NUMERICAL CONTROL

http://www.toolingu.com/definition-300200-12690-tool-offset.html

NC &CNC

- Numeric Control (NC) and Computer Numeric Control (CNC) are means by which machine centers are used to produce repeatable machining process.
- Two types are used:
  - Fixed Automation using mechanical cam
  - Flexible Automation using G Code
- The control programs use either
  - Closed loop control using feedback
  - Or Open loop control

CNC Motion Control

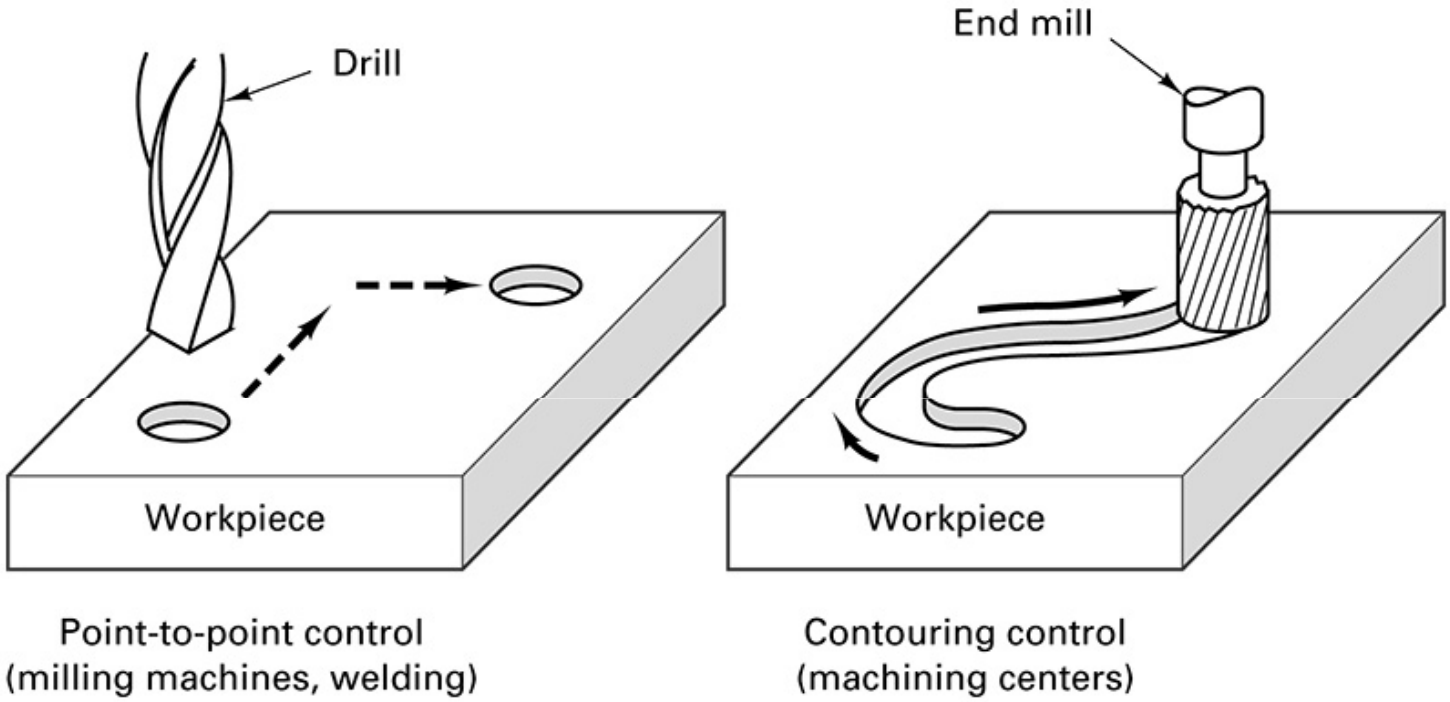


FIGURE 26-8 NC and CNC systems are subdivided into two basic categories: point-to-point controls or contouring controls.

