DOCUMENT RESUME

ED 347 338 CE 061 573

AUTHOR	Leonard, Dennis
TITLE	Computer Programmed Milling Machine Operations. High-Technology Training Module.
INSTITUTION	Wausau District Public Schools, Wis.
SPONS AGENCY	Office of Vocational and Adult Education (ED), Washington, DC.
PUB DATE	6 Jul 90
CONTRACT	V199A90151
NOTE	51p.; Developed as part of the High-Technology Training Model for Rural Based Business and Industry, Technical Colleges and Local and State Educational Agencies. Portions of activity sheets contain marginally legible print.
PUB TYPE	Guides - Classroom Use - Teaching Guides (For Teacher) (052)
EDRS PRICE	MF01/PC03 Plus Postage.
DESCRIPTORS	Behavioral Objectives; Classroom Techniques; *Computer Assisted Manufacturing; Computer Oriented Programs; *Computer Science; Computer Science Education; Course Content; Electronic Control; High Schools; Learning Modules; Machinery Industry; Machine Tools; *Microcomputers; *Numerical Control; Pretests Posttests; Student Evaluation; Technical Education; Test Items

ABSTRACT

This learning module for a high school metals and manufacturing course is designed to introduce the concept of computer-assisted machining (CAM). Through it, students learn how to set up and put data into the controller to machine a part. They also become familiar with computer-aided manufacturing and learn the advantages of computer numerical control (CNC) over conventional machining. The six sections of the module are the following: module objectives, content outline, teaching methods, student activities, resource list, and evaluation (pretest, three exercises, and posttest). Student activities include information sheets and are illustrated with line drawings. (KC)

High-Technology Training Module

Module Title: COMPUTER PROGRAMMED MILLING MACHINE OPERATIONS

Unit: CNC MILLING

Course: METALS I / MANUFACTURING

Grade Level (s): 10 - 12TH GRADE

Developed by: DENNIS LEONARD

Date: JULY 6, 1990

School: WAUSAU EAST HIGH SCHOOL

708 FULTON STREET, WAUSAU, WI 54401

In Touch with High Tech

Developed as a part of the High-Technology Training Model for Rural Based Business and Industry, Technical Colleges and Local and State Educational Agencies under Grant No. V199A90151.

The purpose of this unit is to introduce the concept of CAM. The student will learn how to set up and put data into the controller to machine a part. The student will become familiar with CAM and learn the advantages of CNC machining over "conventional"machining.

OBJECTIVES

At the completion of this unit, the student will be able to:

1) identify the standard X Y & Z axis.

2) understand the advantages of CNC machining.

3) understand the differences of Incremental and Absolute values in dimensioning and machining.

4) be able to punch the commands into the controller and run a sample program.

5) write their own program of a rectangle frame with only the dimensions given.

CONTENT OUTLINE

I. WHAT IS CNC MACHINING?

II. A COMPARISON OF NC AND CNC

   A. Major Components
   B. Intelligence of Each System
   C. Differences of Each System

III. QUADRANTS AND PLUS AND MINUS SIGNS GIVEN TO AXIS

IV. IDENTIFY MACHINE AXIS

   A. Milling Machine
   B. Lathe
   C. Rule of Thumb for Axis

V. ABSOLUTE VALUES COMPARED TO RELATIVE VALUES

   A. Advantages vs. Disadvantages
   B. Incremental vs. Absolute Dimensioning

VI. SAFETY

VII. CNC PROCESS FROM IDEA, PLANNING, TO PRODUCTION

   A. Sketch Idea on Paper
   B. Software and Firmware
   C. Tool Path and Cutter Diameter
   D. Programming on the Controller

METHODOLOGY

1) Introduction
2) Pre Test
3) Instructor Presentation
4) Exercise #1
5) Exercise #2
6) Exercise #3
7) Post Test

ACTIVITIES

- Exercise #1
- Exercise #2
- Exercise #3
- Evaluation

RESOURCES

1. Computer Numerical Controlled Simplified
   M. Fitzpatrick, Glencoe Publishing

2. Programming Manual
   Dyna Electronics

Video Tapes:

1. Personal Computers in Manufacturing (Society of Manufacturing Engineers)

2. CAD/CAM (Society of Manufacturing Engineers)

Transparencies

Demonstrations

1. CNC Milling Machine

Worksheets

Hardware and Software

Jig and Fixtures

Evaluation

UNIT EVALUATION

CNC PRE-TEST	40 POINTS
EXERCISE #1	15 POINTS
EXERCISE #2	20 POINTS
EXERCISE #3	25 POINTS
POST-TEST	40 POINTS

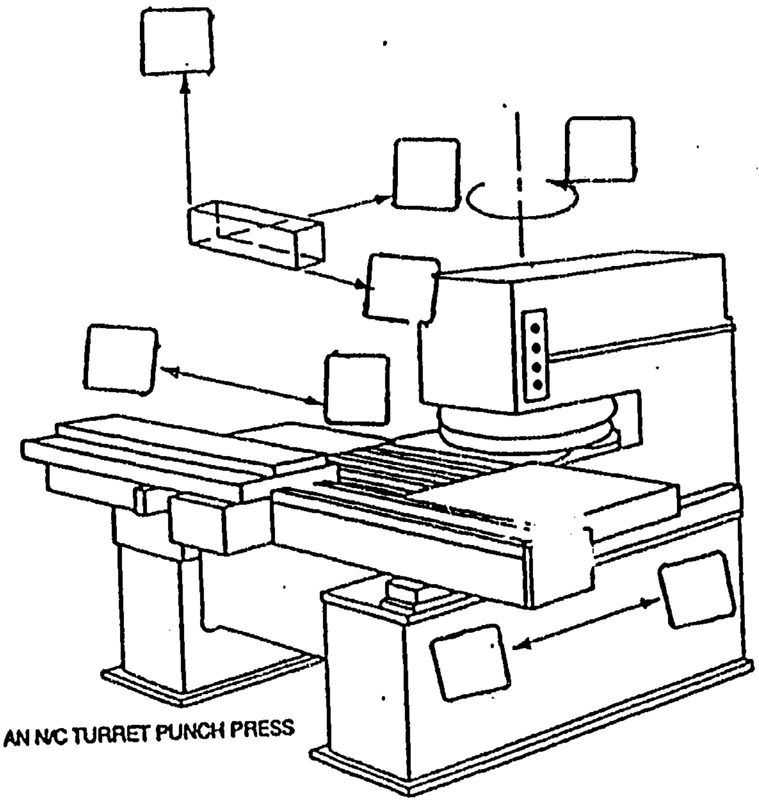
PRE-TEST AXIS DIRECTION

In each box place the axis and a (+) or(-)

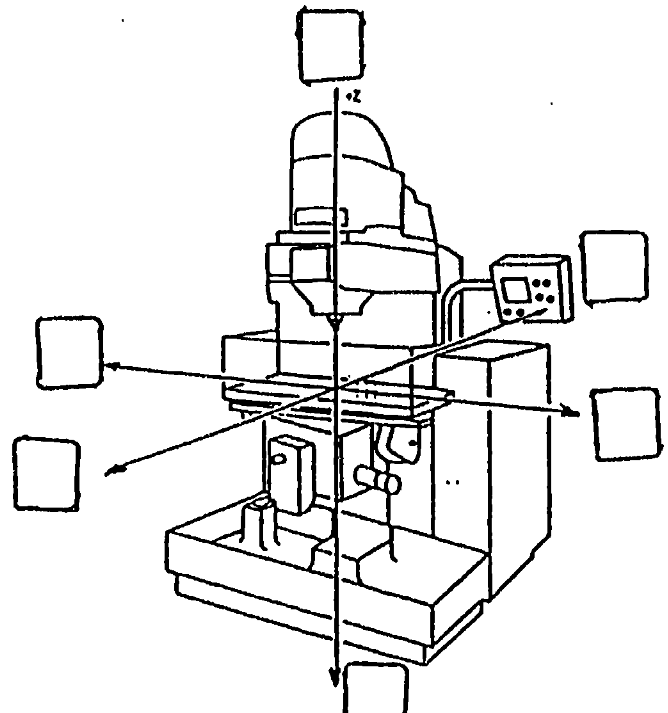
NAME____________________

DATE____________________

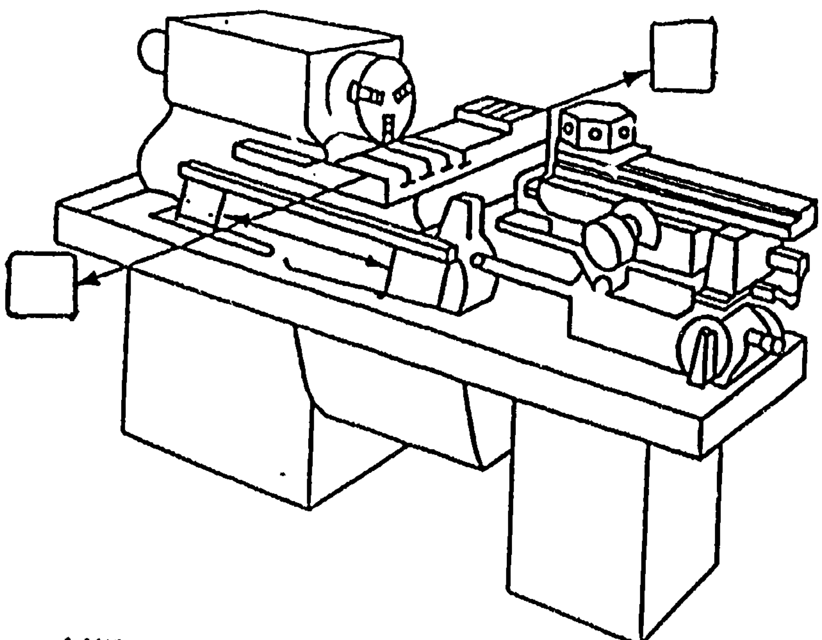
SCORE__________



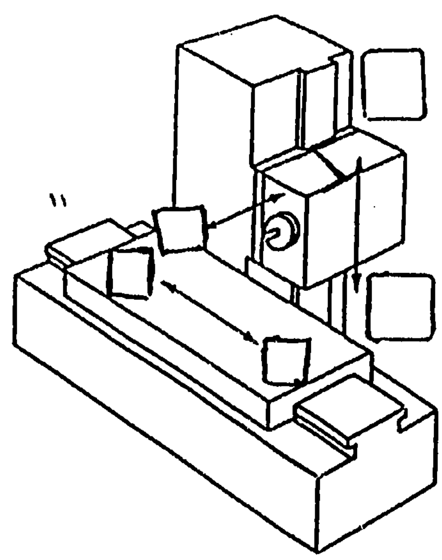
AN N/C TURRET PUNCH PRESS



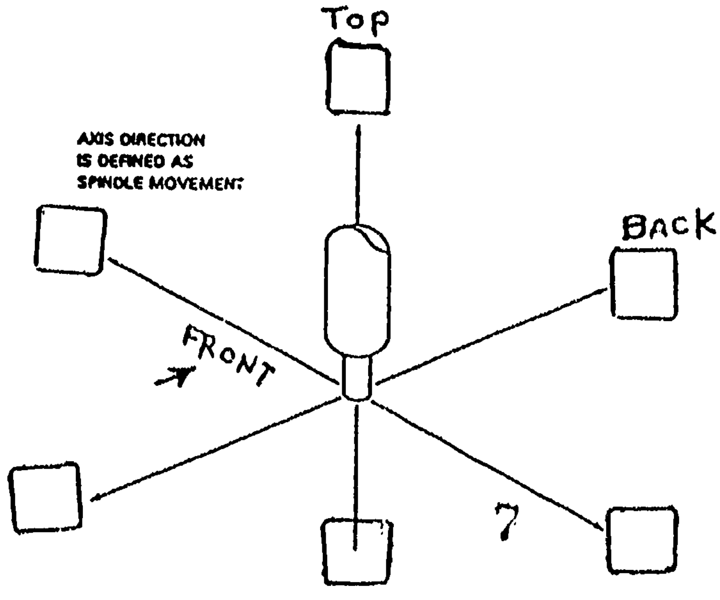
THREE-AXIS VERTICAL MILL



A N/C TURRET LATHE



A HORIZONTAL N/C MACHINE



PRE-TEST

NAME ____________________

DATE __________ SCORE __________

AXIS and NC and CNC ABSOLUTE and INCREMENTAL

1. What does NC stand for and where is it used ? (brief statement)

2. What does CNC stand for and how is it different when programing then NC ?

3. List three (3) advantages of CNC over NC machining ?

a. ____________________

B. ____________________

C. ____________________

4. What effect does the quadrant have when programing the axis direction ?

5. With the right thumb rule :

A. The thumb is which axis __________ and points in which direction ?

B. The first finger points to the __________ axis .

C. The second finger points to the __________ axis.

6. What is meant by absolute value or absolute dimensioning ?

7. What is meant by incremental value or dimensioning ?

8. What is the one BIG error of using relative values when programing ?

THE CONTROLLER

A major area of evolution has been the controller. Control capabilities are the difference between NC and C/NC. We will compare NC to C/NC. The major difference between an NC and C/NC machine is that the C/NC machine has a microprocessor coupled with the NC controller. The C/NC microprocessor has:

- The capability of managing data and making calculations.
- The capability of making decisions based on data input.
- Flexibility in the form of random access memory (RAM).

Refer to Fig. 1-8.

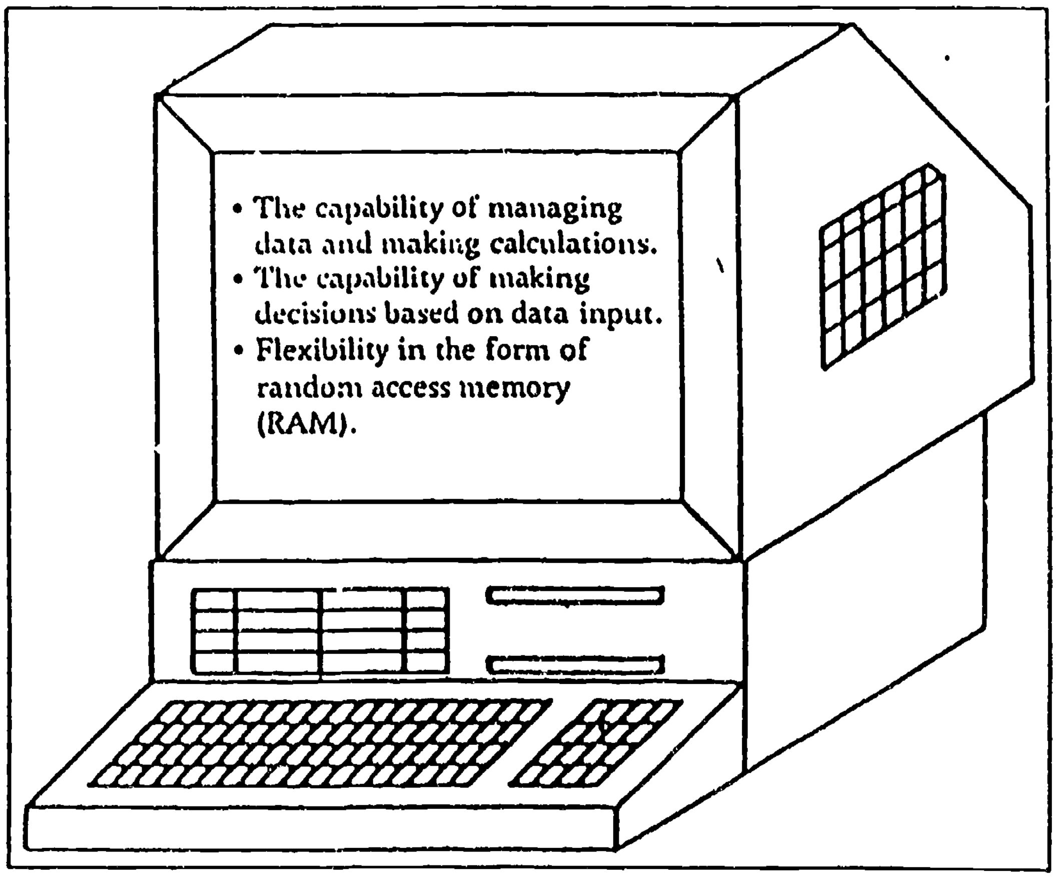
NC Compared with C/NC

NC is the operation of a machine from data stored in a program. The data is a combination of numbers and letters that command machine movements and functions such as "spindle on" to a machine from a *master control unit* (MCU). The MCU directs the machine movements.

