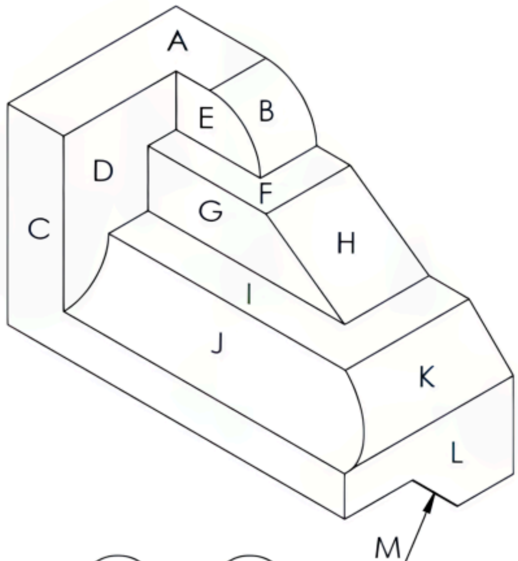
6. 2.75
7. Yes

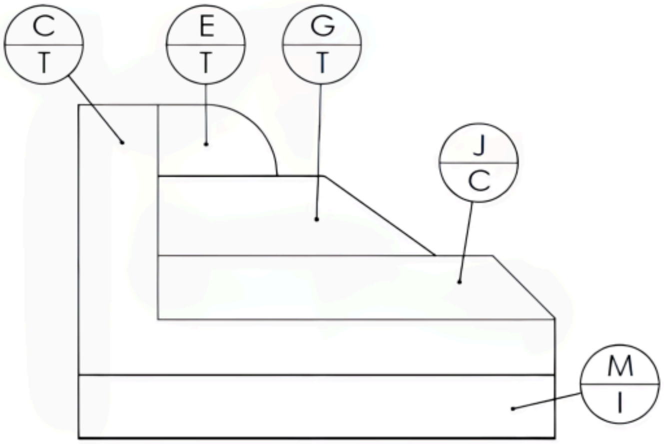
A. 60 deg
   B. .51-.38 = .13
   C. 2.25-.250 = 2.00
   D. 1.25-.30 = .950
   E. 4.-.38-.38 = 3.24
   F. 2.-1.1 = .90
   G. 1.25
   H. 2.50-1.25-(.66-.16) = .75
   I. **MAX** (1.01-.5)/2 = .255 **MIN** (.99-.5003)/2 = .24485