Tool Dimensioning

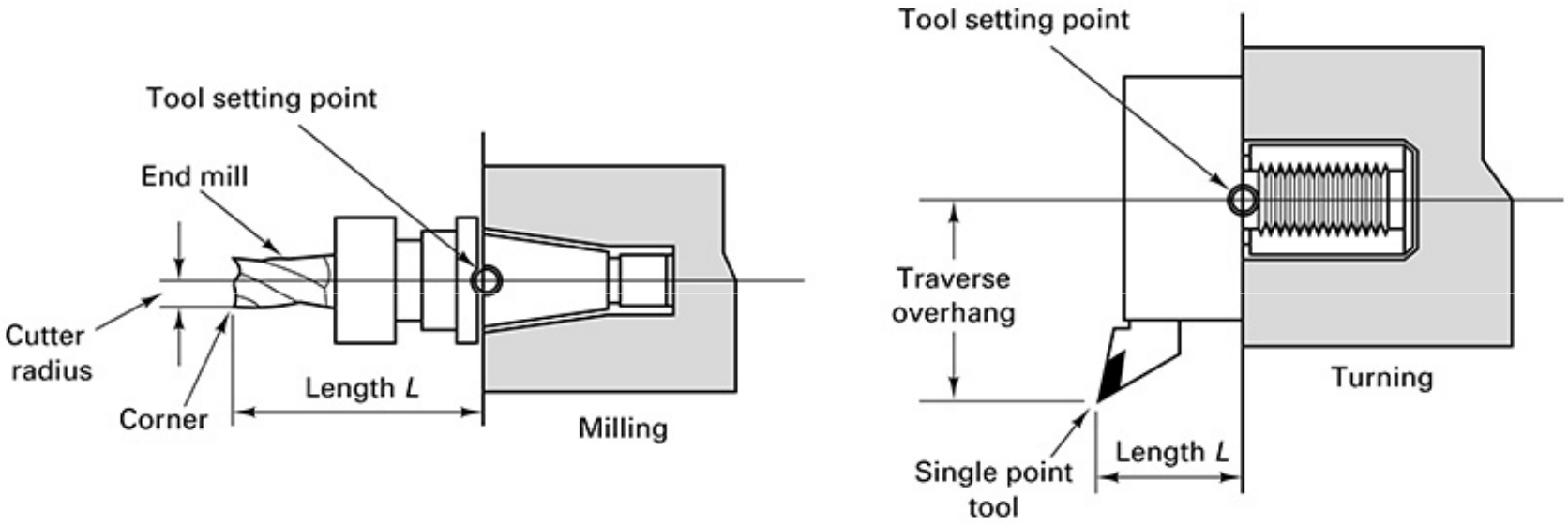


FIGURE 26-12 The location of the corner of the end mill (left) or the tip of a single-point tool (right) must be known with respect to the tool setting points so that tool dimensions are accurately set.

Motion Control

- NC machines use electric motor drives with position feedback provided by transducers.
- Older system used DC motors with analog transducers
- Newer system use AC servomotors, or stepper motors with optical encoders for better accuracy, reliability, lower power consumption and performance to weight ratios.
- Recirculating ball screws drives or linear accelerators help improve accuracy by removing backlash in the drive systems
- Canned program routines are used when repeated common features are used in the part designs

CNC PROGRAMMING

- LINE #1 = SELECT CUTTING TOOL.
- LINE #2 = TURN SPINDLE ON AND SELECT THE RPM.
- LINE #3 = RAPID TO THE STARTING POSITION OF THE PART.
- LINE #4 = TURN COOLANT ON.
- LINE #5 = CHOOSE PROPER FEED RATE AND MAKE THE CUT(S).
- LINE #6 = TURN THE SPINDLE AND COOLANT OFF.
- LINE #7 = RETURN TO CLEARANCE POSITION TO SELECT ANOTHER TOOL.

DEFINITIONS

- 1. CHARACTER : A single alphanumeric character value or the "+" and "-" sign.
- 2. WORD : A series of characters defining a single function
- 3. BLOCK : Series of words defining a single instruction.
- 4. POSITIVE SIGNS : If the value following an address letter command the plus sign need not be programmed in.If it has a minus value it must be programmed in with a minus (-) sign.
- 5. LEADING ZERO'S : If the digits proceeding a number are zero, they need not be

PROGRAM STRUCTURE

- A CNC part program consists of one or more blocks of commands.
- A block is the same as a line of text.
- Blocks shown on the CRT are always
- terminated by the “ ; “ symbol which is called an End Of Block (EOB).
- Blocks are made up of alphabetical address codes which are always an alphabetical character followed by a numeric value
- .

