SECTION A

Introduction

INTRODUCTION

Machining processes are among the most important of the manufacturing processes. The fundamental cutting processes in machining such as milling and turning are still the most prominent. What has changed is the way in which these processes are applied. Changes include the cutting tool materials, the materials the parts are made from, and the methods of material removal. These new methods include the use of lasers, electrical energy, electrochemical processes, and high-pressure water jets.

Computer numerically controlled (CNC) machines have provided an exceptional degree of accuracy, reliability, and repeatability. This application of computer-driven automated equipment is creating many employment opportunities for skilled machinists and machine operators.

The computer has found its way into almost every other phase of manufacturing as well. One important area is computer-aided design (CAD). Product design is done on computers and CNC programs are generated directly from those designs.

In order to be competitive, industry will continue to automate. There will be increased use of manufacturing cells of machines, and increased use of robotics for loading and unloading parts to/from CNC machines. The automation will continue to improve the working conditions in manufacturing, reduce the manual labor required, and increase the need for skilled and knowledgeable workers.

Units in This Section

The units in this section deal with careers, competitive manufacturing, safety, threads and fasteners, blueprint reading and GD&T.

UNIT ONE

Careers and the Machinist's Role in Process Plans

OBJECTIVES

After completing this unit, you should be able to:

- Discuss various types of jobs that are available.
- List some characteristics that employers look for in potential employees.
- Explain what a job packet is used for and what it typically includes.
- Explain what process planning is.

The increased use of computer technology and processes in industry has had a significant effect on the types and numbers of jobs available in manufacturing. Many exciting career opportunities are available in manufacturing and in machining in particular.

Manufacturing creates wealth. Manufacturing is crucial to economic success. There is a large demand for skilled manufacturing people. Wages and benefits for skilled manufacturing people are excellent. Wages and benefits for manufacturing are usually higher than other types of jobs. Working conditions have also dramatically changed. Most modern manufacturing facilities are clean, bright, and well organized. Modern computer-controlled equipment has dramatically changed the role of the machinist from manual dexterity and manual skills to more knowledge-based skills.

Helpers and Limited Machinists

A person's first job in manufacturing might be as a helper or limited machinist. They learn the trade from the floor up. They might be sweeping the floor or cutting stock to rough lengths for further machining. They are given additional responsibilities as they prove themselves in work. Remember that the supervisor is watching. The supervisor is looking to see if the new employee has work ethic and a good attitude. This determines whether they keep the job and how quickly they advance to becoming machinists.

Machine Operator/Production Machinist

A machine operator's responsibilities are to operate computer-driven (CNC) machine tools such as turning or machining centers. The operator observes machine functions and tool performance, inspects parts, and has limited duties in setting up and making minor changes to programs. Figure A-1 shows a machinist programming a part on a CNC turning center.

Machine operators should be familiar with basic machining processes, tooling selection and application, speeds and feeds, basic blueprint reading, basic math calculations, basic machine setup, and basic use of measurement tools. Machine operators are generally taught to use one or more CNC machine controls. Machine operators may receive training from a technical college, industrial training programs, or they may learn on the job.

Figure A-1 Machinist programming a part on a CNC turning center (*Courtesy of Fox Valley Technical College*).

Apprentice Machinist

The apprentice machinist learns the trade by entering a formal training program sponsored by a private company, a trade union, or a government entity. The period of training is typically four years and is a combination of on-the-job experiences and formal classroom education. Serving an apprenticeship represents one of the best methods of learning a skilled trade.

Journeyman Machinist

The journeyman machinist will have the capability to set up and operate most CNC equipment. A person becomes a journeyman by serving an apprenticeship. A machinist apprenticeship is generally four years long. States have apprenticeship divisions that facilitate apprenticeships. An apprenticeship is a contract between the apprentice, the company, and the state. The contract is called an indenture. The role of the state apprenticeship division is to make sure that the apprentice learns a broad range of knowledge and skills during the apprenticeship. The apprenticeship agreement will specify the number of hours to complete the apprenticeship. A four-year apprenticeship would be approximately 8,320 hours. The pay for the apprentice generally increases every six months until the apprentice finishes and gets full pay. The agreement also specifies the hours of required education. The apprentice must attend the required classes during the apprenticeship. Classes are generally one day a week during the normal school year. Classes are generally offered at a technical or community college. Employers might give a beginning apprentice credit for prior experience or classes already taken. This boosts their pay and shortens the length of time required to complete the apprenticeship. When an apprentice finishes, he/she is awarded a journeyman's card by the state. The card is proof that he or she has completed an apprenticeship. A journeyman's card is held in high regard by potential employers if the machinist wants to change employers.

Tool and Die Maker

A tool and die maker is usually an experienced machinist who served a tool and die apprenticeship. The tool and die apprenticeship is usually five years and requires more classes than a machinist apprenticeship. Tool and die makers may receive training through industrial apprenticeships and/or college and trade school programs. Although tool and die makers are often chosen only after several years of on-the-job experience, it is possible to start out in tool and die work through an apprenticeship program. Tool and die makers often receive premium pay for their work and are involved with many high-precision machining applications, tool design, material selection, metallurgy, and general manufacturing processes. Tool and die makers may be involved with making tooling and fixtures for automated processes and production.

Quality Control Technician

Quality control technicians are involved with quality control and inspection. They utilize common measurement equipment as well as computerized measurement machines to inspect parts to make sure they meet the specifications (see Figure A-2).

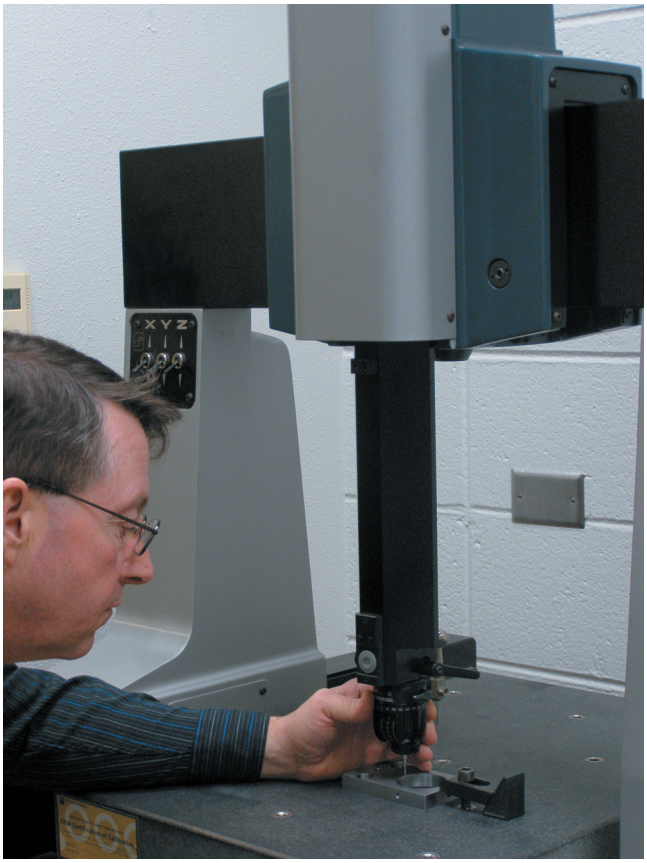


Figure A-2 Quality control technician using a CMM to inspect a part (*Courtesy of Fox Valley Technical College*).

When parts are complex and or costly, the quality control technician often checks the first part coming out of a machine to be sure they meet requirements before more parts are run. Quality control technicians often have the responsibility for measurement tool calibration systems as well. They make sure that measurement equipment is accurate throughout the plant. Many quality control technicians began as machinists and moved into a quality control position. The quality control technician may receive training through college and trade school programs or on the job.

Supervisor

Many shop supervisors begin their careers as machinists. A machinist who proves his or her ability might be chosen to be a supervisor. A supervisor is responsible for a group of people and a part of the production process. They may further their education by taking supervisory management classes at a local technical college, community college, or university.

CNC Programmer

Some shops centralize the programming of CNC machines. Instead of having machinists write their own programs, they have specialized people doing the CNC programming with specialized software (Figure A-3). A machinist is very well prepared to be a CNC programmer. A machinist has knowledge of

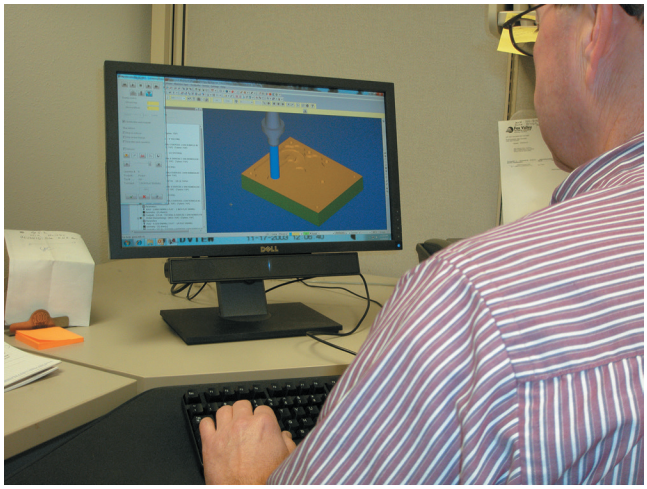


Figure A-3 A CNC programmer utilizing a CAD/CAM system to program a CNC machine (*Courtesy of Fox Valley Technical College*).

tooling, machining operations, feeds and speeds, and materials. To become a CNC programmer, a machinist would take a class on the software or be trained by another employee on the software that the company uses for programming.

Manufacturing Engineer

The manufacturing engineer and/or industrial engineer is involved with the application of manufacturing technology. These individuals may be involved with the design of manufacturing tooling, setting up manufacturing systems, applying computers to manufacturing requirements, new product development, and improving efficiency.

Estimator/Bidder

Most machine shops do not make complete products. They produce parts for larger companies that do produce products. Shops that produce parts for other companies are called job shops. Job shops must bid on work. An estimator/bidder looks at potential parts that need to be made and develops an estimate of how much it will cost to make. They consider costs such as material, tooling, machine, special processes, labor, shipping, and so on. They then develop a quote that is sent to the company that is requesting quotes on the parts. If the quote is accepted, the shop will get an order for the parts. The estimator/bidder must be skilled at developing the quotes. The quote price must be high enough to make a profit, but low enough to beat bids from other shops.

Shop Owner

Most machine shops were started by machinists. Many shop owners were machinists who learned the trade, gained experience, and then bought one or more CNC machines and started their own company. Many manufacturing companies started this way and grew very rapidly to become large, successful businesses.

SPECIAL OCCUPATIONS

Automotive Machinist

The automotive machinist will work in an engine rebuilding shop where engines are overhauled. This person's responsibilities will be somewhat like those of the general machinist, with specialization in engine work, including boring, milling, and some types of grinding applications. Training for this job may be obtained on the job or through college or trade school programs.

Maintenance Machinist

The maintenance machinist has broad responsibilities. The person may be involved in plant equipment maintenance, machine tool rebuilding, or general mechanical repairs, including welding and electrical repairs. The maintenance machinist is often involved with general machining as well as with general industrial mechanical work. The maintenance machinist normally just uses simple manual machines to make or repair parts for machines.

MACHINING AND YOU

There are many exciting occupations available in manufacturing. These occupations provide a good working environment with wonderful wages and benefits as well as opportunities for advancement. Industry really needs people with good attitude and work ethic. There is a real demand for, and shortage of, individuals with good attitude and good work ethic. Industry is more than willing to invest in training and advancement for those individuals. Professional behavior will also serve you well. Appropriate clothing and language help promote a professional personal image. Dealing well with people is also an important attribute. A working knowledge of the machining processes and the related subjects described in this book will provide an excellent basis on which to build a successful career in manufacturing.

FURTHER INFORMATION ABOUT OCCUPATIONS IN MANUFACTURING

If you have further interest in machine shop career opportunities, discuss opportunities with an employment counselor at your school or a program instructor. Study the job advertisements in local newspapers to see what types of jobs are available. Talk to an employment agency about jobs that are in demand and the skills required to get them. Contact local manufacturing companies that employ machinists. Ask them for an opportunity to talk about employment opportunities and a tour of their facility. You may well find an employer who is willing to hire you as an intern while you go to school.

Job Packets and Process Planning

Most manufacturing companies produce parts for other companies. They must bid on jobs to get work for their shop. A larger manufacturer typically solicits bids on work they need done. For example, assume a manufacturer (ACME Manufacturing) that makes and sells snow blowers. They do much of the work themselves, especially the assembly of the parts into the completed machine. But they have other companies produce many of the component parts that go into their final machine. These smaller machine shops are typically called job shops. Job shops do not have their own products. They make parts for other companies. Imagine ACME needs small axle-shafts made for their snow blowers. ACME puts out bid requests to their list of suppliers. The bid request would normally be transmitted electronically to all suppliers. The bid request would include a blueprint, material specifications, number of parts needed, due dates, any special requirements for packing shipping, etc. Each job shop can then evaluate the bid request to determine whether or not to bid on the work.

Imagine how one job shop (XYZ Machining) might evaluate the bid. XYZ receives the bid request in an email. XYZ has a customer service representative (CSR) who works with customers. The CSR looks at the bid request to see how many parts and what type of machining is required to quickly decide if XYZ has the machining capability to make the parts. The CSR must also determine if XYZ can produce the parts by the due date while considering all of the other work XYZ already has to do. Assuming XYZ has the machining capability and the time to make the parts, a price would need to be calculated. The CSR might confer with their CNC turning center supervisor to determine feasibility and cost per part. XYZ would also need to get a price on the steel that is required and make sure it will be available in the needed timeframe. Tooling costs, machine times, packaging, and shipping would all need to be considered. XYZ would also evaluate whether or not this is a customer they want to work for. XYZ would evaluate based on past history, if they have worked for them before. Did they pay their bills on time? Were they difficult to work with, and so on? If everything looks good at this point, the CSR would put together a bid price for the parts. Note that the price would need to be low enough to get the job, but high enough to make a profit. When there is a lot of work, companies tend to bid higher. When work is scarce bids get very competitive. In fact, in slow times a company might take work they normally would not take, just to keep their workers working. Note that ACME might accept bids from more than one company for parts. Their real need might be 10,000 shafts and they might split it between two job shops to reduce ACME's risk.

Assume that ACME Manufacturing accepts the bid from XYZ Machining. The order calls for 1,000 parts a month from January through May. The CSR orders steel once the bid is awarded by ACME. The CSR begins to prepare a job packet and schedules the job. Assume the parts must be shipped three days before the month end and it will take 38 hours of setup and machine time to make them. The CSR must evaluate the shop schedule and determine when the parts should be scheduled so these parts and all others can be made and shipped on time. The CSR might have to consult with the supervisor of that area in the shop. The CSR then puts a job packet together (Figure A-4). The job packet will include a current revision of the blueprint with all of the part specifications. The packet may include

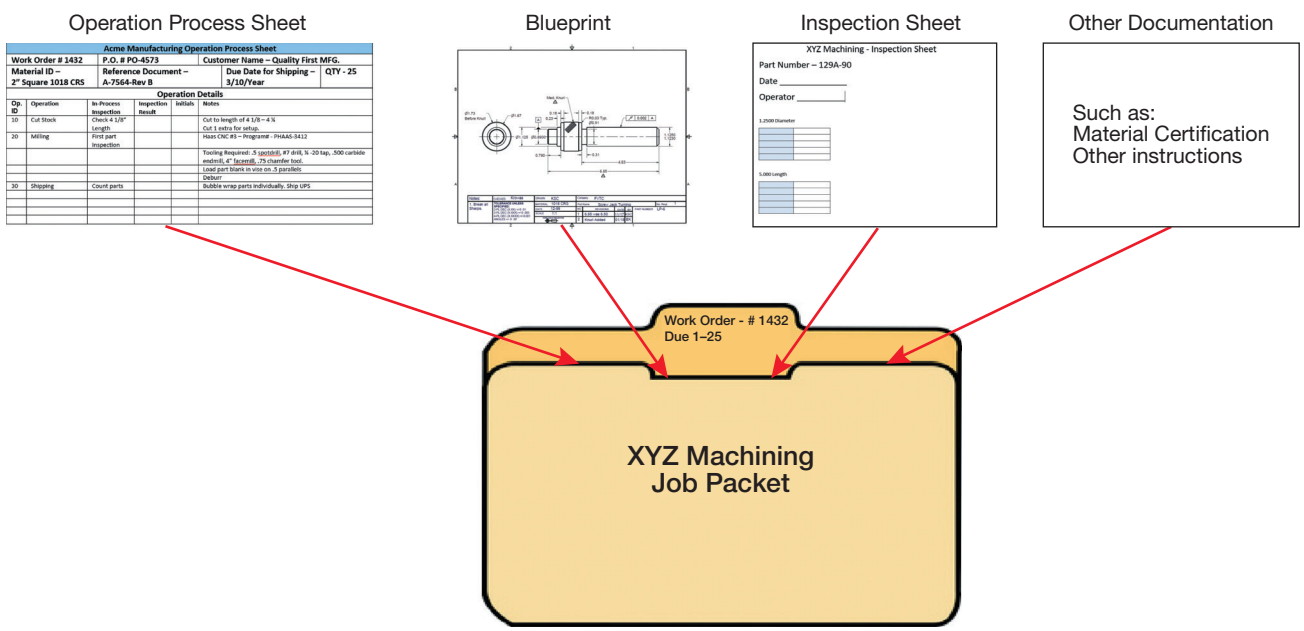


Figure A-4 Job Packet.

inspection sheets if special inspections are required. XYZ has a person who does process plans. The process planner develops an operation process sheet for the job. The process planner might also develop the CNC program for the parts, although the machinists program their own parts in many shops. If XYZ had run these parts before for ACME, the program would have already been written and the program number and machine would be specified on the operation process sheet.

Figure A-4 shows an example of the types of things that might go into a job packet. Job packets may also be called shop packets.

Process Plans

A process plan details all of the things that need to be done to manufacture a part (Figure A-5). They are step-by-step instructions that are done by the machinist. The steps detail each machining operation, inspections that need to be done, deburring operations, and so on. The process plan also references the blueprint and correct revision level that needs to be used. The process plan may list the CNC program to be used, the machine, and any special setup instructions. The process plan would also detail and special handling, packaging, and or shipping requirements.

Process plans are also called routings or routers.

Reference Document

The reference document is typically the customer-supplied blueprint. This is very important information. The machinist must always check to make sure the blueprint is the current revision. In this example, the blueprint number is A-7564-Rev B. The blueprint number must be the same exact number and revision level or the parts will be wrong.

Work Order Number

The work order number is an internal number. The company the machinist works for assigns a work order number to each order it accepts from customers. This number is used for internal tracking of all information about this part run.

P.O. Number

This is the purchase number. This number ties the order to a particular customer and order. It is the number the company will use to get paid by the customer after delivery of the parts.

Material Specification

This is where the material for this part is specified. This specification must match the material specified by the blueprint.

Due Date for Shipping

A machine shop must meet their customers' requirements. One of the most important is on-time delivery. A customer must have parts when they are needed. If a machine shop is not consistently on time with deliveries, the customer will find a new machine shop to work with.

Figure A-5 An operation process plan.

XYZ Manufacturing Operation Process Plan					
Work Order # 1432		P.O. # PO-4573		Customer Name – ACME	
Material Spec. – 2″ Square 1018 CRS		Reference Document – A-7564-Rev B		Due Date for Shipping – 5/25/Year	QTY - 1000
Operation Details					
Op. ID	**Operation**	**In-Process Inspection**	**Inspection Result**	**initials**	**Notes**
10	Cut Stock	Check 4 1/8″ Length			Cut to length of 4 1/8 – 4 ¼ Cut 1 extra for setup.
20	Milling	First part Inspection			Haas CNC #3 – Program# - PHAAS-3412
					Tooling Required: .5 spotdrill, #7 drill, ¼ -20 tap, .500 carbide endmill, 4″ facemill, .75 chamfer tool.
					Load part blank in vise on .5 parallels
					Deburr
30	Shipping	Count parts			Bubble wrap parts individually. Ship UPS

Operation

Operation is used to describe the operation that will be performed in this step. Note that there may be several steps and operations to be performed. In this example, there are only three operations. The first is to cut 25 blanks of 2″ square 1018 CRS (cold roll steel) to a length of 4 1/8″ to 4 $\frac{1}{4}$″. The note also calls for an extra blank to be cut for setting up a first piece.

SELF-TEST

1. Develop a personal career plan. What job would you like to start your career in? What would you like to be doing in five years? Fifteen years?
2. List five things you can do to achieve your 15-year goal.
3. Companies are looking for people with good attitude and work ethic. List at least five things that describe the attributes of good attitude. List at least five things that describe the attributes of work ethic.
4. How does a company get jobs to do?
5. Name some considerations that go into a bid.
6. What is a process control plan?
7. What is a router?
8. What typically is included in a job packet?

UNIT TWO

Manufacturing Competitiveness and Improvement

OBJECTIVES

After completing this unit, you should be able to:

- Explain at least three techniques for improving competitiveness.

MANUFACTURING COMPETITIVENESS

Manufacturing is very competitive. A large percentage of manufacturing is done in job shops. They must bid on jobs to get work. The shop with the lowest bid that meets specifications will generally get the work. One of the techniques that manufacturers have been adopting to increase their efficiency is *lean manufacturing*.

Lean manufacturing is a technique that views the investment of a company's resources for any reason other than the creation of value for the customer to be wasted. The goal is to reduce or eliminate costs that do not create value for the customer. Customer value could be defined as any process that the customer would be willing to pay for. Many of the concepts and techniques of lean manufacturing have been around for a very long time.

Lean manufacturing is concerned with producing the product with fewer resources. Many of the concepts are based on principles from the Toyota Production System (TPS). The TPS was based on reducing waste and it identified seven principal wastes.

Overproduction

Overproduction is defined as making a product before it is actually required. Overproduction is very costly because it prevents the efficient flow of materials and adversely affects productivity and quality. Overproduction often hides the real problems and opportunities for improvement in manufacturing. The goal is to make only what can be sold or shipped immediately. This is often called just-in-time.

Wait Time

If a product is not moving or is being worked on, waste occurs. In a typical company, more than 99 percent of a product's time in the plant might be spent in waiting to be processed. This is often because distances between work centers are excessive, production runs are too long, and material flow is often very inefficient. Waiting can be reduced by having each process feed directly into the next process.

Unnecessary transportation

Transporting products between processes is a cost that does not add any value to the product. Excessive transportation and product handling can cause damage. When people have to excessively transport parts in a plant, it takes time from their real role of adding value to the product. This is another cost that adds no value.

Excess Inventory

Excess inventory or work in progress (WIP) is a direct result of overproduction and waiting. Excess inventory hides problems on the plant floor. Excess inventory wastes floor space, increases lead times, and dramatically increases costs.

Unnecessary Motion

Unnecessary motion by people or equipment is a waste. This waste is related to ergonomics and is seen in all unnecessary movement of people performing an operation or task.

Over Processing

Over processing often involves the use of expensive, high-cost equipment where simpler tools would do the job. The use of high-cost technology often results in higher quantity runs to try and recover the high cost of the equipment. Toyota is famous for using low-cost automation and well-maintained, older machines. If a machine is needed to produce X number of parts, Toyota would typically by a low-cost machine to

produce that many parts. If production demands increase, they would buy another low-cost machine to meet the increased demand. American manufacturers would typically buy a more expensive machine that could produce many more than the required parts. There is less risk involved with the Toyota method. When possible, companies should invest in smaller and more flexible equipment.

Defective Product

Quality defects that cause scrap or rework are a huge cost to manufacturers. When scrap is produced, many additional costs occur. These additional costs include quarantining the defective product, reinspecting it, and rescheduling the line to reproduce the scrapped product. This results in the loss of capacity as the time and resources used to reproduce the product cannot be used to create additional product. The total cost of defects is often a very significant percentage of the total cost of manufacturing.

An eighth waste has recently been added: underutilization of employees. Enterprises often hire employees to physically produce products but do not utilize their brain. Manufacturers must utilize their employees' creativity to eliminate the seven wastes and increase productivity.

LEAN MANUFACTURING TOOLS FOR IMPROVEMENT

Lean manufacturing is a set of tools that assist in the identification and elimination of waste. When waste is eliminated, quality improves, and production time and cost are reduced. Some of the tools that are utilized in lean manufacturing include: Five S, Value Stream Mapping, Kanban (pull systems), and Poka-Yoke (error-proofing).

The Five S Improvement Tool

The term *Five S* comes from the Japanese. The five Japanese words used to describe the method all begin with the letter s.

Sorting (Seiri) All unnecessary parts, tools, and instructions should be eliminated. All tools, materials, instructions, and so on in the work area and overall plant should be thoroughly examined. Only essential items should be kept. Essential tools should be easily accessible in the work station. Everything that is nonessential should be stored or discarded.

Stabilizing or Straightening Out (Seiton) This technique stresses that there should be a place for everything and everything should be in its place. There should be a place for each item in a station and it should be clearly labeled. Parts, tools, supplies, and equipment should be located close to where they are needed. Everything should be arranged to promote efficient work flow.

Cleaning or Shining (Seiso) Clean the workspace and equipment, and then keep it clean and organized. At the end of each shift, clean the work area and be sure everything is restored to its place. Spills, leaks, and other messes also then become a visual signal for equipment or process steps that need attention. Maintaining cleanliness must be an integral part of the daily work.

Standardizing (Seiketsu) All work practices should be consistent and standardized. Work stations for a particular operation should be exactly the same. Employees with the same role should be able to work in any station. The same tools should be in the same location in every station.

Sustaining the Practice (Shitsuke) The new standards that have been implemented as a result of the previous tools must be reviewed, maintained, and improved. After the first four S's have been established, they must continue to be the new way to operate. This new way must be sustained. The plant must not be allowed to slide back to the old ways of operating. Everyone must also commit to continue looking for ways to improve.

Value Stream Mapping

Value stream mapping is a lean manufacturing tool that can be used to analyze and design the flow of materials and information required to bring a product to a customer. It originated at Toyota.

Implementation Steps

1. Identify the product to be studied.
2. Draw a state value stream map based on the shop floor that reflects the current steps, waits, and information flows required to deliver the product. This can be done by drawing it on paper or with special value stream mapping software.
3. Assess the efficiency of the current value stream map in terms of improving flow by eliminating waste.
4. Develop the redesigned value stream map.
5. Implement the redesign.

Shigeo Shingo was a Japanese industrial engineer who is one of the world's leading experts on manufacturing and the TPS. Shigeo Shingo suggested that the value-adding steps be drawn horizontally across the center of the value stream map and the non–value-adding steps be drawn vertically at right angles to the value stream.

This makes it easy to distinguish the value stream from the wasted steps. Shingo calls the value stream the process and the non–value streams the operations. Shingo viewed each vertical line as the story of a person or workstation. He viewed the horizontal line as the story of the product being created. Value stream mapping is a very visual and easy to understand tool to study and improve a process.

Kanban

Kanban is not an inventory control system. It is a scheduling system that helps determine what to produce, when to produce it, and how many to produce.

In the 1940s, Toyota studied supermarkets to try and apply store and shelf-stocking techniques to the factory floor. Toyota observed that in a supermarket, customers get what they need at the needed time, and in the needed quantity. Toyota saw that a supermarket only stocks what it thinks will sell, and customers only buy as many as they need because they are not worried about getting additional products in the future. Toyota saw a process as being a customer of the processes that preceded it. The customer (worker) goes to the store to get the parts they need, and the store restocks with additional product like a store would.

Kanban uses the rate of demand to control the rate of production. Demand is passed from the end customer through the chain of customer (worker) processes. Toyota first applied this technique in 1953 in their main machine shop.

Imagine that one of the components needed to make the product is a ¼−20 bolt and it arrives in a box. There are 100 of the bolts in a box. The worker might have two boxes of bolts available. When the first box is empty, the assembler installing the bolts would take the card that was attached to the box and send it to the bolt making department. The assembler would then begin to use the bolts from the second box of bolts. The card would serve as an order for another box of 100 bolts. The box of bolts would be made and sent to the assembler. A new box of bolts would not be produced until a card is received. In this way the demand drives the rate of production.

Kanban is a pull-type production system. The number of bolts that are produced depends on the actual demand. In this example, the demand is the number of cards received by the bolt manufacturing area from the customer (assembler).

Poka-Yoke Poka-Yoke is a Japanese word that means mistake-proofing or fail-safing. A Poka-Yoke is any mechanism in a manufacturing process that helps an operator avoid mistakes. For example, the addition of a pin in a fixture to prevent the part from being mislocated would be a Poka-Yoke. Poka-Yoke attempts to eliminate product defects by preventing, correcting, or warning the operator an error is about to occur. Shigeo Shingo first used the term Poka-Yoke as part of the TPS.

