SECOND EDITION

Visualization, Modeling, and Graphics for ENGINEERING DESIGN

LIEU & SORBY

Visualization, Modeling, and Graphics for

ENGINEERING DESIGN

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ENGINEERING DESIGN

Second Edition

Dennis K. Lieu
Professor of Mechanical Engineering
University of California, Berkeley

Sheryl Sorby
Professor of Engineering Education
The Ohio State University

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Dennis K. Lieu and Sheryl Sorby

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20 Channel Center Street
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brief contents

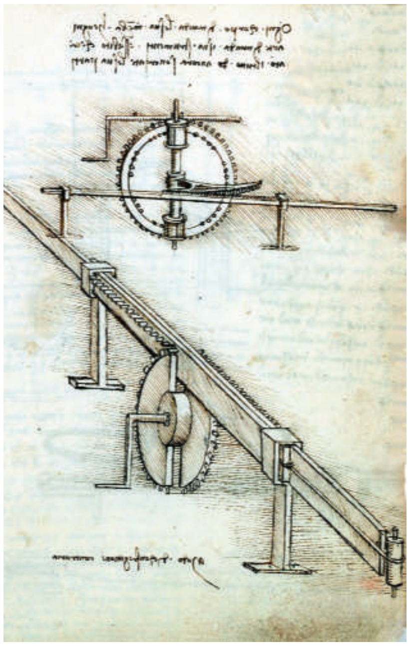
VISUALIZATION, MODELING, AND GRAPHICS FOR ENGINEERING DESIGN, SECOND EDITION

contents

THE FOLLOWING CHAPTERS WILL BE AVAILABLE IN OUR MINDTAP PRODUCT

preface

Leonardo da Vinci. You have probably learned that he was a famous Italian artist during the Renaissance. You may even subscribe to some of the conspiracy theories about him that have surfaced recently regarding secret codes and societies. What you may not know about him is that he was one of the very first engineers. (In fact, many people consider him to be *the* first engineer.) Some even say he was really an engineer who sometimes sold a painting in order to put food on the table. Artists played a prominent role at the birth of modern-day engineering, and some of the first artist-engineers included Francesco di Giorgio, Georg Agricola, and Mariano Taccola. These were the individuals who could visualize new devices that advanced the human condition. Their creativity and willingness to try seemingly "crazy" ideas propelled technology forward at a much faster pace than had occurred in the previous thousand years.



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This marriage between art and engineering has diminished somewhat since the early beginnings of the profession; however, creativity in engineering is still of paramount importance. Would the Apollo spaceship have landed on the moon without the creative thinking of hundreds of engineers who designed and tested the various systems necessary for space travel? Would we be able to instantaneously retrieve information and communicate with one another via the World Wide Web without the vision of the engineers and scientists who turned a crazy idea into reality? Would the modern-day devices that enrich and simplify our lives such as washing machines, televisions, telephones, and automobiles exist without the analytical skills of the engineers and technologists who developed and made successive improvements to these devices? The answer to all of these questions is "no." The ability to think of systems that never were and to design devices to meet the changing needs of the human population is the purview of the engineering profession.

Graphical communication has always played a central role in engineering, perhaps due to engineering's genesis within the arts or perhaps because graphical forms of communication convey design ideas more effectively than do written words. Maybe a picture really *is* worth a thousand words. As you might expect, the face of engineering graphics has evolved dramatically since the time of da Vinci. Traditional engineering graphics focused on two-dimensional graphical mathematics, drawing, and design; knowledge of graphics was considered a key skill for engineers. Early engineering programs included graphics as an integral topic of instruction, and hand-drawn engineering graphics from 50 years ago are works of art in their own right.

However, in the recent past, the ability to create a 2-D engineering drawing by hand has become de-emphasized due to improvements and advances in computer hardware and software. More recently, as computer-based tools have advanced even further, the demand for skills in 3-D geometric modeling, assembly modeling, animation, and data management has defined a new engineering graphics curriculum. Moreover, three-dimensional geometric models have become the foundation for advanced numerical analysis methods, including kinematic analysis, kinetic analysis, and finite element methods for stress, fluid, magnetic, and thermal systems.

The engineering graphics curriculum has also evolved over time to include a focus on developing 3-D spatial visualization ability since this particular skill has been documented as important to the success of engineers in the classroom and in the field. Spatial visualization is also strongly linked to the creative process. Would da Vinci have imagined his various flying machines without well-honed visualization skills?

We have come full circle in engineering education through the inclusion of topics such as creativity, teamwork, and design in the modern-day graphics curriculum. The strong link between creativity, design, and graphics cannot be overstated. Gone is the need for engineers and technicians who robotically reproduce drawings with little thought involved. With modern-day computational tools, we can devise creative solutions to problems without concern about whether a line should be lightly penciled in or drawn thickly and displayed prominently on the page.

It is for this new, back-to-the-future graphics curriculum that *Visualization, Modeling, and Graphics for Engineering Design, Second Edition* has been designed. This text is a mixture of traditional as well as modern-day topics, a mixture of analytical and creative thinking, a mixture of exacting drawing technique and freeform sketching. Enjoy.

Development of the Text

Many of the current graphics textbooks were written several years ago with modern-day topics such as feature-based solid modeling included as a separate add-on to the existing material. In these texts, modern-day computer-based techniques are more of an afterthought: "Oh, by the way, you can also use the computer to help you accomplish some of these common tasks." In fact, some of the more popular texts were written nearly a century ago when computer workstations and CAD software were figments of some forward-looking engineer's imagination. Texts from that era focused on drawing technique and not on graphic communication within the larger context of engineering design and creativity.

Modern engineering graphics curricula—and texts—must follow what is happening in the field. Modern product development techniques allow engineers to use computer hardware and software to examine the proper fit and function of a device. Engineers can "virtually" develop and test a device before producing an actual physical model, which greatly increases the speed and efficiency of the design process. The virtual, computer-based model then facilitates the creation of the engineering drawings used in manufacturing and production—an activity that required many hours of hand drafting just a few short decades ago.

In the real world, modern CAD practices have also allowed us more time to focus on other important aspects of the engineering design cycle, including creative thinking, product ideation, and advanced analysis techniques. Some might argue that these aspects are, in fact, the *most* important aspects of the design process. The engineering graphics curricula at many colleges and universities have evolved to reflect this shift in the design process. However, most engineering graphics textbooks have simply added CAD sections to cover the new topics. Thick textbooks have gotten even thicker. As a wise person has said, "Engineering faculty are really good at addition, but are miserable at subtraction."

When we sat down to plan this text, we wanted to produce the engineering graphics benchmark of the future—an engineering graphics approach that teaches design and design communication rather than a vocational text focused on drafting techniques and standards. We wanted to integrate modern-day design techniques throughout the text, not treat these topics as an afterthought.

A strength of this textbook is its focus not only on "what" to do, but also on "why" you do it (or do not do it) that way—concepts as well as details. This text is intended to be a learning aid as well as a reference book. Step-by-step software-specific tutorials, which are too focused on techniques, are very poor training for students who need to understand the modeling strategies rather than just which buttons to push for a particular task. In fact, we believe that mere *training* should be abandoned in favor of an *education* in the fundamentals. Students need to learn CAD strategy as well as technique. Students need to develop their creative skills and not have these skills stifled through a focus on the minutiae. In order to prepare for a lifelong career in this fast-changing technological world, students will need to understand fundamental concepts. For example, in the current methods-based approach to graphics training, students learn about geometric dimensioning and tolerancing. Many texts describe what the symbols mean, but do not explain how, why, and when they should be used. Yet, these questions of how, why, and when are the questions with which most young engineers struggle and the questions that are directly addressed in this text. They are also the *important* questions—if a student knows the answer to these questions, she or he will understand the fundamental concepts in geometric dimensioning and tolerancing. This fundamental understanding will serve far into the future where techniques, and possibly the symbols themselves, are likely to change.

Organization of the Text

The textbook is organized into 14 main chapters; a number of supplementary chapters are available in our MindTap product to create custom versions for specific course needs. In organizing the chapters of this text, we were careful to group topics in a way that reflects the modern engineering design process. We purposely did not mimic the decades-old graphics classical texts, which were written in an era when the design process was based on drawings and not computer models, an era when physical and not virtual models were analyzed for structural integrity. For this reason, the order of topics in this text will not match that of the traditional graphics texts where Lettering was often the first topic of instruction.

In this text, we start with foundational topics such as sketching and visualization since these are useful in the initial or "brainstorming" step in the design process and since these are fundamental topics on which many other topics hinge. Also included is a chapter on creativity and design. From there, we move to 3-D modeling because, in the real world, design typically begins with the production of a computer model. In the next stage of the modern design process, a computer model is analyzed either virtually or sometimes physically, and these topics are covered next. Once your model is complete and thoroughly analyzed, engineers move into the design documentation stage where drawings are created and annotated. The text is organized into four major sections as described subsequently. Supplemental chapters cover topics in traditional graphics instruction as well as some modern-day, "not quite ready for primetime" topics such as HTML and web utilization.

Section One—Laying the Foundation

The materials presented here focus on the needs of today's first-year engineering students who might have well-developed math and computer skills and less-developed hands-on mechanical skills. Incoming engineering students likely no longer work on their cars or bikes in the garage or may not have taken shop and drafting classes in high school. Hands-on tinkering is probably an activity of the past replaced by hands-on web page design and text messaging. (Engineering students of today in all probability have much greater dexterity in their thumbs from "texting" than do the authors of this text!) Although many engineering students enter college having spent time in a virtual computer environment, the lack of hands-on experiences that involve more than just the thumbs and that also involve real-life physical objects, often results in poorly developed three-dimensional visualization skills. In this section, these skills are explored and developed. This section also includes a project-oriented approach with inclusion of topics in design and creativity to prepare students for a lifetime of professional engineering practice.

Section Two—Modern Design Practice and Tools

The modern topics found in this section reflect the current state of design in industry. Solid modeling has revolutionized engineering graphics. The widespread availability of computers has made three-dimensional modeling the preferred tool for engineering design in nearly all disciplines. Solid modeling allows engineers to easily create mathematical models, parts, and assemblies, visualize and manipulate these models in real time, calculate physical properties, and inspect how they mate with other parts. The modern-day design process is characterized by computer methods that take advantage of the efficiency and advantages offered by workstations and feature-based modeling software. These new technologies have revolutionized the design process and have enabled around-the-clock engineering. By this model, engineers in Europe hand off (via the Internet) a design project when they leave work to American engineers. The Americans, in turn, hand off the design as they leave the office at the end of the day to Asian engineers. The Asian engineers complete the cycle by passing the design back to the Europeans at the end of their day. The sun never sets on an engineering design. Over your lifelong engineering career, the details of the design process may change again in ways that are unimagined today, but the fundamentals, as described in this section, will migrate from system to system with each advance in technology.

Section Three—Setting Up an Engineering Drawing

This section contains material found in most conventional textbooks on engineering graphics; however, the content is presented in novel ways and with a fresh approach to problem solving. The topics and techniques in this section are in wide use in engineering graphics classrooms today and are likely to continue to be invaluable into the foreseeable future. These traditional graphics topics continue to be important for several reasons. First, many legacy designs out there were produced prior to the feature-based solid modeling revolution. You may be asked to examine these designs, so it is important that you thoroughly understand drawings. Second, not every company has the capability to go directly from a computer model to a manufactured part, and drawings are still important in these environments. Finally, while the computer can usually automatically generate a drawing for you, certain conventions and dimensioning practices do not translate well. You will need to be able to verify the integrity of drawings that the computer generates for you and make changes where needed. For all of these reasons, no matter how sophisticated the computer design, hardware and software, or the manufacturing system, engineers must still be able to visualize a three-dimensional object from a set of two-dimensional drawings and vice versa.

Section Four—Drawing Annotation and Design Implementation

The ultimate goal of the engineering design process is to develop devices where everything fits together and functions properly. The sizes of the features that define an object are crucial to the overall functionality of the system. The chapters in this section describe how sizes and geometries of entities are specified. Since no part can be made to an "exact" size even with the best in computer technologies, the allowable errors for part sizes are also described in this section. The final drawings produced in preparation for fabrication must meet exacting criteria to ensure that they are properly cataloged and interpreted for clear communication among all parties. If your drawing includes nonstandard annotations, the machinist or contractor who uses those drawings to produce an engineered system may unknowingly misinterpret the drawing, resulting in higher project costs or even failure. The chapters in this section detail standard practice in drawing annotation to help you avoid making blunders. The ability to make proper, 100 percent correct drawing annotations will likely take you several years to develop. Be patient and keep learning.

Advanced Topics in Engineering Graphics

Additional chapters on topics in graphic communication are available in our MindTap product. A chapter is included to assist you with communicating your thoughts, ideas, analyses, and conclusions through animation, graphs, and charts. You may think that this type of communication is a "no brainer" with modern tools such as a spreadsheet. However, many times the automatically generated graph from a spreadsheet does not follow standard engineering practice for graphic communication and must be edited in order to meet these standards. For example, spreadsheets typically leave axis labels and titles off a graph resulting in a pretty, but meaningless, picture. A picture may be worth a thousand words, but sometimes it takes a few words to describe what a picture is illustrating. For communication with nontechnical (or sometimes even technical) audiences, tremendous amounts of information can be conveyed through the use of animation. If a picture is worth a thousand words, then an animation is surely worth a million.

Chapter Structure

With a few exceptions, each chapter is organized along similar lines. The material is presented with the following outline:

1. Objectives
   Chapter-opening objectives alert students to the chapter's fundamental concepts.
2. Introduction
   This section provides an overview of the material that will be presented in the chapter, and discusses why it is important.
3. The Problem
   Each chapter directly addresses a certain need or problem in graphical communication. That problem is presented here as if the student had to face such a problem in the field. The presence of a real problem that needs to be solved gives a student added incentive to learn the material in the chapter to solve that problem.
4. Explanation and Justification of Methods
   Engineering graphics has evolved and continues to evolve at an increasingly fast pace due to advances in computer hardware and software. Although new methods associated with new technologies exist, these modern methods must remain compatible with conventional graphics practices. This consistency is required to eliminate possible confusion in the interpretation of drawings, maintain sufficient flexibility to create designs unencumbered by the tools available to document them, and reduce the time and effort required to create the drawings.
5. Summary
   This section distills the most important information contained in the chapter.
6. Glossary of Key Terms
   Formal definitions of the most important terms or phrases for the chapter are provided. Each term or phrase is highlighted the first time it is used in the chapter.
7. Questions for Review
   These questions test the student's understanding of the chapter's main concepts.
8. Problems
   A variety of problems and exercises help to develop skill and proficiency of the material covered in the chapter.

Key Features of the Presentation

We believe that this text will have a broad appeal to engineering graphics students across a wide spectrum of institutions. The following are key features of the text:

- *A focus on learning and fundamental skill development, not only on definitions, tools, and techniques.* This approach prepares students to apply the material to unfamiliar problems and situations rather than simply to regurgitate previously memorized material. In the fast-changing world we live in, an understanding of the fundamentals is a key to further learning and the ability to keep pace with new technologies.

- *Formal development of visualization skills as a key element at an early stage of the curriculum.* Development of these skills is important for students who have not had the opportunity to be exposed to a large number of engineering models and physical devices. Further, the link between visualization and creativity is strong—tools for success over a lifelong career.
- *Use of a problem-based approach.* This approach presents the student with real problems at the beginning of each chapter, shows the graphics solutions, and then generalizes the solutions.
- *A casual tone and student-friendly approach.* It is a proven fact that students learn the material better if they are not fast asleep!
- *Several common example threads and a common project that are presented in most chapters.* The text shows how the material contained in each chapter was actually applied in the context of product development.

One of the case studies to be presented, for example, is the Hoyt Aerotec™ Olympic style recurve target bow. The unique geometry of this bow was brought to prominence after it was part of the equipment package used to win the Gold Medal in target archery at the summer Olympic Games in Sydney, Australia. The design history of the development of this product is traced starting from its ideation as an improvement to all other target bows on the market at that time. As a student moves through the chapters of this book, the progress of the development of this product will be documented. This product was selected as an example for these reasons:

1. It was a very successful design that accomplished all of the goals set forth by its engineers.
2. It was also a product that is relatively unencumbered by the complexity of mechanisms or electronics, which are not the focus of this book.
3. The design is mature, having made it to the consumer market; this means it offers an opportunity to study some of the nontechnical issues that play an important role in engineering design.

Final Remarks

This textbook contains a "core" of material covered in a traditional engineering graphics course and also a number of other chapters on modern graphics techniques. The collected material represents over 50 combined years of personal experience in the learning, application, and teaching of engineering graphics. The result is a text that should appeal to both traditional and contemporary graphics curricula. We, the authors, would like to thank you for considering this text.

Dennis K. Lieu
Professor of Mechanical Engineering
University of California, Berkeley

Sheryl Sorby
Professor of Engineering Education
The Ohio State University

New to This Edition

Chapter 1
- The People and Their Skills (Section 1.03) has been removed from the print edition. It can be found in the MindTap course.

Chapter 2
- Strategies for Simple Pictorial Sketches (Section 2.11) has been removed from the print edition. It can be found in the MindTap course. Caution (pp. 2-27 through 2-29) has been removed from the print edition. It can be found in the MindTap course.

Chapter 3
- New section added discussing the importance of spatial skills.
- Strategies for Developing 3-D Visualization Skills (Section 3.12) has been removed from the print edition. It can be found in the MindTap course.
- Caution (pp. 3-48 through 3-50) has been removed from the print edition. It can be found in the MindTap course.

Chapter 4
- Formerly Chapter 5.
- Patents (Section 5.06) has been removed from the print edition. It can be found in the MindTap course.

Chapter 5
- Formerly Chapter 6.
- Strategies for Making a Model (Section 6.11) has been removed from the print edition. It can be found in the MindTap course.
- Caution (pp. 6-71 through 6-78) has been removed from the print edition. It can be found in the MindTap course.
- Problems 5.1 and 5.4 are new.

Chapter 6
- Formerly Chapter 7.
- The vise assembly example has been removed from the book and is located in the MindTap course.
- Caution (pp. 7-30 through 7-31) has been removed from the print edition. It can be found in the MindTap course.
- Problems 6.2 and 6.5 thru 6.14 are new.

Chapter 7
- Formerly Chapter 8.
- The Finite Element Analysis

Process (Section 8.07) has been removed from the print edition. It can be found in the MindTap course.
- All problems are new.

Chapter 8
- Formerly Chapter 10.
- Strategies for Creating Multiviews from Pictorials (Section 10.06) has been removed from the print edition. It can be found in the MindTap course.
- Caution (pp. 10-51 through 10-62) has been removed from the print edition. It can be found in the MindTap course.
- Problems 8.3 and 8.5 through 8.7 are new.

Chapter 9
- Formerly Chapter 12.
- Step-by-step content has been removed from the print edition. It can be found in the MindTap course.

Chapter 10
- Formerly Chapter 13.
- Procedures for the Creation of Section Views (Section 13.08) has been removed from the print edition. It can be found in the MindTap course.
- Caution (pp. 13-44 through 13-51) has been removed from the print edition. It can be found in the MindTap course.
- Problems 10.5 through 10.9 are new.

Chapter 11
- Formerly Chapter 14.
- Caution (pp. 14-13 through 14-14) has been removed from the print edition. It can be found in the MindTap course.
- Sketching Techniques for Auxiliary Views (Section 14.06) has been removed from the print edition. It can be found in the MindTap course.
- All new problems.

Chapter 12
- Formerly Chapter 15.
- Problems 12.2 and 12.3 are new.

Chapter 13
- Formerly Chapter 16.
- Caution (pp. 16-44 through 16-48) has been removed from the print edition. It can be found in the MindTap course.
- Examples of Specifying Fits and Geometric Tolerances (Section 16.07) has been removed from the print edition. It can be found in the MindTap course.

Chapter 14
- Formerly Chapter 18.
- Caution (pp. 18-57 through 18-70) has been removed from the print edition. It can be found in the MindTap course.
- All new problems.

Online Content
- Former Chapter 4: Working in a Team Environment has been removed from the print edition. It can be found in the MindTap course.
- Former Chapter 9: Fabrication Processes has been removed from the print edition. It can be found in the MindTap course.
- Former Chapter 11: Advanced Visualization Techniques has been removed from the print edition. It can be found in the MindTap course.
- Former Chapter 17: Fasteners has been removed from the print edition. It can be found in the MindTap course.

Contributors

Holly K. Ault, Worcester Polytechnic Institute (Chapters 5 and 18)

Ron Barr, The University of Texas at Austin (Chapters 4 and 7)

Judy Birchman, Purdue University (Chapter 20)

Ted Branoff, North Carolina State University (Chapters 13 and 18)

Frank Croft, The Ohio State University (Chapter 9)

La Verne Abe Harris, Purdue University (Chapter 20)

Kathy Holliday-Darr, Penn State University, Erie (Chapter 11)

Tom Krueger, The University of Texas at Austin (Chapters 4 and 7)

Jim Morgan, Texas A&M University (Chapter 15)

Bill Ross, Purdue University (Chapter 19)

Mary Sadowski, Purdue University (Chapter 20)

Kevin Standiford, Consultant (Supplemental Chapters 2 and 3)

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Tamara W. Knott
Virginia Tech

Kellen Maicher
Purdue University

Jim Morgan
Texas A&M University

William A. Ross
Purdue University

Mary Sadowski
Purdue University

James Shahan
Iowa State University

Michael Stewart
Georgia Tech

about the authors

Dennis K. Lieu

Professor Dennis K. Lieu was born in 1957 in San Francisco, where he attended the public schools, including Lowell High School. He pursued his higher education at the University of California at Berkeley, where he received his BSME in 1977, MSME in 1978, and D.Eng. in mechanical engineering in 1982. His major field of study was dynamics and control. His graduate work, under the direction of Professor C. D. Mote, Jr., involved the study of skier/ski mechanics and ski binding function. After graduate studies, Dr. Lieu worked as an advisory engineer with IBM in San Jose CA, where he directed the specification, design, and development of mechanisms and components in the head-disk-assemblies of disk files. In 1988, Dr. Lieu joined the Mechanical Engineering faculty at UC Berkeley. His research laboratory is engaged in research on the mechanics of high-speed electromechanical devices and magnetically generated noise and vibration. His laboratory also studies the design of devices to prevent blunt trauma injuries in sports, medical, and law enforcement applications. Professor Lieu teaches courses in Engineering Graphics and Design of Electromechanical Devices. He was the recipient of a National Science Foundation Presidential Young Investigator Award in 1989, the Pi Tau Sigma Award for Excellence in Teaching in 1990, the Berkeley Distinguished Teaching Award (which is the highest honor for teaching excellence on the UC Berkeley campus) in 1992, and the Distinguished Service Award from the Engineering Design Graphics Division of ASEE in 2015. He is a member of Pi Tau Sigma, Tau Beta Pi, and Phi Beta Kappa. His professional affiliations include ASEE and ASME. Professor Lieu's hobbies include Taekwondo (in which he holds a 4th degree black belt) and Olympic style archery.

Sheryl Sorby

Professor Sheryl Sorby is not willing to divulge the year in which she was born but will state that she is younger than Dennis Lieu. She pursued her higher education at Michigan Technological University receiving a BS in Civil Engineering in 1982, an MS in Engineering Mechanics in 1985, and a Ph.D. in Mechanical Engineering-Engineering Mechanics in 1991. She was a graduate exchange student to the Eidgenoessiche Technische Hochshule in Zurich, Switzerland, studying advanced courses in solid mechanics and civil engineering. She is currently a Professor of Engineering Education at The Ohio State University and a Professor Emerita of Mechanical Engineering-Engineering Mechanics at Michigan Technological University. Dr. Sorby is the former Associate Dean for Academic Programs and the former Department Chair of Engineering Fundamentals at Michigan Tech. She has also served as a Program Director in the Division of Undergraduate Education at the National Science Foundation. She served as a Fulbright Scholar to the Dublin Institute of Technology to conduct research in Engineering Education. Her research interests include various topics in engineering education, with emphasis on graphics and visualization. She was the recipient of the Betty Vetter research award through the Women in Engineering Program Advocates Network (WEPAN) for her work in improving the success of women engineering students through the development of a spatial skills course. She received the Sharon Keillor award for outstanding women in engineering education in 2011. She has also received the Engineering Design Graphics Distinguished Service Award, the Distinguished Teaching Award, and the Dow Outstanding New Faculty Award, all from ASEE.

Dr. Sorby currently serves as an Associate Editor for ASEE's online journal, *Advances in Engineering Education*. She is a member of the Michigan Tech Council of Alumnae. She has been a leader in developing first-year engineering and the Enterprise program at Michigan Tech and is the author of numerous publications and several textbooks. Dr. Sorby's hobbies include golf and knitting.

SECTION ONE

LAYING THE FOUNDATION

The materials presented in this section focus on the needs of today's beginning engineering students, who typically have well-developed math and computer skills but less-developed, hands-on mechanical skills compared to students of earlier generations. Incoming engineering students may no longer be people who work on their cars or bikes in the garage and who took shop and drafting classes in high school. Although many engineering students enter college having spent time in a virtual computer environment, the lack of hands-on experience often results in a lack of three-dimensional visualization skills. So in addition to the classical material on standard engineering graphics practices, these students need to enhance their visualization skills. Prior to the advent of CAD, the graphics classroom featured large tables topped with mechanical drafting machines and drawers full of mechanical drawing instruments. Engineering students and engineers now have additional time to focus on aspects of the engineering design cycle that are more worthy of their talents as engineers. These aspects include creative thinking, product ideation, and advanced analysis techniques to ensure a manufacturable and robust product. Formalization of the design process allows designers to focus their energies on certain areas in the process and gain more meaningful results.

CHAPTER 1

AN INTRODUCTION TO GRAPHICAL COMMUNICATION IN ENGINEERING

OBJECTIVES

After completing this chapter, you should be able to

- Explain and illustrate how engineering graphics is one of the special tools available to an engineer
- Define how engineering visualization, modeling, and graphics are used by engineers in their work
- Provide a short history of how engineering graphics, as a perspective on how it is used today, was used in the past

1.01 INTRODUCTION

Because engineering graphics is one of the first skills formally taught to most engineering students, you are probably a new student enrolled in an engineering program. Welcome!

You may be wondering why you are studying this subject and what it will do for you as an engineering student and, soon, as a professional engineer. This chapter will explain what engineering is, how it has progressed over the years, and how graphics is a tool for engineers.

What exactly is engineering? What does an engineer do? The term *engineer* comes from the Latin *ingenerare*, which means "to create." You may be better able to appreciate what an engineer does if you consider that *ingenious* also is a derivative of *ingenerare*. The following serves well enough as a formal definition of engineering:

The profession in which knowledge of mathematical and natural sciences, gained by study, experience, and practice is applied with judgment to develop and utilize economically the materials and forces of nature for the benefit of humanity.

A modern and informal definition of engineering is "the art of making things work." An engineered part or an engineering system does not occur naturally. It is something that has required knowledge, planning, and effort to create.

So where and how does graphics fit in? Engineering graphics has played three roles through its history:

1. Communication
2. Record keeping
3. Analysis

First, engineering graphics has served as a means of communication. It has been used to convey concepts and ideas quickly and accurately from one person to another without the use of words. As more people became involved in the development of products, accurate and efficient communication became increasingly necessary. Second, engineering graphics has served as a means of recording the history of an idea and its development over time. As designs became more complex, it became necessary to record the ideas or features that worked well in a design so they could be repeated in future applications. And third, engineering graphics has served as a tool for analysis to determine critical shapes and sizes, as well as other variables needed in an engineered system.

These three roles are still vital today, more so than in the past, because of the technical complexity required in making modern products. Computers, three-dimensional modeling, and graphics software have made it increasingly effective to use engineering graphics as an aid in design, visualization, and optimization.

1.02 A Short History

The way things are done today evolved from the way things were done in the past. You can understand the way engineering graphics is used today by examining how it was used in the past. Graphical communications has supported **engineering** throughout history. The nature of engineering graphics has changed with the development of new graphics tools and techniques.

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FIGURE 1.01. Undated cave painting showing hunting and the use of tools.

1.02.01 Ancient History

The earliest documented forms of graphical communication are cave paintings, such as the one shown in Figure 1.01, which showed human beings depicting organized social behavior, such as living and hunting in groups. The use of tools and other **fabricated** items for living comfort and convenience were also communicated in cave paintings. However, these paintings typically depicted a lifestyle, rather than any instructions for the fabrication of tools, products, or structures. How the items were made is still left to conjecture.

The earliest large structures of significance were the Egyptian pyramids and Native American pyramids. Some surviving examples are shown in Figure 1.02. The Egyptian pyramids were constructed as tombs for the Pharaohs. The Native American pyramids were built for religious ceremonies or scientific use, such as observatories. Making these large structures, with precision in the fitting of their parts using the tools that were available at the time, required much time, effort, and planning. Even with modern tools and construction techniques, these structures would be difficult to

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FIGURE 1.02. Mayan pyramid, Yucatan, Mexico (left), and Pharaoh Knufu and Pharaoh Khafre Pyramids, Giza, Egypt (right).

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FIGURE 1.03. Ancient Egyptian hieroglyphics describing a life story.

re-create today. The method of construction for the pyramids is largely unknown—records of the construction have never been found—although there have been several theories over the years.

Egyptian hieroglyphics, which were a form of written record, included the documentation of a few occupational skills, such as papermaking and farming, although, for the most part, they documented lifestyle. An example of a surviving record is shown in Figure 1.03. As a result of those records, papermaking and farming skills could be maintained and improved over time. Even people who were not formally trained in those skills could develop them by consulting the written records.

Two engineering construction methods helped the Roman Empire expand to include much of the civilized European world. These methods were used to create the Roman arch and the Roman road.

The Roman arch, shown in Figure 1.04, was composed of stone that was precut to prescribed dimensions and assembled into an archway. The installation of the keystone at the top of the arch transferred the weight of the arch and the load it carried into the remaining stones that were locked together with friction. This structure took advantage of the compressive strength of stone, leading to the creation of large structures that used much less material. The Roman arch architecture was used to create many large buildings and bridges. Roman-era aqueducts, which still exist today in Spain and other countries in Europe, are evidence of the robustness of this **design**.

The method used to construct Roman roads prescribed successive layers of sand, gravel, and stone (instead of a single layer of the native earth), forming paths wide enough for commercial and military use. In addition to the layered construction methods, these roads were also crowned to shed rain and had gutters to carry away water. This construction method increased the probability that the roads would not become overgrown with vegetation and would remain passable even in adverse weather. As a result, Roman armies had reliable access to all corners of the empire.

Photos by William G. Godden. Reprinted with permission from EERC Library, Univ. of California, Berkeley.

Photos by William G. Godden. Reprinted with permission from EERC Library, Univ. of California, Berkeley.

FIGURE 1.04. Pont-du-Gard Roman aqueduct (left) built in 19 BC to carry water across the Gardon Valley to Nimes. Spans of the first- and second-level arches are 53–80 feet. The Ponte Fabricio Bridge in Rome (right) built in 64 BC spans the bank of the River Tiber and Tiber Island.

The Roman Empire is long gone, but the techniques used for the construction of the Roman arch and the Roman road are still in use today. The reason for the pervasiveness of those designs was probably due to Marcus Vitruvius, who, during the Roman Empire, took the trouble to carefully document how the structures were made.

The Archimedes screw, used to raise water, is an example of a mechanical invention developed during the time of the Greek Empire. Variations of the device were used for many centuries because diagrams depicting its use were (and still are) widely available. One of those diagrams is shown in Figure 1.05. These early documents were precursors to modern engineering **drawings**. Because the documents graphically communicated how to build special devices and structures, neither language nor language translation was necessary.

FIGURE 1.05. An engraving showing the operation of an Archimedes screw to lift water.

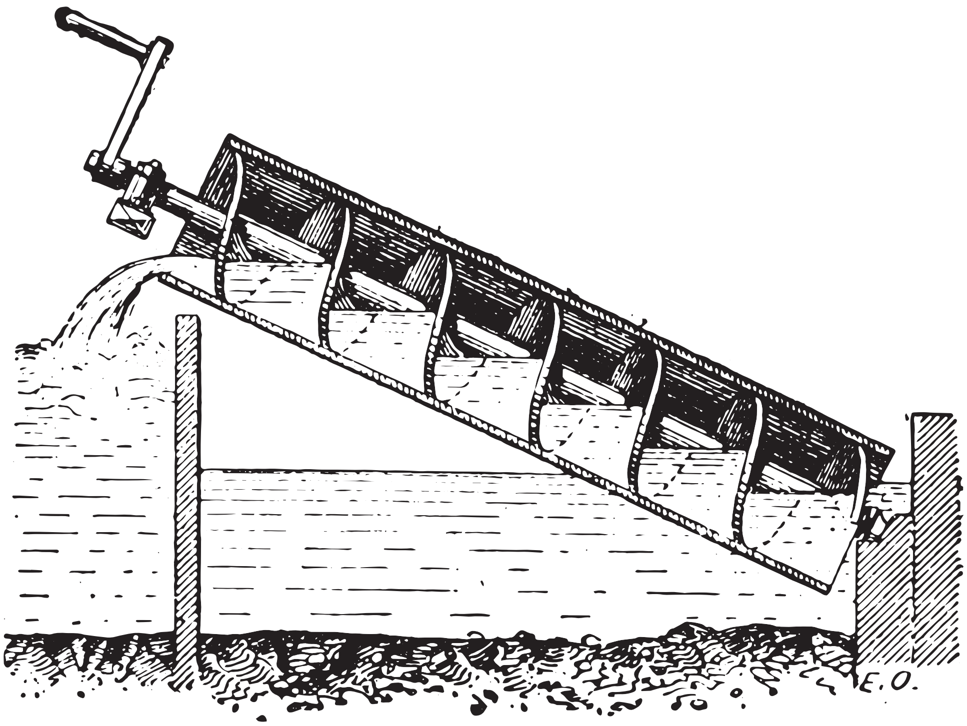

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FIGURE 1.06. Flying buttress construction used to support the exterior walls of Notre Dame Cathedral in Paris.

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1.02.02 The Medieval Period

Large building construction helped define the medieval period in Europe. Its architecture was more complicated than the basic architecture used for the designs of ancient buildings. The flying buttress, a modification of the Roman arch, made it possible to construct larger and taller buildings with cavernous interiors. This type of structure was especially popular in Europe for building cathedrals, such as the one shown in Figure 1.06. The walls of fortresses and castles became higher and thicker. Towers were included as an integral part of the walls, as shown in Figure 1.07, to defend the inhabitants from many directions, even when attackers had reached the base of a wall.

FIGURE 1.07. Warwick Castle, England, circa 1350, is an example of a medieval style fortification.

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FIGURE 1.08. The Great Wall of China, built during the medieval period, used simple engineering principles despite the large scale of the project.

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In Asia, large fortifications, shrines, and temples, as shown in Figure 1.08, were built to last hundreds of years. The complexity of techniques to build those structures required planning and documentation, especially when raw materials had to be transported from long distances. Building structures of such sizes required an understanding of the transmission of forces among the supporting members and the amount of force those members could withstand. That knowledge was especially important when wood was the primary building material.

Large-scale civil engineering **projects** were begun during the medieval era. Those projects were designated by a civilian government to benefit large groups or the general population, as opposed to projects constructed for private or military use. The windmills of Holland, shown in Figure 1.09, are an example of a civil engineering project. The windmills harvested natural wind energy to pump large amounts of water out of vast swampland, making the land suitable for farming and habitation.

Windmills and waterwheels were used for a variety of tasks, such as milling grain and pumping water for irrigation. Both inventions were popular throughout Europe and Asia—a fact that is known because diagrams showing their construction and use have been widely available.

1.02.03 The Renaissance

The beginning of the Renaissance in the 1400s saw the rise of physical scientific thinking, which was used to predict the behavior of physical **systems** based on empirical observation and mathematical relationships. The most prominent person among the scientific physical thinkers at that time was Leonardo da Vinci, who documented his ideas in drawings. Some of those drawings, which are well known today, are shown in Figure 1.10. Many of his proposed devices would not have worked in their original form, but his drawings conveyed new ideas and proposals as well as known facts.

Prior to the Renaissance, nearly all art and diagrams of structures and devices were records of something already in existence or were easily extrapolated from something already tried and known to work. When inventors applied physical science to engineering, they could conceive things that theoretically should have worked without having been previously built. When inventors did not understand the science

FIGURE 1.09. The network of windmills in Holland, used to drain water from flooded land, is an example of an early large-scale civil engineering project.

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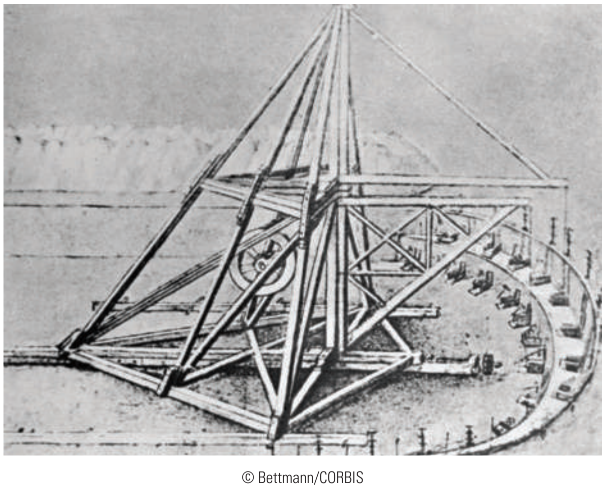

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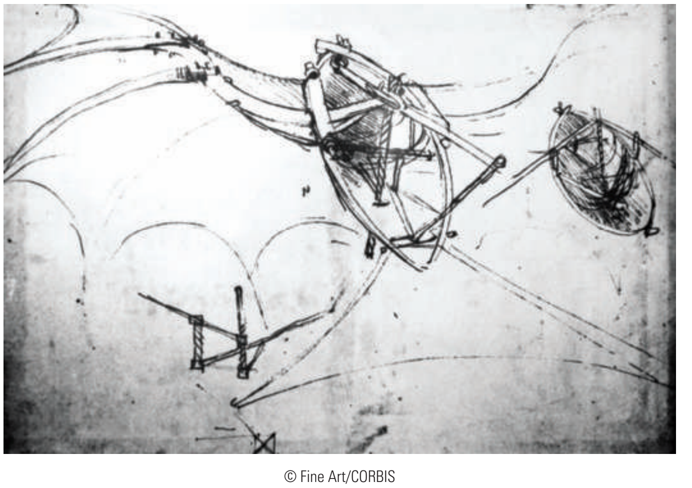

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FIGURE 1.10. Images of original da Vinci drawings: a machine used for canal excavation (left) and a flying ship (right). Codex Atlanticus, folio 860; drawing from Il Codice Atlantico di Leonardo da Vinci nella biblioteca Ambrosiana di Milano, Editore Milano Hoepli 1894–1904; the original drawing is kept in the Biblioteca Ambrosiana in Milan.

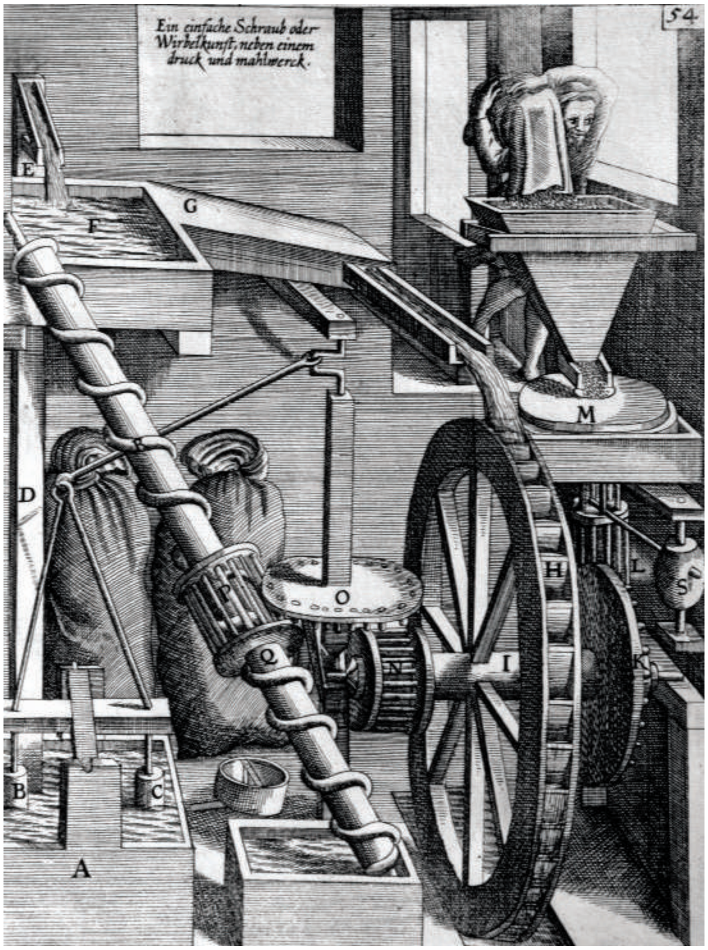


FIGURE 1.11. A perpetual motion machine by medieval inventors: an Archimedes screw driven by a waterwheel is used to mill grain.

Timewatch Images/Alamy

behind the proposed devices, the devices usually did not work. The many perpetual motion machines proposed at that time, as shown in Figure 1.11, are evidence of inventors' lack of understanding of the physical science and their resultant failed attempts to build the machines.

Engineers began to realize that accurate sizing was an element of the function of a structure or device. Diagrams made during the Renaissance paid more attention to accurate depth and perspective than in earlier times. As a result, drawings of both proposed and existing devices looked more realistic than earlier drawings.

Gunpowder was introduced during the Renaissance, as was the cannon. The cannon made obsolete most of the fortresses built during the medieval era. The walls could not withstand the impact from cannon projectiles. Consequently, fortresses needed to be redesigned to survive cannon fire. In France, a new, stronger style of fortification was designed. The fortification was constructed with angled walls that helped to deflect cannon fire and did not crumble as flat vertical walls did when struck head on. The new fortresses were geometrically more complicated to build than their predecessors with vertical walls. Further, the perimeter of the fortress had evolved from a simple rectangular shape to a pentagonal shape with a prominent extension at each apex. That perimeter shape, coupled with the angled walls, resulted in walls that intersected at odd angles that could not be seen and measured easily or directly. Following is a list of questions that builders of earlier fortresses could easily answer but that builders of the angled wall fortresses could not:

- What is the surface area of a wall?
- What is the fill volume?

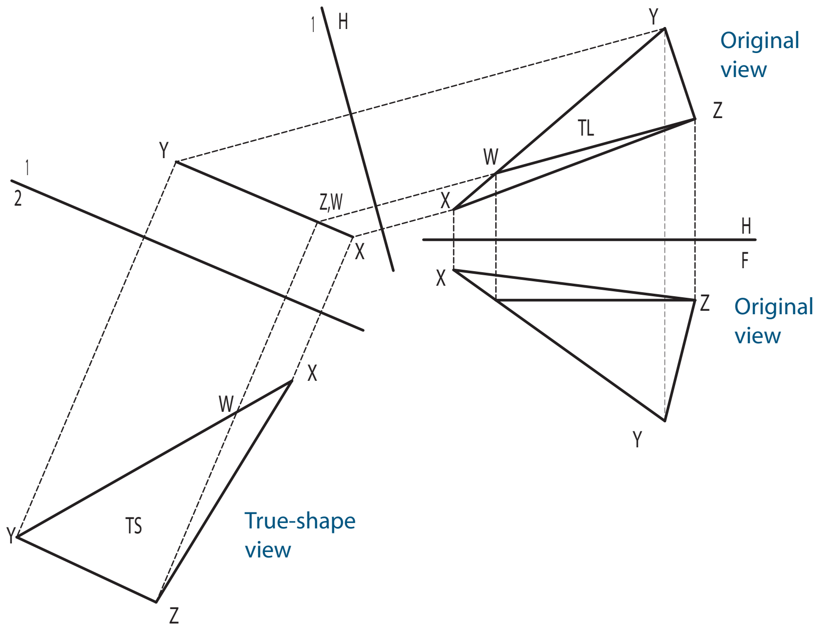
- What are the specific lengths of timbers and beams needed to construct and brace the walls?
- What are the true angles of intersection between certain surfaces?
- What are the distances between lines and other lines, between points and lines, and between points and surfaces?

Fortunately, the French had Gaspard Monge, who developed a graphical analysis technique called **descriptive geometry**. Analytical techniques using mathematics were not very sophisticated at that time, nor were machines available to do mathematical calculations. But mechanical **instruments**, such as compasses, protractors, and rulers, together with the graphical method, were used to analyze problems without the need to do burdensome math. Descriptive geometry techniques enabled engineers to create any view of a geometric object from two existing views. By creating the proper view, engineers could see and measure an object's attributes, such as the true length of its lines, the true shape of planes, and true angles of intersection. Such skills were necessary, especially for the construction of fortifications, as shown in Figure 1.12. The complex geometry, odd angles of intersection, and height of walls were intended to maximize the cross fire on an approaching enemy, while not revealing the interior of the fortress. Another objective was to construct the ramparts and walls by moving the minimum amount of material for maximum economy.

The astuteness of the French at building fortifications kept France the prime military power in Europe until the 1700s. At that time, descriptive geometry was considered a French state secret; divulging it was a crime punishable by death. As a result of the alliance between France and the newly constituted United States, many U.S. fortifications used French designs. An example is Fort McHenry (shown in Figure 1.13), which was built in 1806 and is exquisitely preserved in Baltimore, Maryland. Fort McHenry survived bombardment by the British during the War of 1812 and is significant because it inspired Francis Scott Key to write "The Star Spangled Banner."

By the 1800s, most engineering was either civil engineering or military engineering. Civil engineering specialized in the construction of buildings, bridges, roads, commerce ships, and other structures, primarily for civilian and trade use. Military engineering specialized in the construction of fortifications, warships, cannons, and other items for military use. In both fields of engineering, as projects became more

FIGURE 1.12. Using descriptive geometry to find the area of a plane.



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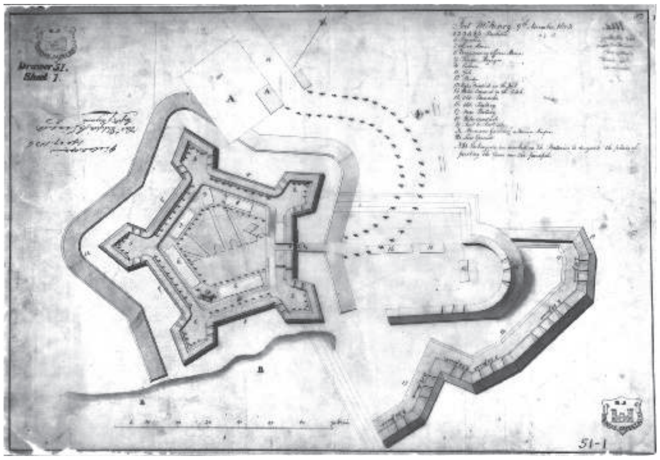


Courtesy of The National Archives, College Park, Maryland

Courtesy of The National Park Service, Fort McHenry NMHS

FIGURE 1.13. French fortification design principles (left) and Fort McHenry (right) in Baltimore, Maryland, whose design was based on those principles.

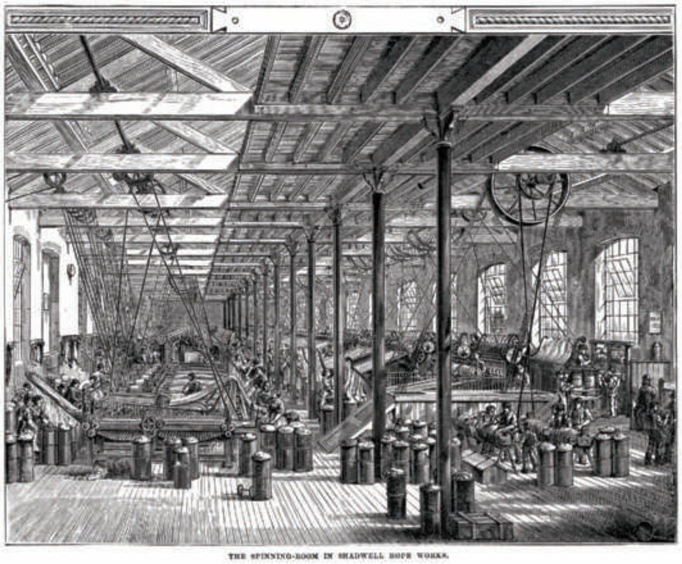
complicated, more people skilled in various subspecialties were needed. Clear, simple, and universal communication was necessary to coordinate and control the efforts of specialists interacting on the same project. Different people needed to know what other people were doing in order for various **parts** and **assemblies** to fit together and function properly. To fill that need, early forms of scaled drawings began to be used as the medium for communications in constructing a building or device.

1.02.04 The Industrial Revolution

The industrial revolution began in the early 1800s with the new field of mechanical engineering. This revolution was, in part, a result of the need for new military weapons. Before the 1800s, ships and guns were fabricated one at a time by skilled craftsmen. No original plans of any ships from the Age of Discovery exist, because shipwrights did not use plans drawn on paper or parchment. The only plans were in the master shipwright's mind, and ships were built by eye. As the demand for ships grew, production methods changed. It was far more economical to build many ships using a single design of common parts than to use a custom design for each ship. Constructing from a common design required accurate specifications of the parts that went into the design.

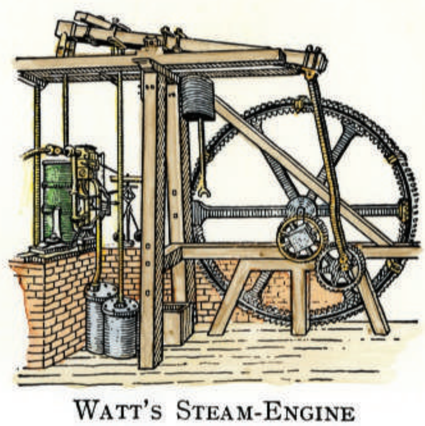
The hardware products that general and military consumers needed then were no longer produced by skilled craftsmen but were mass-produced according the techniques and machines specified by engineers. Mass production meant that each product had to be identical to all other products, had to be fabricated within predictable and short production times, had to be made from parts that were interchangeable, and had to be produced economically in volumes much larger than in the past. The consistent and repetitive motions of machines required efficient, large-scale production which replaced manufacturing operations that had needed the skilled motions of craftsmen. Also, engines, boilers, and pressure vessels were required to provide power to machines. An early manufacturing facility with machine tools and an early steam engine are shown in Figures 1-14 and 1-15, respectively.

Creating not only a product but also the machines to produce it was beyond the abilities of individual craftsmen—each likely to have a different set of skills needed for the production of a single product. The high demand for creating machines as well as products meant that the existing master-apprentice relationship could no longer supply the demand for these skills. To meet the growing demand, engineering schools had to teach courses in basic physics, machine-tool design, physical motion, and energy transfer.



© Duncan Walker/Getty Images

FIGURE 1.14. A photo showing early factory conditions during the industrial revolution.



North Wind Picture Archives/Alamy

FIGURE 1.15. A schematic drawing of a James Watt steam engine; the type commonly used to power production machinery during the early years of the industrial revolution.

Communication was necessary to coordinate and control the efforts of different people with different skills. Each craftsman, as well as each worker, on a project needed to know what others were doing so the various pieces, devices, structures, and/or systems would fit together and function properly. The ideas of the master designer had to be transferred without misinterpretation to those who worked at all levels of supporting roles. In the design stage, before things were actually built, the pictorial diagrams once used were soon found to be insufficient and inaccurate when new structures with new techniques were being built. More accurate representations, which would provide exact sizes, were needed. That need eventually led to the modern engineering drawing, with its multiple-view presentation, identification of sizes, and specification of allowable errors.

Around the time of the industrial revolution, patents started to become important. As a method of stimulating innovation in an industrialized society, many governments offered patents to inventors. The owners of patents were guaranteed exclusive manufacturing rights for the device represented in the patent for a prescribed number of years in exchange for full disclosure of how the device operated. Since a single successful patented invention could make its owner rich, many people were inspired to create new products. From the start, the difference between patent drawings and engineering has been that engineering drawings are made to be viewed by those formally trained in engineering skills and to show precise sizes and locations. Patent drawings, on the other hand, are made to teach others how and why a device operates. Consequently, patent drawings often do not show the actual or scaled sizes of the parts. In fact, sizes are commonly distorted to make the device more difficult for potential competitors to copy. An example of a patent drawing is shown in Figure 1.16.

1.02.05 More Recent History

As technology advanced over time, additional engineering specialties were born. In the late 1800s, as electric power became more popular and more available, electrical engineering was born. Electrical engineering at that time was concerned with the production, distribution, and use of electrical energy. The information derived from the study of electric motors, generators (shown in Figure 1.17), power conversion,

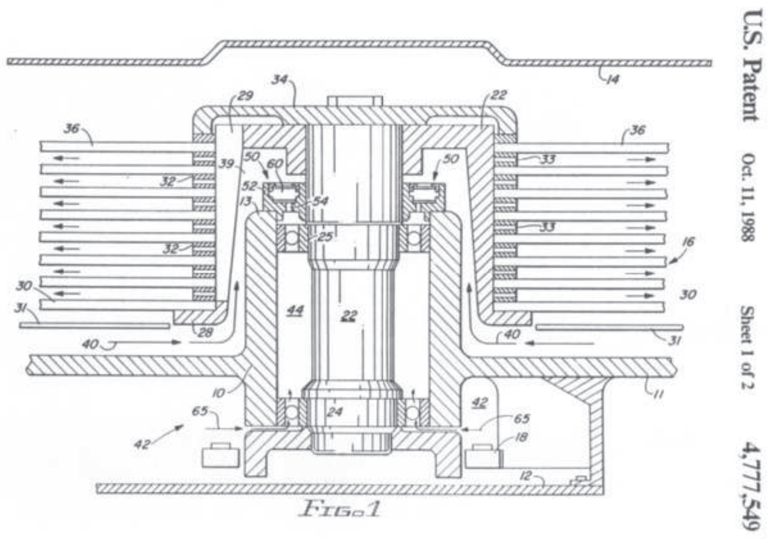


FIGURE 1.16. A U.S. patent drawing showing function but not necessarily the true sizes of the parts.

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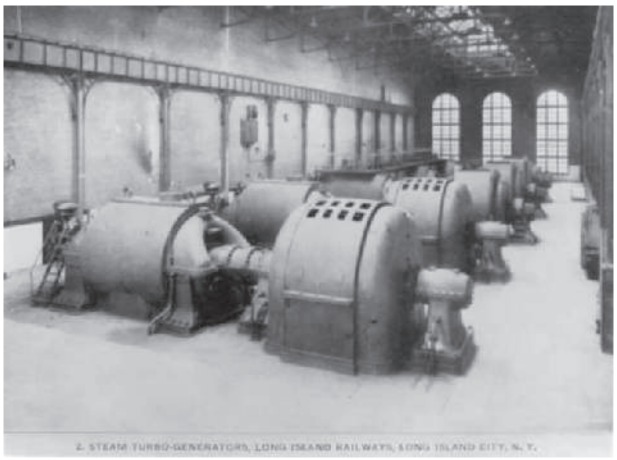
and transmission lines needed for their design was more than other engineers—not specifically electrical engineers—could be expected to know and use. Chemical engineering, as a special engineering discipline, emerged at the beginning of the twentieth century with the need for large-scale production of petroleum products in refineries, as shown in Figure 1.18, and the production of synthetic chemicals.

During the 1950s, industrial engineering and manufacturing engineering emerged from the necessity to improve production quality, control, and efficiency. Nuclear engineering emerged as a result of the nuclear energy and nuclear weapons programs.

Some of the more recent engineering disciplines include bioengineering, information and computational sciences, micro-electro-mechanical systems (MEMS), and nano-engineering. The design of a MEMS device (for example, the valve shown in Figure 1.19) requires skills from both electrical and mechanical engineering. A nano-engineered device cannot be seen with conventional optics. Its presumed appearance, such as that shown in Figure 1.20, and function are based on conjecture using engineering graphics tools.



Courtesy of SMITHSONIAN INSTITUTION Neg. #44191D

FIGURE 1.17. Later during the industrial revolution, steam engines were replaced by electric power supplied, for example, by these generators at the Long Island Railway (circa 1907). Electrical engineering was born.

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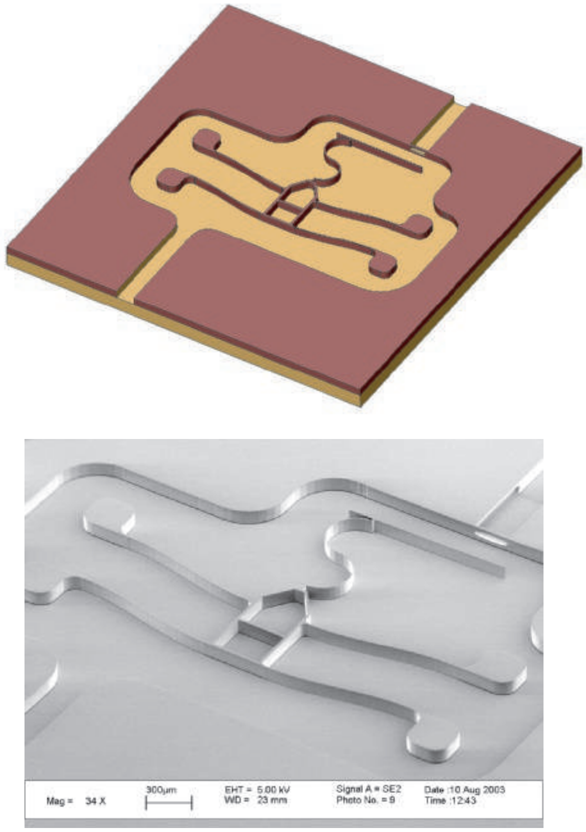
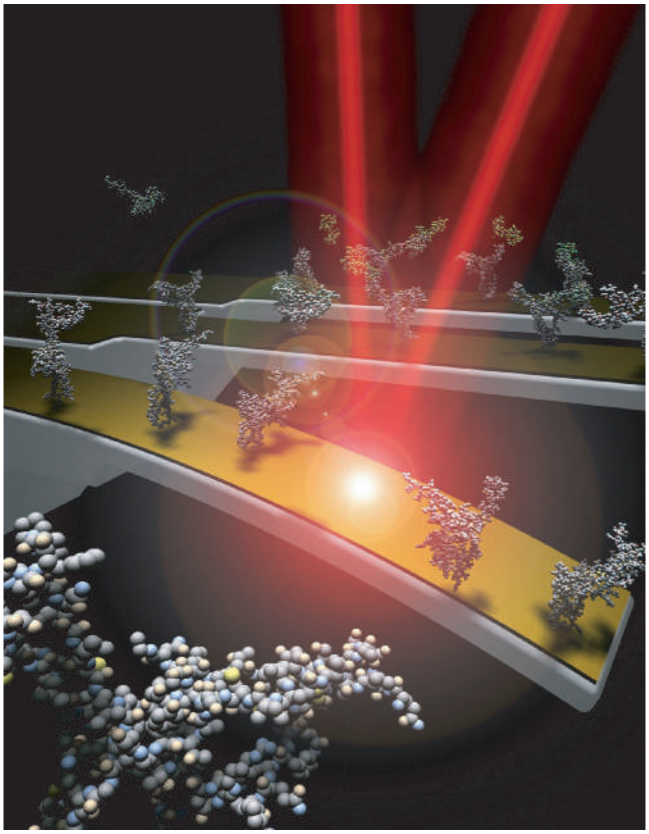
FIGURE 1.18. The demand for chemical and petroleum products led to the construction of sophisticated plants and refineries and the disciplines of chemical and petroleum engineering.



Courtesy of the Berkeley Sensor and Actuator Center, University of California

FIGURE 1.19. This MEMS valve was designed with a solid modeler and was fabricated using semiconductor processing techniques.

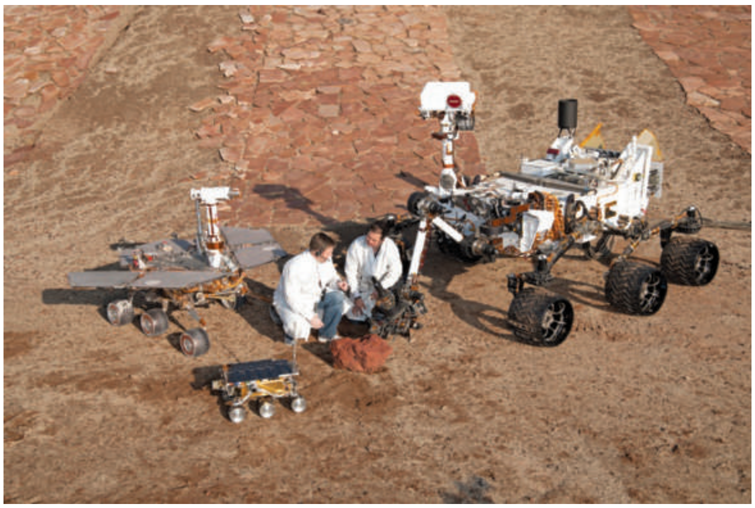


Courtesy of Kenneth Hsu

FIGURE 1.20. This nano-engineered device for sorting molecules does not actually appear as shown, but the use of graphics aids in understanding its operating principles.

With the emergence of a new discipline comes formal intensive training, particularly in the specific discipline, as opposed to subspecialty training within an existing discipline. Most complex engineering projects today require the combined skills of engineers from a variety of disciplines. Engineers from any single discipline cannot accomplish landing an astronaut on the moon or putting a robotic rover, shown in Figure 1.21, on Mars.

FIGURE 1.21. Complex engineering projects, such as interplanetary space missions, require interdisciplinary engineering skills.



Source: NASA/JPL/Caltech

1.03 Engineering Graphics Technology

Mechanical drawing instruments have been a tremendous aid for the creation of engineering graphics. These instruments greatly improve the precision with which graphics can be produced and reproduced, reducing any distortion and making analyses easier and more accurate. The improvement of engineering graphics technology over the years has been a major factor in the improvement of engineering design and communication.

1.03.01 Early Years

Up until the time of the Renaissance, most drawing was done by hand without mechanical devices, because none were available. As a result, many of the drawings that were made to depict some sort of engineering device were distorted. The amount of distortion depended on the skill of the person making the drawing. **Two-dimensional (2-D) drawings** were common because they were easy to make. Attempts at drawing objects showing depth had mixed results. Leonardo da Vinci was one of few people who was good at it, but he was also a skillful artist. In general, though, handmade drawings were good for conveying ideas and some rough sizing. They were poor when precision was necessary, mostly because it was not possible to determine exact sizes from them. In fact, the inch and foot as units of measurement in Europe were not standardized until the twelfth century, and the meter was not defined until the eighteenth century. As a result, when different craftsmen built the same item, the sizes of the parts would be slightly different. Those differences made part interchangeability, and thus mass production, extremely difficult.

1.03.02 Instrument Drawing

Early instruments used to make drawings included straightedges with graduated scales, compasses and dividers, and protractors. They were generally custom-made items for the convenience of those who could afford them. Mechanical instruments for drawing did not become widely available until the industrial revolution, when, for a reasonable cost, machines could produce accurate instruments for both drawing and measuring. Both standardized units and accurate drawings made it possible for different fabricators to make the same part. With careful specifications, those parts would be interchangeable between the devices in which they functioned. Now that engineering drawing made it possible to fabricate the same part at different manufacturers, engineering drawing became a valuable means of communication.

From the industrial revolution to the late twentieth century, drawing instruments slowly improved in quality and became less expensive. Drawing instrument technology reached its most effective and highest level of use during the 1970s. Some companies and individuals today still retain, and even prefer, to use mechanical instruments for making engineering drawings. Classic drawing instruments, some of which are shown in Figure 1.22, are available from architecture, art, and engineering supply shops; these instruments include the following:

- Drafting board—a large, flat table with straight, square edges for alignment of drawing instruments
- Drawing vellum—a tough, dimensionally stable, and age-resistant paper on which drawings are made when placed on the drafting board
- T square—an instrument used to make horizontal and vertical lines by using the edges of the drafting board for reference
- Triangle—an instrument used to make lines at common angles
- Protractor—an instrument used to measure angles or make lines at arbitrary angles
- Scale—an instrument used to measure linear distances

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FIGURE 1.22. Tools for instrument drawing (left) and a drafting machine (right).

- Drafting machine—a special machine used to hold scales at arbitrary angles while the scales are allowed to translate across the drawing, thus replacing many of the previously listed instruments
- Compass—an instrument used to make circles and arcs
- French curve—an instrument used to make curves
- Template—an instrument used to make common shapes

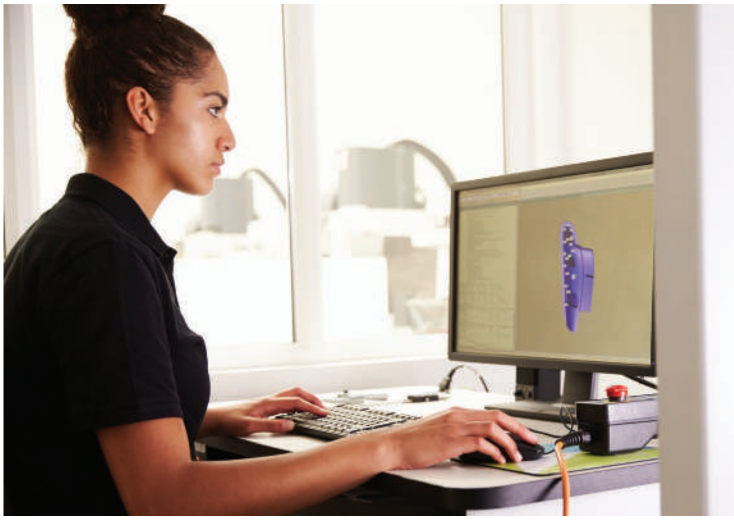
Using pencil or ink, engineers use instruments to draw directly on the desired sized vellum sheet. Large drawings are reproduced on special copy machines. Up until the 1980s, engineering students often were burdened with having to learn how to use the drawing instruments.

1.03.03 The Computer Revolution

During the 1970s, many large companies, particularly those in the automotive and aerospace industries, recognized the advantages of computer-based drawing and graphics: ease of storage and transmission of data, precise drawing data, and ease of data manipulation when drawings needed to be changed. Several large companies began developing computer-aided drawing (**CAD**) tools for their own use. Mainframe computers were just reaching the point where their cost, computation power, and storage capability would support computer-based drawing. The CAD systems consisted of computer terminals connected to a mainframe computer. However, the conversion to computer-based drawing was slow. Mainframe computers were expensive, the user had to have some computer skills, computer hardware and software were not very reliable, and special input and output devices were necessary. Thus, the average engineer or drafter still had a difficult time making the transition from mechanical tools to computer-based tools.

In the late 1970s and early 1980s, several companies specializing in CAD developed freestanding computer-drawing stations based on small independent computers called workstations. Those companies marketed the computer hardware and software as a complete, ready-to-operate unit known as a turnkey system. The workstation approach to CAD made the software more affordable for smaller companies. Also, CAD software became more sophisticated and easier to use. It began to grow in popularity. As personal computers (PCs) began to proliferate in the 1980s, CAD software made specifically to run on PCs became popular.

FIGURE 1.23. Computer graphics stations have replaced mechanical drawing instruments in most applications. A CAD drawing can be created by itself or extracted from a solid model.



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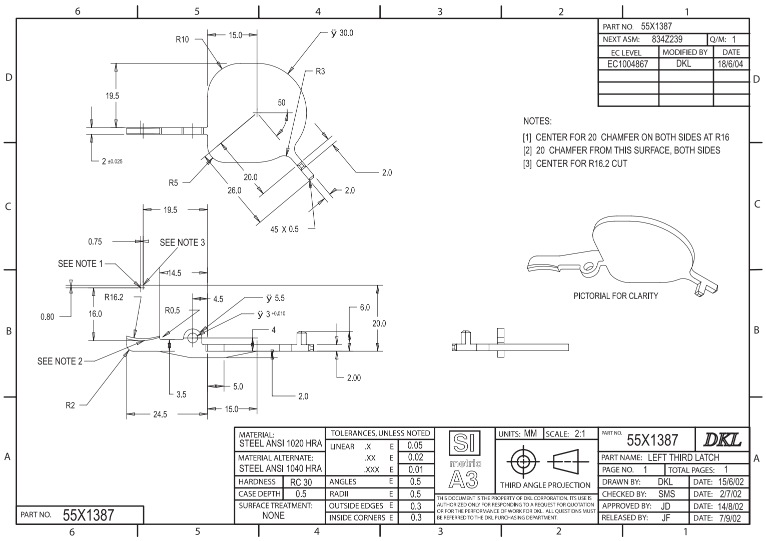

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One company that became a leader in this application was Autodesk, with its AutoCAD software. Companies that formerly supplied mainframe computer-based or turnkey CAD systems either quickly adapted their products for PC use or went out of business. As PCs became more powerful, cheaper, easier to use, and more prolific, CAD software did the same. Drafting boards were quickly replaced by PCs. An example of a PC-based CAD system is shown in Figure 1.23.

1.03.04 Graphics as a Design Tool

Computer-based **three-dimensional (3-D) modeling** as an engineering design tool began in the 1980s. CAD was a great convenience, but it produced only drawings. In this sense, CAD was just a very accurate instrument for making drawings. A drawing's representation of an object in three dimensions had to be visualized by the person reading the drawing. It was the same for any fit or function of an assembly—the person reading the drawing had to visualize it. One problem was that not all readers visualized a drawing the same way. Three-dimensional modeling addressed those problems directly. Unlike a 2-D CAD drawing, which was a collection of 2-D objects used to represent specified views of an object, computer-based solid models had 3-D properties.

The field of mechanical engineering quickly adopted 3-D modeling, calling it **solid modeling**, for the design and analysis of mechanical parts and assemblies. Extrusion or revolution of 2-D shapes created simple 3-D geometries. More complex geometries were created by Boolean operations with simple geometries. The computer calculated a 3-D pictorial **image** of the part, which the engineer could see on a computer monitor. The biggest advantage of solid modeling over CAD was that it permitted viewing a 3-D object from different perspectives, greatly easing the **visualization** of a proposed object. Multiple parts could be viewed together as an assembly and examined for proper fitting. With solid modeling, graphics became more of an engineering design tool, rather than merely a drawing tool. An example of a solid **model** for a single part is shown in Figure 1.24. An assembly model is shown in Figure 1.25.

As you may have realized, solid modeling required more computation power and memory to process files than CAD did. That is why solid modeling was originally introduced on computer workstations using UNIX operating systems, which were relatively costly at the time. In the late 1980s, a new software algorithm increased the utility of solid modeling by making it possible to link the sizes and locations of features on an object to variables that could be input and changed easily. The process was known as parametric design. Those products made it easy for an engineer to add, delete, or change the geometry and sizes of features on a part and see the results almost immediately. Dynamic viewing, which enabled the engineer to twist and turn the part image in real time, was also a powerful software feature. A particular facility of that software—the quick and easy extraction of engineering drawings from the 3-D model—made the total software package a valuable drawing tool as well as a modeling tool.

As PCs continued to become more powerful, in the 1990s solid modeling was introduced as a PC software product. The migration of solid modeling from expensive workstations to less expensive PCs made the software popular among small companies and individuals. The later development of new graphical user interfaces, such as the one shown in Figure 1.26, as opposed to the text menus prevalent at the time, made solid modeling easy to use, even for casual users. PC-based solid modeling with graphical user interfaces soon became a standard.

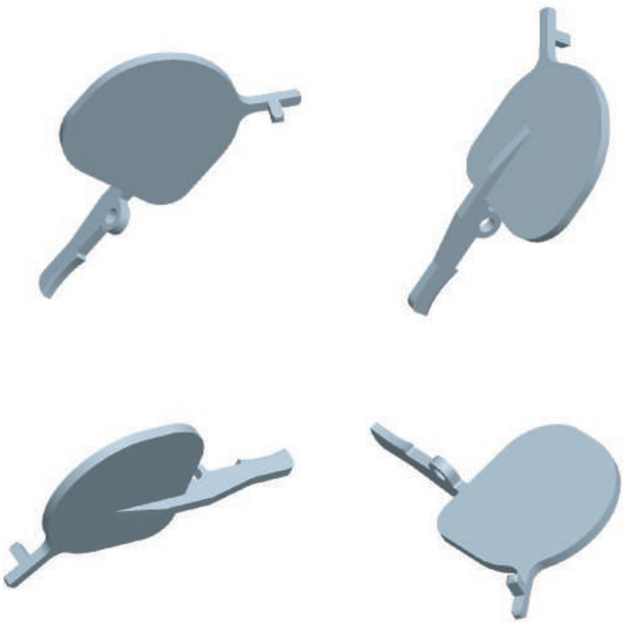

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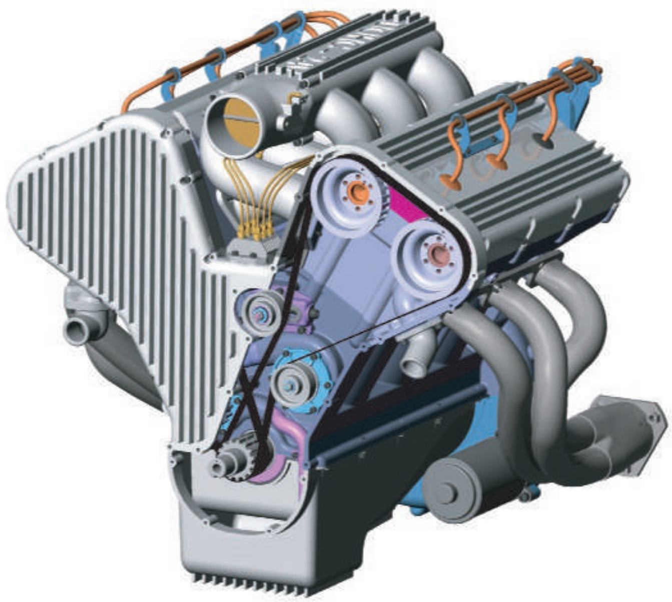
FIGURE 1.24. Solid modeling allows a proposed part to be easily visualized in a variety of orientations.



Courtesy of SolidWorks Corporation

FIGURE 1.25. An assembly model of an Omnica 3.2-liter V-6 engine made from a collection of solid model parts.

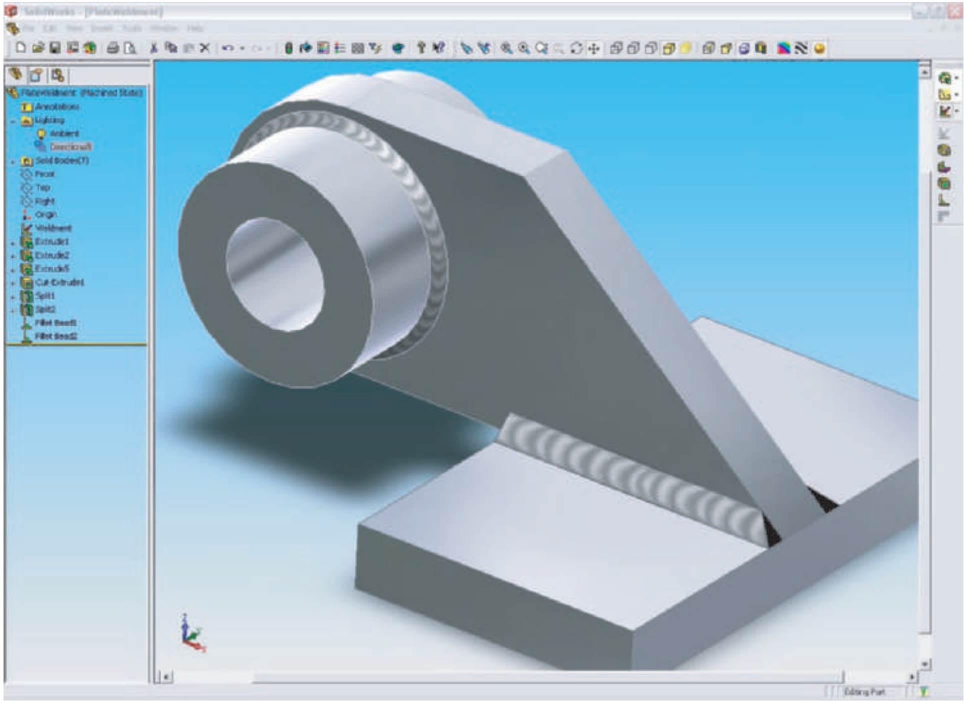


FIGURE 1.26. The graphical user interface of a solid modeling software program.

Courtesy of SolidWorks Corporation

1.03.05 Graphics as an Analysis Tool

Prior to the 1970s, before the days of inexpensive digital computers and handheld calculators, many types of mathematical problems were solved using graphical techniques. Those types of problems included graphical vector analysis, roots and intersections of nonlinear functions, and graphical calculus. Numerical techniques now solve these problems more quickly and easily than graphical techniques, so graphical techniques are not used much anymore. Although solid modeling has decreased the usefulness of descriptive geometry as an analytical tool in many mechanical engineering applications, descriptive geometry still has useful applications in some large-scale civil, architectural, and mining projects. For the most part, drafting boards have been replaced with computers and CAD software, considerably improving accuracy as well as ease of use. However, the classical methods of finding distances, areas, inclines, and intersections used for land characterization and modifications are still used. Many recent large-scale construction and landscaping projects, such as the one shown in Figure 1.27, used classical 2-D graphical analysis and presentation methods.

Using solid modeling, the calculation of important mechanical properties of parts and assemblies can be done easily. The volume that a part or assembly occupies usually can be calculated with a single command after the computer model has been built. Properties of volume, such as mass, center of mass, moments of inertia, products of inertia, and principal axes, can also be calculated. Without a solid modeler, the calculation of these properties would be laborious, especially for complex geometries.

The analysis capability of 3-D modeling also has made it popular for certain types of analyses in civil engineering applications. A two-dimensional topographic map, such as the one shown in Figure 1.28, shows land elevations at development sites for proposed residential areas before and after the addition of roads and building pads. The elevation contours of the land change, because certain locations are excavated while other locations are filled with earth to accommodate the roads and pads.

FIGURE 1.27. Design of many large structures, such as the Forth Road Bridge, Scotland, shown here, still requires the use of classical two-dimensional drawing and analysis techniques.

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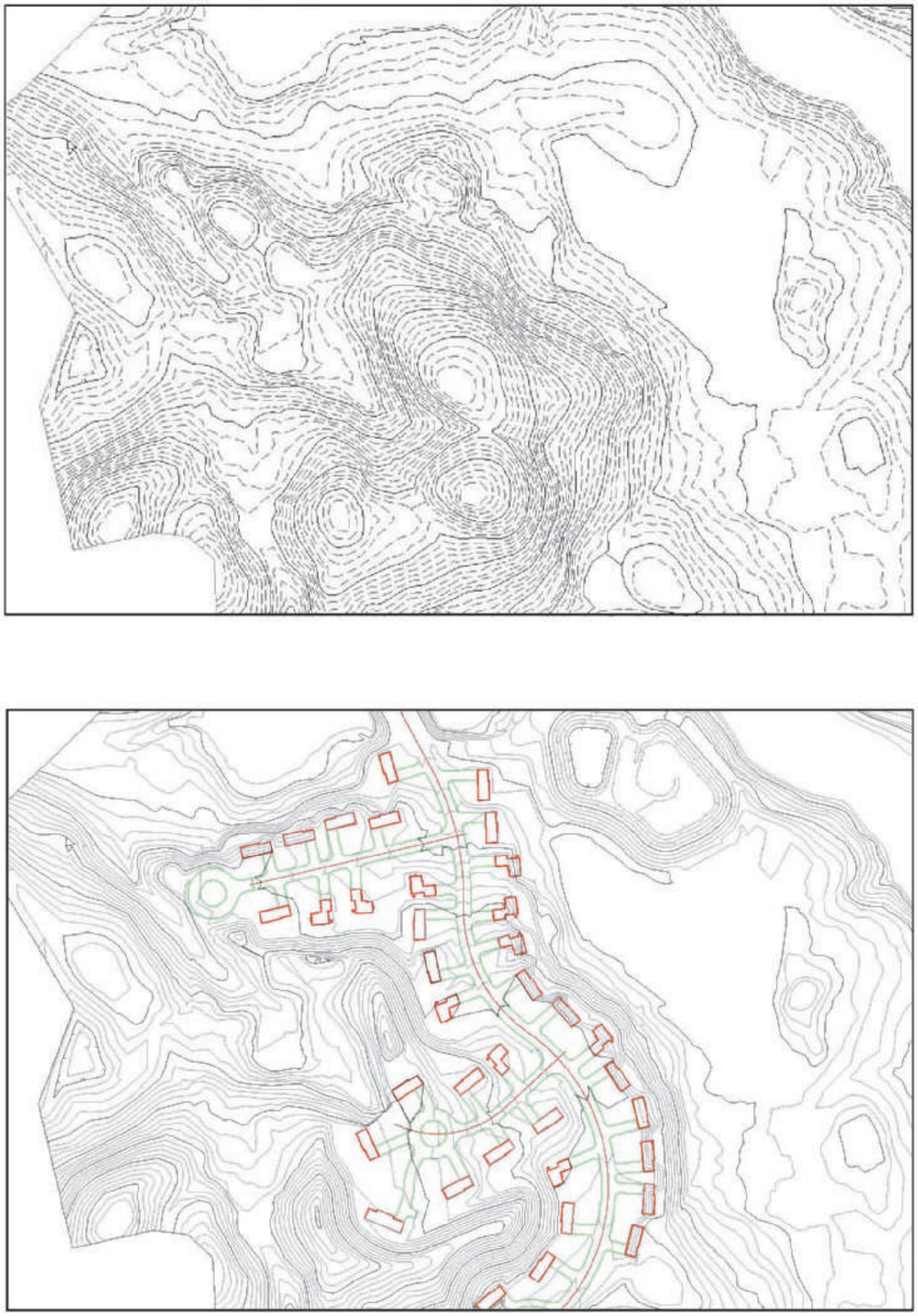
The use of 3-D land models, shown in Figure 1.29, generated from surveying data has made it easier for both engineers and nonengineers to visualize the appearance of a landscape before and after a proposed development. Further, the analytical capability of 3-D modeling in civil engineering applications has made it possible to quickly calculate the volumes of earth that must be removed or added to accommodate the development. It is even possible to match the total addition to the total removal of earth to minimize the volume changed from the site.

FIGURE 1.28. Classical two-dimensional presentation of land-height contours, natural landscape (top), and development for roads and housing (bottom).



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1.03.06 Graphics as a Presentation Tool

An engineer must be able to communicate not only ideas and designs but also precise engineering data. Whether these data are empirical, as collected from experiments, or analytical, as calculated from mathematical models, they must be presented so other people can understand them quickly and easily. Traditional methods of data presentation are in the form of charts and graphs. Charts include familiar items such as pie charts and bar charts commonly used for presenting data to the general public. Graphs, which are usually more technical, show data trends when the relationship

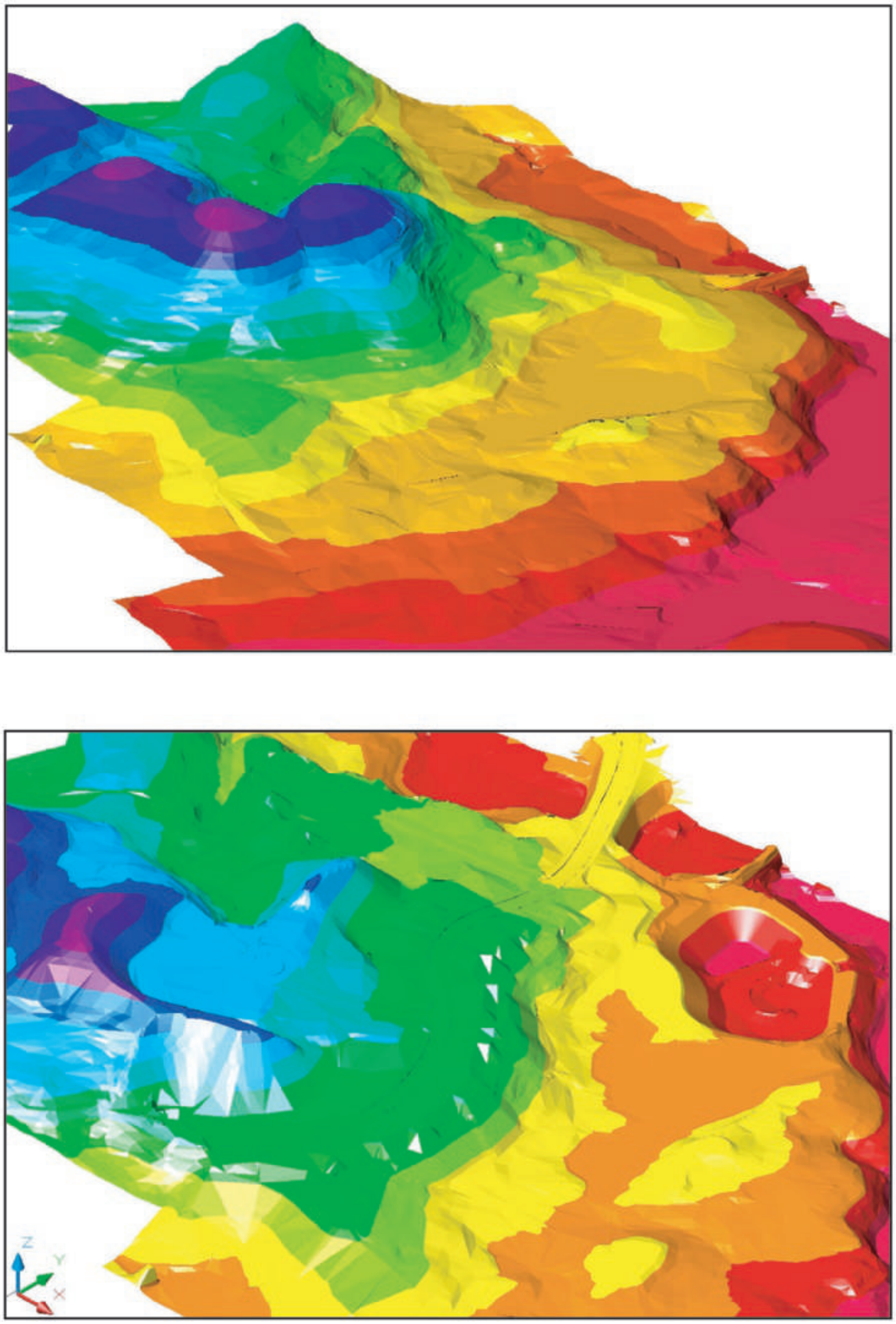

FIGURE 1.29. Three-dimensional images of the contours shown in the previous figure, original land (top) and modifications to accommodate roads and buildings (bottom).

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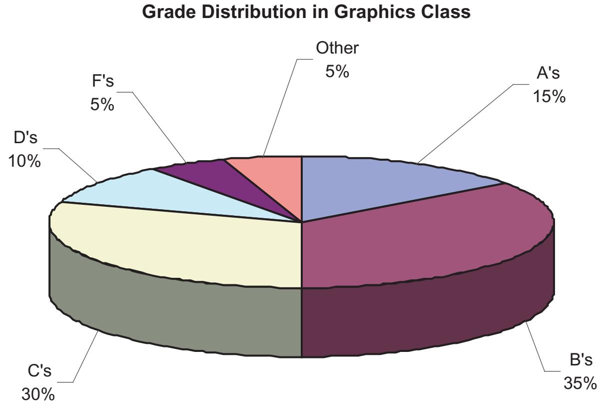
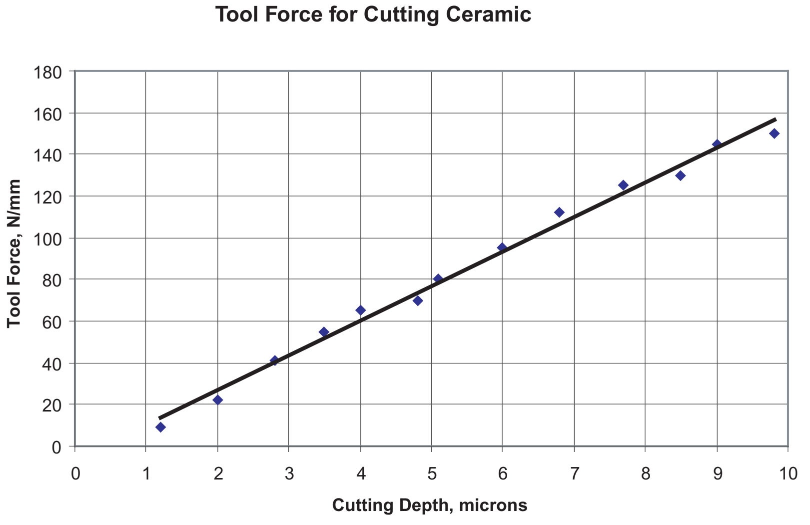
between two or more variables is plotted on orthogonal axes. Examples of these types of data presentation are shown in Figure 1.30.

Three-dimensional modeling software is also used to build geometric models that can be exported for finite element analysis (FEA). FEA is a numerical analysis method used to calculate results such as stress distribution, temperature distribution, or deformation in a part. Although FEA usually is not considered a formal part of engineering graphics, one of the most efficient and effective methods of presenting FEA results is to show the predicted contours of variables such as stress, deflection,

FIGURE 1.30. Data presentation and analysis is a vital part of engineering.





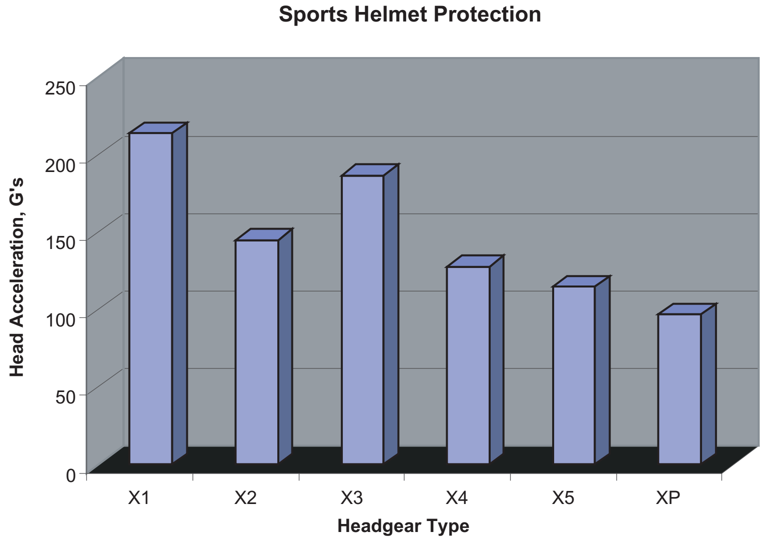


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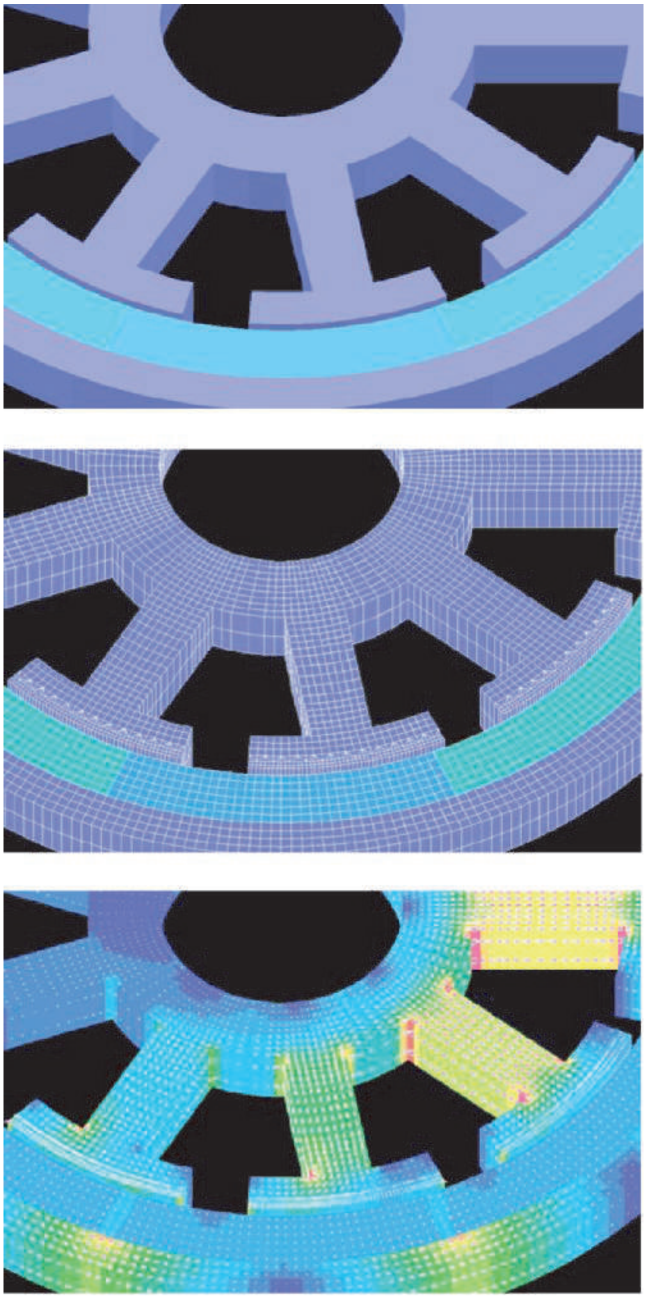
FIGURE 1.31. A three-dimensional model (top) of an electric motor is used to create a FEA mesh (center) from which magnetic flux densities can be calculated and presented (bottom).



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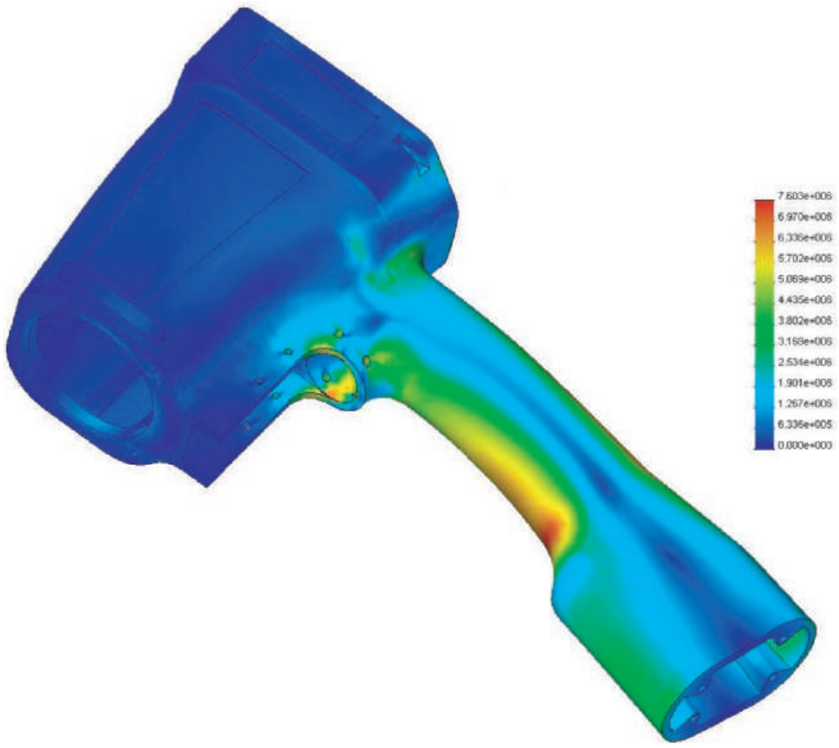
or temperature atop a pictorial of the object. Different colors are used to represent different magnitudes of a variable. For example, Figure 1.31 shows how a solid model of the teeth, steel, and magnets of a small electric motor are created for geometry analysis. The same model is then used to generate a FEA mesh in preparation for an analysis of the magnetic-flux density distribution in the structure. The flux densities are calculated and their contours are plotted directly atop the original solid model image to show the location and magnitude of the flux densities in the motor.

A popular and effective data presentation method is to show the stress distribution in a part by plotting stress contours directly on the part image, as shown in Figure 1.32. In this way, the location and level of the highest stress in the part can be located easily. The same technique can be used for plotting the temperature distribution and magnetic flux densities in a part.

FIGURE 1.32. Graphical representation of stress, such as that produced by forces applied to the part shown, is an important part of presenting the results of a finite element analysis.



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1.04 The Modern Role of Engineering Graphics

Although the role of engineering graphics has evolved over the years, many aspects remain the same. Graphics remains the medium for communicating ideas and technical information. The best way to communicate an idea for a part or device is to show a picture of it. In the past, the pictures were crude handmade drawings, which required time and skill to create. Now pictures are computer-generated images of 3-D models that can be turned and rotated so they are viewable from any direction, providing more accurate depictions. Because the models are easy to create, many variations can be created and viewed in a short time. This advantage makes 3-D modeling useful not only as a means of communication but also as a means of design.

Recording the history of a design also remains an important role of engineering graphics. In the past, recording the history of designs usually meant saving master hard-copy drawings in cabinets in some sort of vault. The smallest change in a design meant changing the master copy and then sending updated copies of the master to whoever needed them. Copies commonly suffered distortion or reduced resolution due to the machines that made the copies. While hard-copy drawings are still necessary, most drawing data are now stored as electronic files. There are enormous advantages in the cataloging, retrieval, and transmission of data stored in this manner. Today, model and drawing data and their updates can be sent across the world in a fraction of a second with no loss in resolution.

Engineering graphics remains an analysis tool, but the type of analysis has changed. Graphical means are no longer used to solve vector algebra, mathematics, or calculus problems. Instead, graphical models are now used to do things like examine the proper fit and function of parts within assemblies. Using 3-D models, engineers can examine parts in their final assembled state for proper motion and location. Engineers can extract the volumetric and inertial properties of the parts and assemblies, to ensure that they fit as specified. Based on externally applied forces, the stresses and deflections in the material also can be examined to ensure that failure of the device does not occur.

Formal engineering drawing remains a part of the overall design process. The traditional role of formal engineering drawings was to ensure that parts would be fabricated to specified sizes, that they would appear as specified, and that various parts would fit together properly. Prior to the 1990s, most engineering graphics classes concentrated on drawing technique and accuracy and on proper use of mechanical drawing instruments. Since engineering drawings can now be created easily and accurately with computers and software tools, the effort required by the formal drawing process is greatly reduced from what it was in the past. Since most computer graphics tools are easy to master, modern graphics classes concentrate mainly on visualization, analysis, function, and **optimization** of designs.

The development of visualization skills is a particular goal of modern engineering graphics courses. Developing visualization skill is necessary for envisioning, specifying, and creating complex designs with functional features in the three spatial dimensions. Traditionally, these skills are developed through hand-eye coordination involving physical parts. Hands-on experience, such as repairing an automobile or a bicycle, constructing models, or playing with building toys, is helpful for developing visualization skills. In an engineering graphics curriculum, these skills can be developed by doing special visualization exercises and by building and working with solid models. Another method of developing visualization skills is to disassemble and reassemble engineered devices in a process known as mechanical dissection. During this process, students examine the operating concepts and their practical implementation, as shown in Figure 1.33.

Sketching also has proven to be a valuable technique for developing visualization skills, as shown in Figure 1.34. Sketches, which can be prepared quickly, provide a simple graphical representation of an idea with a great deal of information on concepts and appearance, without the need for formal drawing tools. For this reason, even though powerful computers and software are available, sketching remains a part of engineering graphics, both as a learning tool and as a practical skill, as you will see throughout this textbook.

FIGURE 1.33. The construction and function of a device can be learned from its disassembly, examination, and reassembly in a process known as mechanical dissection.

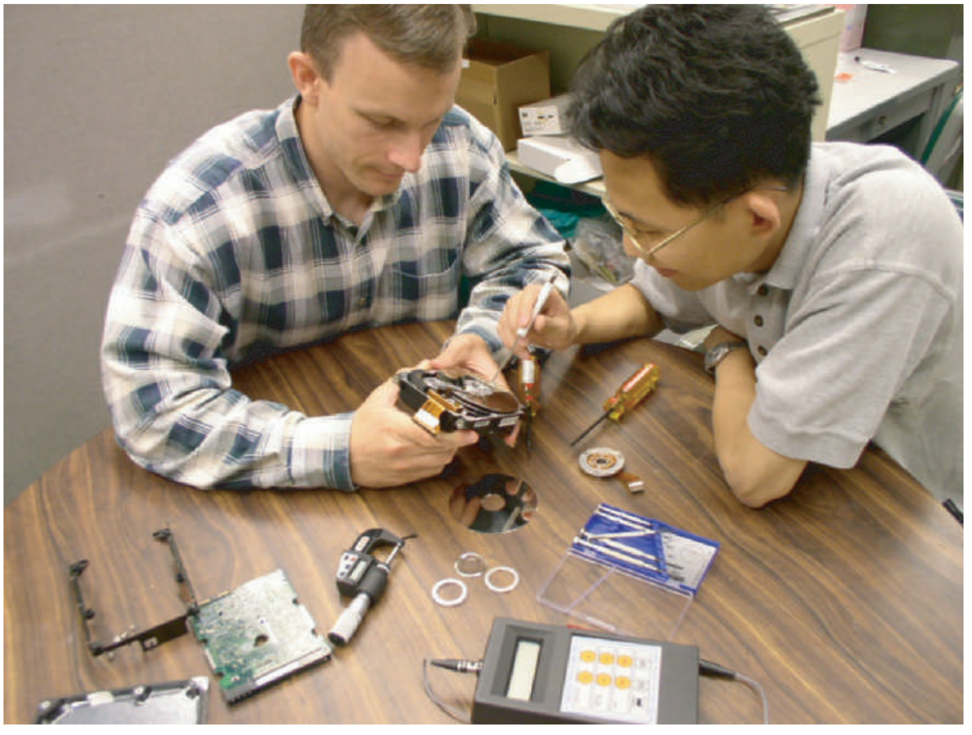

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FIGURE 1.34. Sketching is not only a useful skill but also an excellent exercise for developing spatial reasoning abilities.

1.05 Chapter Summary

The history of graphical communication has shown it to be vital in nearly all aspects of engineering. The development of technology, tools, and techniques used for engineering graphics has advanced, with all of the developments supporting each other. Technological tools have made the tasks associated with classical engineering graphics much easier. The technical sophistication and simple human interface of new tools have enabled engineers to concentrate on learning and developing the techniques offered by the tools, instead of merely operating the tools. Advances in computing, modeling, and display tools have increased the speed and accuracy with which communication, visualization, and analytical problems are performed. More complex designs can be produced more quickly with better functionality and fewer errors than in the past. Engineering drawing has become quicker and simpler; making it possible for engineers to concentrate on what they do best, which is to examine the functionality of a design and to optimize it for its intended environment. Engineers have new responsibilities associated with the new tools, including following protocols for the construction of proper computer models, the electronic transmission of data, and data management.

1.06 GLOSSARY OF KEY TERMS

assembly: A collection of parts that mate together to perform a specified function or functions.

CAD: Computer-aided drawing. The use of computer hardware and software for the purpose of creating, modifying, and storing engineering drawings in an electronic format.

descriptive geometry: A two-dimensional graphical construction technique used for geometric analysis of three-dimensional objects.

design (noun): An original manifestation of a device or method created for performing one or more useful functions.

design (verb): The process of creating a design (noun).

drawing: A collection of images and other detailed graphical specifications intended to represent physical objects or processes for the purpose of accurately re-creating those objects or processes.

engineer (verb): To plan and build a device that does not occur naturally within the environment.

engineer (noun): A person who engages in the art of engineering.

engineering: The profession in which knowledge of mathematical and natural sciences gained by study, experience, and practice is applied with judgment to develop and utilize economically the materials and forces of nature for the benefit of humanity.

fabricate: To make something from existing materials.

image: A collection of printed, displayed, or imagined patterns intended to represent real objects, data, or processes.

instruments: In engineering drawing, mechanical devices used to aid in creating accurate and precise images.

model: A mathematical representation of an object or a device from which information about its function, appearance, or physical properties can be extracted.

optimization: Modification of shapes, sizes, and other variables to achieve the best performance based on predefined criteria.

part: A single object fabricated to perform one or more functions.

project: In engineering, a collection of tasks that must be performed to create, operate, or retire a system or device.

solid modeling: Three-dimensional modeling of parts and assemblies originally developed for mechanical engineering use but presently used in all engineering disciplines.

system: A collection of parts, assemblies, structures, and processes that work together to perform one or more prescribed functions.

three-dimensional (3-D) modeling: Mathematical modeling where the appearance, volumetric, and inertial properties of parts, assemblies, or structures are created with the assistance of computers and display devices.

two-dimensional (2-D) drawing: Mathematical modeling or drawing where the appearance of parts, assemblies, or structures are represented by a collection of two-dimensional geometric shapes.

visualization: The ability to create and manipulate mental images of devices or processes.

1.07 QUESTIONS FOR REVIEW

1. Why are most cave drawings and hieroglyphics not considered to be engineering drawings?
2. In what ways did the design of military fortifications change after the discovery of gunpowder and the invention of the cannon?
3. Why did engineering drawings need to become more precise during the industrial revolution?
4. What were the three traditional roles of engineering graphics?
5. What are some of the new roles of engineering graphics created by computer graphics?
6. What are some of the advantages and disadvantages of using mechanical drawing instruments, as opposed to mathematical tools, for problem solving?
7. What are some of the advantages and disadvantages of using mechanical drawing instruments, as opposed to computational tools, for problem solving?
8. How is solid modeling different from CAD?
9. What is visualization?
10. In what ways can visualization skills be developed?

1.08 PROBLEMS

Graphical communications makes the lives of engineers easier in many ways. The following exercises are intended to give you a feeling of what communication and analysis would be like without the tools and techniques used in engineering graphics. Do not become discouraged if you find these exercises to be difficult or cumbersome or if you find that the results are not accurate, which is the point of these exercises. In the chapters that follow, you will be introduced to methods of addressing the difficulties you encounter here.

1.08 PROBLEMS (CONTINUED)

1a. Do this exercise with one of your classmates. Select one or more of the objects shown in Figure P1.1, but do not show the object(s) to your partner. Using only words, give your partner a complete description of the objects you selected. Then have your partner sketch a picture of the objects based on your verbal descriptions. Reverse roles using different objects. What errors occurred between the objects that were being described and the objects that were envisioned? What can be done to reduce these errors?

1b. Give a third classmate the sketches made in Part A of this exercise. Without revealing what the original objects in the figure look like, give a complete description of the errors in the sketches and have this person make corrections to the sketches. Reverse roles using different objects. How much closer are the sketches to representing the objects shown in the figure? What additional problems occur when a third person is involved?

2. Do this exercise with a group of classmates. Select one or more of the objects shown in Figure P1.2 but do not show the figure(s) to the rest of the group. Make sketches of the object(s) you have selected, give them to the first person in the group, have that person examine them carefully, and then retrieve your sketches. Have that person use the memory of your sketches to make new sketches. Then give the new person's sketches to the second person in the group. Do not show the previous sketches to the new person. Repeat for all of the classmates in the group. When the last person is done, compare the final set of sketches to the objects selected by the first person in the original figure. What errors occurred between the final sketches and the objects that were selected? What happens to the sketches with each revision? What can be done to reduce these errors?

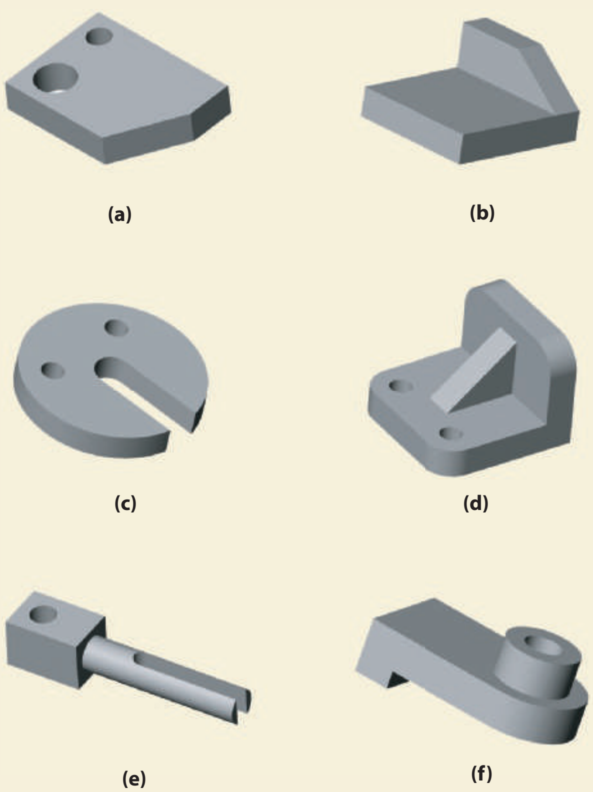


FIGURE P1.1. (a)–(f) Verbally describe these objects to your partner; then have your partner sketch a picture of the object.

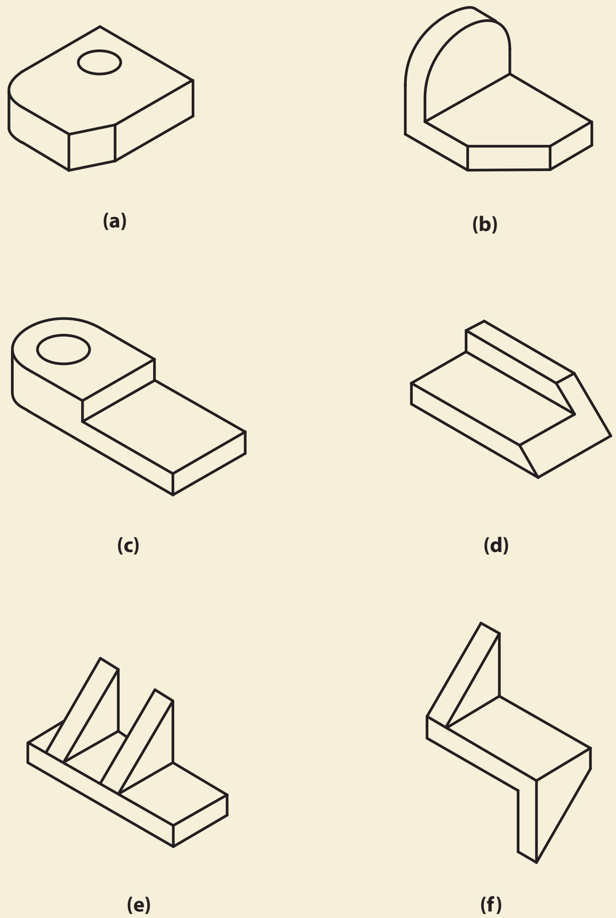
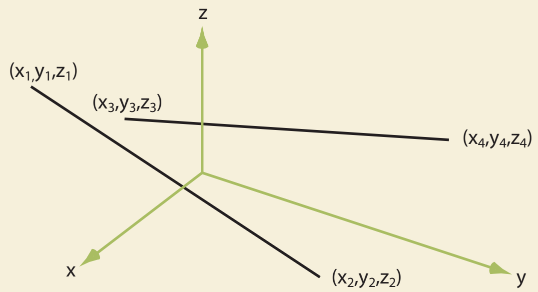


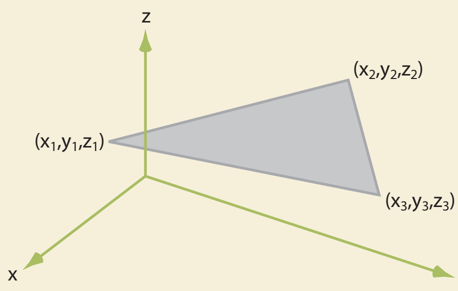
FIGURE P1.2. (a)–(f) Show one or more of these objects to your partner and have your partner sketch the object(s) from memory. Repeat the process with the newly created sketch. Compare the sketch to the original object.

1.08 PROBLEMS (CONTINUED)

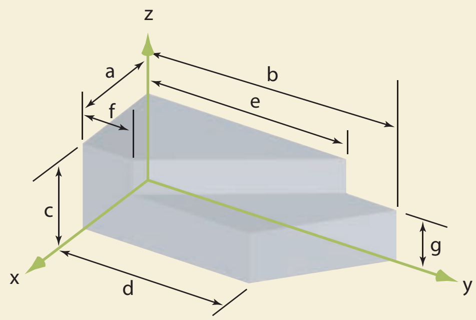
3. For the geometric elements shown in each of the three panels of Figure P1.3, develop formulas for finding the length, angle, area, or volume, whichever is required in each panel, using analytical methods. Generalize the solution in terms of x-, y-, and z-coordinates of the points given. What problems do you envision if the person making the calculations has no access to computers, calculators, or any other computational aids? What happens to the solution formulas as the geometries become more complicated or are rotated and translated in space?



(a) Shortest distance between the lines and equation of the connector



(b) Area and center-of-mass of the plane



(c) Volume and center-of-mass of the solid

FIGURE P1.3. (a)–(c) Find the specified geometric properties of the objects.

CHAPTER 2

SKETCHING

OBJECTIVES

After completing this chapter, you should be able to

- Explain the importance of sketching in the engineering design process
- Make simple sketches of basic shapes such as lines, circles, and ellipses
- Use 3-D coordinate systems, particularly right-handed systems
- Draw simple isometric sketches from coded plans
- Make simple oblique pictorial sketches
- Use advanced sketching skills for complex objects

2.01 INTRODUCTION

Sketching is one of the primary modes of communication in the initial stages of the design process. Sketching also is a means to creative thinking. It has been shown that your mind works more creatively when your hand is sketching as you are engaged in thinking about a problem.

This chapter focuses on one of the fundamental skills required of engineers and technologists—freehand sketching. The importance of sketching in the initial phases of the design process is presented, as are some techniques to help you create sketches that correctly convey your design ideas. The definition of 3-D coordinate systems and the way they are portrayed on a 2-D sheet of paper will be covered, along with the difference between right- and left-handed coordinate systems. The chapter will investigate how to create simple pictorial sketches. Finally, the advanced sketching techniques of shading and cartooning will be presented with a framework for creating sketches of complex objects. You will begin to explore these topics in this chapter and will further refine your sketching abilities as you progress through your graphics course.

2.02 Sketching in the Engineering Design Process

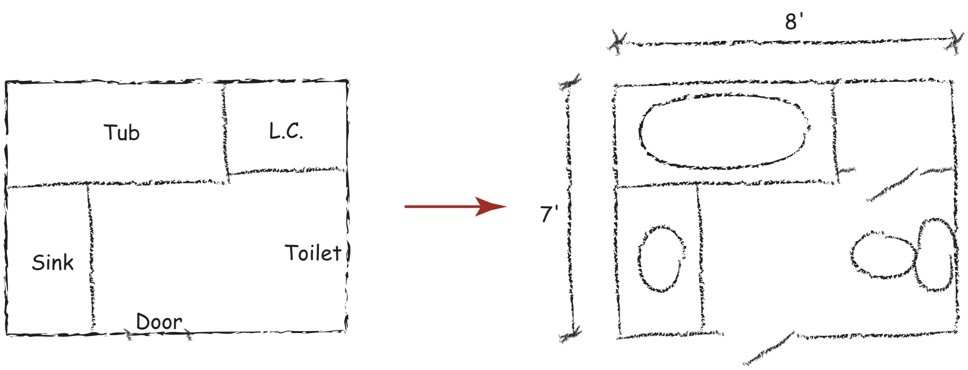
As you may remember from Chapter 1, engineers communicate with one another primarily through graphical means. Those graphical communications take several forms, ranging from precise, complex drawings to simple sketches on the back of an envelope. Most of this text is focused on complex drawings; however, this chapter focuses on simple sketches.

Technically speaking, a sketch is any drawing made without the use of drawing instruments such as triangles and T squares. Some computer graphics packages allow you to create sketches; however, you will probably be more creative (and thus more effective) if you stick to hand sketching, particularly in the initial stages of the design process. In fact, carefully constructed, exact drawings often serve as a hindrance to creativity when they are employed in the initial stages of the design process. Typically, all you need for sketching are a pencil, paper, an eraser, and your imagination.

Your initial sketches may be based on rough ideas. But as you refine your ideas, you will want to refine your sketches, including details that you left out of the originals. For example, suppose you were remodeling the bathroom in your house. Figure 2.01 shows two sketches that define the layout of the bathroom, with details added as ideas evolve. Once you have completed the layout to your satisfaction, you can create an official engineering drawing showing exact dimensions and features that you can give to the contractor who will perform the remodeling work for you.

When engineers sit down to brainstorm solutions to problems, before long, one of them usually takes out a sheet of paper and sketches an idea on it. The others in

FIGURE 2.01. Sketches for a bathroom remodel.



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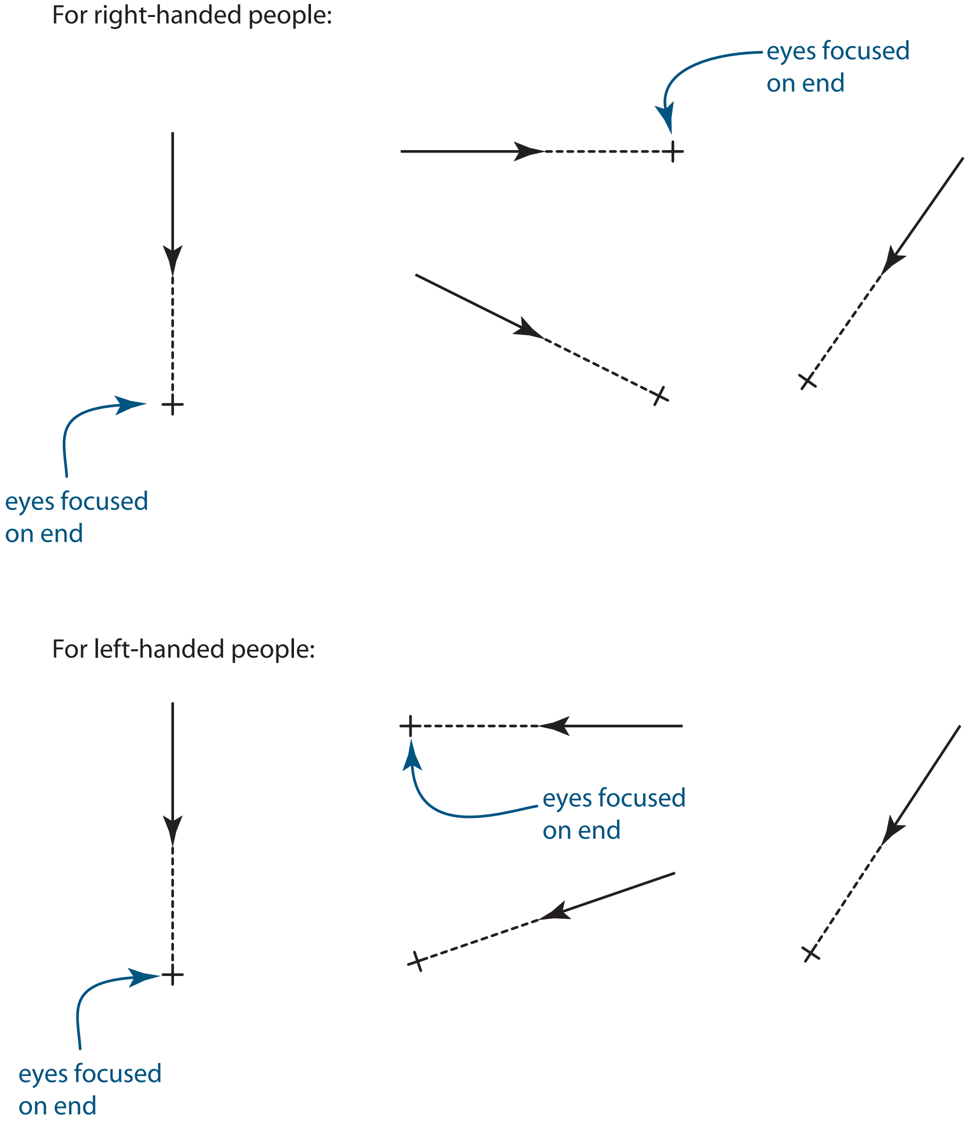
the discussion may add to the original sketch, or they may create sketches of their own. The paper-and-pencil sketches become media for the effective exchange of ideas. Although few "rules" regulate the creation of sketches, you should follow some general guidelines to ensure clarity.

2.03 Sketching Lines

Most of your sketches will involve basic shapes made from lines and circles. Although you are not expected to make perfect sketches, a few simple techniques will enable you to create understandable sketches.

When sketching **lines**, the key is to make them as straight as possible. If you are right-handed, you should sketch your vertical lines from top to bottom and your horizontal lines from left to right. If you are sketching an angled line, choose a direction that matches the general inclination of the line—for angled lines that are mostly vertical, sketch them from top to bottom; for angled lines that are mostly horizontal, sketch them from left to right. If you are left-handed, you should sketch your vertical lines from top to bottom, but your horizontal lines from right to left. For angled lines, left-handed people should sketch from either right to left or top to bottom, again depending on the inclination of the line. To keep your lines straight, focus on the endpoint as you sketch. The best practices for sketching straight lines are illustrated in Figure 2.02.

FIGURE 2.02. Techniques for sketching straight lines.



You also can try rotating the paper on the desk to suit your preferences. For example, if you find that drawing vertical lines is easiest for you and you are confronted with an angled line to sketch, rotate the paper on the desk so you can sketch a "vertical" line. Or you can rotate the paper 90 degrees to sketch a horizontal line. Figure 2.03 illustrates rotation of the paper to create an angled line.

One last point to consider when sketching lines is that you initially may have to create "long" lines as a series of connected segments. Then you can sketch over the segments in a continuous motion to make sure the line appears to be one entity and not several joined end to end. Using segments to define long lines is illustrated in Figure 2.04.

2.04 Sketching Curved Entities

Arcs and **circles** are other types of geometric entities you often will be required to sketch. When sketching arcs and circles, use lightly sketched square **bounding boxes** to define the limits of the curved entities and then construct the curved entities as tangent to the edges of the bounding box. For example, to sketch a circle, you first lightly sketch a square (with straight lines). Note that the length of the sides of the bounding box is equal to the diameter of the circle you are attempting to sketch. At the centers of each edge of the box, you can make a short **tick mark** to establish the point of tangency for the circle, then draw the four arcs that make up the circle. Initially, you may find it easier to sketch one arc at a time to complete the circle; but as you gain experience, you may be able to sketch the entire circle all at once. Figure 2.05 shows the procedure used to sketch a circle by creating a bounding box first.

One problem you may have when using a bounding box to sketch a circle occurs when the radius of the circle is relatively large. In that case, the arcs you create may be too flat or too curved, as shown in Figure 2.06. To avoid this type of error, you might try marking the radius at points halfway between the tick marks included on the bounding box. Using simple geometry, when you draw a line between the center of the circle and the corner of the bounding box, the radius is about two-thirds of the distance

FIGURE 2.03. Rotating the paper to draw an angled line.

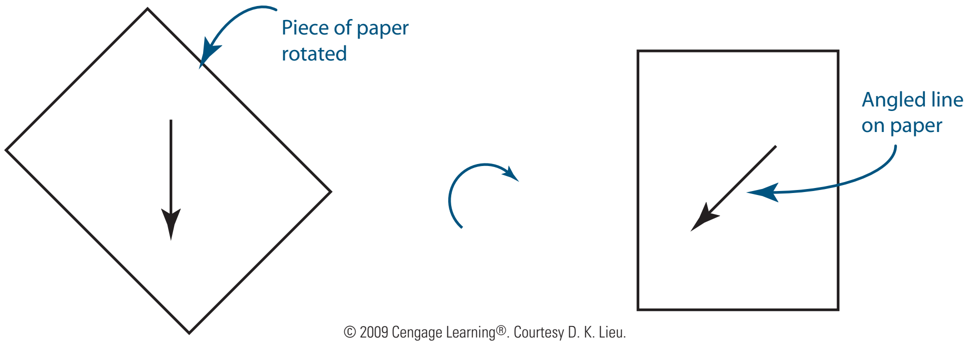


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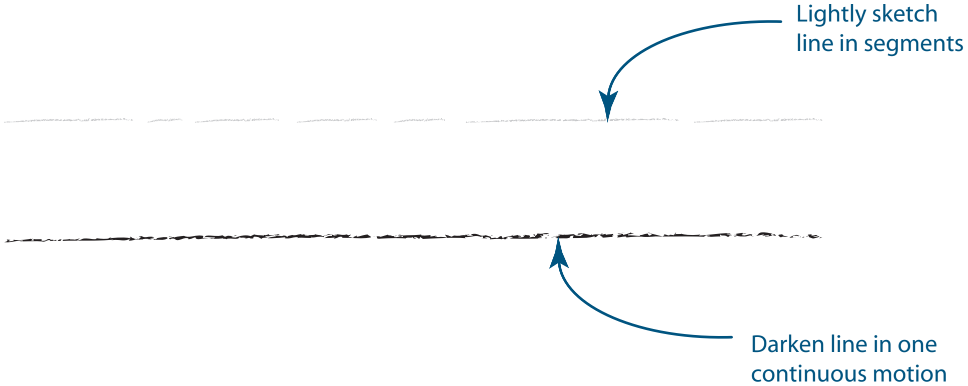
FIGURE 2.04. Sketching long lines in segments.



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(technically, the radius is 0.707, but that number is close enough to two-thirds for your purposes). Then you can include some additional tick marks around the circle to guide your sketching and to improve the appearance of your circles. This technique is illustrated in Figure 2.07.

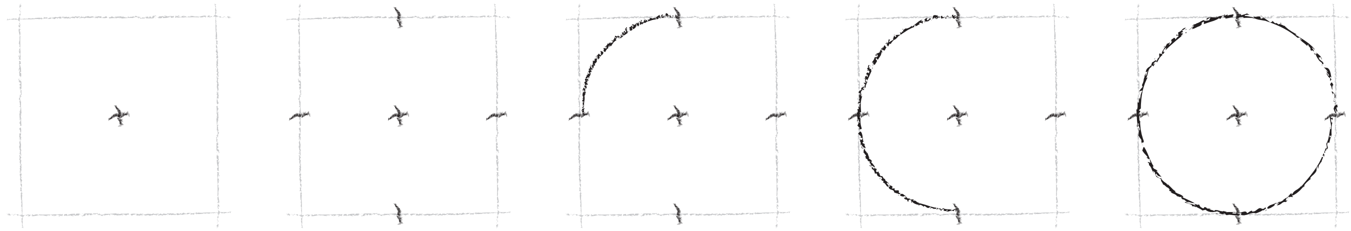


FIGURE 2.05. Sketching a circle using a bounding box.

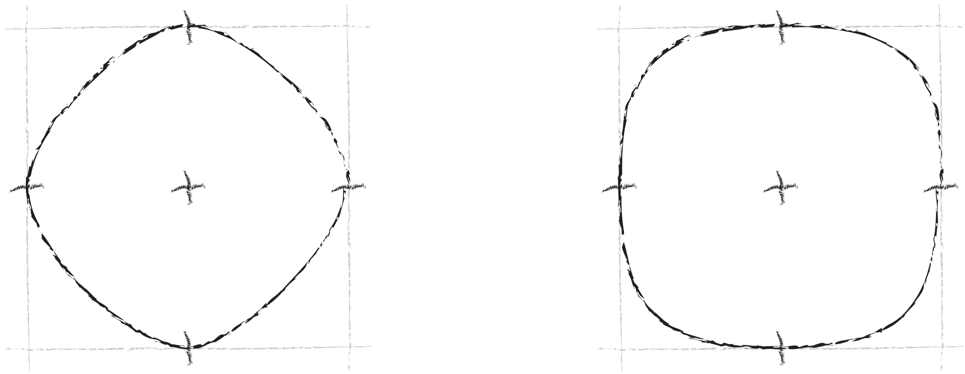


FIGURE 2.06. Circles sketched either too flat or too curved.

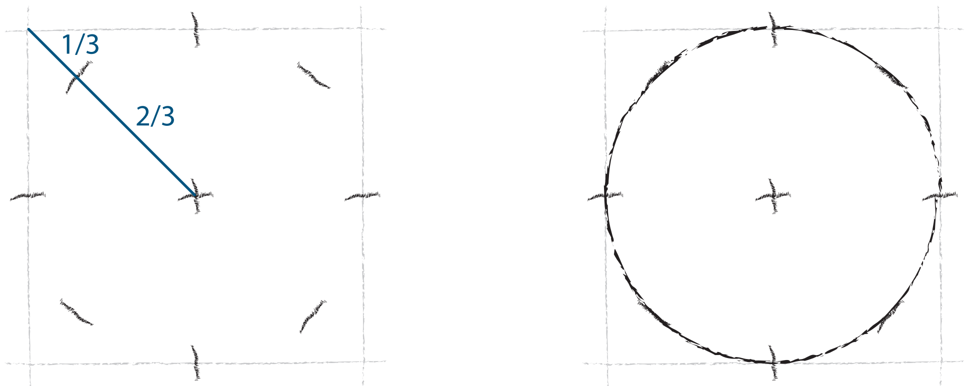
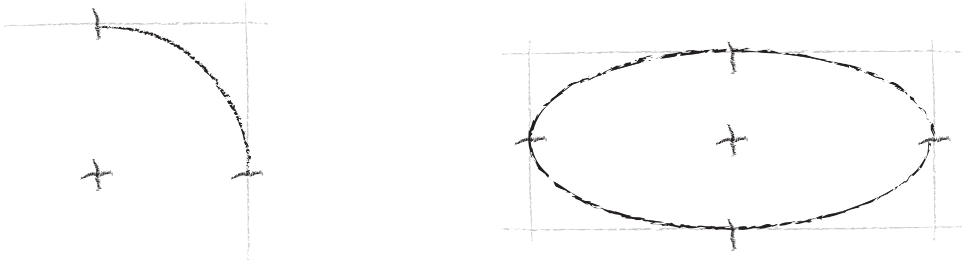


FIGURE 2.07. Using intermediate radial tick marks for large circles.

Sketching an arc follows the same general procedure as sketching a circle, except that your curved entity is only a portion of a circle. Sketching an **ellipse** follows the same general rules as sketching a circle, except that your bounding box is a rectangle and not a square. Sketching arcs and ellipses is illustrated in Figure 2.08.



Sketching an arc

Sketching an ellipse

FIGURE 2.08. Using boundary boxes to sketch arcs and ellipses.

2.05 Construction Lines

Similar to the way you used bounding boxes to create circles and ellipses, other **construction lines** can help with your sketching. Using construction lines, you outline the shape of the object you are trying to sketch. Then you fill in the details of the sketch using the construction lines as a guide. Figure 2.09 shows the front view of an object you need to sketch. To create the sketch, you lightly draw the construction lines that outline the main body of the object and then create the construction lines that define the prominent features of it. One rule of thumb is that construction lines should be drawn so lightly on the page that when it is held at arm's length, the lines are nearly impossible to see. The creation of the relevant construction lines is illustrated in Figure 2.10.

Using construction lines as a guide, you can fill in the details of the front view of the object until it is complete. The final result is shown in Figure 2.11.

Another way you can use construction lines is to locate the center of a square or rectangle. Recall from your geometry class that the diagonals of a box (either a rectangle or a square) intersect at its center. After you create construction lines for the edges of the box, you sketch the two diagonals that intersect at the center. Once you find the center of the box, you can use it to create a new centered box of smaller dimensions—a kind of concentric box. Locating the center of a box and creating construction lines for a newly centered box within the original box are illustrated in Figure 2.12.

Once you have created your centered box within a box, you can sketch a circle using the smaller box as a bounding box, resulting in a circle that is centered within the larger box as shown in Figure 2.13. Or you can use these techniques to create a square with four holes located in the corners of the box as illustrated in Figure 2.14.

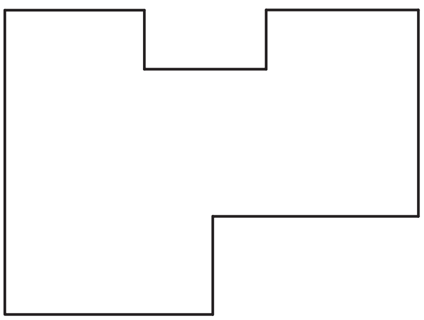


FIGURE 2.09. The front view of an object to sketch.

FIGURE 2.10. Construction lines used to create a sketch.

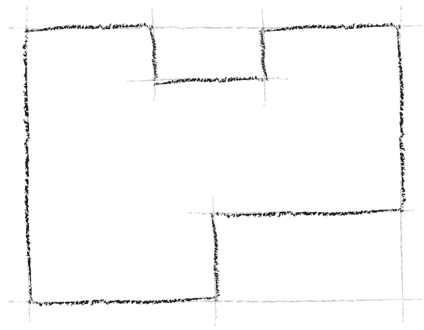


FIGURE 2.11. Completed sketch using construction lines as a guide.

FIGURE 2.12. Creating concentric bounding boxes.

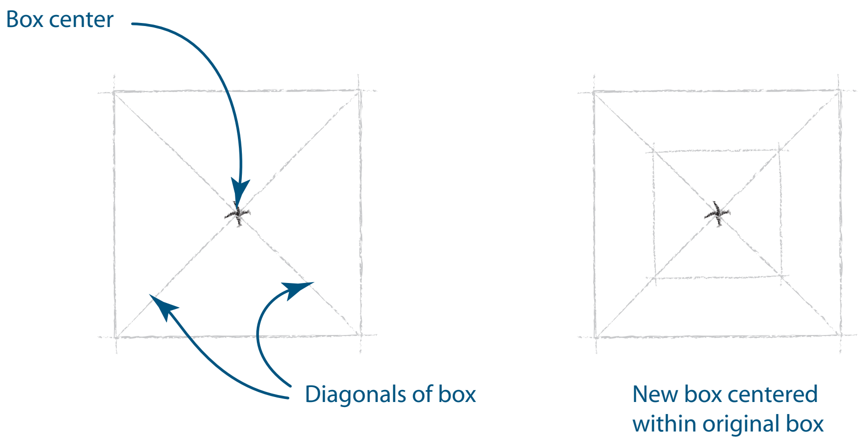


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FIGURE 2.13. Sketching a circle in a box.

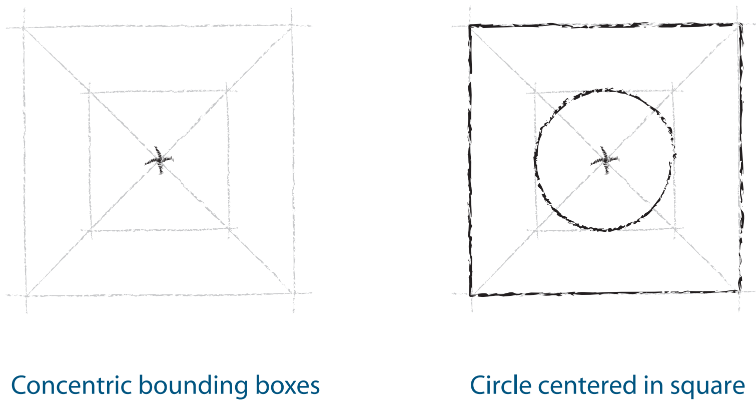

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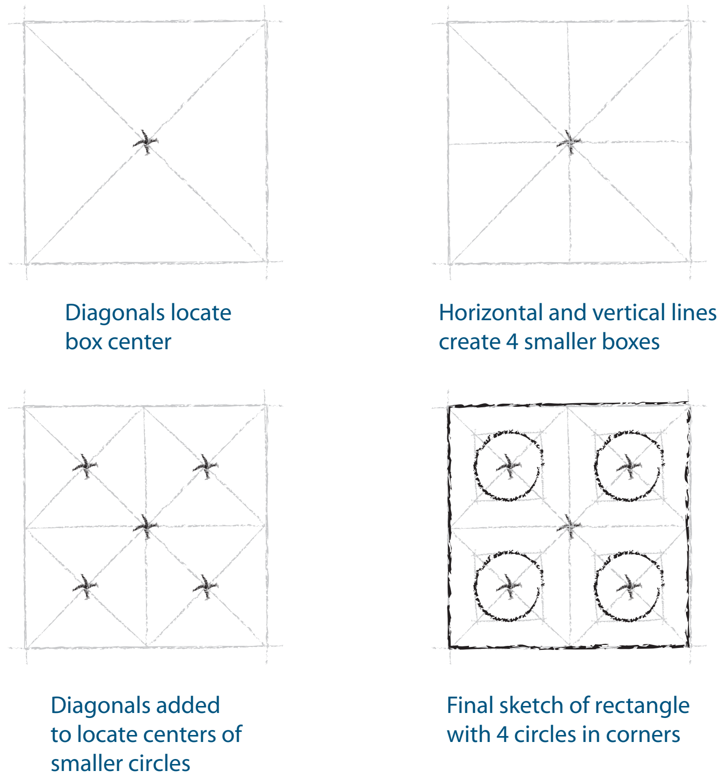
FIGURE 2.14. Using diagonal construction lines to locate centers.



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2.06 Coordinate Systems

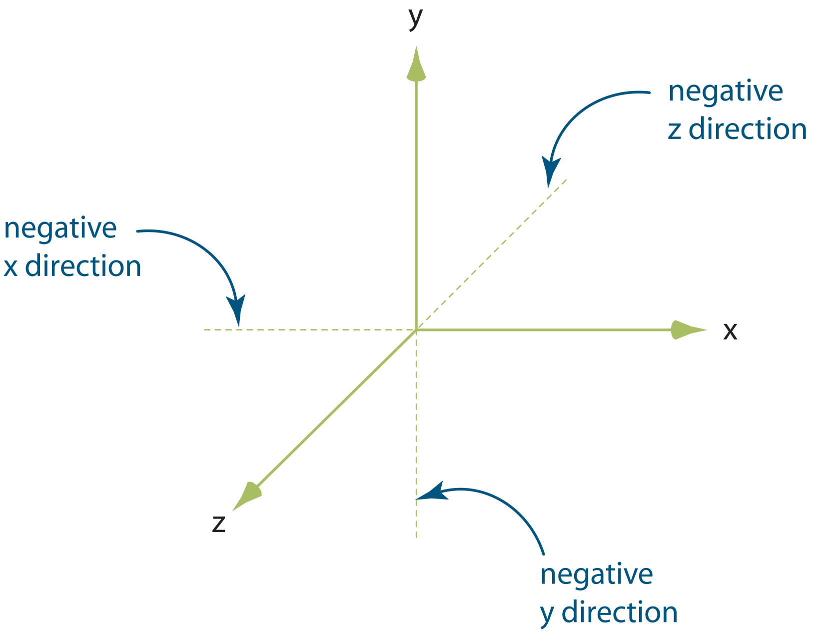
When sketching, you often have to portray 3-D objects on a flat 2-D sheet of paper. As is usually the case with graphical communication, a few conventions have evolved over time for representing 3-D space on a 2-D sheet of paper. One convention, called the **3-D coordinate system**, is that space can be represented by three mutually perpendicular coordinate axes, typically the x-, y-, and z-axes. To visualize those three axes, look at the bottom corner of the room. Notice the lines that are formed by the intersection of each of the two walls with the floor and the line that is formed where the two walls intersect. You can think of these lines of intersection as the x-, y-, and z-coordinate axes. You can define all locations in the room with respect to this corner, just as all points in 3-D space can be defined from an origin where the three axes intersect.

You are probably familiar with the concept of the three coordinate axes from your math classes. In Figure 2.15, a set of coordinate axes, notice the positive and negative directions for each of the axes. Typically, arrows at the ends of the axes denote the positive direction along the axes.

For engineering, the axes usually define a right-handed coordinate system. Since most engineering analysis techniques are defined by a right-handed system, you should learn what this means and how to recognize such a system when you see it. A **right-handed system** means that if you point the fingers of your right hand down the positive x-axis and curl them in the direction of the positive y-axis, your thumb will point in the direction of the positive z-axis, as illustrated in Figure 2.16. This procedure is sometimes referred to as the **right-hand rule**.

Another way to think about the right-hand rule is to point your thumb down the positive x-axis and your index finger down the positive y-axis; your middle finger will then automatically point down the positive z-axis. This technique is illustrated in Figure 2.17. Either method for illustrating the right-hand rule results in the same set of coordinate axes; choose the method that is easiest for you to use.

FIGURE 2.15. The x-, y-, and z-coordinate axes.



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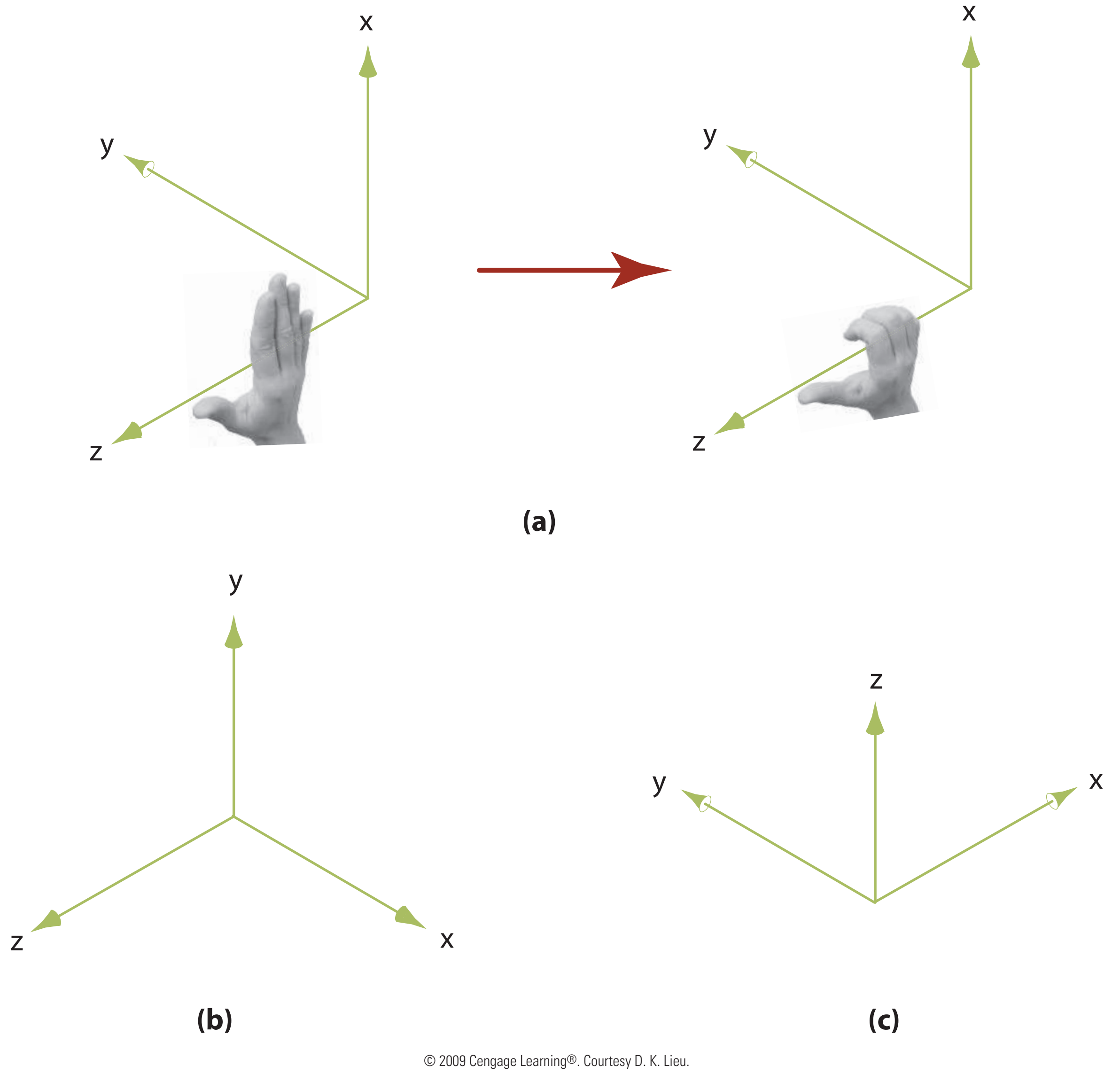

FIGURE 2.16. Curling the fingers to check for a right-handed coordinate system in (a) and alternative presentations of right-handed coordinate systems in (b) and (c).

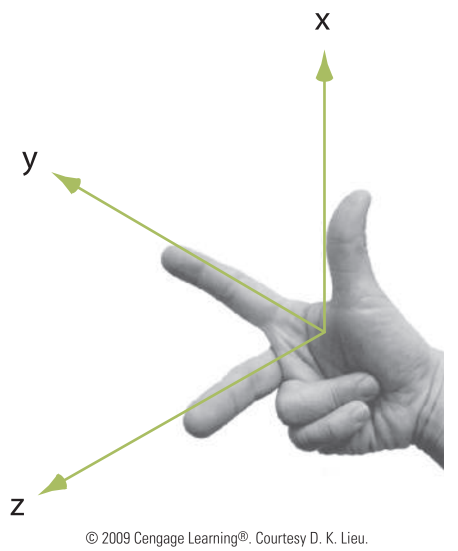

FIGURE 2.17. An alternative method to check for a right-handed coordinate system.

Notice that if you try either technique with your left hand, your thumb (or middle finger) will point down the negative *z*-axis, as illustrated in Figure 2.18.

A **left-handed system** is defined similarly to a right-handed system, except that you use your left hand to show the positive directions of the coordinate axes. Left-handed systems are typically used in engineering applications that are geologically based—positive *z* is defined as going down into the earth. Figure 2.19 illustrates left-handed coordinate systems. (Use the left-hand rule to verify that these are left-handed coordinate systems.)

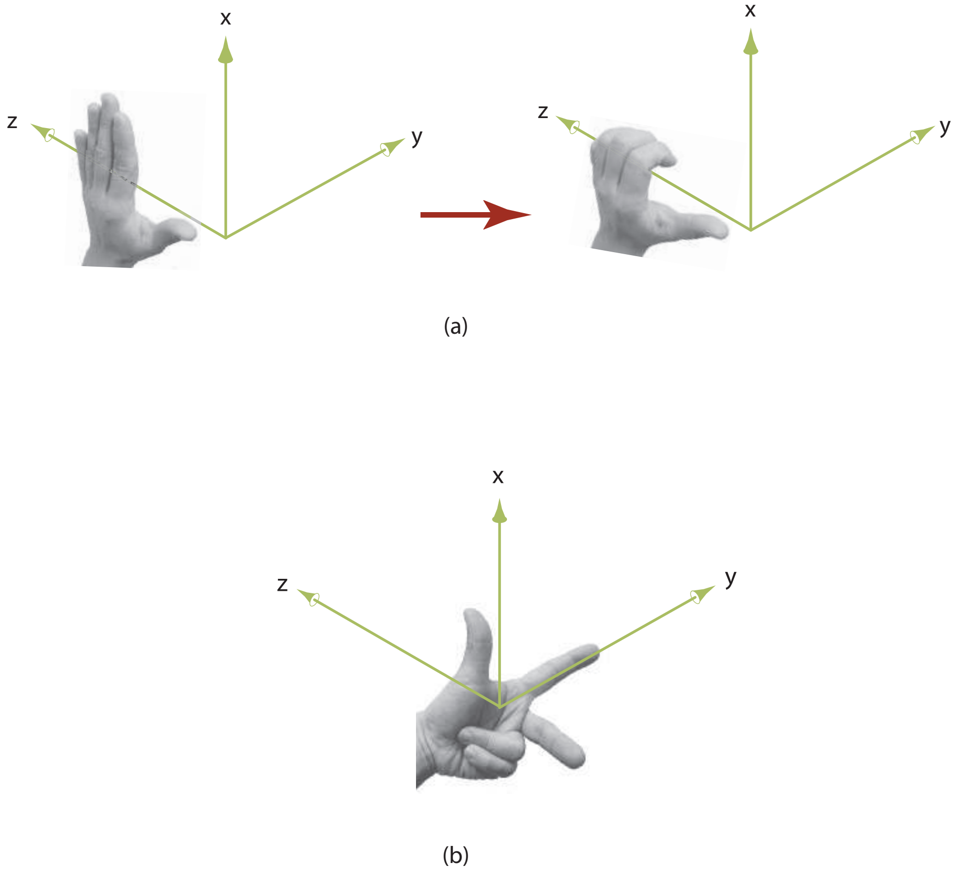

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FIGURE 2.18. The result of using the left hand to test for a right-handed coordinate system.

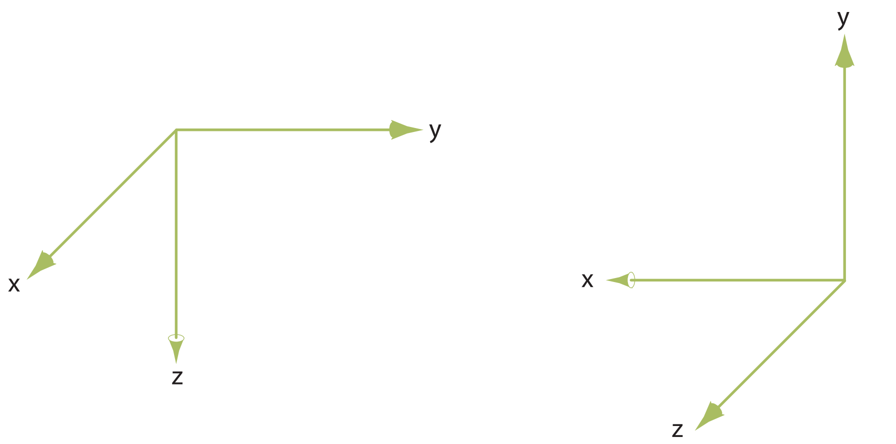

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FIGURE 2.19. Left-handed coordinate systems.

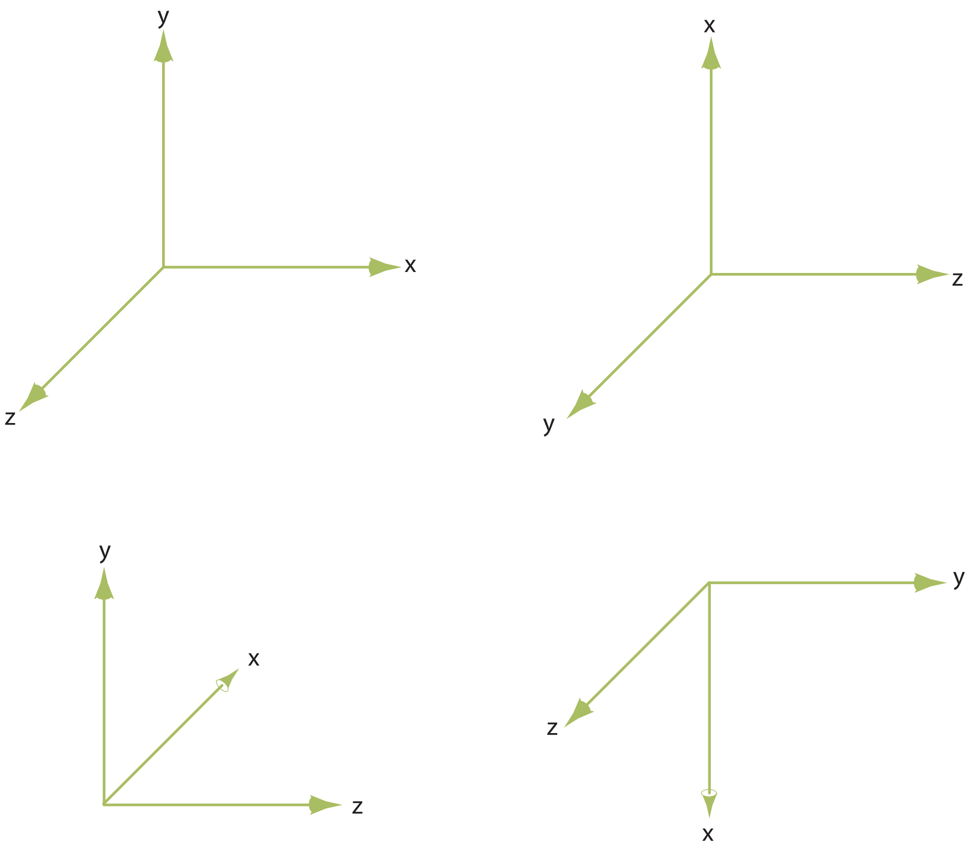
The question remains about how to represent 3-D space on a 2-D sheet of paper when sketching. The answer is that the three coordinate axes are typically represented as oblique or isometric, depending on the preferences of the person making the sketch. You are probably most familiar with oblique representation of the coordinate axes, which seems to be the preferred method of many individuals. With this method, two axes are sketched perpendicular to each other and the third is drawn at an angle, usually 45 degrees to both axes. The angle of the inclined line does not have to be 45 degrees, but it is usually sketched that way. Your math teachers probably sketched the three coordinate axes that way in their classes. Figure 2.20 shows multiple sets of coordinate axes drawn as oblique axes. Notice that all of the coordinate systems are right-handed systems. (Verify this for yourself by using the right-hand rule.)

Another way of portraying the 3-D coordinate axes on a 2-D sheet of paper is through isometric representation. With this method, the axes are projected onto the paper as if you were looking down the diagonal of a cube. When you do this, the axes appear to be 120 degrees apart, as shown in Figure 2.21. In fact, the term *isometric* comes from the Greek *iso* (meaning "the same") and *metric* (meaning "measure"). Notice that for **isometric axes** representations, the right-hand rule still applies.

Isometric axes also can be sketched with one of the axes extending in the "opposite" direction. This results in angles other than 120 degrees, depending on the orientation of the axes with respect to the paper, as shown in Figure 2.22.

Grid or dot paper can help you make isometric sketches. With **isometric dot paper**, the dots are oriented such that when you sketch lines through the dots, you end up with standard 120-degree axes. With grid paper, the lines are already drawn at an angle of 120 degrees with respect to one another. Isometric grid paper and isometric dot paper are illustrated in Figure 2.23.

FIGURE 2.20. An oblique representation of right-handed coordinate systems.



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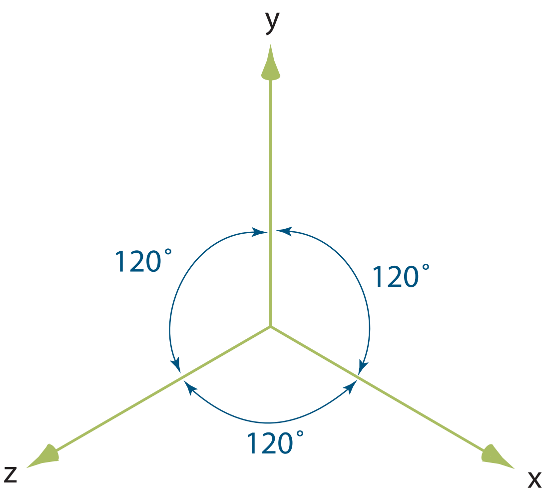

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FIGURE 2.21. An isometric representation of a right-handed coordinate system.

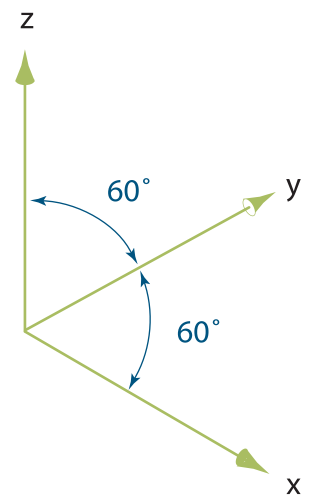

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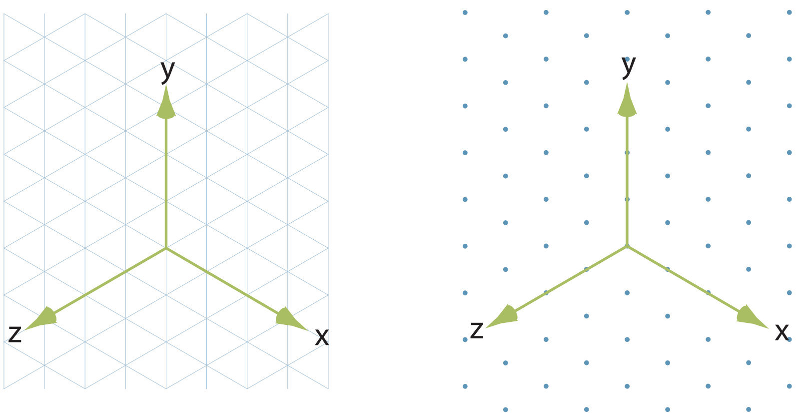
FIGURE 2.22. An isometric representation of axes with angles less than 120 degrees.

FIGURE 2.23. Isometric grid and dot paper.



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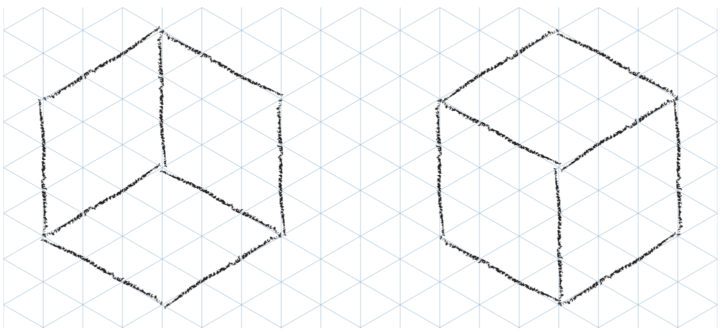
2.07 Isometric Sketches of Simple Objects

Creating isometric drawings and sketches of complex objects will be covered in more detail in a later chapter; however, this section serves as an introduction to the topic for simple objects. Mastering the techniques used to create isometric sketches of simple objects may help as you branch out to tackle increasingly complex objects. Figure 2.24 shows how **isometric grid paper** is used to sketch a $3 \times 3 \times 3$ block. Notice that there

FIGURE 2.24. Using isometric grid paper to sketch a block.



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is more than one orientation from which the block can be sketched on the same sheet of grid paper. Ultimately, the orientation you choose depends on your needs or preferences.

Coded plans can be used to define simple objects that are constructed entirely out of blocks. The numerical values in the coded plan represent the height of the stack of blocks at that location. The object then "grows" up from the plan according to the numbers specified. Figure 2.25 shows a coded plan on isometric grid paper and the object that results from it.

The object shown in Figure 2.25 clearly outlines all of the blocks used to create it. When isometric sketches of an object are made, however, standard practice dictates that lines appear only where two surfaces intersect—lines between blocks on the same surface are not shown. Figure 2.26 shows the object from Figure 2.25 after the unwanted lines have been removed. Notice that the only lines on the sketch are those formed from the intersection of two surfaces. Also notice that object edges hidden from view on the back side are not shown in the sketch. Not showing hidden edges on an **isometric pictorial** also is standard practice in technical sketching.

Sometimes when you are creating an isometric sketch of a simple object, part of one surface is obscured by one of the more prominent features of the object. When creating the sketch, make sure you show only the visible part of the surface in question, as illustrated in Figure 2.27.

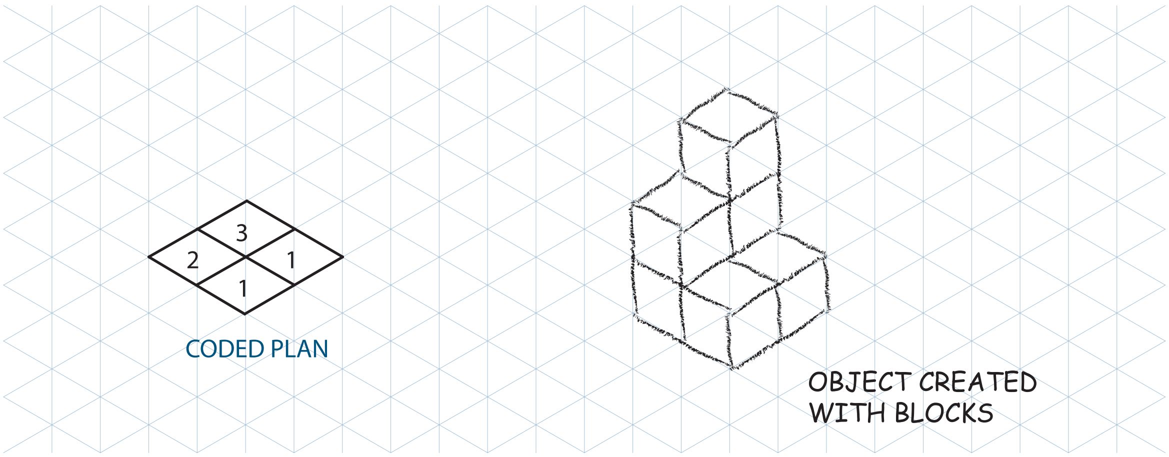

FIGURE 2.25. A coded plan and the resulting object.

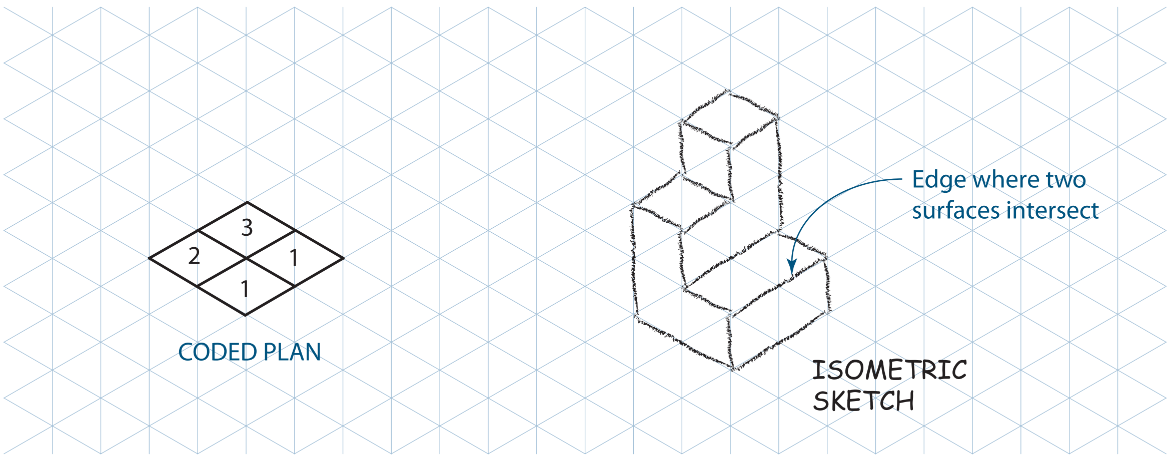

FIGURE 2.26. A properly drawn isometric sketch of the object from the coded plan.

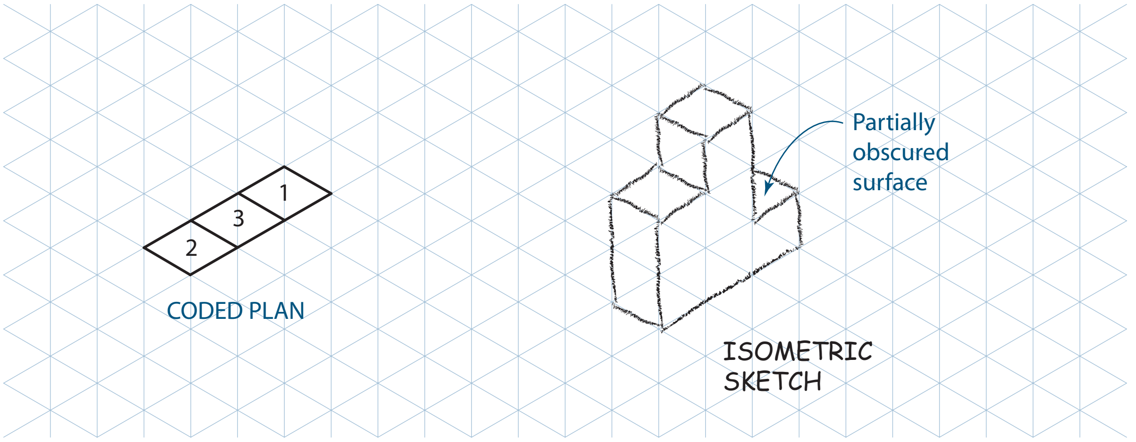

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FIGURE 2.27. The partially obscured surface on an isometric sketch.

Figure 2.28 shows several coded plans and the corresponding isometric sketches. Look at each isometric sketch carefully to verify that it matches the defining coded plan: that lines are shown only at the edges between surfaces (not to define each block), that no hidden edges are shown, and that only the visible portions of partially obscured surfaces are shown.

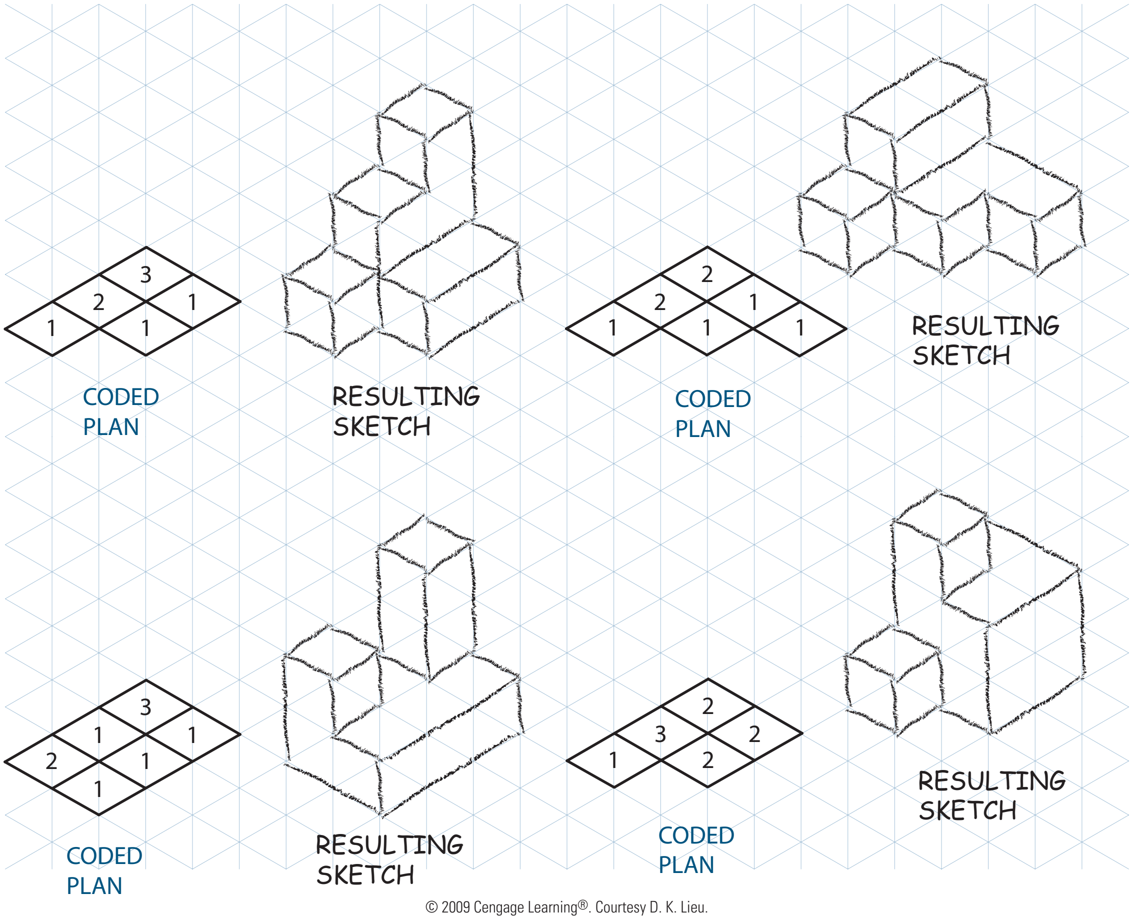

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FIGURE 2.28. Four coded plans and the resulting isometric sketches.

2.07.01 Circles in Isometric Sketches

Look back at the 3 × 3 × 3 block shown in Figure 2.24. In reality, you know that all of the surfaces of the block are 3 × 3 squares; yet in the isometric sketch, each surface is shown as a parallelogram. The distortion of planar surfaces is one disadvantage of creating isometric sketches. The isometric portrayal of circles and arcs is particularly difficult. Circles appear as ellipses in isometric sketches; however, you will not be able to create a rectangular bounding box to sketch the ellipse in isometric as described earlier in this chapter. To create an ellipse that represents a circle in an isometric sketch, you first create a square bounding box as before; however, the bounding box will appear as a parallelogram in the isometric sketch. To create your bounding box, locate the center of the circle first. From the center, locate the four radial points. The direction you move on the grid corresponds to the lines that define the surface. If you are sketching the circle on a rectangular surface, look at the sides of the rectangle as they appear in isometric and move that same direction on the grid. Figure 2.29 shows a 4 × 4 × 4 cube with a circle center and four radial points located on one of the sides.

Once you have located the center of the circle and the four radial points, the next step is to create the bounding box through the radial points. The edges of the bounding box should correspond to the lines that define this particular surface. The edges will be parallel to the edges of the parallelogram that define the surface if that surface is square or rectangular. Figure 2.30 shows the cube with the circle center and the bounding box located on its side.

Four arcs that go through the radial points define the ellipse, just like an ellipse drawn in a regular rectangular bounding box. The difference is that for the isometric ellipse, the arcs are of varying curvatures—two long arcs and two short

FIGURE 2.29. A cube with a circle center and radial points located.

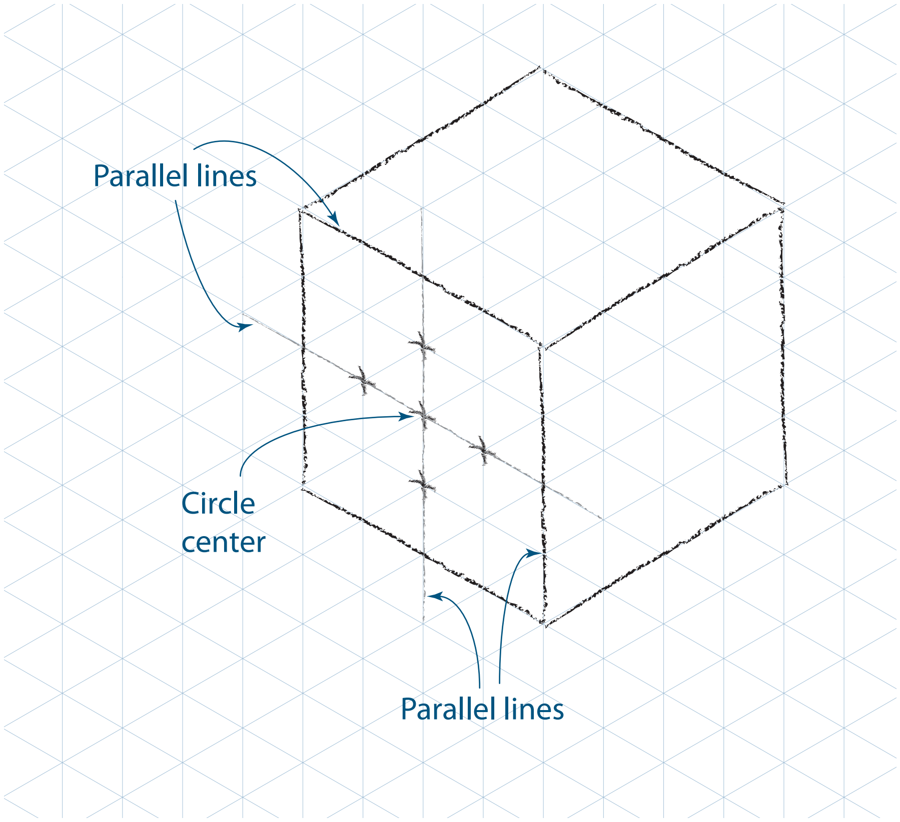

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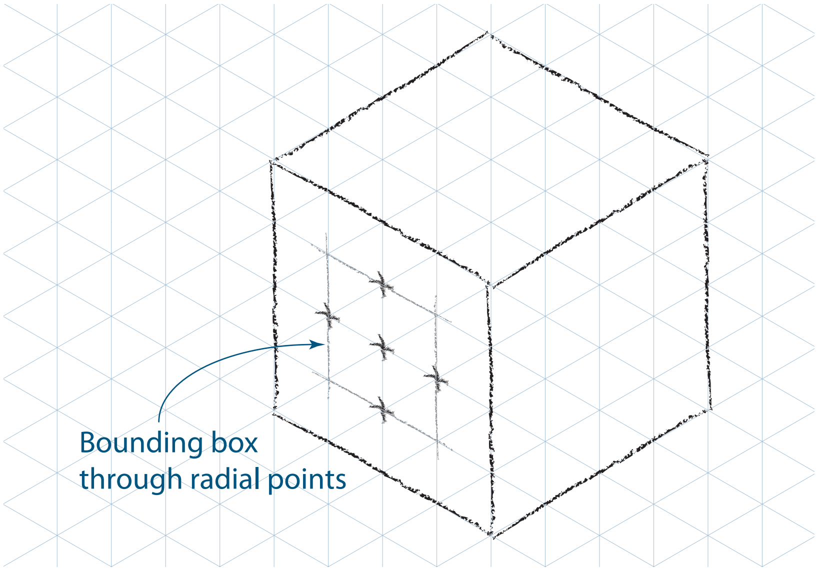
FIGURE 2.30. A cube with the circle center and bounding box on the side.



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arcs in this case. The arcs are tangent to the bounding box at the radial points, as before. It is usually best if you start by sketching the long arcs, and then add the short arcs to complete the ellipse. Sketching the arcs that form the ellipse is illustrated in Figure 2.31.

Creating ellipses that represent circles on the other faces of the cube is accomplished in a similar manner, as illustrated in Figures 2.32 and 2.33.

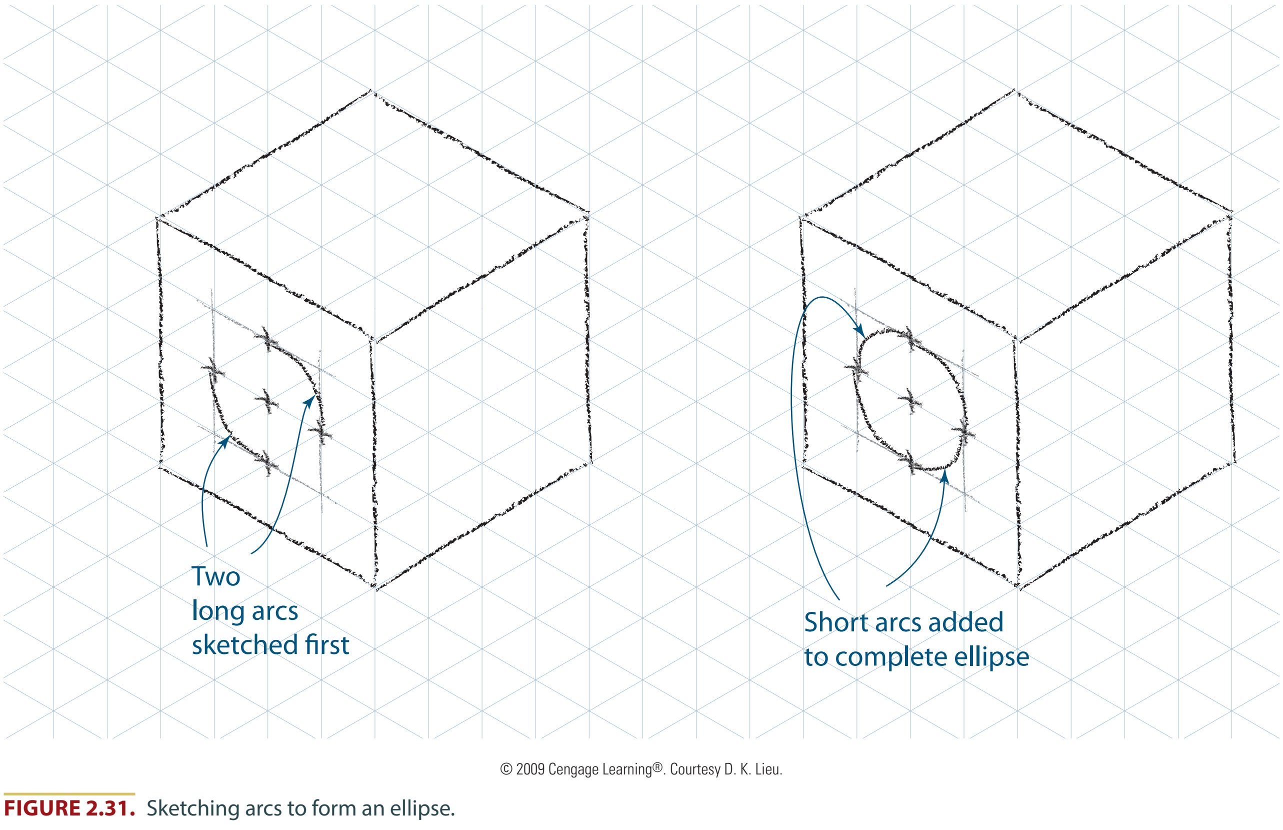

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FIGURE 2.31. Sketching arcs to form an ellipse.

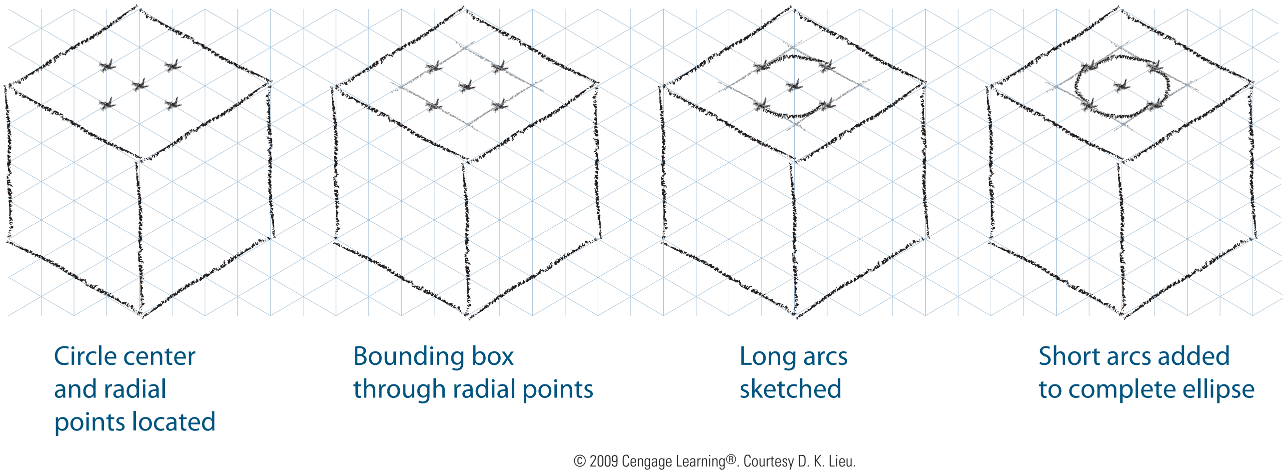

FIGURE 2.32. Sketching an ellipse on the top surface of a cube.

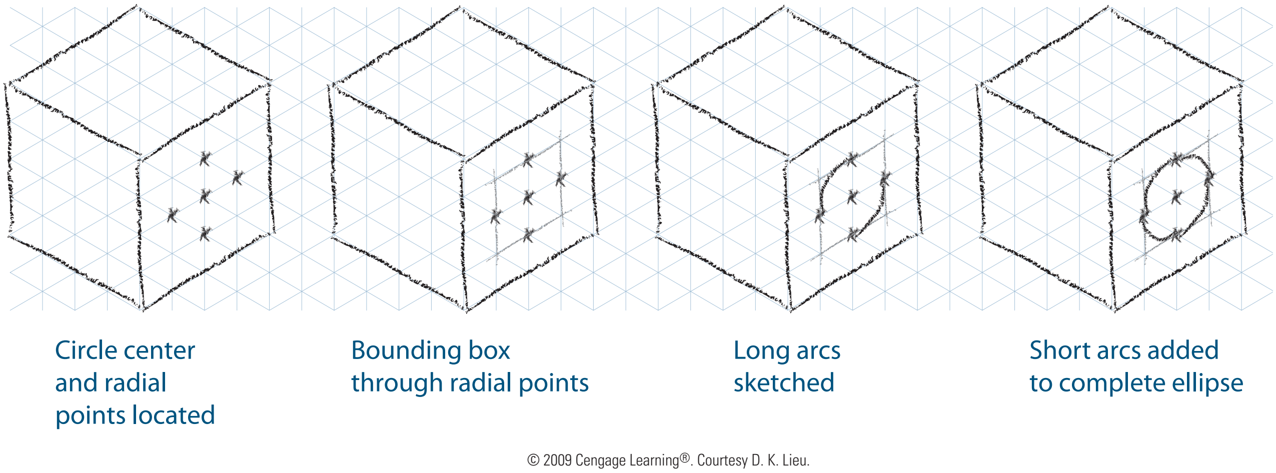

FIGURE 2.33. Sketching an ellipse on the side face of a cube.

2.07.02 Circular Holes in Isometric Sketches

One of the most common occurrences that produces a circular feature in an isometric sketch is a hole in the object. You will learn more about circular holes and object features in a later chapter, but a short introduction follows here. A circular hole usually extends all the way through an object. In an isometric pictorial, a portion of the "back" edge of a circular hole is often visible through the hole and should be included in your sketch. As a rule of thumb, the back edge of a hole is partially visible when the object is relatively thin or the hole is relatively large; when the object is thick or the diameter of the hole is small, the back edge of the hole is not visible. Figure 2.34 shows two blocks with circular holes going through them. Notice in the "thin" block that you can see a portion of the back edge of the hole; in the thicker block, though, the back edge is not visible.

To determine whether a part of the back edge of a hole is visible in an isometric sketch, you first need to locate the center of the back hole. To locate the back center, start from the center of the hole on the front surface and move in a direction perpendicular to the front surface toward the back of the object a distance equal to the object's dimension in that direction. Figure 2.35 shows the location of the center of the two back circles for the objects in Figure 2.34.

FIGURE 2.34. Blocks with circular holes in them.

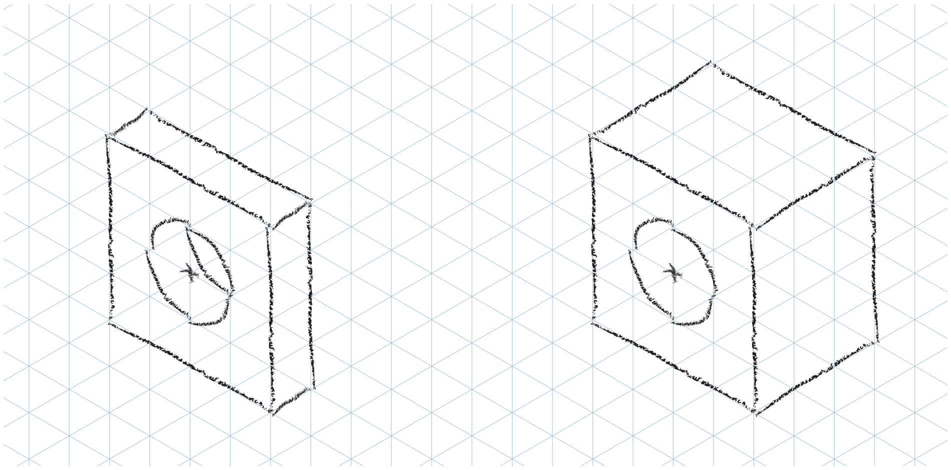

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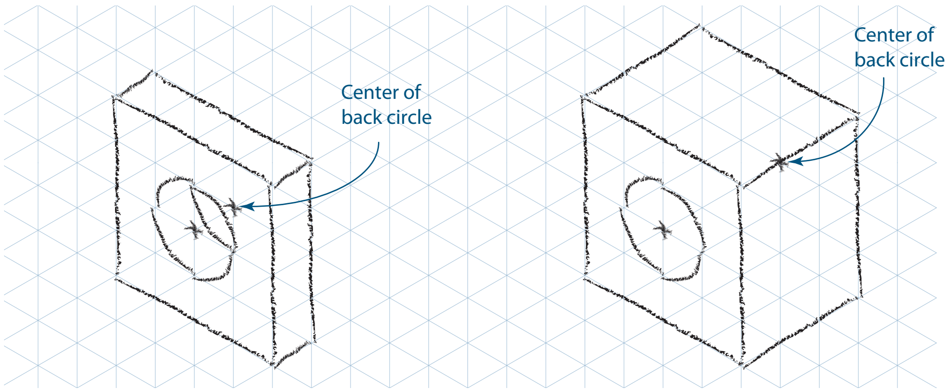
FIGURE 2.35. Centers of back circles located.



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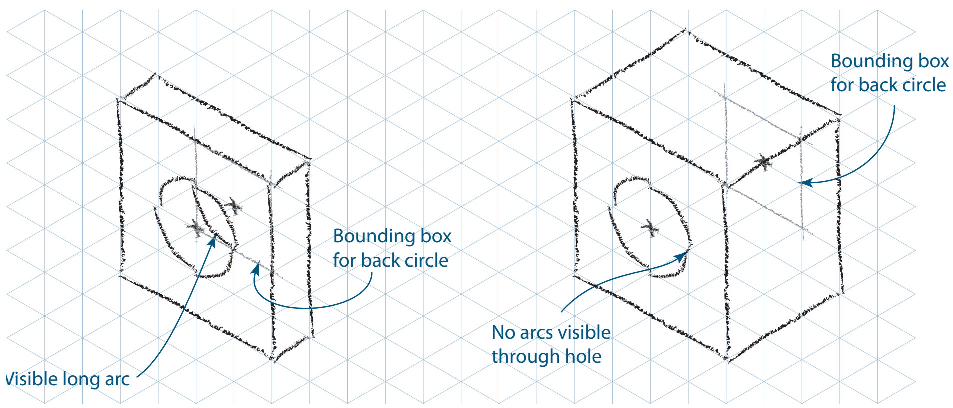
Starting from the back center point, lightly sketch the radial points and the bounding box for the back circle similar to the way you did for the front circle. Then add the long arc that is visible through the hole. (Note that only *one* of the long arcs is typically visible through the hole.) Add segments of the short arcs as needed to complete the visible portion of the back edge of the hole. Conversely, if after you sketch the back bounding box you notice that no portion of the ellipse will be visible on the sketch, do not include any arcs within the hole on the sketch and erase any lines associated with the bounding box. Figure 2.36 illustrates the inclusion and noninclusion of segments of the back edges of holes for the objects in Figures 2.34 and 2.35.

FIGURE 2.36. Determining visibility of back circles.



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2.08 Oblique Pictorials

Oblique pictorials are another type of sketch you can create to show a 3-D object. Oblique pictorials are often preferred for freehand sketching because a specialized grid is not required. With oblique pictorials, as with **oblique axes**, the three dimensions of the object are shown with the height and width of the object in the plane of the paper and the third dimension (the depth) receding off at an angle from the others. Although the angle is usually 45 degrees, it can be any value.

The advantage that oblique pictorials have over isometric pictorials is that when one face of the object is placed in the plane of the paper, the object will appear in its true shape and size in that plane—it will be undistorted. This means that squares remain squares, rectangles remain rectangles, and circles remain circles. Figure 2.37 shows two pictorial representations of simple objects—one in isometric and one in oblique. Notice that the rules established for isometric pictorial sketches also hold true for oblique pictorial sketches—you do not show the hidden back edges, you show lines only where two surfaces intersect to form an edge, and you show only the visible parts of partially obstructed surfaces.

When making oblique sketches, the length of the **receding dimension** is not too important. In fact, oblique pictorials typically look better when the true length of the receding dimension is not shown. When the true length of an object's receding dimension is sketched, the object often appears distorted and unrealistic. Figure 2.38a shows the true length of a cube's receding dimension (use a ruler to make sure), and Figure 2.38b shows the same cube with the receding dimension drawn at about one-half to three-fourths its true length. Notice that the sketch in Figure 2.38a appears distorted—it does not look very much like a cube—whereas the sketch in Figure 2.38b looks like a cube even though the length of the receding dimension is less than the two sides of the square.

FIGURE 2.37. A comparison of isometric and oblique pictorials.

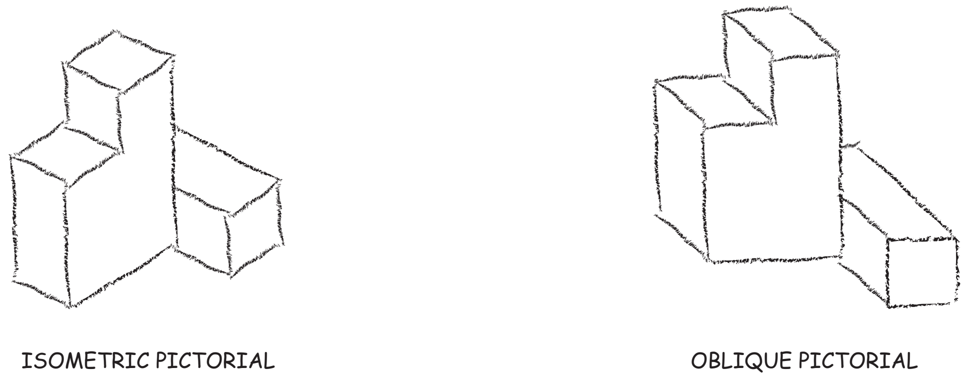
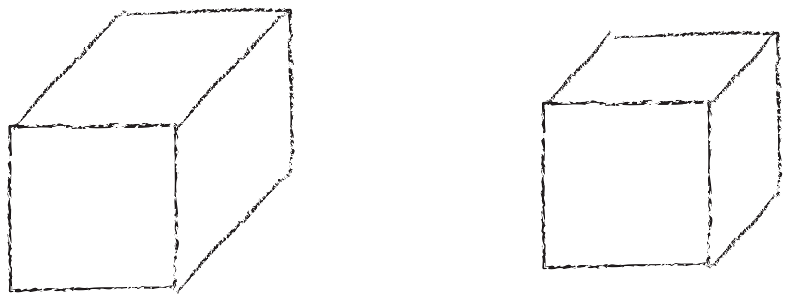


FIGURE 2.38. Oblique pictorials of a cube.



(a) Receding dimension drawn true length.

(b) Receding dimension drawn less than true length.

Other conventions pertain to the way the receding dimension is portrayed in an oblique sketch; you will learn about them in a later chapter. For now, you will concentrate on trying to make a sketch that looks proportionally correct.

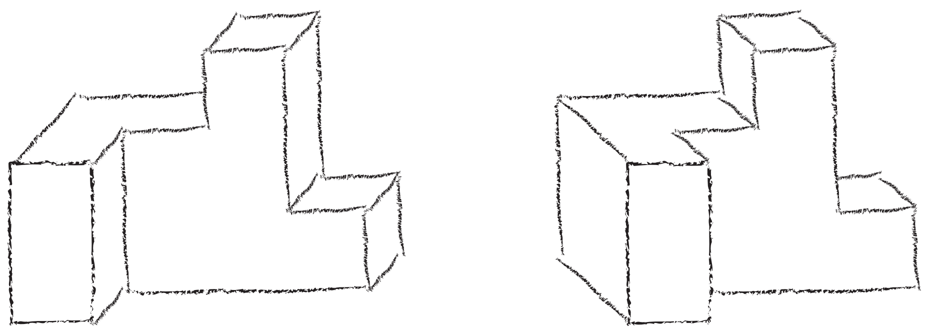
When creating oblique pictorials, you can choose to have the receding dimension going back and to the left or back and to the right. The direction you choose should be the one that produces the fewest obstructed surfaces in the resulting sketch. Figure 2.39 shows two possible sketches of the same object—one with the receding dimension to the left and one with the receding dimension to the right. Notice that the first sketch (Figure 2.39a) is preferable since none of the surfaces are obscured as they are with the second sketch (Figure 2.39b).

When creating an oblique pictorial, you should put the most irregular surface in the plane of the paper. This is particularly true about any surface that has a circular feature on it. Figure 2.40 shows two different oblique pictorials of the same object. In the first sketch (Figure 2.40a), the most irregular surface is placed in the plane of the paper as it should be; in the second sketch (Figure 2.40b), the irregular surface is shown in the receding dimension. Notice that the first sketch shows the features of the object more clearly than the second sketch does.

2.08.01 Circular Holes in Oblique Pictorial Sketches

When circular holes appear in an oblique pictorial sketch, as with isometric sketches, you show the partial edges of the back circle where they are visible through the hole. Once again, partial circles are visible when the object is relatively thin or when the hole has a relatively large diameter; otherwise, partial edges are not shown. Figure 2.41 shows two oblique sketches—one in which a portion of the back edge of the hole is visible and the other in which it is not.

FIGURE 2.39. Two possible orientations for an oblique pictorial.



(a) **(b)**

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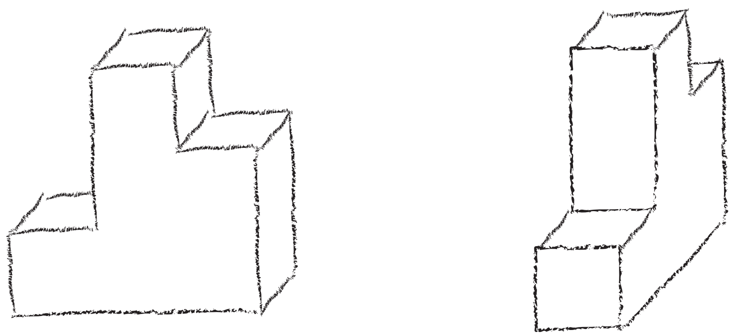
FIGURE 2.40. Two possible orientations for an oblique pictorial.



(a) Irregular surface in plane of paper.

(b) Irregular surface in receding direction.

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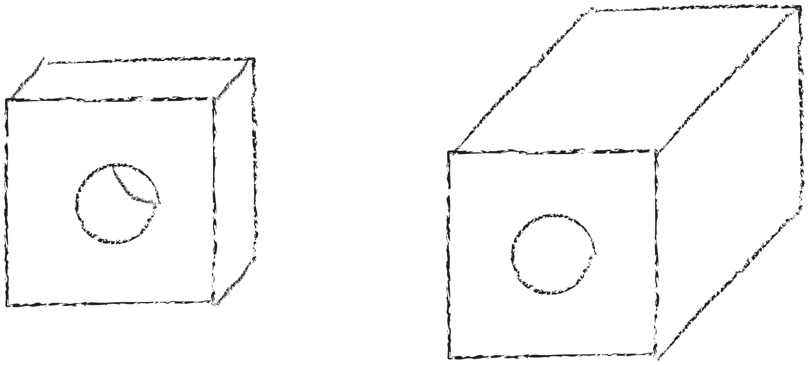

FIGURE 2.41. Oblique pictorials with circular holes in objects.

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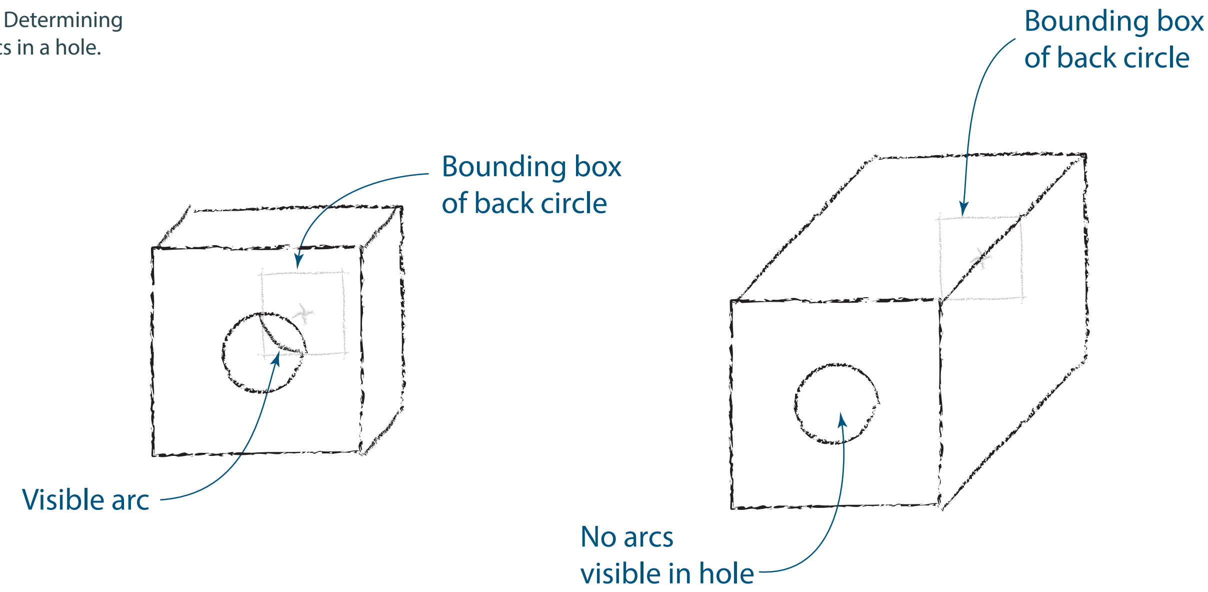

FIGURE 2.42. Determining visible back arcs in a hole.

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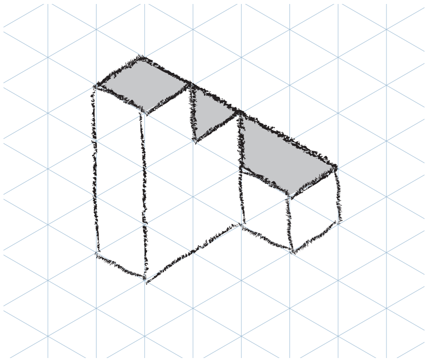
The procedure you use to determine whether a portion of the back circle edge is visible and, if so, which portion is visible follows the procedure outlined for isometric sketches. You start by locating the center of the back edge of the hole and marking off the four radial points. You then lightly sketch the bounding box that defines the circle. Finally, as needed, you sketch the visible portions of arcs within the circular hole. Figure 2.42 shows the procedure used to sketch the visible back edges of a circular hole in an oblique pictorial.

2.09 Shading and Other Special Effects

One thing you can do to improve the quality of your pictorial sketches is to include **shading** on selected surfaces to make them stand out from other surfaces or to provide clarity for the viewer. Figure 2.43 shows an isometric sketch with all of the top surfaces shaded. Notice that the shading better defines the object for the viewer. When including shading on a pictorial sketch, try not to overdo the shading. Too much shading can be confusing or irritating to the viewer—two things you should avoid in effective graphical communication.

Another common use of shading is to show curvature of a surface. For example, the visible portion of a hole's curved surface might be shaded in a pictorial sketch. A curved surface on an exterior corner also might be shaded to highlight its curvature.

FIGURE 2.43. An object with the top surface shaded.



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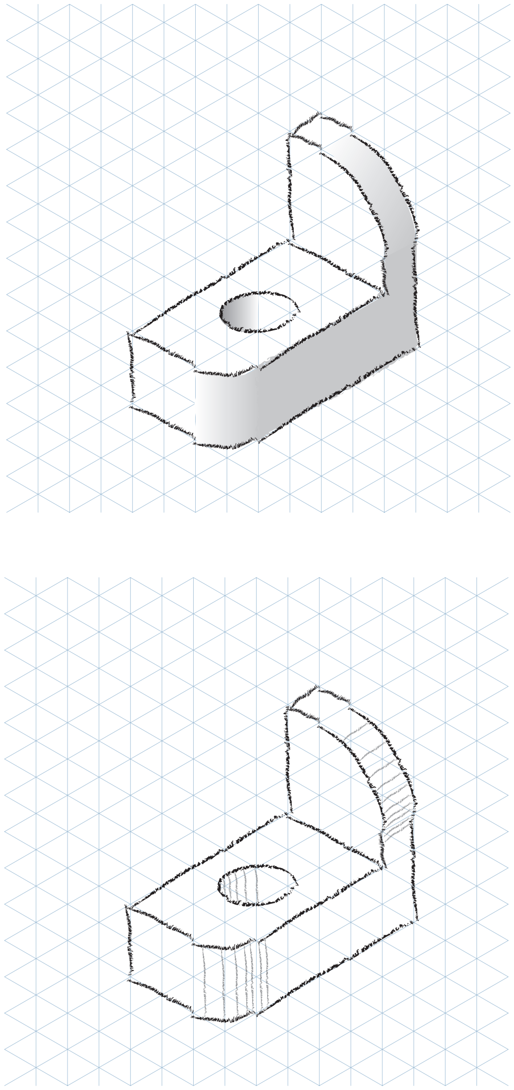

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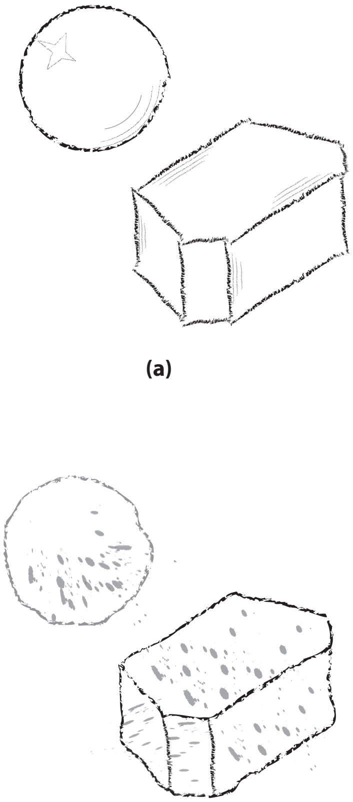
FIGURE 2.44. A simple object with two possible types of progressive shading used to emphasize the curvature of surfaces.

FIGURE 2.45. The addition of surface treatments to convey smooth surfaces (a) and rough surfaces (b).



(a)

(b)

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Figure 2.44 shows a pictorial sketch of a simple object with curved surfaces that are shaded.

Other sketching techniques can be used to convey features such as smooth or rough surfaces. Figure 2.45 shows different types of surface treatments that are possible for sketched objects.

You are probably familiar with techniques used in cartoons to convey ideas such as motion, temperature, and sound. Figure 2.46 shows typical cartooning lines that convey concepts not easily incorporated in a static sketch. Many of these same markings can be used in technical sketches. For example, Figure 2.47 uses action lines to convey motion for the sketch of linkages.

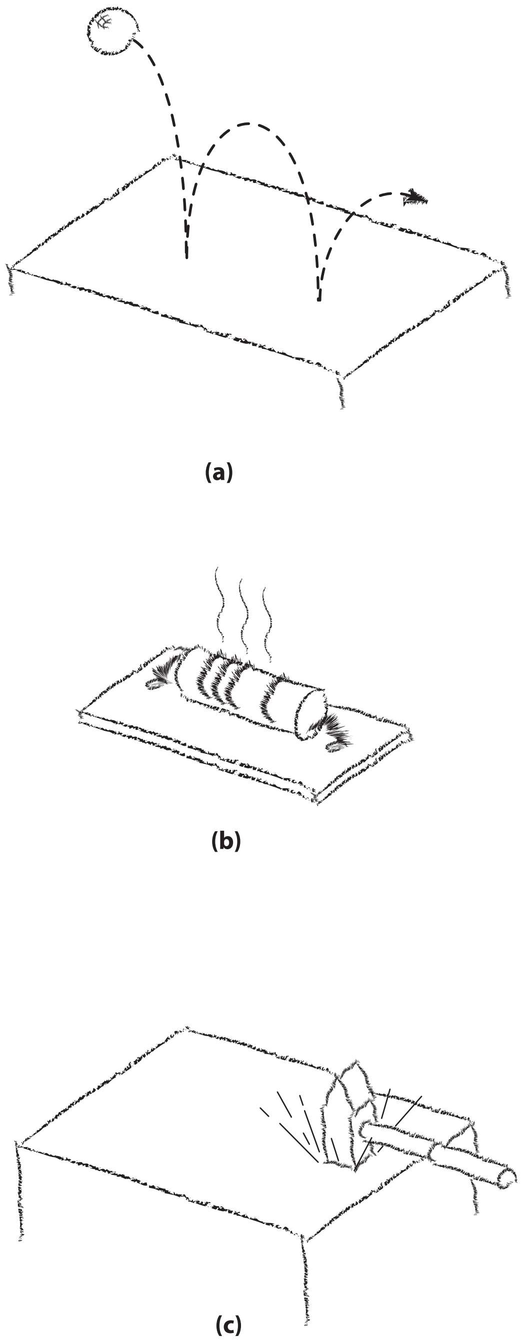

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FIGURE 2.46. Some sketching techniques that can be used to convey motion (a), temperature (b), and sound (c).

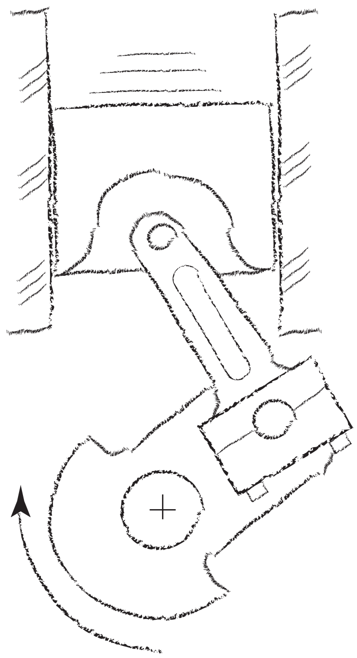

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FIGURE 2.47. Action lines used to convey the motion of linkages.

2.10 Sketching Complex Objects

As you refine your sketching skills, you will be able to tackle increasingly complex objects. Figures 2.48, 2.49, and 2.50 show pictorial sketches of small electronic devices. These sketches were not made to any particular scale, but were constructed so the object features appear proportionally correct with respect to one another. Notice the use of shading to enhance object appearance and to make the objects look more realistic. Being able to sketch relatively complicated objects such as these will improve your ability to communicate with colleagues throughout your career. To develop this

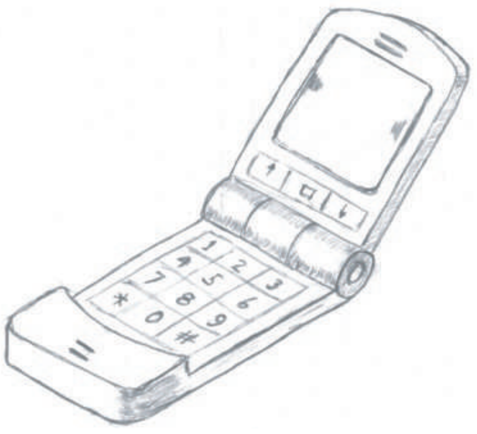

FIGURE 2.48. A sketch of a cell phone.

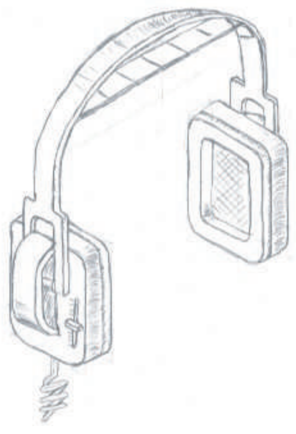

FIGURE 2.49. A sketch of a set of headphones.

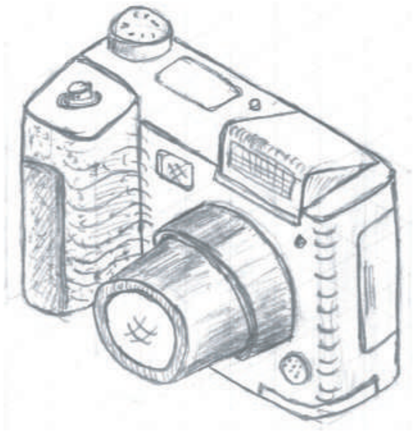

FIGURE 2.50. A sketch of a camera.

important skill, you should practice often. Do not be afraid to make mistakes—just keep trying until you get the results you want.

One way to tackle sketching a complex object is to think about it in the same way that a house is constructed—namely, "foundation, frame, finish." Using this method, you start with the "foundation" of the sketch, which usually consists of multiple guidelines and construction lines. When creating the sketch foundation, think about outlining the volume taken up by the entire object. You next "frame" the object by darkening some of the construction lines to define the basic shape of the object and its features. Once the basic frame is complete, you "finish" the sketch by adding necessary details and special features such as shading, especially on curved surfaces. Figure 2.51 shows a sketch of a flat panel computer monitor by the "foundation, frame, finish" method. Several of the exercises at the end of this chapter ask you to use this technique to develop your skills in sketching complex objects.

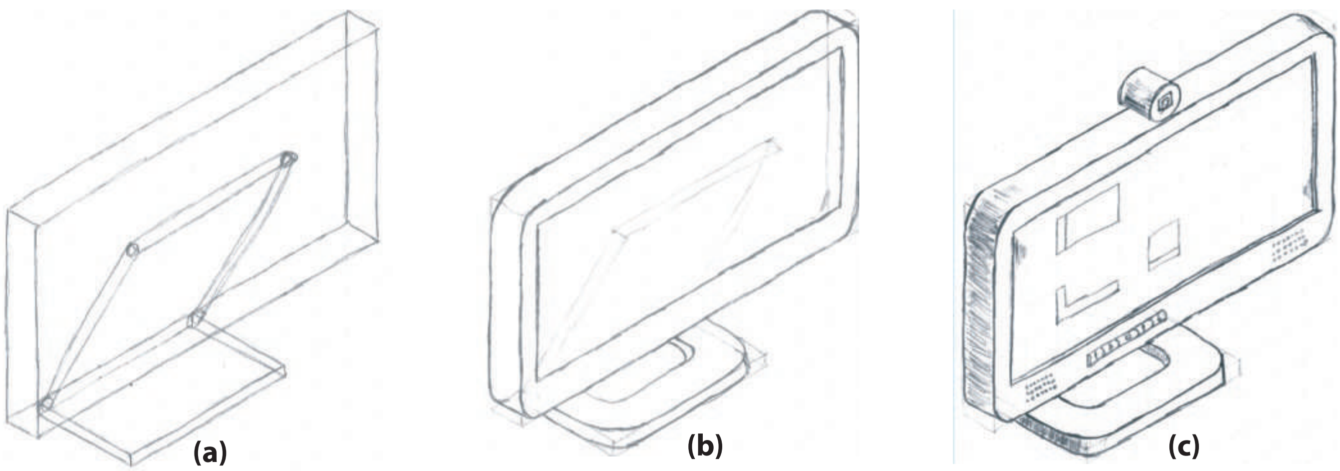

FIGURE 2.51. A sketch of a computer monitor using the method of "foundation (a), frame (b), finish (c)."

2.11 Chapter Summary

In this chapter, you learned about technical sketching and about some techniques to help you master this important form of communication. Specifically, you:

- Learned about the importance of sketching for engineering professionals and the link between creativity and freehand sketching.
- Developed techniques for successfully sketching basic shapes such as lines, arcs, circles, and ellipses.
- Learned about the right-hand rule and the way it is used to define 3-D coordinate systems in space. The axes can be portrayed on paper in either isometric or oblique format.
- Discovered how to make basic isometric sketches of objects from coded plans and about some of the rules that govern the creation of these sketches. You also learned about creating ellipses in isometric to represent circular holes in objects.
- Developed techniques for creating oblique pictorials. You also learned that for this type of pictorial, you should not show the receding dimension of the object true to size in order to avoid a distorted image.

2.12 GLOSSARY OF KEY TERMS

arc: A curved entity that represents a portion of a circle.

bounding box: A square box used to sketch circles or ellipses.

circle: A closed curved figure where all points on it are equidistant from its center point.

construction line: A faint line used in sketching to align items and define shapes.

ellipse: A closed curved figure where the sum of the distance between any point on the figure and its two foci is constant.

isometric axes: A set of three coordinate axes that are portrayed on the paper at 120 degrees relative to one another.

isometric dot paper: Paper used for sketching purposes that includes dots located along lines that meet at 120 degrees.

isometric grid paper: Paper used for sketching purposes that includes grid lines at 120 degrees relative to one another.

isometric pictorial: A sketch of an object that shows its three dimensions where isometric axes were used as the basis for defining the edges of the object.

left-handed system: Any 3-D coordinate system that is defined by the left-hand rule.

line: A spatial feature that marks the shortest distance between two points. A line has location, orientation, and length, but no area.

oblique axes: A set of three coordinate axes that are portrayed on the paper as two perpendicular lines, with the third axis meeting them at an angle, typically 45 degrees.

oblique pictorial: A sketch of an object that shows one face in the plane of the paper and the third dimension receding off at an angle relative to the face.

receding dimension: The portion of the object that appears to go back from the plane of the paper in an oblique pictorial.

right-hand rule: Used to define a 3-D coordinate system whereby by pointing the fingers of the right hand down the x-axis and curling them in the direction of the y-axis, the thumb will point down the z-axis.

right-handed system: Any 3-D coordinate system that is defined by the right-hand rule.

shading: Marks added to surfaces and features of a sketch to highlight 3-D effects.

3-D coordinate system: A set of three mutually perpendicular axes used to define 3-D space.

tick mark: A short dash used in sketching to locate points on the paper.

2.13 QUESTIONS FOR REVIEW

1. What is the role of sketching in engineering design? In creativity?
2. Describe which procedure you should use to sketch straight lines. (Are you right- or left-handed?)
3. How do circles appear on an isometric pictorial? On an oblique pictorial?
4. What is a bounding box?
5. How are construction lines used in sketching?
6. Why is it important to know the right-hand rule?

2.14 PROBLEMS

1. For each of the coordinate axes shown below, indicate whether they are isometric or oblique and whether they represent right- or left-handed systems.

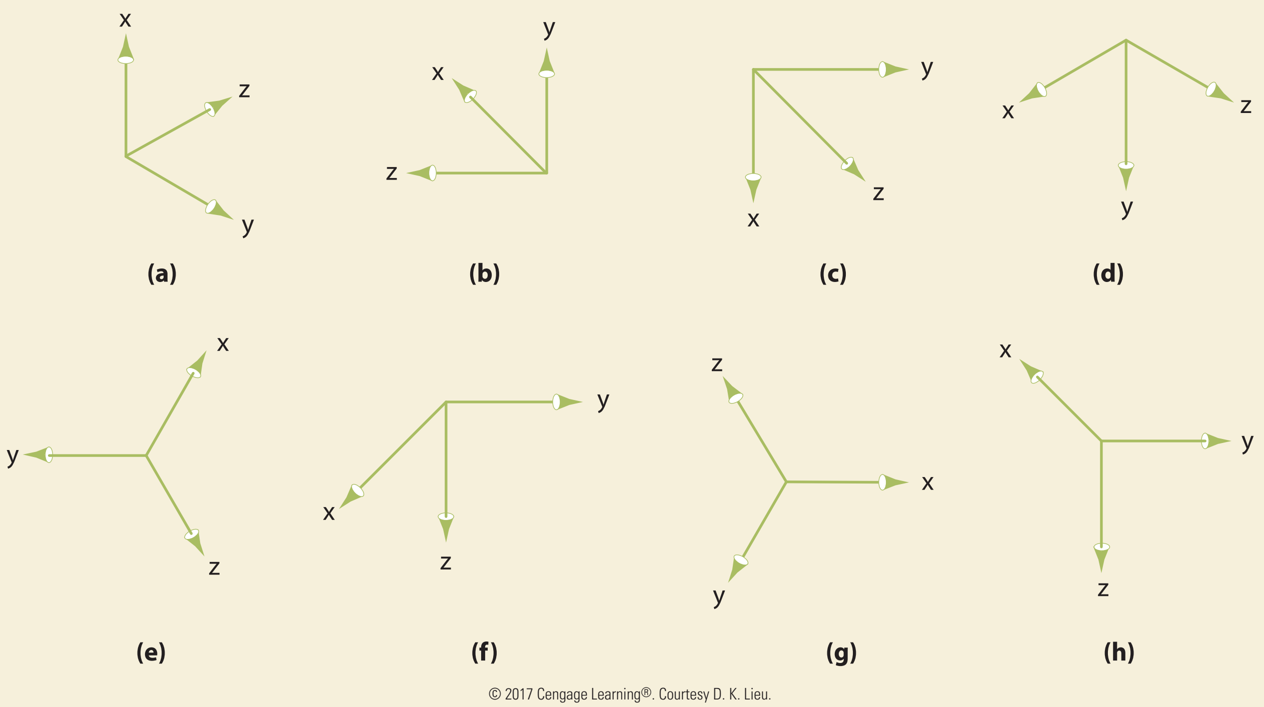

FIGURE P2.1.

2.14 PROBLEMS (CONTINUED)

2. Label the third axis in each of the following figures to define a right-handed system.

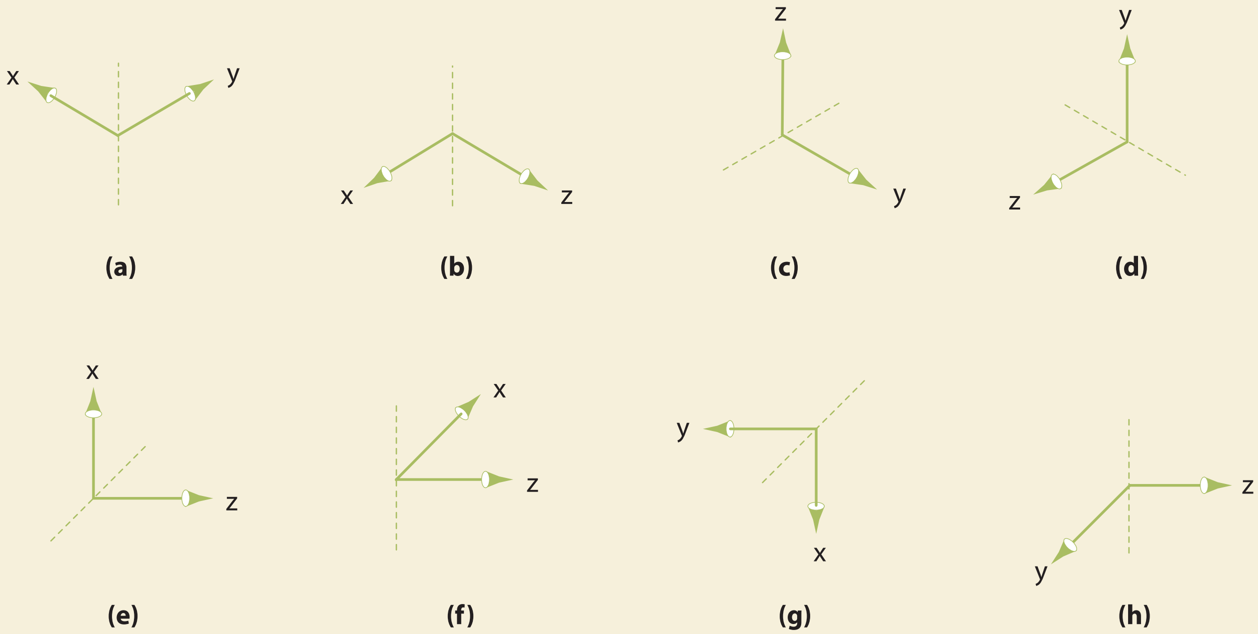

FIGURE P2.2.

2.14 PROBLEMS (CONTINUED)

3. Create isometric sketches from the coded plans shown below.

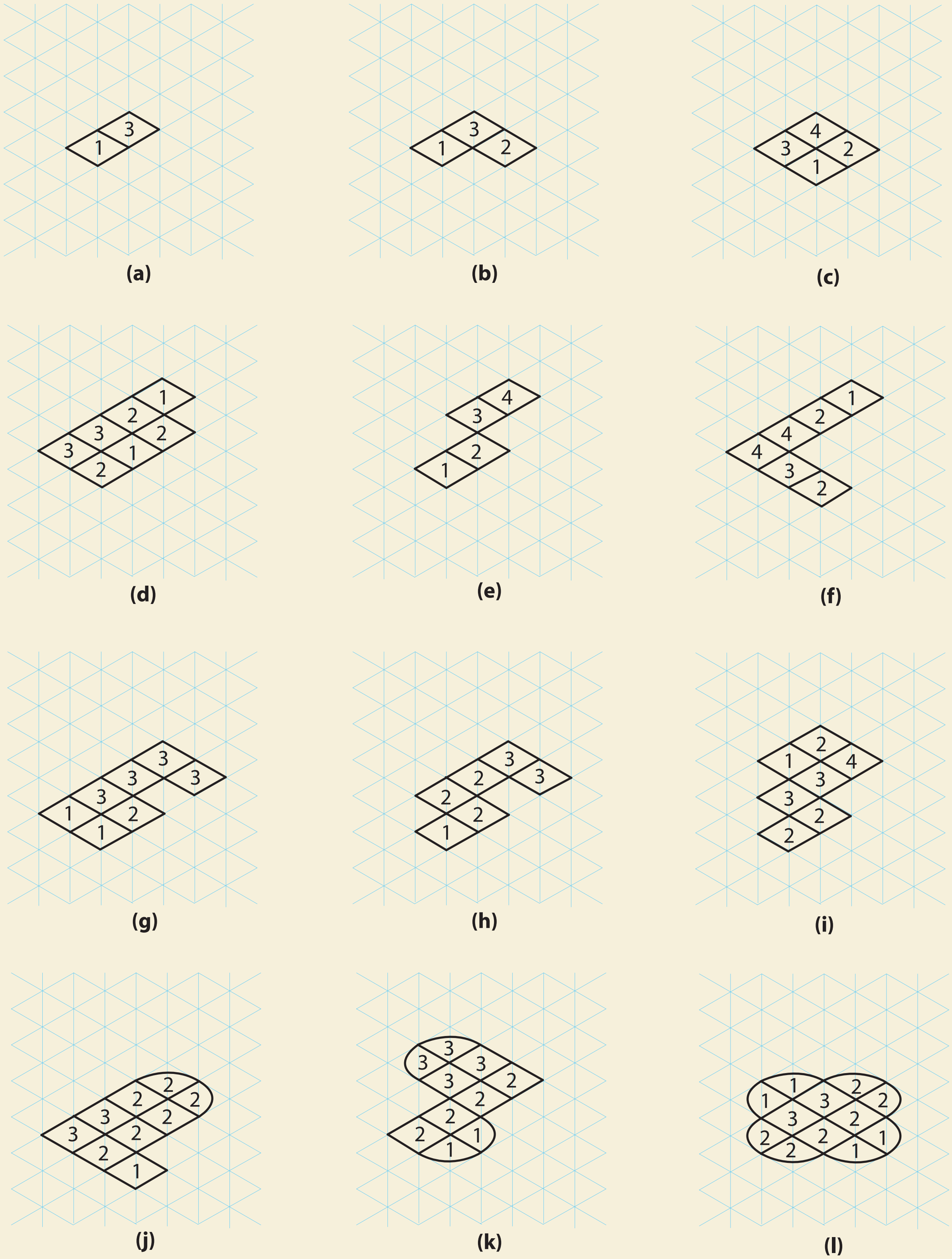

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FIGURE P2.3.

2.14 PROBLEMS (CONTINUED)

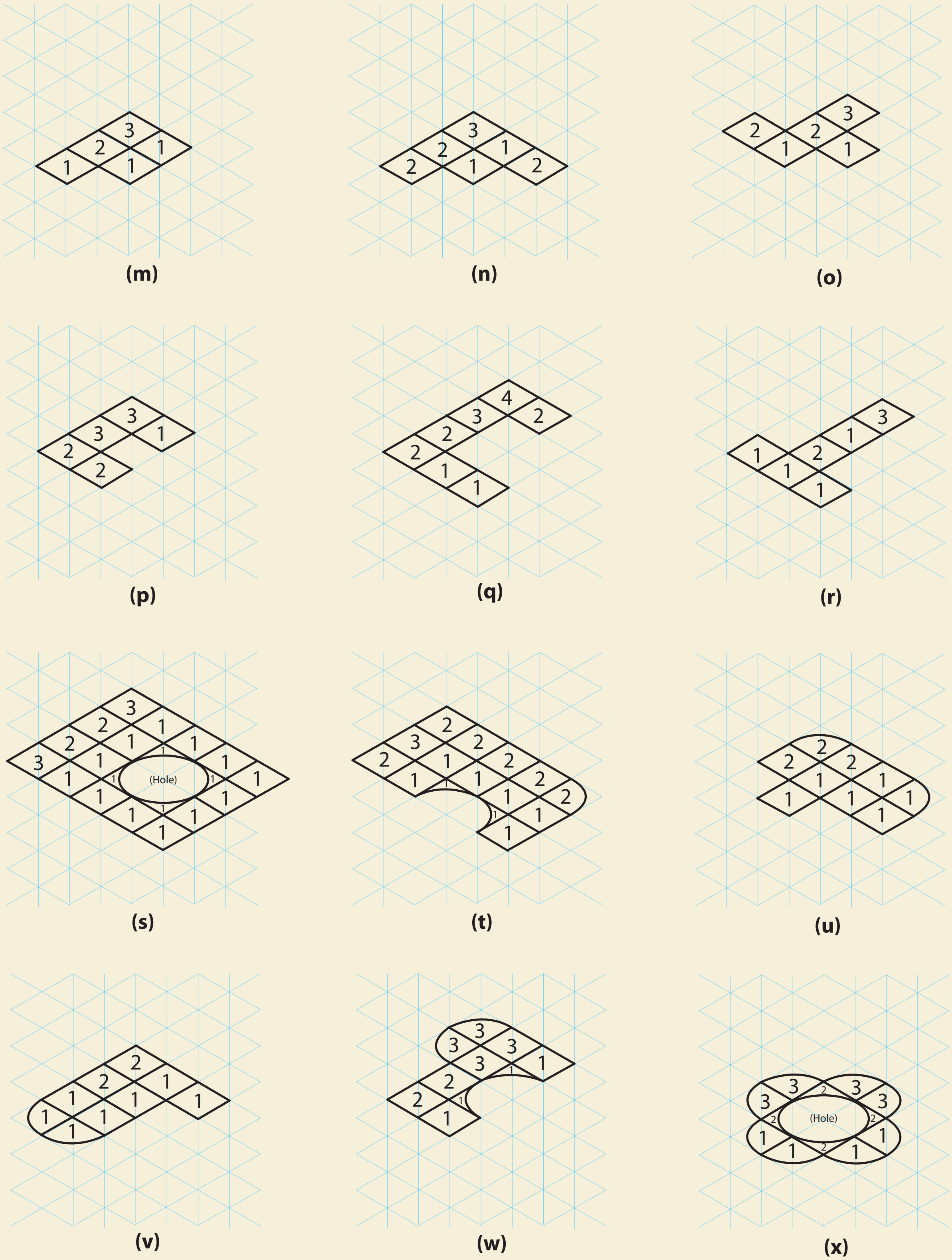

FIGURE P2.3. (Concluded)

2.14 PROBLEMS (CONTINUED)

4. Sketch a 6 × 6 × 2 block in isometric. On the 6 × 6 side, sketch a hole of diameter 4, making sure you include back edges of the hole as appropriate. Also create an oblique pictorial of the block.
5. Sketch a 6 × 6 × 2 block in isometric. On the 6 × 6 side, sketch a hole of diameter 2, making sure you include back edges of the hole as appropriate. Also create an oblique pictorial of the block.
6. Sketch a 6 × 6 × 4 block in isometric. On the 6 × 6 side, sketch a hole of diameter 2, making sure you include back edges of the hole as appropriate. Also create an oblique pictorial of the block.
7. From the isometric pictorials and viewing directions defined in the following sketches, create oblique pictorial sketches that look proportionally correct.

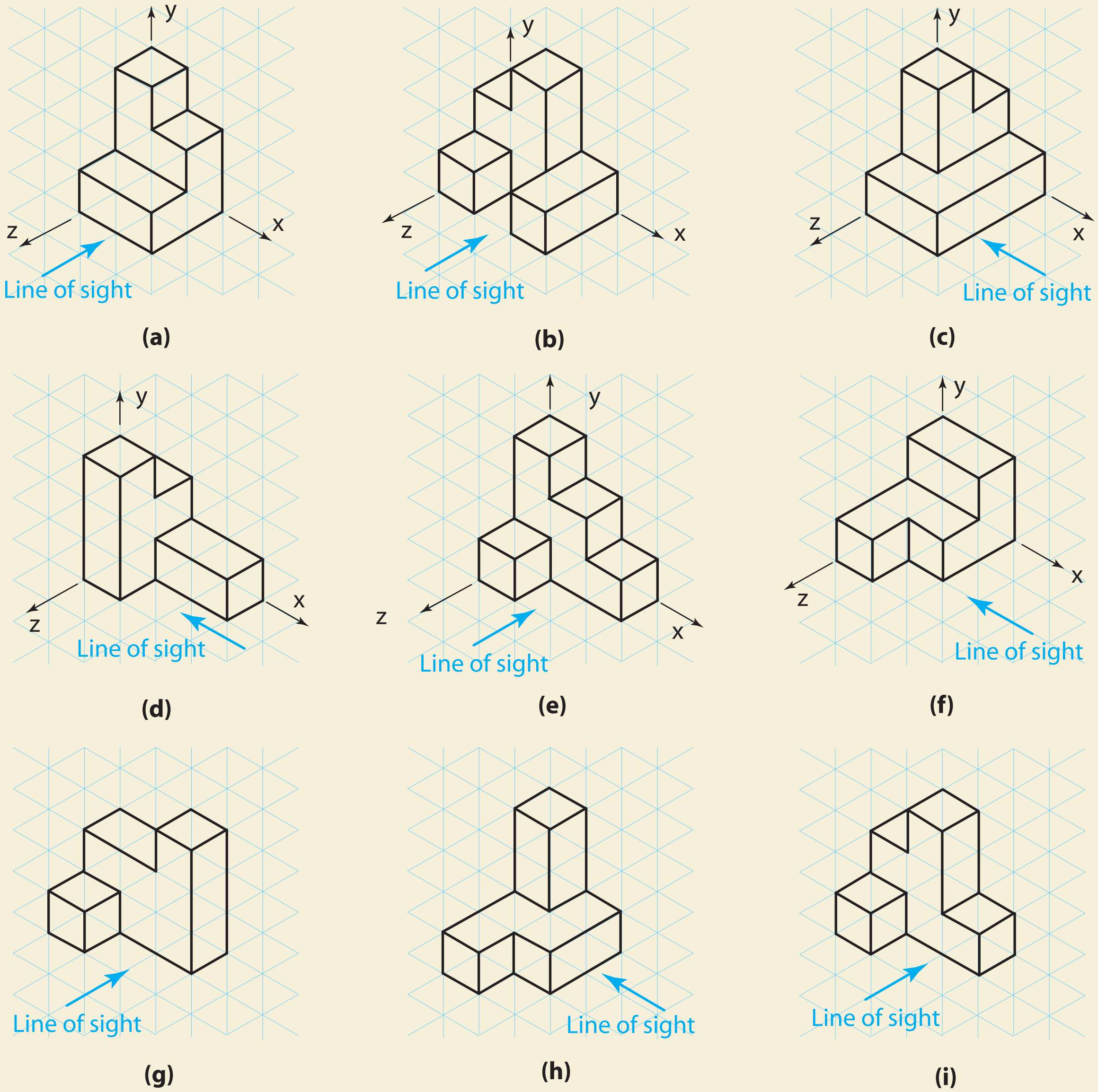

FIGURE P2.4.

2.14 PROBLEMS (CONTINUED)

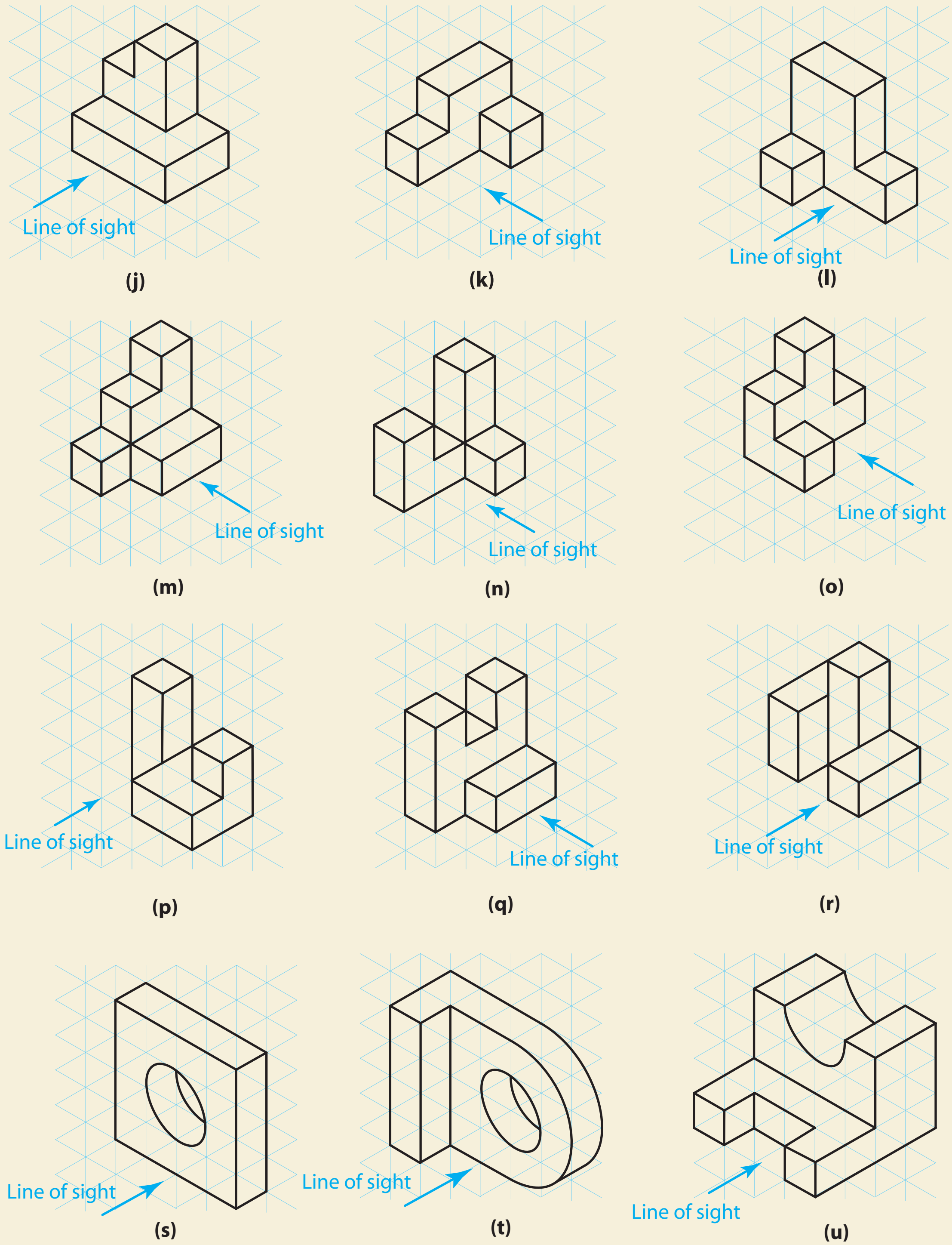

FIGURE P2.4. (Concluded)

8. Use the "foundation, frame, finish" method to create sketches of the following:

a. stapler
b. speedboat
c. coffee mug
d. bicycle
e. calculator
f. laptop computer
g. simple table
h. mobile phone
i. bookshelf

CHAPTER 3

VISUALIZATION

OBJECTIVES

After completing this chapter, you should be able to

- Recognize that 3-D spatial skills are necessary for success in engineering or technology
- Describe how a person's spatial skills develop as they age
- Examine the types of questions used to assess a person's spatial skill level
- Show how you can improve your 3-D spatial skills through techniques that include
  - Drawing different corner views of an object.
  - Rotating objects about one or more axes.
  - Sketching object reflections and making use of symmetries.
  - Considering cross sections of objects.
  - Combining two objects to form a third object through Boolean operations.

3.01 INTRODUCTION

When you start your first job in the real world, an engineer or a technologist is likely to hand you a drawing and expect you to understand what is on the page. Imagine your embarrassment if you have no clue what all of the lines and symbols on the drawing mean. One of the fundamental skills you need to understand is that drawing is the ability to visualize in three dimensions. The ability to visualize in three dimensions is also linked to creativity in design. People who think creatively are able to "see" things in their minds that others cannot. Their imaginations are not confined by traditional boundaries.

In this chapter, you will learn about the different types of three-dimensional (3-D) spatial skills and ways they can be developed through practice. The chapter will begin with an introduction to the background research conducted in education and to 3-D spatial skills. Then the chapter will take you through several types of visualization activities to further develop your 3-D skills through practice.

3.02 Background

Beginning in the early part of the twentieth century, IQ testing was developed to categorize a person based on his or her intelligence quotient. Anyone who took the IQ test was defined by a number that identified a level of intelligence. IQ scores over 140 identified geniuses; scores below 100 identified slow thinkers. Beginning in the 1970s, scholars began to perceive problems with this one-number categorization of a person's ability to think. One scholar in particular, Howard Gardner, theorized that there were multiple human intelligences and the one-number-fits-all theory did not accurately reflect the scope of human thought processes. Although some of his theories might be subject to scrutiny, they have gained acceptance within the scientific and educational communities. Gardner theorized that there are eight distinct human intelligences; he identified them as:

- Linguistic—the ability to use words effectively in speaking or in writing.
- Logical-Mathematical—the ability to use numbers effectively and to reason well.
- Spatial—the ability to perceive the visual-spatial world accurately and to perform transformations on those perceptions.
- Bodily-Kinesthetic—the capacity of a person to use the whole body to express ideas or feelings and the facility to use the hands to produce or transform things.
- Musical—the capacity to perceive, discriminate, transform, and express musical forms.
- Interpersonal—the ability to perceive and make distinctions in the moods, intentions, motivations, and feelings of other people.
- Intrapersonal—self-knowledge and the ability to act adaptively on the basis of that knowledge.
- Naturalist—the ability to recognize plant or animal species within the environment.

You may be acquainted with someone who has a high level of linguistic intelligence but a low level of musical intelligence. Or you might know someone who has a high level of logical-mathematical intelligence but who lacks interpersonal intelligence relationships. You may even have a friend who is generally smart but who lacks intrapersonal intelligence and attempts stunts that are beyond his or her limitations.

Most people are born with one or more of the intelligences listed. As a child, Tiger Woods was gifted with a natural ability in bodily-kinesthetic intelligence.

Mozart was born with a high level of musical intelligence. However, just because a person naturally has a high level of intelligence in one area does not mean that he or she cannot learn and improve his or her abilities in weaker areas. A person might naturally have strength in linguistics and musical intelligences, but he or she can still learn and improve in logical-mathematical endeavors. The goal of this chapter is to help those of you who were not born with a high level of spatial intelligence as defined by Gardner.

Learning in general and spatial skills in particular have been the subjects of education research studies over the past several decades. The following are a few important questions that the research raised in the area of spatial intelligence:

- How does a person develop spatial skills?
- Why does a person need well-developed spatial skills?
- How are spatial skills measured?

The next few sections will examine researchers' answers to these questions.

3.03 Development of Spatial Skills

As a child grows, the brain develops in ways that enable the child to learn. If you think of each of the eight intelligences described by Gardner, you can understand how these skills and abilities develop as a child grows to maturity. Consider kinesthetic intelligence. Newborn infants cannot move on their own during the first few weeks of life. Within a few months, they can hold up their heads without support. By the age of four months, they can roll over; by six months, they can crawl; by one year, they can walk. Children learn to run, skip, and jump within the next year or so. Eventually, they usually develop all sorts of kinesthetic abilities that enable them to enjoy physical activities such as basketball, swimming, ballet, and bike riding. Nearly every child goes through this natural progression. However, some children develop more quickly than others; some even skip a step and go directly from rolling over to walking without ever crawling. As with most types of intelligence, some individuals—such as professional athletes—have exceptional kinesthetic skills, while others have poorly developed skills and struggle to perform the simplest tasks. However, even people who naturally have little kinesthetic ability can improve through practice and perseverance.

The remaining intelligences (mathematics, verbal, etc.) also have a natural progression; for example, to develop your mathematical intelligence, you have to learn addition before you can learn algebra. Children also acquire spatial skills through a natural progression; however, you may not be as aware of that progression of development as you are of the progressions for the other intelligences. Educational psychologists theorize that there are three distinct stages of development for spatial skills.

The first stage of development involves 2-D spatial skills. As children develop these skills, they are able to recognize 2-D shapes and eventually are able to recognize that a 2-D shape has a certain orientation in space. If you watched *Sesame Street* as a child, you may remember the game where four pictures of 2-D objects are shown on the TV screen—three objects are identical; the fourth is different in some way. A song urges you to pick out the object that does not belong with the other three. A child who can accomplish this task has developed some of the spatial skills at the first stage. You also may remember playing with a toy similar to a Tupperware™ ShapeSorter, shown in Figure 3.01. The toy is a ball that is half red and half blue with ten holes in it, each hole a different shape. A child playing this game not only has to recognize that the star-shaped piece corresponds to the star-shaped hole but also has to turn the piece to the correct orientation to fit the piece through the hole. This game challenges different 2-D skills found at the first stage of development of spatial intelligence—a child must recognize the 2-D shape of the object and then must be able to recognize its orientation in 2-D space to complete the task.

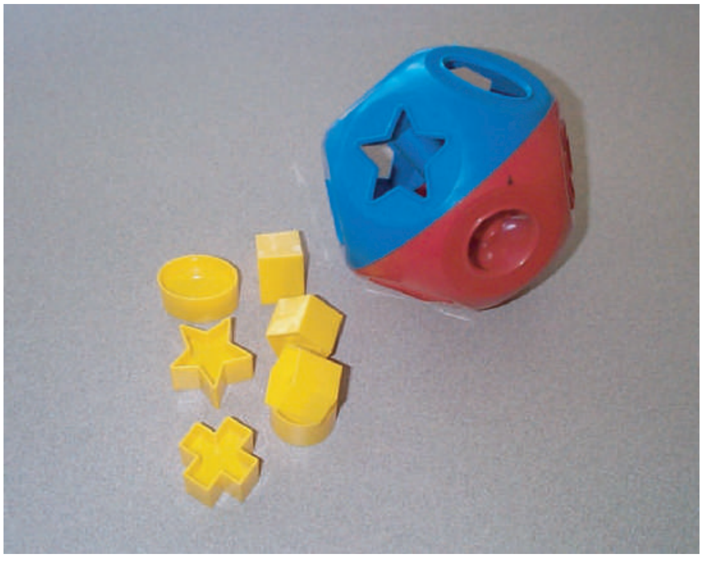

FIGURE 3.01. A Tupperware™ ShapeSorter toy.

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Three-dimensional spatial skills are acquired during the second stage of development. Children at this stage can imagine what a 3-D object looks like when it is rotated in space. They can imagine what an object looks like from a different point of view, or they can imagine what an object would look like when folded up from a 2-D pattern. People who are adept at solving the Rubik's cube puzzle have well-developed 3-D spatial skills. Computer games such as 3D Tetris require well-developed 3-D spatial skills to perform the manipulations required to remain "alive." Soccer players who can imagine the trajectory that puts the ball in the goal from any angle on the playing field typically have well-developed 3-D spatial skills. Children have usually acquired 3-D spatial skills by the time they are in middle school. For some children, it may take a few more years, depending on their natural predisposition toward spatial intelligence and their childhood experiences.

People at an advanced stage of the development of spatial intelligence can combine their 3-D skills with concepts of measurement. Assume you are buying sand for a turtle-shaped sandbox. You go to the local gravel pit where an employee loads the sand in the back of your pickup using a big "scoop." How many scoopfuls will you need? If you can successfully visualize the volume of sand as it is transformed from the 3-D volume of one full scoop to the 3-D volume of the turtle-shaped sandbox, you have acquired this advanced 3-D visualization skill.

Many people never develop the advanced level in spatial intelligence, just like the many people who never achieve advanced skill levels in mathematics or kinesthetic intelligence. Not achieving advanced levels in some of the intelligence areas is not likely to hamper your ability to become a productive, well-adjusted member of society. However, just as a lack of basic development in verbal intelligence is likely to hurt your chances professionally, a lack of basic skills in spatial intelligence may limit your ability to be successful, especially in engineering or a technical field.

Schools help students develop most of the intelligence types; however, schools do not usually provide formal training to develop spatial intelligence. You began learning mathematics in kindergarten and are likely continuing your education in math at the present time. If you get a graduate degree in a technical area, you will probably be developing your mathematical intelligence for many years thereafter. The focus on developing spatial skills from an early age, continuing through high school and beyond, is typically absent in the U.S. educational system. Developing spatial intelligence is largely ignored in schools for a variety of reasons; however, those reasons are not the subject of this text.

The lack of prior spatial training may not be a problem for you—you developed your spatial skills informally through everyday experiences or you naturally have a high level of ability in spatial intelligence. However, poorly developed 3-D spatial skills may hinder your success in fields such as engineering and technology. This is especially true as you embark on a journey through an engineering graphics course. Poorly developed spatial skills will leave you frustrated and possibly discouraged about engineering graphics. The good news is that if you do not have a naturally high level of ability in 3-D spatial skills you can develop them through practice and exercise.

3.04 Types of Spatial Skills

According to McGee (1979), spatial ability is "the ability to mentally manipulate, rotate, twist, or invert pictorially presented visual stimuli." McGee identifies five components of spatial skills:

- **Spatial perception**—the ability to identify horizontal and vertical directions.
- **Spatial visualization**—the ability to mentally transform (rotate, translate, or mirror) or to mentally alter (twist, fold, or invert) 2-D figures and/or 3-D objects.
- **Mental rotations**—the ability to mentally turn a 3-D object in space and then be able to mentally rotate a different 3-D object in the same way.
- **Spatial relations**—the ability to visualize the relationship between two objects in space, i.e., overlapping or nonoverlapping.
- **Spatial orientation**—the ability of a person to mentally determine his or her own location and orientation within a given environment.

A different researcher proposed a classification scheme for spatial skills based on the mental processes that are expected to be used in performing a given task. She believes that there are two distinct categories of 3-D spatial skills—spatial visualization and spatial orientation. Spatial visualization is mentally moving an object. Spatial orientation is mentally shifting the point from which you view the object while it remains fixed in space.

Regardless of the classification scheme you choose to believe, it is clear that more than one component skill makes up the broad category of human intelligence known as spatial visualization. Thus, you cannot do just one type of activity and expect to develop all of the components of spatial skills equally. You need to do a variety of tasks to develop your spatial intelligence, just as developing linguistic intelligence requires you to speak, read, write, and listen.

3.05 Assessing Spatial Skills

As with the seven other intelligence types, standardized tests have been developed to determine your level of achievements in spatial intelligence. There are many different tests—some are for 2-D shapes, and some are for 3-D objects. Some evaluate mental rotation skills, and others measure spatial relations skills. The standardized tests usually measure only one specific component of visualization skill. If you were to take a number of different visualization tests, you might find that you have a high level of ability in one component (perhaps paper folding) relative to a low ability in a different component, such as 3-D object rotations. That is normal. Many educators and psychologists believe there is no one-size-fits-all measure of spatial intelligence, just as a single IQ number does not give a clear indication of a person's overall intelligence.

One of the tests designed to measure your level of 2-D spatial skills is the Minnesota Paper Form Board (MPFB) test. Figure 3.02 shows a visualization problem similar to those found on the MPFB test. This problem tests a person's ability to determine which set of five 2-D shapes, A through E, is the composite of the 2-D fragments given in the upper left corner of the figure. The way to solve this test is to

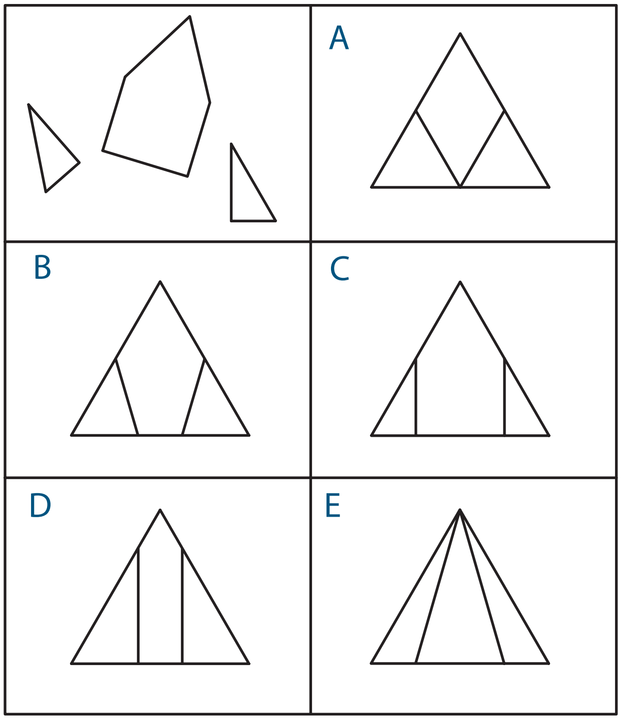

FIGURE 3.02. A problem similar to that found on the Minnesota Paper Form Board test.

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mentally rotate or move the three pieces to visualize how to put them together to coincide with the combined shape that contains the pieces. The test may seem easy, but you should have fully developed the 2-D spatial skills needed to solve this test when you were four or five years old. During the years since then, you should have developed more advanced 2-D visualization skills. For example, you may now be able to follow a map and determine whether to make a right or a left turn without turning the map.

Figure 3.03 shows a visualization problem similar to what is found on the Differential Aptitude Test: Space Relations. This test is designed to measure your ability to move from the 2-D to the 3-D world. The objective is to mentally fold the 2-D pattern along the solid lines, which designate the fold lines, so the object will result in the 3-D shape. You then choose the correct 3-D object from the four possibilities shown in the figure. In your previous math classes, these 2-D patterns may have been referred to as *nets*. In engineering, the 2-D figures are called *flat patterns* or *developments*.

Mental rotation—the ability to visualize the rotation of 3-D objects—is a necessary component skill in engineering graphics and in the use of 3-D modeling software. Figures 3.04 and 3.05 show problems similar to those found on two widely used 3-D spatial tests for rotations.

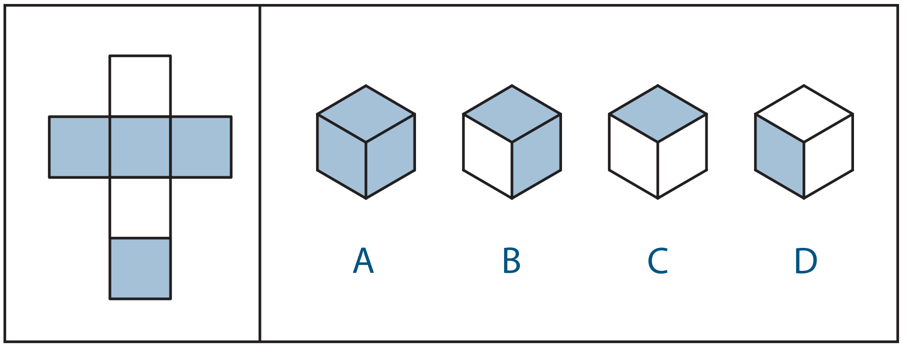

FIGURE 3.03. A problem similar to that found on the Differential Aptitude Test: Space Relations.

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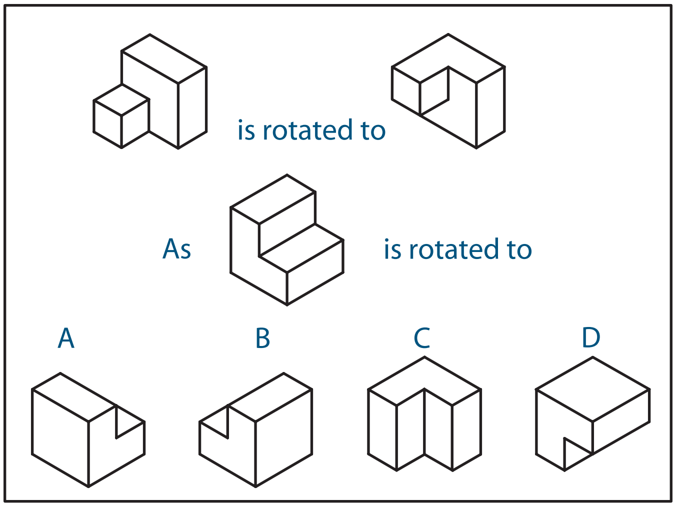

Figure 3.04. A problem similar to that found on the Purdue Spatial Visualization Test: Rotations.

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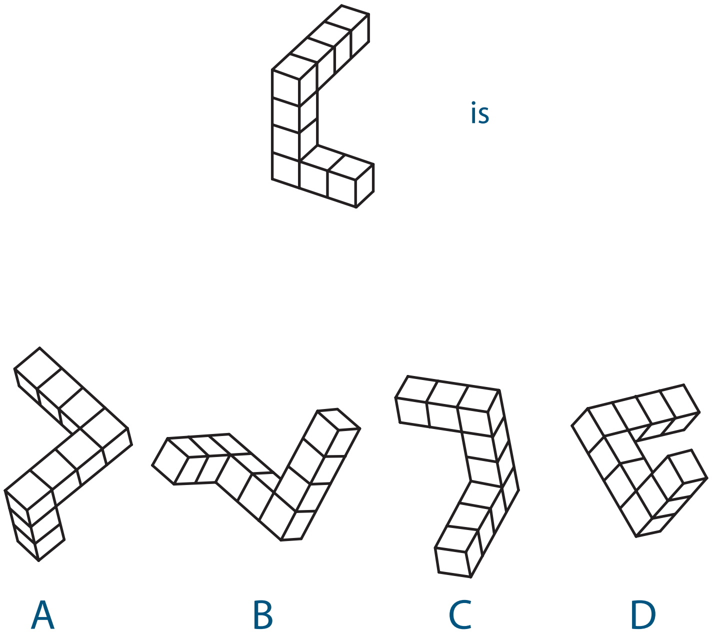

FIGURE 3.05. A problem similar to that found on the Mental Rotation Test.

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In the Purdue Spatial Visualization Test: Rotations, an object such as is shown in Figure 3.04 is given on the top line before and after it has been rotated in 3-D space. You then have to mentally rotate a different object on the second line by the same amount and select the correct result from the choices given on the third line.

In the Mental Rotation Test, you are given an object such as shown in Figure 3.05 at the top. Of the four choices given, you pick the *two* that show correct possible rotations in space of the original object. (Note that two choices are the same object and two choices are different objects.)

Another type of spatial skill that is often tested is the ability to visualize the **cross section** that results from "slicing" a 3-D object with a **cutting plane**. One popular test of this type is the Mental Cutting Test. Figure 3.06 shows the type of problem found on this test, which challenges you to imagine the 2-D shape that is the intersection between the cutting plane and the 3-D object.

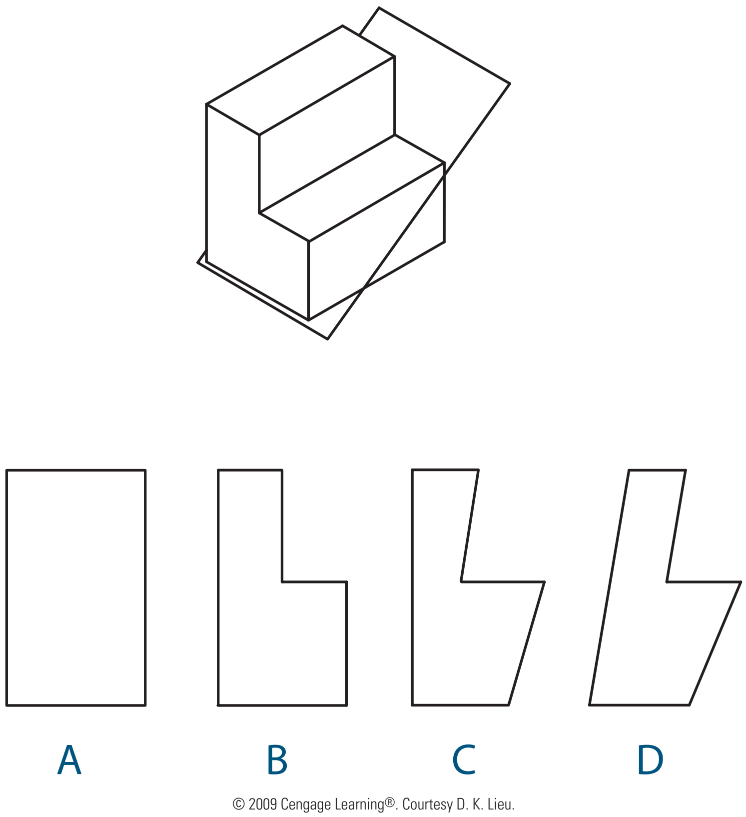

FIGURE 3.06. A problem similar to that found on the Mental Cutting Test.

3.06 The Importance of Spatial Skills

Engineers and technologists communicate with each other largely through graphical means. They use drawings, sketches, charts, graphs, and CAD models to convey ideas. Design solutions commonly have a graphical component that is backed up by pages of calculations and analysis. Your designs will not be complete without graphics. Even chemical and electrical engineers use drawings for the processes and circuits they design.

So to communicate as an engineer, you must be able to visualize and interpret the images represented in the drawings. Besides satisfying the need for effective communication, a side benefit to having well-developed 3-D spatial skills is that your brain works better when *all* parts of it are focused on solving a problem. Sketching and visualization have been shown to improve the creative process. Well-developed spatial skills contribute to your ability to work innovatively, as well as to learn to use 3-D modeling software.

The remaining sections of this chapter will provide exercises for your brain—exercises that develop your 3-D spatial skills; exercises that help you think differently from the way you are thinking in your math and science courses; exercises that will help you improve your sketching skills.

3.07 Isometric Corner Views of Simple Objects

In Chapter 2, you learned how to create a simple isometric sketch of an object made out of blocks as specified by a coded plan. The coded plan is a 2-D portrayal of the object, using numbers to specify the height of the stack of blocks at a given location. Figure 3.07 illustrates the relationship between the coded plan, the object constructed

FIGURE 3.07. A coded plan and its resulting isometric sketch.

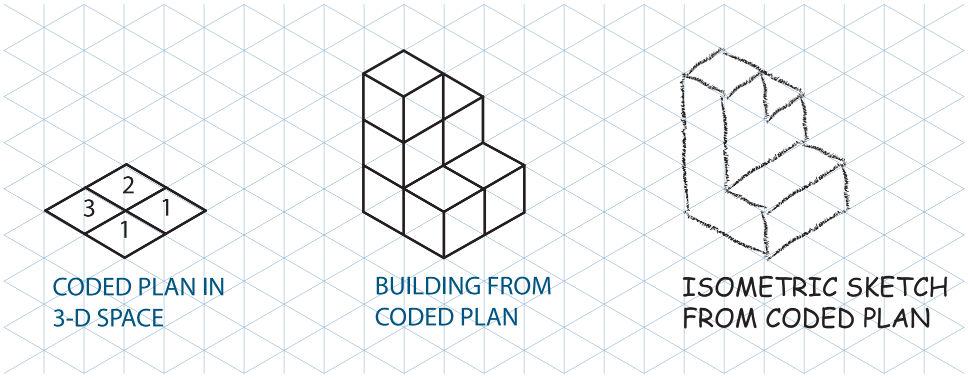

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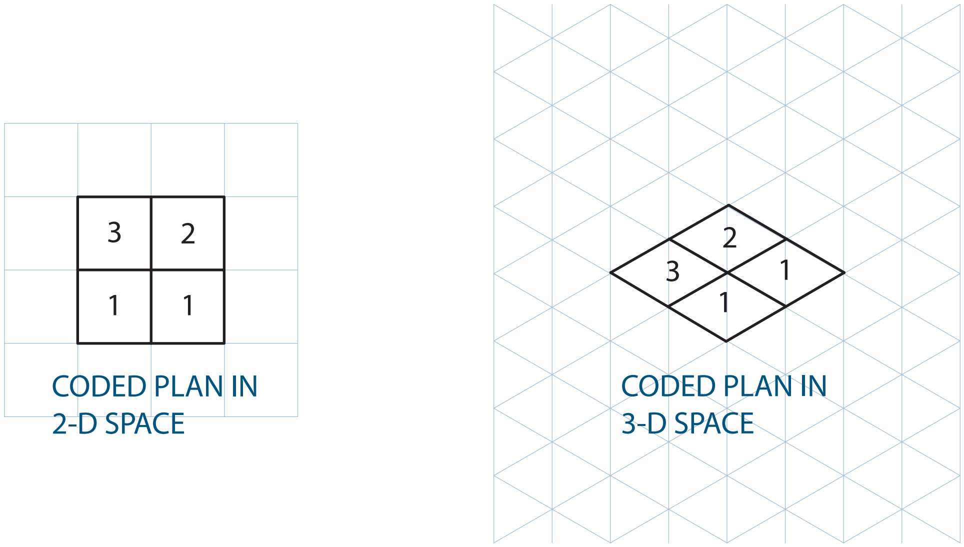
FIGURE 3.08. The relationship between a coded plan in 2-D space and 3-D space.



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out of blocks, and the resulting isometric sketch of the object—remember, you show edges only between surfaces on the isometric sketch.

The coded plans you viewed in Chapter 2 were constructed on isometric grid paper. The building "grew up" from the plan into the isometric grid. In the previous exercises, the coded plan was oriented in 3-D space on the isometric grid, which represents 3-D space. Now think about laying the coded plan flat on a 2-D sheet of paper. Figure 3.08 shows the coded plan for the object shown in Figure 3.07 laid flat in a 2-D orientation. Figure 3.09 shows the relationship between a coded plan in 2-D space, the coded plan in 3-D space, the object made of blocks, and the resulting isometric sketch.

When you orient the coded plan in 2-D space, everything you learned about these plans still applies: you "build up" from the plan. The numbers represent the height of the stack of blocks at a given location, and you show lines only where two surfaces intersect. However, now one more consideration has been introduced into the isometric sketching equation—the orientation of your "eye" with respect to the object itself. (Note that *the orientation of your eye* is often referred to as *your viewpoint*.)

Examine again the coded plans in 2-D space. Figure 3.10 shows a simple coded plan with its four corners labeled as w, x, y, and z. A **corner view** of the object represented by the coded plan in Figure 3.10 is the view from a given corner when

FIGURE 3.09. The relationship between coded plans, a building, and an isometric sketch.

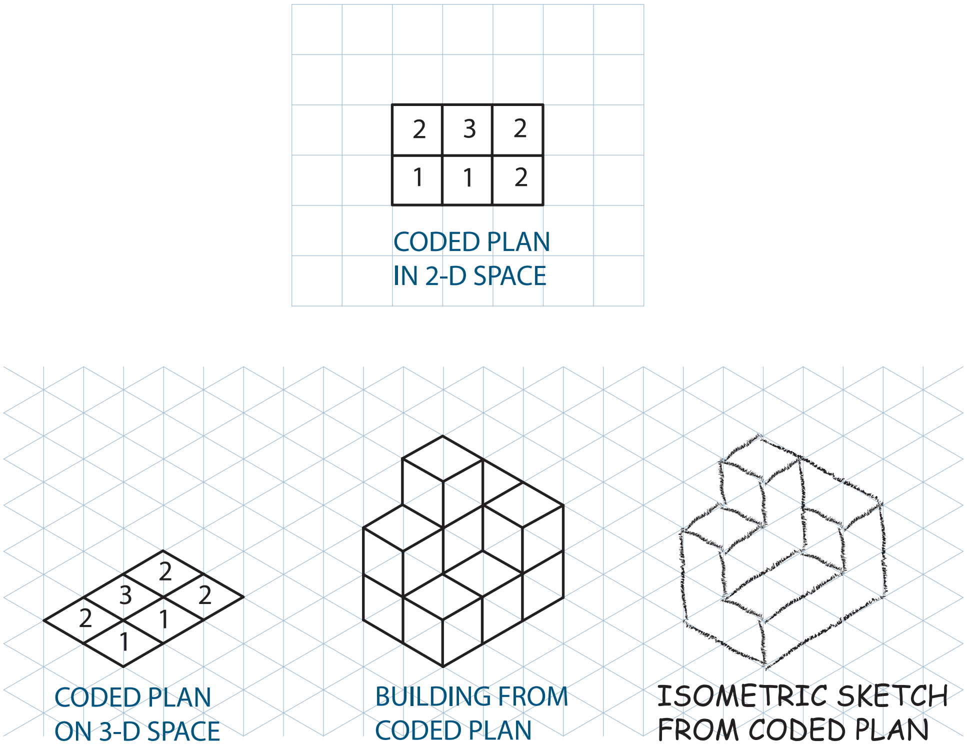
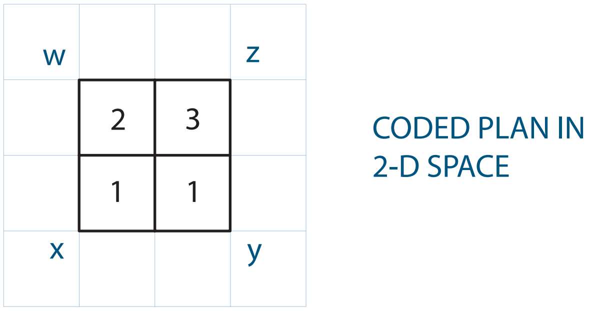


FIGURE 3.10. A simple coded plan with corners labeled.



the viewpoint is *above* the object in question. This view is sometimes referred to as *the bird's-eye view*, because the viewpoint is above the object. A worm's-eye view is the viewpoint from *below* the object. Figure 3.11 shows the four corner views for the coded plan from Figure 3.10.

When the four corner views of the object are created, the object does not change—just your viewpoint of the object. The importance of viewpoint in visualization is readily apparent when you think about a complex system such as an automobile. When you are looking at a car from the front, you may have an entirely different mental image of the car than if you look at it from the side or rear. What you "see" depends largely on where your eye is located relative to the object.

With more practice, you will find it easier to make corner views from coded plans. At first, you may need to turn the paper to visualize what an object will look like from a given corner. With continued practice, however, you should be able to mentally turn the paper to sketch the object from the vantage point of any corner.

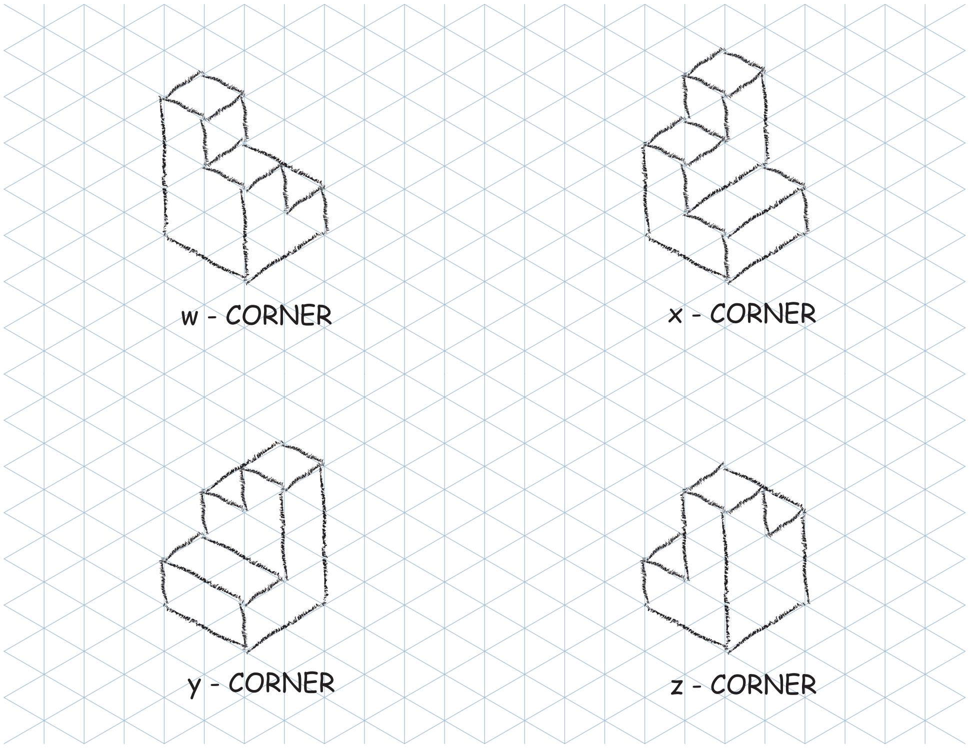


FIGURE 3.11. Sketched isometric views from the corners of the coded plan in Figure 3.10.

3.08 Object Rotations about a Single Axis

Being able to mentally visualize an object as it rotates in space is an important skill for you to acquire as an engineer or a technologist. You already have had limited exposure to the concept of rotating objects through your work with mentally rotating coded plans to obtain different corner views. In the preceding section, you started with the y-corner view to draw the isometric. Having done that, you should be able to imagine what the object will look like from the x-corner view. If you can see in your mind what the object looks like from the x-corner view, you are mentally rotating the object in space. In this section, you will continue to work with object rotations, tackling increasingly complex objects and using increasingly complex manipulations.

You probably learned in your math classes how 2-D shapes are rotated in 2-D space about a pivot point, as illustrated in Figure 3.12. In this figure, the shape has

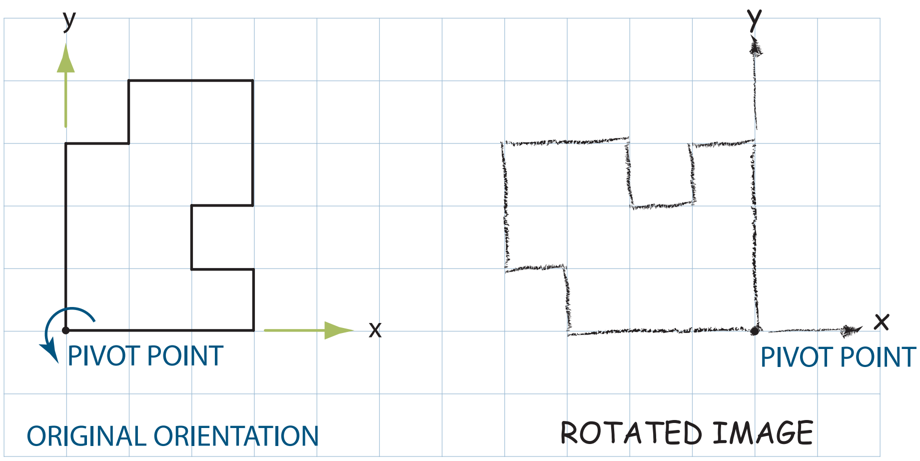


FIGURE 3.12. A shape rotated about a pivot point in 2-D space.

been rotated 90 degrees counterclockwise (CCW) about the pivot point, which is the origin of the 2-D xy-coordinate system. After rotation, the newly oriented shape is referred to as the "image" of the original shape. Notice that when the 2-D shape is rotated about the pivot point, each line on the shape is rotated by the same amount—in this case, 90 degrees CCW about the pivot point. Also notice that the point on the shape that was originally located at the pivot point, the origin, remains at that same location after rotation.

In Chapter 2, you learned about 3-D coordinate systems and how three axes (the x-, y-, and z-axes) can be used to describe 3-D space. When you rotate an object in 3-D space, the same principles apply as for 2-D rotations. In fact, you can reexamine the rotation of the shape in Figure 3.12 from a 3-D perspective. Figure 3.13 shows the 2-D shape drawn in 3-D space before and after it was rotated 90 degrees CCW about the pivot point, which is the origin of the xyz-coordinate system.

Observe and understand how each line on the shape is rotated the same amount—90 degrees CCW about the origin—and that the point on the shape originally in contact with the origin remains at the origin after rotation. One other thing you may notice is that the pivot point is the point view of the z-axis. The point view of a line is what you see as you look down the length of the axis. To illustrate this principle, take a pen or pencil and rotate it so you are looking directly at its point; notice that the length of the pen "vanishes" and only the "point" remains visible, as shown in Figure 3.14. As such, the original rotation of the 2-D shape, as shown in Figure 3.12, could be considered a 90-degree CCW rotation about the z-axis in 3-D space.

Think back to what you learned in Chapter 2 about the right-hand rule. If you point the thumb of your right hand in the positive direction of the z-axis and curl your fingers, you will see that the 90-degree CCW rotation mimics the direction that your fingers curl, as illustrated in Figure 3.15. This CCW rotation of the 2-D shape represents a *positive* 90-degree rotation about the z-axis. The CCW rotation is positive because the thumb of your right hand was pointing in the positive direction of the z-axis as the shape was rotated. If you point the thumb of your right hand in the negative direction of the z-axis and the shape is rotated in the direction the fingers of your right hand curl, your fingers indicate a clockwise (CW) rotation of the shape about the z-axis, as shown in Figure 3.16. A CW

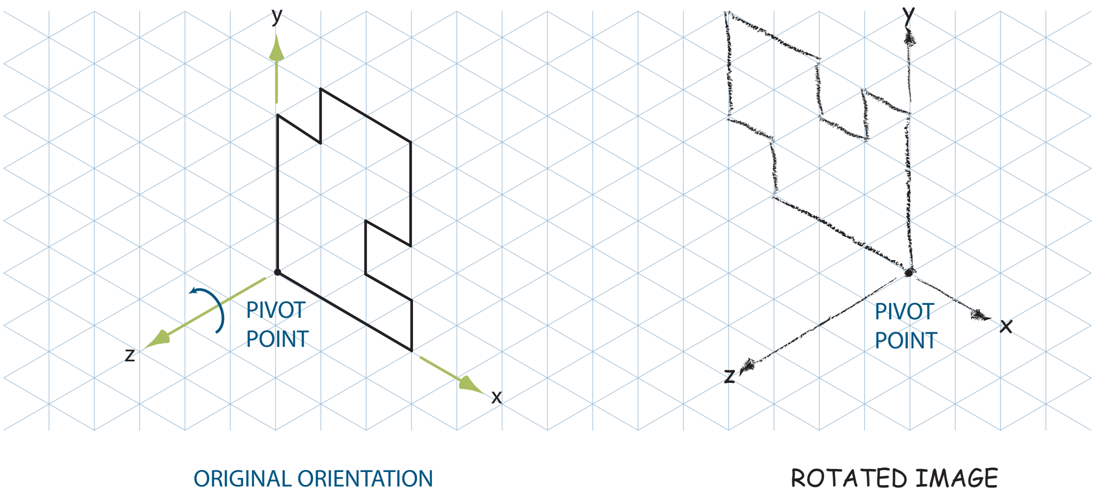


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FIGURE 3.13. A 2-D shape rotated in 3-D space.

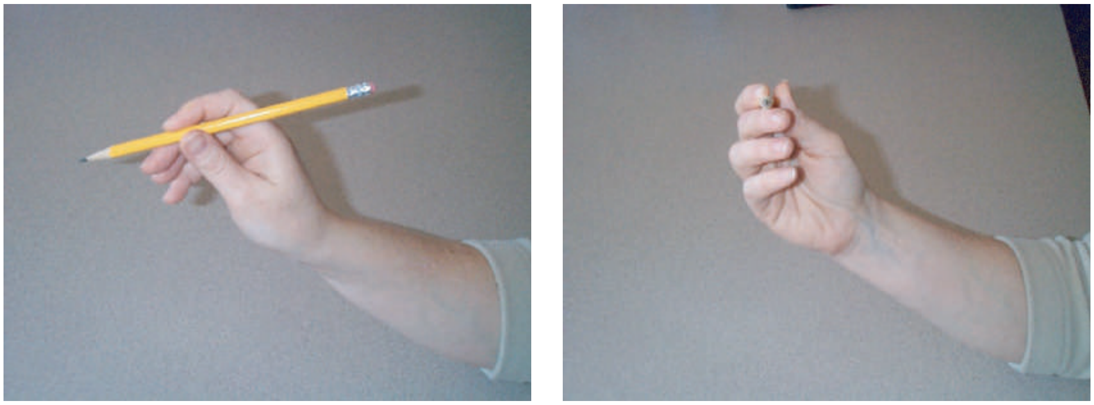


FIGURE 3.14. Looking down the end of a pencil.

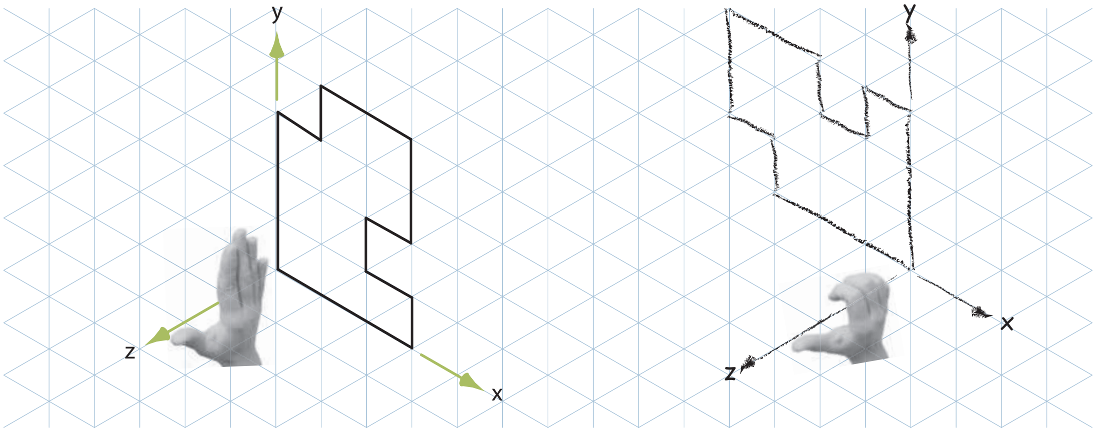


FIGURE 3.15. Positive rotation of a 2-D shape about the z-axis.

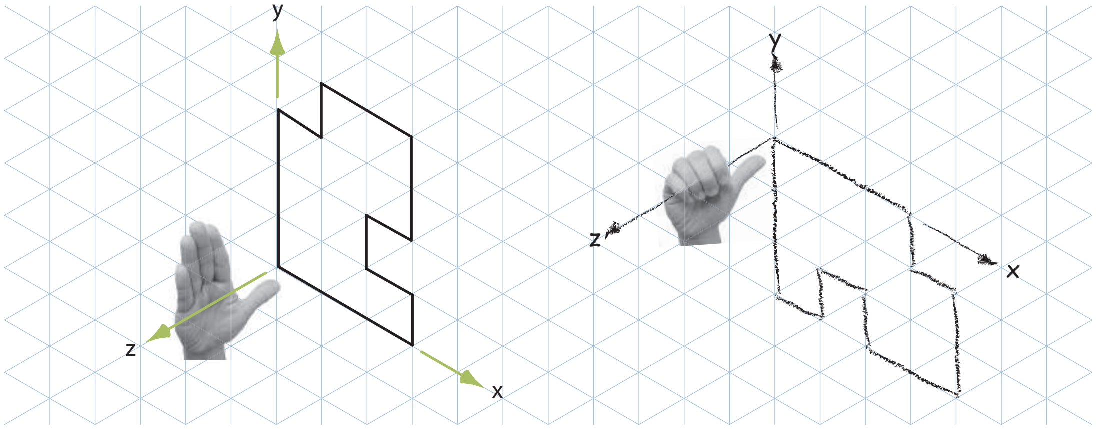


FIGURE 3.16. Negative rotation of a 2-D shape about the z-axis.

rotation about an axis is defined as a negative rotation. Remember that the thumb of your right hand is pointing in the negative z-direction. Also remember that the pivot point of the shape remains at a fixed location in space as it is rotated in the negative z-direction.

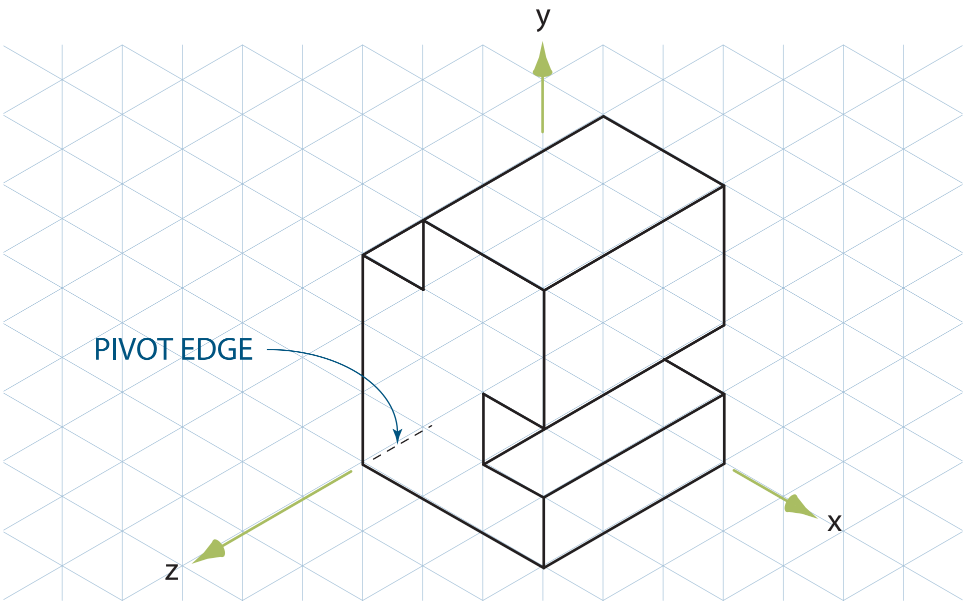
You should now be ready to tackle rotations of 3-D objects in 3-D space. Imagine the 2-D shape from the past several figures is a surface view of a 3-D object. Assume you can extend the surface you have been seeing in the xy-plane into the z-dimension. The result of extending that surface in the third dimension is a solid object. The terminology of 3-D CAD software says that the shape was extruded. You will learn more about extrusion later in this text. If this shape is "extruded" three units into the z-direction, the object will appear as shown in Figure 3.17. In this figure, notice that instead of a single point located on the axis of rotation (the z-axis in the figure), an entire edge of the object is located on that axis. The edge is hidden from sight in this view, but you can imagine it nonetheless. Now think about rotating the entire object about the z-axis in a positive direction (or CCW) 90 degrees from its original position. When this happens, the image shown in Figure 3.18 appears. Instead of a single pivot point, the 3-D rotation has a pivot edge. Throughout the rotation, the edge remained in contact with the axis of rotation. All parts of the object also rotated by the same amount (90 degrees CCW about z) just as all parts of the surface were rotated when you were considering 2-D shapes.

Just as 2-D shapes can be rotated positively (CCW) or negatively (CW) about the z-axis, 3-D objects can be rotated in either direction. Figure 3.19 shows the same object after it has been rotated negative 90 degrees (CW) about the z-axis. This figure also makes clear that the pivot edge of the object, which is now visible in the rotated view, remains in contact with the axis of rotation as the object is rotated.

Any object can be rotated about the x- or y-axis by following the same simple rules established for rotation about the z-axis:

1. The edge of the object originally in contact with the axis of rotation remains in contact after the rotation. This edge is called the pivot edge.
2. Each point, edge, and surface on the object is rotated by exactly the same amount.

FIGURE 3.17. A 2-D shape from Figure 3.12 extruded three units in the z-direction.



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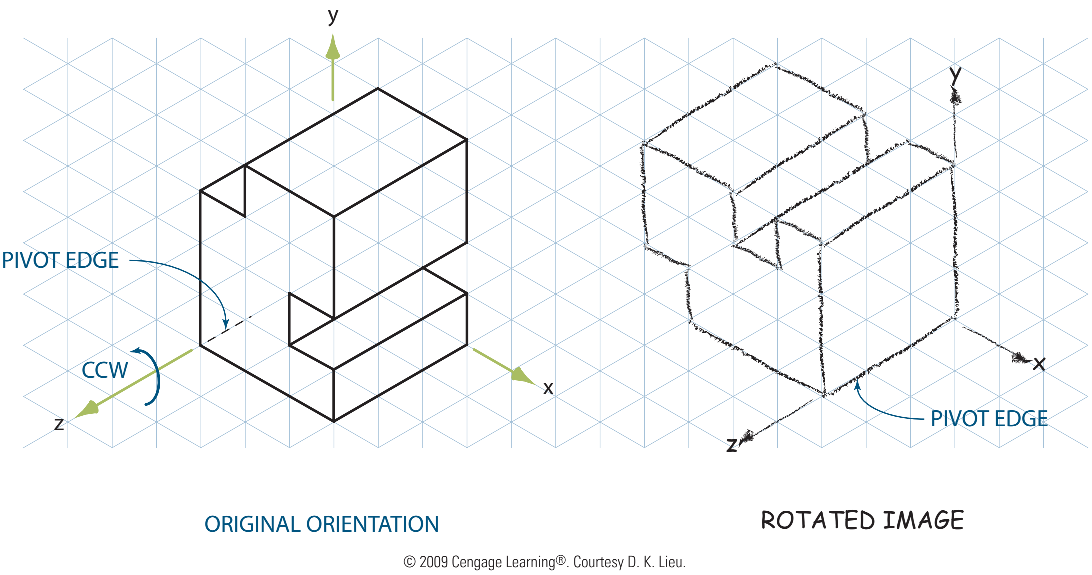

FIGURE 3.18. A 3-D object rotated 90 degrees counterclockwise about the z-axis.

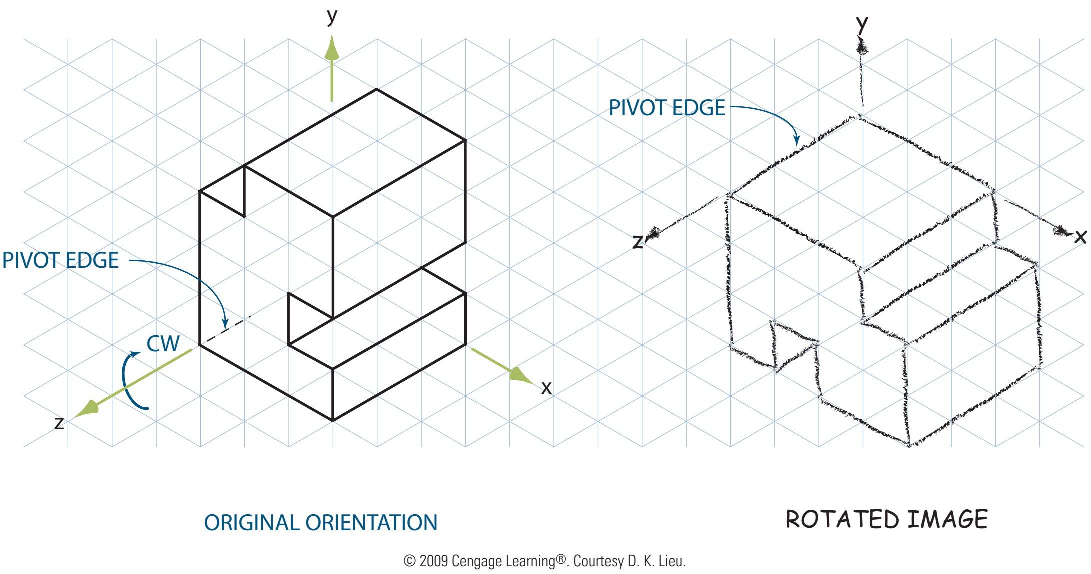

FIGURE 3.19. A 3-D object rotated 90 degrees clockwise about the z-axis.

3. The rotation is positive when it is CCW about an axis and negative when it is CW about an axis. The direction is determined by looking directly down the positive end of the axis of rotation.
4. An alternative method for determining the direction of the rotation is the right-hand rule. Point the thumb of your right hand into the axis of rotation—into either the positive or negative end of the axis of rotation—and curl your fingers in the direction the object is rotated. The direction you obtain from the right-hand rule is the same as the direction defined in number 3 above, positive is CCW and negative is CW.

Figures 3.20 and 3.21 illustrate the positive and negative 90-degree rotations obtained about the x- and y-axis, respectively.

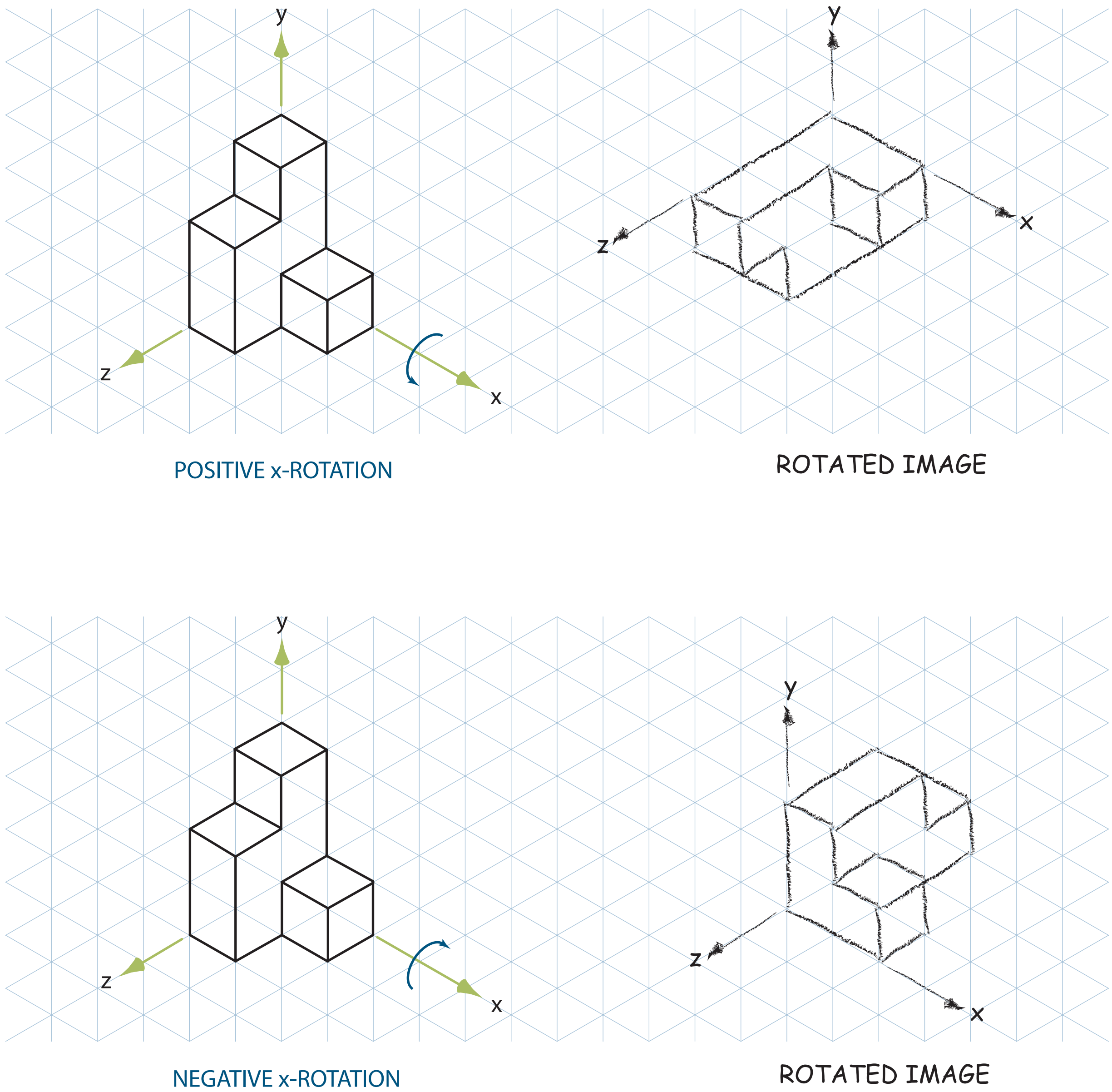

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FIGURE 3.20. Positive and negative rotations about the x-axis.

3.08.01 Notation

Specifying in writing a positive, or CCW, rotation about any axis is cumbersome and time-consuming. For this reason, the following notations will be used to describe object rotations in this text:

- To denote positive rotations of an object about the indicated axis.
- To denote negative rotations of an object about the indicated axis.
- Also, for simplicity in sketching, this text will always rotate an object in increments of 90 degrees about the indicated axis. Figure 3.22 illustrates the result when you rotate the object according to the notation given.

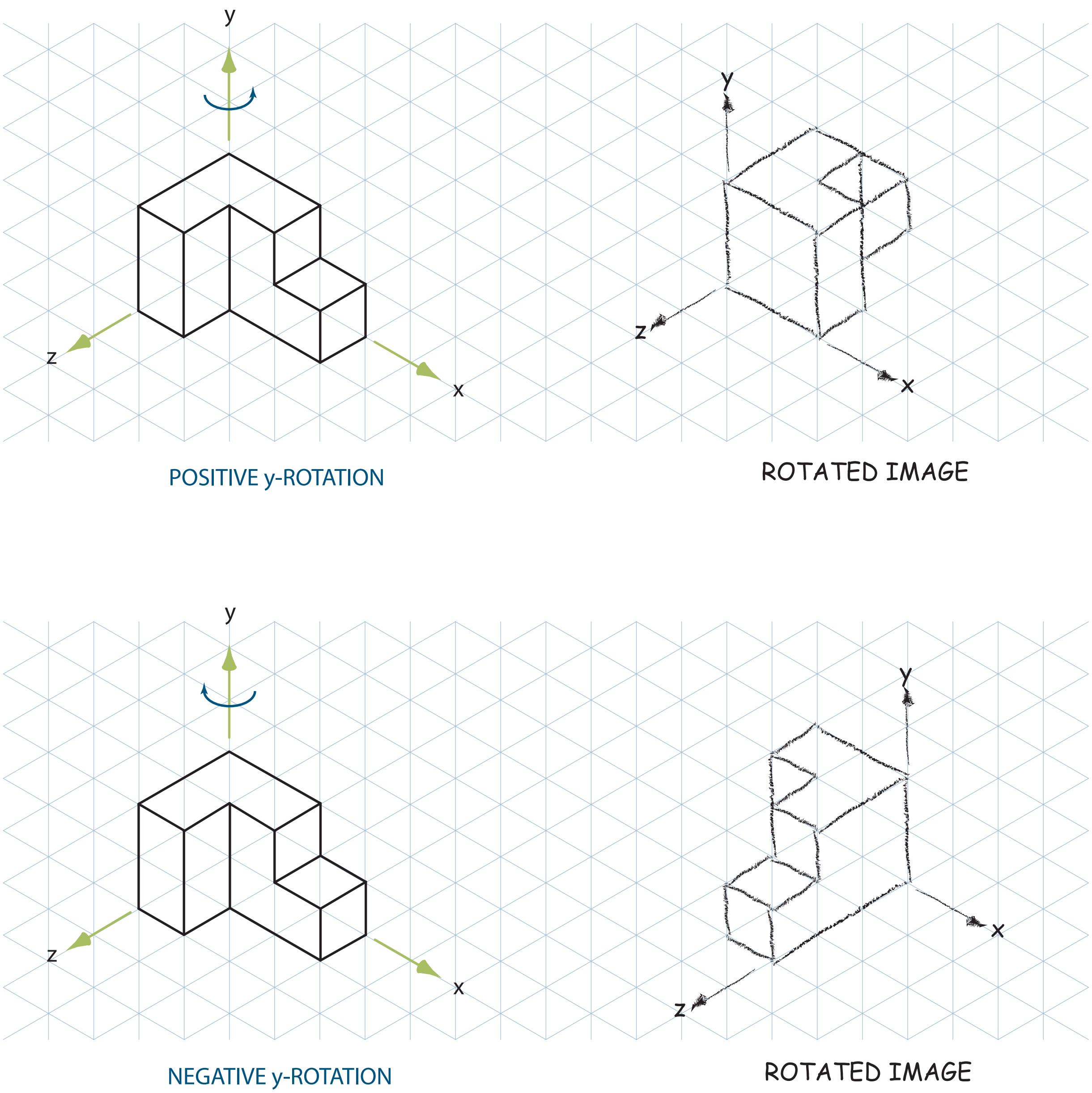

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FIGURE 3.21. Positive and negative rotations about the y-axis.

3.08.02 Rotation of Objects by More Than 90 Degrees about a Single Axis

In all examples and figures in the preceding sections, objects were rotated exactly 90 degrees about a single axis. In reality, you can rotate objects by any number of degrees. If you rotate an object in two increments of 90 degrees about the same axis, the total rotation will be 180 degrees. Similarly, if you rotate an object in three increments, the total rotation will be 270 degrees. Figure 3.23 shows an object that has been rotated 180 degrees about a single axis, along with the symbol denoting the amount and direction of rotation. Notice that the two 90-degree positive x-axis rotations indicate the total 180-degree rotation achieved.

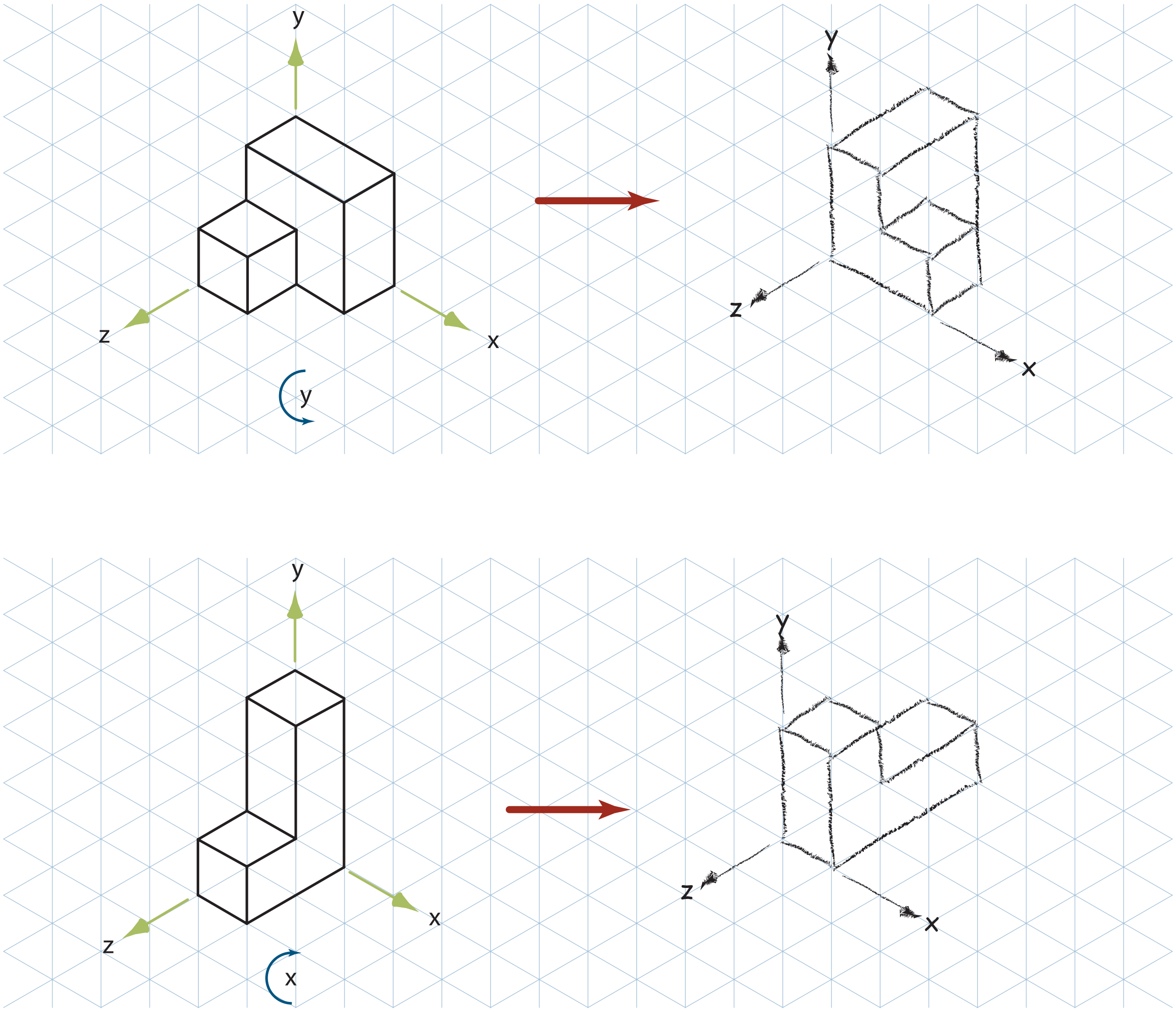

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FIGURE 3.22. Object rotations specified by notation.

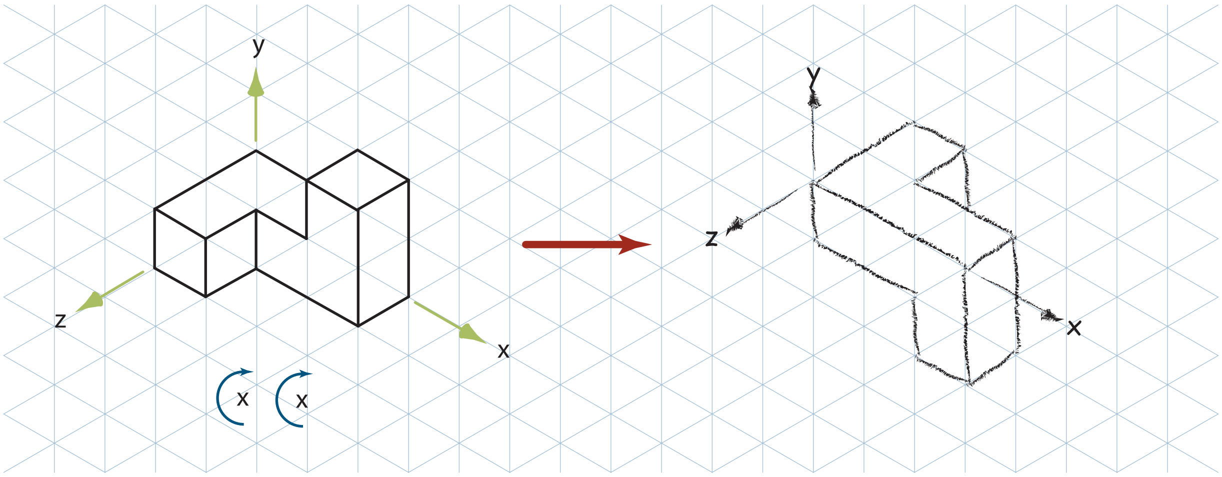

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FIGURE 3.23. An object rotated 180 degrees about an axis.

Once you are free to rotate objects in multiple increments of 90 degrees, you can achieve several equivalent rotations. The term *equivalent rotations* means that two different sets of rotations produce the same result.

3.08.03 Equivalencies for Rotations about a Single Axis

When an object is rotated in multiple increments about an axis, the following equivalencies can be observed:

- A positive 180-degree rotation is equivalent to a negative 180-degree rotation.
- A negative 90-degree rotation is equivalent to a positive 270-degree rotation.
- A positive 90-degree rotation is equivalent to a negative 270-degree rotation.

These equivalencies are illustrated in Figures 3.24, 3.25, and 3.26, respectively.

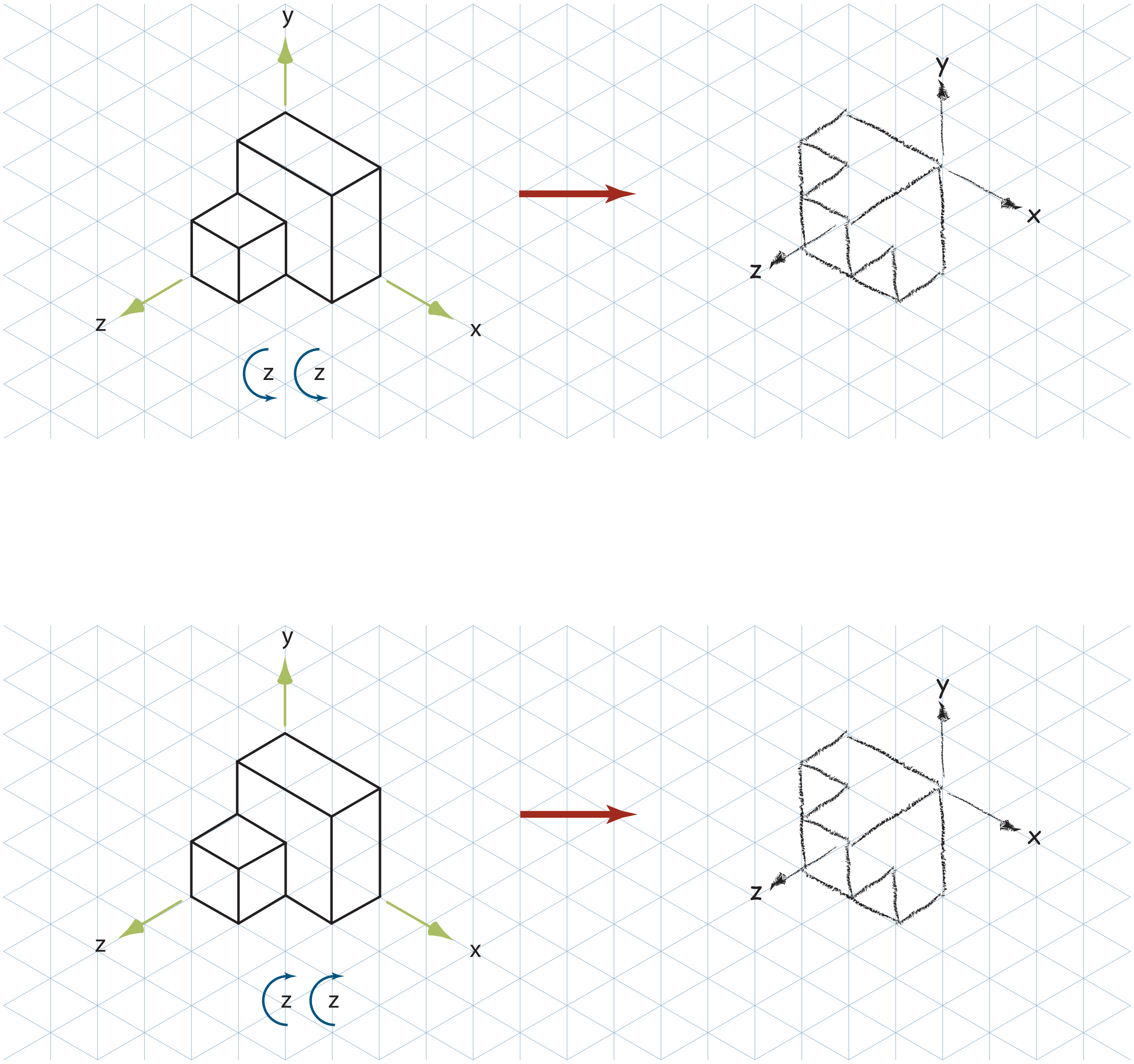

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FIGURE 3.24. A positive 180-degree rotation is equivalent to a negative 180-degree rotation.

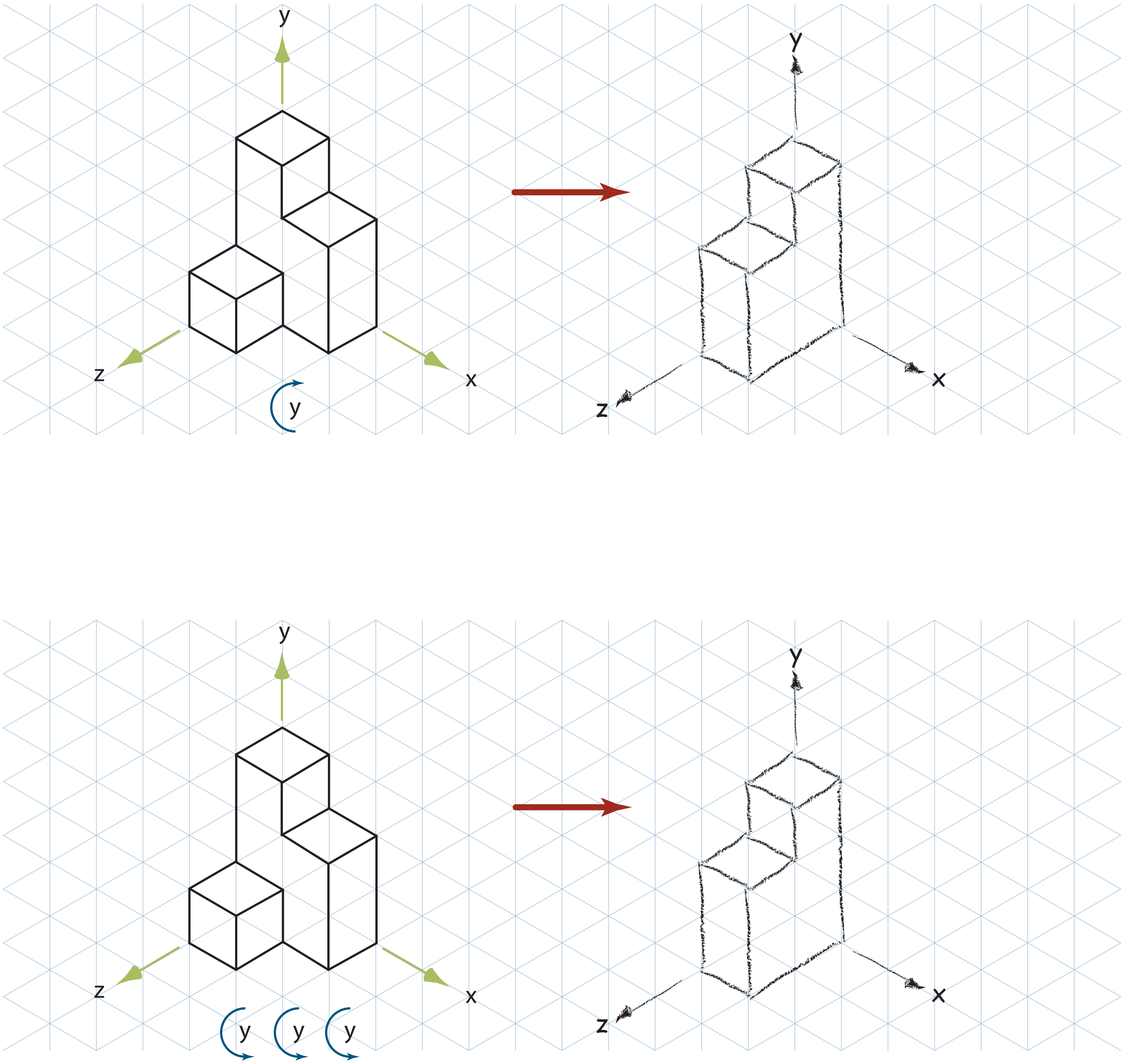

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FIGURE 3.25. A negative 90-degree rotation is equivalent to a positive 270-degree rotation.

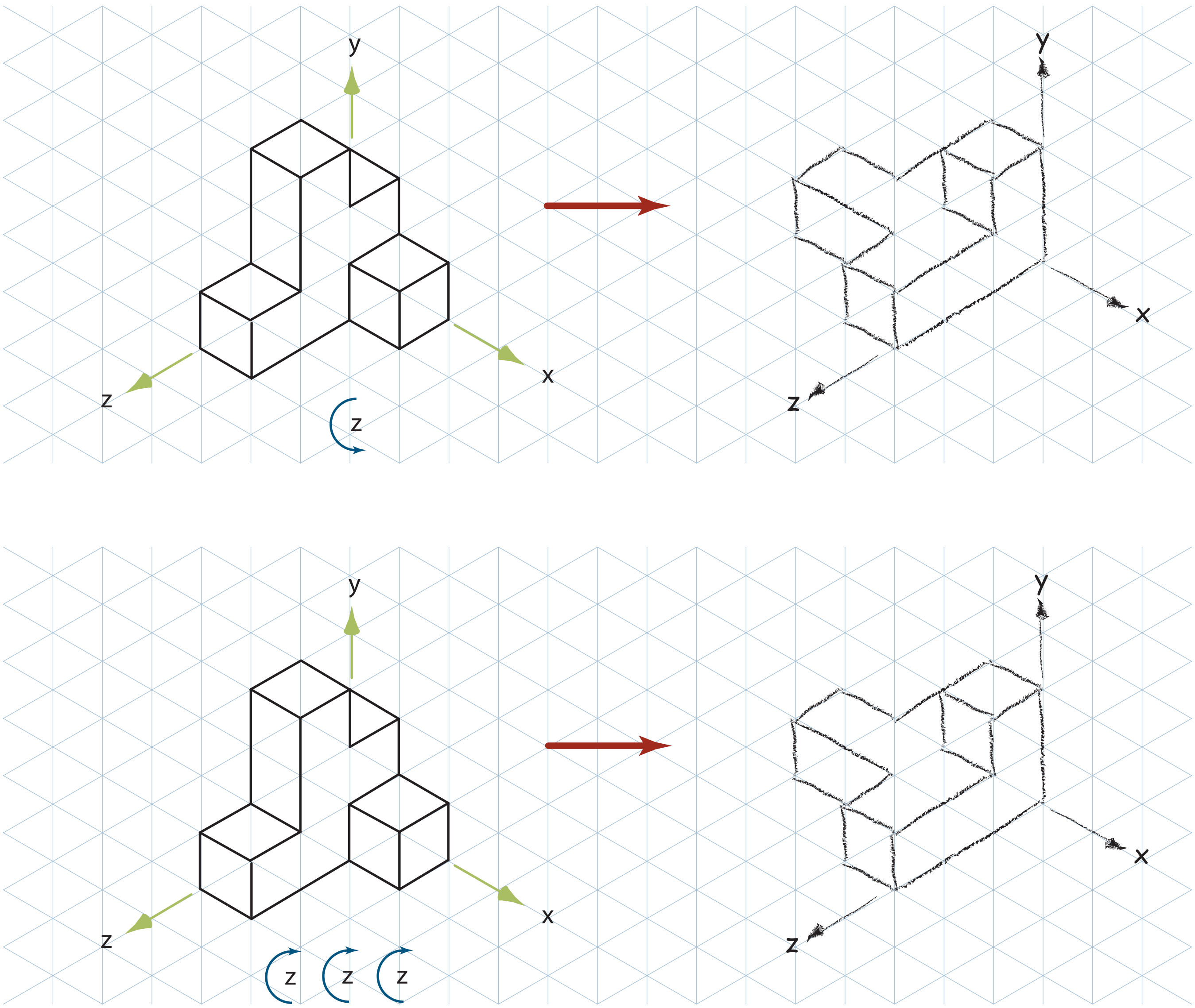

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FIGURE 3.26. A positive 90-degree rotation is equivalent to a negative 270-degree rotation.

3.09 Rotation about Two or More Axes

In the same way you rotated an object about a single axis, you can also rotate the object about more than one axis in a series of steps. Figure 3.27 shows an object that has been rotated in the negative direction about the x-axis and then rotated in the positive direction about the *z*-axis. The rotation notation used in the figure indicates the specified two-step rotation. Figure 3.28 shows the same set of rotations, only this time they are shown in two single steps to achieve the final result. Notice that when an object is rotated about two different axes, a single edge no longer remains in contact with the axis of rotation (since there are now two of them). For rotations about two axes, only a single point remains in its original location, as shown in Figures 3.27 and 3.28.

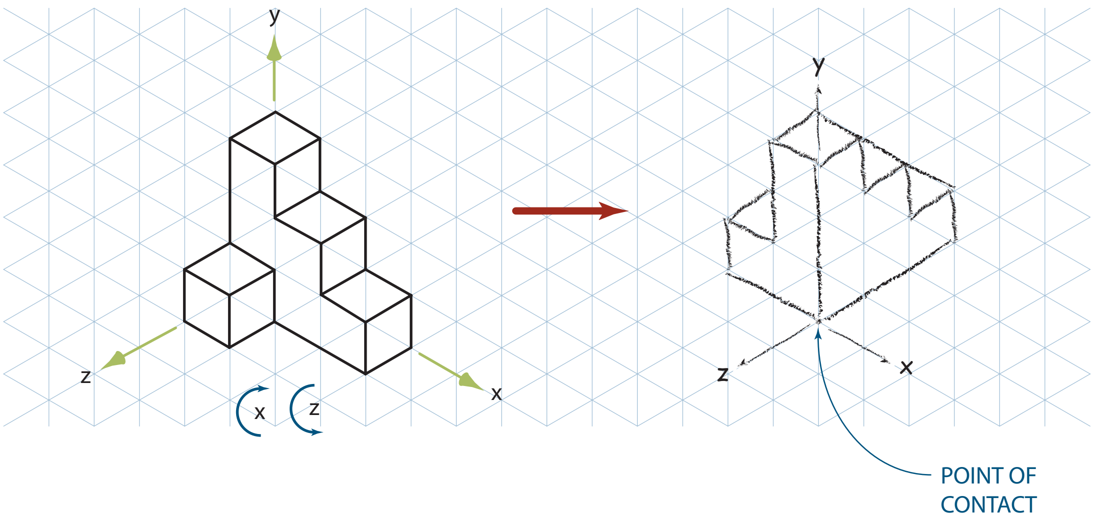

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FIGURE 3.27. An object rotated about two axes.