NC Program Language

TABLE 26-2 Definitions of Common NC Words

NC Word	Use
N	*Sequence number:* identifies the block of information
G	*Preparatory function:* requests different control functions, including preprogrammed machining routines
X, Y, Z, B	*Dimensional coordinate data:* linear and angular motion commands for the axis of the machine
F	*Feed function:* sets feed rate for this operation
S	*Speed function:* sets cutting speed for this operation
T	*Tool function:* tells the machine the location of the tool in the tool holder or tool turret
M	*Miscellaneous function:* turns coolant on or off, opens spindle, reverses spindle, tool change, etc.
EOB	*End of block:* indicates to the MCU that a full block of information has been transmitted and the block can be executed

PREPARATORY FUNCTIONS:G

- "G" makes the machine tool do specific operations, such as :
- .G00- Move the tool at rapid traverse.
- G01- Move the tool at a feedrate along a straight line.
- G02- Move the tool along an arc at a feedrate in a clockwise direction.
- G03- Move the tool along an arc at a feedrate in a counterclockwise direction.

PREPARATORY FUNCTIONS:G

- G04 00 Dwell (P) (P =seconds"."milliseconds)
- G09 00 Exact Stop, Non-Modal
- G10 00 Programmable Offset Setting
- G90-Absolute programming
- G91-Incremental programming

SPACE IDENTICATION

- Absolute zero
- Home position
- Absolute (G90) and incremental position(G91)
- Offset-a displacement in an axial tool direction
- Offset-a correction on an actual tool length

SPEED CODE

- F-feed function, specifying the feed of the cutting tool or dwell time,
- S-cutting speed
- D-acceleration/deceleration

T AND M-CODES

- T- Tool selection, to access a particular tool from a tool changer or turret
- "M" codes are effective or cause an action to occur at the end of the block
- Only one M code is allowed in each block of a program.
- .

M-CODE

- M00 Program Stop
- M01 Optional Program Stop
- M02 Program End
- M03 Spindle On, Clockwise (S) (
- M04 Spindle On, Counterclockwise (S)
- M05 Spindle Stop
- M06 Tool Change (T)
- M08 Coolant On
- M09 Coolant Off

N-CODE

- SEQUENCE NUMBERS : N1 thru N99999 in a program are only used to locate and identify a line or block and its relative position within a CNC program.
- A program can be with or without SEQUENCE NUMBERS

ORDER OF COMMANDS

- Write X first, Y next, then Z.
- G codes has to be in the beginning of a line and M codes has to be at the end.
- Only one M code may be programmed per block
- All M codes are activated after everything else on the line has been executed.

MODAL AND NON-MODAL COMMAND

- MODAL COMMANDS : Codes that are active for more than the line in which they are
- A NON-MODAL command is effective only in the calling block, and then is immediately forgotten by the control.

EXAMPLE

- N216 G03 X7900 Y2500 S716 M04

- Block 216, CCW circular interpolation, 7.5 in positive X direction,2.5 in in Y direction, 16 *10**4 RPM, start spindle in a CCW direction