A Poka-Yoke technique alerts the operator when a mistake is about to be made, or the Poka-Yoke device actually prevents the mistake from being made.

Summary of Lean Manufacturing

Toyota's belief is that *lean* is not the tools, but the actual reduction of three types of waste: non–value-adding work, overburden, and unevenness of production. The tools are used to expose problems and improve processes.

Lean manufacturing attempts to get the correct things to the correct place at the correct time in the correct quantity to achieve an efficient work flow. Lean manufacturing also attempts to minimize waste and make processes flexible and able to react to change. For lean manufacturing techniques to be effective, they must be understood and be actively supported by the employees who build the product and add the value.

SELF-TEST

1. How is lean manufacturing supposed to make a company more efficient and more profitable?
2. What was the basis of improvement in the Toyota Production System?
3. The Toyota Production System originally listed seven wastes. An eighth was added. What is it and what does it mean for you in your career?
4. What is the basis of the Five S tool?
5. How can Value Stream Mapping improve a process?
6. What is Kanban?
7. What is Poka-Yoke?

UNIT THREE

Shop Safety

OBJECTIVES

After completing this unit, you should be able to:

- Identify common shop hazards.
- Identify and use common shop safety equipment.
- Explain the classes of fires that are applicable to a machine shop.
- Given a fire extinguisher, explain what types of fires it can be used on.
- Demonstrate safe working practices in the shop.
- Explain Lockout/Tagout.

SAFETY FIRST

We generally do not think about safety until it is too late. Safety is often not thought about as you do your daily tasks. Often you expose yourself to needless risk because you have experienced no harmful effects in the past. Unsafe habits become almost automatic. You may drive your automobile without wearing a seat belt. You know this to be unsafe, but you have done it before and so far, no harm has resulted. None of us really likes to think about the possible consequences of an unsafe act. However, safety can and does have an important effect on anyone who makes his or her living in a potentially dangerous environment. An accident can end your career as a machinist. Accidents are always unexpected! You may spend several years learning the trade and more years gaining experience. Years spent in training and gaining experience can be wasted in an instant if you have an accident, not to mention a possible permanent physical handicap for you and hardship on your family. Safety is an attitude that should extend far beyond the machine shop and into every facet of your life. You must constantly think about safety in everything you do. Safe habits must be developed and utilized. Safety glasses are just one example. You must develop the habit to wear them at all times in the shop to the point that you feel naked without them.

PERSONAL SAFETY

Grinding Dust, Hazardous Fumes, and Chemicals

Grinding dust is produced by abrasive wheels and consists of extremely fine metal particles and abrasive wheel particles. These should not be inhaled. Many grinding machines have a vacuum dust collector (Figure A-6). Grinding may be done with coolants that aid in dust control. You should wear an approved respirator if you are exposed to grinding dust. Change the respirator filter at regular intervals. Grinding dust can present a great danger to health.

Some metals, such as zinc, give off toxic fumes when heated above their boiling point. When inhaled, some of these fumes cause only temporary sickness, but other fumes can be severe or even fatal. The fumes of mercury and lead are especially dangerous, as their effect is cumulative in the body and can cause irreversible damage. Cadmium and beryllium compounds are also very poisonous. Therefore, when welding, burning, or heat-treating metals, adequate ventilation is an absolute necessity. This is also true when parts are being carburized with compounds containing potassium cyanide. These cyanogen compounds are deadly poisons, and every precaution should be taken when using them. Kasenit, a trade name for a nontoxic carburizing compound, is often found in school shops and in machine shops.

There may also be chemical hazards in the machine shop; lubricating oils, cutting oils, coolants, solvents, and some types of degreasing agents may be used. Any of these chemical agents can cause both short- and long-term health problems. Cutting oils may smoke when heated and give off noxious fumes. Inhaling any type of smoke can have short- and long-term health risks. Coolants may cause contact dermatitis, a skin irritation problem, and prolonged exposure can cause other long-term health problems. You should seek chemical safety data regarding these products and determine what health problems that short- and long-term exposure can cause. Chemical hazard awareness programs label chemicals to make employees aware of particular fire, health, and reactivity issues.

Figure A-6 Dust collector installed on a grinder.

Material Safety Data Sheets (MSDS)

A material safety data sheet or MSDS contains information describing the properties of particular chemicals, materials, and other substances. Technical data are provided defining the chemical and physical properties of materials, such as melting point, boiling point, flash point, and any toxic elements that are likely to be present during processing or handling. Other items that may be included on an MSDS are proper disposal techniques, first-aid issues from exposure to hazardous materials, and protective equipment required to safely handle the material. Material safety data sheets are available from many different sources including the Occupational Safety and Health Administration (OSHA) and manufacturers' published information. MSDS sheets should be readily available in the shop. Modern industrial operations often use many hazardous materials. Safety in using, handling, and disposing of these materials has become extremely important.

Eye Protection

Eye protection is a primary safety consideration in the machine shop. Machine tools produce hot, sharp metal chips that may be ejected from a machine at high velocity. Sometimes they can fly many feet. Furthermore, most

Figure A-7 Safety glasses, face shield, and safety goggles.

cutting tools are made from hard, brittle materials. They can occasionally break or shatter from the stress applied to them during a cut. The result can be flying metal particles.

Eye protection must be worn at all times in the machine shop. Several types of eye protection are available. Plain safety glasses are all that are required in most shops (see Figure A-7). These have shatterproof lenses that may be replaced if they become scratched. The lenses have a high resistance to impact.

Side shield safety glasses should be worn around any grinding operation. The side shield protects the side of the eye from flying particles. Side shield safety glasses may be of the solid or perforated type. The perforated side shield type fits closer to the eye.

Prescription glasses may be covered with safety goggles. The full-face shield may also be used (Figure A-7). Prescription safety glasses are available. In industry, prescription safety glasses are sometimes provided free to employees.

Foot Protection

Generally, the machine shop presents a modest, but real hazard to the feet. There is always a possibility that you could drop something on your foot. Safety shoes with a steel toe shield to protect the foot are available. Some safety shoes also have an instep guard. Shoes must be worn at all times in the machine shop. A solid leather shoe is recommended. Sandals should not be worn as they expose the foot to hot chips produced in machining processes. Many shops provide rubber mats by machines that can dramatically reduce the fatigue on the feet caused by standing on a hard-concrete floor.

Ear Protection

Machine shops are noisy environments. Hearing protection should be worn. Excess noise can cause permanent hearing loss. Usually, this occurs over a period of time, depending on the intensity of the exposure. Noise is considered an industrial hazard if it is continuously above 85 **decibels (dB)**. Decibel is the unit used for measuring the relative intensity of sounds. If the noise level is over 115 dB, even for short periods of time, ear protection must be worn (Figure A-8). Several types of sound suppressors and noise-reducing earplugs may be worn. Earmuffs or earplugs should be used wherever high-intensity noise occurs. Table A-1 shows the decibel level of various sounds; sudden sharp or high-intensity noises are the most harmful to the ears.

Figure A-8 Ear plugs.

Table A-1 Decibel Level of Various Sounds

R	Sound
130	Painful, Jet engine on the ground
120	Airplane on the ground
110	Power saw at 3 feet
100	Motorcycle or snowmobile
90	Truck traffic
80	Telephone dial tone
60–70	Normal conversation at 3–5 feet
30	Whisper or a quiet library
0	Faintest audible sound

Clothing, Hair, and Jewelry

Shirts should be short-sleeved or long sleeves should be rolled up above the elbow. Shirts should be tucked in. If a shop apron is worn, it should be kept tied behind the back. If apron strings were to get entangled in the machine, you may be reeled in as well. Loose, hanging clothing is dangerous. A shop coat may be worn as long as long sleeves are rolled up above the elbow. Fuzzy sweaters should not be worn around machines. Polyester clothing is very susceptible to hot chips and should be avoided. Cotton clothing is a better choice.

If you have long hair, keep it secured properly. In industry, you may be required to keep your hair tied back so that your hair cannot become tangled in a moving machine. Long hair is easily caught up in moving tools such as drills and chucks. The result of this can be disastrous.

Remove your wristwatch and rings before operating any machine tool. These can cause serious injury if they are caught in a moving machine part.

Hand Protection

No device will totally protect your hands from injury. Next to your eyes, your hands are the most important tools you have. It is up to you to keep them out of danger. Use a brush to remove chips from a machine (Figure A-9). Do not use your hands. Chips are razor sharp and can be extremely hot. Never grab chips as they come from a cut. Long chips are extremely dangerous. Long chips can be eliminated by sharpening your cutting tools properly. Chips should not be removed with a rag. The metal particles become embedded in the cloth and they may cut you. Furthermore, the rag may be caught in a moving machine. Gloves should never be worn around moving machinery. If a glove is caught in a moving part, it will be pulled in, along with the hand inside it.

Figure A-9 Use of a brush to remove chips safely.

Do not be careless around machines. Do not put your hands near moving machinery. Do not lean on a machine that is running. Do not lean over the back of a machine and talk to the employee who is running the machine. Accidents happen very quickly and they are never expected.

Various cutting oils, coolants, and solvents may affect your skin. The result may be a rash or an infection. Avoid direct contact with these products as much as possible, and wash your hands as soon as possible after contact.

Lifting

Improper lifting (Figure A-10) can result in a permanent back injury that can limit or end your ability to earn a living and reduce your quality of life by restricting what you are able to do. Back injury can be avoided if you lift properly at all times. If you must lift a large or heavy object, get some help or use a hoist or forklift. Don't try to be a "superman" and lift something that you know is too heavy. It is not worth the risk. Most shops have lifting devices available for heavy objects. Objects within your lifting capability can be lifted safely by using the following procedure:

1. Keep your back straight.
2. Squat down, bending your knees.
3. Lift smoothly, using the muscles in your legs to do the work. Keep your back straight. Bending over the load puts an excessive stress on your spine.
4. Position the load so that it is comfortable to carry. Watch where you are walking when carrying a load.
5. If you replace the load back at floor level, lower it in the same manner in which you picked it up.

Scuffling and Horseplay

The machine shop is no place for scuffling and horseplay. This activity can result in a serious injury to you or those around you. Practical joking is also hazardous. What might appear comical to you could result in a disastrous accident to

Figure A-10 The picture on the left shows the incorrect method to lift object. The picture on the right shows the correct method for lifting objects.

someone else. In industry, horseplay and practical joking are often grounds for firing an employee. Loud noises should also be avoided in the plant. Yelling, whistling, and making loud noises can startle an individual and may cause them to injure themselves.

Injuries

If you should be injured, report it immediately to your instructor. In many schools and companies, there are incident report forms that must be filled out to report injuries to help fix the problem and help prevent it in the future.

IDENTIFYING SHOP HAZARDS

A machine shop is not so much a dangerous place as it is a potentially dangerous place. One of the best ways to be safe is to be able to identify shop hazards before they result in an accident. By being aware of potential dangers, you can be very safe as you do your work in a machine shop.

Compressed Air

Machine shops use compressed air to operate certain machine tools. Often, flexible air hoses hang throughout the shop. There is a large amount of energy stored in a compressed gas such as air. Releasing this energy presents an extreme danger. You may be tempted to blow chips from a machine tool using compressed air. This is not good practice. The air will propel metal particles at high velocity. They can injure you or someone on the other side of the shop. Use a brush to clean chips from the machine. Do not blow compressed air on your clothing or skin. The air may be dirty, and the force can implant dirt and germs into your skin. Air can be a hazard to ears as well. An eardrum can be ruptured.

Should an air hose break or the nozzle on the end come unscrewed, the hose will whip about wildly. This can result in an injury if you happen to be standing nearby. When an air hose is not in use, it is good practice to shut off the supply valve. The air trapped in the hose should be vented. When removing an air hose from its supply valve, be sure that the supply is turned off and the hose has been vented. Removing a charged air hose will result in a sudden venting of air. This can surprise you, and an accident might result.

Housekeeping

Keep the floor and aisles clear of stock and tools. This will ensure that all exits are clear if the building should have to be evacuated. Material on the floor, especially round bars, can cause falls. Clean up oils or coolants that spill on the floor. Several products designed to absorb oil are available. Keep a broom handy when you are machining if chips are flying onto the floor. Keep the floor clean around the machine. Keep oily rags in an approved safety can (Figure A-11). This will prevent possible fire from spontaneous combustion.

Fire Extinguishers

Fire extinguishers work by cooling the fuel, stopping the reaction, or by removing the oxygen from the fire. Fire extinguishers are used to spray a pressurized media out of a nozzle. The spray should not be aimed at the flames. The spray should be aimed at the fuel of the fire, which is at the bottom of the flames.

OSHA has identified five basic types of fires and specified types of extinguishers for each type of fire. Four of the five types of fire may be found in a machine shop.

Class A Class A fires involve common combustibles such as paper, cloth, wood, rubber, many types of plastics, and so on.

Figure A-11 A safety can for oily rags to prevent fires.

An extinguisher rated for a Class A fire uses pressurized water as the media. A Class A fire extinguisher should only be used on a Class A fire, never on any other type of fire.

Class B Class B fires involve oils, gasoline, lacquers, some types of paints, solvents, grease, and most other flammable liquids. Class B fire extinguishers use carbon dioxide or dry chemicals as the media.

Class C Class C fires are electrical fires. These can occur in wiring, energized electrical equipment, fuse boxes, computers, and so on. Class C fire extinguishers utilize carbon dioxide or a dry chemical as the extinguishing media.

There are fire extinguishers available that can be used for a Class A, B, or C fires. They are labeled for Class A, B, and C.

Class D Class D fires involve metals such as powders, flakes, or shavings of combustible metals such as magnesium, titanium, potassium, and sodium. These can be very dangerous fires. Class D fire extinguishers can contain a sodium chloride–based dry powder extinguishing agent, but most of the fire extinguishers labeled Class D have components that are geared to a specific metal.

Class K Class K fires involve combustible cooking fluids such as oils and fats. These are not applicable to a machine shop.

Know where all fire extinguishers are located in the plant. Know what type they are and what types of fires they can be used on. Know how they are operated. You should be fully aware of this information before they are needed. Precious minutes can be saved by knowing this ahead of time.

Carrying Objects

If material is over 6 feet long, it should be carried in the horizontal position. If it must be carried in the vertical position, be careful of electrical buses, light fixtures, and ceilings. If the material is both long and over 40 lb in weight, it should be carried by two people, one at each end.

Machine Hazards

There are many types of machine hazards. Each section of this book will discuss the specific dangers applicable to that type of machine tool. Remember that a machine cannot distinguish between cutting metal and cutting fingers. Do not think that you are strong enough to stop a machine should you become tangled in moving parts. You are not. When operating a machine, think about what you are going to do before you do it. Go over a **safety checklist**:

1. Do I know how to operate this machine?
2. What are the potential hazards involved?
3. Are all guards in place?
4. Are my procedures safe?
5. Am I doing something that I probably should not do?
6. Have I made all the proper adjustments and tightened all locking bolts and clamps?
7. Is the workpiece secured properly?
8. Do I have proper safety equipment?
9. Do I know where the stop switch is?
10. Do I think about safety in everything that I do?

INDUSTRIAL SAFETY AND FEDERAL LAW

In 1970, Congress passed the Williams-Steiger Occupational Safety and Health Act. This act took effect on April 28, 1971. The purpose and policy of the act is "to assure so far as possible every working man and woman in the Nation safe and healthful working conditions and to preserve our human resources."

The Occupational Safety and Health Act is commonly known as OSHA. Prior to its passage, industrial safety was the individual responsibility of each state. The establishment of OSHA added a degree of standardization to industrial safety throughout the nation. OSHA encourages states to assume full responsibility in administration and enforcement of federal occupational safety and health regulations.

Duties of Employers and Employees

Every employer under OSHA has the general duty to furnish places of employment free from recognized hazards causing or likely to cause death or serious physical harm. The

employer has the specific duty of complying with safety and health standards as defined under OSHA. Each employee has the duty to comply with safety and health standards and all rules and regulations established by OSHA.

Occupational Safety and Health Standards

Job safety and health standards consist of rules for avoiding hazards that have been proven by research and experience to be harmful to personal safety and health. These rules may apply to all employees, as in the case of fire protection standards. Many standards apply only to workers engaged in specific types of work. A typical rule might state that aisles and passageways shall be kept clear and in good repair, with no obstruction across or in aisles that could create a hazard.

Complaints of Violations

Any employee who believes that a violation of job safety or health standards exists may request an inspection by sending a signed written notice to OSHA. This includes anything that threatens physical harm or represents an imminent danger. A copy must also be provided to the employer; however, the name of the person complaining need not be revealed to the employer.

Enforcement of OSHA Standards

OSHA inspectors may enter a business or school at any reasonable time and conduct an inspection. They are not permitted to give prior notice of this inspection. They may question any employer, owner, operator, agent, or employee in regard to any safety violation. The employer and a representative of the employees have the right to accompany the inspector during the inspection.

If a violation is discovered, a written citation is issued to the employer. A reasonable time is permitted to correct the condition. The citation must be posted at or near the place of the violation. If after a reasonable time the condition has not been corrected, a fine may be imposed on the employer. If the employer is making an attempt to correct the unsafe condition but has exceeded the time limit, a hearing may be held to determine progress.

Penalties

Willful or repeated violations may incur monetary penalties. Citations issued for serious violations incur mandatory penalties. A serious violation may be penalized for each day of the violation.

OSHA Education and Training Programs

The Occupational Safety and Health Act provides for programs to be conducted by the Department of Labor. These programs provide for education and training of employers and employees in recognizing, avoiding, and preventing unsafe and unhealthful working conditions. The act also provides for training an adequate supply of qualified personnel to carry out OSHA's purpose.

Lockout and Tagout Procedures

On October 30, 1989, the Department of Labor released "The control of hazardous energy sources (Lockout/Tagout)" standard. It is the Lockout/Tagout standard numbered 29 CFR 1910.147. The standard was intended to reduce the number of deaths and injuries related to servicing and maintaining machines and equipment.

The Lockout/Tagout standard covers the servicing and maintenance of machines and equipment in which the unexpected startup or energization of the machines or equipment or the release of stored energy could cause injury to employees. The standard covers energy sources such as electrical, mechanical, hydraulic, chemical, nuclear, and thermal. The standard establishes minimum standards for the control of these hazards.

Normal production operations are excluded from the lockout/tagout restrictions. Normal production operation is the use of a machine or equipment to perform its intended production function. Any work performed to prepare a machine or equipment to perform its normal production operation is called setup.

If an employee is working on cord and plug-connected electrical equipment for which exposure to unexpected energization or start up of the equipment is controlled by the unplugging of the equipment from the energy source, and the plug is under the exclusive control of the employee performing the servicing or maintenance, this activity is also excluded from requirements of the standard.

The Lockout/Tagout standard defines an energy source as any source of electrical, mechanical, hydraulic, pneumatic, chemical, thermal, or other energy. Machinery or equipment is considered to be energized if it is connected to an energy source or contains residual or stored energy. Stored energy can be found in pneumatic and hydraulic systems, capacitors, springs, and even gravity. Heavy objects have stored energy. If they fall they can cause injury.

Servicing and/or maintenance includes activities such as installing, constructing, setting up, adjusting, inspecting, modifying, and maintaining and/or servicing equipment. These activities include lubrication, cleaning or unjamming of machines or equipment, and making adjustments or tool changes, where personnel may be exposed to the unexpected energization or start up of the equipment or a release of hazardous energy.

Employers are required to establish (lockout/tagout) procedures and employee training to ensure that before any employee performs any servicing or maintenance on a machine or equipment where the unexpected energizing, startup, or release of stored energy could occur and cause injury, the machine or equipment shall be isolated, and rendered inoperative. Most companies impose strong sanctions on employees who do not follow the procedures to the letter. Many

companies terminate employees who repeatedly violate the procedures. The procedures are designed to keep personnel safe. Make sure you follow procedures. You should also be aware of the penalties of not following them. The greatest penalty could be severe injury or death.

Employers are also required to conduct periodic inspections of the procedures at least annually to ensure that the procedures and the requirements of the standard are being followed.

Only authorized employees may lock out machines or equipment. An authorized employee is defined as being one who has been trained and has been given the authority to lock or tag out machines or equipment to perform servicing or maintenance on that machine or equipment.

An energy-isolating device is defined as being a mechanical device that can physically prevent the transmission or release of energy. A disconnect on an electrical enclosure is a good example of an energy-isolating device (see Figure A-12). If the disconnect is off, it physically prevents the transmission of electrical energy. Energy-isolating devices include the following: disconnects, manual electrical circuit breakers, manually operated switches by which the conductors of a circuit can be disconnected from all ungrounded supply conductors and, in addition, no pole can be operated independently; line valves (see Figure A-12); locks and any similar device used to block or isolate energy. Push buttons, selector switches and other control circuit type devices are not considered to be energy-isolating devices.

Due to the invisible and potentially fatal hazard of electrical energy, manufacturing companies developed **lockout/tagout** procedures for safely working on electrical equipment. When working on electrical equipment, it is of critical importance to absolutely prevent the accidental energizing of an electrical circuit. In lockout/tagout procedures, the source of the power is turned off and the control switches, circuit breakers, or main switches are physically locked out, often using a keyed lock. The circuit is also tagged and signed off by the electrician or other maintenance workers and can be unlocked and reenergized only by the person directly responsible for the lockout/tagout procedure. Make sure that you follow lockout/tagout procedures. Failure to follow them may cause serious injury or death.

Figure A-12 Electrical disconnect on the left and a pneumatic disconnect on the right (*Courtesy of Fox Valley Technical College*).

Most companies also have severe penalties up to and including, being fired for not following lockout/tagout procedures.

SELF-TEST

1. What is the primary piece of safety equipment in the machine shop?
2. What are side shields and what are they designed to do?
3. Describe proper dress for the machine shop.
4. What can be done to control grinding dust?
5. What hazards exist from coolants, oils, and solvents?
6. Describe proper lifting procedure.
7. Describe at least two compressed-air hazards.
8. Describe good housekeeping procedures.
9. How should long pieces of material be carried?
10. List at least five points from the safety checklist for a machine tool.
11. Describe the purpose of lockout/tagout procedures.

INTERNET REFERENCES

Information on industrial safety, lockout/tagout and safety equipment:

http://www.osha.gov

http://www.seton.com

UNIT FOUR

Threads and Fasteners

Many precision-machined products are useless until assembled into a machine, tool, or other mechanism. This assembly requires many types of fasteners and other mechanical hardware. In this unit you will be introduced to many of these important hardware items.

OBJECTIVES

After completing this unit, you should be able to:

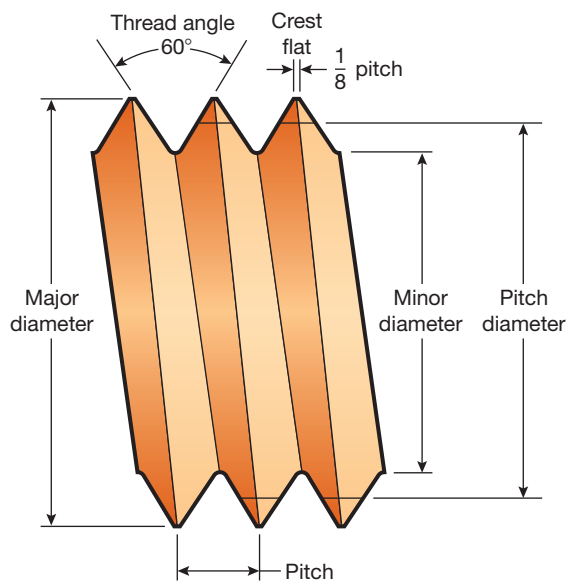
- Identify threads and threaded fasteners.
- Identify thread nomenclature on drawings.
- Discuss standard series of threads.
- Identify and describe applications of common mechanical hardware found in the machine shop.

THREADS

This unit will introduce types of fasteners and their proper use. Threads are mainly used as fastening devices. Threads are also used for adjustment, measurement, and the transmission of power. The thread is an extremely important mechanical device. It derives its usefulness from the principle of the inclined plane, one of the six simple machines. Almost every mechanical device is assembled with threaded fasteners. One of the fundamental tasks of a machinist is to produce external and internal threads using machine tools and hand tools.

THREAD FORMS

There are a number of thread forms. In later units you will examine these in detail, and you will have the opportunity to make several of them on a machine tool. The most common is the unified thread form (Figure A-13). The unified thread form is an outgrowth of the American National Standard form. It was developed to help standardize manufacturing in the United States, Canada, and Great Britain.



UNC	National Coarse
UNF	National Fine
UNS	National Special

Figure A-13 Unified thread form.

In 1948, representatives from America, Canada, and Great Britain agreed on a common standard for threads. It utilized the best of both standards and was called the Unified Screw Thread System.

America has not adopted the metric system. Great Britain adopted the metric system in 1965. The International Standards Organization (ISO) adopted an international screw thread standard in 1969. The ISO metric standard is the most widely used in the world. Unified threads, a combination of the American National and the British Standard Whitworth forms, are divided into the following series: Unified National Course (UNC); Unified National Fine (UNF); and Unified National Special (UNS).

IDENTIFYING THREADED FASTENERS

Unified coarse and *unified fine* refer to the number of threads per inch of length on standard threaded fasteners. A specific diameter of bolt or nut will have a specific number of threads per inch of length. For example, a $\frac{1}{2}$ − inch − 13 Unified National Coarse bolt will have 13 threads per inch of length. This bolt is identified by the following marking:

$$\tfrac{1}{2} \text{ in.} - 13 \text{ UNC}$$

The $\frac{1}{2}$ inch is the **major diameter**, and 13 is the number of threads per inch of length. A $\frac{1}{2}$ inch diameter Unified National Fine bolt will be identified by the following marking:

$$\tfrac{1}{2} \text{ in.} - 20 \text{ UNF}$$

The $\frac{1}{2}$ inch is the major diameter and 20 is the number of threads per inch.

The Unified National Special threads are identified in the same manner. A $\frac{1}{2}$ inch diameter UNS bolt may have 12, 14, or 18 threads per inch. These are less common than the standard UNC and UNF. There are many other series of threads used for different applications. Information and data on these can be found in machinists' handbooks. You might wonder why there needs to be a UNC and a UNF series. A fine thread is stronger in harder materials like steels and a coarse thread is stronger in softer materials like aluminum. The choice of coarse versus fine series also has to do with the application. For example, an adjusting screw may require a fine thread, while a common bolt may require only a coarse thread.

Thread Terminology

Some of the more commonly used thread terms are:

Thread Angle—The angle of the thread is the included angle between the sides of the thread (see Figure A-13). For example, the thread angle for Unified Screw Thread forms is 60 degrees.

Major Diameter—Commonly known as the outside diameter (see Figure A-13). On a straight screw thread, the major diameter is the largest diameter of the thread on the screw or nut.

Minor Diameter—Commonly known as the root diameter (see Figure A-13). On a straight screw thread, the minor diameter is the smallest diameter of the thread on the screw or nut.

Number of Threads—Refers to the number of threads per inch of length.

Pitch—Is the distance from a given point on one thread to a corresponding point on the next thread.

Lead—Lead and pitch are closely related concepts. Lead is how far a thread advances by one complete rotation of the screw (360 degrees). For example, if we turn a bolt one revolution clockwise, the distance it would move into the nut would be the lead of the thread. If a bolt has 10 threads per inch, the lead would be equal to 1 inch/10 or .100 inch. One complete revolution of the bolt would advance the bolt into the nut .100 inch. Lead and pitch are normally the same for a thread. Lead and pitch would be different if the threads are not a single thread. For example, the lead on a double thread would be twice as much as the pitch.

CLASSES OF THREAD FITS

Some applications can tolerate loose threads, while other applications require tight threads. For example, the head of your car's engine is held down by a threaded fastener called a stud bolt, or simply a stud. A stud is threaded on both ends. One end is threaded into the engine block. The other end receives a nut that bears against the cylinder head. When the head is removed, it is desirable to have the stud remain screwed into the engine block. This end requires a tighter thread fit than the end of the stud accepting the nut. If the fit on the nut end is too tight, the stud may unscrew as the nut is removed.

Unified thread fits are classified as 1A, 2A, 3A, or 1B, 2B, 3B. The A symbol indicates an external thread. The B symbol indicates an internal thread. This notation is added to the thread size and number of threads per inch. Let us consider the $\frac{1}{2}$-inch diameter bolt discussed previously. The complete notation reads:

$$\tfrac{1}{2} - 13 \text{ UNC } 2\text{A}$$

On this particular bolt, the class of fit is 2. The symbol A indicates an external thread. If the notation had read:

$$\tfrac{1}{2} - 13 \text{ UNC } 3\text{B}$$

it would indicate an internal thread with a class 3 fit. This could be a nut or a hole threaded with a tap. Taps are a common tool for producing an internal thread.

