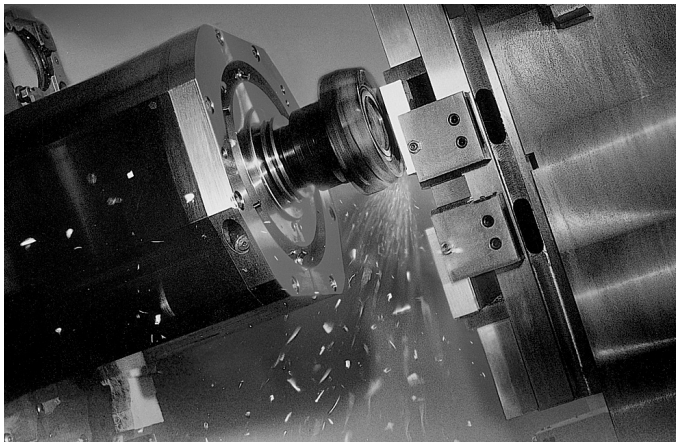


Chapter 21

Computer Numerical Control

LEARNING OBJECTIVES

After studying this chapter, students will be able to:

- Define the term "numerical control."
- Describe the difference between the incremental and absolute positioning methods.
- Explain the operation of NC (numerical control), CNC (computer numerical control), and DNC (direct or distributed numerical control) systems.
- Point out how manual and computer-aided programming is done.

INSTRUCTIONAL MATERIALS

Text: pages 399–422
Test Your Knowledge Questions, page 421
Workbook: pages 117–122
Instructor's Resource: pages 279–292
Guide for Lesson Planning
Research and Development Ideas
Reproducible Masters:
21-1 Direct Numerical Control (DNC)
21-2 Distributed Numerical Control (DNC)
21-3 The Cartesian Coordinate System
21-4 Axes of Machine Movements
21-5 NC Positioning Methods
21-6 Contour or Continuous Path Machining
21-7 Mirror Image Machining
21-8 Test Your Knowledge Questions
Color Transparency (Binder/CD only)

GUIDE FOR LESSON PLANNING

Have the class read and study the chapter. Review the assignment using the reproducible masters as overhead transparencies and/or handouts. Discuss the following:

- The meaning of *Computer-Aided Machining Technology.*
- Numerical Control (NC) and Computer Numerical Control (CNC).
- The difference between *Direct* and *Distributed Numerical Control.* Use Reproducible Masters 21-1 and 21-2.
- The Cartesian Coordinate System. Use Reproducible Master 21-3.
- NC tool positioning methods (*absolute* and *incremental*). Use Reproducible Masters 21-4 and 21-5.
- NC movement systems, including *point-to-point, straight-cut,* and *contour or continuous path.* Use Reproducible Master 21-6.
- Mirror imaging. Use Reproducible Master 21-7.
- Programming NC machines, both manual and computer-aided.
- Computer languages.
- Adaptive control.
- Advantages and disadvantages of NC.

- Other NC applications.
- Setting up and programming the NC machine in the shop/lab.
- Demonstrating the NC machine in the shop/lab.

A brief review of the demonstrations will provide students/trainees the opportunity to ask questions.

Technical Terms

Review the terms introduced in the chapter. New terms can be assigned as a quiz, homework, or extra credit. The following list is also given at the beginning of the chapter.

absolute positioning
Cartesian Coordinate System
circular interpolation
closed loop system
continuous path system
incremental positioning
machine control unit (MCU)
open loop system
point-to-point system
straight-cut system

Review Questions

Assign *Test Your Knowledge* questions. Copy and distribute Reproducible Master 21-8 or have students use the questions on page 421 and write their answers on a separate sheet of paper.

Workbook Assignment

Assign Chapter 21 of the ***Machining Fundamentals Workbook.***

Research and Development

Discuss the following topics in class or have students complete projects on their own.

1. Design and construct a simple machine that will illustrate how *numerical control* works.
2. Review up-to-date technical magazines that have articles on subjects such as NC, CNC, and robotics. Prepare a brief outline of at least one article for class discussion.
3. Visit a plant that uses automated equipment. If such a visit is not possible, show a video or film that illustrates automation.
4. If your shop/lab is fortunate enough to have an NC or CNC machine tool, ask your instructor to assign a programming problem. Prepare the program, edit and proof it, and follow through to the finished machined part.