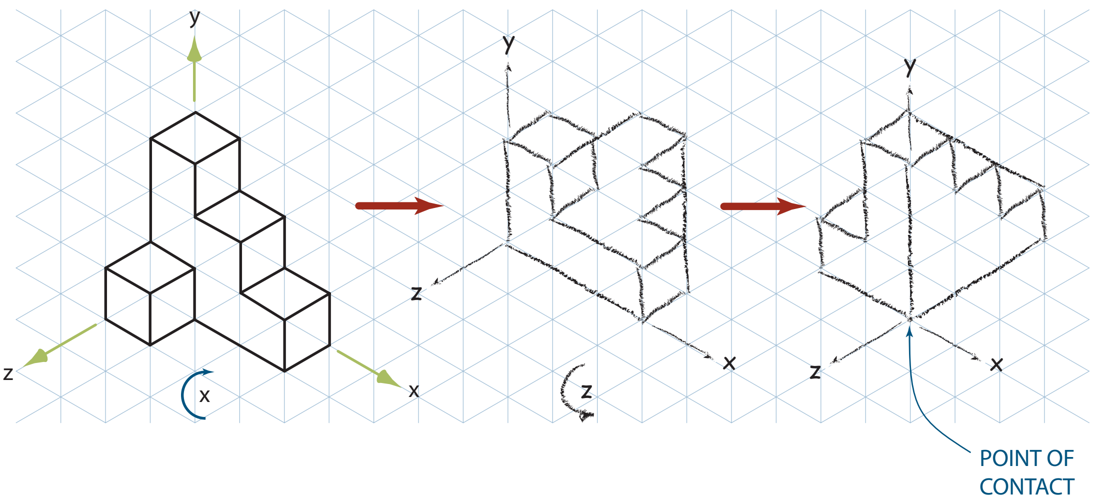

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FIGURE 3.28. An object rotated in two steps.

When rotating an object about two or more axes, you must be careful to perform the rotations in the exact order specified. If the rotations are listed such that you rotate the object CW in the negative direction about the x-axis and then rotate it CCW in the positive direction about the z-axis, you must perform the rotations in that order. Object rotations are not commutative. (Remember that the commutative property in math states that 2 + 3 = 3 + 2.) For object rotations, rotating about the x-axis and then rotating about the y-axis is *not* the same as rotating about the y-axis and then rotating about the x-axis.

In the top portion of Figure 3.29, the object has been rotated about positive y and then rotated about negative z to obtain its image. In the bottom portion of the figure, the object has been rotated about negative z and then rotated about positive y to obtain a new image of the rotated object. The second image is obtained by reversing

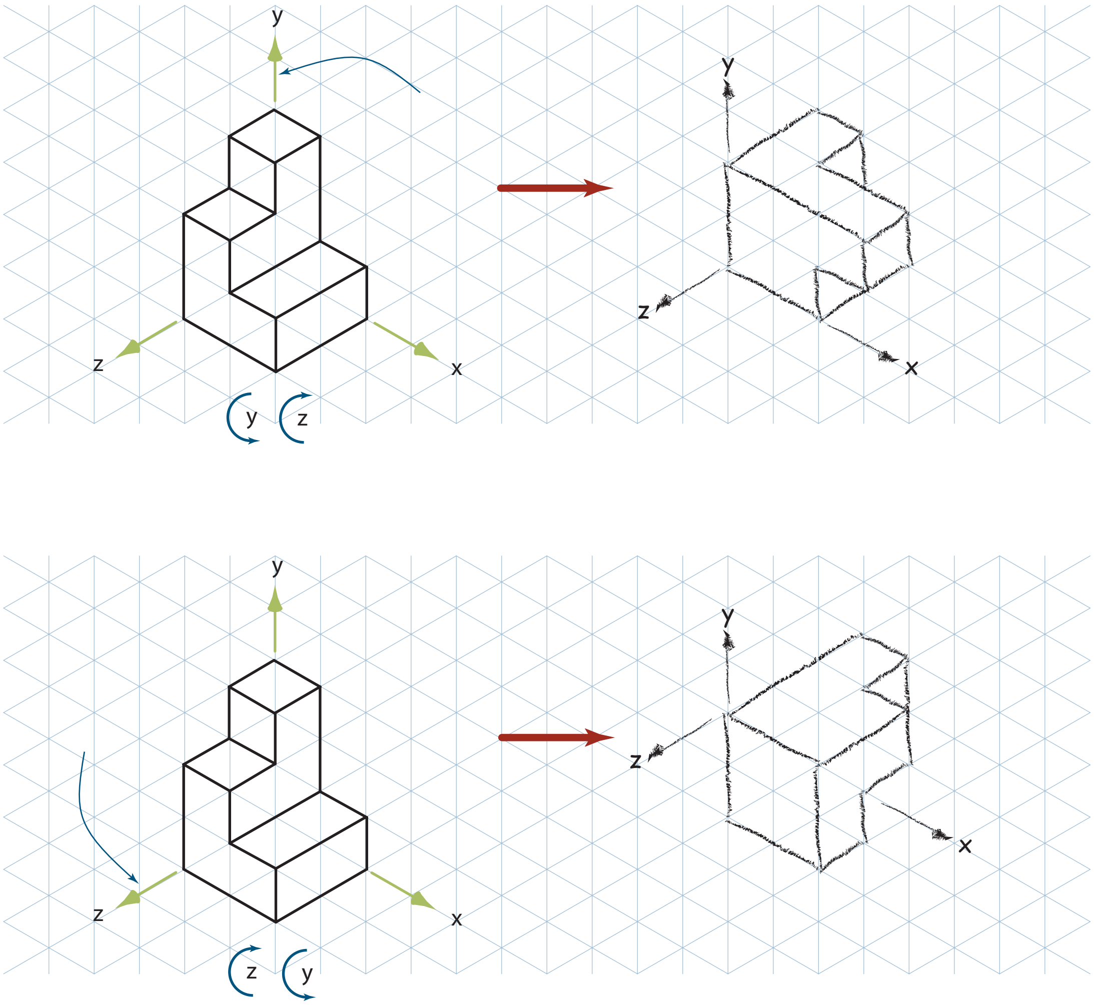

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FIGURE 3.29. Object rotations about two axes—order not commutative.

the order of the rotations. The resulting images are not the same when the order of rotation is changed. Why? Because with the first set of rotations, the edge of the object on the y-axis serves as the pivot line for the first rotation, which is about positive y. For the second set of rotations, the edge of the object on the z-axis serves as the pivot edge for the first of the two rotations. When you rotate first about negative z, you are using an entirely different object edge than the initial pivot line; hence, the difference in rotated images.

3.09.01 Equivalencies for Object Rotations about Two or More Axes

Just as there are equivalencies for rotations of an object about a single axis, there are equivalencies for object rotations about two axes. Figure 3.30 shows one pair of rotational equivalencies. Can you find another set? How about positive x and then negative z? No! Or positive y and then positive z? Yes! There are several possibilities for each pair of rotations. But it is impossible to come up with simple rules for equivalency, as in the previous discussion of equivalent rotations about a single axis. Equivalent rotations for objects about two or more axes are likely to be determined through trial and error and a great deal of practice.

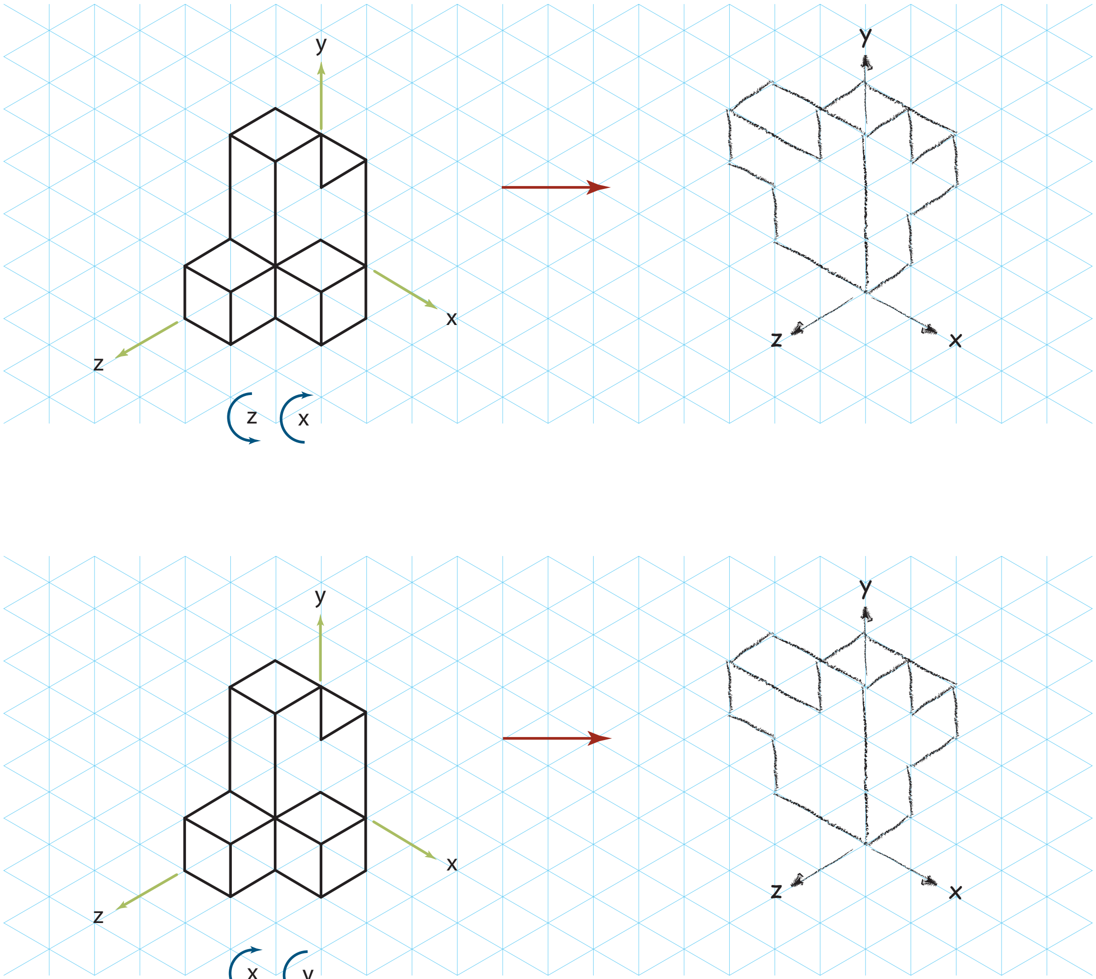

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FIGURE 3.30. Equivalent rotations about two axes.

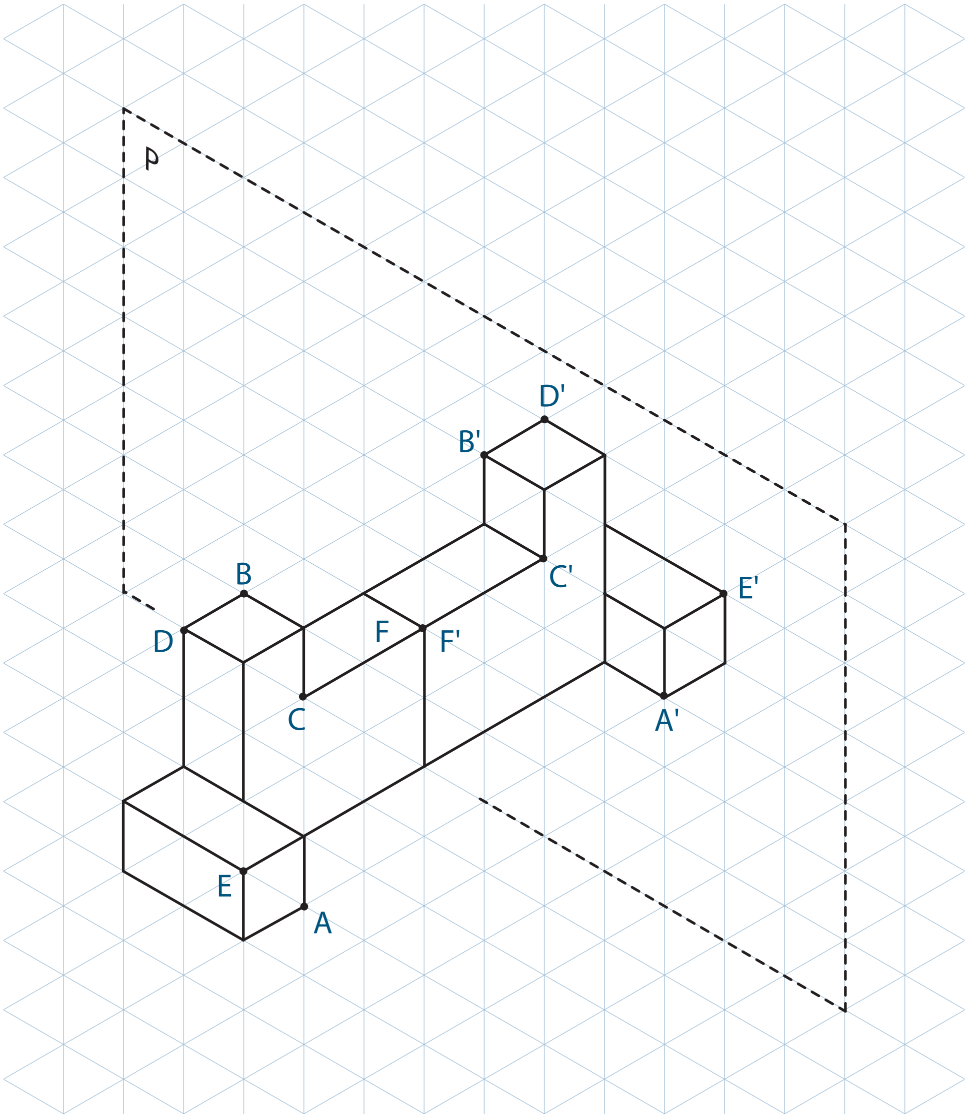
3.10 Reflections and Symmetry

Now that you know the basics of how to visualize an object rotated about an axis, you are ready to move on to visualizing reflections and symmetry. Visualizing planes of symmetry, for example, could save you a great deal of computation time when you are using tools such as Finite Element Analysis or FEA. You will learn more about FEA in later chapters of this text.

You are probably familiar with the concept of **reflections** because you are used to looking at your image reflected back to you from a mirror. With a mirror, you see a reflected 2-D image of your face. If you have a mole on your right cheek, you will see the mole on the right cheek of the reflection. Even though your face is three-dimensional, your face in the mirror is a 2-D reflection—as if your face were projected onto a 2-D plane with your line of sight perpendicular to the plane. You may be able to see somewhat in the third dimension from this mirror plane; however, your depth perception will be a bit off because the image is only two-dimensional. Three-dimensional reflection of objects is different from 2-D reflections with mirrors. For one thing, you reflect a 3-D object *across* the plane so that a 3-D image ends up on the other side of the reflection plane.

Figure 3.31 shows a simple object and its reflection across a reflection plane. Formally stated, in the case of 3-D object reflections, such as shown in Figure 3.31, each point A of the object is associated with an image point A´ in the reflection such that the plane of reflection is a perpendicular bisector of the line segment AA´. What this means is the distance between a point on an object and the reflection plane is equal to the distance between the corresponding point on the image and the reflection plane. The distances are measured along a line perpendicular to the plane of reflection. In this figure, several points on the original object are labeled, as well as their corresponding points on the reflected image. In this case, the plane of reflection coincides with one planar end of the original object; therefore, the corresponding planar end of the reflected image also coincides with the reflection plane. If you measure the distance between point A on the object and the reflection plane, you will find that it is three units. Then if you measure the distance between A´ and the reflection plane, you will find the distance to be three units again. It is also possible to reflect an object across a plane when the object is located some distance from the reflection plane, as illustrated in Figure 3.32.

FIGURE 3.31. An object and its 3-D reflection.



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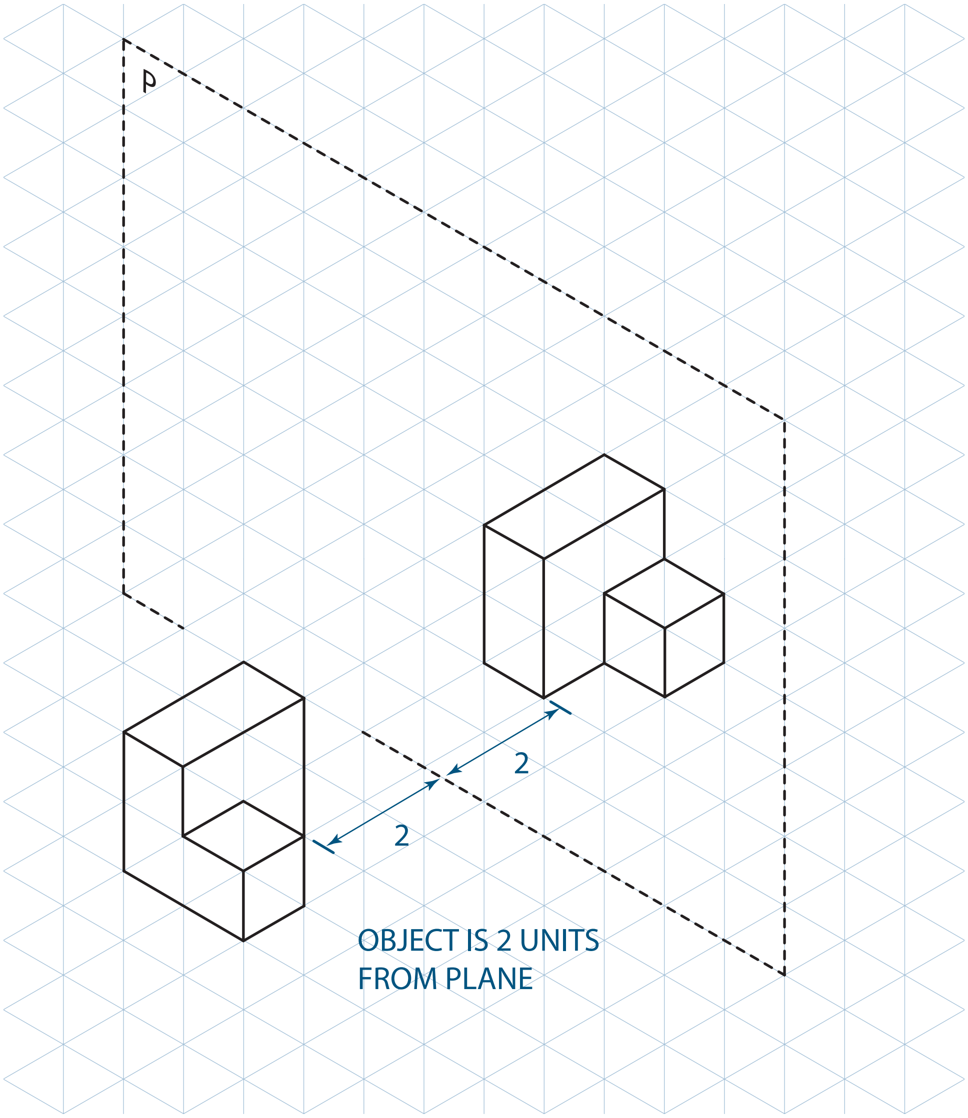

FIGURE 3.32. An object located at a distance from the plane and its reflection.

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3.10.01 Symmetry

Your job as an engineer may be easier if you can recognize planes of symmetry within an object. A plane of **symmetry** is an imaginary plane that cuts through an object such that the two parts, one on either side of the plane, are reflections of each other. Not all objects have inherent symmetry. The human body is roughly symmetrical and has one plane of symmetry—a vertical plane through the tip of the nose and the belly button. The left side is a reflection of the right side. Some objects contain no planes of symmetry, some contain only one plane of symmetry, and still others contain an infinite number of planes of symmetry. Figure 3.33 shows several objects and their planes of symmetry: one object contains no planes of symmetry, one object has just one plane of symmetry, one object has two planes of symmetry, and the last object contains an infinite number of planes of symmetry.

There is one major difference between object reflection and object symmetry. With reflections, you end up with two separate objects (the original and its reflected image); with symmetry, you have a single object that you imagine is being sliced by a plane to form two symmetrical halves. Figure 3.34 illustrates the difference between the two.

FIGURE 3.33. Objects and their planes of symmetry.

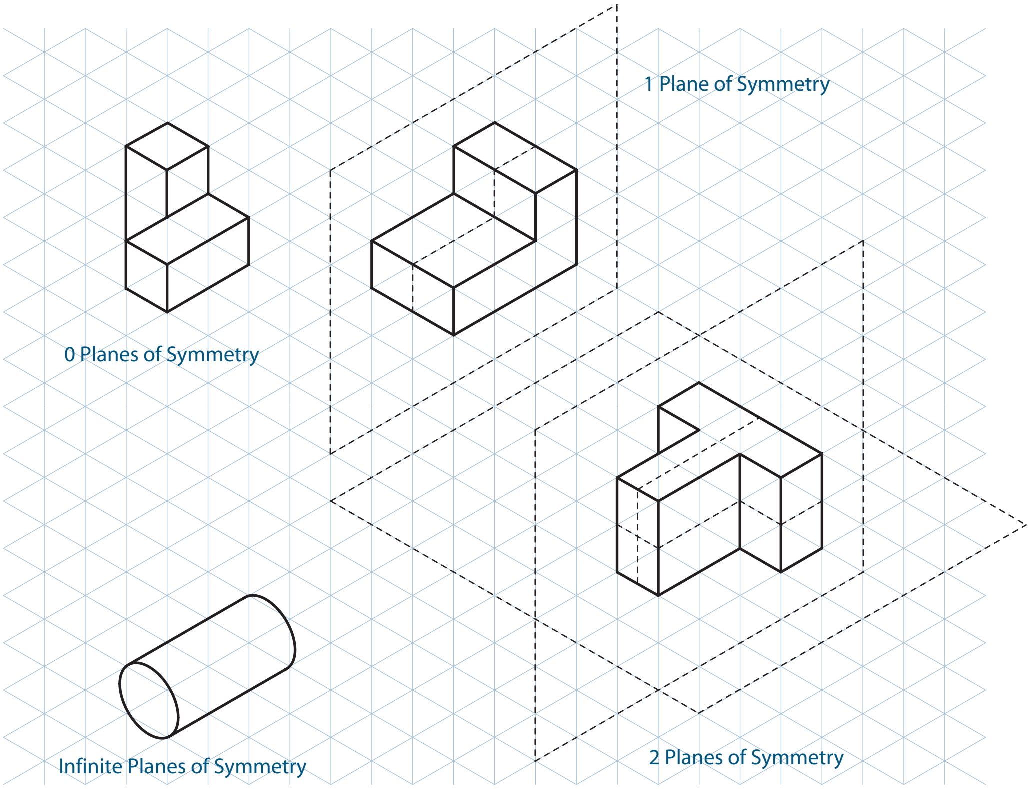

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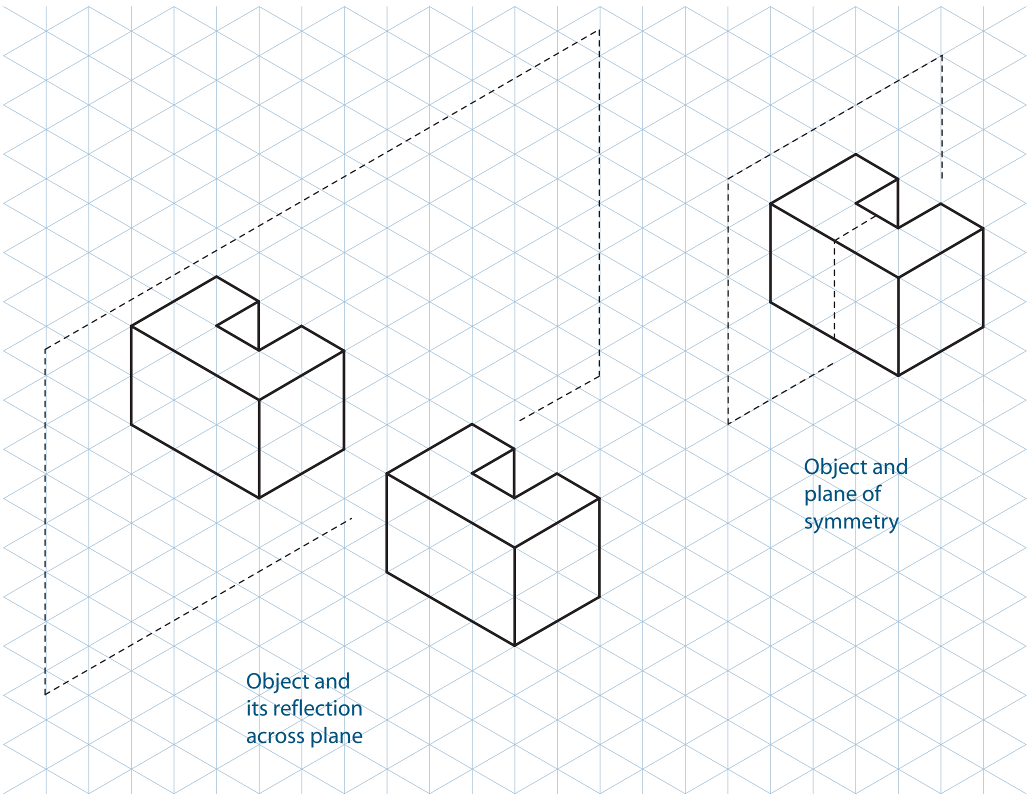
FIGURE 3.34. A comparison of object reflection and symmetry.



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For an object that is symmetrical about a plane, you can sometimes obtain its reflection by rotating the object 180 degrees. To do this, the axis of rotation must be the intersection between the plane of reflection and the plane of symmetry (two planes intersect to form a line). This concept is illustrated in Figure 3.35. Note that a reflection of an object that is not symmetrical cannot be achieved through a simple 180-degree rotation of the object. Hold up your hands in front of you to obtain an object (left hand) and its reflected image (right hand). Note that because your hands have no planes of symmetry, it is impossible to rotate one of them in space to obtain the other one.

3.11 Cross Sections of Solids

Visualizing cross sections enables an engineer to figure out how a building or a mechanical device is put together. Visualizing cross sections enables an electrical engineer to think about how circuit boards stack together within the housing that contains them. Chemical engineers and materials engineers think about the cross sections of molecules and the way those molecules combine with other molecules. Geological engineers and mining engineers visualize cross sections of the earth to determine where veins of rock and ore may be located. Most of the skills described in these examples are at an advanced

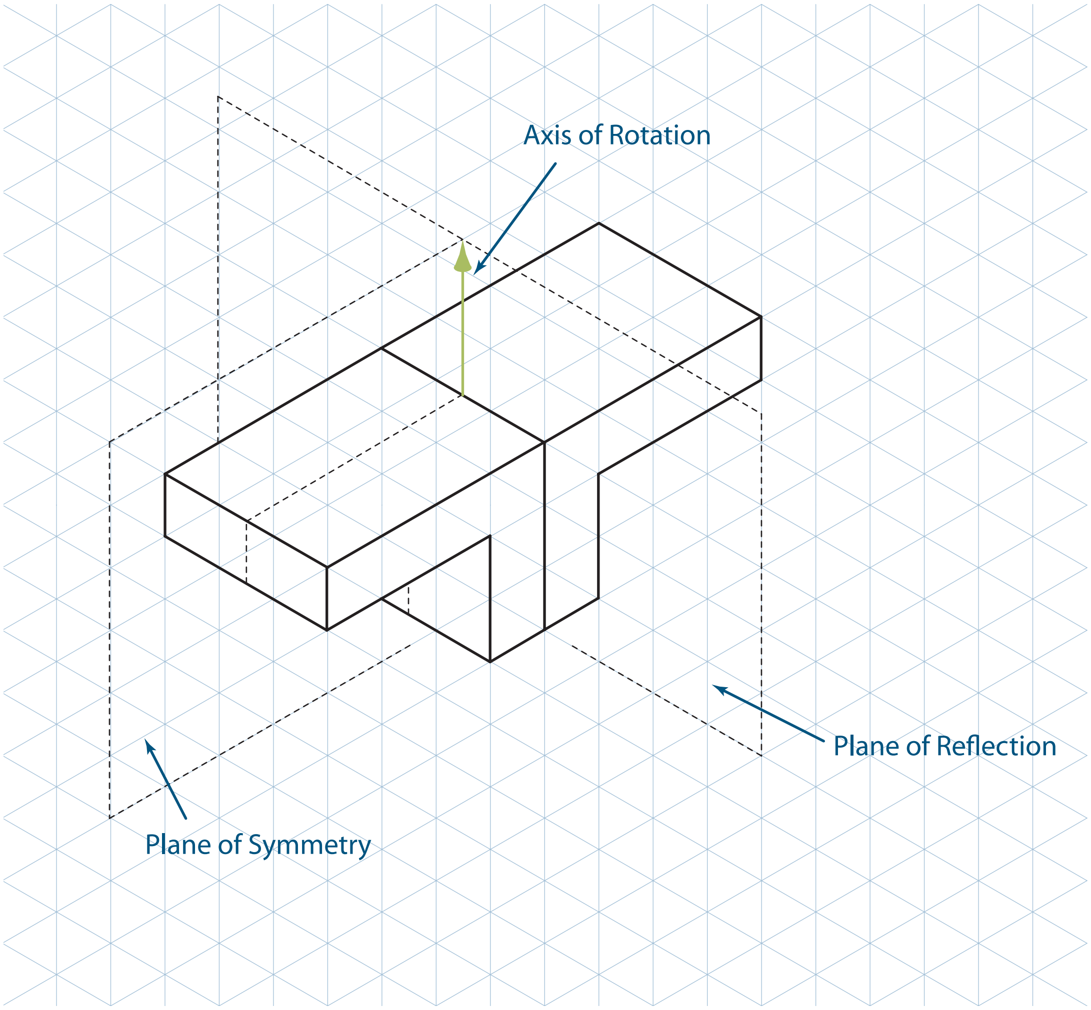

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FIGURE 3.35. Object reflection through rotation.

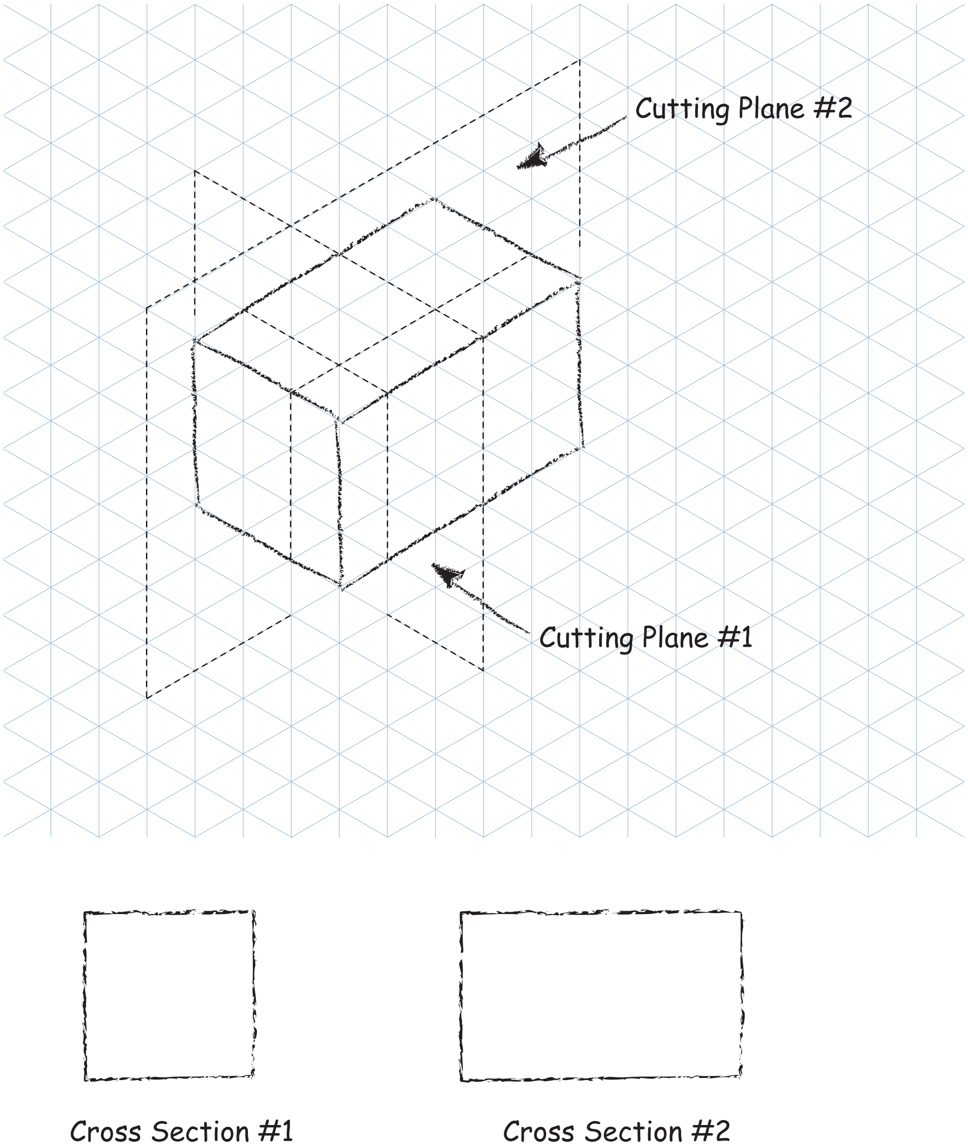
level; in this section, you will learn about cross sections of solids from a fundamental level. Then you can apply the principles to the visualization of more complex parts and systems in later courses and, of course, in your professional work.

Simply stated, a cross section is defined as "the intersection between a solid object and a cutting plane." Because a plane is infinitely thin, the resulting intersection of the two entities is a planar section. The limits of the cross-sectional plane are the edges and the surfaces where the plane cuts through the object. Consider a loaf of bread. Imagine a single slice of infinitely thin bread. One slice of bread would represent the cross section obtained by slicing a vertical plane through the loaf. Because most loaves of bread are not "constant" in shape along their lengths, the cross section changes as you go along the loaf. You know from experience that the cross sections, or slices, on the ends of the loaf are typically smaller than the slices in the middle.

The cross section obtained by intersecting a cutting plane with an object depends on two things: (1) the orientation of the cutting plane with respect to the object and (2) the shape of the original object.

Consider the square prism shown in Figure 3.36. It is cut first by a vertical cutting plane perpendicular to its long axis to obtain the square cross section shown. If the

FIGURE 3.36. Cross sections from a square prism.



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cutting plane is rotated 90 degrees about a vertical axis, the result is the rectangular cross section shown in the figure. The two cross sections are obtained from the same object. The difference in the resulting cross sections is determined by changing the orientation of the cutting plane with respect to the object.

Now consider the cylinder shown in Figure 3.37. If a cutting plane is oriented perpendicular to the axis of the cylinder, a circular cross section results; if the plane is located along the axis of the cylinder, a rectangular cross section is obtained. Observe that this rectangular cross section through the cylinder is identical to the cross section obtained by slicing the rectangular prism along its long axis in Figure 3.36.

Because a resulting cross section through an object depends on the orientation of the cutting plane with respect to the object, most objects may have several cross sections associated with them. Figure 3.38 shows a cylinder with four possible cross sections. Can you imagine the orientation of the cutting plane with respect to the cylinder for each cross section?

FIGURE 3.37. Cross sections of a cylinder.

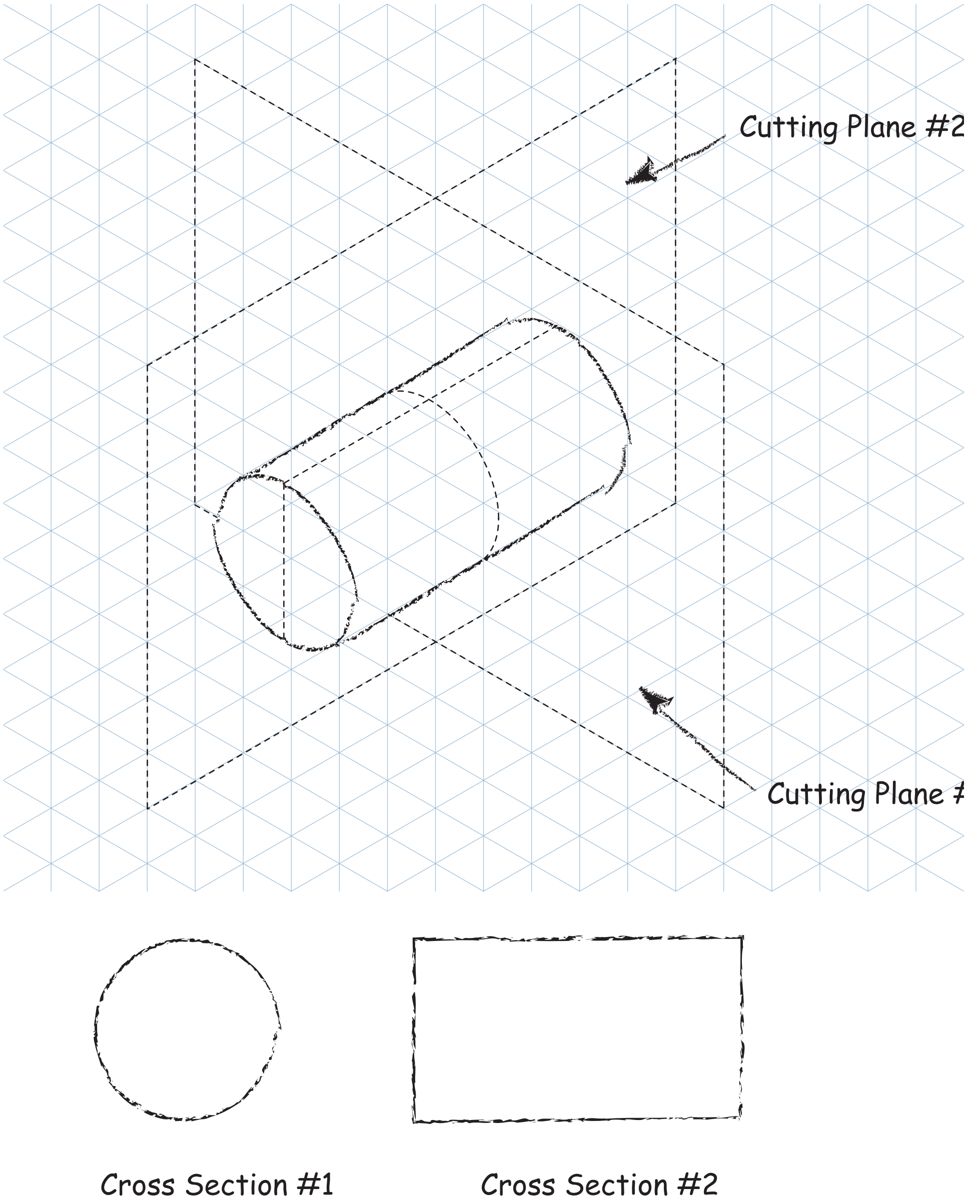

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FIGURE 3.38. Various cross sections of a cylinder.

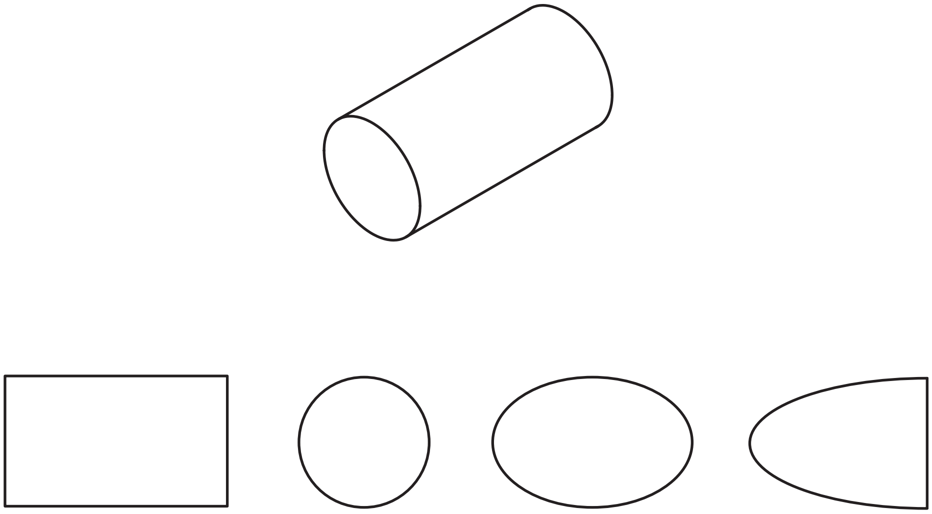

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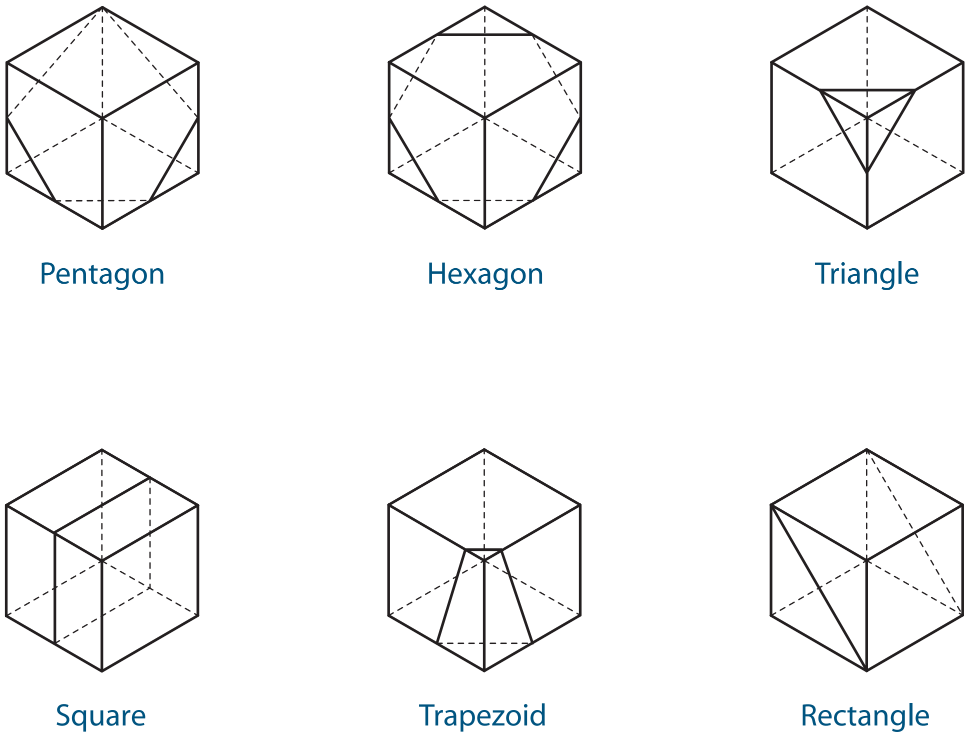
FIGURE 3.39. Various cross sections of a cube.



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You already know that the first two cross sections, rectangle and circle, were obtained by orienting the cutting plane perpendicular to and along the long axis of the cylinder, respectively.

What about the third cross section? It was obtained by orienting the cutting plane at an angle with respect to the axis of the cylinder.

The fourth cross section was also obtained by angling the cutting plane with respect to the cylinder axis, but the angle was such that a portion of the cutting plane went through the flat circular end surface of the cylinder.

Figure 3.39 shows several cross sections obtained by slicing a cube with cutting planes at different orientations.

3.12 Combining Solids

Another skill that will be helpful to you as an engineer or technologist is the ability to visualize how two solids combine to form a third solid. The ability to visualize **combining solids** will be helpful as you learn how to use solid modeling software. In early versions of 3-D CAD software, commands used to combine solids were sometimes known as **Boolean operations**. This terminology was borrowed from mathematics set theory operations, called Booleans, where basic operations include unions, intersections, and complements between sets of numbers. Boolean logic is now the foundation of many modern innovations. In fact, if you have performed a search on the Web using an AND or an OR operator, you have used Boolean logic to help you narrow or expand your search. In terms of 3-D CAD, the Boolean set operations typically correspond to software commands of Join, Intersect, and Cut. To help you become familiar with the terminology since you probably will be building 3-D computer models, this section will use the same terminology.

Two overlapping objects can be combined to form a third object with characteristics of each original object apparent in the final result. To perform any Cut, Join, or Intersect operation to combine objects, the objects must be overlapping initially. What is meant by overlapping is that they share a common volume in 3-D space—called the **volume of interference**. Figure 3.40a shows two objects that overlap; Figure 3.40b shows the volume of interference between the two objects. Notice that the volume of interference takes shape and size characteristics from each of the two initial objects.

When two objects are **joined**, the volume of interference is absorbed into the combined object. The result is a single object that does not have "double" volume in the region of interference. The Boolean Join operation is illustrated in Figure 3.41.

When two objects are combined by **intersecting**, the combined object that results from the intersection is the volume of interference between them, as shown in Figure 3.42.

In the **cutting** of two objects, the combined object that results from the cutting depends on which object serves as the cutting tool and which object is cut by the other object. The result of a cutting operation is that the volume of interference is removed from the object that is cut, as illustrated in Figure 3.43.

FIGURE 3.40. Overlapping objects and volume of interference.

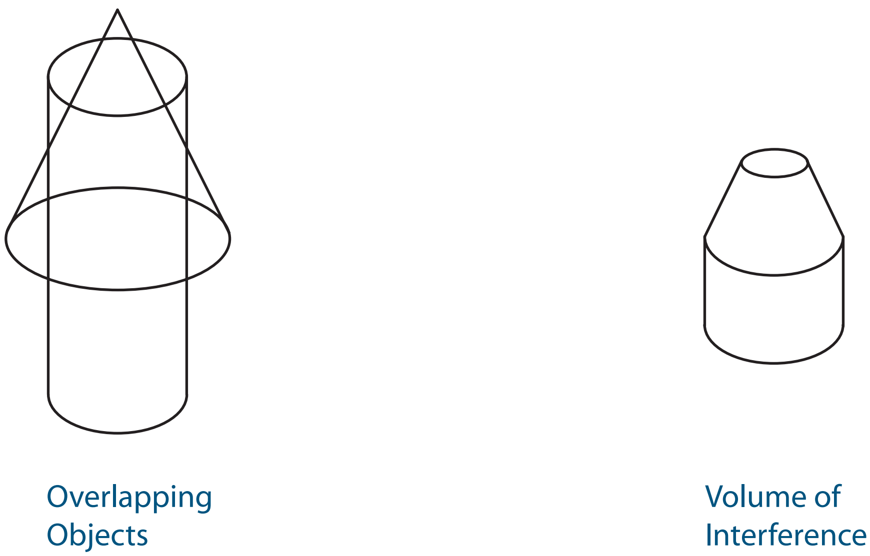

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FIGURE 3.41. Result of two objects joined.

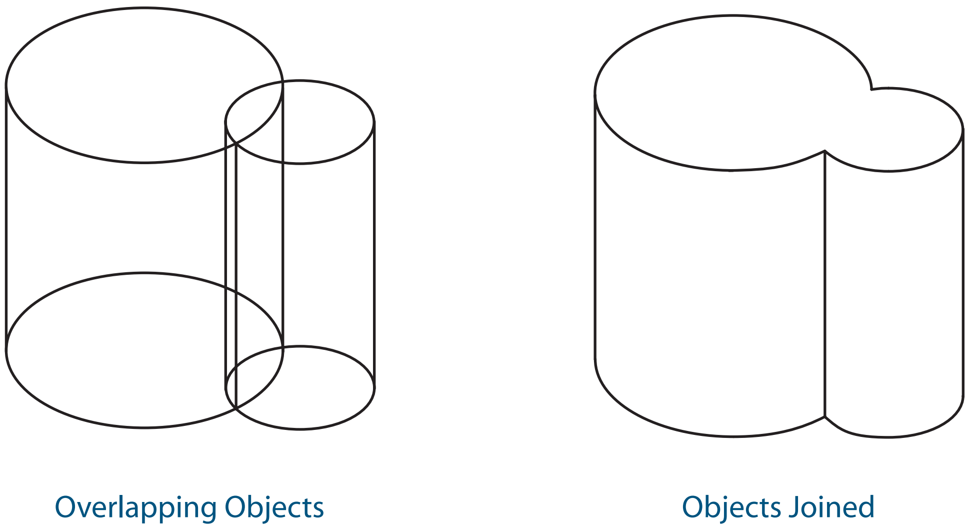


FIGURE 3.42. Result of two objects intersected.

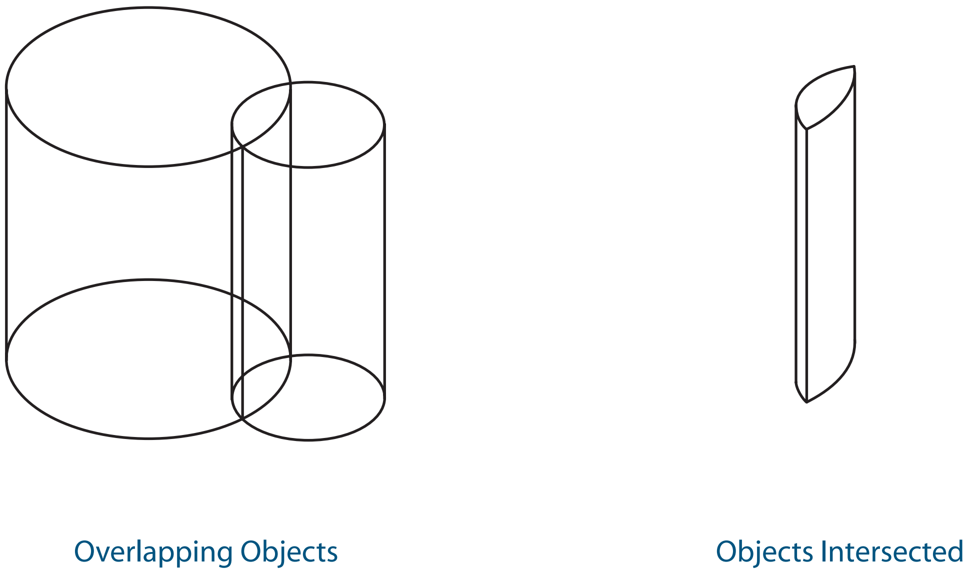
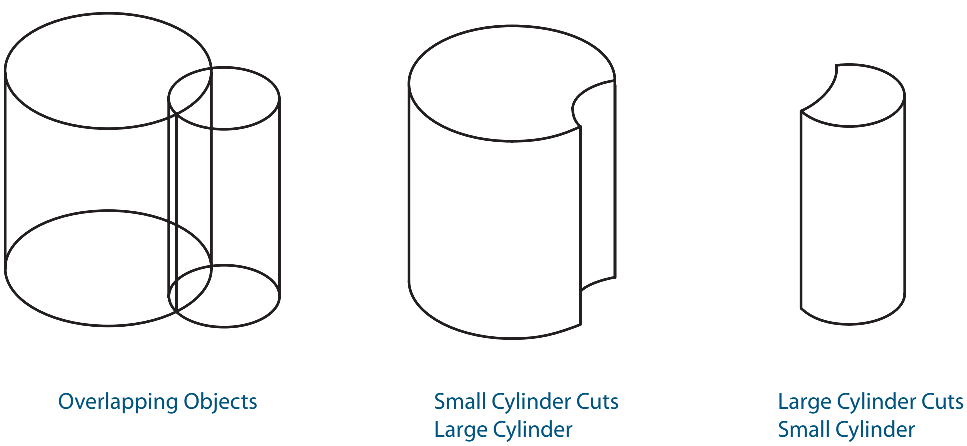


FIGURE 3.43. Result of two objects cutting.



3.13 Chapter Summary

In this chapter, you learned about Gardner's definitions of basic human intelligences (including spatial intelligence) and the way spatial intelligence is developed and assessed. Spatial intelligence is important for engineering success, especially in engineering graphics and solid modeling courses. The chapter outlined several exercises that help develop spatial skills, including:

- Constructing isometric sketches from different corner views.
- Rotating 3-D objects about one or more axes.
- Reflecting objects across a plane and recognizing planes of symmetry.
- Defining cross sections obtained between cutting planes and objects.
- Combining two objects to form a third object by cutting, joining, or intersecting.

3.14 GLOSSARY OF KEY TERMS

Boolean operations: In early versions of 3-D CAD software, commands used to combine solids.

combining solids: The process of cutting, joining, or intersecting two objects to form a third object.

corner views: An isometric view of an object created from the perspective at a given corner of the object.

cross section: The intersection between a cutting plane and a 3-D object.

cut: To remove the volume of interference between two objects from one of the objects.

cutting plane: An imaginary plane that intersects with an object to form a cross section.

intersect: To create a new object that consists of the volume of interference between two objects.

join: To absorb the volume of interference between two objects to form a third object.

mental rotations: The ability to mentally turn an object in space.

reflection: The process of obtaining a mirror image of an object from a plane of reflection.

spatial orientation: The ability of a person to mentally determine his own location and orientation within a given environment.

spatial perception: The ability to identify horizontal and vertical directions.

spatial relations: The ability to visualize the relationship between two objects in space, i.e., overlapping or nonoverlapping.

spatial visualization: The ability to mentally transform (rotate, translate, or mirror) or to mentally alter (twist, fold, or invert) 2-D figures and/or 3-D objects.

symmetry: The characteristic of an object in which one half of the object is a mirror image of the other half.

volume of interference: The volume that is common between two overlapping objects.

3.15 QUESTIONS FOR REVIEW

1. What are some of the basic human intelligences as defined by Gardner?
2. What are the stages of development for spatial intelligence?
3. What are some of the basic spatial skill types?
4. What do the numbers on a coded plan represent?
5. What are some general rules to follow when creating isometric sketches from coded plans?
6. When a person is looking down a coordinate axis, are positive rotations CW or CCW?
7. Describe the right-hand rule in your own words.
8. Are object rotations about two or more axes commutative? Why or why not?
9. What is one difference between object reflection and object symmetry?
10. Are all objects symmetrical about at least one plane? Explain.

11. The shape of a cross section depends on two things. Name them.
12. What is the effect on the resulting cross section of a cutting plane that is tilted?
13. What are the three basic ways to combine solids?
14. In the cutting of two objects, does it matter which object is doing the cutting?

3.16 PROBLEMS

1. For the following objects, sketch a coded plan, labeling the corner marked with an *x* properly.
2. Indicate the coded plan corner view that corresponds to the isometric sketch provided.

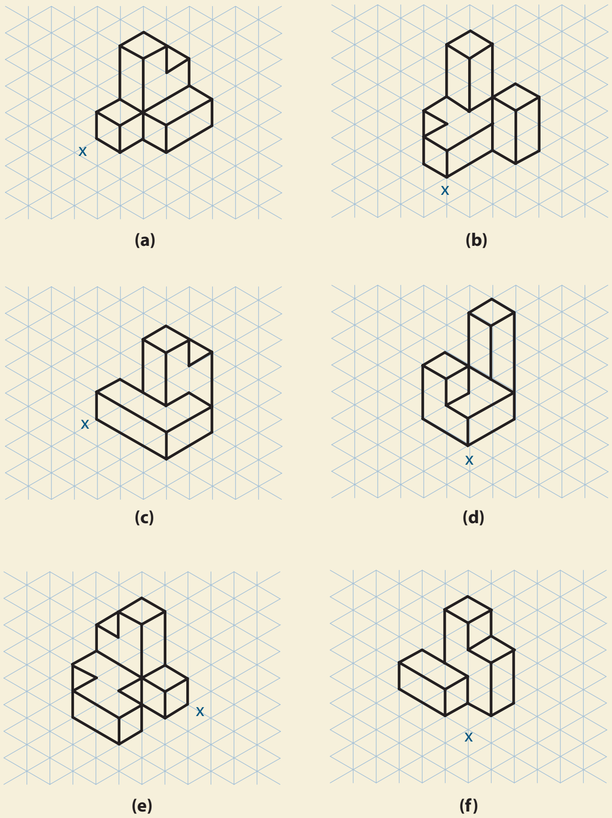


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FIGURE P3.1.

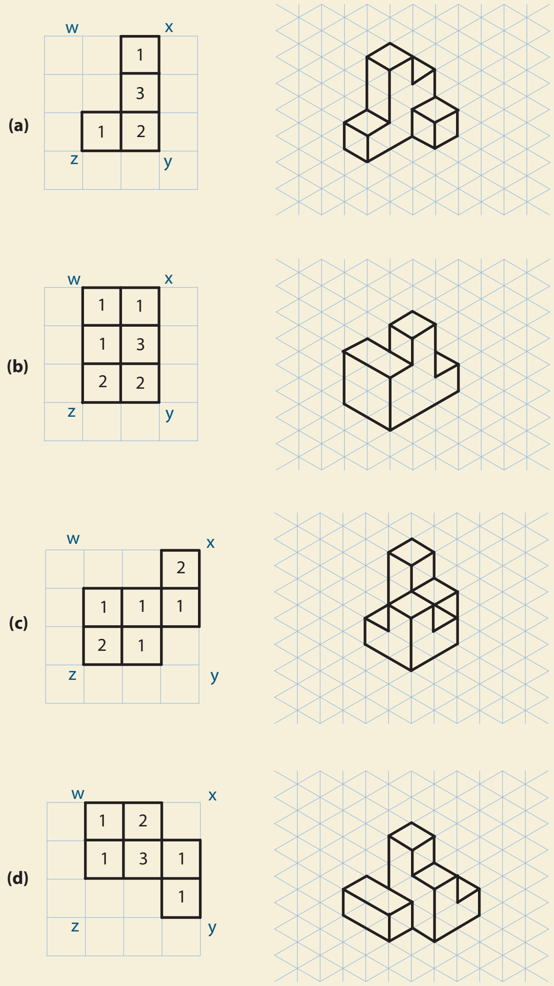

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FIGURE P3.2.

3.16 PROBLEMS (CONTINUED)

3. Use isometric grid paper to sketch the indicated corner view (marked with an *x*) for the coded plan.

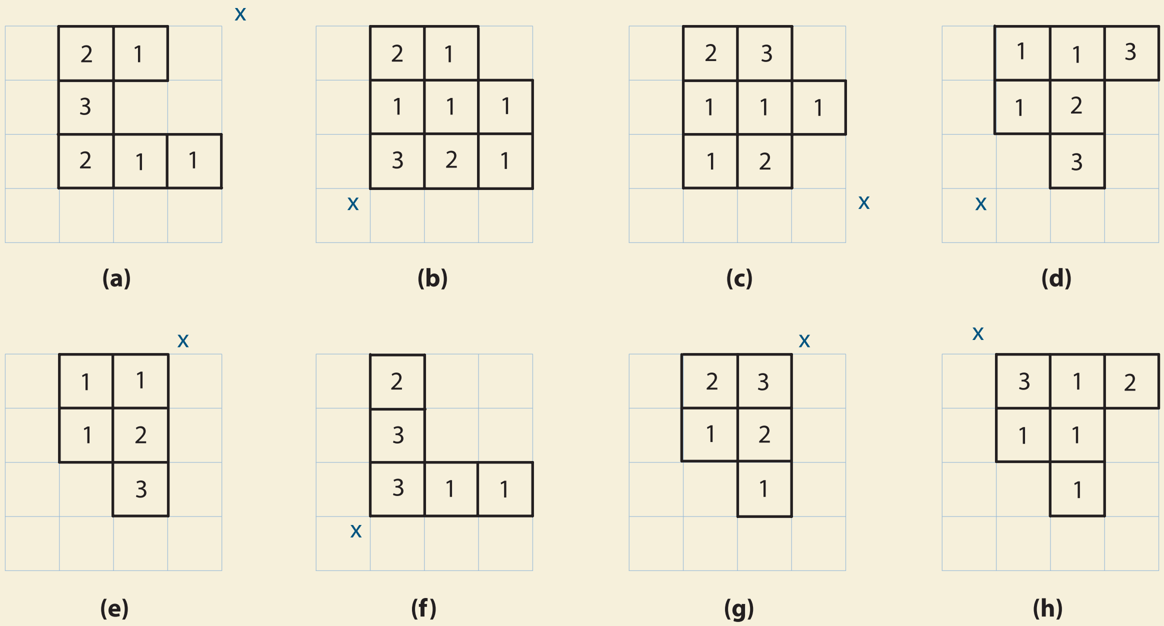

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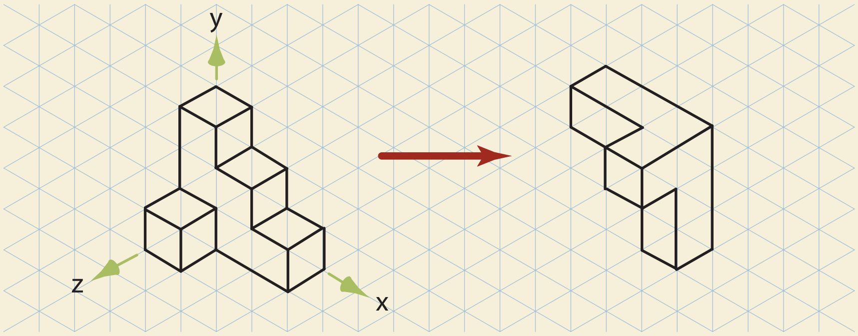
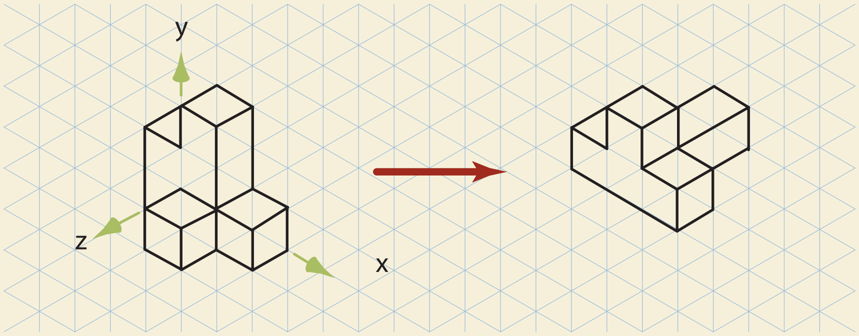
FIGURE P3.3.

3.16 PROBLEMS (CONTINUED)

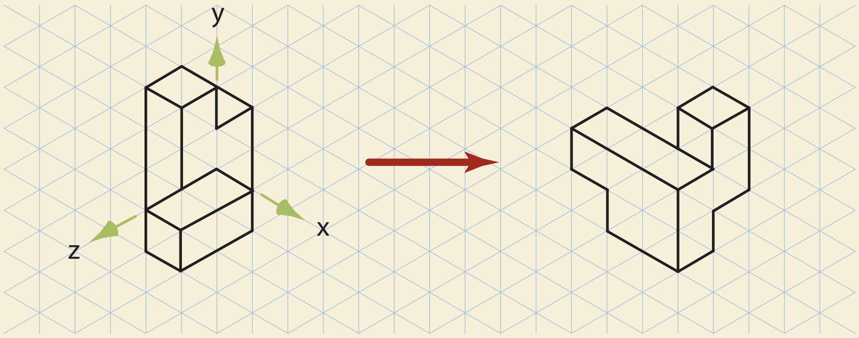
4. Using the notation developed in this chapter, indicate the rotation the following objects have experienced.



(a)



(b)



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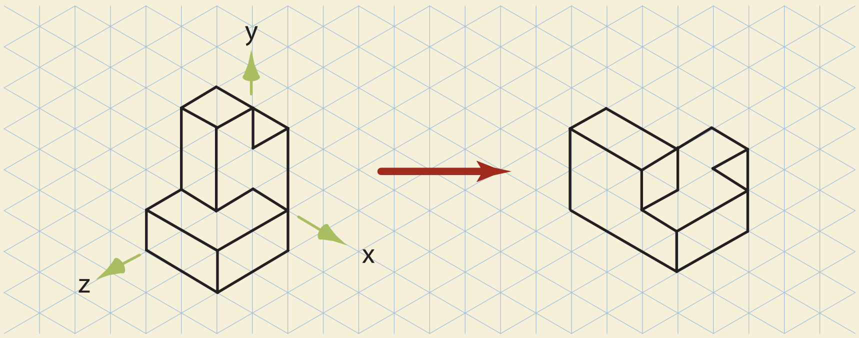

(d)

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FIGURE P3.4.

3.16 PROBLEMS (CONTINUED)

5. Rotate the following objects by the indicated amount and sketch the results on isometric grid paper.

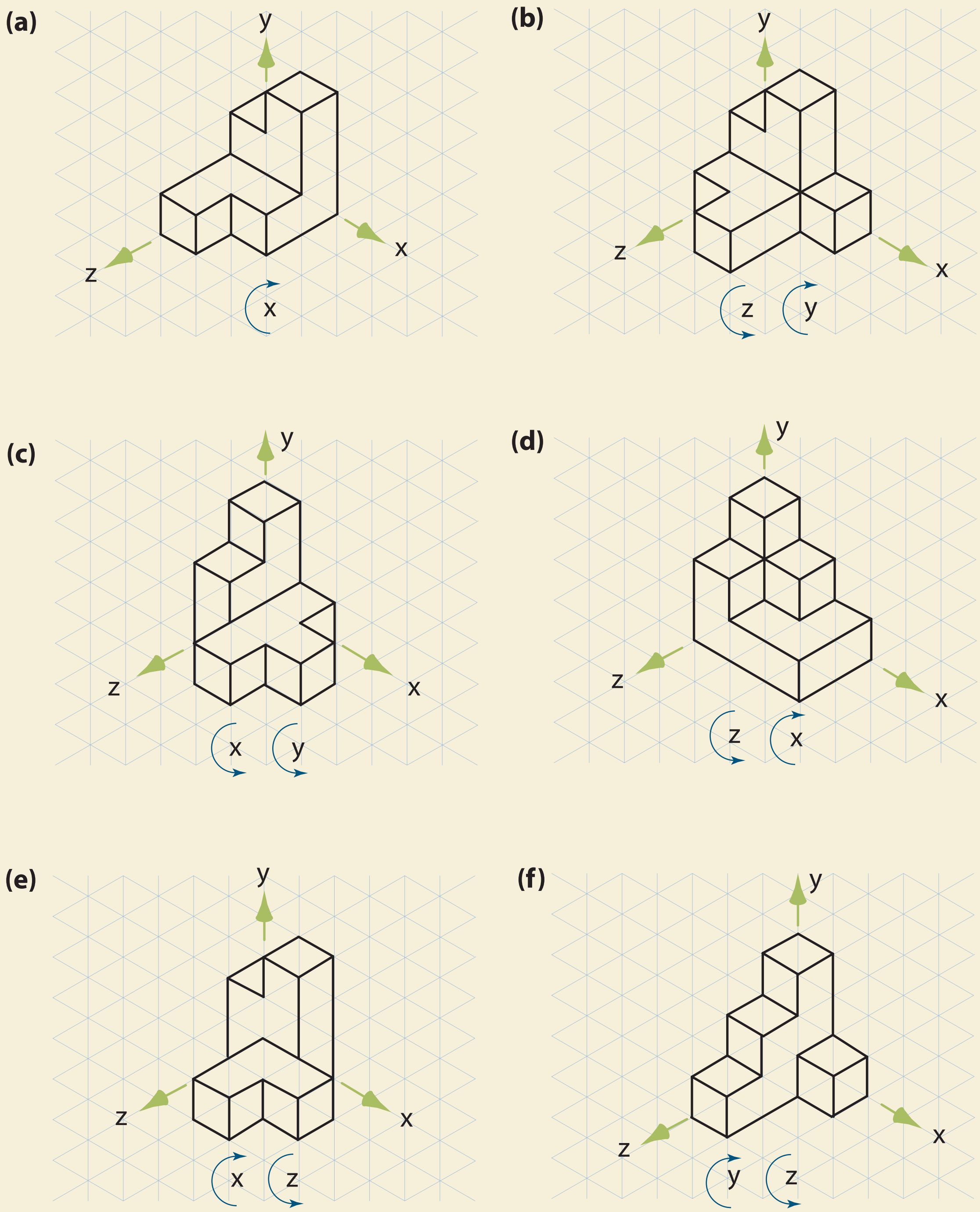

FIGURE P3.5.

3.16 PROBLEMS (CONTINUED)

6. Copy the following object on isometric grid paper and sketch its reflection across the indicated plane. Note that the sketch of the reflection has been started for you.

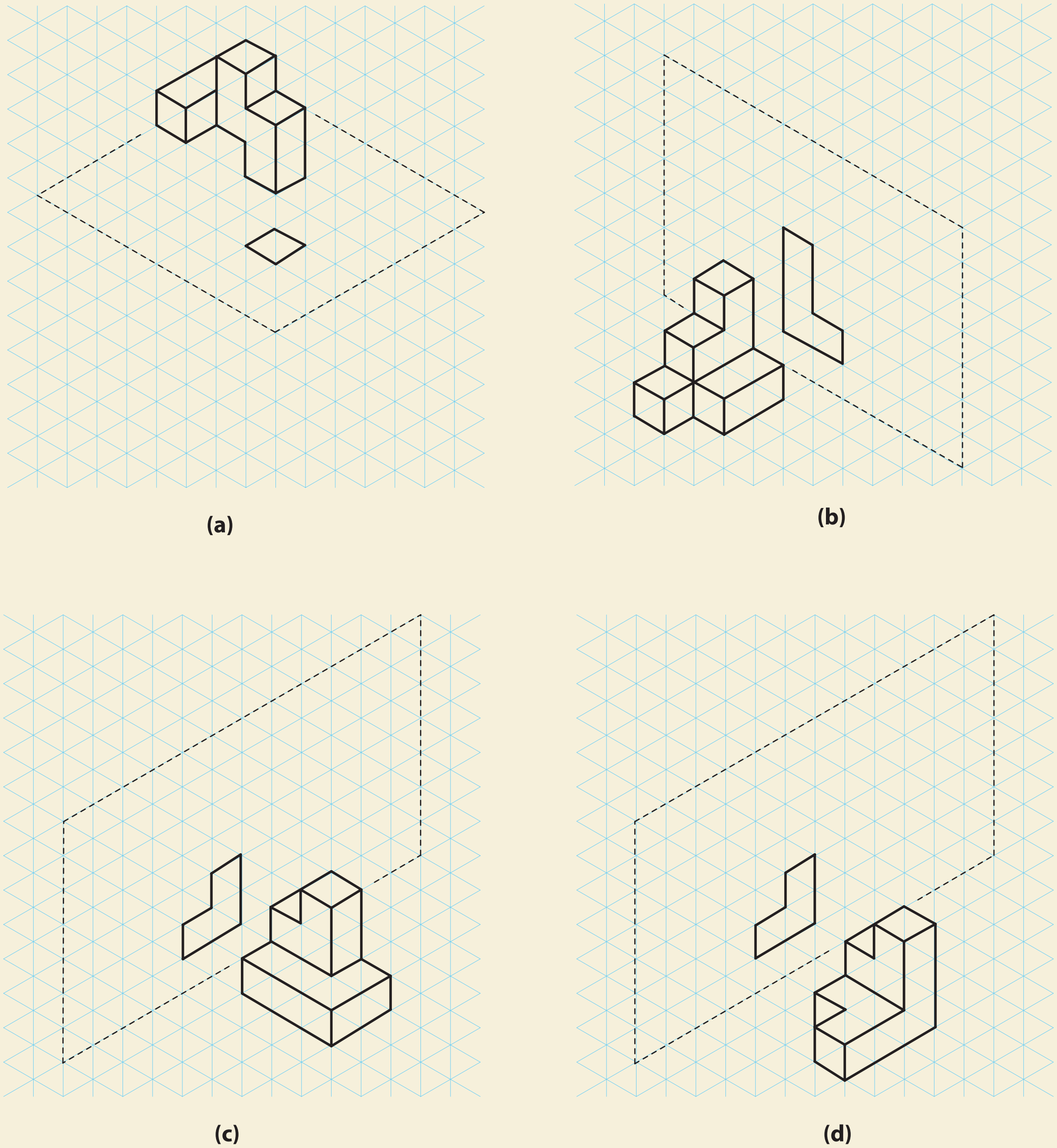

FIGURE P3.6.

3.16 PROBLEMS (CONTINUED)

7. How many planes of symmetry does each of the following objects have?

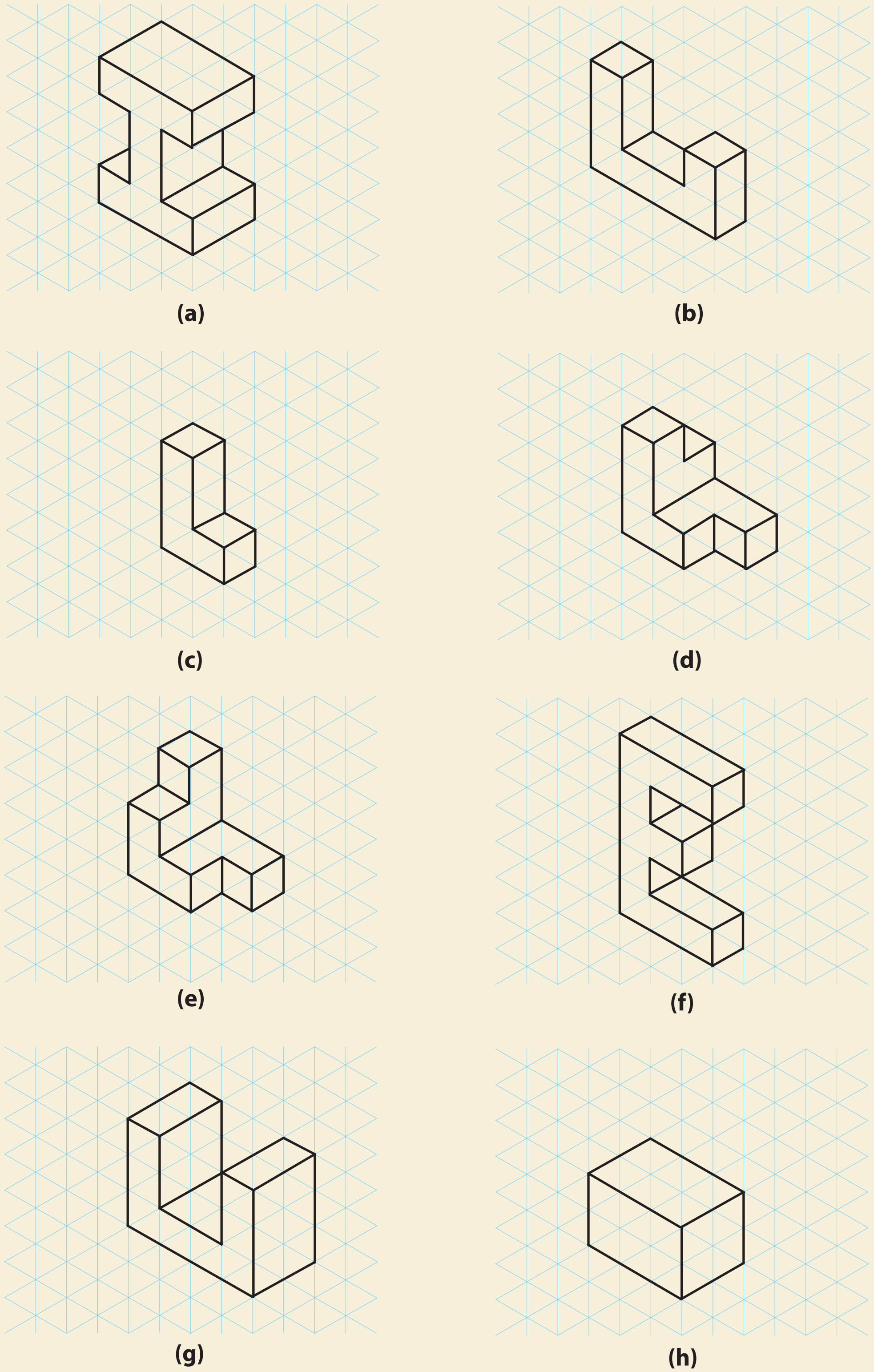

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FIGURE P3.7.

3.16 PROBLEMS (CONTINUED)

8. Sketch the cross section obtained between the intersection of the object and the cutting plane.

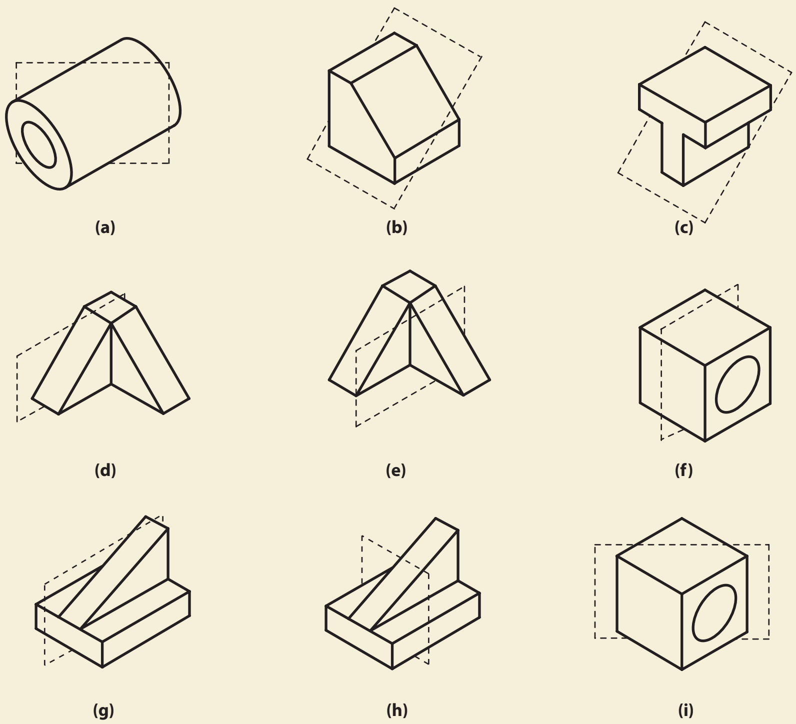


FIGURE P3.8.

9. Sketch the result of combining the following objects by the indicated method.

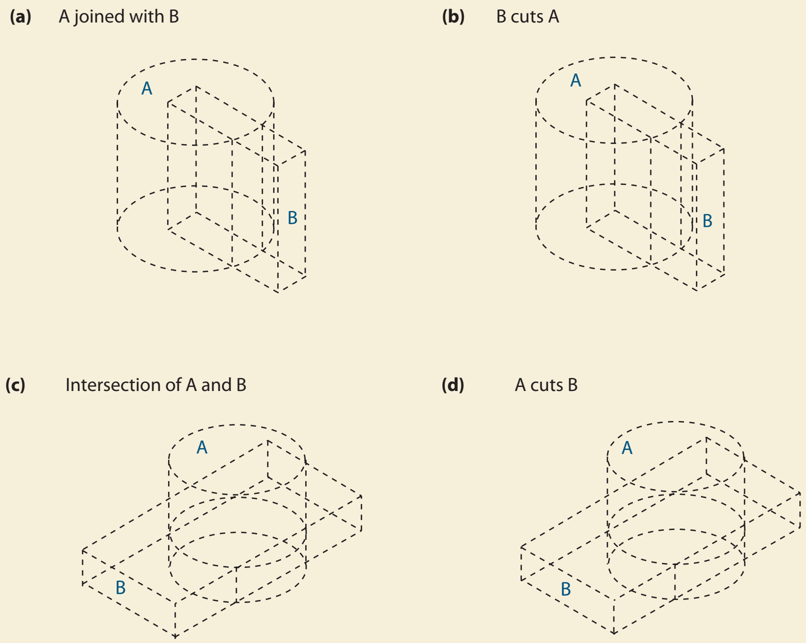


FIGURE P3.9.

3.16 PROBLEMS (CONTINUED)

10. Describe by words and sketches how you would create the following objects by combining basic 3-D shapes.

11. Create isometric sketches from these coded plans using the corner view that is circled or the corner prescribed by your instructor.

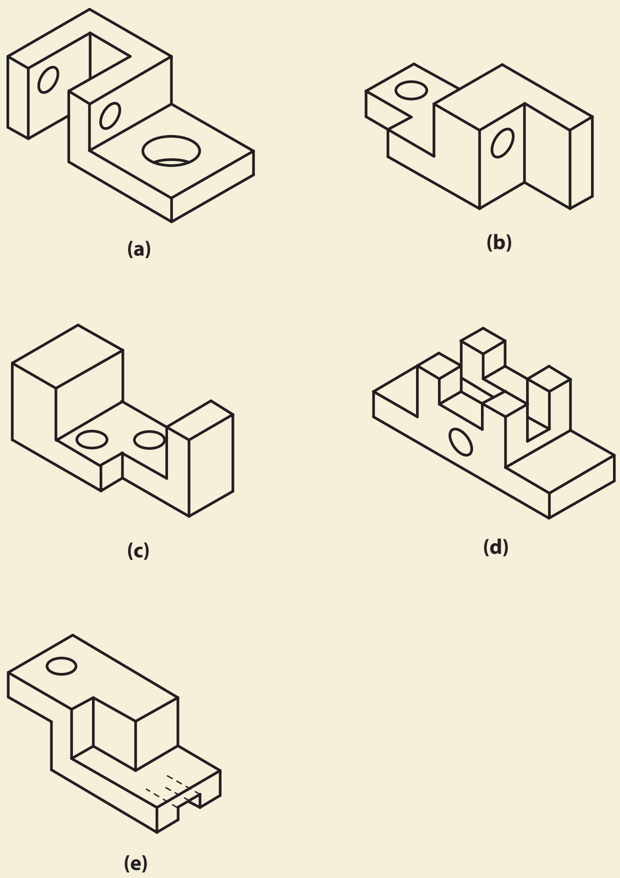
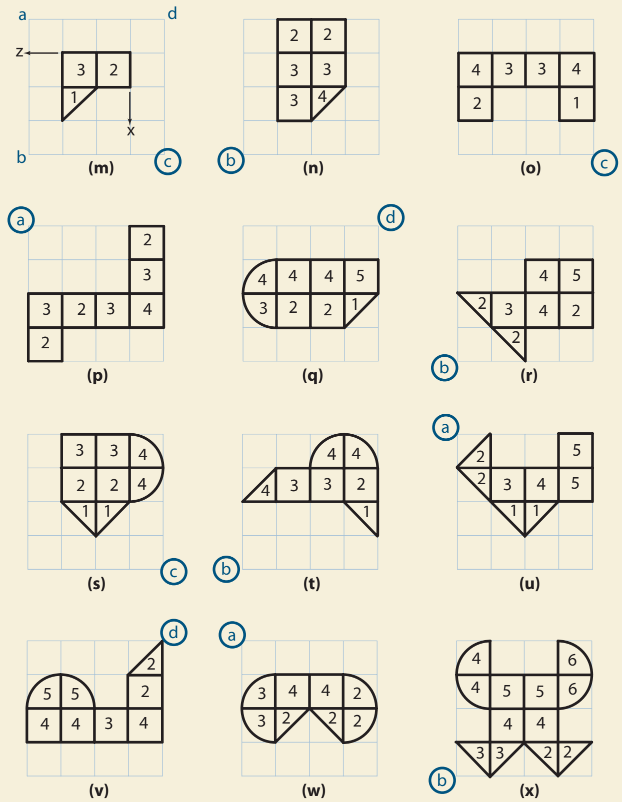


FIGURE P3.10.



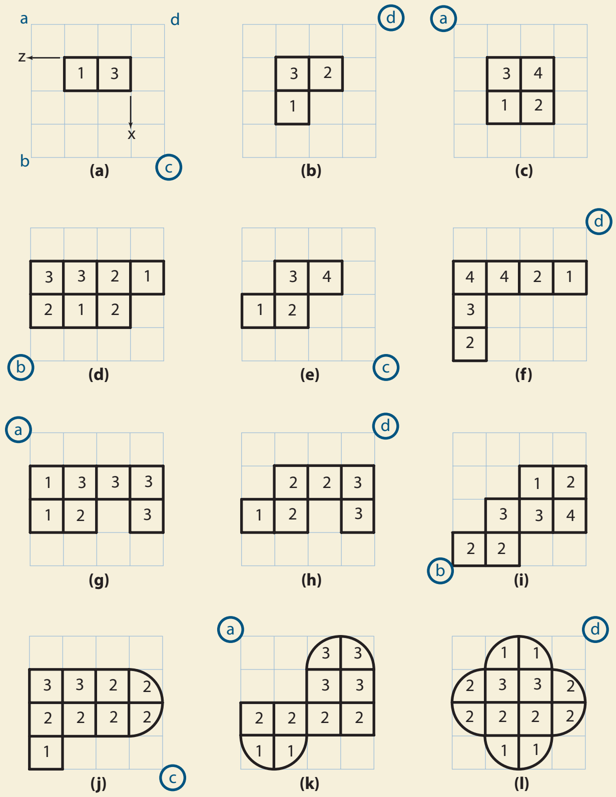


FIGURE P3.11.

3.16 PROBLEMS (CONTINUED)

12. Add the reflected images or redraw these objects with symmetry using the xy-, yz-, or xz-planes as indicated or the planes prescribed by your instructor.

13. The object shown in (a) is show again in (b) rotated by –90 degrees about the y-axis to reveal more detail. Rotate the object sequentially about the axes indicated or about the axes prescribed by your instructor.

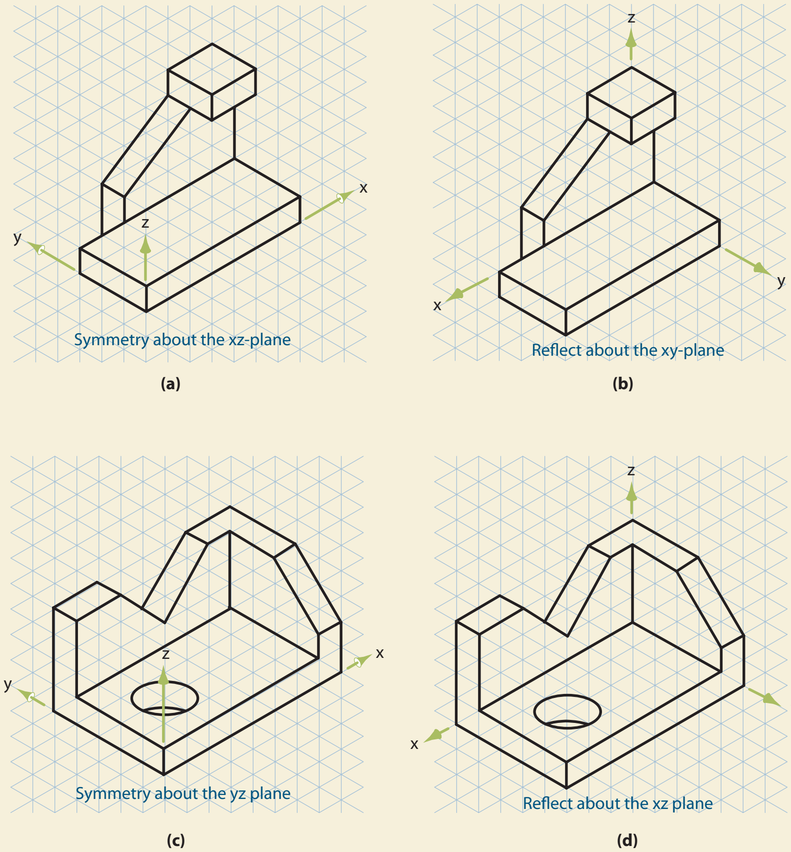


FIGURE P3.12.

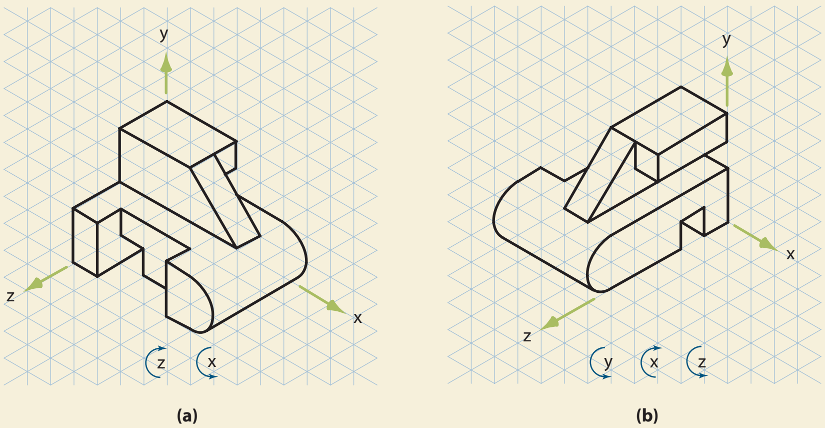


FIGURE P3.13.

3.16 PROBLEMS (CONTINUED)

14. Create an isometric sketch of the objects created from coded plans A and B. Rotate object A sequentially about the axes indicated or about the axes prescribed by your instructor. Show the new object created by the indicated Boolean combination of object A and object B or the Boolean operation prescribed by your instructor when the coordinate axes of A and B are aligned.

15. Triangular volume A, triangular volume B, and rectangular volume C are shown intersecting in space. On the dashed outline drawings, darken and add edges to show all visible edges of the final volume created by the indicated Boolean operations.

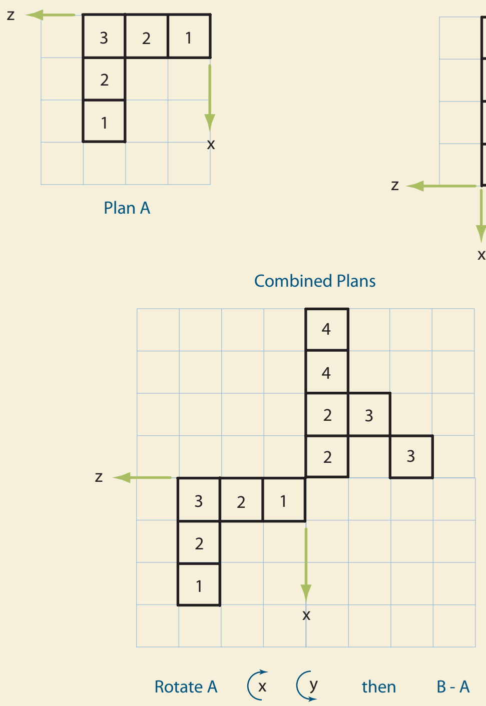


FIGURE P3.14.

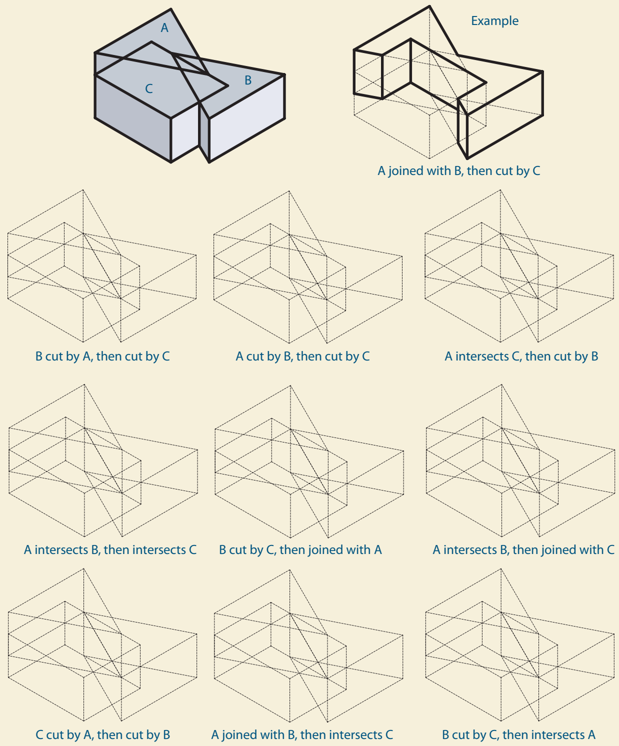


FIGURE P3.15.

CHAPTER 4

CREATIVITY AND THE DESIGN PROCESS

OBJECTIVES

After completing this chapter, you should be able to

- Describe the steps in the design process
- Identify engineering tasks at the various stages of the design process
- Explain the role of creativity in the engineering design process
- Use several techniques that aid in the development of creative thinking

4.01 INTRODUCTION

What distinguishes engineers from scientists is that engineers design and create solutions to the many technical problems of the world, whereas scientists study problems and report their findings in literature. For example, environmental scientists studied Earth's atmosphere several years ago and found that, as a society, the world is creating too many air pollutants—especially from automobile emissions. Certain engineers acted on the results of those studies and designed cars that were more fuel-efficient and that produced less pollution. Other engineers worked on changing the composition of fuel so that fewer pollutants were produced during combustion. Engineers are now working on solar- and electrical-powered vehicles and on hydrogen fuel cells to further reduce the quantity of air pollutants emitted into the atmosphere. Other engineers are working on ways to use biofuels for cars.

Engineering is a design-oriented profession. Engineers design devices or systems and figure out how to mass-produce them, how to package them, and how to ship them to their intended destination. As an engineer, the type of device or system you design may depend on your discipline, but the design procedure you use will be essentially the same regardless of the industry in which you work.

As described in a previous chapter, engineers function in many capacities in business enterprises. Some engineers are involved in manufacturing, some in marketing, some in management, and some in testing. However, design is the central function of engineers. Typically, as you move through an engineering career, you will be responsible for different aspects of the enterprise at different times; but at some point, you will probably play a primary role in the design of new products. At other times, you may oversee the design of new products as a supervisor of a team of engineers. You also may be responsible for making sure the entire enterprise—including designers, manufacturers, and marketers—is running smoothly and the various team members are communicating with one another and working together to achieve the optimal end product. Because design is such a central portion of the engineering endeavor, you need to understand the design process thoroughly. The remainder of this chapter describes the engineering design process and the role creativity plays in the process.

4.02 What Is Design?

Design is a goal-oriented, problem-solving activity that typically takes many iterations—teams rarely come up with the "optimal" design the first time around. As an example, think of the minivan. When Chrysler developed the first minivan, it was considered a revolutionary concept in design and other car manufacturers quickly developed competing models. With each model, improvements were made to the original design such that the minivans of today are much improved compared to the initial product. The key activity in the design process is the development and testing of a descriptive model of the finished product before the product is finally manufactured or constructed. In the case of a manufactured product, the descriptive model usually includes solid 3-D computer models, engineering sketches and drawings, and possibly rapid prototypes. For a civil engineering project, the descriptive model includes drawings, specifications, and sometimes a scale model made of wood or plastic. Three-dimensional CAD models also are becoming more prevalent in the civil/construction industry. **Engineering design** includes a systematic approach to product definition, conceptualization, development, testing, documentation, and production/construction. Design is usually accomplished in a group environment with many people contributing ideas and skills to complete the finished design. Hence, creativity and interpersonal skills are important attributes of the design engineer.

When designing a device or a system, the engineer must keep certain factors in mind. These factors are usually related to the function and cost of the resulting system. For **sustainable design**, life cycle analysis and environmental impact are especially important factors. In most cases, engineers must make a number of choices during the design process. For example, consider the automobile. Engineers may choose from metals, composites, or plastics for the materials that make up the car's body. Each type of material has advantages and disadvantages. Although steel is strong and ductile, it is prone to corrosion (rust) and is relatively heavy, reducing the fuel efficiency of the car, which, in turn, leads to increased pollution. Composites are strong and can absorb a great deal of energy during crashes, but can be brittle and may be more expensive than steel. Plastics are readily formed in almost any shape and are resistant to corrosion, but are relatively weak, making safety a significant concern. Although plastics are widely used in car bumper systems, they typically are not used in the car's body. These are just a few of the factors that engineers must consider as they design an automobile body.

Engineers make choices by weighing the often competing factors associated with function, cost, and environmental impact. A car could be built that causes virtually no pollution and that is perfectly safe; however, the average person may not be able to afford it. So engineers make trade-offs between cost, safety, and environmental impact and design a car that is reasonably safe, is relatively inexpensive, and has minimal emissions.

Design is an aspect in virtually every discipline of engineering; however, chemical engineers typically view design differently than do mechanical engineers. For a chemical engineer, design includes determining the correct chemicals/materials to combine in the correct quantities and in the correct order to achieve the desired final product. Chemical engineers determine when to stir the mixture, heat it, or cool it. Electrical engineers may design computer chips or wiring for a building or the antenna system for a car or a satellite receiver. Civil engineers, like most chemical engineers, typically design one-off systems with features and/or specifications that are unique to a single application or location. They may design a single bridge or roadway, or a water distribution system or sewage system. Because civil and chemical engineering designs usually are not mass-produced, it is often impossible to create a **prototype** for testing before construction begins. Imagine the cost of building a "practice" bridge for every bridge that a civil engineering team designs and constructs.

Mechanical engineers typically design products that will be mass-produced for consumer use—cars, bicycles, washing machines, etc. Therefore, prototyping is an important part of the design process for mechanical engineers. Prototypes are the initial design concepts that are often created so that further design analysis can be performed before machines are retooled to produce, say, 10 million copies of a product. The process of creating and testing prototypes often saves a company money because engineers can work out the kinks or discover flaws early in the design stage. In the past, the design process included the production of several prototypes for testing and analysis. Today, much of the testing can be accomplished using computer software tools, greatly reducing the need for prototypes. However, most manufacturing companies still produce at least one prototype before going into the production of a new product. The foundation for many computer-based testing and analyses in the design process is a 3-D solid model, which is a focal topic of this text.

Although modifications exist in the design process for engineers of different disciplines, there are similarities, too. Almost all designs require drawings, sketches, models, and analysis (calculations). The remainder of this chapter will focus on design in a manufacturing arena—the type of design most familiar to mechanical engineers. This type of design results in products that are mass-produced. Where appropriate, variations to the design process for one-off designs (as in civil and chemical engineering) will be described.

4.02.01 Computers in Design

Computers have been used in engineering design for several decades. In the early years, computers were used primarily for their number-crunching capabilities. In other words, computers were employed to perform the tedious calculations involved in engineering design. Over the years, the role of computers in the design process changed significantly. Graphical computer workstations evolved and with them the ability of engineers to see their designs before building them. Engineers also can do much of the testing of design iterations on-screen, eliminating the need for numerous prototypes. Numerical methods such as **finite element analysis (FEA)**, modal analysis, and thermal analysis have enabled engineers to design systems in a fraction of the time that traditional design methods require. Modern design software often is easily incorporated into the manufacturing process, enabling the designer to establish cutting tool paths on-screen for the efficient manufacture of computer-generated models. Even other manufacturing capabilities, such as rapid prototyping, in which physical prototypes can be created within a matter of hours rather than days, have been a direct result of computer-aided design capabilities.

Today, **computer-aided design (CAD)** is an efficient design method. The basis for CAD is the construction of a graphical 3-D model on the computer. This model can then be tested by any of the available numerical methods. Design modifications can be accomplished on screen and the modified 3-D models tested again. When the engineer is satisfied that the design will meet or exceed all of the design criteria, a 2-D drawing can be created using the 3-D model as its basis. From this drawing, a physical prototype can be created and tested by traditional means to ensure compliance with the design criteria. Then the drawing is usually handed over to the manufacturing division for mass production. Alternatively, when a **computer-aided manufacturing (CAM)** system is available, the part or parts are produced directly from the 3-D computer models.

4.02.02 Classification of Engineering Designers

Engineering design is a broad concept with many integrated stages, competing alternatives, and diverse requirements for success. As such, many design teams in industry have specialty engineers who are responsible for certain aspects of the design process. Most design projects have a team leader, or **chief designer**, who oversees the work of the individual team members. In the early stages of the process for mass-produced designs, **industrial designers** lend their creative skills to develop the product concept and style. For civil engineering projects, **architects** may be employed for their creative talents in **conceptual design**. Specialists in CAD, **CAD designers**, develop the computer geometry for the new design. **Design analysts**, specialists in FEA and other software tools, check the new design for stress and load distribution, fluid flow, heat transfer, and a host of other simulated mechanical properties. **Model builders** are engineers who make physical mock-ups of the design using modern rapid prototyping and CAM equipment. **Detail designers** complete the final design requirements by making engineering drawings and other forms of **design documentation**. Before you begin reading about the **design process**, you should consider the role that creativity plays in the process.

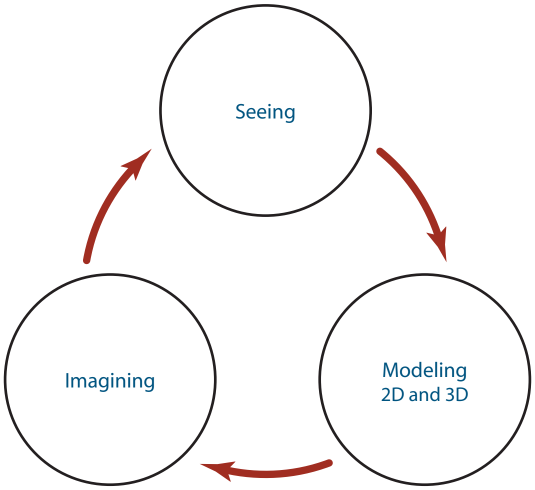
4.03 Creativity in Design

Creativity is an important feature of the design process. Often, engineers get hung up on the "rules" and "constraints" of design and forget to think creatively. Historically, there is a strong link between engineering and art. One of the earliest recognized engineers is Leonardo da Vinci. In fact, some people say that da Vinci was an engineer who sometimes sold a painting so he could make a living. If you examine the sketches and drawings of da Vinci, you will see that he was interested in the development of products to help improve people's lives, including some of the first recorded conceptual designs of airplanes and helicopters. Creativity is at the heart of innovation, and it can be enhanced with both individual and group activities. Some of the more common activities used to facilitate creative thinking are described subsequently. However, psychologists have found that the brain works most creatively when the hands are engaged in completing a mindless task. You probably have experienced occasions when you try to remember something but cannot, then find that the thought pops into your head when you begin another task. So to free your mind for creative thoughts, it may be best to take a break and wash the dishes, do some yard work, lift weights, or do laundry. Performing any of these mindless tasks will help free your mind for creative thoughts.

4.03.01 Visual Thinking

Visual thinking is the process of expanding one's creative ideas using visual cues and feedback. The visual cues can take the form of sketches or computer models; however, in the *initial* stages of design, sketching is often viewed as a necessary ingredient for creative thought. Visual thinking can be thought of as a circular feedback loop, as illustrated in Figure 4.01. The visual thinking process can start at any place in the feedback loop; but for the sake of simplicity, start with the step labeled "Imagining." You first imagine an idea for a new design or product and then sketch the idea in some graphical mode (2-D sketch, 3-D sketch, or computer image). Seeing the idea adds to your understanding of it, which can be extrapolated more deeply. You get a better mental image with a visual cue, which allows you to take the preliminary idea and refine it, sketch it again, etc. You can continue the process until you have a well-defined sketch or idea of the product for formal analysis and design.

FIGURE 4.01. Visual thinking model.



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4.03.02 Brainstorming

Brainstorming is the most common form of group ideation and concept generation and is a process used for generating as many ideas as possible. Brainstorming is typically done as early in the design process as possible (i.e., before you start solving the problem, before breaking the project into tasks, and before deciding who will do each task). Once individuals begin to focus on specific tasks, it may be too late to consider alternatives. Too often, teams will tackle a problem by taking the first idea presented and pursuing that idea without considering any alternatives. Teams also can use brainstorming to generate a list of the tasks that need to be done before the project can be completed. Five simple steps define a brainstorming session:

Step 1: Assemble your project team and make sure you allow ample time for the session. Diversity in the group will enhance the quality and breadth of the ideas generated. Select a group leader to run the session and select a group recorder to take notes.

Step 2: Define the idea of the design project to be discussed. Write down the idea and make sure everyone in the group understands it.

Step 3: Discuss the rules about brainstorming and make sure everyone in the group agrees to abide by them. If necessary, keep the rules on display as a reminder to members who may stray. Rules for brainstorming are intended to help the team generate more ideas. Comments about an idea, whether positive or negative, can stifle the brainstorming process. Although the following rules are simple, you may find them difficult to follow:

1. Everyone participates.
2. Every idea is recorded.
3. Judgment is suspended—there is no such thing as a bad idea.
4. No criticism is permitted.
5. No commentary is permitted.
6. No one dominates the process.

Step 4: Start the brainstorming session by asking everyone in the group to offer an idea. If possible, the recorder writes down all responses so everyone can see them. Alternatively, each member of the team can keep a list of his own ideas. These lists will prevent ideas from being lost and will help restart the process if there is a pause in the flow of ideas.

Step 5: At the end of the session, spend time going through all of the ideas. Combine, categorize, and eliminate the ideas to narrow the list. Once team members are sure that all possible ideas have been included, the team should discuss the advantages and disadvantages of each idea. After talking about the pros and cons, each member should rank the best three ideas on the list. The team should keep the ideas with the highest rank and decide as a group which approach to use.

Instead of conducting the entire brainstorming session by the process outlined previously, you may consider using the following steps for a brainstorming session:

1. Individually spend ten minutes writing down ideas for tackling the assigned project.
2. Combine the individual lists onto a flip-chart, blackboard, or whiteboard. Team members may ask questions to clarify an idea, but no other comments are allowed during this process.
3. Continue brainstorming as a group until all ideas have been exhausted.

4.03.03 Brainwriting (6-3-5 Method)

Brainwriting is an alternative to brainstorming. In brainwriting, each member of the group focuses on sketching his ideas rather than verbalizing them. With brainwriting, you typically start with a team of six people. Each person sketches three ideas on a sheet of paper, leaving ample room for additional graphics and annotations. The idea sketches are then passed around the table so fellow members can add their own comments and ideas, as shown in Figure 4.02. Usually, the idea sketches are passed around the group five times. The expectation is that by the fifth time around, a favorable design idea will have emerged. Brainwriting also is called the *6-3-5 method* (six people, three ideas each, five times around the table).

4.03.04 Morphological Charts

Morphology refers to the study of form and structure. A **morphological chart** can be used to generate ideas about a new design concept. The chart has a leading column that lists the various desirable functions of the proposed design. Along each row, various options for each function are listed, as shown in Figure 4.03. You can use brainstorming techniques to list as many options as possible for each function. The group then reviews and decides on a priority pathway through the options to address each desired function.

FIGURE 4.02. Brainwriting.

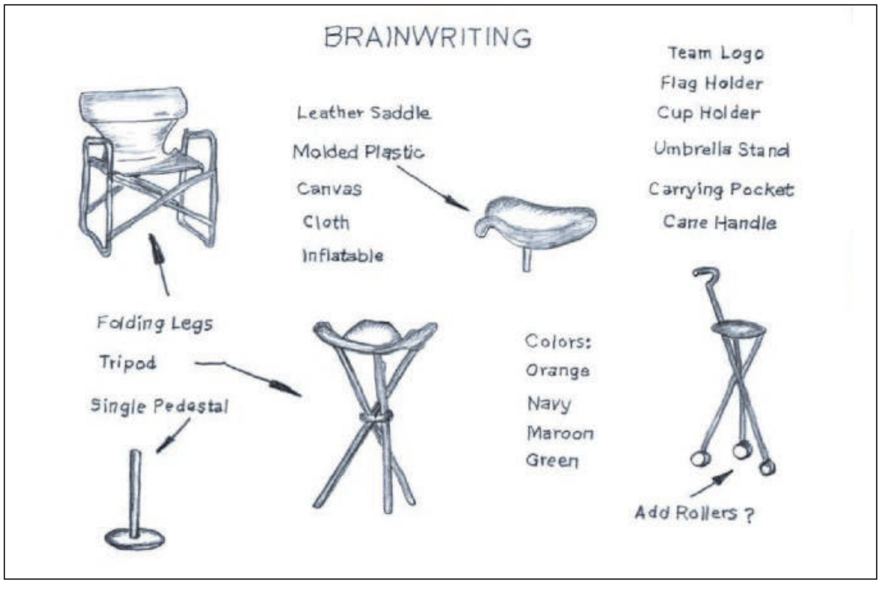

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FIGURE 4.03. A morphological chart.

MORPHOLOGICAL CHART					
Function	**Options**				
Seat Style	Saddle	Molded Dish	Strap	Inflatable	
Seat Materials	Leather	Plastic	Canvas	Rubber	Cloth
Number of Legs	One	Three	Four		
Leg Material	Wood	Aluminum	Plastic	Wrought Iron	
Leg Assembly	Pin	Hinge	Force Fit	Folding	
Accessories	Cup Holder	Beverage Cooler	Umbrella Holder	Flag Post	
Aesthetic Offerings	Team Logo	Choice of Colors			
Carrying Style	Carrying Handle	Handle Like Cane	Strap On Back	Roller Wheels	

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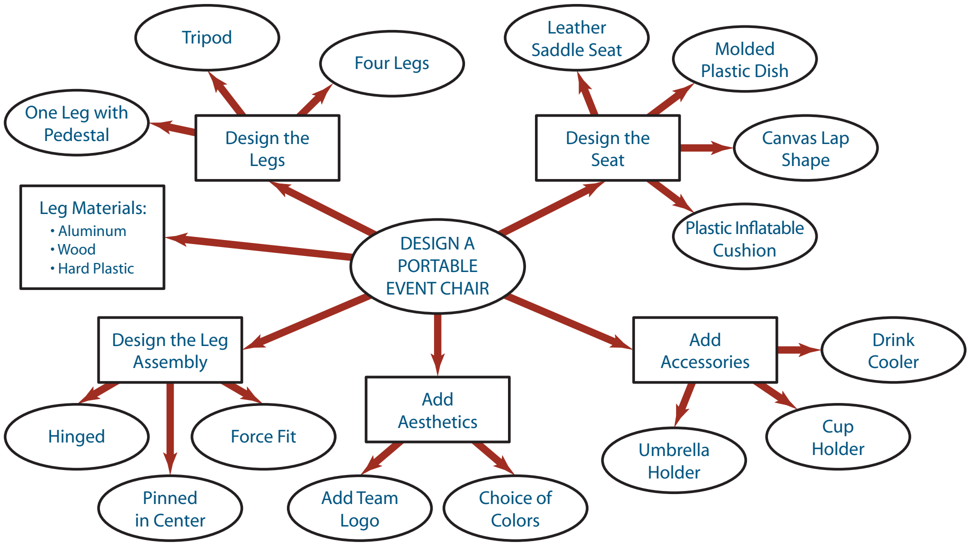
FIGURE 4.04. A concept map.



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4.03.05 Concept Mapping

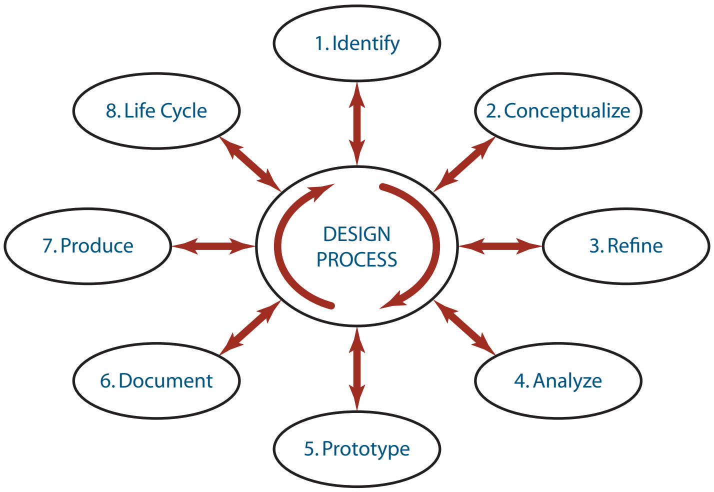
Concept mapping is a technique used to network various ideas together, as shown in Figure 4.04. During idea generation, the main design concept is placed in the center of the map with the various options linked outward in a brainstorming-like session. Each option then serves as a node for other choices. In that manner, the team can explore a big picture of all ideas and see a strong visual image of the connectivity of the different ideas.

4.04 The Engineering Design Process

Design is a multistep process. However, there is considerable disagreement on exactly how many steps are involved in the process. Figure 4.05 shows one example of the sequence of steps in the design process. The design team often starts with stage 1 (Identify) and continues to stage 7 (Produce) and then begins again. Often, the process does not proceed sequentially from stage 1 to stage 7; the design team might discover a serious problem in stage 4 (Analyze) and then return to stage 2 (Conceptualize) before moving on to stage 5 (Prototype).

Textbooks and writings on the design process include many different versions of and names for the stages in the design process. The stages presented previously

FIGURE 4.05. An engineering design process.



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are just one way to look at the design process; therefore, you should not think of them as the definitive word on the subject. However, the stages described in the remainder of this chapter are related to the graphical tools you will study in this textbook. For this reason, they have been adopted here. Knowing the number of stages and the labels for each stage is not nearly as important as understanding the overall process.

4.04.01 Stage 1: Problem Identification

Good design practice starts with a clearly defined need for a new product or system. Alternatively, a revised, improved design for an existing product may be required. A market survey may demonstrate that the new product or system is useful, has market appeal, is producible with today's technology, and will make a profit for the company supporting the design effort. Sometimes a new design idea is simply an alternate solution to a problem answered by an existing competing product. Indeed, many of today's highly successful products are the evolutionary result of free market enterprise. In the civil engineering world, a design project is typically the result of a client requesting a specific structure or system. For example, a governmental agency such as a county may request that a civil engineering firm design a new water distribution system to serve the needs of the county's residents. In this case, the client may have already defined the problem.

In the **problem identification** stage, the design engineer must address questions and answers from the customer's/client's perspective and from the engineer's perspective. For example, Table 4.01 shows two different perspectives for a new urban bicycle design.

When a product is designed, one of the design considerations is how long will it be used before it is no longer effective, which is called the **life cycle** of the product. Some products are designed for replacement on a regular basis. Thus, environmental considerations and disposal of a product must be considered throughout the design process. In the design process called **green engineering**, environmental concerns are considered throughout the process, not just at the end, because an engineer's choices in

Table 4.01. Functional Requirements for a New Urban Bicycle.

CUSTOMER'S PERSPECTIVE	ENGINEER'S PERSPECTIVE
MODERATE STREET SPEED	SUSTAINABLE SPEED OF XX MPH MAXIMUM SPEED OF YY MPH
COMPACT SIZE	DIMENSIONS NOT TO EXCEED A × B × C
SAFE	STRUCTURAL STRENGTH CONTROLLABILITY BRAKING CAPABILITY TIRE PUNCTURE RESISTANCE
COMFORTABLE	ERGONOMICS OF SEAT EFFICIENCY OF POWER TRAIN POSITION(S) OF HANDLEBAR
ATTRACTIVE	CHOICES OF PAINT COLOR LIGHTS AND REFLECTORS
AFFORDABLE	SELECTION OF MATERIALS MANUFACTURING PROCESS NUMBER OF PARTS SALES VOLUME

the problem definition stage often influence the overall environmental impact and life cycle of the product. Once the functional requirements have been identified, the design team can start the design process by generating some concepts.

4.04.02 Stage 2: Concept Generation

Concept generation is the most creative phase of the design process. You learned about some methods for creative thinking and concept generation earlier in this chapter. Typically, concept generation starts with brainstorming, brainwriting, or a similar team meeting where ideas are tossed around and discussed. Criticism is usually limited in the concept generation stage, since maximizing the number of good ideas is desirable. At the end of the concept generation stage, the team should have selected a few main ideas that it will focus on for future refinement and analysis. Sketching is an integral way to develop concepts for a new design. Figure 4.06 shows some examples of concept sketches.

4.04.03 Stage 3: Concept Selection and Refinement

Once a few quality concepts have been identified, the design team must converge on one or two final concepts to further explore in the design process. A common technique for selecting the final concept(s) is to use a **weighted decision table**, as shown in Figure 4.07. With a weighted decision table, all of the common attributes and desirable features of each concept are listed in the first column. A weighting factor for each feature/attribute is then established (e.g., using a scale of 0 to 10). The various design options are listed in subsequent columns in a parallel fashion to the listed features/attributes. The team then conscientiously scores each option for every feature/attribute, each time applying the weighting factor to the score, as illustrated in Figure 4.07. Adding all of the scores for individual attributes yields a final "bottom-line" number that can be used to select the highest-ranked option.

FIGURE 4.06. Concept sketches.

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			DESIGN 1		DESIGN 2		DESIGN 3		DESIGN 4	
FEATURE ATTRIBUTE		WEIGHT 0–10	**OPEN BOTTOM** SCORE–TOTAL 0–10	**S * W**	**THUMB GRIP** SCORE–TOTAL 0–10	**S * W**	**DOUBLE LOOP** SCORE–TOTAL 0–10	**S * W**	**SINGLE LOOP W/FINGER SUPPORT** SCORE–TOTAL 0–10	**S * W**
AESTHETICS	Color	5	3	**15**	3	**15**	3	**15**	3	**15**
	Form	8	7	**56**	7	**56**	7	**56**	7	**56**
ERGONOMICS	Grip Ability	8	7	**56**	6	**48**	2	**16**	9	**72**
	Drinking Ease	6	3	**18**	5	**30**	4	**24**	8	**48**
FUNCTIONALITY	Adapts to Hand	8	5	**40**	7	**56**	2	**16**	8	**56**
STABILITY	Base Size	6	7	**42**	9	**54**	3	**18**	9	**54**
	Height	8	7	**56**	8	**64**	3	**24**	8	**64**
MANUFACTURABILITY	Injection Molding	5	3	**15**	3	**15**	3	**15**	3	**15**
	Slip Molding	5	3	**15**	3	**15**	3	**15**	3	**15**
			Weighted Total	**313**	**Weighted Total**	**353**	**Weighted Total**	**199**	**Weighted Total**	**395**

FIGURE 4.07. A weighted decision table for concept selection.

Sometimes the initial concepts may need to be refined before a final decision can be made. Refinement will likely include the development of 3-D computer models for defining geometry not accurately expressed in the concept sketches. For example, as shown in Figure 4.08, different computer models of a new product can assist the members of the design team in visualizing the specific model that has the marketing appeal they are seeking and in making the final decision.

4.04.04 Stage 4: Design Evaluation and Analysis

In this stage, the selected concept is further analyzed by any number of numerical methods. Before the advent of CAD and analysis tools, this stage of the design process involved building and testing physical models. Now the building and testing can be done on the computer, saving companies a great deal of time and money. The tests are conducted to determine mechanical properties of objects or systems and their performance during simulated conditions.

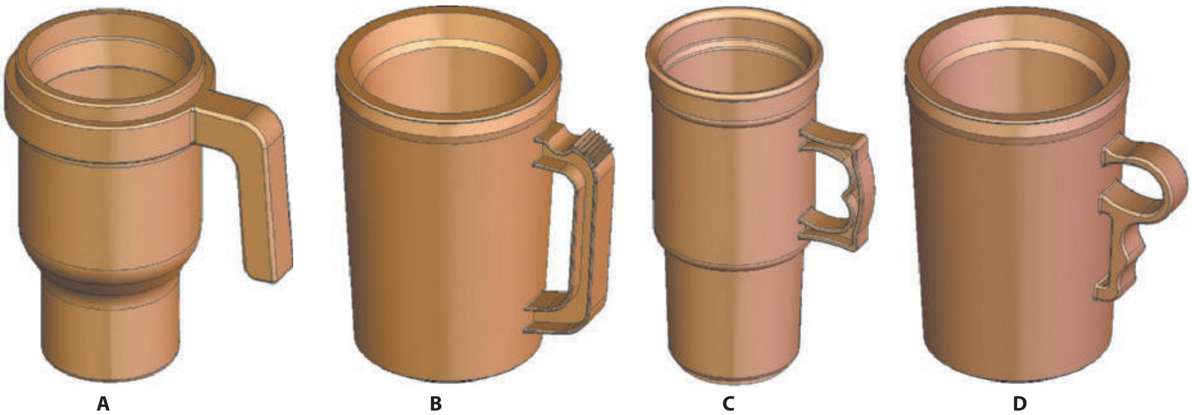

FIGURE 4.08. Computer models for concept selection.

One of the simpler types of analysis that can be performed with a computer model is the computation of the object's mechanical properties, such as mass, center of gravity, and moment of inertia. All of these properties may not be meaningful to you right now; however, they are key quantities used in performing most types of static and dynamic analyses. A **mass properties analysis** report, shown in Figure 4.09, is a useful document for evaluating and presenting the static mechanical conditions of the design.

Further analysis of the design might include an FEA of stress contours and deformation. Heat transfer and aerodynamic flow also can be simulated using modern computational software (Figure 4.10). These numerical methods will be discussed in more detail in a later chapter and are themselves topics of entire texts.

4.04.05 Stage 5: Physical Prototyping

Most designers and clients would like to see a physical model of the design—they want to look at it, hold it in their hand, and show it to other interested parties. Several different types of physical models can be developed during this stage of the design process. Engineers can have the shop people build a *scale model*, an actual *true-size model*, or just a simple *mock-up concept model* that shows the general physical appearance of the design.

FIGURE 4.09. Object mass properties.

MASS PROPERTIES REPORT (Concept 4)

Mass Properties of ergonomic cup (Material – ABS Plastic)

Output coordinate system: -- default --

Density = 0.037 pound per cubic inch

Mass = 0.948 pound

Volume = 25.722 cubic inches

Surface area = 134.693 square inches

Center of mass: (inches)
X = 0.071
Y = 1.648
Z = 0.000

Principal axes of inertia and principal moments of inertia: (pounds * square inches)
Taken at the center of mass.

Ix = (0.033, 0.999, 0.000)	Px = 2.401
Iy = (-0.999, 0.033, 0.000)	Py = 3.770
Iz = (0.000, 0.000, 1.000)	Pz = 3.921

Moments of inertia: (pounds * square inches)
Taken at the center of mass and aligned with the output coordinate system.

Lxx = 3.768	Lxy = 0.046	Lxz = -0.000
Lyx = 0.046	Lyy = 2.402	Lyz = -0.000
Lzx = -0.000	Lzy = -0.000	Lzz = 3.921

Moments of inertia: (pounds * square inches)
Taken at the output coordinate system.

Ixx = 6.341	Ixy = 0.157	Ixz = -0.000
Iyx = 0.157	Iyy = 2.407	Iyz = -0.000
Izx = -0.000	Izy = -0.000	Izz = 6.498

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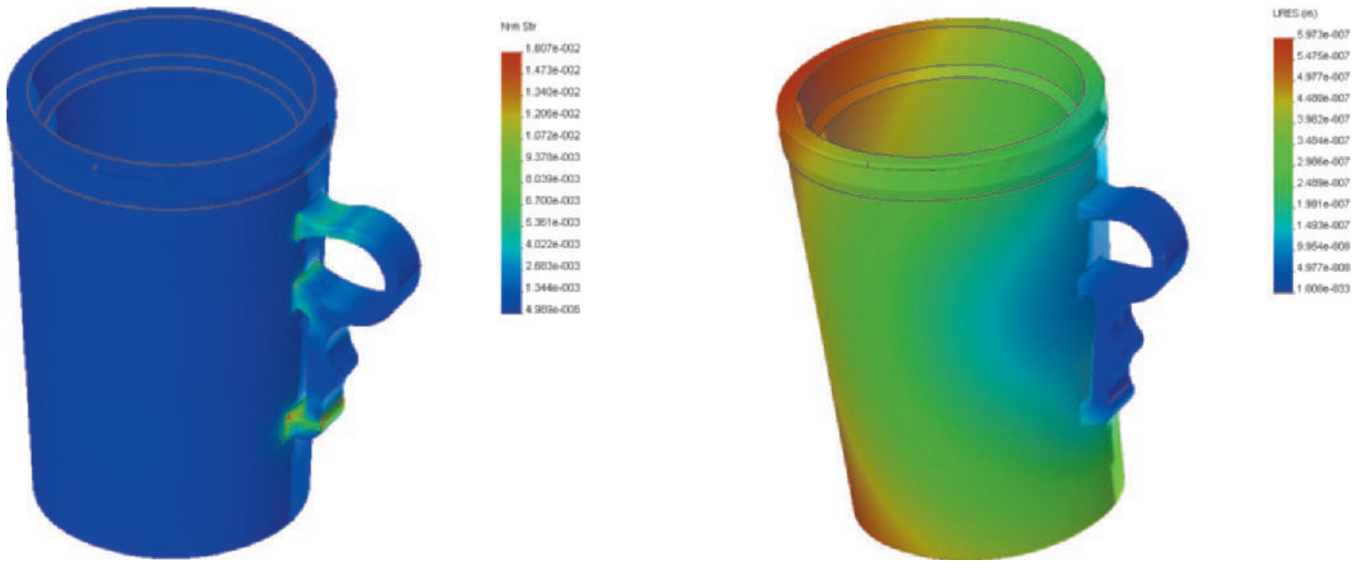

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A

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B

FIGURE 4.10. A numerical analysis model.

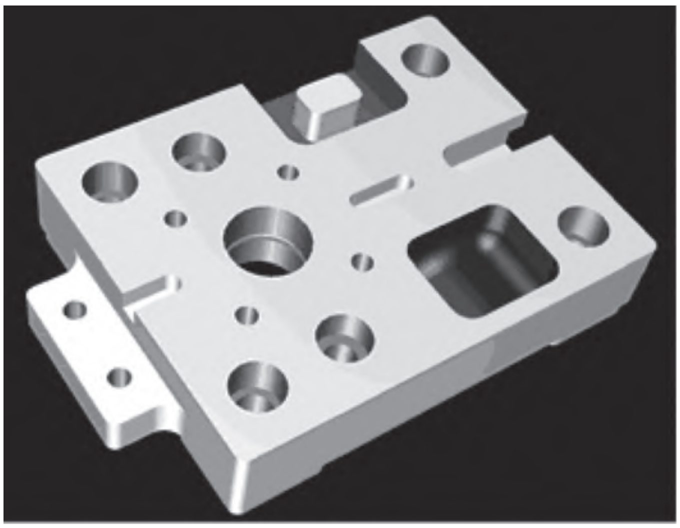

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FIGURE 4.11. A part created through CNC machining.

In recent years, modern technology has accelerated the production of prototype models in the design process. CAM systems can take data from a 3-D solid model and cut the pattern using computer numerical control (CNC) machines. Figure 4.11 shows a part that was created through CNC machining. CNC machining will be covered in more detail in a later chapter of this text.

Today, rapid prototyping systems can perform some of the same functions of traditional machining tools, except that they require far less time (hence, the term *rapid*) and fewer resources than traditional methods. Some of the modern rapid prototyping methods include stereolithography (SLA), Selective Laser Sintering (SLS), Laminated Object Manufacturing (LOM), Fused Deposition Modeling (FDM), Solid Ground Curing (SGC), and inkjet printing techniques. Most recently, 3-D printers have become affordable prototyping alternatives for the office environment. Reasonable 3-D models can be printed using 3-D printers, as shown in Figure 4.12.

FIGURE 4.12. Models created with a 3-D printer.

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4.04.06 Stage 6: Design Documentation

There are many forms of design documentation, but the most common form is a finished detailed drawing, as shown in Figure 4.13. A detailed drawing shows the information needed to manufacture the final part. A good portion of the rest of this text discusses detailed engineering drawings. You will learn how to create drawings and how to interpret them correctly. You will learn about dimensioning and tolerancing for annotation of the drawing. You will learn about conventions developed over the years that provide everyone with the same understanding of what is on the drawing. You also will learn how drawings are created from 3-D computer models of parts and systems.

4.04.07 Stage 7: Production

Once the design documentation is complete, it is time to begin the production stage. For a civil engineering design, production is called the construction stage; for mechanical engineering, production is the manufacturing stage. Many engineers claim

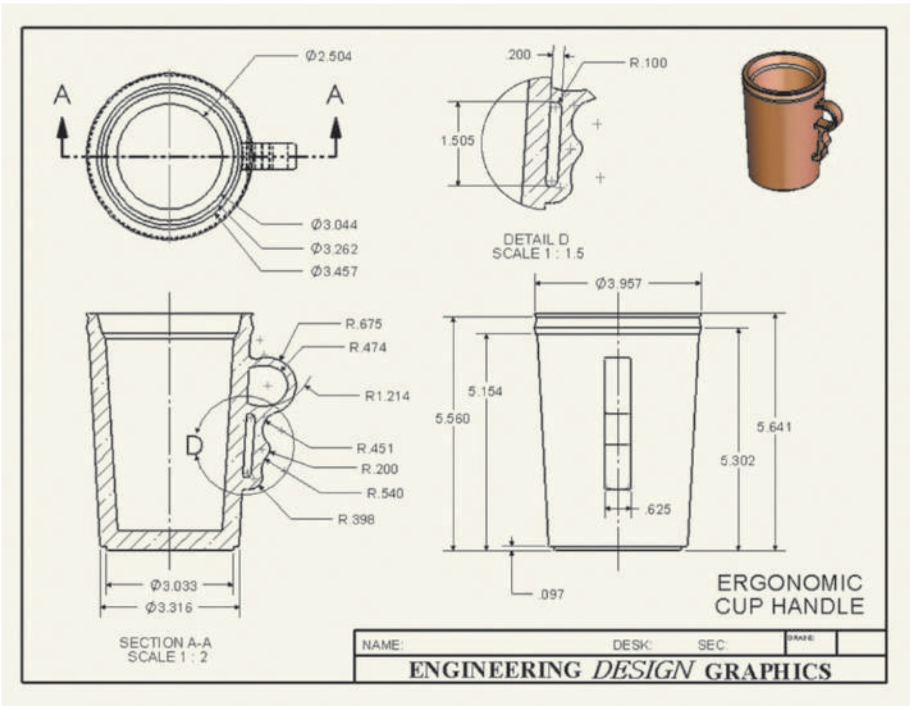

FIGURE 4.13. A detailed design drawing.

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that the design process ends with the documentation stage. However, the way the product is designed may impact its production and distribution processes later on. For mechanical engineering projects in this stage, the goal is usually (but not always) to produce the product in large quantities, to meet performance standards, and to keep manufacturing costs low. Many different methods for manufacturing parts have been developed and are widely used, including machining, casting, rolling, and sheet metal cutting processes. These methods will be covered in more detail in a later chapter.

For civil engineering projects, because no prototypes have been built, design modifications are common in the production stage. Contractors may find, for example, that ductwork does not fit within the space provided on the drawings, or they may find that piping needs to be rerouted around an obstruction. For this reason, in civil engineering projects, it is important to continue to document the design by making notations on the drawings where changes are made. These drawings are called **as-built drawings**, and they reflect the way the project was actually constructed—not just the way it was designed. As-built drawings are an important part of the design process because if piping is rerouted through a building, for example, someone will need to know exactly where the plumbing is in case leaks or other problems arise.

4.05 The Concurrent Engineering Design Process

The new paradigm of **concurrent engineering** is sometimes referred to as "design for manufacturability." In traditional engineering, the part being designed progresses through each stage, moving from one team to the next. At each new stage, the team takes the design from the previous team and applies its own expertise. The first time the manufacturing engineer sees the part for production is when the design and analysis teams finish their work. With concurrent engineering, designers, analysts, and manufacturing engineers work together from the initial stages of the design process. In this way, each person can apply his own expertise to the problem at hand *from the start*. Thus, early in the design process, the manufacturing engineer might say, "If you made this minor modification to the part geometry, we would save $100,000 in retooling costs." The design change in question could be easily implemented during the initial phases of the process; whereas, without concurrent engineering, the change (and related cost savings) might be impossible in the final stage.

Modern computer workstations have enabled concurrent engineering to become a reality in the workplace. With local area networks, wide area networks, and the Internet, data can be moved from one desktop to another almost effortlessly. Members of the concurrent engineering team who work in different countries can share design ideas nearly as easily as engineers who work in the same building. Using the principles of concurrent engineering, manufacturers can save thousands, even millions, of dollars. In addition, computer-aided concurrent engineering design is more efficient and results are often of higher quality compared with designs produced in the past.

4.06 Chapter Summary

In this chapter, you learned that design is an iterative process and that the process has several stages; however, there is no general agreement about the exact number of stages. You learned that engineers must weigh competing factors such as cost, function, and environmental impact when making design decisions. You learned about design in the information age and ways computers are used throughout the process, which greatly reduces costs. You learned about the importance of creativity in engineering design and about several techniques you can use, either as an individual or as part of a team, to foster creative thinking. Finally, you learned about concurrent engineering, which is enabled by computer technologies and can be used to reduce product costs and improve product quality.

CASE STUDY INTEGRATED PROJECT

CONCEPTION OF THE HOYT AEROTEC TARGET BOW

Hoyt USA was founded in 1931 by sportsman and bow maker Earl Hoyt. The company is located in Salt Lake City, Utah, where it has both engineering and fabrication facilities. Hoyt USA has a long-standing reputation as a high-quality maker of bows for sports, recreation, and competition. In 1972, the company revolutionized competition archery when it introduced its first metal-handled collapsible recurve bow at the 1972 Olympic games. Given only to the U.S. team, the metal-handle design offered significant advantages over other bows, which were mostly made of wood at that time. A metal handle was relatively immune to the effects of changing temperature, humidity, and time, which affected the geometry, stiffness, and vibration properties of the bow. These variations made it difficult to use a wooden bow to land arrows shot after shot in the same place on a target. In addition, lighter arrows produce higher stresses in a wooden bow, sometimes causing the bow to break. By contrast, the strength of the handle's metal enabled an archer to use lighter arrows, which reached more distant targets in shorter times. Soon after Hoyt USA introduced the metal-handled bow, other bow manufacturers followed suit.

In the early 1990s, Hoyt USA wanted to improve the design of its metal bow handle to improve its share of the target archery equipment market. This market was very competitive, with a typical recurve bow lasting only three or four years before it needed to be replaced. In the development of a new design, Hoyt USA had to consider several things. For product performance, the main considerations were strength, weight, and vibration. A new product had to be stronger than previous ones. Super-light carbon arrows, light composite bow limbs, and synthetic strings produced increasing levels of stress in bows, to the point where even metal handles were breaking. But additional strength could not be gained at the expense of weight. Many archers already complained about the excessive weight of metal-handled bows. Any new product also needed to be less flexible and have less vibration than existing bows. Target archers considered excessive flexibility, noise, and vibration to be undesirable characteristics.

The new product needed to be developed with analysis and production in mind. Detailed stress analysis was necessary to ensure that there would be no breakage problems, as had been the case with other products. Also, the new product had to be designed so it could be easily produced in state-of-the-art fabrication facilities consisting mostly of computer-controlled, four-axis milling machines, which Hoyt USA was expanding at the time.

The new concept that Hoyt USA developed was a structural support member located behind the grip of the bow handle. The support member was designed with an outward bend so it would not touch the arm of the user. Touching the bow at any location except the grip was a violation of the rules in target archery. The new design allowed the forces in the bow handle to be more widely distributed, resulting in reduced stresses without a significant increase in weight.

Conceptual sketches for the new design were developed first. The sketches enabled Hoyt USA engineers to communicate ideas among themselves, as well as to engineers outside their group, managers, production specialists, and potential customers. About 20 design variations were examined on the drawing board, after which three or four were selected for

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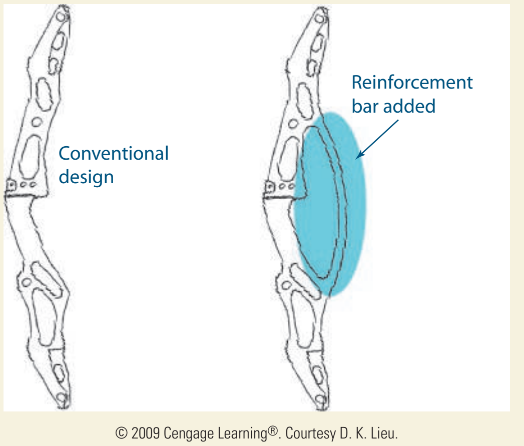
A conventional metal-handled recurve target bow.



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Concept sketches showing the addition of a structural reinforcement bar to a conventional bow handle to reduce its flexibility and vibration.

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A solid model of the new design used for analysis and fabrication.

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The production version of the structurally reinforced handle.

further development. A solid model was built using a computer, because the model could be used for stress analysis with finite element methods and because the model could be exported directly to the fabrication machines on Hoyt USA's production floor.

After final selection and refinement of the design as a result of the stress analyses and field testing, Hoyt USA began full production of the new product less than one year after the concept was first discussed. To protect the innovative design from being copied by competitors, Hoyt USA applied for and was granted a U.S. patent. Patents also were secured in foreign countries. The design concept was trademarked TEC for "Total Engineering Concept." Because of the relatively radical appearance of the product (and the rather conservative nature of archers), at first, the product was slowly and cautiously adopted by the market. But after a few years (and some outstanding performances at the Olympics), TEC products by Hoyt USA were eagerly embraced by the market as a superior technology.

Discussion Questions/Activities

1. Explain the design process that the developers at Hoyt used to engineer the TEC bow.
2. Create a weighted decision table based on the questions and answers that might have been generated during the design process of the TEC bow.
3. In what ways were concurrent engineering techniques used throughout the development of the TEC bow?

4.07 GLOSSARY OF KEY TERMS

architects: Professionals who complete conceptual designs for civil engineering projects.

as-built drawings: The marked-up drawings from a civil engineering project that show any modifications implemented in the field during construction.

brainstorming: The process of group creative thinking used to generate as many ideas as possible for consideration.

brainwriting: A process of group creative thinking where sketching is the primary mode of communication between team members.

CAD designers: Designers who create 3-D computer models for analysis and detailing.

chief designer: The individual who oversees other members of the design team and manages the overall project.

computer-aided design (CAD): The process by which computers are used to model and analyze designed products.

computer-aided manufacturing (CAM): The process by which parts are manufactured directly from 3-D computer models.

concept mapping: The creative process by which the central idea is placed in the middle of a page and related concepts radiate out from that central idea.

conceptual design: The initial idea for a design before analysis has been performed.

concurrent engineering: The process by which designers, analysts, and manufacturers work together from the start to design a product.

design analysts: Individuals who analyze design concepts by computer methods to determine their structural, thermal, or vibration characteristics.

design documentation: The set of drawings and specifications that illustrate and thoroughly describe a designed product.

design process: The multistep, iterative process by which products are conceived and produced.

detail designers: The individuals who create engineering drawings, complete with annotation, from 3-D computer models or from engineering sketches.

engineering design: The process by which many competing factors of a product are weighed to select the best alternative in terms of cost, sustainability, and function.

finite element analysis (FEA): A numerical method used to analyze a product in terms of it structural, thermal, and vibrational performance.

green engineering: The process by which environmental and life cycle considerations are examined from the outset in design.

industrial designers: The individuals who use their creative abilities to develop conceptual designs of potential products.

life cycle: The amount of time a product will be used before it is no longer effective.

mass properties analysis: A computer-generated document that gives the mechanical properties of a 3-D solid model.

model builders: Engineers who make physical mock-ups of designs using modern rapid prototyping and CAM equipment.

morphological chart: A chart used to generate ideas about the desirable qualities of a product and all of the possible options for achieving them.

problem identification: The first stage in the design process where the need for a product or a product modification is clearly defined.

prototype: The initial creation of a product for testing and analysis before it is mass-produced.

sustainable design: A paradigm for making design decisions based on environmental considerations and life cycle analysis.

visual thinking: A method for creative thinking, usually through sketching, where visual feedback assists in the development of creative ideas.

weighted decision table: A matrix used to weigh design options to determine the best possible design characteristics.

4.08 QUESTIONS FOR REVIEW

1. What are the main stages in the design process?
2. Why is creativity important in the engineering design process?
3. How does engineering design differ from the type of design artists perform?
4. What is meant by concurrent engineering?
5. How is a computer used in the modern-day design process?
6. What are some of the differences in design for a civil engineering project versus a mechanical engineering project?

4.09 DESIGN PROJECTS

The following sections will outline specifications for design projects. These projects were tested with students at the University of California at Berkeley over the years and are suitable for use in a first- or second-year design course at a university. The projects are designed for completion by a team of four or five students.

4.09.01 Standard Project Materials

Use the following standard list of materials, in addition to any special items listed in the specific design rules for the project, to construct the device assigned by your instructor. No other materials are permitted.

- Paper, 30# (maximum): 2 square meters maximum; 2 layers maximum
- Poster board, single-ply, medium weight: 1 square meter maximum
- Foam core modeling board, 3/16" nominal thickness: 1 square meter maximum
- Twine, 60 lbs. (maximum) labeled breaking strength: 3 meter length maximum
- Wood dowel, 1/4" nominal diameter: 1 meter length maximum
- Mailing tube, 2" nominal diameter, medium-weight cardboard: 1 meter length maximum; no endcaps
- Rubber bands (sample to be supplied), #62 or #64: 10 maximum
- Elmer's Glue-All glue: 30 cc maximum
- Hot melt adhesive (polyolefin): 30 cc maximum
- Scotch brand transparent cellophane tape: 1 meter maximum

All of the materials can be purchased at local art supply or convenience stores. Equivalent material may be substituted only with the instructor's permission. Paints, markers, flags, and other decorative items not on the list may be used as long as they are purely decorative; for example, paint cannot be used as weight or ballast.

4.09.02 Standard Project Deliverables

The following list provides the standard deliverables for your project. Your instructor may assign additional deliverables and will let you know the due date for each deliverable. When you are organizing your team effort, you can use these deliverables as the milestones to produce a Gantt chart or critical path diagram to help you stay on track and complete the project on time.

Required Drawing Deliverables

1. Conceptual sketches—alternative and final designs
2. Outline assembly drawings
   - Multiview of assembled project
   - Isometric or pictorial view of assembled project
   - Cutaway views as required for clarity
   - Sectional views as required for clarity
   - Overall dimensions only
   - Balloons to identify subassembly or part numbers and names
3. Detail drawings
   - One multiview drawing per part (isometric or pictorial)
   - All dimensions, datums, and tolerances
   - Quantity
   - Material
   - Sectional views as required for clarity
   - Isometric views as required for clarity
4. Exploded assembly drawings
   - Blow-apart pictorial view of all assemblies
   - Blow-apart pictorial view of all subassemblies
   - Balloons to identify part numbers and part and subassembly names
   - Subassemblies as required (highly recommended)
5. Bill of materials
   - List of all parts by PN, showing name, quantity, and material
   - List of all materials needed for assembly (e.g., tape and glue)

Use millimeter dimensions and proper title blocks and borders for your engineering drawings. It is recommended that all drawings be cross-checked by different people. Alternate the functions of drafter, designer, and checker. The team leader must give final approval.

Final Demonstration and Oral Presentation

Each team is expected to give an informative final presentation of its design, as well as a demonstration during the distance contest. Use descriptive graphics slides to complement the presentation. Keep the presentation short and direct.

Written Report

In your written final report detailing the project results, describe the alternatives your team considered, describe which ones you selected, and explain why you selected them. Use drawings to illustrate key points in the design process. Include the results of your product testing and include a section on what you would have done differently.

Design Project #1: Escape!

NASA is once again looking for a few good engineers. This time, the agency is seeking conceptual ideas for an escape device that would allow launchpad crews and astronauts to leave the area quickly in case of a potentially explosive, toxic, or otherwise harmful situation. The device is to be remotely launched and should be designed to place personnel as far away from the launch point as possible. The device must land safely, leaving the personnel unharmed. However, for this project, you will demonstrate the concept of your device using a hard-boiled egg.

The Mission

Your mission is to design and build a device that will launch a hard-boiled egg (USDA Grade A Large, which your team must provide) into the air and have it land as far from the launch point as possible. The device must land the egg totally intact (no cracks in the shell). The design, for example, may be composed of a mechanism for launching the egg and a device attached to the egg for lowering it slowly (like a parachute). You may surround the egg with a protective covering. However, the covering cannot penetrate the shell or be bonded to it. The function of your device will be graded on the distance from the point of launch relative to that of the other teams in the class. A stiff distance penalty will be assessed if the egg is damaged.

Design Rules

The device must be constructed out of the standard project materials listed previously, in addition to *only* the following item: one hard-boiled egg (USDA Grade A Large).

Equivalent material may be substituted only with the permission of the instructor. Any design deemed by the instructor as unsafe will not be acceptable (e.g., no sharp flying objects, no explosive devices, and no raw or rotten eggs).

More Contest Rules

- The device, including the launcher, must initially fit within a 1.0 m × 1.0 m × 1.0 m volume without external support, except for the triggering means.
- Once the egg has been launched, the device may expand to any size.
- The device must be freestanding and may not be taped, glued, or in any other way affixed to the ground.
- The device must be remotely triggered (e.g., by a string or rod). Team members (all parts of the body) must remain a minimum of 1 meter away from the device when the egg is launched.
- The device must be set up within 3 minutes; otherwise, a 3-meter distance penalty on the total distance will be assessed for every 10 seconds of overtime.
- Human power may be used to trigger the device, but not to impart motion to the egg. However, human power may be used to store energy (e.g., into the rubber bands) for any use.
- Distance is measured from the point the egg is completely clear of the launch structure (e.g., completely airborne with no attachment to the launch structure) to the point stops where any part of the device containing the egg touches a solid object connected to the ground.
- The egg must attain a distance of at least 3 meters from the launch position.
- The egg must survive the landing without cracking or sustaining any other visible signs of damage.
- Surviving eggs will be peeled and eaten by the team leader to ensure that they have not been altered in any way. If the team leader does not survive, the entire team will fail the project.
- The egg must be removed from its protective covering within 30 seconds.
- If the egg does not survive, the total distance will be recorded as zero.
- If the egg is damaged, it must be replaced in time for the next launch.
- The maximum distance from three trials will be recorded. A misfire will count as one trial.
- Spare parts are recommended and do not count in the materials inventory of the final assembly.

Design Project #2: Fast Food!

Food Service in the dormitories is experimenting with a new method of feeding students in the morning. Instead of going to the dining commons for breakfast, students will open a window before a prescribed time. Food Service will then deliver breakfast by launching it into the dorm room. That way, students do not need to wake up early just to get breakfast and can sleep late if they so choose. All students need to do is leave the window open in the evening to ensure that breakfast will be delivered in the morning. The dining service has asked your team to demonstrate a conceptual prototype of a device that will perform this function.

The Mission

Your mission is to design and build a device that will launch a bagel with cream cheese into the air and have it land as far from the launch point as possible. The device must land the bagel totally intact and unsoiled. The design, for example, may be composed of a mechanism for launching the bagel and a box around the bagel to help protect it. However, the covering cannot pierce the bagel and cannot be bonded to it. The function of your device will be partially graded on the distance from the point of launch to the point of landing relative to that of the other teams in the class. A stiff distance penalty will be assessed if the bagel is damaged or soiled: you will have to eat it.

Design Rules

The device must be constructed out of the standard project materials listed previously, in addition to *only* the following item: one plain bagel sliced horizontally and smeared with plain soft cream cheese to 1/4" average thickness. The bagel cannot be more than 24 hours old at the time of launch.

All of the materials (except the bagel with cream cheese) can be purchased at local art supply stores. Any design deemed by the instructor as unsafe will not be acceptable (e.g., no sharp flying objects, no explosive devices, and no spoiled food).

More Contest Rules

- Once the bagel has been launched, the device may expand to any size.
- The device, including the launcher, must initially fit within a 1.0 m × 1.0 m × 1.0 m volume without external support, except for the triggering means.
- The device must be freestanding and may not be taped, glued, or in any other way affixed to the ground.
- The device must be remotely triggered (e.g., by a string or rod). Team members (all parts of the body) must remain a minimum of 1 meter away from the device when the bagel is launched.
- The device must be set up within 3 minutes; otherwise, a 3-meter distance penalty on the total distance will be assessed for every 10 seconds of overtime.
- Human power may be used to trigger the device, but not to impart motion to the bagel. However, human power may be used to store energy (e.g., into the rubber bands) for any use.
- Distance is measured from the point the bagel is completely clear of the launch structure (e.g., completely airborne with no attachment to the launch structure) to the point stops where any part of the device containing the bagel touches a solid object connected to the ground.
- The bagel must survive the landing without cracking, opening, soiling, or sustaining any other visible signs of damage.
- The surviving bagel with the longest distance will be eaten by the team leader to ensure that it has not been altered in any way. If the team leader does not survive, the entire team will fail the project.
- The bagel must be removed from its protective covering within 15 seconds.
- If the bagel does not survive, the total distance will be recorded as zero.
- If the bagel is damaged, it must be replaced in time for the next launch.
- The average of the three longest distances from as many launches as can be accomplished within a single 60-second period will be recorded. Thus, you should have multiple bagels and containers ready to launch. A misfire will be considered as zero distance.
- Spare parts are recommended and do not count in the materials inventory of the final assembly.

Design Project #3: Reward!

Several problems are on the horizon for engineering graphics classes of the future. First, the classes are getting larger, requiring that lectures be held in larger rooms. This trend makes it difficult for the instructor to toss candy rewards to specific students in the class, because the instructor's throwing range is limited. Second, the course CD is apparently a flop in the market and customers from all over the country are returning their disks to the publisher. To solve both problems at the same time, someone recommended that candies be strapped to CDs and thrown together. The aerodynamic properties of the CD can be used to increase the range of the candy. This idea was immediately adopted, so here is your project.

The Mission

Your mission is to design and build a device that will launch a Hershey's chocolate Nugget (with almonds) taped to a CD so it passes through an 8′ × 8′ target frame placed as far from the launch point as possible. The target frame will be placed such that the opening faces the launcher, with the bottom of the target frame on the ground. The launching field will be relatively flat. Each team will have 3 minutes to hit the target at least once. The target distance from the launcher will be specified by the team. The single longest distance at which the target is hit will be recorded for each team. If the target is not hit on any of the tries, the final recorded distance will be recorded as zero.

Design Rules

The device must be constructed out of the standard project materials listed previously, in addition to *only* the following items:

- One genuine Hershey's chocolate Nugget (with almonds) at room temperature, still in the wrapper
- One standard 120 mm diameter optical CD
- Any design deemed by the instructor as unsafe will not be acceptable (e.g., no sharp flying objects, no explosive devices, and no spoiled food).

More Contest Rules

- The device, including the launcher, must initially fit within a 1.0 m × 1.0 m × 1.0 m volume without external support, except for the triggering means.
- The device must be freestanding and may not be taped, glued, or in any other way affixed to the ground.
- The device must be remotely triggered (e.g., by a string or rod). Team members (all parts of the body) must remain a minimum of 1 meter away from the device when the Nugget is launched.
- When the launcher is armed and ready to be triggered, it must be entirely self-supporting and stable (e.g., it does not require any external support from team members).
- The device must be set up within 3 minutes; otherwise, a 1-meter distance penalty on the total distance will be assessed for every 10 seconds of overtime.
- Human power may be used to trigger the device, but not to impart motion to the Nugget. However, human power may be used to store energy (e.g., into the rubber bands) for any use.
- Distance is measured from the point the CD is completely clear of the launch structure (e.g., completely airborne with no attachment to the launch structure) to the 8' × 8' target frame.
- The Nugget with the longest distance will be eaten by the team leader to ensure that it has not been altered in any way. If the team leader does not survive, the entire team will fail the project.
- The single longest distance at which you hit the target from as many launches as can be accomplished within a single 3-minute period will be recorded. Thus, you should have multiple Nuggets on CDs ready to launch. A misfire will be considered as zero distance.
- Spare parts are recommended and do not count in the materials inventory of the final assembly.

Design Project #4: Deploy It!

NASA is once again looking for a few good engineers. This time, the agency is seeking conceptual ideas for the deployment of structures such as antennas and solar panels in spacecraft. The device is to be self-deploying and should be designed to extend as far from the base point as possible while still remaining connected to the base point. It is possible that deployment will occur in various environments—from gravity-free space to high gravity on planets and moons. However, you will demonstrate the concept for your device in earth gravity.

The Mission

Your mission is to design and build a device that will deploy a structure to reach as far from the origin point as possible. The device must remain physically connected from the origin point to the point of furthest extension. Deployment must be automatic upon activation of a trigger mechanism, and the base structure is to be fixed to the ground. The design, for example, may be composed of a mechanism for extending a boom, cantilevered structure, or suspended structure from the origin point.

Design Rules

The device must be constructed out of the standard project materials listed previously, in addition to *only* the following item: 4 meters of 3M #2090 Long-Mask masking tape, 2" wide, to fix the base to the ground.

Any design deemed by the instructor as unsafe will not be acceptable (e.g., no sharp flying objects and no explosive devices).

More Contest Rules

- The device, without external support, must initially fit within a 1.0 m × 1.0 m base area on the ground, except for the triggering means.
- The device must initially be less than 0.5 m in height, except for the triggering means.
- Once triggered, the device must deploy automatically to its final state without further assistance and may expand to any size.
- The device must be remotely triggered (e.g., by a string or rod). The trigger may be used only to release energy from the system. The trigger cannot add energy to the system. Team members (all parts of the body) must remain a minimum of 1 meter away from the device when the device is deployed. The means of triggering must be contained on the allowable materials list but will not be counted in the final materials inventory.

- The device may be fixed to the ground only in the original base area, using only the 3M tape specified for this purpose.
- The device must be set up within 3 minutes; otherwise, a 0.1-meter distance penalty on the total distance will be assessed for every 10 seconds of overtime.
- Human power may be used to trigger the device, but not to impart motion to the structure. However, human power may be used during setup to store energy (e.g., into the rubber bands) for any use.
- Distance is measured from the forward-most point of the base prior to deployment to the forward-most connected point of the structure after deployment (in a predefined direction).
- Except within the original base area, no part of the structure may touch the ground in the final deployed position. Incidental (accidental) contact with the floor is permitted during deployment. However, prolonged contact (e.g., using the ground for support, using a wheeled carriage, or bouncing along the ground) is not permitted. No external structures (e.g., wall, ceiling, or pipes) can be used for guidance or support at any time.
- The maximum distance from three trials will be recorded. A misfire will count as one trial.
- Spare parts are recommended and do not count in the materials inventory of the final assembly.

Design Project #5: Vertical Limit

Your local fire department is looking for conceptual ideas for rescuing people in high-rise buildings. The firefighters have asked you to develop and build a test model for their review. The device is to be self-deploying and designed to extend as high as possible while still remaining connected to the ground. The structure is to be freestanding in its original and deployed states.

The Mission

Your mission is to design and build a device that will deploy from a prescribed initial size to a freestanding structure that reaches as high as possible. Deployment must be automatic upon activation of a trigger mechanism. The base structure is to be fixed to the ground. The design, for example, may be composed of a mechanism for extending a boom or truss structure.

Design Rules

The device must be constructed out of the standard project materials listed previously, in addition to *only* the following item: 4 meters of duct tape, 2" wide, to fix the base to the ground.

Any design deemed by the instructor as unsafe will not be acceptable (e.g., no sharp flying objects and no explosive devices).

More Contest Rules

- The device, without external support, must initially fit within a 1.0 m × 1.0 m base area on the ground, except for the triggering means.
- The device must initially be less than 0.5 m in height, except for the triggering means.
- Once triggered, the device must deploy automatically to its final state without further assistance and may expand to any size.
- The device must be remotely triggered (e.g., by a string or rod). The trigger may be used only to release energy from the system. The trigger cannot add energy to the system. Team members (all parts of the body) must remain a minimum of 1 meter away from the device when the device is deployed. The means of triggering must be contained on the allowable materials list but will not be counted in the final materials inventory.
- The device may be fixed to the ground only in the original base area, using only duct tape.
- The device must be set up within 3 minutes; otherwise, a 0.1-meter distance penalty on the total height will be assessed for every 10 seconds of overtime.
- Human power may be used to trigger the device, but not to impart motion to the structure. However, human power may be used during setup to store energy (e.g., into the rubber bands) for any use.
- Distance is measured from the ground to the highest point of the structure when it is fully deployed.
- No external structures (e.g., wall, ceiling, or pipes) can be used for guidance or support at any time.
- The maximum height from three trials will be recorded. The structure height must be maintained for the time it takes to measure the height (approximately 2 minutes). A misfire will count as one trial.
- Spare parts are recommended and do not count in the materials inventory of the final assembly.

Design Project #6: There 'N Back

The problem of air pollution caused by automobiles has plagued cities worldwide for decades. Several solutions have been proposed over the years, including public mass transportation systems, electric vehicles, hybrid vehicles, low-emission fuels, human-powered vehicles, and solar- or wind-powered vehicles. None of these options has been very successful to date. Consequently, it is time to develop new concepts in powered vehicles. Recently, your instructor received an anonymous e-mail stating that energy in a vehicle might be stored in elastic elements. This idea was immediately adopted, and a study was commissioned to investigate the possibility of using a large number of surplus rubber bands to power a commuter vehicle.

The Mission

Your mission is to design and build a small-scale concept vehicle that travels in a linear trajectory as far as possible and then automatically returns along the same trajectory. The device is to be powered by two rubber bands—either #62 or #64. On the day of testing, a travel line will be taped on the floor. Travel distances will be measured in the direction of the line only. Each team will have three launches of their vehicle. *The travel distance to be recorded will be the distance the vehicle travels backward along the trajectory line after the vehicle stops its forward travel.* The backward travel distance cannot exceed the forward travel distance. The single longest distance the vehicle travels backward in three attempts will be recorded for each team. If the vehicle has no forward or backward travel, the final distance will be recorded as zero.

Design Rules

The device must be constructed out of the standard project materials listed previously (and *only* the materials listed).

Any design deemed by the instructor as unsafe will not be acceptable (e.g., no sharp flying objects, no explosive devices, and no burning or combustible materials).

More Contest Rules

- The vehicle must be entirely self-contained (e.g., no external launching or guidance devices).
- The entire vehicle must initially fit within a 1.0 m × 1.0 m × 1.0 m volume without external support. After launching, the vehicle can expand to any size.
- The vehicle can be released by hand or remotely triggered. Any number of team members can be involved with the release. Once released, the vehicle cannot be touched.
- The device must be set up within 3 minutes for each launch; otherwise, a 0.5-meter distance penalty on the total distance will be assessed for every 10 seconds of overtime.
- The vehicle or a part of the vehicle must remain in contact with the ground at all times.
- Human power may be used to trigger the vehicle, but not to impart motion to the vehicle (i.e., no pushing or pulling the vehicle). However, human power may be used to store energy in the rubber bands for any use.
- Gravity cannot be used to produce motion (e.g., no launching from a ramp).
- Travel distance is measured in the direction parallel to the length of the path. If the vehicle hits the side wall of the hallway and stops, all vehicle motion is considered finished.
- The final recorded travel distance will be the distance from the closest point of forward travel (from the starting line) on the vehicle to the closest point of return travel (from the forward mark) on the vehicle.
- Objects expelled from the vehicle are still considered a part of the vehicle for measurement of travel distance.
- Spare parts are recommended and do not count in the materials inventory of the final assembly.

SECTION TWO

MODERN DESIGN PRACTICE AND TOOLS

The widespread availability of computers has made three-dimensional modeling the preferred tool for engineering design in nearly all disciplines. Solid modeling allows engineers to easily create mathematical models parts and assemblies, visualize and manipulate these models in real time, inspect how they mate with other parts, and calculate their physical properties. The geometry of a part, as well as allowable errors, is determined to a large degree on the fabrication method used to make it. Thus, a basic understanding of fabrication processes is important for creating three-dimensional designs. There are many available manufacturing processes, and all have advantages and disadvantages. Three-dimensional modeling also is used as the foundation for many sophisticated computational analysis techniques such as mass properties, interference checking, and finite element analysis. The ease with which this can be done has moved what was formerly a complicated analytical process performed by specialists into the realm of standard practice by many design engineers as part of the design process.

CHAPTER 5

SOLID MODELING

OBJECTIVES

After completing this chapter, you should be able to

- Introduce solid modeling as an engineering design graphics tool
- Explain how solid models are created
- Show how parts and models can be decomposed into features
- Develop strategies for creating a solid model
- Explain how solid models support the entire product life cycle

5.01 INTRODUCTION

Solid modeling is a computer-based simulation that produces a visual display of an object as if it existed in three dimensions. **Solid models** aid in forming a foundation for the product development process by providing an accurate description of a product's geometry and are used in many phases of the design process and life cycle of the product. This chapter will focus on methods for creating robust solid models of mechanical parts; however, these methods can be applied to other domains as well.

Solid models are created with specialized software that generates files for individual as well as assembled parts. These models are then used in a variety of applications throughout the design and manufacturing processes, as shown schematically in Figure 5.01. During the product concept stage, solid models are used to visualize the design. As the product is refined, engineers use solid models to determine physical properties such as the strength of the parts, to study how mechanisms move, and to evaluate how various parts fit together. Manufacturing engineers use solid models to create manufacturing process plans and any special tools or machines needed to fabricate or assemble parts. Solid models also can be used to generate formal engineering drawings to document the design and communicate details of the design to others. People responsible for the product life cycle may depend on solid models to help create images for service manuals and disposal documentation. Even sales and marketing functions use graphics generated from solid models for business presentations and advertising. Thus, it is very important not only to learn how to create solid models but also to understand how others will use the models. Solid models must be built with sound modeling practices if they are to be useful in downstream applications. In this chapter, you will learn how to create robust solid models that not only look like the real thing but also support the entire product life cycle. You also will learn about the history of CAD tools and the importance of solid modeling as part of an engineering design graphics system.

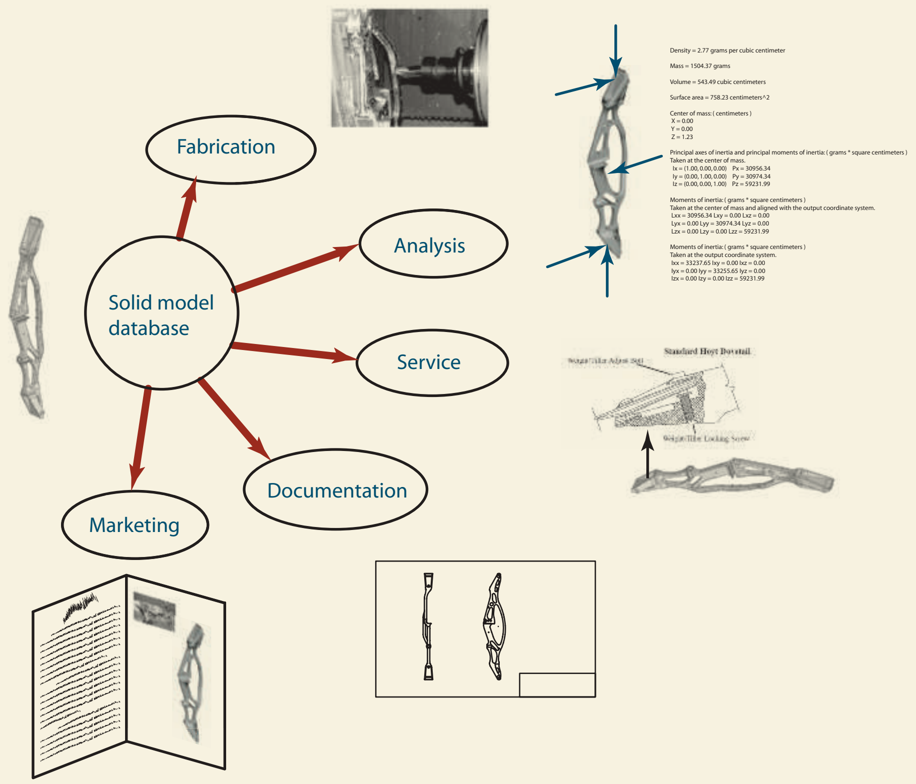

FIGURE 5.01. Uses for a solid model database.

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5.02 Tools for Developing Your Idea

Many tools have been developed for creating accurate images of an object as an aid in analyzing its function, recording its history, or visualizing its appearance. One of the simplest tools is a pencil, which is used to make sketches of an object on paper. More formal tools include rulers, protractors, compasses, and various types of manually operated drafting machines. These tools are used to make more accurate, standardized drawings according to precise rules and conventions, as discussed in a previous chapter.

CAD systems are among the most sophisticated graphics and design tools available to engineers and designers. Many types of CAD systems are on the market today. The simplest systems are general purpose drawing or drafting packages that can be used to create 2-D images, similar to the way pencil images are created on paper (except faster and easier). More complex packages allow you to create simulations of 3-D models that can be used not only to generate conventional 2-D drawings of a design but also to create 3-D images for visualization. The core of a CAD model is a geometric **database**. The database includes information about the geometry and other engineering properties of an object. The CAD software uses the database to display the model and to conduct further engineering analysis. A short discussion of CAD history will demonstrate how these systems evolved and provide some insight into the modeling processes used by designers with various CAD systems.

5.02.01 Two-Dimensional CAD

The first CAD systems were developed in the late 1960s at a time when computational resources were very limited. Graphics displays had refresh times measured in seconds, and the data storage capabilities were limited to fractions of a kilobyte. As a result, only very simple models could be created. Those models were basically electronic versions of conventional pencil-and-paper drawings. The user had to specify the location of each vertex in the model for the particular view desired. If the user wanted another view, she had to start from scratch, just as you would do if you were creating a drawing on paper.

Since CAD models are used to define the geometry or shape and size of objects, the models are composed of geometric entities. In the earliest CAD models, those entities represented the edges of the object, just as you would draw the edges of an object with a pencil. In fact, at that time there was very little distinction between a 2-D drawing of an object and a 2-D CAD model of the object. The 2-D CAD model was simply a database that contains the edges of the object, dimensions, text, and other information that you would find on the drawing, but in electronic form instead of on paper.

The simplest geometric entity is a point. Points in two dimensions are defined according to their location in a coordinate system, usually Cartesian coordinates (x, y). In a CAD system, the coordinate system represents locations on the "paper," or computer screen. Points are generally used to locate or define more complex entities, such as the endpoints of a line segment or the center of a circle. A point on an entity that marks a particular position, such as the endpoint of a line segment or the intersection of two entities, is referred to as a **vertex**.

Two-dimensional geometric entities are those that can be created as a path or curve on a plane. Those entities include lines, circles, **splines**, arcs, polygons, and conic sections, which are shown in Figure 5.02. The entities can be assembled to create images of a desired object as it would be seen from different viewing directions, as shown in Figure 5.03.

One weakness of 2-D CAD systems is that to visualize and manipulate a 3-D model of the object, you must mentally assemble and reform the 2-D views. Another weakness of 2-D CAD (and pencil drawings) is that it is possible to create images of objects that are physically impossible to build, such as the three-pronged fork and the triangle shown in Figure 5.04.

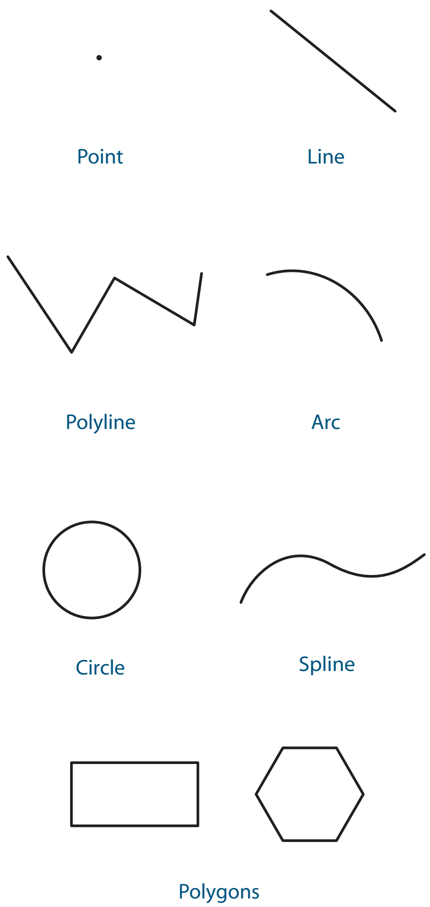


FIGURE 5.02. Some entities used for 2-D CAD.

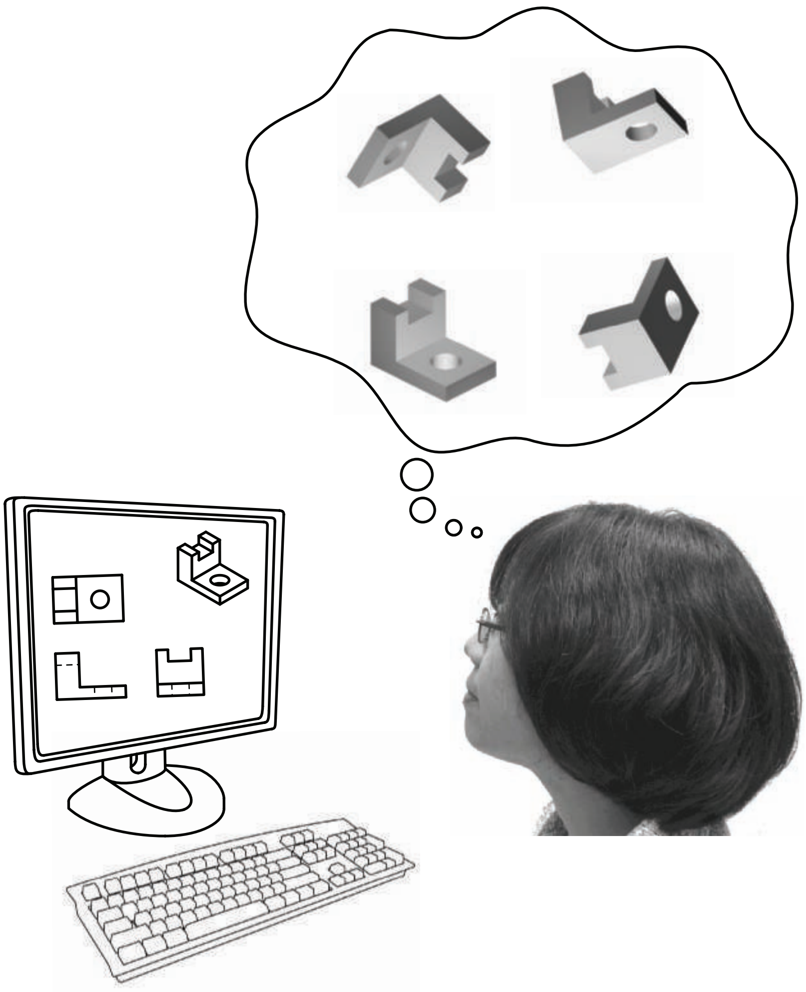
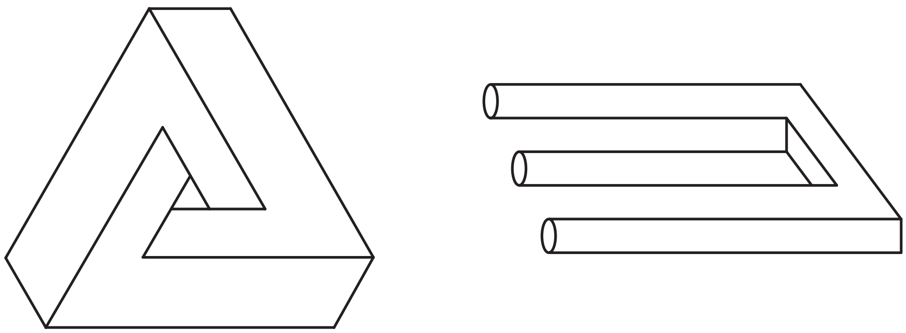


FIGURE 5.03. With 2-D models, visualization of a 3-D object must be done mentally.

FIGURE 5.04. Impossible 3-D objects can be drawn with 2-D elements.



5.02.02 Wireframe Modeling

In the early 1980s, 2-D CAD drafting packages evolved into 3-D modeling systems. In these newer systems, 3-D information could be included for the model. The computer could then perform the calculations needed to create views of an object as if it was seen from different directions. These systems were still limited to using entities such as lines, circles, and arcs; but the assemblage of the entities was no longer restricted to being on a single plane. The geometric entities were represented in a 3-D database

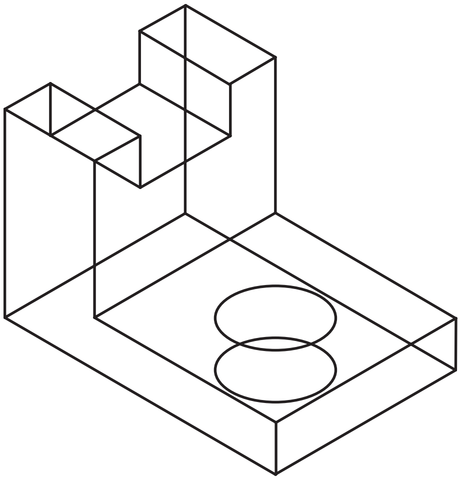


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FIGURE 5.05. A wireframe model of a 3-D object.

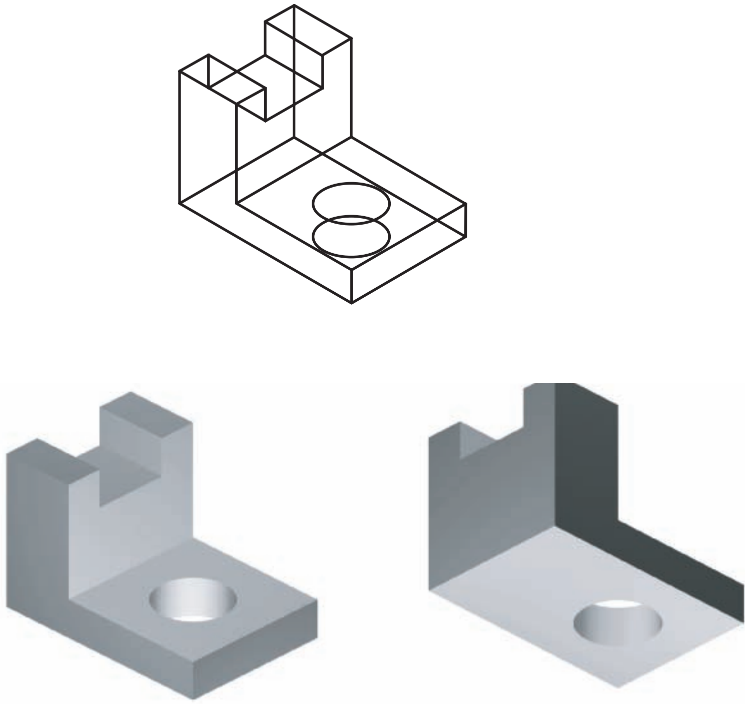

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FIGURE 5.06. Two possible view interpretations of the same wireframe model.

within a 3-D coordinate system with *x*-, *y*-, and *z*-coordinates. Since simple curve or path entities were used to define the edges of an object, such models were called **wireframe models**. Think of a wireframe as being similar to a box kite. A wireframe model of a bracket is shown in Figure 5.05.

Wireframe models are still very limited in their representation of parts. The same wireframe model can represent an object from two different viewing directions, as demonstrated in Figure 5.06. Thus, the models were sometimes difficult to visualize as solids. Some models were ambiguous, being interpreted by viewers as different objects. Look at Figure 5.07 and try to imagine the solid object represented by the wireframe model in (a). Can you visualize the shape of the object? Does this figure represent more than one object? When the hidden edges are removed and the surfaces shaded, as in (b) and (c), it is much easier to see the desired shape.

Another problem with wireframe CAD systems was that the geometry was limited to shapes with simple planar and cylindrical surfaces. Also, parts with cylindrical features, such as the one shown in Figure 5.08, generated wireframe models that did not show the optical limit or silhouette of the cylindrical surface. Even so, wireframe modeling represented a tremendous advance in technology compared to the drafting board. It is estimated that more than 75 percent of all common machined parts can be accurately represented using 3-D wireframe models.

5.02.03 Surface Modeling

As computers became more powerful and data storage capabilities increased, surface modeling techniques were developed. With a **surface model**, the designer could display the surfaces of a part, such as those shown in Figure 5.09, and use the model to perform engineering analyses such as calculating the part's mass properties. Such models also could be used to generate computer programs that controlled the fabrication of parts, for example, on a computer-controlled cutting machine called a mill.

FIGURE 5.07. The wireframe model in (a) can represent the object in (b) or the object in (c).

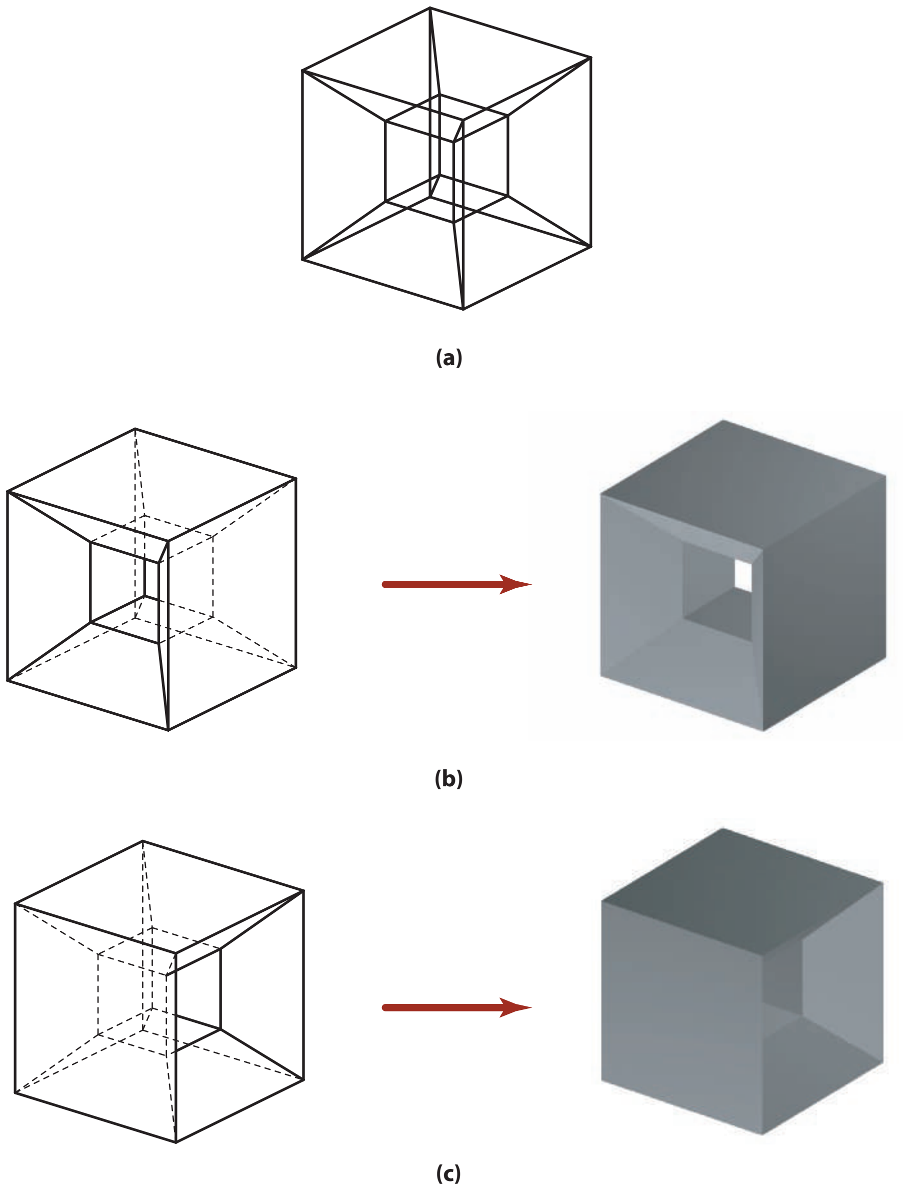

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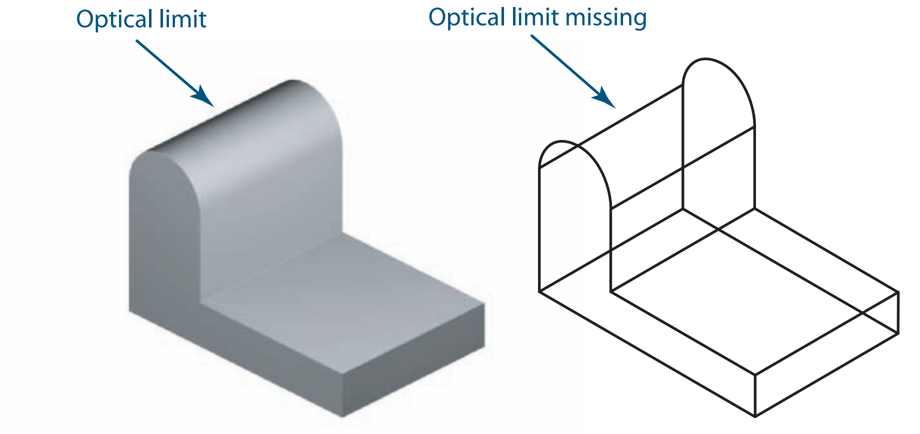
FIGURE 5.08. Wireframe models do not show the optical limit of a curved surface.



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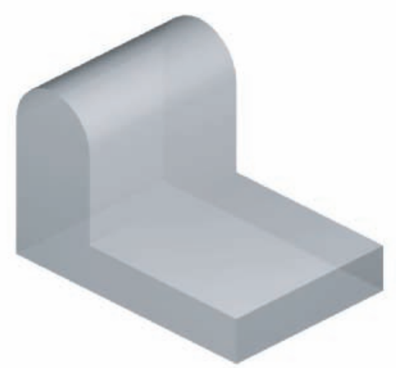

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FIGURE 5.09. A surface model with semitransparent surfaces to reveal detail.

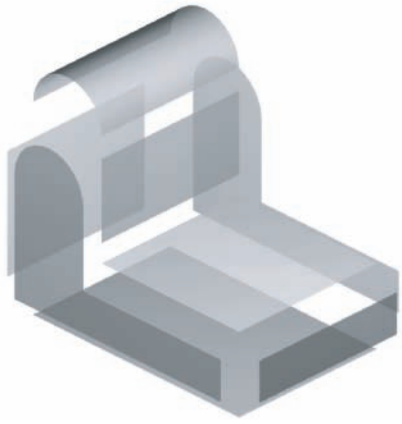

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FIGURE 5.10. A surface model exploded to show individual surfaces.

Surface models evolved from wireframe models by mathematically describing and then displaying surfaces between the edges of the wireframe model. Thus, a surface model is a collection of the individual surfaces of the object. This modeling method is called **boundary representation**, or **b-rep**, because the surfaces "bound" the shape. The bounding entities of a simple part created using boundary representation are shown in Figure 5.10. The bounding entities can be planes, cylinders, and other surfaces in three dimensions. These surfaces are in turn bounded by simpler curve entities such as lines and arcs.

The use of surface models eliminates most of the problems with visual ambiguity encountered with wireframe models.

5.02.04 Solid Modeling

Solid models are visually similar to surface models, so it is sometimes difficult to distinguish between them. With a solid model, however, the software can distinguish between the inside and outside of a part and the objects can have thickness. Thus, the information stored in the 3-D database is sufficient to distinguish between an empty shoe box and a brick. The software also easily computes information such as the object's volume, mass, center of mass, and other inertial properties. Early solid models, developed in the late 1980s, were made using a technique known as **constructive solid geometry (CSG)**. CSG models are composed of standard building blocks in the form of simple solids such as rectangular prisms (bricks), cylinders, and spheres, called **primitives**. The shapes are easy to define using a small number of dimensions. Figure 5.11 shows some of these basic solids. To create more complex solids, the primitives are assembled using Boolean operations such as addition (union), subtraction (difference), and interference. Examples of these operations are shown in Figure 5.12.

Surface and CSG models were very powerful tools for design, but their early versions were rather cumbersome to use. As computational resources improved, so did the capabilities of modeling software. Increasingly more sophisticated modeling methods, such as creating a solid model by moving or rotating a closed 2-D outline on a path through space, as shown in Figure 5.13, were developed. Further developments included software tools for taking many individual solid model parts and simulating their assembly into a larger structure, as explained in Chapter 1, and for easily creating formal engineering drawings for parts and assemblies from their solid models.

A more accurate and efficient modeling tool called **feature-based solid modeling** was developed in the mid-1990s. This modeling method permitted engineers and

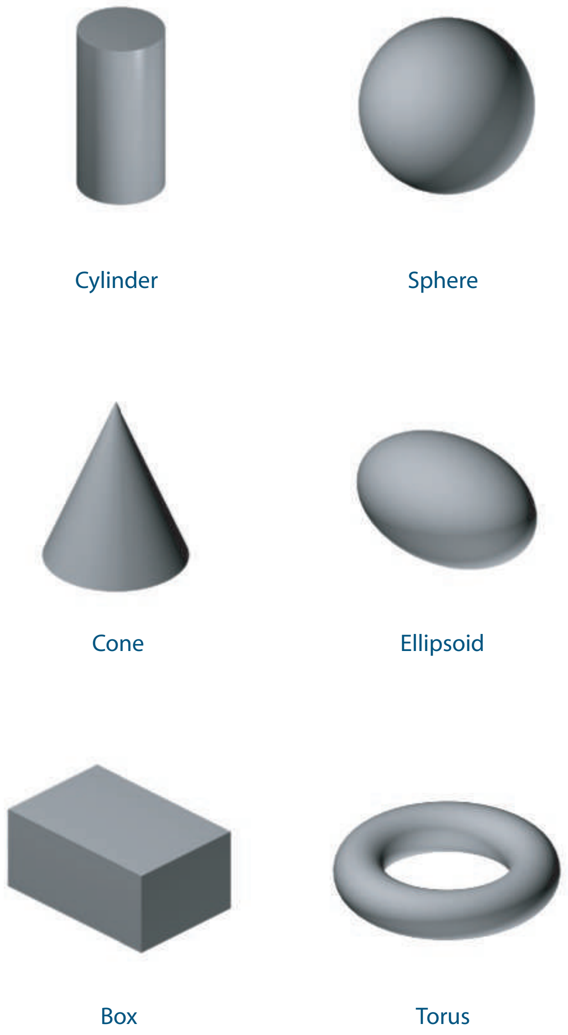

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FIGURE 5.11. Some 3-D primitives used in solid modeling.

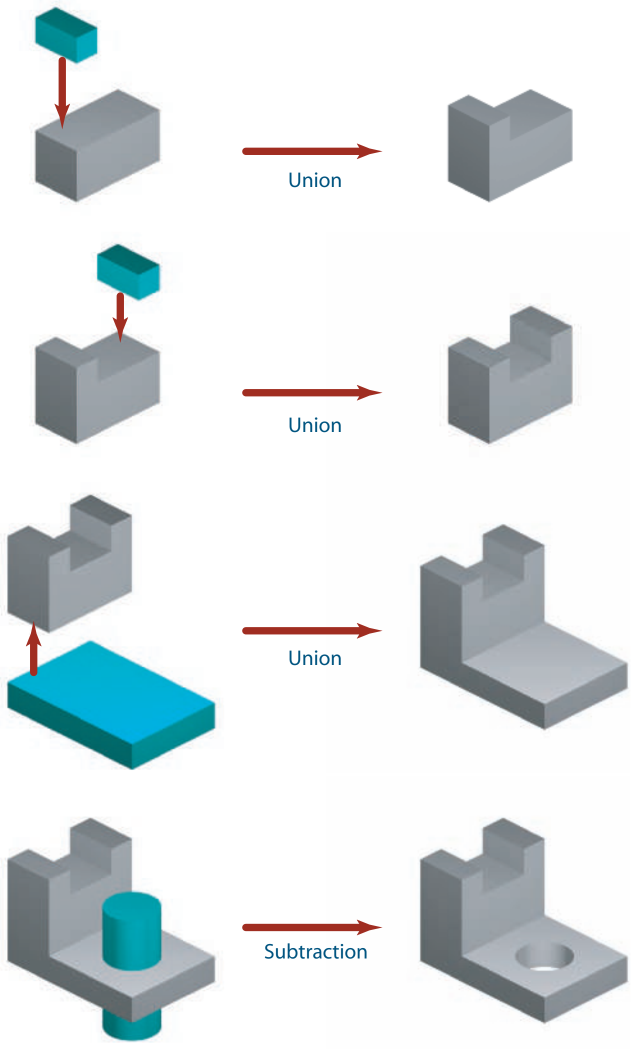

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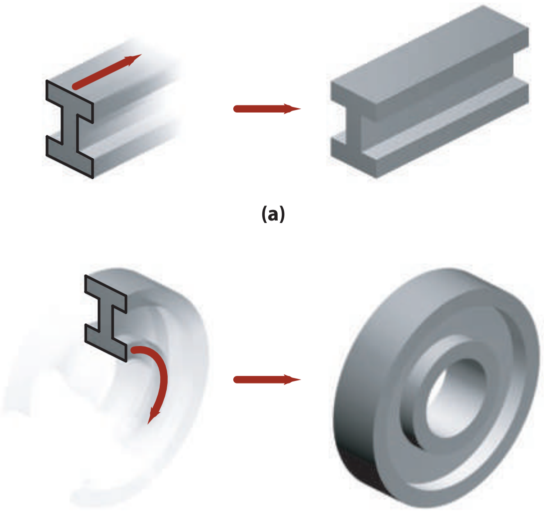
FIGURE 5.12. Steps in using solid primitives to build a more complicated solid model using Boolean operations.

FIGURE 5.13. Solids created by (a) moving and (b) revolving a 2-D outline through space.



(a)

(b)

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designers to create a more complex part model quickly by adding common features to the basic model. Features are 3-D geometric entities that exist to serve some function. One common and easily recognizable feature is a hole. Holes in a part exist to serve some function, whether it is to accommodate a shaft or to make the part lighter. Other features, such as bosses, fillets, and chamfers, will be defined later in this chapter.

Parametric solid modeling is a form of feature-based modeling that allows the designer to change the dimensions of a part or an assembly quickly and easily. Since parametric feature-based solid modeling is currently considered the most powerful 3-D CAD tool for engineers and designers, the remainder of this chapter will be devoted to this modeling method.

5.03 A Parametric Solid Model

So how does one go about creating a parametric feature-based solid model? In this section, a very simple model will be created to demonstrate basic concepts. More detail and sophistication will be presented in subsequent sections of this chapter. The tools that you need to create a parametric model are solid modeling software and a computer that is powerful enough to run the software. As you create the model, the software will display an image of the object which can be turned and viewed from any direction as if it actually existed in three dimensions.

Using the mouse and keyboard, you will interact with the software through a **graphical user interface (GUI)** on the computer's display device (i.e., the computer monitor). The GUI gives you access to various tools for creating and editing your models. GUIs differ slightly in different solid modeling software. However, most of the packages share some common approaches. When creating a new model (i.e., with nothing yet existing), you will probably be presented with a display of 3-D Cartesian coordinate x-, y-, and z-axes and the three **primary modeling planes**, which are sometimes called the **principal viewing planes** or **datum planes**. These planes help you visualize the xy-, yz-, and xz-planes and are usually displayed from a viewing direction from which all three planes can be seen, as shown in Figure 5.14.

Nearly all solid modelers use 2-D **sketches** as a basis for creating solid features. Sketches are made on one of the planes of the model with a 2-D sketching editor

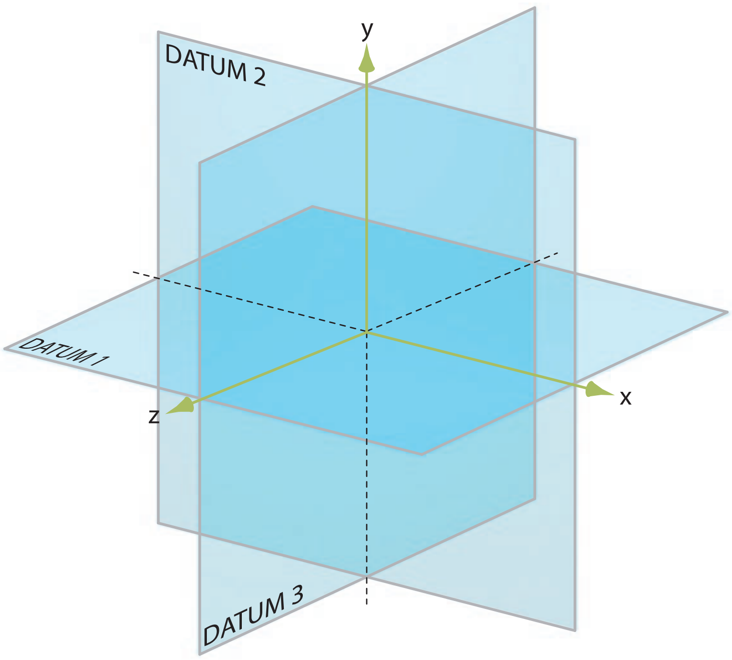

FIGURE 5.14. The primary modeling planes for solid modeling.

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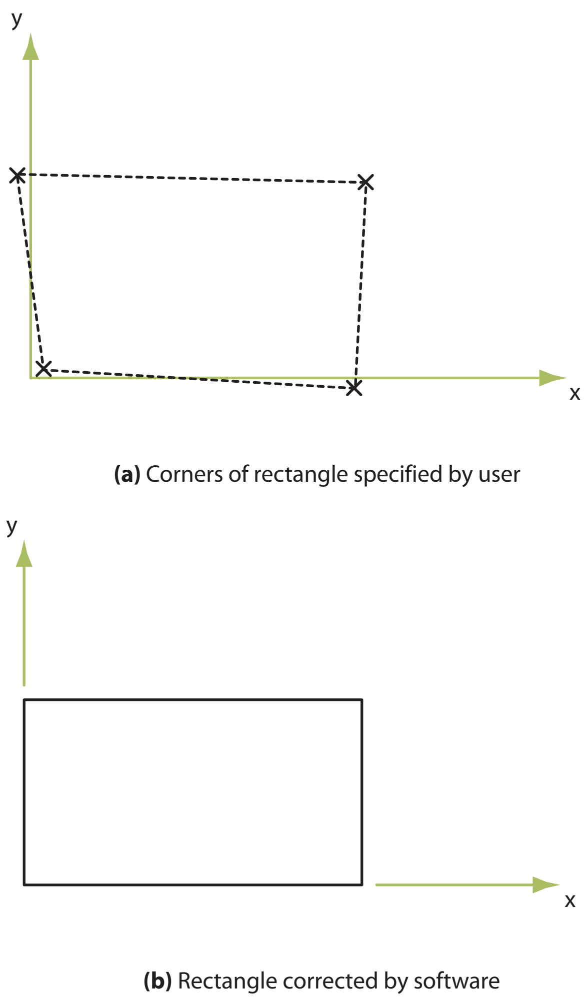
similar to a drawing editor found on most 2-D CAD drafting software. When you begin a new model, you often make a sketch on one of the basic modeling planes. When the sketching plane is chosen, some modelers will reorient the view so you are looking straight at the 2-D sketching plane. You can then begin sketching.

Line segments are usually inserted using mouse clicks, as shown in Figure 5.15a. A sketch is initially created without much attention being paid to precise dimensions and exact orientations of the different segments. For convenience, the **sketching editor** in most solid models automatically corrects sloppy sketches by making assumptions about the intended geometry. For example, if a line segment is sketched almost vertically or almost horizontally, the sketching editor will force the line into a vertical or horizontal orientation. Figure 5.15a shows a sketch of a rectangle created by clicking the four corners, or vertices; Figure 5.15b shows the cleaned-up sketch after the sketching editor corrects the user input and reorients the line segments.

5.03.01 Valid Profiles

Before a solid feature can be created by extrusion or rotation, the final profile of the shape must be a closed loop. Extra line segments, gaps between the line segments, or overlapping lines create problems because the software cannot determine the boundaries of the solid in the model. Samples of proper and improper profiles are shown in Figure 5.16.

FIGURE 5.15. 2-D sketching.



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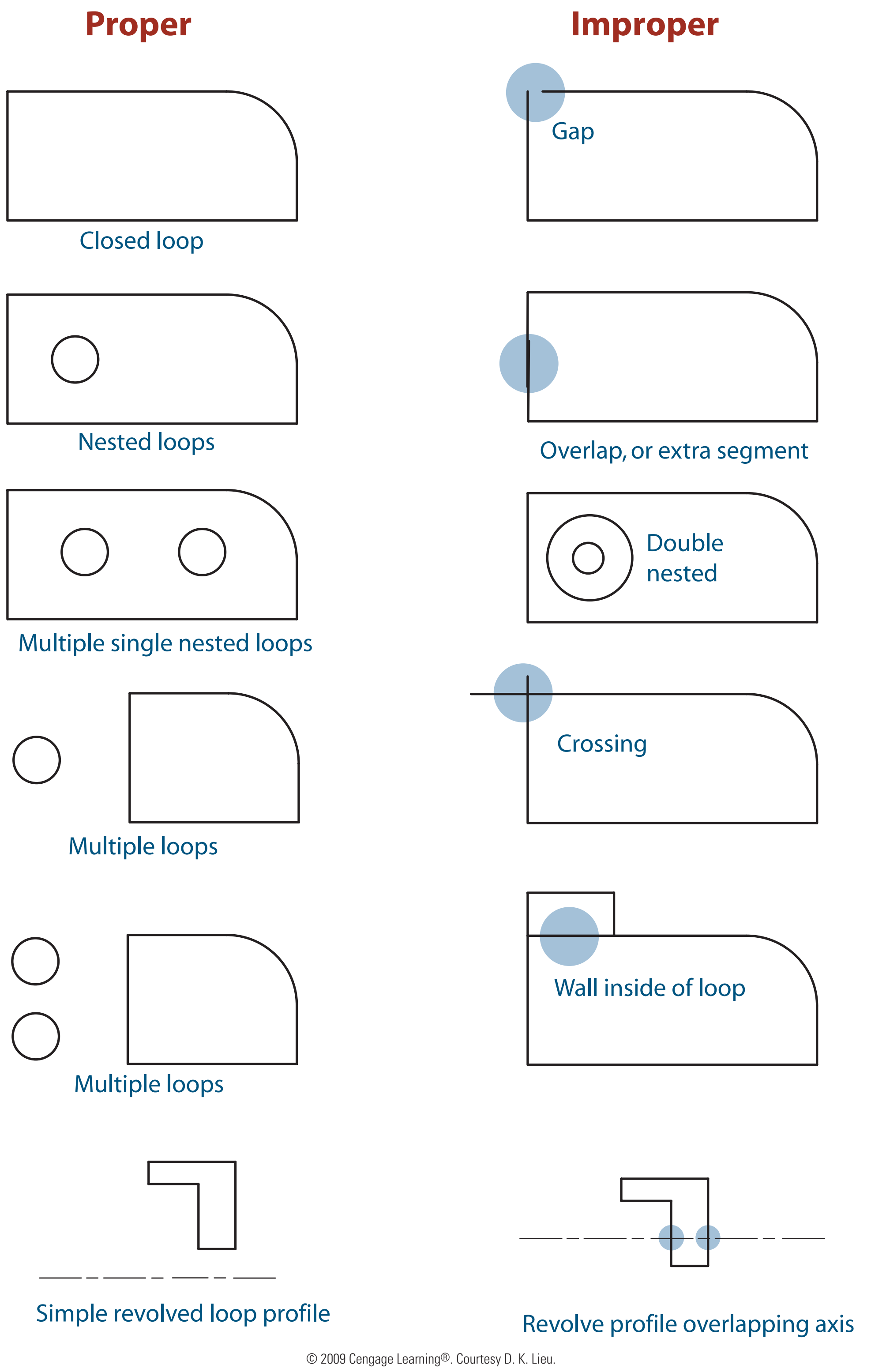
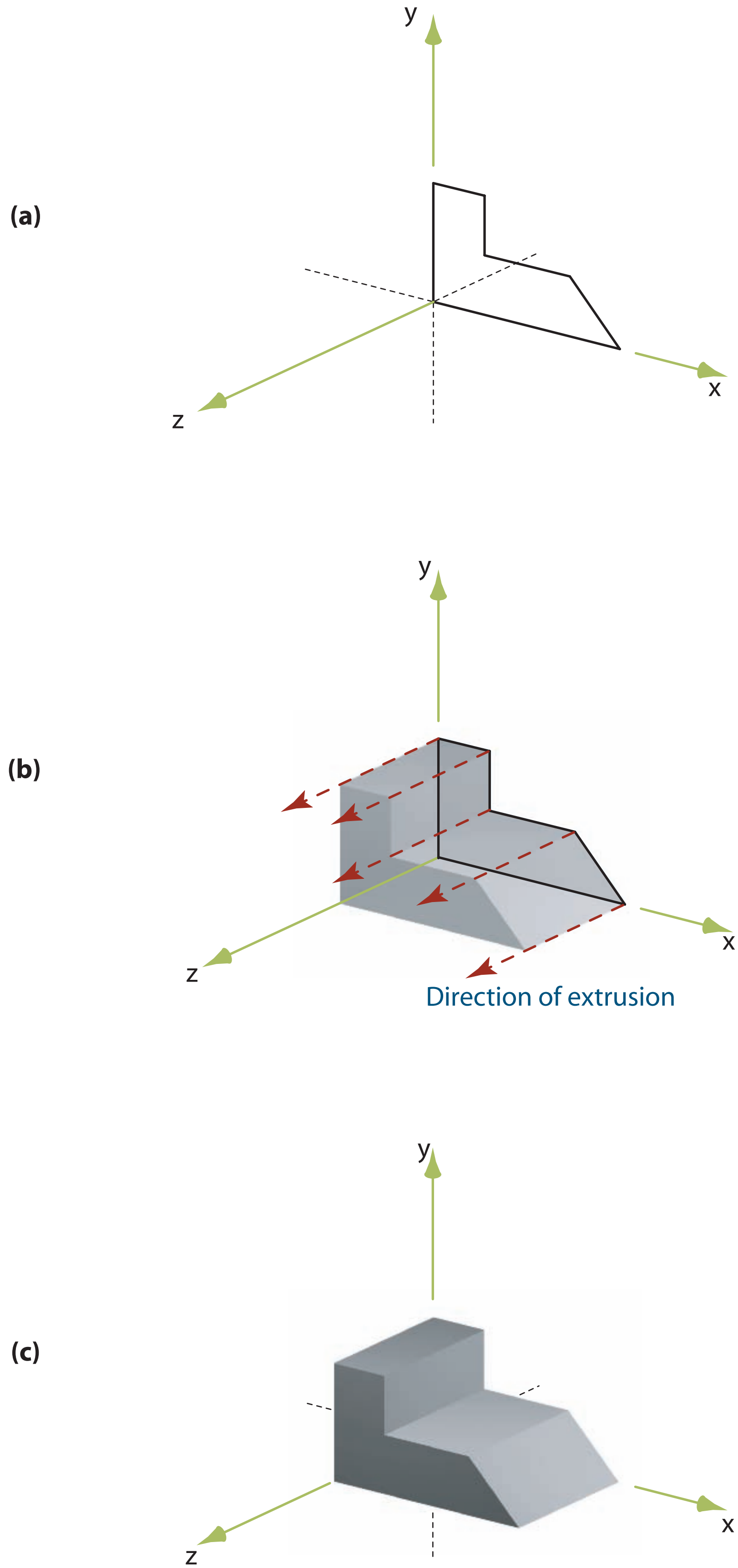


FIGURE 5.16. Examples of proper and improper profiles.

5.03.02 Creation of the Solid

A completed sketch that is used to create a solid is called a **profile**. A simple solid model can be created from the profile by a process known as **extrusion**, as shown in Figure 5.17. Imagine the profile curve being pulled straight out of the sketching plane. The solid that is formed is bound by the surfaces swept out in space by the profile

FIGURE 5.17. A solid created by extrusion of a 2-D profile.



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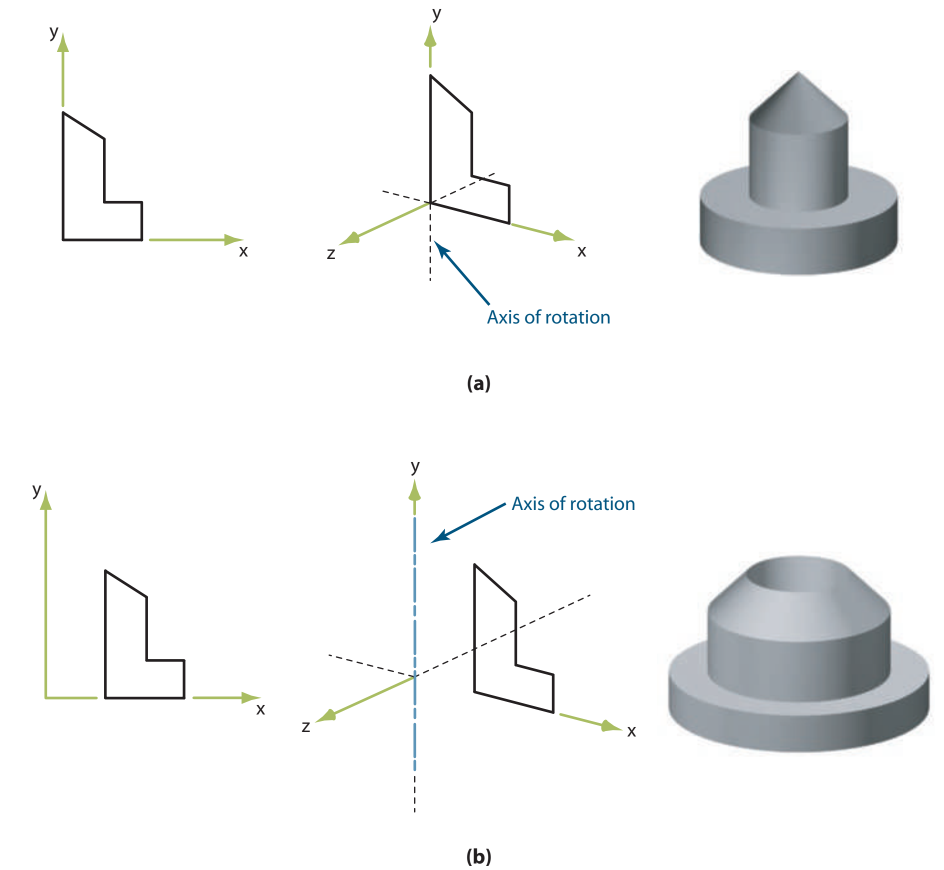


FIGURE 5.18. A solid created by rotation of a 2-D profile, with the axis on the profile in (a) and with the axis off the profile in (b).

as it is pulled along the path. Both the geometry of the profile and the length of the extrusion must be specified to define the model fully.

A different model can be created from the profile by a process called revolution. To create a **revolved solid**, a profile curve is rotated about an axis. The process is similar to creating a clay vase or bowl on a potter's wheel. The profile of a revolved part is also planar, and the axis of revolution lies in the profile plane (sketching plane). One edge of the sketch may lie along the axis of revolution, as shown in Figure 5.18a; or the sketch may be offset from the axis, as shown in Figure 5.18b. It is important to make sure that the profile does not cross over the axis of revolution. This would create a self-intersecting model (i.e., a solid created inside another solid), which most solid modeling software interpret to be a geometric error. The geometry of the profile and the angle of rotation must be specified to define the model fully. The models shown in Figure 5.18 are revolved through a full 360 degrees.

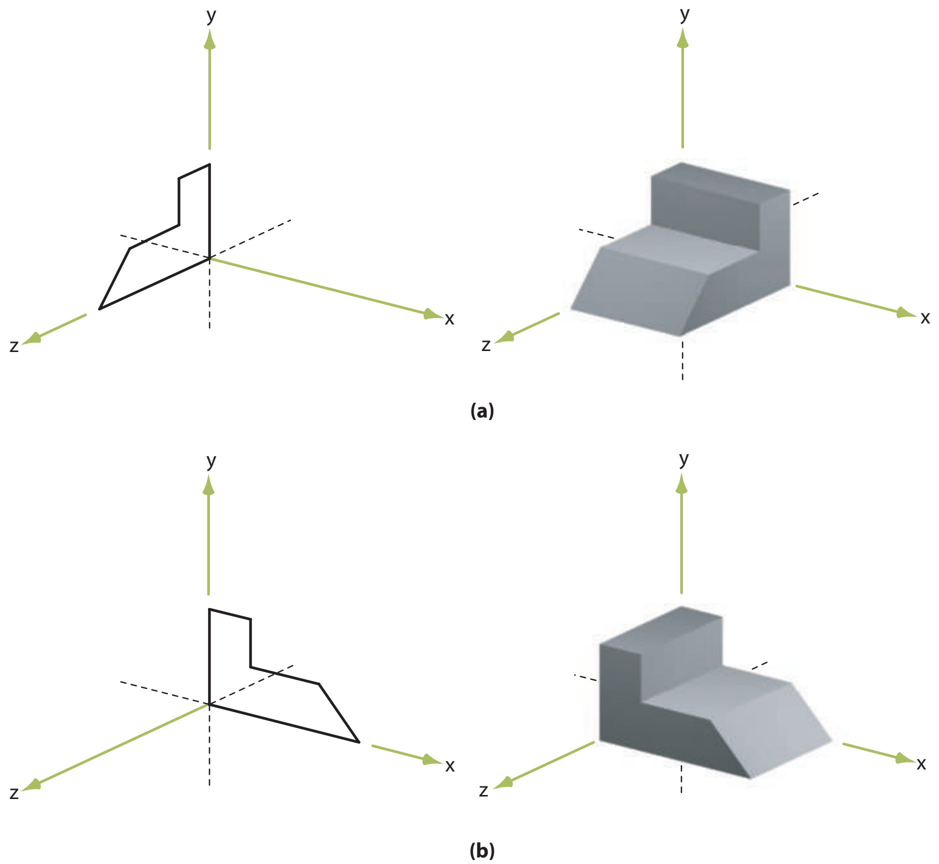
5.04 Making It Precise

Before a part can be submitted for analysis or fabrication, the sizes and locations of all of its features must be completely specified. To see how this is done, let's back up a few steps in our discussion of the creation of the model.

5.04.01 Orientation of the Sketch

Before you begin to create the first extrusion or revolution, you must decide where to place the part in the space relative to the xyz-coordinate system. With the model shown in Figure 5.17, the initial sketch was placed on one of the basic modeling

FIGURE 5.19. The same profile made in different sketching planes produces the same object but in different orientations. In (a), the profile is made in the yz-plane; and in (b), the profile is made in the xy-plane.



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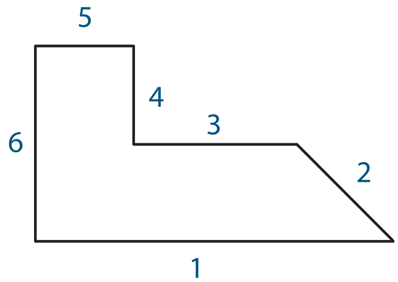
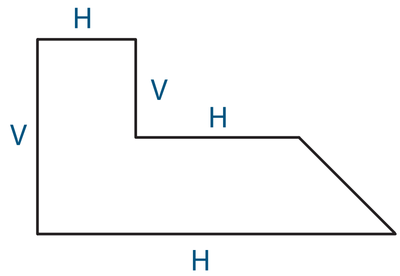
planes. If the sketch was placed on one of the other basic modeling planes instead, the model would have the same geometry but with a different orientation in space, as shown in Figure 5.19.

5.04.02 Geometric Constraints

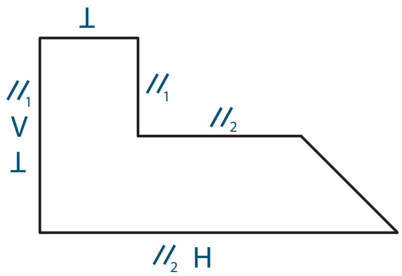
Formally, **constraints** are the geometric relationships, dimensions, or equations that control the size, shape, and/or orientation of entities in the profile sketch and include the assumptions that the CAD sketcher makes about your sloppy sketching. Constraints that define the size of features will be discussed in the following section. The previous section provided a few examples of **geometric constraints** that were applied to a simple sketch: lines that were drawn as nearly horizontal were assumed to be horizontal, and lines that were drawn as nearly vertical were assumed to be vertical. Those assumptions reduce the number of coordinates needed to specify the location of the endpoints. Some solid modelers require you to constrain the profile fully and specify the sizes and locations of all of its elements before allowing the creation of a solid feature; others allow more free-form sketching. Geometric constraints may be either implicitly defined (hidden from the designer) or explicitly displayed so you can modify them. A set of geometric constraints is not unique, as demonstrated in Figure 5.20. In this example, a set of geometric constraints that restricts some lines to being horizontal or vertical is equivalent to another set of constraints that restricts some lines to being either parallel or perpendicular to each other.

FIGURE 5.20. The line segments in a profile are numbered in (a). The implied geometric constraints for each segment are shown in (b), and an equivalent set of applied constraints is shown in (c). A letter or symbol beside a segment signifies the type of geometric constraint applied to it.





Segment	Constraint
1	Horizontal
3	Horizontal
4	Vertical
5	Horizontal
6	Vertical



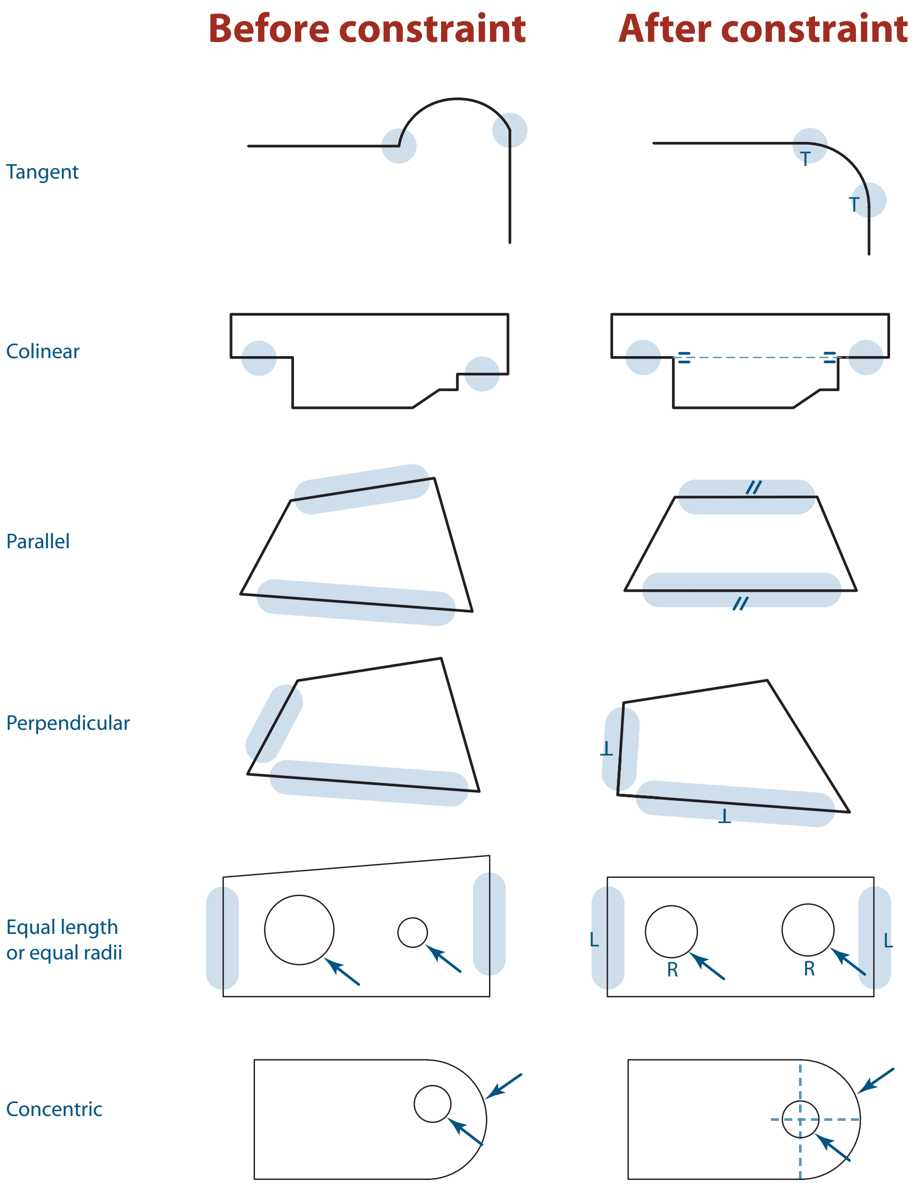
Segment	Constraint
1	Horizontal, parallel to 3
3	Parallel to 1
4	Parallel to 6
5	Perpendicular to 6
6	Vertical, parallel to 4, perpendicular to 5

Geometric constraints specify relationships between points, lines, circles, arcs, or other planar curves. The following is a list of typical geometric constraints. The results of applying the constraints are shown graphically in Figure 5.21.

- Coincident—forces two points to coincide
- Concentric—makes the centers of arcs or circles coincident
- Point on Line—forces a point to lie on a line
- Horizontal/Vertical—forces a line to be horizontal/vertical
- Tangent—makes a line, a circle, or an arc tangent to another curve
- Colinear—forces a line to be colinear to another line
- Parallel—forces a line to be parallel to another line
- Perpendicular—forces a line to be perpendicular to another line
- Symmetric—makes two points symmetric across a centerline

The sketching editors in most solid modeling software are usually configured to try to interpret the user's sketching intent such that certain constraints are created automatically. In addition to adjusting nearly horizontal or vertical lines into true horizontal or vertical lines, if two lines are nearly perpendicular or parallel or an arc and a line are nearly tangent at the common endpoint, the sketching editor will impose the assumed geometric relationship. These automatically applied geometric constraints can be changed at a later time if desired.

FIGURE 5.21. Geometric constraints commonly found in sketching editors.



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5.04.03 Dimensional Constraints

Each of the 2-D entities in the profile must have size and position. **Dimensional constraints** are the measurements used to control the size and position of entities in your sketch. Dimensional constraints are expressed in units of length, such as millimeters, meters, inches, or feet. For example, look at the profile in Figure 5.22, which shows dimensional constraints that define its size. If you, the designer, do not fully specify all of the necessary information, the software will default to some value that you may not want. It is better if you control the model, rather than have the software assign assumed parameters and conditions to the model.

Dimensional constraints can be created interactively while you are sketching, but also automatically as a result of a feature operation, an extrusion, or a revolution. There are three principal types of dimensional constraints:

- Linear dimensional constraints define the distance between two points, the length of a line segment, or the distance between a point and a line. Linear dimensions can be measured horizontally or vertically or aligned with the distance being measured.

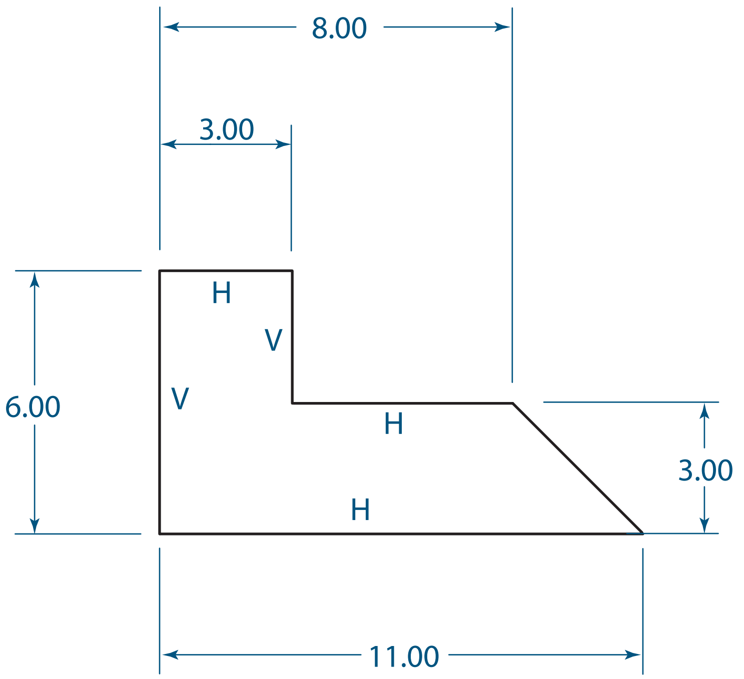

FIGURE 5.22. A profile fully constrained with geometric and dimensional constraints.

- Radial and diametral dimensional constraints specify the radius or diameter of an arc or a circle.
- Angular dimensional constraints measure the angle between two lines. The lines do not need to intersect, but they cannot be parallel.

5.04.04 Uniqueness of Constraints

A set of dimensional constraints is not unique. It is possible to apply a different set of dimensional constraints on a profile to produce exactly the same geometry, as shown in Figure 5.23.

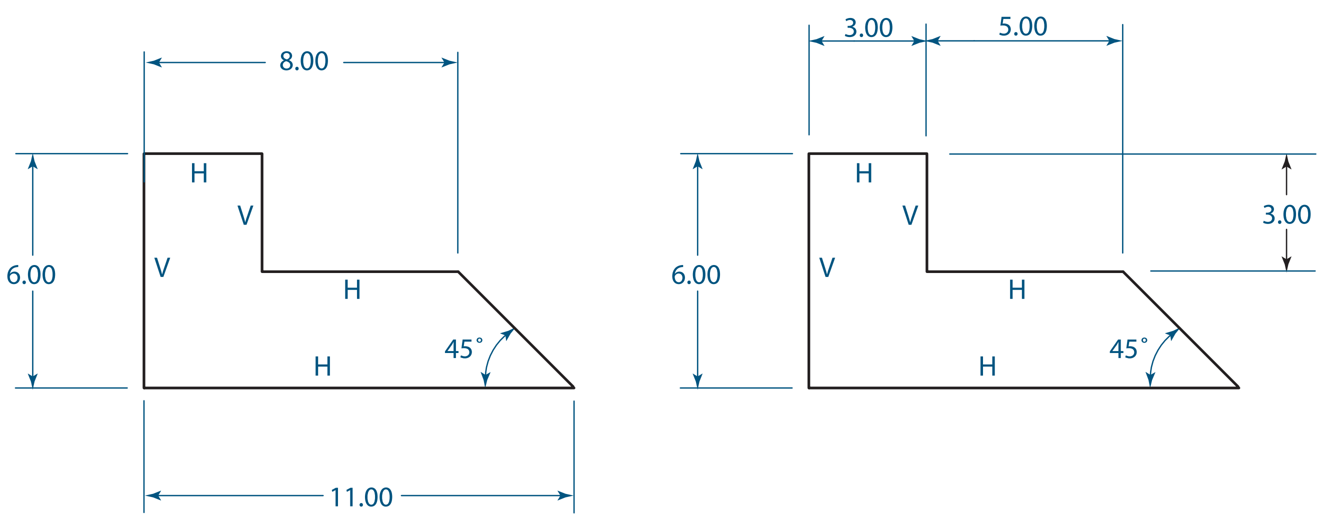

FIGURE 5.23. Two different sets of dimensional constraints that can be used to define the same geometry.

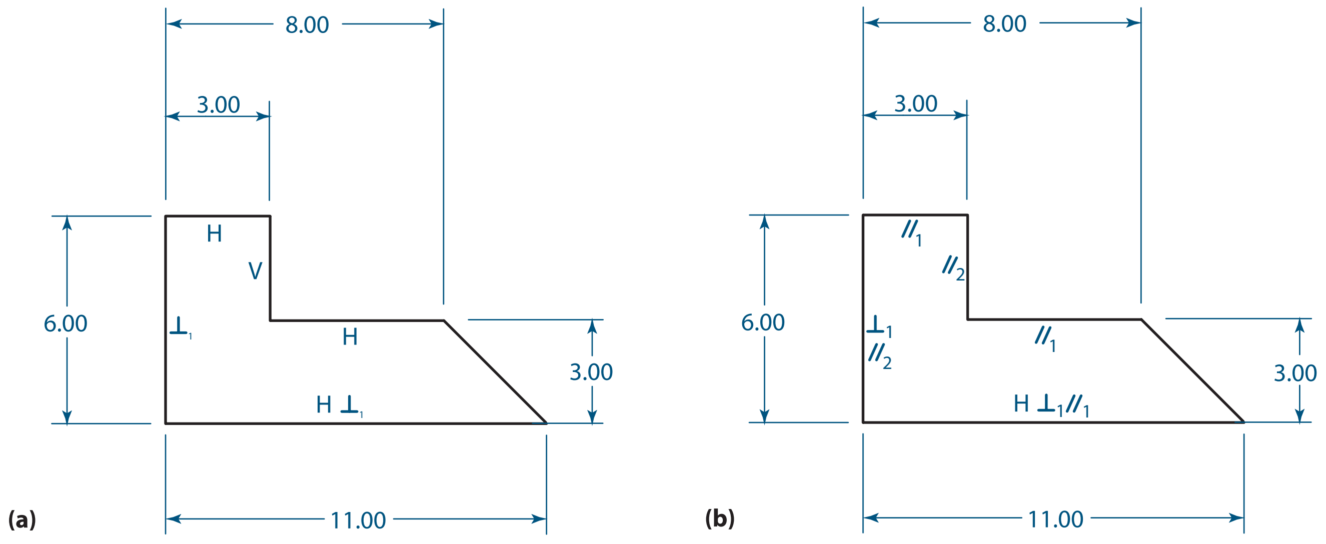

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FIGURE 5.24. Two different sets of geometric constraints that define the same geometry.

Combinations of dimensional and geometric constraints also are not unique. It is possible to have different combinations of geometric constraints and dimensional constraints define exactly the same geometry, as shown in Figure 5.24.

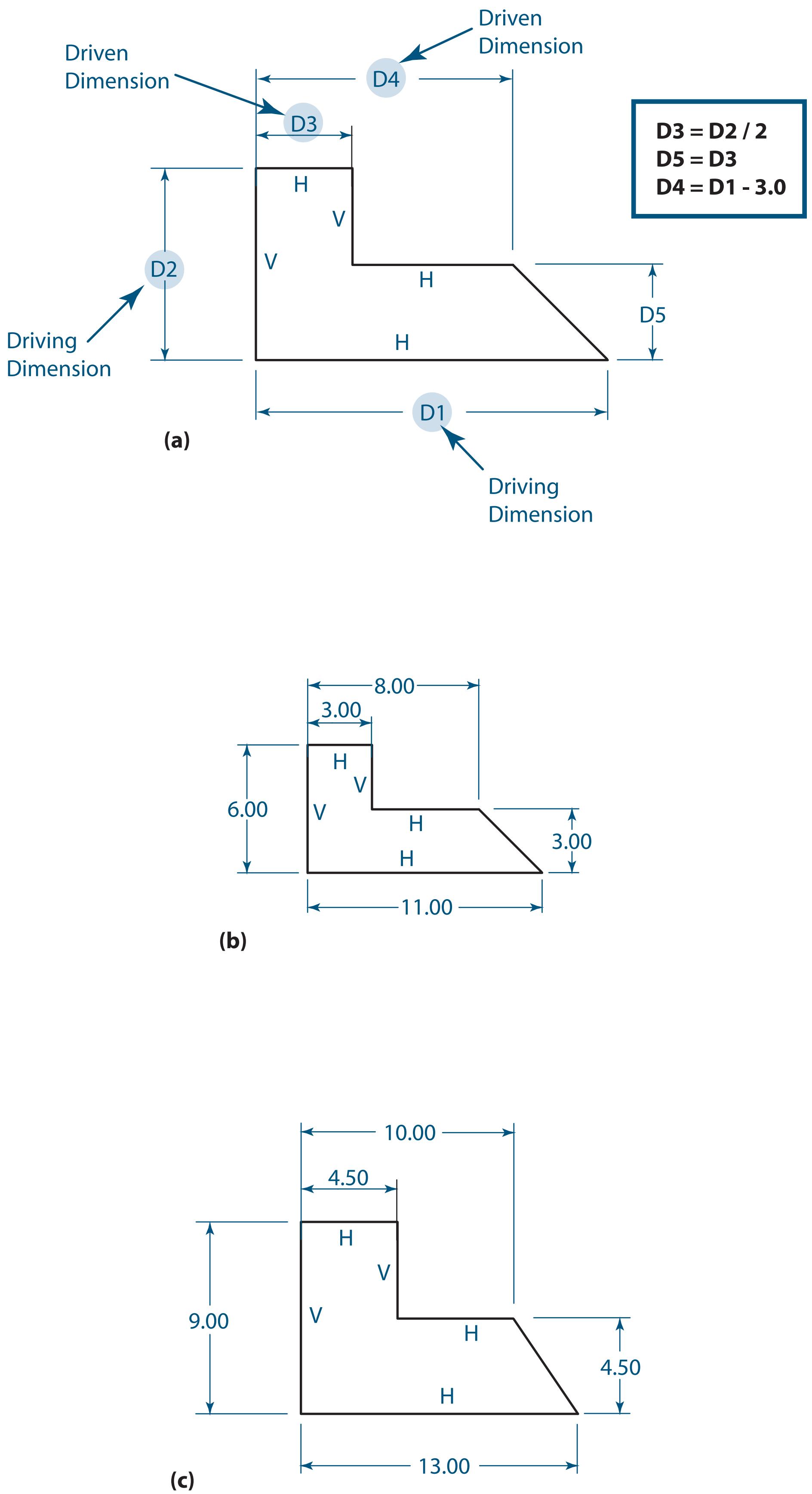
The natural question then becomes, which set of constraints is correct or preferred? The answer depends on what the function of the part and the design intent is or how the designer wants to be able to change the model. You also should consider how the solid model will be used for analysis, manufacturing, and documentation when applying sets of constraints. One of the greatest advantages of a parametric solid model is that the model can be changed easily as the design changes. However, the constraints limit the ways in which the model can be changed.

5.04.05 Associative and Algebraic Constraints

Associative constraints, sometimes called **algebraic constraints**, can be used to relate one dimensional constraint to another. The dimensional constraints on a profile are expressed in terms of variables. Each dimensional constraint is identifiable by a unique variable name, as shown in Figure 5.25. Algebraic constraints can be used to control the values of selected variables as the result of algebraic expressions. Algebraic expressions consist of constants and variables related to each other through the use of arithmetic functions (+, −, ×, absolute value, exponent, logarithm, power, square root, and sometimes minimum and maximum); trigonometric functions; and conditional expressions (if, else, or when) including inequalities comparisons (if A > B then ...).

There are two different methods for solving sets of algebraic constraint equations. Software that uses **variational techniques** solves the equations simultaneously. A compatible solution for all of the variables can be calculated when there are a sufficient number of equations. In a system using **parametric techniques**, the equations are usually solved in sequential order. The equations will have only one unknown variable. All other variables in the algebraic expression must be known for the value of the unknown variable to be calculated, which is called the dependent or **driven dimension**. The known variables are called the **driving dimensions**. As shown in Figure 5.25, when the value of a driving dimensional constraint is changed, the value of its driven dimensional constraints are automatically changed, too.

FIGURE 5.25. Dimensional constraints are shown in terms of variables and a set of algebraic constraints in (a). Dimensions D3, D4, and D5 are automatically specified by specifying dimensions D1 and D2 in (b). Dimensions D3, D4, and D5 change automatically when D1 and D2 are changed in (c).



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5.05 Strategies for Combining Profile Constraints

A completed profile is constrained using a combination of geometric and dimensional constraints and may include algebraic constraints as well. The constraint set must be complete for the geometry to be fully defined. If a profile is overconstrained or underconstrained, it may not be possible to create a solid feature from the profile. Some solid modeling software automatically applies constraints, but these constraints usually need to be changed to reflect the design intent. Furthermore, most software systems expect the user to apply constraints in addition to the automatically generated constraints; in particular, variational modelers allow underconstrained sketches and do not require user-applied constraints, but these systems can yield unpredictable results when the dimension values are changed. By gaining a thorough understanding of constraints (and how and when to apply them), you will be able to control the behavior of your models and capture your design intent. A strategy for applying geometric and dimensional constraints to a profile is demonstrated next.

The first constraint usually applied to a new sketch is a **ground constraint**. Ground constraints serve as anchors to fix the geometry in space. Ground constraints may have various forms. The most common type of ground constraint is a geometric entity such as a line or point on the profile having been made coincident with one of the basic modeling planes or with the origin of the coordinate system. For example, if the first feature of a model is created by extrusion, it may be convenient to place a corner of the profile on the origin of the coordinate system. This is usually done by placing one of the vertices of the sketch exactly at the origin. If the first feature is created by rotation, it may be convenient to place one of the endpoints of the center axis at the origin.

When the profile is closed and the automatically generated constraints have been applied, the interactive constraint definition phase begins. Some software creates a fully constrained sketch, including both geometric and dimensional constraints; but the constraint set chosen by the software is usually not exactly what you want. Other software does not fully constrain the sketch, but leaves this task to the designer. Ground constraints should be specified if this was not already done when the profile was sketched. Next, geometric and dimensional constraints should be added and/or changed until the profile is fully constrained. Typically, your solid modeling software will alert you when the profile is fully constrained or when you try to overconstrain the profile. In particular, you should take care to delete any unwanted geometric and dimensional constraints that may have been automatically added. Finally, the profile should be changed to reflect the design intent and the dimensional constraints adjusted to the desired values. Some sketching editors automatically readjust the profile after each constraint is added; others wait until all of the constraints have been specified before readjusting the profile. Updating the profile to show its new shape after constraints are added or changed is called **regeneration**.

The way dimensional constraints are added depends largely on what the intended function of the part is, how it is to be fabricated, and how the geometry of the part may change in the future. What would a simple L-bracket look like if some of the dimensions were changed, as shown in Figure 5.26? In this case, d1 was changed from 30 to 40 and d2 was changed from 3 to 8. The result is shown in the figure. But if you want to make the bracket by bending a piece of sheet metal, the part should have a uniform thickness throughout. One way to do this is to force the length of line segments that define the thickness of both legs of the *L* to be equal. The geometric constraint shown in Figure 5.27 has this effect. The equal length geometric constraint replaces the dimensional constraint for the thickness of the vertical leg of the bracket. If you tried to apply the equal length constraint and the dimensional constraints on both line segments, the sketch would be overconstrained, a situation the software would not accept. In addition, an associative constraint needs to be added between the radius of the inside corner of the bracket and the radius of the outside corner to ensure that the thickness of the part is constant around the corner.

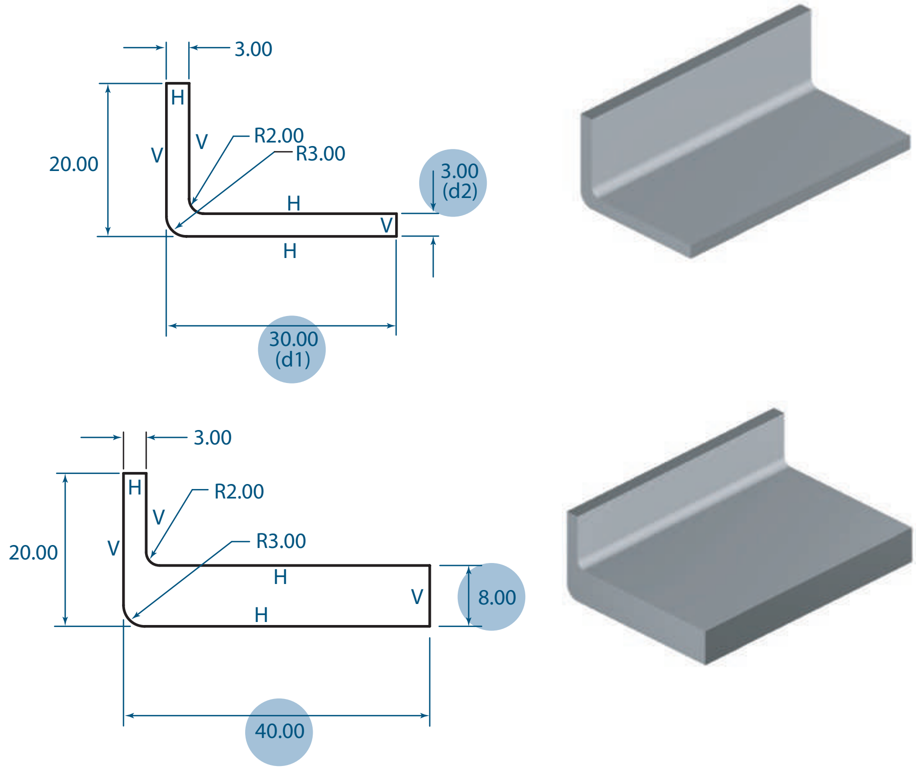

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FIGURE 5.26. Changing the values of the dimensions changes the geometry of the model, without the need to reconstruct the model.

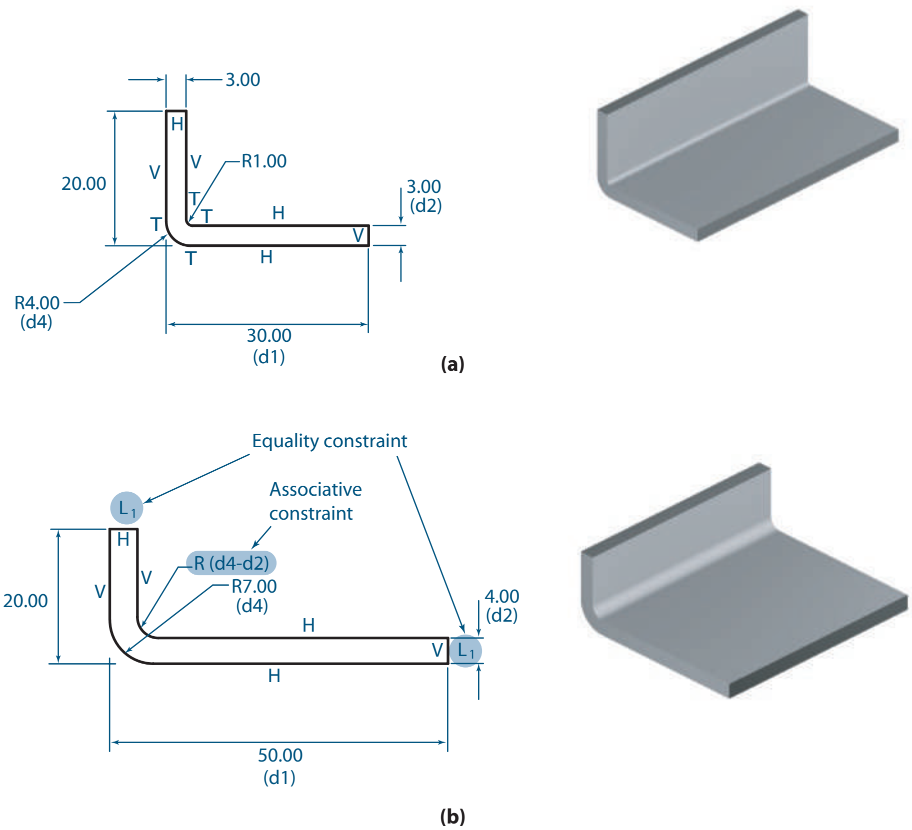

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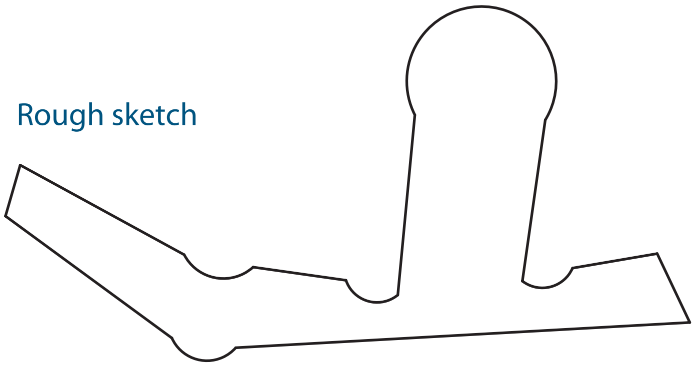
FIGURE 5.27. When compared to the original model in (a), the addition of the equality and associative constraints in (b) ensures a constant material thickness even if the dimensions are changed, thus adding functionality to the model if that is the intent.

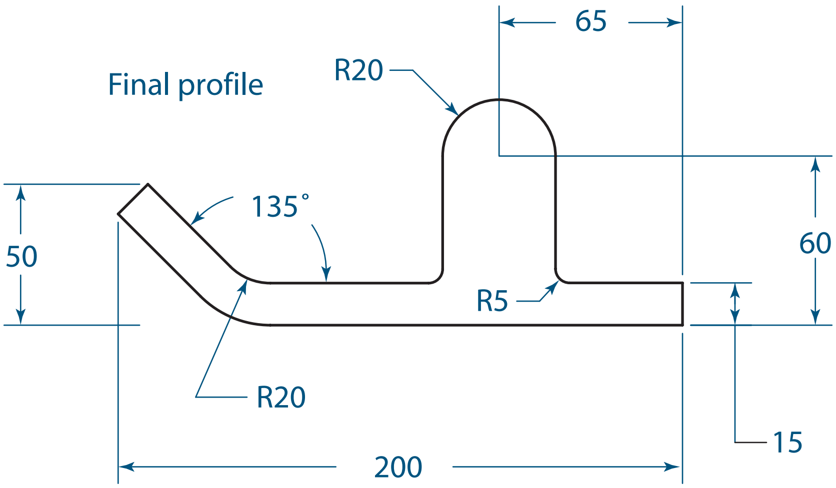
This constraint strategy demonstrates how to make your parts more robust. Through this simple example, you can see the importance of fully understanding the behavior of your model and the effects of your selection of dimensions and constraints. Your choices for geometric, dimensional, and algebraic constraints are not unique; but the decisions you make in selecting a set of constraints will have a big impact on the behavior of your model if you make changes to it. You must choose a modeling strategy that will reflect your design intent.

As an exercise for developing skill in the application of constraints, consider the rough sketch and the finished profile shown in Figure 5.28. For the profile to be fully constrained using only the dimensional constraints shown, certain geometric constraints are needed. Segment 1, for example, needs to be horizontal and tangent to Segment 2. Segment 2 needs to be tangent to Segment 1 as well as to Segment 3. Segment 3 needs to be tangent to Segment 2, perpendicular to Segment 4, and parallel to Segment 5. Segment 4 needs to be perpendicular to Segment 3 and equal in length to Segment 14. These constraints and the required geometric constraints on the remaining segments are shown in Figure 5.29. Keep in mind that a set of geometric constraints may not be unique. Can you specify another set of geometric constraints for this example that would create the same profile with the same dimensional constraints?

FIGURE 5.28. Geometric constraints need to be applied to the rough sketch (a) to produce the desired, fully constrained profile (b).



(a)



(b)

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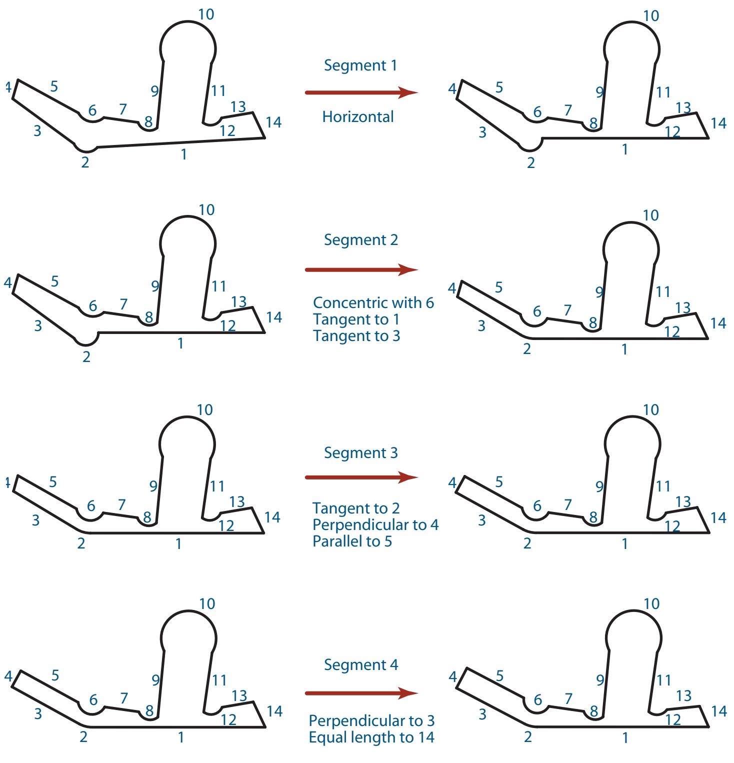
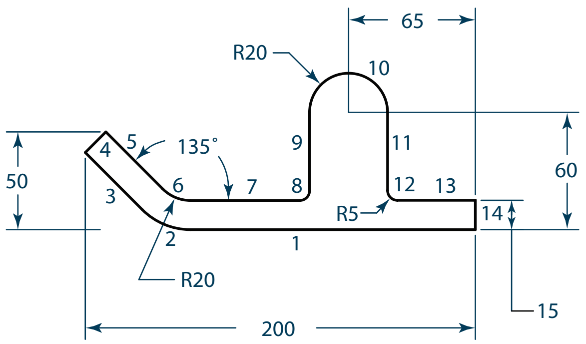
Figure 5.29. Applying geometric constraints to the first four segments of the sketch in Figure 5.28a to produce the finished profile in Figure 5.28b.



(a)



Segment	Constraint
1	Horizontal, Tangent to 2
2	Concentric with 6, Tangent to 1, Tangent to 3
3	Perpendicular to 4, Parallel to 5, Tangent to 2
4	Equal Length to 14, Perpendicular to 3
5	Parallel to 3, Tangent to 6
6	Concentric with 2, Tangent to 5, Tangent to 7
7	Horizontal, Tangent to 6, Tangent to 8
8	Tangent to 7, Tangent to 9
9	Vertical, Tangent to 8, Tangent to 10
10	Tangent to 9, Tangent to 11
11	Vertical, Tangent to 10, Tangent to 12
12	Tangent to 11, Tangent to 13
13	Horizontal, Tangent to 12
14	Vertical, Equal Length to 4

(b)

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5.06 More Complexity Using Constructive Solids

You have seen how to create solid models by sketching a 2-D profile on one of the basic modeling planes and then using a single extrusion or a single rotation to create a 3-D model. Adding material to or removing material from the original model can create a more complex model. When material is added, a **protrusion** feature is created. When material is removed, a **cut** feature is created. Both protrusions and cuts begin with sketched profiles that are then extruded or revolved to form solid shapes that are added to or removed from the existing body of the model. For an extruded feature, the profile lies in the sketch plane and is extruded in a direction perpendicular to the sketching plane. For a revolved feature, the profile and the axis of revolution must be coplanar so both will lie on the sketch plane.

When protrusions or cuts are made on an existing model, sketches and profiles are no longer restricted to be located on one of the basic modeling planes. Instead, any planar surface on the model can be selected and used as a **sketching plane** on which sketches and profiles can be created. Once a sketching plane has been selected, any 2-D element that is created will be forced to lie on that plane. After a sketching plane is selected, the model can be reoriented to look directly into the sketching plane. Although you can sketch when not looking directly into the sketching plane, you need to be very careful when viewing from a different orientation. Edges of your sketch may not be shown in their true shape, and angles may appear distorted. Most people find it easier to create 2-D profiles when they are looking directly into the sketching plane, just as it is easier for someone to draw straight lines and angles with correct measurements when the paper is oriented straight in front of them.

Examples of profiles on various sketching planes on a model and resulting extruded protrusions are shown in Figure 5.30; examples of extruded cuts are shown in Figure 5.31. Examples of revolved protrusions are shown in Figure 5.32, and examples of revolved cuts are shown in Figure 5.33.

As with the first extrusion or revolution that created the main body of the model, the profiles for the added protrusions or cuts must be fully defined by geometric, dimensional, and algebraic constraints before they can be extruded or revolved. A common geometric constraint for protrusions or cut features is to make one or more edges or vertices of the new profile coincident with edges of the surface used as the sketching plane. In Figure 5.30a, notice that one surface of the original object has been selected as a sketching plane and a rectangular profile has been sketched on the selected plane. The top and bottom edges of the sketched profile are coincident with edges of the sketching surface. The direction of extrusion is, by default, perpendicular to the selected sketching plane.

The length of the extrusion or angle of rotation also must be specified. There are several options for defining the length of the extrusion, as shown in Figures 5.30 and 5.31. The simplest is to specify a **blind extrusion**. A blind extrusion is one that is made to a specified length in the selected direction, analogous to specifying a dimensional constraint, as shown in Figure 5.30b. If your extrusion is the first feature used to create your initial model, it will be a blind extrusion. For a cut such as a hole, a blind extrusion creates a hole of a specified depth, as shown in Figure 5.31b.

Another way to determine the length of the extrusion is to use existing geometry. One option for specifying an extrusion length is to **extrude to the next surface**. With this option, the extrusion begins at the profile and the protrusion or cut stops when it intersects the next surface encountered, as shown in Figure 5.30c and Figure 5.31c. Another option is to **extrude to a selected surface**, where the protrusion or cut begins at the profile and stops when it intersects a selected surface, which may not necessarily be the first one encountered. See Figure 5.30d and Figure 5.31d. For extruded cuts, there is an option to **extrude through all**. This option creates a cut or protrusion that starts at the profile and extends in the selected direction through all solid features, as shown in Figure 5.31e. A **double-sided extrusion** permits

FIGURE 5.30. Different ways to terminate an extruded protrusion from the profile in the sketching plane in (a). Blind extrusion in (b), extrude to next surface in (c), extrude to a selected surface in (d).

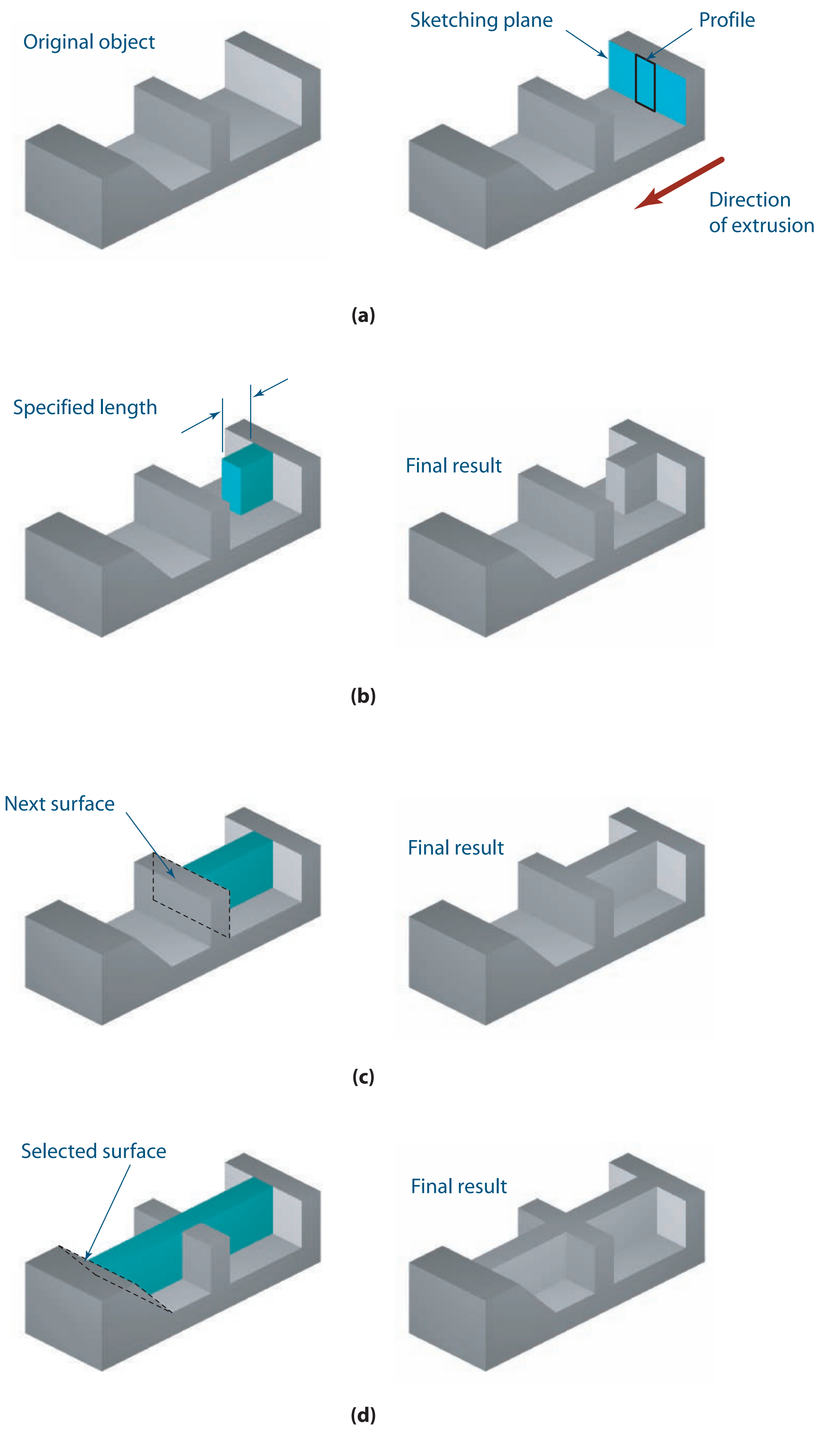
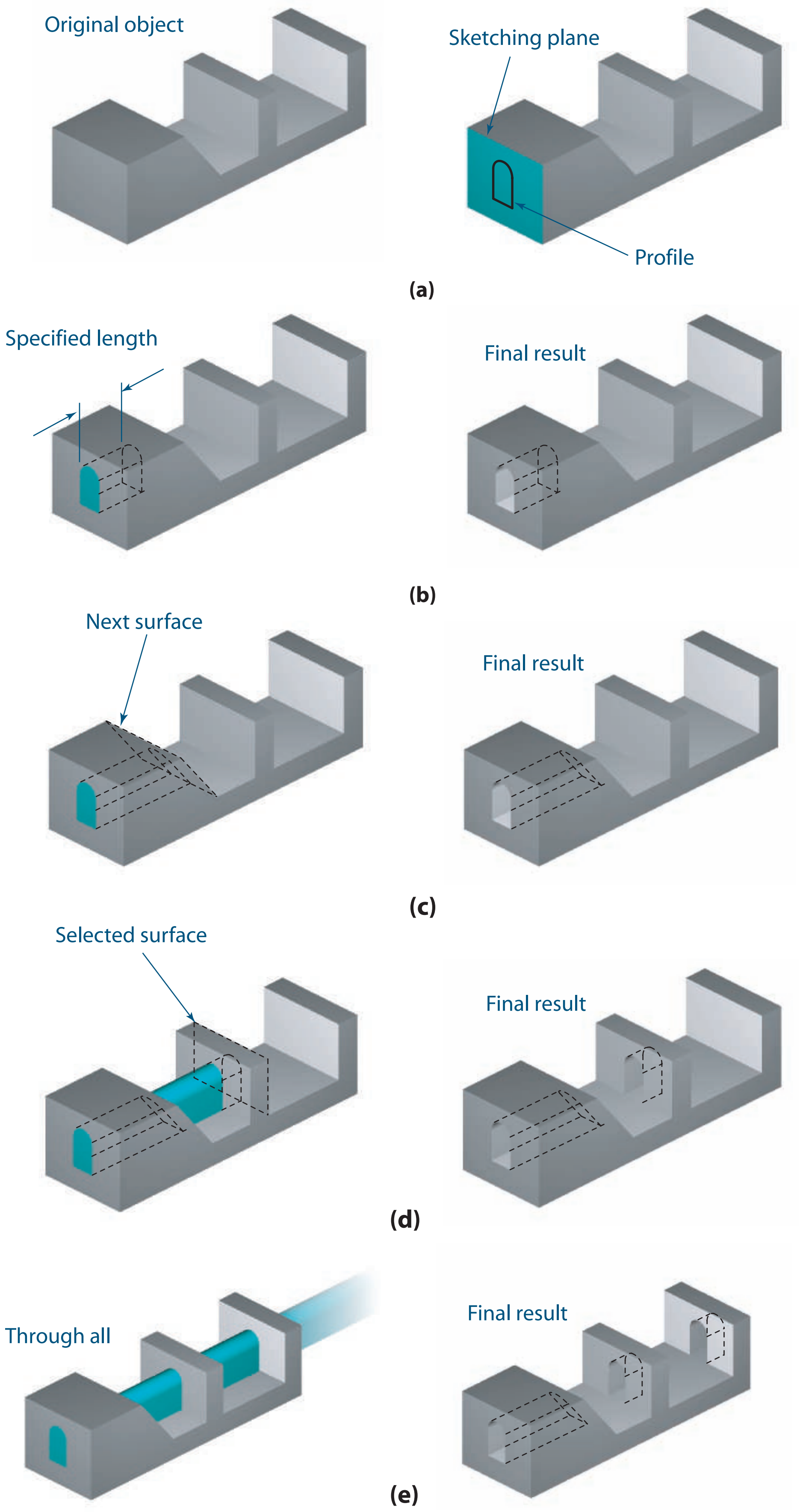


FIGURE 5.31. Different ways to terminate an extruded cut from the profile in the sketching plane in (a). Blind cut in (b); cut to next surface in (c). Cutting to a selected surface (d) and cutting through all (e).



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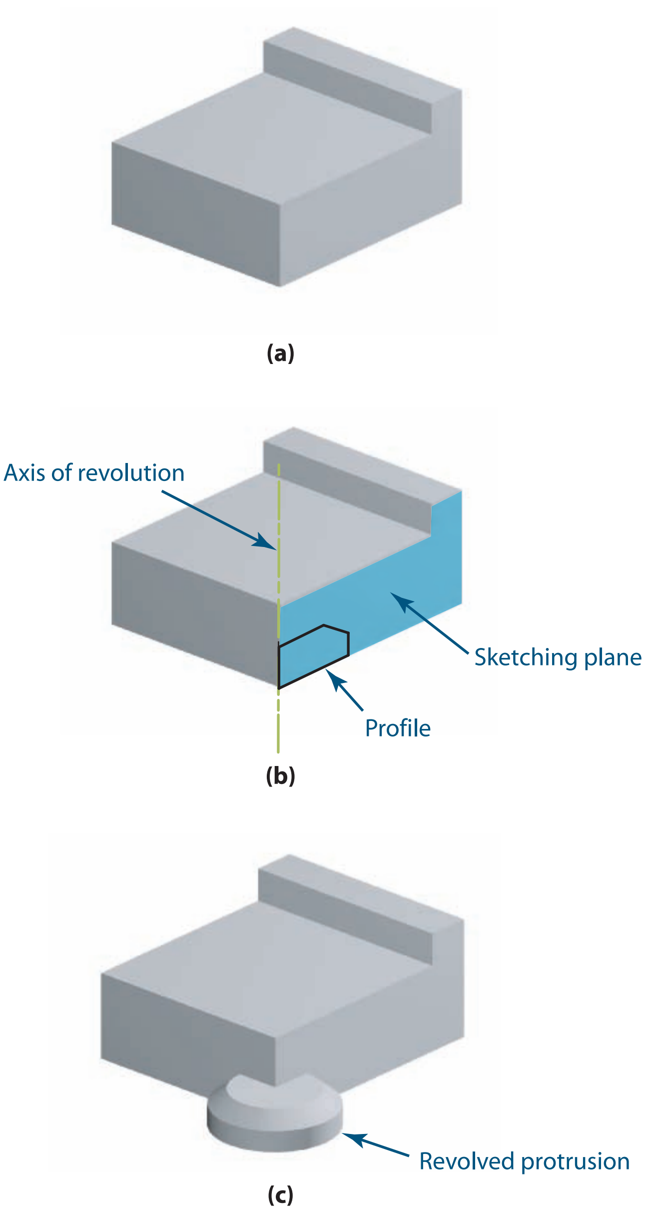

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FIGURE 5.32. The addition of a revolved protrusion to an existing base in (a) by using one of its surfaces as a sketching plane to create a centerline and profile in (b) and revolving it to produce the final result in (c).

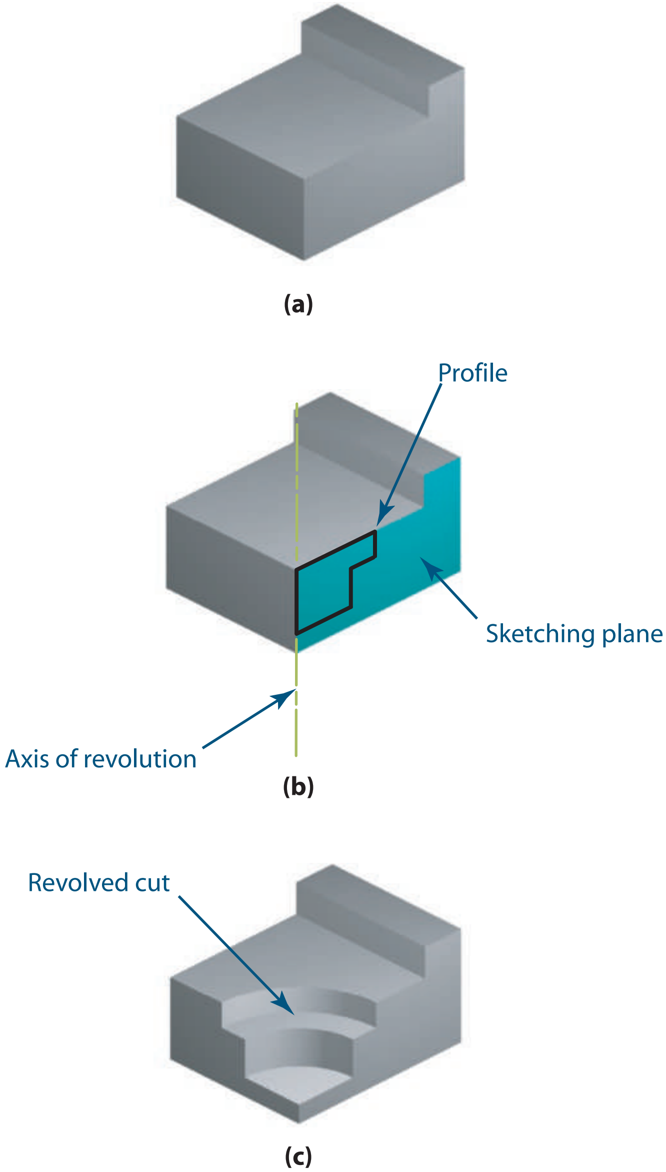

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FIGURE 5.33. The addition of a revolved cut to an existing base in (a) by using one of its surfaces as a sketching plane to create a centerline and profile in (b) and revolving it to produce the final result in (c).

the protrusion or cut to extend in both directions from a profile. The method of termination in each direction can then be specified independently. Other methods of terminating the extrusion length may be available depending on the specific solid modeling software used. You also can specify the angle of rotation of a revolved protrusion or cut in a similar manner by using a specified angle (blind revolution) or by revolving up to next or selected surfaces.

5.07 Breaking It Down into Features

When you build a solid model, you need to decide how to create the various shapes that compose the part. Very few parts can be modeled as a single extrusion or revolution. The various protrusions and cuts on the main body of a model are called **features**. What are features? If you consider your face, you might say that its features are your eyes, nose, lips, and cheeks. It is not much different on a manufactured part; a feature can be any combination of geometric shapes that make up the part and are distinctive in shape, size, or location. Features are characteristic elements of a particular object, things that stand out or make the object unique. Features often have characteristic geometric shapes and specific functions. A simple hole, for example, is a cylindrical cut that is often used as a receptacle for a fastener such as a bolt or screw. A manufactured part may have many different types of features. Since these features are the foundation of contemporary solid modeling systems, you must be able to recognize them.

Engineered parts also have features that are composed of repeated combinations of shapes. Most feature-based modelers have a collection of standard built-in features and may also allow you to define your own features. This can be handy when your products are designed with a particular shape that varies in size, such as gear teeth, airfoils, or turbine blades. The challenge for designers is to identify part features and build solid models that reflect the function of the part and design intent.

5.07.01 The Base Feature

All of your parts will be created from a collection of features, but you need to start your model with a basic shape that represents the general shape of the object. Your first step should be to study the part and identify the shape that you will use as the **base feature**. The base feature should be something that describes the overall shape of the part or something that gives you the greatest amount of functional detail that can be created with a single extrusion or rotation. Figure 5.34 shows several parts with the base features used to create the solid models.

FIGURE 5.34. Parts and their base features.

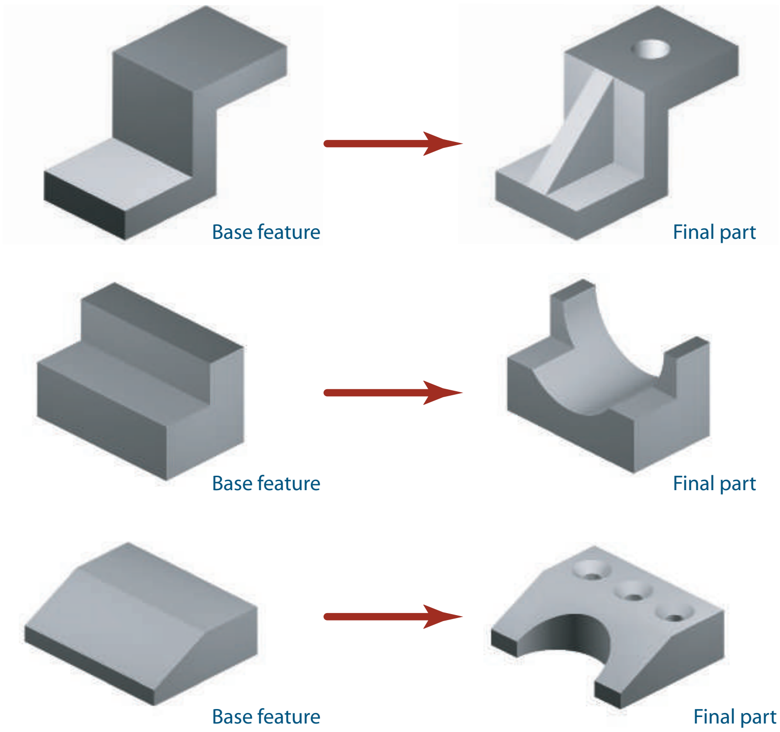

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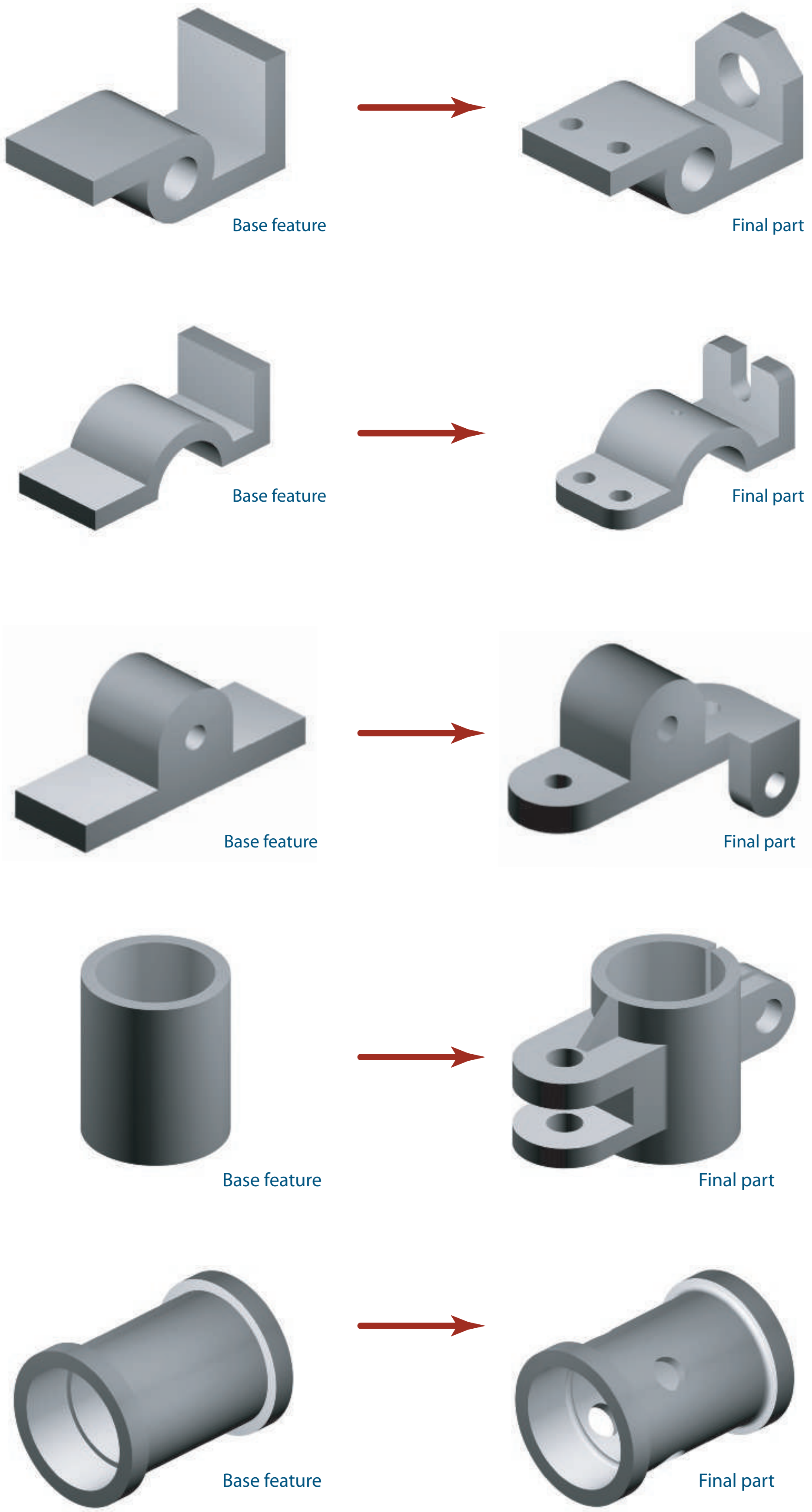
FIGURE 5.34. (CONTINUED)
Parts and their base features.



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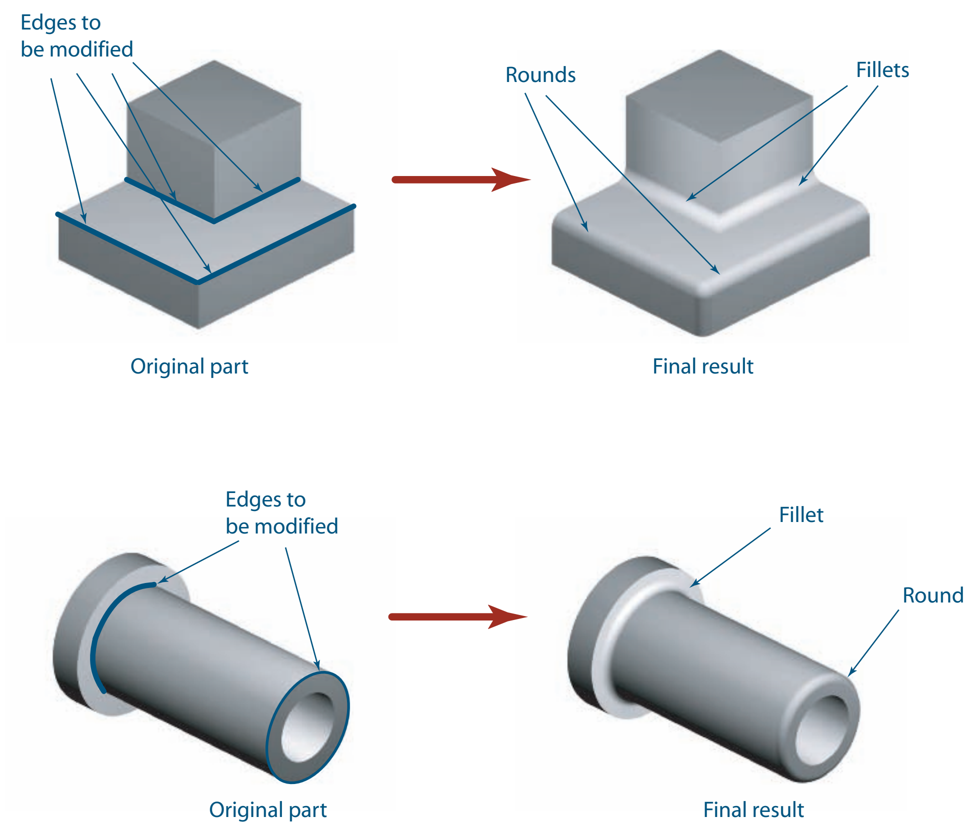
After the base feature is created, you can modify the shape by adding or subtracting material to it to create form features. A **form feature** is a recognizable region or area on the part geometry that may have a specific function and/or method of manufacture. The geometric components or shapes within the feature usually have some geometric relationships or constraints. Different CAD systems use various names for these features, but you should become familiar with some of the common terms. The following section discusses common feature types.

5.07.02 Chamfers, Rounds, and Fillets

Unless otherwise specified, adjoining surfaces on a virtual part can intersect to form sharp corners and edges, but real parts often have smooth transitions along the edges of these surfaces. The most common edge transitions are **rounds**, **fillets**, and **chamfers**. A round is a smooth radius transition of the external edge created by two intersecting surfaces. A fillet is a smooth transition of the internal edge created by two intersecting surfaces. Geometrically, the rounds and fillets are tangent to both intersecting surfaces. Examples of rounds and fillets are shown in Figure 5.35. Fillets and rounds are specified by the size of their radii and the edge(s) that are rounded.

Chamfers also provide a transition between two intersecting surfaces, but the transition is an angled cut instead of a radius. Examples of chamfers are shown in Figure 5.36. Chamfers can be specified by the distance along each intersecting surface to the original edge or by the distance along one of the original surfaces and the angle made with that surface.

FIGURE 5.35. Examples of rounds and fillets applied to the edges of a part.



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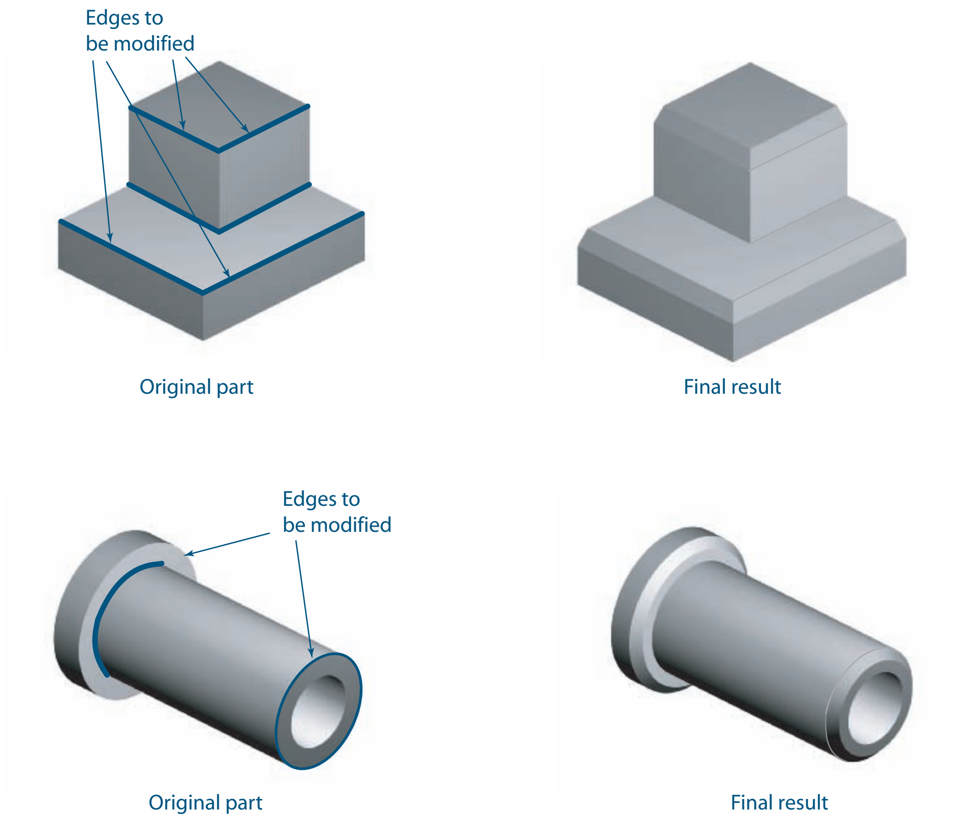

FIGURE 5.36. Examples of chamfering applied to the edges of a part.

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Functionally, on an inside edge, a fillet may be necessary to facilitate fabrication or to reduce stresses at the corner so the part does not break as easily. On an outside edge, rounds and chamfers are usually used to eliminate sharp edges that can be easily damaged or that can cause injury or damage when the part is handled. Rounds, fillets, and chamfers are generally small when compared to the overall size of the associated base or parent feature.

5.07.03 Holes

Holes are ubiquitous in nearly all manufactured parts and, therefore, can be inserted into a model as features by most solid modeling software. Holes are often used with bolts or screws to fasten parts together. Many different types of holes can be used with specific fasteners or can be created using different manufacturing processes. Some special types include holes that are blind, through, tapped, counterbored, or countersunk, as shown in Figure 5.37. Each type of hole has a particular geometry to suit a specific function. You should study the hole types so you recognize them when you model your parts.

Many solid modeling software packages include standard or built-in features to help you with your modeling task. When you use a standard hole feature, the solid modeling software makes certain assumptions about the geometry of a hole so you do not need to specify all of the dimensions and constraints that make up the feature. A countersunk hole, for example, can be made as a revolved cut. What do you need to do to create this feature? You begin by selecting a sketching plane, then create and constrain the sketch and revolve the sketch about a specified axis. Many things can go wrong if you are not careful. There might not be a plane on which to sketch. Your sketch might not have the proper shape, or the axis might not be perpendicular to the desired surface.

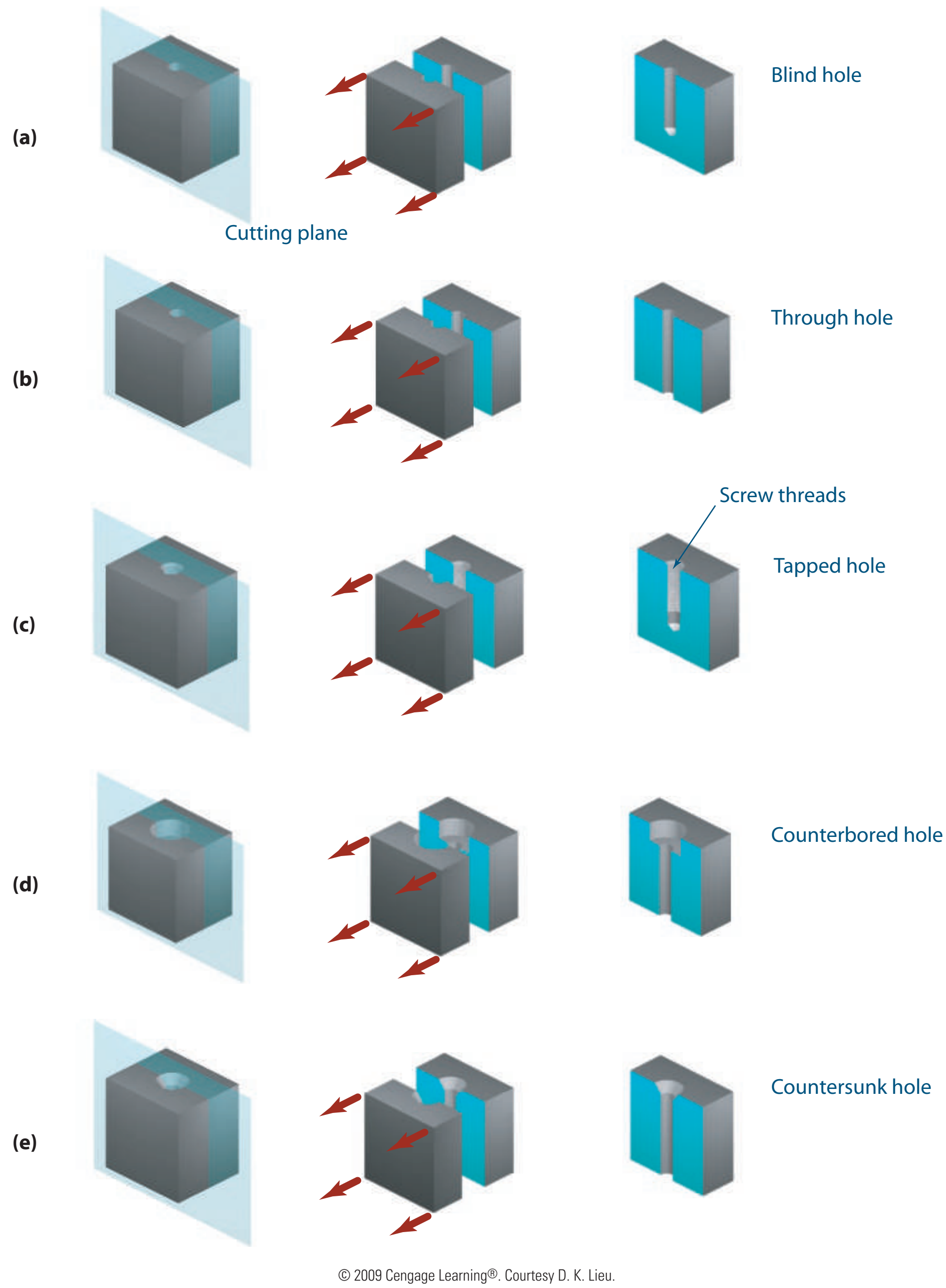

FIGURE 5.37. Cross sections of various types of holes to reveal their geometry.

However, a countersunk hole feature can often be created from a standard feature by selecting the location of the axis of the hole on the desired surface, the diameter of the hole, the diameter and angle of the countersink, and the depth of the hole. The shape of the profile, axis of revolution, and angle of revolution are included automatically in the feature definition. No sketching plane is needed. Figure 5.38 illustrates the use of a cut feature compared to the use of a built-in hole feature to create a countersunk hole. In most cases, it is more desirable to create a hole using the built-in hole feature instead of a general purpose cut feature. Besides being a more natural way to place a hole in a model, you avoid potential errors in creating the desired geometry. Furthermore, using a general cut feature does not incorporate the specific geometry and function of a "hole" in the knowledge base of the model, which may be useful in downstream applications such as process planning for manufacturing the hole.

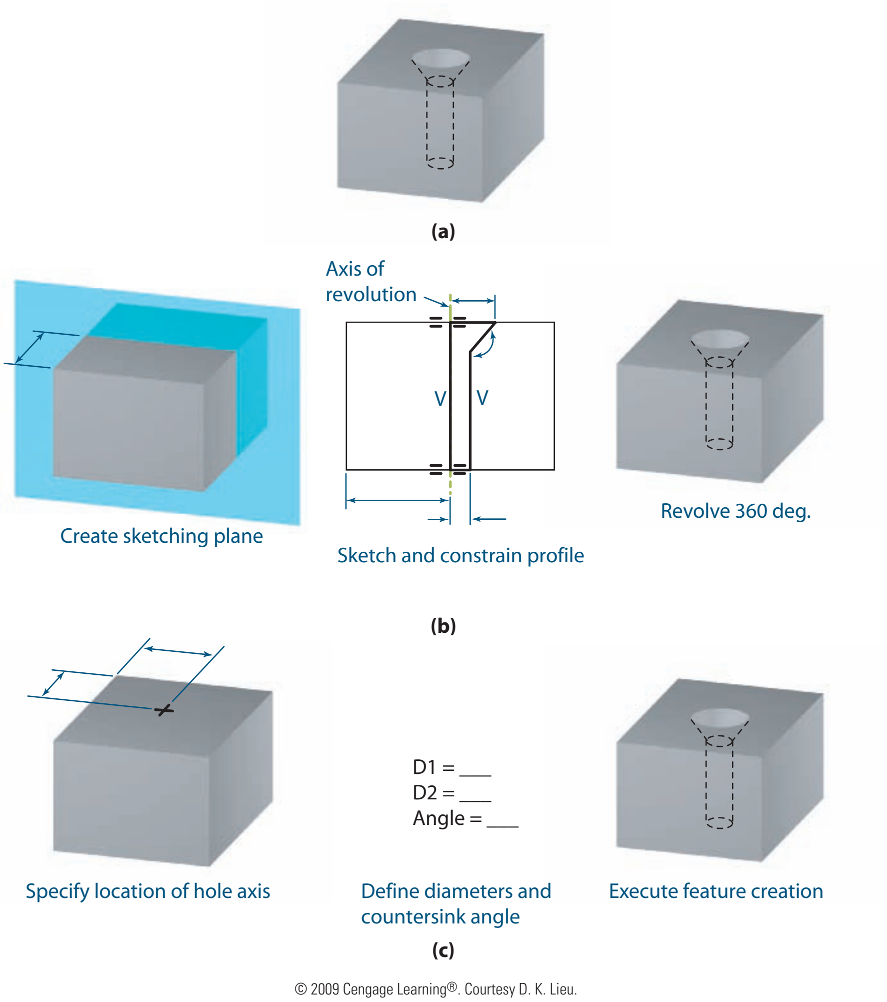

FIGURE 5.38. The countersunk hole shown in (a) can be created by using a general revolved cut, as shown in (b), or by specifying the hole as a built-in feature, as shown in (c).

5.07.04 Shells

The process of creating a shell, or **shelling**, removes most of the interior volume of a solid model, leaving a relatively thin wall of material that closely conforms to the outer surfaces of the original model. Shelled objects are often used to make cases and containers. For example, a soda bottle is a shell, as are cases for electronic products such as cell phones and video displays. The walls of a shell are generally of constant thickness, and at least one of the surfaces of the original object is removed so the interior of the shell is accessible. Figure 5.39 shows examples of a model that has been shelled.

Shelling is sometimes considered an operation rather than a feature. It is usually performed on the entire model, including all of its features, by selecting the surfaces to be removed and the thickness of the shell wall. Any feature not to be shelled should be added to the model after the shelling operation is complete. The order of feature creation and shelling operations may have a dramatic effect on the shape of the part, as will be shown later in this chapter.

5.07.05 Ribs and Webs

Ribs are small, thin, protrusions of constant thickness that extend predominantly from the surface of a part. Ribs are typically added to provide support or to stiffen a part. Sometimes they are added to improve a part's heat transferability. **Webs** are areas of thin material that connect two or more heavier areas on the part. Examples of ribs and webs are shown in Figure 5.40. These features are usually specified by their flat geometry, thickness, and location.

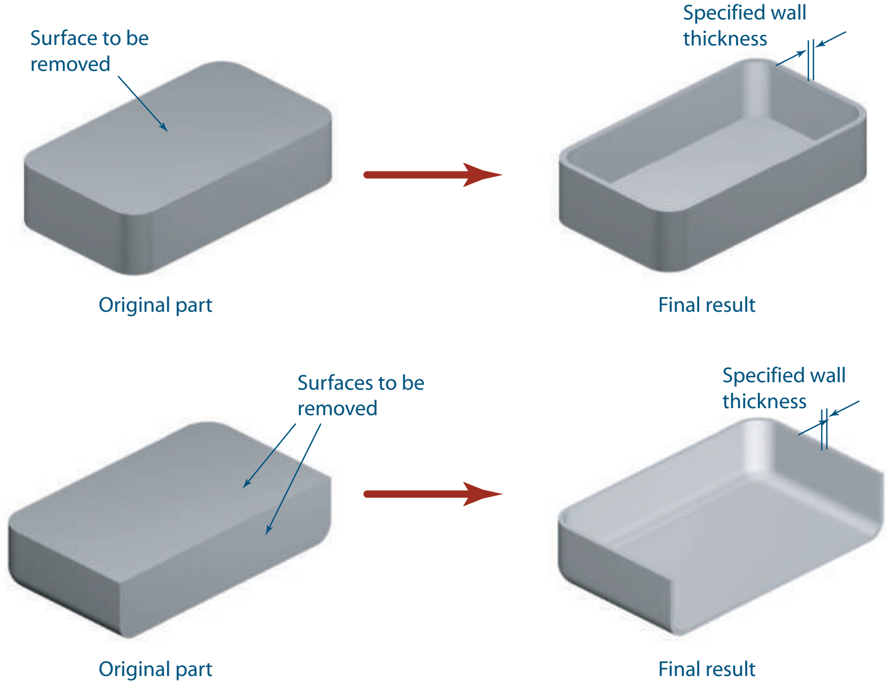

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FIGURE 5.39. Examples of shelling.

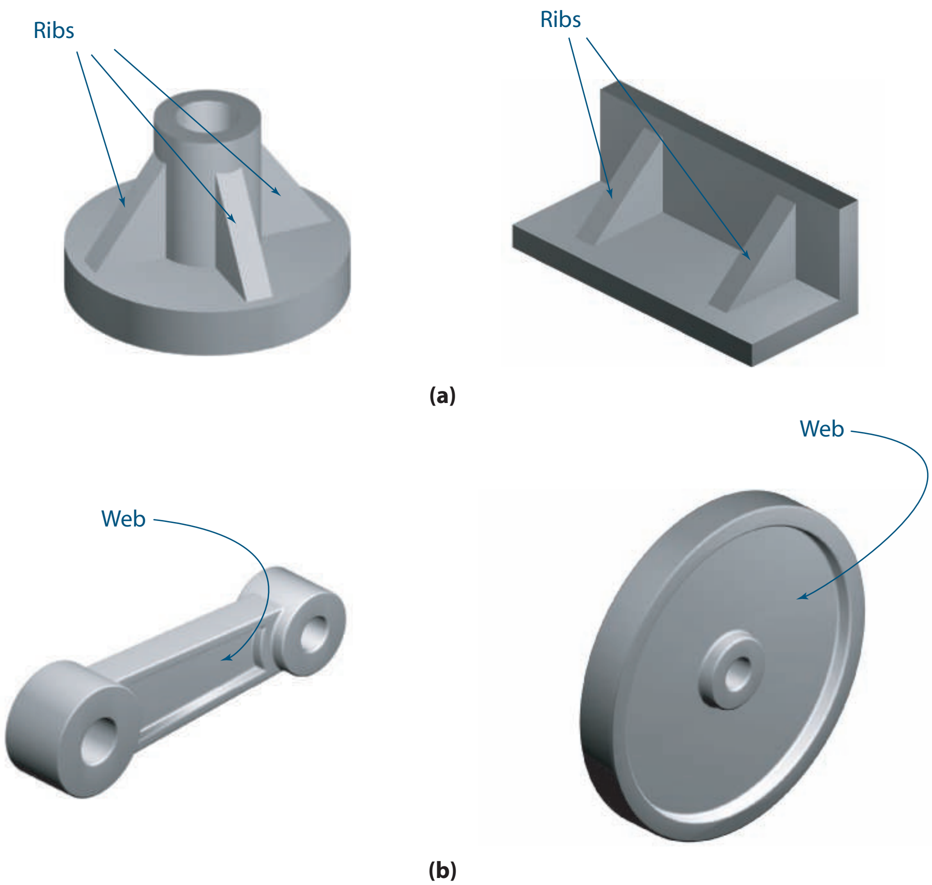

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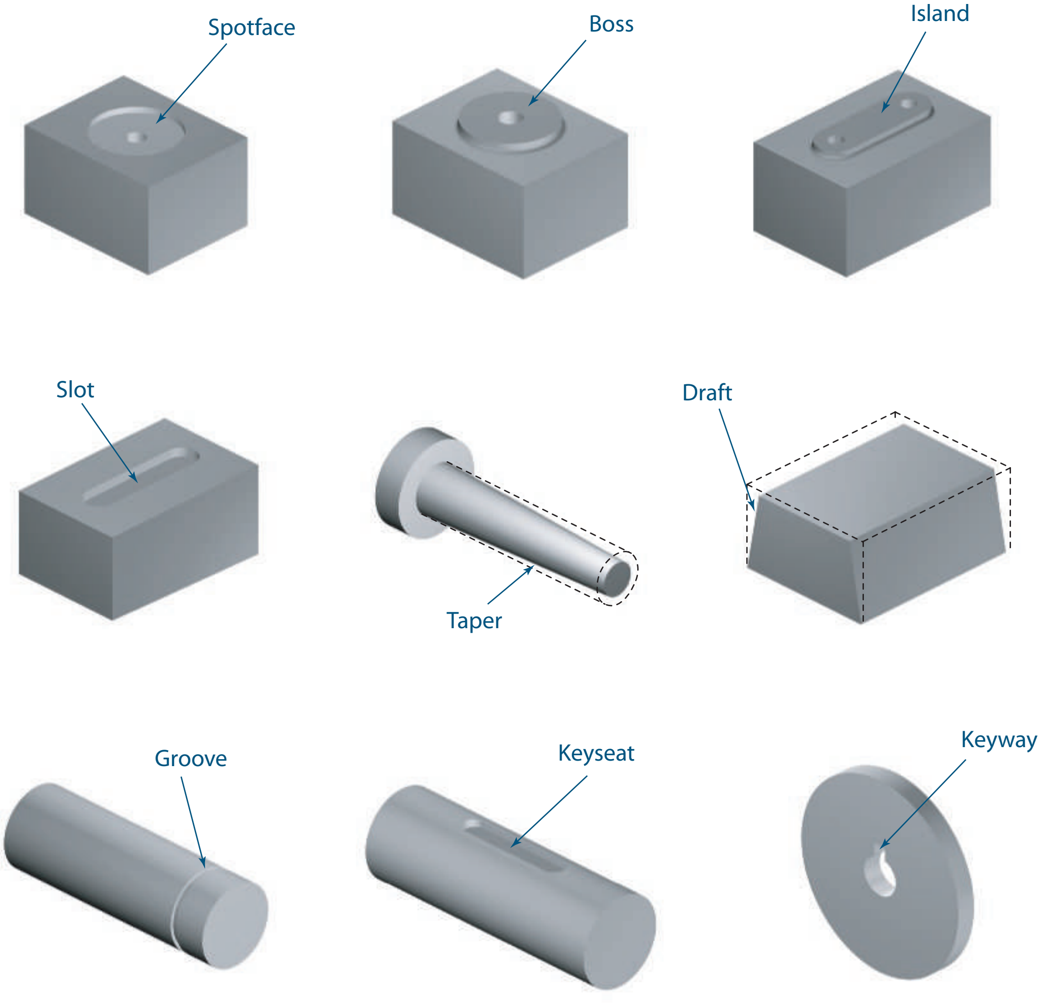
FIGURE 5.40. Ribs (a) added to parts to reinforce them, and webs (b) connect thicker sections on parts.

5.07.06 Other Feature Types

The features that follow (and that are shown in Figure 5.41) are less commonly found in solid modelers. When available, they should be used as needed. When such features are not available in the solid modeler, the geometric shapes can still be created from sketched profiles as protrusions or cuts. Note that special feature types usually imply a particular shape, function, manufacturing process, or other feature attribute.

- Boss—a slightly raised circular area, usually used to provide a small, flat, clean surface
- Draft—a slight angle in the otherwise straight walls of a part, usually used to facilitate its removal from a mold
- Groove—a long, shallow cut or annulus
- Island—an elongated or irregularly shaped raised area, usually used to provide a flat, clean surface
- Keyseat—an axially oriented slot of finite length on the outside of a shaft
- Keyway—an axially oriented slot that extends the entire length of a hole
- Slot—a straight, long cut with deep vertical walls
- Spot face—a shallow circular depression that has been cut, usually used to provide a small, flat, clean surface
- Taper—a slight angle in the otherwise cylindrical walls of a part, usually used to facilitate its insertion or removal into another part

FIGURE 5.41. Various features with specific functions that may be added to a solid model.



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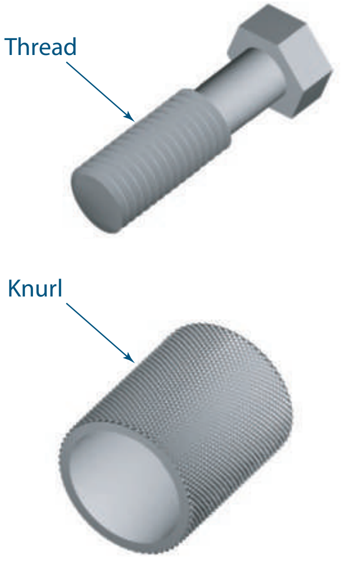
5.07.07 Cosmetic Features

Parts can be modified by altering their surface characteristics. These characteristics are called **cosmetic features** because they generally modify the appearance of the surface but do not alter the size or shape of the object, just like lipstick or hair coloring. Cosmetic features are necessary to the function of the part and may be included in the model so they can be used in later applications, such as fabrication. Some common cosmetic features include threads and knurls. Since the geometric changes are small and detailed, the cosmetic features usually are not modeled in their exact geometric form in the database of the object, but are included as notes or with a simplified geometric representation. You will learn more about simplified representations on drawings in later chapters. Some cosmetic features are shown in Figure 5.42.

5.07.08 An Understanding of Features and Functions

As a design engineer, you need to become familiar with the different types of features on various parts. Doing so will help you communicate with other engineers as well as imbed more of a part's engineering function into your models. For example, if you look at Figure 5.43, you will notice a rectangular cut on the edge of the hole. This cut is a geometric feature called a keyway. Why is it there? What purpose does it serve? In Figure 5.44, the gear is mounted to a shaft, which also has a rectangular cut.

FIGURE 5.42. Some cosmetic features.



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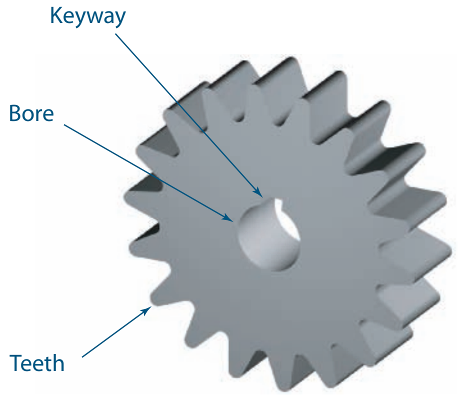

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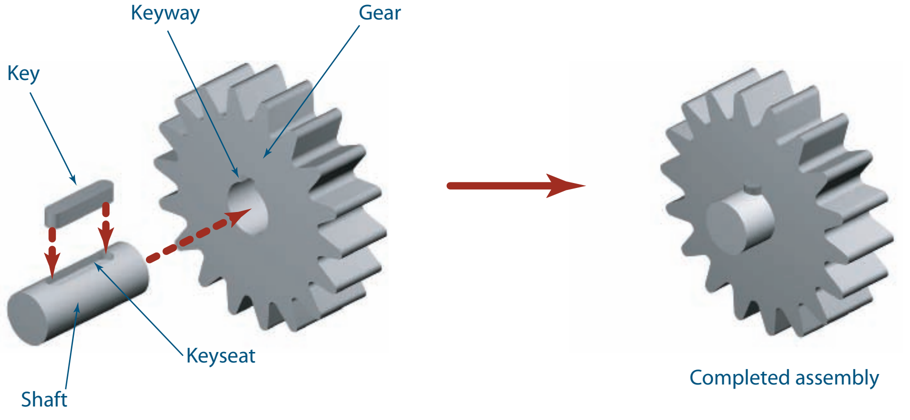
FIGURE 5.43. A gear with teeth, a bore, and a keyway as functional features.

FIGURE 5.44. A gear and shaft assembly. The key functions to transmit torque. The keyseat receives the key in the shaft, and the keyway receives the key in the gear.



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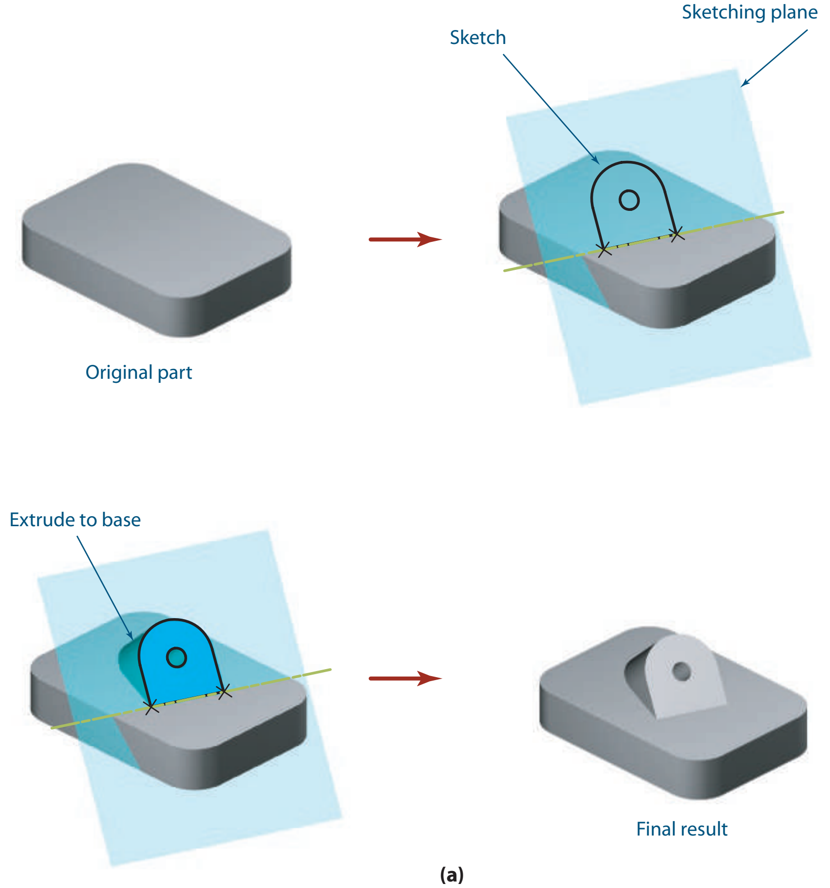
A small part called a key is used to line up the shaft and the gear and transmits torque from the shaft to the gear. If you were to create a feature-based solid model of the gear, you could identify the rectangular cut as a keyway feature. If the model parts were to be assembled with assembly modeling software (which is explained in detail in a subsequent chapter), the computer and software would recognize the models as mating parts and orient the gear, key, and shaft automatically.

5.08 More Ways to Create Sophisticated Geometry

Creating protrusions and cuts by extending the sketch profiles made on either the basic modeling plane or one of the existing surfaces of the model results in a wide variety of possible models. Even more sophisticated models, however, can be created by using reference geometries called datums, which can be added to the model, displayed, and used to create features. Generally, solid modelers offer at least three types of **datum geometries** that can be placed into a model: datum points, datum axes, and datum planes. These datum geometries do not actually exist on the real part (i.e., they cannot be seen or felt) but are used to help locate and define features. Consider, for example, the part shown in Figure 5.45a. The angled protrusion with the hole would be easy to create if an angled sketching plane could be defined as shown. The extrusion could be made to extend from the sketching plane to the surface of the base feature. This feature would be more difficult, although not impossible, to define using extruded protrusions and cuts that extended only from the basic modeling planes or one of the surfaces on the existing model. In Figure 5.45b, the uniquely shaped web would be easy to create if a sketching plane could be placed between the connected features as shown. An extruded protrusion could extend from both sides of the sketching plane to the surfaces of the connected features.

FIGURE 5.45. Using a sketching plane and profile, which are not on an existing surface of the object, to create a protrusion feature (a).



(a)

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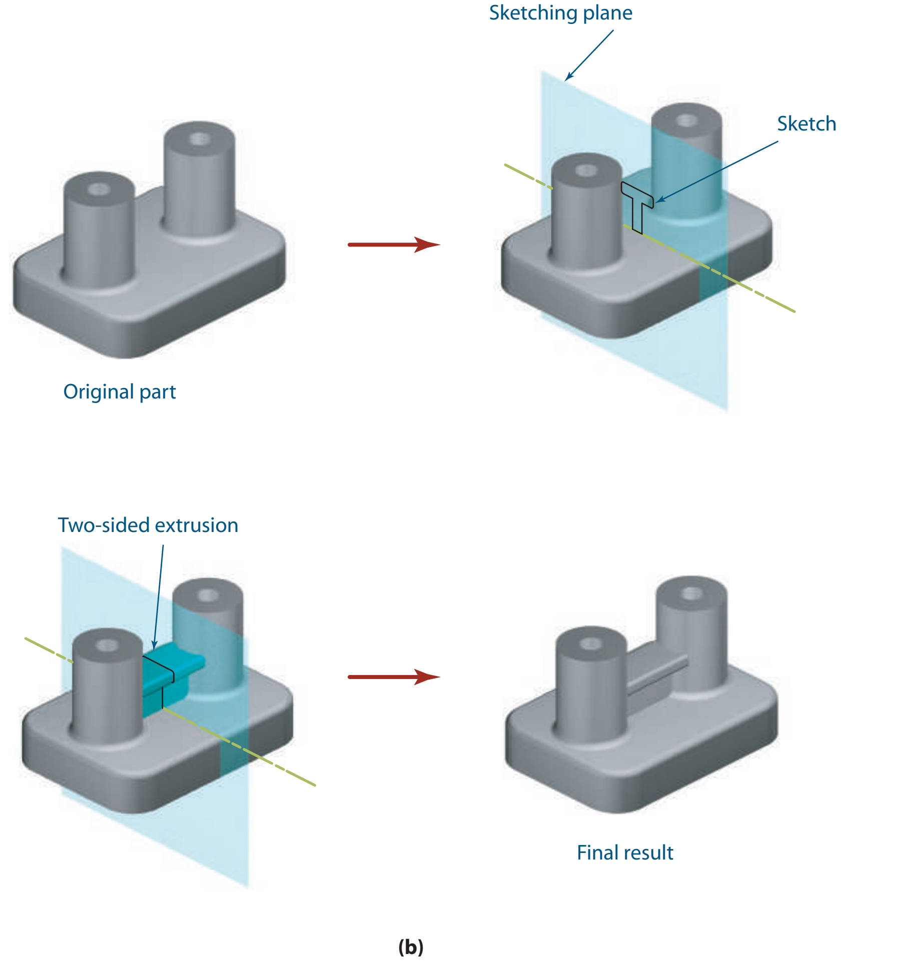
FIGURE 5.45. (CONTINUED) Using a sketching plane and profile, which are not on an existing surface of the object, to create a web feature (b).



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The next few sections will describe methods in which the three different types of datums can be defined geometrically and how the datums can be used to create a variety of new types of features. Depending on the specific solid modeling software being used, some of the methods described here for datum definition may or may not be available.

5.08.01 Defining Datum Points

Following are some of the different ways a datum point can be defined and created. The definitions are shown graphically in Figure 5.46.

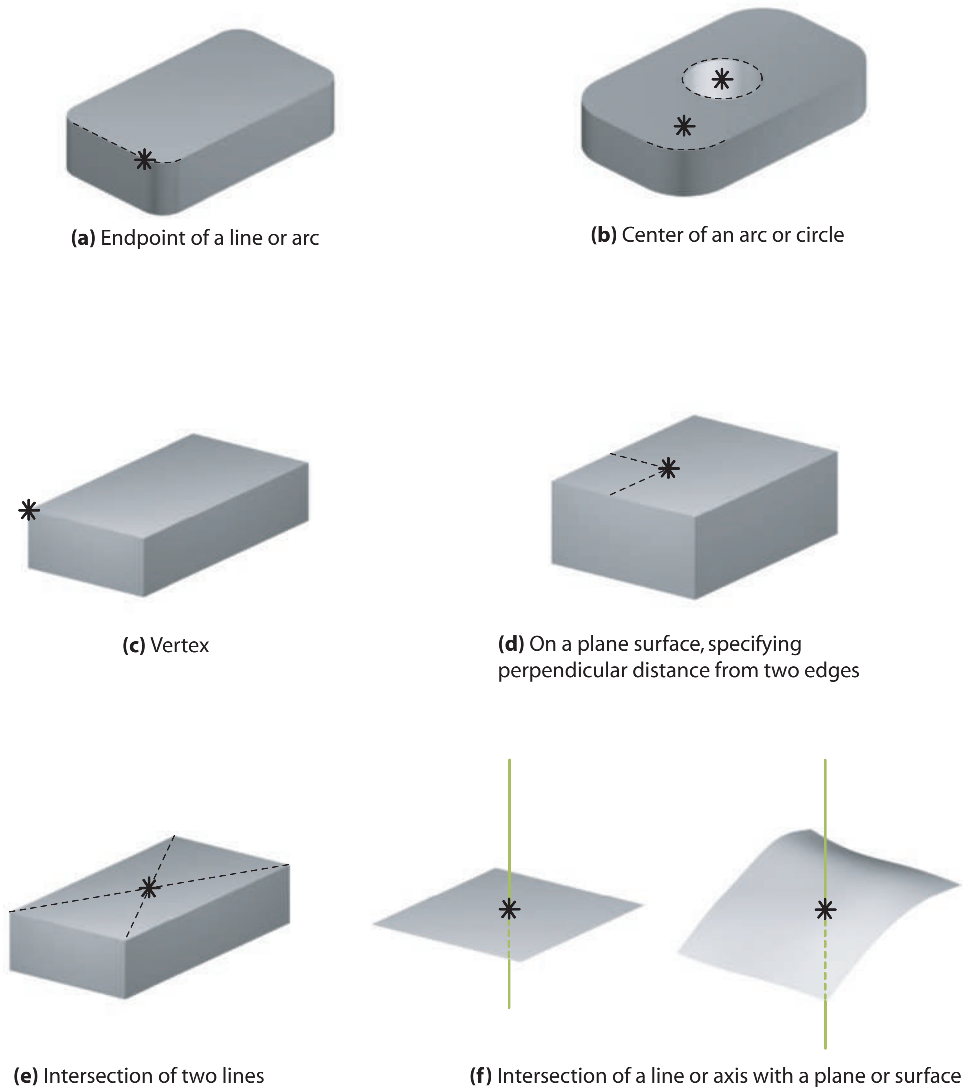
- At a vertex
- On a planar surface at specified perpendicular distances from two edges
- At the intersection of a line or an axis and a surface that does not contain the line

5.08.02 Defining Datum Axes

Following are some of the different ways a datum axis can be defined and created. The definitions are shown graphically in Figure 5.47.

- Between two points (or vertices)
- Along a linear edge
- At the intersection of two planar surfaces

FIGURE 5.46. Various ways to define a datum point.



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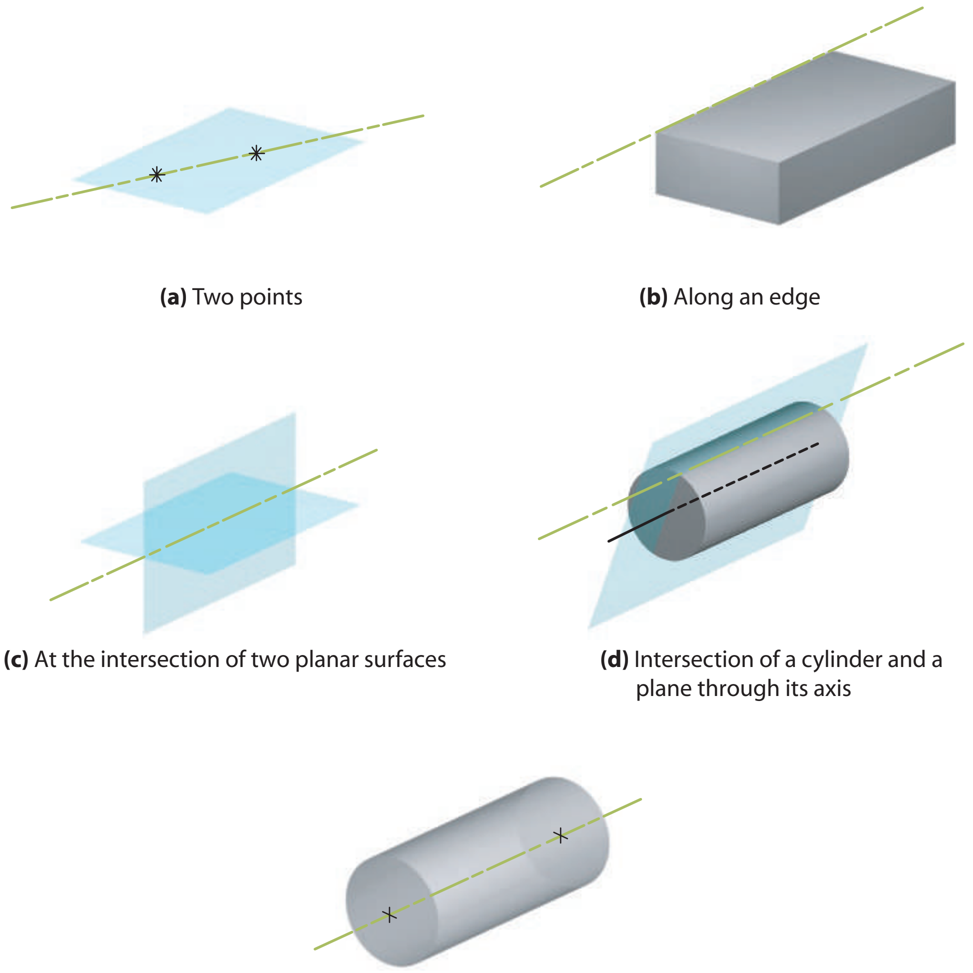
- At the intersection of a cylinder and a plane through its axis
- Along the centerline of a cylinder or cylindrical surface

5.08.03 Defining Datum Planes

Following are some of the different ways a datum plane can be defined and created. The definitions are shown graphically in Figure 5.48.

- Through three noncolinear points
- Through two intersecting lines
- Through a line and a noncolinear point
- Offset from an existing flat surface at a specified distance
- Through an edge or axis on a flat surface at an angle from that surface
- Tangent to a surface at a point on that surface
- Perpendicular to a flat surface and through a line parallel to that surface
- Perpendicular to a flat or cylindrical surface through a line on that surface
- Tangent to a cylindrical surface at a line on that surface

FIGURE 5.47. Various ways to define a datum axis.



(a) Two points

(b) Along an edge

(c) At the intersection of two planar surfaces

(d) Intersection of a cylinder and a plane through its axis

(e) At the centerline of a cylinder or cylindrical surface

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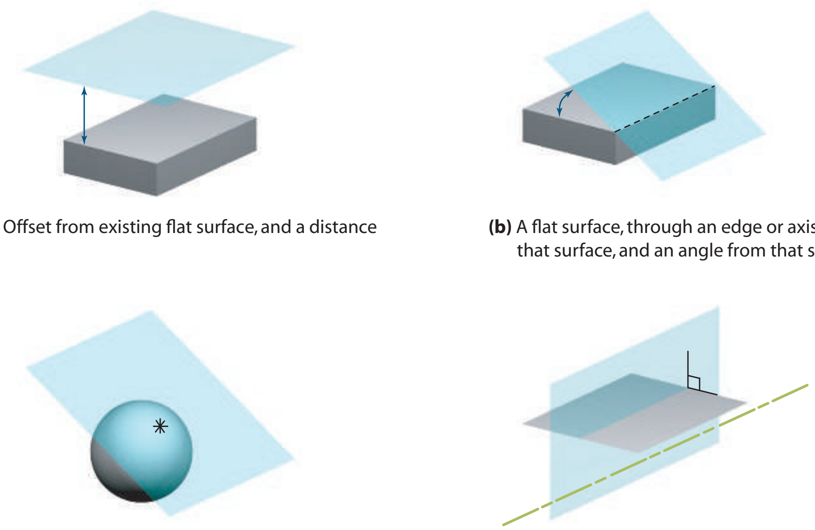
FIGURE 5.48. Various ways to define a datum plane. More ways to define a datum plane.



