BPR – 111, Print Reading

Orientation and Introduction

Introduction

Concept Content:

In this section you will give an introduction of yourself to your class. This is an opportunity to state your relevant experiences and credentials to teach this subject along with your personal background. This can help connect with students. You can make a video introduction and upload it to this page as well.

Also, this is where you will give a brief overview of the course and what its contents will be. There is a section later in this module where you will give more detail about the course.

Course Syllabus

Concept Goals:

Insert the student learning outcomes for the course here.

Concept Content:

This is where you will upload the syllabus. You can do this either by uploading the syllabus text here or you can upload a copy of the syllabus under the resources tab for this section. If you do upload it to the resources, please be sure to give instructions to your students to look for the syllabus there.

Course Resources

Concept Goals:

You can leave this section blank provided you uploaded the student learning outcomes to the previous section.

Concept Content:

This is where you would outline student support resources such as tutoring services, listing your office hours, contact info for support for your college's learning management system, etc. If there are documents you wish to upload, be sure to upload them to the resources tab and give instructions for the students to find the documents there.

Course Overview

Concept Goals:

Student Learning Outcomes
1. Interpret symbols, abbreviations, and line types.
2. Identify and describe types of projection and use of views.
3. Draw freehand sketches.
4. Calculate measurements of features.
5. Identify and interpret dimensioning and tolerancing.

Concept Content:

Course Overview

This course introduces the basic principles of print reading. Topics include line types, orthographic projections, dimensioning methods, and notes. Upon completion, students should be able to interpret basic prints and visualize the features of a part or system.

Module	Module Learning Objectives
Module 1: Introduction	• Identify the types of prints (SLO 1) • Recognize how prints are produced (SLO 1) • Explain the importance of prints (SLO 1) • Define the types of lines in a print (SLO 1)
Module 2: Sketching and Views	• Learn basic sketching techniques. (SLO 3) • Become familiar with isometric sketches. (SLO 3) • Become familiar with orthographic sketches. (SLO 3) • Become familiar with projection views. (SLO 3)
Module 3: Scales and Measurement	• Identify various measuring tools (SLO 4) • Become familiar with scale in sketching (SLO 3) • Review basic math (SLO 4)

Module 4: Blueprint Reading - Lines and Letterings	• Identify the different types of blueprint lines (SLO 1) • Know the functions of the different types of blueprint lines (SLO 1) • Know how to draw the different types of blueprint lines (SLO 3)
Module 5: Reading Multi-View Drawings	• Know how to read a multi-view drawing (SLO 1, SLO 2)
Module 6: Reading Dimensions	• Understand numerals and other dimension figures (SLO 1) • Understand isometric dimensioning (SLO 5) • Understand orthographic dimensioning (SLO 5)
Module 7: Mid-Term Exam	• Demonstrate an understanding of the first six weeks' material
Module 8: Manufacturing Process	• Become familiar with manufacturing processes (SLO 1) • Be able to read blueprints to decipher what manufacturing processes are called for (SLO 1)
Module 9: Fasteners and Springs	• Learn the parts of an external thread (SLO 4) • Learn thread classifications in both standard and metric measurements (SLO 4) • Learn about thread milling (SLO 4) • Begin learning about torque (SLO 4)
Module 10: Working Drawings	• Become familiar with the types of working drawings. (SLO 2, SLO 3)
Module 11: Pictorial Drawings	• Identify and describe each of the members of the axonometric family of pictorial views. (SLO 2) • Identify and describe each of the members of the oblique family of pictorial views. (SLO 2) • Understand how to read pictorial drawings. (SLO 2)
Module 12: Surfacing Finishes	• Learn the purpose of surface finish. (SLO 1) • Recognize and understand surface finish symbols. (SLO 1)
Module 13: Geometric Dimensions and Tolerancing (GD&T)	• Learn the definitions of GD&T (SLO 5) • Become familiar with the basic concepts of GD&T (SLO 5) • Understand why GD&T is vital for engineering and machining (SLO 5)
Module 14: More GD&T	• Learn the GD&T symbols for angularity, parallelism, and perpendicularity (SLO 5) • Understand what angularity, parallelism, and perpendicularity are used for in GD&T (SLO 5)
Module 15: Final Exam	• Demonstrate an understanding of the course material

Instructor Note: This is a 15 week course. If you need a 16th week due to your semesters being 16 weeks, there are additional materials related to GD&T located in the instructor resources module.

Notes/Helpful Tips

Next Steps…

Your Census assignments are REQUIRED in order to remain in the class and they MUST be completed prior to the Census Date **[insert census date here]**. **If you do not have a census date requirement, you can delete this section.**

Effective note taking is also important for not only this course, but for your career as well. Note taking is a great way to retain information. The process of taking notes can keep you alert and focused on the information being presented. It also keeps your mind engaged with what you are hearing, increasing the likelihood you will retain that information. Note taking can also allow you to better organize your thoughts on the information being discussed.

Here is a video that provides some tips for effective note taking.

Module 1 - Introduction

1.1 Module Overview

Concept Goals:

By the end of this module the student should know how to:

- Identify the types of prints (SLO 1)
- Recognize how prints are produced (SLO 1)
- Explain the importance of prints (SLO 1)
- Define the types of lines in a print (SLO 1)

Concept Content:

This section begins our overview into blueprint reading. During this module, we will go over some basics of how to read blueprints, a brief history of blueprint drawing, and the topics outlined in the student learning outcomes above.

This week at a glance:

Lectures:

Blueprint Reading Intro PowerPoint - 35 slides

Videos:

Intro to Blueprint Reading- 24 Minutes

Reading:

Chapter 2 of Basic Blueprint Reading - Lines - Pages 1-7

Assignments:

Module 1 Worksheet

Types of Drawings Quiz - 5 Questions

1.2 Module Content Resources

Concept Content:

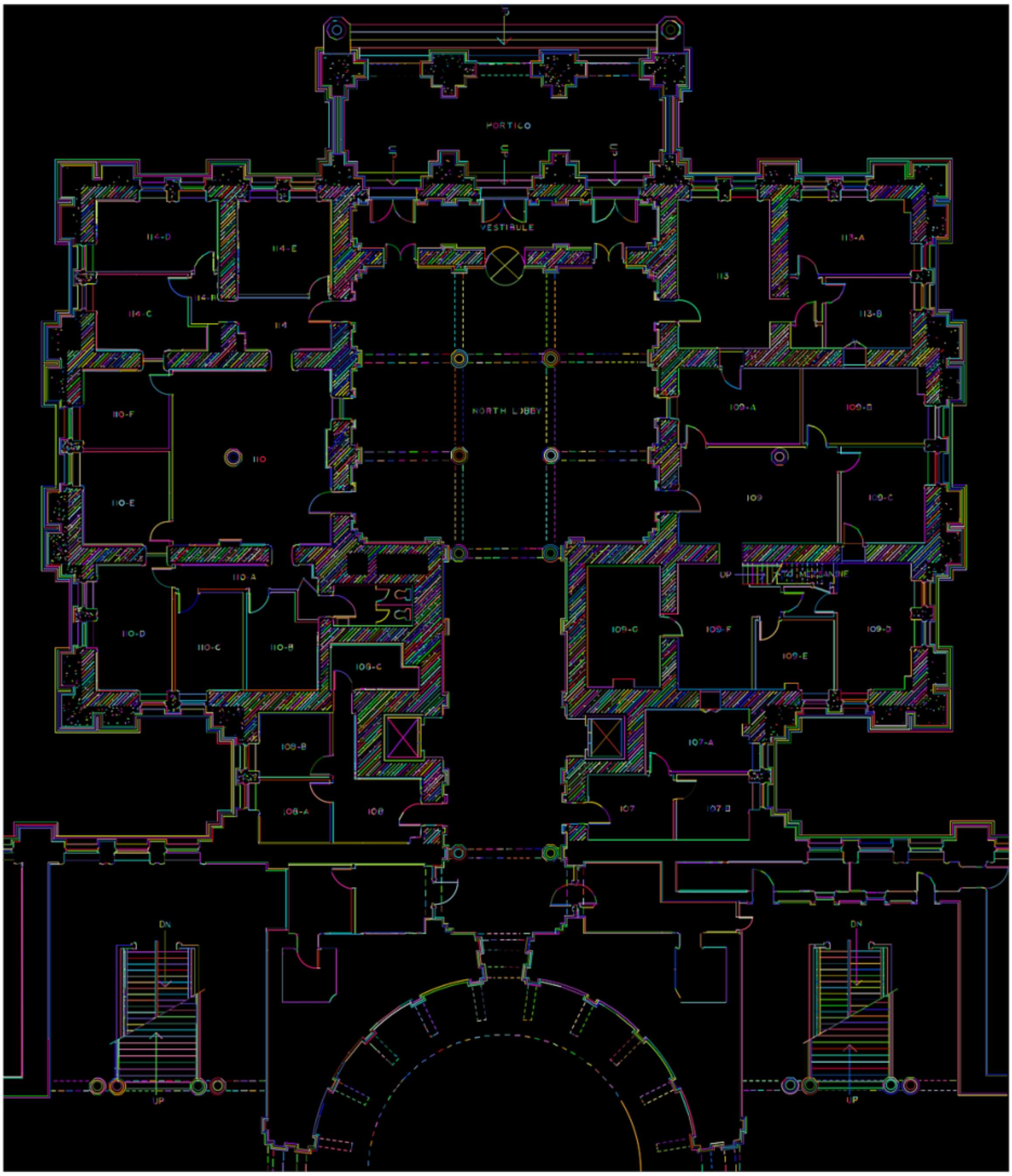


Figure 1: Floor Plan Layout. Image by [Gordon Johnson](#) from [Pixabay](#)

Look at the image above. This is a blueprint. A very complicated blueprint, but a good example of one. Please try to analyze the blueprint and see if you can make sense of it.

While we will not be working with construction blueprints, the principles that guide drafting construction prints apply to the parts prints you will run across in the machine shop. While they might not get as complicated as the above drawing, machine part blueprints can be complex. It is important to be able to understand the information they relay so that you can run the machine correctly when it comes time to make them. By the end of this course, you should have a basic understanding of how blueprint reading works. This module will introduce you to the course, give you a basic overview of blueprint reading, and give you some vocabulary words to study and remember.

Lectures:

Blueprint Reading Intro PowerPoint - 35 slides - Be sure to read the notes under each slide.

Videos:

Intro to Blueprint Reading- 24 Minutes

Reading:

Chapter 2 of Basic Blueprint Reading - Pages 1-7

Vocabulary Terms to Remember:

The types of lines outlined in the textbook pages linked above.

As you take notes for this week's module, please pay special attention to the various types of lines in a blueprint. Knowing the difference between the lines is crucial for understanding how to read a blueprint. Knowing the various types of drawings is also important.

1.3 Module Assessment/Assignment

Concept Content:

This week we will have two assignments:

Module 1 Worksheet - Located in the Resources Tab. Download and complete the worksheet. Upload to the Assignments tab under quizzes.

Types of Drawings Quiz - Questions 2-6 under quizzes.

1.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

1.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

1.6 Module Wrap-Up

Concept Goals:

By the end of this module the student should know how to:

- Identify the types of prints
- Recognize how prints are produced
- Explain the importance of prints
- Define the types of lines in a print

Concept Content:

This first week we discussed the basics of blueprint reading. Topics covered included the above learning outcomes as well as going over the class rules. This is the beginning of building a foundational skill that will be critical to your success in the industry. Be sure to have written down your notes and keep them for later review.

A look into what this week entailed.

Lectures:

Blueprint Reading Intro PowerPoint - 35 slides

Videos:

Intro to Blueprint Reading- 24 Minutes

Reading:

Chapter 2 of Basic Blueprint Reading - Pages 1-7

Assignments:

Module 1 Worksheet

Types of Drawings Quiz - 5 Questions

Be sure to have completed the assignments by **[insert due date here].**

Module 2 - Sketching and Views

2.1 Module Overview

Concept Goals:

By the end of this moudle you should:

- Learn basic sketching techniques. (SLO 3)
- Become familiar with isometric sketches. (SLO 3)
- Become familiar with orthographic sketches. (SLO 3)
- Become familiar with projection views. (SLO 3)

Concept Content:

This week we will go over basic sketching and drawing views. Content will cover the learning objectives listed above and more. Knowing how to sketch is an important skill for a machinist.

This week at a glance:

Lectures:

Basic Sketching Presentation - 38 Slides

Videos:

Orthographic Projection from Isometric View in Engineering Drawing - 9 Minutes

Isometric Projection in Engineering Drawing - 7 Minutes

Reading:

Technical Sketching Chapter - 21 Pages

Blueprint Views Chapter - 4 pages

Assignments:

Practice Drawing - Outlined in module 2.3

Module 2 Quiz - 3 Questions

2.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

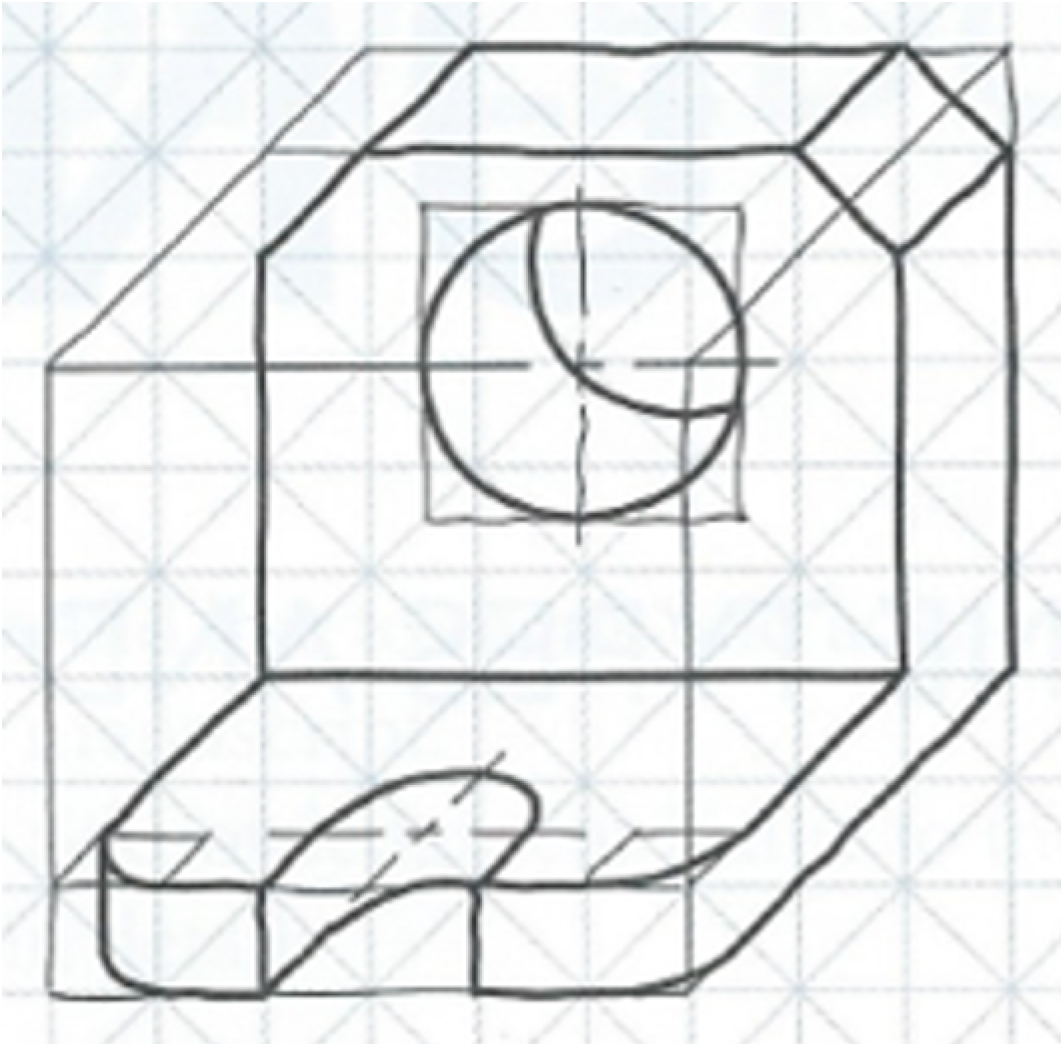


Figure 1: An example of a part sketch

To warm our brains up this week, take a look at the image above. Without looking through this week's material, that type of sketch do you think this is? Take a moment and guess?

Did you guess that it was an orthographic sketch? If so, you are correct.

This week we will go into an overview of basic sketching. This will include sketching techniques and knowing the difference between orthographic and isometric projections.

Sketching is an important practice as it is how you will create your own blueprints. This skill is essential in order to do well in machining.

Lectures:

Basic Sketching Presentation - 38 Slides

Videos:

Orthographic Projection from Isometric View in Engineering Drawing - 9 Minutes

Isometric Projection in Engineering Drawing - 7 Minutes

Reading:

Technical Sketching Chapter - 21 Pages

Blueprint Views Chapter - 4 pages

As you take notes for this week, be sure to pay special attention to the various sketching techniques and what the definitions of isometric sketching and orthographic sketching are.

2.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week's assignments:

Practice Drawing - Make an isometric view of the capital letters E, F, H, I, O, and Z. Make them approximately 3" high and 2" wide and 1" deep. Upload your drawings under quizzes for the first question.

Module 2 Quiz - Also located under quizzes - 3 Questions

2.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

2.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

2.6 Module Wrap-Up

Concept Goals:

Module Learning Outcomes

- Learn basic sketching techniques. (SLO 3)
- Become familiar with isometric sketches. (SLO 3)
- Become familiar with orthographic sketches. (SLO 3)
- Become familiar with projection views. (SLO 3)

Concept Content:

This concludes our module on basic sketching and different views. As a reminder, sketching is an important practice as it is how you will create your own blueprints. Be sure to keep practicing your sketching. The textbook chapter this week provides many opportunities to practice various types of sketching in your spare time if you wish.

This week in review:

Lectures:

Basic Sketching Presentation - 38 Slides

Videos:

Orthographic Projection from Isometric View in Engineering Drawing - 9 Minutes

Isometric Projection in Engineering Drawing - 7 Minutes

Reading:

Technical Sketching Chapter - 21 Pages

Blueprint Views Chapter - 4 pages

Assignments:

Practice Drawing - Outlined in module 2.3

Module 2 Quiz - 3 Questions

Be sure to have completed the assignments by **[insert due date here].**

Module 3 - Scales and Measurement

3.1 Module Overview

Concept Goals:

By the end of this moudle, you should be able to:

- Identify various measuring tools (SLO 4)
- Become familiar with scale in sketching (SLO 3)
- Review basic math (SLO 4)

Concept Content:

For this week, we will go over both basic math and the use of scale in blueprints. See the next sub-module for more details.

This week at a glance:

Lectures:

Measurement Tools Presentation - 15 Slides

Math Practice Questions - 32 Slides

Videos:

How to read a tape measurer - 7 Minutes

Reading:

Chapter 5 Scaling - 6 Pages

Assignments:

Module Quiz - 5 Questions

3.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will both review basic math skills along with introducing the concept of scales. Math is important for being able to decipher blueprint directions correctly. It is also important for being able to sketch them correctly. Another important ability is being able to utilize scale when both reading and sketching blueprints. As most products would be hard to draw in a 1:1 scale on a blueprint while relaying all of the important information pertaining to it. Using scale can allow us to scale up small projects and scale down large projects.

This week's content:

Lectures:

Measurement Tools Presentation - 15 Slides **(Instructor Note: You will have to demonstrate how to use these tools in class as the presentation does not have written instructions for students)**

Math Practice Questions - 32 Slides **(Instructor Note: answer key is located in the resources module)**

Videos:

How to read a tape measurer - 7 Minutes

Reading:

Chapter 5 Scaling - 6 Pages

As you take notes for this week, pay special attention to the various uses of scale when reading/sketching a blueprint.

3.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week's assignment:

Module Quiz - 5 Questions

3.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

3.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

3.6 Module Wrap-Up

Concept Goals:

Module Learning Outcomes:

- Identify various measuring tools (SLO 4)
- Become familiar with scale in sketching (SLO 3)
- Review basic math (SLO 4)

Concept Content:

This ends our look into basic math and use of scale. Thank you for keeping up with the coursework thus far.

This week in review:

Lectures:

Measurement Tools Presentation - 15 Slides

Math Practice Questions - 32 Slides

Videos:

How to read a tape measurer - 7 Minutes

Reading:

Chapter 5 Scaling - 6 Pages

Assignments:

Module Quiz - 5 Questions

Be sure to have completed the quiz by **[insert due date here].**

Module 4 - Blueprint Reading - Lines and Letterings

4.1 Moudle Overview

Concept Goals:

By the enod of this moudle you should be able to:

- Identify the different types of blueprint lines (SLO 1)
- Know the functions of the different types of blueprint lines (SLO 1)
- Know how to draw the different types of blueprint lines (SLO 3)

Concept Content:

This week we will go over the lines and lettering you see in a blueprint. We will go into more detail in the next sub-module, but just know that lines and lettering are almost like the alphabet when it comes to blueprint reading.

This week at a glance:

Lectures:

Blueprint Lines and Lettering PowerPoint - 34 Slides

Videos:

How to Read and Draw Blueprint Lines - 6 Minutes

Assignment:

Blueprint Line Drawing Assignment

Module 4 Quiz - 2 Questions

4.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

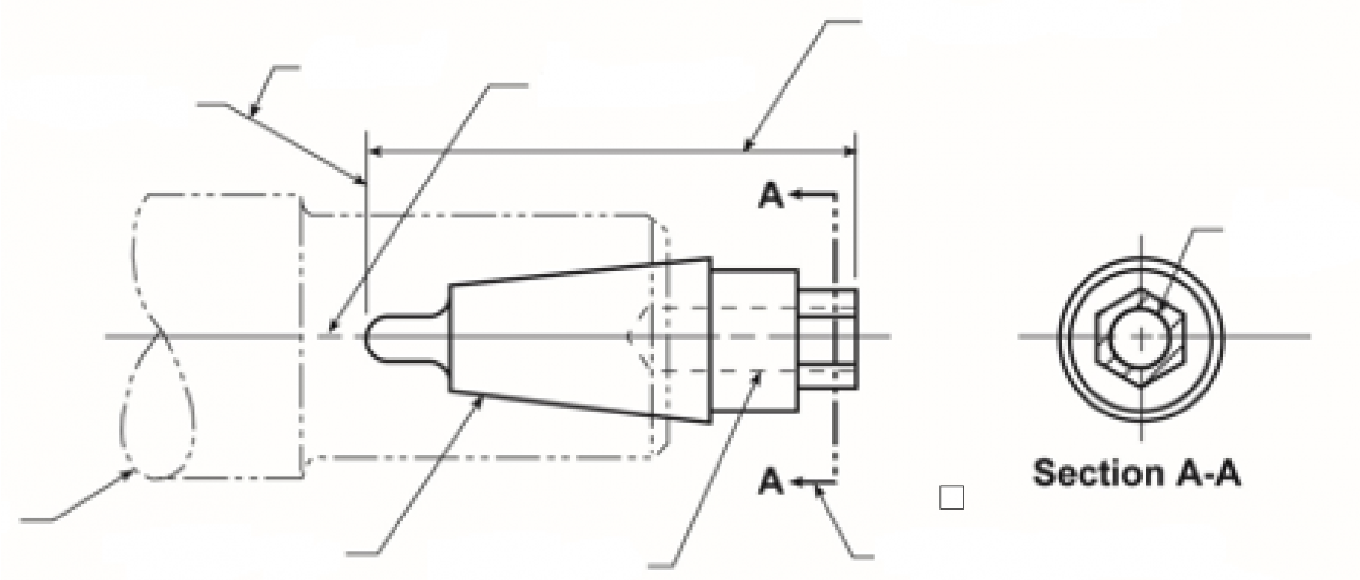


Figure 1: An example of a part blueprint

For a fun exercise, look at the above image. There are various lines that are being pointed to by the arrows. The following are the types of lines that are shown above: Leader Line, Extension Line, Sectioning Line, Cutting Plane Line, Dimension Line, Center Line, Hidden Line, Object Line, Phantom Line.

These are the types of lines that we will discuss in this week's material. However, as a warmup, try to guess what lines each arrow is pointing at. After we go over the material, check back, and see how many of them you got right.

This week we will discuss the basics of blueprint reading. Particularly, we will be focusing on blueprint lines, lettering, and going over a bit about views. Lines and letterings are the alphabet for a print. You need to know the lines and what they mean in order to glean all the information a print contains. Without that understanding, a blueprint really would be just a bunch of lines. Knowing the lines is foundational to reading a print properly.

This week's content:

Lectures:

Blueprint Lines and Lettering PowerPoint - 34 Slides

Videos:

How to Read and Draw Blueprint Lines - 6 Minutes

As you take notes for this week, be sure to take extensive notice of the types of lines. There are quite a few different ones and some of them look similar to each other. It may help to draw examples of the lines as you make your notes. The writing and drawing may help you better remember what each line looks like and how they function.

4.3 Moudle Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week's assignment:

Blueprint Line Drawing Assignment - On a piece of graph paper, draw two examples of the following lines:

Short Break, Long Break, Phantom Line, Hidden Line, and Object Line

Line straightness will be part of the grading for this assignment.

Module 4 Quiz - 2 Questions. However, these questions are long and multi-part questions.

4.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

4.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

4.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives

- Identify the different types of blueprint lines (SLO 1)
- Know the functions of the different types of blueprint lines (SLO 1)
- Know how to draw the different types of blueprint lines (SLO 3)

Concept Content:

This week we delved into lines and lettering, the "alphabet" of blueprint reading. Knowledge of these types of lines is important when it comes to being able to read blueprints.

This week in review:

Lectures:

Blueprint Lines and Lettering PowerPoint - 34 Slides

Videos:

How to Read and Draw Blueprint Lines - 6 Minutes

Assignment:

Blueprint Line Drawing Assignment

Module 4 Quiz - 2 Questions

Both assignments are due by **[insert due date here].**

Module 5 - Reading Multi-View Drawings

5.1 Module Overview

Concept Goals:

By the end of this module, you should be able to:

- Know how to read a multi-view drawing (SLO 1, SLO 2)

Concept Content:

This week it is time to go over multi-view drawings and how to read them. Go to the next sub-module for more detail on what we will be covering this week.