TEST YOUR KNOWLEDGE ANSWERS, Page 421

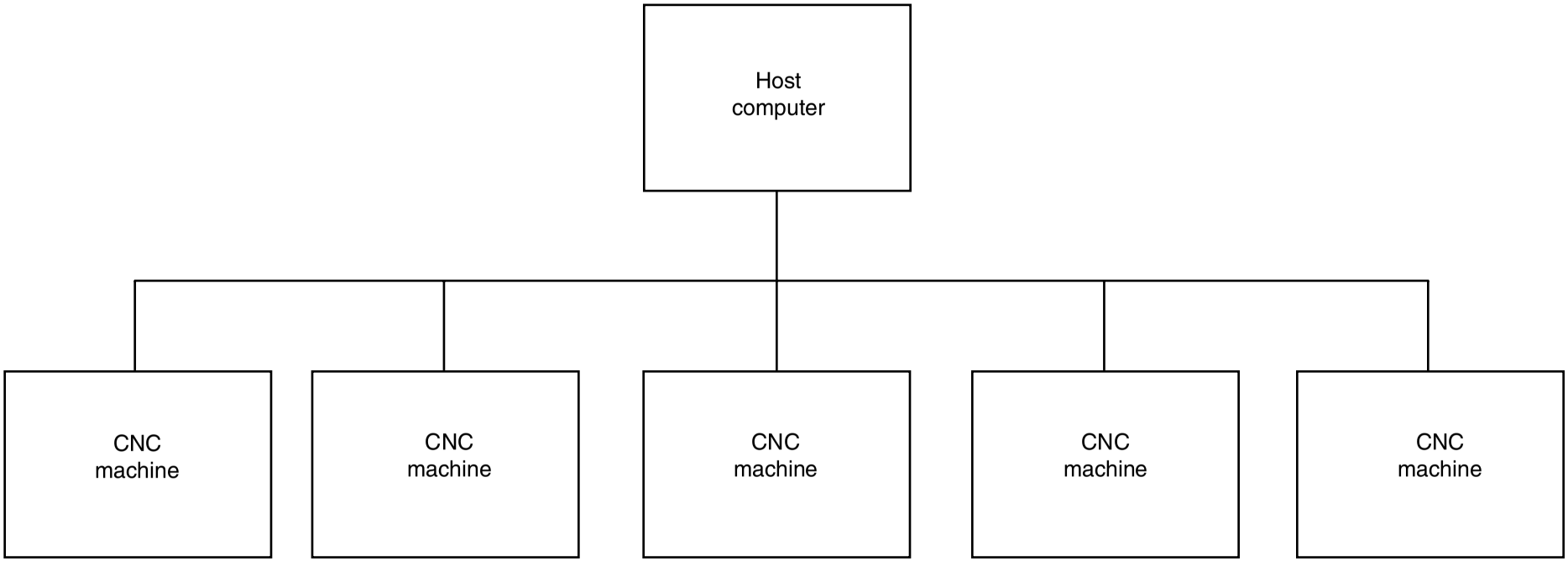
1. Evaluate individually. Refer to Section 21.1.
2. Evaluate individually. Refer to Section 21.2.1 and Figure 21-6.
3. Evaluate individually. Refer to Section 21.2.2 and Figures 21-10 and 21-11.
4. A sequence of instructions that tells the machine what operations to perform, and where on the material they are to be done.
5. Point-to-point, straight-cut, and contour or continuous path.
6. Evaluate individually. Refer to Figures 21-13, 21-14, and 21-15.
7. Contour or continuous path system. Geometrical complexity makes a computer mandatory when preparing programs to machine two- and three-dimensional shapes.
8. a. Machine Control Unit.
   b. A series of letters, numbers, punctuation marks, and special characters used to instruct the machine what operations to perform and where to perform them.
   c. A script containing lines of information blocks.
9. By producing a prototype made of plastic or wax. Refer to Figure 21-41.
10. Evaluate individually. Refer to Section 21.6.

