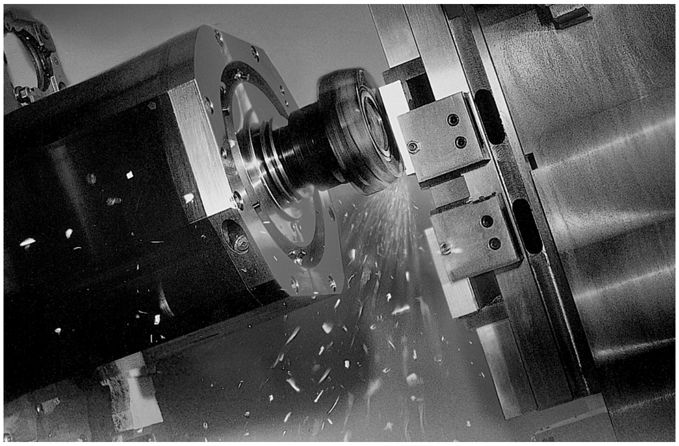


Chapter 3

Understanding Drawings

LEARNING OBJECTIVES

After studying this chapter, students will be able to:
- ❍ Read drawings that are dimensioned in fractional inches, decimal inches, and in metric units.
- ❍ Explain the information found on a typical drawing.
- ❍ Describe how detail, subassembly, and assembly drawings differ.
- ❍ Point out why drawings are numbered.
- ❍ Explain the basics of geometric dimensioning and tolerancing.

INSTRUCTIONAL MATERIALS

Text: pages 29–54
Test Your Knowledge Questions, pages 52–53
Workbook: pages 15–22
Instructor's Resource: pages 57–70
Guide for Lesson Planning
Research and Development Ideas
Reproducible Masters:
3-1 Typical Assembly Drawing
3-2 Alphabet of Lines
3-3 Threads (*how depicted on a drawing*)
3-4 Information on a Typical Drawing
3-5 Dual Dimensioning
3-6 Metric Drawing
3-7 Geometric Dimensioning and Tolerancing (*application of*)
3-8 Test Your Knowledge Questions
Color Transparencies (Binder/CD only)

GUIDE FOR LESSON PLANNING

This chapter introduces and explains the basics of drawings used in industry. Since it would not be possible to manufacture complex products without them, the machinist must know how to obtain and understand all of the information provided on drawings.

With the increasing use of computer-generated machining programs, drawings may not always be available to the machinist at the work station. Often times, the machinist only sees a "drawing" on the computer monitor and the computer program makes corrections and adjustments or alerts the machinist to possible problems. However, the machinist may have to refer to the drawings to determine what adjustments and changes are acceptable. For this reason, it is of vital importance that a machinist be able to read and understand drawings.

Have students read and study Chapter 3. Review the assignment and discuss the following:
- Importance of drawings to ensure that parts, no matter where they are made, will be interchangeable and fit properly in new assemblies and in similar assemblies made at an earlier date.
- Reason for standardized symbols, lines, and figures.
- The importance of the American National Standards Institute (ANSI).
- The Alphabet of Lines.

- Symbols revised by ANSI and the symbols they replace.
- Information found on drawings and how it is used.
- Types of drawings used in shops.
- Methods used to reproduce drawings.
- Drawing sizes.
- Geometric Tolerancing and Dimensioning and why it is used.

Emphasize that a machinist:

- *Always* works to the dimensions, tolerances, and surface finishes specified on a drawing.
- *Never* scales a dimension from a drawing.

Technical Terms

Review the terms introduced in the chapter. New terms can be assigned as a quiz, homework, or extra credit. These terms are also listed at the beginning of the chapter.

actual size
American National Standards Institute (ANSI)
bill of materials
dual dimensioning
geometric dimensioning and tolerancing
revisions
scale drawings
SI Metric
US Conventional
working drawings

Review Questions

Assign *Test Your Knowledge* questions. Copy and distribute Reproducible Master 3-8 or have students use the questions on pages 52–53 in the text and write their answers on a separate sheet of paper.

Workbook Assignment

Assign Chapter 3 of the ***Machining Fundamentals Workbook.***

Research and Development

Discuss the following topics in class or allow students to choose topics for individual or group projects.

1. Make a tracing and reproduce it using the diazo and electrostatic processes.
2. Prepare a display of the microfilming technique of print reproduction. Include prints, samples of film cards and photographs, or magazine advertisements illustrating the equipment used to make them.
3. Secure sample prints from industry.
4. Secure prints produced by the CAD (Computer-Aided Design) technique.
5. Prepare a display panel that shows a simple project from print to finished product.
6. Prepare transparencies for the overhead projector that show the title block, parts list, and material list from an actual industrial drawing. Use them to explain or describe an industrial drawing to the class. If possible, borrow a sample of the part shown on the drawing.
7. Contact a local industry and borrow prints of a simple assembly. If possible, also secure a sample of the object shown on the print. Develop a display.