Classes 1A and 1B have the greatest manufacturing tolerance. They are used where ease of assembly is desired and a loose thread is acceptable. Class 2 fits are used on the largest percentage of threaded fasteners. Class 3 fits will be tight when assembled. Each class of fit has a specific tolerance on major diameter and pitch diameter. These specifications can be found in machinists' handbooks and are required for the manufacture of threaded fasteners.

STANDARD SERIES OF THREADED FASTENERS

Threaded fasteners range from very small machine screws through very large bolts. Fasteners under $\frac{1}{4}$-inch diameter are given a number. Common UNC and UNF series threaded fasteners are listed in Table A-2. Above size 12, the major diameter is expressed in fractional form. Both series continue up to about 4 inches.

Metric Threads

With all of the importation of foreign products, metric threads have become much more common.

Table A-2 UNC and UNF Threaded Fasteners

UNC			UNF		
Size	Major Diameter (inches)	Threads/ Inch	Size	Major Diameter (inches)	Threads/ Inch
			0	.059	80
1	.072	64	1	.072	72
2	.085	56	2	.085	64
3	.098	48	3	.098	56
4	.111	40	4	.111	48
5	.124	40	5	.124	44
6	.137	32	6	.137	40
8	.163	32	8	.163	36
10	.189	24	10	.189	32
12	.215	24	12	.215	28
$\frac{1}{4}$ inch	.248	20	$\frac{1}{4}$ inch	.249	28
$\frac{5}{16}$ inch	.311	18	$\frac{5}{16}$ inch	.311	24
$\frac{3}{8}$ inch	.373	16	$\frac{3}{8}$ inch	.373	24
$\frac{7}{16}$ inch	.436	14	$\frac{7}{16}$ inch	.436	20
$\frac{1}{2}$ inch	.498	13	$\frac{1}{2}$ inch	.498	20
$\frac{9}{16}$ inch	.560	12	$\frac{9}{16}$ inch	.561	18
$\frac{5}{8}$ inch	.623	11	$\frac{5}{8}$ inch	.623	18
$\frac{3}{4}$ inch	.748	10	$\frac{3}{4}$ inch	.748	16
$\frac{7}{8}$ inch	.873	9	$\frac{7}{8}$ inch	.873	14
1 inch	.998	8	1 inch	.998	12

The metric thread form is similar to the unified and based on an equilateral triangle. The root may be rounded and the depth somewhat greater. An attempt has been made through international efforts (ISO) to standardize metric threads. The ISO metric thread series now has 25 thread sizes with major diameters ranging from 1.6 to 100 mm.

Metric thread notation takes the following form:

$$M\ 10 \times 1.5$$

M is the major diameter and 1.5 is the thread pitch in millimeters. This thread would have a major diameter of 10 mm and a pitch (or lead) of 1.5 mm. ISO metric thread major diameters and respective pitches are shown in Table A-3.

ISO METRIC THREADS CLASSES OF FIT

Numbers and letters after the pitch number in the notation specify the tolerance for both the pitch and the major diameter of external threads and the pitch and the minor diameter of internal threads. The numbers indicate the amount of tolerance allowed. The smaller the number, the smaller the amount of tolerance allowed. The letters indicate the position of the thread tolerance in relation to its basic diameter. Lowercase letters are used for external threads, with the letters e indicating a large allowance, g a small allowance, and h no allowance. Conversely, uppercase letters are used for internal threads, with a G used to indicate a small allowance and an H used to indicate no allowance.

Table A-3 ISO Metric Threads

Diameter (mm)	Pitch (mm)	Diameter (mm)	Pitch (mm)
1.6	.35	20.0	2.5
2.0	.40	24.0	3.0
2.5	.45	30.0	3.5
3.0	.50	36.0	4.0
3.5	.60	42.0	4.5
4.0	.70	48.0	5.0
5.0	.80	56.0	5.5
6.3	1.00	64.0	6.0
8.0	1.25	72.0	6.0
10.0	1.50	80.0	6.0
12.0	1.75	90.0	6.0
14.0	2.00	100.0	6.0
16.0	2.00		

The tolerance classes 6H/6g are used for general-purpose applications. They are comparable to the Unified National 2A/2B classes of fit. The designation 4H5H/4h5h is approximately equal to the Unified National class 3A/3B.

Taper Pipe Threads

Standard taper pipe threads (NPT) are unlike standard threads in the way they are cut and in their use. Taper pipe threads are one of the more challenging thread-cutting operations. Cutting taper pipe threads requires greater accuracy than standard threads. Pipe threads are designed to mechanically seal a threaded joint for pressure and to prevent leakage. To create the mechanical seal, 100 percent of the thread height must be cut by the tap to maintain the standard thread profile. The NPT standard limits for both external and internal are identical, so that hand-tight engagement can result in thread flank contact only or crest and root contact only. Wrench tightening normally produces an assembly with crest and root clearances. Excessive tightening with a wrench without the use of some kind of tape or chemical sealer is not advisable.

Taper threads are also unlike standard threads in their designations. A machinist looking at a $\frac{1}{4}$–18 NPT would immediately notice that the outside diameter of the tap is much bigger than $\frac{1}{4}$ of an inch. This is due to the fact that the tap size designation refers to the inside diameter of the standard iron pipe it is designed to fit. Another major difference between taper pipe tapping and standard thread tapping is the way in which the machinist must control the thread diameter. Since the tap is tapered, the machinist can control the thread diameter by adjusting the depth that the tap threads

into the hole. To get the proper thread depth, a machinist drives the tap into the work 12 full turns. More turns may result in the tapped hole being too large. Less turns may result in the tapped hole being too small. There is no one particular class of fit for NPT threads, such as 3A or 3B. Proper taper thread gauging is the only acceptable method for determining proper depth for taper pipe tapping. One method of monitoring tap depth is to wrap a piece of wire or tape around the tap as a line to serve as a depth indicator. This practice works well to get you "close." This trick works well for hand tapping when cutting fluids are used.

COMMON EXTERNALLY THREADED FASTENERS

Common mechanical hardware includes threaded fasteners such as bolts, screws, nuts, and thread inserts. All these are used in a variety of ways to hold parts and assemblies together.

Bolts and Screws

A general definition of a *bolt* is an externally threaded fastener that is inserted through holes in an assembly. A bolt is tightened with a nut (Figure A-14, right). A *screw* is an externally threaded fastener that is inserted into a threaded hole and tightened or released by turning the head (Figure A-14, left). From these definitions, it is apparent that a bolt can become a screw or vice versa. This depends on how it is used. Bolts and screws are the most common of the threaded fasteners. Threaded fasteners are used to assemble parts quickly, and they also make disassembly possible.

The strength of an assembly of parts depends to a large extent on the diameter of the screws or bolts used. In the case of screws, strength depends on the amount of thread engagement. *Thread engagement* is the distance that a screw extends into a threaded hole. The minimum thread engagement should be a distance equal to the diameter of the screw used; preferably, it should be a minimum of $1\frac{1}{2}$ times the screw diameter. Should an assembly fail, it is better to have the screw break than to have the internal thread stripped from the hole. It is generally easier to remove a broken screw than to drill and tap for a larger screw size. With a screw engagement of $1\frac{1}{2}$ times its diameter, the screw will usually break rather than strip the threads in the hole.

Machine bolts are made with hexagonal or square heads. These bolts are often used in the assembly of parts that do not require a precision bolt. The body diameter of machine bolts is usually slightly larger than the nominal or standard size of the bolt. Body diameter is the diameter of the unthreaded portion of a bolt below the head. A hole that is to accept a common bolt must be slightly larger than the body diameter. Common bolts are made with a class 2A unified thread and come in both UNC and UNF series. Hexagonal head machine bolt sizes range from $\frac{1}{4}$-inch diameter to 4 inches. The size of the head on a square-head machine bolt is typically $1\frac{1}{2}$ times the thread diameter.

Stud bolts (Figure A-15) have threads on both ends. Stud bolts are used where one end is semipermanently screwed into a threaded hole. A good example of the use of stud bolts is in an automobile engine. The stud bolts are tightly held in the cylinder block, and easily changed nuts hold the cylinder heads in place. The end of the stud bolt screwed into the tapped hole has a class 3A thread, while the nut end is a class 2A thread. Studs are also widely used to hold vises and or parts down on machine tables.

Machine screws are made with either coarse or fine thread and are used for general assembly work. Machine screws are available in many sizes and lengths (Figure A-16). Several head styles are also available (Figure A-17). Machine screw sizes fall into two categories. Fraction sizes range from diameters of $\frac{1}{5}$ to $\frac{3}{4}$ inch. Screws that are under $\frac{1}{4}$-inch diameter are identified by numbers from 0 to 12. A No. 0 machine screw has a diameter of .060 inch (60 thousandths of an inch). For each number above zero, add .013 in. to the diameter.

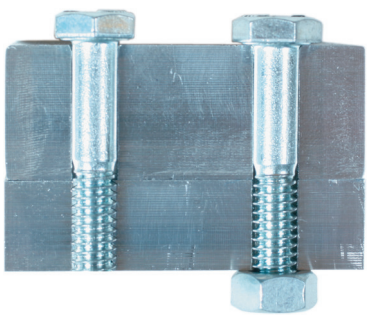


Figure A-14 On the left a fastener is used as a screw and on the right the screw is used as a bolt because it is used with a nut.

Figure A-15 Stud bolts, T-nuts, and a clamping nut (*Courtesy of Fox Valley Technical College*).

Figure A-16 Machine screw head styles.

Figure A-17 Various cap screws. From the left: a stainless-steel socket-head cap screw, a regular socket-head cap screw, a button-head cap screw, and a flat-head cap screw (*Courtesy of Fox Valley Technical College*).

EXAMPLE

Find the diameter of a No. 6 machine screw:

$$\text{No. 0 diameter} = .060 \text{ in.}$$

$$\text{No. 6 diameter} = .060 \text{ in.} + (6 \times .013 \text{ in.})$$

$$= .060 \text{ in.} + .078 \text{ in.}$$

$$= .138 \text{ in.}$$

Machine screws 2 inches long or less typically have threads extending all the way to the head. Longer machine screws have a $1\frac{3}{4}$-inch thread length.

Machine screws (Figure A-16) are made with a variety of different head shapes and are used where precision bolts or screws are needed. Machine screws are made with coarse, fine, or special threads. Machine screws with a 1-inch diameter have a class 3A thread. Those with greater than a 1-inch diameter have a class 2A thread. The strength of screws depends mainly on the kind of material used to make the screw. Materials that screws are made from include aluminum, brass, bronze, low-carbon steel, medium-carbon steel, alloy steel, stainless steel, titanium, and various plastics.

Socket Head Cap Screws

Socket head screws are made with a variety of different head shapes and are used where precision bolts or screws are needed. Socket head screws are manufactured to close tolerances. Cap screws can have round heads, low heads, flat heads, button head, or can be in a shoulder screw configuration (see Figure A-17).

Socket head cap screws are the strongest of the head styles. The height of the head is equal to the shank diameter.

Low Head Cap Screw

These are designed for applications in which head height is limited. These are not as strong as the regular head. The head height is approximately half the shank diameter.

Flat Head Cap Screw

These are designed for flush mount applications. Note that the inch and metric screws have different countersink angles. You must use the correct countersink or the screw may fail.

Button Head Cap Screw

These are well suited to holding thin materials because of the large head. They are good for tamper proofing because of the hex drive style makes it less susceptible to tampering with a screwdriver. Also good for applications where a regular screw (slotted or Phillips) is prone to stripping.

Socket Shoulder Screw

These are typically used as pivot points or axles for other parts to rotate around. The shoulders are ground to tight tolerances.

Bolt Grades and Torque Factors

In some instances, bolts need to be fastened with the correct amount of pressure (torque). In these instances, the manufacturer of the product will specify a certain torque be applied to a particular fastener. Insufficient torque will usually result in parts working loose. Overtightening, on the other hand, can cause stress or warping which also might disturb alignment on assemblies. The "armstrong" (overtightening) method of tightening fasteners can also cause broken castings, broken bolts, or stretching of the fastener.

Steel has excellent elasticity. Elasticity is the ability to stretch slightly, like a spring, and then return to its original shape. Any fastener must reach its limits of stretching to exert clamping force. But also, like a spring, an overstretched fastener takes on a set, loses its elasticity, and cannot snap back to its original shape. Proper torqueing will prevent this condition.

A popping or snapping sound is sometimes heard during the final tightening of a fastener. This popping sound indicates that the fastener is undergoing set. When a new fastener is being used and the popping occurs, the remedy is to back it off and retighten to the proper torque specifications. When an old fastener is being used and you hear popping, take the fastener out, and clean the bolt and the internal threads out completely. The safer, more economical thing to do is replace the old fastener with a new one.

SCREW STRENGTH GRADES

The selection of the right grade of fastener is just as important as proper torqueing. The strength grades of bolts are identified by the markings on the heads. The grade indicates the strength of the fastener. Figure A-18 shows a few examples of grade markings. Use a manufacturer's chart as a guide for the proper torque of fasteners.

Metric Strength Property Classes

The correct terminology for the grade of a metric fastener is property class. The metric system designates strength capabilities via property classes rather than grades. The numbering system is simple. Figure A-19 shows an example of the strength class marking on a metric bolt. The number that appears before the decimal, when multiplied by 100, will provide the approximate minimum tensile strength of the bolt. In this example, the 8 in 8.8 multiplied by 100 tells the user that this bolt has an approximate minimum tensile strength of 800 MPa (Mega (million) Pascals). The

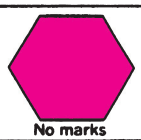
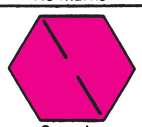
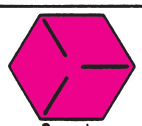
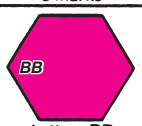
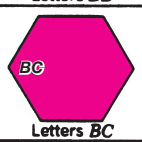
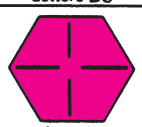
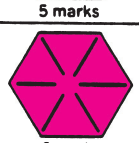
Bolt head marking	SAE – Society of Automotive Engineers ASTM – American Society for Testing and Materials ASTM Definitions	Material	Minimum tensile strength in pounds per square inch (PSI)
 No marks	SAE grade 1 SAE grade 2 Indeterminate quality	Low carbon steel Low carbon steel	65,000 PSI
 2 marks	SAE grade 3	Medium carbon steel, cold worked	110,000 PSI
 3 marks	SAE grade 5 ASTM – A 325 Common commercial quality	Medium carbon steel, quenched and tempered	120,000 PSI
 Letters *BB*	ASTM – A 354	Low alloy steel or medium carbon steel, quenched and tempered	105,000 PSI
 Letters *BC*	ASTM – A 354	Low alloy steel or medium carbon steel, quenched and tempered	125,000 PSI
 4 marks	SAE grade 6 Better commercial quality	Medium carbon steel, quenched and tempered	140,000 PSI
 5 marks	SAE grade 7	Medium carbon alloy steel, quenched and tempered, roll threaded after heat treatment	133,000 PSI
 6 marks	SAE grade 8 ASTM – A 345 Best commercial quality	Medium carbon alloy steel, quenched and tempered	150,000 PSI

Figure A-18 Fastener strength grade chart.

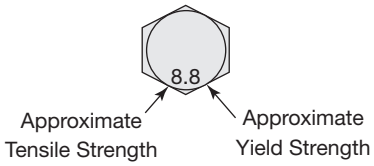


Figure A-19 Strength class marking for metric bolts (*Courtesy of Fox Valley Technical College*).

Pascal (symbol: **Pa**) is the unit of pressure, stress, and tensile strength, named after the French physicist, mathematician, inventor, and philosopher Blaise Pascal. It is a measure of force per unit area, defined as one newton per square meter.

The number which appears after the decimal, when multiplied by 10, will provide the approximate yield strength percentage in relation to the minimum tensile strength. For the 8.8 bolt, the 8 after the decimal point tells the user that the yield strength of the bolt is approximately 80 percent of the first number: 800 MPa. Thus, the 8.8 bolt has a yield strength of approximately 640 MPa.

There are nine common metric property classes: 4.6, 4.8, 5.8, 8.8, 9.8, 10.9, and 12.9. In the United States, property classes 8.8.3 and 10.9.3 also defined. The 3 after the second decimal point indicates the fastener is made of weather resistant steel. Hex head bolts M5 and larger have the property class marked on the head of the bolt. Fasteners smaller than M5, and those with slotted or recessed heads do not have to be marked.

Setscrews are used to lock pulleys or collars on shafts. Setscrews can have square heads with the head extending above the surface; more often, the setscrews are slotted or have socket heads. The heads of slotted or socket head setscrews are usually set below the surface of the part to be fastened. A pulley or collar with the setscrews below the surface is much safer for persons working around it. Socket head setscrews may have hex socket heads or spline socket heads. Setscrews are manufactured in number sizes from 0 to 10 and in fractional sizes from $\frac{1}{4}$ to 2 inches. Setscrews are usually made from carbon or alloy steel and hardened.

Setscrews can have various point configurations (Figure A-20). The flat point setscrew will make the least amount of indentation on a shaft and is used where frequent adjustments are made. A flat point setscrew is also used to provide a jam screw action when a second setscrew is tightened on another setscrew to prevent its release through vibration. The oval point setscrew will make a slight indentation as compared with the cone point. With a half-dog or full-dog point setscrew holding a collar to a shaft, alignment between shaft and collar will be maintained even when the parts are disassembled and reassembled. This is because the shaft is drilled with a hole of the same diameter as the dog point. Cup-pointed setscrews will make a ring-shaped depression in the shaft and will give a slip-resistant connection. Square head setscrews have a class 2A thread and are usually supplied with a coarse thread. Slotted and socket head setscrews have a class 3A UNC or UNF thread.

Thread-forming screws form their own threads and eliminate the need for tapping. These screws are used in the assembly of sheet metal parts, plastics, and nonferrous material. Thread-forming screws form threads by displacing material with no cutting action. These screws require an existing hole of the correct size.

Thread-cutting screws make threads by cutting and producing chips. Because of the cutting action these screws need less driving torque than thread-forming screws. Applications are similar to those for thread-forming screws. These include fastening sheet metal, aluminum, brass, die castings, and plastics.

COMMON INTERNALLY THREADED FASTENERS

Nuts

Common nuts (Figure A-21) are manufactured in as many sizes as bolts are. Most nuts are either hex (hexagonal) in shape. Nuts are identified by the size of the bolt they fit, not by their outside size. Common hex nuts are made in different thicknesses. A thin hex nut is called a *jam nut*. It is used where space is limited or where the strength of a regular nut is not required. Jam nuts are often used to lock other nuts. Regular hex nuts are slightly thinner than their size designation. A $\frac{1}{2}$-inch regular hex nut is $\frac{7}{16}$-inch thick. A $\frac{1}{2}$-inch heavy hex nut is $\frac{31}{64}$-inch thick. A $\frac{1}{2}$-inch-high hex nut measures $\frac{11}{16}$ inch in thickness. Other common nuts include various stop nuts or locknuts. Two common types are the elastic stop nut

Figure A-20 Setscrew points.

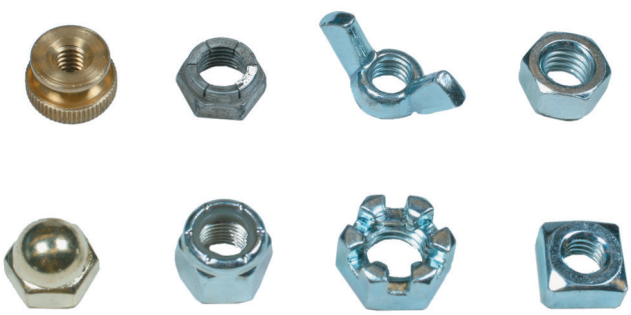


Figure A-21 Common nuts.

and the compression stop nut. They are used in applications where the nut might vibrate off the bolt. Wing nuts and thumb nuts are used where quick assembly or disassembly by hand is desired. Other hex nuts are slotted or castle nuts. These nuts have slots cut into them. When the slots are aligned with holes in a bolt, a cotter pin may be used to prevent the nut from turning. Axles and spindles on vehicles have slotted nuts with cotter pins to prevent wheel bearing adjustments from slipping.

Cap or acorn nuts are often used where decorative nuts are needed. These nuts also protect projecting threads from accidental damage. Nuts are made from many different materials, depending on their application and strength requirements.

INTERNAL THREAD INSERTS

Thread inserts can be used when an internal thread is damaged or stripped and it is not possible to drill and tap for a larger size. A thread insert retains the original thread size; however, it is necessary to drill and tap a somewhat larger hole to accept the thread insert.

One common type of internal thread insert is the key–lock type. The thread insert has both external and internal threads. This type of thread insert is screwed into a hole tapped to the same size as the thread on the outside of the insert. The four keys are driven in using a special driver (Figure A-22). This holds the insert in place. The internal thread in the insert is the same as the original hole.

A second type of thread insert is also used in repair applications as well as in new installations. Threaded holes are often required in products made from soft metals such as aluminum. If bolts, screws, or studs were to be screwed directly into the softer material, excessive wear could result, especially if the bolt was taken in and out a number of times. To overcome this problem, a thread insert made from a more durable material may be used. Stainless steel inserts are frequently used in aluminum (Figure A-23). This type of thread insert requires an insert tap, an insert driver, and a thread insert (Figure A-24). After the hole for the thread insert is tapped, the insert driver is used to screw the insert into the hole (Figure A-25). The end of the insert coil must be broken off and removed after the insert is screwed into place. The insert in the illustration is used to repair spark plug threads in engine blocks.

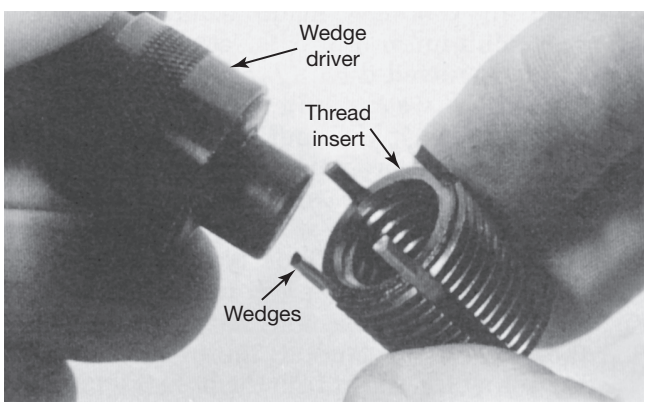


Figure A-22 Key–lock internal thread insert.

Figure A-23 Stainless steel thread insert used in an aluminum valve housing.

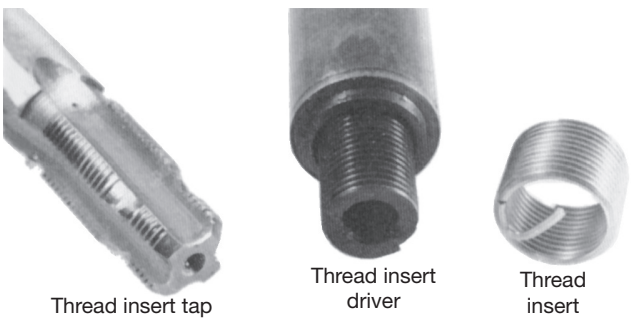


Figure A-24 Thread insert tap, driver, and repair insert for spark plug hole repair.

Figure A-25 Thread insert driver.

WASHERS, PINS, RETAINING RINGS, AND KEYS

Washers

Flat washers (Figure A-26) are used under nuts and bolt heads to distribute the pressure over a larger area. Washers also prevent the marring of a finished surface when nuts or screws are tightened. Washers can be manufactured from many different materials. The nominal size of a washer is intended to be used with the same nominal-size bolt or screw. Standard series of washers are narrow, regular, and wide. For example, the outside diameter of a $\frac{1}{4}$-inch narrow washer is $\frac{1}{2}$ inch the outside diameter of a $\frac{1}{4}$-inch regular washer is almost $\frac{3}{4}$ inch and the diameter of a wide $\frac{1}{4}$-inch washer measures 1 inch.

Lock washers (Figure A-27) are manufactured in many styles. The helical spring lock washer provides hardened bearing surfaces between a nut or bolt head and the components of an assembly. The spring-type construction of this lock washer will hold the tension between a nut and bolt assembly even if a small amount of looseness should develop. Helical spring lock washers are manufactured in series: light, regular, heavy, extra duty, and hi-collar. The hi-collar lock washer has an outside diameter equal to the same nominal-size socket head cap screw. This makes the use of these lock washers in a counterbored bolt hole possible. Counterbored holes have the end enlarged to accept the bolt head. A variety of standard tooth lock washers are produced, the external type providing the greatest amount of friction or locking effect between fastener and assembly. For use with small head screws and where a smooth appearance is desired, an internal tooth lock washer is used. When a large bearing area is desired or where the assembly holes are oversized, an internal–external tooth lock washer is available. A countersunk tooth lock washer is used for a locking action with flat head screws.

Pins

Pins (Figure A-28) find many applications in the assembly of parts. **Dowel pins** are heat-treated and precision ground. Their diameter varies from the nominal dimension by only plus or minus .0001 inch (1/10,000th of an inch). Dowel pins are used where accurate alignments must be maintained between two or more parts. Holes for dowel pins are reamed to provide a slight press fit. Reaming is a machining process during which a drilled hole is slightly enlarged to provide a smooth finish and accurate diameter. Dowel pins only

Figure A-26 A few examples of flat washers (*Courtesy of Fox Valley Technical College*).

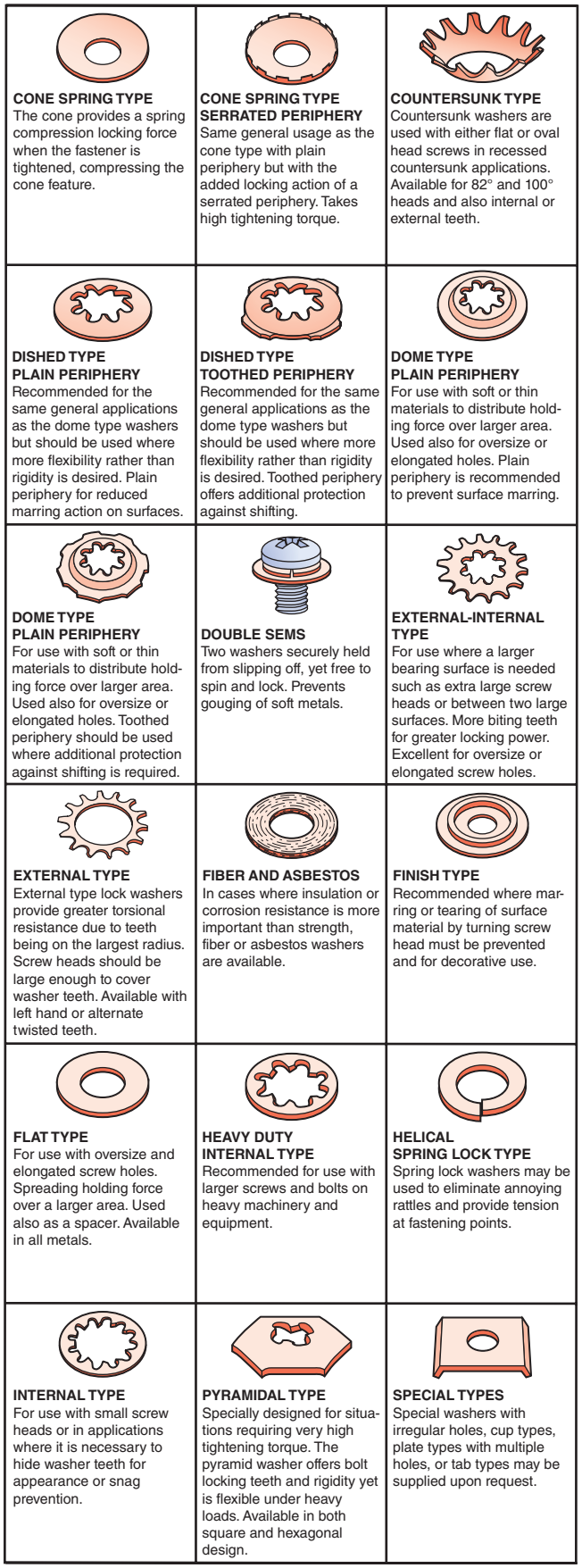


Figure A-27 Lock washers.