WORKBOOK ANSWERS, Pages 117–122

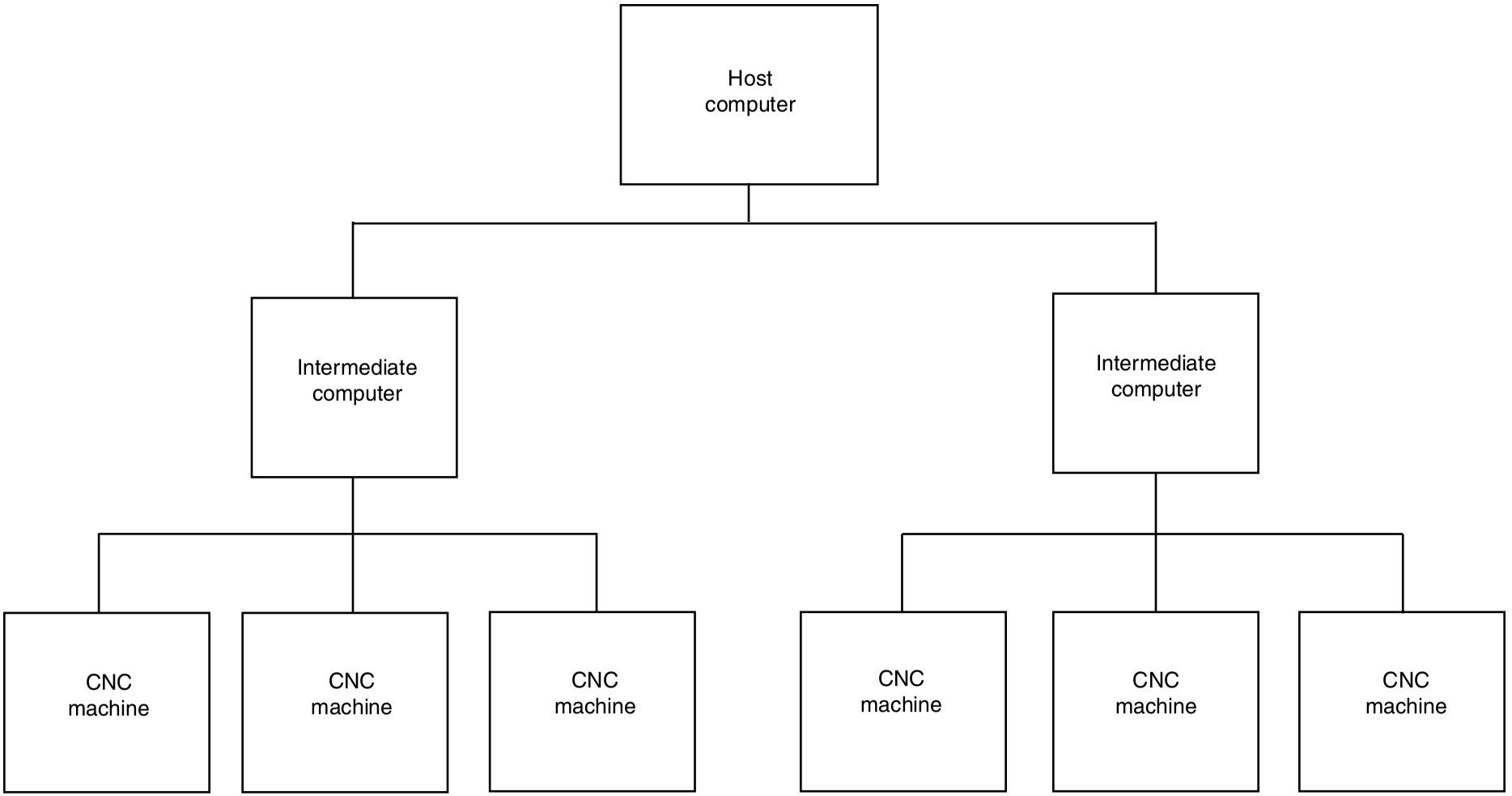
1. Manual machining is done by the machinist moving one or more of the machine's lead and feed screws and guiding it through the various machining operations.
2. a. is the operation of the machine tool by a series of coded instructions
3. d. All of the above.
4. c. Both a and b.
5. d. All of the above.
6. The closed loop system uses an electronic feed-back device, called a transducer, to continually monitor tool position. The open loop system has no feedback for monitoring for comparing purposes. The system relies on the integrity of the control unit for accuracy.