This Week at A Glance:

Lectures:

Multi-View Drawing Reading Pt. 1 - 10 Slides

Multi-View Drawing Reading Pt. 2 - 16 Slides

Videos:

Multi-View Drawings Video - 20 Minutes

Sketching Dimensioned Multi-View Drawing Video - 12.5 Minutes

Assignment:

Multi-View Drawing Assignment

5.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

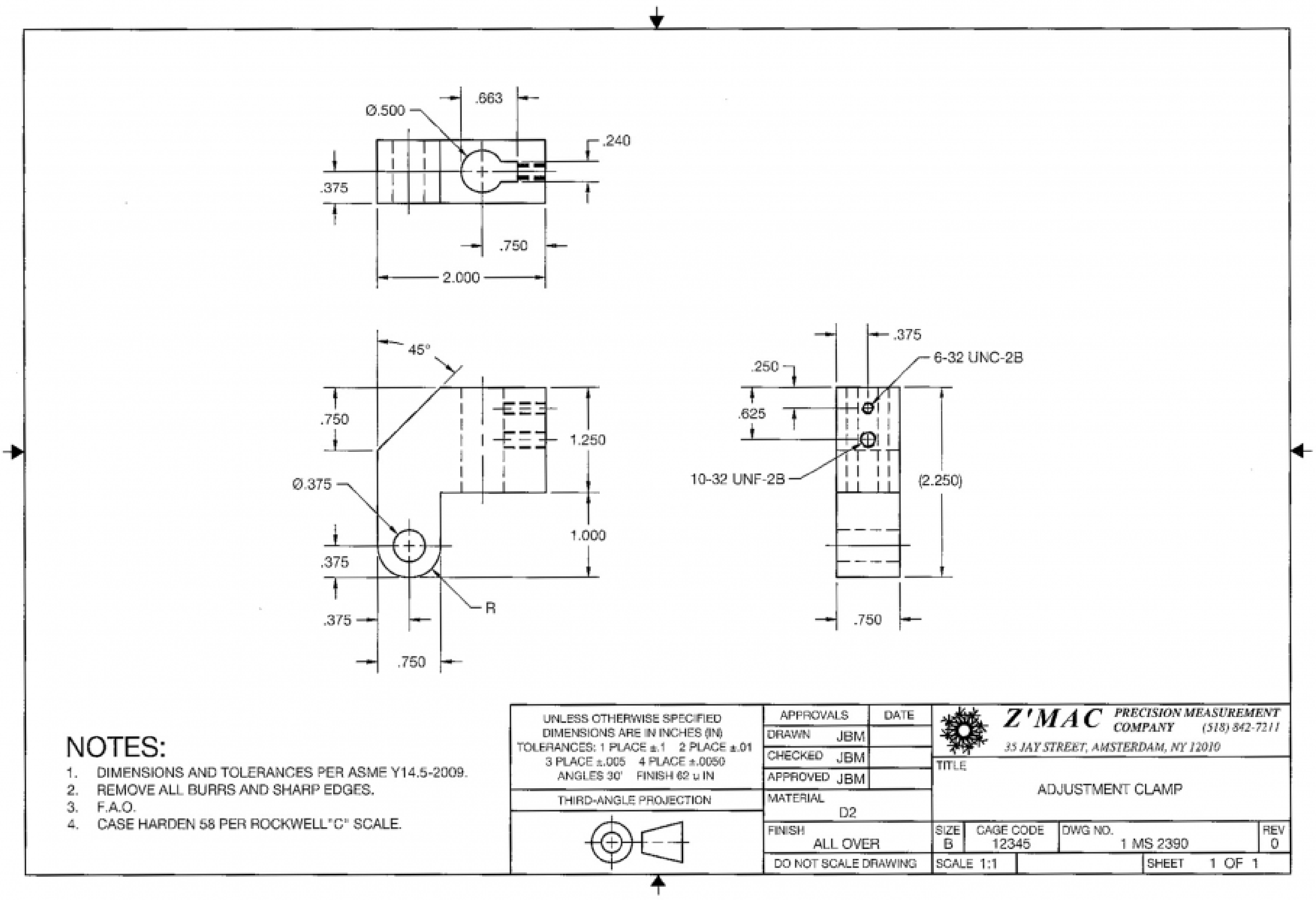


Figure: Multi-View Drawing.

The above image is a multi-view drawing of a machine part. Most blueprints you will encounter in a machine shop setting will be drawn this way. It is important to know how to properly read a multi-view drawing, so you know what part of the part is needed for the task. To that end, this week we will discuss how to read multi-view drawings. There is also a video about sketching a multi-view drawing. While drawing multi-view sketches is not something we will go in-depth with this week, seeing how a multi-view sketch is done can help you better understand how to read one.

This Week's Content:

Lectures:

Multi-View Drawing Reading Pt. 1 - 10 Slides

Multi-View Drawing Reading Pt. 2 - 16 Slides

Videos:

Multi-View Drawings Video - 20 Minutes

Sketching Dimensioned Multi-View Drawing Video - 12.5 Minutes

Blueprint Reading Intro and Views Video - 10 Minutes

5.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This Week's Assignment:

Multi-View Drawing Assignment - This assignment is for practice. Select an object in the machine lab or your house and do an orthographic drawing that shows the top, front, and right-side views of the object.

Upload your drawings to the quiz section.

5.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

5.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

5.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Know how to read a multi-view drawing (SLO 1, SLO 2)

Concept Content:

This week we covered multi-view drawings. Both the reading and drawing of them. Thank you for persisting through the course thus far. Remember, if you have any questions about the content or issues with the course, please reach out to me.

This Week In Summary:

Lectures:

Multi-View Drawing Reading Pt. 1 - 10 Slides

Multi-View Drawing Reading Pt. 2 - 16 Slides

Videos:

Multi-View Drawings Video - 20 Minutes

Sketching Dimensioned Multi-View Drawing Video - 12.5 Minutes

Assignment:

Multi-View Drawing Assignment

Module 6 - Reading Dimensions

6.1 Module Overview

Concept Goals:

By the end of this module you should be able to:

- Understand numerals and other dimension figures (SLO 1)
- Understand isometric dimensioning (SLO 5)
- Understand orthographic dimensioning (SLO 5)

Concept Content:

This week we will cover dimensioning. Please view the next sub-module for more information as to why this is important.

This week at a glance:

Textbook:

Dimensioning - 22 Pages

Lectures:

Dimensioning Interactive Lecture - 31 slides

Videos:

Dimension Calculation - 4 Minutes

Assignments:

Dimensioning Assignment

6.2 Module Content Resources

Concept Content:

This week we will go into reading blueprint dimensions. Two things that a blueprint must communicate are the shape and size of the object. We've looked into viewing the shape of an object previously but now it's time to look at the size of the object. Dimensions are how we communicate that information. To properly read and sketch a blueprint one must understand how dimensions work. To this end we will cover how dimensioning works.

Textbook:

Dimensioning - 22 Pages - Read this before viewing the lecture.

Lectures:

Dimensioning Interactive Lecture - 31 slides

Videos:

Dimension Calculation - 4 Minutes

6.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week's assignment:

Dimensioning Assignment - The worksheet and pictures that go with it are located in the resources tab. Download and complete the worksheet. Once you have completed the worksheet, upload a copy of the completed worksheet to assignments tab under quiz.

6.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

6.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

6.6 Module Wrap-Up

Concept Goals:

Module Learning Outcomes

- Understand numerals and other dimension figures (SLO 1)
- Understand isometric dimensioning (SLO 5)
- Understand orthographic dimensioning (SLO 5)

Concept Content:

Now that you have had the chance to learn about dimensioning, hopefully you will be better equipped to read industrial blueprints.

This week in review:

Textbook:

Dimensioning - 22 Pages - Read this before viewing the lecture.

Lectures:

Dimensioning Interactive Lecture - 31 slides

Videos:

Dimension Caculation - 4 Minutes

Assignments:

Dimensioning Assignment

Be sure to complete the worksheet by **[insert due date here].** Also, be sure to review your notes and course materials as next week will be the midterm for this course.

Module 7 - Mid-Term Exam

7.1 Module Overview

Concept Goals:

- Demonstrate an understanding of the first six weeks' content.

Concept Content:

This week we will have our mid-term exam. This exam tests your understanding of the first six module's content. It will also test your ability to apply that knowledge.

The exam is located under the assignments tab under tests. It has 15 Questions.

7.2 Module Wrap-Up

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

Congratulations on completing your mid-term exam. Next week we will start the second half of this course. Thank you for keeping up with the course so far!

Module 8 - Manufacturing Process

8.1 Module Overview

Concept Goals:

By the end of this module you should be able to:

- Become familiar with manufacturing processes (SLO 1)
- Be able to read blueprints to decipher what manufacturing processes are called for (SLO 1)

Concept Content:

This week we will go into manufacturing processes and how they relate to blueprints.

This week at a glance:

Lectures:

Manufacturing Process Presentation - 25 Slides

Textbook:

Chapter 9 Machined Features - 15 Pages

Videos:

Basic Blueprint Reading Chapter 9 Pt 1 - 5.5 Minutes

Basic Blueprint Reading Chapter 9 Pt 2 - 4.5 Minutes

Basic Blueprint Reading Chapter 9 Pt 3 - 8 Minutes

Assignment:

Module 8 Quiz - 5 Questions

8.2 Module Content Resources

Concept Content:

As a brain teaser, try to define what boss, counterbore, kerf, and neck would mean in a blueprint reading context. These terms and more are ones we will go over this week. Write down what you might think they would be and we can compare your initial definition to the actual one once the module is over.

This week we will discuss manufacturing processes. Oftentimes, blueprints communicate information regarding what kinds of manufacturing processes need to take place during a job. In order to effectively read a blueprint, one needs to know what these processes are. Many of the processes we will discuss this week are ones you would run into at a machine shop.

This week's content:

Lectures:

Manufacturing Process Presentation - 25 Slides

Textbook:

Chapter 9 Machined Features - 15 Pages

Videos:

Basic Blueprint Reading Chapter 9 Pt 1 - 5.5 Minutes

Basic Blueprint Reading Chapter 9 Pt 2 - 4.5 Minutes

Basic Blueprint Reading Chapter 9 Pt 3 - 8 Minutes

8.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This Week's Assignment:

Module 8 Quiz - 5 Questions

8.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

8.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

8.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives

- Become familiar with manufacturing processes (SLO 1)
- Be able to read blueprints to decipher what manufacturing processes are called for (SLO 1)

Concept Content:

This week we went into manufacturing processes and how they relate to blueprints. As you can see, blueprints relay a lot of vital information. This includes what types of manufacturing processes are called for during a project. Knowing what these processes are will help make blueprint reading easier.

This week in review:

Lectures:

Manufacturing Process Presentation - 25 Slides

Textbook:

Chapter 9 Machined Features - 15 Pages

Videos:

Basic Blueprint Reading Chapter 9 Pt 1 - 5.5 Minutes

Basic Blueprint Reading Chapter 9 Pt 2 - 4.5 Minutes

Basic Blueprint Reading Chapter 9 Pt 3 - 8 Minutes

Assignment:

Module 8 Quiz - 5 Questions

As a reminder, the module quiz is due by **[insert due date here].**

Module 9 - Fasteners and Springs

9.1 Module Overview

Concept Goals:

By the end of this module you should:

- Learn the parts of an external thread (SLO 4)
- Learn thread classifications in both standard and metric measurements (SLO 4)
- Learn about thread milling (SLO 4)
- Begin learning about torque (SLO 4)

Concept Content:

This week we will go over threads and fasteners.

This week at a glance:

Lectures:

Threads and Fasteners - 38 Slides

Reading:

Embedded in the next sub-module.

Assignments:

Module 9 Quiz - 8 Questions

9.2 Module Content Resources

Concept Content:

This week we will discuss threads and fasteners. Much of machining revolves around the use of threads and fasteners. The exact type of treads and fasteners is communicated on most blueprints. Understanding threads and fasteners, their uses, and the different types will help you better understand the information presented in a blueprint.

Lectures:

Threads and Fasteners - 38 Slides

Reading:

Screw Thread Nomenclature

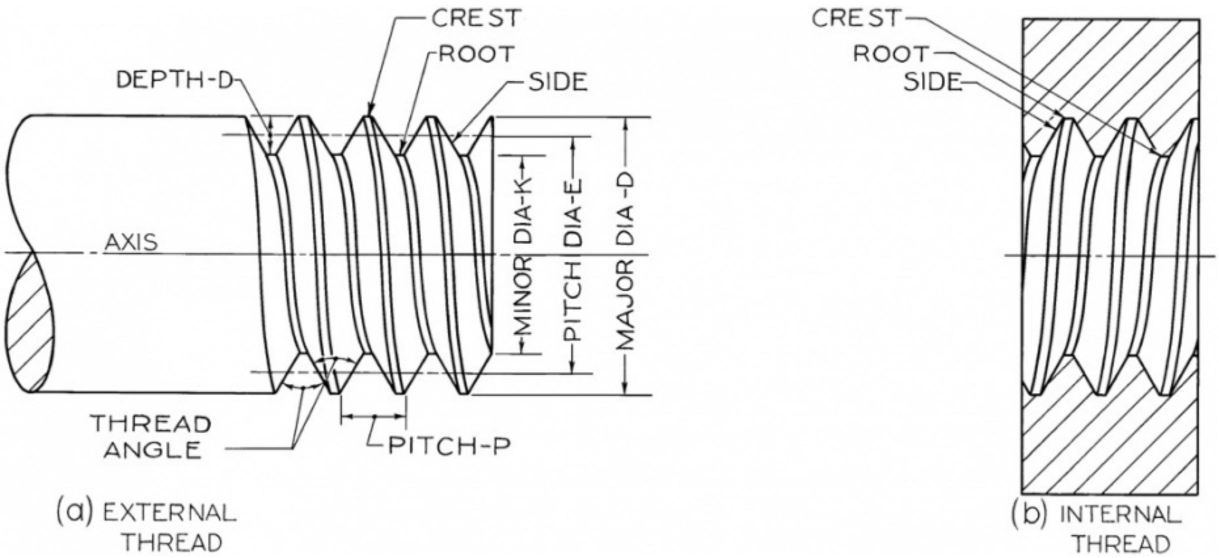


figure: **Screw Thread Nomenclature**

Common Thread Forms

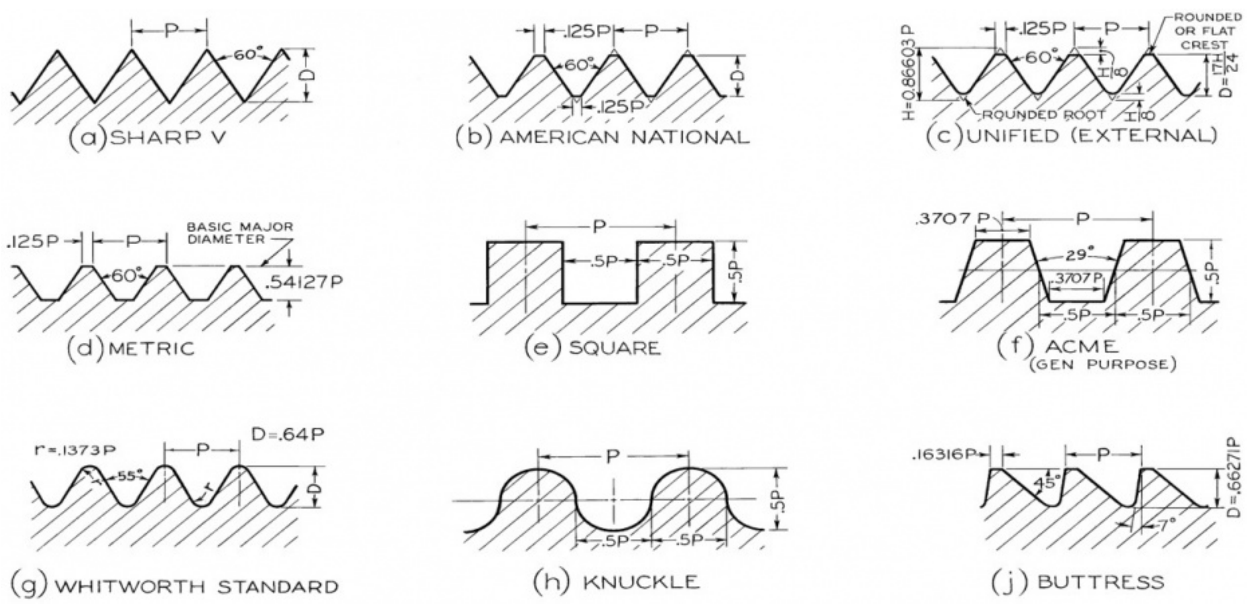


figure: **Common Thread Forms**

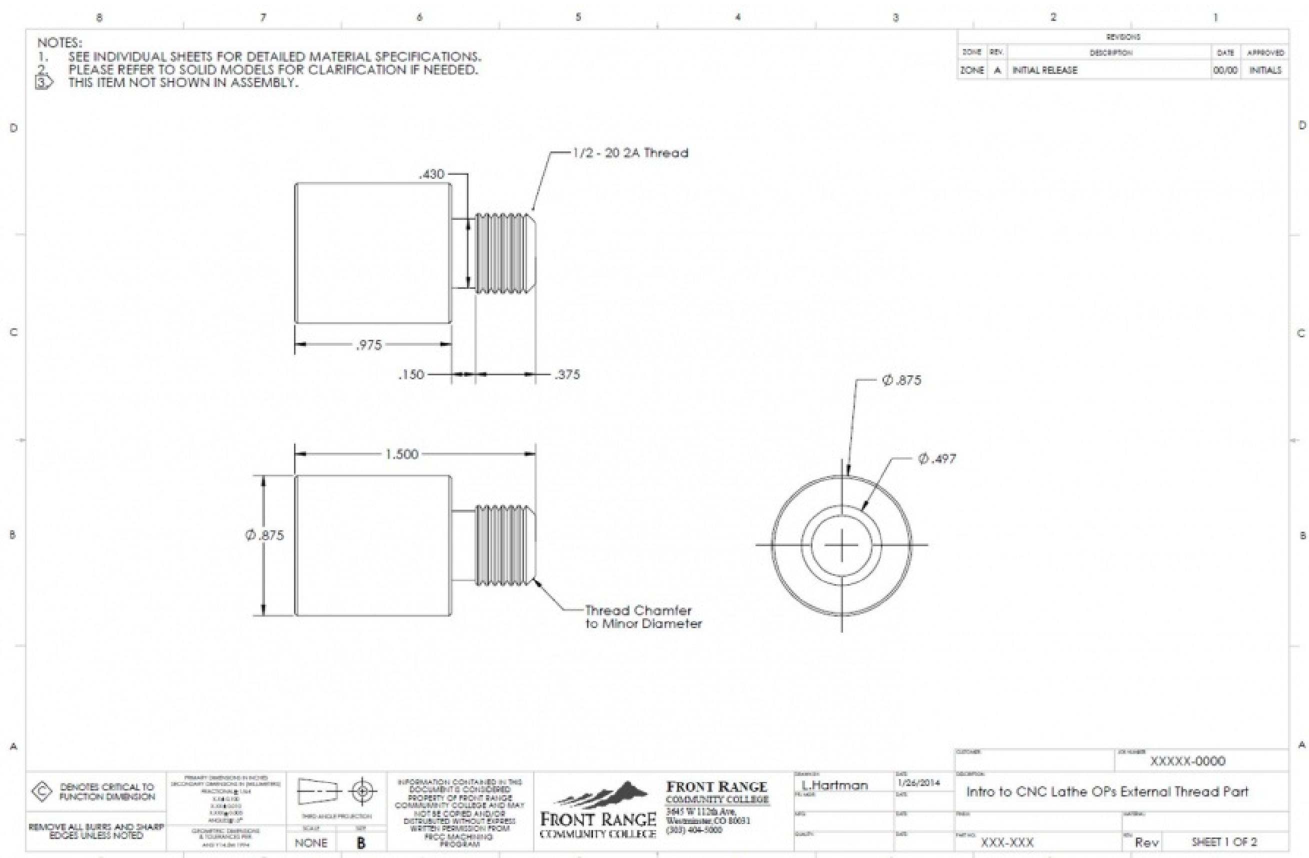


figure: **External Thread Part**

CNC Lathe Thread Calculations

For American Standard Threads:

- Single Side Depth of Thread = .6495 / No. of Threads per inch
- The Thread on Print is ½ - 20 (threads per inch)

  .6495/20 = 0.032475

- Minor Diameter = Major Diameter – 2 times Single Side Depth
- 0.500 – (0.032475 * 2) = 0.500 – 0.06495 = 0.43505 (Minor Diameter)

G92 Threading Cycle

MUST USE G97 DIRECT RPM

- Format for Threading Cycle (1/threads per in)
- G92 X (major diameter start) Z (length of thread) F (feed=thread lead)
  - X (subsequent cut diameter to minor)
  - X (subsequent cut diameter to minor)
  - X (subsequent cut diameter to minor)

G92 Thread Cycle Example from Print Dims

```
G97 S500 M03
G00 X0.500 Z0.100
G92 X0.490 Z-0.390 F0.050
        X0.485
        X0.480
        X0.475
(Repeat to Calculated 0.43505)
```

Thread Measurement

- Threading Calculations are for Minor Diameter for programming starting points.
- Threads need to be measured at Pitch Diameter.
- Thread pitch diameter should be obtained based on class of fit.
- Measured with two methods:
  - Measurement with a Screw Pitch Micrometer
  - Measured with 3 Wire Method

Standards

ASME B1.1, *Unified Inch Screw Threads* (UN and UNR Thread Form)

ASME B1.3M, *Screw Thread Gaging Systems for Dimensional Acceptability—Inch and Metric Screw Threads* (UN, UNR, M, and MJ)

ASME B1.5, *Acme Screw Threads*

ASME B 1.7M, *Nomenclature, Definitions, and Letter*

ASME B1.8, *Stub Acme Screw Threads*

ASME B1.9, *Buttress Inch Screw Threads Symbols for Screw Threads*

ASME B1.10M, *Unified Miniature Screw Threads*

ASME B 1.11, *Microscope Objective Thread*

ASME B1.12, *Class 5 Interference-Fit Thread*

ASME B1.13M, *Metric Screw Threads—M Profile*

ASME B1.15, *Unified Inch Screw Threads (UNJ Thread)*

ASME B1.20.1, *Pipe Threads, General Purpose (Inch)*

ASME B1.20.3, *Dryseal Pipe Threads (Inch)*

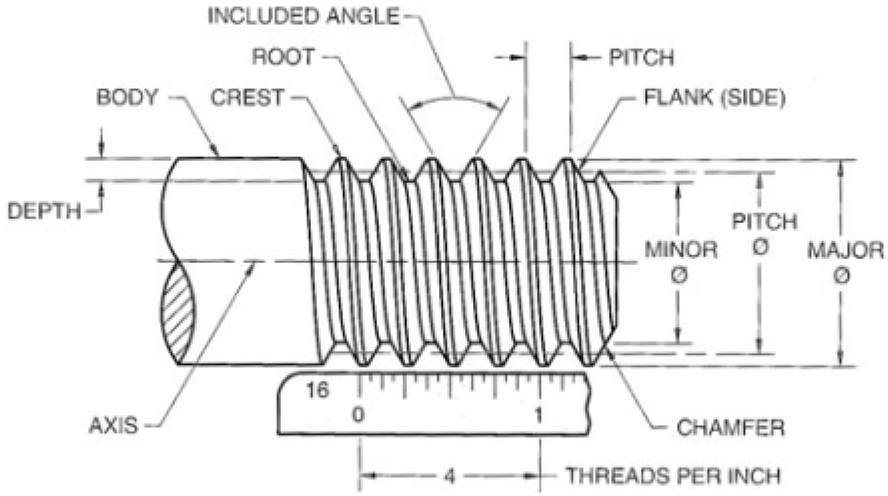
ASME B1.20.7, *Hose Coupling Screw Threads (Inch)*

ASME B1.21M, *Metric Screw Threads: MJ Profile*

ASME B18.29.1, *Helical Coil Screw Thread Inserts*

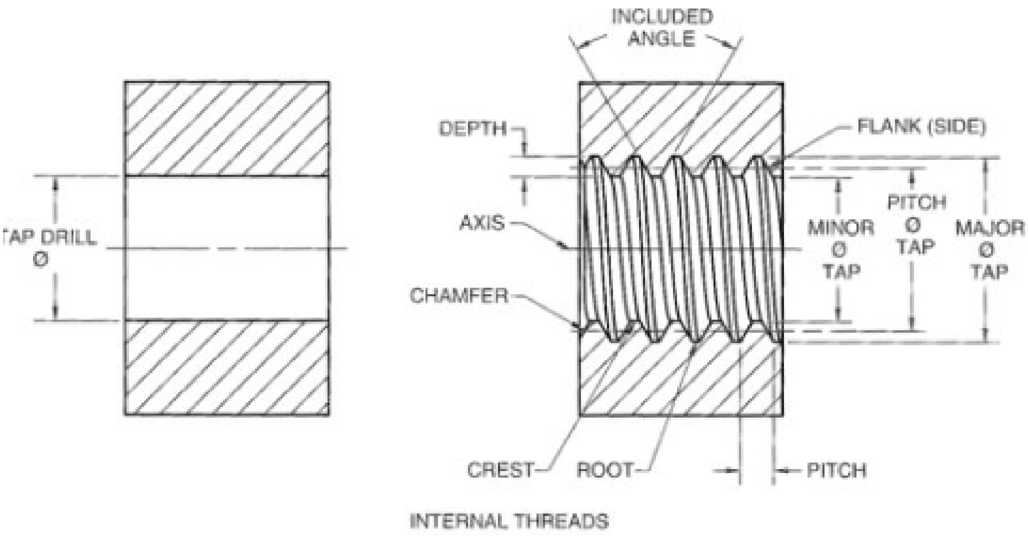
ASME B18.29.2M, *Helical Coil Screw Thread Inserts*