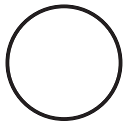
TEST YOUR KNOWLEDGE ANSWERS, Pages 52–53

1. d. All of the above.
2. language of industry
3. one-millionth
4. one-millionth
5. Use a surface roughness comparison standard.
   Profilometer or electronic roughness gage.
6. bilateral
7. unilateral
8. b. Allowances in either oversize or undersize that a part can be made and still be acceptable.
9. scale drawings
10. a. Showing only a small portion of the complete object.
11. They sometimes get lost, damaged, or destroyed.
12. detail
13. It shows where and how the parts described on a detail drawing fit into the completed assembly.
14. Convenience in filing and locating drawings.
15. basic
16. reference
17. A feature control frame is used when a location or form tolerance is related to a datum.

18.

Straightness


Flatness


Circularity (roundness)


Cylindricity

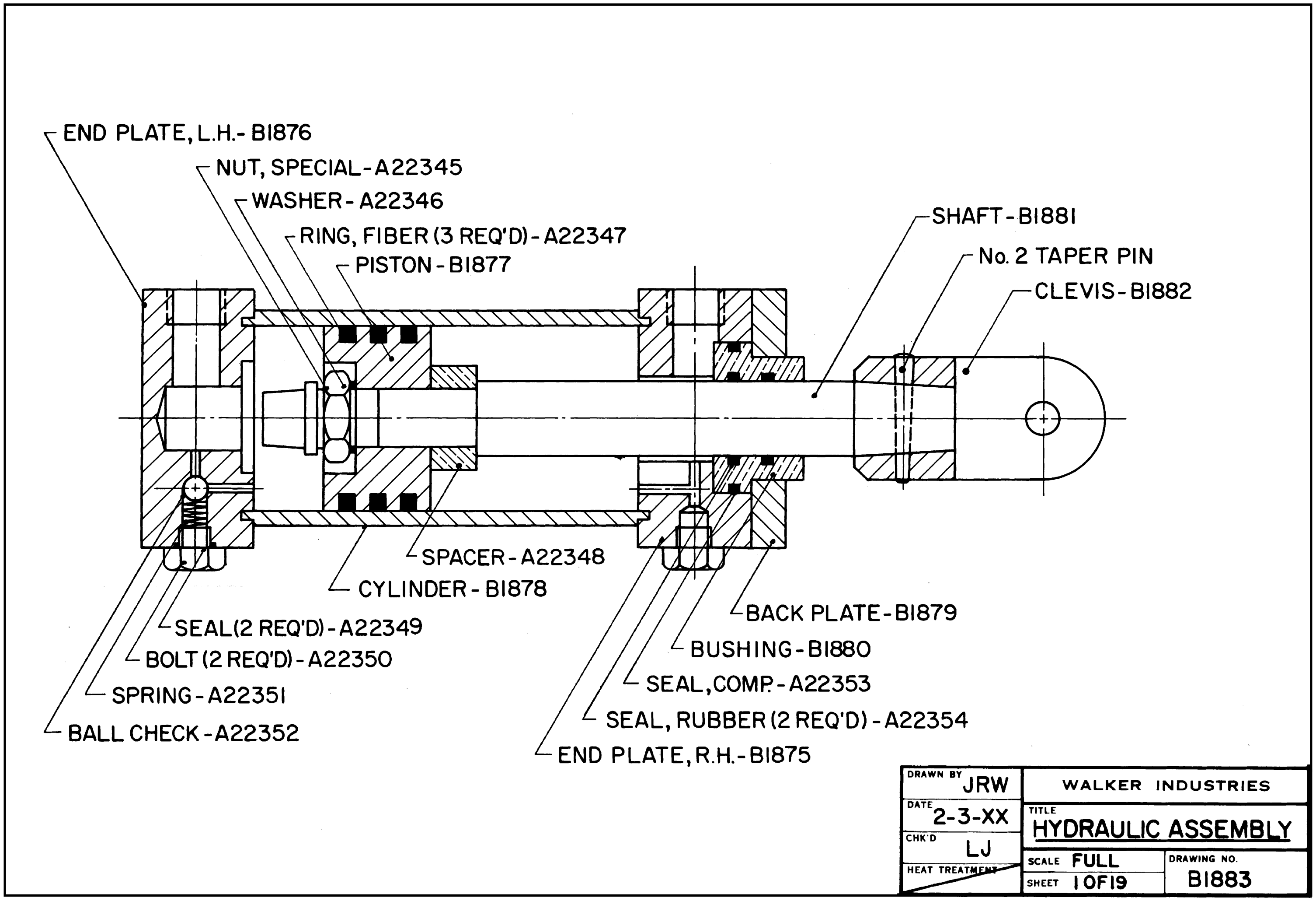
19. Maximum material condition (MMC) is the condition in which the size of a feature contains the maximum amount of material within the stated limits of size. Also refer to Figure 3-31.
20. Least material condition (LMC) is the condition in which the size of a feature contains the least amount of material within the stated tolerance limits. Also refer to Figure 3-32.