7. The basis for NC programming. It provides a way to define movement.
8. c. Z axis
9. CNC (computer numerical control)
10. c. a fixed point of origin, or zero point
11. b. the prior tool position
12. absolute
13. MCU (Machine Control Unit)
14. A. Tool movement from one point to the next does not have to follow a specific path.
    B. Permits controlled tool travel along one axis at a time.
    C. Controls machine and tool movement as the cutter moves along the programmed path. Cutting is continuous and can be in six axes simultaneously.
15. c. Six
16. d. All of the above.
17. c. Both a and b.
18. point-to-point; the cutting tool must be fed a constantly changing series of instructions
19. d. All of the above.
20. end of block (EOB)
21. For straight line cutting, drilling, and spot welding.
22. each machining sequence and machine function into a coded block of information the MCU can understand
23. Computer-aided programming reduces and simplifies the numerical calculations that the programmer must perform when programming the machining of more complex parts.
24. languages
25. The rules used for combining the vocabulary of words, numbers, and other symbols used when writing programs.
26. geometry, machining
27. Any of the following: increased productivity, reduced machining costs, extended cutter life, reduced scrap, improved work quality, greater machine utilization.
28. Coordinates
29. Evaluate individually. Refer to Section 21.8.
30. Any two of the following: high initial cost of equipment; shortage of skilled technicians to service equipment; increased maintenance costs over traditional machine tools; machine capabilities must be fully utilized.

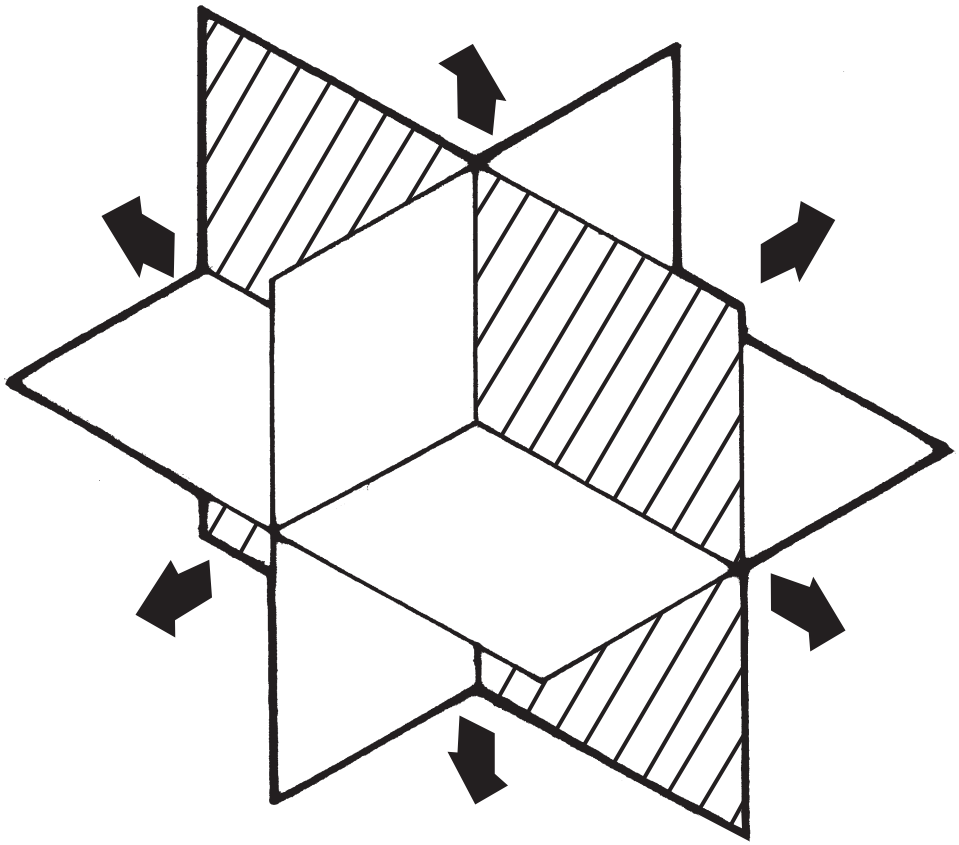
Direct Numerical Control (DNC)