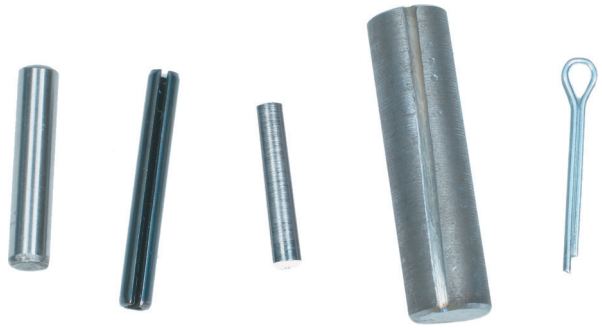


Figure A-28 Pins.

locate. Clamping pressure is supplied by the screws. Dowel pins may be driven into a blind hole. A blind hole is closed at one end. When this kind of hole is used, provision must be made to let the air that is displaced by the pin escape. This can be done by drilling a small through hole or by grinding a narrow flat the full length of the pin.

One disadvantage of dowel pins is that they tend to enlarge the hole in an unhardened workpiece if they are driven in and out several times. When parts are intended to be disassembled frequently, **taper pins** can provide accurate alignment. Taper pins have a taper (diminishing diameter) of $\frac{1}{4}$ inch per foot of length and are fitted into reamed taper holes. If a taper pin hole wears larger because of frequent disassembly, the hole can be reamed larger to receive the next larger size of taper pin. Diameters of taper pins range in size from $\frac{1}{16}$ inch to $\frac{11}{16}$ inch measured at the large end. Taper pins are identified by a number from $\frac{7}{0}$ (small diameter) to number 10 (large diameter) as well as by their length. The large end diameter is constant for a given size pin, but the small diameter changes with the length of the pin. Some taper pins have a threaded portion on the large end. A nut can be threaded on the pin and used to pull the pin from the hole much like a screw jack. This facilitates removal of the pin.

A **grooved pin** is either a cylindrical or a tapered pin with longitudinal grooves pressed into the pin body. This causes the pin to deform. A grooved pin will hold securely in a drilled hole even after repeated removal.

Roll pins can also be used in drilled holes with no reaming required. These pins are manufactured from flat steel bands and rolled into cylindrical shape. Roll pins, because of their spring action, will stay tight in a hole even after repeated disassembly.

Cotter pins are used to retain parts on a shaft or to lock a nut or bolt as a safety precaution. Cotter pins make quick assembly and disassembly possible.

Retaining Rings

Retaining rings are fasteners that are used in many assemblies. Retaining rings can be easily installed in machined grooves, internally in housings, or externally on shafts or pins (Figure A-29). Some types of retaining rings do not require grooves but have a self-locking spring-type action. The most common application of a retaining ring is to provide a shoulder to hold and retain a bearing or any other part on an otherwise smooth shaft. They may also be used in a bearing housing (Figure A-30). Special pliers are used to install and remove retaining rings.

Keys

Keys (Figure A-31) are used to prevent the rotation of gears or pulleys on a shaft. Keys are fitted into key seats in both the shaft and the external part. Keys should fit the key seats

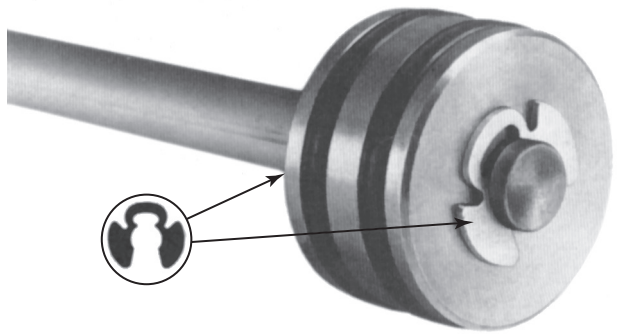


Figure A-29 External retaining ring used on a shaft (Rotor Clip Company, Inc.).

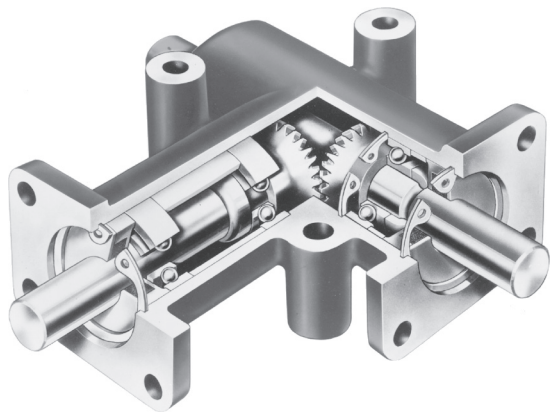


Figure A-30 Internal retaining rings used to retain bearings (Rotor Clip Company, Inc.).

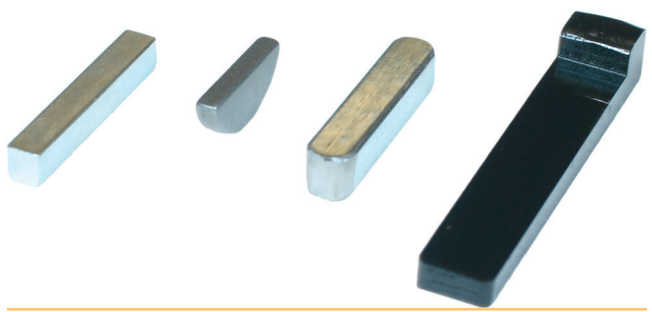


Figure A-31 Keys. A square key, a Woodruff key, a rectangular key, and a gib-head taper key.

rather snugly. **Square keys**, where the width and the height are equal, are preferred on shaft sizes up to a $6\frac{1}{2}$-inch diameter. Above a $6\frac{1}{2}$-inch diameter, rectangular keys are recommended. Woodruff keys, which almost form a half circle, are used where relatively light loads are transmitted. One advantage of **Woodruff keys** is that they cannot change their axial location on a shaft because they are retained in a pocket. A key fitted into an end-milled pocket will also retain its axial position on the shaft. Most of these keys are held under tension with one or more setscrews threaded through the hub of the pulley or gear. Where extremely heavy shock loads or high torques are encountered, a taper key is used. **Taper keys** have a taper (diminishing diameter) of $\frac{1}{8}$ inch per foot. When a tapered key is used, the key seat in the shaft is parallel to the shaft axis, and a taper to match the key is in the hub. Where only one side of an assembly is accessible, a gib-head taper key is used instead of a plain taper key. When a gib-head taper key is driven into the key seat as far as possible, a gap remains between the gib and the hub of the pulley or gear. The key is removed for disassembly by driving a wedge into the gap to push the key out.

SELF-TEST

1. Define the term pitch diameter.
2. Name two ways to measure a thread.
3. What is the rule of thumb for the length of internal thread to maximize the strength of the bolt? (In other words, that the bolt will break before the threads pull out.)
4. Describe when class two fits are used.
5. Describe UNC and UNF.
6. What is the formula for calculating the OD of a machine screw? What are setscrews used for?
7. When are stud bolts used?
8. Describe the strength classification system and marking for bolts.
9. Explain two reasons why flat washers are used.
10. What is the purpose of a helical spring lock washer?
11. When is an internal–external tooth lock washer used?
12. When are dowel pins used?
13. When are taper pins used?
14. When are roll pins used?
15. What are retaining rings?
16. What is the purpose of a key?
17. When is a Woodruff key used?
18. When is a gib-head key used?

UNIT FIVE

Blueprint Reading Fundamentals

From earliest times, people have communicated their thoughts through drawings. The pictorial representation of an idea is vital communication between the designer and the people who produce the final product. The drawing can also provide an important testing phase for an idea. Many times, an idea may be rejected at the drawing board or CAD stage before a large investment is made.

OBJECTIVES

After completing this unit, you should be able to:

- Find information in a title block.
- Explain terms such as surface finish, isometric, oblique, first angle projection, and third angle projection.
- Make isometric and oblique sketches of objects.
- Find information on blue prints.

Almost anything can be represented by a drawing. It is important that the designer be aware machining methods and limitations. On the other hand, a machinist must fully understand all the symbols and terminology on the designer's drawing. The machinist must be able to understand the drawings to transform the ideas of the designer into useful products.

The name "blueprint" originated from one of the first processes used to make copies of drawings. One of the first processes developed to duplicate drawings produced white lines on a blue background, hence the name "blueprint."

TITLE BLOCK

The title block is normally located in the lower-right corner (Figure A-32). The title block contains the drawing number, the name of the part or assembly, and the material to be used. It also includes the scale of the drawing, drafting record, authentication, date, and may include more information depending on the complexity of the part.

Drawing Number

Each blueprint has a drawing number in the title block (Figure A-32). On blueprints with more than one sheet, the information in the number block shows the sheet number and the number of sheets in the series. For example, note that the title blocks shown in Figure A-32 shows sheet 1 of A complex part or assembly might have several sheets.

Revision Block

If a revision has been made, there will be a revision block on the drawing, as shown in Figure A-32. Note that each revision is numbered and the revisions are shown with the number on the actual drawing where the feature was changed. The revision block may be attached to the title block or be in a separate location on the print. All revisions in this block are identified by a number and a brief description of the revision. A revised drawing is shown by the addition of a letter to the original number as shown in the figure. When the print is revised, the revision will use the next available letter. For example, if the first revision was labelled A, the next revision will be labelled B. The letter is used to identify the current revision of the drawing.

Reference Number

Reference numbers that appear in the title block refer to numbers of other blueprints. On some prints you may see a notation above the title block such as "34512 RH shown; 34512-1 LH opposite." RH means right hand, and LH means left hand. Note that both the right-hand and left-hand parts carry the same number, but the left-hand part number has a "–1."

Zone Number

Zone numbers serve the same purpose as the letters and numbers on the borders of maps. They are used to locate a particular feature on a drawing. It makes it easier to convey information when you are communicating with

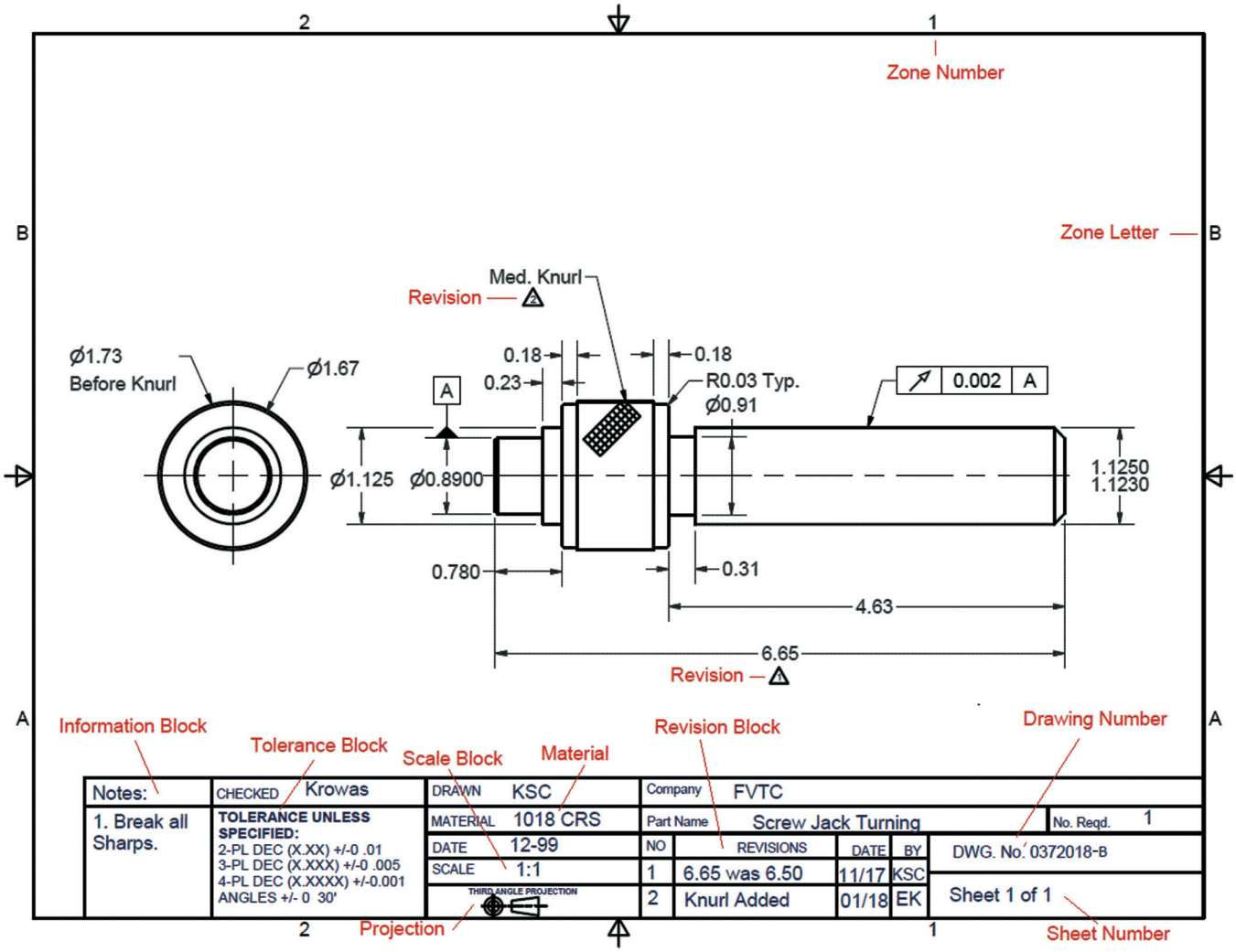


Figure A-32 An example of a typical blueprint.

someone on the phone or through email. You can give them the letter and number that helps them find the feature you are interested in. To find the feature, they can mentally draw a horizontal and vertical line from the letters and numbers. The lines will intersect at the feature you want them to find.

Scale Block

Parts may be too large to draw them full-size on the print. They may also be small and it might be advantageous to draw them larger for clarity. The scale block in the title block of the blueprint shows the size of the drawing compared with the actual size of the part. For example, the part may be drawn $\frac{1}{2}$ scale. That would mean that the drawing is $\frac{1}{2}$ the size of the actual part. If it were $\frac{2}{1}$, the drawing would twice the size of the part. The scale is chosen to fit the object being drawn and space available on the print.

Never measure a drawing to find a dimension; use the dimensions that are shown on the print. The print may have been reduced in size from the original drawing.

Bill of Material Block

The bill of material block specifies the material to be used for the part (Figure A-32). If there are multiple parts that go into an assembly of parts, they are also listed in the material block, as well as the number required for the assembly. Parts and materials may be specified by their stock number or other appropriate number.

Information Blocks

Information blocks are used to give the machinist information about materials, specifications, that may not be on the blueprint or that may need additional explanation. Heat treat specification would be one example of information that might be in an information block.

Standard Tolerance Information Blocks

Not all features will have specific tolerances shown on a blueprint. For those features that do not have a specific tolerance shown, shop tolerances are shown in an information block. For example, if a feature on a print does not have a specified tolerance, the general tolerance is used. For example, if the feature's size has two decimal places, the general tolerance might be plus or minus .015. A three-place decimal feature might have a general tolerance of plus or minus .005. These tolerances vary from shop to shop and print to print. Always look for general tolerances in an information block (Figure A-32).

Finish Marks

A machinist often works on parts that are already partially shaped. An example of this might be a casting or forging. Finish marks (Figure A-33) are used to indicate which surfaces are to be machined. Finish marks may also indicate a required degree of surface finish. For example, a finish mark notation of 4, 32, or 64 refers to a specific surface finish. The number with the surface finish symbol indicates the average height of the surface deviations measured in micro inches. A micro inch equals one millionth of an inch (.000001 inch). The smaller the number, the better the finish. Higher cost is usually associated with finer finishes. Figure A-34 shows some manufacturing processes and what finishes are possible with each. Machining provides a better surface appearance and a better fit with closely mated parts, but is also adds cost. Cast parts, for example, often do not have every surface machined. Figure A-35 shows various surface finish symbols.

Figures A-36 and A-37 show explanations of surface finish specifications. Surface finish, also known as surface

✓	Basic surface finish symbol
✓	Material removal by machining required
✓	Material removal not allowed
✓	Same finish required for all surfaces

Figure A-35 Surface finish symbols.

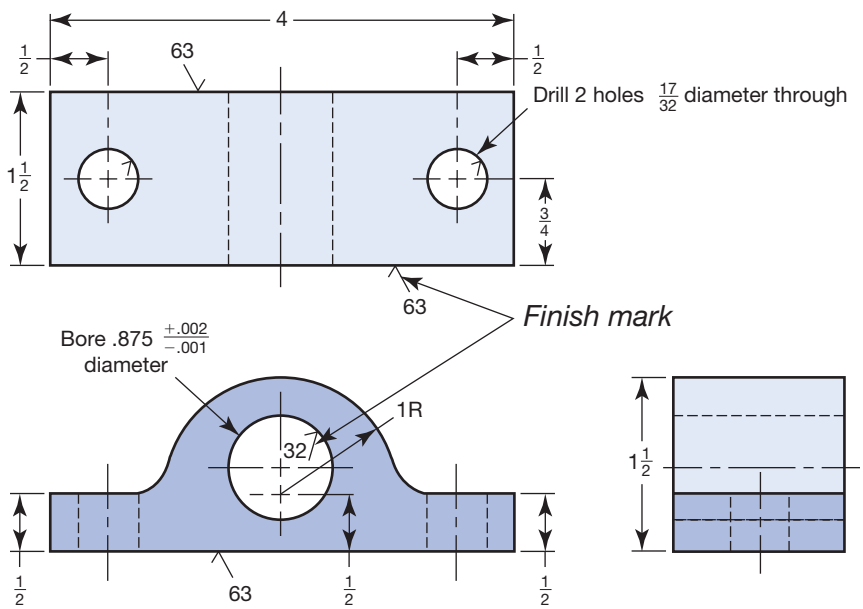


Figure A-33 Drawing showing finish marks.

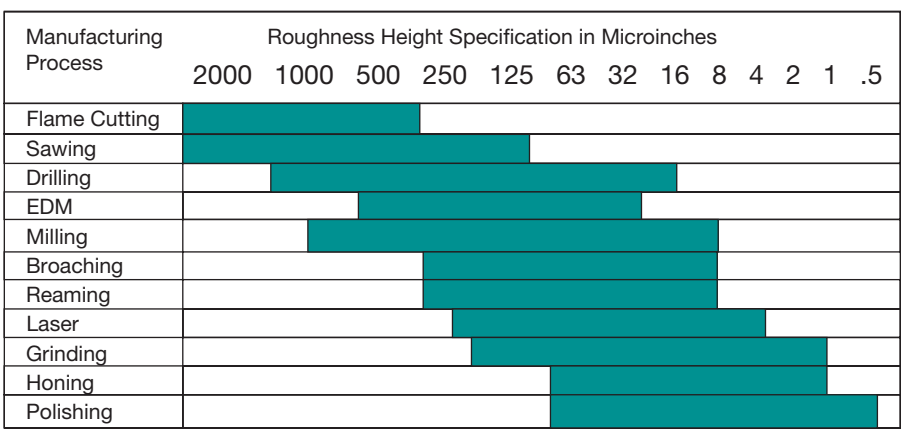


Figure A-34 Roughness for various machining methods.

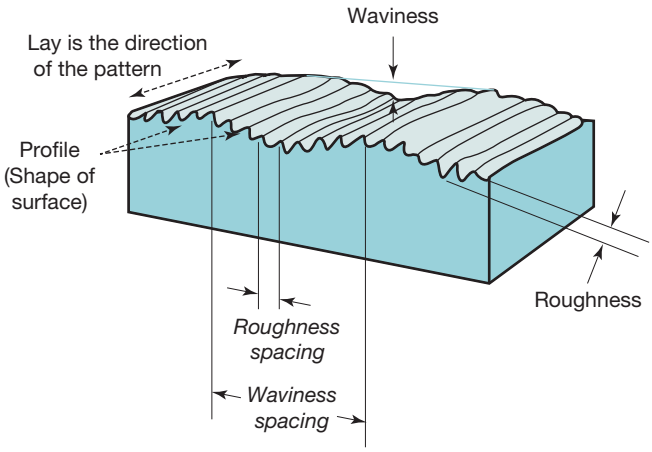


Figure A-36 Surface finish characteristics.

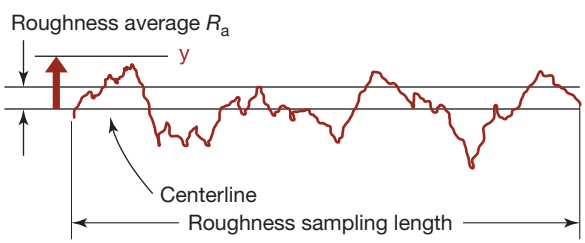


Figure A-37 Roughness (Ra) illustration.

texture, is the characteristic of a surface. Surface finish has three components: lay, surface roughness, and waviness.

On a machined part, form is the result of fluctuations when the machine produces the part. These could be called straightness errors. Waviness is a result of vibrations in the machine tool and from the surroundings. Roughness results from the chosen tool geometry, the condition of the tool, the feed-rate, and variations in material and its hardness.

The most common measure of surface finish is average roughness (Ra). Ra is calculated by calculating the average difference between the peaks and valleys and the deviation from the mean line within the sampling length (see Figure A-37). Ra averages the peaks and valleys of the part surface.

Figure A-38 shows the symbols that are used when direction of lay is specified.

Figure A-39 shows several surface finish characteristics that can be specified with a finish symbol.

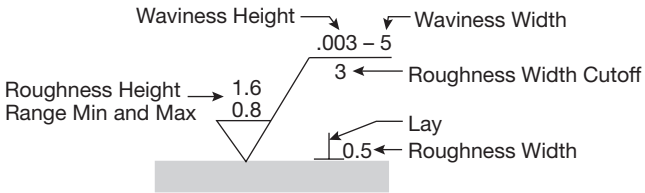


Figure A-39 How surface finish is represented on a blueprint.

Isometric and Oblique (Perspective) Drawings

An isometric drawing (Figure A-40) is one method used to represent an object in three dimensions. In the isometric format, the lines of the object remain parallel and the object is drawn about three isometric axes that are 120 degrees apart. In an isometric drawing the baselines of the part are drawn at a 30-degree angle to horizontal.

Figure A-41 shows an oblique drawing of a cube. Note that the front view is drawn horizontally. Object lines in the oblique drawing are parallel. The oblique differs from the isometric in that one axis of the object is parallel to the plane of the drawing. Isometric and oblique are not used as working drawings for the machinist. You may occasionally see them in the machine shop.

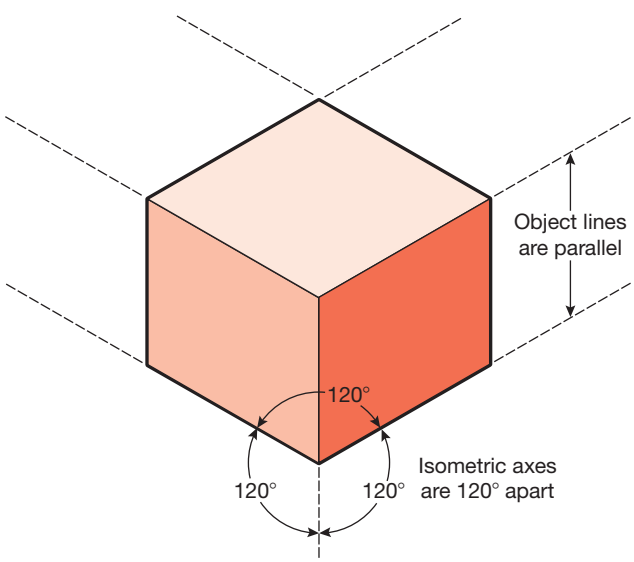
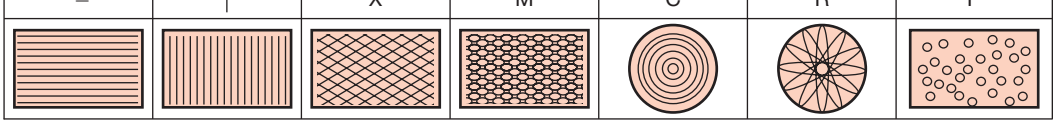


Figure A-40 An isometric drawing.

Figure A-38 Illustrations of lay and their symbols.

Parallel	Across	Crossed	Different	Concentric	Radial	Non-Directional
=	|	X	M	C	R	P



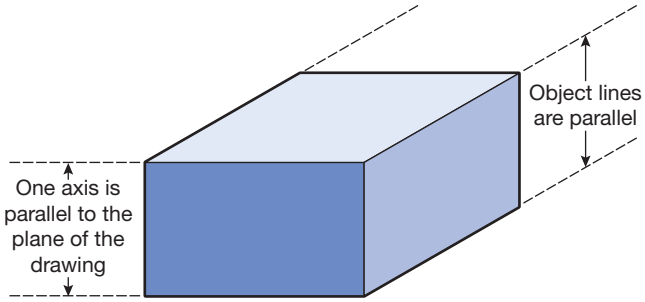


Figure A-41 Oblique drawing of a cube.

SELF-TEST

1. Draw the symbol for third-angle projection.
2. What is the difference between first and third angle projection?
3. Draw an oblique sketch of a shoe box.
4. Draw an isometric sketch of a shoe box.
5. List at least five things that are found in a title block.
6. Explain where revisions are shown on a blueprint and how they are indicated on the feature that was changed.

UNIT SIX

Views and Line Types

The basis of blueprints is orthographic projection. This unit will examine how views of objects are chosen and projected to create blueprints. This unit will also examine line types. Different types and weights of lines are used to convey information on blueprints.

OBJECTIVES

After completing this unit, you should be able to:

- Explain terms such as orthographic projection, phantom line, cutting plane line, hidden line, section line, auxiliary view.
- Identify views on a blueprint.
- Sketch views of a part.
- Identify surfaces of a part in different views.
- Explain features of a part on a blueprint based on line type.

VIEWS

Imagine you are a mechanical designer. You have a part that you need to draw. You can pick up the part, turn it around, and look at the features of the part. You can look at the object so that certain features are visible and also true size and shape. You must determine how many views of the object you need to draw to make every detail of the part understandable. Determining which views to provide on the drawing is a critical decision.

Too many views can make a drawing difficult to read. Omitting views that provide true size views of features will make it impossible to correctly dimension the drawing. The designer should provide views that will provide the part information needed in production and a clear representation of the geometry of the part. The machinist must have the ability to look at the views provided on a blueprint and visualize what the part will actually look like.

Orthographic Projection

In almost every case, the working drawing will be a three-view or orthographic drawing. The typical orthographic format always shows an object in the three-view combination of front, side, and top (Figure A-42). In many cases, an object can be completely shown by a combination of only two orthographic views. However, any orthographic drawing must have a minimum of two views to show an object completely. The top, front, and right-side views are the most common on typical drawings. Left-side, rear, and bottom views are also seen occasionally, depending on the complexity of the part being illustrated.

Third-Angle Projection versus First-Angle Projection

There are two main methods to project the part into views. Third-angle projection is the one used in America. A blueprint will have a symbol on it to declare which method was used. Figure A-43 shows the symbols for third-angle and first-angle projection. First angle is common in Europe and

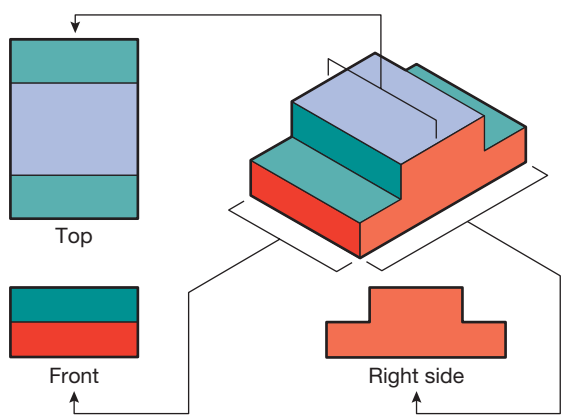


Figure A-42 An orthographic drawing.

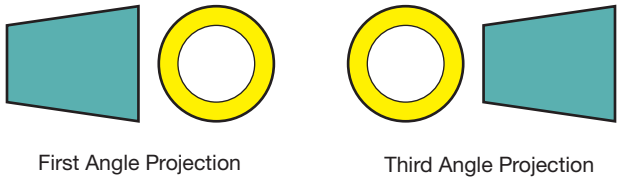


Figure A-43 Symbols for first-angle and third-angle projection.

many other countries. It is always wise to look for the projection symbol on a blueprint to be sure which system is being used. If you think the print was drawn in third angle projection but it was really drawn in first angle projection you will make parts that are essentially left-handed instead of right-handed and the parts will be scrap.

There are six possible standard views. Two or three are usually enough to show all the required information for a part. Figure A-44 shows how the three most common views are projected to create the front, top, and right-side views for a blueprint.