WORKBOOK ANSWERS, Pages 15–22

1. d. All of the above.
2. fractional
3. English, metric
4. Student answers will vary but may include any four of the following: material(s) to be used; surface finish required; tolerances; quantity of units per assembly; scale of drawing; next assembly or subassembly; revisions; the name of the object.
5. Tolerances are allowances, either undersize or oversize, permitted when machining or making an object.
6. roughness comparison
7. profilometer
8. d. Dimensions should *never* be scaled off a drawing.
9. b. only a small portion
10. drawings might be lost, damaged, or destroyed; same print may be needed in different places at same time
11. d. All of the above.
12. Evaluate individually.
13. When the amount of variation (tolerances) in form (shape and size) and position (location) needs to be more strictly defined, it provides the precision needed to allow for the most economical manufacture of parts.
14. Geometric dimensioning and tolerancing is a system that provides additional precision compared to conventional dimensioning. It ensures that parts can be easily interchanged.
15. They are employed to provide clarity and precision in communicating design specifications.
16. d. All of the above.
17. b. basic dimension
18. a. reference dimension
19. measured size of a part after it is manufactured
20. feature control frame
21. Maximum material (MMC)
22. Least material (LMC)
23. A. Material to be used
    B. Tolerances
    C. Quantity
    D. Scale
    E. Next assembly
    F. Revisions
    G. Name of object
    H. Drawing number
24. A. 3.000″
    B. 2.000″
    C. 1.625″
    D. 0.7503″
    E. +0.0003″
    F. 0.266″
    G. 0.265″
    H. 0.391″
    I. Remove burrs. Break sharp edges .010″ Max. Finish 125 √ all over except as noted.
25. A. Clamp, Alignment
    B. Full size
    C. B123456
    D. D45678
    E. Dual dimensioning
    F. Aluminum 6061-T4
    G. 12
    H. Distance from centerline to flat on top of part was changed from 1.50″ (38.0 mm) to 1.62″ (41.14 mm).

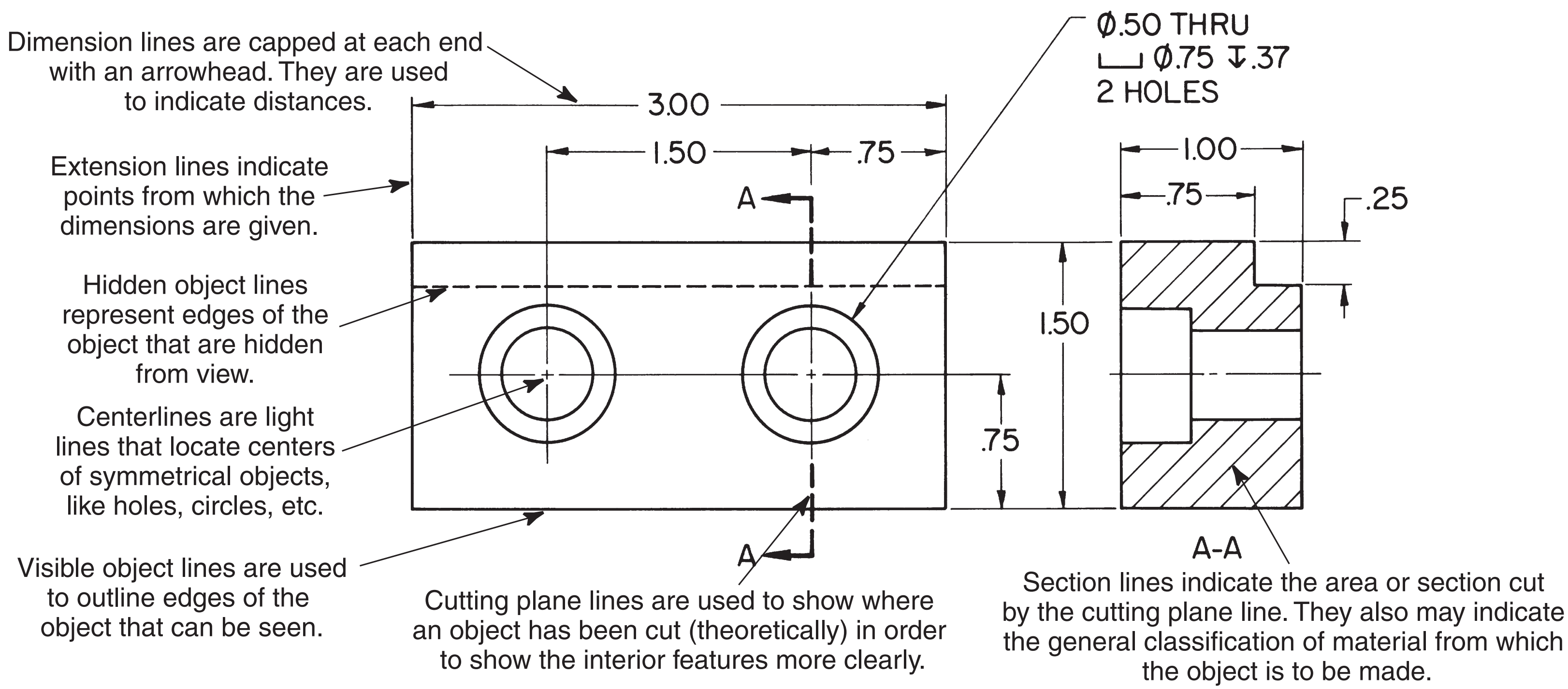
26. L. 4.25″ (107.85 mm)
    W. 2.62″ (66.54 mm)
    T. 0.75″ (19.0 mm)
27. 1. 125√ [3.2√] all over.
    2. Break all sharp edges 0.01 [0.5] MAX.
    3. Dimensions in [ ] are millimeters.
28. A. 1.50″ (38.1 mm)
    B. 0.500″
29. No standard metric tool available this size.
30. A. 3.25″ (82.45 mm)
    B. 1/4-20UNC-2
    C. There is no metric thread this size.
    D. 0.37″ (9.5 mm)
    E. 0.26″ (6.7 mm)
    F. 0.13″ (3.5 mm)
    G. 0.75″ (19.0 mm)
    H. +0.001″
    I. 0.75″ (19.0 mm)
    J. 1.00″ (25.4 mm)
    K. 1.62″ (41.14 mm)