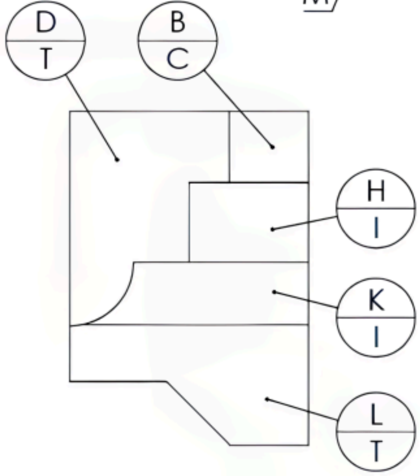
CHAPTER 12: AUXILIARY VIEWS

Exercise 12.3-1 Surface Identification

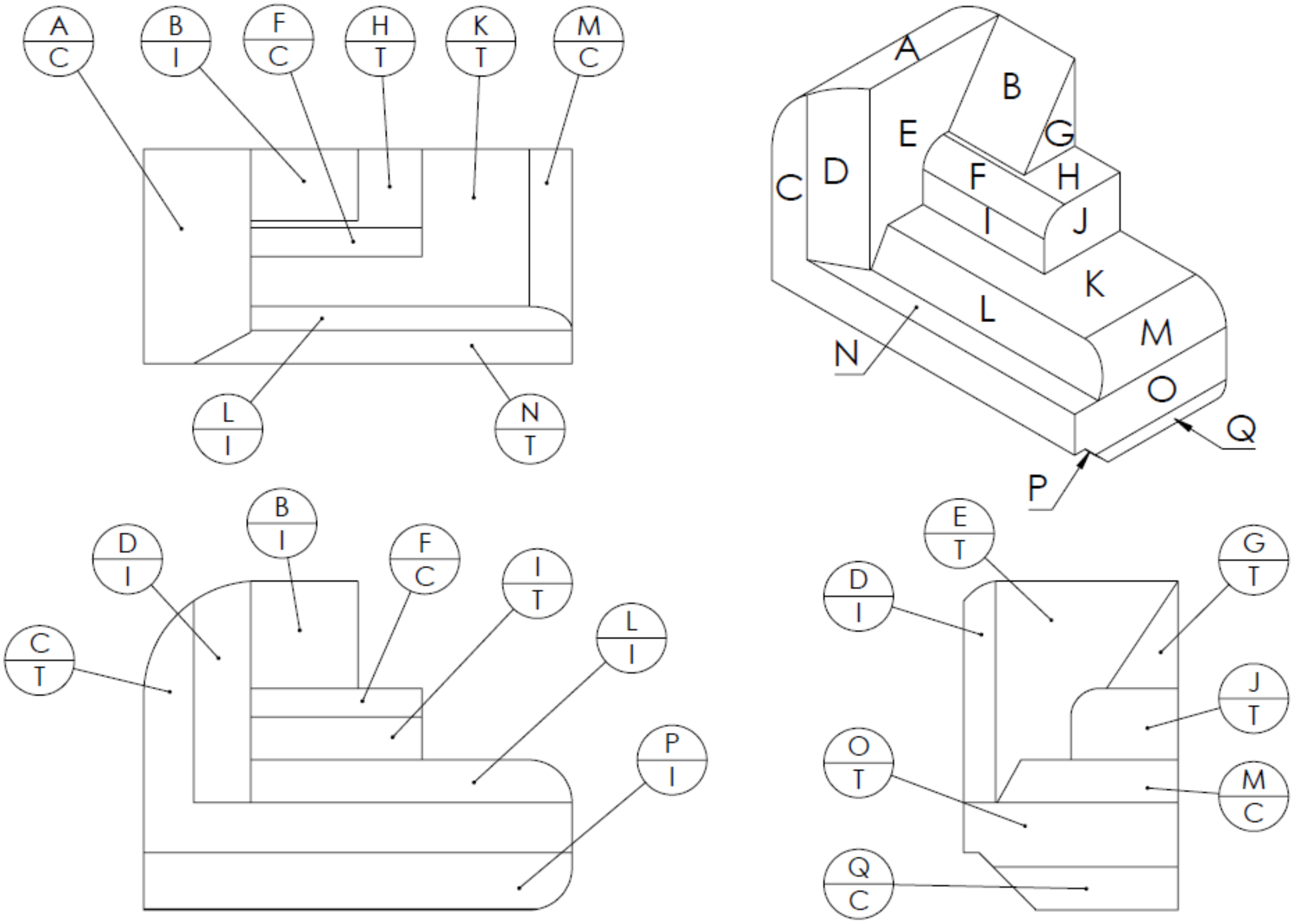








Exercise 12.3-2 Surface Identification



Exercise 12.4-1 Auxiliary View

1. Removed
2. ⅜ or .375
3. 1:1
4. 1.20-1.08 = .12
5. Broken-out
6. 8.00
7. 1.5
8. 11
9. 1/11 = .091
10. 1/16 = .0625

11. 2B (used when no classification is listed)
12. .52
13. .750+.04+.04 = .83
14. 1.13+(.250/2) = 1.255
15. 1.89+1.01+2.76+2.39 = 8.05

Exercise 12.4-2 Auxiliary View

1. 13
2. 1/20 = .05
3. 1:3
4. Right
5. 4.000 / 3.998
6. .748
7. Auxiliary
8. .225
9. 63
10. 3

A. 2.63 ± .020
B. .50 ± .025
C. 3.54 ± .025
D. 0.55 ± .010
E. 0.75 ± .010
F. .31 ± .01
G. 1.38 ± .01
H. **MAX** 0.524 **MIN** 0.502
I. **MAX** 3.500 **MIN** 3.458