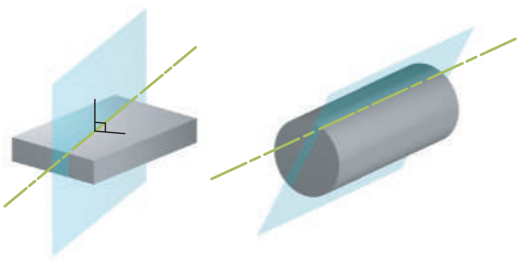
(a) Offset from existing flat surface, and a distance

(b) A flat surface, through an edge or axis on that surface, and an angle from that surface

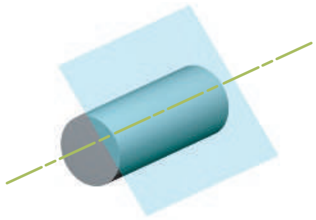
(c) Tangent to a surface at a point on that surface

(d) Perpendicular to a flat surface, and through a line parallel to that surface

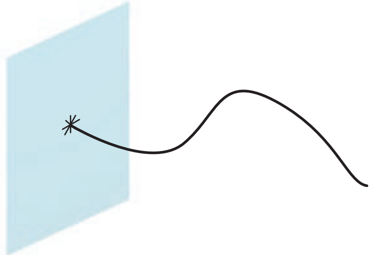
FIGURE 5.48. (CONTINUED) Various ways to define a datum plane. More ways to define a datum plane.



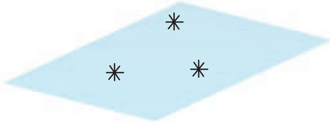
(e) Perpendicular to a flat or cylindrical surface and through a line on that surface



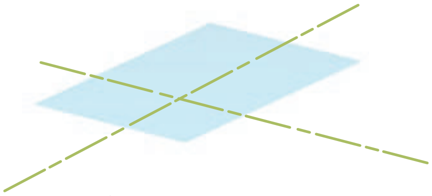
(f) Tangent to a cylindrical surface at a line on that surface



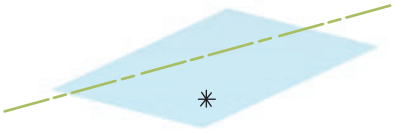
(g) Perpendicular to a curve at a point on that curve



(h) Three points



(i) Two intersecting lines



(j) A line and a point

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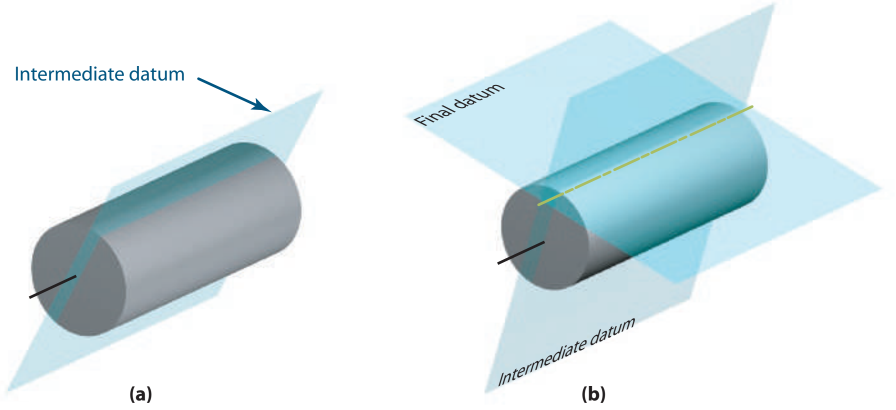
5.08.04 Chaining Datums

Series of simply defined datums are often used for creating more complex datums. In the example shown in Figure 5.45a, the angled protrusion was created in this manner: On the top surface of the base extrusion, two datum points were created by defining each of their locations from the edges of the base extrusion. A datum axis was then created using the two datum points as the endpoints of the axis. Finally, the desired datum plane was defined using the top surface of the base extrusion, using the datum axis created in that plane, and specifying the angle that the new datum plane makes with the top surface.

Another example is shown in Figure 5.49, where a datum plane is created to be tangent to the surface of a cylindrical extrusion. An intermediate datum plane is defined by one of the basic planes; the axis of the cylinder, which lies on that basic plane; and the angle the intermediate datum plane makes with the basic plane. The final datum plane is then created to be tangent to the surface of the cylindrical extrusion at its intersection with the intermediate datum plane. A datum plane tangent to the surface of a cylinder is commonly used to create cuts that extend radially into a cylindrical surface, such as holes or slots, and protrusions that extend radially from the cylindrical surface, such as spokes or vanes. Note that with protrusion from a tangent datum plane, the extrusion must be specified to extend in both directions from that datum; otherwise, there will be a gap between the extrusion and the curved surface.

FIGURE 5.49. To create a datum plane that is tangent to a cylindrical surface at a specific location, an intermediate datum plane, shown in (a), can be created through the centerline of the cylinder. The intersection of the intermediate datum with the cylinder creates a datums axis that is used to locate the final datum plane.



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5.08.05 Using Arrays (Rectangular and Circular)

One method of creating multiple identical copies of a feature in a model is to create a **feature array**, which is sometimes called a **feature pattern**. A feature array takes one feature, called the **master feature**, and places copies of it on the model at a specified spacing. The copied features are identical to the master feature, and changing the geometry of the master at a later time also changes the geometry of the copies at that time. Including features in this manner can save time and effort in creating the entire model, especially when the features are rather complex. An example of a model with a rectangular array of features is shown in Figure 5.50. An array of rectangular cuts is shown in Figure 5.51. As shown, rectangular arrays can generate copied features

FIGURE 5.50. A rectangular array of protrusions created from a master feature.

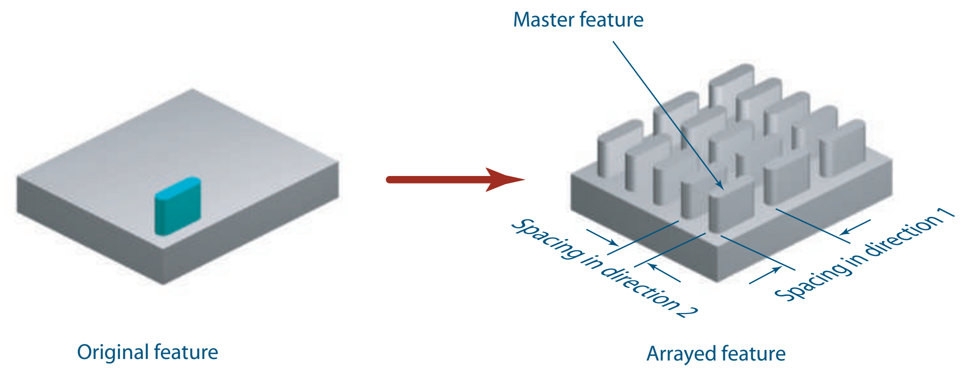

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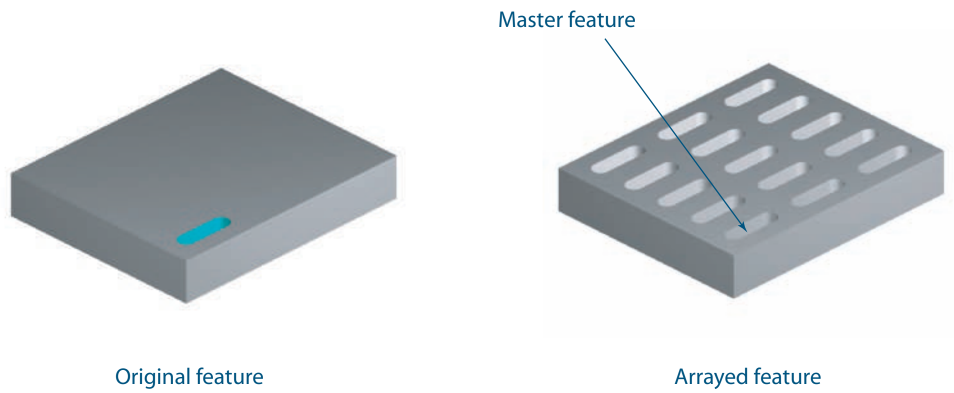
FIGURE 5.51. A rectangular array of cuts created from a master feature.



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in two directions. These directions must be specified, as well as the spacing of the copied features in each direction. Finally, the number of copies in each direction must be specified. Care must be taken to ensure that there is enough room on the model to accommodate all of the copied features.

Examples of models with radial arrays of protrusions are shown in Figure 5.52. Radial arrays can extend radially or axially. For radial arrays, in addition to the master feature being selected, the axis of revolution for the array must be selected. If such an axis does not already exist on the model, one must be created from an added datum axis. The number of copies, the direction of the array, and the radial and axial spacing of the copies must be specified.

5.08.06 Using Mirrored Features

Another method of creating a feature, when applicable, is to create its mirrored image. To create a **mirrored feature**, you must first identify a mirror plane. You can use an existing plane or define a new datum plane to use as the mirror plane, as shown in

FIGURE 5.52. A circular array of protrusions created from master features in the axial direction (a) and (b), both in the axial and radial direction (c).

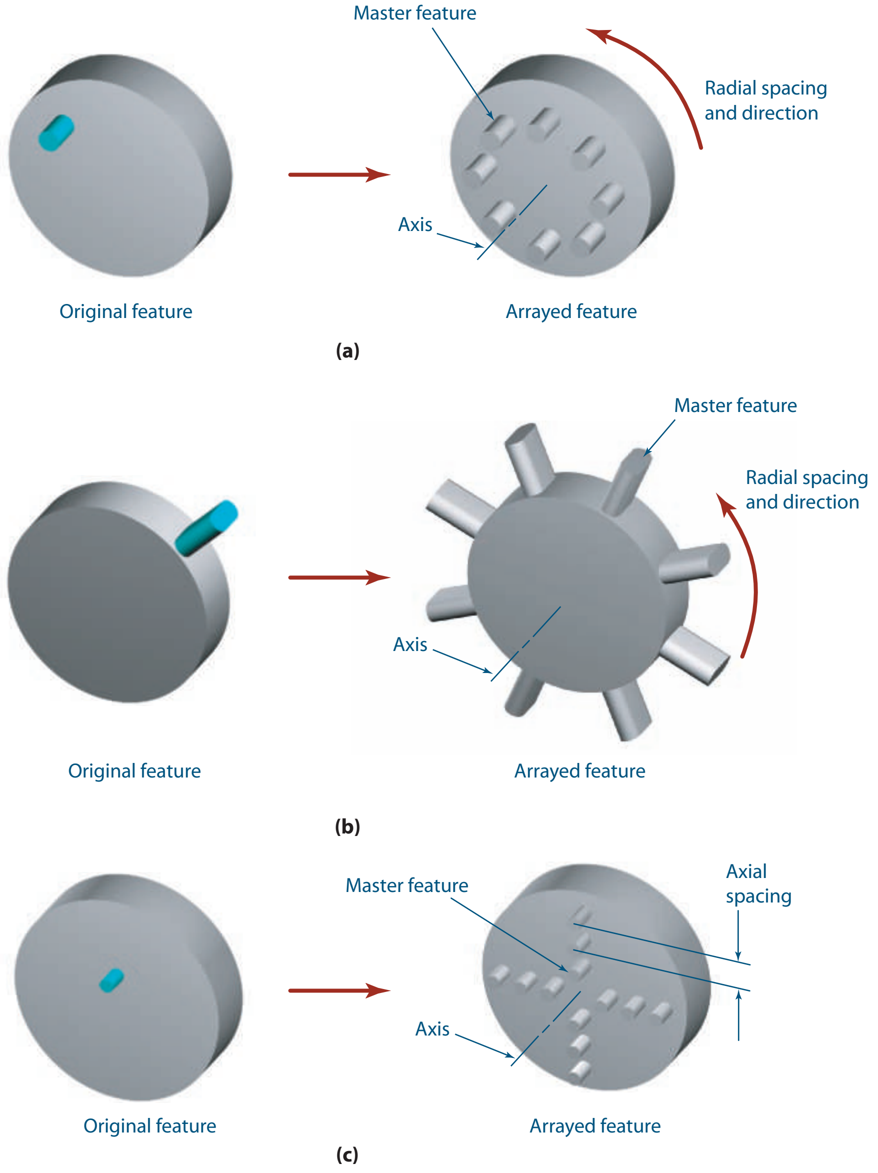

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Figure 5.53. A mirrored duplicate of the master feature can then be created on the model on the opposite side of the mirror plane. Mirrored features can be cuts or protrusions; however, keep in mind that the copied feature will be a mirror image of the master, not an identical copy. Changing the master feature at a later time also will change the mirrored feature correspondingly. As with arrayed features, using mirrored features can save a great deal of time in model creation, especially when the mirrored feature is complex.

5.08.07 Using Blends

Not all models can be created using just extruded or revolved features. One complex feature is a **blend**. Figure 5.54 shows models with blended surfaces. A blend requires at least two profile sketches, and the model is formed by a smooth transition between these profiles. The profiles can be sketched on the basic modeling planes, on surfaces of an existing model, or on datum planes. In the simplest blends, the profiles are sketched on parallel planes. Many software packages require the number of vertices on each of the sketched profiles to be equal. If your profiles do not have the same number of vertices, you will have to divide one or more of the entities to create additional vertices. In some sketching editors, circles include four vertices by default. The vertices in all profiles are usually numbered sequentially, and the software usually tries to match the vertices to create an edge between vertices with the same number, as shown in Figures 5.54a and 5.54b. Rotating the profiles or redefining the vertex numbering can control twisting of the blended transition, as shown in Figure 5.54c. Further control on the model transition usually can be performed by specifying the slope of the transition at each vertex for each shape.

FIGURE 5.53. Creation of a mirrored feature. The master feature in (a) is mirrored by creating a datum plane as a mirror plane (b). A mirror image of the feature is produced on the opposite side of the datum plane in (c), and the final result is shown in (d).

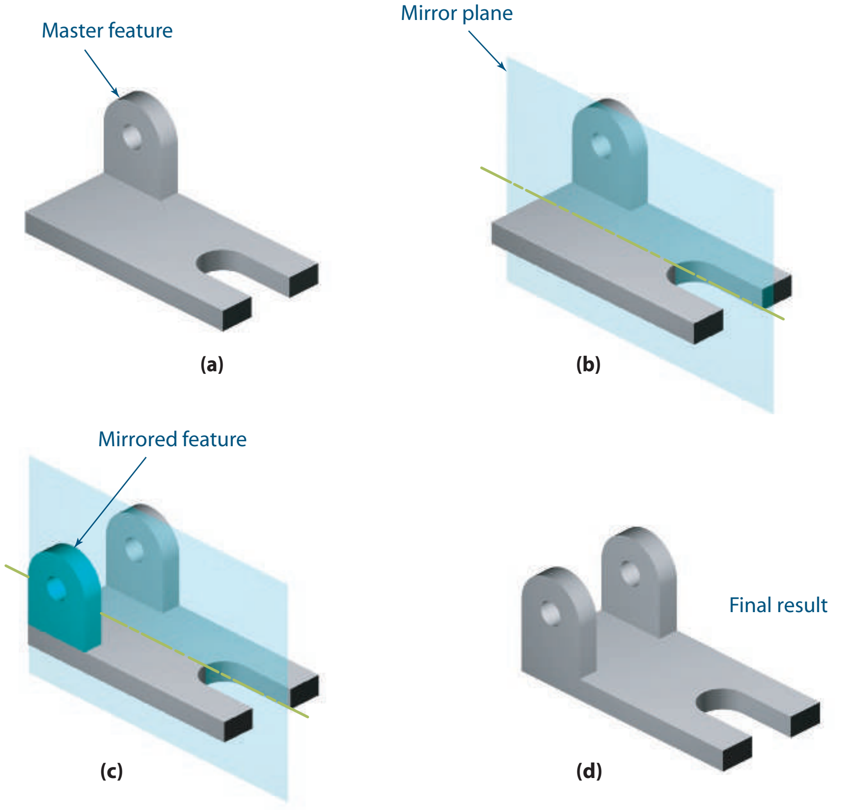

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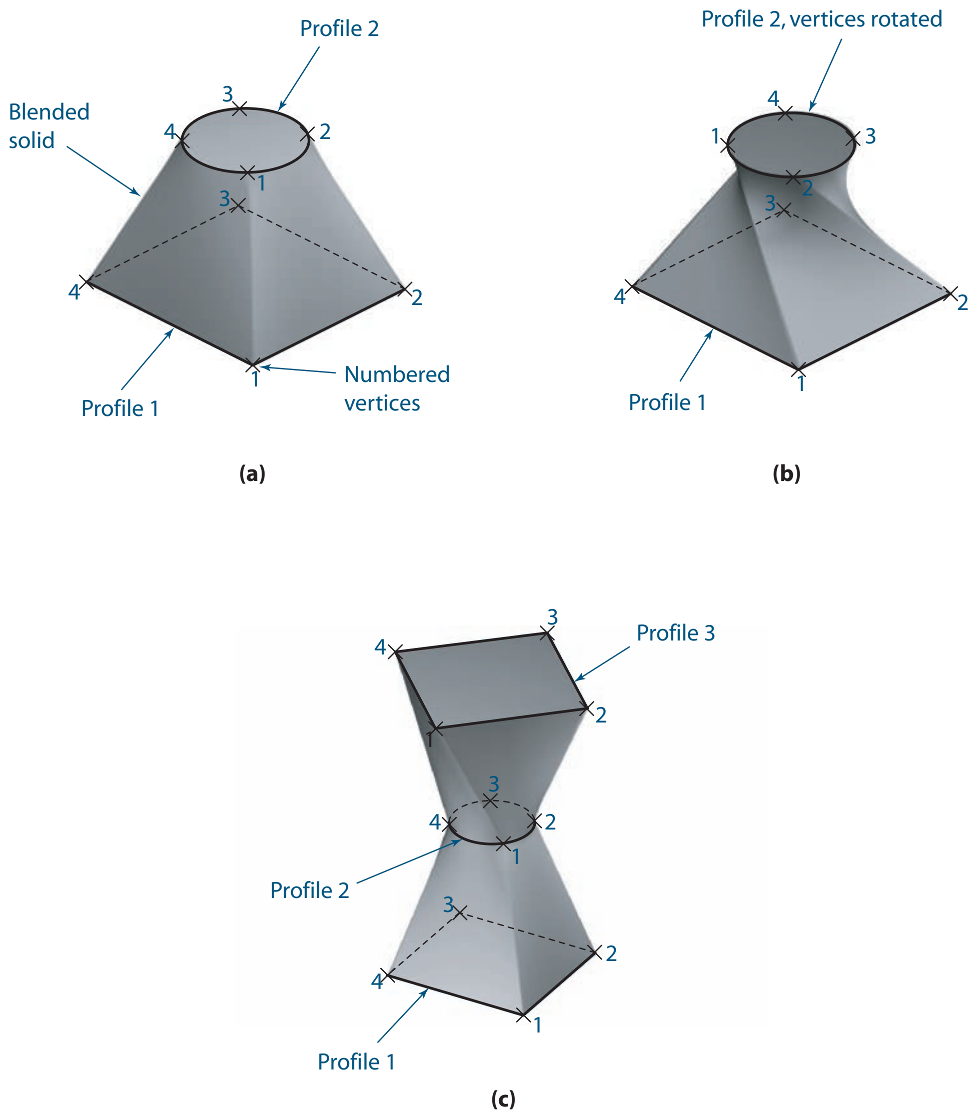
FIGURE 5.54. Blended solids created with two profiles in (a), with the same profiles but rotated vertices in (b), and with three profiles in (c).



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5.08.08 Sweeps

Swept features, as with simply extruded or revolved features, are created with a single profile. The difference is that a swept feature does not need to follow a linear or circular path, but can follow a specified curve called a **path** or **trajectory**. The profile is created at an endpoint of the path on a sketching plane that is perpendicular to the path at that endpoint. In sweeping out a solid volume, the profile is imagined to travel along the path. Usually, the profile is constrained to remain perpendicular to the path. A good example of a swept solid is a garden hose. The cross section or profile is a simple circle, but the path can be curved. Figure 5.55a shows the path and profile of a swept feature where the path is open. Figure 5.55b shows a swept feature where the path is closed. Care must be taken in defining the profile and path of a swept solid. Just as you cannot bend a garden hose around a sharp corner without creating a kink, if the path of your sweep contains a sharp corner or a small radius, the feature may fail by trying to create a self-intersecting solid. A special case of a swept solid is a coil

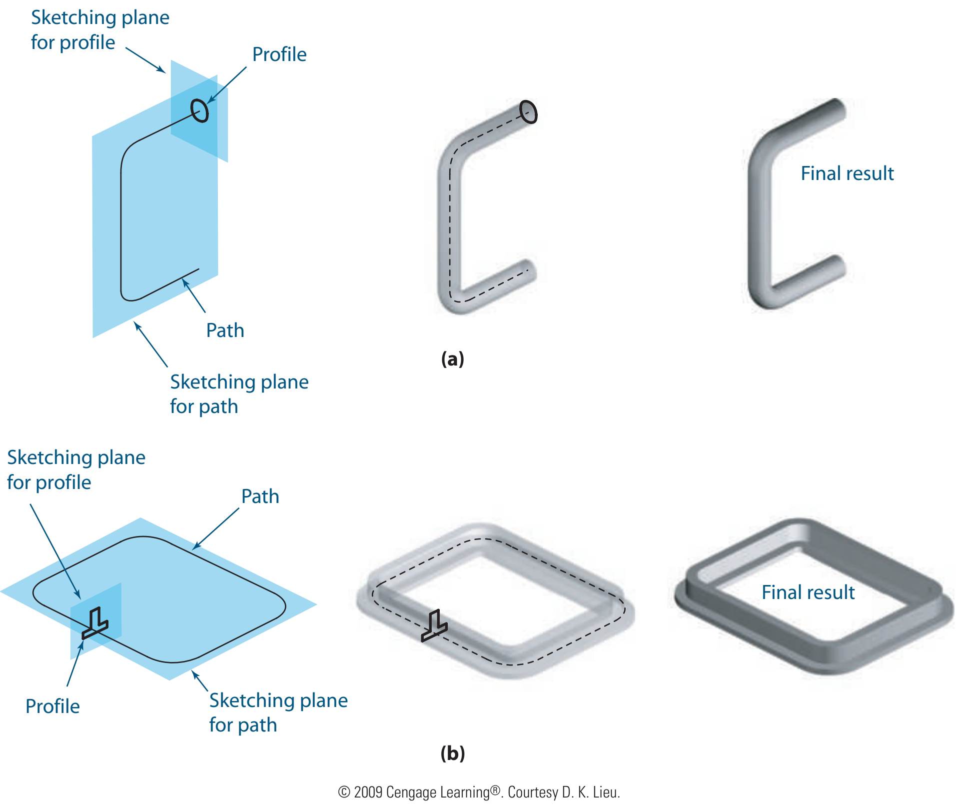


FIGURE 5.55. Features created by sweeps. The sketching plane is perpendicular to the path. The path in (a) is open, and the path in (b) is closed.

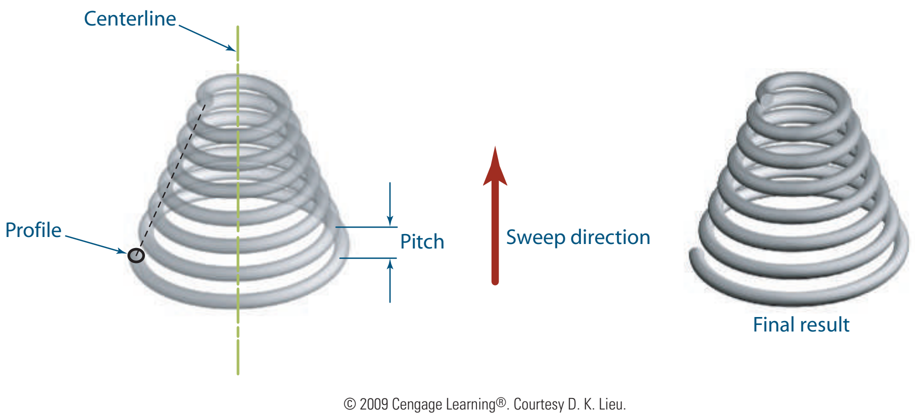


FIGURE 5.56. A tapered spring created by sweeping a circular profile on a helical path.

spring. In this case, the path is a helix, as shown in Figure 5.56. Many solid modelers include a helical sweep as a special feature so you do not have to sketch the helix. In this case, you sketch the profile and specify an axis on the sketching plane. The helix is specified by a pitch dimension, which is the distance between coils, and the direction of the sweep. To avoid self-intersection, the pitch must be larger than the maximum size of the profile in the sweep direction.

5.09 The Model Tree

An extremely useful editing tool included in most solid modeling software is the **model tree**, sometimes called the **feature tree**, **design tree**, or **history tree**. The model tree lists all of the features of a solid model in the order in which they were created, providing a "history" of the sequence of feature creation. Further, any feature in the model tree can be selected individually to allow the designer to edit the feature.

An example of a model tree and its associated solid model are shown in Figure 5.57. Usually, new features are added at the bottom of the model tree. Some software allows the designer to "roll back" the model and insert new features in the middle of the tree. In this case, the model reverts to its appearance just before the insertion point, so any inserted feature cannot have its geometry or location based on features that will be created after it.

The order in which features are created may have a profound effect on the results. In the previous example, a shell feature, which has the effect of hollowing out a part, was performed with the top surface of the part removed from the feature. A hole was then added to the model after the shelling operation. If the hole was added to the block before the shelling operation, the result would be different, as shown in Figure 5.58, because the surface around the hole through the block would have been considered a part of the shell. In most solid modeling software, removing the feature from one location in the model tree and inserting it in a new location changes the order of creation of the feature.

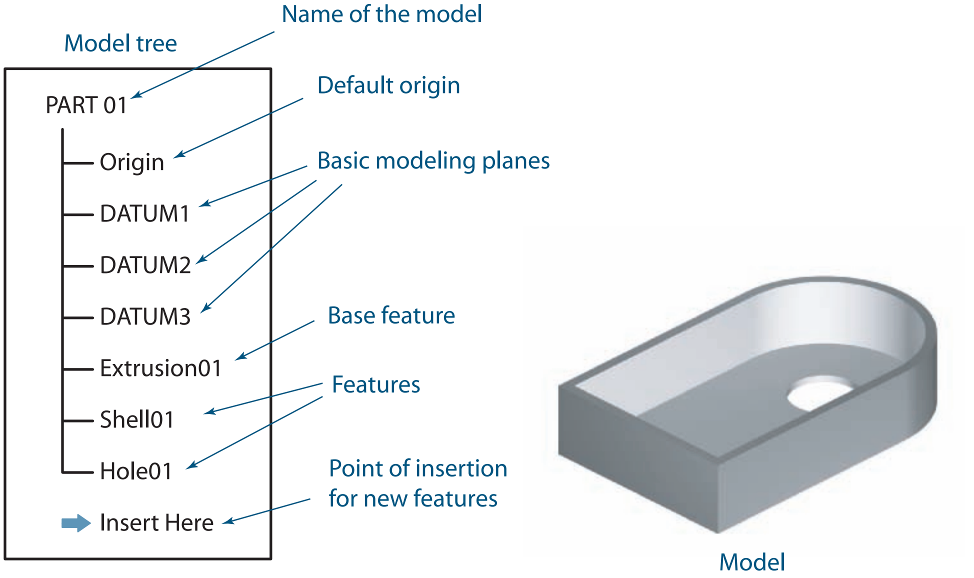


FIGURE 5.57. A typical model tree showing the features of a model in the order in which they were created.

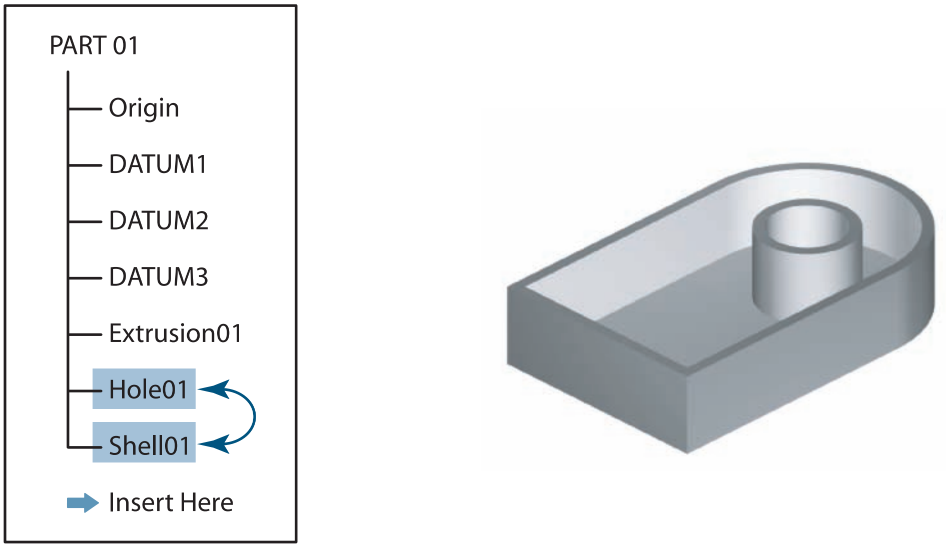
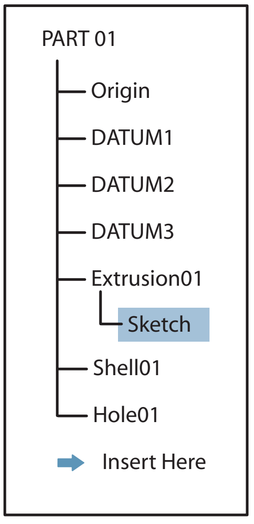


FIGURE 5.58. The result of reversing the order of creating the hole and shell features.

The model tree also provides access to the editing of features. Each feature item on the model tree can be expanded. The base extrusion in the previous example is composed of a fully constrained rectangular sketch profile that has been extruded to a specified length. The feature can be expanded in the model tree, as shown in Figure 5.59, to give access to the profile so it can be selected for editing. The sketch can then be edited by restarting the sketching editor. The dimensional constraints can be changed by selecting and editing their numerical values. Access to the sketching editor and feature parameters may vary with different software, and changes made through the model tree may be one of several different ways to modify your model.

In many models, certain features are dependent upon the existence of other features. For example, consider the features shown in the model in Figure 5.60. The location of the counterbored hole is measured from the edges of the rectangular base. However, the location of the countersunk hole is measured from the location of the counterbored hole and the location of the straight hole is measured from the location of the countersunk hole. Imagine what would happen to the straight hole if the countersunk hole were deleted. There would be no reference for placing the straight hole; therefore, it could not be created. Similarly, if the counterbored hole was deleted, neither the countersunk hole nor the straight hole could be created. This relationship is often referred to as a parent-child relationship. The straight hole is considered the **child feature** of the countersunk hole, and the countersunk hole is considered the child of the counterbored hole. The counterbored hole is considered the **parent feature** of the countersunk hole, and the countersunk hole is considered the parent of the straight hole. Just as you would not be reading this text if your parents did not

FIGURE 5.59. Use of the model tree to access and edit the sketch used to create the base feature (Extrusion01).



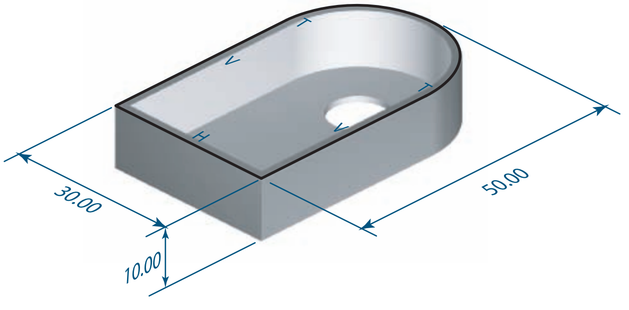
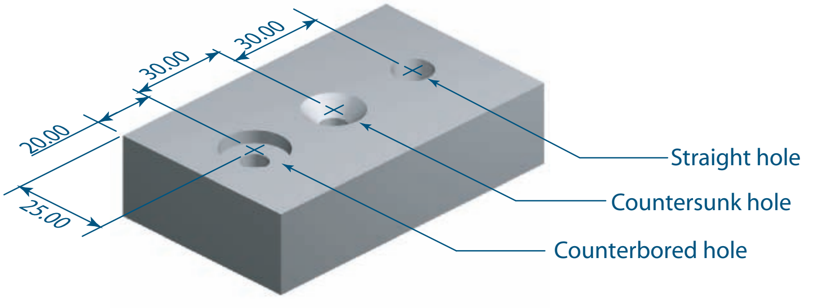


FIGURE 5.60. The holes in the model show parent-child dependencies. The existence of the straight hole depends on the existence of the countersunk hole, which depends on the existence of the counterbored hole. Elimination of a parent also eliminates its child.



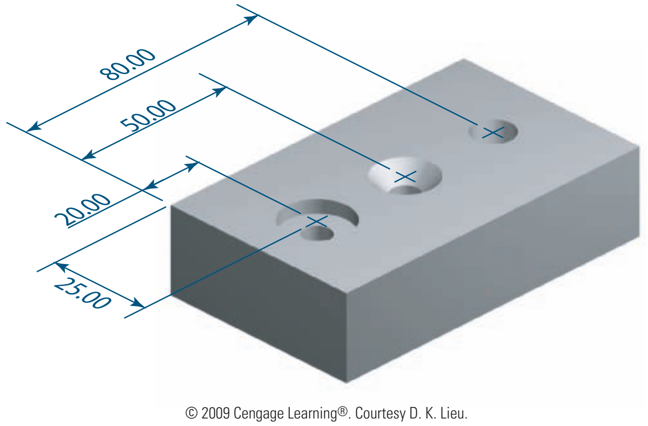


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FIGURE 5.61. This model demonstrates horizontal modeling. Each hole has no parent-child dependencies except to the base feature.

exist, neither can features in a solid model exist without their parent (or grandparent) features. On the model tree, if you try to delete a particular feature, its progeny also will be deleted. However, different software behaves differently; while some software provides specific warnings about the deletion of features, other software does not.

Understanding parent-child relationships in solid models is important if your model needs to be flexible and robust. As a designer, you undoubtedly will want to change the model at some time. You might need to add or delete features to accommodate a new function for the part or reuse the model as the basis of a new design. If you minimize the number of dependencies in the feature tree (like a family tree), it will be easier to make changes to your model. When it is likely that some features will be deleted or suppressed in a future modification of the part, those features should not be used as parents for other features that must remain present. The most extreme example of this strategy is called **horizontal modeling**, where the feature tree is completely flat; that is, there are no parent features except the base feature. This type of modeling strategy was patented by Delphi and has been used successfully by many companies. In Figure 5.61, the locations of three holes have been redefined so they are measured from the edge of the rectangular base instead of relative to one another. The base then becomes the parent to all three holes, and deleting any one of the holes does not affect the others.

5.10 Families of Parts

Groups of engineered parts often have very similar geometry. An everyday example is bolts and screws. A group of bolts may have the same head and thread geometries, but differ in their available length. Another example is the family of support brackets shown in Figure 5.62. Each bracket has a rough L-shaped base feature, holes, and a support rib (except for Version 3). Only the size and number of holes are different for each version.

When a group of parts is similar, it is possible to represent the entire group with a **family model**, with different versions of that model selected to specify particular parts. Such a model includes a **master model**, which has all of the features that are in any of the members of the group, and a **design table**, which lists all of the versions of that model and the dimensional constraints or features that may change in any of its versions. The attributes that may change are sometimes called **parameters**. The first step in building a family model is to identify all of the features and parameters that can be varied in the members of the family. In addition to a numerical value, every dimensional constraint in a model has a unique **dimension name**, which can be shown by selecting the appropriate display option. In Figure 5.63, all of the dimensional constraints have been changed to show their dimension names and the features have been identified by the feature names that appear on the design tree.

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FIGURE 5.62. A family of three parts with similar features and geometry.

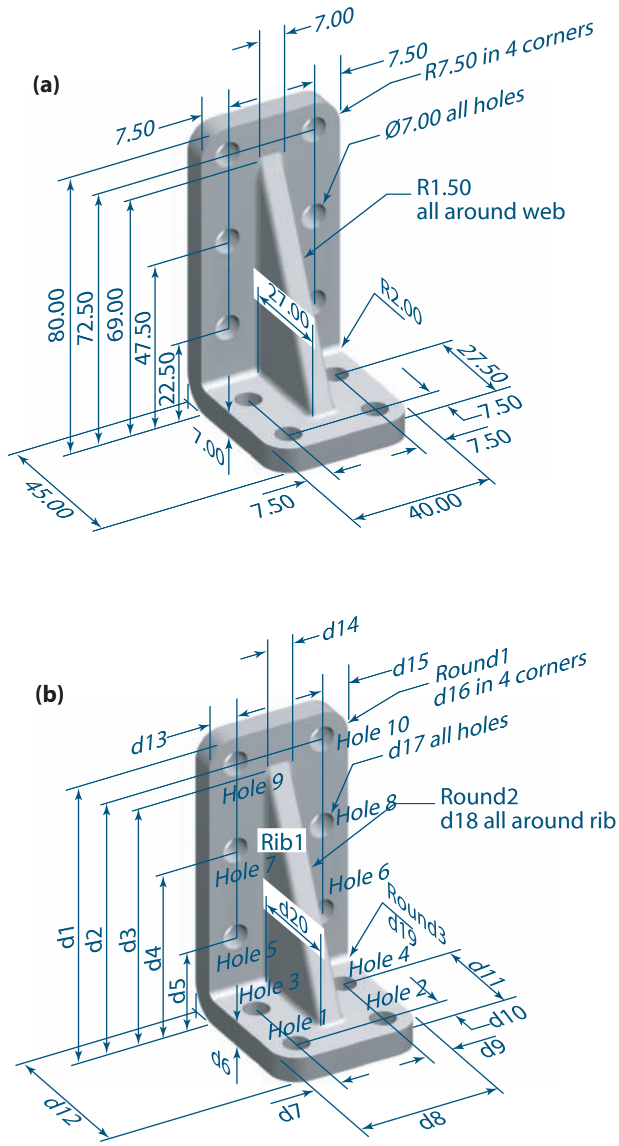

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FIGURE 5.63. The master model showing the numerical values of its dimensions in (a) and the names of the features and dimensions in (b).

The next step is to select the option for the construction of a design table, which is usually an internal or external spreadsheet, in the solid modeling software. The spreadsheet table should look similar to that shown in Figure 5.64. The first column usually contains the names of the different versions of the model. In Figure 5.64, these versions are called Version 1, Version 2, and Version 3 for convenience. The first row

FIGURE 5.64. The design table for the parameters that change within the three versions of the family of parts in Figures 5.62 and 5.63.

	Hole 3	Hole 4	Hole 7	Hole 8	Hole 9	Hole 10	Rib1	d1	d3	d12
Version 1	U	U	U	U	U	U	U	80.00	69.00	45.00
Version 2	U	U	U	U	S	S	U	60.00	50.00	45.00
Version 3	S	S	S	S	S	S	S	30.00	N/A	30.00

S = Suppressed
U = Unsuppressed

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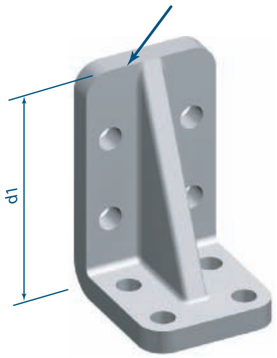
usually contains the names of the parameters that can change with each version. The individual cells of the spreadsheet show what the corresponding numerical values are of the dimensional constraints for each version and whether a particular feature is present in that version. When a particular feature is present, it is specified as being **unsuppressed**. When that feature is not present, it is specified as being **suppressed**. When the version of the part to be displayed has been selected, the corresponding model with its specified parameters is shown.

With the existence of a design table, editing the values in the table can change the numerical values of those dimensional constraints for any model version. In Figure 5.65, selecting and editing the appropriate cell in the design table changed the height of the L-bracket. In Figure 5.66, the support rib is no longer present because it was suppressed in the design table. When suppressing a feature, remember to be cautious, because suppressing a feature will also suppress its entire progeny.

FIGURE 5.65. The height of the L-bracket in the model has been changed by changing the value of the cell in the design table associated with this parameter.



	Hole 3	Hole 4	Hole 7	Hole 8	Hole 9	Hole 10	Rib1	d1	d3	d12
Version 1	U	U	U	U	U	U	U	69.00	69.00	45.00
Version 2	U	U	U	U	S	S	U	60.00	50.00	45.00
Version 3	S	S	S	S	S	S	S	30.00	N/A	30.00

S = Suppressed
U = Unsuppressed

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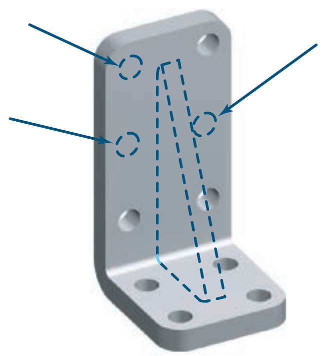
FIGURE 5.66. Features in the model can appear or not appear by changing their suppression states in the design table.



	Hole 3	Hole 4	Hole 7	Hole 8	Hole 9	Hole 10	Rib1	d1	d3	d12
Version 1	U	U	S	S	S	U	S	80.00	69.00	45.00
Version 2	U	U	U	U	S	S	U	60.00	50.00	45.00
Version 3	S	S	S	S	S	S	S	30.00	N/A	30.00

S = Suppressed
U = Unsuppressed

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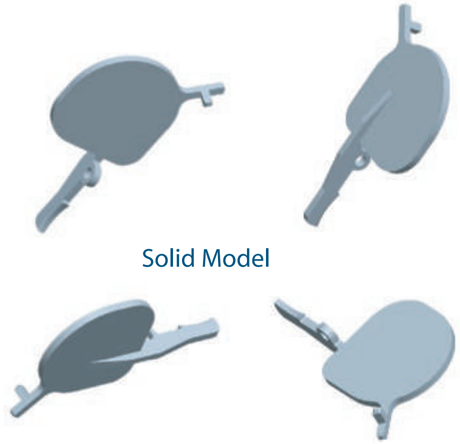
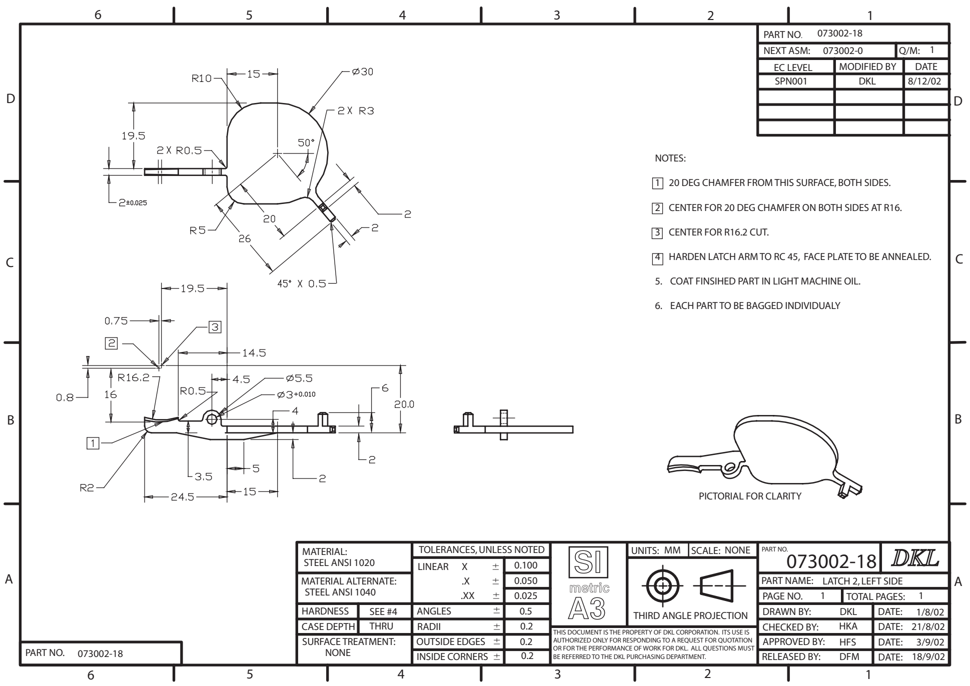
5.11 Extraction of 2-D Drawings

Nearly all solid modeling software packages have a facility for easily creating 2-D engineering drawings from solid models. Formal engineering drawing, which is covered in detail in later chapters, displays the part with all of its features in multiple predesignated views. It also displays the sizes and locations of the features. Solid modelers, which display a model from any viewpoint, can easily create the required views and display dimensions, thus greatly reducing the time and effort required to produce a drawing. Note that the dimensional constraints used in creating the solid model may be different from the dimension values that should be displayed on the engineering drawing. For example, the drawing is required to display all of the dimensions that are necessary for manufacturing the part. Some of these dimension values may be controlled by geometric constraints and are not included in the model as dimensional constraints. You will learn more about proper dimensioning practices in later chapters of this text. An example of a 2-D engineering drawing produced from a solid model is shown in Figure 5.67.

FIGURE 5.67. A typical solid model and a formal working drawing extracted from the model.



Solid Model



Formal Working Drawing

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5.12 Chapter Summary

In this chapter, you learned about features and parametric solid modeling. Features are distinctive shapes that compose a solid model. Parametric models have the capability to be modified by changing the sizes and other attributes of the features in the model. The history of solid modeling shows how CAD has evolved from wireframe to solid models and provides some insight regarding the strategies used to create solid models.

Part modeling can be a very complicated process, but some general strategies make it easier to create good, robust solid models. But before you go to the computer, you need to consider how the part model will be used. Later applications such as manufacturing and documentation will be easier when the part is modeled properly. Thus, you need to plan carefully and ask yourself some questions before the first feature is created: How can you decompose that complicated widget into simpler features that are available on your solid modeler? Which feature should you create first? Which features are related to each other? How is the part used? Manufactured? Can standard features such as holes be modeled to imbed design intent and/or manufacturing information in addition to simple geometric characteristics? How does the part fit into an assembly? What will the engineering drawing look like? These are just a few questions you need to consider before you begin. For now, you may not know the answers to all of these questions; but as you gain experience, you will develop an appreciation for the importance of building a robust solid model that captures your design intent.

The solid modeling process begins with identification of the features of the part, followed by selection of the base feature. Profiles for extrusion, rotation, sweeps, and blends are created with sketches and are controlled using different types of geometric, dimensional, and associative constraints. After the base feature is created, other features are added to the model; these features are dependent upon the base feature or other previously created features. Care must be taken in the creation of solid models to make flexible models that are robust and that can be used for purposes such as analysis, manufacturing, and documentation.

As modeling systems continue to develop, designers and engineers want to include more information in the model to better simulate the physical characteristics of the parts. These models, called behavioral models, might include features such as physical properties, manufacturing tolerances, surface finish, and other characteristics of the parts. Besides a person's appearance, you would need to know something about his or her education or physical abilities to determine whether the person might be able to do a particular job. Likewise, a designer or an engineer may need to know more than just the shape of an object or assembly model to determine whether it will perform its intended function. As they become more realistic and can simulate the actual behavior of the parts and assemblies, product models of the future will contain even more characteristics and features.

5.13 GLOSSARY OF KEY TERMS

algebraic constraints: Constraint that define the value of a selected variable as the result of an algebraic expression containing other variables from the solid model.

associative constraints: *See* algebraic constraints.

base feature: The first feature created for a part, usually a protrusion.

blend: A solid formed by a smooth transition between two or more profiles.

blind extrusion: An extrusion made to a specified length in a selected direction.

boundary representation (b-rep): A method used to build solid models from their bounding surfaces.

chamfers: Angled cut transitions between two intersecting surfaces.

child feature: A feature that is dependent upon the existence of a previously created feature.

constraints: Geometric relationships, dimensions, or equations that control the size, shape, and/or orientation of entities in a sketch or solid model.

constructive solid geometry (CSG): A method used to build solid models from primitive shapes based on Boolean set theory.

cosmetic features: Features that modify the appearance of the surface but do not alter the size or shape of the object.

cut: A feature created by the removal of solid volume from a model.

database: A collection of information for a computer and a method for interpretation of the information from which the original model can be re-created.

datum geometries: Geometric entities such as points, axes, and planes that do not actually exist on real parts, but are used to help locate and define other features.

datum planes: The planes used to define the locations of features and entities in the construction of a solid model.

design table: A table or spreadsheet that lists all of the versions of a family model, the dimensions or features that may change, and the values in any of its versions.

design tree: *See* model tree.

dimensional constraints: Measurements used to control the size or position of entities in a sketch.

dimension name: The unique alphanumeric designation of a variable dimension.

driven dimension: A variable connected to an algebraic constraint that can be modified only by user changes to the driving dimensions.

driving dimension: A variable used in an algebraic constraint to control the values of another (driven) dimension.

double-sided extrusion: A solid formed by the extrusion of a profile in both directions from its sketching plane.

extrude through all: An extrusion that begins on the sketching plane and protrudes or cuts through all portions of the solid model that it encounters.

extrude to a selected surface: An extrusion where the protrusion or cut begins on the sketching plane and stops when it intersects a selected surface.

extrude to the next surface: An extrusion begins at the profile and the protrusion or cut stops when it intersects the next surface encountered.

extrusion: A solid that is bounded by the surfaces swept out in space by a planar profile as it is pulled along a path perpendicular to the plane of the profile.

family model: A collection of different versions of a part in a single model that can display any of the versions.

feature array: A method for making additional features by placing copies of a master feature on the model at a specified equal spacing.

feature pattern: *See* feature array.

feature tree: *See* model tree.

feature-based solid modeling: A solid modeling system that uses features to build models.

features: Distinctive geometric shapes on solid parts; 3-D geometric entities that exist to serve some function.

fillets: Smooth transitions of the internal edge created by two intersecting surfaces and tangent to both intersecting surfaces.

form feature: A recognizable area on a solid model that has a specific function.

geometric constraints: Definitions used to control the shape of a profile sketch through geometric relationships.

graphical user interface (GUI): The format of information on the visual display of a computer, giving its user control of the input, output, and editing of the information.

ground constraint: A constraint usually applied to a new sketch to fix the location of the sketch in space.

history tree: *See* model tree.

holes: A cut feature added to a model that will often receive a fastener for system assembly.

horizontal modeling: A strategy for creating solid models that reduces parent-child relationships within the feature tree.

master feature: A feature or collection of features that is to be copied for placement at other locations in a model.

master model: In a collection of similar parts, the model that includes all of the features that may appear in any of the other parts.

mirrored feature: A feature that is created as a mirror image of a master feature.

model tree: A list of all of the features of a solid model in the order in which they were created, providing a "history" of the sequence of feature creation.

parameters: The attributes of features, such as dimensions, that can be modified.

parametric solid modeling: A solid modeling system that allows the user to vary the dimensions and other parameters of the model.

parametric techniques: Modeling techniques where all driven dimensions in algebraic expressions must be known for the value of the dependent variables to be calculated.

parent feature: A feature used in the creation of another feature, which is called its child feature.

path: The specified curve on which a profile is placed to create a swept solid.

primary modeling planes: The planes representing the xy-, xz-, and yz-planes in a Cartesian coordinate system.

primitives: The set of regular shapes, such as boxes, spheres, or cylinders that are used to build solid models with constructive solid geometry methods (CSG).

principal viewing planes: The planes in space on which the top, bottom, front, back, and right and left side views are projected.

profile: A planar sketch that is used to create a solid.

protrusion: A feature created by the addition of solid volume to a model.

regeneration: The process of updating the profile or part to show its new shape after constraints are added or changed.

revolved solid: A solid formed when a profile curve is rotated about an axis.

ribs: Constant thickness protrusions that extend from the surface of a part and are used to strengthen or stiffen the part.

rounds: Smooth radius transitions of external edges created by two intersecting surfaces and tangent to both intersecting surfaces.

shelling: Removing most of the interior volume of a solid model, leaving a relatively thin wall of material that closely conforms to the outer surfaces of the original model.

sketches: Collections of 2-D entities.

sketching editor: A software tool used to create and edit sketches.

sketching plane: A plane where 2-D sketches and profiles can be created.

solid model: A mathematical representation of a physical object that includes the surfaces and the interior material, usually including a computer-based simulation that produces a visual display of an object as if it existed in three dimensions.

splines: Polynomial curves that pass through multiple data points.

suppressed: Refers to the option for not displaying a selected feature.

surface model: A CAD-generated model created to show a part as a collection of intersecting surfaces that bound a solid.

swept feature: A solid that is bound by the surfaces swept out in space as a profile is pulled along a path.

trajectory: *See* path.

unsuppressed: Refers to the option for displaying a selected feature.

variational techniques: Modeling techniques in which algebraic expressions or equations that express relationships between a number of variables and constants, any one of which can be calculated when all of the others are known.

vertex: A point that is used to define the endpoint of an entity such as a line segment or the intersection of two geometric entities.

webs: Small, thin protrusions that connect two or more thicker regions on a part.

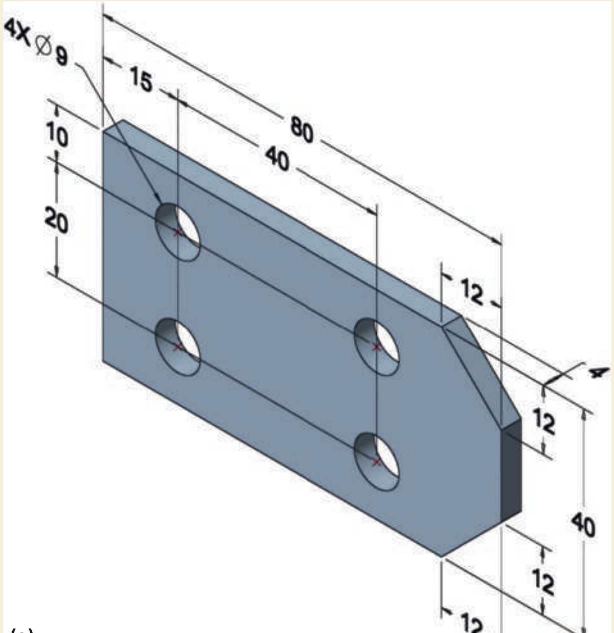
wireframe models: CAD models created using lines, arcs, and other 2-D entities to represent the edges of the part; surfaces or solid volumes are not defined.

5.14 QUESTIONS FOR REVIEW

1. What are some of the uses of solid models?
2. What is a feature?
3. Why are features important in solid modeling?
4. What types of features can be used as base features for your solid models?
5. Why are wireframe models inferior to solid models?
6. What are the steps in creating a solid model?
7. What are some errors that make a sketch invalid for creating a solid?
8. Why is it necessary to constrain a 2-D sketch?
9. What are the different types of geometric constraints?
10. What are associative constraints?
11. What are dimensional constraints?
12. What does it mean when a feature is a child of another feature? A parent of another feature?
13. What are some errors that constitute poor modeling practices?
14. What are some examples of good modeling strategies?

5.15 PROBLEMS

1. Using a single extrusion, create each the following object with a single closed-loop profile using the 2-D drawing capabilities of your solid modeling software. Define the geometry and sizes precisely as shown, using the necessary geometric constraints. Do not over- or underconstrain the profiles.



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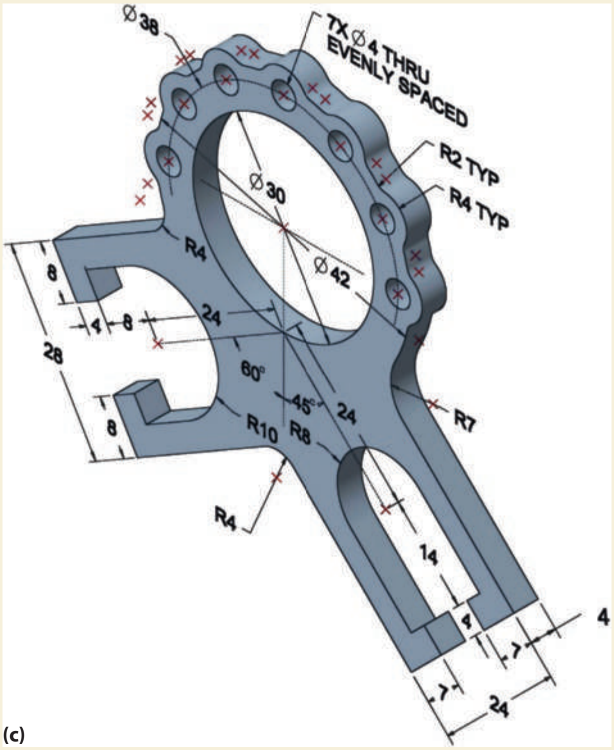

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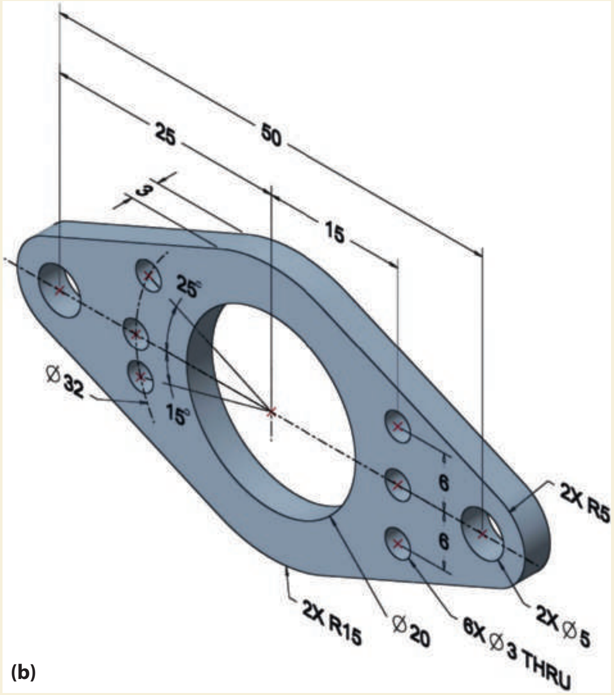

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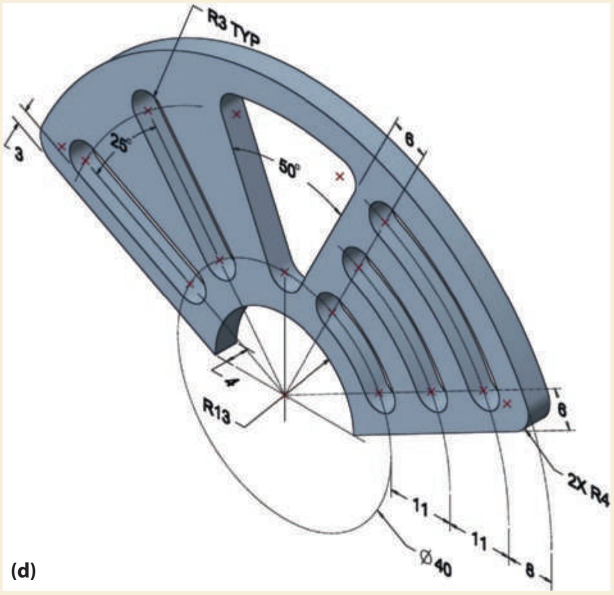

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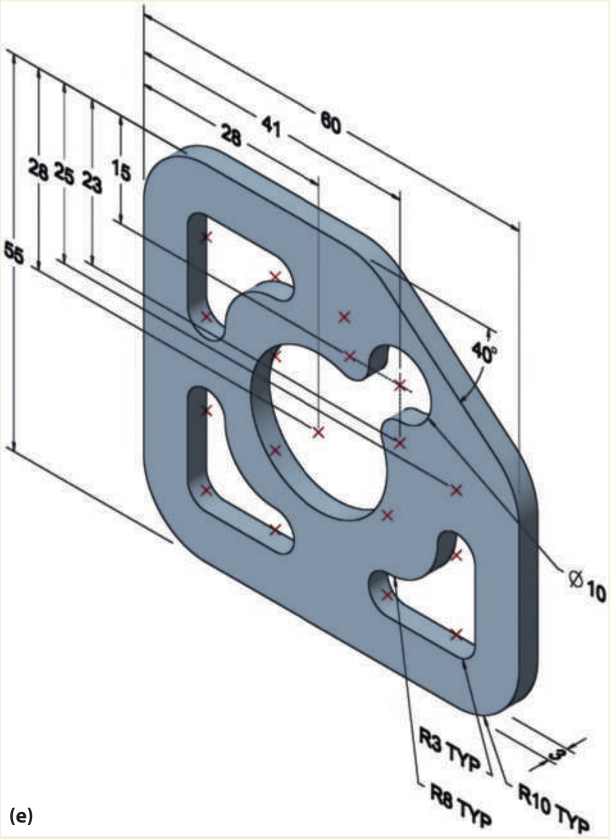
5.15 PROBLEMS (CONTINUED)



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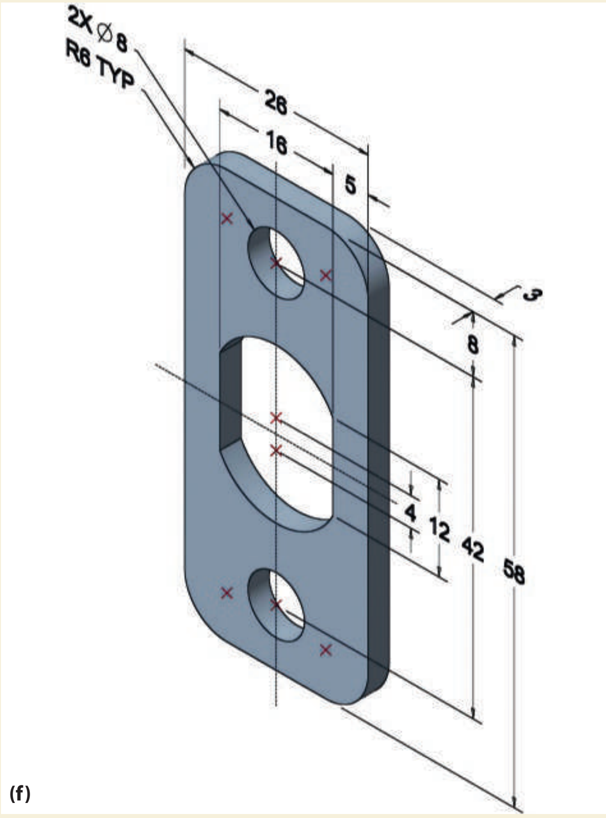

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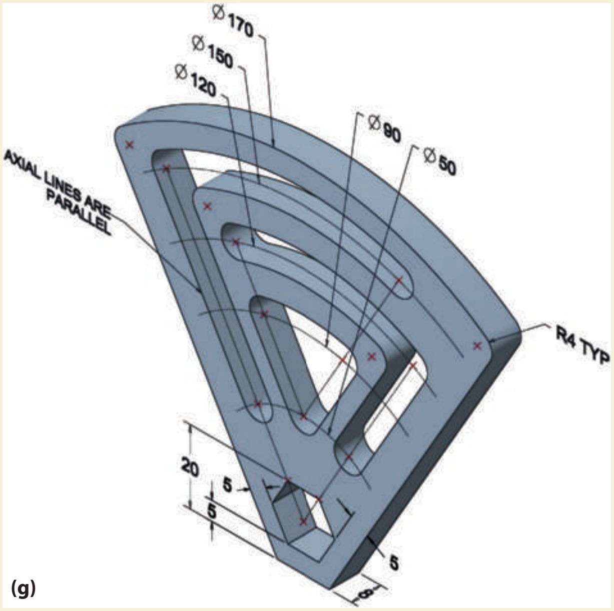

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FIGURE P5.1.

5.15 PROBLEMS (CONTINUED)

2. Study the following closed-loop profiles for which geometric constraints have not been added. Number each segment of the profiles and specify the necessary geometric constraints on each segment to create the final profile. Do not over- or underconstrain the profiles.

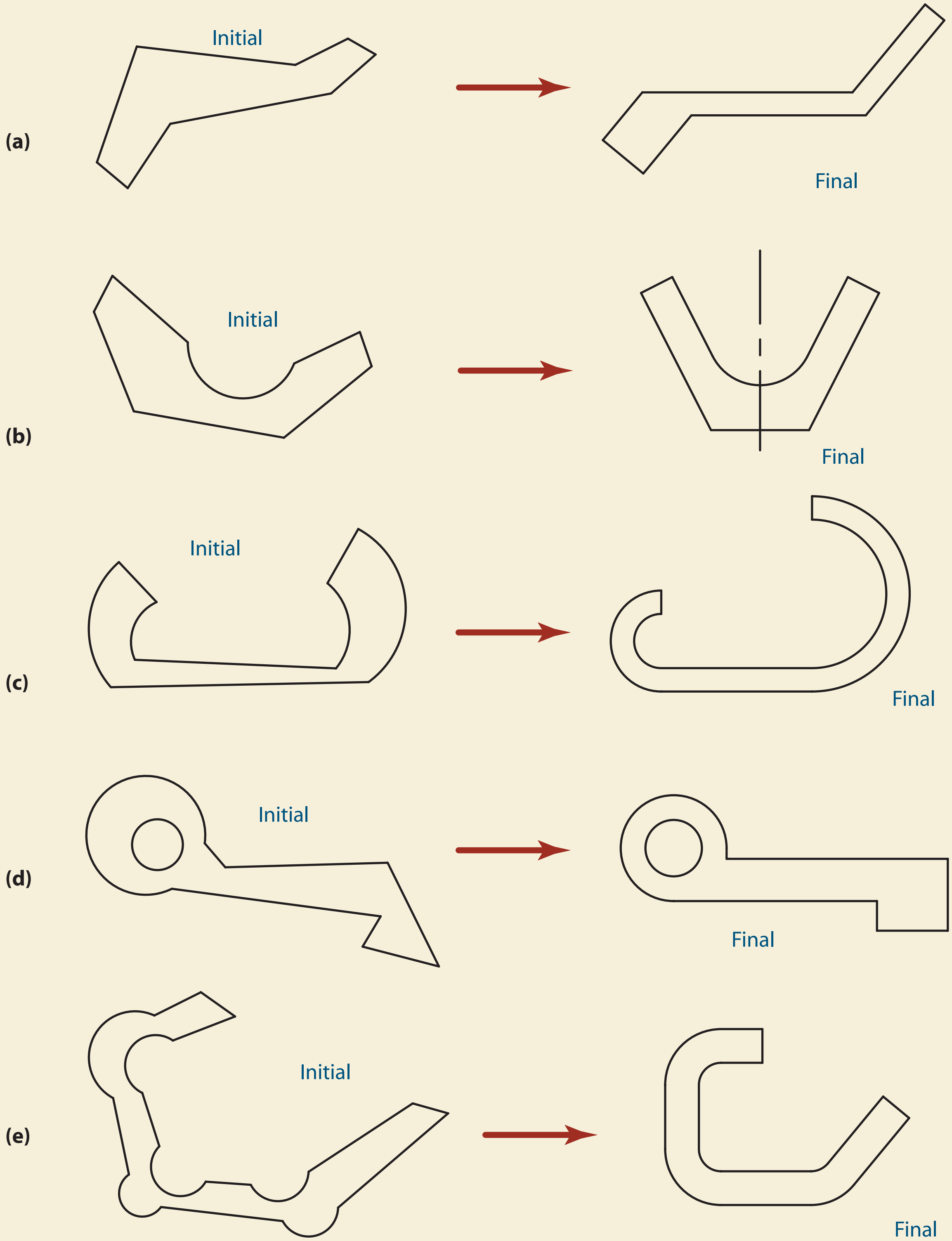

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FIGURE P5.2.

5.15 PROBLEMS (CONTINUED)

3. At first glance, these profiles may appear to be missing key dimensions. However, they are fully constrained by the addition of geometric constraints. Number each segment of the profiles. What were the geometric constraints used for each segment?

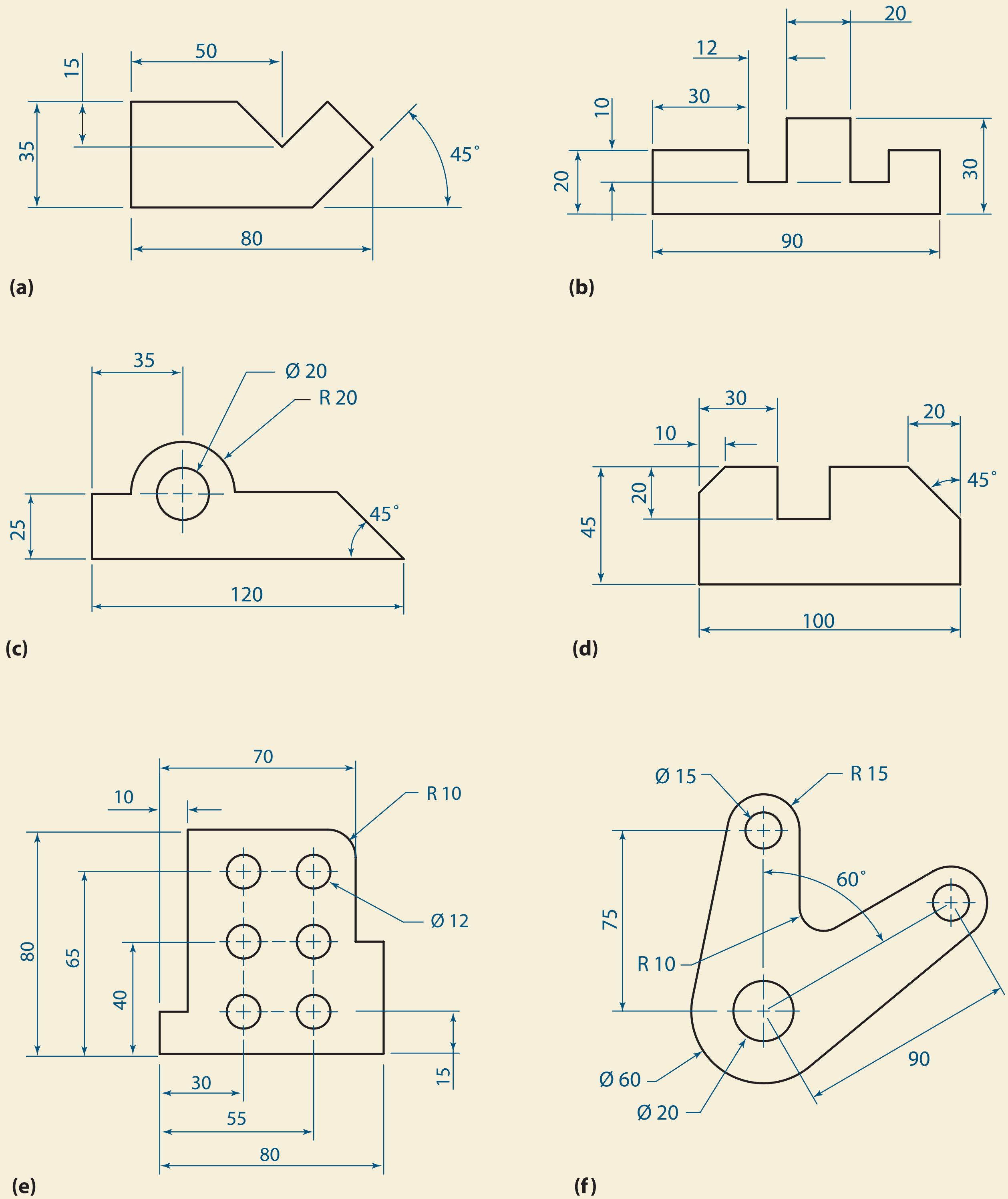

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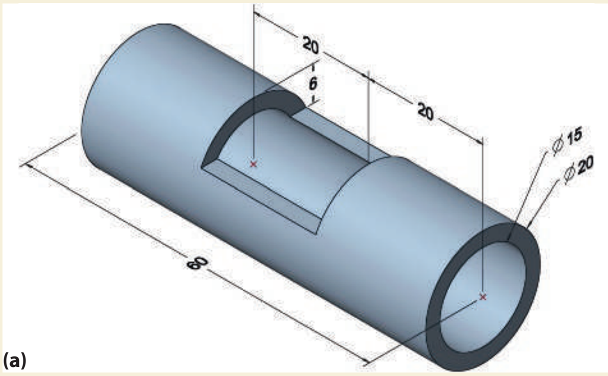
FIGURE P5.3.

5.15 PROBLEMS (CONTINUED)

4. Create solid models of the following parts in your CAD system. Identify what you consider to be the base geometry for each part. Are any (child) features dependent upon the existence of other (parent) features? If so, specify the hierarchy.



(a)

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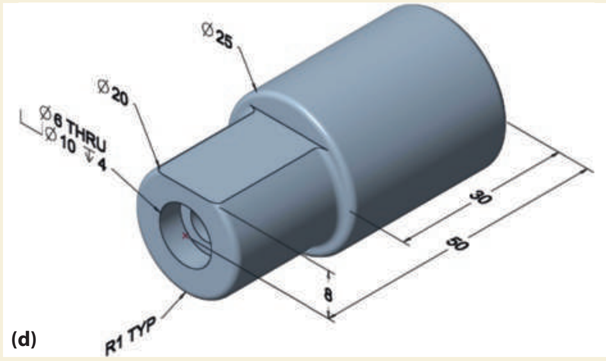

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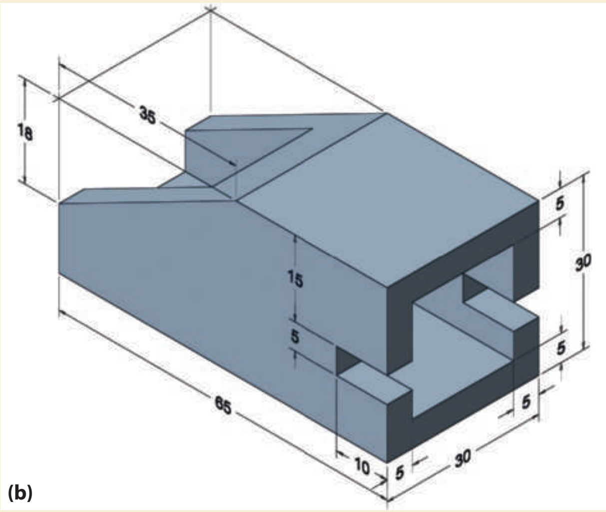

(b)

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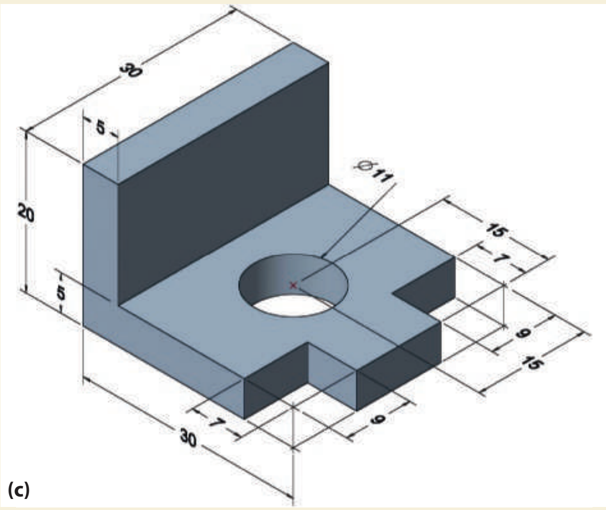

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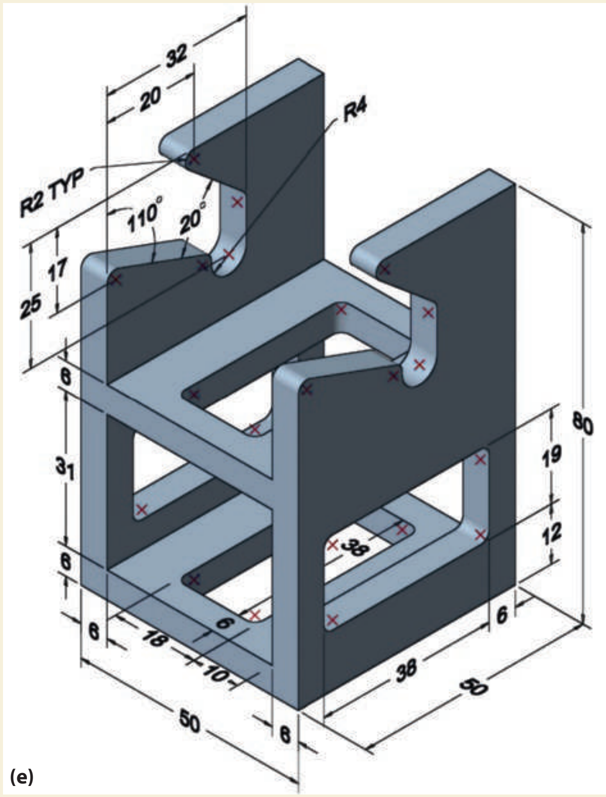

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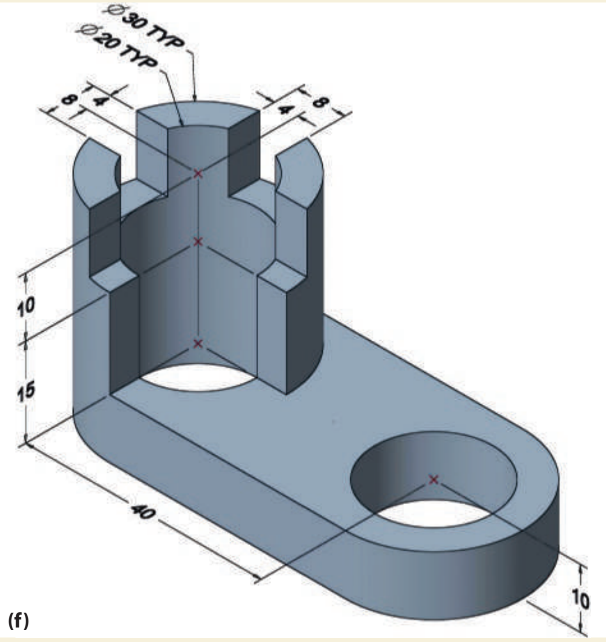
5.15 PROBLEMS (CONTINUED)



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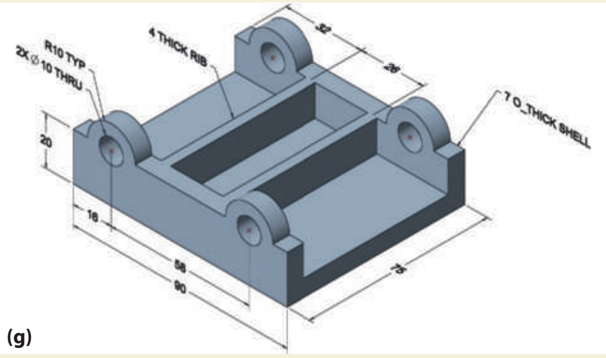

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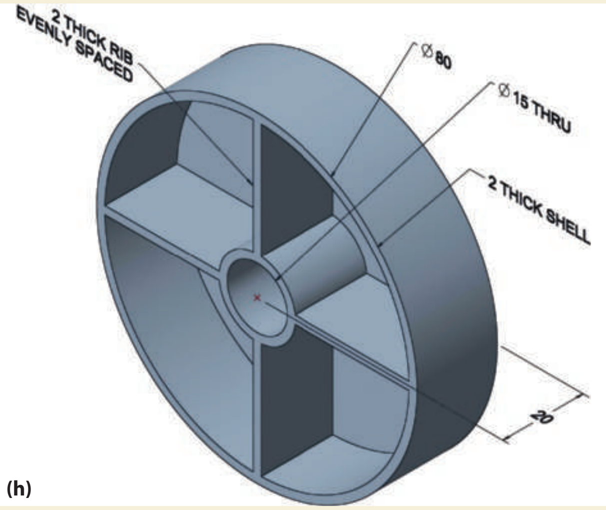

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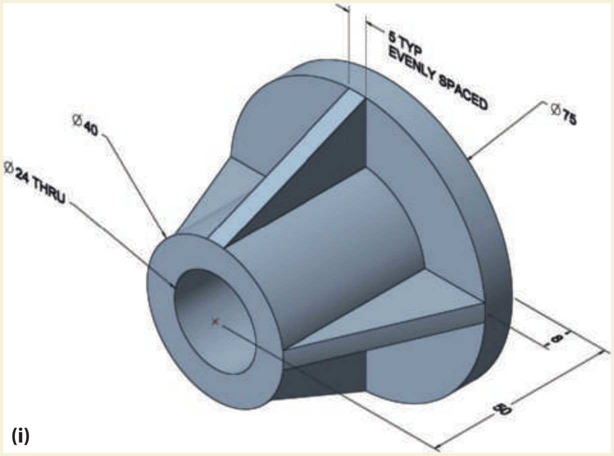

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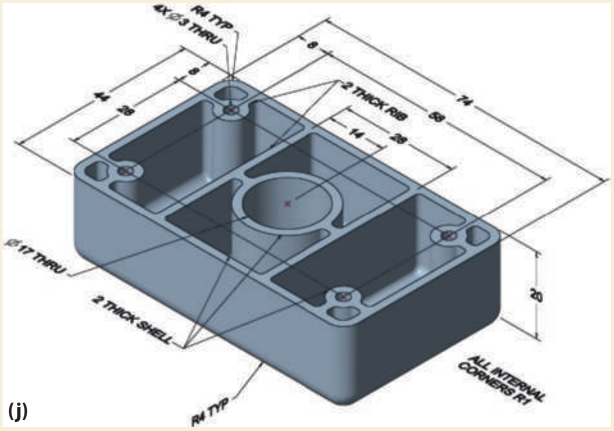

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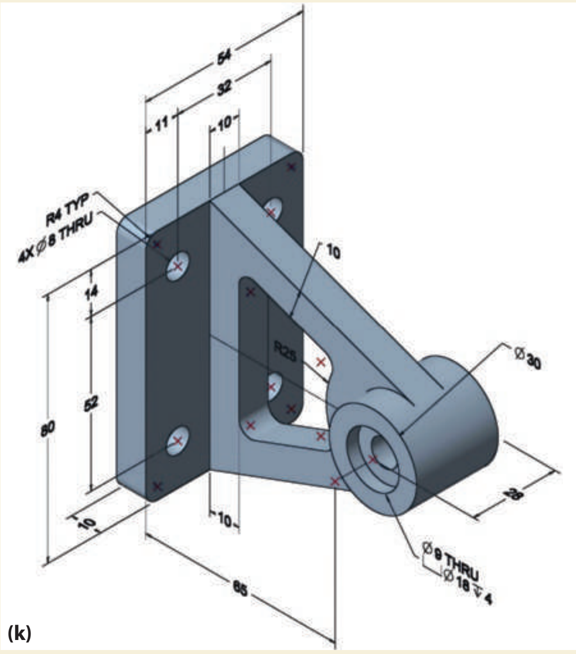

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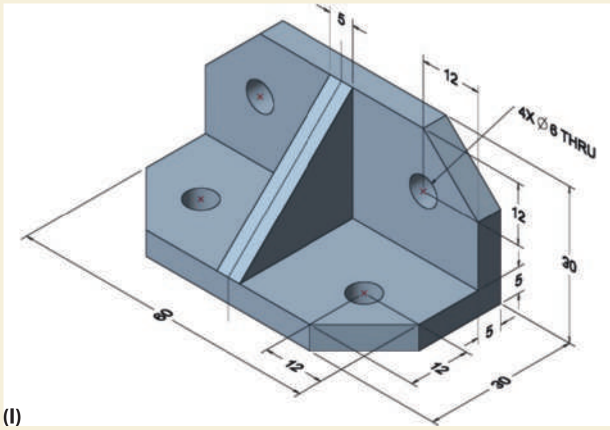
5.15 PROBLEMS (CONTINUED)



(l)

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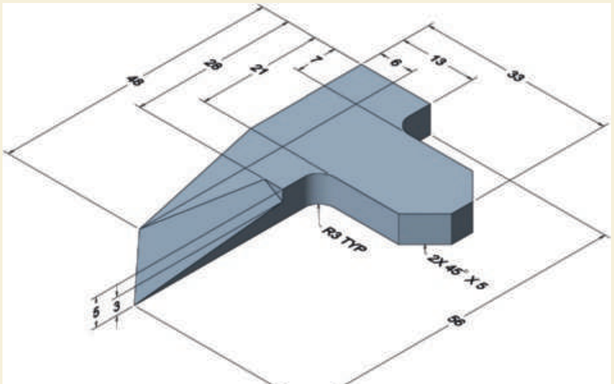

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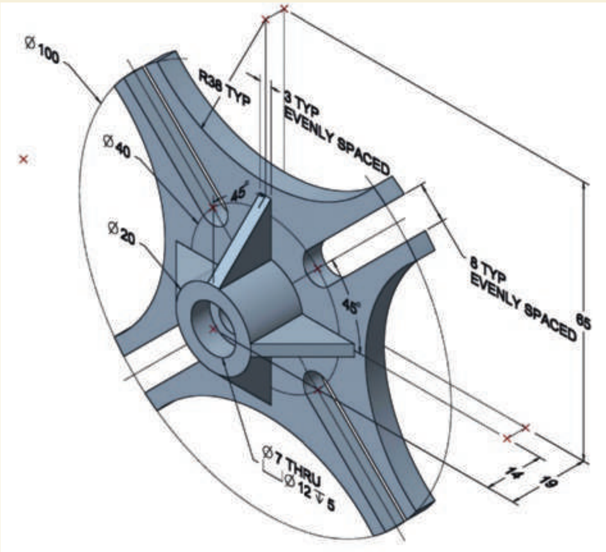

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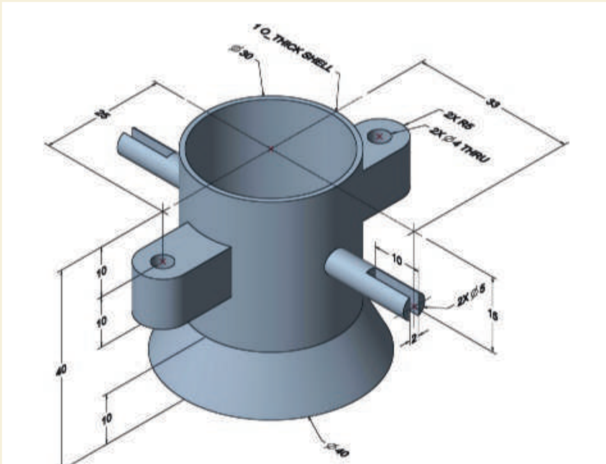

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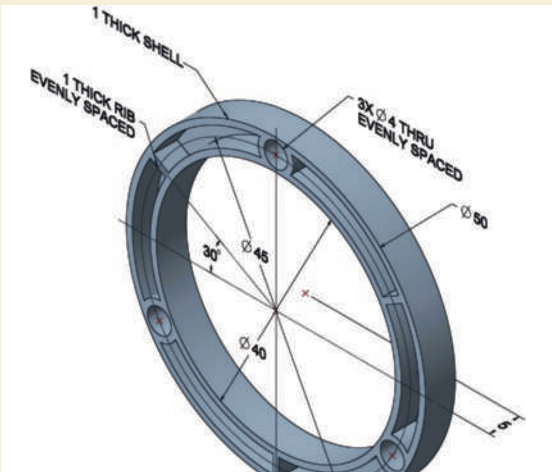

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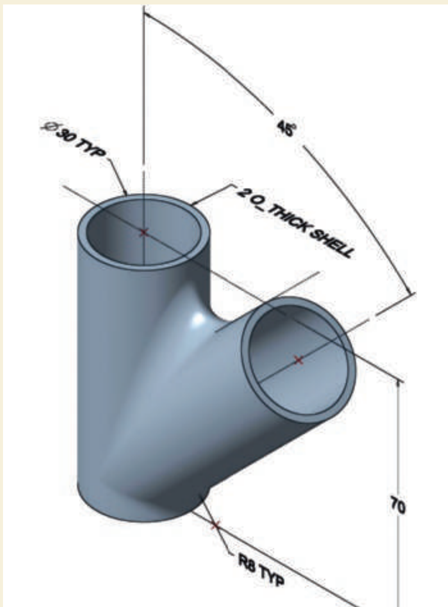

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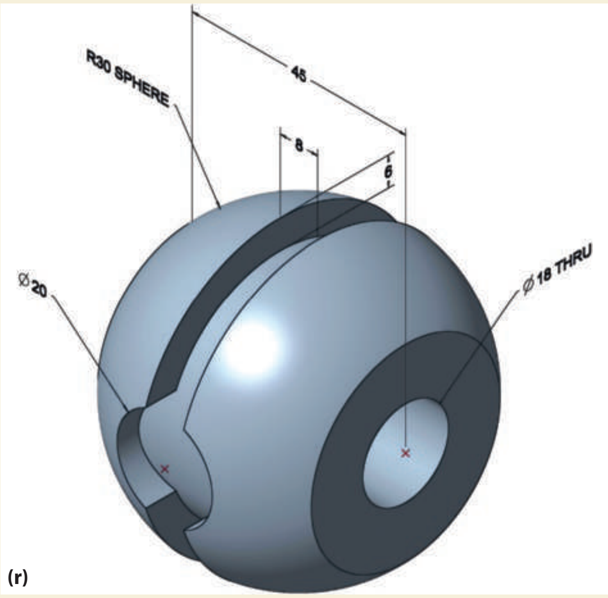

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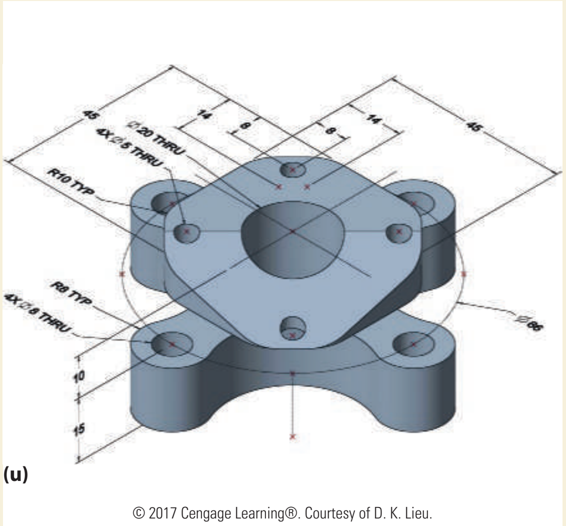

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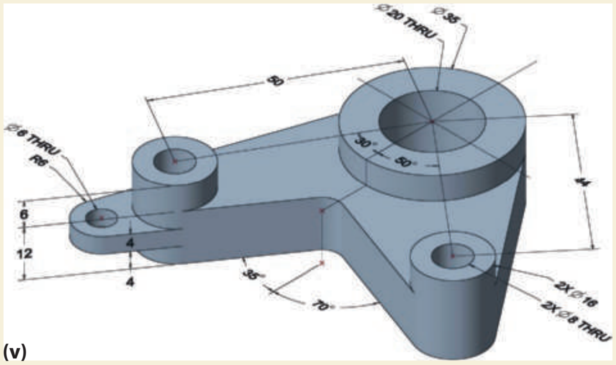

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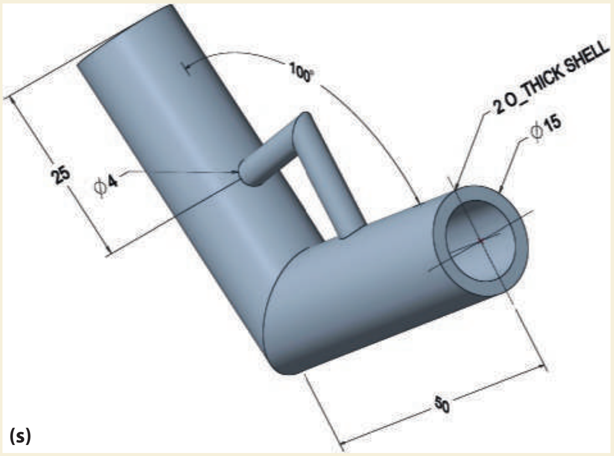

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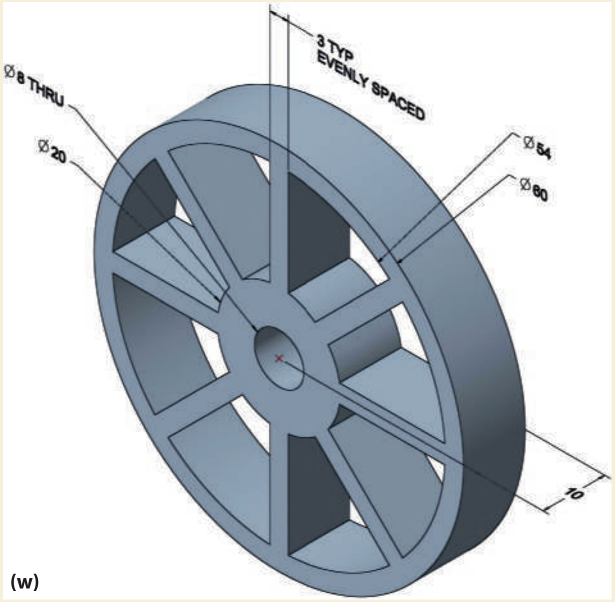

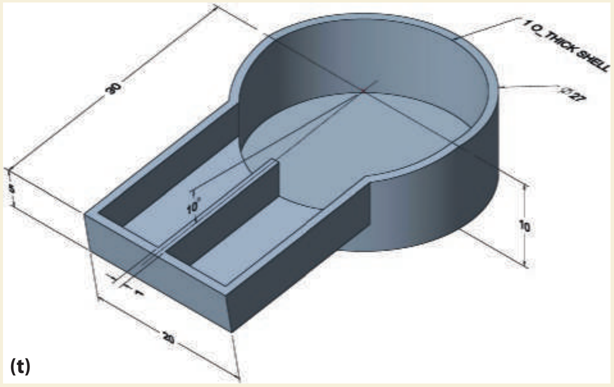

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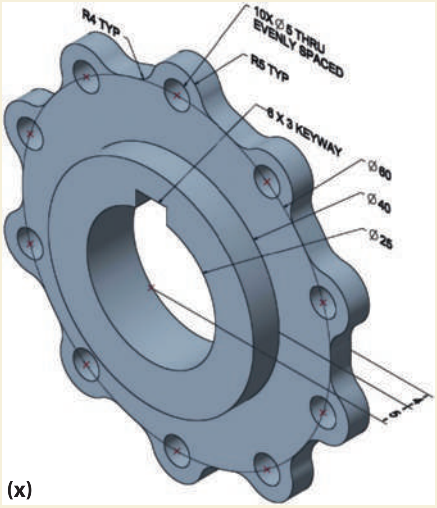

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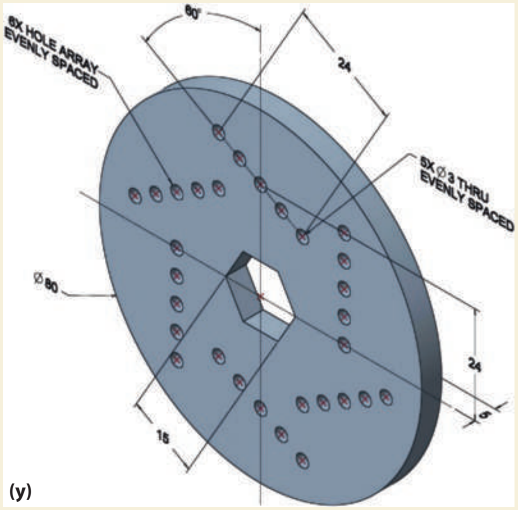

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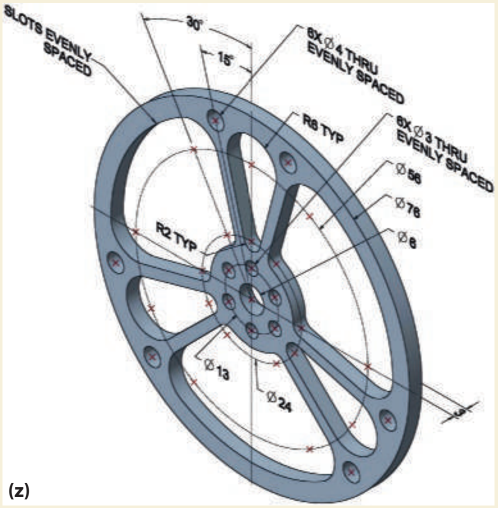

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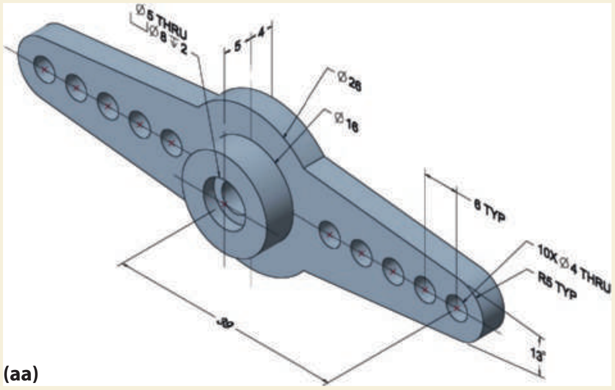

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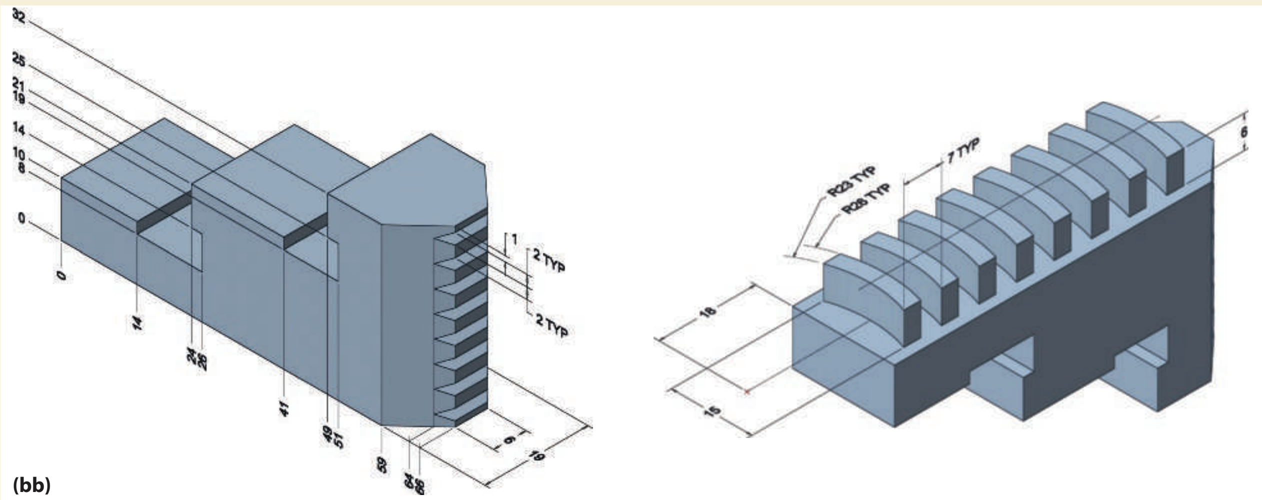

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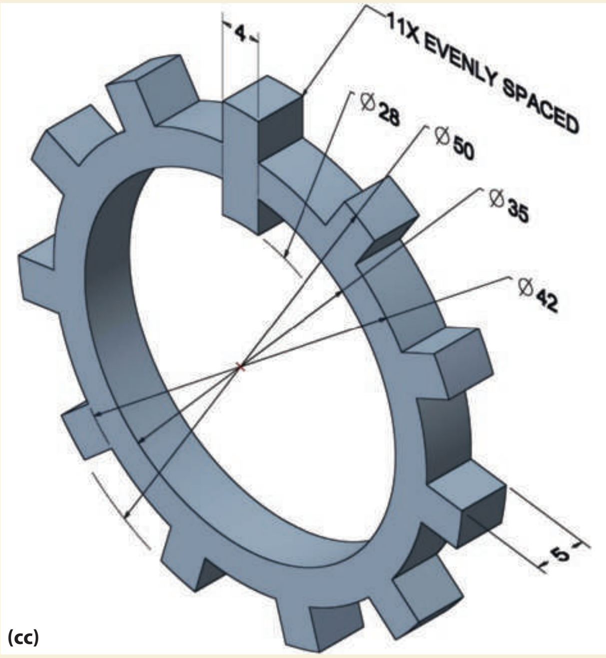

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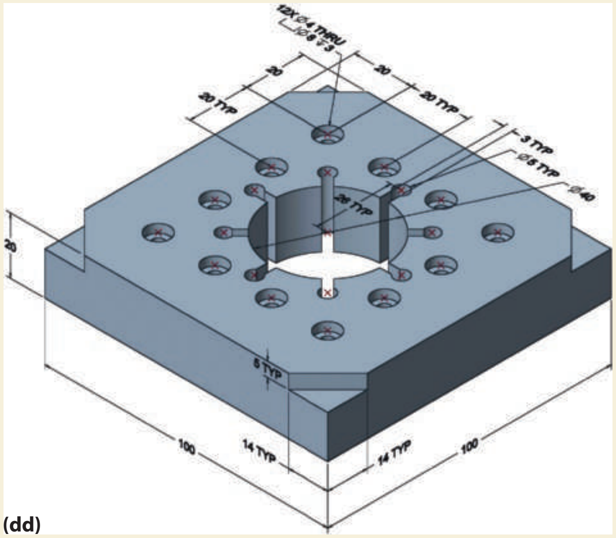

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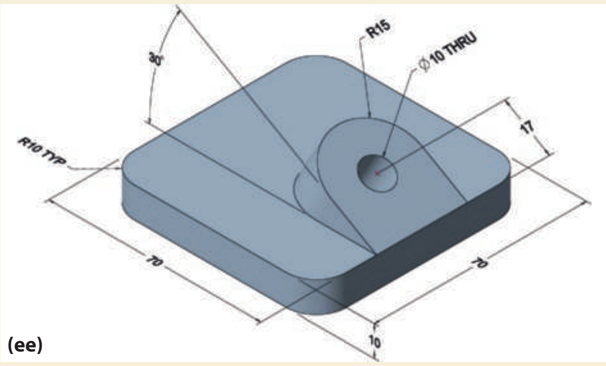
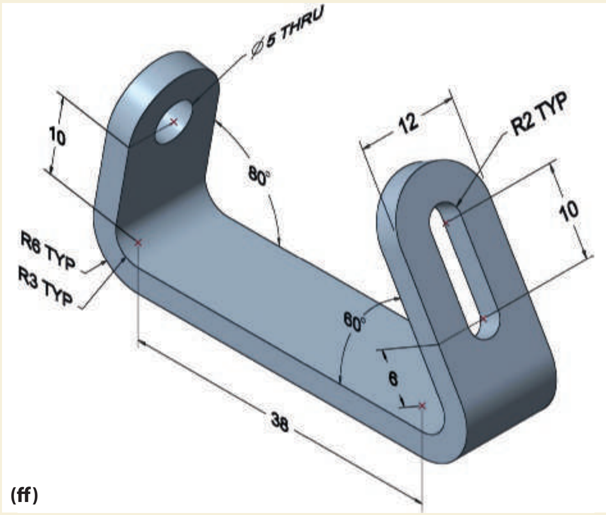


(ee)



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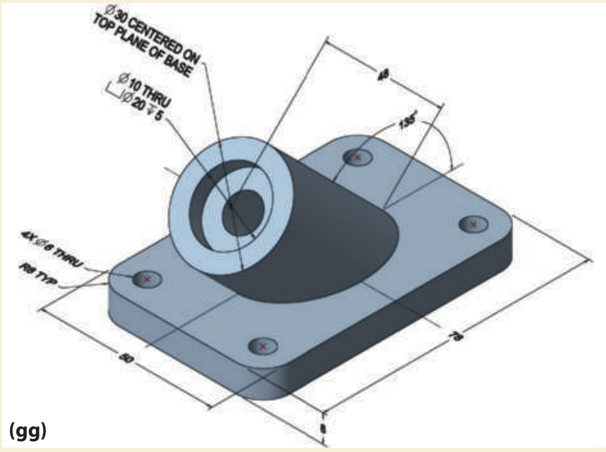
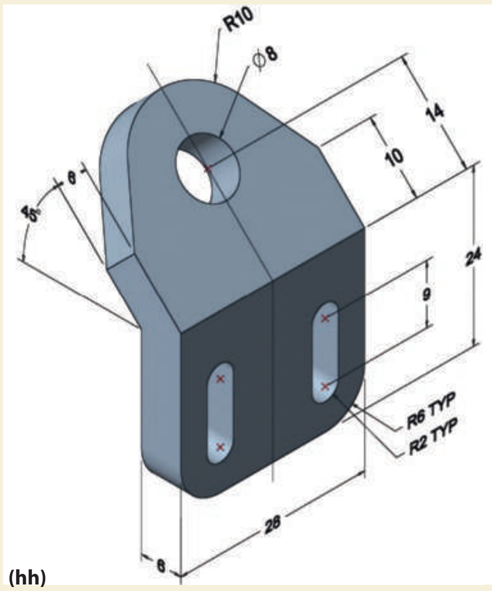
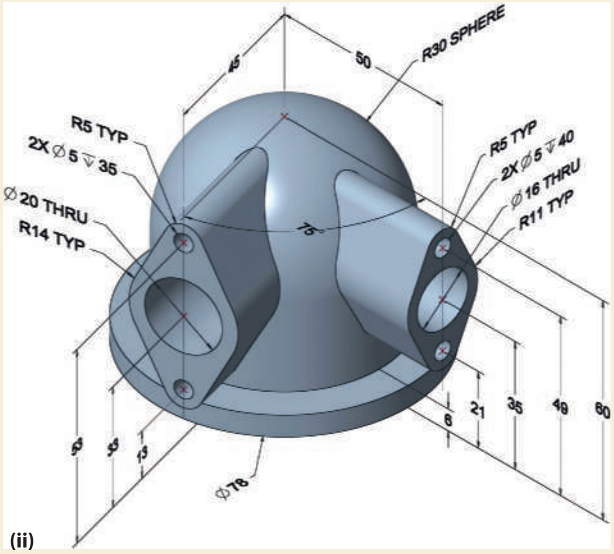


(gg)



(hh)

5.15 PROBLEMS (CONTINUED)



(ii)

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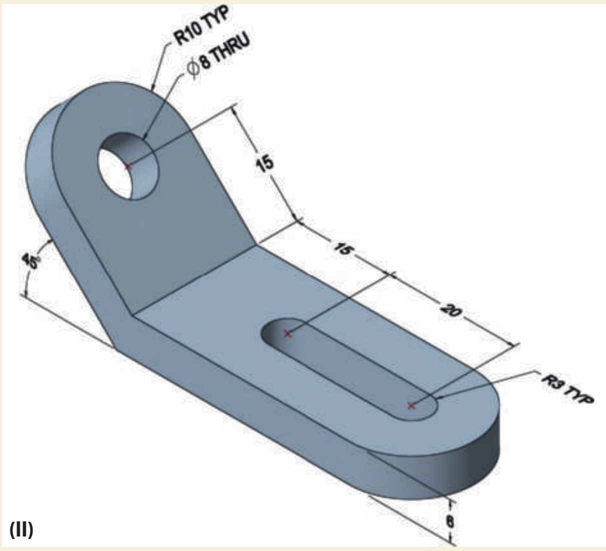

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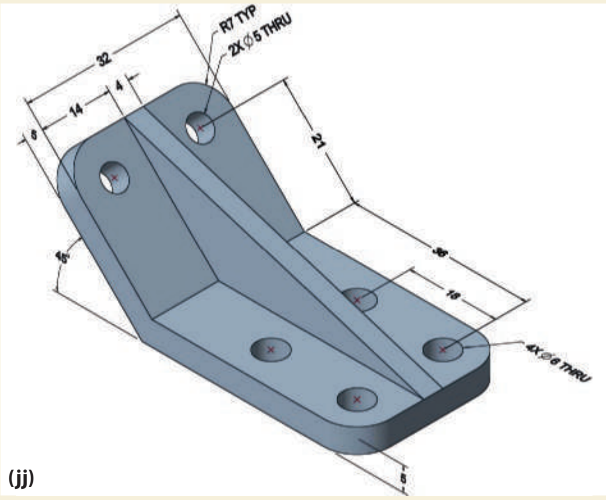

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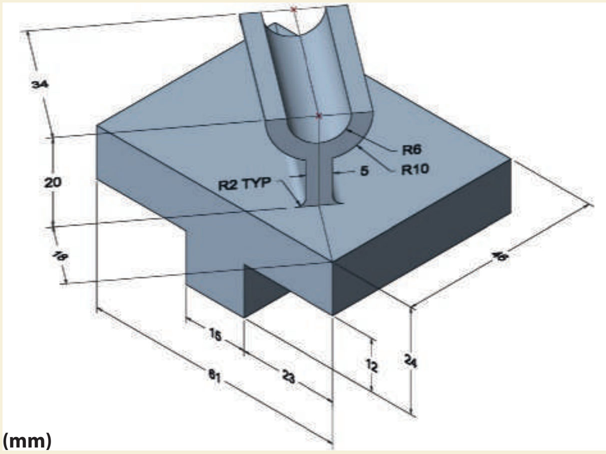

(mm)

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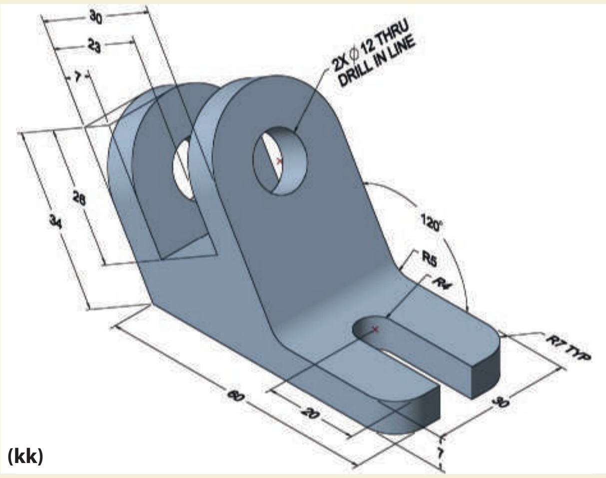

(kk)

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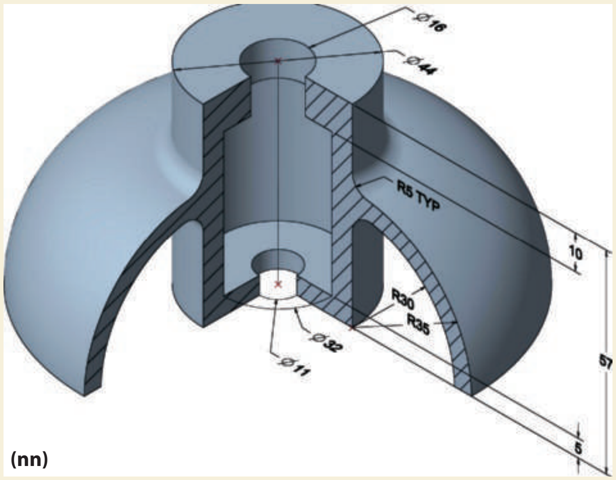

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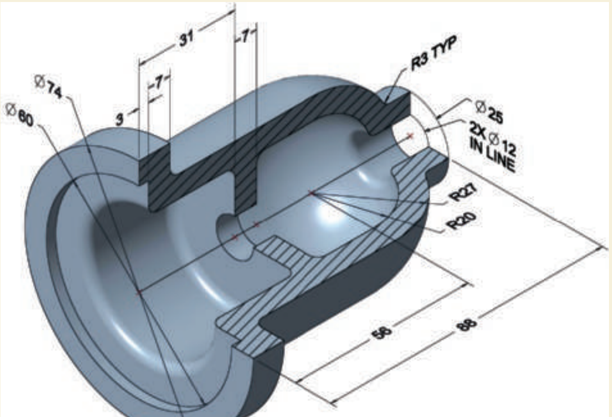

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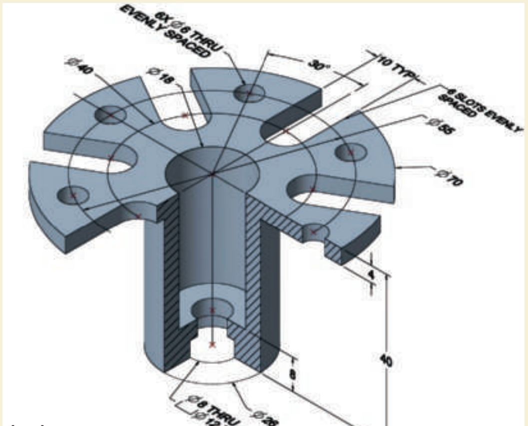

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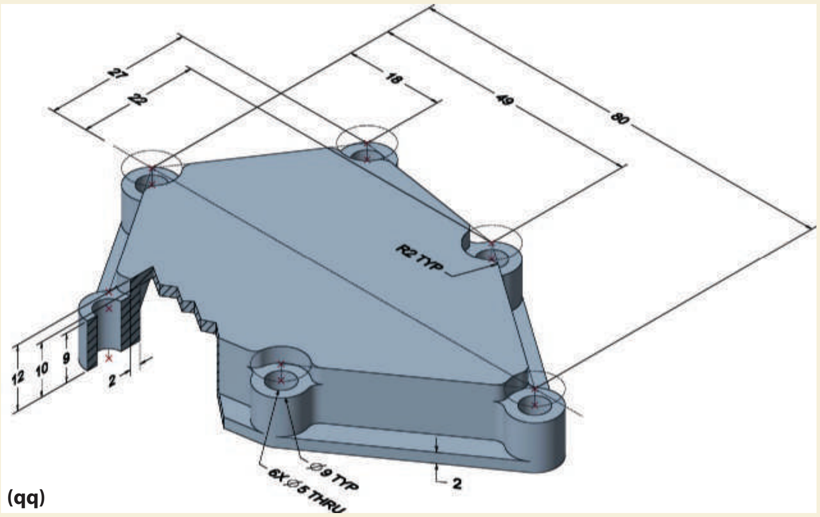

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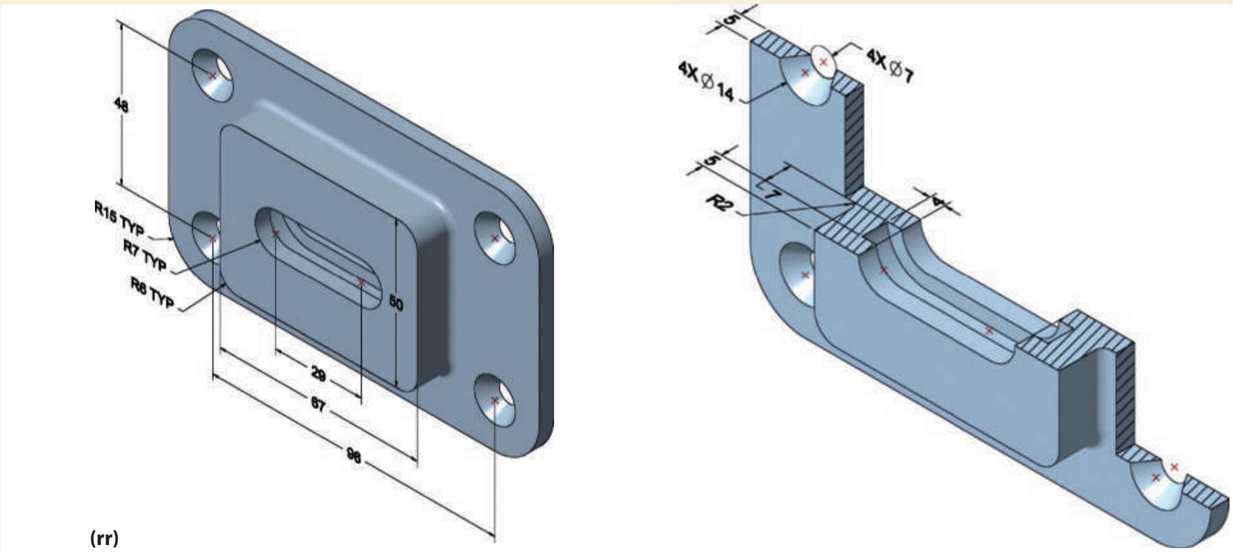

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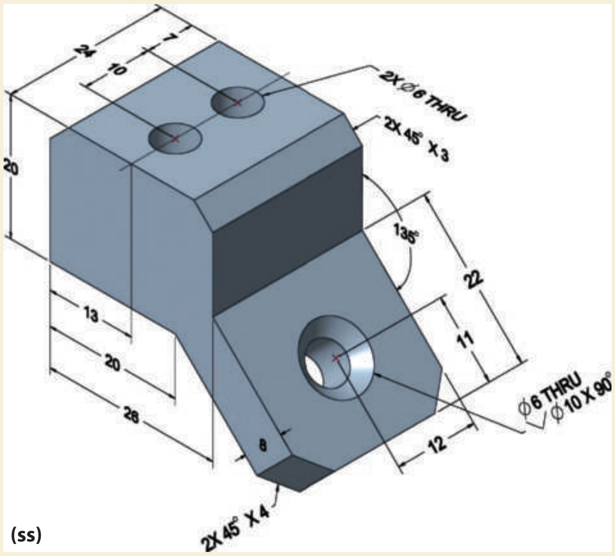

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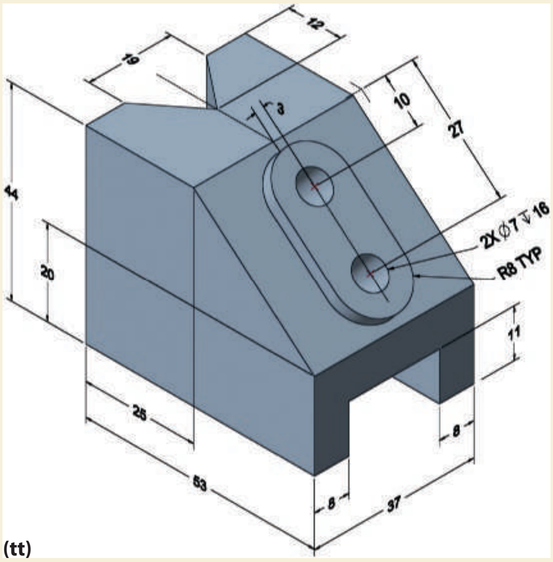

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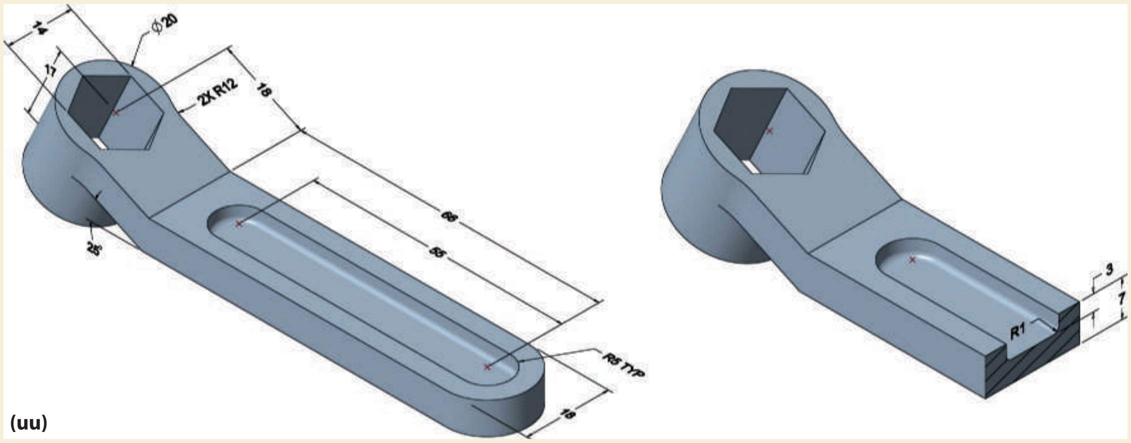

(uu)

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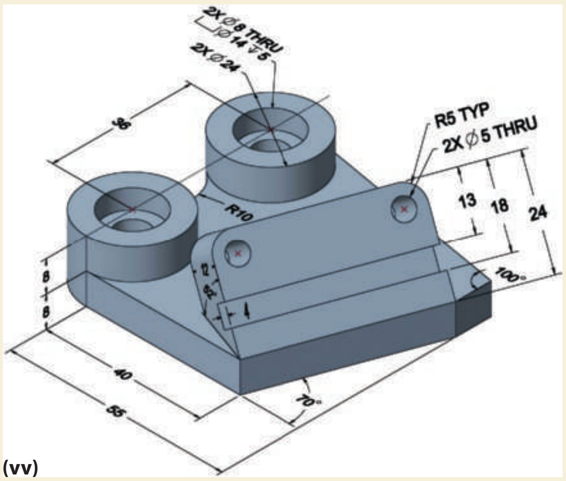

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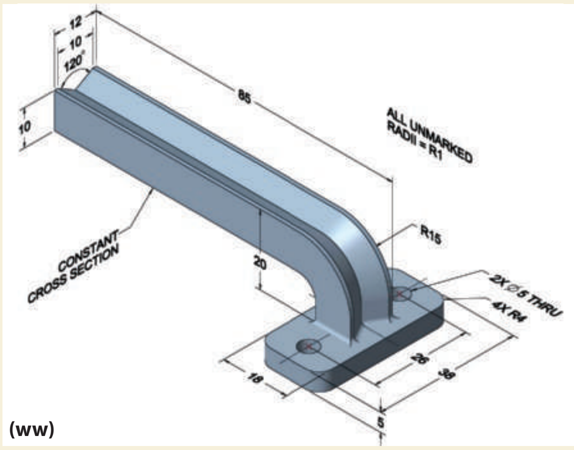

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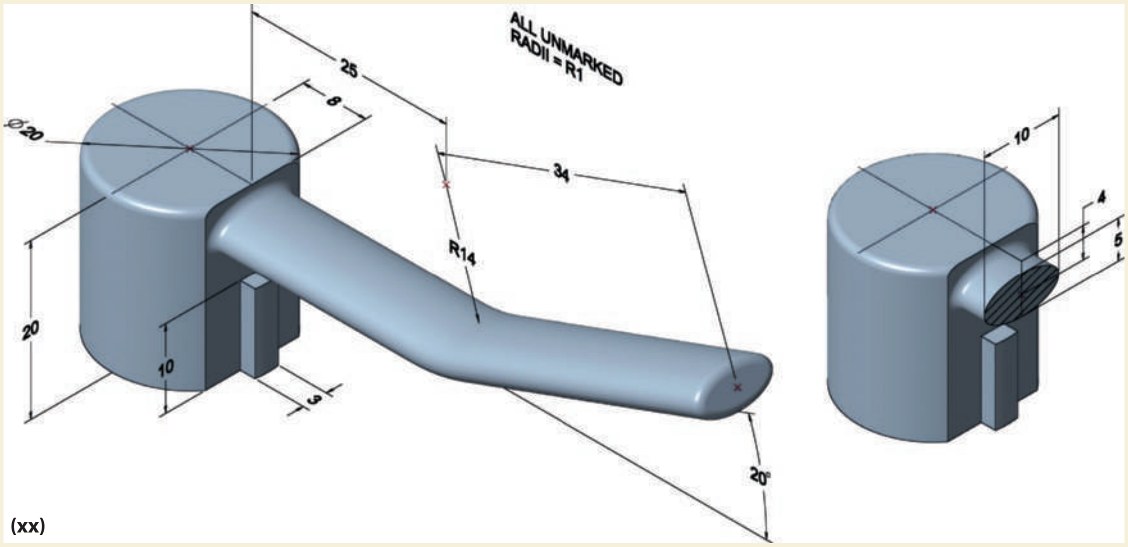

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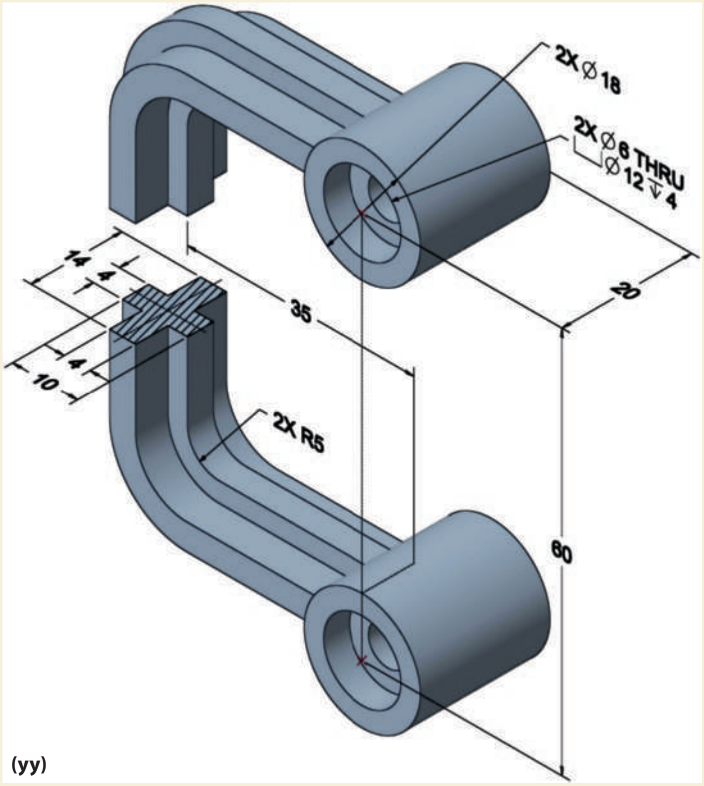

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FIGURE P5.4.

CHAPTER 6

ASSEMBLY MODELING

OBJECTIVES

After completing this chapter, you should be able to

- Apply new terminology in the context of assembly modeling
- Create an appropriate hierarchy for effective assembly modeling
- Apply assembly constraints between instances
- Create a bill of materials and an assembly drawing
- Determine interferences and clearances between instances in an assembly

6.01 INTRODUCTION

In a previous chapter, you learned about the techniques used to create solid models with computer tools. Most engineered systems, however, do not consist of a single part, but comprise multiple parts that work together and form the system or assembly. Think of a bicycle. It is used for personal transportation, and its purpose is to allow an individual to get from point A to point B through pedaling. It is composed of several subsystems, each of which serves a distinct purpose in allowing a person to operate the bicycle in a safe and consistent manner. Some of the subsystems found on a bicycle are the pedal system, the gear system, the tire system, and the frame. Each subsystem is composed of other subsystems or parts. For example, the gear system contains many single parts (e.g., the individual gears and the cables), as well as other subsystems (e.g., the derailleur).

This chapter describes how systems or assemblies are created with the use of CAD software and what type of information can be extracted from the assembly models. Because the foundation of assembly modeling is most often the initial creation of the components that make up the assembly, the assumption is that you are already familiar with the creation of solids.

6.02 Assembly Terminology

As with most categories in engineering design and analysis, one of the first things you must learn is the terminology particular to the topic. For assembly modeling, you need to learn a few new terms so you can work productively. Some of these terms will be discussed in the subsequent paragraphs.

In the creation of solid models, computer-generated geometry was referred to as parts, features, or objects; however, the objects that make up a system are referred to as **components**. A system component is identical to its referenced object geometry, and the change in terminology is to avoid confusion between two closely related modeling tasks. Think of a component as a 3-D part that has been brought into an assembly model. For example, consider the footbridge shown in Figure 6.01. Notice that the assembly model consists of abutments, approaches, a bridge deck, and handrails.

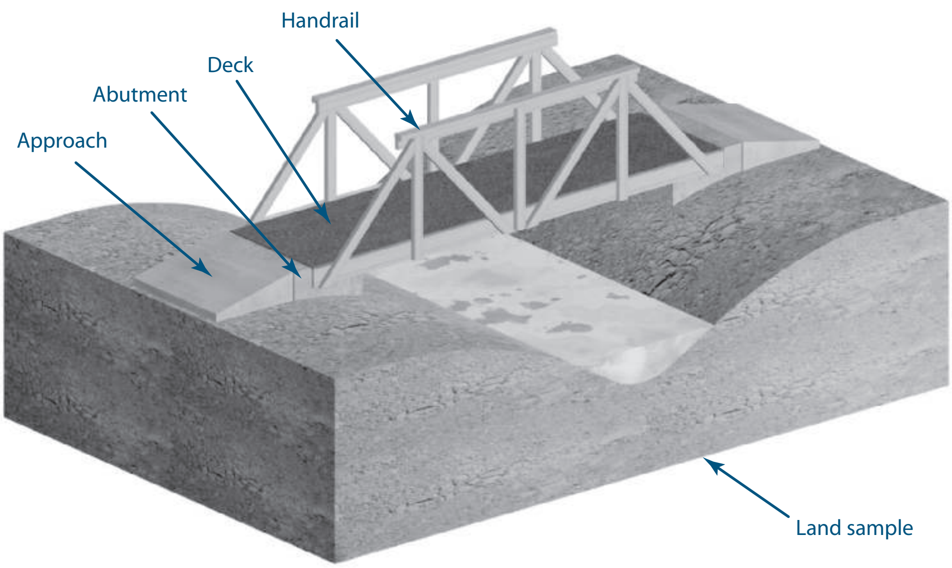

FIGURE 6.01. An assembled footbridge placed on a sample cutout section of land.

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You can create a solid model of the abutment within solid modeling software. In this case, the solid model of the abutment is the desired final outcome of the modeling process. When you are working on assembly modeling where the desired final outcome is a bridge assembly, the parts brought into the model that you are working on (the assembly) are called components. When the solid model of the abutment is brought into the assembly model, it is referred to as a component and is no longer thought of as a "part."

In the case of the bridge assembly model, two abutments are in the desired final result. It would be cumbersome to bring a new abutment component into the assembly each time it is required. Further, the abutments are identical to each other (they are just oriented differently in space), so you want it to be clear that the abutments in the final assembly are copies of the same component. In assembly modeling, the copies of a component within the system are called **instances**. Thus, for the bridge, an abutment can be an overall component associated with the system with two instances of the particular component within the assembly. Figure 6.02 illustrates the concept of instances of components in an assembly model.

In reality, system components do not actually appear on the computer screen—they are stored in memory within the assembly model, waiting for you to instance them into the system, but are not displayed in the work area. It is similar to having certain vocabulary words stored in your brain. You can put the words on a sheet of paper to form sentences and paragraphs, but the words do not physically exist anywhere in space—they exist only in your memory. When you want to put a copy of a component in an assembly, you *instance* it into your workspace.

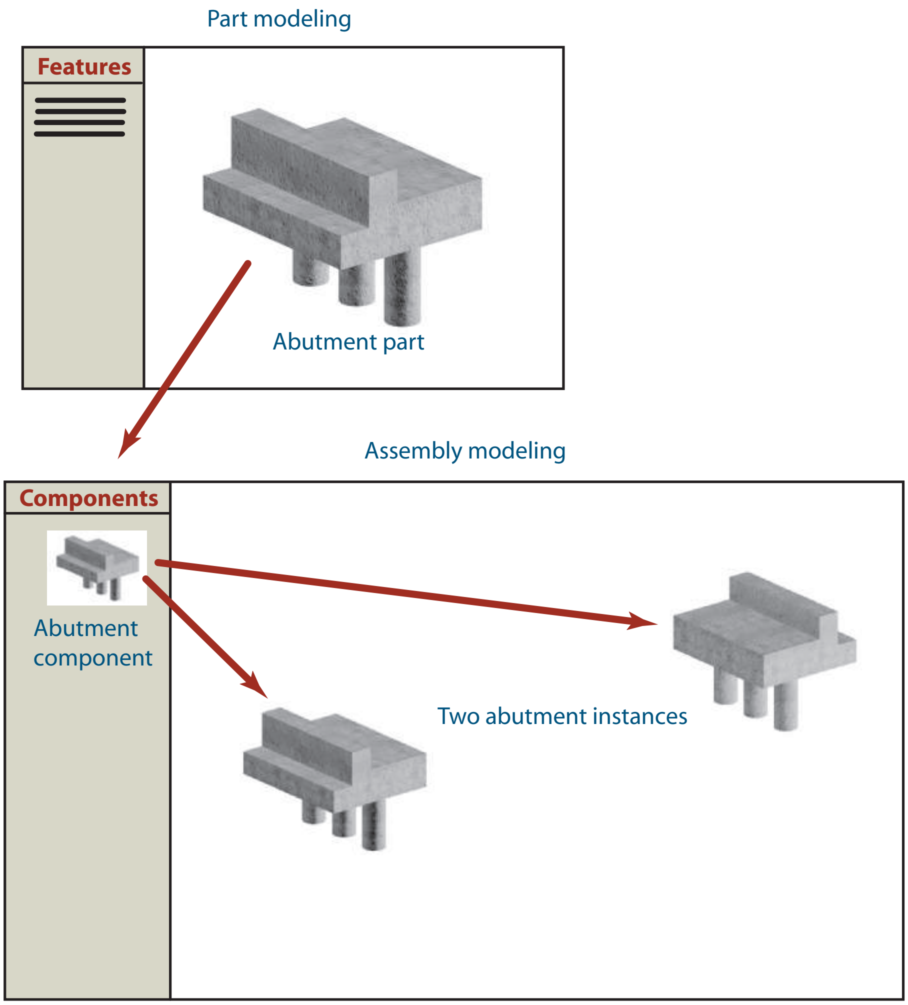

FIGURE 6.02. The concept of parts, components, and instances in assembly modeling.

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Another new term for you to consider in assembly modeling is the concept of a **subassembly**. A subassembly is a grouping of components that serves a single purpose within the overall assembly. Going back to the bridge example, a subassembly could be created to represent the handrail. Notice that the handrail consists of vertical members, angled members, and the horizontal crossbar at the top, as shown in Figure 6.03.

Instead of instancing each of the individual members that make up the handrail into the overall assembly, you could create one subassembly of the handrail that includes appropriate instances of all of its components—including the vertical, angled, and horizontal members—and just instance that subassembly into the overall system two times. The completed handrail subassembly is shown in Figure 6.04.

Since it is unlikely that the horizontal member of the handrail will act independently of the vertical or angled members, it makes sense to put them into a subassembly together so you can work with them as a single unit. You might think it would be easier to make the handrail a single part. But because the individual components may be made of different materials, you want to keep them as separate objects for any analysis you perform later on. Linking them in a subassembly allows you the flexibility to alter properties of components independent of one another, but also affords you the efficiency of treating them as one unit in the assembly modeling process.

Subassemblies are unique in that they are composed of instances, but subassemblies also can be instances. When you are creating the overall system, you can insert instances that are individual components or that are subassemblies made up of several individual components. Subassemblies also can contain other subassembly instances. For the handrail subassembly, you could choose to combine two angled members and one vertical member as one subassembly, as shown in Figure 6.05. The subassembly

FIGURE 6.03. The handrail subassembly (with the bridge deck shown in phantom for reference only).

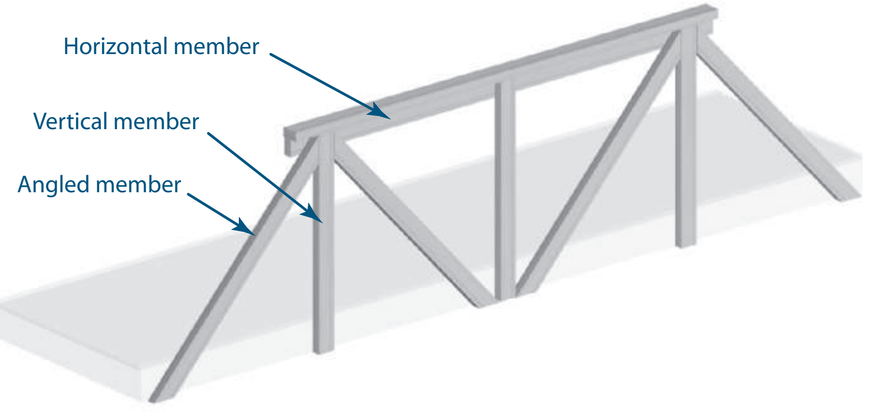

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FIGURE 6.04. The second handrail subassembly as an instance of the first handrail subassembly.

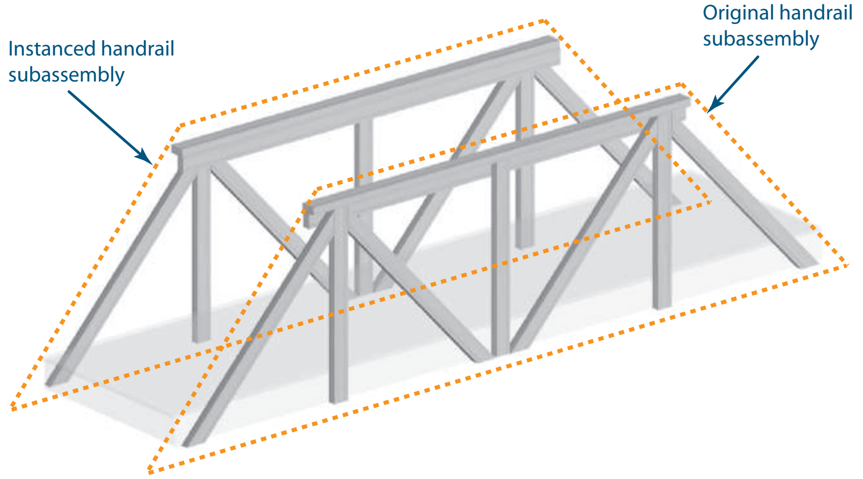

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FIGURE 6.05. The web subassembly composed of two angled components and one vertical component.

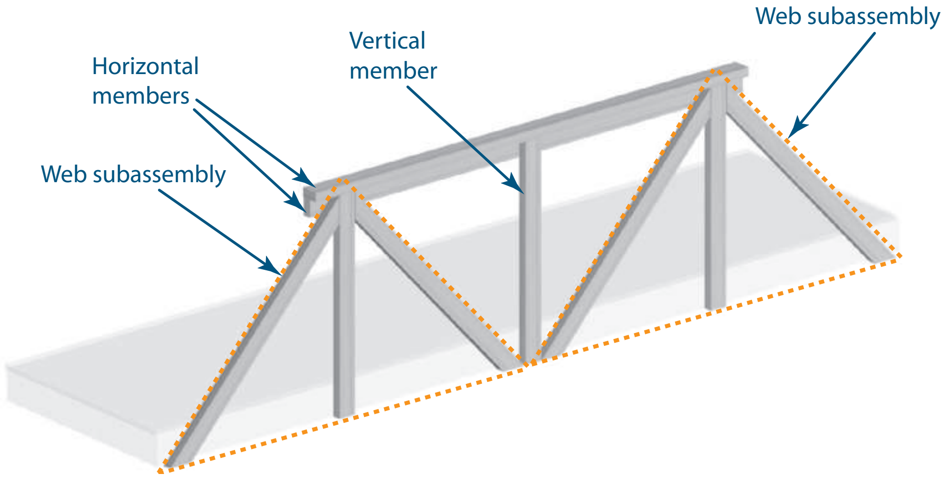

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FIGURE 6.06. The complete handrail subassembly composed of one vertical member, two horizontal members, and two web subassemblies (with the bridge deck shown in phantom for reference only).

could be instanced into the overall handrail subassembly two times, along with one additional vertical member and the horizontal bar, to make up the overall handrail subassembly shown in Figure 6.06.

It may seem that the change in terminology between solid modeling and assembly modeling is picky and overly complicated. Why not just put multiple copies of the abutment part within the assembly? Why do you have to learn about components and instances? Why are subassemblies that are made up of instances and that are instances themselves important? The main reasons for the changes are to improve assembly modeling efficiency and to save computer storage space. Saving computer storage space may seem like a trivial matter; however, the less memory a model takes up, the faster your computer modeling software will work. Files that are too large (in terms of memory) are difficult and slow to work with. The improvement in assembly modeling efficiency through the use of subassemblies should be apparent. But how do you save on computer storage space or memory?

To answer that question, consider just the horizontal member of the handrail subassembly. Depending on the design of the member, the computer must "remember" many points to define the model. Each time there is a vertex where two or three edges intersect, the software must remember where the point is located in 3-D space. If the edges of the member are rounded, not sharp, corners, several points on the rounded edge must be remembered for the part geometry to be defined completely. Because some designs for a horizontal handrail are fairly complex, the number of geometric data points used to define it can be large; and the number of 3-D locations in space for the part the computer model must remember also is large. If you were to put the horizontal member directly into the assembly model, each time you moved the member in space within the assembly model, the computer would have to remember the new location of each multiple point that defines the horizontal member. In the case of a bridge with two handrails, the number of points to be remembered is relatively large.

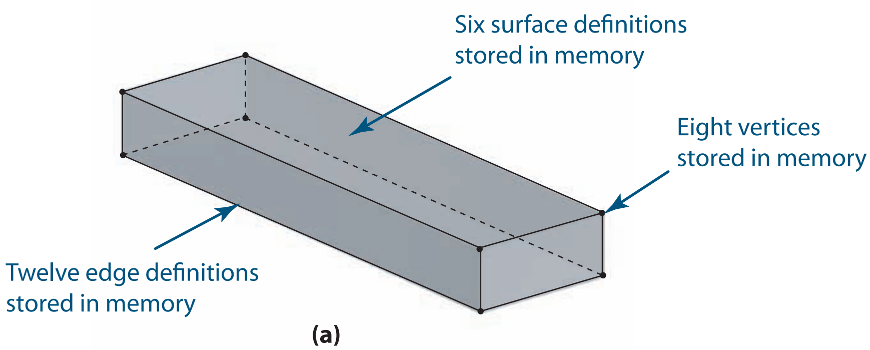
By organizing with components and instances, however, the data to be stored are significantly reduced. The component is referenced to an object, and all the component has to remember is that the component looks exactly like the object. When an instance of the component is placed in the assembly (or subassembly), all that has to be remembered is location of the instance relative to that of the component. In other words, when the instance is located 6 units along the x-axis and 0 units along both the y- and z-axes and it is rotated 90 degrees about Z relative to the component definition, the only thing to be remembered is the location of the instance compared to the location of the component. Thus, for each instance, only six pieces of data must be remembered by the computer—translations in X, Y, and Z and rotations about X, Y, and Z. The amount of data stored under this scheme is significantly less than what would be stored if copies of an object were placed directly into the system. Figure 6.07 shows the object for the horizontal handrail member and an instance of it in a subassembly. Also shown in the figure are the data points defining the location of the instance within the subassembly.

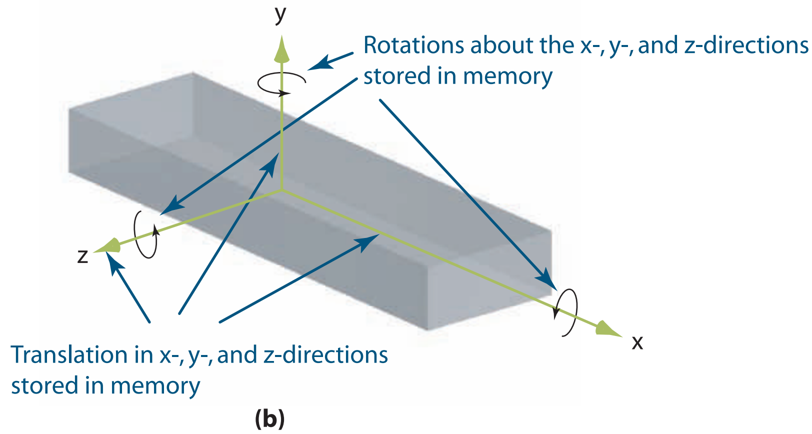
6.02.01 Associativity

In the solid modeling chapter, you learned about associativity between parts and drawings. When an object changed, any associated drawings also changed in the same way. Associativity also can exist between parts, components, and assemblies. In assembly modeling, **associativity** means that if you change the geometry of a part, the component and all instances of it also will change by the same amount. With some software, when you change a component within assembly modeling, the object geometry also changes by the same amount. That also means that any drawings, finite element models, or manufacturing models associated with the part also will be updated.

Associativity greatly increases your computer modeling efficiency for complex parts and systems. Thinking back on the footbridge example, imagine you are working for a company that designs and produces footbridges for parks and recreation areas. The company might have several bridge models that use the same bridge deck. If the bridge deck design division of the company changes its design, the changes can be reflected throughout all bridge models simultaneously as long as the associative relationships between parts and assemblies have been maintained throughout the company's product line. If the associations have been broken, the engineers working with the various models must be aware of the changes in the bridge deck design and then make the required changes model by model. If the engineers forget one of the models, the overall design will be incorrect, which could have a significant impact when the production phase of the project is initiated.

FIGURE 6.07. A schematic representation of a horizontal member part (a) and an instance (b).





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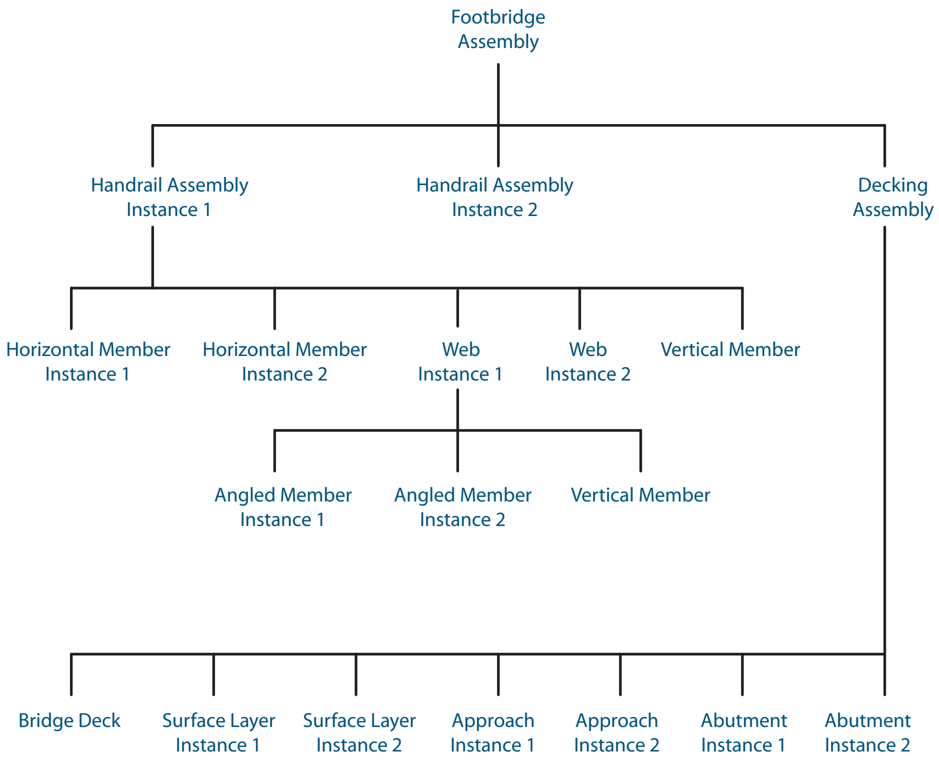
6.03 Assembly Hierarchy

Assembly models are easier to work with when they are organized in a logical manner. An assembly is usually thought of as a composition of several smaller subassemblies, each of which may consist of other subassemblies or individual components. The organization or structure of a system is referred to as its **hierarchy**. The hierarchy is similar to an inverted family tree. Another way to think of an assembly hierarchy is as a corporate structure. The president of the company is at the top of the hierarchy and has several vice presidents at the next level reporting to him. Each vice president, in turn, is responsible for several managers; the structure of the organization continues until you reach the lowest level in the company hierarchy, where the laborers are located. In this analogy, the individual laborers can be thought of as subassembly instances, the managers would be the first level of subassemblies, the vice presidents would be the next level of subassemblies, etc., until the entire assembly is defined. The top-level assembly could be considered the company president.

As with objects and features, the associations between components and subassemblies are often called parent-child relationships. Going back to the handrail subassembly, the handrail would be a parent whose children are the horizontal member, the subassembly of two angled members and a vertical one, etc. The overall bridge would be the parent whose children consist of two handrails, two approaches, two abutments, the bridge deck, etc. As with objects and features, to work effectively, you must understand the various parent-child relationships defined in assembly modeling. A schematic of a possible hierarchy for the footbridge is shown in Figure 6.08.

Organizing an assembly in a logical hierarchy enables you to work more efficiently with the assembly. The technique is similar to the way you organize files on a computer, establishing folders to group files together in your work space. As stated previously, one advantage of creating an assembly hierarchy is that subassemblies can be dealt with as a whole rather than as separate components. For example, one subassembly can be moved as a single unit within the system, rather than individual components of the system being moved separately, just like the way you move folders and all of the files in them on your computer workstation.

FIGURE 6.08. The footbridge assembly hierarchy.



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6.04 Assembly Constraints

The first thing you want to do after setting up your system hierarchy is to orient the instances so they are properly located in space relative to one another. To do that, you need to select one component to serve as the **base instance**. The base instance remains stationary with the other instances moving into place around it. For example, if your assembly model was a car, you might choose the chassis as the base instance. Usually, the choice of a base instance is fairly obvious; however, in some cases, you may have to choose one from among several choices.

After selecting the base component, you should establish reference planes for the assembly that are connected to the base component. These reference planes will serve as the coordinate planes for the space defined by the system. For the footbridge assembly, the bridge deck was selected as the base component, with the reference planes established as shown in Figure 6.09. You will use these reference planes as you orient the various components within the system.

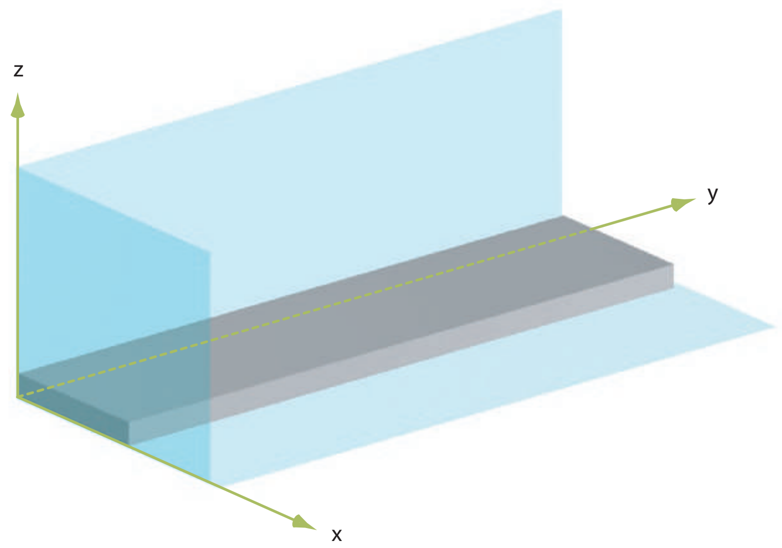
When creating parts, you used constraints to establish geometric and dimensional relationships between 2-D entities or features. Thus, you constrained two lines to be parallel or perpendicular to each other. Or you constrained the diameter of a circle to be a specific size and its center to be located given distances from lines on the drawing or from edges on an object. In assembly modeling, you can apply **assembly constraints** between two 3-D instances so the instances maintain dimensional or geometric relationships with respect to each other within the assembly.

Each time you bring an instance of a component into an assembly, you introduce six degrees of freedom (DOF). The instance will have a set of coordinate axes associated with it, and the six new DOFs will correspond to the three translational DOFs (distance along X, Y, and Z from the instance origin to the base component origin) and to the three rotational DOFs (rotations about X, Y, and Z) relative to the coordinate planes associated with the base component. Each time you apply a constraint to the assembly, you remove one or more DOFs. As with part modeling, you can continue to apply constraints until all of the DOFs are removed from the assembly model.

6.04.01 Concentric Constraints

One of the more useful applications of assembly constraints is to define two different instances to be concentric with each other. This is especially useful in dealing with cylindrical shafts that fit within a cylindrical hole in another part. You can constrain the centerline of one instance to coincide with the centerline of a different instance, forcing

FIGURE 6.09. The bridge deck as the base instance, with reference planes and axes.



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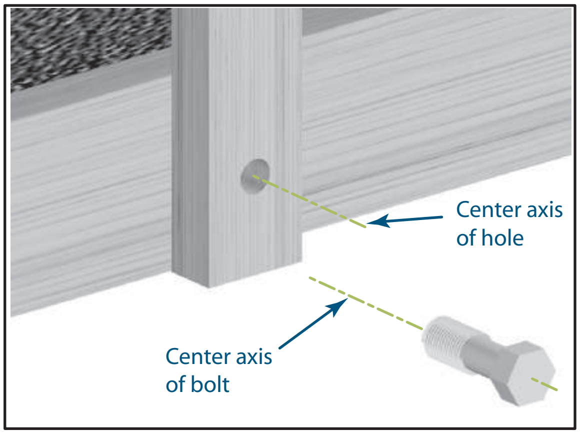
the two instances to be concentric. After constraining the instances to be concentric, you only need to determine how far along each centerline the instances are located relative to each other. For the bridge assembly, nuts and bolts are used to fasten the instances of the system together. (The nuts and bolts were not shown in previous figures of the bridge assembly to simplify the discussion.) Figure 6.10 shows a concentric constraint applied between two instances in the bridge assembly. Note that when the concentric constraint is applied to the system, several DOFs are removed, since the *x*-, *y*-, and *z*-rotations are now fixed and one of the translation DOFs also has been removed.

In some modeling packages, an "insert" constraint is a special type of concentric constraint used to insert a fastener such as a bolt into a hole in a different part. When the bolt is inserted into the hole, the system automatically applies a concentric constraint between the two components.

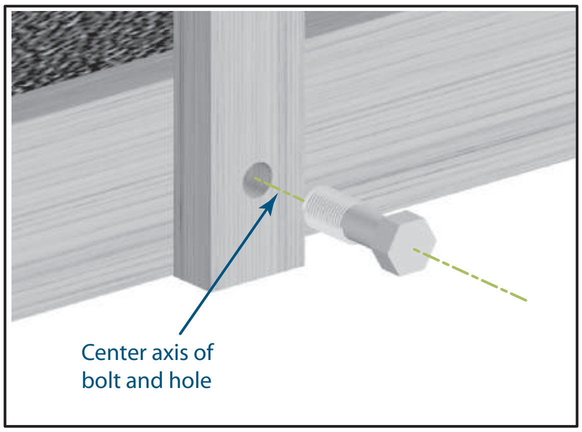
6.04.02 Mating Surfaces Constraints

Another useful constraint available in most assembly modeling software is used to define two surfaces as mating surfaces. Mating surfaces coincide with each other—in other words, they line up on top of each other. Usually, that type of constraint works only with flat or planar surfaces. When creating a mating surface constraint, you are essentially working with the surface normals. A surface normal is defined as "a vector that is perpendicular to the surface and points away from it." So when you are applying a mating surface constraint, you are forcing the normals of the two surfaces to be parallel

FIGURE 6.10. Concentric constraint used to locate a bolt in its hole. Before application of constraint (a) and after constraint (b).



(a)



(b)

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to each other. Depending on the final desired result, you may have to flip one of the instances around in order to apply the mating surface constraint to achieve the correct orientation between instances. Figure 6.11a shows a mating surface constraint applied between the inner face of the hex head at the end of the bolt and the outer surface of the handrail member for the bridge. Figure 6.11b shows the result of incorrect mating surface orientation. Figure 6.11c shows the correct surface orientation, with bolt flipped in the final result. Also notice that when the constraint is applied, several DOFs for the system are removed, including both rotational and translational DOFs.

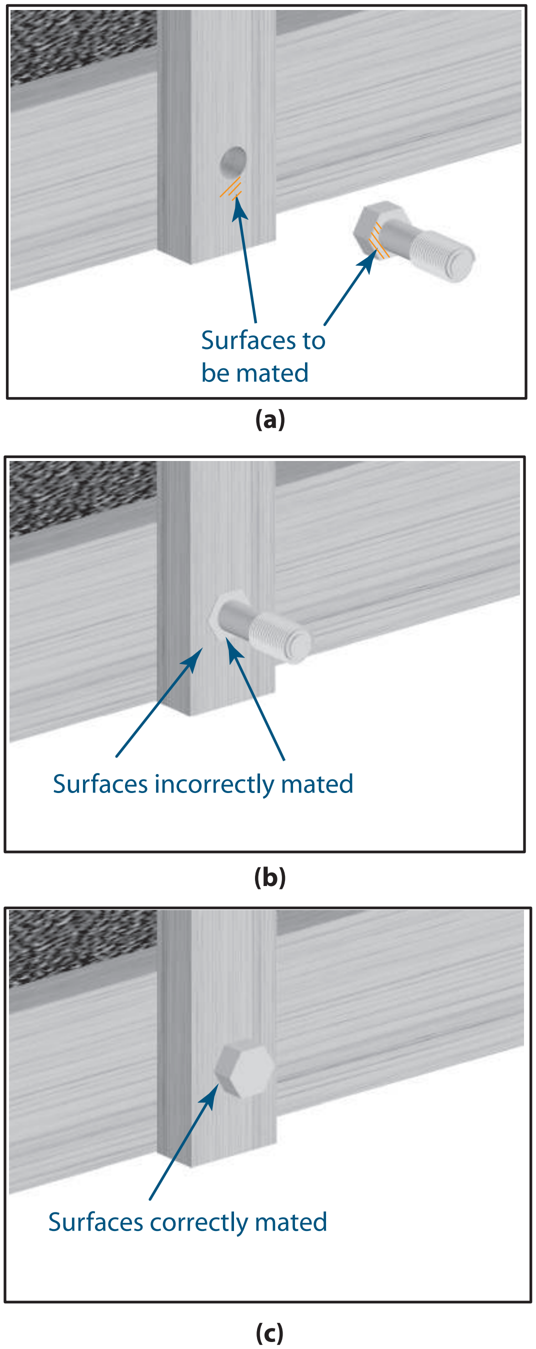

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FIGURE 6.11. Applying mating surfaces constraint after concentric constraint to the bolt and hole. The mating surfaces are identified in (a). The incorrect mating orientation is shown in (b); the correct orientation in (c).

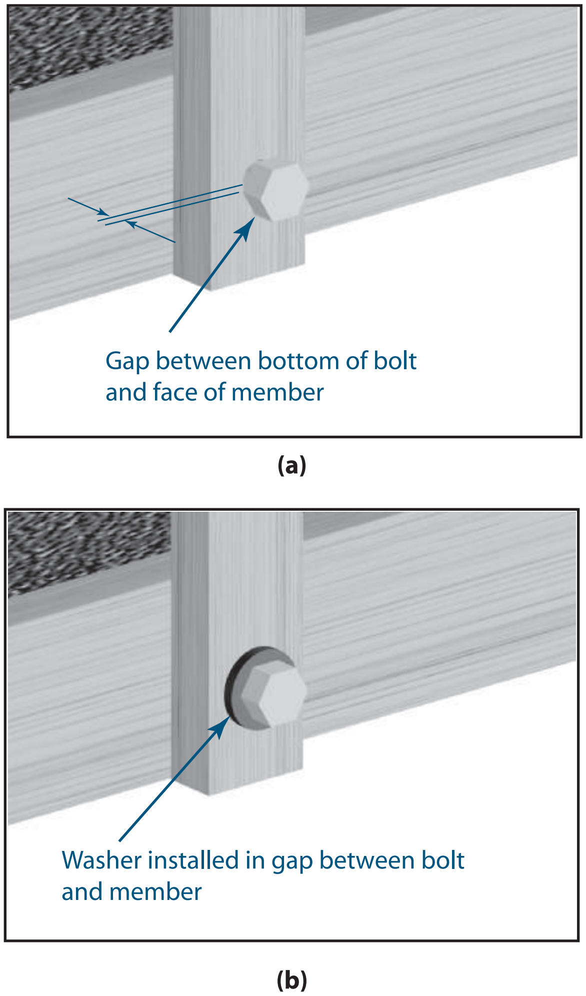

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FIGURE 6.12. Applying an offset constraint between the bolt and member to permit a washer instance.

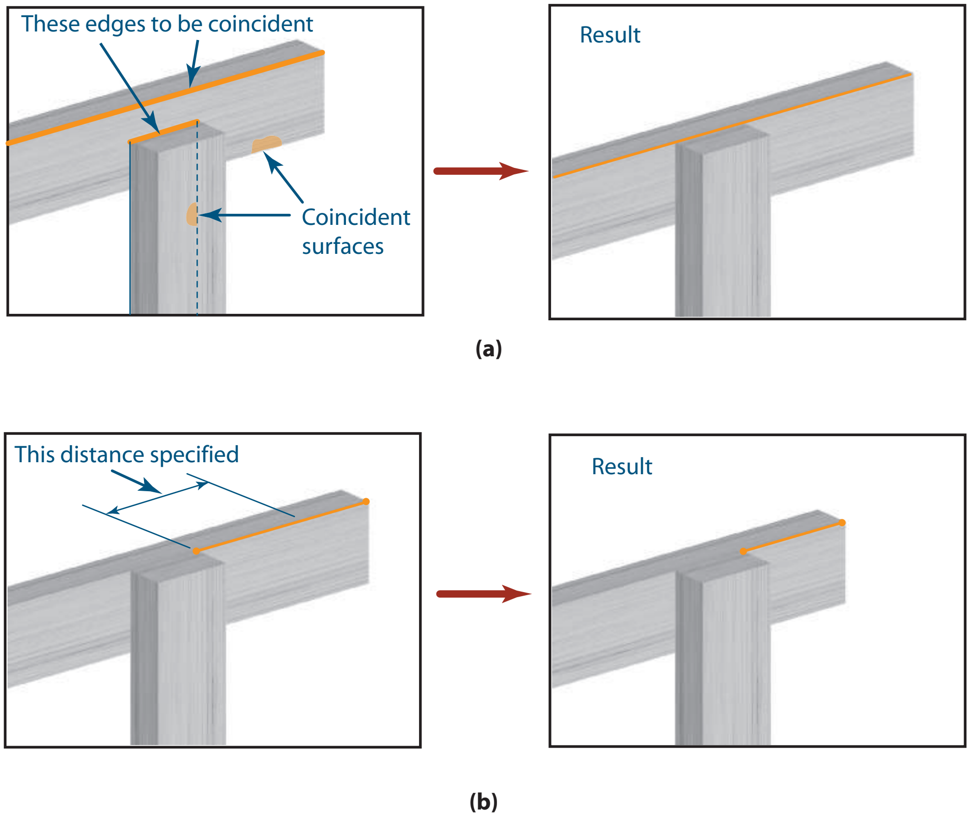
Another type of constraint that is related to the mating surfaces constraint is one whereby you include an "offset" from one surface to the other. Basically, you are saying that the surface normals for the two surfaces are still parallel to each other, but now the surfaces do not line up on top of each other. Figure 6.12a shows the bolt and handrail member after an offset constraint has been included between the instances to make room for the washer that is included in Figure 6.12b.

6.04.03 Coincident Constraints

One definition of *coincidence* is that two entities take up the same space. For that type of constraint in assembly modeling, coincidence can occur between two lines, between two points, or between a point and a line. (Note that coincidence of two planes has already been defined with the mating surfaces constraint.) In assembly modeling, defining coincident constraints often means that you select a corner or an edge on one instance to be coincident with a corner or an edge on another instance. When applying this type of constraint, you typically are required to input offset distances between points in order to achieve your final desired result. Figure 6.13a shows the effect of making one edge on a vertical handrail member coincident with an edge on the horizontal member, and Figure 6.13b shows the final result achieved after including a dimensional constraint between corresponding endpoints of the edges. Figures 6.14a and 6.14b show the effect of making the edges of two members coincident and then making the two corresponding corner points coincident to achieve the desired final result.

When applying coincident constraints, you may again have to flip one instance around to achieve the desired final result. Figure 6.15a shows two instances with a coincident constraint applied, and Figure 6.15b shows the same two instances after one has been flipped to put it into its desired orientation.

FIGURE 6.13. A coincident edge constraint is used in (a) to align the edges of the horizontal and vertical members. A dimensional constraint is used in (b) to perform the final location of the horizontal member with respect to the vertical member.



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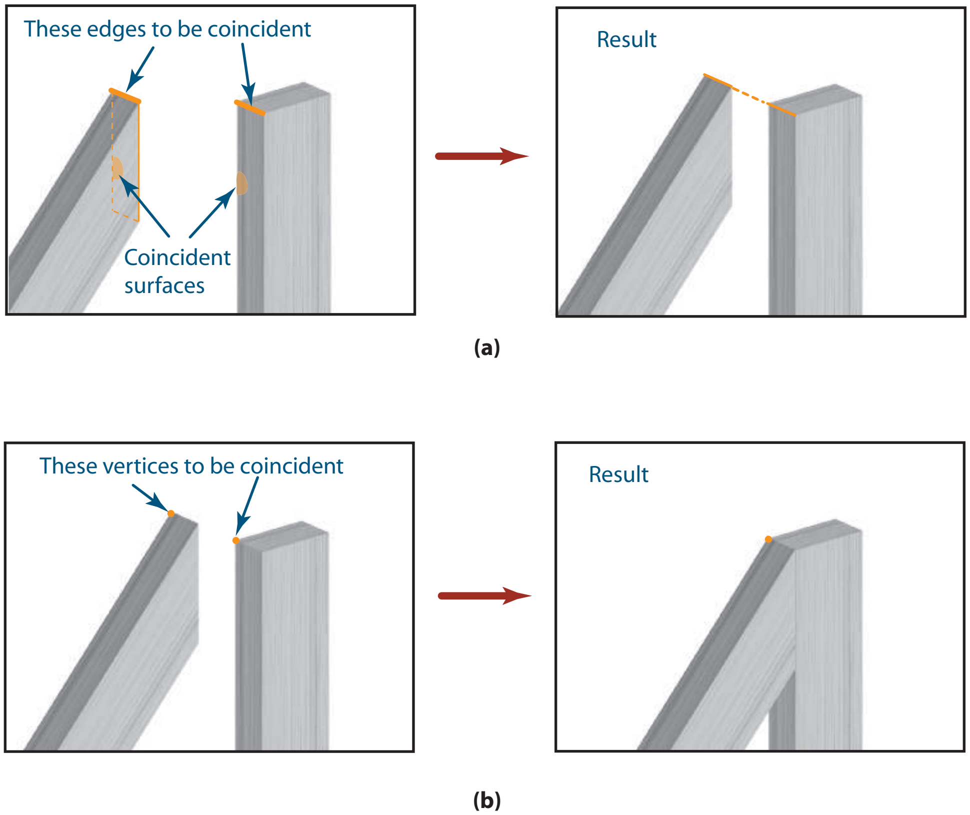

FIGURE 6.14. A coincident edge constraint is used in (a) to align the edges of the vertical and angled members. A coincident vertices constraint is used on the vertices shown in (b) to perform the final location of the angled member with respect to the vertical member.

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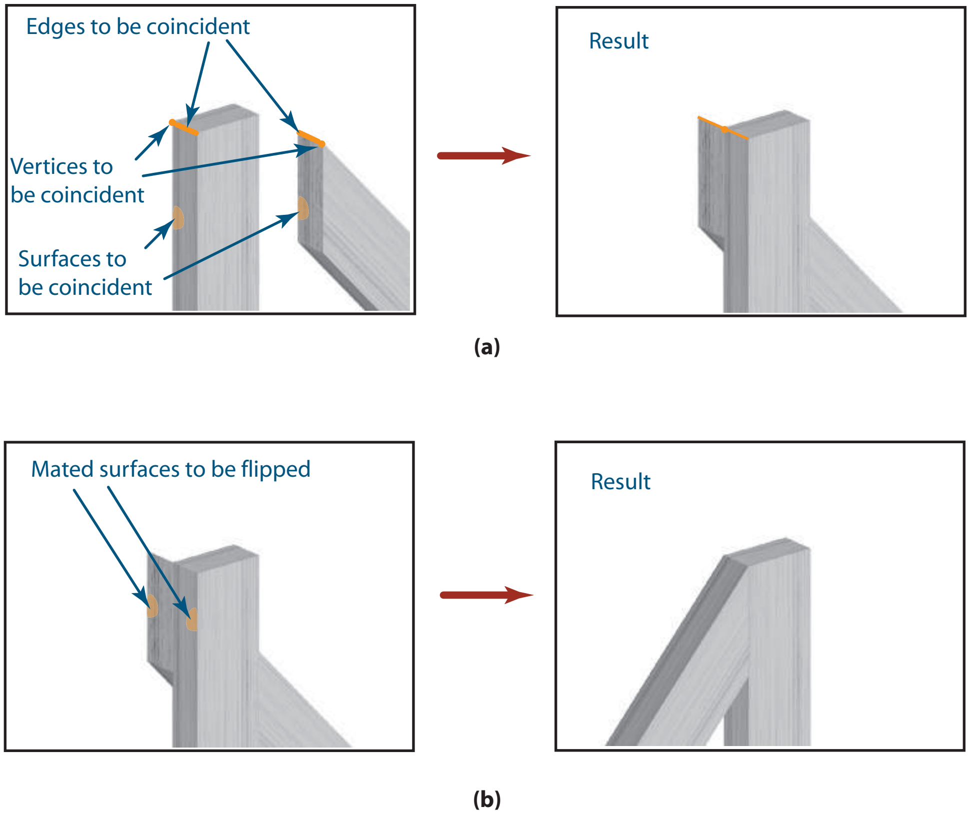

FIGURE 6.15. The effect of flipping a coincident surface constraint. Coincident surfaces, edges, and vertices are used in (a) to align the vertical and angled members. The coincident surface constraint is flipped in (b) to realign the members.

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6.04.04 Distance Constraints

When creating a 3-D solid model, you often add dimensions between features to define sizes of parts. With assembly models, you add distance constraints to define the relationship between two instances. When adding the distance constraints, you use points, edges, and surfaces to define the distances between the instances. Often, a distance constraint is used in conjunction with a second or third assembly constraint. For example, for the two instances shown in Figure 6.16, a coincident constraint was applied between the point on the angled member and the edge on the horizontal crossbar. A distance constraint was then applied between the endpoint of the two instances, as shown in the figure.

A distance constraint also can be applied between parallel surfaces on two separate instances, as shown in Figure 6.17.

6.04.05 Adding Constraints to Your Assembly

In addition to putting your parts together to see how they look, certain types of analysis that are best performed on an assembled system will aid you in your engineering design tasks. One predominant type of analysis that you can perform with an assembled model is to model it as a mechanism. With this type of analysis, you add joints such that two parts are able to rotate freely about an axis or you add a joint such that two parts slide relative to each other. Consider a piston in a cylinder. The piston can slide in and out; and as is illustrated schematically in Figure 6.18, the cylinder and the connecting rod are also free to pivot about the piston pin and the connecting pin.

Mechanism analysis is important for a piston assembly. The analysis allows you to determine whether the connecting rod bumps up against the walls of the cylinder as the mechanism goes through its cycle of motion or whether the piston bumps up against the fixed end of the cylinder. For you to accomplish this type of analysis, however, your assembly must be properly constrained.

When you learned about including constraints with sketches or models as you were studying part creation, you learned that sketches could be unconstrained, fully constrained, or partially constrained. Similarly, assemblies can be unconstrained, fully constrained, or partially constrained. You should avoid unconstrained systems since all instances in the assembly are free to move relative to one another. If you have an unconstrained assembly, you need to move each instance and subassembly one at

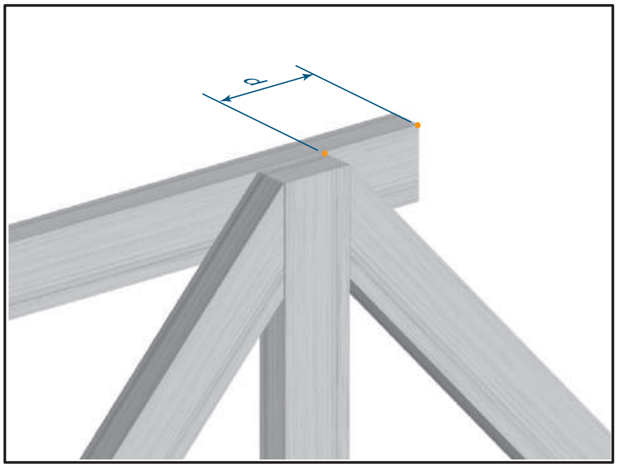

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FIGURE 6.16. A distance constraint between vertices to locate the horizontal member with respect to the angled member.

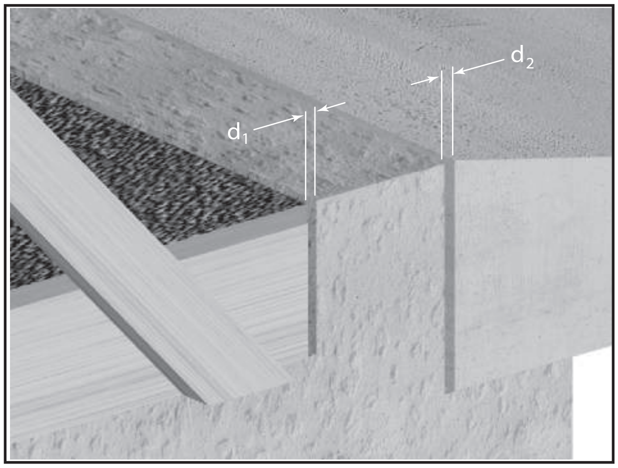

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FIGURE 6.17. Parallel distance constraints used to create clearance gaps between the deck and the abutment and between the abutment and the approach.

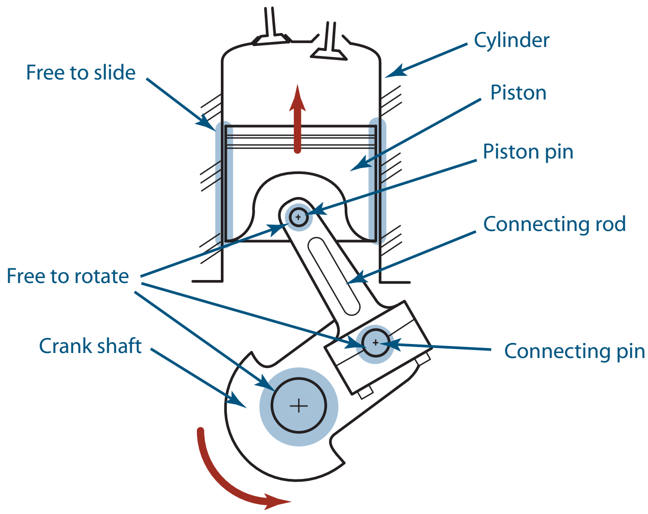

FIGURE 6.18. Schematic drawing of piston/cylinder assembly.

a time—a tedious task—to perform your analysis. An assembly that is fully constrained is rigid and unable to move. Each time you try to move one instance, the entire system reverts to its original orientation to satisfy all of the constraints on it. Like Goldilocks, you want a system that is "just right"—with just enough constraints to define the permissible motion in the assembly but not too many constraints that its ability to function properly is hindered.

Going back to the piston/cylinder assembly, you want the constraint between the piston and the pin to be rigidly defined—these two instances should not be able to move relative to each other (ever!). However, the cylinder and piston should be partially constrained such that they remain concentric to each other but the piston is still free to move up and down along the centerline of the cylinder. Likewise, the connecting rod should be partially constrained such that the pin remains concentric to the pin hole but is free to rotate about its axis as the system moves. To achieve the balance between systems that are over- or underconstrained, you should think about the types of motion your assembly will go through in computer analysis as well as in real life. If you think about the bridge assembly presented earlier in this chapter, the instances in this system should never move relative to one another—a bridge is a rigid structure that is built to remain rigid its entire life cycle. In the case of the footbridge, having a fully constrained assembly is permissible.

Some software requires assembly models to be fully constrained. In this case, you are usually permitted to include "assumptions" with your constraints in order to satisfy the need for system flexibility and movement. For example, in the case of the connecting rod and piston pin, you could include the concentric constraint and then add an "assumption" that the two instances are able to rotate relative to each other. In this case, the assumption will eliminate DOFs in the system and the software will be satisfied that the assembly is "fully" constrained.

6.05 Exploded Configurations

For most mechanical systems, an assembly drawing is necessary in order to put the system together. You probably saw that kind of drawing if you ever put together a furniture kit or a model. Once you assemble your system, you can create an **exploded configuration** that will essentially be an assembly drawing for the system. Figure 6.19 shows an exploded configuration for a small flashlight assembly. Note that the configuration shows how all of the parts in the system will be put together, or assembled.

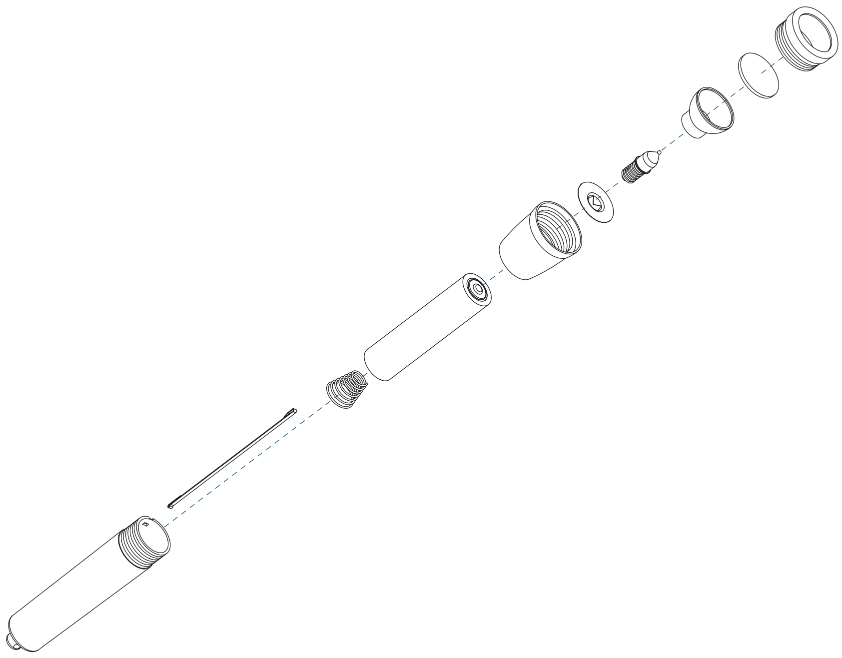

FIGURE 6.19. An exploded view of a small flashlight assembly.

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6.06 Interferences and Clearances

One of the main advantages to be gained from assembling a system of parts is that you are able to determine whether two parts overlap or interfere with each other. Since different engineering teams often work on separate parts of an assembly and since communication between groups may not always be clear, being able to determine overlap is especially important when working collaboratively on a large design. The amount that two instances overlap is referred to as the **interference** between them. Another advantage is that you can determine **clearances**, or minimum distances, between parts. You can use that information to optimize the assembly. You can change the size of features as appropriate to remove interference or to increase clearances between parts. For some systems, you may choose to perform a kinematic analysis of the parts in the assembly. You can then check each part for interference or clearance problems as the parts move through the kinematic analysis. For example, if you are designing a piston and cylinder system, in the initial position, none of the parts will overlap. However, as the crankshaft is rotated and the piston slides in or out of the cylinder, you may find that the connecting rod interferes with the end of the cylinder, as illustrated in Figure 6.20. Since that is a problem with the design, you can modify the design to alleviate the problem and re-check the interferences and clearances.

One other consideration in accomplishing interference or clearance checks for an assembly is the *number* of checks you want to perform. If you have a system with 25 parts, you probably will not want to check for interference between all parts in the assembly; that would require $(25 - 1)!$ (24 factorial, or more than 6×10^{23}) checks. With most CAD software, you can specify two groups of parts. The software will perform the interference or clearance checks, checking all of the parts in one group against all of the parts in the second group.

Only one of the two, interference or clearance, is possible between two instances. If two parts overlap, there is an interference; a clearance is not defined between the two. Conversely, if two instances have a clearance, or distance, between them, they do not overlap; an interference is not defined.

One common place where you need to check interferences and clearances is between shafts and circular holes. When you align shafts in holes by making them concentric, you may not realize that the shaft is too large or too small for the hole. In a later chapter, you will learn about classes of fits between holes and shafts. In that chapter, you will learn that sometimes you want the shaft to be larger than the hole and other times you want the shaft to be smaller than the hole. The class of fit you need depends on your design intent. Figure 6.21 shows an end view of two shafts in holes—in one case, the shaft is larger than the hole; in the other case, the shaft is smaller than the hole. The clearance and interference between the shaft and the hole in each case are shown in the figure.

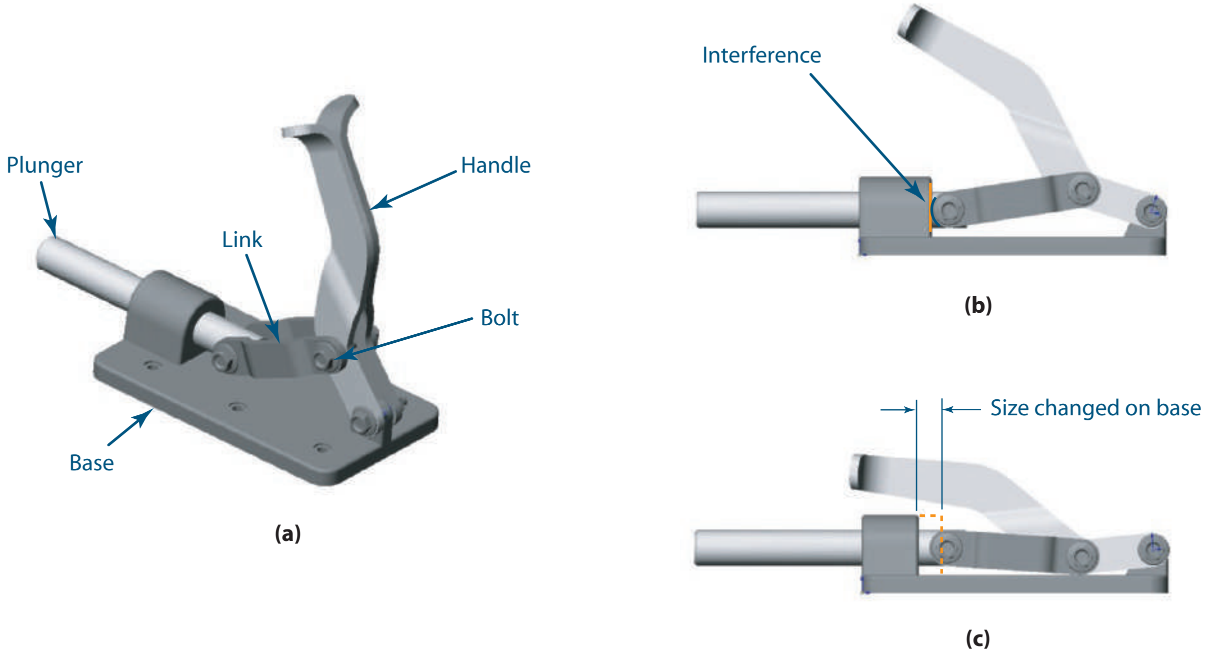
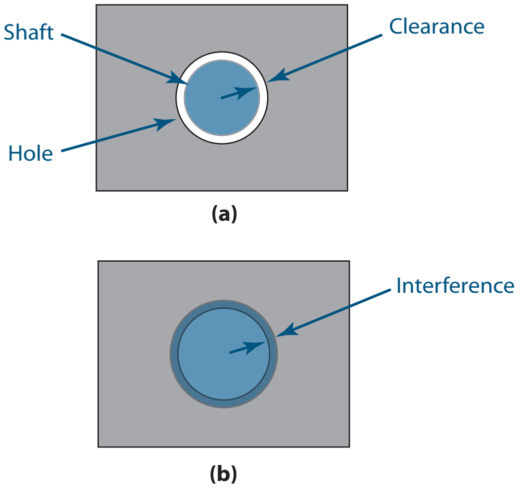


FIGURE 6.20. The completed toggle clamp assembly is shown in (a). In its original form, the link and the base interfere, as shown in (b), preventing the handle from being pushed all the way down into a closed and locked position. The base has been modified in (c) to eliminate the interference, allowing the toggle to work properly.

FIGURE 6.21. When the shaft has a smaller diameter than the hole, as in (a), there is clearance between the two parts. If the shaft has a larger diameter than the hole, as in (b), there is interference between the parts.

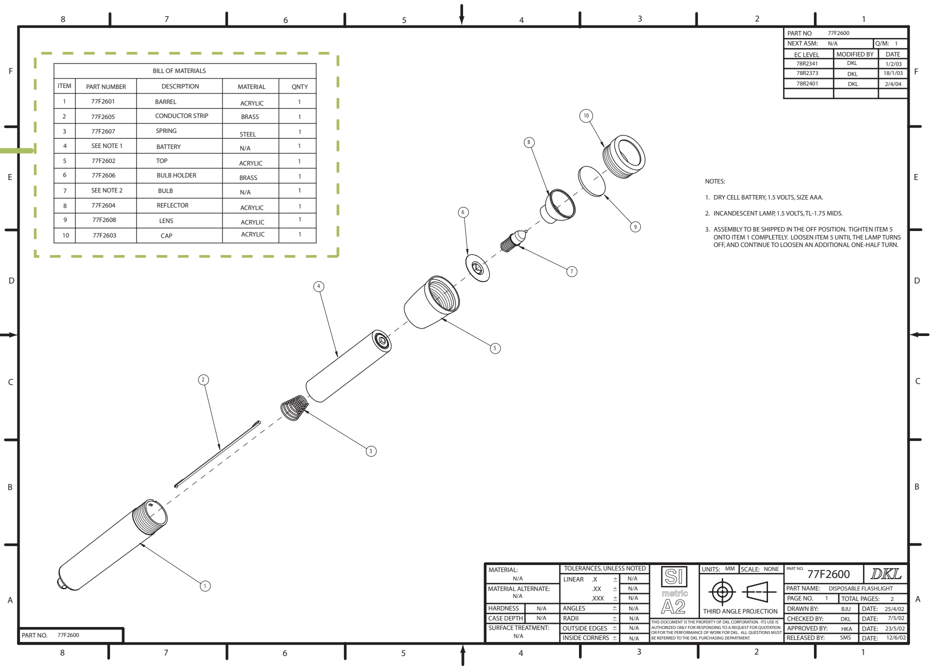


6.07 Bill of Materials

Most real-life assembly drawings contain a **bill of materials**, which consists of a table listing all parts in the assembly as well as the quantity of each part required to put the assembly together. If you were to begin assembling a system, you could look at the bill of materials and easily determine how many screws, how many washers, etc., you needed to gather to assemble the system. Figure 6.22 shows an assembly drawing (exploded configuration) for a system, along with the associated bill of materials.

In some cases, a bill of materials lists subassembly items, as shown in Figure 6.23. In that case, the assembly drawing for the subassembly is usually shown on a different sheet of paper and you are expected to have constructed the subassembly before you begin putting the entire assembly together.

FIGURE 6.22. The assembly drawing for a small flashlight, with its bill of materials on the drawing.



BILL OF MATERIALS				
ITEM	PART NUMBER	DESCRIPTION	MATERIAL	QNTY
1	77F2601	BARREL	ACRYLIC	1
2	77F2605	CONDUCTOR STRIP	BRASS	1
3	77F2607	SPRING	STEEL	1
4	SEE NOTE 1	BATTERY	N/A	1
5	77F2602	TOP	ACRYLIC	1
6	77F2606	BULB HOLDER	BRASS	1
7	SEE NOTE 2	BULB	N/A	1
8	77F2604	REFLECTOR	ACRYLIC	1
9	77F2608	LENS	ACRYLIC	1
10	77F2603	CAP	ACRYLIC	1

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FIGURE 6.23. The bill of materials for two levels of subassembly, leading to the final assembly of the footbridge.

BILL OF MATERIALS, PART NUMBER XKZ0030, WEB ASSEMBLY				
ITEM	PART NUMBER	DESCRIPTION	MATERIAL	QNTY
1	XKZ0001	VERTICAL MEMBER	WOOD	1
2	XKZ0002	ANGLED MEMBER	WOOD	2

BILL OF MATERIALS, PART NUMBER XKZ0015, HANDRAIL ASSEMBLY				
ITEM	PART NUMBER	DESCRIPTION	MATERIAL	QNTY
1	XKZ0001	VERTICAL MEMBER	WOOD	1
2	XKZ0003	HORIZONTAL MEMBER	WOOD	2
3	XKZ0030	WEB ASSEMBLY	WOOD	2

BILL OF MATERIALS, PART NUMBER XKZ0009, DECKING ASSEMBLY				
ITEM	PART NUMBER	DESCRIPTION	MATERIAL	QNTY
1	XKZ0006	DECK	WOOD	1
2	XKZ0007	SURFACE LAYER	FIBER REINFORCED CONCRETE BOARD	2

BILL OF MATERIALS, PART NUMBER XKZ0001, FOOTBRIDGE ASSEMBLY				
ITEM	PART NUMBER	DESCRIPTION	MATERIAL	QNTY
1	XKZ0023	ABUTMENT	POURED CONCRETE	2
2	XKZ0013	APPROACH	POURED CONCRETE	2
3	XKX0009	DECKING ASSEMBLY	WOOD, CONCRETE	1
4	XKZ0015	HANDRAIL ASSEMBLY	WOOD	2

6.08 Assembly Strategy

When creating an assembly, you can choose from two modeling methods—top-down or bottom-up. Bottom-up assembly modeling was used extensively in the past; however, top-down modeling is now gaining in popularity. A description of each method follows.

6.08.01 Bottom-Up Assembly Modeling

With **bottom-up modeling**, you create all of the parts required for the system. When more than one copy of a given part is in an assembly, you do not have to create more than one copy of the part—you include multiple instances of the component in the system. You then establish the assembly hierarchy, making sure you organize instances into smaller subassemblies as appropriate. You orient all of the instances using constraints to establish relationships between instances. Next, you check for clearances and interferences between instances to make sure your constraints were properly applied and you achieved your design intent. If you so choose, you obtain a bill of materials for the assembly. Your final step is to create an exploded view of the system as necessary to create its assembly drawing.

6.08.02 Top-Down Assembly Modeling

Top-down assembly modeling has evolved in recent years in much the same way concurrent engineering evolved—both were facilitated through the modern computer tools and CAD systems. With **top-down modeling**, the system is first defined, including its hierarchy. In some cases, physical space may be assigned to the assembly as well as to all subassemblies and components contained in it. The function of the system and its components also is articulated at this time. After the framework of the system has been established, engineering teams then work on creating the individual parts and subassemblies that go into the overall system. For example, because the engineers working on subassembly A know that their subassembly connects and interacts with subassemblies B, D, and G, they can collaborate through e-mails or other means as they create their assigned subassembly. Further, the team members working on subassembly A know that the space they occupy cannot encroach on the space occupied by subassembly C and must take that fact into account when doing their design work. Teams working on subassembly B know that they must work with teams working on subassemblies A and C as they complete their assigned task. In this way, efficiency in assembly design is achieved—problems can be solved before they arise. With the bottom-up design approach, problems between subassemblies A and D might not be apparent until the entire design is completed, meaning that all teams need to start from scratch to solve the problem. Top-down assembly modeling is especially effective when multiple teams, many times in different parts of the world, are working on a single design. Most modern-day CAD systems have the tools necessary to implement a system of top-down assembly modeling; however, as a student, you may not have the opportunity to work in this efficient environment.

6.09 Strategy for Bottom-up Assembly Modeling

Suppose you needed to model the assembly shown in Figure 6.24. The distinct components that make up the assembly are shown in Figure 6.25. Note that the bottom-up approach will be employed in the assembly modeling of this case, since all of the parts have already been created and must simply be put together to form the assembly model.

FIGURE 6.24. A fully assembled model of a bearing block.

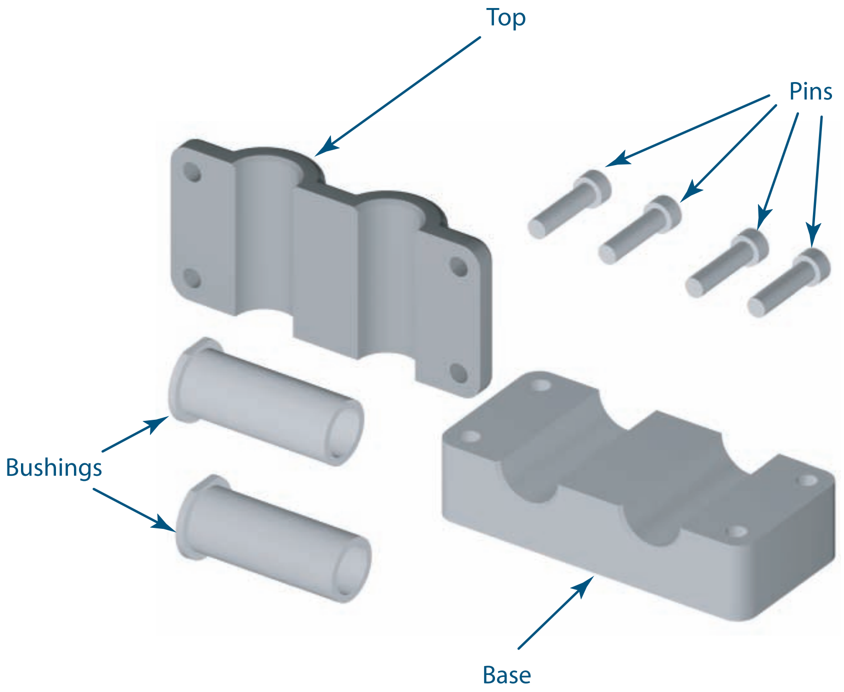


FIGURE 6.25. An exploded view of the parts that form the bearing block assembly.

With the bottom-up assembly modeling approach, after the individual components have been modeled, you need to define a hierarchy for the system. For this particular assembly, it might make sense to link the four pins into one subassembly; however, this is probably not necessary. Since the bushings have a flat surface for alignment, you probably need to know if they should be aligned independently of each other before you decide whether they make up a subassembly. Since the base block and the cap block will likely be considered a single unit, the two components should be put into a subassembly. Two possible hierarchies for the system are shown in Figure 6.26.

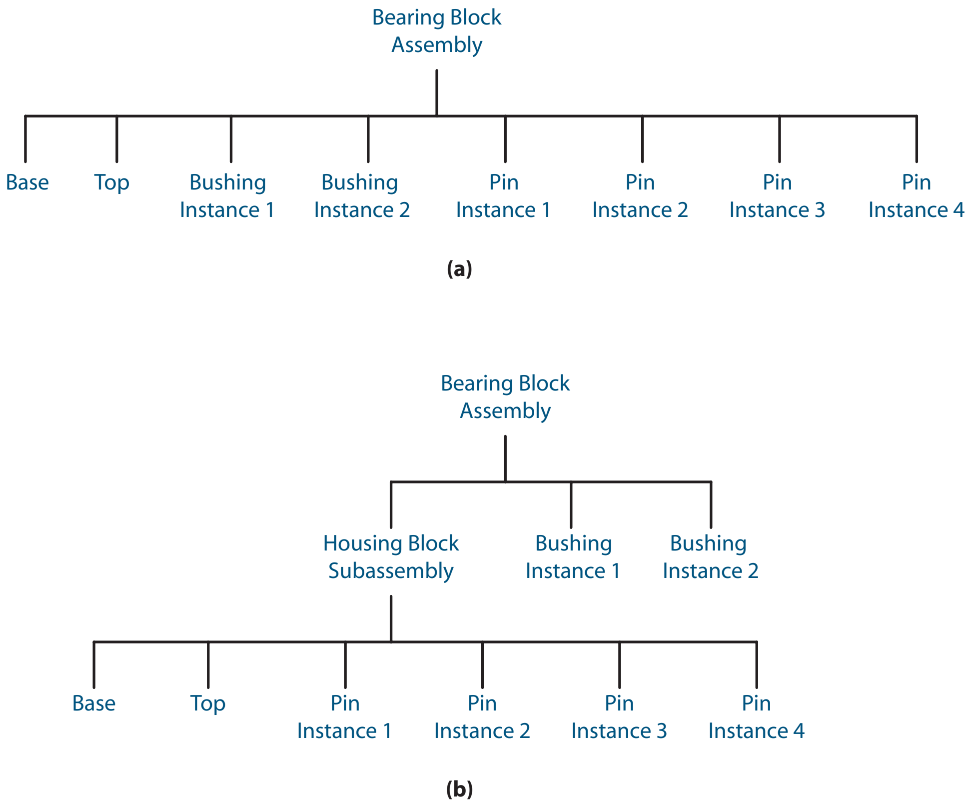
For purposes of demonstration, you will work with the hierarchy established in Figure 6.26a. In this case, your first step is to assemble the base subassembly. This subassembly consists of the base block and the cap block. You should establish the base block as the base instance for the assembly, setting the coordinate planes for the system as shown in Figure 6.27.

If you bring the cap block onto the screen, you can put it in place within the system through the use of coincident constraints. For this subassembly, the edges of the half circles of the cap and base should coincide. If this constraint is applied, notice the two possible orientations for the cap where this constraint is satisfied, as shown in Figure 6.28.

To achieve the desired orientation for the cap block, you need to apply one more coincident constraint using any of the remaining edges, or you can use the endpoints of the corresponding half circles. Figure 6.29 shows the second coincident constraint applied, which leads to the desired final result.

Now you put the pin subassembly in place within the overall system. The pin subassembly consists of four instances of the pin component. You can bring the pins on-screen one at a time and locate each of them relative to the base subassembly that is already in place. If you bring the first instance of the pin on-screen, you need to establish two constraints to put it in its final location. The two constraints to be

FIGURE 6.26. Two possible hierarchies for the bearing block assembly, with no subassemblies (a) and with a housing block subassembly (b).



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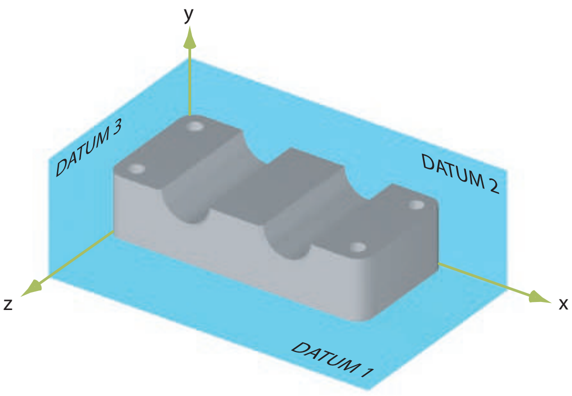

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FIGURE 6.27. Definitions of coordinate axes and datum planes for the base as the first object in the assembly model.

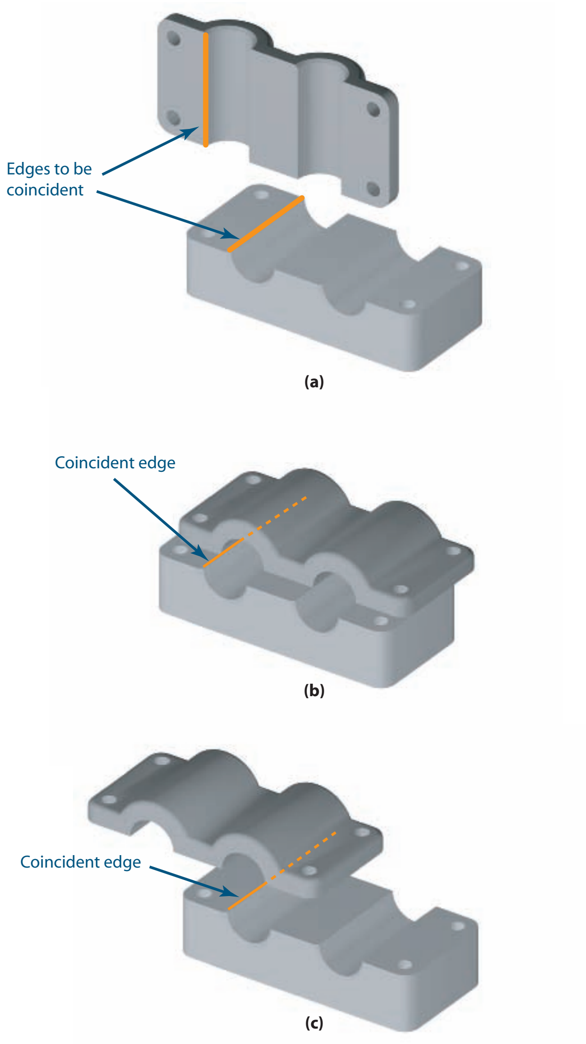

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FIGURE 6.28. A coincident constraint applied to the edges shown in (a) can result in two possible orientations of the top, as shown in (b) and (c).

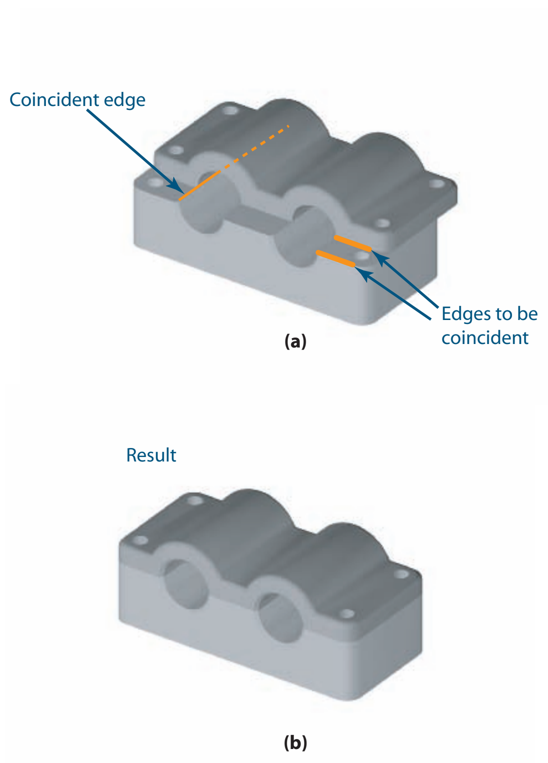

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FIGURE 6.29. Adding a coincident edge constraint to the edges shown in (a) is one method used to create the final alignment between the base and the top, as shown in (b).

added are a concentric constraint between the axis of the pin and the axis of the corresponding hole, and a mating surfaces constraint can be applied between the “bottom” surface of the pinhead and the top surface of the cap, block. These two constraints are shown in Figure 6.30a, with the final result shown in Figure 6.30b.

Using a similar strategy, the remaining three instances of the pin component can be oriented within the assembly, with the result shown in Figure 6.31.

Finally, the two bushings can be inserted one at a time into the assembly using a strategy similar to the one used in locating the pins—applying a concentric constraint between the centerlines of the bushing and the base-block semicircular cutout and a mating surfaces constraint between the corresponding surfaces of the bushing and the base block. Figure 6.32a shows the constraints applied to the leftmost bushing, and Figure 6.32b shows the result of applying these constraints.

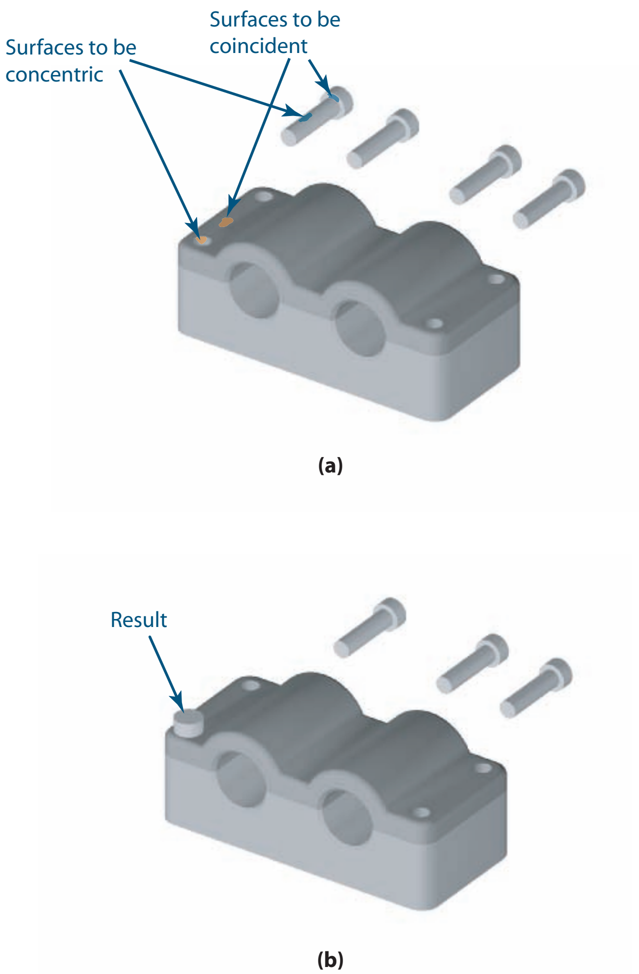

FIGURE 6.30. Applying a concentric constraint between the shaft of the pin and the hole in the top and a coincident constraint between the bottom of the pinhead and the top, as shown in (a), locates the pin in the hole, as shown in (b).

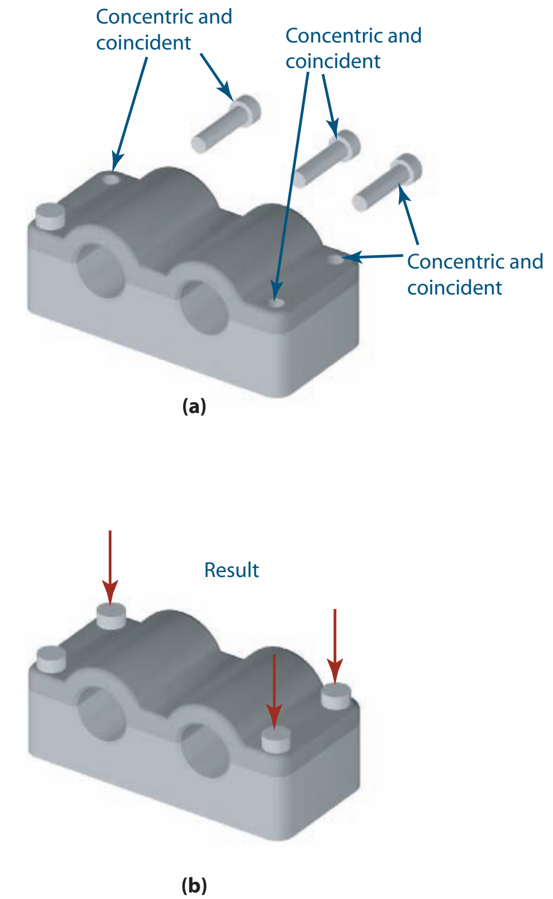

FIGURE 6.31. The remaining three pins are inserted in their holes by applying the same types of concentric and coincident constraints used for the first pin.

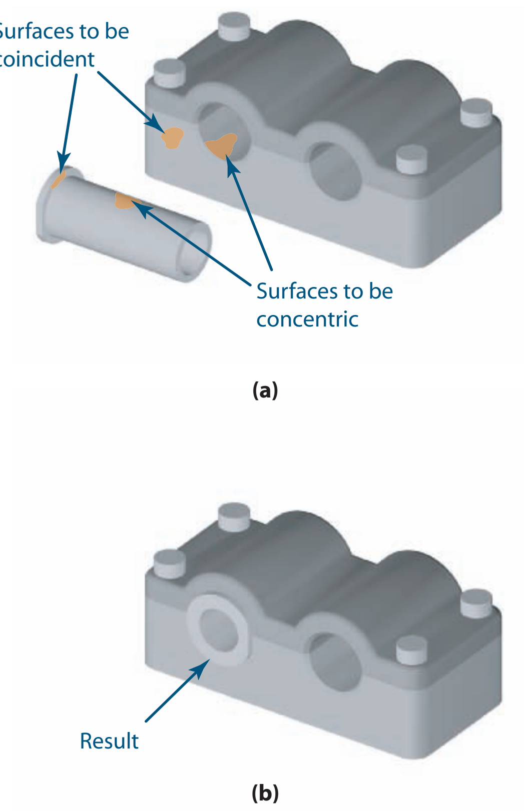

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FIGURE 6.32. The bushing is placed by applying the concentric and coincident constraints to the surfaces indicated in (a) to produce the result shown in (b).

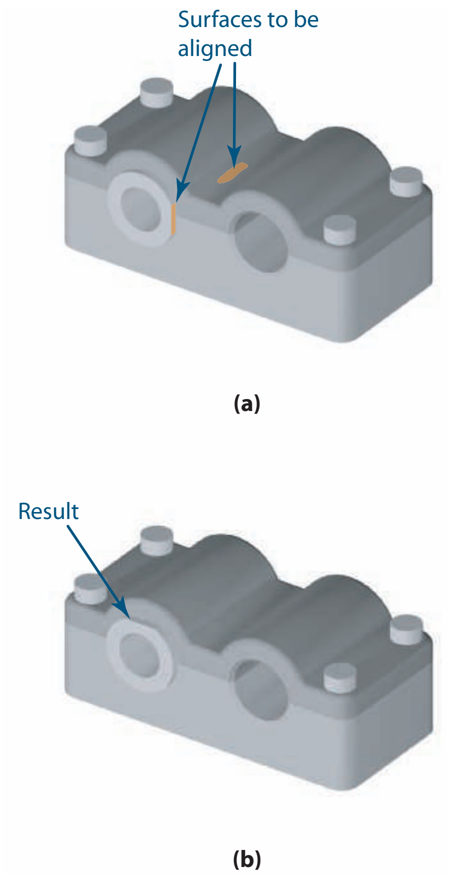

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FIGURE 6.33. An alignment constraint is applied between the flat on the bushing and the top surface, as shown in (a), to create the desired orientation of the bushing, as shown in (b).

When the pins were added to the assembly, it did not matter what angular orientation they had with respect to the axis of the pin; however, the bushing instances include a flat surface that can be used for alignment. To align the surface properly, you want to include a distance constraint between the flat surface on the bushing and the bottom (or top) flat surface on the base. In this case, the distance constraint forces the two surfaces to be parallel to each other. Figure 6.33a shows the constraint applied between the flat surface of the bushing and the upper flat surface of the top, and Figure 6.33b shows the result of applying this constraint.

Finally, the second instance of the bushing can be brought into the assembly and oriented with the use of appropriate constraints. The final assembly model is shown in Figure 6.34.

Once your assembly model is complete, you may want to check for interferences or clearances between instances. Figure 6.35 includes the results from an interference and clearance check for the instances in the assembly. Since there are relatively few instances in the assembly, all instances were checked against all others.

Other items you might need from this model include an assembly drawing and a bill of materials. Most modern-day software can generate these automatically. Figure 6.36 shows an assembly drawing with the bill of materials for the block assembly you have been working with thus far.

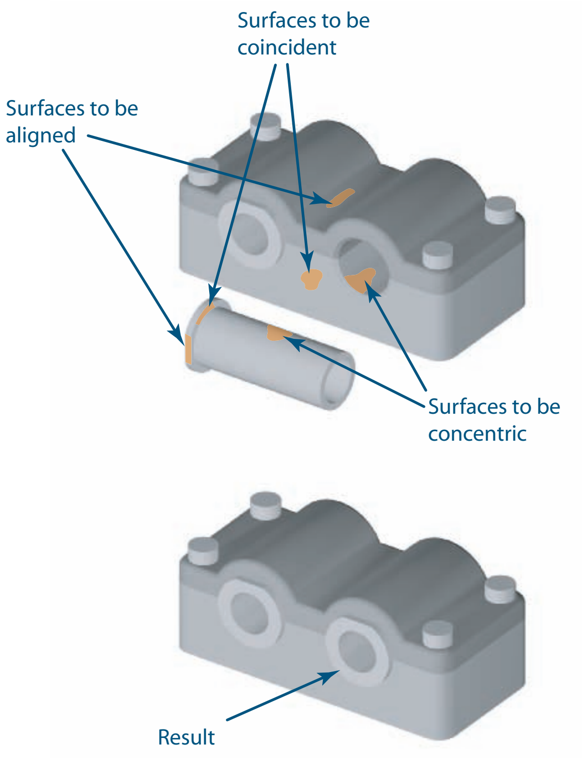

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FIGURE 6.34. The second bushing is placed by applying the same type of concentric, coincident, and aligned constraints as were used to place the first bushing, as shown in (a). The final position of the second bushing is shown in (b).

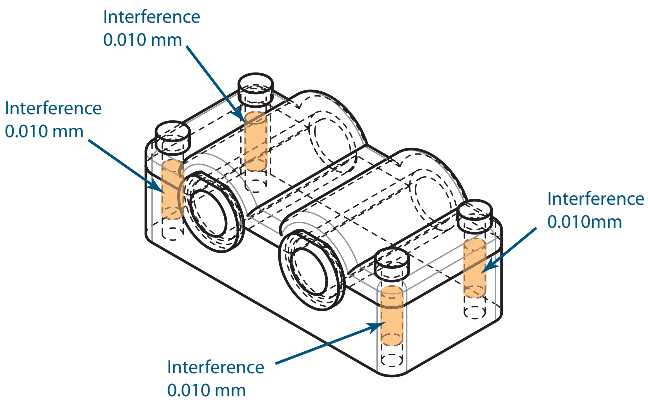

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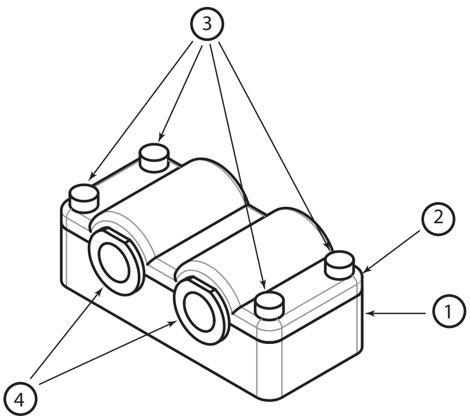
FIGURE 6.35. The result of an interference check between all of the parts in the bearing block assembly, showing the interference from the forced fit between the pins and the base block.

FIGURE 6.36. An assembly drawing of the block assembly (removed from its drawing header) identifying its parts in its bill of materials.

BILL OF MATERIALS, PART NUMBER CDX010, BEARING BLOCK ASSEMBLY				
ITEM	PART NUMBER	DESCRIPTION	MATERIAL	QNTY
1	CDX011	BASE BLOCK	ALUMINUM, 6061 T6	1
2	CDX012	CAP BLOCK	ALUMINUM, 6061 T6	1
3	CDX089	PIN	STEEL, 1060	4
4	CDX076	BUSHING	BRONZE, SINTERED	2



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6.10 Chapter Summary

Most engineered devices are composed of many different parts, all of which must fit together properly in order for the devices to function in their intended manner. To create an assembly model, the individual part models of a system must be created first. The fitting of the individual part models together is simulated using assembly constraints, much in the same manner as putting actual parts together. Assembly models are invaluable tools for ensuring that the various parts and pieces of a system fit and function properly. Assembly models also have the benefit of creating a database from which important documents such as assembly drawing and bills-of-material can be easily created.

6.11 GLOSSARY OF KEY TERMS

assembly constraints: Used to establish relationships between instances in the development of a flexible assembly model.

associativity: The situation whereby parts can be modified and the components referenced to the parts will be modified accordingly.

base instance: The one fixed instance within an assembly.

bill of materials: A tabular list of the components, with quantities of each for the parts, that make up an assembly.

bottom-up modeling: The process of creating individual parts and then creating an assembly from them.

clearances: The minimum distances between two instances in an assembly.

components: References of object geometry used in assembly models.

exploded configuration: A configuration of an assembly that shows instances separated from one another. An exploded configuration is used as the basis for an assembly drawing.

hierarchy: The parent-child relationships between instances in an assembly.

instances: Copies of components that are included within an assembly model.

interference: The amount of overlap between two instances in an assembly.

subassembly: A logical grouping of assembly instances that is treated as a single entity within the overall assembly model.

top-down modeling: The process of establishing the assembly and hierarchy before individual components are created.

6.12 QUESTIONS FOR REVIEW

1. Describe the differences between an object and a component.
2. What is an instance?
3. What does the term *associativity* mean in the context of assembly modeling?
4. What types of relationships are made when an assembly model is established?
5. Name and describe three types of assembly constraints.
6. Define *interference* and *clearance* in the context of assembly modeling.
7. What is a bill of materials?
8. What is the primary difference between bottom-up assembly modeling and top-down assembly modeling?

6.13 PROBLEMS

1. The two parts in Figure P6.1 are to be mated together as shown. Using only the features labeled, apply assembly constraints to mate the two pieces so that the top part is fully constrained and assembled correctly with the bottom part. Assume the bottom part is already fixed in position. Specify five ways of doing that using only coincident and concentric constraints. An example follows:

   Constraint set 1: hole 1 concentric with hole 2
   surface 1 coincident with surface 3
   surface 2 coincident with surface 4

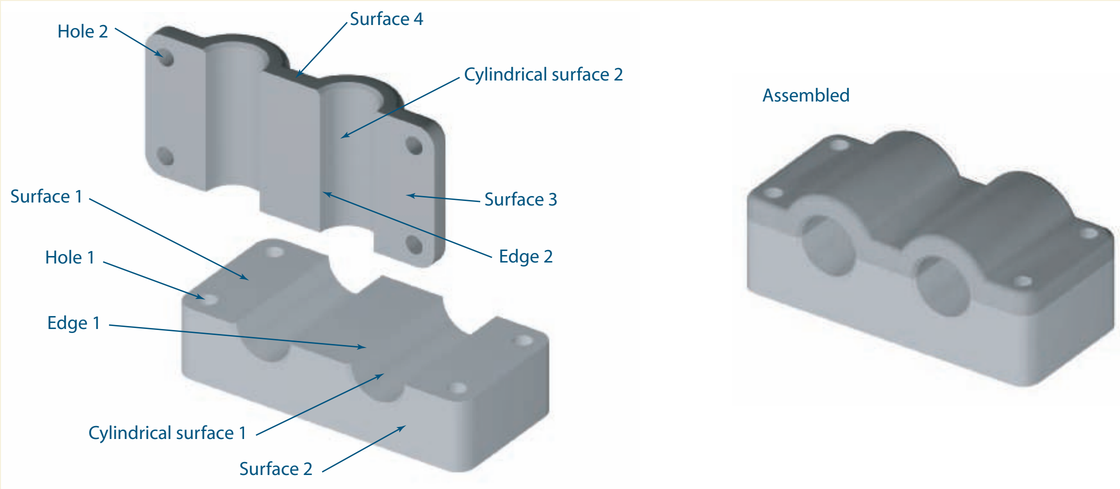

FIGURE P6.1. Constrain the edges and surfaces of (a) to create the assembled position (b).

2. Assembled and exploded views for a staple remover are shown in Figure P6.2. Using reasonable materials and dimensions of your choice, create solid models of the individual parts. Using assembly constraints, create an assembly model of the staple remover.

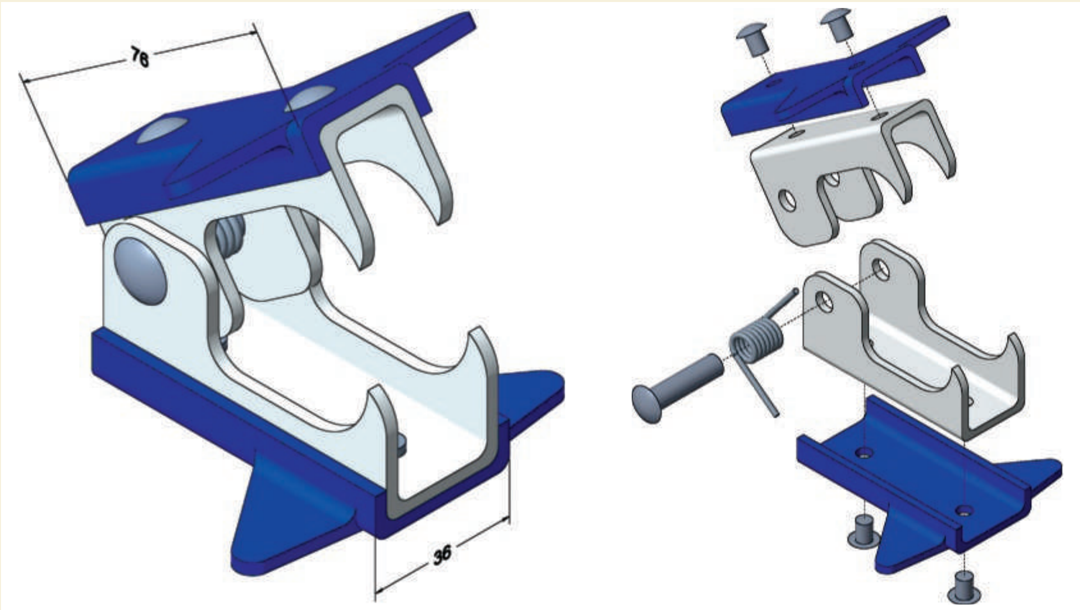

FIGURE P6.2.

6.13 PROBLEMS (CONTINUED)

3. Figure P6.3 shows a conceptual model for a pen-type eraser in whole and in cutaway view. Using reasonable materials and dimensions of your choice, expand the concept to create solid models of the individual parts. Using assembly constraints, create an assembly model of the eraser.

FIGURE P6.3.

4. Figure P6.4 shows a conceptual model for a garden hose nozzle in whole, cutaway, and exploded views. Using reasonable materials and dimensions of your choice, expand the concept to create solid models of the individual parts. Using assembly constraints, create an assembly model of the nozzle.

FIGURE P6.4.

5. Assembled and exploded views for a toggle clamp are shown in Figure P6.5. Using reasonable materials and dimensions of your choice, create solid models of the individual parts. Using assembly constraints, create an assembly model of the clamp.

FIGURE P6.5.

6.13 PROBLEMS (CONTINUED)

6. Assembled and exploded views for a spring carabineer are shown in Figure P6.6. Using reasonable materials and dimensions of your choice, create solid models of the individual parts. Using assembly constraints, create an assembly model of the carabineer.

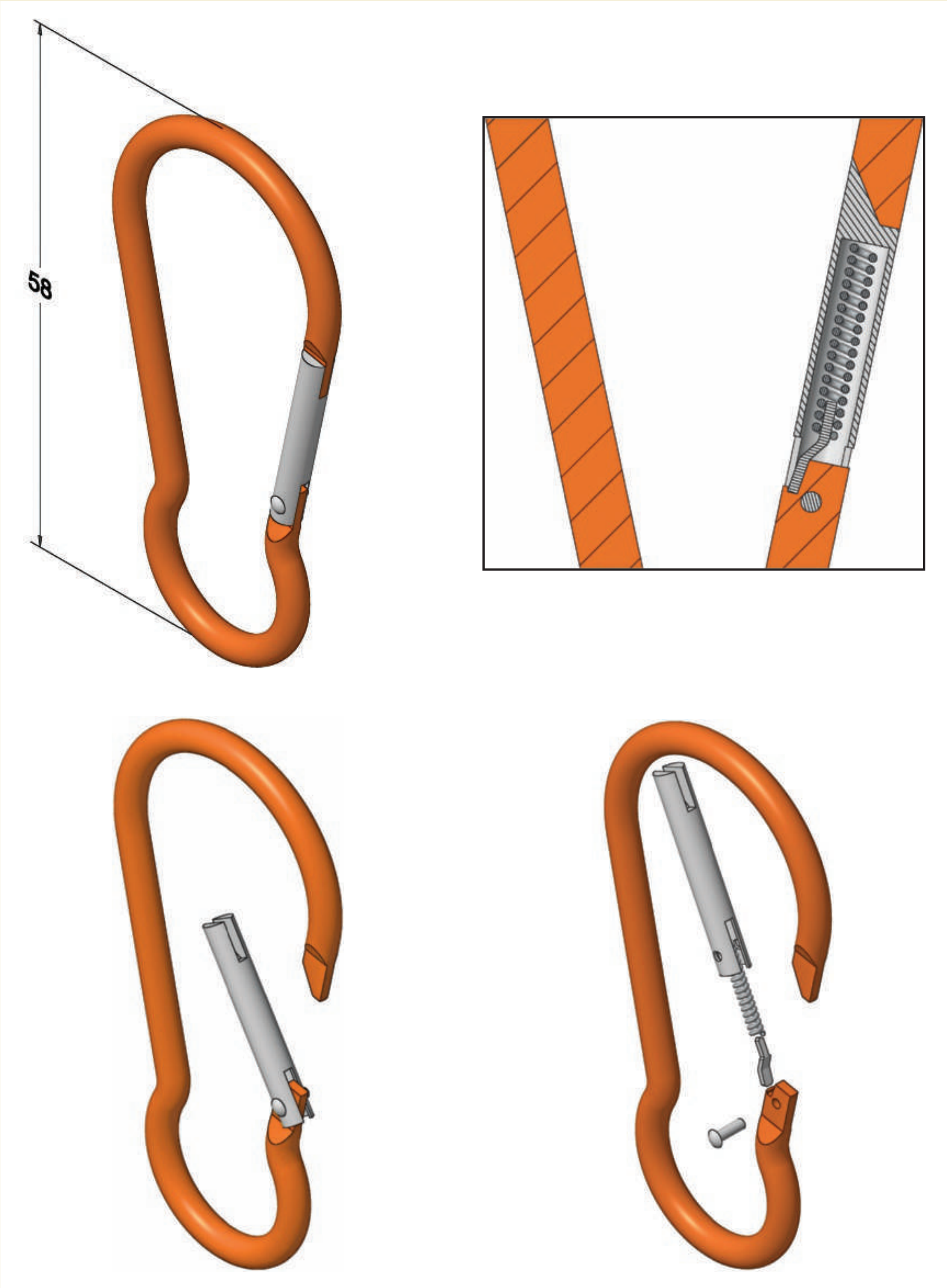

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FIGURE P6.6.

6.13 PROBLEMS (CONTINUED)

7. Assembled and exploded views for a universal joint are shown in Figure P6.7. Using reasonable materials and dimensions of your choice, create solid models of the individual parts. Using assembly constraints, create an assembly model of the joint.

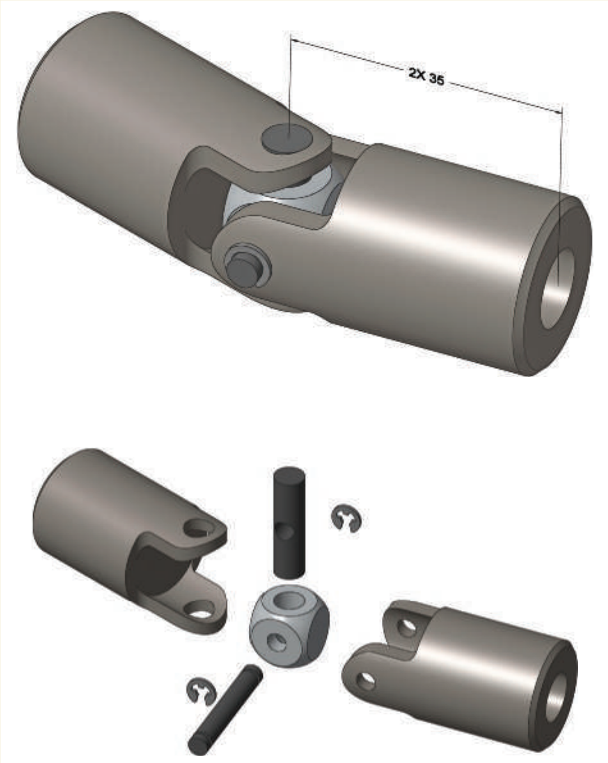

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FIGURE P6.7.

8. Assembled and exploded views for a gear puller are shown in Figure P6.8. Using reasonable materials and dimensions of your choice, create solid models of the individual parts. Using assembly constraints, create an assembly model of the puller.

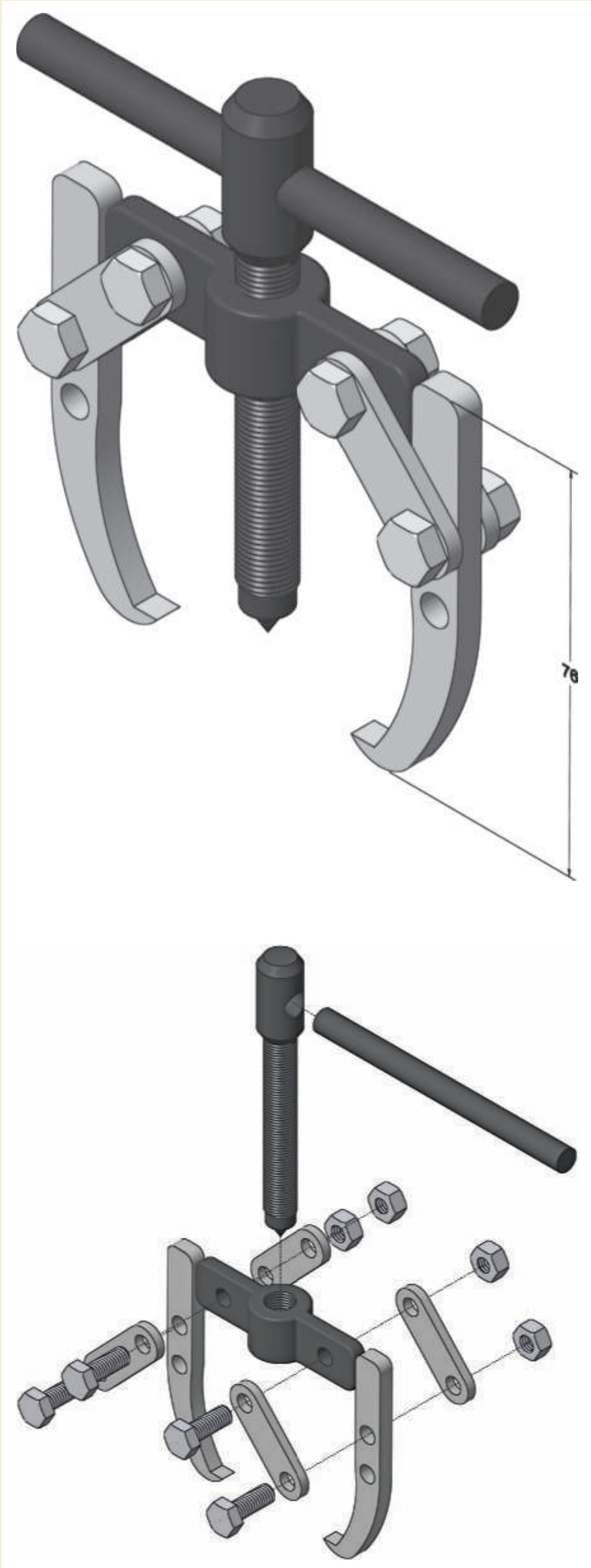

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FIGURE P6.8.

6.13 PROBLEMS (CONTINUED)

9. A concept drawing for a flexible shaft coupling is shown in Figure P6.9. Using reasonable materials and dimensions of your choice, create solid models of the individual parts. Using assembly constraints, create an assembly model of the coupling. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

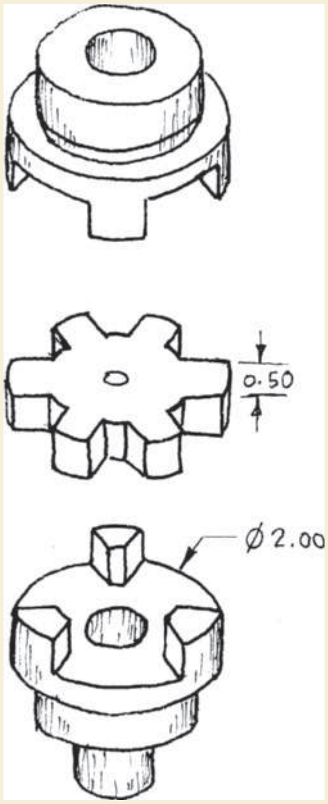

FIGURE P6.9.

10. A concept drawing for a clock stand is shown in Figure P6.10. Using reasonable materials and dimensions of your choice, create solid models of the individual parts. Using assembly constraints, create an assembly model of the stand. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

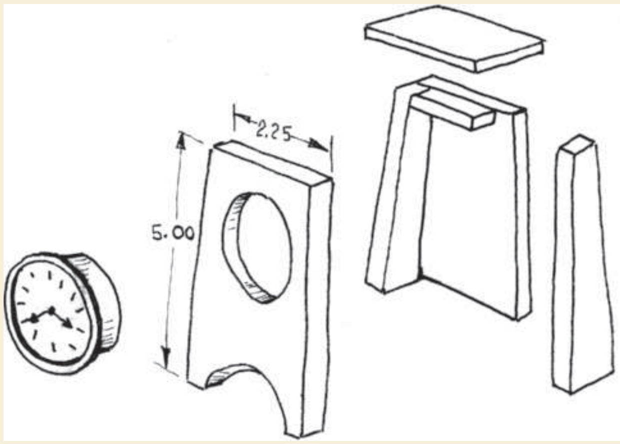

FIGURE P6.10.

11. A concept drawing for a work table is shown in Figure P6.11. Using reasonable materials and dimensions of your choice, create solid models of the individual parts. Using assembly constraints, create an assembly model of the table. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

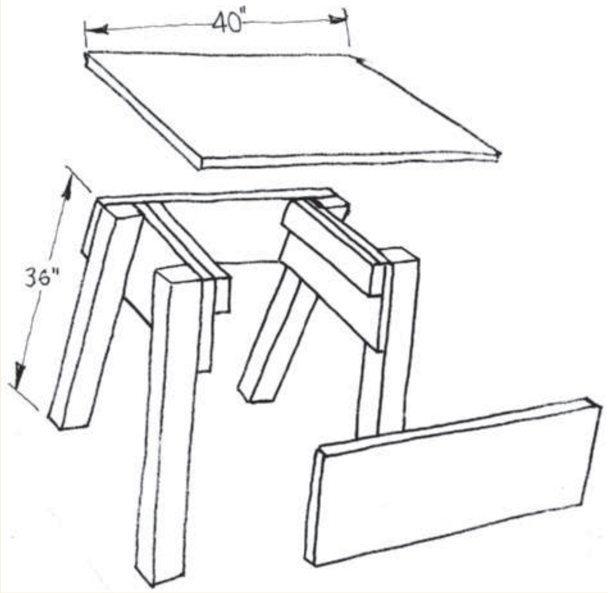

FIGURE P6.11.

6.13 PROBLEMS (CONTINUED)

12. A concept drawing for a utility clamp is shown in Figure P6.12. Using reasonable materials and dimensions of your choice, create solid models of the individual parts. Using assembly constraints, create an assembly model of the clamp. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

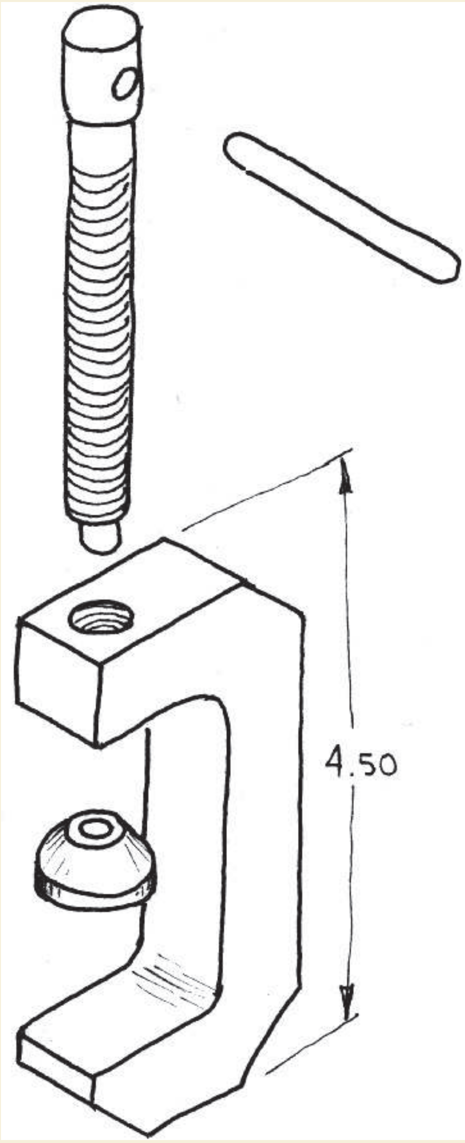

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FIGURE P6.12.

6.13 PROBLEMS (CONTINUED)

13. A concept drawing for a stop guide is shown in Figure P6.13. Using reasonable materials and dimensions of your choice, create solid models of the individual parts. Using assembly constraints, create an assembly model of the guide. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

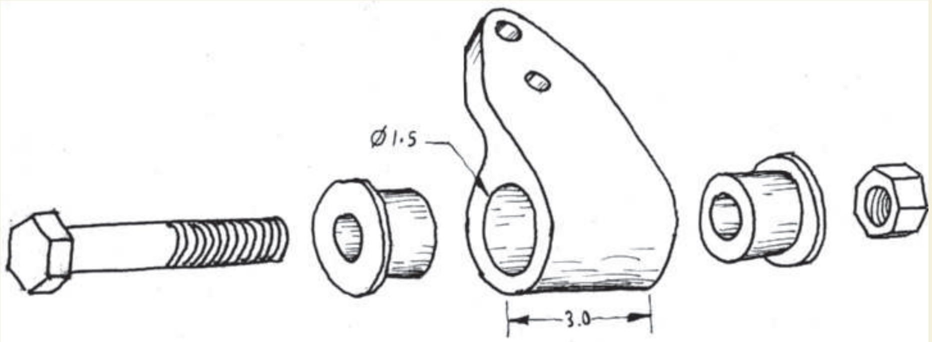

FIGURE P6.13.

14. A concept drawing for a caster wheel assembly is shown in Figure P6.14. Using reasonable materials and dimensions of your choice, create solid models of the individual parts. Using assembly constraints, create an assembly model of the caster. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

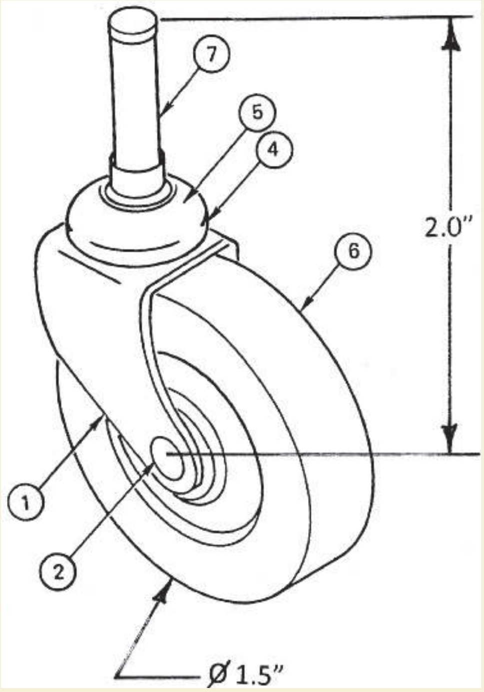

FIGURE P6.14.

CHAPTER 7

DESIGN ANALYSIS

OBJECTIVES

After completing this chapter, you should be able to

- Explain the importance of analysis in the design process
- Apply the reverse engineering process to a simple device
- Perform a mass properties analysis for a computer-generated model
- Describe the fundamental principles in performing a finite element analysis

7.01 INTRODUCTION

In an earlier chapter, you learned about the stages in the design process and about the role of the computer in the process. You also learned that some engineers focus on the design stage, some on the manufacturing stage, and some on the analysis stage. Most of the computer-based analysis techniques described in this chapter require a high level of mathematical understanding and are beyond the scope of this text. In fact, for some analysis techniques, such as the finite element method, entire books are written on the topic. In this chapter, you will learn about analysis techniques that are typically found in an introductory design graphics course. You should understand that many other types of analysis techniques are available to engineers as they design devices and systems. You will probably learn about many of these techniques as you make your way through your engineering curriculum. This chapter introduces three analysis techniques: reverse engineering, mass properties analysis, and the finite element method.

Analysis is a broad term that involves the study of the behavior of a physical system under certain imposed conditions. Analysis usually involves quantitative reasoning and techniques, but it might not. As you learned previously, engineering analysis characterizes a significant part of the design process, since all design ideas have to be verified for feasibility.

Analysis can be conducted in many different ways, but generally an underlying physical principle (equation) is involved. Although modern analysis is typically done using special computer programs, conducting an experiment in a laboratory setting or reverse engineering a product also is a feasible analysis method. Many of the numerical analysis techniques are based on advanced-level mathematical descriptions and are highly computerized. A few numerical techniques are as follows: structures can be analyzed for stresses and deformations; bodies, for motion; engines, for temperature distributions; and dynamic systems, for their vibrations. In performing lab experiments, engineers can determine the strength of a new design component through tensile testing. Through reverse engineering, systems and parts can be analyzed for their function and possible modification. These are just a few of the analysis tools available to engineers as they design products or infrastructure for the general public.

7.02 Reverse Engineering

Reverse engineering is a systematic methodology for analyzing the design of an existing device or system, either as an approach to study the design or as a prerequisite for redesign. Reverse engineering essentially is a process used to gain information about the functionality and sizes of existing design components. For instance, you might have an idea or a concept concerning a unique addition to an existing product or system. It would make sense that you first analyze the products or systems already on the market to see how they work and what features they have that could be used or improved for the new design. A careful dissection and study of several similar designs could contribute much toward the new process or design. In industry, the most common reasons for reverse engineering are these:

- Existing prototypes are available but no design data, CAD computer files, or drawings exist.
- The original manufacturer of the product or part is hesitant, unwilling, or unable to provide replacement parts for a system in use or the data to reproduce them.

- Old, worn, or broken parts for a system exist for which there are no known data sources.
- Existing geometry data needs to be modified to improve a part's functionality, dimensionality, or appearance.

It should be noted that reverse engineering is a legitimate activity in industry and is not the same as industrial espionage. Determining "how something works" is not stealing someone's ideas and manufacturing counterfeit products. Indeed, some engineering companies offer reverse engineering services using sophisticated measuring systems.

Reverse engineering is a technique within the practice of engineering design that can be useful in several ways. Reverse engineering can save time because there is no need to "reinvent the wheel" when you can start from existing geometric data. The reverse engineering technique also can help an engineer develop a systematic approach to thinking about and improving the design of devices and systems. Seeing and holding the existing parts, noting their dimensions, and understanding their mating relationships improve an engineer's design and visualization abilities. By repeated practice of the reverse engineering process, you can gradually acquire a mental data bank of many design solutions for use in future products.

Reverse engineering is sometimes called **mechanical dissection** because it involves taking apart, or dissecting, a mechanical system. As you dissect the system, you carefully sketch the parts and show how they fit and work together so the system can be reassembled at a later date. You also need to measure all of the features on each part carefully during the dissection process so you can create solid models of them at a later time. Since accurate measurements are a significant part of the reverse engineering process, this chapter will discuss some of the more common measurement tools available. In engineering, the practice of measuring parts is called **metrology**.

7.03 Metrology Tools for Reverse Engineering

Reverse engineering is essentially a measurement and documentation process. As you take systems apart, you *document* how the parts work together and *measure* the sizes of the features of the parts. Tools used to do this measuring vary from simple handheld devices to the most sophisticated computer-driven machines. Accuracy will usually be a direct function of the cost of the metrology device, with less expensive tools providing less accurate measurements. Traditional inexpensive **engineering scales** or a ruler can be used for linear measurements with 0.25 mm "eyeball" accuracy; however, this is not always accurate enough for reverse engineering purposes. The following sections outline some common reverse engineering metrology tools.

7.03.01 Handheld Calipers

Other than engineering scales or rulers, the most economical tool for reverse engineering is a handheld **caliper**. Fairly accurate measurements can be obtained using a set of calipers such as those shown in Figure 7.01. With calipers such as these, the jaw, shown on the left, expands to fit around the part being measured. The distance between the flat surfaces of the caliper—and thus the distance you are trying to measure—can be determined by reading the scale on the long side of the device in combination with the digital readout. Calipers also can be used to determine the inside diameters of holes and other features through use of the opposite ends of the jaws. Typical calibers have a jaw-gap range of up to 100 mm, with a resolution accuracy of 0.025 to 0.010 mm. The numerical readout of calipers can be either a dial gauge needle or a digital LCD display, and they come in both English and metric units.

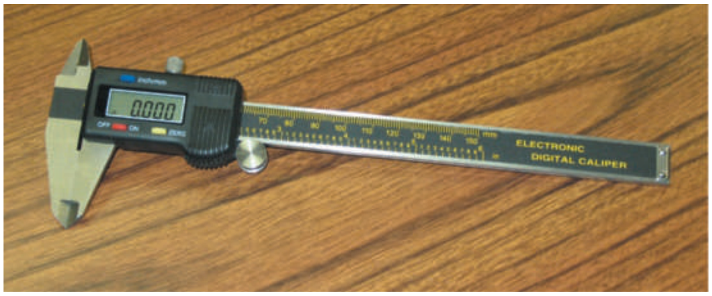

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FIGURE 7.01. Handheld calipers.

Courtesy Hexagon Metrology, Inc.

FIGURE 7.02. A single probe coordinate measuring machine.

7.03.02 Coordinate Measuring Machine (CMM)

Coordinate measuring machines, such as the one shown in Figure 7.02, use a probe to touch the part. The computer system built into the probe senses the x-, y-, and z-coordinates of the location of the probe and records that information into memory. The probe then lifts and touches the next point on the part, once again recording the coordinates of the endpoint of the probe. The system repeats the process until the whole part has been systematically scanned. This process is sometimes called digitizing the model since it converts the 3-D continuous part geometry to digital data. The points determined by the probe can then be used to build a 3-D computer model of the part. High-density scans, where the probe moves a small distance each time, may take a long time, but the result is a data set with accuracy in the 0.001-mm range. Typical volume capacities (measurement space) for CMM systems are 0.6 m × 0.6 m × 0.6 m and larger, with larger volumes greatly increasing the cost of the system.

7.03.03 3-D Laser Scanner

Laser scanning systems use a laser projector and cameras to determine part dimensional coordinates to a high degree of accuracy. During a scan, the projector shines a series of white-light stripes on the object. The two cameras pick up the light and use the principle of triangulation to determine the point's x-, y-, and z-coordinates. As many as 1.3 million coordinate points can be obtained in one scan session in less than 24 hours with an accuracy of 0.025 mm per point. Similar to working with a CMM system, the points can then be downloaded to a CAD system for solid modeling creation. A two-camera laser scanning system is shown in Figure 7.03. Due to the high cost of both CMM and laser scanning systems, you will likely be using rulers, scales, and calipers for any mechanical dissection projects you are involved in at the university, at least in your initial years of study.

Courtesy of Z Corporation

FIGURE 7.03. A handheld 3-D laser scanner used to extract geometry.

7.04 The Reverse Engineering Process

Like many technical activities, reverse engineering can be viewed as a process. In the following sections, the reverse engineering process is illustrated through a simple example—a common kitchen scale such as the one shown in Figure 7.04.

7.04.01 Defining the Reverse Engineering Project

Just like design projects, the first step in a reverse engineering project is to clearly define the project and determine the relevant factors and parameters. Careful planning of the project begins with assessing the customer needs and the engineering specifications. The overall functionality of the system, such as input and output relationships, also must be defined. Finally, subsystems and individual components should be estimated and outlined, perhaps through the use of a product dissection approach. After you have determined the relevant parameters and factors, you should summarize your findings in a problem statement so you have a clear picture of the reverse engineering project—what you hope to discover and what you hope to show. If your goal is to make improvements to a product, you should include that goal in the problem statement.

The problem statement can be in narrative or bulleted format, but you may find it easier to list the items rather than write a narrative statement. The problem statement should include a clear definition of the project/problem, a look at what a customer might be expecting in the product, a view of what an engineer will have to consider, as well as a definition of how the product must function. Figure 7.05 illustrates a problem statement for the kitchen scale project.

FIGURE 7.04. A kitchen scale.

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The Small Kitchen Scale

The Problem Statement

The Kitchen Scale is a useful device to measure out small amounts of food that are to be prepared. There are many people who need to be careful of the quantity of food that they take in. Diabetics must be careful of the amount of certain foods that they eat in order to maintain a proper blood sugar level. People on weight loss diets must also be careful with the number of calories that they consume at every meal. The Kitchen Scale is a device that can help to control the amount of food consumed by an individual.

Customer's Perspective

Some of the things that the customer might look for in a kitchen scale are the cost of the item and its appearance. Some other factors he/she would be concerned about would be how sturdy the device is, the accuracy of the weighing process, and its size. The customer may even ask if he/she is getting quality for the price.

Engineer's Perspective

The engineer has a totally different perspective from which she operates. The engineer will be looking at things like the material that the object is made of, the mechanisms that are required, the strength of the materials used, etc. In relation to the kitchen scale: Is the riveting process of the plate to the stem adequate? Are the materials used strong enough to hold 1 pound of weight? Does the scale have any sharp edges that could cause injury? What mechanism is necessary to give an accurate reading on the scale?

Functional Requirements

The kitchen scale must be able to accommodate 1 pound of food.

The plate must be large enough to hold 1 pound of food.

The stem must be strong enough to support the plate and the food.

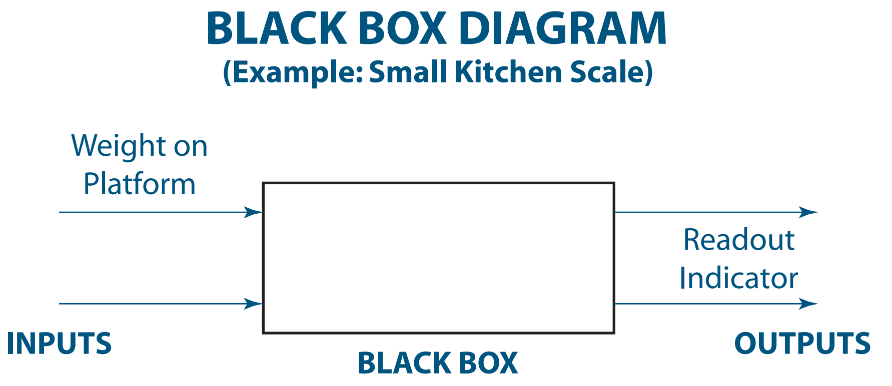
The linkage must be designed so that the reading is accurate to the nearest .5 ounce.

The read-out must be easily read.

The design must lend itself for easy cleaning and sanitizing.

FIGURE 7.05. A problem statement for the kitchen scale reverse engineering project.

FIGURE 7.06. A black box diagram for the small kitchen scale.



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As part of this initial stage in the reverse engineering project, you may find it helpful to create a black box diagram. With a **black box diagram**, you specify the overall inputs and outputs on the system, ignoring the inner workings of the device. Sometimes this black box diagram is referred to as the "view from 10,000 feet." A black box diagram of the small kitchen scale is shown in Figure 7.06.

At this time, you also might want to develop a sketch of the device for reference later. Figure 7.07 shows a sketch of the kitchen scale.

7.04.02 Dissecting a System

Once the preliminary examination of the project is completed, the problem statement is completed, and the black box diagram has been created, it is time to dissect the device. As you take apart the device, you need to document how the mechanism was dissected. Write down detailed notes for every step in the dissection process, because you may need the directions to reassemble the device at a later date. Figure 7.08 shows the dissection notes obtained as the kitchen scale is dissected. As part of

FIGURE 7.07. A sketch of the kitchen scale.

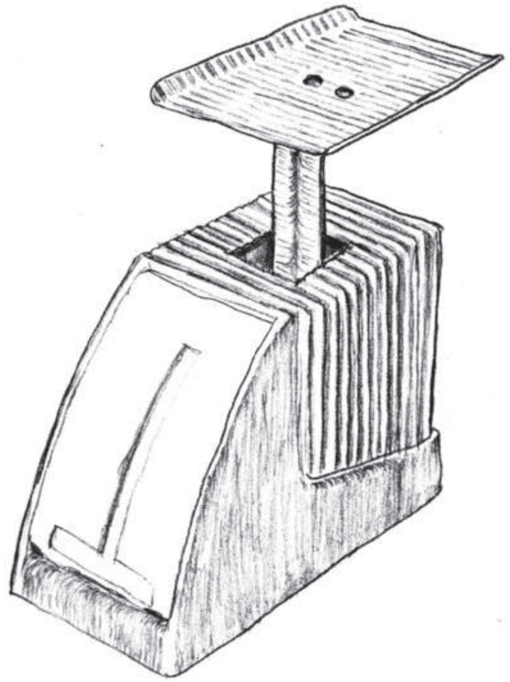

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FIGURE 7.08. Dissection notes.

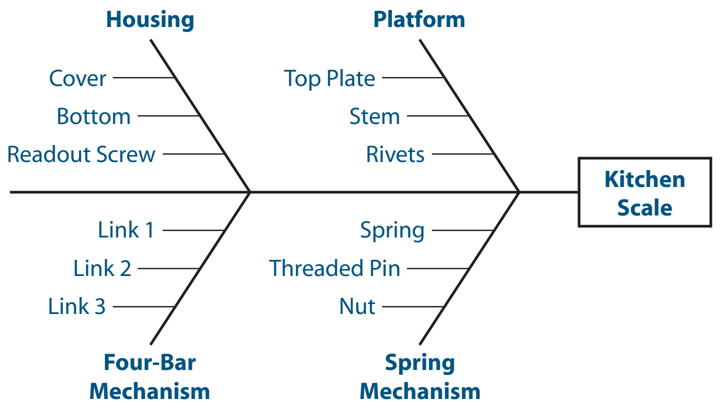
Dissection Notes for a Small Kitchen Scale

1. Remove the small Phillips-Head screw from the bottom of the scale.
2. Remove the cover plate from the bottom of the scale.
3. Remove the small brass knurled nut located under the weighing plate.
4. Release the spring from the lower end of the metal stem inside the scale, and remove the spring and threaded stem subassembly.
5. The four-bar linkage will now release from the back wall of the scale body.
6. Gently remove the metal stem from the plastic four-bar linkage.
7. The weighing plate and the stem can now be removed from the scale.
8. The read-out scale can now be removed from the front of the scale body.
9. If deemed necessary, the four-bar linkage can be taken apart and the spring can be removed from the threaded shaft.

the documentation of the dissection, you also should create a fishbone diagram. A **fishbone diagram** shows each of the subassemblies as a major "stem" from the "backbone" with all of the individual parts within a subassembly shown as branches from the corresponding stem. Figure 7.09 shows a fishbone diagram for the kitchen scale. Notice that for this diagram, four major subassemblies are identified: the housing, the platform, the four-bar mechanism, and the spring mechanism.

You also should sketch the subassemblies as you take them apart so you can refer to them later as needed. Figure 7.10 shows a sketch of the housing subassembly; Figure 7.11 shows the platform and spring mechanism subassemblies; and Figure 7.12 shows the sketch of the four-bar linkage for the kitchen scale. It may not seem important to document each small step of dissection on a simple product; however, on a more complicated mechanism, it is essential to document each step for reassembly at a later time.

FIGURE 7.09. A fishbone diagram for the dissection of the kitchen scale.



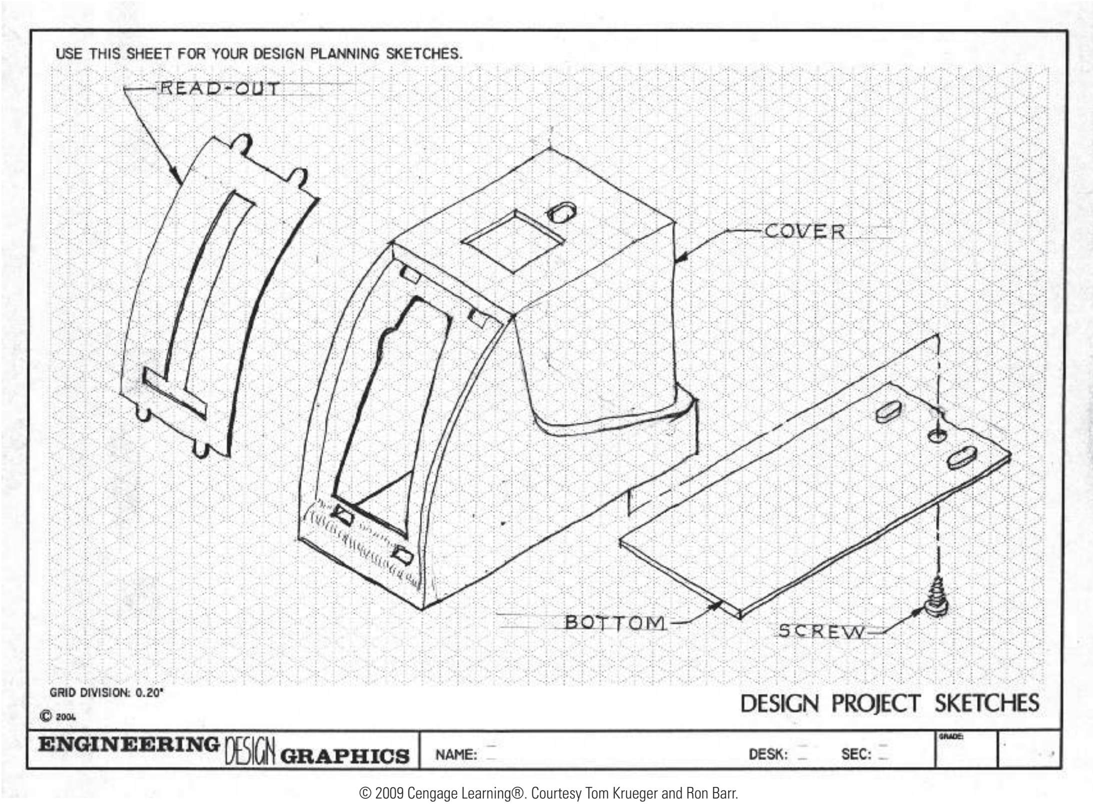


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FIGURE 7.10. A sketch of a housing subassembly.

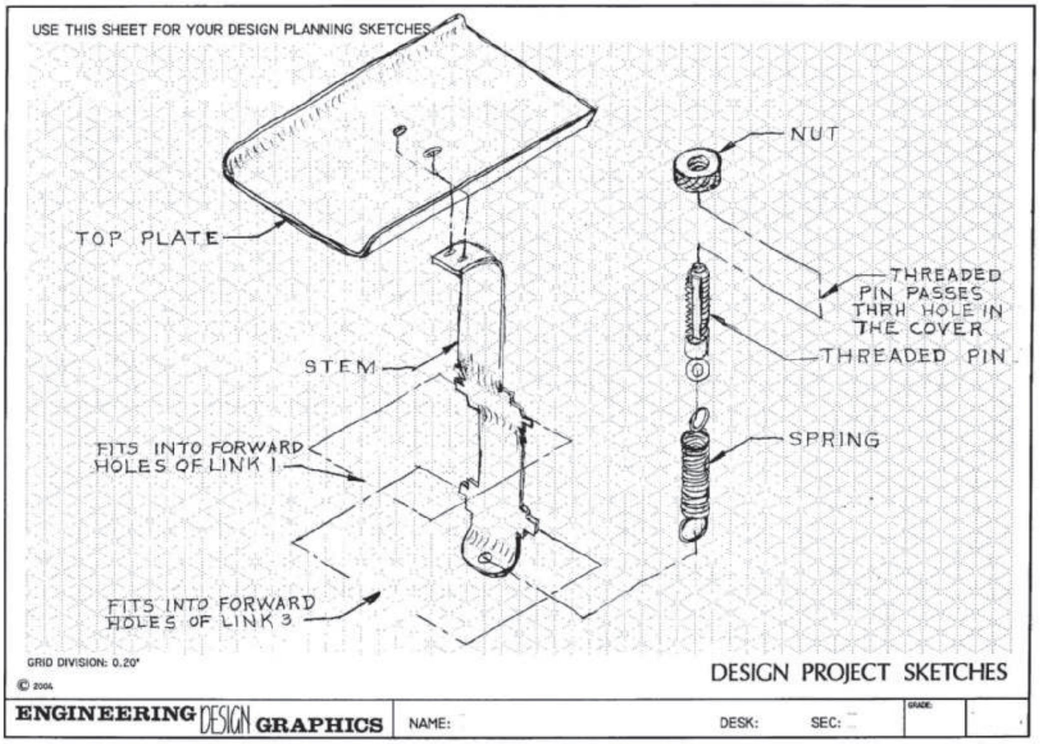

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FIGURE 7.11. A sketch of platform and spring subassemblies.

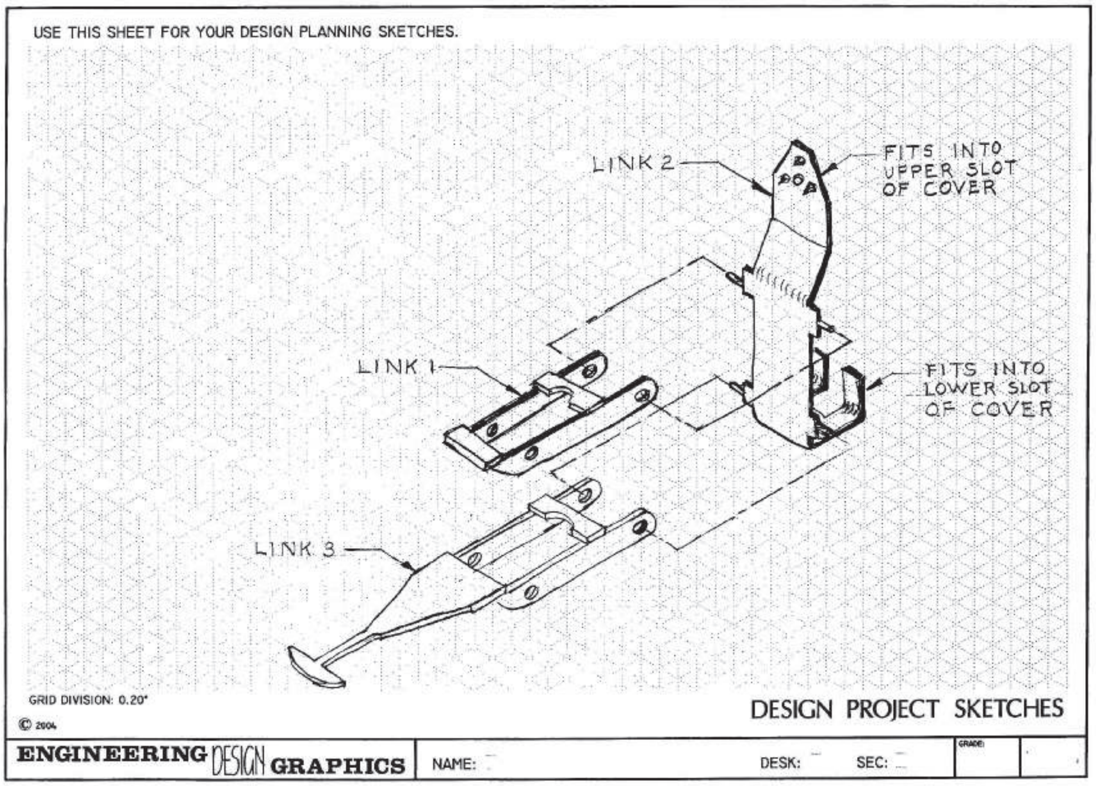

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FIGURE 7.12. A sketch of a four-bar linkage subassembly.

7.04.03 Obtaining Part Sizes

Through the use of metrology equipment, the individual parts are now studied for size, shape, and position data; this is usually done with simple handheld devices such as a set of scales or calipers. You can use more accurate and sophisticated measurement systems such as a coordinate measuring machine if you have one available or if greater accuracy is needed.

Sometimes this dimensional data can be written down in a set of notes or in an inspection report. Making sketches of the part and applying dimensions directly to the sketch also can be used to document the part. Note that the assembly sketches you made in the previous step may not be suitable for this part of the exercise, since they are essentially assembly pictorials. You may need to make new sketches of individual parts for this step.

7.04.04 Developing a 3-D CAD Model

Some believe that the most important phase of the reverse engineering process is to obtain an accurate 3-D computer model of the devices being dissected. You have learned about 3-D solid modeling and assembly modeling in previous chapters of this text. To create accurate 3-D solid models, you may need to refine the CAD data once the preliminary model is completed, adjusting the data as needed. If you used a coordinate measuring machine for the part measurements, the computer system might be able to download the data automatically to a CAD file and create the parts for you. If not, you will need to build the 3-D CAD parts and systems individually using the available software commands. Figure 7.13 shows a CAD model of the kitchen scale. In this figure, the front plate of the model is not shown, exposing the interior of the assembly.

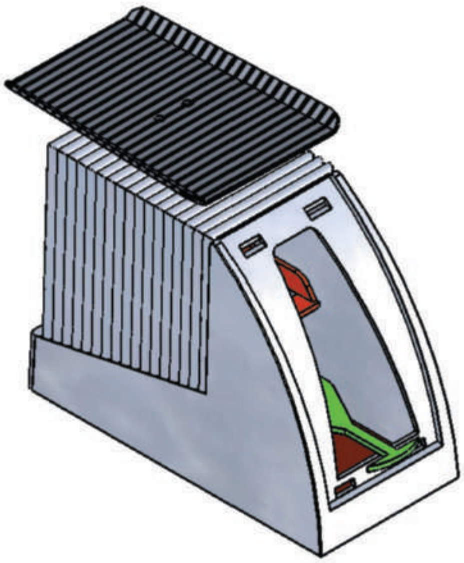

FIGURE 7.13. A computer model of the kitchen scale assembly.

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Once you have created each of the 3-D CAD models, you can generate engineering drawings from them. These drawings will contain dimensions and could be used by a machinist to create the individual parts. Figure 7.14 shows an engineering drawing of the kitchen scale body that was created from its 3-D CAD model. You will learn more about engineering drawings in later chapters of this text.

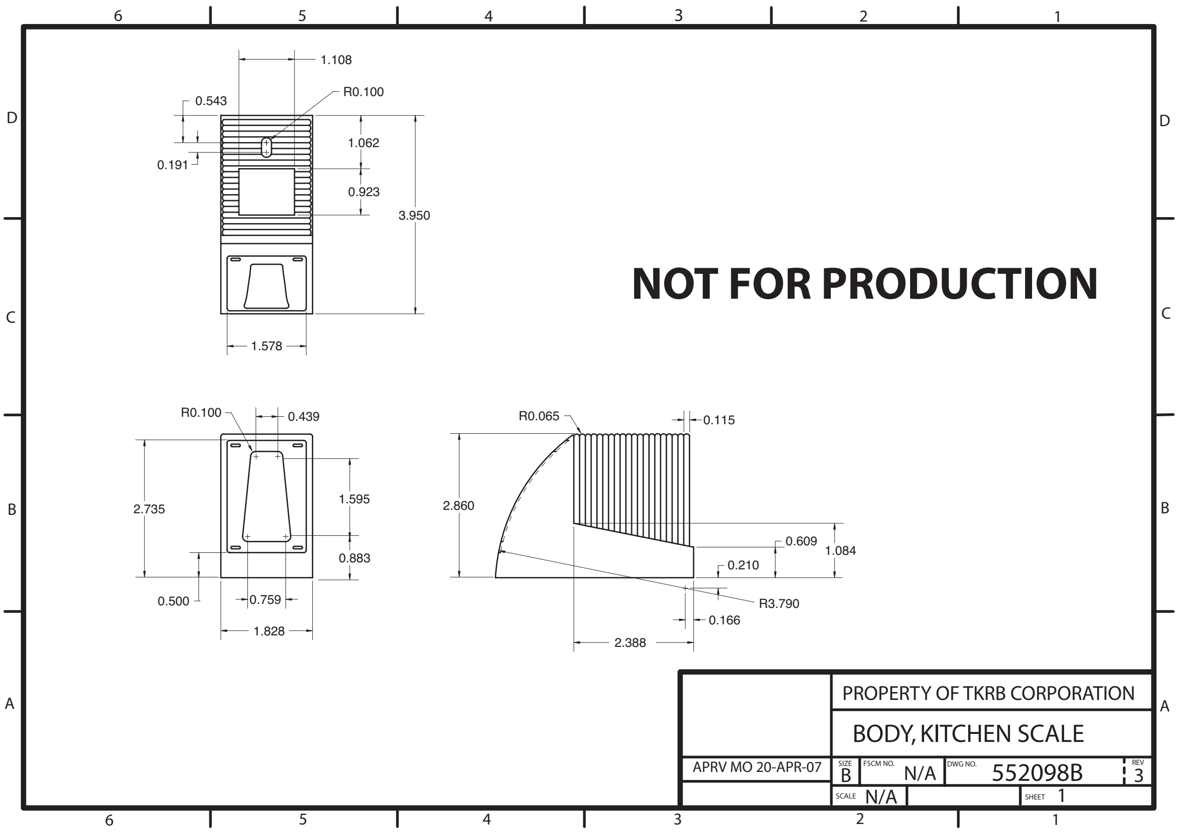

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FIGURE 7.14. An engineering drawing of the kitchen scale body.

7.04.05 Considering Potential Redesign

Since one purpose of the reverse engineering process is to study an existing design and suggest improvements to it, the final task of the dissection project should be to consider potential modifications to the product. During your thorough study of the design, some weaknesses may have become apparent. For example, the hole on the top of the body of the kitchen scale may be too large, causing the stem and weighing plate to be unstable. For the model studied here, the weighing plate and the stem are made of stamped metal and have relatively sharp edges. A different process of manufacturing or a different selection of materials might be desirable. The materials used in the spring may be prone to creep, meaning that the spring "stretches out" slightly but permanently with repeated use and thus will not accurately weigh food over the long term. You also may decide that the capacity of the scale should be increased from 0.5 kg to 1 kg. Some of these modifications would require that the entire system be redesigned; other modifications would merely require a different specification for materials. In any case, if you are asked to suggest design modifications for the product you dissected, you should outline them in a memo or report, including new drawings or 3-D CAD models as needed to illustrate your new product.

7.05 Geometric Properties Analysis

In the preceding sections of this chapter, you learned about one type of analysis—reverse engineering. Although 3-D CAD played a role in the reverse engineering process, solid modeling was not an absolute necessity in performing reverse engineering. You could have stopped with the creation of sketches and drawings of the physical parts in the device and not created 3-D CAD models. Your analysis of the device would have been complete at that time. For the remainder of this chapter, you will learn about two types of analysis for which 3-D CAD modeling is absolutely essential. For these analysis activities, the *basis* for the analysis is a 3-D solid model. In other words, in order to perform the analysis, you must first create the solid model and then proceed from there.

In some instances, you may want to compute the physical properties of your 3-D models. For example, you may want to minimize an object's volume or weight. Or you may need to know where a system's **center-of-mass** is so you can increase the device's stability. If you check the physical properties of the object as you make changes to it, you should be able to optimize your design through an iterative process. The types of physical properties that are typically calculated from a 3-D part definition include the **surface area**, **volume**, **mass**, **density**, and center-of-mass. Distances and dimensions on the parts also can be measured or computed. In addition, inertial properties (such as **radii-of-gyration**, **moments-of-inertia**, principal-axes of rotation, and products-of-inertia) also can be calculated from the models. You learn more about these types of properties in courses such as Statics and Dynamics; but for now, you merely need to understand that these are calculated from the geometry, mass, and mass distribution of the part.

The physical properties of a part can be computed about any set of axes—the global, the local, or some other set of user-defined axes. Properties can be input by material type for computational purposes. For example, in specifying that a given part is composed of steel, the software will automatically use the standard density of steel in its calculation of physical properties.

7.05.01 Measurement Analysis

With measurement analysis, you can determine the dimensions and other geometric parameters of a CAD model. Most modern 3-D modeling software can display the dimensions of the object one way or the other, especially the dimensions you input to create the model. In addition, the software can compute dimensions that are not shown. For example, most software has a Measure function that can be applied to a 2-D sketch or to a 3-D solid model. When using this type of command in the software, you query the solid model on-screen and the software returns the value you desire.

- *Measure Point:* This command returns the coordinates (x-, y-, and z-) of a specific point on a sketch or model. Usually, this point has to be a pickable point, such as a corner or circle center. You could, however, interpolate between two pickable points to return the coordinates of a point that is halfway between two defined points. Figure 7.15a illustrates the Measure Point command.
- *Measure Line Length:* This measure command returns the length of a line, as shown in Figure 7.15b. The line can but does not have to be a specific edge on the model. In other words, the software can be used to determine the distance between two points on the model regardless of whether a true edge exists between the points.
- *Measure Line Distance:* This measure command returns the shortest (perpendicular) distance between a line and another identified entity, such as a point. The Measure Line Distance command is illustrated in Figure 7.15c.
- *Measure Circle:* This measure command returns the center and diameter of a circle, such as a hole diameter. Figure 7.15d illustrates the use of the Measure Circle command.
- *Measure Arc:* The Measure Arc command returns the center and radius of an arc, as shown in Figure 7.15e.
- *Measure Surface:* Measure Surface returns the area of a specified surface and the length of the perimeter surrounding that surface. This command is illustrated in Figure 7.15f.

7.05.02 Mass Properties Analysis

Mass properties are the static properties of a solid body. Mass properties depend on two things: the geometry of the part and its density, where density is defined as the mass per unit volume. Table 7.01 includes the fundamental mass properties that can typically be computed with 3-D CAD software. Your computer system may include other properties; however, the properties provided here are the "basics." For now, you may not understand all of the terminology found in the table; you will likely learn about some of these mass properties in more advanced-level engineering and science courses.

The mass properties of an object are related to one another through various formulas. For example, the density is the mass divided by the volume. Likewise, the radius-of-gyration about a given axis is derived directly from the object's moment-of-inertia about that same axis. For most CAD systems, densities of standard materials are internally stored and are available for assignment to the parts you create. So for an assembly model, you could assign some parts to be made from steel, others from aluminum, and still others from plastic. The software would automatically insert the correct density value in its calculations. After computing the volume of the object, the software would simply multiply the volume by the density and return the object's mass for your further calculations.

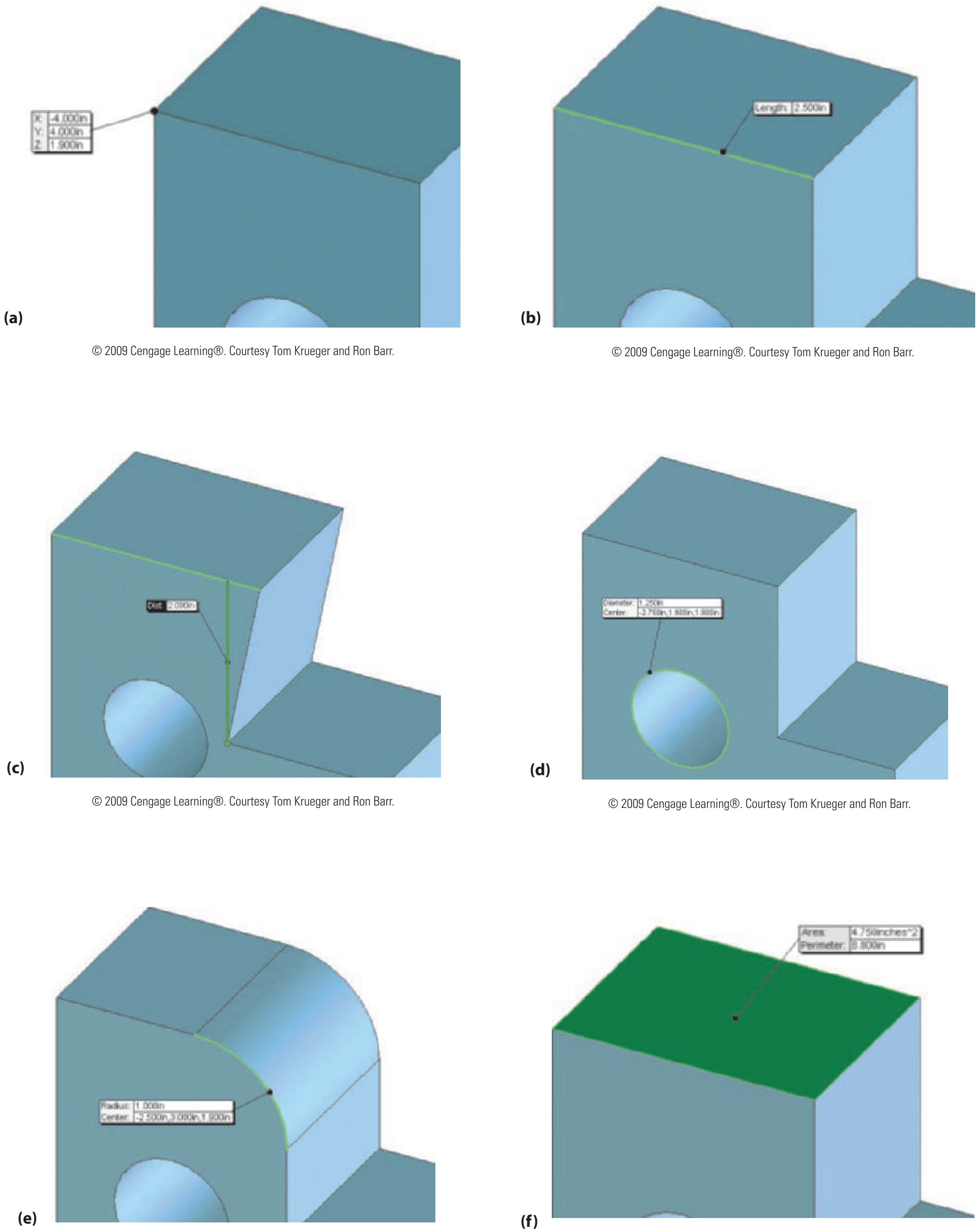

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FIGURE 7.15. Various measure commands. (a) point, (b) line length, (c) line distance, (d) circle, (e) arc, (f) surface.

Table 7.01.

MASS PROPERTIES OF A SOLID BODY.	
PROPERTY	**DEFINITION**
DENSITY	DENSITY IS THE WEIGHT PER UNIT VOLUME FOR THE MATERIAL FROM WHICH THE PART IS MADE.
MASS	THE MASS OF A BODY IS THE MEASURE OF THE BODY'S PROPERTY TO RESIST CHANGE IN ITS STEADY MOTION. THE MASS DEPENDS ON THE VOLUME OF THE BODY AND THE DENSITY OF THE MATERIAL FROM WHICH THE BODY IS MADE.
VOLUME	THE VOLUME OF A BODY IS THE TOTAL VOLUME OF SPACE ENCLOSED BY THE BODY'S BOUNDARY SURFACES.
SURFACE AREA	THE SURFACE AREA IS THE TOTAL AREA OF THE BOUNDARY SURFACES DEFINING THE SOLID.
CENTER-OF-MASS OR CENTROID	CENTER-OF-MASS (OR CENTROID) OF A VOLUME IS THE ORIGIN OF COORDINATE AXES FOR WHICH FIRST MOMENTS ARE ZERO. IT IS CONSIDERED THE CENTER OF A VOLUME. FOR A PARALLEL GRAVITY FIELD, THE CENTER OF GRAVITY COINCIDES WITH THE CENTROID.
PRINCIPAL AXES-OF-INERTIA AND PRINCIPAL MOMENTS-OF-INERTIA	PRINCIPAL MOMENTS-OF-INERTIA ARE EXTREME (MAXIMAL, MINIMAL) MOMENTS OF INERTIA FOR A BODY. THEY ARE ASSOCIATED WITH PRINCIPAL AXES-OF-INERTIA THAT HAVE THEIR ORIGIN AT THE CENTROID, AND THE DIRECTION OF EACH IS USUALLY GIVEN BY THE THREE UNIT VECTOR COMPONENTS.
MOMENTS-OF-INERTIA	A MOMENT-OF-INERTIA IS THE SECOND MOMENT OF MASS OF A BODY RELATIVE TO AN AXIS, USUALLY X-, Y-, OR Z-. IT IS A MEASURE OF THE BODY'S PROPERTY TO RESIST CHANGE IN ITS STEADY ROTATION ABOUT THE AXIS. IT DEPENDS ON THE BODY'S MASS AND ITS DISTRIBUTION AROUND THE AXIS.
RADIUS-OF-GYRATION	THE RADIUS-OF-GYRATION IS THE DISTANCE FROM THE AXIS OF INTEREST WHERE ALL OF THE MASS CAN BE CONCENTRATED WHILE STILL YIELDING THE SAME MOMENT-OF-INERTIA.

Many engineering calculations require data on mass properties of the component parts of the mechanical system for further analysis. The data required might include the mass, the centroid, moments-of-inertia, products-of-inertia, or radii-of-gyration. The moments-of-inertia, products-of-inertia, or radii-of-gyration may be calculated for any set of axes of rotation. The most commonly used sets of axes for these calculations are either the Cartesian axes (standard x-, y-, or z-axes) or the centroidal axes, with the centroidal axes being the preferred set for this type of calculation. The origin of the centroidal axes is located at the centroid of the object and is the "mass center" of the object. For simple objects such as a sphere, the centroid coincides with the exact center of the sphere. For an object such as a cube, the centroid would be at a point in the cube's interior that is halfway between all six surfaces that make up the cube. For more complex objects, the software can compute the exact centroid (and the various inertial properties) for you.

As stated previously, you will learn about most of these mass properties in other engineering courses; but consider just one example now. To spin a wheel about an axis, you have to apply a moment or torque to it. The level of resistance of the wheel to rotate faster and faster under the action of the moment is governed by its moment-of-inertia. If the moment-of-inertia is large, the wheel will require a higher torque to accelerate to a desired rotation rate; conversely, if the moment-of-inertia is small, the wheel will easily accelerate to the same rotation rate with a smaller torque applied to it. In essence, the moment-of-inertia is the weight of the mass multiplied by a distance the mass is away from the rotation axis. Thus, a wheel with a large diameter will be harder to accelerate to a desired rotation rate than a wheel with a small diameter.

Moments-of-inertia are easily computed for standard shapes by simple formulas; however, most objects are not made up of simple shapes and the computer is an effective tool for computing moments-of-inertia for complex shapes. The computer software determines an object's moment-of-inertia by dividing the body into small pieces. The software then multiplies the mass of each piece by the distance the small piece is away from the axis of interest. Adding up all of the individual moments-of-inertia, you obtain the moment-of-inertia for the entire body. An example of this procedure is shown in Figure 7.16. In this figure, a solid model of a cam wheel is shown in 7.16a and the wheel has been subdivided into small pieces in Figure 7.16b. The moment-of-inertia of each small piece (one of which is shown in Figure 7.16c) is calculated by multiplying the mass of the piece by the distance from the axis; in this case, ρx. Summing all of the individual moments-of-inertia from each piece produces the overall moment-of-inertia for the cam wheel. You should realize that trying to calculate by hand the moment-of-inertia about X for the cam wheel would be a tedious, painstaking, and error-prone procedure. With the use of computer software tools, quantities such as the moment-of-inertia can be calculated easily with a few mouse clicks, provided you already created the solid model of the part.

To see how an engineer might use a property such as a moment-of-inertia in the design process, consider the two flutter plates shown in Figures 7.17 and 7.18 (A and B). The mass properties for design A are given in Table 7.02, and the mass properties of design B are given in Table 7.03.

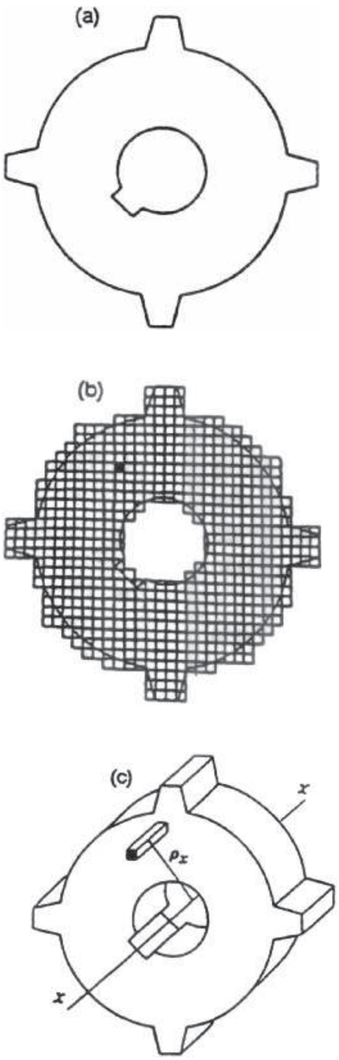

FIGURE 7.16. Numerical computation of the moment-of-inertia.

Flutter Plate A

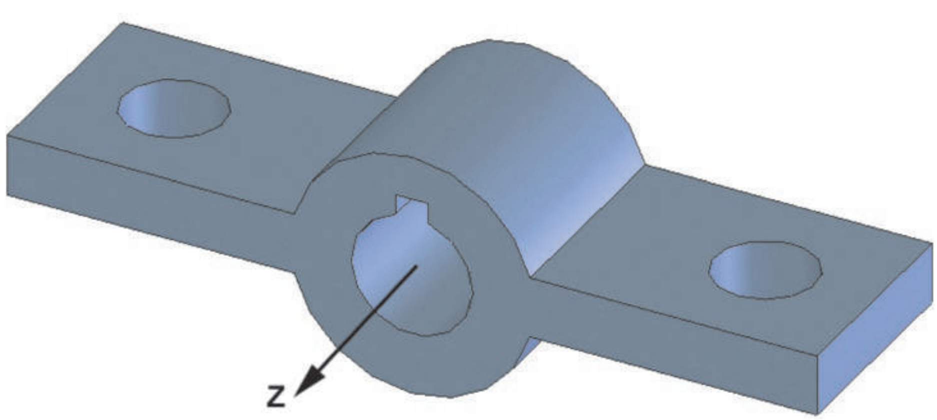

FIGURE 7.17. Flutter plate design A.

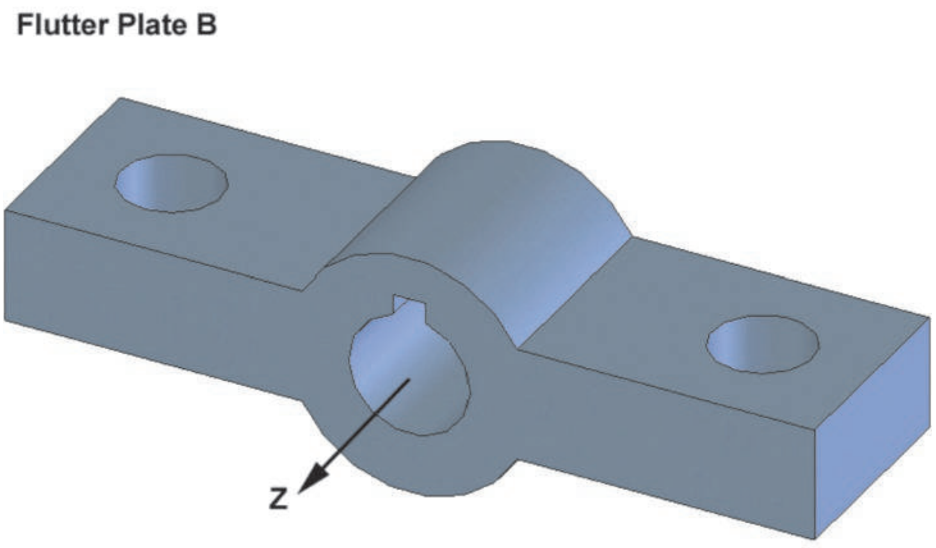

FIGURE 7.18. Flutter plate design B.

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Table 7.02. MASS PROPERTIES OF FLUTTER PLATE A.

DENSITY = 0.10 POUND PER CUBIC INCH

MASS = 2.58 POUNDS

VOLUME = 26.46 CUBIC INCHES

SURFACE AREA = 99.72 SQUARE INCHES

CENTER-OF-MASS: (INCHES)

X = 0.00

Y = –0.01

Z = 0.00

PRINCIPAL AXES-OF-INERTIA AND PRINCIPAL MOMENTS-OF-INERTIA: (POUNDS × SQUARE INCHES) TAKEN AT THE CENTER-OF-MASS

IX = (1.00, 0.00, 0.00)	PX = 2.68
IY = (0.00, 0.00, –1.00)	PY = 15.62
IZ = (0.00, 1.00, 0.00)	PZ = 16.26

MOMENTS-OF-INERTIA: (POUNDS × SQUARE INCHES) TAKEN AT THE CENTER-OF-MASS AND ALIGNED WITH THE OUTPUT COORDINATE SYSTEM

LXX = 2.68	LXY = 0.00	LXZ = 0.00
LYX = 0.00	LYY = 16.26	LYZ = 0.00
LZX = 0.00	LZY = 0.00	LZZ = 15.62

MOMENTS-OF-INERTIA: (POUNDS × SQUARE INCHES) TAKEN AT THE OUTPUT COORDINATE SYSTEM

IXX = 2.68	IXY = 0.00	IXZ = 0.00
IYX = 0.00	IYY = 16.26	IYZ = 0.00
IZX = 0.00	IZY = 0.00	IZZ = 15.62

Table 7.03. MASS PROPERTIES OF FLUTTER PLATE B.

DENSITY = 0.10 POUND PER CUBIC INCH		
MASS = 3.66 POUNDS		
VOLUME = 37.51 CUBIC INCHES		
SURFACE AREA = 115.29 SQUARE INCHES		
CENTER-OF-MASS: (INCHES)		
X = 0.00		
Y = −0.01		
Z = 0.00		
PRINCIPAL AXES-OF-INERTIA AND PRINCIPAL MOMENTS-OF-INERTIA: (POUNDS × SQUARE INCHES) TAKEN AT THE CENTER-OF-MASS		
IX = (1.00, 0.00, 0.00)	PX = 3.74	
IY = (0.00, 0.00, −1.00)	PY = 27.81	
IZ = (0.00, 1.00, 0.00)	PZ = 28.86	
MOMENTS-OF-INERTIA: (POUNDS × SQUARE INCHES) TAKEN AT THE CENTER-OF-MASS AND ALIGNED WITH THE OUTPUT COORDINATE SYSTEM		
LXX = 3.74	LXY = 0.00	LXZ = 0.00
LYX = 0.00	LYY = 28.86	LYZ = 0.00
LZX = 0.00	LZY = 0.00	LZZ = 27.81
MOMENTS-OF-INERTIA: (POUNDS × SQUARE INCHES) TAKEN AT THE OUTPUT COORDINATE SYSTEM		
IXX = 3.74	IXY = 0.00	IXZ = 0.00
IYX = 0.00	IYY = 28.86	IYZ = 0.00
IZX = 0.00	IZY = 0.00	IZZ = 27.81

The primary motion of the flutter plate is rotation about the given *z*-axis. In this case, the thickness of the arm plates is the key difference in the two designs, with a larger thickness used for design B. A check of the moment-of-inertia about the *z*-axis (IZZ) for design A has a value of 15.62 lbs/in^2, and the moment-of-inertia about the *z*-axis for design B has a value of 27.81 lbs/in^2. This means that design B would require more torque to accelerate the flutter plate to a desired rotation rate, making A the better choice if required torque were the only design consideration. In some cases, however, a strength requirement for the flutter plate arms might supersede the torque requirement; and design B, with its thicker and thus stronger arms, might be the better choice overall.

Mass properties analysis offers fast insight into the performance of a design concept without requiring significant additional effort. Once you have created the CAD model, you can obtain the mass properties with a few simple mouse clicks. Hence, computing mass properties is an initial design analysis tool that allows you to identify possibilities that can be ruled out or possibilities that require further exploration. More advanced analysis tools, such as the finite element method described in the next section of this chapter, require more sophisticated input parameters and take more time to complete, but they also yield a better in-depth insight into the mechanical behavior of the designed object.

7.06 Finite Element Analysis

Finite element analysis (FEA) is an advanced design analysis technique that has been made possible through the development of sophisticated 3-D CAD solid modeling tools. In engineering practice, there are some cases where desired quantities can be calculated by simple mathematical expressions. For example, for a simple circular rod that is pulled on either end, the stress in the member can be computed as the pulling force applied to the end of the shaft divided by the cross-sectional area of the member. Armed with knowledge of the physical properties of the rod's material, an engineer can calculate whether the stress applied to the rod is too large, resulting in permanent deformation. Design changes can then be made to reduce the stress in the rod if desired. One simple method for reducing the stress is to increase the cross-sectional area of the rod.

Unfortunately, most real-life objects are not simple circular rods and the determination of stresses within these objects is a complicated process. Thus, determining whether an object will be permanently deformed through applied loads would be a nearly impossible proposition without the use of tools such as FEA. For complex objects, engineers can still model their behavior mathematically under various loading conditions; however, for these objects, the equations are virtually impossible to solve using simple mathematics. In these cases, the behavior of the object is governed by a set of differential equations for which no simple solution exists. Similar to the way the computer solved for the moment-of-inertia of a complex object, the FEA method is founded on dividing the part into small pieces, which are together called a **mesh**. The governing differential equation can be easily solved for each of these individual pieces, called elements. The results are then compiled by the computer across all of the elements to produce an accurate picture of the stresses throughout the object.

7.06.01 Classes of FEA Problems

In essence, FEA is in the category of engineering analysis tools that apply computational approximations to classic field problems. By dividing an object into small elements, solving for stresses within the individual elements, and then compiling the results across the entire object, you have obtained an approximate solution to the problem. Generally speaking, this approximate solution is "close enough." In fact, since an exact solution is virtually impossible to achieve, the FEA approximation is far better than no solution at all. The types of field problems for which the FEA method is a viable analysis technique are presented in the following paragraphs.

Mechanical stress and displacement fields. Objects that are subjected to loads experience **mechanical stresses**. On a microscopic level, these stresses tend to pull molecules apart or push them closer together. A shear stress, for example, tries to slide molecules apart in a motion similar to when you slide your hands against each other. The stresses experienced by an object will result in slight changes in shape as the molecules move relative to one another. If you are pulling on an object, it will tend to elongate or stretch. If you are pushing on the object, it will compress or squash together. If you are applying a shear stress to the object, it will shear and become distorted. The FEA method can be used to solve for all of these types of stresses and deformations for objects subjected to various types of loads.

Figure 7.19 shows the stresses experienced by a simple cantilever beam that has been loaded by applying a downward force along its top surface. In this case, the lower stresses are found at the end of the beam where the load is applied, with higher stresses found at the end of the beam where it is attached to the wall.

Fluid flow and pressure fields. Another application of the finite element method is in the computation of fluid flow and pressure fields, and one of the primary uses is in analyzing airflow. (In engineering analysis, air is considered to be a fluid.) This type of analysis is used to compute the airflow around airplane wings, enabling engineers to design bigger and better jets that can develop sufficient lift with bigger payloads. Air

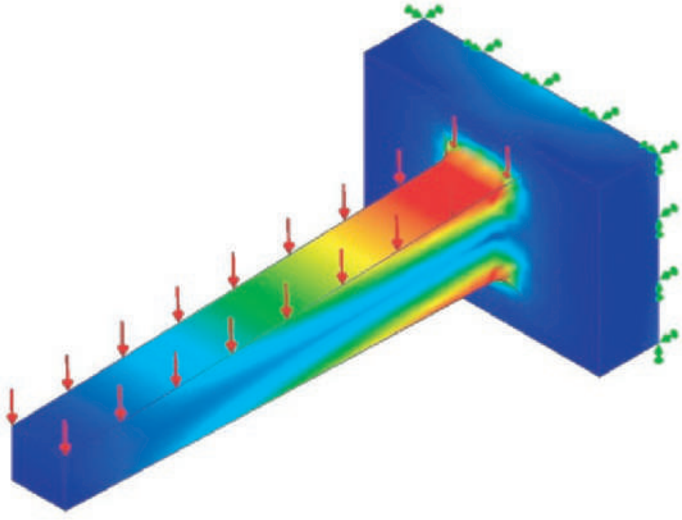

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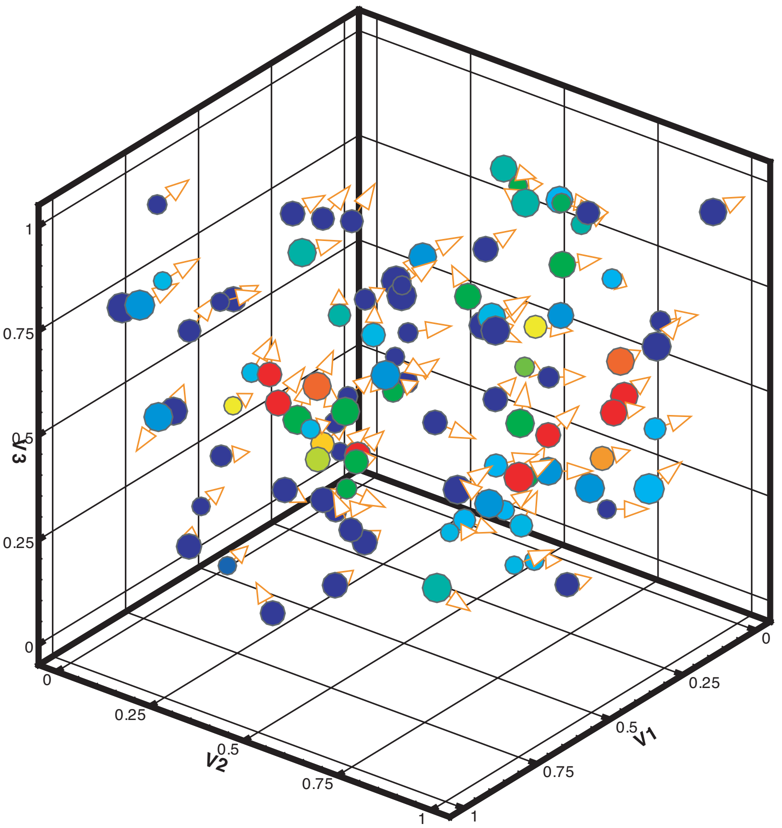
FIGURE 7.19. Stress analysis of a cantilever beam.

pressure fields also can be determined through this type of analysis. Figure 7.20 shows the movement of particles suspended in a moving fluid. The particles, for example, could be blood cells or nano-particles in a blood vessel. By predicting and altering the way the fluid flows, or altering the geometry of the particles, a desired motion of the particles can be produced.

Thermal flow and temperature fields. Another class of problems that can be solved using the finite element method is the computation of thermal flow and temperature fields. For a satellite system, one side of the system is typically exposed to the sun



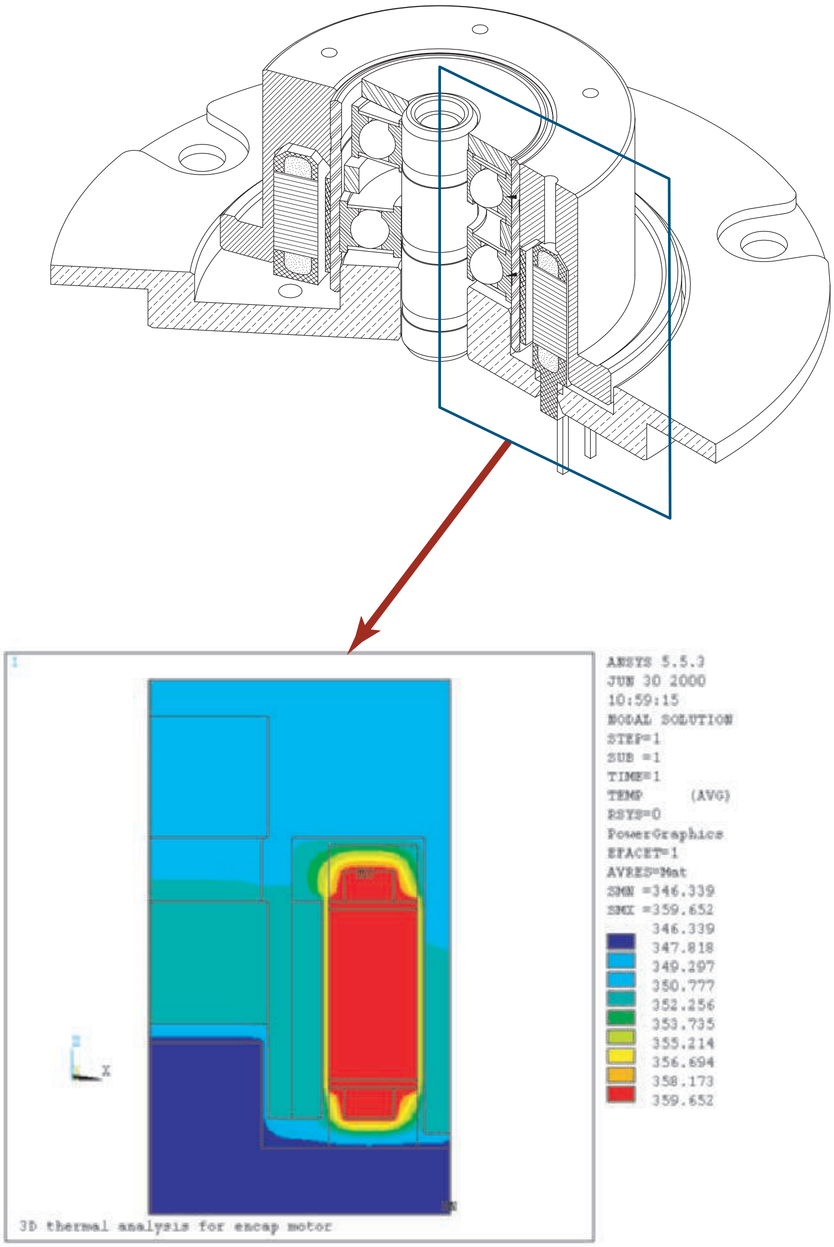
Courtesy of Tarek Zohdi, University of California.

FIGURE 7.20. Particle motion is a moving flow field.

and the other is not. As you can imagine, the side exposed to the sun is much warmer than the other side. It is important to determine how the temperature flows from one side to the other, because temperature may affect the performance of the satellite. For example, large variations in temperature can cause thermal stresses, because the warmer side expands and the colder side contracts. If the stresses are too large, permanent deformation or even fracture can result. Figure 7.21 shows a spindle motor for a disk drive that has been analyzed for its temperature distribution.

Electromagnetic fields. When electricity flows through wire, an electromagnetic field is created. Electromagnetic fields are also a significant consideration in the design of antennas. The size and shape of the electromagnetic field depends on the type of antenna used. If the steel in the body of the automobile interferes with the electromagnetic field from the antenna, radio reception will be poor and owners will

FIGURE 7.21. Thermal analysis of the inside of a disk drive spindle motor.



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not be satisfied with their cars. Due to poor electromagnetic performance, the simple vertical mast antenna has been virtually eliminated in new car models—new antennas are more sophisticated and are less likely to have poor performance issues.

The finite element method can be used to solve for electromagnetic fields so that antennas for automobiles can be optimized. Figure 7.22 shows the output from a finite element model that was used to determine an electromagnetic field for a linear step-motor.

7.06.02 Finite Element Meshes

The key to successful numerical solution of engineering field problems is the division of the solid volume into small, finite-sized elements. The process of subdividing the solid into elements is referred to as meshing. In general, the smaller the elements, the more accurate the solution; however, if the elements are too small, you may introduce error into your model due to computational approximations or round-off error. Further, smaller element sizes mean more elements and computation time will significantly increase. Element sizes should be reduced wherever there is curvature in the model to compute desired quantities accurately in those regions. In general, you can use larger elements in areas where stresses (or temperatures, etc.) are relatively static and you need small elements in areas where the stresses are rapidly changing. Knowing what sizes to use for elements will take practice.

Fortunately, most of the tedium in meshing has been eliminated or greatly reduced with modern FEA software applications. In most cases, you merely specify a nominal element size and the software will automatically create the elements for you; the software also will automatically reduce the element sizes when it encounters curved regions or changes in geometry. As you gain experience with the finite element method, you will get a better feel for reasonable nominal element sizes; but for now, you should rely on your instructor to provide insight into the modeling procedure. Figure 7.23 shows a simple part for which a finite element mesh has been created.

7.06.03 Finite Element Boundary Conditions

In the FEA meshing scheme, the finite elements are connected through their edges and nodal points. It is at these nodal points and along the element edges that the **boundary conditions** are specified. In the case of stress analysis, boundary conditions

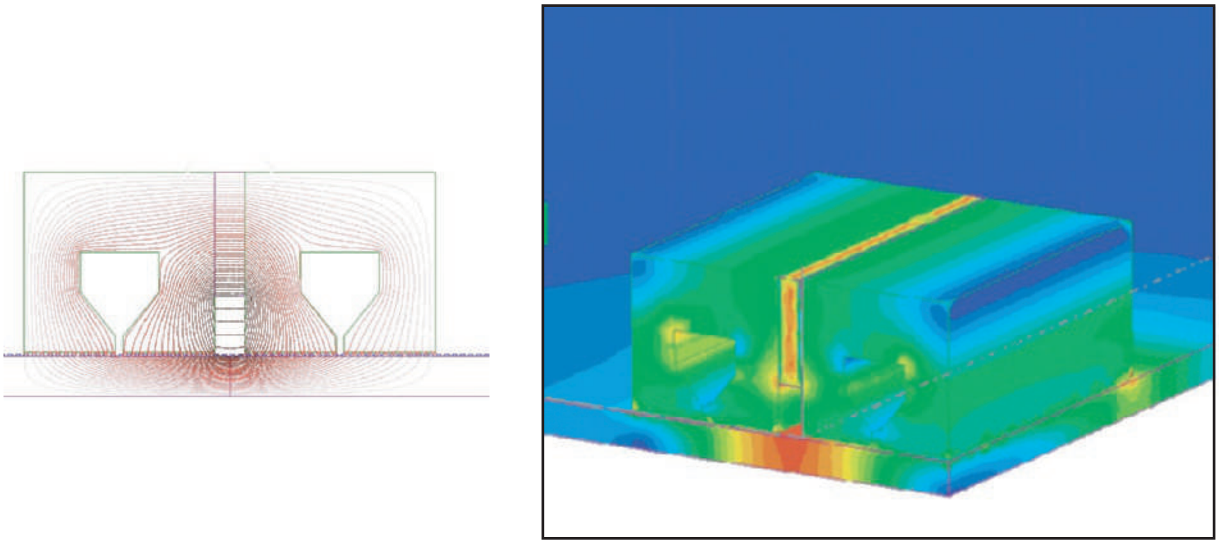


FIGURE 7.22. Electromagnetic analysis of a linear step-motor.

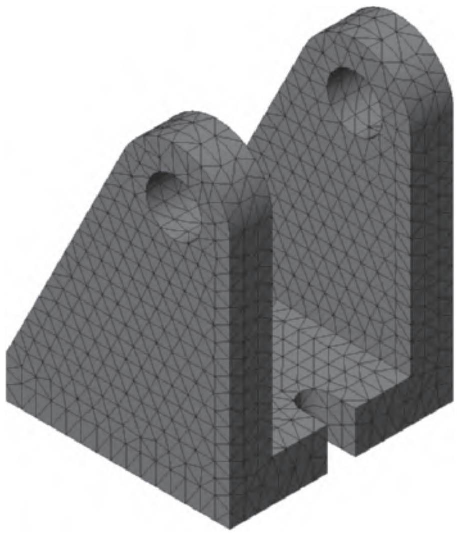


FIGURE 7.23. A finite element mesh applied to a 3-D CAD model.

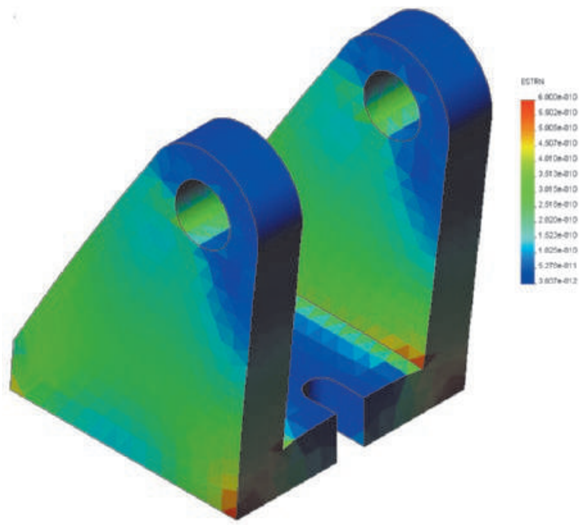


FIGURE 7.24. Stress distribution contours obtained through FEA.

might be forces or defined displacements; for thermal analysis, boundary conditions might be input as a temperature. In the case of objects subjected to external forces for stress analysis, you must fix some of the nodal points in space so the object is not able to move at those points.

7.06.04 Finite Element Output

During the analysis, each finite element is treated as a simple solid body with limited properties, interacting with its surrounding region only through the nodal points and element edges. Although this approximation seems very rough, proper selection of the element properties and sizes will ensure an accurate solution to the problem. When the analysis results are displayed, deformations are usually given to an enlarged scale and colored contours are used to show the stress distribution, as shown in Figure 7.24. Thermal gradients and fluid pressure fields also can be shown as colored contours on the output plots.

7.07 Chapter Summary

In this chapter, you learned about the importance of analysis in the engineering design process. You learned that many of the analysis methods are founded in mathematical and scientific principles. You learned that reverse engineering is a type of analysis where devices are dissected and the functions of the parts that make up the devices are studied. The dissection process is documented through notes and sketches so the device can be reassembled at a later time. Parts in the device are measured, and computer models are created and assembled. You also learned about mass properties analysis as a technique for computing various quantities from part geometry. Computing mass properties provides a quick look at the part geometry and is an analysis technique that helps rule out possible designs or identify potential solutions that are worth pursuing in the initial stages of the design process. Finally, you learned about the FEA technique and the way it is used to solve complex engineering problems. You learned about the classes of problems that can be solved through FEA, including stress analysis, fluid flow, thermal flow, and electromagnetic fields.

7.08 GLOSSARY OF KEY TERMS

analysis: The study of the behavior of a physical system under certain imposed conditions.

black box diagram: A diagram that shows the major inputs and outputs from a system.

boundary conditions: The constraints and loads added to the boundaries of a finite element model.

caliper: A handheld device used to measure objects with a fair degree of accuracy.

center-of-mass (centroid): The origin of the coordinate axes for which the first moments are zero.

coordinate measuring machine: A computer-based tool used to digitize object geometry for direct input to a 3-D CAD system.

density: The mass per unit volume for a given material.

engineering scale: A device used to make measurements in much the same way a ruler is used.

finite element analysis (FEA): An advanced computer-based design analysis technique that involves subdividing an object into several small elements to determine stresses, displacements, pressure fields, thermal distributions, or electromagnetic fields.

fishbone diagram: A diagram that shows the various subsystems in a device and the parts that make up each subsystem.

laser scanning (three-dimensional): A process where cameras and lasers are used to digitize an object based on the principle of triangulation.

mass: A property of an object's ability to resist a change in acceleration.

mechanical dissection: The process of taking apart a device to determine the function of each part.

mechanical stress: Developed force applied per unit area that tries to deform an object.

mesh: The series of elements and nodal points on a finite element model.

metrology: The practice of measuring parts.

moment-of-inertia: The measure of an object's ability to resist rotational acceleration about an axis.

radius-of-gyration: The distance from an axis where all of the mass can be concentrated and still produce the same moment-of-inertia.

reverse engineering: A systematic methodology for analyzing the design of an existing device.

surface area: The total area of the surfaces that bound an object.

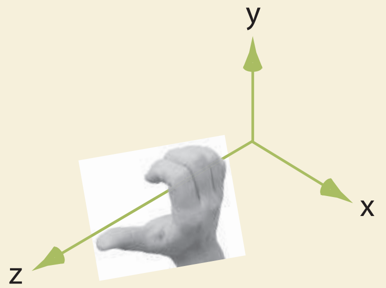
volume: The quantity of space enclosed within an object's boundary surfaces.

7.09 QUESTIONS FOR REVIEW

1. How does reverse engineering differ from industrial espionage?
2. Describe the following metrology tools: scales, calipers, coordinate measuring machine, and 3-D laser scanner. Which is least expensive? Which is most accurate?
3. How are volume, density, and mass for an object related to one another?
4. What are some of the mass properties that can be computed for a CAD model?
5. What is a moment-of-inertia? Is it easier to spin a wheel with a large diameter or a small diameter?
6. What are constraints in an FEA?
7. What kinds of boundary conditions can be applied to a finite element model?
8. Describe a finite element mesh. In general, will accuracy improve as element size gets smaller or as it gets larger?
9. What is one disadvantage in using a small element size for a finite element model?

7.10 PROBLEMS

1. For the objects shown in Figure P7.1, the given coordinate system is to be placed at the lower rear corner of each object. Create a solid model for each object and calculate its volume using the units shown. Relative to the given coordinate system, find the x-, y-, and z-coordinates of the centroid for each object.



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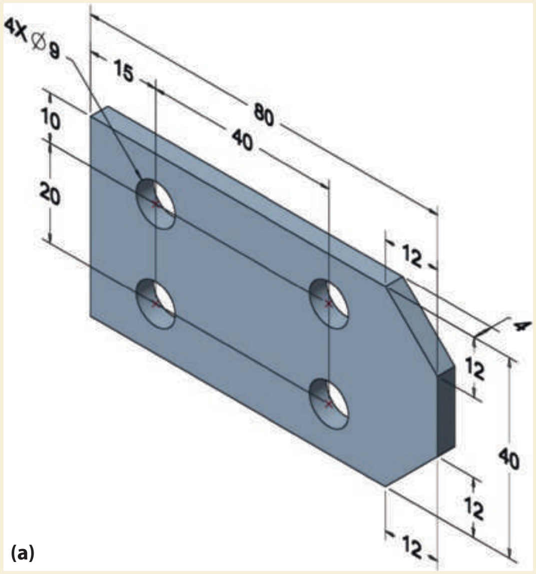
Coordinate system used from Problem 1.