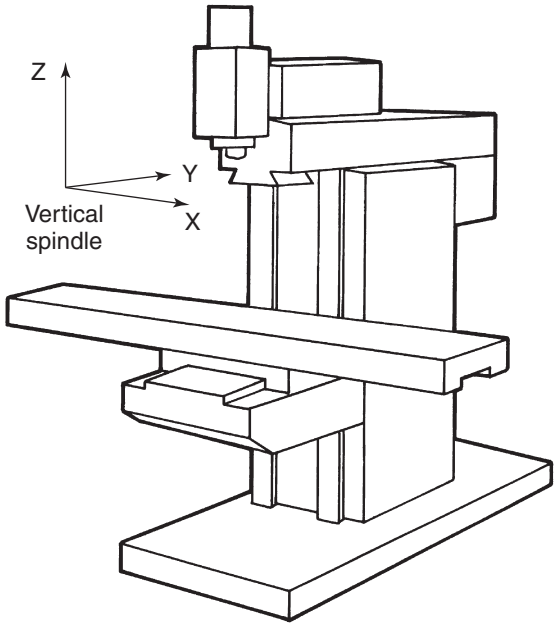
Distributed Numerical Control (DNC)



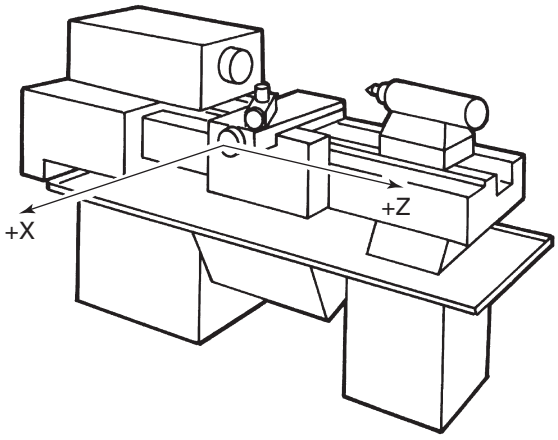
The Cartesian Coordinate System



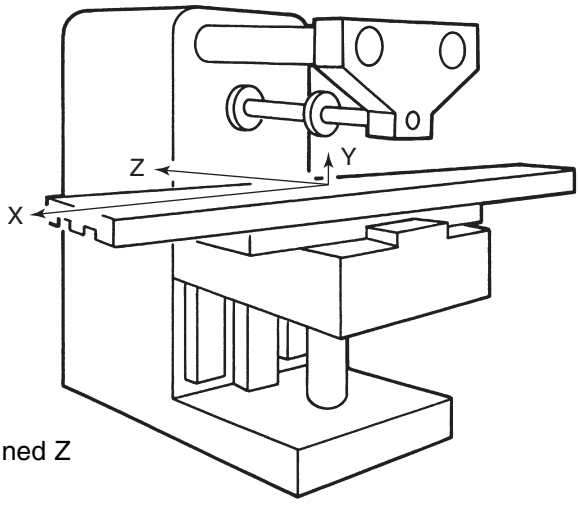
Axes of Machine Movements



Vertical Milling Machine



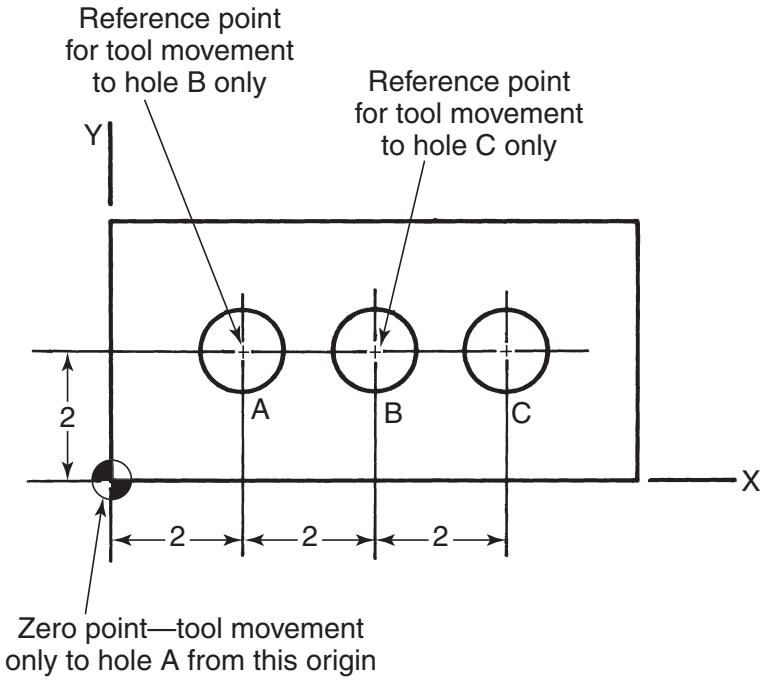