Cutter Offset

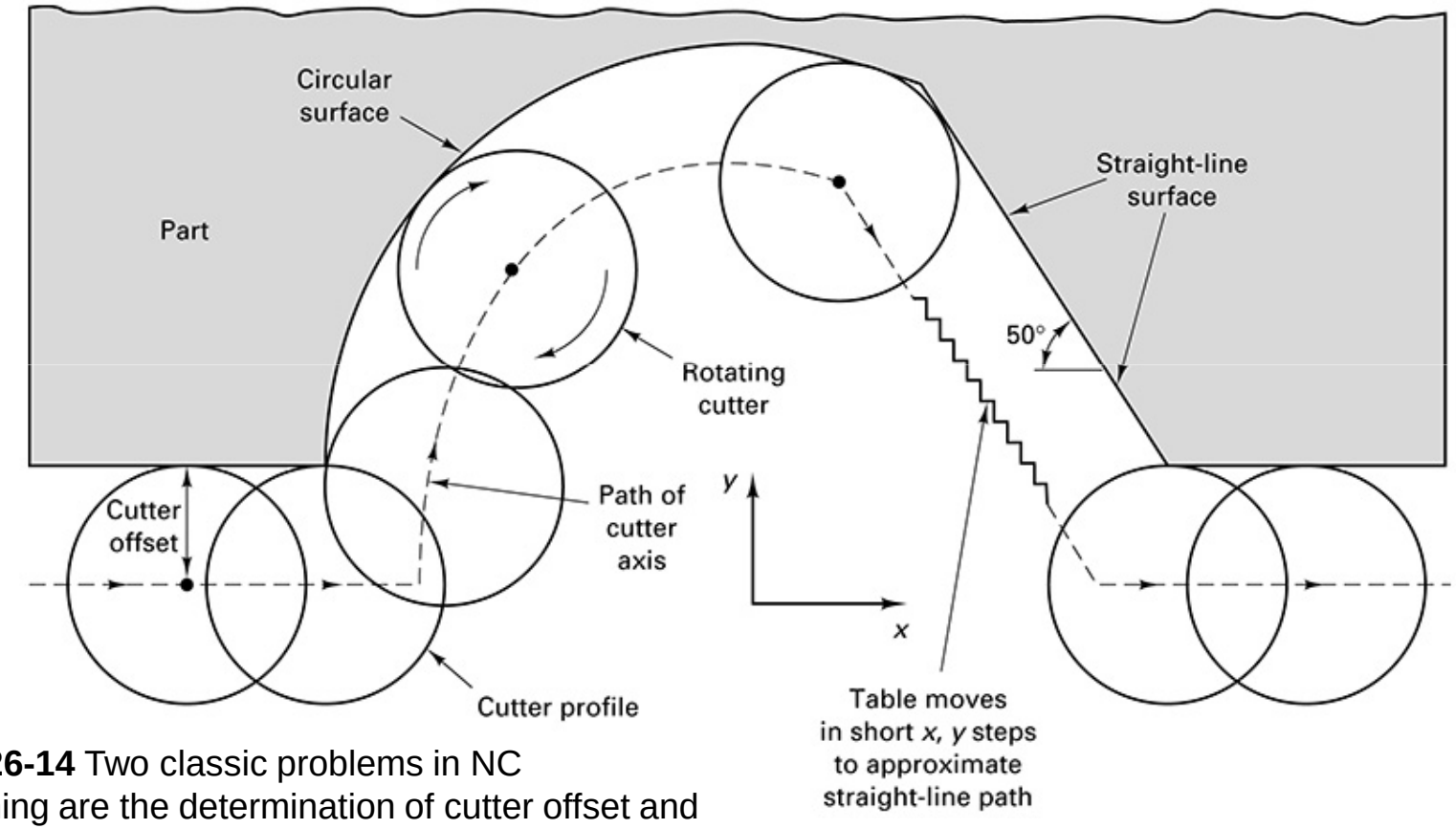


FIGURE 26-14 Two classic problems in NC programming are the determination of cutter offset and interpolation of cutter parts.