An NC machine simply follows directions from a program; it is a slave. There is no flexibility nor authority over the work being done. The program is external to the control such as a punched tape. The program is fixed—all information must be in the program.

Computer numerical control (C/NC) means that data stored within the MCU memory *directs* the operation of a machine. The microprocessor allows the machine to exercise authority over the operation. A C/NC control can make calculations and decisions.

A C/NC machine is more flexible than an NC machine. Similar to NC, a C/NC follows program directions. In addition, it assumes responsibility for calculations and decisions and contains a memory in which the program resides. This means that the program can be changed easily. The memory that can be edited is called random access memory (RAM). A C/NC can detect and help correct problems and communicate with the operator and with external devices such as robots and central programming computers.



NC Intelligence

The NC intelligence was mostly in the program supplied by the programmer. An NC control had a buffer area for the next command. It could follow customizable pre-planned cycles, but that is the limit of NC intelligence. The program had to be specific for each move and operation. Perhaps the biggest single difference was in machining curved shapes. If a curved shape was to be made on an NC, each point for the path of the cutter had to be programmed. Many points were needed to make the curve smooth. Fig. 1-10. This led to long and difficult programs. Errors were common. To avoid these errors, offline programming languages were developed to generate the tool path data. The most common of these languages was *automatic programmed tooling language* (APT). Much math or an offline computer was necessary to produce an NC tape.

NC Curves

Curves were difficult to program in NC. Refer to the NC curve shown in Fig. 1-10. This shows how an NC control can be used to machine a complex shape. It is possible to approximate a circle (or a line of any shape) with a series of short straight lines. If the lines are close enough together, the curve will be very close to a true curve. An NC curve had to have each point in the program. This created long sets of point coordinates. In programming an NC curve, the programmer determines the amount of tolerable divergence from a true curve. Then the APT computer or the programmer calculates points that are close enough to give the correct results.

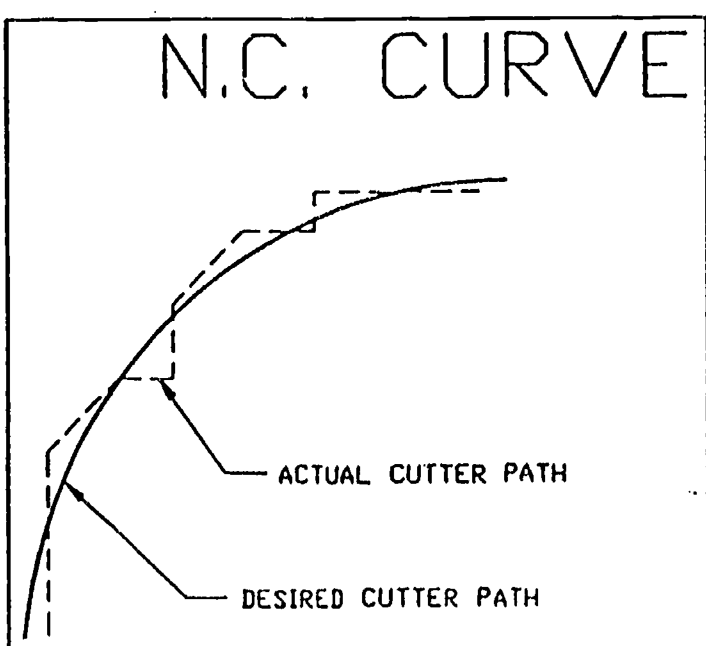


Fig. 1-10. To produce a curve on an NC machine, many short line segments had to be programmed. These short lines approximated the desired shape.

A CNC machine, however, is capable of producing a smooth, exact curve. The programmer inputs only the parameters of the curve. The *parameters* are the conditions of the curve's shape. Thus, the CNC needs the radius, length, center point, and start/stop points or number of degrees in the curve.

CNC Intelligence

On the CNC controller, it was natural to let the computer assume some of the work required of the programmer and the offline computer. CNC made several features available. The following ten features are found only on a CNC machine. These features are possible because of the microprocessor.

1. Random access memory.
2. Tool path compensation.
3. On-line programming and editing MDI.
4. Full geometry programming—true curves, not approximations. Program information describes the shape. The microprocessor generates the machine axis moves perfectly.
5. Tool compensation for size, shape, length, and wear.
6. Constant surface speed on lathe cuts. The rim speed is maintained no matter what diameter is being cut. On a face cut, as the tool moves inward, the spindle speeds up in direct proportion.
7. Part scaling. The computer is requested to generate a bigger or smaller version of the part program. This is often used in mold work where a shrink factor must be included in the mold.
8. Graphic representations of the program. Controls are capable of displaying the program on a screen before running it. This is a safety factor and eliminates errors before actual tryout.
9. Program error analysis. Many controllers are capable of finding certain program errors and alerting the user. The safe work envelope is designated to help avoid cutter-to-machine contact.
10. Advanced control authority. This allows the following in-process inspection; part loading is triggered by the control to a robot; tools are replaced by wear factors or a timed number of parts; tools and work fixtures are set up by probes in the machine, rather than by an operator using an indicator.

Because each of these operations requires a computer, each of the above is possible only with a CNC machine controller.

BEST COPY AVAILABLE

An NC or CNC machine is:

"A SYSTEM IN WHICH ACTIONS ARE CONTROLLED BY THE DIRECT INSERTION OF NUMERICAL DATA AT SOME POINT. THE SYSTEM MUST AUTOMATICALLY INTERPRET SOME PORTION OF THE DATA"

The Major System Components

NC

1. MACHINE TOOL

   SHAPES CUTS & FORMS MATERIAL AT THE DIRECTION OF THE PROGRAM (USUALLY PUNCHED TAPE)

2. CONTROLLER

   READS PROGRAM, DIRECTS MACHINE MOVEMENTS, ALLOWS LIMITED MANUAL MOVEMENT AND CONTROLS SET-UP.

3. TAPE READER

   USUALLY PART OF CONTROLLER READS TAPE AND FEEDS PROGRAM TO CONTROL MECHANISM.

4. TAPE PUNCH

   A TYPEWRITER-LIKE DEVICE THAT PUNCHES THE PROGRAM. THIS IS THE ONLY PLACE AN NC MACHINE MAY BE PROGRAMMED (USUALLY)

CNC

1. MACHINE TOOL

   SHAPES, FORMS & CUTS MATERIAL AT THE DIRECTION OF PROGRAM IN THE CONTROLLER MEMORY.

2. CONTROLLER - (microprocessor)

   A. DIRECTS MACHINE MOVEMENTS FROM PROGRAM STORED IN MEMORY.
   B. ALLOWS ACCESS TO PROGRAM MEMORY FOR WRITING OR EDIT.
   C. ALLOWS MANUAL OPERATION FOR SET-UPS

3. DATA STORAGE

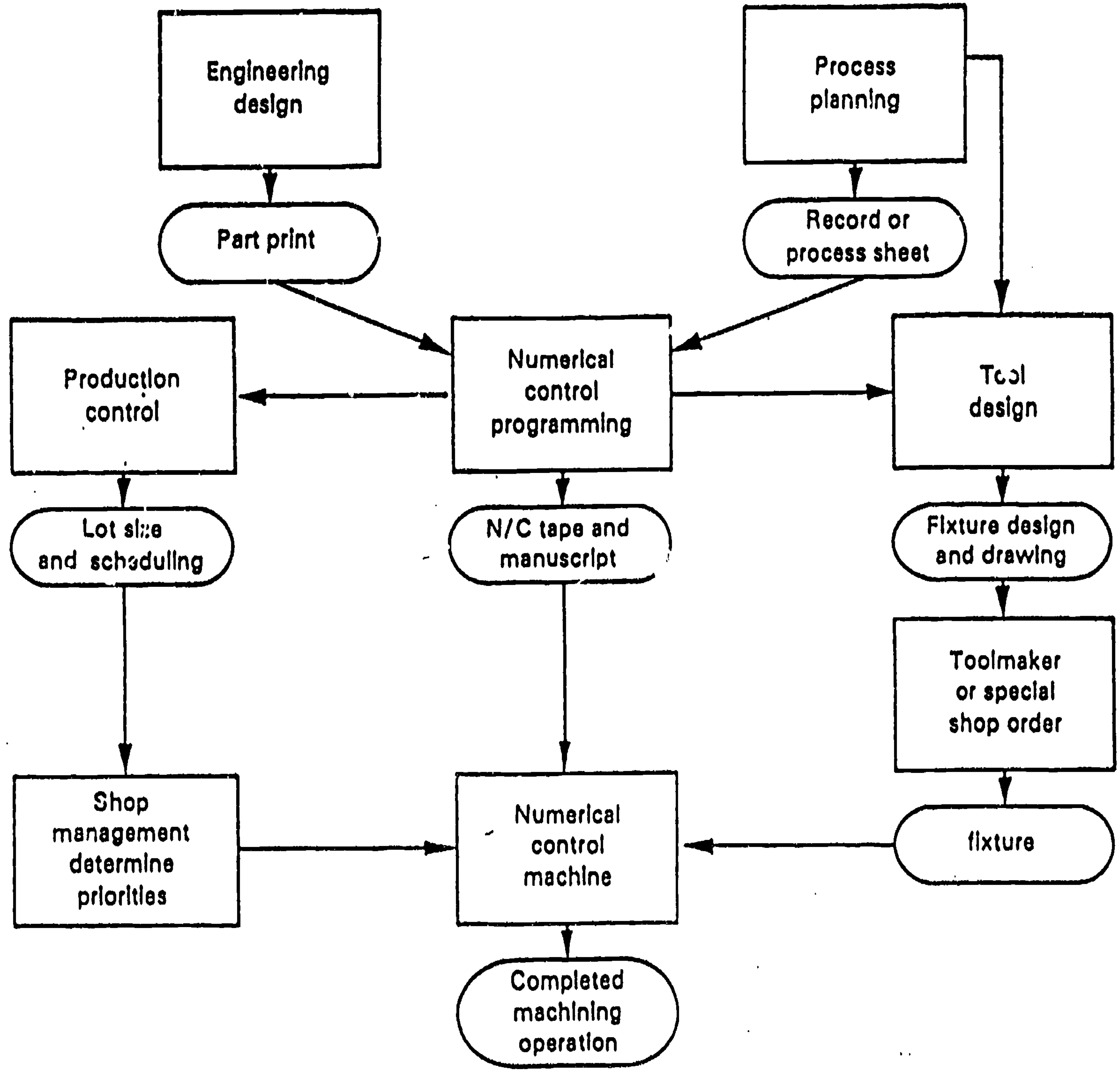
   A. STORES PROGRAMS NOT BEING USED ON PUNCHED TAPE, COMPUTER DISK OR CASSETTE TAPE.
   B. WILL PUT PROGRAM INTO CONTROL UNIT OR RECORD ONE INTO CONTROL This is called program storage & retrieval

4. OFFLINE PROGRAM UNIT

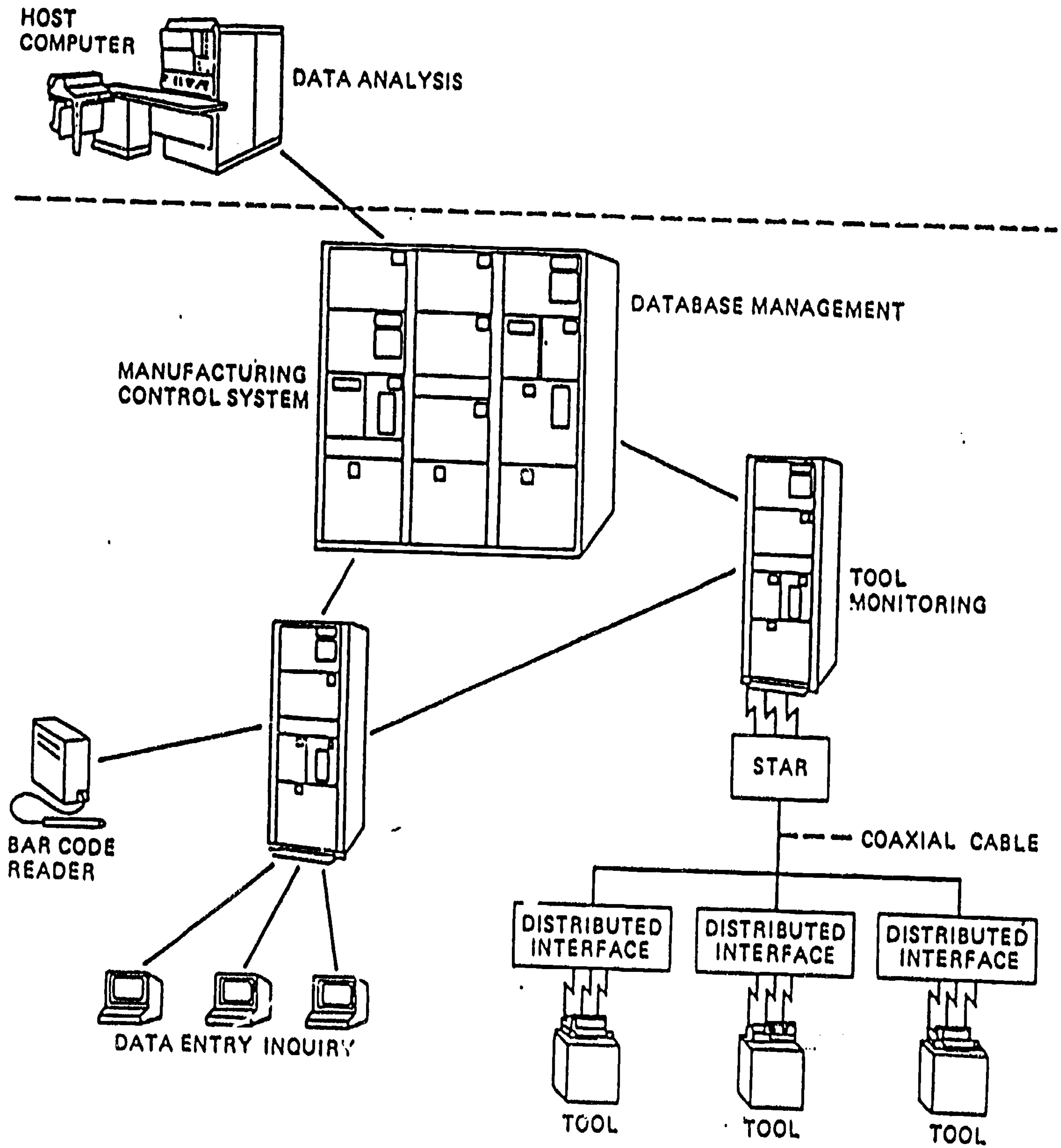
   A. PREPARES PROGRAMS
   B. AIDS PROGRAMMER WITH MATH AND GRAPHIC DISPLAY OF ACTUAL MACHINE MOVEMENTS.
   C. THIS IS USUALLY A MICROCOMPUTER

