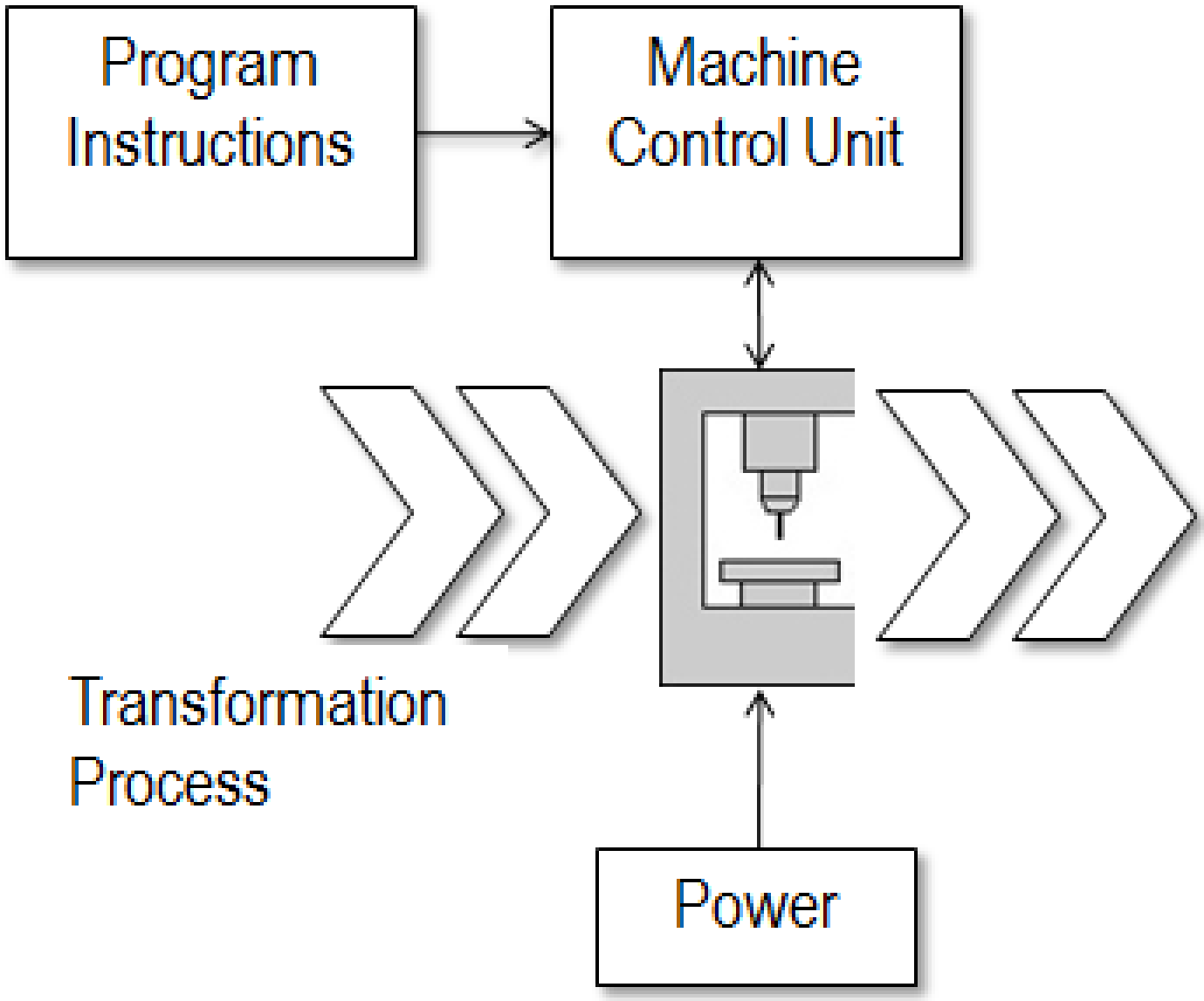
NUMERICAL CONTROL

DEFINITION

Programmable automation in which the mechanical actions of a 'machine tool' are controlled by a program containing coded alphanumeric data that represents relative positions between a work head (e.g., cutting tool) and a work part

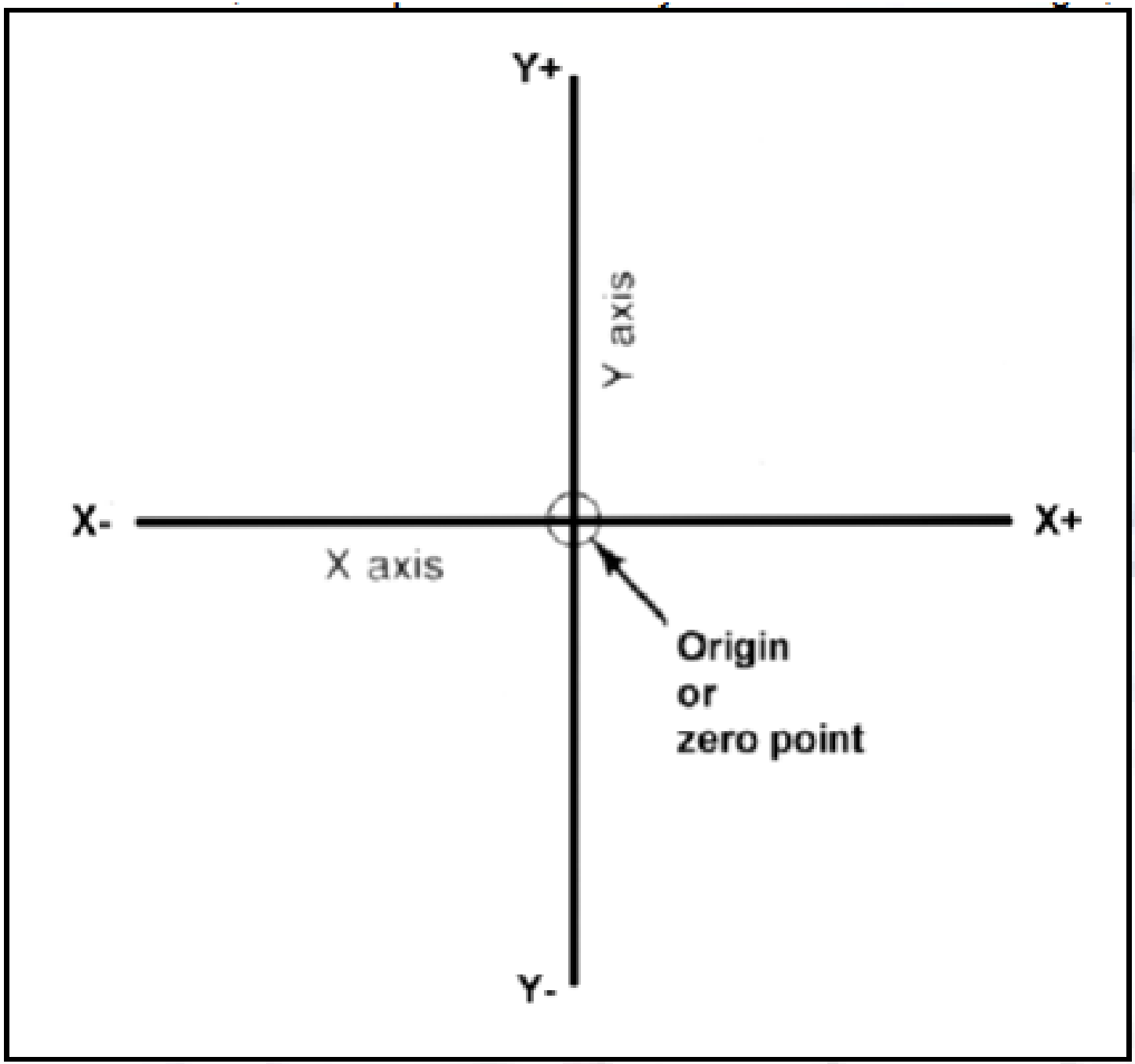
THREE BASIC COMPONENTS OF NC

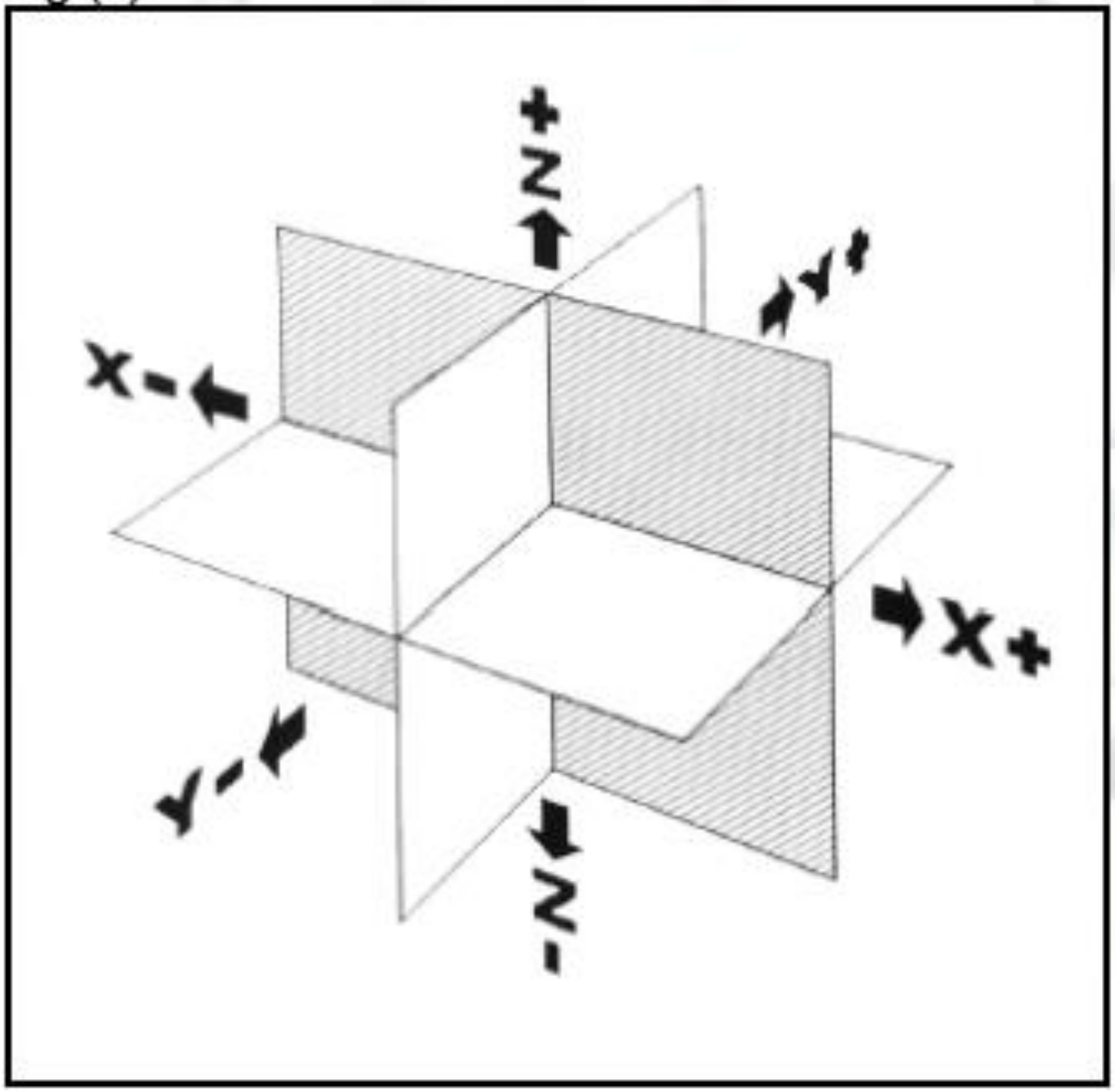
1. INPUT MEDIUM
2. MACHINE CONTROL UNIT
3. MACHINE TOOL