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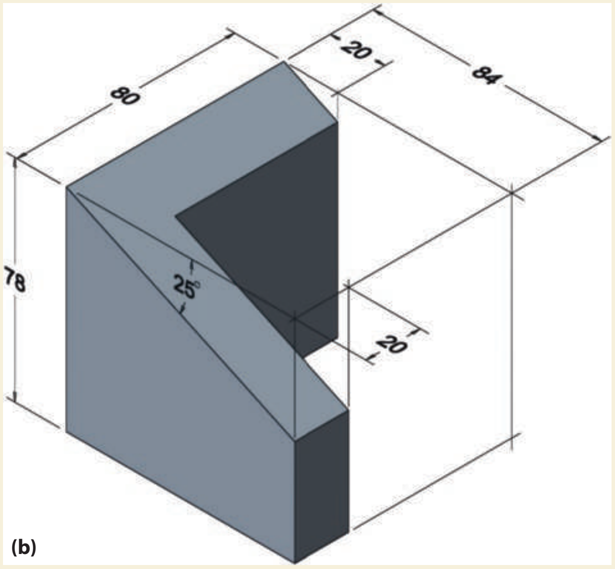

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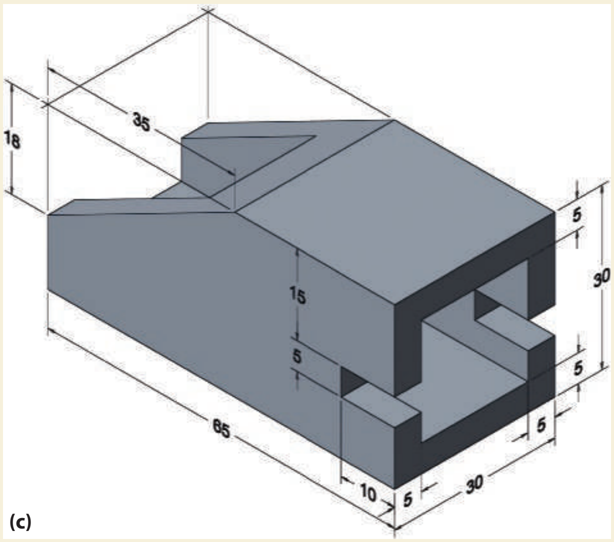

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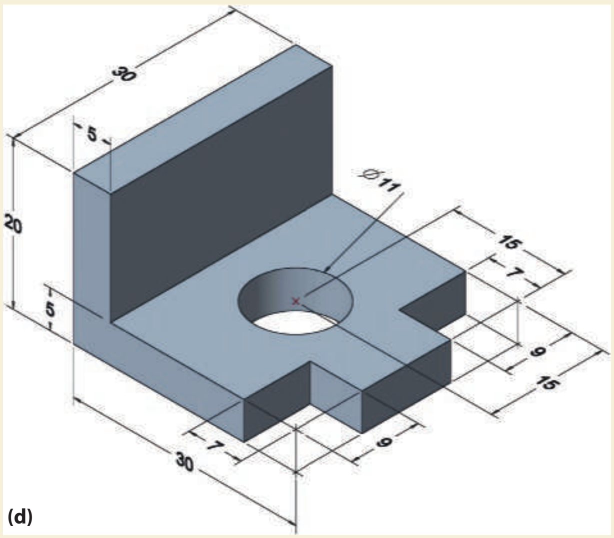

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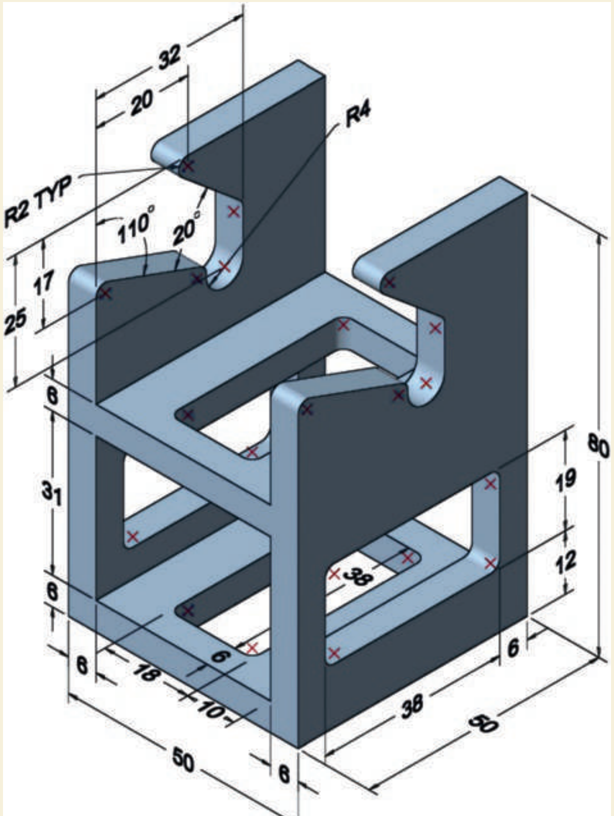

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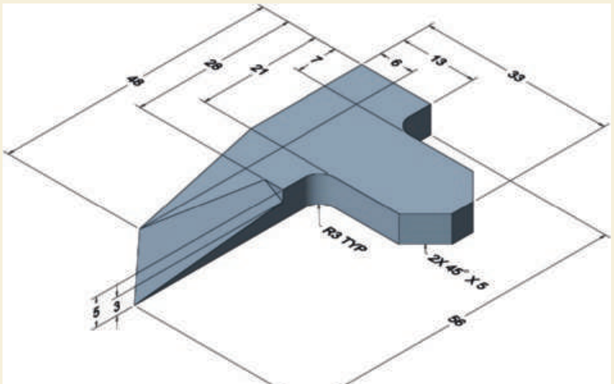

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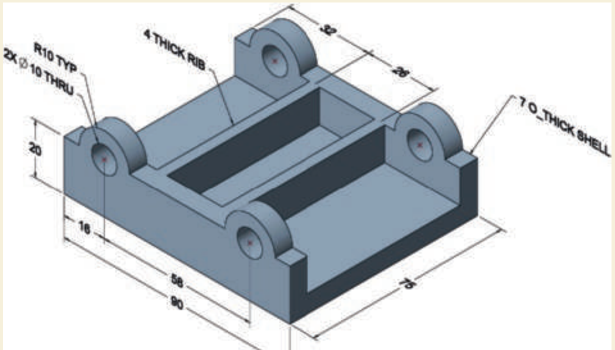

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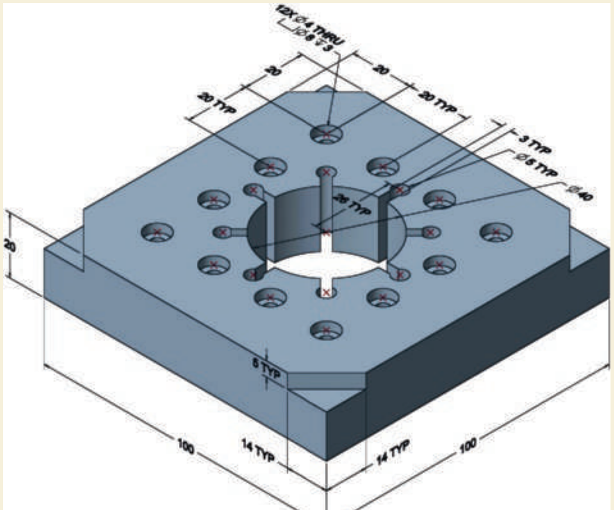

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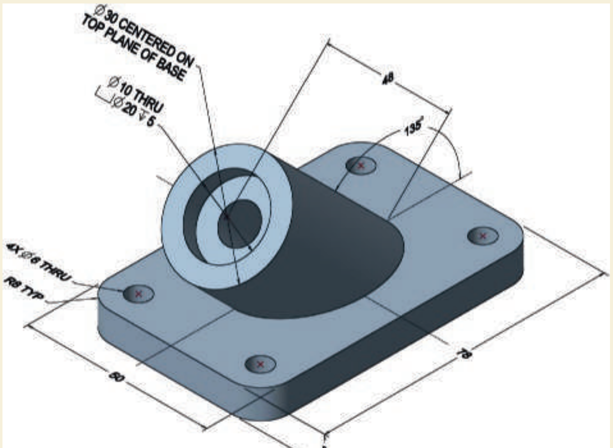

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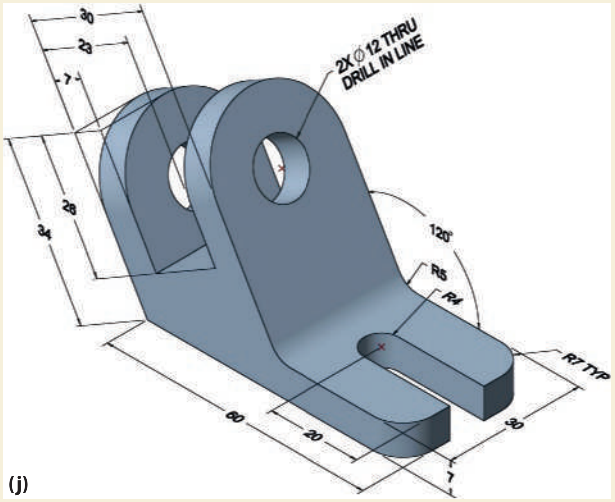

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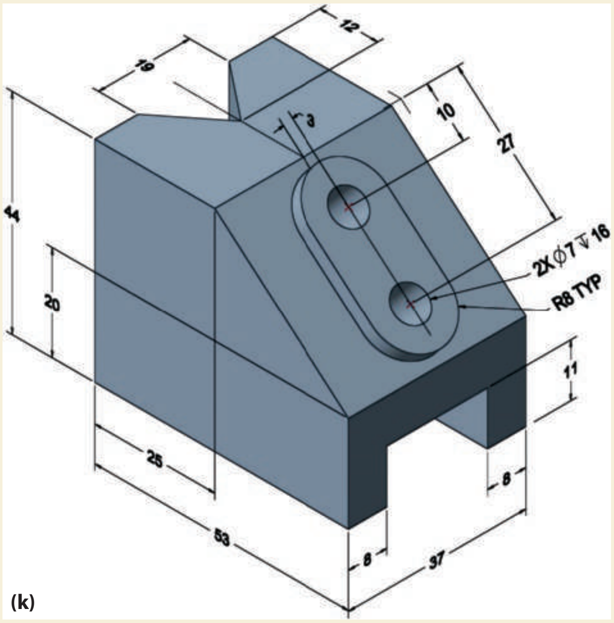

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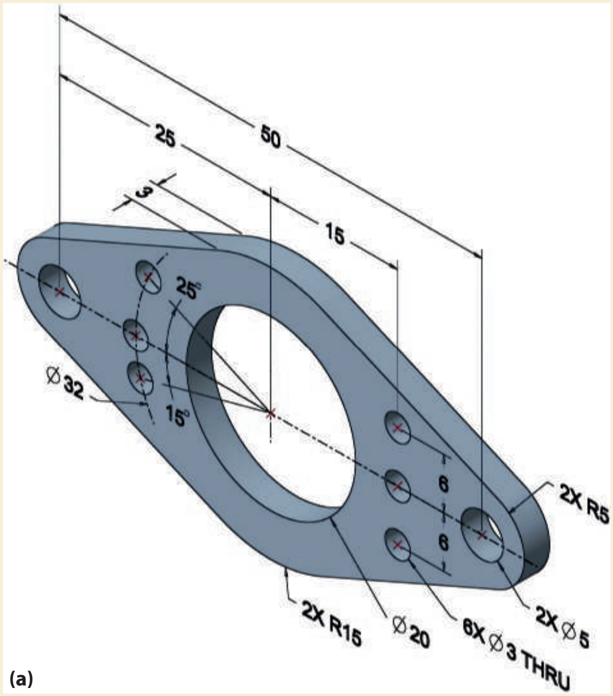
FIGURE P7.1.

2. For each object shown in Figure P7.2, create a solid model and calculate its mass and moment-of-inertia about the main symmetry axis. For some objects, a portion has been cut away to reveal interior detail. Assume first the object is made of steel, then aluminum.



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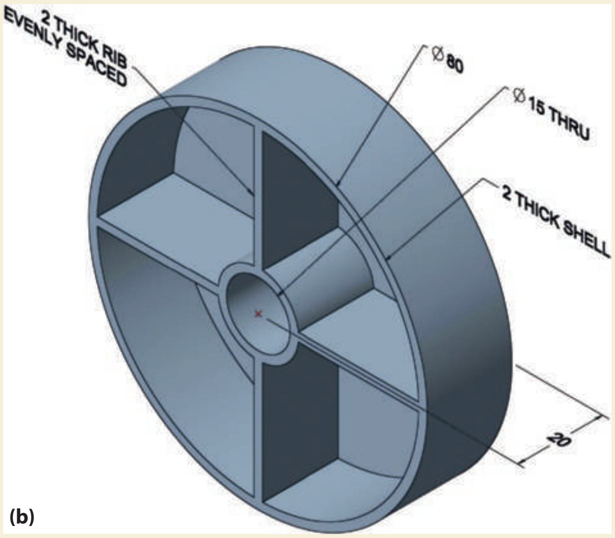

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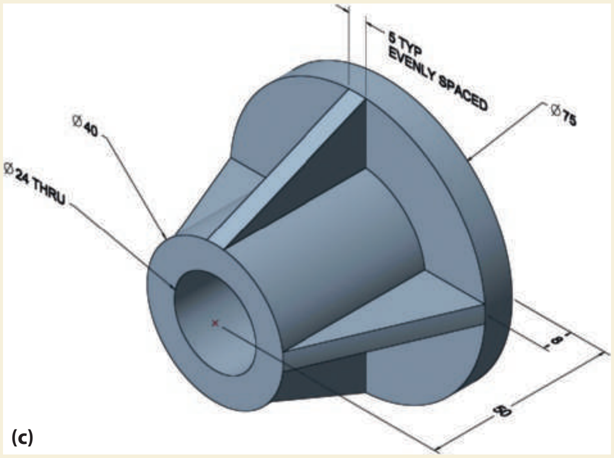

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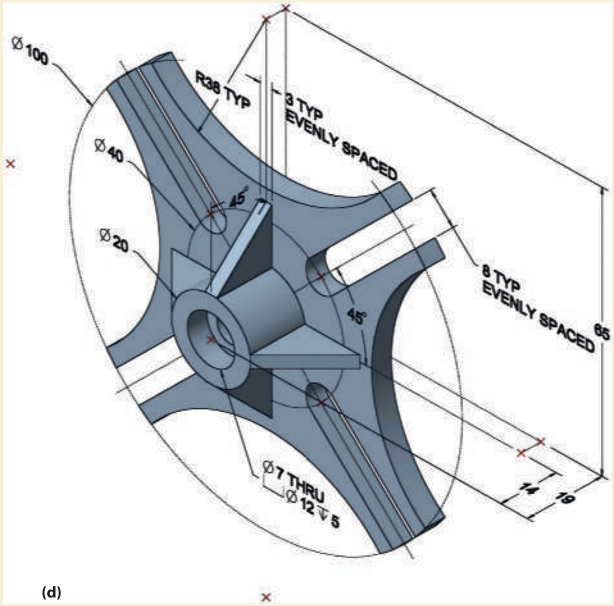

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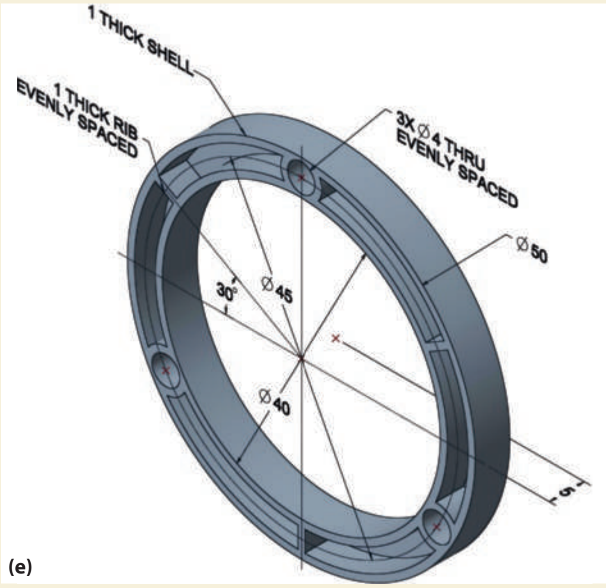

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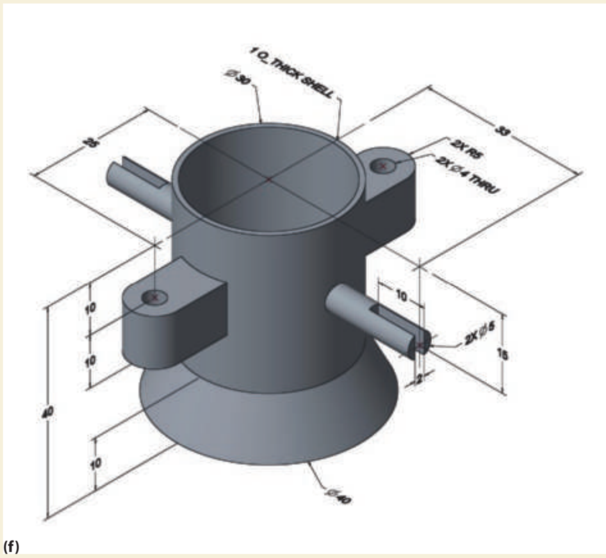

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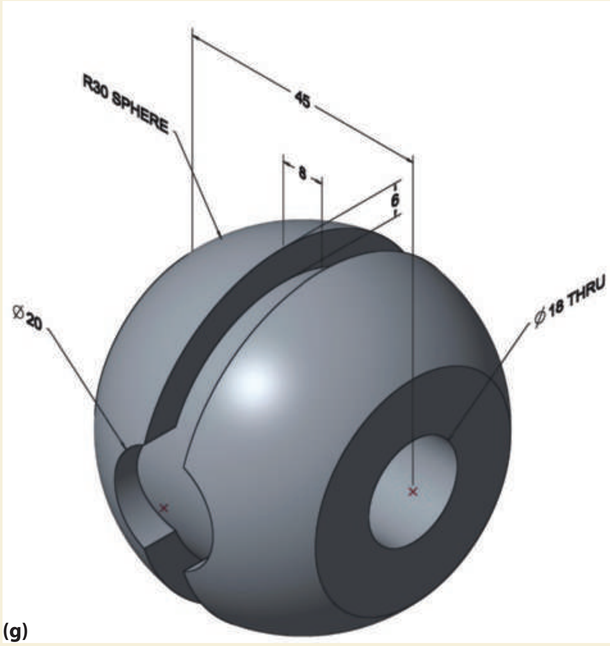
7.10 PROBLEMS (CONTINUED)



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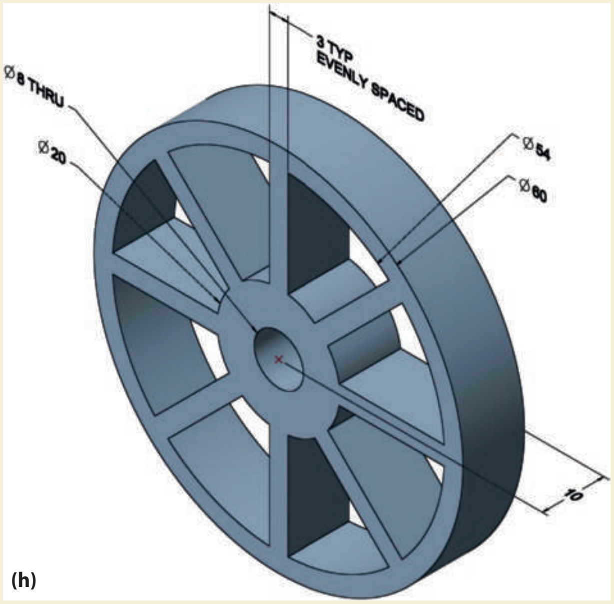

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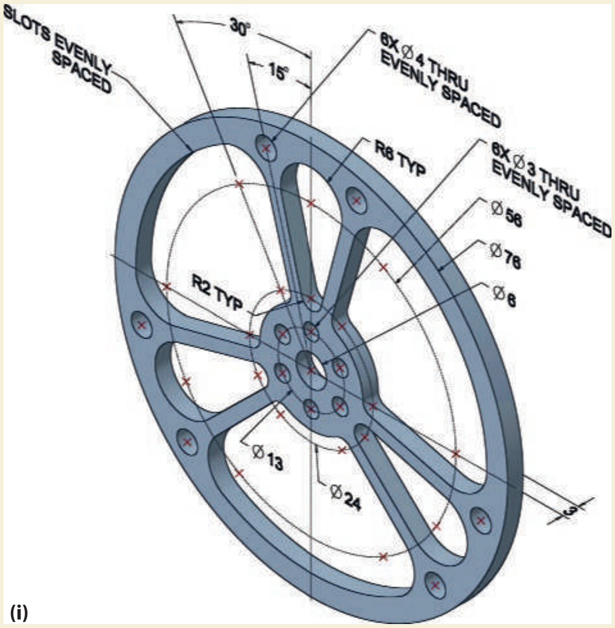

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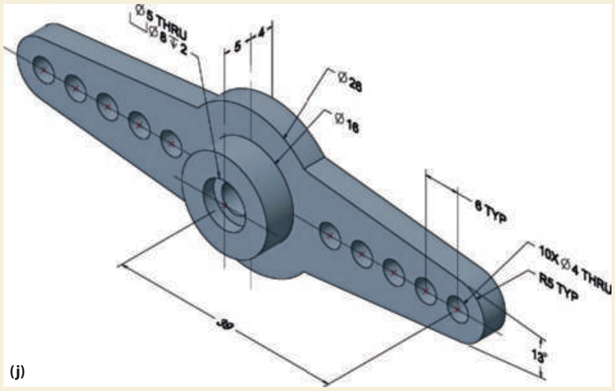

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FIGURE P7.2.

7.10 PROBLEMS (CONTINUED)

3. The object shown in Figure P7.3 is made of steel. The rectangular portion is secured to a vertical wall using four bolts in the 8 mm holes. A 9 mm steel pin is placed in the bore, and a downward force of 4000 N is exerted on the pin such that the force is evenly distributed inside the 9 mm bore. Create a solid model of the object, mesh it, and apply the proper boundary conditions for an FEA of the stress and deflection of the object under the given load. Calculate the magnitude and location of the maximum von Mises stress developed in the object. Create a color contour plot that shows the distribution of the stress on the part.

4. The object shown in Figure P7.4 is made of aluminum. The part is secured to the ceiling with bolts through the two 5 mm holes. A pinching force of 150 N is applied to the "wings" at the 4 mm hole. Create a solid model of the object, mesh it, and apply the proper boundary conditions for an FEA of the stress and deflection of the object under the given load. Calculate the magnitude and location of the maximum von Mises stress developed in the object. Does the 3 mm gap close under the applied force?

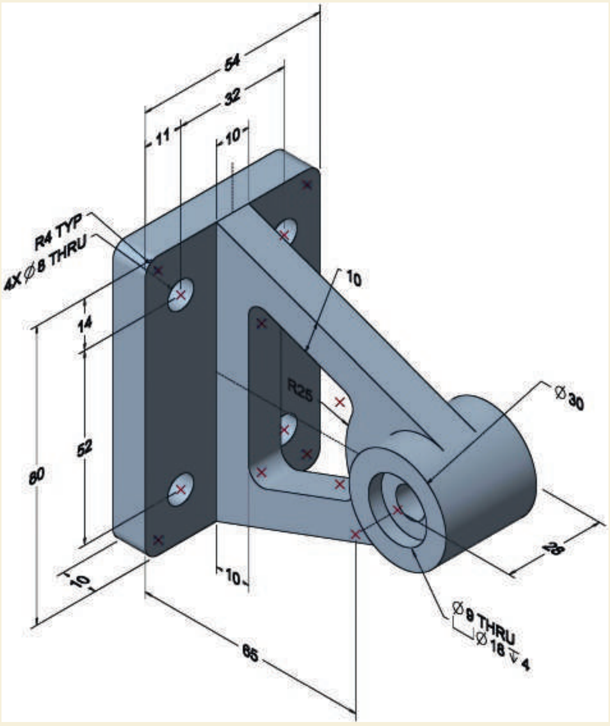

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FIGURE P7.3.

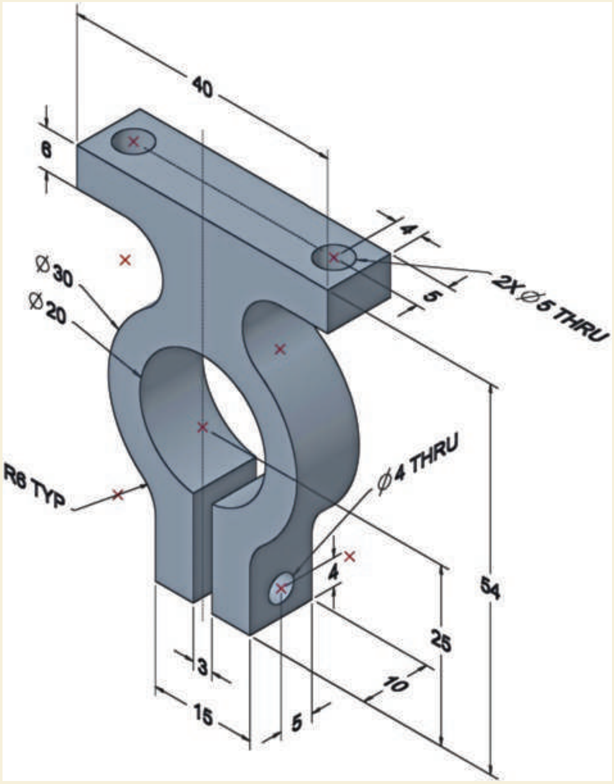

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FIGURE P7.4.

7.10 PROBLEMS (CONTINUED)

5. The object shown in Figure P7.5 is made of aluminum. The part is spun about its main symmetry axis at a rotation speed of 5000 RPM. Create a solid model of the object, mesh it, and apply the proper boundary conditions for an FEA of the stress and deflection of the object under the given load. Calculate the magnitude and location of the maximum von Mises stress developed in the object.

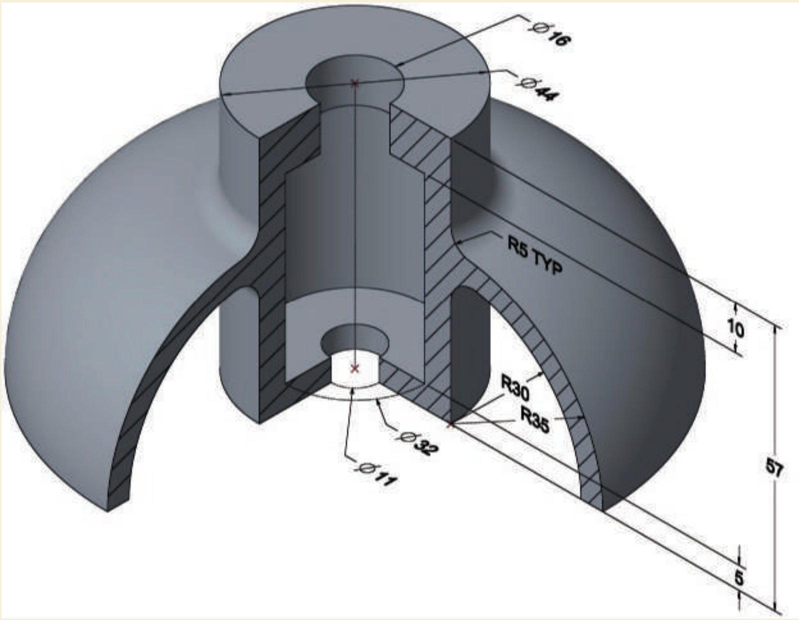

FIGURE P7.5.

SECTION THREE

SETTING UP AN ENGINEERING DRAWING

Almost all formal engineering drawings are presented in a multiview format using orthogonal projection. Most of the topics and techniques presented in this section are in wide use today and will continue to be invaluable in the foreseeable future. No matter how sophisticated the computer design hardware and software or the manufacturing system, engineers still must be able to create two-dimensional drawings of three-dimensional objects and be able to visualize three-dimensional objects from two-dimensional drawings.

CHAPTER 8

ORTHOGONAL PROJECTION AND MULTIVIEW REPRESENTATION

OBJECTIVES

After completing this chapter, you should be able to

- Discuss the principles of orthogonal projection
- Show how orthogonal projection is used to create multiple views of an object for formal engineering drawing
- Explain why orthogonal projection is necessary to represent objects in formal engineering drawing
- Create a multiview drawing from a 3-D object

8.01 INTRODUCTION

The best way to communicate the appearance of an object (short of showing the object itself) is to show its image. For the purposes of the object's fabrication, analysis, or record keeping, this image must be precise. A precise description of an object begins with an accurate graphical representation of that object, which is what a formal engineering drawing is all about. It is a series of images that show the object viewed from different angles, every view accurately depicting what that object would look like from each view.

Whether you originated a drawing or you received one from the originator, the images represented in any engineering drawing must be interpreted the same way. Consistency is achieved by adhering to nationally and internationally accepted methods for creating and interpreting the images. Pictorial images, such as the isometric drawings first presented in Chapter 2 (and detailed in Chapter 9), quickly convey large amounts of qualitative information. However, pictorial images have the disadvantage of distorting the true size, configuration, and geometry of their features.

For an object to be represented without distortion or ambiguity, enough views must be provided such that all of the object's features can be clearly seen and accurately measured. In an engineering drawing, the choice of views is not arbitrary. Also, the views are carefully chosen such that the features on the object are aligned between the views and the geometries of the features are shown without distortion. With these views, size specifications can be added later to complete the description of the object.

8.02 A More Precise Way to Communicate Your Ideas

You have a wonderful idea for a new device. You believe in your idea. You want to have it fabricated. However, you must communicate to another party your thoughts about what the parts in the device will look like when they are fabricated. The other party may be another engineer who subjects your device to a more detailed analysis of what it should look like. The other party may be a fabricator who makes the device to your exact specifications. The other parties may be located in another area of the country or in another country. With the international scope of business today, design, analyses, and fabrication are commonly done in different locations around the world.

If questions arise concerning your idea, you may not be around to answer them. That is why all other parties involved in fabricating the object must envision it exactly as you do. One of your goals as the engineer or designer of a product, device, or structure is to represent it graphically in such a way (i.e., accurately) that it can be fabricated without any party misinterpreting how you want it to appear.

During the development of the AeroTec riser, the engineers at Hoyt USA faced the possibility that the product's geometry would be misinterpreted due to insufficient representation of what it would look like after fabrication. Creating a graphical image of the object in the form of a sketch or drawing as seen from only a single direction was not a good idea. The riser, which is shown in Figure 8.01, contains many features, such as cutouts and protrusions that could remain hidden when viewed from only one direction. The object had to be viewed from multiple directions to ensure that all of its features were revealed. If you were the engineer responsible for the design and manufacturing of a similar product, what would you do? How would you communicate what you want built to those who build it? How would you ensure that different people interpret and build the product the same way every time?

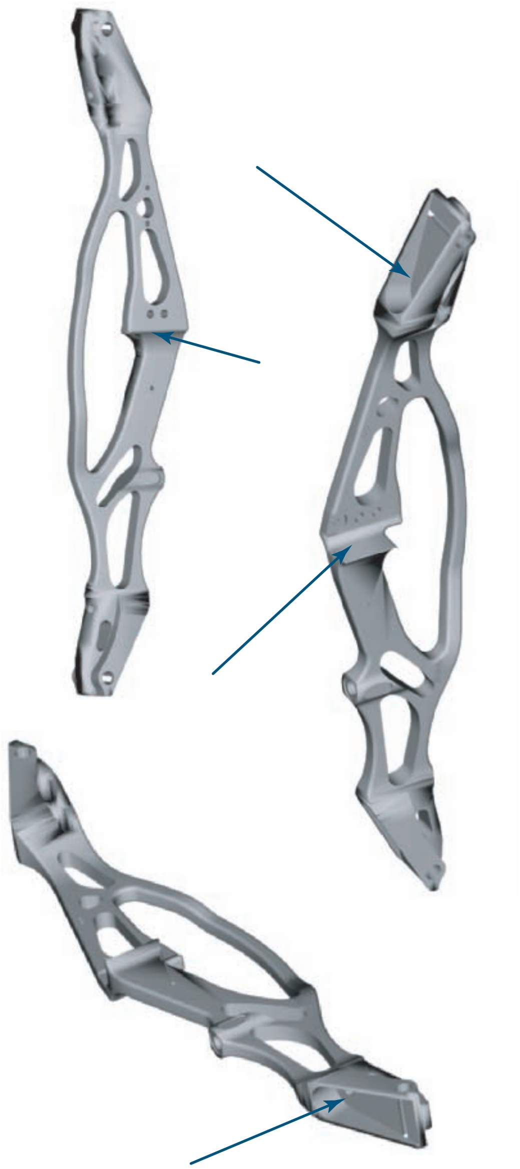

FIGURE 8.01. Viewing the AeroTec riser from different directions reveals previously hidden features. The arrows indicate details in each view that cannot be fully seen and described in the other views.

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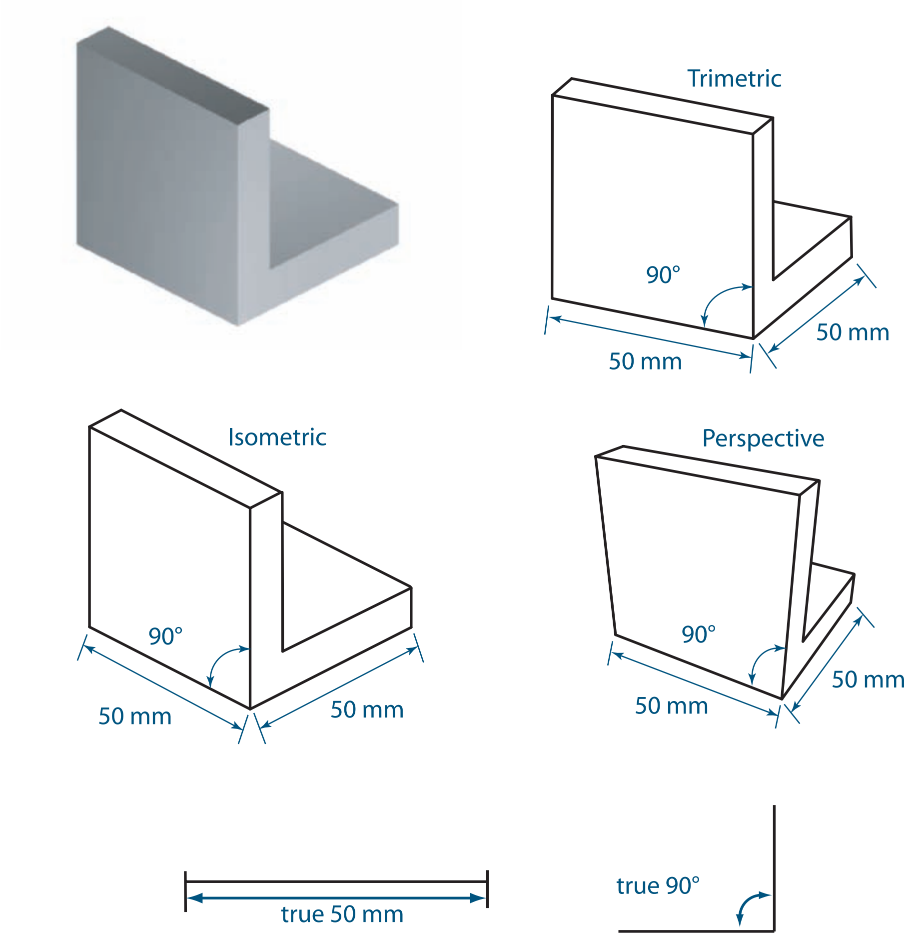
8.02.01 Problems with Pictorials

One solution would be to use pictorials such as an isometric or perspective view. These types of representations of an object offer the advantage of quickly conveying the object's 3-D aspects from one view. Even people who do not have a technical background can easily and quickly understand pictorials.

However, pictorial representations present problems that are inherent in the use of one view of an object's three dimensions. One problem is the distortion of angles, as shown in Figure 8.02. The use of right angles and perpendicularity between surfaces is common on many fabricated objects because surfaces having those relationships are easy to construct with machine tools. However, on pictorials, 90-degree angles do not appear as 90-degree angles. In fact, depending on the angle of viewing, a 90-degree angle can appear as more or less than 90 degrees. On a pictorial, it is difficult to depict an object's angles correctly when angles are not 90 degrees.

Another problem with pictorials is the distortion of true lengths. In any pictorial, a length of 1 m on an object, for example, is neither depicted nor clearly perceived as a 1 m length.

FIGURE 8.02. Distortion of true lengths and angles in pictorial presentations.



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In some cases, such as an object with only rectilinear edges seen in the isometric view, this length distortion is the same in every direction. In this case, the real length can be obtained by multiplying the distorted edge length by a single correction factor. In Figure 8.02, for example, the length of each edge of the object shown in the isometric would need to be the actual edge length multiplied by a scaling factor of 0.612 if the object were drawn its full size. The formula for getting this particular scaling factor is complicated, so do not worry about it for now. In general, however, the correction is not just a simple scaling factor. In a dimetric view, the correction factor for an edge of the object is dependent upon direction. The correction factor is more complicated for a trimetric representation, and the correction factor is even more complicated for a perspective representation.

Internal measurements also are distorted in pictorials. This distortion is dependent upon the direction of measurement, as shown in Figure 8.03. The location of the center of a hole placed at the center of a square face is, in reality, equidistant from each vertex of the square. However, in an isometric pictorial, the center of the hole must be drawn such that it is located a different distance from one vertex than from its adjacent vertices.

Figure 8.03 also shows the problem of curve distortion. The simplest curve—a circle or an arc of a circle—appears elliptical on a pictorial. On an isometric pictorial, the conversion from a circle to its representation as an ellipse is a matter of figuring out the scaling factors to calculate the major and minor axes and the orientation of the ellipse, both of which are dependent upon the circle's orientation in space. The calculation or construction is more complicated on a trimetric view because the scaling and orientation

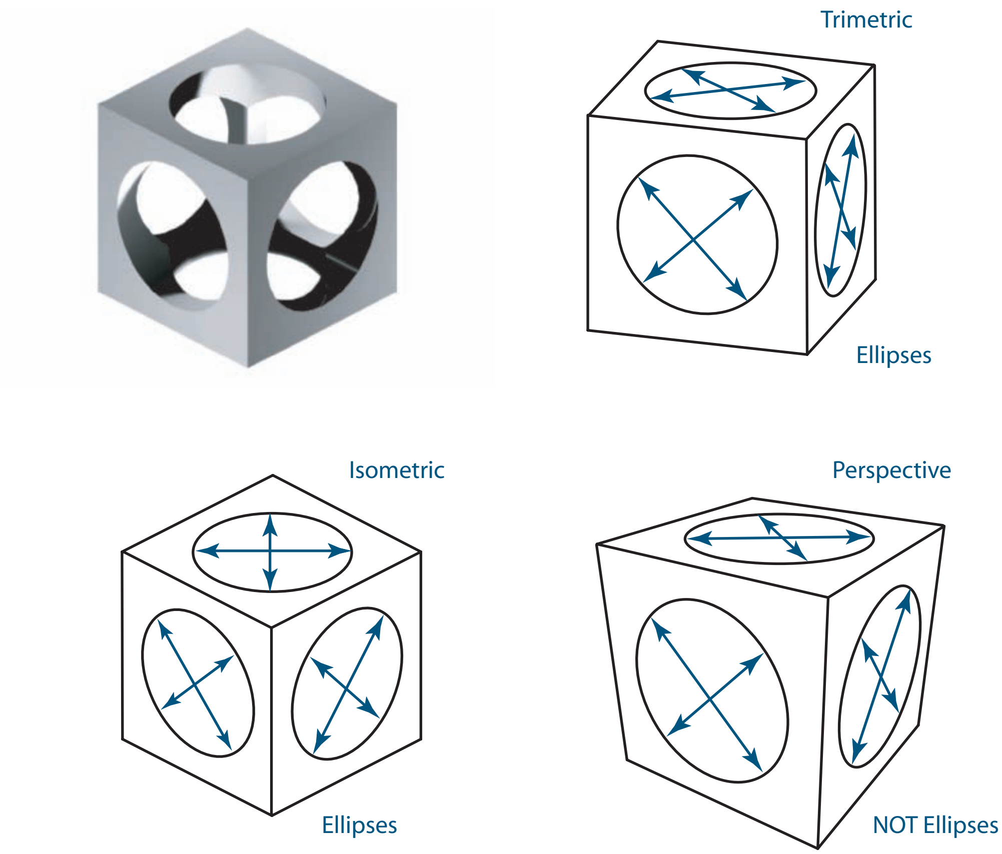

FIGURE 8.03. Distortion of internal lengths in pictorials. These different lengths on the same object represent the same length, which is the diameter of the holes in the cube.

factors are different for different plane orientations on the object. On a perspective representation, the construction is more complicated because the circle does not appear as an ellipse, but rather as an oval, or egg shape. (Remember, an egg shape is not an ellipse.)

The sum of the previous discussion is that although pictorials have the advantage of looking realistic, it is difficult or impractical to create an object with precision from them. Pictorials are subject to misinterpretation and errors in analysis and fabrication because the angles and distances are distorted. The most universally accepted solution to these problems is to use multiview representations, which are explained next.

8.02.02 Viewing Planes

A **multiview** representation depicts in one plane, such as a sheet of paper, many images of the same object, each viewed from a different direction. Pictorials can be used to enhance the clarity of the 3-D perception of an object; but the sizes of the object and its details are shown in a series of views, each view showing the sizes in their true length or shape. Any fabrication or analysis of the object's measurements can then be based on what is shown in the multiview projections, not on what is shown in the pictorial.

When you visualize an object in space, its appearance changes depending on the direction from which you view it. The lines and curves that form the graphical presentation of the object, such as the lines and curves shown in Figure 8.03, represent edges that are the intersections of surfaces. Now visualize a transparent plane, perhaps a sheet of glass, fixed in space between you and the object. This plane is called a **viewing plane**. Imagine the image of the object as seen through the plane is somehow painted onto the plane. Continuing to imagine, remove the object and look at the image painted on the viewing plane. What you see on that plane is a 2-D image of a 3-D object. The appearance of the image, however, would depend on the viewing angle of your head in front of that plane when you created the image. The simplest and most accurate view is from your head looking directly forward at the object. In general, to be accurate about the appearance of the object as seen through the plane, you would need to define the locations and orientations of the object, the viewing plane, and the viewer. This is a great deal of information. But you would not need all of that information if you defined the image as one created by orthogonal projection, which is explained next.

8.02.03 Orthogonal Projection

In **orthogonal projection**, the image of an object is composed of points projected from individual points on the object onto the viewing plane such that the projection of each point is perpendicular to the viewing plane. Orthogonal projection of an object onto a transparent viewing plane is shown in Figure 8.04, where you can see the perpendicular relationship between the projection lines and the viewing plane when the plane is turned on edge.

An image created in this manner has two advantages. One advantage is that such an image is easy to create because you do not have to worry about defining the location or orientation of the viewing plane relative to the line-of-sight. The line-of-sight from a point on the object to the viewing plane is like the projection path; that is, it is always perpendicular to the viewing plane. The other advantage is that by turning the object such that an edge of the object is parallel to the viewing plane, the image of that edge shows its true length. Furthermore, the length of a projected edge is independent of its distance from the viewing plane. Both of these properties are shown in Figure 8.05.

8.02.04 A Distorted Reality

An image created by orthogonal projection is merely a convenience that allows you to analyze the image more easily when you are ready to make the object depicted. In the strictest sense, orthogonal projection does not accurately represent an image of the way a real object looks. In reality, parts of an object that are farther away appear smaller than the same-sized parts of an object that are closer. With orthogonal projection, all parts of the object appear in the same scale no matter how far the object is placed from the

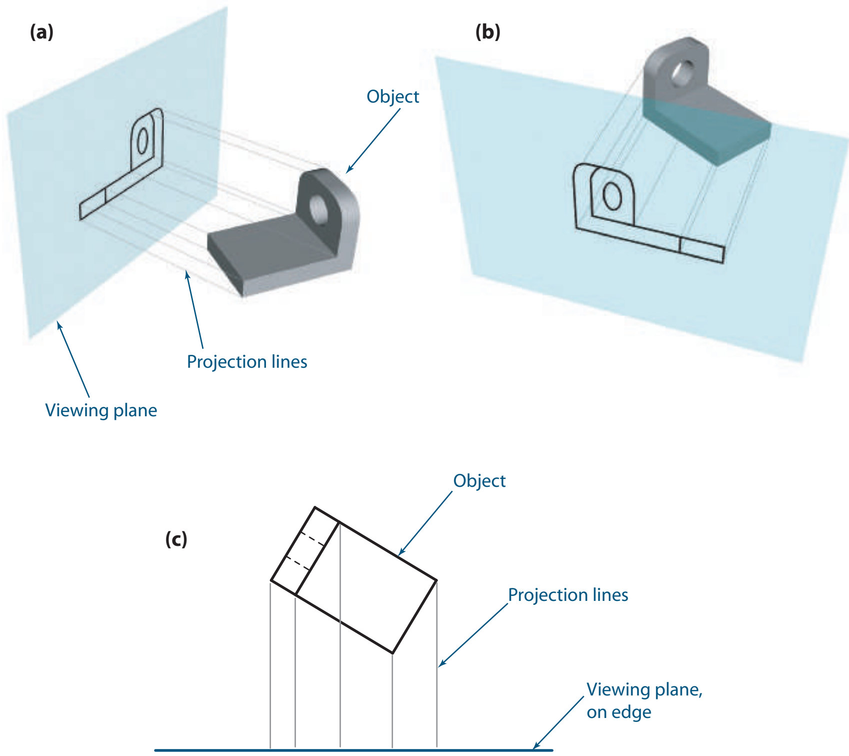

FIGURE 8.04. Using orthogonal projection to create an image of an object on a viewing plane. The object in (a) is in front of the viewing plane. The object in (b) is behind the viewing plane. In either case, the projection lines are perpendicular to the viewing plane, as shown in (c).

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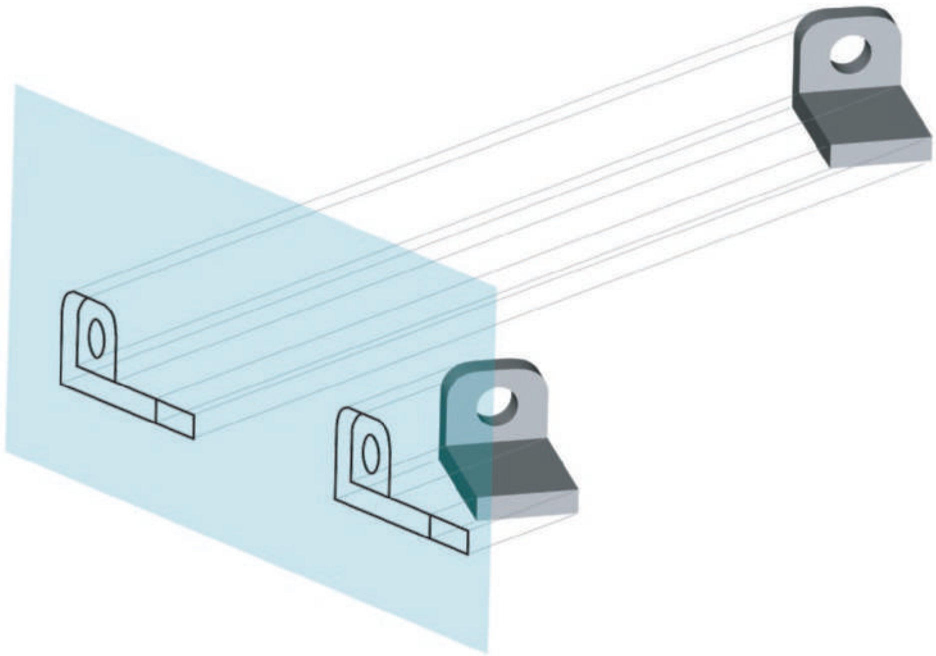
FIGURE 8.05. With orthogonal projection, the projected length of an edge is independent of its distance to the viewing plane. For this particular object orientation, the true lengths of the vertical edges are shown.



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viewing plane. But as in the case that follows and in most cases, the approximation is close and the convenience and ease of image creation and analysis far outweigh the need to see the image as it really appears.

The effect of an image created by orthogonal projection is similar to a photograph of an object taken at a long distance using a powerful telephoto lens. That type of picture lacks depth; that is, the object appears flat. This lack of depth is attributable to the fact that although the light rays actually extend radially from the surface of an object, the reflected light rays appear less like radial rays and more like parallel rays viewed at a great distances from the object. The greater the distance, the more parallel the light rays. At a long distance, where the light rays compose the image of the object, such as at a camera lens, the light rays are very nearly parallel to each other and very nearly perpendicular to the plane of the lens. This effect is shown in the bottom photograph in Figure 8.06; both photographs are the same object, each taken from a different camera distance. Even though the overall image size of the object is about the same, in the close-up photo, you should be able to see that the parts (for example, the wheels) of the object that are closer appear magnified when compared to the parts that are farther away.

8.02.05 Choice of Viewing Planes

From what was just explained, you should understand that an orthogonal projection of an object is a 2-D drawing of that object as it would appear on a viewing plane. To get a different view of the object, you need to move the object and/or the viewing plane to a different location.

Consider the case of keeping the viewing plane in the same place and rotating the object. One advantage of orthogonal projection is that an object's lines and curves can be seen in their true shape. For example, when the viewing plane is parallel to a circle, the circle actually appears as a circle rather than an ellipse. This may be important, for example, when you want to see how close the edge of a hole in an object actually comes to the edge of the object. It makes sense, therefore, to rotate the object into an orientation where the measurements, such as the diameter of the hole or its distance to the edge, can be seen to represent the true shape, distance, and size. Figure 8.07 shows an object rotated into the best position for this specific analysis versus the same

FIGURE 8.06. The top photograph was taken from up close. The bottom photo was taken from a long distance and enlarged so feature sizes could be compared. Can you see the lack of perspective in the long-distance photo?

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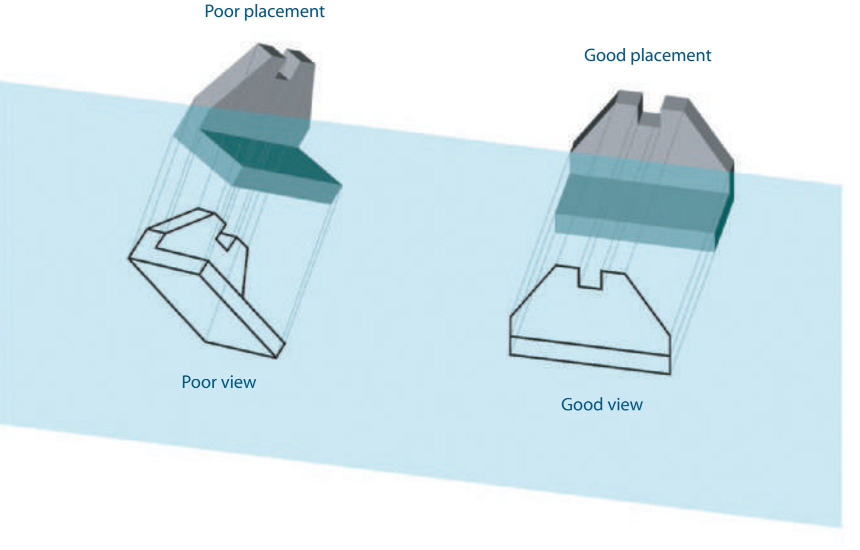
object in a poor orientation. In general, in the creation of the first view of an object, it has become common practice to orient the object in a position that shows as many as possible of its lines and curves in their true shape.

However, a single view of an object is usually insufficient to specify all of its features and measurements fully. Figure 8.08 shows how different objects can appear the same using a single view only.

FIGURE 8.07. Good part placement shows most of the part edges in their true length.



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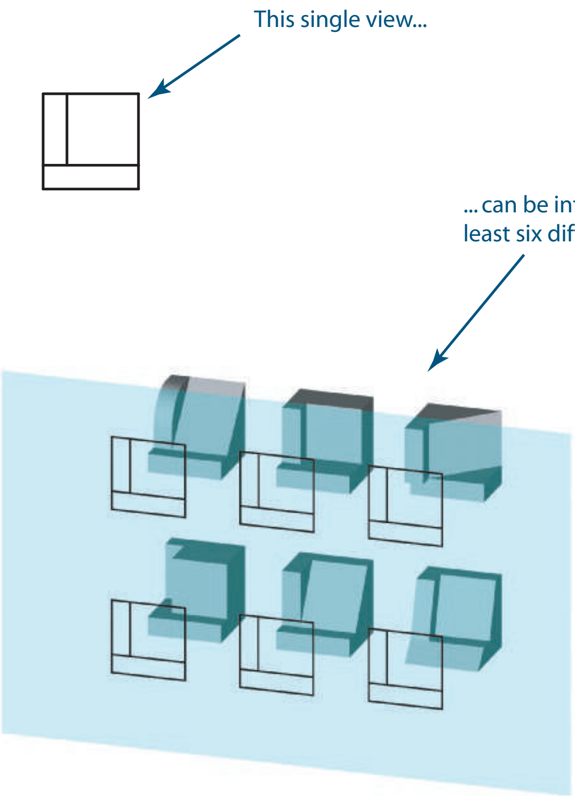

FIGURE 8.08. A single view of a part may have many different interpretations.

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To fully define the 3-D geometry of an object, it is necessary to depict the object in **multiple views**. This means there must be a viewing plane for each of the views. Specifying the location and orientation of each of the additional viewing planes must be done in a standardized way so that 2-D images can be extracted from the object easily. Also, the multiple 2-D images must contain enough information so that the original 3-D image can be re-created from them. One way to do this is to locate and orient the additional viewing planes so that each is orthogonal to the first viewing plane, as shown with a second and third viewing plane in Figure 8.09a. The images on all of these viewing planes are created using orthogonal projection.

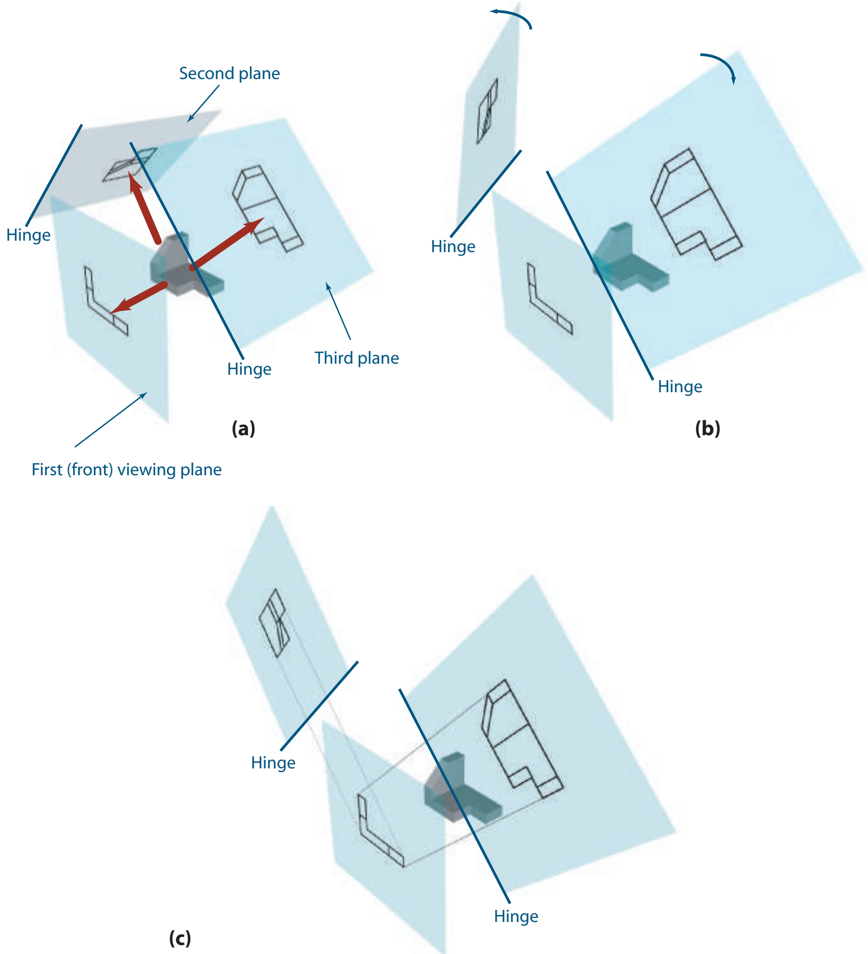
When the location and orientation of the intersection line between the first viewing plane and any one of the additional viewing planes are known, the location and orientation of each of the other additional images can be specified. The intersection line between the first viewing plane and any of the additional viewing planes can be imagined as a hinge between the two planes. By "unfolding" the additional planes at their imaginary hinges, as shown in Figure 8.09b, the images on all of the viewing planes can be shown on a single plane, or in other words, a 2-D drawing.

Used this way, orthogonal projection and viewing planes offer you the advantage of seeing multiple views of the same object at the same time on a single sheet of paper. Orthogonal projection also can precisely identify the position and orientation of the viewing planes used to create those views by specifying on the single sheet the location of the intersection lines between the viewing planes.

8.02.06 Size and Alignment

When the second and third planes are completely unfolded and are coplanar with the first viewing plane, as shown in Figure 8.09c, three images can be seen on a single plane. The images from the second and third planes are considered adjacent to (i.e., created immediately next to) the image from the first plane. Note that the size and orientation of the images are not arbitrary. Each image has the same scale (or magnification); and

FIGURE 8.09. Two viewing planes that are orthogonal to the first (front) viewing plane (a) can be unfolded (b) to present the images on a single plane (c). The imaginary hinges for the two viewing planes are at the intersections of these planes with the front viewing plane.



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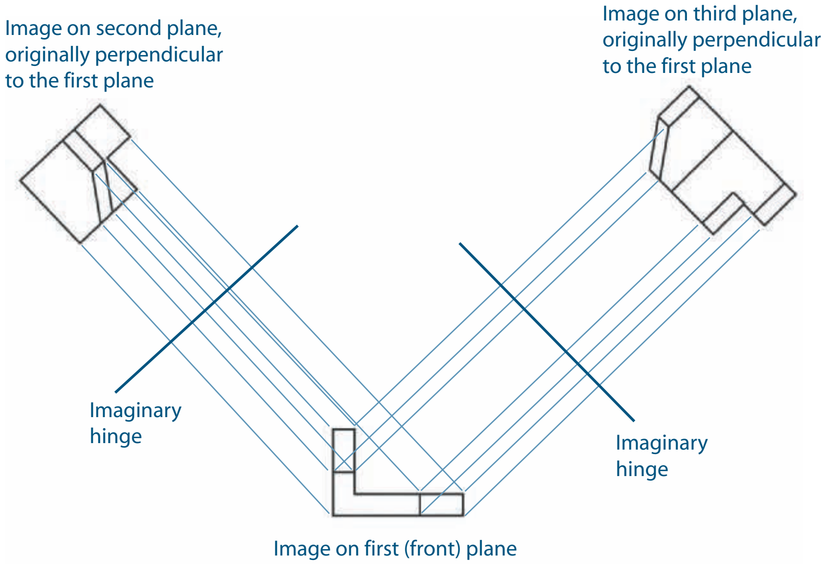
the orientation of the image is dependent upon the original location of its viewing plane as defined by the location of the intersection line between the viewing planes, or their hinge. This alignment of the vertices of the object images in **adjacent views** is shown in Figure 8.10, where the three views are presented on a single sheet.

FIGURE 8.10. Viewing planes completely unfolded showing proper size, location, and orientation of the images on a single plane.



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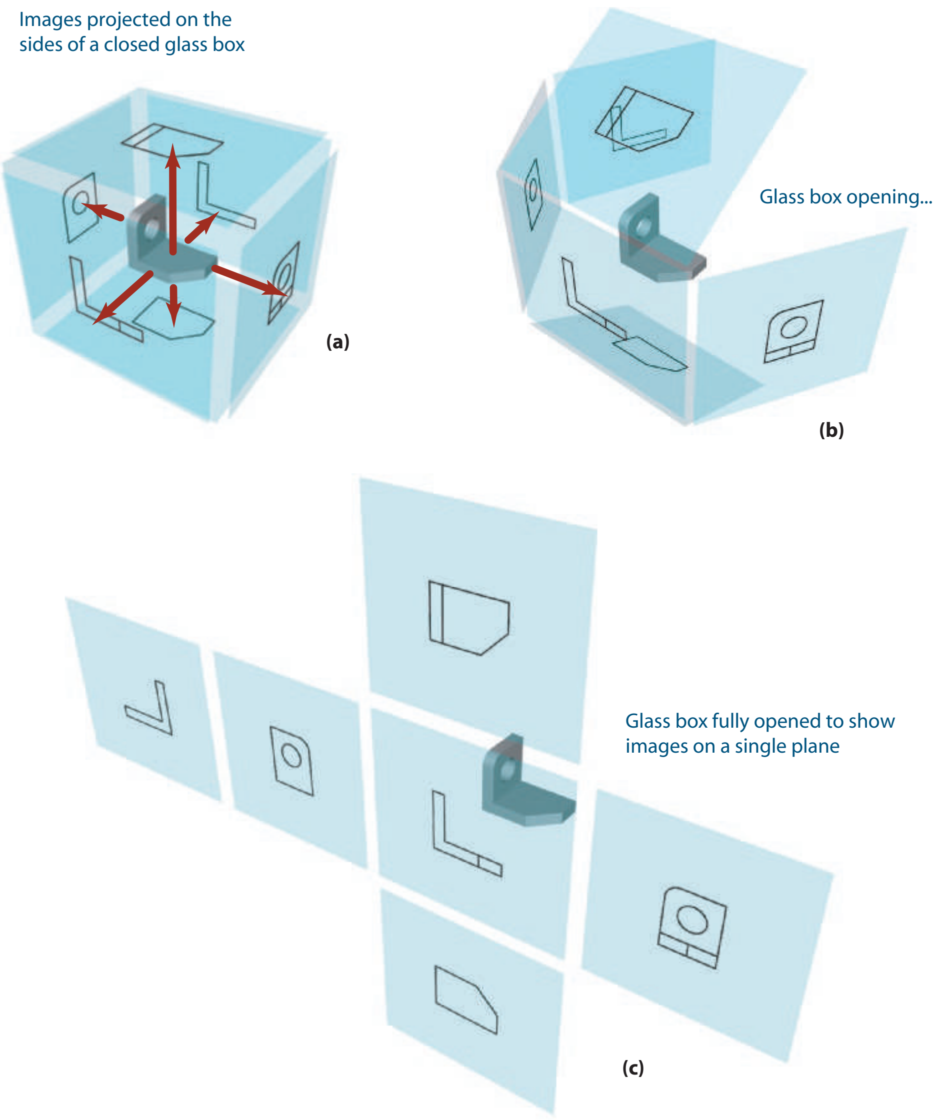
8.03 The Glass Box

Only three or four views are required to fully define most objects. Simple objects may require only one view; complicated objects may require six or more views. Objects such as engineered parts can usually be fully defined when they are viewed through a set of six viewing planes that together form a **glass box**, as shown in Figure 8.11a. The glass box has the unique property that for any viewing plane, all of its adjacent planes are perpendicular to each other and opposite viewing planes are parallel to each other.

When you open (or unfold) the panels of the box, as shown in Figure 8.11b, you can view all six sides of the object simultaneously on a single plane, as shown in Figure 8.11c. There is more than one way to unfold the box. Unfolding in the manner shown in Figure 8.11 is the standard way to do it according to accepted drawing practices. The top and bottom and right- and left-side views open about the front view; and the rear view is attached to the left-side view.

Make sure you see and understand that when the viewing planes are completely unfolded, the size and orientation of each image are not arbitrary. The scale in each view is the same. In the case of the complete glass box, each viewing plane is orthogonal to its adjacent viewing planes. When the box is unfolded and presented

FIGURE 8.11. Viewing an engineered part through a glass box (a) that opens (b) to present the images on a single plane (c).



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on a single sheet, as in Figure 8.12, adjacent images are aligned horizontally for horizontally adjacent views or vertically for vertically adjacent views. These alignment properties are very important when the object is analyzed. If you select any point on the object (assume point A on Figure 8.12), the images of that point will be horizontally aligned with each other on horizontally adjacent views and those images will be vertically aligned with each other on vertically adjacent views.

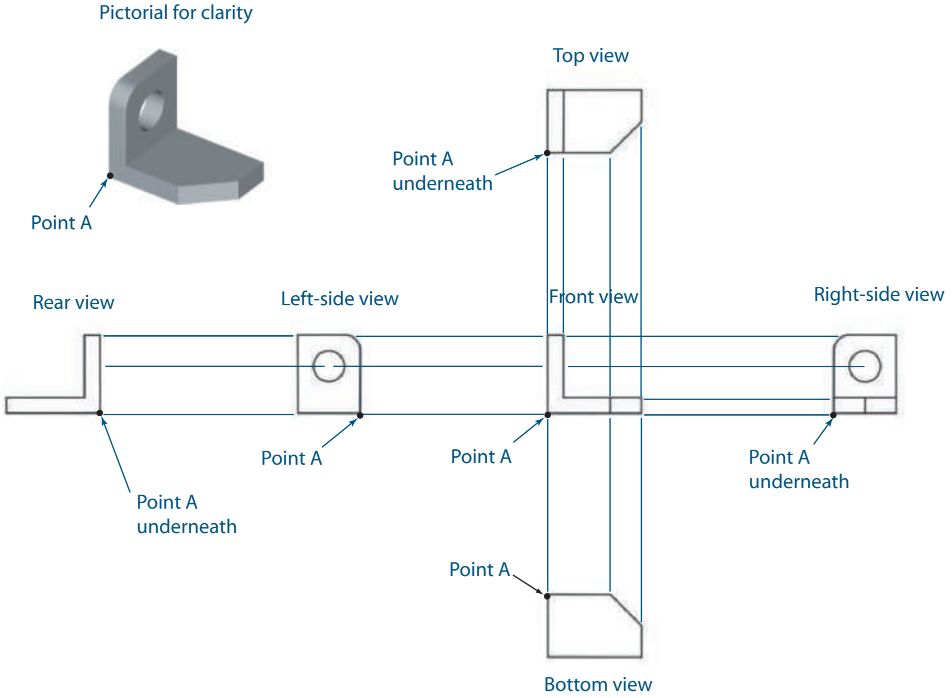
In general, the same point in space seen in adjacent views is aligned along a path that is perpendicular to the intersection line of the viewing planes, as shown in Figure 8.10. What this means for engineering drawing is that features on an object, such as edges or holes, shown in one view can be easily located on the adjacent view because the features are aligned between adjacent views. For complex objects with many features, the ability to identify the same feature on adjacent views is of tremendous utility.

8.03.01 Standard Views

The glass box yields six different views of an object. For a large percentage of engineered parts, six views are more than sufficient. Engineers typically like to design things that are easy and therefore inexpensive to make. Three-axis milling machines and single-axis lathes are common machines in any fabrication shop. These machines easily create surfaces on the workpiece that are parallel, perpendicular, or concentric to each other and that easily cut holes, slots, or other features that are perpendicular to the working surface.

The six views represented by the glass box are the front, top, left-side, right-side, bottom, and rear views. These views are known as the six standard orthogonal views or the six principal orthogonal views or more simply as the **six standard views** or the **six principal views**, respectively. When a formal drawing is created showing these views, the intersection lines and projection lines between views are not shown because these lines do not add much information to the drawing when it is already understood that adjacent views are orthogonal to each other. Also, each view does not need to be labeled as the front, top, right-side, etc., views.

FIGURE 8.12. Alignment of points on adjacent views for all six standard views.



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8.03.02 The Preferred Configuration

Are all six views necessary? Usually not. The great percentage of engineered parts can be fully defined geometrically with fewer than all six of the standard views. In fact, most engineered parts can be completely defined for fabrication using only three views.

Although there are no defined rules as to which views must be included or excluded in a formal engineering drawing, there is a **preferred configuration**—the front, top, and right-side views. Additional views are presented only when necessary to reveal and define features that cannot be shown in the preferred views. The preferred configuration for the object in Figure 8.12 is shown in Figure 8.13. Only the front, top, and side views are shown. Make sure every edge of the object can be seen in its true length in at least one view.

It is becoming increasingly popular to include an isometric or trimetric pictorial of the object somewhere on the drawing. When a pictorial is included, it serves only to aid in clarity; it does not need to be properly aligned or scaled with the standard views.

8.04 The Necessary Details

Only the minimum number of views needed to quickly and accurately communicate the geometry of an object should be created. Whenever possible, the preferred configuration of a front, top, and side view should be used unless fewer than three views are needed to see and define all of the features of the object. To minimize the number of required views on complicated objects and to reduce any possible ambiguity, some shorthand notation that describes common geometries such as certain types of holes and screw threads is used in drawing practice. Such notation is detailed in later chapters in this book. There will, however, be cases where additional views become necessary or when the preferred configuration may not be the best.

8.04.01 Hidden Lines and Centerlines

The dashed lines you see on the views shown in Figure 8.13 represent internal features or edges that are obscured by the object. These obscured features or edges are called **hidden lines** in these views. Hidden lines, which are denoted as equally spaced dashed lines on a drawing, represent the edges of an object or its features that cannot be seen on the real object but would be visible if the object were partially transparent. Hidden

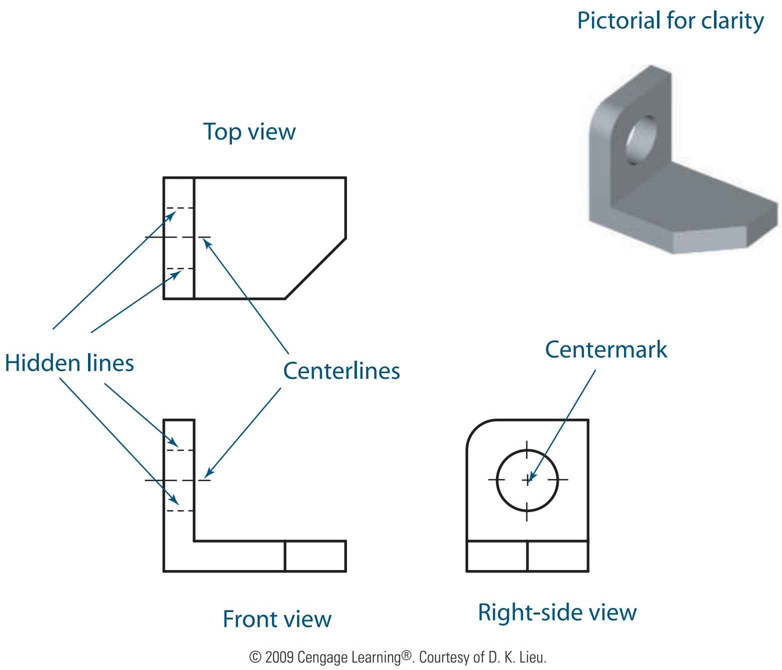

FIGURE 8.13. The preferred presentation configuration showing the front, top, and right-side views of an object. Other views are added only when necessary to show features that cannot be defined in the preferred configuration.

lines are used to emphasize an object's unseen geometry and thus speed the interpretation of its presentation. Hidden lines also are used to reduce the need for creating additional views. Although hidden edges cannot be seen on an opaque object, they are represented graphically the same way hidden lines are included in a view to emphasize that a feature cannot be seen in that view or to show that a feature cannot be seen from any of the other views. Later in this chapter, hidden lines will be discussed further as you encounter examples of the advantages and problems associated with them.

Looking closely at Figure 8.13, you will see lines located at the center axis of the hole. These are not hidden lines. They are **centerlines**, which are represented graphically by alternating short and long dashes along the length of the center of the circular hole. Centerlines cannot be seen on the real object, but they must be included on the drawing to identify where the center of the circular hole is located on the object. More generally, centerlines are used where there is a cylindrical surface such as a hole or a tube.

The reason for including centerlines is to make it easier for the reader to distinguish between edges, visible or hidden, that are part of a cylindrical surface and edges that result from the intersections of planes. Using centerlines also makes it easier to locate features such as holes, which are commonly defined by their diameters and center locations.

A **centermark**, the end view of a centerline, is identified by a right-angle cross such as that shown in the center of the circular hole in the right-side view of the object in Figure 8.13. Typically, centerlines and centermarks are used where the arc of a cylindrical surface is 180 degrees or greater, although they can be used for lesser arcs as required for clarity in a drawing.

8.04.02 The Necessary Views

How many views should be created to fully define an object? In engineering practice, it is considered poor practice to create more views than are needed. Creating unneeded views means more work for which there is no payoff. However, having too few views can create problems when the fabricator tries to make the part. In the worst-case scenario, the fabricator will try to guess what you want, get it wrong, and deliver a potentially expensive part that cannot be used. In that case, the creator of the drawing would be at fault, not the fabricator. The party responsible for creating the drawing also may be legally responsible for paying for the services of the fabricator.

So how many views are needed to fully define an object? The number depends on how complicated the object is to depict in three dimensions. Start by creating the front, top, and right-side views. Remember, they represent the preferred configuration, which all engineering personnel like to see. Try to orient the object in such a way that these three views reveal as much of the object's features as possible. If you are lucky, these three views will fully define the object; but that is not always the case.

You should ask yourself the following two questions when you finish creating the drawing views:

1. Can the true size of all of the measurements needed to define all of the features of this object be seen in at least one of the views just created?
2. Is it impossible for the geometry of any feature to be misinterpreted as another type of geometry?

Yes to both questions means you have enough views. No to either question means you have more work to do.

The multiview production drawing for the Hoyt AeroTec riser is shown in Figure 8.14. The complexity of this object requires that all six standard views be used because it has features that can be seen only from each of the six viewing directions.

Objects that are flat can be defined with a single view along with some sort of note specifying the thickness of the object. Flat sheet metal objects and objects that can be cut from a plate of uniform thickness fall into this category. The cuts must be through

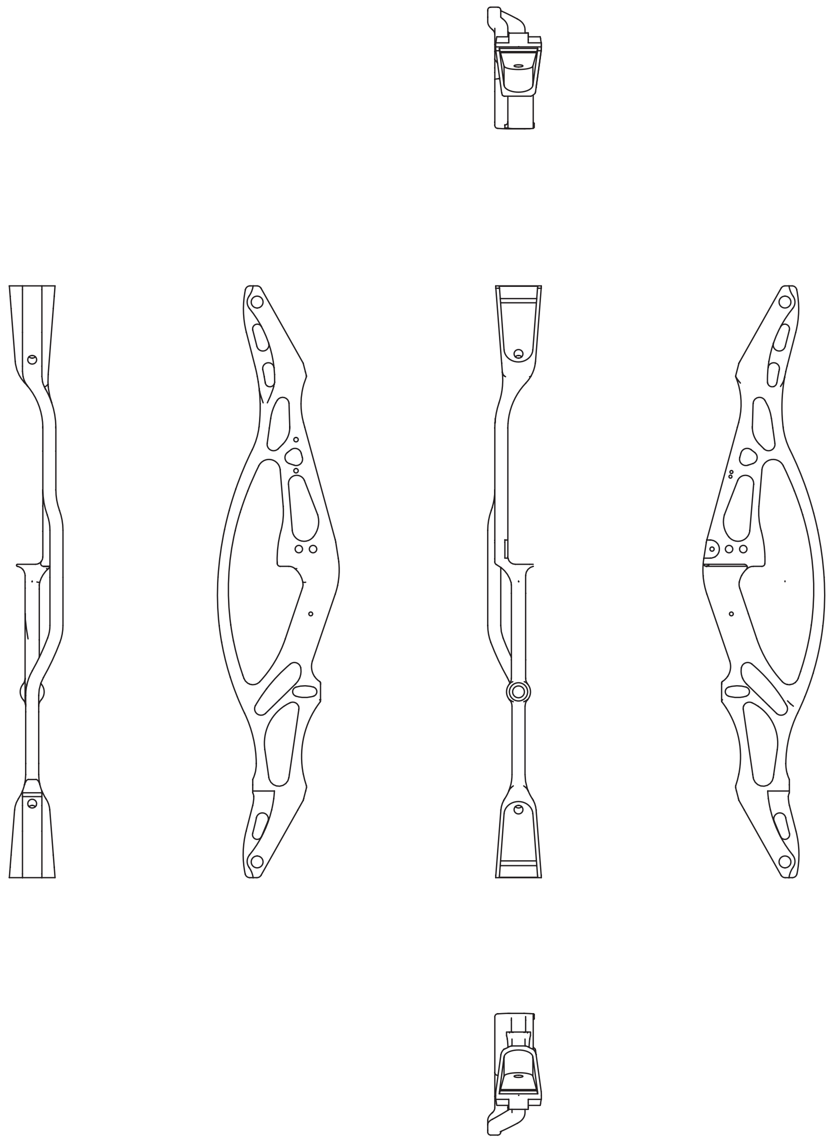

FIGURE 8.14. Formal multiview presentation of the AeroTec riser as would be seen in an engineering drawing. The measurements, text, and other specifications are not shown in this example so you can see the views more clearly.

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the entire thickness of the sheet or plate. An example of this type of object is shown in Figure 8.15. Because this object is made of very thin material, the adjacent orthogonal views would appear as lines.

Even when the thickness of the object is constant, a fabricator may find it helpful to see a second view; for example, to emphasize that the thickness of the object is a significant fraction of the object's planar geometry. See how the second view in Figure 8.16 helps depict the relatively large and uniform thickness of the object.

For objects that have 3-D features such as protrusions and cuts, each with a different depth, the problem of finding the proper number of views for a drawing becomes more difficult. Figure 8.17 shows an example of a drawing with two views. In this case, more than one interpretation of the object is possible. The addition of a third view is necessary to completely specify the desired object.

Figure 8.18 shows three original views that, in the absence of hidden lines, could be used to represent two possible objects. A fourth orthogonal view, a bottom view, is required in this case to distinguish between the two possibilities.

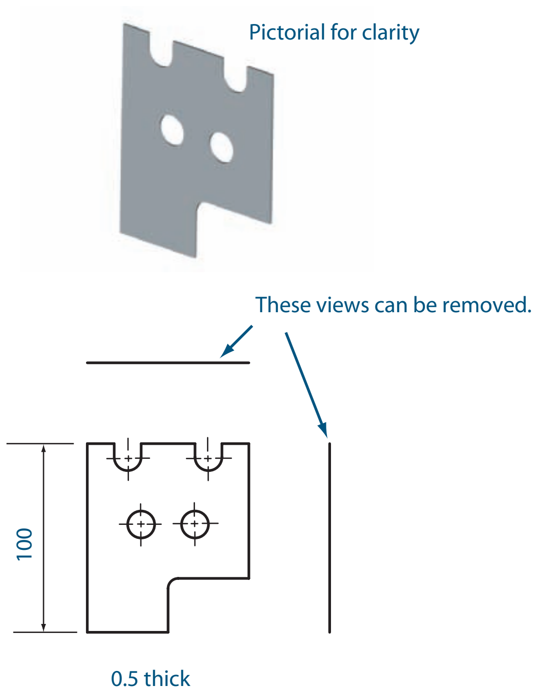


FIGURE 8.15. Additional views for very thin parts, such as sheet metal, add little information.

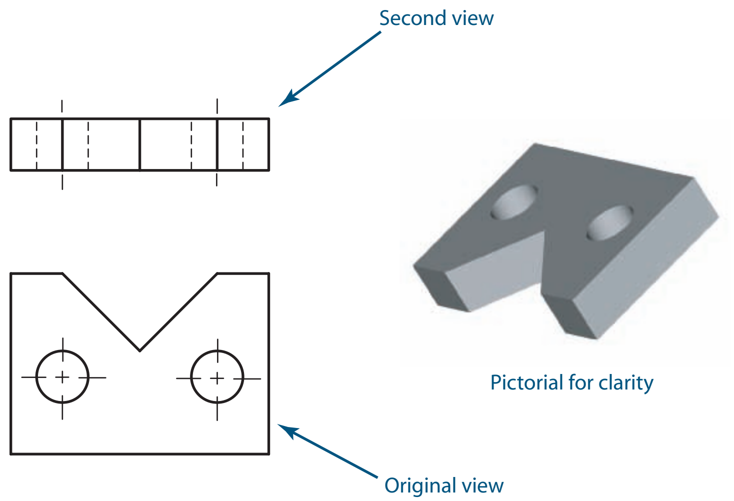


FIGURE 8.16. For a part with a constant but significant thickness, including a second view is a good idea to emphasize the 3-D nature of the part.

FIGURE 8.17. Different interpretations of a drawing with two views. A third view is necessary.

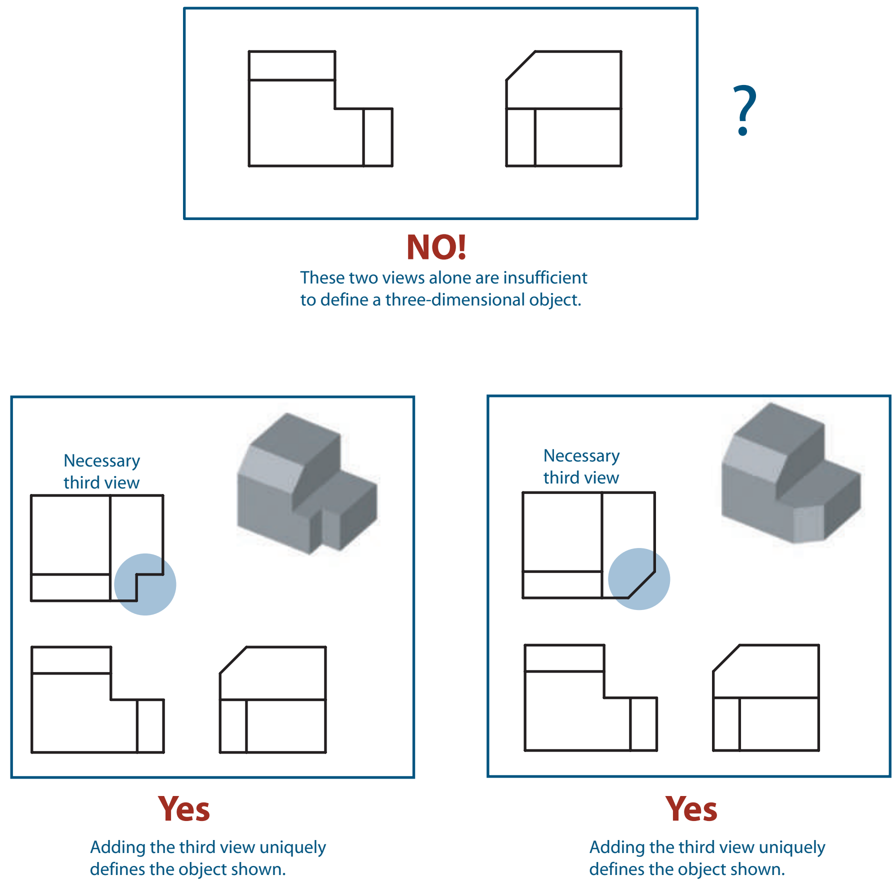
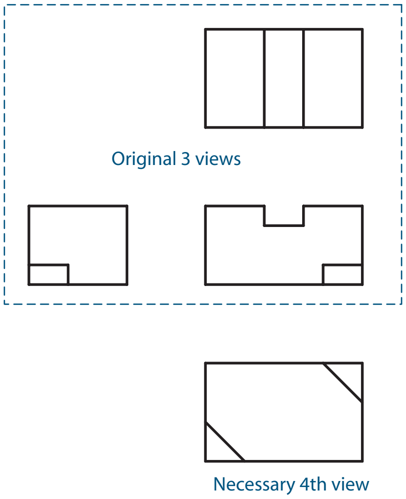


FIGURE 8.18. In the absence of hidden lines, four views are required to distinguish between these two parts. The fourth view is needed to distinguish the cutout on the underside as being diagonal instead of square.



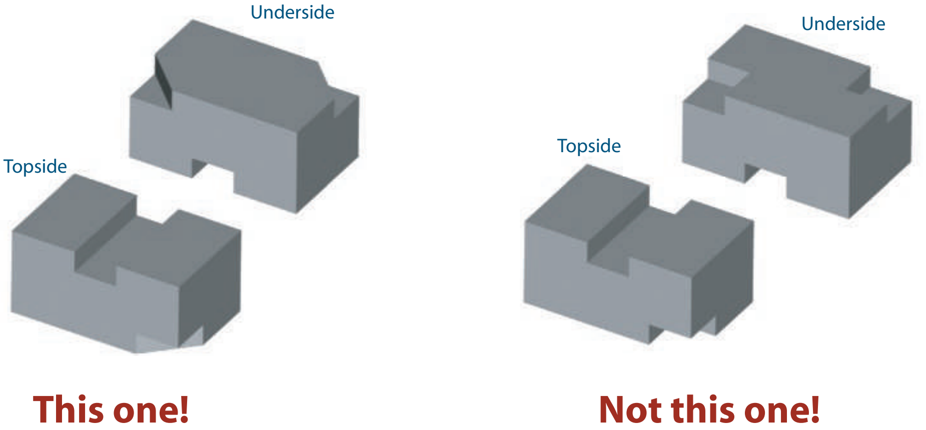


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Figure 8.19 shows an example of an object where, in the absence of hidden lines, five views are necessary.

As a rule of thumb, when an object contains inclined surfaces with respect to the standard viewing directions, each of those inclined surfaces must appear inclined in at least one of the orthogonal views representing the object. When the inclined surface is not shown in one of the orthogonal views, a view needs to show the surface as being inclined (i.e., with at least one of its edges at an angle that is not 0 degrees or 90 degrees).

8.04.03 Hidden Lines versus More Views

One way to reduce the number of required views is to use hidden lines. The object shown in Figure 8.20 has some unique features. Try to imagine representing the object without using hidden lines. Without the hidden lines, five views would be required to define all of its features. With only the front, top, and right-side views and no hidden lines, the geometry of the keyway seen on the underside of the object cannot be defined. Moreover, without hidden lines, additional views would be required to show that the hole and slot extend all the way through the object.

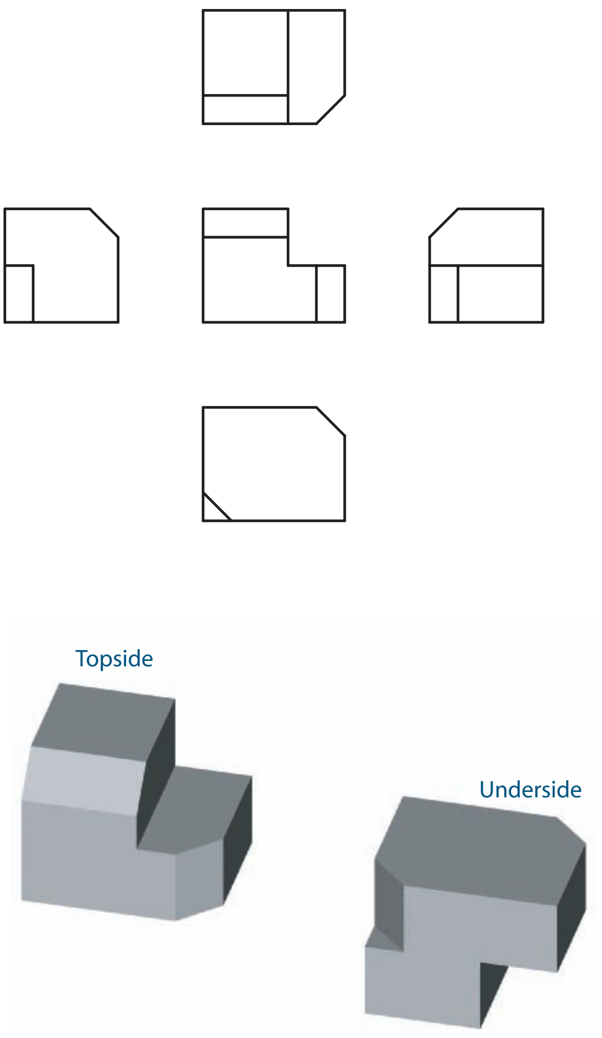

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FIGURE 8.19. Without the benefit of hidden lines, five views are required to describe this object.

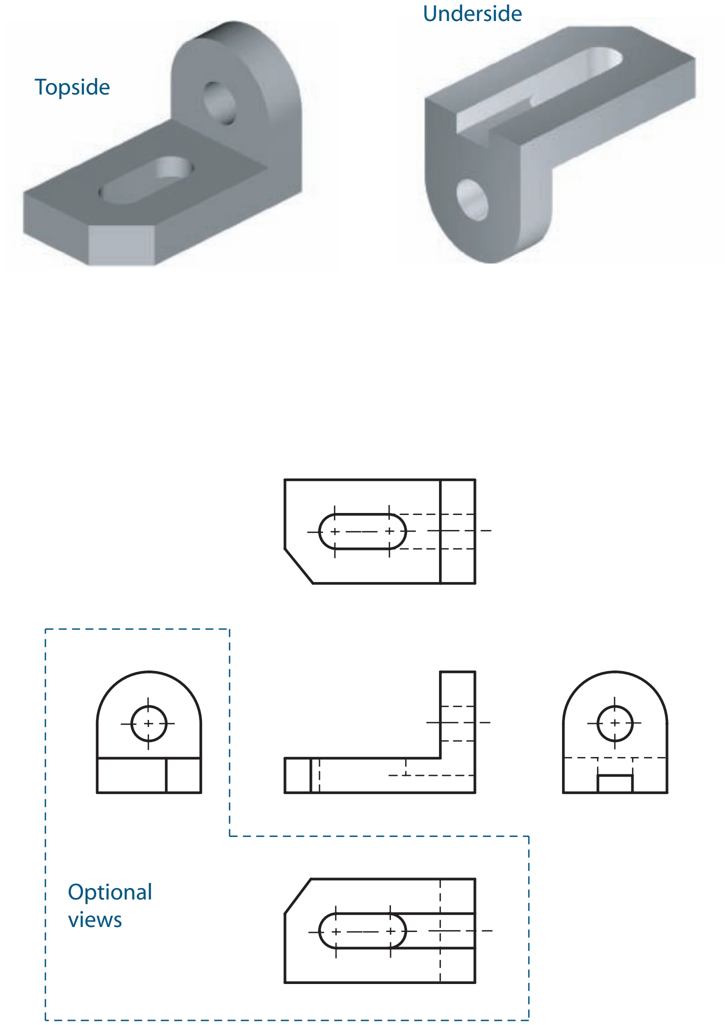

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FIGURE 8.20. Hidden and internal features on a part. Using hidden lines makes the left-side and bottom views optional.

By using hidden lines, only three views are required—the preferred configuration of front, top, and side views. Whether you use all five views shown in Figure 8.20 or the preferred three-view presentation depends on your answer to this question: Which presentation would be clearer? You always select the presentation that has, in your opinion, the least ambiguity (and not necessarily the least amount of work to produce). In the case of the five-view presentation, although it would not be an absolute requirement, adding hidden lines would emphasize the internal geometry of the object. For the three-view presentation, adding the hidden lines would be an absolute necessity.

Another use of hidden lines is to reveal the details of internal features that cannot be easily seen in any of the standard orthogonal views. Such details would be, for example, the depth or the profile of holes and slots, as shown in Figure 8.21. Figure 8.20 demonstrates how hidden lines can be used instead of additional views, making the drawing easier to create and more compact without the loss of any information. For the object shown in Figure 8.21, the depth of the slot cannot be seen in any of the standard orthogonal views. If you look carefully at the views for the object shown in Figure 8.21, you see that hidden lines for different features can be separated into different views. But if all of the hidden lines were shown on all of the views, the result would be a jumble of so many hidden lines that it would be difficult to distinguish the different features that they represent.

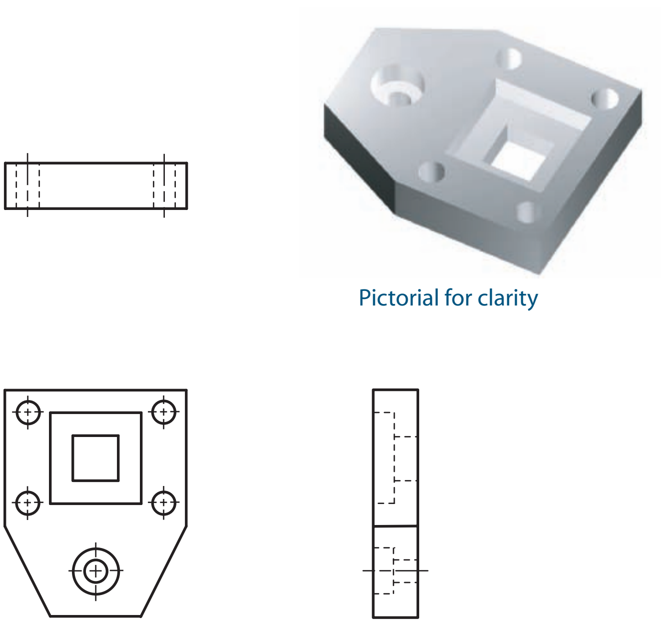

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FIGURE 8.21. Use of hidden lines to reveal internal features.

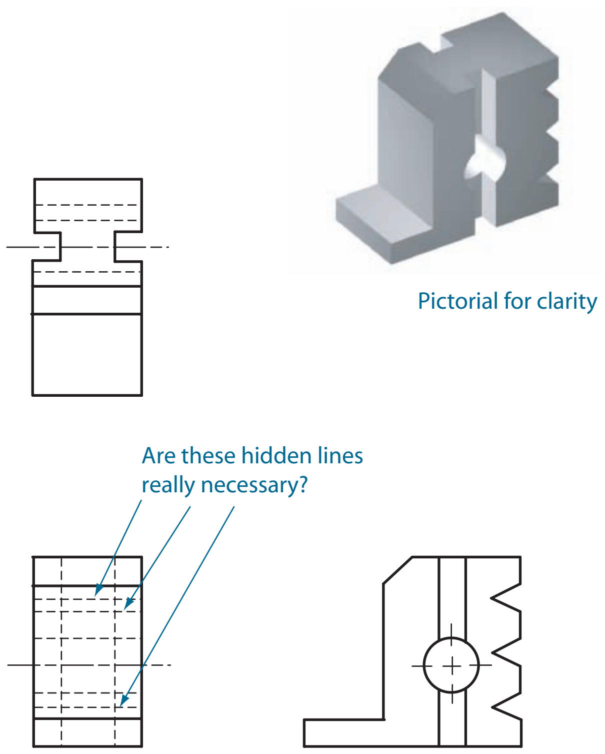

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FIGURE 8.22. Overuse of hidden lines causes confusion. Exercise judgment. It might be better to create another view, such as a rear view in this case.

A common problem for inexperienced designers is deciding when to use hidden lines. Hidden lines should be used to add clarity to a drawing. Hidden lines should be used to emphasize a feature, even if that feature can be seen and defined in the existing orthogonal views. The goal of the creator or the drawing is to increase the speed at which the drawing can be interpreted. However, hidden lines must be used to add information when there is no way to obtain this information from the rest of the drawing.

Because hidden lines can be used to avoid creating another view, it is sometimes tempting to do just that, even when using another view would be better. Figure 8.22 shows that adding too many hidden lines creates a complex, confusing drawing. With this result, it would be better to create extra views. When deciding whether to use hidden lines or to add additional views, simply do whatever will cause less confusion for the reader of the drawing. However, it is usually not a good idea to create hidden lines of different features such that the hidden lines cross each other, lie on top of each other, or even come close to each other.

The purpose of hidden lines is to define or emphasize details that cannot be seen, which is accepted as standard practice. Deleting unnecessary hidden lines and adding additional views are considered optional methods of reducing ambiguity when the use of hidden lines makes the presentation confusing. There must be no confusion as to which feature a hidden line represents.

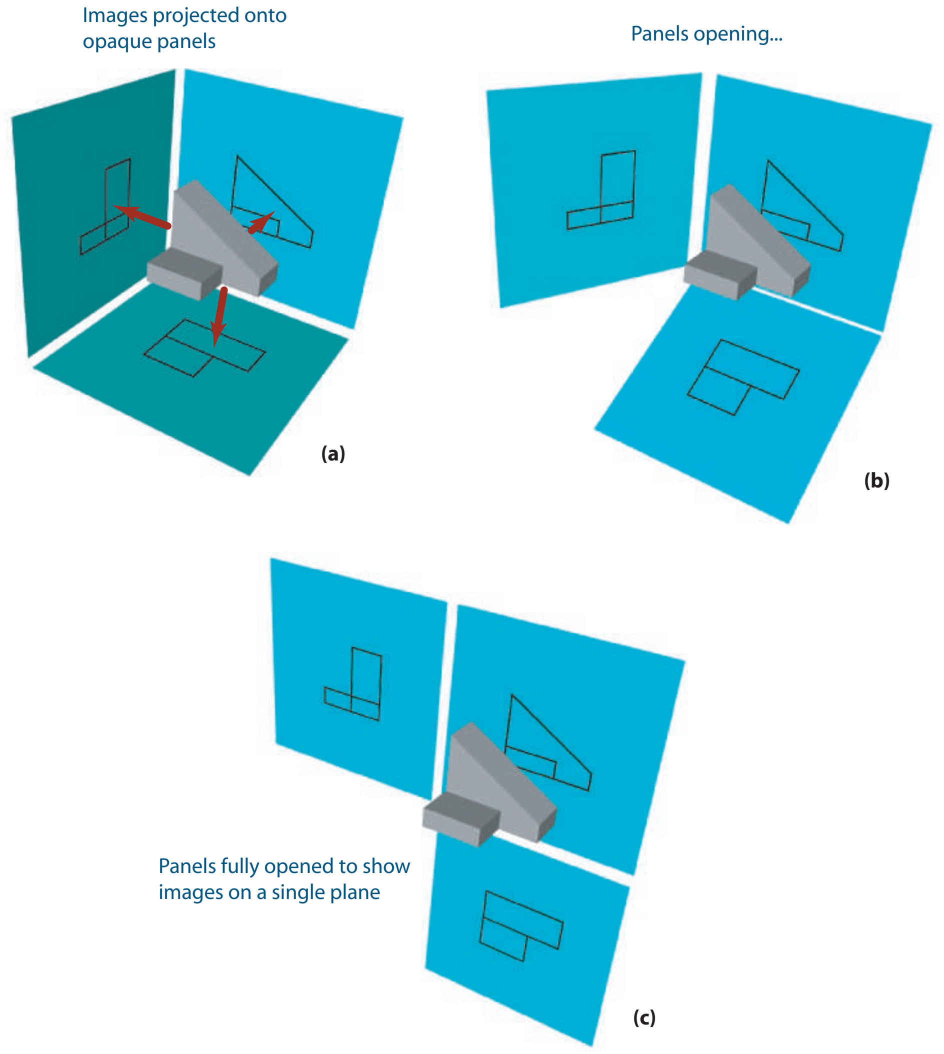
8.05 First-Angle Projection versus Third-Angle Projection

The glass box representation of multiviews of an object is formally referred to as **third-angle projection**. Whenever third-angle projection is specified on a drawing, each view of the object was created by projecting the image of the object onto the glass box's transparent viewing plane between you and the object—the object is behind the transparent viewing plane. The viewing planes are then rotated about their intersection lines until all of the views are shown on a single plane or sheet. This interpretation is the one most commonly used in the United States.

However, in some parts of Asia and Europe, **first-angle projection** is commonly used. In first-angle projection, each viewing plane is behind the object, which means the object is between you and the viewing plane. With first-angle projection, the viewing plane is opaque and the image is projected back and transferred onto the viewing plane. One way to interpret first-angle projection is to imagine the object in front of the opaque panels, as shown in Figure 8.23a. The image of the object, as seen by a viewer located directly in front of each panel (with the object directly in line between the two panels), is transferred to that panel. Opening the panels, as in Figure 8.23b, begins to show how the front, top, and right-side views are presented on a single plane, as shown in Figure 8.23c.

For drawings created using either first- or third-angle projection, the primary view is considered to be the front view. The front view in either projection is usually selected as the view containing the most features in their true sizes and shapes, thereby allowing for the most measurement extraction. As you saw earlier, for the six standard views using third-angle projection, the top view appears above the front view and the bottom view appears below the front view. The right-side view appears

FIGURE 8.23. Viewing an object in front of opaque panels for first-angle projection. The images are projected onto the panels (a), which open (b) to present the images on a single plane (c).



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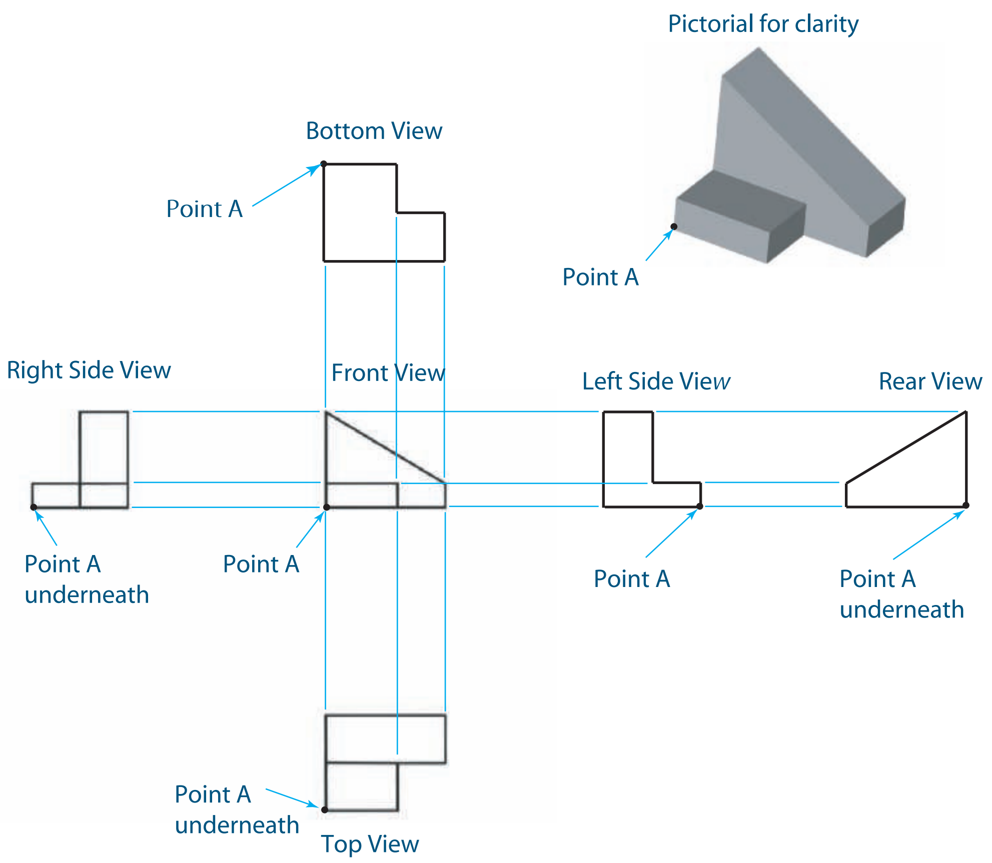
to the right of the front view, and the left-side view appears to the left of the front view. The rear view appears, by practice and convention, attached to the left-side view and appears to its left.

Using first-angle projection, the top view of an object appears below the front view and the bottom view appears above the front view. The right-side view appears to the left of the front view, and the left-side view appears to the right of the front view. The rear view appears, by practice and convention, attached to the left-side view and appears on its right. The location of the first-angle projection views is shown in Figure 8.24.

The differences between first- and third-angle projections are sometimes subtle and confusing, particularly because the front view of the object is the same in both cases. To add further confusion, for a large percentage of engineered parts, the left- and right-side views or the top and bottom views are identical. These reasons explain why drawings need to clearly specify whether first- or third-angle projection must be used to interpret the views. Many large companies operate internationally, with engineering and fabrication facilities worldwide. In international business, drawings are often created in one country and the parts fabricated in another country. When a drawing is interpreted incorrectly, the resulting fabricated part may be the mirror image of what was desired. Figure 8.25 shows a multiview drawing of an object and the two different objects that are created when the drawing is interpreted using first- and third-angle projection.

The symbol added to a drawing to specify first- or third-angle projection is two views of a truncated cone, as shown in Figure 8.26. This symbol depicts how a truncated cone would appear if a drawing of it were made using the projection method used for the entire drawing. The appropriate symbol and/or wording must be added to a formal drawing, usually somewhere in the title block (for which more detail can be found in Chapter 14) to eliminate ambiguities that may arise from misinterpreting which projection was used.

FIGURE 8.24. The six standard views, using first-angle projection, presented on a single sheet.



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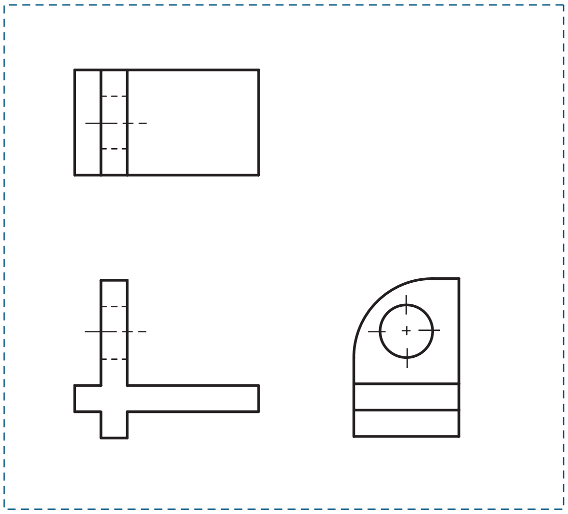
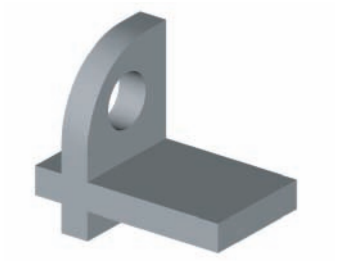
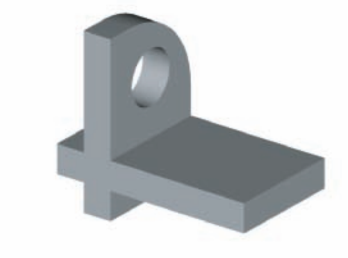


Multiview drawing

Object represented



Interpretation using third-angle projection



Interpretation using first-angle projection

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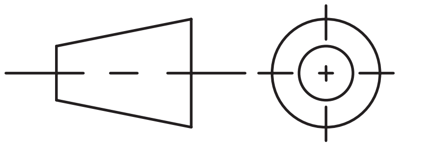
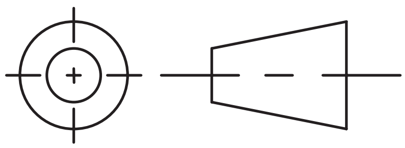
FIGURE 8.25. Drawing interpretation using first- or third-angle projection may lead to different parts.



Symbol for first-angle projection



Symbol for third-angle projection

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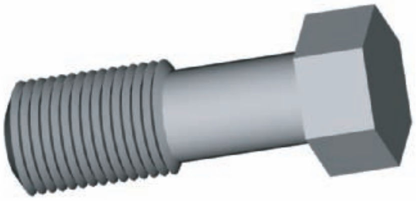
FIGURE 8.26. Drafting symbols for specifying the use of either first- or third-angle projection in a drawing.

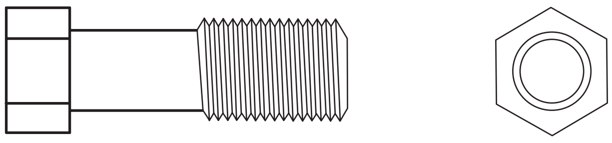
8.06 Breaking the Rules—and Why It Is Good to Break Them Sometimes

Creating an engineering drawing using orthogonal views is sometimes a balance between how accurately the drawing can be interpreted and how easily the drawing can be created. Strictly following some of the guidelines presented so far may lead to problems. To avoid those problems, you should consider some generally accepted exceptions to the guidelines, which are usually graphical shortcuts or approximations. These exceptions can reduce the time it takes to create a drawing and/or minimize possible misinterpretation of a drawing. With all of the exceptions that follow, the main question you need to ask yourself before using any of them is whether the approximation or shortcut could lead to misinterpretation of the drawing. If the answer is yes, the exception should not be used.

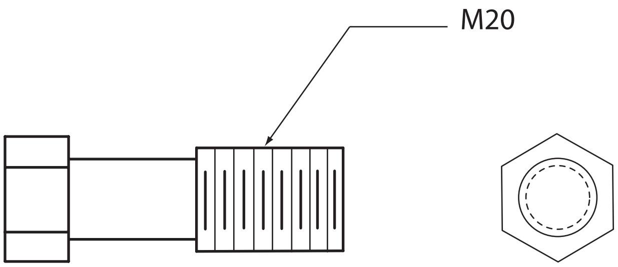
8.06.01 Threaded Parts

The first shortcut is in the representation of a threaded part, such as the bolt shown in Figure 8.27. A thread is essentially a helical mating surface for a fastener. The thread may be external, such as on the outside of a bolt or screw, as shown in Figure 8.27, or internal, such as on the inside of a nut. An accurate drawing of all surfaces on such an





Accurate projection



Schematic presentation

FIGURE 8.27. The schematic representation of an externally threaded part. The note specifies the metric size of the thread.

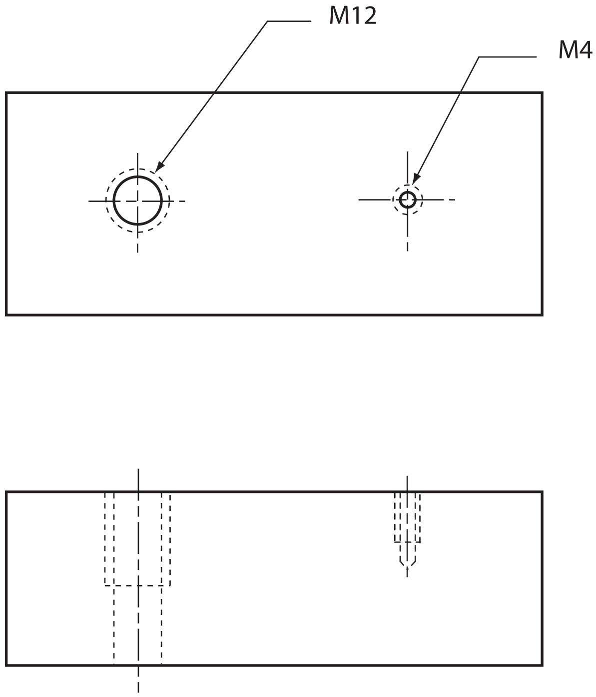


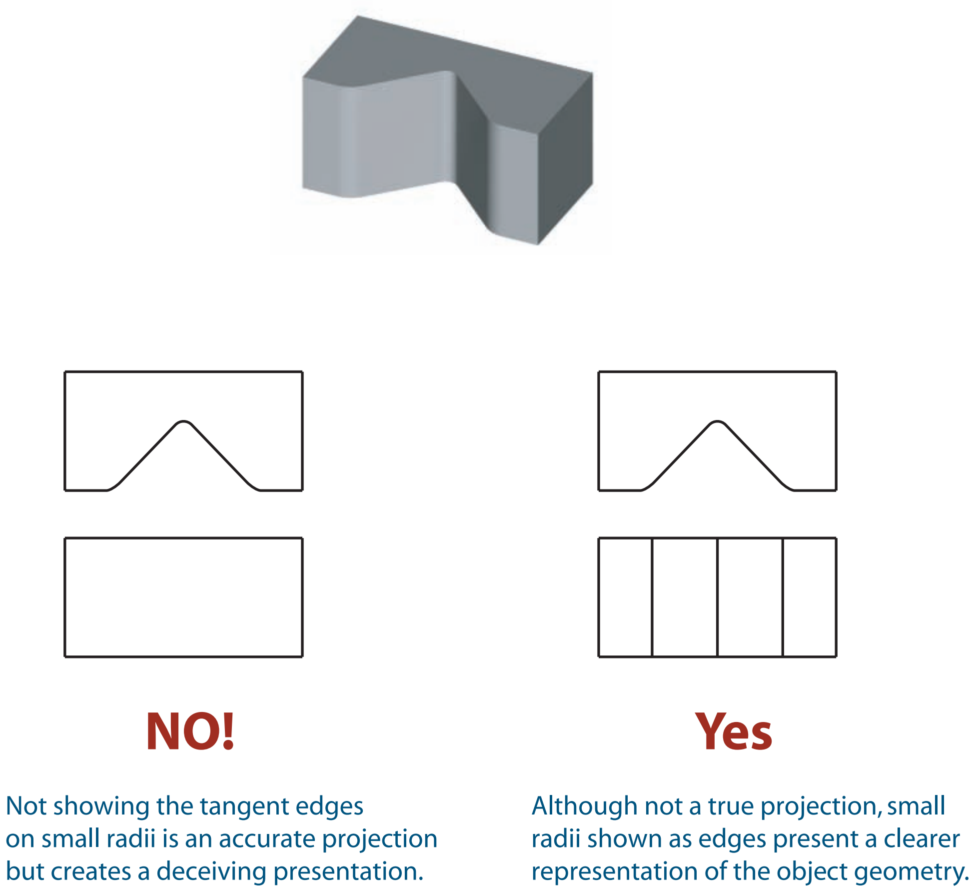
FIGURE 8.28. The schematic presentation of internal threads. The notes specify the metric sizes of the threads.

object would result in a very complicated drawing, especially if the drawing had to be created with manual instruments or 2-D CAD software. A much simpler representation of the external thread is shown as the schematic representation in Figure 8.27. For internal threads, the schematic representation is shown in Figure 8.28. These schematic representations are much simpler to construct with very little loss of information, especially since thread sizes are, for the most part, standardized based on the diameter of the part. A note (and arrow) is required to specify the precise thread sizes. Methods for the complete specification of thread sizes are found in Chapter 18 of this book. You can also find thread specifications in most machinists' or engineers' handbooks.

8.06.02 Features with Small Radii

An exception to the guidelines is in the representation of edges with small radii. Consider the object shown in Figure 8.29, which has small rounds on some of its edges. Based on the guidelines established in this chapter, a multiview drawing of the object should look like the drawing in Figure 8.29. Recall that cylindrical surfaces have

FIGURE 8.29. The representation of small radii on a part.



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no defined edges and the tangent lines between curved and planar surfaces are not shown. Following the established guidelines, the top view should look like a featureless plane. Such a presentation, however, would likely cause confusion because, upon initial inspection, the front view contains features that are absent in the top view.

A better, albeit not accurate, representation would be a presentation where the small rounds are represented as if they were true edges. The rounded edges are still shown in the front view, where their measurements can be specified. However, the approximation of the small rounds as edges enables the reader of the drawing to grasp the larger shape of the object more quickly. But what exactly is a "small" radius, and when should a small round be approximated as an edge on a drawing? The purpose of the approximation is to clarify the drawing. When the approximation clarifies the drawing, it should be used. As a rule of thumb, when the radius is less than about 5 percent of the overall size of the object, consider using the approximation.

8.06.03 Small Cutouts on Curved Surfaces

An approximation also is allowed when there is a small hole or another cutout on a curved surface. Figure 8.30, for example, shows a small hole and slot on a tube as compared to larger cutouts.

If a true projection were made of these features, the orthogonal views would show a curved depression on the surface of the tube. The shape of this curve is complex and would take time to create. In most applications, the size of the depression on the surface is unimportant; so the depression is not shown on the orthogonal views. The true projection of these features and the accepted shortcut are shown in Figure 8.30. This approximation makes the drawing easier to create, with very little loss of information.

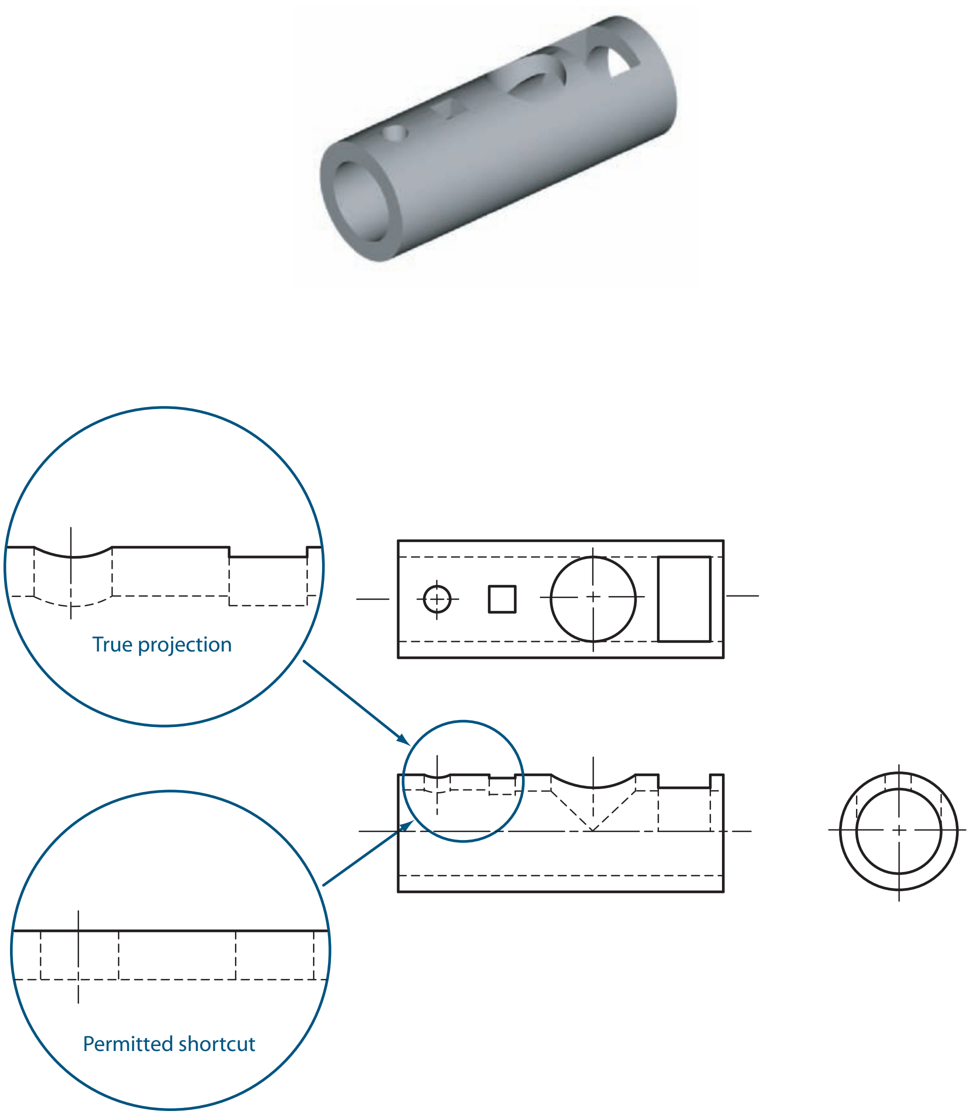

FIGURE 8.30. The true projection and an acceptable shortcut for small holes and slots on a curved surface. The shortcuts should not be used for large holes and slots because the geometric inaccuracies would be too obvious.

However, when the cutouts are large or the size of the depression cannot be ignored in the function of the object, the true projection should be used. Within these guidelines, what is considered "small" is up to whoever is creating the drawing. The question that must be asked is this: Will this approximation possibly lead to misinterpretation of the drawing? If the answer is yes, the shortcut should not be used.

8.06.04 Small Intersections with Curved Surfaces

A similar approximation is allowed for small protrusions that extend from a curved surface, as shown in Figure 8.31. As with small cutouts on a curved surface, the appropriate use of this approximation is subjective. When the protrusions are small relative to the arc of the surface, their intersections on the curved surface can be shown as lines without affecting the intended representation of those features. When the protrusions are large relative to the arc of the surface, the approximation cannot be made. Again, the question that must be asked is whether this approximation could lead to misinterpretation of the drawing. If the answer is yes, the shortcut should not be used.

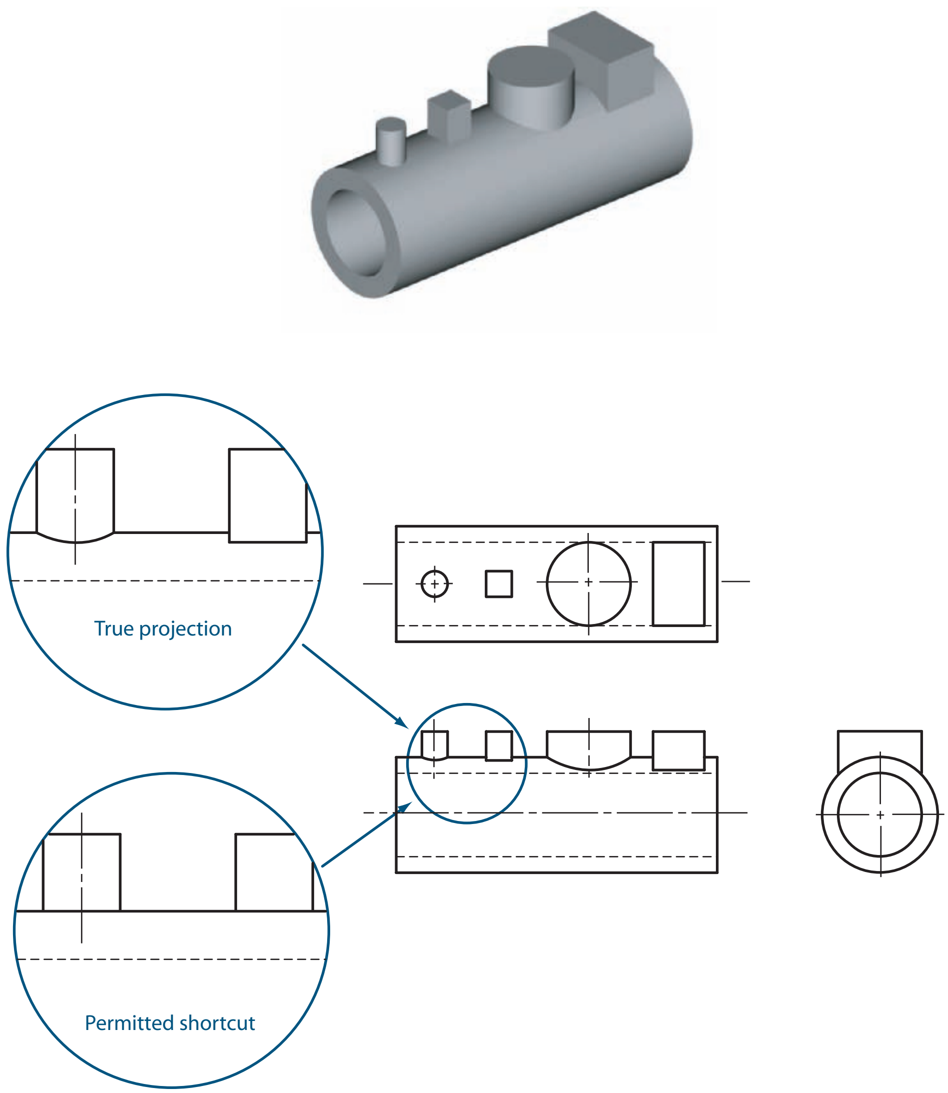

FIGURE 8.31. The true projection and an acceptable shortcut for small protrusions from a curved surface. The shortcuts should not be used for large protrusions because the geometric inaccuracies would be too obvious.

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8.06.05 Symmetrical Features

An interesting exception to the rules of true projection occurs in the representation of objects with symmetry, as shown in Figure 8.32. This object has one-third rotational symmetry, which means the object can be divided into three identical sections about its axis of rotation, with three support ribs about the center tube.

An accurate multiview drawing would be the true projection drawing shown in Figure 8.32 using a front and top view. However, using a true projection for the front view in this case has two problems. One problem is that when instruments or 2-D CAD is used, an accurate projection is difficult to create. The other problem is that the true projection of the side view may be incorrectly interpreted as representing a nonsymmetrical object.

A preferred presentation for this drawing is shown in Figure 8.32. This drawing is easier to create and gives the impression that the object is symmetrical. The top view clarifies any possible misinterpretation about the number and locations of the support ribs. This may seem strange, but if the object had one-quarter rotational symmetry, for example, with four equally spaced support ribs instead of three, the front view would be the same as the view for the three support ribs.

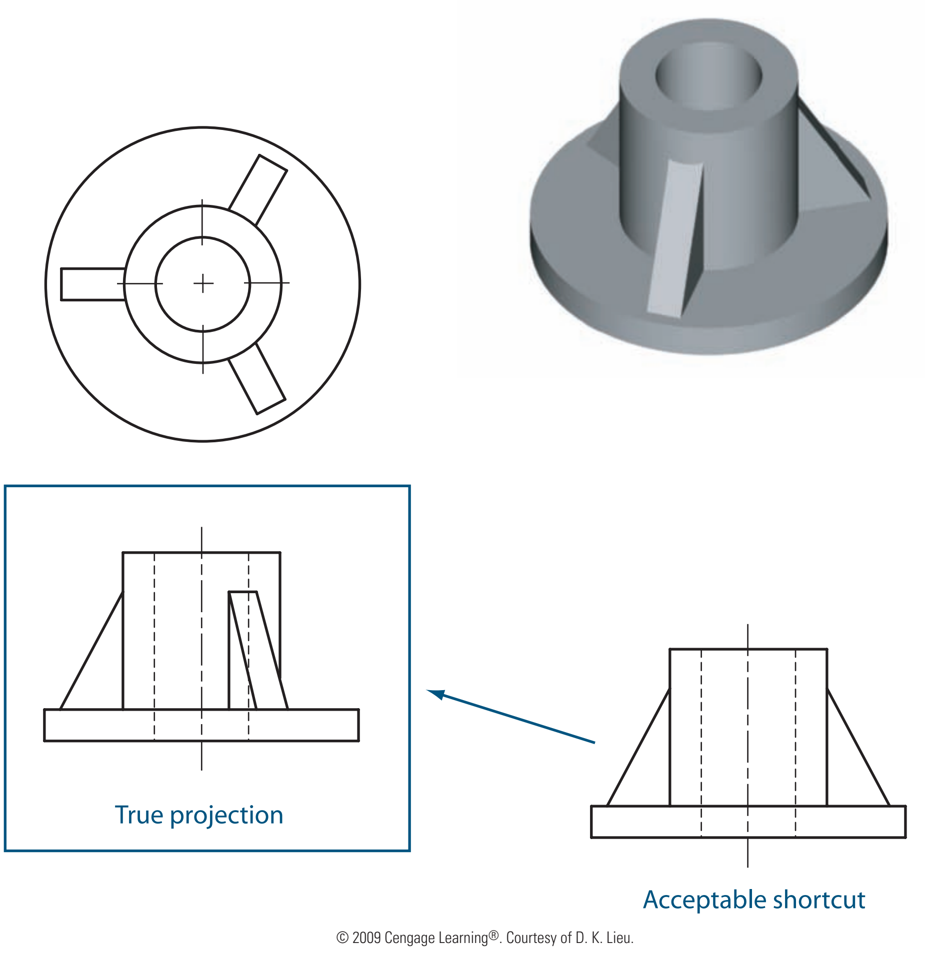

FIGURE 8.32. The true projection and an acceptable shortcut for an object with prominent symmetry. This property is emphasized by the use of a projected view that is modified to appear symmetrical.

8.06.06 Representation of Welds

Objects that contain welds, which are very common in civil engineering and some mechanical engineering applications, use special notation to specify the geometry of the weld. The use of this notation increases the speed of drawing creation with little loss of information. A simple object made from individual pieces that are welded together is shown in Figure 8.33. Even though a welded object is composed of two or more smaller pieces, it is common that such an object be fabricated at a single shop and delivered as a single unit. Thus, a single drawing showing the final welded configuration is often desirable.

Drawing the geometry of the welds on the multiview drawing takes time and effort, especially when the object contains many welds. So instead of the weld being drawn, a shorthand symbol is used. The notation specifies the geometry and locations of the weld, as well as any necessary modifications to the individual pieces in preparation for welding.

8.07 When Six Views Are Not Enough

It would seem that the six orthogonal views provided by the glass box would be sufficient to specify the geometry of any object. But the views are not sufficient for every object.

8.07.01 Features at Odd Angles

An example of an object requiring more than six views or nonstandard views is shown in Figure 8.34, where features are located on surfaces that are inclined or oblique.

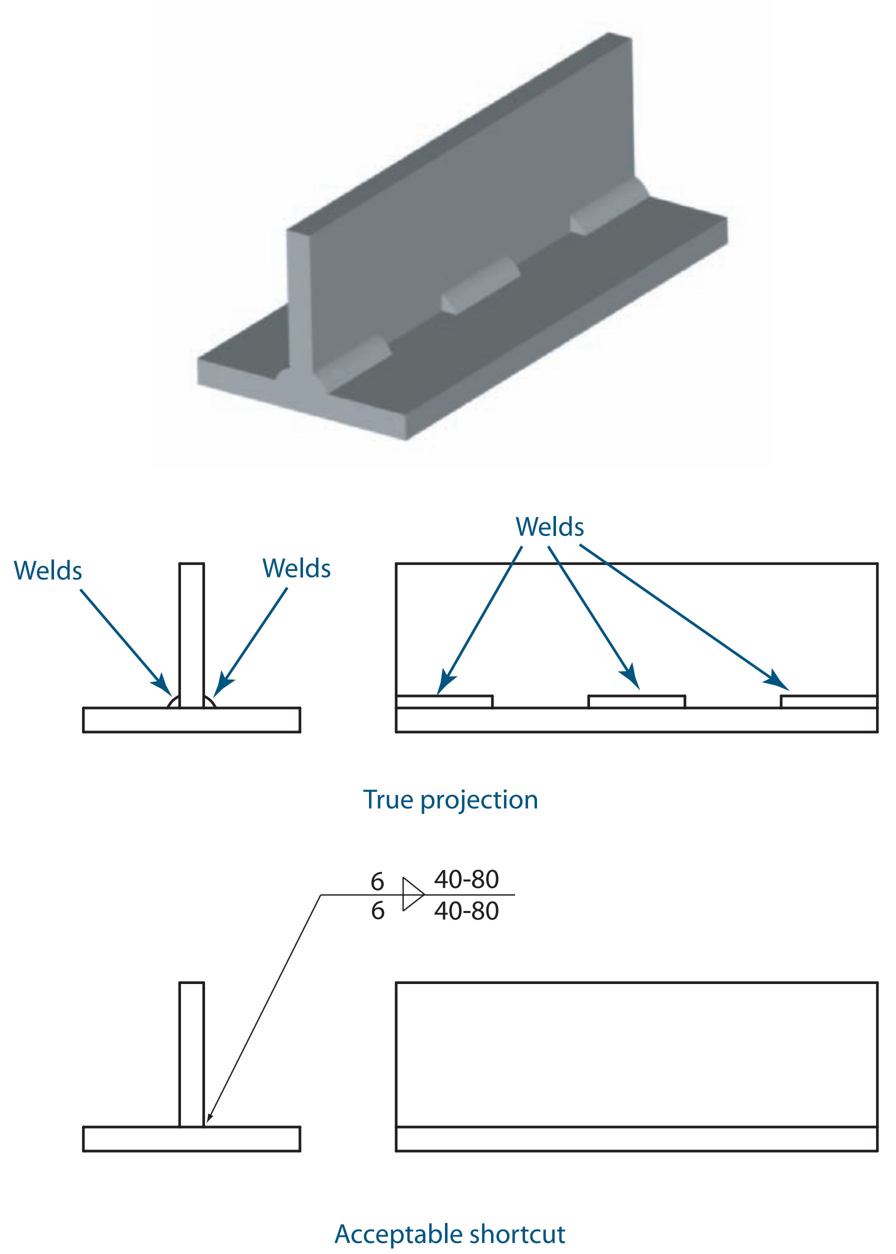

FIGURE 8.33. The acceptable presentation of two parts that are welded together to make a single part. The note specifies the size and location of the welds.

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For this object, none of the six standard orthogonal views would show these features in their true shape. A supplementary view, known as an auxiliary view, must be created before measurements can be specified for the feature represented in that view. Auxiliary views are covered in detail in Chapter 11 of this book.

8.07.02 Internal Features

Certain internal features, such as holes, bores, or cutouts with an irregular wall profile and details that are hidden from view, cannot be seen in any of the six standard views. An example of an object with internal features is the wheel shown in Figure 8.35. For this object, the geometry of spokes cannot be seen because the rim of the wheel obscures it. Although hidden lines can sometimes be used to show such features, those features will appear more clearly in a cutaway, or section view. Section views of all sorts are covered in detail in Chapter 10 of this book.

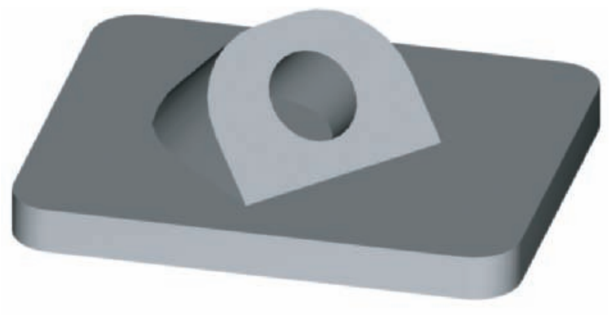

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FIGURE 8.34. An object such as this one cannot be fully described by the six standard views.

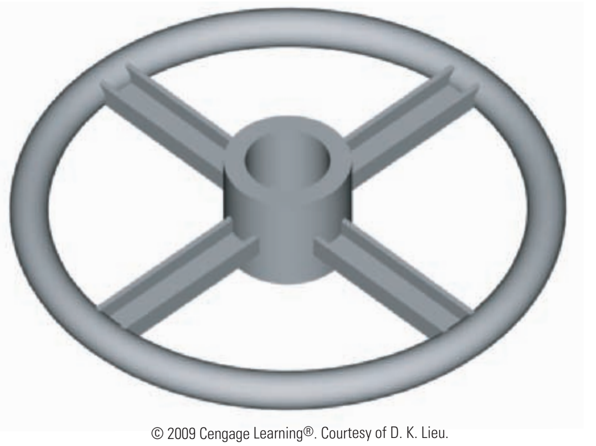

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FIGURE 8.35. An object with internal features such as this one cannot be fully described by the six standard views.

8.08 Considerations for 3-D Modeling

The proliferation of 3-D solids modeling software, especially in mechanical engineering applications, has made the process of creating drawings much easier than in the past. Typically, with solids modeling software, objects are initially modeled as a series of protrusions and cuts to create their 3-D graphical representation. The solids modeling software creates a mathematical model of the geometry from which the projections of the object are used to create drawings. The model can be scaled and rotated for viewing from any orientation direction. Once the solids model is created, it usually is simple to specify the viewing directions needed for the software to create isometric and other pictorial views. It also is easy to extract a front view, side view, or any of the other orthogonal views from a solids model.

The ease with which pictorials and multiview drawings can be created from a solids model has many advantages, but also some disadvantages. The greatest advantage is the speed and accuracy with which orthogonal views can be created. With most software, additional views can be created by specifying the location of the viewing plane and then picking a location on the drawing where the additional view is to appear. Usually, this is done by striking a few keys on a keyboard or making a few clicks with a mouse or another pointing device. The time required to produce the additional view is usually only a few seconds. Hidden lines can be added or removed for individual features or for an entire view. Also, accurate orthogonal projections of features that were previously represented by shortcut practices, such as small cutouts in curved surfaces or thin symmetric features, are easily created. In fact, with most software, it would be difficult to create a view that is *not* an accurate projection.

But there is a disadvantage to having so much ease in creating drawings. Remember, the original process of manually creating projected views from pictorials and mental images and pictorials from projected views depended on the drawer's developed skills of spatial reasoning and mental imaging. When software makes the process of creating drawings too automatic, a person may not be able to apply these skills in the absence of the software because she did not develop adequate drawing skills. In other words, the person may have become too dependent on the software. That person, when faced with a multiview drawing in the shop, may not be able to create a mental image of the object or may not develop the skills necessary to interpret standard drawings. Eventually, the person will develop these skills, but it may require experience with many solids models and their drawings. Whether you are working with instruments, 2-D CAD, or solids modelers, the key to successful development of mental imaging skills is simply to practice—a lot.

8.09 Chapter Summary

Orthogonal projection and the use of the standard views of an object are accepted nationally and internationally as the formal means of creating and presenting images for the purpose of producing the original object. Constructed correctly, these views are used to re-create the same 3-D object, no matter who is viewing the images. Care must be taken to ensure that the rules for view creation, orientation, scale, and alignment are followed. Hidden lines are used for completing the description or for additional emphasis of certain features on the object. Extra views are used as necessary for completing the description of these features. From these formal views, the original 3-D object can be re-created. When done successfully, whether you are the person making the drawing or the person reading the drawing, you will find that the interpretation of the views and the object they represent are the same.

8.10 GLOSSARY OF KEY TERMS

adjacent views: Orthogonal views presented on a single plane that are created immediately next to each other.

centerline: A series of alternating long and short dashed lines used to identify an axis of rotational symmetry.

centermark: A small right-angle cross that is used to identify the end view of an axis of rotational symmetry.

first-angle projection: The process of creating a view of an object by imprinting its image, using orthogonal projection, on an opaque surface behind that object.

glass box: A visualization aid for understanding the locations and orientations of images of an object produced by third-angle projection on a drawing. The images of an object are projected, using orthogonal projection, on the sides of a hypothetical transparent box that is then unfolded into a single plane.

hidden lines: The representation, using dashed lines, on a drawing of an object of the edges that cannot be seen because the object is opaque.

multiple views: The presentation of an object using more than one image on the same drawing, each image representing a different orientation of the object.

multiview: Refers to a drawing that contains more than one image of an object and whose adjacent images are generated from orthogonal viewing planes.

orthogonal projection: The process by which the image of an object is created on a viewing plane by rays from the object that are perpendicular to that plane.

preferred configuration: The drawing presentation of an object using its top, front, and right-side views.

six standard views (or six principal views): The drawing presentation of an object using the views produced by the glass box (i.e., the top, front, bottom, rear, left-side, and right-side views).

third-angle projection: The process of creating a view of an object by imprinting its image, using orthogonal projection, on a translucent surface in front of that object.

viewing plane: A hypothetical plane between an object and its viewer onto which the image of the object, as seen by the viewer, is imprinted.

8.11 QUESTIONS FOR REVIEW

1. What is orthogonal projection?
2. What are the advantages and disadvantages of using pictorial images, such as isometric images, for the graphical representation of an object?
3. What is a multiview presentation?
4. What are the advantages and disadvantages of using a multiview presentation for the graphical representation of an object?
5. How are different views located with respect to each other on the same drawing?
6. Why should features be aligned between views in a multiview presentation?
7. Why is it important that different views have the same scale?
8. What are the advantages of having features of an object aligned between views?

9. What are the standard (or principal) views?
10. What is the preferred configuration?
11. When should extra orthogonal views be used?
12. When should hidden lines be used?
13. When should hidden lines not be shown?
14. What is the difference between first- and third-angle projection?
15. When can the rules of orthogonal projection be bent? What are the advantages of doing so?

8.12 PROBLEMS

1. From the isometric pictorials shown in Figure P8.1, create accurate multiview drawings with a sufficient number of views to specify all details of the object completely. Do not use hidden lines.

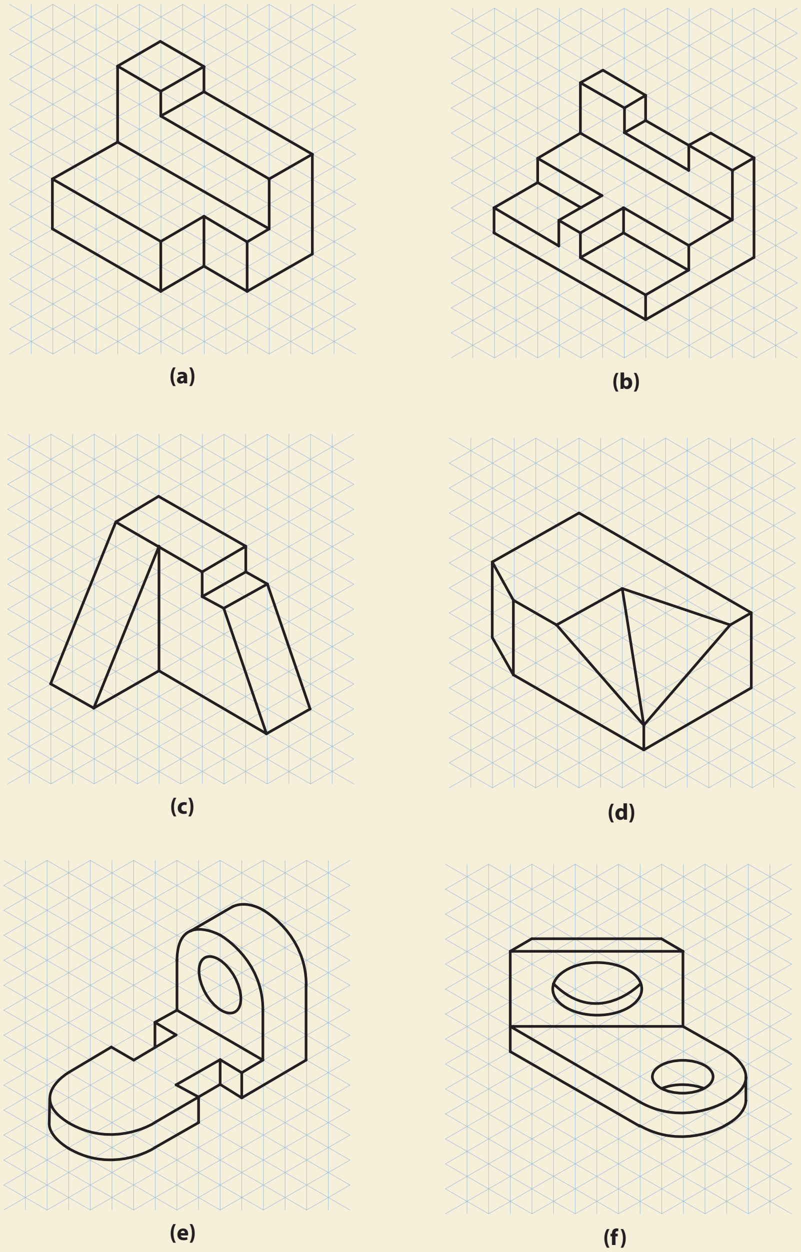

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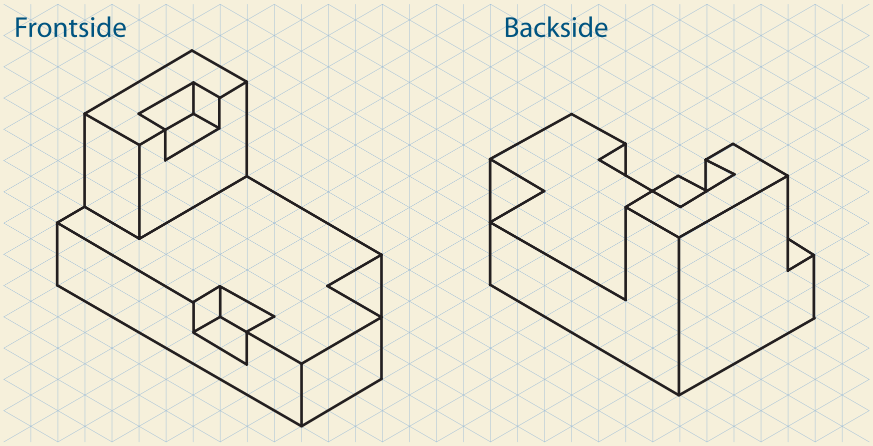
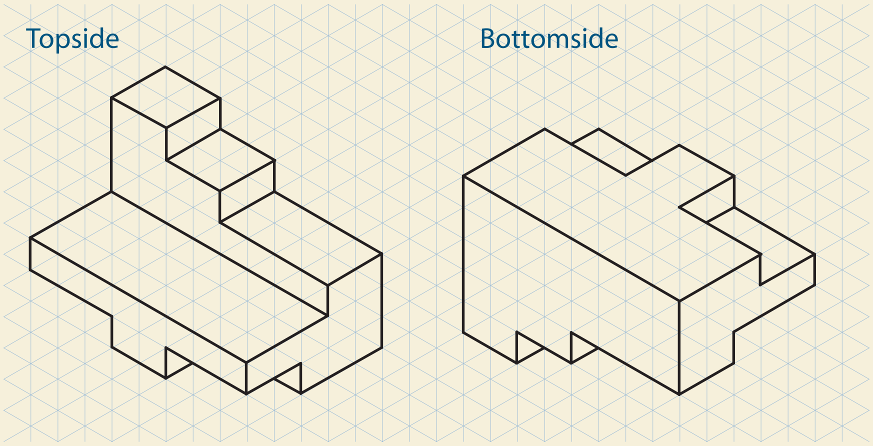
FIGURE P8.1.

8.12 PROBLEMS (CONTINUED)

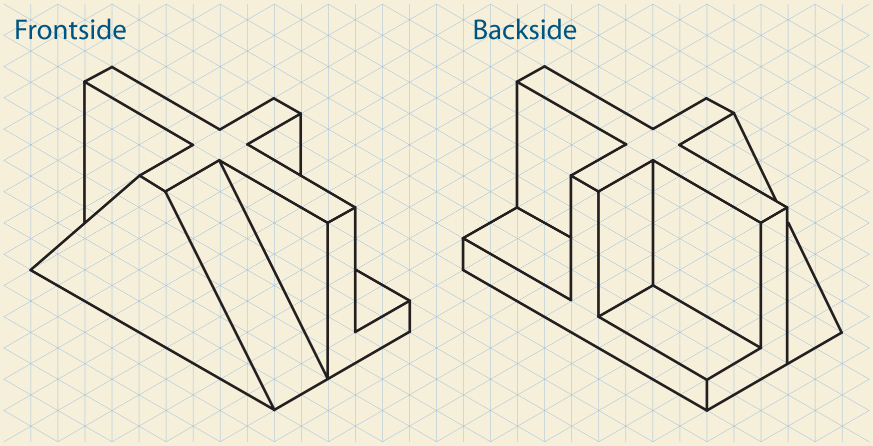
2. From the isometric pictorials shown in Figure P8.2, create accurate multiview drawings in the preferred format of front, top, and right-side views. Use hidden lines as necessary to specify all details of the object completely.



(a)



(b)



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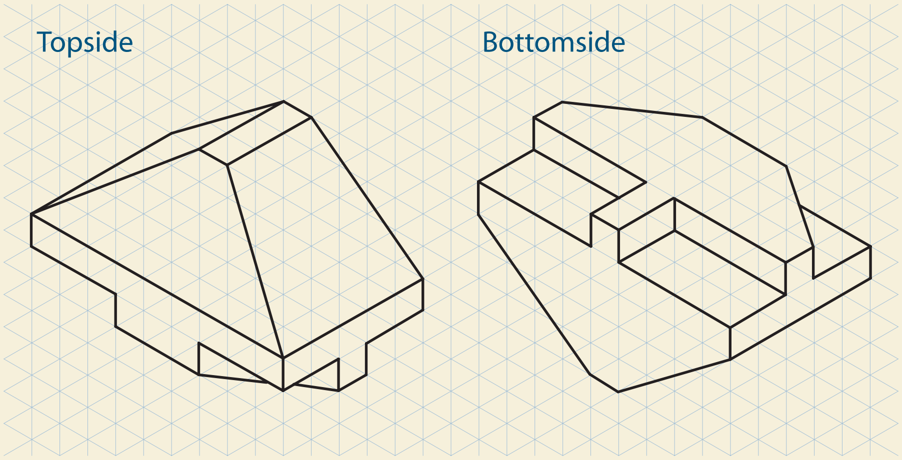
8.12 PROBLEMS (CONTINUED)



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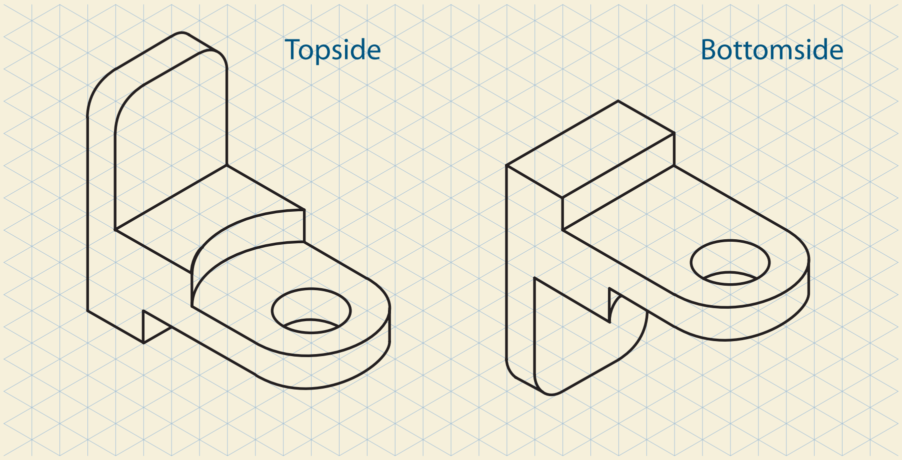

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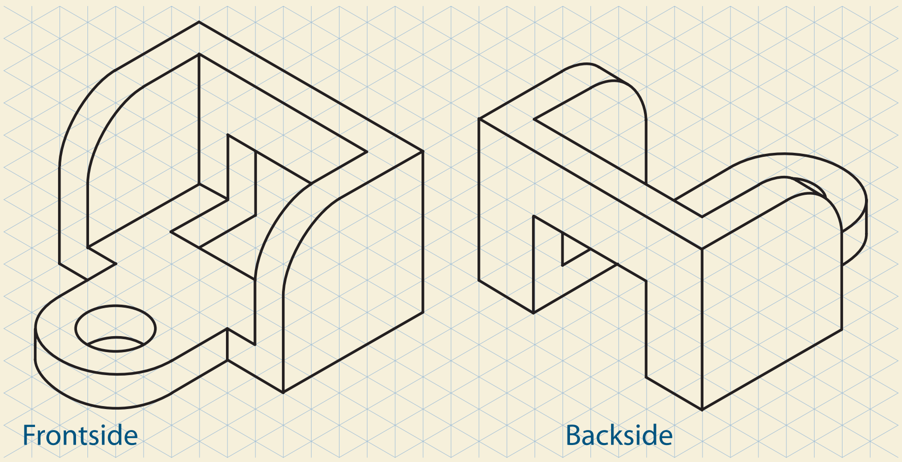
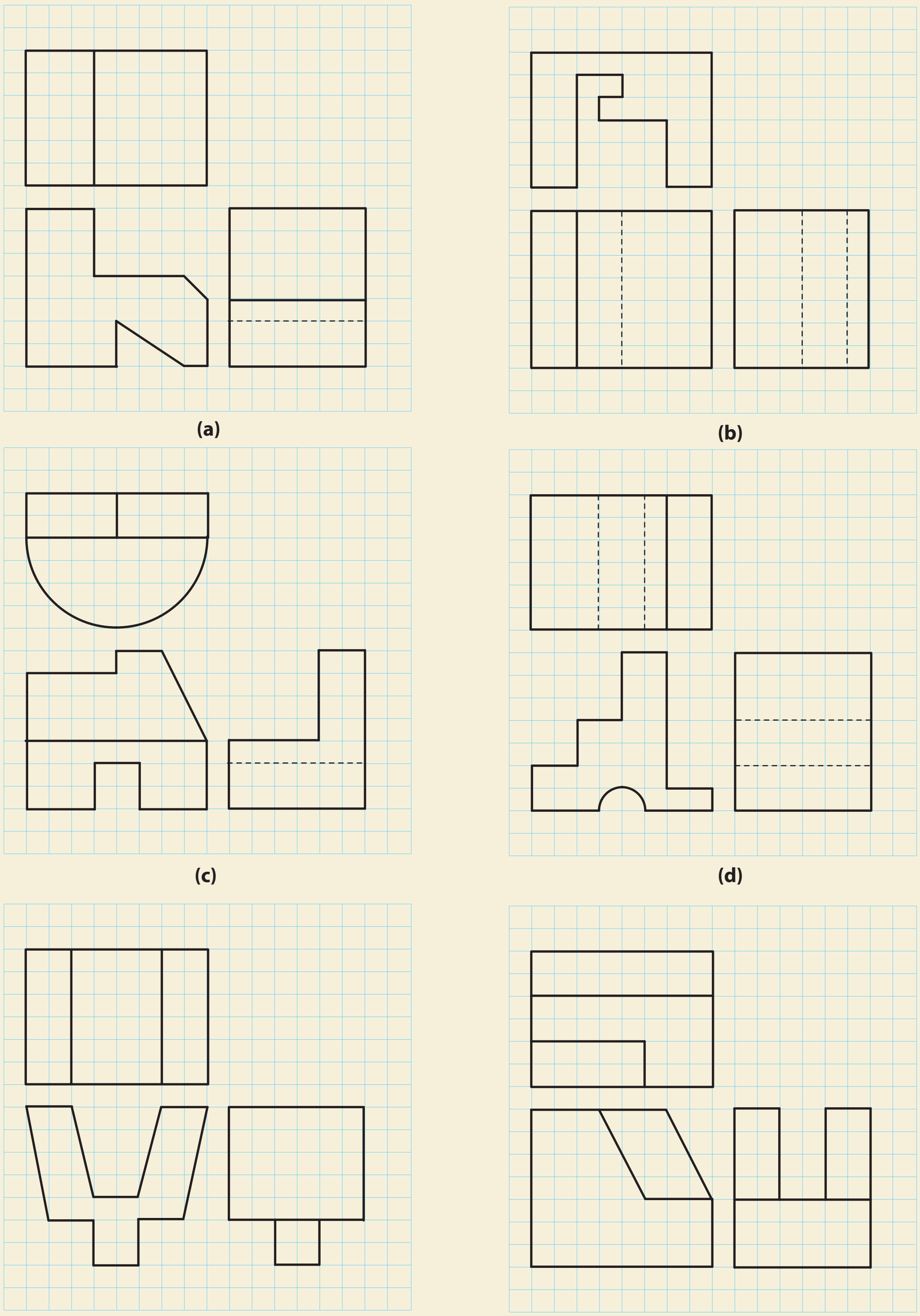


(f)

FIGURE P8.2.

8.12 PROBLEMS (CONTINUED)

3. Each set of multiview drawings shown in Figure P8.3 may have visible or hidden lines missing. Add the missing lines to the drawing.



(a) (b) (c) (d) (e) (f)

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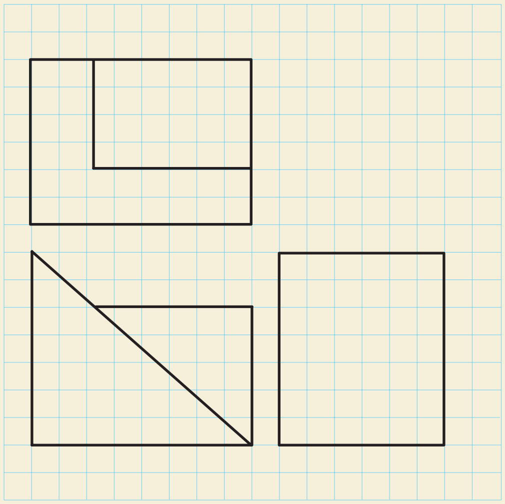
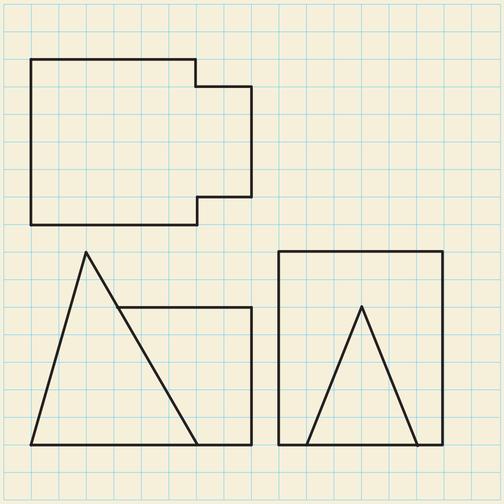
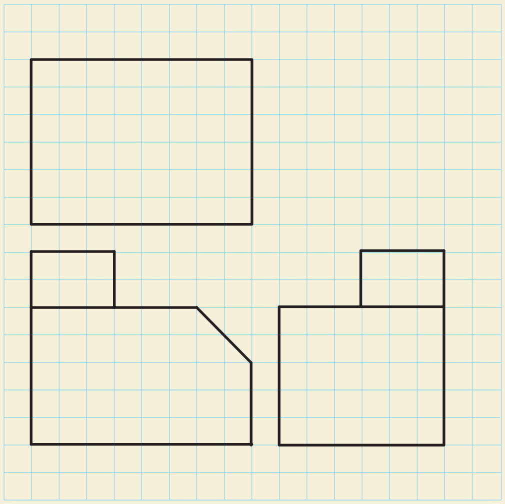
8.12 PROBLEMS (CONTINUED)



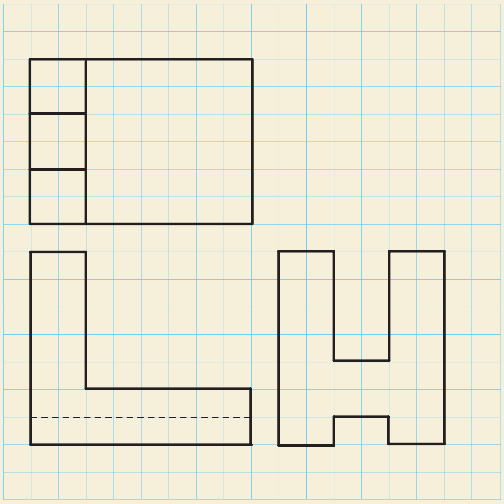
(g)



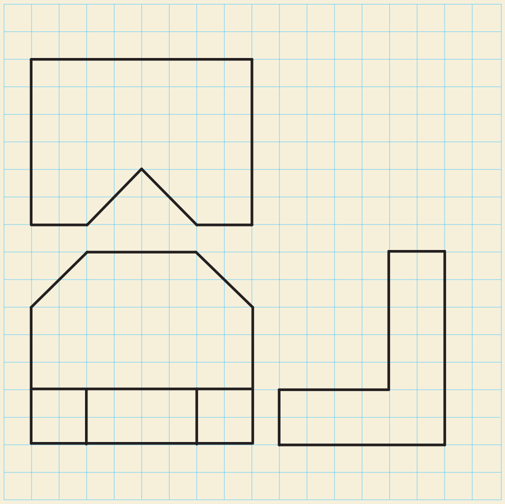
(h)



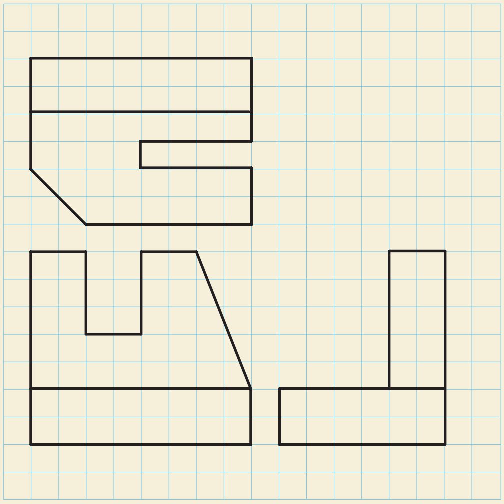
(i)



(j)



(k)



(l)

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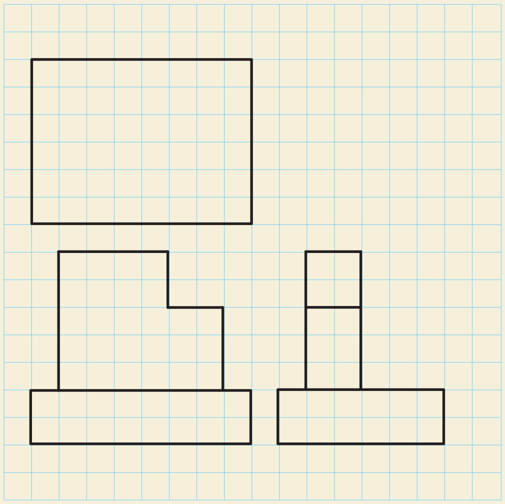
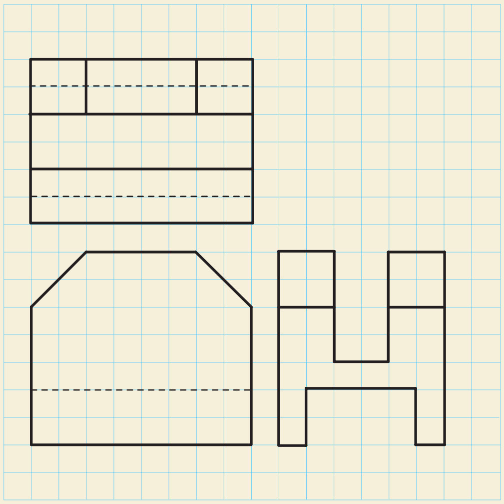
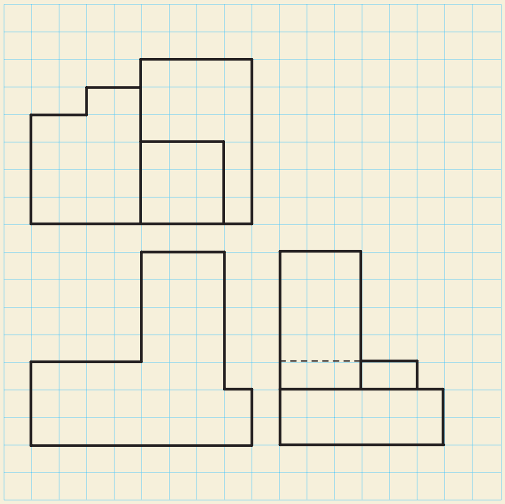
8.12 PROBLEMS (CONTINUED)



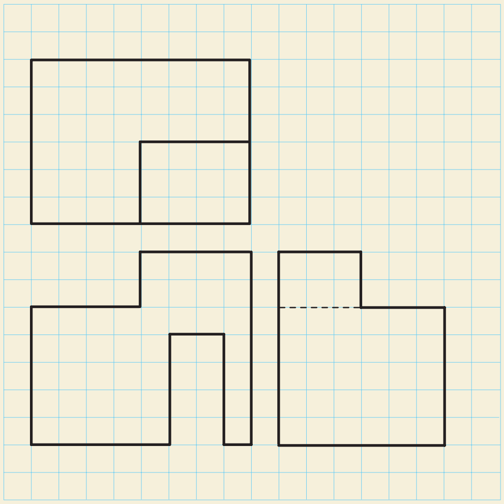
(m)



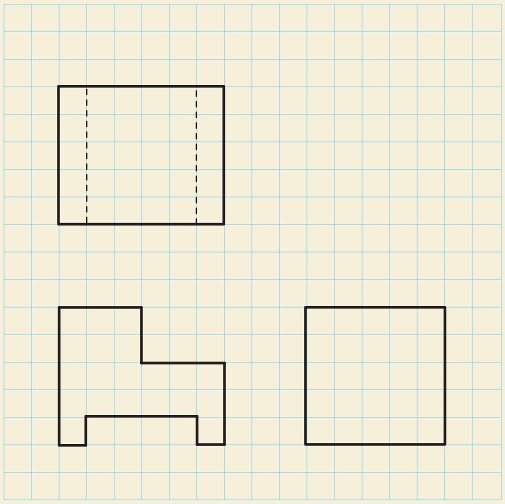
(n)



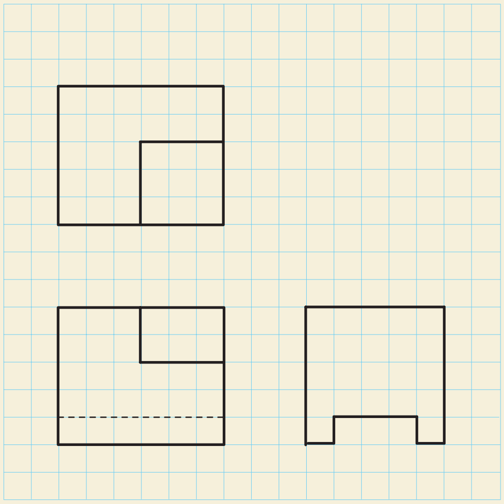
(o)



(p)



(q)



(r)

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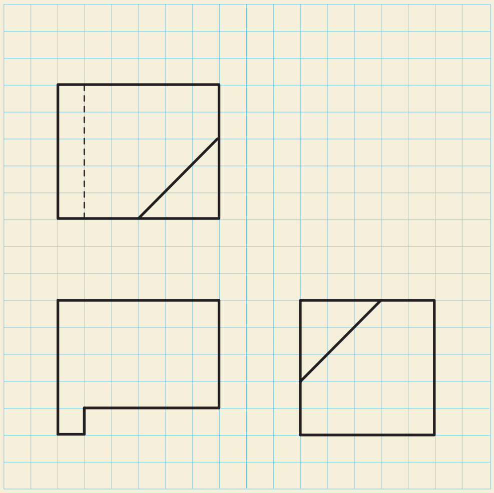
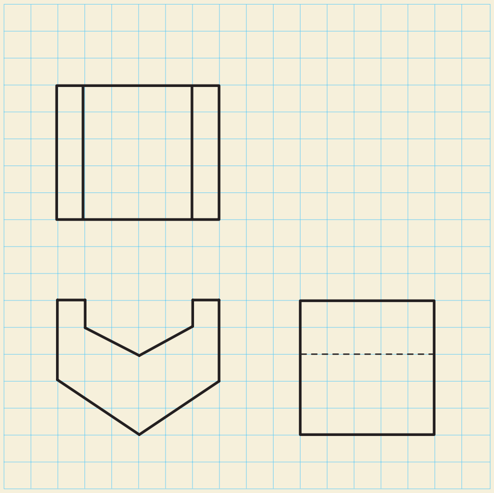
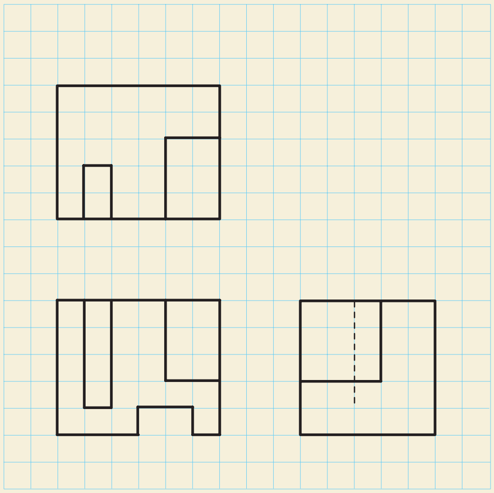
8.12 PROBLEMS (CONTINUED)



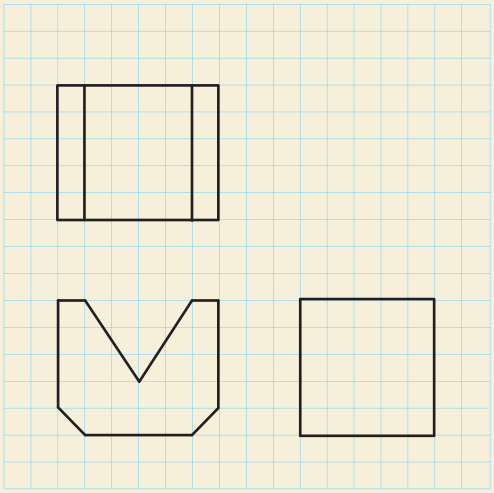
(s)



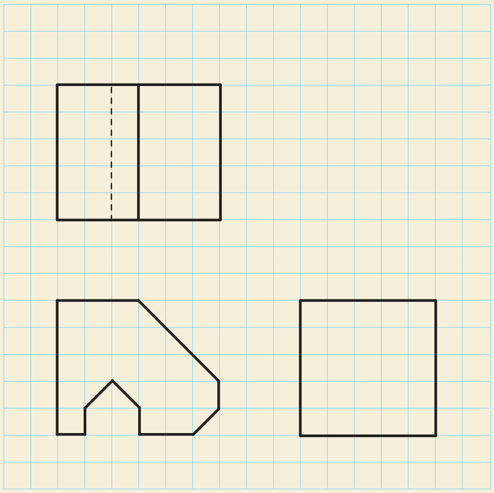
(t)



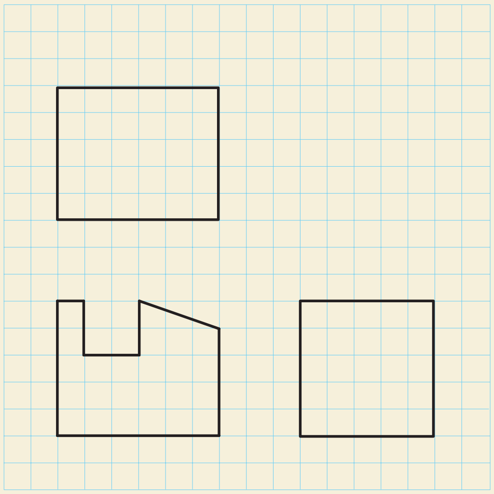
(u)



(v)



(w)



(x)

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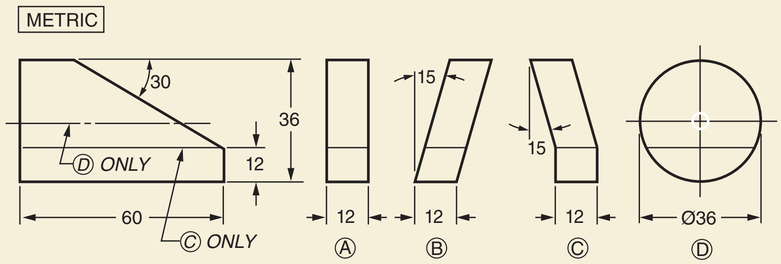
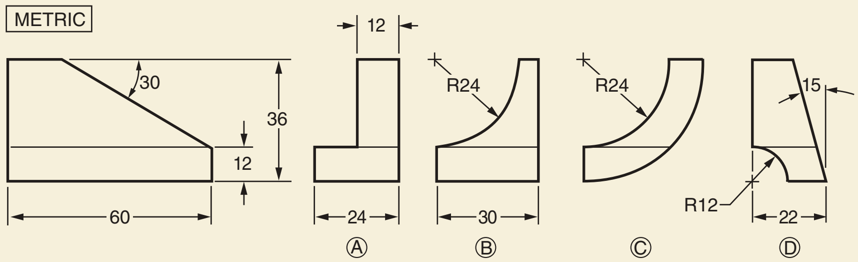
FIGURE P8.3. Add the missing solid or hidden lines for the multiview drawing.

8.12 PROBLEMS (CONTINUED)

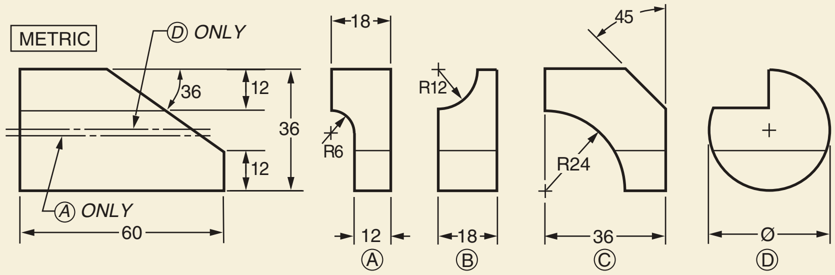
4. For each front view shown in Figure P8.4, draw the top view (in the correct scale, location, and orientation) that corresponds to each of the possible side views that are shown.



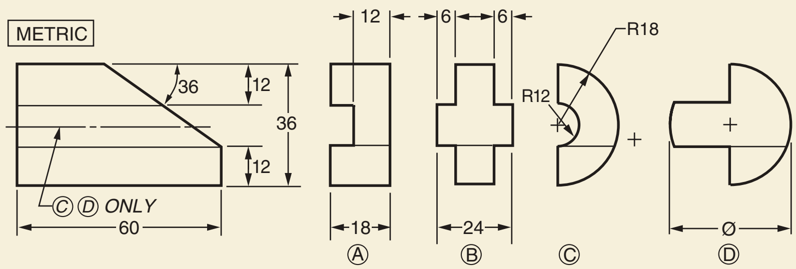
(a)



(b)

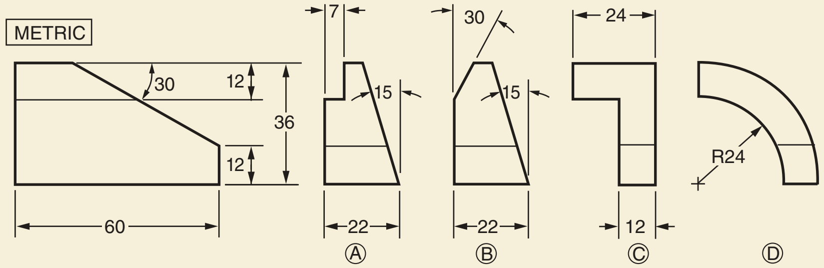


(c)

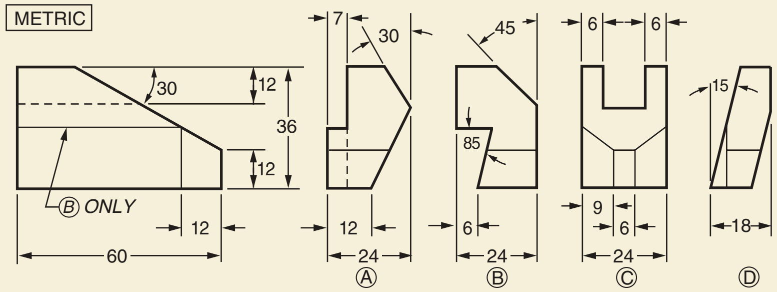


(d)

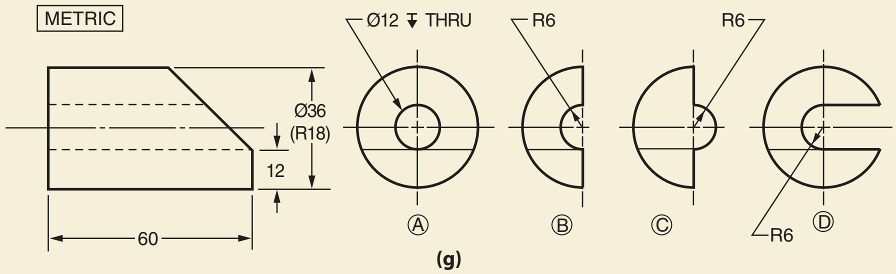
8.12 PROBLEMS (CONTINUED)



(e)



(f)



(g)

FIGURE P8.4.

8.12 PROBLEMS (CONTINUED)

5. For each set of multiview drawings shown in Figure P8.5, add a third view to the drawing in its correct location, size, and orientation.

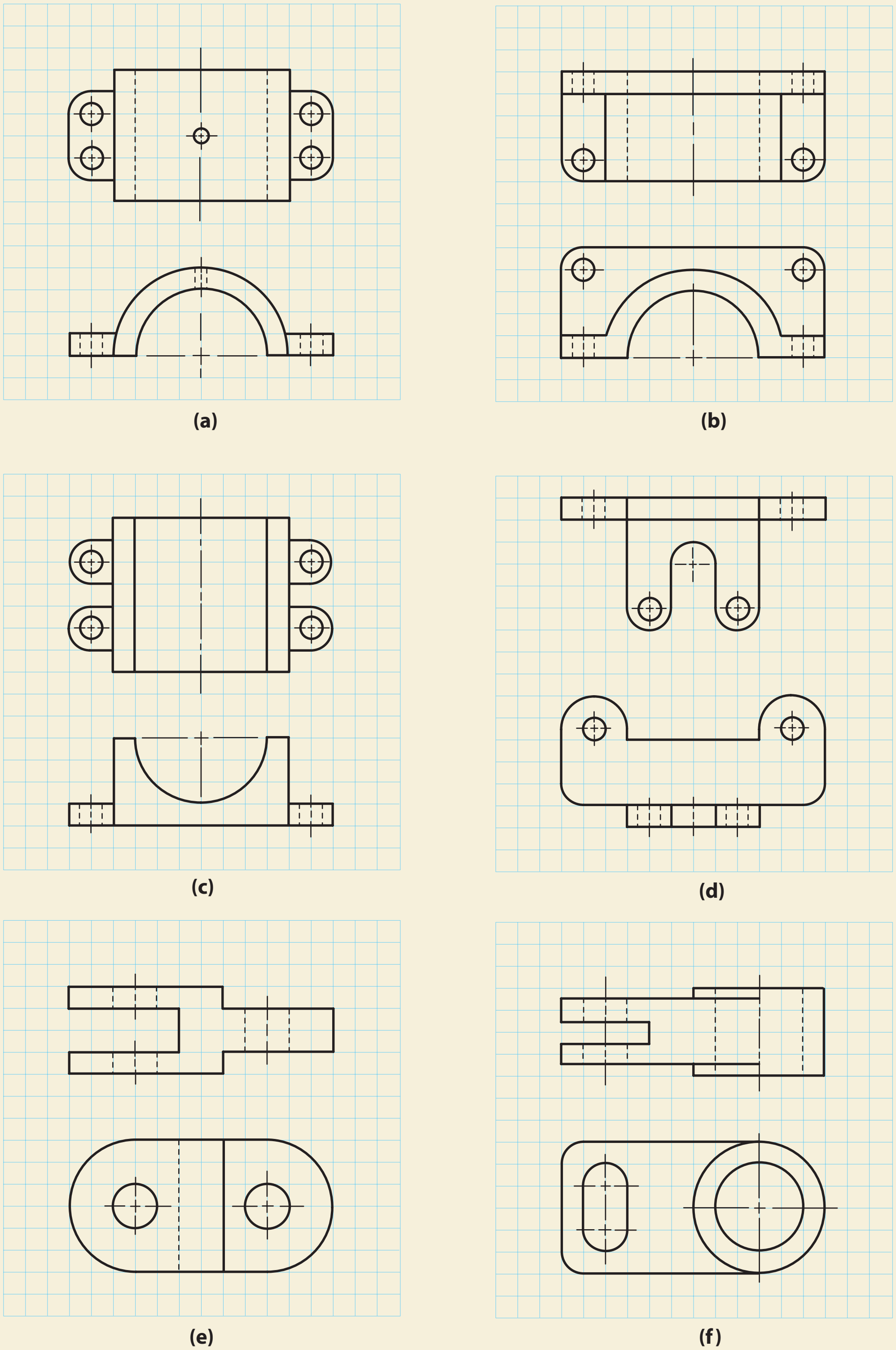
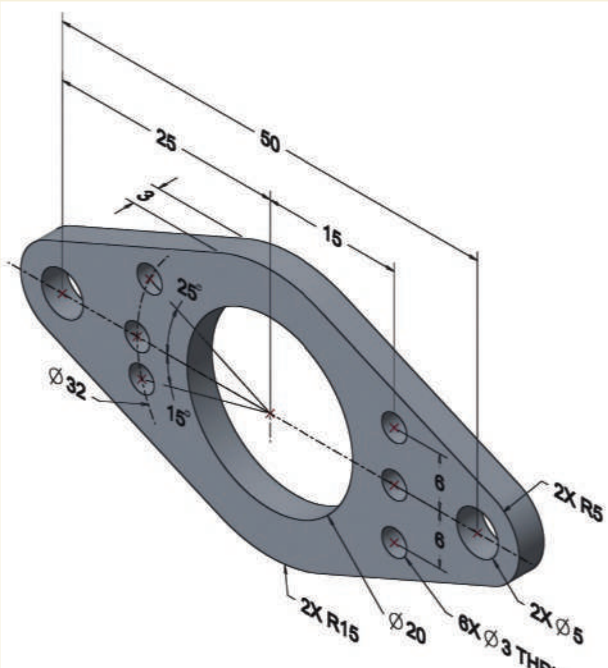


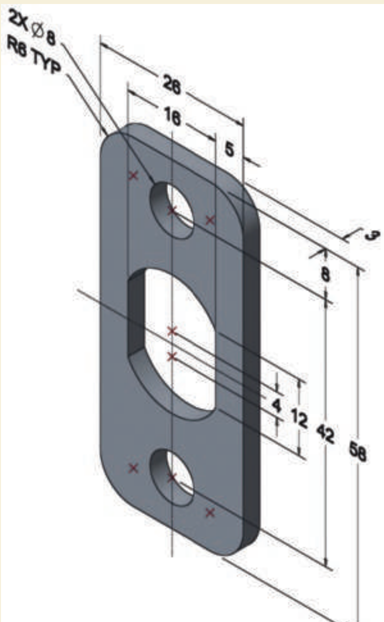
FIGURE P8.5. Add a third view to the multiview drawing.

8.12 PROBLEMS (CONTINUED)

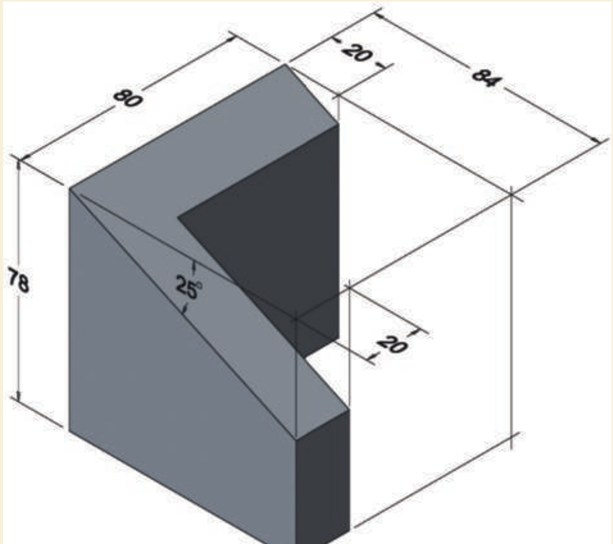
6. Create correctly scaled multiview orthogonal drawings of the objects shown in Figure P8.6. Show at least the front, top, and right-side views. Include hidden lines. Recommend when additional views would be useful to clarify the presentation and add these views to the drawing.



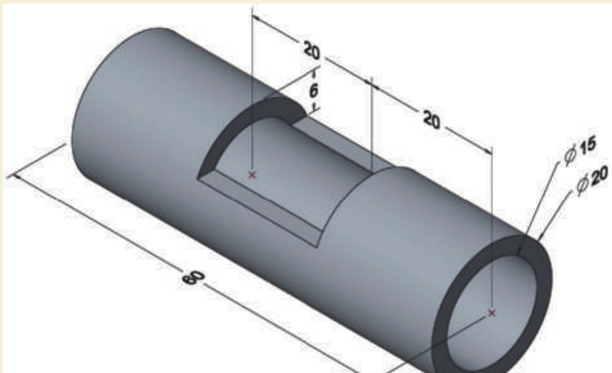
(a)



(b)

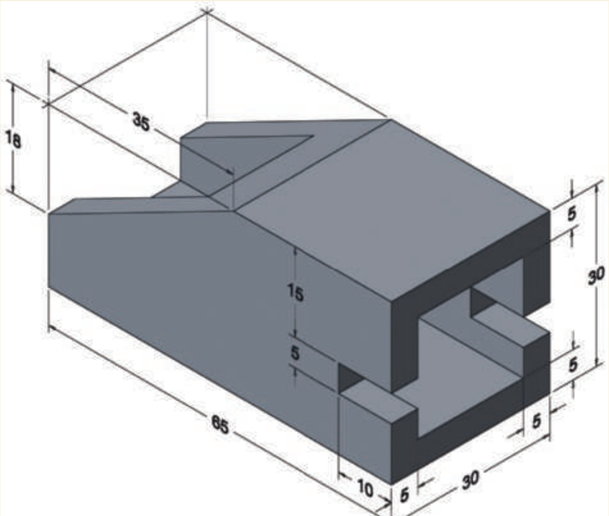


(c)



(d)

8.12 PROBLEMS (CONTINUED)



(e)

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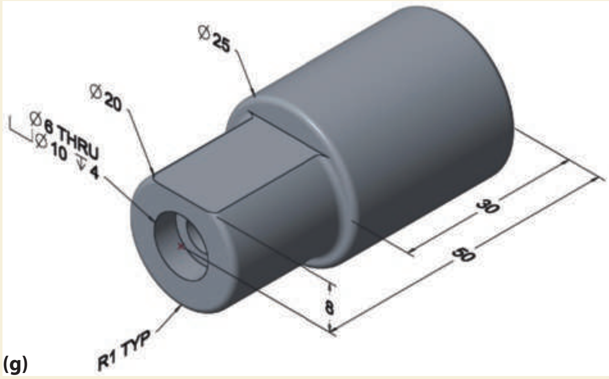

(g)

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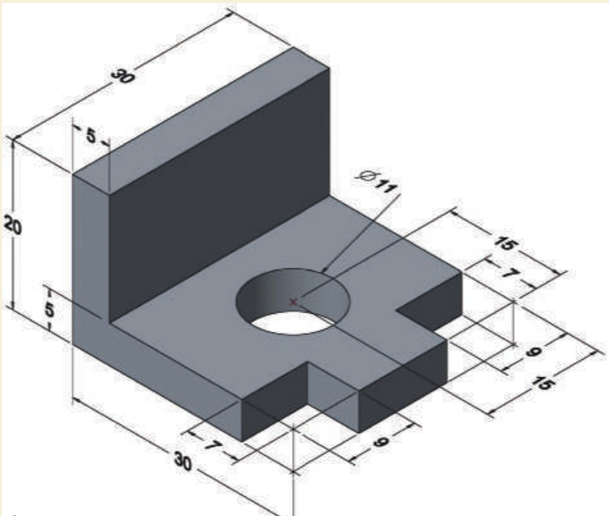

(f)

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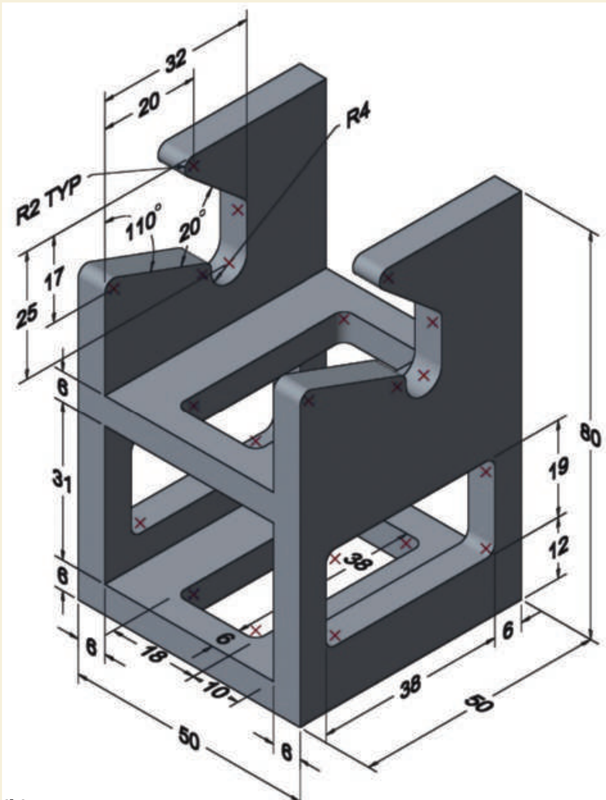

(h)

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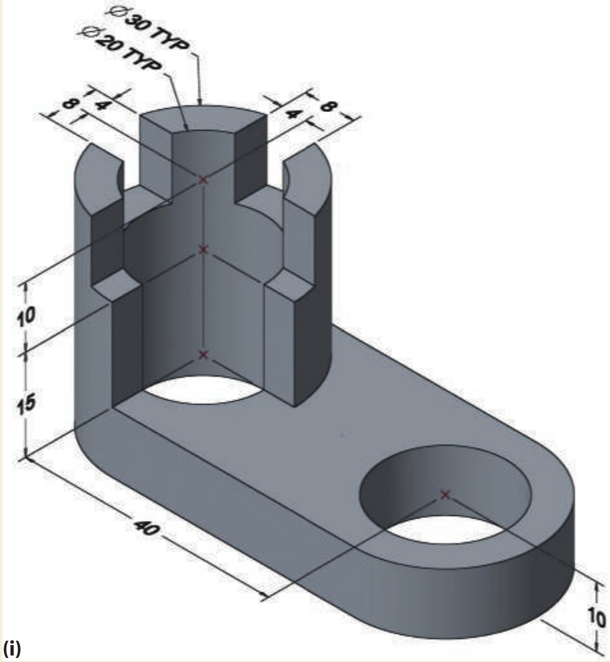
8.12 PROBLEMS (CONTINUED)



(i)

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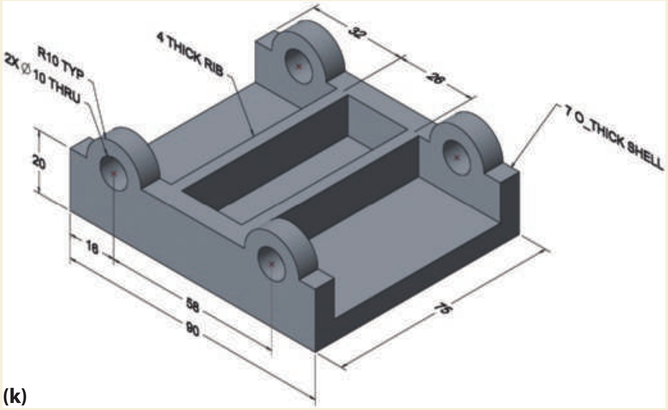

(k)

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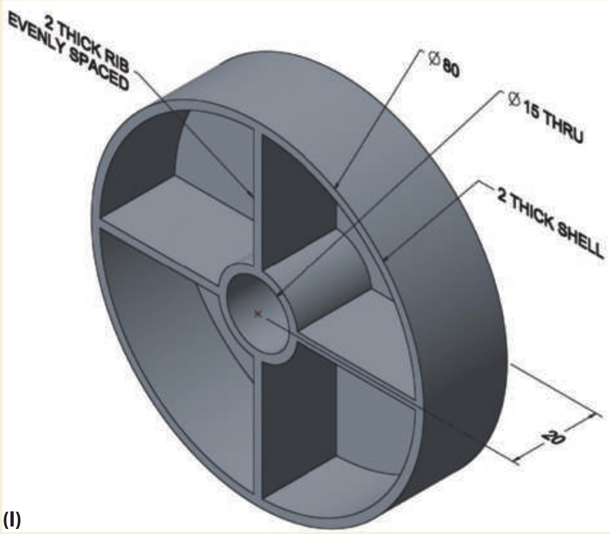

(l)

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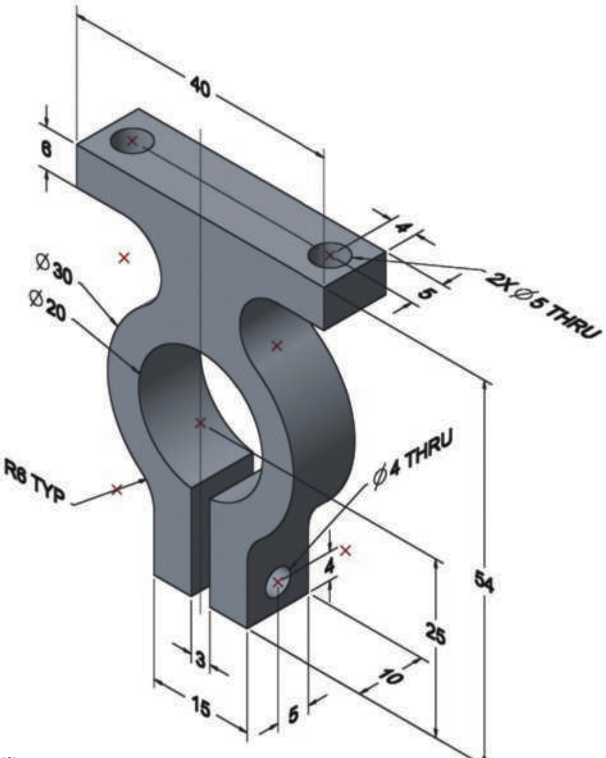

(j)

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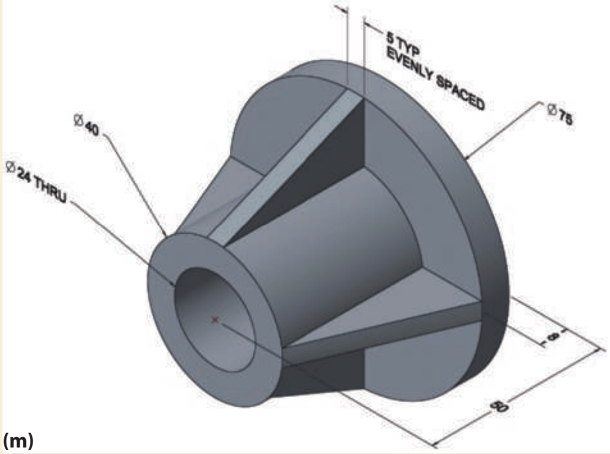

(m)

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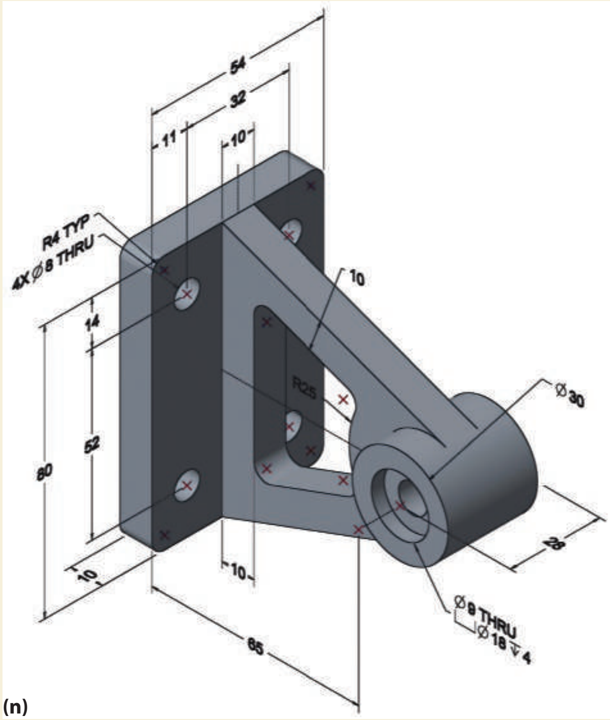
8.12 PROBLEMS (CONTINUED)



(n)

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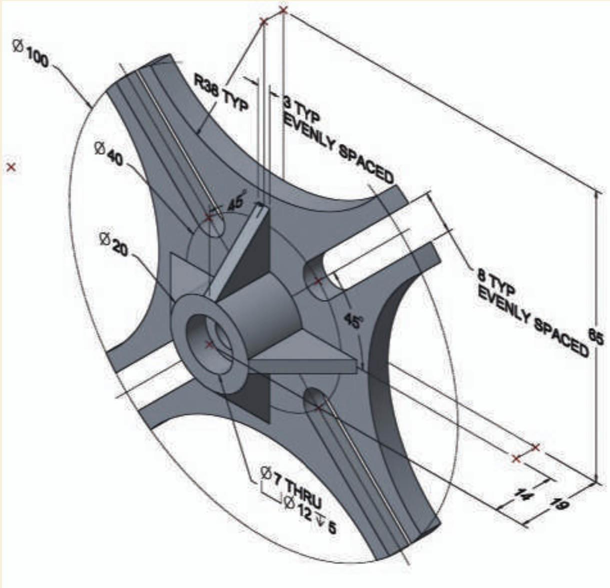

(p)

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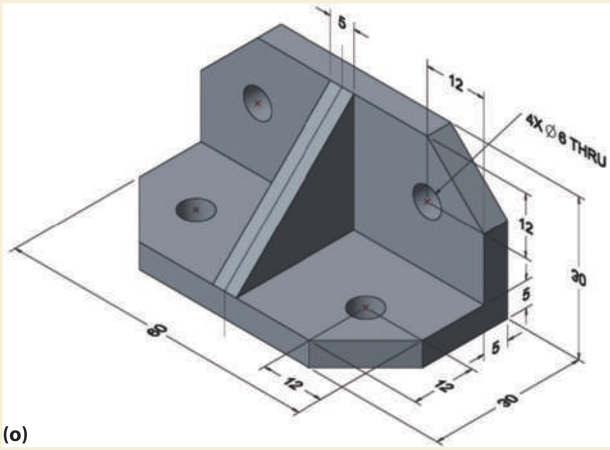

(o)

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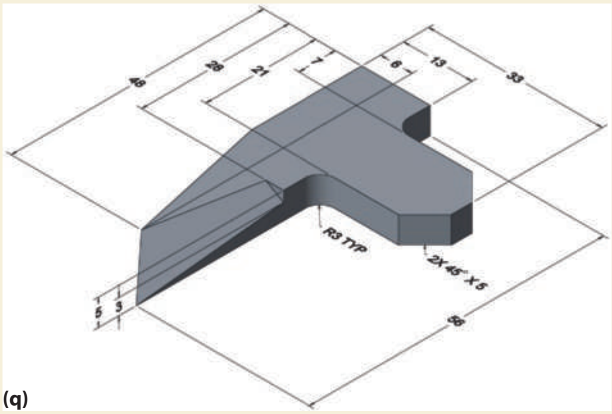

(q)

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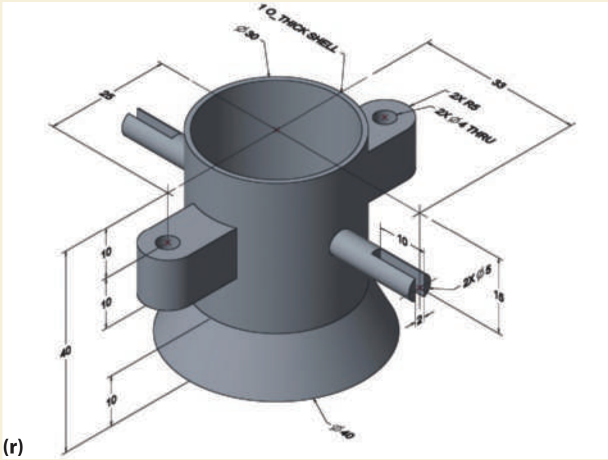
8.12 PROBLEMS (CONTINUED)



(r)

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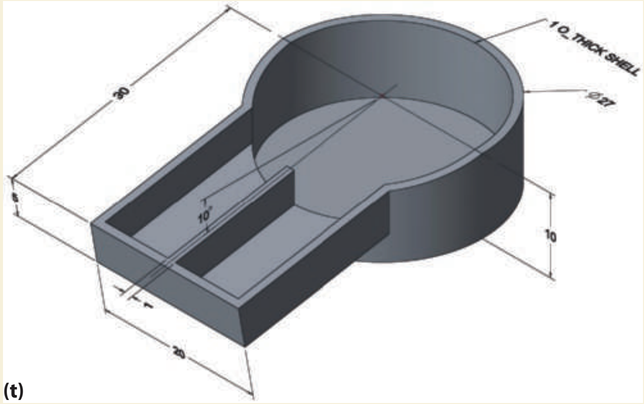

(t)

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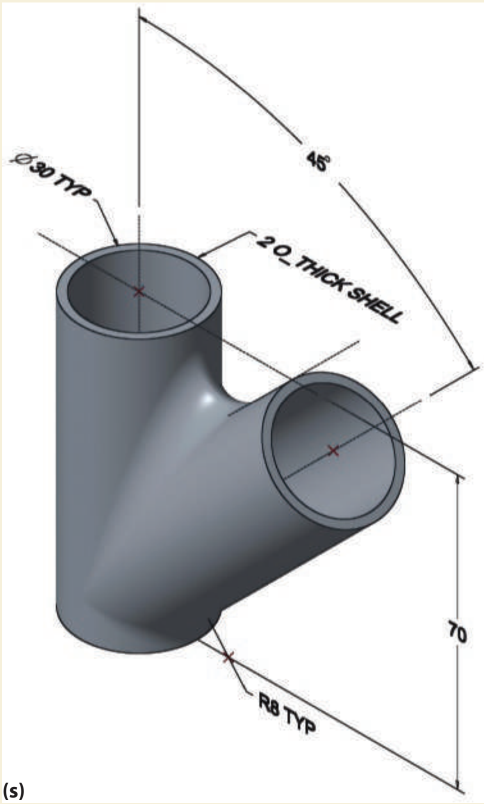

(s)

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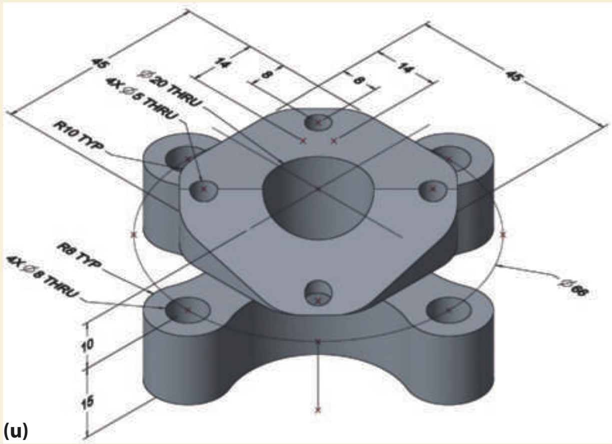

(u)

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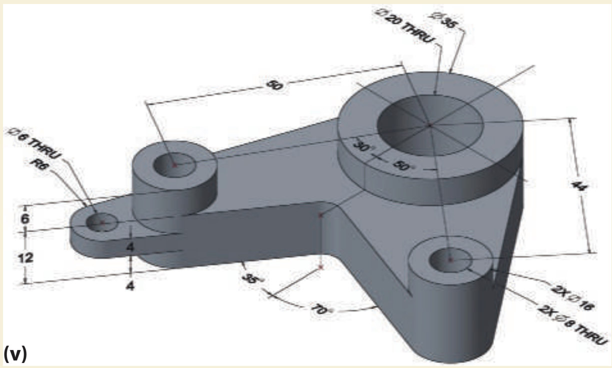

(v)

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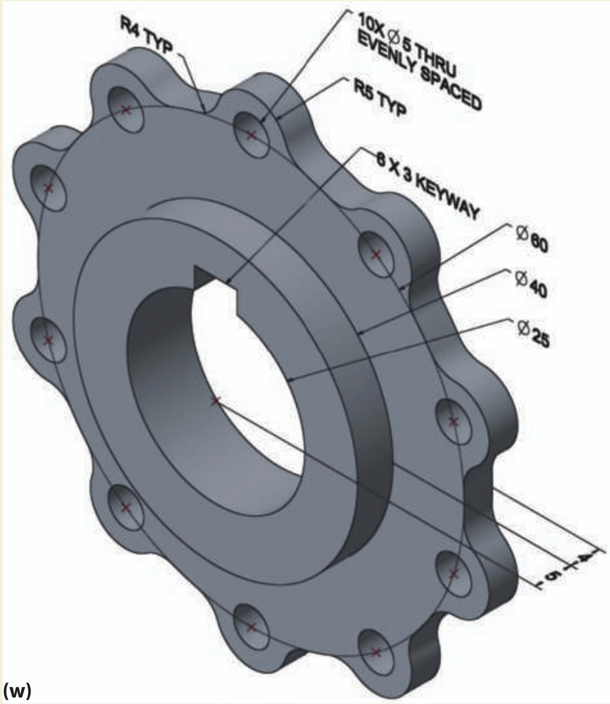
8.12 PROBLEMS (CONTINUED)



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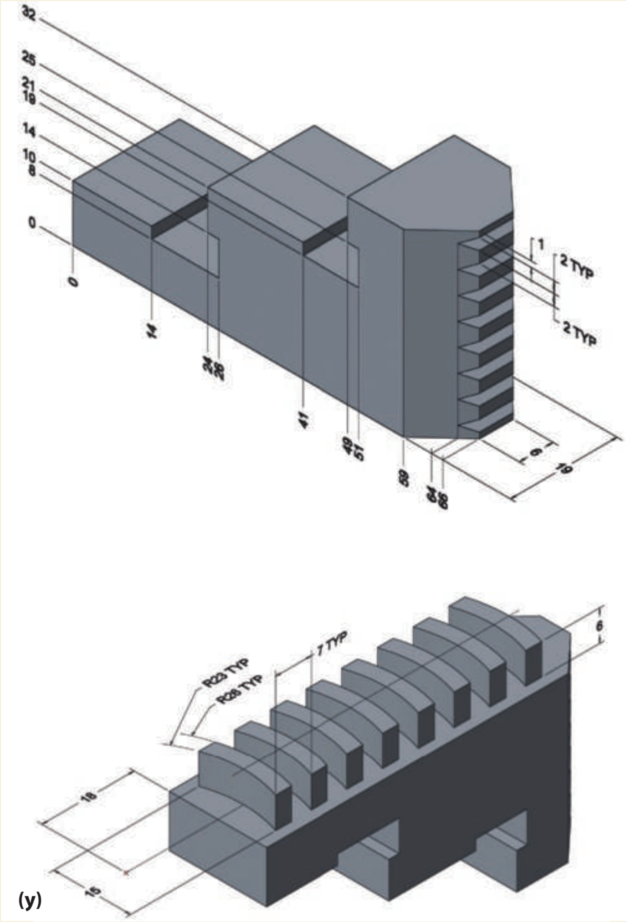

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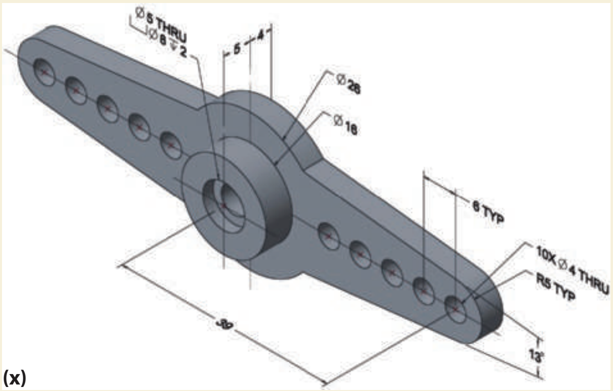

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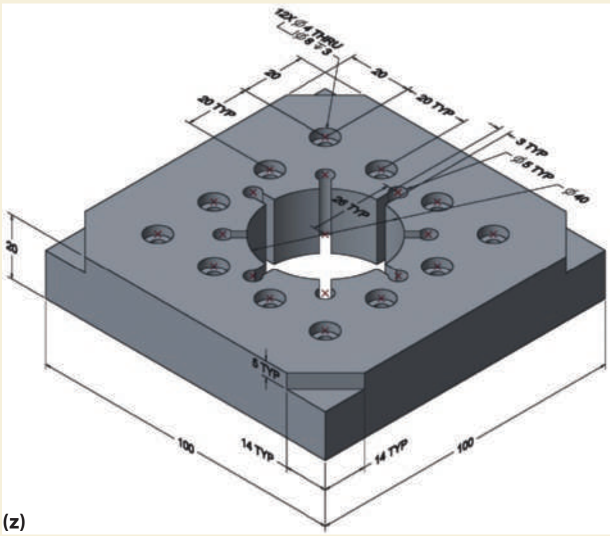

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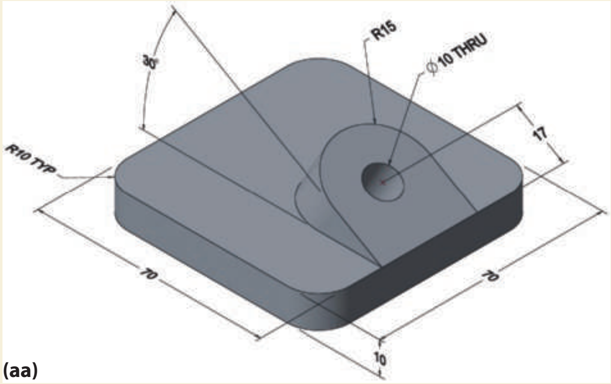
8.12 PROBLEMS (CONTINUED)



(aa)

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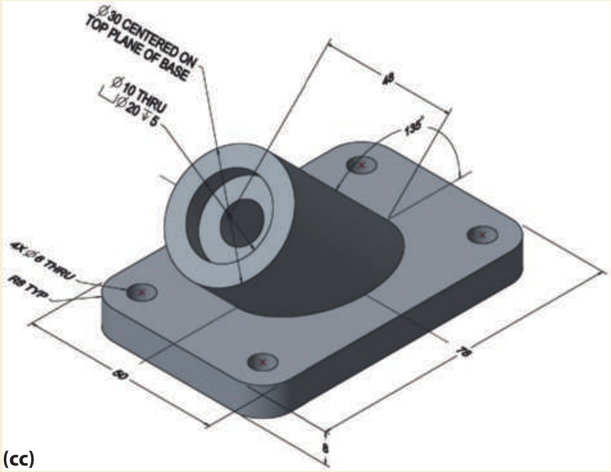

(cc)

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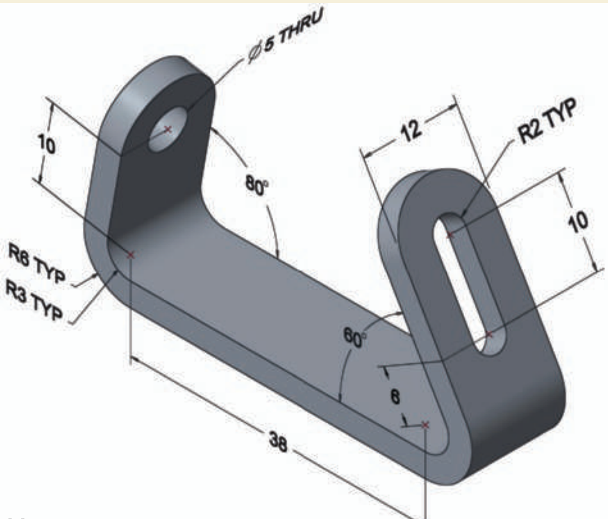

(bb)

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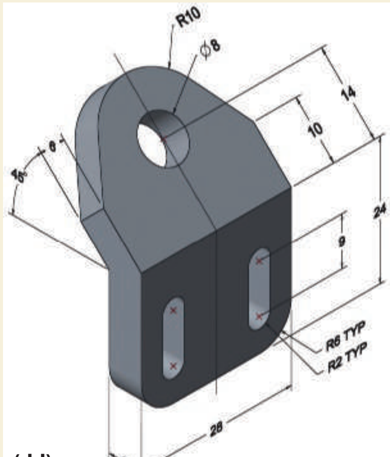

(dd)

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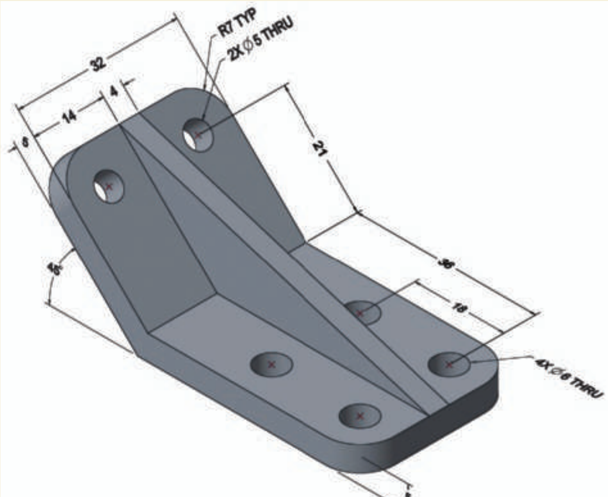
8.12 PROBLEMS (CONTINUED)



(ee)

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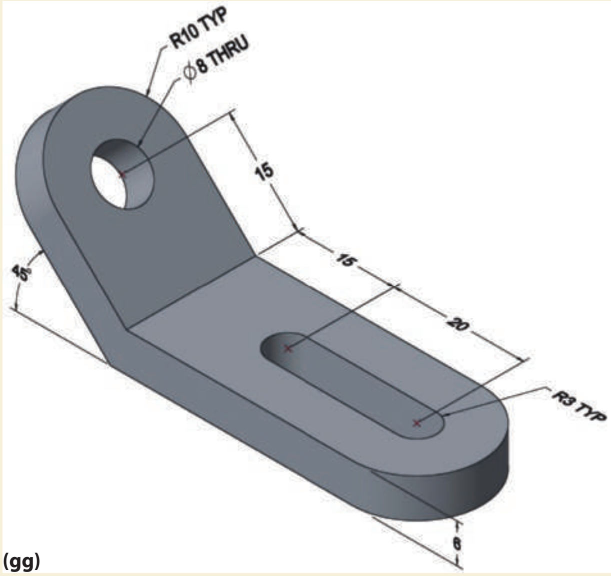

(gg)

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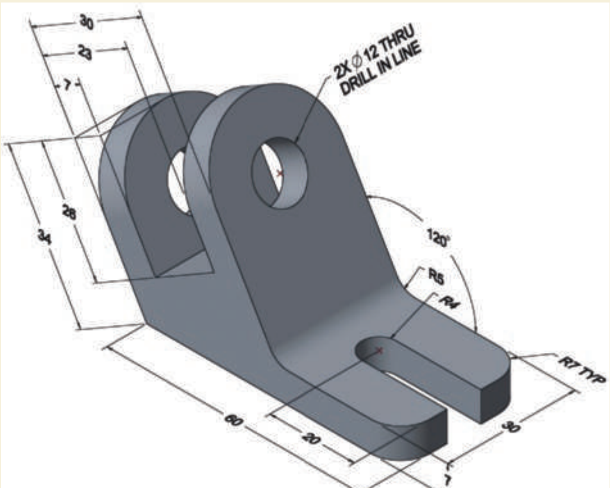

(ff)

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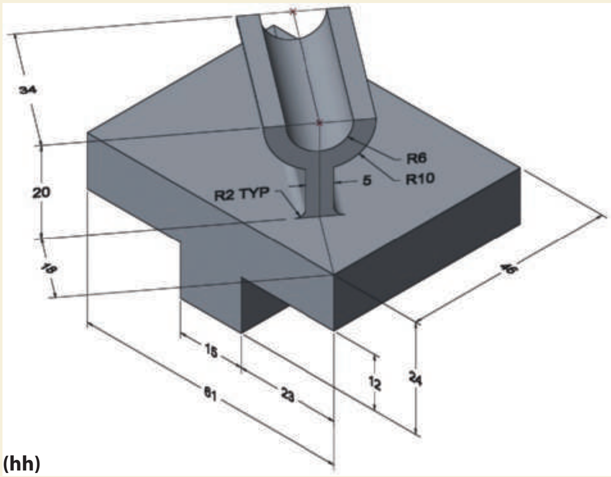

(hh)

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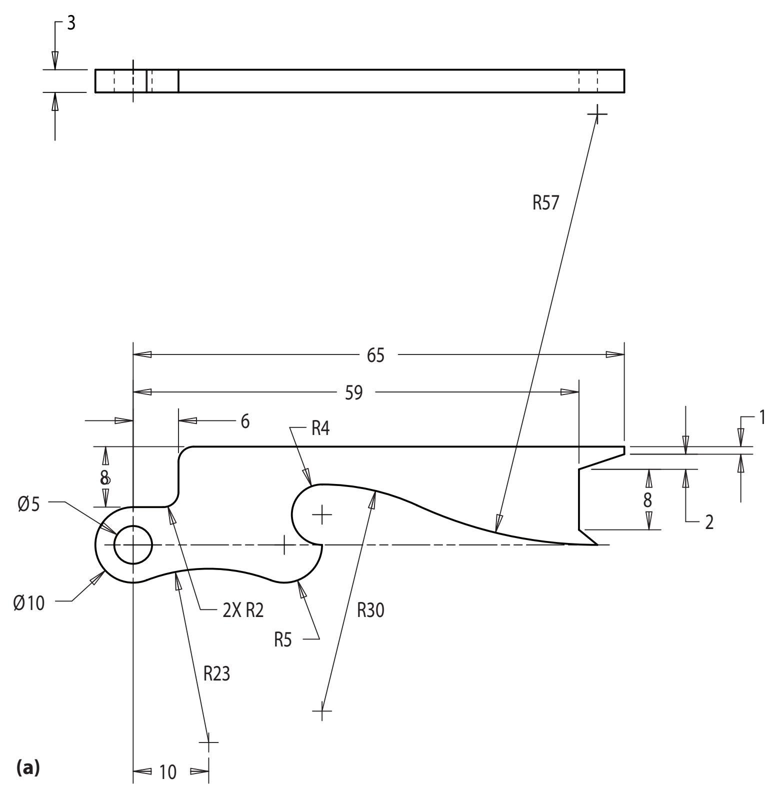
FIGURE P8.6.

8.12 PROBLEMS (CONTINUED)

7. For the multiview drawings shown in Figure P8.7, create a pictorial sketch of the object or create the solid model using CAD, as indicated by your instructor.



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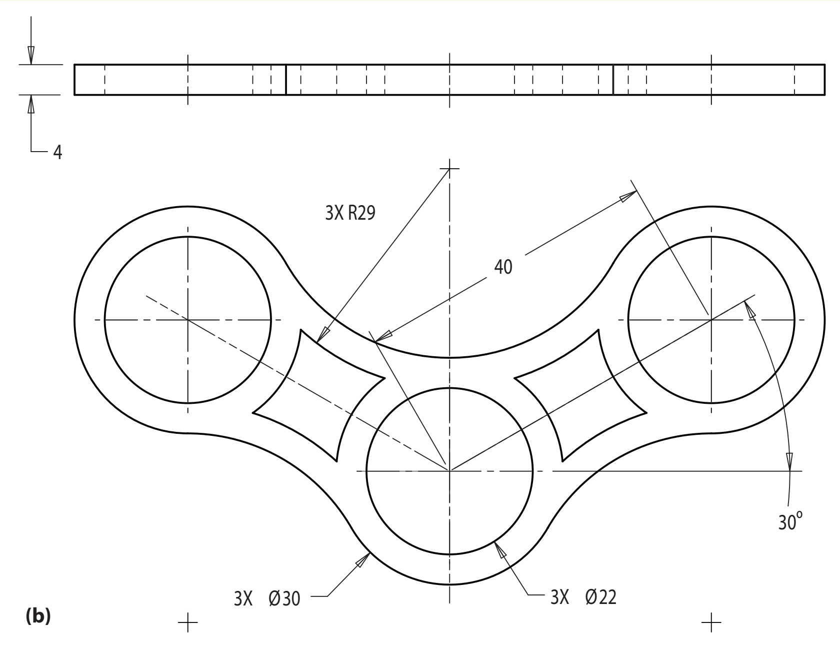

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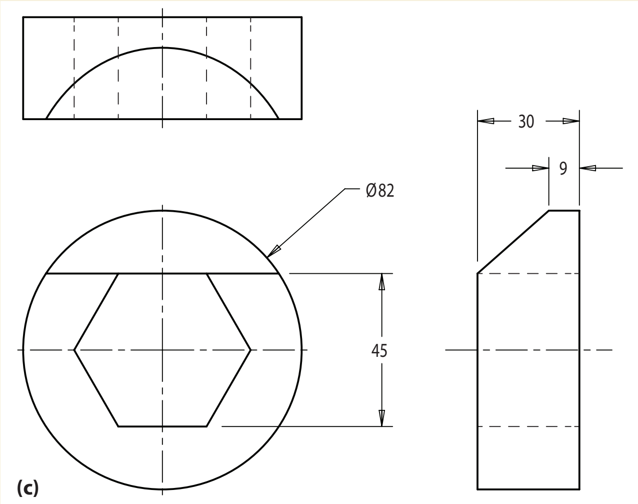
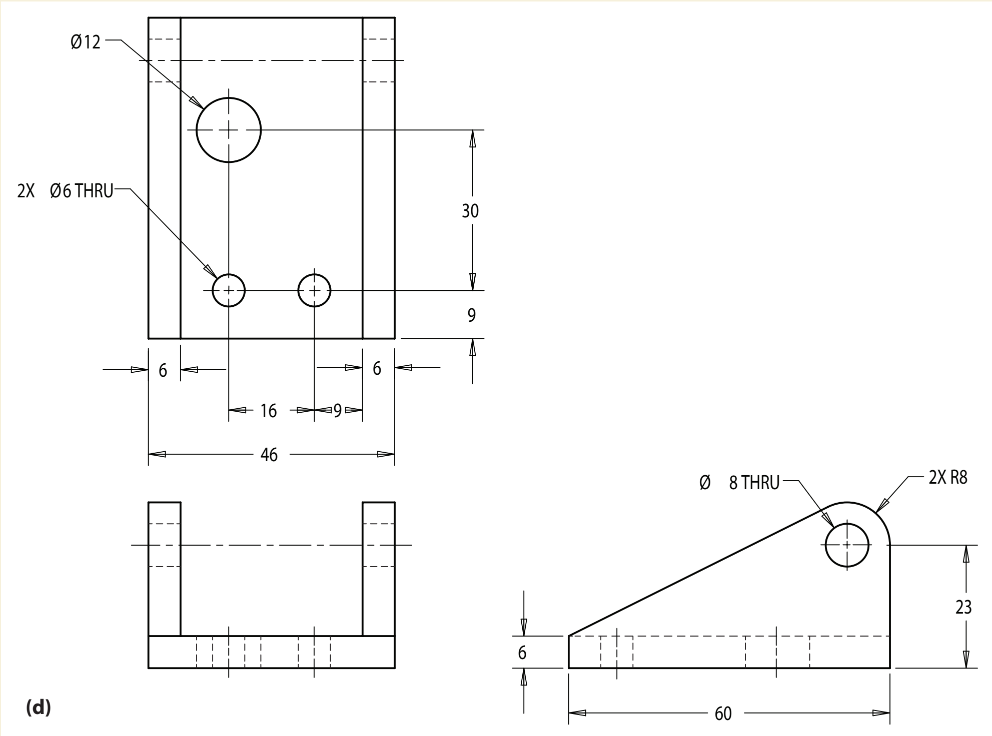
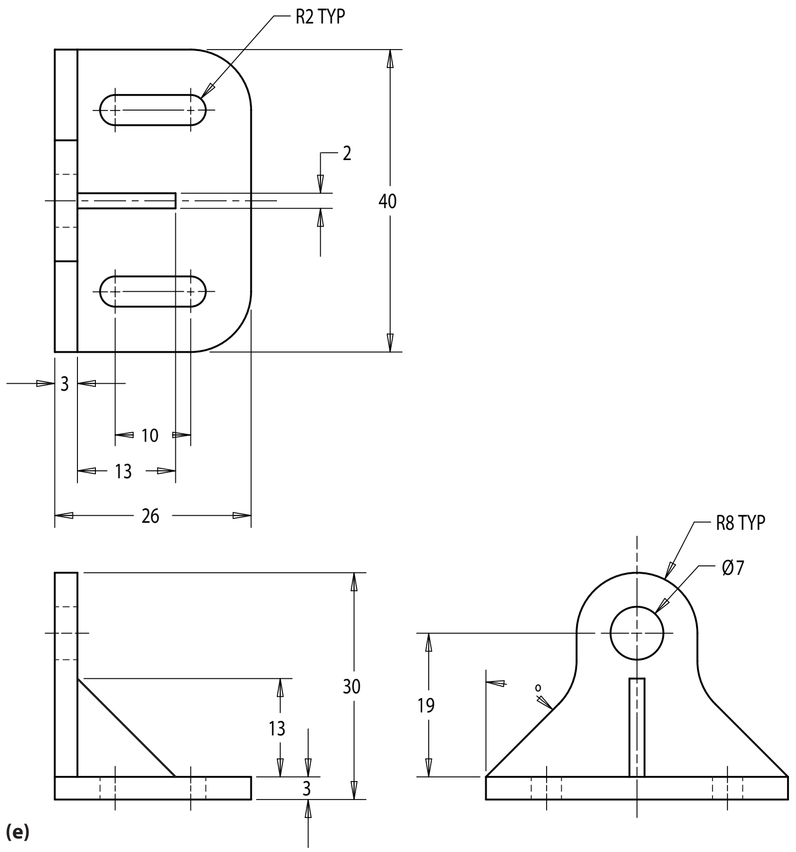
8.12 PROBLEMS (CONTINUED)





8.12 PROBLEMS (CONTINUED)



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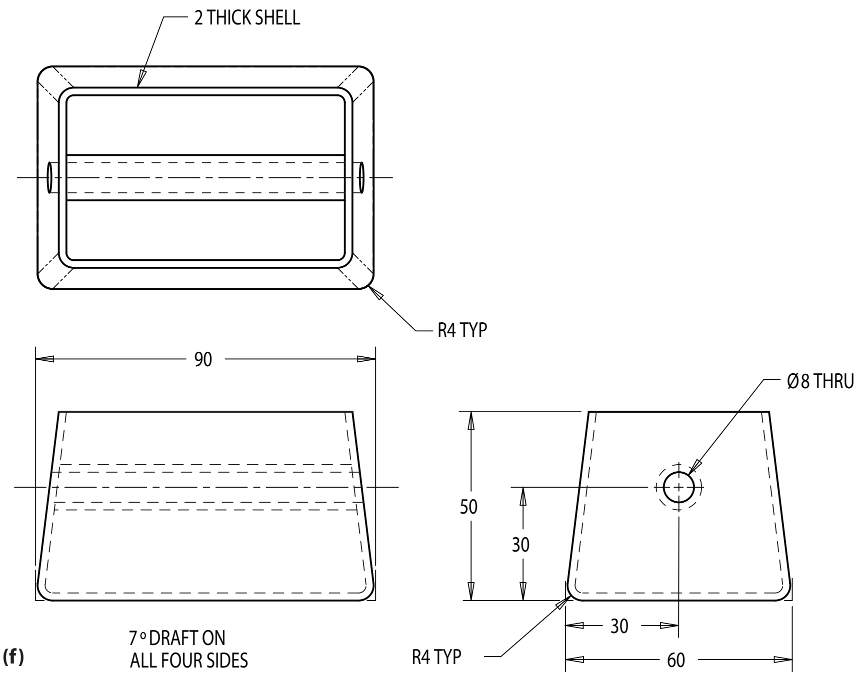

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8.12 PROBLEMS (CONTINUED)

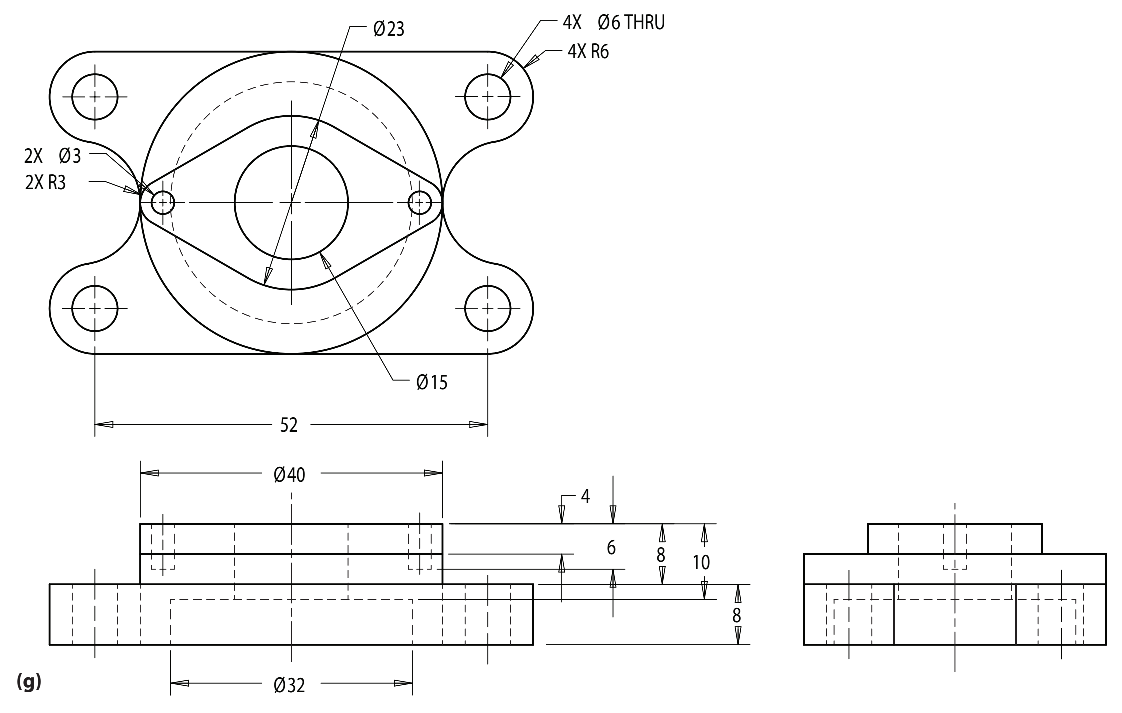

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FIGURE P8.7.

CHAPTER 9

PICTORIAL DRAWINGS

OBJECTIVES

After completing this chapter, you should be able to

- Explain the importance of pictorial drawings as an aid in visualization
- Create an isometric drawing of an object composed of principal, inclined, and oblique surfaces
- Draw ellipses on the front, top, and right faces of the isometric to represent cylinders and holes
- Explain the difference between a cavalier and cabinet oblique drawing
- Create an oblique drawing given the orthographic drawing of an object
- Create a two-point perspective drawing given the orientation of the plan view and the location of the elevation view

9.01 INTRODUCTION

A major problem in engineering graphics is the need to represent a 3-D object using a 2-D sheet of paper. Generally, this is done using orthographic views that show the top, front, and side views of the object as separate entities. Each view represents two dimensions of the object. The top view displays the width and depth dimensions, the front view shows the width and height dimensions, and the side view displays the width and height dimensions. Since each of these views represents only two dimensions, you have to move the views around in your mind's eye to figure out what the 3-D object looks like. Sometimes you will get it right, and sometimes you will get it wrong. A pictorial drawing or sketch can be effective in helping you visualize the 3-D shape. The **pictorial** shows all three dimensions (height, width, and depth) in a single view of the object. You already have been exposed to pictorials in previous chapters of this book, but in a rather informal way. Sketching pictorials helped you develop your visualization skills. The earlier chapters also used pictorials as a visualization aid for making formal multiview drawings. There are cases, however, when pictorial sketches are inappropriate for certain applications; for example, in formal engineering drawings and documents. In these cases, a more formal and more accurate method of creating pictorial drawings is needed.

Pictorials help you visualize, but they also are important when you attempt to assemble parts into mechanisms or need to purchase replacement parts for a tool. For example, when you purchase a lawn mower, the manufacturer will furnish a user's manual. In the user's manual, you are likely to find a series of pictorial drawings that show every lawn mower part with some identifying information. If you need to replace a part, you can use the pictorial to install the part. Pictorial drawings can be an effective way to ensure that a mechanism gets reassembled properly after a part is replaced.

9.02 Types of Pictorial Drawings

Pictorial drawings are used mainly as visualization aids; they are not used as working drawings from which a part is produced. Historically, before the industrial revolution, craftsmen would sketch a design as a pictorial and use it to produce a product. If part of the design were to fail or break, they would have to custom-make another part in order for the product to be useful again. After the industrial revolution, products were manufactured using mass production and the concept of interchangeable parts came into being. With interchangeable parts, a detail drawing showing lines in true length and angles as true angles was necessary to guarantee that parts could be used interchangeably. Also, proper dimensions and tolerances were required to ensure that the parts would be interchangeable. Because a pictorial drawing shows all three dimensions on a 2-D sheet of paper, lines that define surfaces may not be true length and angles may not be true angles, as shown in Figure 9.01. The figure shows a pictorial representation of a cube. Note that the lines that define the cube are not shown in true length. Also, since a cube is composed of six surfaces that are squares, the angle between the lines that form the edges of the cube should be 90 degrees; but it is not. For the object to be produced by mechanical processes, it is important that the shape be interpreted correctly. An effective way to do that is to use a multiview orthographic drawing (each view contains only two dimensions), which shows the true

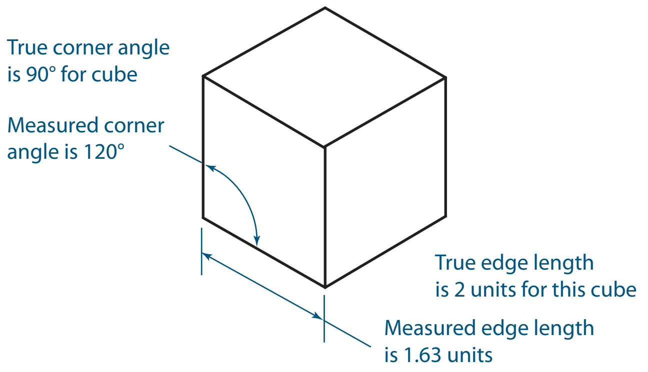

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FIGURE 9.01. An isometric pictorial of a cube.

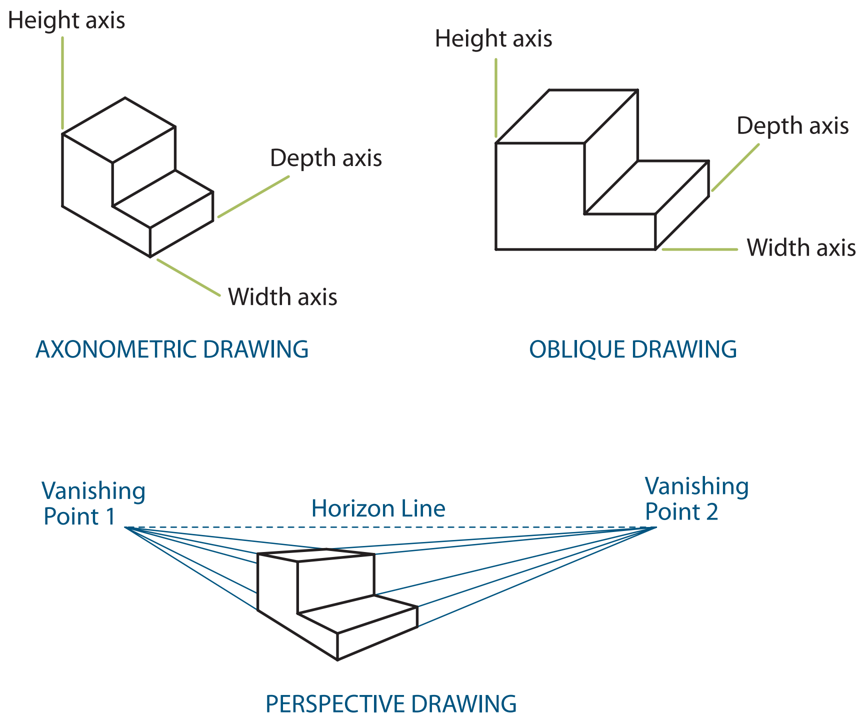

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FIGURE 9.02. Samples of pictorial drawing types of the same object.

lengths of lines and the true angles. After the object is produced, the pictorial drawing can be used to compare the 3-D shape of the object with the actual object and the pictorial drawing will verify that the shape is correct.

There are many types and variations of pictorial drawings since they must serve different purposes. Some of the major types of pictorials are axonometric drawings, oblique drawings, and perspective drawings. An example of each is shown in Figure 9.02. The term *axonometric* is a broad category of pictorial drawings that includes isometric, dimetric, and trimetric drawings. The term *axonometric* refers to the angle that the coordinate axes make with each other when the three dimensions are defined for the object. Figure 9.02 shows the coordinate axes relative to the type of pictorial drawing represented. An oblique drawing is a pictorial in which the irregular surface (often a circular cylinder) is drawn in the 2-D plane of the paper and the third dimension is drawn at some receding angle to this plane (Figure 9.02). A perspective drawing is one where the receding axis(es) converge at vanishing point(s) located on what is referred to as a horizon. When done correctly, a perspective drawing gives the impression that you are looking at a photograph. Each of these types of pictorial drawings will be discussed in detail later in this chapter.

Most of the following techniques for creating a pictorial drawing are traditional methods based on 2-D CAD and drafting techniques. These techniques can still be used when 3-D modeling tools are not available. For speed, efficiency, and accuracy, pictorial drawings are most easily created when they are extracted from 3-D models.

9.03 Axonometric Drawings

The word *axonometric* has its origin in Greek from the word *axon*, which means "axis," and the word *metric*, which means "measure." An **axonometric drawing** refers to three types of pictorial drawings: isometric, dimetric, and trimetric drawings that are created by measuring along three axes representing width, depth, and height. Isometric drawings are relatively easy to produce and usually can be done quickly. In an **isometric drawing**, the three dimensions of width, depth, and height are shown along the three isometric axes, as shown in Figure 9.03. When the dimensions of width, height, and depth are plotted in this fashion, the three normal surfaces (frontal, horizontal, and profile) of a rectangular solid object have equal angles between them (120 degrees). In **dimetric drawings**, the three dimensions of width, depth, and height are shown along the three dimetric axes, as shown in Figure 9.04. When plotted correctly along these axes, two of the normal surfaces (frontal and profile) have equal angles between them, while the third normal surface (horizontal) has a different angle. In a **trimetric drawing**, the dimensions of width, height, and depth are plotted along axes so the normal surfaces of a rectangular solid (frontal, horizontal, and profile) have none of the three angles equal, as shown in Figure 9.05.

FIGURE 9.03. Three visible normal surfaces (frontal, horizontal, and profile) of a prism have equal angles between them (120 degrees) in an isometric drawing.

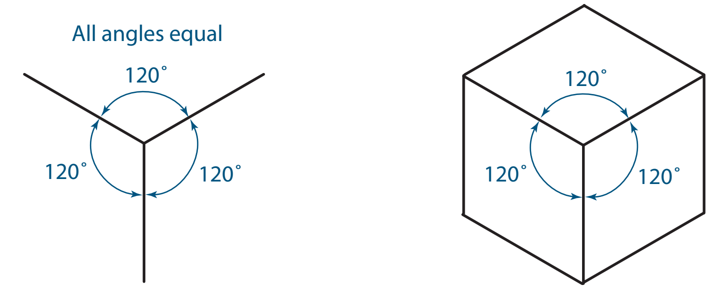

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FIGURE 9.04. In a dimetric drawing, two of the three visible normal surfaces (frontal and profile) of a prism have edges presented at equal angles, not equal to 120 degrees.

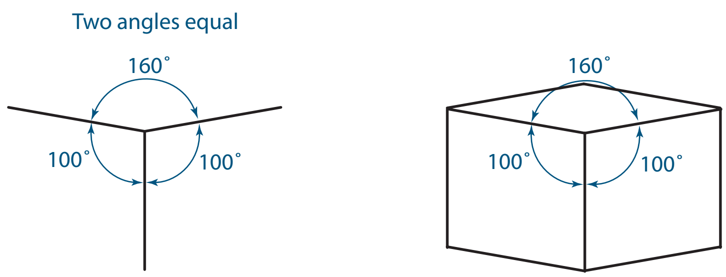

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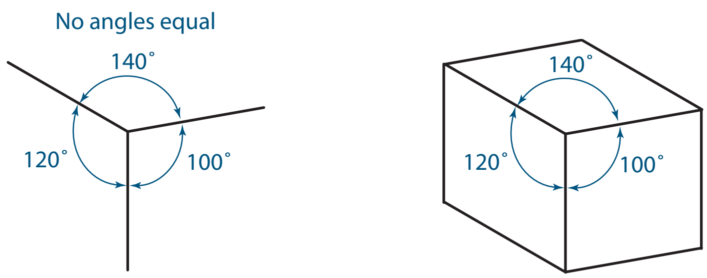
FIGURE 9.05. A trimetric drawing presents the three visible normal surfaces (frontal, horizontal, and profile) of a prism in a position where none of the angles between the surface edges is equal.



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The drawing of dimetric and trimetic pictorials takes a great deal of time because of the uncommon angles that are used. Therefore, the time it takes to complete this type of pictorial may offset the benefits there may be in producing it. Dimetric and trimetric pictorials are seldom used in engineering work; so they will not be discussed in detail. Instead, the chapter will focus on the easiest and most popular form of axonometric drawing, the isometric drawing.

9.03.01 Isometric Drawings

The best way to learn about formal isometric drawings is to study a simple example. Figure 9.06 shows the orthographic views of a step block. For this object, all of the surfaces are normal surfaces. This means that each surface is viewed in its true size and shape in one of the principal views and will appear as an edge view in the other principal views. A frontal surface appears in true size and shape in the frontal view, while a horizontal surface appears in true size and shape in the horizontal or top view. A profile surface appears in true size and shape in the right side or right profile view. Note that in the front view, you see the width and height dimensions; in the top view, you see the width and depth dimensions; and in the profile view, you see the depth and height dimensions. To draw the isometric pictorial of the step block, you must first set up the isometric axes that will define where to measure the width, height, and depth dimensions.

One way to define the isometric axes is shown in Figure 9.07. Two receding axes intersect at point A and are at 30 degrees to an imaginary horizontal line. The width dimensions will be plotted along the receding axis extending to the left of point A, and

FIGURE 9.06. Orthographic views of a step block.

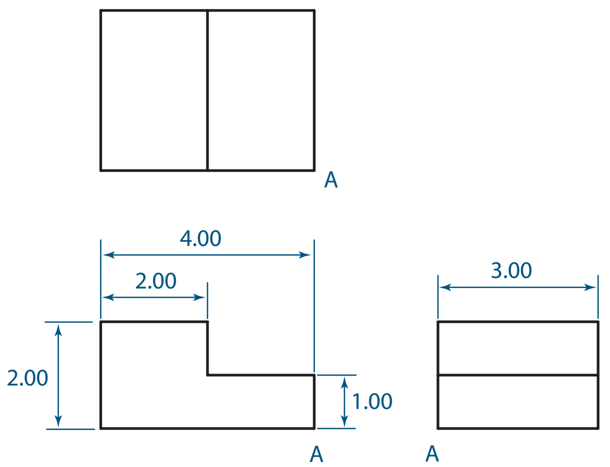

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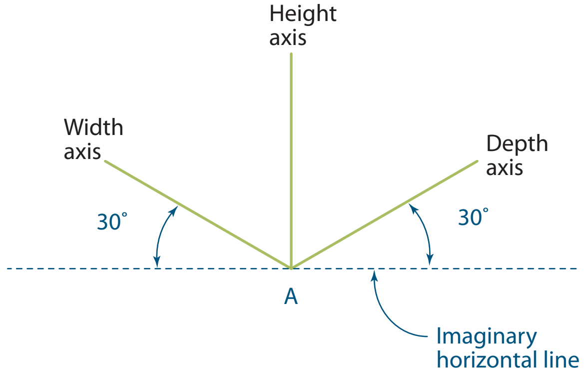
FIGURE 9.07. The isometric axes.



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the depth dimensions will be plotted along the receding axis extending to the right of point A. The height dimension will be plotted along the vertical axis that extends upward from point A. When you look at the orthographic drawing in Figure 9.06, you see that the maximum width, depth, and height are 4 units, 3 units, and 2 units, respectively. When beginning to draw the isometric, you want to frame the step block on the isometric axes and then take care of the details to finish the isometric.

To create this isometric drawing, you first create a block of the overall size of the object noting that the width will go up and to the left from the origin, the depth will go up and to the right, and the height will be a vertical line from the origin. Once you have this block created, you can measure off distances for the step and fill in those lines. Finally, you will erase unwanted lines and darken lines on the drawing to make sure it is well-defined (see Figure 9.08).

The step block is relatively easy to draw because all of the lines that define it are called **isometric lines**. All of the lines defining the frontal surface are parallel or perpendicular, as are the lines defining the horizontal and profile surfaces. Isometric drawings can get rather complex when surfaces that need to be drawn are not defined as frontal, horizontal, or profile surfaces because the lines that form the surfaces are not parallel or perpendicular. Therefore, these lines are "nonisometric lines" and need to be plotted using their endpoints.

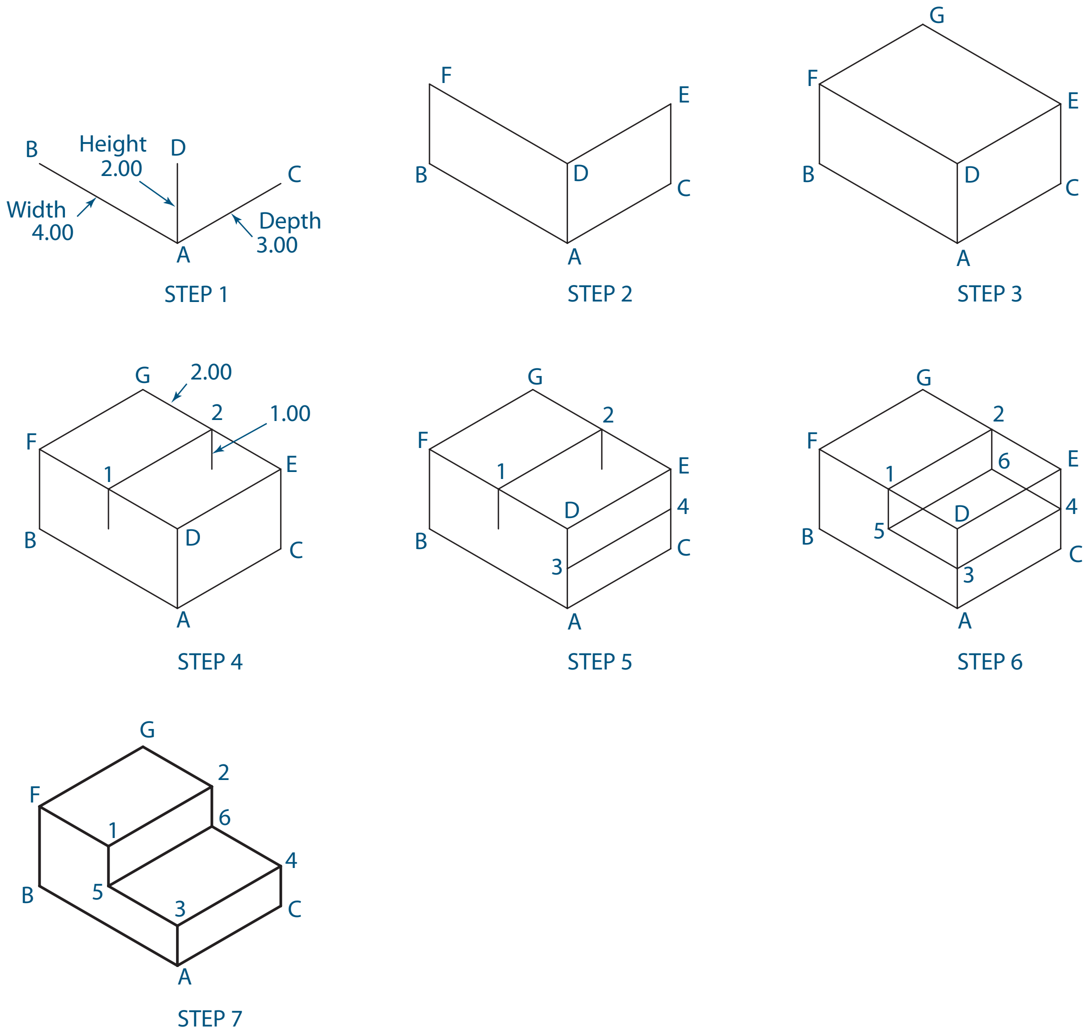

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FIGURE 9.08. The steps to draw an isometric pictorial of the step block.

9.03.02 Inclined Surfaces

Figure 9.09 is an orthographic drawing showing a variation of the step block that has an inclined surface. The procedure to draw the isometric pictorial of this object is similar to drawing the step block in Figure 9.08 except that you have to account for the inclined surface. Figure 9.10 illustrates the procedure used to create this isometric drawing. You would start with the step block as before, but then locate the endpoints defining the inclined surface. You then "connect the dots" for these endpoints to define the edges of the surface, erasing lines and darkening edges as needed for clarity.

Lines 5-4 and 7-C in Figure 9.10 are defined as nonisometric lines because they are not parallel to any isometric lines. They must be drawn by plotting the endpoints and then connecting them properly, as shown in Figure 9.10.

FIGURE 9.09. Orthographic views of a step block with an inclined surface.

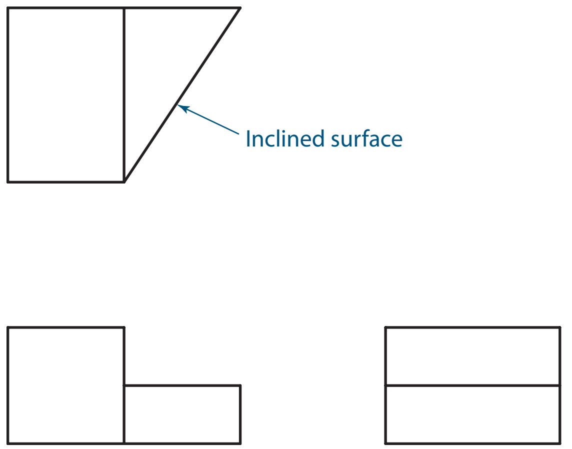

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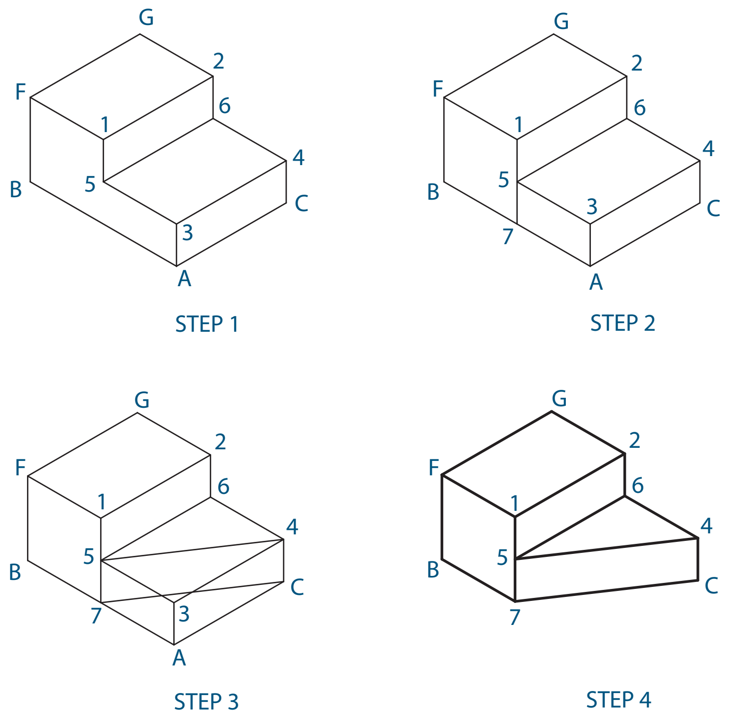
FIGURE 9.10. The steps involved to create an isometric pictorial of a step block with an inclined surface.



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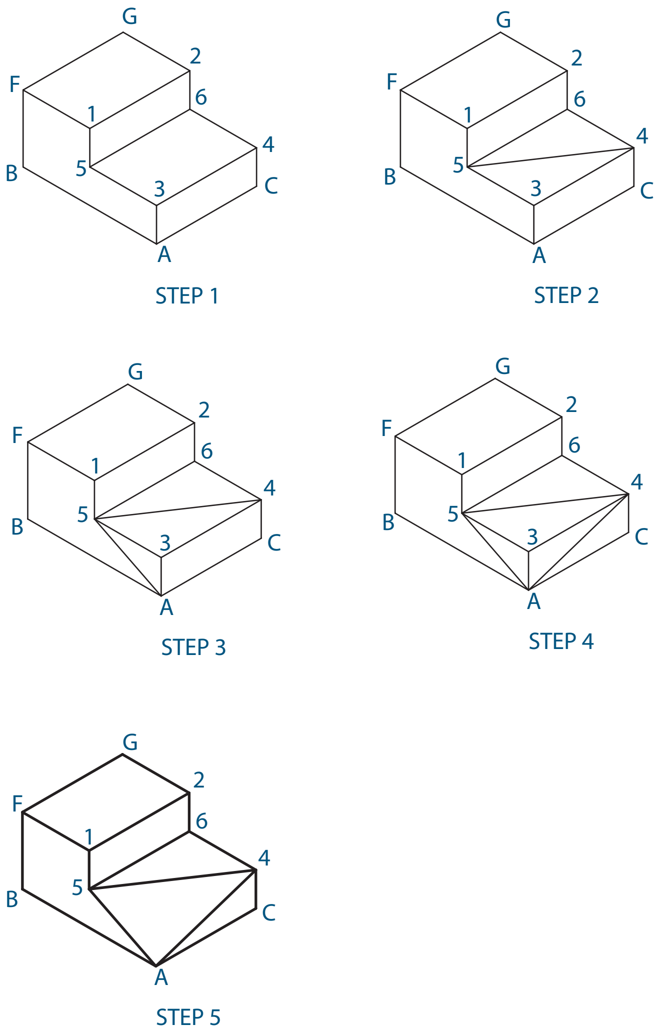
9.03.03 Oblique Surfaces

An even more complex example involving isometric drawings includes a drawing that has an oblique surface. An oblique surface is a surface that is neither parallel nor perpendicular to the frontal, horizontal, or profile projection planes; and the oblique surface will appear in all three views in its characteristic shape. That is, if the oblique surface is a triangle, it will be a triangle in all three views; it will not appear as an edge in any of the three views. Knowing this, it is reasonable to assume that an isometric drawing showing an oblique surface will show the surface in its characteristic shape as well. Figure 9.11 shows three orthographic views of a step block that includes an oblique surface.

Figure 9.12 illustrates the procedure used to create this drawing. It is similar to the way the drawing of the object with the inclined surface was created—you first start with the step block, locate the endpoints that define the oblique surface, and then "connect the dots" to form the edges of the surface. Once again, you erase unwanted lines and darken edges to clearly define the object.

In Figure 9.12, the surface formed by 5-4-A is an oblique surface. The lines 5-4, 4-A, and 5-A are nonisometric lines. All of the remaining lines of the step block are isometric lines.



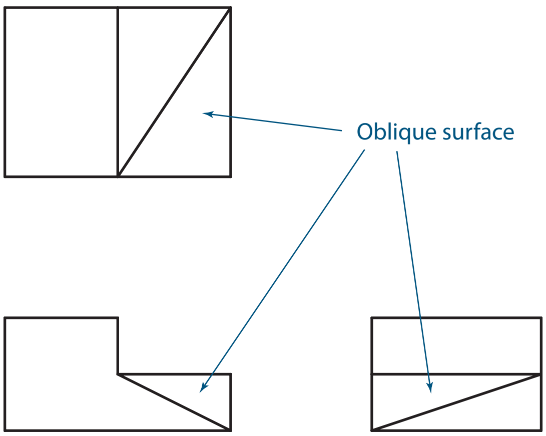


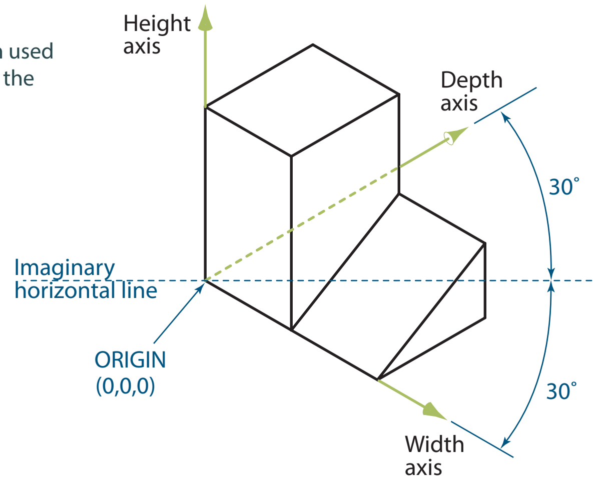
FIGURE 9.11. Orthographic views of a step block with an oblique surface.

FIGURE 9.12. The steps involved to create an isometric pictorial of a step block with an oblique surface.

In an isometric drawing, the orientation of the object depends on the placement of the axes, which locates the origin. In Figure 9.08, point A on the lower-right corner of the object was chosen as the location of the origin. From this point, you measured back along the left receding axis to establish the width and you measured along the right receding axis to find the depth. Then you measured vertically to establish the height. For all measurements, you assumed point A to be the origin, or the 0,0,0 point. Selection of the origin (or point 0,0,0) in 3-D coordinates establishes the relative position of each point that makes up the object. The origin can be placed anywhere on the object, and the isometric can be drawn from this reference. For example, Figure 9.13 shows an origin that would be located at the lower-left corner of the object. The width, depth, and height measurements would be measured along the appropriate axes as shown in Figure 9.13. Notice that the width and depth axes are still 30 degrees from an imaginary horizontal line, as explained previously. No matter where the origin is located, all points that define the object can be plotted in 3-D space; then the points are connected to show the pictorial.

Refer back to Figure 9.06, the multiview drawing of the step block. This set of views shows a front view, a top or horizontal view, and a right view. You can also orient the views for the step block to show a front view, a top or horizontal view, and a left view (Figure 9.14). Each of these orthographic arrangements allows for two different orientations of the pictorial. The pictorial can open to the right or it can open to the left. The term *open* refers to how the orthographic views are interpreted. When the orthographic drawing is oriented to show a front, top, and right-side view, the isometric pictorial opens to the right (Figure 9.15). When the orthographic drawing is oriented to show a front, top, and left-side view, the isometric pictorial opens to the left (Figure 9.15).

FIGURE 9.13. The origin used to locate an object using the lower-left corner.



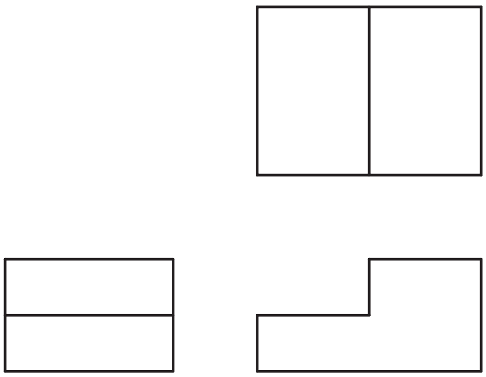
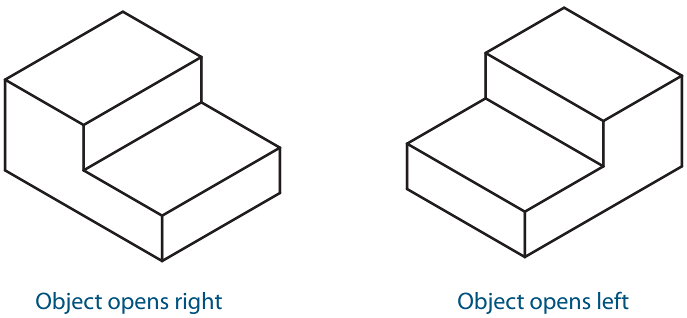


FIGURE 9.14. Orthographic views of the step block oriented as the front, top, and left profile views.

FIGURE 9.15. An isometric drawing has two primary positions for viewing. The object can be oriented so that it "opens" to the right or to the left.



Orthographic views generally show all kinds of lines that define the object. Lines may be visible, or they may be hidden. Hidden lines are represented in orthographic views as dashed lines. In isometric pictorial views, hidden lines are generally not shown unless they are necessary for interpretation of the object. For example, Figure 9.16 shows an orthographic drawing of a step block on the left that has a square hole going all the way through it. In the orthographic views, hidden lines in the front view and in the right view define the hole. The isometric pictorial of this block is shown on the right in the figure. Notice that no hidden lines are shown in the pictorial. The square hole shown is assumed to go through the entire object. This is a correct representation of the step block in isometric. Figure 9.17 shows an orthographic drawing of a step block on the left that has a square hole that does not go all the way through. This is often referred to as a blind hole. As before, hidden lines are shown in the orthographic views; however, since the hole does not go all the way through the object, the isometric pictorial shown on the right must include hidden lines to define how deep the square hole goes into the step block. In this case, hidden lines are necessary to define the depth of the hole.

FIGURE 9.16. Orthographic views with a square hole through the base. Hidden lines are shown in the front and right profile views. The isometric pictorial does not show hidden lines.

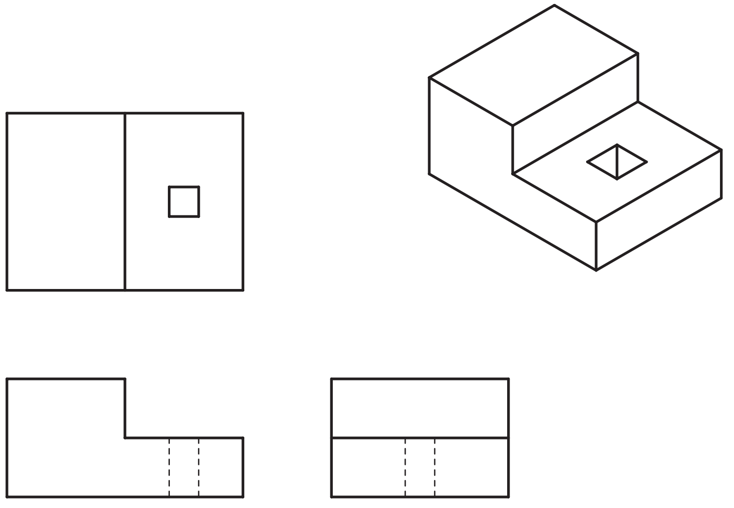
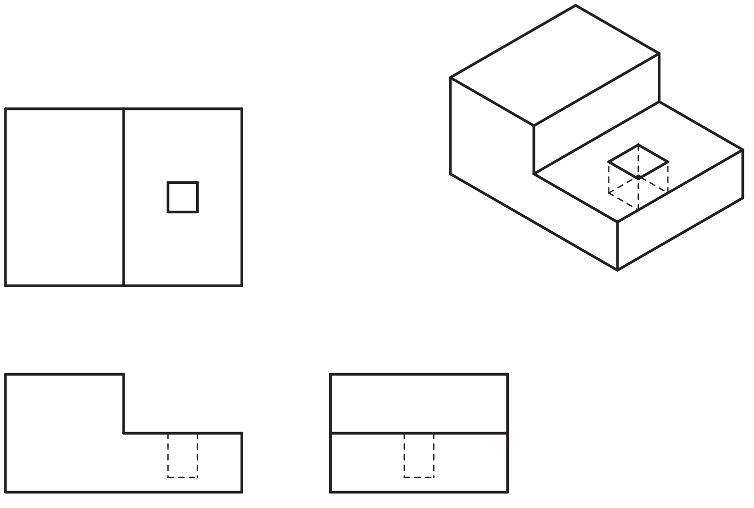


FIGURE 9.17. Orthographic views showing a square hole that does not go all the way through the base. Hidden lines are shown in the front and right profile views. Hidden lines are shown in the isometric pictorial.



9.03.04 Cylindrical Surfaces

You have learned that 3-D objects can be composed of normal, inclined, and oblique surfaces, and you have learned how to represent each surface in an isometric pictorial. Another type of surface that is associated with 3-D objects is the cylindrical surface. The cylindrical surface may be positive (such as a post) or negative (such as a hole). When you are looking directly at the "edge" view of the cylindrical, it will be seen as a circle. Since the surfaces of an isometric pictorial are distorted and you are looking at all three dimensions on one sheet of paper, the cylinder is represented on an isometric drawing by an ellipse. The orientation of the ellipse is dependent upon whether it appears on the top face, the right face, or the left face. An isometric cube that has an ellipse on the right face, the left face, and the top face is shown in Figure 9.18. Note the orientation of the long axis for the ellipses on each of the three normal faces of the cube.

Figure 9.19 illustrates how to construct an ellipse on a horizontal (top) isometric surface. The first step is to establish the limits of the ellipse by drawing a bounding box, which appears as a parallelogram in the pictorial view. For normal surfaces on the

FIGURE 9.18. An isometric cube showing ellipses on each face. Note the orientation of the long (major) axis for the ellipse on each of the three normal faces.

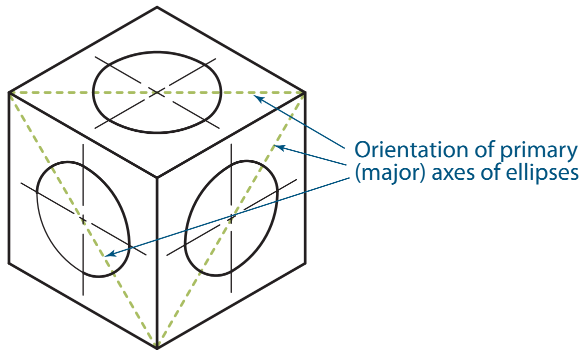
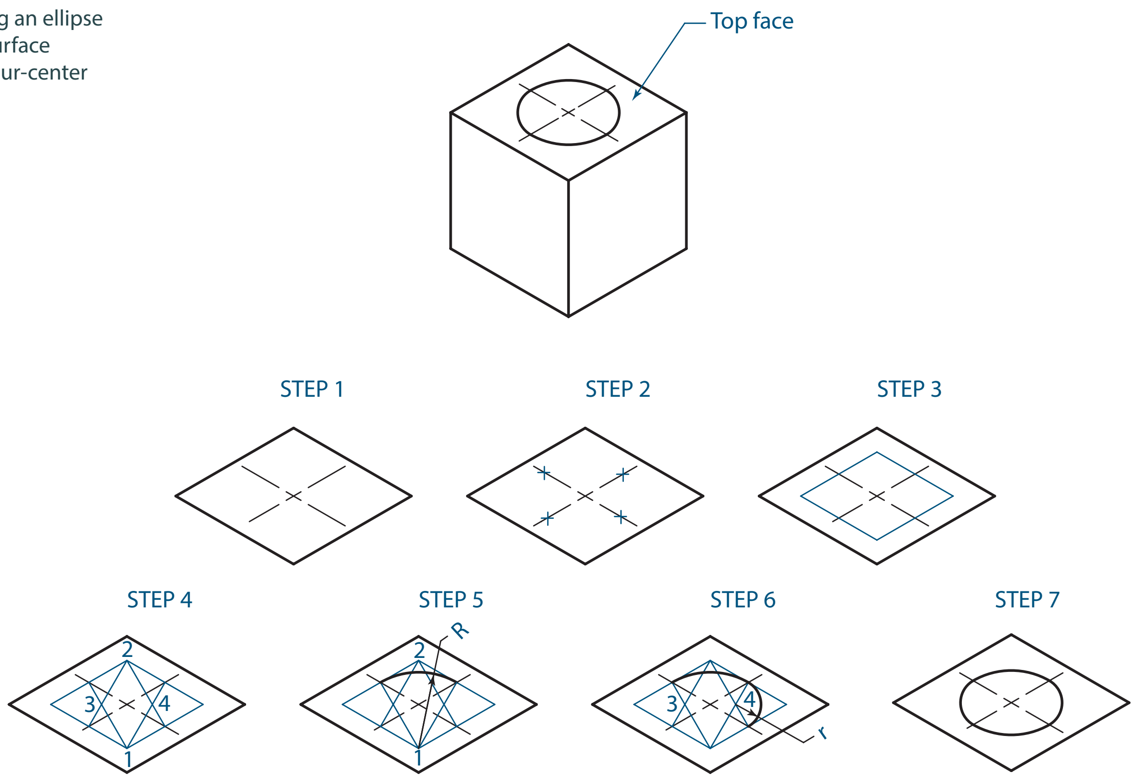


FIGURE 9.19. Drawing an ellipse on the top isometric surface using the traditional four-center method.



object, the sides of the parallelogram are parallel to the isometric lines. The easiest way to create the ellipse is to use an ellipse template (when the drawing is done by hand) and select the ellipse that is tangent to the sides of the limiting box. When a 2-D CAD tool is used, an ellipse is created by specifying its major and minor diameters in the ellipse creation tool; the ellipse is then rotated into the correct orientation.

The ellipse also can be constructed using the four-center method. This construction locates four centers that can be used for drawing four arcs (two large arcs and two small arcs) that approximate an ellipse. These arcs may be sketched or may be drawn with a compass. The technique used to create the ellipse is shown in Figure 9.19.

To create the ellipse, you first create the bounding box that defines the limits of the circle. The bounding box will look like a parallelogram on the isometric drawing. You are going to be designating four center points—one for each of the four arcs that define the ellipse. Note that the near point (point 1) and the far point (point 2) of the diamond are the first two points of the four centers. From these points, draw light construction lines across the diamond where the centerline intersects the box. You should have four light construction lines on the surface. Where these light construction lines cross is the location of points 3 and 4 of the four centers. Once you have located the four center points, you can draw the two long arcs and then the two short arcs that define the ellipse.

The steps required to draw an ellipse on the right face are shown in Figure 9.20, and the steps required to draw an ellipse on the left face are shown in Figure 9.21. Essentially, they are the same as drawing the ellipse on the top face.

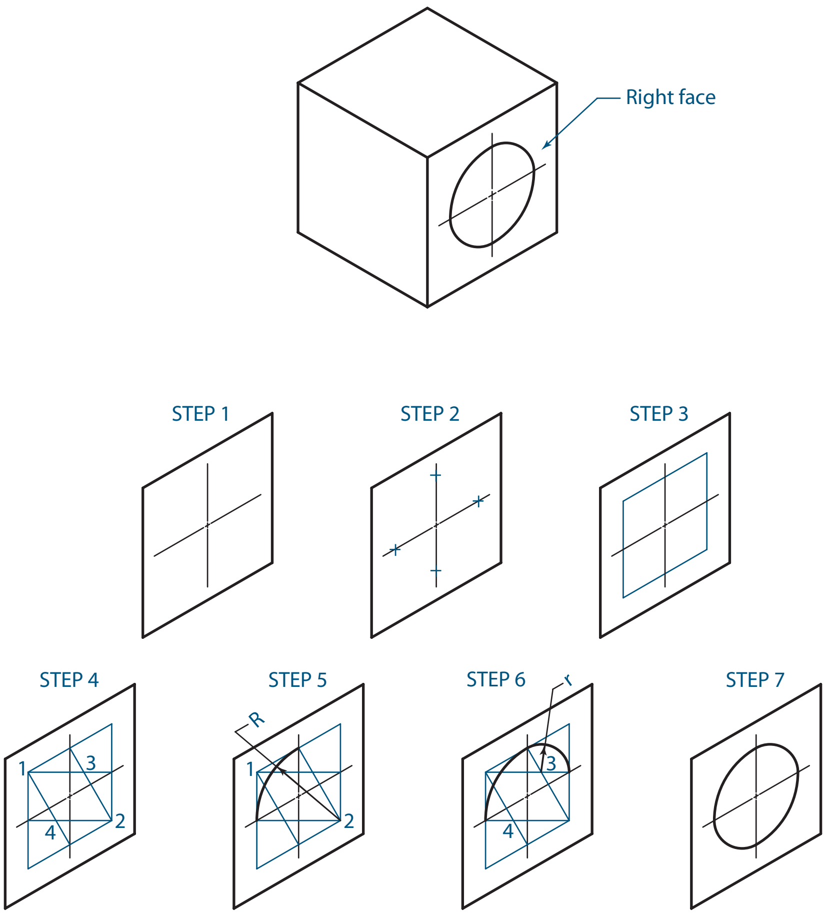


FIGURE 9.20. Drawing an ellipse on the right isometric surface using the traditional four-center method.

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FIGURE 9.21. Drawing an ellipse on the left isometric surface using the traditional four-center method.

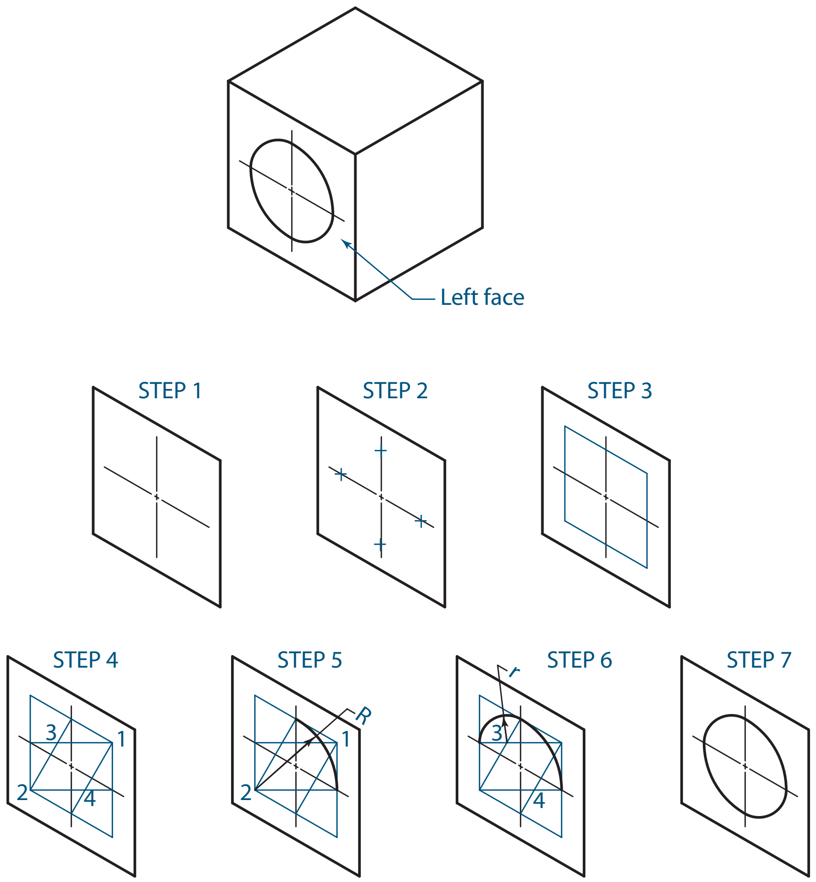
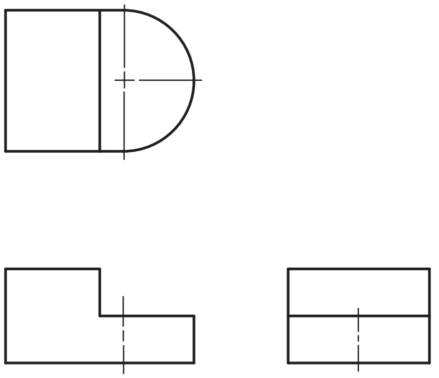


Figure 9.22 shows the orthographic views of a step block that has one of the surfaces represented by a semicircle. When drawing the isometric pictorial of this object, you need to incorporate the techniques involved in drawing an ellipse; but you are going to draw only half of it. Construct the isometric drawing of the step block shown in Figure 9.08. Figure 9.23 illustrates the technique required to draw the lower semicircular surface.

To create this semicircular end, you first locate the center and construct the bounding box as before and locate the centers to draw the arcs. However, this time there are only two centers you have to locate, since you will only be drawing two arcs to define the half-ellipse. You then repeat this process to create the two arcs that

FIGURE 9.22. Orthographic views of a step block with a semicircular base.



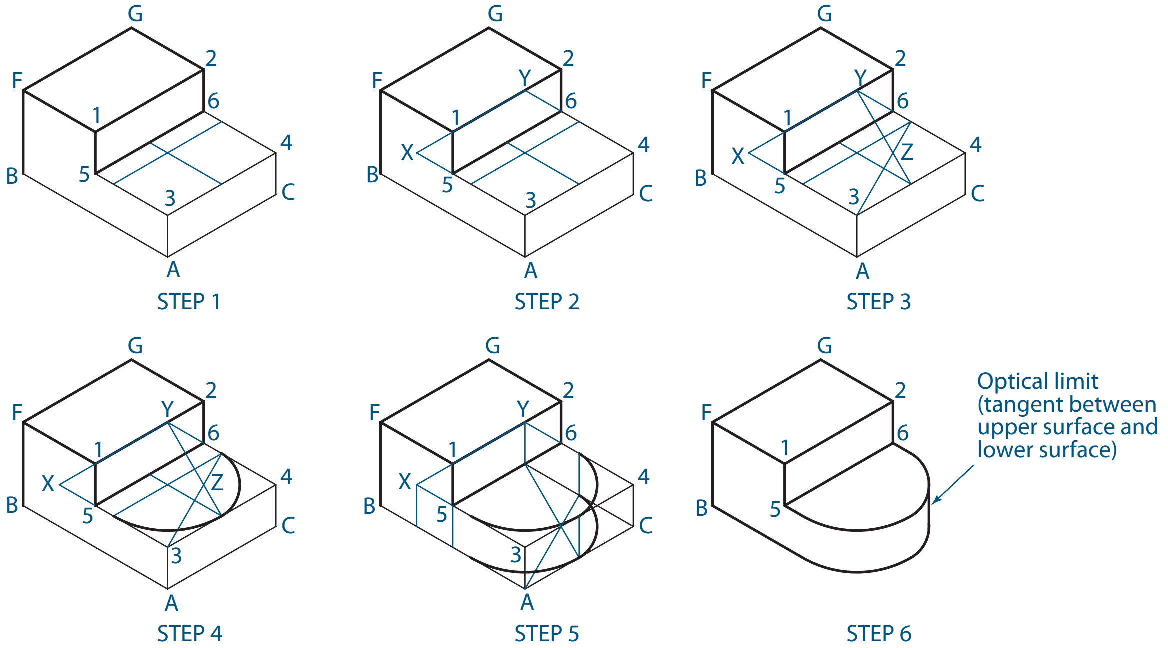


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FIGURE 9.23. Drawing an isometric pictorial of a step block with a semicircular base.

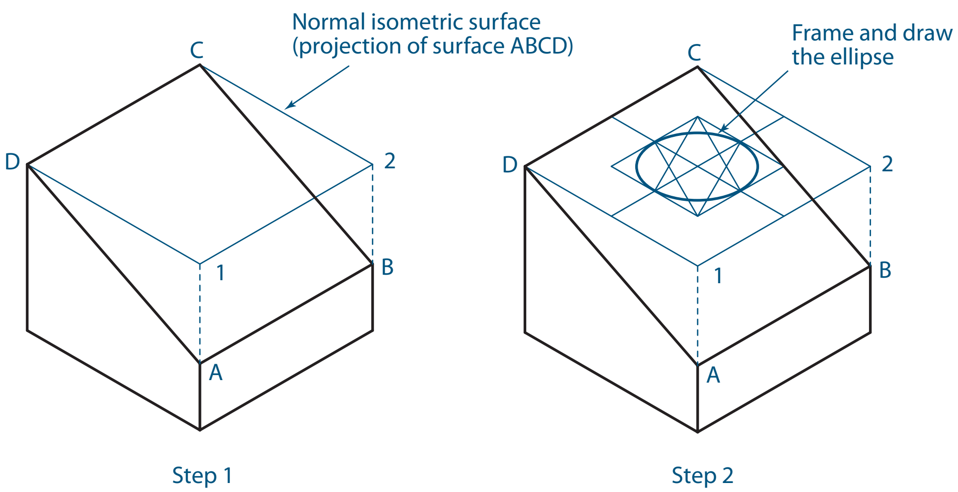
define the "bottom" of the curved surface. Finally, to complete the pictorial, you draw a tangent line to define the visible extent of the curved surface. Finish by erasing unneeded lines and darkening edges as before.

9.03.05 Ellipses on Inclined Surfaces

Sometimes the need arises to draw an ellipse on an inclined surface that appears in an isometric drawing. To do this, you must project some points to ascertain the location of the ellipse on the inclined surface. To begin this task, you first create a normal isometric surface, or phantom surface, by extending up from points A and B and over from points C and D as shown in Figure 9.24, Step 1. You then construct an ellipse on this surface as you have done in the past. This is shown in Figure 9.24, Step 2.

You will then draw several construction lines defining the circle. These construction lines should extend to the edge of the surface you initially created and

FIGURE 9.24. Steps 1 and 2 to create an isometric pictorial showing a vertical circular hole in an inclined surface.



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then should extend down to the true edge on the object. This procedure is illustrated in Figure 9.25, Steps 3 and 4. Using these three illustrations, you will create a circular hole in the center of the inclined surface labeled as A-B-C-D.

Now that you have the locations where these projection lines intersect with the true edge of the object, extend these lines over onto the inclined surface itself as shown in Figure 9.26, Step 5. Project the points defining the circle straight down from the normal phantom surface you created initially. The points where these vertical extension lines intersect with the lines you drew on the inclined surface are the points that define the ellipse on the inclined surface. You can now either carefully sketch or use a tool for irregular curves to finish the ellipse on the inclined surface. Again, erase unnecessary lines and darken edges as appropriate. Your final drawing should look like that shown in Figure 9.27.

FIGURE 9.25. Steps 3 and 4 to create an isometric pictorial showing a vertical circular hole in an inclined surface.

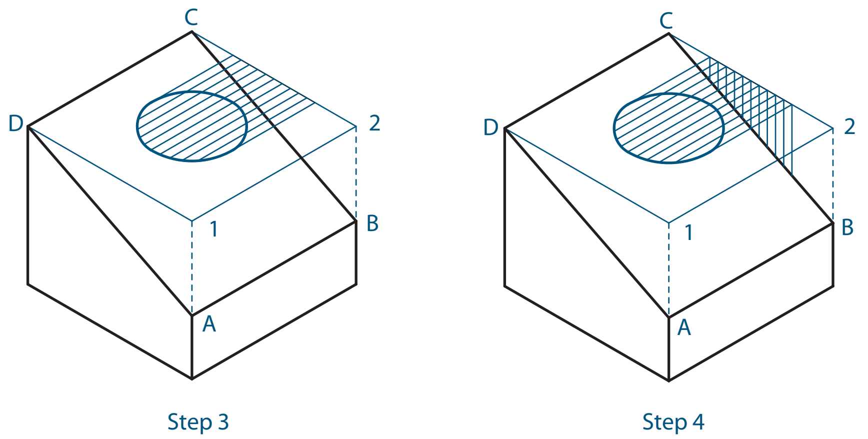

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FIGURE 9.26. Steps 5 and 6 to create an isometric pictorial showing a vertical circular hole in an inclined surface.

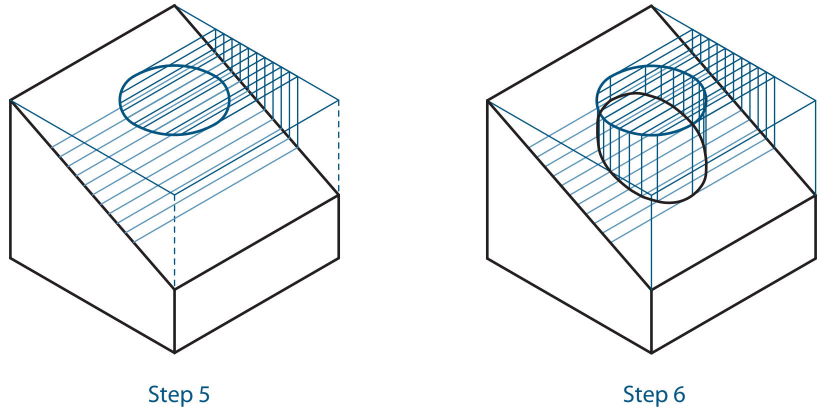

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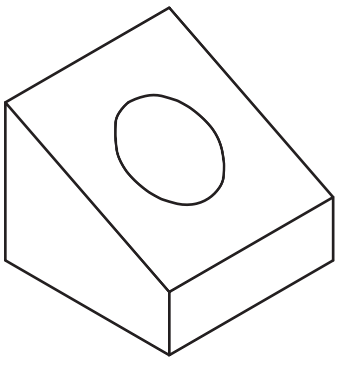
FIGURE 9.27. The finished construction of the vertical circular hole in an inclined surface.



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9.04 Oblique Drawings

Oblique drawings are forms of pictorial drawings that enable the viewer to see the most descriptive view of the object as a front view projected directly onto the plane of the paper. The depth of the object is shown at a receding angle. An advantage of oblique pictorials is that one surface appears as its true size and shape and is not distorted. This means that circular features such as holes and cylinders appear as circles in the plane of the paper and do not appear as ellipses. Objects with cylinders and holes are easier to show in a pictorial representation when the drawing is an oblique pictorial. A disadvantage of oblique pictorials is that they tend to be distorted and appear elongated when viewed because they are not a "true projection" even though they are dimensionally correct. Although any receding angle from 0 degrees to 90 degrees may be used for an oblique drawing, angles between 30 degrees and 60 degrees should be used, which minimizes the distortion and elongation, as shown in Figure 9.28.

9.04.01 Types of Oblique Drawings

There are generally two types of oblique drawings: **cavalier** and **cabinet**. Both types generate a pictorial drawing in a similar manner by showing the most descriptive view of the object in the plane of the paper in true size and shape and showing a receding dimension along an axis at some angle between 30 degrees and 60 degrees (45 degrees is preferred) to minimize distortion. The difference between cavalier and cabinet oblique drawings lies in the measurements made along the receding depth axes. A cavalier oblique drawing is generated when the true length of the depth dimension is measured along the receding axes, as shown in Figure 9.29. The cabinet oblique is generated when half the true length of the depth is measured along the receding axes,

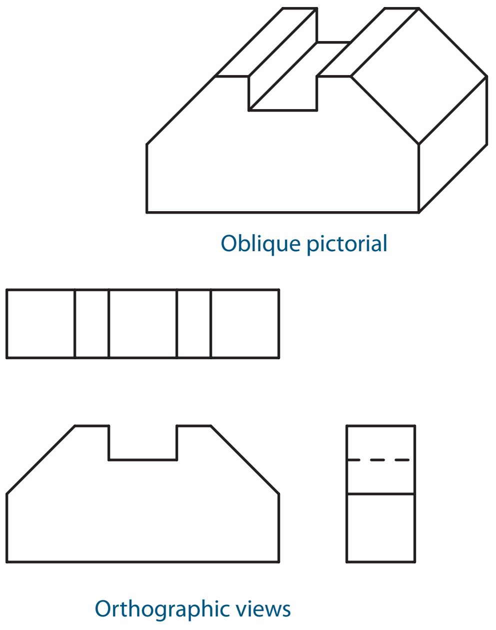

FIGURE 9.28. Example of an oblique pictorial.

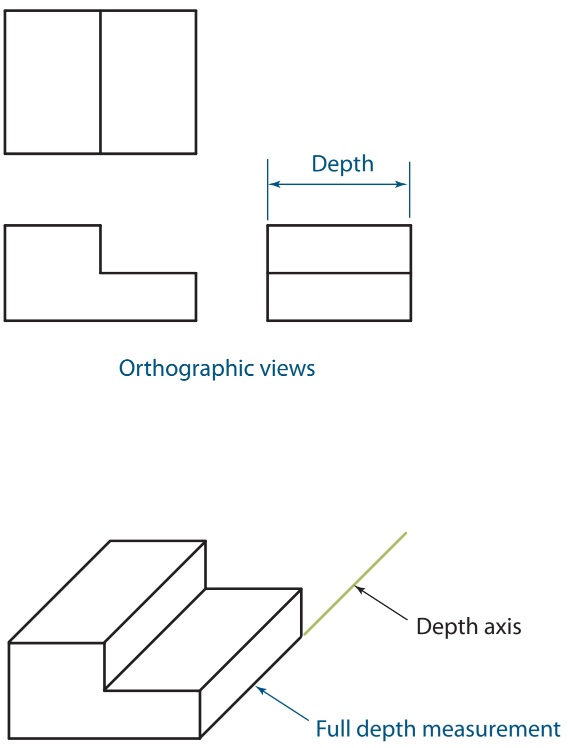

FIGURE 9.29. A cavalier oblique pictorial.

as shown in Figure 9.30. The term "cabinet" stems from the fact that this technique was first developed by furniture makers who thought that showing their products full size in the receding dimension made them appear distorted and who thus settled on half-size for the receding dimension. The cavalier oblique shows the most distortion and elongation, while the cabinet oblique shows the least. For this reason, the cabinet oblique tends to be selected most often.

9.04.02 Construction of Oblique Drawings

An oblique drawing can be constructed using the "framing" technique, which is similar to the technique used when creating an isometric drawing. Essentially, an oblique prism is constructed and then the features are cut away to show the 3-D aspects of the object. Figure 9.31 illustrates the construction of an oblique drawing.

Similar to the way that you started with an isometric pictorial, you first create a block of the overall size of the object that you are drawing. The receding axis should be drawn at about 45 degrees going back in either the right or left direction from the front. You should then draw the front view of the object and extend lines in the

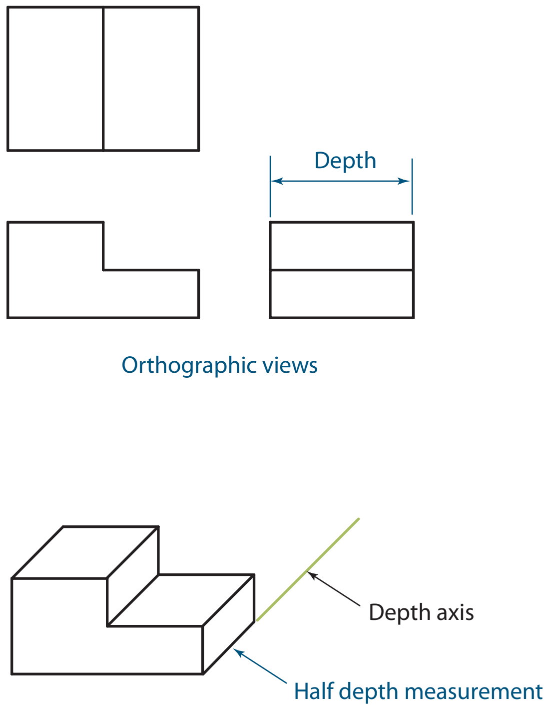

FIGURE 9.30. A cabinet oblique pictorial.

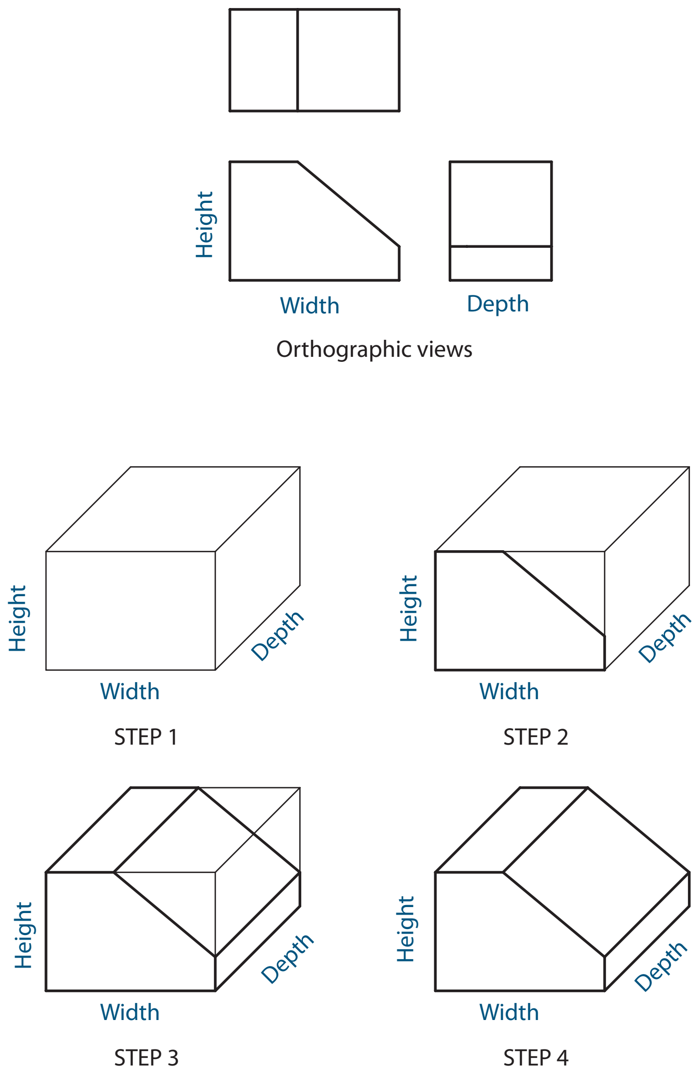

FIGURE 9.31. The steps required to construct an oblique pictorial.

receding direction from each corner on this view. You should note that at least one corner on the object (in this case, the lower-left corner from the front view) will not be seen in the receding dimension. Add the lines necessary at the "back" of the object to define the corners there.

9.04.03 Construction of an Object with Circular Features

The "boxing-in" technique can be used to create an oblique drawing of an object that has circular features such as holes or cylinders. The orthographic views shown in Figure 9.32 indicate that you will have to frame the cylinder with the hole on the left portion of the object as one step and frame the rectangular prism that also has a hole as another step. This object will need to have two "box-in" steps. Figure 9.32 illustrates how the oblique pictorial for this object is created in two parts.

In order to create this oblique pictorial, you first frame the left portion of the object and then locate the hole and cylinder on the front face of this box. Extend this back in the receding dimension and create a partial circle to define the back circular surface of that portion of the object. Draw tangent receding lines that show the visible extent of the object. You would then frame the right portion of the object and where this intersects the cylinder just created. To locate this block, you would measure back along the receding dimension by the appropriate amount. You can then create the true-size front surface of this part of the object and create the oblique pictorial receding back from this.

When a circular feature such as a hole or cylinder appears in a view that is not in the plane of the paper, an ellipse must be drawn using the four-center method discussed in the section describing isometric drawings. However, creating an oblique pictorial with irregular features placed in the receding dimension is rare and should be avoided if at all possible.

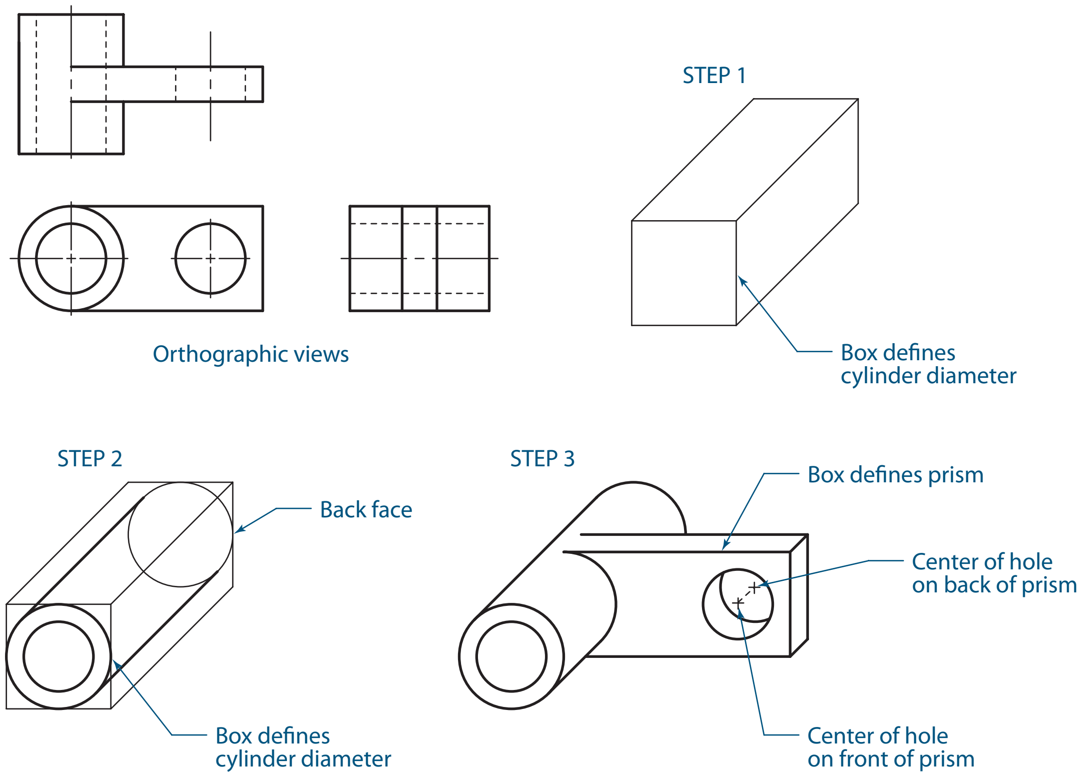

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FIGURE 9.32. The steps required to construct an oblique pictorial with circular features.

9.05 Perspective Drawings

Perspective drawings and sketches are some of the most lifelike pictorials you will create. Someone with a great deal of skill can create a perspective drawing that looks as detailed as a photograph. Perspectives incorporate the concept of vanishing points to produce the 3-D shape of an object on the plane of the paper or the computer screen. The concept of a vanishing point is very simple. Suppose you are standing in the middle of a railroad track in flat terrain so you can see where the sky intersects the terrain. The line where the sky intersects the terrain is called the horizon. As you look down the tracks toward the horizon, the outside rails seem to converge to a single point as the tracks intersect the horizon. This point is called the **vanishing point (VP)**, as shown in Figure 9.33. Perspective pictorials employ the use of vanishing points to create a 3-D effect.

9.05.01 Types of Perspective Drawings

There are three types of perspective drawings: one-point perspectives, two-point perspectives, and three-point perspectives. A one-point perspective is illustrated in Figure 9.33. There is one vanishing point at the horizon, and all of the lines converge to this vanishing point. Figure 9.34 shows a two-point perspective. In this type of perspective, there are two vanishing points—one on the right and one on the left.

FIGURE 9.33. An illustration showing railroad tracks converging to a vanishing point on the horizon.

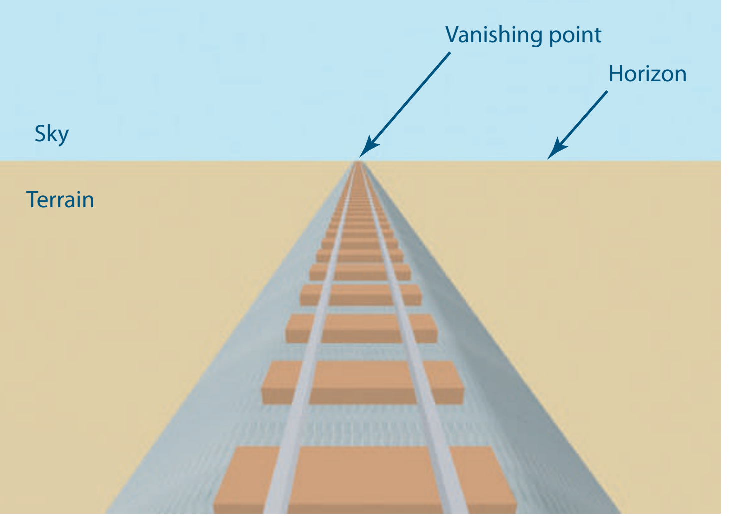

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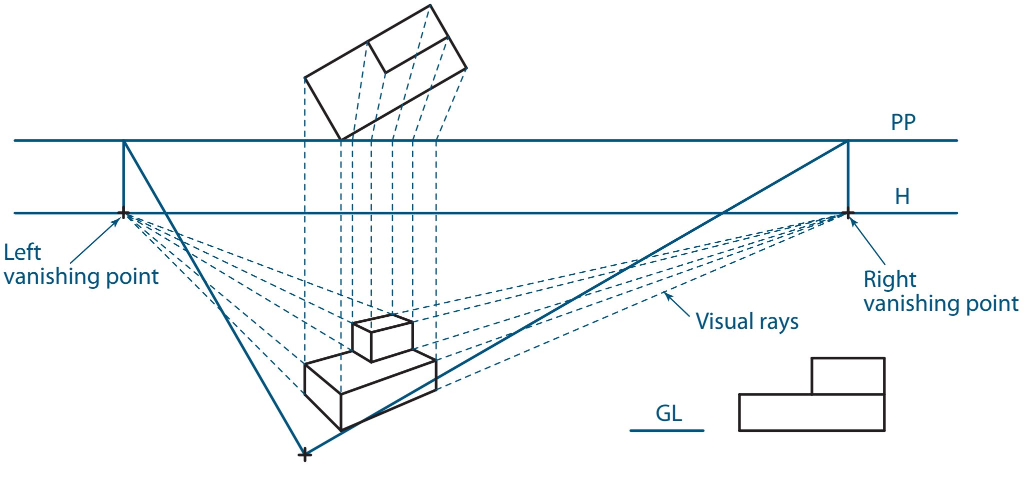
FIGURE 9.34. An example of a two-point perspective drawing.



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Lines that define the 3-D object converge at the vanishing points. A three-point perspective is shown in Figure 9.35. In this type of perspective, there are right and left vanishing points and there is a central vanishing point.

Of the three types of perspectives that can be drawn, the two-point perspective is the one most often used in engineering and architecture to show an object as a pictorial. The one- and three-point perspectives are limited in their use and do not convey an image that the eye would be likely to perceive. Therefore, the two-point perspective will be the only type discussed in detail in this text.

9.05.02 Two-Point Perspective Drawings

Generally, a two-point perspective is generated using the top (plan) orthographic view of the object and an elevation view. The **plan view** is rotated at an appropriate angle to enhance the 3-D aspects of the perspective and the **elevation view** is shown as it normally would be shown in its orthographic position.

An important feature in developing the two-point perspective drawing is the **picture plane (PP)**. The location of this vertical plane (shown as an edge) defines the size and position of the perspective when viewed from the **station point (SP)**. The PP can be located anywhere relative to the object. The simplest position is shown in Figure 9.36, where the PP goes through the corner of the object. When the PP is

FIGURE 9.35. An example of a three-point perspective drawing.

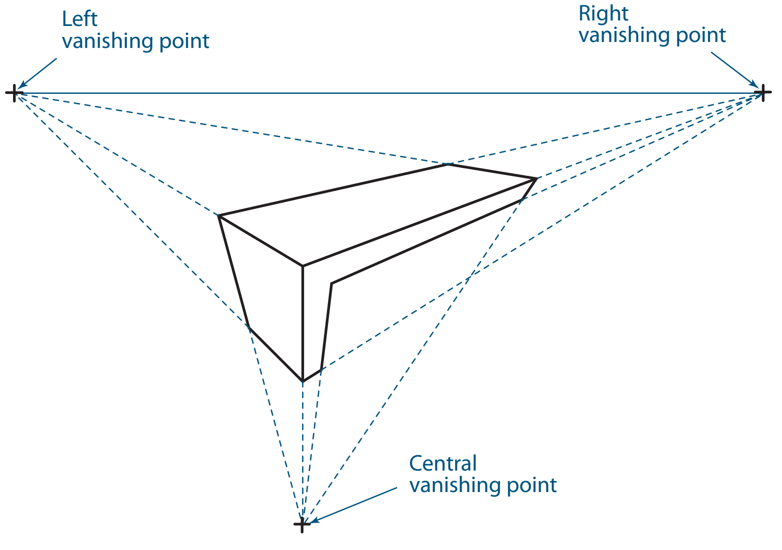

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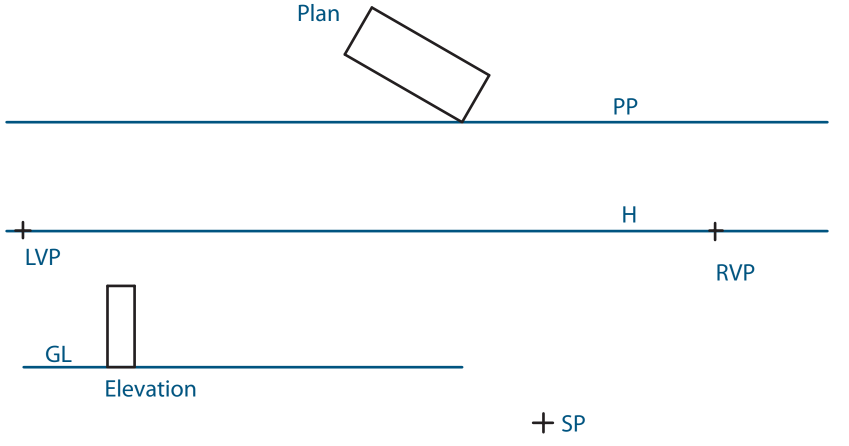
FIGURE 9.36. The relationship between the PP and the object.



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located in front of the object, the perspective appears farther away from the observer. When the PP is located behind the object, the perspective appears closer to the observer. Selection of the position of the PP controls the size of the perspective. At times, illustrators may choose to place the PP through the middle of the object, giving the appearance that one portion of the object is closer to the observer and one portion is farther away from the observer.

The **horizon line (HL)** is the line that defines where the ground meets the sky. It can be placed anywhere between the PP and the SP. The HL is important because it establishes the position of the left and right vanishing points that locate the position of the perspective drawing.

The **ground line (GL)** defines the position of the elevation view of the object. The GL is important because it determines the vertical location of the perspective drawing and serves as a starting point for the drawing, as shown in Figure 9.36.

9.05.03 Construction of a Two-Point Perspective Drawing

The two-point perspective drawing is constructed using steps similar to those for constructing the isometric pictorial and the oblique pictorial. The first thing you should do is to establish the two vanishing points for the pictorial. To do this, you draw lines from the SP to the PP at angles parallel to the left and right visible planes in the plan view and then draw lines from these intersections with the PP vertically downward to the horizon. These intersections become the left vanishing point (LVP) and the right vanishing point (RVP), as shown in Figure 9.37.

The next thing you do is to establish the **measuring line (ML)**. When the corner of the plan view intersects the picture plane, draw the measuring line downward vertically from the intersection of the two, as shown in Figure 9.38. You should now extend the corner of the plan view to the picture plane by drawing parallel to the line used when establishing the vanishing point (SP to PP). Then draw the measuring line down vertically from this intersection with the PP, as shown in Figure 9.39.

Note that these instructions are given for when the PP is in front of the object. When the object is behind the picture plane, you extend the corner of the plan view to the picture plane by drawing parallel to the line used when establishing the vanishing points (SP to PP). Then draw the measuring line downward vertically from the intersection with the picture plane, as shown in Figure 9.40.

The next thing you will do is to project the height measurements to the ML from the elevation view. This is done by projecting horizontally from the elevation view.

FIGURE 9.37. Establishing and marking the left and right vanishing points.

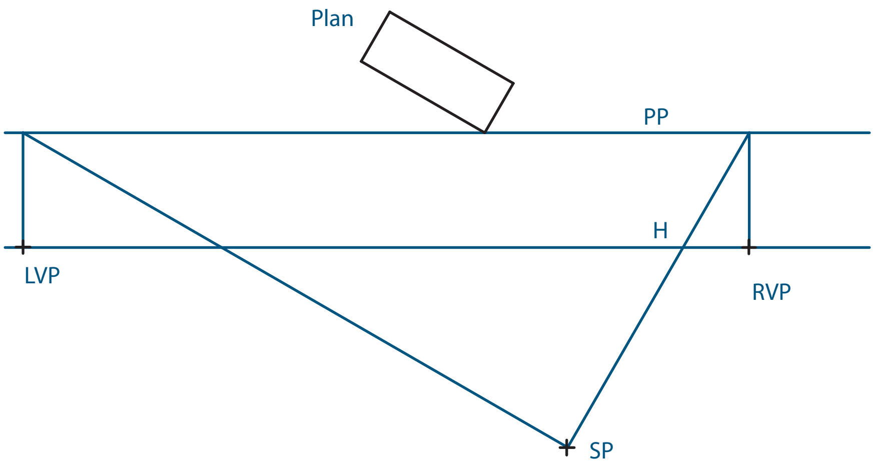

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FIGURE 9.38. Establishing the ML.

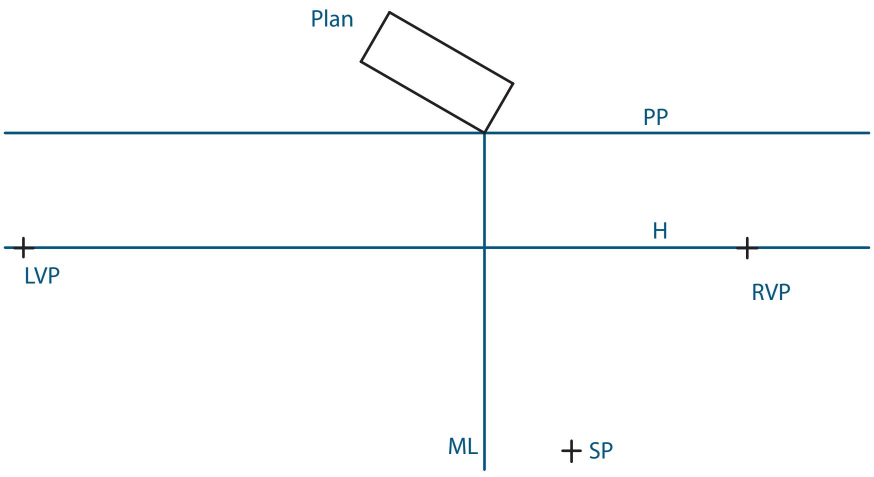


FIGURE 9.39. Establishing the ML when the object is in front of the PP.

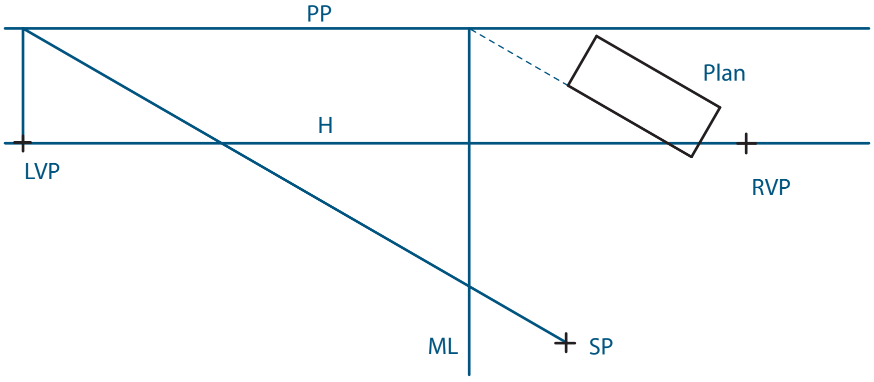
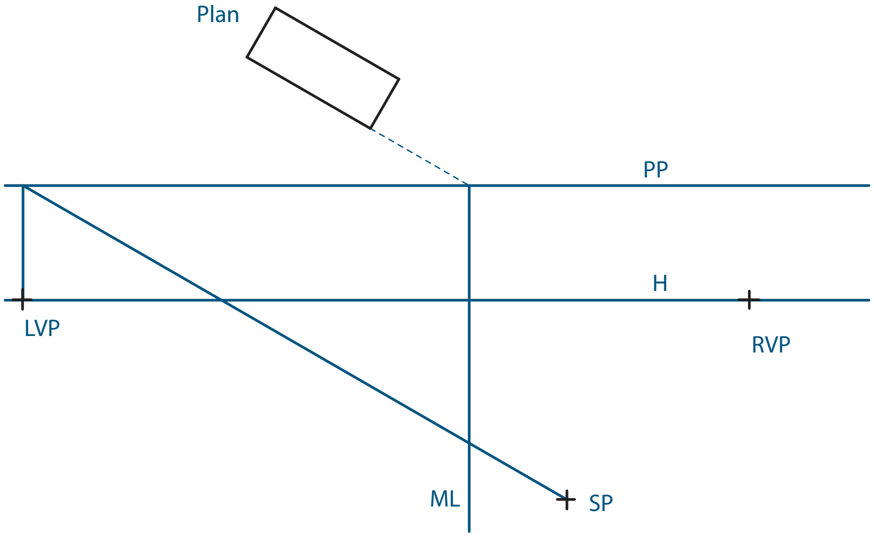


FIGURE 9.40. Establishing the ML when the object is behind the PP.



Note the position of the GL to the elevation view, as shown in Figure 9.41. You will next establish the **measuring walls**. When the visible corner between the left and right visible planes touches the PP, the measuring walls originate at the ML. The lines may be drawn from the top and bottom of the ML to the left and right vanishing points to establish the left and right measuring walls, as shown in Figure 9.42. When the corner that will be represented by the measuring line falls behind or in front of the PP, the measuring wall representing the extended visible plane in the plan view must be drawn first, as shown in Figure 9.43. *Special Note:* The front-near corner must be projected before the measuring wall that represents the other visible plane can be drawn, as shown in Figure 9.44.

You will now draw the visual rays. To do this, align the straight edge from each object intersection in the plan view with the SP, but draw only the plan view intersections to the PP, as shown in Figure 9.45. You will then project the visual rays by drawing the PP intersections downward perpendicular from the PP to the

FIGURE 9.41. Establishing the height of the object on the ML.

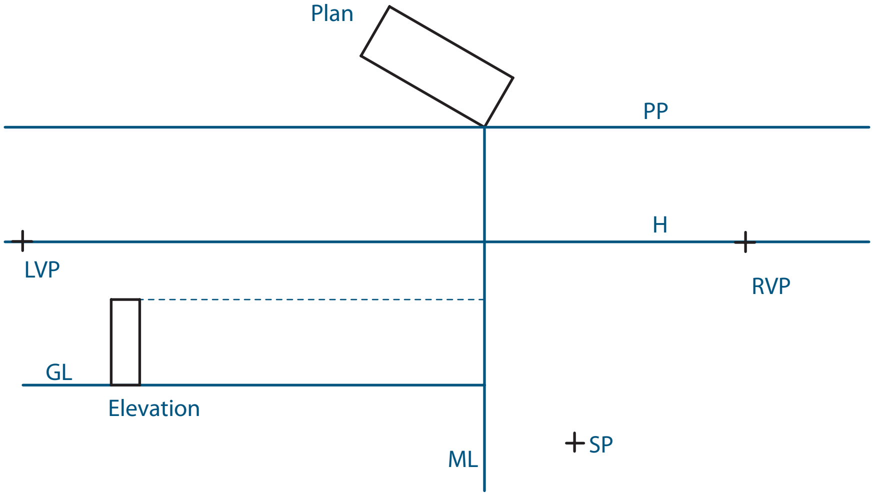


FIGURE 9.42. Establishing the measuring walls when the PP intersects the corner of the object.

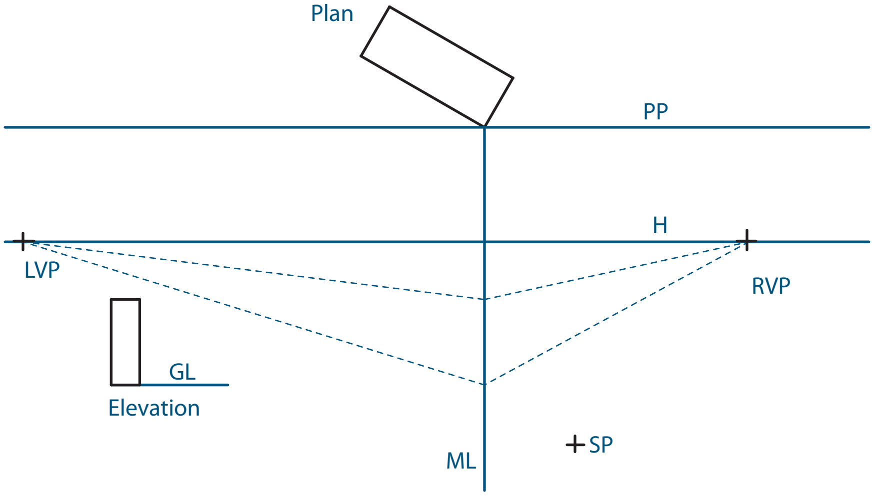


FIGURE 9.43. Establishing the measuring walls when the object is behind or in front of the PP.

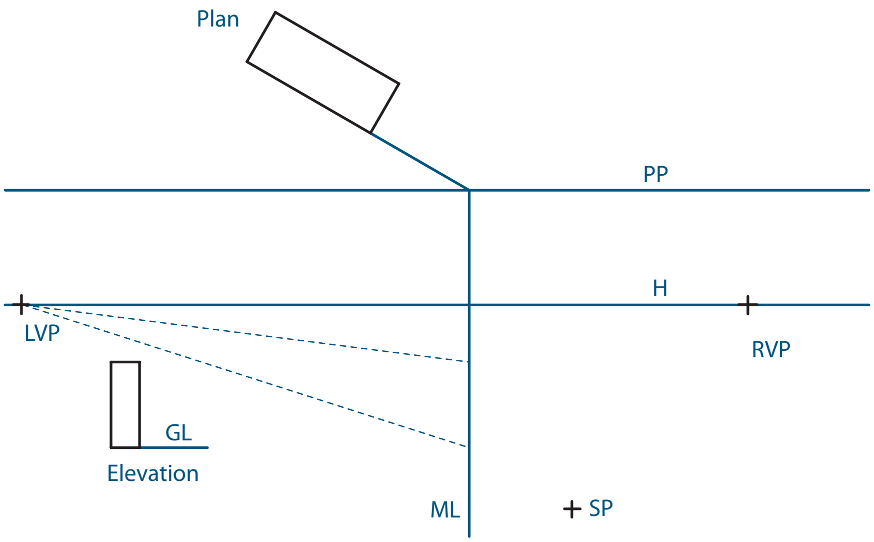


FIGURE 9.44. Projecting the front corner before the measuring wall can be drawn.

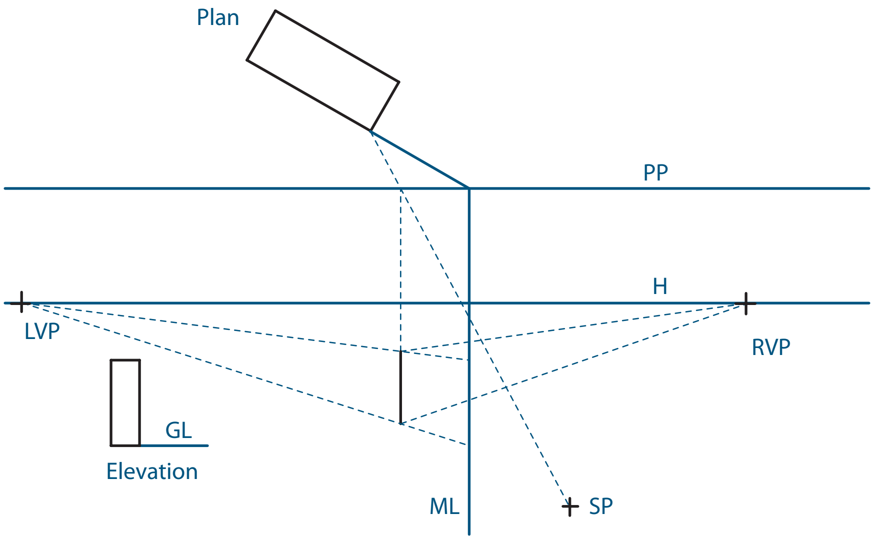
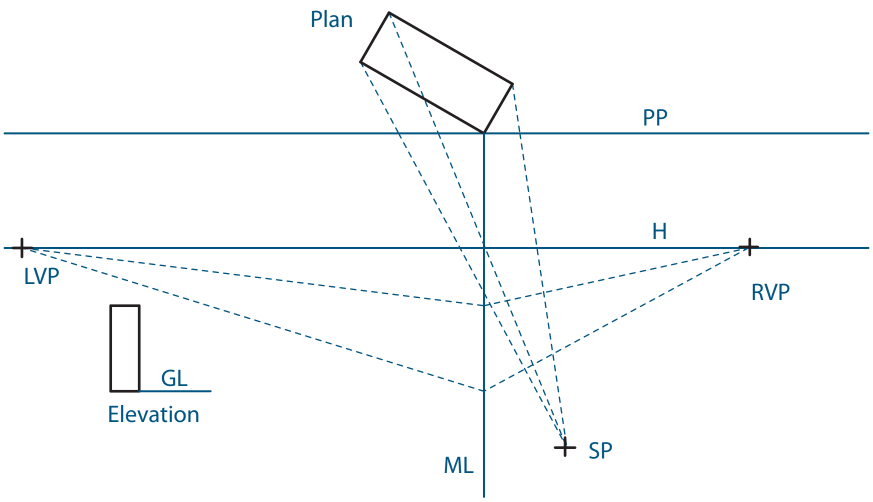


FIGURE 9.45. Drawing the visual rays of the object to the PP through the SP.



measuring walls to establish the side details of the object, as shown in Figure 9.46. Finally, you now need to lay out the details on the drawing. Continue the process outlined here to include the details of the drawing by projecting the appropriate points on the object to the vanishing points, as shown in Figure 9.47.