MAJOR DIFFERENCES

NUMERICAL CONTROLLED MACHINERY	COMPUTER NUMERICAL CONTROLLED MACHINERY
NO MEMORY FOR PROGRAM STORAGE	PROGRAMS MAY BE STORED WITHIN THE CONTROLLER MEMORY
CURVES ARE CONT. PATH APPROXIMATION	CURVES ARE EXACT
PROGRAMS CAN BE EDITED ONLY BY REPROGRAMMING TAPE	COMPUTER HAS RANDOM ACCESS MEMORY (RAM) ALLOWING OPERATOR EDITING ON SITE
CONTROLLER MAKES NO DECISIONS OR ADJUSTMENTS WHILE OPERATING	CONTROLLER CAN MAKE DECISIONS SUCH AS: IS PART TO TOLERANCE OR COMPENSATE BACKLASH TO AVOID CLIMBING IN INTERNAL POCKET CORNERS
MUST BE PROGRAMMED AWAY FROM MACHINE	MAY BE PROGRAMMED ON THE MACHINE (MDI) MANUAL DATA INSERT PROGRAMMING



A FLOW DIAGRAM OF THE STEPS IN PROCESSING AN N/C PROGRAM

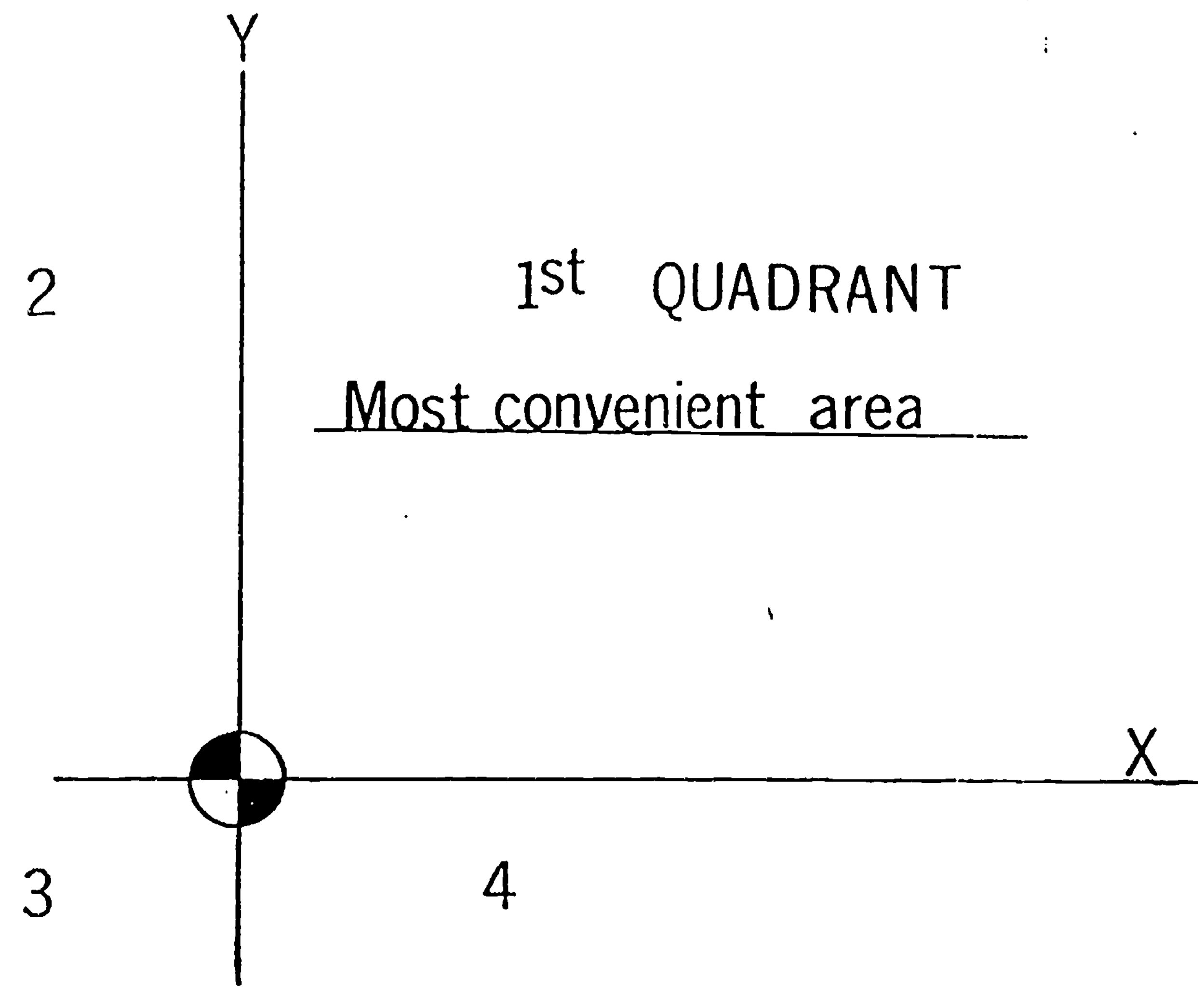


A DISTRIBUTED HEIRARCHICAL SYSTEM (*International Business Machines*)

PROGRAM REFERENCE ZERO The Quadrants

If we divide the X-Y plane into four parts starting from the zero point, each fourth is called a quadrant The quadrants are numbered in a counter-clockwise direction. We place the PRZ at the center of the quadrants and make all the programmed moves to the right and up in the Y axis. We try to always program in the first quadrant.

You may choose any location for the Program Reference Zero. However the lower left corner of the part is ideal because all of the points in absolute will be positive. If the PRZ is the farthest left in the X axis and farthest toward the operator in the Y axis then all moves will stay in the first quadrant.



We try to machine in the first quadrant by choosing the PRZ at the far left, lower corner. This is the most minus X and minus Y point on the part. The PRZ need not be located on the actual part. Often it is more convienient to choose a PRZ that is on a fixture or tooling so we can put multiple parts in for machining. This tooling point is usually a hole that is easily located by the machine operator. This point would also be chosen so that all program moves will be positive using absolute values. While this does not affect relative values, we still choose the PRZ to the lower left for consistency.

What Is a Machine Axis?

MOTION AND DIRECTION

Axes are used to identify motion. All C/NC machines require some method of identifying which motion is needed in the program. For example, on a vertical milling machine, if the programmer requires motion left or right, the X axis is called upon, while in and out motion of the table would require the Y axis.

Direction is determined by sign value. Once the axis is identified, the sign of the axis, plus or minus, determines the direction of its movement: left or right, in or out, up or down, clockwise or counterclockwise.

Refer to the milling machine in Fig. 3-1. The X axis of this milling machine table is the left and right movement. If the programmer needs a tool movement to the right, he or she would call out a "plus X movement." A movement to the left would be "minus X." Movement of the tool inward (away from the operator) is plus Y, and tool-up would be plus Z.

Refer to the C/NC lathe shown in Fig. 3-2. A C/NC lathe has two common axes. They are the X and Z axis. Plus X movement of the tool would be the cross slide moving away from the work. A plus Z movement would also be away from the work, with the saddle moving. Note that the Z axis is perpendicular to the X axis.

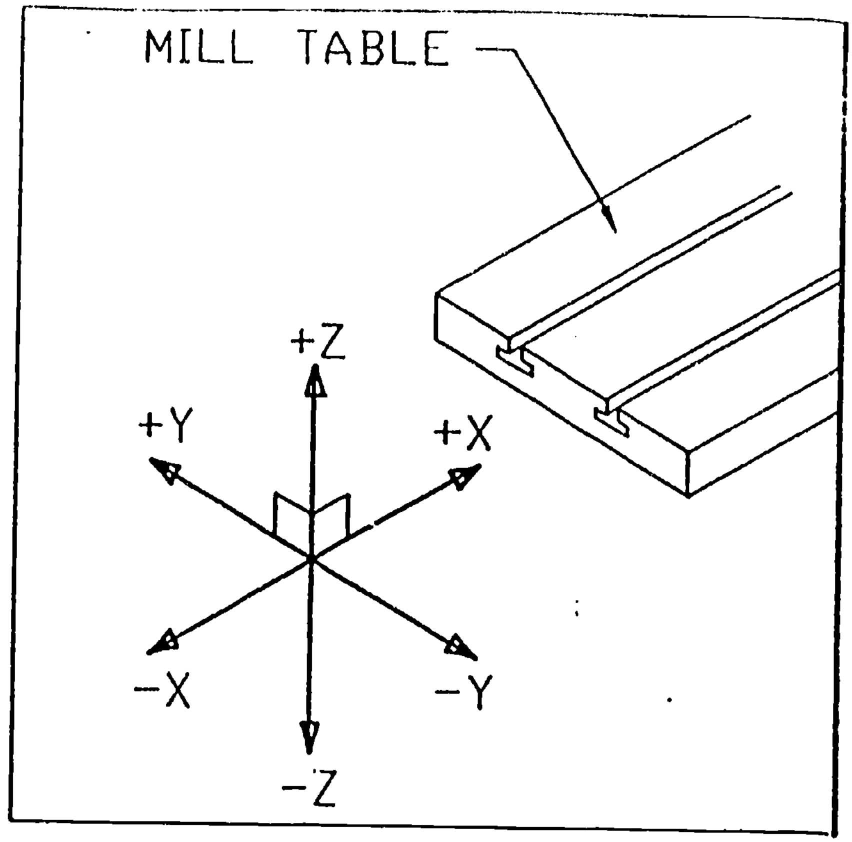


Fig. 3-1. The three axes of a vertical milling machine. Th[illegible] axis is usually parallel to the floor on most C/NC machines

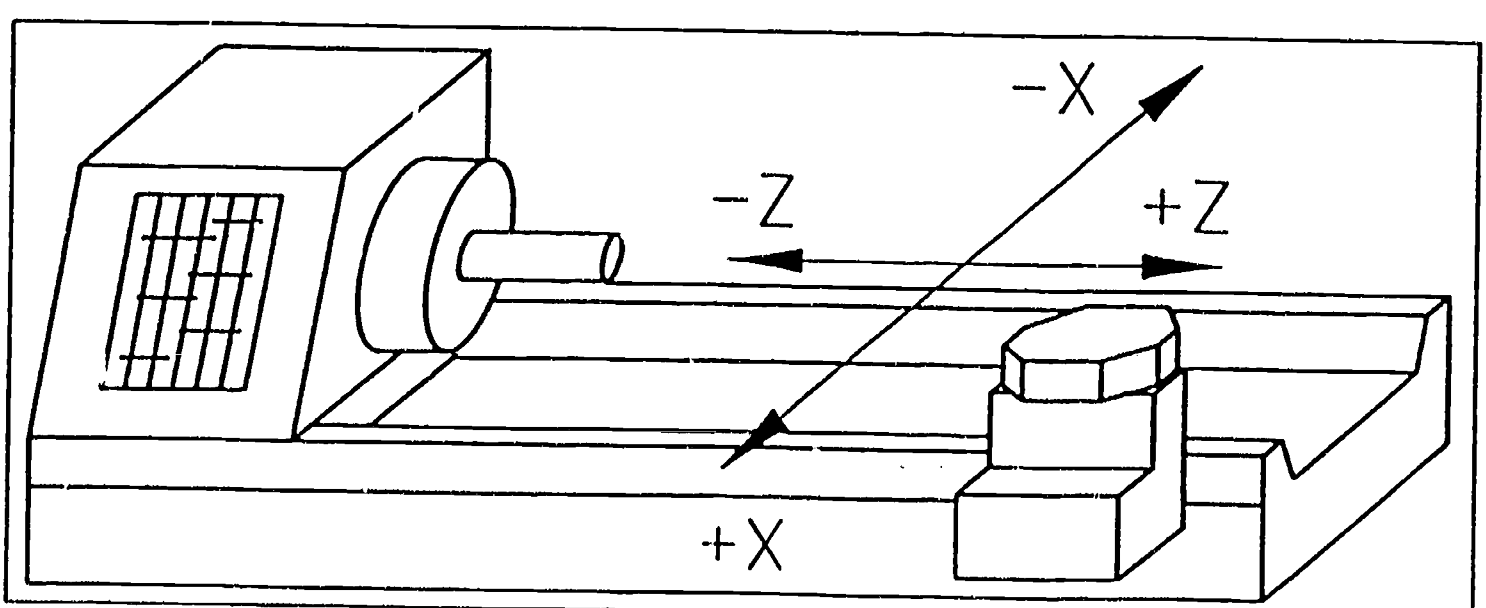
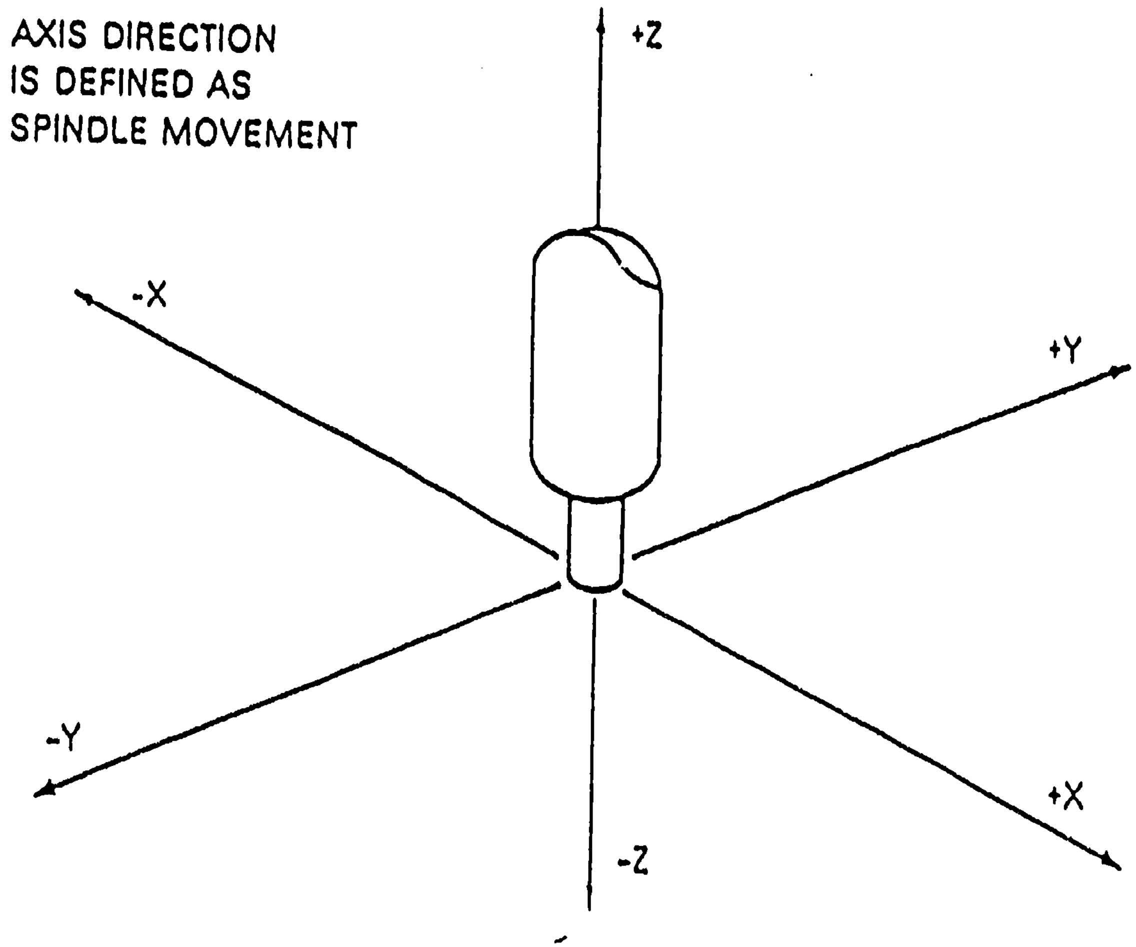


Fig. 3-2. The X and Z axes on a C/NC lathe.