External Threads

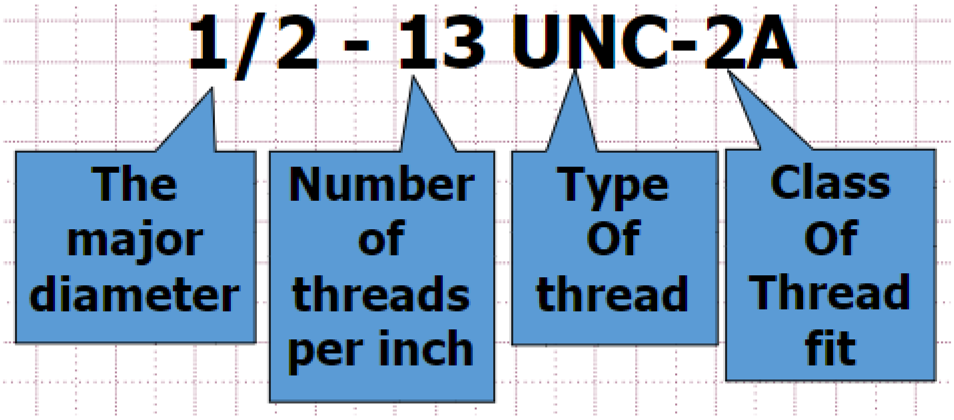


EXTERNAL THREADS

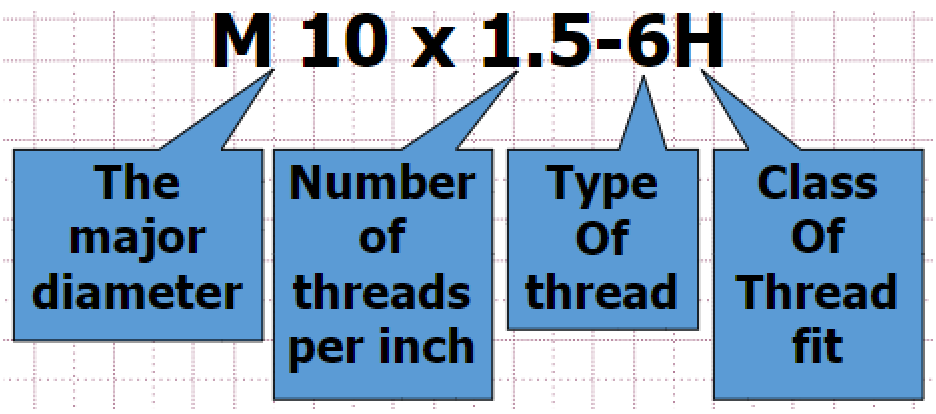
Internal Threads



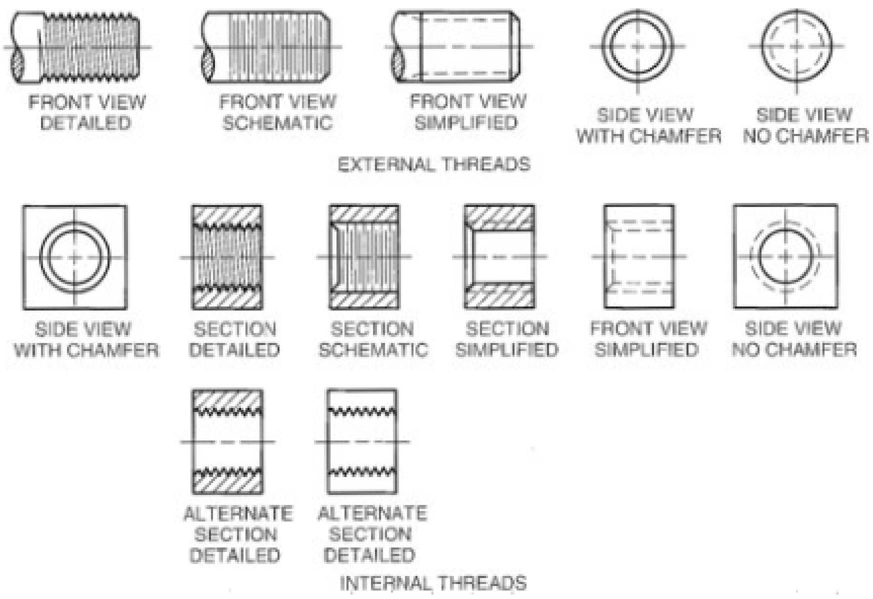
Reading a Thread Note (Inch)



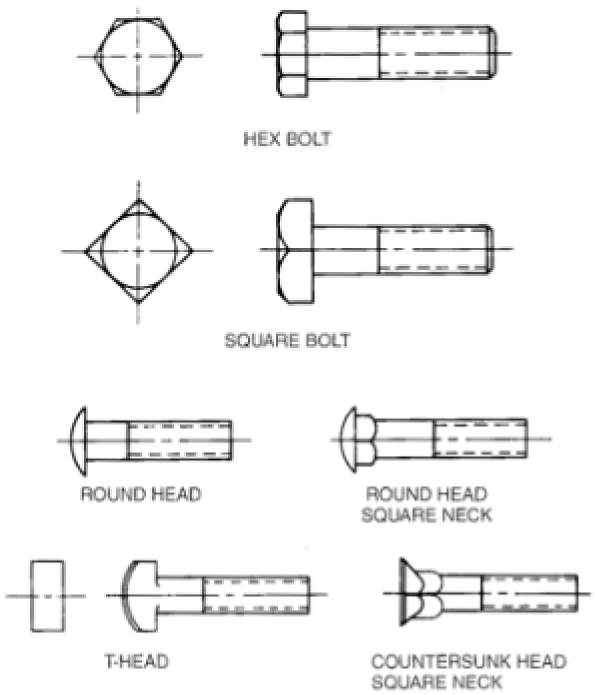
Reading a Thread Note (MM)



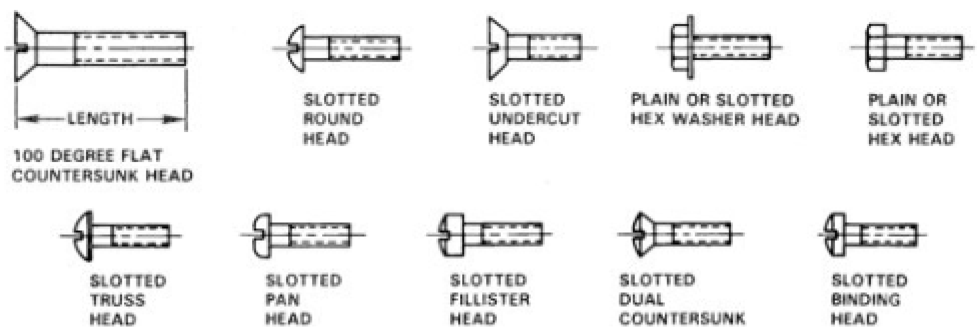
Thread Representation



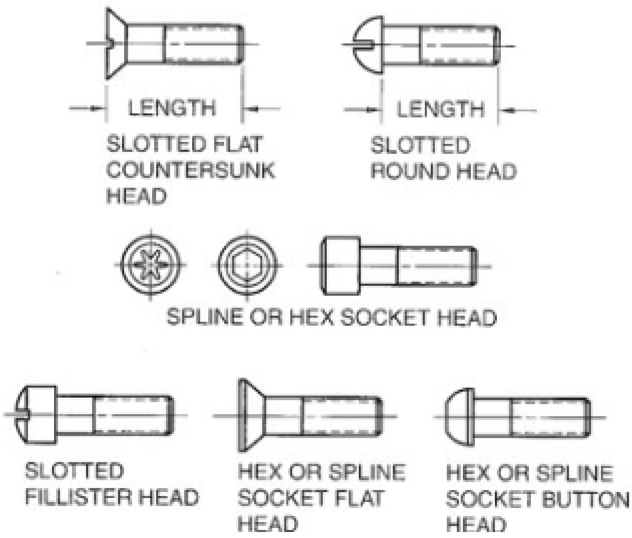
Bolt Types



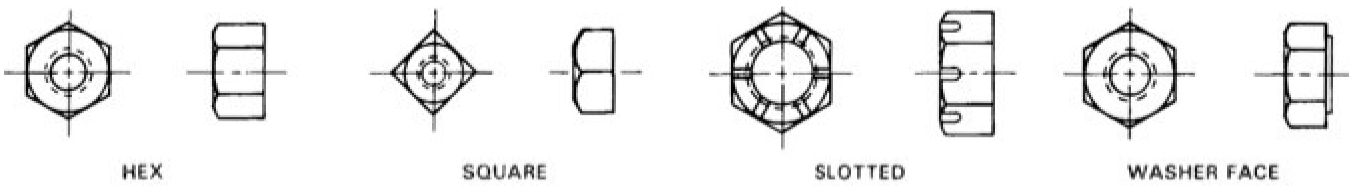
Machine Screw Types



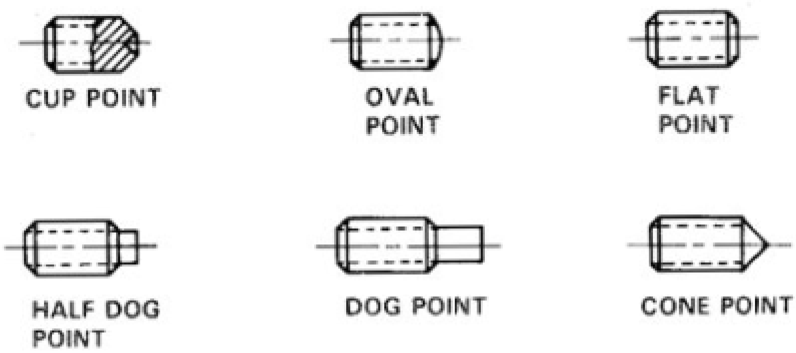
Cap Screw Types



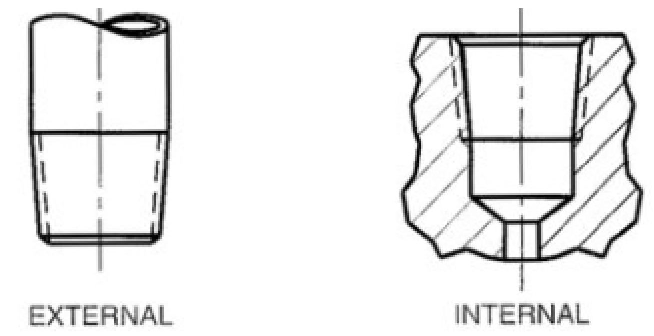
Nut Types



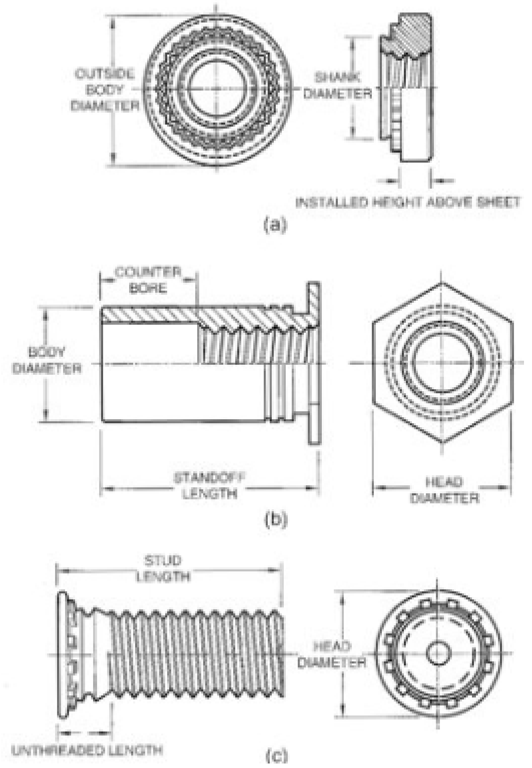
Set Screws



Pipe Threads

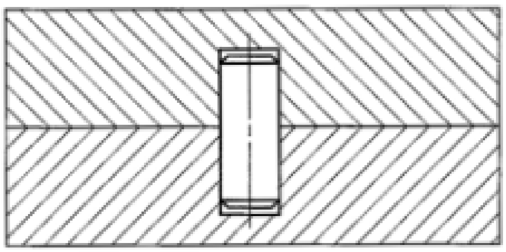


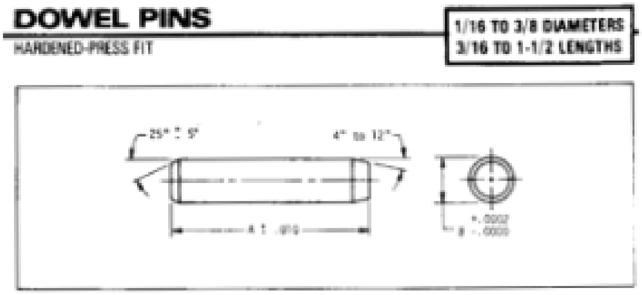
PEM



Common Washers

Dowels





Taper Pins

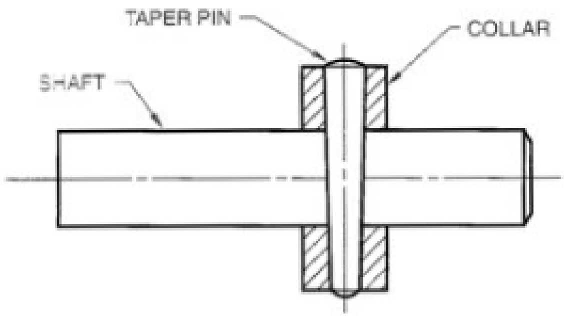
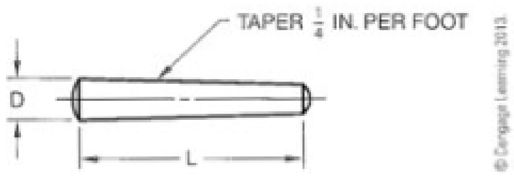
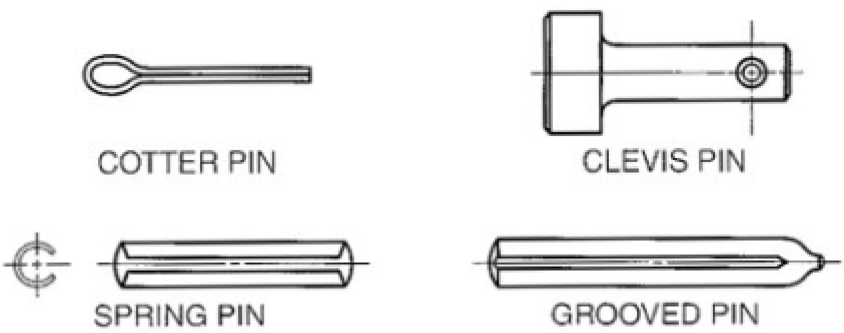


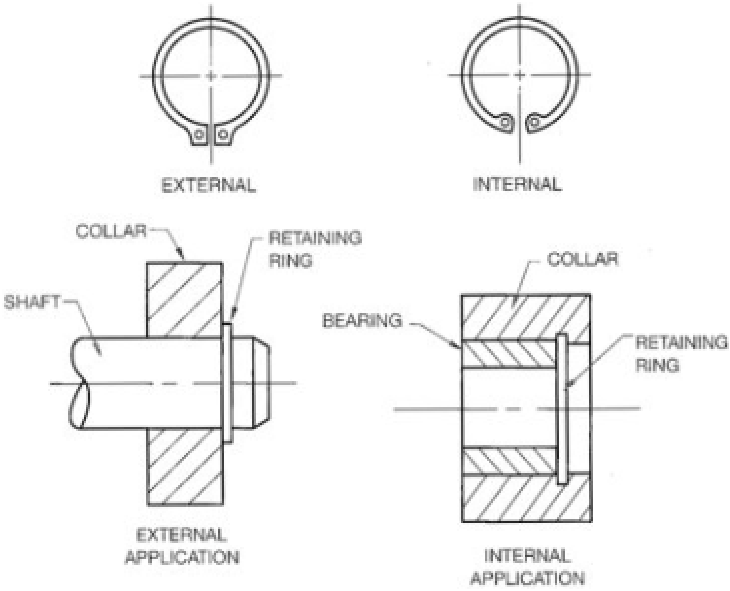
FIGURE 9.39 Taper pin in assembly, sectional view.



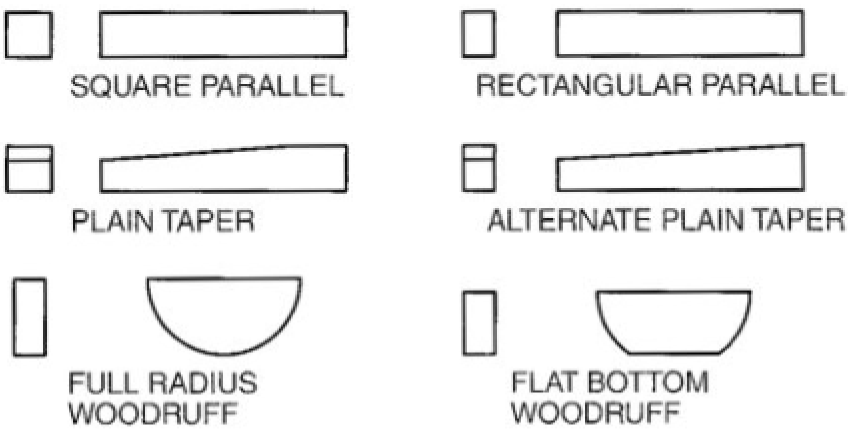
Other Pin Types



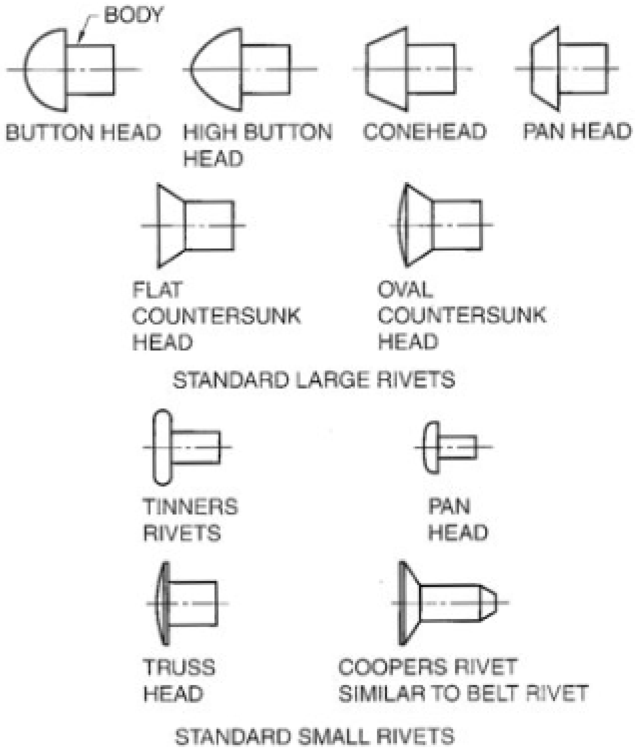
Snap or Retaining Rings



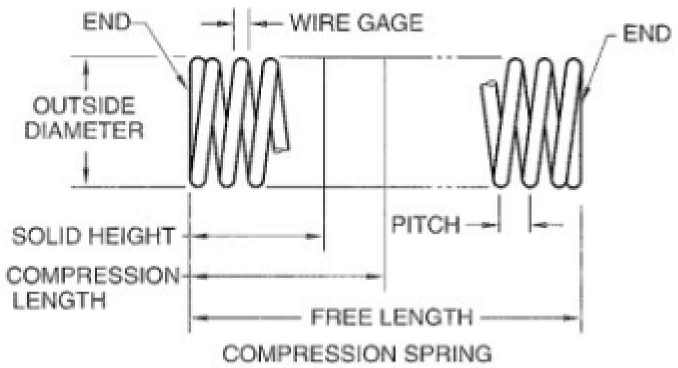
Key Types

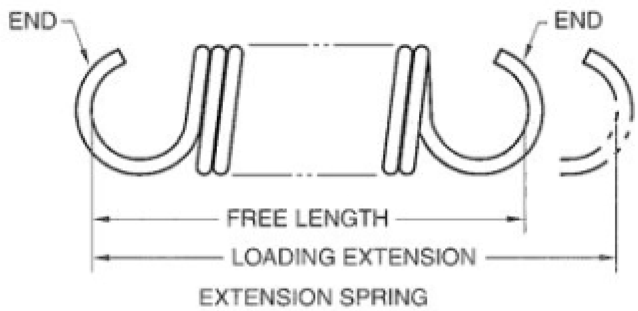


Rivets

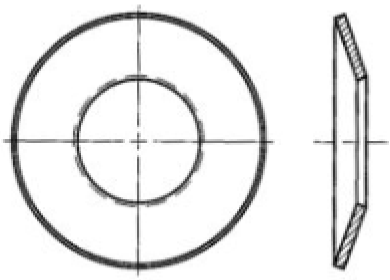


Springs

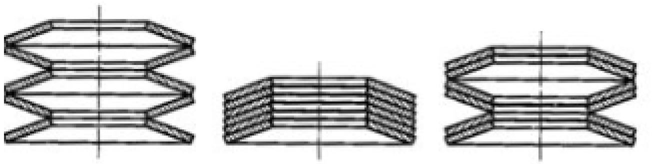




Cone Washers



CONED DISK SPRING



STACKING METHODS

9.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week's assignment:

Module 9 Quiz - 8 Questions

9.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

9.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

9.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Learn the parts of an external thread (SLO 4)
- Learn thread classifications in both standard and metric measurements (SLO 4)
- Learn about thread milling (SLO 4)
- Begin learning about torque (SLO 4)

Concept Content:

This week we went over threads and fasteners. As you can see, there is much to know about the different types. Each one is needed for different tasks.

This week in review:

Lectures:

Threads and Fasteners - 38 Slides

Reading:

Embedded in the next sub-module.

Assignments:

Module 9 Quiz - 8 Questions

Module 10 - Working Drawings

10.1 Module Overview

Concept Goals:

By the end of this module, you should:

- Become familiar with the types of working drawings. (SLO 2, SLO 3)

Concept Content:

This week we will look into working drawings. Go to the next sub-module for more detail.

This week at a glance:

Lectures:

Working Drawings Lecture - 23 Slides

Videos:

Blueprint Reading Video - 24 Minutes

Multidetail Drawings - First 2 minutes

Assignments:

Module 10 Quiz - 4 Questions

10.2 Module Content Resources

Concept Goals:

- Become familiar with the types of working drawings.

Concept Content:

This week we will go into reading a working drawing. This will include the different types of drawings and what those drawings mean.

Lectures:

Working Drawings Lecture - 23 Slides

Videos:

Blueprint Reading Video - 24 Minutes

Multidetail Drawings - First 2 minutes

10.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This Week's Assignment:

Module 10 Quiz - 4 Questions

10.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

10.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

10.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Become familiar with the types of working drawings. (SLO 2, SLO 3)

Concept Content:

This week in review:

Lectures:

Working Drawings Lecture - 23 Slides

Videos:

Blueprint Reading Video - 24 Minutes

Multidetail Drawings - First 2 minutes

Assignments:

Module 10 Quiz - 4 Questions

Module 11 - Pictorial Drawings

11.1 Module Overview

Concept Goals:

By the end of this module, you should be able to:

- Identify and describe each of the members of the axonometric family of pictorial views. (SLO 2)
- Identify and describe each of the members of the oblique family of pictorial views. (SLO 2)
- Understand how to read pictorial drawings. (SLO 2)

Concept Content:

This week we will discuss pictorial drawings. See the next sub-module for more detail.

This week at a glance:

Lectures:

Pictorial Drawing - 16 Slides

Reading:

Pictorial Drawings - 17 Pages

Videos:

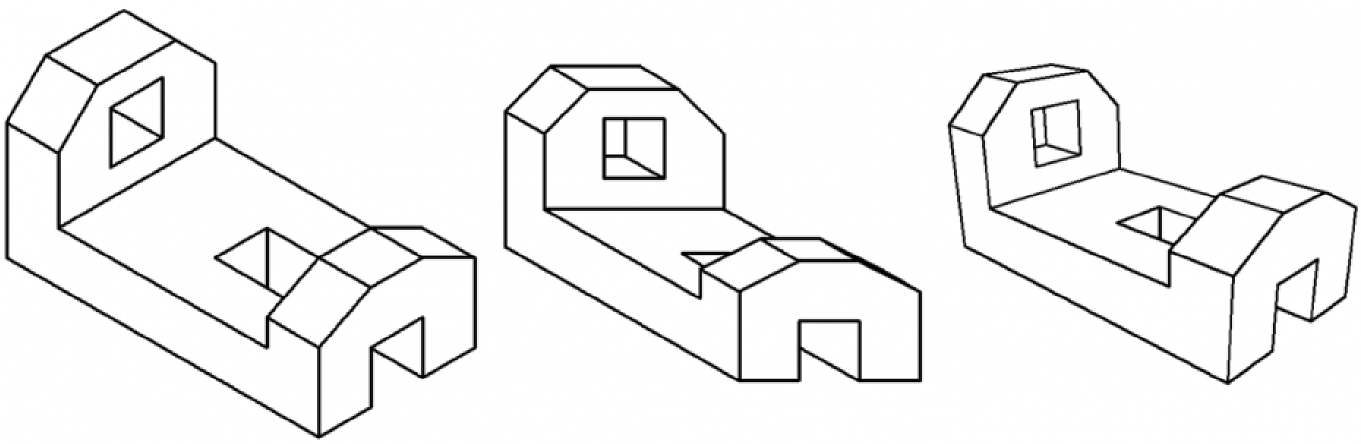
Understanding Pictorial Drawings - 15 Minutes

Assignments:

Module Worksheets

11.2 Module Content Resources

Concept Content:



The above image showcases an example of the three different families of pictorial views. These would be perspective, oblique, and axonometric. Without looking at this week's material, try to guess which image represents what family of drawing. Later we will discuss this image and will see if your initial guesses were correct.

This week we will discuss pictorial drawings. We will cover the different types of pictorial views, the most used pictorial drawings, and how to read and understand them. Special attention will be given to axonometric pictorial views as this is a very common type of view in industrial blueprints.

Lectures:

Pictorial Drawing - 16 Slides

Reading:

Pictorial Drawings - 17 Pages

Videos:

Understanding Pictorial Drawings - 15 Minutes

11.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This Week's Assignment:

Module 11 Worksheets - Once you have completed the worksheets, upload them in the assignments tab under quiz.

11.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

11.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

11.6 Module Wrap-Up

Concept Goals:

Module Learning Outcomes:

- Identify and describe each of the members of the axonometric family of pictorial views. (SLO 2)
- Identify and describe each of the members of the oblique family of pictorial views. (SLO 2)
- Understand how to read pictorial drawings. (SLO 2)

Concept Content:

This week we discussed pictorial drawings. As shown, there are a variety of different views, each with a different purpose. Next week we will go into surfacing finishes.

This week in review:

Lectures:

Pictorial Drawing - 16 Slides

Reading:

Pictorial Drawings - 17 Pages

Videos:

Understanding Pictorial Drawings - 15 Minutes

Assignments:

Module Worksheets

Please be sure to have your worksheets completed by **[insert due date here].**

Module 12 - Surfacing Finishes

12.1 Module Overview

Concept Goals:

After this module, you should:

- Learn the purpose of surface finish. (SLO 1)
- Recognize and understand surface finish symbols. (SLO 1)

Concept Content:

This week we will go into surface finish and surface finish symbols.

This week at a glance:

Lectures:

Videos:

Surface Finish Video - 5 Minutes

Surface Finish Symbols Video - 18 Minutes

Assignments:

Review - Dimension Assignment

12.2 Module Content Resources

Concept Content:

This week we will go over surface finish and surface finish symbols. Blueprints will often communicate what kind of surface finish is needed for a particular job. This is important as surface finish/texture helps control friction and transfer layer formation during sliding. During this week we will go over different types of surface finishes and going over how to read and interpret surface finish symbols.

Lectures:

Surface Finish Lecture - 30 Slides

Videos:

Surface Finish Video - 5 Minutes

Surface Finish Symbols Video - 18 Minutes

12.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This Week's Assignment:

Dimensioning Assignment - Download and complete the worksheet and upload the completed assignment to the assignments tab under quiz.

This week we will have a review of dimensioning.

12.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

12.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

12.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Learn the purpose of surface finish. (SLO 1)
- Recognize and understand surface finish symbols. (SLO 1)

Concept Content:

This week we went into surface finish and surface finish symbols.