Exercise 12.6-1 Auxiliary View

1. 340 SS
2. Partially enlarged
3. 3:1

4. 1/4-28 hole
5. Equal bilateral

A. 0.20
  B. 0.1875 ± .0055
  C. 0.125 ± .003
  D. 0.270 ± .015
  E. 0.425 ± .021
  F. 0.485 ± .021
  G. 0.565 ± .021

CHAPTER 13: ASSEMBLY PRINTS

Exercise 13.5-1

1. B
2. 6
3. SHCS 0.5-13x2x2
4. Exploded view
5. Threaded
6. 1:1.5
7. 38726
8. A

Exercise 13.5-2

1. 11 x 17
2. 23
3. Slide Wear Plate
4. Multi-view

5. 14
6. 1:2
7. 64587FN
8. Twice

Exercise 13.5-3

1. A
2. 9
3. Top clamp
4. Broken-out
5. 1:5
6. 8.5 x 11
7. Tee nut
8. 3

CHAPTER 14: PRINT SPECIFICATIONS

Answer keys for these activities are not available.

Images:

All images by Mark Lorier and licensed under CC BY-NC 4.0 except where otherwise noted.

Abbreviations and Acronyms Glossary

A

ADJ: Adjustable
AISC: American Institute of Steel Construction
AISI: American Iron and Steel Institute
AL: Aluminum
ANSI: American National Standards Institute
Approx: Approximate
ASME: American Society of Mechanical Engineers
ASTM: American Society for Testing and Materials
AWG: American Wire Gauge

B

BC: Bolt Circle
BOM: Bill of Material
BOT: Bottom
BRZ: Bronze

C

CAD: Computer-Aided Drafting

CAM: Computer-Aided Manufacturing
CBORE: Counterbore
C.I.: Cast Iron
CL: Center Line
CONT: Continuous
CMM: Coordinate Measuring Machine
CRS: Colled Rolled Steel
CTR: Center

D

DB: Double
DET: Detail
DIA: Diameter
DIM: Dimension
DP: Deep
DR: Drill
DWG: Drawing

F

FAO: Finish All Over
FIN: Finish
FOS: Feature of Size
FT: Foot or Feet

G

GA: Gauge
GALV: Galvanized
GD&T: Geometric Dimensioning and Tolerancing

H

HRS: Hot Rolled Steel
HT TR: Heat Treat

I

ID: Inner Diameter
ISO: International Organization for Standardization

M

MATL: Material
MAX: Maximum
MECH: Mechanical
MIN: Minimum
MISC: Miscellaneous

N

NTS: Not to Scale

O

OD: Outer Diameter
OPP: Opposite

P

PD: Pitch Diameter

R

R: Radius
REF: Reference
REQD: Required

S

SAE: Society of Automotive Engineers
SPEC: Specification
SS: Stainless Steel
STD: Standard
STL: Steel

T

TOL: Tolerance
TPF: Taper Per Foot
TYP: Typical

U

UNC: Unified National Coarse
UNEF: Unified National Extra Fine
UNF: Unified National Fine

Glossary

0-9

3D Printer: A machine allowing the creation of a physical object from a digital model.

A

Allowance: The minimum clearance (or maximum interference) of two parts.

Alloy: A mixture of two or more elements, one or more being metallic, melted together to create a type of metal.

Angular Dimension: A dimension used to measure an angle in decimal degrees or degrees, minutes, and seconds.

Anneal: The process of heating iron-based metal to a specific temperature then slowly cooling to reduce hardness and remove stress.

Assembly: The process of joining individual parts to create a complete product.

Assembly Drawing: A print or drawing showing how individual parts come together to form a complete product.

Annotation: Textual information on a blueprint, such as notes, dimensions, or symbols.

Auxiliary View: A secondary view of an object showing a specific detail not clearly visible in the main view.

Axis: A central line that a part or parts are symmetrical about.

B

Blind Hole: A hole that does not pass through the material.

Bill of Materials (BOM): A list of all components required to assemble a product, including quantities and specifications.

Bolt Circle: Circular centerline in which two or more holes are located.