AXIS DIRECTION
IS DEFINED AS
SPINDLE MOVEMENT
+Z
-X
+Y
-Y
+X
-Z

RELATIVE TOOL MOVEMENT

Always think of the tool as moving. When programming or setting up a C/NC machine, view the axis movement as though the tool is moving. This is called *relative tool movement.*

As an example, refer to Fig. 3-3. After drilling hole A on this NC drilling machine, we wish to drill hole B 2.5" farther to the right on the part.

The programmer would call out a plus X movement of 2.5". Would the tool actually move 2.5" to the right? Actually, the table moves to the left. Thus, the tool position *relative to the part*, is shifted to the right. We ignore the table movement and think about the tool as moving. From this point on we will always envision the tool as the moving object. This is a standard in C/NC programming. If the tool moves or the machine slide moves, always think of the tool as the moving object. By following this rule, programming is simplified. Also, there is no confusion as to the movement that will be produced.

For another example, refer to Fig. 3-1. Using the concept of relative tool movement, we see that the axis directions on a vertical mill are:

Tool to the right	= plus X
Tool left	= minus X
Tool away from front	= plus Y
Tool toward	= minus Y
Tool up from table	= plus Z
Tool down	= minus Z

Refer to Fig. 3-2.

Crossslide tool toward work	= minus X
Crossslide away from work	= plus X
Saddle away from chuck	= plus Z
Saddle toward chuck	= minus Z

The Axis System

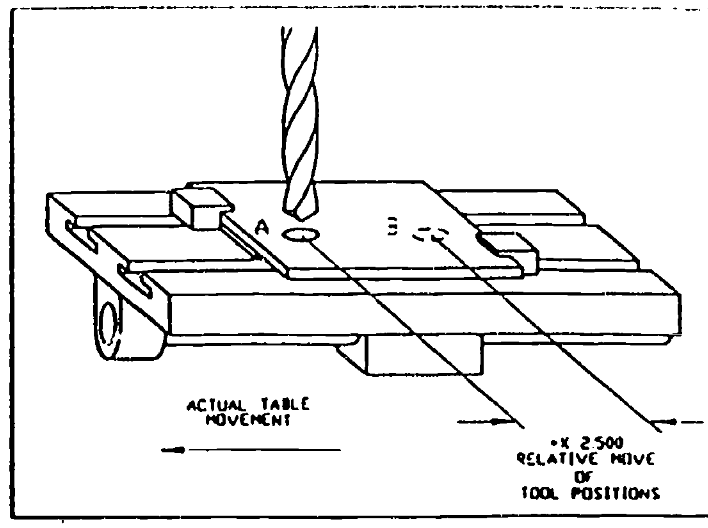
THE AXIS FRAMEWORK

Note that the three axes on the mill (Fig. 3-1) and the two on the lathe (Fig. 3-2) are at 90° to each other. This is called an *orthogonal axis frame. Orthogonal* means "at 90°." The standard axes on most C/NC machines are orthogonal. Fig. 3-4.

RIGHT-HAND RULE

You can identify the axis framework on most C/NC machines by the right-hand rule illustrated in Fig. 3-5.

If the thumb of your right hand points along the positive X axis, the first finger will point out the positive Y axis and the second finger will identify the positive Z axis.



Relative tool movement means that we always think of the tool as the moving object. Hole B is to the right if you think of the tool as the moving object.

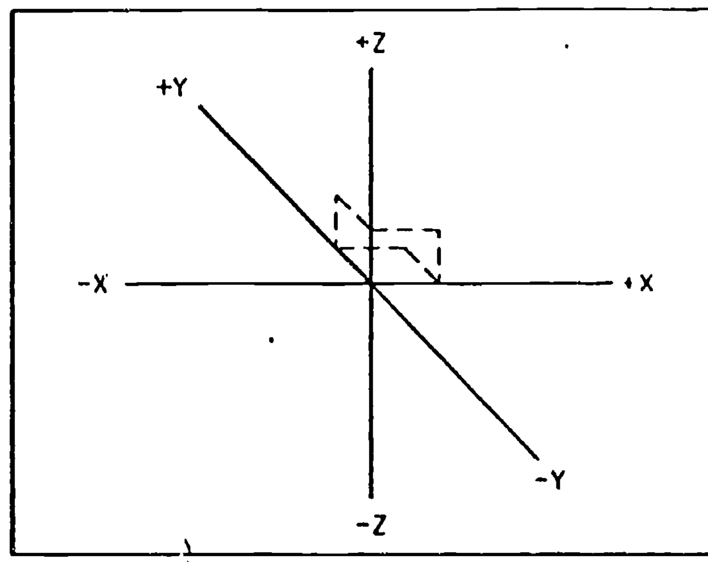


Fig. 3-4. An orthogonal axis framework.

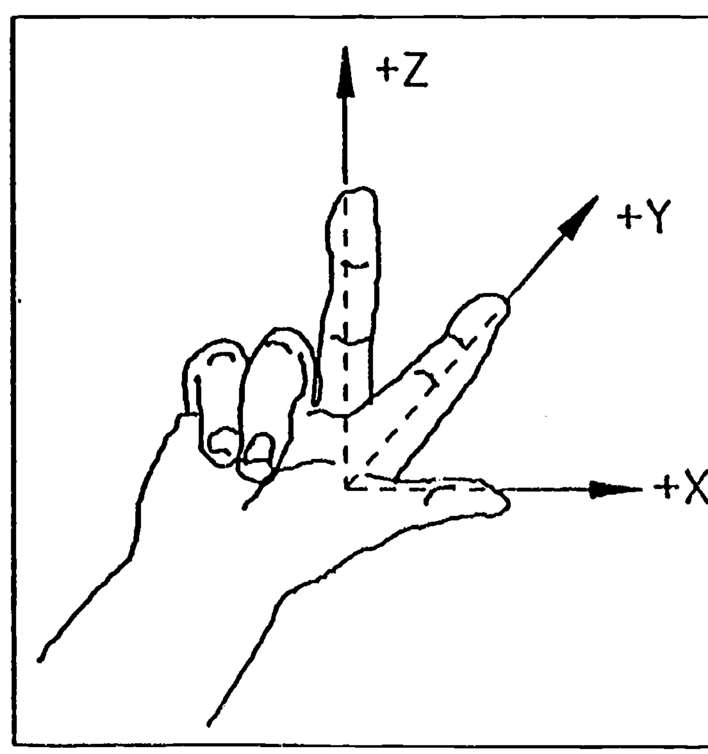


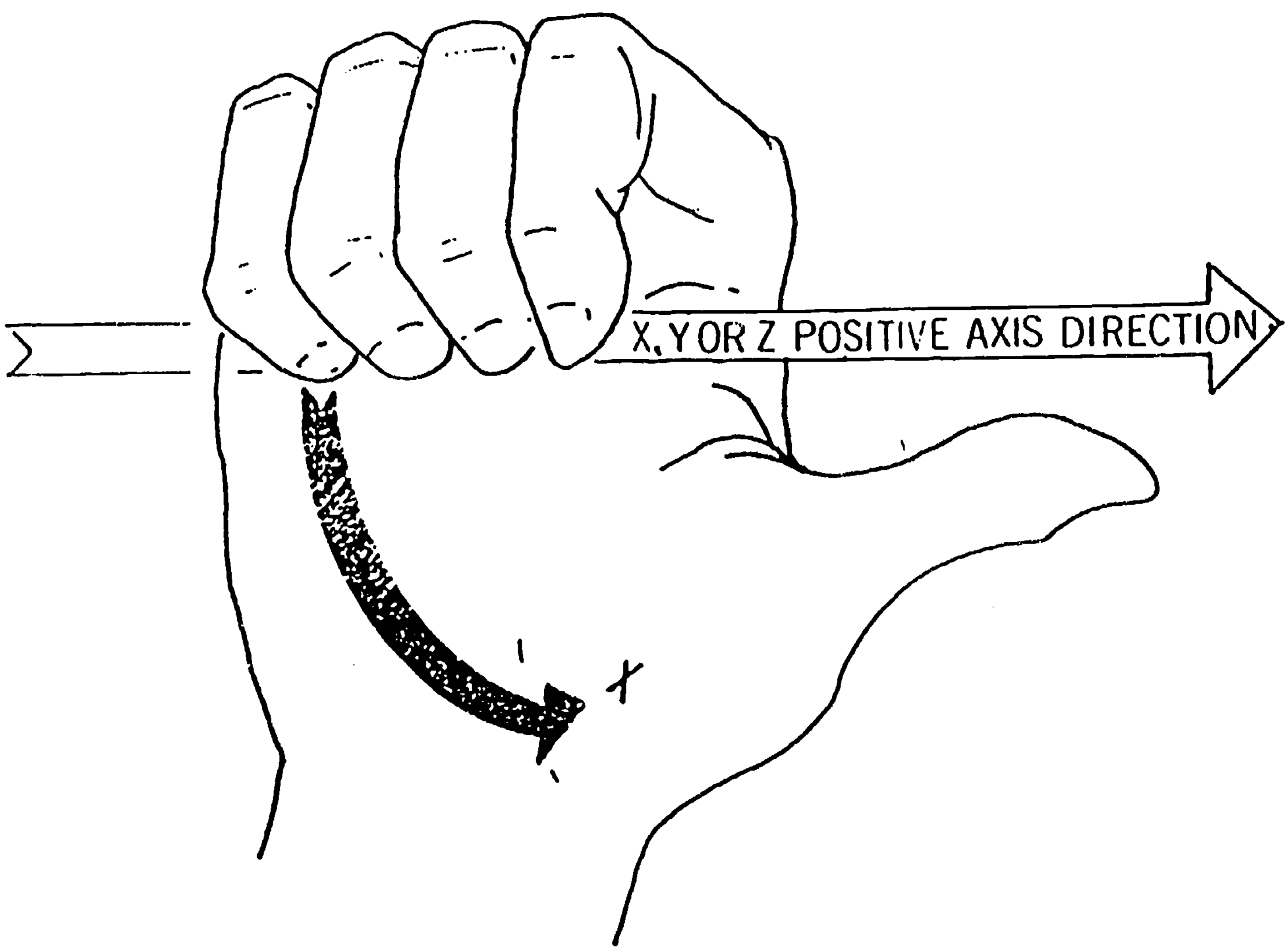
Fig. 3-5. The right-hand rule shows you the relationship of the primary linear axes of C/NC machines.

"RULE OF THUMB"

For Auxiliary Motion and Polar Arcs

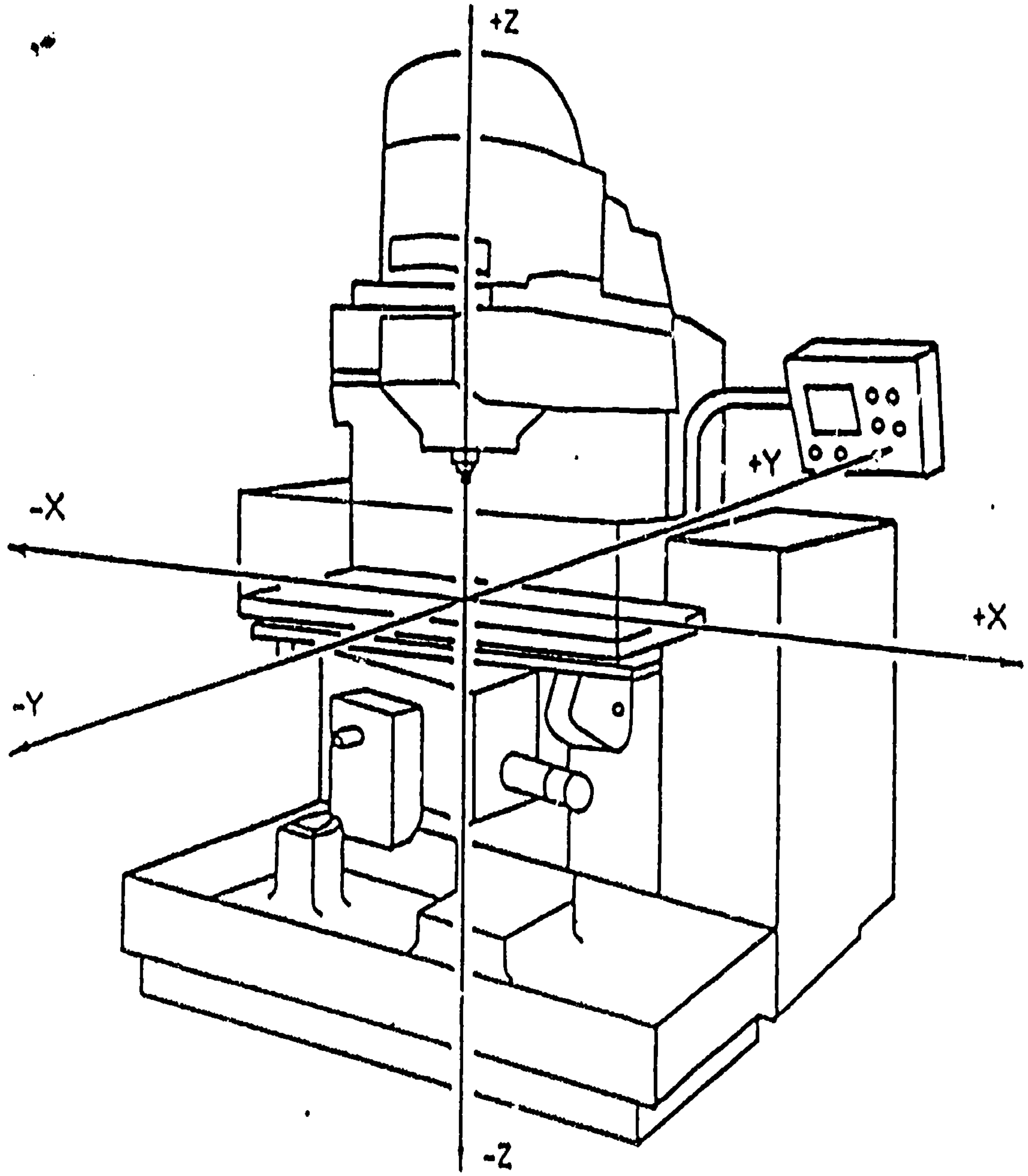
To determine the sign of a rotary motion - positive or negative:

POINT THE THUMB OF YOUR RIGHT HAND IN THE DIRECTION OF THE POSITIVE AXIS OF ROTATION.