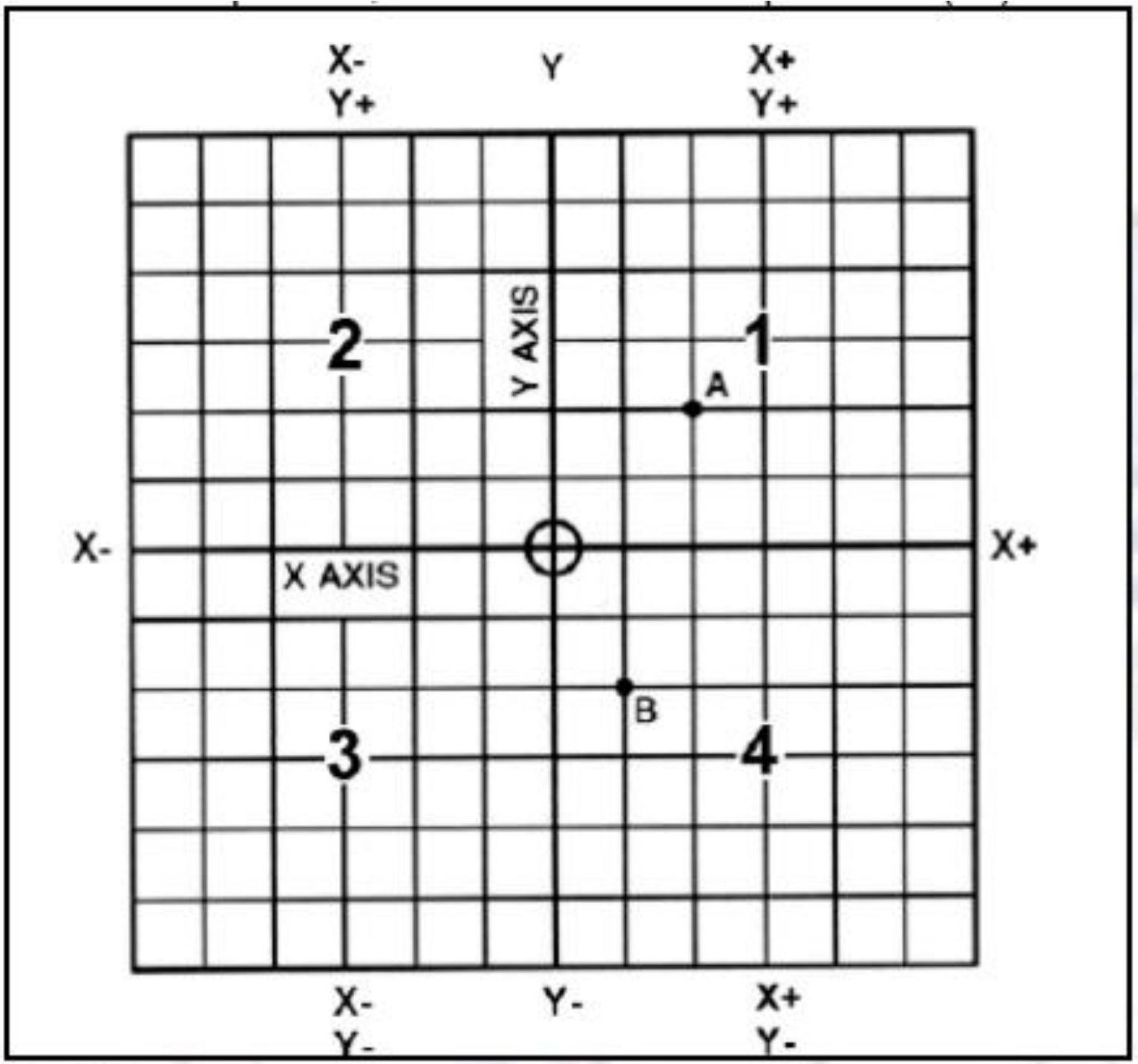

Program Instructions
Machine Control Unit
Transformation Process
Power

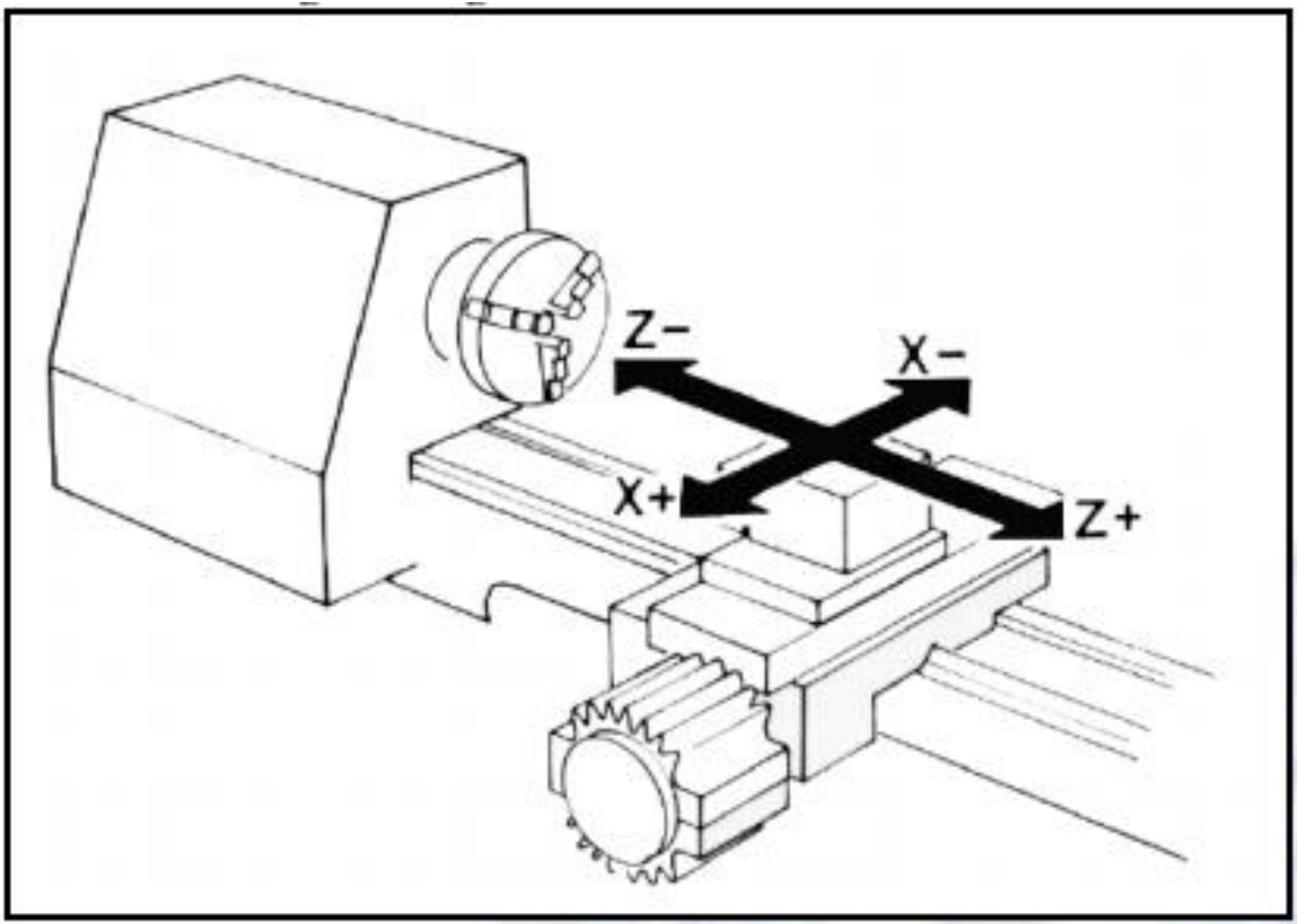
Principle

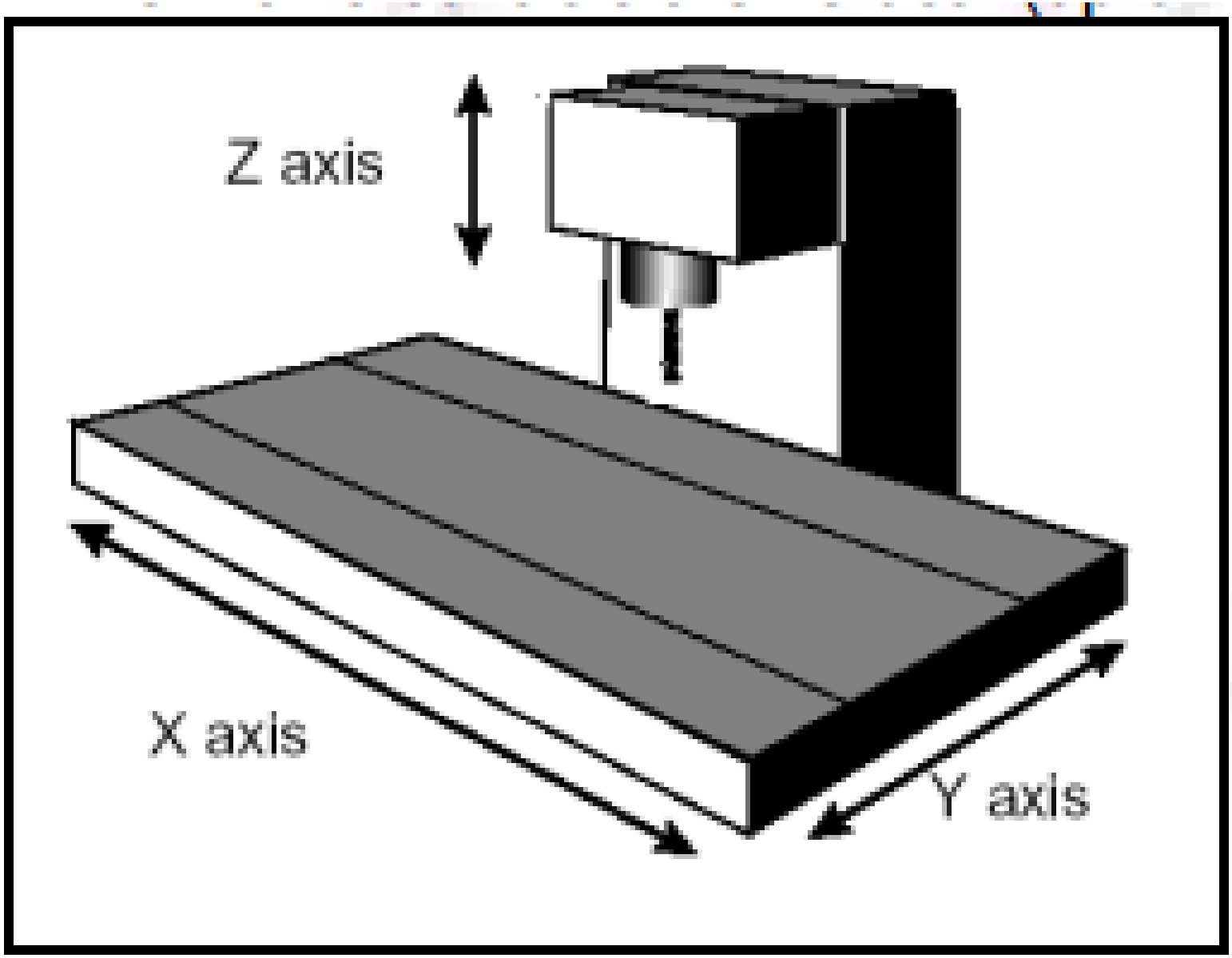
A system in which actions are controlled by direct insertion of numerical data.