Bore: Enlarging a hole to an accurate size.

Boss: A circular portion raised from a surface.

Burr: A sharp or jagged edge created by many metal cutting and forming processes.

C

Casting: A metal object created by molten metal poured or injected in a mold.

Chamfer: A beveled edge on a corner, often specified by angle or dimensions.

Chamfer Dimension: Specifies the angle and/or size of a chamfered edge.

Clearance: The space between two mating parts.

Clearance Hole: A hole with a diameter larger than the mating part's diameter, allowing for clearance.

Computer-Aided Drafting (CAD): A software-based process that allows users to create and analyze designs before committing to production.

Core: A part of a mold used to create interior shapes of the casting.

Contour: The overall outline or shape of an object.

Counterbore: A cylindrical enlargement at the end of a hole for a fastener head to sit flush.

Countersink: A conical depression at the end of a hole for a screw head to sit below the surface.

CNC (Computer Numerical Control) Machine: A machine using pre-programmed software to control the movement of tools and machinery in manufacturing.

D

Datum: A theoretical reference surface used for dimensioning and tolerancing.

Datum Plane: A theoretically flat surface used as a reference for dimensioning and tolerancing.

Detail Drawing: A print or drawing focusing on a specific part, providing all the details and information necessary to manufacture the part.

Detail Specification: Additional information specific to a particular part beyond the general blueprint.

Dimension: The numerical measurement of an object’s features, such as length, width, or height.

Dimension Line: A thin line with arrowheads at both ends indicating the measurement location.

Dimensioning System: The method used to specify dimensions on a blueprint (e.g., parallel, coordinate).

Draft Angle: A slight taper on the sides of a molded part to facilitate removal from the mold.

E

Extension Line: A thin line extending beyond the object to accommodate dimension lines and arrowheads.

Extrusion: An object created by pushing material through dies to form the desired shape.

F

Fillet: A concave curved edge where two surfaces meet creating an internal corner, often specified by radius.

Finish: The surface texture or treatment applied to a component.

Finish Symbol: A symbol on a blueprint indicating the desired surface finish.

First-Angle Projection: A method of creating orthographic drawings where the views are projected onto planes behind the object.

Fit: The relationship between mating parts, such as clearance fit or interference fit.

Fold Line: A line on a blueprint indicating where sheet metal should be folded, also called bend lines.

G

General Tolerance: A tolerance applied to all dimensions on a blueprint unless otherwise specified.

Geometric Dimensioning and Tolerancing (GD&T): A system for specifying exact geometry and allowable variations on blueprints.

Geometric Tolerance: A tolerance applied to the form, orientation, profile, location, or run-out of a feature.

Gusset: An angled piece placed between two joined pieces to provide additional strength.

H

Hidden Line: A dashed line representing features not visible from the current viewing angle.

Hole: A circular opening through a material.

Hole Callout: Specifies the size, depth, and other relevant information for a hole.

I

Inspection: The process of verifying that a manufactured part meets blueprint specifications.

Inspection Report: A document where inspection results are recorded.

Interference Fit: A tight fit where the mating parts require force to assemble.

Isometric Drawing: A three-dimensional drawing showing the object at a specific angle, allowing the viewer to see the front, top, and side of the object simultaneously.

L

Landing: A machined surface designed to provide a bearing surface for another part.

Least Material Condition (LMC): When a feature contains the minimum amount of material allowed by the feature tolerance.

Leader Line: A thin line with an arrowhead pointing to a specific feature and connecting to a note or symbol.

Limits: The maximum and minimum allowable size of a feature.

Line Type: The style of line used on a blueprint to convey different information (e.g., solid, dashed, dotted).

M

Machining Symbol: A symbol on a blueprint specifying the machining process required for a feature.

Machining Tolerance: A tolerance applied to a feature created by a machining process.

Material: The substance used to manufacture a component.

Material List: A list of materials required for a project, including types, quantities, and specifications.

Maximum Material Condition (MMC): When a feature contains the maximum amount of material allowed by the feature tolerance.

Metric System: A system of measurement based on meters and grams.