Lathe



Horizontal Milling Machine

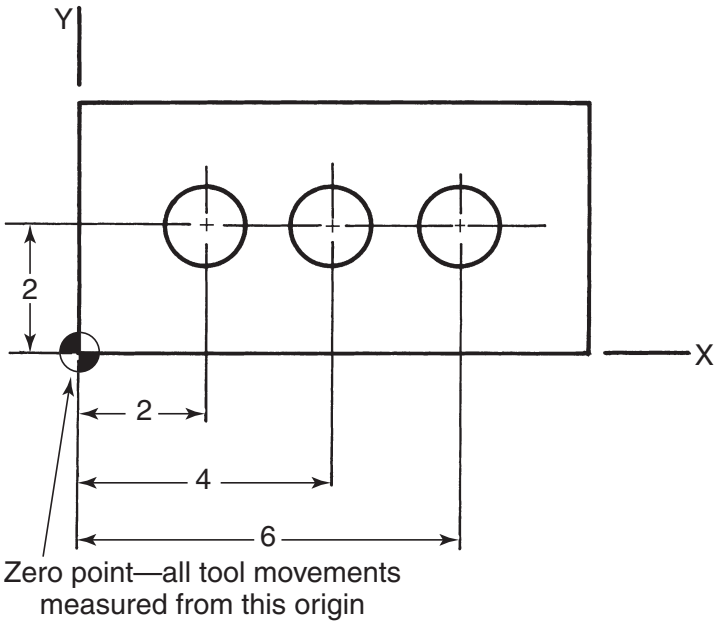
Note: Spindle motion is assigned Z axis

NC Positioning Methods



Incremental Positioning System

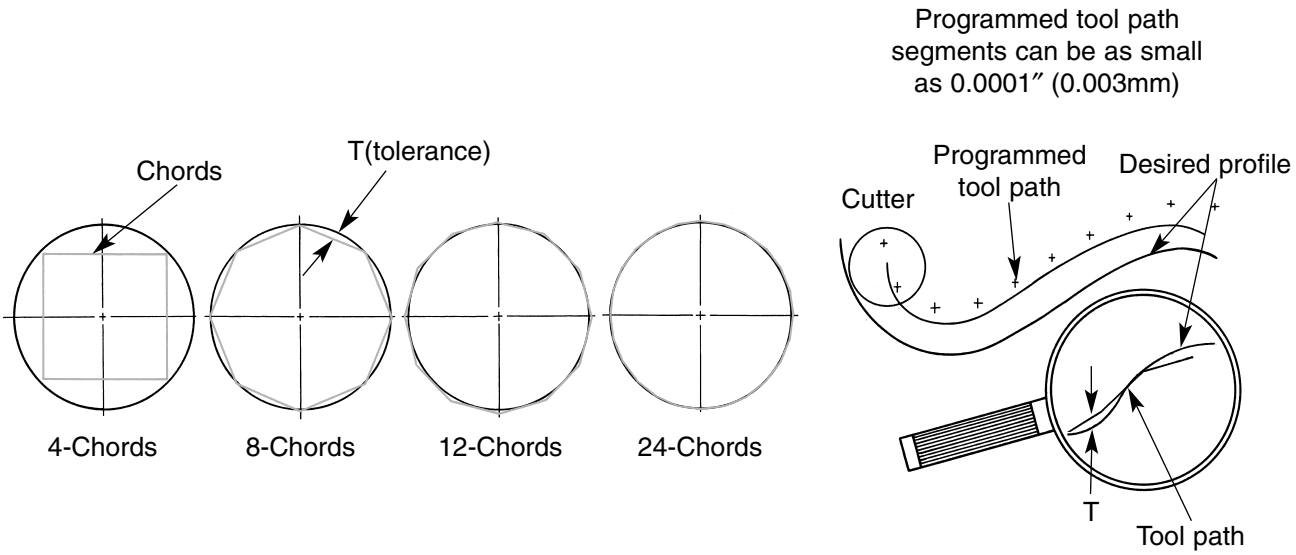
Each set of coordinates has its point of origin from last point established.