9.05.04 Complex Object in Two-Point Perspective

When a complex object is composed of more than one prism, the two-point perspective may require more than one measuring line. Figure 9.48 shows an object composed of two prisms, with one of the prisms behind the PP. When this happens, a second measuring line is required to establish the proper height of the prism located behind the PP. Note in Figure 9.48 that ML-1 establishes the height of the rectangular prism forming the base and ML-2 establishes the height of the prism containing the inclined surface that is behind the PP. The heights projected from the elevation view to ML-2 are transferred along a line from ML-2 to the right vanishing point. The projectors of the prism located behind the PP from the plan view are then drawn to the PP and projected downward to these lines to establish the position of the prism that is behind the PP.

FIGURE 9.46. Projecting the intersection of the visual rays with the PP to the measuring walls.

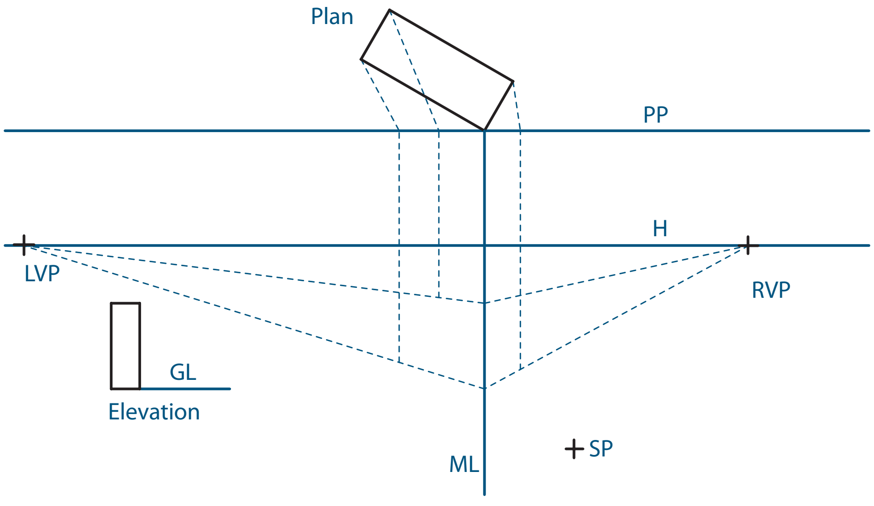
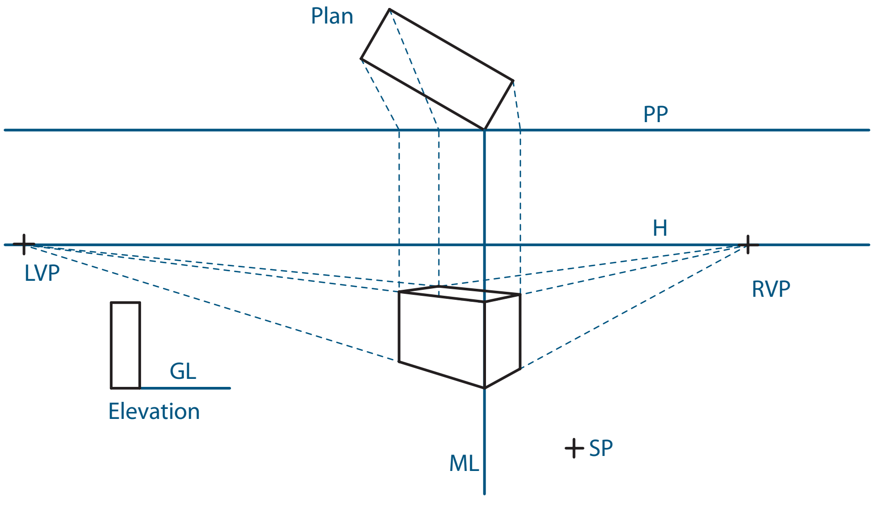


FIGURE 9.47. Completion of the perspective.



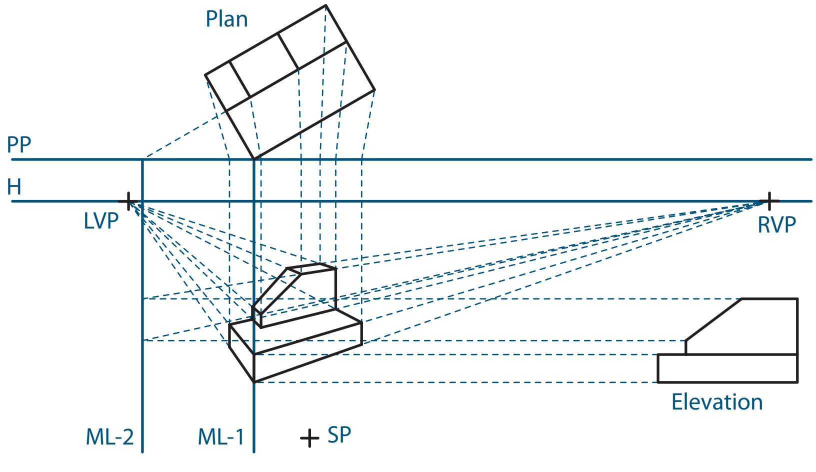


FIGURE 9.48. A two-point perspective drawing of a complex object.

9.06 Considerations for 3-D Modeling

The easiest way to create pictorial drawings is to extract them from solid models. Most solid modeling software has the capability to create engineering drawings from models. These drawings usually include not only traditional orthographic views but also pictorial views to increase the speed with which the parts and assemblies can be visualized. Creating a pictorial from a solid model usually is a matter of specifying the viewing orientation (many times predetermined to give a choice of an isometric or trimetric view) and the amount of perspective when a perspective view is desired. In fact, pictorial drawings are so easily extracted from solid models that it might be foolish not to include them with the orthographic views on working engineering drawings.

9.07 Chapter Summary

Pictorial drawings are designed to assist in providing a clear picture of an object that may be difficult to visualize from just a set of multiview drawings. Since pictorial drawings describe all three dimensions on the plane of the paper, they are less likely to show the detail that would be expected in the orthographic drawings used in working drawings. Visualization and an understanding of the 3-D relationships of objects are greatly enhanced through the use of pictorial drawings. Different levels of complexity are involved in creating different types of pictorial drawings. For simple communication, isometric drawings usually can be created easily for most objects. Oblique drawings, albeit less realistic in their appearance, are even quicker and simpler to create. For applications that demand the most realistic appearance, especially for large objects such as buildings, perspective drawings can be used. With a solid modeler, pictorial drawings of any type can be created quickly (after the solid model is created) with a few commands. When 2-D CAD or manual drafting instruments are the only tools available, the traditional techniques presented in this chapter may need to be used. Regardless of the graphics tools available, pictorial drawings are now commonly included in formal engineering drawing to add clarity to the traditional orthographic multiview presentation.

9.08 GLOSSARY OF KEY TERMS

axonometric drawing: A drawing in which all three dimensional axes on an object can be seen, with the scaling factor constant in each direction. Usually, one axis is shown as being vertical.

cabinet oblique drawing: An oblique drawing where one half the true length of the depth dimension is measured along the receding axes.

cavalier oblique drawing: An oblique drawing where the true length of the depth dimension is measured along the receding axes.

dimetric drawing: An axonometric drawing in which the scaling factor is the same for two of the axes.

elevation view: In the construction of a perspective view, the object as viewed from the front, as if created by orthogonal projection.

ground line (GL): In the construction of a perspective view, a line on the elevation view that represents the height of the ground.

horizon line (HL): In the construction of a perspective view, the line that represents the horizon, which is the separation between the earth and the sky at a long distance. The left and right vanishing points are located on the HL. The PP and the HL are usually parallel to each other.

isometric drawing: An axonometric drawing in which the scaling factor is the same for all three axes.

isometric lines: Lines on an isometric drawing that are parallel or perpendicular to the front, top, or profile viewing planes.

measuring line (ML): In the construction of a perspective view, a vertical line used in conjunction with the elevation view to locate vertical points on the perspective drawing.

measuring wall: In the construction of a perspective view, a line that extends from the object to the vanishing point to help establish the location of horizontal points on the drawing.

oblique pictorial: A sketch of an object that shows one face in the plane of the paper and the third dimension receding off at an angle relative to the face.

perspective drawing: A drawing in which all three-dimensional axes on an object can be seen, with the scaling factor linearly increasing or decreasing in each direction. Usually, one axis is shown as being vertical. This type of drawing generally offers the most realistic presentation of an object.

pictorial: A drawing that shows the 3-D aspects and features of an object.

picture plane (PP): In the construction of a perspective view, the viewing plane through which the object is seen. The PP appears as a line (edge view of the viewing plane) in the plan view.

plan view: In the construction of a perspective view, the object as viewed from the top, as if created by orthogonal projection.

station point (SP): In the construction of a perspective view, the theoretical location of the observer who looks at the object through the picture plane.

trimetric drawing: An axonometric drawing in which the scaling factor is different for all three axes.

vanishing point (VP): In the construction of a perspective view, the point on the horizon where all parallel lines in a single direction converge.

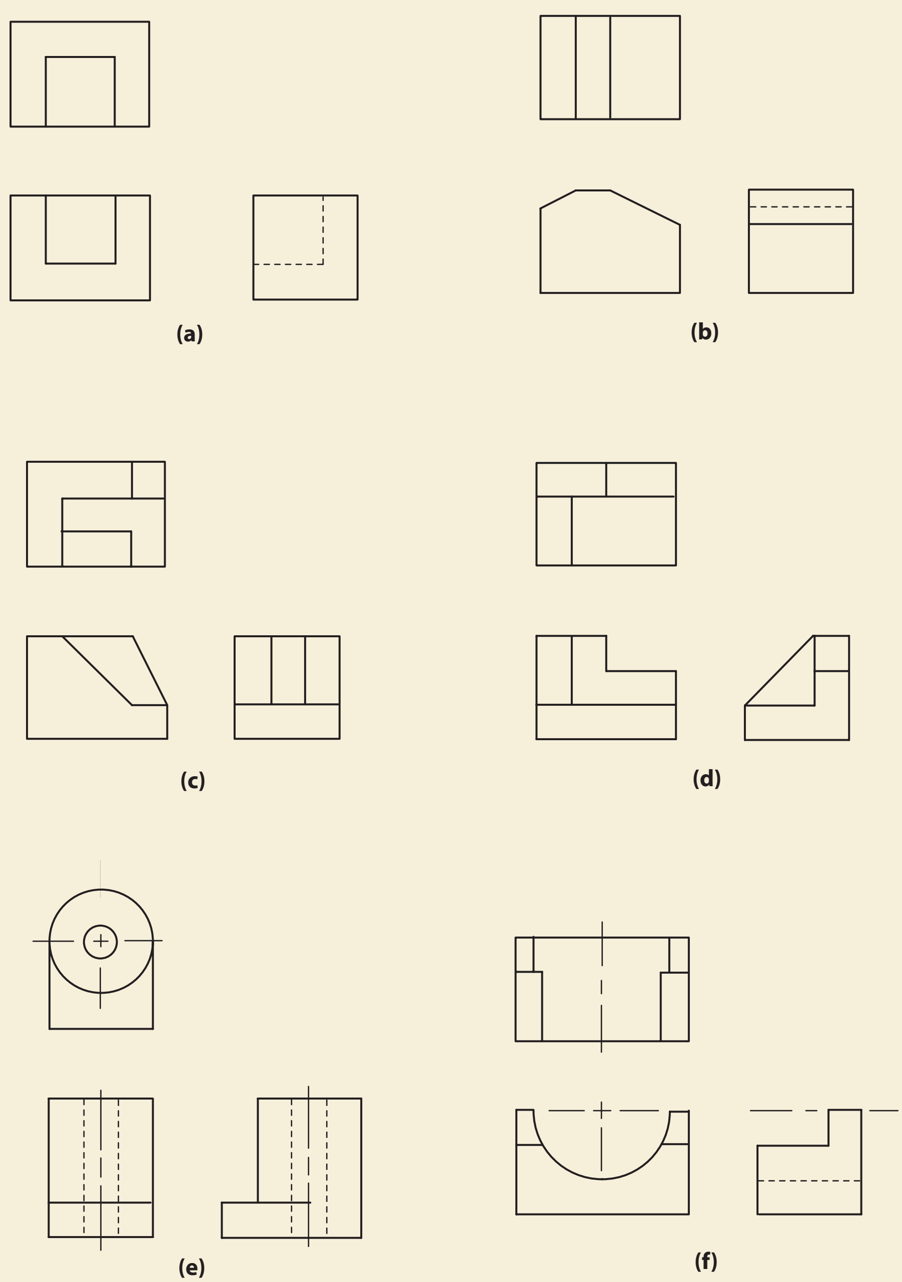
9.09 QUESTIONS FOR REVIEW

1. Why are pictorial drawings useful?
2. When should pictorial drawing be used instead of pictorial sketches?
3. Why should pictorial drawings not be used with working drawings to produce parts?
4. What is an axonometric drawing?
5. How do isometric, dimetric, and trimetric drawings differ?
6. How do isometric and oblique drawings differ?
7. In what way is an oblique drawing nonrealistic?
8. How do cabinet and cavalier oblique drawings differ?
9. How do isometric and perspective drawings differ?
10. In what way are perspective drawings more realistic than isometric drawings?
11. When should perspective drawings be used in favor of axonometric drawings?
12. Why are two-point perspective drawings more common than one- or three-point perspective drawings?

9.10 PROBLEMS

Measure the features shown in the front, top, and right-side views of the objects shown in Figure P9.1. Using drafting instruments or CAD, create the following scaled pictorials of each object that is represented.

1. An isometric drawing
2. A cabinet oblique drawing
3. A cavalier oblique drawing
4. A trimetric drawing using your choice of axes angles (one axis must be vertical).
5. A two-point perspective drawing using your choice of plan location and orientation, station point, and vanishing points. The height axis must be vertical, and the two vanishing points must be on the same HL.



9.10 PROBLEMS (CONTINUED)

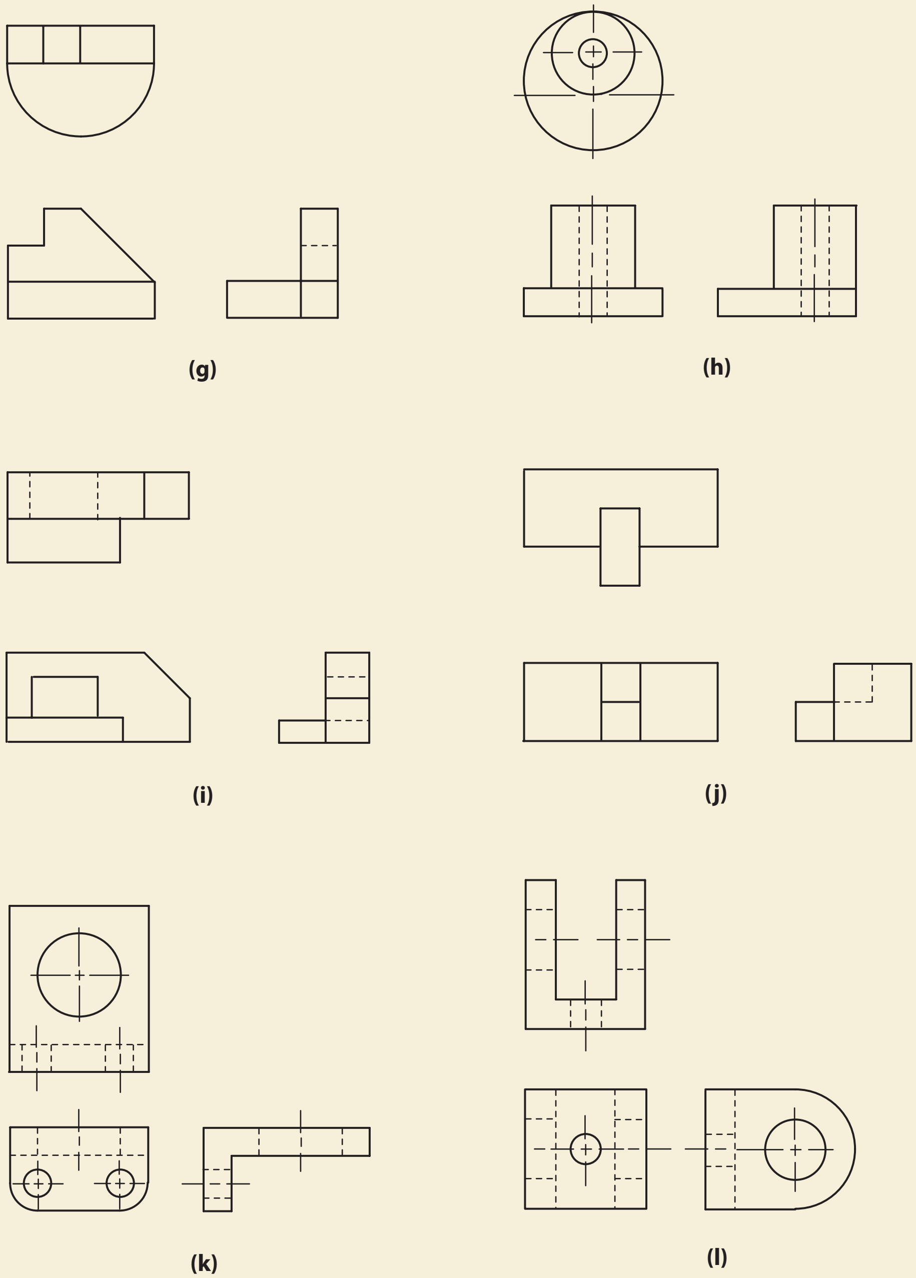


FIGURE P9.1.

CHAPTER 10

SECTION VIEWS

OBJECTIVES

After completing this chapter, you should be able to

- Use cutaway, or section, views as a method for showing the features of a part that are normally hidden when presented on a multiview drawing
- Decide when a section view is necessary
- Decide what category of section view should be used for particular circumstances
- Create a desired section view such that it adheres to accepted engineering drawing practices

10.01 INTRODUCTION

The precisely aligned images in a multiview drawing offer an excellent start in defining the exact geometry needed for a part that you want to build. However, this description alone may not be adequate to define all of the features in many types of parts. Some features may be partially or fully obscured in the standard views. The use of hidden lines can alleviate the problem, but too many hidden lines may cause confusion. In these cases, it is useful to have a means of revealing proposed interior detail. This is done by showing cross sections, or section views, at important locations. As with multiview drawings, to minimize ambiguity, you must follow certain guidelines when you want to present a section view.

10.02 A Look Inside

Pick up an everyday object (for example, a coffee mug) and look at it from all directions. If you cannot find a coffee mug, some images have been provided for your convenience in Figure 10.01. You will notice that you cannot view the mug from a direction where the inside depth of the cup or the thickness of the bottom can be directly measured (unless it is made of a clear material). If the mug has a handle on it, look at that as well. Are the edges of the handle rounded? Can you look at the handle from a direction where you get an undistorted view of the radius of the edge? These features are simple examples of measurements that cannot be made from looking at an object in a multiview drawing. Yet there must be some means of showing these types of features so a fabricator will know what to make and what sizes are required. A coffee mug is a very simple example.

Here is an industrial example. Consider the Hoyt AeroTec bow handle again. Its image and multiview engineering drawing are shown in Figure 10.02, with its cross brace highlighted. Note that the edges of the cross brace are rounded. Can these rounded edges be seen on the drawing? Assume the edges of the cross brace are not rounded (i.e., the surfaces meet at a 90-degree angle). How would the drawing change? The answer is that in its current state, with all of the complexity and exquisite detail, the drawing cannot show the existence of rounded edges. Clearly, something must be added to the drawing to show that these edges are rounded and to what size they are rounded.

FIGURE 10.01. Two views of an object (a coffee mug) with interior detail.

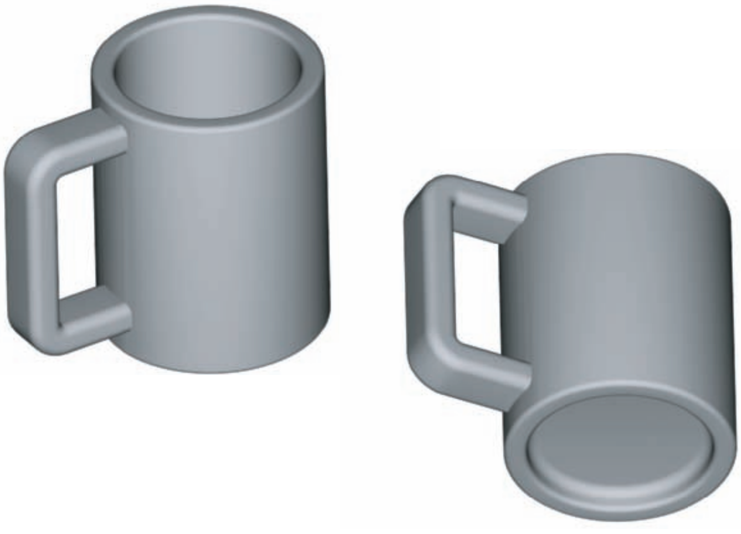


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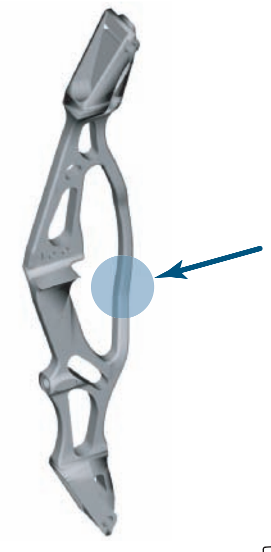
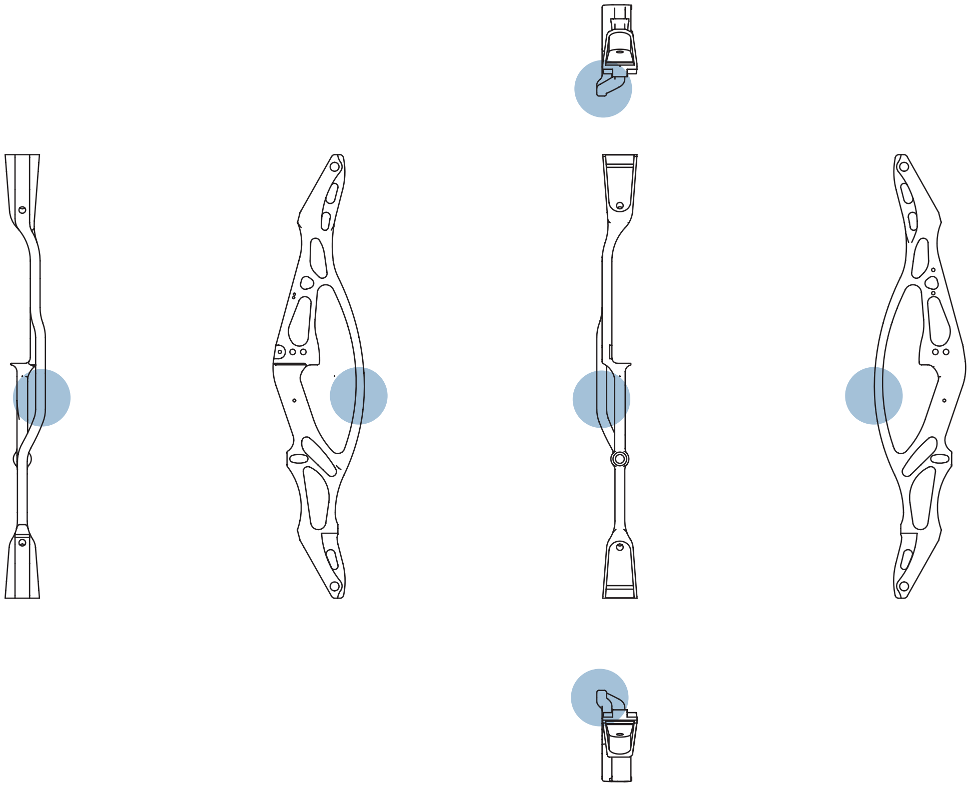
FIGURE 10.02. The geometry of the cross brace on the AeroTec riser cannot be seen in the multiview drawing.





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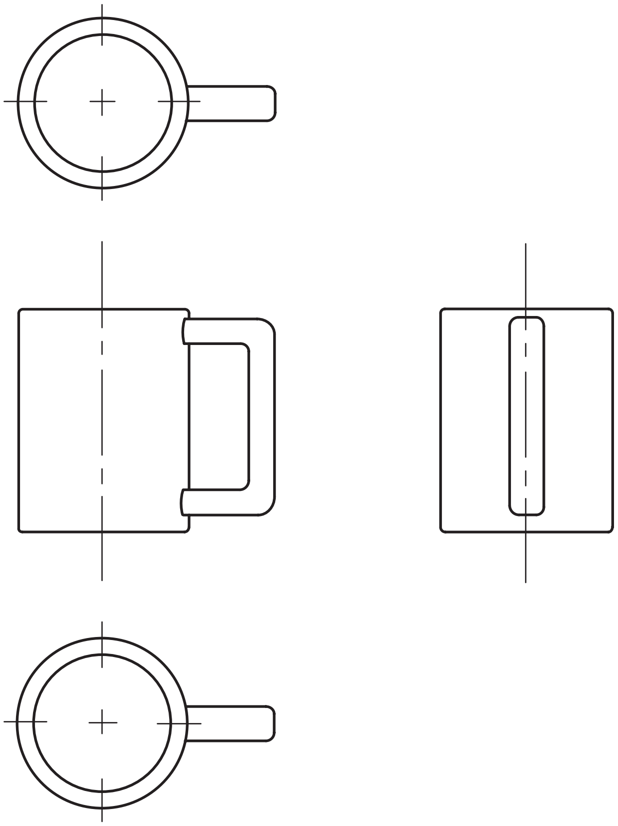
Backing up a bit, look at the mug again to find out what is causing the problem. Figure 10.03 shows the multiview engineering drawing of the mug. Note that the depth of the mug and the radius of the edges of the handle cannot be seen on this drawing. The reason is because the object gets in the way of itself. Portions of the object obscure other portions of the same object. The outside of the mug hides the inside.

A possible solution to this problem is to use hidden lines, as shown in Figure 10.04. The hidden lines show the depth and geometry of the inside of the mug, as well as the geometry of the edges on the handle. However, the use of hidden lines is not always an ideal solution. As objects become more complicated, too many hidden lines make the views confusing, particularly when the images of different features start to fall atop one another.



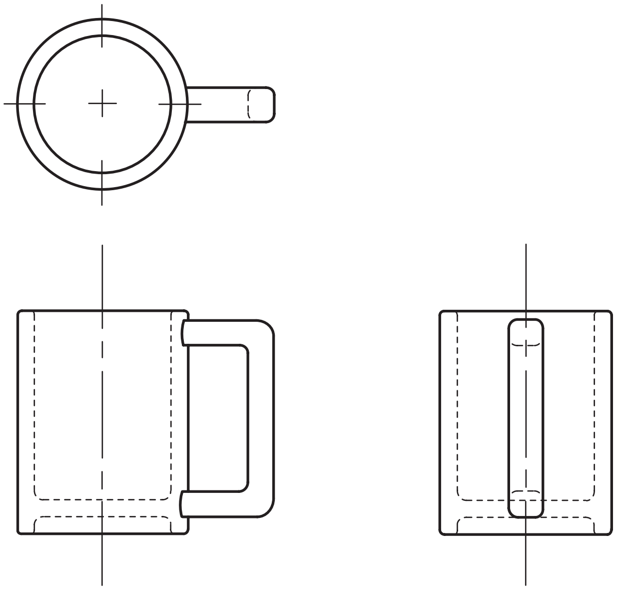


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FIGURE 10.04. A multiview drawing of the coffee mug using hidden lines to show interior detail.

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FIGURE 10.03. Orthographic views of the coffee mug fail to define interior detail.

In Chapter 3, you learned about cross sections of 3-D objects. If there were a way to cut the mug open, as shown in Figure 10.05, you could take the sliced part and turn it around until you were able to see the desired geometry. This hypothetical slicing is the essence of creating a cross section of the object, to create what is called a **section view**. The slicing, however, must be done following certain rules to ensure that the person who sees a section view on a drawing knows exactly where the slicing has occurred and how it was performed.

A drawing of the mug with three orthogonal views and two types of section views is shown in Figure 10.06. Do not worry if you have difficulty understanding the extra views in this figure. The following sections discuss in detail how various types of section views are made and how they should be interpreted.

FIGURE 10.05. Hypothetical cutting of the object to reveal interior detail.

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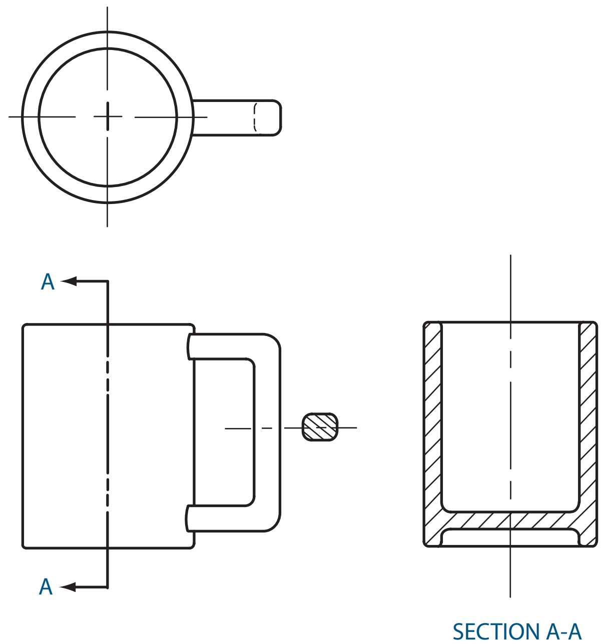
FIGURE 10.06. A multiview drawing of the coffee mug using section views to show interior detail.



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10.03 Full Sections

The simplest section view is the **full section**. In a full section, the object is cut completely apart by a **cutting plane** that is perpendicular to one of the standard viewing planes, such as the front, top, or side views. The image of the original whole object is made on the viewing plane using orthogonal projection, and the cutting plane is seen in edge view. A good way to think of a cutting plane is as a very thin knife with the blade held perpendicular to the viewing plane, which hypothetically splits the part into two pieces. This process is shown in Figure 10.07. Note that the cutting plane has an associated **viewing direction**, identified by a set of arrows pointing in the direction of the freshly cut surface that is to be viewed. To create the section view, the image of the split part is imprinted on the cutting plane. The cutting plane and the image are then rotated away from the split part until it is coplanar with the viewing plane. The hinge for this rotation of the cutting plane is its intersection with the viewing plane. With this definition of a section view and its location on the viewing plane, the alignment and orientation of the section view is the same as that used to create an orthogonal view. The section view is the image of the cut object as seen through the cutting plane, and this image is then placed on the viewing plane. In essence, a full section view is just another orthogonal view, but one that reveals the interior of the object.

On an engineering drawing, the original images of the object are not cut apart, as shown in Figure 10.08. A heavy line that extends across the entire part, with alternating short-short-long dashes, represents the edge view of the cutting plane. This line is called a **cutting plane line**. The orientation of the section view relative to the original view of the object is the same as if the viewing plane and the cutting plane were orthogonal viewing plans that had been unfolded. **Section lines**, which are a form of shading, are used to identify areas on the section view that are solid on the original whole object.

There are some important things to note in a full section on an engineering drawing. First, the cutting of the part is imaginary. The part is not to be split into separate pieces. The use of a full section is similar to saying, "If we imagine that the part was cut here, this is what we would see." The cutting plane is flat and goes all the way through the object. Notice the pairs of large, bold capital letters on the cutting plane line next to the arrows. These are used for unique identification of cutting plane lines and their associated section views on a drawing. If the letter *A* is used beside both arrows on

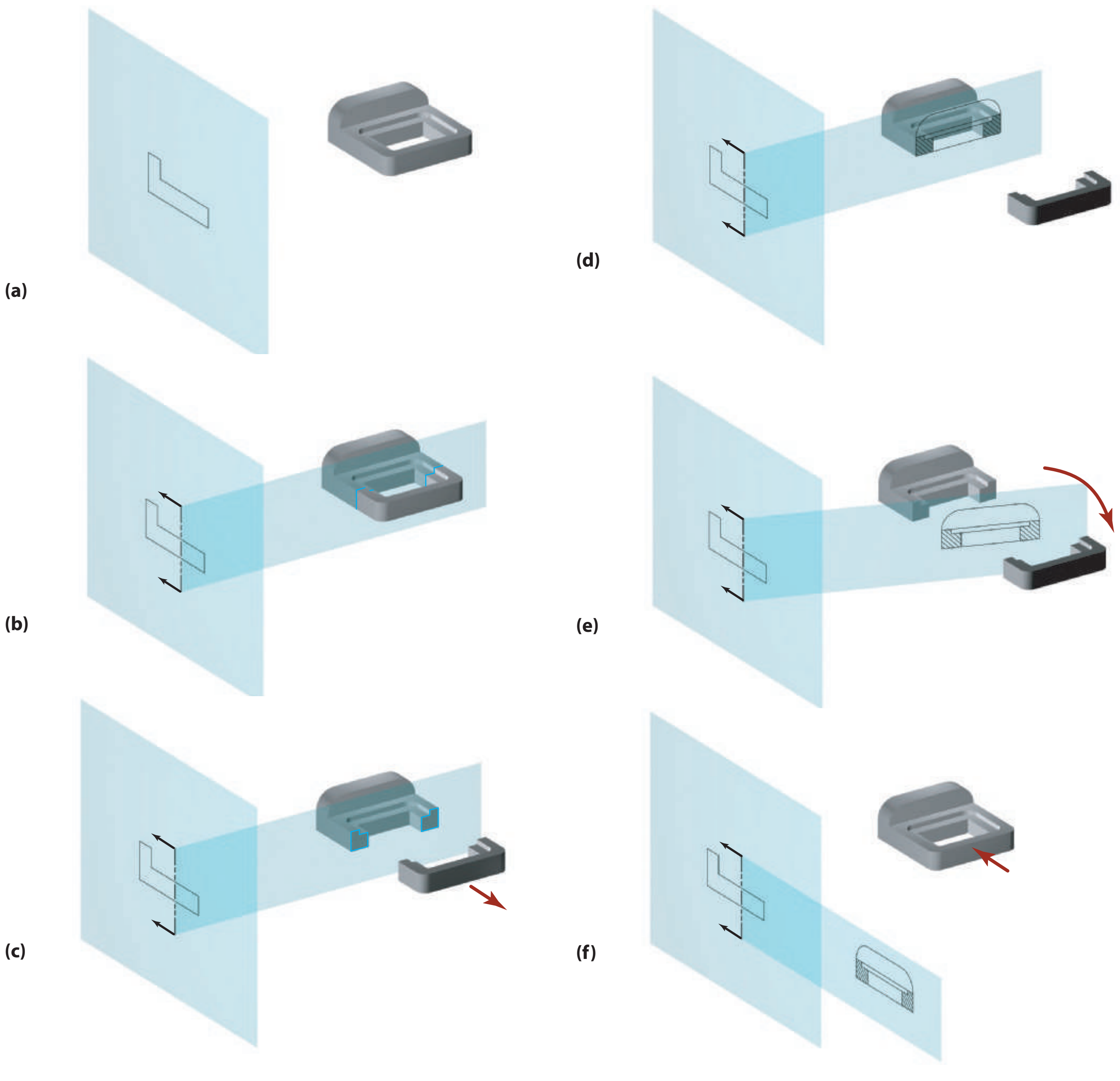
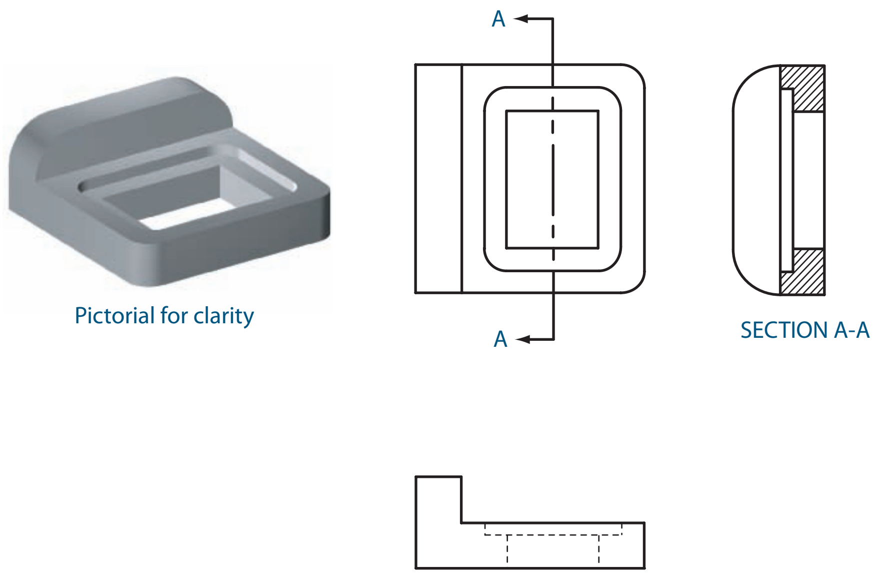


FIGURE 10.07. Creating a full section. An object is projected onto a viewing plane in (a). A cutting plane orthogonal to the viewing plane slices the object in (b). The piece to be viewed remains, while the other piece is removed in (c). The projection of the sliced object is made on the cutting plane in (d). The cutting plane and image are rotated about the section line in (e). The section view is coplanar with the viewing plane in (e).

a cutting plane line, there must be a note immediately below the corresponding section view that identifies it as "SECTION A-A." The arrows on the cutting plane line point in the direction of viewing. This last point is important because the viewing direction and the orientation of the section view are not arbitrary. An error in either may cause confusion for the reader. Try to visualize the cutting process by comparing Figure 10.07 with Figure 10.08. Correlate the 3-D cutting process in Figure 10.07 with what is shown on the 2-D representation in Figure 10.08. The arrows on the cutting plane point are in the same direction as the arrows on the cutting plane line.

FIGURE 10.08. An engineering drawing with a section view to reveal interior detail.



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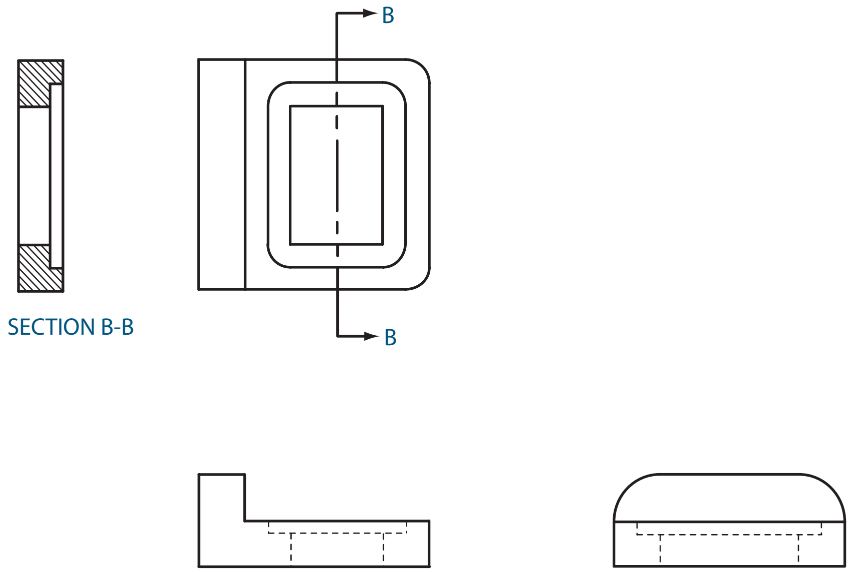
If the arrows on the cutting plane and its corresponding cutting plane line were reversed, as shown in Figure 10.09, the section view would be slightly different. Although the surface that is created by the cutting operation would be the same, the background image of the part would be different. This change is due to the fact that you would be retaining and looking at the other piece that was created when the part was hypothetically split compared to the case in Figure 10.08. When working with multiview drawings, you can remember the proper orientation of the section view by noting that it has the same orientation as the orthogonal view opposite to which the cutting plane line arrows point. For example, if the cutting plane line was located on the front view (and its arrows pointed away from the right-side view), the associated section view would have the same orientation and alignment as the right-side view.

FIGURE 10.09. An engineering drawing with a section view to reveal interior detail.



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The drawing in Figure 10.10 shows a part with multiple section views. If a drawing has multiple section views, a pair of letters must uniquely identify each set of cutting plane lines and corresponding section views. So if there is a second cutting plane line and corresponding section view on the drawing, it may be identified as "SECTION B-B" if "SECTION A-A" already exists. The third set may be called "SECTION C-C," and so on. These identification labels are customarily used even when a drawing has only one section view. The hypothetical interpretation of the multiple sections on the object in Figure 10.10 is shown in Figure 10.11.

One way section views differ from conventional orthogonal views is that, in practice, section views are not required to remain aligned with their adjacent orthogonal views. Although breaking this alignment may violate the rules of orthogonal projection used to create a section view, it is allowed for convenience. Figure 10.12, for example, shows multiple section views of the part. One section is aligned with the view in which it was created. The other section views are nonaligned, but this is permitted in engineering drawing. However, note that even when the section views are nonaligned, they still are required to maintain the same rotational orientation as if they were aligned.

FIGURE 10.10. Multiple section views on a single object.

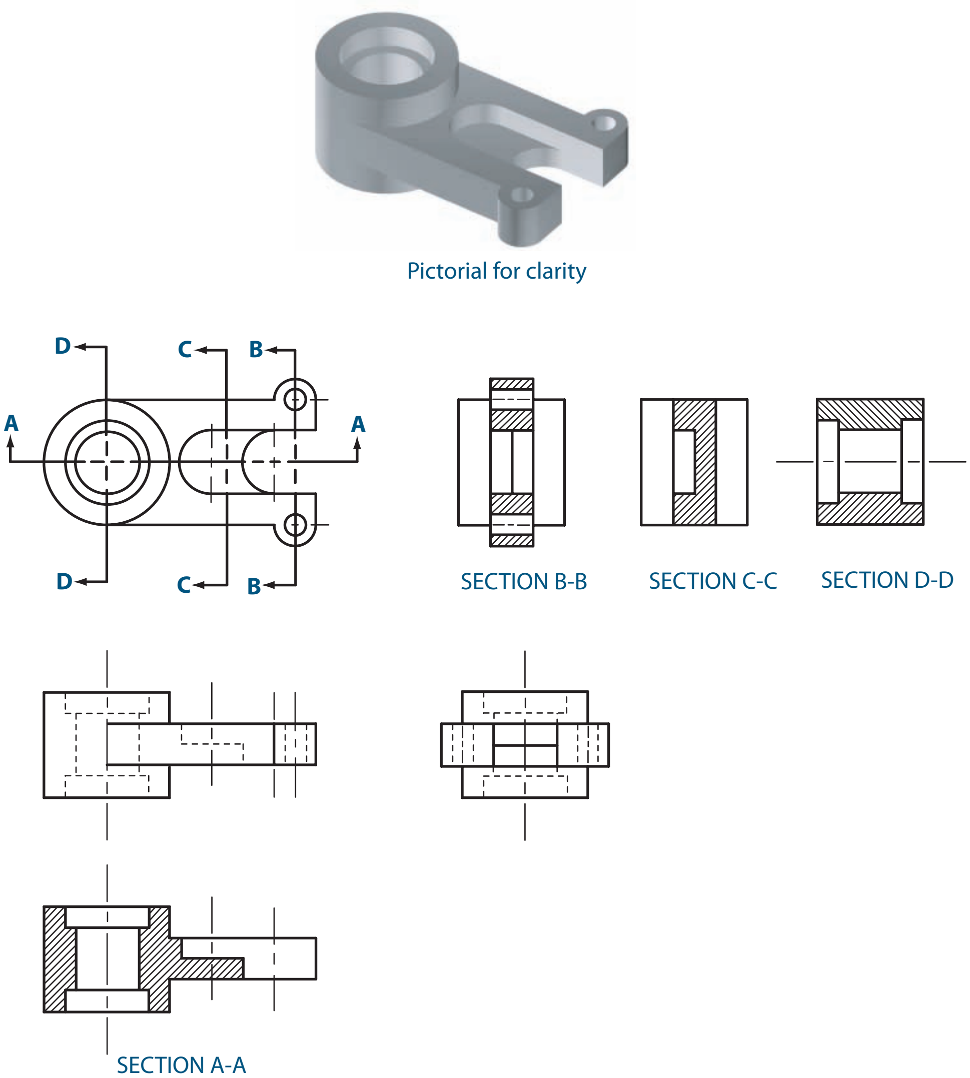

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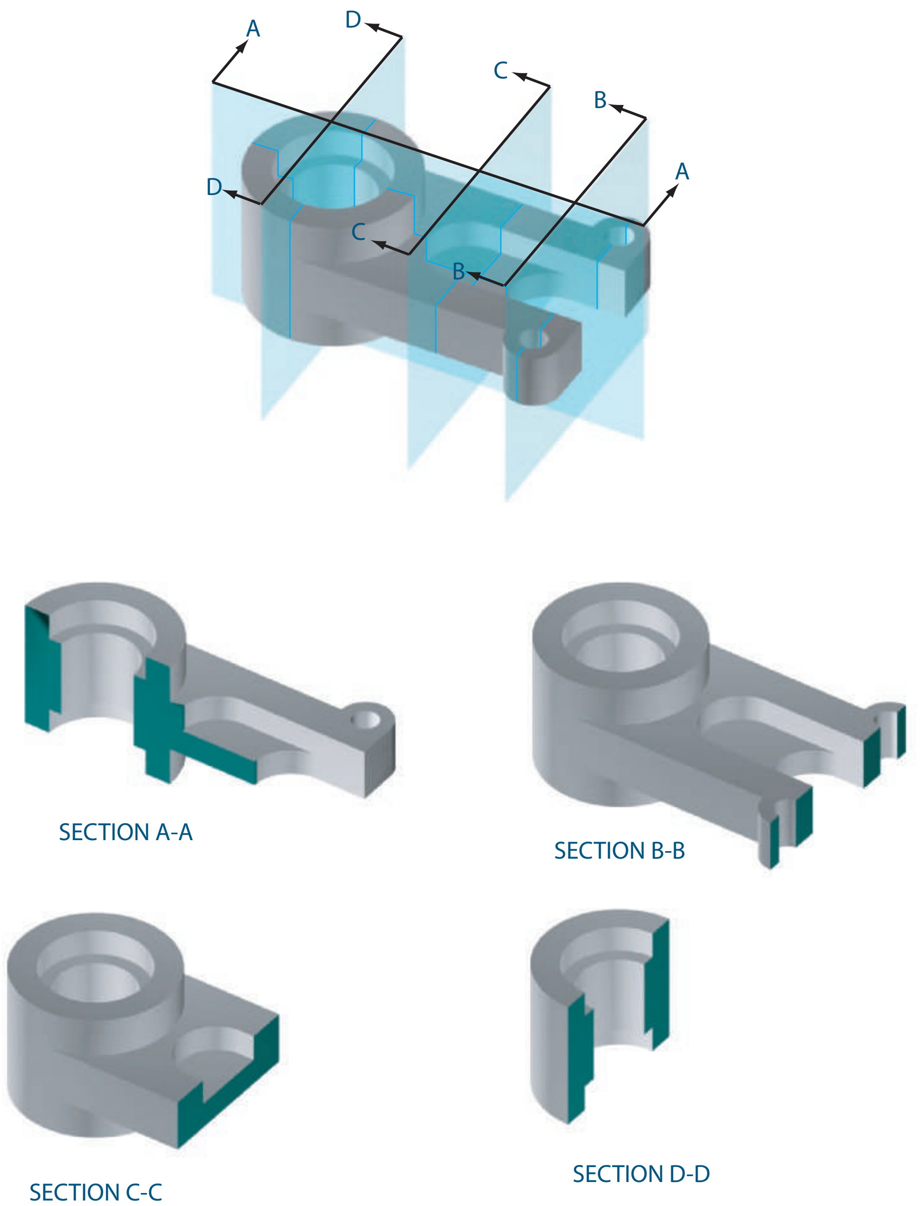
FIGURE 10.11. A hypothetical interpretation of the cutting planes for the previous figure.



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Another difference between a section view and a conventional orthogonal view is that a section view is permitted to have a different scale, or magnification, than the view in which it was created. An example of this property is shown in Figure 10.12, where three of the section views are magnified to reveal detail inside the part that would otherwise be difficult to see. When a section view uses a scale that is different from that of the principal views, the new scale must be clearly marked below the note used to identify the section view, as shown in Figure 10.12.

When a section view is created, even though the cutting plane is perpendicular to the viewing plane, there is no requirement that the cutting plane be parallel or perpendicular to any of the other orthogonal views. This property of section views makes it convenient to view features that may be placed at odd angles with respect to the principal views, as with the part shown in Figure 10.13.

FIGURE 10.12. Multiple sections with nonaligned section views and different scales.

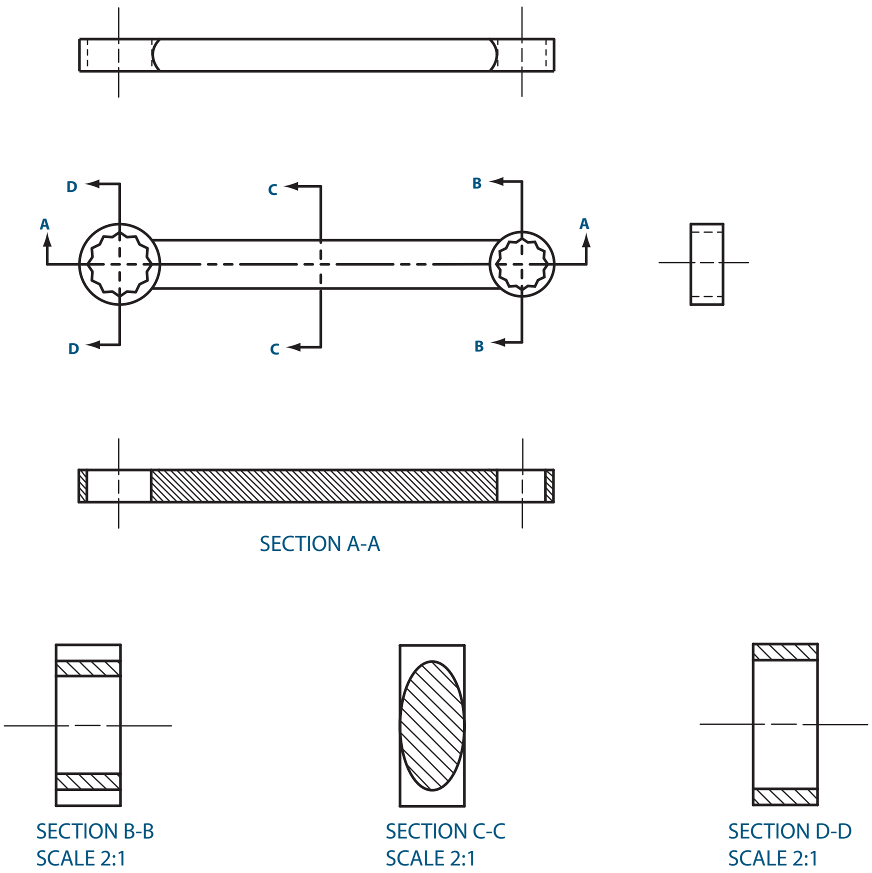

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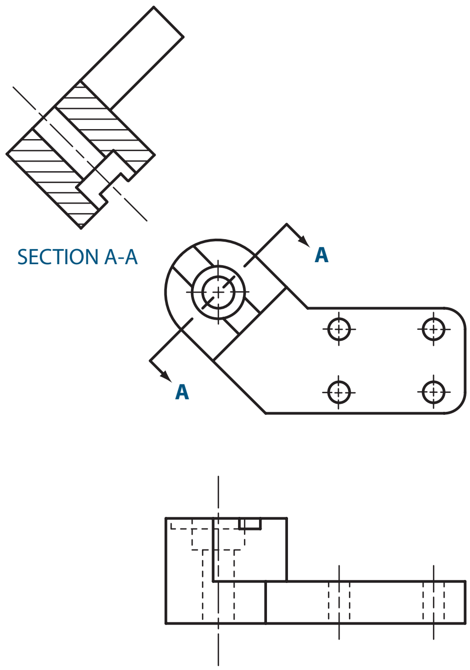
FIGURE 10.13. A full section through a feature placed at an angle.



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10.04 What Happens to the Hidden Lines?

One of the main incentives for using section views is to reduce the use of hidden lines, which until now has been the only method available for revealing the interior and hidden features of many types of objects. When there are too many hidden lines on the view of an object, the drawing becomes confusing. Replacing those hidden lines with one or more section views greatly clarifies the drawing. When section views are used in this manner, there is no longer any need to retain the hidden lines; and they can be removed from the drawing. Hidden lines are typically not shown on the section-line-filled portions of a section view except to indicate the presence of screw threads. Figure 10.14 compares an example of a drawing that originally contained many hidden lines with a revised drawing that replaces one of the orthogonal views with a section view. The improvement in clarity is substantial.

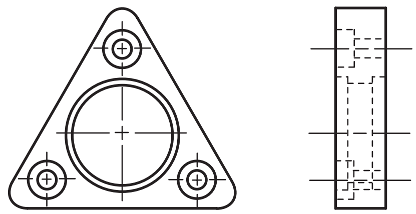
10.05 The Finer Points of Section Lines

Section lines are used to improve the clarity of a section view by indicating the portions of the part that had been solid at the location it was hypothetically cut. However, indiscriminate section line patterns may cause more confusion than clarification. The most basic pattern is a set of lines with a common inclination angle, thickness, and spacing, as shown in Figure 10.15. The line thickness for the pattern is usually no thicker than that used for the part edges. Even with this simple set of variables, the pattern requires some thought. The pattern must be discernible as being section lines when the drawing is read, and the pattern must be reproducible without significant distortion occurring when the drawing is copied. For example, optical copiers, scanners, and fax machines can greatly distort a high-density pattern. A low-density pattern may not appear as section line patterns at all, and the section lines may be misinterpreted as edges on the part.

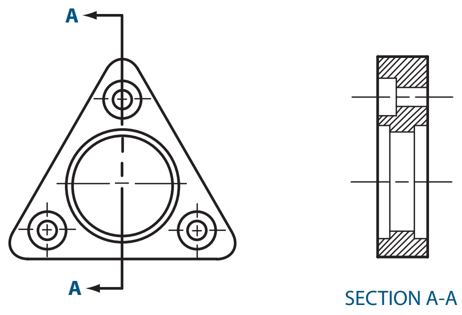
The pattern should not be parallel or perpendicular to any of the major feature edges of the part; otherwise, there may be some confusion about which lines are part edges and which are section lines. Vertical and horizontal lines are rarely used for section lines.

Different section line patterns can be used to represent different materials. Sample patterns are shown in Figure 10.16. For some materials used in construction, such as concrete or earth, section line patterns are more of a texture than a simple geometric pattern.

FIGURE 10.14. The need for many hidden lines in the original drawing (a) is reduced by the use of a section view (b).



(a)



(b)

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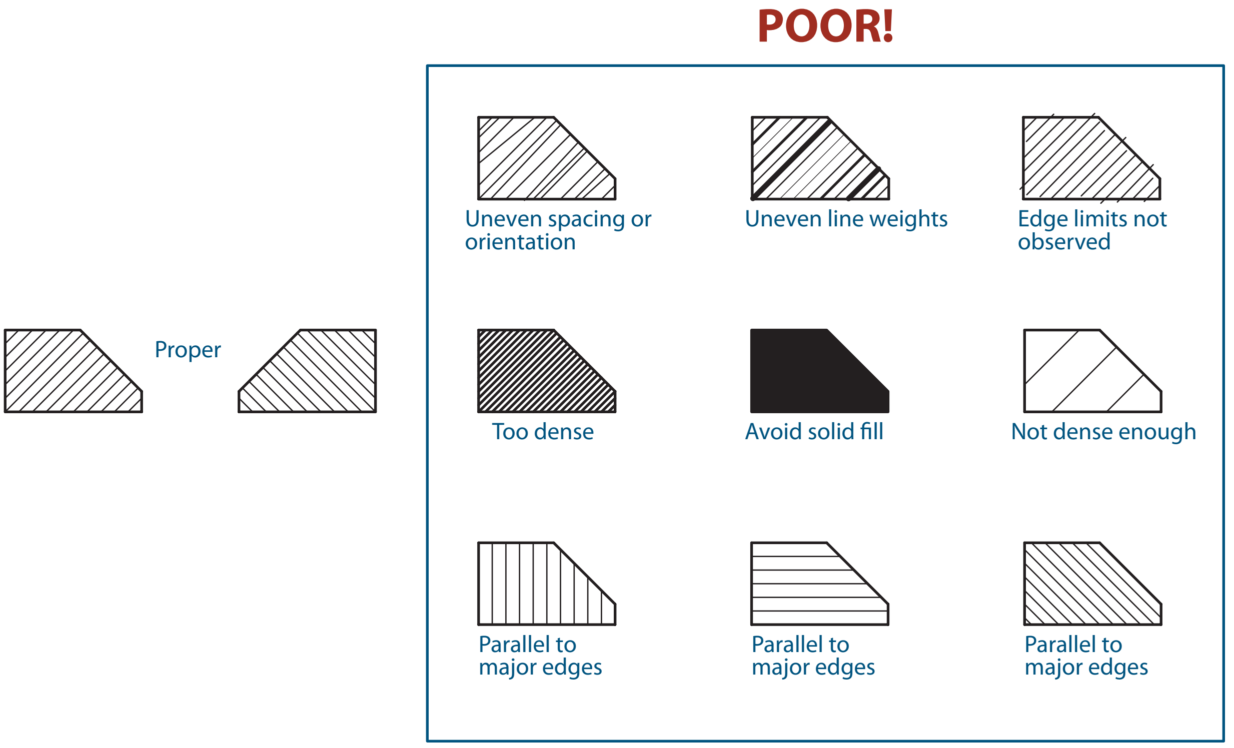

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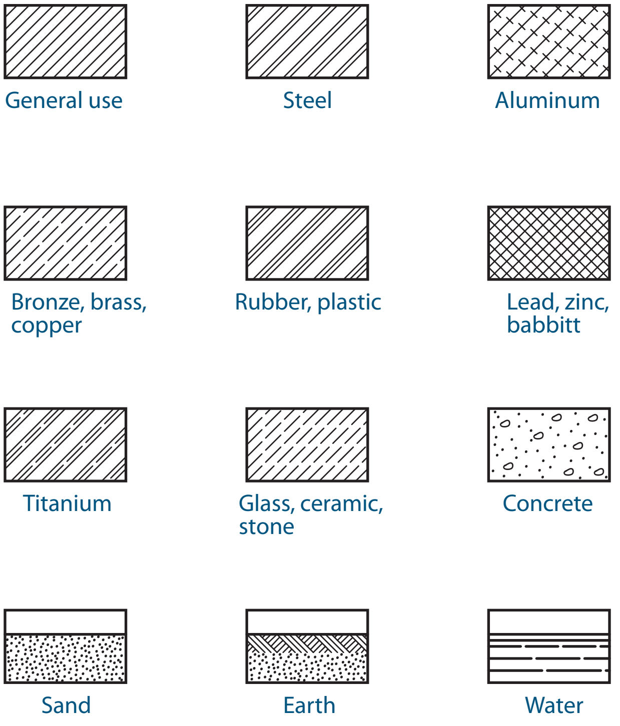
FIGURE 10.15. Examples of proper and poor cross-hatching techniques.

FIGURE 10.16. ANSI standard cross-hatch patterns for various materials.



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10.06 Offset Sections

Offset sections can be considered modifications of full sections. An offset section allows multiple features, which normally require multiple section views, to be captured on a single view. As with a full section, an external surface hypothetically cuts through an entire part. However, instead of the part being divided with a single flat cutting plane, the cutting surface is stepped. The size and location of each step are chosen to best capture the features to be displayed. Also, as with a full section, an offset section has its viewing direction indicated by arrows that point at the cut surface to be seen. When the offset cutting surface is rotated onto the viewing plane, the cross sections of multiple features, which could not be shown otherwise with a single cutting plane, can be displayed on a single view. This process is shown in Figure 10.17.

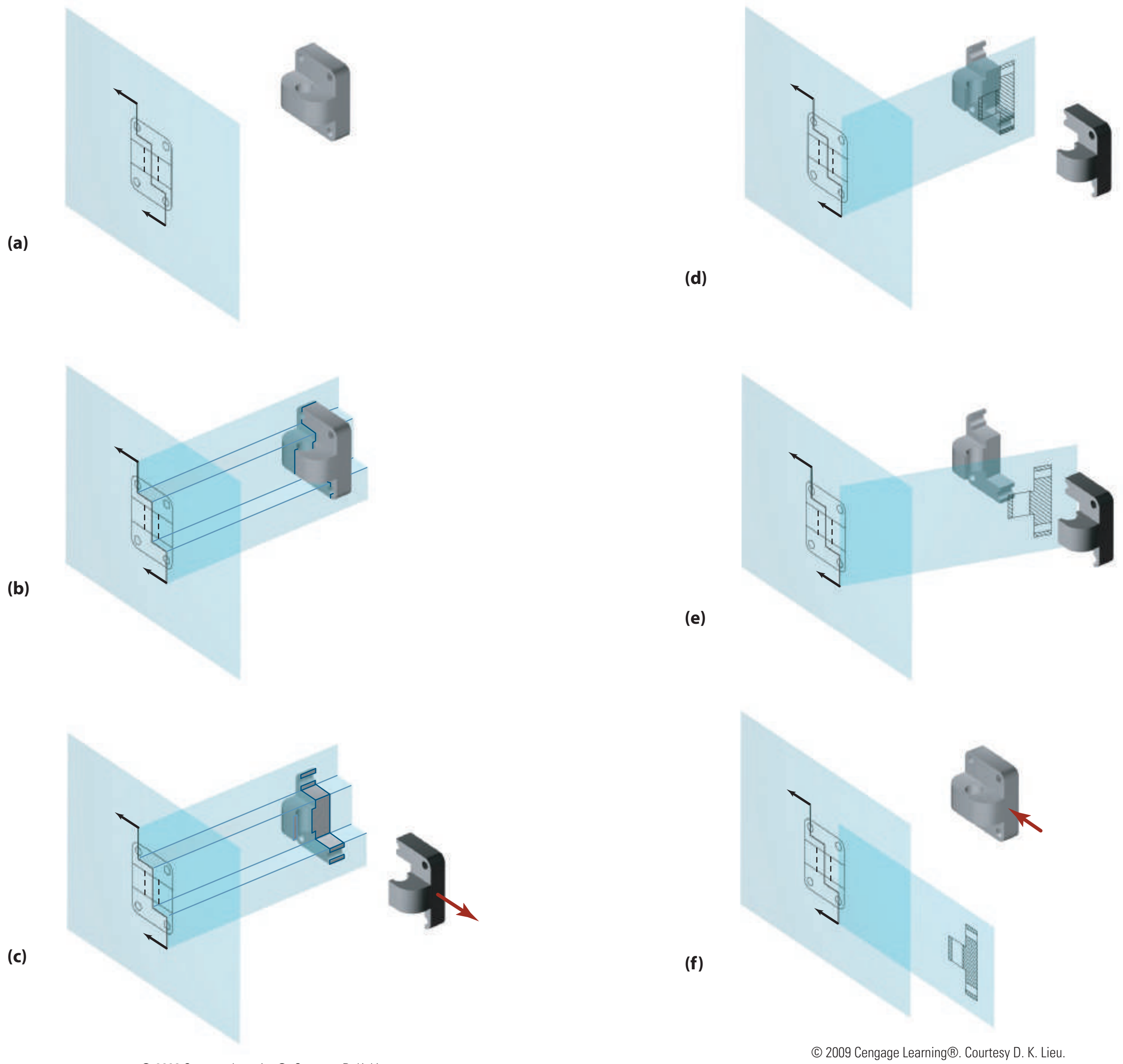


FIGURE 10.17. Creating an offset section. An object is projected onto a viewing plane in (a). A stepped cutting plane orthogonal to the viewing plane slices the object in (b). The piece to be viewed remains, while the other piece is removed in (c). The projection of the sliced part is made on the outermost segment of the stepped cutting plane in (d). The cutting plane and image are rotated about the section line in (e). The section view is coplanar with the viewing plane in (f).

On an engineering drawing such as the one shown in Figure 10.18, the edge view of the stepped cutting surface is represented by a heavy stepped line with alternating short-short-long dashes. This is still called a cutting plane line, although technically it is no longer a straight line. The arrows point in the direction of viewing, and the cutting plane line and its associated offset section view are uniquely identified in each drawing with a pair of capital letters, as before.

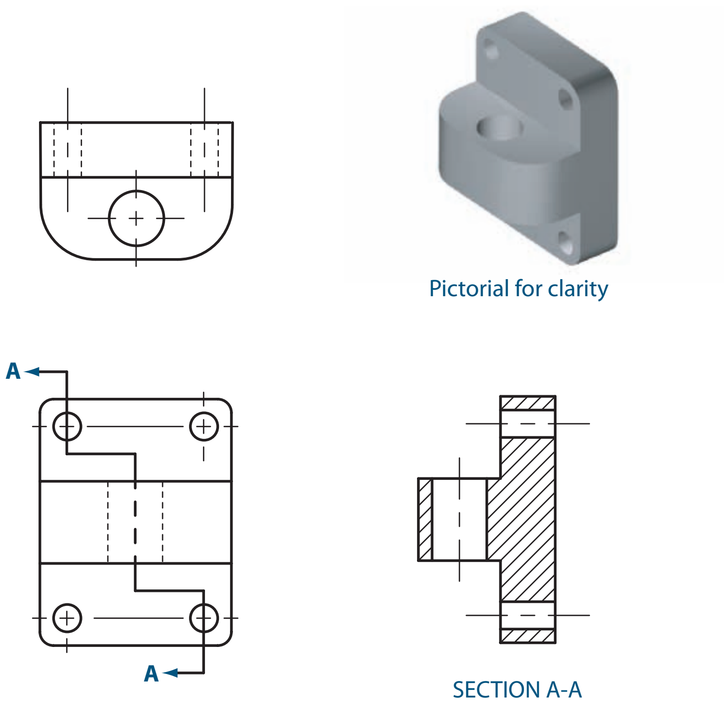
As with full sections, the rotation orientation of an offset section view must be consistent with the creation of an orthogonal view; but the location and scale of the view are left to the discretion of the person creating the drawing. Note that in an offset section view, it is customary not to show the locations of the steps on the view. The reason is because this information is already available by inspecting the cutting plane line and because adding step lines may cause confusion by showing edges that do not actually exist on the part.

10.07 Half Sections

Half sections are used to save space and labor on an engineering drawing, especially for symmetrical parts. Recall what you learned about symmetry in Chapter 3. When an object is symmetrical about a plane or an axis, it is acceptable to present the object partially in its original state and partially in a sectioned state on the same orthogonal view. The plane of symmetry separates the two states. Another way of visualizing a half section is to imagine a part that is cut such that one-quarter of it is removed to reveal the interior detail. This hypothetical process is shown in Figure 10.19.

In the engineering drawing for this half section, the cutting plane line extends across the object only to the plane of symmetry. The cutting plane line extends partway across the object. A single arrow on the cutting plane line points in the direction of viewing. The absence of a second arrow is an indication that the cutting plane line is for a half section. There is no separate section view. Instead, the orthogonal view and the section view are combined such that the exterior of the part is shown on one half and the interior of the part is shown on the other half. In Figure 10.20, the view types change at the plane of symmetry, which is shown as a centerline. Note that hidden lines are not shown on the unsectioned half of the part.

FIGURE 10.18. An engineering drawing using an offset section view to reveal multiple interior detail.



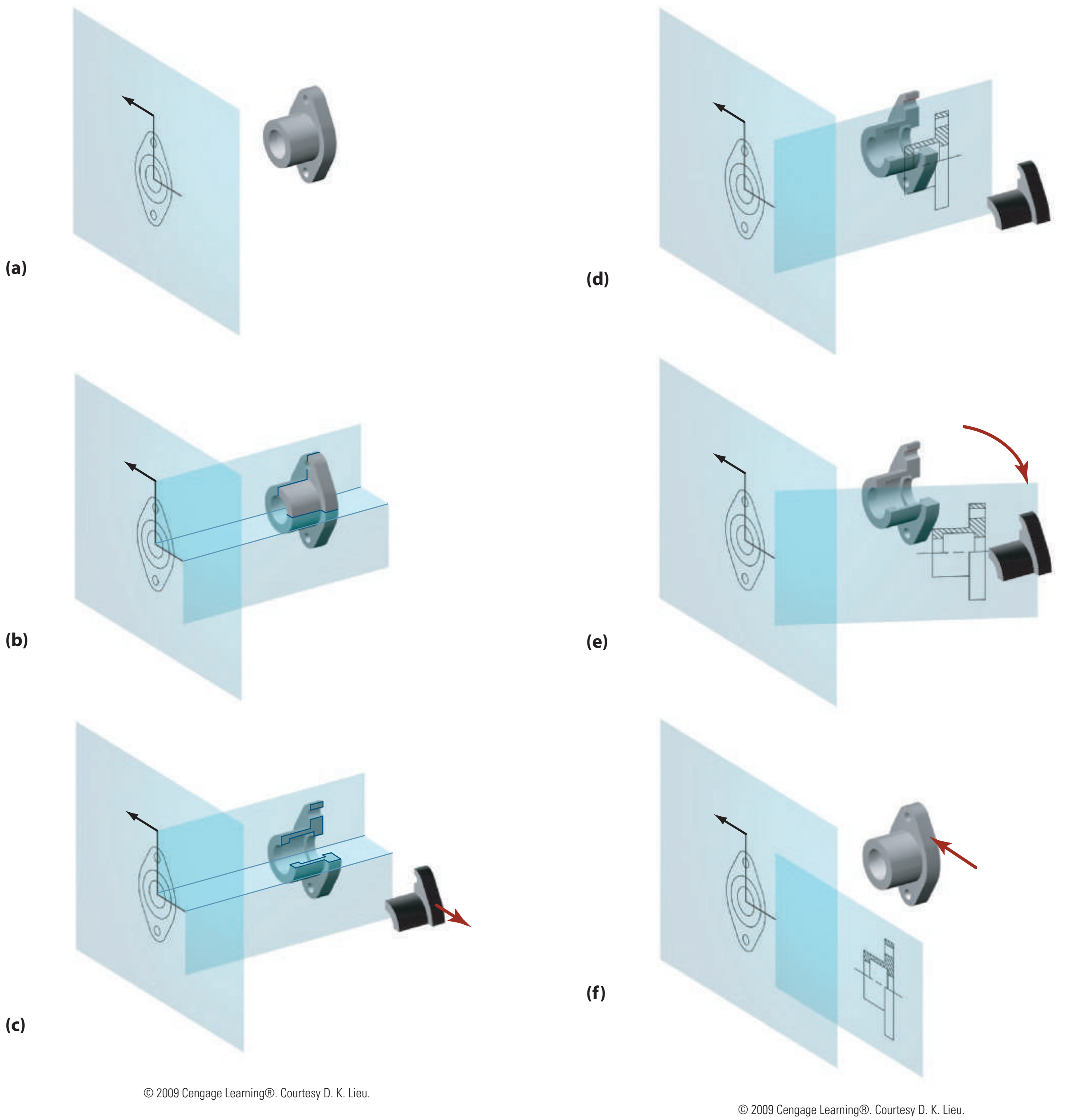
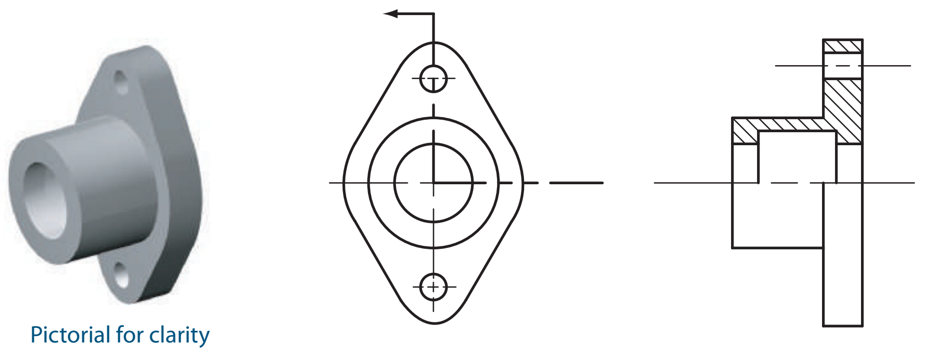


FIGURE 10.19. Creating a half section. An object is projected onto a viewing plane in (a). A stepped cutting plane slices through the object to the plane of symmetry in (b). The piece to be viewed remains, while the other piece is removed in (c). The image of the sliced object is projected onto an orthogonal viewing plane in (d). The viewing plane and image are rotated about the intersection line in (e). The section view is coplanar with the original viewing plane in (f).

10.08 Removed Sections

In certain cases, it is convenient to use a removed section instead of a full section. A **removed section** offers the convenience of showing only the new surfaces created by a cutting plane, without the complexity of showing the remaining surfaces on an object. A hypothetical procedure for creating a removed section image is shown in Figure 10.21.

FIGURE 10.20. An engineering drawing shows the use of a half section to reveal interior as well as exterior detail.



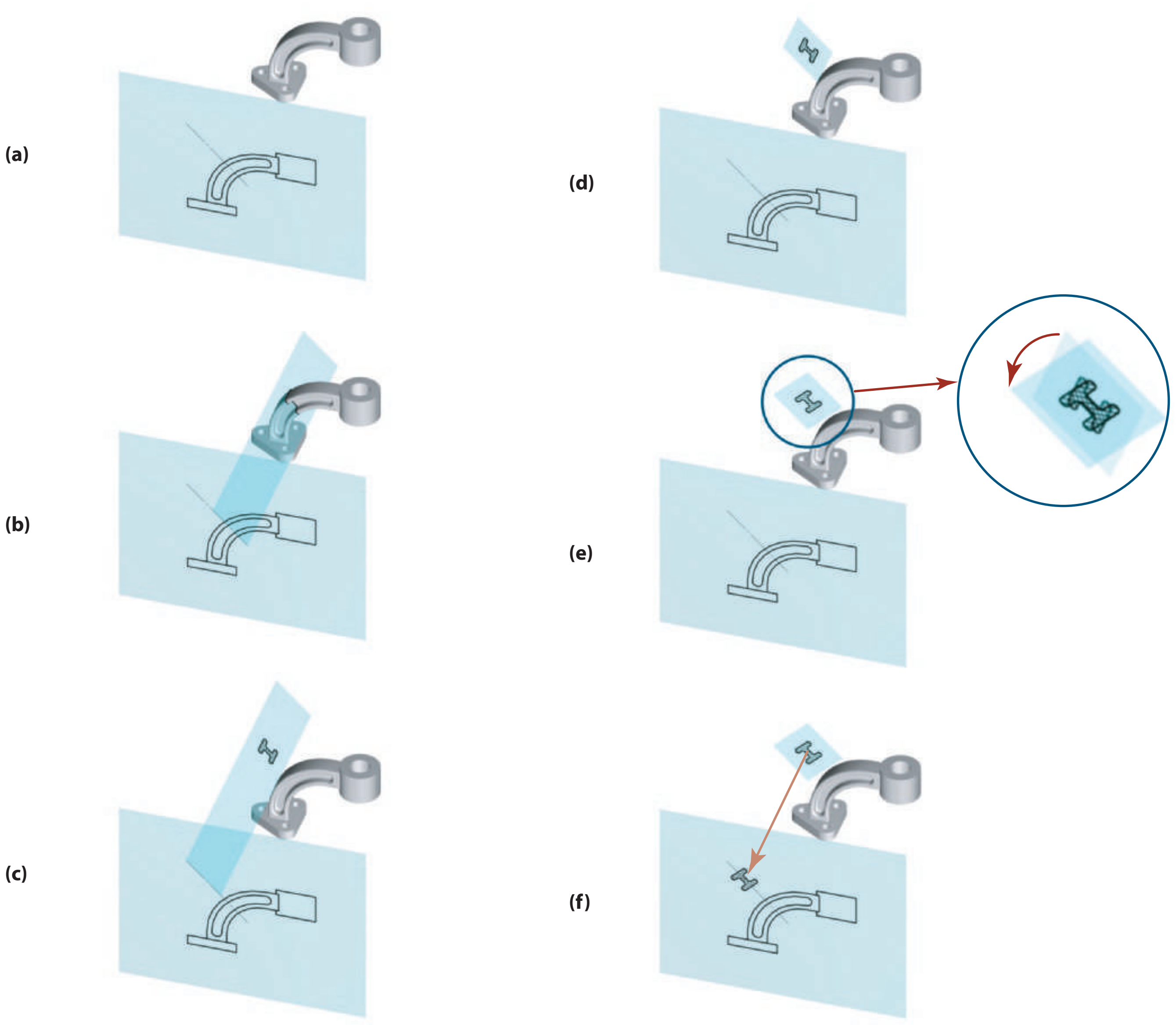


FIGURE 10.21. Creating a removed section. An object is projected into a viewing plane in (a). The cutting plane slices the object in (b). The image of the intersection is removed from the object in (c). The removed image is initially perpendicular to the viewing plane in (d) but is then rotated to be parallel to the viewing plane in (e). The removed image is finally projected onto the viewing plane in (f).

In this figure, a cutting plane intersects the object in the area of interest in (a). The cutting plane should be parallel to one of the principal views or perpendicular to the major surfaces of the part where the cut is made in order to reveal its true sizes. In (b), the cutting plane is removed from the part with the image of the intersection imprinted on the plane. The cutting plane and image are then rotated by 90 degrees in (c) such that the arrows point into the page on a drawing (d). The complete removed section view, as would be seen in an engineering drawing, is shown in Figure 10.22.

There is no need for view alignment for the removed section, although its orientation must still follow the rules of multiview presentation and the section view should be located near the cutting plane line. If the scale of the section view is different from that used on the multiview projections, the new scale must be included with the labeling of the section view.

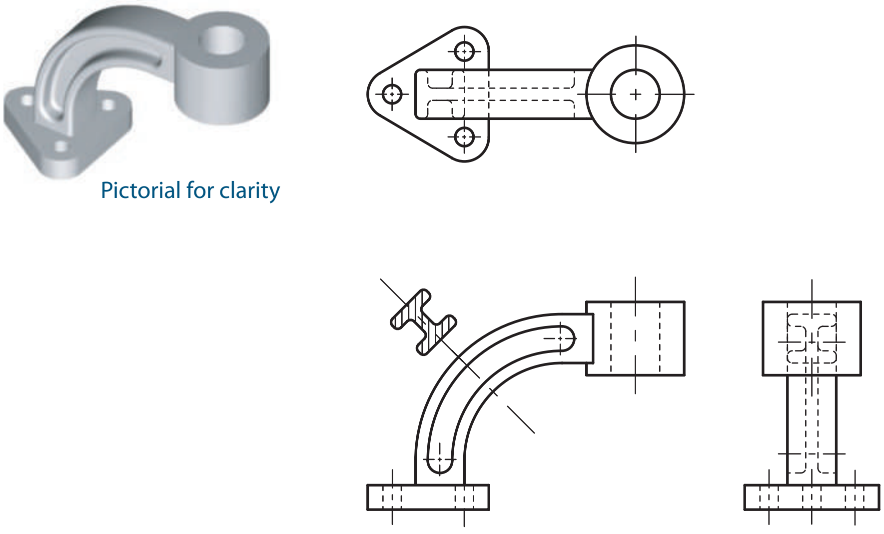
For views where the surfaces created by a hypothetical cut are relatively small compared to the remaining surfaces of the view or where full sections may create unnecessarily large or confusing views, removed sections are a good option for improving clarity while reducing effort and complexity in a drawing. Figure 10.23 shows how removed sections were used for defining various parts of the Hoyt AeroTec bow handle. In this case, full sections or offset sections would have created unnecessary complexity in the drawing.

10.09 Revolved Sections

Revolved sections are created in a manner similar to that of removed sections. A hypothetical procedure for creating a revolved section image is shown in Figure 10.24.

In this figure, a cutting plane intersects the object in the area of interest and the image of the intersection is imprinted on the plane. The cutting plane should be parallel to one of the principal views or perpendicular to the major surfaces of the part where the cut is made in order to reveal its true sizes. The cutting plane is rotated by 90 degrees such that the arrows point into the page on a drawing. The axis of rotation of the intersection image is on the cutting plane, parallel to the cutting plane line, and through the geometric center of the image. Unlike the removed section, the cutting plane is not removed from the object. Thus, the image of the intersection is

FIGURE 10.22. A removed section as it would be placed on an engineering drawing.



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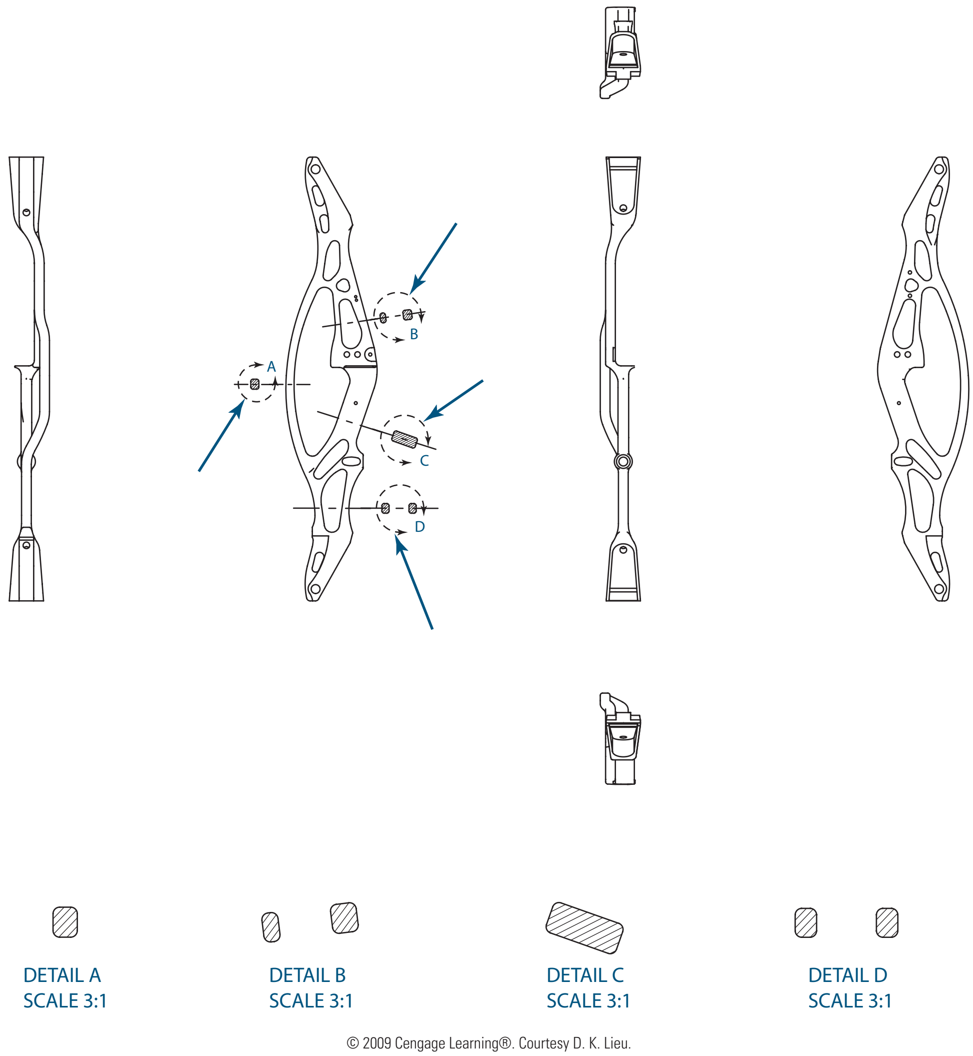


FIGURE 10.23. Use of removed sections on the Hoyt AeroTec bow example.

superimposed on the orthogonal view. The complete revolved section view, as would be seen in an engineering drawing, is shown in Figure 10.25. Since a revolved section view is constructed at the location of the cutting plane line, there is no need to label the view.

The scale of the revolved section must be the same as for the principal views, and its orientation must follow the rules of multiview presentation. For views where the surfaces created by a hypothetical cut are relatively small compared to the remaining surfaces of the view, revolved sections are another good option for improving clarity while reducing effort and complexity in a drawing. Revolved sections should not be used when the section image interferes significantly with other features in the principal view. Figure 10.26 shows another example where the use of revolved sections is convenient.

10.10 Broken-Out Sections

A **broken-out section** can be used when the internal feature to be revealed is a small portion of the entire object and a full section would not reveal additional details of interest. Use of a broken-out section in this manner would decrease the size and complexity of a drawing, as well as reduce the effort required to make it.

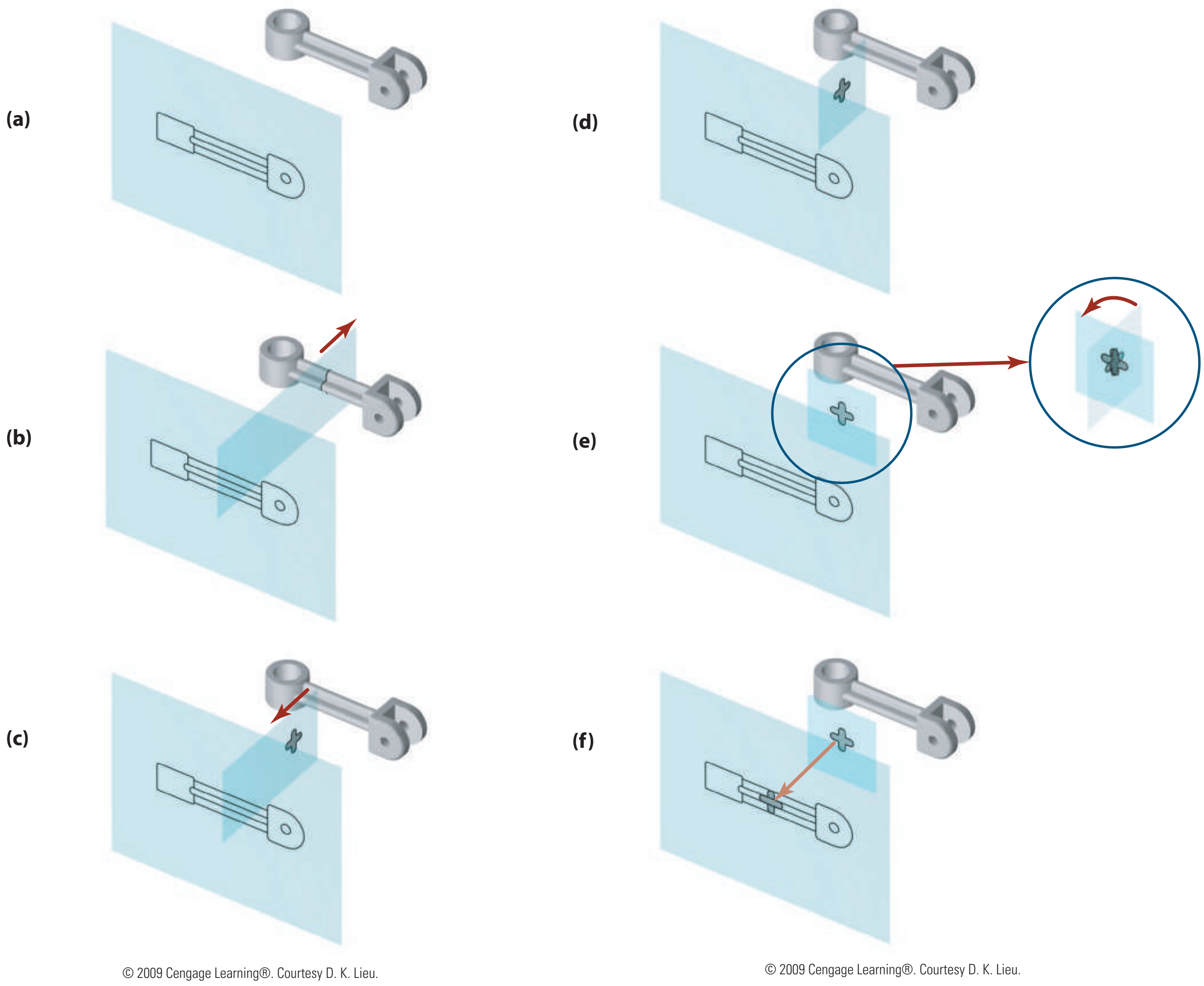
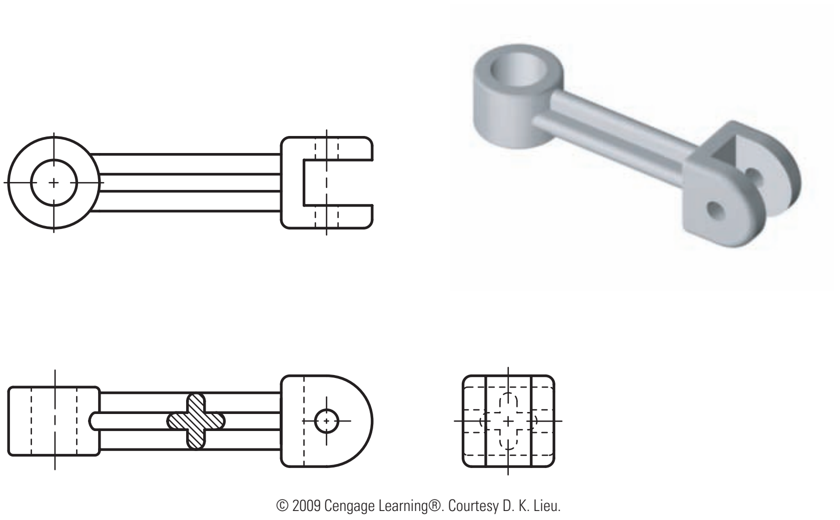


FIGURE 10.24. Creating a revolved section. An object is projected onto a viewing plane in (a). The cutting plane slices the object in (b). An image of the intersection is removed in (c). The intersection image is initially perpendicular to the viewing plane in (d) and then rotated to be parallel to the viewing plane in (e). The image is projected onto the viewing plane in (f).

FIGURE 10.25. A revolved section as it would be placed on an engineering drawing.



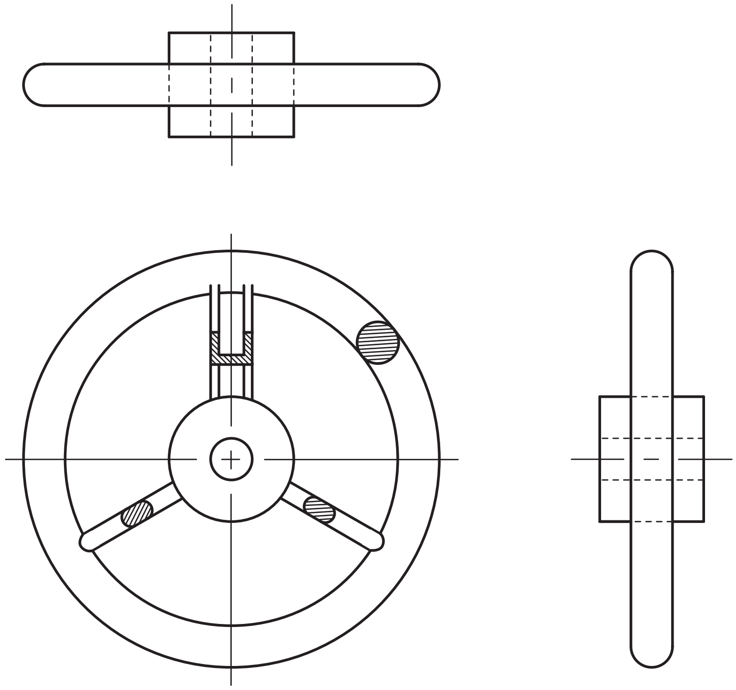


FIGURE 10.26. An example of a part with multiple revolved sections.

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A broken-out section, as with the revolved and removed sections, offers the convenience of slicing only a fraction of the entire object when only a small slice is needed to define an internal geometry. However, a broken-out section offers the added convenience of not requiring the cutting plane to go all the way through the part. With all of the other sections you have studied so far, cutting plane lines start in space, go through the part, and end in space. With a broken-out section, the ends of the cutting plane line can be wholly or partially embedded in the part, as shown in Figure 10.27, where a cutting plane that has its extent limited to the area immediately surrounding a feature is imbedded into the part. A piece of the object that is opposite the viewing direction is then hypothetically broken off to reveal the interior details of the feature.

The portion of the cutting plane that is embedded in the part is shown on the section view as an irregular edge to emphasize that the part would hypothetically be broken to reveal the interior details shown at that location. The broken-out-section view may be shown on the corresponding orthogonal view, as shown in Figure 10.28, or in a separate detail view, as shown in another example in Figure 10.29.

10.11 Sections of Assemblies

Section views are commonly used in drawings that show multiple parts in their intended mating configuration to illustrate proper alignment of different features between the parts. When multiple parts are sectioned, as in Figure 10.30, it is advisable to use a different section line pattern for each part in order to distinguish the different parts easily. In everyday practice, assemblies that include pins, keys, shafts, or bolts usually do not show these items sectioned even though the cutting plane line may pass through them. These items usually have standardized geometries and sizes; thus, their sections add little information to a drawing and may even detract from the information presented by parts of greater interest.

FIGURE 10.27. Creating a broken-out section. The object is projected onto a viewing plane on (a). A cutting plane slices through the feature of interest (but not the entire part) in (b). The portion in front of the cutting is broken out and removed in (c). The interior details of the feature are shown in (d). The image of these features is projected forward in (e) and placed directly on the part image in (f).

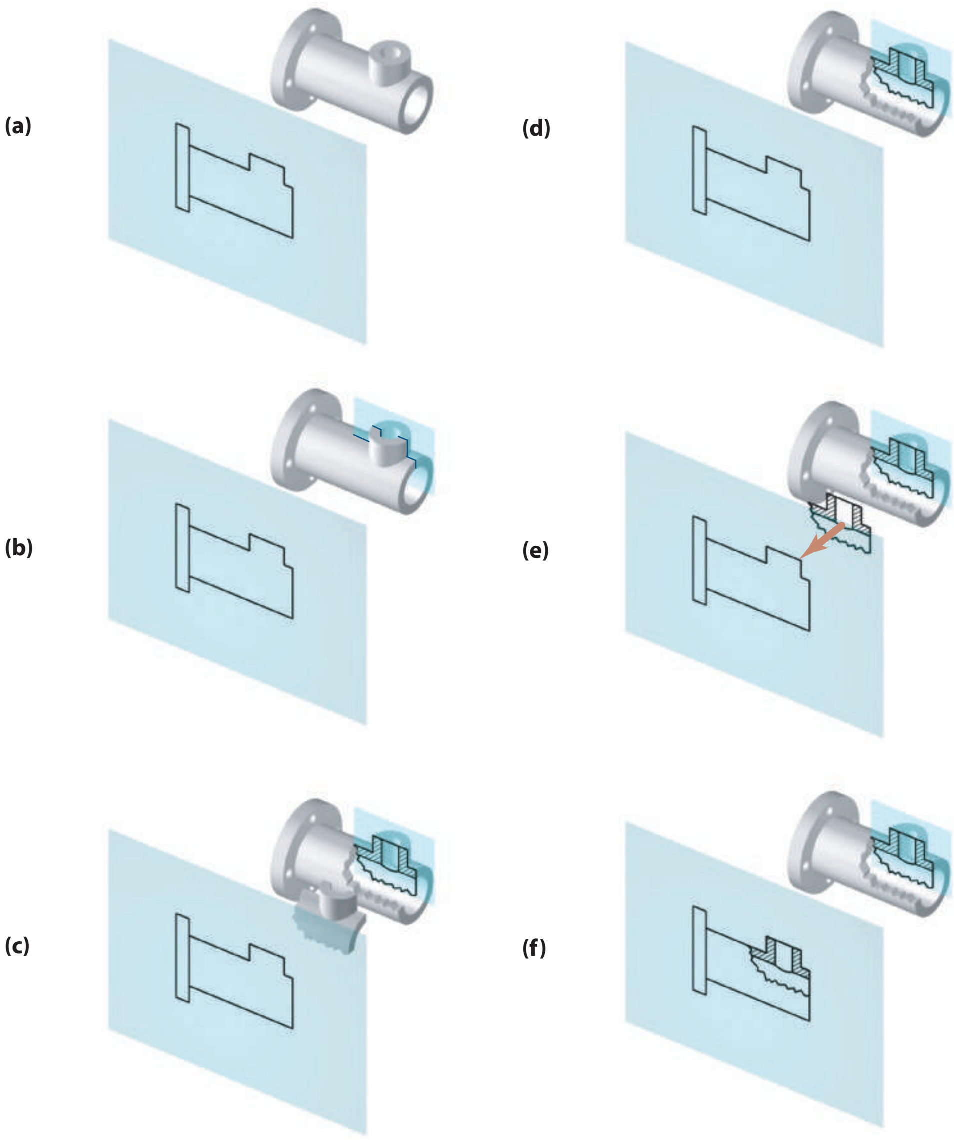

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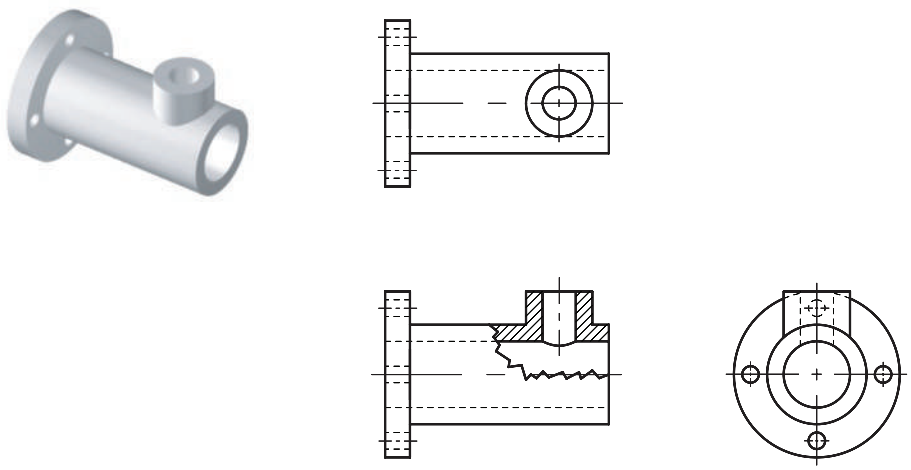
FIGURE 10.28. A broken-out section as it would be placed on an engineering drawing.



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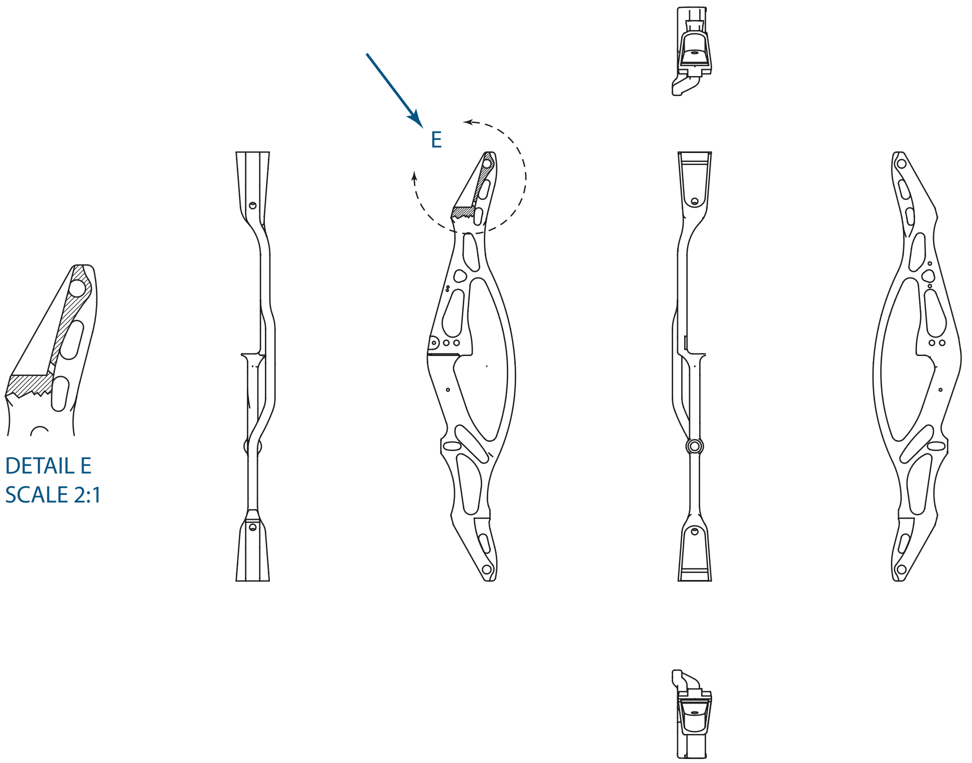

FIGURE 10.29. A broken-out section used to reveal some pocket details of the Hoyt AeroTec bow.

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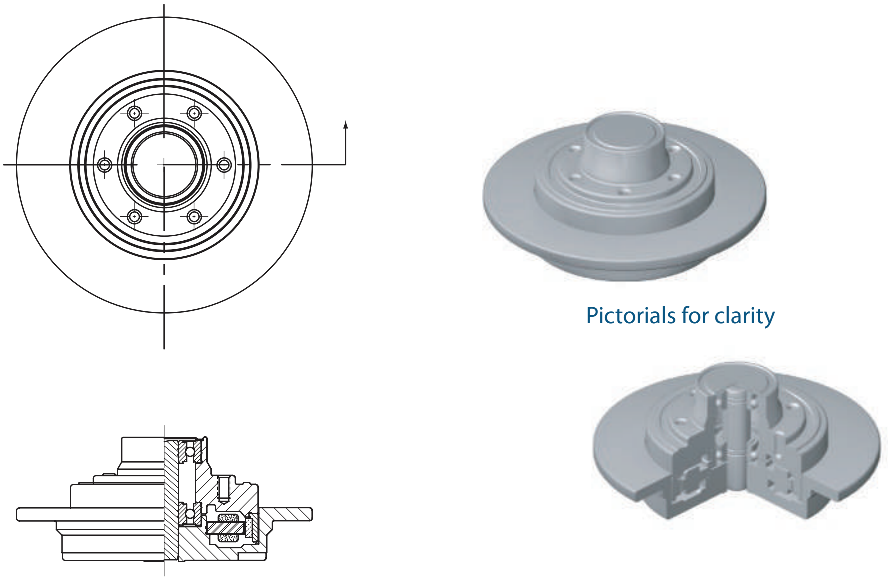

FIGURE 10.30. The method for showing the assembly of many parts.

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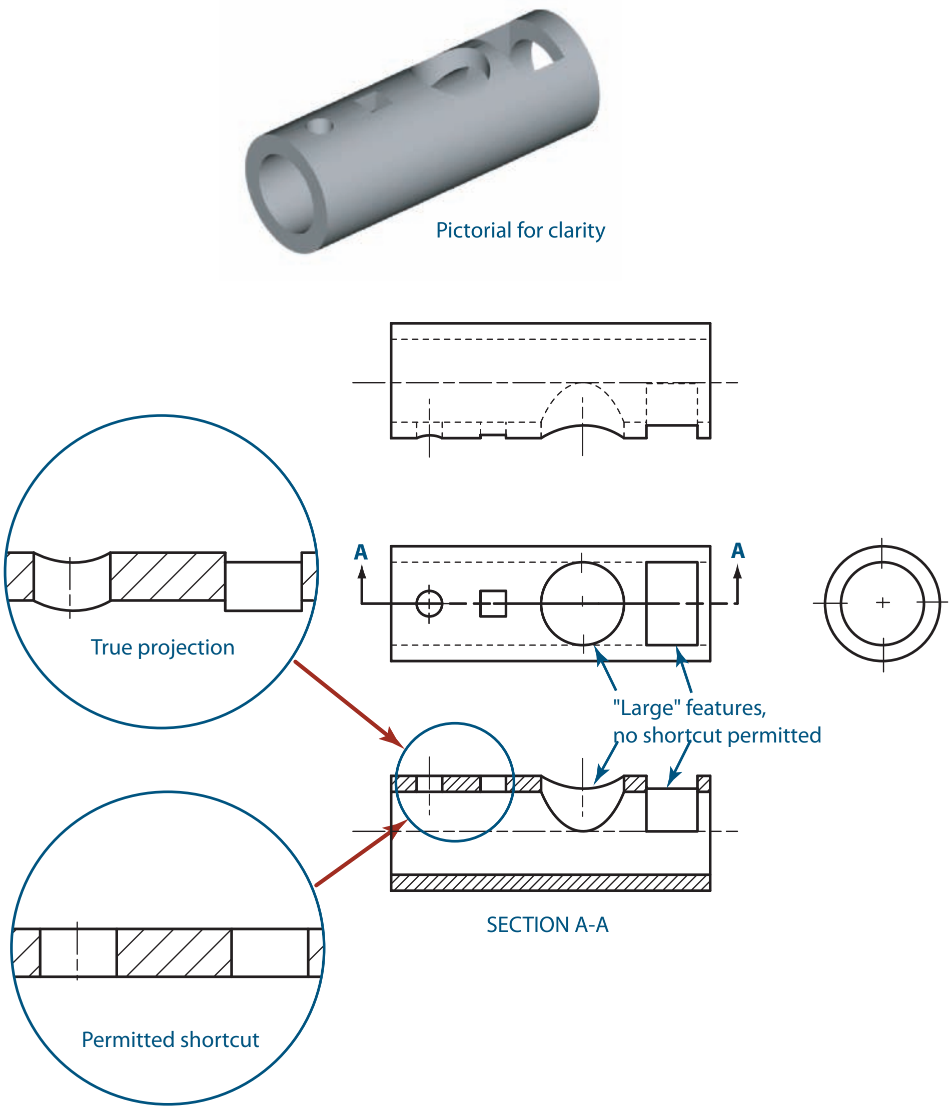
10.12 A Few Shortcuts to Simplify Your Life

As with many other engineering drawing practices, acceptable shortcuts for creating section views can be used to reduce the time it takes to create a drawing and/or to minimize possible misinterpretation of a drawing. With all of the shortcuts presented next, the main question you need to ask yourself before using any of them is, "Will this approximation or shortcut increase or decrease the speed and accuracy of interpretation of the drawing?" If the speed or accuracy of interpretation decreases, the shortcuts should not be used.

10.12.01 Small Cutouts on Curved Surfaces

A shortcut is allowed when there is a small hole or another cutout on a curved surface. Figure 10.31, for example, shows a small hole and slot on a tube compared to larger cutouts. If a true projection of these features were made, the orthogonal views would show a curved depression on the surface of the tube. The shape of this curve is complex and would take some time to create. Since in most applications the size of the depression on the surface is unimportant, the depression is not shown on the orthogonal views. The true projection of these features and the accepted shortcut are shown in Figure 10.31. This approximation makes the drawing easier to create, with very little loss of information. However, when the cutouts are large or the size of the depression cannot be ignored in the function of the part, the true projection should be used. What is considered "small" is rather subjective.

FIGURE 10.31. A permitted shortcut for small holes and slots in curved surfaces.



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10.12.02 Threaded Parts

Another shortcut is in the representation of a threaded part, such as the pneumatic fitting shown in Figure 10.32. A thread on the outside of a bolt or screw or the inside of a nut has many complex curved surfaces that would result in a very complicated drawing, especially if it were created with manual instruments or 2-D CAD. Much simpler representations of internal and external threads are included in Figure 10.32. These schematic representations are easier to construct, with very little loss of information, especially since thread sizes are mostly standardized based on the diameter of the part. A note and arrow are required to specify the precise thread sizes. Methods for the complete specification of thread sizes can be found in most machinists' or engineers' handbooks.

10.12.03 Thin Features

For sectioned features that have relatively small thickness when compared to the remainder of a sectioned part, it is acceptable not to fill these features with section lines even when cutting plane lines pass directly through them. As an example, consider the objects in Figure 10.33. These two objects are composed of the same main body but with mounting flanges turned differently. In both cases, the cutting plane line goes through both the main body and the flanges. For the part in (a), the thickness of the two flanges in the section view is about the same as the depth of the main body. In this case, the flanges are filled with section lines, as normal. For the part in (b), the thickness of the two flanges in the section view is a fraction of the depth of the main body. In this latter case, it is acceptable not to fill the flanges with section lines because doing so may give an immediate false impression that the flanges are about the same thickness as the main body. As an alternative to not filling thin features with section lines, it is permissible to use a different section line pattern for spokes and vanes than is used for the main body of the object. Note that for this shortcut, an extra edge must exist to separate the thin feature from the main body in the section view. Webs and fins, such as those shown in Figure 10.34, are generally treated in this manner.

10.12.04 Vanes, Fins, Spokes, and the Like

Objects with axially symmetric features such as vanes and spokes, as shown in the two parts in Figure 10.35, also are not filled with section lines even when cutting plane lines pass directly through them. Filling such features with section lines may give the false impression that the features are solid throughout the part. It also is permissible to

FIGURE 10.32. A section of a threaded part.

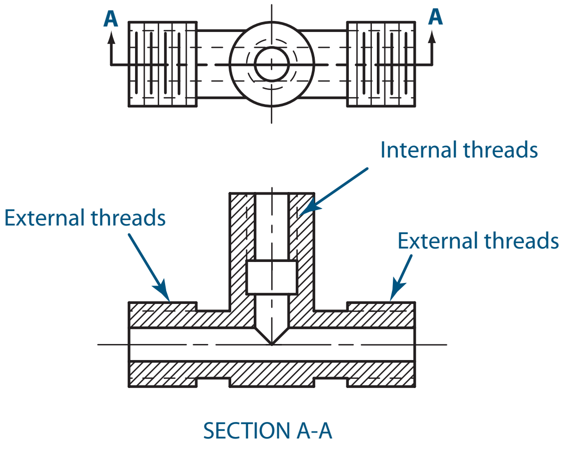

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FIGURE 10.33. The conventional section (a) and recommended variation for a thin feature (b).

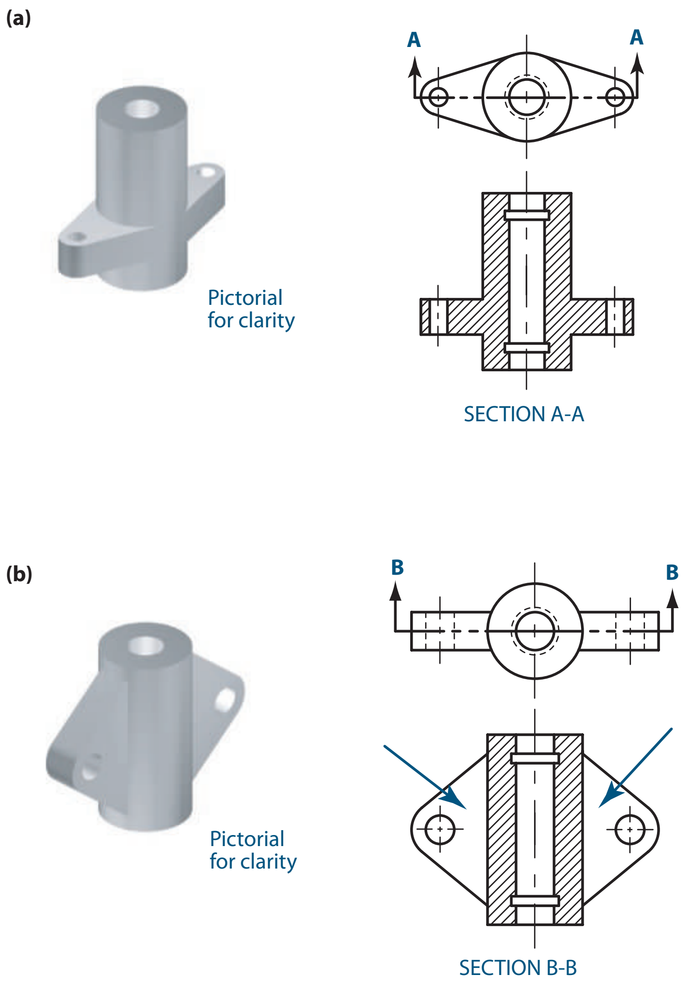

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FIGURE 10.34. The recommended presentation of thin webs.

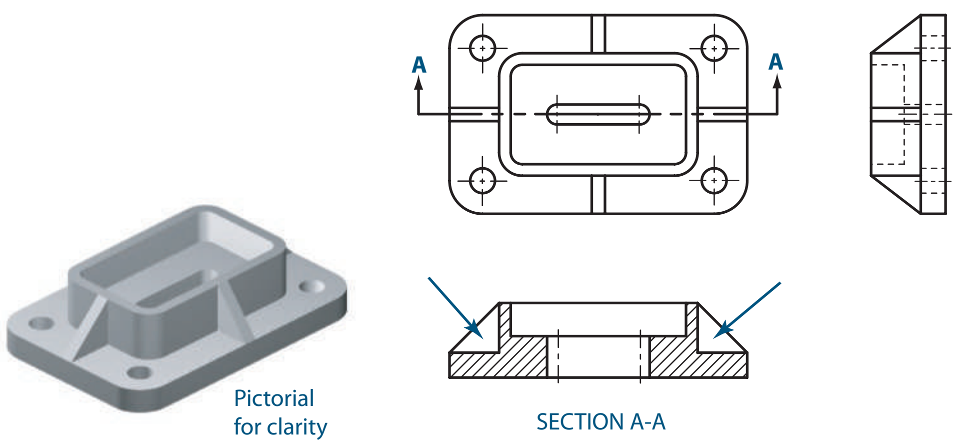

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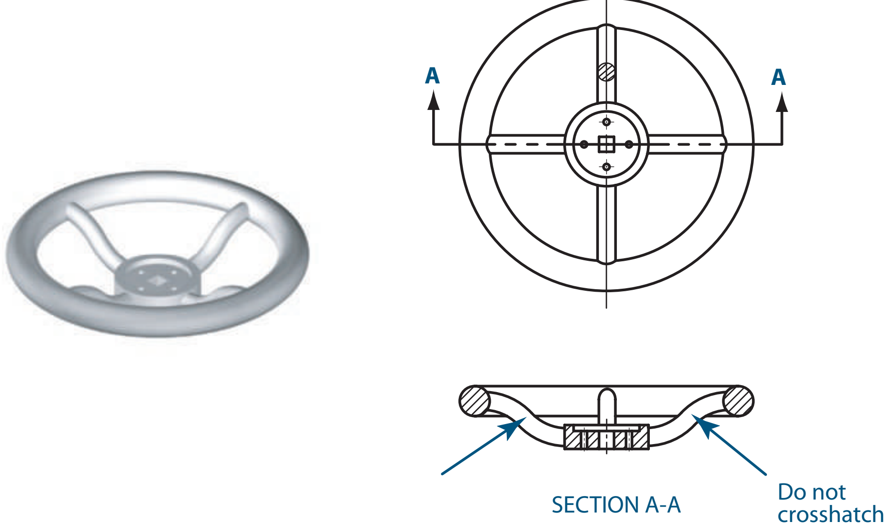
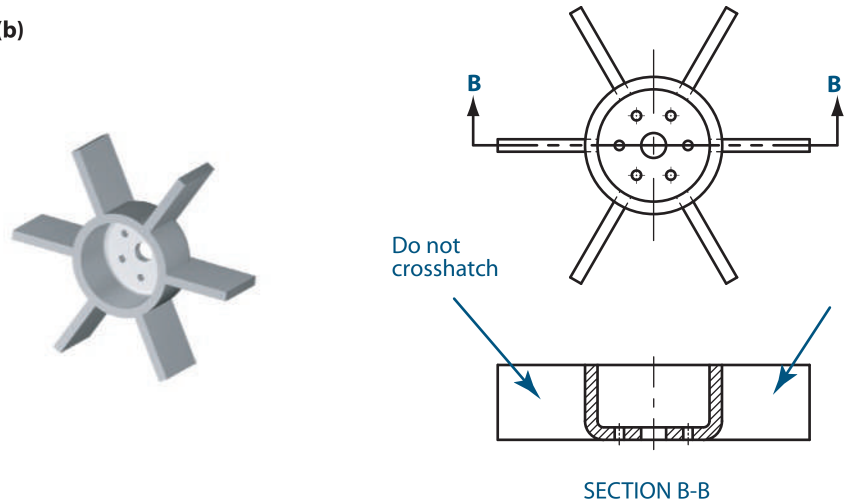
FIGURE 10.35. Two examples of the recommended presentation of spoke, vanes, and fins.

(a)



(b)



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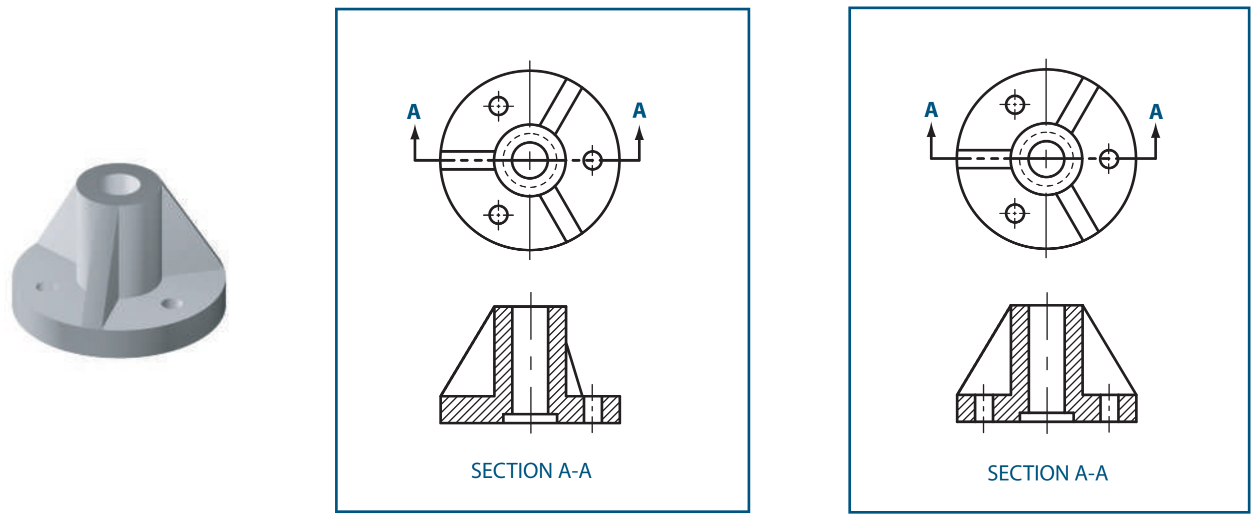
use a different section line pattern for spokes and vanes than is used for the main body of the object. Note that for this shortcut, an extra edge must exist to separate spokes and vanes from the main body in the section view.

10.12.05 Symmetry

An interesting exception to the rules of true projection occurs when parts with rotational symmetry, which means that the part can be divided into identical wedges along an axis, are sectioned. Note that rotational symmetry is different from the planar symmetry discussed in Chapter 3, where the image of an entire object can be created by reflecting a portion of it on a plane. For example, examine the part shown in Figure 10.36. This part has one-third rotational symmetry, with three thin support ribs and three holes about the center tube.



True projection

Preferred presentation

FIGURE 10.36. The preferred presentation of symmetrical features.

A multiview drawing created using true projection would be like that shown in Figure 10.36. Using a true projection for the front view in this case has some problems. First, using instruments or 2-D CAD, the projection is rather difficult to create. Also, the true projection of the side view may have the negative effect of representing the part as being nonsymmetrical.

An acceptable shortcut for this drawing also is included in Figure 10.36. This drawing is easier to create and gives the impression that the part is symmetrical. The top view clarifies any possible misinterpretation about the number and locations of the support ribs. Interestingly, if the part had one-quarter (or higher) symmetry, for example, four (or more) support ribs instead of three, the front view would be exactly the same as the part with one-third symmetry.

10.13 Considerations for 3-D Modeling

With solids modeling software, parts are initially modeled as a series of protrusions and cuts to create a 3-D graphical model of a part. The solids modeling software creates a mathematical model of the geometry from which the projections of the object are used to create drawings. Once the solids model is created, it is usually a simple matter to extract a front view, side view, or any of the other orthogonal views from the model. A section view is created merely as another orthogonal view, but with a portion of the object removed. The ease with which section views can be created from a solids model has many advantages, but also some disadvantages. The greatest advantage is the speed and accuracy with which section views can be created. With most software, creating additional views is simply a matter of specifying the cutting plane and viewing direction and then picking a location on the drawing where the new view is to appear. Cutting planes can be specified as existing or newly created reference planes. Creating stepped cutting plane lines in the views of interest usually specifies offset cutting planes. The process is often a matter of a few strokes on a keyboard or a few clicks with a mouse or another pointing device. The time required is usually only a few seconds. Also, accurate orthogonal projection of features that were previously represented by shortcut practices, such as small cutouts in curved surfaces or thin symmetrical features, are very easy to create. In fact, with most software, it would be difficult to create a view that is *not* an accurate projection. Using section lines to fill areas that were formerly solid is also a rather simple matter. The software identifies the newly cut surfaces and automatically fills them. All the software user needs to do is specify the section line pattern to be used and modify it if necessary.

The selection of where to section an object to view its interior or the type of section to use is still up to the person making the drawing. One disadvantage of the nearly automatic section creation offered by 3-D modeling is that in some cases, the modeling becomes too accurate. Many of the shortcuts and clarification practices used in traditional drafting are no longer available in some software. For example, a section through a spoke or vane used in a 3-D model would show the spoke or vane filled with section lines, not blank as would be preferred. Also, all projections would be true projections. With an object of odd rotational symmetry, there would be no opportunity to modify the projection to create a symmetrical presentation, as would be preferred. With some software, the step edges of an offset section may be visible in the section view, and not removed as is practiced.

Another disadvantage of 3-D modeling is that manual creation of section views has been a traditional method of developing spatial reasoning and mental imaging skills. When the process is too automatic with software, a person may not adequately develop these skills in the absence of the software and may become too dependent on the software. When faced with multiple section views in a shop drawing, that person may not be able to create a mental image of the part or may not develop the skills necessary to interpret the drawings. Eventually, the person will develop these skills, but it may require exposure to many solids models and their drawings.

10.14 Chapter Summary

With many complex objects, looking only at the exterior may not fully reveal all of their features. The use of section views is a method of looking at the internal details of such objects. The section process involves using a hypothetical cutting plane to hypothetically cut an object into pieces so the interior details one or more features. These features can then be examined more closely, specified in such a manner that the details can be fabricated and inspected to ensure that they meet the desired specifications. On an engineering drawing, the cutting plane appears in an edge view called a cutting plane line. Several types of section views are available for use at the discretion of the drafter, depending on the desired presentation. Whichever type is used, certain rules and practices must be followed to ensure that these views can be interpreted easily and quickly without ambiguity. Of primary importance is that the rules of orthogonal projection and multiview presentation be used.

10.15 GLOSSARY OF KEY TERMS

broken-out section: The section view produced when the cutting plane is partially imbedded into the object, requiring an irregular portion of the object to be removed before the hypothetically cut surface can be seen.

cutting plane: A theoretical plane used to hypothetically cut and remove a portion of an object to reveal its interior details.

cutting plane line: On an orthographic view of an object, the presentation of the edge view of a cutting plane used to hypothetically cut and remove a portion of that object for viewing.

full section: The section view produced when a single cutting plane is used to hypothetically cut an object completely into two pieces.

half section: The section view produced when a single cutting plane is used to hypothetically cut an object up to a plane or axis of symmetry, leaving that portion beyond the plane or axis intact.

offset section: The section view produced by a stepped cutting plane that is used to hypothetically cut an object completely into two pieces. Different portions of the plane are used to reveal the interior details of different features of interest.

removed section: The section view produced when a cutting plane is used to hypothetically remove an infinitesimally thin slice of an object for viewing.

revolved section: The section view produced when a cutting plane is used to hypothetically create an infinitesimally thin slice, which is rotated 90 degrees for viewing, on an object.

section lines: Shading used to indicate newly formed or cut surfaces that result when an object is hypothetically cut.

section view: A general term for any view that presents an object that has been hypothetically cut to reveal the interior details of its features, with the cut surfaces perpendicular to the viewing direction and filled with section lines for improved presentation.

viewing direction: The direction indicated by arrows on the cutting plane line from the eye to the object of interest that corresponds to the tail and point of the arrow, respectively.

10.16 QUESTIONS FOR REVIEW

1. When should a section view be used?
2. What does a cutting plane line represent?
3. What does the area filled with section lines on a section view represent?
4. What are some guidelines concerning good drafting practice in creating section line patterns?
5. What is the significance of the direction of the arrow on a cutting plane line?
6. Why is it important that the rotational orientation of a section view, even if it is moved, be maintained as if it were an orthogonal view?
7. When should an offset section be used instead of a full section?
8. When should revolved or removed sections be used instead of full or offset sections?
9. Under what conditions should certain areas on a section view not be filled with section lines even though the cut is through solid material?

10.17 PROBLEMS

1. In the problem shown in Figure P10.1, the views indicated by the balloons are to be changed to full section views taken along the centerline in the direction indicated by the arrows in the remaining view. For each set of views, select the correct section view from the 24 proposed views shown at the right. A section view choice may be used more than once. A correct answer may not be available as a choice.

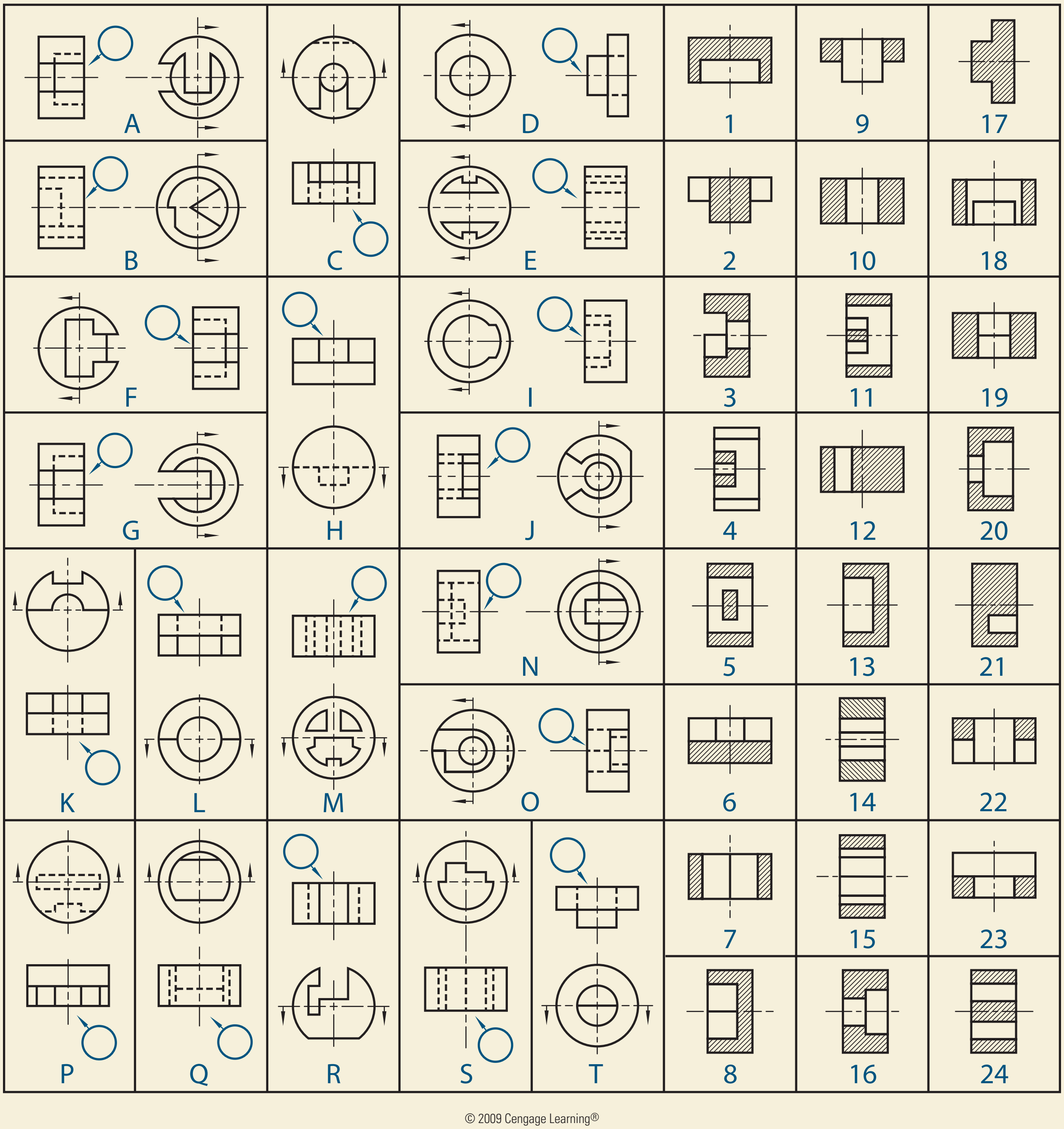


FIGURE P10.1.

10.17 PROBLEMS (CONTINUED)

2. In the problem shown in Figure P10.2, the views indicated by the balloons are to be changed to half section views as indicated by the letters A-A taken along the centerline in the direction in the remaining view. For each set of views, select the correct section view from the 24 proposed views shown at the right. A section view choice may be used more than once. A correct answer may not be available as a choice.

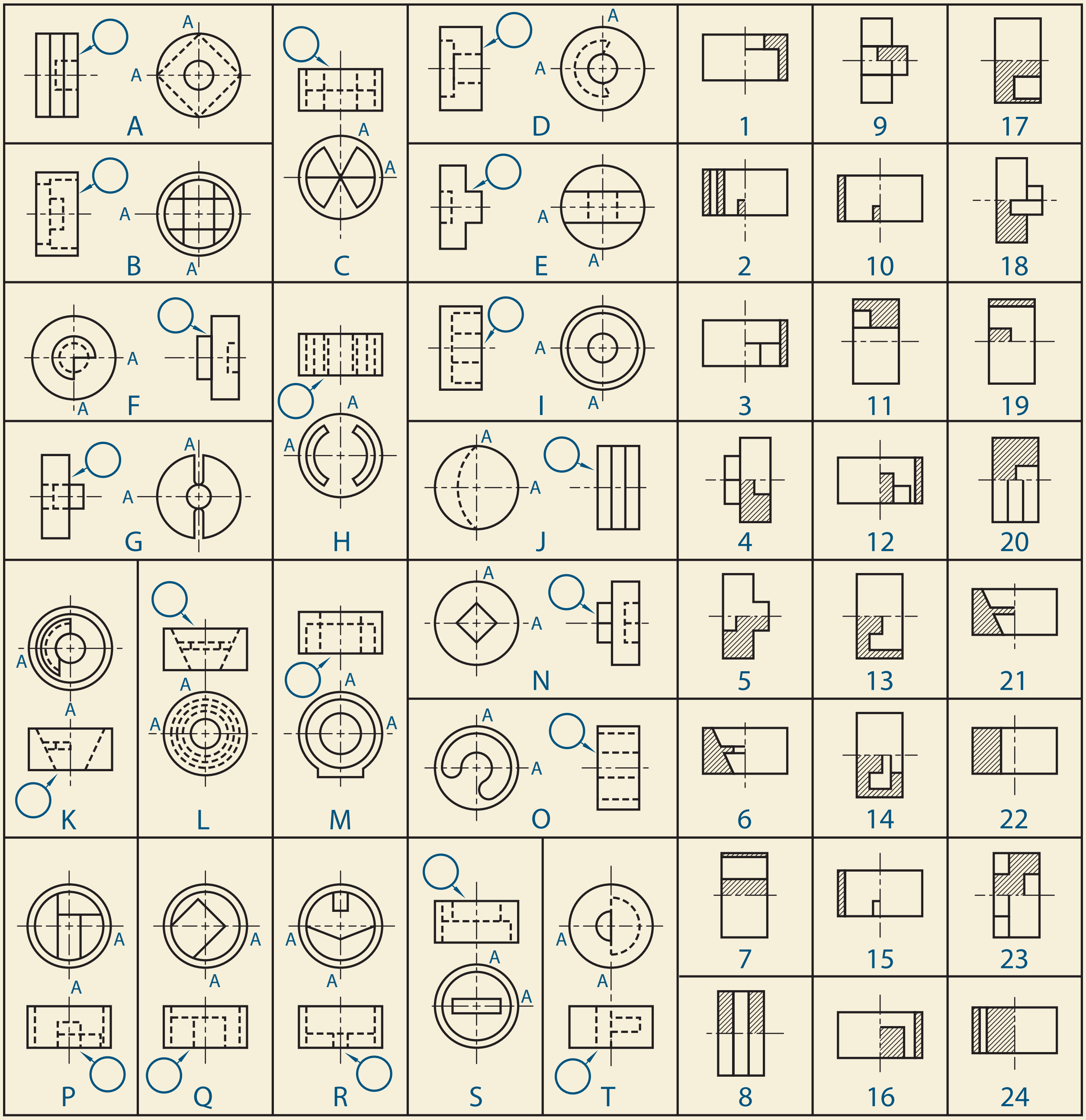


FIGURE P10.2.

10.17 PROBLEMS (CONTINUED)

3. In the problem shown in Figure P10.3, the views indicated by the circles are to be the location of section views. Select the correct section view to complete each problem from the 30 proposed views shown. A section view choice may be used more than once. A correct answer may not be available as a choice.

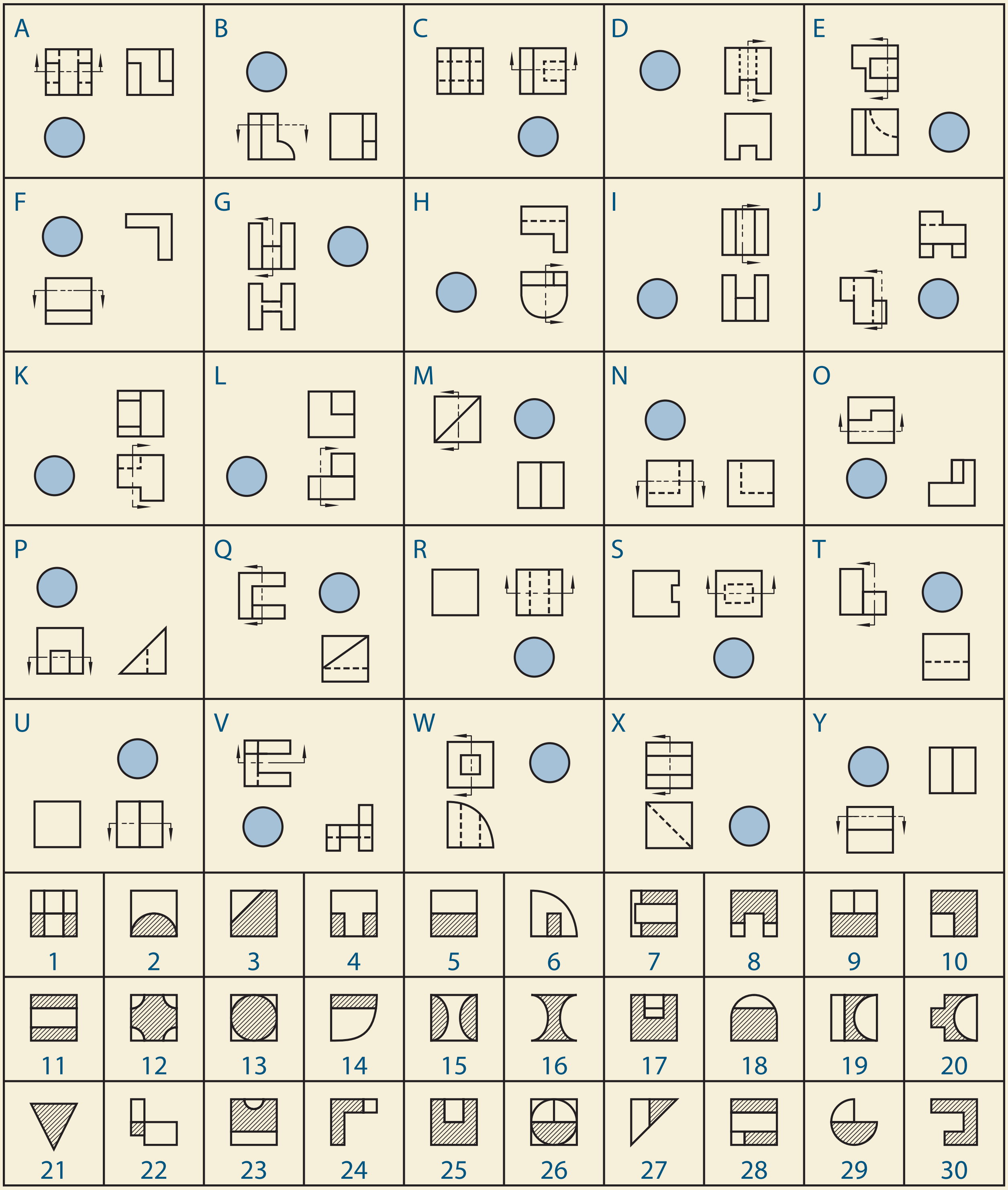


FIGURE P10.3.

10.17 PROBLEMS (CONTINUED)

4. In the problem shown in Figure P10.4, the views indicated by the circles are to be the location of offset section views. Select the correct section view to complete each problem from the 30 proposed views shown. A section view choice may be used more than once. A correct answer may not be available as a choice.

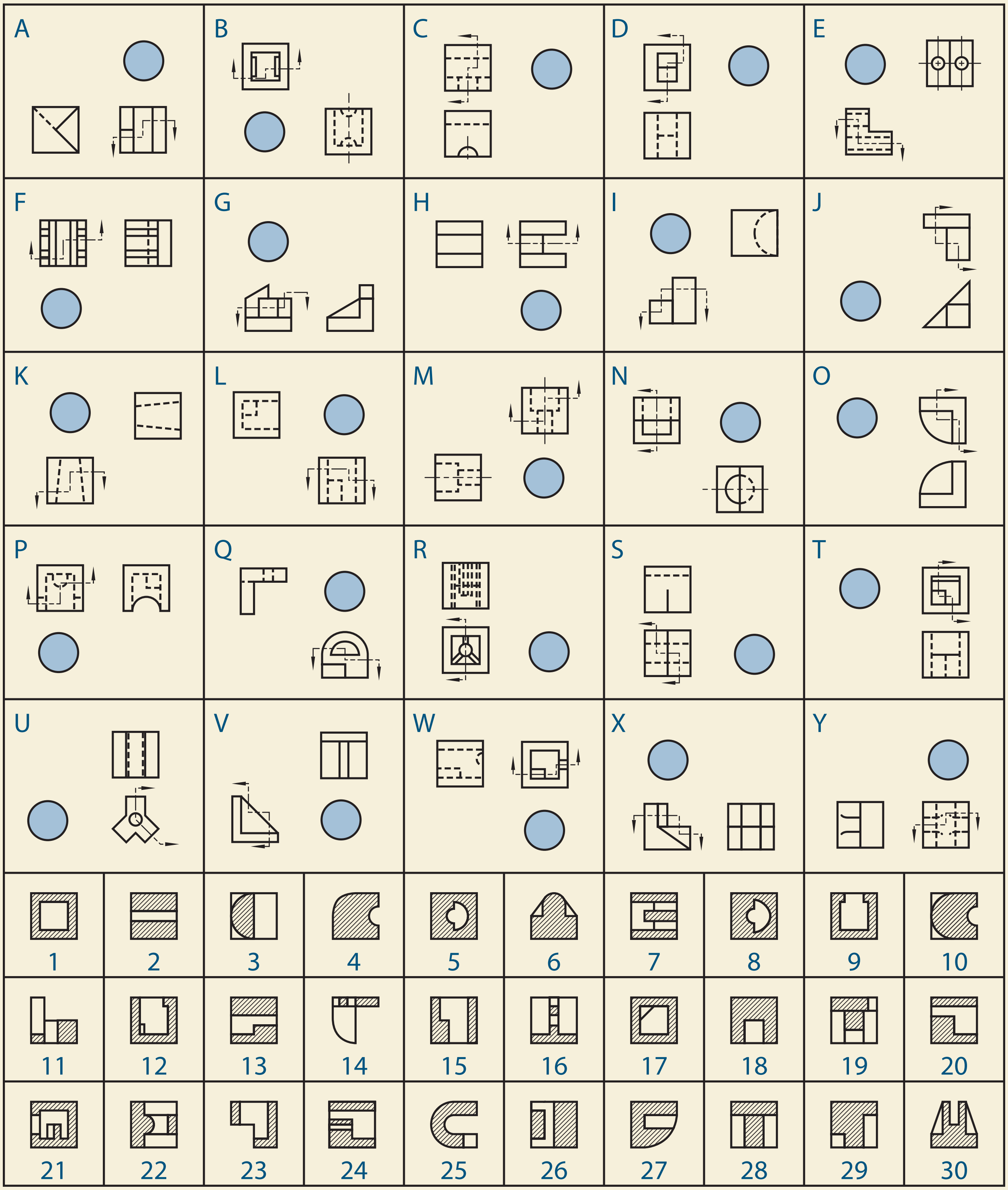
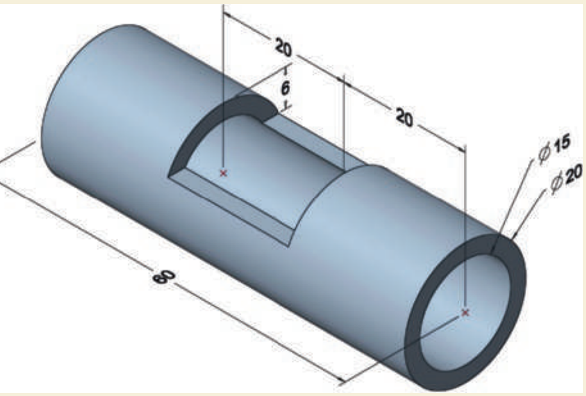


FIGURE P10.4.

10.17 PROBLEMS (CONTINUED)

5. For each object represented in Figure P10.5, create a multiview drawing to fully describe the object, including the indicated full section views to reveal interior detail. When the precise location of the cutting plane line for the full section is not specified, choose the location to best reveal the interior detail.



(a)

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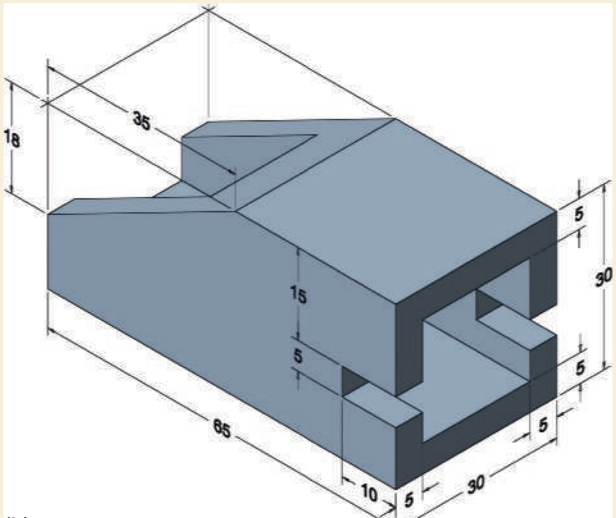

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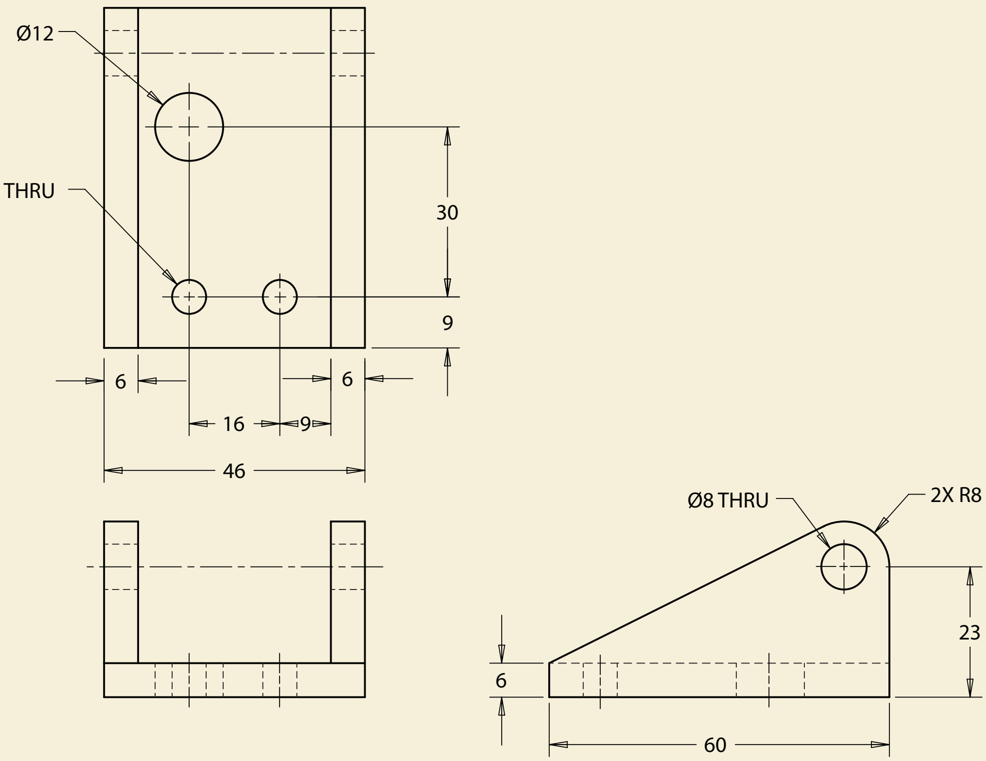

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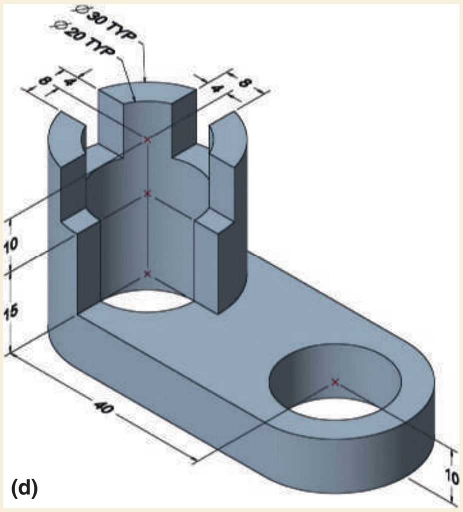
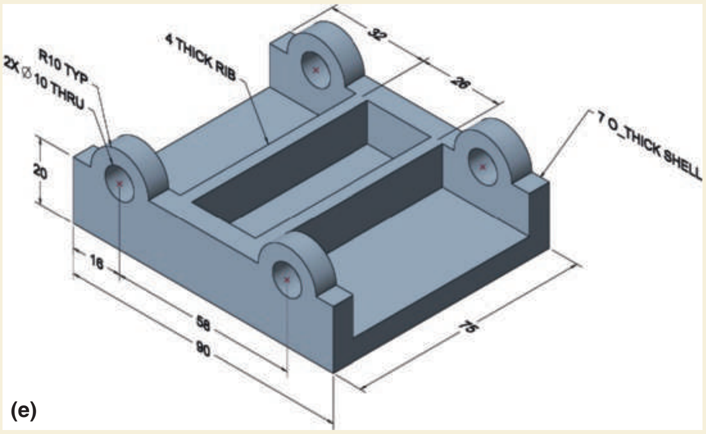
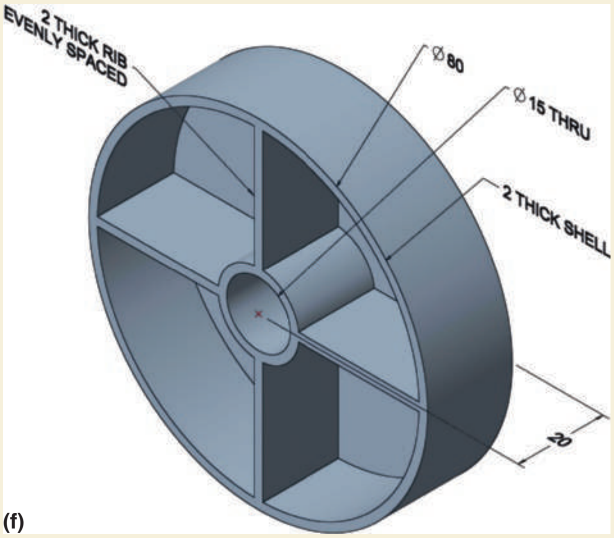
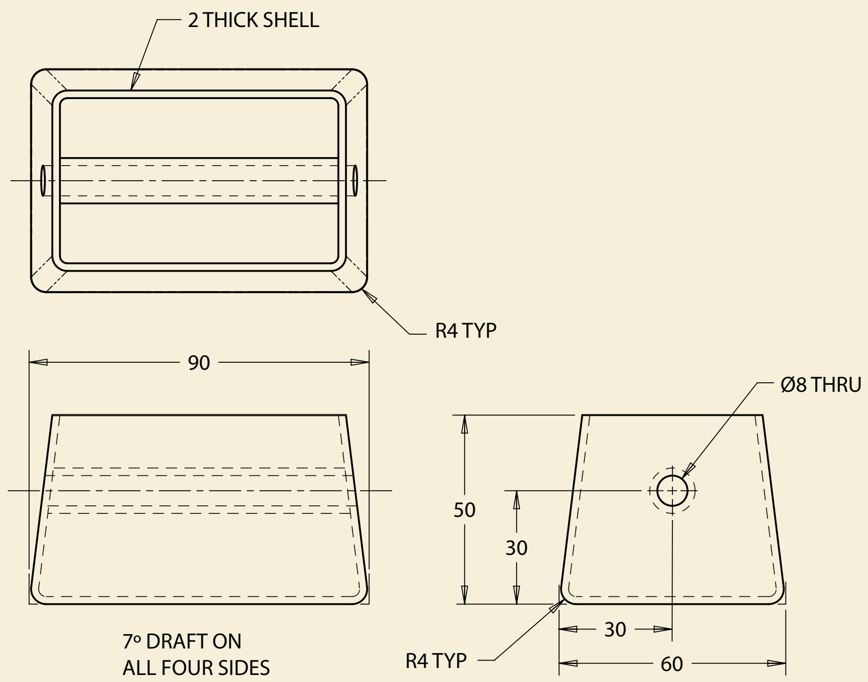
10.17 PROBLEMS (CONTINUED)







10.17 PROBLEMS (CONTINUED)



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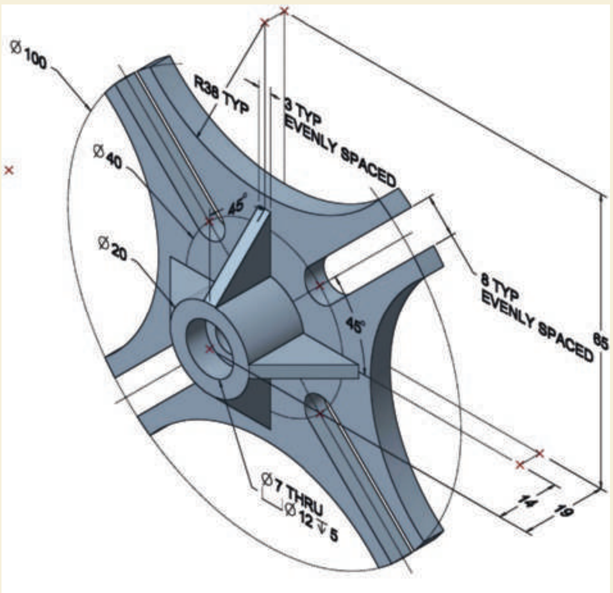

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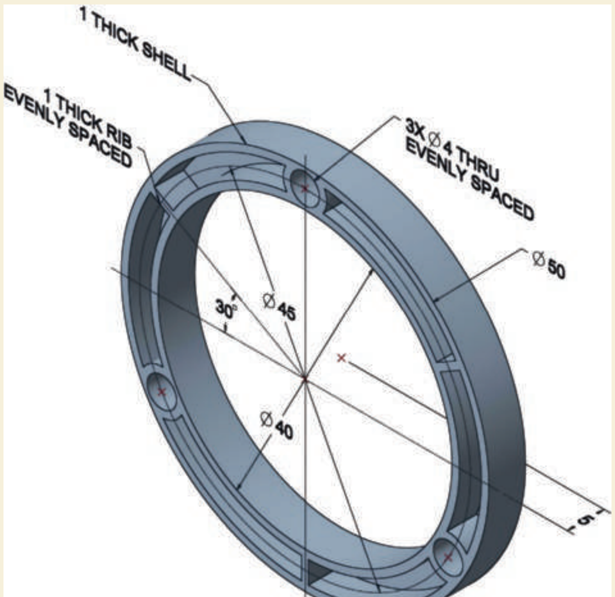

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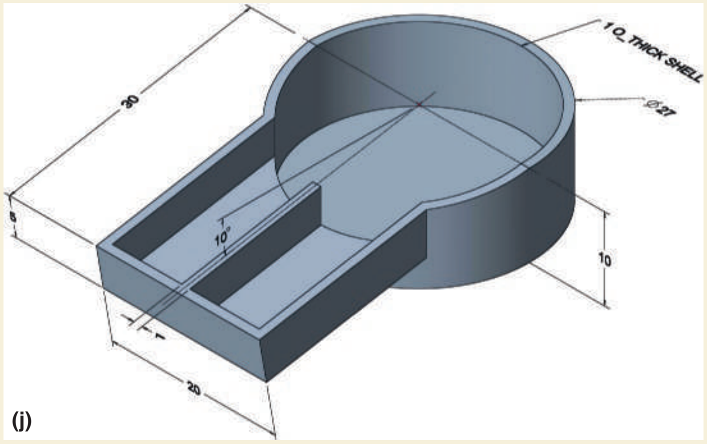
10.17 PROBLEMS (CONTINUED)



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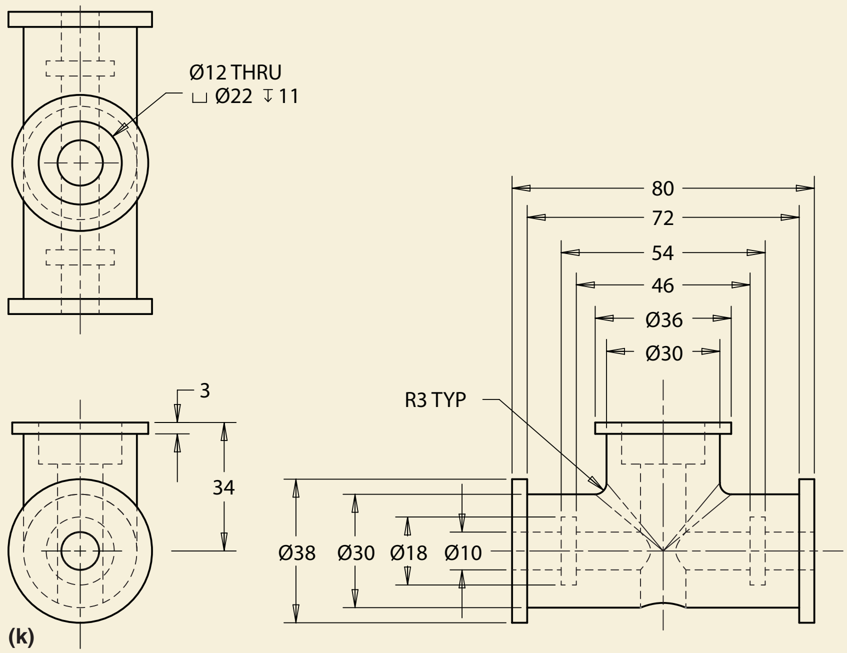

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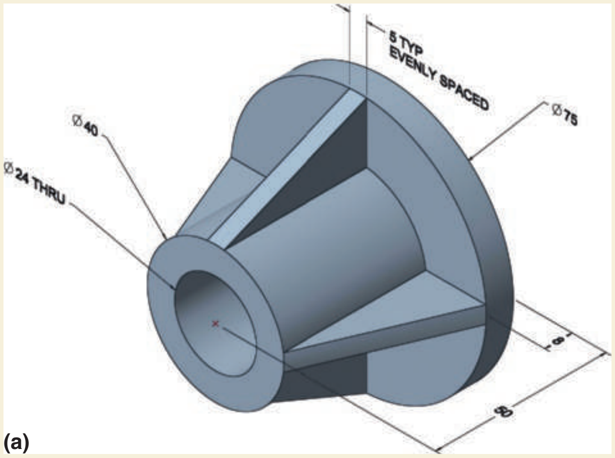
FIGURE P10.5.

10.17 PROBLEMS (CONTINUED)

6. For each object represented in Figure P10.6, create a multiview drawing to fully describe the object, including the indicated half section views to reveal interior detail. When the precise location of the cutting plane line for the half section is not specified, choose the location to best reveal the interior detail.



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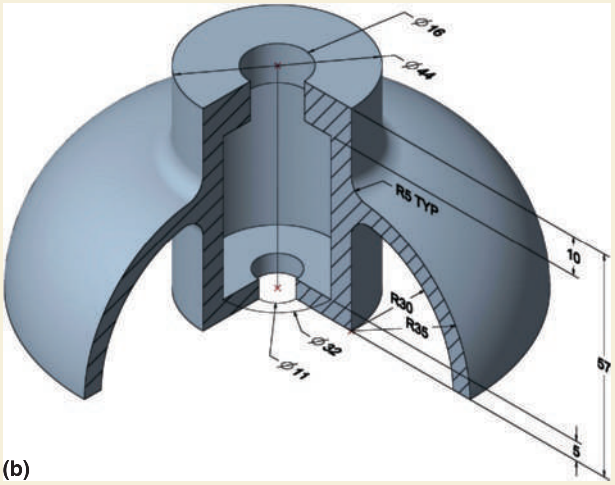

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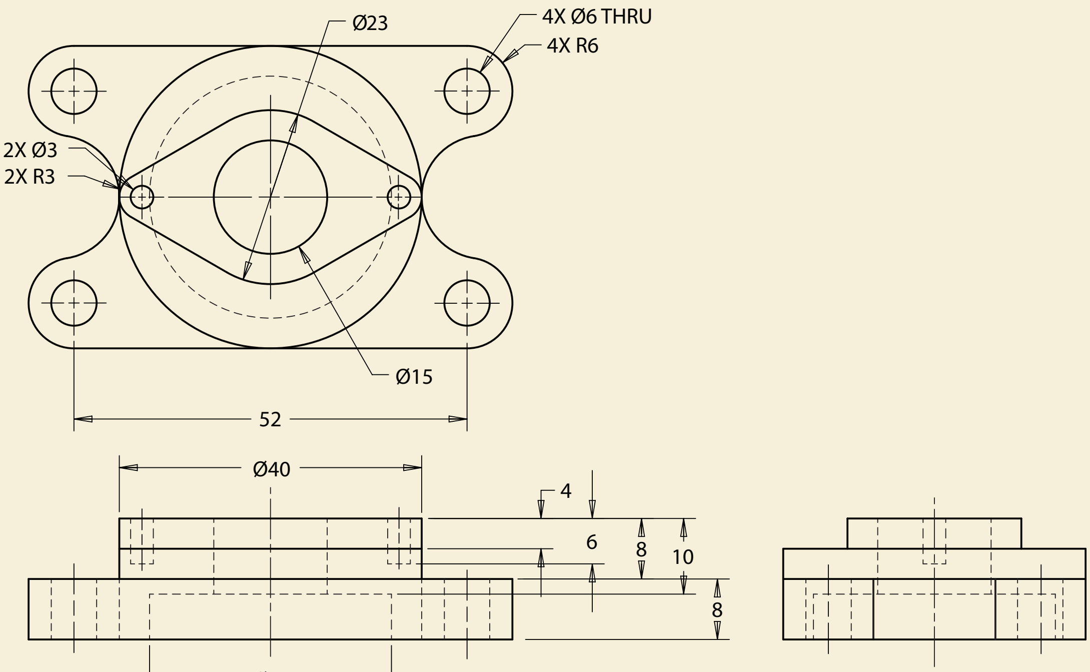

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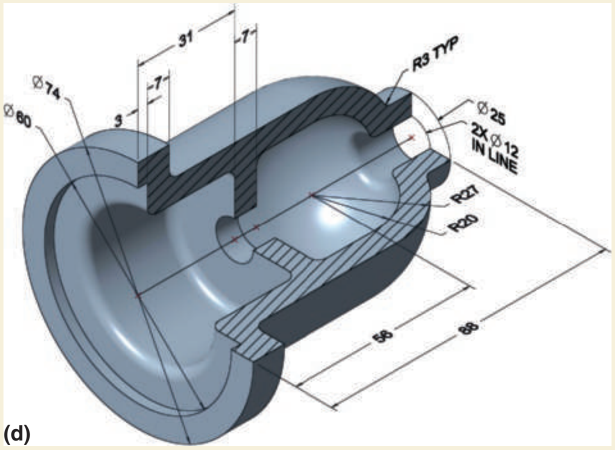
10.17 PROBLEMS (CONTINUED)



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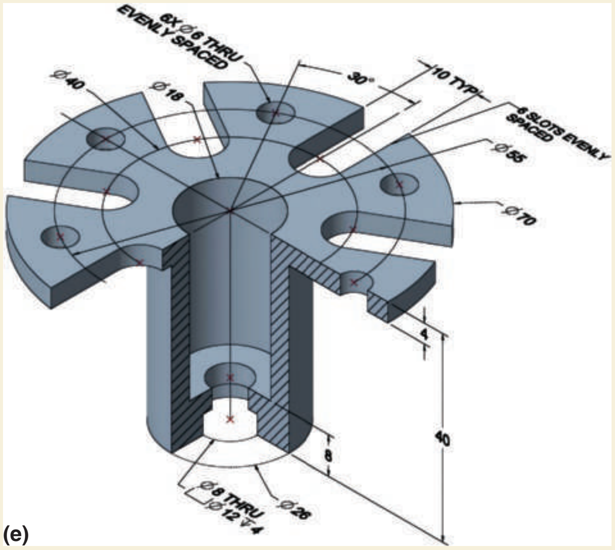

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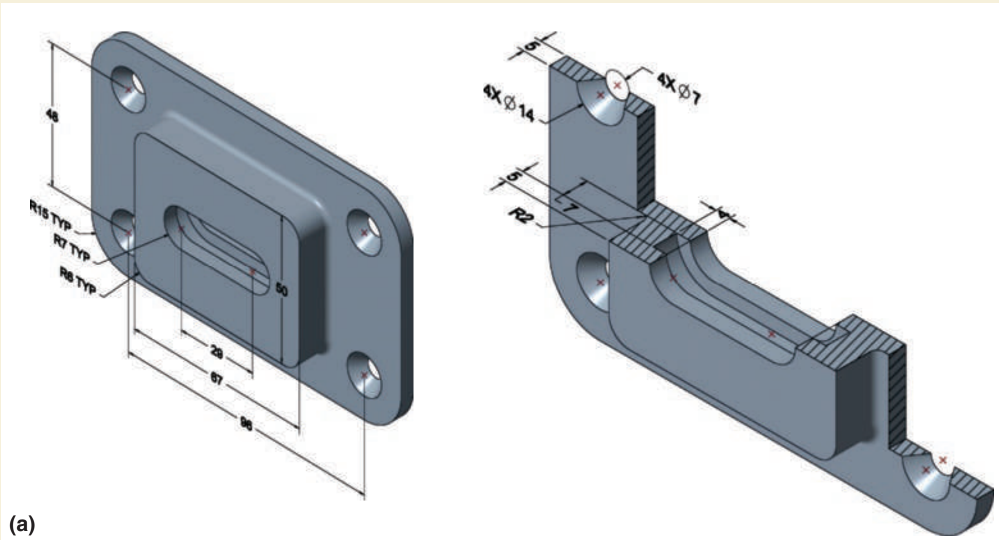
FIGURE P10.6.

7. For each object represented in Figure P10.7, create a multiview drawing to fully describe the object, including the indicated offset section views to reveal interior detail. When the precise location of the cutting plane lines for the offset sections are not specified, choose the locations to best reveal the interior detail.



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10.17 PROBLEMS (CONTINUED)

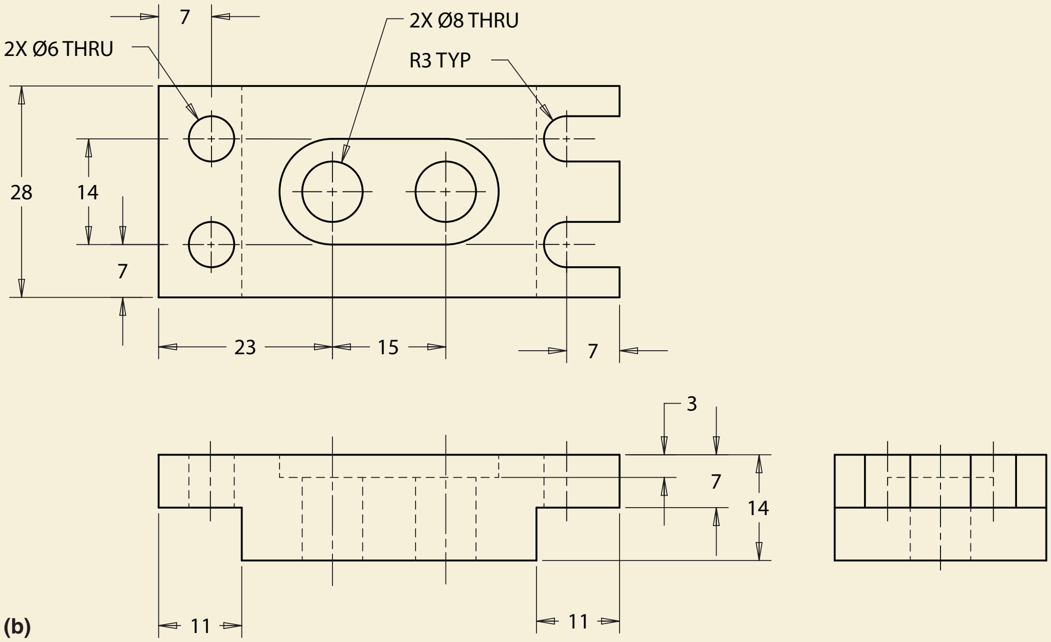

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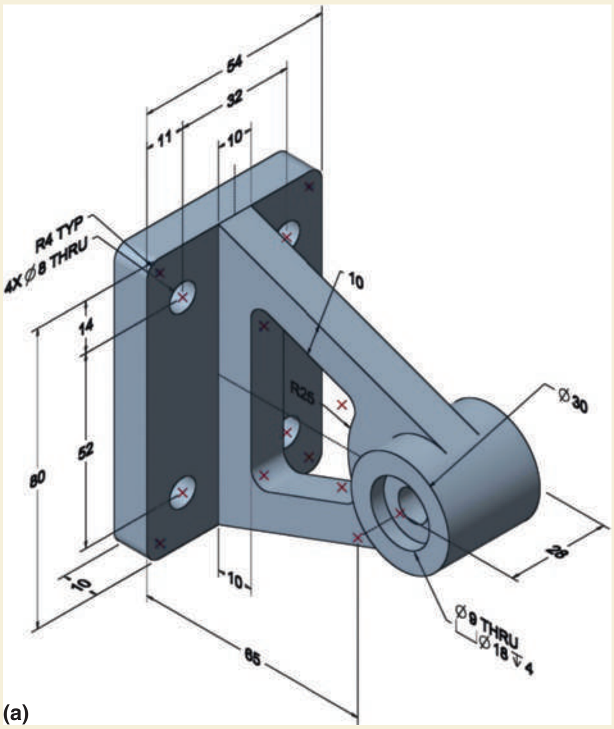
FIGURE P10.7.

8. For each object represented in Figure P10.8, create a multiview drawing to fully describe the object, including the indicated removed or revolved section views to reveal interior detail. When the precise location of the cutting plane line for the removed or revolved section is not specified, choose the location to best reveal the interior detail.



(a)

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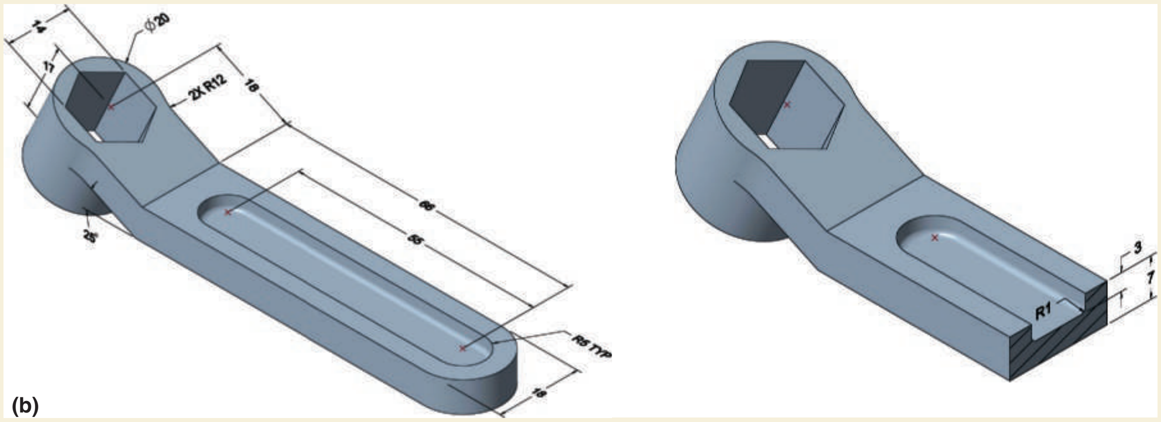
10.17 PROBLEMS (CONTINUED)



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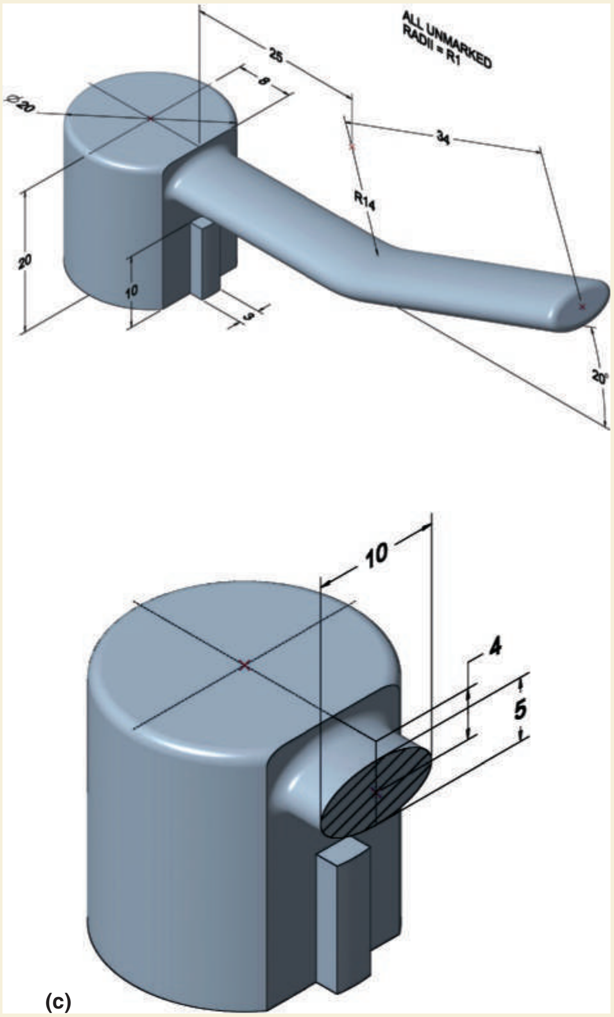

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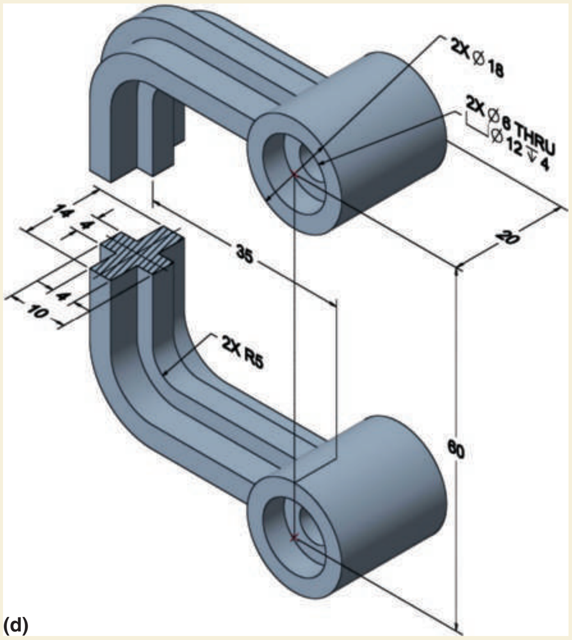

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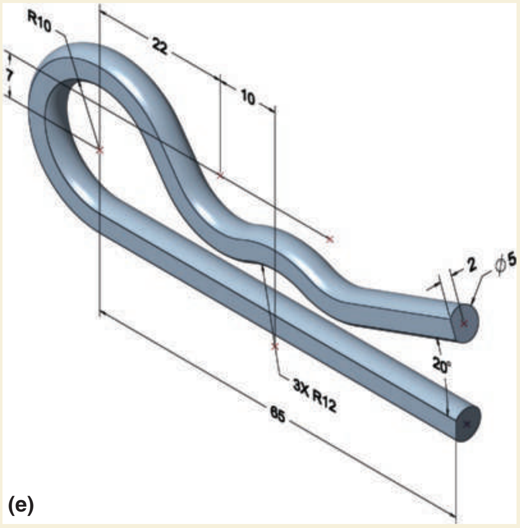

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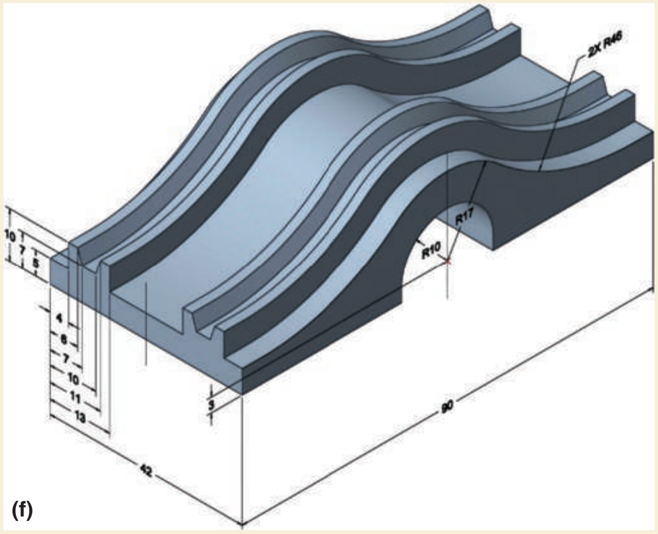

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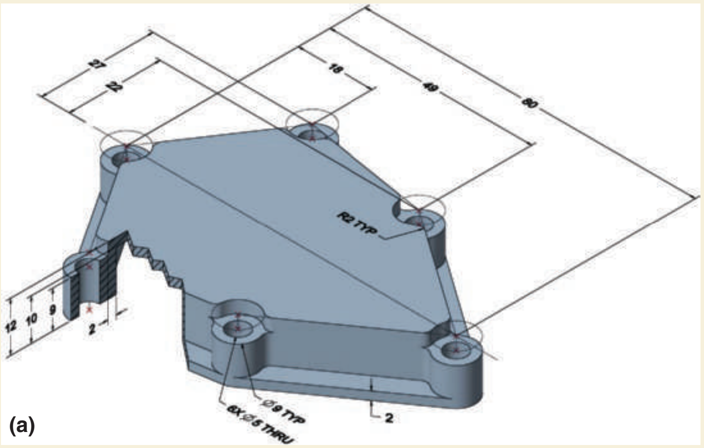
FIGURE P10.8.

9. For each object represented in Figure P10.9, create a multiview drawing to fully describe the object, including a broken-out-section view to reveal interior detail. When the precise location of the broken-out section is not specified, choose the location to best reveal the interior detail.



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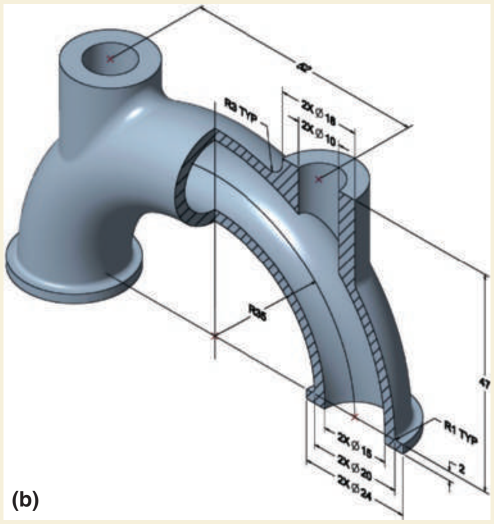

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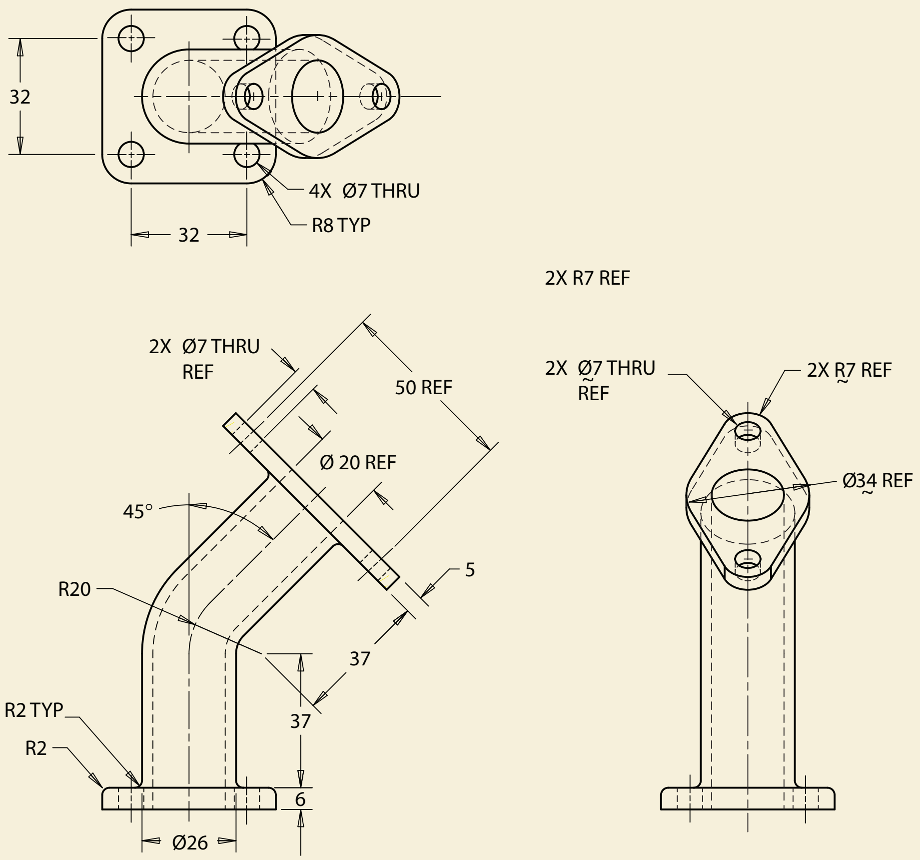

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FIGURE P10.9.

10. For each object presented in a multiview format in Figures P10.5–P10.9, create an isometric pictorial of the remaining object after it has been sectioned.

CHAPTER 11

AUXILIARY VIEWS

OBJECTIVES

After completing this chapter, you should be able to

- Describe an auxiliary view
- Describe situations where an auxiliary view is desired
- Locate top-adjacent, front-adjacent, and side-adjacent auxiliary views constructed from primary views
- Create an auxiliary view of an inclined surface

11.01 INTRODUCTION

Auxiliary views are most commonly used to determine the true shape of inclined or oblique surfaces. Auxiliary views also are used to locate characteristics and relationships between lines and planes, such as:

- Visibility of lines and planes. For example, at a chemical facility, you will find buildings containing pumps and pipes crossing over and under each other. Auxiliary views and visibility principles are used to determine whether a pipe is on top or in front of another pipe.
- The shortest distance between two lines; for example, designing a brace to separate two pipes at their closest point.
- The shortest distance from a point to a plane. For example, in an ore mine, the entrance tunnel leading to a vein of ore should be short for economic reasons, but it also should have the optimal slope for the transportation of the ore.
- The slope of a line or plane; for example, the downward angle of a feeder that delivers parts to a conveyor belt.
- The angle between two planes, also called a dihedral angle. For example, the angle between the face and flank of a tool bit is ground to 62° to cut steel and 71° to cut cast iron.
- The intersection of two planes; for example, the intersection of one tube with another on a bicycle frame.

In this chapter, you will learn about the basics of using auxiliary views for examining inclined surfaces on solid objects. A supplemental chapter covers fundamentals in descriptive geometry, a graphical technique that was developed to explore characteristics and relationships between lines and planes. In descriptive geometry, auxiliary views are constructed to define relationships between points, lines, and planes in space; however, here you will focus on the fundamentals in creating auxiliary views of object surfaces, leaving the more advanced techniques for your later exploration.

11.02 Auxiliary Views for Solid Objects

In a previous chapter, you learned about multiview drawings of objects. You learned how to construct the top, front, and side views of an object by projecting these views onto the surfaces of a glass box surrounding the object and then unfolding the panes of glass so all of them were in the same plane. You also learned that normal surfaces are parallel to one of the six primary views, **inclined surfaces** are perpendicular to one of the primary views but are not parallel to any of the primary views, and **oblique surfaces** are neither parallel nor perpendicular to any of the primary views.

For normal surfaces, a frontal surface is parallel to the front view (or front pane on the glass box). A frontal surface is seen as an edge in the top and side views and is seen in its **true shape** and size in the front view. Likewise, horizontal surfaces are seen in true shape and size in the top view (they are parallel to the top view), and profile surfaces are seen in true shape and size in the side view (they are parallel to the side view). Thus, for a surface to be seen in its true shape and size, it must be parallel to the given view.

What about an inclined or oblique surface? You learned previously that these types of surfaces are not parallel to any of the primary views. Because they are not parallel to any of the primary views, they are seen **foreshortened** in the primary views—they are not seen in their true shape or size in the top, front, or side views. In the remainder of this chapter, you will learn about the creation of auxiliary views that show inclined surfaces in true shape and size. The creation of auxiliary views to determine the true size and shape of oblique surfaces will not be covered here. If you want to create successive auxiliary views to obtain true size and shape of an oblique surface, you can refer to the supplementary chapter on descriptive geometry.

Before you dive into understanding how auxiliary views are created, step back and think about the creation of the primary views. A horizontal surface is parallel to the top view. When you constructed the top view, you projected it onto the top pane of the glass box using **projection rays** that were perpendicular to the pane. In the front view, these projection rays extended perpendicularly from the horizontal **edge view** of the normal surface toward the top viewing plane. When the top pane was unfolded, the projection rays from the front view defined the outer limits of the surface as seen in the top view. The horizontal pane of glass also appears as an edge in the front view—an edge that is parallel to the surface in question. Figure 11.01a shows a simple object with the projection rays used to create the top view indicated, and Figure 11.01b shows the top and front views of the object after the imaginary glass box has been unfolded. The two horizontal surfaces—seen here as two rectangles, one large and one small—are shown in their true shape and size in the top view.

Figure 11.02 shows the top, front, and right-side views of a drill jig; a pictorial view of the jig is also shown for clarity. Surface A on the drill jig is seen as an edge in the front and side views and is seen in its true shape in the top view, signifying that surface A is parallel to the top plane. The edge view of inclined surface B is seen in the front view as highlighted in Figure 11.03; however, since neither the top nor right-side view of the object is parallel to the surface, you are not seeing surface B in its true shape and size in either view. Surface B is foreshortened in both the top and side views.

To manufacture this part, a machinist will need to know exactly where the hole through the inclined surface B is located. However, since none of the views shows the surface in its true shape and size, it would be difficult for the machinist to make this part accurately, based on the given information. If a view that is parallel to this inclined surface could somehow be constructed, surface B would be seen in its true shape and size in this new view, solving the problem of accurately locating the hole for the machinist. Because this new view is not one of the primary views of the object, it is called an auxiliary view—essentially any extra view, other than a pictorial, which has been constructed for overall clarity.

FIGURE 11.01. The image of the object is projected onto a horizontal surface to create the top view in (a). The front and top views are shown in (b).

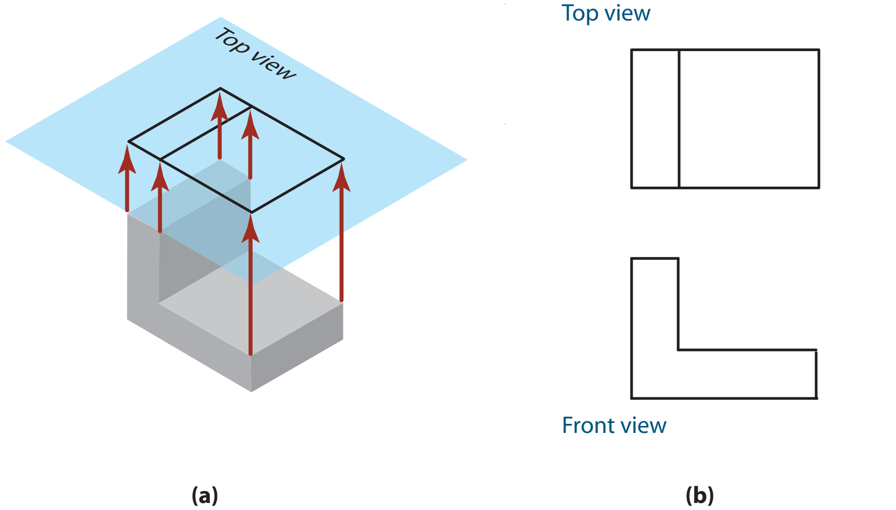

FIGURE 11.02. An object with an inclined surface shown in the preferred orthogonal view configuration with top, front, and right-side views. Surface A is parallel to the top viewing plane (and, therefore, will be shown in its true shape in the top view) and in its edge view in the front and right-side views.

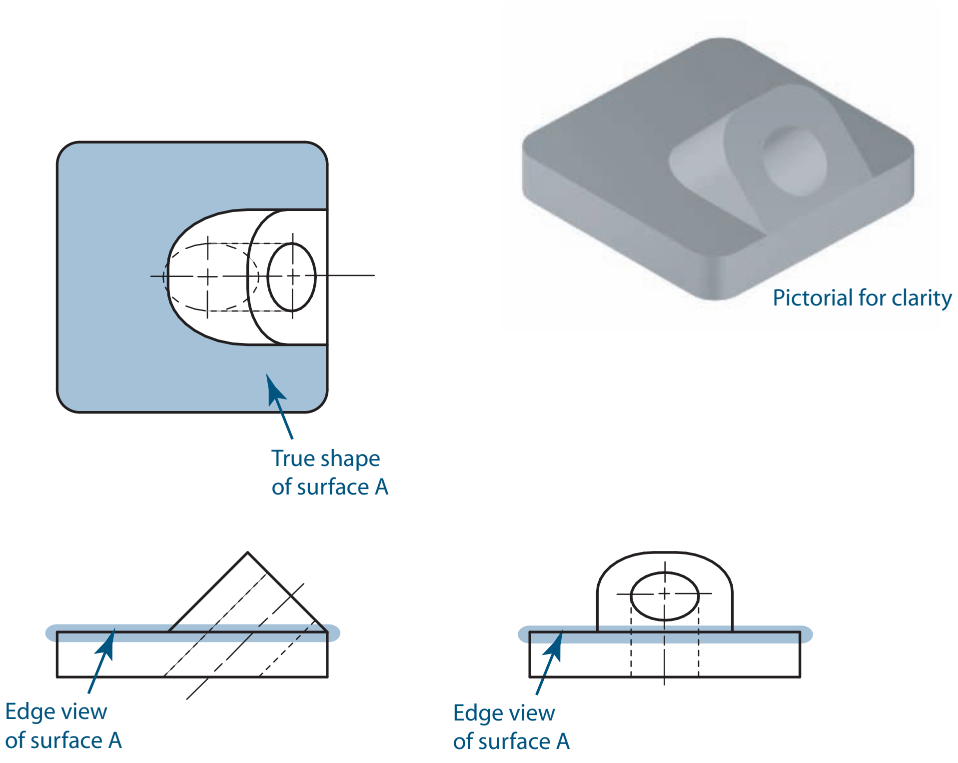

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FIGURE 11.03. Surface B is seen in an edge view in the front view but is not seen in its true shape in either the top or right-side views.

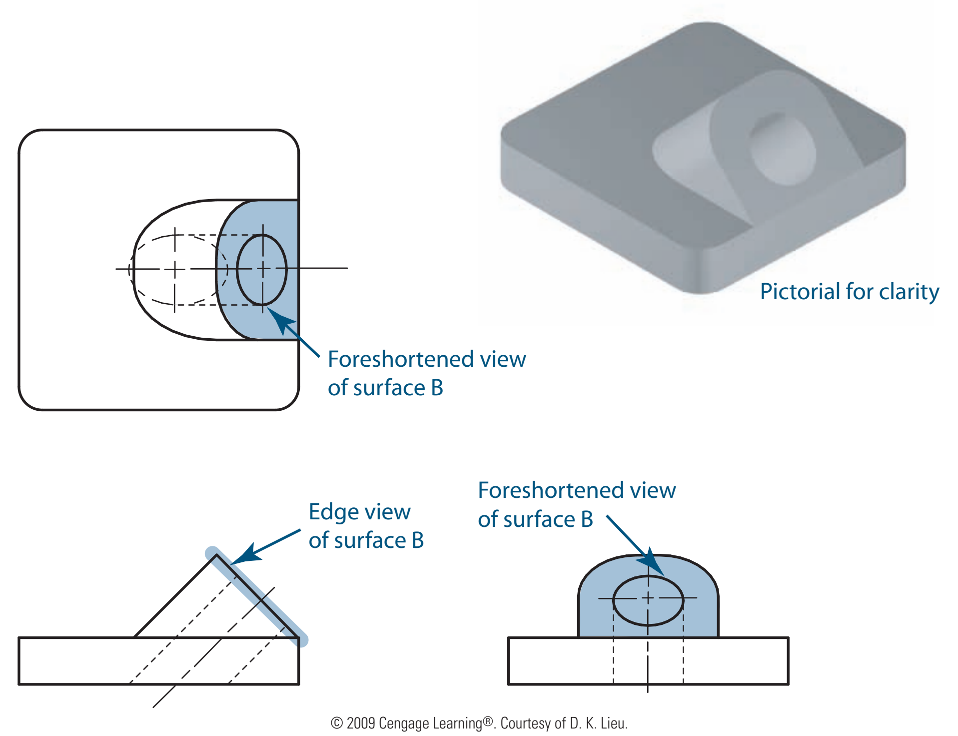

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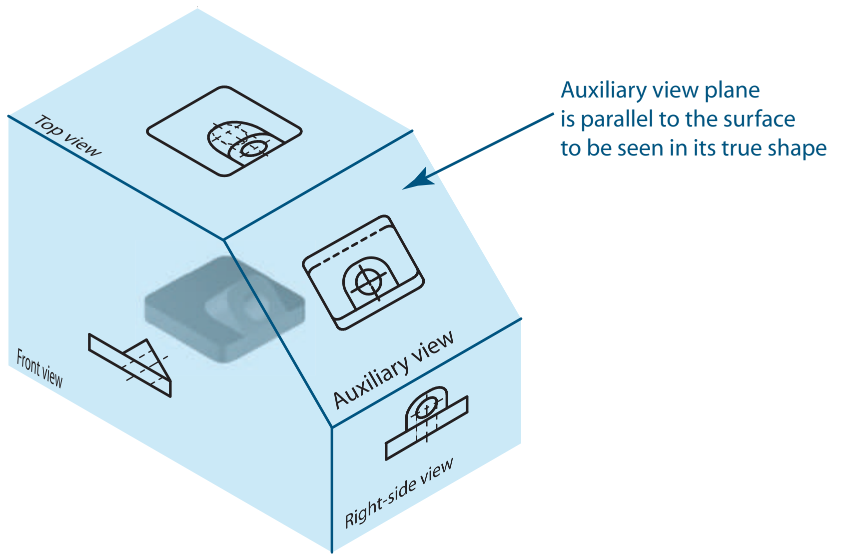
FIGURE 11.04. An additional plane (pane) added to the glass box to show the true shape of the inclined surface.



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To understand how auxiliary views are created, it is useful once again to think about the object as if it were surrounded by a glass box. This time, however, imagine an extra pane of glass has been added to the glass box that is parallel to the inclined plane as shown in Figure 11.04. The inclined surface can be projected perpendicularly onto this angled pane of glass, resulting in a view of its true shape and size. The glass box can now be unfolded to show all views, including the auxiliary view, on a single plane. Figure 11.05 shows the glass box with the panes of glass unfolded, including the pane with the auxiliary view on it.

FIGURE 11.05. The glass box unfolded, showing the top, front, right-side, and auxiliary viewing planes on a single plane.

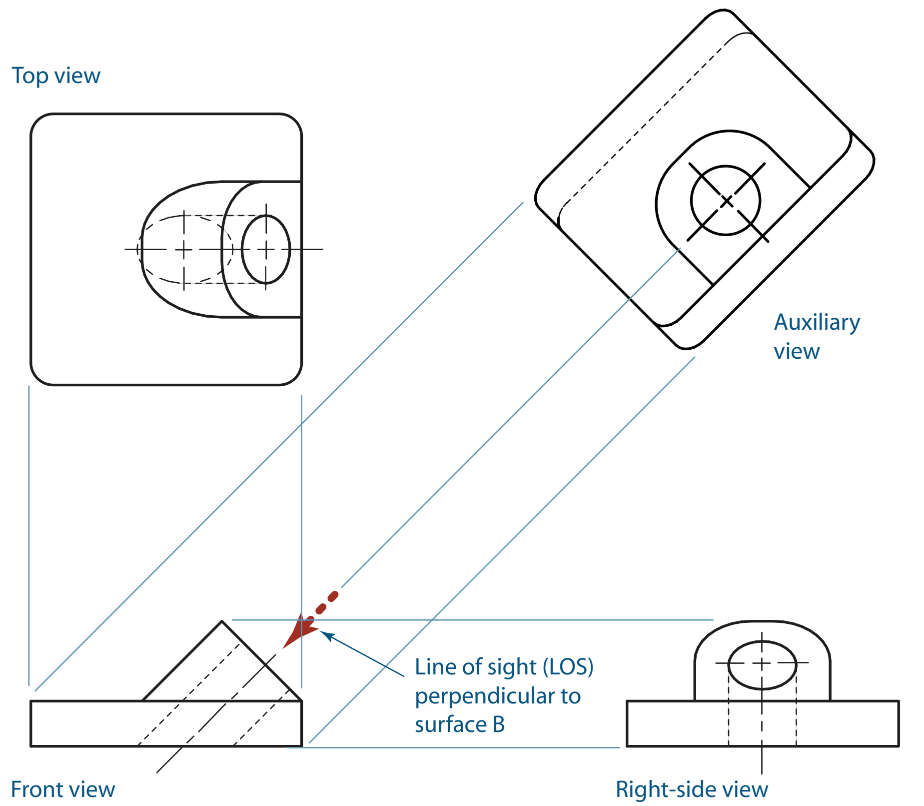

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Figure 11.06a shows a full auxiliary view of the drill jig, while Figure 11.06b shows a partial auxiliary view of the drill jig. The difference is that the full auxiliary view shows all surfaces of the object, whether they are true shape or not, whereas the partial auxiliary view shows only the surface for which the true shape and size are required—the inclined surface. Full auxiliary views are usually not necessary to construct because you do not gain additional information from including other surfaces, which may even serve to clutter the drawing in such a view—the primary views show you what you need to know about the other surfaces on the object. Since your primary purpose in creating an auxiliary view is for clarity and not for confusion, partial auxiliary views of objects are the most common type of auxiliary views you will create.

Some conventions have been developed to enable you to organize your work when you are working with auxiliary views. One of the conventions is in labeling views. You know the glass box is composed of six principal planes—two horizontal planes (the top and bottom planes), two frontal planes (the front and back planes), and two profile planes (the right- and left-side planes). To help identify which views you are working with, when creating auxiliary views, it is standard practice to label the top, front, and side planes with the capital letters *H*, *F*, and *P*, respectively. (In this case, *H* represents the horizontal, or top plane; *F* represents the front plane; and *P* represents the side, or profile plane.) Any auxiliary planes you create should be numbered sequentially.

Figure 11.07 shows the three principal planes of the glass box labeled H, F, and P and the angled glass plane for the auxiliary view labeled 1. Figure 11.08 shows what these planes look like after the glass box has been unfolded. The glass box hinges are referred to as **reference lines** or fold lines and will be used to transfer measurements as you construct your auxiliary views. One other convention that has been developed for work with auxiliary views is that the fold lines are usually shown in the multiview drawings. This is in contrast with standard practice for constructing multiview drawings that you learned about in a previous chapter, where fold lines are not usually shown.

FIGURE 11.06. A full auxiliary view is shown in (a). A partial auxiliary view, showing only the inclined surface, is shown in (b).

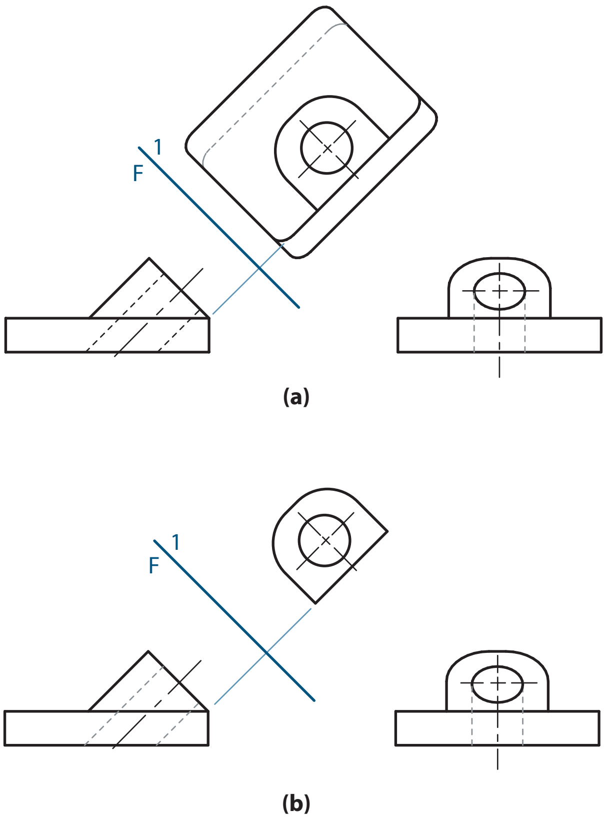

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FIGURE 11.07. The glass box planes labeled.

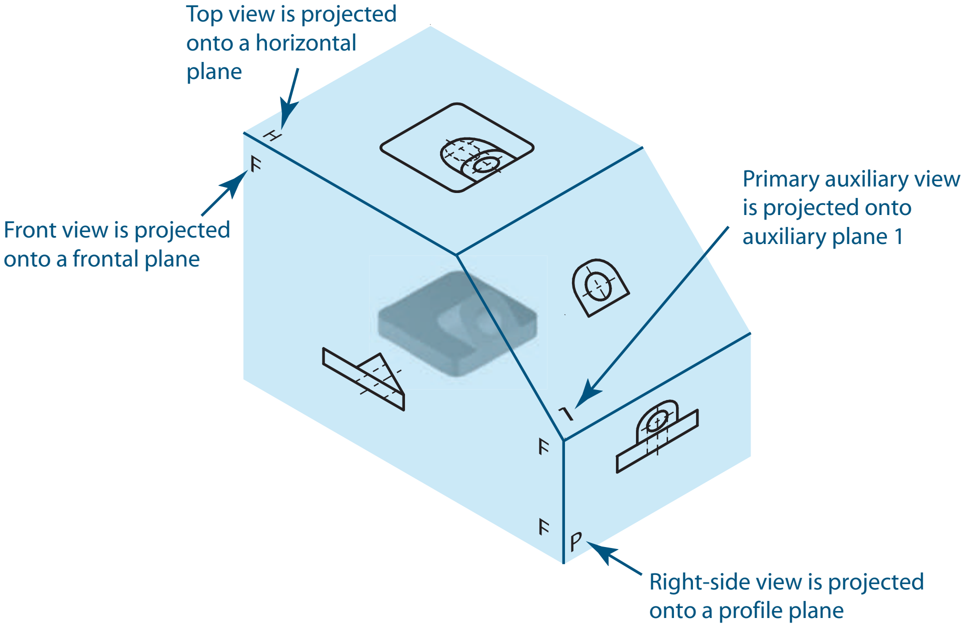

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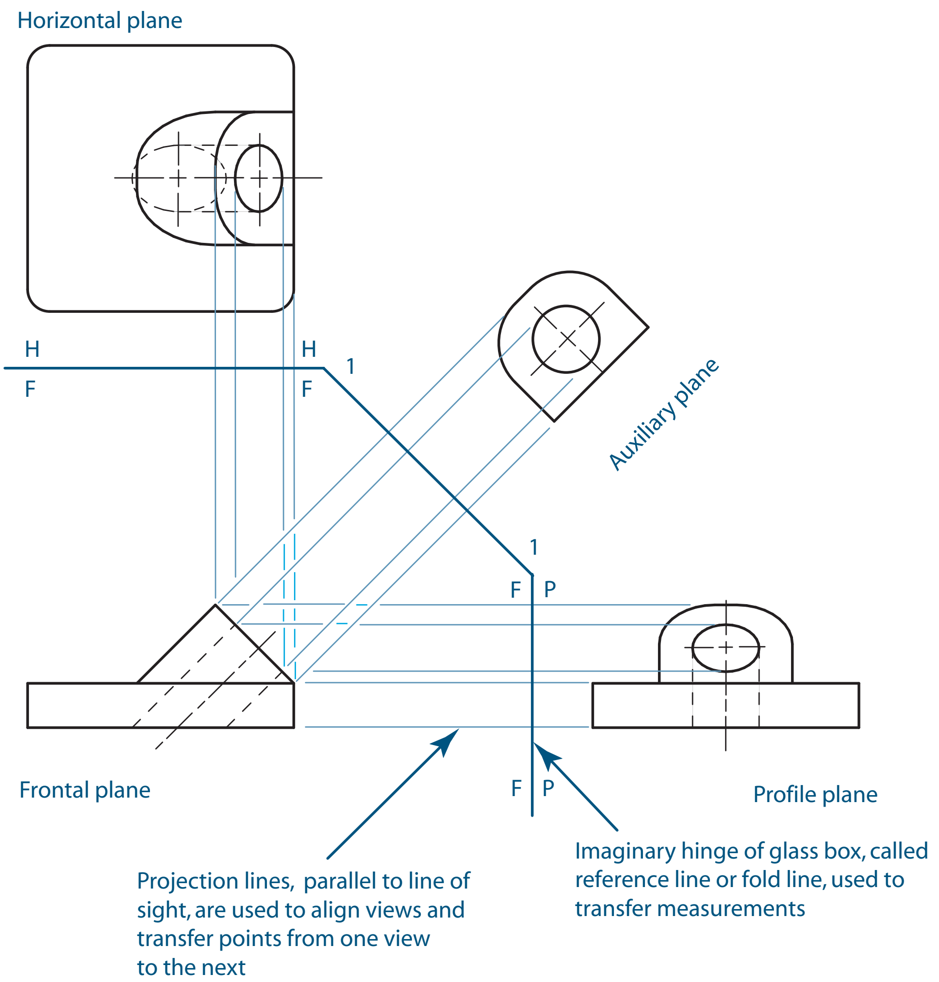
FIGURE 11.08. The glass box opened to show proper view alignment.



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Understanding an auxiliary view is simple if you remember the principles of orthographic projection. Any view that is projected from one view into the next is *adjacent* to the first view. The auxiliary view showing the inclined surface B in true size and shape is adjacent to the front view; the top and side views are also adjacent to the front view. In general, **adjacent views** are aligned side by side and share a common dimension. For example, the right and front views that are adjacent to each other show the height of the object in common. **Related views** are adjacent to the same view and share a common dimension. For the drill jig, the top and auxiliary views are adjacent to the front view. In this case, the top and auxiliary views are related to each other and share a common dimension—the object depth. The depth that point A is away from the fold line is visible in the top view, and this distance is preserved in the related auxiliary view. Figure 11.09 shows how the depth dimension for point A is preserved from the top into the auxiliary view.

The auxiliary view you have been working with is called a front-adjacent view because it was constructed adjacent to the front view. It also is possible to draw top-adjacent and side-adjacent views, depending on which primary view shows the inclined surface in question as an edge. For example, if the inclined surface is an edge view in the top view, you would project the auxiliary view from the top view to obtain its true size and shape, thus creating a top-adjacent view. Similarly, if the inclined surface is seen as an edge in the side view, a side-adjacent auxiliary view would be used to show the surface in its true size and shape. Figure 11.10 illustrates a side-adjacent auxiliary view that shows the true size and shape of the indicated inclined surface.

FIGURE 11.09. Comparison between adjacent and related views.

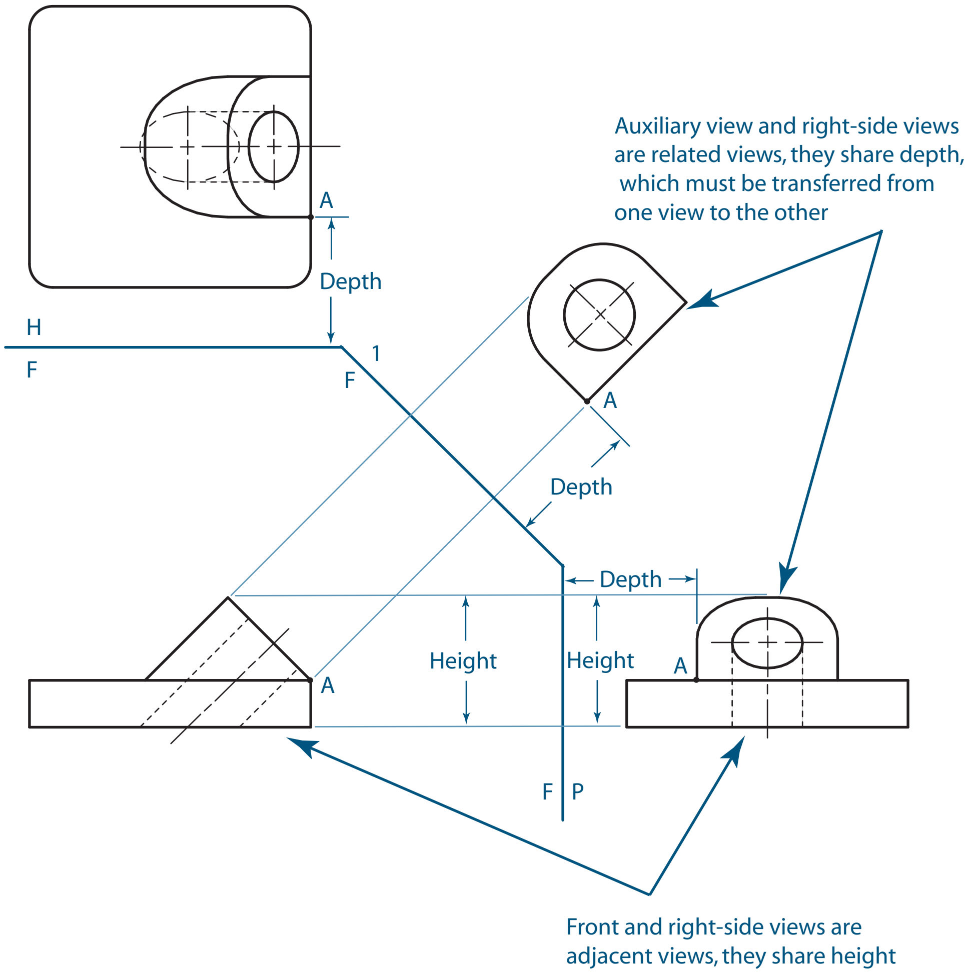

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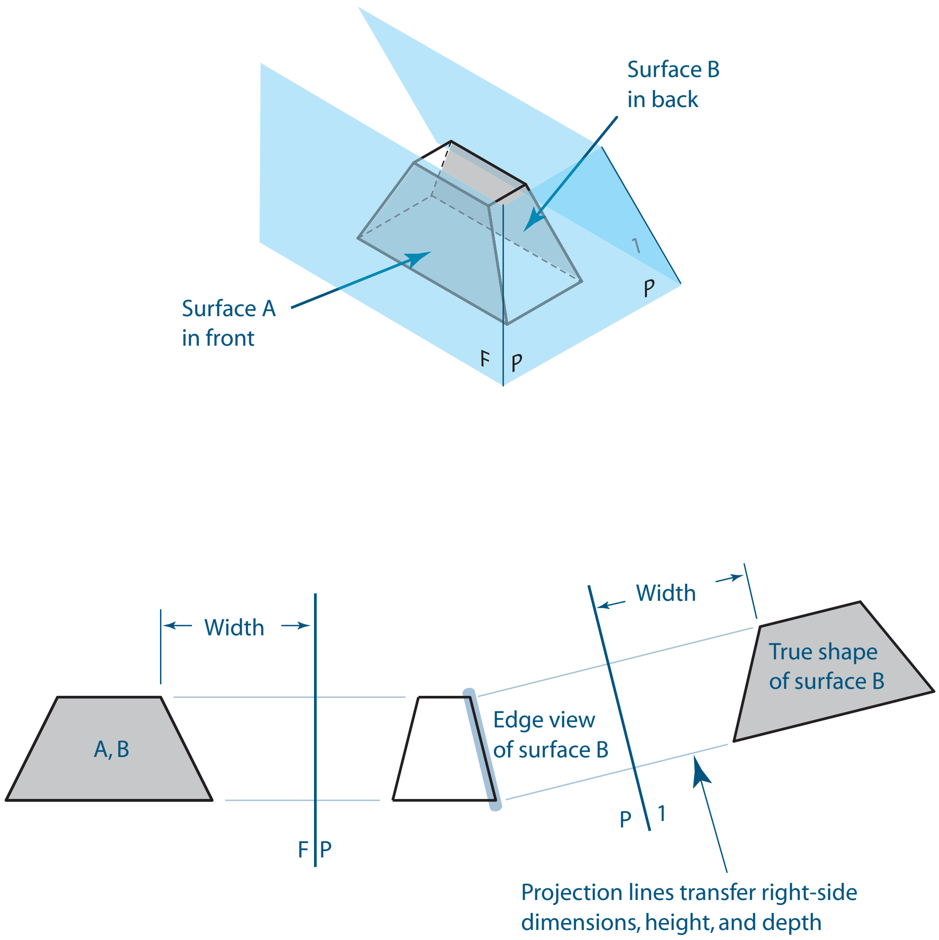
FIGURE 11.10. A profile-adjacent auxiliary view of surface B.



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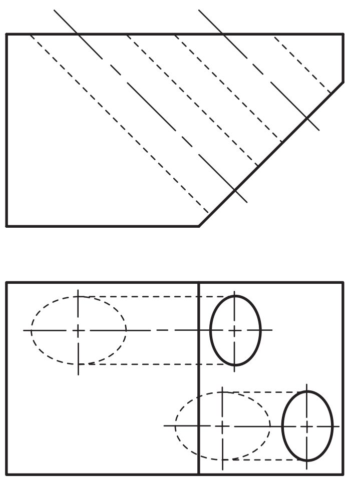
11.03 Auxiliary Views of Irregular or Curved Surfaces

Most of the time, you will need to construct an auxiliary view of a surface on an object that shows a curved or irregular feature, like the drill jig from the previous example. Consider the object shown in Figure 11.11. This object contains two holes through an inclined surface. The inclined surface is seen in edge view in the top view, and the holes appear as ellipses in the front view. The holes would also appear as ellipses in the side view, if it were constructed.

FIGURE 11.11. An object with two holes through an inclined surface.



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When creating an auxiliary view of this surface, you should project several points on the curved edges to define them in the new view. For a circular hole, usually four radial points are sufficient; but, for an irregular curve, you may need to locate several points to obtain an accurate projection. Figure 11.12 shows an auxiliary view of the inclined surface for the object shown in Figure 11.11. In this case, four radial points were transferred into the auxiliary view for each circular hole on the surface.

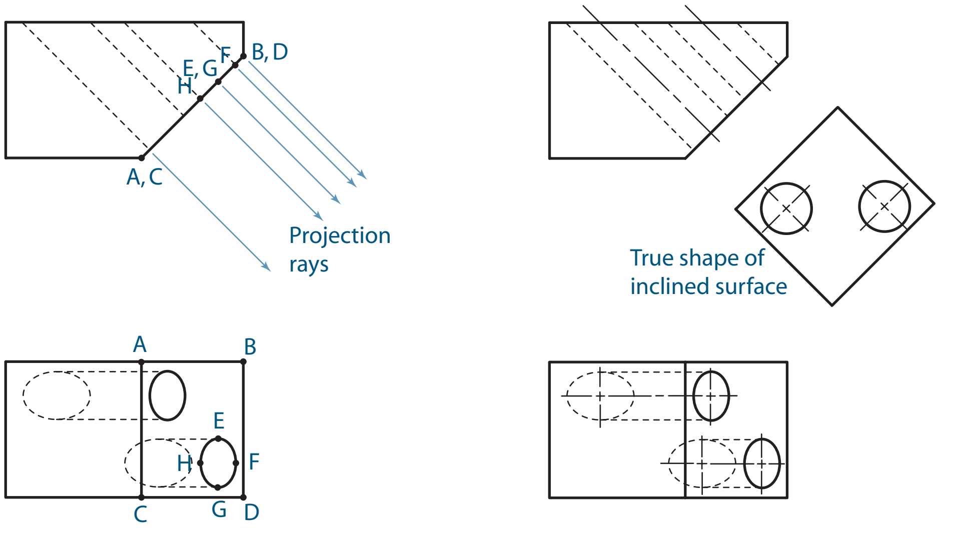
11.04 Creating Auxiliary Views

How do you construct an auxiliary view that shows an inclined surface in true size without the aid of a glass cube? Let's review what you know so far. To see a surface in true size and shape, *you must view it from a plane that is parallel to the surface*. If a viewing plane is parallel to the surface, the edge view of the pane of glass defining the new view will also be parallel to the surface. In addition, *the points on the surface will project perpendicularly into this new view*, similar to the way points are projected perpendicularly from the front into the top view (or from the front into the side view).

In general, the procedure used to create an auxiliary view that shows an inclined surface in true size and shape is as follows:

1. Identify the edge view of the inclined surface in one of the primary views. Note that for an inclined surface, it will only appear as an edge in one view (it will appear as a foreshortened surface in the other two views).
2. Sketch a "fold" line parallel to the edge view of the surface, making sure that there is enough room on the drawing to accommodate the new auxiliary view.
3. Label all of the fold lines (H, F, and P), including the one you just created. Note that if this is your first auxiliary view projected on the drawing that it will likely be labeled surface "1", and thus will appear as H/1, F/1, or P/1 on your drawing.
4. You should now project the points that define the inclined surface along rays that are perpendicular to the fold line for the auxiliary view (H/1, F/1, or P/1, depending on your drawing).
5. Obtain the projected dimensions of the surface into the auxiliary view by observing the same dimension in a related view. (Another way to think of this is that the dimensions are found in a view that is two views back from the auxiliary view in which you are working.)

FIGURE 11.12. An auxiliary view of an inclined surface with holes in it.



(a) Projection of four points on each circle **(b)** Completed auxiliary view

In the case of curved or irregular surfaces, the first step in the procedure is to segment the curved edge into several points and to locate these in the primary views. The points defining the segments will then be projected into the auxiliary view along with the other points defining the surface in Step 4 of this procedure.

Figure 11.13 shows a truncated pyramid, and you want to sketch an auxiliary view of the plane A that shows its true size and shape. Figure 11.14 shows the horizontal and front views of the pyramid. Notice that the front view includes the edge view of the surface in question. Also notice that for this application the fold line is included between the top and front views and is labeled H/F.

To begin the construction of the auxiliary view, sketch a line parallel to the edge view of the surface (surface A) in the front view. This line represents the fold line for the auxiliary view—the edge view of the pane added to the glass box. After you sketch the fold line, you should label the auxiliary view appropriately (F/1 since it will be adjacent to the front view). Figure 11.15 shows the two views of the object with the added fold line and the labels for the fold lines.

The surface will be projected into the auxiliary view along perpendicular projection rays. Only four points define the surface; but in the front view, these points are on top of each other—the endpoints of the line representing the edge view of the plane. Lightly sketch the projection rays from the endpoints of the line into the auxiliary view, keeping in mind that the direction of the projection rays is perpendicular to the fold line (F/1). Since the fold line was drawn parallel to the edge view of the plane, the projection rays also are perpendicular to the edge view of the inclined surface. Figure 11.16 shows the two views of the object with the projection rays extending into the auxiliary view.

Transfer the depth dimensions for each of the points that define the surface in question by looking in the related view (two views back). In this case, the depth is obtained in the top view for transfer into the auxiliary view. Figure 11.17 shows the first point for the surface, point "a," transferred into the auxiliary view.

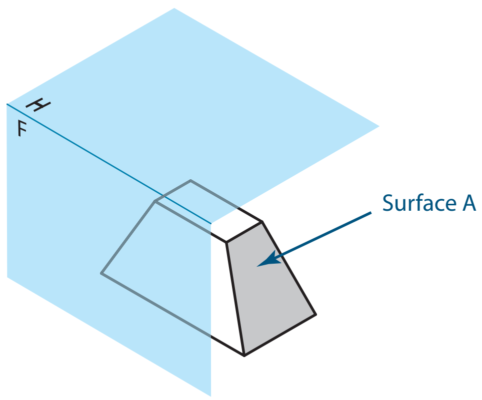


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FIGURE 11.13. A truncated pyramid for constructing an auxiliary view.

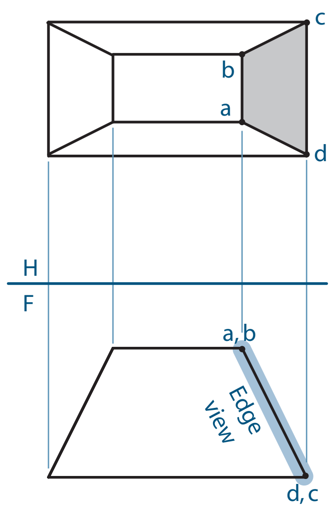

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FIGURE 11.14. Top and front views of the truncated pyramid.

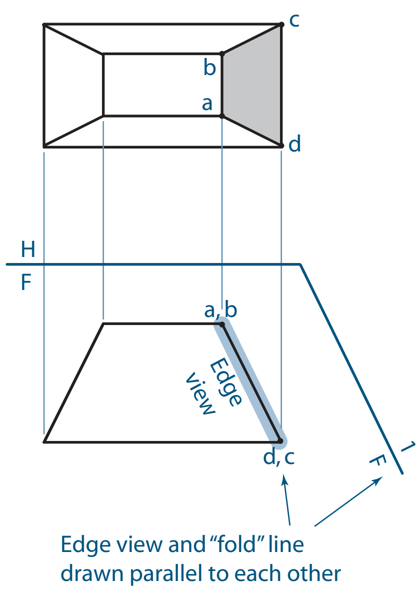

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FIGURE 11.15. The fold line for the auxiliary view, with labels added.

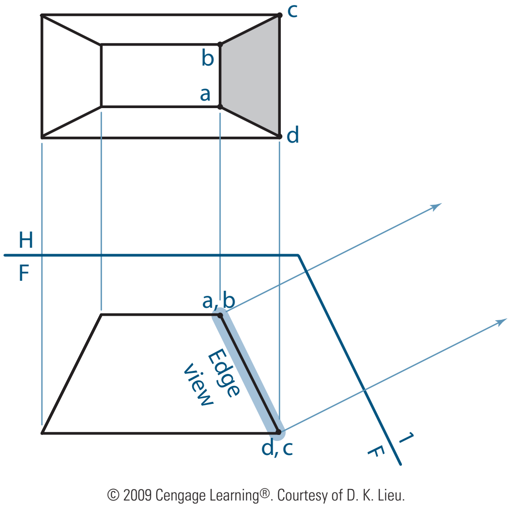

FIGURE 11.16. Projection rays added, extending from the edge view into the auxiliary view.

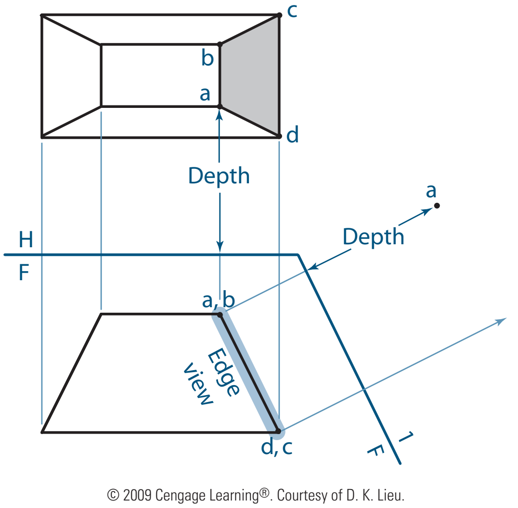

FIGURE 11.17. Point "a" transferred into the auxiliary view.

Continue transferring the remaining points that define the surface into the auxiliary view, each time transferring the depth dimension from the top view. Figure 11.18 illustrates the completed auxiliary view showing the true size and shape of the inclined surface.

As a second example, let's follow this same procedure to obtain the true size and shape of the irregular inclined surface on the object shown in Figure 11.19.

Recall that for projecting an irregular surface into an auxiliary view, you need to first segment the curved edge and locate the points defining the segments in each of

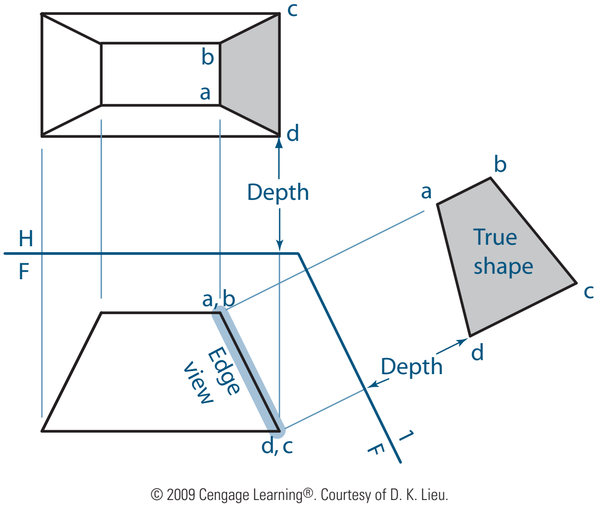

FIGURE 11.18. The auxiliary view showing surface A in true shape.

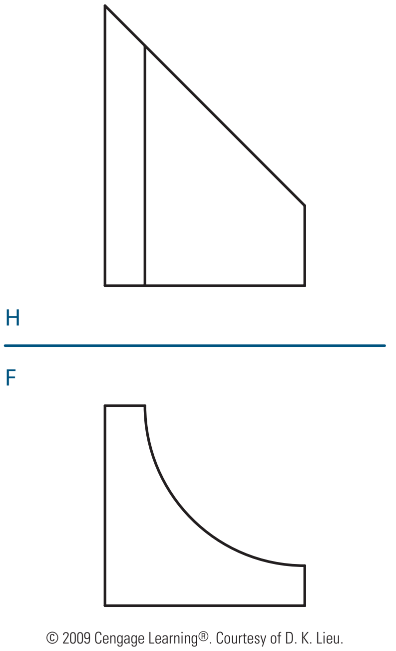

FIGURE 11.19. Top and front views of an object with an irregularly shaped surface for the auxiliary view.

the primary views. Figure 11.20 shows the same object except that the curved edge has been divided into four segments defined by five points. These points have been located and labeled A–E in the top and front views.

Make sure you label the fold lines appropriately according to the conventions described earlier, noting that the label for the auxiliary view will now be H/1 since it is constructed adjacent to the top, or horizontal, view. You now sketch the fold line parallel to the edge view of the inclined surface and project the points into the auxiliary view using perpendicular projection rays as you did before. The result is shown in Figure 11.21. When projecting the points, make sure you also project each of the points from the segments defining the irregular curved edge.

Locate each of the points in the auxiliary view by measuring the distance each point is from the fold line for the related view (in this case, the front view) and transferring those distances into the view you are creating. Figure 11.22a shows the distance transferred for point A, and Figure 11.22b shows the remainder of the points defining the curved edge transferred into the auxiliary view. Figure 11.22c shows the remaining points defining the surface transferred into the auxiliary view.

Finally, connect the dots to create a smooth curved edge and complete the auxiliary view that shows the surface in true shape and size. The completed true size view of the inclined surface is shown in Figure 11.23.

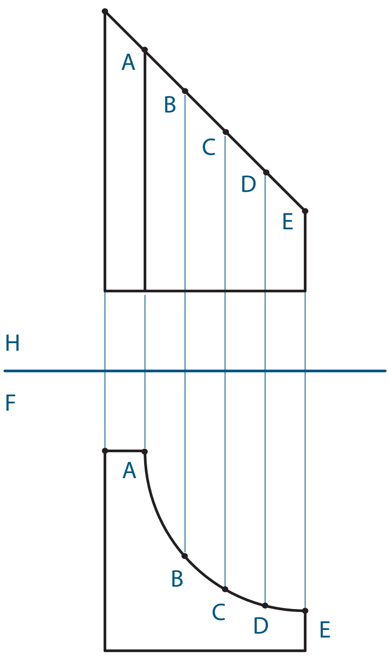

FIGURE 11.20. A curved edge with points A–E located.

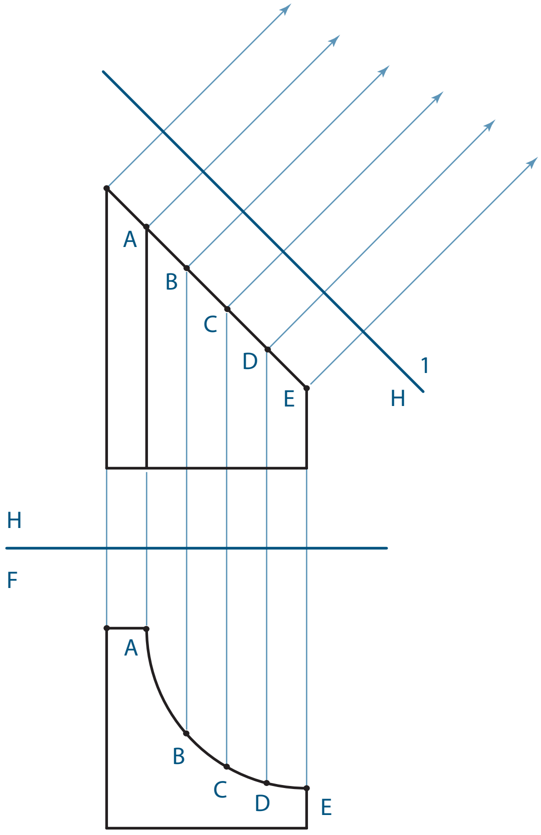

FIGURE 11.21. The fold line and projection rays added.

FIGURE 11.22. Points defining an irregular surface projected into the auxiliary view.

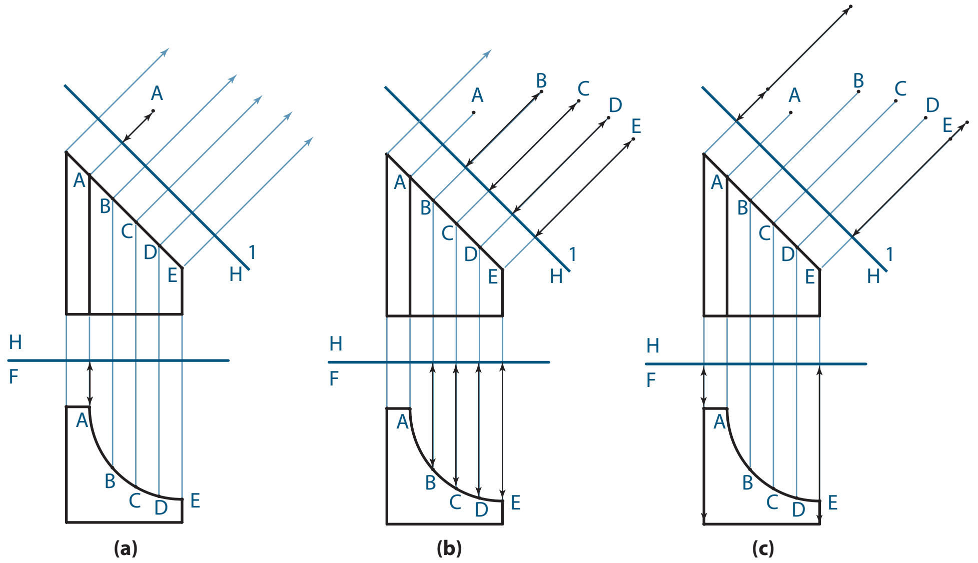

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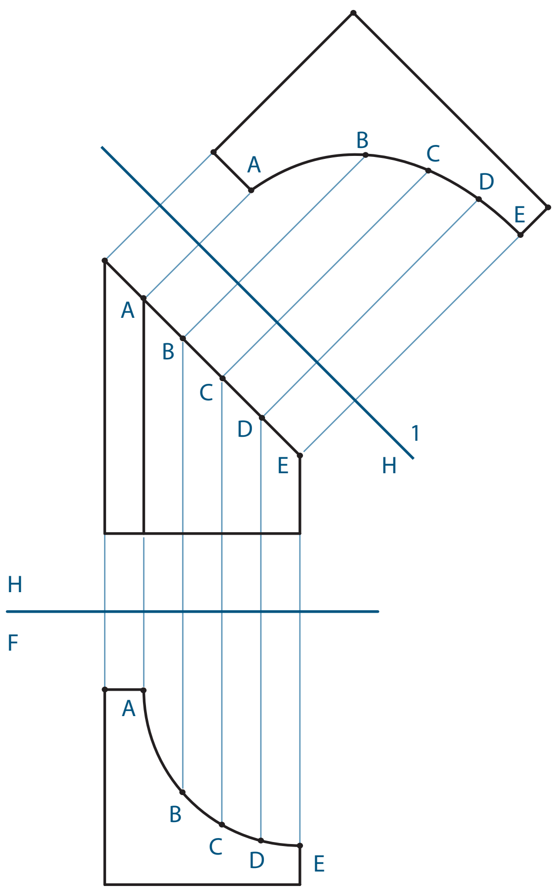
FIGURE 11.23. The completed auxiliary view of the irregular surface.



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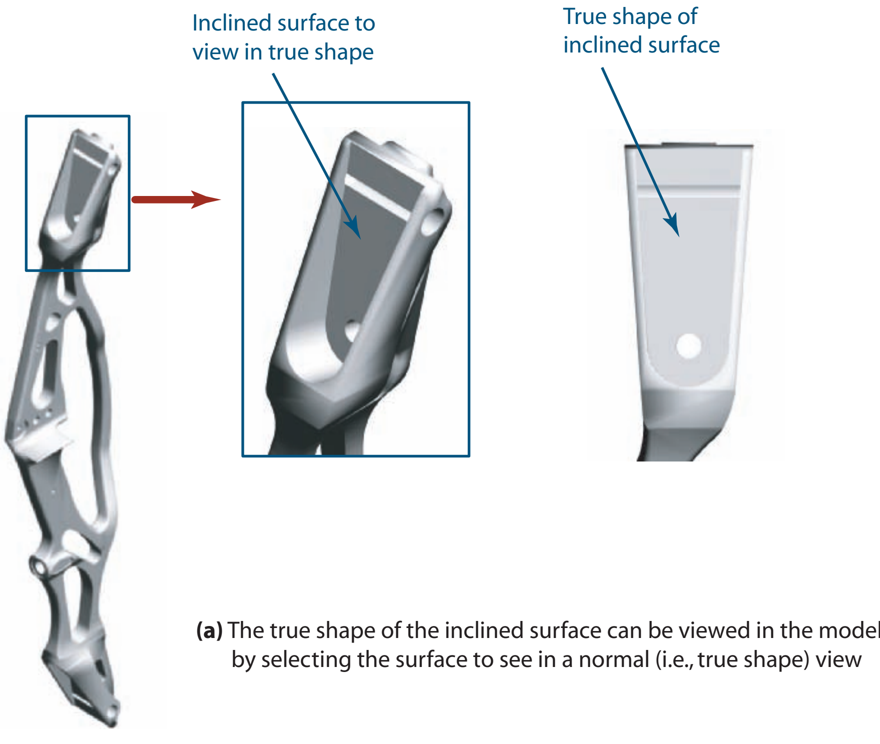
11.05 Solid Modeling Considerations in Creating Auxiliary Views

The procedure followed to create an auxiliary view by hand is sometimes tedious and often prone to errors. In the era of 3-D solid modeling, the need for auxiliary views may be somewhat diminished, and the difficulty in creating auxiliary views is greatly reduced. First of all, since the 3-D model is often sent directly to a CAM system for fabrication, it may not be necessary to create an auxiliary view to locate holes and other features accurately on an inclined surface. Because the 3-D model contains all of the necessary information for the creation of the part, an auxiliary view of an inclined surface might not provide any additional information and, therefore, may not be needed for manufacturing the part.

Second, when an auxiliary view from a 3-D solid model is needed, creating it is a relatively easy task. Recall that an auxiliary view is created by "looking" perpendicular to the inclined surface. With 3-D modeling software, you usually can select a plane on the object to define the viewing plane. The software will then rotate the object in space so that the selected plane is parallel to the computer screen, meaning the plane will appear in true size and shape in this view. With some software, you will be able to show only the plane; with other software, you will be forced to show the entire object from this viewpoint. However, since this auxiliary view is so simple to create, showing the entire object is a small price to pay. Figure 11.24 illustrates a 3-D solid model and an auxiliary view showing the inclined surface in true shape and size.

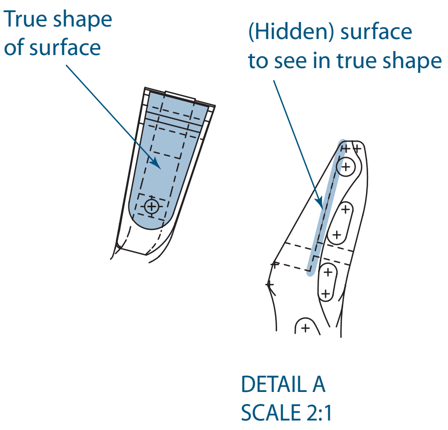
Also, note that the software can just as easily show the true size and shape of an oblique surface and that successive auxiliary views are not required in this application. Figure 11.25 shows a 3-D model of an object containing an oblique surface and the corresponding auxiliary view in true size and shape.

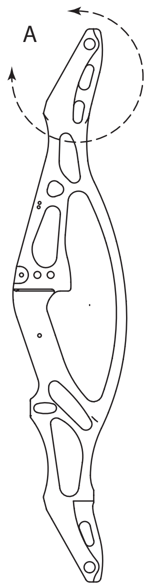
FIGURE 11.24. The creation of auxiliary views of the Aerotec bow handle from its solid model.



(a) The true shape of the inclined surface can be viewed in the model by selecting the surface to see in a normal (i.e., true shape) view

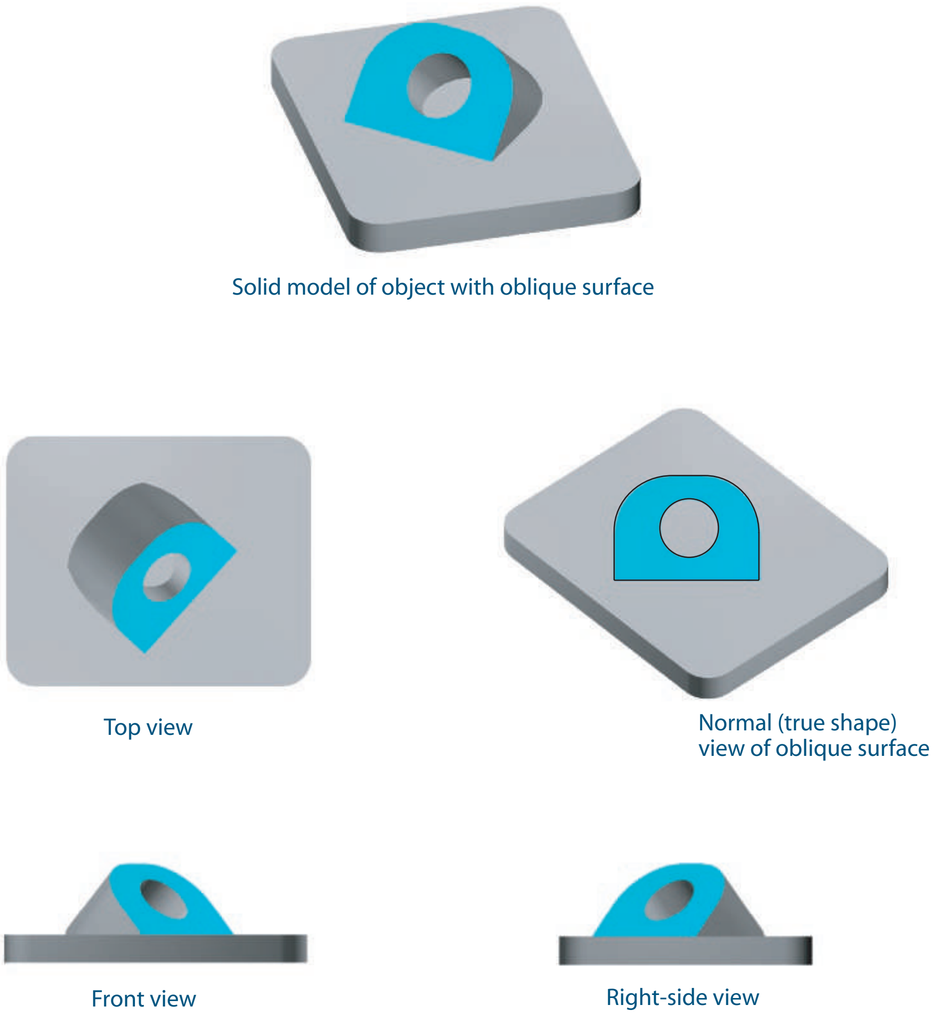
FIGURE 11.24. (CONTINUED) The creation of auxiliary views of the Aerotec bow handle from its solid model.





(b) The true shape of the inclined surface can be viewed in the drawing by selecting the surface to see in an auxilary view

FIGURE 11.25. An object with an oblique surface created by a solid model can easily be presented in any view orientation, including a normal (i.e., true shape) view of the oblique surface.



11.06 Chapter Summary

Auxiliary views allow you to set your viewpoint perpendicular to an inclined surface and project it in its true shape and size. When you are looking perpendicular to the surface, it is important to note that your viewing plane is parallel to the plane in question. Auxiliary views also are used to find information between two lines, a line and a plane, and two planes; however, these applications are left for your exploration in a supplementary chapter on descriptive geometry. The procedure you use to create an auxiliary view is based on the principles of orthographic projection described in detail in previous chapters of this text. In essence, when creating an auxiliary view, you are inserting a pane of glass into the imaginary glass cube that is parallel to the inclined surface. When the glass cube is unfolded, including the extra pane, the auxiliary view shows the inclined surface in true size and shape. In the age of 3-D solid modeling, the need for auxiliary views may be diminished; however, knowing the basics of creating this type of view is important for your understanding of graphic communication. Three-dimensional solid modeling software also enables you to easily create an auxiliary view of an inclined or an oblique surface using a few clicks of the mouse button.

11.07 GLOSSARY OF KEY TERMS

adjacent views: Views that are aligned side by side to share a common dimension.

auxiliary views: Views on any projection plane other than a primary or principal projection plane.

edge view (of a plane): A view in which the given plane appears as a straight line.

foreshortened (line or plane): Appearing shorter than its actual length or size in one of the primary views.

inclined surface: A plane that appears as an edge view in one primary view but is not parallel to any of the principal views.

oblique surface: A plane that does not appear as an edge view in any of the six principal planes.

projection ray: A line perpendicular to the projection plane. It transfers the 2-D shape from the object to an adjacent view. Projection rays are drawn lightly or are not shown at all on a finished drawing.

reference line: Edges of the glass box or the intersection of the perpendicular planes. The reference line is drawn only when needed to aid in constructing additional views. The reference line should be labeled in constructing auxiliary views to show its association between the planes it is representing; for example, H/F for the hinged line between the frontal and horizontal planes. A reference line is also referred to as a fold line or a hinged line.

related views: Views adjacent to the same view that share a common dimension that must be transferred in creating auxiliary views.

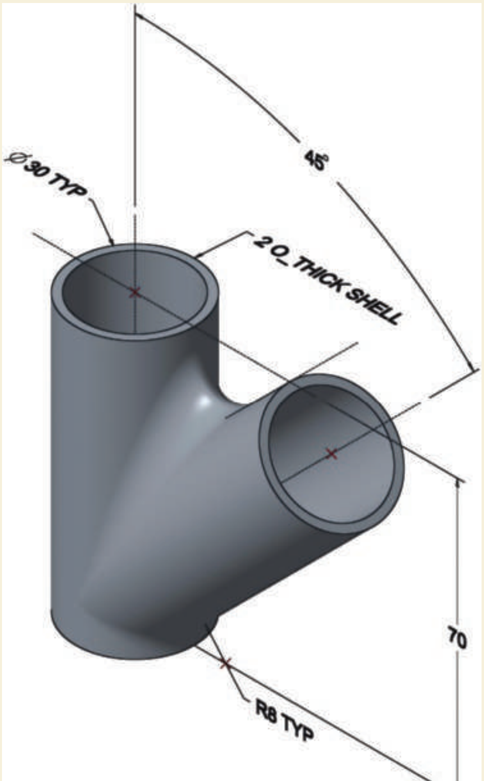
true shape (of a plane): The actual shape and size of a plane surface as seen in a view that is parallel to the surface in question.

11.08 QUESTIONS FOR REVIEW

1. What is a primary auxiliary view?
2. What is the purpose of an auxiliary view?
3. How is a full auxiliary view different from a partial auxiliary view?
4. List the five basic steps or procedure for drawing an auxiliary view.
5. Why might the creation of auxiliary views not be necessary with 3-D solid modeling software?
6. How does 3-D solid modeling software make the job of auxiliary view creation easier and less prone to errors?

11.09 PROBLEMS

1. For each of the pictorials shown in Figure P11.1, create the top, front, and right-side views. Then, create an auxiliary view to present the true shape of the inclined surface.



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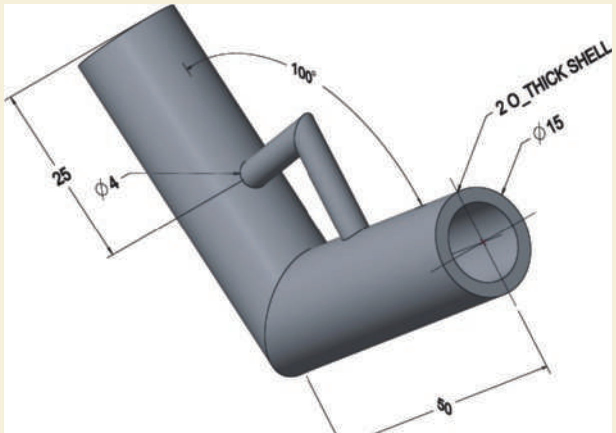

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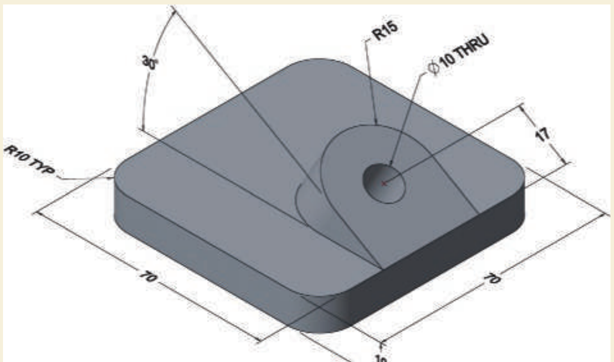

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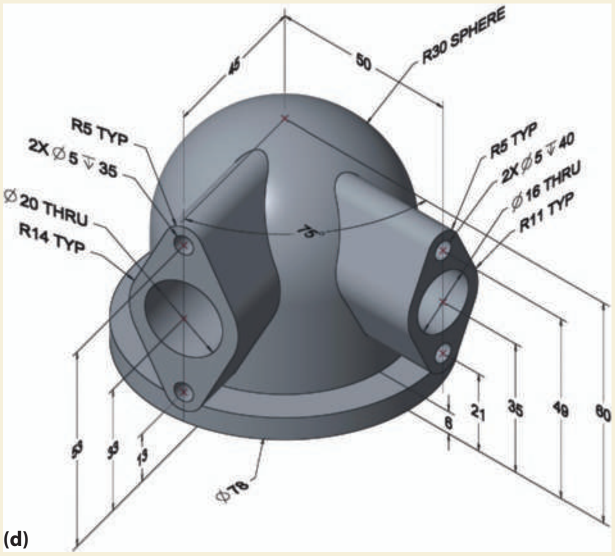

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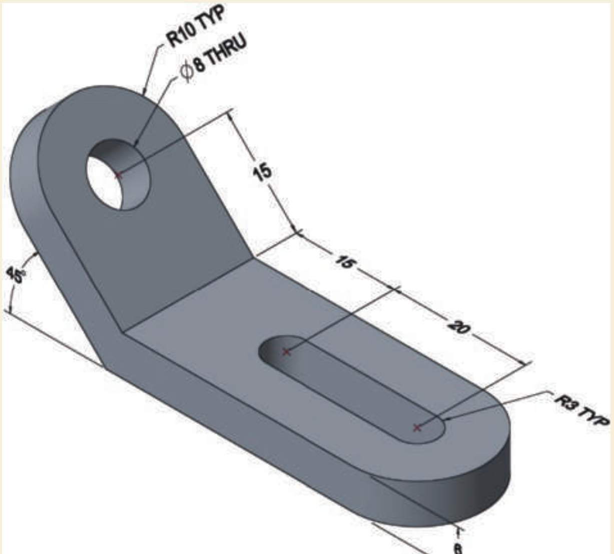
11.09 PROBLEMS (CONTINUED)



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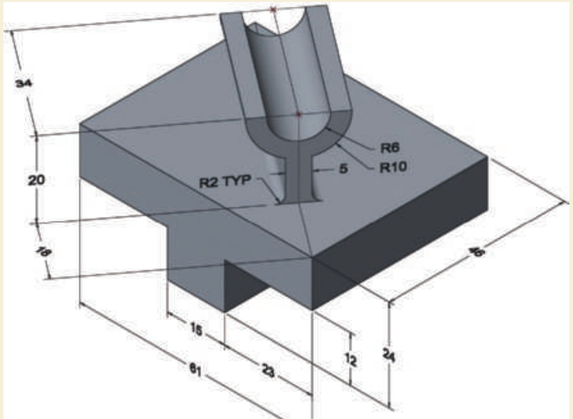

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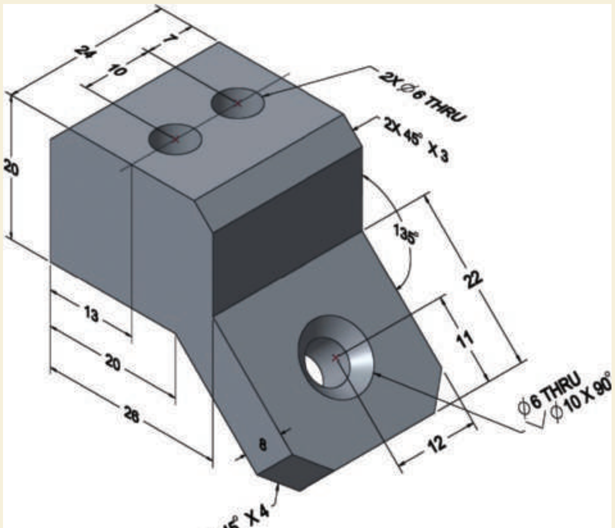

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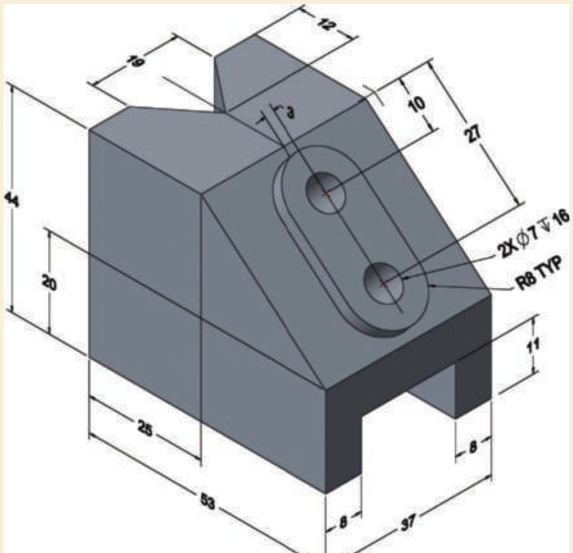

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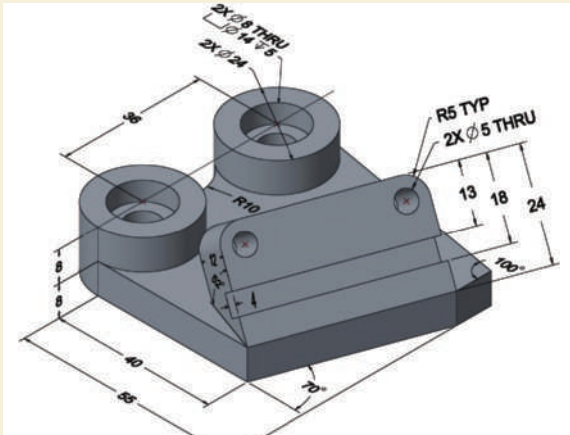

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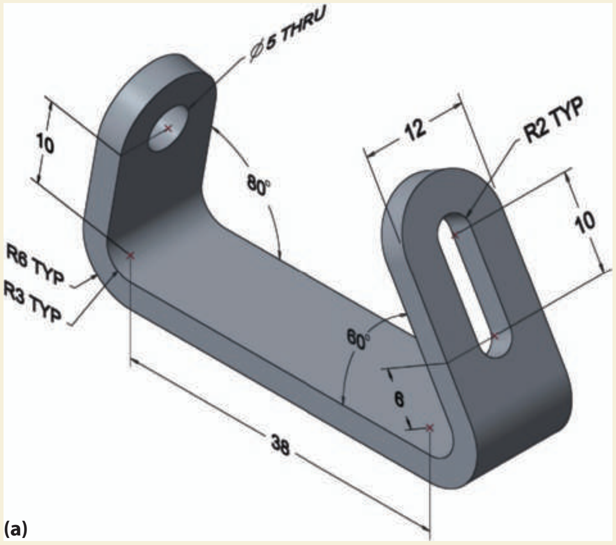
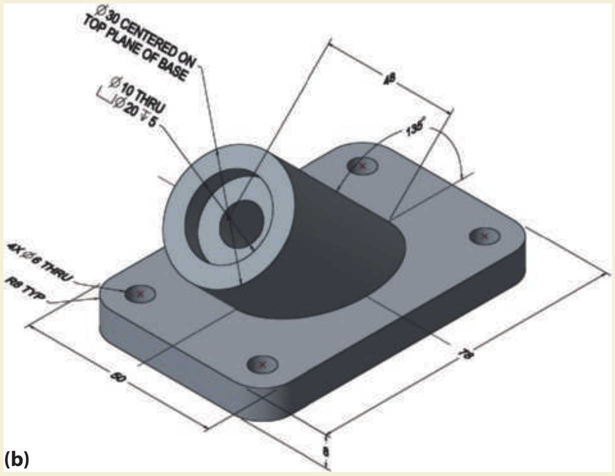
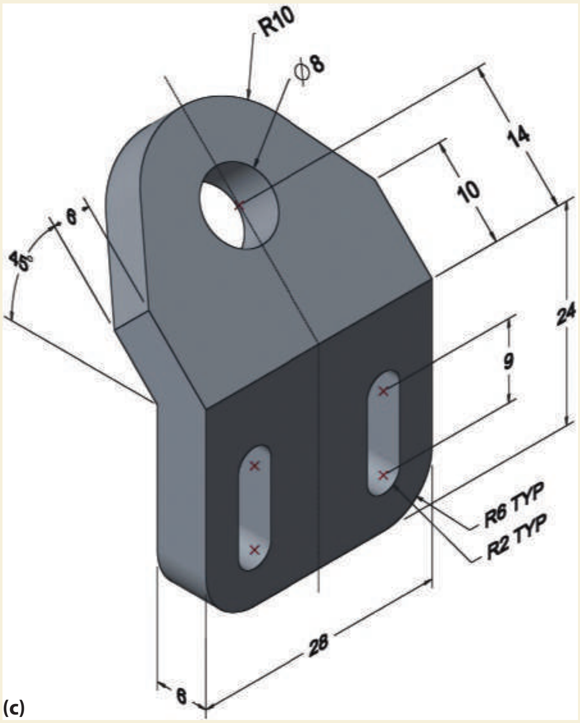
FIGURE P11.1.

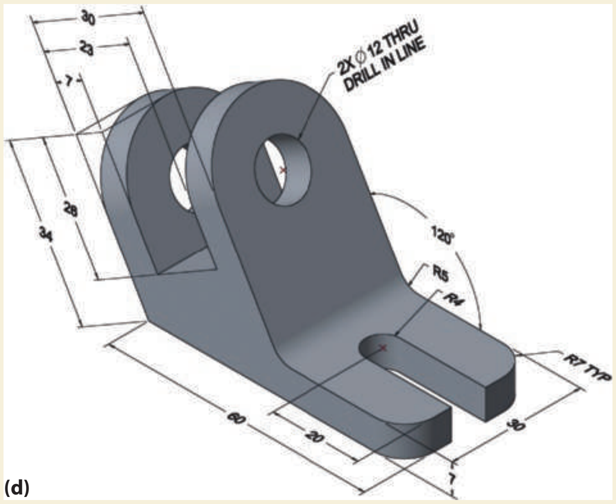
11.09 PROBLEMS (CONTINUED)

2. For each of the pictorials shown in Figure P11.2, create the top, front, and right-side views. Then, create an auxiliary view to present the entire object with the inclined surface shown in its true shape.









11.09 PROBLEMS (CONTINUED)

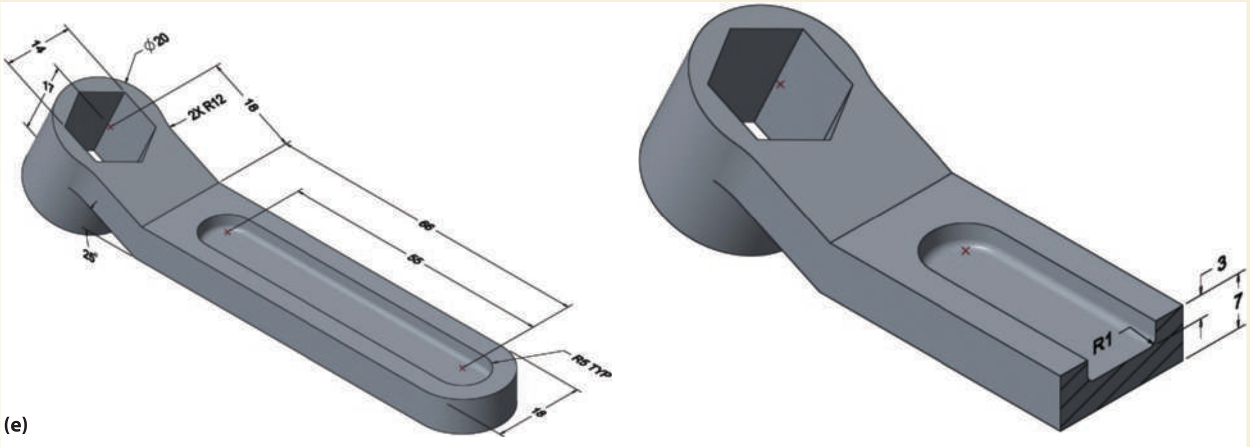


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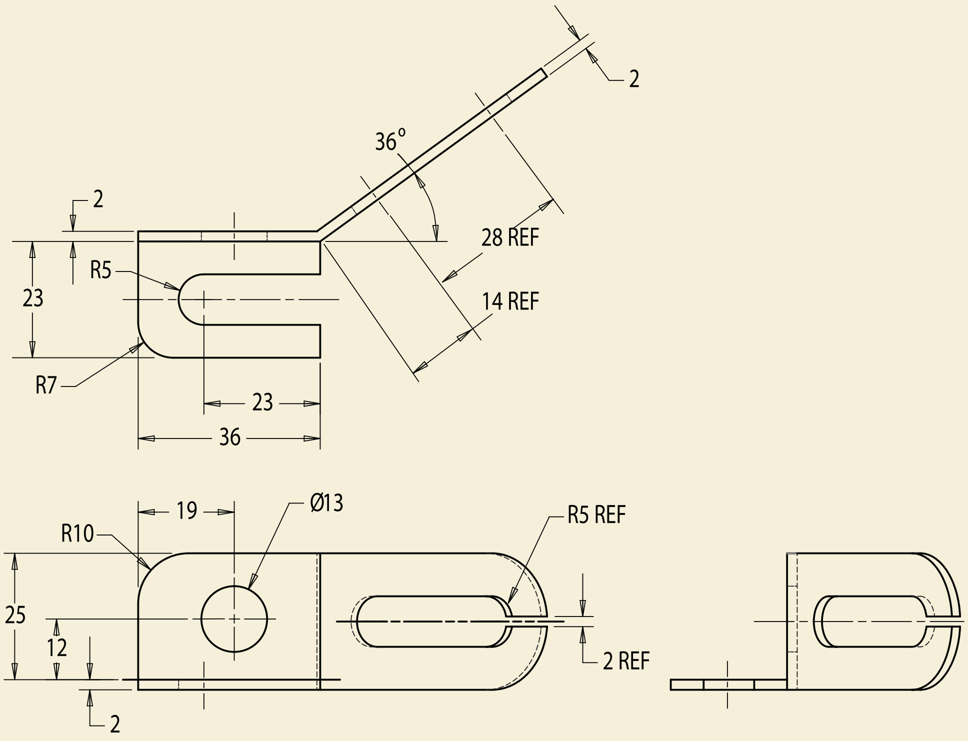
FIGURE P11.2.

3. For each of the multiview drawings shown in Figure P11.3, create an auxiliary view to present the inclined surface shown in its true shape. The dimensions marked REF should be moved to the auxiliary view.



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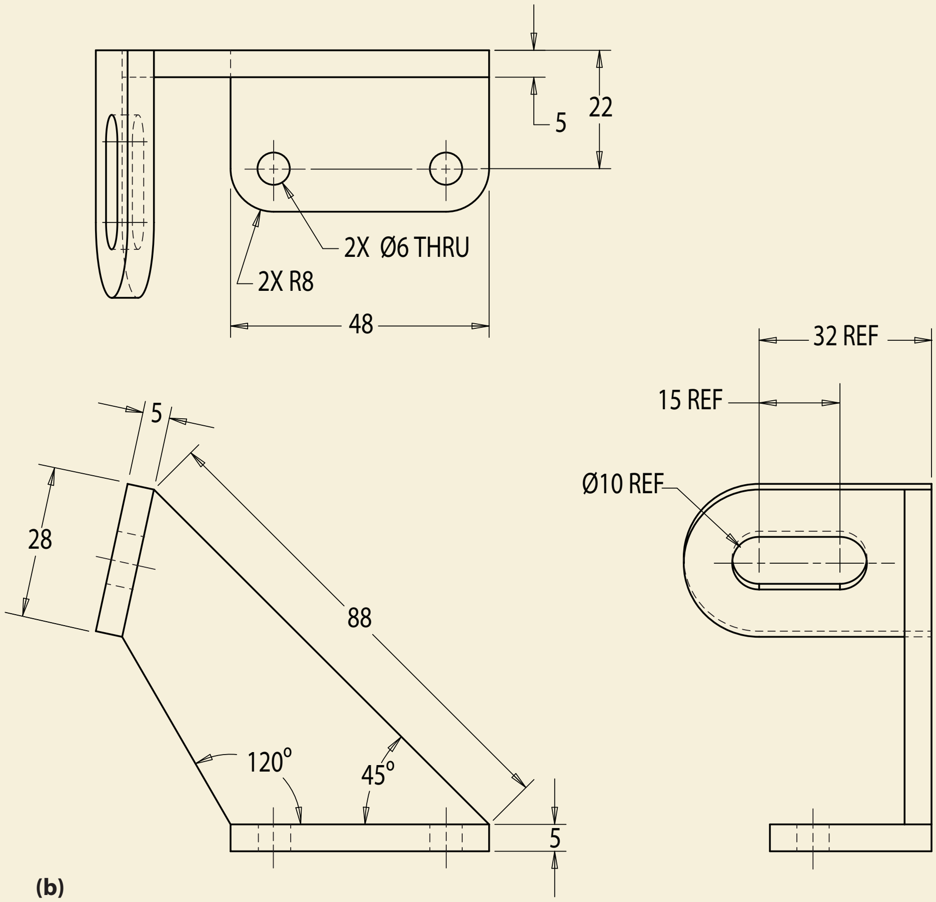
11.09 PROBLEMS (CONTINUED)



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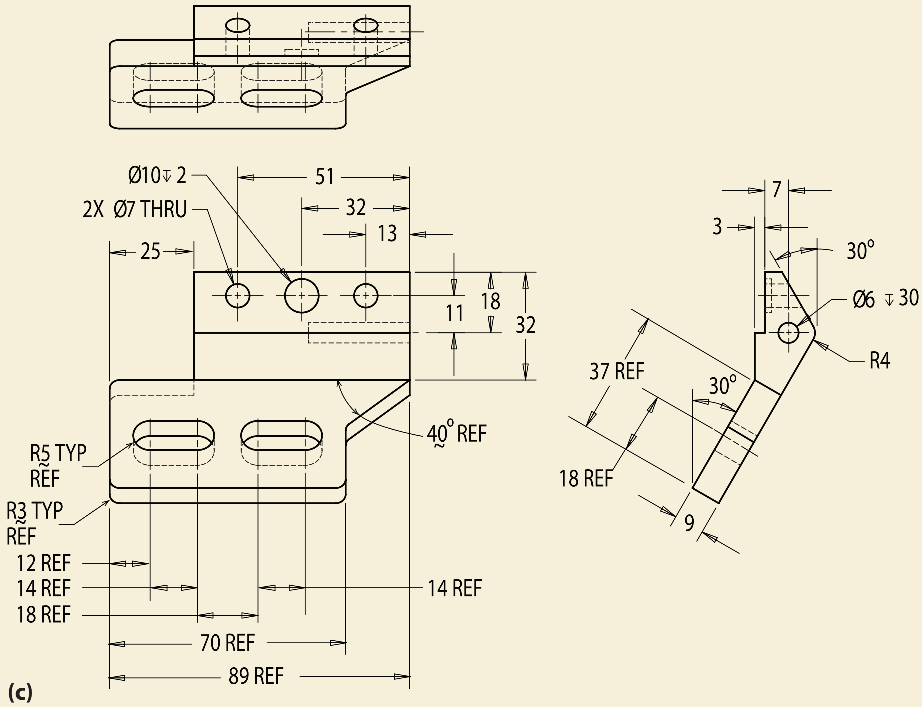

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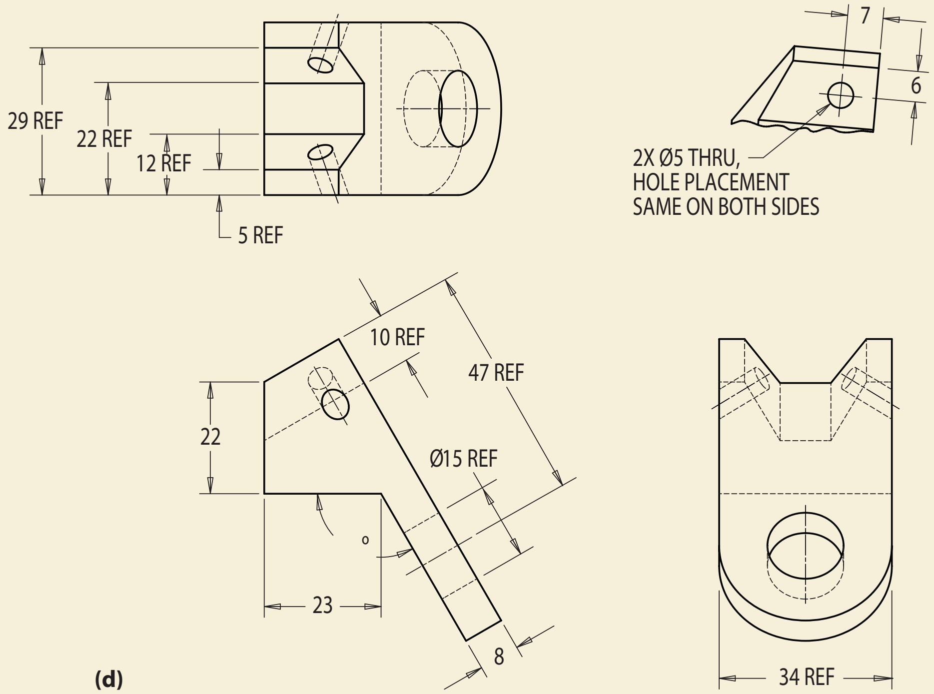
11.09 PROBLEMS (CONTINUED)



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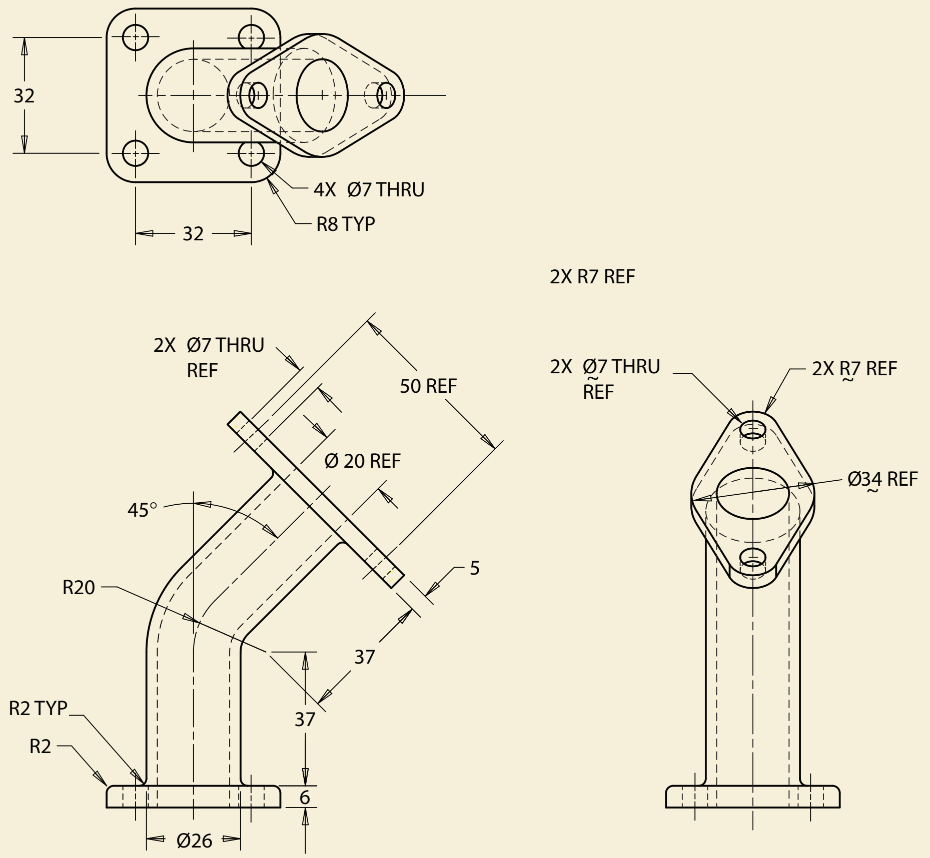

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FIGURE P11.3.

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SECTION FOUR

DRAWING ANNOTATION AND DESIGN IMPLEMENTATION

The majority of engineered parts require the specification of measurements, sizes, and allowable errors of features on the parts. Engineers still need to be able to specify part sizes so that everything fits together and functions as intended. This specification must be completed before the parts can be fabricated. Procedures for size specification must be followed to ensure that these specifications can be easily interpreted, checked, and controlled for proper function of the parts. When an engineer is presented with a formal engineering drawing, whether it is a mechanical device or a construction project, that engineer must be able to read all of its contents correctly. Drawings are legal documents and, as such, are required to contain certain information to ensure that the creators and the receivers interpret them properly. Guidelines must be followed to ensure that completed drawings are created, updated, and approved in a manner that establishes a line of accountability.

CHAPTER 12

DIMENSIONING

OBJECTIVES

After completing this chapter, you should be able to

- Use the concept of dimensioning
- Explain the idea of tolerance in dimensioning
- Recall the fundamental rules and apply the techniques for dimensioning
- Select appropriate dimensions for a moderately complex part and correctly apply them to a drawing of that part

12.01 INTRODUCTION

In the previous chapters, you learned how to represent the shape of objects in various ways. You learned about standard ways of representing objects with orthogonal projection techniques, pictorials, sectional views, and auxiliary views. You learned about different techniques for creating solid models and the way parametric, feature-based modeling is used to create 3-D representations of objects. All of this information is great for representing the shape of objects; but at some point, you will want to communicate size information to someone who will construct or manufacture your design. Until your designs are built, you will not be making any money. As with orthogonal projection, there are standard ways of displaying this size information, or dimensions, on drawings. In this chapter, you will examine some of these standards as well as look at some reasons for dimensioning objects in certain ways.

To begin, it is critical that you understand something about how dimensions are formally presented in a design. As you learned in your work with 3-D solid modeling, objects are a combination of features such as rectangular prisms, cylinders, holes, fillets, and chamfers. Recall from solid modeling that most of these features require that they be defined by their sizes and their locations. For example, the hole in the object shown in Figure 12.01 is considered a feature. The **size** (.500 diameter) must be given so the person manufacturing the part can select the correct drill bit or cutting tool to machine the hole to the proper size. For engineering drawings, diameters (such as the .500 for this hole) are preceded with the Ø symbol. Location dimensions (1.250 from the right and .750 from the top) are given from the sides of the part to the center of the hole so the machinist can accurately locate the center of the drill bit on that point.

Similarly, the slot in the top of the object shown in Figure 12.02 must also be defined by its size and location. The size of the slot feature is defined by its width (.500) and its height (.250). A **location** dimension is given from the left side (1.000).

Dimensioning is much like creating constraint-based solid models—you define the size and location of the features within the software, and the part is created "virtually" to your size and location specifications. Figure 12.03 illustrates how the size and location dimensions for a rectangle are used to define an extruded cut in a constraint-based solid modeling program for the object shown in Figure 12.02.

One of the key points you should learn from this chapter is a strategy for determining the types of dimensions required to define a part. You already have a head start through your experiences with 3-D solid modeling software. Following the correct standards for representing dimensions on a drawing is important, but being able to apply the best dimensions to a drawing will impress your boss more than your knowing the standards and applying bad dimensions. For example, imagine you work for a company that manufactures hardware for household doors. Your current project involves the deadbolt lock assembly shown in Figure 12.04. Your boss asks you to design a cover plate where the door meets the doorjamb. You are familiar with the standard ways to represent the shape of the part. One or two views will be enough to describe the shape of the plate, but what are the dimensions needed to manufacture the plate? What are the critical dimensions that must be given? What are some of the standard dimensions that exist on other parts or previous parts? Think about the assembly and these questions because you will return to this example later in the chapter.

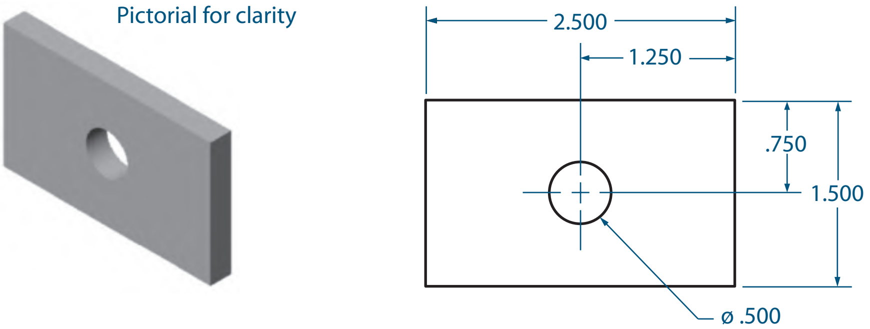

FIGURE 12.01. The size and location of a hole feature.

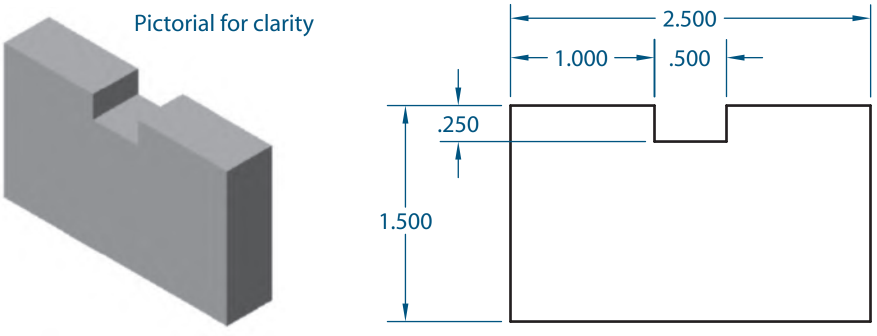

FIGURE 12.02. The size and location of a slot feature.

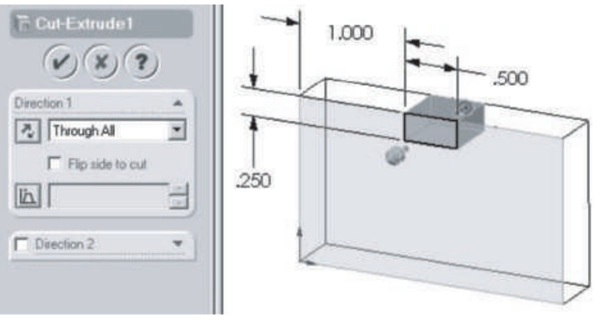

FIGURE 12.03. Defining features in a constraint-based modeling program.

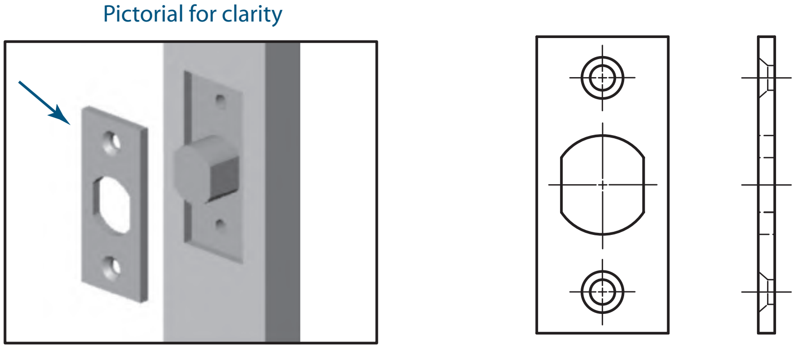

FIGURE 12.04. A deadbolt lock plate.

12.02 Is the Dimension I See on a Drawing Exact?

People are not perfect. When they fabricate metal objects, mold plastic parts, and build houses, some room must be made to account for their imperfection. Even when robots are used to machine parts, there may be some slight imperfection in the resulting object. For machined parts, the amount of variation, or **tolerance**, might be relatively small. There are various ways of including allowable tolerances for a part on a drawing. An example of a note appearing on an engineering drawing might be this:

ALL LINEAR DIMENSIONS ± .010 UNLESS OTHERWISE SPECIFIED.

In the construction industry, tolerances for laying brick or pouring a concrete foundation are typically larger than tolerances on hand-held sized machined parts.

Tolerance dimensions also help with effective size control of finished parts. Examine the assembly of parts in Figure 12.05. For this design, the BUSHING is not supposed to spin inside the WHEEL, but the SHAFT is designed to spin inside the BUSHING. Therefore, the largest diameter of the SHAFT must be just a little smaller than the diameter of the hole in the BUSHING, and the outside diameter of the BUSHING must be just a little larger than the hole in the WHEEL. Dimensioning the hole in the BUSHING and the diameter of the SHAFT as .750 would not communicate the intended type of fit between the two parts. The person putting the parts together would not know whether you wanted the parts to spin freely or be jammed together.

Detail drawings of the BUSHING and the SHAFT are shown in Figures 12.06 and 12.07, respectively. Notice that tolerance dimensions (specifically, limit dimensions) are given to ensure effective size control between the parts. The hole in the BUSHING is dimensioned as .7500–.7512, and the diameter of the SHAFT is given as .7484–.7492. If the parts are manufactured within these specifications, the SHAFT will spin freely within the BUSHING.

FIGURE 12.05. Parts requiring effective size control.

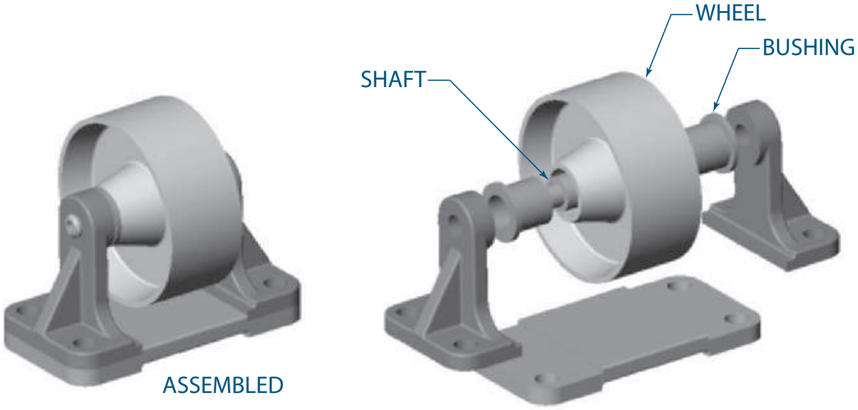

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FIGURE 12.06. The detail drawing of the BUSHING.

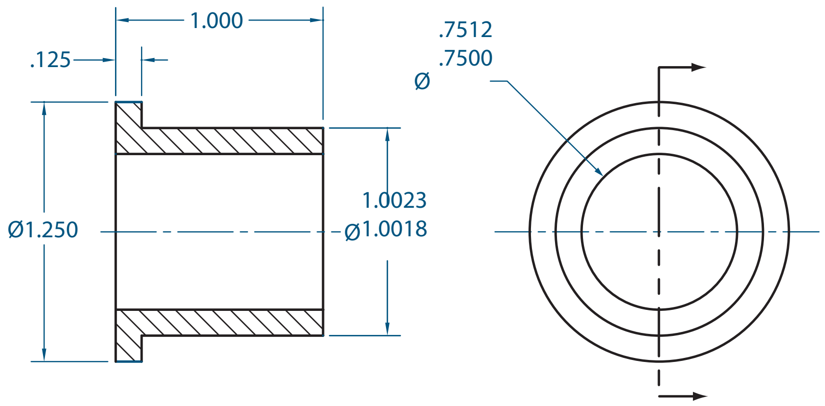

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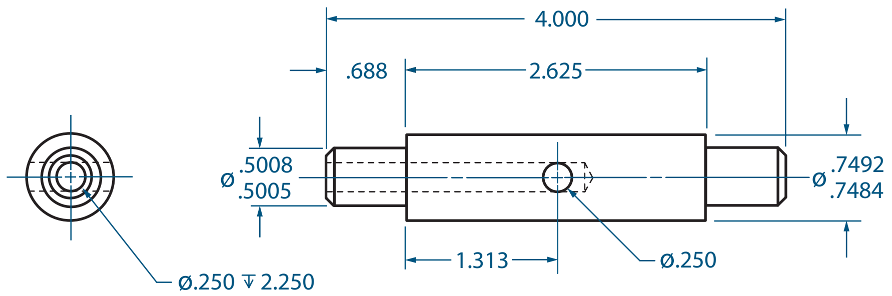
FIGURE 12.07. The detail drawing of the SHAFT.



12.03 What Are the Rules for Dimensioning?

As with most topics in engineering graphics, dimensions conform to national, international, and individual company standards. The accepted national standard in the United States for Dimensioning and Tolerancing is **ANSI Y14.5** (currently referenced as *ASME Y14.5M-1994*), which is published by the American Society of Mechanical Engineers (ASME). This standard outlines uniform practices for displaying and interpreting dimensions and related information on drawings and other forms of engineering documentation. The information in *ASME Y14.5M-1994* is important, but do not be too concerned with it right now. Remember, keep trying to figure out what the critical dimensions are, and you will worry about standard dimensioning technique later.

12.03.01 Millimeters, Inches, or Angstroms?

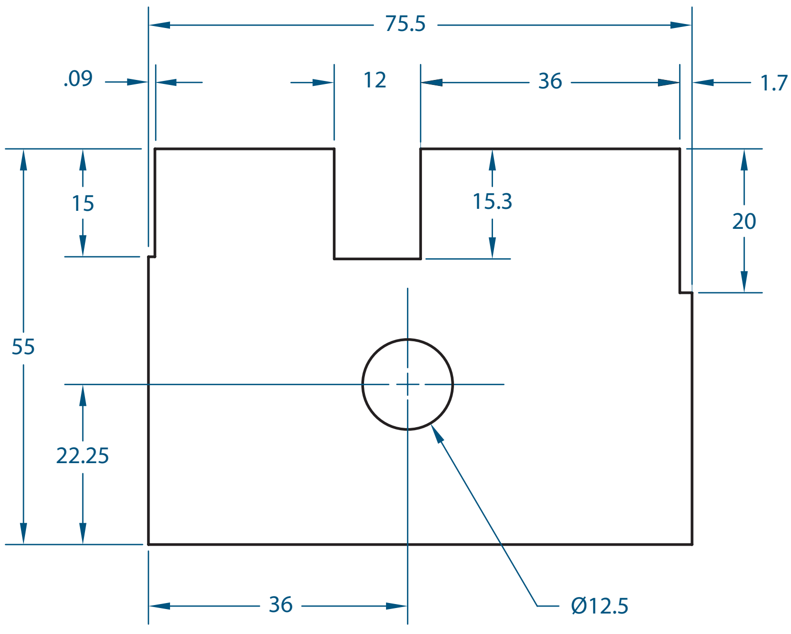
"The 200 meter dash." "First down and 10 yards to go." "Hand me that 2 × 4." These are all examples of length measurements that are familiar to most people. For most track and field events, lengths are defined in meters. In baseball and football, lengths are measured in feet and yards. In the construction industry, decimal or fractional inch measurements are the standard way lengths are defined. Engineering drawings also have standard units of measure. Most drawings conform to the International System of Units (SI), which is metric and uses the millimeter as the standard unit; or they conform to U.S. customary units with a standard unit of the decimal inch. Throughout this chapter, you will see examples using both millimeter and inch dimensions. The next section will discuss how to recognize the differences between the two. Since both standards are used throughout the United States, it is important that you be able to work with each type. You should be familiar with both standards by the end of the chapter.

12.03.02 Types of Dimensioning

At this point in your class, you may have noticed that your instructor or professor is fairly picky about the way things look on sketches or drawings, mostly because, as was mentioned earlier, engineering drawings do follow standards. Well, here is the first really picky thing about dimensioning that will help you recognize the differences between metric- and inch-based drawings. For metric drawings where millimeters are the standard unit (see Figure 12.08), the following rules apply (*ASME Y14.5M-1994*, p. 5):

1. Where the dimension is less than one millimeter, a zero precedes the decimal point.
2. Where the dimension is a whole number, neither the decimal point nor the zero is shown.
3. Where the dimension exceeds a whole number by a decimal fraction of one millimeter, the last digit to the right of the decimal point is not followed by a zero.
4. Neither commas nor spaces shall be used to separate digits into groups in specifying millimeter dimensions on drawings (e.g., 1000 not 1,000).

FIGURE 12.08. Millimeter dimensions.



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To help distinguish between the two systems, the following rules have been established for decimal inches (*ASME Y14.5M-1994*, pp. 5–6) (see Figure 12.09):

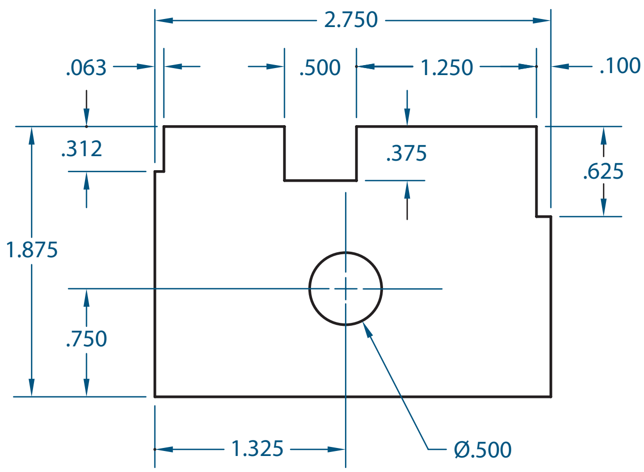
1. A zero is not used before the decimal point for values less than one inch.
2. A dimension is expressed to the same number of decimal places as its tolerance. Zeros are added to the right of the decimal point where necessary.

What does this mean? When dimensioning in millimeters, show leading zeros for values less than 1, but do not show trailing zeros. When using inches, do not show leading zeros for values less than 1, but do show trailing zeros equal to the precision on the drawing.

12.03.03 Fundamental Rules for Dimensioning

As you can imagine, making sure that a drawing created by a designer in Raleigh, North Carolina, can be read by a manufacturer in Detroit, Michigan, or Taipei, Taiwan, requires that some standards be established. The main reason for having standards is

FIGURE 12.09. Dimensioning in inches.



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to ensure consistency in the way things are done. Over the last 100 years, individuals in the automotive, aircraft, and military industries and in other industries have refined the standards for dimensioning objects. As mentioned previously, fundamental rules and standards for dimensioning and tolerancing are published in *ASME Y14.5M*. These rules define engineering and design intent clearly. Some of the rules are listed here. (A complete list of the fundamental rules is given at the end of the chapter.)

1. *Each dimension shall have a tolerance*. As was mentioned earlier in the chapter, tolerance dimensioning is necessary to account for human imperfection and to allow for effective size control. If a dimension does not appear as a limit dimension, the tolerance is usually covered by a general note on the drawing or in the title block.
2. *Dimensioning and tolerancing shall be complete so there is full understanding of the characteristic of each feature*. Drawings need to be dimensioned so the manufacturer or construction worker does not have to guess at anything. It is your responsibility to provide all necessary information to produce, manufacture, or build the design.
3. *Each necessary dimension of an end product shall be shown. No more dimensions than those necessary for complete definition shall be given*. As you will see later in the chapter, you do not want to give more dimensions than necessary to describe your design. Show only the dimensions that the person producing the design will need. Taking rule 3 together with rule 2 means you need "just enough" dimensions to define the part—not too many and not too few.
4. *The drawing should define a part without specifying manufacturing methods*. Do not specify that a hole is to be drilled, reamed, punched, or made by any other operation. The person manufacturing your design is responsible for determining the best method for producing the hole.
5. *Dimensions should be arranged to provide required information for optimum readability. Dimensions should be shown in true profile views and refer to visible outlines*. Show the size and location of a hole in the view where the hole shows up as a circle. When the hole is created in the part, it will be located and drilled from that same view. This ensures consistency between the design and the manufacturing of the part. Also, do not dimension hidden features on a part. Find a view where the feature is visible, and dimension in that view.

12.04 Definitions

The following terms are used in the remainder of this chapter. Studying them now will help you better understand the dimensioning concepts that follow. (*Definitions are from *ASME Y14.5M-1994*.)

- **Dimension**—*A numerical value expressed in appropriate units of measure and used to define the size, location, geometric characteristic, or surface texture of a part or part feature.
- **Arrowhead**—A small triangle at the end of dimension lines and leaders to indicate the direction and extent of a dimension (see Figure 12.10).
- **Dimension Line**—A thin, dark, solid line that terminates at each end with arrowheads. The value of a dimension typically is shown in the center of the dimension line. *A dimension line, along with its arrowheads, shows the direction and extent of a dimension (see Figure 12.10).
- **Extension Line**—A thin, dark, solid line extending from a point on an object, perpendicular to a dimension line. *Extension lines are used to indicate the extension of a surface or point to a location preferably outside the part outline (see Figure 12.10). There should be a visible gap between extension lines and visible lines so the person

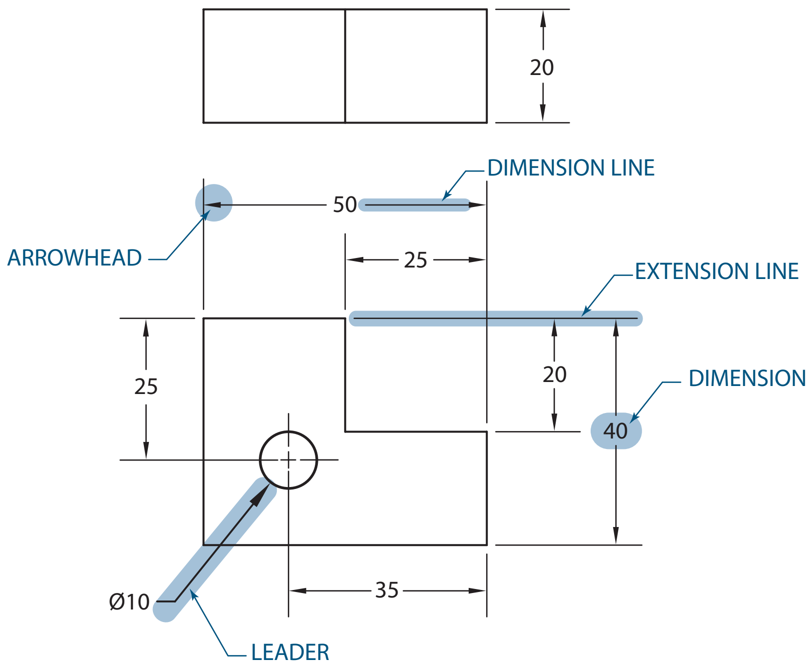

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FIGURE 12.10. Dimensioning terminology.

reading the drawing can distinguish between the part and the dimensions describing the part.

- **Leader**—A thin, dark, solid line terminating with an arrowhead at one end and a dimension, note, or symbol at the other end. *Leaders are used to direct a dimension, note, or symbol to the intended place on a drawing.

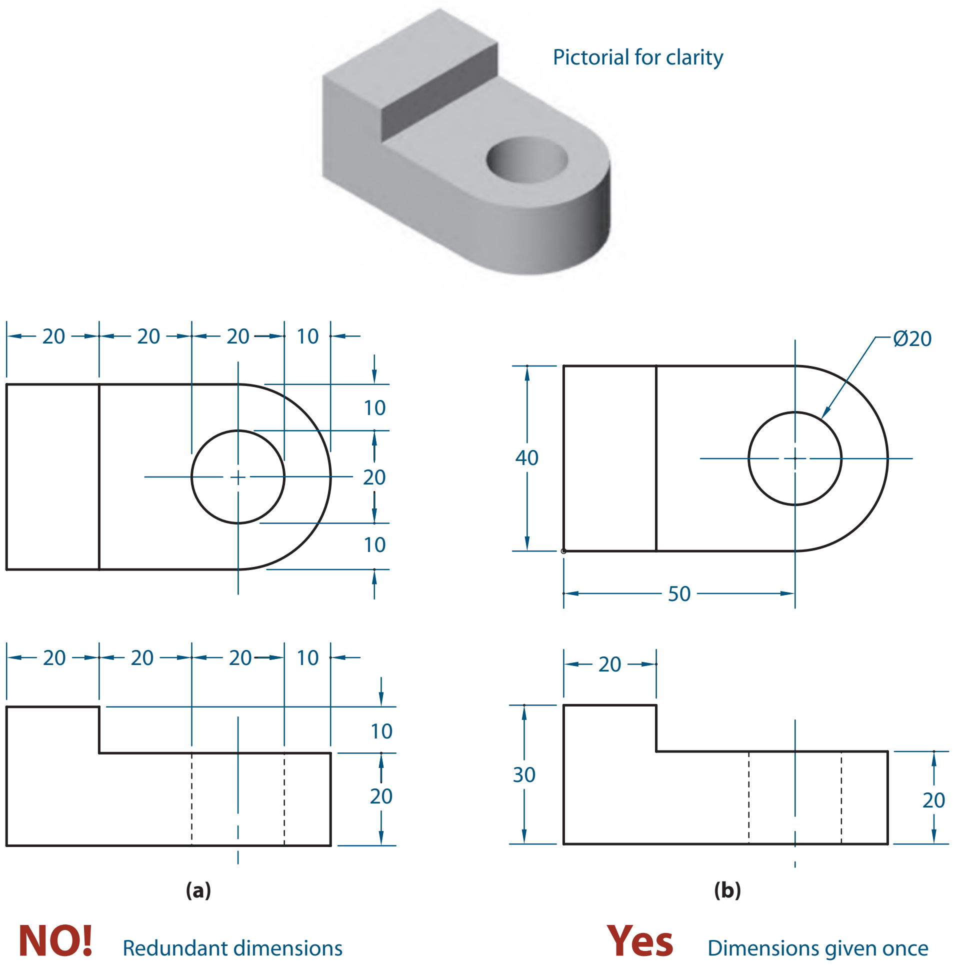
12.05 Redundancy Is Dumb

As you learn more about dimensioning parts, you will discover that clarity is very important and that a certain amount of economy goes a long way. The machinist is not going to be too happy if you dimension every point in every view on a drawing. He or she is expecting to see only the dimensions that are necessary to manufacture the part. Dimensions should appear only once on a drawing. In addition, each dimension should be placed in the view where the contour shape is best shown. This is known as the **contour rule** or **contour dimensioning**. Examine the part and dimensions shown in Figure 12.11. In (a), too many dimensions are given. It is not necessary to give dimensions to every point in each view. Notice the dimensions in (b). Each dimension is shown only once in the view where the contour or shape for that particular dimension shows up the best. For example, the hole shows up the best in the top view; therefore, it is best to show the size and location of the hole in that view rather than in the front view.

Another example of redundancy that should be avoided is shown in Figure 12.12. This is a very simple example, but notice that one of the horizontal dimensions can be omitted since 20 + 15 + 20 = 55. The same is true for the vertical dimensions since 10 + 20 = 30. The task here is to determine which dimensions are needed most and include just those.

To help you determine which dimensions are most critical, imagine a similar part in a couple of situations. In Figure 12.13a, notice that the spacer must fit correctly with respect to a couple of different features within the larger part. The tab in the larger part fits into the slot, and the left side of the spacer fits against the right side of the larger part. For the drawing in (b), there is no need to include the dimension of 23 on the right of the part since it is not really critical. The overall dimension is more

FIGURE 12.11. Redundant dimensions in (a) are poor practice. Dimensions in (b) are shown once in the view best suited for viewing.



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important in this context. Overall dimensions help to define outer boundaries of parts quickly. Figure 12.13c is not appropriate in this situation. The critical dimension of 20 is omitted, and the noncritical dimension of 23 is included. It is being left to chance that the feature whose size is 20 will be correct based on the other three dimensions being manufactured to their exact correct sizes.

FIGURE 12.12. Redundant dimensions.

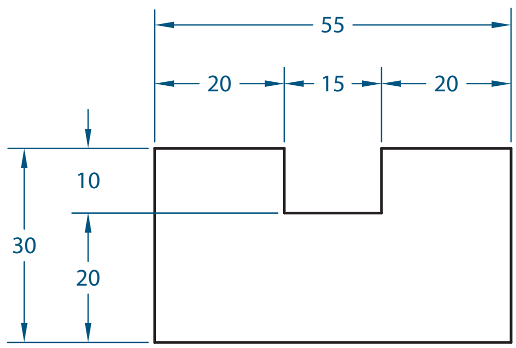

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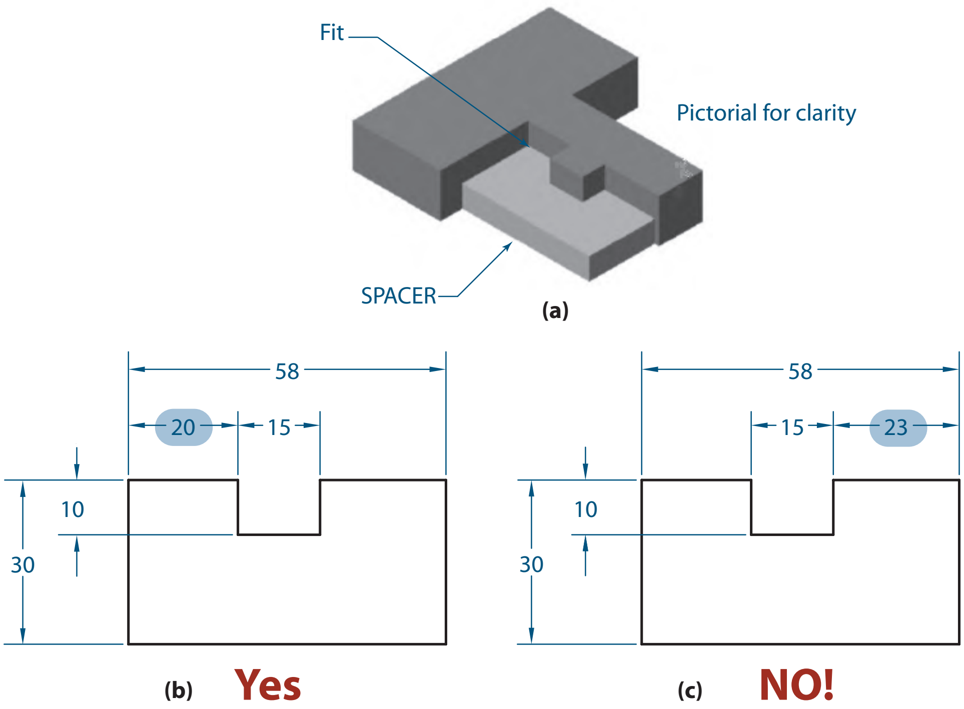
FIGURE 12.13. Dimensions applied, considering the fit and function of the part named SPACER.



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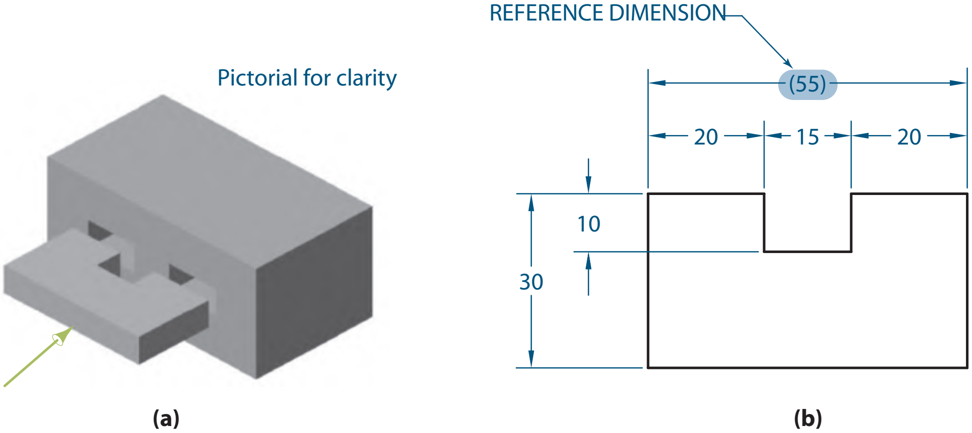
When the "inside" dimensions are more important or more critical than the overall dimension, the overall dimension should be identified as a reference dimension. In Figure 12.14a, the two tabs on the spacer fit into two holes in the larger part. In this case, the sizes of the tabs *and* the space between them are critical for the parts to fit together. In this case, the overall dimension is given as a reference dimension so the person making the part does not have to add the three dimensions to figure out the overall size. Reference dimensions, like the overall dimension in (b), are identified by enclosing them in parentheses. Here the person inspecting the parts can use the overall dimension as a quick check.

FIGURE 12.14. Reference dimensions.



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12.06 Geometrically Correct, but Still Wrong!

Why does it matter what dimensions are given if all of the geometry is defined? It may not matter if you are converting 3-D CAD data *directly* to produce molds for plastic parts, STL (stereolithography) file data for rapid prototyping, or tool paths for CNC machining. However, when drawings are being used to document parts for manufacture, accepted rules and practices must be followed to ensure acceptable results.

12.06.01 Different Ways of Specifying the Same Geometry

One of the first things to recognize when dimensioning objects is that there are standards for specifying particular types of geometry. For example, circles are typically dimensioned as diameters (Ø), and arcs are dimensioned as radii (see Figure 12.15). Circles are dimensioned with diameters since they typically represent machined holes, and machined holes are produced with standard tools that are defined by diameter dimensions. If the part in Figure 12.15 is a gasket, the three holes must line up with three holes on mating parts. Therefore, the 35 and the 70 dimensions are important dimensions to include since they identify the centers of the holes. Although the 94 dimension might be of interest to someone knowing the overall width of the object, it is not critical for defining the geometry. For the part shown in (a), the overall width can be determined by adding the 70 to the radii on the ends if necessary.

12.06.02 Identifying and Specifying the Critical Dimensions for Part Function

As you get more experience in engineering and design, one skill you will acquire is the ability to identify the critical dimensions on parts. In fact, by carefully planning the way a part is dimensioned, you may be able to eliminate potential errors in assembly. Examine the SPACER shown in Figure 12.16. The purpose of this part is to make sure the vise assembly stays together when the vise is opened to its maximum width. What are the critical dimensions on the SPACER? Are the overall height and width dimensions critical? To some extent yes, but the most important dimensions are the size of the machined holes and the distance between the two holes. The size of the holes is critical because the cylindrical bars must fit correctly in the holes. The location dimension between the holes is important because it ensures that both bars line up with both holes. For this example, you are going to concentrate on dimensioning the location of the holes.

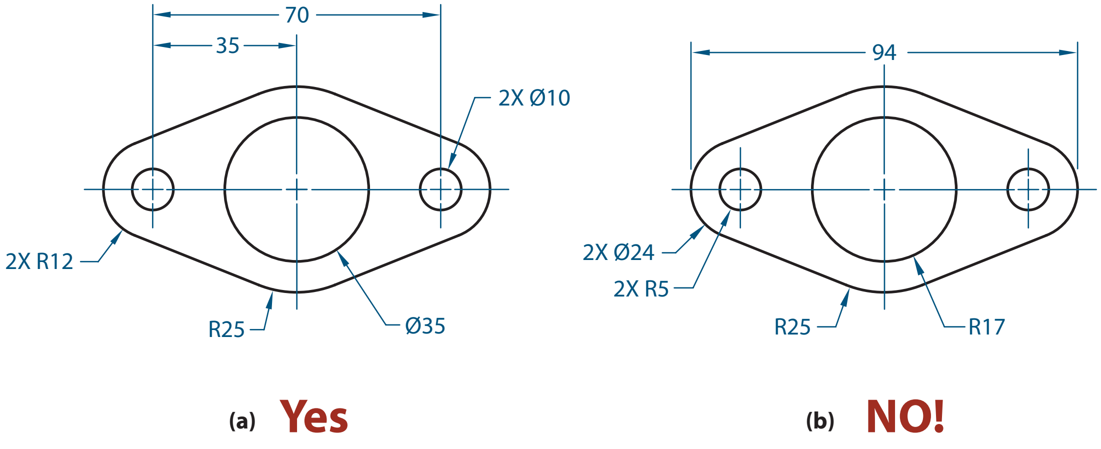

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FIGURE 12.15. Proper dimensioning of circles and arcs.

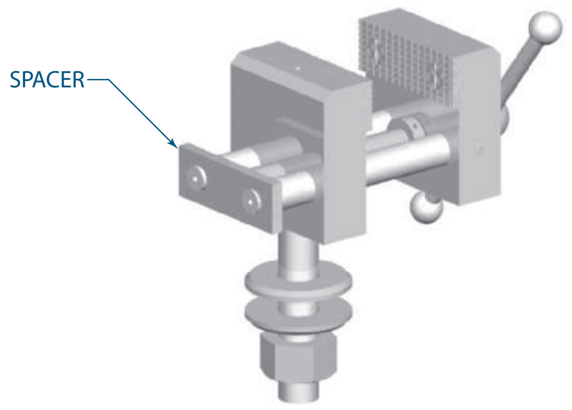

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FIGURE 12.16. The vise assembly SPACER.

Figure 12.17 includes two examples of dimensioning the location of the holes on the spacer from the assembly shown in Figure 12.16. You might be asking yourself, why does it matter whether you dimension the holes from the ends of the part (b) or give the dimension between the centers (a)?

Imagine the parts are manufactured according to the fixture shown in Figure 12.18. A jig has been set up such that the SPACER stock material is slid into the jig, held down, and then cut to its overall length of 70 mm. Next, a machinist uses the dimensions on the drawing you prepared to locate and drill the two holes. What happens if during the day of manufacturing parts, the jig begins to slip? By the end of the day, the overall length of the parts is coming out to be 72 mm instead of 70 mm. If the machinist used the drawing

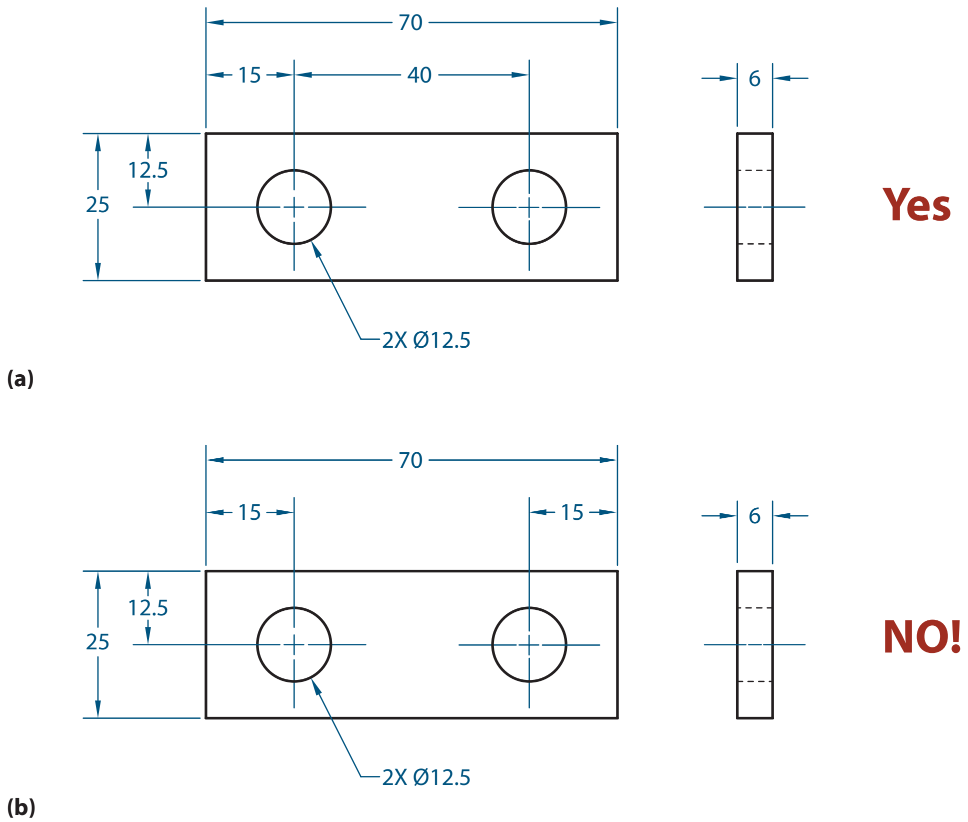

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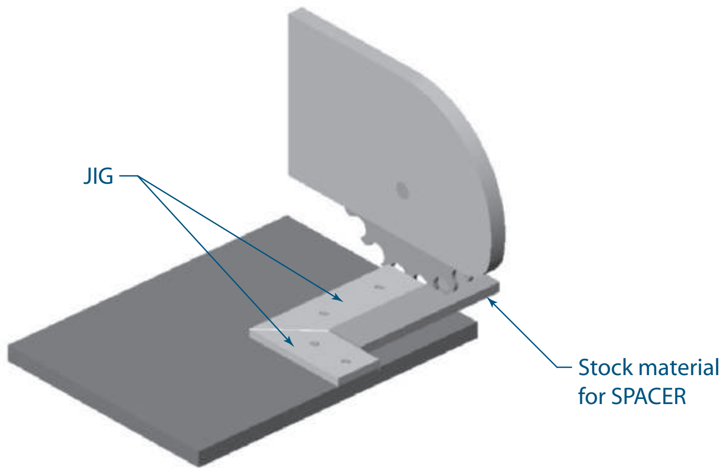
FIGURE 12.17. Two possible dimensioned drawings of the SPACER.

FIGURE 12.18. Cutting the SPACER.



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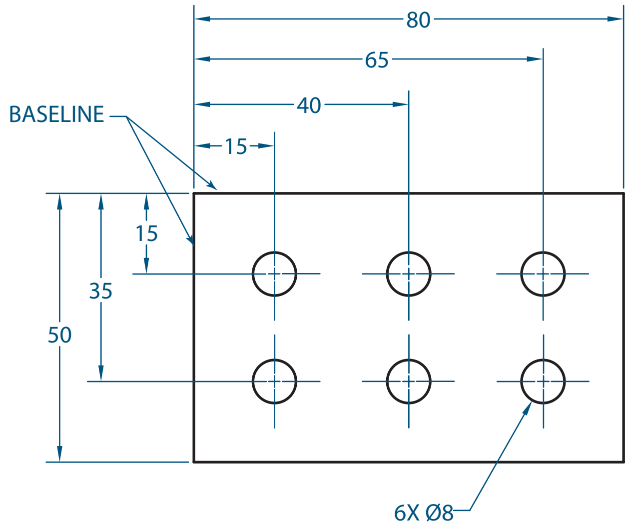
in Figure 12.17a to locate and drill the holes, the parts would not function since the distance between the holes is probably 42 mm instead of 40 mm (72 – 30 = 42). If the drawing in Figure 12.17b was used to machine the parts, the overall length would still be incorrect, but the distance between the holes would be right. The SPACER would still function, and the additional material could be removed if necessary.

12.06.03 Baseline versus Chain Dimensioning

There are many different ways to locate features. As mentioned already, starting with an examination of how the part will function within the assembly is the best way to begin determining which dimensions are most important. Two of the main types of dimensioning techniques are baseline and chain.

Baseline dimensioning is illustrated in Figure 12.19. Notice how all of the dimensions in a given direction originate from a base or datum. This type of dimensioning is frequently used for CNC machines that work from a rectangular coordinate system.

FIGURE 12.19. Baseline dimensioning.



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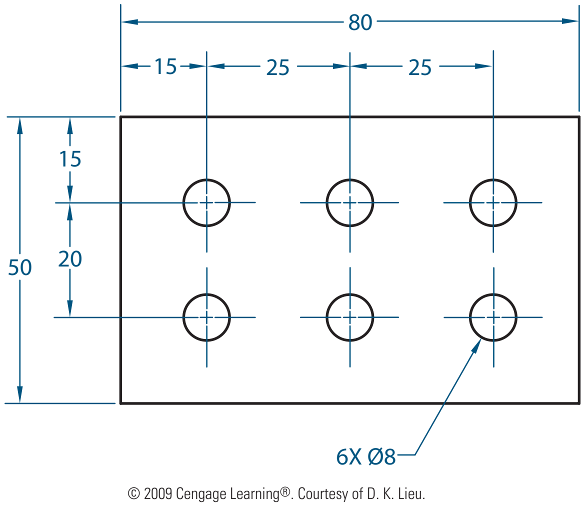


FIGURE 12.20. Chain dimensioning.

Chain dimensioning is shown in Figure 12.20. In this system, features are dimensioned relative to one another. This is appropriate when part function requires that features be related to one another, as discussed previously for the part in Figure 12.17b. As you will see in the next chapter, chain dimensioning can cause problems with tolerance accumulation; so baseline dimensioning is often preferred. However, there are times when chain dimensioning is appropriate.

12.06.04 What Types of Dimensions Can Be Measured and Checked?

As was discussed earlier, it is important that you give dimensions that make sense to the person who is manufacturing or constructing the object you are designing. When dimensioning holes, you dimension to their centers because the machinist will locate the same points and center the drill bit at that location. When dimensioning parts, you also should select dimensions that can be measured. The object in Figure 12.21a is dimensioned to locate the center of the R50 arc. It would be very difficult for the person inspecting the part to locate the center of the arc since it is not on the object. In Figure 12.21b, the ends of the arc are dimensioned, as well as the radius. This is better practice because the linear dimensions on the final part can be easily checked with standard measuring tools.