This week in review:

Lectures:

Videos:

Surface Finish Video - 5 Minutes

Surface Finish Symbols Video - 18 Minutes

Assignments:

Review - Dimension Assignment

Be sure to have your dimensioning worksheet completed by **[insert due date here].**

Module 13 - Geometric Dimensions and Tolerancing (GD&T)

13.1 Module Overview

Concept Goals:

By the end of this module, you should:

- Learn the definitions of GD&T (SLO 5)
- Become familiar with the basic concepts of GD&T (SLO 5)
- Understand why GD&T is vital for engineering and machining (SLO 5)

Concept Content:

This week we will begin our two week series on GD&T.

This week at a glance:

Videos:

[What is GD&T in 10 Minutes]- 10 Minutes.

Readings:

See the next sub-module, this week's content is embedded there.

Assignment:

Module 13 Quiz - 5 Questions

13.2 Module Content Resources

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

This week we will start going over Geometric Dimensioning and Tolerancing (GD&T). There is a lot of information that goes along with these concepts so we will be spending the last two weeks before the final on this subject. This week is part one and we will wrap up our section on GD&T next week.

GD&T is a symbolic language. It is used to specify the size, shape, form, orientation, and location of features on a part. Features toleranced with GD&T reflect the actual relationship between mating parts. Drawings with properly applied geometric tolerancing provide the best opportunity for uniform interpretation and cost-effective assembly. GD&T was created to insure the proper assembly of mating parts, to improve quality, and to reduce cost. GD&T is a design tool. Before designers can properly apply geometric tolerancing, they must carefully consider the fit and function of each feature of every part. GD&T, in effect, serves as a checklist to remind the designers to consider all aspects of each feature. Properly applied geometric tolerancing insures that every part will assemble every time. Geometric tolerancing allows the designers to specify the maximum available tolerance and, consequently, design the most economical parts. GD&T communicates design intent. This tolerancing scheme identifies all applicable datums, which are reference surfaces, and the features being controlled to these datums. A properly toleranced drawing is not only a picture that communicates the size and shape of the part, but it also tells a story that explains the tolerance relationships between features.

Videos:

What is GD&T in 10 Minutes - 10 Minutes. This video provides a good summary of the concepts outlined below and provides visual examples.

Readings:

Symbols and Terms

Introduction: Geometric Dimensioning and Tolerancing is an engineering and manufacturing convention for specifying engineering design and drawing requirements that, when followed, dictate and ensure the proper description, function and relationship of hardware features.

Additionally, when properly used and followed, it guarantees the most effective and economical control of the manufacture of that hardware.

However, where the material requires a more complex assembly of definitions with translations of concepts, this material is meant to supplement and assist the instructor's presentation as a more structured format, if they care to use it.

Also note that there are some locations in the material where it says to refer to an illustration, which is to be chosen to suit by the instructor.

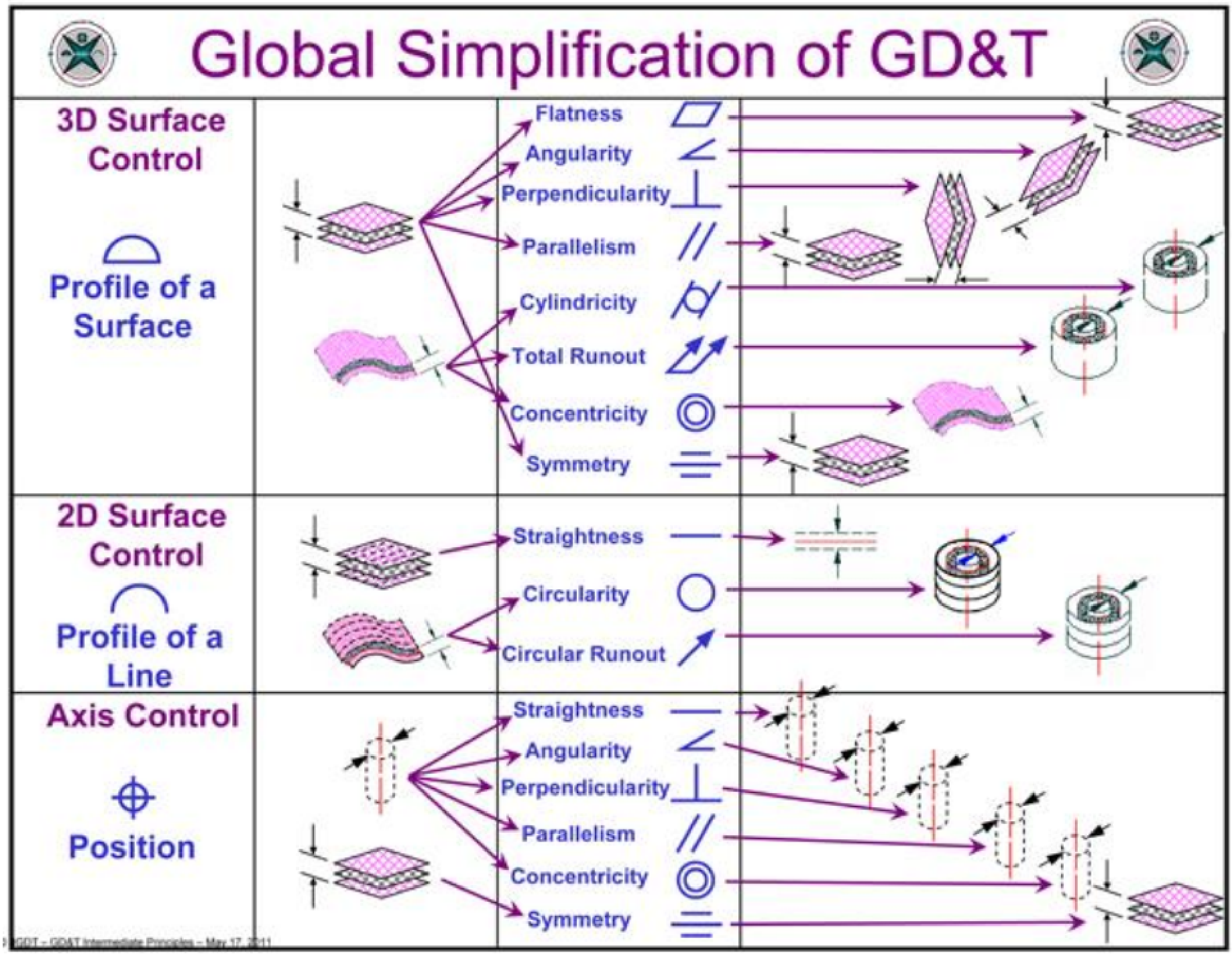
Coordinate or limit tolerancing vs. Geometric tolerancing: Coordinate drawing methodology has been used for centuries if not millennia to convey an engineer's or designer's intent for components and assemblies. Its limitations have been a growing lack of specificity as the improvements of mankind's capabilities in sciences like aeronautics, automotive, medical technology and many more avenues of manufacturing took place.

The older methods of drawing and engineering simply could not provide the necessary controls between manufacturing organizations to produce hardware that did not rely on mouth to mouth communications and corporate tribal knowledge that this convention required. GD&T is an effective replacement for these shortcomings and provides a repeatable control of engineering specifications from organization to organization as well as country to country.

Geometric Characteristic Symbols: There are 14 symbols used in GD&T to define the geometry attributes of parts. These 14 symbols, which will be studied more in depth later are further defined by their type, which divides each of them into one of five categories, which are form, profile, orientation, location and runout. (Instructor illustrations will briefly describe each one.)

GD&T tolerances control the allowable deviation of four of the five possible geometry attributes of a feature. Those are size, location, orientation, form, and the fifth attribute, which is surface texture, has its own symbology and feature controls.

For this class, we will focuse on Angularity, Perpendicularity, and Parallelism. Please review those three symbols below.



[GD&T Orientation Video:](#) 5.5 Minutes

13.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

Assignment:

Module 13 Quiz - 5 Questions

13.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

13.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

13.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Learn the definitions of GD&T (SLO 5)
- Become familiar with the basic concepts of GD&T (SLO 5)
- Understand why GD&T is vital for engineering and machining (SLO 5)

Concept Content:

This week we began our two week series on GD&T.

This week in review:

Videos:

What is GD&T in 10 Minutes - 10 Minutes.

Readings:

See the next sub-module, this week's content is embedded there.

Assignment:

Module 13 Quiz - 5 Questions

Be sure to have your worksheet completed by **[insert due date here].**

Module 14 - More GD&T

14.1 Module Overview

Concept Goals:

By the end of this module, you should:

- Learn the GD&T symbols for angularity, parallelism, and perpendicularity (SLO 5)
- Understand what angularity, parallelism, and perpendicularity are used for in GD&T (SLO 5)

Concept Content:

This week we will continue our section on GD&T.

This week at a glance:

Reading: Embedded into the next submodule

Videos:

Orientation Tolerances Video - 24 Minutes

Anglularity Introduction Video - 2 Minutes

Angularity Inspection Procedure - 7 Minutes

Perpendicularity and Parallelism Inspection Video - Start at the 3:25 Mark

Assignment:

Module 14 Lab Handout

☑ 14.2 Module Content Resources

Concept Content:

Orientation Control

Orientation Control is one of the four types of tolerances in GD&T. It contains 3 basic types of control: Perpendicularity, parallelism and angularity. Orientation is another important building block in the GD&T language.

Orientation controls the angularity, (tilt,) of surfaces, axes and median planes for features of size and non-features of size but never location of those features. Datum references are always required.

Perpendicularity allows the drawing to communicate tolerance zones for perpendicular features and implied 90° angles. Perpendicularity is the condition where a planar surface, axis or center plane is exactly 90° to its referenced datum.

Perpendicular features are the most common relationship created on machine tools but perfect perpendicularity is a concept more than a reality. This is why perpendicularity tolerances are necessary. They take the form of different tolerance zones.

When the tolerance is applied to most features it is two parallel planes separated by the distance of the tolerance in the control frame. When it is used with a diameter symbol is takes the form of a cylinder whose diameter is the tolerance in the control frame. That tolerance zone is always perpendicular to the primary datum frame.

Parallelism is the second of the three orientation controls in GD&T. It is the condition where a planar surface, axis or center plane is equidistant at all points to a datum plane or axis and it must always be specified to a datum plane or axis.

When it is applied to a planar surface it is located by the surface and its tolerance zone is equally disposed about that surface as two planes separated by the tolerance stated in the control frame.

When it is applied to a feature of size the tolerance zone applies to the axis or center plane of that feature of size and is either a cylinder around the axis or two parallel planes disposed about the surface of the feature, the size or separation of which is the tolerance stated in the control frame.

Angularity is the third of the three orientation controls. It is the condition where a planar surface, axis or center plane is at a specified angle to a datum plane or axis.

When it is applied to a planar surface, like parallelism, it is located by the surface and its tolerance zone is equally disposed about that surface as two planes separated by the tolerance stated in the control frame. Except here it is at a specified angle to the datum instead of being parallel to it. Remember that the specified angle is the stated perfect condition and the tolerance allows for the feature to have planar differences and surface irregularities that differ from the perfect callout, within the limits of the tolerance zone.

When it is applied to a feature of size the tolerance zone applies to the axis or center plane of that feature of size and is either a cylinder around the axis or two parallel planes disposed about the surface of the feature, the size or separation of which is the tolerance stated in the control frame, again like parallelism, except the disposition of the feature is at a specified angle instead of being parallel to it.

Videos:

Orientation Tolerances Video - 24 Minutes

Anglularity Introduction Video - 2 Minutes

Angularity Inspection Procedure - 7 Minutes

Perpendicularity and Parallelism Inspection Video - Start at the 3:25 Mark

14.3 Module Assessment/Assignment

Concept Goals:

Outline the learning goals for this module here.

Concept Content:

Assignment:

Module 14 Quiz - 5 Questions

14.4 Module Reflection

Concept Content:

This is a completely optional section. The purpose of this section is to ask your students to reflect on the material they have learned in this course. Or, if there is a specific area of the content you wanted to make sure students understood, you could guide them to discuss that in their response to your reflection question(s). You could also use this section to discuss case studies related to the content this section went over. However, if you feel that this would not be an appropriate assignment/task for your specific subject, please feel free to delete this section from your class.

14.5 Module Discussion Board

Concept Content:

This is a completely optional section. The prupose of this section is to invite your students to discuss the week's content and what they learned from it with each other. If you feel this would not be approriate for your class or at least this week's content, feel free to delete it. If you are interested in doing a discussion board, a good idea would be to come up with a question related to the week's content for the students to answer. From there, require them to answer the quesion and respond to a least one other student's answer to foster discussion.

14.6 Module Wrap-Up

Concept Goals:

Module Learning Objectives:

- Learn the GD&T symbols for angularity, parallelism, and perpendicularity (SLO 5)
- Understand what angularity, parallelism, and perpendicularity are used for in GD&T (SLO 5)

Concept Content:

This week we completed our section on GD&T.

This week in review:

Readings: Embedded in module 14.2

Videos:

Orientation Tolerances Video - 24 Minutes

Anglularity Introduction Video - 2 Minutes

Angularity Inspection Procedure - 7 Minutes

Perpendicularity and Parallelism Inspection Video - Start at the 3:25 Mark

Assignment:

Module 14 Lab Handout

Be sure to have completed the lab assignment by **[insert due date here].**

Next week we will be having our final exam. Thank you all for sticking through the class!

Module 15 - Final Exam

15.1 Final Exam

Concept Goals:

- Demonstrate an understanding of the course material

Concept Content:

This week is our final exam. Please click on the assignments tab and look under test to locate it.

This exam has 25 questions however, some questions have multiple parts to answer.

15.2 Course Wrap-Up

Concept Content:

Thank you all for your participation throughout the course! I hope this course has been enlightening regarding blueprint reading, the uses of blueprints, and how they are used in industry.

Knowing how to read blueprints is a foundational skill if you wish to get hired as a machinist pretty much anywhere.

I wish you all the best of luck in your future classes!

Faculty Resources (For Instructor Only, Do Not Publish Live)

Odigia Guide

Concept Content:

Click on the resources tab to find the guide sheet for instructors.

Module 11 Worksheet Answer Keys

Math Practice Questions Answer Key

Concept Content:

Math Practice Questions Answer Key - 4 Pages

Bonus GD&T Material (Could use to make a 16th week if your semesters are 16 weeks)

Concept Content:

This is a collection of other GD&T Materials that can be used in your course. If you are in need of a 16th week, the materials from here could be used to build that module. There is a lot to sort through and much more than what could be taught in a week so please take a look and choose what looks good for you.

Basic dimensions and feature control frames: GD&T is specified on drawings through the use of feature control frames. Feature control frames include information that identifies first, one of the 14 geometric characteristics, second, the specific tolerance information and possibly modifier information and third, (and perhaps fourth and fifth,) datum reference information.

Basic dimensions are used to convey numerical values that control the size, profile, orientation or position of features and features of size. The values are placed inside rectangular boxes and state the exact theoretical size, profile, orientation or position of a part feature. The term theoretical is used here because a feature's exact size is theoretical at only one perfect condition and varies from that by the allowances of its GD&T definitions and controls.

Modifiers: Material modifiers are a set of symbols that add additional information about how tolerancing is applied to control the size and relationship of features on engineering drawings. Modifiers are especially important to machinists because they can be used to determine process control choices that allow the greatest latitude in producing tighter tolerance parts that ultimately will meet drawing spec's using wider tolerance margins through the correct translation of feature sizes and relationships.

Form Control

Form Control is one of the 5 types of geometric controls and within this type there are four form controls that are each one of the 14 geometric characteristics that define a part's geometry. The four form controls are flatness, straightness, circularity ad cylindricity, which will each be explained herein.

Form controls are the least used of GD&T characteristics simply because they are usually used as a refinement to the characteristics of features, after those features have been positioned and located. Form controls are not used with datums and can be found attached to features with leaders and never associated with feature tolerances. Modifiers are not allowed to be used with form tolerances so bonus tolerances are not gained in their use.

Straightness, for the purposes of this class, is a form control used only on cylindrical features that further limits the allowable deviation of surface elements or the "derived median line" of cylindrical features of size.

Straightness, when applied to the surface of a cylinder creates a tolerance zone between 2 parallel lines on the surface of the cylinder, which by necessity is smaller than the size tolerance of the feature. Straightness, when applied to the median line, (centerline,) of a cylinder, creates a diameter tolerance zone at the derived median line of the feature. This tolerance zone must also be smaller than the size tolerance of the feature.
Flatness is defined as being the condition of a surface as having all the line elements in one plane. A

perfectly flat plane's surface has only perfectly straight line elements in it. The tolerance zone for a feature with controlled flatness is 2 parallel planes separated by the flatness tolerance callout. Circularity controls the surface of features that have surfaces of revolution like cylinders, cones, torus, (plural tori,) other than a sphere, where all points on the surface intersected by a plane perpendicular to that surface are equidistant from its axis of revolution.

Where circularity controls the surface of a sphere, all points on that surface intersected by any plane passing through the center must be equidistant to that center.

Datum System

The Datum System is another very important building block in the GD&T language. The datum system is a set of symbols governed by rules and procedures that control the relationships of parts and the geometric tolerances that define those parts.
The word relationship is important here because as it was noted earlier, the old system of limit or coordinate tolerancing was lacking a method that established a priority or importance of how dimensions and relationships of parts were established, making it difficult to properly control part geometry on individual parts and assemblies.

The use of datum systems in part design and manufacture guarantees more precise communication of tolerances and the relationship of features as well as a more thorough and repeatable way to measure those parts and assemblies.

Understanding the datum system from a machinist's standpoint will require learning some terms and how their use establishes relationships and how they are used on drawings. First there are basically two kinds of datums machinists will need to understand to setup and manufacture parts correctly. Those are "datum features" and "datums."

Something to note here is that there are more terms and definitions concerning datums than are included in this material, however because of the scope and limited time available the ones covered here are considered basic and important for machinists to understand in their efforts to learn how to use GD&T to manufacture and control the quality of their methods in the shop.

Datum features are part features or features of size that are identified on drawings to control the translation of tolerances and relationships. Parts may have a number of features of size, like holes, bosses or shoulders, of which, the part designer was most concerned with one, perhaps a certain hole, that was important to the function and assembly of not only that part but others too.

Datums are theoretical planes, axes, lines or points on a part, also located on the part, used to control the translation of tolerances and relationships.

That feature is identified as a datum and it along with other features and characteristics of the part, establish the important relationship of the orientation and position of the part and its features. Understanding datums is understanding constraints.

The main purpose of a datum system is to constrain the movement of that part so that repeatable measurements can be made when inspecting the part. The simplest way to look at that is to understand how a typical part, such as a part with 3 datum planes, like a cube, is constrained or controlled. (See illustration.)

A part such as this cube has what is known as 6 degrees of freedom. Note in the illustration that the part has 3 planes, labeled A, B and C. The planes are at 90 degree angles to each other in three dimensional space. These planes are known as a “datum reference frame.” This datum reference frame establishes axes in 3 dimensions which will be called X, Y and Z.

The 6 degrees of freedom associated to this part can now be identified as rotation around the X, Y and Z axes and movement along those same axes. Understanding how these datum planes constrain the part is key to understanding how a machinist must setup, hold and cut the part that will become that cube.

Datums are specified on a drawing in the feature control frame. Most parts will have 3 datums and many will have more than that. Complex parts may have multi part feature control frames with numerous datums, all with complex relationships.

The 3-2-1 rule defines a basic place to start on how datums are used in the shop and in the quality lab. It states that a part will have a minimum of 3 points of contact with its primary datum, 2 points of contact with its secondary datum and 1 point of contact with its third or tertiary datum.
The need for this rule becomes evident in real world situations where parts are imperfect, at least more often than not parts are far less perfect than the machines and inspection equipment they are made and inspected on. And that’s where the 3-2-1 rule is needed. A flat surface is often not really flat, so when a person sets up that plane on a surface plate it needs to have at least 3 point of contact for that plane to be as normal as possible to that plate.

And so it goes with the second and third datums. The second needs to be as close to normal with the machine or inspection setup as possible and so it is with the third datum. Each datum, in its priority of importance to the drawing will become less likely to match the more perfect world of the machine it’s made on or the inspection devices it is checked on.

Position Tolerance

Position Tolerance is perhaps the most important individual subject in GD&T because it is the most commonly employed tool used by designers and engineers to orient and locate features. Because this is true it is the most common tolerance found on drawings and hence it is the most studied and analyzed callout in GD&T.

Machinists and inspectors spend a great deal of their time ensuring their setups, gages, methods, and understanding of the parts they are producing meet the intentions of the drawings the are working on.

Position tolerances include 3 basic characteristics: true position, profile of a surface and profile of a line. These tolerances control the theoretical exact position of a feature of size. This is basic to understanding position and furthermore how to understand how the basic components of GD&T affect part geometry.

Position tolerances control only features of size. They limit how much a center point, axis or center plane of a feature is allowed to deviate from the exact position dictated in the position callout. Position may or may not use datum references. These tolerances directly control the location of features and may indirectly control orientation and form.

When we say position controls the exact location of features it is necessary to review how the condition of something perfect cannot realistically be achieved in manufacturing. Since this is true we look at all of the features, dimensions, tolerances, control frames, characteristic symbols and part geometry on a drawing and try to conceive the perfect component as it existed in the designer's imagination. As machinists we break all those components down into scenarios that our machines can control and keep up with to produce the part within those parameters.

Often machinists perform their setups and cutting methods with too narrow of an understanding of what the drawing is requiring. We focus on one setup to produce some features and in our haste we complete the task perhaps correctly for the things we accomplished only to find that subsequent features cannot be correctly added to our part because the interrelationship of all the features was more complex than what was allowed for in the initial efforts to get work accomplished on the part.

Position gives us the prescriptions for just how much a dimension can vary from perfect and still meet all the requirements of the drawing. It gives the features relationship to each other and part datums. It can control form in the GD&T sense IE: flatness and straightness etc. but also in the sense of shape regarding how the designer envisions and requires the part to function perhaps because it needs to fit with other parts or in its eventual use in how it responds to conditions of physics and how that may require it to be a certain shape.

There are four requirements for specifying a position tolerance: It can only be applied to a feature of size or a pattern of features of size. Basic dimensions must be employed to define the exact location of tolerance features of size. Position tolerances require datum references except when controlling coaxial features of size. When no modifiers are used in the control frame RFS, (regardless of feature size,) and RMB, (regardless of material boundary,) are the default modifiers.

MMC which means maximum material condition, (and its opposite but related modifier LMC, least material condition,) are used to allow more tolerance in the location and orientation of features.

When MMC is applied to a feature or pattern of features it creates a virtual condition that may add to the tolerance of a feature under certain conditions. In such a case the drawing will specify a feature of size like a hole. That hole will have a required size dimension and depth controlled by the circumstances of the drawing. The position feature control frame will dictate the feature's required position tolerance refined by the basic callouts and datums in the control frame.

This is the basic recipe for position. When the designer knows that the requirements of these features can vary from their near perfect positions to something less demanding under certain conditions, then they can use a modifier like MMC to tell the machinist what they need.

For instance: a drawing shows a hole at some diameter and it has a stated size tolerance. The control frame and basic dimensions tell the machinist to machine the hole located by the callouts. Because the control frame contains the M inside a circle, telling the machinist that this instance allows for him to choose to refine the size of the hole to give him more tolerance in where the hole is located.

The definition for this condition works like this: The control frame tells the machinist that the feature must be located in its true position to .014". This means that the exact location of its axis, as stated by all the basic dimensions and datums on the drawing must be a cylinder with a .014" diameter tolerance zone. When the control frame contains the M modifier it means that axis must be located in that spot when the hole is at its smallest, or when there is the maximum amount of material contained in the part.

When the machinist machines the hole to a larger diameter, the part has less material in by witness of the chips that have come out of the hole as it became larger. When the hole is larger the part has been made to have less than its maximum material as was the case when the hole was at its smallest.

Now the rules that define this condition say that for every thousandth of an inch that the hole deviates from MMC there is a bonus of that same amount added to the position tolerance. The hole or feature must remain with the limits of its stated size on the drawing but for any given amount that the hole moves away from MMC then that same amount of bonus tolerance is added to the position tolerance.

In the case of our .014" position requirement, if the machinist can safely increase the size of that hole from it smallest to .008" larger than the .014" position tolerance becomes .022". when all the mathis calculated the difference between a true position of .014" and .022" is an increase in the deviation of the rectilinear location of the hole's axis of nearly .003".

This is considerable knowing that the original tolerance required the hole to deviate no more than approximately .005" under the same conditions as mentioned above. The new acceptable deviation in X and Y in this case would be about .008 instead of .005.

(Instructor will provide the needed graphics and explanations to cover this subject and how it can be expanded to include modified datums and patterns.)

Position Tolerance – Special Applications

Position Tolerance specifications in special applications. Position tolerancing is often applied to patterns of features as opposed to singular features. When this happens the need to control those features becomes more complicated. At this point GD&T contains expanded definitions and terms that meet these requirements.

When the position of a feature becomes the position of features, the standard position control frame is no longer definitive enough to constrain all the aspects of the part. In other words the size limits of the features and their allowable deviations from the exact location requirements to the stated datums do not provide enough information to serve the purposes of why a pattern of features exists.

A callout on a drawing may specify four holes at a certain size and to a specified location. The drawing with position tolerance controls where each of those holes is located. The machinist machines each of the four holes and in turn they will be inspected by the same rules that they were located during machining.

However, in some cases, where a pattern of features, such as four holes, is to be used to locate a separate part or assembly, the specific location of the part to be located by the pattern requires its own tolerance of location, which will be stated separately from the features within the pattern. In this case the system to control location of the four holes considers both the location of the pattern, which can be looked at as one feature, and the location of each of the four holes with respect to each other, which is a requirement of the part or assembly that they will eventually hold in place.
This requires an expanded system of control which incorporates a dual feature control frame with one frame positioned above the other secondary frame. These are known as composite feature control frames.

The upper frame specifies the applicable datums and the positional tolerance for the pattern of features. The second or lower frame specifies the datum for the pattern, (sometimes,) and the positional tolerance for the features in the pattern.

From the machinist's perspective the system usually resolves itself well because the more difficult task of locating the position of the pattern to its three datums would be the most demanding part of the machine setup, which is the least difficult of the requirements because that position is usually a looser tolerance. The tighter positioning of the second frame tolerance controls the holes as they relate to themselves, which is a function of the machine axes encoders or scales which will control the position of the holes within the pattern with little difficulty.

From the machinist's perspective composite position tolerances can be understood more simply by remembering that the lower segment cannot control position of the features. Depending on whether and how datums are used in the frame, it can only control orientation, alignment and spacing.

This leaves the control of the location of the features to the upper frame, which will contain the tolerance and applicable datums required for the function of the features.

Calculating True Position

The material in plan #6 showed that position tolerances require a number of things to be valid. One of those requirements is the inclusion of a modifier. Even when there is no modifier evident in the control frame the default rule specifies that when none are present there is by definition a default modifier present in their absence.(Instructor provides illustrations.)

Understanding zero tolerance at MMC. Sometimes designers use this method to tolerance features whose eventual function is assembly. A pattern of holes with an allowable position deviation can be stated at zero position tolerance at MMC so that the most tolerance for the assembly of features can be expressed as a bonus tolerance regarding the size of the features.