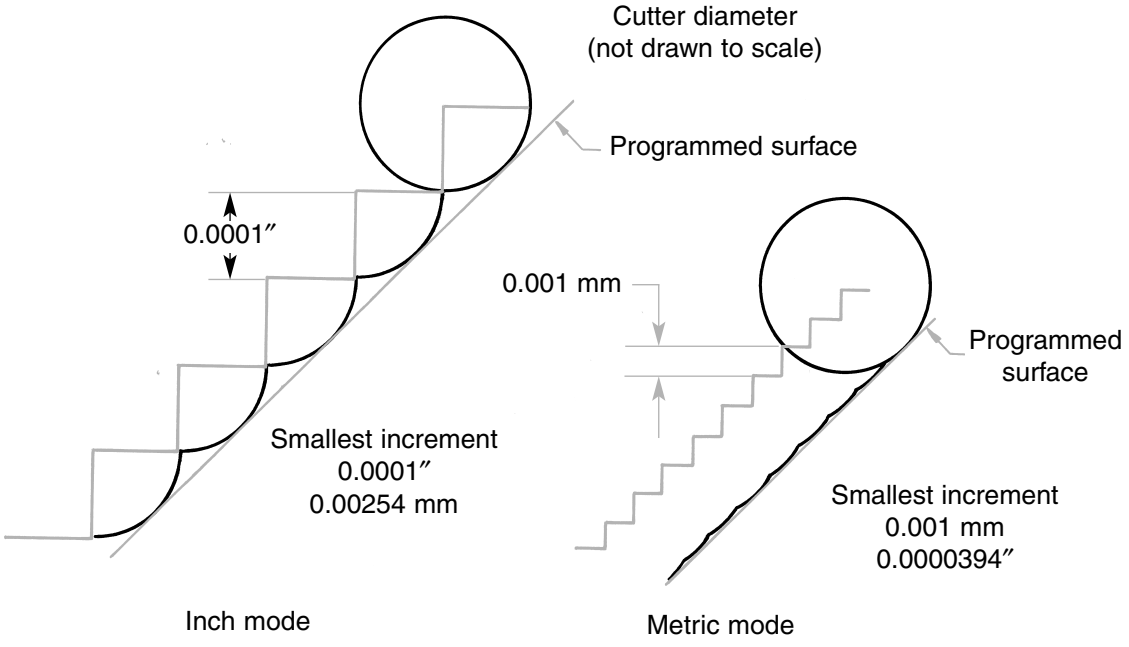
Absolute Positioning System

In this system, all coordinates are measured from fixed point (zero point) of origin.

Contour or Continuous Machining

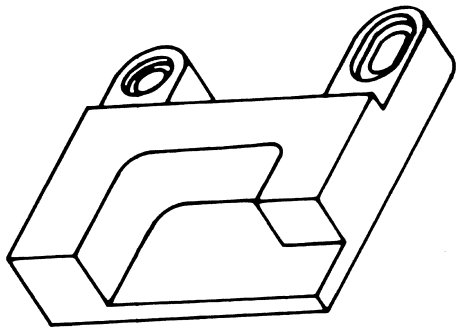


Contours obtained from contour or continuous path machining are result of a series of straight-line movements. The degree to which a contour corresponds with specified curve depends upon how many movements or chords are used. Note how, as number of chords increase, the closer the contour is to a perfect circle. The actual number of lines or points needed is determined by the tolerance allowed between design of the curved surface and one actually machined.

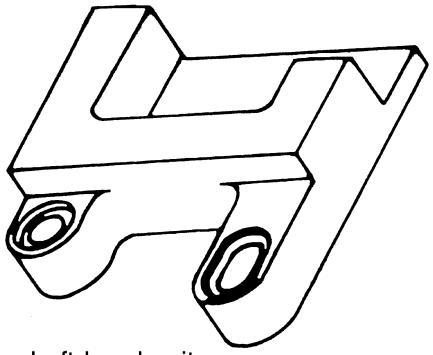


This exaggerated illustration shows why metric machine movement increments are often preferred when contour machining. The benefit has to do with the least input increment allowed in the metric mode. In the inch mode, the least input increment is 0.0001″, which means you can input program coordinates and tool offsets down to 0.0001″. In the metric mode, the least input increment is 0.001 mm, which is less than one-half the least input increment when using the inch mode. The coordinates going into the program will then be much closer to what is desired for accurately machined parts.