Figure A-44 also shows examples of third-angle and first-angle projection. In these examples the views were projected onto the walls of an imaginary glass box. In third-angle projection (normally used in America) the front of the glass box shows the front view. The right side of the glass box shows the right-side view. The top of the glass box shows the top view. Note also the arrangement of the views. If there was a hinge on the edge of the box between the front view and the right-side view, we could open the side of the box out to the front and it would be in the correct position for the drawing. If there was a hinge on the edge of the glass box between the front view and top view, we could open the top of the glass box and the top view would be in the correct position for the drawing.

LINE TYPES AND MEANINGS

There are several types of lines used on blueprints. The type and weight of the line are used to convey information. Figure A-45 shows the use of standard lines in a simple drawing. Line characteristics such as width, breaks in the line, and zigzags have meaning.

Object Lines

Object lines (Figure A-45) are thick solid lines that outline all surfaces of a part that are visible to the eye.

Hidden Lines

Hidden or invisible lines are lines that have short, evenly spaced dashes. Hidden lines are used to outline invisible or hidden surfaces (Figure A-45). They are thin lines, about half the weight of visible lines. Hidden lines begin with a dash, except if the dash would create a continuation of a solid line.

Extension Lines

Extension lines are short, solid, thin lines used to show the limits of dimensions (Figure A-45). They may be placed inside or outside the outline of an object. They extend from the outline or surface of a part. Extension lines do not touch the part.

Dimension Lines

A dimension line is a thin solid line that shows the extent and direction of a dimension. Arrows are used at the ends of dimension lines to show the limits of the dimension. Dimension lines are broken to insert the dimension (Figure A-45).

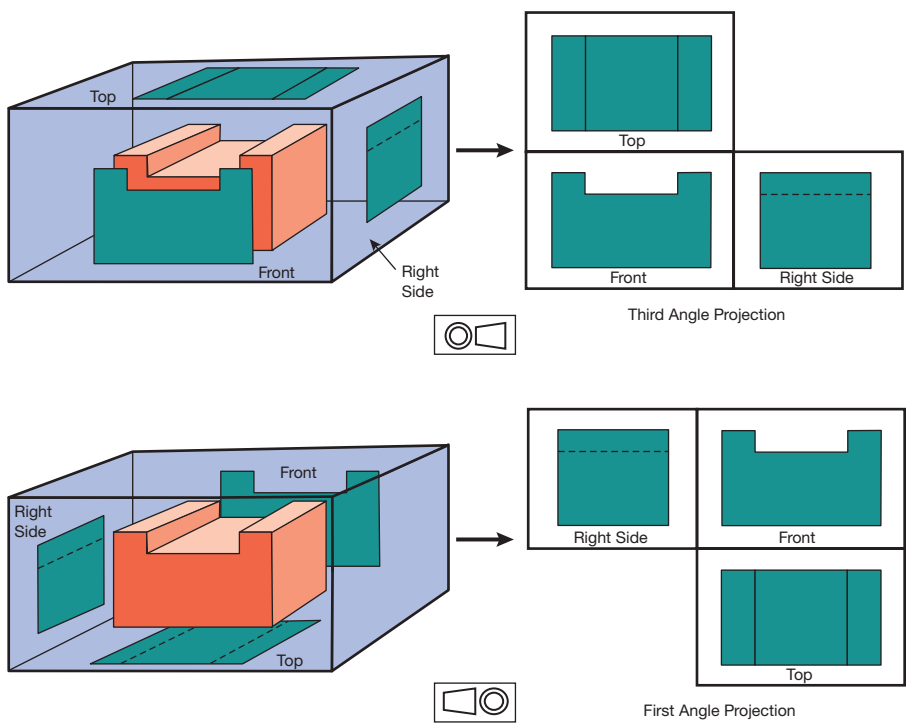


Figure A-44 The example on the top shows a third-angle projection. Note that the projections on the glass box are between the viewer and the object. The example on the bottom shows a first-angle projection. Note that the object is between the viewer and the projections on the glass box. First-angle projects different views. Make sure you understand which system was used before making parts.

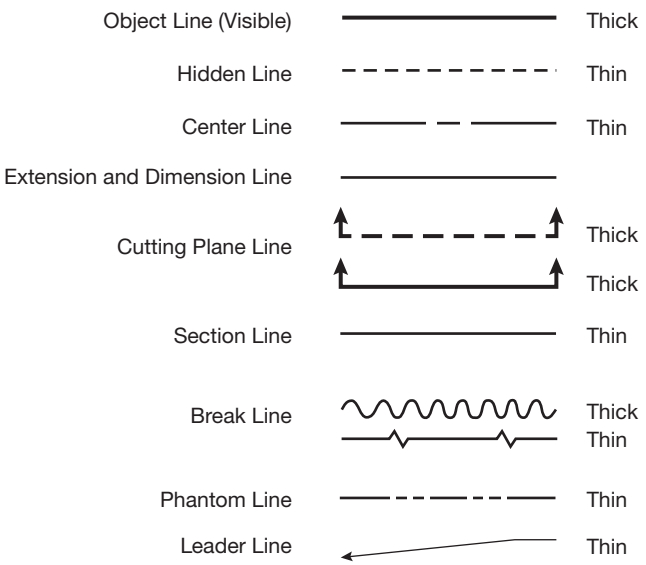


Figure A-45 Line types.

Leader Lines

A Leader line is a thin solid line that may have an arrow that is used to indicate a feature on a part (Figure A-45). Leader lines indicate the part or feature to which a number, note, or other reference applies.

Centerlines

Centerlines have alternating long and short evenly spaced dashes. Centerlines have the same weight as invisible lines. They have a long dash at each end and short dashes at points of intersection (Figure A-45). Centerlines are used to show the central axis of an object or part. If a circle, such as a bolt circle or hole is shown, horizontal and vertical centerlines are used to show the center point.

Break Lines

There are two types of lines for long and short breaks (see Figure A-45). Short breaks are shown by solid, thick, freehand lines. Long breaks are shown by solid, thin, lines broken by freehand zigzags. These can be used to indicate that a part is broken out or removed. They also are used to reduce the size of the drawing of a long part having a uniform cross section. For example, a shaft that is 10 feet long with no details between the ends would be drawn with a break in the middle, rather than having to show 10 feet of the shaft. Breaks on shafts, rods, tubes, and pipes are curved (Figure A-45).

Phantom Lines

Phantom lines (Figure A-45) are thin lines that can be used to indicate the locations of absent parts, alternate positions of the parts of an object, or repeated details.

Cutting Plane Lines

Cutting plane lines bisect a part and provide a view of the part's interior features (Figures A-45 and A-46). A cutting plane is a plane that cuts through a part drawing to create a sectional view. The sectional view is used to show internal features that cannot be seen from the outside. Imagine cutting a candy bar in half to see what is inside. The portion of the part that is in front of the cutting plane surface is removed to expose the internal details on the section view. Figure A-46 shows an example of a full- and half-section view. In the section view, features that would be cut to produce the section view are indicated by section lines.

Section Lines

Section lines, also called crosshatch lines, are used to show the portion of a part that would be cut to create a section view (Figure A-46). Section lines can also be used to distinguish between two separate parts that meet at a given point. Each part is lined or hatched in opposite directions with thin parallel lines at 30, 45, or 60 degrees on the exposed cut surface.

When internal features are complex and showing them on a drawing as hidden lines would be confusing, a section drawing may be used. Two common styles of sections are used. In the full section (Figure A-46, Section AA), the object has been cut completely through. In the half section (Figure A-46, Section BB), one quarter of the object is removed. The section indicator line shows the plane at which the section is taken. For example (Figure A-46, Section AA), the end view of the object shows the section line marked AA. Section line BB (Figure A-46, Section BB) indicates the portion removed in the half section. An object may be sectioned at any plane as long as the section plane is indicated on the drawing.

Figure A-47 shows an example of a drawing with various types of lines and their use identified.

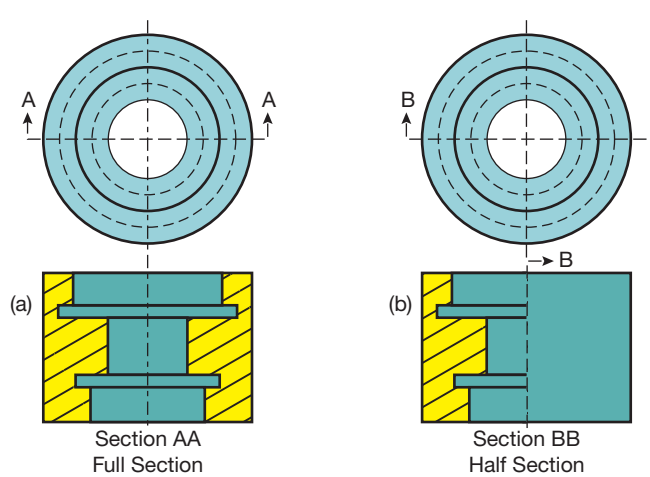


Figure A-46 Full- and half-section views.

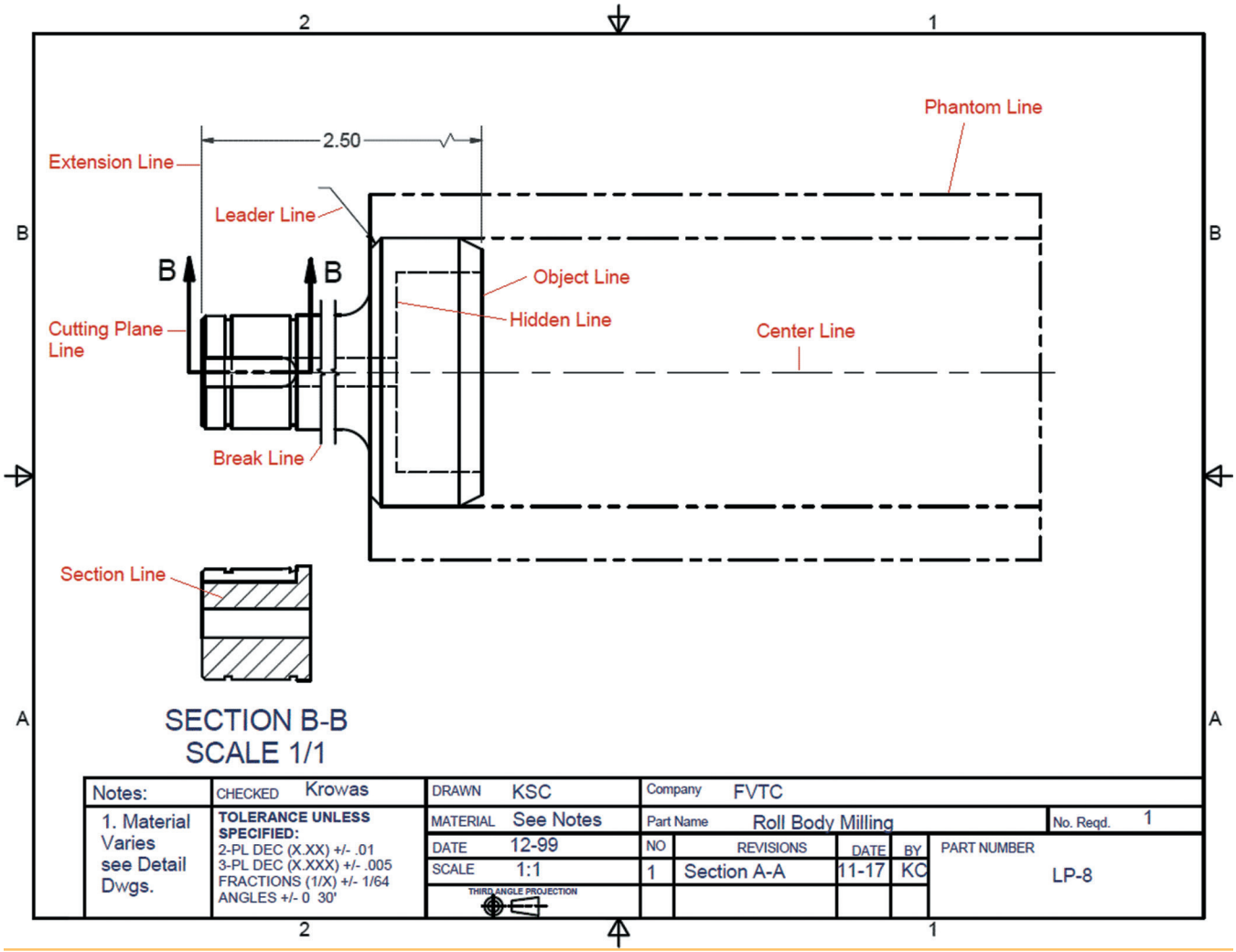


Figure A-47 Drawing showing various types of lines.

Auxiliary Views

When creating drawings, features should be shown in a view where they appear to be true to their size so that they can be dimensioned. The object is normally positioned such that the major surfaces and features are either parallel or perpendicular to the X, Y, and Z planes. For example, normally the front, top, and right-side views are shown on a part drawing. Views of the part are normally chosen so that most of the features will be visible in the front, top, and right-side views.

Study the angular surface in Figure A-48. To get a true view of the angled surface and its size, the view had to be projected at an angle. If lines had been projected from the part vertically or horizontally, the view and size of the surface would be distorted. Lines had to be projected at the angle perpendicular to the surface to provide a true view of the surface.

If a part is complex, three views may not be enough to show all of the part's features. Some part features may not appear to be true size and shape in those views, or may be hidden. In examples like this, it may be necessary to draw one or more auxiliary views. Primary auxiliary views are projected off one of the main part views. If a secondary auxiliary view is required, it is projected off a primary auxiliary view.

Exploded Views

An exploded drawing (Figure A-49) is a type of pictorial drawing designed to show several parts in their proper location prior to assembly. Exploded views appear extensively in manuals used for repair and assembly of machines and other mechanisms.

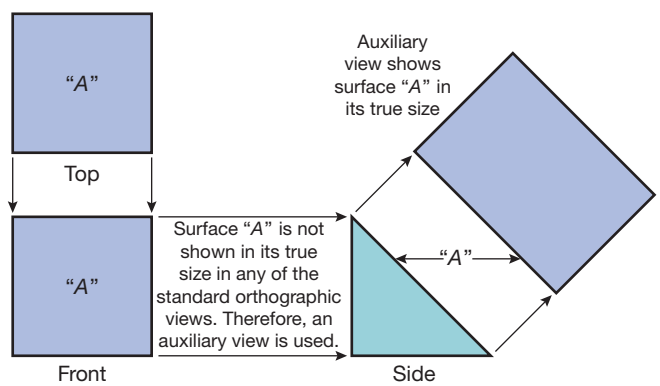


Figure A-48 Drawing with an auxiliary view.

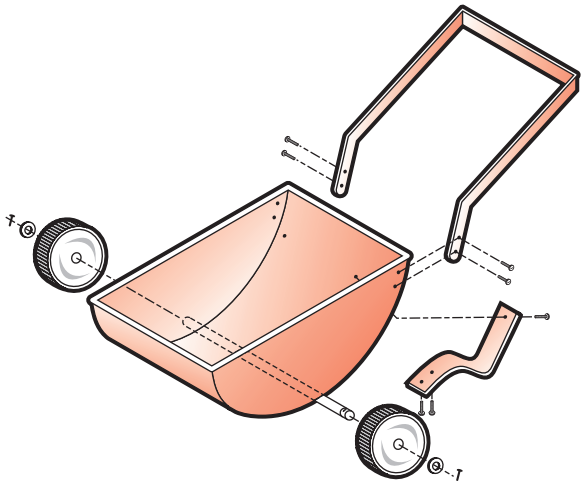


Figure A-49 Example of an exploded view.

SELF-TEST

1. Identify the types of lines in the drawing in Figure A-50 (Hint: object, invisible, phantom, extension, dimension, broken, hidden, center, cutting plane line, part, heavy, break, section, parting)
2. Assume third-angle projection. Label the views in Figure A-51.
3. Draw the three third-angle projection views for the part shown in Figure A-52.
4. Draw the missing lines in the three views shown in Figure A-53.
5. Fill in the numbers that represent the surface on the part shown in Figure A-54 in each view.
6. Fill in the numbers that represent the surface on the part shown in Figure A-55 in each view.
7. Draw the missing lines in the views shown in Figure A-56.
8. Draw the missing lines in the views shown in Figure A-57.
9. Draw the missing lines in the views shown in Figure A-58.
10. Draw the missing lines in the views shown in Figure A-59.

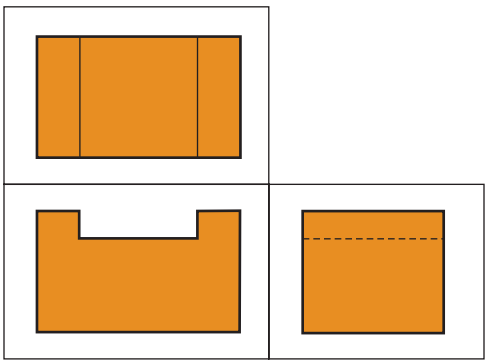


Figure A-51 Drawing for question 2.

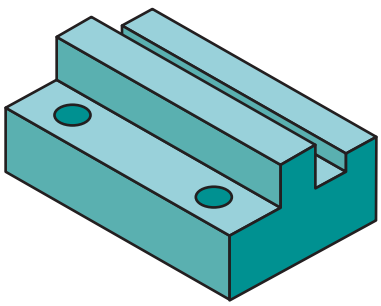


Figure A-52 Drawing for question 3.

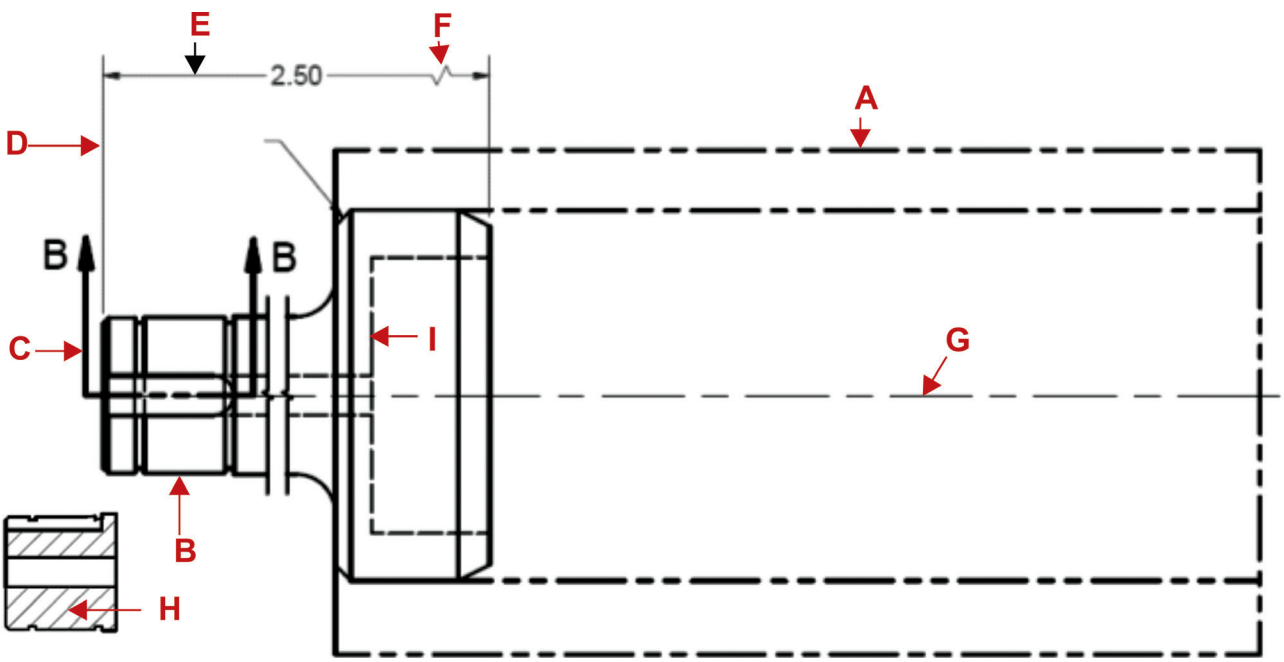


Figure A-50 Part drawing for the self check.

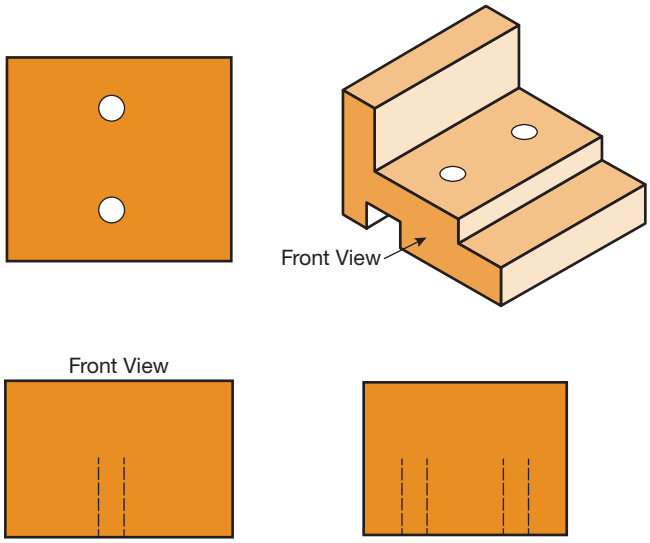


Figure A-53 Drawing for question 4.

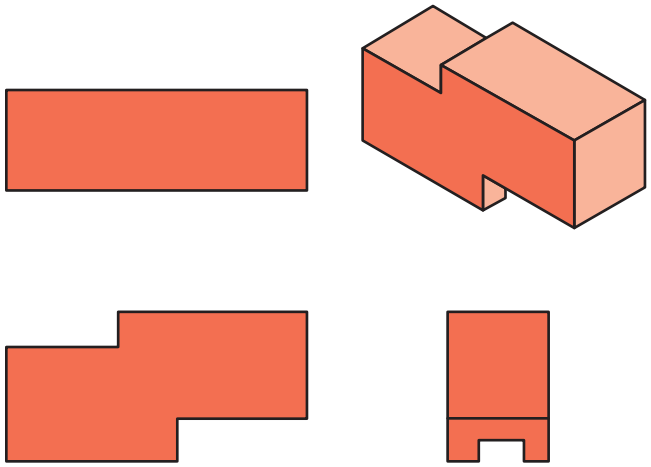


Figure A-56 Drawing for question 7.

Figure A-54 Drawing for question 5.

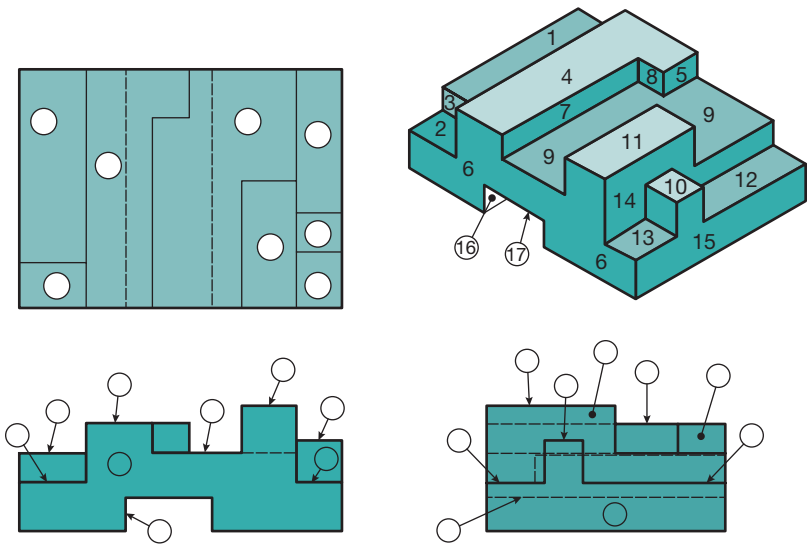
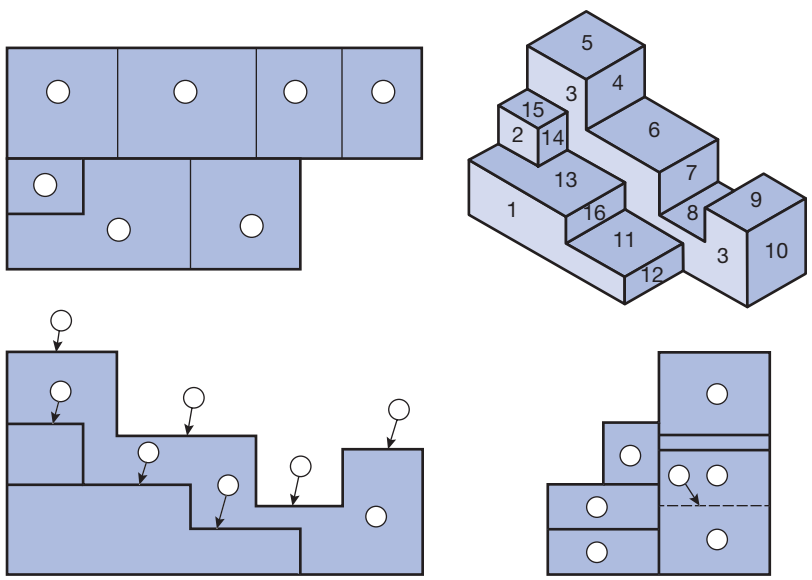


Figure A-55 Drawing for question 6.



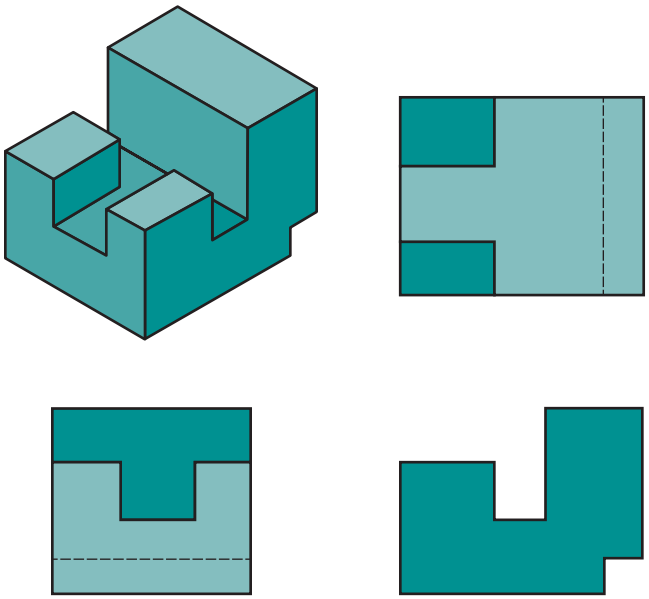


Figure A-57 Drawing for question 8.

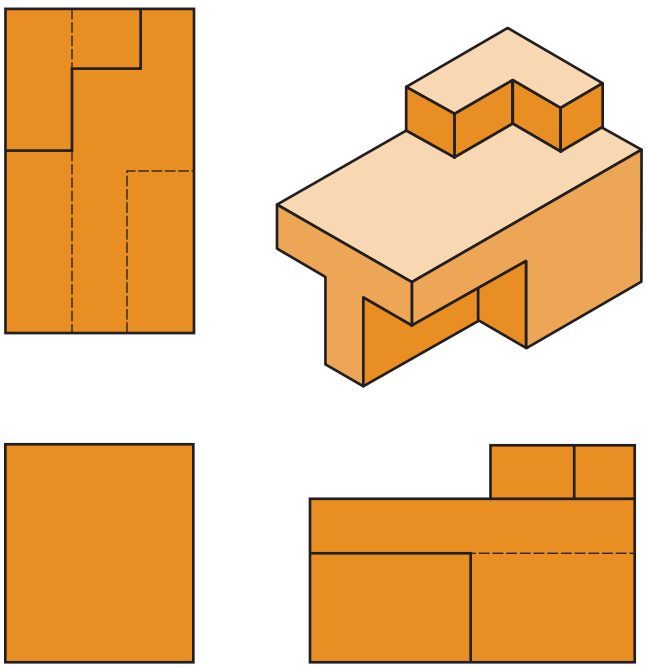


Figure A-58 Drawing for question 9.

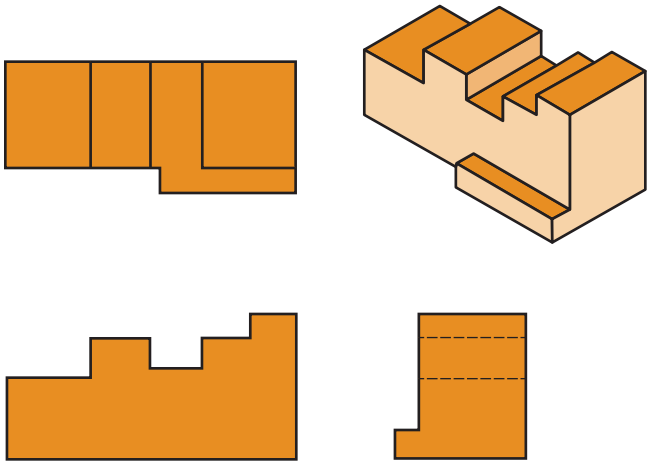


Figure A-59 Drawing for question 10.