In cases like these, such as a hole or pattern of holes, the size of the features is allowed to increase until their size allows for the variations that are needed to facilitate the assembly of the part to another part. In this case the size of the hole can be increased until the part will meet tolerance for a given machining scenario, where the additional size, (as it deviates from MMC,) adds tolerance to allow the setup to produce the part to tolerance.
Calculating the true position of a feature in a real world application is somewhat complicated but when approached step by step it is easy to understand. Earlier in plan #6 an example of how MMC affects a true position callout is explained. Here we will use that example to break down an actual TP calculation like a machinist would perform to check if their setup is producing their part to tolerance.

In this case the machinist has setup a part that has a hole toleranced to a true position of .014" to datums A,B and C. The have machined their first piece and after measuring it in the quality lab they find their hole to be .005" from its perfect location in X and .005" from its perfect location in Y. Illustration #101 shows a graphic of the .014" diameter tolerance zone and how the maximum allowable deviation in two axes from the perfect center of the tolerance zone is laid out geometrically. The formula below that graphic shows how a machinist or inspector would consider the actual deviations in X and Y and put them in a formula that would tell them what the true position is for that part, using the formula 2 times the square root of the sum of the squares.

The results show how a true position tolerance of .014", when applied to a real world part, will allow a deviation in two axes of .005”. Looked at from the opposite direction the example shows how a feature machined at a deviation of .005” in two axes calculates to a true position of .014”. This shows that a machinist must understand position tolerancing from the standpoint of how it translates on a drawing. They must also know how to translate the results of their methods on machines into usable information about how their parts relate to the requirements on the drawings. Earlier in plan #6 we learned that position tolerancing is made up of 3 geometric characteristics: position, profile of a line and profile of a surface. These 2 latter characteristics referring to profile will be studied in a later learning plan.

Run-out, Concentricity and Symmetry Tolerances

Runout Tolerances: control the deviation of surfaces of revolution over the full 360 degrees of rotation of the part on a datum axis. Circular runout controls elements of a surface independent of each other. Total runout controls the entire surface of revolution, all elements of that surface together.

Circular and total runout are geometric tolerances that simultaneously limit the form, location, and orientation of parts containing surfaces of revolution. These tolerances always require a datum axis. Feature surfaces can be concentric or at right angles to the datum axis.
Circular runout is a geometric tolerance that controls the high to low point deviation of circular elements of any surface of revolution. That also includes the circular elements of planar surfaces that are perpendicular to and intersect the datum axis. The circular runout tolerance zone applies independently to each circular element of the applicable feature.

Total runout is a geometric tolerance that controls the high to low point deviation of all surface elements simultaneously of cylindrical, conical and sometimes irregular surfaces of revolution or planar surfaces at right angles to the datum axis.

Runout tolerances control the geometry attributes of form and orientation and when axis offset is controlled, they control the location of features.
The tolerance zones for circular runout are two coaxial circles. The tolerance zones for total runout are two coaxial cylinders. Because this is true, total runout is always more complex to verify.

Concentricity and Symmetry are two tolerances of the fourteen geometric characteristic tolerances. While once they were not uncommon, they are becoming scarce. That scarceness can be attributed to how difficult they are to actually prove in real world situations, and because the purpose for which they were often used in the past is being managed by other characteristic tolerances that serve the designer’s purposes and are easier to employ.

These tolerances must always be used with datum references and both are considered tolerances of location.

Concentricity defines that a surface of revolution must have all of its axial median points of all diametrically opposite points on that surface within the boundary of the part’s concentricity tolerance zone, which is a cylinder.

In other words, if you take a cylinder and measure all the points on its surface that are opposite each other, where the line between each set of points bisects the axis of the feature, that line’s midpoints will represent the data needed to completely define the condition. That is a considerable task considering that the geometry definition of those points on the feature’s surface says that there are an infinite number of them.

The median point of each set of opposite surface points will create a set of points that make up the axis of the feature, running the length of that feature. That axis, because the cylinder itself is imperfect, creates not an axis line but an axis cylinder. The variations in the surface of the feature create that cylinder, which becomes the construction of the tolerance zone for those median points and it must be within the size limits of the concentricity tolerance to be in tolerance. (Instructor to provide example graphics showing how the variations in the surface of the feature create an imperfect axis for the feature.)

Symmetry is very similar to concentricity. This tolerance controls the space between two parallel planes centered on a datum axis or datum center plane. Usually used with slots or tabs, the tolerance controls the disposition of planar features about a center axis or plane. Symmetry tolerances can control the geometry attributes of form, orientation and location.

Profile Tolerances

Profile tolerances. These tolerances control shapes made up of one or more features that are collectively called true profiles. True profiles describe features of parts that designers need to serve purposes that perhaps simpler or standard shapes can't perform; hence the more complex profile callout is employed to serve the designer's purpose.

Complex, in these cases, can take the shape of antenna parts hidden under the skins of jet aircraft with undulating 3D contours that reflect the complex shapes of the aircraft to electronic chassis with pockets made up of rectilinear lines and curves attached end to end. If it can be described mathematically it can be toleranced on a drawing with profile callouts.

Profile tolerances control the deviations of line elements and surfaces and depending on how they are used they can affect size, form, orientation and location and often all four of these at once.

Profile tolerances come in two varieties: Profile of a line, which is a two-dimensional tolerance and profile of a surface which is a three-dimensional tolerance

Profile of a line is most often seen in a section view and controls individual line elements of a surface. The tolerance zone is centered along the true profile of the described feature and may be used with or without datum references. This tolerance only applies in the view in which it is used because it is two-dimensional.

Profile of a surface is a tolerance that controls three dimensional shapes and describes tolerance zones that extend over the width and length or sometimes circumference of a part's features. This tolerance applies to the entire surface simultaneously and is centered along the true profile of the feature. These tolerances may also be used with or without datums and control deviations of surface elements.

As mentioned profile tolerances describe tolerance zones centered on the true profile of a feature and by default the zone is bilateral, defined as being equal on both sides of the true profile. However, under different circumstances a feature may require the disposition of the tolerance to be divided about the true profile as all on the inside or outside or in some cases unequally divided on one side or the other.

When a unilateral, (using all the tolerance on one side of the true profile,) or unequal, (part but not all of the tolerance used on one side or the other,) tolerance is needed, the drawing must specify the condition.

When the tolerance is to be applied to the feature either all on the outside or inside of the feature, the control frame will simply contain the total profile tolerance and the drawing will show a phantom line on one side of the true profile or the other, and the amount of the tolerance disposed to the inside or outside of the feature, which designates the amount and direction of the feature that the tolerance is to be applied.

A second method is acceptable that states the condition in the feature control frame. The control fame will contain the total tolerance followed by an offset modifier. The offset of the tolerance is then identified by placing a value of either the total tolerance, which signifies that the tolerance will be placed on the outside of the feature or the value will be zero, which signifies that the tolerance will be placed on the inside of the feature. This method allows the control frame to contain all the information necessary without additions to the drawing.

Another set of terms that are good to learn are “in space,” which refers to the outside of the profile and “in material,” which refers to the inside of the profile. “In space” and “in material” are perhaps more descriptive in more circumstances.

When the tolerance is intended to be disposed unequally about the true profile then the drawing can identify this in one of two methods. In one method the control frame will contain the profile symbol and total tolerance. The drawing view of that feature will then contain a phantom line showing the amount of the tolerance and to which side of the feature it is to be placed.

A second method is acceptable that states the total value of the tolerance before an offset modifier and a value after the modifier that represents the portion of the tolerance to be offset to the outside of the feature, leaving the remaining portion of the total, (when the offset portion is subtracted from the total,) to be offset to the inside of the feature. This method allows the control frame to contain all the information necessary without additions to the drawing.

Lectures:

GD&T - Gauges, Dimensioning, and Errors - 27 Minutes

Reading:

- **Tolerance** is the total amount a dimension may vary and is the difference between the **upper** (maximum) and **lower** (minimum) limits.
- Tolerances are used to control the amount of variation inherent in all manufactured parts. In particular, tolerances are assigned to mating parts in an assembly.
- One of the great advantages of using tolerances is that it allows for **interchangeable parts**, thus permitting the replacement of individual parts.
- Tolerances are used in production drawings to control the manufacturing process more accurately and control the variation between parts.
- Tolerance representation

- Direct limits or as tolerance values applied directly to a dimension.
- Geometric tolerances
- Notes referring to specific condition.

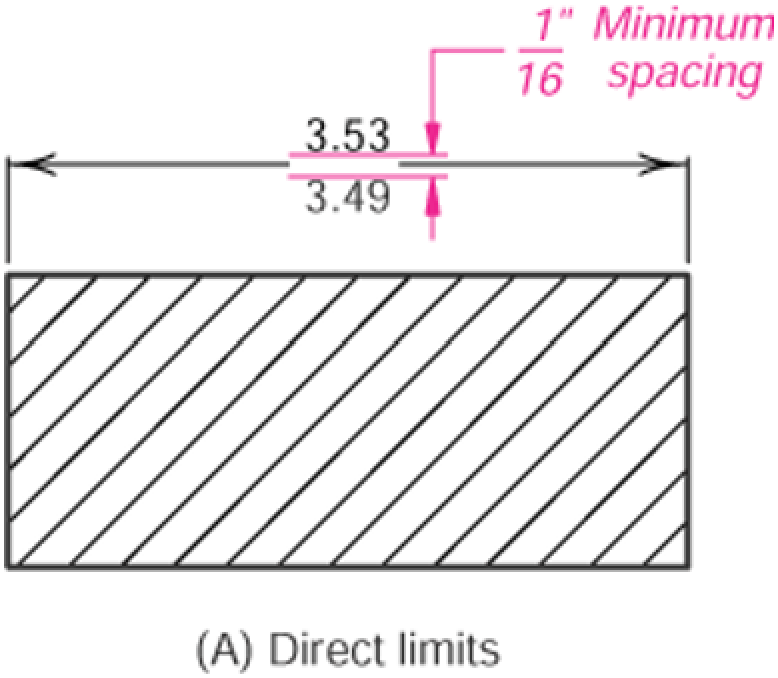


figure: Direct Limits

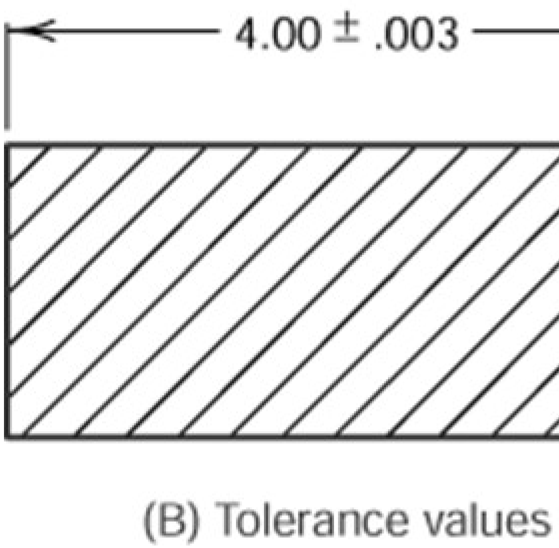


figure: Tolerance Values

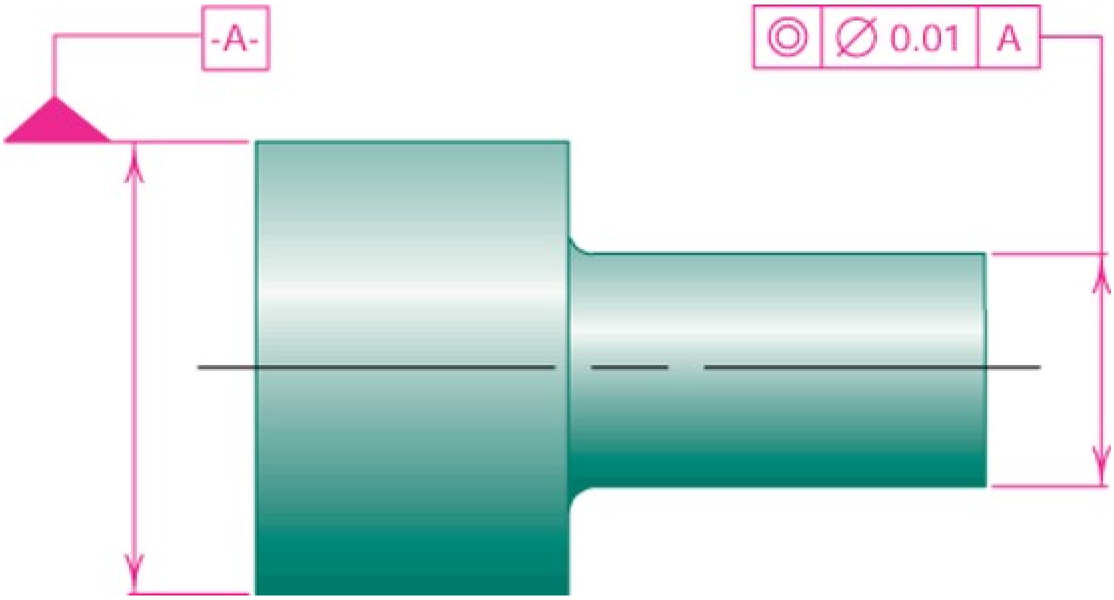


figure: Geometric tolerance

- **Tolerance representation**
    - Plus/Minus

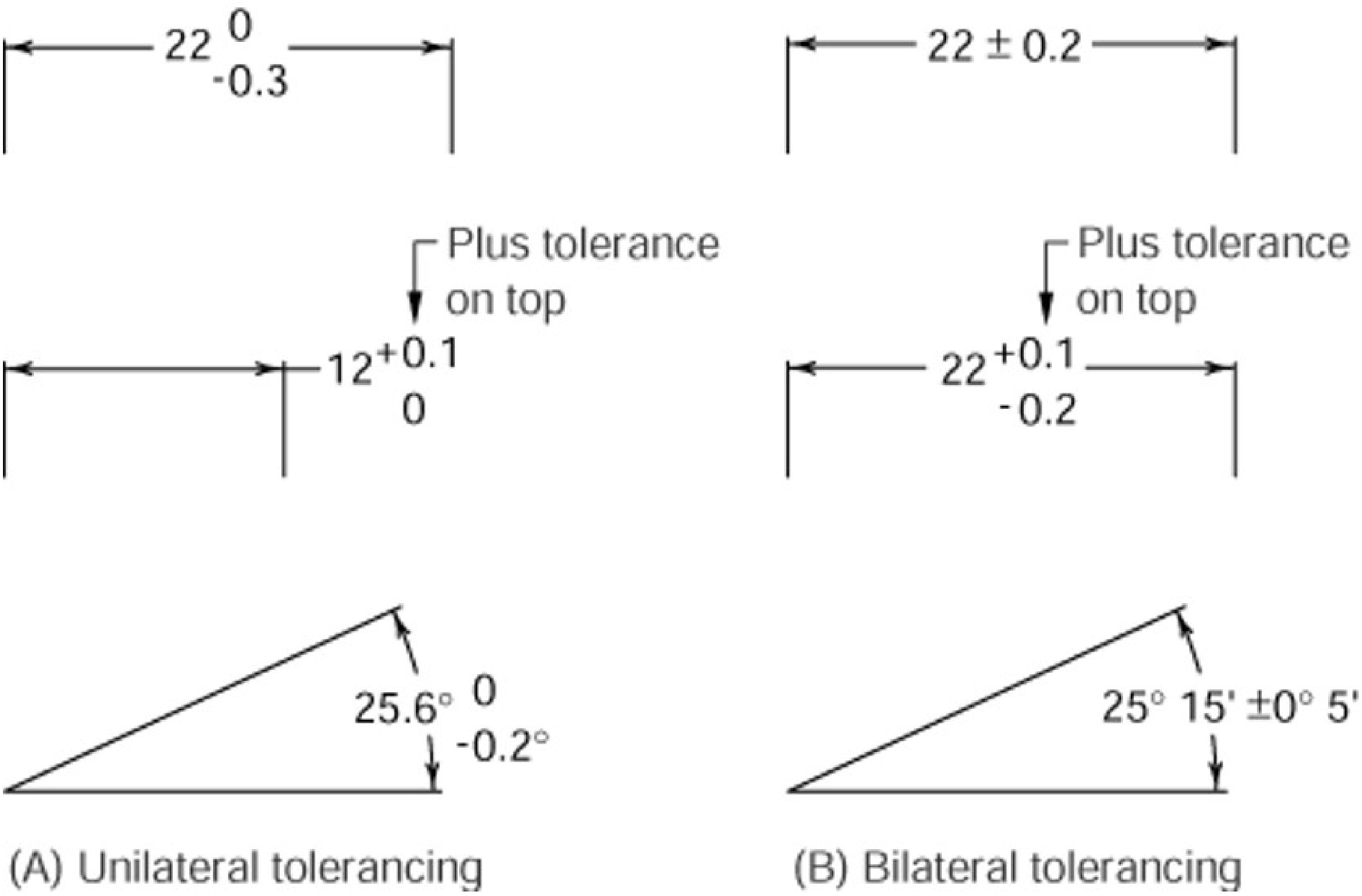


figure: Unilateral and Bilateral toleracing

Important terms

Nominal size a dimension used to describe the general size usually expressed in common fractions.

Basic size the theoretical size used as a starting point for the application of tolerances.

Actual size the measured size of the finished part after machining

Limits the maximum and minimum sizes shown by the toleranced dimension.

Allowance is the minimum clearance or maximum interference between parts.

Tolerance is the total variance in a dimension which is the difference between the upper and lower limits. The tolerance of the slot in Figure 14.50 is .004" and the tolerance of the mating part is .002".

Maximum material condition (MMC) is the condition of a part when it contains the most amount of material. The MMC of an external feature such as a shaft is the upper limit. The MMC of an internal feature such as a hole is the lower limit.

Least material condition (LMC) is the condition of a part when it contains the least amount of material possible. The LMC of an external feature is the lower limit of the part. The LMC of an internal feature is the upper limit of the part.

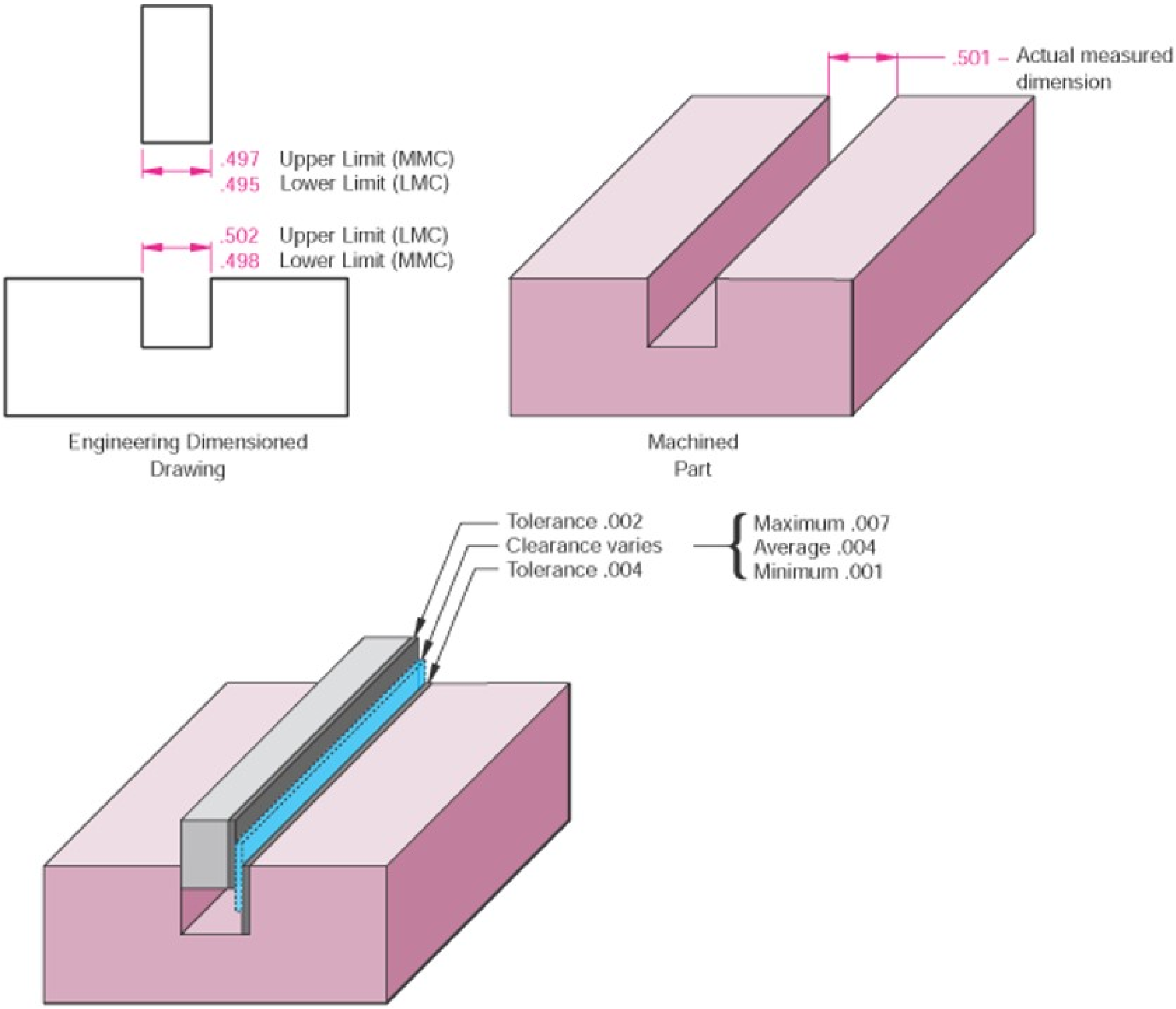


figure: Dimensioned drawing

Fit types

- **Clearance fit** occurs when two toleranced mating parts will always leave a space or clearance when assembled.
- **Interference fit** occurs when two toleranced mating parts will always interfere when assembled.
- **Transition fit** occurs when two toleranced mating parts will sometimes be an interference fit and sometimes be a clearance fit when assembled.