Typical Assembly Drawing

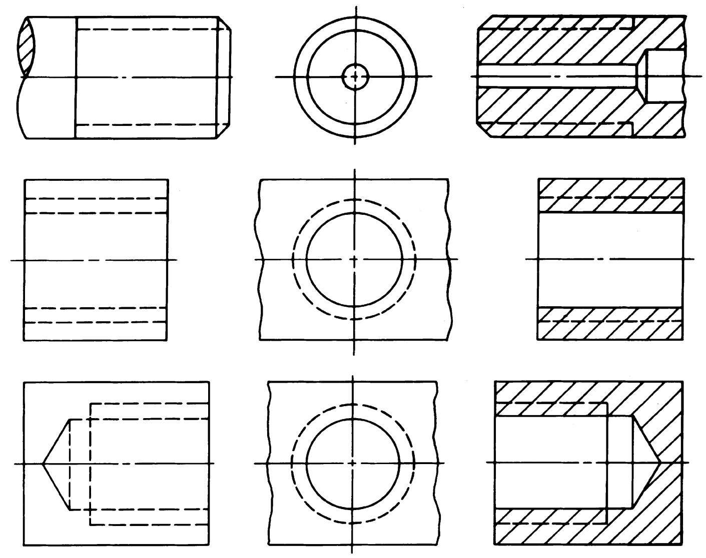


3-1

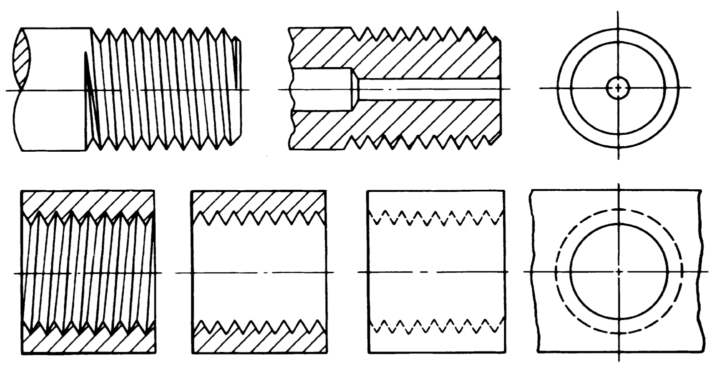
Alphabet of Lines



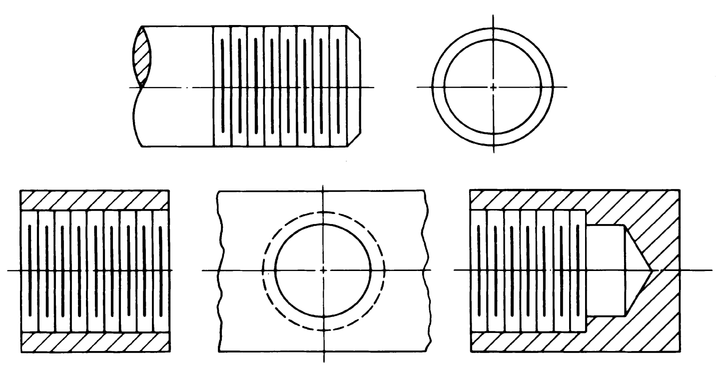
Threads



Simplified Representation

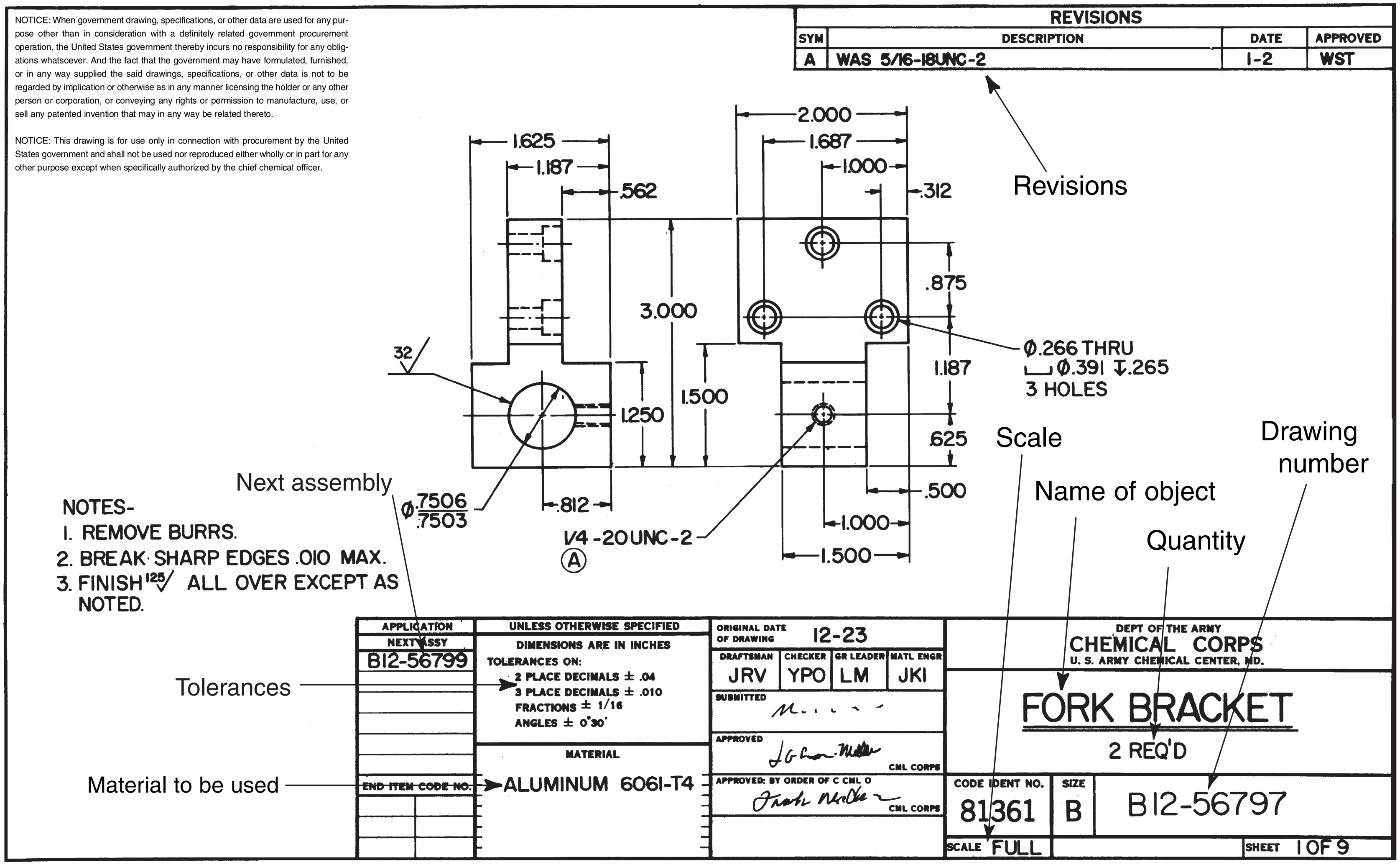


Detailed Representation



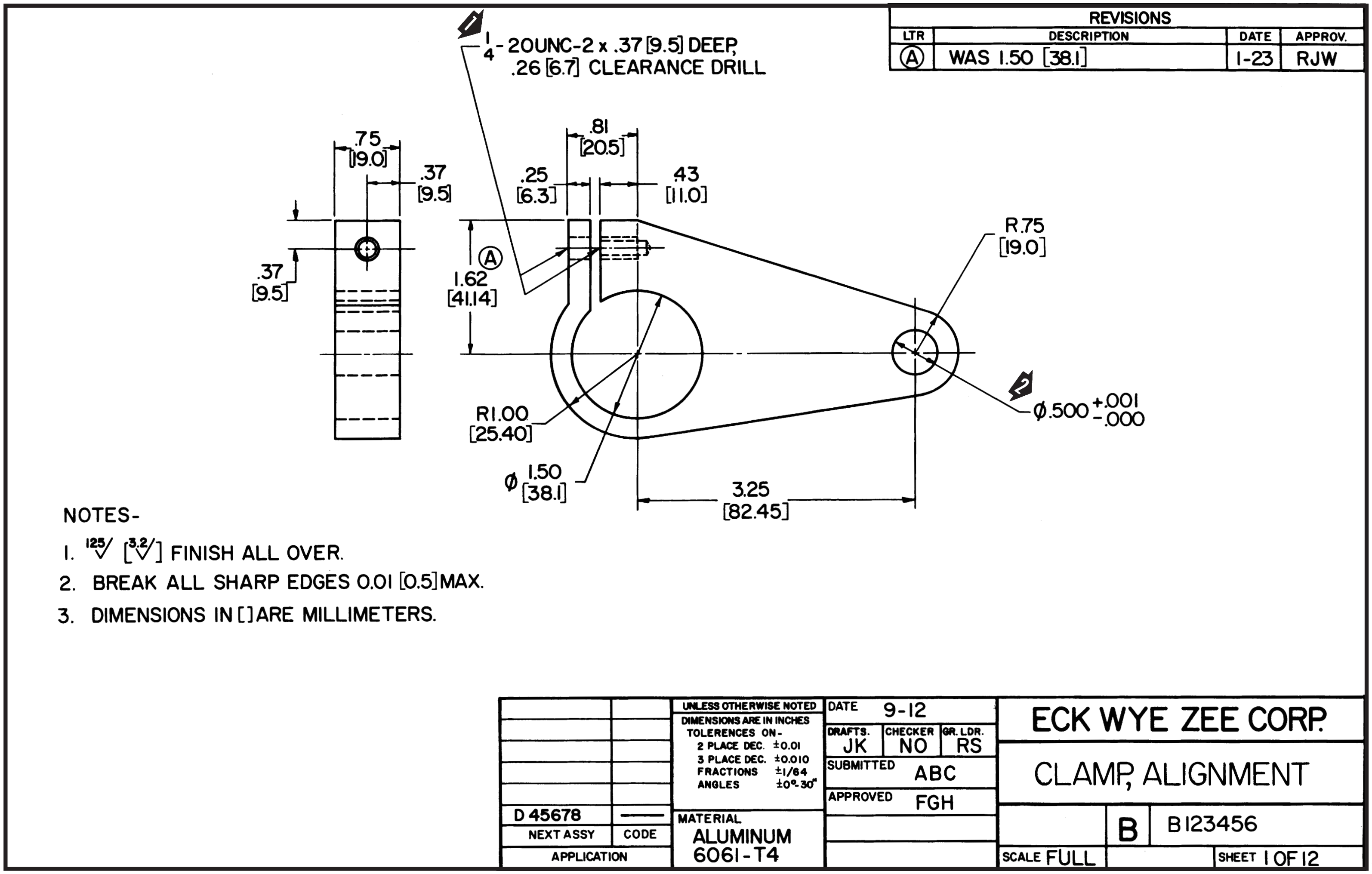