UNIT SEVEN

Dimensions, Tolerances, and Fits

A key to producing functional parts is the development of dimensions for key features and their tolerances. Prints can be dimensioned using absolute dimensioning or incremental dimensions. Both are used in some cases. Mating parts also need to be dimensioned and toleranced. Parts have various functions. In one application one part, like a shaft, needs to be turned by a gear. In this example, a drive fit would be needed so that the gear is tight on the shaft and would turn the shaft. There are a variety of types of fits. This unit will cover the system of fits and their specification.

OBJECTIVES

After completing this unit, you should be able to:

- Explain terms such as absolute, incremental, detail drawing, bilateral, unilateral, limits, counterbore, spotface, and countersink.
- Develop unilateral and bilateral tolerances for part features.
- Given an application that involves fit, specify the tolerances for the clearance and the mating parts.

DIMENSIONS

A machinist typically works with a drawing type known as detail drawing. A detail drawing is a drawing of an individual part. A detail drawing contains all the essential information a machinist needs to make a particular part. Most important are the dimensions. Dimensioning refers generally to the sizes specified for the part and the locations of its features. Furthermore, dimensions reflect many design considerations, such as the fit of mating parts so they will function correctly.

A dimension defines the size or specification of a feature on a part. A basic dimension is the numerical value that would be the theoretically exact size (true size) of a feature. A reference dimension is a numerical value provided for information only. Reference dimensions are enclosed in parentheses on blueprints. They are for reference only and are not used in the manufacturing of the part.

The coordinate or absolute system of dimensioning (Figure A-60) is used for most part drawings. In this system, all dimensions are specified from the same zero point. This is very important. If some part features were specified from the left side of the part and some from the right, that part might not be functional because the length of the part will vary. If the part is longer, the features specified from the right side will be further away from the part features specified from the left side. If all of the part feature dimensions are taken from one side, the part should be functional because all of the part features will be located in relation to each other. Note that this method of dimensioning also makes it easier to program the part on a CNC machine. The other method of dimensioning is called incremental dimensioning. In incremental dimensioning, one feature is specified as the distance and direction from another feature, not from a zero point.

The figure shows the dimensions expressed in decimal form. General tolerances that would be shown in the title block apply to the print unless specific tolerances are shown.

With the increasing use of the metric system in recent years, some companies put dual dimensions on drawings with both metric and inch notation. In Figure A-61, metric dimensions appear above the line and inch dimensions appear below the line.

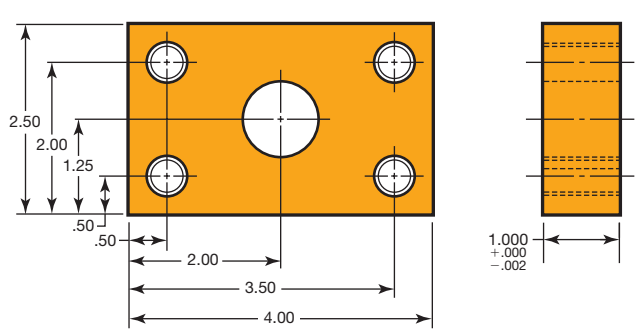


Figure A-60 Example of absolute dimensioning.

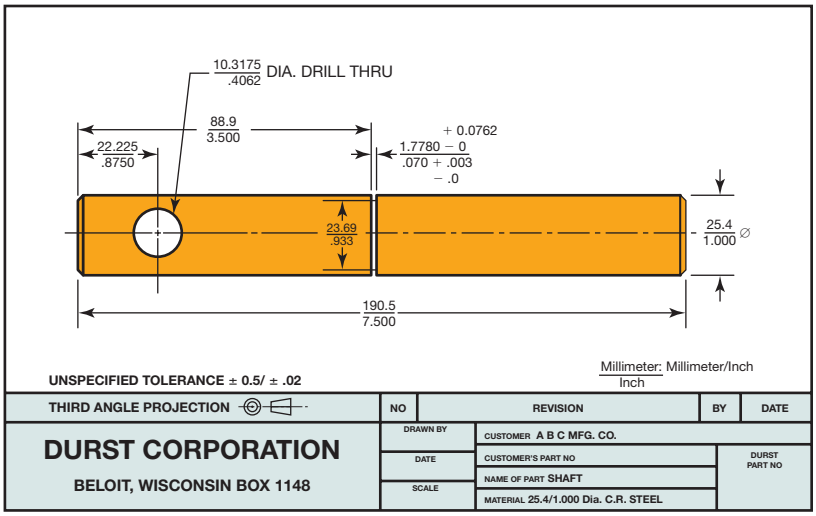


Figure A-61 Dual dimensioning in metric and English.

Limit and Tolerance

Since it is impossible to machine a part to an exact size, a designer must specify an acceptable range of sizes that will still permit the part to function as designed. Tolerances also make it possible for different manufacturers to make parts and have them work together. The difference between the maximum and minimum limits is the tolerance, or the total amount by which a part dimension may vary. Tolerances on drawings are often indicated by specifying a limit, or by plus and minus notations (Figure A-62). A limit dimension shows the upper and lower limit for the allowable size. With plus/minus tolerance, when the tolerance is both above and below the nominal (true theoretical) size, it is said to be bilateral (two-sided). When the tolerance is indicated all on one side of nominal, it is said to be unilateral (one-sided).

Plus and minus dimensioning is the allowable positive and negative variance from the dimension specified. Note that the tolerance does not have to be the same in the positive and negative specification for a feature. For example, a bore might be specified as 1.000 +.002 −.001.

Standard Tolerances

On many drawings, tolerances will be specified at the specific feature's dimension. If no specific tolerances are specified at the feature's dimension, general tolerances are used. These are often listed in the title block.

Note that in Figure A-63, no tolerance is specified for the overall dimensions of the part or the locations of the various features. In this case the general tolerances are applied. In this example the general tolerances are shown at the top of the figure. The general tolerances apply when no specific tolerance is shown for a particular part feature. Consider the overall length of the part (4.00 in). Since there is no specific tolerance shown for that dimension on the print, the general tolerance would apply. In this example from Figure A-63 the tolerance for 4.00 (a 2-place decimal) would be ± .01.

How Tolerance Affects Mating Parts

When parts mate in an assembly, tolerance becomes crucial. Consider the following example shown in Figure A-64. The shaft must fit the bore in the bearing and be able to turn freely. The diameter of the shaft is specified as 1.000 + or − .001. This means that the maximum limit of the shaft is 1.001 and the minimum is .999. The total tolerance is .002 and bilateral. The maximum limit of the bearing bore is 1.002 and the minimum limit is 1.000. The total tolerance is .002. Will the shaft made by one machine shop fit the bearing made by another machine shop using the tolerances specified? If the shaft is turned to the maximum limit of 1.001 and the bearing is bored to its minimum limit of 1.000, both parts will be within acceptable

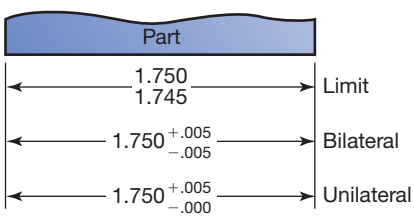


Figure A-62 Types of tolerance.

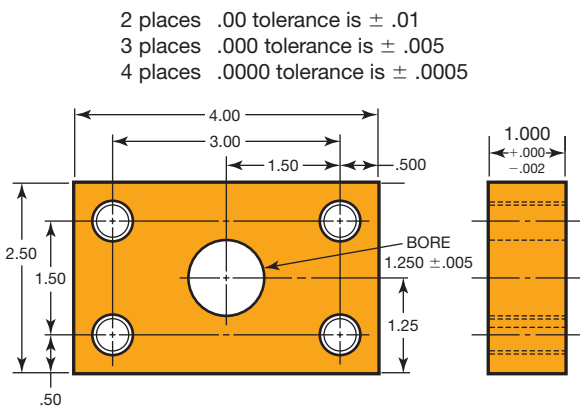


Figure A-63 Illustration of tolerancing a part.

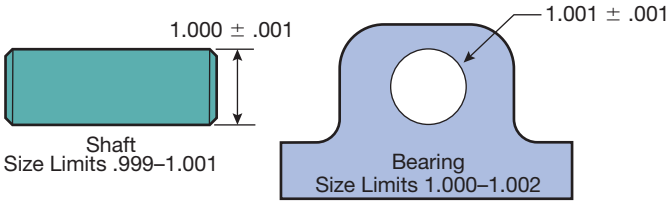


Figure A-64 The shaft diameter is specified to be 1.000 ± .001. The maximum diameter would be 1.001 and the minimum diameter would be .999. The bore could be 1.000 and still be in tolerance. The shaft could be larger than the bore, thus preventing proper fit.

tolerance but will not fit, since the shaft is .001 larger than the bearing. However, if the bearing bore was specified in limit form or unilateral tolerance of 1.002 + .002 − .000, the parts would fit as intended. Even if the shaft was turned to the high limit of 1.001, it would still fit the bearing even though the bore was machined to the low limit of 1.002.

FITS

Fit refers to the amount of clearance or lack of clearance between two mating parts. Fits can range from free running or sliding, where a certain amount of clearance exists between mating parts, to press or interference fits, where parts are forced together under pressure. Clearance fits can range from a few millionths of an inch, such as in the parts of a ball or roller bearing, to a clearance of several thousandths of an inch, for a low-speed drive bushing. There are charts in the Machinery's Handbook and other references that have classes of fits and the allowances for each.

There are three basic types of fit: clearance; transition, and interference. A clearance fit is a fit that has clearance between the hole and shaft. A transition fit is a fit where, depending on the size of the shaft and hole, clearance or interference may occur in the coupling. An interference fit is a fit that always has some interference between the hole and shaft in the coupling.

Running and Sliding Fits (RC)

Running and sliding fit ranges are designated with the classifications of RC1 to RC9. The lower RC numbers are, the tighter the fit. The higher the number, the looser the fit.

Close Sliding Fit (RC1)

A close sliding fit is intended for the accurate location of parts which must assemble without noticeable play.

Sliding Fit (RC2)

RC2 fits are designed for accurate location but with greater maximum clearance than class RC1. Parts made to this fit, turn and move easily. RC2 fits are still relatively close and in larger sizes may seize with temperature changes.

Precision Running Fit (RC3)

An RC3 fit is about the closest fit that will run freely. Precision running fits are intended for precision applications at low speed with low bearing pressures. RC3 fits should not be used where noticeable temperature differences can occur.

Close Running Fit (RC4)

RC4 fits are used for running fits on accurate machinery with moderate speed and moderate bearing pressures where accurate location and minimal play is required.

Medium Running Fit (RC5 and R6)

RC5 and RC6 fits are designed for machines running at higher speeds with large bearing pressures.

Free Running Fit (RC7)

RC7 fits are used where accuracy is not crucial. An RC7 fit allows large temperature variations.

Loose Running Fit (RC8 and RC9)

RC8 and RC9 fits are used where wide tolerances are acceptable for the shaft. These loose running fits may be used where the parts are exposed to the effects of corrosion and contamination.

Locational Fits (LC, LT, LN)

Locational fits are divided into three groups; locational clearance fits (LC), locational transition fits (LT), and interference fits (LN). These fits are used to accurately locate mating parts. They provide accurate location with an interference fit or a minor amount of location variation with a clearance fit. Locational fits are not used for moving parts.

Locational Clearance Fit (LC)

Locational fits are used for non-moving parts that need to be easily assembled and disassembled. Locational clearance fits can be snug fits for parts that require accurate location or looser fits. The tolerance of mating parts and clearance increases with increasing class of the fit. The lower classes such as LC1 and LC2 are for precise location of parts. Higher classes such as LC10 and LC11 are for free fits with greater clearances and maximum tolerances. LC fits are tighter than RC fits.

Locational Transition Fit (LT)

Locational transition fits lie between clearance and interference. Locational interference fits are for applications where accuracy of location is important. A small amount of clearance or interference is permissible with a locational transition fit. These fits are a compromise between LC and LN (interference/force) fits. LT1 and LT2 fits can be used for bushings, hubs of gears or pulleys, etc. LT3 and LT4 fits can be used for pulleys, wheels, etc. The parts can be assembled or disassembled

without any great force using a rubber hammer. LT5 and LT6 can be used for motor armatures, driven bushings, shafts and so on. LT fit parts can be assembled using low force.

Locational Interference Fit (LN)

Locational Interference fits are used where accuracy of location is very important. Locational interference fits are used for parts requiring rigidity and alignment. The parts can be assembled or disassembled using a press. These fits are not designed for applications where one part is used to drive another part. For example, a shaft and a pulley. The fit would be insufficient to drive the pulley without additional parts such as a key or setscrew.

Force or Shrink Fit (FN)

Force or shrink fits are used to move parts using the friction between holes and shafts. Force and shrink fits are a type of interference fit used to maintain constant hole pressure for all sizes of components. The amount of interference varies with the diameter of the parts. The amount of interference of the fit increases with increasing class of the fit. During assembly, steel parts are usually greased to prevent seizing or galling of the components. High pressures are normally required to assemble parts of this class of fit.

Light Drive Fit (FN1)

Light drive fits are used for parts that require light assembly pressure. Light drive fits are used for more permanent assemblies.

Medium Drive Fit (FN2)

Medium drive fits are used for steel parts. They are also used for shrink fits on light, fragile sections of material. Medium drive fits are about the tightest fits that can be used with cast-iron parts. Cast iron is quite brittle.

Heavy Drive Fit (FN3)

Heavy drive fits are used for heavy steel parts or can be used for shrink fits in medium sections of material.

Force Drive Fits (FN4, FN5)

Force drive fits are for parts which can be heavily stressed. They can also be used for shrink fits where heavy pressing forces are not practical.

HOW TO DETERMINE SIZES FOR A GIVEN FIT

Table A-4 shows a partial chart showing sizes for a few of the running and sliding (RC) fits. Note that only RC1–RC3 are shown and only a few diameters are included. Comprehensive charts that show all classes and types of fits for all diameters are readily available in reference texts. The first step for a machinist or designer is to choose the correct type of fit for the application. Let's assume an RC3 fit is chosen of our example. An RC3 fit is a close fit that will run freely. Precision running fits are used for precision applications at low speed with low bearing pressures and light journal pressures. Assume we have a 1.5-inch shaft that turns slowly in a bronze bearing on a conveyor. We find that a 1.5-inch part would be in the bottom row of the chart in Table A-4 in the basic size ranges column of the chart. Then we go to the RC3 columns on the right side of the chart. We see that the clearance between the shaft and bore should be between .001 and .0026 of an inch. The values shown in the chart are in thousandths of an inch. From the chart we see that tolerance of the bore should be 0 to .001. For our 1.5-inch application, the bore should be toleranced as 1.500 to 1.501. The shaft tolerance from the table is −.001 to −.0016. The shaft specification should be 1.4984 to 1.499.

A machinist often deals with press or interference fits. For a press or interference fit, two parts are forced together, usually by mechanical or hydraulic pressing. The frictional forces hold the parts together without any additional hardware such as keys or setscrews. Tolerances for press fits are critical because parts can easily be damaged if there is an excessive difference in their mating dimensions. In addition, press fitting physically deforms the parts to some extent. This can result in damage, mechanical binding, or the need for a secondary resizing operation such as hand reaming or honing after the parts are pressed together.

Table A-4 Partial Chart of RC Fits for Various Diameters of Shafts and Bores

Values Below Are in Thousandths of Inches										
Basic Size Ranges		RC1			RC2			RC3		
		Clearance Limits	Standard Tolerance Limits		Clearance Limits			Clearance Limits		
Over	To		Hole	Shaft		Hole	Shaft		Hole	Shaft
0	0.12	.1 to .45	0 to +.2	−.1 to−.25	.1 to .55	.25 to 0	−.1 to −.3	.3 to .95	.4 to 0	−.3 to −.55
0.12	0.24	.15 to .5	.2 to 0	−.15 to −.3	.15 to .65	.3 to 0	−.15 to −.35	.4 to 1.12	.5 to 0	−.4 to −.7
0.24	0.40	.2 to .6	.25 to 0	−.2 to −.35	.2 to .85	.35 to 0	−.2 to −.45	.5 to 1.5	.6 to 0	−.5 to −.9
0.40	0.71	.25 to .75	.3 to 0	−.2 to −.45	.25 to .95	.4 to 0	−.25 to −.55	.6 to 1.7	.7 to 0	−.6 to −1
0.71	1.19	.3 to .95	.4 to 0	−.3 to −.55	.3 to 1.2	.5 to 0	−.3 to −.7	.8 to 2.1	.8 to 0	−.8 to −1.3
1.19	1.97	.4 to 1.1	.4 to 0	−.4 to −.7	.4 to 1.4	.6 to 0	−.4 to −.8	1 to 2.6	1 to 0	−1 to −1.6

A typical example of press fit is the pressing of a ball bearing inner race onto a shaft, or an outer race into a bore. Thus, the bearing is retained by friction, and the free-running feature is obtained within the bearing itself. Ball-to-race clearance is only a few millionths of an inch in precision bearings. If a bearing is pressed into a bore or onto a shaft with excessive force, the bearing may be physically deformed and cause mechanical binding. This would cause excess heat and friction and result in early failure of the bearing. On the other hand, insufficient frictional retention of the part resulting from a poor press fit can cause the wrong part to turn under load or cause the mechanism to come loose during operation.

PRESS FIT ALLOWANCES

Press fit allowances depend on factors such as the diameter, the length of engagement, the material, and the need for later disassembly of parts. Soft materials, such as aluminum, may deform, and may not stand up to repeated pressings. Parts made of the same material pressed without lubrication may gall, making them difficult if not impossible to press apart. Thin parts such as thin-walled tubing may bend or deform to such a degree that the press fit fails to hold the parts together under load.

Generally, pressing tolerances range from a few ten thousandths to a few thousandths of an inch depending on the diameter of the parts and other factors. Proper measurement tools and techniques must be used to make accurate measurements. For additional dimensions on pressing allowances, consult a Machinery's Handbook or other reference.

PRESS FITS AND SURFACE FINISHES

The surface finish (texture) of parts being pressed is an important factor. Smooth-finished parts will press together more easily than rough-finished parts. If the roughness height of the surface texture is large, more frictional forces will be generated, and the chances for misalignment, galling, and seizing will be increased, especially if no lubricant is used. Lubrication will improve this situation. Lubrication, however, can be detrimental to press fit retention in some cases. Lubricant between fit surfaces, especially if they are quite smooth, can cause the parts to slip when subjected to force.

SHRINK AND EXPANSION FITS

Parts can be fitted by using the characteristics of metals to expand or contract when heated or cooled. Heating parts expand them, and they can then be slipped onto a mating part. When the part cools, the heated part will contract and grip the mating part with tremendous force. Parts may also be mated by cooling one so that it shrinks. When the part warms to ambient temperature, it will expand to meet the mating part. Shrink and expansion fits can have superior holding power over press fits. As with press fits, allowances are extremely important. Consult a machining handbook for proper allowance specifications.

A clearance fit exists when two mating parts have a clearance between them when assembled. An interference fit exists when two mating parts interfere when assembled. A transition fit exists when two mating parts may have an interference fit or a clearance fit when assembled.

ABBREVIATIONS FOR MACHINE OPERATIONS

Drawings contain symbols and abbreviations that convey important information to the machinist (Figure A-65). Many machining operations can be abbreviated. Countersinking is a machining operation in which the end of a hole is shaped to accept a flat head screw. On a drawing, countersinking may be abbreviated as CS or CSK. The desired angle will also be specified. A counterbore is a hole that is enlarged in diameter so that a bolt head may be recessed. Counterboring may be abbreviated as C'BORE. Spot facing is usually spelled out. This operation is similar to counterboring except that the spot facing depth is only enough to provide a smooth and flat surface around a hole. Figure A-65 shows the symbols for counterbore, spotface, and countersink. Note that counterbore and spotface use the same symbol. The depth would be different for spotface and counterbore.

OTHER COMMON SYMBOLS AND ABBREVIATIONS

External and internal radii are generally indicated by the symbol R and the specified size. Chamfers may be indicated by size and angle (Figure A-66). Threads on drawings are generally represented by symbols (Figure A-66). Thread notations on drawings include form, size, number of threads per inch or per millimeter, and the class of fit.

Bolt circles are indicated by the abbreviation B.C. or D.B.C (diameter bolt circle), meaning bolt circle (see Figure A-66). The size of the diameter of a bolt circle is often indicated with an abbreviation such as 2 1/2 B.C.

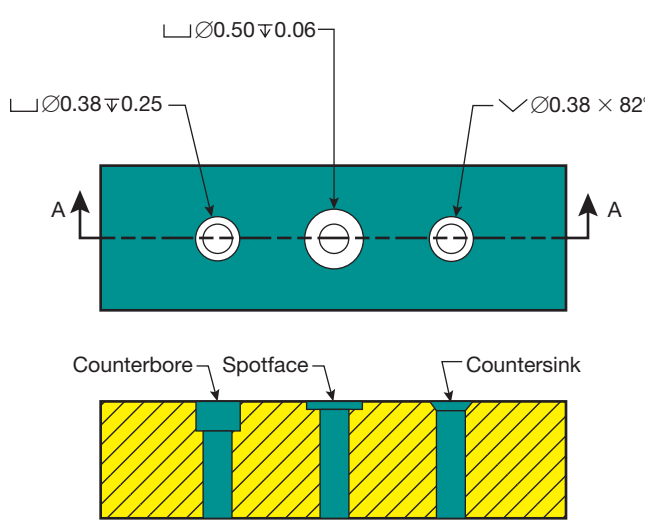


Figure A-65 Symbols for various machining operations.

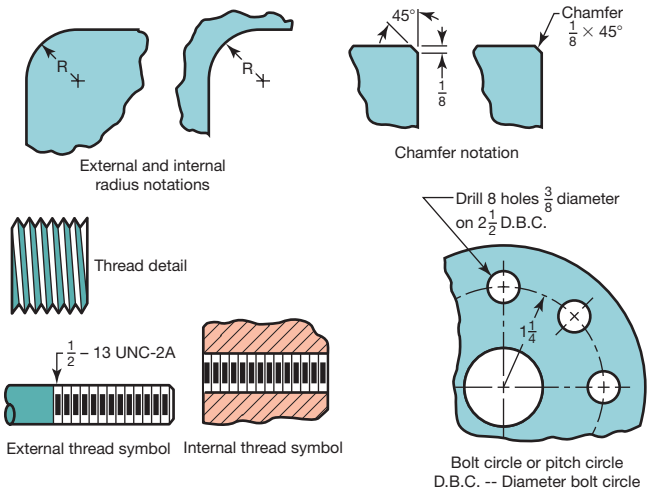


Figure A-66 Common machining symbols and abbreviations.

SELF-TEST

1. What is absolute dimensioning?
2. What is incremental dimensioning?
3. What is a general tolerance and where are they found?
4. Write down an example of a bilateral tolerance.
5. Write down an example of a unilateral tolerance.
6. What are the three basic types of fits?
7. Describe an RC3 fit.
8. Describe an LT fit.
9. Describe a LN fit.
10. An RC3 fit is desired for a one inch shaft and bore. Give the tolerance specification for the clearance between the shaft and bore, the tolerance for the shaft, and the tolerance for the bore in thousandths of an inch.

UNIT EIGHT

Fundamentals of GD&T

Geometric dimensioning and tolerancing (GD&T) has had profound effects on manufacturing. GD&T actually enables a part to have more tolerance and still assure that it will function as designed. One interesting fact about GD&T is that a tolerance can actually change based on the size of a part. For example, if a bored hole is at the upper end of its size the location tolerance might be larger, and yet assure that it will function.

OBJECTIVES

After completing this unit, you should be able to:

- Explain the difference between traditional rectangular tolerances and GD&T tolerances.
- Explain the use and function of datums.
- Explain maximum material condition, least material condition, and regardless of feature size as they affect tolerances.
- Calculate tolerances based on MMC and LMC.
- Explain what a feature control frame is.

GEOMETRIC DIMENSIONING AND TOLERANCING

It is often more important to control the form, orientation, location, profile, or runout of a part than the basic tolerance. In other words, the function of the parts is the most important. Geometric tolerancing helps establish specifications so that parts will function as designed.

Geometric Dimensioning and Tolerancing (GD&T)

Tolerances allow parts to vary from their perfect sizes, but only within defined limits. The difference with GD&T tolerancing is that it is based on the part's function. Tolerance limits allow a part's size to vary, but ensure that the part will function as intended. A designer must always balance between high cost, narrow tolerances and lower cost, wide tolerances.

GD&T can actually allow larger tolerances while ensuring the parts will function properly. Figure A-67 shows an example. A hole position is specified using normal rectangular tolerances on the left of the drawing. The hole position is 1.000 plus or minus .005 in the X and the Y axes. This tolerance zone is illustrated in gray on the right-hand side of the figure. The hole center's location could be anywhere in the gray area and it would be in tolerance for location. A hole could be located at the extreme corner positions and still be within the + or − .005 tolerance. But as the figure shows, the hole position would actually be located .007 from the center on the diagonal. That means that the hole location (if properly engineered) would actually be functional if it were .007 out of location. The problem is that with rectangular tolerancing there is no way to show that, so the stated tolerance must be + or − .005. Geometric tolerancing utilizes a circular tolerance zone. Note that any hole location in the blue would also be a good part. In effect the tolerance has been increased with the use of GD&T while still ensuring the proper function of the part.

Geometric tolerances are used to control the form and shape of parts. They are used when the shape or form has a particular function and inaccuracies would result in poor performance. In these cases, GD&T tolerances are applied in addition to dimensional tolerances.

Datums and Basic Dimensions

Datums are reference points, lines, areas, and planes that are theoretically exact for the purpose of calculations and measurements. The first machined surface on a casting, for example, might be selected as a datum surface and used as a reference from which to locate other part features. Datums are usually not changed by subsequent machining operations and are identified by letters inside a rectangular frame (see Figure A-68). In this figure, the bottom of the plate is Datum B and the left side of the part is Datum A. In this example, it would be important to take dimensions from Datum A and Datum B.

The term basic dimension (or true position) represents a theoretically exact dimension describing the location or

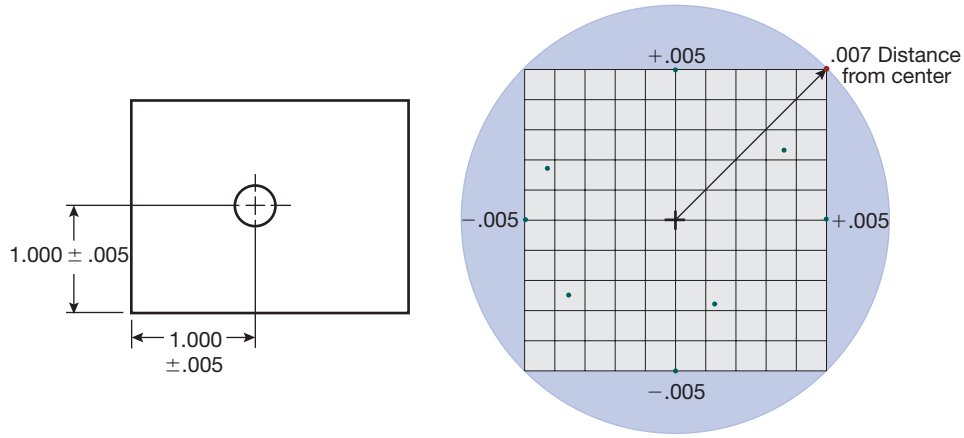


Figure A-67 This figure compares rectangular tolerances versus GD&T tolerances.

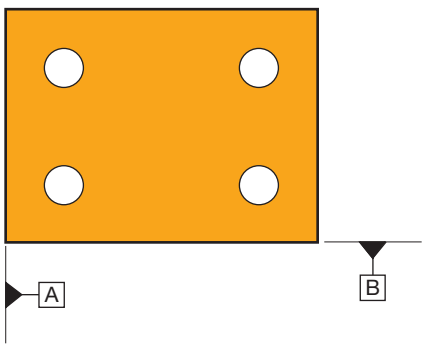


Figure A-68 Example of datum specification.