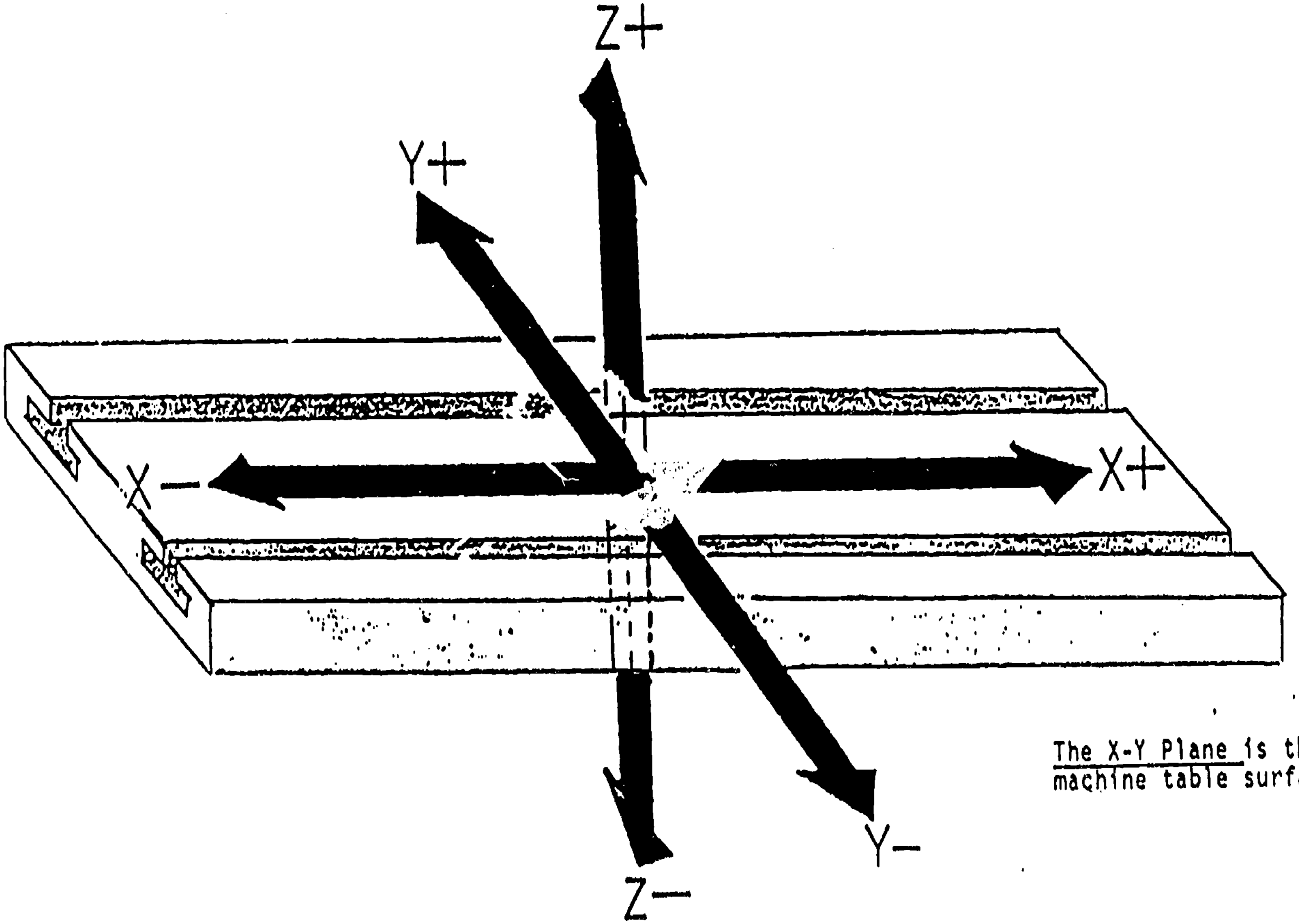
YOUR FINGERS WILL POINT IN THE POSITIVE ROTARY DIRECTION.

The rule of thumb works for polar arcs and for auxiliary axis motion. If we wish to call out an arc or a axial move of 20 degrees, we use the RULE OF THUMB to determine the sign of the entry.

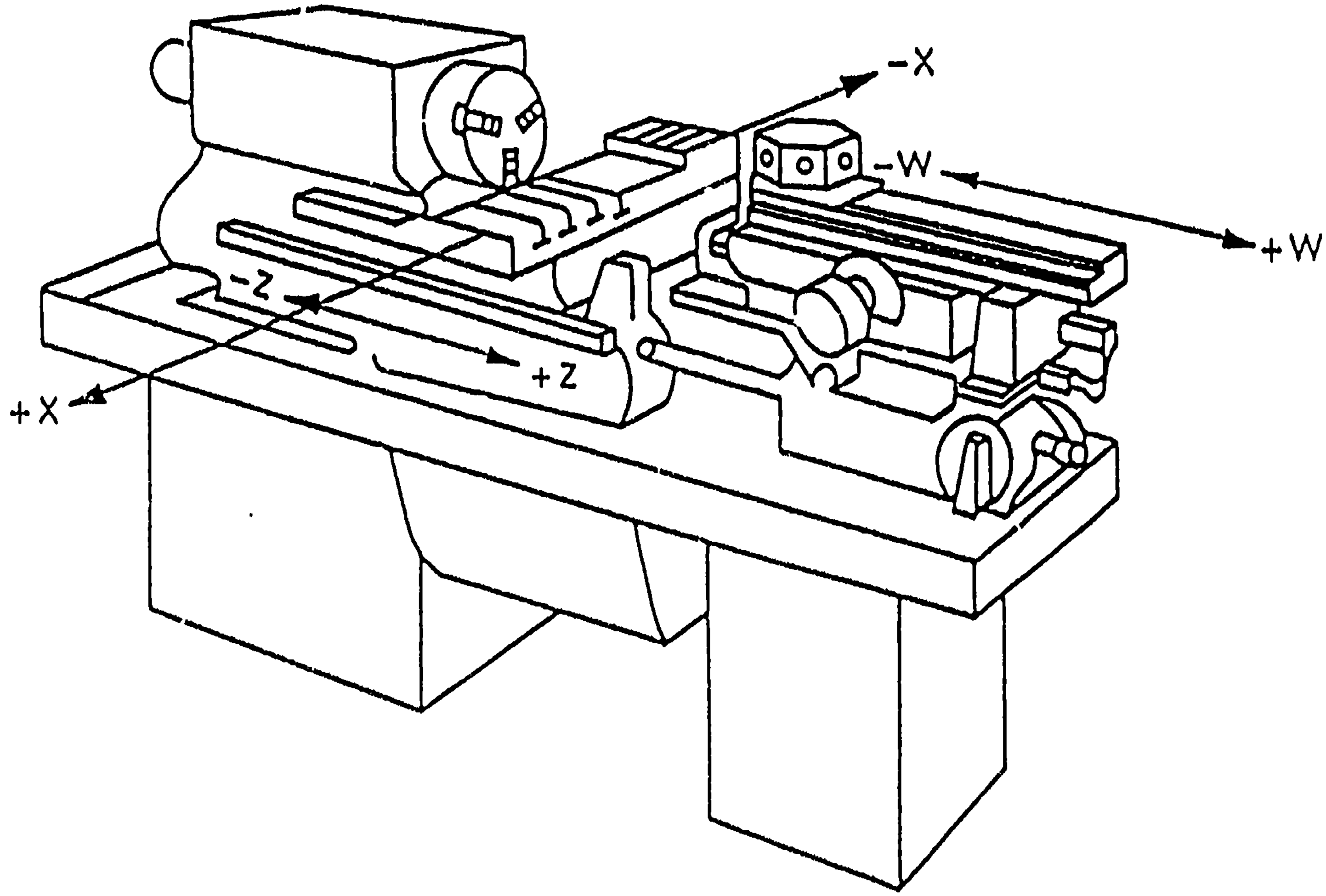


THREE-AXIS VERTICAL MILL

Milling Machine Axis



The X-Y Plane is the machine table surface



A N/C TURRET LATHE

ABSOLUTE VALUES COMPARED TO RELATIVE VALUES

Both systems have usefulness. Both systems have advantages and disadvantages. A smart programmer takes full advantage of both by being flexible and using whatever unit is correct for:

THE ENGINEERING DRAWING - BLUEPRINT

It is the drawing that will dictate whether a certain unit should be used to write the program. Often the most efficient way to program is with a mixture. By choosing the point identifications well, a good programmer may save much needless math and chance for errors.

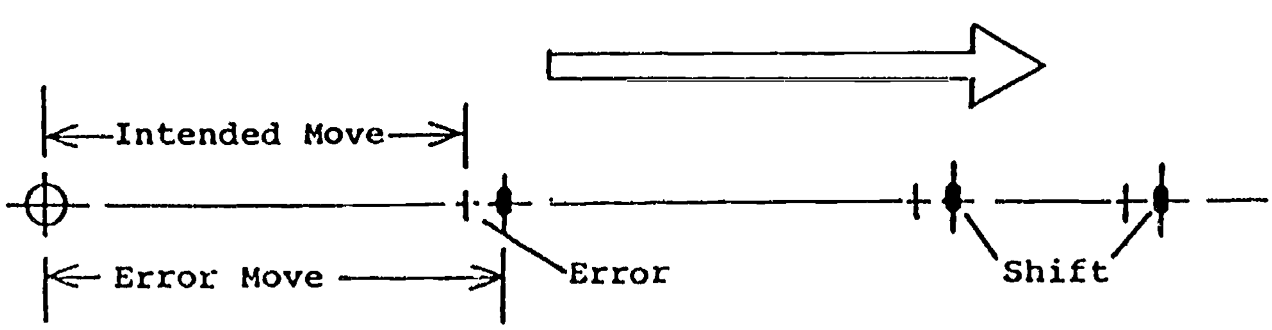
RELATIVE VALUES

ADVANTAGES:
Relative values are the easier to understand because we tend to think of the tool movement as increments from one point to the next. Each new move is a new task and the point values match this thinking. Often, math is less complicated using relative values. Relative values are easier for beginning programmers to understand.

DISADVANTAGES:
Relative values have one very big danger. If we make an error between two points, that error will affect each move after that.

Example, We write 2.300 instead of 2.030 for a certain step in a program. The error amount would be a plus .270 too much. If we continue to program in relative values, each point after will be shifted to the plus side by an additional .270

EACH POINT IS SHIFTED TO THE RIGHT BY .270



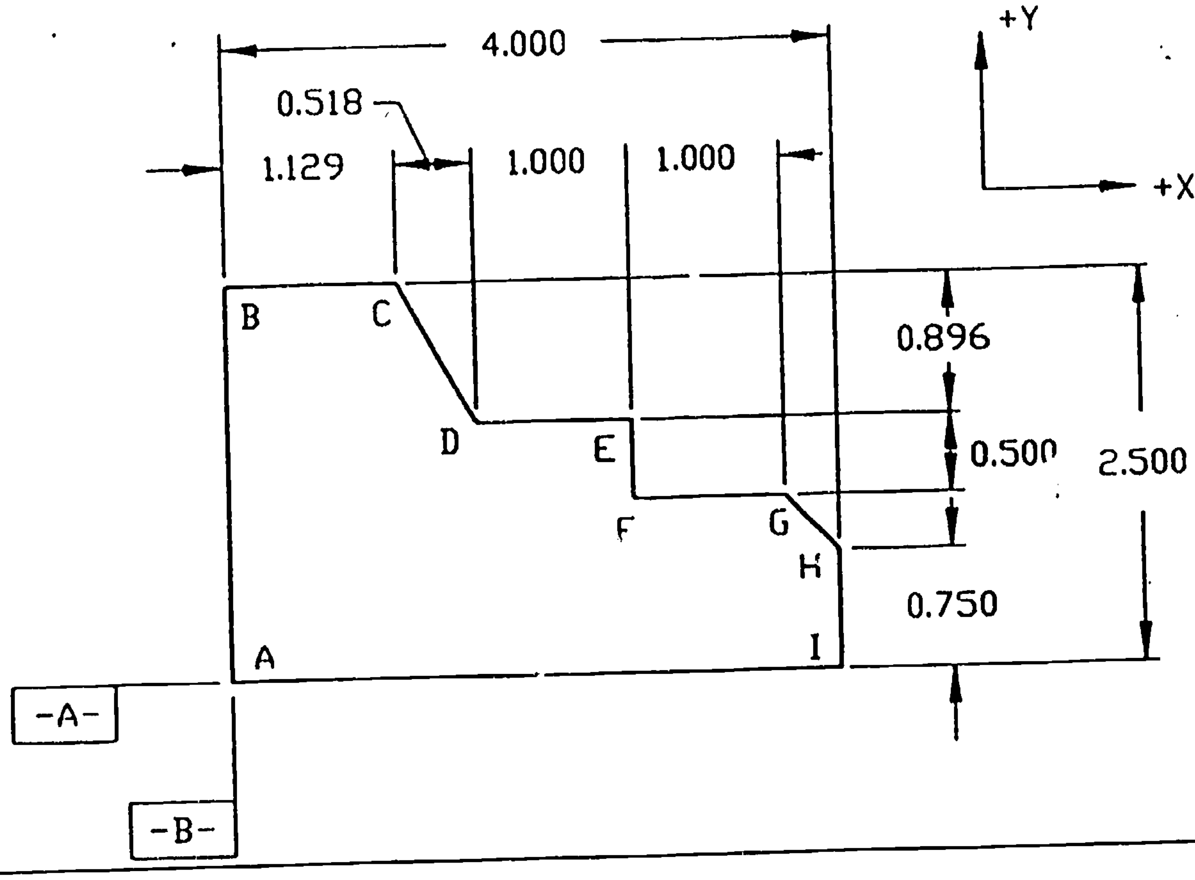
This is known as accumulative error. Accumulative error means that the errors will continue to add up (accumulate) and affect each new movement. This is a real danger with relative point values.

ABSOLUTE VALUES

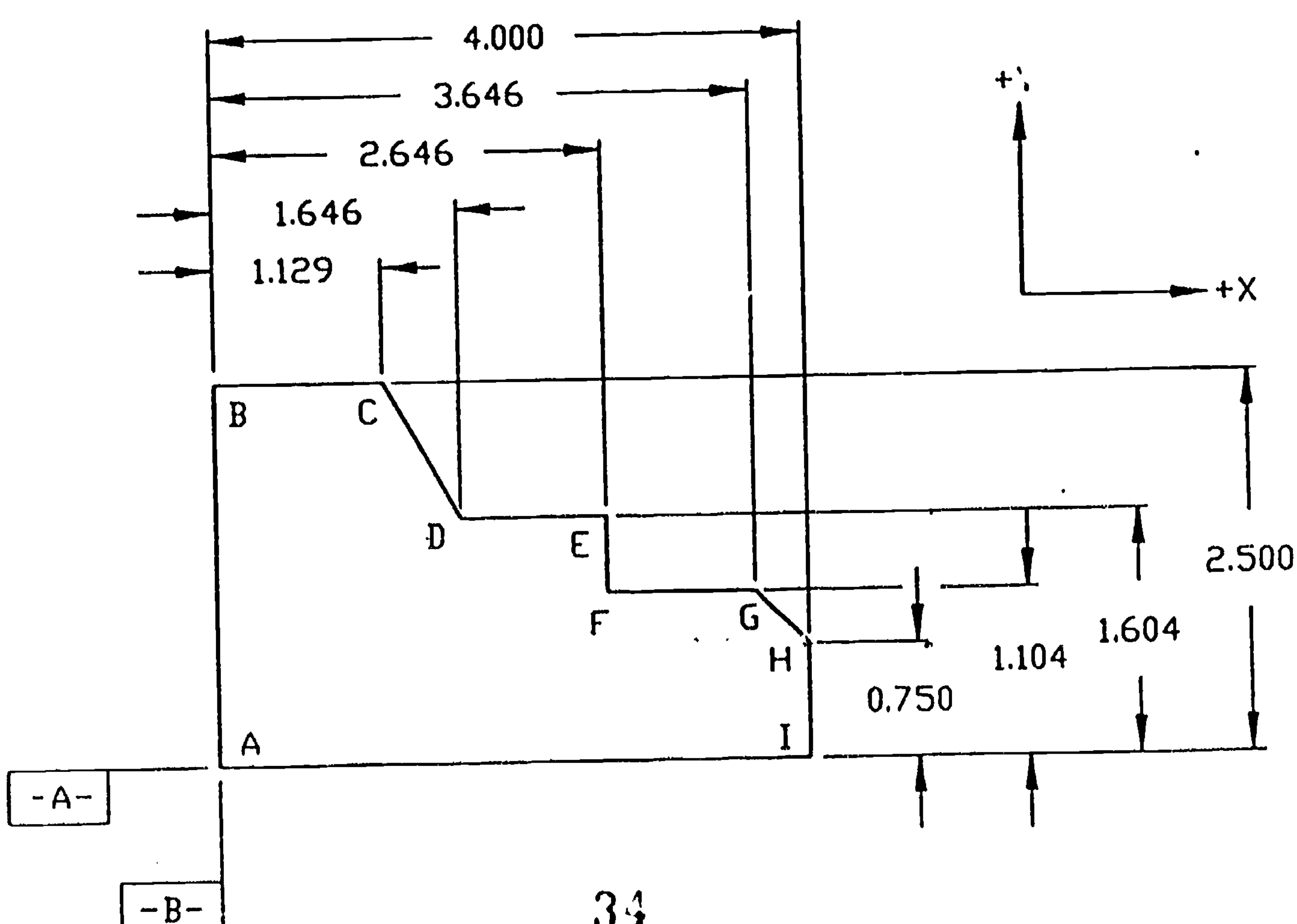
ADVANTAGES:
Absolute values have the advantage of not producing accumulative errors. Thus, they can only effect the single movement in which the error was entered. Absolute values also simplify programming if the drawing uses base line dimensioning.