Mirror Image Machining

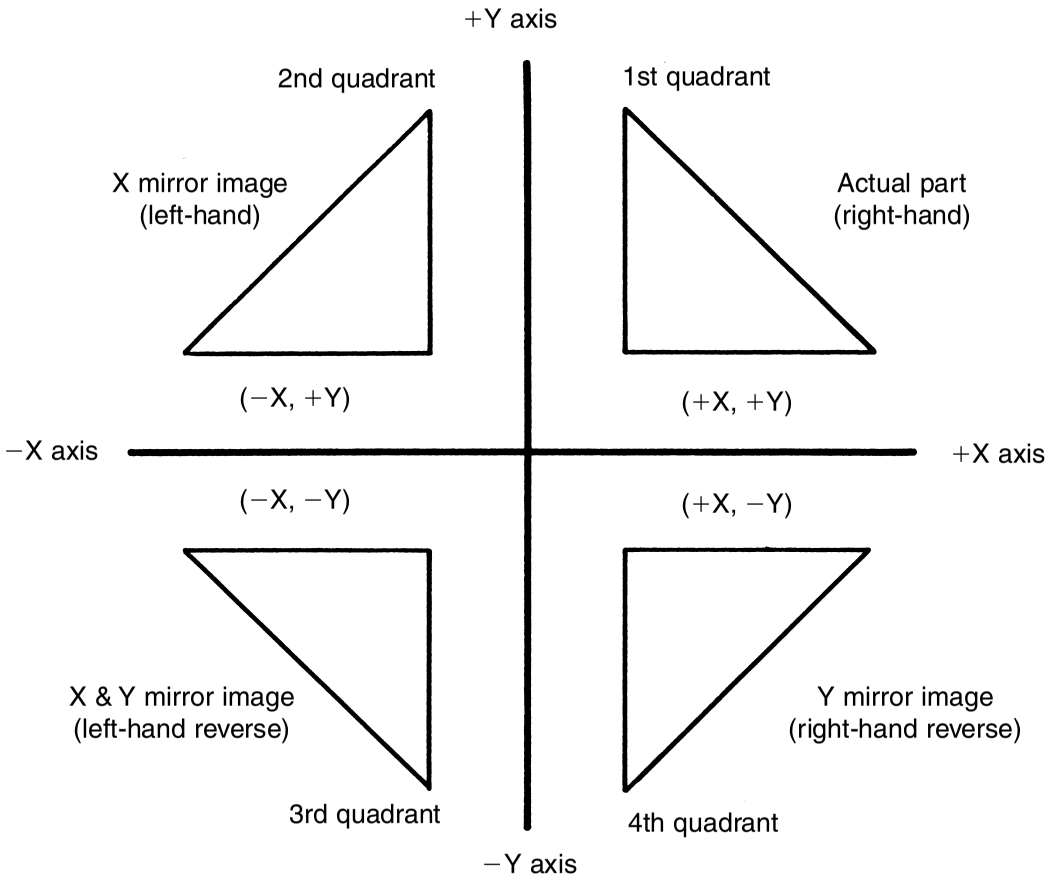


Right-hand unit



Left-hand unit

On many NC machine tools, mirror image parts can be machined using the same program.



Mirror image to produce a reverse duplicate part. The technique is known as axis inversion.

Computer Numerical Control

Name: _____Date: _____ Score: _____

1. Describe the differences between manual machining techniques and NC methods. _____

_____

_____

_____

_____

_____

2. Prepare a sketch showing the Cartesian Coordinate System.

3. Prepare two similar sketches. Show incremental dimensioning on the first sketch and absolute dimensioning on the second sketch.

4. What is an NC program? _____

_____

_____

5. List the three basic NC systems. _____

_____

_____

Name _____

6. Draw sketches showing how the three NC systems differ.

7. Which of the three NC methods require the use of a computer? Why is a computer required?

_____

_____

8. What do the following terms mean?

a. MCU: _____

_____

b. Alphanumeric data: _____

_____

c. Program sheet: _____

_____

9. How is an NC program often verified? _____

_____

_____

10. Briefly explain adaptive control (AC). _____

_____

_____

_____