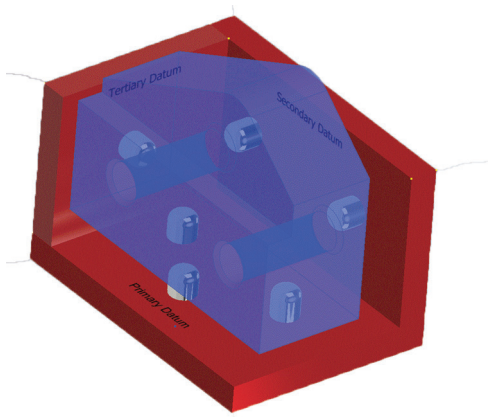


Figure A-69 Example of how part datums are established.

shape of a part feature. Basic dimensions have no tolerance. The tolerance comes from the associated geometric tolerances used with the basic dimension.

Primary, Secondary, and Tertiary Datum Planes

Figure A-69 shows how datums are established. Datum A is the primary, Datum B is the secondary, and Datum C is the tertiary datum. The primary datum (Datum A) is selected to provide functional relationships, standardizations, and repeatability between surfaces. Datum B is a secondary datum and is perpendicular to the primary datum so measurements can be referenced from them. Datum C, when needed, is perpendicular to both the primary and secondary datums ensuring the part's fixed position from three related datums. A datum does not have to be a surface. A datum can be a feature, such as the center of a bore.

After the primary datum is located using three points (see Figure A-69), the secondary datum will typically use two points to define a line. The third (tertiary) datum is defined by a single point.

Maximum Material Condition, Least Material Condition, and Regardless of Feature Size

Three material condition modifier symbols are shown in Figure A-70. These symbols identify at which size the geometric tolerance applies. MMC refers to the maximum amount of material remaining on a feature or size. On an external cylindrical feature this would be the high limit of the feature tolerance. For example, a shaft with a diameter of .750+ or −0.010 would have an MMC diameter of .760, since this would leave maximum material remaining on the part (largest diameter allowed for the shaft). An internal cylindrical feature such as a hole with a diameter of .750 Plus or minus 0.010 would have an MMC diameter of .740, since this would leave maximum material remaining on the part (smallest hole, most remaining material).

Least material condition (LMC) is the opposite. The LMC of the shaft would be .740 diameter, and the LMC of the hole would be .760 diameter. These sizes would have the least amount of material remaining on the part. RFS (regardless of feature size) means that the geometric tolerance applies no matter what the feature size is. The tolerance zone size remains the same, unlike MMC or LMC, which allows a tolerance zone size growth as the feature size changes. RFS is the default condition for all geometric tolerances. Unless

MMC or Ⓜ for Maximum Material Condition
LMC or Ⓛ for Least Material Condition
RFS or Ⓢ for Regardless of Feature Size

Figure A-70 Symbols for maximum material condition, least material condition, and regardless of feature size.

MMC or LMC is specified on the drawing by M or L, the tolerance zone size remains the same even though the feature size changes. Remember that the RFS symbol is very seldom used. If there is no MMC or LMC, RFS is implied. LMC is not used very much.

Limits of Size

Many different GD&T tolerances may be used to control a feature's position or location. No datum reference is needed for limits of size.

Feature Control Frames

Figure A-71 shows a GD&T feature control frame. The leftmost box would hold a geometric symbol that describes what type of tolerance this is. For example, it might be the symbol for perpendicularity. The next box would hold the tolerance zone size. For example, if the tolerance is .005, the tolerance zone would be .005 and the feature controlled by this specification would have to be perpendicular within the .005 tolerance zone. The third box would have a datum reference. This will all be clarified later in this section.

Figure A-72 shows an example of a feature control frame with three datums specified. The location control symbol is used in this example. The location of this feature would have to be within .005 at MMC in relation to Datum A, Datum B, and Datum C.

A sample drawing that includes a feature control frame (FCF) is shown in Figure A-73. In this example the flatness symbol is shown in the left rectangle of the FCF. The right box of the FCF has the tolerance for flatness for this example. The FCF denotes a flatness tolerance of .002s. Tolerance values in FCFs are a total tolerance, not a plus or minus value. In this example, you could think of it as a ±.001 tolerance of flatness. This flatness tolerance controls how flat this surface is.

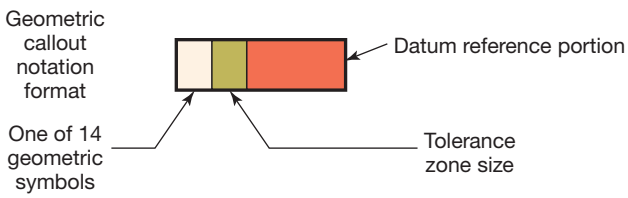


Figure A-71 A feature control frame.

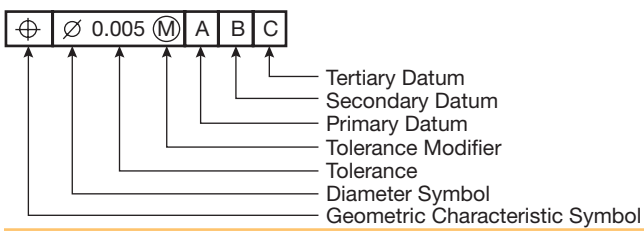


Figure A-72 Feature control frame with three datums specified.

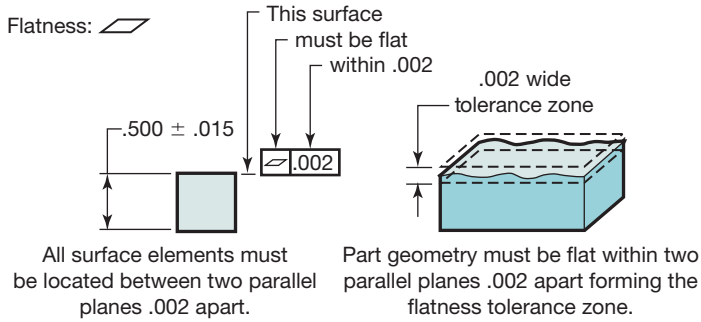


Figure A-73 Example of flatness specification.

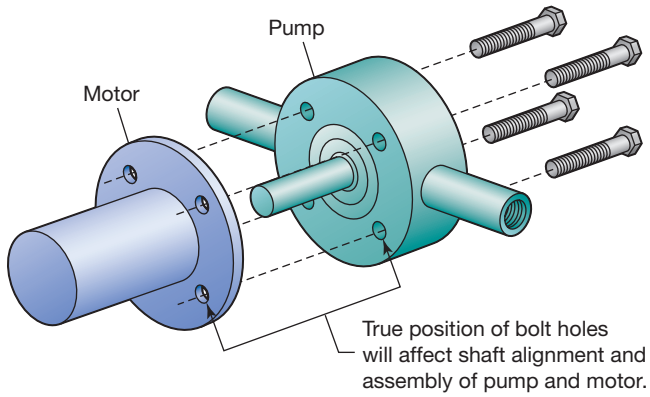


Figure A-74 Pump and motor assembly drawing illustrating the importance of true position.

True Position

Study the pump and motor shown in Figure A-74. For these to fit together, the holes in the pump and the threaded holes in the motor need to be accurately located.

SELF-TEST

1. How is it possible that GD&T can have larger tolerances than traditional rectangular tolerancing and still assure that the part will function as designed?
2. What is a datum?
3. How do you decide where a datum should be?
4. Describe how a part would be located based on primary, secondary and tertiary datums.
5. Describe the term MMC as it would apply to a shaft.
6. Describe the term MMC as it would apply to a bore.
7. Describe the term LMC as it would apply to a shaft.
8. Describe the term LMC as it would apply to a bore.
9. Write down the symbol for MMC and LMC.
10. What is RFS and why isn't it used much?
11. Draw a typical feature control frame for location with a primary datum, a secondary datum, a tolerance, and a MMC condition.

UNIT NINE

Geometric Tolerancing

The real power of GD&T is in its ability to describe the required relationships between features. For example, maybe the top and bottom of a part need to be parallel within .001. GD&T enables relationships like these to be specified. This unit will cover the ways in which GD&T can specify these relationships between features.

OBJECTIVES

After completing this unit, you should be able to:

- Explain terms such as form, profile, orientation, location, and runout.
- Evaluate feature control frames that specify control features.
- Apply MMC and LMC to feature control frame tolerances.
- Read blueprints that utilize GD&T feature control frames.

GEOMETRIC CONTROLS

Form tolerances are for individual features of a part. Figure A-75 shows the four forms of tolerance types and symbols.

Tolerance Type	Characteristic	Symbol
Form	Straightness	—
	Flatness	▱
	Circularity (Roundness)	○
	Cylindricity	⌭

Figure A-75 Forms of tolerance symbols.

Straightness

A straightness tolerance specifies a tolerance zone within which an axis or element must lie (Figure A-76). Straightness is a condition in which a feature is a straight line. Straightness is a two-dimensional geometric tolerance. Straightness is a condition where one-line element of a surface or an axis must lie in a straight line. It controls how much one line on a surface can deviate from a straight line. The feature control frame is attached to the surface with an extension line or a leader. Tolerance zone for both cylindrical and flat surfaces are applied along the entire surface. When straightness is inspected each line checked on the surface is independent.

Figure A-77 shows how the straightness tolerance is applied. In this example the tolerance for straightness is .005.

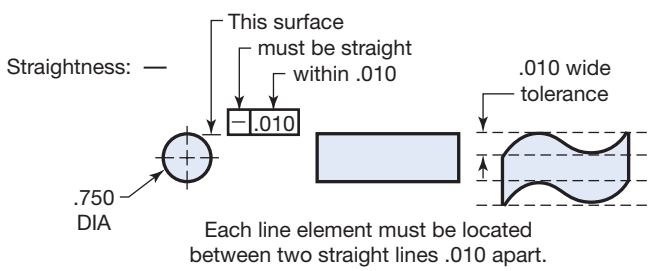


Figure A-76 Straightness specification example.

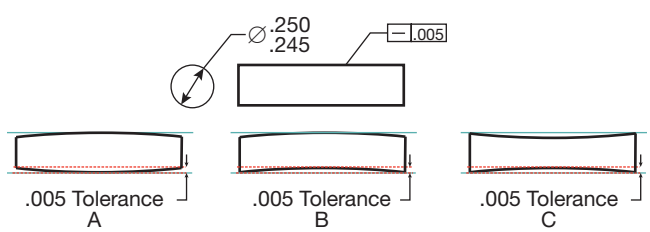
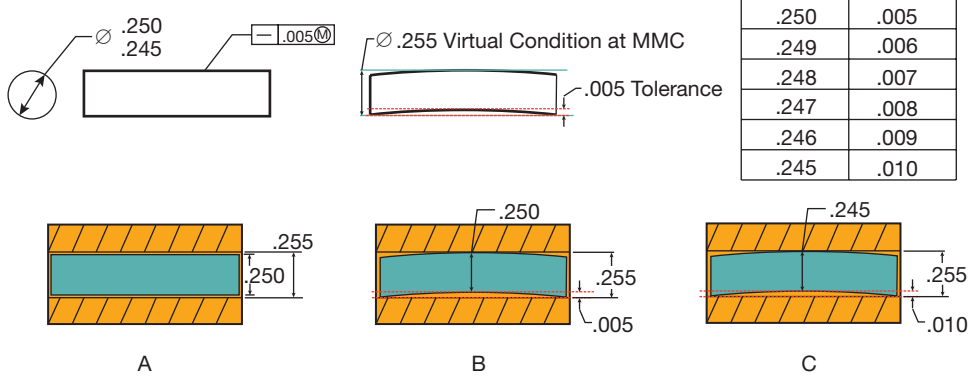


Figure A-77 Illustration of straightness tolerance.



Diameter	Tolerance
.250	.005
.249	.006
.248	.007
.247	.008
.246	.009
.245	.010

Figure A-78 Straightness tolerance with MMC.

There is no modifier (MMC or LMC) to the tolerance so the tolerance is regardless of feature size (RFS). The diameter of the part must be .245 to .250. Note examples A, B, and C in the figure. Note that the tolerance zone applies to individual lines on the part. Each line would be independent. Note also that the part can have different shapes as long as each line falls within the .005 straightness tolerance zone.

Figure A-78 shows how the straightness tolerance is applied. In this example the tolerance for straightness is .005. There is a maximum material condition (MMC) modifier to the tolerance. The diameter of the part must be .245 to .250. Note that the tolerance zone applies to individual lines on the part. Each line would be independent. Note also that the part can have different shapes as long as each line falls within the .005 straightness tolerance zone. In this example there is a MMC modifier so the tolerance zone changes depending on the diameter of the part. The table shows the size of the tolerance zone for each possible diameter. Note that if the part is at the maximum diameter of .250, the tolerance is the specified .005 (MMC). If the part diameter is at the smallest allowable size of .245, the tolerance zone is .010 (larger).

A gage could be used to inspect this part. The gage is illustrated in gray in A, B, and C in Figure A-78. Note that the bore though the gage is .255 in all three examples. In illustration A, the part diameter is .250 and the part appears to be straight. It fits in the gage with .005 clearance. In illustration B, the part diameter is .250 and we can see that the straightness tolerance allows the part to be bent (not straight) up to the .005 tolerance (part at MMC of .250). In illustration C, the part is at its smallest allowable size. In illustration C, the part can be bent (not straight) up to .010. This should make sense. If the part is smaller, it can be less straight and still fit into the gage.

Flatness

A flatness tolerance specifies a tolerance zone defined by two parallel planes within which all points on the surface must lie (Figure A-79). The tolerance is specified by a zone formed by two parallel planes.

A flatness tolerance does not have to be related to a datum. Flatness is applied to an individual surface. A feature control frame points to the surface with a leader line. When a feature control frame with a flatness tolerance is applied with a size dimension, the flatness tolerance applies to the median plane.

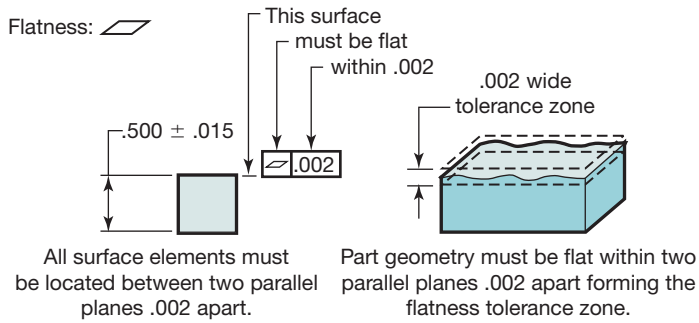


Figure A-79 Flatness specification example.

Roundness or Circularity

A roundness (circularity) tolerance specifies a tolerance zone bounded by two concentric circles within which each circular element of the surface must lie. Circularity tolerance is used to control the roundness of circular features. Circularity tolerance is applied to an individual surface. Circular features can be defined by cylinders, cones, or spheres.

Figure A-80 shows an example of a circularity tolerance. Note that the circularity (or roundness) symbol is

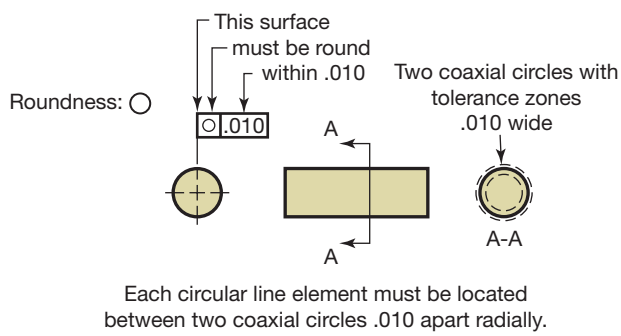


Figure A-80 Circularity specification example.

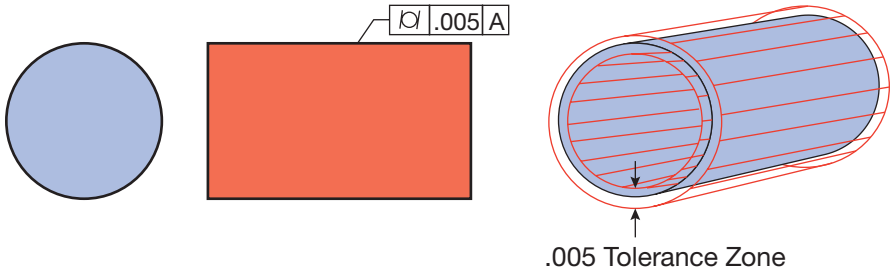


Figure A-81 Cylindricity specification example.

used in the feature control frame and a tolerance of .010 is specified. Section A-A shows the two concentric circles spaced .010 apart. All points on that circle on the part must fall within the zone shown by the two concentric circles. Note that more than one circle might be checked on a diameter, but all checks are independent for circularity.

Cylindricity

Figure A-81 shows the concept of cylindricity. The tolerance zone for cylindricity is the red inner and outer cylinder. All points on the surface of the part (gray) must fall within the inner and outer tolerance zones.

QUICK CHECK 1

1. Explain the form tolerance shown in Figure A-82.
2. Explain the form tolerance shown in Figure A-83.
3. Explain the difference between a flatness tolerance and a straightness form tolerance.
4. Explain the form tolerance shown and complete the tolerance table shown in Figure A-84.
5. Explain the form tolerance shown in Figure A-85.
6. Explain the form tolerance shown in Figure A-86.

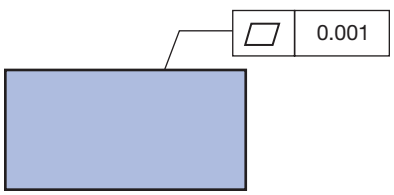


Figure A-82 Drawing for question 1.

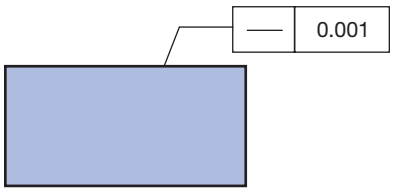


Figure A-83 Drawing for question 2.

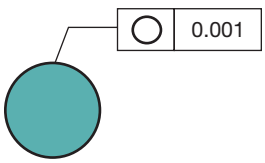


Figure A-85 Drawing for question 5.

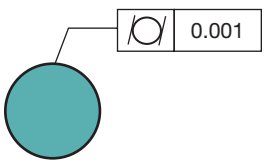


Figure A-86 Drawing for question 6.

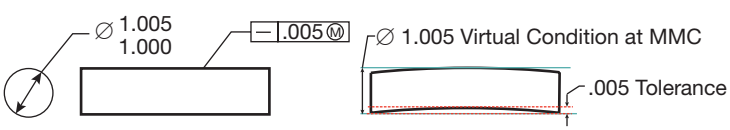
PROFILE TOLERANCES

A profile tolerance specifies a tolerance zone along the true profile within which all elements of the surface must lie. Figure A-87 shows the profile symbols.

Tolerance Type	Characteristic	Symbol
Profile	Profile of a Line	⌒
	Profile of a Surface	⌓

Figure A-87 Symbols for profile of a line and surface.

Figure A-84 Drawing for question 4.



Diameter	Tolerance

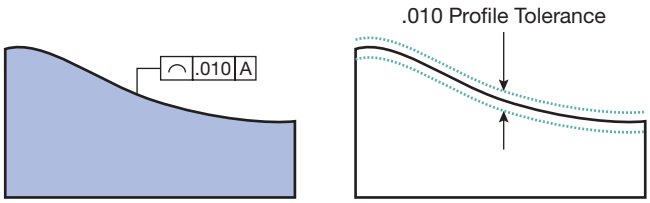


Figure A-88 Line profile specification example.

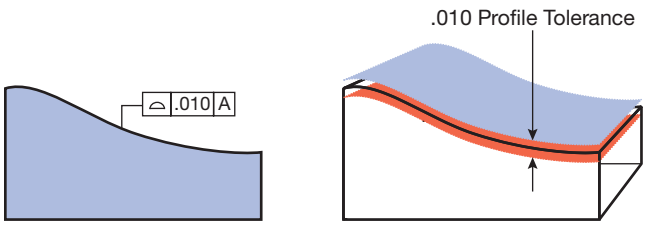


Figure A-89 Surface profile specification example.

Profile of a Line

The profile of a line describes the tolerance zone around a line, usually of a curved shape. Profile of a line is a tolerance zone that can be applied to any linear tolerance. A profile of a line takes a cross section at any point along the surface and sets a tolerance zone on either side of the specified profile (Figure A-88). This would be inspected by checking lines on the surface to make sure all points on the line were within the tolerance zone. With a line profile, the tolerance applies to each checked line independently. The points on one line are not compared with the points on a different line.

Profile of a Surface

Figure A-89 shows the tolerance zone for a surface. The tolerance is .010. The tolerance zone is illustrated on the right of the figure. In a surface profile all points on the surface must fall within the tolerance zone.

QUICK CHECK 2

1. Explain the profile tolerance shown in Figure A-90.
2. Explain the profile tolerance shown in Figure A-91.

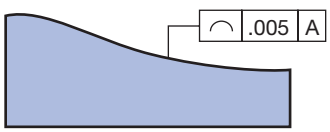


Figure A-90 Drawing for question 1.

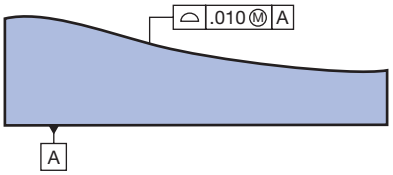


Figure A-91 Drawing for question 2.

ORIENTATION, LOCATION, AND RUNOUT TOLERANCING

Orientation, location, and runout tolerances are for related features of a part. Figure A-92 shows the symbols for orientation, location, and runout.

Angularity

The angularity tolerance symbol is shown as "∠." An angularity tolerance specifies a tolerance zone defined by two parallel planes at the specified angle. In this example, the top surface must be at a 15-degree angle in relation to the bottom (Datum A) within a tolerance zone of .010. LMC or MMC can apply to features of size (Figure A-93).

Perpendicularity

The perpendicularity tolerance symbol is "⊥." A perpendicularity tolerance is a three-dimensional geometric tolerance that controls how much a surface, axis, or plane can

Tolerance Type	Characteristic	Symbol
Orientation	Angularity	∠
	Perpendicularity	⊥
	Parallelism	//
Location	Position	⌖
	Concentricity	◎
	Symmetry	⌯
Runout	Circular Runout	↗
	Total Runout	⌰

Figure A-92 Orientation, location, and runout symbols.

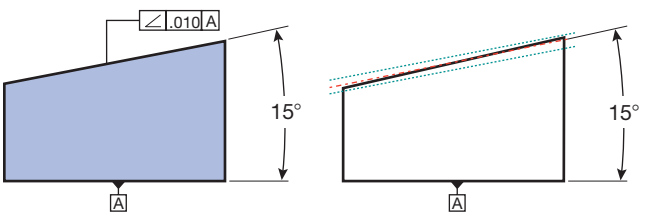


Figure A-93 Angularity specification example.

deviate from a 90-degree angle. A perpendicularity tolerance zone can be defined by two parallel planes or a cylindrical zone perpendicular to a datum. LMC or MMC can apply to features of size. Figure A-94 shows an example of a pin that must be perpendicular within .005 to a surface (Datum A). Note the cylindrical tolerance zone.

Figure A-95 shows another example of perpendicularity. The left side of the figure shows the perpendicularity specification for the surface. The right side of the figure shows the .010 MMC tolerance. The tolerance at MMC is .010. If the width is 1.105, the tolerance zone will be .010. The table in the figure shows the tolerance zone for different size widths. Note that the smaller the width is the larger the tolerance zone becomes.

Parallelism

The parallelism symbol is used in the feature control frame in Figure A-96. A parallelism tolerance zone is the condition such that a surface is equidistant at all points from a datum plane, or an axis. The distance between the parallel lines, or surfaces, is specified by the geometric tolerance zone. In this example all the points in the top surface must be within a .002 tolerance zone that is parallel to the bottom surface (Datum B). LMC or MMC can apply to features of size.

Figure A-97 shows an example of a parallelism specification for a bored hole. In this example the hole must be parallel to surface Datum A within .002. Note the tolerance zone in the figure.

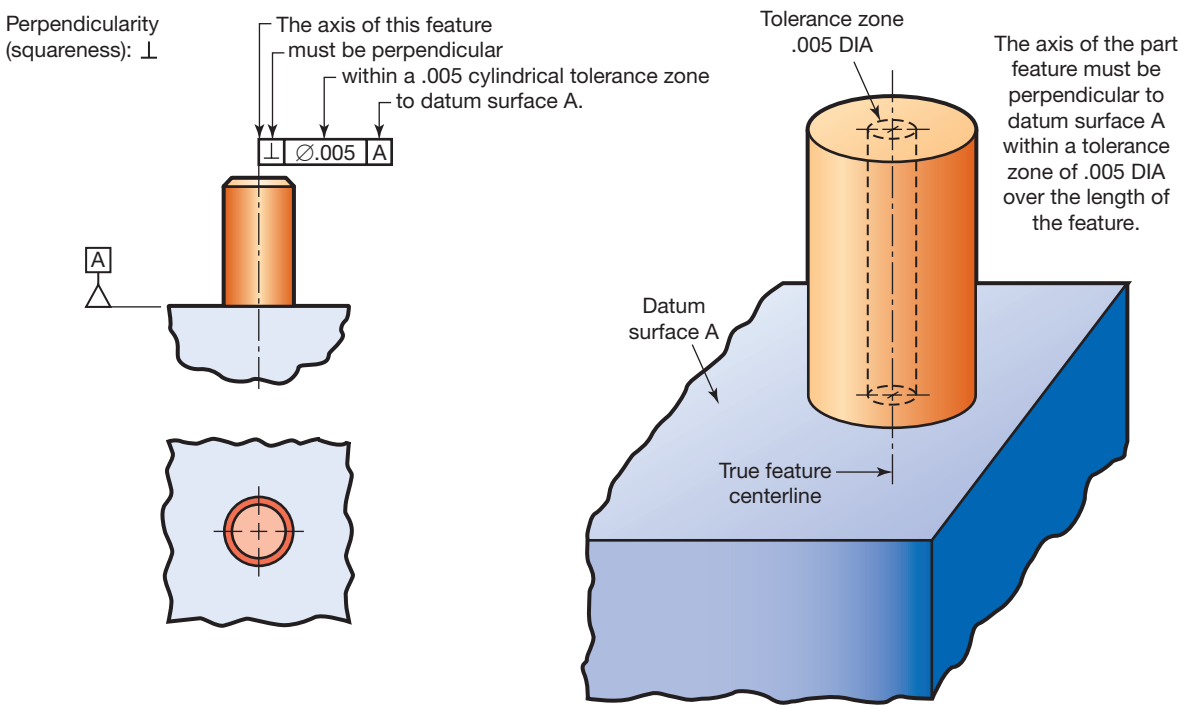
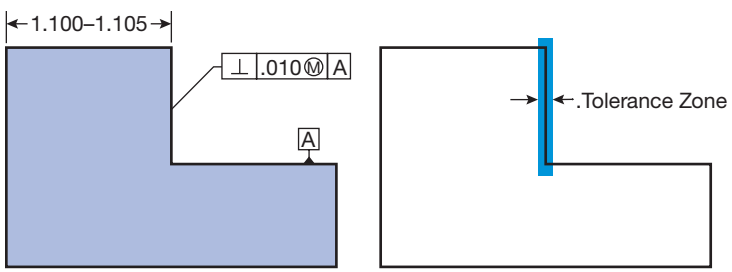


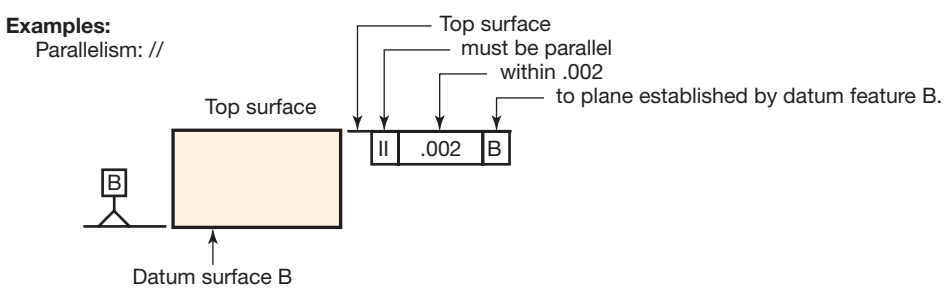
Figure A-94 Perpendicularity example.

Figure A-95 Example of perpendicularity of a surface.



Diameter	Tolerance
1.105	.010
1.104	.011
1.103	.012
1.102	.013
1.101	.014
1.100	.015

Figure A-96 Parallelism example.