Schematic Representation

Information on a Typical Drawing



3-4

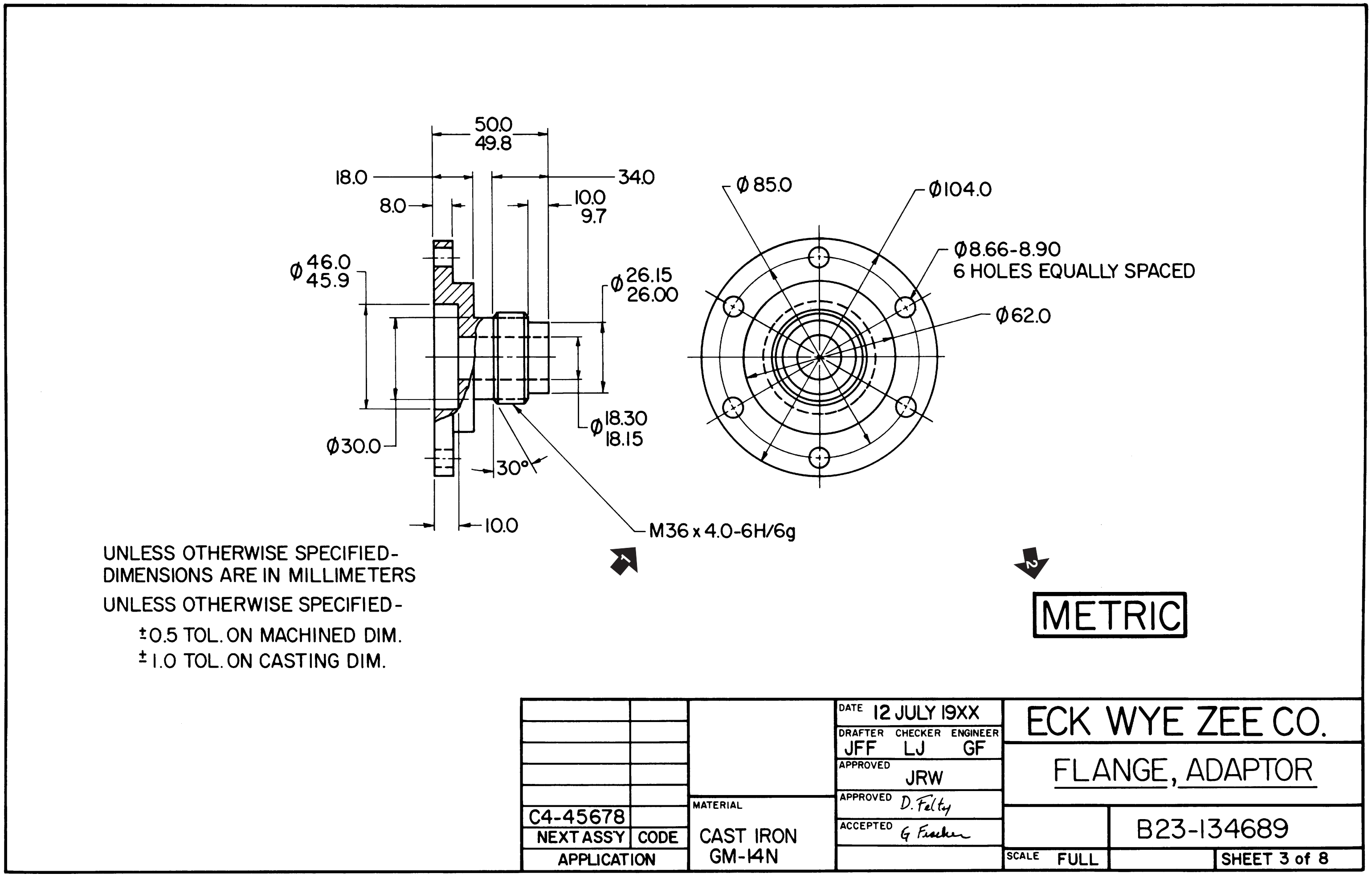
Dual Dimensioning



1. A metric thread size has not been given because there is none equal to this size fractional thread.

2. There is no metric reamer equal to this size.

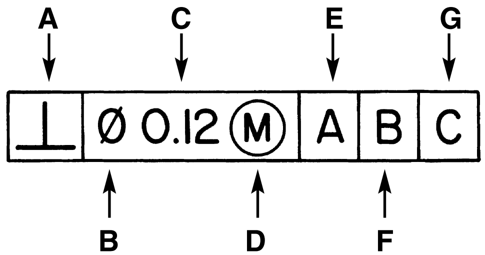
Metric Drawing



1. Note that metric thread specifications are different from the more familiar UNC (*coarse*) and the UNF (*fine*) series threads. The letter *M* denotes standard metric screw threads. The 36 indicates the nominal thread diameter in millimeters. The 4.0 denotes thread pitch in millimeters. The 6H and 6g ate tolerance calss designations.