DISADVANTAGES
Often even with baseline drawings, we must think and work in terms of increments for the programming of some shapes so we end up doing a lot of adding and subtracting to find the next distance using absolute. There is more math to do in general using absolute. Also the concept of always referring back to a base position is somewhat abstract and not always comfortable especially for new programmers.

INCREMENTAL DIMENSIONING



ABSOLUTE DIMENSIONING

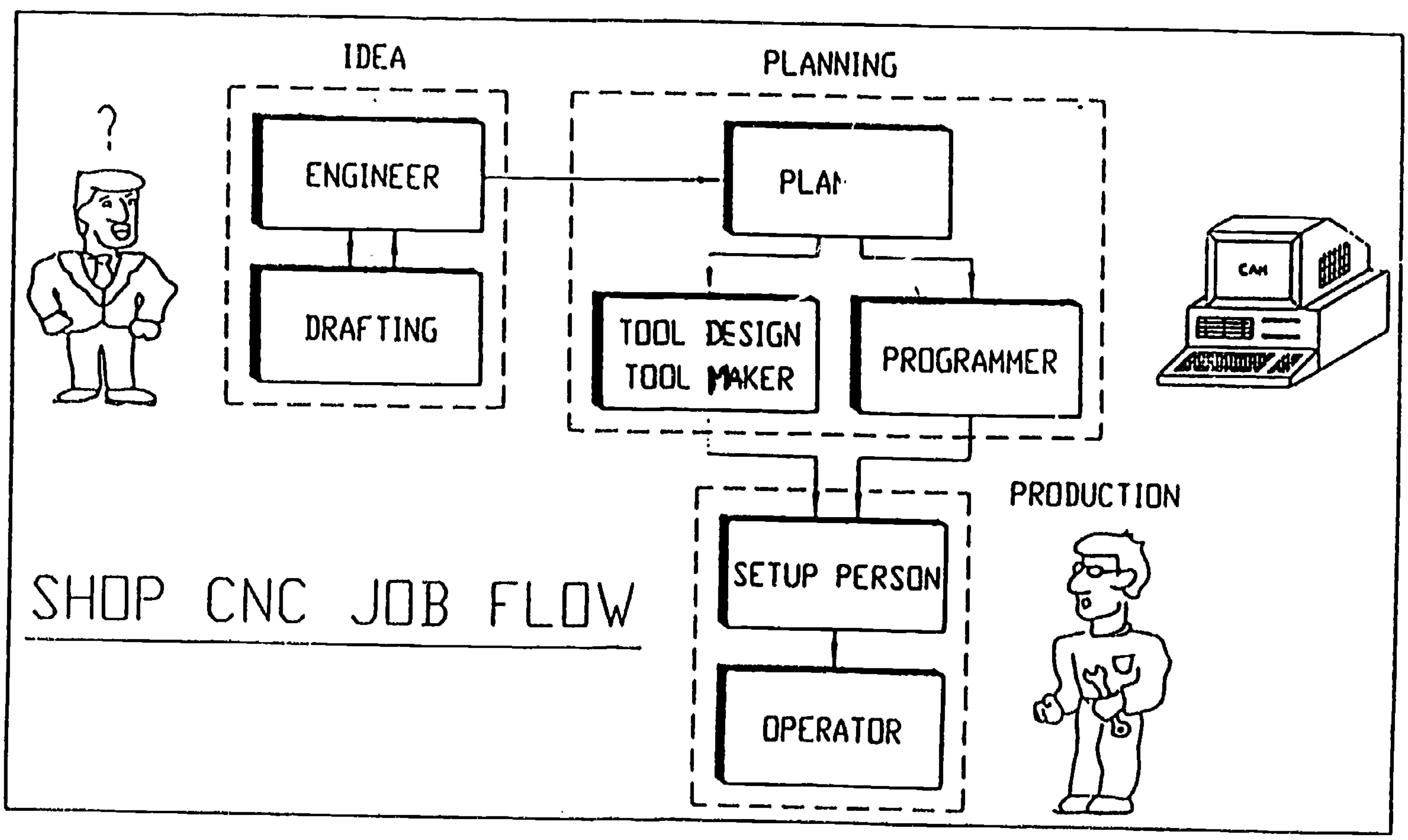


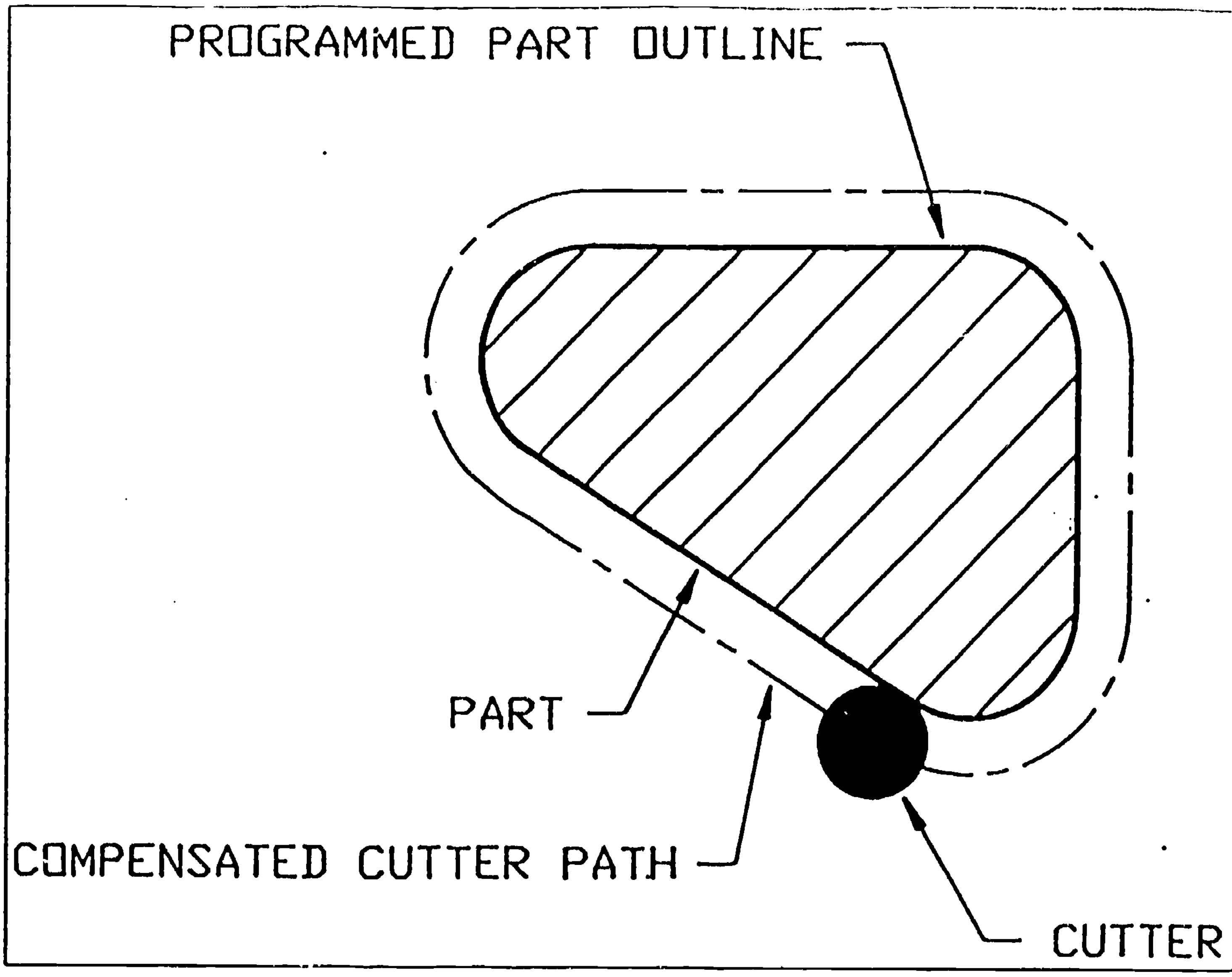
SAFETY RULES AND PRECAUTIONS

1. KNOW YOUR MACHINE-Read the Operating Manual CAREFULLY. Learn the machine features, applications and limitations. Follow all recommended operating procedures.
2. GROUND THE MACHINE-Follow instructions for grounding as described in the manual.
3. KEEP WORK AREA CLEAN-Cluttered areas and benches invite accidents.
4. AVOID DANGEROUS ENVIRONMENT-Do not use this machine in damp, wet, gaseous or explosive locations. Keep work area well lighted.
5. KEEP SMALL CHILDREN AWAY-Small visitors should be kept away from the work area.
6. DO NOT FORCE THE TOOL-It will not do a better and faster job in removing material.
7. USE THE RIGHT TOOL-It will do a better and faster job in removing material.
8. WEAR PROPER CLOTHES-Loose clothing and ties can get caught in moving parts.
9. USE SAFETY GLASSES-Most cutting tools can throw dangerous and hot chips. Wear a face and dust mask if the cutting operation creates dust.
10. SECURE WORK-Use a clamp or vise to hold work. It is safer than using your hands and it frees both hands to operate the machine.
11. MAINTAIN TOOLS WITH CARE-Keep tools sharp and clean at all times for best and safer preformance.
12. DISCONNECT MACHINE-When not in use.
13. REMOVE ADJUSTING KEYS AND WRENCHES-Form a habit of removing adjusting wrenches and keys before operating the machine. Do not leave parts or warenches on the table.
14. KEEP HANDS AWAY FROM CUTTING EDGES AND MOVING PARTS.
15. DO NOT USE AN AIR GUN TO CLEAR CHIPS-This can blast the chips in between the slides and through the louvers into the electronics... VACUUM OR BRUSH ONLY.

MACHINE SHOP
C/NC PROCESS

The work needed to complete a C/NC job fall into three main phases: creating the idea, planning, and production. Fig. 2-1. Each one of these tasks requires distinct job skills.





Computer tool radius compensated cutter path. The input program is the actual part shape. The generated path is the cutter centerline to produce the part.

Tool Compensation

Tool compensation is one of the most useful aspects of CNC. We will look further into tool compensation in later units.

Inputting the Part Shape

To input the part shape, the programmer writes the part shape into the program. Then he requests the computer to adjust for the actual path the tool should follow to compensate for the size of the tool. On a milling machine, the control can allow for a new diameter or length of the cutter. On a lathe, the radius on the nose of the cutter may be changed. Yet the control can still recalculate the path the tool is to take to generate the original part. This eliminates calculations. Fig. 1-9.

If the tool becomes dull and must be replaced, it need not be the same size. Once the controller is informed of the changed size, it can compensate (adjust) the tool path for the new tool size or shape.

An NC machine program is locked to a cutter of a certain size or shape. If that cutter is replaced, it must be replaced with an exact duplicate. If it is not, the part will be bigger or smaller than intended due to the size of the new tool.

SOFTWARE AND FIRMWARE EVOLUTION

Of equal importance to controller evolution are the operating instructions and capabilities built into the control. *Firmware* suggests intelligence you cannot change, while *software* is changeable. Either term is used to indicate the intelligence level of the control. From previous discussions we have learned NC controls had little intelligence because they were a controller only and did not have the microprocessor as part of their system.

STEPS FOR RUNNING AN EXISTING PROGRAM ON A CNC MILL

1. Install a 1/4" end mill cutter.

2. Loosen quill clamp and return quill to the highest position and tighten.

3. Have workpiece centered and mounted on table.

4. Make sure SPINDLE ON/OFF CONTROL is switched to the PROGRAM postion.

5. Turn on main power.

6. Display will read "READY?"

7. Answer by pressing YES.

8. Machine will initialize and return to machine "hard home".

9. Display will read "MODE?"

10. Answer by pressing LINE NO.

11. Type in first step of program. The first step of the program must include the "START" command. Example: 000 START INS 01 You would enter 000.

12. Change mode to PROGRAM RUN. Push this twice. Machine will run a pre-check for error detection and then ask "NONSTOP?"

13. Answer YES. Display will read "SETUP--->zcxyu". Now you establish your PRZ (program reference zero). This is the point on your work where you want the machine to start from. The front left corner is a common PRZ for programs on this machine.

    The axis to set-up are: z - vertical
    c - z clearance
    x - longitudinal
    y - crossfeed
    u - disregard

    By pressing the NO button, you rotate the order of the axes until the one you want is in front of the >. The letter in front is the active axis when you use the jog keys. Press the NO key until the z is in front.

14. With the z in front, move the cutter down to about 1/2" above the workpiece by using the jog keys. Loosen the quill clamp, lower cutter to touch top of workpiece and tighten quill clamp. Press SET UP REF key for setting that reference. The z becomes capital Z.

15. Press the NO button to select c. This is to set the z clearance reference. Use the z up jog key and bring the cutter up to clear the workpiece. Press SET UP REF key and c becomes capital C.