Example of Part Programming

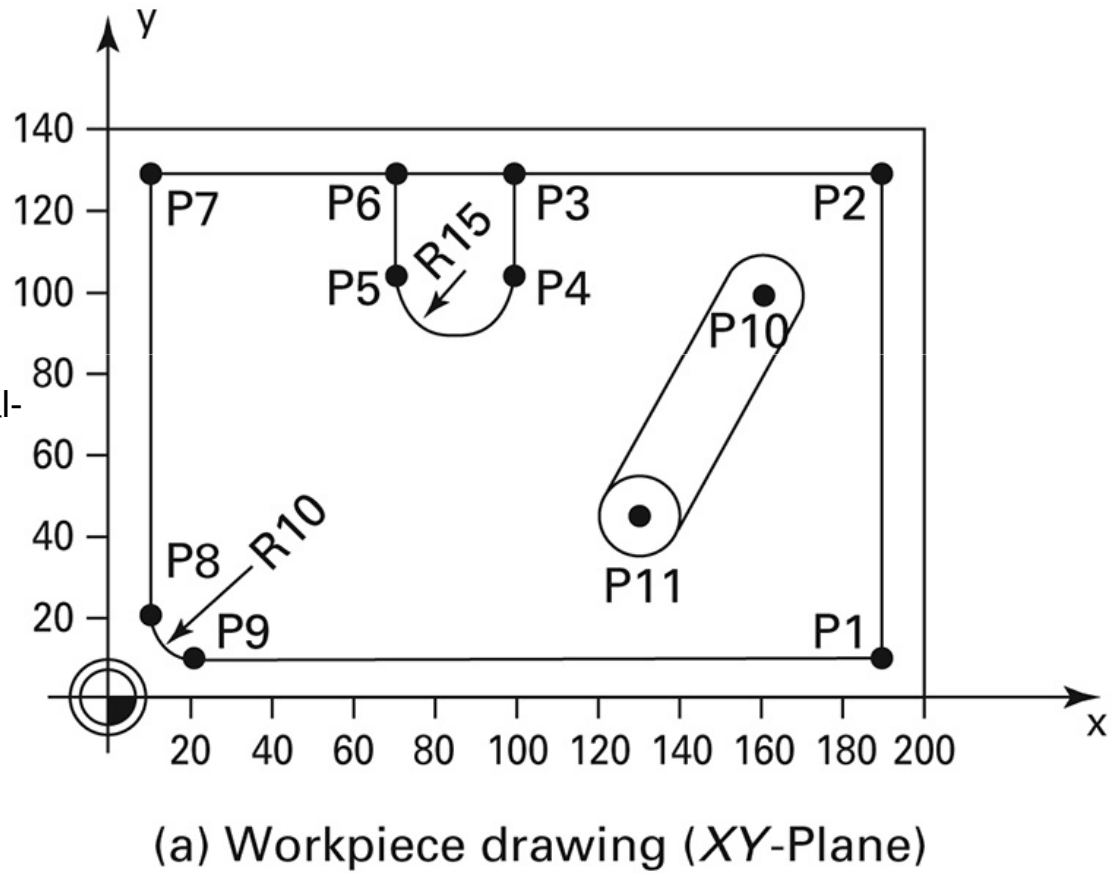


FIGURE 26-13 Example of programming a part in a vertical-spindle NC machine.

Example of Part Programming

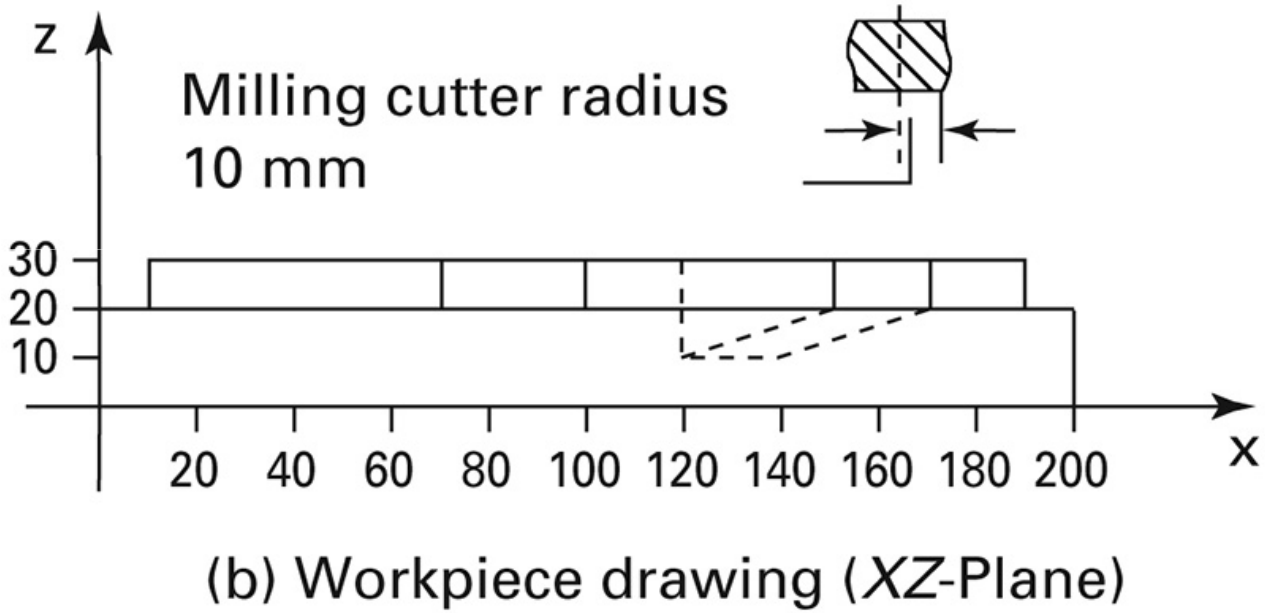


FIGURE 26-13 Example of programming a part in a vertical-spindle NC machine.