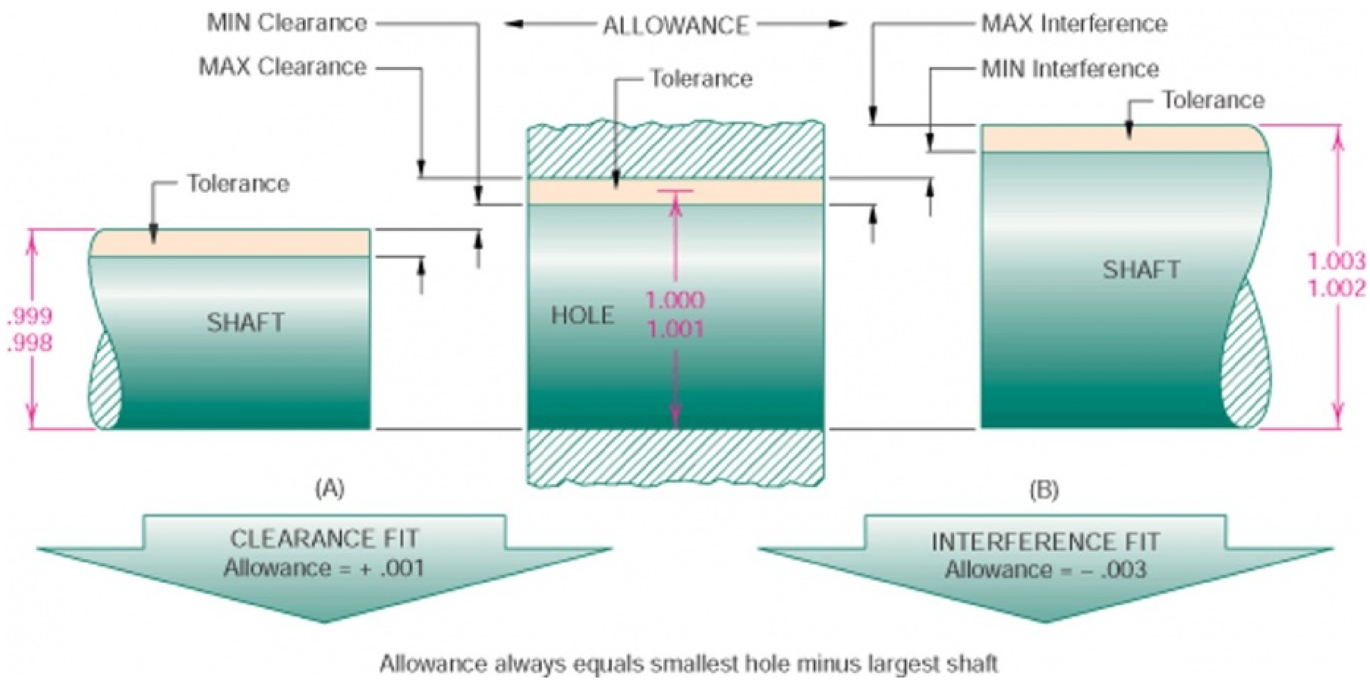


figure: Allowance A

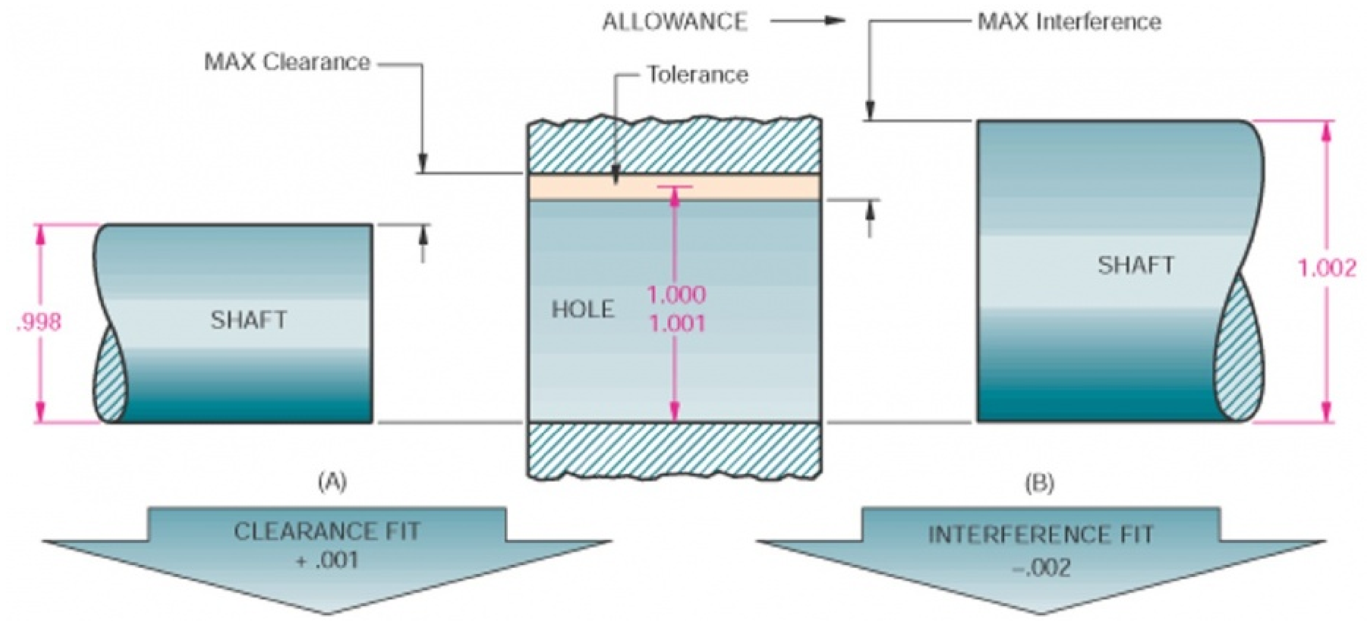


figure: Allowance B

Metric Limits and Fits

- Basic size
- Deviation
- Upper Deviation
- Lower Deviation

- Fundamental Deviation

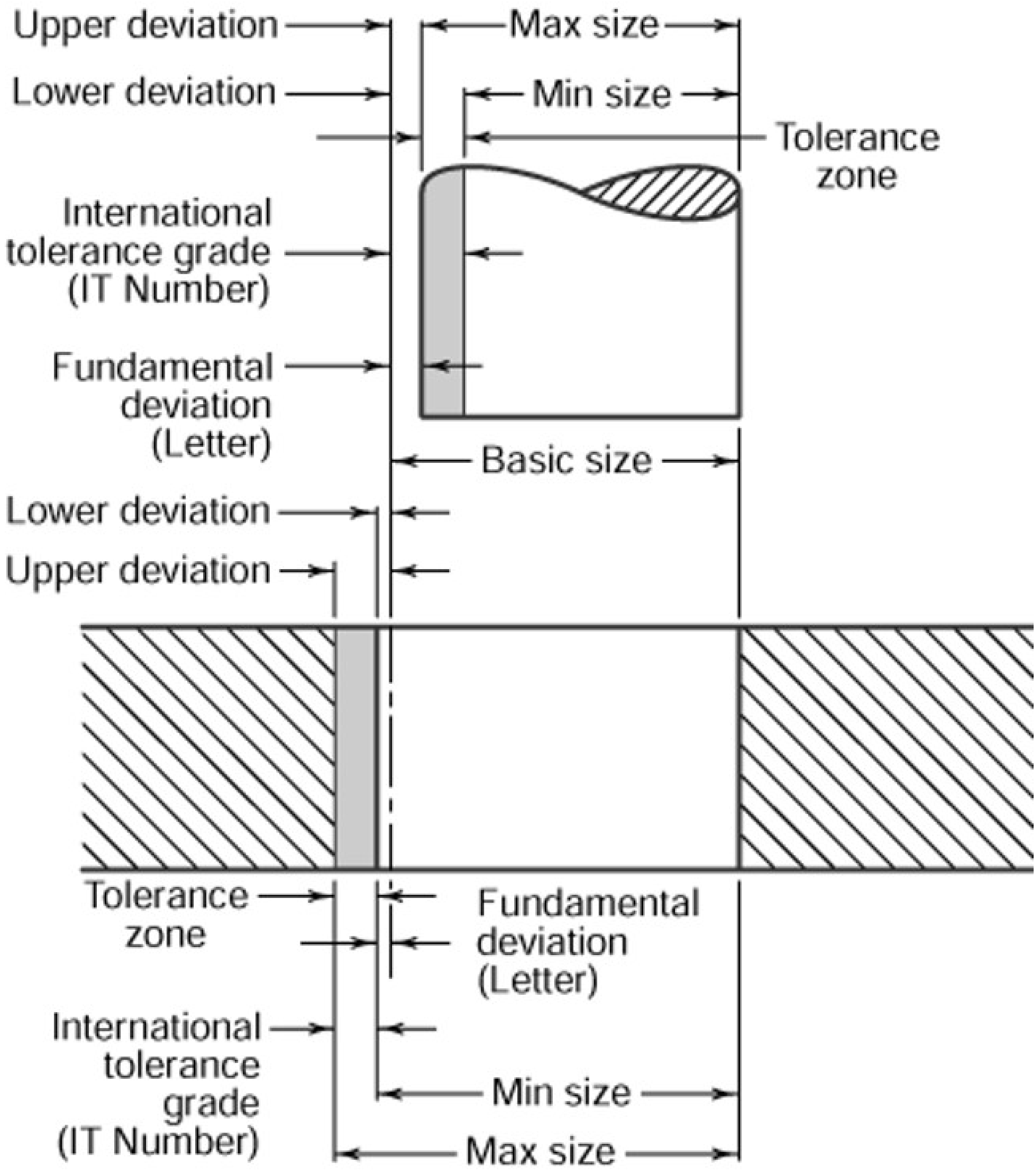


figure: Deviation

- Tolerance
- Tolerance zone
- International tolerance grade
- Hole basis
- Shaft basis

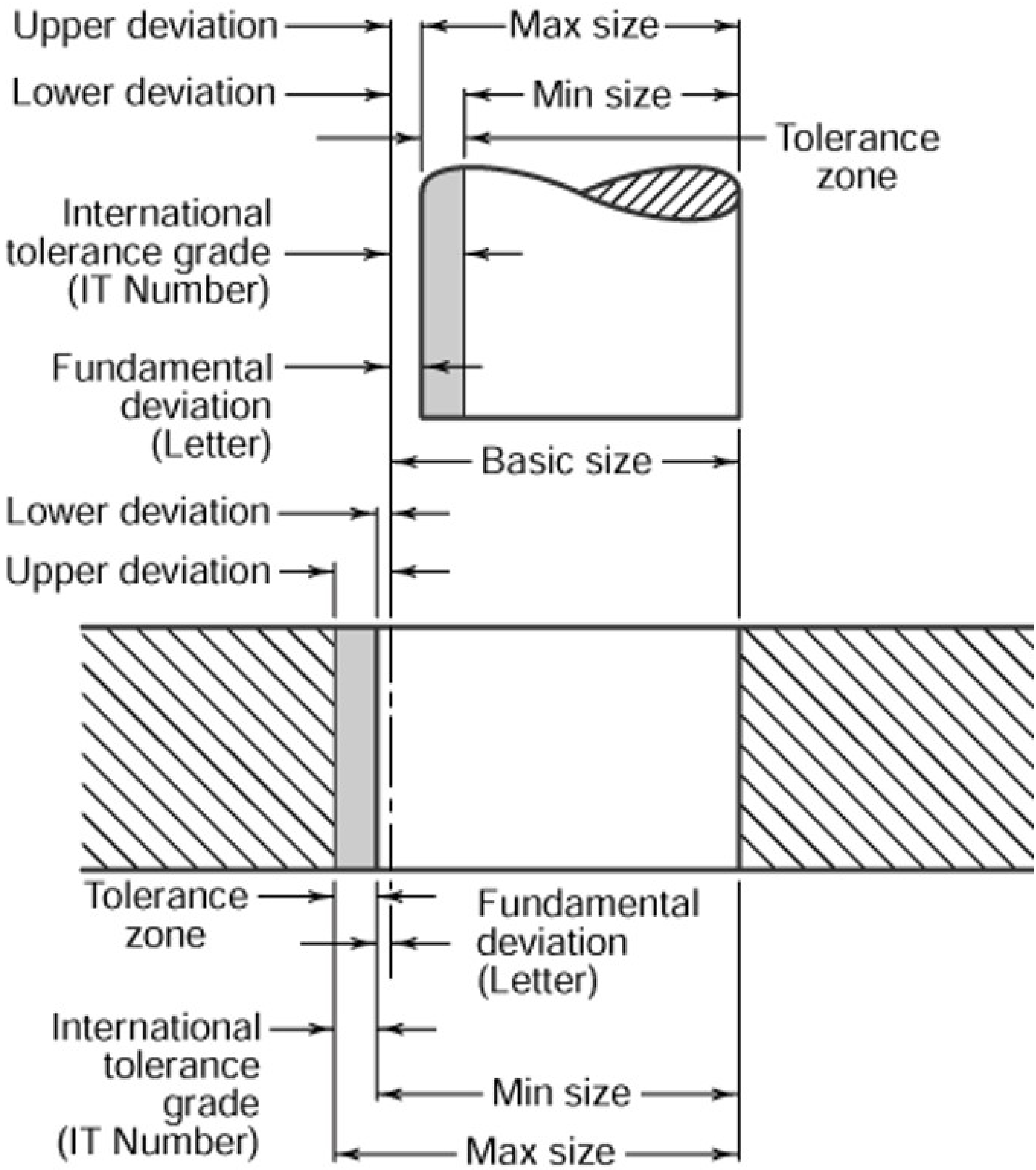


figure: Deviation B

- Symbols and Definitions
- Methods

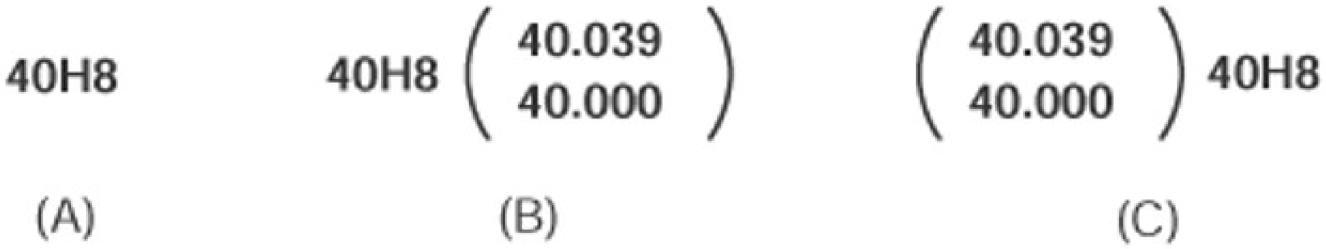


figure: Symbols and Definitions A

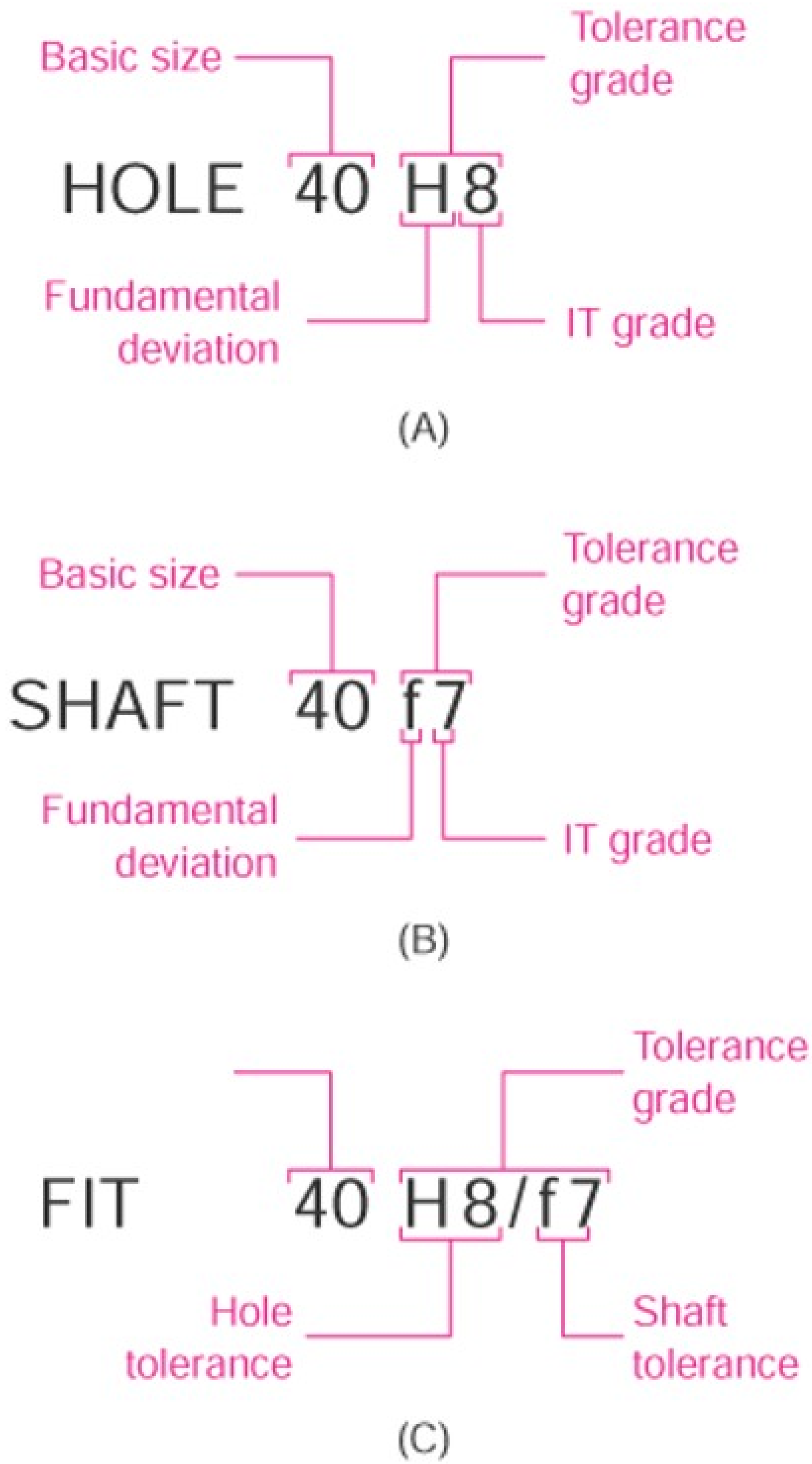


figure: Symbols and Definitions B

- Standard Hole basis table; limits

BASIC SIZE		LOOSE RUNNING Hole H11	LOOSE RUNNING Shaft c11	LOOSE RUNNING Fit	FREE RUNNING Hole H9	FREE RUNNING Shaft d9	FREE RUNNING Fit	CLOSE RUNNING Hole H8	CLOSE RUNNING Shaft f7	CLOSE RUNNING Fit	SLIDING Hole H7	SLIDING Shaft g6	SLIDING Fit	LOCATIONAL CLEARANCE Hole H7	LOCATIONAL CLEARANCE Shaft h6	LOCATIONAL CLEARANCE Fit
40	MAX	40.160	39.880	0.440	40.062	39.920	0.204	40.039	39.975	0.029	40.025	39.991	0.050	40.025	40.000	0.041
	MIN	40.000	39.720	0.120	40.000	39.858	0.060	40.000	39.950	0.025	40.000	39.975	0.009	40.000	39.984	0.000
50	MAX	50.160	49.870	0.450	50.062	49.920	0.204	50.039	49.975	0.089	50.025	49.991	0.050	50.025	50.000	0.041
	MIN	50.000	49.710	0.130	50.000	49.858	0.080	50.000	49.950	0.025	50.000	49.975	0.009	50.000	49.984	0.000
60	MAX	60.190	59.860	0.520	60.074	59.900	0.248	60.046	59.970	0.106	60.030	59.990	0.059	60.030	60.000	0.049
	MIN	60.000	59.670	0.140	60.000	59.826	0.100	60.000	59.940	0.030	60.000	59.971	0.010	60.000	59.981	0.000
80	MAX	80.190	79.550	0.530	80.074	79.900	0.248	80.046	79.970	0.106	80.030	79.990	0.059	80.030	80.000	0.049
	MIN	80.000	79.660	0.150	80.000	79.826	0.100	80.000	79.940	0.030	80.000	79.971	0.010	80.000	79.981	0.000
100	MAX	100.220	99.830	0.610	100.087	99.880	0.294	100.054	99.964	0.125	100.035	99.988	0.069	100.035	100.000	0.057
	MIN	100.000	99.610	0.170	100.000	99.793	0.120	100.000	99.929	0.036	100.000	99.966	0.012	100.000	99.978	0.000
120	MAX	120.220	119.820	0.620	120.087	119.880	0.294	120.054	119.964	0.125	120.035	119.988	0.069	120.035	120.000	0.057
	MIN	120.000	119.600	0.180	120.000	119.793	0.120	120.000	119.929	0.036	120.000	119.966	0.012	120.000	119.978	0.000
160	MAX	160.250	159.790	0.710	160.100	159.855	0.345	160.063	159.957	0.146	160.040	159.986	0.078	160.040	160.000	0.065
	MIN	160.000	159.540	0.210	160.000	159.755	0.145	160.000	159.917	0.043	160.000	159.961	0.014	160.000	159.975	0.000
200	MAX	200.290	199.760	0.820	200.115	199.830	0.400	200.072	199.950	0.168	200.046	199.985	0.040	200.046	200.000	0.075
	MIN	200.000	199.470	0.240	200.000	199.715	0.170	200.000	199.904	0.050	200.000	199.956	0.015	200.000	199.971	0.000
250	MAX	250.290	249.720	0.860	250.115	249.830	0.400	250.072	249.950	0.168	250.046	249.985	0.090	250.046	250.000	0.075
	MIN	250.000	249.430	0.280	250.000	249.715	0.170	250.000	249.904	0.050	250.000	249.956	0.015	250.000	249.971	0.000
300	MAX	300.320	299.670	0.970	300.130	299.810	0.450	300.081	299.944	0.189	300.052	299.983	0.101	300.052	300.000	0.084
	MIN	300.000	299.350	0.330	300.000	299.680	0.190	300.000	299.892	0.056	300.000	299.951	0.017	300.000	299.968	0.000
400	MAX	400.360	399.600	1.120	400.140	399.790	0.490	400.089	399.938	0.208	400.057	399.982	0.111	400.057	400.000	0.093
	MIN	400.000	399.240	0.400	400.000	399.650	0.210	400.000	399.881	0.062	400.000	399.946	0.018	400.000	399.964	0.000
500	MAX	500.400	499.520	1.280	500.155	499.770	0.540	500.097	499.932	0.228	500.063	499.980	0.123	500.063	500.000	0.103
	MIN	500.000	499.120	0.480	500.000	499.615	0.230	500.000	499.869	0.068	500.000	499.940	0.020	500.000	499.960	0.000

figure: Standard Hole basis table; limits

- Hole basis system; fits

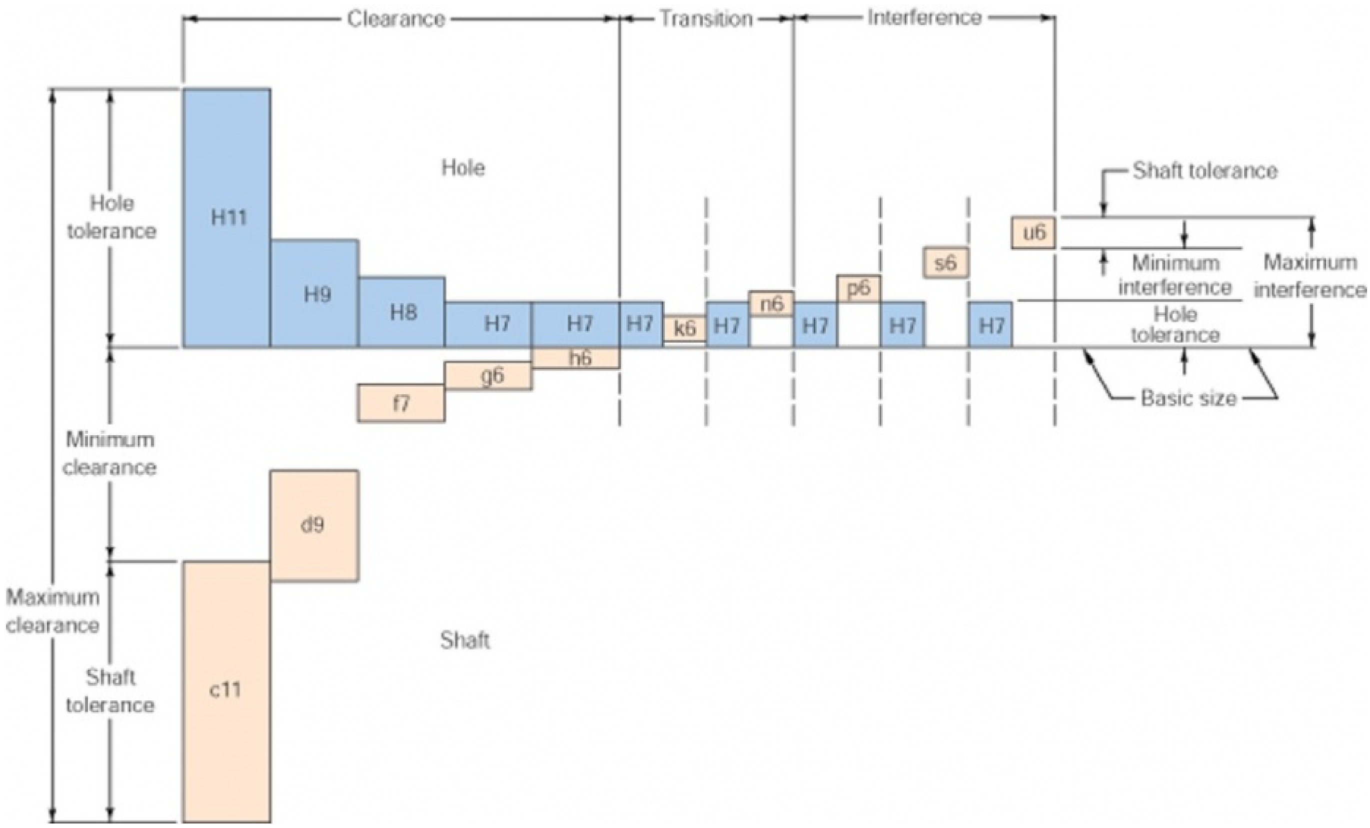


figure: Hole basis system; fits

- Shaft basis system; fits

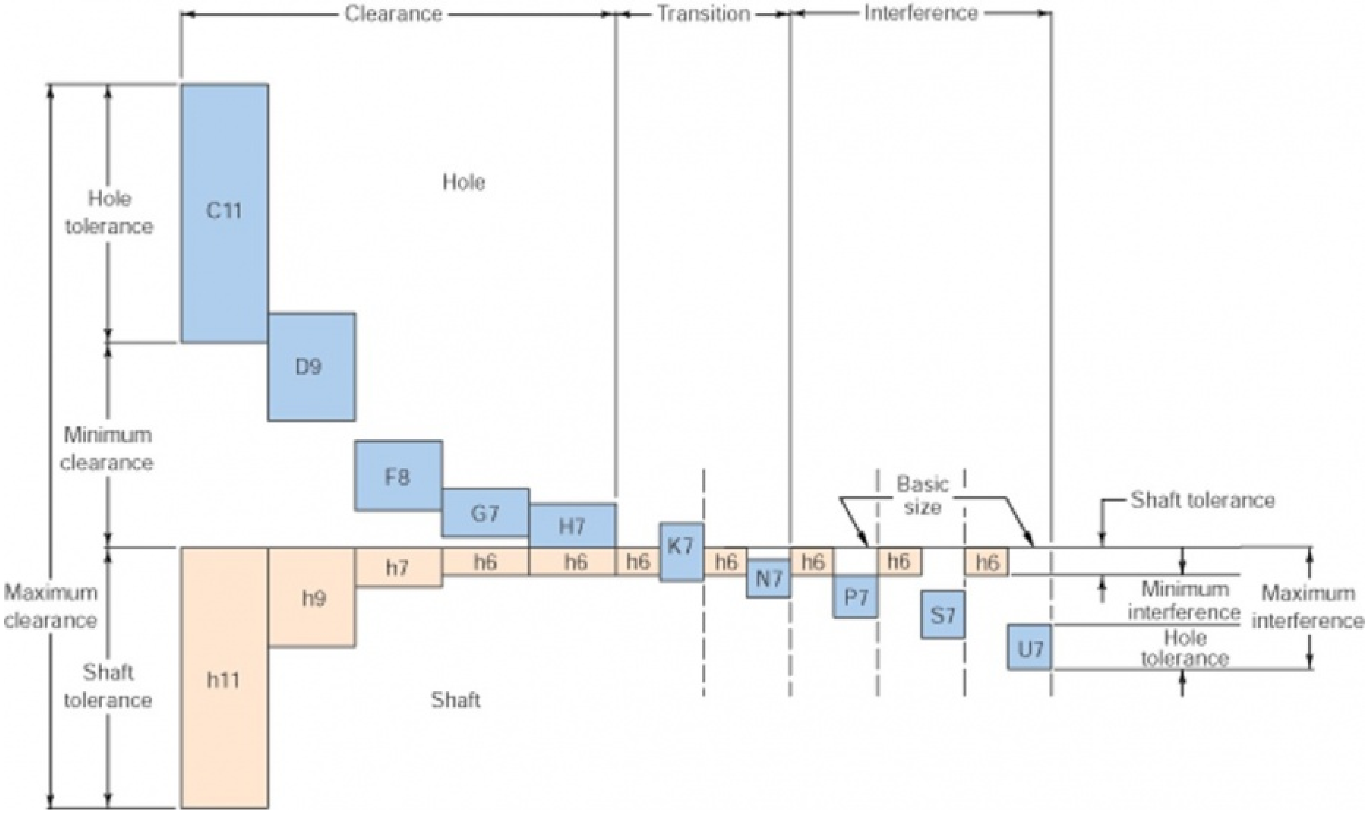


figure: Shaft basis system; fits

Standard Precision Fit; English Units

- Running and Sliding (RC)
- Clearance Locational (LC)
- Transition Locational (LT)
- Interference Locational (LN)
- Force and Shrinks (FN)

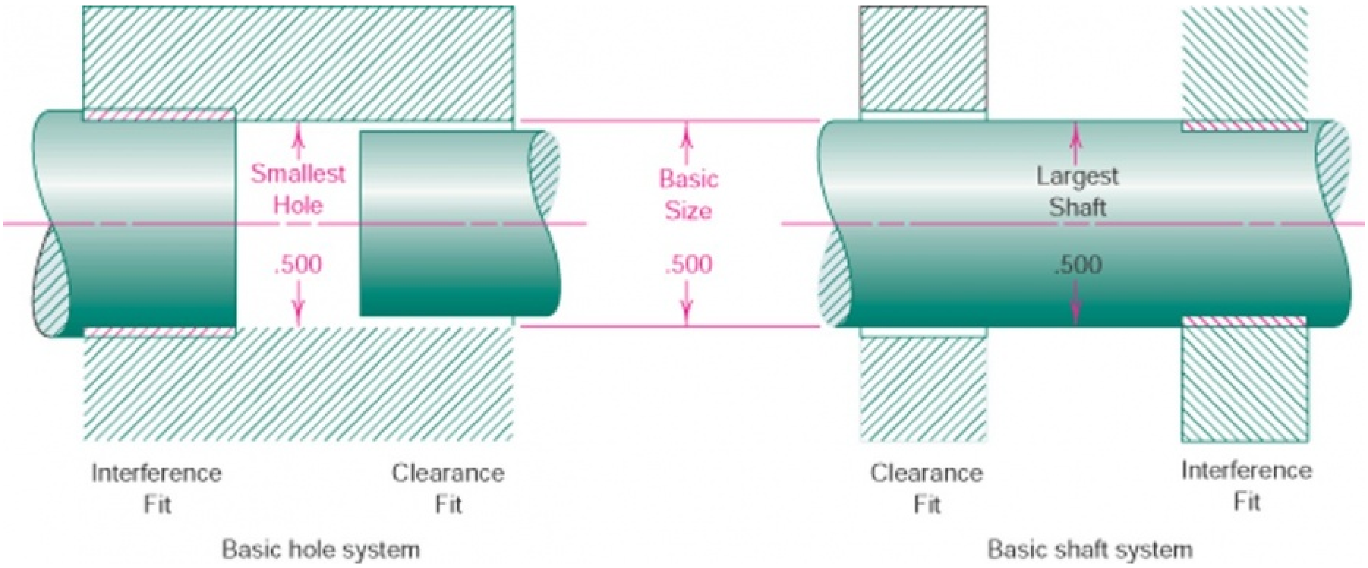


figure: Basic hole / shaft systems

Geometric Dimensioning and Tolerancing

GDT is a method of defining parts based on how they function, using standard ASME ANSI symbols.