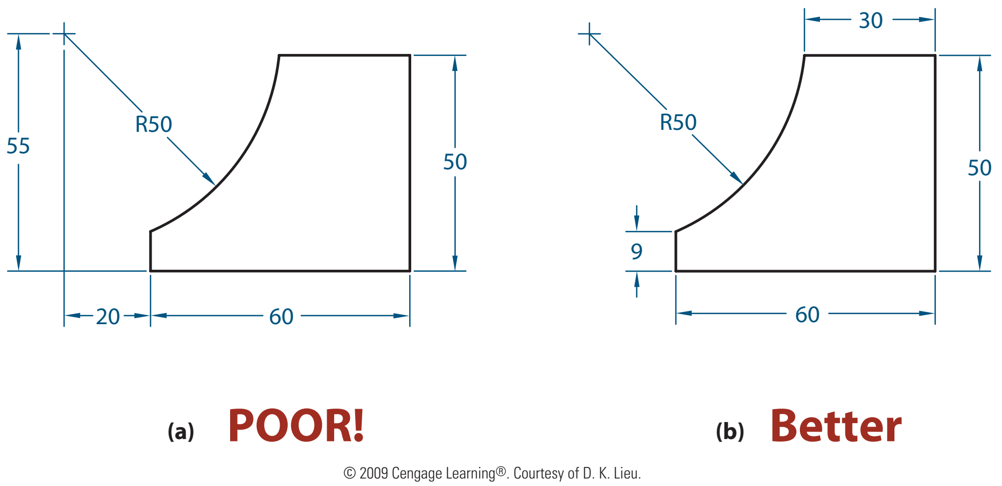
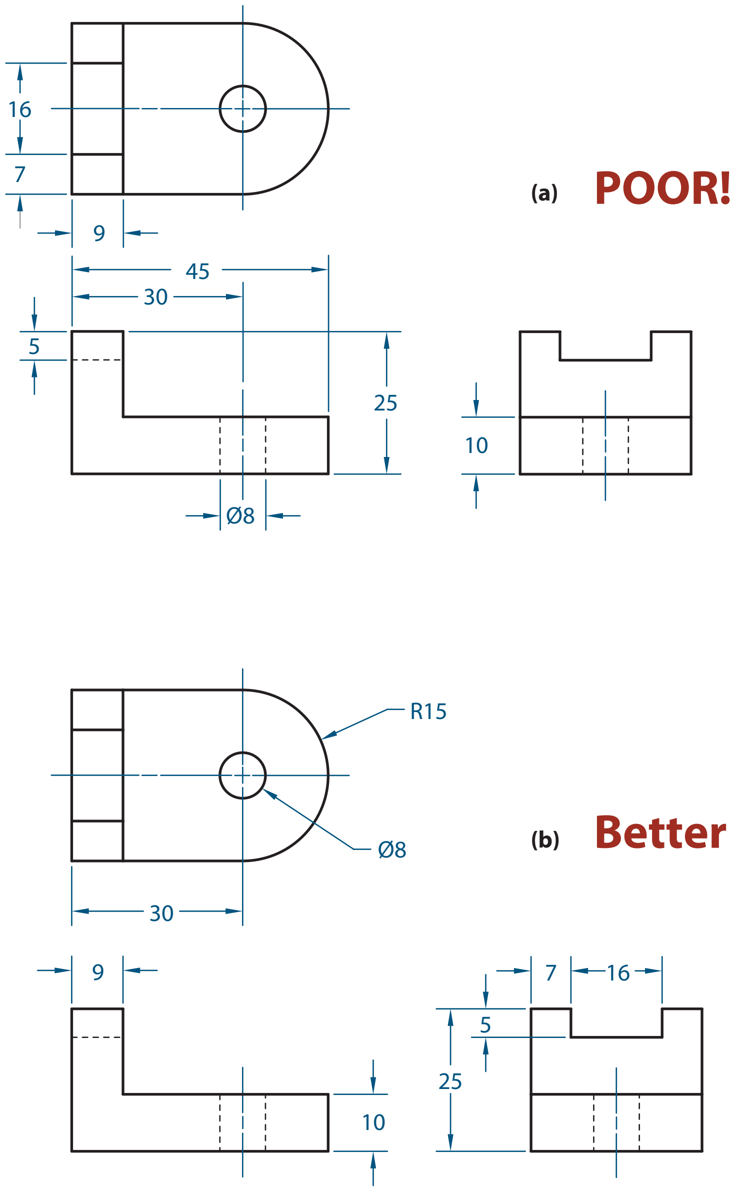


FIGURE 12.21. Checking the location of an arc center.

12.07 Guidelines to Guide Your Lines

As you dimension more parts, you will get a better idea about where to place particular dimensions. As mentioned earlier in the chapter, showing dimensions in the view where the contour or shape of the object shows up the best is a good global rule to follow. There are some exceptions; but for most parts, following this contour rule is good practice. Figure 12.22a shows an example of poor dimensioning. Notice that the contour of the slot shows up the best in the right-side view; but the depth dimension of 16 is given in the top view, and the height dimension of 5 is given in the front. The size and location dimensions for the hole also are not clear in Figure 12.22a. The diameter of 8 and the location dimension of 30 from the left side should both be in the top view, not the front view. Figure 12.22b shows the hole correctly dimensioned in the view where its size, location, and shape show up the best (top view).

FIGURE 12.22. Contour dimensioning.



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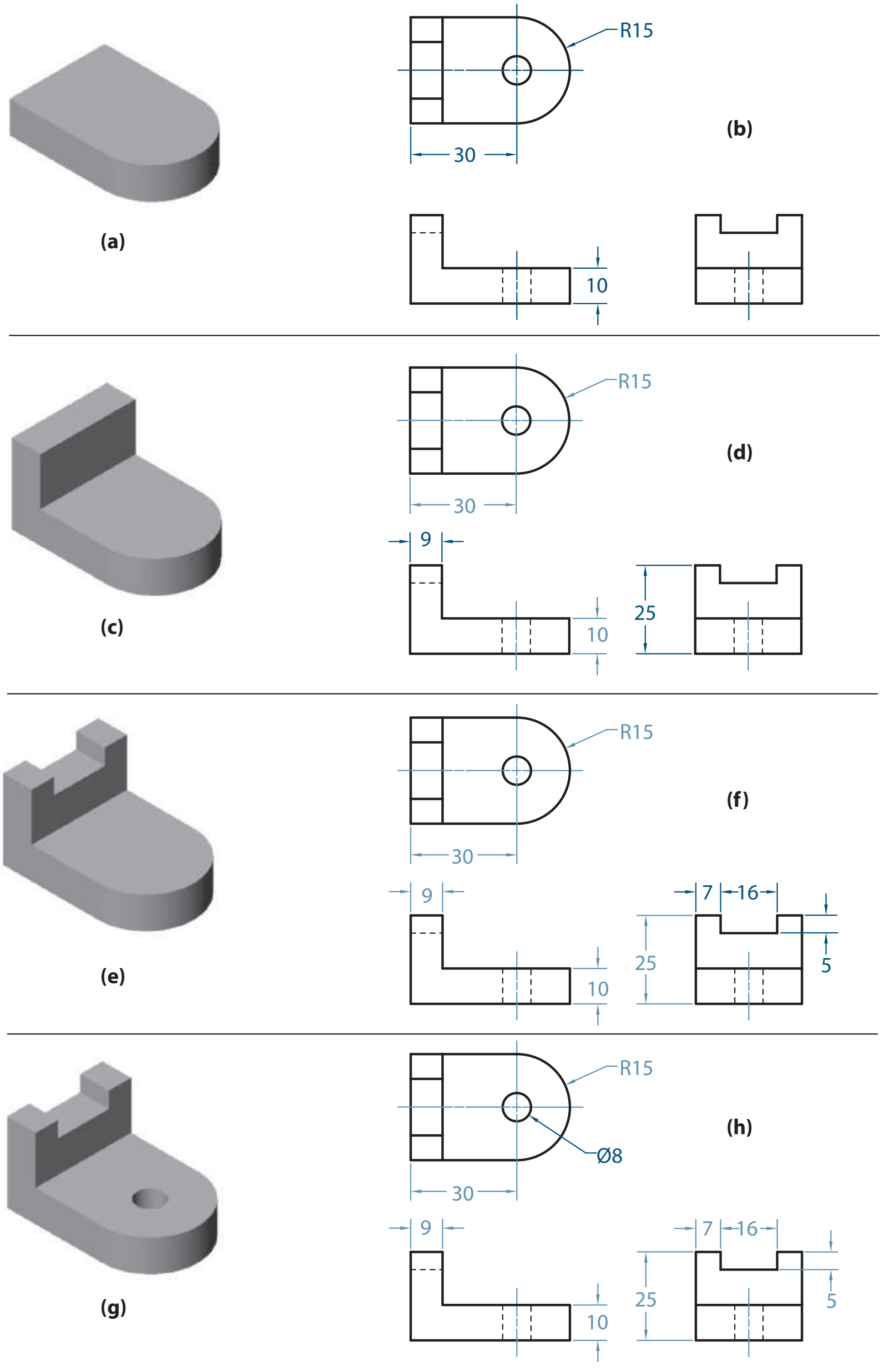
As an illustration of the process used in picking the correct view for dimensions, this example can be broken down into smaller steps. Figure 12.23 illustrates a step-by-step feature breakdown of the CONTOUR BLOCK. A solid model of the first feature is shown in Figure 12.23a. This feature is defined by three dimensions shown in 12.23b: the radius of the arc (R15), the distance from the left side of the part to the center of the arc (30), and the height of the feature (10). Based on following

FIGURE 12.23. Dimensioning breakdown of the CONTOUR BLOCK.



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contour dimensioning, the R15 and 30 dimensions show up best in the top view; but the 10 dimension is best placed in the front view. The next feature is the extruded piece on the left side of the part shown in Figure 12.23c. This feature is defined by the two dimensions in Figure 12.23d: the height of the feature from the bottom of the part (25) and the width of the part (9). The front view is the only view where the 9 dimension clearly shows the width of the extrusion. The height of the extrusion (25) can be shown in either the front or right-side views; but since you will be putting other dimensions in the right-side view, it is better to group them. The rectangular cut feature is shown in Figure 12.23e. This feature is defined by three dimensions in Figure 12.23f: the height of the cut (5), the depth of the cut (16), and a location dimension for the cut (7). The contour of this feature is best seen in the side view, so that is where these three dimensions should be located. The last feature of this part is the hole shown in Figure 12.23g. Since the hole has the same center as the arc, there is no need for a location dimension. The only dimension necessary for the hole feature is the diameter. Figure 12.23h illustrates how this dimension shows up best in the top view, where the hole's contour is most clearly seen.

12.07.01 Solid Lines Only

Another good rule of thumb to follow when dimensioning is to dimension only to visible or solid lines. This is related to the contour rule. In Figure 12.24a, notice how the extension line of the 10 dimension is related to the hidden line. The dimension is much clearer in Figure 12.24b where the extension line extends from a visible or solid line. Also notice the illustration of this rule on the drawing shown in Figure 12.22.

12.07.02 Placement and Spacing

As you place dimensions on drawings, you should follow established guidelines for the distance that dimensions should be located from views, gaps between extension lines and visible lines, lengths of arrowheads, etc. Figure 12.25 shows the standard practice related to dimension placement and spacing. When someone is looking at your drawing, the first thing that will be noticed will be the object itself. Several conventions and standards help distinguish dimensions from object geometry. As mentioned earlier in the book, visible lines are thick and dark to make the outline and visible edges of the object stand out. Dimension lines, extension lines, and leader lines should be thin and dark. Dimension lines also should be at least 10 mm (.375 inch) from any view, helping to avoid clutter. When dimensions are placed outside other dimensions, there should be at least 6 mm (.25 inch) between dimension lines. The standards for dimensioning also require a visible gap between extension lines and object geometry. Typically, 1 mm (.0625 inch) is a good rule of thumb. Also, extension lines should extend just past their corresponding dimension line (2–3 mm or .125 inch).

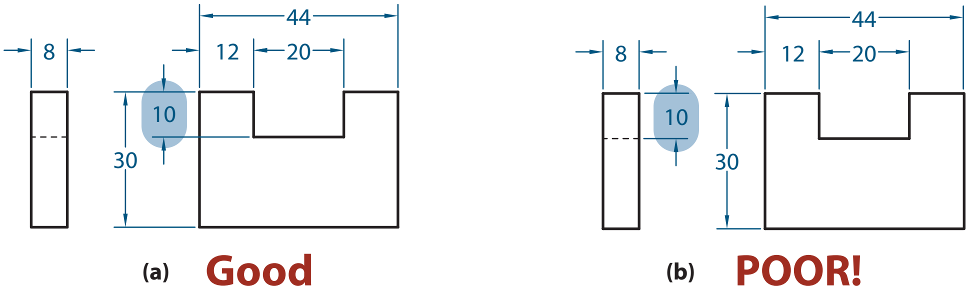

FIGURE 12.24. Dimensioning to solid lines.

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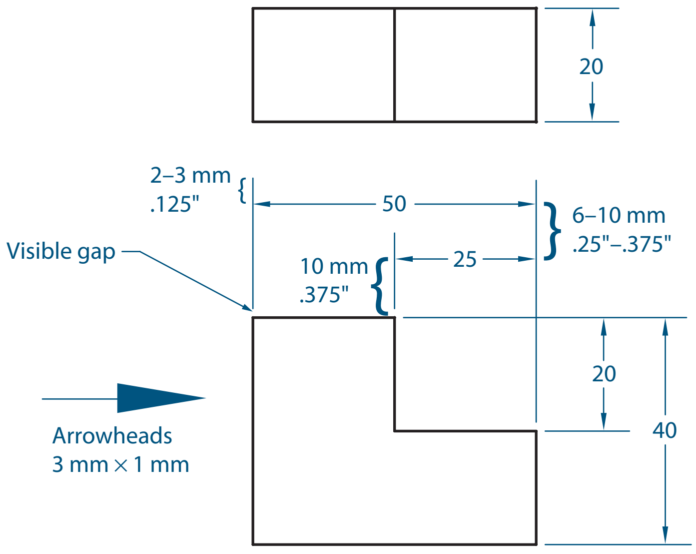

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FIGURE 12.25. Dimension placement and spacing.

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FIGURE 12.26. Fonts for dimensioning.

12.07.03 Font

Engineering drawings require the use of single-stroke gothic lettering. In addition, letters should be uppercase since few languages worldwide have an upper- and a lowercase. Typical fonts used in CAD software are Century Gothic and Romans.shx (see Figure 12.26).

12.08 Shortcuts

The last 20 years have seen a shift toward the use of symbols to define features on drawings rather than notes written in English. Since many companies have adopted international standards for design and production and they must be able to communicate in a universal language that everyone understands, symbols often lend themselves to clarity of design intent. Some of these symbols are used in the shortcuts representing dimensions for diameters, radii, chamfers, machined holes, threads, and standard features as described in the following sections.

12.08.01 Diameters and Radii

As shown earlier in the chapter in Figure 12.15, circles are dimensioned as diameters using the Ø symbol and arcs are dimensioned as radii using the R symbol. Both symbols are shown preceding the dimension value. For holes, diameter dimensions are usually shown in the view where the hole appears as a circle since that is the view of the part a machinist will see when the hole is being produced. When the diameter of a cylinder is dimensioned, however, the dimension should be placed in the rectangular view of the feature (see Figure 12.27). This helps distinguish holes from positive space cylinders.

Figure 12.28 illustrates several options for dimensioning arcs. When the arc is large enough, as in (a), the leader line and text can be placed on the inside of the arc. In (b), the arc is not large enough to place the text on the inside. In this case, the leader line should extend through the arc with the text on the outside. With small arcs such as (c) and (d), the leader line and the text should be placed on the outside.

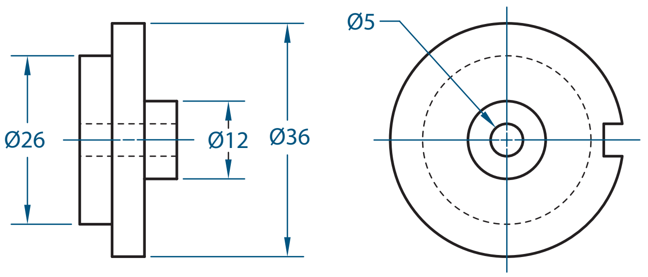


FIGURE 12.27. Dimensioning cylinders and holes.

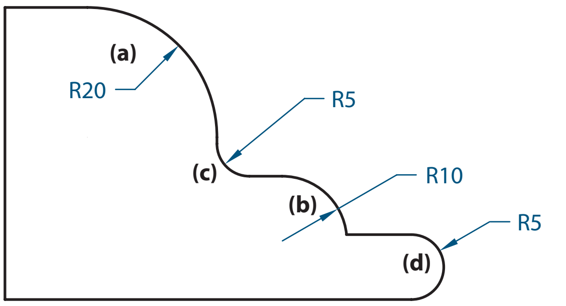
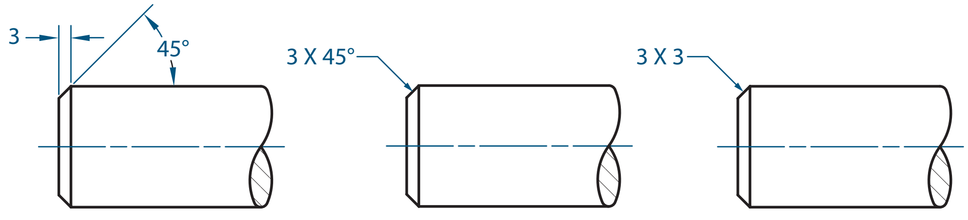


FIGURE 12.28. Dimensioning arcs.

FIGURE 12.29. Dimensioning chamfers.



12.08.02 Chamfers

Chamfers are beveled or angled edges that typically appear on the ends of shafts or fasteners to aid in assembling parts or to smooth out rough edges. They are dimensioned by giving a length from the end of the part and an angle or by specifying two distances. Figure 12.29 illustrates the different options for dimensioning a chamfer.

12.08.03 Standard Machined Holes: Countersinks and Counterbores

The use of symbols also is very important when you are dimensioning the sizes of machined holes such as counterbores, countersinks, spotfaced holes, and blind holes. Take a look at Figure 12.30. Symbols used in the top view represent the different types of machined holes. In these examples, ⌴ represents a counterbore, ⌵ represents a countersink, and ↧ is the symbol used for specifying depth. Note that according to standard practice, no manufacturing processes are specified (e.g., drill, ream, or bore).

- Figure 12.30a illustrates a standard *drill* hole with a diameter of 8 mm.
- Figure 12.30b illustrates a *blind* hole with the same diameter. The depth of 15 is measured from the top surface to the horizontal line at the bottom of the cylindrical portion of the hole, not the point.
- A *counterbore* hole is shown in Figure 12.30c. The 8 diameter indicates the original drill size, the 15 diameter is the size of the counterbore, and the 7 is the depth of the counterbore. Counterbore holes are used to accept fillister head and hex socket head screws.
- Figure 12.30d illustrates a *countersink*. The 8 indicates the original drill diameter, the 14 is the diameter of the countersink, and the 82 degrees is the angle of the countersink bit. Countersunk holes are used for applications with flat head and oval head screws.
- A *spotface* hole is shown in Figure 12.30e. Spotfacing is used to clean off the rough surface of a cast part typically to accept a hexagon head type screw. The format of the dimension is the same as the counterbore; however, the depth dimension may be left off if a company uses a standard spotface depth.

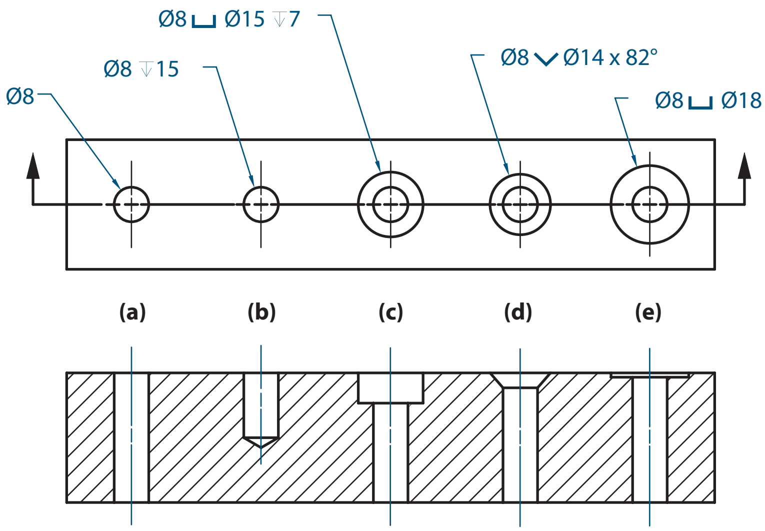


FIGURE 12.30. Dimensioning the sizes of machined holes.

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It should be noted that you string the symbols together in the order that a machinist would perform the operations. For example, in Figure 12.30c, the diameter of the through hole is given first, followed by the diameter and depth of the counterbore. These symbols are included in that order because a machinist would first drill the hole and then make the counterbore at that location.

When multiple holes with the same size are present, only dimension one of the holes. The X symbol is used to indicate how many times that particular hole is machined. In Figure 12.31, 4X is placed before the counterbored hole dimension to indicate that four holes require that size dimension.

12.08.04 Slots

Slots are produced with standard tools such as milling bits. Since these tools are specified by their diameters, slots also should be dimensioned by their diameters. Figure 12.32 shows several acceptable ways that slots can be dimensioned. Notice that in each case, the end radii are indicated but not dimensioned.

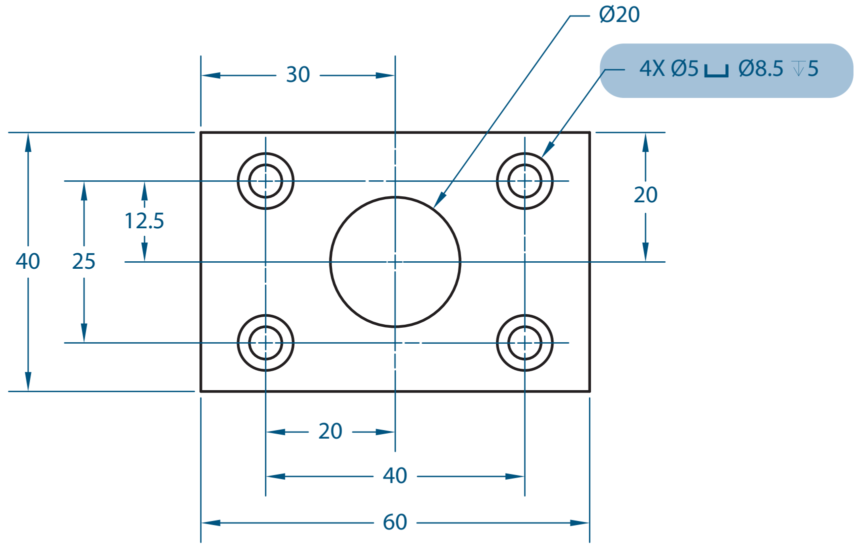

FIGURE 12.31. Dimensioning multiple holes.

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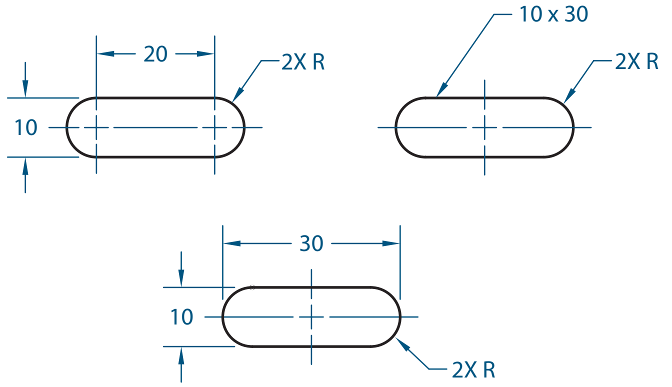
FIGURE 12.32. Dimensioning slots.



12.09 Notes

Most drawings require some type of note or notes in addition to the dimensions on the drawing in order to fully define the part. Since the purpose of your drawing is to give all of the information necessary to manufacture the part, some pieces of information cannot easily be shown in typical dimensions. No matter what type of note is being shown, all of them should be placed so they are read from the bottom of the sheet of paper.

12.09.01 General Notes

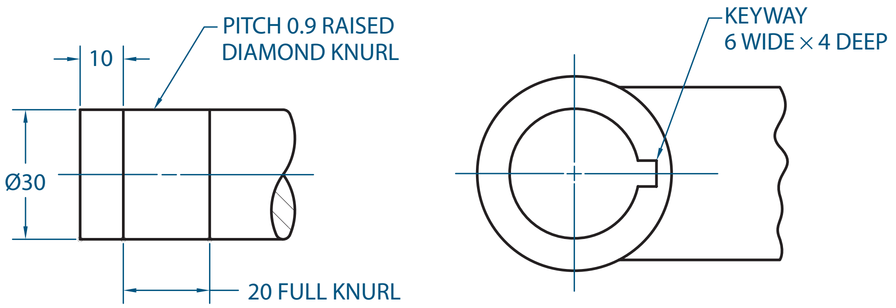
General notes typically appear in the lower right-hand corner of a drawing and apply to the whole drawing. Some may be located in the titleblock. Examples of general notes are as follows:

MATERIAL: CAST IRON
FAO (*finish all over*)
ALL DIMENSIONS ARE IN MILLIMETERS
ALL DIMENSIONS `0.1 UNLESS OTHERWISE SPECIFIED
BREAK ALL SHARP EDGES

12.09.02 Local Notes

Local notes appear on the drawing views and are usually specified with a leader line. Like general notes, local notes are used to specify information that cannot be shown with regular dimensions. Figure 12.33 includes examples of local notes.

FIGURE 12.33. Using local notes.



12.10 Considerations for 3-D Modeling

Now that you have covered all of the rules and guidelines for dimensioning, take some time to think about what all of this means as you are creating parts using a 3-D modeler. If you have already been creating solid models, you probably noticed that drawings sometimes require more dimensions than what you would use when modeling. This happens because you can imbed certain geometric relations or constraints within a 3-D model that must be explicitly pointed out on a drawing. Figure 12.34 illustrates this idea. Notice that the drawing includes dimensions from the center of the hole to the ends of the part, but the 3-D model sketch does not. A machinist would need to know this information to locate the hole in the center of the part. The sketch incorporates symmetric constraints between the outside lines and their corresponding center line.

Since these differences between the dimensions are required in 3-D models and the drawings are required for documenting the parts, drawings with dimensions for manufacturing are typically done at the end of the design process instead of at the beginning. As you model parts, you want to add geometric and dimensional constraints that capture the design intent for each part. Documentation drawings can then be completed with dimensions for manufacture when the design is complete.

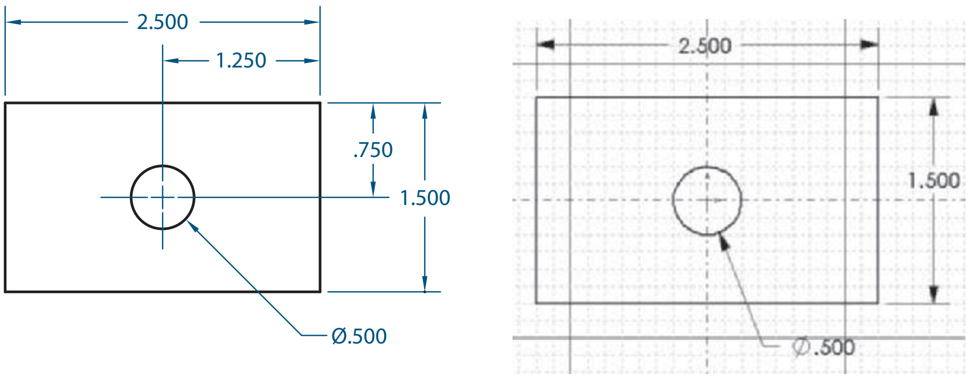
One of the nice features of constraint-based modelers is the ability of the software to let you know when geometry has been underdimensioned or overdimensioned. Your goal should be to fully define the geometry with geometric and dimensional constraints. If a constraint or constraints are missing, the software usually has some type of indicator that the geometry is underdefined. When geometry is underdefined, you should be able to grab entities and move them. When too many dimensions or geometric constraints are present, the software lets you know that the geometry is overdefined or overconstrained. To correct this problem, you must delete a dimension or geometric constraint that is in conflict with other constraints.

12.11 Dimensions for the Plate Example

Return to the scenario where you work for the company that manufactures hardware for household doors. You were asked to think about several questions. What are the dimensions needed to manufacture the plate in Figure 12.35? What are the critical dimensions that must be given? What are some of the standard dimensions that exist on other parts or previous parts?

Figure 12.36 shows an example of how the part might be dimensioned. The critical dimensions on the plate are the distances between the countersunk holes and the center hole, the sizes of the holes, and the overall size of the plate (since it must fit in the door properly).

FIGURE 12.34. Differences between dimensioning drawings and 3-D models.



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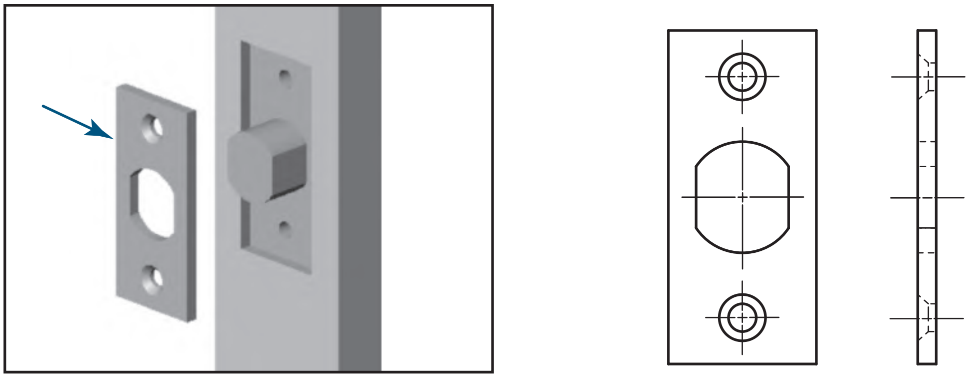

FIGURE 12.35. A deadbolt lock plate.

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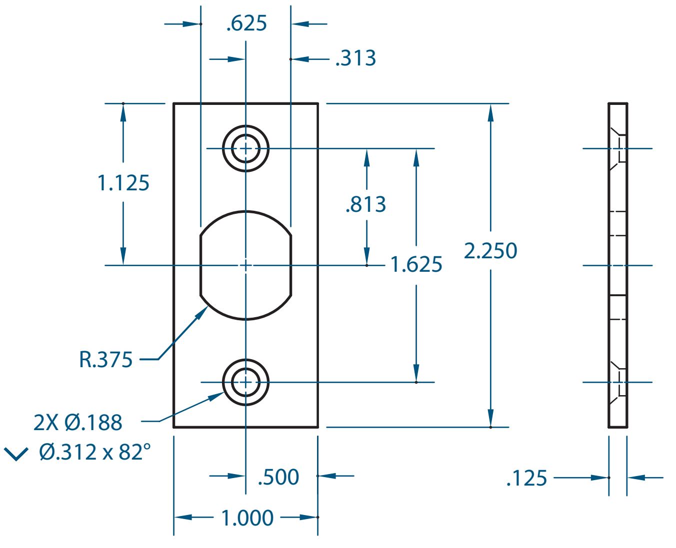

FIGURE 12.36. Plate dimensions.

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The plate drawing includes dimensions that may not be present in the constraint-based solid model. Since symmetric geometric constraints may have been incorporated into the model, the highlighted dimensions may not exist in the 3-D solid model database. These dimensions will need to be specified on the drawing to ensure that the part is manufactured properly.

12.12 Fundamental Rules for Dimensioning

This chapter contains best practice suggestions for dimensioning a part; however, as stated previously, *ASME Y14.5M-1994* is the accepted standard for dimensioning practice. The following fundamental rules are quoted from the *ASME Y14.5M-1994* standards for Dimensioning and Tolerancing. For a complete listing of the standards, see *ASME Y14.5M-1994*.

- **a.** Each dimension shall have a tolerance, except for those dimensions specifically identified as reference, maximum, minimum, or stock (commercial stock size). The tolerance may be applied directly to a dimension (or indirectly in the case of basic dimensions), indicated by a general note, or located in a supplementary block of the drawing format.
- **b.** Dimensioning and tolerancing shall be complete so there is full understanding of the characteristic of each feature. Neither scaling (measuring the size of a feature directly

from an engineering drawing) nor assumption of a distance of size is permitted, except as follows: Undimensioned drawings, such as loft, printed wiring, templates, and master layouts prepared on stable material, are excluded provided the necessary control dimensions are specified.

c. Each necessary dimension of an end product shall be shown. No more dimensions than those necessary for complete definition shall be given. The use of reference dimensions on a drawing should be minimized.

d. Dimensions shall be selected and arranged to suit the function and mating relationship of a part and shall not be subject to more than one interpretation.

e. The drawing should define a part without specifying manufacturing methods. Thus, only the diameter of a hole is given without indicating whether it is to be drilled, reamed, punched, or made by any other operation. However, in those instances where manufacturing, processing, quality assurance, or environmental information is essential to the definition of engineering requirements, it shall be specified on the drawing or in a document referenced on the drawing.

f. It is permissible to identify as nonmandatory certain processing dimensions that provide for finish allowance, shrink allowance, and other requirements, provided the final dimensions are given on the drawing. Nonmandatory processing dimensions shall be identified by an appropriate note, such as NONMANDATORY (MFG DATA).

g. Dimensions should be arranged to provide required information for optimum readability. Dimensions should be shown in true profile views and refer to visible outlines.

h. Wires, cables, sheets, rods, and other materials manufactured to gage or code numbers shall be specified by linear dimensions indicating the diameter or thickness. Gage or code numbers may be shown in parentheses following the dimension.

i. A 90-degree angle applies where center lines and lines depicting features are shown on a drawing at right angles and no angle is specified.

j. A 90-degree basic angle applies where centerlines of features in a pattern or surfaces shown at right angles on the drawing are located or defined by basic dimensions and no angle is specified.

k. Unless otherwise specified, all dimensions are applicable at 20°C (68°F). Compensation may be made for measurements made at other temperatures.

l. All dimensions and tolerances apply in a free state condition. This principle does not apply to non-rigid parts.

m. Unless otherwise specified, all geometric tolerances apply for full depth, length, and width of the feature.

n. Dimensions and tolerances apply only at the drawing level where they are specified. A dimension specified for a given feature on one level of drawing (e.g., a detail drawing) is not mandatory for that feature at any other level (e.g., an assembly drawing).

12.13 Chapter Summary

This chapter provided an introduction to dimensioning. The chapter discussed how all dimensions have a tolerance and how tolerances are important for the function of designs. Dimensioning, like other drawing topics, follows fairly specific standards or rules. Whether dimensioning in inches or millimeters, you must follow these standards. This chapter also covered techniques for dimensioning different features, such as standard parts, machined holes, and notes.

The next chapter will discuss tolerance dimensioning in more detail, as well as introduce the topic of geometric dimensioning and tolerancing. These topics are key to the production of parts that are based on the specific intent of the designer.

12.14 GLOSSARY OF KEY TERMS

ANSI Y14.5 (*ASME Y14.5M-1994*): Industry standard document that outlines uniform practices for displaying and interpreting dimensions and related information on drawings and other forms of engineering documentation.

arrowhead: A small triangle at the end of dimension lines and leaders to indicate the direction and extent of a dimension.

baseline dimensioning: A system of dimensioning where each feature is dimensioned from the same origin.

chain dimensioning: A system of dimensioning where features are dimensioned from one another instead of from an origin.

contour dimensioning: Placing each dimension in the view where the contour or shape of the feature shows up best.

contour rule: A drawing practice where each dimension should be placed in the view where the contour shape is best shown.

dimension: A numerical value expressed in appropriate units of measure and used to define the size, location, geometric characteristic, or surface texture of a part or part feature.

dimension line: A thin, dark, solid line that terminates at each end with arrowheads. The value of a dimension typically is shown in the center of the dimension line.

extension line: A thin, dark, solid line extending from a point on an object, perpendicular to a dimension line used to indicate the extension of a surface or point to a location preferably outside the part outline.

leader: A thin, dark, solid line terminating with an arrowhead at one end and a dimension, note, or symbol at the other end.

location: A dimension associated with the position of a feature on a part.

size: The general term for the size of a feature, such as a hole, cylinder, or set of opposed parallel surfaces.

tolerance: The total amount a specific dimension is permitted to vary. It is the difference between the upper and lower limits of the dimension.

12.15 QUESTIONS FOR REVIEW

1. What is the current standard for dimensioning and tolerancing in the United States?
2. Explain the difference between dimensioning standards for inches and the standards for millimeters.
3. List at least four fundamental rules for dimensioning.
4. What are the correct line types and darkness for dimension lines, extension lines, and leaders?
5. When a two-view drawing of a simple rectangular block is given, what dimensions are necessary?
6. Explain the difference between baseline and chain dimensioning.
7. What is contour dimensioning?
8. What are the standard symbols for diameter, radius, counterbore, countersink, and depth?
9. Explain why the dimensions for a constraint-based solid model of a design may be different from the dimensions that appear on a detail drawing of the part.

12.16 PROBLEMS

1. Sketch the necessary dimensions to fully define each object shown in Figure P12.1. Do not use redundant or reference dimensions.

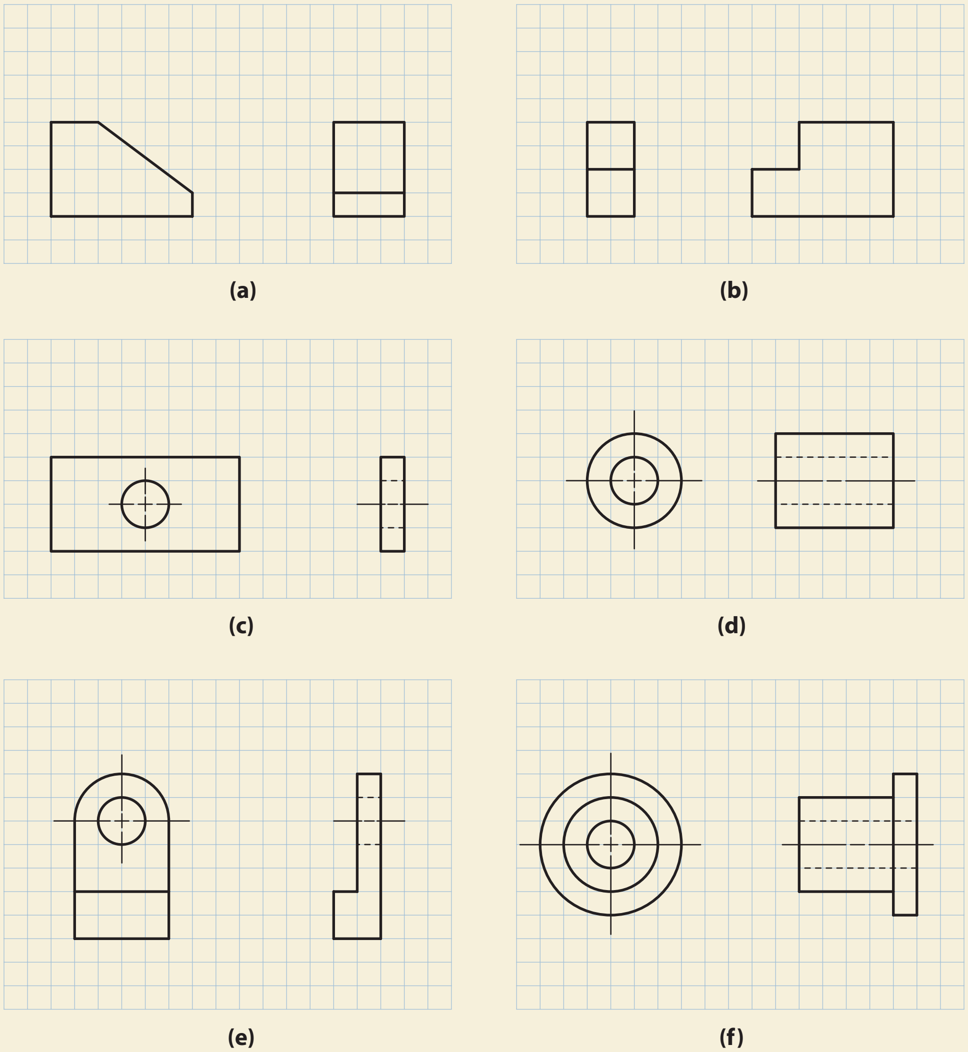

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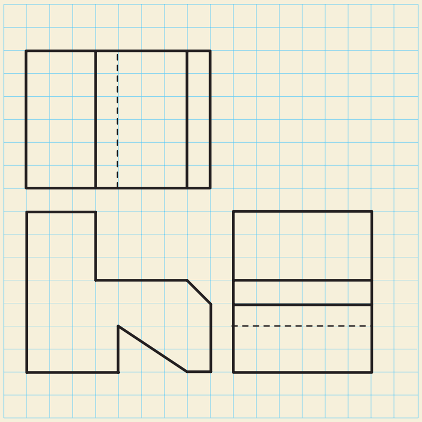
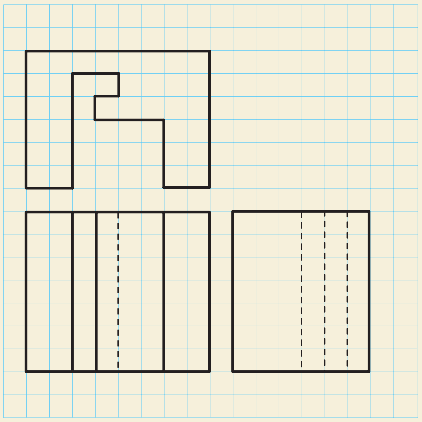
FIGURE P12.1.

12.16 PROBLEMS (CONTINUED)

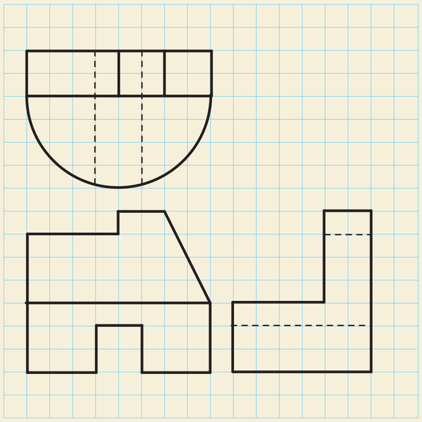
2. Scale and copy the drawings shown in Figure P12.2, leaving sufficient space between the views to add dimensions. Add the necessary dimensions to fully define each object. Add additional views as necessary to conform to the dimensioning guidelines in this chapter. Do not use redundant or reference dimensions.



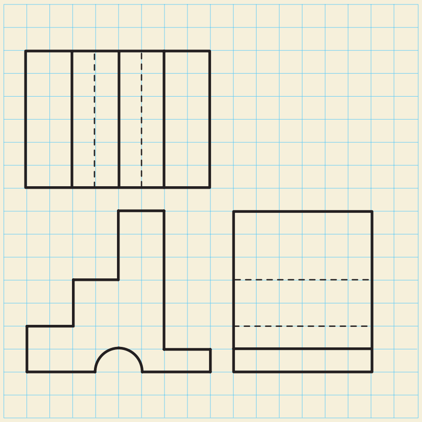
(a)



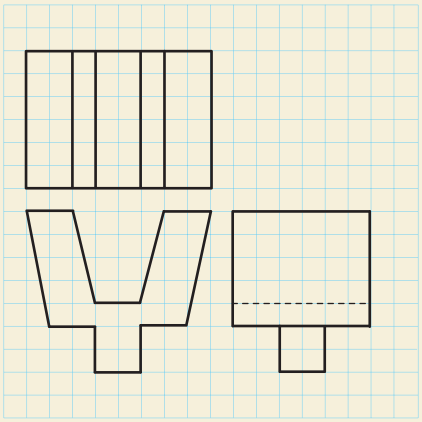
(b)



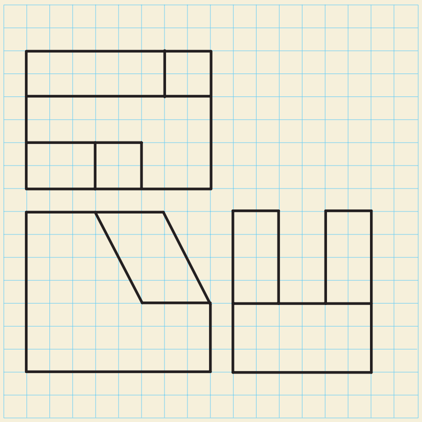
(c)



(d)



(e)



(f)

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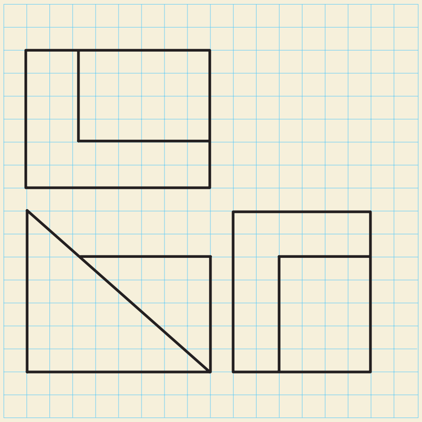
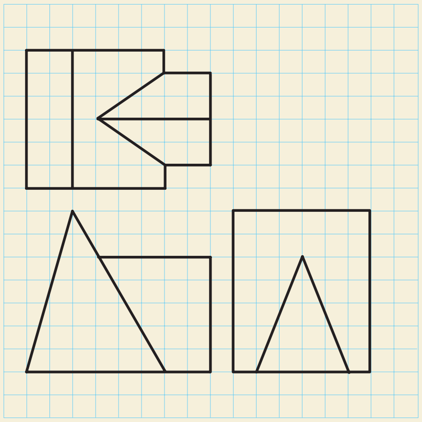
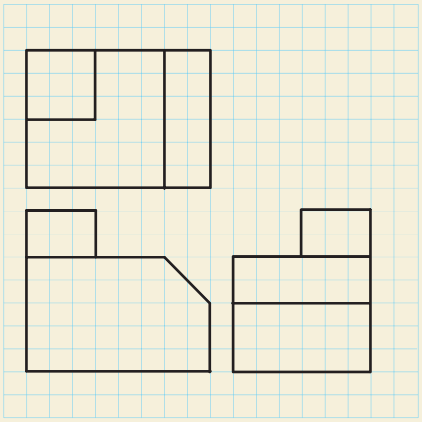
12.16 PROBLEMS (CONTINUED)



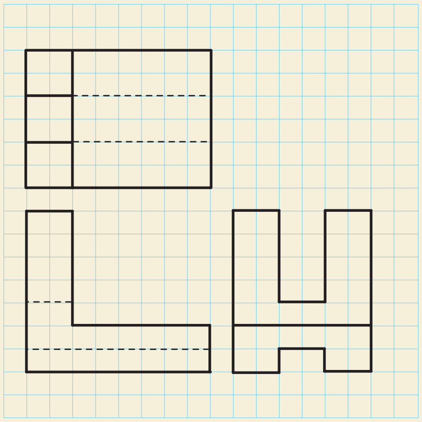
(g)



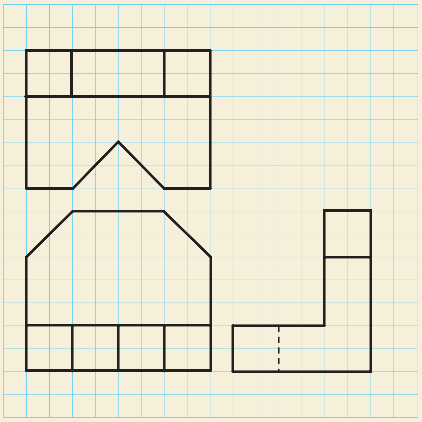
(h)



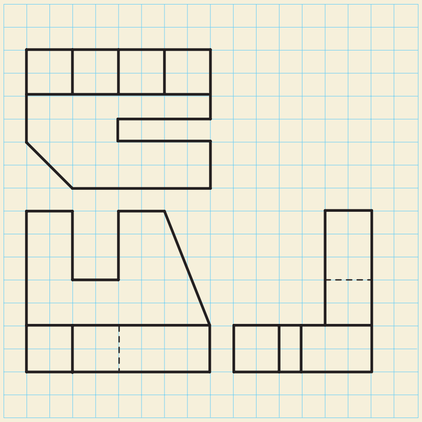
(i)



(j)

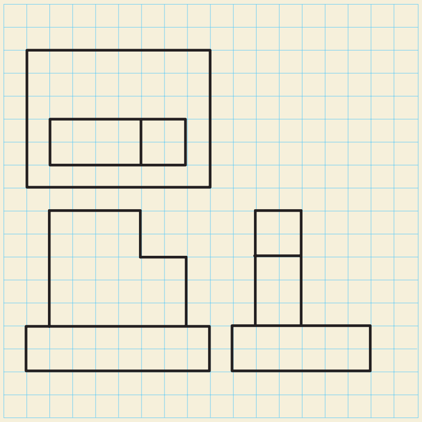


(k)

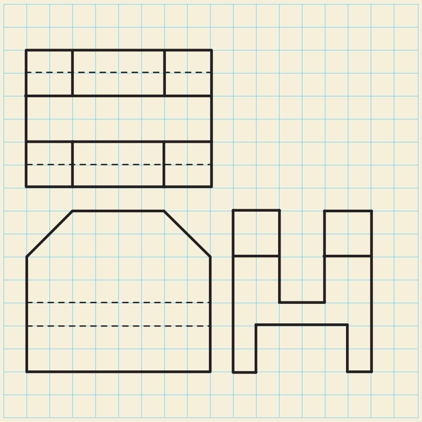


(l)

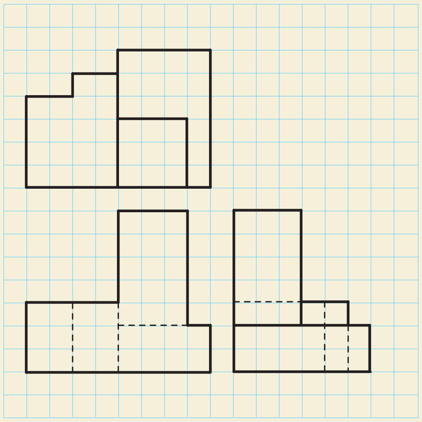
12.16 PROBLEMS (CONTINUED)



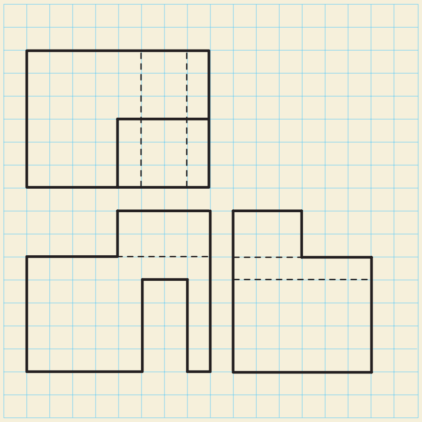
(m)



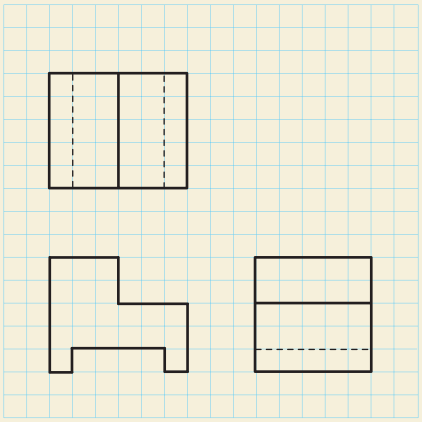
(n)



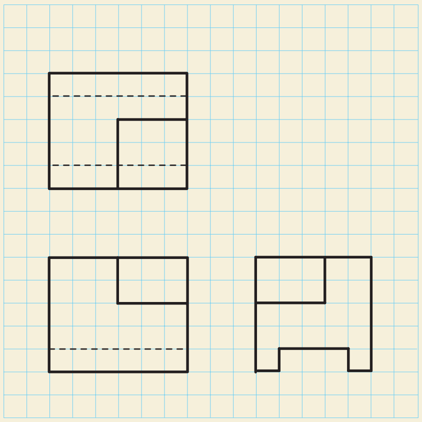
(o)



(p)

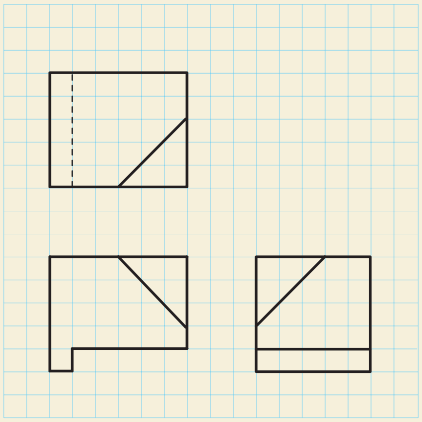


(q)

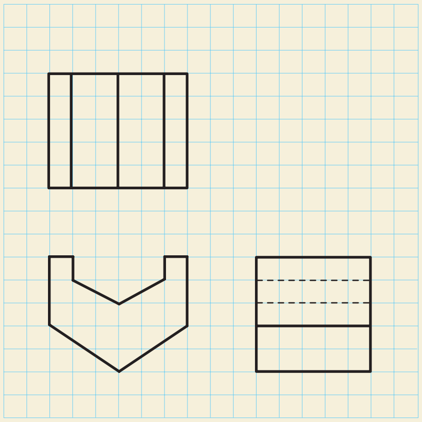


(r)

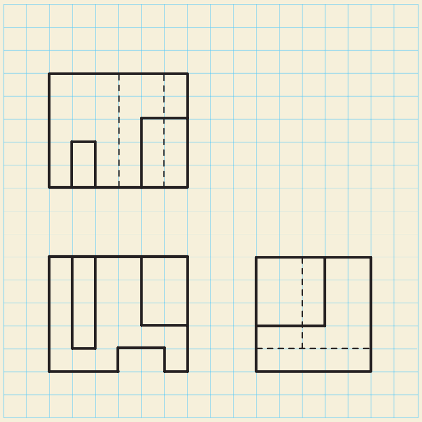
12.16 PROBLEMS (CONTINUED)



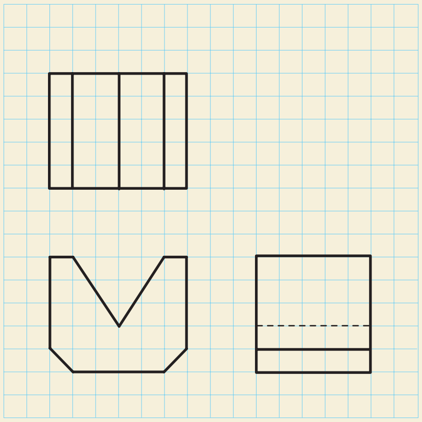
(s)



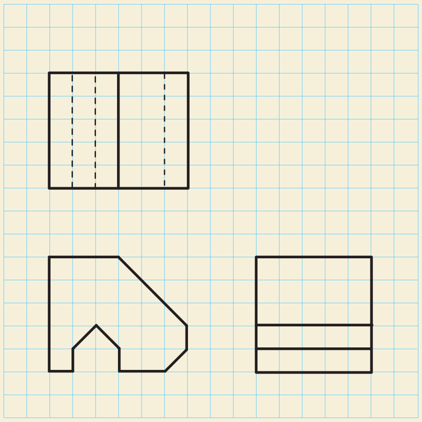
(t)



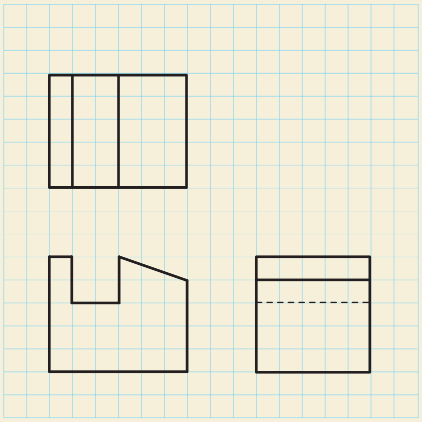
(u)



(v)



(w)



(x)

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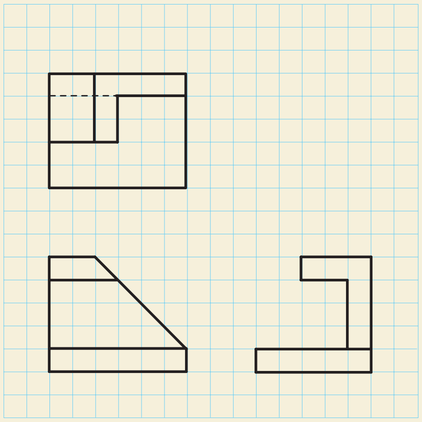
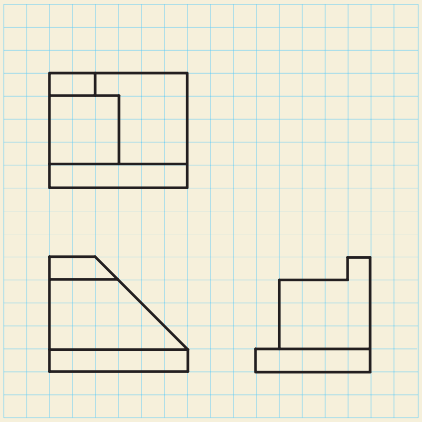
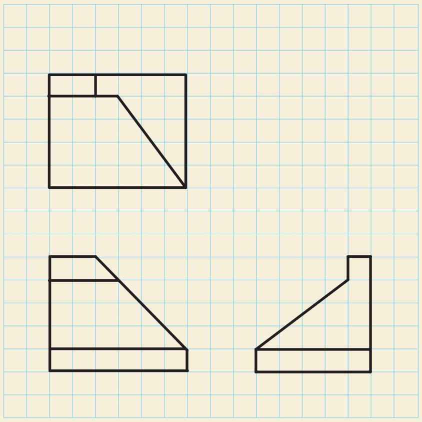
12.16 PROBLEMS (CONTINUED)



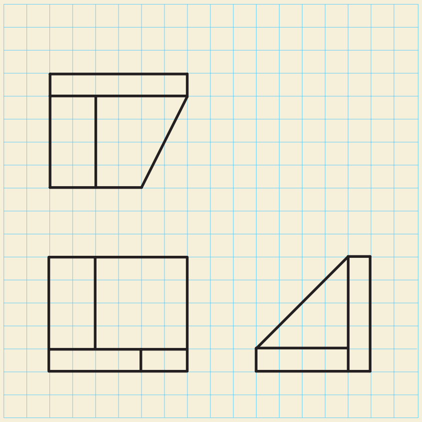
(y)



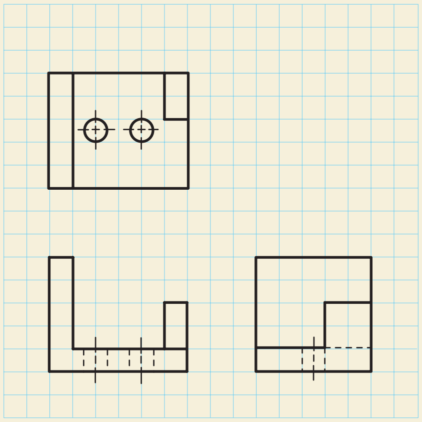
(z)



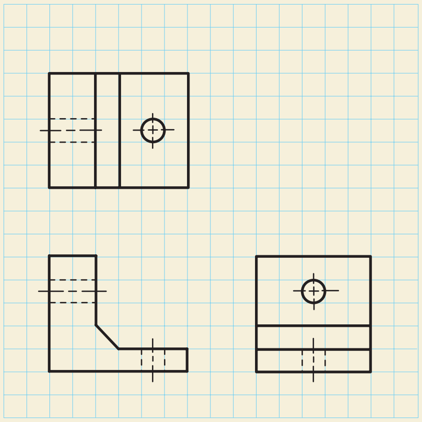
(aa)



(bb)

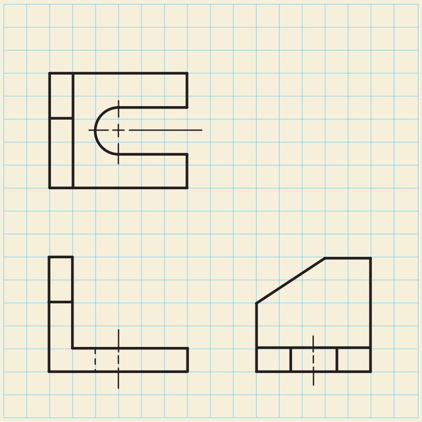


(cc)

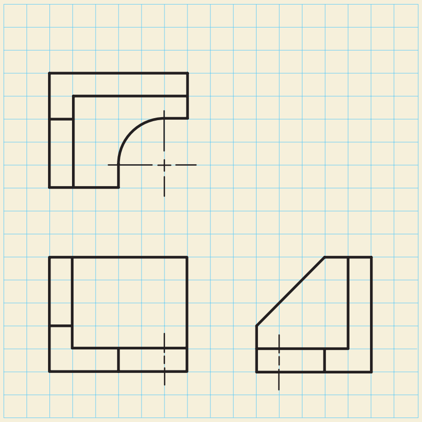


(dd)

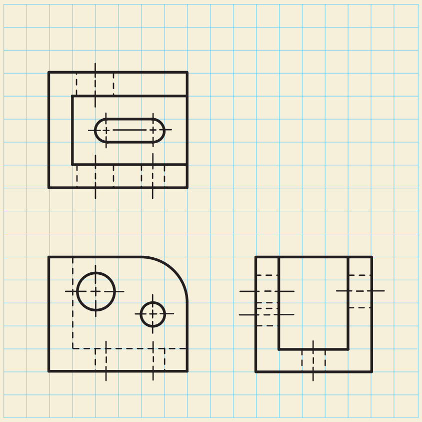
12.16 PROBLEMS (CONTINUED)



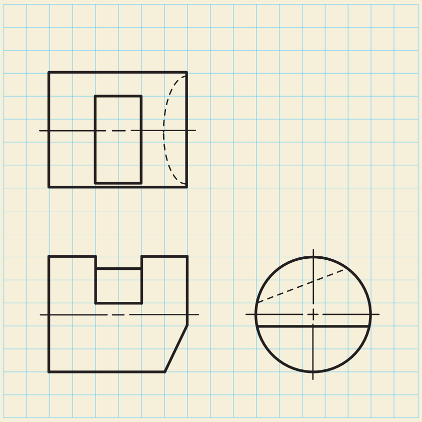
(ee)



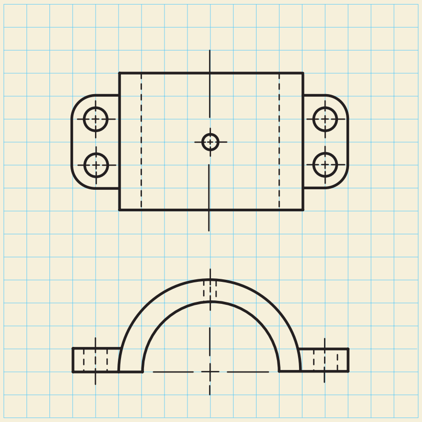
(ff)



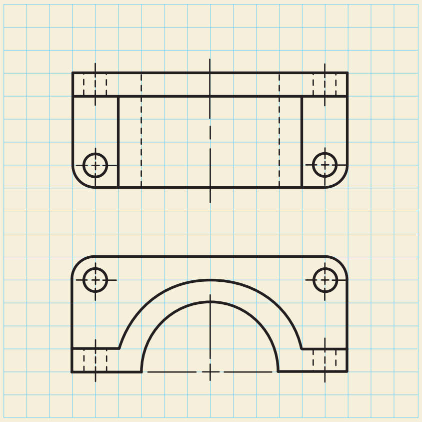
(gg)



(hh)



(ii)



(jj)

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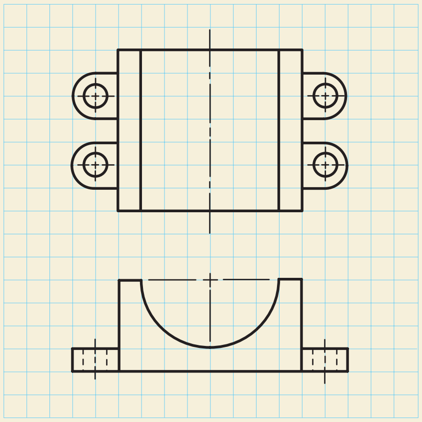
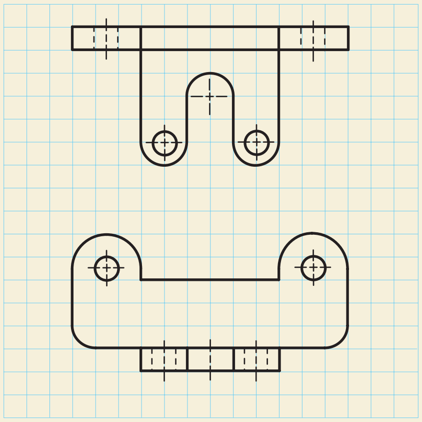
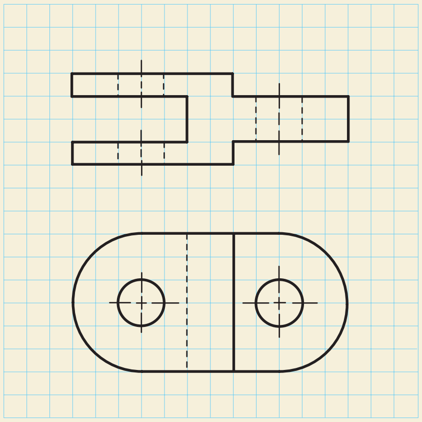
12.16 PROBLEMS (CONTINUED)



(kk)



(ll)



(mm)

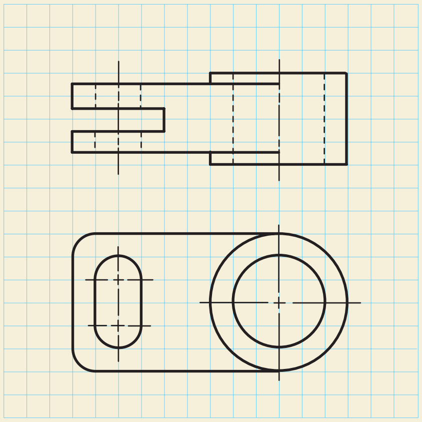


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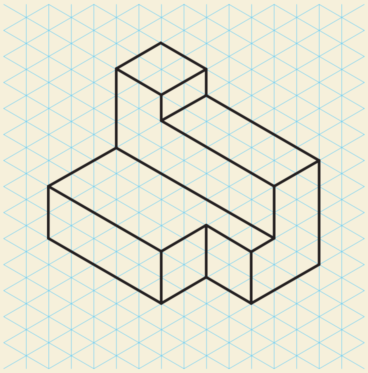
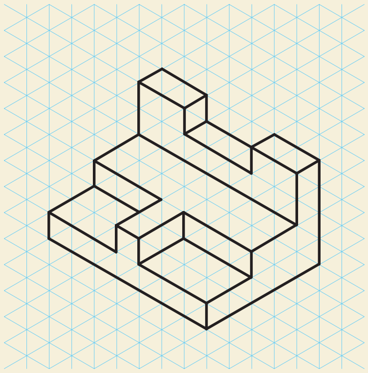
FIGURE P12.2. Add the necessary dimensions to fully define each object.

12.16 PROBLEMS (CONTINUED)

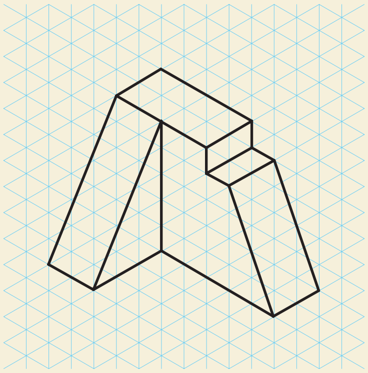
3. Scale and copy the drawings shown in Figure P12.3, leaving sufficient space between the views to add dimensions. Add the necessary dimensions to fully define each object. Add additional views and section views as necessary to conform to the dimensioning guidelines in this chapter. Whenever possible, apply accepted shortcut practices to describe appropriate features. Do not use redundant or reference dimensions.



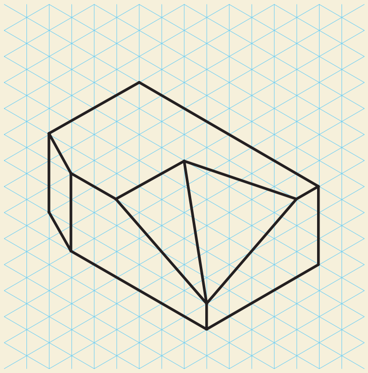
(a)



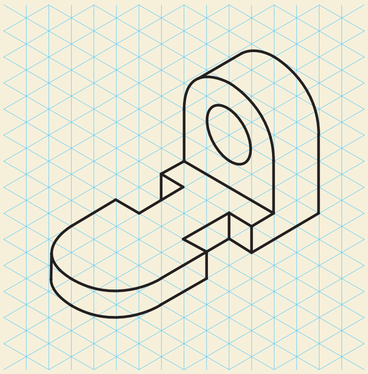
(b)



(c)



(d)



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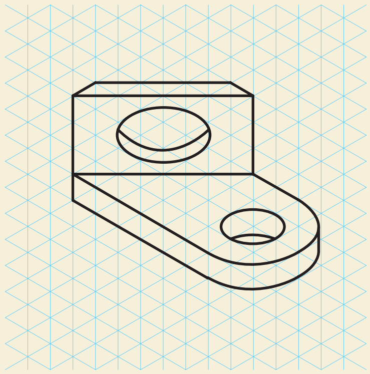

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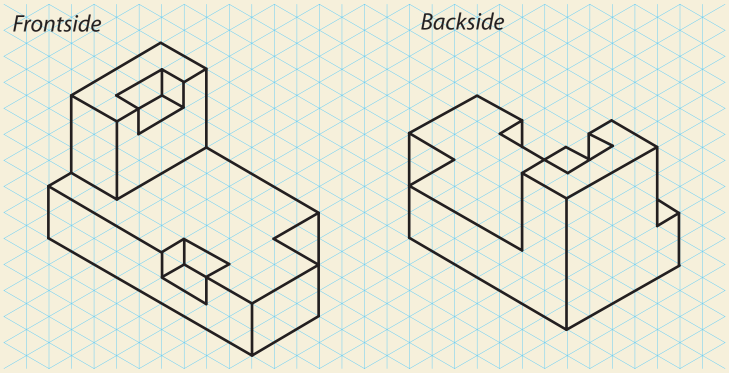

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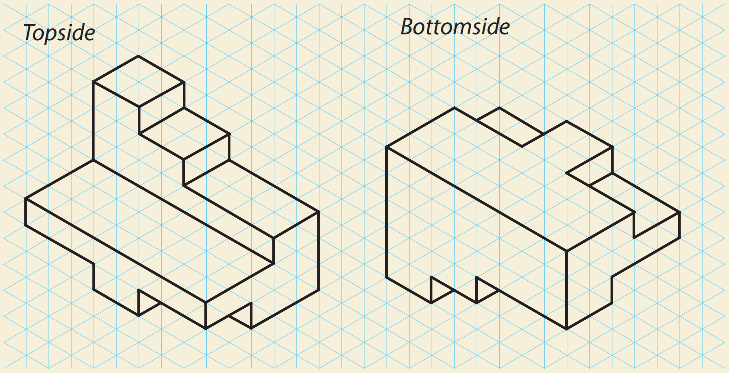

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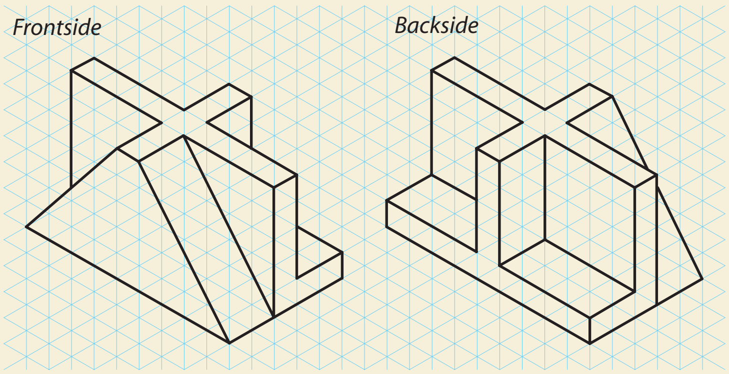

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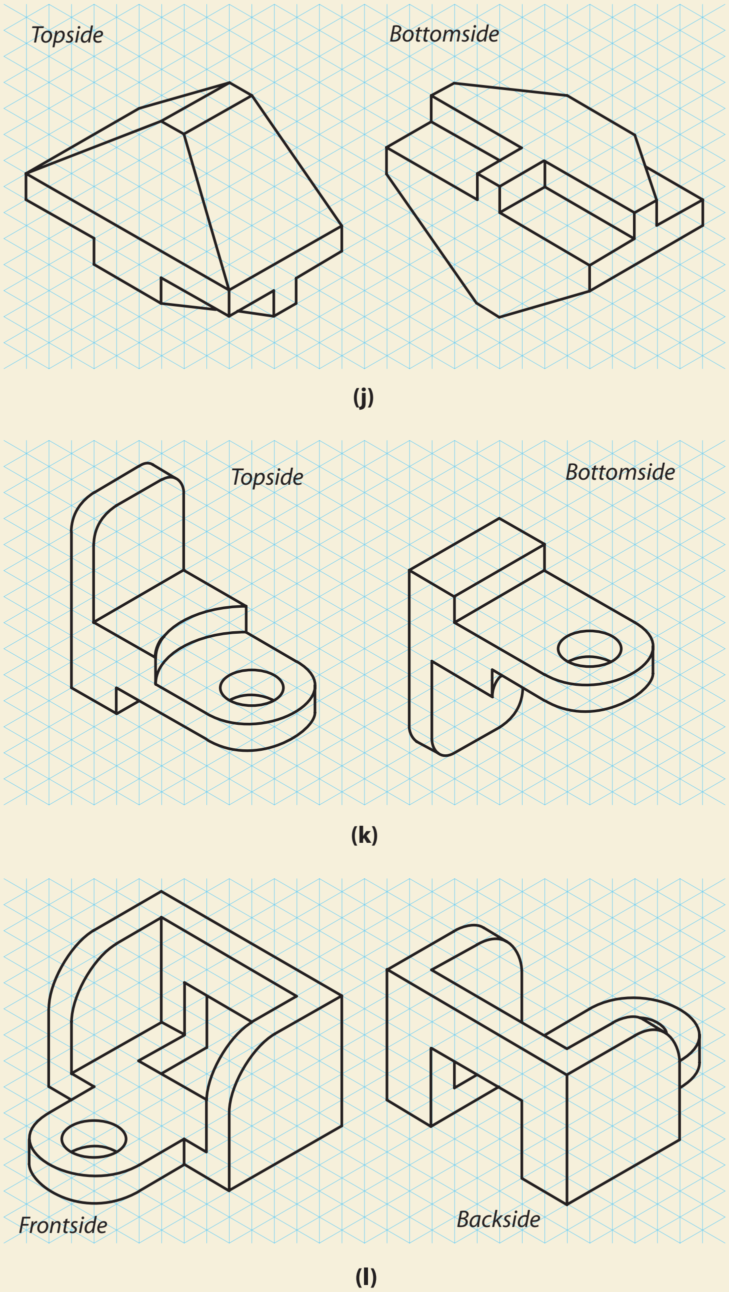
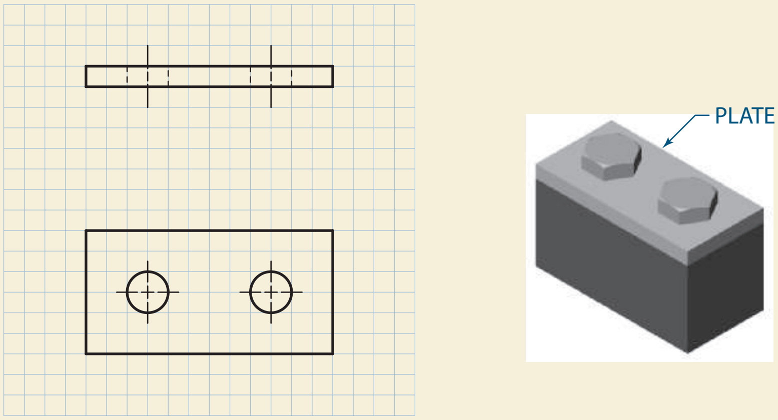


FIGURE P12.3. Create multiview drawings and add the necessary dimensions to define these objects.

12.16 PROBLEMS (CONTINUED)

4. In Figure P12.4, consider the function of the indicated parts shown in their intended assemblies. For each drawing, add the necessary dimensions to fully define the object, giving consideration to the critical dimensions necessary for each part to fit and function in its intended assembly.



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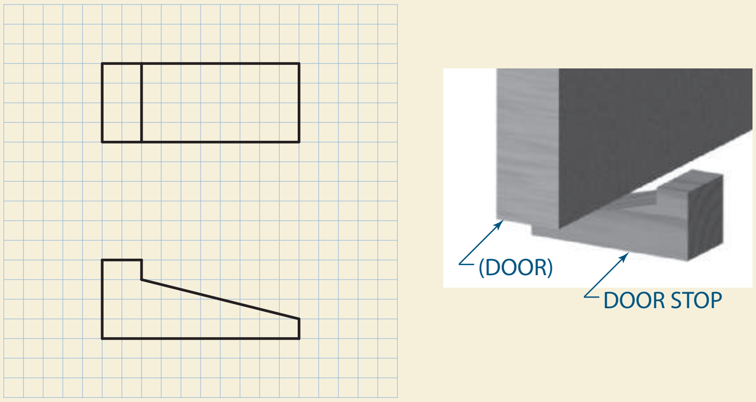

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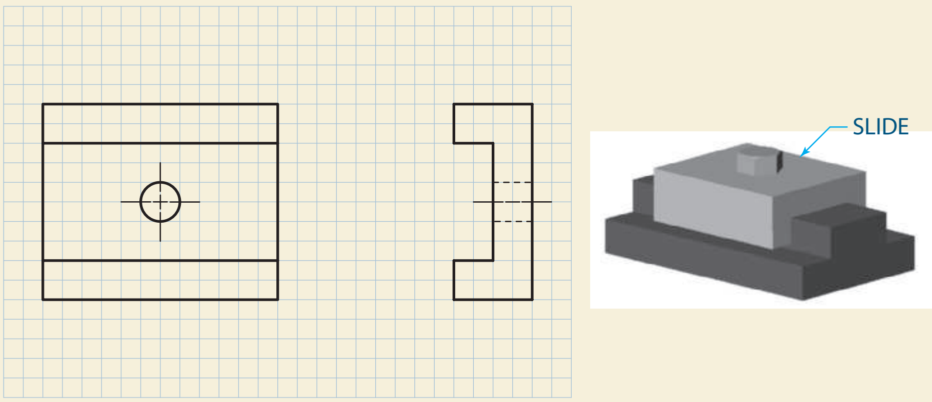

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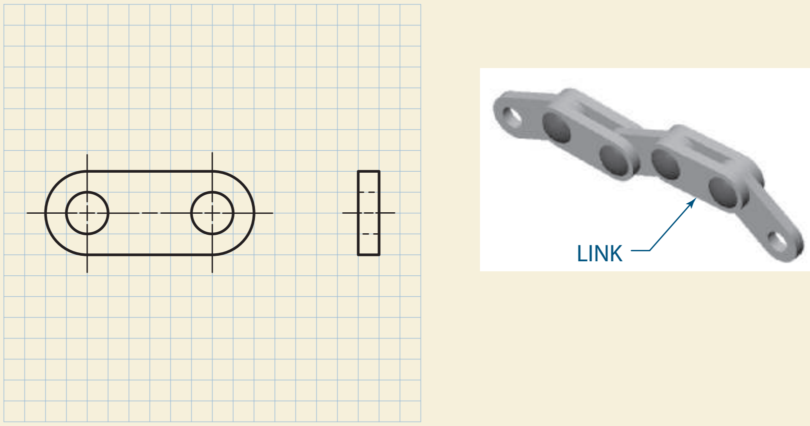

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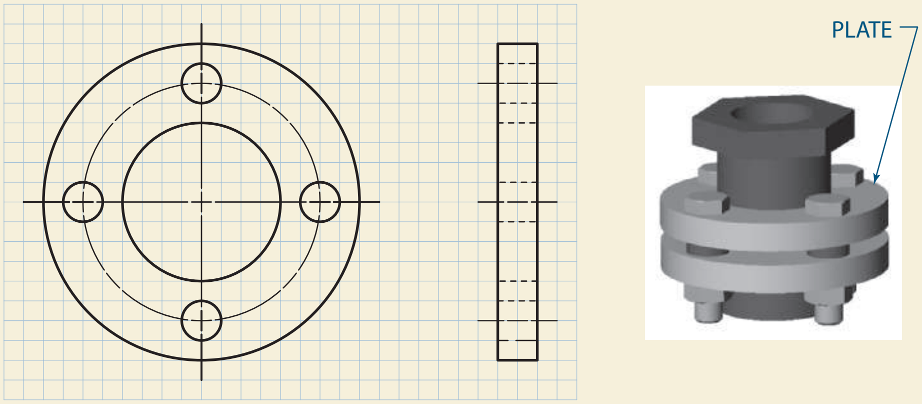

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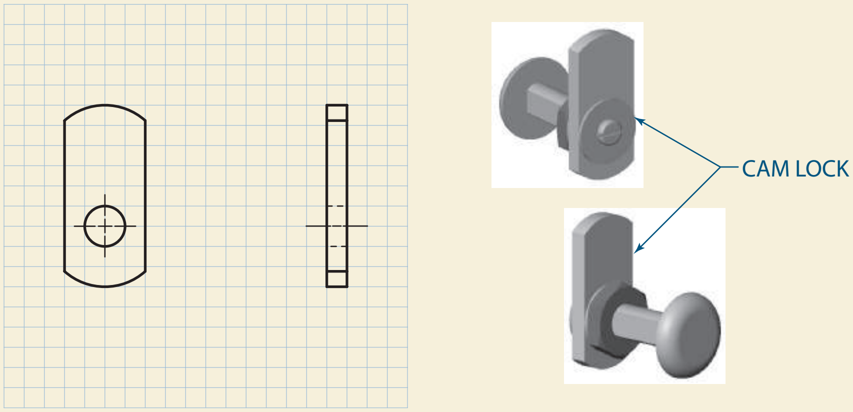

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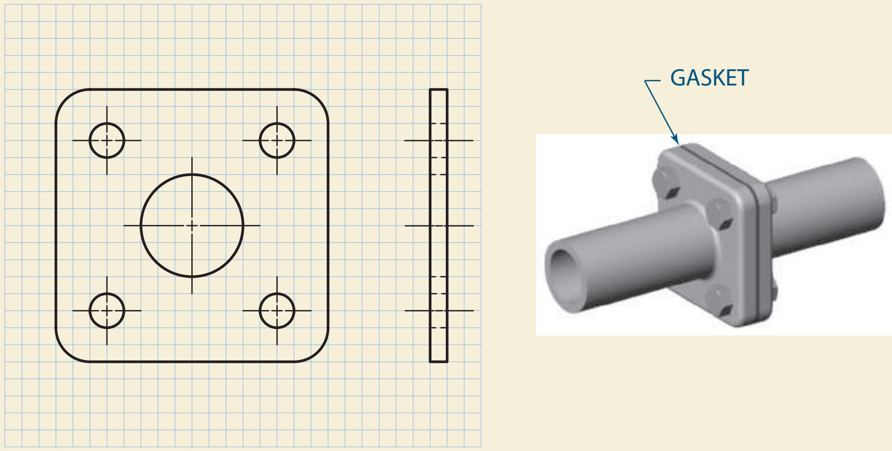
12.16 PROBLEMS (CONTINUED)



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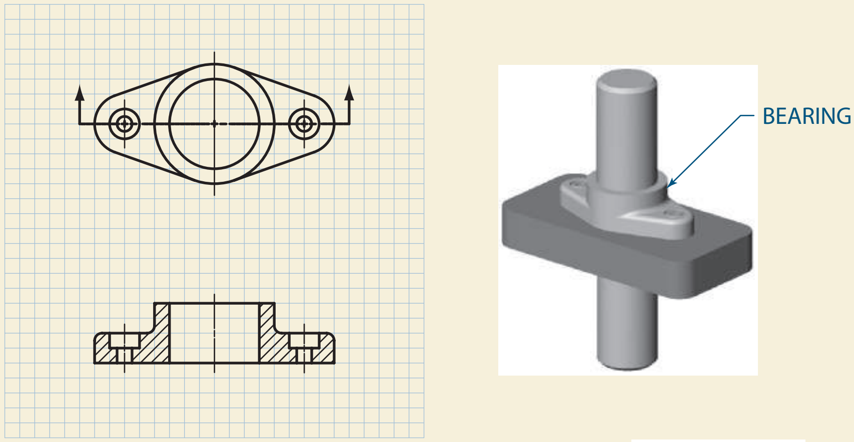

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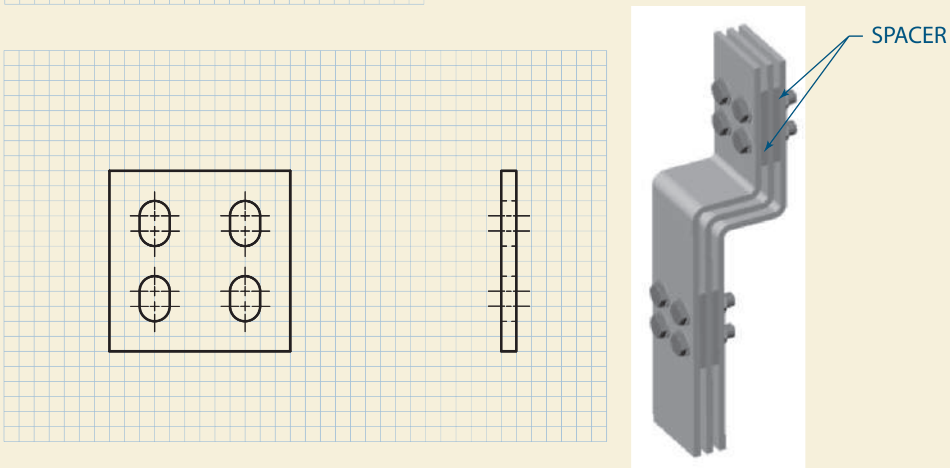

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FIGURE P12.4.

CHAPTER 13

TOLERANCING

OBJECTIVES

After completing this chapter, you should be able to

- Describe the purpose of conventional tolerancing and its limitations
- Use standard tables to specify an appropriate fit between two mating parts
- Explain the advantages of using geometric dimensioning and tolerancing (GD&T) over conventional tolerancing
- Recognize the datum reference frame on a drawing with geometric dimensions and tolerances
- Describe the tolerance zone shape for each geometric tolerance
- Correctly read the feature control frames on a drawing with geometric dimensions and tolerances

13.01 INTRODUCTION

In the previous chapter, you learned some of the basics for displaying dimensions correctly and, to some extent, how to select appropriate dimensions to describe the size and location of features on objects. This chapter will look at dimensioning objects for interchangeable manufacturing. **Interchangeable manufacturing** is the process by which parts are made at different locations and brought together for assembly. For many industries, this process enables third-party companies to produce replacement parts or custom parts.

The first topic in this chapter deals with the amount of tolerance required. In previous chapters, the tolerance of a dimension was defined as the total amount the tolerance could vary. Since it is impossible to make anything perfectly, design engineers must define a range of acceptable tolerance for manufacturing. If you specify a small value for a tolerance, such as .0001 mm, the machining cost will be high because of the required accuracy. As the tolerance value gets larger, the cost of fabrication usually gets smaller. If you are in the business of making children's toys from plastic materials, it is unlikely that you will be specifying tolerance values such as .0001 mm. If you are designing engine parts for space missions, you may need to require very small tolerances.

It is rare to find companies that manufacture products where all parts in the final assembly have been produced at the same location. It is more likely that parts for the final assembly are manufactured at different facilities around the world. By using standard practices for tolerance dimensioning, manufacturers can be confident that parts will fit together as intended. This is critical in the specification of parts that might be manufactured by subcontractors or by other divisions of a company.

13.01.01 Relationships between Different Parts

When doing engineering design work, it is rare to design parts that do not interact with other parts. In most cases, it is necessary to specify the intended fit between parts. Examine the WHEEL ASSEMBLY in Figure 13.01. The intent of the design is to have the WHEEL spin freely as it rolls against the ground. For this to happen, there must be a clearance of material, or space, between some of the parts in the assembly. In the case of the WHEEL ASSEMBLY, the intent is for the BUSHINGS to fit tightly into the WHEEL, the SHAFT to fit tightly within the side supports, and the BUSHINGS to spin freely around the SHAFT. The largest diameter of the SHAFT and the hole through the BUSHING are both about 3/4". To ensure that the BUSHINGS spin about the SHAFT, the designer must specify a size range for each part. Tolerance is the specific amount a particular dimension can vary. The SHAFT and BUSHING drawings appear in Figure 13.02. The size range for the largest diameter of the SHAFT is .7435–.7455, and the range for the hole in the BUSHING is .7500–.7535. If each part is machined within the stated size range, the BUSHING diameter will be larger than the SHAFT diameter and the BUSHING will be free to spin about the SHAFT.

13.01.02 Problems with Inexperience in New Engineers

One of the main obstacles for new engineers is their lack of experience. Until they gain some valuable experience on the job, they are likely to have a difficult time making all of the correct design decisions. Lack of experience can be a problem in several areas.

First, not knowing the history or function of a product can put young engineers at a disadvantage. For example, the WHEEL ASSEMBLY in Figure 13.01 has several intended fits. It is important for an engineer to know why certain fits exist between

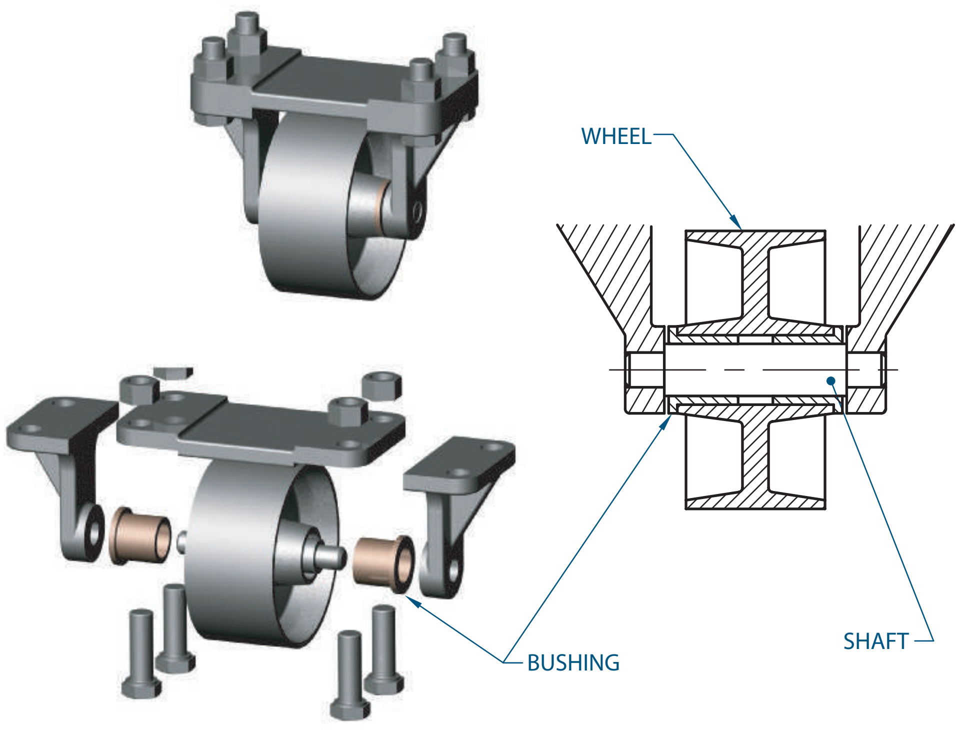

FIGURE 13.01. A WHEEL assembly.

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parts so the assembly will function properly. It is important for the engineer to know what materials will yield the best results within the assembly. Here are some other questions the engineer might ask:

- How will the parts be manufactured?
- Several parts in the WHEEL ASSEMBLY need to be cast. Does the company have a foundry?
- What type of machining operations can the company complete?
- What subcontractors does the company typically use?

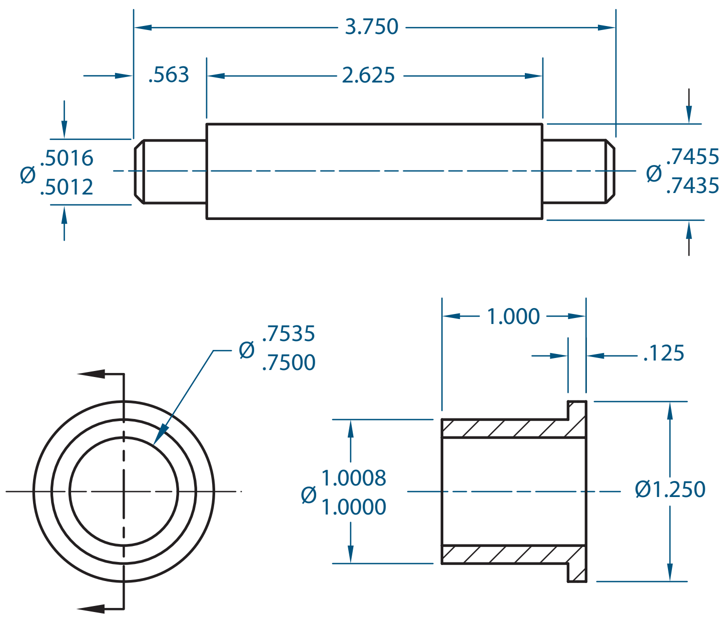

FIGURE 13.02. Detail drawings of the SHAFT and BUSHING.

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All of these issues concerning product history and function tend to put young engineers at a disadvantage.

Another area that can create problems for young engineers is applying inappropriate tolerance values to dimensions. Applying too small of a tolerance value might cause problems for a machinist. For example, if an engineer applies a toleranced dimension of .750000–.750035 to the BUSHING in Figure 13.02, the machinist might not be able to machine the hole with a tolerance that small. If the machinist can machine the hole with that small of a tolerance, the cost will probably be higher than that of a larger tolerance value. However, if the engineer applies a tolerance value that is too loose (e.g., .700–.750), the part may not function properly.

Inexperience also can be a problem when dealing with geometric dimensions and tolerances. As you will see later in the chapter, selecting an appropriate datum reference frame or coordinate system for geometric tolerancing is critical. Selecting appropriate dimensions is something that requires time and meaningful experiences on the job.

13.02 Formats for Tolerances

Tolerance dimensions can be displayed in several common formats: unilateral, bilateral, and limit dimensions. Figure 13.03 illustrates the differences between metric and inch conventions for displaying the number of decimal places. For each of these types of tolerance dimensions, a range is given for a specified basic size from which the limits were derived. For all of the metric examples in Figure 13.03, the basic size is 35. The basic size for the unilateral inch example is .500.

In a unilateral tolerance, all of the deviation is in one direction from the basic size. The tolerance is either all above or all below the basic size of the dimension. For metric dimensions, a single zero is shown without a plus or minus sign. When a designer is dimensioning in inches, the tolerance value is expressed with the same number of decimal places as the basic size and the appropriate plus or minus size is added.

Bilateral tolerances are tolerances where the deviation is divided in some way above and below the basic size of the dimension. The tolerance can be equally or unequally distributed about the basic size.

Limit dimensions are displayed with the high limit above the low limit. If the dimension is displayed on a single line, the low limit appears before the high limit.

FIGURE 13.03. Formats for tolerance dimensioning in millimeters and inches.

		METRIC	INCHES
UNILATERAL		$35^{+0.05}_{0}$ or $35^{0}_{-0.05}$	$.500^{+.005}_{-.000}$ *not* $.500^{+.005}_{0}$
BILATERAL	Equal	35 ±0.05 *not* 35.00 ±0.05	.750 ±.005 *not* .75 ±.005
	Unequal	$35^{+0.25}_{-0.10}$ *not* $35^{+0.25}_{-0.1}$	$1.000^{+.008}_{-.010}$ *not* $1.000^{+.008}_{-.01}$
LIMIT		35.05 / 35.00 *not* 35.05 / 35	.250 / .248 *not* .25 / .248

13.03 Tolerance Buildup Problems

As a designer working with tolerance dimensions, one thing you must consider is the buildup, or accumulation, of tolerances. When you are using tolerance dimensions, accumulation can occur in several ways.

13.03.01 Tolerance Buildup with Chain, Baseline, and Direct Dimensioning

Tolerance buildup, or accumulation, between features can be minimized depending on the type of dimensioning used. **Chain dimensioning** usually yields the largest accumulation of tolerance between features. The maximum variation or distance between features is equal to the sum of the intermediate distances. In Figure 13.04, the total tolerance accumulation between points X and Y is ±0.15. The distance between points X and Y is the sum of three dimensions: 20±0.05 + 25±0.05 + 25±0.05. If all three dimensions are machined to their maximum values, the result is 70.15. If they are machined to their minimum values, the result is 69.85.

Baseline dimensioning can eliminate some of the accumulation of tolerances. In this system, the maximum variation between two features is the sum of the tolerances on the two dimensions from their origin to the two features. In Figure 13.05, the feature at X is located from the baseline with a 25±0.05 dimension, and the feature at Y is located from the baseline with a 95±0.05 dimension. The tolerance buildup between the surfaces at X and Y is ±0.1. The distance between points X and Y is the difference between two dimensions: 95±0.05–25±0.05. If the 95±0.05 dimension is machined at its maximum value and the 25±0.05 dimension is machined at its minimum value, the result is 70.1. If the 95±0.05 dimension is machined at its minimum value and the 25±0.05 dimension is machined at its maximum value, the result is 69.9.

Direct dimensioning offers the best way to eliminate tolerance accumulation. This method involves placing a single dimension between two key points to minimize the tolerance accumulation. As shown in Figure 13.06, the total tolerance between features X and Y is only the tolerance on the one dimension between the two features. In this case, that tolerance is ±0.05.

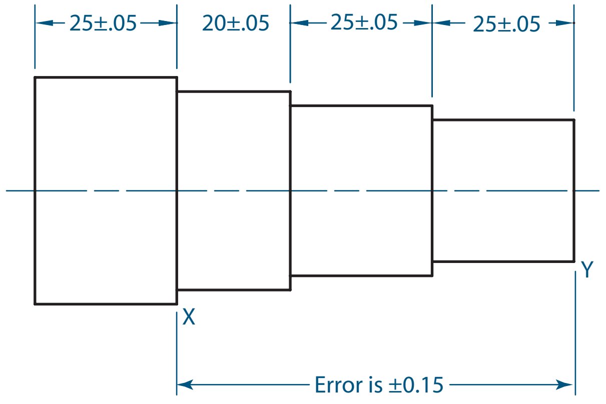


FIGURE 13.04. Tolerance accumulation with CHAIN dimensioning.

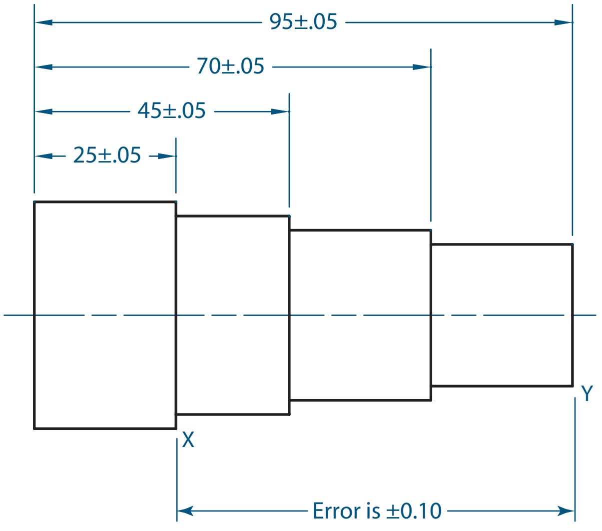
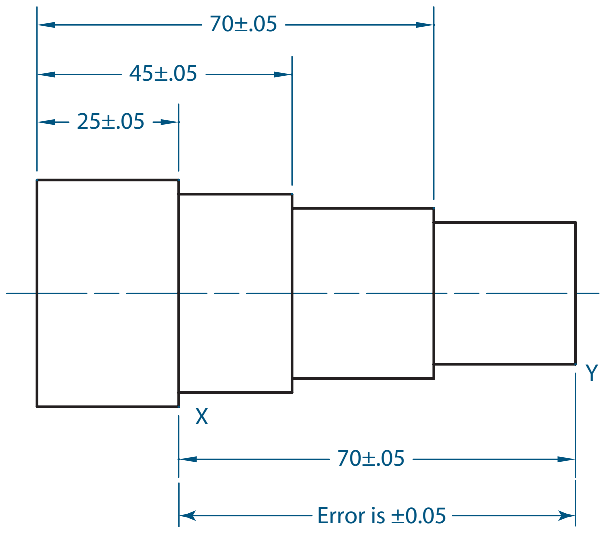


FIGURE 13.05. Tolerance accumulation with BASELINE dimensioning.

FIGURE 13.06. Tolerance accumulation with DIRECT dimensioning.



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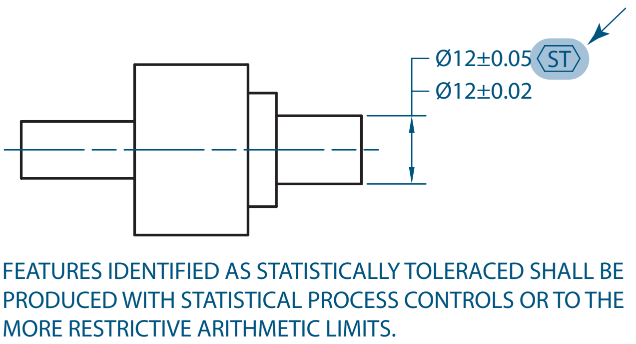
13.03.02 Statistical Tolerance Control

Statistical tolerancing is a way to assign tolerances based on sound statistical practices rather than conventional tolerancing practices. It can be applied only when appropriate statistical process control methods are used for manufacturing. When conventional tolerancing methods are used, often the total assembly tolerance is divided by the number of individual parts in the assembly. A portion of this assembly tolerance is then assigned to each component. The problem with this method is that it usually results in tolerance values being more restrictive than necessary. When manufacturing processes are monitored by statistical process controls, technicians and engineers are better informed about processes for which tolerance values can be increased to reduce manufacturing costs. Figure 13.07 illustrates an example where a statistical tolerance is given with an arithmetic tolerance. In this case, the 12 ±0.05 dimension is appropriate when statistical process controls are in place. When those controls are not in place, the 12 ±0.02 dimension is applied.

13.04 Use of Tables for Fits

As mentioned at the beginning of the chapter, the intent of a design requires that you accurately specify fits between mating parts. For just about every application, you will be defining the looseness or tightness of the fit. As you design parts, you will be specifying fits using standard tables and recognized types of fits.

FIGURE 13.07. Tolerancing with statistical process control.



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13.04.01 Types of Fits

Fits can be classified as one of the following:

- *Clearance fit*—Specifying the limits of size in such a way that a clearance or space always exists between mating parts. Figure 13.08 shows a machined hole with a limit dimension of 1.5000–1.5016 and a machined shaft with a limit dimension of 1.4990–1.4980. If both parts are machined within the stated limits of size, space will always exist between the two parts.
- *Interference fit*—Specifying the limits of size in such a way that an interference of material always exists between mating parts. Figure 13.09 shows a machined hole with a limit dimension of 1.5000–1.5006 and a machined shaft with a limit dimension of 1.5009–1.5013. If both parts are machined within the stated limits of size, material interference will always exist between the two parts.
- *Transition fit*—Specifying the limits of size in such a way that either a clearance or interference fit will exist when mating parts are assembled. Figure 13.10 shows a machined hole with a limit dimension of 1.5000–1.5012 and a machined shaft with a limit dimension of 1.5008–1.5015. If the hole is machined at its upper limit (1.5012) and the shaft is machined at its lower limit (1.5008), the result will be a clearance fit. On the other hand, if the hole is machined at its lower limit (1.5000) and the shaft is machined at its upper limit (1.5015), the result will be an interference fit.

You may be wondering why anyone would specify such a fit. It is almost as if the person cannot make up his or her mind about the type of fit that is necessary. Transition fits are typically associated with selective assembly. Selective assembly involves measuring parts after they are machined and matching them up with appropriate mating parts. Manufacturing parts to tight or small tolerances is expensive. With transition fits, tolerances can be "opened up," or made larger, so that manufacturing the individual parts is less expensive.

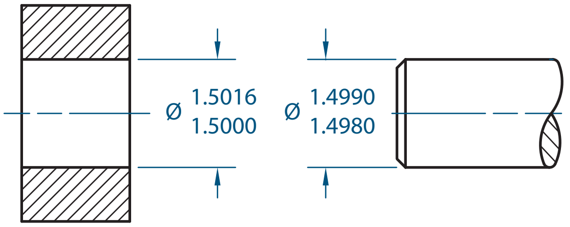

FIGURE 13.08. Specifying a CLEARANCE FIT with limit dimensioning.

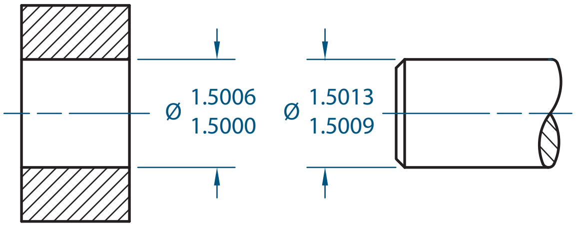

FIGURE 13.09. Specifying an INTERFERENCE FIT with limit dimensioning.

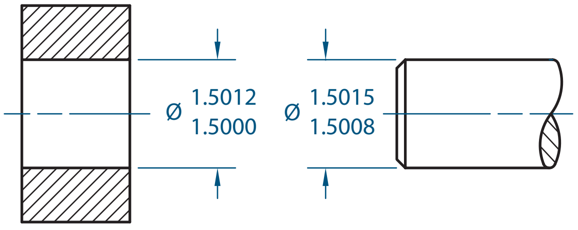

FIGURE 13.10. Specifying a TRANSITION FIT with limit dimensioning.

13.04.02 Fit Terminology

You need to be familiar with some terms as you read standard fit tables or specify fits between parts. Figures 13.11 and 13.12 illustrate some of these terms.

- ***Allowance***—Allowance is the difference between the maximum material limits of mating parts. It is the minimum clearance or maximum interference between parts. To calculate allowance, subtract the upper limit of the shaft dimension (largest cylinder) from the lower limit of the hole dimension (smallest hole). In Figure 13.8, the allowance is 1.5000 – 1.4990 = .0010.
- *Tolerance*—Tolerance is the total permissible variation of a size. It is the difference between the upper limit and the lower limit.
- *Basic size*—The basic size is the size from which the limit dimensions were derived. The basic size of the parts in Figures 16.8 through 16.10 is 1.500.
- ***Clearance***—Clearance refers to a fit where there is space between the two mating parts. The intent is that when assembled, the shaft will spin within the hole (see Figure 13.11).
- ***Interference***—Interference is a fit where the two mating parts have intersecting nominal volumes, requiring the deformation of the parts. For example, the diameter of the shaft is larger than the diameter of the hole. When assembled, the intent is that the shaft will not spin in the hole (see Figure 13.12).
- *Hole basis or basic hole system*—In this system, the basic size is applied to the lower limit of the hole. This system is used quite often since standard tools such as reamers and broaches are designed to machine holes no less than a particular size. The shaft can then be machined to create the desired type of fit (see Figure 13.13).

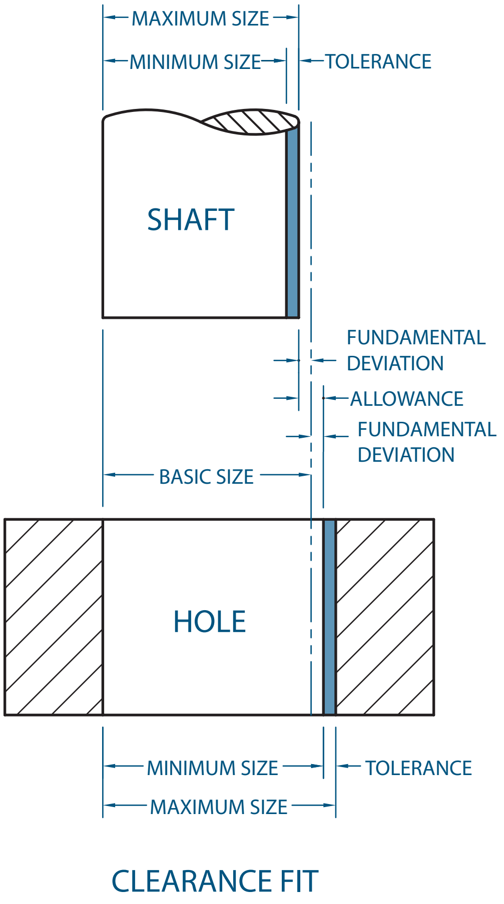

FIGURE 13.11. Clearance fit terminology.

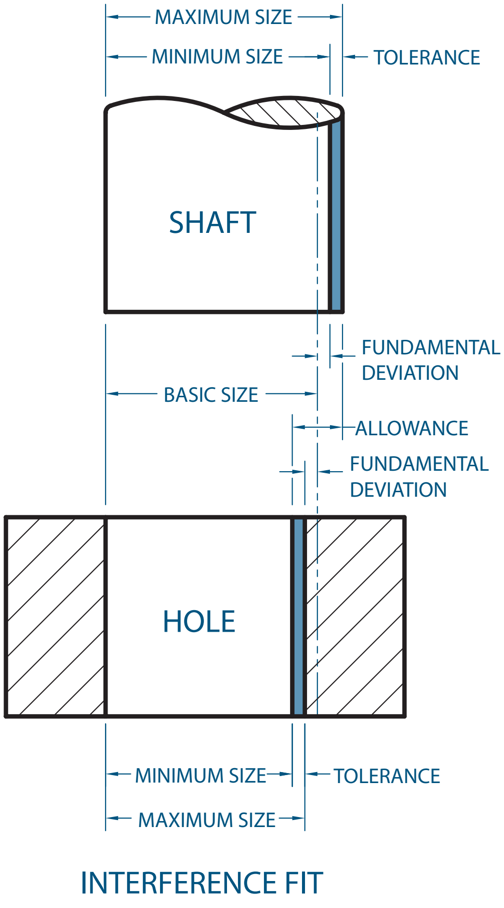

FIGURE 13.12. Interference fit terminology.

FIGURE 13.13. A basic hole system.

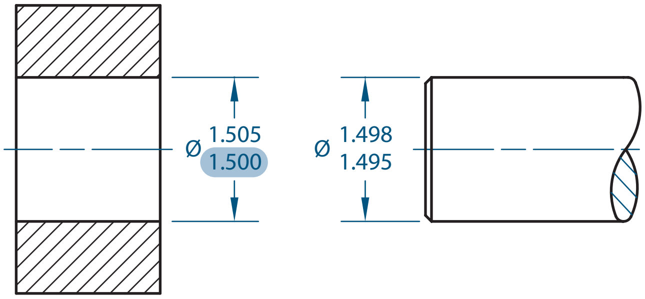

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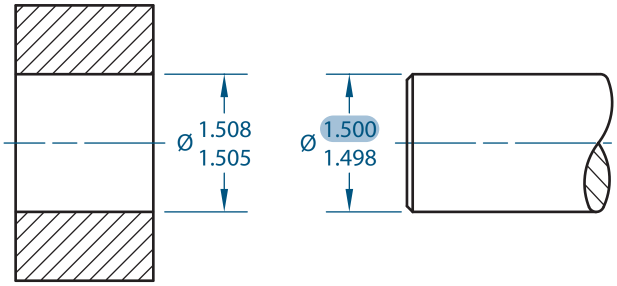
FIGURE 13.14. A basic shaft system.



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- *Shaft basis or basic shaft system*—In this system, the basic size is applied to the upper limit of the shaft. The hole is then machined to create the desired type of fit. This is used when several parts with different fits are required to fit on a particular shaft (see Figure 13.14).

13.04.03 English Fits

There are five types of fits within the English, or inch, system. These fits, for which the clearance or interference are depicted graphically in Figures 16.15 through 16.18, are established as a starting point for determining appropriate fits between mating parts [ANSI B4.1 – 1967 (R1994)]:

- RC—*Running or sliding clearance fit*—These fits provide a similar running performance, with suitable lubrication allowance, throughout the range of sizes. The clearances for the first two classes (RC1 and RC2), used chiefly as slide fits, increase more slowly with the diameter than the other two classes do; thus, accurate location is maintained even at the expense of free relative motion.
- LC—*Locational clearance fit*—These fits are intended for parts that are normally stationary but can be freely assembled or disassembled. They range from snug fits for parts requiring accuracy of location to medium clearance fits for parts such as spigots to looser fastener fits where freedom of assembly is of prime importance.
- LT—*Locational transition fit*—These fits are intended where accuracy of location is important but a small amount of clearance or interference is permissible. They are a compromise between clearance and interference fits.
- LN—*Locational interference fit*—These fits are intended where accuracy of location is of prime importance and where parts require rigidity and alignment with no special requirement for bore pressure. Such fits are not intended for parts designed to transmit frictional loads from one part to another by virtue of the tightness of fit, as these conditions are covered by force fits.
- FN—*Force or shrink fit*—These types of interference fits are usually characterized by maintenance of constant bore pressures throughout the range of sizes. Therefore, the interference varies almost directly with diameter and the difference between its minimum and maximum value is small to maintain the resulting pressures within reasonable limits.

FIGURE 13.15. Running and sliding fits.

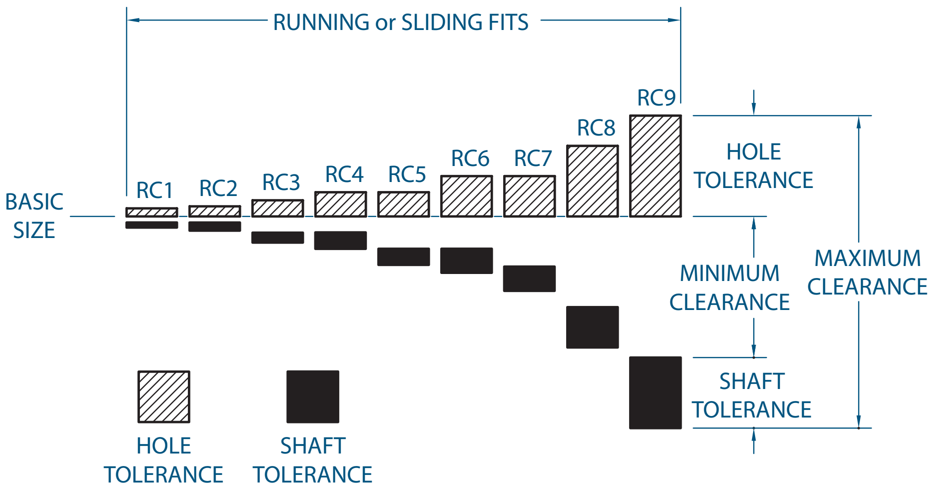

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FIGURE 13.16. Locational clearance fits.

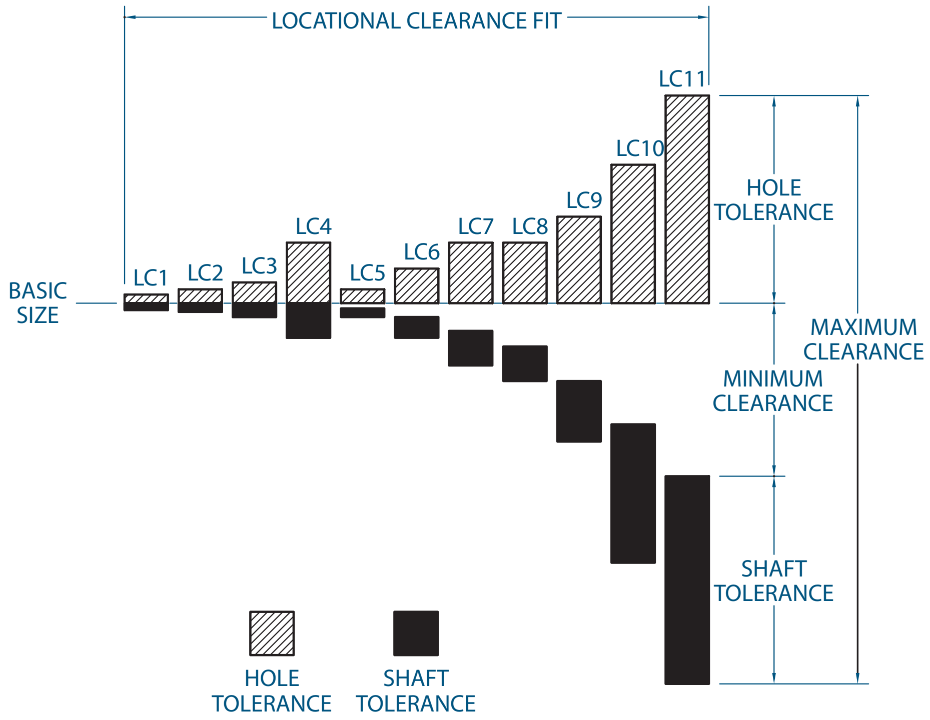

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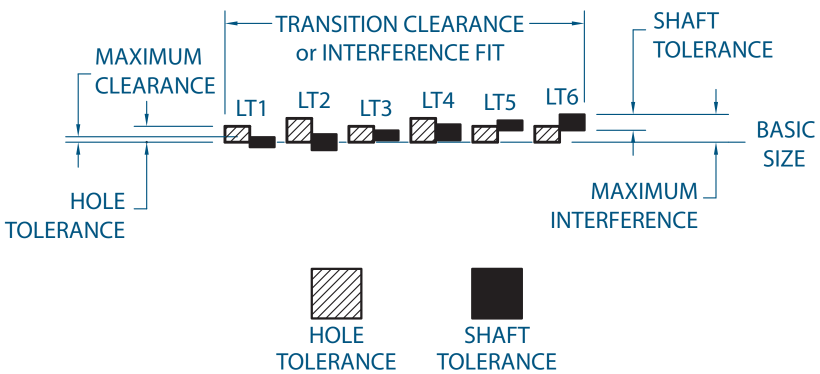
FIGURE 13.17. Locational transition fits.



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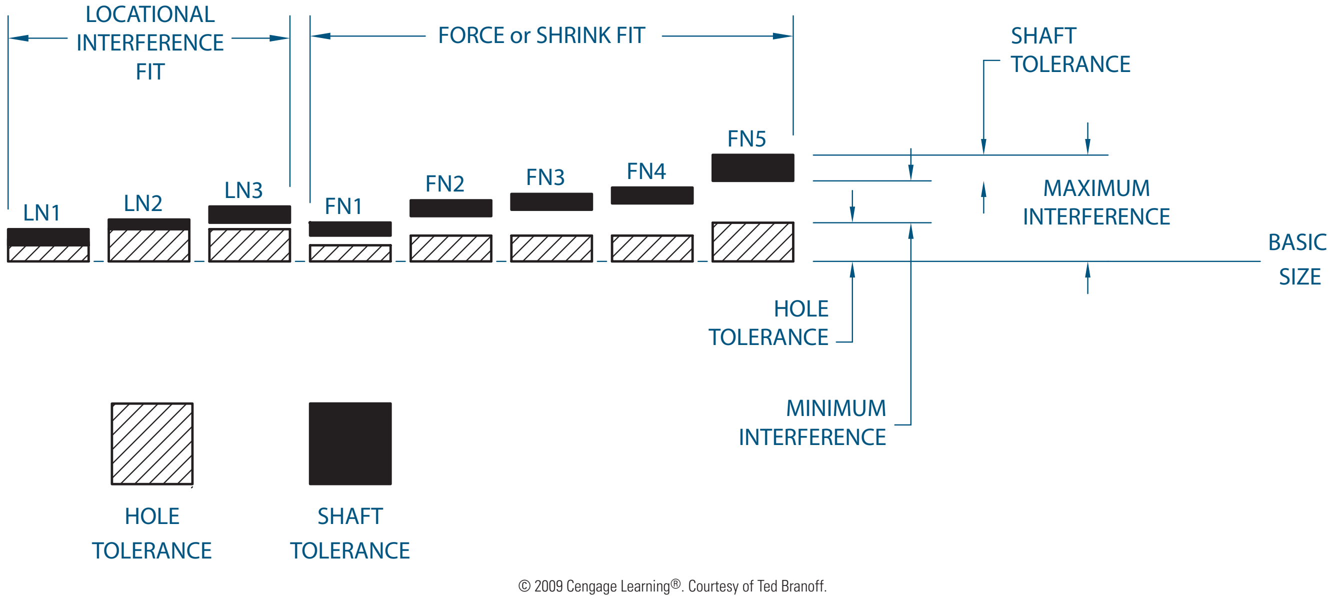

FIGURE 13.18. Locational interference and force fits.

13.04.04 Metric Fits

There are nine types of fits within the metric system. These fits are established as a starting point for determining appropriate fits between mating parts. Figure 13.19 shows the symbol designations for hole basis and shaft basis fits. The clearance and interference for these fits are depicted graphically in Figures 13.20 through 13.21.

- *Loose running fit*—These fits are for wide commercial tolerances or allowances on external members.
- *Free running fit*—These fits are not for use where accuracy is essential, but are good for large temperature variations, high running speeds, and heavy journal pressures.
- *Close running fit*—These fits are for running on accurate machines and for ensuring accurate location at moderate speeds and journal pressures.
- *Sliding fit*—These fits are not intended to run freely, but are intended to move and turn freely and locate accurately.
- *Locational clearance fit*—These fits provide snug fit for locating stationary parts; they can be freely assembled and disassembled.
- *Locational transition fit*—These fits provide for accurate location and a compromise between clearance and interference.
- *Locational interference fit*—These fits are for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements.
- *Medium drive fit*—These fits are for ordinary steel parts or shrink fits on light sections—the tightest fit usable with cast iron.
- *Force fit*—These fits are suitable for parts that can be highly stressed or for shrink fits where the heavy pressure forces required are impractical.

FIGURE 13.19. Metric fit table.

	ISO SYMBOL		DESCRIPTION	
	Hole Basis	Shaft Basis		
Clearance Fits	H11/c11	C11/h11	Loose Running	More Clearance
	H9/d9	D9/h9	Free Running	
	H8/f7	F8/h7	Close Running	
	H7/g6	G7/h6	Sliding	
	H7/h6	H7/h6	Locational Clearance	
Transition Fits	H7/k6	K7/h6	Locational Transition	
	H7/n6	N7/h6	Locational Transition	
Interference Fits	H7/p6	P7/h6	Locational Interference	More Interference
	H7/s6	S7/h6	Medium Drive	
	H7/u6	U7/h6	Force	

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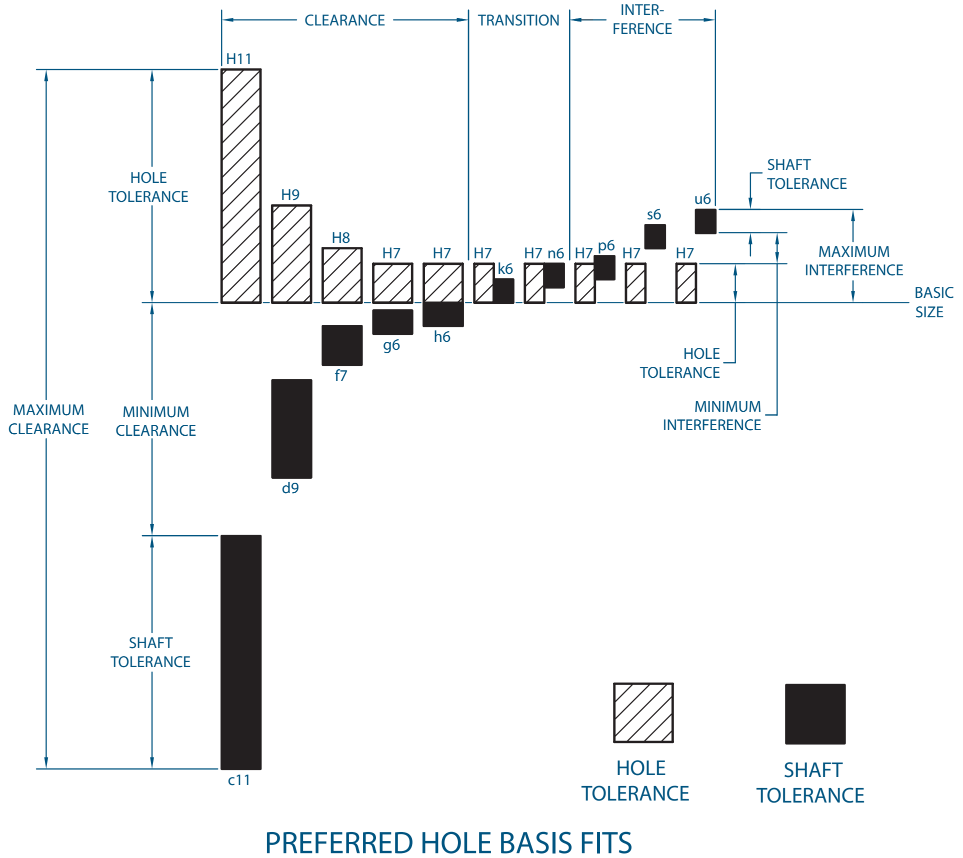
FIGURE 13.20. Metric hole basis fits.



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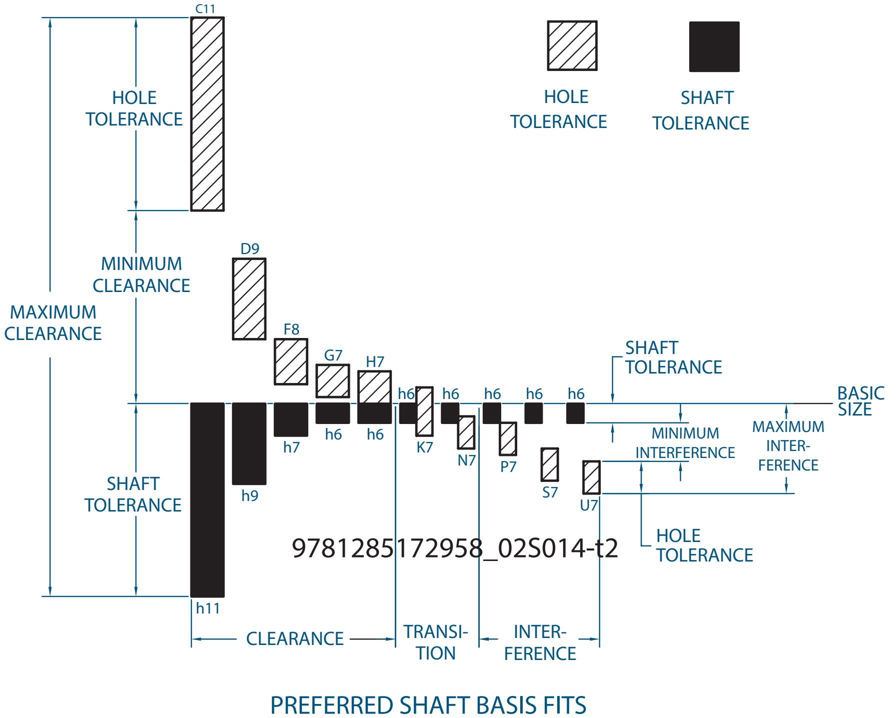


FIGURE 13.21. Metric shaft basis fits.

13.04.05 Fits Tables

Figure 13.22 shows a standard table for fits in the English system of units. When specifying an inch, or English, fit between a hole and a shaft from a standard table, use the following guidelines:

- Determine the type of fit appropriate for the design and locate the corresponding table.
- Determine the basic size of the parts.
- Find the size range on the table.
- Determine the tolerances for the hole and the shaft.
- Remember that values on the English tables are in *thousandths* of an inch.

Refer to the following example to practice looking up a fit in the inch system. For this application, a *close sliding fit* is appropriate (RC1 fit) and the basic size for the parts is 1.500. On the table (see Figure 13.22), the nominal size of 1.500 falls between 1.19–1.97. The limits on the hole are –0 to +0.4. The limits on the shaft are –0.4 and –0.7. One of the most common mistakes when working with inch tables is forgetting that these limits are in *thousandths* of an inch. The upper tolerance on the hole is really +0.0004. The values for the shaft are really –0.0004 and –0.0007. When these values from the table are added or subtracted from the basic size, the results are the dimensions shown in Figure 13.23.

Figure 13.24 shows a standard table for fits in the metric system of units. Specifying fits from ISO, or metric, tables is a little easier. The tables provide the direct values, so there is no need to add or subtract. Follow these guidelines to determine metric fits:

- Determine the type of fit appropriate for the design and locate the corresponding table.
- Determine the basic size of the parts.
- Find the size range on the table.
- Determine the tolerances for the hole and the shaft.

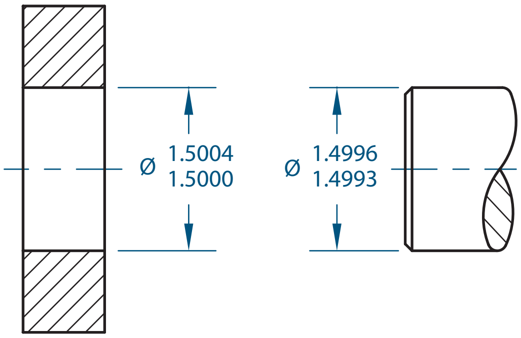
To practice looking up a metric fit, you will use a *loose running fit* with a basic size of 25. In Figure 13.24, find 25 on the left side of the table. Look to the right under the Hole and Shaft columns. The result is the dimensions shown in Figure 13.25.

FIGURE 13.22. American National Standard running and sliding fits.

Nominal Size Range, Inches Over To	Class RC 1 Limits of Clearance	Class RC 1 Standard Limits Hole H5	Class RC 1 Standard Limits Shaft g4	Class RC 2 Limits of Clearance	Class RC 2 Standard Limits Hole H6	Class RC 2 Standard Limits Shaft g5	Class RC 3 Limits of Clearance	Class RC 3 Standard Limits Hole H7	Class RC 3 Standard Limits Shaft f6	Class RC 4 Limits of Clearance	Class RC 4 Standard Limits Hole H8	Class RC 4 Standard Limits Shaft f7
0-0.12	0.1 0.45	+0.2 −0	−0.1 −0.25	0.1 0.55	+0.25 −0	−0.1 −0.3	0.3 0.95	+0.4 −0	−0.3 −0.55	0.3 1.3	+0.6 −0	−0.3 −0.07
0.12-0.24	0.15 0.5	+0.2 −0	−0.15 −0.3	0.15 0.65	+0.3 −0	−0.15 −0.35	0.4 1.12	+0.5 −0	−0.4 −0.7	0.4 1.6	+0.7 −0	−0.4 −0.9
0.24-0.40	0.2 0.6	+0.25 −0	−0.2 −0.35	0.2 0.85	+0.4 −0	−0.2 −0.45	0.5 1.5	+0.6 −0	−0.5 −0.9	0.5 2.0	+0.9 −0	−0.5 −1.1
0.40-0.71	0.25 0.75	+0.3 −0	−0.25 −0.45	0.25 0.95	+0.4 −0	−0.25 −0.55	0.6 1.7	+0.7 −0	−0.6 −1.0	0.6 2.3	+1.0 −0	−0.6 −1.3
0.71-1.19	0.3 0.95	+0.4 −0	−0.3 −0.55	0.3 1.2	+0.5 −0	−0.3 −0.7	0.8 2.1	+0.8 −0	−0.8 −1.3	0.8 2.8	+1.2 −0	−0.8 −1.6
1.19-1.97	0.4 1.1	+0.4 −0	−0.4 −0.7	0.4 1.4	+0.6 −0	−0.4 −0.8	1.0 2.6	+1.0 −0	−1.0 −1.6	1.0 3.6	+1.6 −0	−1.0 −2.0
1.97-3.15	0.4 1.2	+0.5 −0	−0.4 −0.7	0.4 1.6	+0.7 −0	−0.4 −0.9	1.2 3.1	+1.2 −0	−1.2 −1.9	1.2 4.2	+1.8 −0	−1.2 −2.4
3.15-4.73	0.5 1.5	+0.6 −0	−0.5 −0.9	0.5 2.0	+0.9 −0	−0.5 −1.1	1.4 3.7	+1.4 −0	−1.4 −2.3	1.4 5.0	+2.2 −0	−0.4 −2.8
4.73-7.09	0.6 1.8	+0.7 −0	−0.6 −1.1	0.6 2.3	+1.0 −0	−0.6 −1.3	1.6 4.2	+1.6 −0	−1.6 −2.6	1.6 5.7	+2.5 −0	−1.6 −3.2
7.09-9.85	0.6 2.0	+0.8 −0	−0.6 −1.2	0.6 2.6	+1.2 −0	−0.6 −1.4	2.0 5.0	+1.8 −0	−2.0 −3.2	2.0 6.6	+2.8 −0	−2.0 −3.8
9.85-12.41	0.8 2.3	+0.9 −0	−0.8 −1.4	0.8 2.9	+1.2 −0	−0.8 −1.7	2.5 5.7	+2.0 −0	−2.5 −3.7	2.5 7.5	+3.0 −0	−2.5 −4.5
12.41- 15.75	1.0 2.7	+1.0 −0	−1.0 −1.7	1.0 3.4	+1.4 −0	−1.0 −2.0	3.0 6.6	+2.2 −0	−3.0 −4.4	3.0 8.7	+3.5 −0	−3.0 −5.2

* From ANSI B4.1 – 1967 (R1994). For larger diameters, see the standard.

FIGURE 13.23. A close sliding fit.



Preferred Metric Hole Basis Clearance Fits – American National Standard

Dimensions are in millimeters.

Basic Size		Loose Running Hole H11	Loose Running Shaft c11	Loose Running Fit	Free Running Hole H9	Free Running Shaft d9	Free Running Fit	Close Running Hole H8	Close Running Shaft f7	Close Running Fit
1	Max	1.060	0.940	0.180	1.025	0.980	0.070	1.014	0.994	0.030
	Min	1.060	0.880	0.060	1.000	0.955	0.020	1.000	0.984	0.006
20	Max	20.130	19.890	0.370	20.052	19.935	0.169	20.033	19.980	0.074
	Min	20.000	19.760	0.110	20.000	19.883	0.065	20.000	19.959	0.020
25	Max	25.130	24.890	0.370	25.052	24.935	0.169	25.033	24.980	0.074
	Min	25.000	24.760	0.110	25.000	24.833	0.065	25.000	24.959	0.020

From ANSI B4.2 – 1978 (R1984). For larger diameters, see the standard.

FIGURE 13.24. Metric hole basis clearance fits table.

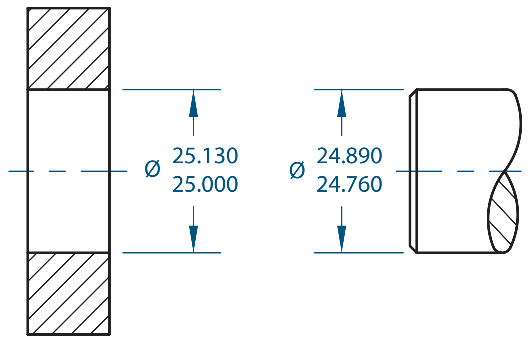


FIGURE 13.25. A loose running fit.

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13.05 Conventional Tolerancing versus Geometric Tolerancing

Before you move on to geometric dimensioning and tolerancing (GD&T), it is important to understand some of the limitations of conventional tolerancing—the limit or plus/minus system of tolerancing. To begin, you will look at some terminology that is used throughout the rest of the chapter.

13.05.01 Features With and Without Size

Professionals who deal with engineering parts use a specific language, especially concerning dimensioning and tolerancing. When talking about drawings and 3-D models, they must be able to identify features with size and features without size (see Figure 13.26). A feature is a general term that applies to an actual portion of a part, such as a surface, pin, tab, hole, or slot. A **feature with size** is a cylindrical or spherical surface or a set of two opposed elements or opposed parallel surfaces associated with a size dimension. The 40 mm wide slot in Figure 13.27 is a feature of size. The feature contains two equal and opposing parallel surfaces. Notice how the normal vectors from each surface point in opposite directions. The feature defined by the 30 mm dimension is not a feature of size. Even though the normal vectors are opposing, the surfaces are not of equal size.

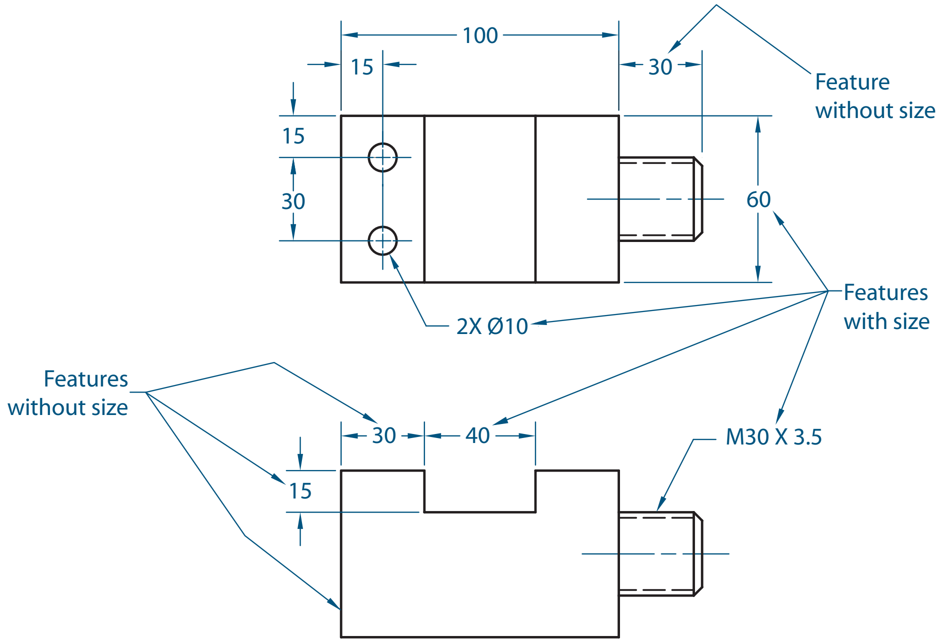

FIGURE 13.26. Features with and without size.

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A **feature without size** is typically a planar surface or a feature where the normal vectors point in the same direction. The feature defined by the 30 mm dimension in Figure 13.28 is not a feature of size since the normal vectors of the two surfaces point in the same direction.

13.05.02 Conventional Tolerancing and Form

Sometimes it is difficult to think about imperfect parts when all of the models and drawings that you generate have sharp corners and flat surfaces like those in Figure 13.29. When parts are finally produced, imperfections exist, even if you cannot see them with the naked eye. Surfaces may be wavy or bumpy, may have dips in them, or may be angled slightly (see Figure 13.30). If you use conventional tolerancing to control form, you must understand the extent to which this works. Where only a conventional tolerance of size is specified, the limits of size of an individual feature prescribe the extent to which variations in its geometric form, as well as size, are allowed (Rule 1). Figure 13.29 shows a rectangular block with two size dimensions. When you are using conventional tolerancing, Rule 1 states that the actual size of the object must be within the boundaries defined by the limit dimensions at *each cross section*. What this means is that conventional tolerancing is a 2-D system. When the rectangular block in Figure 13.30 is inspected, it must be checked at the front, at the back, and at all cross sections. Since the drawing does not

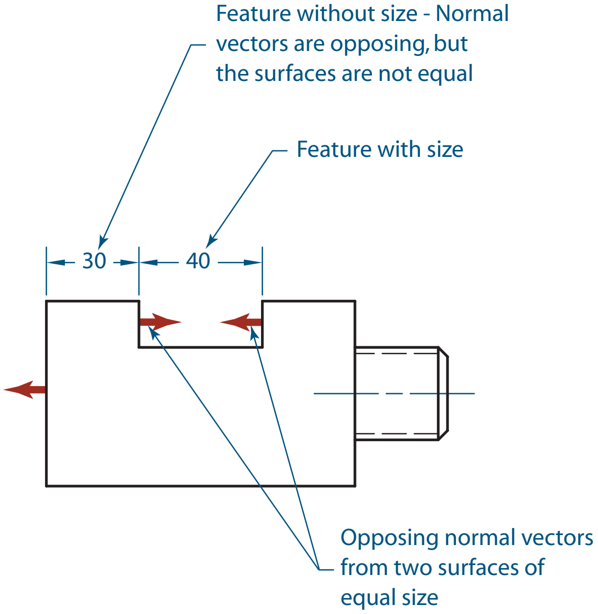


FIGURE 13.27. Features with and without size.

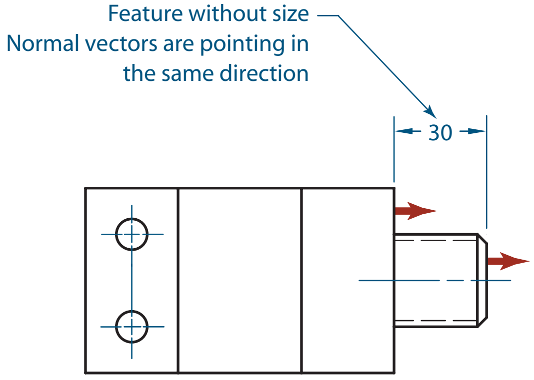
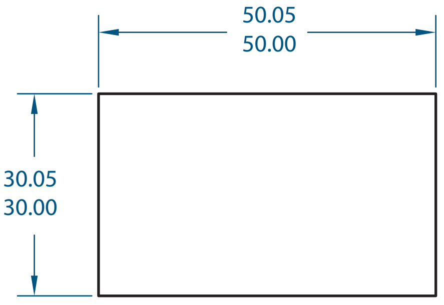


FIGURE 13.28. A feature without size.

FIGURE 13.29. Conventional tolerance dimensioning of a block.



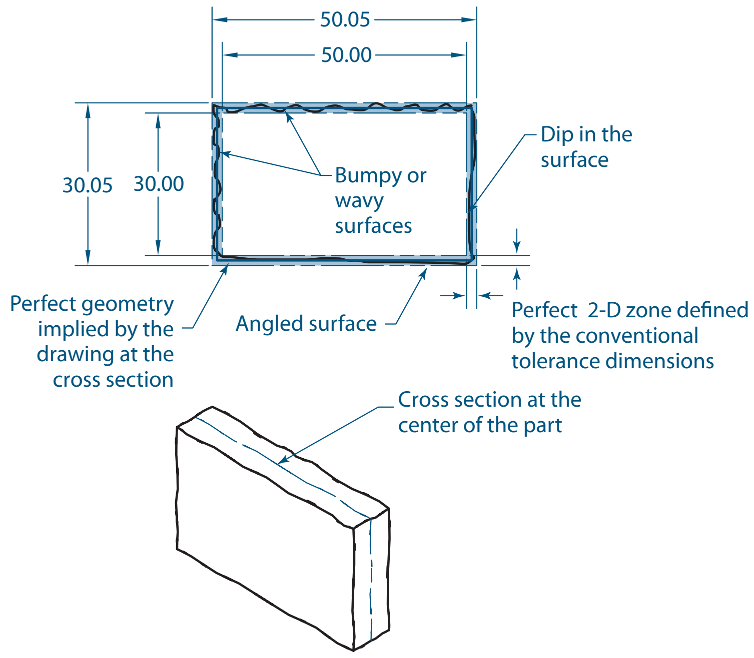


FIGURE 13.30. How conventional tolerancing controls surfaces.

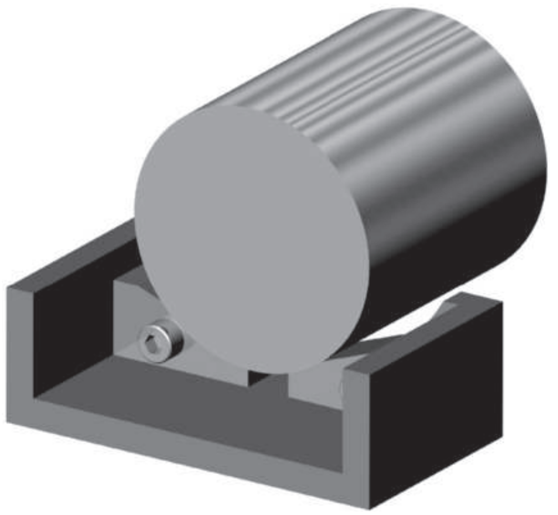
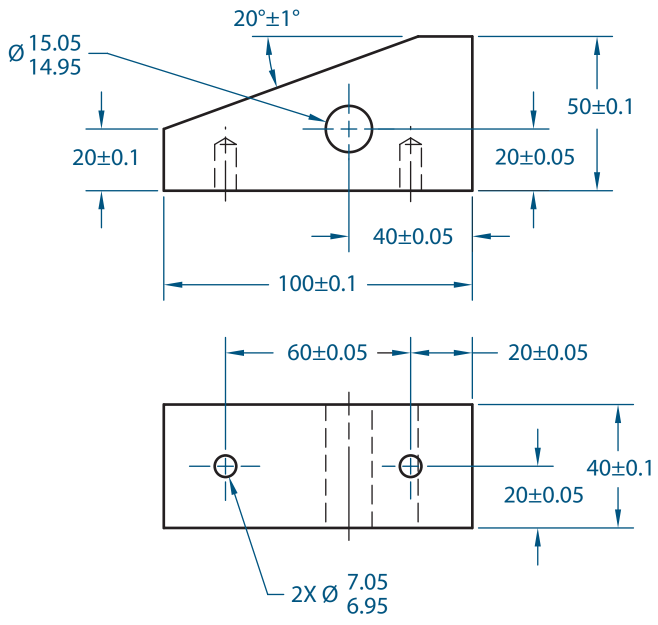


FIGURE 13.31. A WEDGE BLOCK assembly.

specify where dimensions originate, two inspectors may measure the part differently. This could lead to accepting parts that do not function or rejecting parts that actually work.

Examine the WEDGE BLOCK assembly in Figure 13.31. The purpose of the blocks is to keep the cylinder centered. The blocks mate against three surfaces and have three holes machined in them (see Figure 13.32). When the dimensions that describe the outside shape of the part are inspected, measurements can be taken at each cross-sectional area moving from the front of the part to the back. A measurement taken from the front of the part may yield a different result than

FIGURE 13.32. A WEDGE BLOCK drawing with conventional tolerancing.



one taken at the back of the part (see the imperfect part in Figure 13.33). Like the rectangular block in Figure 13.29, an inspector might accept parts that do not actually work in the assembly or reject parts that would work.

13.05.03 Location of Holes and Pins with Conventional Tolerancing

Another potential problem occurs when holes are located using conventional tolerance dimensions. Examine the two parts in Figure 13.34. The parts are designed so that there will be a clearance fit between the pins and the holes. The location dimensions for the holes (1.500 ±.006) will yield square tolerance zones for each part (see Figure 13.35). Now you will take a closer look at some examples where the holes and pins fall at the extreme edges of their tolerance zones.

FIGURE 13.33. Imperfect geometry of the WEDGE BLOCK.

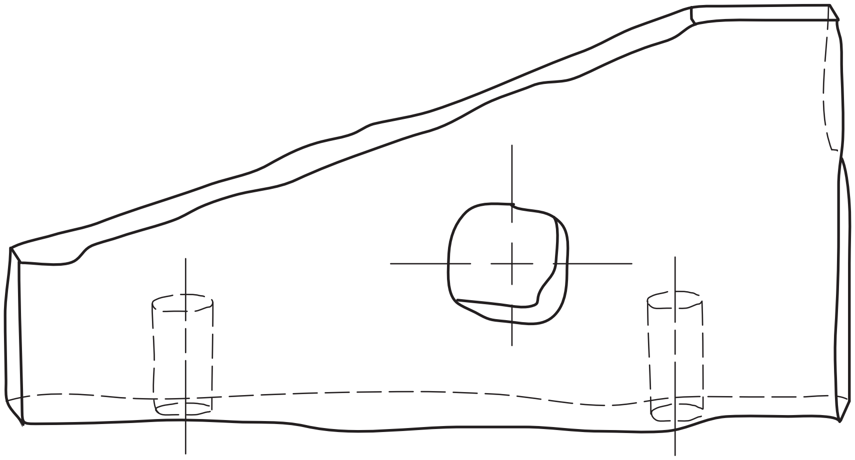


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FIGURE 13.34. Conventional tolerancing when locating holes.

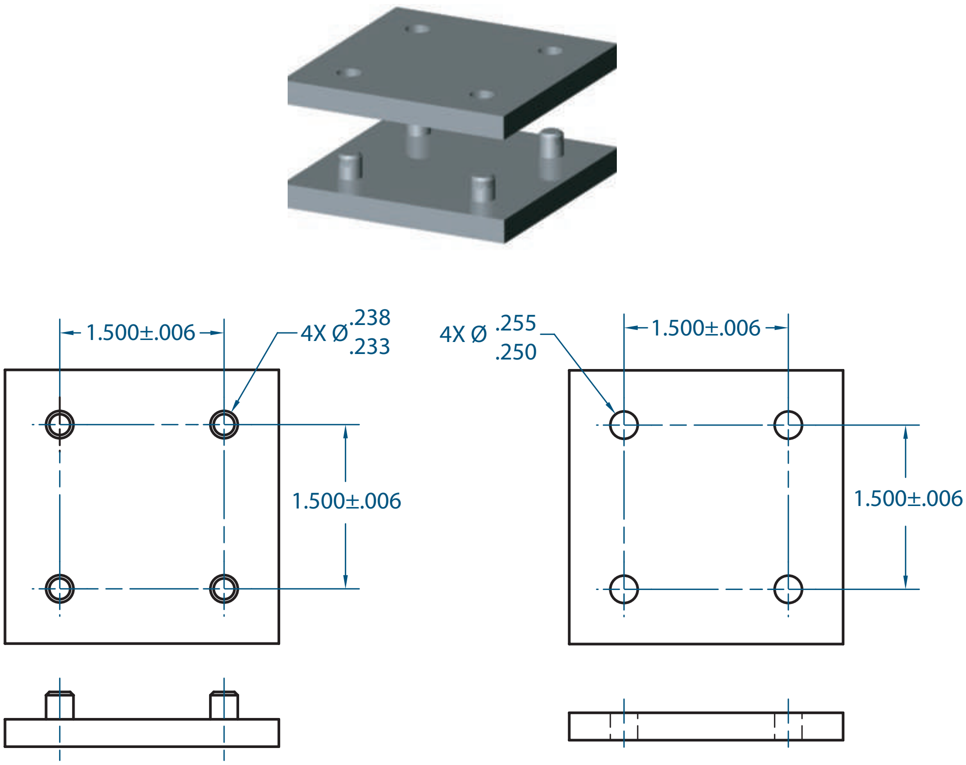

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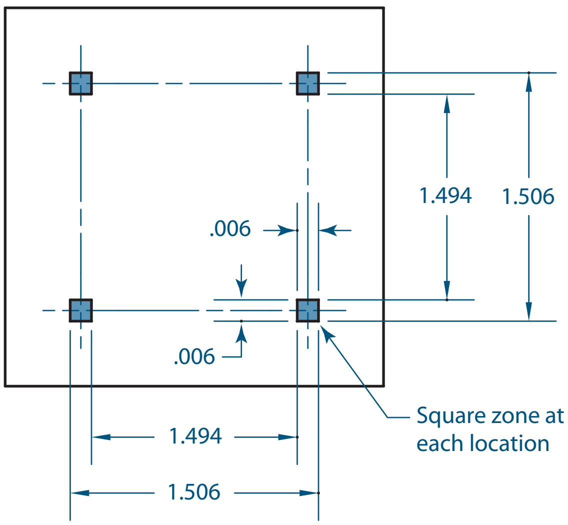
FIGURE 13.35. Square tolerance zones from conventional tolerancing.



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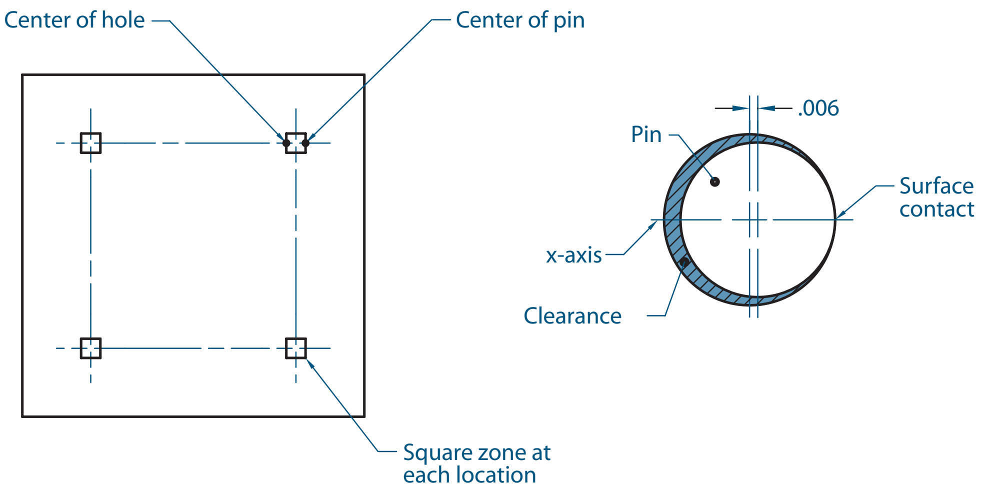
When a pin is machined to the far right of the square zone and the hole is machined to the far left of the square zone, the result is surface contact between the two parts (see Figure 13.36). This is acceptable for the assembly since the parts will still function under these conditions. Surface contact is also the result when the two features are located at the extreme ends of the *y*-axis (see Figure 13.37). Again, this is acceptable. The parts will still work.

A problem occurs when the center of the hole and the center of the pin are located at the extreme diagonals within the tolerance zone (see Figure 13.38). In this situation, material interference will result, which will not work for the assembly. The parts will not fit or function properly. You may be thinking that all that needs to be done to make things work is to specify a larger hole size or a smaller pin size. Usually, this is not a good idea since tolerances between holes and pins are taken from standard tables. The only other way to correct this problem when using limit dimensions to locate the center of holes is to reduce the size of the square tolerance zone. Any time you need to reduce a tolerance value, it usually costs more to produce the parts. If you stay with the same values for the diameters of the holes and pins and continue to use conventional tolerancing to locate the centers, you must reduce the tolerances on the location dimensions from ±.006 to ±.004. The result is that you always pass parts that will function. Unfortunately, you also reject

FIGURE 13.36. A pin and a hole at the extreme horizontal positions of the square tolerance zone.



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parts that will function properly. When a hole is drilled at position 1 in Figure 13.39 within the ±.004 square zone, the part passes inspection and works properly. A hole drilled at position 2 falls outside the ±.004 square zone, so it fails inspection. You know from Figures 13.36 and 13.37 that the parts will function in this situation. You will come back to these parts after some discussion about GD&T.

FIGURE 13.37. A pin and a hole at the extreme vertical positions of the square tolerance zone.

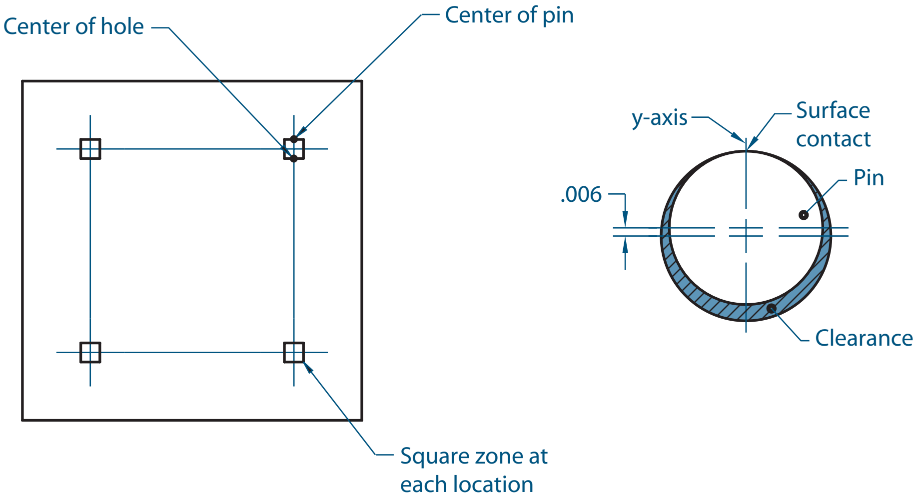

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FIGURE 13.38. A pin and a hole at the extreme diagonal positions of the square tolerance zone.

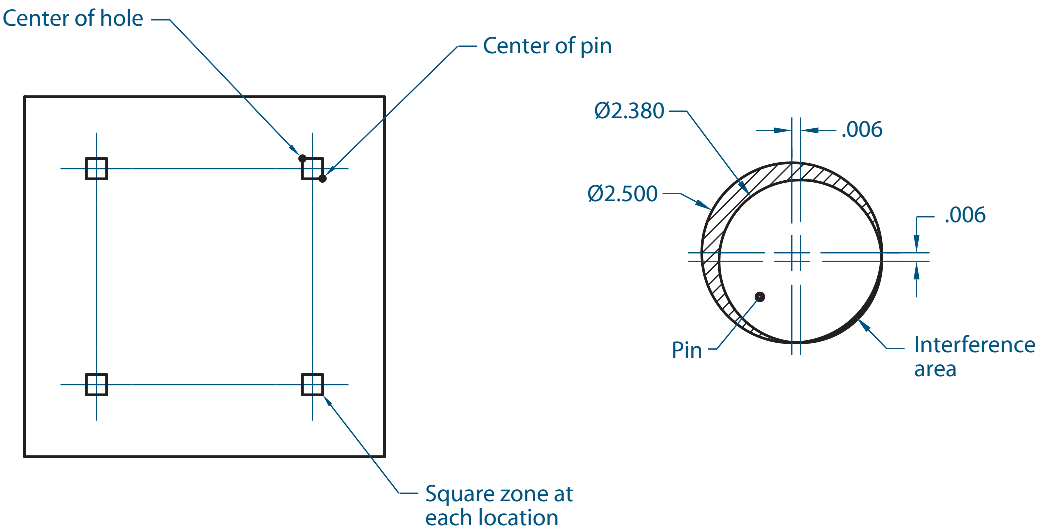

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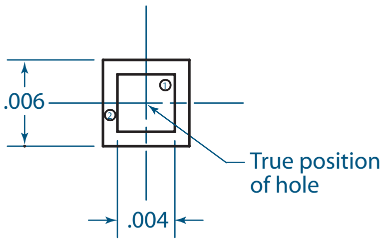
FIGURE 13.39. The results of changing the size of the square tolerance zone.



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13.06 Geometric Dimensioning and Tolerancing (GD&T)

Many students have trouble understanding geometric dimensioning and tolerancing (GD&T). GD&T has even been referred to as Gloom, Doom, and Terror. If you try to keep things simple in the beginning, you should be able to apply appropriate tolerances to objects without confusing your boss, a machinist, or a quality control person.

Geometric dimensioning and tolerancing (GD&T) is a 3-D mathematical system that allows a designer to describe the form, orientation, and location of features on a part within precise tolerance zones. Because it uses symbols instead of words, GD&T is an international language that is understood by technicians and engineers around the world. For these reasons, there are several advantages of using GD&T instead of conventional tolerancing:

- *The system allows the designer to clearly specify design intent*. The location of datums allows manufacturing engineers, machinists, and quality control personnel to easily recognize part functionality.
- *There is better communication throughout the design process*. Manufacturing engineers and machinists are able to make better decisions about how things are made. Quality control personnel are able to make better choices for inspection.
- *The system is set up so that almost nothing can be interpreted in more than one way*. This is extremely valuable when hiring a subcontractor to manufacture parts. You want to make sure everything works when the parts are assembled.

Just two items are required within this system:

- A datum reference frame to immobilize and orient the part.
- Specific form, orientation, and location tolerances that describe 2-D and 3-D tolerance zones within which all part geometry must fall.

13.06.01 The Datum Reference Frame

Usually, errors related to GD&T have something to do with how the **datum reference frame** was established (or not established). The datum reference frame is a theoretical system made up of three mutually perpendicular planes, or **datums**, established by real features on the object. This concept of a 3-D system should not be new to you. From the time you began plotting points on graph paper in mathematics, science, or graphics class, you were using a 3-D coordinate system. When working with GD&T, you need to keep two things in mind:

- *The things you cannot see*: The imaginary or theoretical coordinate system and perfect part geometry that exists within the CAD software, CNC manufacturing software, or inspection software
- *The things you can see*: The real features on the finished object

Figure 13.40 shows the items that make up the theoretical datum reference frame. As stated earlier, the key part of the system is the three mutually perpendicular planes. The origin or datum point is defined by the intersection of the three planes. Datum axes are defined by the intersection of two of the planes. As you begin applying geometric tolerances, the goal is to add just enough datums to immobilize the part or eliminate all degrees-of-freedom, or DOFs (translational and rotational).

Figure 13.41 illustrates some terminology related to datums. The previous paragraph talked about the datum plane, which is a theoretical perfect plane that exists in one's mind and in the CAD, CNC, and inspection software. To establish these theoretical datums, datum simulators must be used. Datum simulators can be a number of different things. When parts are being manufactured, machine beds, lathe chucks or collets, gage pins, and vises are used to establish datums. When parts are being inspected, datums are established using granite tabletops, surface plates, and angle plates. All of these datum simulators must be at least *ten times better in quality*

FIGURE 13.40. Components of the theoretical datum system.

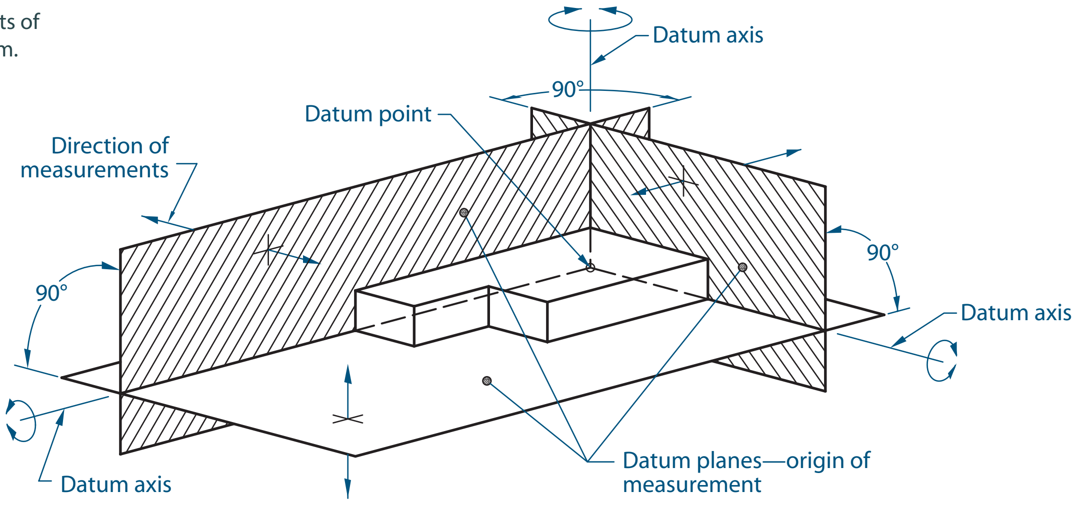

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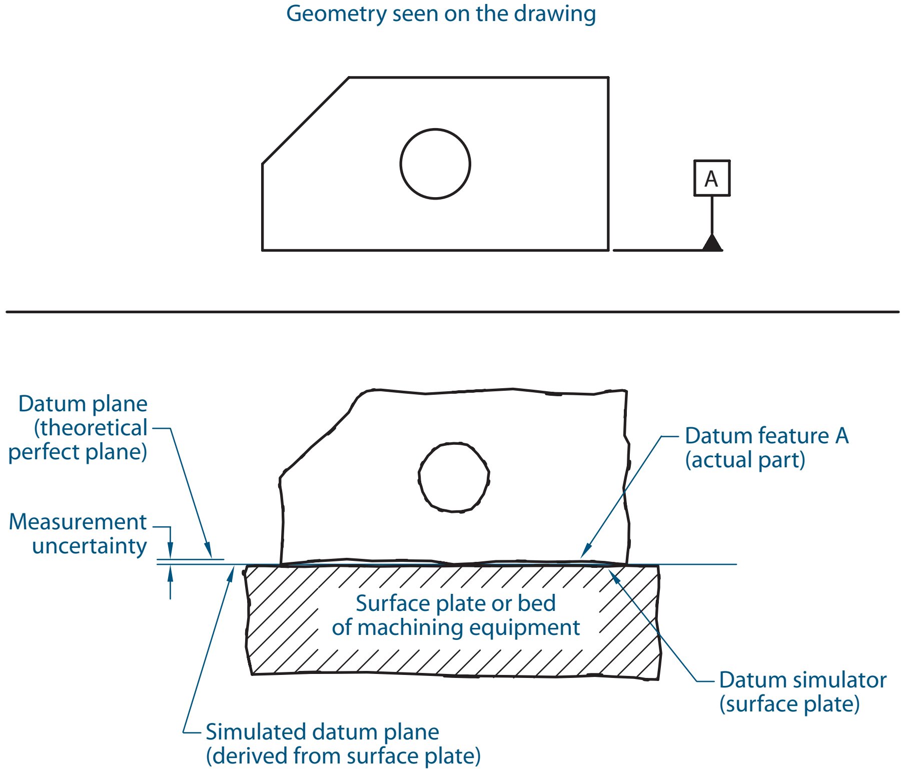
FIGURE 13.41. Datum terminology.



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than the tolerances specified on the drawings to be considered datum simulators. Simulated datums are derived from the datum simulators coming in contact with features on the actual part or on the datum features.

13.06.02 Geometry Characteristic Symbols and Feature Control Frames

Before considering the specific tolerances, take a look at Figure 13.42. Notice how the geometry characteristic symbols are organized. The form tolerances are for individual features. They are not related to any datums. The profile tolerances may or may not be related to datums. The orientation, location, and runout tolerances must be related to datums. As you explore the individual symbols, you will see why certain tolerances should be related to datums and why others should not.

FIGURE 13.42. Geometric characteristic symbols.

	TYPE OF TOLERANCE	CHARACTERISTIC	SYM
FOR INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	—
		FLATNESS	⏥
		CIRCULARITY	○
		CYLINDRICITY	⌭
FOR INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A LINE	⌒
		PROFILE OF A SURFACE	⌓
FOR RELATED FEATURES	ORIENTATION	ANGULARITY	∠
		PERPENDICULARITY	⊥
		PARALLELISM	//
	LOCATION	POSITION	⌖
		CONCENTRICITY	◎
		SYMMETRY	⌯
	RUNOUT	CIRCULAR RUNOUT	↗
		TOTAL RUNOUT	⌰

GD&T is a technical language. Like any language, some people use it in a conversational way. Others can read it but not write it, and some designers are experts who can read and write the language fluently.

The main focus of the language is the **feature control frame**. The feature control frame contains the geometric characteristic symbol, the geometric tolerance, and the relative datums (see Figure 13.43). To understand the language, you must be able to put the symbols in the feature control frame into a form that you can read. Figure 13.43 shows the different parts of the feature control frame. One of the 14 geometric characteristic symbols will appear in the first section of the frame. In the second section of the feature, control frame is where the tolerance information is displayed. This section will include information about the shape of the zone (such as a diameter symbol for cylindrical zones), the size of the zone, and any material condition modifiers (such as the maximum material condition, MMC, or the least material condition, LMC).

Now you will practice reading a couple of feature control frames before looking at each of the 14 geometric characteristic symbols. Figure 13.44 shows a feature control frame with a perpendicularity tolerance of 0.05 that is related to one datum. You read the feature control frame and try to take each segment separately. You might read the first example as follows: The feature must be *perpendicular* within a *five-hundredths of a millimeter* tolerance zone relative to *datum feature A*.

The example in Figure 13.45 can be read as follows: The features must be *positioned* within a *five-thousandths of an inch cylindrical* tolerance zone at *maximum material condition* relative to primary *datum feature A*, secondary *datum feature B*, and tertiary *datum feature C*.

FIGURE 13.43. A feature control frame.

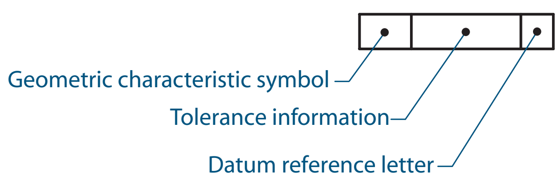


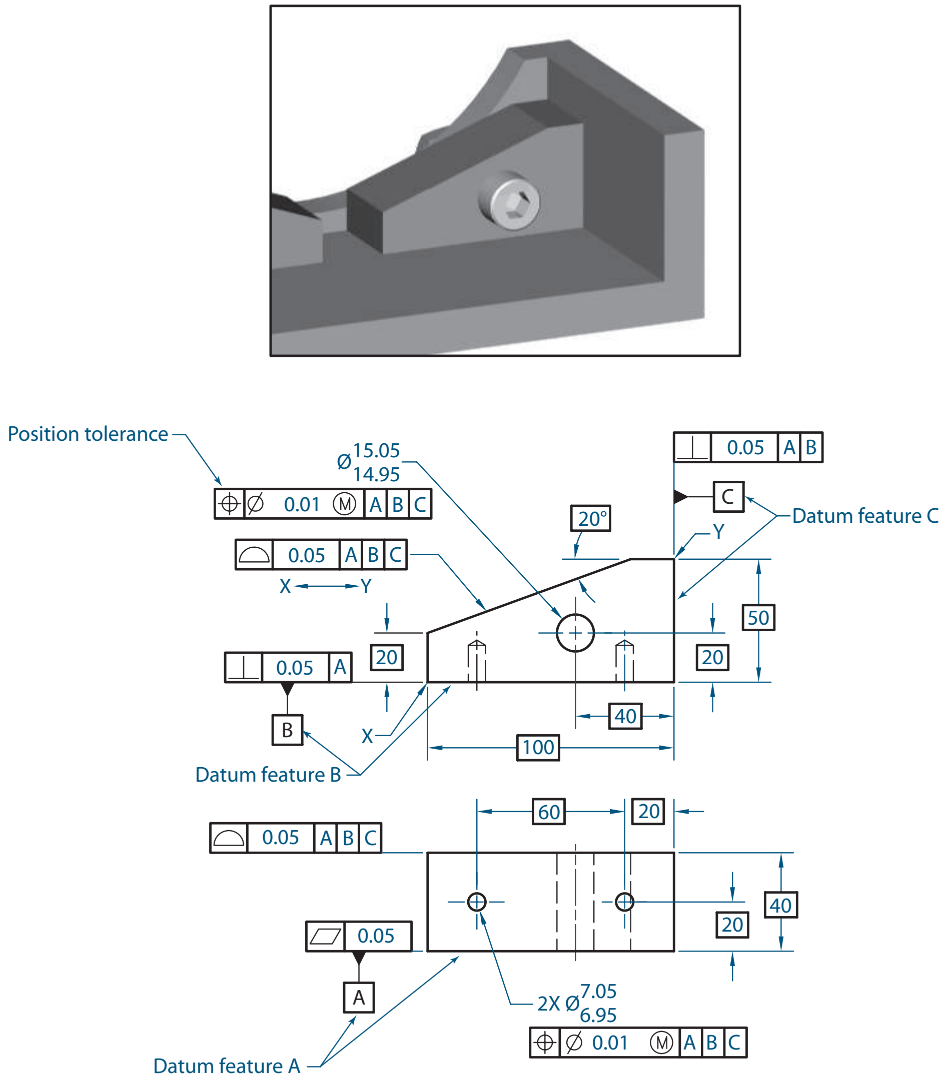
FIGURE 13.44. A feature control frame with the perpendicularity tolerance.

FIGURE 13.45. A feature control frame with the position tolerance.

13.06.03 Order of Precedence for Datums

As you apply datums to a part, you must specify the order of precedence. Examine the WEDGE BLOCK in Figure 13.46. Since there is a large hole passing through the part for the socket head screw, it is critical that the hole be perpendicular to the back surface of the part and that the hole be located properly. Now examine the drawing for the WEDGE BLOCK. Directly under the size tolerance for the hole (14.95–15.05) is a feature control frame indicating that a position tolerance is used to control the orientation and location of the axis of the hole. The precedence of the datums for this geometric tolerance is datum feature A, datum feature B, and datum feature C. Datum feature A is the back surface (see the bottom view of the part), datum feature B is the bottom surface of the part, and datum feature C is the right-side surface of the part.

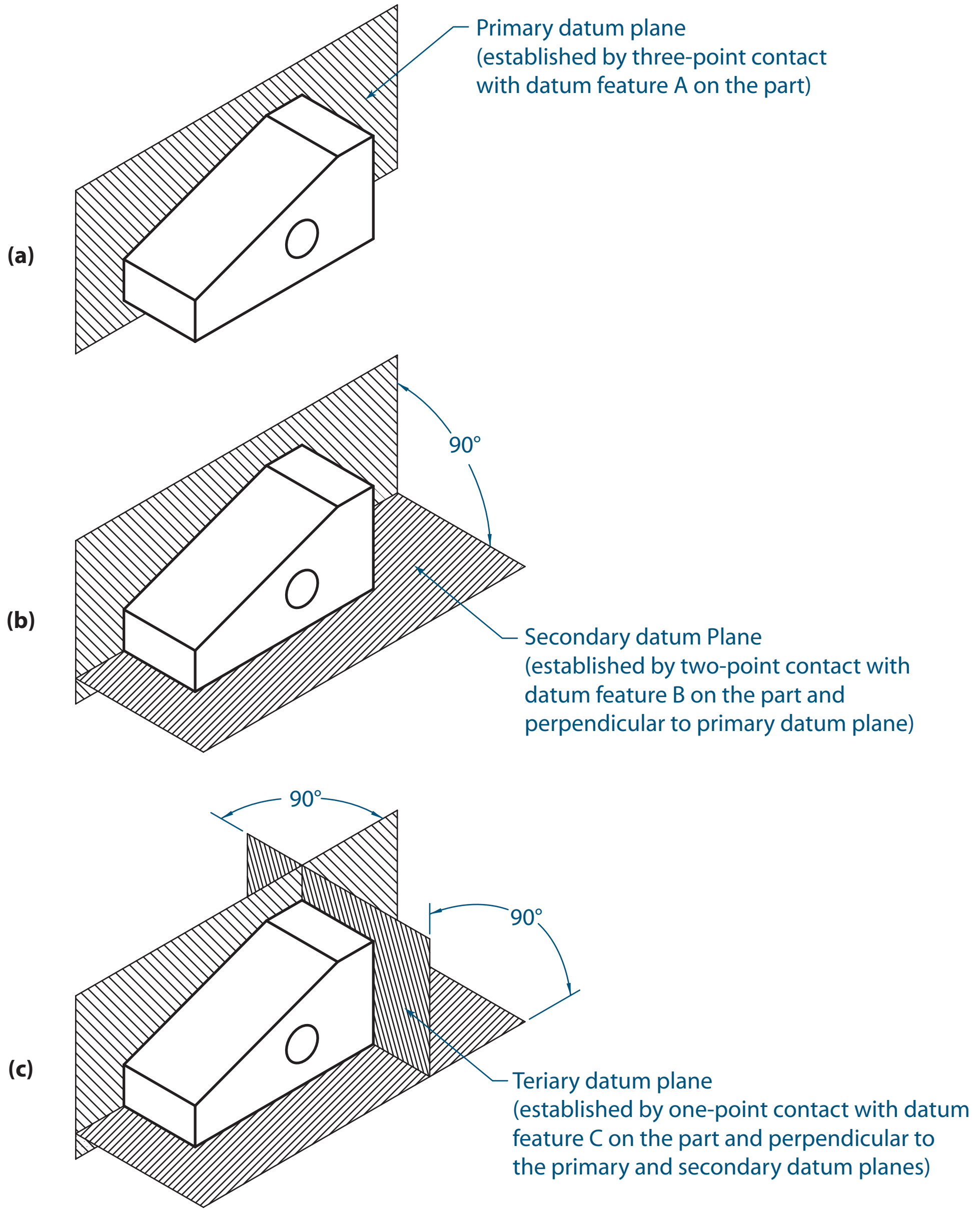
FIGURE 13.46. A WEDGE BLOCK assembly and detail drawing.



To establish the datum reference frame for manufacturing or inspection, a part has to be placed in the machine or loaded in a specific order. Assume for this example that a quality control person is inspecting the position of the large hole through the part. The datums are listed in the order A, B, and C. The primary datum plane is established by a minimum three-point contact with the back surface of the object (datum feature A in Figure 13.47a). Once the primary datum plane is set, the secondary datum plane must be perpendicular to it. Therefore, to establish the secondary datum plane, only a two-point contact with the bottom surface (datum feature B in Figure 13.47b) is necessary. Finally, the tertiary datum plane, which is mutually perpendicular to the first two datum planes, can be established by a one-point contact with the right-hand surface (datum feature C in Figure 13.47c).

Once the datum reference frame is established, the inspector can check the position of the hole. If the hole is machined at maximum material condition (14.95—the smallest hole will generate the most material), the size of the cylindrical tolerance zone is 0.01. The axis of the hole must fall within this cylinder, which is located from

FIGURE 13.47. The sequence of datum features.



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the datums by the 20 and 40 basic dimensions (Figure 13.48). The imperfections in the geometry of the part must fall within the theoretically perfect tolerance zones located from the established datums.

13.06.04 Position Tolerances versus Conventional Tolerances

Section 13.05.03 looked at using conventional tolerancing to locate the centers of holes and pins. That section discussed some problems that might result from the square tolerance zones established with this type of tolerance method. An alternative to using conventional tolerancing is using the geometric tolerance of position.

Figure 13.49 shows the same part as Figure 13.34 with the four holes except that it is dimensioned using geometric dimensions. Look at how the holes are dimensioned on

FIGURE 13.48. The cylindrical tolerance zone of the position tolerance.

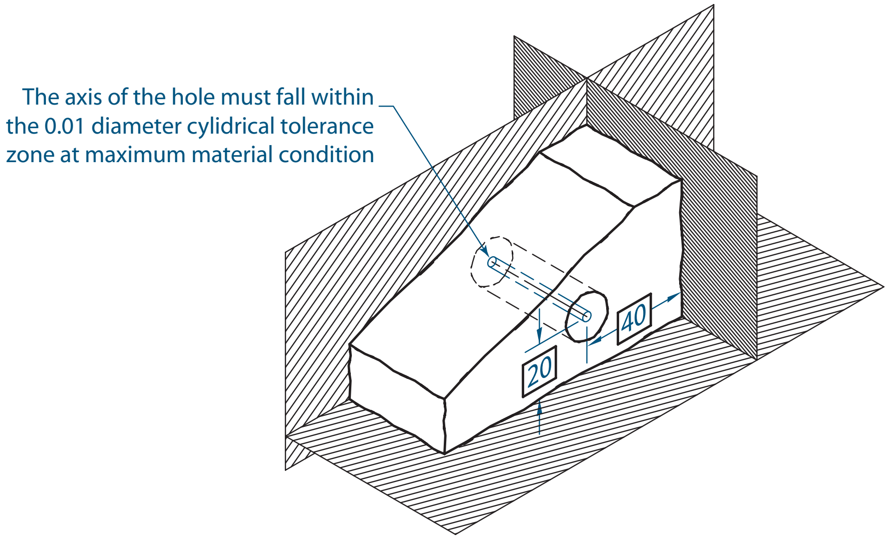

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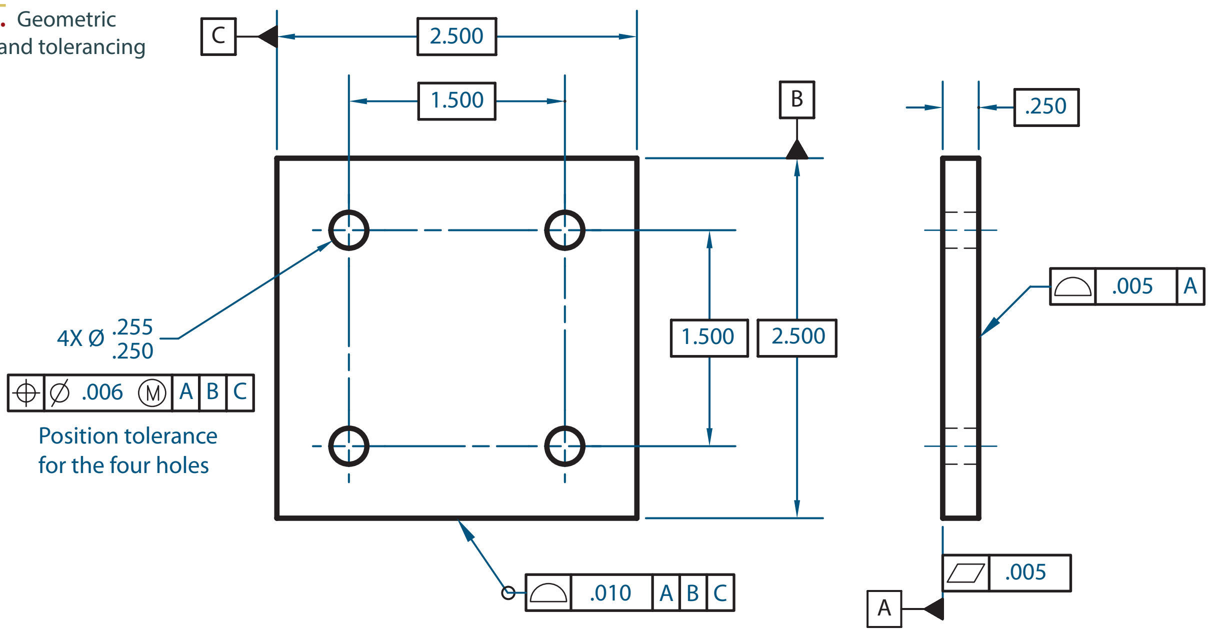
FIGURE 13.49. Geometric dimensioning and tolerancing of the PLATE.



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this part. Notice that the size of the holes are dimensioned the same as before (.250–.255). Conventional tolerancing is still an excellent way to specify the sizes of holes. The main difference here is that the locations of the holes are not dimensioned with limit dimensions, but with **basic dimensions**, which are sometimes called true position dimensions. Basic dimensions (those with a box around them) are theoretically exact. They locate the perfect position of features from clearly identified datums. For the part in Figure 13.49, three datums are used (A, B, and C). The tolerance for the location of the holes is given in a feature control frame (see Figure 13.50). This particular feature control frame indicates that *the four holes must be positioned within a .006 cylindrical tolerance zone at maximum material condition relative to primary datum feature A, secondary datum feature B, and tertiary datum feature C*. Instead of a square tolerance zone, you now have a circular or cylindrical zone (see Figure 13.51).

The **maximum material condition (MMC)** modifier allows the size of the zone to change if the size of the hole changes. MMC is the condition in which a feature of size contains the maximum amount of material within the stated limits of size. The counterpart of MMC is the *least material condition* (LMC)—the condition in which a feature of size contains the minimum amount of material within the stated limits of size. As mentioned earlier, the size of the four holes is .250–.255. The feature control frame states that at MMC, the size of the cylindrical zone is .006. MMC for a hole is the smallest hole; in this case, .250. As the hole departs from MMC, you can add *bonus tolerance* to .006. Figure 13.52 illustrates the potential tolerance zone sizes based on the actual hole size after it is machined.

Now compare the differences between conventional tolerancing square zones and the cylindrical zones of geometric dimensioning for some potential hole locations. Figure 13.53 shows the tolerance zones discussed previously (see Figure 13.39)

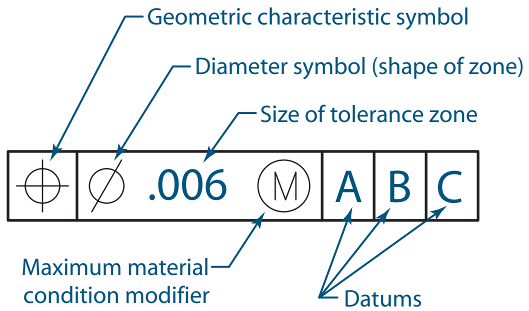

FIGURE 13.50. The feature control frame for the PLATE position tolerance.

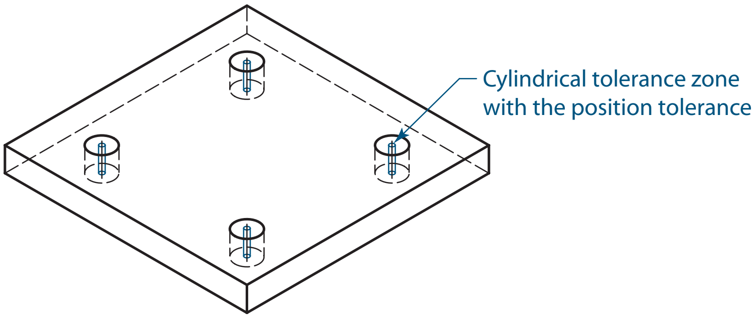

FIGURE 13.51. Cylindrical tolerance zones for the position tolerance.

ACTUAL SIZE OF HOLE	SIZE OF CYLINDRICAL TOLERANCE ZONE
.250	.006
.251	.007
.252	.008
.253	.009
.254	.010
.255	.011

FIGURE 13.52. Effects of the maximum material condition modifier on the position tolerance.

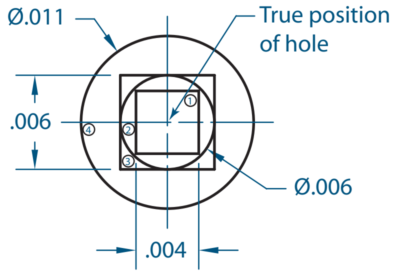

FIGURE 13.53. Cylindrical tolerance zones for the position tolerance.

combined in one figure. It includes the original square zone of .006 and the modified square zone of .004. It also includes the two cylindrical zones from the geometric tolerancing: n.006 (MMC) and n.011 (LMC). Also in the figure are four potential hole locations. If a hole is machined at location 1, it would pass inspection under all methods of tolerancing discussed since it falls within the .004 square zone. A hole machined with its center at position 2 would pass inspection under the original drawing, which specified a square zone of .006, or under the drawing that uses geometric tolerances. Notice, however, that if the zone size is reduced to .004, under the conventional tolerancing method, the part would not pass inspection. This is a problem since the part would work under the initial design conditions. The last two potential hole positions, 3 and 4, represent locations that would fail inspection if the material condition was not considered. With the MMC modifier specified in the feature control frame, these two possible hole centers would pass inspection if the size of the hole departed enough from the MMC.

13.06.05 Form Tolerances

Form tolerances are for individual features and are not related to any datums. This group of geometric tolerances includes straightness, flatness, circularity (roundness), and cylindricity. They control individual features such as surface or line elements within the surface. The shape of the tolerance zones might be two-dimensional (space between two parallel lines or two concentric circles) or three-dimensional (space between two parallel planes, between two concentric cylinders, or within a cylindrical zone).

Flatness-Flatness specifies a 3-D tolerance zone defined by two parallel planes. All points on the specified surface must fall between the two imaginary planes. The tolerance specified in the feature control frame must be less than the tolerance on the size dimension for the part. For the geometric tolerance in Figure 13.54, the feature control frame might be read as follows: The feature must be *flat* within *one-hundredth of a millimeter*.

Figure 13.55 illustrates one method for inspecting flatness. Moving a dial indicator over the surface is time-consuming, but this type of inspection method is fairly good. The full indicator movement (FIM) reading must not exceed the total flatness tolerance.

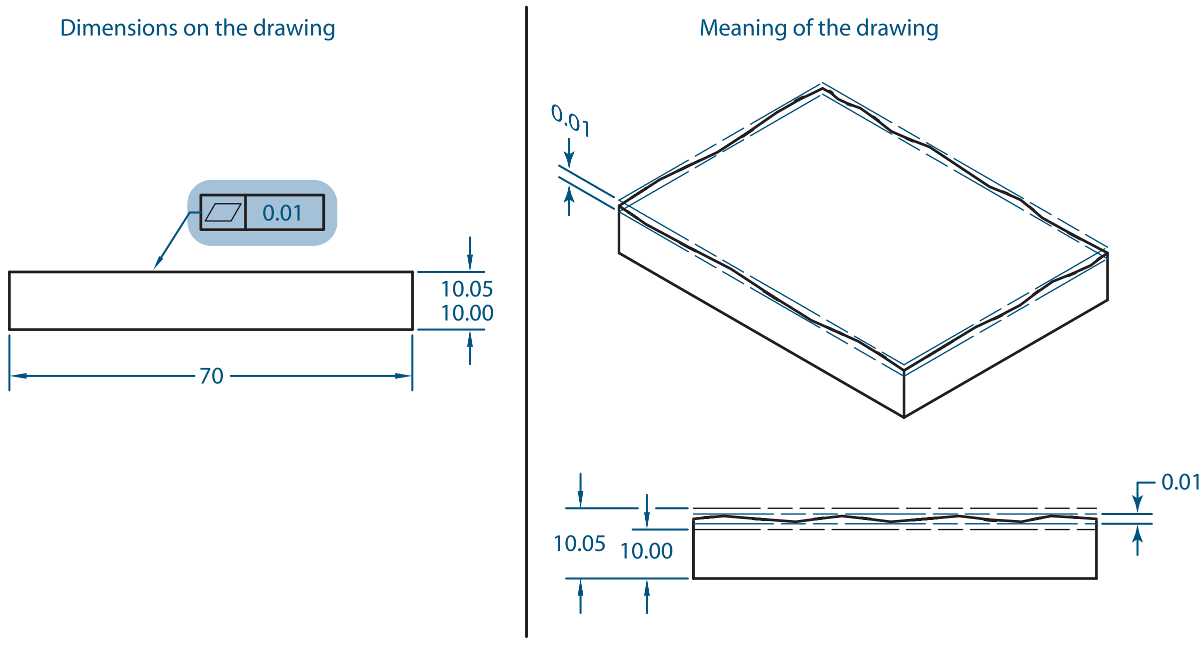

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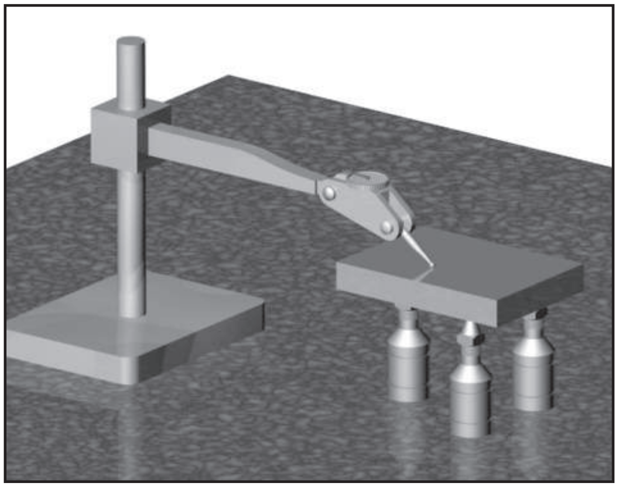
FIGURE 13.54. The flatness tolerance.

FIGURE 13.55. Inspecting the flatness of a feature.



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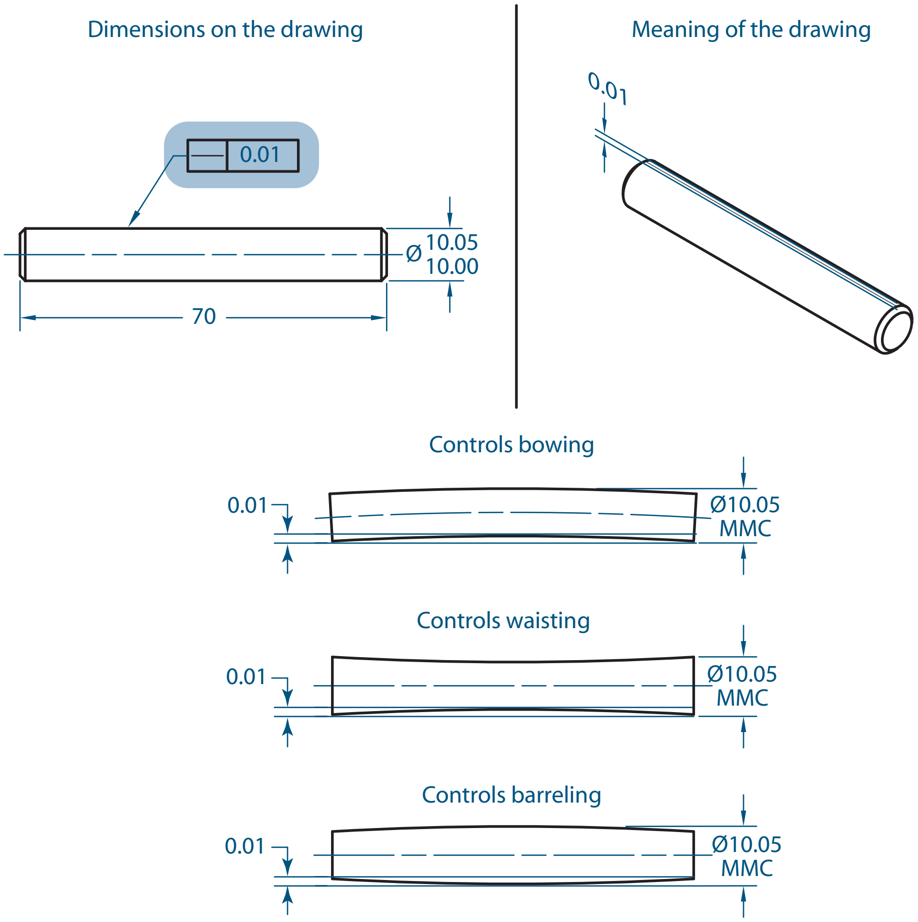
Straightness

The straightness tolerance can be applied in several different ways. If applied as shown in Figure 13.56, the zone is defined by two parallel lines to create a 2-D zone. The tolerance is view-specific, meaning the two parallel lines must move from left to right, not from front to back. The feature control frame reads as follows: Each line element on the surface must be *straight* within *five-hundredths of a millimeter*.

FIGURE 13.56. Straightness of a surface.



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Inspecting straightness is done similarly to the way flatness is checked. The main difference is that the dial indicator is passed over a surface in a straight line. After the FIM is checked, the dial is reset and the indicator is passed over the surface again using a different line element. This is repeated until the whole surface has been checked.

Another way to apply straightness is when you are concerned about the size of a cylindrical feature, such as a machined shaft, but you want to allow the shaft to bend or bow beyond the perfect form limits. Figure 13.57 shows straightness applied to an axis. Because the diameter symbol is specified in the feature control, you know that the tolerance applies to the axis of the shaft instead of to the cylindrical surface. For this tolerance, all derived points on the axis of the shaft must fall within the cylindrical zone. Since the straightness is applied to a feature of size, material condition modifiers such as MMC and LMC can be applied. The feature control frame can be read as follows: The feature (axis of the shaft) must be *straight* within a *one-hundredth of a millimeter cylindrical* tolerance zone at *maximum material condition*. Because a MMC modifier is used, the size of the tolerance zone can increase proportionally as the size of the shaft departs from the MMC.

Because this tolerance is concerned with the axis of a feature (something that cannot be physically touched), this type of straightness is difficult to inspect. If you are inspecting the straightness of an axis for a shaft, a **functional gage** like the one in Figure 13.58 is used. The size of the hole in the gage block is the virtual size of the shaft (MMC plus the tolerance of 0.01). If the shaft can pass through the hole, the part is acceptable.

FIGURE 13.57. Straightness of an axis.

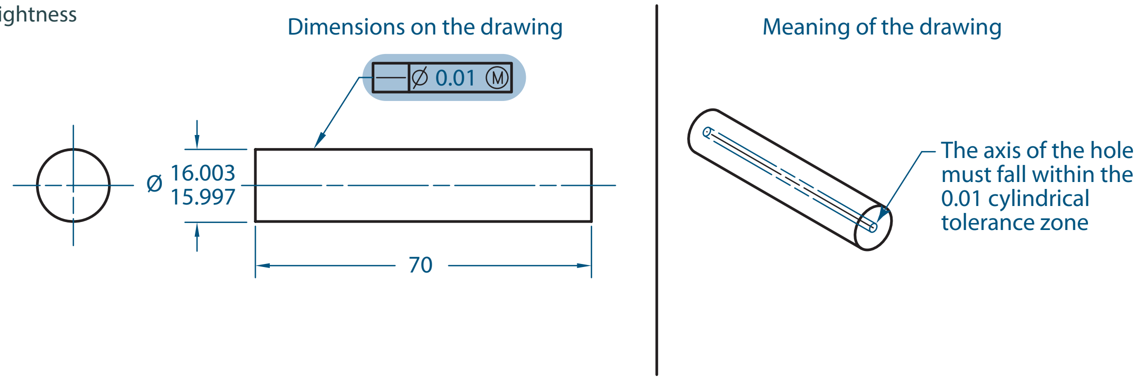
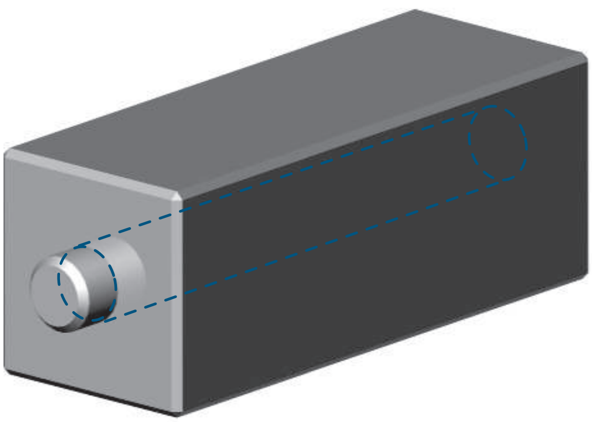


FIGURE 13.58. Inspecting the straightness of an axis with a functional gage.



Circularity (Roundness)

Circularity or roundness is a 2-D control. For shafts, all points within any plane perpendicular to the axis of the shaft must be equidistant from that axis. Circularity specifies a tolerance zone bounded by two concentric circles. The circularity tolerance must be less than the size tolerance. A good analogy of this control is a stack of pennies. The circularity tolerance would control how round each penny is but not the straightness of the whole stack. This tolerance keeps the cylinder from barreling, tapering, or wasting (see Figure 13.56).

Figure 13.59 illustrates the correct method for applying circularity to a cylinder. The feature control frame reads as follows: The feature (each circular element) must be *round* within *one-tenth of a millimeter* tolerance zone.

Inspecting circularity for cylinders is typically done with a V-Block and dial indicator (see Figure 13.60) or with a v-anvil micrometer.

Cylindricity

The cylindricity tolerance controls a cylindrical surface so that all points are equidistant from a common axis. It is the most complex form of tolerance. Inspecting the tolerance also is very difficult. The tolerance zone is a 3-D zone defined by two concentric cylinders.

FIGURE 13.59. The circularity or roundness of a feature.

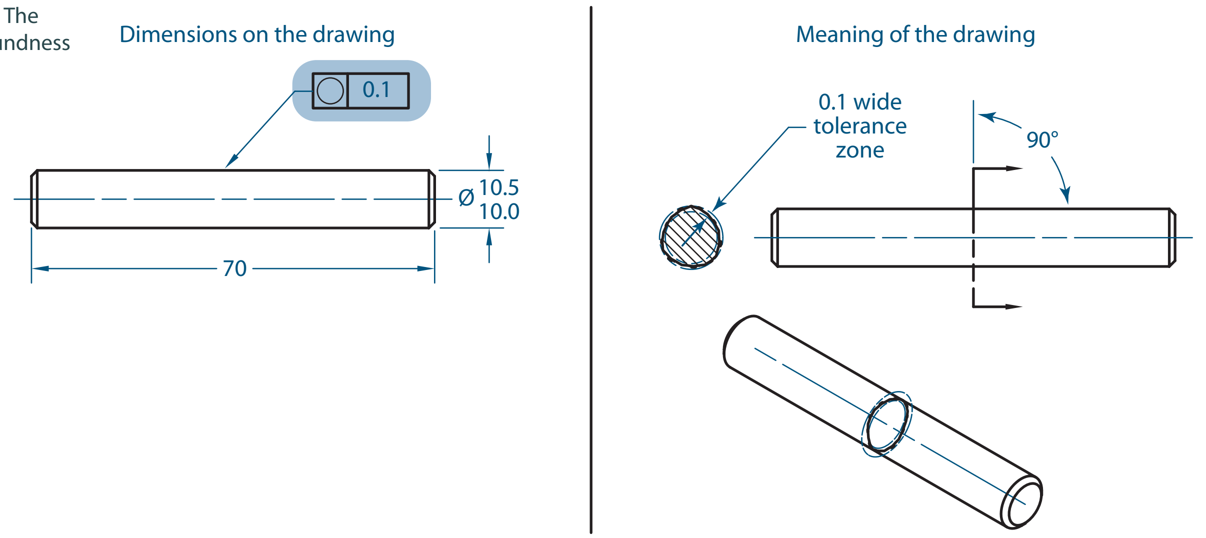


FIGURE 13.60. Inspecting circularity.

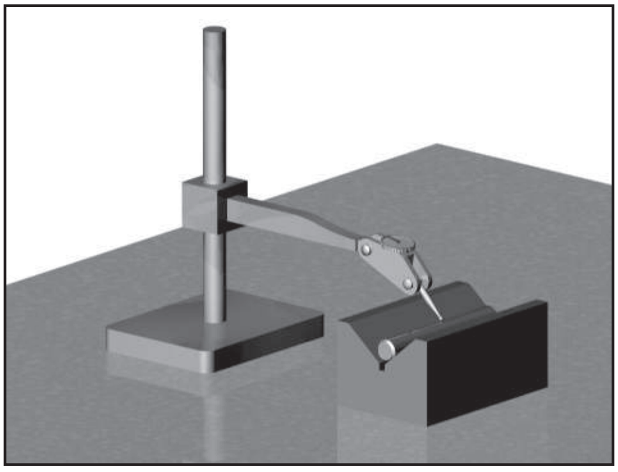
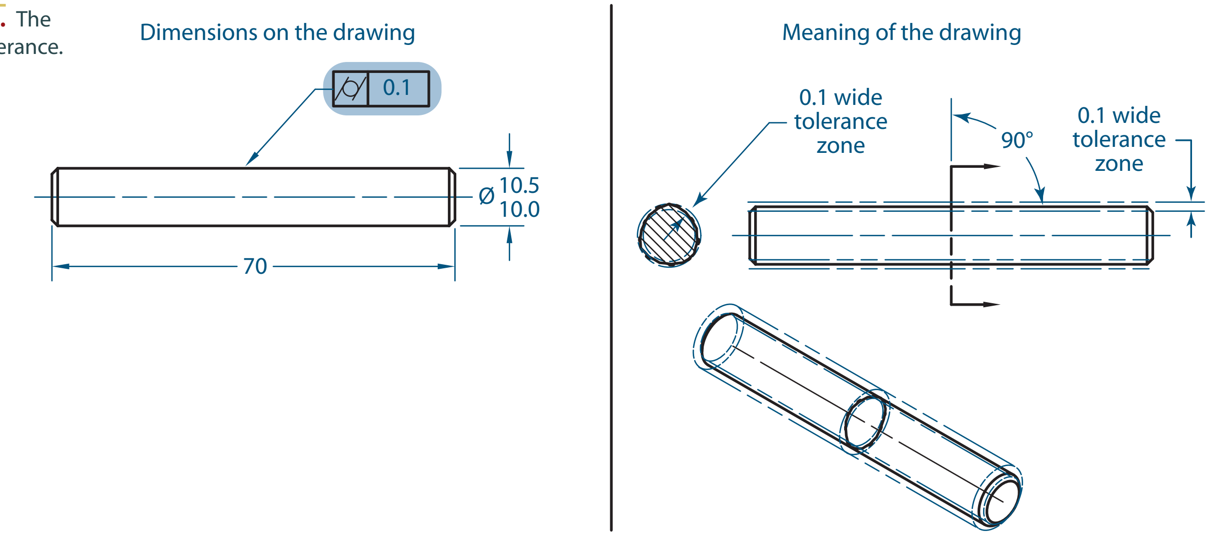


FIGURE 13.61. The cylindricity tolerance.



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All points on the surface of the shaft must fall between the two concentric cylinders. The feature control frame in Figure 13.61 reads as follows: The cylindricity of the feature must be within a twenty-five hundredths of a millimeter tolerance zone.

There are a couple of different accepted techniques for inspecting cylindricity. Figure 13.62 shows a way to inspect cylindricity using a total runout technique. The shaft is spun about two center points. After one complete rotation, the point of the indicator is moved in a straight line parallel to the axis of the shaft. After the point of the indicator moves over the complete surface of interest, FIM is recorded.

13.06.06 Profile Tolerances

Profile of a Line

Profile of a line specifies a *2-D tolerance zone* defined by two contours. The tolerance may specify a datum reference. When a datum reference is not specified, the tolerance controls only the shape of the contour. When datums are specified, the tolerance controls the shape of the contour as well as the size and/or location of the contour.

FIGURE 13.62. Inspection technique for cylindricity.

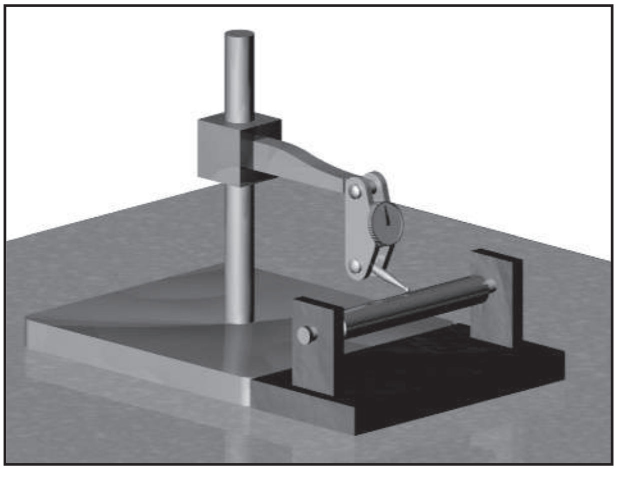

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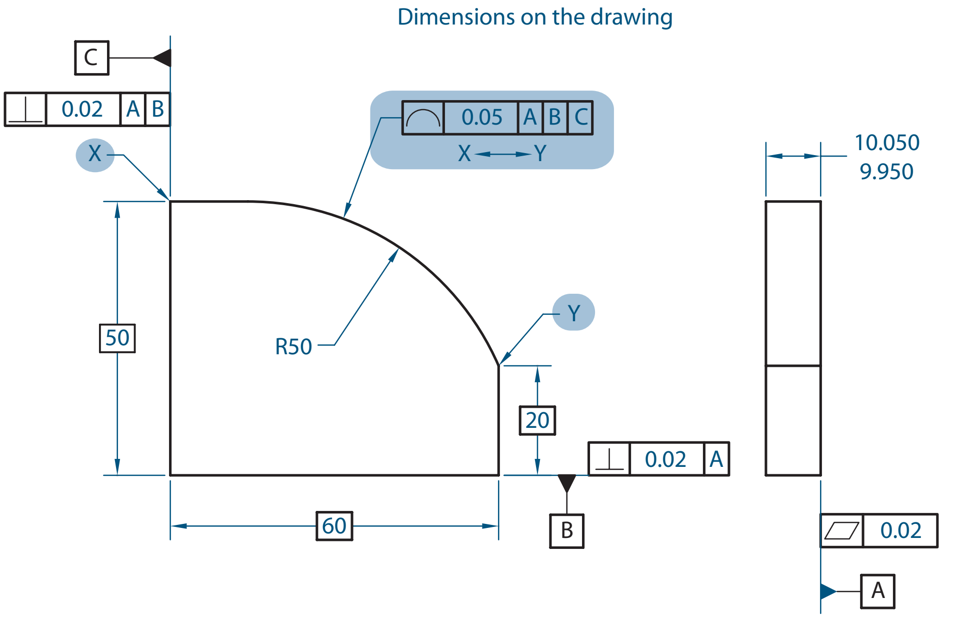
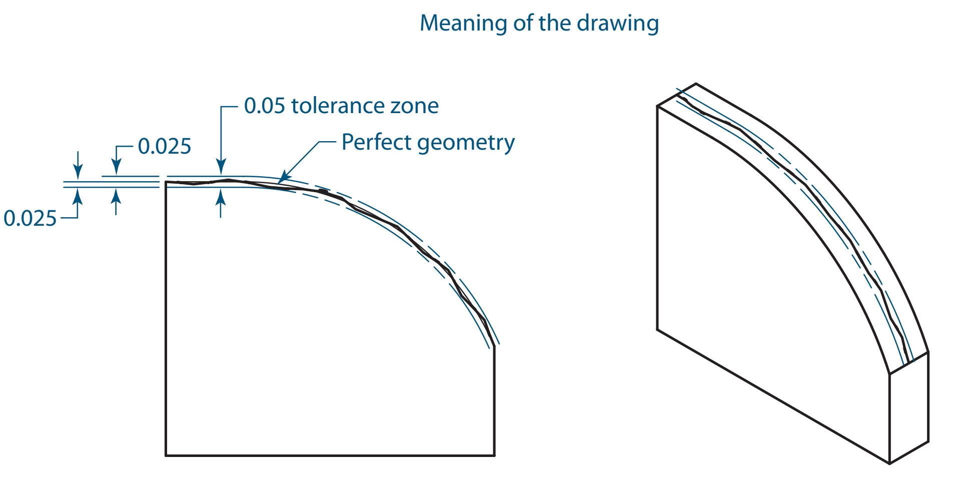
Figure 13.63 illustrates profile of a line. The feature control frame reads as follows: Each feature line profile must be within a five-hundredths of a millimeter tolerance zone relative to primary datum feature A, secondary datum feature B, and tertiary datum feature C. Unless otherwise specified, the tolerance specified in the feature control frame is equally disposed on either side of the perfect geometry (0.025 above and 0.025 below).

Profile of a Surface

Profile of a surface specifies a *3-D tolerance zone* defined by two contoured surfaces. Like profile of a line, the tolerance may specify a datum reference. When a datum reference is not specified, the tolerance controls only the shape of the contour. When datums are specified, the tolerance controls the shape of the contour as well as the size and/or location of the contour. Figure 13.64 illustrates profile of a surface. The feature control frame reads as follows: The surface profile must be within a five-hundredths of a millimeter tolerance zone relative to primary datum feature A, secondary datum feature B, and tertiary datum feature C. Like profile of a line, the tolerance specified in the feature control frame is equally disposed on either side of the perfect geometry (0.025 above and 0.025 below) unless specified otherwise.

FIGURE 13.63. Profile of a line.





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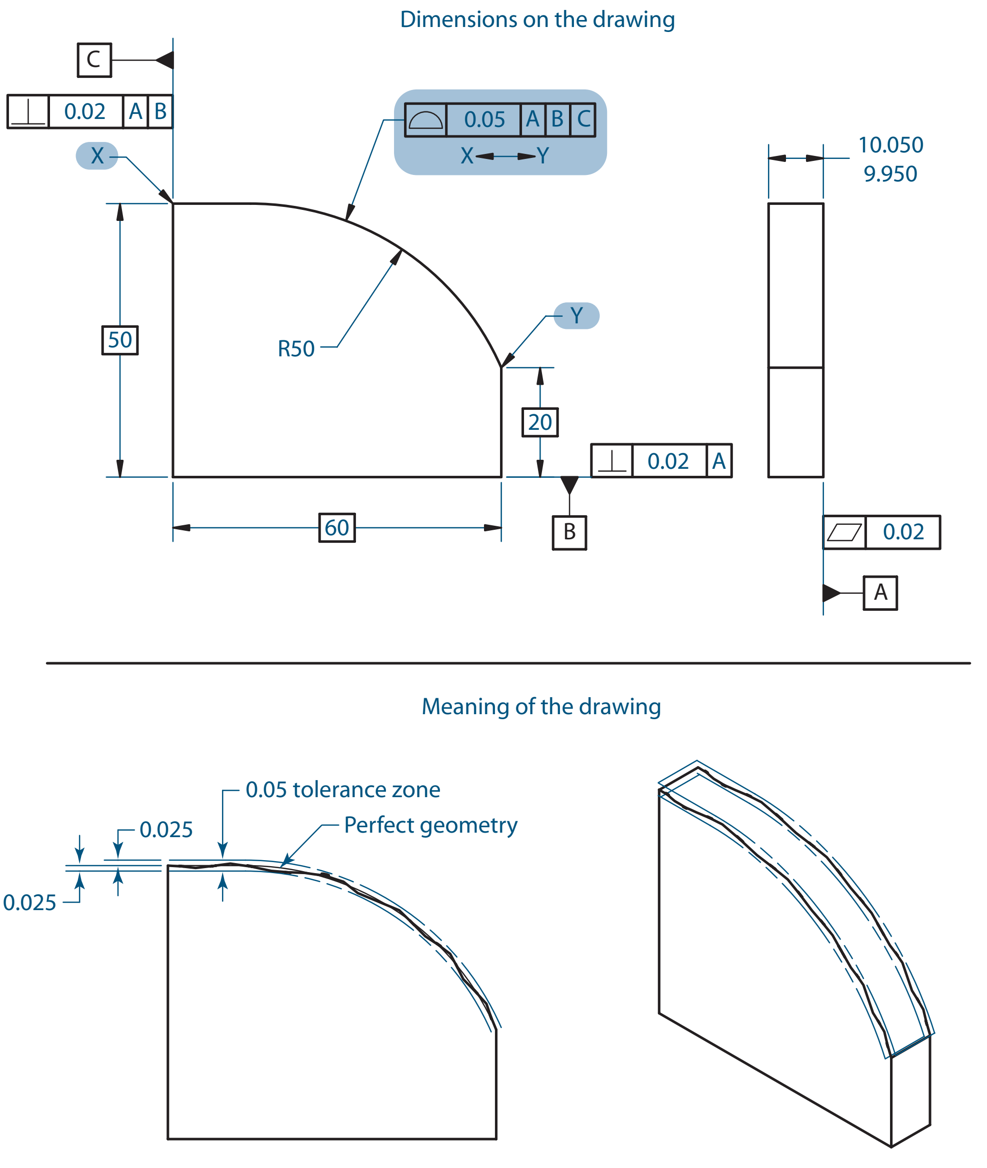

FIGURE 13.64. Profile of a surface.

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Both profile of a line and profile of a surface can control surfaces in other ways. As indicated earlier, profile tolerances are equally distributed about the perfect geometry of the feature. This is referred to as a *bilateral-equal* distribution. Figure 13.65 illustrates three additional ways to describe the tolerance zones for the profile of a line or the profile of a surface. In Figure 13.65a, the 0.05 tolerance zone is on the outside of the perfect geometry. This is called *unilateral-outside*. Figure 13.65b shows an example of *unilateral-inside* since all of the tolerance zone is specified inside the perfect geometry. The last example in Figure 13.65c can be used when an unequal distribution is desired. The profile tolerance here specifies a *bilateral-unequal* distribution where 0.02 is indicated outside the perfect geometry using a basic dimension and the remaining 0.03 is inside the perfect geometry.

Profile of a surface also can be used to make sure two or more surfaces are coplanar. This can be specified for an object such as the one in Figure 13.66 when datums are applied (see Figure 13.67) or when datums are not applied (see Figure 13.68).

FIGURE 13.65. Tolerance distributions for the profile tolerance.

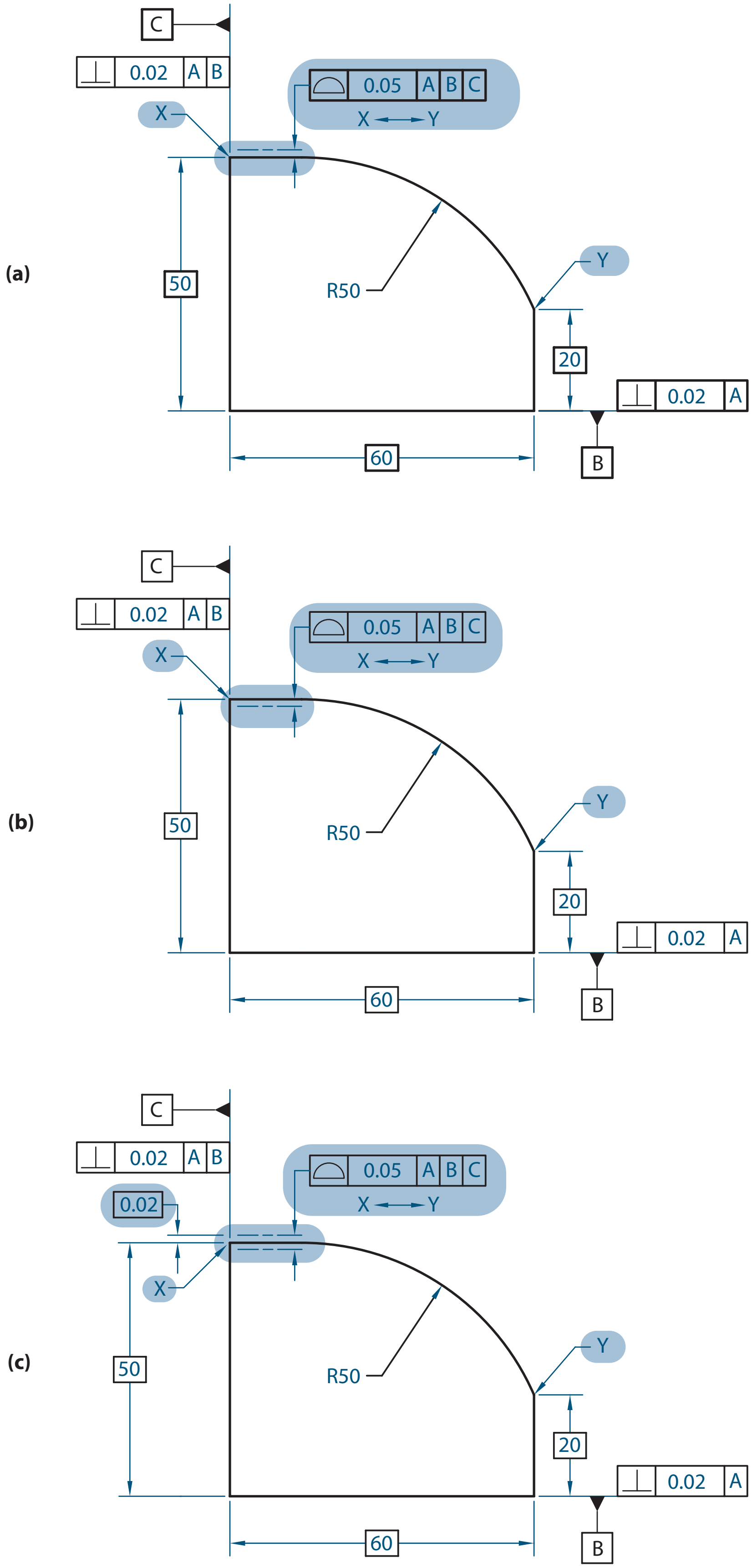

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FIGURE 13.66. Profile of a surface also can be used to make sure two or more surfaces are coplanar.

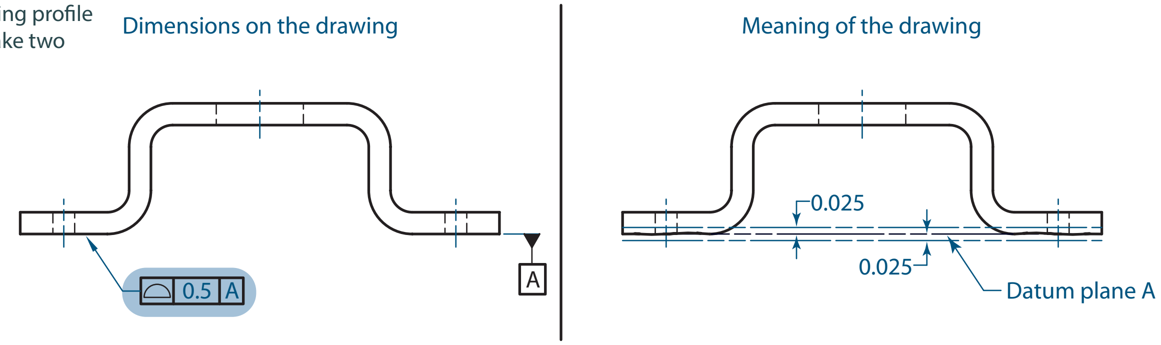

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FIGURE 13.67. Using profile with a datum to make two surfaces coplanar.

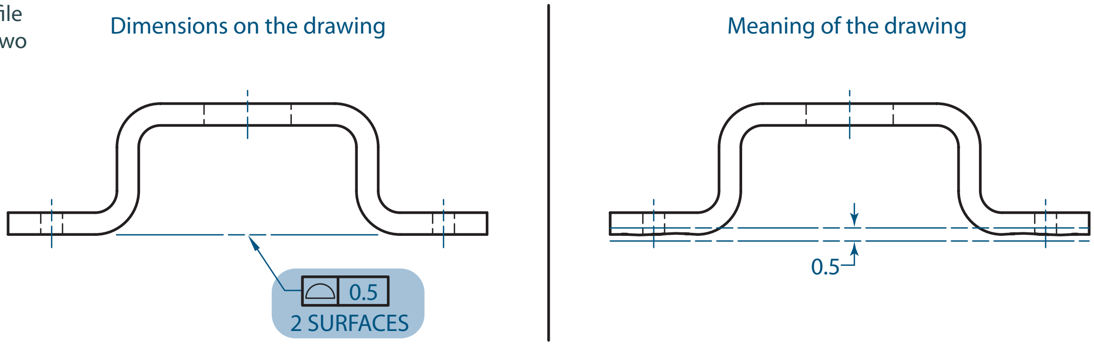

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FIGURE 13.68. Using profile without a datum to make two surfaces coplanar.

Inspection of Profile Tolerances

Profile tolerances can be inspected in a couple of different ways. Optical comparators along with overlay charts are frequently used to inspect profile tolerances. These tools work best for parts such as gaskets and plates and for other thin parts. When a datum reference frame is applied to a tolerance, mechanical gaging can be used to inspect parts. For this method, a master part or gage is created to guide a dial indicator as it traces over the surface of the part being inspected.

13.06.07 Orientation Tolerances

Parallelism

Parallelism specifies a 3-D tolerance zone that can control the orientation of a surface or axis of a hole or cylinder relative to a datum. When applied to a surface, the 3-D zone is defined by the area between two parallel planes. When applied to an axis, the 3-D zone is defined by the area within a cylinder. In either case, the tolerance must include a datum reference. Figure 13.69 illustrates parallelism applied to a surface. The feature control frame reads as follows: The feature must be parallel within five-hundredths of a millimeter tolerance zone relative to datum feature A.

When parallelism is applied to an axis (see Figure 13.70), the tolerance zone is a cylinder. Since the tolerance is applied to a feature of size (a hole), a material condition modifier may be used. In this case, the axis of the hole must fall within the 0.05 cylindrical tolerance zone. The axis of the cylindrical tolerance zone is parallel to datum axis A.

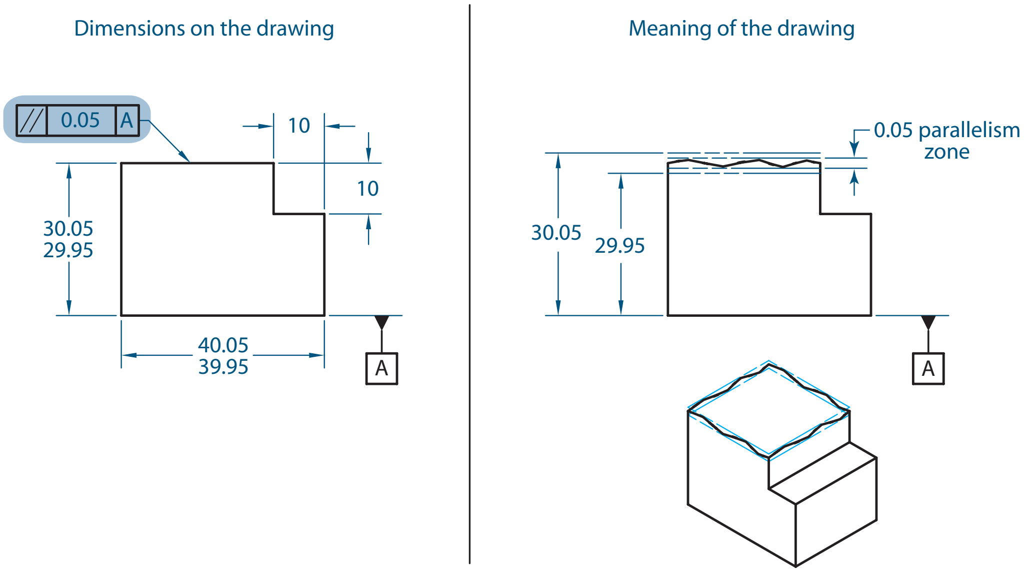


FIGURE 13.69. Parallelism tolerance used to control a surface.

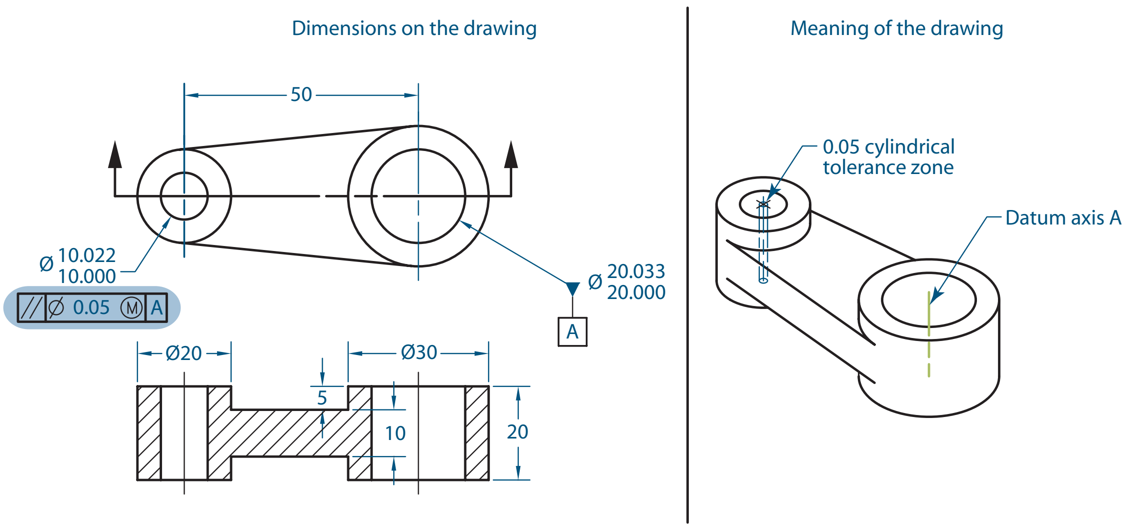


FIGURE 13.70. Parallelism tolerance used to control an axis.

When parallelism for a flat surface is inspected, a surface plate is typically used with a dial indicator (see Figure 13.71). The FIM of the dial must not exceed the tolerance specified on the drawing. When axis-to-axis parallelism is inspected, gage pins must be inserted into the datum hole and the controlled hole. The datum hole is locked into a V-BLOCK, and a dial indicator is used to check the parallelism (see Figure 13.72).

Perpendicularity

Perpendicularity specifies a 3-D tolerance zone that can control the orientation of a surface or axis of a hole or cylinder relative to a datum. When applied to a surface, the 3-D zone is defined by the area between two parallel planes. When applied to an axis, the 3-D zone is defined by the area within a cylinder. In either case, the tolerance must include a datum reference. Figure 13.73 illustrates perpendicularity applied to a surface. The feature control frame reads as follows: The feature must be perpendicular within five-hundredths of a millimeter tolerance zone relative to datum feature A.

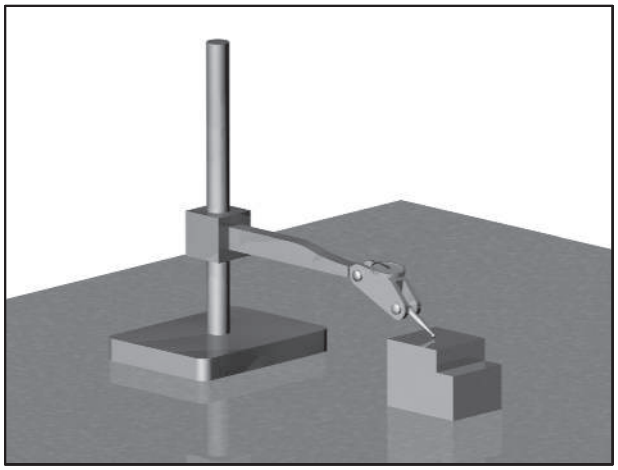


FIGURE 13.71. Inspecting the parallelism of a surface.

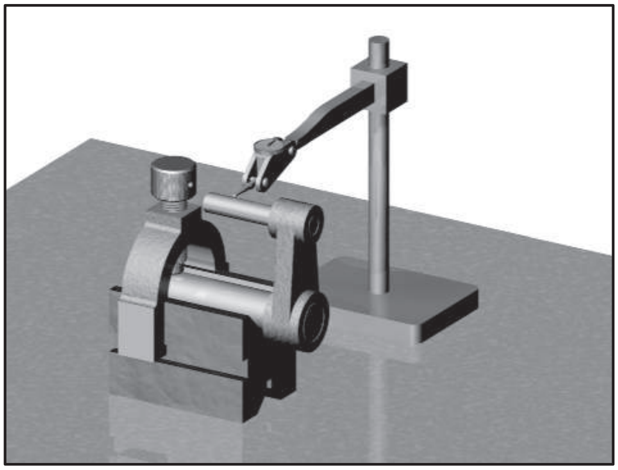


FIGURE 13.72. Inspecting the parallelism of an axis.

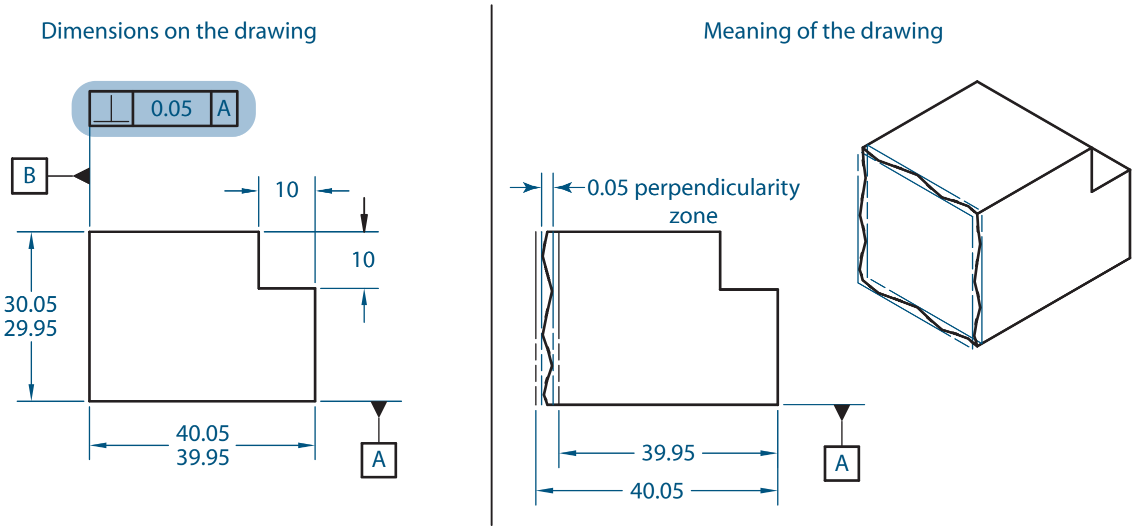


FIGURE 13.73. Perpendicularity tolerance used to control a surface.

When perpendicularity is applied to an axis (see Figure 13.74), the tolerance zone is a cylinder. Since the tolerance is applied to a feature of size (a cylinder), a material condition modifier may be used. In this case, the axis of the cylinder must fall within the 0.05 cylindrical tolerance zone. The axis of the cylindrical tolerance zone is perpendicular to datum axis A.

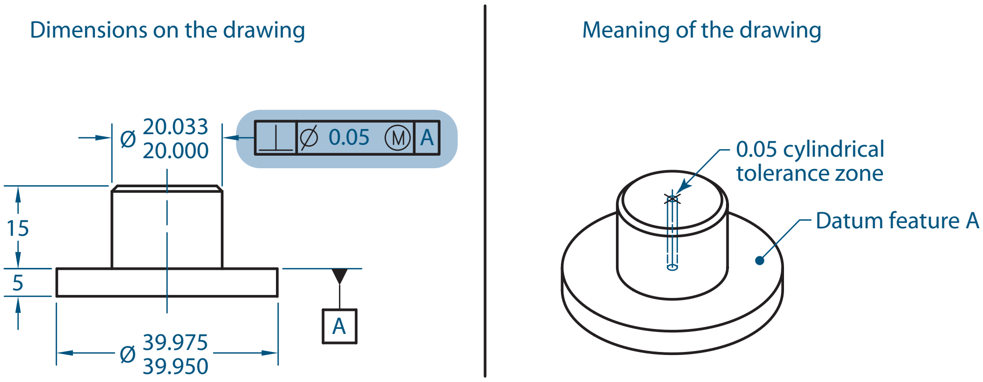
Inspecting perpendicularity can be accomplished in many ways. A common method for inspecting perpendicularity between two surfaces is to use a right-angle plate method (see Figure 13.75). Inspecting the perpendicularity of an axis is similar to inspecting the parallelism of an axis. Gage pins can be inserted into holes, and a dial caliper can be used.

Angularity

One of the disadvantages of using plus-minus dimensioning to control angular surfaces has to do with the shape of the resultant zone. Figure 13.76 illustrates the zone that is defined by the note ALL ANGULAR DIMENSIONS +/– 1° UNLESS OTHERWISE SPECIFIED. Notice that a wedge-shaped zone is created. This is a problem since more tolerance is accepted to the top left of the surface than to the bottom right.

Angularity specifies a 3-D tolerance zone that can control the orientation of a surface or axis of a hole or cylinder relative to a datum. When applied to a surface, the 3-D zone is defined by the area between two parallel planes. When applied to an

FIGURE 13.74. Perpendicularity tolerance used to control an axis.



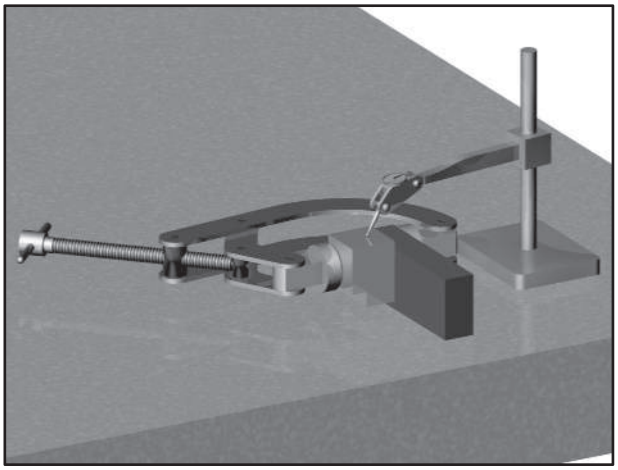


FIGURE 13.75. Inspecting the perpendicularity between two surfaces.

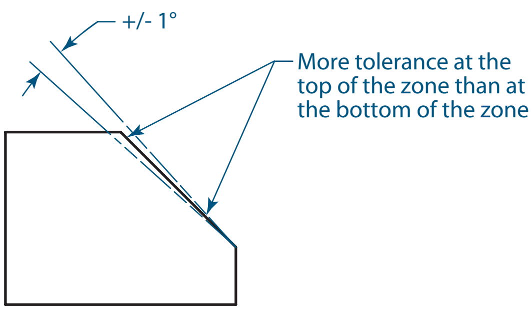


FIGURE 13.76. The tolerance zone when conventional tolerancing is used to control angles.

axis, the 3-D zone is defined by the area within a cylinder. In either case, the tolerance must include a datum reference and a basic dimension specifying the angle from one or more datums. The feature control frame in Figure 13.77 reads as follows: The feature must be at an angle of 45degrees within five-hundredths of a millimeter tolerance zone relative to datum feature A.

Angularity also can be inspected many different ways. Figure 13.78 illustrates one popular way. A sine bar is used in combination with precision cylinders and blocks to orient the angled surface parallel to the tabletop. A dial indictor is then used to investigate the FIM.

13.06.08 Location Tolerances

Position

The position tolerance is one of the most frequently used geometric tolerances. It can be used to control the orientation and location of a center, an axis, or a center plane of a feature of size. When the location of a hole or cylindrical feature of size needs to be controlled, position establishes a 3-D cylindrical tolerance zone within which the axis of the feature must fall. When the location of a center plane needs to be controlled, the position tolerance establishes a 3-D zone defined by two parallel planes. In any case, the zones are located using basic dimensions from specified datum features. There are other uses of the position tolerance, but the most common ones are described above.

FIGURE 13.77. Angularity tolerance used to control a surface.

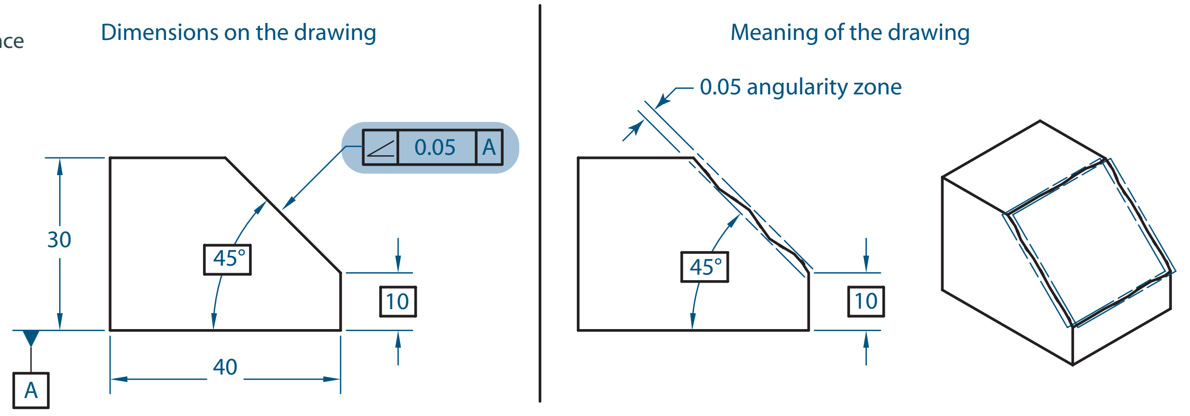
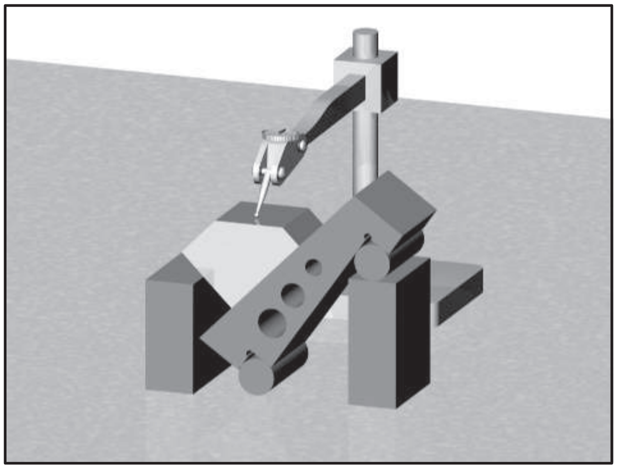


FIGURE 13.78. Inspecting the angularity between two surfaces.



The feature control frame in Figure 13.79 reads as follows: The feature must be positioned within a one-thousandth of a millimeter cylindrical tolerance zone at maximum material condition relative to primary datum feature A, secondary datum feature B, and tertiary datum feature C.

Figure 13.80 illustrates one method for inspecting the position tolerance when it is applied to a hole feature. In this case, an open setup technique is used with surface plates, a gage pin, a clamp, and a dial indicator. Similar techniques are used when the location of more than one feature is inspected.

Concentricity

Concentricity is a geometric tolerance used to control the *axis-to-axis* relationship between two features. The feature control frame in Figure 13.81 reads as follows: The feature must be concentric within a four-tenths of a millimeter cylindrical tolerance zone relative to datum feature A.

FIGURE 13.79. Using the position tolerance to locate the axis of a hole.

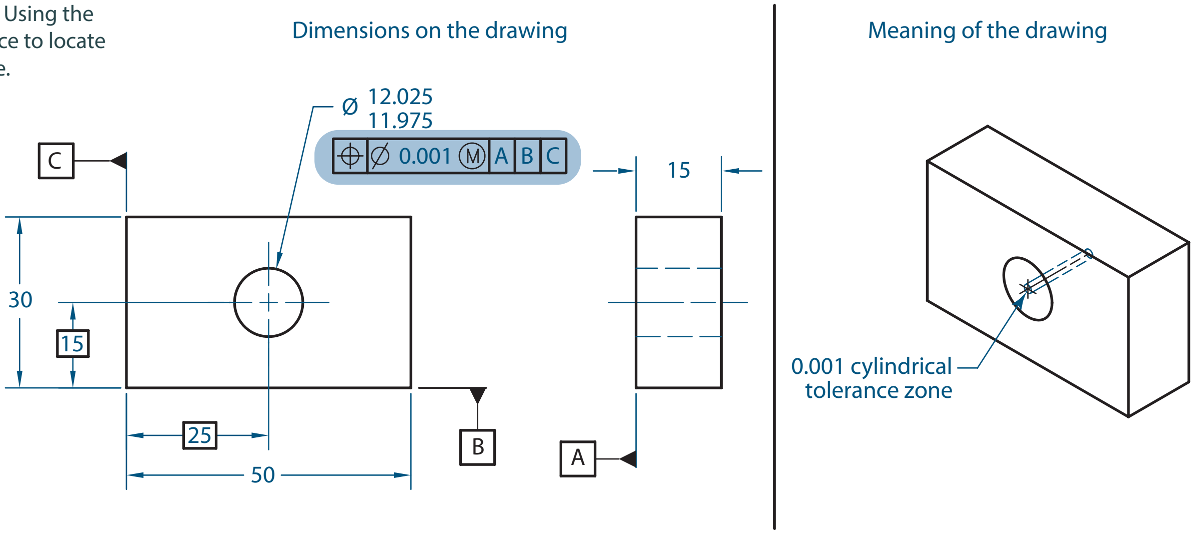


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FIGURE 13.80. Inspecting the position of a hole using a gage pin.

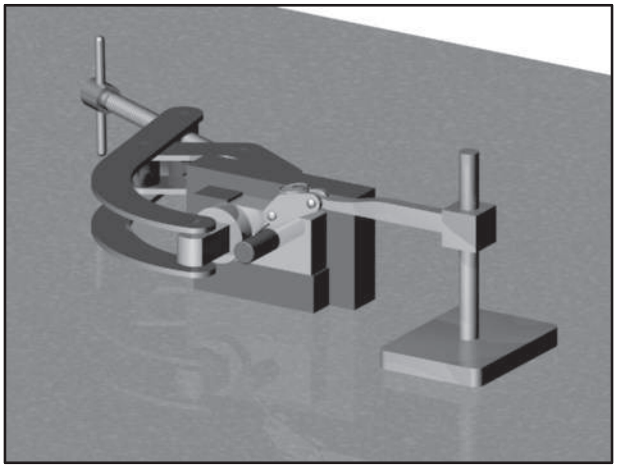

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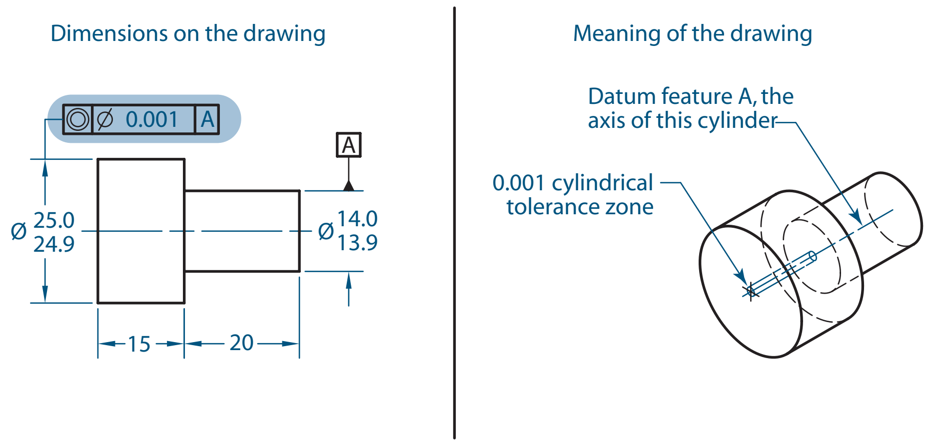
FIGURE 13.81. Concentricity applied to a cylinder.



In theory, this may be easier to understand than it is to inspect. Since concentricity requires an inspector to determine all median points along a feature, it requires some complicated inspection techniques. Diametrically opposed indicators are required to accurately determine concentricity (see Figure 13.82). For objects such as the one in Figure 13.81, runout tolerances provide better inspection methods (see the section on inspecting runout later in the chapter). For an object like the one in Figure 13.83, runout would be a bad choice since a dial indicator would need to be used to make contact with the hexagonal surfaces.

Symmetry

In some ways, symmetry and concentricity are very similar. The main difference is that symmetry is typically concerned with the position of a *center plane* relative to an axis or a center plane of a datum feature. Figure 13.84 illustrates a part with the symmetry tolerance. The feature control frame in the figure reads as follows: The center plane of the feature must be symmetric within a five-hundredths of a millimeter tolerance zone relative to datum feature A.

Since it is necessary to determine the median points on the feature, symmetry is difficult to inspect. For this reason, position and profile tolerances are frequently used instead of symmetry.

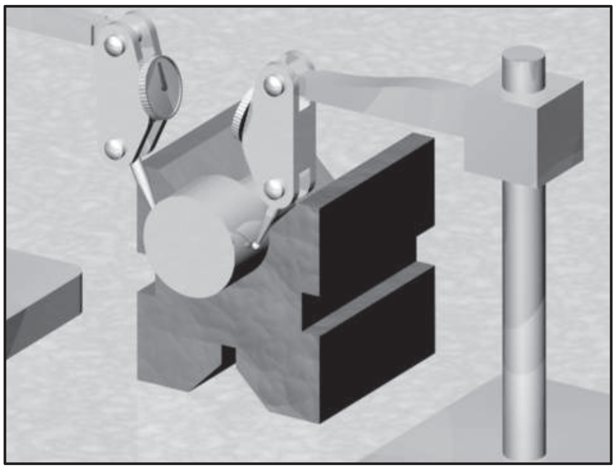


FIGURE 13.82. Inspecting concentricity with diametrically opposed indicators.

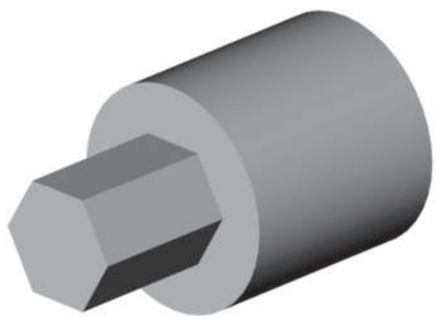
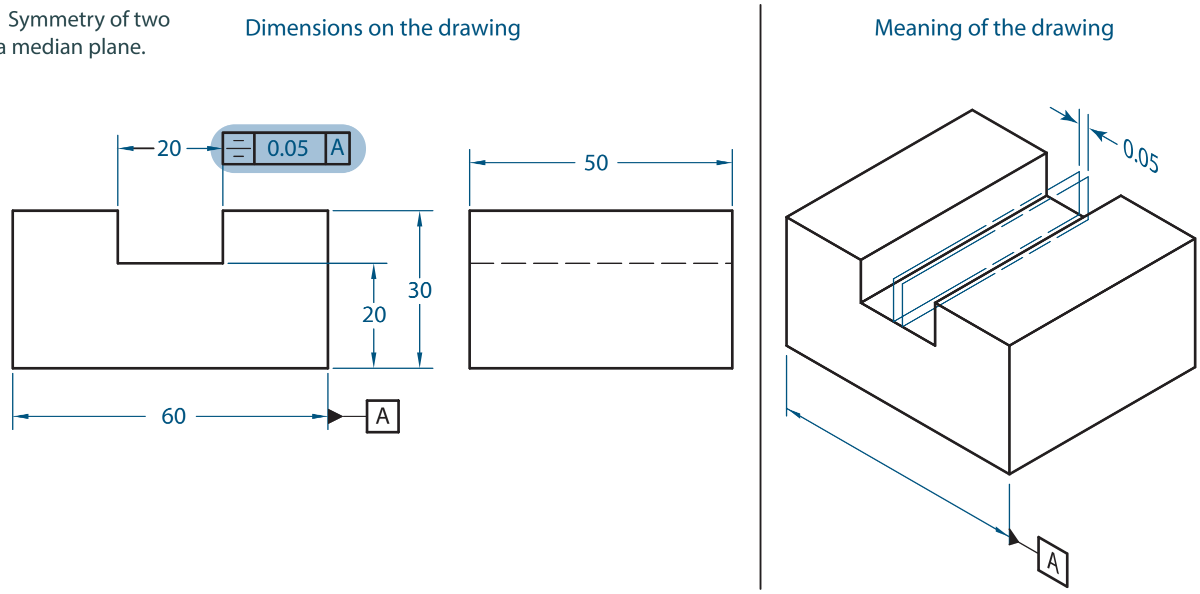


FIGURE 13.83. Concentricity applied to a hexagonal feature.

FIGURE 13.84. Symmetry of two surfaces about a median plane.



13.06.09 Runout Tolerances

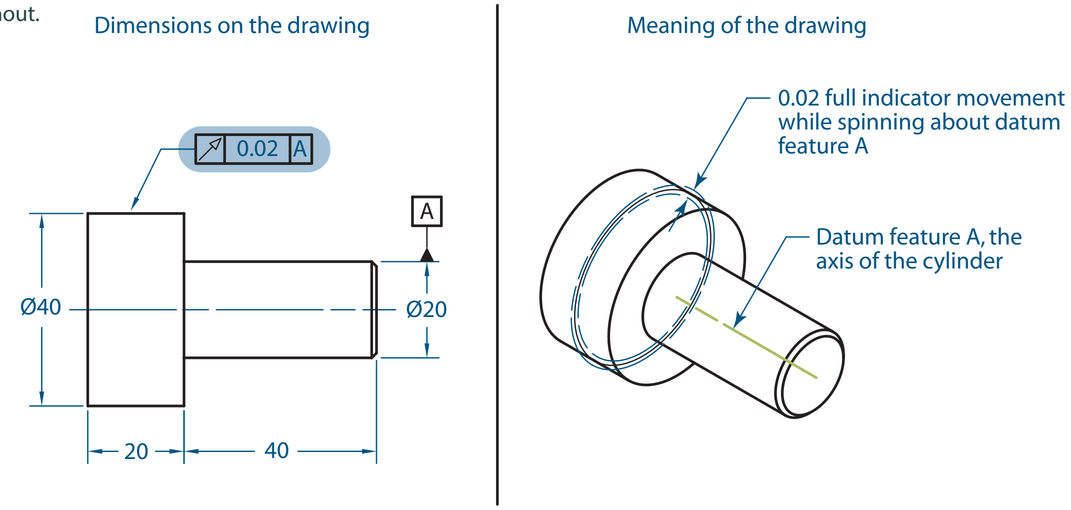
Circular Runout

Circular runout is a 2-D control similar to circularity or roundness. The main difference is that circular runout controls a surface relative to a datum axis. The feature control frame in Figure 13.85 reads as follows: The circular runout of the feature must be within a two-hundredths of a millimeter tolerance zone relative to datum feature A.

Total Runout

Total runout is a 3-D control for rotating parts relative to a datum axis. When applied to a cylindrical surface, it controls circularity, concentricity, straightness, taper, and surface profile (see Figure 13.86). It also can be applied to a flat surface to control

FIGURE 13.85. Circular runout.



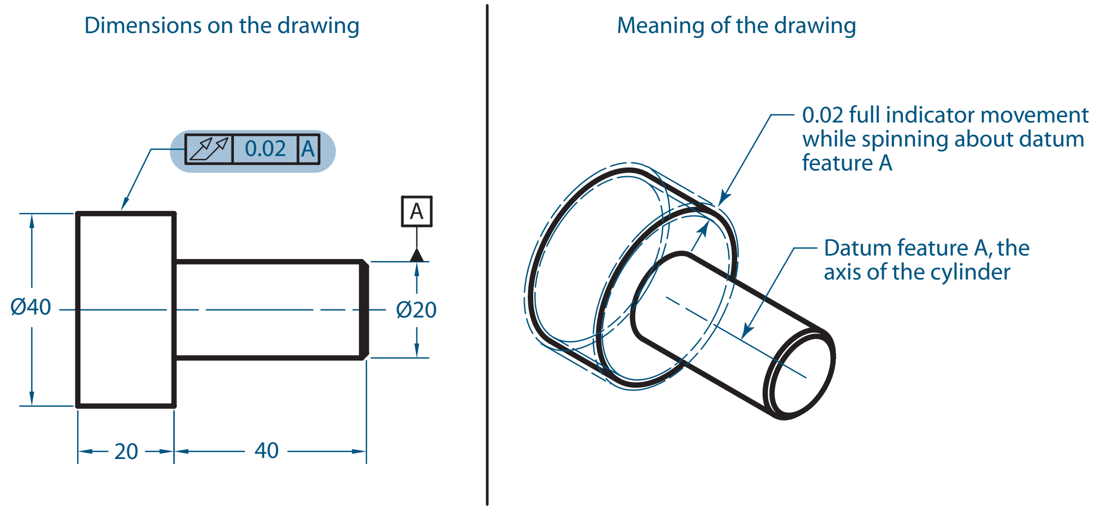


FIGURE 13.86. Total runout applied to a cylindrical surface.

wobble, perpendicularity, and flatness (see Figure 13.87). The feature control frame in Figure 13.86 reads as follows: The total runout of the cylindrical feature must be within a two-hundredths of a millimeter tolerance zone relative to datum feature A.

When using a V-Block and a dial indicator to inspect runout, the FIM must be within the specified tolerance as the part is spinning about the datum feature (see Figure 13.88). For circular runout, the dial is reset after each revolution of the part. For total runout, the indicator is passed over the entire surface before the FIM is examined.

Multiple Datums

Since one of the main goals of geometric tolerancing is to apply tolerances based on how parts function, careful specification of datums is necessary. Figure 13.89 illustrates how a multiple datum can be specified when the runout tolerance is used. Since the SHAFT spins about both SUPPORTS, the datum axis is determined by both ends of the SHAFT. Figure 13.90 illustrates an inspection technique for a multiple datum.

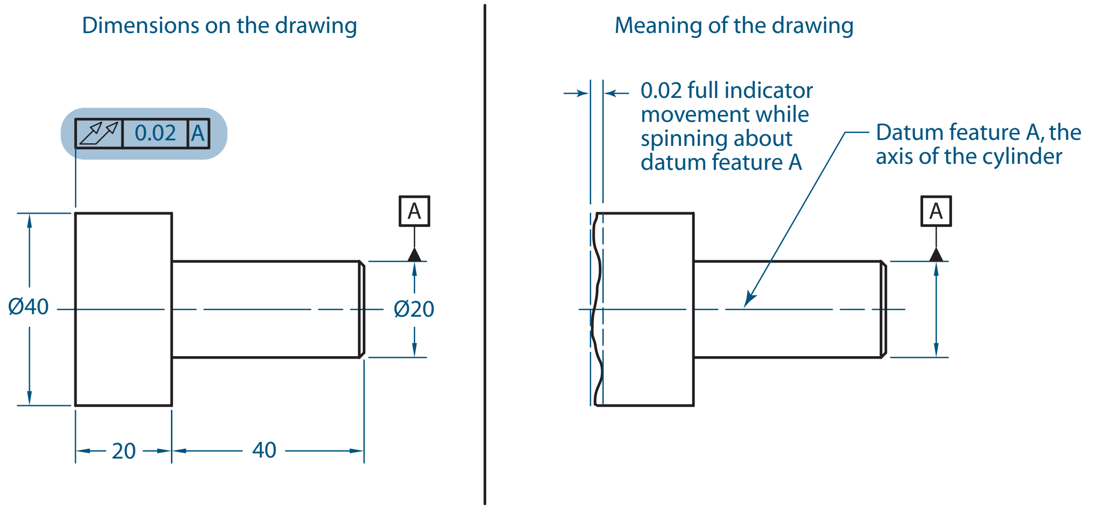


FIGURE 13.87. Total runout applied to a flat surface.

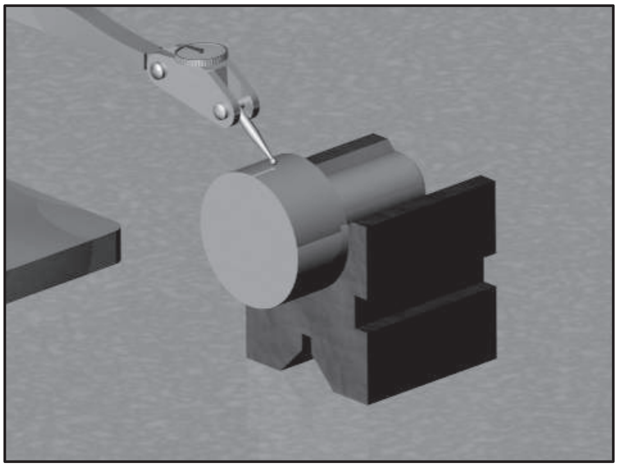


FIGURE 13.88. Inspecting runout using a V-Block.

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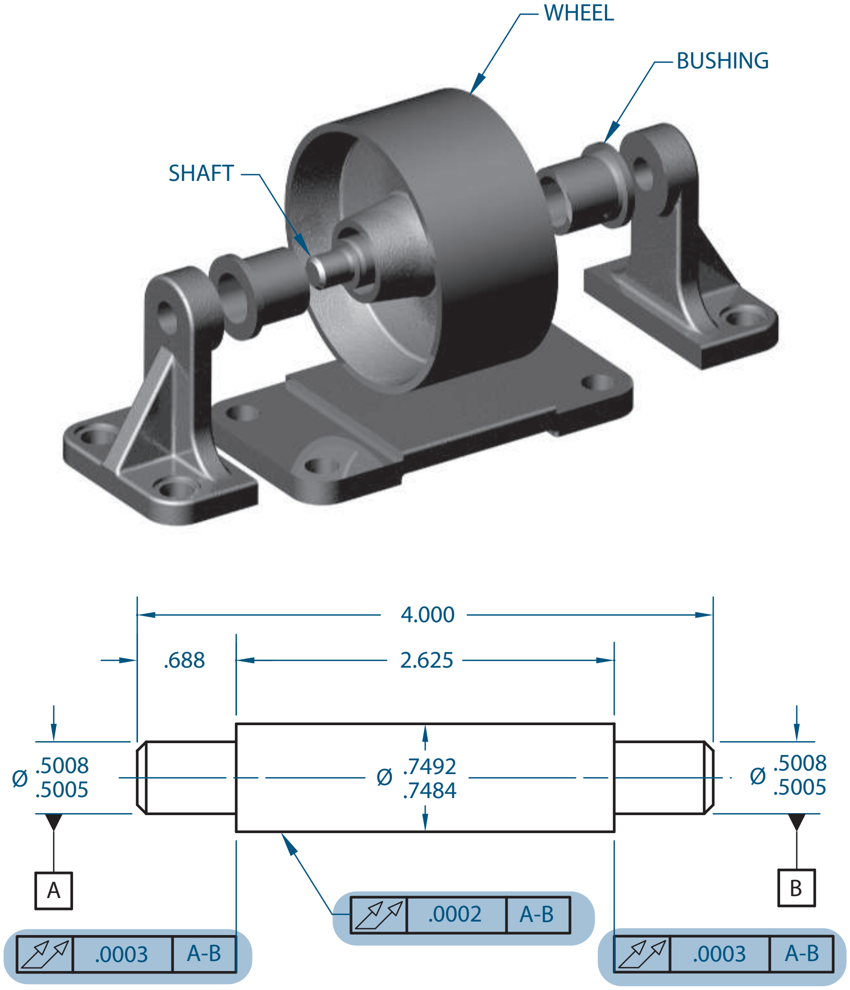

FIGURE 13.89. Using a multiple datum with the runout tolerance.

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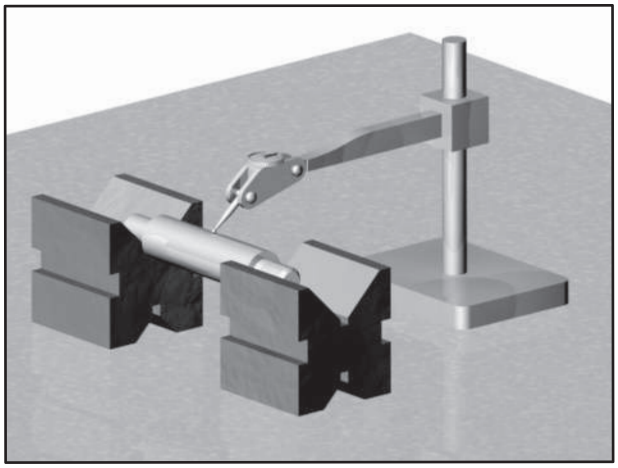
FIGURE 13.90. Inspecting a part with a multiple datum.



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13.07 Chapter Summary

This chapter covered basic information related to conventional tolerancing and GD&T. It began with a discussion of interchangeable manufacturing and explained why it is important to the way modern industry functions. The chapter looked at several topics within conventional tolerancing, such as what tolerance stack-up is, how to specify English and metric fits, and how tolerance dimensions control form and location. With these topics as a foundation, the chapter covered the basics of geometric dimensioning and tolerancing. Included was a discussion of the advantages of geometric tolerancing over conventional tolerancing, the importance of the datum reference frame to establish a coordinate system for design manufacturing and inspection, an explanation of how to read feature control frames, and a description of each geometric tolerance with an example of how to inspect each one.

13.08 GLOSSARY OF KEY TERMS

allowance: The difference between the maximum material limits of mating parts. It is the minimum clearance or maximum interference between parts.

baseline dimensioning: A method for specifying the location of features on a part whereby all the locations are relative to a common feature or edge.

basic dimension: A dimension that is theoretically exact. It is identified by a box around the dimension. It locates the perfect position of features from clearly identified datums.

chain dimensioning: A method for specifying the location of features on a part whereby the location of each feature is successively specified relative to the location of the previous feature.

clearance: A type of fit where space exists between two mating parts.

datum: A theoretical plane or axis established by real features on an object for the purpose of defining the datum reference frame.

datum reference frame: A system of three mutually perpendicular planes used as the coordinate system for geometric dimensioning.

direct dimensioning: Dimensioning between two key points to minimize tolerance accumulation.

feature control frame: The main alphabet of the language of geometric dimensioning and tolerancing. These boxes contain the geometric characteristic symbol, the geometric tolerances, and the relative datums.

feature with size: A cylindrical or spherical surface or a set of two opposed elements or opposed parallel surfaces associated with a size dimension. Typical features with size are holes, cylinders, spheres, and opposite sides of a rectangular block.

feature without size: A planar surface or a feature where the normal vectors point in the same direction.

functional gage: An inspection tool built uniquely for the purpose of quickly checking a specific dimension or geometric condition on a part to determine whether or not it falls within tolerance limits.

geometric dimensioning and tolerancing (GD&T): A 3-D mathematical system that allows a designer to describe the form, orientation, and location of features on a part within precise tolerance zones.

interchangeable manufacturing: A process by which parts are made at different locations and brought together for assembly. For many industries, this process opens the door for third-party companies to produce replacement parts or custom parts.

interference: A fit where two mating parts have intersecting nominal volumes, requiring the deformation of the parts. For example, the diameter of the shaft is larger than the diameter of the hole. When assembled, the intent is that the shaft will not spin in the hole.

maximum material condition (MMC): The condition in which a feature of size contains the maximum amount of material within the stated limits of size.

statistical tolerancing: A way to assign tolerances based on sound statistical practices rather than conventional tolerancing practices.

13.09 QUESTIONS FOR REVIEW

1. Describe interchangeable manufacturing. List two products that you use daily that rely on interchangeable manufacturing.
2. Describe the difference between baseline and chain dimensioning. Make two sketches of the object in Figure 13.106 and dimension it using the two methods of dimensioning.
3. How can you tell if a dimension on a drawing is appropriate for statistical process control applications?
4. What are the differences between clearance, interference, and transition fits? Give an example of a design that requires a clearance fit. Give an example of a design that requires an interference fit.
5. What is allowance as it relates to tolerance dimensioning? If the diameter of a hole is dimensioned as .500–.505 and the diameter of the mating shaft is .495–.498, what is the allowance?
6. Define tolerance as it relates to limit dimensioning. What is the tolerance for the hole in question 5? for the shaft?
7. Define the following terms: feature, feature with size, and feature without size.
8. List three advantages of GD&T over conventional tolerancing.
9. What is the theoretical coordinate system used in geometric dimensioning and tolerancing that consists of three mutually perpendicular planes?
10. Define the following: datum plane, datum simulator, and simulated datum. Give three examples of datum simulators.
11. What are the parts of a feature control frame?
12. Describe how the maximum material condition modifier is used to allow bonus tolerance for a feature.
13. Describe the difference between a flatness tolerance and a parallelism tolerance for a surface. Do these tolerances control form? orientation? location?
14. Describe the difference between the tolerance zone shape for straightness of a surface element and straightness of an axis.
15. What are the three ways in which profile of a line and profile of a surface can be specified relative to the perfect geometry implied on the drawing?
16. Describe the difference between circularity and circular runout.
17. Identify the geometric tolerances that require datum references.

13.10 PROBLEMS

1. Using the tables in the back of the book, add the limit dimensions for the machined hole and shaft shown in Figure P13.1 per the following specifications. If instructed, determine the allowance for each system.
   - **a.** RC1 fit with a basic size of .7500 inch
   - **b.** RC9 fit with a basic size of 1.0000 inch

13.10 PROBLEMS (CONTINUED)

c. FN3 fit with a basic size of .5000 inch
d. FN5 fit with a basic size of 2.0000 inches
e. LT1 fit with a basic size of .8750 inch
f. LT6 fit with a basic size of 3.0000 inches
g. LN1 fit with a basic size of .3750 inch
h. LN3 fit with a basic size of 4.0000 inches
i. LC1 fit with a basic size of .6250 inch
j. LC11 fit with a basic size of .5000 inch
k. Loose running fit (H11/c11) with a basic size of 5.000 millimeters
l. Close running fit (H8/f7) with a basic size of 25.000 millimeters
m. Locational transition fit (H7/n6) with a basic size of 10.000 millimeters
n. Medium drive fit (H7/s6) with a basic size of 15.000 millimeters
o. Sliding fit (G7/h6) with a basic size of 20.000 millimeters

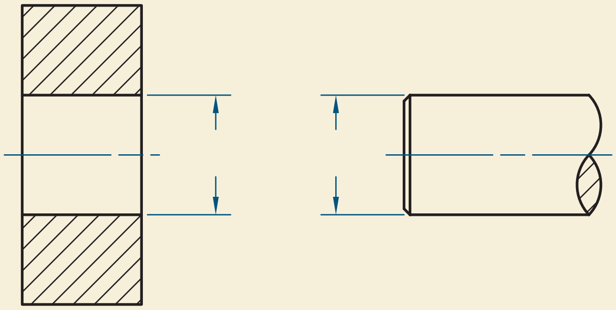

FIGURE P13.1.

2. Given the dimensions in the drawing shown in Figure P13.2, circle the value that represents the maximum material condition value for each dimension.

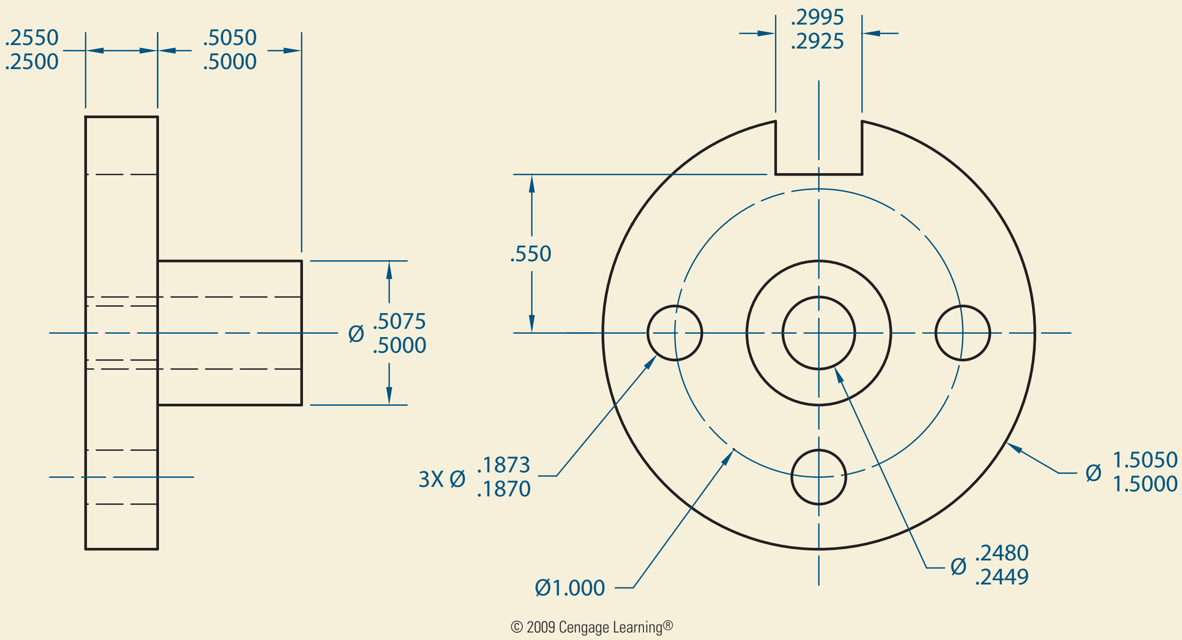

FIGURE P13.2.

13.10 PROBLEMS (CONTINUED)

3. On the machined block shown in Figure P13.3, circle the dimensions that represent features with size.

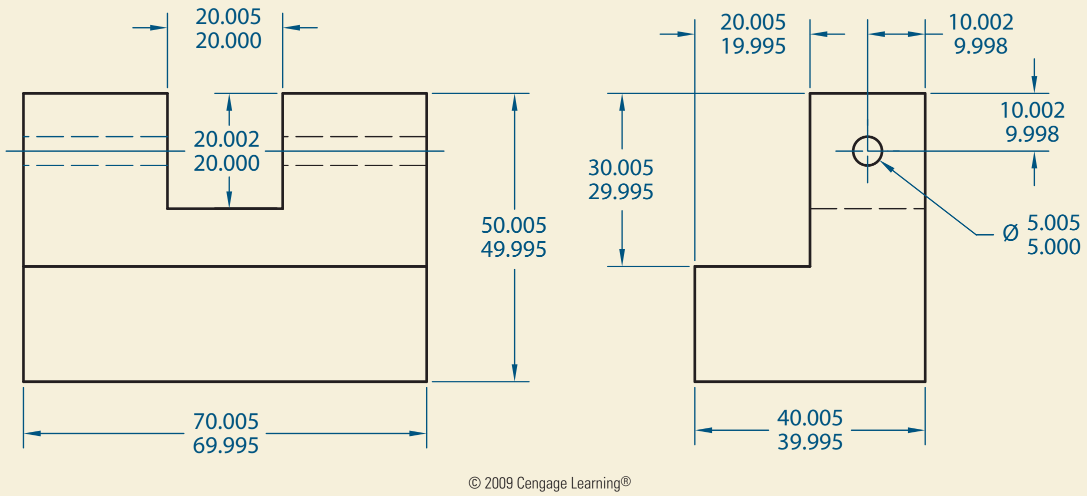

FIGURE P13.3.

4. On the drawing shown in Figure P13.4, apply datum feature symbols per the following specifications:

- **a.** Identify the left-hand face in the left-side view as datum feature A.
- **b.** Identify the bottom surface in the front view as datum feature B.
- **c.** Identify the median plane of the slot in the front view as datum feature C.
- **d.** Identify the axis of the large hole as datum feature D.

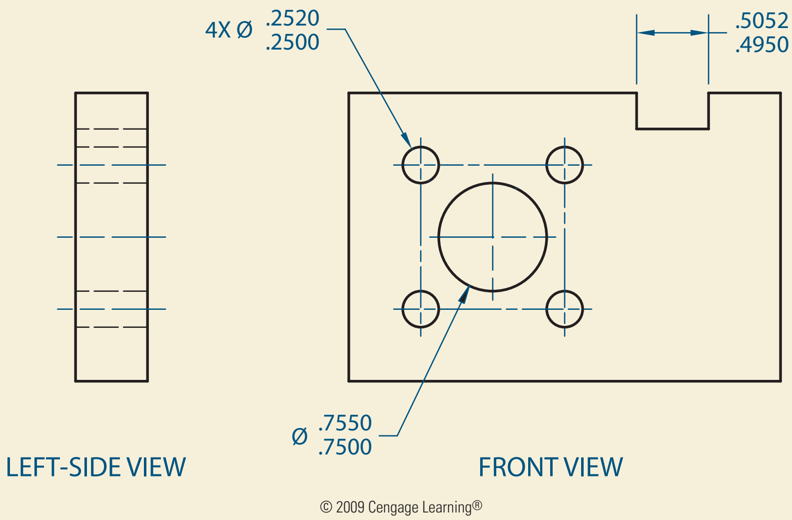

FIGURE P13.4.

13.10 PROBLEMS (CONTINUED)

5. Given the following sentence descriptions of the geometric tolerance information, correctly sketch the feature control frames for each.

- **a.** The feature must be flat within a tenth of a millimeter tolerance zone.
- **b.** The feature must be straight within a thousandth of an inch tolerance zone.
- **c.** The feature must be round within five-tenths of a millimeter tolerance zone.
- **d.** The feature (planar surface) must be perpendicular within a five-thousandths of an inch tolerance zone relative to datum feature A.
- **e.** The feature (axis of a cylinder) must be perpendicular within a one-hundredth of a millimeter cylindrical tolerance zone at maximum material condition relative to datum feature D.
- **f.** The feature (planar surface) must be parallel within a one-thousandth of an inch tolerance zone relative to datum feature A.
- **g.** The feature (axis of a cylinder) must be parallel within a three-hundredths of a millimeter cylindrical tolerance zone at maximum material condition relative to datum feature M.
- **h.** The total runout of the surface must be within a five-hundredths of a millimeter tolerance zone relative to datum feature A.
- **i.** The total surface profile of the surface must be within a two-thousandths of an inch tolerance zone relative to datum feature A.
- **j.** The features (axis of a hole) must be positioned within a five-thousandths of a millimeter cylindrical tolerance zone at maximum material condition relative to primary datum feature D, secondary datum feature E, and tertiary datum feature F.

6. Given the following feature control frames, write the sentence descriptions for each.

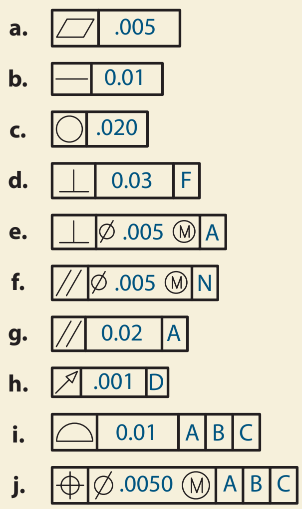

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FIGURE P13.5.

13.10 PROBLEMS (CONTINUED)

7. Place the item number of the following terms in the bold circles on the drawing in Figure P13.6 to identify each symbol.

- **a.** Basic dimension
- **b.** Feature control frame
- **c.** Maximum material condition modifier
- **d.** Datum feature symbol
- **e.** Statistical tolerance symbol

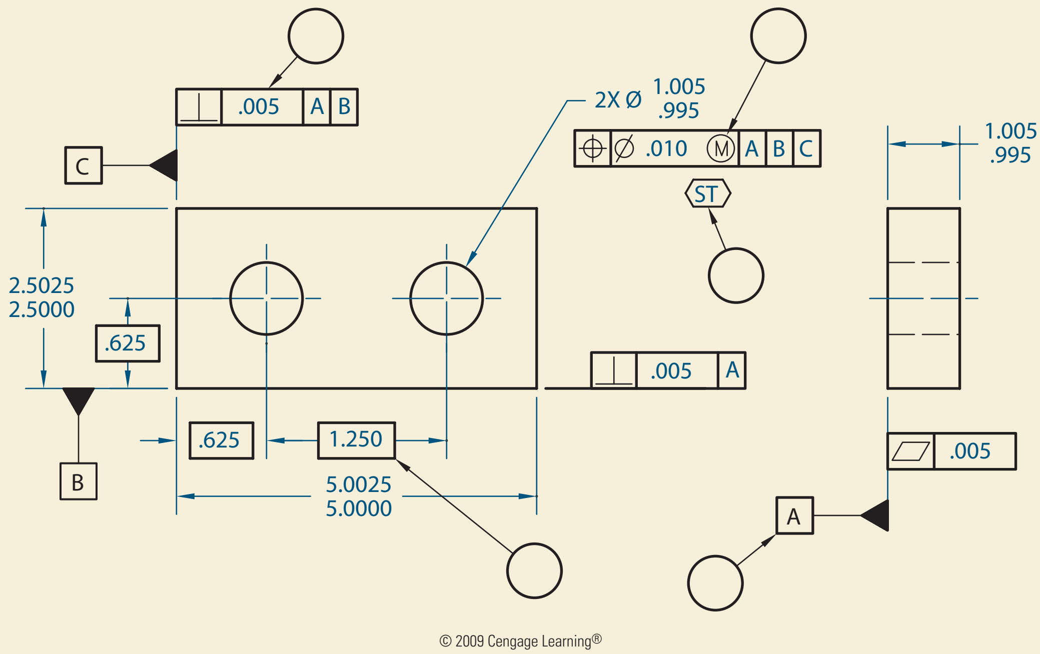

FIGURE P13.6.

8. Given the drawing in Figure P13.7 and the sentence descriptions of the geometric tolerance information below, correctly dimension the drawing.

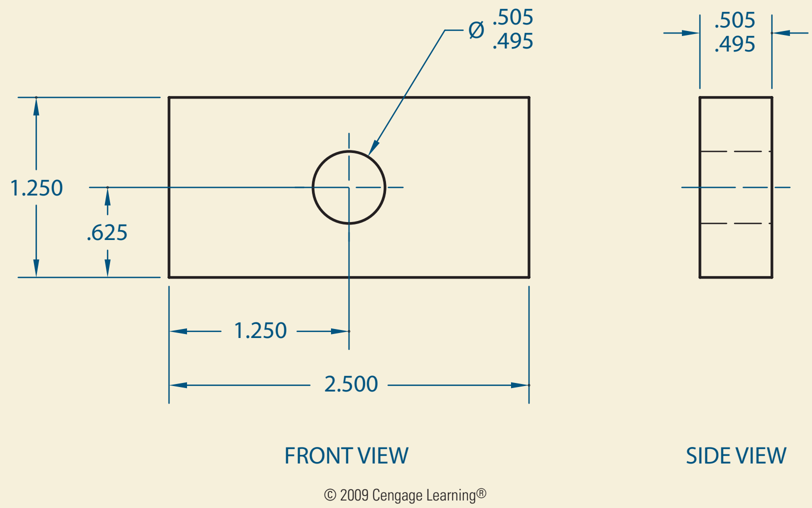

FIGURE P13.7.

13.10 PROBLEMS (CONTINUED)

a. Identify the left-hand surface in the SIDE VIEW as datum feature A. Control it with a flatness tolerance of .005.
b. Identify the bottom surface in the FRONT VIEW as datum feature B. Control it with a perpendicularity tolerance of .005 relative to datum feature A.
c. Identify the left-hand surface in the FRONT VIEW as datum feature C. Control it with a perpendicularity tolerance of .005 relative to primary datum feature A and secondary datum feature B.
d. Make all dimensions basic except for the two limit dimensions.
e. Add a position tolerance for the machined hole. The hole must be positioned within a ten-thousandths of an inch cylindrical tolerance zone at maximum material condition relative to primary datum feature A, secondary datum feature B, and tertiary datum feature C.
f. In the FRONT VIEW, identify the top-left corner as point X and the lower-right corner as point Y. On either the top surface or the right-hand surface, add a profile of a surface tolerance of .010 relative to primary datum feature A, secondary datum feature B, and tertiary datum feature C. Under the feature control frame, identify that the tolerance applies between points X and Y.

9. Given the drawing in Figure P13.8 and the sentence descriptions of the geometric tolerance information below, correctly dimension the drawing.
   a. Identify the right-hand surface of the large cylinder in the SIDE VIEW as datum feature A. Control it with a flatness tolerance of .0050.
   b. Apply a perpendicularity tolerance to the axis of the .5000–.5075 cylinder in the SIDE VIEW. The feature must be perpendicular with a .0030 cylindrical tolerance zone at maximum material condition relative to datum feature A. Identify the axis of this cylinder as datum feature B.
   c. Identify the median plane of the .2925–.2995 slot in the FRONT VIEW as datum feature C. Control this with a position tolerance of .0035 at maximum material condition relative to primary datum feature A and secondary datum feature B at maximum material condition.
   d. Make the .5500, 1.0000, and 1.5000 dimensions basic.

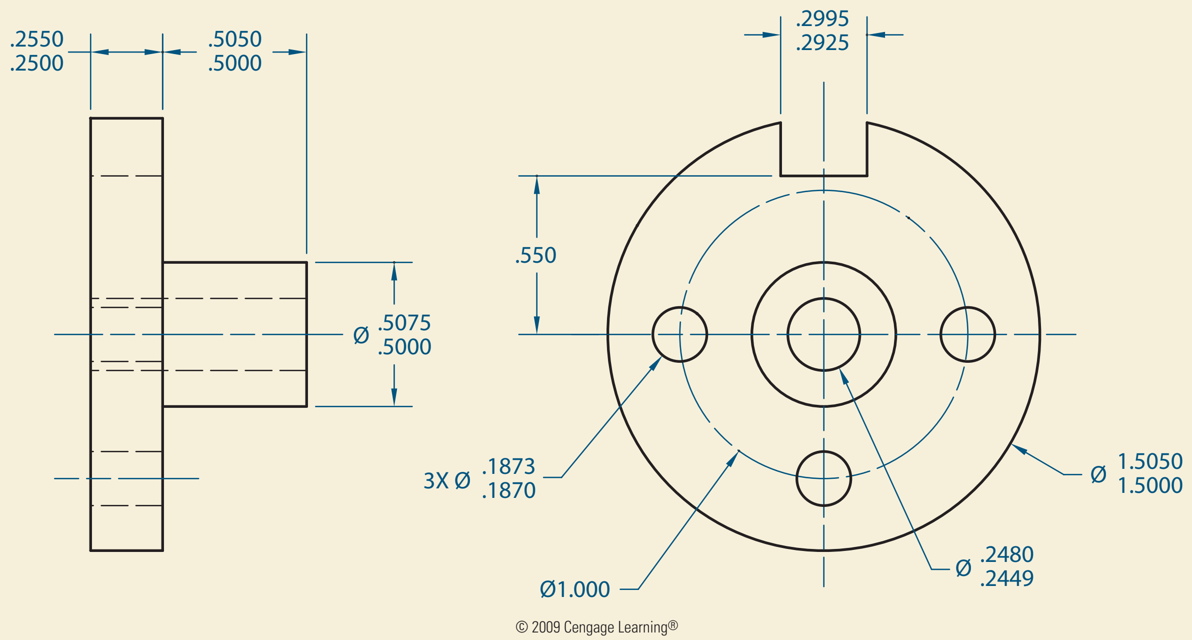

FIGURE P13.8.

13.10 PROBLEMS (CONTINUED)

e. Add a position tolerance for the .2449–.2480 hole. The hole must be positioned within a .0025 of an inch cylindrical tolerance zone at maximum material condition relative to primary datum feature A and secondary datum feature B at maximum material condition.

f. Add a position tolerance for the .1870–.1873 holes. The holes must be positioned within a .0001 of an inch cylindrical tolerance zone at maximum material condition relative to primary datum feature A, secondary datum feature B at maximum material condition, and tertiary datum feature C at maximum material condition.

g. In the FRONT VIEW on the horizontal surface of the slot, add a profile of a surface tolerance of .0020 relative to primary datum feature A, secondary datum feature B at maximum material condition, and tertiary datum feature C at maximum material condition.

h. In the FRONT VIEW on the largest diameter, add a profile of a surface tolerance of .0020 relative to primary datum feature A and secondary datum feature B at maximum material condition.

CHAPTER 14

WORKING DRAWINGS

OBJECTIVES

After completing this chapter, you should be able to

- Specify the contents, formatting, and organization of engineering drawings
- Correctly prepare and interpret formal, professional engineering drawings
- Discuss the primary differences between drawings used in manufacturing projects and those used in construction projects
- Effectively use scales to measure the length of lines on a drawing

14.01 INTRODUCTION

In this chapter, you will learn about conventions and practices used in real-world engineering drawings. You will learn about two primary types of drawings—manufacturing and construction. Manufacturing drawings are used by engineers other than just mechanical engineers, and construction drawings are used by engineers other than just civil engineers; however, for the sake of simplicity in terminology, this chapter will sometimes refer to working drawings as "mechanical engineering drawings" or as "civil engineering drawings" since those two disciplines are concerned primarily with manufacturing and construction drawings, respectively. As a further simplification, the term *object* or *part* will refer to the mechanical parts that have been designed as well as a civil infrastructure project. **Manufacturing drawings** are used for products such as bicycles and toasters. **Construction drawings** are used for roads and bridges. Mechanical drawings depict products that are mass-produced; civil drawings represent unique projects that are known as one-offs. (A **one-off** is a system for which only one such system is constructed.) Although there are similarities between manufacturing and construction drawings, there are also significant differences. In the following sections, you will learn about manufacturing working drawings and construction working drawings. In particular, in the discussion of construction drawings, you will learn about the characteristics that set them apart from typical manufacturing drawings.

When parts and assemblies are ready for fabrication or when structures are ready for construction, the drawings must be presented in a format that is considered formal and professional. There are several reasons for this formality. First and foremost, the drawings must be able to stand on their own without any vagueness or ambiguity. They must be interpreted the same way when viewed by different people. In fact, for complex projects (in particular, for construction projects), many people will view the drawings. Also, the engineer responsible for the design may not be available to answer questions that arise during manufacturing or construction. Formal drawings need to be formatted in such a way that relevant information concerning the specifications, records, and identification of the part is easy to locate and is included with the drawing.

Next, an engineering drawing is considered a legal document and, as such, must contain a certain amount of information concerning the history of and responsibilities for the design. In the case of a set of construction drawings, the seal and signature of a registered **professional engineer (PE)** are typically required. Finally, since engineering drawings are usually presented to third parties for cost estimation or fabrication, the presentation of the drawing is a reflection of the quality of the originator. Drawings that are presented well reflect favorably on the person or company that made the drawings.

When fabricated parts and assemblies are later used in the field or when infrastructure projects are constructed in the field, additional people may need to see the design drawings. These people sometimes include technicians and assemblers who install the parts in the final working environment; salespeople who ensure that the parts are compatible with other products produced by different companies; subcontractors who construct specific systems in the project, such as the wiring or plumbing; maintenance people who repair or replace the parts in the field; government inspectors who monitor the progress of a civil works project; and for certain types of systems, engineers or technicians who are responsible for the removal, recycling, or disposal of the parts at their end of life.

14.02 Making It Formal

If you have decided that you would like someone else to make your parts or device for you, you must produce a set of formal documents known as **working drawings** to send to the fabricator. In civil engineering projects, contractors bid on projects based on the working drawings and specifications. Working drawings show each part or structure in all of the views necessary to fully define their features, their sizes and tolerances, and the way they are to be assembled into the completed product. Consequently, much of what you have learned in the previous chapters concerning orthogonal projection, pictorial views, dimensioning, and tolerancing is used extensively in working drawings. In the ideal case, once you have produced a set of working drawings and they are delivered to the fabricator or contractor, your systems should be able to be fabricated correctly without any further intervention from you. No one should need to call you with questions concerning any feature of the part or the way the part is to be made. In large construction projects, this is rarely the case and the engineer who designed the structure is often extensively involved in overseeing final construction of the project.

Once a drawing leaves your hands as the responsible engineer, it is likely to be reproduced many times and viewed many times by different people. You must have confidence that the information contained on the drawing will be interpreted correctly by every person who views it. In a complex manufacturing project, for example, the first person who will likely see the drawing is a buyer who must evaluate the operations required to fabricate the system and the degree of difficulty of fabrication. Thus, a fabricator with the capability to produce the part will be selected. For construction projects, the first people to see the drawings are usually the contractors who bid on the project. Contractors estimate project costs based on the drawings and specifications; the person with the lowest estimate, or bid, is typically awarded the project. The selected fabricator or contractor must then produce the part or **assembly** as specified on the drawings. Inspectors measure and test the part or materials to ensure that they meet the criteria defined in the drawings and specifications. The engineers and technicians who are responsible for installing the part in the final product must know the sizes of the part's features and their allowable variation so that any special tooling required for the installation can be built. The subcontractors who install various systems on a construction project must know how their portion fits in with the overall structure. This process of design and specification, fabrication, inspection, and installation is shown in Figure 14.01 for the prototype production of a computer disk drive spindle, a typical manufacturing project. Not shown is the special tooling required to ensure the proper alignment of the parts when they are assembled.

When you finish making a working drawing, you have created part of a legal document. Engineering drawings are, in fact, legally binding documents. Once an agreement between you and the fabricator or contractor has been reached for the manufacture or construction of a part or system, the engineering drawing becomes the focal point of the agreement. A working drawing is part of a contract in which a fabricator or contractor agrees to make the specified part in accordance with all of the requirements indicated on the drawing in exchange for an agreed amount of money, products, or services. For manufactured products, an additional agreement usually outlines what information, if any, can be shared with others besides the fabricator. If any information required to make the part is missing on the working drawing, the contract may not be able to be completed. In the worst-case scenario, if any information on the working drawing can be easily misinterpreted, an error in the part may result. In either case, as the originator of the drawing, the fault would lie with you and you may be required to compensate the fabricator or contractor for whatever time and effort was expended in the attempt to make the part.

Manufactured parts that meet all of the requirements specified on the drawing must be purchased for the agreed-upon volume, delivery schedule, and price. Contractors must be paid the agreed-upon amount when the structure is completed. On the other hand, for manufactured products, you can reject delivered parts that fail

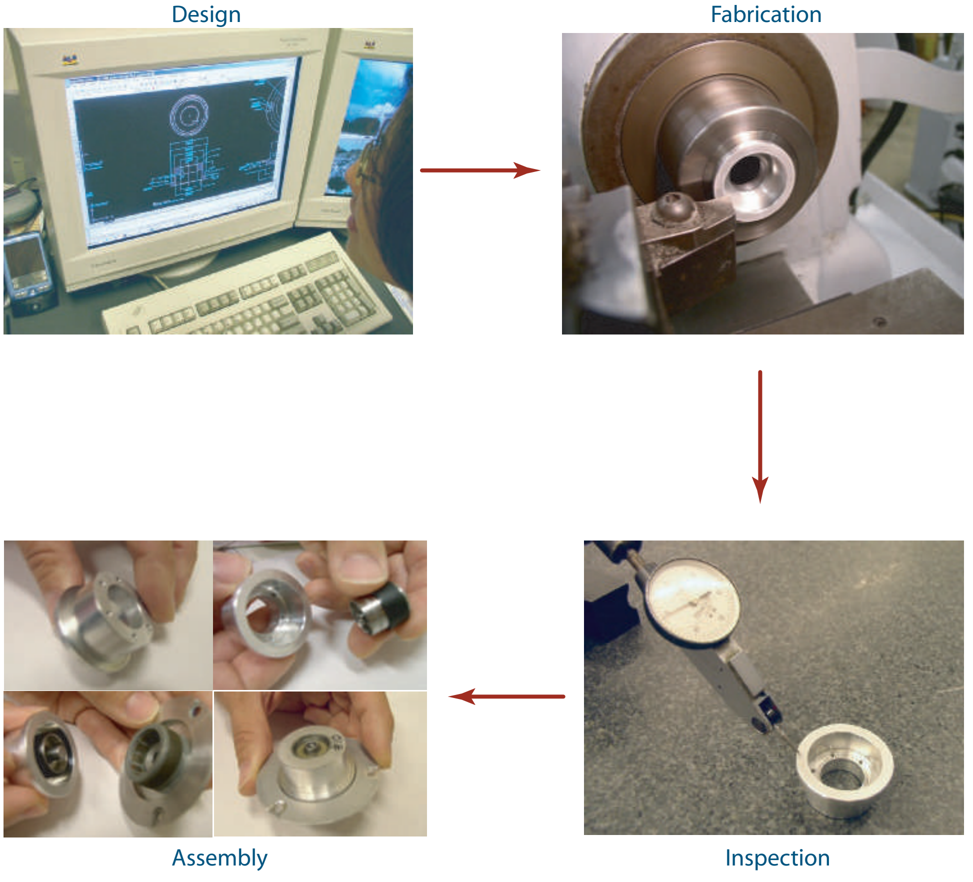

FIGURE 14.01. Some steps in the product development cycle that require the use of working drawings.

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to meet any specification of the drawing. For construction projects, contractors must typically redo the portion of the structure that is not in keeping with the drawings and specifications—eating into their profit margin for the project. If there is an error on the drawing, the buyer must still purchase the manufactured parts if they have been made according to the drawing given to the fabricator. For construction engineering projects, the design firm may have to pay for cost overruns due to incorrect design information on the drawings. Information that is missing on a drawing (and then misinterpreted by the fabricator) is most often considered the fault of the designer. For example, if the numerical dimensions for a part to be manufactured are meant to be in centimeters, this information is missing on the drawing, and the numerical dimensions are interpreted as inches, the error is considered the buyer's fault. Still, these parts must be purchased. Since construction projects are one-off designs, missing dimensions or missing information is a relatively frequent occurrence. Contractors and engineers typically remain in close contact during the construction phase so that these issues can be easily resolved in the field.

Working drawings can usually be distinguished from less formal drawings by their formatting. Just as courts require all submitted legal documents to adhere to a required format and colleges and universities require graduate theses to have a uniform appearance, engineering working drawings also have a prescribed presentation form. Informal drawings, such as those shown in Figure 14.02, have no required formatting and can appear on any size paper; multiview presentations are not required, and dimensions frequently appear on pictorials. Informal drawings can be sketches, can be made with mechanical instruments or CAD, and can include many parts on a single page. Working drawings, on the other hand, are to be of specific sizes and include borders and headers containing specific information. Specific views and presentation techniques are expected. Most of all, working drawings must be complete in providing

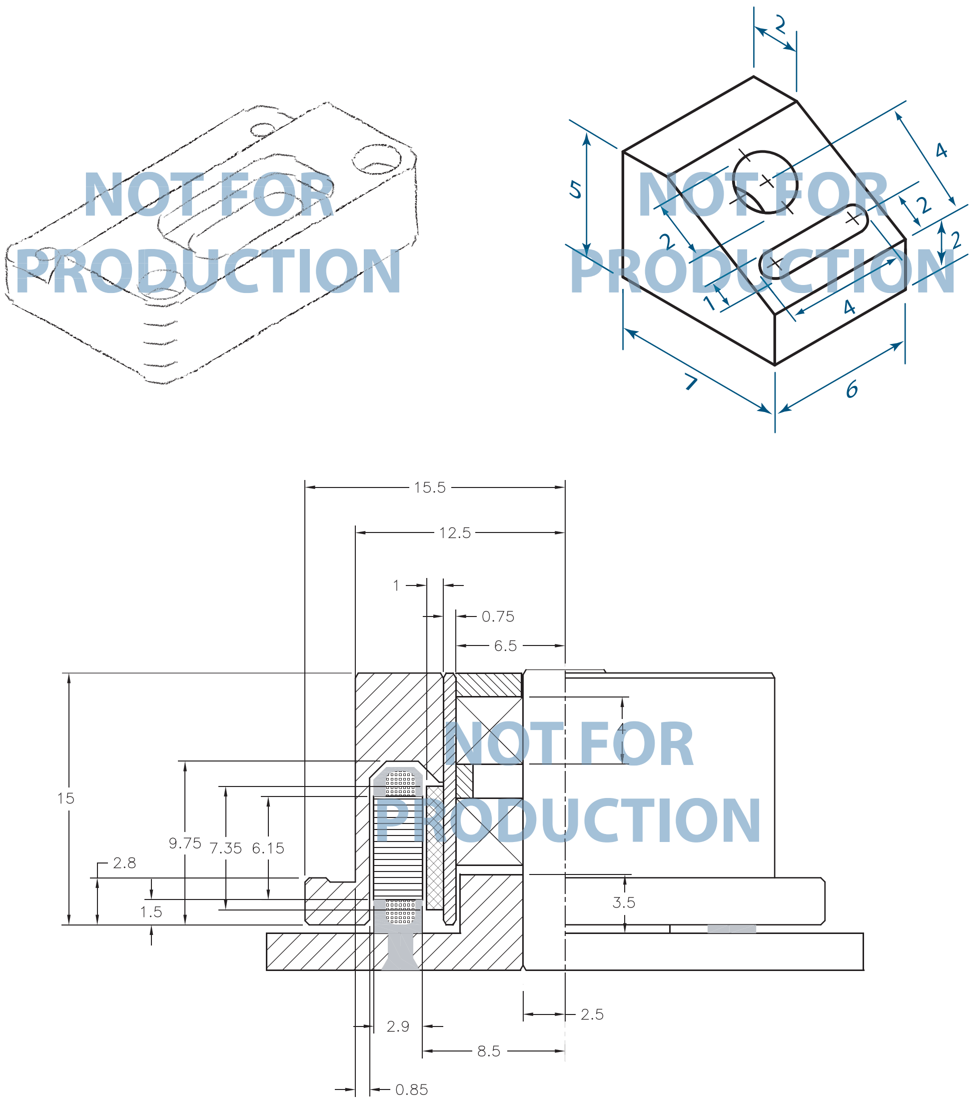


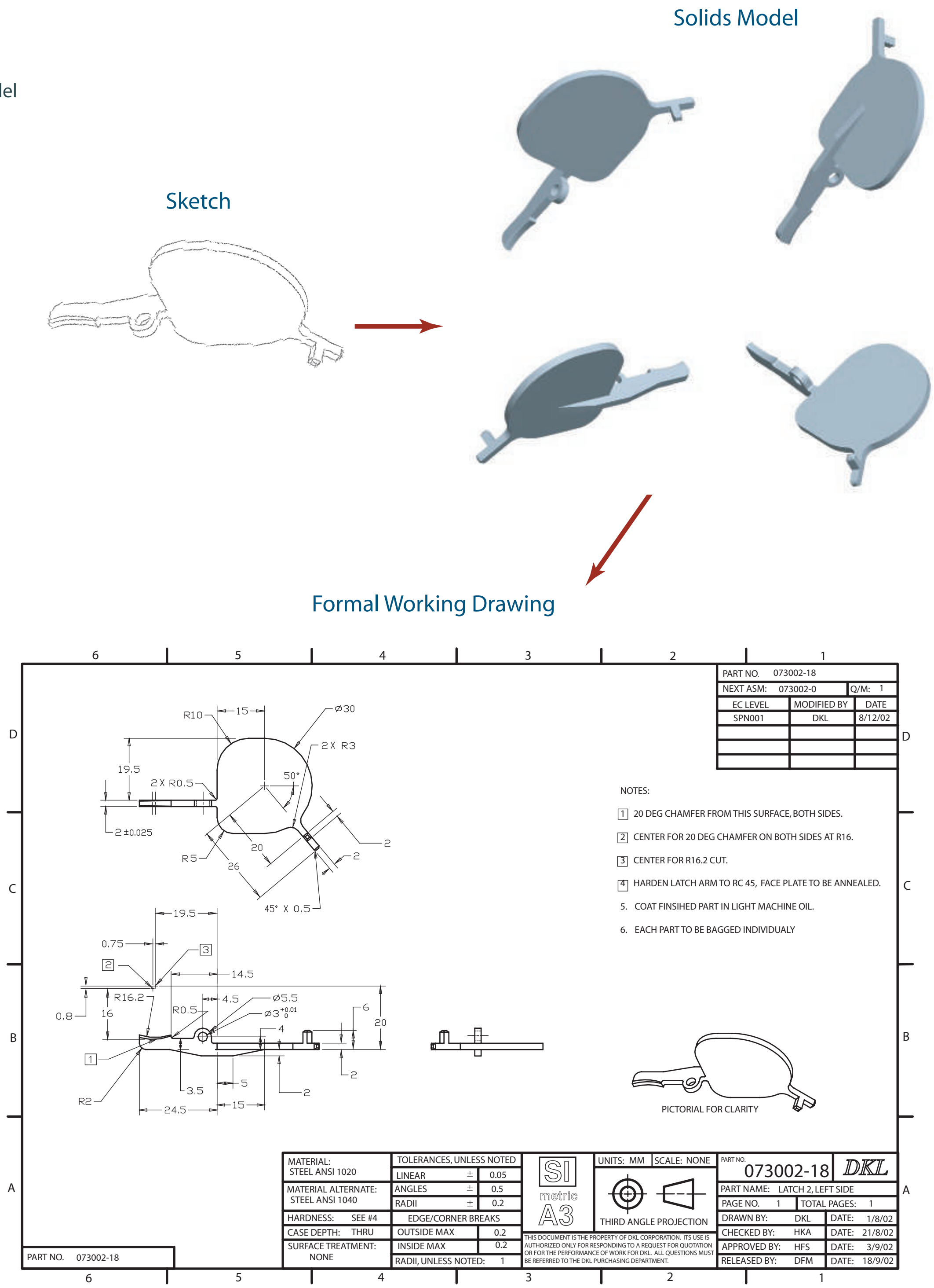
FIGURE 14.02. Sketches, pictorials, and layouts are helpful for visualization and initial sizing but are usually not considered complete, formal drawings.

the information required to make the parts they describe. This is not to say, however, that only working drawings can be legal documents. In civil engineering practice, written specifications usually accompany the drawings and are considered part of the project's legal documentation.

Under certain conditions, informal drawings (even sketches) can be considered legal documents. If, for example, you or your buyer gives an informal drawing to a fabricator with instructions to make the part, even the informal drawing becomes part of the legal contract. The drawing just will not look very nice—or professional. Another example involves patent disputes. The courts may consider the notes and sketches you make in your engineering notebook to be legal documents for establishing the date of conception of an idea. For this purpose, you should have a witness sign and date any notes, sketches, or drawings that you produce that may lead to a patent.

In the engineering and business worlds, appearances are important. The progression from informal drawings to formal drawings, as shown in Figure 14.03, is in many ways a transition in appearance and presentation. Many people consider a formal engineering drawing to be not only a means of information transfer but also a work of art. The presentation of this document can reflect well or poorly on its originator. For that reason, the formal drawings you submit should be well organized, neat, and polished—a part of engineering professionalism.

FIGURE 14.03. The typical progression of the design of a part from a conceptual hand sketch to a computer 3-D model to a formal working drawing extracted from the model.



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14.03 Sheet Sizes

The first step in making a formal working drawing is to choose an appropriate sheet size. This statement might sound strange today, when computers can generate a drawing of almost any size with the part views shown in any scale. However, most working drawings need to be printed for easy viewing, perhaps by a machinist trying to make the parts in a shop, a contractor examining the drawing in the preparation of a bid, or a group of engineers sitting around a conference table reviewing the design. When a working drawing is printed to its intended size, it must be readable.

Most of the world, with the exception of the United States, uses **international sheet sizes**. The most common international sizes are A4, A3, A2, A1, and A0. Size A4 is 297 mm × 210 mm. (For anyone not well-versed in metric sizes, this is approximately the size of the paper used in a computer printer.) If the horizontal dimension size is larger than the vertical size, the paper orientation is known as **landscape**; otherwise, the orientation is called **portrait**. Landscape paper orientation is used almost exclusively in engineering working drawings. The next largest paper size, A3, is generated by attaching two A4 "sheets" along their lengths, producing a sheet that is twice the area of an A4 sheet, or 420 mm × 297 mm. The A2 sheet (594 mm × 420 mm) is similarly produced by putting together two A3 sheets. The A1 (840 mm × 594 mm) and A0 (1188 mm × 840 mm) sizes are generated similarly. The A0 size is generally accepted as the largest size that will fit, without rolling or folding, inside available cabinets made for drawing storage.

U.S. sheet sizes, which are designated A, B, C, D, and E, are close to the international sheet sizes. Size A paper is 11" × 8.5" and is commonly called letter size. As with the international paper sizes, each increasing U.S. sheet size is generated by attaching its two smaller sizes along their lengths. Thus, a B size sheet is 17" × 11", a C size sheet is 22" × 17", etc. An E size sheet, which is 44" × 34", is the largest drawing size that will fit easily inside a common filing cabinet for drawings. Civil engineering drawings are usually drawn on E size paper unless a bound book of B size drawings is created for a project. The common international and U.S. sheet sizes are shown in Figure 14.04.

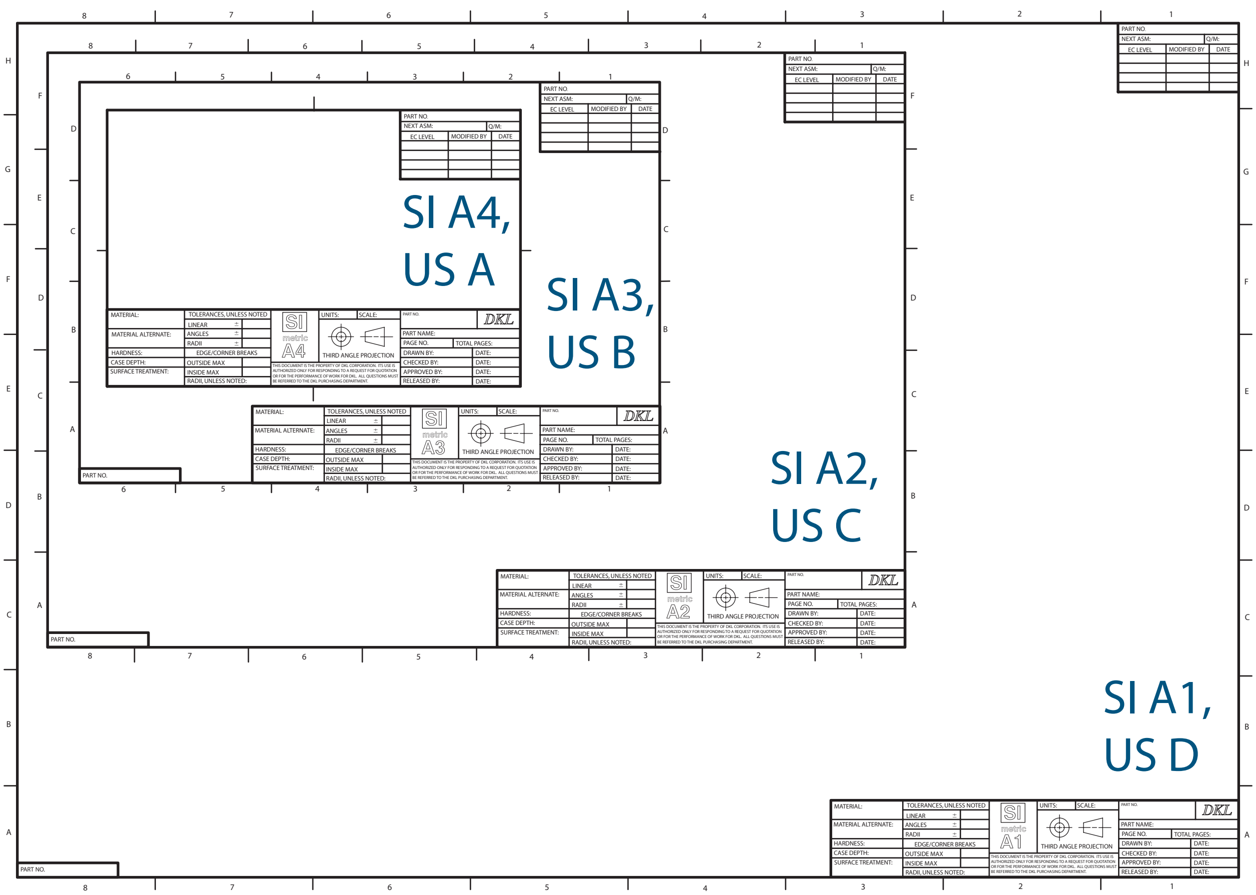

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FIGURE 14.04. Some relative standard sheet sizes, international and U.S., used for formal drawings.

Some caution is necessary when you are printing a drawing less than its full size, which provides convenience in printing, copying, and handling. In large construction projects, printing a drawing to less than full size is a necessity. The font size used for the dimensions and notes on a working drawing is usually 3 mm to 6 mm in height and is independent of the size of the drawing; that is, the font size on an A4 drawing is the same as that on an A0 drawing. If you want to see the notes and dimensions printed to their full size, the drawing needs to be printed to its full size. If, for example, an A1 or A0 size drawing is reduced to an A4 size, the notes and dimensions may be reduced to the point where they are no longer legible. This effect is demonstrated in Figure 14.05 as larger drawings are reduced to a smaller sheet size. Fortunately, a larger printer, such as the one shown in Figure 14.06, is fairly easy to find, and drawings can be easily printed to a size that is legible and convenient for handling.

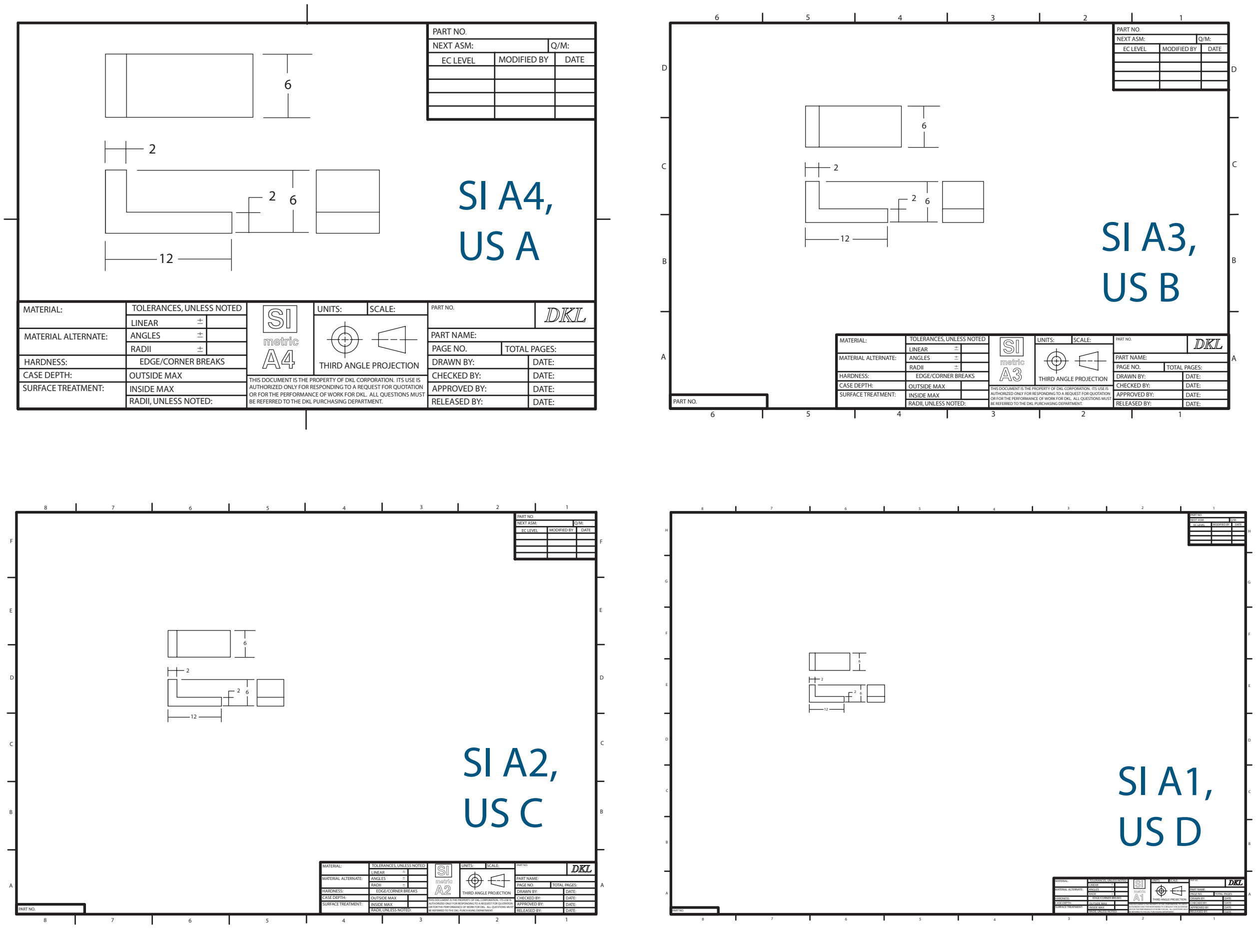

FIGURE 14.05. Geometry and letter font size reduction when larger drawings are printed to smaller sheets.

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FIGURE 14.06. An ink-jet printer capable of creating a full size international A0 or U.S. size E drawing.

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The previous discussion covered working drawings in a generic way. The similarities and some differences between manufacturing and construction drawings were discussed. In the following sections, the discussion of manufacturing and construction drawings will diverge. You will first learn about manufacturing drawings and then about construction drawings.

14.04 The Formal Drawing Header in Manufacturing Drawings

In addition to the information about the geometries of the parts or assemblies, working drawings need to contain other information. There must be information, for example, on how each part can be uniquely identified; otherwise, it may be difficult to locate specific parts and drawings among the vast numbers of parts being manufactured or assembled at any given time. Also, there must be some information about the history of a part or an assembly; otherwise, it may be difficult to distinguish modified new parts from original old parts as the design progresses. If the design for a part does change with time, there must be a way of recording those changes so that everyone working with the part knows what it looks like, how it fits into other parts, and how it performs in the completed device. Is it a new style part or an old style part? If it is an old style part, how old is it? How many changes has it undergone? Can the old style be used instead of the new style?

There also must be a sense of accountability for the design of a part. Who made the drawing? Which engineer was responsible for the part's proper function? Who approved the release of the drawing outside the company? When were those things done? Is the information contained on a drawing considered confidential to the originator? Most of those questions can be answered by examining the drawing header, which is described next.

A **header** is a printed frame or outline on which a drawing is created. Drawing headers are usually unique to the company that produces the drawing, but they follow a similar format and contain the same type of information. A typical header for a manufacturing drawing is shown in Figure 14.07. A heavy line **border** defines the limit of the formal drawing area. Any added markings that are to be a part of the drawing must be inside this border. On some headers, evenly spaced **location grid** marks appear in the horizontal and vertical directions outside the border. The location grid on a drawing, similar to the location grid on a street map, helps readers of the drawing locate areas on the sheet where specific features can be found. For large drawings with many features, the location grid is particularly useful. If, for example, you were told to look for a specific feature at location C5 on the drawing, you would immediately begin looking at the double highlighted area shown in Figure 14.07. A major part of the header is the **main title block**. The main title block contains most of the information required to identify the part on the drawing as well as to track its progress in the design cycle. The main title block provides space for specifying the material and the material processing required to fabricate the part. Some companies provide a **secondary title block** for additional information a company would like to see included on its manufacturing drawings.

The main title block contains information on how to interpret what is seen on the drawing, as highlighted in the magnified portion shown in Figure 14.08. The definition of the units for the dimensions is specified there. Usually the units are specified as MM (millimeters), CM (centimeters), M (meters), IN (inches), or FT (feet). In addition, words and/or graphics specify whether the orthogonal views on the drawing are produced using first-angle projection, which is popular internationally, or using third-angle projection, which is used in the United States. The scale of the drawing is defined as "the ratio of the size of the actual part to the size of the image of the part shown on the drawing when the drawing is printed to its full sheet size."