Example of Part Programming

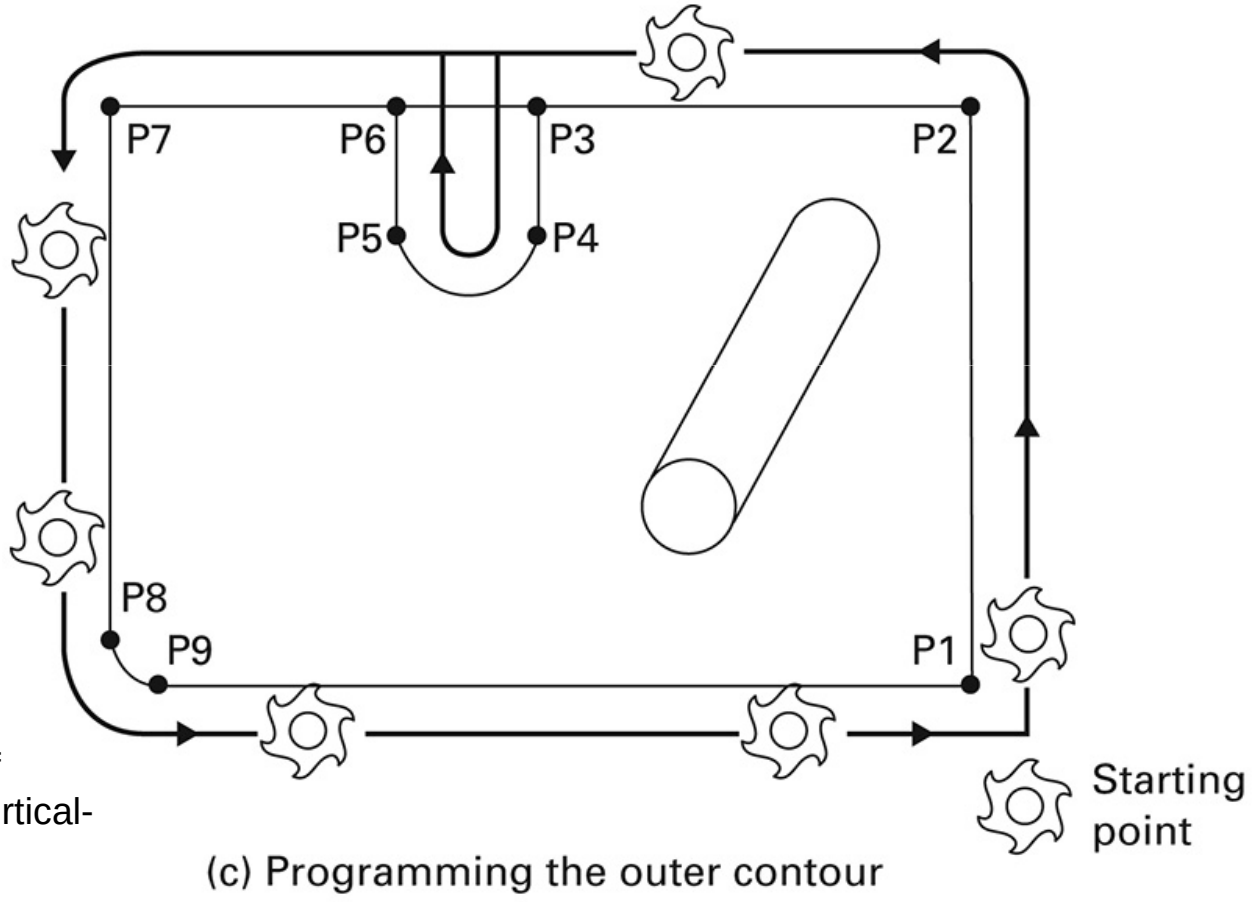
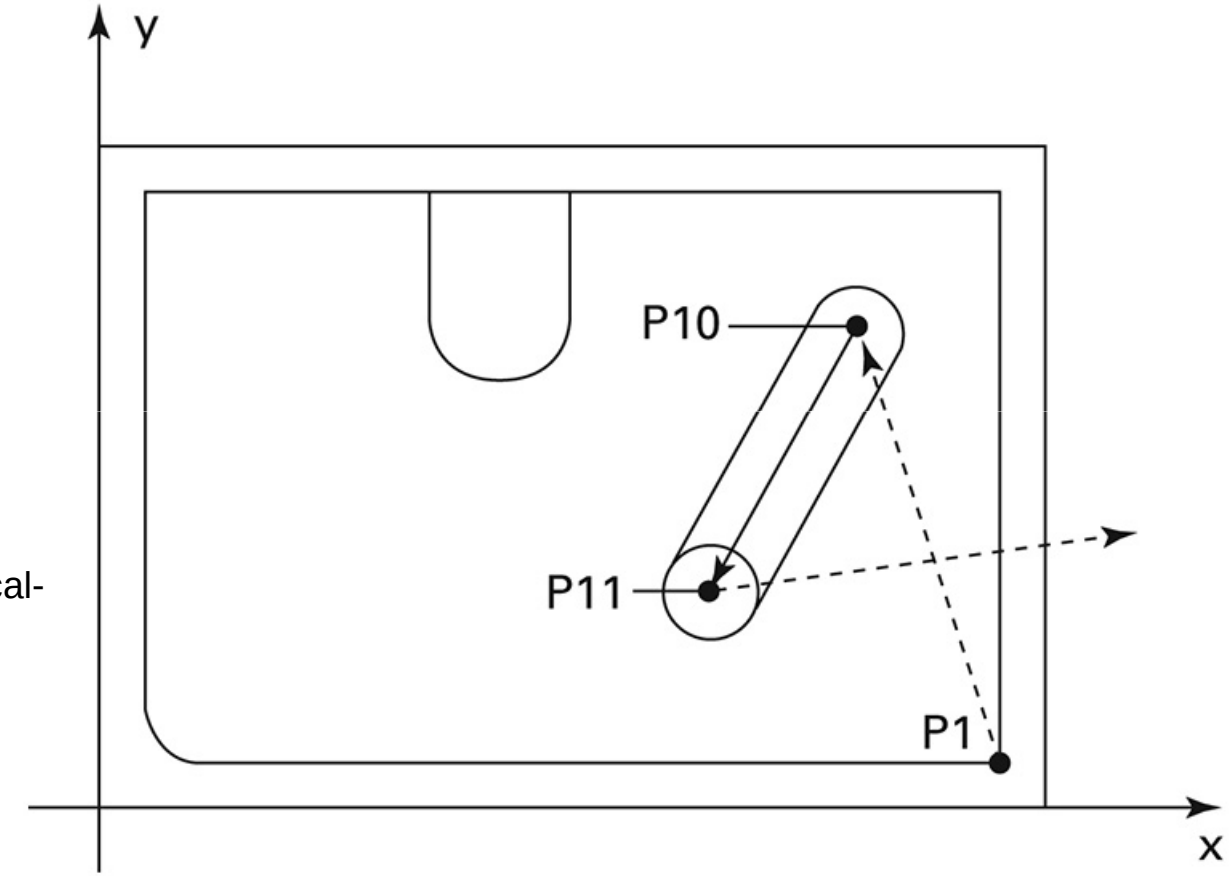