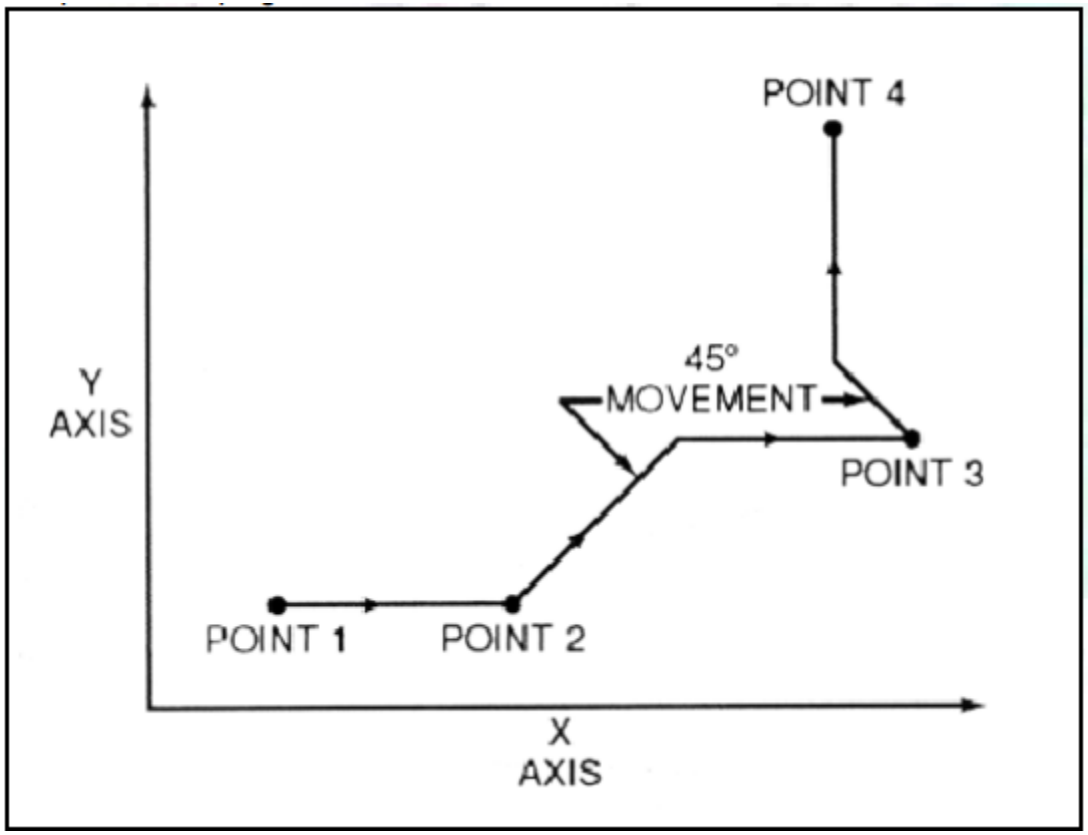

Y+
Y axis
X-
X+
X axis
Origin
or
zero point
Y-


+Z
+Y
X-
X+
Y-
-Z


X-
Y+
Y
X+
Y+
2
Y AXIS
1
A
X-
X AXIS
X+
B
3
4
X-
Y-
Y-
X+
Y-


Z–
X–
X+
Z+

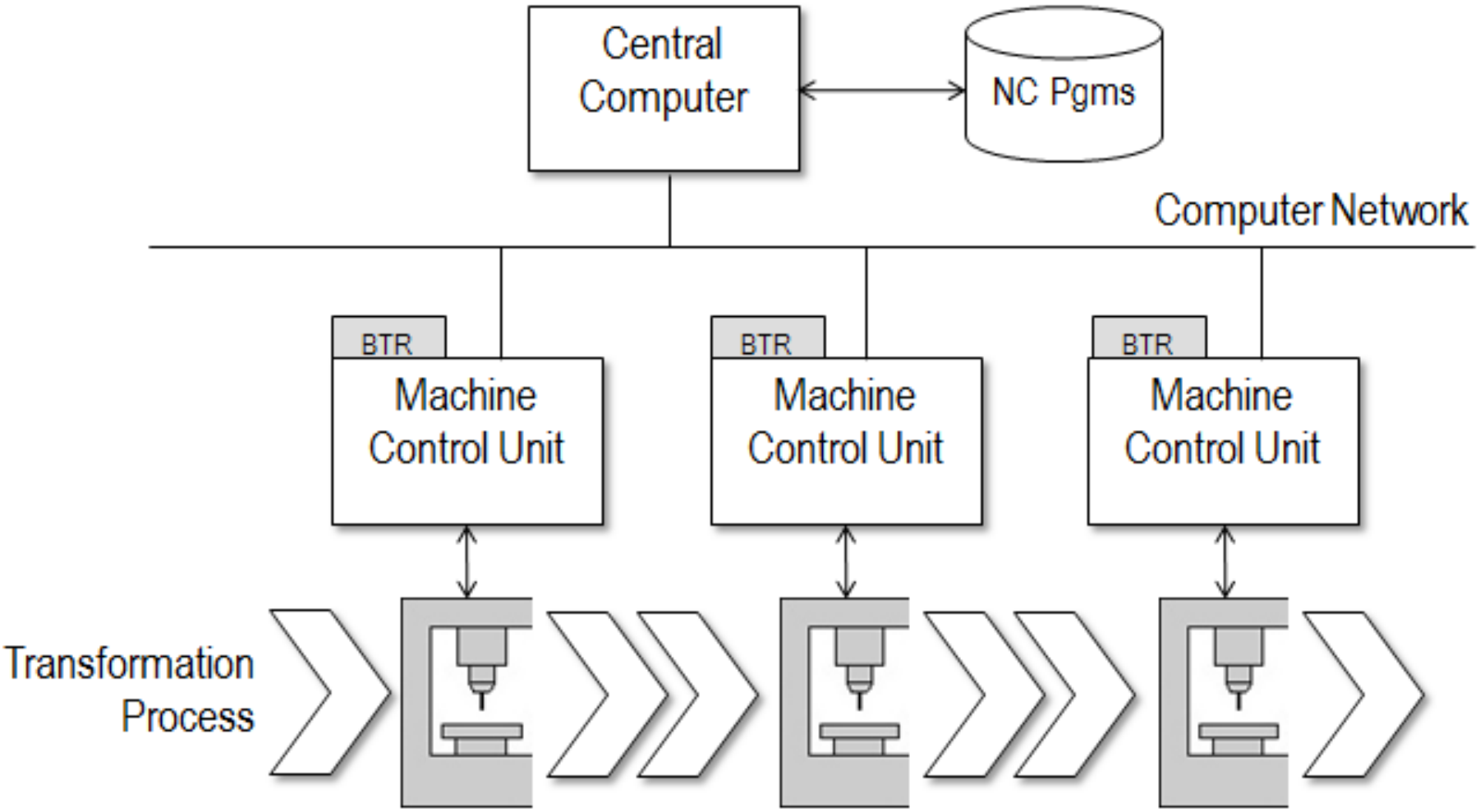

Z axis
X axis
Y axis


POINT 4
45°
MOVEMENT
Y
AXIS
POINT 3
POINT 1
POINT 2
X
AXIS

COMPUTER NUMERICAL CONTROL (CNC)

- Storage of more than one part program
- Various forms of program input
- Program editing at the machine tool
- Fixed cycles and programming subroutines
- Interpolation
- Acceleration and deceleration computations
- Communications interface
- Diagnostics

DIRECT NUMERICAL CONTROL (DNC)



NC Position Movement

Incremental

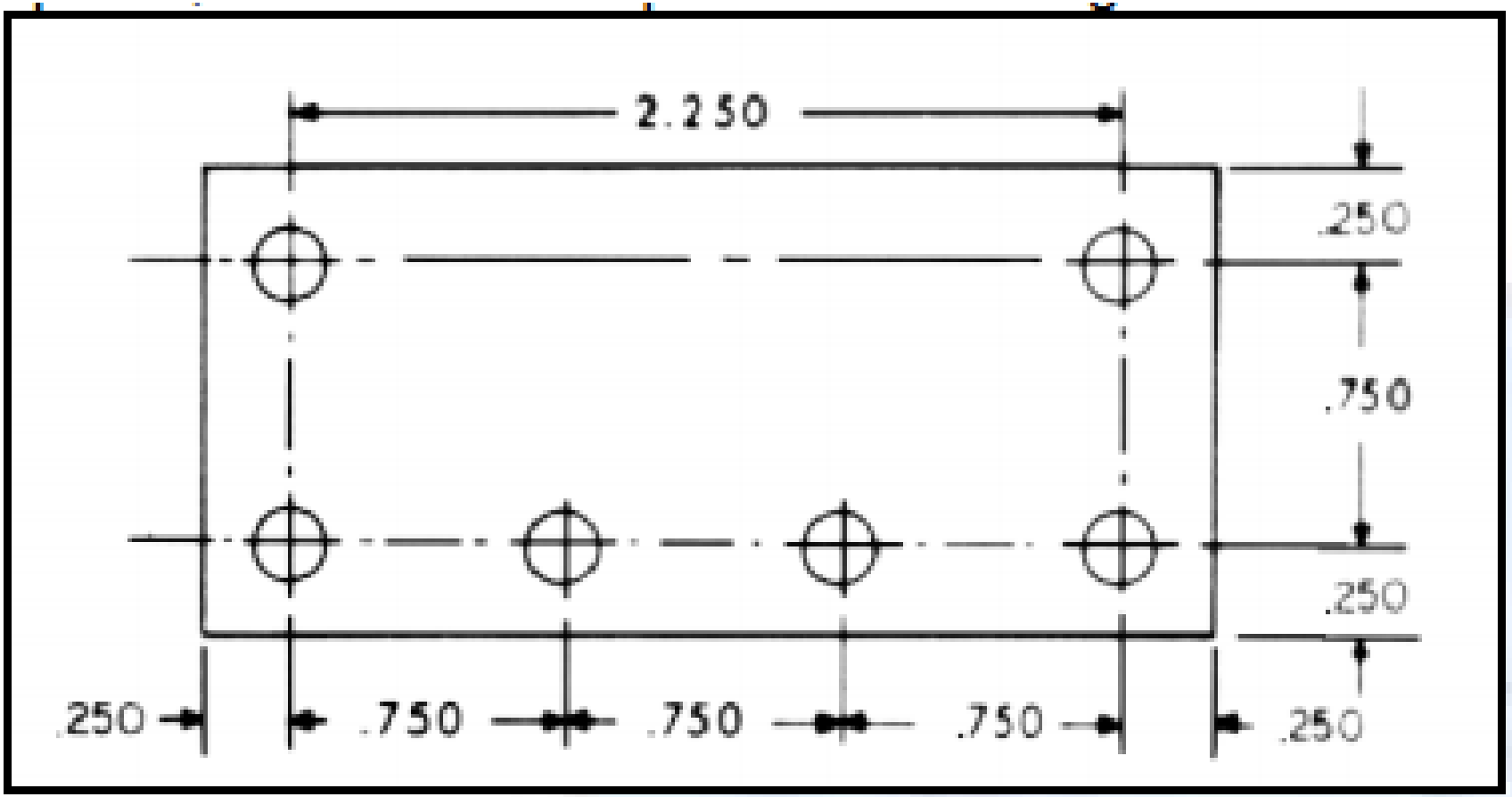
> Taking the “Last” position as the zero position.

Absolute

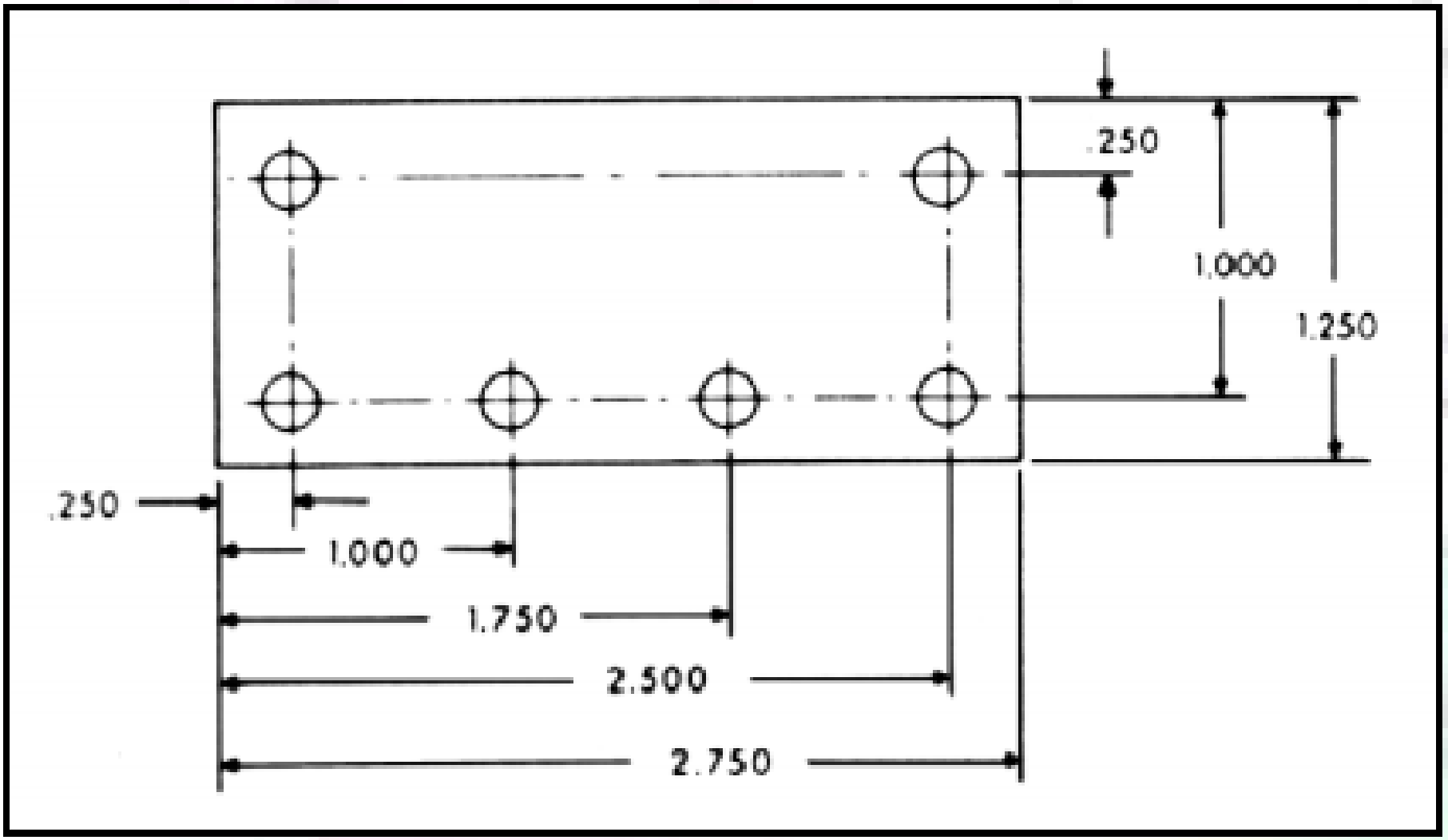
> Locations on Part – Fixed Reference Frame with

Home position for reference.

REFERENCE POSTIONING SYSTEMS



INCREMENTAL



ABSOLUTE

Degree of Motion Control

Point-to-Point (PTP)

> Good for holes & slots

> Position tool over point.

Contouring

> Complex curved surfaces

> Computers needed for complex calculations

> Motion control to motors: varying voltages to DC servo motors.

Motion Control Systems

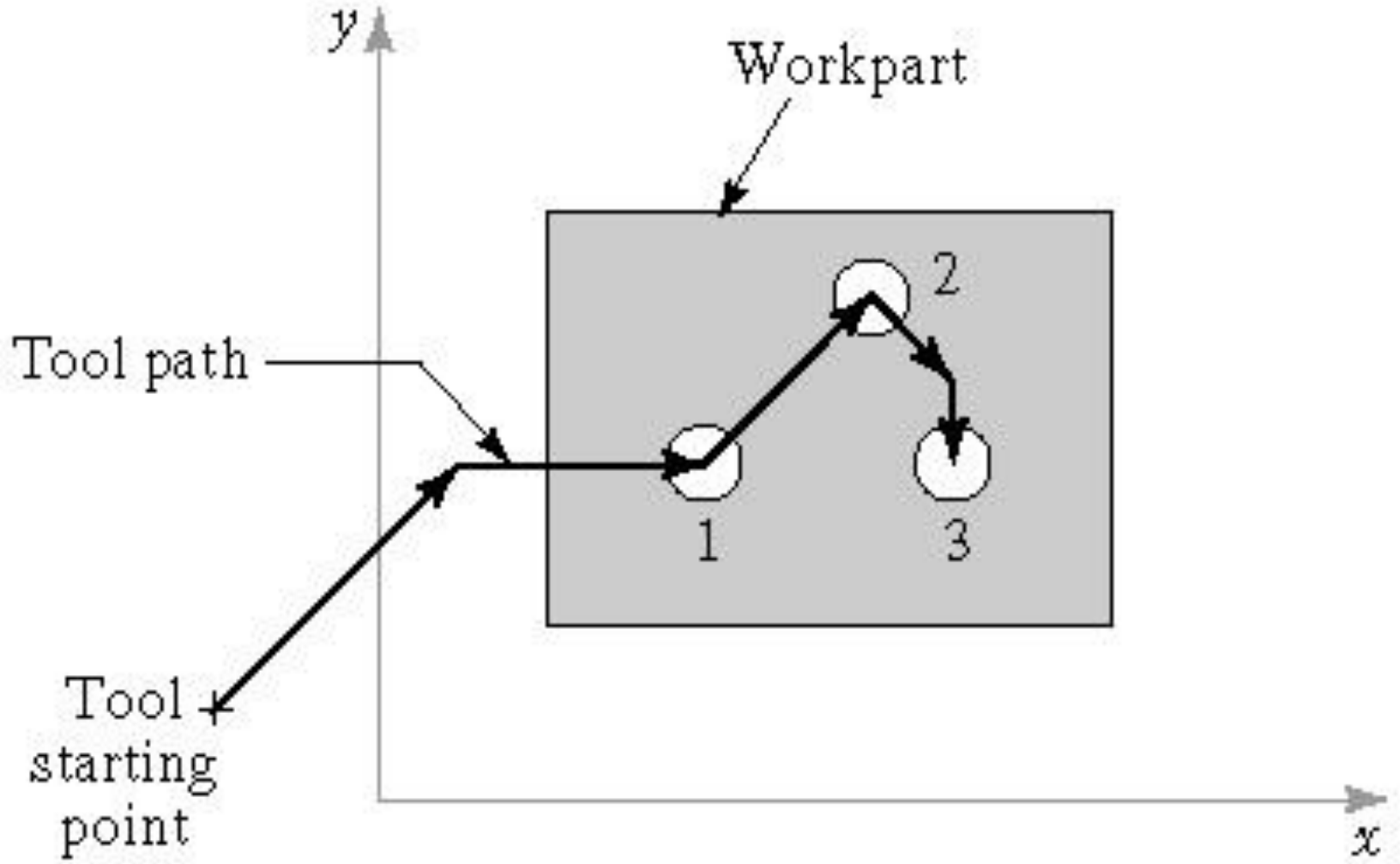
Point-to-Point systems

- Also called position systems
- System moves to a location and performs an operation at that location (e.g., drilling)
- Also applicable in robotics