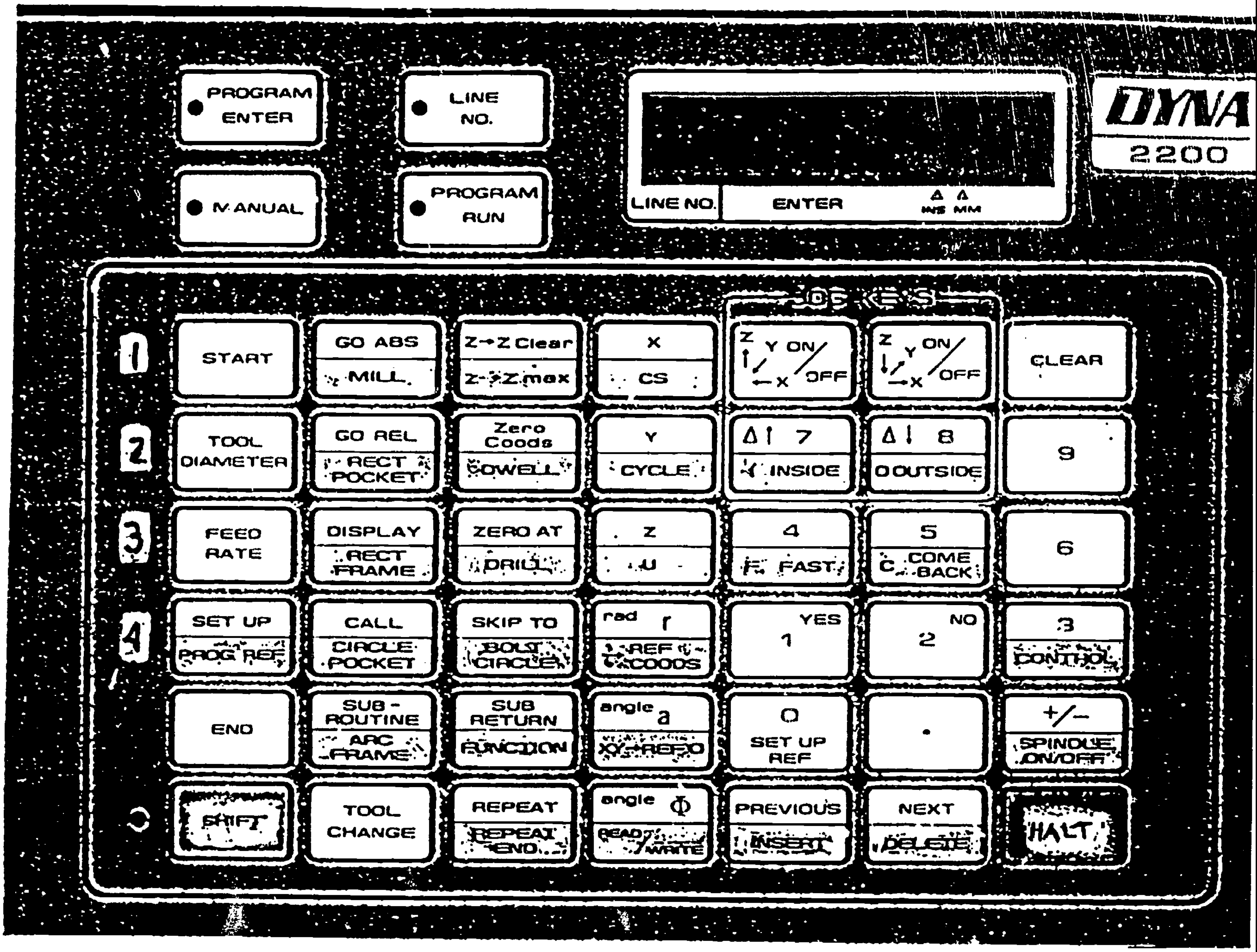
16. Press the NO key to select x. Move the x axis by using the jog keys until the center of the cutter lines up approximately with PRZ. Press SET UP REF to set x reference and x becomes capital X.

17. Press the NO key to select y. Move the y axis by using the jog keys until the center of the cutter lines up approximately with PRZ. Press SET UP REF to set y reference and y becomes capital Y.

18. ZCXY should all be capital letters.

19. Press the NEXT key to execute the program. Machine will run program and stop at the setup line for another workpiece.

CONTROLLER


PROGRAM ENTER
LINE NO.
MANUAL
PROGRAM RUN
DYNA
2200
LINE NO.
ENTER
INS MM
START
GO ABS
MILL
Z→Z Clear
Z→Z max
X
CS
Y ON/OFF
CLEAR
TOOL DIAMETER
GO REL
RECT POCKET
Zero Coods
DOWELL
Y
CYCLE
7
INSIDE
8
OUTSIDE
9
FEED RATE
DISPLAY
RECT FRAME
ZERO AT
DRILL
Z
U
4
FAST
5
COME BACK
6
SET UP
PROG REF
CALL
CIRCLE POCKET
SKIP TO
BOLT CIRCLE
rad r
REF COODS
YES
1
NO
2
3
CONTROL
END
SUB-ROUTINE
ARC FRAME
SUB RETURN
FUNCTION
angle a
XY→REFO
0
SET UP REF
.
+/−
SPINDLE ON/OFF
SHIFT
TOOL CHANGE
REPEAT
REPEAT END
angle Φ
READ/WRITE
PREVIOUS
INSERT
NEXT
DELETE
HALT

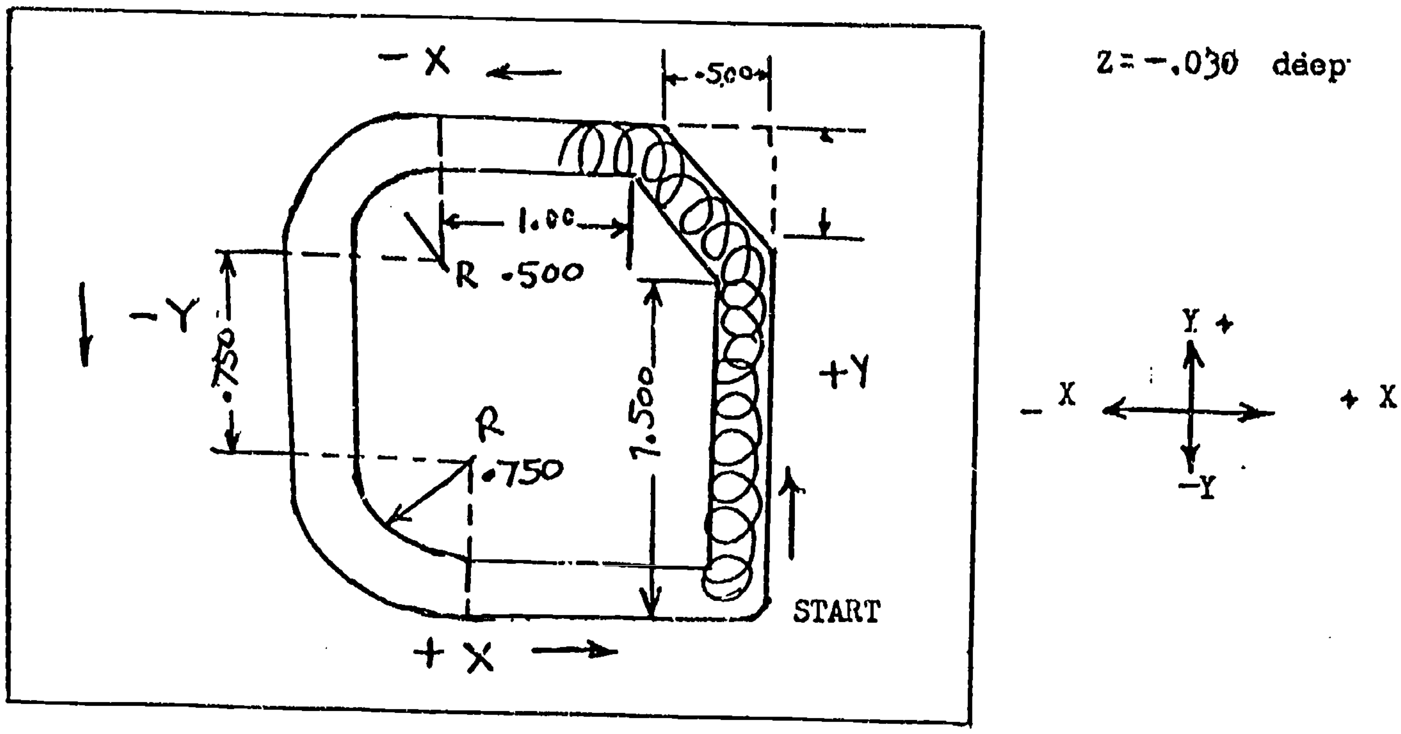
EXERCISE NO 1#

Name ______________________

Date ______________________

Score ______________________

MACHINE THIS DESIGN IN A BLOCK



REMEMBER: Before you do any machining always make a rough sketch of the part that you want to machine.

CHECK : Are all the locks loose ?

Are the hold downs far enough away from the cutter ?

Do you know where the emergency button is ?

Is the spindle locked in the proper Z depth ?

Did you double check all the feeds and speeds ?

You are now ready to program in the CNC language into the controller. Start from line 000 and punch in all the commands through line 032. There is an explaination after each line so that you can visualize the path that the cutter is taking. GOOD LUCK !

THIS IS WHAT A PROGRAM SHOULD

LOOK LIKE

```
000  START INS 16
001  TD =  .250
002  FR  X Y = 16
003  FR  Z  = 8
004  SETUP → zcxyu
005  SPINDLE ON
006  GO ABS  X — .030
007  GO ABS  Y  1.500
008  GO ABS  Y  2.00
009  X  =  — .500
010  GO ABS  X  = — 1.500
011  ZERO AT
012  Y  = 1.500
013  ZERO  X
014  GO ANGLE a 180
015  GO ABS  Y  = — .750
016  ZERO AT
017  X  =  .250
018  ZERO  Y
019  GO ANGLE a 270
020  REF COODS
021  GO  X = 0
022  Z → C
023  SPINDLE OFF
024  END NEWPART
```

ON THE NEXT FEW PAGES YOU WILL LEARN WHICH KEYS TO PRESS TO PUT THIS INFORMATION INTO THE CONTROLLER.

PROGRAMING THE CONTROLLER

A. CLAMP WORK ON THE BED USING THE HOLD DOWNS

B. MAKE SURE THE WORK PIECE IS LARGE ENOUGH FOR THE JOB.

C. LOOSEN ALL LOCKS ON THE SPINDLE AND BED.

D. PLACE $\frac{1}{4}$" .250 MILLING CUTTER IN THE SPINDLE CHUCK.

E. TURN ON THE MACHINE IN THE UPPER LEFT HAND CORNER.

READY ? WILL COME ON THE SCREEN. PRESS YES

INITALIZATING WILL COME ON THE SCREEN TO TAKE OUT BACKLASH.

MODE ? COMES ON THE SCREEN. PRESS PROGRAM ENTER ON THE LEFT SIDE OF THE CONTROLLER PRESS START BUTTON. THIS WILL BRING UP 000 START INS ANSWER YES IF IN INCHES OR NO IF YOUR WANT MM. IN THIS EXERCISE YOU WILL PUSH YES AND 16

press NEXT

001 PRESS TOOL DIAMETER TD ? WILL COME UP ON THE SCREEN PUNCH IN .250 THIS IS THE SIZE OF THE CUTTER.

press NEXT

002 PRESS FEED RATE FR AXIS ? WILL COME ON THE SCREEN. PRESS X & Y & 16 FOR FEED RATE.

PRESS FEED RATE 003 FR AXIS ? WILL APEAR ON THE SCREEN AGAIN . THIS TIME PRESS Z & 8 FOR FEED RATE.

PRESS NEXT

004 PRESS SET UP SET UP → zcxyu WILL

APPEAR ON THE SCREEN. PRESS NEXT

005 PRESS SHIFT PRESS SPINDLE ON/OFF (twice)

PRESS NEXT

006 PRESS GO ABS PRESS Z - .030 (depth)

PRESS NEXT

007 PRESS GO ABS PRESS Y 1.500 THIS WILL

MOVE THE CUTTER FROM HOME POSITION BACK $1\frac{1}{2}$" .

PRESS NEXT

008 PRESS GO ABS PRESS Y 2.00

PRESS NEXT

009 PRESS X — .500 THIS MAKES THE CORNER

PRESS NEXT

010 PRESS GO ABS PRESS X —1.50 THIS

WILL MOVE THE CUTTER ALONG THE BACK OF THE WORK TO THE LEFT CORNER.

PRESS NEXT

011 PRESS ZERO AT THIS WILL PUT AN IMAGINARY

RADIUS POINT INTO THE CONTROLLER TO SWING THE CORNER CURVE.

PRESS NEXT

012 PRESS Y 1.500 THIS WILL MOVE THE CUTTER DOWN THE LEFT SIDE TOWARD THE OPERATER.

PRESS NEXT

013 PRESS ZERO AT AXIS ? WILL COME ON THE SCREEN PRESS X

PRESS NEXT

014 PRESS GO ABS PRESS ANGLE a 180.

THIS WILL TURN THE CORNER IN THE UPPER LEFT HAND CORNER.

PRESS NEXT

015 PRESS GO ABS Y —.750

PRESS NEXT

016 PRESS ZERO AT THIS MAKES AN IMAGINARY POINT FOR THE RADIUS OF THE LOWER LEFT CORNER.

PRESS NEXT

017 PRESS X PRESS .250

PRESS NEXT

018 PRESS ZERO COODS AXIS ? WILL COME ON THE SCREEN PRESS Y

PRESS NEXT

019 PRESS GO ABS PRESS ANGLE a 270

THIS WILL TURN THE CORNER IN THE LOWER LEFT HAND CORNER

PRESS NEXT

020 PRESS SHIFT PRESS REF COODS

PRESS NEXT

021 PRESS GO ABS PRESS X = 0

PRESS NEXT

022 PRESS Z → Z CLEAR THIS BRINGS CUTTER UP.

PRESS NEXT

023 PRESS SHIFT PRESS SPINDLE ON / OFF

THIS STOPS THE CUTTER FROM ROTATING. YOU ARE NOW AT HOME POSITION.

PRESS NEXT

024 PRESS END END NEW REF ? COMES

ON THE SCREEN. PRESS NO. END NEW PART ? WILL COME

ON THE SCREEN PRESS YES

PRESS NEXT

NOW PRESS PROGRAM RUN (twice for red light) THE CONTROLLER WILL

ASK YOU NON STOP ? PRESS YES NOW ON THE

SCREEN YOU WILL SEE SET UP ⟶ zcxyu

NOW FIND THE TWO SETS OF JOG KEYS ON THE CONTROLLER. PRESS THE JOG KEY FOR Z UNTIL THE CUTTER IS LOWERED TO ABOUT 1 ABOVE THE WORK. BRING THE QUILL DOWN UNTIL IT JUST TOUCHES THE SURFSCE OF THE MATERIAL. NOW LOCK THE QUILL DOWN. PRESS THE SET UP REF KEY. THIS WILL CAPITALIZE THE Z ON THE SCREEN. PRESS THE NO KEY AND THE c WILL APPEAR ON THE SCREEN. MOVE THE JOG KEY SO THE Z GOES UP ABOUT $\frac{1}{2}$" ABOVE THE MATERIAL. NOW PRESS THE SET UP REF KEY. THIS WILL CAPITALIZE THE C AND GIVE THE CUTTER CLEARANCE. PRESS THE NO KEY THE SMALL x WILL APPEAR. WITH THE JOG KEY MOVE THE X AXIS OVER TO YOUR STARTING POSITION AND PRESS THE SET UP REF KEY. THIS WILL CAPITALIZE THE X. PRESS THE NO KEY AND THE Y WILL BE NEXT IN LINE. WITH THE JOG KEY MOVE THE Y AXIS BACK TO THE STARTING POSITION. PRESS THE SET UP REF KEY TO CAPITALIZE THE Y AXIS. YOU DO NOT HAVE TO USE THE u SO DISREGUARD. PRESS NEXT THE SPINDLE SHOULD COME ON AND START DOWN INTO THE WORK PIECE AND MACHINE THE PART.