The main title block also contains information about who owns the drawing and the information it contains, as indicated in Figure 14.09. A manufacturing drawing

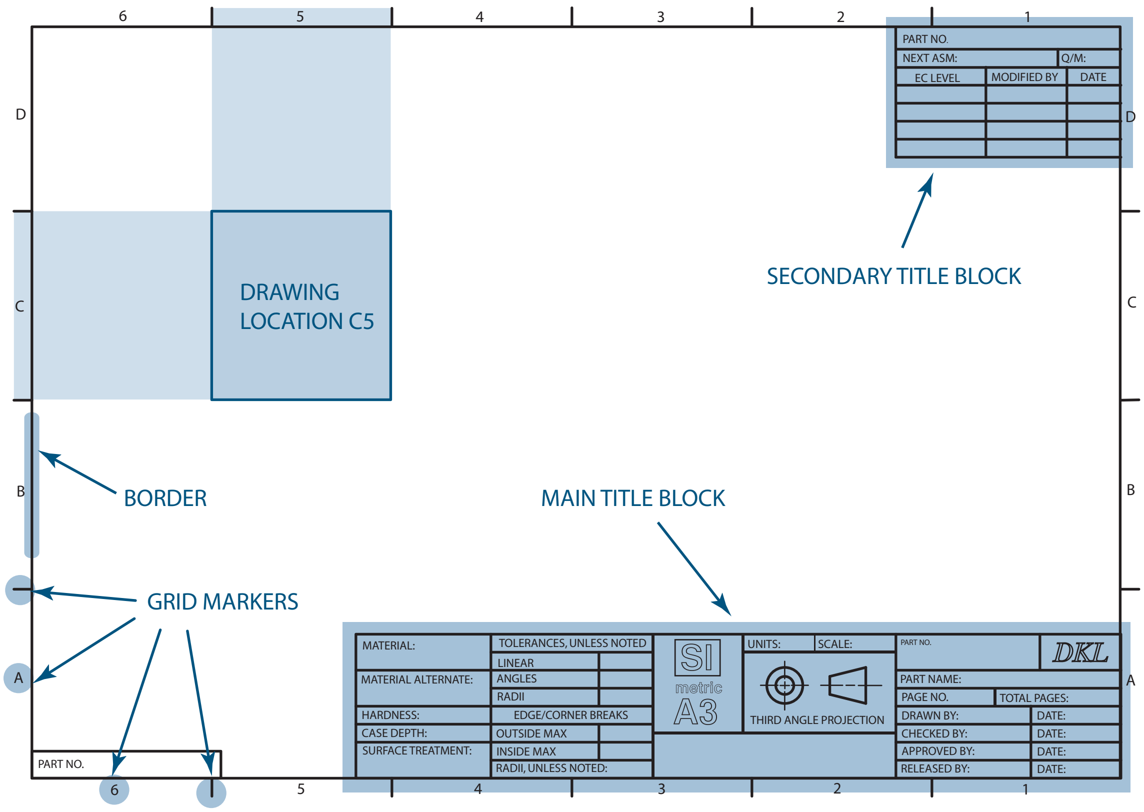

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FIGURE 14.07. A typical header for a formal engineering drawing.

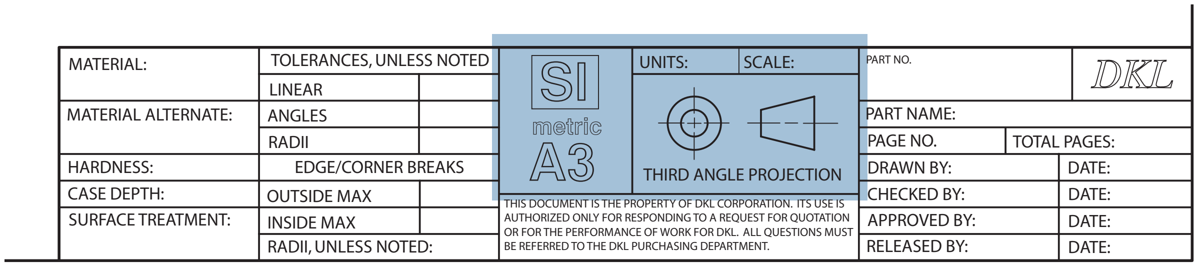

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FIGURE 14.08. Inside the main title block, the definitions of the units, scale, and projection angle help the reader to correctly interpret the placement and orientation of the views and the dimensions that are shown on them.

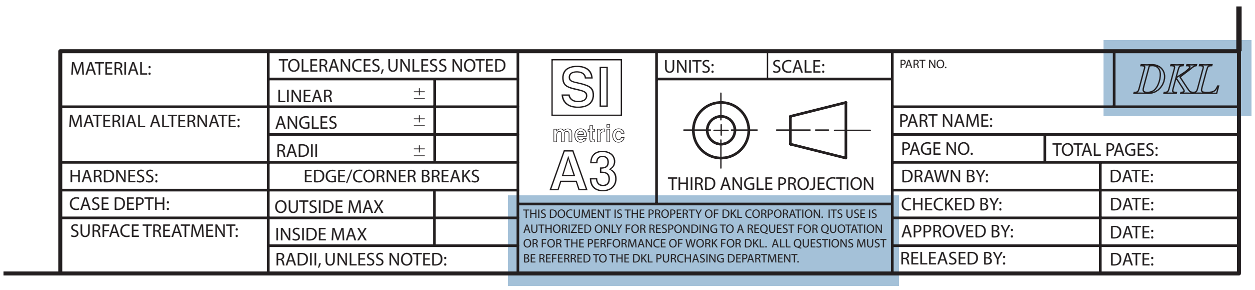

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FIGURE 14.09. The name of the company, a statement of ownership of the drawing contents, and conditions of use are permanently printed in the title block.

and any of its copies are usually considered the property of the company for which the part is made. Usually, the name, initial, or logo of the company (in this case, DKL Corporation) is displayed and some sort of message expresses how the information contained in the drawing can be used and distributed. If this information is considered confidential, or proprietary, it is clearly marked as such. The purpose of these statements is to ensure that the information contained in the drawings is not freely distributed, especially to the company's competitors.

The main means of identification of a part is through its **part number**, as highlighted in Figure 14.10. Every individual part that is fabricated according to the same drawing (or a copy of it) carries the same part number. Ideally, all parts with the same part number should be interchangeable. When it is important to identify each part fabricated from the same drawing, those parts can be assigned unique serial numbers. Since assemblies and subassemblies are often handled and transported as complete units, they also are assigned part numbers. Every company has its own method of assigning part numbers. Some are alphanumeric strings, some include information on the date the number was assigned, and some include coded information on the project type or the location of the engineering facility. Whether an individual part, a subassembly, or a full assembly is defined, part numbers must be unique within a company. A firm usually has an internal accounting system for assigning and tracking part numbers to ensure that no two unique parts are given the same number. In addition to the part number, a part, subassembly, or full assembly is usually given a **part name**. A part name is given for convenience and is usually based on a part's function or appearance, such as L-bracket, or Base Plate, or Pillow Block A. Part names do not have to be unique since they are meant to provide temporary convenience for identifying parts while they are in fabrication or use. A company could have several L-brackets defined for use in various assemblies; however, each unique L-bracket is defined by a unique number.

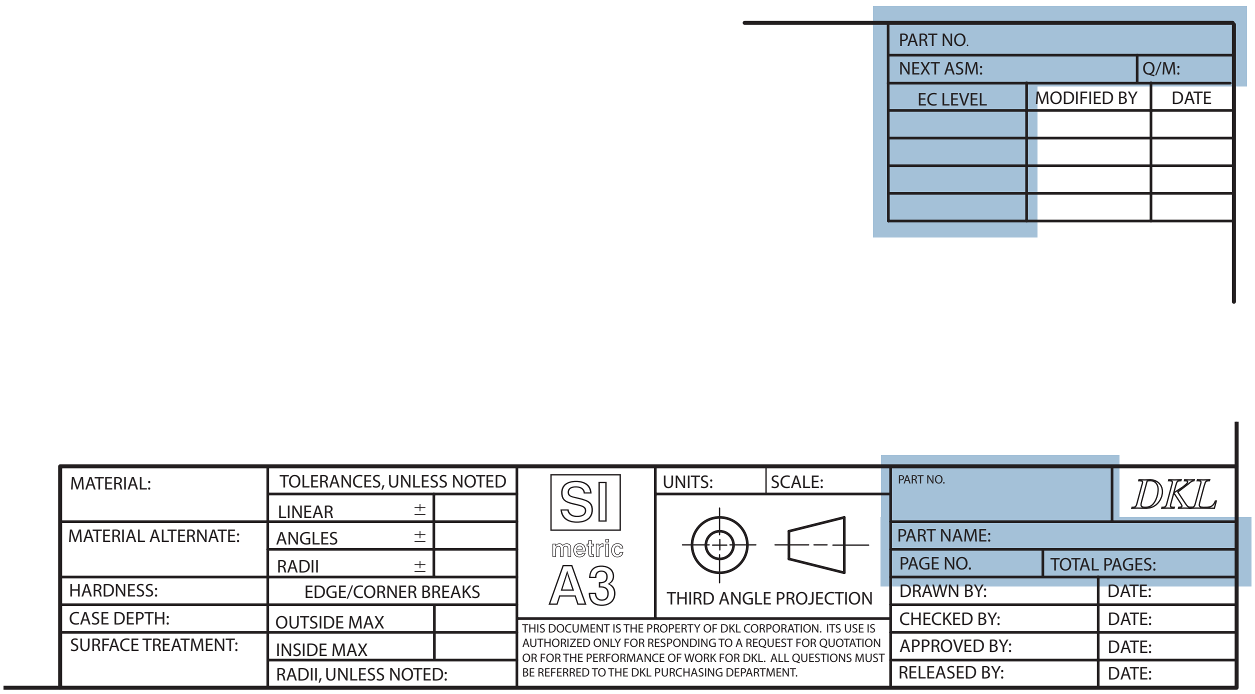

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FIGURE 14.10. The part name, part number, and revision (or EC) number uniquely distinguishes this part from different parts or earlier versions of the same part.

If a part is modified during its life, the drawing is given an **EC Level** number. EC is an acronym for Engineering Change; it also may be called Revision Level or something similar depending on the terminology a company uses. The existence of an EC Level number is an indication that the original design has been updated in some way (e.g., changes in the material or in one or more dimensions or tolerances). In Figure 14.10, the EC Level appears on the secondary title block. Different parts and assemblies can have the same EC Level number if they are from the same product and were updated at the same time. As with part numbers, EC Level numbers can be an alphanumeric string and cannot be reused on the same part or assembly after they have been assigned. If the design of a part has changed significantly to the point where it is no longer interchangeable with the older versions of the part, that part should be assigned a new part number rather than a new EC Level number.

Drawings with large numbers of detail views and notes may require more than one sheet. In this case, each sheet must have a page number and specify the total pages in the entire drawing. Some companies require that each part specifies its next assembly, which is the part number of the assembly or subassembly into which the part is to be immediately installed. If the next assembly requires more than one of a particular part, that quantity is specified as the **quantity per machine (Q/M)**.

A chain of responsibility is required for all manufacturing drawings. The people responsible for the creation of a drawing must be identifiable should any questions arise about the drawing's contents. Every formal drawing has areas for **approval signatures** in the main title block, as shown in Figure 14.11, where the appropriate people can initial and date the drawing. (However, with most CAD drawings, the initials are no longer handwritten; rather, they are inserted as a drawing note.) The required signatures usually include those of the drafter who made the drawing (drawn by…), the person who reviewed the drawing to make sure it was free from errors (checked by …), the designer or engineer who checked to make sure the fabricated

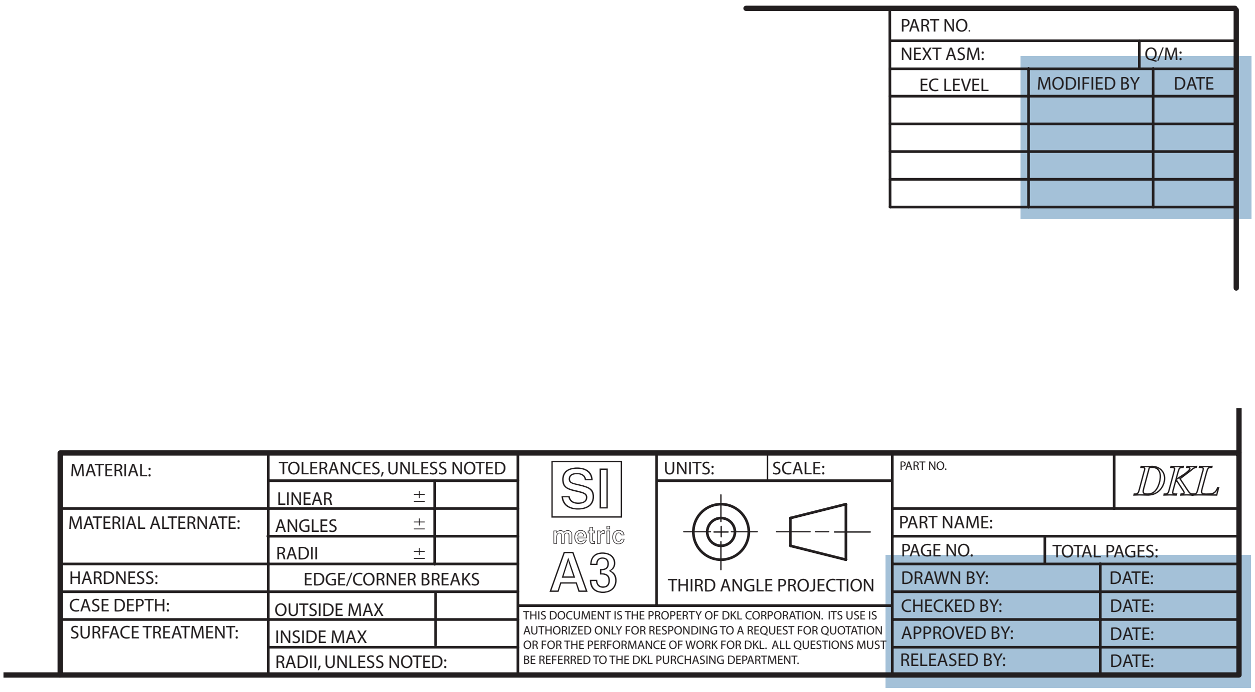

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FIGURE 14.11. Signatures and dates help establish the history of development and leave a trail of accountability.

part would fit and function in its intended manner (approved by …), and a manager who checked that the formal drawing would meet all accounting and security requirements when delivered to a fabricator outside the company (released by …). In smaller companies, it is common to see two or more of these functions performed by the same person.

The main title block usually includes spaces where additional information required for the fabrication of the part is contained, as shown in Figure 14.12. The reason for these spaces is to prompt the entry of additional information; usually, the information required for the fabrication of the parts is included on the drawing. Typical additional information might include the material from which the part is made and any special heat treatment or surface treatments that are required. Other important information includes the **default tolerances**, which are the dimensional tolerances that may be assumed when no tolerance appears with the dimension. Using default tolerance in this manner saves effort in assigning a tolerance to every dimension when the tolerances are the same and generally gives the drawing a neater appearance. Default tolerances can be specified to be different according to the number of decimal places shown on the dimension. Dimensions that have one, two, or three decimal places can be assigned different default tolerances, usually with stricter tolerances used as the number of decimal places increases.

If the drawing extends to multiple pages, the header blocks on the subsequent pages can be simplified, as shown in Figure 14.13. These simplified blocks are called **continuation blocks** and usually contain information that identifies the sheets as being part of a larger drawing. This information includes the part number, EC number, and sheet number of the drawing. The size of the sheet is also included in case the sheets of the drawing are of different sizes.

If the company has no preferred standard **title blocks** for its drawings, ANSI has recommended the use of some generic title blocks, which are shown in Figure 14.14.

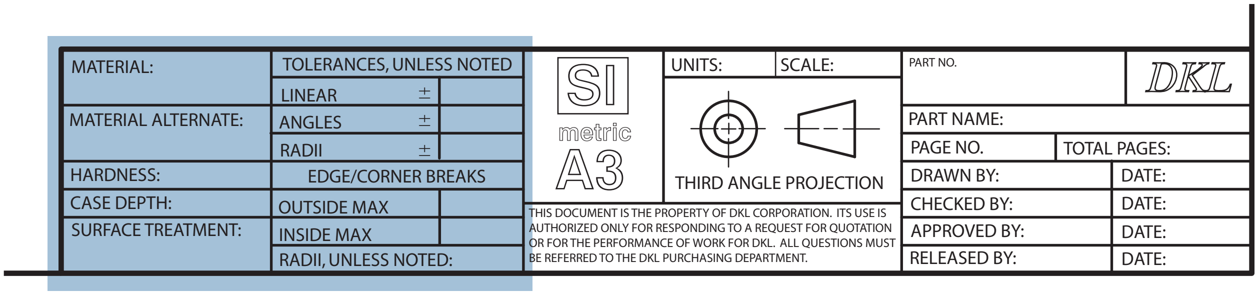

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FIGURE 14.12. Areas for entry of information about material, hardening, surface treatment, and tolerances ensure that this information is not neglected.

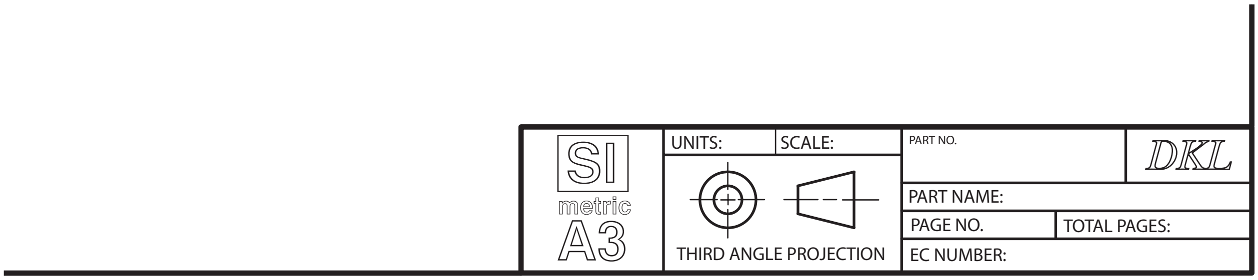

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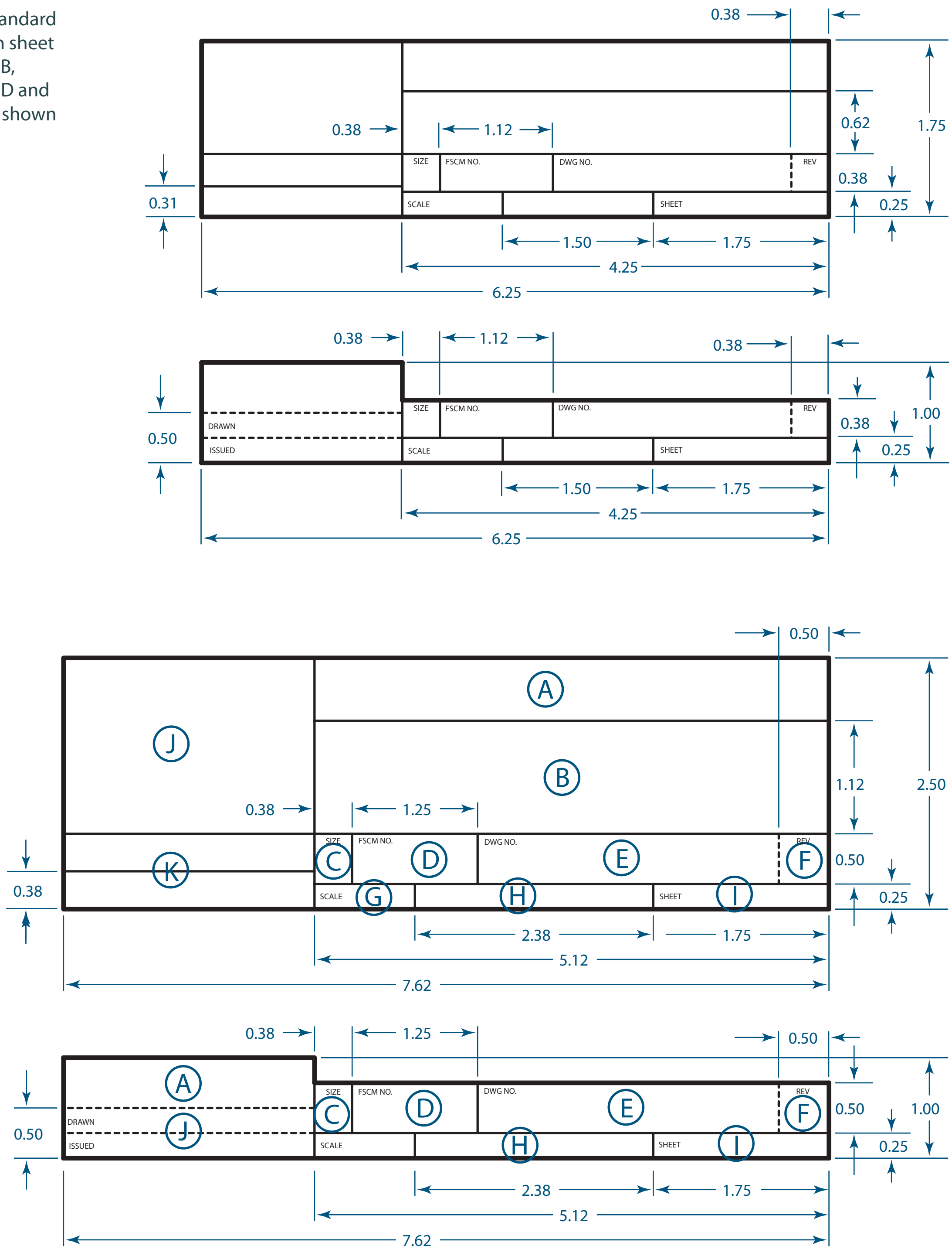
FIGURE 14.13. When a drawing is composed of multiple pages, the title block on the second and subsequent pages may be simplified.

FIGURE 14.14. An ANSI standard title block and continuation sheet block for U.S. sheet sizes A, B, and C (above) and for sizes D and larger (below). Dimensions shown are in inches.



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The information contained in the generic ANSI title blocks is representative of that expected to be contained in most formal drawing title blocks. This information is entered in the proper areas of the ANSI title block as indicated in Figure 14.14 and as listed here:

A. A statement of origin or ownership of the drawing
B. The title of the part or drawing
C. The size of the sheet when the drawing is printed to its full size

D. The Federal Supply Code for Manufacturers (FSCM) number if the work is being done for the federal government

E. The drawing number or part number

F. The revision, or EC, number

G. The ratio of the item size shown on the drawing versus that of the actual item

H. The approximate weight of the item if it is heavy

I. The page number (for drawings with multiple pages)

J. Names of the drafter and checker, with dates

K. Names of any additional people needed to approve the drawing, with dates

14.05 The Drawing Area for Manufactured Parts

The drawing area is defined as the area inside the border of the drawing. It is informally subdivided in an area reserved for showing geometry and an area for the **notes**, as shown in Figure 14.15. Notes are usually listed in one corner of the drawing, and the rest of the drawing area is dedicated to showing the views of the part.

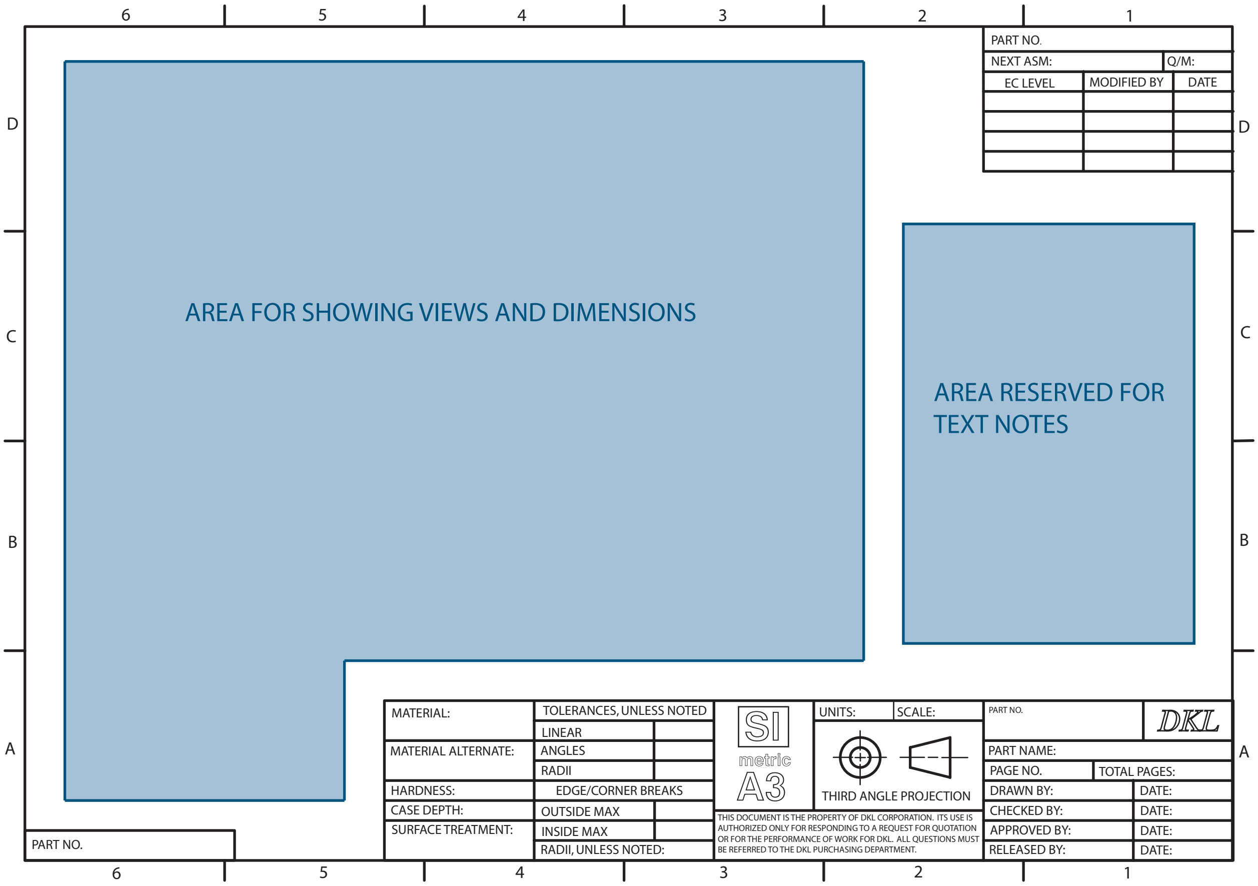

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FIGURE 14.15. For clarity, text notes are kept in a separate area on a formal drawing so a reader can look in a single location to find all of them.

14.05.01 Geometry Presentation

The size of the sheet and the scale of the object should be selected so that the area reserved for showing the object's geometry is uncluttered, even after the dimensions are added. These choices are rather subjective, and looking at how other (good) formal drawings have been prepared gives you some clues about how to make these choices. Simple objects with few geometric features and dimensions generally require smaller sheets. Complicated objects with many features and dimensions require larger sheets or perhaps multiple sheets. Some consideration, however, should be given toward handling convenience. Smaller sheets are easier to store and carry, whereas larger sheets are more difficult to store but are better for showing to a group of people.

14.05.02 Object Views

Whether the object is a part or an assembly, its geometry must be presented using the rule of orthogonal projection and multiview representation. Further, unless there is a good reason not to do so, the views to be shown are those of the preferred configuration (i.e., the front, top, and right-side views, as shown in Figure 14.16). Cases where more or fewer than three orthographic views should be used are shown in the chapter on orthogonal projection and multiview drawings. A more complex part requiring more views is shown in Figure 14.17. For this part, a left-side view has been added to show a cutout and threaded holes on that side of the object. Using the preferred configuration and hidden lines alone to show those features would have

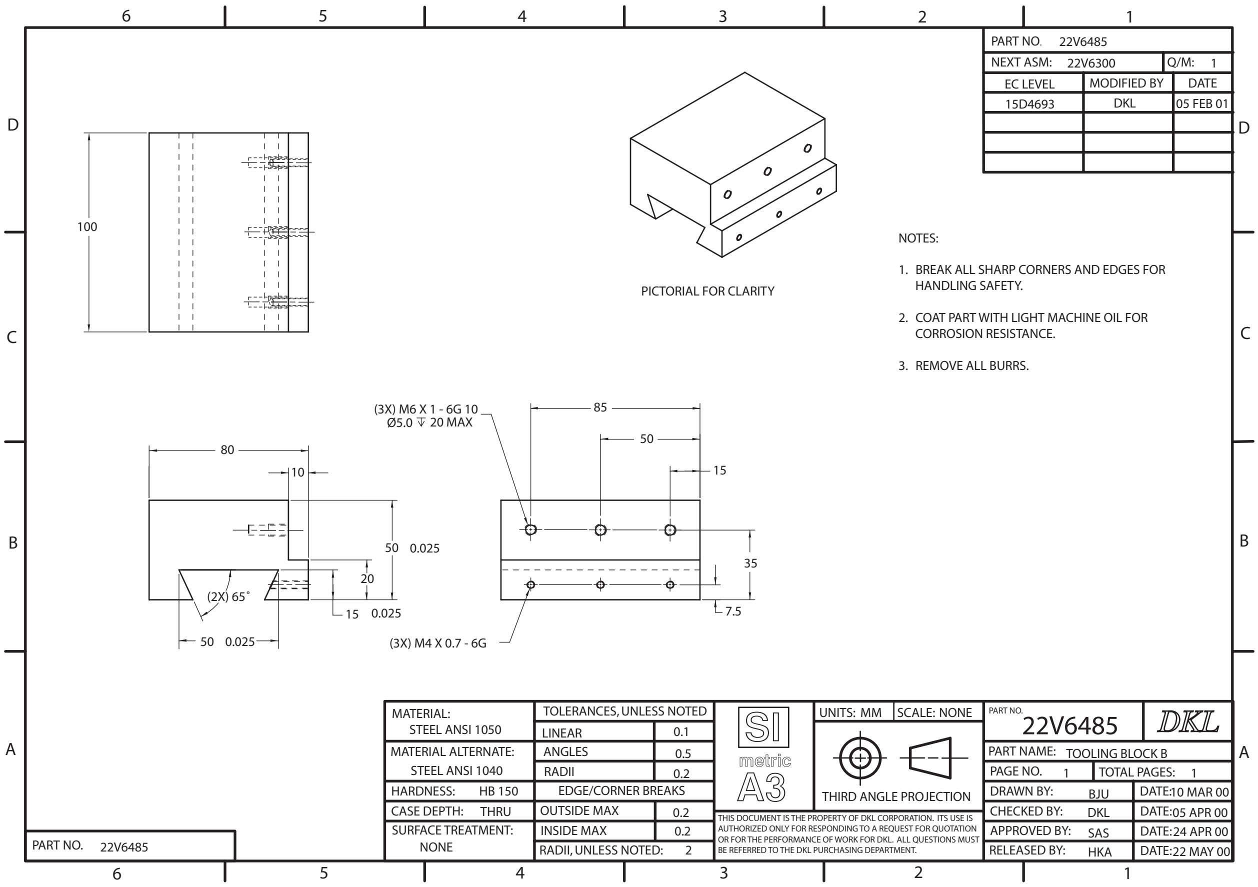

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FIGURE 14.16. A formal drawing of a tooling block presented in the preferred multiview configuration showing all hidden lines. All of its features can be shown using the top, front, and right-side views.

required that dimensions be applied to the hidden lines, which produces views that are more complex and difficult to interpret.

The orientation of the object must be such that these three views show as many visible edges in their true length as possible. The guidelines for adding additional drawing details and additional views are as follows:

1. Start by showing the object in the preferred configuration (i.e., the top, front, and right-side views). Orient the object such that as many edges as possible are shown in their true length in these views. If a view adds no additional information to the presentation, it may be removed.
2. Add more of the standard orthogonal views (e.g., left-side, bottom, and/or back view) as necessary so that dimensioning can be applied to visible edges or features only.
3. Add all hidden lines from the exterior edges and interior detail that are not visible.
4. If there are too many hidden lines and the views are confusing, remove the hidden lines that are not necessary for fully defining the geometry or features of the object.
5. If there are still too many hidden lines and the views are still confusing, add more of the standard orthogonal views as necessary to reduce the number of hidden lines and to maintain full definition of the object's geometry.
6. Use shorthand notation to define screw threads and the size and depth of counterbored and countersunk holes and slots. Otherwise, add section views to clarify the interior details.
7. If the hidden lines from different interior details cross or overlap, add section views to clarify the interior details.
8. If any edges cannot be seen in their true lengths in the standard orthogonal views, add auxiliary views so these true lengths can be seen.

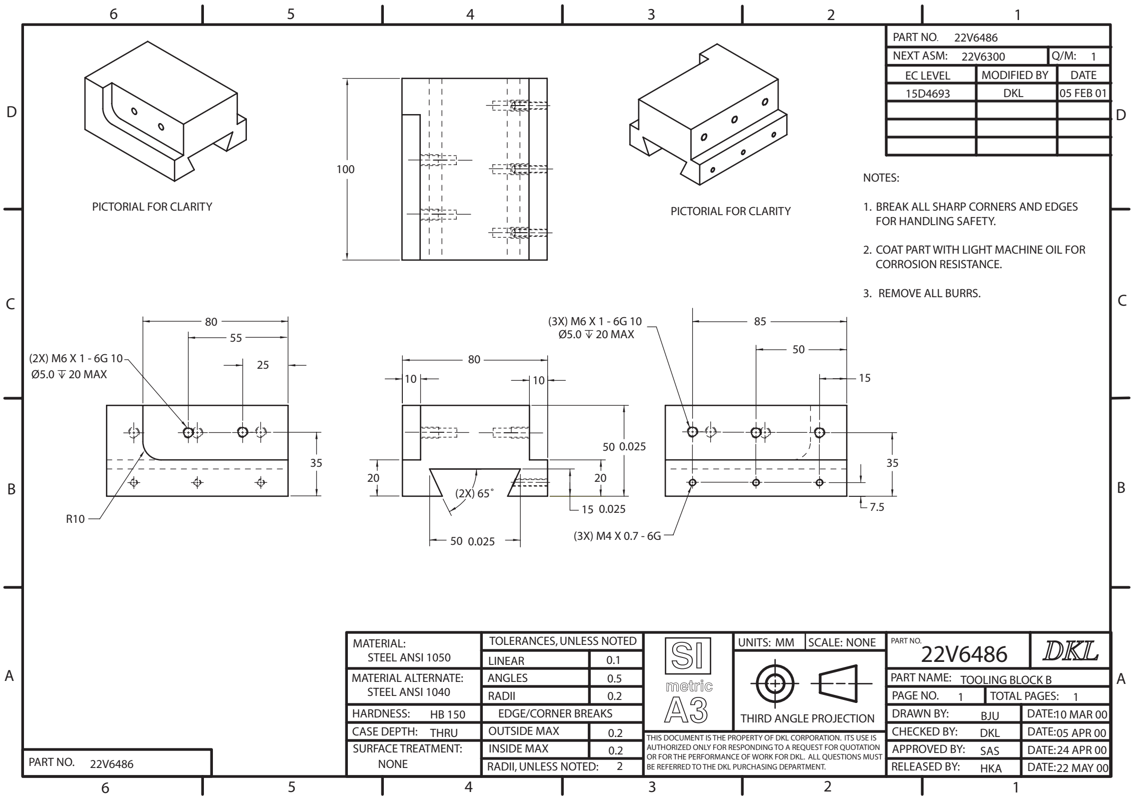

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FIGURE 14.17. This modified tooling block requires an additional left-side view in order to show the features on that side.

9. If features cannot be seen clearly and cannot be defined because they are small, add detail views with a magnified scale.
10. Add the appropriate dimensions and tolerance specifications for the object. If the drawing starts to look crowded, transfer everything to a larger-sized sheet.

For people with a great deal of drawing experience, most of those steps can be done mentally. Then the general rule for presenting the object becomes simply, "Start with the preferred configuration; then add or subtract whatever views are necessary to best show all of the geometry."

14.05.03 Notes

The notes on a formal drawing refer to special processing, handling, or assembly procedures that are required on a part or an assembly that cannot be specified by the dimensions and tolerances on the part or by the materials specifications in the title block. The notes are usually numbered according to each specific requirement and listed together in the same area of the drawing. Notes on a drawing might include "This surface to be free of plating" or "Part to be cleaned and degreased when completed" or "All internal edges to be free of burrs." Generally speaking, a note is added whenever you want something done to a part and you do not know how else to specify it.

14.06 Parts, Subassemblies, and Assemblies

An engineered device is composed of one or more pieces, and one of the purposes of manufacturing drawings is to document how those pieces should be made and how the pieces should be put together to make the device. A set of drawings must contain enough information so that any manufacturer with the proper fabrication tools and skills can make all of the pieces and put them together properly. This fabrication must be possible from the information available on the drawings, without consultation with the engineers, designers, or drafters who made the drawings. A set of manufacturing drawings is structured like the roots of a tree, as shown in Figure 14.18.

At the very top of the root structure is the **main assembly** for the device. Assembly drawings are required to show how to put the main assembly together and how all of the pieces look and fit when this is done. The main assembly is composed of smaller individual parts that can be made especially for the device or purchased as commercially available parts. Examples of common commercial parts include screws, bolts, washers, nuts, and rivets.

Assembly drawings contain information that identifies the parts or subassemblies in the assembly. This is done with **balloons** and arrows pointing to each part. Balloons are closed geometric shapes, not necessarily circular, that contain a number. Depending on individual company practices, the number inside the balloon can be the part number or an **item number** that is referenced to the part number and listed with the drawing's notes. Some companies like to include the quantity of each part inside the balloon along with the item or part number.

The main assembly also can be composed of smaller **subassemblies**, which are collections of custom-made and/or commercially available parts that have already been put together and installed in the main assembly as a single piece. A large project may have several levels of subassemblies in the main assembly. Each subassembly needs its own layout drawing and assembly drawing for the various sub-subassemblies, or parts, that go into it. Finally, each custom-made part requires a **detail drawing** that shows the geometry, dimensions, tolerances, materials, and all other information needed to fabricate the part.

At this time, a more precise definition of a part is needed. The most common interpretation of a **part** is that it is a single object made from a single, contiguous material. Most metal, plastic, or wooden objects, for example, fall under this definition.

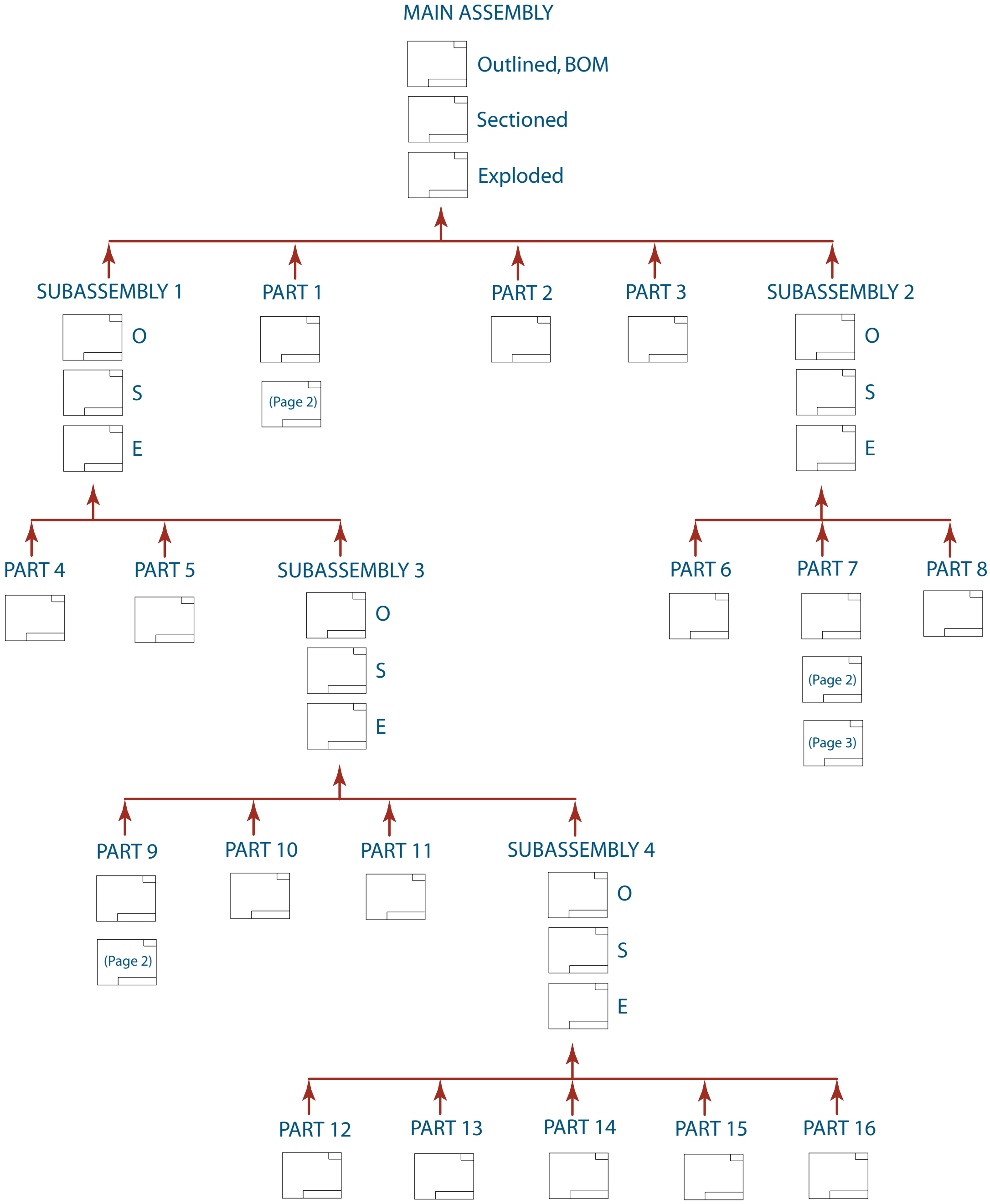

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FIGURE 14.18. A set of drawings for a project is organized like the roots of a tree. Individual parts fit into subassemblies or the main assembly. Subassemblies fit into higher subassemblies or the main assembly.

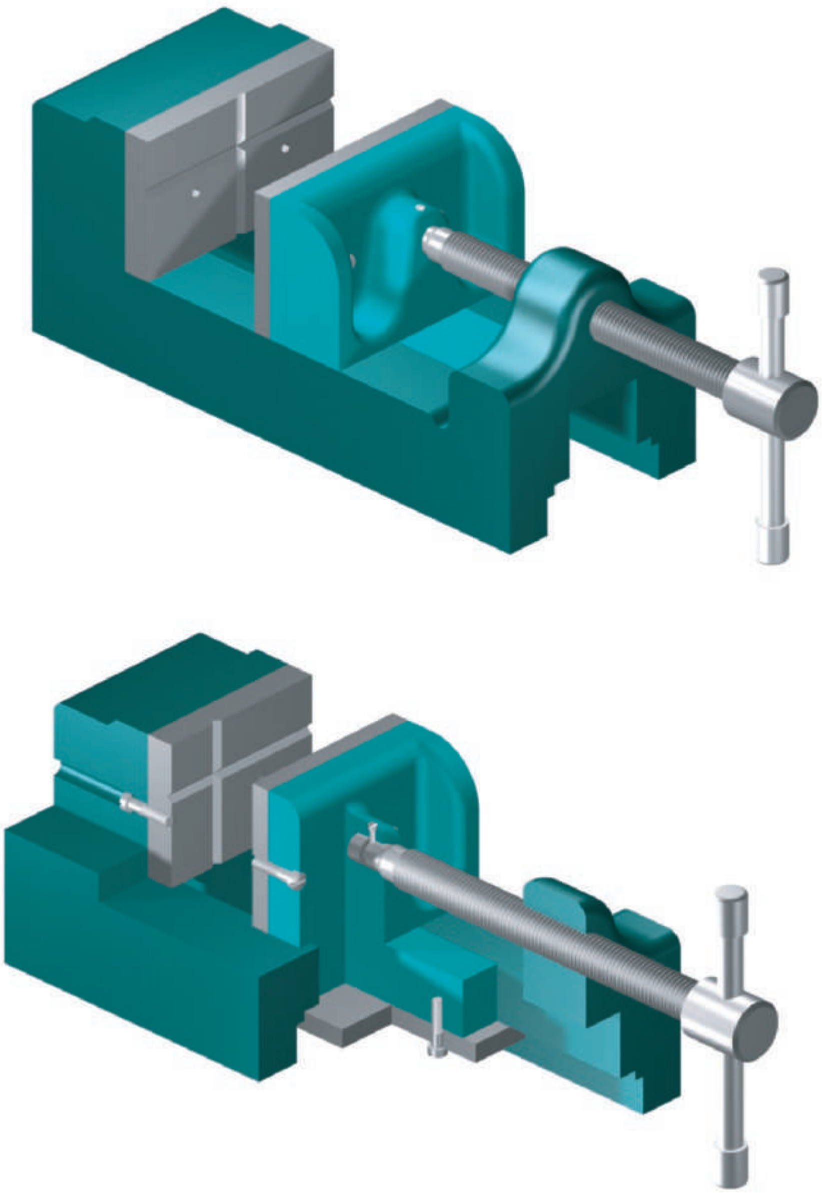
Up until the twentieth century, this definition was correct. Since that time, products have become increasingly complex in their construction and use of different materials. Is an electric motor a single part; or is it a collection of parts, otherwise known as an assembly? What about the headlight module on a car? Imagine an electric circuit board with various electronic components installed on it. If you were the user of the board, you would probably consider it to be a single part. However, if you were its manufacturer, you would probably consider it to be an assembly. Therefore, the definition of a part depends on how you expect to receive it from the manufacturer making it for you. You can generally refer to an object as a single part, even though it may be composed of many different pieces and be made of many different materials, when you expect the fabricator to deliver it to you as a single unit with only its external dimensions and functional requirements specified.

Consider the drawings that would be necessary for the product shown in Figure 14.19, which shows a vise clamp used for holding work pieces during machining operations. All of the drawings necessary to fabricate this device will be discussed, with more detail on the type and method of presentation of information for each type of drawing.

14.06.01 Exploded Assembly Drawings

Exploded assembly drawings show how various parts and pieces that compose an assembly or subassembly are put together. Rather than showing everything in their final position, as with a layout drawing, an assembly drawing shows the parts of a

FIGURE 14.19. A 3-D computer-generated model of a machine vise.



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device in a disassembled state. The assembly drawing for the vise clamp is shown in Figure 14.20. The various parts are shown in their final resting orientation, but not necessarily in their final location. Instead, the parts are located such that they are removed from their final location in the opposite direction of manufacturing insertion. The path of insertion for each part is then shown using a dashed path called a **trail**. Therefore, the trail of each part shows the fabricator how that part is to be placed in the device to create the final configuration.

Since assembly drawings are used to show a process rather than precise geometry, using a pictorial presentation is preferred to using a multiview presentation. Neither part nor assembly dimensions are shown, except for occasional reference dimensions for convenience only. As with a layout drawing, assembly drawings use numbered balloons and arrows pointing to each piece to identify their parts or subassemblies. The item number in each balloon must correspond to the same item number in the layout drawing and is then listed with its corresponding part number and part name in the assembly drawing's notes.

14.06.02 Outline Assembly Drawings

Outline assembly drawings, sometimes called **layout drawings**, are used to show the fit and function of all of the various pieces that go into a completed assembly or subassembly. The main outline assembly drawing for the vise clamp is shown in Figure 14.21. An outline assembly drawing shows the final product in its final configuration using the multiview format required for all working drawings.

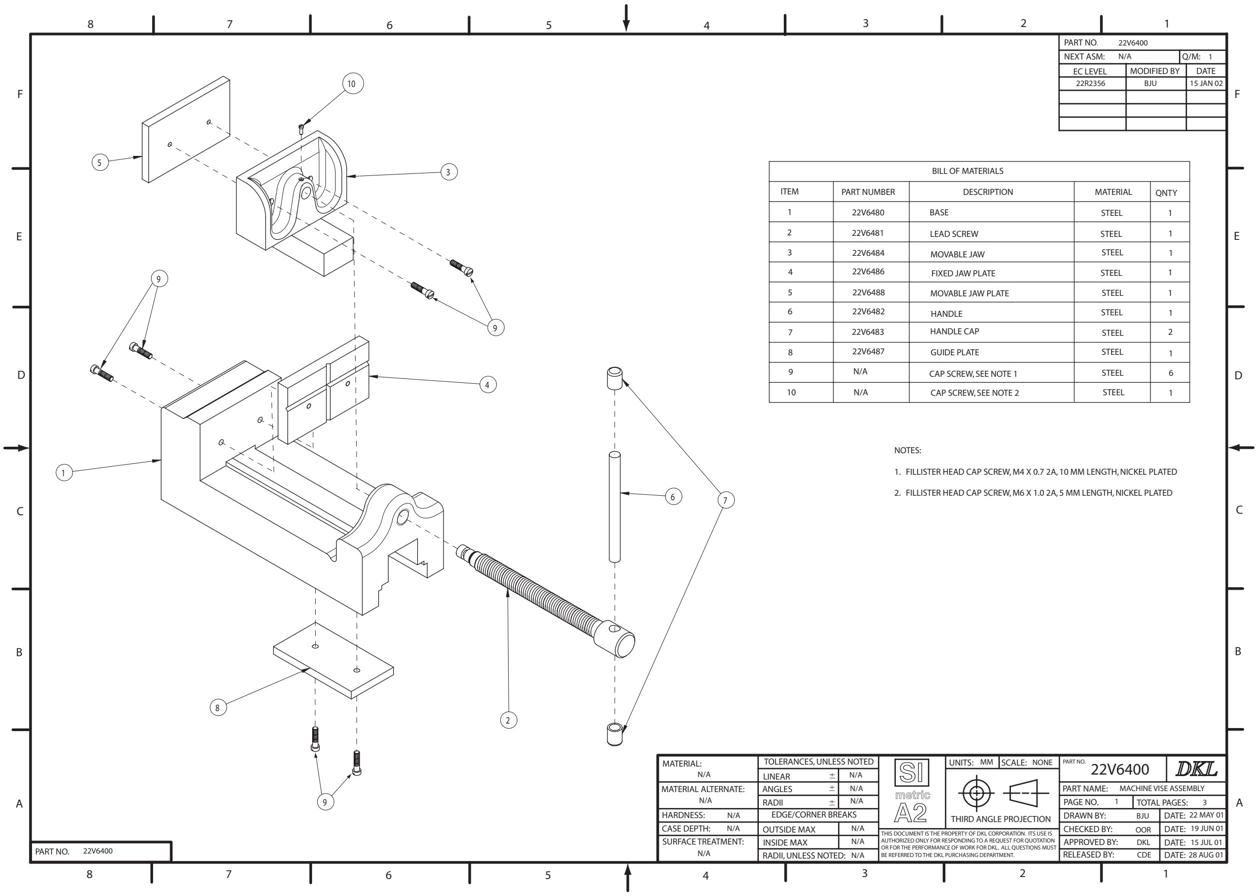

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FIGURE 14.20. An exploded assembly drawing including a bill of materials for a machine vise.

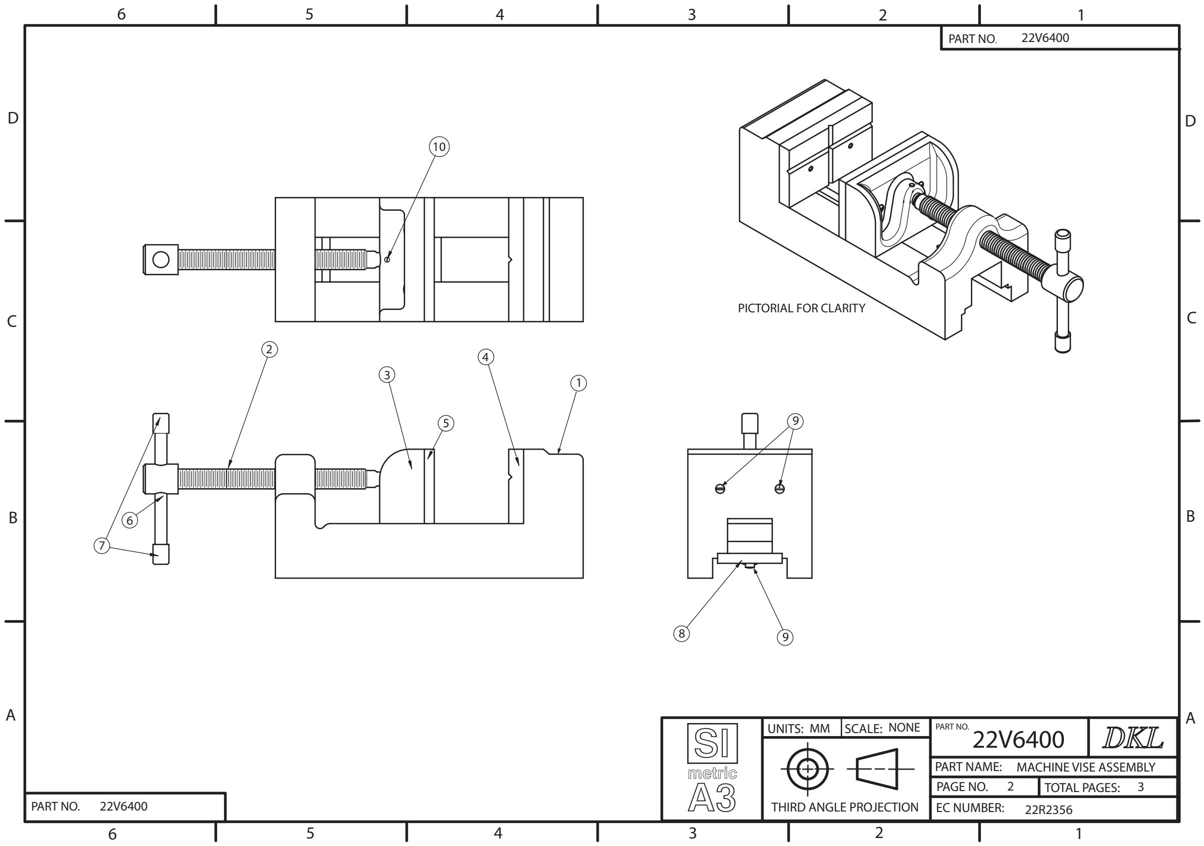

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FIGURE 14.21. An outline assembly drawing of the machine vise.

Sometimes isometric or other pictorial views are included for additional clarity. Section views are added to reveal parts that cannot be seen externally, and magnified detail views are used to show parts that are too small to recognize. Note that the dimensions for the individual parts are not shown in any assembly drawing. That information is contained in the detail drawings. Information pertaining to tolerances and materials on the drawing header are left blank since that information is contained in the detail drawing for the individual part and does not need to be repeated for the assembled device.

If any **assembly dimensions** are required, they must be shown with their tolerances on the layout drawing. Assembly dimensions show where parts must be placed relative to other parts when the device is being put together (e.g., when a special alignment between parts is required and no features on the individual parts provide for this alignment). An example of when assembly dimensions are required is when a smaller block is to be welded or bonded to a larger plate. Unless there are features on the plate for locating the block, assembly dimensions must be supplied. If the parts are to be welded together, specification for the welds must be placed on the layout drawing.

Reference dimensions, if used at all, should be used sparingly and must be clearly identified and placed inside parentheses. These dimensions already exist on or can be extracted from other drawings. They are shown mostly for the convenience of the reader and are usually used to show gross sizes, such as the overall width, height, and length of a device. The reason reference dimensions are used sparingly is because errors sometimes occur when the dimension on a part changes and the change is forgotten on the reference dimension.

The notes on a layout drawing are used to specify any special procedures or processes needed to put the device together and any tests that are necessary to ensure that the device will work in its intended manner. The notes also list the definitions of the item numbers referenced in the multiview presentation.

14.06.03 Sectioned Assembly Drawings

A **sectioned assembly drawing** is a pictorial or orthogonal view(s) that shows all of the various pieces of an assembly or a subassembly in their final resting position. For purposes of revealing otherwise hidden parts, some parts have been cut away. The cut surfaces are indicated by the use of section lines. A section line pattern is usually used for each part to aid in its distinction. The various pieces within the assembly are identified using numbered balloons with the same item numbers used in the other assembly drawings. A sectioned assembly drawing for the vise clamp is shown in Figure 14.22. Although sectioned assembly drawings are difficult to create, especially without the use of 3-D modeling, such drawings offer unparalleled clarity for showing how various parts fit together to make a complete device. Whenever possible, a sectioned assembly drawing should be included in a drawing set.

14.06.04 The Bill of Materials

The **bill of materials (BOM)** for a device is not an actual drawing, but rather a text list of its parts, subassemblies, and subassembly parts. The BOM, which can appear on any one of the assembly drawings or as a separate drawing or document, is used

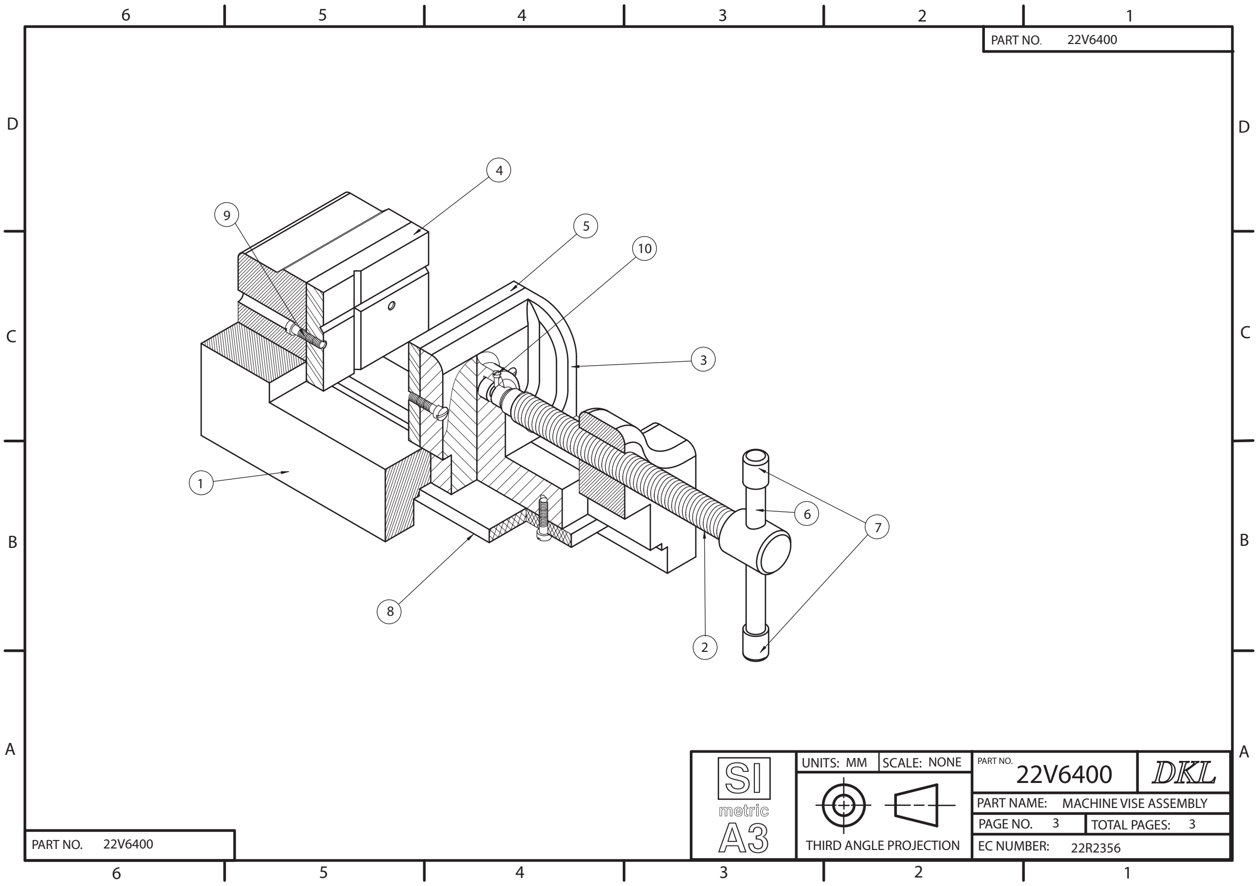

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FIGURE 14.22. A sectioned assembly drawing of the machine vise.

mainly by a fabricator to check that all of the drawings and materials needed to make the device are available. Commercial parts are also included on this list. Although not a drawing, a BOM is usually printed as a table on an assembly drawing to emphasize that this list is considered a member of the set of drawings for the device. The BOM is often included as part of the layout or assembly drawing.

The typical information included on a BOM of an assembly includes the item number (if the bill is included as part of the layout or assembly drawing), the corresponding part number, its part name, the material from which the part is made, and the number of times the part is used in the assembly. Subassemblies have their own BOMs. When the main assembly contains subassemblies, the part numbers and names for those subassemblies are listed on the main BOM. The various parts and sub-subassemblies are listed on the BOM for each respective subassembly. If commercial parts are used, their descriptions must be included on the BOM. These descriptions must contain enough information for the parts to be acquired without subsequent explanation.

14.06.05 Manufacturing Detail Drawings

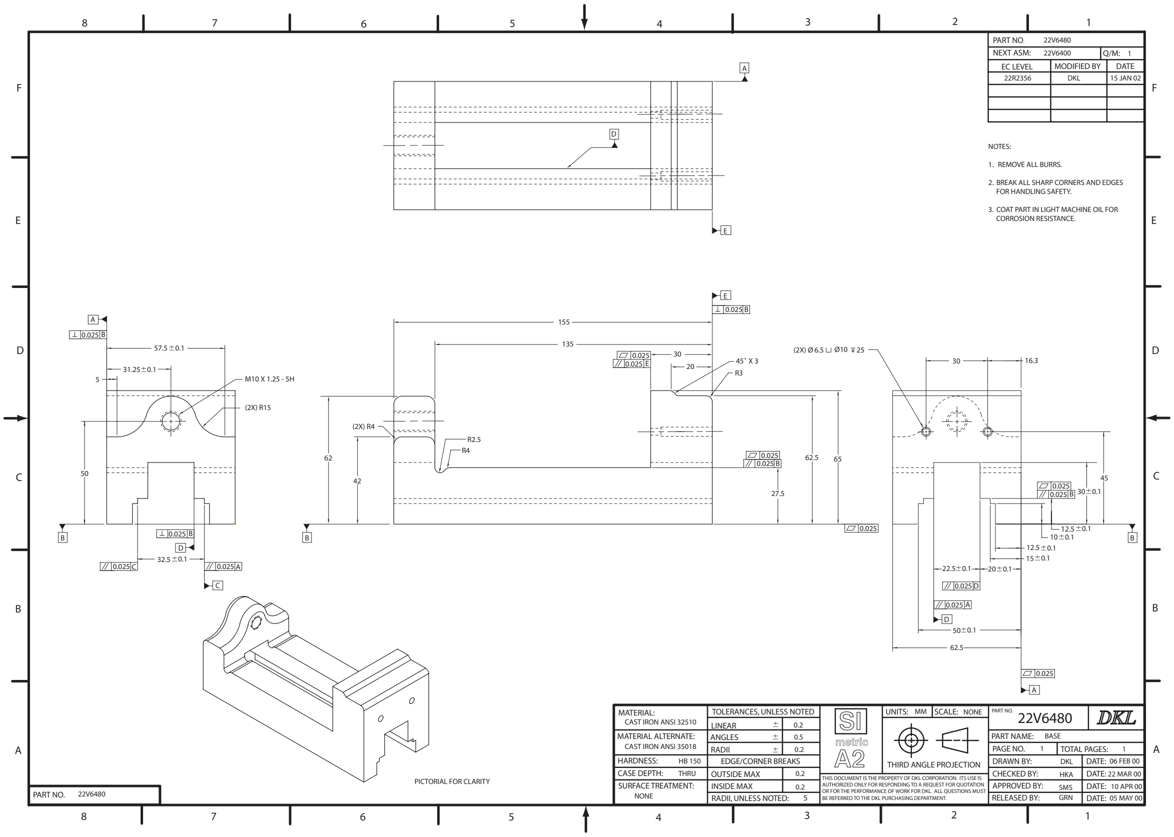
A detail drawing shows all of the geometry, dimensions, tolerances, material, and processes needed to fabricate a single part. Each custom-made part must have its own detail drawing. Some companies even require that commercially available parts have their own detail drawings placed on that company's header to ensure that these parts will fit and function properly with the custom-made parts. The detail drawings for the parts of the vise clamp are shown in Figure 14.23. The drawings shown in Figures 14.16 and 14.17 are also classified as detail drawings.

The detail drawing for each part shows it using the multiview format required for all manufacturing drawings. Sometimes isometric or other pictorial views are included for additional clarity. Section views are added as necessary to reveal interior features. Magnified detail views are added as necessary to show features that would otherwise be too small to dimension. All of the dimensions for each part must be shown. Information pertaining to tolerances and materials on the drawing header is also included.

The notes on a detail drawing are used to specify any special procedures or processes needed to fabricate or finish the part that are not evident from the dimensions and tolerances and any tests that are necessary to ensure that the part works in its intended manner. Putting a closed geometric shape, usually a box, around the note number highlights a note that refers to a particular feature on the part. A leader arrow then points to that feature and is annotated "See note X," which tells the reader of the drawing that a special instruction in the notes is associated with that feature.

14.06.06 More Examples of Manufacturing Drawings

The 3-D model of a small, inexpensive flashlight is shown in a complete state and in a sectioned state in Figure 14.24. The working drawings for this product are shown in Figure 14.25. The assembly drawings show how the individual pieces fit together, and the detail drawings show the required sizes of the custom parts. The complete assembly drawing is three pages long and includes exploded, outline, and sectioned views as well as a BOM. Many of the pieces (more than 100,000 per month) for this intended low-cost, high-volume product will be molded from plastic. The product has been designed such that stringent tolerances on the dimensions are not required for the parts to fit together and function. Two of the parts, the battery and the light bulb, are commercially available; thus, detail drawings are not needed for them. The assembly drawings, however, show all of the parts, including the battery and bulb. The BOM, located on the first page of the assembly drawing, specifies enough information about these two parts so that they may be purchased.



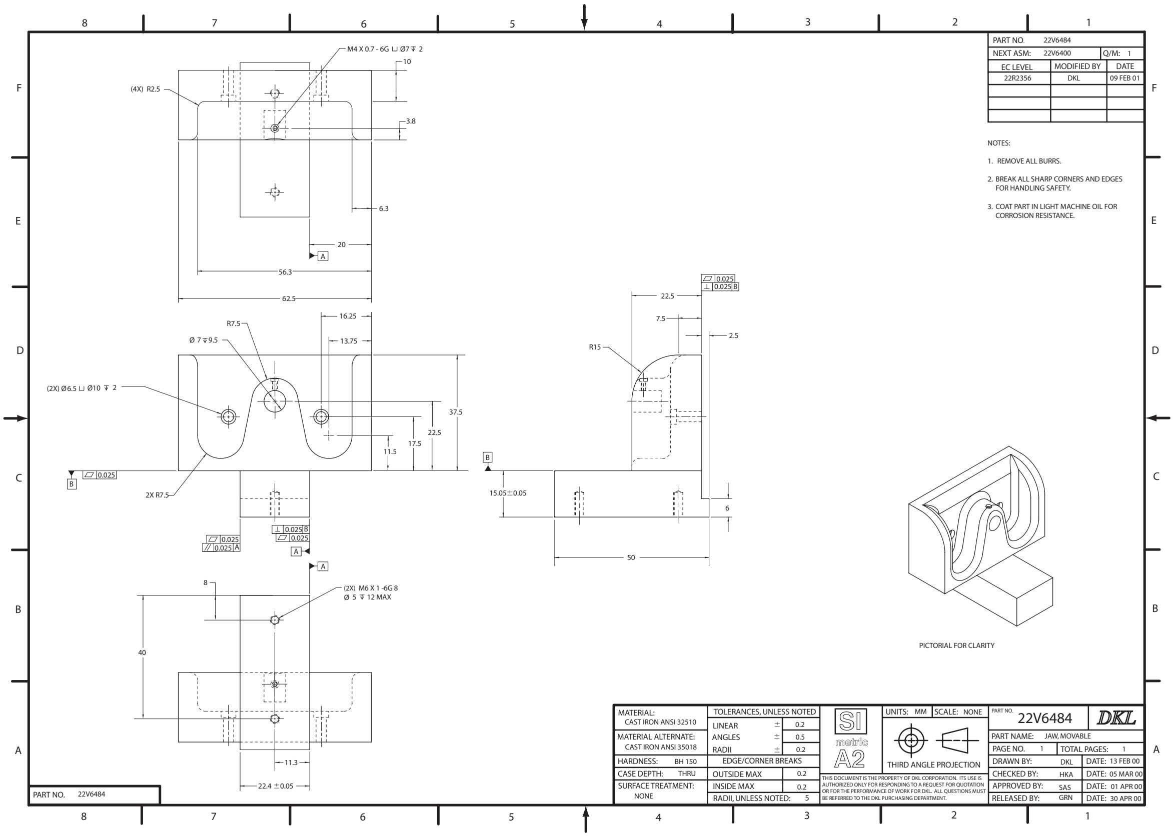


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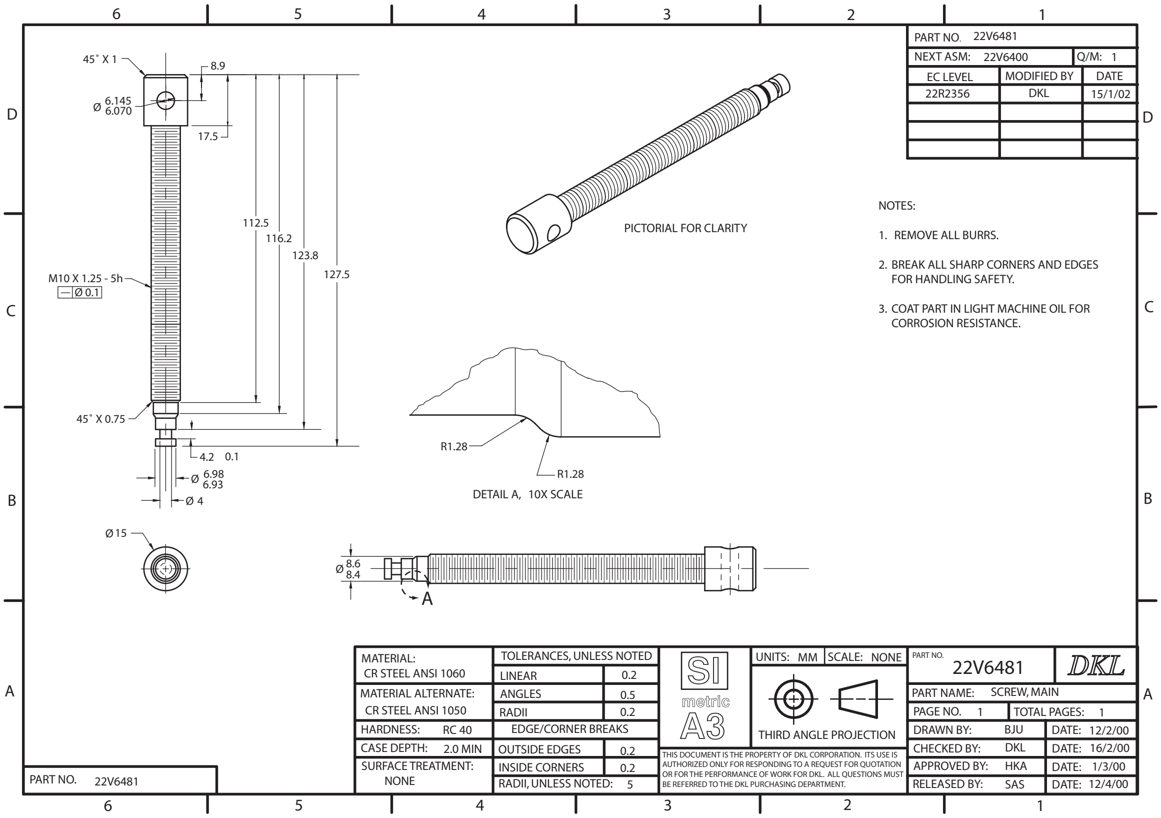
FIGURE 14.23. Detail drawings for the machine vise.



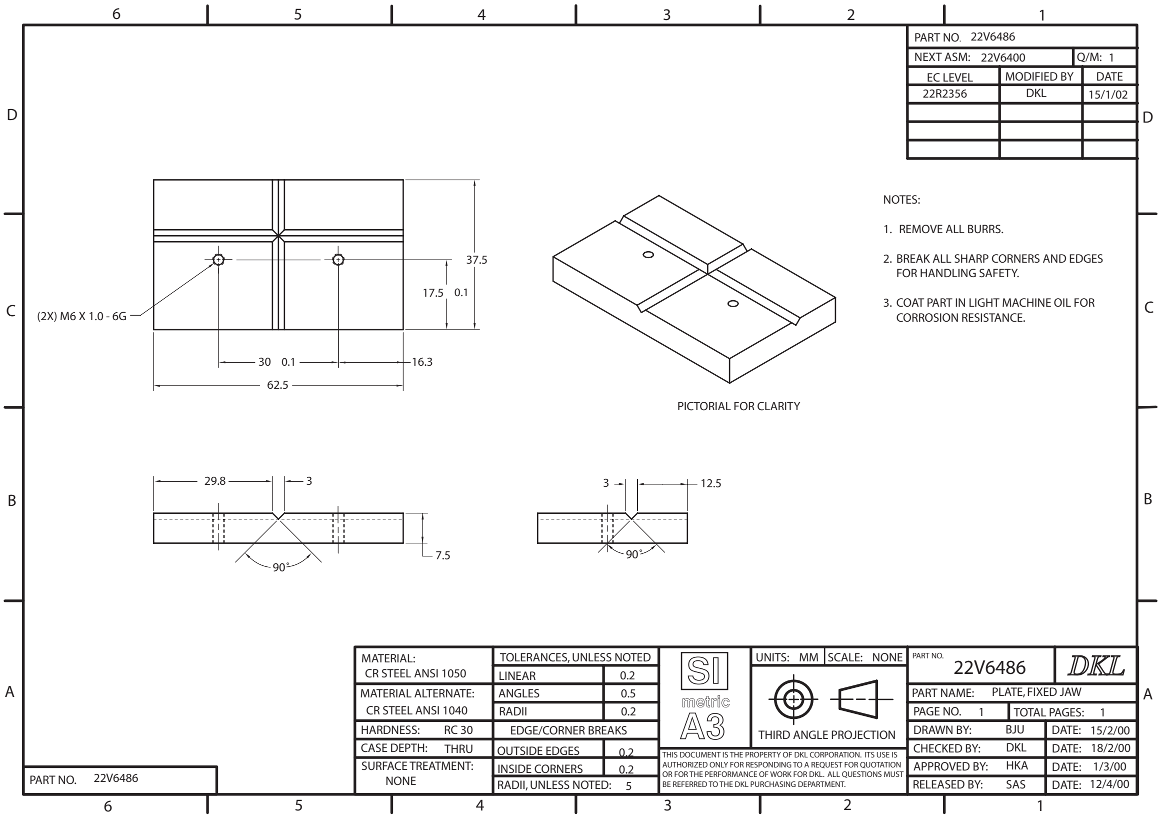


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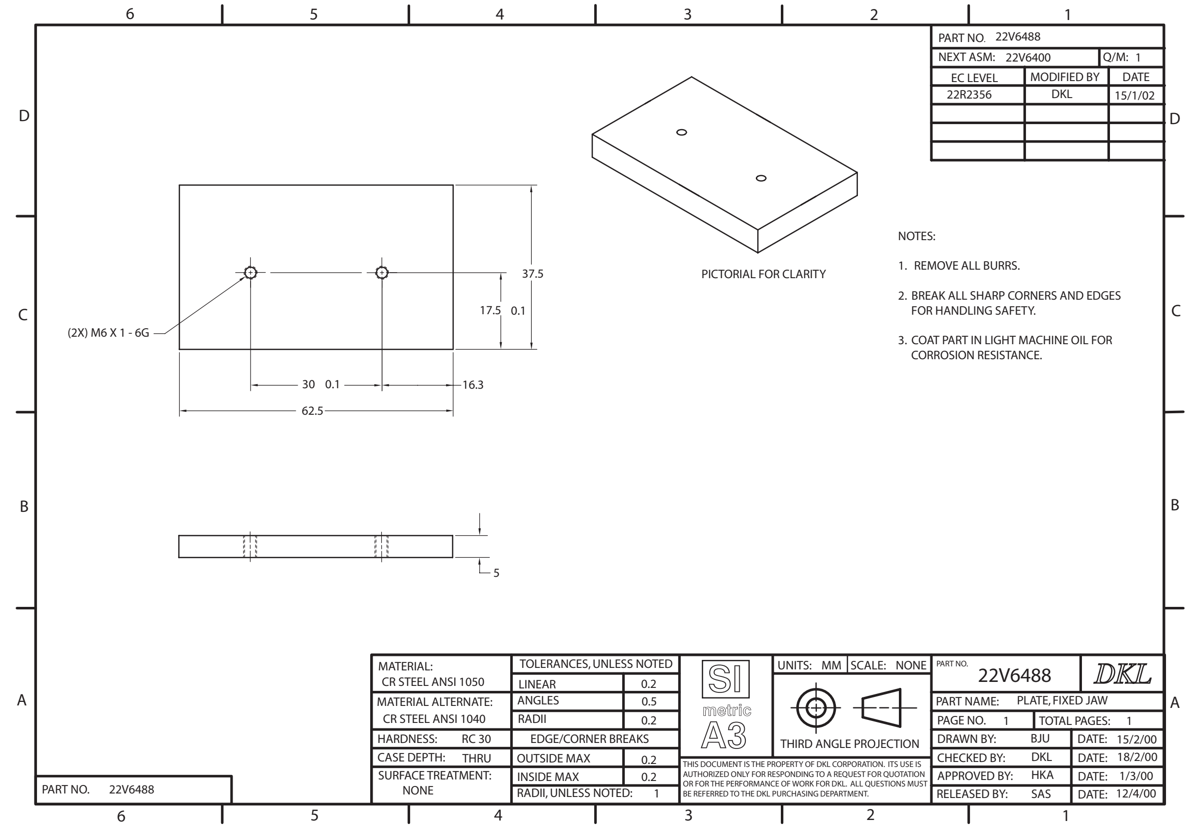
FIGURE 14.23. (CONTINUED) Detail drawings for the machine vise.



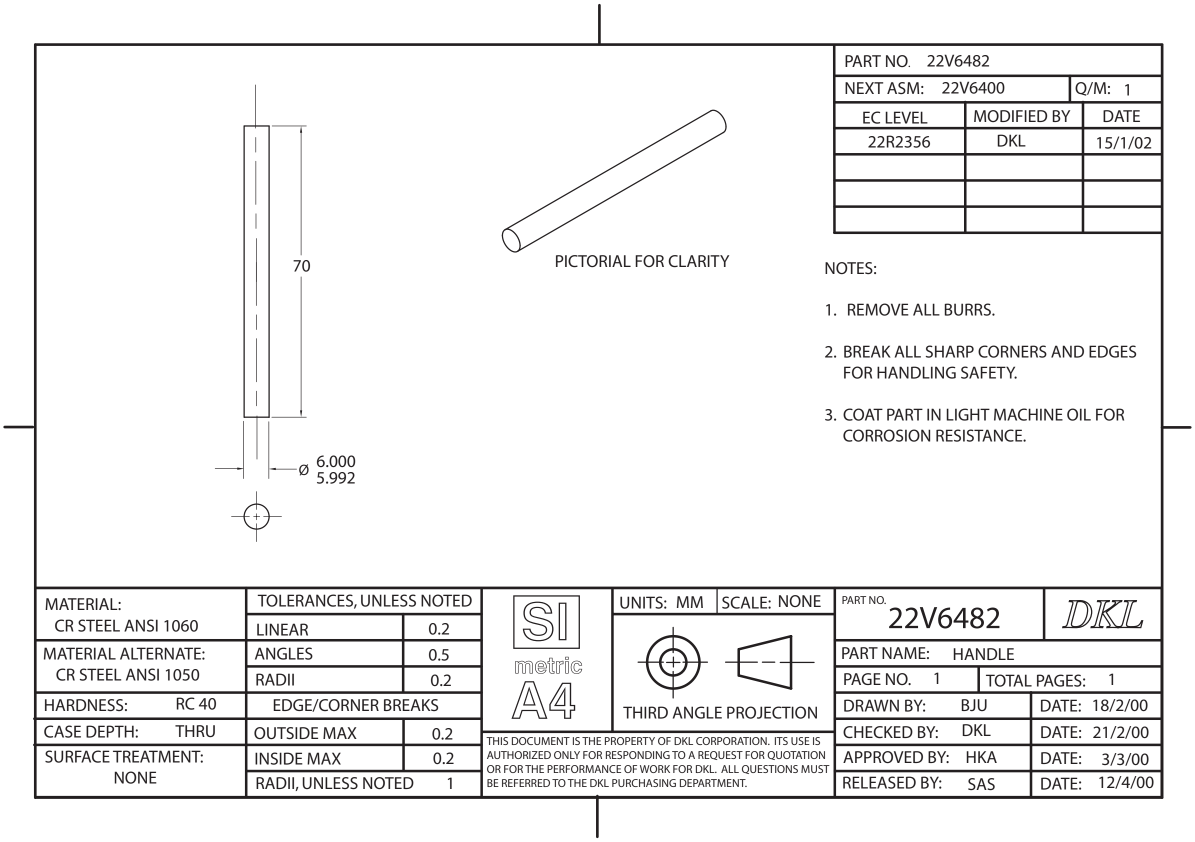


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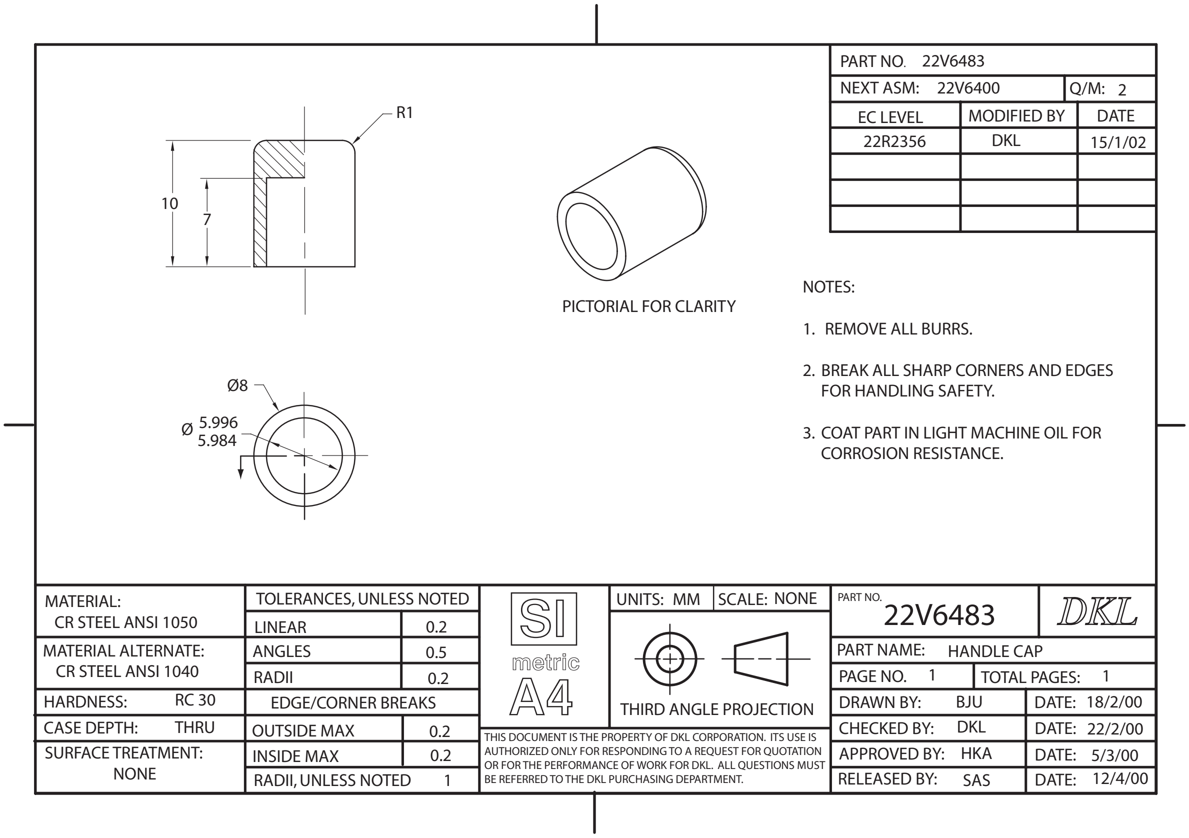
FIGURE 14.23. (CONTINUED) Detail drawings for the machine vise.



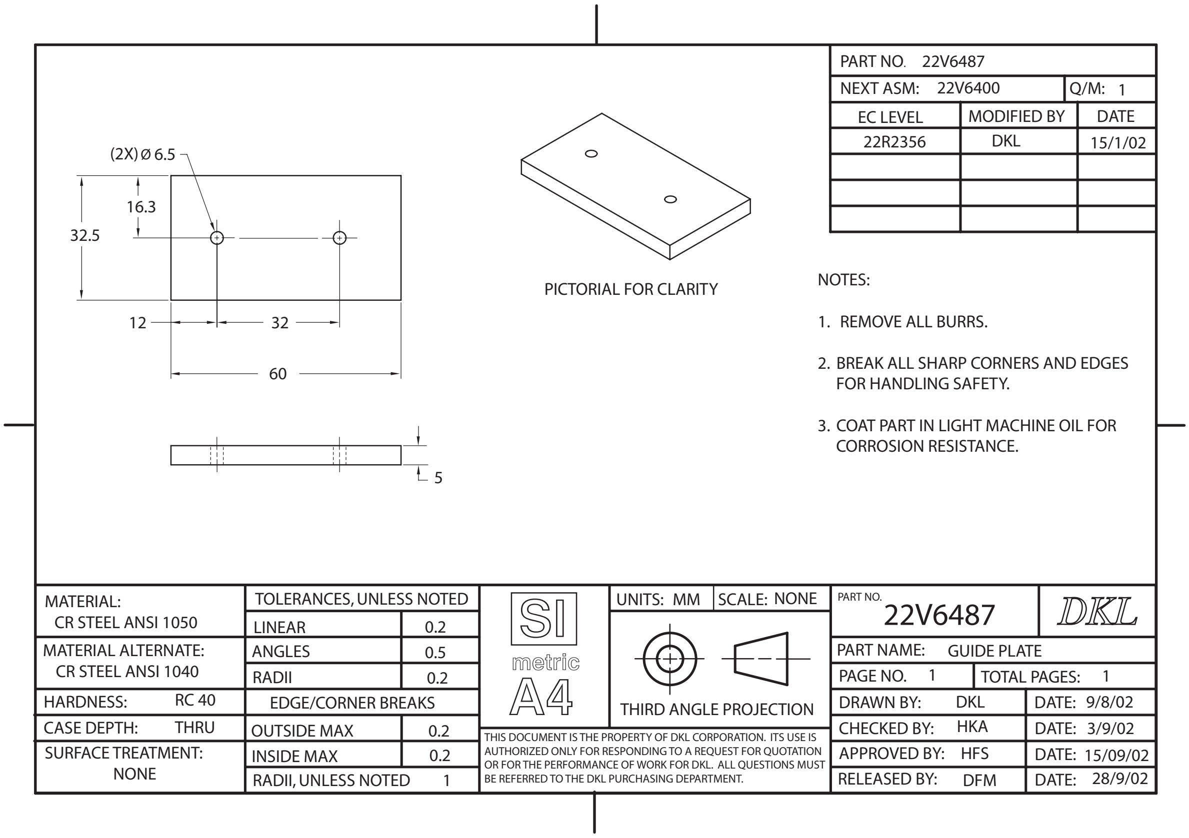


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FIGURE 14.23. (CONTINUED) Detail drawings for the machine vise.

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FIGURE 14.24. Three-dimensional computer-generated models of a disposable one-cell flashlight.

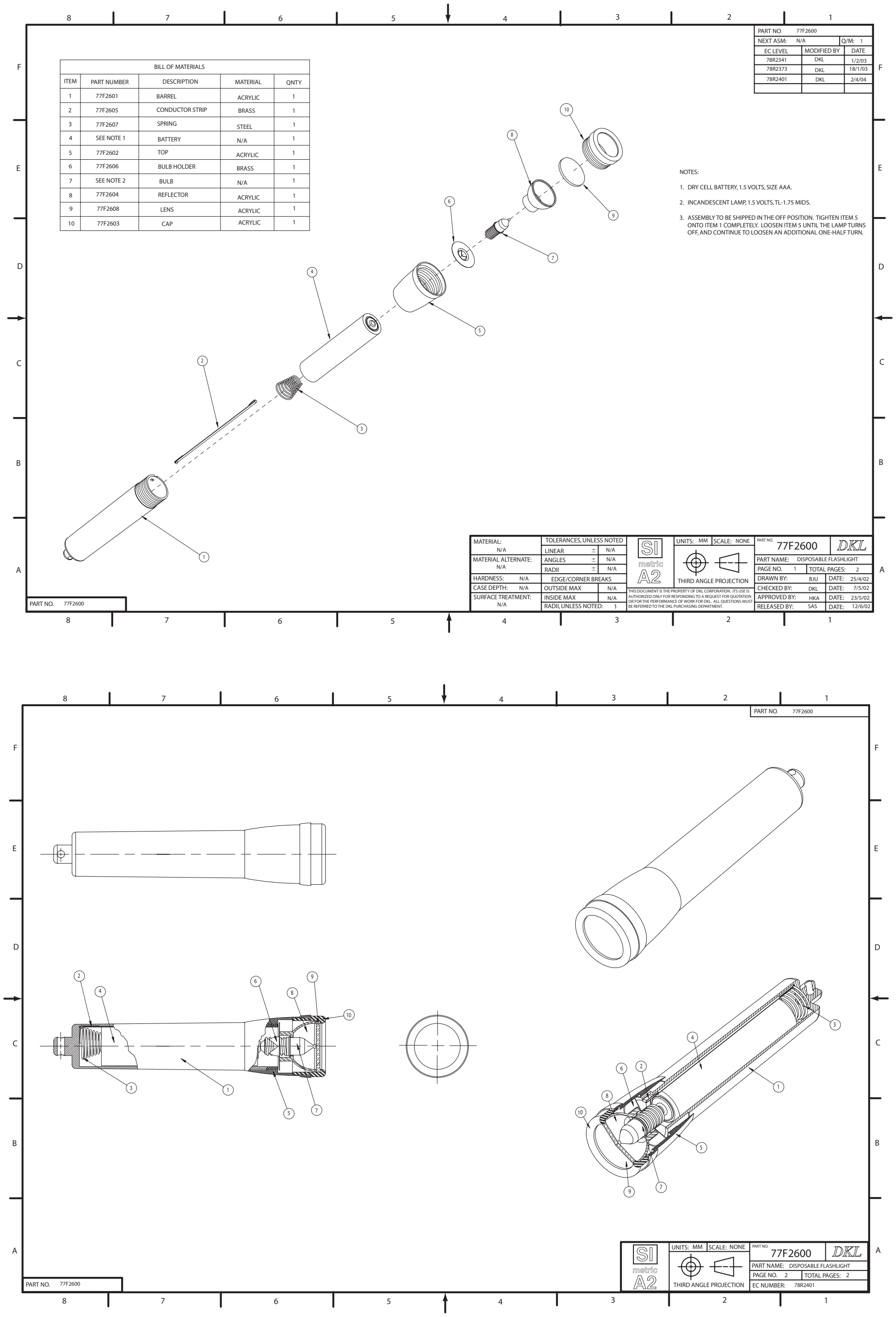

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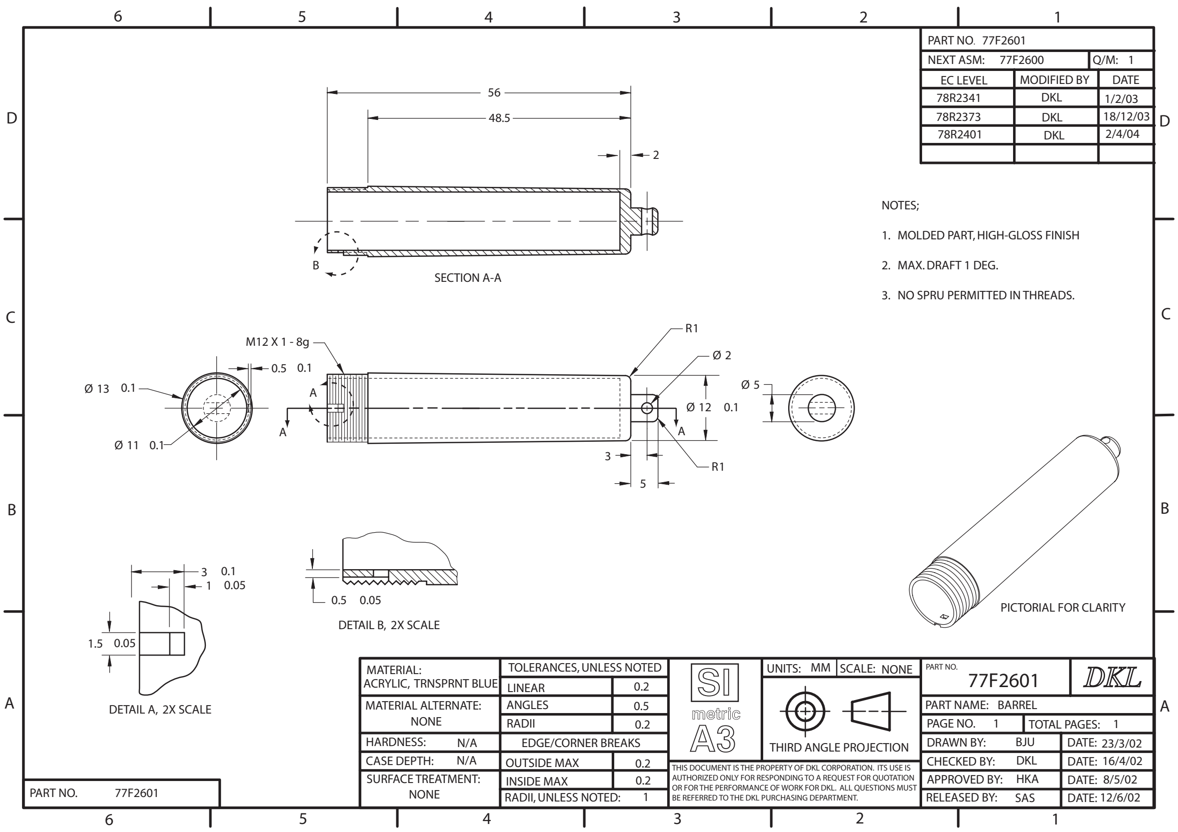
FIGURE 14.25. Working drawings for the disposable flashlight.



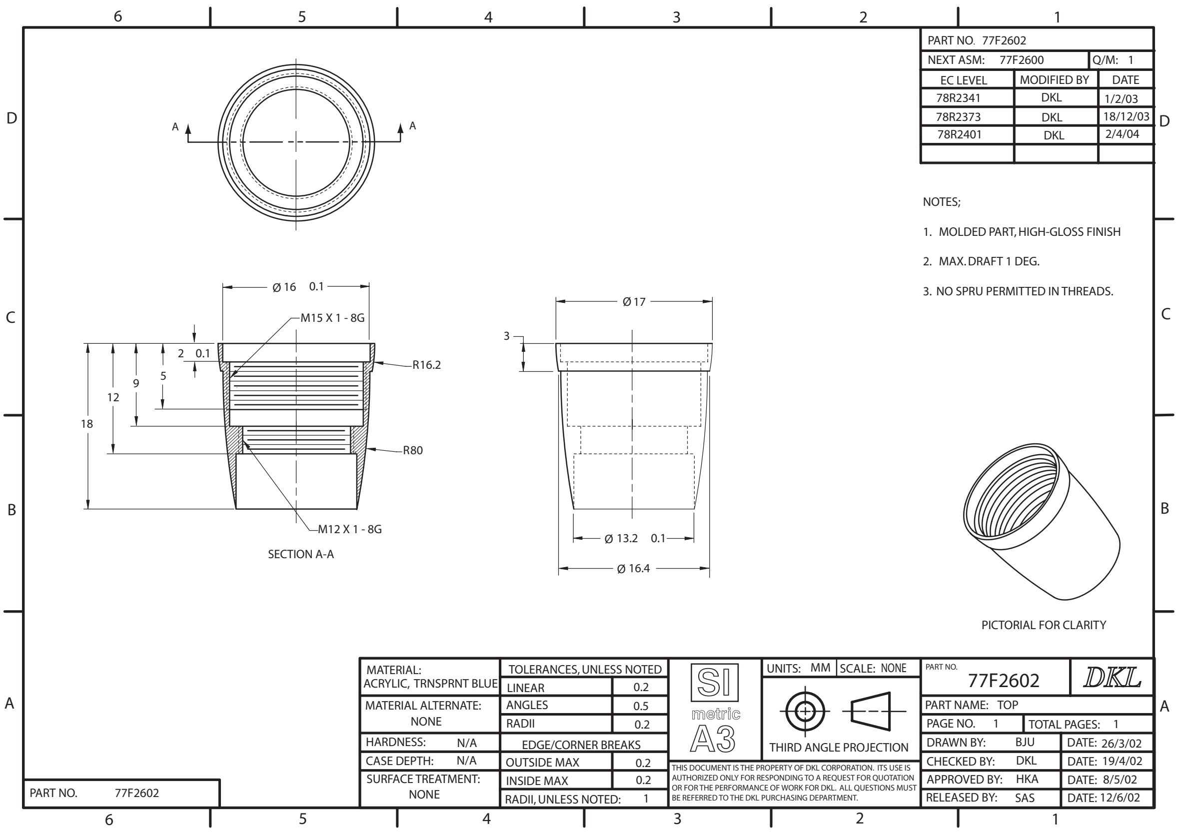


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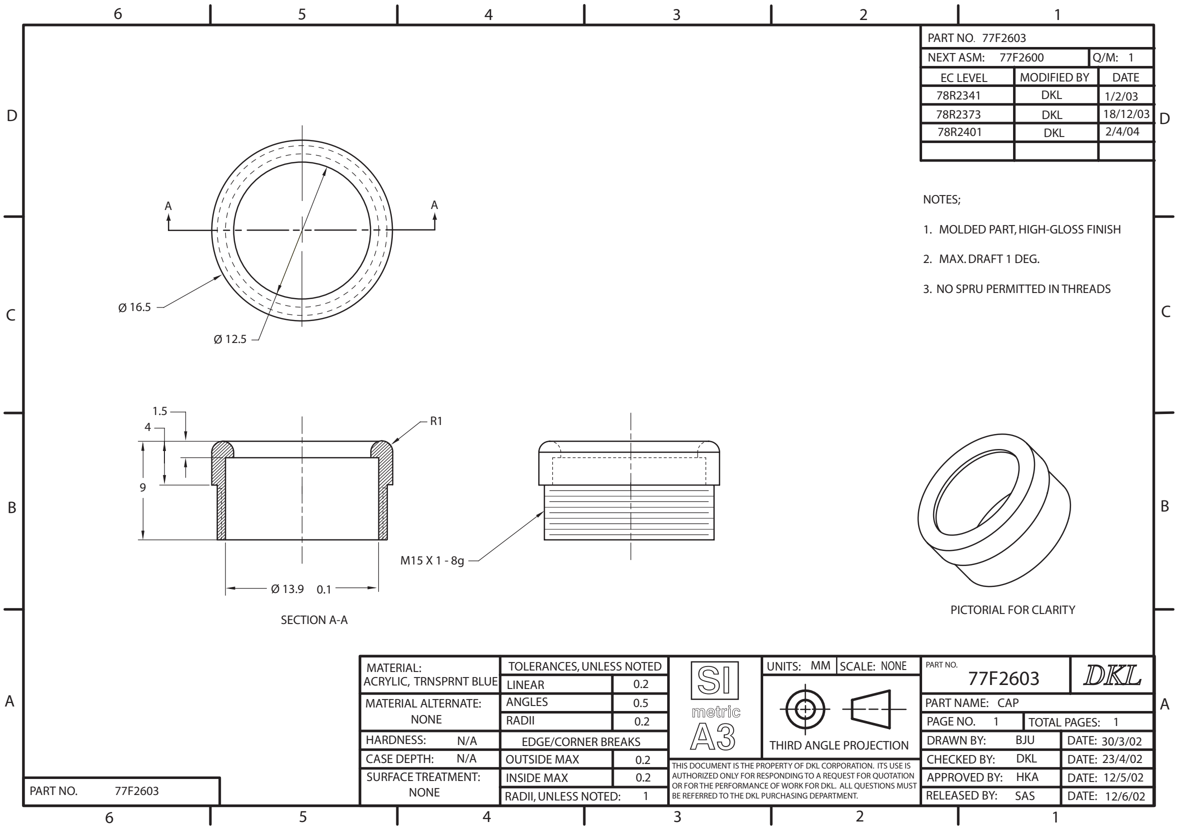
FIGURE 14.25. (CONTINUED) Working drawings for the disposable flashlight.



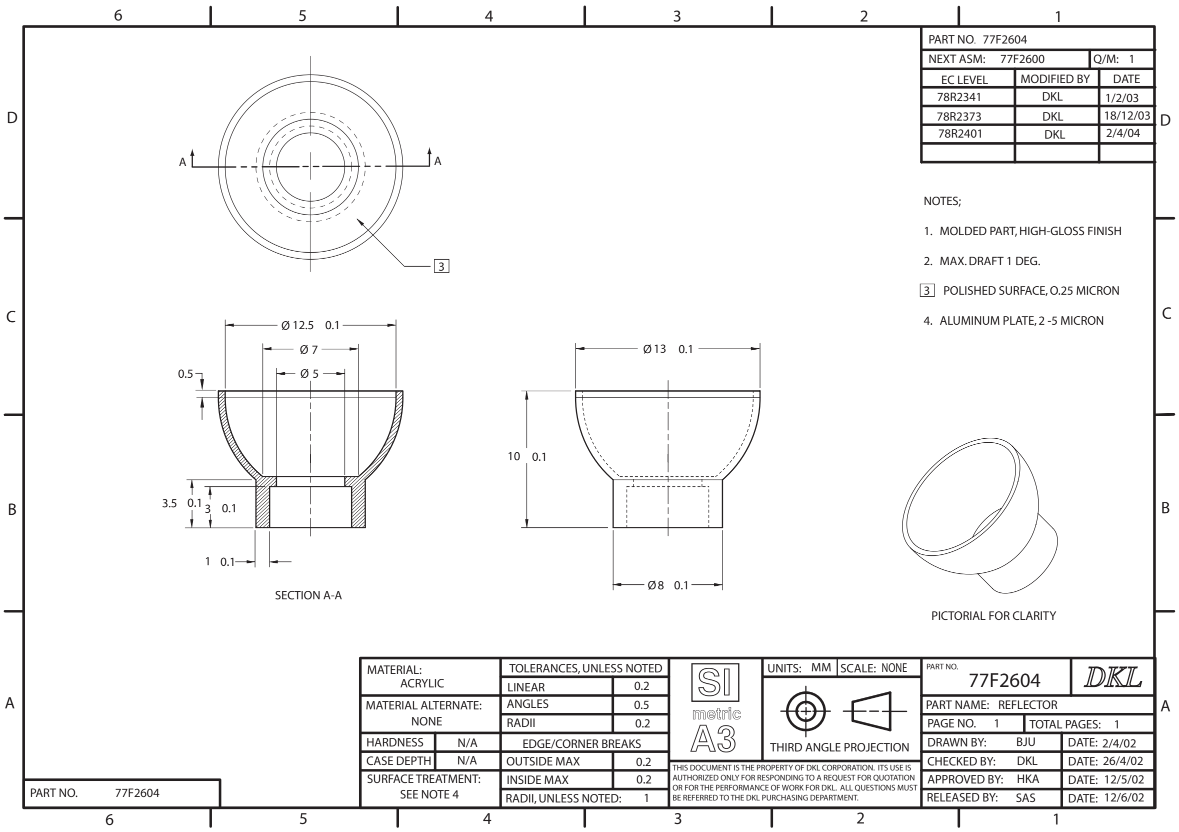
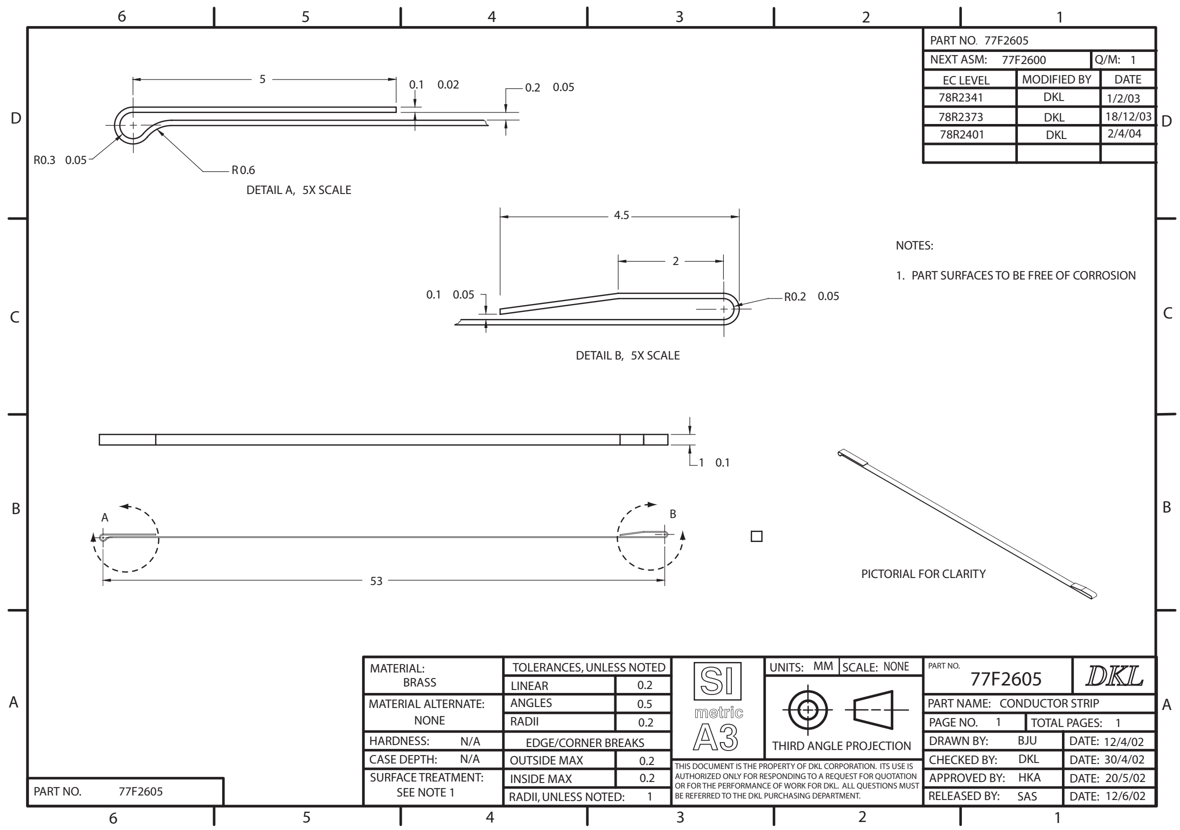


FIGURE 14.25. (CONTINUED) Working drawings for the disposable flashlight.



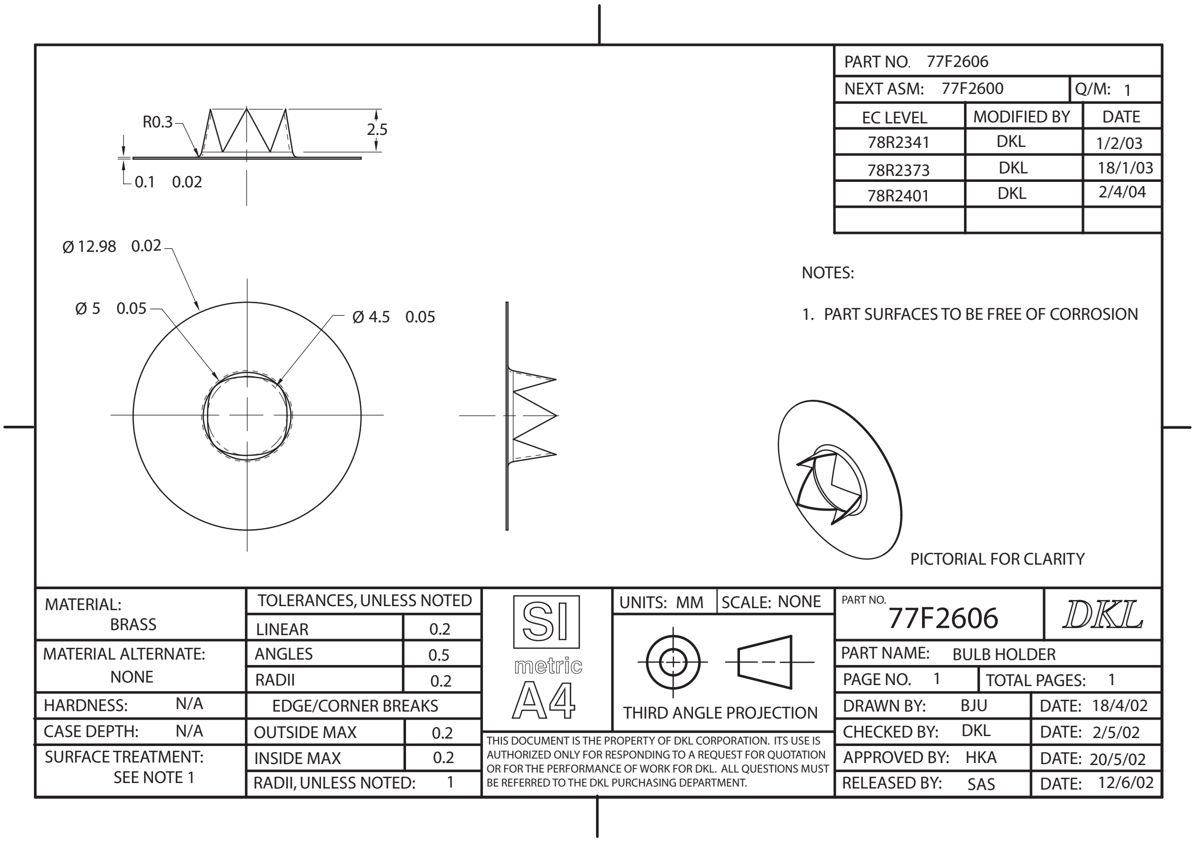


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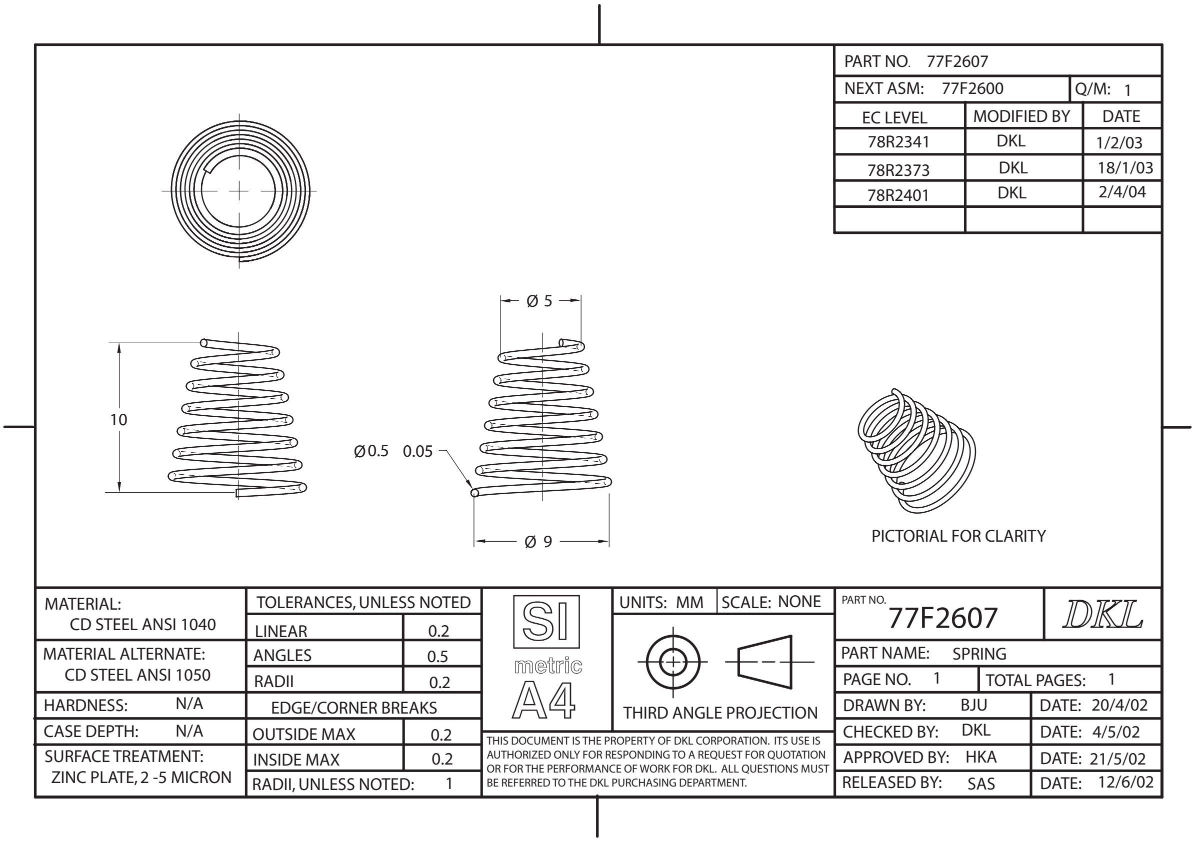
FIGURE 14.25. (CONTINUED) Working drawings for the disposable flashlight.



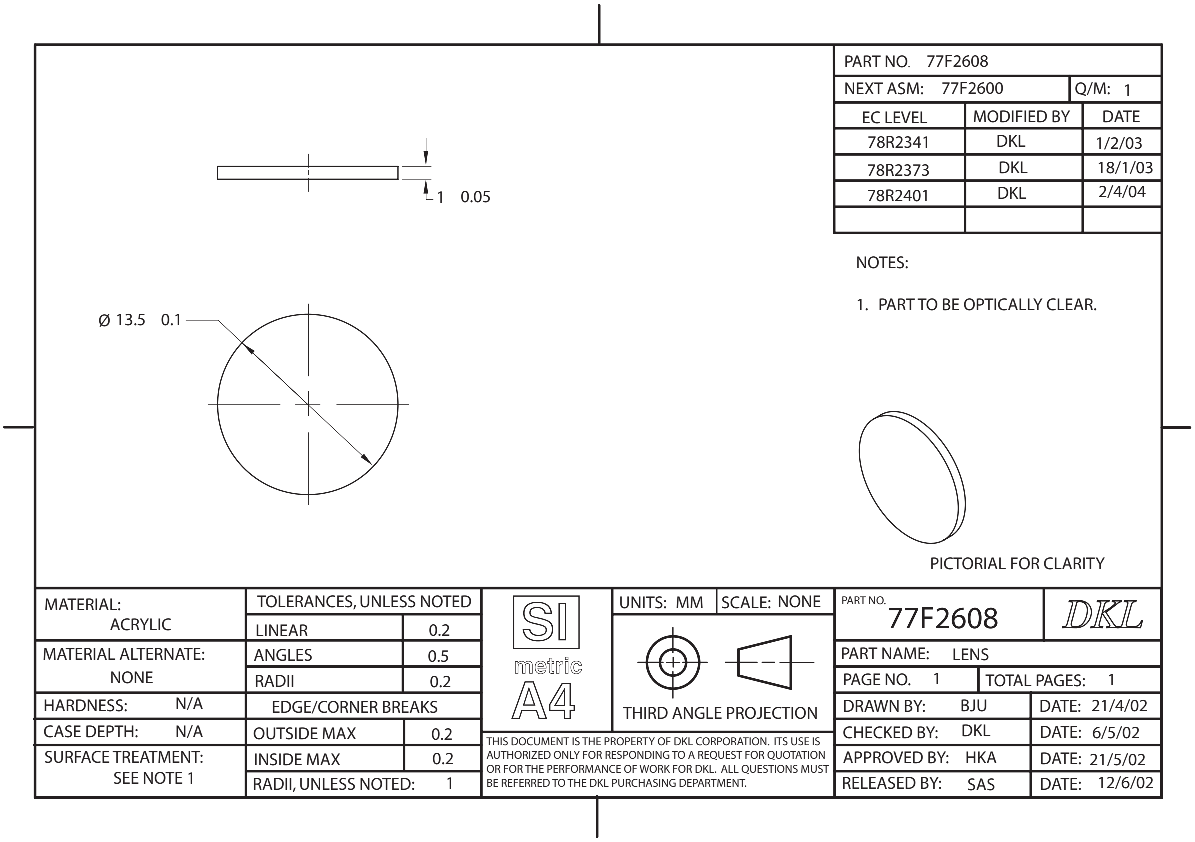


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FIGURE 14.25. (CONTINUED) Working drawings for the disposable flashlight.

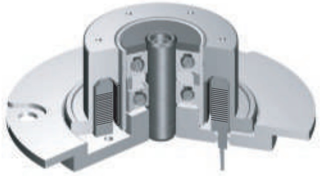

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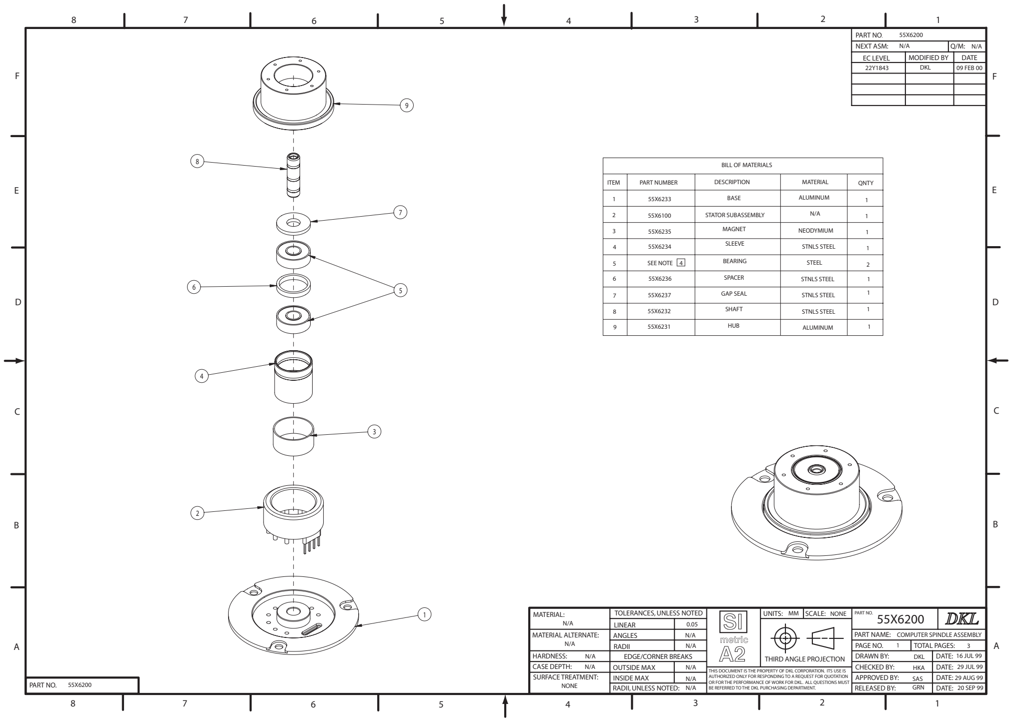
FIGURE 14.26. A 3-D computer-generated model of a computer disk drive spindle.

The 3-D model of a computer disk drive spindle is shown in Figure 14.26. The working drawings for this spindle are shown in Figure 14.27. The anticipated production volume of this product (more than 250,000 per month) is very high; however, performance requirements for its application demand very strict tolerance control on many of its part dimensions. Close attention needs to be paid to fabrication techniques to ensure that the required tolerances can be met with a minimum of manufacturing cost. The computer spindle uses two commercially



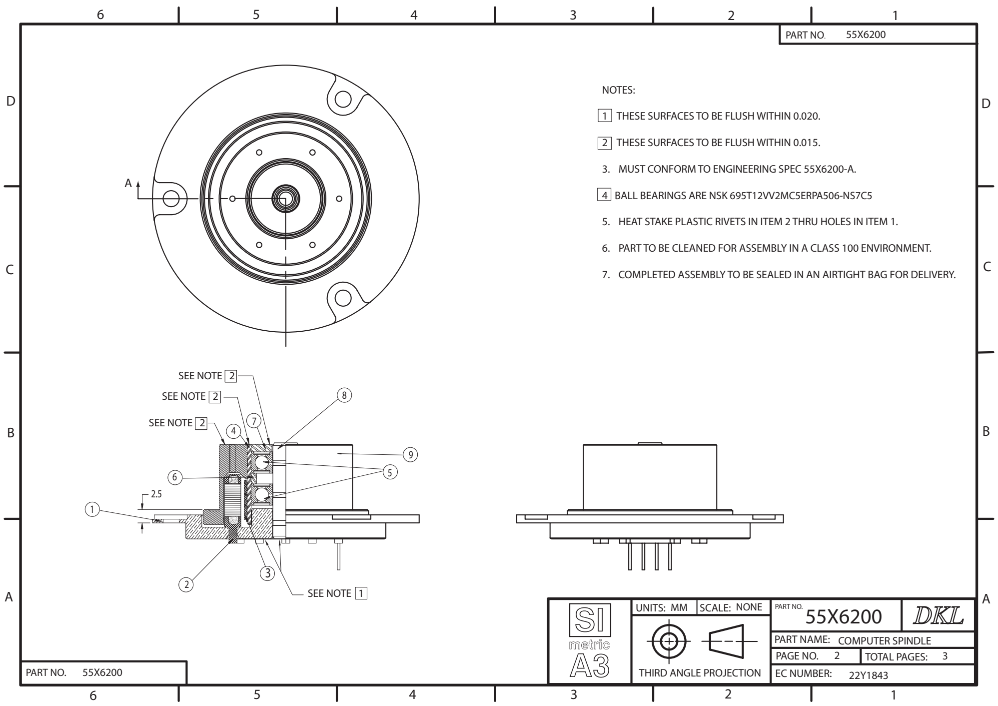


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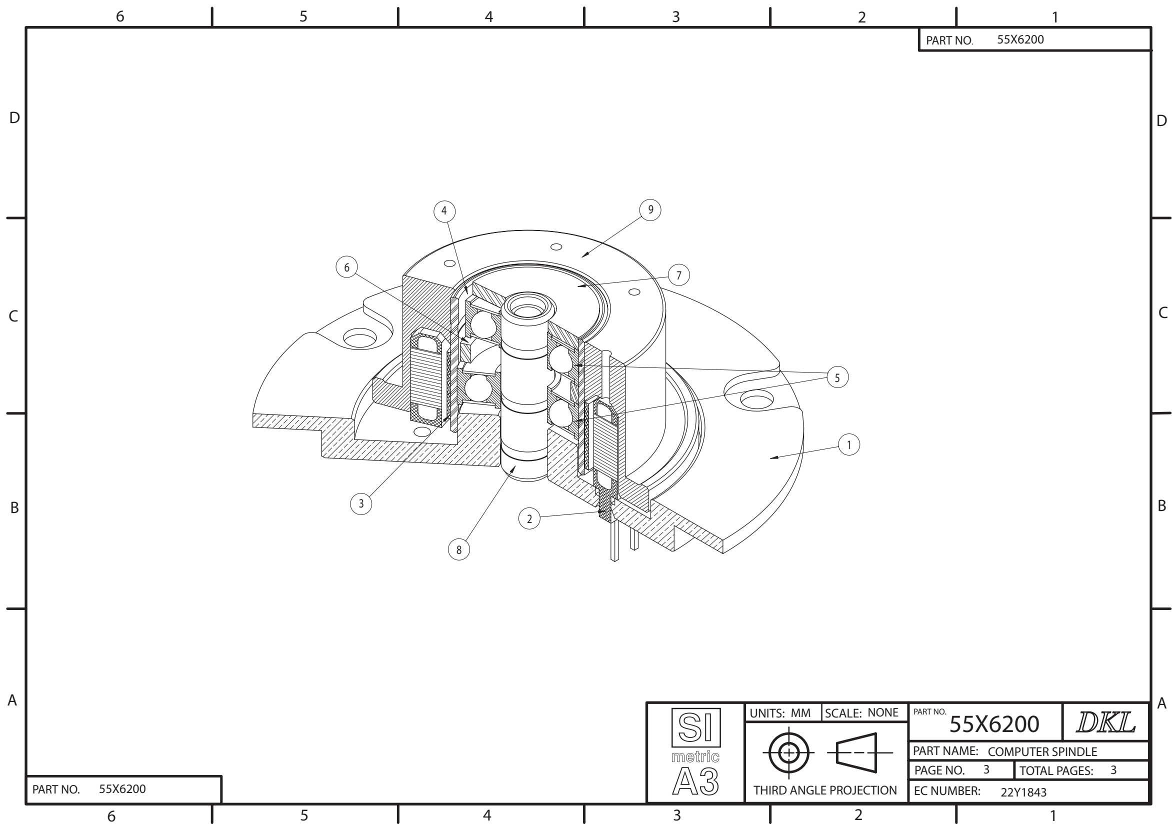
FIGURE 14.27. Working drawings for the computer disk drive spindle.



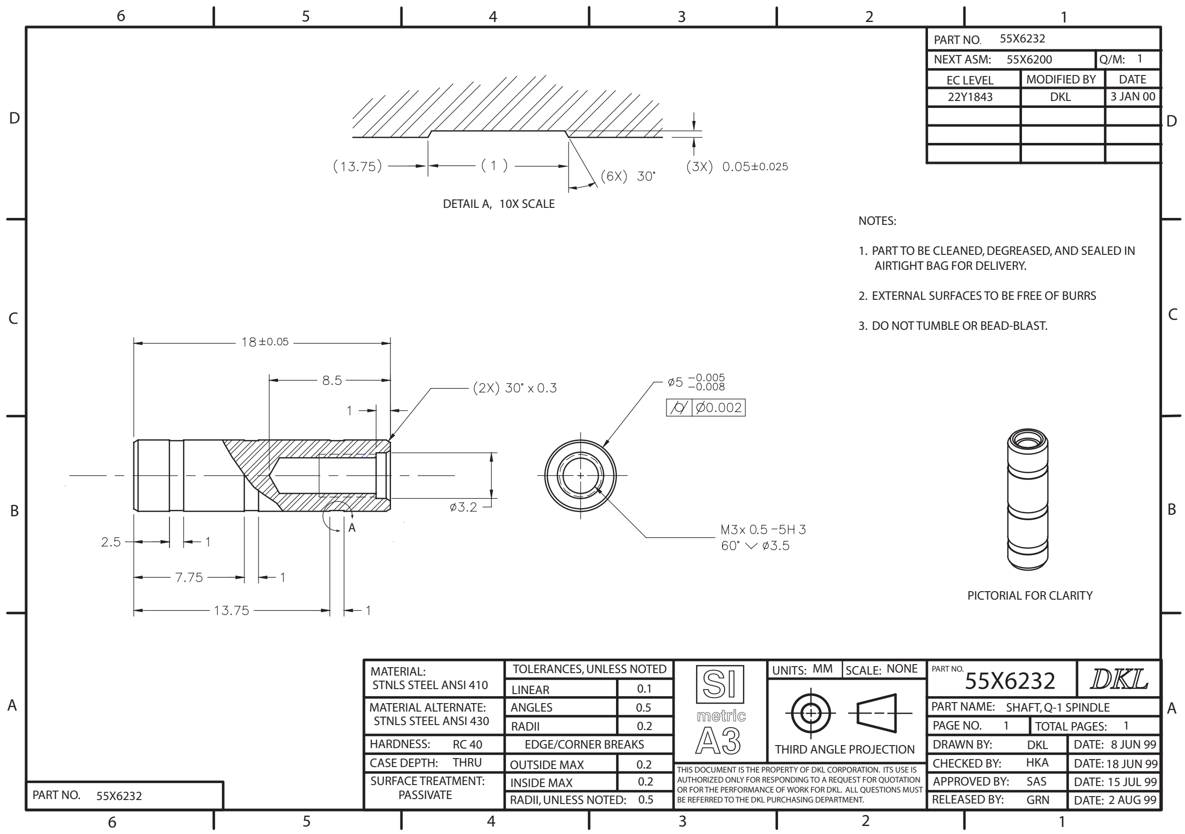


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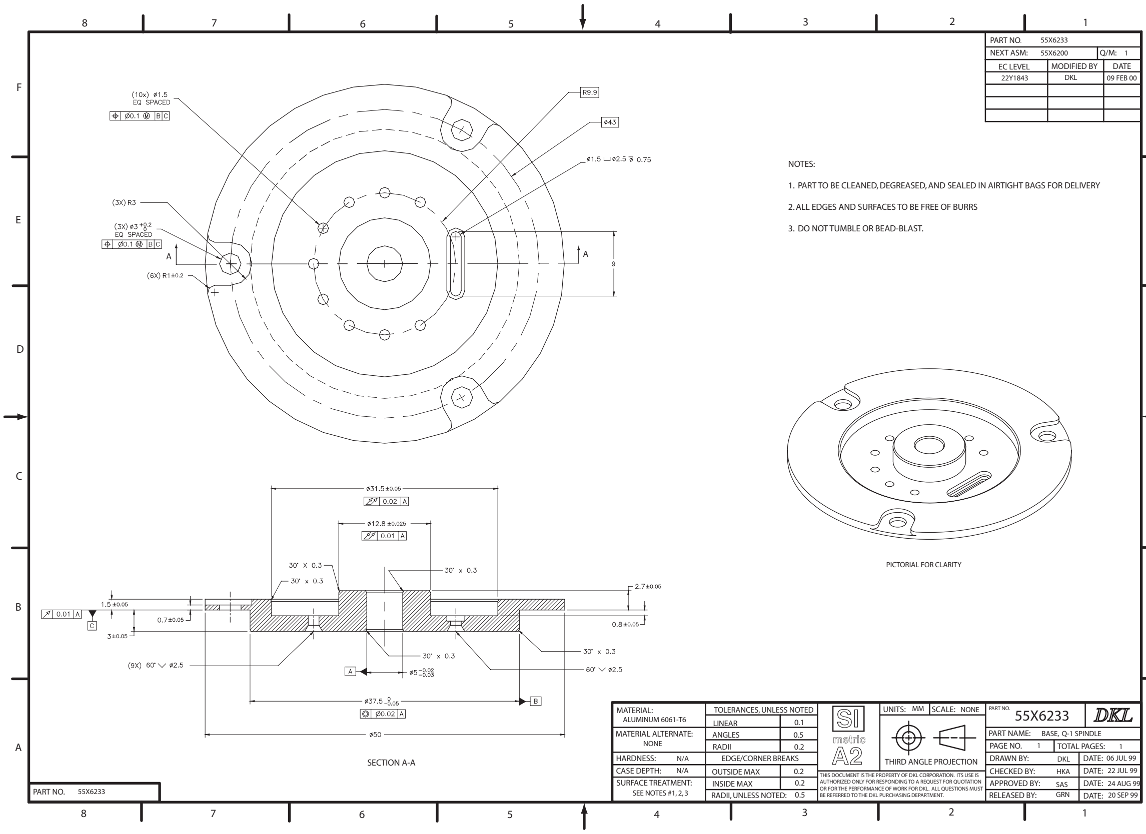
FIGURE 14.27. (CONTINUED) Working drawings for the computer disk drive spindle.



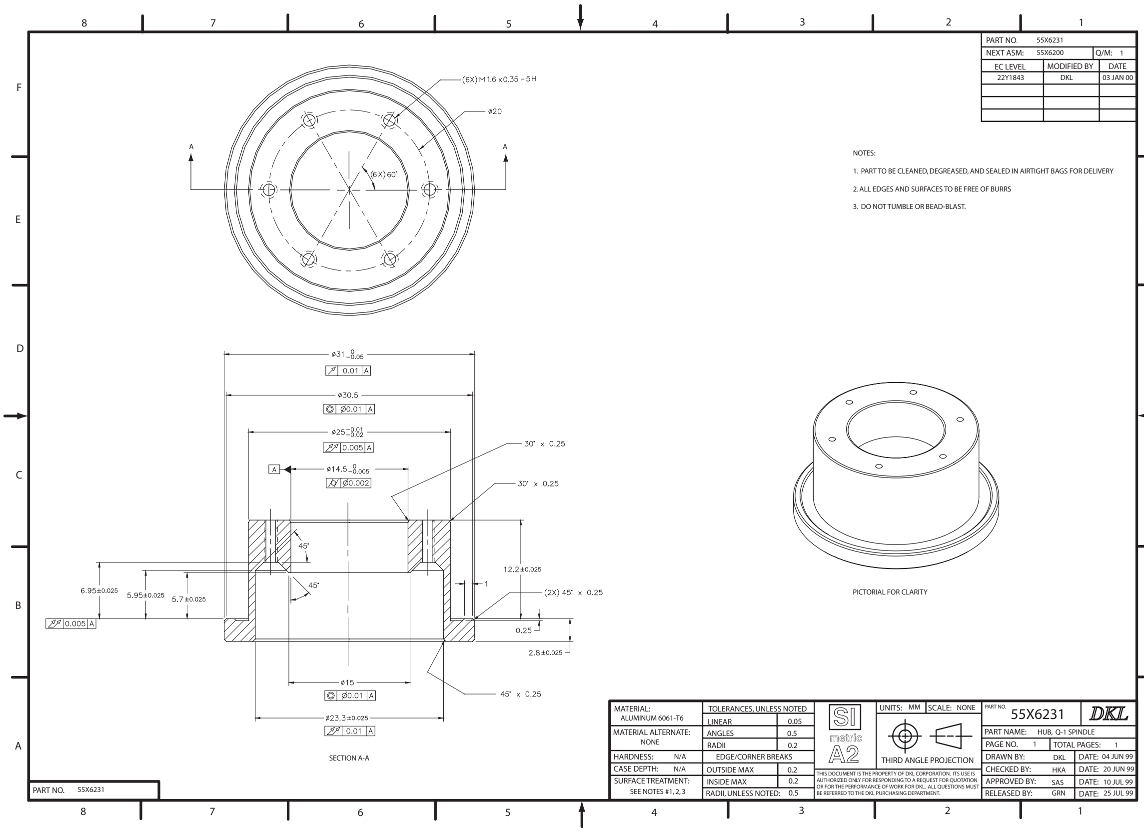


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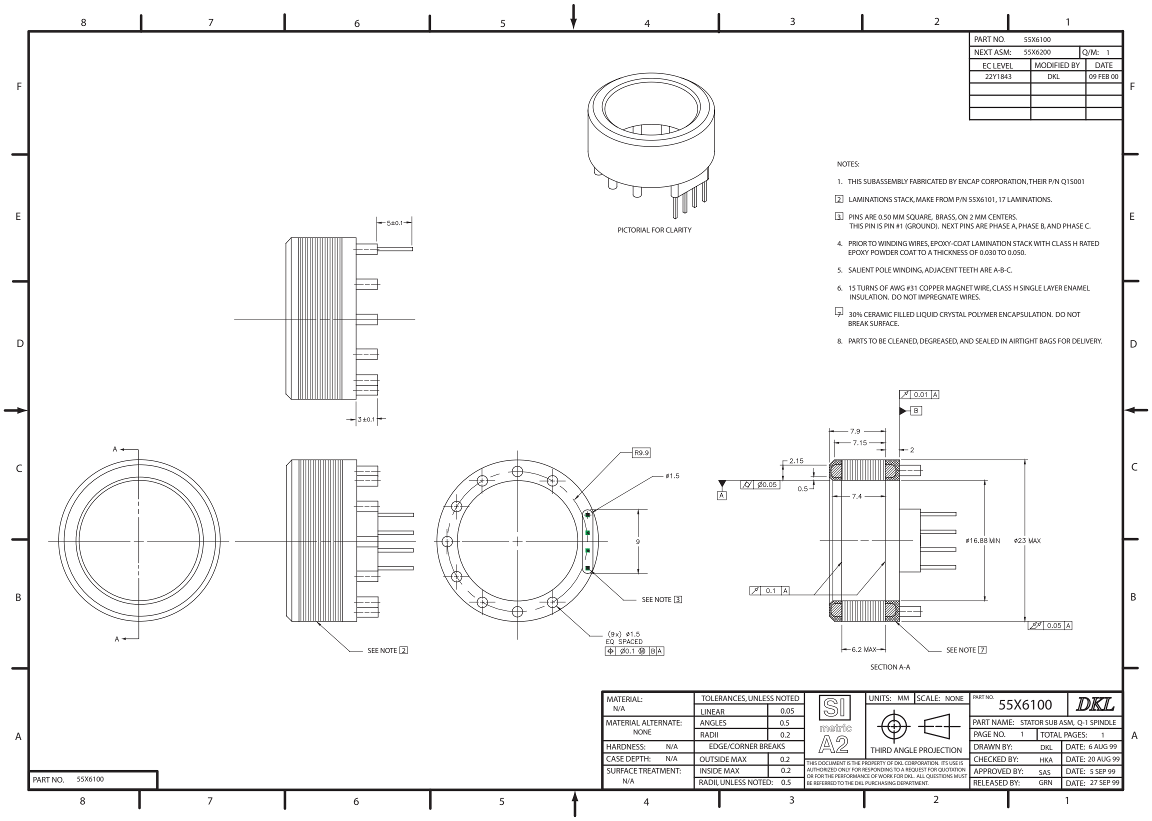
FIGURE 14.27. (CONTINUED) Working drawings for the computer disk drive spindle.



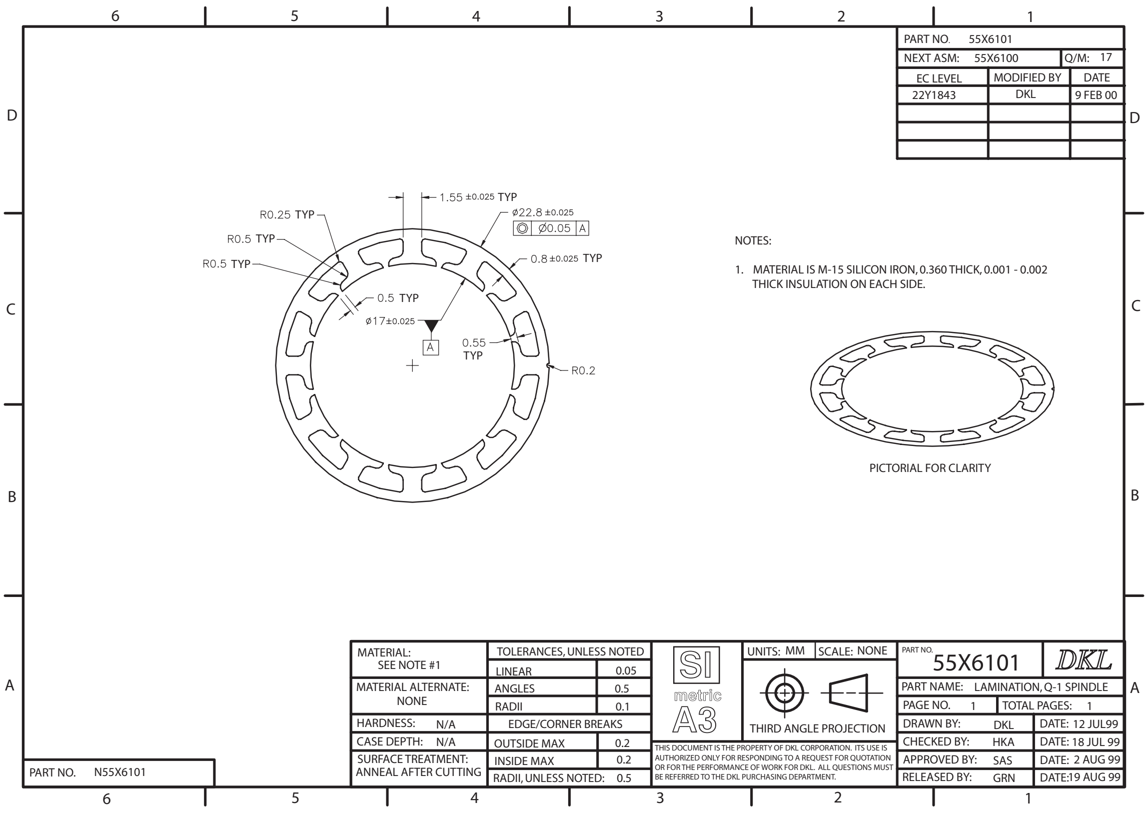


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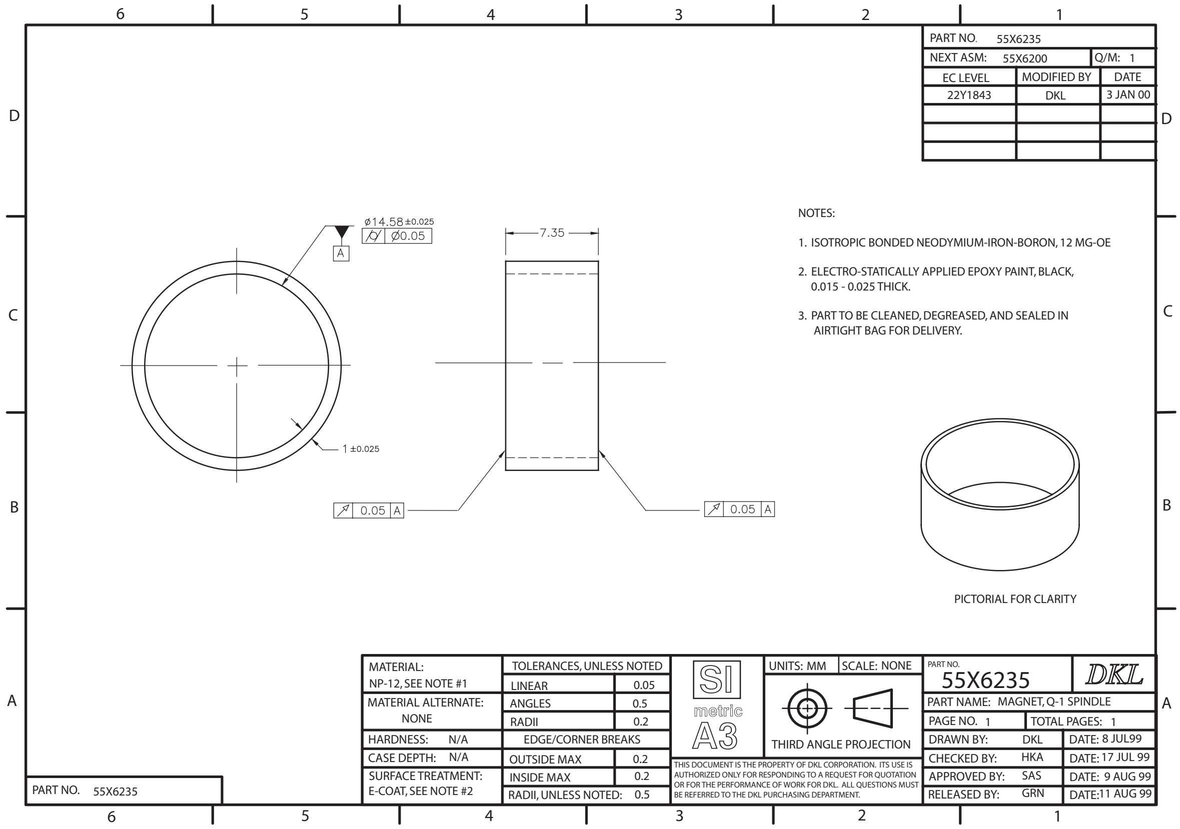
FIGURE 14.27. (CONTINUED) Working drawings for the computer disk drive spindle.



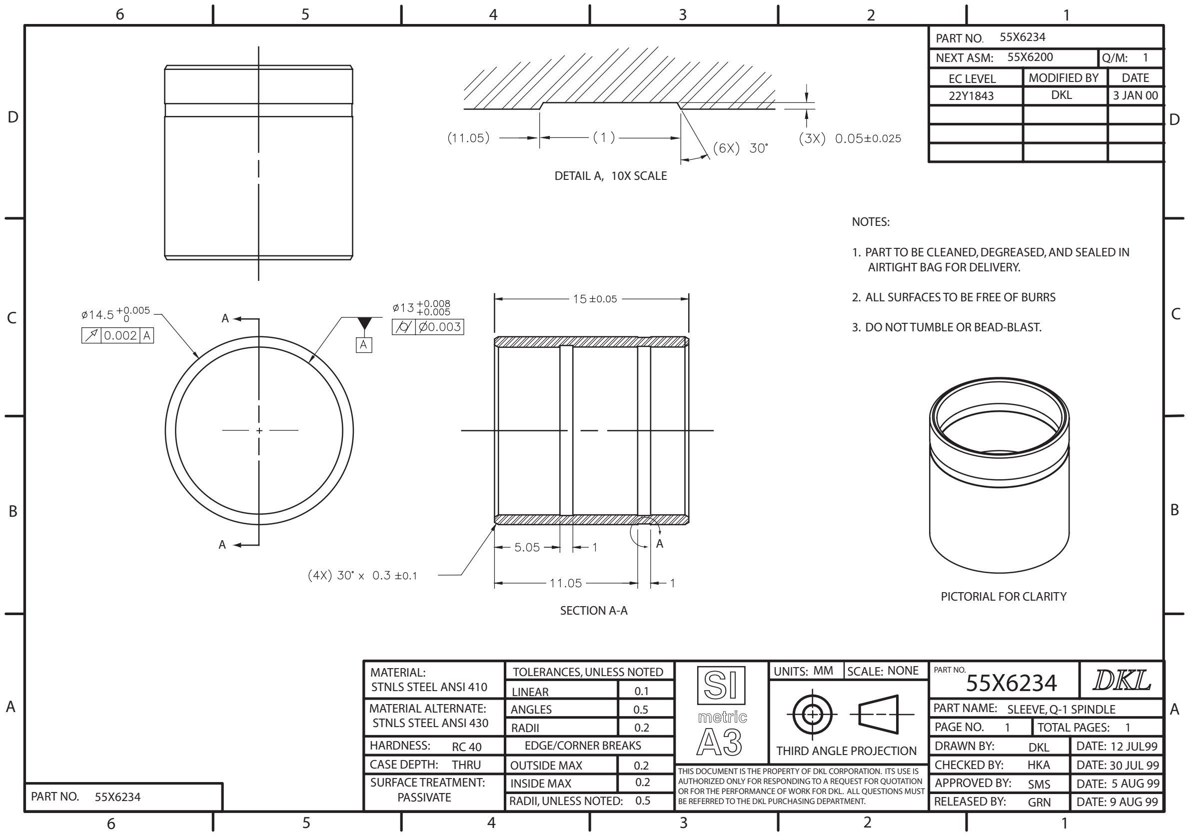


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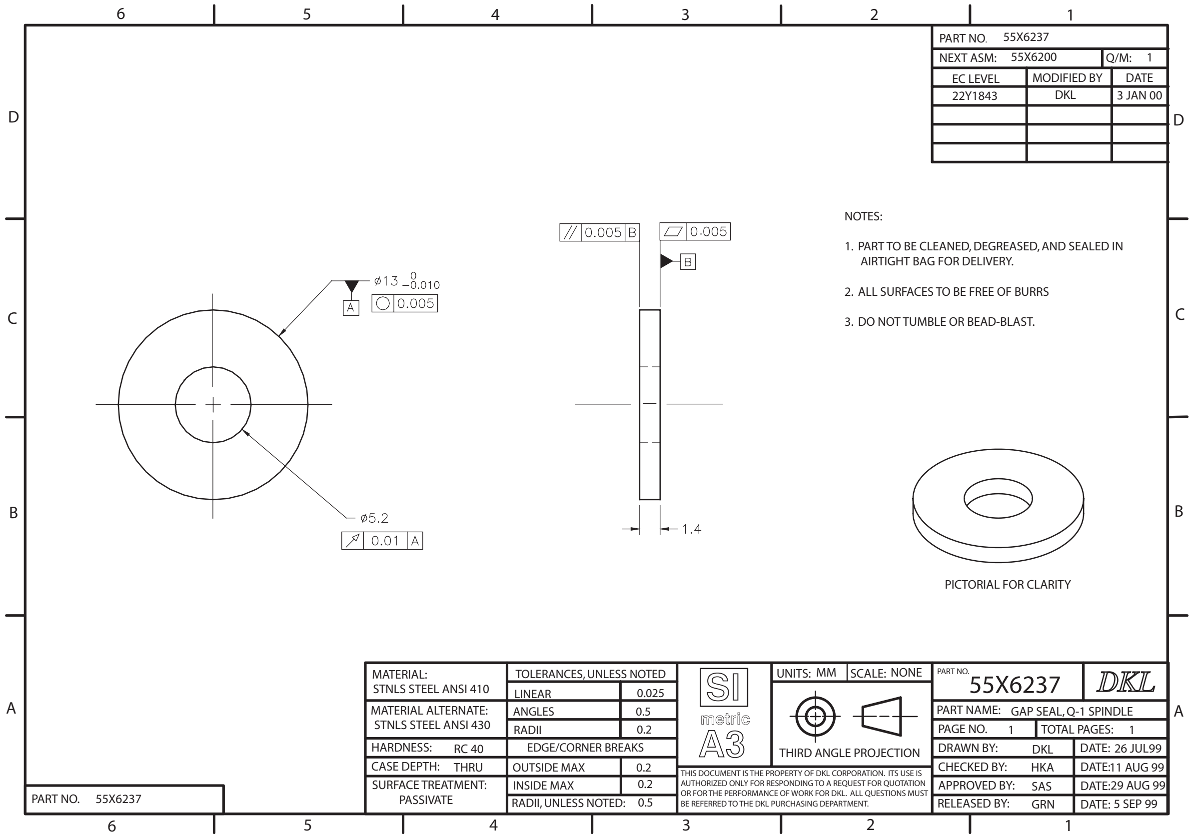
FIGURE 14.27. (CONTINUED) Working drawings for the computer disk drive spindle.



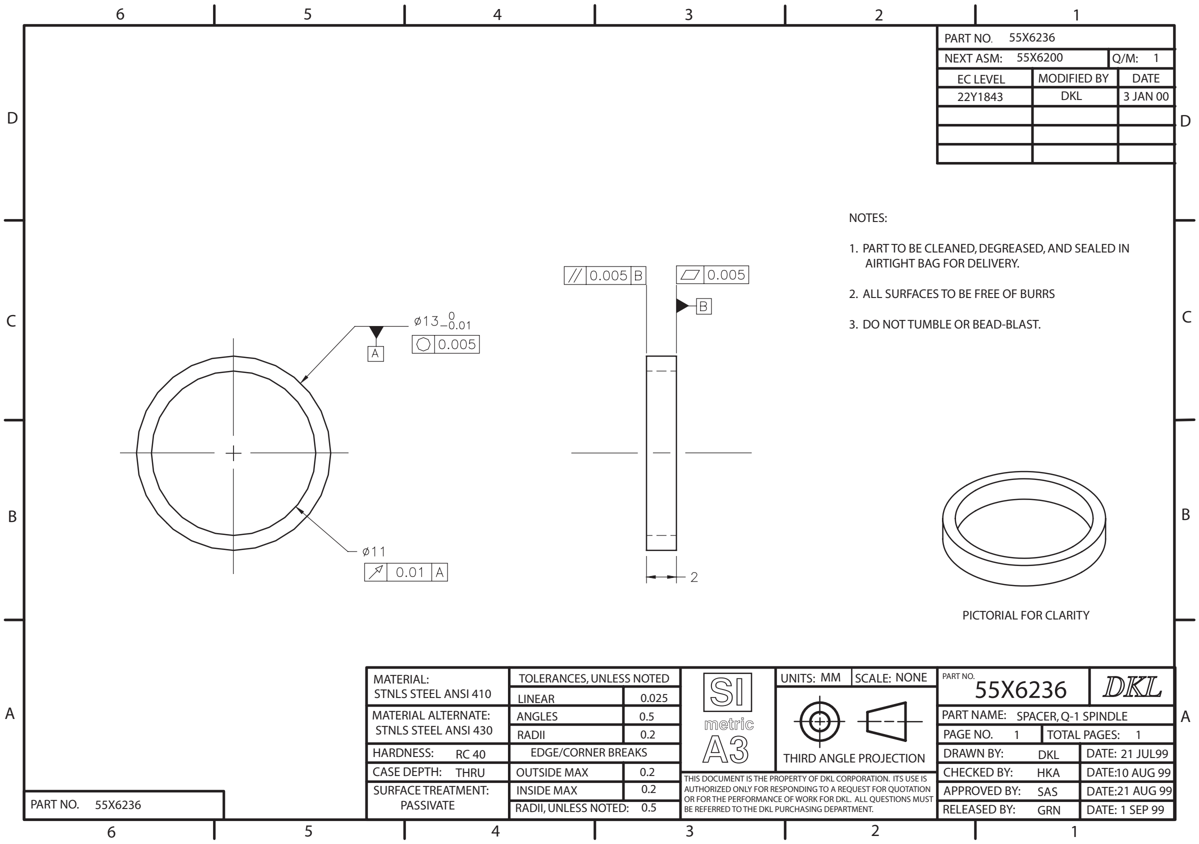


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FIGURE 14.27. (CONTINUED) Working drawings for the computer disk drive spindle.

available ball bearings; detail drawings are not necessary, but they are shown and specified in the assembly drawing and BOM. Note also that one of the parts, the stator, in the main assembly is actually a subassembly. The stator is composed of custom laminations that are stacked, insulated, wound, and insert-molded with connector pins. The part is then delivered as a single piece, ready to be installed into the spindle main assembly.

14.07 Construction Drawings

In the previous sections, you learned about working drawings in general and about manufacturing drawings specifically. In the following sections, you will learn about a different type of drawings—those used primarily in the construction of large civil engineering structures. In these sections, the term *structure* refers to any type of large infrastructure project, such as roads, bridges, buildings, and dams.

14.07.01 Why Construction Drawings Are Different from Manufacturing Drawings

Earlier in this chapter, you learned that one of the primary differences between manufacturing and construction engineering projects is that mechanical designs are developed for mass production, whereas civil designs represent a single, one-off system. Another significant difference between the two types of engineering projects is that construction projects are typically site-specific and manufacturing projects typically are not. Bridges are constructed at specific locations. Water treatment facilities are located on specific property. Mechanical products are manufactured and shipped to various locations throughout the world—it does not matter where they are manufactured or where they are used.

Another difference between construction and manufacturing projects is their relative size. Construction projects are typically large-scale. Bridges can be several miles in length; buildings can be several stories high; dams can be massively large; sewage treatment plants can cover several acres. Manufactured products are typically shipped from one location to another for eventual use, and their size is relatively small when compared to construction projects.

Yet another difference between construction and manufacturing projects is that in the United States, construction projects are often designed in the English system of units. Although the government has encouraged the construction industry to adopt the metric system of measurement (and in some cases has required that civil designs include metric dimensions), the metric system is generally not used for this type of project. In fact, in many cases where engineers working on infrastructure projects were required to include metric dimensions, they merely converted the dimensions from English to metric and did not actually *design* the system in metric. Thus, a dimension might be given on a construction drawing as 25.4 cm, having been converted from something originally designed to be 10 inches. In contrast, manufacturing projects are often designed from the start in the metric system; so dimensions will appear as whole numbers such as 25 cm or 30 cm.

Infrastructure projects are often designed and constructed for the members of society. In fact, the name *civil engineering* comes from the profession's origins in France where citizens demanded roads, water, and sanitary systems for the *civilians*—these facilities were already in place for the armed forces. Because construction projects are designed for use by the general public, they are usually required to be approved by a registered PE. A PE must pass two tests that assess his or her level of proficiency in solving engineering problems. In addition, an engineer must work for several years under the supervision of a PE before being eligible to attempt the second day-long test. After passing the exam, the new PE is legally and ethically responsible for the integrity of the designs developed under his or her supervision and can be sued if a structure fails. In contrast, manufacturing projects rarely have PEs working on them, although the senior engineer on a project usually has several years of experience and would not have been trusted to verify the integrity of the design without the demonstrated ability to perform this function.

Although 3-D computer modeling predominates in manufactured systems, its use in civil engineering design practice is still fairly limited; this trend will likely continue for the foreseeable future. In some of the larger civil engineering firms, 3-D models of projects are created, but these computer models are typically specialized for civil applications and cannot be used to generate 2-D drawings directly from the models. The 2-D drawings are still created independently from the 3-D models. For manufactured products, drawings are becoming less important, especially as modern software enhances the ability to send 3-D computer models electronically to CNC lathes for production. This is not the case for civil and architectural applications. In civil engineering practice, design and construction are still accomplished primarily through drawings. Construction projects are not built in climate-controlled, clean environments. Projects are constructed outside with exposure to the elements, often far away from electric power sources or network connections. Physical drawings are still far more practical in this environment than are 3-D computer models.

Finally, construction projects are like large-scale assembly projects that are always built from the ground up. With a construction project, the contractor performs the site excavation first; then the foundation is poured. The first floor of a structure must be built before the second floor can be built, and the second floor must be constructed before the third floor can be constructed. All of the floors must be complete before the roof can be added. Also, wiring, plumbing, and ductwork must be in place before the walls and ceilings can be completed. Further, each subsystem on the project, like the wiring and plumbing, is typically put in place by a subcontractor who is hired by the general contractor on the project. Thus, there is a specific order and timing in which the various parts of the project are completed; and significant communication and coordination are required between the client, engineer, contractor, and subcontractors. Scheduling is a significant part of a construction project. Manufacturing projects typically do not require this complex level of communication, scheduling, or coordination and do not typically have a rigid order for assembly. (Although, of course, some subassemblies must be put together before other subassemblies.)

14.07.02 How Construction Drawings Are Different from Manufacturing Drawings

Due to the some of the differences between construction and manufacturing practices described in the previous paragraphs, several differences in the development of working drawings have evolved over time. In the following paragraphs, these differences will be described and illustrated.

Terminology

Drawings used in the design and construction of civil or architectural projects are frequently referred to as **blueprints**. The name *blueprint* is derived from an earlier era when construction drawings were reproduced by a method that resulted in a blue background with white lines. The original drawings were made using ink on large sheets of paper; but there was no such thing as copiers or printers, especially of this size. Special blueprint machines were developed so that multiple copies of the handmade ink drawings could be produced. Although modern-day computer hardware with large printers has enabled the creation of construction drawings with black lines on a white background, they are still often referred to as blueprints. Figure 14.28 shows a blueprint of a hand-drawn sketch of a mixer plate design. Note that the title block shows the date that this blueprint was drawn.

In construction applications, **plan views** are views made from a vantage point above the "object." Thus, plan views can be thought of as top views. You are probably familiar with the term **floor plan**; in fact, you may have seen a floor plan in a newspaper or magazine. A floor plan is a drawing made from a vantage

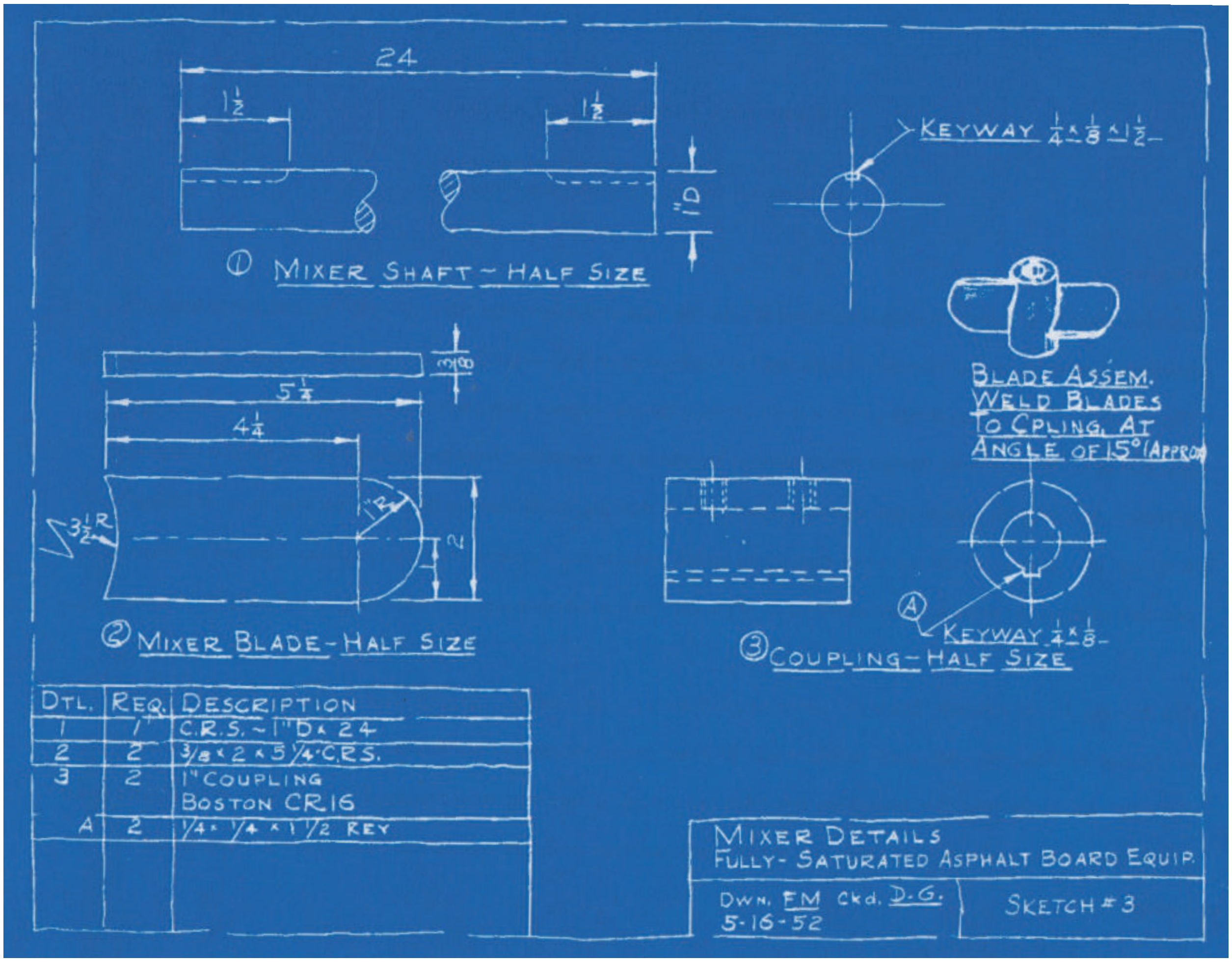

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FIGURE 14.28. Hand-drawn blueprint of a mixer plate.

point above a building that shows the layout of all of the rooms on a particular floor. Similarly, a **foundation plan** shows the building foundation from above, the **electrical plan** shows the wiring diagram from above, and the **heating and ventilation plan** shows the location of ducts and equipment from above. **Profile views** show the building or project from the front or the side. In other words, they are views where the top of the structure is seen as an edge. **Elevation views** are drawings that show differences in elevations on a structure. Since changes in elevation can be seen in any view where the top is an edge, elevation views are essentially the same as profile views.

In the design and construction of large infrastructure projects, several drawings are necessary to describe the facility completely so that it can be built. The entire set of drawings is called the **set of construction plans**, or "the plans," even though not all of the drawings represent plan (top) views of the structure. The **specifications**, or **specs**, consist of written instructions regarding the construction of the facility. Together, the plans and specs make up the entire construction documentation. In this text, you will focus on understanding the drawings—specifications are beyond its scope.

Size Considerations

Construction drawings are almost always created or printed on E size sheets. Recall that E size sheets are 34" × 44"; however, smaller sheets may be used for drawings that are brought into the field. Large sheets are used for construction drawings due to the relative large size of the projects. For a bridge that is three miles long, it would be difficult to show the entire structure on an 11" × 17" sheet of paper (B size) or to show it in sufficient detail for understanding. Construction drawings are always made to scale, with 1 inch sometimes equaling hundreds of feet. It would be impossible (and impractical) to draw a large structure true size—imagine the size of the sheet of paper that would be required.

The relatively large size of construction projects also makes strict tolerancing relatively meaningless. When a slab of concrete is specified as 10'-6" × 40'-9", no one expects the slab to be *exactly* that size; plus or minus a few tenths of an inch is probably acceptable. For concrete slabs, making sure that the surface is level is far more important than its overall surface area. Smaller tolerances may be needed when bolt holes are located on a steel structural member; however, once again, the tolerances are nowhere near the precision found in manufacturing projects where tolerances as small as 0.001" are acceptable and routine. A contractor would likely laugh out loud if a tolerance of 0.001" were ever specified on the design drawings for an infrastructure project.

Another difference in the working drawings that results due to the large size of construction projects is that views typically do not project orthogonally from one view to the next. Often, the plan view (or top view) is on one sheet with the elevation view (or front view) located several sheets away. Sometimes even the scale used to draw the plan view differs from the scale used to draw the elevation view, meaning you could not separate the sheets and try to line them up if you wanted to see how features projected from one view to the next. In this respect, the need for well-developed 3-D spatial skills may be even more important for engineers working on construction projects. You often must remember what the plan view looks like as you search for the elevation view on a separate sheet.

Site-Specific Considerations

In the construction industry, the orientation and location of a project with respect to its surroundings are extremely important. Imagine the problems that would develop if a building were constructed on someone else's property. Several methods used on drawings help the contractor locate the structure properly. **Bearings** of lines may be shown on a drawing. The bearing of a line is the angle that the line makes with a North-South line, as illustrated in Figure 14.29. Bearings of lines are seen only in plan views (i.e., from above). On the construction site, bearings of lines can be obtained by any of several surveying techniques and the building can then be accurately located on the property. Alternatively, a North line may be placed on the drawing to show the relative orientation of the structure.

Control points are often provided on construction drawings to help locate features of the project accurately. With this method, an "origin" for the construction site is designated and all points are referenced north, south, east, or west from it. Thus, a point on a drawing might have coordinates N13750 and E7895, for example. Similar to bearings, the coordinates of the control points are seen only in plan views. The origin for the coordinate system is usually referred to as a **benchmark**. Benchmarks have been established across the United States by the U.S. Geological Survey (USGS) and typically consist of a concrete cylinder with a brass, circular medal on top imbedded in the earth. The location of each benchmark was determined with a high degree of accuracy. Many times, job benchmarks are established on construction sites when a USGS benchmark is not located within the vicinity of the project.

Benchmarks are also used to determine the elevation of points on a construction site. Elevations are used to establish vertical distances between points on a building. For example, the top of a floor slab might be specified as having an elevation of 556 feet. Elevations are seen only in profile views (or elevation views) and are usually referenced to true elevations (i.e., the height of the point above sea level) or to job elevations. With job elevations, a benchmark is established and given an arbitrary elevation of, for example, 100 feet. All other elevations for the project are then specified relative to that point. A benchmark elevation of 0 is usually not specified to ensure that job elevations are never negative.

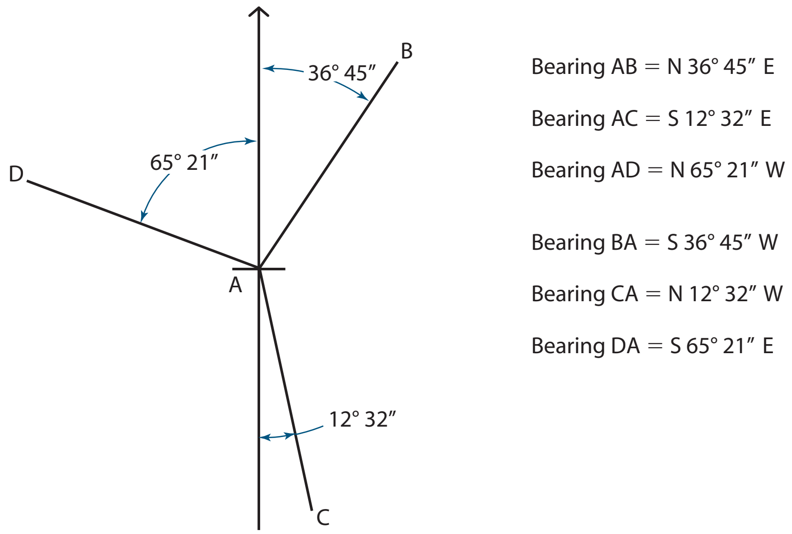

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FIGURE 14.29. Illustration of the definition of a bearing of a line. Bearings are only seen in the plan views.

One of the first steps in the design of a structure is to send a crew out to do a **site survey**. Usually, there are existing structures or features that must be noted on the plans before the design can proceed. Modern survey equipment is computerized such that the survey data are automatically stored and later easily converted to a drawing. Figure 14.30 shows the survey data for a site taken in the field; Figure 14.31 shows the survey data after they have been converted to a site plan drawing.

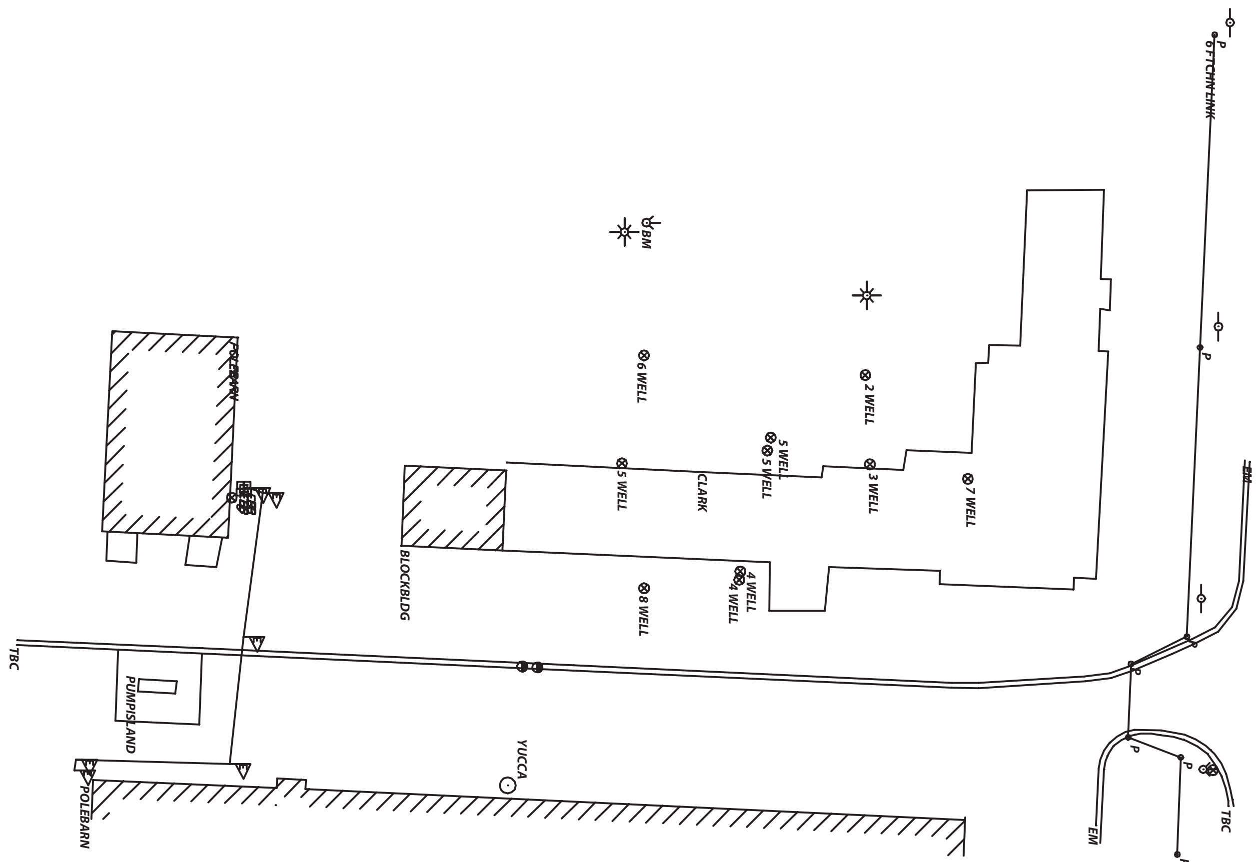

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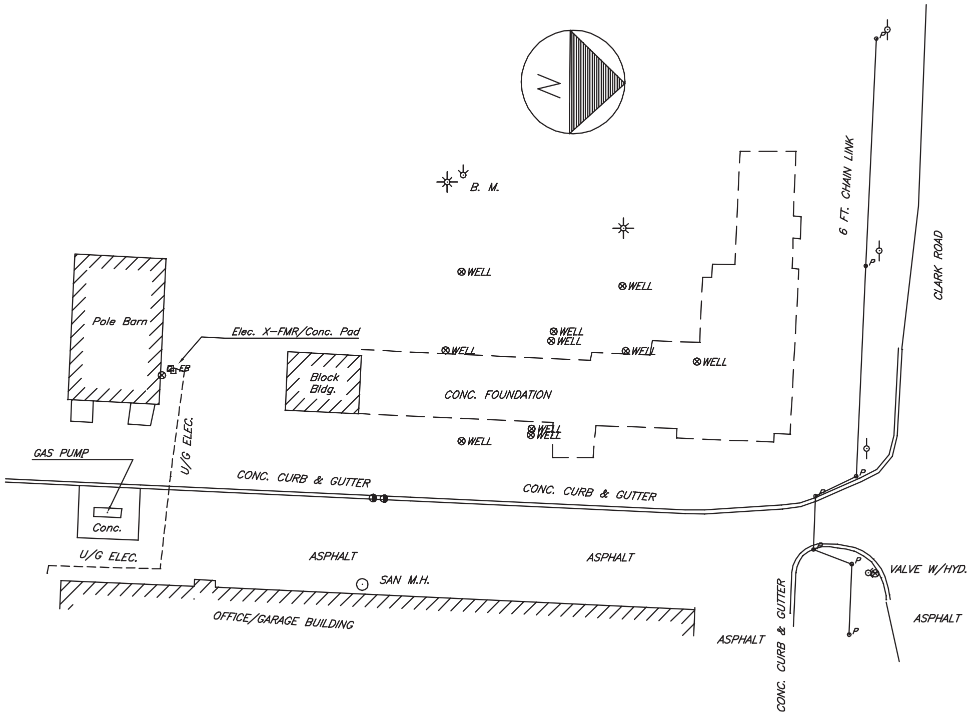
FIGURE 14.30. Data from site survey showing existing structures and other entities.

FIGURE 14.31. Survey data converted to site drawing.



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One-of-a-Kind Considerations

In manufacturing projects, prototypes of products are created and tested for their integrity and functionality. The prototypes can be virtual or actual physical models. Construction projects are too large and expensive to warrant the building of prototypes; however, sometimes small-scale models are developed to allow clients and others to visualize what a structure will look like when completed. The models built in construction practice have little value in analyzing the structure—they are merely used for display purposes, especially when dealing with a client who may not be able to visualize a project based on the plans. Virtual computer models may also be available for analysis, especially in larger engineering firms. Because large structures are one-offs, unforeseen problems are likely to occur during construction. Ductwork may interfere with plumbing pipes and need to be rerouted. Dimensions may have inadvertently been left off the drawing. The engineer may discover that walls or doorways need to be moved. Because changes to the original design may need to be made during construction, the contractor keeps track of them on a set of **as-built plans**. As-built drawings graphically show any changes from the original design and are important for future maintenance and operation of the facility.

14.08 Construction Plans

A set of construction plans usually consists of a large number of drawings. The drawings in the plans usually include the cover sheet, site plan, elevation views, foundation plan, floor plans, electrical plans, roofing plans, sections, detail drawings, and any other drawings needed to describe the project completely. The drawings are included in the set of plans in the order in which they are needed for the construction of the project. This means that the foundation plan appears before the first-floor plan, the floor plans appear before the roofing plan, etc.

For a complete set of plans, several sheets are often required for each type of drawing. In addition to the drawings, **schedules of materials**, which list, for example, the types of doors and windows to be used in the construction of the facility, are included on the drawings. In this case, a schedule of materials is much like a BOM for manufacturing projects. Most public projects require the seal and signature of a PE, which often appears on the cover sheet of the drawing set. The PE is legally responsible for the design and construction of the facility even if he or she did not complete all of the analysis for the project. A PE will usually meticulously check the calculations and analysis before signing off on the plans. The name of the PE appears in the title block of each drawing in the set of plans. The title block also typically includes the name of the drafter who made the drawings and others involved with the design; however, title blocks on construction drawings do not typically contain all of the information found on the title block for manufacturing drawings.

14.08.01 Cover Sheet

The **cover sheet** for the plans typically contains a map of the area surrounding the project site. The map is not overly detailed, but shows the general location of the project. Since the set of drawings for a project usually consists of several sheets, an **index** of all drawings in the set is included on the cover sheet or on the first page following the cover. Figure 14.32 shows the cover sheet from a set of plans drawn for an expansion project for a water treatment plant in the village of Baraga in the state of Michigan. Note the map showing part of the state with the portion of the state near the project site enlarged to show the area in greater detail. Also note the index listing all of the drawings in the set of plans and the area reserved on the cover sheet for the seal of the PE.

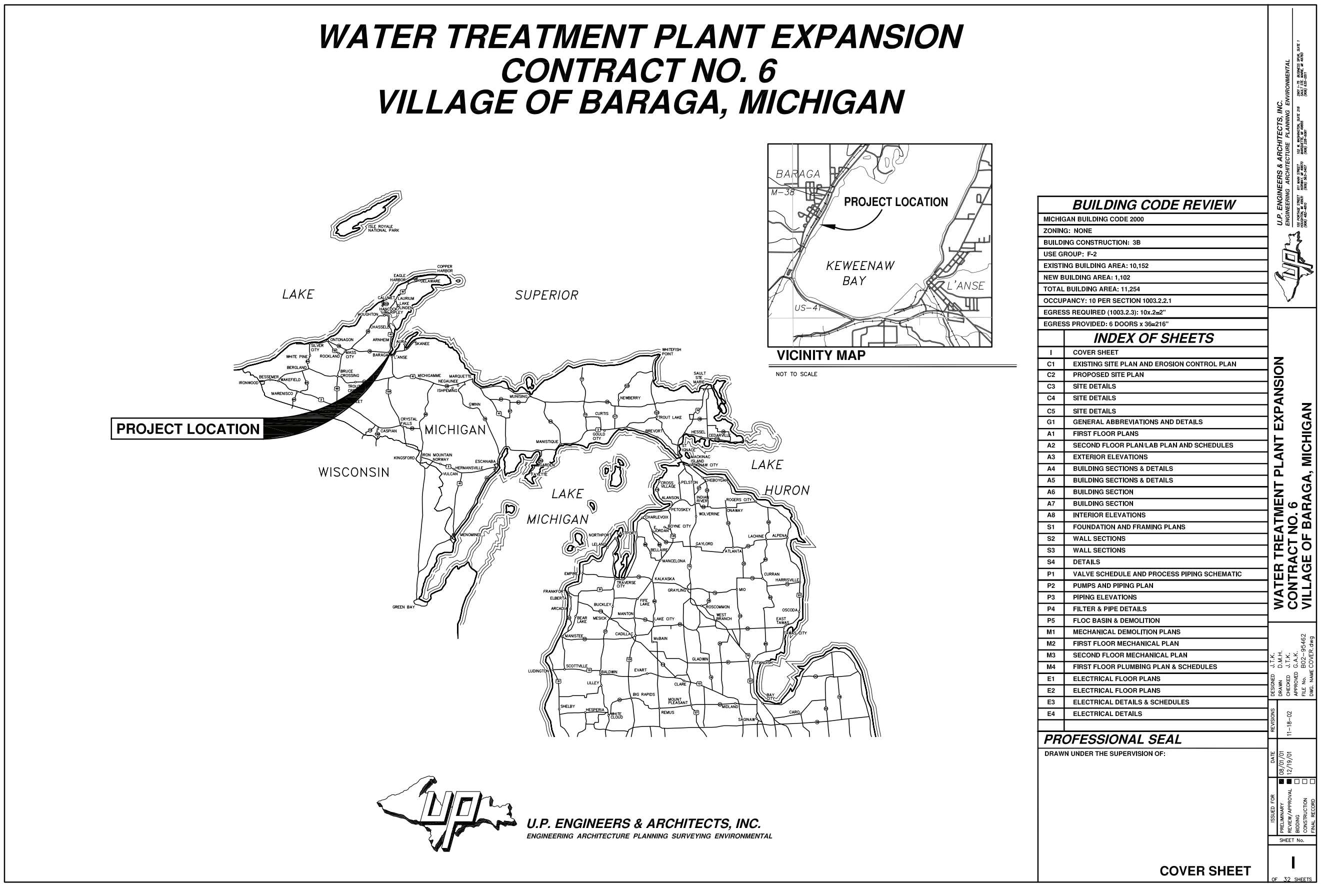

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FIGURE 14.32. Cover sheet and index for the Baraga water treatment plant expansion project.

14.08.02 Site Plan

One of the first drawings in the set of construction plans is a site map or a **site plan**. Figure 14.33 shows the proposed site plan for the Baraga water treatment facility. Note that this plan shows the highway as well as an arrow indicating north. Since this project constitutes an addition to an existing structure, the outline of that building is shown on the site plan as well. The scale for the drawing is shown in the upper-left corner both graphically and numerically (1 inch = 20 feet). Contour lines showing changes in existing ground elevation are also shown. You will learn more about contour lines and topographic maps in a later chapter.

14.08.03 Elevation Views

As stated previously, elevation views show the structure from a vantage point where changes in elevation are visible. Elevation views can be thought of as front or side views; but for a large structure, the terms *front* and *side* are fairly meaningless. Elevation views are defined by their orientation with respect to the compass points of North, South, East, and West. A North Elevation shows what the structure would like if you stood to the north of it and looked back, a South Elevation shows what it would look like from the south, etc. Although elevation views do not contain a great deal of detail or many dimensions about the actual construction of the facility, they do help contractors and owners visualize the resulting project. Figure 14.34 shows the South Elevation view for the overflow and drain portion of the lagoon for the Baraga water

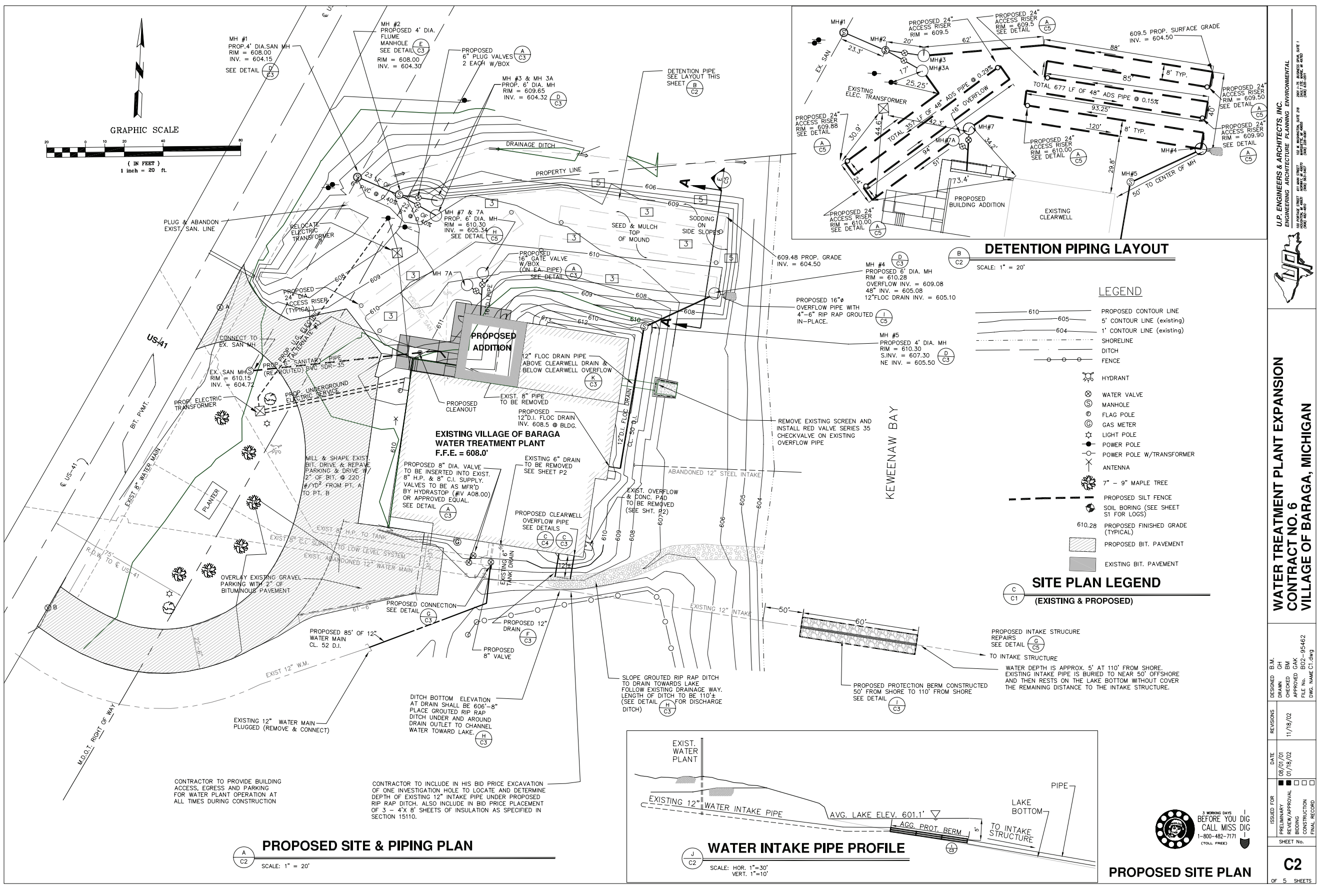

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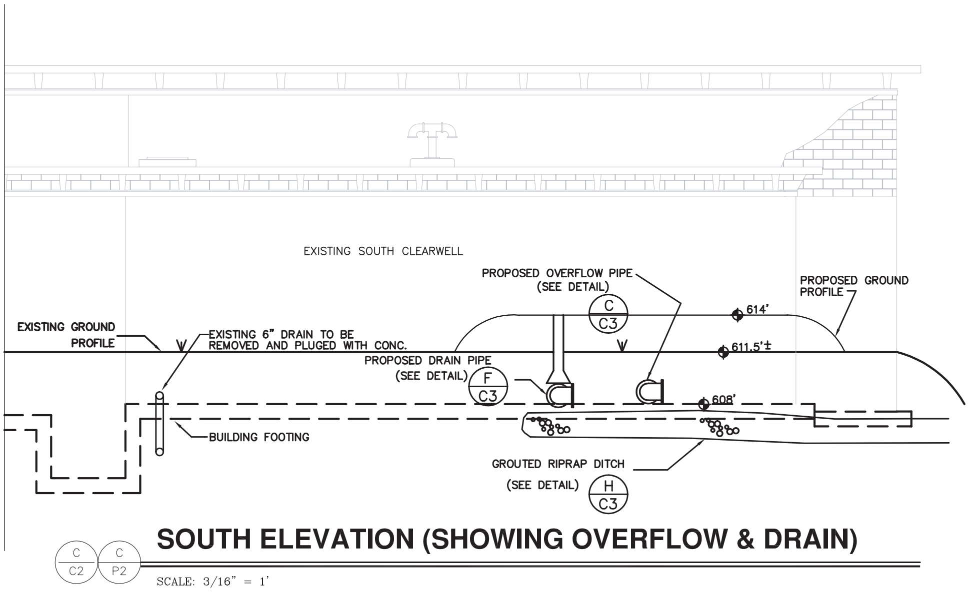
FIGURE 14.33. Site plan for Baraga water treatment facility showing existing structure as well as nearby highway.

FIGURE 14.34. Elevation view for overflow and drain portion of lagoon.



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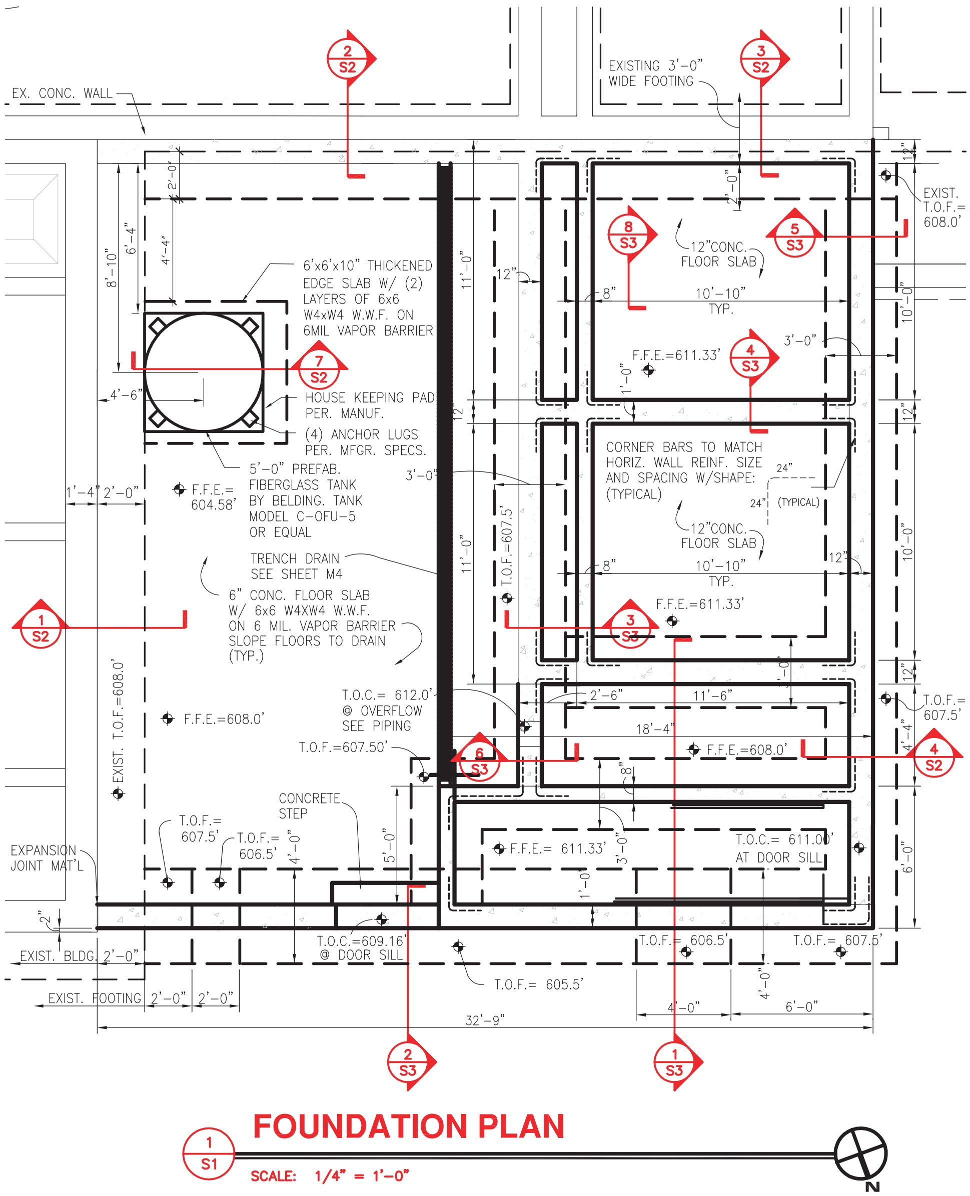
treatment facility. Note that there are no dimensions on this drawing, but elevations of some features are included—the existing ground profile has an elevation specified of 811.5' ±, and the proposed ground profile shows an elevation of 814'. The symbol for elevation on the drawing is a circle with a cross through it with the horizontal "crosshair" on the surface whose elevation is being specified.

14.08.04 Foundation and Floor Plans

Because a building is constructed from the foundation up, the foundation plans are among the first drawings in the set of plans. A building foundation is usually constructed out of concrete that has been reinforced with steel bars, or **rebars** (reinforcing bars). Concrete footings support the walls and columns in a building, and the foundation walls are often made of reinforced concrete. Details about the size of the footings and the size and location of rebars are usually included in a wall section drawing. Sometimes a reinforced concrete slab is constructed for the building and included as part of the foundation plan or in a wall section view. Concrete slabs typically contain reinforcing bars or a steel mesh for controlling the thermal expansion and contraction of the slab.

Figure 14.35 shows the foundation plan for a portion of the Baraga water treatment facility. In this drawing, the right portion of the foundation includes a 12" slab and the left portion includes a 6" slab. The reinforcing for the 6" slab is specified on the foundation plan as 6X6 W4XW4 W.W.F. This specification means that the spacing of the bars is 6" × 6" and that the diameter of the steel wire is a gage of 4 (approximately two-tenths of an inch). The WWF in this specification refers to welded wire fabric. The dashed lines around both sides of the outer walls of the structure define the footings. Since this project is an addition to an existing structure, the existing footing sizes are given—3'-0" along one wall and 2'-0" along another wall. The new footings to be constructed are 4'-0" and 3'-0" on each of the remaining two walls, respectively. Note the specification of the elevation of the T.O.F. (top of footing) in various locations on the plan. Also notice the various cutting plane lines for sectional views through the walls. The sectional views will be found on various sheets that are labeled in this view. For example, for the wall located nearest the top of the page, two section lines are shown and both section views will be located on S2 (section sheet 2); they will be drawing 2 and drawing 3, respectively.

FIGURE 14.35. Foundation plan for portion of the Baraga water treatment facility.



The type of construction drawing with which you are probably most familiar is the floor plan. A floor plan shows the layout of the rooms in a building. Doors between rooms are shown, as is the location of windows, closets, plumbing fixtures, and any other pertinent information about the drawings. The dimensions of the rooms as well as the thickness of walls are usually shown on the floor plans. Figure 14.36 shows the first-floor plan for the Baraga water treatment facility. Notice how the new construction fits within the existing structures on two sides. For this floor, the new walls will be constructed from concrete. (Dotted cross-hatching is used to show this graphically.) A concrete landing also surrounds the new construction, connecting to the existing generator; and spaces for planting in the landing are included to avoid the "concrete jungle" look. Two doors will be installed in the new construction. (Several are shown on the existing structure.) One of the doors leads to the stairway; the other door leads directly into the pipe gallery. Notice that the way the doors swing is also shown on the plan.

Figure 14.37 shows a different type of floor plan for the first floor—the electrical plan. According to the legend provided with this electrical plan, seven types of electrical devices are to be installed on this floor—duplex convenience receptacles,

FIGURE 14.36. First-floor plan for the Baraga water treatment facility.

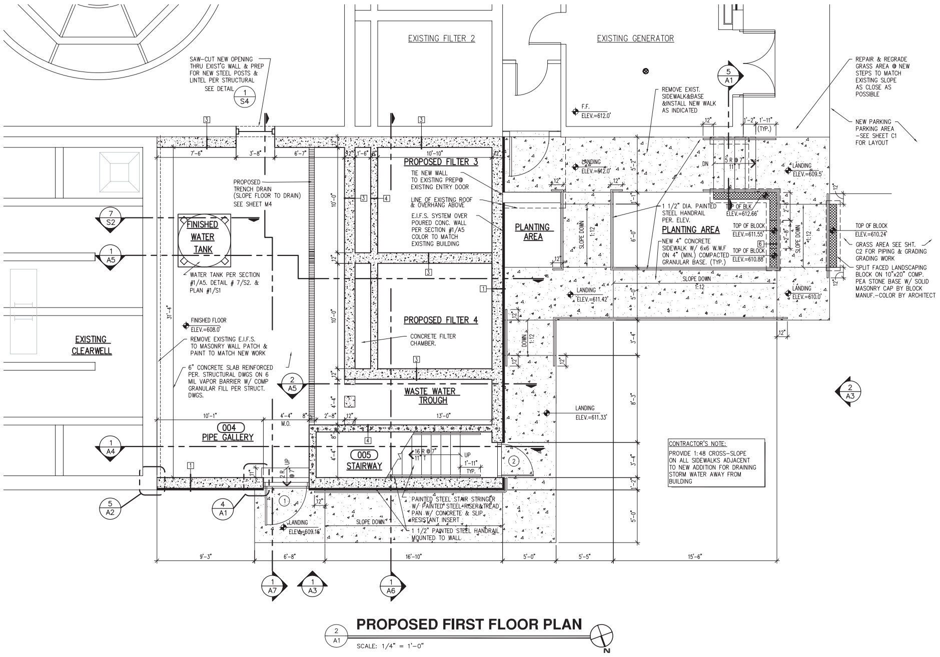


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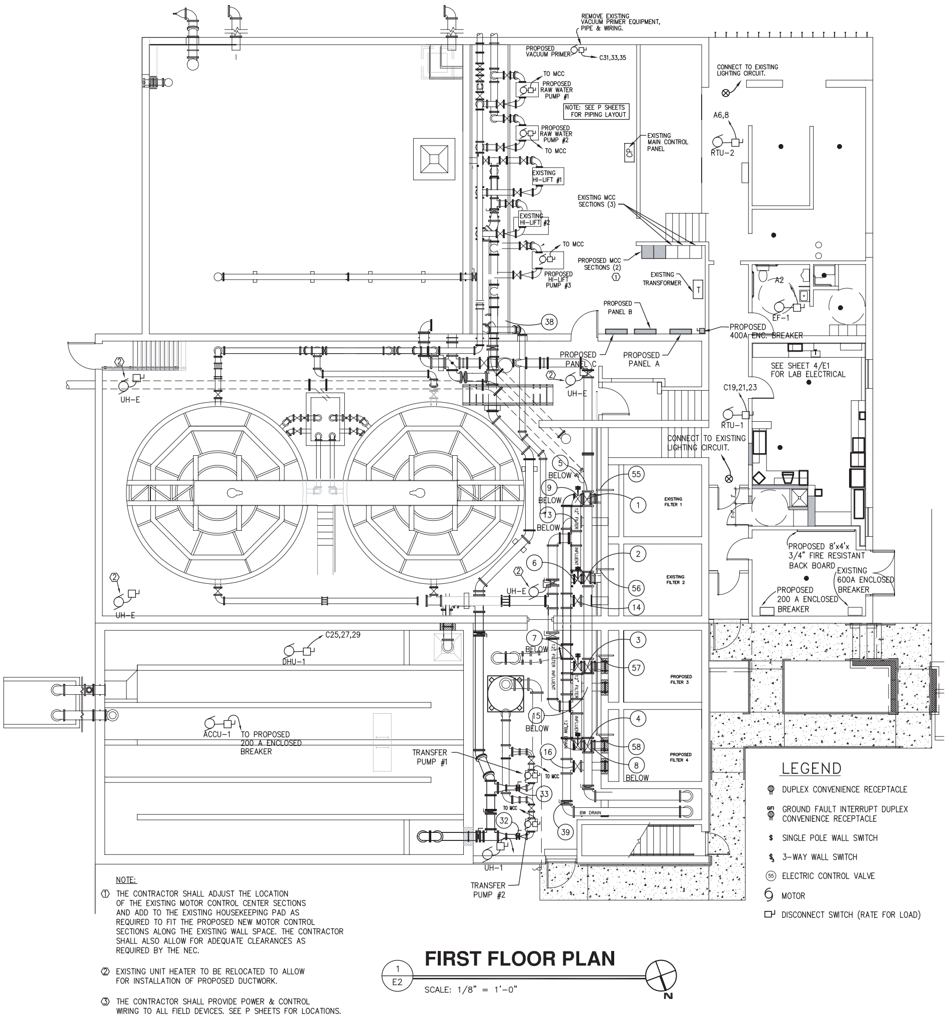
FIGURE 14.37. Electrical plan for the first floor of the Baraga water treatment facility.



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ground fault interrupt duplex convenience receptacles, single pole wall switches, three-way wall switches, electric control valves, motors, and disconnect switches. Notice that the exact locations of these devices are not shown—just approximations. Their locations are not critical to the integrity of the building, so the electrical subcontractor is free to put them wherever it makes the most sense in the field.

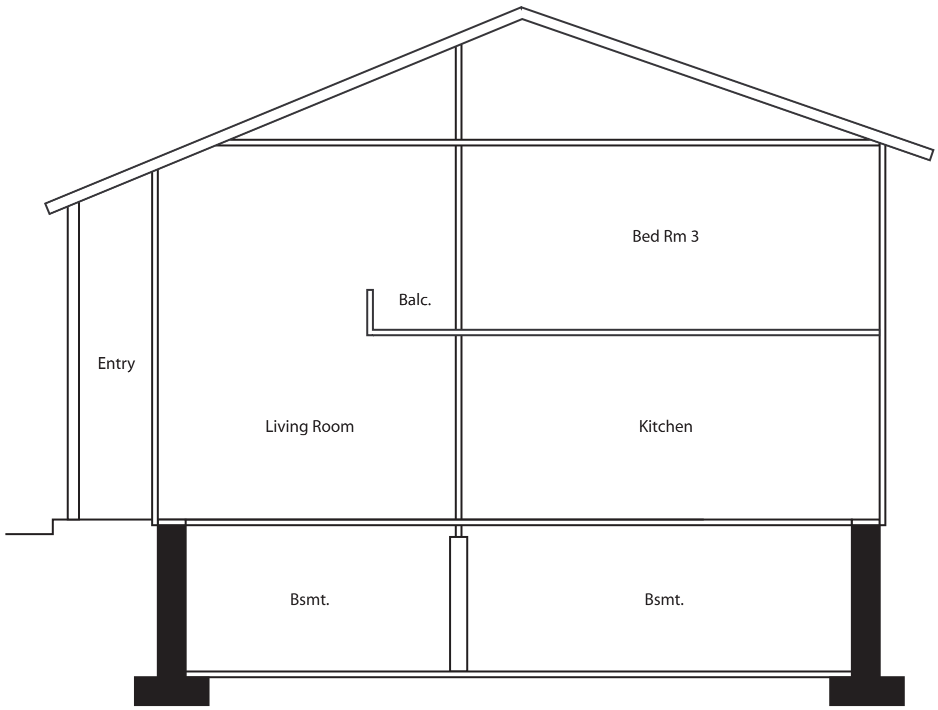
14.08.05 Sections

Sections in construction drawings can be organized into two types: **general sections**, which show room or floor layouts for buildings, or **detail sections**, which show cross sections with enough detail for construction purposes. In fact, a floor plan also can be thought of as a horizontal section through a building. Figure 14.38 shows a vertical section through a house. Note that with this type of general section, not enough detail is included for construction purposes; but the detail that is given is helpful because it provides a general idea about the layout of the rooms and the floors within the house.

Figure 14.39 shows a general section for the Baraga water treatment facility upgrade. This section shows the general building layout, clearly indicating where the pipe gallery is in relation to the hall above it. Notice that not much detail and very few dimensions are provided in this section; however, the drawing is helpful in understanding the overall design of the facility.

Detail sections provide a great deal of information. They show how the different components in a building system fit together, and they provide information that cannot be shown in large-scale drawings, such as floor plans or elevation views. **Wall sections** are among the most prevalent types of detail sectional drawings in a set of construction plans, although roof framing and foundation sections are also common. Refer to the foundation plan shown for the Baraga project in Figure 14.35. On the wall of the foundation plan toward the top of the drawing, a cutting plane line is shown with an arrow pointing toward the right side of the page. The label for this section line is given as 2/S2. This means that the cross section indicated by the cutting plane line is drawing number 2 found on section sheet 2. Figure 14.40 shows the

FIGURE 14.38. General section through a house showing room layouts.



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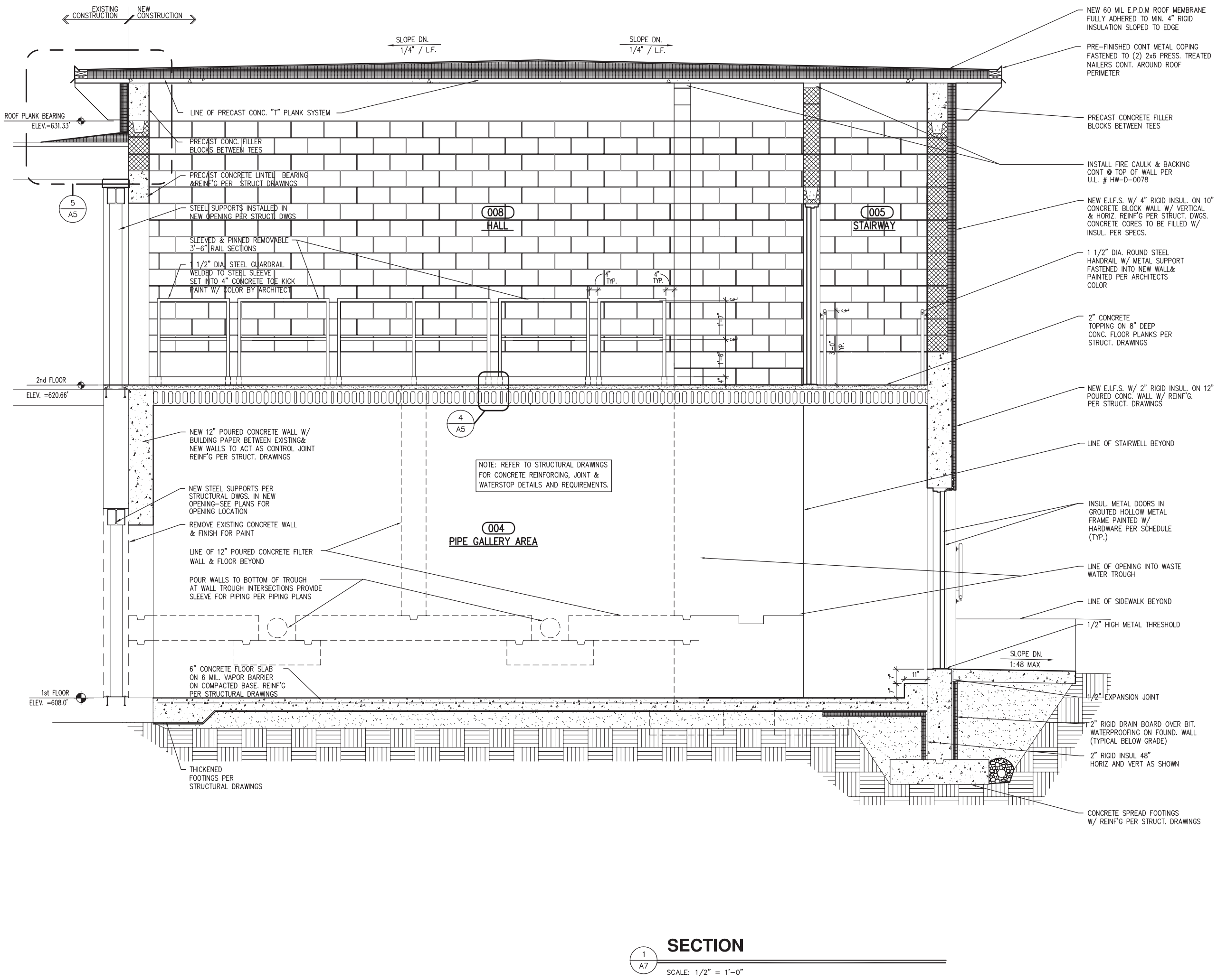

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FIGURE 14.39. General section for Baraga water treatment facility.

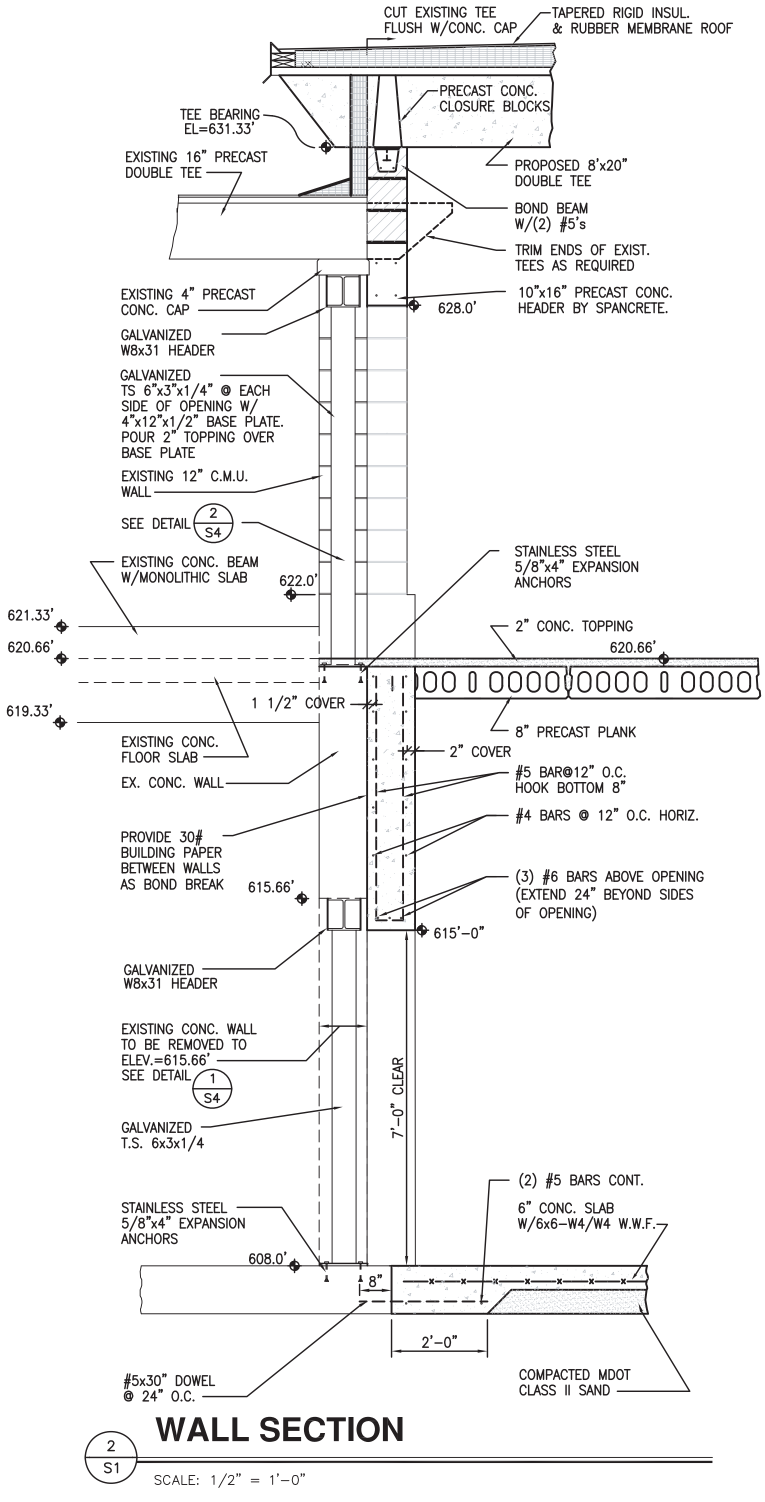
wall section that corresponds to that cutting plane line. You should note that unlike manufacturing section views, this section did not project orthographically (it was even on another sheet); further, the section view is not drawn to the same scale as the top view from which it is projecting. The plan view is drawn at a scale of ¼" = 1'-0"; the wall section is drawn at a scale of ½" = 1'-0".

The wall section shown in Figure 14.40 includes details about the reinforcing (welded wire fabric, dowels, stainless steel expansion anchors, and #6 bars); it includes information about elevations of various portions of the wall (the top of the foundation slab is at 608.0', the top of the slab for the first floor is at 620.66", and the bottom of the existing precast concrete cap is at 628.0'); it also shows the various components that extend out from the walls (the slabs, the double tee, the existing structure, and the roof).

14.08.06 Detail Construction Drawings

Detail drawings are made to show one or two particular features on the constructed facility so that it can be built. Because constructed facilities are typically large, some of the finer details of the construction cannot be shown adequately on other types of drawings. Thus, detail views show one specific area on a drawing that has been

FIGURE 14.40. Detailed wall section for Baraga water treatment facility.



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enlarged. Detail views are referenced from existing drawings, and all detail drawings may be shown on one sheet. Figure 14.41 shows the door frame detail for the Baraga project. In the upper-right corner of the detail drawing for the door frame, two new details are referenced—3/S4 and 4/S4, which are also shown in the figure. Detail 3/S4 shows how the cap plate will be bolted to the crossbeam from a profile viewpoint, and Detail 4/S4 shows the connection from a plan viewpoint. You should note that the

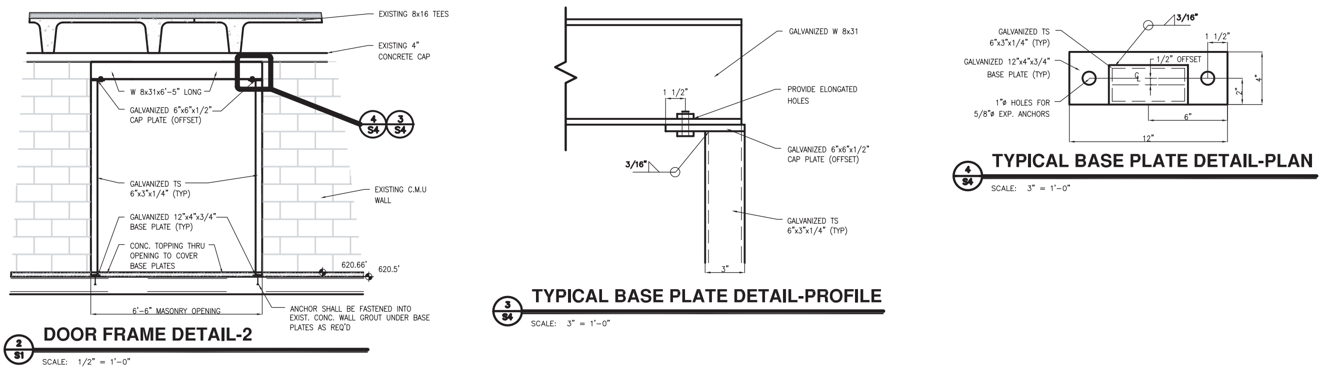

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FIGURE 14.41. Detail drawings showing how a door will be installed in the Baraga water treatment facility.

plan and profile views of the detail do not project orthographically on the sheet. You must mentally line these drawings up in order to understand how the door frame is to be constructed.

14.08.07 Plan and Profile Drawings

Another common type of construction drawing is a **plan and profile drawing**. Recall that plan views show a structure from above and that profile views show the structure from the side or front. In other words, plan views show changes in bearings of lines and profile views show changes in elevations of features. Figure 14.42 shows a plan and

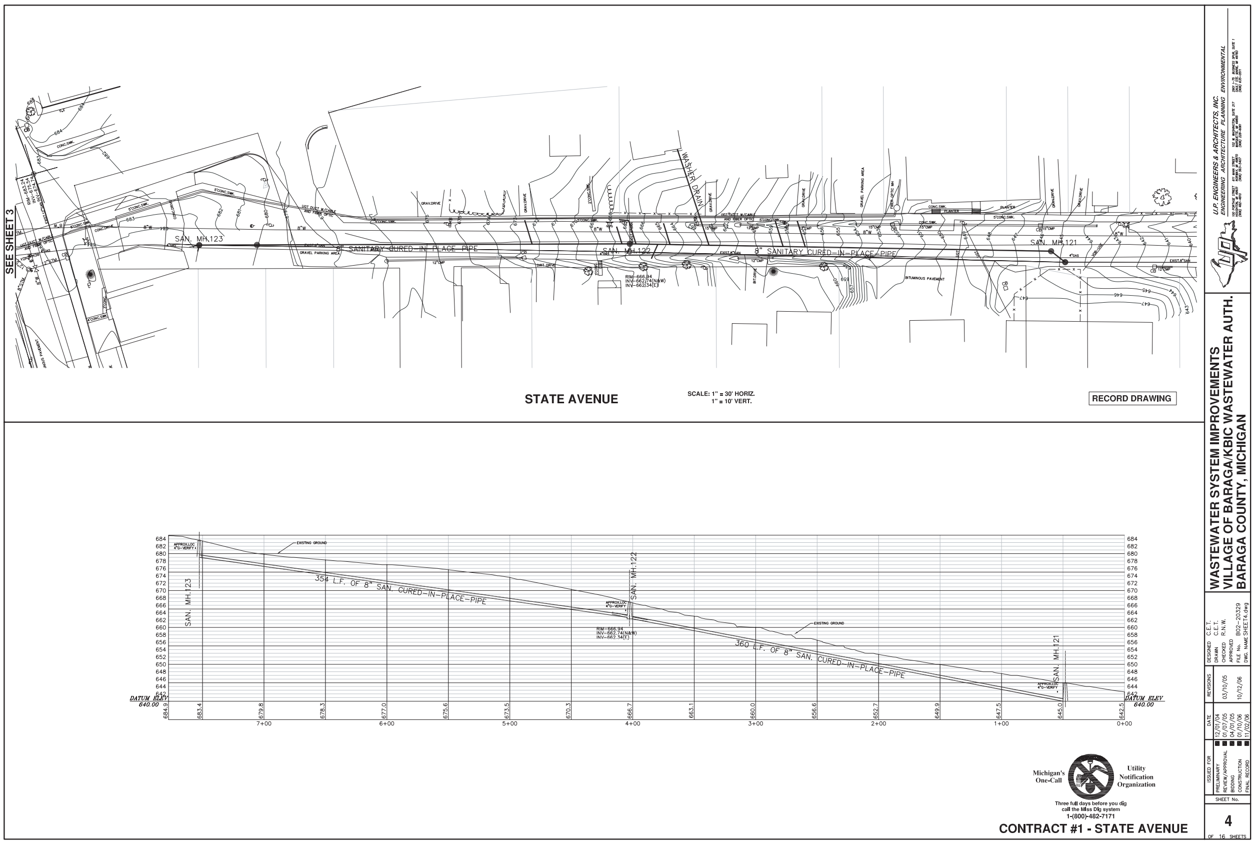

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FIGURE 14.42. Plan and profile drawing showing the street and corresponding sewage pipe for wastewater system in Baraga.

profile drawing for a street and a corresponding sewage pipe for a wastewater system in Baraga. Note that in this drawing, things do project orthographically between the plan and the profile views; however, the drawing scales are different between views. For the plan view, the entire view is drawn at a scale of 1" = 30'. For the profile view, the scale is 1" = 30' on the horizontal dimension and 1" = 10' on the vertical dimension. The reason for this change in scale is to show the changes in elevation in greater detail.

14.09 Engineering Scales

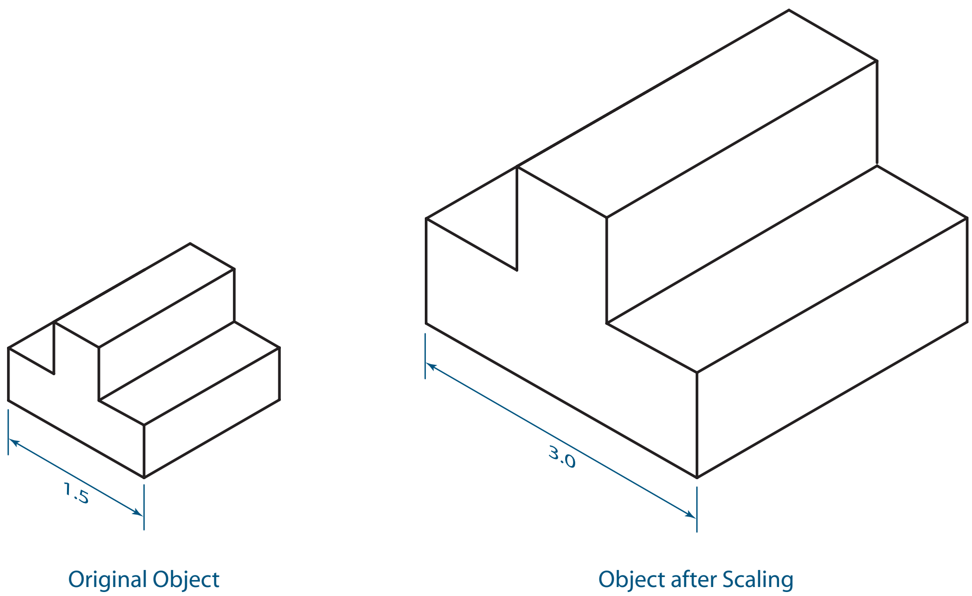
As you learned previously, virtually all engineering drawings are made to scale. This is especially true with construction drawings that represent large projects that do not fit on a single sheet of paper. In this age of CAD, engineering designs can be printed to any scale desired. Further, if you decide that you need the drawing at a scale that differs from the one you first chose, you merely adjust the font sizes and reprint the drawing at the new scale. Previously, if you needed the drawing at a new scale, you were required to re-create the entire drawing from the beginning—a tedious task.

It is important to understand that when an object is drawn to scale, its actual size does not change—just its appearance on the paper. Figure 14.43 illustrates the concept of drawing something to scale. Here the objects are the same size; the one on the right just appears to be twice as large as the one on the left. The notation of a 2:1 scale means that 2 inches on paper equals 1 inch on the object. Therefore, the drawing looks larger than the actual object. Conversely, a 1:2 scale would mean that 1 inch on paper represents 2 inches on the physical object; hence, the drawing would look smaller than the actual object.

Another way to think of drawing objects to scale is that the scale indicates how close you are to the object. If you are a substantial distance from the object, it appears very small; whereas if you are very close to the object, it appears large. However, the true object size does not change. Similarly, from an airplane, a house on the ground looks tiny; but if you are standing a few inches from the same house, it appears enormous. It is your perception of the house that changes, not the size of the house.

Scales for drawings are usually reported as ratios. In denoting scales, the first number in the ratio corresponds to the drawing and the second number corresponds to the physical object. However, sometimes drawing scales are denoted with an equal

FIGURE 14.43. Scaled drawing of object.



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sign rather than a ratio. This is particularly true when scales are given in the English system of units. Thus, a scale may be reported as 1" = 50' or ¼" = 1'-0". The first scale (1" = 50') means that 1 inch on the drawing corresponds to 50 feet on the actual object. These drawing scales can also relate back to their ratio equivalents, i.e., 1" = 50' corresponds to a scale of 1:600 (there are 12 inches in a foot, so 50 feet = 600 inches, resulting in a 1:600 ratio), and a scale of ¼" = 1'-0" corresponds to 1:48.

A scale is a device that was developed over the years to aid in making a drawing to scale; it is usually a triangular prism with six to 12 different drawing scales depicted on one piece of equipment. Figure 14.44 shows three common scales used by engineers—an Engineer's scale, an Architect's scale, and a Metric scale. With modern-day mechanical CAD systems, you typically create a 3-D object in true size and print it out to the scale you need. In civil engineering applications, you create your drawings full-size and print them out to scale. In either case, you must ensure that the text on the drawings is legible when printed to the desired scale.

Due to advances in computer software, physical scales such as those shown in Figure 14.44 are, for the most part, a relic of the past. However, in a few instances, knowledge of scales is helpful—and possibly necessary. Because construction projects are built as one-of-a-kind structures, there may be times when dimensions are inadvertently left off drawings; and despite the diligence of the engineers who check the designs, the lack of a dimension is not known until the project is under construction. In this case, scales can be used in the field to quickly determine the dimension and construction can continue. Since tolerances in civil engineering projects are typically large (or nonexistent), reading a dimension using a scale is often "close enough." Further, in civil engineering projects, scales can be used in the field to measure a dimension on a given sheet rather than looking through a large set of drawings to find the specific sheet where the dimension is "officially" located. In manufacturing applications, scales can be used to quickly estimate dimensions as needed.

In determining a dimension from a drawing that has been drawn to scale, many novices use a calculator. For example, if a drawing has been made at a scale of 1" = 40' and you measured a line that is ⅝" long, you could calculate that the line represents 25' on the actual object (⅝ of 40 is 25). Using a calculator to figure out dimension is extremely tedious and would likely result in error. Fortunately, this tedium can be avoided with the use of an appropriate scale. In the following sections, you will learn about the three primary types of scales used in engineering.

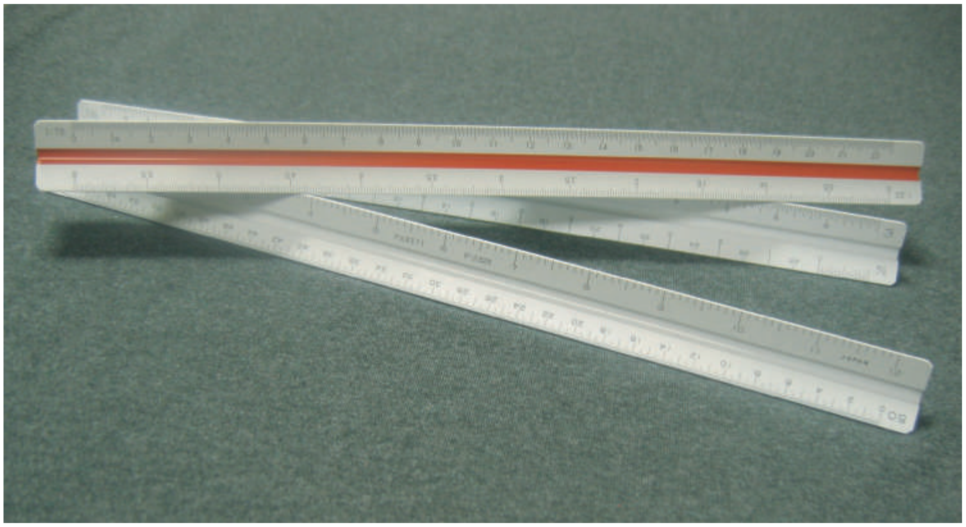

FIGURE 14.44. Three common scales used in creating or measuring dimension on engineering drawings.

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14.09.01 Engineer's Scale

An **Engineer's scale** (sometimes called a Civil Engineer's scale) usually consists of 10, 20, 30, 40, 50, and 60 scales. These scales are based on the English system of units, with the inch as the basis for measurement. The divisions on the scales are in increments of tenths of an inch, not eighths of an inch as on ordinary rulers. Engineer's scales can be used to measure a line in any multiple of ten of the basic unit. For example, the 30 scale can be used for reading the following scales from drawings: 1" = 3', 1" = 300', or 1" = 30 mi. Similarly, the 50 scale can be used for drawings with scales of 1" = 50', 1" = 5 yds, or 1" = 500 mi.

Figure 14.45 shows a line being measured with a 20 scale. The actual length of the line on paper is 3.5"; but since it is drawn at a scale of 1" = 200', this line represents a length of 700' on the actual object. By reading the scale in this figure, what is the length of the actual line to point A? (The actual line to point A is 580' long.)

Figure 14.46 shows lines drawn at a scale of 1" = 40' and a corresponding 40 Engineer's scale. What is the length of each line segment (OA, OB, OC)? These values can be read directly from the scale, making proper adjustments for decimal places. The line is drawn at a scale of 1" = 40'; therefore, the first 2 on the scale represents 20', the 4 represents 40', the 6 represents 60', etc. The unlabeled long tick marks represent 10', 30', 50', etc. The intermediate-length tick marks occur at 5' intervals, and the smaller tick marks represent 1' intervals. Therefore, the length of line OA on this scale can be read as 67'. The line OB is read as 103'. Many times, novice scale readers will incorrectly interpret the length of OB as 130'. When the scale is read, care should be taken to put the decimal in the correct place. The distance OB is read directly as 10.3; but since the scale is 1" = 40', the decimal is moved to the right one unit and the correct scale reading becomes 103'. The length of the line OC can be read from the scale as 132'.

In Figure 14.46, what are the lengths of the lines if they are drawn at a scale of 1" = 4000'? Note that you still use the same Engineer's scale (the 40 scale) to make this reading but you add more zeros to the number that you read on the scale. Thus, the line OA has a length of 6700', OB has a length of 10,300', and OC has a length of 13,400' at a scale of 1" = 4000'. What if the lines are drawn at a scale of 1" = 4 yds? At

FIGURE 14.45. Line to be measured at a scale of 1"=200' and a 20 Engineer's scale.

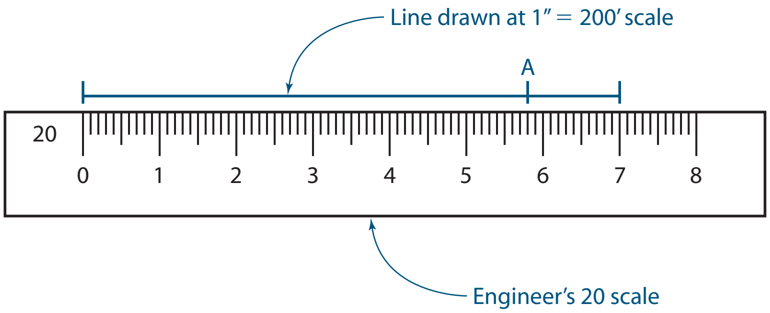
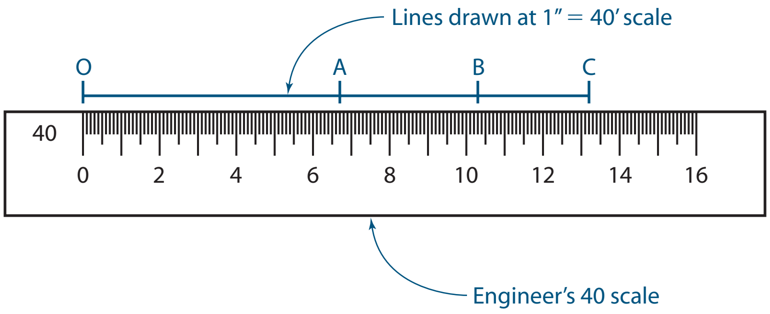


FIGURE 14.46. Lines drawn at a scale of 1"=40' and a 40 Engineer's scale.



that scale, the line OA has a length of 6.7 yds, OB has a length of 10.3 yds, and OC has a length of 13.4 yds.

14.09.02 Metric Scale

Metric scales are similar to Engineer's scales except that they are based on the metric system of units. Because metric units are based on decimals, unlike the English system of units (12" per foot, 3' per yard, 1,760 yds per mile, etc.), Metric scales are reported as ratios. Thus, typical Metric scales are reported as 1:1, 1:2, 1:5, and 1:10, for example. The same principle used to measure distances with an Engineer's scale is used for Metric scales. Like an Engineer's scale, the Metric scales can be used for multiples of ten of the basic unit. Thus, a 1:5 scale can also be used to measure 1:50, 1:500, and 1:5000 scales. Figure 14.47 shows a line drawn at a 1:2000 scale and a 1:20 Metric scale. On a Metric scale, the numbers (0.5, 1.0, 1.5, etc.) generally represent meters; and you adjust the decimal according to the specific scale at which you are measuring. If you read the scale directly, the length of the line is 1.84 m. But since the scale depicts a drawing scale of 1:20 and the line was drawn at a scale of 1:2000, you must move the decimal two units to the right to account for the difference. Thus, the length of the line is equal to 184 m. Similarly, if the line was drawn at a 1:200 scale, the length of the line would be 18.4 m.

Figure 14.48 shows a Metric scale of 1:100 and a set of lines drawn at a 1:1 scale. What are the lengths of each line? If you read the length of OA directly from the scale, you see that it is 7.7 m. Since the scale is 1:100 and the line is drawn at 1:1, you move the decimal place two units to the left; therefore, the length of the line is 0.077 m. Alternatively, you could report the length of the line as 7.7 cm or 77 mm. What is the length of OB? The value read from the scale is 10.7 m. If you move the decimal place two units to the left, the length is determined as 0.107 m (or 10.7 cm or 107 mm). Similarly, the length of line OC is 0.124 m (12.4 cm).

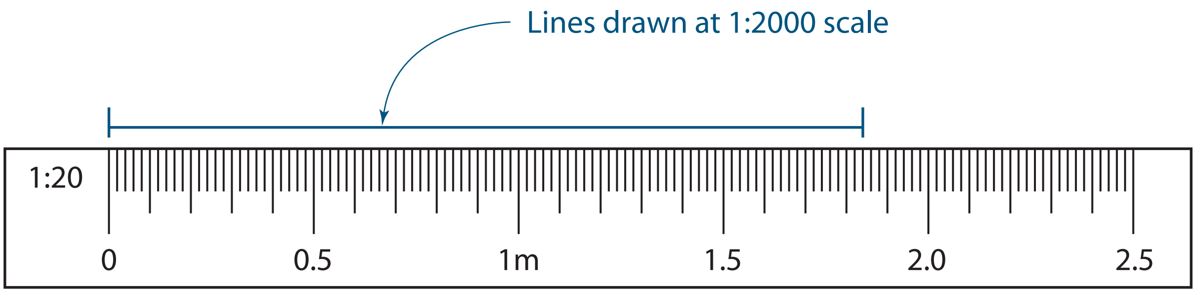


FIGURE 14.47. Line drawn at a scale of 1:2000 and a 1:20 Metric scale.

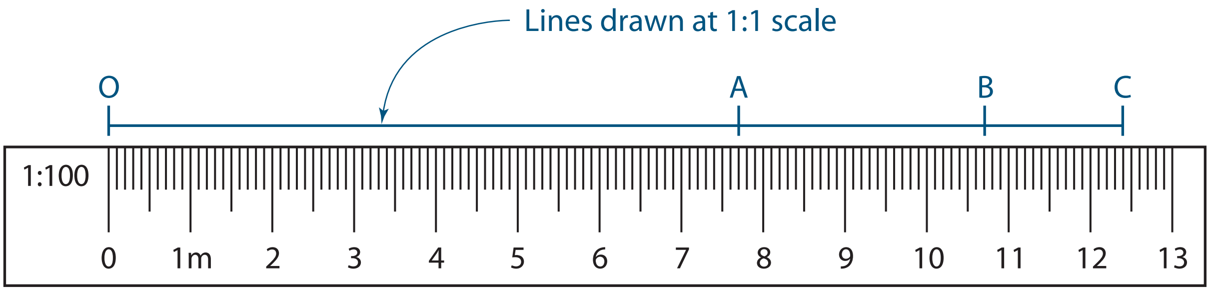


FIGURE 14.48. Lines drawn at 1:1 scale and a 1:100 Metric scale.

14.09.03 Architect's Scale

An **Architect's scale** is similar to an Engineer's scale in that it is based on the English system of units. One of the differences between the two scales is that the Architect's scale is based on fractions of an inch. (Recall that the Engineer's scale is based on tenths.) Another significant difference is that with an Architect's scale, drawing scales are always reported as something = 1'-0". Thus, a scale might be reported as ¼" = 1'-0" or as ⅜" = 1'-0". Some of the more common scales depicted on an Architect's scale are as follows:

12" = 1'0" (full size)	6" = 1'-0" (half size)	3" = 1'-0" (quarter size)
1½" = 1'-0" (⅛ size)	1" = 1'-0" (1/12 size)	¾" = 1'-0" (1/16 size)
½" = 1'-0" (1/24 size)	⅜" = 1'-0" (1/32 size)	¼" = 1'-0" (1/48 size)
3/16" = 1'-0" (1/64 size)	⅛" = 1'-0" (1/96 size)	3/32" = 1'-0" (1/128 size)

Architect's scales usually look significantly different than Engineer's or Metric scales. The biggest difference is that each edge of the Architect's scales typically depicts two scales—one reading from left to right and the other reading from right to left. Thus, twice as many scales (12 versus six) are depicted on an Architect's scale when compared to an Engineer's or Metric scale. The other difference is that fractional gradations are shown only at the ends of the scale. Thus, when you are measuring a distance with an Architect's scale, you must place one end of the line at the nearest whole number foot on the scale and read the fractional foot at the end with the gradation. Figure 14.49 shows an Architect's ¼ scale (this means the ¼" = 1'-0") and line to be measured with this scale. In this figure, the ¼ scale is read from right to left on the scale and the ⅛ scale is read from left to right. To read the length of this line, you place one end of the line on the nearest even foot mark of the scale. In this case, it is 11'. (Remember that you are reading from right to left for the ¼ scale.) Notice that the closest foot mark is not labeled for you. For the ¼ scale, the even foot markers are labeled but the odd ones are not. The smaller tick marks for this scale represent one-half foot divisions. Be careful not to line up the end of the line with the half-foot marks instead of the foot markers. The fractional feet are shown in the last foot of the scale (past the 0). This last foot is divided into 12 gradations, so each tick mark on the scale represents 1" because there are 12 inches in a foot. Thus, the length of this line is 11'-7".

What happens if you read the length of this line using the ⅛ scale? Figure 14.50 shows the same line being measured with the ⅛ scale. The same procedure is followed to read the length of the line at this scale. In this case, the nearest even foot mark is 23'. Notice that on this scale, only every fourth (4, 8, 12, etc.) foot marker is labeled and that the long tick marks from left to right represent the odd foot marks. The short tick marks from left to right represent the intermediate even foot markers. The final foot of this scale is divided into six increments compared to 12 for the ¼ scale. Thus, each division in the final foot represents 2". As shown in this figure, the length of the line at

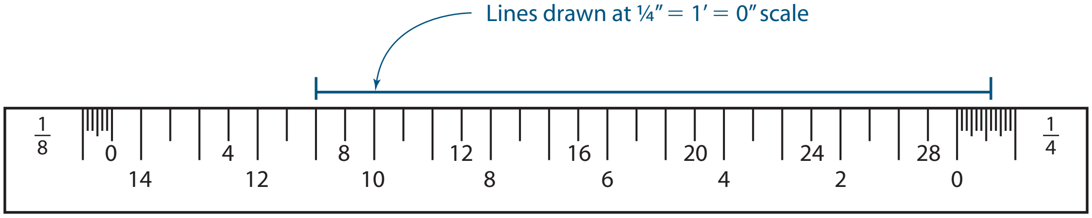


FIGURE 14.49. Line drawn at a scale of ¼" = 1'-0" and a ¼" Architect's scale.

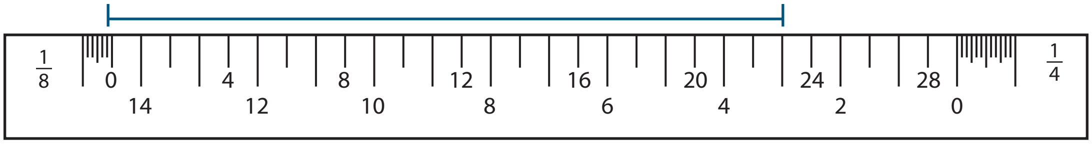


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FIGURE 14.50. Line drawn at a scale of ⅛"=1'-0" and a ⅛" Architect's scale.

a ⅛ scale is 23'-2". (Note that this makes sense because 2 × 11'-7" = 22'-14", or 23'-2"). The Architect's scale usually requires a great deal of practice on your part to be able to read it with confidence.

As you may have realized from the previous discussion of drawing scales (Engineer's, Architect's, and Metric), these devices are time-saving and relatively easy to use. Unfortunately, many students have a tendency to use calculators when working with scaled drawings for the first time. It is important that you learn to use a scale for working with engineering drawings.

14.10 Considerations for 3-D Modeling

The development of 3-D computer modeling has greatly reduced the time and effort required to produce a manufacturing drawing; however, as stated previously, 3-D modeling is not predominant in the construction industry and is not likely to be so for several years. In the following paragraphs, the considerations for 3-D modeling are given for projects in the realm of manufacturing.

There are provisions on most 3-D modeling software to easily generate different views of an object or assembly, including multiviews, section and detail views, auxiliary views, and pictorials. Dimensions can also be made to appear on any of the views. Because of the ease with which these graphics are generated, there is no excuse not to include as many different views as needed to communicate the geometry of the part or device. Formal drawing headers also are easily created, usually recalled from a library of premade headers with different complexities as demanded by the application of the device to be made.

Three-dimensional modeling software, however, cannot fully interpret the functional requirements of the device that is to be built. That knowledge resides with the designer or engineer. For example, depending on the functional requirements of a part, only the engineer or designer can know what dimension tolerances are acceptable for the part to work properly. Since most numerically controlled machine tools (described in the chapter on fabrication processes) can easily hold tolerances to within 25 microns for most small parts, it is sometimes tempting to use this number as a default tolerance for every dimension. However, if tighter tolerances are required by the engineer, special manufacturing processes may be required. An example is in the computer disk drive spindle detailed in Figure 14.27; it shows that tolerances in the range of 5 microns are required for some dimensions. Only the engineer or designer can know those requirements. The software can present only what the engineer or designer wants.

14.11 Chapter Summary

With any type of engineering drawing, the key word is *communication*. The drawing must be able to communicate to the reader the desires of the engineer or designer for a part or assembly. When working drawings are created, this communication must occur even in the absence of the drawing's originator. Use of the formal drawing format—with proper views, sheet sizes, headers, and drawing organization—serves to maximize the probability that the device will be built correctly. Almost as important is the fact that the quality of a working drawing is a reflection of the person, company, or organization that produced it. A high-quality and professional presentation must be maintained.

A formal working drawing succeeds not because it looks good, but because it clearly and unambiguously tells a fabricator or contractor how to produce a desired product. What you have designed may well be beautiful once it is produced, but that is irrelevant initially. The only goal at the start is to produce a working drawing that allows your conception to be made into reality.

14.12 GLOSSARY OF KEY TERMS

approval signatures: The dated signatures or initials of the people responsible for certain aspects of a formal drawing, such as the people who did the drafting or the engineer responsible for the function of the part.

Architect's scale: A device used to measure or draw lines in the English system of units with a base unit of inches and fractions of an inch.

as-built plans: Drawings that show exactly how buildings were constructed, especially when variations exist between the final building and the plans created during the design phase.

assembly: A collection of parts and/or subassemblies that have been put together to make a device or structure that performs a specific function.

assembly dimensions: Dimensions that show where parts must be placed relative to other parts when the device is being put together.

balloons: Closed geometric shapes, usually circles, containing identification numbers and placed beside parts on a layout or assembly drawing to help identify those parts.

bearing: The angle that a line makes with a North-South line as seen in a plan view.

benchmarks: Points established by the U.S. Geological Survey that can be used to accurately locate control points on a construction site.

bill of materials (BOM): A drawing or table in a drawing that lists all of the parts needed to build a device by (at least) the part number, part name, type of material used, and number of times the part is used in the device.

blueprints: The name sometimes given to construction drawings based on historical blue-on-white drawings that were produced from ink drawings.

border: A thick line that defines the perimeter of a drawing.

construction drawings: Working drawings, often created by civil engineers, that are used to build large-scale, one-of-a-kind structures.

continuation blocks: Header blocks used on the second and subsequent pages of multipage drawings.

control points: Points at a construction site that are referenced to an origin by north, south, east, or west coordinates.

cover sheet: The first page in a set of construction drawings showing a map of the location of the project and possibly an index.

default tolerances: Usually appearing in the drawing header, the tolerances to be assumed for any dimension show on a part when that dimension does not specify any tolerances.

detail drawing: A formal drawing that shows the geometry, dimensions, tolerances, materials, and any processes needed to fabricate a part.

detail sections: Drawings included in a set of construction plans that show how the various components are assembled.

EC Level: A number included in the title block of a drawing indicating that the part has undergone a revision.

electrical plan: A plan view showing the layout of electrical devices on a floor in a building.

elevation views: Views of a structure that show changes in elevation (side or front views).

Engineer's scale: A device used to measure or draw lines in the English system of units with a base unit of inches and tenths of an inch.

engineering change (EC) number: A dated number that defines the degree to which the specifications of a part have been updated.

exploded assembly drawing: A formal drawing, usually in pictorial form, that shows the orientation and sequence in which parts are put together to make a device.

floor plan: A plan view of a single floor in a building that shows the layout of the rooms.

foundation plan: A plan view of the foundation of a building showing footings and other support structures.

general sections: Sections through entire structures that show the layout of rooms but provide little detail.

header: A premade outline on which working drawings are created to ensure that all information required for fabrication and record keeping is entered.

heating and ventilation plan: A plan view of the ventilation systems on a specific floor of a building, including ductwork and devices such as air conditioning units.

index: A list of all sheets of drawings contained in a set of construction plans.

item number: A number used to identify a part on a layout or assembly drawing.

international sheet sizes: The internationally accepted paper dimensions used when drawings are created or printed to their full intended size.

landscape: The drawing orientation in which the horizontal size is larger than the vertical size.

layout drawing: A formal drawing that shows a device in its assembled state with all of its parts identified.

location grid: An imaginary alphanumeric grid, similar to that of a street map, on a drawing that is used to specify area locations on the drawing.

main assembly: A completed device usually composed of multiple smaller parts and/or subassemblies.

main title block: A bordered area of a drawing (and part of the drawing header) that contains important information about the identification, fabrication, history, and ownership of the item shown on the drawing.

manufacturing drawings: Working drawings, often created by mechanical engineers, that are used to mass-produce products for consumers.

Metric scale: A device used to measure or draw lines in the metric system of units with drawings scales reported as ratios.

notes: Additional information or instructions placed on a drawing that are not contained on the dimensions, tolerances, or header.

one-off: A one-of-a-kind engineering project for which no physical prototypes are created.

outline assembly drawing: *See* layout drawing.

part: An object expected to be delivered from a fabricator as a single unit with only its external dimensions and functional requirements specified.

part name: A very short descriptive title given to a part, subassembly, or device.

part number: Within a company, a string of alphanumeric characters used to identify a part, a subassembly, an assembly, or a device.

parts list: *See* bill of materials.

plan and profile drawings: Construction drawings typically used for roads or other linear entities that show the road from above as well as from the side, with the profile view usually drawn with an exaggerated vertical scale.

plan views: Drawings created from a viewpoint above the structure (top view).

portrait: The drawing orientation in which the vertical size is larger than the horizontal size.

professional engineer (PE): An individual who has received an engineering degree, who has worked under the supervision of a PE for a number of years, and who has passed two examinations certifying knowledge of engineering practice.

profile views: Views of a structure that show horizontal surfaces in edge view (side or front views).

quantity per machine (Q/M): The number of times a part is required to build its next highest assembly.

rebars: Steel bars added to concrete for reinforcement or for temperature control.

reference dimensions: Unneeded dimensions shown for the convenience of the reader used to show overall dimensions that could be extracted from other dimensions on the part or from other drawings.

schedule of materials: A list of the materials, such as doors and windows, necessary for a construction project.

secondary title block: An additional bordered area of a drawing (and part of the drawing header) that contains important information about the identification, fabrication, and history of the item shown on the drawing.

sectioned assembly drawing: A formal drawing, usually in pictorial form, that shows the device in its assembled form but with sections removed from obscuring parts to reveal formerly hidden parts.

set of construction plans: A collection of drawings, not necessarily all of them plan views, needed to construct a building or infrastructure project.

site plan: A plan view showing the construction site for an infrastructure project.

site survey: Data regarding the existing topography and structures gathered during the preliminary design stages by trained surveying crews.

specifications (specs): The written instructions that accompany a set of construction plans used to build an infrastructure project.

subassemblies: Collections of parts that have been put together for the purpose of installing the collections as single units into larger assemblies.

title block: Usually the main title block, which is a bordered area of the drawing (and part of the drawing header) that contains important information about the identification, fabrication, history, and ownership of the item shown on the drawing.

trail: Dashed lines on an assembly drawing that show how various parts or subassemblies are inserted to create a larger assembly.

U.S. sheet sizes: The accepted paper dimensions used in the United States when drawings are created or printed to their intended size.

wall sections: Sectional views of walls from foundation to roof for a construction project.

working drawings: A collection of all drawings needed to fabricate and put together a device or structure.

14.13 QUESTIONS FOR REVIEW

1. What is the purpose of a header on a formal engineering drawing?
2. What type of information is typically included on a drawing header?
3. What signatures (or initials) typically appear in a drawing header?
4. Why is it important that dates be included on a drawing?
5. Why is it important that part numbers be unique to each part?
6. When should the part number for a particular part be changed?
7. What considerations need to be made in the selection of a sheet size for a drawing?
8. What considerations need to be made in the selection of a scale for a drawing?
9. What are the three different types of assembly drawings? How do they differ?
10. What is a revision (or engineering change) to a drawing?
11. How is a subassembly different from a main assembly?
12. What sort of information is typically included in a bill of materials?
13. What types of dimensions are permitted on an assembly drawing?
14. List three ways that manufacturing drawings differ from construction drawings.
15. Construction projects are site-specific. What does that mean?
16. What is the bearing of a line? In which view is it seen?
17. What does the term *professional engineer* mean?
18. What are plan, profile, and elevation views?
19. What is meant by the term *one-off*?
20. For construction drawings, what is a general section?
21. What are the three types of scales used in engineering?
22. Which scales are based on the English system of units?

14.14 PROBLEMS

1. Assembled and exploded views for a staple remover are shown in Figure P14.1. Using reasonable materials and dimensions of your choice, create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a bill of materials, and all detailed part drawings. Specify appropriate tolerances for all dimensions. You may use metric dimensions or convert the metric dimensions to their nearest inch equivalents.

14.14 PROBLEMS (CONTINUED)

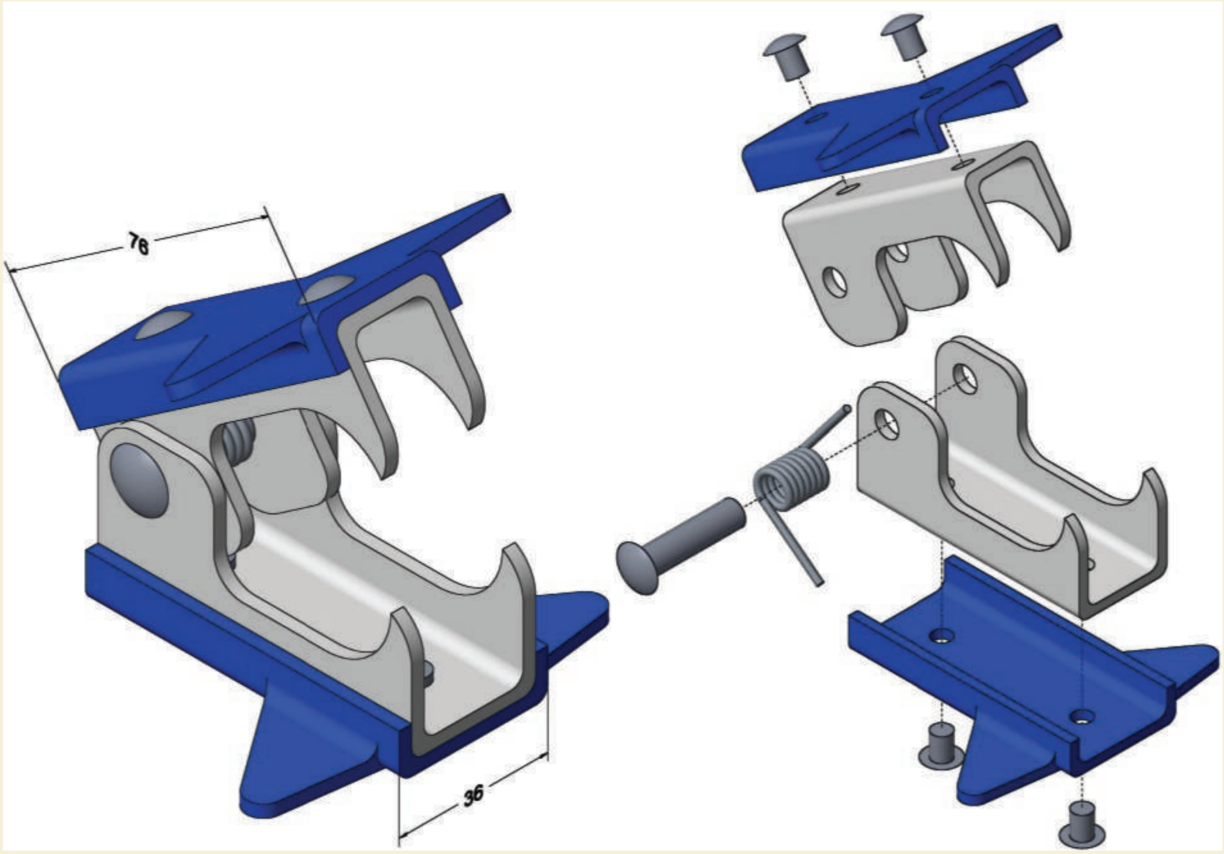

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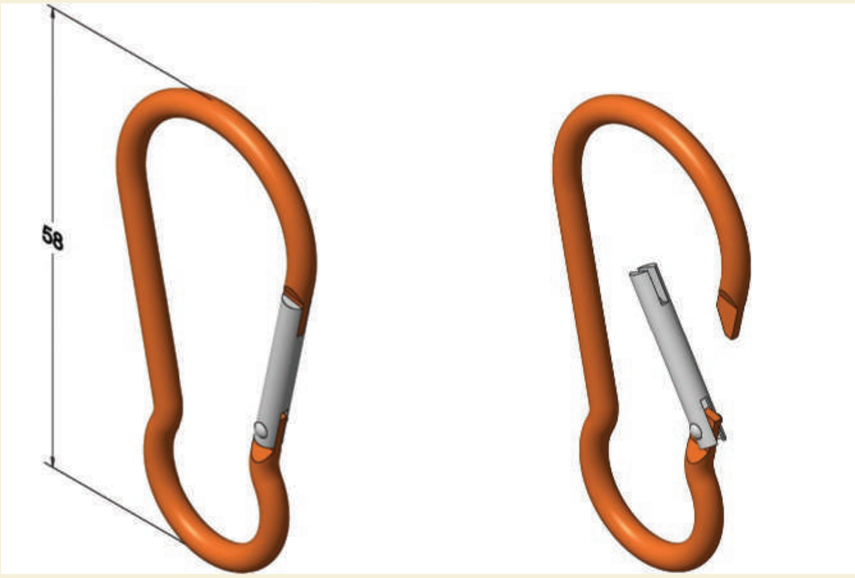
FIGURE P14.1.

2. Assembled and exploded views for a spring carabineer are shown in Figure P14.2. Using reasonable materials and dimensions of your choice, create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a bill of materials, and all detailed part drawings. Specify appropriate tolerances for all dimensions. You may use metric dimensions or convert the metric dimensions to their nearest inch equivalents.



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14.14 PROBLEMS (CONTINUED)

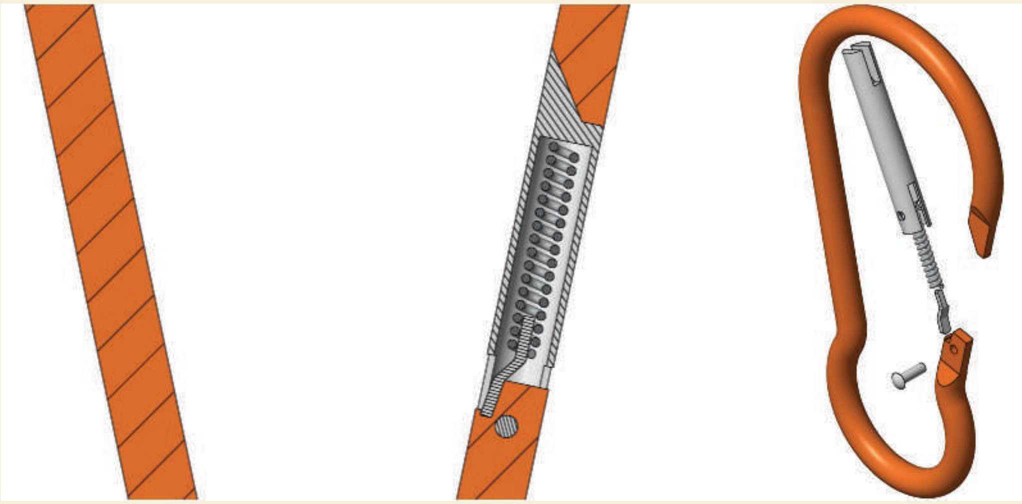

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FIGURE P14.2.

3. Assembled and exploded views for a universal joint are shown in Figure P14.3. Using reasonable materials and dimensions of your choice, create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a bill of materials, and all detailed part drawings. Specify appropriate tolerances for all dimensions. You may use metric dimensions or convert the metric dimensions to their nearest inch equivalents.

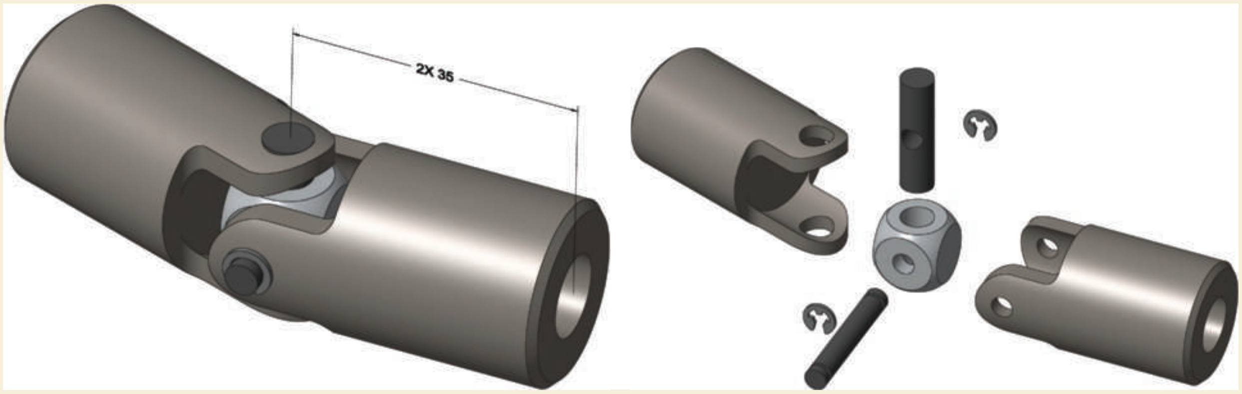

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FIGURE P14.3.

4. Assembled and exploded views for a gear puller are shown in Figure P14.4. Detail the design by specifying appropriate materials, dimensions, and tolerances for each part. Create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a bill of materials, and all detailed part drawings. You may use metric dimensions or convert the metric dimensions to their nearest inch equivalents.

14.14 PROBLEMS (CONTINUED)

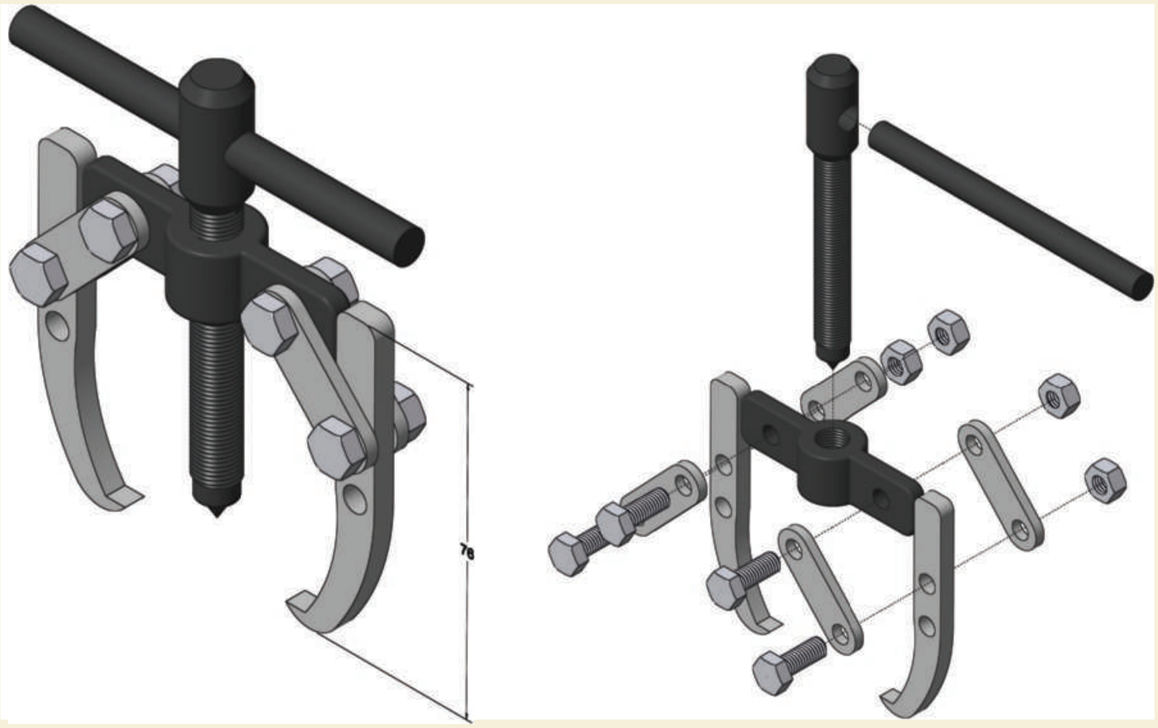
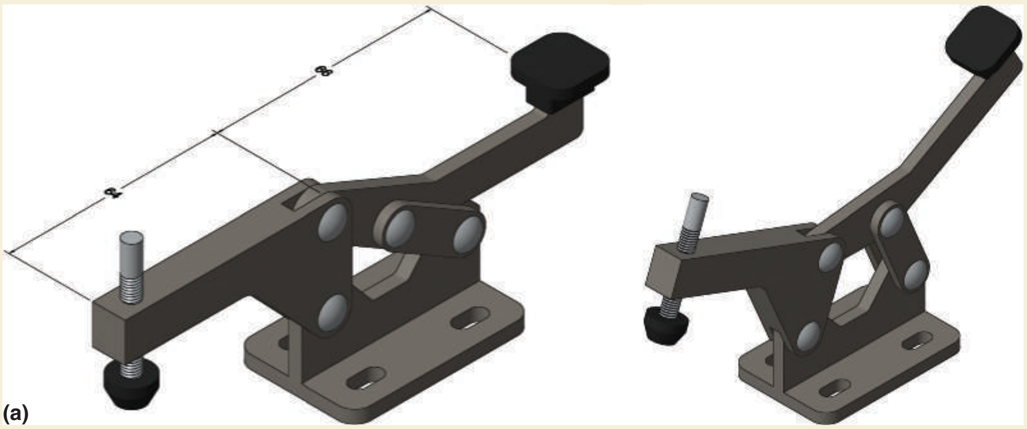


FIGURE P14.4.

5. Assembled and exploded views for a toggle clamp are shown in Figure P14.5. Detail the design by specifying appropriate materials, dimensions, and tolerances for each part. Create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a bill of materials, and all detailed part drawings. You may use metric dimensions or convert the metric dimensions to their nearest inch equivalents.



14.14 PROBLEMS (CONTINUED)

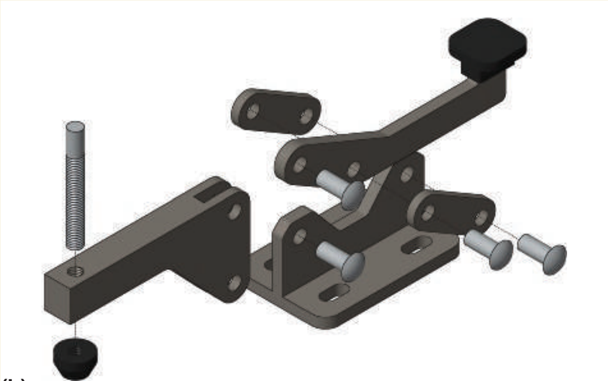

(b)

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FIGURE P14.5.

6. A conceptual model for a pen-type eraser is shown in whole and in cutaway view in Figure P14.6. Using reasonable materials and dimensions of your choice, expand the concept to create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a sectioned assembly drawing, a bill of materials, and all detailed part drawings. Specify appropriate tolerances for all dimensions.

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FIGURE P14.6.

14.14 PROBLEMS (CONTINUED)

7. A conceptual model for a garden hose nozzle is shown in whole, cutaway, and exploded views in Figure P14.7. Using reasonable materials and dimensions of your choice, expand the concept to create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a sectioned assembly drawing, a bill of materials, and all detailed part drawings. Specify appropriate tolerances for all dimensions.

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FIGURE P14.7.

8. A concept drawing for a flexible shaft coupling is shown in Figure P14.8. Using reasonable materials and dimensions of your choice, expand the concept to create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a sectioned assembly drawing, a bill of materials, and all detailed part drawings. Specify appropriate tolerances for all dimensions. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

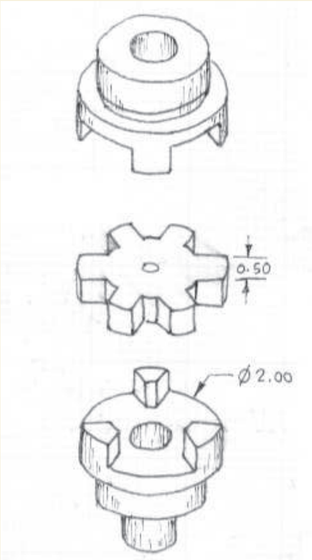

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FIGURE P14.8.

14.14 PROBLEMS (CONTINUED)

9. A concept drawing for a clock stand is shown in Figure P14.9. Using reasonable materials and dimensions of your choice, expand the concept to create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a sectioned assembly drawing, a bill of materials, and all detailed part drawings. Specify appropriate tolerances for all dimensions. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

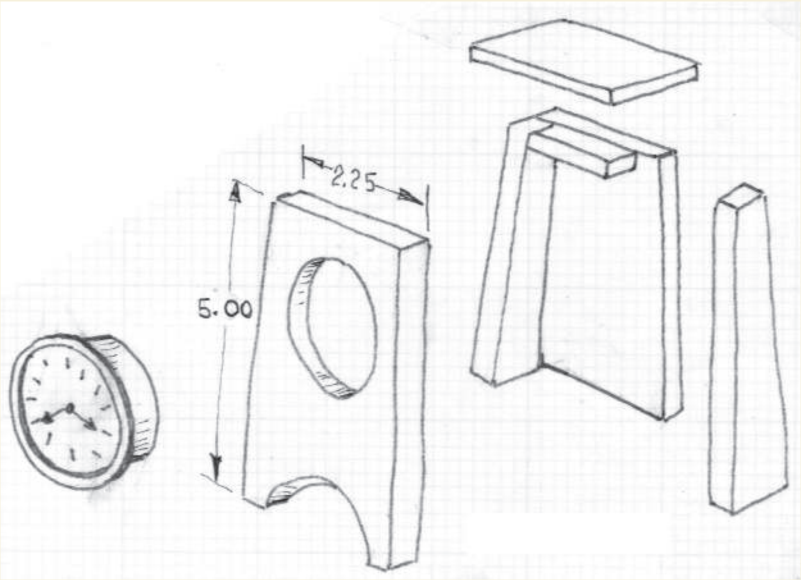

FIGURE P14.9.

10. A concept drawing for a work table is shown in Figure P14.10. Using reasonable materials and dimensions of your choice, expand the concept to create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a sectioned assembly drawing, a bill of materials, and all detailed part drawings. Specify appropriate tolerances for all dimensions. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

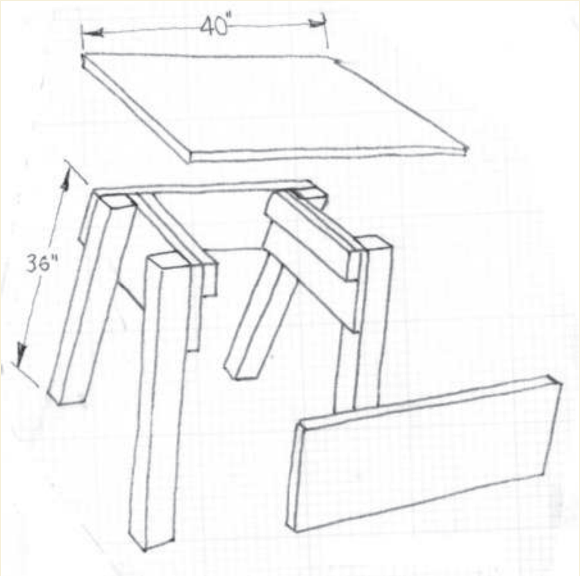

FIGURE P14.10.

14.14 PROBLEMS (CONTINUED)

11. A concept drawing for a utility clamp is shown in Figure P14.11. Using reasonable materials and dimensions of your choice, expand the concept to create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a sectioned assembly drawing, a bill of materials, and all detailed part drawings. Specify appropriate tolerances for all dimensions. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

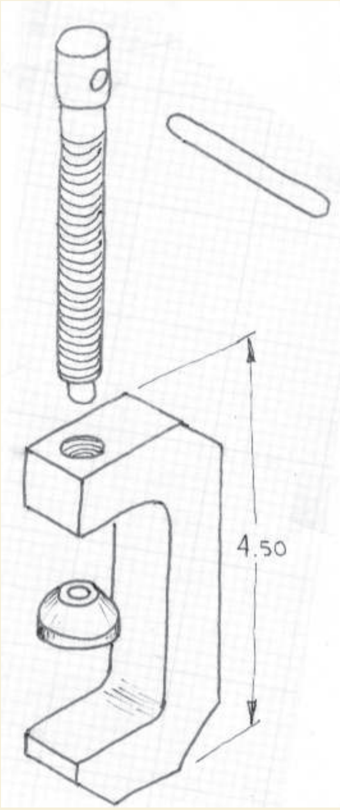

FIGURE P14.11.

12. A concept drawing for a stop guide is shown in Figure P14.12. Using reasonable materials and dimensions of your choice, expand the concept to create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a sectioned assembly drawing, a bill of materials, and all detailed part drawings. Specify appropriate tolerances for all dimensions. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

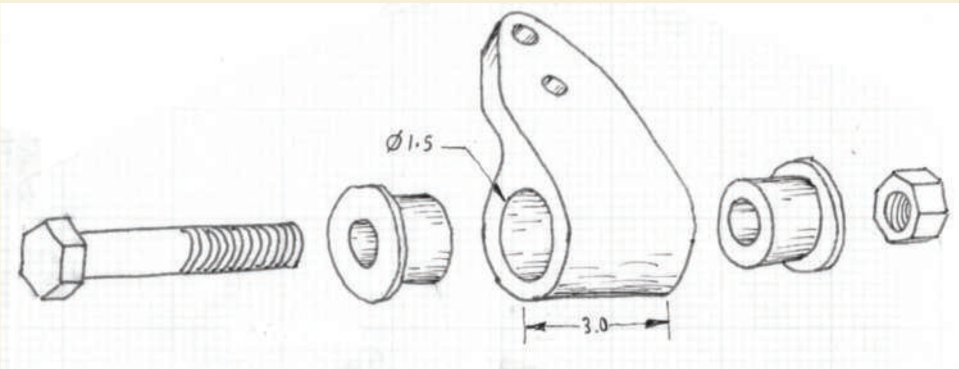

FIGURE P14.12.

14.14 PROBLEMS (CONTINUED)

13. A concept drawing for a caster wheel assembly is shown in Figure P14.13. Using reasonable materials and dimensions of your choice, expand the concept to create a complete set of working drawings for the device, including an outline assembly drawing, an exploded assembly drawing, a sectioned assembly drawing, a bill of materials, and all detailed part drawings. Specify appropriate tolerances for all dimensions. The dimension shown is in inches, but you may use equivalent millimeter dimensions.

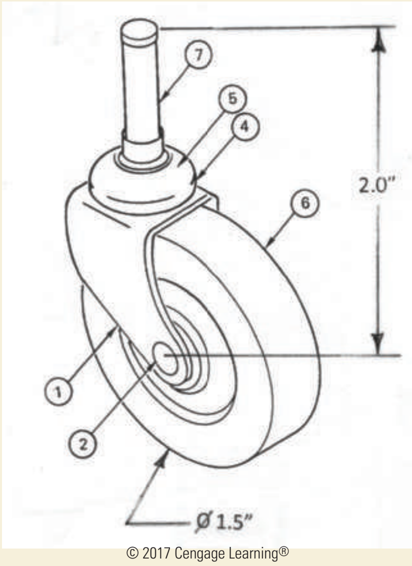

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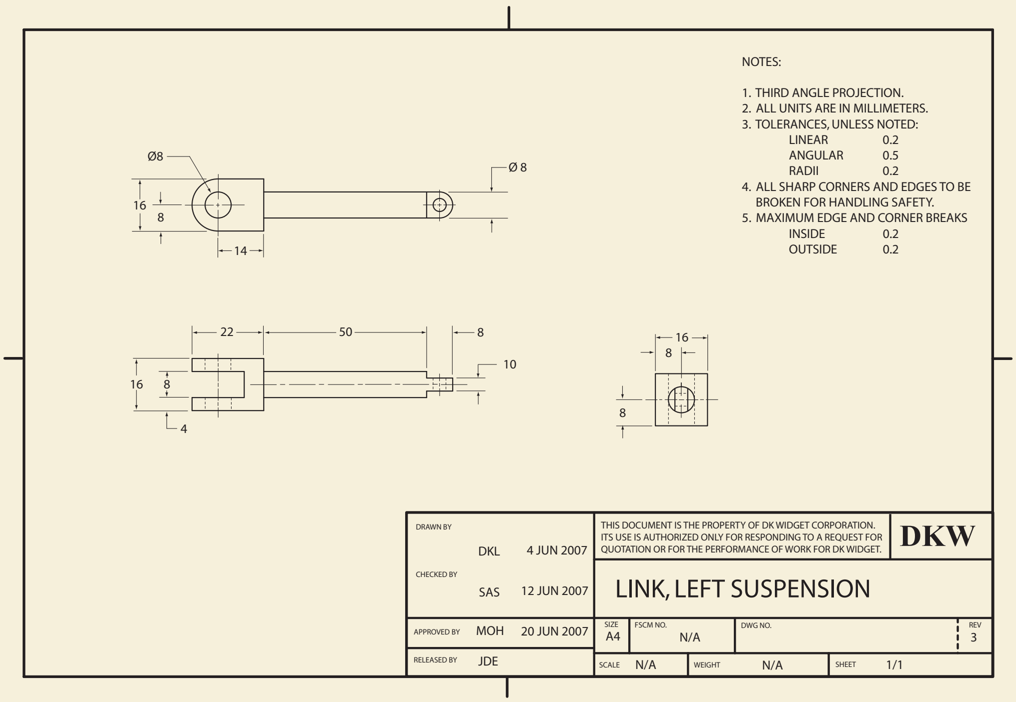
FIGURE P14.13.

14. Find the errors and poor practices in these drawings.

(a)



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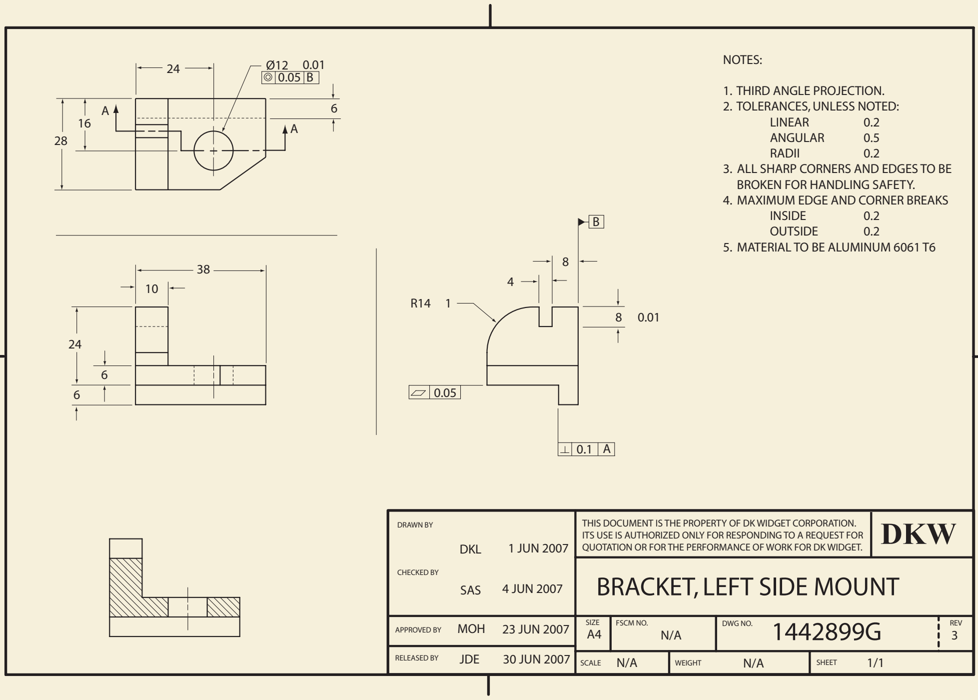
14.14 PROBLEMS (CONTINUED)

(b)



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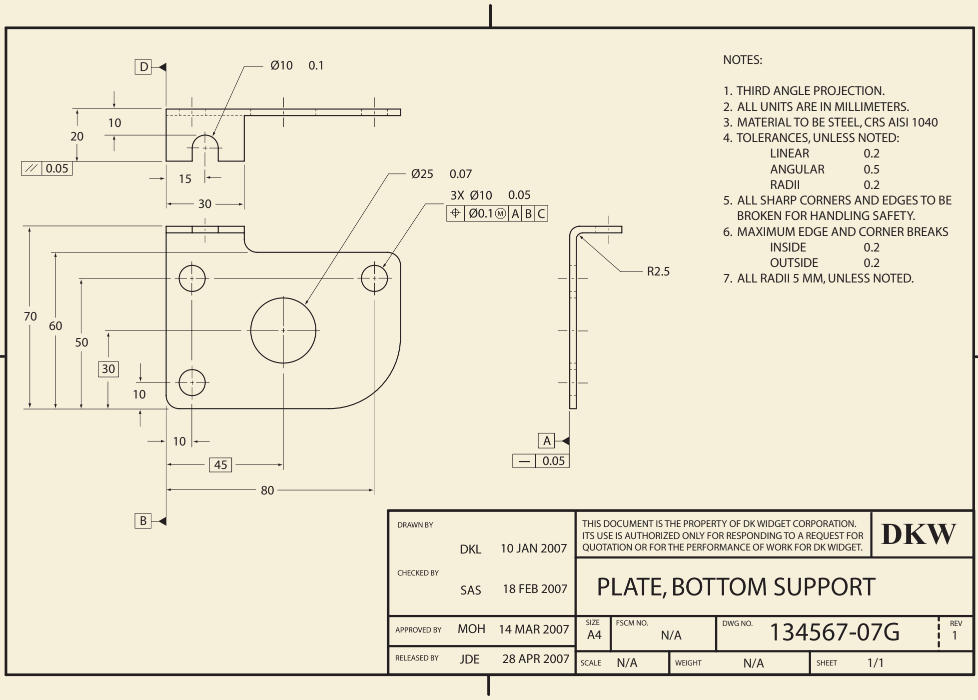
(c)



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14.14 PROBLEMS (CONTINUED)

(d)

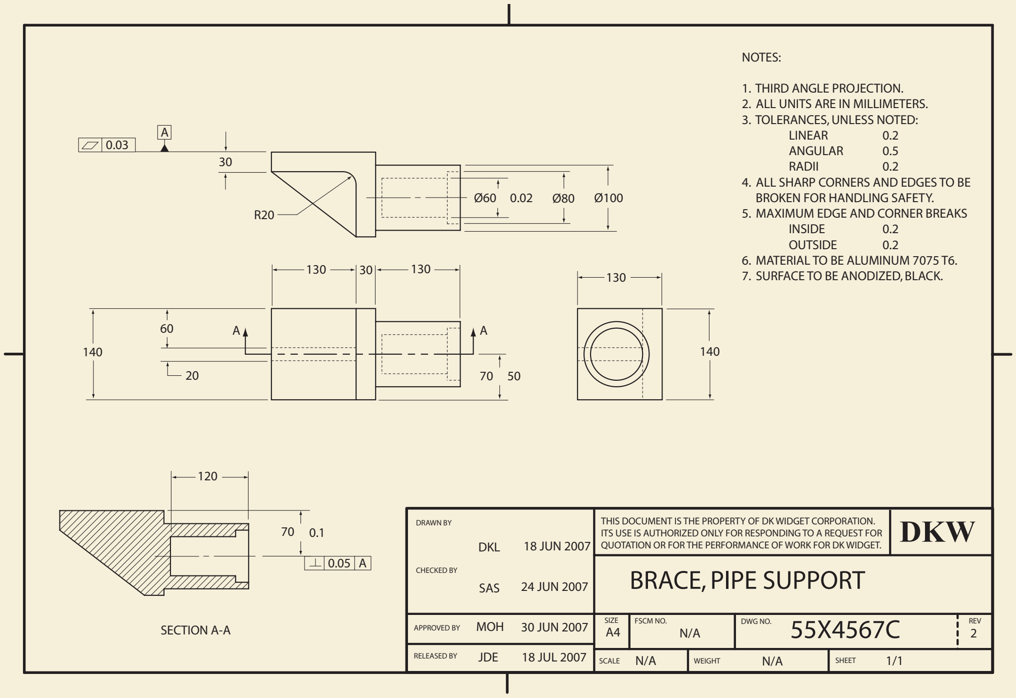

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FIGURE P14.14.

15. Answer the following questions regarding the site plan for the water main extension project in the village of Baraga shown here.

a. What are the names of the four streets bordering the area under consideration?

b. How many existing fire hydrants are shown on the plan?

c. How many proposed fire hydrants are shown on the plan?

d. Counting all houses and separate garages, how many buildings are in the area?

e. What is the diameter of the supply main to be abandoned?

f. What is the diameter of the new water main? How many total linear feet of it is required?

g. For the houses and garages shown on the plan, how many have gravel driveways and how many have bituminous driveways?

14.14 PROBLEMS (CONTINUED)

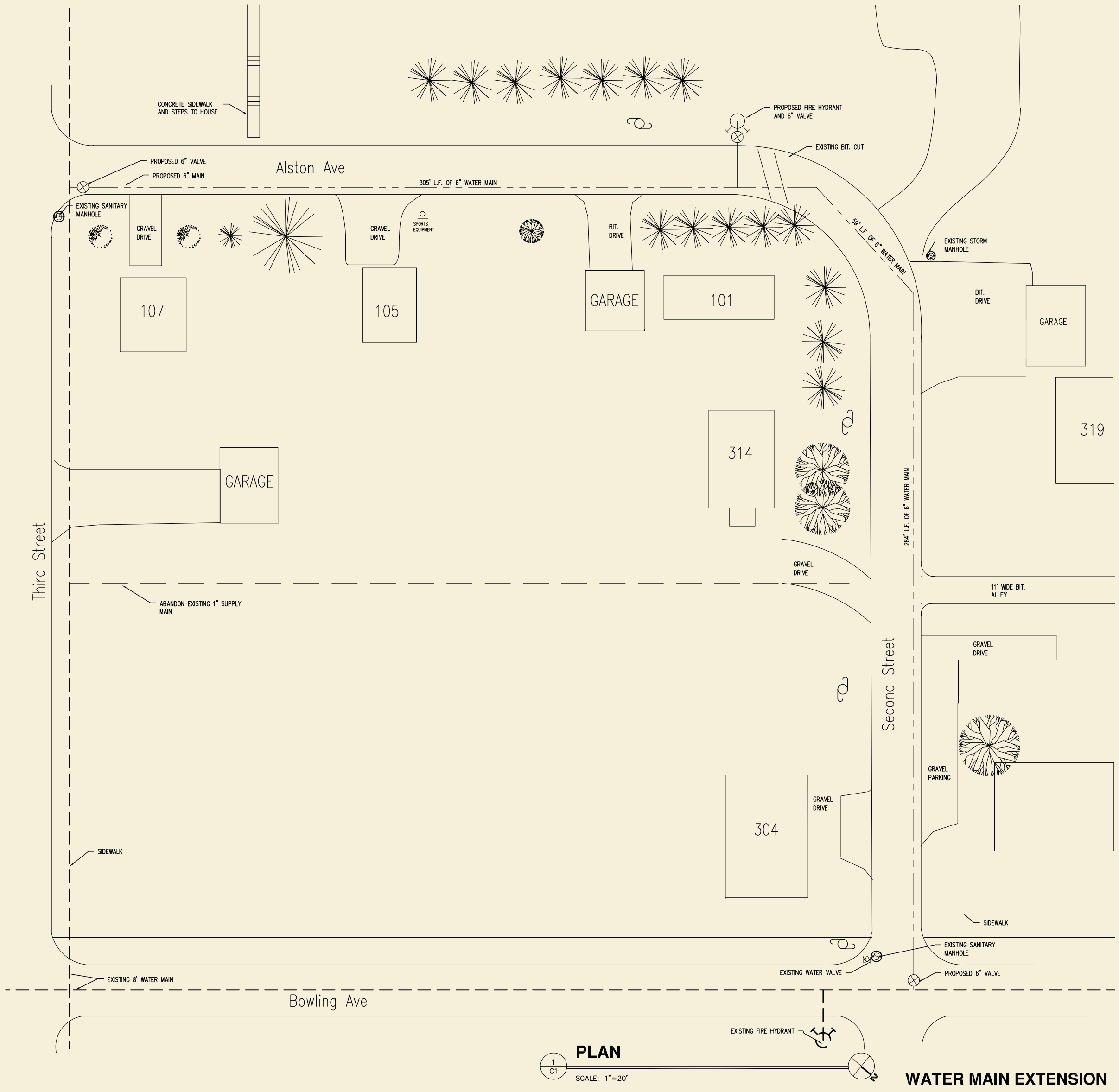

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FIGURE P14.15.

14.14 PROBLEMS (CONTINUED)

16. Answer the following questions regarding the trench detail for the water main extension project in the village of Baraga shown here.

a. What is the trench width for an 8" pipe? for a 12" pipe?

b. What is the minimum depth from the ground to the top of the pipe? Who can approve a smaller minimum depth?

c. What is the slope of the sides of the trench?

d. Where will the contractor find the specifications for the bedding requirements?

e. What is the minimum distance between the bottom of the pipe and the bottom of the trench?

f. What type of compacted material surrounds the pipe?

g. This drawing is made N.T.S. (meaning "not to scale"). Why do you suppose this is an acceptable practice?

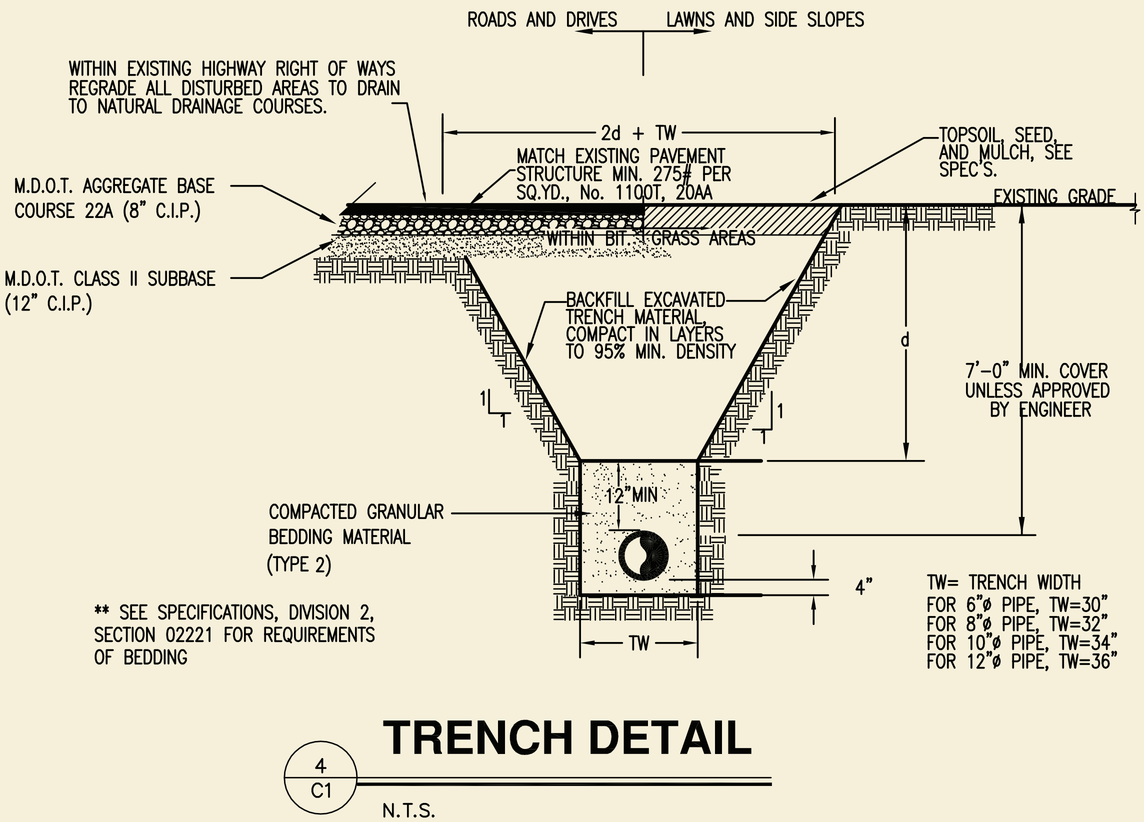

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FIGURE P14.16.

14.14 PROBLEMS (CONTINUED)

17. Answer the following questions regarding the wall section for the water treatment plant expansion project in the village of Baraga shown here.

a. What is the elevation of the top of the footing?

b. What is the elevation of the top of the concrete slab?

c. What is the elevation of the top of the concrete sidewalk?

d. What is the elevation of the bottom of the proposed double tee?

e. What is the depth of the concrete topping that covers the 8 " precast plank?

f. What is the size and type of insulation for the upper portion of the wall?

g. What is the minimum concrete cover for the steel rebars in the wall?

h. How many rebars are required at the top of the door opening? What is the size of the bars there?

i. What is the width of the footing?

j. #4 and #5 rebars are used as reinforcement throughout the wall and footing. What size is used in the footing? What size is used for horizontal reinforcement? What size is used for vertical reinforcement?

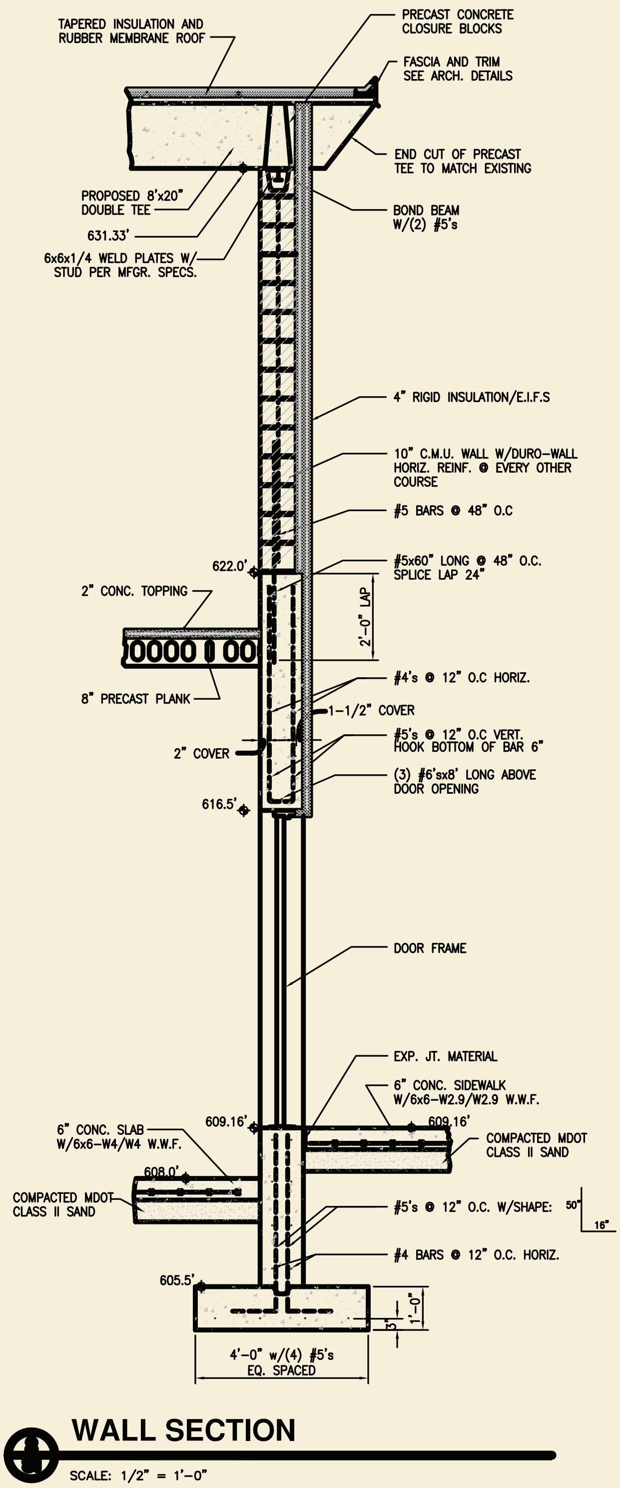


FIGURE P14.17.

14.14 PROBLEMS (CONTINUED)

18. For the figure shown here, measure the lengths of the lines at the indicated scales. (Do not use a calculator for this exercise.) What are the lengths of lines A through F if they are drawn at the indicated scales?

a. 1" = 4000'

b. 1" = 5 yds

c. 1" = 60'

d. 1" = 2'

e. ¼" = 1'-0"

f. ⅜" = 1'-0"

g. ¾" = 1'-0"

h. ½" = 1'-0"

i. 1:2 (use Metric scale)

j. 1:500 (use Metric scale)

k. 1:75 (use Metric scale)

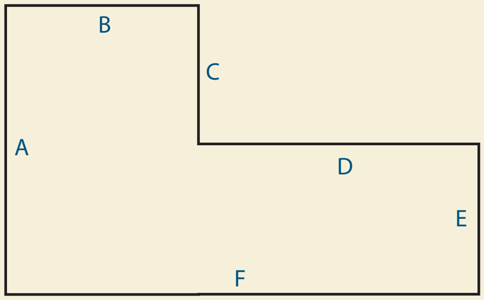


FIGURE P14.18.

APPENDIX

CASE STUDY

TILITE WHEELCHAIRS—"THE ULTIMATE RIDE"

The freedom to get around is important to everyone, but especially to people with disabilities. Users of manual wheelchairs must be independently mobile to enjoy work, travel, sports, and other social activities. They require comfortable, lightweight wheelchairs with features to suit their active lifestyle. A properly designed and fitted wheelchair not only is comfortable but also minimizes energy expenditure and reduces stresses on the user's body. Today people with disabilities can benefit from breakthroughs in design, materials, and manufacturing methods to obtain a unique, customized wheelchair that fits their personal abilities and active lifestyle.

Founded in 1998, TiLite's goal is to provide a twenty-first-century solution to the age-old problem of mobility. TiLite has successfully combined the unique material properties of titanium; traditional fabrication methods; and modern design tools such as parametric modeling, finite element analysis, and rapid prototyping to provide a unique line of affordable, lightweight, and custom-fit manual wheelchairs. TiLite wheelchairs are fabricated from a titanium alloy that is lightweight and has superior strength. Unlike steel, titanium does not corrode and is very durable. The unique combination of these properties is ideal for wheelchair design.

TiLite designs wheelchairs for a variety of users. Ultra-lightweight chairs are ideal for sports enthusiasts such as wheelchair basketball players and marathon racers. A TiLite chair has even carried the Olympic Torch. Children require chairs that are lightweight and adapt to children's growth. Elderly users also benefit from lightweight chairs that are easy to propel. Lightweight folding models are easy to transport. TiLite wheelchairs are designed not only to be functional but also to be stylish, displaying the beautiful patina of polished titanium metal. The ability to custom-design and fabricate a unique chair to fit each individual means that there is a TiLite chair for every user. Modern CAD tools enable designers to modify their designs for custom fabrication.

The TiLite XC Ultralightweight wheelchair.

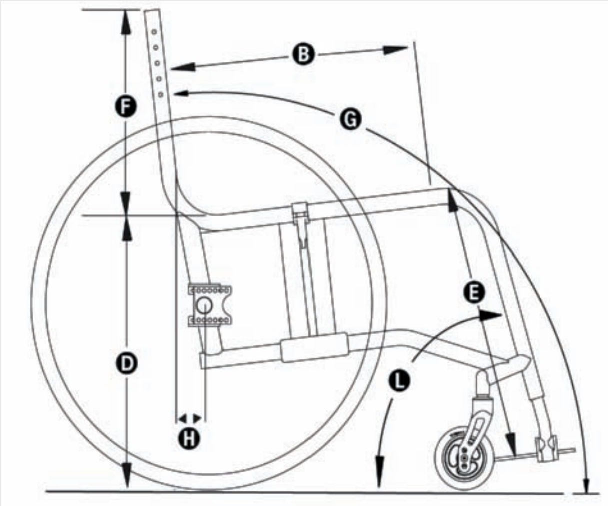
VARIATIONAL DESIGN OF THE TILITE WHEELCHAIR

Over half of all manual wheelchair users suffer from repetitive motion injuries such as carpal tunnel syndrome, chronic shoulder injury, pressure sores, back pain, postural deformities, and reduced heart and lung function. Proper fit and low weight are critical to wheelchair design to reduce these injuries. Studies have shown that wheelchair users suffer fewer long-term health problems with a properly fitted wheelchair. The rear axle of the wheelchair should be positioned forward of the user's shoulder, and the center of gravity of the user and chair should be just forward of the rear axle. The seat should be as narrow as possible to keep the rear wheels close to the frame and the arms close to the body during propulsion. The wheelchair frame must be sized to fit the user's body measurements. These requirements translate into specific dimensions on the wheelchair frame.

For the TiLite TX model, the user must specify 12 measurements or parameters, such as seat width, back angle, footrest width, and wheel camber; there is a choice of between three and 25 possible values for each measurement. With all of these geometric variables and constraints, the designers of TiLite chairs must utilize parametric models to create solid models and drawings of the custom-fit wheelchair design. Based on the user's desired dimensions, the Design Table function in the solid modeling software allows the designer to create all of the necessary

CASE STUDY CONT.

TiLITE WHEELCHAIRS—"THE ULTIMATE RIDE"



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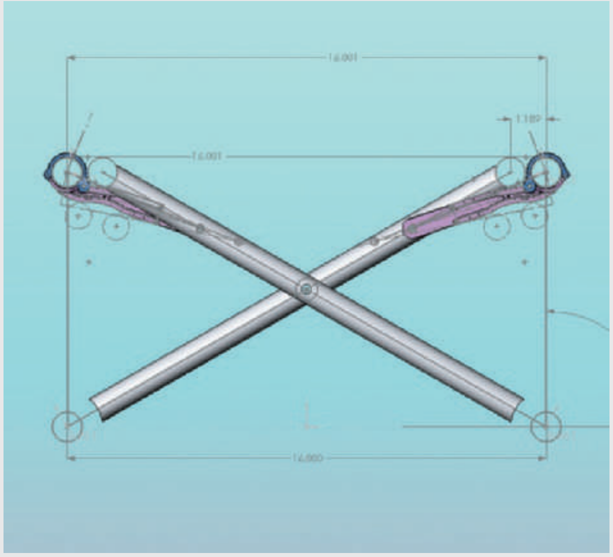
Measurements for ordering a custom wheelchair design

configurations of basic wheelchair frame members. When the user's desired dimensions are inserted into the design tables, all of the remaining dimensions of the parts are automatically adjusted to ensure that the parts fit together and fit the user. From these solid models, all of the necessary drawings for a custom wheelchair can be created in only a few minutes. This saves time for the designer and speeds up delivery of the wheelchair to the user.

ASSEMBLY MODELING OF THE TILITE WHEELCHAIR

For a custom-fit wheelchair design, a vast number of different configurations are available depending on the user's body dimensions and seating preferences. Therefore, it is impossible and impractical to build a physical prototype of each wheelchair frame configuration. Nonetheless, the wheelchair must function properly regardless of the size of the frame members. Solid modeling can be used to create virtual prototypes instead of physical models of any wheelchair design.

A folding wheelchair based on a familiar x-frame design is composed of two cross tubes (shown in gray), seat tubes (blue), and hinge members (pink). When designing a folding wheelchair, the designer must check for interference between parts in the open and folded configurations as well as all positions in between. When the sizes of the parts vary for different users, the model must be carefully checked for each configuration. This can be done most efficiently using a solid model. Stick figure models are created to represent the centerlines of each moving part, measured from their attachment points at the pin joints. These skeleton models are manipulated to



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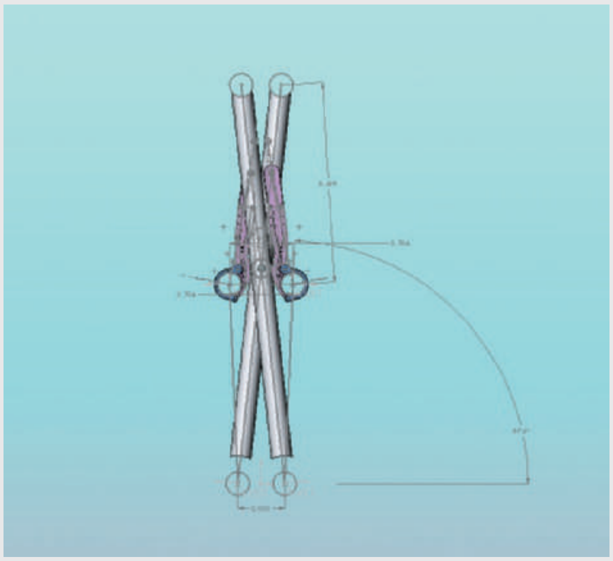
An assembly model of an open wheelchair frame.

make sure that the mechanism does not lock up, invert, or toggle. A trial-and-error method is used to move the locations of the hinge pins until an acceptable design is found that works for all possible sizes of the design.

Although a range of sizes is available for seat tubes and cross tubes, the goal is to have a single design for the hinge members, which may be manufactured by an outside vendor. After the positions of the hinge pins are



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An assembly model of a folded wheelchair frame.

established with the stick figure models, the hinge members are fleshed out and given a shape that will avoid interference with the cross tubes and seat tubes. A variational solid model allows the designer to check multiple configurations quickly and to ensure that all sizes of the wheelchair frame will function as desired.

Another important design consideration is that the wheelchair can be folded compactly. Solid models can be used to ensure that none of the parts interfere during folding. Changes in the design are easily checked for multiple sizes and configurations of the wheelchair. With the use of solid models, the designer can be assured that the wheelchair will fold to the most compact form possible and function smoothly in the folding operation.

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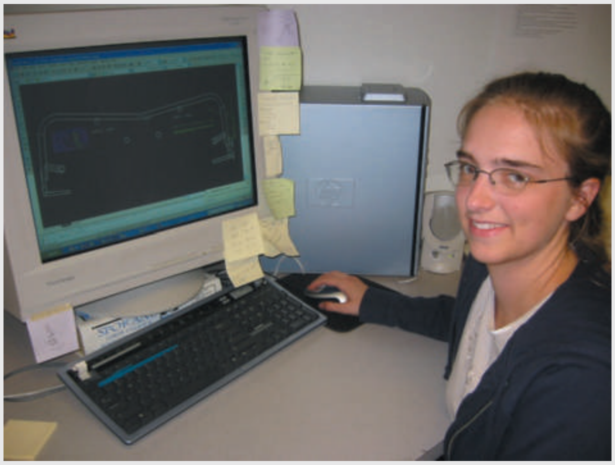
FABRICATION OF THE TILITE WHEELCHAIR

Each custom-fitted wheelchair frame is unique and must be manufactured individually. To begin the fabrication process, TiLite designer Lindy Anderlini uses the parametric models of the wheelchair frame to generate its full-scale layout. The frame drawing is then laid on the surface of a modular fixture, and the stop blocks on the fixture are bolted down in the proper positions to hold each piece of the tubing. Setting up the modular fixture takes only about ten minutes.

Each tube member is carefully bent to the proper angle using a special bending machine. The tubing is then measured and cut to the correct length based on the scale drawing, and the ends are shaped to fit snugly to the mating parts. Proper sizing and positioning of the frame members is critical to obtain sturdy weld joints. By using the full-scale drawing as a template, the manufacturer can ensure a perfect fit for every customer. After each piece of tubing has been formed and cut, they are laid in the fixture and clamped in the proper position. The parts are tack-welded together, then removed from the fixture and finish-welded. In completion of the fabrication process, the frame is drilled in the necessary locations for assembly of other parts and bead-blasted, hand-buffed, or painted according to the user's preference for surface finish.

DISCUSSION QUESTIONS/ACTIVITIES

1. Explain how a parametric solid modeler can be used to customize the design of wheelchairs to fit individual users. How is the model used in the design process? the manufacturing process?
2. Compare a custom design to a standard wheelchair with adjustable components. What are the advantages and disadvantages of each design?
3. Make a list of the important design and performance specifications for a lightweight wheelchair. How can a solid model be used to check wheelchair design to ensure that the design meets the desired specifications?



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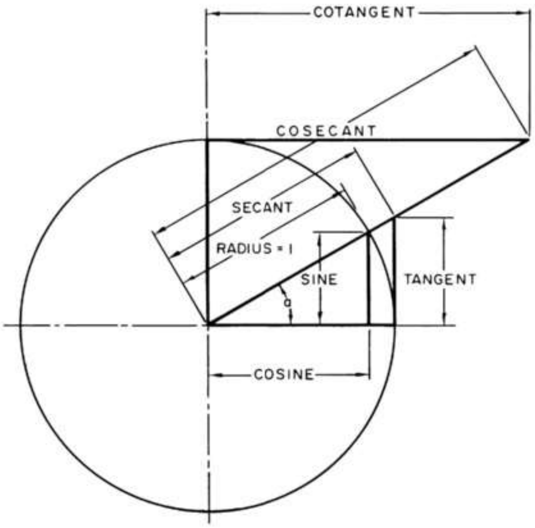
TiLite designer Lindy Anderlini creates full-scale drawings of wheelchair frames.

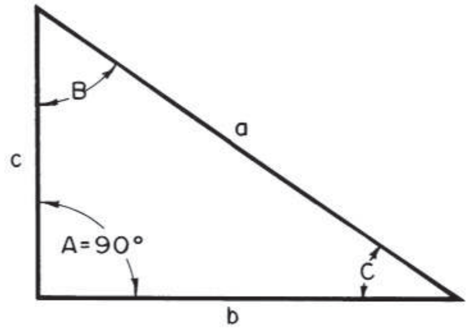
TRIGONOMETRIC FORMULAS



FORMULAS FOR FINDING FUNCTIONS OF ANGLES	
$\frac{\text{Side opposite}}{\text{Hypotenuse}}$	= SINE
$\frac{\text{Side adjacent}}{\text{Hypotenuse}}$	= COSINE
$\frac{\text{Side opposite}}{\text{Side adjacent}}$	= TANGENT
$\frac{\text{Side adjacent}}{\text{Side opposite}}$	= COTANGENT
$\frac{\text{Hypotenuse}}{\text{Side adjacent}}$	= SECANT
$\frac{\text{Hypotenuse}}{\text{Side opposite}}$	= COSECANT
FORMULAS FOR FINDING THE LENGTH OF SIDES FOR RIGHT-ANGLE TRIANGLES WHEN AN ANGLE AND SIDE ARE KNOWN	
Length of side opposite	Hypotenuse × Sine Hypotenuse ÷ Cosecant Side adjacent × Tangent Side adjacent ÷ Cotangent
Length of side adjacent	Hypotenuse × Cosine Hypotenuse ÷ Secant Side opposite × Cotangent Side opposite ÷ Tangent
Length of hypotenuse	Side opposite × Cosecant Side opposite ÷ Sine Side adjacent × Secant Side adjacent ÷ Cosine

Trigonometric Formulas for Triangles

RIGHT-TRIANGLE FORMULAS

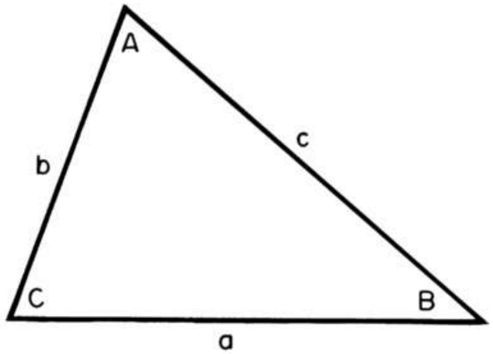


TO FIND ANGLES	FORMULAS	
C	$\frac{c}{a}$ = Sine C	90°−B
C	$\frac{b}{a}$ = Cosine C	90°−B
C	$\frac{c}{b}$ = Tan C	90°−B
C	$\frac{b}{c}$ = Cot C	90°−B
C	$\frac{a}{b}$ = Secant C	90°−B
C	$\frac{a}{c}$ = Csc C	90°−B
B	$\frac{b}{a}$ = Sine B	90°−C
B	$\frac{c}{a}$ = Cosine B	90°−C
B	$\frac{b}{c}$ = Tan B	90°−C
B	$\frac{c}{b}$ = Cot B	90°−C
B	$\frac{a}{c}$ = Secant B	90°−C
B	$\frac{a}{b}$ = Csc B	90°−C

TO FIND SIDES	FORMULAS	
a	$\sqrt{b^2 + c^2}$	
a	c × Csc C	$\frac{c}{\text{sine C}}$
a	c × Secant B	$\frac{c}{\text{Cosine B}}$
a	b × Csc B	$\frac{b}{\text{Sine B}}$
a	b × Secant C	$\frac{b}{\text{Cosine C}}$
b	$\sqrt{a^2 - c^2}$	
b	a × Sine B	$\frac{a}{\text{Cosecant B}}$
b	a × Cos C	$\frac{a}{\text{Secant C}}$
b	c × Tan B	$\frac{c}{\text{Cotangent B}}$
b	c × Cot C	$\frac{c}{\text{Tangent C}}$
c	$\sqrt{a^2 - b^2}$	
c	a × Cos B	$\frac{a}{\text{Secant B}}$
c	a × Sine C	$\frac{a}{\text{Cosecant C}}$
c	b × Cot B	$\frac{b}{\text{Tangent B}}$
c	b × Tan C	$\frac{b}{\text{Cotangent C}}$

Trigonometric Formulas for Triangles (CONTINUED)

OBLIQUE-ANGLED TRIANGLE FORMULAS

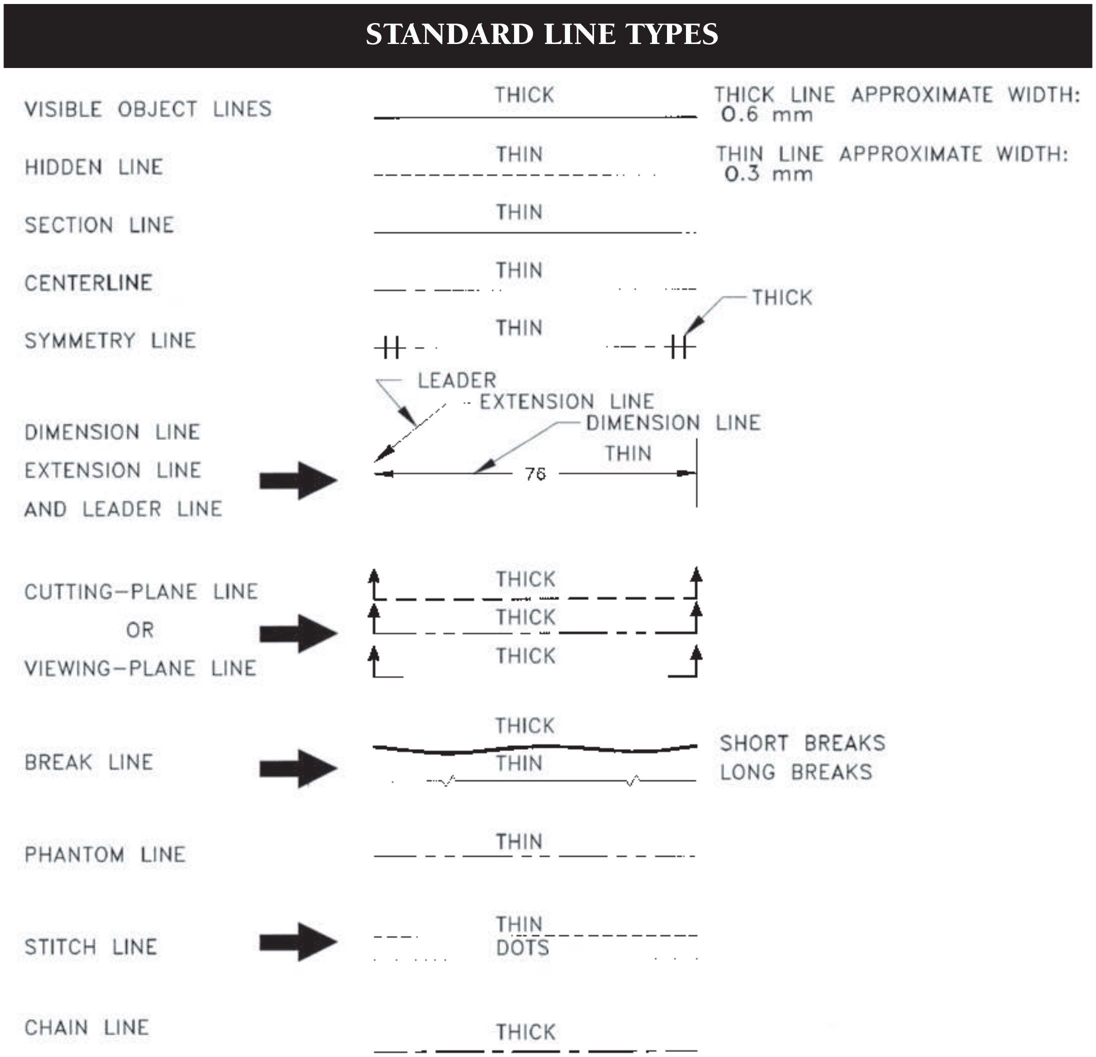


TO FIND	KNOWN	SOLUTION
C	A-B	$180° - (A + B)$
b	a-B-A	$\frac{a \times \text{Sin } B}{\text{Sin } A}$
c	a-A-C	$\frac{a \times \text{Sin } C}{\text{Sin } A}$
Tan A	a-C-b	$\frac{a \times \text{Sin } C}{b - (a \times \text{Cos } C)}$
B	A-C	$180° - (A + C)$
Sin B	b-A-a	$\frac{b \times \text{Sin } A}{a}$
A	B-C	$180° - (B + C)$
Cos A	a-b-c	$\frac{b^2 + C^2 - a^2}{2bc}$
Sin C	c-A-a	$\frac{c \times \text{Sin } A}{a}$
Cot B	a-C-b	$\frac{a \times \text{Csc } C}{b} - \text{Cot } C$
c	b-C-B	$b \times \text{Sin } C \times \text{Csc } B$

Trigonometric Formulas for Triangles (CONTINUED)

RULES RELATIVE TO THE CIRCLE			
To Find Circumference—			
Multiply diameter by	3.1416	Or divide diameter by	0.3183
To Find Diameter—			
Multiply circumference by	0.3183	Or divide circumference by	3.1416
To Find Radius—			
Multiply circumference by	0.15915	Or divide circumference by	6.28318
To Find Side of an Inscribed Square—			
Multiply diameter by	0.7071		
Or multiply circumference by	0.2251	Or divide circumference by	4.4428
To Find Side of an Equal Square—			
Multiply diameter by	0.8862	Or divide diameter by	1.1284
Or multiply circumference by	0.2821	Or divide circumference by	3.545
Square—			
A side multiplied by	1.4142	equals diameter of its circumscribing circle.	
A side multiplied by	4.443	equals circumference of its circumscribing circle.	
A side multiplied by	1.128	equals diameter of an equal circle.	
A side multiplied by	3.547	equals circumference of an equal circle.	
To Find the Area of a Circle—			
Multiply circumference by one-quarter of the diameter.			
Or multiply the square of diameter by	0.7854		
Or multiply the square of circumference by	.07958		
Or multiply the square of 1/2 diameter by	3.1416		
To Find the Surface of a Sphere or Globe—			
Multiply the diameter by the circumference.			
Or multiply the square of diameter by	3.1416		
Or multiply four times the square of radius by	3.1416		

Trigonometric Formulas for Circles.



Standard Line Types for Drafting.

Inches to Millimeters Conversion.