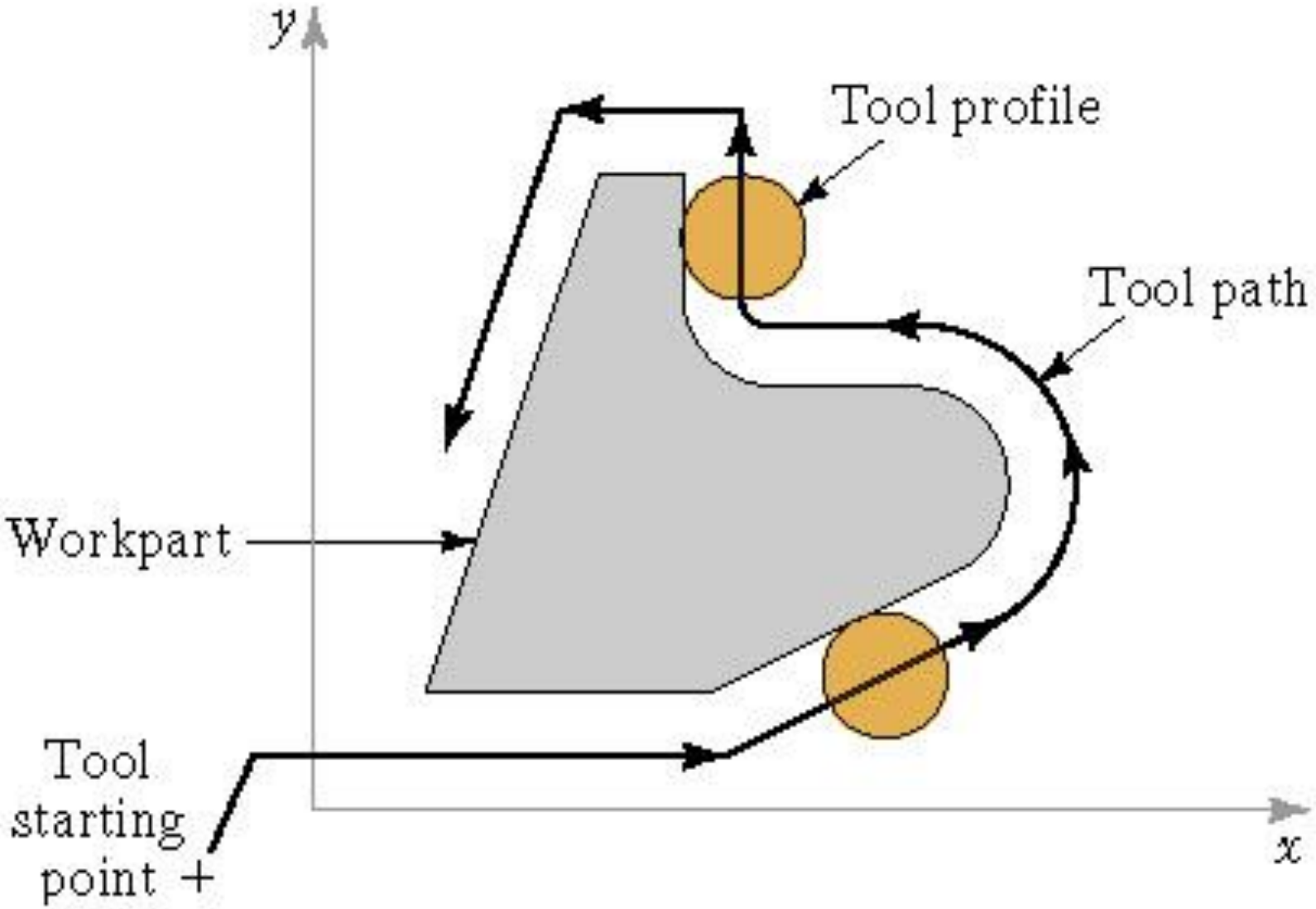
Continuous path systems

- Also called contouring systems in machining
- System performs an operation during movement (e.g., milling and turning)

PTP

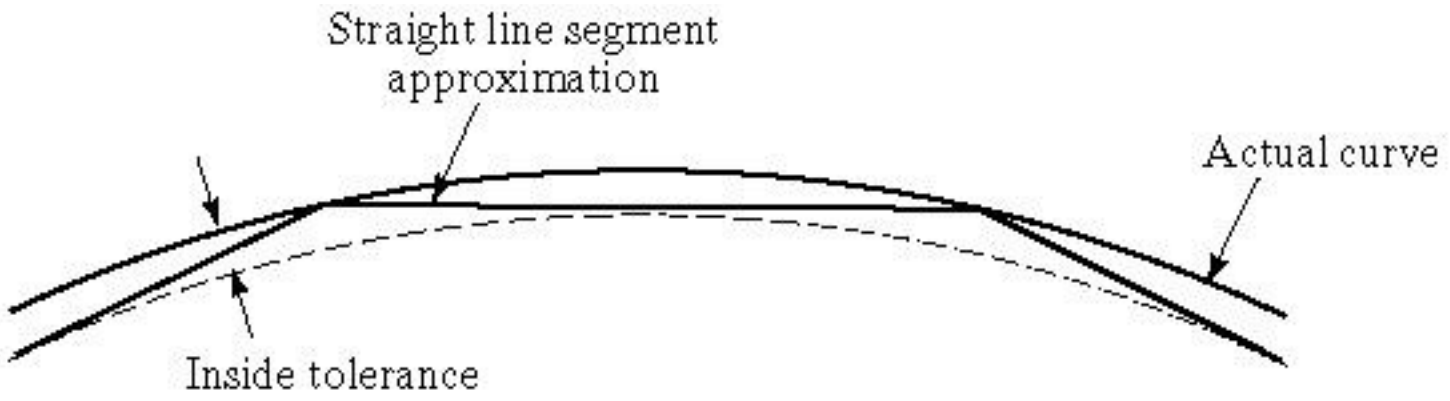


CONTOURING



Interpolation Methods

1. **Linear interpolation**
   Straight line between two points in space
2. **Circular interpolation**
   Circular arc defined by starting point, end point, center or radius, and direction
3. **Helical interpolation**
   Circular plus linear motion
4. **Parabolic and cubic interpolation**
   Free form curves using higher order equations



NC Part Programming

Bit - 0 or 1 = absence or presence of hole in the tape

Character - row of bits across the tape

Word - sequence of characters (e.g., y-axis position)

Block - collection of words to form one complete instruction

Part program - sequence of instructions (blocks)

Block Format

Organization of words within a block in NC part program

Also known as tape format because the original formats were designed for punched tape

Word address format - used on all modern CNC controllers

- Uses a letter prefix to identify each type of word
- Spaces to separate words within the block
- Allows any order of words in a block
- Words can be omitted if their values do not change from the previous block

N - sequence number prefix

G - preparatory words

Example: G00 = PTP rapid traverse move

X, Y, Z - prefixes for x, y, and z-axes

F - feed rate prefix

S - spindle speed

T - tool selection

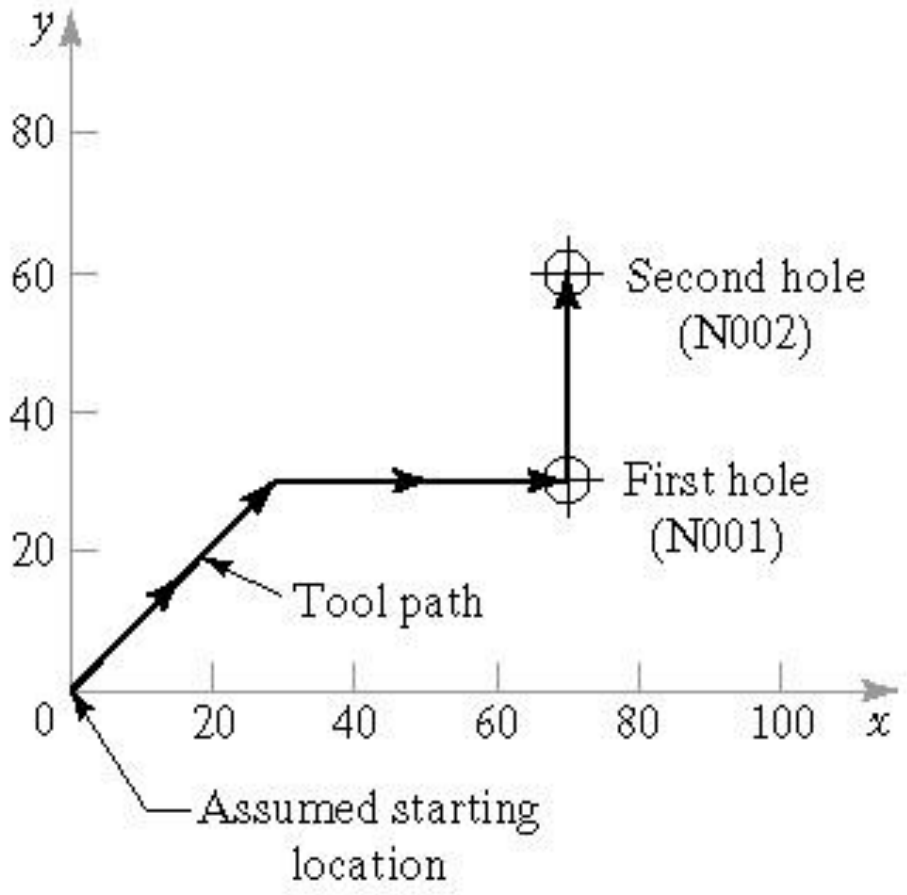
M - miscellaneous command

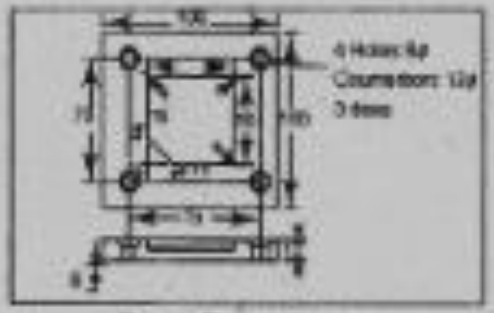
Example: M07 = turn cutting fluid on

Example: Word Address Format

N001 G00 X07000 Y03000 M03

N002 Y06000





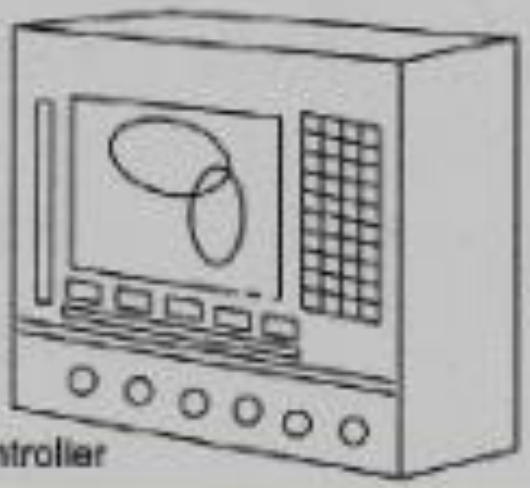
Part Drawing

```
N030 G00 X34.43 Y12.5
N035 Z2.0
N040 G01 X105.0 Y35.5
N045 X55.0 Y65.0
N050 G00 Z50
```