FIGURE 26-13 Example of programming a part in a vertical-spindle NC machine.

Example of Part Programming



(d) Programming the slot

FIGURE 26-13 Example of programming a part in a vertical-spindle NC machine.

Example Code for Part in Figure

	P1	P2	P3	P4	P5	P6	P7	P8	P9	P10	P11
x	190	190	100	100	70	70	10	10	20	160	130
y	10	130	130	105	105	130	130	20	10	100	45
z	20	20	20	20	20	20	20	20	20	20	10

- G01 Y130 F200 — Straight line from starting point to P2
- G01 X100 — Straight line from P2 to P3
- G01 X105 F150 — Straight line from P3 to P4
- G02 X70 Y105 R15 — Radial arc, clockwise, with 15 radius
- G01 Y130 F200 — Straight line from P5 to P6
- G01 X10 — Straight line from P6 to P7
- G01 Y20 — Straight line from P7 to P8
- G03 X20 Y10 R10 F150 — Radial arc, counterclockwise with 10 radius
- G01 X190 F200 — Straight line from P9 to P1
- G00 X160 X100 — Rapid traverse to point P10
- G01 Z20 F150 — Down feed at point P10
- G01 X130 Y45 Z10 — Straight line from P10 to P11
- G01 Z35 F200 — Retraction from workpiece
- G00 X300 Y300 — Rapid traverse away from workpiece

ACTUAL PROGRAM

- Programs must begin and end with a percent (%) sign.
- The next line in a program must have a program number beginning
- with the letter O (not zero) and then the number that defines that program.
- The % sign will "not" be seen on the control, but they must
- be in the program when you load a program into the control.

ACTUAL PROGRAM

- This program will drill four holes and mill a two-inch hole in a four-inch square plate with X and Y zero at the center

ACTUAL PROGRAM

- G90 G54 G00 X-2.35 Y2.35 S1604 M03 ;
- :ABS POSIT, WORK OFFSET#, RAPID X Y, SPINDLE ON CW

- G43 H01 Z0.1 M08 ; :
- TOOL LENGTH COMP #2, Z POSITION, COOLANT ON

- G01 Z-0.625 F50. ;
- :FAST FEED TO DEPTH

- G41 Y2. D02 F16. ; :
- CUTTER COMP. LEFT OF LINE WITH DIA. COMP D02

ACTUAL PROGRAM

- X2.0 ; :CUT A 4.0 IN. SQUARE
- Y-2.0 ; :" " "
- X-2.0 ; :" " "
- Y2.25 ; :" " "