Within the last 15 years there has been considerable interest in GDT, in part because of the increased popularity of statistical process control. This control process, when combined with GDT, helps reduce or eliminate inspection of features on the manufactured object. The flipside is that the part must be toleranced very efficiently; this is where GDT comes in.

Another reason for the increased popularity of GDT is the rise of worldwide standards, such as ISO 9000, which require universally understood and accepted methods of documentation.

GDT-Symbols

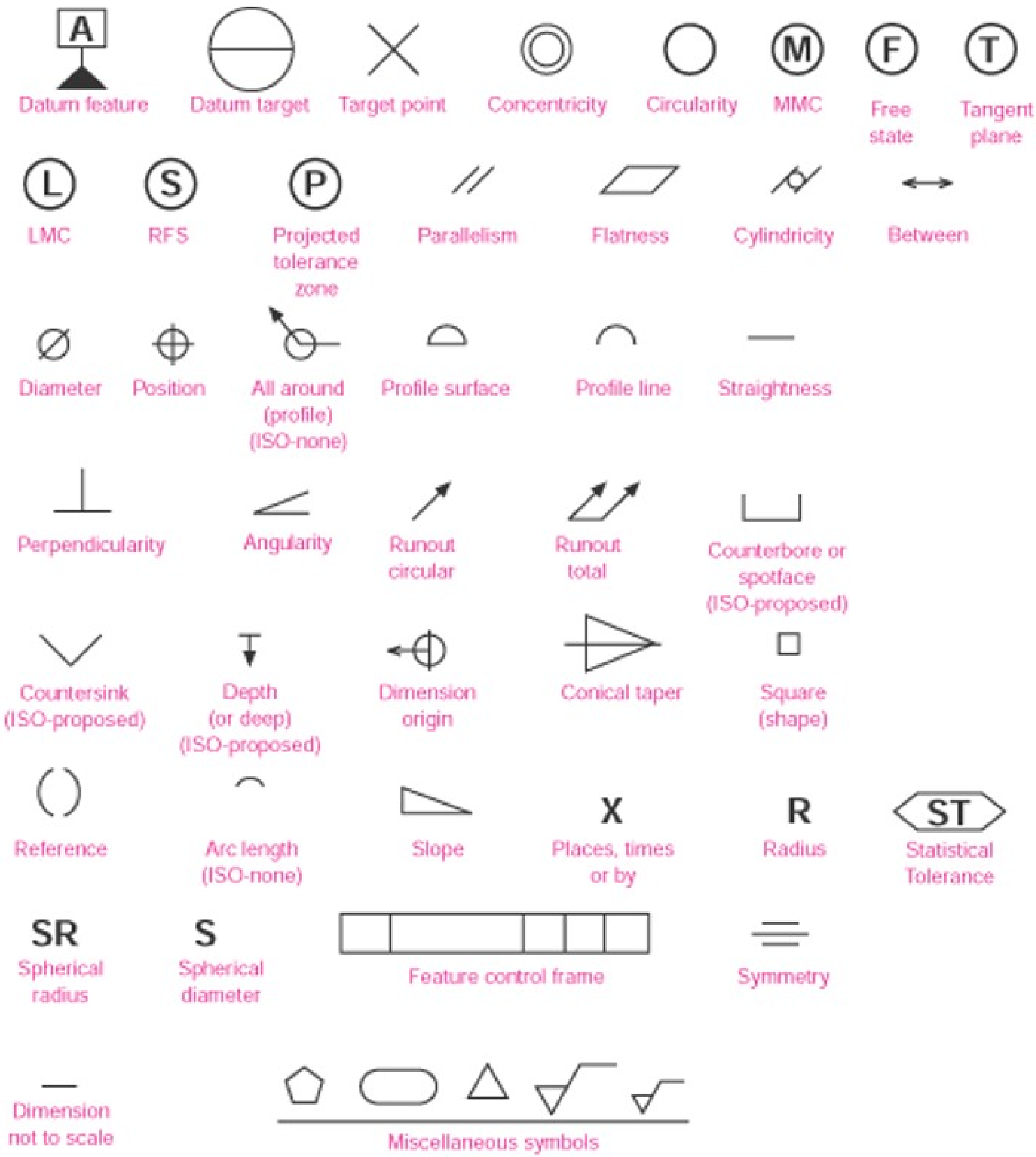


figure: GDT-Symbols

- Feature control frames

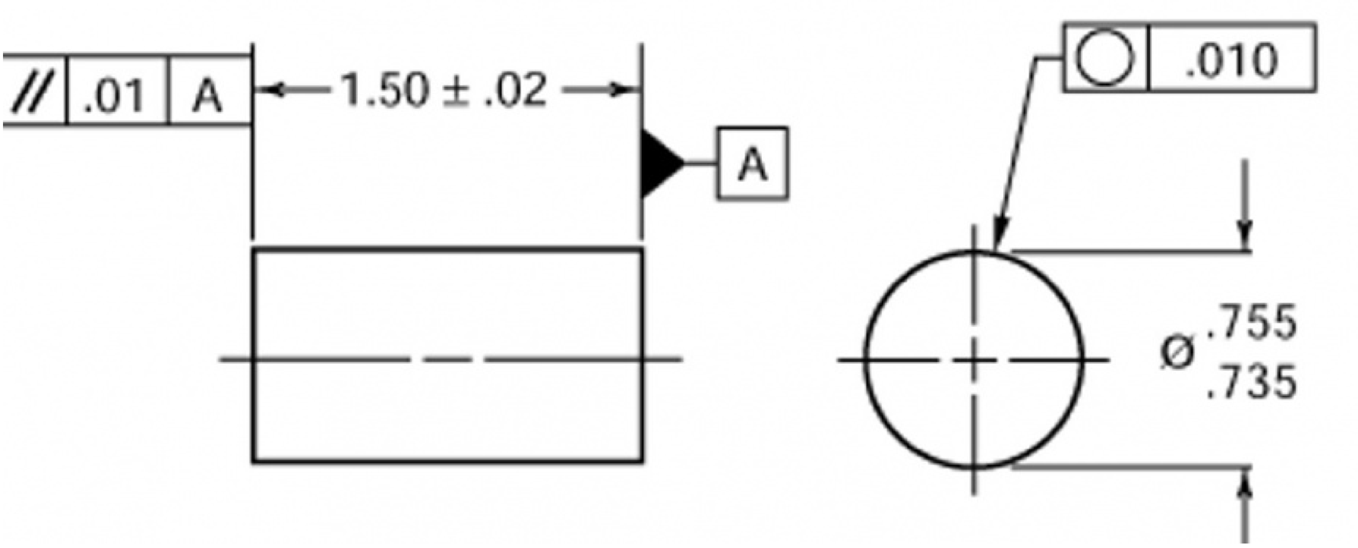


figure: Feature control frames

- MMC/LMC
- Datums
- Geometric Controls
    - Form
    - Orientation
    - Position