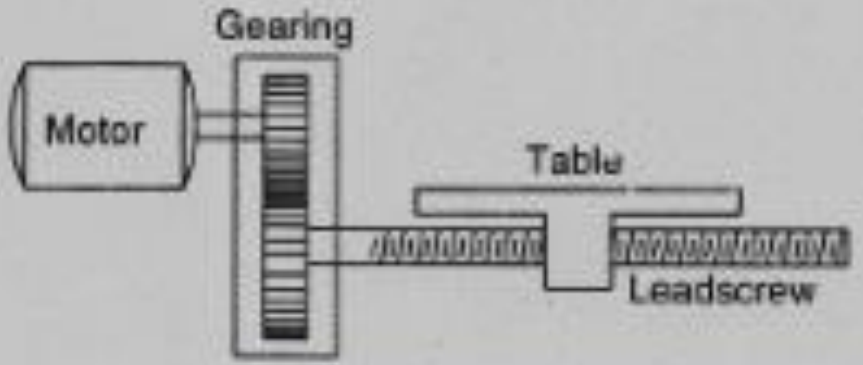
Part Program

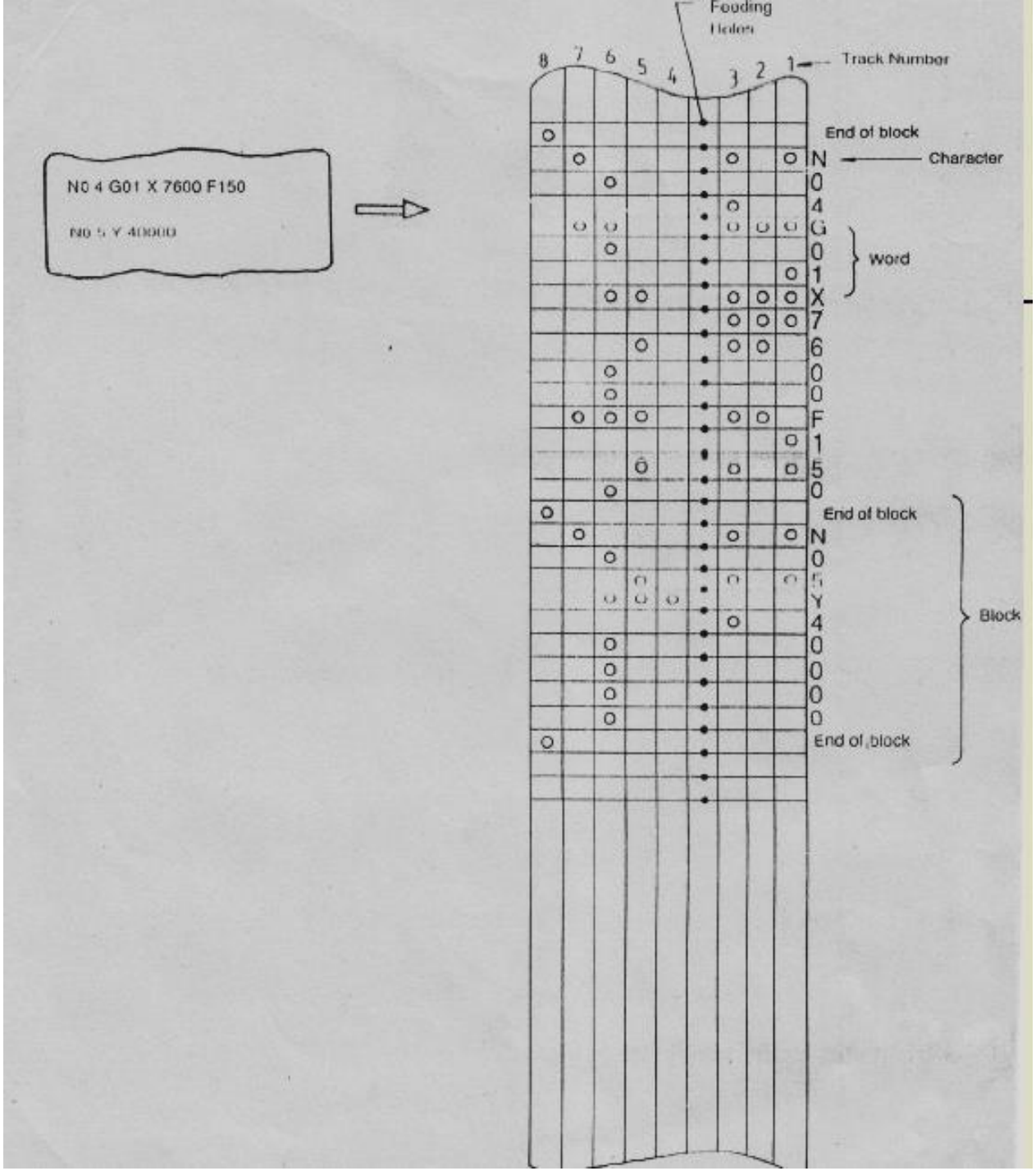
Paper
Tape Reader

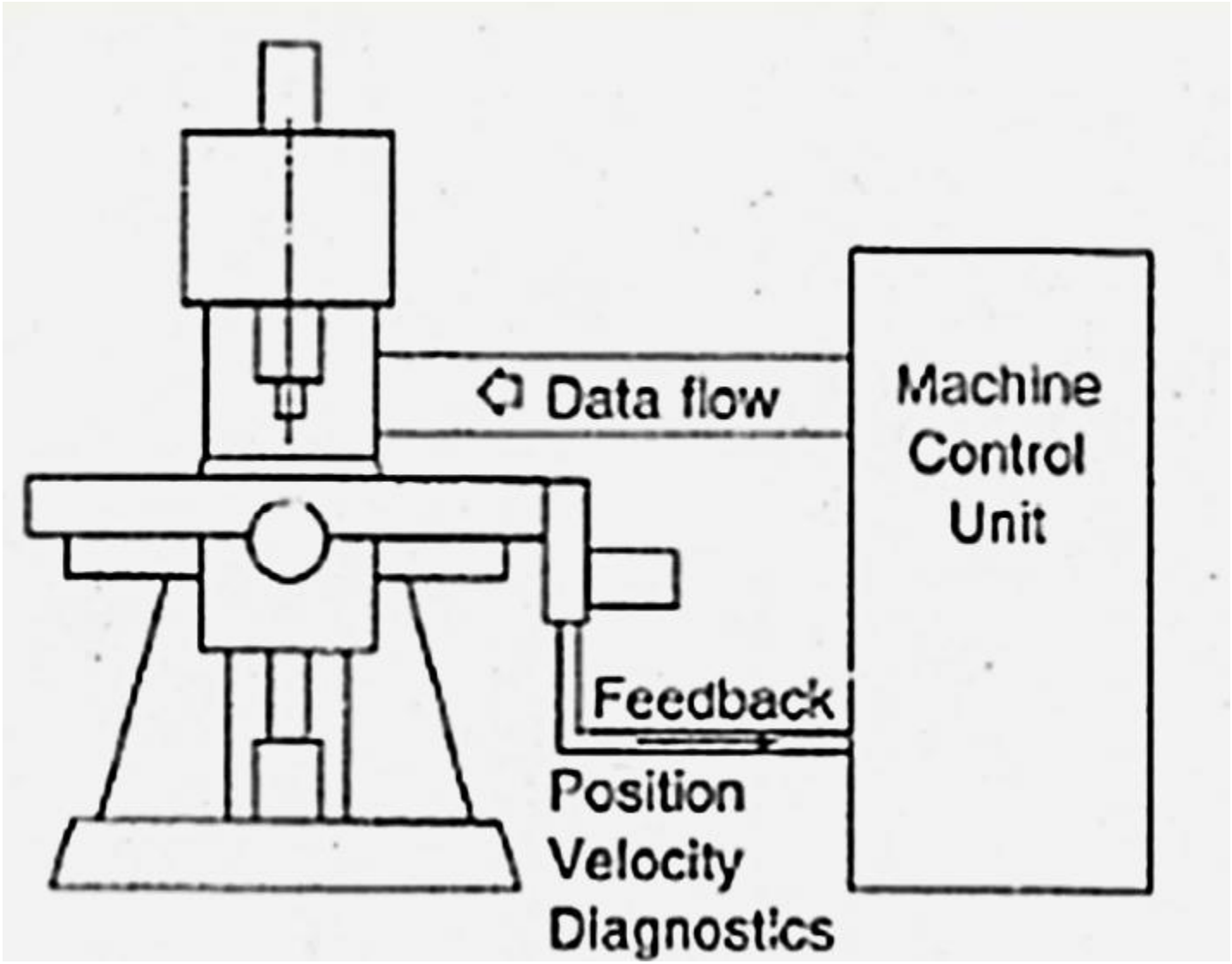
Program
Tape



Controller




N0 4 G01 X 7600 F150
N0 5 Y 40000
Feeding Holes
8 7 6 5 4 3 2 1
Track Number
End of block
N
Character
0
4
G
0
1
X
7
6
0
0
F
1
5
0
Word
End of block
N
0
5
Y
4
0
0
0
0
End of block
Block


Data flow
Machine
Control
Unit
Feedback
Position
Velocity
Diagnostics

Advantages of NC

Flexibility

Capability for Complex

Work-pieces

Manage Large Work-pieces

Reduced Jig & Fixture Cost

Higher Quality

Direct Numerical Control

Advantages:

> Library of programs

> Instant modifications

> Links with CAD

> Increase Information

Response

> Instant Reports

Computer Numerical Control (CNC)

Advantages:

> CRT allows review/editing

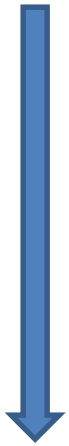
> Pre-check/simulation

> Interface allows more capability

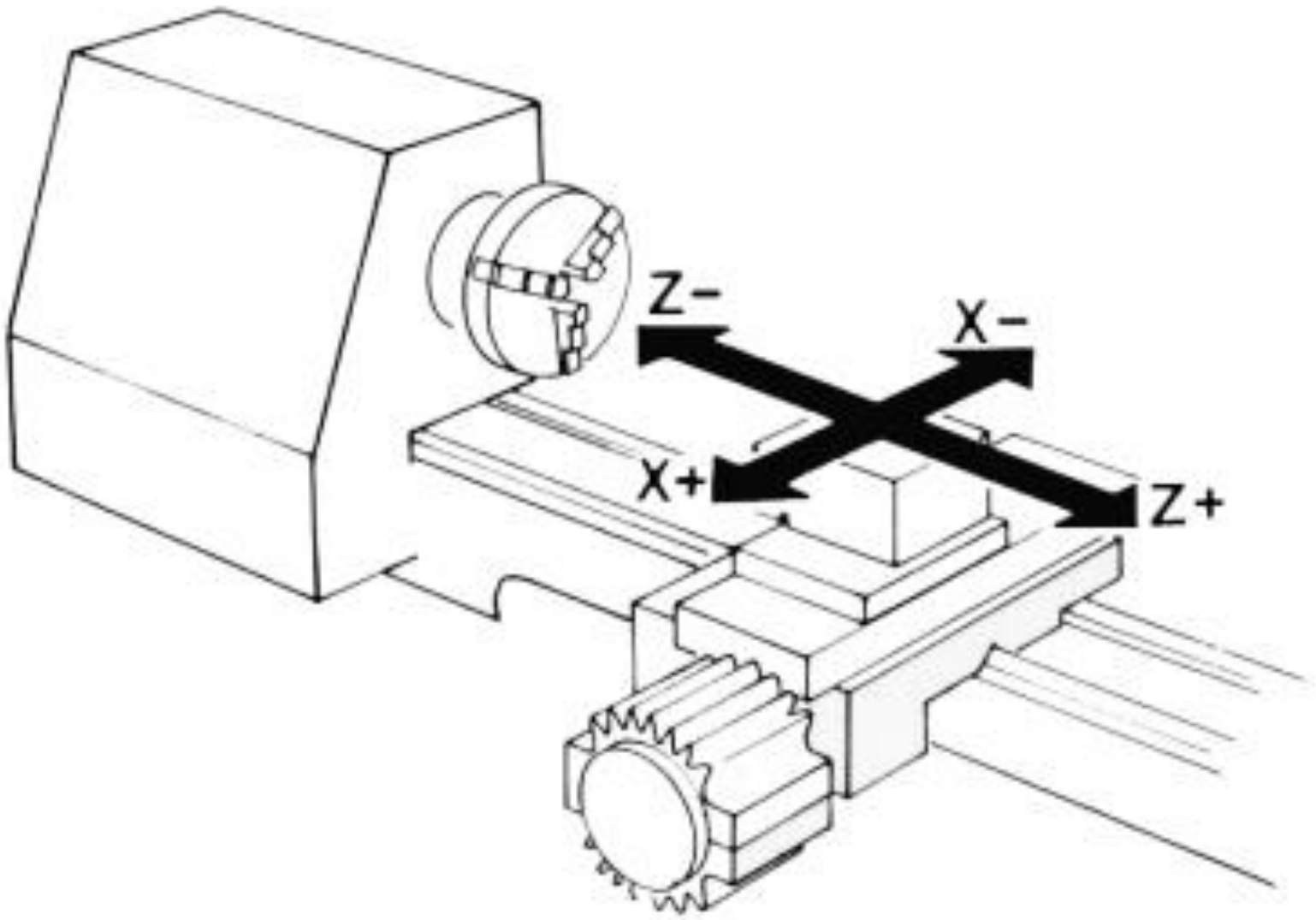
> Accurate positioning

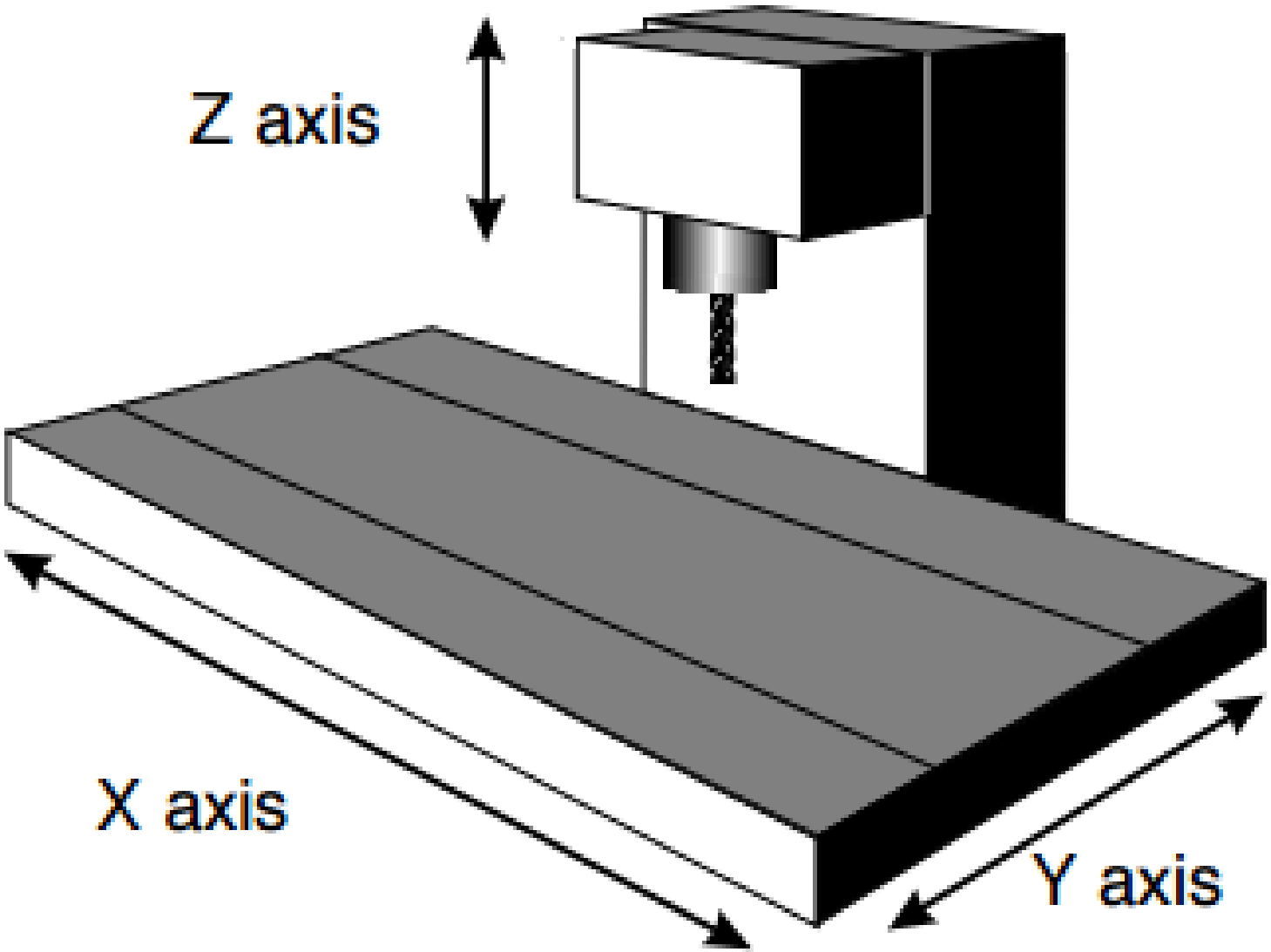
> More functions

NUMERICAL CONTROL PROGRAMMING



COMPUTER NUMERICAL CONTROL PROGRAMMING

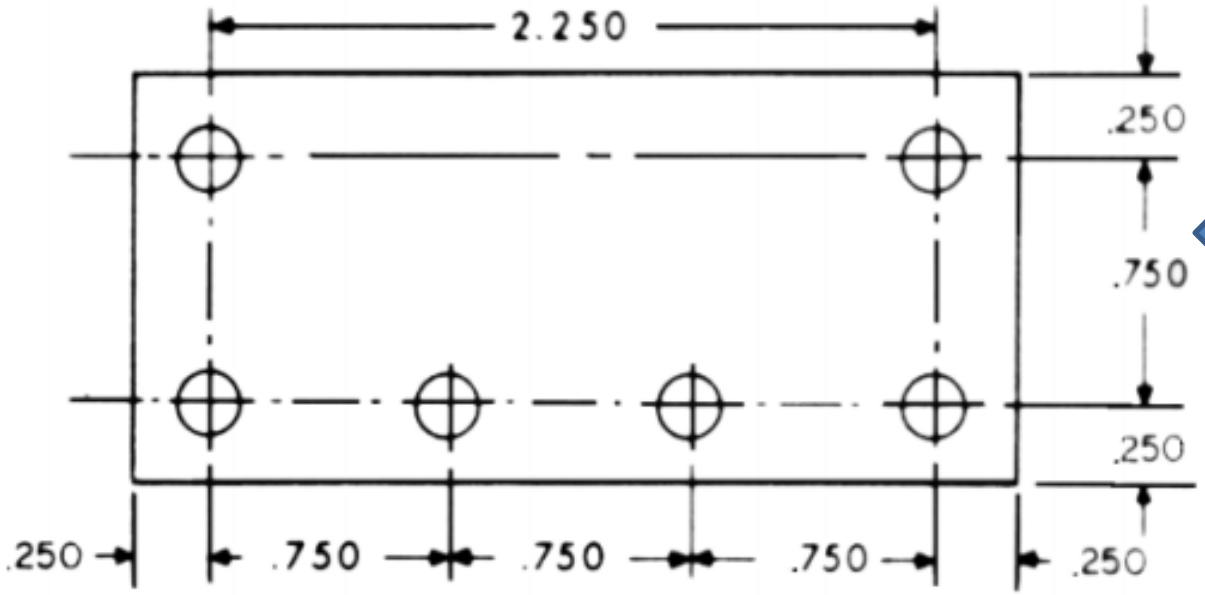

Z-
X-
X+
Z+


Z axis
X axis
Y axis

- X axis controls the table movement left or right.

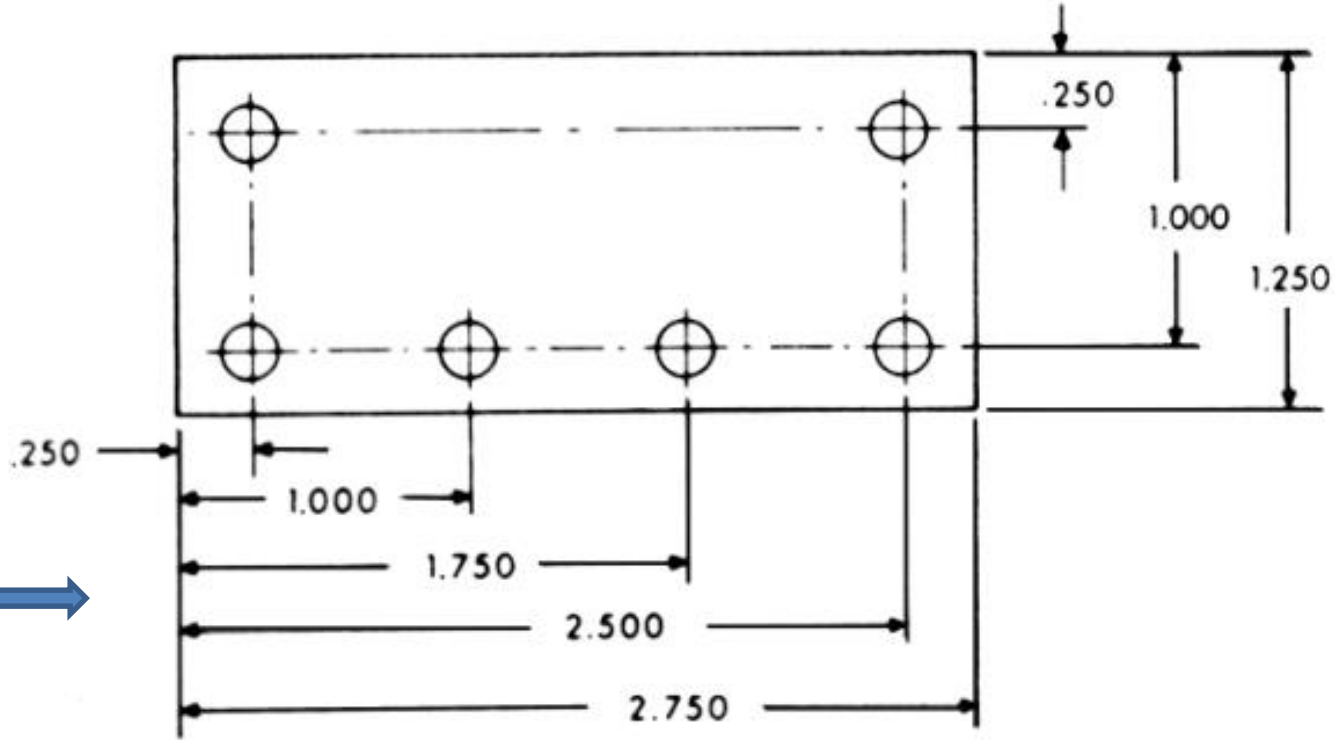
- Y axis controls the table movement toward or away from the column.

- Z axis controls the vertical (up or down) movement of the knee or spindle.

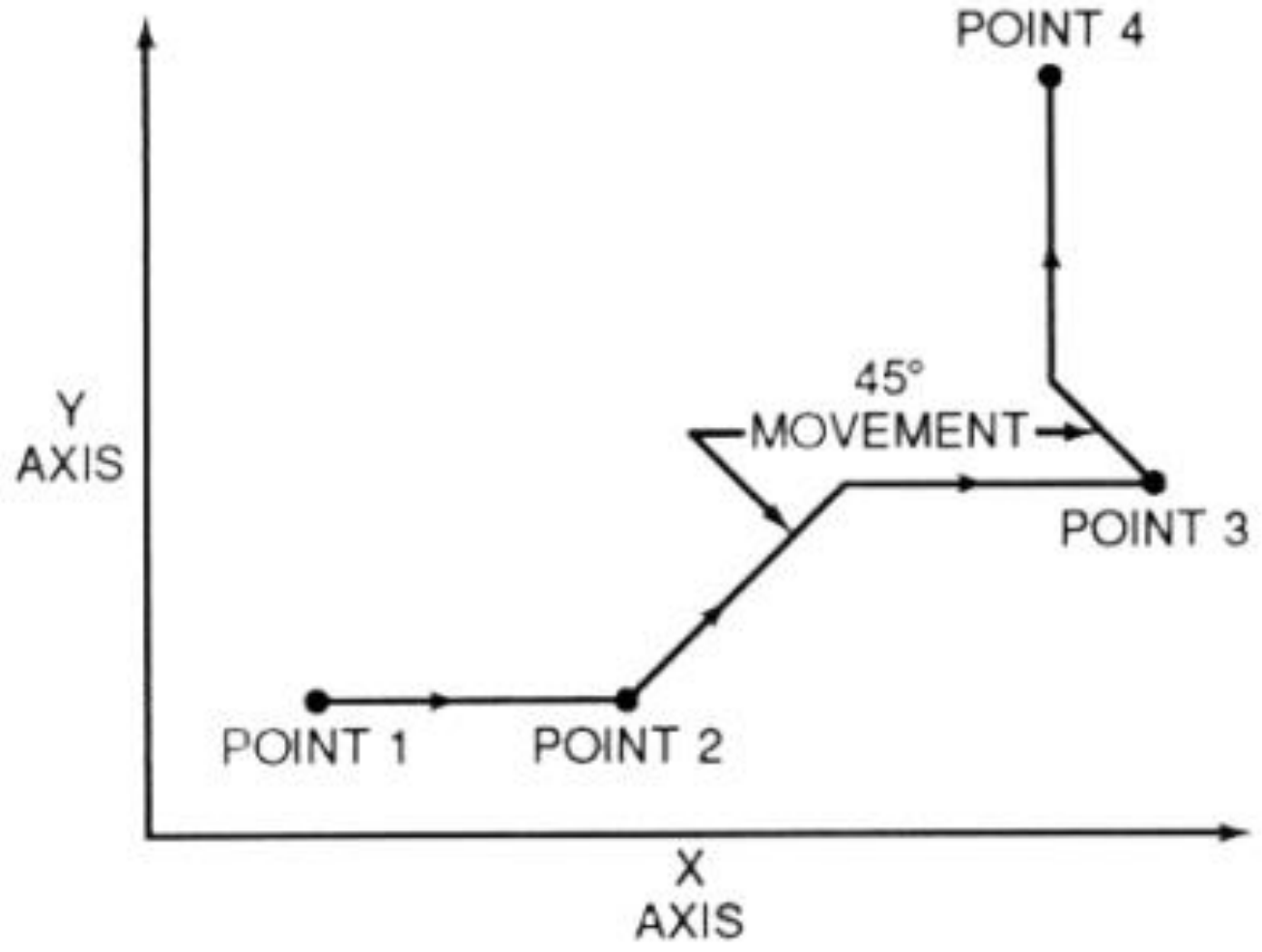


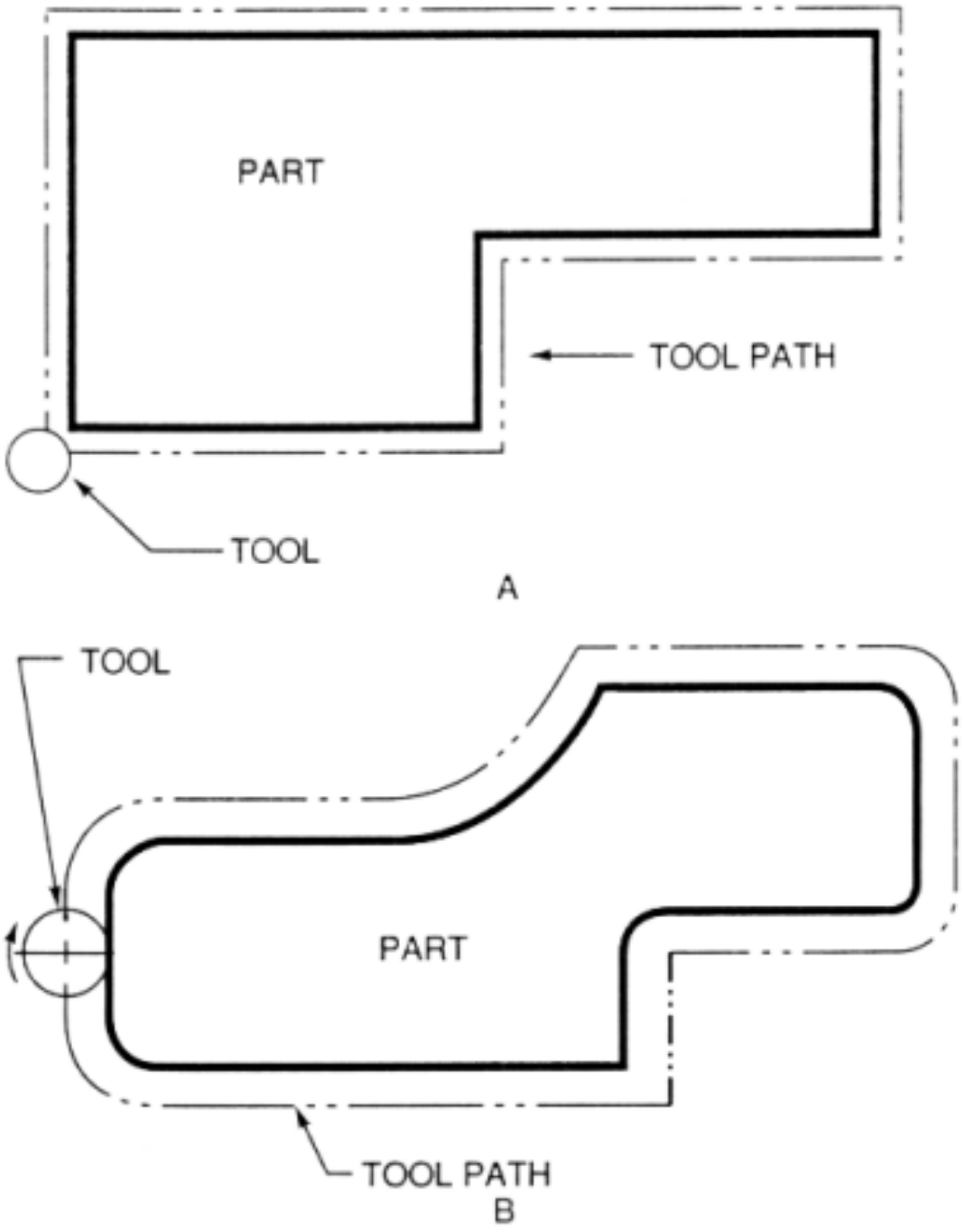
Positioning Reference Point Systems - INCREMENTAL