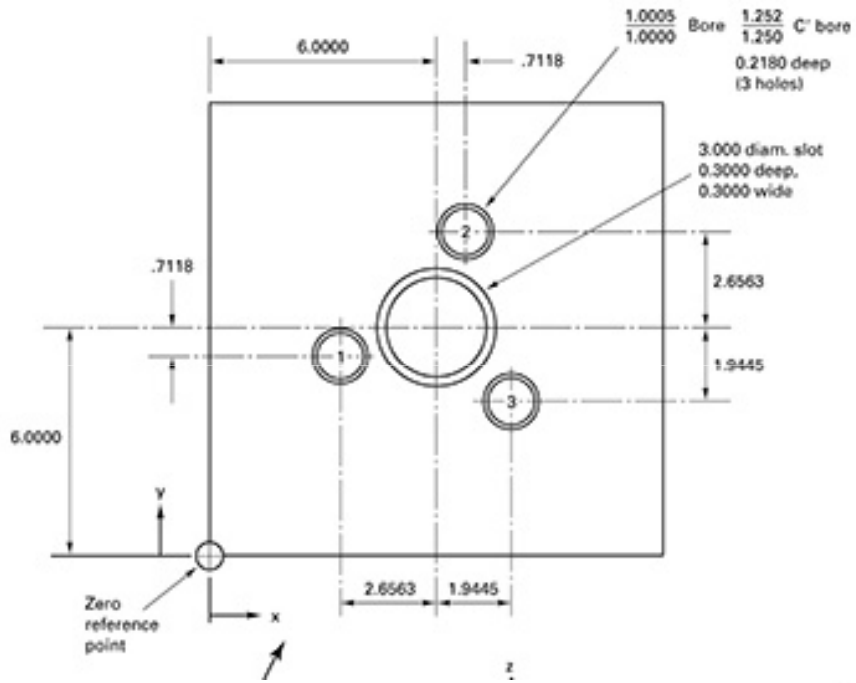
- G40 X-2.3 Y2.3 ; :
- G40 CANCELS CUTTER COMP MOVING AWAY FROM PART

- G00 Z1. M09 ; :
- RAPID Z1., COOLANT OFF

- G28 G91 Z0. M05 ; :
- RETURN Z TO MACHINE ZERO, SPINDLE OFF

- M00 (CHECK PART) ; :
- PROGRAM STOP COMMAND TO PERFORM A TASK

Motion Control in NC Machines



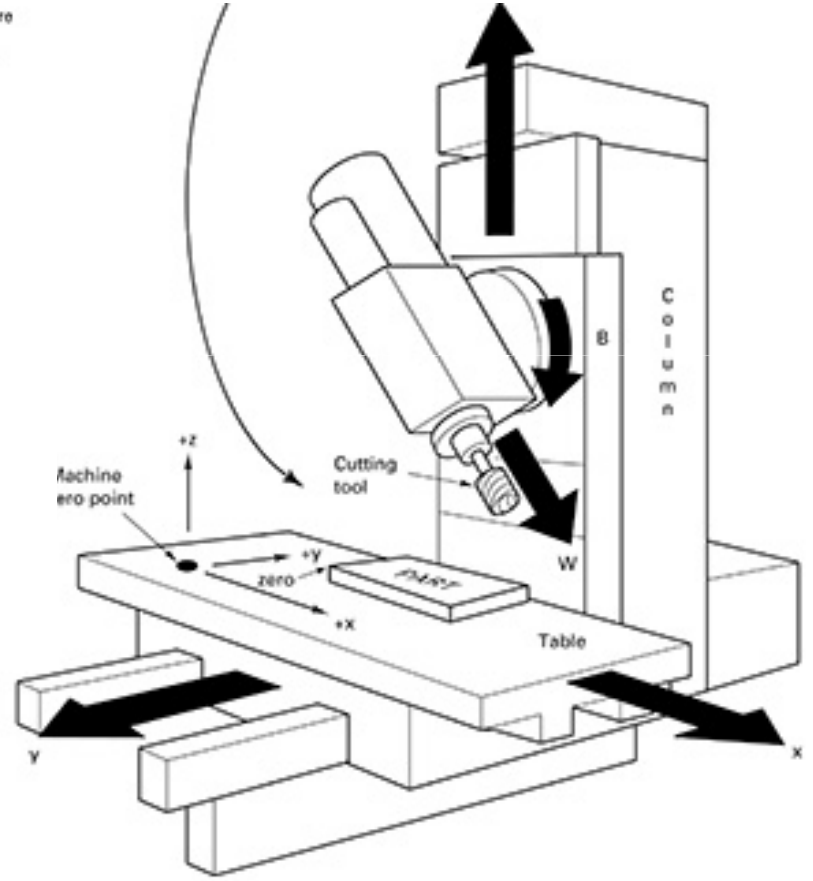


FIGURE 26-7 The part (above) to be machined on the NC machine (below) has a zero reference point. The machine also has a zero reference point.