2. To avoid possible misunderstanding, *metric* is shown on the drawing in large letters.

Geometric Dimensioning and Tolerancing



A Geometric characteristic symbol

B Diameter symbol (when used) Zone descriptor

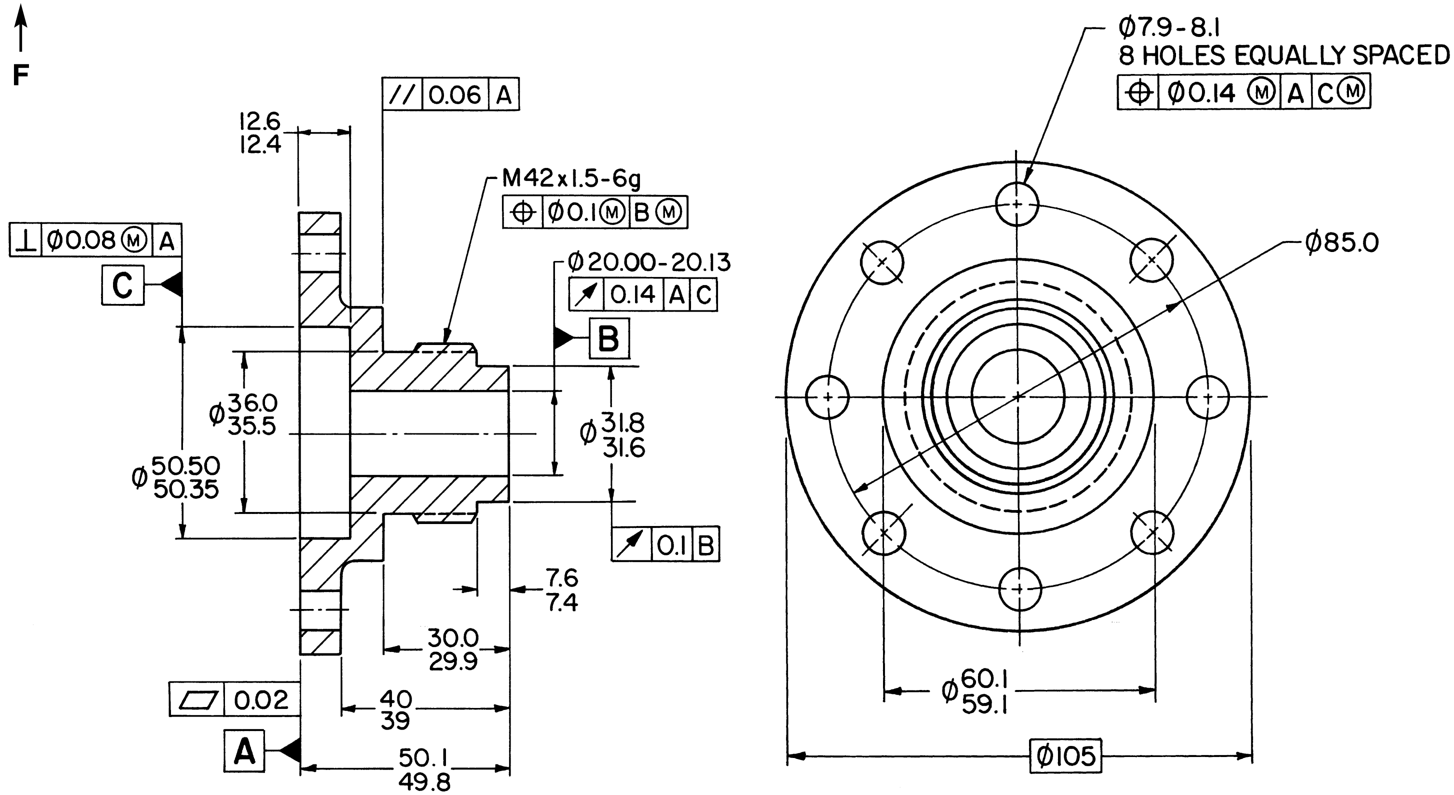
C Geometric tolerance

D Material condition symbol

E Primary datum reference

F Secondary datum reference

G Tertiary datum reference



DIMENSIONS ARE IN MILLIMETERS

3-7

Understanding Drawings

Name: ____Date: ______________ Score: ________

1. Drawings are used to: 1. ______________________________
   a. Show, in multiview, what an object looks like before it is made.
   b. Standardize parts.
   c. Show what to make and the sizes to make it.
   d. All of the above.
   e. None of the above.
2. The symbols, lines, and figures that make up a drawing are frequently called the _____. 2. ______________________________
3. A microinch is _____ of an inch. 3. ______________________________
4. A micrometer is _____ of a meter. 4. ______________________________
5. How can surface roughness of a machined part be checked against specifications on the drawing?

   ____

   How can it be measured electronically? ____

   ____

   ____
6. When tolerances are plus and minus, it is called a _____ tolerance. 6. ______________________________
7. When tolerances are only plus or only minus, it is called a _____ tolerance. 7. ______________________________
8. Tolerances are: 8. ______________________________
   a. The different materials that can be used.
   b. Allowances in either oversize or undersize that a part can be made and still be acceptable.
   c. Dimensions.
   d. All of the above.
   e. None of the above.
9. Drawings made other than actual size are called _____. 9. ______________________________
10. A subassembly drawing differs from an assembly drawing by: 10. ______________________________
    a. Showing only a small portion of the complete object.
    b. Making it possible to use smaller drawings.
    c. Showing the object without all needed dimensions.
    d. All of the above.
    e. None of the above.

Name ______________________________

11. Why are prints used in place of the original drawings? ______________________________
____
____

12. The craft worker is given all of the information needed to make a part on a _____ drawing. 12. ______________________________

13. What does an assembly drawing show? ______________________________
____
____

14. Why are standard size drawing sheets used? ______________________________
____
____

15. All dimensions have a tolerance except _____ dimensions. 15. ______________________________

16. Dimensions placed between parentheses are _____ dimensions. 16. ______________________________

17. When is a feature control frame employed? ______________________________
____
____

18. Sketch the form geometric tolerance symbols and indicate what they mean.

Name ______________________________

19. Define the term *maximum material condition* (MMC). Use a sketch if necessary. ______________

____

____

____

20 Define the term *least material condition* (LMC). Use a sketch if necessary. ______________

____

____

____