Positioning Reference Point Systems - ABSOLUTE



CNC Positioning Systems - **Point-to-Point or Positioning**





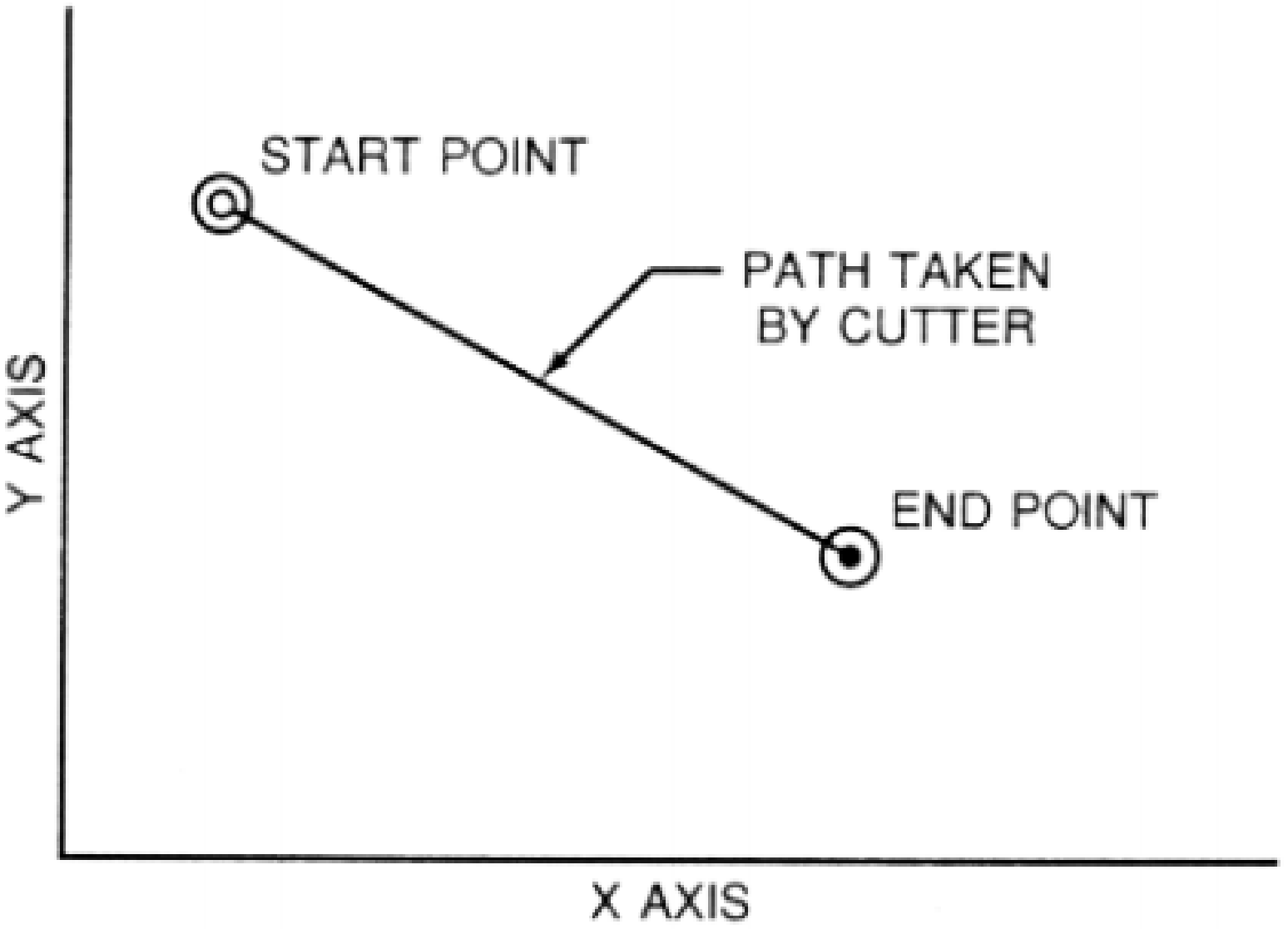
Types of contour machining (A) Simple contour; (B) complex contour

Interpolation

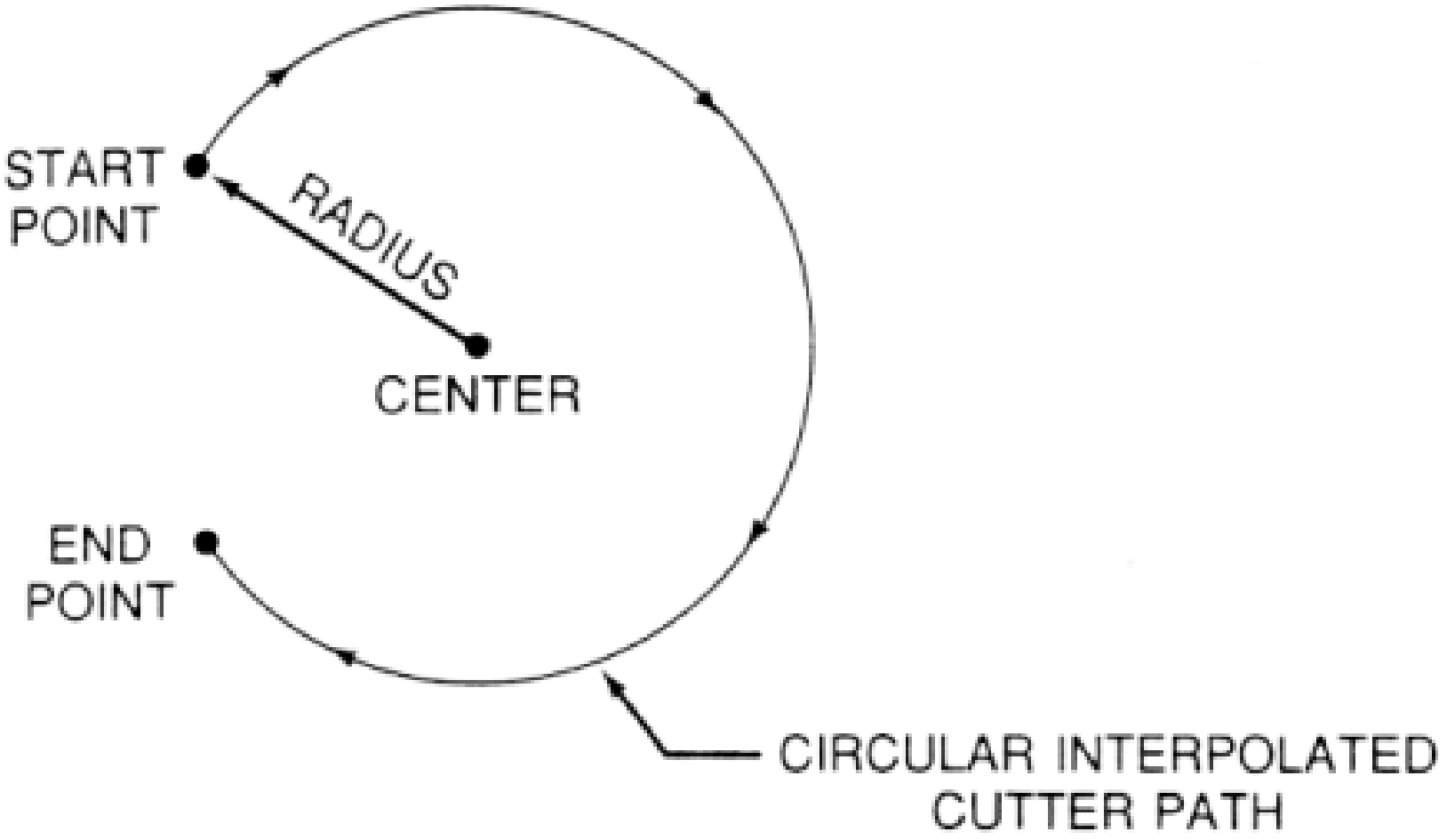
Method by which contouring machine tools move from one programmed point to the next is called interpolation

1. Linear
2. Circular
3. Helical
4. Parabolic
5. cubic

Linear Interpolation



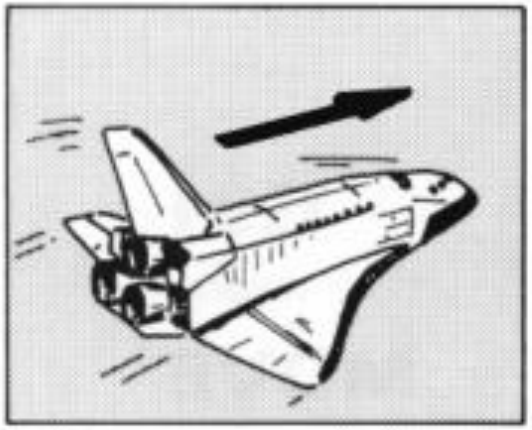
Circular Interpolation



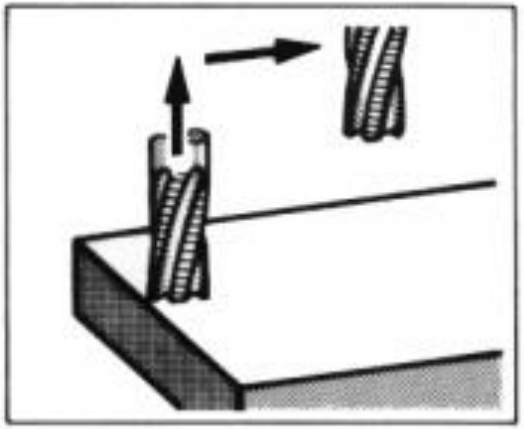
PROGRAMMING CODES

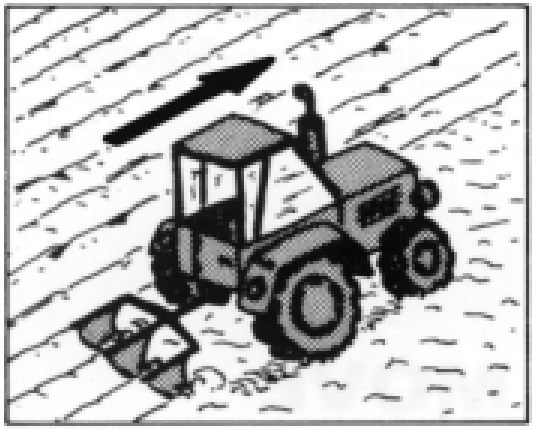
G Codes and **M Codes**

G CODES - preparatory functions

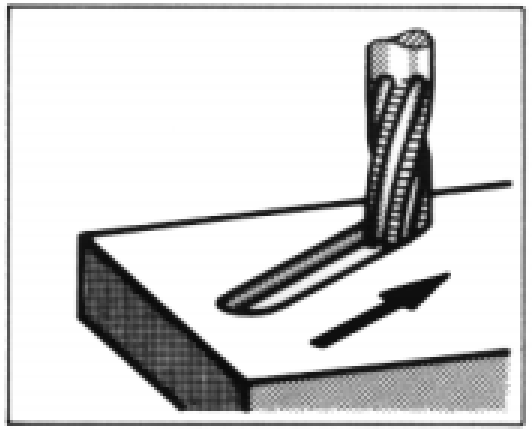


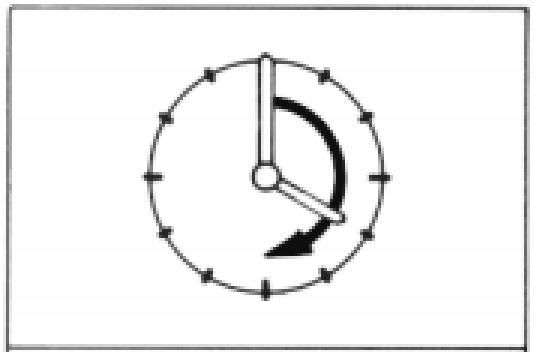
G00
RAPID TRAVERSE



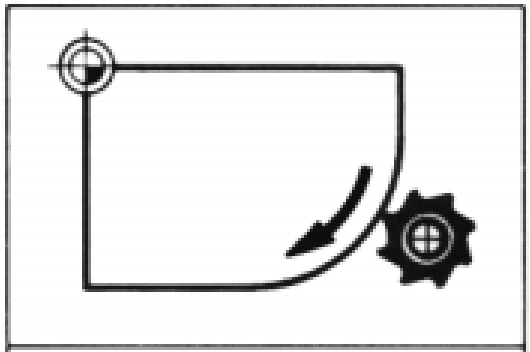


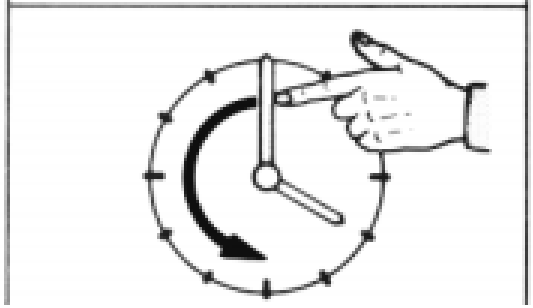
G01
LINEAR INTERPOLATION
(STRAIGHT LINE MOVEMENT)



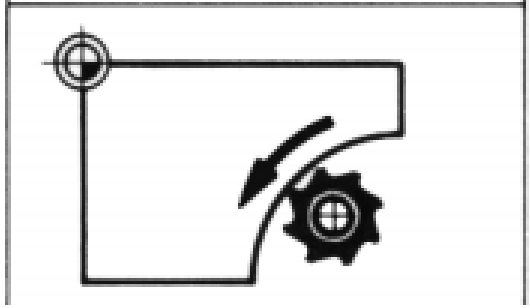


G02
CIRCULAR INTERPOLATION
(CLOCKWISE)