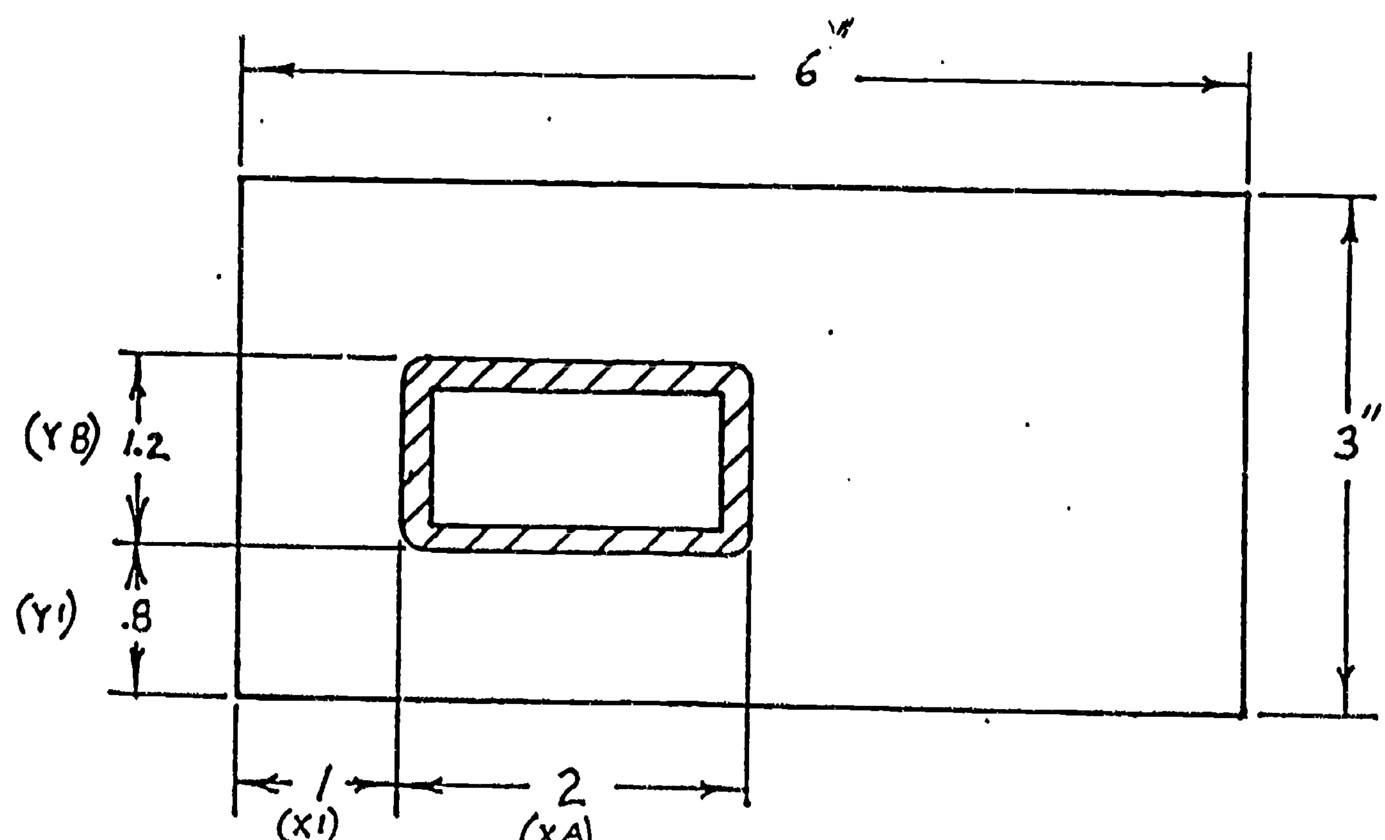
EXERCISE NO 2#

Name ______________ ________

Date __________ Score __________

RECTANGLE FRAME - SAMPLE PROGRAM

```
001  START INS  01
002  TD = 0.2500
003  FR XYZ = 15
004  SETUP —>zcxyu
005  SPINDLE ON
006  FRAM F I Z% 050      (Rectangle frame function.  F means finish cut.
                           I means cut will be inside of dimension given.
                           Z% 050 means that the depth will be no more
                           than 50% of tool diameter.)
007    ZH = 0.0000        (Z reference offset.  Should be 0 for programs.)
008    Zd = 0.0400        (Zd = depth that cutter will go.)
009    X1 - 1.0000  \
010    Y1 = 0.8000  /     (Location of lower left corner of frame cut.)
011    XA = 2.0000        (Length of rectangle along X axis.)
012    YB = 1.2000        (Length of rectangle along Y axis.)
013  SPINDLE OFF
014  END NEWPART
```



WITH THE INFORMATION ABOVE PROGRAM THIS IN THE CONTROLLER AND RUN IT

Name ____________ ____________

EXERCISE NO 3#

Date ____________ Score ____________

RECTANGLE FRAME

WRITE THE CORRECT PROGRAM BELOW WITH THE FOLLOWING INFORMATION

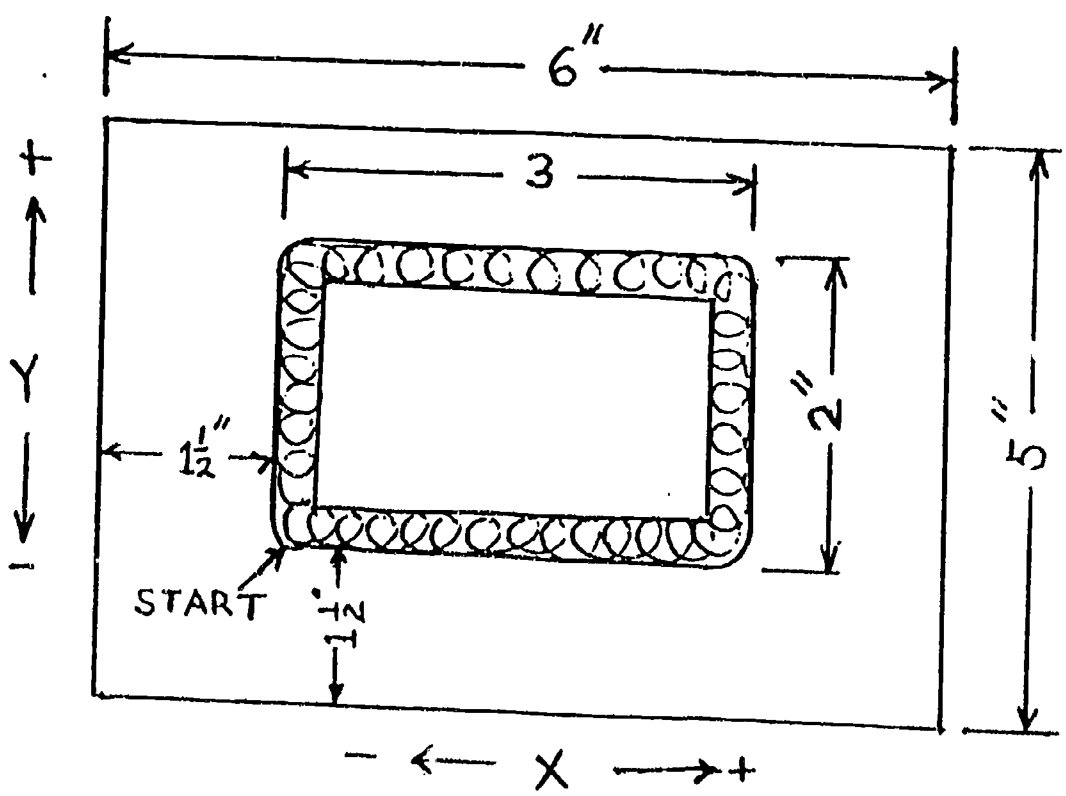
DEPTH = .050 Z

LOWER LEFT CORNER X= $1\frac{1}{2}''$ and Y = $1\frac{1}{2}''$ START

LENGTH OF X = 3"

LENGTH OF Y = 2"

000	____________	009	____________
001	____________	010	____________
002	____________	011	____________
003	____________	012	____________
004	____________	013	____________
005	____________	014	____________
006	____________	015	____________
007	____________	016	____________
008	____________	017	____________



POST-TEST

NAME ______________________

DATE __________ SCORE ________

AXIS and NC and CNC ABSOLUTE and INCREMENTAL

1. What does NC stand for and where is it used ? (brief statement)

2. What does CNC stand for and how is it different when programing then NC ?

3. List three (3) advantages of CNC over NC machining ?

a. ______________________________

B. ______________________________

C. ______________________________

4. What effect does the quadrant have when programing the axis direction ?

5. With the right thumb rule :

A. The thumb is which axis ________ and points in which direction ?

B. The first finger points to the ________ axis .

C. The second finger points to the ________ axis.

6. What is meant by absolute value or absolute dimensioning ?

7. What is meant by incremental value or dimensioning ?

8. What is the one BIG error of using relative values when programing ?

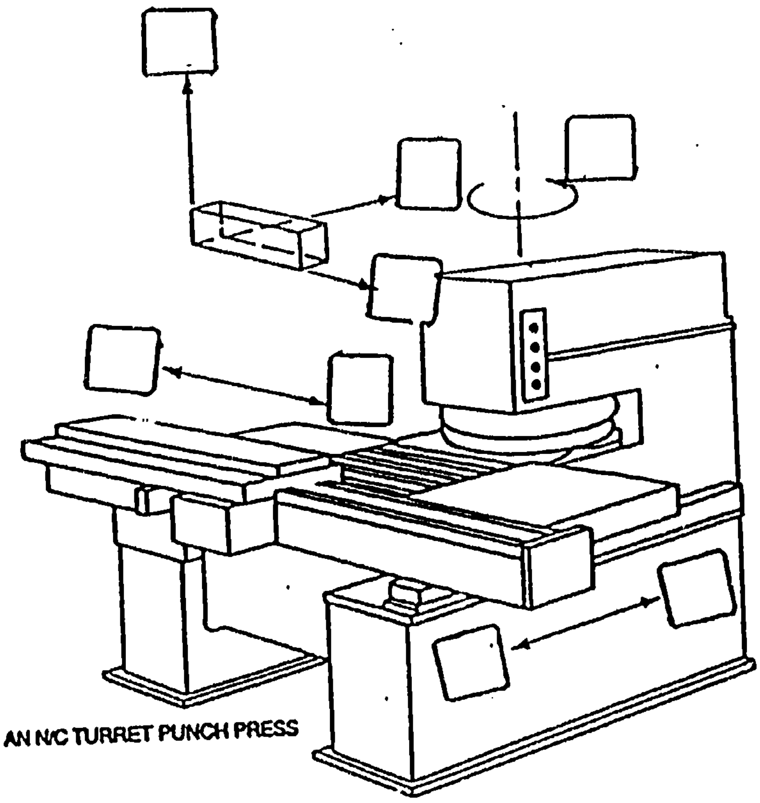
POST-TEST AXIS DIRECTION

In each box place the axis and a (+) or(-)

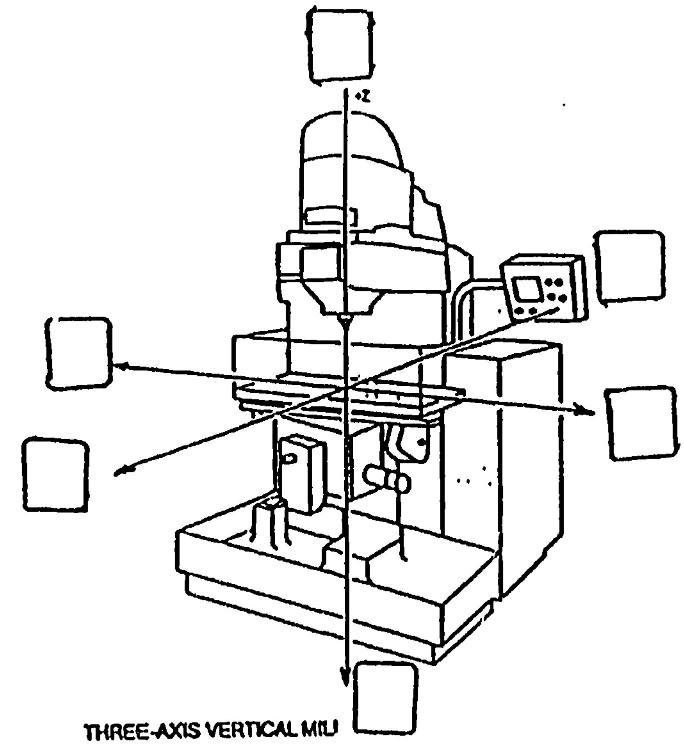
NAME________________

DATE________________

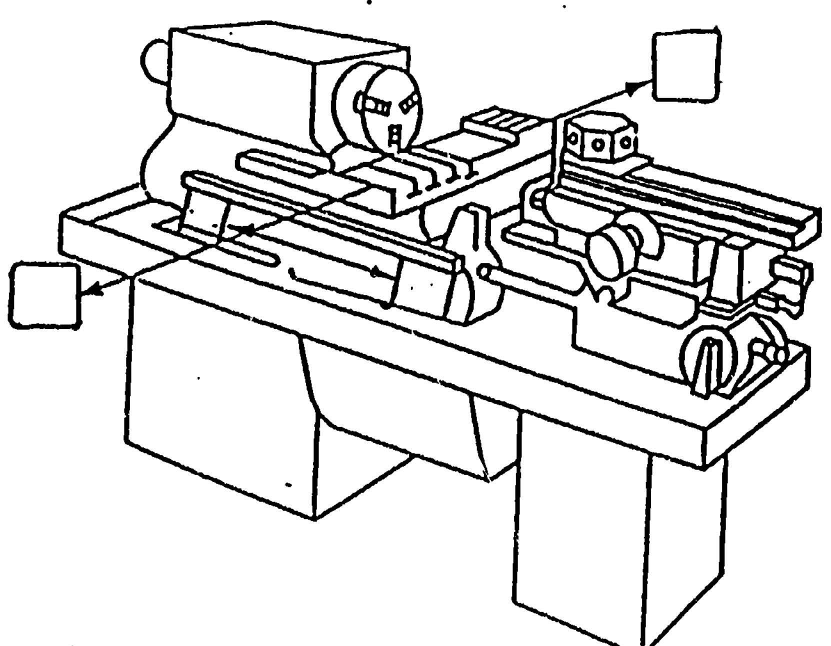
SCORE__________



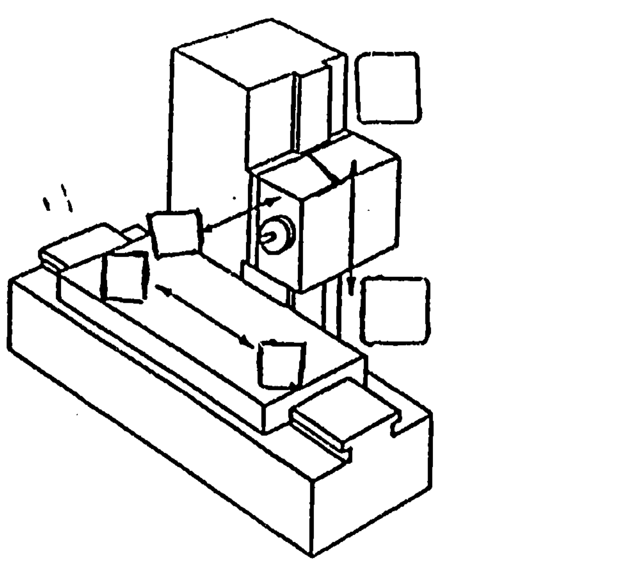
AN N/C TURRET PUNCH PRESS



THREE-AXIS VERTICAL MILL



A N/C TURRET LATHE



A HORIZONTAL N/C MACHINE