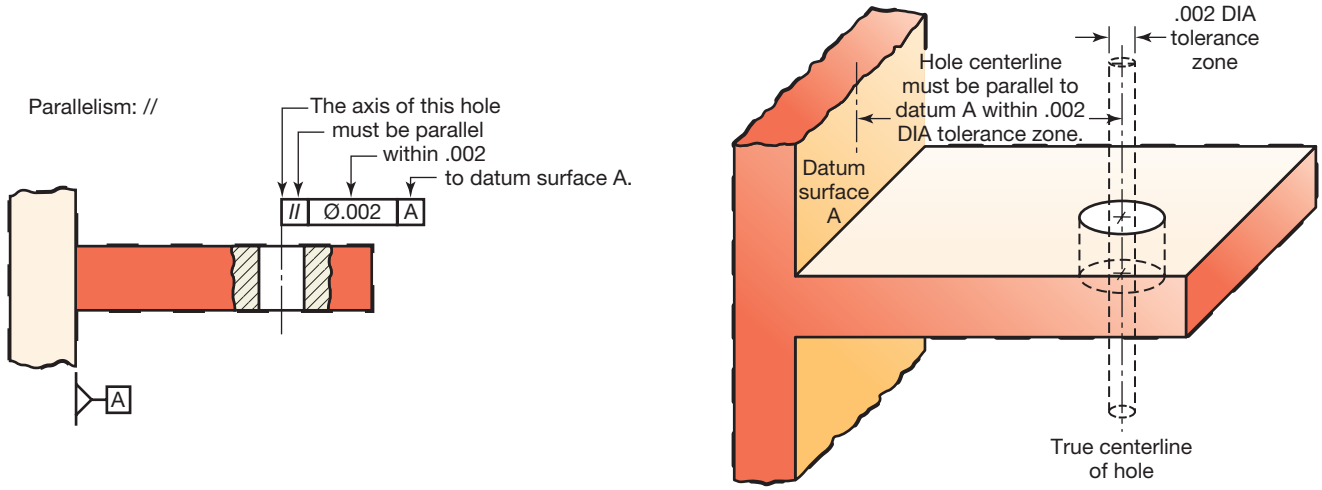


Figure A-97 Parallelism example for a bored hole.

QUICK CHECK 3

1. List the three types of orientation tolerances.
2. Explain the orientation tolerance and complete the tolerance table in Figure A-98.

CONTROLLING THE LOCATION OF FEATURES

Figure A-77 shows an exploded view of a motor and pump that are bolted together. The pattern of bolt holes in the pump must match the pattern of bolt holes in the motor for them to fit together. Also, the bore in the pump housing must match the boss on the motor so that the shaft and the motor will align. If the position of either bolt pattern deviates far from the specified dimensions, the parts will not fit together. This is an example of a situation in which the location or the true position of the holes could be more critical than the size of the holes themselves.

Location Tolerances

Location tolerances can be stated by three tolerance zones. These are position, concentricity, and symmetry tolerances (Figure A-99). Concentricity and symmetry are used to control the center distance of feature elements. Position is used to control coaxial features, the center distance between features, and the location of features as a group. Position, concentricity, and symmetry tolerances are associated with datums. LMC or MMC can apply to features of size.

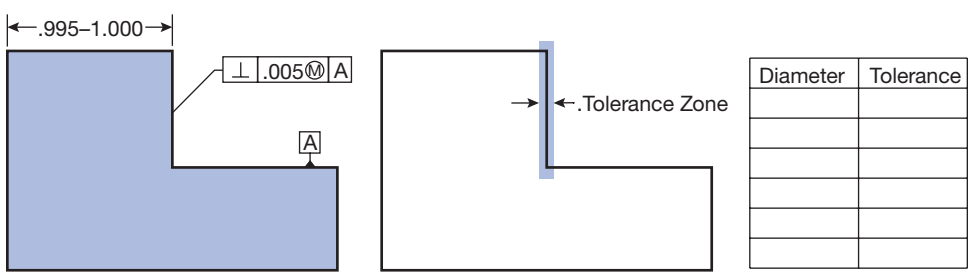
Position Tolerance Position tolerances control how much the location of a feature can vary from its true position. Positional tolerances are used to locate features of size from datum planes. A position tolerance is the total permissible variation in the location of a feature about its exact true position. For every part feature there is an exact position (true position) where the feature would be if it was perfect. A tolerance zone allows deviation from perfection. The exact, or true, position of the feature is indicated by its basic dimensions.

For cylindrical features, the position tolerance zone is typically a cylinder within which the axis of the feature must lie (Figure A-100). The tolerance defines a zone that the axis or center plane of a feature of size may vary within. LMC or MMC can apply to features of size.

Tolerance Type	Characteristic	Symbol
Location	Position	⌖
	Concentricity	◎
	Symmetry	⌯

Figure A-99 Location tolerance symbols.

Figure A-98 Drawing for question 2.



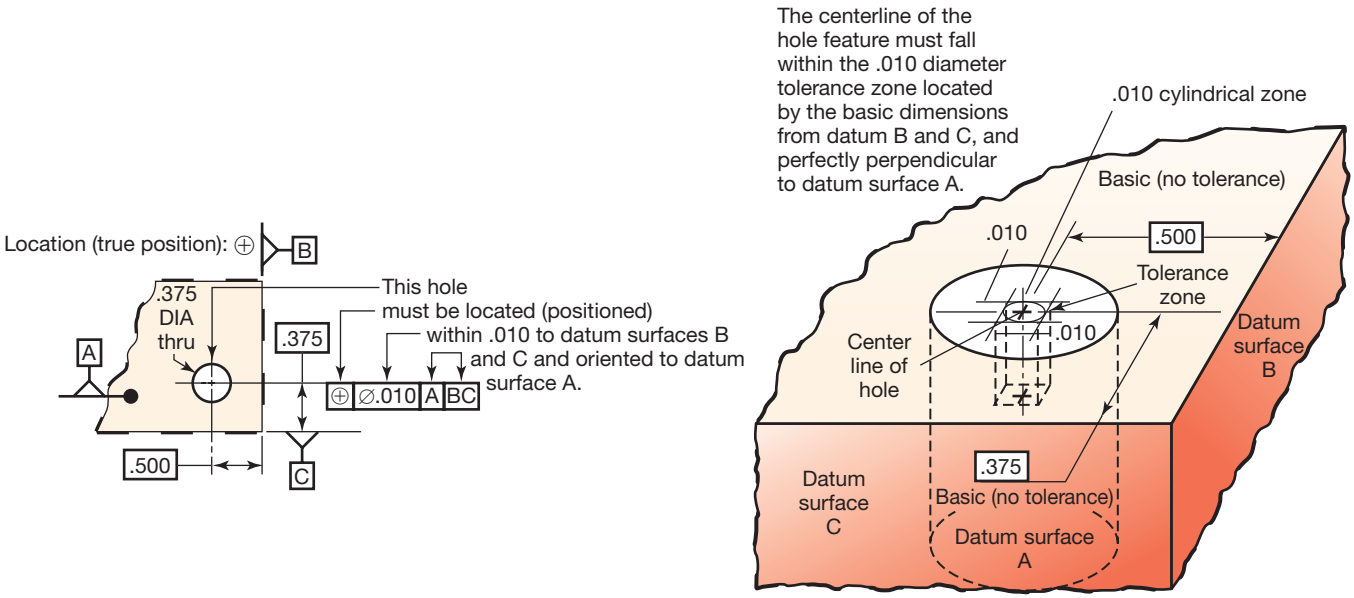


Figure A-100 Position (location) tolerance.

True Position Using MMC True position controls where the datum locations need to be. MMC specifies a minimum size and positional location of the hole. It does this by allowing a bonus tolerance to be added to the feature location tolerance. As a hole size gets closer to the MMC, the hole must be closer to its specified location. If the hole is larger, it can vary from its true position more and still function as designed.

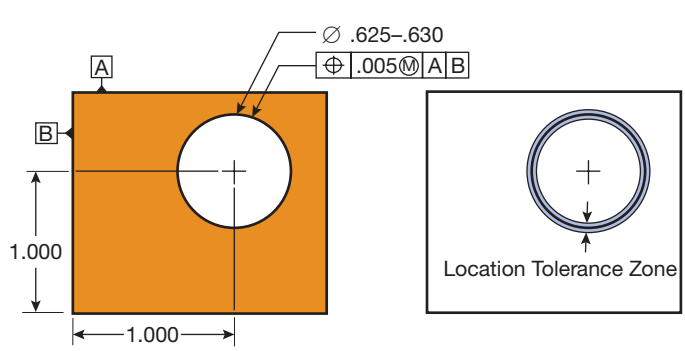
Consider the bored hole in Figure A-101. The diameter of the hole must be between .625 and .630. The location of the bore is 1.000 from the left edge and 1.000 from the bottom edge of the part. There is also a GD&T specification for the hole. The feature control symbol is a location symbol. The feature control frame specifies a location tolerance of .005 at MMC. MMC for a hole is the smallest hole size allowed (most material left). The MMC specification adds a bonus tolerance to location based on the hole size. If the hole is larger than minimum size, the location tolerance increases. Study the table in Figure A-101. If the hole size is .625 (smallest allowed) the location tolerance is .005. If the hole is .001 larger (.626), the location tolerance increases to .006. If the hole is at its largest allowed size the location tolerance is .010. Basically, this means that if the hole size is larger (but within specifications) the location can vary more and the part will still be functional.

This adds some complexity to the inspection of parts. In industry a company may require first piece inspection on a coordinate measuring machine or measurement arm. If the parts are made in high quantities, the company might have a gage made to inspect parts.

Concentricity A concentricity tolerance specifies a cylindrical tolerance zone whose axis coincides with the datum axis. A concentricity tolerance indicates that a cylinder, cone, hex, square, or surface of revolution shares a common axis with a datum feature. It controls the location for the axis of the indicated feature within a cylindrical tolerance zone whose axis coincides with the datum axis.

An example of a concentricity tolerance is shown in Figure A-102. The larger diameter of the part is specified to be Datum A. The smaller diameter must be concentric with Datum A within .005. Note that there was no modifier so the tolerance is regardless of feature size (RFS). Note that the tolerance zone is cylindrical around the centerline of Datum A. Runout is easier to inspect and is used more often than concentricity.

Figure A-101 Use of MMC to control a hole location.



Diameter	Tolerance
.625	.005
.626	.006
.627	.007
.628	.008
.629	.009
.630	.010

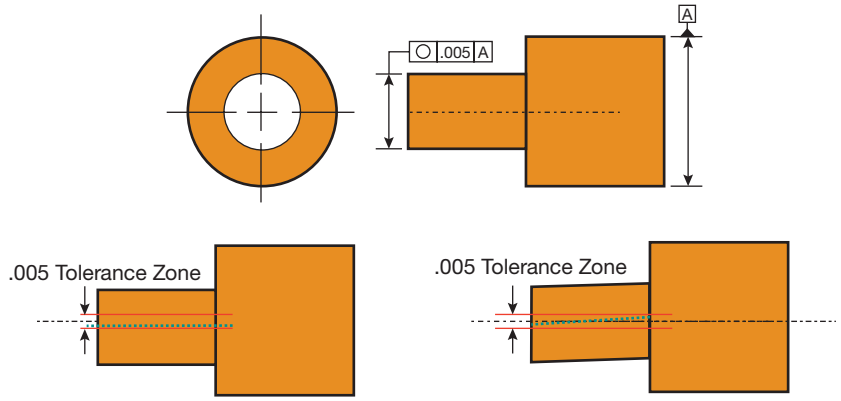


Figure A-102 Concentricity example.

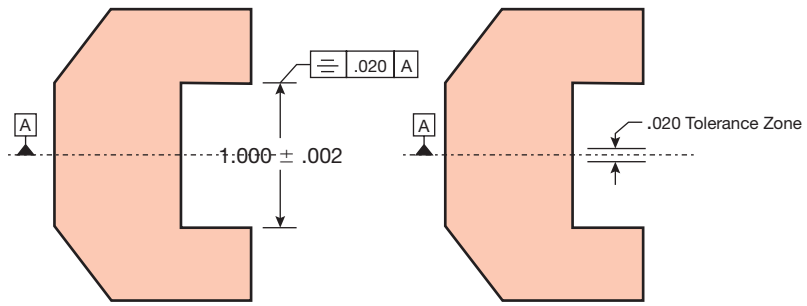


Figure A-103 Symmetry specification example.

Symmetry A symmetry tolerance controls the median points of a feature of size. Symmetry tolerances are applied to non-circular features. A symmetry tolerance is a three-dimensional geometric tolerance that controls how much the median points between two features may deviate from a specified center plane or axis. It is generally recommended to use position, parallelism, or straightness instead of a symmetry tolerance, when possible.

An example of a symmetry tolerance is shown in Figure A-103. The left half of the figure shows the symmetry symbol applied to a slot in the part. The symmetry symbol in this example specifies that the median points of the opposing surfaces of the slot must be symmetric about Datum A within a tolerance zone of .020. In this example, the slot width must be 1.000+ or −.002 and it must be centered around Datum A within a tolerance zone of .020.

QUICK CHECK 4

1. Explain the location tolerance and complete the tolerance table in Figure A-104.
2. Complete the feature control frame for a location tolerance. The tolerance should be related to datum A and B and should be at maximum material condition. Complete the tolerance table shown in Figure A-105.
3. Explain what a concentricity tolerance is used for.

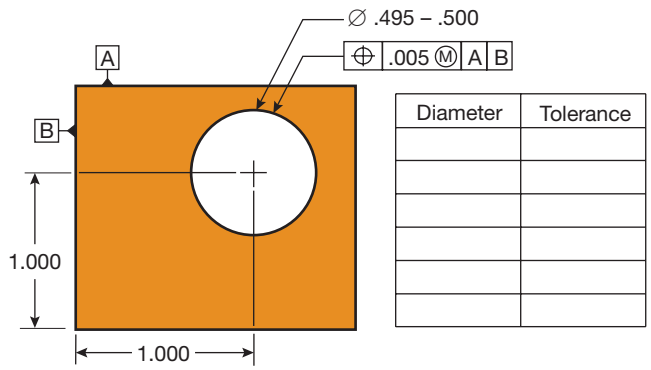


Figure A-104 Drawing for question 1.

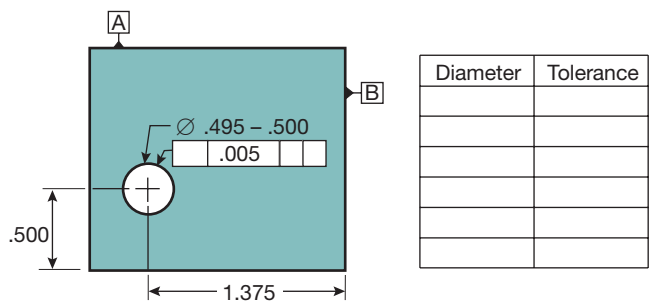


Figure A-105 Drawing for question 2.

Runout

Runout is how much one feature varies with respect to a datum when the part is rotated 360° around the datum axis. It is essentially a control of a circular feature, and how much variation it has with the rotational axis. Runout can be called out on any feature that is rotated about an axis. There are two symbols used for runout. They are shown in Figure A-106.

Circular Runout Circular runout is used to provide control of the circular elements of a surface (Figure A-107). The tolerance is applied independently at any circular measuring position as the part is rotated 360 degrees. One way this part could be inspected would be to put the large diameter in a V-block and an indicator at one point on the small diameter (Figure A-108). The part would then be rotated and the smallest and largest readings on the indicator would be used to determine the runout. More than one circle can be checked, but the checks are independent. When applied to surfaces at right angles to the datum axis, it controls the circular elements of the plane surface (Figure A-109).

Tolerance Type	Characteristic	Symbol
Runout	Circular Runout	↗
	Total Runout	⌰

Figure A-106 Runout symbols.

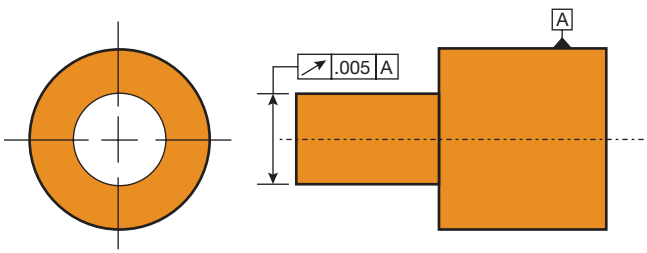


Figure A-107 Circular runout specification.

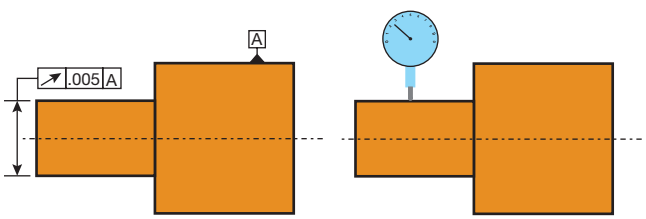


Figure A-108 Method to check runout. Note that the diameter specified by Datum A would be put in a precision V-block and the part would be rotated.

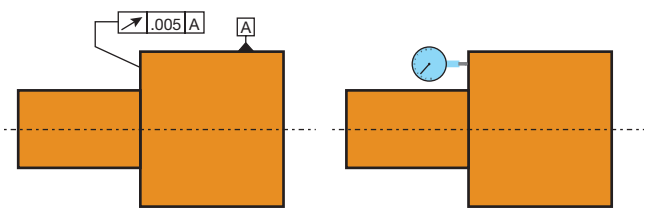


Figure A-109 Inspecting for runout on a surface.

Total Runout Total runout is very similar to circular runout except that total runout applies to the whole controlled feature, not just one circle. See Figure A-110. The GD&T symbol for total runout is used. This means that the whole length of the small diameter would need to be within .005 when checked. To inspect this part the operator would need to check several circles along the whole length of the small diameter (see Figure A-111). All points of the checks would need to have .005 or less runout.

Runout and total runout can also be on a surface of a cylindrical part. In the example shown in Figure A-111 the surface must have runout of less than .005 in relation to Datum A.

SELF-TEST

1. Explain a circular runout tolerance.
2. How is circular runout checked?
3. How is total runout checked?
4. Thoroughly explain the numbered features and tolerances (red 1-5) shown on the print in Figure A-112.
5. Thoroughly explain the numbered features and tolerances (red 1-5) shown on the print in Figure A-113. Complete the tolerance table.

Size	Tolerance
.203	
.202	
.200	
.201	
.200	
.199	
.198	
.197	

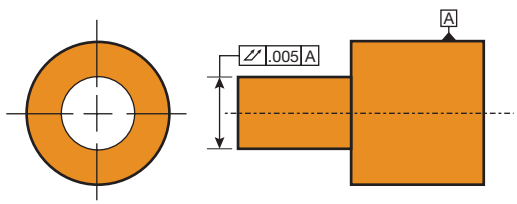


Figure A-110 Total runout specification.

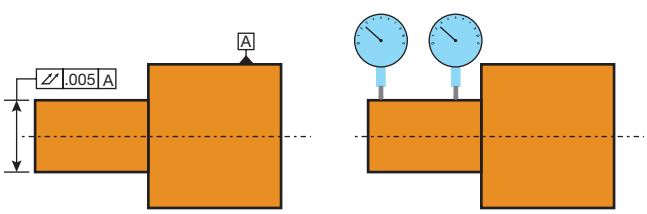


Figure A-111 Inspecting total runout.

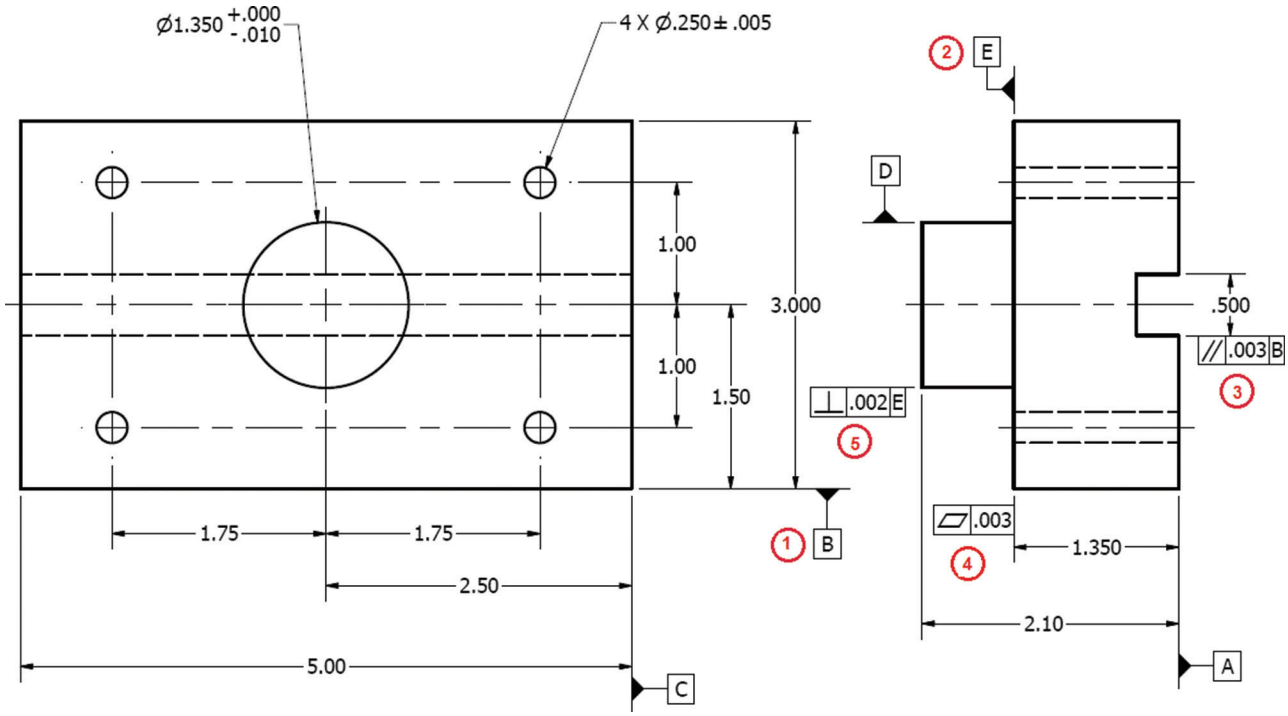


Figure A-112 Mill part.

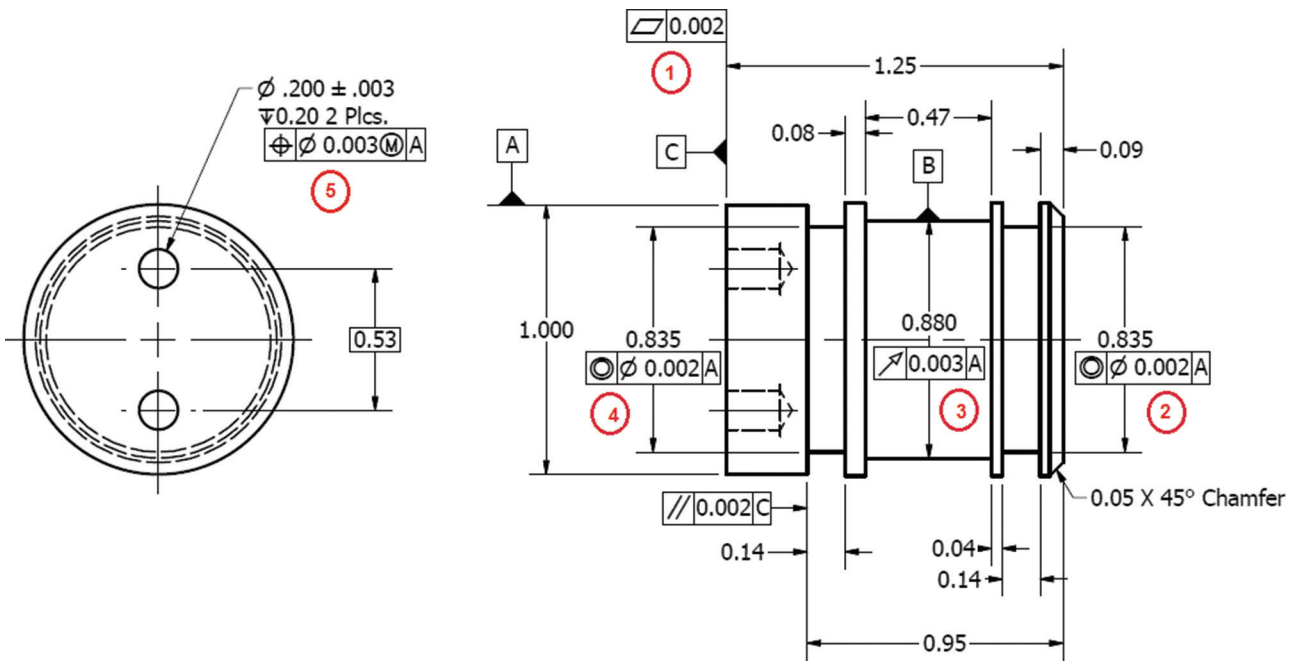


Figure A-113 Lathe part.

6. Explain the following based on the blueprint in Figure A-114.

1.
2.
3.
4.
5.
6.
7.
8.

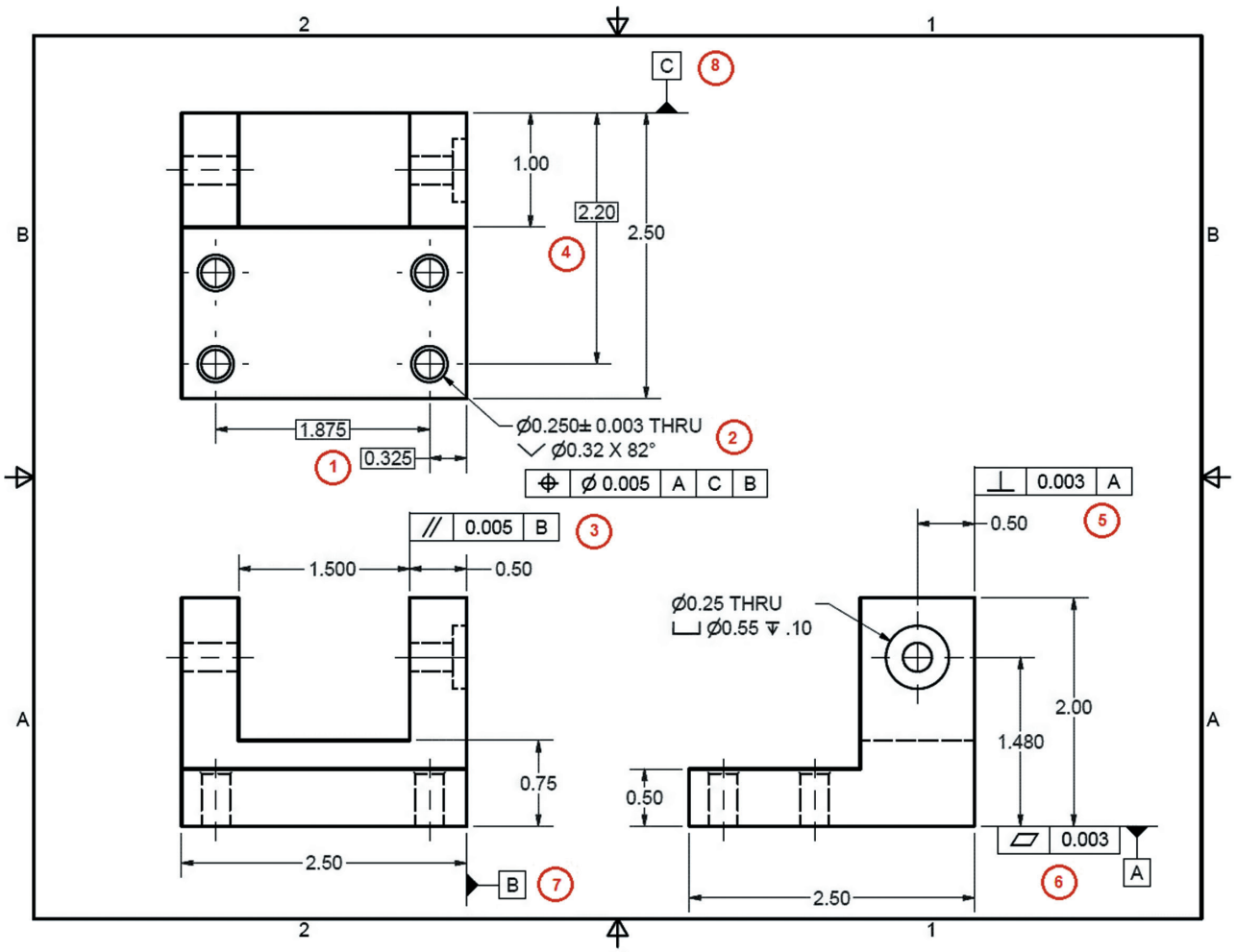


Figure A-114 Drawing for question 6.