Metric Thread: A screw thread with dimensions specified in millimeters.

Mold Line: A line on a print or drawing indicating the parting line of a mold used to create a part.

Multi-View Drawing: A two-dimensional representation of a three-dimensional object using multiple views to convey the shape and size of the object.

N

Nominal Size: The basic size of a component, not necessarily the exact manufactured size.

O

Oblique Drawing: A method of drawing that shows three sides of an object by using lines of sight at an angle to the plane of projection.

Offset: The displacement of a feature from a reference point.

Orthographic Projection: A method of drawing that creates multiple two-dimensional views of an object on a flat surface.

Orthographic Projection Views: Standard views (front, top, side) of an object created using orthographic projection.

P

Pattern: A template used to create repetitive features on a part.

Pattern Development: The process of creating a flat pattern for a three-dimensional sheet metal part.

Perspective Drawing: A three-dimensional drawing showing the object as it appears to the eye.

Pictorial Drawing: A three-dimensional representation of an object.

Pilot Hole: A small hole drilled before creating a larger hole, which allows for better accuracy and alignment.

Pin: A cylindrical rod used to locate or secure parts.

Principal View: One of the six views of an object that are used in multi-view orthographic drawings.

Print Reading: The ability to read text and images to understand engineering drawings.

Projection: The representation of a three-dimensional object on a two-dimensional plane.

Projected View: A view of an object created by projecting its features onto a plane.

R

Radius: The distance from the center of a circle or arc to a point on its circumference.

Reference Dimension: A dimension that does not carry any tolerance.

Regardless of Feature Size (RFS): Where a tolerance zone is not affected by the size of the feature.

Revision Cloud: A wavy circle on a blueprint highlighting changes made to the design.

S

Scale: The ratio between the actual size of an object and its representation on a blueprint.

Screw: A threaded fastener that screws into a tapped hole.

Screw Thread Designation: Specifies the thread type, diameter, and pitch.

Section Line: Thin parallel lines at regular intervals indicating material that has been cut through by the cutting-plane line, often placed at a 45° angle.

Sectional View Types: Different types of sectional views (full, half, offset) depending on the information needed.

Section View: A view of an object showing its internal details by cutting through it along an imaginary plane.

Sheet Metal: Thin flat metal sections that are less than 3/16 inch (5 mm) thick.

Sheet Metal Gauge: A system for specifying the thickness of sheet metal.

Shop Drawing: A print or drawing used on the shop floor to guide manufacturing processes.

Shop Note: Additional instructions or clarifications added to a blueprint for manufacturing.

Slot: A long, narrow opening in a part.

Slot Dimension: Specifies the length, width, depth, and location of a slot.

Spotface (SF): To flatten a rough surface around a hole for a bolt head to contact.

Standard: Established specifications for materials, processes, or components.

Standard Hole Sizes: Standardized sizes for commonly used drilled holes.

Surface Finish: The smoothness or roughness of a machined surface.

Surface Texture Symbol: A symbol on a blueprint specifying the desired surface roughness.

Supplement: An additional sheet attached to a blueprint providing further details.

Symbol: A graphical representation on a blueprint conveying specific information (e.g., weld symbol, surface texture symbol).

Symmetrical: Equal halves, usually split with a centerline.

T

Tolerance: The allowable deviation from a specified dimension.

Tolerance Stack-Up: The cumulative effect of multiple tolerances on a final dimension.

Thread: A helical groove on a cylindrical surface used to create a fastening mechanism.

Three-Dimensional (3D) Model: A digital representation of an object that is created using computer software.

Third-Angle Projection: A method for creating two-dimensional drawings of three-dimensional objects projecting each view onto the plane closest to it.

Title Block: A section on a blueprint containing information like title, scale, revision number, and creator.

U

Uniform Tolerance: A single tolerance applied to all dimensions of a specific type (e.g., all hole diameters).

V

Visible Object Line: The thickest line and defines the visible edges and outlines of the object being depicted.

W

Weld Symbol: A symbol on a blueprint specifying the type and details of a weld.

Weld Zone: The area where two pieces of metal are joined by welding.