INCHES TO MILLIMETERS

in.	mm	in.	mm	in.	mm	in.	mm
1	25.4	26	660.4	51	1295.4	76	1930.4
2	50.8	27	685.8	52	1320.8	77	1955.8
3	76.2	28	711.2	53	1346.2	78	1981.2
4	101.6	29	736.6	54	1371.6	79	2006.6
5	127.0	30	762.0	55	1397.0	80	2032.0
6	152.4	31	787.4	56	1422.4	81	2057.4
7	177.8	32	812.8	57	1447.8	82	2082.8
8	203.2	33	838.2	58	1473.2	83	2108.2
9	228.6	34	863.6	59	1498.6	84	2133.6
10	254.0	35	889.0	60	1524.0	85	2159.0
11	279.4	36	914.4	61	1549.4	86	2184.4
12	304.8	37	939.8	62	1574.8	87	2209.8
13	330.2	38	965.2	63	1600.2	88	2235.2
14	355.6	39	990.6	64	1625.6	89	2260.6
15	381.0	40	1016.0	65	1651.0	90	2286.0
16	406.4	41	1041.4	66	1676.4	91	2311.4
17	431.8	42	1066.8	67	1701.8	92	2336.8
18	457.2	43	1092.2	68	1727.2	93	2362.2
19	482.6	44	1117.6	69	1752.6	94	2387.6
20	508.0	45	1143.0	70	1778.0	95	2413.0
21	533.4	46	1168.4	71	1803.4	96	2438.4
22	558.8	47	1193.8	72	1828.8	97	2463.8
23	584.2	48	1219.2	73	1854.2	98	2489.2
24	609.6	49	1244.6	74	1879.6	99	2514.6
25	635.0	50	1270.0	75	1905.0	100	2540.0

The above table is exact on the basis: 1 in. = 25.4 mm

Millimeters to Inches Conversion.

MILLIMETERS TO INCHES

in.	mm	in.	mm	in.	mm	in.	mm
1	0.039370	26	1.023622	51	2.007874	76	2.992126
2	0.078740	27	1.062992	52	2.047244	77	3.031496
3	0.118110	28	1.102362	53	2.086614	78	3.070866
4	0.157480	29	1.141732	54	2.125984	79	3.110236
5	0.196850	30	1.181102	55	2.165354	80	3.149606
6	0.236220	31	1.220472	56	2.204724	81	3.188976
7	0.275591	32	1.259843	57	2.244094	82	3.228346
8	0.314961	33	1.299213	58	2.283465	83	3.267717
9	0.354331	34	1.338583	59	2.322835	84	3.307087
10	0.393701	35	1.377953	60	2.362205	85	3.346457
11	0.433071	36	1.417323	61	2.401575	86	3.385827
12	0.472441	37	1.456693	62	2.440945	87	3.425197
13	0.511811	38	1.496063	63	2.480315	88	3.464567
14	0.551181	39	1.535433	64	2.519685	89	3.503937
15	0.590551	40	1.574803	65	2.559055	90	3.543307
16	0.629921	41	1.614173	66	2.598425	91	3.582677
17	0.669291	42	1.653543	67	2.637795	92	3.622047
18	0.708661	43	1.692913	68	2.677165	93	3.661417
19	0.748031	44	1.732283	69	2.716535	94	3.700787
20	0.787402	45	1.771654	70	2.755906	95	3.740157
21	0.826772	46	1.811024	71	2.795276	96	3.779528
22	0.866142	47	1.850394	72	2.834646	97	3.818898
23	0.905512	48	1.889764	73	2.874016	98	3.858268
24	0.944882	49	1.929134	74	2.913386	99	3.897638
25	0.984252	50	1.968504	75	2.952756	100	3.937008

The above table is approximate on the basis: 1 in. = 25.4 mm, 1/25.4 = 0.039370078740+

INCH/METRIC EQUIVALENTS					
Fraction	Decimal Equivalent Customary (in.)	Decimal Equivalent Metric (mm)	Fraction	Decimal Equivalent Customary (in.)	Decimal Equivalent Metric (mm)
1/64	.015625	0.3969	33/64	.515625	13.0969
1/32	.03125	0.7938	17/32	.53125	13.4938
3/64	.046875	1.1906	35/64	.546875	13.8906
1/16	.0625	1.5875	9/16	.5625	14.2875
5/64	.078125	1.9844	37/64	.578125	14.6844
3/32	.09375	2.3813	19/32	.59375	15.0813
7/64	.109375	2.7781	39/64	.609375	15.4781
1/8	.1250	3.1750	5/8	.6250	15.8750
9/64	.140625	3.5719	41/64	.640625	16.2719
5/32	.15625	3.9688	21/32	.65625	16.6688
11/64	.171875	4.3656	43/64	.671875	17.0656
3/16	.1875	4.7625	11/16	.6875	17.4625
13/64	.203125	5.1594	45/64	.703125	17.8594
7/32	.21875	5.5563	23/32	.71875	18.2563
15/64	.234375	5.9531	47/64	.734375	18.6531
1/4	.250	6.3500	3/4	.750	19.0500
17/64	.265625	6.7469	49/64	.765625	19.4469
9/32	.28125	7.1438	25/32	.78125	19.8438
19/64	.296875	7.5406	51/64	.796875	20.2406
5/16	.3125	7.9375	13/16	.8125	20.6375
21/64	.328125	8.3384	53/64	.828125	21.0344
11/32	.34375	8.7313	27/32	.84375	21.4313
23/64	.359375	9.1281	55/64	.859375	21.8281
3/8	.3750	9.5250	7/8	.8750	22.2250
25/64	.390625	9.9219	57/64	.890625	22.6219
13/32	.40625	10.3188	29/32	.90625	23.0188
27/64	.421875	10.7156	59/64	.921875	23.4156
7/16	.4375	11.1125	15/16	.9375	23.8125
29/64	.453125	11.5094	61/64	.953125	24.2094
15/32	.46875	11.9063	31/32	.96875	24.6063
31/64	.484375	12.3031	63/64	.984375	25.0031
1/2	.500	12.7000	1	1.000	25.4000

Inch/Metric Equivalents.

Measures of Length

1 millimeter (mm) = 0.03937 inch
1 centimeter (cm) = 0.39370 inch
1 meter (m) = 39.37008 inches
= 3.2808 feet
= 1.0936 yards
1 kilometer (km) = 0.6214 mile
1 inch = 25.4 millimeters (mm)
= 2.54 centimeters (cm)
1 foot = 304.8 millimeters (mm)
= 0.3048 meter (m)
1 yard = 0.9144 meter (m)
1 mile = 1.609 kilometers (km)

Measures of Area

1 square millimeter = 0.00155 square inch
1 square centimeter = 0.155 square inch
1 square meter = 10.764 square feet
= 1.196 square yards
1 square kilometer = 0.3861 square mile
1 square inch = 645.2 square millimeters
= 6.452 square centimeters
1 square foot = 929 square centimeters
= 0.0929 square meter
1 square yard = 0.836 square meter
1 square mile = 2.5899 square kilometers

Measures of Capacity (Dry)

1 cubic centimeter (cm^3) = 0.061 cubic inch
1 liter = 0.0353 cubic foot
= 61.023 cubic inches
1 cubic meter (m^3) = 35.315 cubic feet
= 1.308 cubic yards
1 cubic inch = 16.38706 cubic centimeters (cm^3)
1 cubic foot = 0.02832 cubic meter (m^3)
= 28.317 liters
1 cubic yard = 0.7646 cubic meter (m^3)

Measures of Capacity (Liquid)

1 liter = 1.0567 U.S. quarts
= 0.2642 U.S. gallon
= 0.2200 Imperial gallon
1 cubic meter (m^3) = 264.2 U.S. gallons
= 219.969 Imperial gallons
1 U.S. quart = 0.946 liter
1 Imperial quart = 1.136 liters
1 U.S. gallon = 3.785 liters
1 Imperial gallon = 4.546 liters

Measures of Weight

1 gram (g) = 15.432 grains
= 0.03215 ounce troy
= 0.03527 ounce avoirdupois
1 kilogram (kg) = 35.274 ounces avoirdupois
= 2.2046 pounds
1000 kilograms (kg) = 1 metric ton (t)
= 1.1023 tons of 2000 pounds
= 0.9842 ton of 2240 pounds
1 ounce avoirdupois = 28.35 grams (g)
1 ounce troy = 31.103 grams (g)
1 pound = 453.6 grams
= 0.4536 kilogram (kg)
1 ton of 2240 pounds = 1016 kilograms (kg)
= 1.016 metric tons
1 grain = 0.0648 gram (g)
1 metric ton = 0.9842 ton of 2240 pounds
= 2204.6 pounds

Inch/Metric Conversion.

Grade Marking	Specification	Material
 NO MARK	SAE–Grade 1	Low or Medium Carbon Steel
	ASTM–A307	Low Carbon Steel
	SAE–Grade 2	Low or Medium Carbon Steel
	SAE–Grade 5	Medium Carbon Steel, Quenched and Tempered
	ASTM–A 449	
	SAE–Grade 5.2	Low Carbon Martensite Steel, Quenched and Tempered
	ASTM–A 325 Type 1	Medium Carbon Steel, Quenched and Tempered Radial dashes optional
	ASTM–A 325 Type 2	Low Carbon Martensite Steel, Quenched and Tempered
	ASTM–A 325 Type 3	Atmospheric Corrosion (Weathering) Steel, Quenched and Tempered
	ASTM–A 354 Grade BC	Alloy Steel, Quenched and Tempered
	SAE–Grade 7	Medium Carbon Alloy Steel, Quenched and Tempered, Roll Threaded After Heat Treatment
	SAE–Grade 8	Medium Carbon Alloy Steel, Quenched and Tempered
	ASTM–A 354 Grade BD	Alloy Steel, Quenched and Tempered
	SAE–Grade 8.2	Low Carbon Martensite Steel, Quenched and Tempered
	ASTM–A 490 Type 1	Alloy Steel, Quenched and Tempered
	ASTM–A 490 Type 3	Atmospheric Corrosion (Weathering) Steel, Quenched and Tempered

ASTM and SAE Grade Markings for Steel Bolts and Screws.

Sizes		Basic Major Diameter	THREADS PER INCH											Sizes
			Series with graded pitches			Series with constant pitches								
Primary	Secondary		Coarse UNC	Fine UNF	Extra fine UNEF	4UN	6UN	8UN	12UN	16UN	20UN	28UN	32UN	
0		0.0600	—	80	—	—	—	—	—	—	—	—	—	0
	1	0.0730	64	72	—	—	—	—	—	—	—	—	—	1
2		0.0860	56	64	—	—	—	—	—	—	—	—	—	2
	3	0.0990	48	56	—	—	—	—	—	—	—	—	—	3
4		0.1120	40	48	—	—	—	—	—	—	—	—	—	4
5		0.1250	40	44	—	—	—	—	—	—	—	—	—	5
6		0.1380	32	40	—	—	—	—	—	—	—	—	UNC	6
8		0.1640	32	36	—	—	—	—	—	—	—	—	UNC	8
10		0.1900	24	32	—	—	—	—	—	—	—	—	UNF	10
	12	0.2160	24	28	32	—	—	—	—	—	—	UNF	UNEF	12
1/4		0.2500	20	28	32	—	—	—	—	—	UNC	UNF	UNEF	1/4
5/16		0.3125	18	24	32	—	—	—	—	—	20	28	UNEF	5/16
3/8		0.3750	16	24	32	—	—	—	—	UNC	20	28	UNEF	3/8
7/16		0.4375	14	20	28	—	—	—	—	16	UNF	UNEF	32	7/16
1/2		0.5000	13	20	28	—	—	—	—	16	UNF	UNEF	32	1/2
9/16		0.5625	12	18	24	—	—	—	UNC	16	20	28	32	9/16
5/8		0.6250	11	18	24	—	—	—	12	16	20	28	32	5/8
	11/16	0.6875	—	—	24	—	—	—	12	16	20	28	32	11/16
3/4		0.7500	10	16	20	—	—	—	12	UNF	UNEF	28	32	3/4
	13/16	0.8125	—	—	20	—	—	—	12	16	UNEF	28	32	13/16
7/8		0.8750	9	14	20	—	—	—	12	16	UNEF	28	32	7/8
	15/16	0.9375	—	—	20	—	—	—	12	16	UNEF	28	32	15/16
1		1.0000	8	12	20	—	—	UNC	UNF	16	UNEF	28	32	1
	1 1/16	1.0625	—	—	18	—	—	8	12	16	20	28	—	1 1/16
1 1/8		1.1250	7	12	18	—	—	8	UNF	16	20	28	—	1 1/8
	1 3/16	1.1875	—	—	18	—	—	8	12	16	20	28	—	1 3/16
1 1/4		1.2500	7	12	18	—	—	8	UNF	16	20	28	—	1 1/4
	1 5/16	1.3125	—	—	18	—	—	8	12	16	20	28	—	1 5/16
1 3/8		1.3750	6	12	18	—	UNC	8	UNF	16	20	28	—	1 3/8
	1 7/16	1.4375	—	—	18	—	6	8	12	16	20	28	—	1 7/16
1 1/2		1.5000	6	12	18	—	UNC	8	UNF	16	20	28	—	1 1/2
	1 9/16	1.5625	—	—	18	—	6	8	12	16	20	—	—	1 9/16
1 5/8		1.6250	—	—	18	—	6	8	12	16	20	—	—	1 5/8
	1 11/16	1.6875	—	—	18	—	6	8	12	16	20	—	—	1 11/16
1 3/4		1.7500	5	—	—	—	6	8	12	16	20	—	—	1 3/4
	1 13/16	1.8125	—	—	—	—	6	8	12	16	20	—	—	1 13/16
1 7/8		1.8750	—	—	—	—	6	8	12	16	20	—	—	1 7/8
	1 15/16	1.9375	—	—	—	—	6	8	12	16	20	—	—	1 15/16
2		2.0000	4 1/2	—	—	—	6	8	12	16	20	—	—	2
	2 1/8	2.1250	—	—	—	—	6	8	12	16	20	—	—	2 1/8
2 1/4		2.2500	4 1/2	—	—	—	6	8	12	16	20	—	—	2 1/4
	2 3/8	2.3750	—	—	—	—	6	8	12	16	20	—	—	2 3/8
2 1/2		2.5000	4	—	—	UNC	6	8	12	16	20	—	—	2 1/2
	2 5/8	2.6250	—	—	—	4	6	8	12	16	20	—	—	2 5/8
2 3/4		2.7500	4	—	—	UNC	6	8	12	16	20	—	—	2 3/4
	2 7/8	2.8750	—	—	—	4	6	8	12	16	20	—	—	2 7/8
3		3.0000	4	—	—	UNC	6	8	12	16	20	—	—	3
	3 1/8	3.1250	—	—	—	4	6	8	12	16	—	—	—	3 1/8
3 1/4		3.2500	4	—	—	UNC	6	8	12	16	—	—	—	3 1/4
	3 3/8	3.3750	—	—	—	4	6	8	12	16	—	—	—	3 3/8
3 1/2		3.5000	4	—	—	UNC	6	8	12	16	—	—	—	3 1/2
	3 5/8	3.6250	—	—	—	4	6	8	12	16	—	—	—	3 5/8
3 3/4		3.7500	4	—	—	UNC	6	8	12	16	—	—	—	3 3/4
	3 7/8	3.8750	—	—	—	4	6	8	12	16	—	—	—	3 7/8
4		4.0000	4	—	—	UNC	6	8	12	16	—	—	—	4
	4 1/8	4.1250	—	—	—	4	6	8	12	16	—	—	—	4 1/8
4 1/4		4.2500	—	—	—	4	6	8	12	16	—	—	—	4 1/4
	4 3/8	4.3750	—	—	—	4	6	8	12	16	—	—	—	4 3/8
4 1/2		4.5000	—	—	—	4	6	8	12	16	—	—	—	4 1/2
	4 5/8	4.6250	—	—	—	4	6	8	12	16	—	—	—	4 5/8
4 3/4		4.7500	—	—	—	4	6	8	12	16	—	—	—	4 3/4
	4 7/8	4.8750	—	—	—	4	6	8	12	16	—	—	—	4 7/8
5		5.0000	—	—	—	4	6	8	12	16	—	—	—	5
	5 1/8	5.1250	—	—	—	4	6	8	12	16	—	—	—	5 1/8
5 1/4		5.2500	—	—	—	4	6	8	12	16	—	—	—	5 1/4
	5 3/8	5.3750	—	—	—	4	6	8	12	16	—	—	—	5 3/8
5 1/2		5.5000	—	—	—	4	6	8	12	16	—	—	—	5 1/2
	5 5/8	5.6250	—	—	—	4	6	8	12	16	—	—	—	5 5/8
5 3/4		5.7500	—	—	—	4	6	8	12	16	—	—	—	5 3/4
	5 7/8	5.8750	—	—	—	4	6	8	12	16	—	—	—	5 7/8
6		6.0000	—	—	—	4	6	8	12	16	—	—	—	6

Unified Standard Screw Thread Series.

ISO BASIC METRIC THREAD INFORMATION

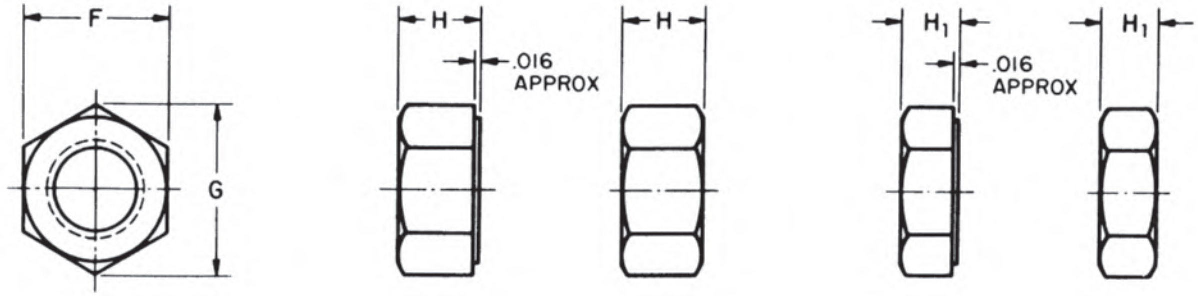
	INTERNAL THREADS			EXTERNAL THREADS		
Basic Major DIA & Pitch	Tap Drill DIA	Minor DIA MAX	Minor DIA MIN	Major DIA MAX	Major DIA MIN	Clearance Hole
M1.6 × 0.35	1.25	1.321	1.221	1.576	1.491	1.9
M2 × 0.4	1.60	1.679	1.567	1.976	1.881	2.4
M2.5 × 0.45	2.05	2.138	2.013	2.476	2.013	2.9
M3 × 0.5	2.50	2.599	2.459	2.976	2.870	3.4
M3.5 × 0.6	2.90	3.010	2.850	3.476	3.351	4.0
M4 × 0.7	3.30	3.422	3.242	3.976	3.836	4.5
M5 × 0.8	4.20	4.334	4.134	4.976	4.826	5.5
M6 × 1	5.00	5.153	4.917	5.974	5.794	6.6
M8 × 1.25	6.80	6.912	6.647	7.972	7.760	9.0
M10 × 1.5	8.50	8.676	8.376	9.968	9.732	11.0
M12 × 1.75	10.20	10.441	10.106	11.966	11.701	13.5
M14 × 2	12.00	12.210	11.835	13.962	13.682	15.5
M16 × 2	14.00	14.210	13.835	15.962	15.682	17.5
M20 × 2.5	17.50	17.744	17.294	19.958	19.623	22.0
M24 × 3	21.00	21.252	20.752	23.952	23.577	26.0
M30 × 3.5	26.50	26.771	26.211	29.947	29.522	33.0
M36 × 4	32.00	32.270	31.670	35.940	35.465	39.0
M42 × 4.5	37.50	37.799	37.129	41.937	41.437	45.0
M48 × 5	43.00	43.297	42.587	47.929	47.399	52.0
M56 × 5.5	50.50	50.796	50.046	55.925	55.365	62.0
M64 × 6	58.00	58.305	57.505	63.920	63.320	70.0
M72 × 6	66.00	66.305	65.505	71.920	71.320	78.0
M80 × 6	74.00	74.305	73.505	79.920	79.320	86.0
M90 × 6	84.00	84.305	83.505	89.920	89.320	96.0
M100 × 6	94.00	94.305	93.505	99.920	99.320	107.0

ISO Metric Thread Information.

INCH–METRIC THREAD COMPARISON

INCH SERIES			METRIC			
Size	Dia.(In.)	TPI	Size	Dia. (In.)	Pitch (mm)	TPI (Approx)
			M1.4	.055	.3 .2	85 127
#0	.060	80				
			M1.6	.063	.35 .2	74 127
#1	.073	64 72				
			M2	.079	.4 .25	64 101
#2	.086	56 64				
			M2.5	.098	.45 .35	56 74
#3	.099	48 56				
#4	.112	40 48				
			M3	.118	.5 .35	51 74
#5	.125	40 44				
#6	.138	32 40				
			M4	.157	.7 .5	36 51
#8	.164	32 36				
#10	.190	24 32				
			M5	.196	.8 .5	32 51
			M6	.236	1.0 .75	25 34
¼	.250	20 28				
5/16	.312	18 24				
			M8	.315	1.25 1.0	20 25
⅜	.375	16 24				
			M10	.393	1.5 1.25	17 20
7/16	.437	14 20				
			M12	.472	1.75 1.25	14.5 20
½	.500	13 20				
			M14	.551	2 1.5	12.5 17
⅝	.625	11 18				
			M16	.630	2 1.5	12.5 17
			M18	.709	2.5 1.5	10 17
¾	.750	10 16				
			M20	.787	2.5 1.5	10 17
			M22	.866	2.5 1.5	10 17
⅞	.875	9 14				
			M24	.945	3 2	8.5 12.5
1"	1.000	8 12				
			M27	1.063	3 2	8.5 12.5

Inch-Metric Thread Comparison.



Nominal Size or Basic Major Dia. of Thread		F Width Across Flats			G Width Across Corners		H Thickness Hex Nuts			H_1 Thickness Hex Jam Nuts			Hex Nuts Specified Proof Load: Up to 150,000 psi	Hex Nuts Specified Proof Load: 150,000 psi and Greater	Jam Nuts All Strength Levels
		Basic	Max	Min	Max	Min	Basic	Max	Min	Basic	Max	Min	Runout of Bearing Face, FIR Max		
1/4	0.2500	7/16	0.438	0.428	0.505	0.488	7/32	0.226	0.212	5/32	0.163	0.150	0.015	0.010	0.015
5/16	0.3125	1/2	0.500	0.489	0.577	0.557	17/64	0.273	0.258	3/16	0.195	0.180	0.016	0.011	0.016
3/8	0.3750	9/16	0.562	0.551	0.650	0.628	21/64	0.337	0.320	7/32	0.227	0.210	0.017	0.012	0.017
7/16	0.4375	11/16	0.688	0.675	0.794	0.768	3/8	0.385	0.365	1/4	0.260	0.240	0.018	0.013	0.018
1/2	0.5000	3/4	0.750	0.736	0.866	0.840	7/16	0.448	0.427	5/16	0.323	0.302	0.019	0.014	0.019
9/16	0.5625	7/8	0.875	0.861	1.010	0.982	31/64	0.496	0.473	5/16	0.324	0.301	0.020	0.015	0.020
5/8	0.6250	15/16	0.938	0.922	1.083	1.051	35/64	0.559	0.535	3/8	0.387	0.363	0.021	0.016	0.021
3/4	0.7500	1 1/8	1.125	1.088	1.299	1.240	41/64	0.665	0.617	27/64	0.446	0.398	0.023	0.018	0.023
7/8	0.8750	1 5/16	1.312	1.269	1.516	1.447	3/4	0.776	0.724	31/64	0.510	0.458	0.025	0.020	0.025
1	1.0000	1 1/2	1.500	1.450	1.732	1.653	55/64	0.887	0.831	35/64	0.575	0.519	0.027	0.022	0.027
1 1/8	1.1250	1 11/16	1.688	1.631	1.949	1.859	31/32	0.999	0.939	39/64	0.639	0.579	0.030	0.025	0.030
1 1/4	1.2500	1 7/8	1.875	1.812	2.165	2.066	1 1/16	1.094	1.030	23/32	0.751	0.687	0.033	0.028	0.033
1 3/8	1.3750	2 1/16	2.062	1.994	2.382	2.273	1 11/64	1.206	1.138	25/32	0.815	0.747	0.036	0.031	0.036
1 1/2	1.5000	2 1/4	2.250	2.175	2.598	2.480	1 9/32	1.317	1.245	27/32	0.880	0.808	0.039	0.034	0.039
See Notes	9	3			4								2		

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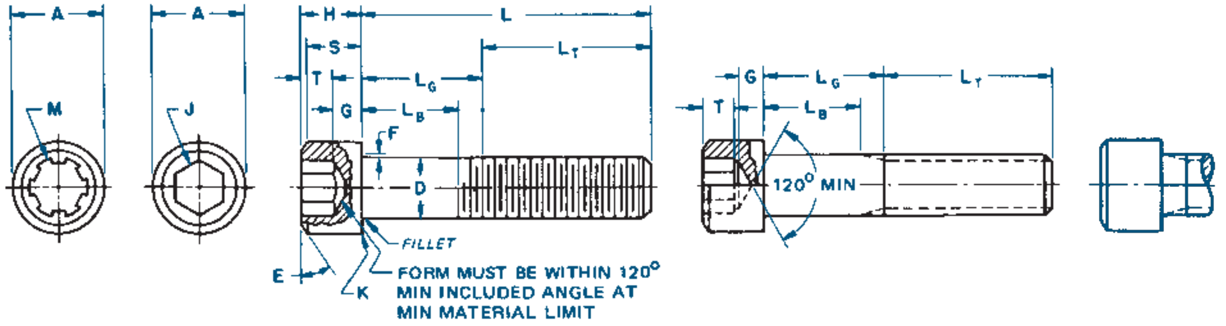
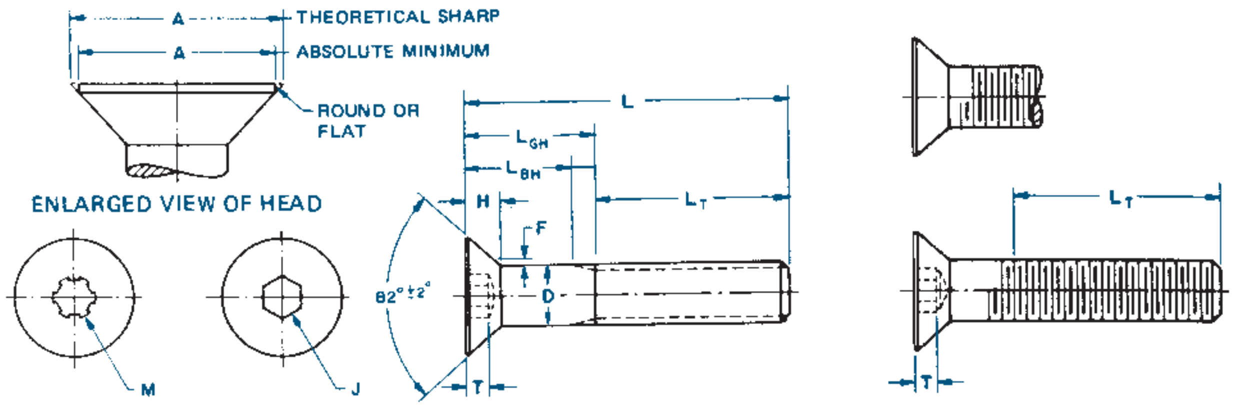
Inch-Metric Thread Comparison. (CONTINUED)



Nominal Size or Basic Screw Diameter		D Body Diameter		A Head Diameter		H Head Height		S Head Side Height	M Spline Socket Size	J Hexagon Socket Size	T Key Engagement	G Wall Thick-ness	K Chamfer or Radius
		Max	Min	Max	Min	Max	Min	Min	Nom	Nom	Min	Min	Max
0	0.0600	0.0600	0.0568	0.096	0.091	0.060	0.057	0.054	0.060	0.050	0.025	0.020	0.003
1	0.0730	0.0730	0.0695	0.118	0.112	0.073	0.070	0.066	0.072	1/16 0.062	0.031	0.025	0.003
2	0.0860	0.0860	0.0822	0.140	0.134	0.086	0.083	0.077	0.096	5/64 0.078	0.038	0.029	0.003
3	0.0990	0.0990	0.0949	0.161	0.154	0.099	0.095	0.089	0.096	5/64 0.078	0.044	0.034	0.003
4	0.1120	0.1120	0.1075	0.183	0.176	0.112	0.108	0.101	0.111	3/32 0.094	0.051	0.038	0.005
5	0.1250	0.1250	0.1202	0.205	0.198	0.125	0.121	0.112	0.111	3/32 0.094	0.057	0.043	0.005
6	0.1380	0.1380	0.1329	0.226	0.218	0.138	0.134	0.124	0.133	7/64 0.109	0.064	0.047	0.005
8	0.1640	0.1640	0.1585	0.270	0.262	0.164	0.159	0.148	0.168	9/64 0.141	0.077	0.056	0.005
10	0.1900	0.1900	0.1840	0.312	0.303	0.190	0.185	0.171	0.183	5/52 0.156	0.090	0.065	0.005
1/4	0.2500	0.2500	0.2435	0.375	0.365	0.250	0.244	0.225	0.216	3/16 0.188	0.120	0.095	0.008
5/16	0.3125	0.3125	0.3053	0.469	0.457	0.312	0.306	0.281	0.291	1/4 0.250	0.151	0.119	0.008
3/8	0.3750	0.3750	0.3678	0.562	0.550	0.375	0.368	0.337	0.372	5/16 0.312	0.182	0.143	0.008
7/16	0.4375	0.4375	0.4294	0.656	0.642	0.438	0.430	0.394	0.454	3/8 0.375	0.213	0.166	0.010
1/2	0.5000	0.5000	0.4919	0.750	0.735	0.500	0.492	0.450	0.454	3/8 0.375	0.245	0.190	0.010
5/8	0.6250	0.6250	0.6163	0.938	0.921	0.625	0.616	0.562	0.595	1/2 0.500	0.307	0.238	0.010
3/4	0.7500	0.7500	0.7406	1.125	1.107	0.750	0.740	0.675	0.620	5/8 0.625	0.370	0.285	0.010
7/8	0.8750	0.8750	0.8647	1.312	1.293	0.875	0.864	0.787	0.698	3/4 0.750	0.432	0.333	0.015
1	1.0000	1.0000	0.9886	1.500	1.479	1.000	0.988	0.900	0.790	3/4 0.750	0.495	0.380	0.015
1 1/8	1.1250	1.1250	1.1086	1.688	1.665	1.125	1.111	1.012	. . . .	7/8 0.875	0.557	0.428	0.015
1 1/4	1.2500	1.2500	1.2336	1.875	1.852	1.250	1.236	1.125	. . . .	7/8 0.875	0.620	0.475	0.015
1 3/8	1.3750	1.3750	1.3568	2.062	2.038	1.375	1.360	1.237	. . . .	1 1.000	0.682	0.523	0.015
1 1/2	1.5000	1.5000	1.4818	2.250	2.224	1.500	1.485	1.350	. . . .	1 1.000	0.745	0.570	0.015
1 3/4	1.7500	1.7500	1.7295	2.625	2.597	1.750	1.734	1.575	. . . .	1 1/4 1.250	0.870	0.665	0.015
2	2.0000	2.0000	1.9780	3.000	2.970	2.000	1.983	1.800	. . . .	1 1/2 1.500	0.995	0.760	0.015
2 1/4	2.2500	2.2500	2.2280	3.375	3.344	2.250	2.232	2.025	. . . .	1 3/4 1.750	1.120	0.855	0.031
2 1/2	2.5000	2.5000	2.4762	3.750	3.717	2.500	2.481	2.250	. . . .	1 3/4 1.750	1.245	0.950	0.031
2 3/4	2.7500	2.7500	2.7262	4.125	4.090	2.750	2.730	2.475	. . . .	2 2.000	1.370	1.045	0.031
3	3.0000	3.0000	2.9762	4.500	4.464	3.000	2.979	2.700	. . . .	2 1/4 2.250	1.495	1.140	0.031
3 1/4	3.2500	3.2500	3.2262	4.875	4.837	3.250	3.228	2.925	. . . .	2 1/4 2.250	1.620	1.235	0.031
3 1/2	3.5000	3.5000	3.4762	5.250	5.211	3.500	3.478	3.150	. . . .	2 3/4 2.750	1.745	1.330	0.031
3 3/4	3.7500	3.7500	3.7262	5.625	5.584	3.750	3.727	3.375	. . . .	2 3/4 2.750	1.870	1.425	0.031
4	4.0000	4.0000	3.9762	6.000	5.958	4.000	3.976	3.600	. . . .	3 3.000	1.995	1.520	0.031

Cap Screw Specifications.



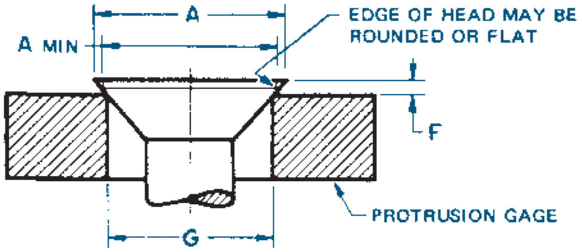
Nominal Size or Basic Screw Diameter		D		A		H		M	J	T	F
		Body Diameter		Head Diameter		Heat Height		Spline Socket Size	Hexagon Socket Size	Key Engagement	Fillet Extension Above D Max
				Theoretical			Flushness				
		Max	Min	Sharp Max	Abs. Min	Reference	Tolerance		Nom	Min	Max
0	0.0600	0.0600	0.0568	0.138	0.117	0.044	0.006	0.048	0.035	0.025	0.006
1	0.0730	0.0730	0.0695	0.168	0.143	0.054	0.007	0.060	0.050	0.031	0.008
2	0.0860	0.0860	0.0822	0.197	0.168	0.064	0.008	0.060	0.050	0.038	0.010
3	0.0990	0.0990	0.0949	0.226	0.193	0.073	0.010	0.072	1/16 0.062	0.044	0.010
4	0.1120	0.1120	0. 1075	0. 255	0.218	0.083	0.011	0.072	1/16 0.062	0.055	0.012
5	0.1250	0.1250	0.1202	0.281	0.240	0.090	0.012	0.096	5/64 0.078	0.061	0.014
6	0.1380	0.1380	0.1329	0.307	0.263	0.097	0.013	0.096	5/64 0.078	0.066	0.015
8	0.1640	0.1640	0.1585	0.359	0.311	0.112	0.014	0.111	3/32 0.094	0.076	0.015
10	0.1900	0.1900	0.1840	0.411	0.359	0.127	0.015	0.145	1/8 0.125	0.087	0.015
1/4	0.2500	0.2500	0.2435	0.531	0.480	0.161	0.016	0.183	5/32 0.156	0.111	0.015
5/16	0.3125	0.3125	0.3053	0.656	0.600	0.198	0.017	0.216	3/16 0.188	0.135	0.015
3/8	0.3750	0.3750	0.3678	0.781	0.720	0.234	0.018	0.251	7/32 0.219	0.159	0.015
7/16	0.4375	0.4375	0.4294	0.844	0.781	0.234	0.018	0.291	1/4 0.250	0.159	0.015
1/2	0.5000	0.5000	0.4919	0.938	0.872	0.251	0.018	0.372	5/16 0.312	0.172	0.015
5/8	0.6250	0.6250	0.6163	1.188	1.112	0.324	0.022	0.454	3/8 0.375	0.220	0.015
3/4	0.7500	0.7500	0.7406	1.438	1.355	0.396	0.024	0.454	1/2 0.500	0.220	0.015
7/8	0.8750	0.8750	0.8647	1.688	1.604	0.468	0.025	. . .	9/16 0.562	0.248	0.015
1	1.0000	1.0000	0.9886	1.938	1.841	0.540	0.028	. . .	5/8 0.625	0.297	0.015
1 1/8	1.1250	1.1250	1.1086	2.188	2.079	0.611	0.031	. . .	3/4 0.750	0.325	0.031
1 1/4	1.2500	1.2500	1.2336	2.438	2.316	0.683	0.035	. . .	7/8 0.875	0.358	0.031
1 3/8	1.3750	1.3750	1.3568	2.688	2.553	0.755	0.038	. . .	7/8 0.875	0.402	0.031
1 1/2	1.5000	1.5000	1.4818	2.938	2.791	0.827	0.042	. . .	1 1.000	0.435	0.031

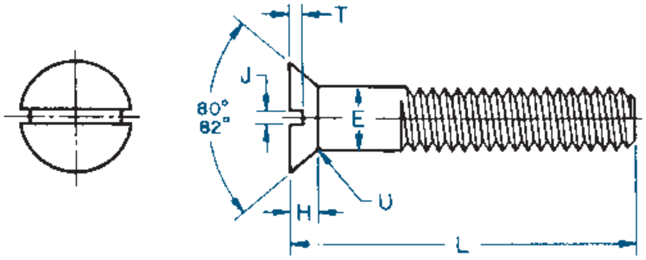
Cap Screw Specifications. (CONTINUED)

CAP SCREWS

FLAT

Type of Head





Nominal Size[1] or Basic Screw Diameter	E		A		H[2]	J		T		U	F[3]		G[3]
	Body Diameter		Head Diameter		Head Height	Slot Width		Slot Depth		Fillet Radius	Protrusion Above Gaging Diameter		Gaging Diameter
	Max	Min	Max, Edge Sharp	Min, Edge Rounded or Flat	Ref	Max	Min	Max	Min	Max	Max	Min	
1/4 0.2500	0.2500	0.2450	0.500	0.452	0.140	0.075	0.064	0.068	0.045	0.100	0.046	0.030	0.424
5/16 0.3125	0.3125	0.3070	0.625	0.567	0.177	0.084	0.072	0.086	0.057	0.125	0.053	0.035	0.538
3/8 0.3750	0.3750	0.3690	0.750	0.682	0.210	0.094	0.081	0.103	0.068	0.150	0.060	0.040	0.651
7/16 0.4375	0.4375	0.4310	0.812	0.736	0.210	0.094	0.081	0.103	0.068	0.175	0.065	0.044	0.703
1/2 0.5000	0.5000	0.4930	0.875	0.791	0.210	0.106	0.091	0.103	0.068	0.200	0.071	0.049	0.756
9/16 0.5625	0.5625	0.5550	1.000	0.906	0.244	0.118	0.102	0.120	0.080	0.225	0.078	0.054	0.869
5/8 0.6250	0.6250	0.6170	1.125	1.020	0.281	0.133	0.116	0.137	0.091	0.250	0.085	0.058	0.982
3/4 0.7500	0.7500	0.7420	1.375	1.251	0.352	0.149	0.131	0.171	0.115	0.300	0.099	0.068	1.208
7/8 0.8750	0.8750	0.8660	1.625	1.480	0.423	0.167	0.147	0.206	0.138	0.350	0.113	0.077	1.435
1 1.0000	1.0000	0.9900	1.875	1.711	0.494	0.188	0.166	0.240	0.162	0.400	0.127	0.087	1.661
1 1/8 1.1250	1.1250	1.1140	2.062	1.880	0.529	0.196	0.178	0.257	0.173	0.450	0.141	0.096	1.826
1 1/4 1.2500	1.2500	1.2390	2.312	2.110	0.600	0.211	0.193	0.291	0.197	0.500	0.155	0.105	2.052
1 3/8 1.3750	1.3750	1.3630	2.562	2.340	0.665	0.226	0.208	0.326	0.220	0.550	0.169	0.115	2.279
1 1/2 1.5000	1.5000	1.4880	2.812	2.570	0.742	0.258	0.240	0.360	0.244	0.600	0.183	0.124	2.505

1 Where specifying nominal size in decimals, zeros preceding decimal and in the fourth decimal place shall be omitted.
2 Tabulated values determined from formula for maximum H, Appendix III.
3 No tolerance for gaging diameter is given. If the gaging diameter of the gage used differs from tabulated value, the protrusion will be affected accordingly and the proper protrusion values must be recalculated using the formulas shown in Appendix II.

FOOTNOTES REFER TO ANSI B18.6.2–1972 (R1993).

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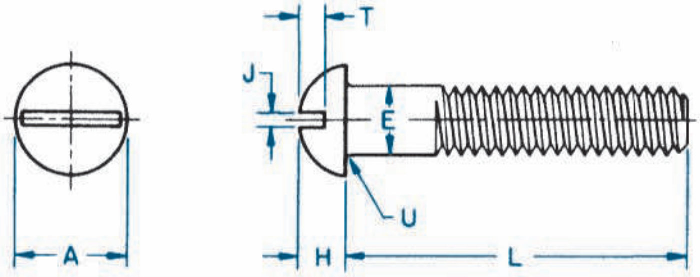
Cap Screw Specifications. (CONTINUED)

CAP SCREWS

ROUND

Type of Head



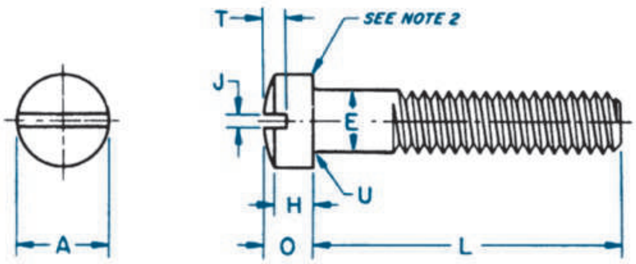
Nominal Size[1] or Basic Screw Diameter		E		A		H		J		T		U	
		Body Diameter		Head Diameter		Head Height		Slot Width		Slot Depth		Fillet Radius	
		Max	Min	Max	Min	Max	Min	Max	Min	Max	Min	Max	Min
1/4	0.2500	0.2500	0.2450	0.437	0.418	0.191	0.175	0.075	0.064	0.117	0.097	0.031	0.016
5/16	0.3125	0.3125	0.3070	0.562	0.540	0.245	0.226	0.084	0.072	0.151	0.126	0.031	0.016
3/8	0.3750	0.3750	0.3690	0.625	0.603	0.273	0.252	0.094	0.081	0.168	0.138	0.031	0.016
7/16	0.4375	0.4375	0.4310	0.750	0.725	0.328	0.302	0.094	0.081	0.202	0.167	0.047	0.016
1/2	0.5000	0.5000	0.4930	0.812	0.786	0.354	0.327	0.106	0.091	0.218	0.178	0.047	0.016
9/16	0.5625	0.5625	0.5550	0.937	0.909	0.409	0.378	0.118	0.102	0.252	0.207	0.047	0.016
5/8	0.6250	0.6250	0.6170	1.000	0.970	0.437	0.405	0.133	0.116	0.270	0.220	0.062	0.031
3/4	0.7500	0.7500	0.7420	1.250	1.215	0.546	0.507	0.149	0.131	0.338	0.278	0.062	0.031

1 Where specifying nominal size in decimals, zeros preceding decimal and in the fourth decimal place shall be omitted.

CAP SCREWS

FILLISTER

Type of Head

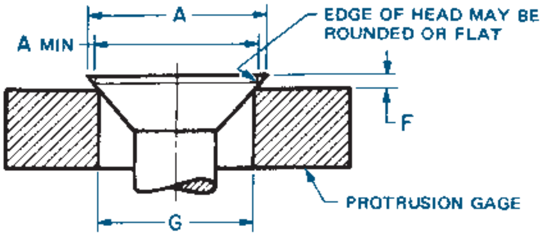


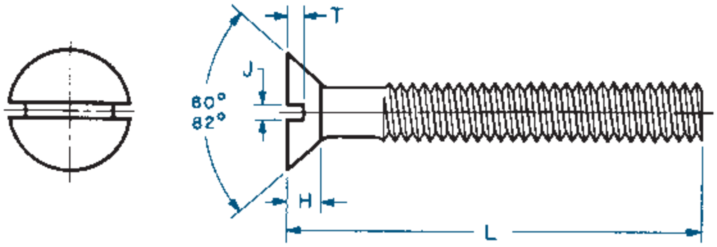
Nominal Size[1] or Basic Screw Diameter		E		A		H		O		J		T		U	
		Body Diameter		Head Diameter		Head Side Height		Total Head Height		Slot Width		Slot Depth		Fillet Radius	
		Max	Min	Max	Min	Max	Min	Max	Min	Max	Min	Max	Min	Max	Min
1/4	0.2500	0.2500	0.2450	0.375	0.363	0.172	0.157	0.216	0.194	0.075	0.064	0.097	0.077	0.031	0.016
5/16	0.3125	0.3125	0.3070	0.437	0.424	0.203	0.186	0.253	0.230	0.084	0.072	0.115	0.090	0.031	0.016
3/8	0.3750	0.3750	0.3690	0.562	0.547	0.250	0.229	0.314	0.284	0.094	0.081	0.142	0.112	0.031	0.016
7/16	0.4375	0.4375	0.4310	0.625	0.608	0.297	0.274	0.368	0.336	0.094	0.081	0.168	0.133	0.047	0.016
1/2	0.5000	0.5000	0.4930	0.750	0.731	0.328	0.301	0.413	0.376	0.106	0.091	0.193	0.153	0.047	0.016
9/16	0.5625	0.5625	0.5550	0.812	0.792	0.375	0.346	0.467	0.427	0.118	0.102	0.213	0.168	0.047	0.016
5/8	0.6250	0.6250	0.6170	0.875	0.853	0.422	0.391	0.521	0.478	0.133	0.116	0.239	0.189	0.062	0.031
3/4	0.7500	0.7500	0.7420	1.000	0.976	0.500	0.466	0.612	0.566	0.149	0.131	0.283	0.223	0.062	0.031
7/8	0.8750	0.8750	0.8660	1.125	1.098	0.594	0.556	0.720	0.668	0.167	0.147	0.334	0.264	0.062	0.031
1	1.0000	1.0000	0.9900	1.312	1.282	0.656	0.612	0.803	0.743	0.188	0.166	0.371	0.291	0.062	0.031

1 Where specifying nominal size in decimals, zeros preceding decimal and in the fourth decimal place shall be omitted.
2 A slight rounding of the edges at periphery of head shall be permissible provided the diameter of the bearing circle is equal to no less than 90 percent of the specified minimum head diameter.

Cap Screw Specifications. (CONTINUED)

Type of Head





Nominal Size[1] or Basic Screw Diameter		L[2] These Lengths or Shorter are Undercut	A Head Diameter Max. Edge Sharp	A Head Diameter Min, Edge Rounded or Flat	H[3] Head Height Ref	J Slot Width Max	J Slot Width Min	T Slot Depth Max	T Slot Depth Min	F[4] Protrusion Above Gaging Diameter Max	F[4] Protrusion Above Gaging Diameter Min	G[4] Gaging Diameter
0000	0.0210	—	0.043	0.037	0.011	0.008	0.004	0.007	0.003	*	*	*
000	0.0340	—	0.064	0.058	0.016	0.011	0.007	0.009	0.005	*	*	*
00	0.0470	—	0.093	0.085	0.028	0.017	0.010	0.014	0.009	*	*	*
0	0.0600	1/8	0.119	0.099	0.035	0.023	0.016	0.015	0.010	0.026	0.016	0.078
1	0.0730	1/8	0.146	0.123	0.043	0.026	0.019	0.019	0.012	0.028	0.016	0.101
2	0.0860	1/8	0.172	0.147	0.051	0.031	0.023	0.023	0.015	0.029	0.017	0.124
3	0.0990	1/8	0.199	0.171	0.059	0.035	0.027	0.027	0.017	0.031	0.018	0.148
4	0.1120	3/16	0.225	0.195	0.067	0.039	0.031	0.030	0.020	0.032	0.019	0.172
5	0.1250	3/16	0.252	0.220	0.075	0.043	0.035	0.034	0.022	0.034	0.020	0.196
6	0.1380	3/16	0.279	0.244	0.083	0.048	0.039	0.038	0.024	0.036	0.021	0.220
8	0.1640	1/4	0.332	0.292	0.100	0.054	0.045	0.045	0.029	0.039	0.023	0.267
10	0.1900	5/16	0.385	0.340	0.116	0.060	0.050	0.053	0.034	0.042	0.025	0.313
12	0.2160	3/8	0.438	0.389	0.132	0.067	0.056	0.060	0.039	0.045	0.027	0.362
1/4	0.2500	7/16	0.507	0.452	0.153	0.075	0.064	0.070	0.046	0.050	0.029	0.424
5/16	0.3125	1/2	0.635	0.568	0.191	0.084	0.072	0.088	0.058	0.057	0.034	0.539
3/8	0.3750	9/16	0.762	0.685	0.230	0.094	0.081	0.106	0.070	0.065	0.039	0.653
7/16	0.4375	5/8	0.812	0.723	0.223	0.094	0.081	0.103	0.066	0.073	0.044	0.690
1/2	0.5000	3/4	0.875	0.775	0.223	0.106	0.091	0.103	0.065	0.081	0.049	0.739
9/16	0.5625	—	1.000	0.889	0.260	0.118	0.102	0.120	0.077	0.089	0.053	0.851
5/8	0.6250	—	1.125	1.002	0.298	0.133	0.116	0.137	0.088	0.097	0.058	0.962
3/4	0.7500	—	1.375	1.230	0.372	0.149	0.131	0.171	0.111	0.112	0.067	1.186

[1] Where specifying nominal size in decimals, zeros preceding decimal and in the fourth decimal place shall be omitted.
[2] Screws of these lengths and shorter shall have undercut heads as shown in Table 5.
[3] Tabulated values determined from formula for maximum H, Appendix V.
[4] No tolerance for gaging diameter is given. If the gaging diameter of the gage used differs from tabulated value, the protrusion will be affected accordingly and the proper protrusion values must be recalculated using the formulas shown in Appendix I.
* Not practical to gage.

For additional requirements refer to General Data on Pages 3, 4 and 5.

FOOTNOTES REFER TO ANSI B18.6.3-1972 (R1991).

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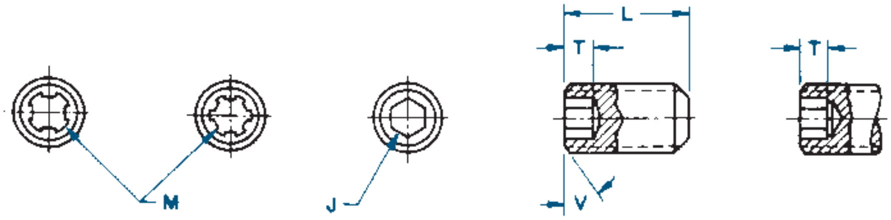
Machine Screw Specifications.



Nominal Size or Basic Screw Diameter		P		Q		B			B_1		
		Half Dog Point				Shortest Optimum Nominal Length To Which Column I_H Applies			Shortest Optimum Nominal Length To Which Column I_S Applies		
		Diameter		Length		Cup and Flat Points	90° Cone and Oval Points	Half Dog Points	Cup and Flat Points	90° Cone and Oval Points	Half Dog Point
		Max	Min	Max	Min						
0	0.0600	0.040	0.037	0.017	0.013	7/64	1/8	7/64	1/16	1/8	7/64
1	0.0730	0.049	0.045	0.021	0.017	1/8	9/64	1/8	3/32	9/64	1/8
2	0.0860	0.057	0.053	0.024	0.020	1/8	9/64	9/64	3/32	9/64	9/64
3	0.0990	0.066	0.062	0.027	0.023	9/64	5/32	5/32	3/32	5/32	5/32
4	0.1120	0.075	0.070	0.030	0.026	9/64	11/64	5/32	3/32	11/64	5/32
5	0.1250	0.083	0.078	0.033	0.027	3/16	3/16	11/64	1/8	3/16	11/64
6	0.1380	0.092	0.087	0.038	0.032	11/64	13/64	3/16	1/8	13/64	3/16
8	0.1640	0.109	0.103	0.043	0.037	3/16	7/32	13/64	3/16	7/32	13/64
10	0.1900	0.127	0.120	0.049	0.041	3/16	1/4	15/64	3/16	1/4	15/64
1/4	0.2500	0.156	0.149	0.067	0.059	1/4	5/16	19/64	1/4	5/16	19/64
5/16	0.3125	0.203	0.195	0.082	0.074	5/16	25/64	23/64	5/16	25/64	23/64
3/8	0.3750	0.250	0.241	0.099	0.089	3/8	7/16	7/16	3/8	7/16	7/16
7/16	0.4375	0.297	0.287	0.114	0.104	7/16	35/64	31/64	7/16	35/64	31/64
1/2	0.5000	0.344	0.334	0.130	0.120	1/2	39/64	35/64	1/2	39/64	35/64
5/8	0.6250	0.469	0.456	0.164	0.148	5/8	49/64	43/64	5/8	49/64	43/64
3/4	0.7500	0.562	0.549	0.196	0.180	3/4	29/32	51/64	3/4	29/32	51/64
7/8	0.8750	0.656	0.642	0.227	0.211	7/8	1 1/8	63/64	7/8	1 1/8	63/64
1	1.0000	0.750	0.734	0.260	0.240	1	1 17/64	1 1/8	. . .	. . .	. . .
1 1/8	1.1250	0.844	0.826	0.291	0.271	1 1/8	1 25/64	1 3/16	. . .	. . .	. . .
1 1/4	1.2500	0.938	0.920	0.323	0.303	1 1/4	1 1/2	1 5/16	. . .	. . .	. . .
1 3/8	1.3750	1.031	1.011	0.354	0.334	1 3/8	1 21/32	1 7/16	. . .	. . .	. . .
1 1/2	1.5000	1.125	1.105	0.385	0.365	1 1/2	1 51/64	1 9/16	. . .	. . .	. . .
1 3/4	1.7500	1.312	1.289	0.448	0.428	1 3/4	2 7/32	1 61/64	. . .	. . .	. . .
2	2.0000	1.500	1.474	0.510	0.490	2	2 25/64	2 5/64	. . .	. . .	. . .

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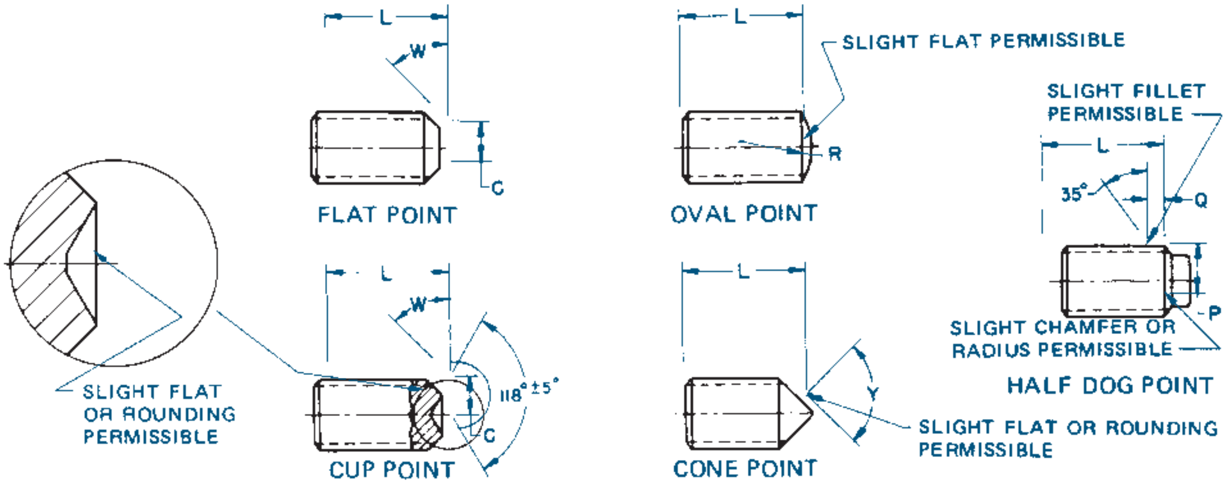
Set Screw Specifications.



Nominal Size or Basic Screw Diameter		P		Q		B			B_1		
		Half Dog Point				Shortest Optimum Nominal Length To Which Column T_H Applies			Shortest Optimum Nominal Length To Which Column T_S Applies		
		Diameter		Length		Cup and Flat Points	90° Cone and Oval Points	Half Dog Points	Cup and Flat Points	90° Cone and Oval Points	Half Dog Point
		Max	Min	Max	Min						
0	0.0600	0.040	0.037	0.017	0.013	7/64	1/8	7/64	1/16	1/8	7/64
1	0.0730	0.049	0.045	0.021	0.017	1/8	9/64	1/8	3/32	9/64	1/8
2	0.0860	0.057	0.053	0.024	0.020	1/8	9/64	9/64	3/32	9/64	9/64
3	0.0990	0.066	0.062	0.027	0.023	9/64	5/32	5/32	3/32	5/32	5/32
4	0.1120	0.075	0.070	0.030	0.026	9/64	11/64	5/32	3/32	11/64	5/32
5	0.1250	0.083	0.078	0.033	0.027	3/16	3/16	11/64	1/8	3/16	11/64
6	0.1380	0.092	0.087	0.038	0.032	11/64	13/64	3/16	1/8	13/64	3/16
8	0.1640	0.109	0.103	0.043	0.037	3/16	7/32	13/64	3/16	7/32	13/64
10	0.1900	0.127	0.120	0.049	0.041	3/16	1/4	15/64	3/16	1/4	15/64
1/4	0.2500	0.156	0.149	0.067	0.059	1/4	5/16	19/64	1/4	5/16	19/64
5/16	0.3125	0.203	0.195	0.082	0.074	5/16	25/64	23/64	5/16	25/64	23/64
3/8	0.3750	0.250	0.241	0.099	0.089	3/8	7/16	7/16	3/8	7/16	7/16
7/16	0.4375	0.297	0.287	0.114	0.104	7/16	35/64	31/64	7/16	35/64	31/64
1/2	0.5000	0.344	0.334	0.130	0.120	1/2	39/64	35/64	1/2	39/64	35/64
5/8	0.6250	0.469	0.456	0.164	0.148	5/8	49/64	43/64	5/8	49/64	43/64
3/4	0.7500	0.562	0.549	0.196	0.180	3/4	29/32	51/64	3/4	29/32	51/64
7/8	0.8750	0.656	0.642	0.227	0.211	7/8	1 1/8	63/64	7/8	1 1/8	63/64
1	1.0000	0.750	0.734	0.260	0.240	1	1 17/64	1 1/8	...	...	...
1 1/8	1.1250	0.844	0.826	0.291	0.271	1 1/8	1 25/64	1 3/16	...	...	...
1 1/4	1.2500	0.938	0.920	0.323	0.303	1 1/4	1 1/2	1 5/16	...	...	...
1 3/8	1.3750	1.031	1.011	0.354	0.334	1 3/8	1 21/32	1 7/16	...	...	...
1 1/2	1.5000	1.125	1.105	0.385	0.365	1 1/2	1 51/64	1 9/16	...	...	...
1 3/4	1.7500	1.312	1.289	0.448	0.428	1 3/4	2 7/32	1 61/64	...	...	...
2	2.0000	1.500	1.474	0.510	0.490	2	2 25/64	2 5/64	...	...	...

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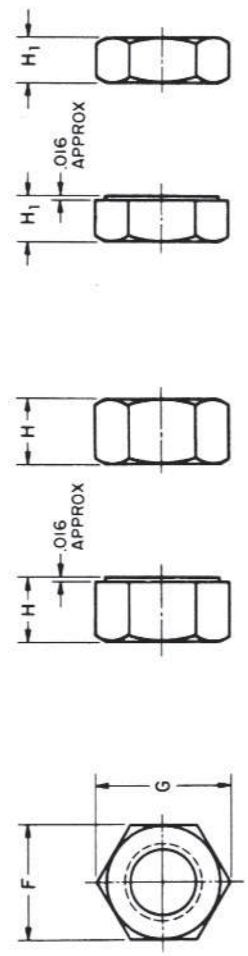
Set Screw Specifications. (CONTINUED)



	F			G		H			H_1					
Nominal Size or Basic Major Dia of Thread	Width Across Flats			Width Across Corners		Thickness Hex Nuts			Thickness Hex Jam Nuts			Hex Nuts Specified Proof Load: Up to 150,000 psi	Hex Nuts Specified Proof Load: 150,000 psi and Greater	Jam Nuts All Strength Levels
	Basic	Max	Min	Max	Min	Basic	Max	Min	Basic	Max	Min	Runout of Bearing Face, FIR Max		
1/4 0.2500	7/16	0.438	0.428	0.505	0.488	7/32	0.226	0.212	5/32	0.163	0.150	0.015	0.010	0.015
5/16 0.3125	1/2	0.500	0.489	0.577	0.557	17/64	0.273	0.258	3/16	0.195	0.180	0.016	0.011	0.016
3/8 0.3750	9/16	0.562	0.551	0.650	0.628	21/64	0.337	0.320	7/32	0.227	0.210	0.017	0.012	0.017
7/16 0.4375	11/16	0.688	0.675	0.794	0.768	3/8	0.385	0.365	1/4	0.260	0.240	0.018	0.013	0.018
1/2 0.5000	3/4	0.750	0.736	0.866	0.840	7/16	0.448	0.427	5/16	0.323	0.302	0.019	0.014	0.019
9/16 0.5625	7/8	0.875	0.861	1.010	0.982	31/64	0.496	0.473	5/16	0.324	0.301	0.020	0.015	0.020
5/8 0.6250	15/16	0.938	0.922	1.083	1.051	35/64	0.559	0.535	3/8	0.387	0.363	0.021	0.016	0.021
3/4 0.7500	1 1/8	1.125	1.088	1.299	1.240	41/64	0.665	0.617	27/64	0.446	0.398	0.023	0.018	0.023
7/8 0.8750	1 5/16	1.312	1.269	1.516	1.447	3/4	0.776	0.724	31/64	0.510	0.458	0.025	0.020	0.025
1 1.0000	1 1/2	1.500	1.450	1.732	1.653	55/64	0.887	0.831	35/64	0.575	0.519	0.027	0.022	0.027
1 1/8 1.1250	1 11/16	1.688	1.631	1.949	1.859	31/32	0.999	0.939	39/64	0.639	0.579	0.030	0.025	0.030
1 1/4 1.2500	1 7/8	1.875	1.812	2.165	2.066	1 1/16	1.094	1.030	23/32	0.751	0.687	0.033	0.028	0.033
1 3/8 1.3750	2 1/16	2.062	1.994	2.382	2.273	1 11/64	1.206	1.138	25/32	0.815	0.747	0.036	0.031	0.036
1 1/2 1.5000	2 1/4	2.250	2.175	2.598	2.480	1 9/32	1.317	1.245	27/32	0.880	0.808	0.039	0.034	0.039

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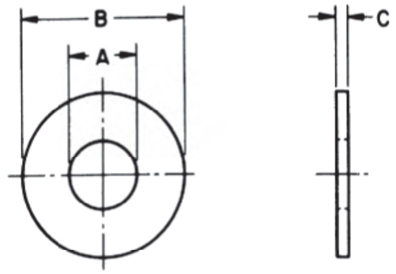
Hex Nut Specifications.

AMERICAN STANDARD



DIMENSIONS OF PREFERRED SIZES OF TYPE A PLAIN WASHERS **

Nominal Washer Size***			Inside Diameter A			Outside Diameter B			Thickness C		
			Basic	Tolerance		Basic	Tolerance		Basic	Max	Min
				Plus	Minus		Plus	Minus			
—	—		0.078	0.000	0.005	0.188	0.000	0.005	0.020	0.025	0.016
—	—		0.094	0.000	0.005	0.250	0.000	0.005	0.020	0.025	0.016
—	—		0.125	0.008	0.005	0.312	0.008	0.005	0.032	0.040	0.025
No. 6	0.138		0.156	0.008	0.005	0.375	0.015	0.005	0.049	0.065	0.036
No. 8	0.164		0.188	0.008	0.005	0.438	0.015	0.005	0.049	0.065	0.036
No. 10	0.190		0.219	0.008	0.005	0.500	0.015	0.005	0.049	0.065	0.036
3/16	0.188		0.250	0.015	0.005	0.562	0.015	0.005	0.049	0.065	0.036
No. 12	0.216		0.250	0.015	0.005	0.562	0.015	0.005	0.065	0.080	0.051
1/4	0.250	N	0.281	0.015	0.005	0.625	0.015	0.005	0.065	0.080	0.051
1/4	0.250	W	0.312	0.015	0.005	0.734*	0.015	0.007	0.065	0.080	0.051
5/16	0.312	N	0.344	0.015	0.005	0.688	0.015	0.007	0.065	0.080	0.051
5/16	0.312	W	0.375	0.015	0.005	0.875	0.030	0.007	0.083	0.104	0.064
3/8	0.375	N	0.406	0.015	0.005	0.812	0.015	0.007	0.065	0.080	0.051
3/8	0.375	W	0.438	0.015	0.005	1.000	0.030	0.007	0.083	0.104	0.064
7/16	0.438	N	0.469	0.015	0.005	0.922	0.015	0.007	0.065	0.080	0.051
7/16	0.438	W	0.500	0.015	0.005	1.250	0.030	0.007	0.083	0.104	0.064
1/2	0.500	N	0.531	0.015	0.005	1.062	0.030	0.007	0.095	0.121	0.074
1/2	0.500	W	0.562	0.015	0.005	1.375	0.030	0.007	0.109	0.132	0.086
9/16	0.562	N	0.594	0.015	0.005	1.156*	0.030	0.007	0.095	0.121	0.074
9/16	0.562	W	0.625	0.015	0.005	1.469*	0.030	0.007	0.109	0.132	0.086
5/8	0.625	N	0.656	0.030	0.007	1.312	0.030	0.007	0.095	0.121	0.074
5/8	0.625	W	0.688	0.030	0.007	1.750	0.030	0.007	0.134	0.160	0.108
3/4	0.750	N	0.812	0.030	0.007	1.469	0.030	0.007	0.134	0.160	0.108
3/4	0.750	W	0.812	0.030	0.007	2.000	0.030	0.007	0.148	0.177	0.122
7/8	0.875	N	0.938	0.030	0.007	1.750	0.030	0.007	0.134	0.160	0.108
7/8	0.875	W	0.938	0.030	0.007	2.250	0.030	0.007	0.165	0.192	0.136
1	1.000	N	1.062	0.030	0.007	2.000	0.030	0.007	0.134	0.160	0.108
1	1.000	W	1.062	0.030	0.007	2.500	0.030	0.007	0.165	0.192	0.136
1 1/8	1.125	N	1.250	0.030	0.007	2.250	0.030	0.007	0.134	0.160	0.108
1 1/8	1.125	W	1.250	0.030	0.007	2.750	0.030	0.007	0.165	0.192	0.136
1 1/4	1.250	N	1.375	0.030	0.007	2.500	0.030	0.007	0.165	0.192	0.136
1 1/4	1.250	W	1.375	0.030	0.007	3.000	0.030	0.007	0.165	0.192	0.136
1 3/8	1.375	N	1.500	0.030	0.007	2.750	0.030	0.007	0.165	0.192	0.136
1 3/8	1.375	W	1.500	0.045	0.010	3.250	0.045	0.010	0.180	0.213	0.153
1 1/2	1.500	N	1.625	0.030	0.007	3.000	0.030	0.007	0.165	0.192	0.136
1 1/2	1.500	W	1.625	0.045	0.010	3.500	0.045	0.010	0.180	0.213	0.153
1 5/8	1.625		1.750	0.045	0.010	3.750	0.045	0.010	0.180	0.213	0.153
1 3/4	1.750		1.875	0.045	0.010	4.000	0.045	0.010	0.180	0.213	0.153
1 7/8	1.875		2.000	0.045	0.010	4.250	0.045	0.010	0.180	0.213	0.153
2	2.000		2.125	0.045	0.010	4.500	0.045	0.010	0.180	0.213	0.153
2 1/4	2.250		2.375	0.045	0.010	4.750	0.045	0.010	0.220	0.248	0.193
2 1/2	2.500		2.625	0.045	0.010	5.000	0.045	0.010	0.238	0.280	0.210
2 3/4	2.750		2.875	0.065	0.010	5.250	0.065	0.010	0.259	0.310	0.228
3	3.000		3.125	0.065	0.010	5.500	0.065	0.010	0.284	0.327	0.249

*The 0.734 in., 1.156 in., and 1.469 in. outside diameters avoid washers which could be used in coin operated devices.

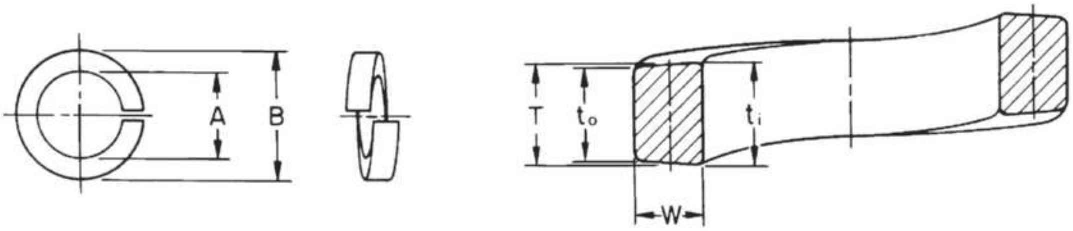
**Preferred sizes are for the most part from series previously designated "Standard Plate" and "SAE." Where common sizes existed in the two series, the SAE size is designated "N" (narrow) and the Standard Plate "W" (wide). These sizes as well as all other sizes of Type A Plain Washers are to be ordered by ID, OD, and thickness dimensions.

***Nominal washer sizes are intended for use with comparable nominal screw or bolt sizes.

Dimensions of Preferred Sizes of Type A Plain Washers.

AMERICAN NATIONAL STANDARD
LOCK WASHERS

ANSI B18.21.1-1990



ENLARGED SECTION

DIMENSIONS OF REGULAR HELICAL SPRING LOCK WASHERS[1]

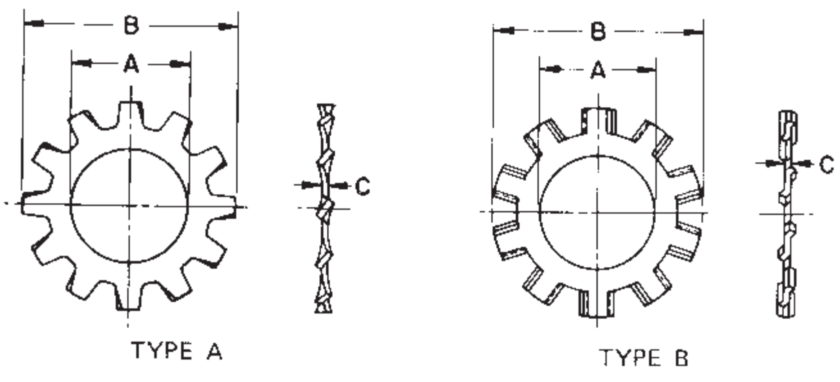
Nominal Washer Size		A Inside Diameter		B Outside Diameter	T Mean Section Thickness $\left(\frac{t_i + t_o}{2}\right)$	W Section Width
		Max	Min	Max[2]	Min	Min
No. 4	0.112	0.120	0.114	0.173	0.022	0.022
No. 5	0.125	0.133	0.127	0.202	0.030	0.030
No. 6	0.138	0.148	0.141	0.216	0.030	0.030
No. 8	0.164	0.174	0.167	0.267	0.047	0.042
No. 10	0.190	0.200	0.193	0.294	0.047	0.042
1/4	0.250	0.262	0.254	0.365	0.078	0.047
5/16	0.312	0.326	0.317	0.460	0.093	0.062
3/8	0.375	0.390	0.380	0.553	0.125	0.076
7/16	0.438	0.455	0.443	0.647	0.140	0.090
1/2	0.500	0.518	0.506	0.737	0.172	0.103
5/8	0.625	0.650	0.635	0.923	0.203	0.125
3/4	0.750	0.775	0.760	1.111	0.218	0.154
7/8	0.875	0.905	0.887	1.296	0.234	0.182
1	1.000	1.042	1.017	1.483	0.250	0.208
1 1/8	1.125	1.172	1.144	1.669	0.313	0.236
1 1/4	1.250	1.302	1.271	1.799	0.313	0.236
1 3/8	1.375	1.432	1.398	2.041	0.375	0.292
1 1/2	1.500	1.561	1.525	2.170	0.375	0.292
1 3/4	1.750	1.811	1.775	2.602	0.469	0.383
2	2.000	2.061	2.025	2.852	0.469	0.383
2 1/4	2.250	2.311	2.275	3.352	0.508	0.508
2 1/2	2.500	2.561	2.525	3.602	0.508	0.508
2 3/4	2.750	2.811	2.775	4.102	0.633	0.633
3	3.000	3.061	3.025	4.352	0.633	0.633

[1]For use with 1960 Series Socket Head Cap Screws specified in American National Standard, ANSI B18.3.
[2]The maximum outside diameters specified allow for the commercial tolerances on cold-drawn wire.

Dimensions of Spring Lock Washers.

AMERICAN NATIONAL STANDARD
LOCK WASHERS

ASME/ANSI B18.21.1-1994



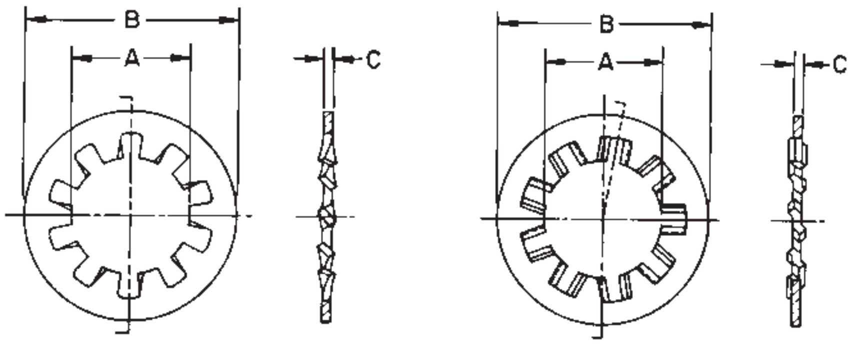
DIMENSIONS OF EXTERNAL TOOTH LOCK WASHERS

Nominal Washer Size		*A* Inside Diameter		*B* Outside Diameter		*C* Thickness	
		Max	Min	Max	Min	Max	Min
No. 3	0.099	0.109	0.102	0.235	0.220	0.015	0.012
No. 4	0.112	0.123	0.115	0.260	0.245	0.019	0.015
No. 5	0.125	0.136	0.129	0.285	0.270	0.019	0.014
No. 6	0.138	0.150	0.141	0.320	0.305	0.022	0.016
No. 8	0.164	0.176	0.168	0.381	0.365	0.023	0.018
No. 10	0.190	0.204	0.195	0.410	0.395	0.025	0.020
No. 12	0.216	0.231	0.221	0.475	0.460	0.028	0.023
1/4	0.250	0.267	0.256	0.510	0.494	0.028	0.023
5/16	0.312	0.332	0.320	0.610	0.588	0.034	0.028
3/8	0.375	0.398	0.384	0.694	0.670	0.040	0.032
7/16	0.438	0.464	0.448	0.760	0.740	0.040	0.032
1/2	0.500	0.530	0.514	0.900	0.880	0.045	0.037
9/16	0.562	0.596	0.576	0.985	0.960	0.045	0.037
5/8	0.625	0.663	0.641	1.070	1.045	0.050	0.042
11/16	0.688	0.728	0.704	1.155	1.130	0.050	0.042
3/4	0.750	0.795	0.768	1.260	1.220	0.055	0.047
13/16	0.812	0.861	0.833	1.315	1.290	0.055	0.047
7/8	0.875	0.927	0.897	1.410	1.380	0.060	0.052
1	1.000	1.060	1.025	1.620	1.590	0.067	0.059

Dimensions of Internal and External Tooth Lock Washers.

AMERICAN NATIONAL STANDARD
LOCK WASHERS

ASME/ANSI B18.21.1-1994



TYPE A TYPE B

DIMENSIONS OF INTERNAL TOOTH LOCK WASHERS

Nominal Washer Size		A Inside Diameter		B Outside Diameter		C Thickness	
		Max	Min	Max	Min	Max	Min
No. 2	0.086	0.095	0.089	0.200	0.175	0.015	0.010
No. 3	0.099	0.109	0.102	0.232	0.215	0.019	0.012
No. 4	0.112	0.123	0.115	0.270	0.255	0.019	0.015
No. 5	0.125	0.136	0.129	0.280	0.245	0.021	0.017
No. 6	0.138	0.150	0.141	0.295	0.275	0.021	0.017
No. 8	0.164	0.176	0.168	0.340	0.325	0.023	0.018
No. 10	0.190	0.204	0.195	0.381	0.365	0.025	0.020
No. 12	0.216	0.231	0.221	0.410	0.394	0.025	0.020
1/4	0.250	0.267	0.256	0.478	0.460	0.028	0.023
5/16	0.312	0.332	0.320	0.610	0.594	0.034	0.028
3/8	0.375	0.398	0.384	0.692	0.670	0.040	0.032
7/16	0.438	0.464	0.448	0.789	0.740	0.040	0.032
1/2	0.500	0.530	0.512	0.900	0.867	0.045	0.037
9/16	0.562	0.596	0.576	0.985	0.957	0.045	0.037
5/8	0.625	0.663	0.640	1.071	1.045	0.050	0.042
11/16	0.688	0.728	0.704	1.166	1.130	0.050	0.042
3/4	0.750	0.795	0.769	1.245	1.220	0.055	0.047
13/16	0.812	0.861	0.832	1.315	1.290	0.055	0.047
7/8	0.875	0.927	0.894	1.410	1.364	0.060	0.052
1	1.000	1.060	1.019	1.637	1.590	0.067	0.059
1 1/8	1.125	1.192	1.144	1.830	1.799	0.067	0.059
1 1/4	1.250	1.325	1.275	1.975	1.921	0.067	0.059

Dimensions of Internal and External Tooth Lock Washers. (CONTINUED)

Fraction or Drill Size	Decimal Equivalent	Tap Size
80 (Number Size Drills)	.0135	
79	.0145	
1/64	.0156	
78	.0160	
77	.0180	
76	.0200	
75	.0210	
74	.0225	
73	.0240	
72	.0250	
71	.0260	
70	.0280	
69	.0292	
68	.0310	
1/32	.0312	
67	.0320	
66	.0330	
65	.0350	
64	.0360	
63	.0370	
62	.0380	
61	.0390	
60	.0400	
59	.0410	
58	.0420	
57	.0430	
56	.0465	
3/64	.0469	0–80
55	.0520	
54	.0550	1–56
53	.0595	1–64, 72
1/16	.0625	
52	.0635	
51	.0670	
50	.0700	2–56, 64
49	.0730	
48	.0760	
5/64	.0781	
47	.0785	3–48
46	.0810	
45	.0820	3–56, 4–32
44	.0860	4–36
43	.0890	4–40
42	.0935	4–48
3/32	.0938	
41	.0960	
40	.0980	

Fraction or Drill Size	Decimal Equivalent	Tap Size
39	.0995	
38	.1015	5–40
37	.1040	5–44
36	.1065	6–32
7/64	.1094	
35	.1100	
34	.1110	6–36
33	.1130	6–40
32	.1160	
31	.1200	
1/8	.1250	
30	.1285	
29	.1360	8–32,36
28	.1405	8–40
9/64	.1406	
27	.1440	
26	.1470	
25	.1495	10–24
24	.1520	
23	.1540	
5/32	.1562	
22	.1570	10–30
21	.1590	10–32
20	.1610	
19	.1660	
18	.1695	
11/64	.1719	
17	.1730	
16	.1770	12–24
15	.1800	
14	.1820	12–28
13	.1850	12–32
3/16	.1875	
12	.1890	
11	.1910	
10	.1935	
9	.1960	
8	.1990	
7	.2010	1/4–20
13/64	.2031	
6	.2040	
5	.2055	
4	.2090	
3	.2130	1/4–28
7/32	.2188	
2	.2210	
1	.2280	
A (Letter Size Drills)	.2340	

Fraction or Drill Size	Decimal Equivalent	Tap Size
15/64	.2344	
B (Letter Size Drills)	.2380	
C	.2420	
D	.2460	
1/4 — E	.2500	
F	.2570	5/16–18
G	2610	
17/64	.2656	
H	.2660	
I	.2720	5/16–24
J	.2770	
K	.2810	
9/32	.2812	
L	.2900	
M	.2950	
19/64	.2969	
N	.3020	
5/16	.3125	3/8–16
O	.3160	
P	.3230	
21/64	.3281	
Q	.3320	3/8–24
R	.3390	
11/32	.3438	
S	.3480	
T	.3580	
23/64	.3594	
U	.3680	7/16–14
3/8	.3750	
V	.3770	
W	.3860	
25/64	.3906	7/16–20
X	.3970	
Y	.4040	
13/32	.4062	
Z	.4130	
27/64	.4219	1/2–13
7/16	.4375	
29/64	.4531	1/2–20
15/32	.4688	
31/64	.4844	9/16–12
1/2	.5000	
33/64	.5156	9/16–18
17/32	.5312	5/8–11
35/64	.5469	
9/16	.5625	
37/64	.5781	5/8–18

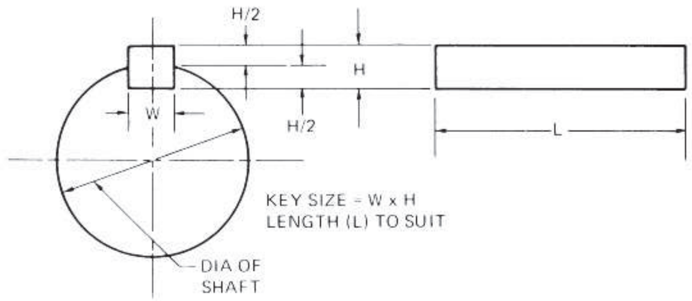
Fraction or Drill Size	Decimal Equivalent	Tap Size
19/32	.5938	11/16–11
39/64	.6094	
5/8	.6250	11/16–16
41/64	.6406	
21/32	.6562	3/4–10
43/64	.6719	
11/16	.6875	3/4–16
45/64	.7031	
23/32	.7188	
47/64	.7344	
3/4	.7500	
49/64	.7656	7/8–9
25/32	.7812	
51/64	.7969	
13/16	.8125	7/8–14
53/64	.8281	
27/32	.8438	
55/64	.8594	
7/8	.8750	1–8
57/64	.8906	
29/32	.9062	
59/64	.9219	
15/16	.9375	1–12, 14
61/64	.9531	
31/32	.9688	
63/64	.9844	1 1/8–7
1	1.0000	
1 3/64	1.0469	1 1/8–12
1 7/64	1.1094	1 1/4–7
1 1/8	1.1250	
1 11/64	1.1719	1 1/4–12
1 7/32	1.2188	1 3/8–6
1 1/4	1.2500	
1 19/64	1.2969	1 3/8–12
1 11/32	1.3438	1 1/2–6
1 3/8	1.3750	
1 27/64	1.4219	1 1/2–12
1 1/2	1.5000	

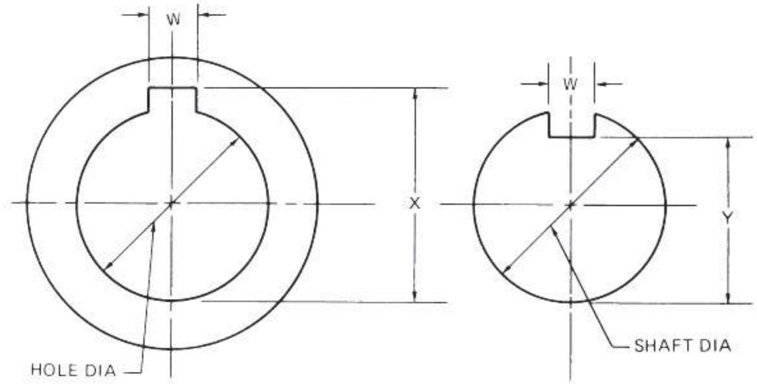
Pipe Thread Sizes

Thread	Drill	Thread	Drill
1/8–27	R	1 1/2–11 1/2	1 47/64
1/4–18	7/16	2–11 1/2	2 7/32
3/8–18	37/64	2 1/2–8	2 5/8
1/2–14	23/32	3–8	3 1/4
3/4–14	59/64	3 1/2–8	3 3/4
1–11 1/2	1 5/32	4–8	4 1/4
1 1/4–11 1/2	1 1/2		

Tap Drill Sizes.

KEY & KEYWAY SIZES





Shaft Nom. Size – DIA. – From	Shaft Nom. Size – DIA. – To & Incl.	Square (W = H)	Type	Square Key From	Square Key To & Incl.	Tolerance
5/16 (8)	7/16 (11)	3/32 (2.38)	Bar Stock	— —	3/4 (19.05)	+.000 - .002 (+.0000 - .0254)
7/16 (11)	9/16 (14)	1/8 (3.175)	Bar Stock	3/4 (19.05)	1 1/2 (38.1)	+.000 - .003 (+.0000 - .0762)
9/16 (14)	7/8 (22)	3/16 (4.76)	Bar Stock	1 1/2 (38.1)	2 1/2 (63.5)	+.000 - .004 (+.0000 - .1016)
7/8 (22)	1 1/4 (32)	1/4 (6.35)	Bar Stock	2 1/2 (63.5)	3 1/2 (88.9)	+.000 - .006 (+.0000 - .1524)
1 1/4 (32)	1 3/8 (35)	5/16 (7.94)	Keystock	— —	1 1/4 (31.75)	+.001 - .000 (+.0254 - .0000)
1 3/8 (35)	1 3/4 (44)	3/8 (9.53)	Keystock	1 1/4 (31.75)	3 (76.2)	+.002 - .000 (+.0508 - .0000)
1 3/4 (44)	2 1/4 (57)	1/2 (12.7)	Keystock	3 (76.2)	3 1/2 (88.9)	+.003 - .000 (+.0762 - .0000)
2 1/4 (57)	2 3/4 (70)	5/8 (15.88)				
2 3/4 (70)	3 1/4 (82)	3/4 (19.05)				
3 1/4 (82)	3 3/4 (95)	7/8 (22.23)				

(Figures in parenthesis = mm)

Nom. Size	– DIA. – (Shaft)		'X' (Collar)		'Y' (Shaft)	
(Inch)	Inch	mm	Inch	mm	Inch	mm
1/2	.500	12.700	.560	14.224	.430	10.922
9/16	.562	14.290	.623	15.824	.493	12.522
5/8	.625	15.875	.709	18.008	.517	13.132
11/16	.688	17.470	.773	18.618	.581	14.757
3/4	.750	19.050	.837	21.259	.644	16.357
13/16	.812	20.640	.900	22.860	.708	17.983
7/8	.875	22.225	.964	24.485	.771	19.583
15/16	.938	23.820	1.051	26.695	.791	20.091
1	1.000	25.400	1.114	28.295	.859	21.818
1 1/16	1.062	26.985	1.178	29.921	.923	23.444
1 1/8	1.125	28.575	1.241	31.521	.986	25.044
1 3/16	1.188	30.165	1.304	33.121	1.049	26.644
1 1/4	1.250	31.750	1.367	34.722	1.112	28.244
1 5/16	1.312	33.340	1.455	36.957	1.137	28.879
1 3/8	1.375	34.923	1.518	38.557	1.201	30.505

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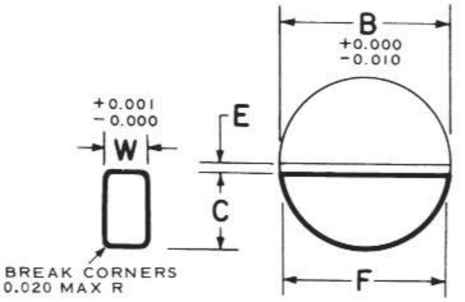
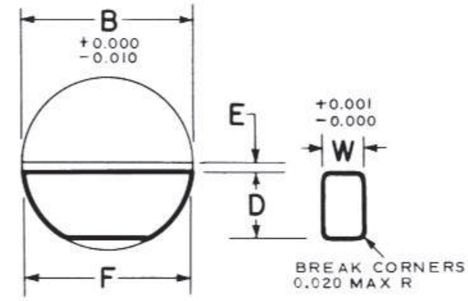
Dimensions of Keys and Slots.

USA STANDARD



FULL RADIUS TYPE



FLAT BOTTOM TYPE

WOODRUFF KEYS

Key No.	Nominal Key Size W × B	Actual Length F +0.000-0.010	Height of Key C Max	Height of Key C Min	Height of Key D Max	Height of Key D Min	Distance Below Center E
202	1/16 × 1/4	0.248	0.109	0.104	0.109	0.104	1/64
202.5	1/16 × 5/16	0.311	0.140	0.135	0.140	0.135	1/64
302.5	3/32 × 5/16	0.311	0.140	0.135	0.140	0.135	1/64
203	1/16 × 3/8	0.374	0.172	0.167	0.172	0.167	1/64
303	3/32 × 3/8	0.374	0.172	0.167	0.172	0.167	1/64
403	1/8 × 3/8	0.374	0.172	0.167	0.172	0.167	1/64
204	1/16 × 1/2	0.491	0.203	0.198	0.194	0.188	3/64
304	3/32 × 1/2	0.491	0.203	0.198	0.194	0.188	3/64
404	1/8 × 1/2	0.491	0.203	0.198	0.194	0.188	3/64
305	3/32 × 5/8	0.612	0.250	0.245	0.240	0.234	1/16
405	1/8 × 5/8	0.612	0.250	0.245	0.240	0.234	1/16
505	5/32 × 5/8	0.612	0.250	0.245	0.240	0.234	1/16
605	3/16 × 5/8	0.612	0.250	0.245	0.240	0.234	1/16
406	1/8 × 3/4	0.740	0.313	0.308	0.303	0.297	1/16
506	5/32 × 3/4	0.740	0.313	0.308	0.303	0.297	1/16
606	3/16 × 3/4	0.740	0.313	0.308	0.303	0.297	1/16
806	1/4 × 3/4	0.740	0.313	0.308	0.303	0.297	1/16
507	5/32 × 7/8	0.866	0.375	0.370	0.365	0.359	1/16
607	3/16 × 7/8	0.866	0.375	0.370	0.365	0.359	1/16
707	7/32 × 7/8	0.866	0.375	0.370	0.365	0.359	1/16
807	1/4 × 7/8	0.866	0.375	0.370	0.365	0.359	1/16
608	3/16 × 1	0.992	0.438	0.433	0.428	0.422	1/16
708	7/32 × 1	0.992	0.438	0.433	0.428	0.422	1/16
808	1/4 × 1	0.992	0.438	0.433	0.428	0.422	1/16
1008	5/16 × 1	0.992	0.438	0.433	0.428	0.422	1/16
1208	3/8 × 1	0.992	0.438	0.433	0.428	0.422	1/16
609	3/16 × 1 1/8	1.114	0.484	0.479	0.475	0.469	5/64
709	7/32 × 1 1/8	1.114	0.484	0.479	0.475	0.469	5/64
809	1/4 × 1 1/8	1.114	0.484	0.479	0.475	0.469	5/64
1009	5/16 × 1 1/8	1.114	0.484	0.479	0.475	0.469	5/64

Dimensions of Keys and Slots. (CONTINUED)

WOODRUFF KEYS (CONCLUDED)

Key No.	Nominal Key Size W × B	Actual Length F +0.000-0.010	Height of Key				Distance Below Center E
			C		D		
			Max	Min	Max	Min	
610	$\frac{3}{16} \times 1\frac{1}{4}$	1.240	0.547	0.542	0.537	0.531	$\frac{5}{64}$
710	$\frac{7}{32} \times 1\frac{1}{4}$	1.240	0.547	0.542	0.537	0.531	$\frac{5}{64}$
810	$\frac{1}{4} \times 1\frac{1}{4}$	1.240	0.547	0.542	0.537	0.531	$\frac{5}{64}$
1010	$\frac{5}{16} \times 1\frac{1}{4}$	1.240	0.547	0.542	0.537	0.531	$\frac{5}{64}$
1210	$\frac{3}{8} \times 1\frac{1}{4}$	1.240	0.547	0.542	0.537	0.531	$\frac{5}{64}$
811	$\frac{1}{4} \times 1\frac{3}{8}$	1.362	0.594	0.589	0.584	0.578	$\frac{3}{32}$
1011	$\frac{5}{16} \times 1\frac{3}{8}$	1.362	0.594	0.589	0.584	0.578	$\frac{3}{32}$
1211	$\frac{3}{8} \times 1\frac{3}{8}$	1.362	0.594	0.589	0.584	0.578	$\frac{3}{32}$
812	$\frac{1}{4} \times 1\frac{1}{2}$	1.484	0.641	0.636	0.631	0.625	$\frac{7}{64}$
1012	$\frac{5}{16} \times 1\frac{1}{2}$	1.484	0.641	0.636	0.631	0.625	$\frac{7}{64}$
1212	$\frac{3}{8} \times 1\frac{1}{2}$	1.484	0.641	0.636	0.631	0.625	$\frac{7}{64}$

All dimensions given are in inches.

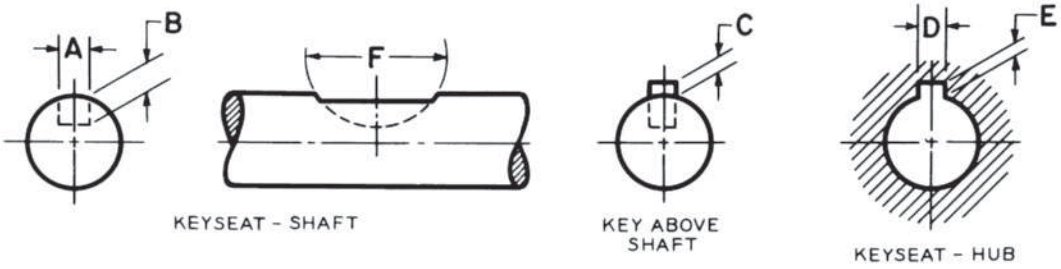
The key numbers indicate nominal key dimensions. The last two digits give the nominal diameter B in eighths of an inch and the digits preceding the last two give the nominal width W in thirty-seconds of an inch.

Example:

No. 204 indicates a key $\frac{2}{32} \times \frac{4}{8}$ or $\frac{1}{16} \times \frac{1}{2}$.
No. 808 indicates a key $\frac{8}{32} \times \frac{8}{8}$ or $\frac{1}{4} \times 1$.
No. 1212 indicates a key $\frac{12}{32} \times \frac{12}{8}$ or $\frac{3}{8} \times 1\frac{1}{2}$.

Dimensions of Keys and Slots. (CONTINUED)

WOODRUFF KEYS AND KEYSEATS

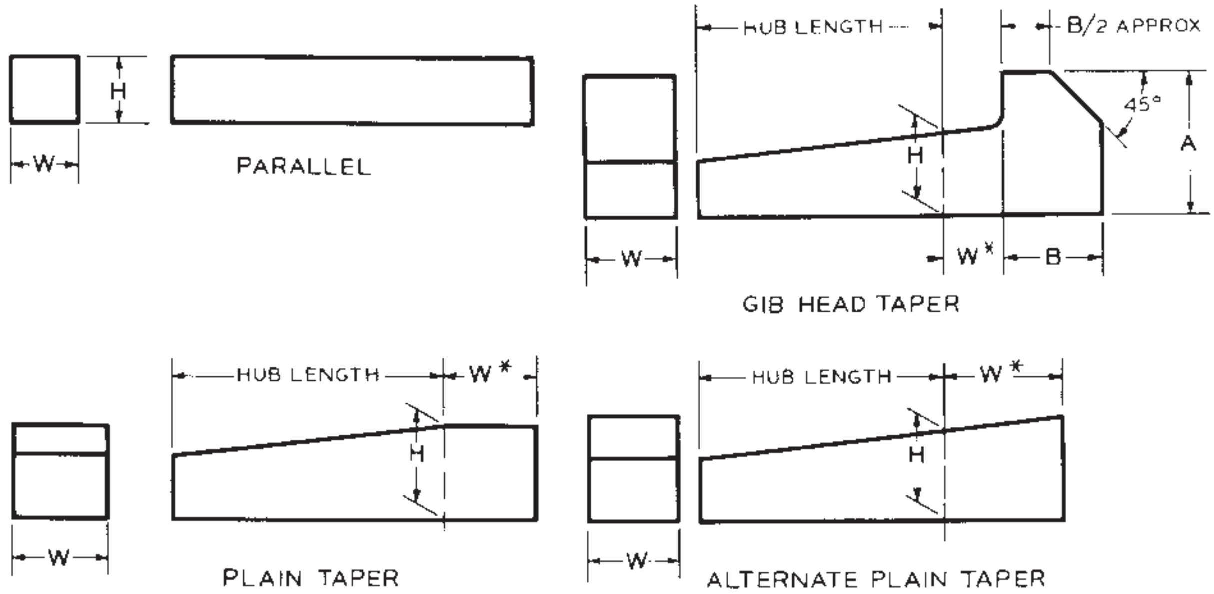


Key Number	Nominal Size Key	Keyseat – Shaft: Width A* Min	Keyseat – Shaft: Width A* Max	Keyseat – Shaft: Depth B +0.005 -0.000	Keyseat – Shaft: Diameter F Min	Keyseat – Shaft: Diameter F Max	Key Above Shaft: Height C +0.005 -0.005	Keyseat – Hub: Width D +0.002 -0.000	Keyseat – Hub: Depth E +0.005 -0.000
202	1/16 × 1/4	0.0615	0.0630	0.0728	0.250	0.268	0.0312	0.0635	0.0372
202.5	1/16 × 5/16	0.0615	0.0630	0.1038	0.312	0.330	0.0312	0.0635	0.0372
302.5	3/32 × 5/16	0.0928	0.0943	0.0882	0.312	0.330	0.0469	0.0948	0.0529
203	1/16 × 3/8	0.0615	0.0630	0.1358	0.375	0.393	0.0312	0.0635	0.0372
303	3/32 × 3/8	0.0928	0.0943	0.1202	0.375	0.393	0.0469	0.0948	0.0529
403	1/8 × 3/8	0.1240	0.1255	0.1045	0.375	0.393	0.0625	0.1260	0.0685
204	1/16 × 1/2	0.0615	0.0630	0.1668	0.500	0.518	0.0312	0.0635	0.0372
304	3/32 × 1/2	0.0928	0.0943	0.1511	0.500	0.518	0.0469	0.0948	0.0529
404	1/8 × 1/2	0.1240	0.1255	0.1355	0.500	0.518	0.0625	0.1260	0.0685
305	3/32 × 5/8	0.0928	0.0943	0.1981	0.625	0.643	0.0469	0.0948	0.0529
405	1/8 × 5/8	0.1240	0.1255	0.1825	0.625	0.643	0.0625	0.1260	0.0685
505	5/32 × 5/8	0.1553	0.1568	0.1669	0.625	0.643	0.0781	0.1573	0.0841
605	3/16 × 5/8	0.1863	0.1880	0.1513	0.625	0.643	0.0937	0.1885	0.0997
406	1/8 × 3/4	0.1240	0.1255	0.2455	0.750	0.768	0.0625	0.1260	0.0685
506	5/32 × 3/4	0.1553	0.1568	0.2299	0.750	0.768	0.0781	0.1573	0.0841
606	3/16 × 3/4	0.1863	0.1880	0.2143	0.750	0.768	0.0937	0.1885	0.0997
806	1/4 × 3/4	0.2487	0.2505	0.1830	0.750	0.768	0.1250	0.2510	0.1310
507	5/32 × 7/8	0.1553	0.1568	0.2919	0.875	0.895	0.0781	0.1573	0.0841
607	3/16 × 7/8	0.1863	0.1880	0.2763	0.875	0.895	0.0937	0.1885	0.0997
707	7/32 × 7/8	0.2175	0.2193	0.2607	0.875	0.895	0.1093	0.2198	0.1153
807	1/4 × 7/8	0.2487	0.2505	0.2450	0.875	0.895	0.1250	0.2510	0.1310
608	3/16 × 1	0.1863	0.1880	0.3393	1.000	1.020	0.0937	0.1885	0.0997
708	7/32 × 1	0.2175	0.2193	0.3237	1.000	1.020	0.1093	0.2198	0.1153
808	1/4 × 1	0.2487	0.2505	0.3080	1.000	1.020	0.1250	0.2510	0.1310
1008	5/16 × 1	0.3111	0.3130	0.2768	1.000	1.020	0.1562	0.3135	0.1622
1208	3/8 × 1	0.3735	0.3755	0.2455	1.000	1.020	0.1875	0.3760	0.1935
609	3/16 × 1 1/8	0.1863	0.1880	0.3853	1.125	1.145	0.0937	0.1885	0.0997
709	7/32 × 1 1/8	0.2175	0.2193	0.3697	1.125	1.145	0.1093	0.2198	0.1153
809	1/4 × 1 1/8	0.2487	0.2505	0.3540	1.125	1.145	0.1250	0.2510	0.1310
1009	5/16 × 1 1/8	0.3111	0.3130	0.3228	1.125	1.145	0.1562	0.3135	0.1622

From The American Society of Mechanical Engineers–ANSI B17.2–1967–R1990

Dimensions of Keys and Slots. (CONTINUED)

KEYS AND KEYSEATS



Plain and Gib Head Taper Keys Have a 1/8" Taper in 12"

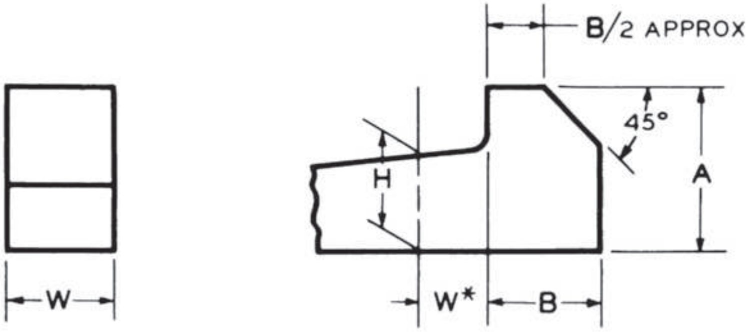
KEY DIMENSIONS AND TOLERANCES

KEY			Nominal Key Size, Width, W: Over	Nominal Key Size, Width, W: To (Incl)	Tolerance: Width, W	Tolerance: Height, H
Parallel	Square	Bar Stock	−	3/4	+0.000 −0.002	+0.000 −0.002
			3/4	1-1/2	+0.000 −0.003	+0.000 −0.003
			1-1/2	2-1/2	+0.000 −0.004	+0.000 −0.004
			2-1/2	3-1/2	+0.000 −0.006	+0.000 −0.006
		Keystock	−	1-1/4	+0.001 −0.000	+0.001 −0.000
			1-1/4	3	+0.002 −0.000	+0.002 −0.000
			3	3-1/2	+0.003 −0.000	+0.003 −0.000
	Rectangular	Bar Stock	−	3/4	+0.000 −0.003	+0.000 −0.003
			3/4	1-1/2	+0.000 −0.004	+0.000 −0.004
			1-1/2	3	+0.000 −0.005	+0.000 −0.005
			3	4	+0.000 −0.006	+0.000 −0.006
			4	6	+0.000 −0.008	+0.000 −0.008
			6	7	+0.000 −0.013	+0.000 −0.013
		Keystock	−	1-1/4	+0.001 −0.000	+0.005 −0.005
			1-1/4	3	+0.002 −0.000	+0.005 −0.005
			3	7	+0.003 −0.000	+0.005 −0.005
Taper	Plain or Gib Head Square or Rectangular		−	1-1/4	+0.001 −0.000	+0.005 −0.000
			1-1/4	3	+0.002 −0.000	+0.005 −0.000
			3	7	+0.003 −0.000	+0.005 −0.000

*For locating position of dimension H. Tolerance does not apply.
See Table 41 for dimensions on gib heads.
All dimensions given in inches.

Dimensions of Keys and Slots. (CONTINUED)

USA STANDARD



GIB HEAD NOMINAL DIMENSIONS

Nominal Key Size Width, *W*	SQUARE			RECTANGULAR		
	H	*A*	*B*	*H*	*A*	*B*
1/8	1/8	1/4	1/4	3/32	3/16	1/8
3/16	3/16	5/16	5/16	1/8	1/4	1/4
1/4	1/4	7/16	3/8	3/16	5/16	5/16
5/16	5/16	1/2	7/16	1/4	7/16	3/8
3/8	3/8	5/8	1/2	1/4	7/16	3/8
1/2	1/2	7/8	5/8	3/8	5/8	1/2
5/8	5/8	1	3/4	7/16	3/4	9/16
3/4	3/4	1-1/4	7/8	1/2	7/8	5/8
7/8	7/8	1-3/8	1	5/8	1	3/4
1	1	1-5/8	1-1/8	3/4	1-1/4	7/8
1-1/4	1-1/4	2	1-7/16	7/8	1-3/8	1
1-1/2	1-1/2	2-3/8	1-3/4	1	1-5/8	1-1/8
1-3/4	1-3/4	2-3/4	2	1-1/2	2-3/8	1-3/4
2	2	3-1/2	2-1/4	1-1/2	2-3/8	1-3/4
2-1/2	2-1/2	4	3	1-3/4	2-3/4	2
3	3	5	3-1/2	2	3-1/2	2-1/4
3-1/2	3-1/2	6	4	2-1/2	4	3

*For locating position of dimension *H*.

For larger sizes the following relationships are suggested as guides for establishing *A* and *B*.

$A = 1.8\,H \qquad B = 1.2\,H$

All dimensions given in inches.

Dimensions of Keys and Slots. (CONTINUED)

WOODRUFF KEY SIZES FOR DIFFERENT SHAFT DIAMETERS

Shaft Diameter	5/16 to 3/8	7/16 to 1/2	9/16 to 3/4	13/16 to 15/16	1 to 1 3/16	1 1/4 to 1 7/16	1 1/2 to 1 3/4	1 13/16 to 2 1/8	2 3/16 to 2 1/2
Key Numbers	204	304 305	404 405 406	505 506 507	606 607 608 609	807 808 809	810 811 812	1011 1012	1211 1212

Dimensions of Keys and Slots. (CONTINUED)

Application	SAE No.	Application	SAE No.
Adapters	1145	Chain pins, transmission	4320
Agricultural steel	1070	" " "	4815
" "	1080	" " "	4820
Aircraft forgings	4140	Chains, transmission	3135
Axles, front or rear	1040	" "	3140
" " "	4140	Clutch disks	1060
Axle shafts	1045	" "	1070
" "	2340	" "	1085
" "	2345	Clutch springs	1060
" "	3135	Coil springs	4063
" "	3140	Cold-headed bolts	4042
" "	3141	Cold-heading steel	30905
" "	4063	Cold-heading wire or rod	rimmed*
" "	4340	" " "	1035
Ball-bearing races	52100	Cold-rolled steel	1070
Balls for ball bearings	52100	Connecting-rods	1040
Body stock for cars	rimmed*	" "	3141
Bolts, anchor	1040	Connecting-rod bolts	3130
Bolts and screws	1035	Corrosion resisting	51710
Bolts, cold-headed	4042	" "	30805
Bolts, connecting-rod	3130	Covers, transmission	rimmed*
Bolts, heat-treated	2330	Crankshafts	1045
Bolts, heavy-duty	4815	"	1145
" " "	4820	"	3135
Bolts, steering-arm	3130	"	3140
Brake levers	1030	"	3141
" "	1040	Crankshafts, Diesel engine	4340
Bumper bars	1085	Cushion, springs	1060
Cams, free-wheeling	4615	Cutlery, stainless	51335
" " "	4620	Cylinder studs	3130
Camshafts	1020	Deep-drawing steel	rimmed*
"	1040	" " "	30905
Carburized parts	1020	Differential gears	4023
" "	1022	Disks, clutch	1070
" "	1024	" "	1060
" "	1320	Ductile steel	30905
" "	2317	Fan blades	1020
" "	2515	Fatigue resisting	4340
" "	3310	" "	4640
" "	3115	Fender stock for cars	rimmed*
" "	3120	Forgings, aircraft	4140
" "	4023	Forgings, carbon steel	1040
" "	4032	" " "	1045
" "	1117	Forgings, heat-treated	3240
" "	1118	" " "	5140

General Applications of SAE Steels.

Application	SAE No.	Application	SAE No.
Forgings, heat-treated	6150	Key stock	1030
Forgings, high-duty	6150	" "	2330
Forgings, small or medium .	1035	" "	3130
Forgings, large .	1036	Leaf springs	1085
Free-cutting carbon steel.....	1111	" "	9260
" " " "	1113	Levers, brake	1030
Free-cutting chro.-ni.steel	30615	" "	1040
Free-cutting mang. steel......	1132	Levers, gear shift	1030
" " " "	1137	Levers, heat-treated	2330
Gears, carburized	1320	Lock-washers	1060
" "	2317	Mower knives	1085
" "	3115	Mower sections	1070
" "	3120	Music wire..........................	1085
" "	3310	Nuts	3130
" "	4119	Nuts, heat-treated	2330
" "	4125	Oil-pans, automobile	rimmed*
" "	4320	Pinions, carburized	3115
" "	4615	" "	3120
" "	4620	" "	4320
" "	4815	Piston-pins	3115
" "	4820	" "	3120
Gears, heat-treated............	2345	Plow beams	1070
Gears, car and truck	4027	Plow disks	1080
" " " "...........	4032	Plow shares	1080
Gears, cyanide-hardening ..	5140	Propeller shafts....................	2340
Gears, differential	4023	" "	2345
Gears, high duty	4640	" "	4140
" " "	6150	Races, ball-bearing	52100
Gears, oil-hardening..........	3145	Ring gears	3115
" " "	3150	" "	3120
" " "	4340	" "	4119
" " "	5150	Rings, snap.........................	1060
Gears, ring.......................	1045	Rivets	rimmed*
" "	3115	Rod and wire......................	killed*
" "	3120	Rod, cold-heading	1035
" "	4119	Roller bearings....................	4815
Gears, transmission	3115	Rollers for bearings..............	52100
" "	3120	Screws and bolts	1035
" "	4119	Screw stock, Bessemer	1111
Gears, truck and bus	3310	" " "	1112
" " " "..........	4320	" " "	1113
Gear shift levers	1030	Screw stock, open hearth	1115
Harrow disks	1080	Screws, heat-treated.............	2330
" "	1095	Seat springs........................	1095
Hay-rake teeth	1095	Shafts, axle.........................	1045

General Applications of SAE Steels. (CONTINUED)

Application	SAE No.	Application	SAE No.
Shafts, cyanide-hardening	5140	Steel, cold-heading	30905
Shafts, heavy-duty	4340	Steel, free-cutting carbon	11111
" " "	6150	" " " "	1113
" " "	4615	Steel, free-cutting chro.-ni.	30615
" " "	4620	Steel, free-cutting mang.	1132
Shafts, oil-hardening	5150	" " " "	0000
Shafts, propeller	2340	Steel, minimum distortion	4615
" "	2345	" " "	4620
" "	4140	" " "	4640
Shafts, transmission	4140	Steel, soft ductile	30905
Sheets and strips	rimmed*	Steering arms	4042
Snap rings	1060	Steering-arm bolts	3130
Spline shafts	1045	Steering knuckles	3141
" "	1320	Steering-knuckle pins	4815
" "	2340	" " "	4820
" "	2345	Studs	1040
" "	3115	"	1111
" "	3120	Studs, cold-headed	4042
" "	3135	Studs, cylinder	3130
" "	3140	Studs, heat-treated	2330
" "	4023	Studs, heavy-duty	4815
Spring clips	1060	" " "	4820
Springs, coil	1095	Tacks	rimmed*
" "	4063	Thrust washers	1060
" "	6150	Thrust washers, oil-harden	5150
Springs, clutch	1060	Transmission shafts	4140
Springs, cushion	1060	Tubing	1040
Springs, leaf	1085	Tubing, front axle	4140
" "	1095	Tubing, seamless	1030
" "	4063	Tubing, welded	1020
" "	4068	Universal joints	1145
" "	9260	Valve springs	1060
" "	6150	Washers, lock	1060
Springs, hard-drawn coiled	1066	Welded structures	30705
Springs, oil-hardening	5150	Wire and rod	killed*
Springs, oil-tempered wire	1066	Wire, cold-heading	rimmed*
Springs, seat	1095	" " "	1035
Springs, valve	1060	Wire, hard-drawn spring	1045
Spring wire	1045	" " " "	1055
Spring wire, hard-drawn	1055	Wire, music	1085
Spring wire, oil-tempered	1055	Wire, oil-tempered spring	1055
Stainless irons	51210	Wrist-pins, automobile	1020
" "	51710	Yokes	1145
Steel, cold-rolled	1070		

General Applications of SAE Steels. (CONTINUED)

Element	Symbol	Melting point, °F	Boiling point, °F	Specific heat,[a] cal/g/°C	Thermal conductivity,[a] Btu/hr/sq ft/°F/ft	Density,[a] g/cm³	Modulus of elasticity in tension, million psi	Coefficient of linear thermal expansion,[a] μ in./in./°F	Electrical resistivity, microhm-cm	Crystal structure
Aluminum	Al	1220	4442	0.215	128.	2.70	9	13.1	2.65	f.c.c
Antimony	Sb	1167	2516	0.049	10.8	6.62	11.3	4.7	39	Rhomb.
Arsenic	As	1503 (28 atm)	1135[b]	0.082	.....	5.72	.....	2.6	33.3	Rhomb.
Barium	Ba	1317	2980	0.068	.....	3.5	.....	....	....	b.c.c.
Beryllium	Be	2332	5020	0.45	84.4	1.85	42	6.4	4	h.c.p.
Bismuth	Bi	520	2840	0.029	4.8	9.80	4.6	7.4	107	Rhomb.
Boron	B	3690	.....	0.309	.....	2.34	.....	4.6	10^6	Orthorhomb
Cadmium	Cd	610	1409	0.055	53.	8.65	8	16.55	6.83	h.c.p.
Calcium	Ca	1540	2625	0.149	72.3	1.55	3.5	12.4	3.91	f.c.c.
Carbon (graphite)	C	6740[b]	8730	0.165	13.8	2.25	0.7	0.3 to 2.4	1375	Hexag.
Cerium	Ce	1479	6280	0.045	6.6	6.77	6	4.4	75	f.c.c.
Cesium	Cs	84	1273	0.048	.....	1.90	.....	54	20	b.c.c.
Chromium	Cr	3407	4829	0.11	40.3	7.19	36	3.4	12.9	b.c.c.
Cobalt	Co	2723	5250	0.099	41.5	8.85	30	7.66	6.24	h.c.p.
Columbium	Cb	4474	8901	0.065	31.5	8.57	.....	4.06	12.5	b.c.c.
Copper	Cu	1981	4703	0.092	226.	8.96	16	9.2	1.67	f.c.c.
Gallium	Ga	86	4059	0.079	19.4	5.91	.....	10	17.4	Orthorhomb.
Germanium[c]	Ge	1719	5125	0.073	33.7	5.32	.....	3.19	46×10^5	Diam. cubic
Gold	Au	1954	5380	0.031	171.	19.32	11.6	7.9	2.35	f.c.c.
Indium	In	313	3632	0.057	13.8	7.31	1.57	18	8.37	f.c.tetr.
Iridium	Ir	4449	9570	0.031	33.7	22.50	76	3.8	5.3	f.c.c.
Iron	Fe	2798	5430	0.11	43.3	7.87	28.5	6.53	9.71	b.c.c.
Lanthanum	La	1688	6280	0.048	8.	6.19	10.5	2.77	57	Hexag.
Lead	Pb	621	3137	0.031	20.	11.36	2	16.3	20.6	f.c.c.
Lithium	Li	357	2426	0.79	41.	0.534	.....	31	8.55	b.c.c.
Magnesium	Mg	1202	2025	0.245	88.5	1.74	6.35	15.05	4.45	h.c.p.
Manganese	Mn	2273	3900	0.115	.....	7.73	23	12.22	185	Complex cubic
Mercury	Hg	−37	675	0.033	4.7	13.55	.....	....	98.4	Rhomb.
Molybdenum	Mo	4730	10040	0.000	82.	10.22	47	2.7	5.2	b.c.c.
Nickel	Ni	2647	4950	0.105	53.	8.90	30	7.39	6.84	f.c.c.
Osmium	Os	4900	9950	0.031	.....	22.57	81	2.6	9.5	h.c.p.
Palladium	Pd	2826	7200	0.058	40.5	12.02	16.3	6.53	10.8	f.c.c.
Phosphorus (white)	P	112	536	0.177	.....	1.83	.....	70	10	Cubic
Platinum	Pt	3217	8185	0.0314	39.8	21.45	21.3	4.9	10.6	f.c.c.
Plutonium	Pu	1184	6000	0.033	4.8	19.00	14	30.55	141.4	Monoclinic
Potassium	K	147	1400	0.177	58.	0.86	.....	46	6.15	b.c.c.
Rhenium	Re	5755	10650	0.033	41.	21.04	66.7	3.7	19.3	h.c.p.
Rhodium	Rh	3571	8130	0.059	50.6	12.44	42.5	4.6	4.51	f.c.c.
Rubidium	Rb	102	1270	0.080	.....	1.53	.....	50	12.5	b.c.c.
Ruthenium	Ru	4530	8850	0.057	.....	12.20	60	5.1	7.6	h.c.p.
Selenium	Se	423	1265	0.084	.....	4.70	8.4	21	12	Hexag.
Silicon[c]	Si	2570	48660	0.162	48.2	2.33	16.35	1.6 to 1.4	10^5	Diam. cubic
Silver	Ag	1761	4010	0.056	242.	10.49	11	10.9	1.59	f.c.c.
Sodium	Na	208	1638	0.295	77.2	0.971	.....	39	4.2	b.c.c.
Strontium	Sr	1414	2520	0.176	.....	2.60	.....	.....	23	f.c.c.
Tantalum	Ta	5425	9800	0.034	31.3	16.60	27	3.6	12.45	b.c.c.
Tellurium	Te	841	1814	0.047	3.3	6.24	6	9.3	46×10^5	Hexag.
Thallium	Tl	577	2655	0.031	22.5	11.85	.....	16	18	h.c.p.
Thorium	Th	3182	7000	0.034	21.7	11.66	.....	6.9	13	f.c.c.
Tin	Sn	449	4120	0.054	36.2	7.30	6.3	13	11	Tetrag
Titanium	Ti	3035	5900	0.124	9.8	4.51	16.8	4.67	42	h.c.p.
Tungsten	W	6170	10706	0.033	96.	19.30	50	2.55	5.65	b.c.c.
Uranium	U	2070	6904	0.028	17.1	19.07	24	3.8 to 7.8	30	Orthorhomb.
Vanadium	V	3450	6150	0.119	16.9	6.1	19	4.6	26	h.c.c.
Yttrium	Y	2748	5490	0.071	8.5	4.47	17	....	57	h.c.p.
Zinc	Zn	787	1663	0.092	65.	7.13	.....	22	5.92	h.c.p.
Zirconium	Zr	3366	6470	0.067	9.6	6.49	13.7	3.2	40	h.c.p.

[a] Near 68°F (20°C)
[b] Sublimes—triple point at 2028 atm.
[c] Semiconductor.
Courtesy of "Metals Handbook," vol. 1, 8th ed., American Society for Metals, Cleveland, 1961.

Source: American Society of Metals

Properties of Common Metals (density, Young's modules, coefficient of thermal expansion).

Metal	Modulus of elasticity E, million psi	Ultimate tensile strength σ_u, thousand psi	Yield strength σ_y, thousand psi	Endurance limit σ_{end}, thousand psi	Hardness Brinell
Gray cast iron, ASTM 20, med. sec.	12	22	....	10	180
Gray cast iron, ASTM 50, med. sec.	19	53	....	25	240
Nodular ductile cast iron:					
Type 60-45-10	22–25	60–80	45–60	35	140–190
Type 120-90-02	22–25	120–150	90–125	52	240–325
Austenitic	18.5	58–68	32–38	32	140–200
Malleable cast iron, ferritic 32510	25	50	33.5	28	110–156
Malleable cast iron, pearlitic 60003	28	80–100	60–80	39–40	197–269
Ingot iron, hot rolled	29.8	44	23	28	83
Ingot iron, cold drawn	29.8	73	69	33	142
Wrought iron, hot rolled longit.	29.5	48	27	23	97–105
Cast carbon steel, normalized 70000	30	70	38	31	140
Cast steel, low alloy, 100,000 norm. and temp	29–30	100	68	45	209
Cast steel, low alloy, 200,000 quench. and temp	29–30	200	170	85	400
Wrought plain C steel:					
C1020 hot rolled	29–30	66	44	32	143
C1045 hard. and temp. 100°F	29–30	118	88	....	277
C1095 hard. and temp. 700°F	29–30	180	118	....	375
Low-alloy steels:					
Wrought 1330, HT and temp 1000°F	29–30	122	100	....	248
Wrought 2317, HT and temp 1000°F	29–30	107	72	....	222
Wrought 4340, HT and temp 800°F	29–30	220	200	....	445
Wrought 6150, HT and temp 1000°F	29–30	187	179	....	444
Wrought 2317, HT and temp 800°F	29–30	214	194	....	423
Ultra high strength steel H11, HT	30	295–311	241–247	132	
300M HT and temp. 500°F	....	289	242	116	
4340 HT and temp. 400°F	30	287	270	107	
25 Ni Maraging	24	319	284		
Austenitic Stainless Steel 302, cold worked	28	110	75	34	240
Ferritic Stainless Steel 302, cold worked	29	75–90	45–80		
Martensitic Stainless Steel 410, HT	29	90–190	60–145	40	180–390
Martensitic Stainless Steel 440A, HT	29	260	240	....	510
Nitriding Steels, 135 Mod, hard and temp. (core properties)	29–30	145–159	125–141	45–90	285–320
Nitiding Steels 5Ni-2A, hard. and temp	29–30	206	202	90	
Structural Steel	30	50–65	30–40	....	120
Aluminum Alloys, cast:					
195 SHT and aged	10.1	36	24	8	75
220 SHT	9.5	48	26	8	75
142 SHT and aged	10.3	28–47	25–42	9.5	75–110
355 SHT and aged	10.2	35–42	25–27	9–10	80–90
A13 as cast	10.3	39	27	19	

Source: American National Standards Institute

ANSI Grades for Steel and Aluminum Alloys (yield strength and ultimate strength for alloys for various conditions including Cold Rolled and Hot Rolled).

Metal	Modulus of elasticity E, million psi	Ultimate tensile strength σ_u, thousand psi	Yield strength σ_y, thousand psi	Endurance limit σ_{end}, thousand psi	Hardness Brinell
Aluminum Alloys, wrought:					
EC ann	10	12	4		
EC H 19, hard	10	27	24	7	
3003 H 18 hard	10	29	27	10	55
2024 H T (T3)	10.6	70	50	20	120
5052 H 38, hard	10.2	42	37	20	77
7075 HT (T6)	10.4	83	73	23	150
7079 HT (T6)	10.3	78	68	23	145
Copper alloys, cast:					
Leaded red brass BB11-4A	9–14.8	33–46	17–24	....	55
Leaded tin bronze BB11-2A	12–16	36–48	16–21	....	60–72
Yellow brass BB11-7A	12–14	60–78	25–40	....	80–95
Aluminum bronze BB11-9BHT	15	90	40	....	180
Copper alloys, wrought:					
Oxygen-free 102 ann	17	32–35	10	11	
Hard	....	50–55	45	13	
Beryllium copper, 172 HT	19	165–183	150–170	35–40	
Cartridge Brass, 260 hard	16	76	63	21	
Muntz metal, 280 ann	15	54	21		
Admiralty, 442 ann	16	53	22		
Manganese bronze, 675 hard	15	84	60		
Phosphor bronze, 521 spring	16	112	70		
Silicon bronze, 647 HT	18	100	88		
Cupro-Nickel, 715 hard	22	80	73		
Magnesium alloys, cast:					
AZ63A, aged	6.5	30–40	14-19	11–15	55–73
AZ92A, aged	6.5	36–40	16-21	11–15	80–84
HK31A, T6	6.5	31	16	9–11	55
Magnesium Alloys, wrought:					
AZ61AF, forged	6.5	43	26	17–22	55
ZE-10A-H24	6.5	34-38	19-28	20–24	
HM31A-T5	6.5	42	33	12–14	
Nickel-alloy castings 210	21.5	45–60	20–30	....	80–125
Monel 411 cast	19	65–90	32–45	....	125–150
Inconel 610 cast	23	70–95	30–45	....	190
Nickel alloys, wrought:					
200 Spring	30	90–130	70–115		
Duranickel, 301 Spring	30	155–190			
K Monel, K500 Spring	26	145–165	130–180		
Titanium alloys, wrought, unalloyed	15–16	60–110	40–95	60–70	
5A1-2.5 Sn	16–17	115–140	110–135	95	
13V-11 Cr3A1 HT	14.5–16	190–240	170–220	50–55	
Zinc, wrought, comm. rolled	....	25–31	....	4.1	
Zinconium, wrought:	14	64	53		
Reactor grade	14	49	29		
Zircaloy 2	13.8	68	61		
Pure metals, wrought:					
Beryllium, ann	44	60–90	45–55		
Hafnium, ann	20	77	32		
Thorium, ann	10	34	26		
Vanadium, ann	20	72	64		
Uranium, ann	30	90	25		

Source: American National Standards Institute

ANSI Grades for Steel and Aluminum Alloys (yield strength and ultimate strength for alloys for various conditions including Cold Rolled and Hot Rolled). (CONTINUED)

Metal	Modulus of elasticity E, million psi	Ultimate tensile strength σ_u, thousand psi	Yield strength σ_y, thousand psi	Endurance limit σ_{end}, thousand psi	Hardness Brinell
Precious metals:	12	19		46	25
Gold, ann	11	22	8		25–35
Silver, ann	21	17–26	2–5.5		38–52
Platinum, ann	17	30	5		46
Rhodium, ann	42	73			55–156
Osmium, cast	80	....			350
Iridium, ann	74	....			170

Babbitt has a compressive elastic limit of 1.3 to 2.5 ksi and a Brinell hardness of 20.

Compressive yield strength of all metals, except those cold-worked = tensile yield strength.

Poisson's ratio is in the range 0.25 to 0.35 for metals.

Yield strength is determined at 0.2 per cent permanent deformation.

Modulus of elasticity in shear for metals is approximately 0.4 of modulus of elasticity in tension E.

Compressive yield strength of cast iron 80,000 to 150,000 ksi.

From Materials in Design Engineering, Materials Selector Issue, vol. 56, No. 5, 1962.
Courtesy of McGraw-Hill Companies.

Source: Harold A. Rothbart, Mechanical Design and Systems Handbook, © 1985 McGraw-Hill.

ANSI Grades for Steel and Aluminum Alloys (yield strength and ultimate strength for alloys for various conditions including Cold Rolled and Hot Rolled). (CONTINUED)

Type	Specific gravity	Coefficient of thermal expansion, 10^{-6} °F	Thermal conductivity, Btu/hr/sq ft/°F/ft	Volume resistivity, ohm-cm	Dialectric strength [a], volts/mil	Modulus of elasticity in tension 10^6 psi	Tensile strength, 10^3 psi
Acrylic, general purpose, type I	1.17–1.19	4.5	0.12	$>10^{15}$	450–530	3.5–4.5	6–9
Cellulose acetnte, type I (med.)	1.24–1.34	4.4–9.0	0.1–0.19	10^{12}	250–600	....	2.7–6.5
Epoxy, general purpose	1.12–2.4	1.7–5.0	0.1–0.8	10^{13}	350–550	....	2–12
Nylon 6	1.13–1.14	4.6–5.4	0.1–0.14	10^{14}	420–485	2.5–3.4	10.2–12
Phenolic, type I (mech.)	1.31	3.3–4.4	....	1.7×10^{12}	350–400	4–5	6–9
Polyester, Allyl type	1.30–1.45	2.8–5.6	0.12	$>10^{13}$	330–500	2–3	4.5–7
Silicone, general (mineral)	1.80–2.0	2.8–3.2	0.09	$>10^{13}$	350–400	....	4.2
Polystyrene, general purpose	1.04–1.07	3.3–4.8	0.00–0.09	10^{18}	>500	4–5	5–8
Polyethylene, low density	0.92	8.9–11	0.19	10^{18}	480	0.22	1.4–2
Polyethylene, medium density	0.93	8.3–16.7	0.19	$>10^{15}$	480	....	2
Polyethylene, high density	0.96	8.3–16.7	0.19	$>10^{15}$	480	....	4.4
Polypropylene	0.89–0.91	6.2	0.08	10^{16}	769–820	1.4–1.7	5

[a] Short time.

Harold A. Rothbart, Mechanical Design and Systems Handbook, © 1985, McGraw-Hill.
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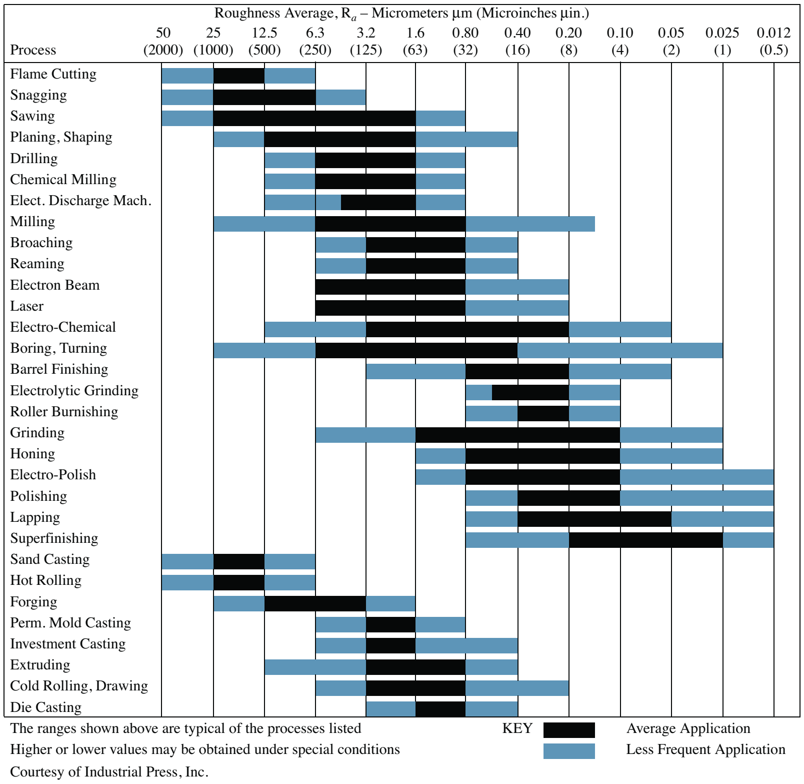
Source: Harold A. Rothbart, Mechanical Design and Systems Handbook, © 1985 McGraw-Hill.

Properties of Common Plastics (density, Young's Modulus, coefficient of thermal expansion, strength).

SHEET METAL AND WIRE GAGE DESIGNATION

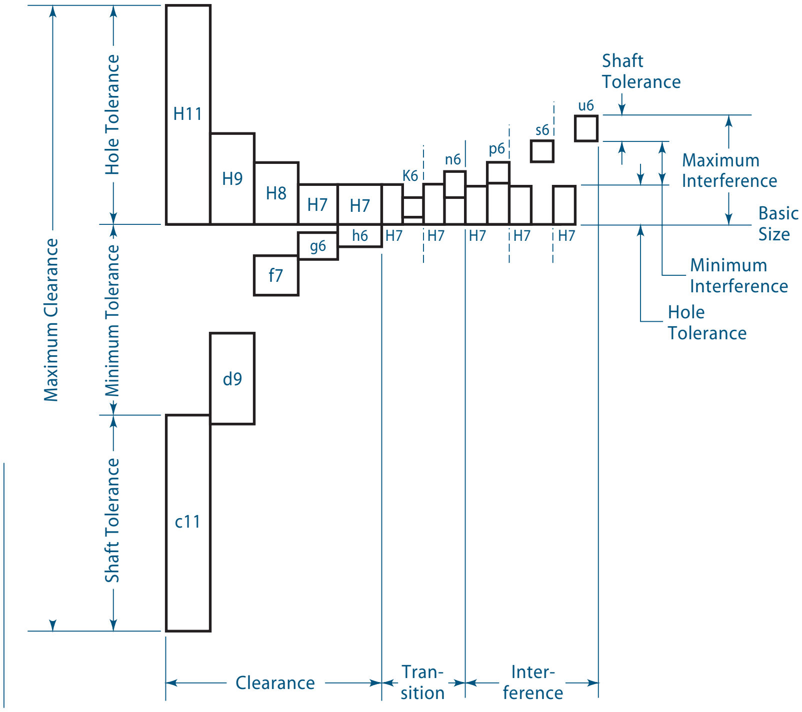
GAGE NO.	AMERICAN OR BROWN & SHARPE'S A.W.G. OR B. & S.	UNITED STATES STANDARD	MANU-FACTURERS' STANDARD FOR SHEET STEEL	GAGE NO.
0000000	....	.500	....	0000000
000000	.5800	.469	....	000000
00000	.5165	.438	....	00000
0000	.4600	.406	....	0000
000	.4096	.375	....	000
00	.3648	.344	....	00
0	.3249	.312	....	0
1	.2893	.281	....	1
2	.2576	.266	....	2
3	.2294	.250	.2391	3
4	.2043	.234	.2242	4
5	.1819	.219	.2092	5
6	.1620	.203	.1943	6
7	.1443	.188	.1793	7
8	.1285	.172	.1644	8
9	.1144	.156	.1495	9
10	.1019	.141	.1345	10
11	.0907	.125	.1196	11
12	.0808	.109	.1046	12
13	.0720	.0938	.0897	13
14	.0642	.0781	.0747	14
15	.0571	.0703	.0673	15
16	.0508	.0625	.0598	16
17	.0453	.0562	.0538	17
18	.0403	.0500	.0478	18
19	.0359	.0438	.0418	19
20	.0320	.0375	.0359	20
21	.0285	.0344	.0329	21
22	.0253	.0312	.0299	22
23	.0226	.0281	.0269	23
24	.0201	.0250	.0239	24
25	.0179	.0219	.0209	25
26	.0159	.0188	.0179	26
27	.0142	.0172	.0164	27
28	.0126	.0156	.0149	28
29	.0113	.0141	.0135	29
30	.0100	.0125	.0120	30
31	.0089	.0109	.0105	31
32	.0080	.0102	.0097	32
33	.0071	.00938	.0090	33
34	.0063	.00859	.0082	34
35	.0056	.00781	.0075	35
36	.0050	.00703	.0067	36

Sheet Metal and Wire Gage Designation.



Source: Industrial Press Inc.

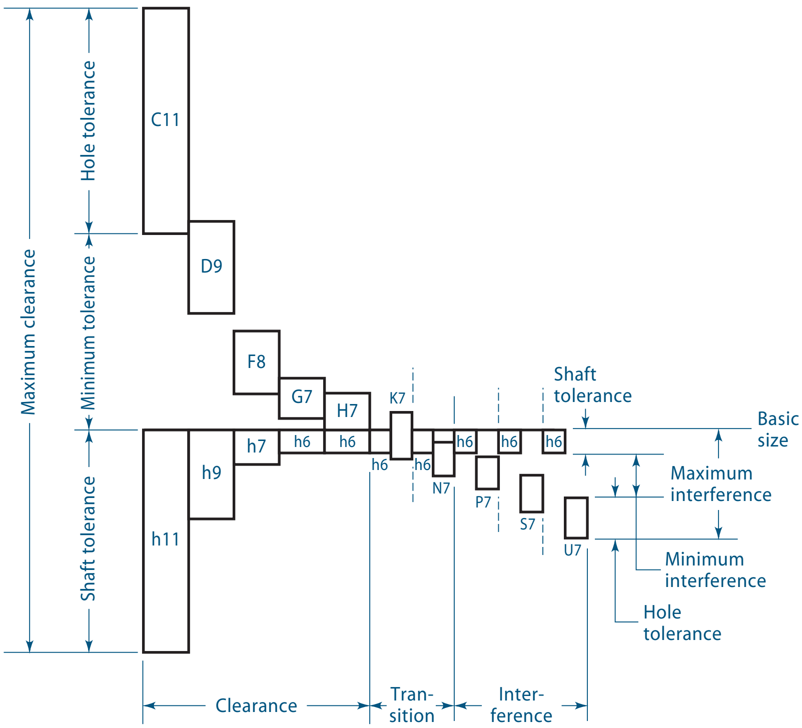
Surface Roughness Produced by Common Production Methods.



Courtesy of Industrial Press, Inc.

Source: Industrial Press Inc.

Standard Allowances, Tolerances, and Fits.



Courtesy of Industrial Press, Inc.

Source: Industrial Press Inc.

Standard Allowances, Tolerances, and Fits. (CONTINUED)

	ISO SYMBOL Hole Basis	ISO SYMBOL Shaft Basis	DESCRIPTION	
Clearance Fits	H11/c11	C11/h11	*Loose running* fit for wide commercial tolerances or allowances on external members.	↑ More Clearance
	H9/d9	D9/h9	*Free running* fit not for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures.	
	H8/f7	F8/h7	*Close Running* fit for running on accurate machines and for accurate moderate speeds and journal pressures.	
	H7/g6	G7/h6	*Sliding fit* not intended to run freely, but to move and turn freely and locate accurately.	
	H7/h6	H7/h6	*Locational clearance* fit provides snug fit for locating stationary parts; but can be freely assembled and disassembled.	
Transition Fits	H7/k6	K7/h6	*Locational transition* fit for accurate location, a compromise between clearance and interferance.	
	H7/n6	N7/h6	*Locational transition* fit for more accurate location where greater interference is permissible.	
Interference Fits	H7/p6[a]	P7/h6	*Locational interference* fit for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements.	More Interferance ↓
	H7/s6	S7/h6	*Medium drive* fit for ordinary steel parts or shrink fits on light sections, the tightest fit usable with cast iron.	
	H7/u6	U7/h6	*Force* fit suitable for parts which can be highly stressed or for shrink fits where the heavy pressing forces required are impractical.	

[a] Transition fit for basic sizes in range from 0 through 3 mm.

Courtesy of Industrial Press, Inc.

Source: Industrial Press Inc.

Standard Allowances, Tolerances, and Fits. (CONTINUED)

AMERICAN NATIONAL STANDARD
PREFERRED METRIC LIMITS AND FITS

ANSI B4.2–1978

PREFERRED HOLE BASIS CLEARANCE FITS

Dimensions in mm.

BASIC SIZE		LOOSE RUNNING Hole H11	LOOSE RUNNING Shaft c11	LOOSE RUNNING Fit	FREE RUNNING Hole H9	FREE RUNNING Shaft d9	FREE RUNNING Fit	CLOSE RUNNING Hole H8	CLOSE RUNNING Shaft f7	CLOSE RUNNING Fit	SLIDING Hole H7	SLIDING Shaft g6	SLIDING Fit	LOCATIONAL CLEARANCE Hole H7	LOCATIONAL CLEARANCE Shaft h6	LOCATIONAL CLEARANCE Fit
1	MAX	1·060	0·940	0·180	1·025	0·980	0·070	1·014	0·994	0·030	1·010	0·998	0·018	1·010	1·000	0·016
	MIN	1·000	0·880	0·060	1·000	0·955	0·020	1·000	0·984	0·006	1·000	0·992	0·002	1·000	0·994	0·000
1·2	MAX	1·260	1·140	0·180	1·225	1·180	0·070	1·214	1·194	0·030	1·210	1·198	0·018	1·210	1·200	0·016
	MIN	1·200	1·080	0·060	1·200	1·155	0·020	1·200	1·184	0·006	1·200	1·192	0·002	1·200	1·194	0·000
1·6	MAX	1·660	1·540	0·180	1·625	1·580	0·070	1·614	1·594	0·030	1·610	1·598	0·018	1·610	1·600	0·016
	MIN	1·600	1·480	0·060	1·600	1·555	0·020	1·600	1·584	0·006	1·600	1·592	0·002	1·600	1·594	0·000
2	MAX	2·060	1·940	0·180	2·025	1·980	0·070	2·014	1·994	0·030	2·010	1·998	0·018	2·010	2·000	0·016
	MIN	2·000	1·880	0·060	2·000	1·955	0·020	2·000	1·984	0·006	2·000	1·992	0·002	2·000	1·994	0·000
2·5	MAX	2·560	2·440	0·180	2·525	2·480	0·070	2·514	2·494	0·030	2·510	2·498	0·018	2·510	2·500	0·016
	MIN	2·500	2·380	0·060	2·500	2·455	0·020	2·500	2·484	0·006	2·500	2·492	0·002	2·500	2·494	0·000
3	MAX	3·060	2·940	0·180	3·025	2·980	0·070	3·014	2·994	0·030	3·010	2·998	0·018	3·010	3·000	0·016
	MIN	3·000	2·880	0·060	3·000	2·955	0·020	3·000	2·984	0·006	3·000	2·992	0·002	3·000	2·994	0·000
4	MAX	4·075	3·930	0·220	4·030	3·970	0·090	4·018	3·990	0·040	4·012	3·996	0·024	4·012	4·000	0·020
	MIN	4·000	3·855	0·070	4·000	3·940	0·030	4·000	3·978	0·010	4·000	3·988	0·004	4·000	3·992	0·000
5	MAX	5·075	4·930	0·220	5·030	4·970	0·090	5·018	4·990	0·040	5·012	4·996	0·024	5·012	5·000	0·020
	MIN	5·000	4·855	0·070	5·000	4·940	0·030	5·000	4·978	0·010	5·000	4·988	0·004	5·000	4·992	0·000
6	MAX	6·075	5·930	0·220	6·030	5·970	0·090	6·018	5·990	0·040	6·012	5·996	0·024	6·012	6·000	0·020
	MIN	6·000	5·855	0·070	6·000	5·940	0·030	6·000	5·978	0·010	6·000	5·988	0·004	6·000	5·992	0·000
8	MAX	8·090	7·920	0·260	8·036	7·960	0·112	8·022	7·987	0·050	8·015	7·995	0·029	8·015	8·000	0·024
	MIN	8·000	7·830	0·080	8·000	7·924	0·040	8·000	7·972	0·013	8·000	7·986	0·005	8·000	7·991	0·000
10	MAX	10·090	9·920	0·260	10·036	9·960	0·112	10·022	9·987	0·050	10·015	9·995	0·029	10·015	10·000	0·024
	MIN	10·000	9·830	0·080	10·000	9·924	0·040	10·000	9·972	0·013	10·000	9·986	0·005	10·000	9·991	0·000
12	MAX	12·110	11·905	0·315	12·043	11·950	0·136	12·027	11·984	0·061	12·018	11·994	0·035	12·018	12·000	0·029
	MIN	12·000	11·795	0·095	12·000	11·907	0·050	12·000	11·966	0·016	12·000	11·983	0·006	12·000	11·989	0·000
16	MAX	16·110	15·905	0·315	16·043	15·950	0·136	16·027	15·984	0·061	16·018	15·994	0·035	16·018	16·000	0·029
	MIN	16·000	15·795	0·095	16·000	15·907	0·050	16·000	15·966	0·016	16·000	15·983	0·006	16·000	15·989	0·000
20	MAX	20·130	19·890	0·370	20·052	19·935	0·169	20·033	19·980	0·074	20·021	19·993	0·041	20·021	20·000	0·034
	MIN	20·000	19·760	0·110	20·000	19·883	0·065	20·000	19·959	0·020	20·000	19·980	0·007	20·000	19·987	0·000
25	MAX	25·130	24·890	0·370	25·052	24·935	0·169	25·033	24·980	0·074	25·021	24·993	0·041	25·021	25·000	0·034
	MIN	25·000	24·760	0·110	25·000	24·883	0·065	25·000	24·959	0·020	25·000	24·980	0·007	25·000	24·987	0·000
30	MAX	30·130	29·890	0·370	30·052	29·935	0·169	30·033	29·980	0·074	30·021	29·993	0·041	30·021	30·000	0·034
	MIN	30·000	29·760	0·110	30·000	29·883	0·065	30·000	29·959	0·020	30·000	29·980	0·007	30·000	29·987	0·000

Metric Limits and Fits.

RUNNING AND SLIDING FITS

VALUES IN THOUSANDTHS OF AN INCH

Nominal Size Range Inches		Class RC1 Precision Sliding			Class RC2 Sliding Fit			Class RC3 Precision Running			Class RC4 Close Running			Class RC5 Medium Running		
		Hole Tol. GR5	Minimum Clearance	Shaft Tol. GR4	Hole Tol. GR6	Minimum Clearance	Shaft Tol. GR5	Hole Tol. GR7	Minimum Clearance	Shaft Tol. GR6	Hole Tol. GR8	Minimum Clearance	Shaft Tol. GR7	Hole Tol. GR8	Minimum Clearance	Shaft Tol. GR7
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	.12	+0.15	0.10	-0.12	+0.25	0.10	-0.15	+0.40	0.30	-0.25	+0.60	0.30	-0.40	+0.60	0.60	-0.40
.12	.24	+0.20	0.15	-0.15	+0.30	0.15	-0.20	+0.50	0.40	-0.30	+0.70	0.40	-0.50	+0.70	0.80	-0.50
.24	.40	+0.25	0.20	-0.15	+0.40	0.20	-0.25	+0.60	0.50	-0.40	+0.90	0.50	-0.60	+0.90	1.00	-0.60
.40	.71	+0.30	0.25	-0.20	+0.40	0.25	-0.30	+0.70	0.60	-0.40	+1.00	0.60	-0.70	+1.00	1.20	-0.70
.71	1.19	+0.40	0.30	-0.25	+0.50	0.30	-0.40	+0.80	0.80	-0.50	+1.20	0.80	-0.80	+1.20	1.60	-0.50
1.19	1.97	+0.40	0.40	-0.30	+0.60	0.40	-0.40	+1.00	1.00	-0.60	+1.60	1.00	-1.00	+1.60	2.00	-1.00
1.97	3.15	+0.50	0.40	-0.30	+0.70	0.40	-0.50	+1.20	1.20	-0.70	+1.80	1.20	-1.20	+1.80	2.50	-1.20
3.15	4.73	+0.60	0.50	-0.40	+0.90	0.50	-0.60	+1.40	1.40	-0.90	+2.20	1.40	-1.40	+2.20	3.00	-1.40
4.73	7.09	+0.70	0.60	-0.50	+1.00	0.60	-0.70	+1.60	1.60	-1.00	+2.50	1.60	-1.60	+2.50	3.50	-1.60
7.09	9.85	+0.80	0.60	-0.60	+1.20	0.60	-0.80	+1.80	2.00	-1.20	+2.80	2.00	-1.80	+2.80	4.50	-1.80
9.85	12.41	+0.90	0.80	-0.60	+1.20	0.80	-0.90	+2.00	2.50	-1.20	+3.00	2.50	-2.00	+3.00	5.00	-2.00
12.41	15.75	+1.00	1.00	-0.70	+1.40	1.00	-1.00	+2.20	3.00	-1.40	+3.50	3.00	-2.20	+3.50	6.00	-2.20

Nominal Size Range Inches		Class RC6 Medium Running			Class RC7 Free Running			Class RC8 Loose Running			Class RC9 Loose Running		
		Hole Tol. GR9	Minimum Clearance	Shaft Tol. GR8	Hole Tol. GR9	Minimum Clearance	Shaft Tol. GR8	Hole Tol. GR10	Minimum Clearance	Shaft Tol. GR9	Hole Tol. GR11	Minimum Clearance	Shaft Tol. GR10
Over	To	-0		+0	-0		+0	-0		+0	-0		+0
0	.12	+1.00	0.60	-0.60	+1.00	1.00	-0.60	+1.60	2.50	-1.00	+2.50	4.00	-1.60
.12	.24	+1.20	0.80	-0.70	+1.20	1.20	-0.70	+1.80	2.80	-1.20	+3.00	4.50	-1.80
.24	.40	+1.40	1.00	-0.90	+1.40	1.60	-0.90	+2.20	3.00	-1.40	+3.50	6.00	-2.20
.40	.71	+1.60	1.20	-1.00	+1.60	2.00	-1.00	+2.80	3.50	-1.60	+4.00	6.00	-2.80
.71	1.19	+2.00	1.60	-1.20	+2.00	2.50	-1.20	+3.50	4.50	-2.00	+5.00	7.00	-3.50
1.19	1.97	+2.50	2.00	-1.60	+2.50	3.00	-1.60	+4.00	5.00	-2.50	+6.00	8.00	-4.00
1.97	3.15	+3.00	2.50	-1.80	+3.00	4.00	-1.80	+4.50	6.00	-3.00	+7.00	9.00	-4.50
3.15	4.73	+3.50	3.00	-2.20	+3.50	5.00	-2.20	+5.00	7.00	-3.50	+9.00	10.00	-5.00
4.73	7.09	+4.00	3.50	-2.50	+4.00	6.00	-2.50	+6.00	8.00	-4.00	+10.00	12.00	-6.00
7.09	9.85	+4.50	4.00	-2.80	+4.50	7.00	-2.80	+7.00	10.00	-4.50	+12.00	15.00	-7.00
9.85	12.41	+5.00	5.00	-3.00	+5.00	8.00	-3.00	+8.00	12.00	-5.00	+12.00	18.00	-8.00
12.41	15.75	+6.00	6.00	-3.50	+6.00	10.00	-3.50	+9.00	14.00	-6.00	+14.00	22.00	-9.00

VALUES IN MILLIMETERS

Nominal Size Range Millimeters		Class RC1 Precision Sliding			Class RC2 Sliding Fit			Class RC3 Precision Running			Class RC4 Close Running			Class RC5 Medium Running		
		Hole Tol. H5	Minimum Clearance	Shaft Tol. g4	Hole Tol. H6	Minimum Clearance	Shaft Tol. g5	Hole Tol. H7	Minimum Clearance	Shaft Tol. f6	Hole Tol. H8	Minimum Clearance	Shaft Tol. f7	Hole Tol. H8	Minimum Clearance	Shaft Tol. e7
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	3	+0.004	0.003	-0.003	+0.006	0.003	-0.004	+0.010	0.008	-0.006	+0.015	0.008	-0.010	+0.015	0.015	-0.010
3	6	+0.005	0.004	-0.004	+0.008	0.004	-0.005	+0.013	0.010	-0.008	+0.018	0.010	-0.013	+0.018	0.020	-0.013
6	10	+0.006	0.005	-0.004	+0.010	0.005	-0.006	+0.015	0.013	-0.010	+0.023	0.013	-0.015	+0.023	0.025	-0.015
10	18	+0.008	0.006	-0.005	+0.010	0.006	-0.008	+0.018	0.015	-0.010	+0.025	0.015	-0.018	+0.025	0.030	-0.018
18	30	+0.010	0.008	-0.006	+0.013	0.008	-0.010	+0.020	0.020	-0.013	+0.030	0.020	-0.020	+0.030	0.040	-0.020
30	50	+0.010	0.010	-0.008	+0.015	0.010	-0.010	+0.030	0.030	-0.015	+0.040	0.030	-0.030	+0.040	0.050	-0.030
50	80	+0.013	0.010	-0.008	+0.018	0.010	-0.013	+0.030	0.030	-0.020	+0.050	0.030	-0.030	+0.050	0.060	-0.030
80	120	+0.015	0.013	-0.010	+0.023	0.013	-0.015	+0.040	0.040	-0.020	+0.060	0.040	-0.040	+0.060	0.080	-0.040
120	180	+0.018	0.015	-0.013	+0.025	0.015	-0.018	+0.040	0.040	-0.030	+0.060	0.040	-0.040	+0.060	0.090	-0.040
180	250	+0.020	0.015	-0.015	+0.030	0.015	-0.020	+0.050	0.050	-0.030	+0.070	0.050	-0.050	+0.070	0.110	-0.050
250	315	+0.023	0.020	-0.015	+0.030	0.020	-0.023	+0.050	0.060	-0.030	+0.080	0.060	-0.050	+0.080	0.130	-0.050
315	400	+0.025	0.025	-0.018	+0.036	0.025	-0.025	+0.060	0.080	-0.040	+0.090	0.080	-0.060	+0.090	0.150	-0.060

Nominal Size Range Millimeters		Class RC6 Medium Running			Class RC7 Free Running			Class RC8 Loose Running			Class RC9 Loose Running		
		Hole Tol. H9	Minimum Clearance	Shaft Tol. e8	Hole Tol. H9	Minimum Clearance	Shaft Tol. d8	Hole Tol. H10	Minimum Clearance	Shaft Tol. e9	Hole Tol. GR11	Minimum Clearance	Shaft Tol. gr10
Over	To	-0		+0	-0		+0	-0		+0	-0		+0
0	3	+0.025	0.015	-0.015	+0.025	0.025	-0.015	+0.041	0.064	-0.025	+0.060	0.100	-0.040
3	6	+0.030	0.015	-0.018	+0.030	0.030	-0.018	+0.046	0.071	-0.030	+0.080	0.110	-0.050
6	10	+0.036	0.025	-0.023	+0.036	0.040	-0.023	+0.056	0.076	-0.036	+0.070	0.130	-0.060
10	18	+0.040	0.030	-0.025	+0.040	0.050	-0.025	+0.070	0.090	-0.040	+0.100	0.150	-0.070
18	30	+0.050	0.040	-0.030	+0.050	0.060	-0.030	+0.090	0.110	-0.050	+0.130	0.180	-0.090
30	50	+0.060	0.050	-0.040	+0.060	0.080	-0.040	+0.100	0.130	-0.060	+0.150	0.200	-0.100
50	80	+0.080	0.060	-0.050	+0.080	0.100	-0.050	+0.110	0.150	-0.080	+0.180	0.230	-0.120
80	120	+0.090	0.080	-0.060	+0.090	0.130	-0.060	+0.130	0.180	-0.090	+0.230	0.250	-0.130
120	180	+0.100	0.090	-0.060	+0.100	0.150	-0.060	+0.150	0.200	-0.100	+0.250	0.300	-0.150
180	250	+0.110	0.100	-0.070	+0.110	0.180	-0.070	+0.180	0.250	-0.110	+0.300	0.380	-0.180
250	315	+0.130	0.130	-0.080	+0.130	0.200	-0.080	+0.200	0.300	-0.130	+0.300	0.460	-0.200
315	400	+0.150	0.150	-0.090	+0.150	0.250	-0.090	+0.230	0.360	-0.150	+0.360	0.560	-0.230

Running and Sliding Fits.

LOCATIONAL CLEARANCE FITS

VALUES IN THOUSANDTHS OF AN INCH

Nominal Size Range Inches		Class LC1			Class LC2			Class LC3			Class LC4			Class LC5			Class LC6		
		Hole Tol. GR6	Minimum Clearance	Shaft Tol. GR5	Hole Tol. GR8	Minimum Clearance	Shaft Tol. GR7	Hole Tol. GR10	Minimum Clearance	Shaft Tol. GR9	Hole Tol. GR7	Minimum Clearance	Shaft Tol. GR6	Hole Tol. GR9	Minimum Clearance	Shaft Tol. GR8	Hole Tol. GR9	Minimum Clearance	Shaft Tol. GR8
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	.12	+0.25	0	-0.15	+0.4	0	-0.25	+0.6	0	-0.4	+1.6	0	-1.0	+0.4	0.10	-0.25	+1.0	0.3	-0.6
.12	.24	+0.30	0	-0.20	+0.5	0	-0.30	+0.7	0	-0.5	+1.8	0	-1.2	+0.5	0.15	-0.30	+1.2	0.4	-0.7
.24	.40	+0.40	0	-0.25	+0.6	0	-0.40	+0.9	0	-0.6	+2.2	0	-1.4	+0.6	0.20	-0.40	+1.4	0.5	-0.9
.40	.71	+0.40	0	-0.30	+0.7	0	-0.40	+1.0	0	-0.7	+2.8	0	-1.6	+0.7	0.25	-0.40	+1.6	0.6	-1.0
.71	1.19	+0.50	0	-0.40	+0.8	0	-0.50	+1.2	0	-0.8	+3.5	0	-2.0	+0.8	0.30	-0.50	+2.0	0.8	-1.2
1.19	1.97	+0.60	0	-0.40	+1.0	0	-0.60	+1.6	0	-1.0	+4.0	0	-2.5	+1.0	0.40	-0.60	+2.5	1.0	-1.6
1.97	3.15	+0.70	0	-0.50	+1.2	0	-0.70	+1.8	0	-1.2	+4.5	0	-3.0	+1.2	0.40	-0.70	+3.0	1.2	-1.8
3.15	4.73	+0.90	0	-0.60	+1.4	0	-0.90	+2.7	0	-1.4	+5.0	0	-3.5	+1.4	0.50	-0.90	+3.5	1.4	-2.2
4.73	7.09	+1.00	0	-0.70	+1.6	0	-1.00	+2.5	0	-1.6	+6.0	0	-4.0	+1.6	0.60	-1.00	+4.0	1.6	-2.5
7.09	9.85	+1.20	0	-0.80	+1.8	0	-1.20	+2.8	0	-1.8	+7.0	0	-4.5	+1.8	0.60	-1.20	+4.5	2.0	-2.8
9.85	12.41	+1.20	0	-0.90	+2.0	0	-1.20	+3.0	0	-2.0	+8.0	0	-5.0	+2.0	0.70	-1.20	+5.0	2.2	-3.0
12.41	15.75	+1.40	0	-1.00	+2.2	0	-1.40	+3.5	0	-2.2	+9.0	0	-6.0	+2.2	0.70	-1.40	+6.0	2.5	-3.5

Nominal Size Range Inches		Class LC7			Class LC8			Class LC9			Class LC10			Class LC11		
		Hole Tol. GR10	Minimum Clearance	Shaft Tol. GR9	Hole Tol. GR10	Minimum Clearance	Shaft Tol. GR9	Hole Tol. GR11	Minimum Clearance	Shaft Tol. GR10	Hole Tol. GR12	Minimum Clearance	Shaft Tol. GR11	Hole Tol. GR13	Minimum Clearance	Shaft Tol. GR12
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	.12	+1.6	0.6	-1.0	+1.6	1.0	-1.0	+2.5	2.5	-1.6	+1.0	4.0	-2.5	+6.0	5.0	-4.0
.12	.24	+1.8	0.8	-1.2	+1.8	1.2	-1.2	+3.0	2.8	-1.8	+5.0	4.5	-3.0	+7.0	6.0	5.0
.24	.40	+2.2	1.0	-1.4	+2.2	1.6	-1.4	+3.5	3.0	-2.2	+6.0	5.0	-3.5	+9.0	7.0	-6.0
.40	.71	+2.8	1.2	-1.6	+2.8	2.0	-1.6	+4.0	3.5	-2.8	+7.0	6.0	-4.0	+10.0	8.0	-7.0
.71	1.19	+3.5	1.6	-2.0	+3.5	2.5	-2.0	+5.0	4.5	-3.5	+8.0	7.0	-5.0	+12.0	10.0	-8.0
1.19	1.97	+4.0	2.0	-2.5	+4.0	3.6	-2.5	+6.0	5.0	-4.0	+10.0	8.0	-6.0	+16.0	12.0	-10.0
1.97	3.15	+4.5	2.5	-3.0	+4.5	4.0	-3.0	+7.0	6.0	-4.5	+12.0	10.0	-7.0	+18.0	14.0	12.0
3.15	4.73	+5.0	3.0	-3.5	+5.0	5.0	-3.5	+9.0	7.0	-5.0	+14.0	11.0	9.0	+22.0	16.0	-14.0
4.73	7.09	+6.0	3.5	-4.0	+6.0	6.0	-4.0	+10.0	8.0	-6.0	+16.0	12.0	-10.0	+25.0	18.0	-16.0
7.09	9.85	+7.0	4.0	4.5	+7.0	7.0	4.5	+12.0	10.0	-7.0	+18.0	16.0	-12.0	+28.0	22.0	-18.0
9.85	12.41	+8.0	4.5	-5.0	+8.0	7.0	-5.0	+12.0	12.0	-8.0	+20.0	20.0	-12.0	+30.0	25.0	-20.0
12.41	15.75	+9.0	5.0	-6.0	+9.0	6.0	-6.0	+14.0	14.0	-9.0	+22.0	22.0	-14.0	+35.0	30.0	-22.0

VALUES IN MILLIMETERS

Nominal Size Range Millimeters		Class LC1			Class LC2			Class LC3			Class LC4			Class LC5			Class LC6		
		Hole Tol. H6	Minimum Clearance	Shaft Tol. h5	Hole Tol. H7	Minimum Clearance	Shaft Tol. h6	Hole Tol. H8	Minimum Clearance	Shaft Tol. h7	Hole Tol. H10	Minimum Clearance	Shaft Tol. h9	Hole Tol. H7	Minimum Clearance	Shaft Tol. g6	Hole Tol. H9	Minimum Clearance	Shaft Tol. f8
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	3	+0.006	0	-0.004	+0.010	0	-0.006	+0.015	0	-0.010	+0.041	0	-0.025	+0.010	0.002	-0.006	+0.025	0.008	-0.015
3	6	+0.008	0	-0.005	+0.013	0	-0.008	+0.018	0	-0.013	+0.046	0	-0.030	+0.013	0.004	-0.008	+0.030	0.010	-0.018
6	10	+0.010	0	-0.006	+0.015	0	-0.010	+0.023	0	-0.015	+0.056	0	-0.036	+0.015	0.005	-0.010	+0.036	0.013	-0.023
10	18	+0.010	0	-0.008	+0.018	0	-0.010	+0.025	0	-0.018	+0.070	0	-0.040	+0.018	0.006	-0.010	+0.041	0.015	-0.025
18	30	+0.013	0	-0.010	+0.020	0	-0.013	+0.030	0	-0.020	+0.090	0	-0.050	+0.020	0.008	-0.013	+0.050	0.020	-0.030
30	50	+0.015	0	-0.010	+0.025	0	-0.015	+0.041	0	-0.025	+0.100	0	-0.060	+0.025	0.010	-0.015	+0.060	0.030	-0.040
50	80	+0.018	0	-0.013	+0.030	0	-0.018	+0.046	0	-0.030	+0.110	0	-0.080	+0.030	0.010	-0.018	+0.080	0.030	-0.050
80	120	+0.023	0	-0.015	+0.036	0	-0.023	+0.056	0	-0.036	+0.130	0	-0.080	+0.036	0.013	-0.023	+0.090	0.040	-0.060
120	180	+0.025	0	-0.018	+0.041	0	-0.025	+0.064	0	-0.041	+0.150	0	-0.100	+0.041	0.015	-0.025	+0.100	0.040	-0.060
180	250	+0.030	0	-0.020	+0.046	0	-0.030	+0.071	0	-0.046	+0.180	0	-0.110	+0.046	0.015	-0.030	+0.110	0.050	-0.070
250	315	+0.020	0	-0.023	+0.051	0	-0.030	+0.076	0	-0.051	+0.200	0	-0.130	+0.051	0.018	-0.030	+0.130	0.060	-0.080
315	400	+0.036	0	-0.025	+0.056	0	-0.036	+0.089	0	-0.056	+0.230	0	-0.150	+0.056	0.018	-0.036	+0.150	0.060	-0.090

Nominal Size Range Millimeters		Class LC7			Class LC8			Class LC9			Class LC10			Class LC11		
		Hole Tol. H10	Minimum Clearance	Shaft Tol. e9	Hole Tol. H10	Minimum Clearance	Shaft Tol. d9	Hole Tol. H11	Minimum Clearance	Shaft Tol. c10	Hole Tol. GR12	Minimum Clearance	Shaft Tol. gr11	Hole Tol. GR13	Minimum Clearance	Shaft Tol. gr12
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	3	+0.041	0.015	-0.025	+0.041	0.025	-0.025	+0.064	0.06	-0.041	+0.10	0.10	-0.06	+0.15	0.13	-0.10
3	6	+0.046	0.020	-0.030	+0.046	0.030	-0.030	+0.076	0.07	-0.46	+0.13	0.11	-0.08	+0.18	0.15	-0.13
6	10	+0.056	0.025	-0.036	+0.056	0.041	-0.036	+0.089	0.08	-0.56	+0.15	0.13	-0.09	+0.23	0.18	-0.15
10	18	+0.070	0.030	-0.040	+0.070	0.050	-0.040	+0.100	0.09	-0.70	+0.18	0.15	-0.10	+0.25	0.20	-0.18
18	30	+0.090	0.040	-0.050	+0.090	0.060	-0.050	+0.130	0.11	-0.90	+0.20	0.18	-0.13	+0.31	0.25	-0.20
30	50	+0.100	0.050	-0.060	+0.100	0.090	-0.060	+0.150	0.13	-0.100	+0.25	0.20	-0.15	+0.41	0.31	-0.25
50	80	+0.110	0.060	-0.080	+0.110	0.100	-0.080	+0.180	0.15	-0.110	+0.31	0.25	-0.18	+0.46	0.36	-0.31
80	120	+0.130	0.080	-0.090	+0.130	0.130	-0.090	+0.230	0.18	-0.130	+0.36	0.28	-0.23	+0.56	0.41	-0.36
120	180	+0.150	0.090	-0.100	+0.150	0.150	-0.100	+0.250	0.20	-0.150	+0.41	0.31	-0.25	+0.64	0.46	-0.41
180	250	+0.180	0.100	-0.110	+0.180	0.180	-0.110	+0.310	0.25	-0.180	+0.46	0.41	-0.31	+0.71	0.56	-0.46
250	315	+0.200	0.110	-0.130	+0.200	0.180	-0.130	+0.310	0.31	-0.200	+0.51	0.51	-0.31	+0.76	0.71	-0.51
315	400	+0.230	0.130	-0.150	+0.230	0.200	-0.150	+0.360	0.36	-0.230	+0.56	0.56	-0.36	+0.89	0.76	-0.56

Running and Sliding Fits. (CONTINUED)

LOCATIONAL TRANSITION FITS

VALUES IN THOUSANDTHS OF AN INCH

Nominal Size Range Inches		Class LT1			Class LT2			Class LT3			Class LT4			Class LT5			Class LT6		
		Hole Tol. GR7	Maximum Interference	Shaft Tol. GR6	Hole Tol. GR8	Maximum Interference	Shaft Tol. GR7	Hole Tol. GR7	Maximum Interference	Shaft Tol. GR6	Hole Tol. GR8	Maximum Interference	Shaft Tol. GR7	Hole Tol. GR7	Maximum Interference	Shaft Tol. GR6	Hole Tol. GR8	Maximum Interference	Shaft Tol. GR7
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	.12	+0.4	0.10	-0.25	+0.6	0.20	-0.4	+0.4	0.25	-0.25	+0.6	0.4	-0.4	+0.4	0.5	-0.25	+0.6	0.65	-0.4
.12	.24	+0.5	0.15	-0.30	+0.7	0.25	-0.5	+0.5	0.40	-0.30	+0.7	0.6	-0.5	+0.5	0.6	-0.30	+0.7	0.80	-0.5
.24	.40	+0.6	0.20	-0.40	+0.9	0.30	-0.6	+0.6	0.50	-0.40	+0.9	0.7	-0.6	+0.6	0.8	-0.40	+0.9	1.00	-0.6
.40	.71	+0.7	0.20	-0.40	+1.0	0.30	-0.7	+0.7	0.50	-0.40	+1.0	0.8	-0.7	+0.7	0.9	-0.40	+1.0	1.20	-0.7
.71	1.19	+0.8	0.25	-0.50	+1.2	0.40	-0.8	+0.8	0.60	-0.50	+1.2	0.9	-0.8	+0.8	1.1	-0.50	+1.2	1.40	-0.8
1.19	1.97	+1.0	0.30	-0.60	+1.6	0.50	-1.0	+1.0	0.70	-0.60	+1.6	1.1	-1.0	+1.0	1.3	-0.60	+1.6	1.70	-1.0
1.97	3.15	+1.2	0.30	-0.70	+1.8	0.60	-1.2	+1.2	0.80	-0.70	+1.8	1.3	-1.2	+1.2	1.5	-0.70	+1.8	2.00	-1.2
3.15	4.73	+1.4	0.40	-0.90	+2.2	0.70	-1.4	+1.4	1.00	-0.90	+2.2	1.5	-1.4	+1.4	1.9	-0.90	+2.2	2.40	-1.4
4.73	7.09	+1.6	0.50	-1.00	+2.5	0.80	-1.6	+1.6	1.10	-1.00	+2.5	1.7	-1.6	+1.6	2.2	-1.00	+2.5	2.80	-1.6
7.09	9.85	+1.8	0.60	-1.20	+2.8	0.90	-1.8	+1.8	1.40	-1.20	+2.8	2.0	-1.8	+1.8	2.6	-1.20	+2.8	3.20	-1.8
9.85	12.41	+2.0	0.60	-1.20	+3.0	1.00	-2.0	+2.0	1.40	-1.20	+3.0	2.2	-2.0	+2.0	2.6	-1.20	+3.0	3.40	-2.0
12.41	15.75	+2.2	0.70	-1.40	+3.5	1.00	-2.2	+2.2	1.60	-1.40	+3.5	2.4	-2.2	+2.2	3.0	-1.40	+3.5	3.80	-2.2

VALUES IN MILLIMETERS

Nominal Size Range Millimeters		Class LT1			Class LT2			Class LT3			Class LT4			Class LT5			Class LT6		
		Hole Tol. H7	Maximum Clearance	Shaft Tol. js6	Hole Tol. H8	Maximum Clearance	Shaft Tol. js7	Hole Tol. H7	Maximum Clearance	Shaft Tol. k6	Hole Tol. H8	Maximum Clearance	Shaft Tol. k7	Hole Tol. H7	Maximum Clearance	Shaft Tol. n6	Hole Tol. H8	Maximum Clearance	Shaft Tol. n7
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	3	+0.010	0.002	-0.006	+0.015	0.005	-0.010	+0.010	0.006	-0.006	+0.015	0.010	-0.010	+0.010	0.013	-0.006	+0.015	0.016	-0.010
3	6	+0.013	0.004	-0.008	+0.018	0.006	-0.013	+0.013	0.010	-0.008	+0.018	0.015	-0.013	+0.013	0.015	-0.008	+0.018	0.020	-0.013
6	10	+0.015	0.005	-0.010	+0.023	0.008	-0.015	+0.015	0.013	-0.010	+0.023	0.018	-0.015	+0.015	0.020	-0.010	+0.023	0.025	-0.015
10	18	+0.018	0.005	-0.010	+0.025	0.008	-0.018	+0.018	0.013	-0.010	+0.025	0.020	-0.018	+0.018	0.023	-0.010	+0.025	0.030	-0.018
18	30	+0.020	0.006	-0.013	+0.030	0.010	-0.020	+0.020	0.015	-0.013	+0.030	0.023	-0.020	+0.020	0.028	-0.013	+0.030	0.036	-0.020
30	50	+0.025	0.008	-0.015	+0.041	0.013	-0.025	+0.025	0.018	-0.015	+0.041	0.028	-0.025	+0.025	0.033	-0.015	+0.041	0.044	-0.025
50	80	+0.030	0.008	-0.018	+0.046	0.015	-0.030	+0.030	0.020	-0.018	+0.046	0.033	-0.030	+0.030	0.038	-0.018	+0.046	0.051	-0.030
80	120	+0.036	0.010	-0.023	+0.056	0.018	-0.036	+0.036	0.025	-0.023	+0.056	0.038	-0.036	+0.036	0.048	-0.023	+0.056	0.062	-0.036
120	180	+0.041	0.013	-0.025	+0.064	0.020	-0.041	+0.041	0.028	-0.025	+0.064	0.044	-0.041	+0.041	0.056	-0.025	+0.064	0.071	-0.041
180	250	+0.046	0.015	-0.030	+0.071	0.023	-0.046	+0.046	0.036	-0.030	+0.071	0.051	-0.046	+0.046	0.066	-0.030	+0.071	0.081	-0.046
250	315	+0.051	0.015	-0.030	+0.076	0.025	-0.051	+0.051	0.036	-0.030	+0.076	0.056	-0.051	+0.051	0.066	-0.030	+0.076	0.086	-0.051
315	400	+0.056	0.018	-0.036	+0.089	0.025	-0.056	+0.056	0.041	-0.036	+0.089	0.062	-0.056	+0.056	0.076	-0.036	+0.089	0.096	-0.056

Running and Sliding Fits. (CONTINUED)

LOCATIONAL INTERFERENCE FITS

VALUES IN THOUSANDTHS OF AN INCH

Nominal Size Range Inches		Class LN1 Light Press Fit			Class LN2 Medium Press Fit			Class LN3 Heavy Press Fit			Class LN4			Class LN5			Class LN6		
		Hole Tol. GR6	Maximum Interference	Shaft Tol. GR5	Hole Tol. GR7	Maximum Interference	Shaft Tol. GR6	Hole Tol. GR7	Maximum Interference	Shaft Tol. GR6	Hole Tol. GR8	Maximum Interference	Shaft Tol. GR7	Hole Tol. GR9	Maximum Interference	Shaft Tol. GR8	Hole Tol. GR10	Maximum Interference	Shaft Tol. GR9
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	.12	+0.25	0.40	-0.15	+0.4	0.65	-0.25	+0.4	0.75	-0.25	+0.6	1.2	-0.4	+1.0	1.8	-0.6	+1.6	3.0	-1.0
.12	.24	+0.30	0.50	-0.20	+0.5	0.80	-0.30	+0.5	0.90	-0.30	+0.7	1.5	-0.5	+1.2	2.3	-0.7	+1.8	3.6	-1.2
.24	.40	+0.40	0.65	-0.25	+0.6	1.00	-0.40	+0.6	1.20	-0.40	+0.9	1.8	-0.6	+1.4	2.8	-0.9	+2.2	4.4	-1.4
.40	.71	+0.40	0.70	-0.30	+0.7	1.10	-0.40	+0.7	1.40	-0.40	+1.0	2.2	-0.7	+1.6	3.4	-1.0	+2.8	5.6	-1.6
.71	1.19	+0.50	0.90	-0.40	+0.8	1.30	-0.50	+0.8	1.70	-0.50	+1.2	2.6	-0.8	+2.0	4.2	-1.2	+3.5	7.0	-2.0
1.19	1.97	+0.60	1.00	-0.40	+1.0	1.60	-0.60	+1.0	2.00	-0.60	+1.6	3.4	-1.0	+2.5	5.3	-1.6	+4.0	8.5	-2.5
1.97	3.15	+0.70	1.30	-0.50	+1.2	2.10	-0.70	+1.2	2.30	-0.70	+1.8	4.0	-1.2	+3.0	6.3	-1.8	+4.5	10.0	-3.0
3.15	4.73	+0.90	1.60	-0.60	+1.4	2.50	-0.90	+1.4	2.90	-0.90	+2.2	4.8	-1.4	+4.0	7.7	-2.2	+5.0	11.5	-3.5
4.73	7.09	+1.00	1.90	-0.70	+1.6	2.80	-1.00	+1.6	3.50	-1.00	+2.5	5.6	-1.6	+4.5	8.7	-2.5	+6.0	13.5	-4.0
7.09	9.85	+1.20	2.20	-0.80	+1.8	3.20	-1.20	+1.8	4.20	-1.20	+2.8	6.6	-1.8	+5.0	10.3	-2.8	+7.0	16.5	-4.5
9.85	12.41	+1.20	2.30	-0.90	+2.0	3.40	-1.20	+2.0	4.70	-1.20	+3.0	7.5	-2.0	+6.0	12.0	-3.0	+8.0	19.0	-5.0
12.41	15.75	+1.40	2.60	-1.00	+2.2	3.90	-1.40	+2.2	5.90	-1.40	+3.5	8.7	-2.2	+6.0	14.5	-3.5	+9.0	23.0	-6.0

VALUES IN MILLIMETERS

Nominal Size Range Millimeters		Class LN1 Light Press Fit			Class LN2 Medium Press Fit			Class LN3 Heavy Press Fit			Class LN4			Class LN5			Class LN6		
		Hole Tol. GR6	Maximum Interference	Shaft Tol. gr5	Hole Tol. H7	Maximum Interference	Shaft Tol. p6	Hole Tol. H7	Maximum Interference	Shaft Tol. t6	Hole Tol. GR8	Maximum Interference	Shaft Tol. gr7	Hole Tol. GR9	Maximum Interference	Shaft Tol. gr8	Hole Tol. GR10	Maximum Interference	Shaft Tol. gr9
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	3	+0.006		-0.004	+0.010	0.016	-0.006	+0.010	0.019	-0.006	+0.015	0.030	-0.010	+0.025	0.046	-0.015	+0.041	0.076	-0.025
3	6	+0.008		-0.005	+0.013	0.020	-0.008	+0.013	0.023	-0.008	+0.018	0.038	-0.013	+0.030	0.059	-0.018	+0.046	0.091	-0.030
6	10	+0.010		-0.006	+0.015	0.025	-0.010	+0.015	0.030	-0.010	+0.023	0.046	-0.015	+0.036	0.071	-0.023	+0.056	0.112	-0.036
10	18	+0.010		-0.008	+0.018	0.028	-0.010	+0.018	0.036	-0.010	+0.025	0.056	-0.018	+0.041	0.086	-0.025	+0.071	0.142	-0.041
18	30	+0.013		-0.010	+0.020	0.033	-0.013	+0.020	0.044	-0.013	+0.030	0.066	-0.020	+0.051	0.107	-0.030	+0.089	0.178	-0.051
30	50	+0.015		-0.010	+0.025	0.041	-0.015	+0.025	0.051	-0.015	+0.041	0.086	-0.025	+0.064	0.135	-0.041	+0.102	0.216	-0.064
50	80	+0.018		-0.013	+0.030	0.054	-0.018	+0.030	0.059	-0.018	+0.046	0.102	-0.030	+0.076	0.160	-0.046	+0.114	0.254	-0.076
80	120	+0.023		-0.015	+0.036	0.064	-0.023	+0.036	0.074	-0.023	+0.056	0.122	-0.036	+0.102	0.196	-0.056	+0.127	0.292	-0.102
120	180	+0.025		-0.018	+0.041	0.071	-0.025	+0.041	0.089	-0.025	+0.064	0.142	-0.041	+0.114	0.221	-0.064	+0.152	0.343	-0.114
180	250	+0.030		-0.020	+0.046	0.081	-0.030	+0.046	0.107	-0.030	+0.071	0.168	-0.046	+0.127	0.262	-0.071	+0.178	0.419	-0.127
250	315	+0.030		-0.023	+0.051	0.086	-0.030	+0.051	0.119	-0.030	+0.076	0.191	-0.051	+0.152	0.305	-0.076	+0.203	0.483	-0.152
315	400	+0.036		-0.025	+0.056	0.099	-0.036	+0.056	0.150	-0.036	+0.089	0.221	-0.056	+0.152	0.368	-0.089	+0.229	0.584	-0.152

Running and Sliding Fits. (CONTINUED)

FORCE AND SHRINK FITS

VALUES IN THOUSANDTHS OF AN INCH

Nominal Size Range Inches		Class FN1 Light Drive Fit			Class FN2 Medium Drive Fit			Class FN3 Heavy Drive Fit			Class FN4 Shrink Fit			Class FN5 Heavy Shrink Fit		
		Hole Tol. GR6	Maximum Interference	Shaft Tol. GR5	Hole Tol. GR7	Maximum Interference	Shaft Tol. GR6	Hole Tol. GR7	Maximum Interference	Shaft Tol. GR6	Hole Tol. GR7	Maximum Interference	Shaft Tol. GR6	Hole Tol. GR8	Maximum Interference	Shaft Tol. GR7
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	.12	+0.25	0.50	-0.15	+0.40	0.85	-0.25				+0.40	0.95	-0.25	+0.60	1.30	-0.40
.12	.24	+0.30	0.60	-0.20	+0.50	1.00	-0.30				+0.50	1.20	-0.30	+0.70	1.70	-0.50
.24	.40	+0.40	0.75	-0.25	+0.60	1.40	-0.40				+0.60	1.60	-0.40	+0.90	2.00	-0.60
.40	.56	+0.40	0.80	-0.30	+0.70	1.60	-0.40				+0.70	1.80	-0.40	+1.00	2.30	-0.70
.56	.71	+0.40	0.90	-0.30	+0.70	1.60	-0.40				+0.70	1.80	-0.40	+1.00	2.50	-0.70
.71	.95	+0.50	1.10	-0.40	+0.80	1.90	-0.50				+0.80	2.10	-0.50	+1.20	3.00	-0.80
.95	1.19	+0.50	1.20	-0.40	+0.80	1.90	-0.50	+0.80	2.10	-0.50	+0.80	2.30	-0.50	+1.20	3.30	-0.80
1.19	1.58	+0.60	1.30	-0.40	+1.00	2.40	-0.60	+1.00	2.60	-0.60	+1.00	3.10	-0.60	+1.60	4.00	-1.00
1.58	1.97	+0.60	1.40	-0.40	+1.00	2.40	-0.60	+1.00	2.80	-0.60	+1.00	3.40	-0.60	+1.60	5.00	-1.00
1.97	2.56	+0.70	1.80	-0.50	+1.20	2.70	-0.70	+1.20	3.20	-0.70	+1.20	4.20	-0.70	+1.80	6.20	-1.20
2.56	3.15	+0.70	1.90	-0.50	+1.20	2.90	-0.70	+1.20	3.70	-0.70	+1.20	4.70	-0.70	+1.80	7.20	-1.20
3.15	3.94	+0.90	2.40	-0.60	+1.40	3.70	-0.90	+1.40	4.40	-0.70	+1.40	5.90	-0.90	+2.20	8.40	-1.40

VALUES IN MILLIMETERS

Nominal Size Range Millimeters		Class FN1 Light Drive Fit			Class FN2 Medium Drive Fit			Class FN3 Heavy Drive Fit			Class FN4 Shrink Fit			Class FN5 Heavy Shrink Fit		
		Hole Tol. GR6	Maximum Interference	Shaft Tol. gr5	Hole Tol. H7	Maximum Interference	Shaft Tol. s6	Hole Tol. H7	Maximum Interference	Shaft Tol. t6	Hole Tol. GR8	Maximum Interference	Shaft Tol. gr7	Hole Tol. H8	Maximum Interference	Shaft Tol. t7
Over	To	-0		+0	-0		+0	-0		+0	-0		+0	-0		+0
0	3	+0.006	0.013	-0.004	+0.010	0.216	-0.006				+0.010	0.024	-0.006	+0.015	0.033	-0.010
3	6	+0.007	0.015	-0.005	+0.013	0.025	-0.007				+0.013	0.030	-0.007	+0.018	0.043	-0.013
6	10	+0.010	0.019	-0.006	+0.015	0.036	-0.010				+0.015	0.041	-0.010	+0.023	0.051	-0.015
10	14	+0.010	0.020	-0.008	+0.018	0.041	-0.010				+0.018	0.046	-0.010	+0.025	0.058	-0.018
14	18	+0.010	0.023	-0.008	+0.018	0.041	-0.010				+0.018	0.046	-0.010	+0.025	0.064	-0.018
18	24	+0.013	0.028	-0.010	+0.020	0.048	-0.013				+0.020	0.053	-0.013	+0.030	0.076	-0.020
24	30	+0.013	0.030	-0.010	+0.020	0.048	-0.013	+0.020	0.053	-0.013	+0.020	0.058	-0.013	+0.030	0.084	-0.020
30	40	+0.015	0.033	-0.010	+0.025	0.061	-0.015	+0.025	0.066	-0.015	+0.025	0.079	-0.015	+0.041	0.102	-0.025
40	50	+0.015	0.036	-0.010	+0.025	0.061	-0.015	+0.025	0.071	-0.015	+0.025	0.086	-0.015	+0.041	0.127	-0.025
50	65	+0.018	0.046	-0.013	+0.030	0.069	-0.018	+0.030	0.082	-0.018	+0.030	0.107	-0.018	+0.046	0.157	-0.030
65	80	+0.018	0.048	-0.013	+0.030	0.074	-0.018	+0.030	0.094	-0.018	+0.030	0.119	-0.018	+0.046	0.183	-0.030
80	100	+0.023	0.061	-0.015	+0.035	0.094	-0.023	+0.035	0.112	-0.023	+0.036	0.150	-0.023	+0.056	0.213	-0.036

Running and Sliding Fits. (CONTINUED)

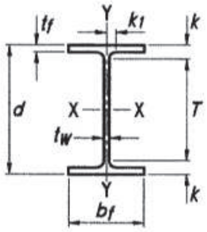


Table 1–1
W Shapes
Dimensions

Shape	Area, A	Depth, d		Web: Thickness, t_w		Web: $\frac{t_w}{2}$	Flange: Width, b_f		Flange: Thickness, t_f		Distance: k: k_{des}	Distance: k: k_{det}	Distance: k_1	Distance: T	Distance: Workable Gage
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.	in.
W36×800[h]	236	42.6	42 1/2	2.38	2 3/8	1 3/16	18.0	18	4.29	4 5/16	5.24	5 9/16	2 3/8	31 3/8	7 1/2
×652[h]	192	41.1	41	1.97	2	1	17.6	17 5/8	3.54	3 9/16	4.49	4 13/16	2 3/16		
×529[h]	156	39.8	39 3/4	1.61	1 5/8	13/16	17.2	17 1/4	2.91	2 15/16	3.86	4 3/16	2		
×487[h]	143	39.3	39 3/8	1.50	1 1/2	3/4	17.1	17 1/8	2.68	2 11/16	3.63	4	1 15/16		
×441[h]	130	38.9	38 7/8	1.36	1 3/8	11/16	17.0	17	2.44	2 7/16	3.39	3 3/4	1 7/8		
×395[h]	116	38.4	38 3/8	1.22	1 1/4	5/8	16.8	16 7/8	2.20	2 3/16	3.15	3 7/16	1 13/16		
×361[h]	106	38.0	38	1.12	1 1/8	9/16	16.7	16 3/4	2.01	2	2.96	3 5/16	1 3/4		
×330	97.0	37.7	37 5/8	1.02	1	1/2	16.6	16 5/8	1.85	1 7/8	2.80	3 1/8	1 3/4		
×302	88.8	37.3	37 3/8	0.945	15/16	1/2	16.7	16 5/8	1.68	1 11/16	2.63	3	1 11/16		
×282[c]	82.9	37.1	37 1/8	0.885	7/8	7/16	16.6	16 5/8	1.57	1 9/16	2.52	2 7/8	1 5/8		
×262[c]	77.0	36.9	36 7/8	0.840	13/16	7/16	16.6	16 1/2	1.44	1 7/16	2.39	2 3/4	1 5/8		
×247[c]	72.5	36.7	36 5/8	0.800	13/16	7/16	16.5	16 1/2	1.35	1 3/8	2.30	2 5/8	1 5/8		
×231[c]	68.1	36.5	36 1/2	0.760	3/4	3/8	16.5	16 1/2	1.26	1 1/4	2.21	2 9/16	1 9/16		
W36×256	75.4	37.4	37 3/8	0.960	15/16	1/2	12.2	12 1/4	1.73	1 3/4	2.48	2 5/8	1 5/16	32 1/8	5 1/2
×232[c]	68.1	37.1	37 1/8	0.870	7/8	7/16	12.1	12 1/8	1.57	1 9/16	2.32	2 7/16	1 1/4		
×210[c]	61.8	36.7	36 3/4	0.830	13/16	7/16	12.2	12 1/8	1.36	1 3/8	2.11	2 5/16	1 1/4		
×194[c]	57.0	36.5	36 1/2	0.765	3/4	3/8	12.1	12 1/8	1.26	1 1/4	2.01	2 3/16	1 3/16		
×182[c]	53.6	36.3	36 3/8	0.725	3/4	3/8	12.1	12 1/8	1.18	1 3/16	1.93	2 1/8	1 3/16		
×170[c]	50.1	36.2	36 1/8	0.680	11/16	3/8	12.0	12	1.10	1 1/8	1.85	2	1 3/16		
×160[c]	47.0	36.0	36	0.650	5/8	5/16	12.0	12	1.02	1	1.77	1 15/16	1 1/8		
×150[c]	44.2	35.9	35 7/8	0.625	5/8	5/16	12.0	12	0.940	15/16	1.69	1 7/8	1 1/8		
×135[c,v]	39.7	35.6	35 1/2	0.600	5/8	5/16	12.0	12	0.790	13/16	1.54	1 11/16	1 1/8		
W33×387[h]	114	36.0	36	1.26	1 1/4	5/8	16.2	16 1/4	2.28	2 1/4	3.07	3 3/16	1 7/16	29 5/8	5 1/2
×354[h]	104	35.6	35 1/2	1.16	1 3/16	5/8	16.1	16 1/8	2.09	2 1/16	2.88	2 15/16	1 3/8		
×318	93.6	35.2	35 1/8	1.04	1 1/16	9/16	16.0	16	1.89	1 7/8	2.68	2 3/4	1 5/16		
×291	85.7	34.8	34 7/8	0.960	15/16	1/2	15.9	15 7/8	1.73	1 3/4	2.52	2 5/8	1 5/16		
×263	77.5	34.5	34 1/2	0.870	7/8	7/16	15.8	15 3/4	1.57	1 9/16	2.36	2 7/16	1 1/4		
×241[c]	71.0	34.2	34 1/8	0.830	13/16	7/16	15.9	15 7/8	1.40	1 3/8	2.19	2 1/4	1 1/4		
×221[c]	65.2	33.9	33 7/8	0.775	3/4	3/8	15.8	15 3/4	1.28	1 1/4	2.06	2 1/8	1 3/16		
×201[c]	59.2	33.7	33 5/8	0.715	11/16	3/8	15.7	15 3/4	1.15	1 1/8	1.94	2	1 3/16		
W33×169[c]	49.5	33.8	33 7/8	0.670	11/16	3/8	11.5	11 1/2	1.22	1 1/4	1.92	2 1/8	1 3/16	29 5/8	5 1/2
×152[c]	44.8	33.5	33 1/2	0.635	5/8	5/16	11.6	11 5/8	1.06	1 1/16	1.76	1 15/16	1 1/8		
×141[c]	41.6	33.3	33 1/4	0.605	5/8	5/16	11.5	11 1/2	0.960	15/16	1.66	1 13/16	1 1/8		
×130[c]	38.3	33.1	33 1/8	0.580	9/16	5/16	11.5	11 1/2	0.855	7/8	1.56	1 3/4	1 1/8		
×118[c,v]	34.7	32.9	32 7/8	0.550	9/16	5/16	11.5	11 1/2	0.740	3/4	1.44	1 5/8	1 1/8		

[c] Shape is slender for compression with $F_y = 50$ ksi.
[h] Flange thickness greater than 2 in. Special requirements may apply per AISC Specification Section A3.1c.
[v] Shape does not meet the h/t_w limit for shear in Specification Section G2.1a with $F_y = 50$ ksi.

Source: American Institute of Steel Construction

Structural Metal Shape Designations.

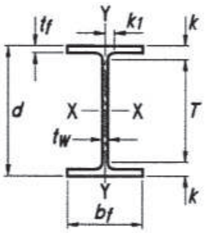


Table 1–1 (continued)
W Shapes
Dimensions

Shape	Area, A	Depth, d		Web Thickness, t_w		Web $\frac{t_w}{2}$	Flange Width, b_f		Flange Thickness, t_f		Distance k k_{des}	Distance k k_{det}	k_1	T	Workable Gage
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.	in.
W30×391[h]	115	33.2	33 1/4	1.36	1 3/8	11/16	15.6	15 5/8	2.44	2 7/16	3.23	3 3/8	1 1/2	26 1/2	5 1/2
×357[h]	105	32.8	32 3/4	1.24	1 1/4	5/8	15.5	15 1/2	2.24	2 1/4	3.03	3 1/8	1 7/16		
×326[h]	95.8	32.4	32 3/8	1.14	1 1/8	9/16	15.4	15 3/8	2.05	2 1/16	2.84	2 15/16	1 3/8		
×292	85.9	32.0	32	1.02	1	1/2	15.3	15 1/4	1.85	1 7/8	2.64	2 3/4	1 5/16		
×261	76.9	31.6	31 5/8	0.930	15/16	1/2	15.2	15 1/8	1.65	1 5/8	2.44	2 9/16	1 5/16		
×235	69.2	31.3	31 1/4	0.830	13/16	7/16	15.1	15	1.50	1 1/2	2.29	2 3/8	1 1/4		
×211	62.2	30.9	31	0.775	3/4	3/8	15.1	15 1/8	1.32	1 5/16	2.10	2 1/4	1 3/16		
×191[c]	56.3	30.7	30 5/8	0.710	11/16	3/8	15.0	15	1.19	1 3/16	1.97	2 1/16	1 3/16		
×173[c]	51.0	30.4	30 1/2	0.655	5/8	5/16	15.0	15	1.07	1 1/16	1.85	2	1 1/8		
W30×148[c]	43.5	30.7	30 5/8	0.650	5/8	5/16	10.5	10 1/2	1.18	1 3/16	1.83	2 1/16	1 1/8	26 1/2	5 1/2
×132[c]	38.9	30.3	30 1/4	0.615	5/8	5/16	10.5	10 1/2	1.00	1	1.65	1 7/8	1 1/8		
×124[c]	36.5	30.2	30 1/8	0.585	9/16	5/16	10.5	10 1/2	0.930	15/16	1.58	1 13/16	1 1/8		
×116[c]	34.2	30.0	30	0.565	9/16	5/16	10.5	10 1/2	0.850	7/8	1.50	1 3/4	1 1/8		
×108[c]	31.7	29.8	29 7/8	0.545	9/16	5/16	10.5	10 1/2	0.760	3/4	1.41	1 11/16	1 1/8		
×99[c]	29.1	29.7	29 5/8	0.520	1/2	1/4	10.5	10 1/2	0.670	11/16	1.32	1 9/16	1 1/16		
×90[c,v]	26.4	29.5	29 1/2	0.470	1/2	1/4	10.4	10 3/8	0.610	5/8	1.26	1 1/2	1 1/16		
W27×539[h]	159	32.5	32 1/2	1.97	2	1	15.3	15 1/4	3.54	3 9/16	4.33	4 7/16	1 13/16	23 5/8	5 1/2[g]
×368[h]	108	30.4	30 3/8	1.38	1 3/8	11/16	14.7	14 5/8	2.48	2 1/2	3.27	3 3/8	1 1/2		5 1/2
×336[h]	98.9	30.0	30	1.26	1 1/4	5/8	14.6	14 1/2	2.28	2 1/4	3.07	3 3/16	1 7/16		
×307[h]	90.4	29.6	29 5/8	1.16	1 3/16	5/8	14.4	14 1/2	2.09	2 1/16	2.88	3	1 7/16		
×281	82.9	29.3	29 1/4	1.06	1 1/16	9/16	14.4	14 3/8	1.93	1 15/16	2.72	2 13/16	1 3/8		
×258	76.0	29.0	29	0.980	1	1/2	14.3	14 1/4	1.77	1 3/4	2.56	2 11/16	1 5/16		
×235	69.4	28.7	28 5/8	0.910	15/16	1/2	14.2	14 1/4	1.61	1 5/8	2.40	2 1/2	1 5/16		
×217	64.0	28.4	28 3/8	0.830	13/16	7/16	14.1	14 1/8	1.50	1 1/2	2.29	2 3/8	1 1/4		
×194	57.2	28.1	28 1/8	0.750	3/4	3/8	14.0	14	1.34	1 5/16	2.13	2 1/4	1 3/16		
×178	52.5	27.8	27 3/4	0.725	3/4	3/8	14.1	14 1/8	1.19	1 3/16	1.98	2 1/16	1 3/16		
×161[c]	47.6	27.6	27 5/8	0.660	11/16	3/8	14.0	14	1.08	1 1/16	1.87	2	1 3/16		
×146[c]	43.1	27.4	27 3/8	0.605	5/8	5/16	14.0	14	0.975	1	1.76	1 7/8	1 1/8		
W27×129[c]	37.8	27.6	27 5/8	0.610	5/8	5/16	10.0	10	1.10	1 1/8	1.70	2	1 1/8	23 5/8	5 1/2
×114[c]	33.5	27.3	27 1/4	0.570	9/16	5/16	10.1	10 1/8	0.930	15/16	1.53	1 13/16	1 1/8		
×102[c]	30.0	27.1	27 1/8	0.515	1/2	1/4	10.0	10	0.830	13/16	1.43	1 3/4	1 1/16		
×94[c]	27.7	26.9	26 7/8	0.490	1/2	1/4	10.0	10	0.745	3/4	1.34	1 5/8	1 1/16		
×84[c]	24.8	26.7	26 3/4	0.460	7/16	1/4	10.0	10	0.640	5/8	1.24	1 9/16	1 1/16		

[c] Shape is slender for compression with F_y = 50 ksi.
[g] The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.
[h] Flange thickness greater than 2 in. Special requirements may apply per AISC Specification Section A3.1c.
[v] Shape does not meet the h/t_w limit for shear in Specification Section G2.1a with F_y = 50 ksi.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

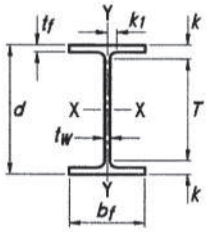


Table 1–1 (continued)
W Shapes
Dimensions

Shape	Area, A	Depth, d		Web Thickness, t_w		$\frac{t_w}{2}$	Flange Width, b_f		Flange Thickness, t_f		Distance k: k_{des}	k_{det}	k_1	T	Workable Gage
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.	in.
W24×370[h]	109	28.0	28	1.52	1 1/2	3/4	13.7	13 5/8	2.72	2 3/4	3.22	3 5/8	1 9/16	20 3/4	5 1/2
×335[h]	98.4	27.5	27 1/2	1.38	1 3/8	11/16	13.5	13 1/2	2.48	2 1/2	2.98	3 3/8	1 1/2		
×306[h]	89.8	27.1	27 1/8	1.26	1 1/4	5/8	13.4	13 3/8	2.28	2 1/4	2.78	3 3/16	1 7/16		
x279[h]	82.0	26.7	26 3/4	1.16	1 3/16	5/8	13.3	13 1/4	2.09	2 1/16	2.59	3	1 7/16		
×250	73.5	26.3	26 3/8	1.04	1 1/16	9/16	13.2	13 1/8	1.89	1 7/8	2.39	2 13/16	1 3/8		
×229	67.2	26.0	26	0.960	15/16	1/2	13.1	13 1/8	1.73	1 3/4	2.23	2 5/8	1 5/16		
×207	60.7	25.7	25 3/4	0.870	7/8	7/16	13.0	13	1.57	1 9/16	2.07	2 1/2	1 1/4		
×192	56.3	25.5	25 1/2	0.810	13/16	7/16	13.0	13	1.46	1 7/16	1.96	2 3/8	1 1/4		
×176	51.7	25.2	25 1/4	0.750	3/4	3/8	12.9	12 7/8	1.34	1 5/16	1.84	2 1/4	1 3/16		
×162	47.7	25.0	25	0.705	11/16	3/8	13.0	13	1.22	1 1/4	1.72	2 1/8	1 3/16		
×146	43.0	24.7	24 3/4	0.650	5/8	5/16	12.9	12 7/8	1.09	1 1/16	1.59	2	1 1/8		
×131	38.5	24.5	24 1/2	0.605	5/8	5/16	12.9	12 7/8	0.960	15/16	1.46	1 7/8	1 1/8		
×117[c]	34.4	24.3	24 1/4	0.550	9/16	5/16	12.8	12 3/4	0.850	7/8	1.35	1 3/4	1 1/8		
×104[c]	30.6	24.1	24	0.500	1/2	1/4	12.8	12 3/4	0.750	3/4	1.25	1 5/8	1 1/16		
W24×103[c]	30.3	24.5	24 1/2	0.550	9/16	5/16	9.00	9	0.980	1	1.48	1 7/8	1 1/8	20 3/4	5 1/2
×94[c]	27.7	24.3	24 1/4	0.515	1/2	1/4	9.07	9 1/8	0.875	7/8	1.38	1 3/4	1 1/16		
×84[c]	24.7	24.1	24 1/8	0.470	1/2	1/4	9.02	9	0.770	3/4	1.27	1 11/16	1 1/16		
×76[c]	22.4	23.9	23 7/8	0.440	7/16	1/4	8.99	9	0.680	11/16	1.18	1 9/16	1 1/16		
×68[c]	20.1	23.7	23 3/4	0.415	7/16	1/4	8.97	9	0.585	9/16	1.09	1 1/2	1 1/16		
W24×62[c]	18.2	23.7	23 3/4	0.430	7/16	1/4	7.04	7	0.590	9/16	1.09	1 1/2	1 1/16	20 3/4	3 1/2[g]
×55[c,v]	16.2	23.6	23 5/8	0.395	3/8	3/16	7.01	7	0.505	1/2	1.01	1 7/16	1	20 3/4	3 1/2[g]
W21×201	59.2	23.0	23	0.910	15/16	1/2	12.6	12 5/8	1.63	1 5/8	2.13	2 1/2	1 5/16	18	5 1/2
×182	53.6	22.7	22 3/4	0.830	13/16	7/16	12.5	12 1/2	1.48	1 1/2	1.98	2 3/8	1 1/4		
×166	48.8	22.5	22 1/2	0.750	3/4	3/8	12.4	12 3/8	1.36	1 3/8	1.86	2 1/4	1 3/16		
×147	43.2	22.1	22	0.720	3/4	3/8	12.5	12 1/2	1.15	1 1/8	1.65	2	1 3/16		
×132	38.8	21.8	21 7/8	0.650	5/8	5/16	12.4	12 1/2	1.04	1 1/16	1.54	1 15/16	1 1/8		
×122	35.9	21.7	21 5/8	0.600	5/8	5/16	12.4	12 3/8	0.960	15/16	1.46	1 13/16	1 1/8		
×111	32.7	21.5	21 1/2	0.550	9/16	5/16	12.3	12 3/8	0.875	7/8	1.38	1 3/4	1 1/8		
×101[c]	29.8	21.4	21 3/8	0.500	1/2	1/4	12.3	12 1/4	0.800	13/16	1.30	1 11/16	1 1/16		

[c] Shape is slender for compression with $F_y = 50$ ksi.
[g] The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.
[h] Flange thickness greater than 2 in. Special requirements may apply per AISC Specification Section A3.1c.
[v] Shape does not meet the h/t_w limit for shear in Specification Section G2.1a with $F_y = 50$ ksi.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

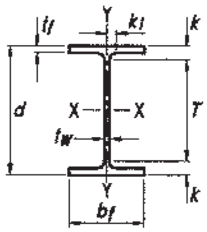


Table 1–1 (continued)
W Shapes
Dimensions

Shape	Area, A	Depth, d		Web			Flange				Distance				
				Thickness, t_w		$\frac{t_w}{2}$	Width, b_f		Thickness, t_f		k		k_1	T	Workable Gage
											k_{des}	k_{det}			
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.	in.
W21×93	27.3	21.6	21 5/8	0.580	9/16	5/16	8.42	8 3/8	0.930	15/16	1.43	1 5/8	15/16	18 3/8	5 1/2
×83[c]	24.3	21.4	21 3/8	0.515	1/2	1/4	8.36	8 3/8	0.835	13/16	1.34	1 1/2	7/8		
×73[c]	21.5	21.2	21 1/4	0.455	7/16	1/4	8.30	8 1/4	0.740	3/4	1.24	1 7/16	7/8		
×68[c]	20.0	21.1	21 1/8	0.430	7/16	1/4	8.27	8 1/4	0.685	11/16	1.19	1 3/8	7/8		
×62[c]	18.3	21.0	21	0.400	3/8	3/16	8.24	8 1/4	0.615	5/8	1.12	1 5/16	13/16		
×55[c]	16.2	20.8	20 3/4	0.375	3/8	3/16	8.22	8 1/4	0.522	1/2	1.02	1 3/16	13/16		
×48[c,f]	14.1	20.6	20 5/8	0.350	3/8	3/16	8.14	8 1/8	0.430	7/16	0.930	1 1/8	13/16		
W21×57[c]	16.7	21.1	21	0.405	3/8	3/16	6.56	6 1/2	0.650	5/8	1.15	1 5/16	13/16	18 3/8	3 1/2
×50[c]	14.7	20.8	20 7/8	0.380	3/8	3/16	6.53	6 1/2	0.535	9/16	1.04	1 1/4	13/16		
×44[c]	13.0	20.7	20 5/8	0.350	3/8	3/16	6.50	6 1/2	0.450	7/16	0.950	1 1/8	13/16		
W18×311[h]	91.6	22.3	22 3/8	1.52	1 1/2	3/4	12.0	12	2.74	2 3/4	3.24	3 7/16	1 3/8	15 1/2	5 1/2
×283[h]	83.3	21.9	21 7/8	1.40	1 3/8	11/16	11.9	11 7/8	2.50	2 1/2	3.00	3 3/16	1 5/16		
×258[h]	75.9	21.5	21 1/2	1.28	1 1/4	5/8	11.8	11 3/4	2.30	2 5/16	2.70	3	1 1/4		
×234[h]	68.8	21.1	21	1.16	1 3/16	5/8	11.7	11 5/8	2.11	2 1/8	2.51	2 3/4	1 3/16		
×211	62.1	20.7	20 5/8	1.06	1 1/16	9/16	11.6	11 1/2	1.91	1 15/16	2.31	2 9/16	1 3/16		
×192	56.4	20.4	20 3/8	0.960	15/16	1/2	11.5	11 1/2	1.75	1 3/4	2.15	2 7/16	1 1/8		
×175	51.3	20.0	20	0.890	7/8	7/16	11.4	11 3/8	1.59	1 9/16	1.99	2 7/16	1 1/4	15 1/8	
×158	46.3	19.7	19 3/4	0.810	13/16	7/16	11.3	11 1/4	1.44	1 7/16	1.84	2 3/8	1 1/4		
×143	42.1	19.5	19 1/2	0.730	3/4	3/8	11.2	11 1/4	1.32	1 5/16	1.72	2 3/16	1 3/16		
×130	38.2	19.3	19 1/4	0.670	11/16	3/8	11.2	11 1/8	1.20	1 3/16	1.60	2 1/16	1 3/16		
×119	35.1	19.0	19	0.655	5/8	5/16	11.3	11 1/4	1.06	1 1/16	1.46	1 15/16	1 3/16		
×106	31.1	18.7	18 3/4	0.590	9/16	5/16	11.2	11 1/4	0.940	15/16	1.34	1 13/16	1 1/8		
×97	28.5	18.6	18 5/8	0.535	9/16	5/16	11.1	11 1/8	0.870	7/8	1.27	1 3/4	1 1/8		
×86	25.3	18.4	18 3/8	0.480	1/2	1/4	11.1	11 1/8	0.770	3/4	1.17	1 5/8	1 1/16		
×76[c]	22.3	18.2	18 1/4	0.425	7/16	1/4	11.0	11	0.680	11/16	1.08	1 9/16	1 1/16		
W18×71	20.8	18.5	18 1/2	0.495	1/2	1/4	7.64	7 5/8	0.810	13/16	1.21	1 1/2	7/8	15 1/2	3 1/2[g]
×65	19.1	18.4	18 3/8	0.450	7/16	1/4	7.59	7 5/8	0.750	3/4	1.15	1 7/16	7/8		
×60[c]	17.6	18.2	18 1/4	0.415	7/16	1/4	7.56	7 1/2	0.695	11/16	1.10	1 3/8	13/16		
×55[c]	16.2	18.1	18 1/8	0.390	3/8	3/16	7.53	7 1/2	0.630	5/8	1.03	1 5/16	13/16		
×50[c]	14.7	18.0	18	0.355	3/8	3/16	7.50	7 1/2	0.570	9/16	0.972	1 1/4	13/16		
W18×46[c]	13.5	18.1	18	0.360	3/8	3/16	6.06	6	0.605	5/8	1.01	1 1/4	13/16	15 1/2	3 1/2[g]
×40[c]	11.8	17.9	17 7/8	0.315	5/16	3/16	6.02	6	0.525	1/2	0.927	1 3/16	13/16		
×35[c]	10.3	17.7	17 3/4	0.300	5/16	3/16	6.00	6	0.425	7/16	0.827	1 1/8	3/4		

[c] Shape is slender for compression with $F_y = 50$ ksi.
[f] Shape exceeds compact limit for flexure with $F_y = 50$ ksi.
[g] The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.
[h] Flange thickness greater than 2 in. Special requirements may apply per AISC Specification Section A3.1c.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

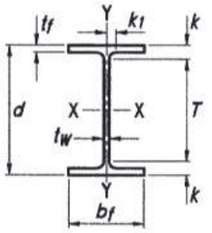


Table 1–1 (continued)
W Shapes
Dimensions

Shape	Area, A	Depth, d		Web: Thickness, t_w		Web: $\frac{t_w}{2}$	Flange: Width, b_f		Flange: Thickness, t_f		Distance: k_{des}	Distance: k_{det}	Distance: k_1	Distance: T	Distance: Workable Gage
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.	in.
W16×100	29.5	17.0	17	0.585	9/16	5/16	10.4	10 3/8	0.985	1	1.39	1 7/8	1 1/8	13 1/4	5 1/2
×89	26.2	16.8	16 3/4	0.525	1/2	1/4	10.4	10 3/8	0.875	7/8	1.28	1 3/4	1 1/16	↓	↓
×77	22.6	16.5	16 1/2	0.455	7/16	1/4	10.3	10 1/4	0.760	3/4	1.16	1 5/8	1 1/16	↓	↓
×67^{c}	19.7	16.3	16 3/8	0.395	3/8	3/16	10.2	10 1/4	0.665	11/16	1.07	1 9/16	1	↓	↓
W16×57	16.8	16.4	16 3/8	0.430	7/16	1/4	7.12	7 1/8	0.715	11/16	1.12	1 3/8	7/8	13 5/8	3 1/2^{g}
×50^{c}	14.7	16.3	16 1/4	0.380	3/8	3/16	7.07	7 1/8	0.630	5/8	1.03	1 5/16	13/16	↓	↓
×45^{c}	13.3	16.1	16 1/8	0.345	3/8	3/16	7.04	7	0.565	9/16	0.967	1 1/4	13/16	↓	↓
×40^{c}	11.8	16.0	16	0.305	5/16	3/16	7.00	7	0.505	1/2	0.907	1 3/16	13/16	↓	↓
×36^{c}	10.6	15.9	15 7/8	0.295	5/16	3/16	6.99	7	0.430	7/16	0.832	1 1/8	3/4	↓	↓
W16×31^{c}	9.13	15.9	15 7/8	0.275	1/4	1/8	5.53	5 1/2	0.440	7/16	0.842	1 1/8	3/4	13 5/8	3 1/2
×26^{c,v}	7.68	15.7	15 3/4	0.250	1/4	1/8	5.50	5 1/2	0.345	3/8	0.747	1 1/16	3/4	13 5/8	3 1/2
W14×730^{h}	215	22.4	22 3/8	3.07	3 1/16	1 9/16	17.9	17 7/8	4.91	4 15/16	5.51	6 3/16	2 3/4	10	3-7 1/2-3^{g}
×665^{h}	196	21.6	21 5/8	2.83	2 13/16	1 7/16	17.7	17 5/8	4.52	4 1/2	5.12	5 13/16	2 5/8	↓	3-7 1/2-3^{g}
×605^{h}	178	20.9	20 7/8	2.60	2 5/8	1 5/16	17.4	17 3/8	4.16	4 3/16	4.76	5 7/16	2 1/2	↓	3-7 1/2-3
×550^{h}	162	20.2	20 1/4	2.38	2 3/8	1 3/16	17.2	17 1/4	3.82	3 13/16	4.42	5 1/8	2 3/8	↓	↓
×500^{h}	147	19.6	19 5/8	2.19	2 3/16	1 1/8	17.0	17	3.50	3 1/2	4.10	4 13/16	2 5/16	↓	↓
×455^{h}	134	19.0	19	2.02	2	1	16.8	16 7/8	3.21	3 3/16	3.81	4 1/2	2 1/4	↓	↓
×426^{h}	125	18.7	18 5/8	1.88	1 7/8	15/16	16.7	16 3/4	3.04	3 1/16	3.63	4 5/16	2 1/8	↓	↓
×398^{h}	117	18.3	18 1/4	1.77	1 3/4	7/8	16.6	16 5/8	2.85	2 7/8	3.44	4 1/8	2 1/8	↓	↓
×370^{h}	109	17.9	17 7/8	1.66	1 5/8	13/16	16.5	16 1/2	2.66	2 11/16	3.26	3 15/16	2 1/16	↓	↓
×342^{h}	101	17.5	17 1/2	1.54	1 9/16	13/16	16.4	16 3/8	2.47	2 1/2	3.07	3 3/4	2	↓	↓
×311^{h}	91.4	17.1	17 1/8	1.41	1 7/16	3/4	16.2	16 1/4	2.26	2 1/4	2.86	3 9/16	1 15/16	↓	↓
×283^{h}	83.3	16.7	16 3/4	1.29	1 5/16	11/16	16.1	16 1/8	2.07	2 1/16	2.67	3 3/8	1 7/8	↓	↓
×257	75.6	16.4	16 3/8	1.18	1 3/16	5/8	16.0	16	1.89	1 7/8	2.49	3 3/16	1 13/16	↓	↓
×233	68.5	16.0	16	1.07	1 1/16	9/16	15.9	15 7/8	1.72	1 3/4	2.32	3	1 3/4	↓	↓
×211	62.0	15.7	15 3/4	0.980	1	1/2	15.8	15 3/4	1.56	1 9/16	2.16	2 7/8	1 11/16	↓	↓
×193	56.8	15.5	15 1/2	0.890	7/8	7/16	15.7	15 3/4	1.44	1 7/16	2.04	2 3/4	1 11/16	↓	↓
×176	51.8	15.2	15 1/4	0.830	13/16	7/16	15.7	15 5/8	1.31	1 5/16	1.91	2 5/8	1 5/8	↓	↓
×159	46.7	15.0	15	0.745	3/4	3/8	15.6	15 5/8	1.19	1 3/16	1.79	2 1/2	1 9/16	↓	↓
×145	42.7	14.8	14 3/4	0.680	11/16	3/8	15.5	15 1/2	1.09	1 1/16	1.69	2 3/8	1 9/16	↓	↓

c Shape is slender for compression with F_y = 50 ksi.
g The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.
h Flange thickness greater than 2 in. Special requirements may apply per AISC Specification Section A3.1c.
v Shape does not meet the h/t_w limit for shear in Specification Section G2.1a with F_y = 50 ksi.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

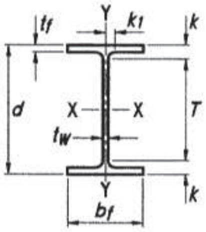


Table 1–1 (continued)
W Shapes
Dimensions

Shape	Area, A	Depth, d		Web Thickness, t_w		Web $\frac{t_w}{2}$	Flange Width, b_f		Flange Thickness, t_f		Distance k k_{des}	Distance k k_{det}	Distance k_1	Distance T	Distance Workable Gage
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.	in.
W14×132	38.8	14.7	14 5/8	0.645	5/8	5/16	14.7	14 3/4	1.03	1	1.63	2 5/16	1 9/16	10	5 1/2
×120	35.3	14.5	14 1/2	0.590	9/16	5/16	14.7	14 5/8	0.940	15/16	1.54	2 1/4	1 1/2		
×109	32.0	14.3	14 3/8	0.525	1/2	1/4	14.6	14 5/8	0.860	7/8	1.46	2 3/16	1 1/2		
×99[f]	29.1	14.2	14 1/8	0.485	1/2	1/4	14.6	14 5/8	0.780	3/4	1.38	2 1/16	1 7/16		
×90[f]	26.5	14.0	14	0.440	7/16	1/4	14.5	14 1/2	0.710	11/16	1.31	2	1 7/16		
W14×82	24.0	14.3	14 1/4	0.510	1/2	1/4	10.1	10 1/8	0.855	7/8	1.45	1 11/16	1 1/16	10 7/8	5 1/2
×74	21.8	14.2	14 1/8	0.450	7/16	1/4	10.1	10 1/8	0.785	13/16	1.38	1 5/8	1 1/16		
×68	20.0	14.0	14	0.415	7/16	1/4	10.0	10	0.720	3/4	1.31	1 9/16	1 1/16		
×61	17.9	13.9	13 7/8	0.375	3/8	3/16	10.0	10	0.645	5/8	1.24	1 1/2	1		
W14×53	15.6	13.9	13 7/8	0.370	3/8	3/16	8.06	8	0.660	11/16	1.25	1 1/2	1	10 7/8	5 1/2
×48	14.1	13.8	13 3/4	0.340	5/16	3/16	8.03	8	0.595	5/8	1.19	1 7/16	1		
×43[c]	12.6	13.7	13 5/8	0.305	5/16	3/16	8.00	8	0.530	1/2	1.12	1 3/8	1		
W14×38[c]	11.2	14.1	14 1/8	0.310	5/16	3/16	6.77	6 3/4	0.515	1/2	0.915	1 1/4	13/16	11 5/8	3 1/2[g]
×34[c]	10.0	14.0	14	0.285	5/16	3/16	6.75	6 3/4	0.455	7/16	0.855	1 3/16	3/4		3 1/2
×30[c]	8.85	13.8	13 7/8	0.270	1/4	1/8	6.73	6 3/4	0.385	3/8	0.785	1 1/8	3/4		3 1/2
W14×26[c]	7.69	13.9	13 7/8	0.255	1/4	1/8	5.03	5	0.420	7/16	0.820	1 1/8	3/4	11 5/8	2 3/4[g]
×22[c]	6.49	13.7	13 3/4	0.230	1/4	1/8	5.00	5	0.335	5/16	0.735	1 1/16	3/4	11 5/8	2 3/4[g]
W12×336[h]	98.8	16.8	16 7/8	1.78	1 3/4	7/8	13.4	13 3/8	2.96	2 15/16	3.55	3 7/8	1 11/16	9 1/8	5 1/2
×305[h]	89.6	16.3	16 3/8	1.63	1 5/8	13/16	13.2	13 1/4	2.71	2 11/16	3.30	3 5/8	1 5/8		
×279[h]	81.9	15.9	15 7/8	1.53	1 1/2	3/4	13.1	13 1/8	2.47	2 1/2	3.07	3 3/8	1 5/8		
×252[h]	74.0	15.4	15 3/8	1.40	1 3/8	11/16	13.0	13	2.25	2 1/4	2.85	3 1/8	1 1/2		
×230[h]	67.7	15.1	15	1.29	1 5/16	11/16	12.9	12 7/8	2.07	2 1/16	2.67	2 15/16	1 1/2		
×210	61.8	14.7	14 3/4	1.18	1 3/16	5/8	12.8	12 3/4	1.90	1 7/8	2.50	2 13/16	1 7/16		
×190	55.8	14.4	14 3/8	1.06	1 1/16	9/16	12.7	12 5/8	1.74	1 3/4	2.33	2 5/8	1 3/8		
×170	50.0	14.0	14	0.960	15/16	1/2	12.6	12 5/8	1.56	1 9/16	2.16	2 7/16	1 5/16		
×152	44.7	13.7	13 3/4	0.870	7/8	7/16	12.5	12 1/2	1.40	1 3/8	2.00	2 5/16	1 1/4		
×136	39.9	13.4	13 3/8	0.790	13/16	7/16	12.4	12 3/8	1.25	1 1/4	1.85	2 1/8	1 1/4		
×120	35.3	13.1	13 1/8	0.710	11/16	3/8	12.3	12 3/8	1.11	1 1/8	1.70	2	1 3/16		
×106	31.2	12.9	12 7/8	0.610	5/8	5/16	12.2	12 1/4	0.990	1	1.59	1 7/8	1 1/8		
×96	28.2	12.7	12 3/4	0.550	9/16	5/16	12.2	12 1/8	0.900	7/8	1.50	1 13/16	1 1/8		
×87	25.6	12.5	12 1/2	0.515	1/2	1/4	12.1	12 1/8	0.810	13/16	1.41	1 11/16	1 1/16		
×79	23.2	12.4	12 3/8	0.470	1/2	1/4	12.1	12 1/8	0.735	3/4	1.33	1 5/8	1 1/16		
×72	21.1	12.3	12 1/4	0.430	7/16	1/4	12.0	12	0.670	11/16	1.27	1 9/16	1 1/16		
×65[f]	19.1	12.1	12 1/8	0.390	3/8	3/16	12.0	12	0.605	5/8	1.20	1 1/2	1		

[c] Shape is slender for compression with $F_y = 50$ ksi.
[f] Shape exceeds compact limit for flexure with $F_y = 50$ ksi.
[g] The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.
[h] Flange thickness greater than 2 in. Special requirements may apply per AISC Specification Section A3.1c.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

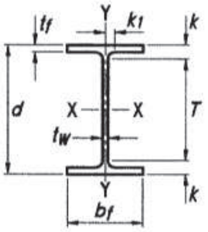


Table 1–1 (continued)
W Shapes
Dimensions

Shape	Area, A	Depth, d		Web Thickness, t_w		Web $\frac{t_w}{2}$	Flange Width, b_f		Flange Thickness, t_f		Distance k k_{des}	Distance k k_{det}	k_1	T	Workable Gage
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.	in.
W12×58	17.0	12.2	12 1/4	0.360	3/8	3/16	10.0	10	0.640	5/8	1.24	1 1/2	15/16	9 1/4	5 1/2
×53	15.6	12.1	12	0.345	3/8	3/16	10.0	10	0.575	9/16	1.18	1 3/8	15/16	9 1/4	5 1/2
W12×50	14.6	12.2	12 1/4	0.370	3/8	3/16	8.08	8 1/8	0.640	5/8	1.14	1 1/2	15/16	9 1/4	5 1/2
×45	13.1	12.1	12	0.335	5/16	3/16	8.05	8	0.575	9/16	1.08	1 3/8	15/16	↓	↓
×40	11.7	11.9	12	0.295	5/16	3/16	8.01	8	0.515	1/2	1.02	1 3/8	7/8	↓	↓
W12×35[c]	10.3	12.5	12 1/2	0.300	5/16	3/16	6.56	6 1/2	0.520	1/2	0.820	1 3/16	3/4	10 1/8	3 1/2
×30[c]	8.79	12.3	12 3/8	0.260	1/4	1/8	6.52	6 1/2	0.440	7/16	0.740	1 1/8	3/4	↓	↓
×26[c]	7.65	12.2	12 1/4	0.230	1/4	1/8	6.49	6 1/2	0.380	3/8	0.680	1 1/16	3/4	↓	↓
W12×22[c]	6.48	12.3	12 1/4	0.260	1/4	1/8	4.03	4	0.425	7/16	0.725	15/16	5/8	10 3/8	2 1/4[g]
×19[c]	5.57	12.2	12 1/8	0.235	1/4	1/8	4.01	4	0.350	3/8	0.650	7/8	9/16	↓	↓
×16[c]	4.71	12.0	12	0.220	1/4	1/8	3.99	4	0.265	1/4	0.565	13/16	9/16	↓	↓
×14[c,v]	4.16	11.9	11 7/8	0.200	3/16	1/8	3.97	4	0.225	1/4	0.525	3/4	9/16	↓	↓
W10×112	32.9	11.4	11 3/8	0.755	3/4	3/8	10.4	10 3/8	1.25	1 1/4	1.75	1 15/16	1	7 1/2	5 1/2
×100	29.4	11.1	11 1/8	0.680	11/16	3/8	10.3	10 3/8	1.12	1 1/8	1.62	1 13/16	1	↓	↓
×88	25.9	10.8	10 7/8	0.605	5/8	5/16	10.3	10 1/4	0.990	1	1.49	1 11/16	15/16	↓	↓
×77	22.6	10.6	10 5/8	0.530	1/2	1/4	10.2	10 1/4	0.870	7/8	1.37	1 9/16	7/8	↓	↓
×68	20.0	10.4	10 3/8	0.470	1/2	1/4	10.1	10 1/8	0.770	3/4	1.27	1 7/16	7/8	↓	↓
×60	17.6	10.2	10 1/4	0.420	7/16	1/4	10.1	10 1/8	0.680	11/16	1.18	1 3/8	13/16	↓	↓
×54	15.8	10.1	10 1/8	0.370	3/8	3/16	10.0	10	0.615	5/8	1.12	1 5/16	13/16	↓	↓
×49	14.4	10.0	10	0.340	5/16	3/16	10.0	10	0.560	9/16	1.06	1 1/4	13/16	↓	↓
W10×45	13.3	10.1	10 1/8	0.350	3/8	3/16	8.02	8	0.620	5/8	1.12	1 5/16	13/16	7 1/2	5 1/2
×39	11.5	9.92	9 7/8	0.315	5/16	3/16	7.99	8	0.530	1/2	1.03	1 3/16	13/16	↓	↓
×33	9.71	9.73	9 3/4	0.290	5/16	3/16	7.96	8	0.435	7/16	0.935	1 1/8	3/4	↓	↓
W10×30	8.84	10.5	10 1/2	0.300	5/16	3/16	5.81	5 3/4	0.510	1/2	0.810	1 1/8	11/16	8 1/4	2 3/4[g]
×26	7.61	10.3	10 3/8	0.260	1/4	1/8	5.77	5 3/4	0.440	7/16	0.740	1 1/16	11/16	↓	↓
×22[c]	6.49	10.2	10 1/8	0.240	1/4	1/8	5.75	5 3/4	0.360	3/8	0.660	15/16	5/8	↓	↓
W10×19	5.62	10.2	10 1/4	0.250	1/4	1/8	4.02	4	0.395	3/8	0.695	15/16	5/8	8 3/8	2 1/4[g]
×17[c]	4.99	10.1	10 1/8	0.240	1/4	1/8	4.01	4	0.330	5/16	0.630	7/8	9/16	↓	↓
×15[c]	4.41	10.0	10	0.230	1/4	1/8	4.00	4	0.270	1/4	0.570	13/16	9/16	↓	↓
×12[c,f]	3.54	9.87	9 7/8	0.190	3/16	1/8	3.96	4	0.210	3/16	0.510	3/4	9/16	↓	↓

[c] Shape is slender for compression with $F_y = 50$ ksi.
[f] Shape exceeds compact limit for flexure with $F_y = 50$ ksi.
[g] The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.
[v] Shape does not meet the h/t_w limit for shear in Specification Section G2.1a with $F_y = 50$ ksi.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

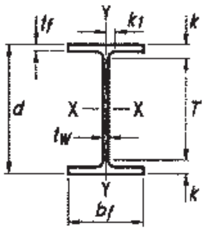


Table 1–1 (continued)
W Shapes
Dimensions

Shape	Area, A	Depth, d		Web: Thickness, t_w		Web: $\frac{t_w}{2}$	Flange: Width, b_f		Flange: Thickness, t_f		Distance: k, k_{des}	Distance: k, k_{det}	Distance: k_1	Distance: T	Distance: Workable Gage
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.	in.
W8×67	19.7	9.00	9	0.570	9/16	5/16	8.28	8 1/4	0.935	15/16	1.33	1 5/8	15/16	5 3/4	5 1/2
×58	17.1	8.75	8 3/4	0.510	1/2	1/4	8.22	8 1/4	0.810	13/16	1.20	1 1/2	7/8		
×48	14.1	8.50	8 1/2	0.400	3/8	3/16	8.11	8 1/8	0.685	11/16	1.08	1 3/8	13/16		
×40	11.7	8.25	8 1/4	0.360	3/8	3/16	8.07	8 1/8	0.560	9/16	0.954	1 1/4	13/16		
×35	10.3	8.12	8 1/8	0.310	5/16	3/16	8.02	8	0.495	1/2	0.889	1 3/16	13/16		
×31[f]	9.12	8.00	8	0.285	5/16	3/16	8.00	8	0.435	7/16	0.829	1 1/8	3/4		
W8×28	8.24	8.06	8	0.285	5/16	3/16	6.54	6 1/2	0.465	7/16	0.859	15/16	5/8	6 1/8	4
×24	7.08	7.93	7 7/8	0.245	1/4	1/8	6.50	6 1/2	0.400	3/8	0.794	7/8	9/16	6 1/8	4
W8×21	6.16	8.28	8 1/4	0.250	1/4	1/8	5.27	5 1/4	0.400	3/8	0.700	7/8	9/16	6 1/2	2 3/4[g]
×18	5.26	8.14	8 1/8	0.230	1/4	1/8	5.25	5 1/4	0.330	5/16	0.630	13/16	9/16	6 1/2	2 3/4[g]
W8×15	4.44	8.11	8 1/8	0.245	1/4	1/8	4.02	4	0.315	5/16	0.615	13/16	9/16	6 1/2	2 1/4[g]
×13	3.84	7.99	8	0.230	1/4	1/8	4.00	4	0.255	1/4	0.555	3/4	9/16		
×10[c,f]	2.96	7.89	7 7/8	0.170	3/16	1/8	3.94	4	0.205	3/16	0.505	11/16	1/2		
W6×25	7.34	6.38	6 3/8	0.320	5/16	3/16	6.08	6 1/8	0.455	7/16	0.705	15/16	9/16	4 1/2	3 1/2
×20	5.87	6.20	6 1/4	0.260	1/4	1/8	6.02	6	0.365	3/8	0.615	7/8	9/16		
×15[f]	4.43	5.99	6	0.230	1/4	1/8	5.99	6	0.260	1/4	0.510	3/4	9/16		
W6×16	4.74	6.28	6 1/4	0.260	1/4	1/8	4.03	4	0.405	3/8	0.655	7/8	9/16	4 1/2	2 1/4[g]
×12	3.55	6.03	6	0.230	1/4	1/8	4.00	4	0.280	1/4	0.530	3/4	9/16		
×9[f]	2.68	5.90	5 7/8	0.170	3/16	1/8	3.94	4	0.215	3/16	0.465	11/16	1/2		
×8.5[f]	2.52	5.83	5 7/8	0.170	3/16	1/8	3.94	4	0.195	3/16	0.445	11/16	1/2		
W5×19	5.56	5.15	5 1/8	0.270	1/4	1/8	5.03	5	0.430	7/16	0.730	13/16	7/16	3 1/2	2 3/4[g]
×16	4.71	5.01	5	0.240	1/4	1/8	5.00	5	0.360	3/8	0.660	3/4	7/16	3 1/2	2 3/4[g]
W4×13	3.83	4.16	4 1/8	0.280	1/4	1/8	4.06	4	0.345	3/8	0.595	3/4	1/2	2 5/8	2 1/4[g]

[c] Shape is slender for compression with F_y = 50 ksi.
[f] Shape exceeds compact limit for flexure with F_y = 50 ksi.
[g] The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

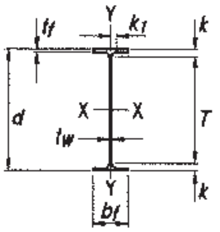


Table 1–2
M Shapes
Dimensions

Shape	Area, A	Depth, d		Web Thickness, t_w		Web $\frac{t_w}{2}$	Flange Width, b_f		Flange Thickness, t_f		Distance k	Distance k_1	Distance T	Workable Gage
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.
M12.5×12.4^{c,v}	3.63	12.5	12 1/2	0.155	1/8	1/16	3.75	3 3/4	0.228	1/4	9/16	3/8	11 3/8	—
×11.6^{c,v}	3.40	12.5	12 1/2	0.155	1/8	1/16	3.50	3 1/2	0.211	3/16	9/16	3/8	11 3/8	—
M12×11.8^{c}	3.47	12.0	12	0.177	3/16	1/8	3.07	3 1/8	0.225	1/4	9/16	3/8	10 7/8	—
×10.8^{c}	3.18	12.0	12	0.160	3/16	1/8	3.07	3 1/8	0.210	3/16	9/16	3/8	10 7/8	—
M12×10^{c,v}	2.95	12.0	12	0.149	1/8	1/16	3.25	3 1/4	0.180	3/16	1/2	3/8	11	—
M10×9^{c}	2.65	10.0	10	0.157	3/16	1/8	2.69	2 3/4	0.206	3/16	9/16	3/8	8 7/8	—
×8^{c}	2.37	9.95	10	0.141	1/8	1/16	2.69	2 3/4	0.182	3/16	9/16	3/8	8 7/8	—
M10×7.5^{c,v}	2.22	9.99	10	0.130	1/8	1/16	2.69	2 3/4	0.173	3/16	7/16	5/16	9 1/8	—
M8×6.5^{c}	1.92	8.00	8	0.135	1/8	1/16	2.28	2 1/4	0.189	3/16	9/16	3/8	6 7/8	—
×6.2^{c}	1.82	8.00	8	0.129	1/8	1/16	2.28	2 1/4	0.177	3/16	7/16	1/4	7 1/8	—
M6×4.4^{c}	1.29	6.00	6	0.114	1/8	1/16	1.84	1 7/8	0.171	3/16	3/8	1/4	5 1/4	—
×3.7^{c}	1.09	5.92	5 7/8	0.0980	1/8	1/16	2.00	2	0.129	1/8	5/16	1/4	5 1/4	—
M5×18.9^{t}	5.56	5.00	5	0.316	5/16	3/16	5.00	5	0.416	7/16	13/16	1/2	3 3/8	2 3/4^{g}
M4×6^{f}	1.75	3.80	3 3/4	0.130	1/8	1/16	3.80	3 3/4	0.160	3/16	1/2	3/8	2 3/4	—
×4.08	1.27	4.00	4	0.115	1/8	1/16	2.25	2 1/4	0.170	3/16	9/16	3/8	2 7/8	—
×3.45	1.01	4.00	4	0.0920	1/16	1/16	2.25	2 1/4	0.130	1/8	1/2	3/8	3	—
×3.2	1.01	4.00	4	0.0920	1/16	1/16	2.25	2 1/4	0.130	1/8	1/2	3/8	3	—
M3×2.9	0.914	3.00	3	0.0900	1/16	1/16	2.25	2 1/4	0.130	1/8	1/2	3/8	2	—

[c] Shape is slender for compression with F_y = 36 ksi.
[f] Shape exceeds compact limit for flexure with F_y = 36 ksi.
[g] The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.
[t] Shape has tapered flanges while other M-shapes have parallel flange surfaces.
[v] Shape does not meet the h/t_w limit for shear in Specification Section G2.1b(i) with F_y = 36 ksi.
— Flange is too narrow to establish a workable gage.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

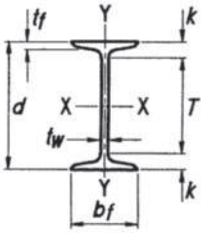


Table 1–3
S Shapes
Dimensions

Shape	Area, A	Depth, d		Web			Flange				Distance		
				Thickness, t_w		$\frac{t_w}{2}$	Width, b_f		Thickness, t_f		k	T	Workable Gage
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.
S24×121	35.5	24.5	$24\frac{1}{2}$	0.800	$\frac{13}{16}$	$\frac{7}{16}$	8.05	8	1.09	$1\frac{1}{16}$	2	$20\frac{1}{2}$	4
×106	31.1	24.5	$24\frac{1}{2}$	0.620	$\frac{5}{8}$	$\frac{5}{16}$	7.87	$7\frac{7}{8}$	1.09	$1\frac{1}{16}$	2	$20\frac{1}{2}$	4
S24×100	29.3	24.0	24	0.745	$\frac{3}{4}$	$\frac{3}{8}$	7.25	$7\frac{1}{4}$	0.870	$\frac{7}{8}$	$1\frac{3}{4}$	$20\frac{1}{2}$	4
×90	26.5	24.0	24	0.625	$\frac{5}{8}$	$\frac{5}{16}$	7.13	$7\frac{1}{8}$	0.870	$\frac{7}{8}$	$1\frac{3}{4}$	$20\frac{1}{2}$	4
×80	23.5	24.0	24	0.500	$\frac{1}{2}$	$\frac{1}{4}$	7.00	7	0.870	$\frac{7}{8}$	$1\frac{3}{4}$	$20\frac{1}{2}$	4
S20×96	28.2	20.3	$20\frac{1}{4}$	0.800	$\frac{13}{16}$	$\frac{7}{16}$	7.20	$7\frac{1}{4}$	0.920	$\frac{15}{16}$	$1\frac{3}{4}$	$16\frac{3}{4}$	4
×86	25.3	20.3	$20\frac{1}{4}$	0.660	$\frac{11}{16}$	$\frac{3}{8}$	7.06	7	0.920	$\frac{15}{16}$	$1\frac{3}{4}$	$16\frac{3}{4}$	4
S20×75	22.0	20.0	20	0.635	$\frac{5}{8}$	$\frac{5}{16}$	6.39	$6\frac{3}{8}$	0.795	$\frac{13}{16}$	$1\frac{5}{8}$	$16\frac{3}{4}$	$3\frac{1}{2}$[g]
×66	19.4	20.0	20	0.505	$\frac{1}{2}$	$\frac{1}{4}$	6.26	$6\frac{1}{4}$	0.795	$\frac{13}{16}$	$1\frac{5}{8}$	$16\frac{3}{4}$	$3\frac{1}{2}$[g]
S18×70	20.5	18.0	18	0.711	$\frac{11}{16}$	$\frac{3}{8}$	6.25	$6\frac{1}{4}$	0.691	$\frac{11}{16}$	$1\frac{1}{2}$	15	$3\frac{1}{2}$[g]
×54.7	16.0	18.0	18	0.461	$\frac{7}{16}$	$\frac{1}{4}$	6.00	6	0.691	$\frac{11}{16}$	$1\frac{1}{2}$	15	$3\frac{1}{2}$[g]
S15×50	14.7	15.0	15	0.550	$\frac{9}{16}$	$\frac{5}{16}$	5.64	$5\frac{5}{8}$	0.622	$\frac{5}{8}$	$1\frac{3}{8}$	$12\frac{1}{4}$	$3\frac{1}{2}$[g]
×42.9	12.6	15.0	15	0.411	$\frac{7}{16}$	$\frac{1}{4}$	5.50	$5\frac{1}{2}$	0.622	$\frac{5}{8}$	$1\frac{3}{8}$	$12\frac{1}{4}$	$3\frac{1}{2}$[g]
S12×50	14.6	12.0	12	0.687	$\frac{11}{16}$	$\frac{3}{8}$	5.48	$5\frac{1}{2}$	0.659	$\frac{11}{16}$	$1\frac{7}{16}$	$9\frac{1}{8}$	3[g]
×40.8	11.9	12.0	12	0.462	$\frac{7}{16}$	$\frac{1}{4}$	5.25	$5\frac{1}{4}$	0.659	$\frac{11}{16}$	$1\frac{7}{16}$	$9\frac{1}{8}$	3[g]
S12×35	10.2	12.0	12	0.428	$\frac{7}{16}$	$\frac{1}{4}$	5.08	$5\frac{1}{8}$	0.544	$\frac{9}{16}$	$1\frac{3}{16}$	$9\frac{5}{8}$	3[g]
×31.8	9.31	12.0	12	0.350	$\frac{3}{8}$	$\frac{3}{16}$	5.00	5	0.544	$\frac{9}{16}$	$1\frac{3}{16}$	$9\frac{5}{8}$	3[g]
S10×35	10.3	10.0	10	0.594	$\frac{5}{8}$	$\frac{5}{16}$	4.94	5	0.491	$\frac{1}{2}$	$1\frac{1}{8}$	$7\frac{3}{4}$	$2\frac{3}{4}$[g]
×25.4	7.45	10.0	10	0.311	$\frac{5}{16}$	$\frac{3}{16}$	4.66	$4\frac{5}{8}$	0.491	$\frac{1}{2}$	$1\frac{1}{8}$	$7\frac{3}{4}$	$2\frac{3}{4}$[g]
S8×23	6.76	8.00	8	0.441	$\frac{7}{16}$	$\frac{1}{4}$	4.17	$4\frac{1}{8}$	0.425	$\frac{7}{16}$	1	6	$2\frac{1}{4}$[g]
×18.4	5.40	8.00	8	0.271	$\frac{1}{4}$	$\frac{1}{8}$	4.00	4	0.425	$\frac{7}{16}$	1	6	$2\frac{1}{4}$[g]
S6×17.2	5.06	6.00	6	0.465	$\frac{7}{16}$	$\frac{1}{4}$	3.57	$3\frac{5}{8}$	0.359	$\frac{3}{8}$	$\frac{13}{16}$	$4\frac{3}{8}$	—
×12.5	3.66	6.00	6	0.232	$\frac{1}{4}$	$\frac{1}{8}$	3.33	$3\frac{3}{8}$	0.359	$\frac{3}{8}$	$\frac{13}{16}$	$4\frac{3}{8}$	—
S5×10	2.93	5.00	5	0.214	$\frac{3}{16}$	$\frac{1}{8}$	3.00	3	0.326	$\frac{5}{16}$	$\frac{3}{4}$	$3\frac{1}{2}$	—
S4×9.5	2.79	4.00	4	0.326	$\frac{5}{16}$	$\frac{3}{16}$	2.80	$2\frac{3}{4}$	0.293	$\frac{5}{16}$	$\frac{3}{4}$	$2\frac{1}{2}$	—
×7.7	2.26	4.00	4	0.193	$\frac{3}{16}$	$\frac{1}{8}$	2.66	$2\frac{5}{8}$	0.293	$\frac{5}{16}$	$\frac{3}{4}$	$2\frac{1}{2}$	—
S3×7.5	2.20	3.00	3	0.349	$\frac{3}{8}$	$\frac{3}{16}$	2.51	$2\frac{1}{2}$	0.260	$\frac{1}{4}$	$\frac{5}{8}$	$1\frac{3}{4}$	—
×5.7	1.66	3.00	3	0.170	$\frac{3}{16}$	$\frac{1}{8}$	2.33	$2\frac{3}{8}$	0.260	$\frac{1}{4}$	$\frac{5}{8}$	$1\frac{3}{4}$	—

[g] The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.
— Flange is too narrow to establish a workable gage.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

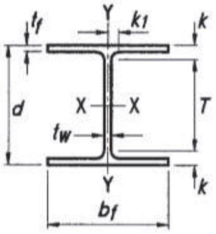


Table 1–4
HP Shapes
Dimensions

Shape	Area, A	Depth, d		Web: Thickness, t_w		Web: $\frac{t_w}{2}$	Flange: Width, b_f		Flange: Thickness, t_f		Distance: k	Distance: k_1	Distance: T	Distance: Workable Gage
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.
HP14×117^f	34.4	14.2	14 1/4	0.805	13/16	7/16	14.9	14 7/8	0.805	13/16	1 1/2	1 1/16	11 1/4	5 1/2
×102^f	30.0	14.0	14	0.705	11/16	3/8	14.8	14 3/4	0.705	11/16	1 3/8	1		
×89^f	26.1	13.8	13 7/8	0.615	5/8	5/16	14.7	14 3/4	0.615	5/8	1 5/16	15/16		
×73^{c,f}	21.4	13.6	13 5/8	0.505	1/2	1/4	14.6	14 5/8	0.505	1/2	1 3/16	7/8		
HP12×84	24.6	12.3	12 1/4	0.685	11/16	3/8	12.3	12 1/4	0.685	11/16	1 3/8	1	9 1/2	5 1/2
×74^f	21.8	12.1	12 1/8	0.605	5/8	5/16	12.2	12 1/4	0.610	5/8	1 5/16	15/16		
×63^f	18.4	11.9	12	0.515	1/2	1/4	12.1	12 1/8	0.515	1/2	1 1/4	7/8		
×53^f	15.5	11.8	11 3/4	0.435	7/16	1/4	12.0	12	0.435	7/16	1 1/8	7/8		
HP10×57^f	16.8	9.99	10	0.565	9/16	5/16	10.2	10 1/4	0.565	9/16	1 1/4	15/16	7 1/2	5 1/2
×42^f	12.4	9.70	9 3/4	0.415	7/16	1/4	10.1	10 1/8	0.420	7/16	1 1/8	13/16	7 1/2	5 1/2
HP8×36^f	10.6	8.02	8	0.445	7/16	1/4	8.16	8 1/8	0.445	7/16	1 1/8	7/8	5 3/4	5 1/2

[c] Shape is slender for compression with $F_y = 50$ ksi.
[f] Shape exceeds compact limit for flexure with $F_y = 50$ ksi.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

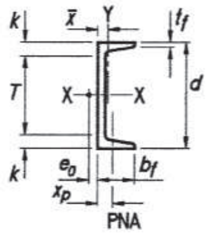


Table 1–5
C Shapes
Dimensions

Shape	Area, A	Depth, d		Web			Flange				Distance			r_{ts}	h_o
				Thickness, t_w		$\frac{t_w}{2}$	Width, b_f		Thickness, t_f		k	T	Workable Gage		
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.	in.
C15×50	14.7	15.0	15	0.716	$\frac{11}{16}$	$\frac{3}{8}$	3.72	$3\frac{3}{4}$	0.650	$\frac{5}{8}$	$1\frac{7}{16}$	$12\frac{1}{8}$	$2\frac{1}{4}$	1.17	14.4
×40	11.8	15.0	15	0.520	$\frac{1}{2}$	$\frac{1}{4}$	3.52	$3\frac{1}{2}$	0.650	$\frac{5}{8}$	$1\frac{7}{16}$	$12\frac{1}{8}$	2	1.15	14.4
×33.9	10.0	15.0	15	0.400	$\frac{3}{8}$	$\frac{3}{16}$	3.40	$3\frac{3}{8}$	0.650	$\frac{5}{8}$	$1\frac{7}{16}$	$12\frac{1}{8}$	2	1.13	14.4
C12×30	8.81	12.0	12	0.510	$\frac{1}{2}$	$\frac{1}{4}$	3.17	$3\frac{1}{8}$	0.501	$\frac{1}{2}$	$1\frac{1}{8}$	$9\frac{3}{4}$	$1\frac{3}{4}$[g]	1.01	11.5
×25	7.34	12.0	12	0.387	$\frac{3}{8}$	$\frac{3}{16}$	3.05	3	0.501	$\frac{1}{2}$	$1\frac{1}{8}$	$9\frac{3}{4}$	$1\frac{3}{4}$[g]	1.00	11.5
×20.7	6.08	12.0	12	0.282	$\frac{5}{16}$	$\frac{3}{16}$	2.94	3	0.501	$\frac{1}{2}$	$1\frac{1}{8}$	$9\frac{3}{4}$	$1\frac{3}{4}$[g]	0.983	11.5
C10×30	8.81	10.0	10	0.673	$\frac{11}{16}$	$\frac{3}{8}$	3.03	3	0.436	$\frac{7}{16}$	1	8	$1\frac{3}{4}$[g]	0.925	9.56
×25	7.34	10.0	10	0.526	$\frac{1}{2}$	$\frac{1}{4}$	2.89	$2\frac{7}{8}$	0.436	$\frac{7}{16}$	1	8	$1\frac{3}{4}$[g]	0.911	9.56
×20	5.87	10.0	10	0.379	$\frac{3}{8}$	$\frac{3}{16}$	2.74	$2\frac{3}{4}$	0.436	$\frac{7}{16}$	1	8	$1\frac{1}{2}$[g]	0.894	9.56
×15.3	4.48	10.0	10	0.240	$\frac{1}{4}$	$\frac{1}{8}$	2.60	$2\frac{5}{8}$	0.436	$\frac{7}{16}$	1	8	$1\frac{1}{2}$[g]	0.869	9.56
C9×20	5.87	9.00	9	0.448	$\frac{7}{16}$	$\frac{1}{4}$	2.65	$2\frac{5}{8}$	0.413	$\frac{7}{16}$	1	7	$1\frac{1}{2}$[g]	0.848	8.59
×15	4.41	9.00	9	0.285	$\frac{5}{16}$	$\frac{3}{16}$	2.49	$2\frac{1}{2}$	0.413	$\frac{7}{16}$	1	7	$1\frac{3}{8}$[g]	0.824	8.59
×13.4	3.94	9.00	9	0.233	$\frac{1}{4}$	$\frac{1}{8}$	2.43	$2\frac{3}{8}$	0.413	$\frac{7}{16}$	1	7	$1\frac{3}{8}$[g]	0.813	8.59
C8×18.7	5.51	8.00	8	0.487	$\frac{1}{2}$	$\frac{1}{4}$	2.53	$2\frac{1}{2}$	0.390	$\frac{3}{8}$	$\frac{15}{16}$	$6\frac{1}{8}$	$1\frac{1}{2}$[g]	0.800	7.61
×13.7	4.04	8.00	8	0.303	$\frac{5}{16}$	$\frac{3}{16}$	2.34	$2\frac{3}{8}$	0.390	$\frac{3}{8}$	$\frac{15}{16}$	$6\frac{1}{8}$	$1\frac{3}{8}$[g]	0.774	7.61
×11.5	3.37	8.00	8	0.220	$\frac{1}{4}$	$\frac{1}{8}$	2.26	$2\frac{1}{4}$	0.390	$\frac{3}{8}$	$\frac{15}{16}$	$6\frac{1}{8}$	$1\frac{3}{8}$[g]	0.756	7.61
C7×14.7	4.33	7.00	7	0.419	$\frac{7}{16}$	$\frac{1}{4}$	2.30	$2\frac{1}{4}$	0.366	$\frac{3}{8}$	$\frac{7}{8}$	$5\frac{1}{4}$	$1\frac{1}{4}$[g]	0.738	6.63
×12.2	3.60	7.00	7	0.314	$\frac{5}{16}$	$\frac{3}{16}$	2.19	$2\frac{1}{4}$	0.366	$\frac{3}{8}$	$\frac{7}{8}$	$5\frac{1}{4}$	$1\frac{1}{4}$[g]	0.721	6.63
×9.8	2.87	7.00	7	0.210	$\frac{3}{16}$	$\frac{1}{8}$	2.09	$2\frac{1}{8}$	0.366	$\frac{3}{8}$	$\frac{7}{8}$	$5\frac{1}{4}$	$1\frac{1}{4}$[g]	0.698	6.63
C6×13	3.81	6.00	6	0.437	$\frac{7}{16}$	$\frac{1}{4}$	2.16	$2\frac{1}{8}$	0.343	$\frac{5}{16}$	$\frac{13}{16}$	4 /8	$1\frac{3}{8}$[g]	0.689	5.66
×10.5	3.08	6.00	6	0.314	$\frac{5}{16}$	$\frac{3}{16}$	2.03	2	0.343	$\frac{5}{16}$	$\frac{13}{16}$	$4\frac{3}{8}$	$1\frac{1}{8}$[g]	0.669	5.66
×8.2	2.39	6.00	6	0.200	$\frac{3}{16}$	$\frac{1}{8}$	1.92	$1\frac{7}{8}$	0.343	$\frac{5}{16}$	$\frac{13}{16}$	$4\frac{3}{8}$	$1\frac{1}{8}$[g]	0.643	5.66
C5×9	2.64	5.00	5	0.325	$\frac{5}{16}$	$\frac{3}{16}$	1.89	$1\frac{7}{8}$	0.320	$\frac{5}{16}$	$\frac{3}{4}$	$3\frac{1}{2}$	$1\frac{1}{8}$[g]	0.617	4.68
×6.7	1.97	5.00	5	0.190	$\frac{3}{16}$	$\frac{1}{8}$	1.75	$1\frac{3}{4}$	0.320	$\frac{5}{16}$	$\frac{3}{4}$	$3\frac{1}{2}$	—	0.584	4.68
C4×7.2	2.13	4.00	4	0.321	$\frac{5}{16}$	$\frac{3}{16}$	1.72	$1\frac{3}{4}$	0.296	$\frac{5}{16}$	$\frac{3}{4}$	$2\frac{1}{2}$	1[g]	0.563	3.70
×5.4	1.58	4.00	4	0.184	$\frac{3}{16}$	$\frac{1}{8}$	1.58	$1\frac{5}{8}$	0.296	$\frac{5}{16}$	$\frac{3}{4}$	$2\frac{1}{2}$	—	0.528	3.70
×4.5	1.38	4.00	4	0.125	$\frac{1}{8}$	$\frac{1}{16}$	1.58	$1\frac{5}{8}$	0.296	$\frac{5}{16}$	$\frac{3}{4}$	$2\frac{1}{2}$	—	0.524	3.70
C3×6	1.76	3.00	3	0.356	$\frac{3}{8}$	$\frac{3}{16}$	1.60	$1\frac{5}{8}$	0.273	$\frac{1}{4}$	$\frac{11}{16}$	$1\frac{5}{8}$	—	0.519	2.73
×5	1.47	3.00	3	0.258	$\frac{1}{4}$	$\frac{1}{8}$	1.50	$1\frac{1}{2}$	0.273	$\frac{1}{4}$	$\frac{11}{16}$	$1\frac{5}{8}$	—	0.495	2.73
×4.1	1.20	3.00	3	0.170	$\frac{3}{16}$	$\frac{1}{8}$	1.41	$1\frac{3}{8}$	0.273	$\frac{1}{4}$	$\frac{11}{16}$	$1\frac{5}{8}$	—	0.469	2.73
×3.5	1.09	3.00	3	0.132	$\frac{1}{8}$	$\frac{1}{16}$	1.37	$1\frac{3}{8}$	0.273	$\frac{1}{4}$	$\frac{11}{16}$	$1\frac{5}{8}$	—	0.455	2.73

[g] The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.
— Flange is too narrow to establish a workable gage.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

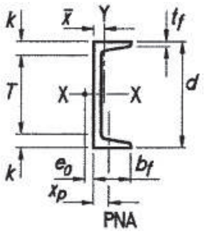


Table 1–6
MC Shapes
Dimensions

Shape	Area, A	Depth, d		Web: Thickness, t_w		Web: $\frac{t_w}{2}$	Flange: Width, b_f		Flange: Average Thickness, t_f		Distance: k	Distance: T	Distance: Workable Gage	r_{ts}	h_o
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.	in.
MC18×58	17.1	18.0	18	0.700	11/16	3/8	4.20	4 1/4	0.625	5/8	1 7/16	15 1/8	2 1/2	1.35	17.4
×51.9	15.3	18.0	18	0.600	5/8	5/16	4.10	4 1/8	0.625	5/8	1 7/16	↓	↓	1.35	17.4
×45.8	13.5	18.0	18	0.500	1/2	1/4	4.00	4	0.625	5/8	1 7/16	↓	↓	1.34	17.4
×42.7	12.6	18.0	18	0.450	7/16	1/4	3.95	4	0.625	5/8	1 7/16	↓	↓	1.34	17.4
MC13×50	14.7	13.0	13	0.787	13/16	7/16	4.41	4 3/8	0.610	5/8	1 7/16	10 1/8	2 1/2	1.41	12.4
×40	11.8	13.0	13	0.560	9/16	5/16	4.19	4 1/8	0.610	5/8	1 7/16	↓	↓	1.38	12.4
×35	10.3	13.0	13	0.447	7/16	1/4	4.07	4 1/8	0.610	5/8	1 7/16	↓	↓	1.35	12.4
×31.8	9.35	13.0	13	0.375	3/8	3/16	4.00	4	0.610	5/8	1 7/16	↓	↓	1.34	12.4
MC12×50	14.7	12.0	12	0.835	13/16	7/16	4.14	4 1/8	0.700	11/16	1 5/16	9 3/8	2 1/2	1.37	11.3
×45	13.2	12.0	12	0.710	11/16	3/8	4.01	4	0.700	11/16	1 5/16	↓	↓	1.35	11.3
×40	11.8	12.0	12	0.590	9/16	5/16	3.89	3 7/8	0.700	11/16	1 5/16	↓	↓	1.33	11.3
×35	10.3	12.0	12	0.465	7/16	1/4	3.77	3 3/4	0.700	11/16	1 5/16	↓	↓	1.30	11.3
×31	9.12	12.0	12	0.370	3/8	3/16	3.67	3 5/8	0.700	11/16	1 5/16	↓	2 1/4	1.28	11.3
MC12×10.6[c]	3.10	12.0	12	0.190	3/16	1/8	1.50	1 1/2	0.309	5/16	3/4	10 1/2	—	0.477	11.7
MC10×41.1	12.1	10.0	10	0.796	13/16	7/16	4.32	4 3/8	0.575	9/16	1 5/16	7 3/8	2 1/2[g]	1.44	9.43
×33.6	9.87	10.0	10	0.575	9/16	5/16	4.10	4 1/8	0.575	9/16	1 5/16	7 3/8	2 1/2[g]	1.40	9.43
×28.5	8.37	10.0	10	0.425	7/16	1/4	3.95	4	0.575	9/16	1 5/16	7 3/8	2 1/2[g]	1.36	9.43
MC10×25	7.35	10.0	10	0.380	3/8	3/16	3.41	3 3/8	0.575	9/16	1 5/16	7 3/8	2[g]	1.17	9.43
×22	6.45	10.0	10	0.290	5/16	3/16	3.32	3 3/8	0.575	9/16	1 5/16	7 3/8	2[g]	1.14	9.43
MC10×8.4[c]	2.46	10.0	10	0.170	3/16	1/8	1.50	1 1/2	0.280	1/4	3/4	8 1/2	—	0.486	9.72
×6.5[c]	1.95	10.0	10	0.152	1/8	1/16	1.17	1 1/8	0.202	3/16	9/16	8 7/8	—	0.364	9.80
MC9×25.4	7.47	9.00	9	0.450	7/16	1/4	3.50	3 1/2	0.550	9/16	1 1/4	6 1/2	2[g]	1.20	8.45
×23.9	7.02	9.00	9	0.400	3/8	3/16	3.45	3 1/2	0.550	9/16	1 1/4	6 1/2	2[g]	1.18	8.45
MC8×22.8	6.70	8.00	8	0.427	7/16	1/4	3.50	3 1/2	0.525	1/2	1 3/16	5 5/8	2[g]	1.20	7.48
×21.4	6.28	8.00	8	0.375	3/8	3/16	3.45	3 1/2	0.525	1/2	1 3/16	5 5/8	2[g]	1.18	7.48
MC8×20	5.88	8.00	8	0.400	3/8	3/16	3.03	3	0.500	1/2	1 1/8	5 3/4	2[g]	1.03	7.50
×18.7	5.50	8.00	8	0.353	3/8	3/16	2.98	3	0.500	1/2	1 1/8	5 3/4	2[g]	1.02	7.50
MC8×8.5	2.50	8.00	8	0.179	3/16	1/8	1.87	1 7/8	0.311	5/16	13/16	6 3/8	1 1/8[g]	0.624	7.69

[c] Shape is slender for compression with F_y = 36 ksi.
[g] The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.
— Flange is too narrow to establish a workable gage.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

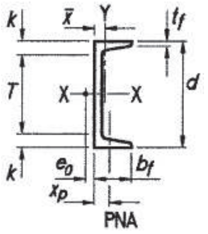


Table 1–6 (continued)
MC Shapes
Dimensions

Shape	Area, A	Depth, d		Web			Flange				Distance			r_{ts}	h_o
				Thickness, t_w		$\frac{t_w}{2}$	Width, b_f		Average Thickness, t_f		k	T	Work-able Gage		
	in.^{2}	in.		in.		in.	in.		in.		in.	in.	in.	in.	in.
MC7×22.7	6.67	7.00	7	0.503	$\frac{1}{2}$	$\frac{1}{4}$	3.60	$3\frac{5}{8}$	0.500	$\frac{1}{2}$	$1\frac{1}{8}$	$4\frac{3}{4}$	2[g]	1.23	6.50
×19.1	5.61	7.00	7	0.352	$\frac{3}{8}$	$\frac{3}{16}$	3.45	$3\frac{1}{2}$	0.500	$\frac{1}{2}$	$1\frac{1}{8}$	$4\frac{3}{4}$	2[g]	1.18	6.50
MC6×18	5.29	6.00	6	0.379	$\frac{3}{8}$	$\frac{3}{16}$	3.50	$3\frac{1}{2}$	0.475	$\frac{1}{2}$	$1\frac{1}{16}$	$3\frac{7}{8}$	2[g]	1.20	5.53
×15.3	4.49	6.00	6	0.340	$\frac{5}{16}$	$\frac{3}{16}$	3.50	$3\frac{1}{2}$	0.385	$\frac{3}{8}$	$\frac{7}{8}$	$4\frac{1}{4}$	2[g]	1.20	5.62
MC6×16.3	4.79	6.00	6	0.375	$\frac{3}{8}$	$\frac{3}{16}$	3.00	3	0.475	$\frac{1}{2}$	$\frac{11}{16}$	$3\frac{7}{8}$	$1\frac{3}{4}$[g]	1.03	5.53
×15.1	4.44	6.00	6	0.316	$\frac{5}{16}$	$\frac{3}{16}$	2.94	3	0.475	$\frac{1}{2}$	$\frac{11}{16}$	$3\frac{7}{8}$	$1\frac{3}{4}$[g]	1.01	5.53
MC6×12	3.53	6.00	6	0.310	$\frac{5}{16}$	$\frac{3}{16}$	2.50	$2\frac{1}{2}$	0.375	$\frac{3}{8}$	$\frac{7}{8}$	$4\frac{1}{4}$	$1\frac{1}{2}$[g]	0.856	5.63
MC6×7	2.09	6.00	6	0.179	$\frac{3}{16}$	$\frac{1}{8}$	1.88	$1\frac{7}{8}$	0.291	$\frac{5}{16}$	$\frac{3}{4}$	$4\frac{1}{2}$	—	0.638	5.71
×6.5	1.95	6.00	6	0.155	$\frac{1}{8}$	$\frac{1}{16}$	1.85	$1\frac{7}{8}$	0.291	$\frac{5}{16}$	$\frac{3}{4}$	$4\frac{1}{2}$	—	0.630	5.71
MC4×13.8	4.03	4.00	4	0.500	$\frac{1}{2}$	$\frac{1}{4}$	2.50	$2\frac{1}{2}$	0.500	$\frac{1}{2}$	1	2	—	0.852	3.50
MC3×7.1	2.11	3.00	3	0.312	$\frac{5}{16}$	$\frac{3}{16}$	1.94	2	0.351	$\frac{3}{8}$	$\frac{13}{16}$	$1\frac{3}{8}$	—	0.657	2.65

[g] The actual size, combination, and orientation of fastener components should be compared with the geometry of the cross-section to ensure compatibility.
— Flange is too narrow to establish a workable gage.

Source: American Institute of Steel Construction

Structural Metal Shape Designations. (CONTINUED)

GLOSSARY OF KEY TERMS

active animation: An animation in which the observer (camera) as well as objects in the scene actively move around and through the scene.

additive color model: The RGB color system in which the primary colors of red, green, and blue are added together to create white.

adjacent views: Orthogonal views created immediately next to each other, aligned side by side to share a common dimension, and presented on a single plane.

agenda: The list of topics for discussion/action at a team meeting.

algebraic constraints: Constraint that define the value of a selected variable as the result of an algebraic expression containing other variables from the solid model.

allowance: The difference between the maximum material limits of mating parts. It is the minimum clearance or maximum interference between parts.

alpha channel: An optional layer of image data containing an additional 8 bits of grayscale data that can be used to control transparency affecting the entire image.

ambient light: Indirect light in a scene that does not come directly from a light source, but arrives at a surface by bouncing around or reflecting off other surfaces in the scene.

analysis: The study of the behavior of a physical system under certain imposed conditions.

anchor edge: The same edge that can be easily and confidently located on multiple views and on a pictorial for an object.

anchor point: The same point, usually a vertex, which can be located easily and confidently on multiple views for an object.

anchor surface: The same surface that can be located easily and confidently on multiple views for an object.

angle of thread: The angle between the side of a thread and a line perpendicular to the axis of the thread.

ANSI Y14.5 (*ASME Y14.5M-1994*): Industry standard document that outlines uniform practices for displaying and interpreting dimensions and related information on drawings and other forms of engineering documentation.

approval signatures: The dated signatures or initials of the people responsible for certain aspects of a formal drawing, such as the people who did the drafting or the engineer responsible for the function of the part.

arc: A curved entity that represents a portion of a circle.

architects: Professionals who complete conceptual designs for civil engineering projects.

Architect's scale: A device used to measure or draw lines in the English system of units with a base unit of inches and fractions of an inch.

arrowhead: A small triangle at the end of dimension lines and leaders to indicate the direction and extent of a dimension.

as-built drawings: The marked-up drawings from a civil engineering project that show any modifications implemented in the field during construction.

as-built plans: Drawings that show exactly how buildings were constructed, especially when variations exist between the final building and the plans created during the design phase.

aspect: A quantitative measure of the direction of a slope face.

assembly: A collection of parts and/or subassemblies that have been put together to make a device or structure that performs a specific function.

assembly constraints: Used to establish relationships between instances in the development of a flexible assembly model.

assembly dimensions: Dimensions that show where parts must be placed relative to other parts when the device is being put together.

associative constraints: *See* algebraic constraints.

associativity: The situation whereby parts can be modified and the components referenced to the parts will be modified accordingly.

attribute: Spatial information that describes the characteristics of spatial features.

auxiliary views: Views on any projection plane other than a primary or principal projection plane.

axis: The longitudinal centerline that passes through a screw.

axonometric drawing: A drawing in which all three-dimensional axes on an object can be seen, with the scaling factor constant in each direction. Usually, one axis is shown as being vertical.

back light: A scene light, usually located behind objects in the scene, which is used to create a defining edge that visually separates foreground objects from the background.

balloons: Closed geometric shapes, usually circles, containing identification numbers and placed beside parts on a layout or assembly drawing to help identify those parts.

bar chart: A chart using bars of varying heights and widths to represent quantitative data.

base feature: The first feature created for a part, usually a protrusion.

base instance: The one fixed instance within an assembly.

baseline dimensioning: A method for specifying the location of features on a part whereby all the locations are relative to a common feature or edge.

basic dimension: A dimension that is theoretically exact. It is identified by a box around the dimension. It locates the perfect position of features from clearly identified datums.

bearing: The angle that a line makes with a North-South line as seen in a plan view.

benchmarks: Points established by the U.S. Geological Survey that can be used to accurately locate control points on a construction site.

bill of materials (BOM): A drawing or table in a drawing that lists all of the parts needed to build a device by (at least) the part number, part name, type of material used, and number of times the part is used in the device.

bitmap textures: Texture mapping routines that are based on referencing external image files.

black box diagram: A diagram that shows the major inputs and outputs from a system.

blend: A solid formed by a smooth transition between two or more profiles.

blind extrusion: An extrusion made to a specified length in a selected direction.

blind hole: A hole that does not pass completely through a part.

blueprints: The name sometimes given to construction drawings based on historical blue-on-white drawings that were produced from ink drawings.

bolt: A threaded fastener that passes completely through parts and holds them together using a nut.

Boolean operations: In early versions of 3-D CAD software, commands used to combine solids.

border: A thick line that defines the perimeter of a drawing.

boring: The general process of making a hole in a part by plunging a rotating tool bit into a part, moving a rotating part into a stationary tool bit, or moving a part into a rotating tool bit.

bottom-up modeling: The process of creating individual parts and then creating an assembly from them.

boundary conditions: The constraints and loads added to the boundaries of a finite element model.

boundary representation (b-rep): A method used to build solid models from their bounding surfaces.

bounding box: A square box used to sketch circles or ellipses.

brainstorming: The process of group creative thinking used to generate as many ideas as possible for consideration.

brainwriting: A process of group creative thinking where sketching is the primary mode of communication between team members.

brazing: A method for joining separate metal parts by heating the parts, flowing a molten metal (solder) between them, and allowing the unit to cool and harden.

brief: A small graphic using word content alone.

broach: A long, shaped cutting tool that moves along the length of a part when placed against it. It is used to create uniquely shaped holes and slots.

broaching: A process of creating uniquely shaped holes and slots using a long, shaped cutting tool that moves along the length of a part in a single stroke when placed against the part.

broken-out section: The section view produced when the cutting plane is partially imbedded into the object, requiring an irregular portion of the object to be removed before the hypothetically cut surface can be seen.

buffer: Measured in units of distance or time; a zone around a map feature. A buffer is useful for proximity analysis.

bump mapping: A technique used to create the illusion of rough or bumpy surface detail through surface normal perturbation.

business diagram: A diagram used in an organization to show organizational hierarchy, task planning or analysis, or relationships between different groups or sets of information.

butt joint: A joint between two parts wherein the parts are butted, or placed next to each other.

cabinet oblique drawing: An oblique drawing where one half the true length of the depth dimension is measured along the receding axes.

CAD: Computer-aided drawing. The use of computer hardware and software for the purpose of creating, modifying, and storing engineering drawings in an electronic format.

CAD designers: Designers who create 3-D computer models for analysis and detailing.

caliper: A handheld device used to measure objects with a fair degree of accuracy.

cap screw: A small threaded fastener that mates with a threaded hole.

casting: A method of creating a part by pouring or injecting molten material into the cavity of a mold, allowing it to harden, and then removing it from the mold.

cavalier oblique drawing: An oblique drawing where the true length of the depth dimension is measured along the receding axes.

center-of-mass (centroid): The origin of the coordinate axes for which the first moments are zero.

centerline: A series of alternating long and short dashed lines used to identify an axis of rotational symmetry.

centermark: A small right-angle cross that is used to identify the end view of an axis of rotational symmetry.

Central Meridian: The line of longitude that defines the center and is often the x-origin of a projected coordinate system.

Central Parallel: The line of latitude that divides a map into north and south halves and is often the y-origin of a projected coordinate system.

chain dimensioning: A method for specifying the location of features on a part whereby the location of each feature is successively specified relative to the location of the previous feature.

chamfers: Angled cut transitions between two intersecting surfaces.

charts: Charts, graphs, tables, and diagrams of ideas and quantitative data.

chief designer: The individual who oversees other members of the design team and manages the overall project.

child feature: A feature that is dependent upon the existence of a previously created feature.

circle: A closed curved figure where all points on it are equidistant from its center point.

Clarke Ellipsoid of 1866: A reference ellipsoid having a semimajor axis of approximately 6,378,206.4 meters and a flattening of 1/294.9786982. It is the basis for the North American Datum of 1927 (NAD27) and other datums.

clearance: A type of fit where space exists between two mating parts.

clearances: The minimum distances between two instances in an assembly.

clip: A geoprocessing command that extracts the features from a coverage that reside entirely within a boundary defined by features in another coverage.

clipping plane: A 3-D virtual camera technique that allows you to selectively exclude, and not view or render, unnecessary objects in a scene that are either too close or too far away.

CODEC: Video compression-decompression algorithm.

collision detection: A built-in software capability for calculating and graphically animating the results of collisions between multiple objects based on object properties of speed, mass, and gravity.

color mapping: Sometimes called diffuse mapping, color mapping replaces the main surface color of a model with an external image map or texture.

combining solids: The process of cutting, joining, or intersecting two objects to form a third object.

components: References of object geometry used in assembly models.

compositing: The technique and art of rendering in layers or passes, editing the image on each layer as needed, and compiling the edited layers or images into a single unified final image.

computer-aided design (CAD): The process by which computers are used to model and analyze designed products.

computer-aided manufacturing (CAM): The process by which parts are manufactured directly from 3-D computer models.

concept mapping: The creative process by which the central idea is placed in the middle of a page and related concepts radiate out from that central idea.

conceptual design: The initial idea for a design before analysis has been performed.

concurrent engineering: The process by which designers, analysts, and manufacturers work together from the start to design a product.

consensus: A process of decision making where an option is chosen that everyone supports.

constraint: A boundary condition applied to a finite element model to prevent it from moving through space.

constraints: Geometric relationships, dimensions, or equations that control the size, shape, and/or orientation of entities in a sketch or solid model.

construction drawings: Working drawings, often created by civil engineers, that are used to build large-scale, one-of-a-kind structures.

construction line: A faint line used in sketching to align items and define shapes.

constructive solid geometry (CSG): A method used to build solid models from primitive shapes based on Boolean set theory.

continuation blocks: Header blocks used on the second and subsequent pages of multipage drawings.

contour dimensioning: Placing each dimension in the view where the contour or shape of the feature shows up best.

contour interval: The vertical distance between contours.

contour rule: A drawing practice where each dimension should be placed in the view where the contour shape is best shown.

contours: Lines or curves that represent the same elevation across the landscape.

control points: Points at a construction site that are referenced to an origin by north, south, east, or west coordinates.

coordinate measuring machine: A computer-based tool used to digitize object geometry for direct input to a 3-D CAD system.

corner views: An isometric view of an object created from the perspective at a given corner of the object.

cosmetic features: Features that modify the appearance of the surface but do not alter the size or shape of the object.

cover sheet: The first page in a set of construction drawings showing a map of the location of the project and possibly an index.

crest: The top surface or point joining the sides of a thread.

critical path: The sequence of activities in a project that have the longest duration.

critical path method (CPM): A tool for determining the least amount of time in which a project can be completed.

cross section: The intersection between a cutting plane and a 3-D object.

curved surface: Any nonflat surface on an object.

cut (noun): A feature created by the removal of solid volume from a model.

cut (verb): To remove the volume of interference between two objects from one of the objects.

cutaway diagram: A diagram that allows the reader to see a slice of an object.

cutting plane: A theoretical plane used to hypothetically cut and remove a portion of an object to reveal its interior details.

cutting plane line: On an orthographic view of an object, the presentation of the edge view of a cutting plane used to hypothetically cut and remove a portion of that object for viewing.

cutting segment: On a stepped cutting plane for an offset section view, that portion of the plane that hypothetically cuts and reveals the interior detail of a feature of interest.

data structure: The organization of data within a specific computer system that allows the information to be stored and manipulated effectively.

database: A collection of information for a computer and a method for interpretation of the information from which the original model can be re-created.

datum: A theoretical plane or axis established by real features on an object for the purpose of defining the datum reference frame.

datum geometries: Geometric entities such as points, axes, and planes that do not actually exist on real parts, but are used to help locate and define other features.

datum planes: The planes used to define the locations of features and entities in the construction of a solid model.

datum reference frame: A system of three mutually perpendicular planes used as the coordinate system for geometric dimensioning.

decimal degrees (DD): A measuring system in which values of latitude and longitude are expressed in decimal format rather than in degrees, minutes, and seconds, such as 87.5 degrees.

deep drawing: Creating a thin-shelled part by pressing sheet metal into a deep cavity mold.

default tolerances: Usually appearing in the drawing header, the tolerances to be assumed for any dimension show on a part when that dimension does not specify any tolerances.

degrees, minutes, seconds (DMS): A measuring system for longitude and latitude values, such as 87° 30′ 00″, in which 60 seconds equals 1 minute and 60 minutes equals 1 degree.

density: The mass per unit volume for a given material.

depiction: An illustration describing and simplifying factual information on a real-world system.

depth of thread: The distance between the crest and the root of a thread, measured normal to the axis.

depth-mapped shadows: Also called shadow-mapped shadows, depth-mapped shadows use a precalculated depth map to determine the location, density, and edge sharpness of shadows.

descriptive geometry: A two-dimensional graphical construction technique used for geometric analysis of three-dimensional objects.

design (noun): An original manifestation of a device or method created for performing one or more useful functions.

design (verb): The process of creating a design (noun).

design analysts: Individuals who analyze design concepts by computer methods to determine their structural, thermal, or vibration characteristics.

design documentation: The set of drawings and specifications that illustrate and thoroughly describe a designed product.

design process: The multistep, iterative process by which products are conceived and produced.

design table: A table or spreadsheet that lists all of the versions of a family model, the dimensions or features that may change, and the values in any of its versions.

design tree: *See* model tree.

detail designers: The individuals who create engineering drawings, complete with annotation, from 3-D computer models or from engineering sketches.

detail drawing: A formal drawing that shows the geometry, dimensions, tolerances, materials, and any processes needed to fabricate a part.

detail sections: Drawings included in a set of construction plans that show how the various components are assembled.

devil's advocate: The team member who challenges ideas to ensure that all options are considered by the group.

diagram: An illustration that explains information, represents a process, or shows how pieces are put together.

die: A special tool made specifically to deform raw or stock material into a desired outline of a part or feature in a single operation.

die casting: A method of casting where the mold is formed by cutting a cavity into steel or another hard material. *See* casting.

digital elevation model (DEM): The representation of continuous elevation values over a topographic surface by a regular array of *z*-values referenced to a common datum. It is typically used to represent terrain relief.

dimension: A numerical value expressed in appropriate units of measure and used to define the size, location, geometric characteristic, or surface texture of a part or part feature.

dimension line: A thin, dark, solid line that terminates at each end with arrowheads. The value of a dimension typically is shown in the center of the dimension line.

dimension name: The unique alphanumeric designation of a variable dimension.

dimensional constraints: Measurements used to control the size or position of entities in a sketch.

dimetric drawing: An axonometric drawing in which the scaling factor is the same for two of the axes.

direct dimensioning: Dimensioning between two key points to minimize tolerance accumulation.

directional light: A computer-generated light source designed to simulate the effect of light sources, such as the sun, that are so far away from objects in the scene that lighting and shadow patterns in the scene appear to be parallel.

displacement: A change in the location of points on an object after it has been subjected to external loads.

dissolve: A geoprocessing command that removes boundaries between adjacent polygons that have the same value for a specified attribute.

draft: The slight angling of the walls of a cast, forged, drawn, or stamped part to enable the part to be removed from the mold more easily.

drawing: A collection of images and other detailed graphical specifications intended to represent physical objects or processes for the purpose of accurately re-creating those objects or processes.

drill bit: A long, rotating cutting tool with a sharpened tip used to make holes.

drilling: A process of making a hole by plunging a rotating tool bit into a part.

drill press: A machine that holds, spins, and plunges a rotating tool bit into a part to make holes.

driven dimension: A variable connected to an algebraic constraint that can be modified only by user changes to the driving dimensions.

driving dimension: A variable used in an algebraic constraint to control the values of another (driven) dimension.

double-sided extrusion: A solid formed by the extrusion of a profile in both directions from its sketching plane.

EC Level: A number included in the title block of a drawing indicating that the part has undergone a revision.

edge tracking: A procedure by which successive edges on an object are simultaneously located on a pictorial image and on a multiview image of that object.

edge view (of a plane): A view in which the given plane appears as a straight line.

electric discharge machining (EDM): A process by which material is eroded from a part by passing an electric current between the part and an electrode (or a wire) through an electrolytic fluid.

electrical plan: A plan view showing the layout of electrical devices on a floor in a building.

elevation view: In the construction of a perspective view, the object as viewed from the front, as if created by orthogonal projection.

elevation views: Views of a structure that show changes in elevation (side or front views).

ellipse: A closed curve figure where the sum of the distance between any point on the figure and its two foci is constant.

end mill: A rotating cutting tool that, when placed in the spindle of a milling machine, can remove material in directions parallel or perpendicular to its rotation axis.

engineer (noun): A person who engages in the art of engineering.

engineer (verb): To plan and build a device that does not occur naturally within the environment.

Engineer's scale: A device used to measure or draw lines in the English system of units with a base unit of inches and tenths of an inch.

engineering: The profession in which knowledge of mathematical and natural sciences gained by study, experience, and practice is applied with judgment to develop and utilize economically the materials and forces of nature for the benefit of humanity.

engineering animation: A dynamic virtual 3-D prototype of a mechanism or system that can be assembled (usually from preexisting or newly created 3-D CAD part models) and/or shown in operation over a period of time.

engineering change (EC) number: A dated number that defines the degree to which the specifications of a part have been updated.

engineering design: The process by which many competing factors of a product are weighed to select the best alternative in terms of cost, sustainability, and function.

engineering scale: A device used to make measurements in much the same way a ruler is used.

explanation diagram: An illustration explaining the way something works; a basic process; or the deconstruction of an object, a plan, or a drawing.

exploded assembly drawing: A formal drawing, usually in pictorial form, that shows the orientation and sequence in which parts are put together to make a device.

exploded configuration: A configuration of an assembly that shows instances separated from one another. An exploded configuration is used as the basis for an assembly drawing.

extension line: A thin, dark, solid line extending from a point on an object, perpendicular to a dimension line used to indicate the extension of a surface or point to a location preferably outside the part outline.

external thread: Threads that are formed on the outside of a cylindrical feature, such as on a bolt or stud.

extrude through all: An extrusion that begins on the sketching plane and protrudes or cuts through all portions of the solid model that it encounters.

extrude to a selected surface: An extrusion where the protrusion or cut begins on the sketching plane and stops when it intersects a selected surface.

extrude to the next surface: An extrusion begins at the profile and the protrusion or cut stops when it intersects the next surface encountered.

extrusion (in fabrication): A process for making long, solid shapes with a constant cross section by squeezing raw material under elevated temperatures and pressure through an orifice shaped with that cross section.

extrusion (in 3-D modeling): A solid that is bounded by the surfaces swept out in space by a planar profile as it is pulled along a path perpendicular to the plane of the profile.

fabricate: To make something from existing materials.

false easting: A value applied to the origin of a coordinate system to modify the x-coordinate readings, usually to make all of the coordinate values positive.

false northing: A value applied to the origin of a coordinate system to modify the y-coordinate readings, usually to make all of the coordinate values positive.

family model: A collection of different versions of a part in a single model that can display any of the versions.

fastener: A manufactured part whose primary function is to join two or more parts.

feature array: A method for making additional features by placing copies of a master feature on the model at a specified equal spacing.

feature control frame: The main alphabet of the language of geometric dimensioning and tolerancing. These boxes contain the geometric characteristic symbol, the geometric tolerances, and the relative datums.

feature generalization: The process of going from the specific to the general in analyzing data.

feature-based solid modeling: A solid modeling system that uses features to build models.

feature pattern: *See* feature array.

feature tree: *See* model tree.

feature with size: A cylindrical or spherical surface or a set of two opposed elements or opposed parallel surfaces associated with a size dimension. Typical features with size are holes, cylinders, spheres, and opposite sides of a rectangular block.

feature without size: A planar surface or a feature where the normal vectors point in the same direction.

features: Distinctive geometric shapes on solid parts; 3-D geometric entities that exist to serve some function.

field: A column in a table that stores the values for a single attribute.

fill light: A light that softens and extends the illumination of the objects provided by the key light.

fillets: Smooth transitions of the internal edge created by two intersecting surfaces and tangent to both intersecting surfaces.

finite element analysis (FEA): An advanced computer-based design analysis technique that involves subdividing an object into several small elements to determine stresses, displacements, pressure fields, thermal distributions, or electromagnetic fields.

first-angle projection: The process of creating a view of an object by imprinting its image, using orthogonal projection, on an opaque surface behind that object.

fishbone diagram: A diagram that shows the various subsystems in a device and the parts that make up each subsystem.

fixture: A mechanical device, such as a clamp or bracket, used for holding a workpiece in place while it is being modified.

flash: Bits of material that are left on a part from a casting or molding operation and found along the seams where the mold pieces separate to allow removal of the part.

floor plan: A plan view of a single floor in a building that shows the layout of the rooms.

flowchart: A quality improvement tool used to document, plan, or analyze a process or series of tasks.

foreshortened (line or plane): Appearing shorter than its actual length or size in one of the primary views.

forging: A process of deforming metal with a common shape at room temperature into a new but similar shape by pressing it into a mold under elevated pressure.

form: The shape of the thread cross section when cut through the axis of the thread cylinder.

form feature: A recognizable area on a solid model that has a specific function.

forward kinematics: In a hierarchical link, total motion in which the motion of the parent is transferred to the motion of the child.

foundation plan: A plan view of the foundation of a building showing footings and other support structures.

foundation space: The rectilinear volume that represents the limits of the volume occupied by an object.

frame rate: The rate of speed, usually in frames per second, in which individual images or frames are played when an animation is viewed.

frontal surface: A surface on an object being viewed that is parallel to the front viewing plane.

full section: The section view produced when a single cutting plane is used to hypothetically cut an object completely into two pieces.

function curve: A graphical method of displaying and controlling an object's transformations.

functional gage: An inspection tool built uniquely for the purpose of quickly checking a specific dimension or geometric condition on a part to determine whether or not it falls within tolerance limits.

fused deposition (FD): A process where parts are gradually built up by bits of molten plastic that are deposited by a heated tip at selected locations and then solidified by cooling.

Gantt chart: A tool for scheduling a project timeline.

general sections: Sections through entire structures that show the layout of rooms but provide little detail.

geographic coordinate system: A spatial reference system using a grid network of parallels and meridians to locate spatial features on the earth's surface.

geographically referenced data: Information that is referenced to a specific geographic location, usually on the earth's surface.

geometric constraints: Definitions used to control the shape of a profile sketch through geometric relationships.

geometric dimensioning and tolerancing (GD&T): A 3-D mathematical system that allows a designer to describe the form, orientation, and location of features on a part within precise tolerance zones.

geometric transformation: Transformations used to alter the position, size, or orientation of a part, camera, or light over a specified period of time.

georeferenced data: *See* geographically referenced data.

glass box: A visualization aid for understanding the locations and orientations of images of an object produced by third-angle projection on a drawing. The images of an object are projected, using orthogonal projection, on the sides of a hypothetical transparent box that is then unfolded into a single plane.

global positioning systems (GPS): A system of geosynchronous, radio-emitting and receiving satellites used for determining positions on the earth.

graphical user interface (GUI): The format of information on the visual display of a computer, giving its user control of the input, output, and editing of the information.

green engineering: The process by which environmental and life cycle considerations are examined from the outset in design.

grinding: A method of removing small amounts of material from a part using a rotation abrasive wheel, thus creating surfaces of very accurate planar or cylindrical geometries.

ground constraint: A constraint usually applied to a new sketch to fix the location of the sketch in space.

ground line (GL): In the construction of a perspective view, a line on the elevation view that represents the height of the ground.

GRS80 spheroid: The satellite-based spheroid for the Geodetic Reference System 1980.

half section: The section view produced when a single cutting plane is used to hypothetically cut an object up to a plane or axis of symmetry, leaving that portion beyond the plane or axis intact.

header: A premade outline on which working drawings are created to ensure that all information required for fabrication and record keeping is entered.

heating and ventilation plan: A plan view of the ventilation systems on a specific floor of a building, including ductwork and devices such as air conditioning units.

hidden lines: The representation, using dashed lines, on a drawing of an object of the edges that cannot be seen because the object is opaque.

hierarchical link: A series of user-defined or linked objects that have a parent-child-grandchild relationship.

hierarchy: The parent-child relationships between instances in an assembly.

history tree: *See* model tree.

holes: A cut feature added to a model that will often receive a fastener for system assembly.

horizon line (HL): In the construction of a perspective view, the line that represents the horizon, which is the separation between the earth and the sky at a long distance. The left and right vanishing points are located on the HL. The PP and the HL are usually parallel to each other.

horizontal modeling: A strategy for creating solid models that reduces parent-child relationships within the feature tree.

horizontal surface: A surface on an object being viewed that is parallel to the top viewing plane.

identity: A geometric integration of spatial datasets that preserves only the geographic features from the first input layer; the second layer merely adds more information to the dataset.

image: A collection of printed, displayed, or imagined patterns intended to represent real objects, data, or processes.

inclined surface: A flat surface on an object being viewed that is perpendicular to one primary view and angled with respect to the other two views; in other words, a plane that appears as an edge view in one primary view but is not parallel to any of the principal views.

index: A list of all sheets of drawings contained in a set of construction plans.

industrial designers: The individuals who use their creative abilities to develop conceptual designs of potential products.

infographic: A shortened form of *informational graphic* or *information graphic*.

information graphics: Often referred to as *infographic*, visual explanations.

injection molding: A process for creating a plastic part by injecting molten plastic into a mold under pressure, allowing the material to solidify, and removing the part from the mold.

instances: Copies of components that are included within an assembly model.

instructional diagram: A diagram showing how a specific action within an object occurs.

instruments: In engineering drawing, mechanical devices used to aid in creating accurate and precise images.

interchangeable manufacturing: A process by which parts are made at different locations and brought together for assembly. For many industries, this process opens the door for third-party companies to produce replacement parts or custom parts.

interference: A fit where two mating parts have intersecting nominal volumes, requiring the deformation of the parts. For example, the diameter of the shaft is larger than the diameter of the hole. When assembled, the intent is that the shaft will not spin in the hole. Also, the amount of overlap between two instances in an assembly.

internal thread: Threads that are formed on the inside of a hole.

international sheet sizes: The internationally accepted paper dimensions used when drawings are created or printed to their full intended size.

intersect: To create a new object that consists of the volume of interference between two objects.

intersection: A geometric integration of spatial datasets that preserves features or portions of features that fall within areas common to all input datasets.

inverse kinematics: A bidirectional set of constraints that allows motion of a set of linked objects by moving the very end of the hierarchically linked chain and having the rest of the links move in response.

investment casting: A method of casting where the mold is formed by successive dipping of a master form into progressively coarser slurries, allowing each layer to harden between each dipping. *See* casting.

isometric axes: A set of three coordinate axes that are portrayed on the paper at 120 degrees relative to one another.

isometric dot paper: Paper used for sketching purposes that includes dots located along lines that meet at 120 degrees.

isometric drawing: An axonometric drawing in which the scaling factor is the same for all three axes.

isometric grid paper: Paper used for sketching purposes that includes grid lines at 120 degrees relative to one another.

isometric lines: Lines on an isometric drawing that are parallel or perpendicular to the front, top, or profile viewing planes.

isometric pictorial: A sketch of an object that shows its three dimensions where isometric axes were used as the basis for defining the edges of the object.

item number: A number used to identify a part on a layout or assembly drawing.

join: To absorb the volume of interference between two objects to form a third object.

key: A small removable part similar to a wedge that provides a positive means of transferring torque between a shaft and a hub.

keyframe: A specific frame located at a specified time within an animation where an object's location, orientation, and scale are defined perfectly.

key light: A light that creates an object's main illumination, defines the dominant angle of the lighting, and is responsible for major highlights on objects in a scene.

keyseat: A rectangular groove cut in a shaft to position a key.

keyway: A rectangular groove cut in a hub to position a key.

landscape: The drawing orientation in which the horizontal size is larger than the vertical size.

lap joint: A joint between two parts wherein the parts are overlapped.

laser scanning (three-dimensional): A process where cameras and lasers are used to digitize an object based on the principle of triangulation.

lathe: A machine used to make axially symmetric parts or features using a material removal process known as turning.

latitude: An imaginary line around the earth's surface in which all of the points on the line are equidistant from the equator.

layout drawing: A formal drawing that shows a device in its assembled state with all of its parts identified.

lead: The distance a screw thread advances axially in one full turn.

leader: A thin, dark, solid line terminating with an arrowhead at one end and a dimension, note, or symbol at the other end.

left-handed system: Any 3-D coordinate system that is defined by the left-hand rule.

level of detail: The number of polygon mesh triangles used to define the surface shape of a 3-D model. For rendering speed, as a general case, objects close to the camera in a scene require a higher number of polygons to more accurately define their surfaces while more distant objects can be effectively rendered with fewer polygons.

life cycle: The amount of time a product will be used before it is no longer effective.

line: A spatial feature that has location and length but no area and is represented by a series of nodes, points, and arcs.

line chart: A graph showing the relationship between two sets of data, where line segments are used to link the data to show trends in their changes.

list: A boxed series of components, definitions, tips, etc.

location: A dimension associated with the position of a feature on a part.

location grid: An imaginary alphanumeric grid, similar to that of a street map, on a drawing that is used to specify area locations on the drawing.

longitude: An imaginary north-south line on the earth's surface that extends from the North Pole to the South Pole.

machine screw: A threaded fastener wherein the threads are cut along the entire length of the cylindrical shaft. Machine screws can mate with a threaded hole or nut.

major diameter: The largest diameter on an internal or external thread.

main assembly: A completed device usually composed of multiple smaller parts and/or subassemblies.

main title block: A bordered area of a drawing (and part of the drawing header) that contains important information about the identification, fabrication, history, and ownership of the item shown on the drawing.

manufacturing drawings: Working drawings, often created by mechanical engineers, that are used to mass-produce products for consumers.

map: A diagram of the location of events with geography.

map projection: A systematic arrangement of parallels and meridians on a plane surface representing the geographic coordinate system.

mapping coordinates: Also called UVW coordinates, mapping coordinates are special coordinate systems designed to correctly place and control the shape of external and procedurally generated images on the surfaces of 3-D models.

mass: A property of an object's ability to resist a change in acceleration.

mass properties analysis: A computer-generated document that gives the mechanical properties of a 3-D solid model.

master feature: A feature or collection of features that is to be copied for placement at other locations in a model.

master model: In a collection of similar parts, the model that includes all of the features that may appear in any of the other parts.

matt object: An object with a combined material and alpha channel map.

maximum material condition (MMC): The condition in which a feature of size contains the maximum amount of material within the stated limits of size.

measuring line (ML): In the construction of a perspective view, a vertical line used in conjunction with the elevation view to locate vertical points on the perspective drawing.

measuring wall: In the construction of a perspective view, a line that extends from the object to the vanishing point to help establish the location of horizontal points on the drawing.

mechanical dissection: The process of taking apart a device to determine the function of each part.

mechanical stress: Developed force applied per unit area that tries to deform an object.

mental rotations: The ability to mentally turn an object in space.

meridian: A line of longitude through the North and South Poles that measure either E or W in a geographic coordinate system.

mesh: The series of elements and nodal points on a finite element model.

Metric scale: A device used to measure or draw lines in the metric system of units with drawings scales reported as ratios.

metrology: The practice of measuring parts.

milling: A process of removing material from a part using a rotating tool bit that can remove material in directions parallel or perpendicular to the tool bit's rotation axis.

milling machine: A machine used to make parts through a material removal process known as milling.

minor diameter: The smallest diameter on an internal or external thread.

mirrored feature: A feature that is created as a mirror image of a master feature.

model: A mathematical representation of an object or a device from which information about its function, appearance, or physical properties can be extracted.

model builders: Engineers who make physical mock-ups of designs using modern rapid prototyping and CAM equipment.

model tree: A list of all of the features of a solid model in the order in which they were created, providing a "history" of the sequence of feature creation.

mold: A supported cavity shaped like a desired part into which molten material is poured or injected.

moment-of-inertia: The measure of an object's ability to resist rotational acceleration about an axis.

morphological chart: A chart used to generate ideas about the desirable qualities of a product and all of the possible options for achieving them.

motion blur: The amount of movement of a high-speed object recorded as it moves through a single frame.

motion path: Spline curves that serve as a trajectory for the motion of objects in animation.

multiple thread: A thread made up of two or more continuous ridges side by side.

multiple views: The presentation of an object using more than one image on the same drawing, each image representing a different orientation of the object.

multiview: Refers to a drawing that contains more than one image of an object and whose adjacent images are generated from orthogonal viewing planes.

node: A point at the beginning and end of a line feature or a point that defines a polygon feature.

normal surface: A surface on an object being viewed that is parallel to one of the primary viewing planes.

note taker: The person who records the actions discussed and taken at team meetings and then prepares the formal written notes for the meeting.

notes: Additional information or instructions placed on a drawing that are not contained on the dimensions, tolerances, or header.

nut: The threaded mate to a bolt used to hold two or more pieces of material together.

oblate ellipsoid: An ellipsoid created by rotating an ellipse around its minor axis.

oblique axes: A set of three coordinate axes that are portrayed on the paper as two perpendicular lines, with the third axis meeting them at an angle, typically 45 degrees.

oblique pictorial: A sketch of an object that shows one face in the plane of the paper and the third dimension receding off at an angle relative to the face.

oblique surface: A flat surface on an object being viewed that is neither parallel nor perpendicular to any of the primary views.

offset section: The section view produced by a stepped cutting plane that is used to hypothetically cut an object completely into two pieces. Different portions of the plane are used to reveal the interior details of different features of interest.

one-off: A one-of-a-kind engineering project for which no physical prototypes are created.

optimization: Modification of shapes, sizes, and other variables to achieve the best performance based on predefined criteria.

organizational chart: A chart representing the relationships of entities of an organization in terms of responsibility or authority.

orthogonal projection: The process by which the image of an object is created on a viewing plane by rays from the object that are perpendicular to that plane.

outline assembly drawing: *See* layout drawing.

parallel: An imaginary line parallel to the equator that corresponds to a measurement of latitude either N or S in a geographic coordinate system.

parameters: The attributes of features, such as dimensions, that can be modified.

parametric solid modeling: A solid modeling system that allows the user to vary the dimensions and other parameters of the model.

parametric techniques: Modeling techniques where all driven dimensions in algebraic expressions must be known for the value of the dependent variables to be calculated.

parent feature: A feature used in the creation of another feature, which is called its child feature.

part: An object expected to be delivered from a fabricator as a single unit with only its external dimensions and functional requirements specified.

part name: A very short descriptive title given to a part, subassembly, or device.

part number: Within a company, a string of alphanumeric characters used to identify a part, a subassembly, an assembly, or a device.

particle system: Specialized software modules used to generate, control, and animate very large numbers of small objects involved in complex events.

parts list: *See* bill of materials.

passive animation: An animation in which the observer remains still while the action occurs around him or her.

patents: A formal way to protect intellectual property rights for a new product.

path: The specified curve on which a profile is placed to create a swept solid.

pattern: A master part from which molds can be made for casting final parts.

perspective drawing: A drawing in which all three-dimensional axes on an object can be seen, with the scaling factor linearly increasing or decreasing in each direction. Usually, one axis is shown as being vertical. This type of drawing generally offers the most realistic presentation of an object.

pictorial: A drawing that shows the 3-D aspects and features of an object.

picture plane (PP): In the construction of a perspective view, the viewing plane through which the object is seen. The PP appears as a line (edge view of the viewing plane) in the plan view.

pie chart: A circular chart that is divided into wedges like a pie, representing a piece of the whole.

pin: A cylindrical (or slightly tapered) fastener typically used to maintain a desired position or orientation between parts.

pitch: The distance from one point on a thread to the corresponding point on the adjacent thread as measured parallel to its axis.

pitch diameter: The diameter of an imaginary cylinder that is halfway between the major and minor diameters of the screw thread.

pivot point: An independent, movable coordinate system on an object that can be used for location, orientation, and scale transformations.

pixel: The contraction for "picture element"; the smallest unit of information within a grid or raster data set.

plan and profile drawings: Construction drawings typically used for roads or other linear entities that show the road from above as well as from the side, with the profile view usually drawn with an exaggerated vertical scale.

plan view: In the construction of a perspective view, the object as viewed from the top, as if created by orthogonal projection.

plan views: Drawings created from a viewpoint above the structure (top view).

planar coordinate system: A 2-D measurement system that locates features on a plane based on their distance from an origin (0,0) along two perpendicular axes.

point: A spatial feature that has only location, has neither length nor area, and is represented by a pair of xy-coordinates.

point light: A computer-generated light source, also called an omni light, that emits light rays and casts shadows uniformly in all directions. Also called an omnidirectional light.

point tracking: A procedure by which successive vertices on an object are simultaneously located on a pictorial image and a multiview image of that object.

polygon: A spatial feature that has location, area, and perimeter and is represented by a series of nodes, points, and arcs that must form a closed boundary.

portrait: The drawing orientation in which the vertical size is larger than the horizontal size.

preferred configuration: The drawing presentation of an object using its top, front, and right-side views.

primary modeling planes: The planes representing the xy-, xz-, and yz-planes in a Cartesian coordinate system.

primitives: The set of regular shapes, such as boxes, spheres, or cylinders that are used to build solid models with constructive solid geometry methods (CSG).

principal viewing planes: The planes in space on which the top, bottom, front, back, and right- and left-side views are projected.

problem identification: The first stage in the design process where the need for a product or a product modification is clearly defined.

procedural textures: Texture mapping routines based on algorithms written into the rendering software that can generate a specialized colored pattern such as wood, water, a checker pattern, a tile pattern, stucco, and many others without reference to external image files.

process check: A method for resolving differences and making adjustments in team performance.

process diagram: An illustration that explains how system elements work and how interactions occur.

professional engineer (PE): An individual who has received an engineering degree, who has worked under the supervision of a PE for a number of years, and who has passed two examinations certifying knowledge of engineering practice.

profile: A planar sketch that is used to create a solid.

profile surface: A surface on an object being viewed that is parallel to a side viewing plane.

profile views: Views of a structure that show horizontal surfaces in edge view (side or front views).

project: In engineering, a collection of tasks that must be performed to create, operate, or retire a system or device.

projection ray: A line perpendicular to the projection plane. It transfers the 2-D shape from the object to an adjacent view. Projection rays are drawn lightly or are not shown at all on a finished drawing.

prototype: The initial creation of a product for testing and analysis before it is mass-produced.

protrusion: A feature created by the addition of solid volume to a model.

qualitative data: Information collected using words and ideas.

quantitative data: Numeric information.

quantity per machine (Q/M): The number of times a part is required to build its next highest assembly.

radius-of-gyration: The distance from an axis where all of the mass can be concentrated and still produce the same moment-of-inertia.

rapid prototyping: Various methods for creating a part quickly by selective hardening of a powder or liquid raw material at room temperature.

raster data model: A representation of the geographic location as a surface divided into a regular grid of cells or pixels.

raytraced shadows: Shadows calculated by a process called raytracing, which traces the path that a ray of light would take from the light source to illuminate or shade each point on an object.

raytracing: A method of rendering that builds an image by tracing rays from the observer, bouncing them off the surfaces of objects in the scene, and tracing them back to the light sources that illuminate the scene.

reaming: A process for creating a hole with a very accurate final diameter using an accurately made cylindrical cutting tool similar to a drill bit to remove final bits of material after a smaller initial hole is created.

rebars: Steel bars added to concrete for reinforcement or for temperature control.

receding dimension: The portion of the object that appears to go back from the plane of the paper in an oblique pictorial.

record: A set of related data fields, often a row in a database, containing the attribute values for a single feature.

reference dimensions: Unneeded dimensions shown for the convenience of the reader used to show overall dimensions that could be extracted from other dimensions on the part or from other drawings.

reference line: Edges of the glass box or the intersection of the perpendicular planes. The reference line is drawn only when needed to aid in constructing additional views. The reference line should be labeled in constructing auxiliary views to show its association between the planes it is representing; for example, H/F for the hinged line between the frontal and horizontal planes. A reference line is also referred to as a fold line or a hinged line.

reflection: The process of obtaining a mirror image of an object from a plane of reflection.

reflection mapping: Mapping that allows the use of grayscale values in an image file to create the illusion of a reflection on the surface of a part. White creates reflective highlights, while black is transparent to the underlying color of the surface.

regeneration: The process of updating the profile or part to show its new shape after constraints are added or changed.

related views: Views adjacent to the same view that share a common dimension that must be transferred in creating auxiliary views.

removed section: The section view produced when a cutting plane is used to hypothetically remove an infinitesimally thin slice of an object for viewing.

rendering: The process where a software program uses all of the 3-D geometric object and lighting data to calculate and display a finished image of a 3-D scene in a 2-D viewport.

retaining rings: Precision-engineered fasteners that provide removable shoulders for positioning or limiting movement in an assembly.

reverse engineering: A systematic methodology for analyzing the design of an existing device.

revolved section: The section view produced when a cutting plane is used to hypothetically create an infinitesimally thin slice, which is rotated 90 degrees for viewing, on an object.

revolved solid: A solid formed when a profile curve is rotated about an axis.

ribs: Constant thickness protrusions that extend from the surface of a part and are used to strengthen or stiffen the part.

right-hand rule: Used to define a 3-D coordinate system whereby by pointing the fingers of the right hand down the *x*-axis and curling them in the direction of the *y*-axis, the thumb will point down the *z*-axis.

right-handed system: Any 3-D coordinate system that is defined by the right-hand rule.

rivet: A cylindrical pin with heads at both ends, one head being formed during the assembly process, forming a permanent fastener often used to hold sheet metal together.

rolling: A process for creating long bars with flat, round, or rectangular cross sections by squeezing solid raw material between large rollers. This can be done when the material is in a hot, soft state (hot rolling) or when the material is near room temperature (cold rolling).

root: The bottom surface or point of a screw thread.

rounds: Smooth radius transitions of external edges created by two intersecting surfaces and tangent to both intersecting surfaces.

sand casting: A casting process where the mold is made of sand and binder material hardened around a master pattern that is subsequently removed to form the cavity. *See* casting.

sawing: A cutting process that uses a multitoothed blade that moves rapidly across and then through the part.

scatter plot: A graph using a pattern of dots showing the relationship between two sets of data.

schedule of materials: A list of the materials, such as doors and windows, necessary for a construction project.

schematic diagram: A diagram explaining how components work together, what the measurements are, how components are set up, or how pieces are connected.

screw thread: A helix or conical spiral formed on the external surface of a shaft or on the internal surface of a cylindrical hole.

scripting: A programming capability that allows a user to access and write code at or near the source code level of the software.

secondary title block: An additional bordered area of a drawing (and part of the drawing header) that contains important information about the identification, fabrication, and history of the item shown on the drawing.

section lines: Shading used to indicate newly formed or cut surfaces that result when an object is hypothetically cut.

section view: A general term for any view that presents an object that has been hypothetically cut to reveal the interior details of its features, with the cut surfaces perpendicular to the viewing direction and filled with section lines for improved presentation.

sectioned assembly drawing: A formal drawing, usually in pictorial form, that shows the device in its assembled form but with sections removed from obscuring parts to reveal formerly hidden parts.

selective laser sintering (SLS): A process where a high-powered laser is used to selectively melt together the particles on a bed of powdered metal to form the shape of a desired part.

self-tapping screw: A fastener that creates its own mating thread.

sequence diagram: A group of diagrams that includes process diagrams, timelines, and step-by-step diagrams.

set of construction plans: A collection of drawings, not necessarily all of them plan views, needed to construct a building or infrastructure project.

set screw: A small screw used to prevent parts from moving due to vibration or rotation, such as to hold a hub on a shaft.

shading: Marks added to surfaces and features of a sketch to highlight 3-D effects.

shading algorithms: Algorithms designed to deal with the diffuse and specular light transmission on the surface of an object.

shelling: Removing most of the interior volume of a solid model, leaving a relatively thin wall of material that closely conforms to the outer surfaces of the original model.

sidebar: Small infographics used within a body of text that are subdivided into briefs, lists, and bio profiles.

single thread: A thread that is formed as one continuous ridge.

sintering: A process where a part is formed by placing powdered metal into a mold and then applying heat and pressure to fuse the powder into a single solid shape.

site plan: A plan view showing the construction site for an infrastructure project.

site survey: Data regarding the existing topography and structures gathered during the preliminary design stages by trained surveying crews.

six standard views (or six principal views): The drawing presentation of an object using the views produced by the glass box (i.e., the top, front, bottom, rear, left-side, and right-side views).

size: The general term for the size of a feature, such as a hole, cylinder, or set of opposed parallel surfaces.

sketches: Collections of 2-D entities.

sketching editor: A software tool used to create and edit sketches.

sketching plane: A plane where 2-D sketches and profiles can be created.

slope: The rate of change of elevation (rise) over a specified distance (run). Measured in percent or degrees.

solid model: A mathematical representation of a physical object that includes the surfaces and the interior material, usually including a computer-based simulation that produces a visual display of an object as if it existed in three dimensions.

solid modeling: Three-dimensional modeling of parts and assemblies originally developed for mechanical engineering use but presently used in all engineering disciplines.

spatial data: A formalized schema for representing data that has both geographic location and descriptive information.

spatial orientation: The ability of a person to mentally determine his own location and orientation within a given environment.

spatial perception: The ability to identify horizontal and vertical directions.

spatial relations: The ability to visualize the relationship between two objects in space, i.e., overlapping or nonoverlapping.

spatial visualization: The ability to mentally transform (rotate, translate, or mirror) or to mentally alter (twist, fold, or invert) 2-D figures and/or 3-D objects.

specifications (specs): The written instructions that accompany a set of construction plans used to build an infrastructure project.

spheroid: *See* oblate ellipsoid.

spindle: That part of a production cutting machine that spins rapidly, usually holding a cutting tool or a workpiece.

splines: Polynomial curves that pass through multiple data points.

split line: The location where a mold can be disassembled for removal of a part once the molten raw material inside has solidified.

spotlight: A computer-generated light that simulates light being emitted from a point in space through a cone or beam, with the angle and direction of light controlled by the user.

spring pin: A hollow pin that is manufactured by cold-forming strip metal in a progressive roll-forming operation. Spring pins are slightly larger in diameter than the hole into which they are inserted and must be radially compressed for assembly.

sprue: Bits of material that are left on the part from a casting or molding operation and found at the ports where the molten material is injected into the mold or at the ports where air is allowed to escape.

stage: That part of a machine that secures and slowly moves a cutting tool or workpiece in one or more directions.

stamping: A process for cutting and shaping sheet metal by shearing and bending it inside forms with closely fitting cutouts and protrusions.

standard commercial shape: A common shape for raw material as would be delivered from a material manufacturer.

State Plane Coordinate System: The planar coordinate system developed in the 1930s for each state to permanently record the locations of the original land survey monuments in the United States.

station point (SP): In the construction of a perspective view, the theoretical location of the observer who looks at the object through the picture plane.

statistical tolerancing: A way to assign tolerances based on sound statistical practices rather than conventional tolerancing practices.

step segment: On a stepped cutting plane for an offset section view, that portion of the plane that connects the cutting segments and is usually perpendicular to them but does not intersect any interior features.

step-by-step diagram: An illustration that visually explains a complex process; it is a type of a sequence diagram.

stereolithography (SLA): A process for creating solid parts from a liquid resin by selectively focusing heat or ultraviolet light into a pool of the resin, causing it to harden and cure in the selected areas.

storyboard: A sequential set of keyframe sketches or drawings, including brief descriptions, indications of object and camera movement, lighting, proposed frame numbers, and timelines sufficient to produce a complete animation project.

stud: A fastener that is a steel rod with threads at both ends.

subassemblies: Collections of parts that have been put together for the purpose of installing the collections as single units into larger assemblies.

subassembly: A logical grouping of assembly instances that is treated as a single entity within the overall assembly model.

successive cuts: A method of forming an object with a complex shape by starting with a basic shape and removing parts of it through subtraction of other basic shapes.

suppressed: Refers to the option for not displaying a selected feature.

surface area: The total area of the surfaces that bound an object.

surface model: A CAD-generated model created to show a part as a collection of intersecting surfaces that bound a solid.

surface modeling: The technique of creating a 3-D computer model to show a part or an object as a collection of intersecting surfaces that bound the part's solid shape.

surface normal: A vector that is perpendicular to each polygon contained in a polygon mesh model.

surface tracking: A procedure by which successive surfaces on an object are simultaneously located on a pictorial image and a multiview image of that object.

sustainable design: A paradigm for making design decisions based on environmental considerations and life cycle analysis.

swept feature: A solid that is bound by the surfaces swept out in space as a profile is pulled along a path.

symmetry: The characteristic of an object in which one half of the object is a mirror image of the other half.

system: A collection of parts, assemblies, structures, and processes that work together to perform one or more prescribed functions.

table: Data organized in columns and rows.

tangent edge: The intersection line between two surfaces that are tangent to each other.

tap: A cutting tool similar to a drill bit used to create screw threads inside a hole

tap drill: A drill used to make a hole in material before the internal threads are cut.

tapped hole: A hole that has screw threads inside it.

task credit matrix: A table that lists all team members and their efforts on project tasks.

team contract: The rules under which a team agrees to operate (also known as a code of conduct, an agreement to cooperate, or rules of engagement).

team leader: The person who calls the meetings, sets the agenda, and maintains the focus of team meetings.

team roles: The roles that team members fill to ensure maximum effectiveness for a team.

technical diagram: A diagram depicting a technical illustration's measurements, movement, dissection, or relationship of parts.

telephoto: As seen through a camera lens with a focal length longer than 80 degrees, creating a narrow field of view and resulting in a flattened perspective.

texture mapping: The technique of adding variation and detail to a surface that goes beyond the level of detail modeled in the geometry of an object.

thematic layer: Features of one type that are generally placed together in a single georeferenced data layer.

thematic layer overlay: The process of combining spatial information from two thematic layers.

third-angle projection: The process of creating a view of an object by imprinting its image, using orthogonal projection, on a translucent surface in front of that object.

thread note: Information on a drawing that clearly and completely identifies a thread.

thread series: The number of threads per inch on a standard thread.

3-D coordinate system: A set of three mutually perpendicular axes used to define 3-D space.

3-D printing: A process for creating solid objects from a powder material by spraying a controlled stream of a binding fluid into a bed of that powder, thus fusing the powder in the selected areas.

three-axis mill: A milling machine whose spindle, which holds the rotating cutting tool, can be oriented along any one of three Cartesian axes.

three-dimensional (3-D) modeling: Mathematical modeling where the appearance, volumetric, and inertial properties of parts, assemblies, or structures are created with the assistance of computers and display devices.

through hole: A hole that extends all the way through a part.

tick mark: A short dash used in sketching to locate points on the paper.

timekeeper: The person who keeps track of the meeting agenda, keeping the team on track to complete all necessary items within the allotted time frame.

timeline: A specific type of sequence diagram used to highlight significant moments in history.

title block: Usually the main title block, which is a bordered area of the drawing (and part of the drawing header) that contains important information about the identification, fabrication, history, and ownership of the item shown on the drawing.

tolerance: The total amount a specific dimension is permitted to vary. It is the difference between the upper and lower limits of the dimension.

tool bit: A fixed or moving replaceable cutting implement with one or more sharpened edges used to remove material from a part.

tool runout: The distance a tool may go beyond the required full thread length.

tooling: Tools and fixtures used to hold, align, create, or transport a part during its production.

top-down modeling: The process of establishing the assembly and hierarchy before individual components are created.

trail: Dashed lines on an assembly drawing that show how various parts or subassemblies are inserted to create a larger assembly.

trajectory: *See* path.

transparency/opacity mapping: A technique used to create areas of differing transparency on a surface or an object.

trimetric drawing: An axonometric drawing in which the scaling factor is different for all three axes.

true shape (of a plane): The actual shape and size of a plane surface as seen in a view that is parallel to the surface in question.

tumbling: A process for removing sharp external edges and extraneous bits of material from a part by surrounding it in a pool of fine abrasive pellets and then shaking the combination.

turning: A process for making axially symmetric parts or features by rotating the part on a spindle and applying a cutting to the part.

two-dimensional (2-D) drawing: Mathematical modeling or drawing where the appearance of parts, assemblies, or structures are represented by a collection of two-dimensional geometric shapes.

undercut feature: A concave feature in which the removed material expands outward anywhere along its depth.

union: A topological overlay of two or more polygon spatial datasets that preserves the features that fall within the spatial extent of either input dataset; that is, all features from both datasets are retained and extracted into a new polygon dataset.

Universal Transverse Mercator (UTM): The planar coordinate system that divides the earth's surface between 84° N and 80° S into 60 zones, each 6° longitude wide.

unsuppressed: Refers to the option for displaying a selected feature.

U.S. sheet sizes: The accepted paper dimensions used in the United States when drawings are created or printed to their intended size.

vanishing point (VP): In the construction of a perspective view, the point on the horizon where all parallel lines in a single direction converge.

variational techniques: Modeling techniques in which algebraic expressions or equations that express relationships between a number of variables and constants, any one of which can be calculated when all of the others are known.

vector data model: A data model that uses nodes and their associated geographic coordinates to construct and define spatial features.

Venn diagram: A type of business diagram that shows the mathematical or logical relationships and overlapping connections between different groups or sets of information.

vertex: A point that is used to define the endpoint of an entity such as a line segment or the intersection of two geometric entities.

video compression: One of a number of algorithms designed to reduce the size and storage requirements of video content.

viewing direction: The direction indicated by arrows on the cutting plane line from the eye to the object of interest that corresponds to the tail and point of the arrow, respectively.

viewing plane: A hypothetical plane between an object and its viewer onto which the image of the object, as seen by the viewer, is imprinted.

visual storytelling diagram: An illustration that displays empirical data or clarification of ideas.

visual thinking: A method for creative thinking, usually through sketching, where visual feedback assists in the development of creative ideas.

visualization: The ability to create and manipulate mental images of devices or processes.

volume: The quantity of space enclosed within an object's boundary surfaces.

volume of interference: The volume that is common between two overlapping objects.

wall sections: Sectional views of walls from foundation to roof for a construction project.

washer: A flat disk with a center hole to allow a fastener to pass through it.

webs: Small, thin protrusions that connect two or more thicker regions on a part.

weighted decision table: A matrix used to weigh design options to determine the best possible design characteristics.

welding: A method for joining two or more separate parts by applying heat to the edges where they meet and melting the edges together along with a filler of essentially the same material composition as the parts.

wide-angle: As seen through a camera lens with a focal length shorter than 30 degrees, creating a wide field of view and resulting in a distorted and exaggerated perspective.

wire drawing: The process of reducing the diameter of a solid wire by pulling it through a nozzle with a reducing aperture.

wireframe models: CAD models created using lines, arcs, and other 2-D entities to represent the edges of the part; surfaces or solid volumes are not defined.

working drawings: A collection of all drawings needed to fabricate and put together a device or structure.

workpiece: A common name for a part while it is still in the fabrication process, that is, before it is a finished part.

z-buffer rendering: A scene-rendering technique that uses visible-surface determination in which each pixel records (in addition to color) its distance from the camera, its angle, light source orientation, and other information defining the visible structure of the scene.

z-value: The value for a given surface location that represents an attribute other than position. In an elevation or terrain model, the z-value represents elevation.

INDEX

B

C

D

G

N

O

Q

R

S

V

W

X

Y

Z