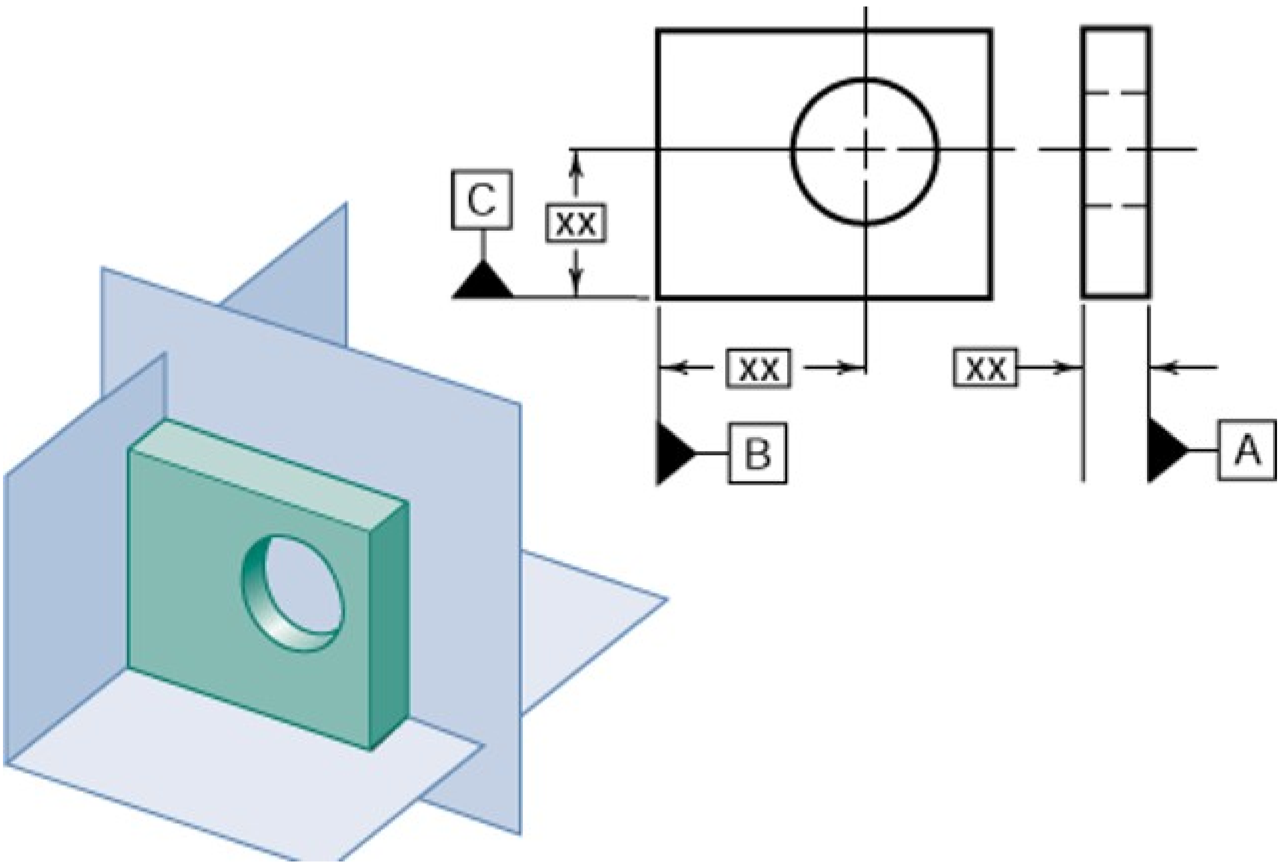


figure: GDT

Forms

- Straightness
  - Line element
  - Axis

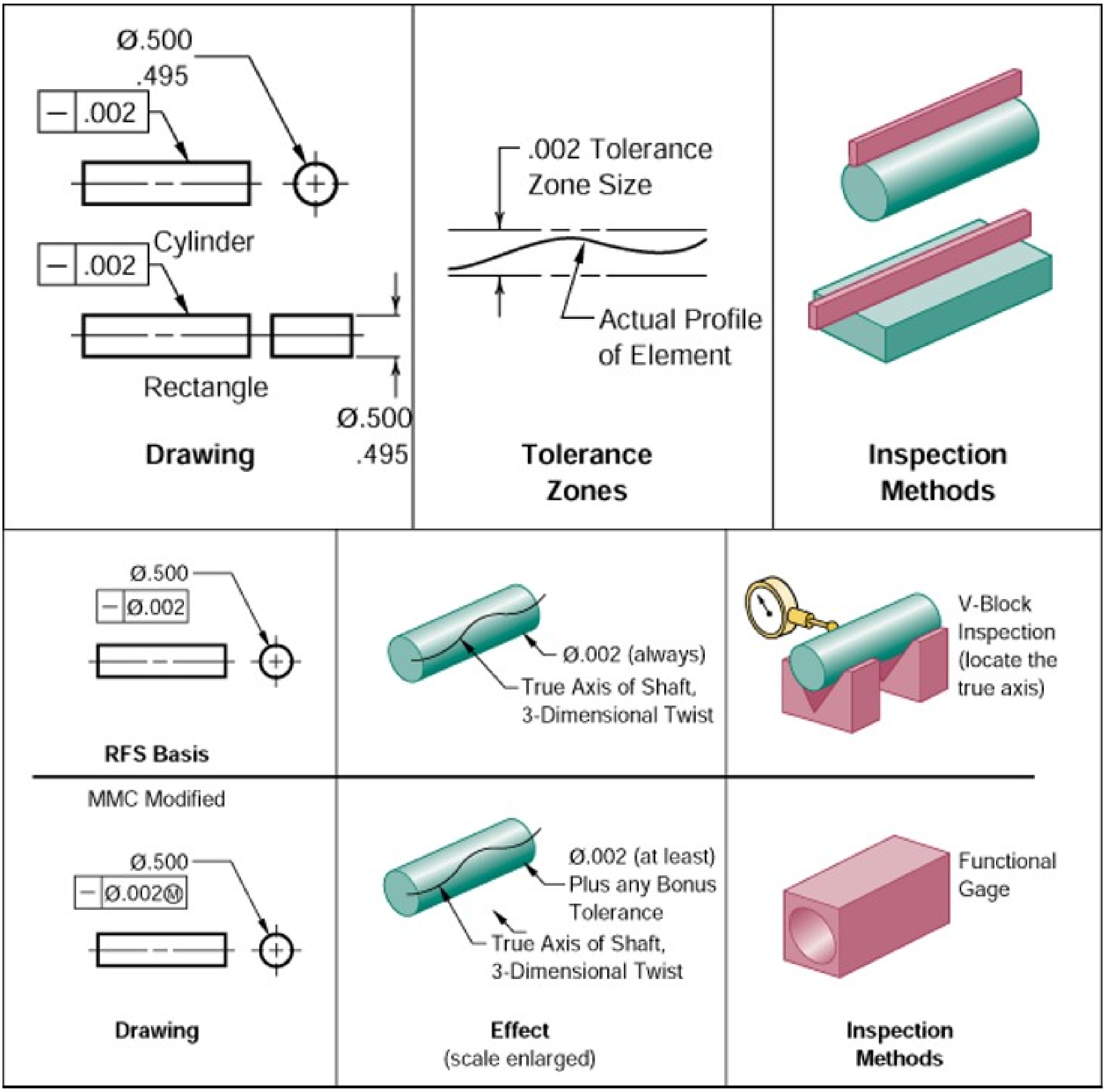


figure: GDT Forms Straightness

- Circularity

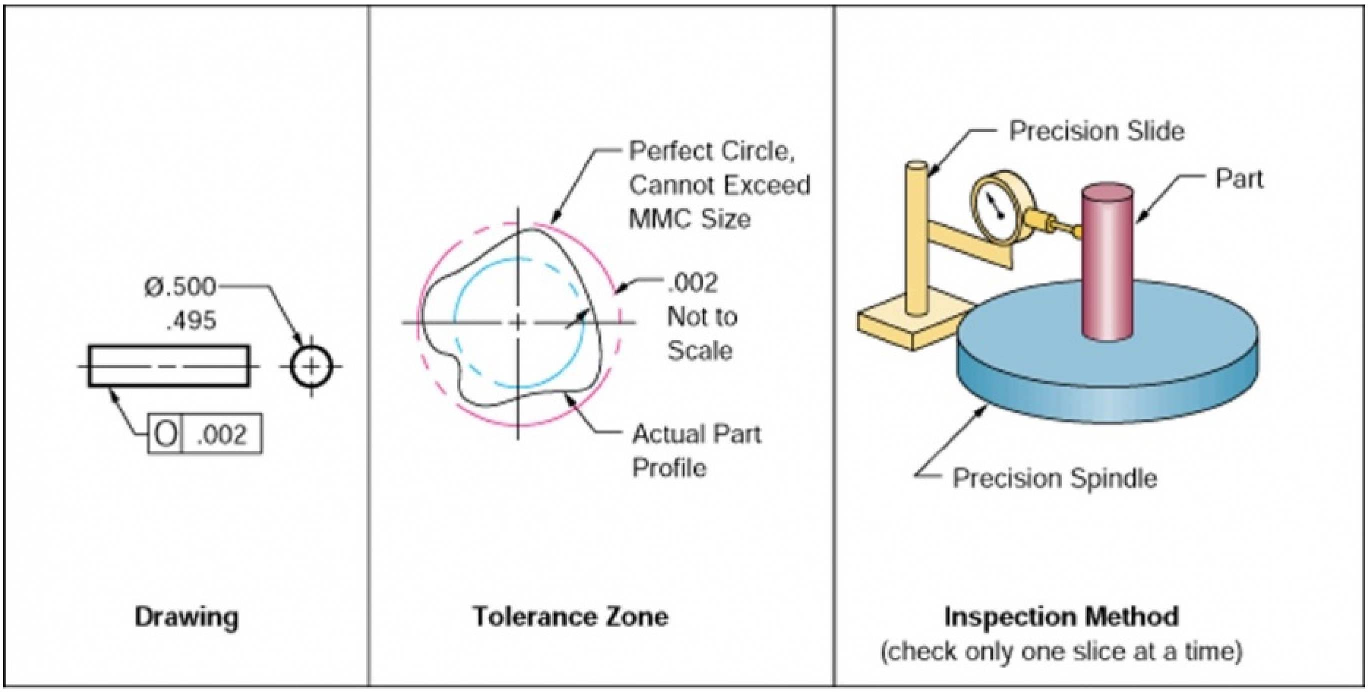


figure: GDT Forms Circularity

- Flatness

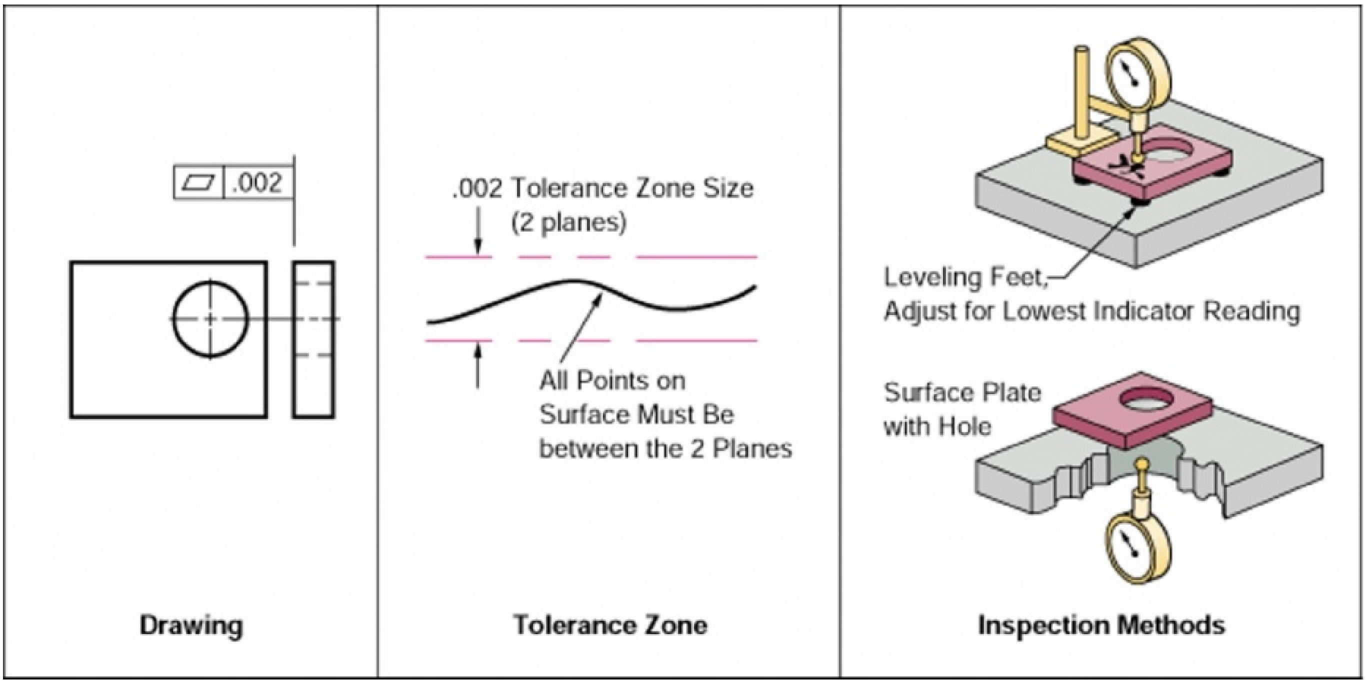


figure: GDT Forms Flatness

- Cylindricity

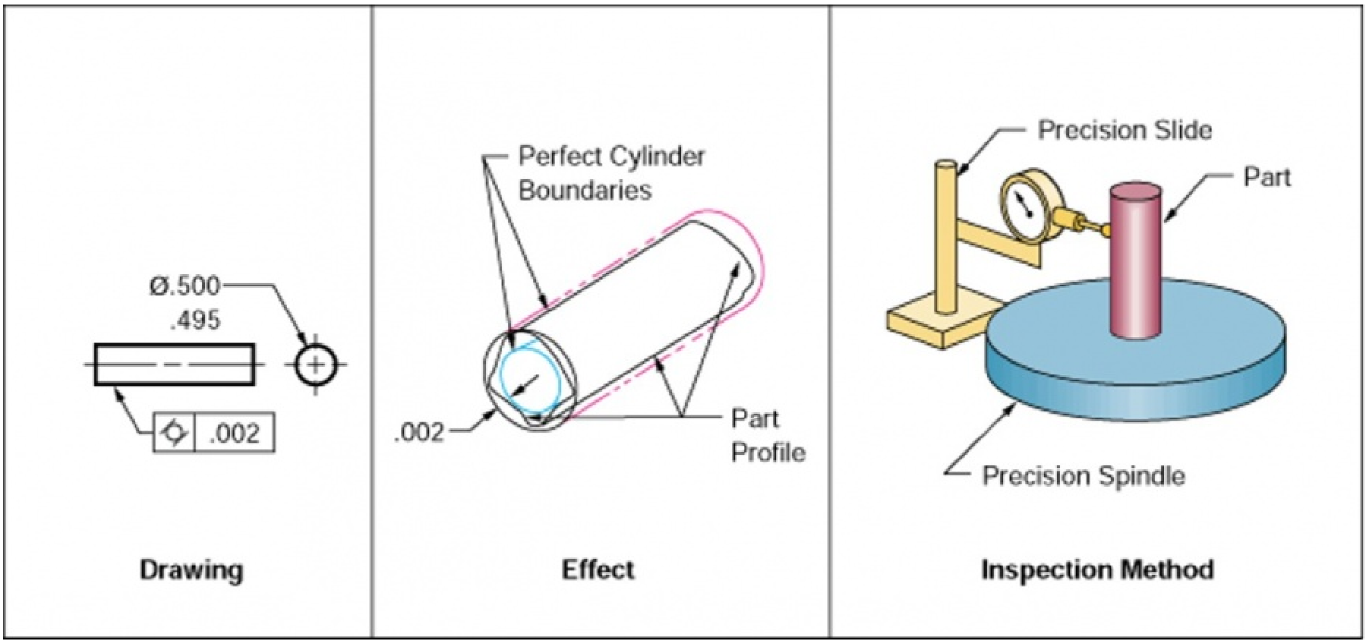


figure: GDT Forms Cylindricity

- Orientation Parallelism

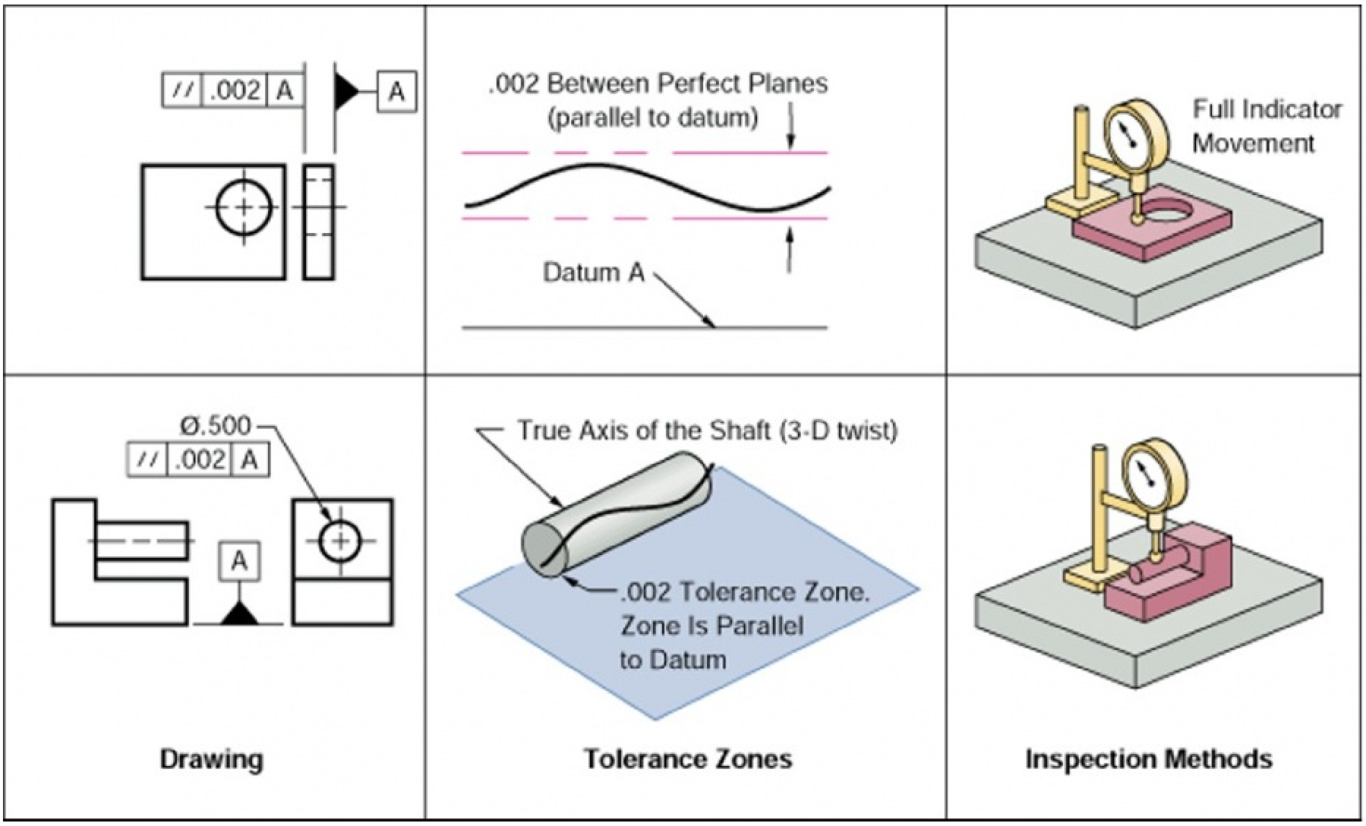


figure: GDT Orientation Parallelism

- Orientation Perpendicularity

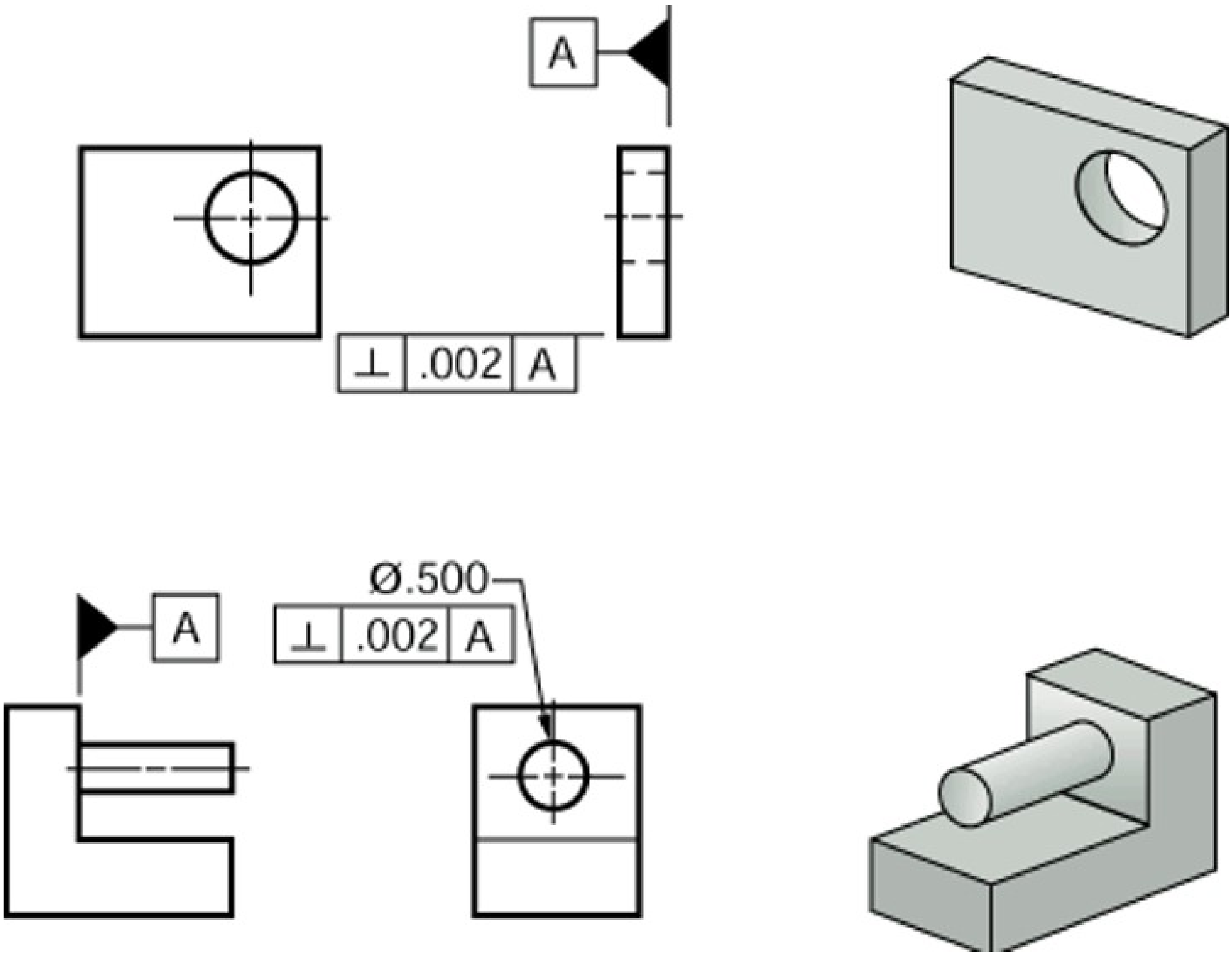


figure: GDT Orientation Perpendicularity

- Orientation Angularity

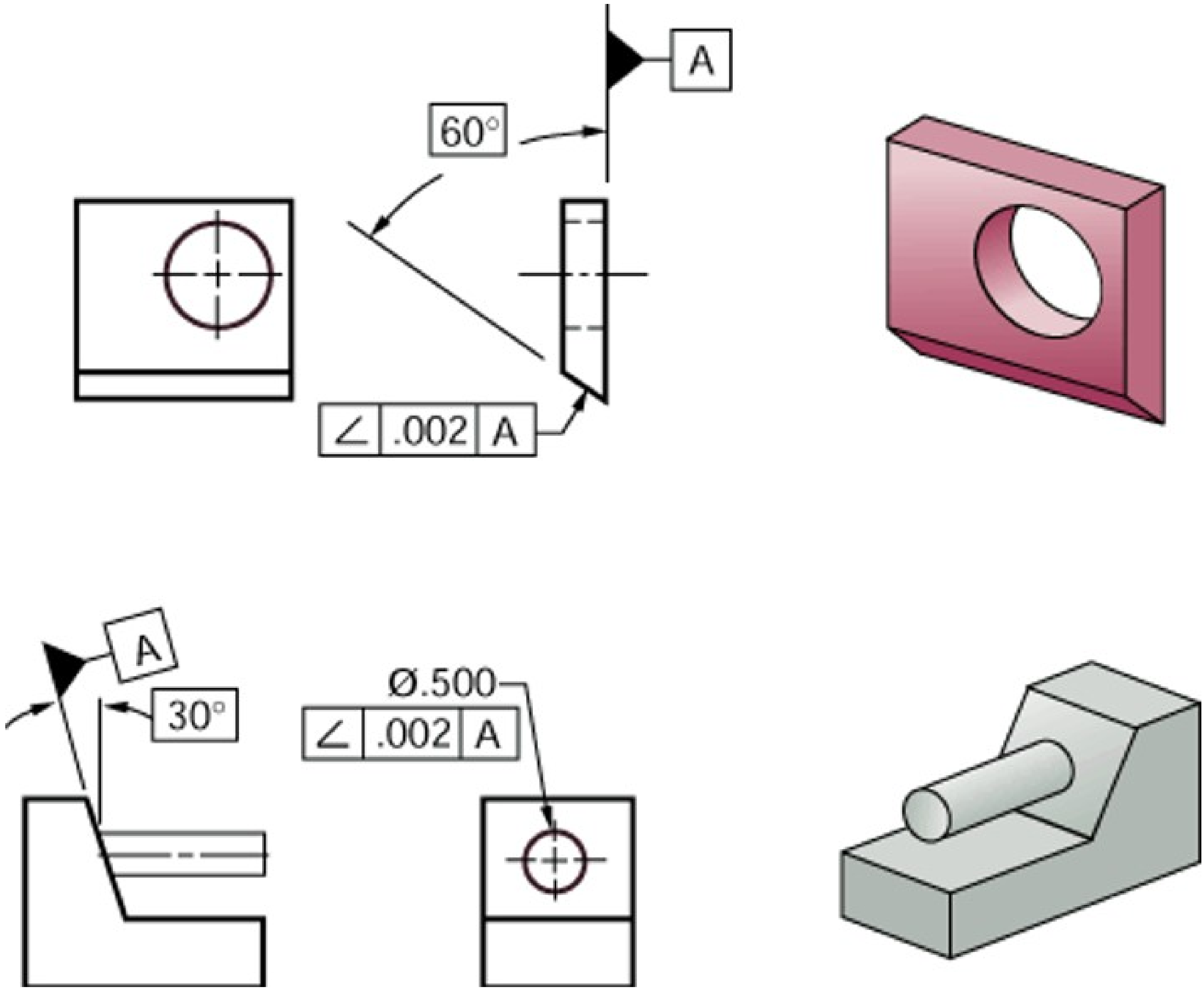


figure: GDT Orientation Angularity

- Orientation Line Profile

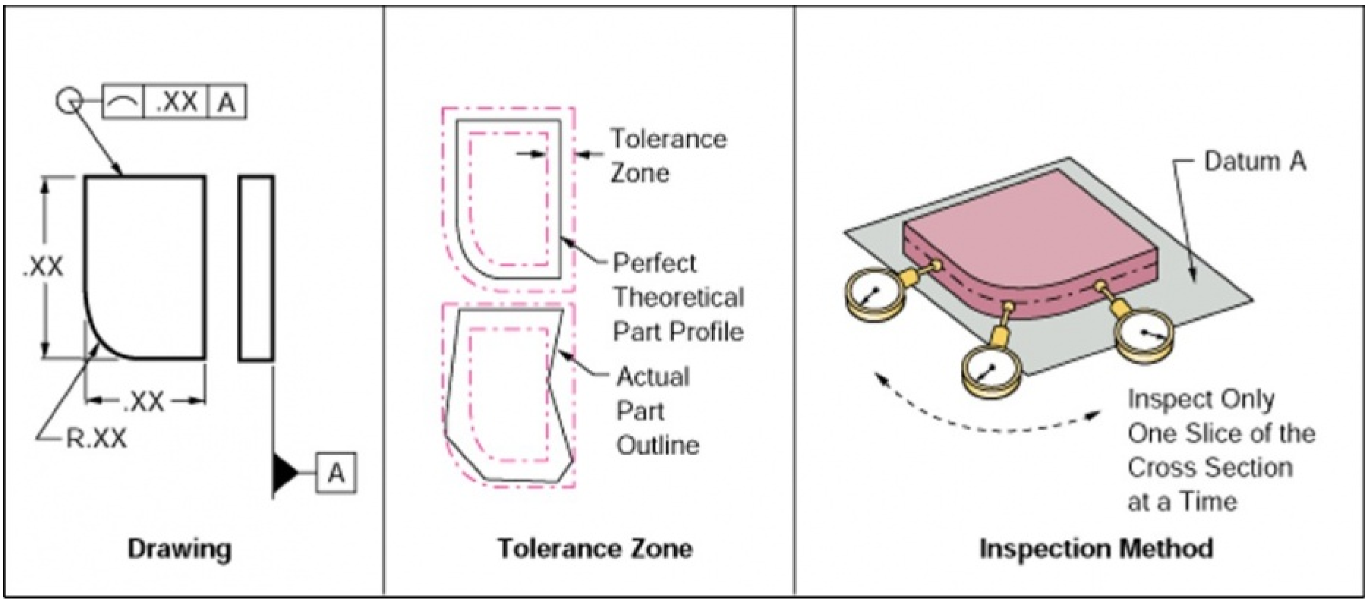


figure: GDT Orientation Line Profile

- Orientation Surface Profile

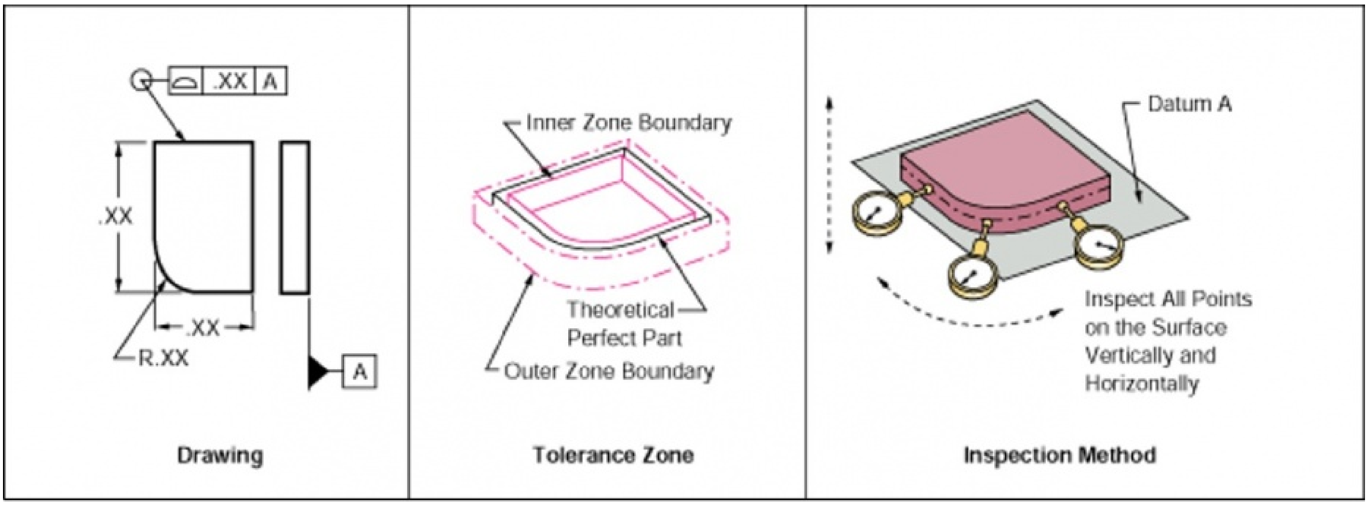


figure: GDT Orientation Surface Profile

- Location Concentricity

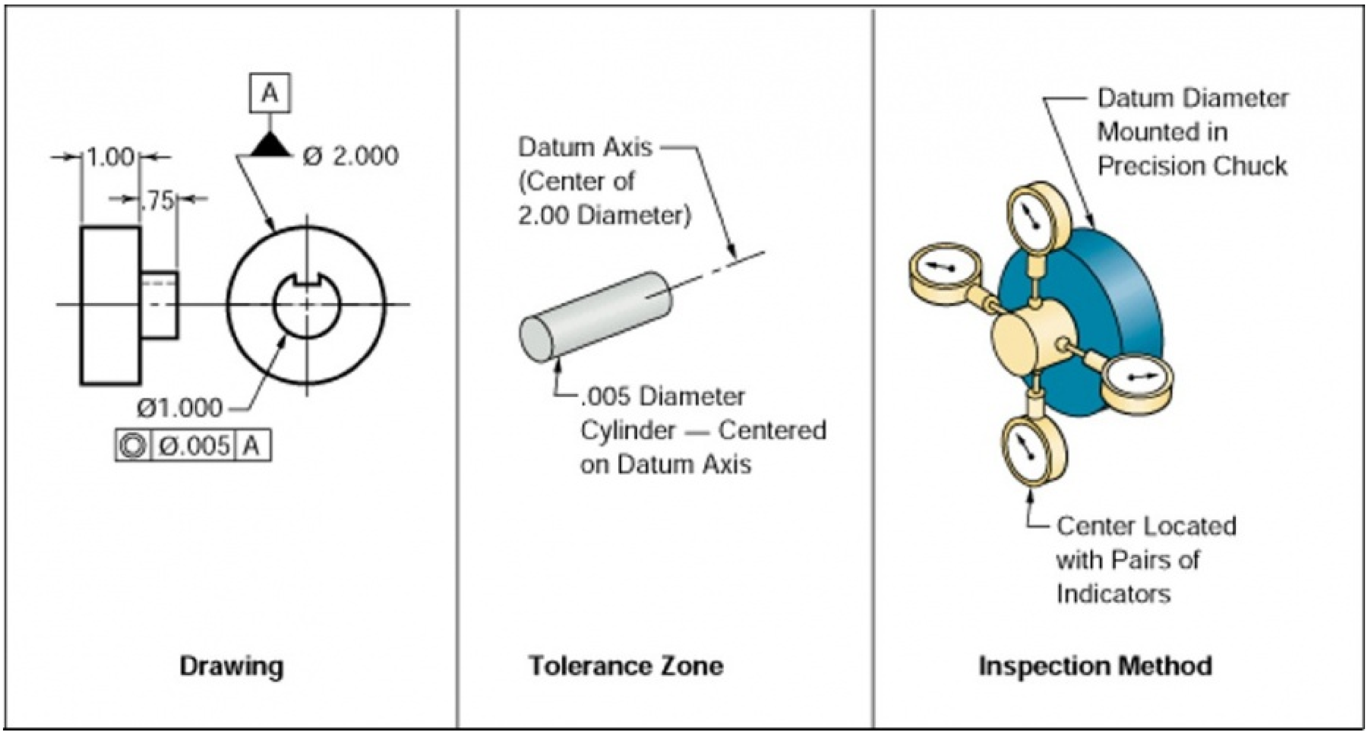


figure: GDT Location Concentricity

- Location Runout

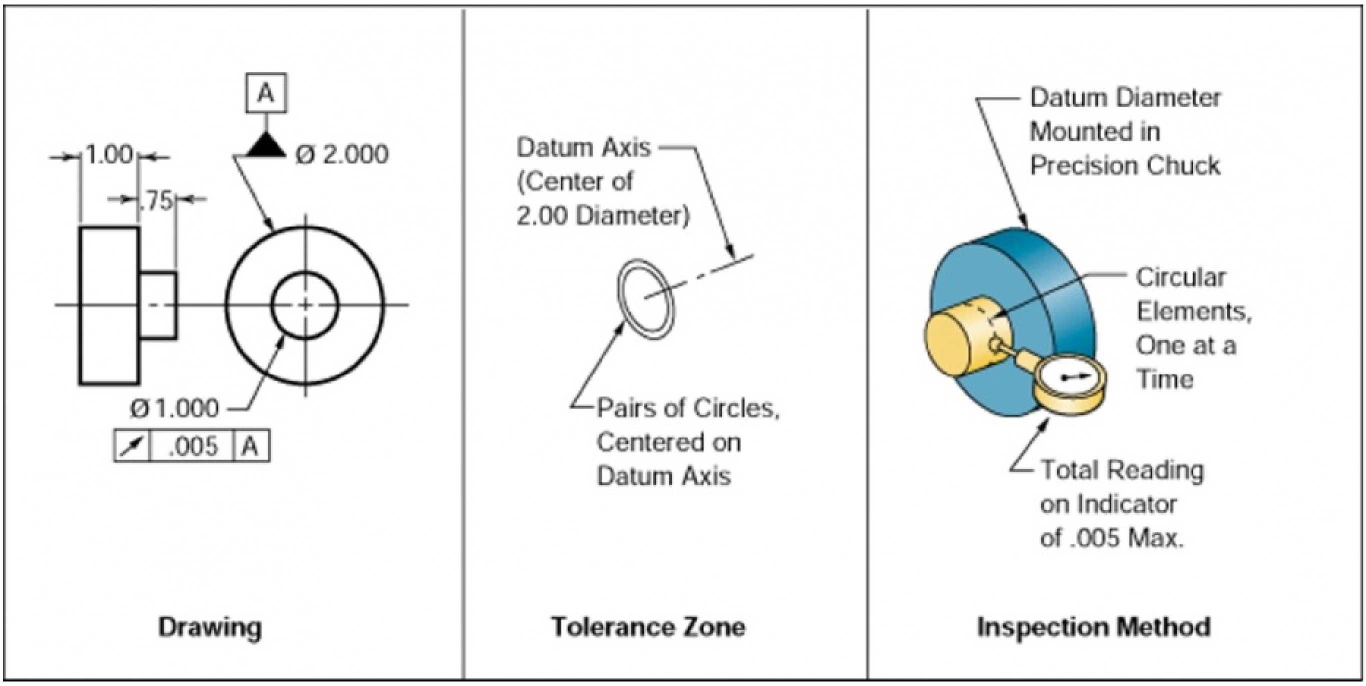


figure: GDT Location Runout

- Location Position

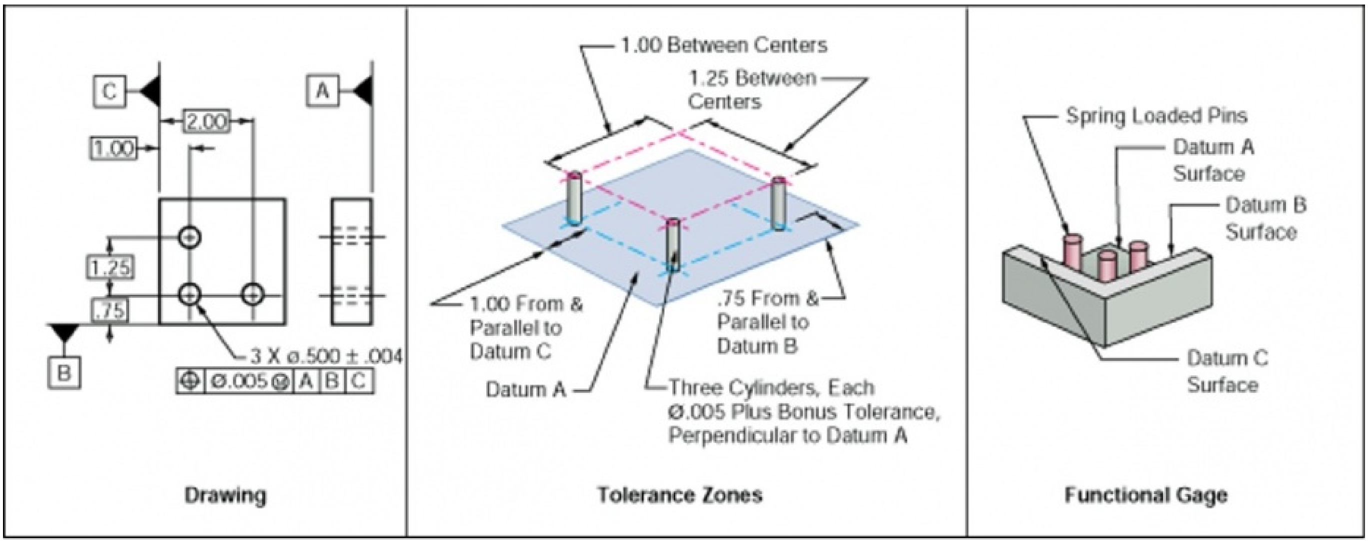


figure: GDT Location Position

- Location Position

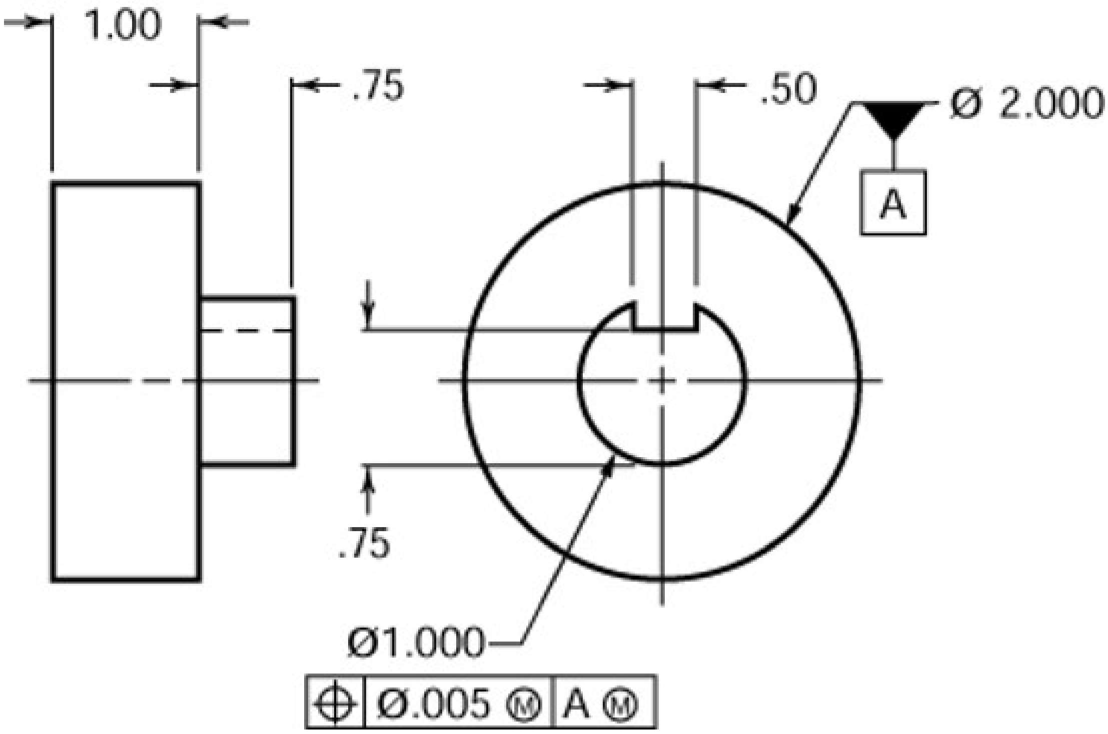


figure: GDT Location Position B

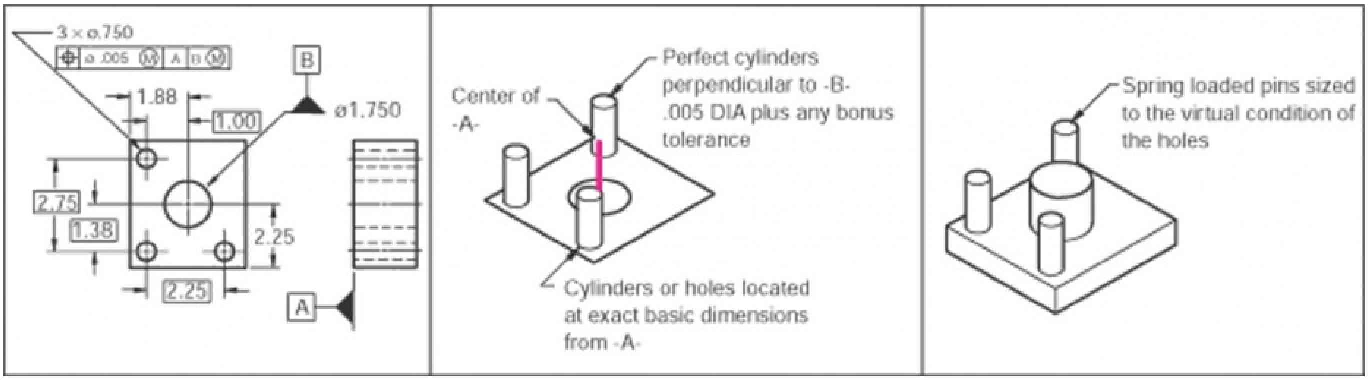


figure: GDT Location Position C

Tolerance Calculation

- **Floating fastener tolerancing** is used to confirm that loose bolts, screws or other fasteners have the standard clearance in their holes.
- **Fixed fastener tolerancing** is measured the same as with floating fasteners except that the fastener is already fixed/located on one of the mating parts and the tolerance is now divided between the parts.
- **Hole diameter tolerancing** is used to calculate the MMC of the hole.

Design Application

- Five-Step
    - Isolate and define the functions of the features/part.
    - Prioritize the functions.

- Identify the datum reference frame based on functional priorities.
- Select the proper control(s).
- Calculate the tolerance values.