G03
CIRCULAR INTERPOLATION
(COUNTERCLOCKWISE)

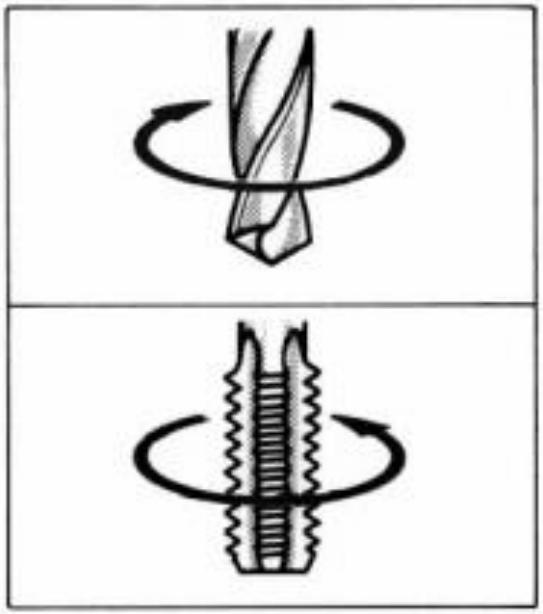


Group	Code	Function
01	G00	Rapid positioning
01	G01	Linear interpolation
01	G02	Circular interpolation clockwise (CW)
01	G03	Circular interpolation counterclockwise (CCW)
06	G20*	Inch input (in.)
06	G21*	Metric input (mm)
	G24	Radius programming (**)
00	G28	Return to reference point
00	G29	Return from reference point
	G32	Thread cutting (**)
07	G40	Cutter compensation cancel
07	G41	Cutter compensation left
07	G42	Cutter compensation right
08	G43	Tool length compensation positive (+) direction
08	G44	Tool length compensation minus (-) direction
08	G49	Tool length compensation cancel
	G84	Canned turning cycle (**)
03	G90	Absolute programming
03	G91	Incremental programming

(*) - on some machines and controls, these may be G70 (inch) and G71 (metric)

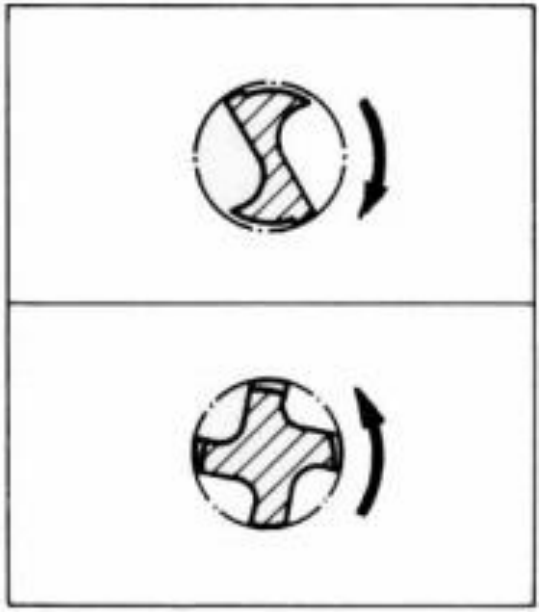
(**) - refers only to CNC lathes and turning centers.

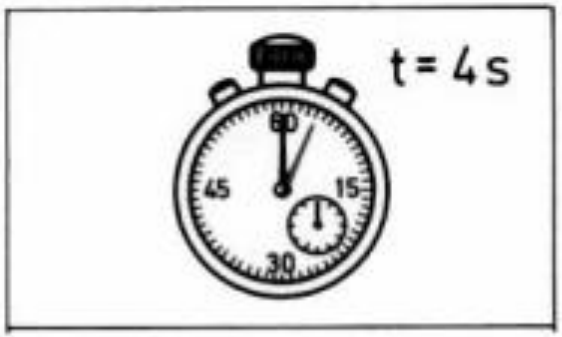
M CODES - miscellaneous functions



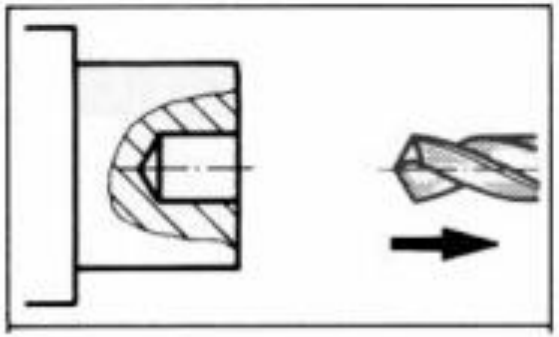
M03
DIRECTION OF ROTATION
(CLOCKWISE)

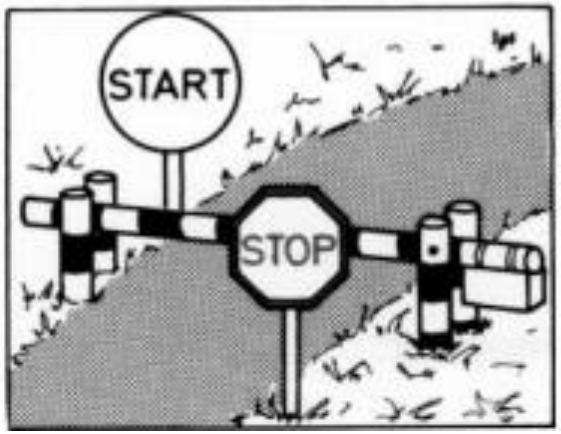
M04
DIRECTION OF ROTATION
(COUNTERCLOCKWISE)



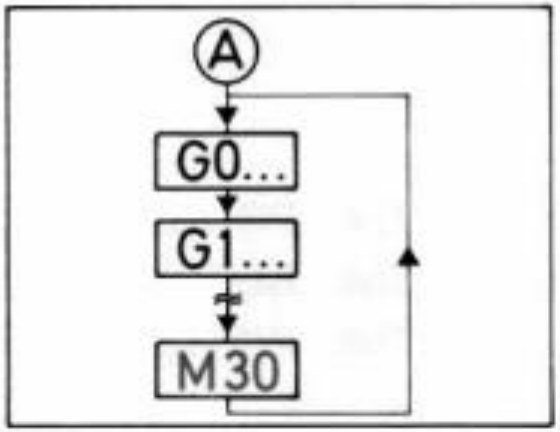


M06
TOOL CHANGE WITH
AUTOMATIC RETRACTION





M30
END OF PROGRAM
AND
RETURN TO BEGINNING
OF PROGRAM



Code	Function
M00	Program stop
M02	End of program
M03	Spindle start (forward CW)
M04	Spindle start (reverse CCW)
M05	Spindle stop
M06	Tool change
M08	Coolant on
M09	Coolant off
M10	Chuck - clamping (**)
M11	Chuck - unclamping (**)
M12	Tailstock spindle out (**)
M13	Tailstock spindle in (**)
M17	Toolpost rotation normal (**)
M18	Toolpost rotation reverse (**)
M30	End of tape and rewind
M98	Transfer to subprogram
M99	End of subprogram

(**) - refers only to CNC lathes and turning centers.

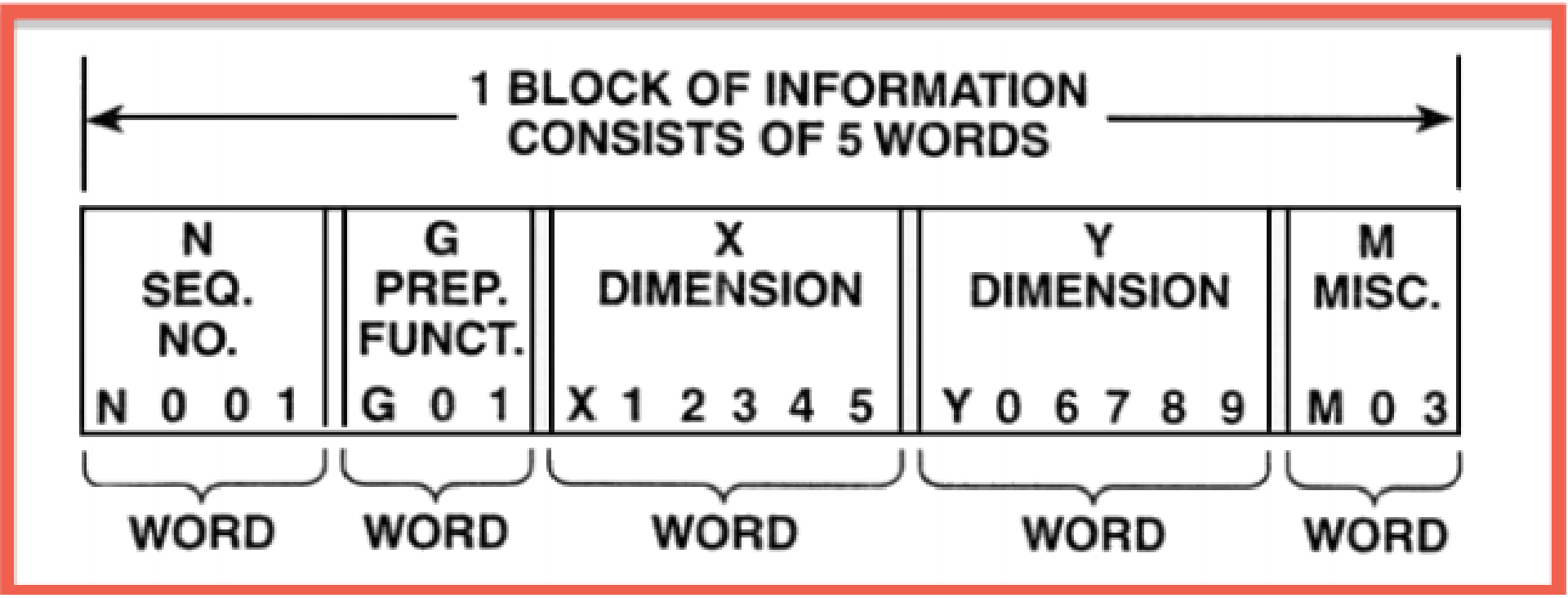
F – Feed

S – Speed

D – Cutter diameter offset

T – Tool Number

Block of Information

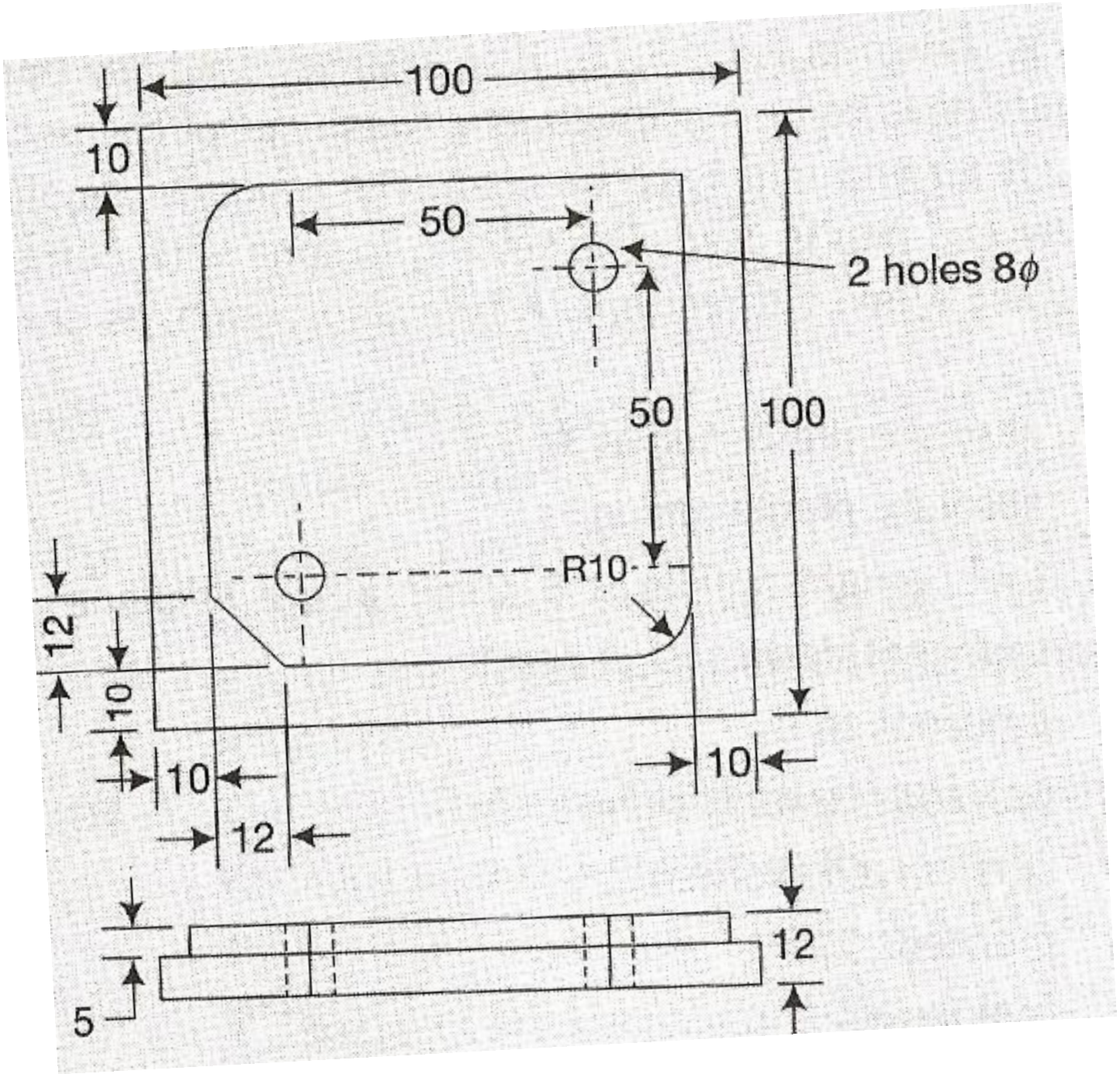


complete block of information consists of five words

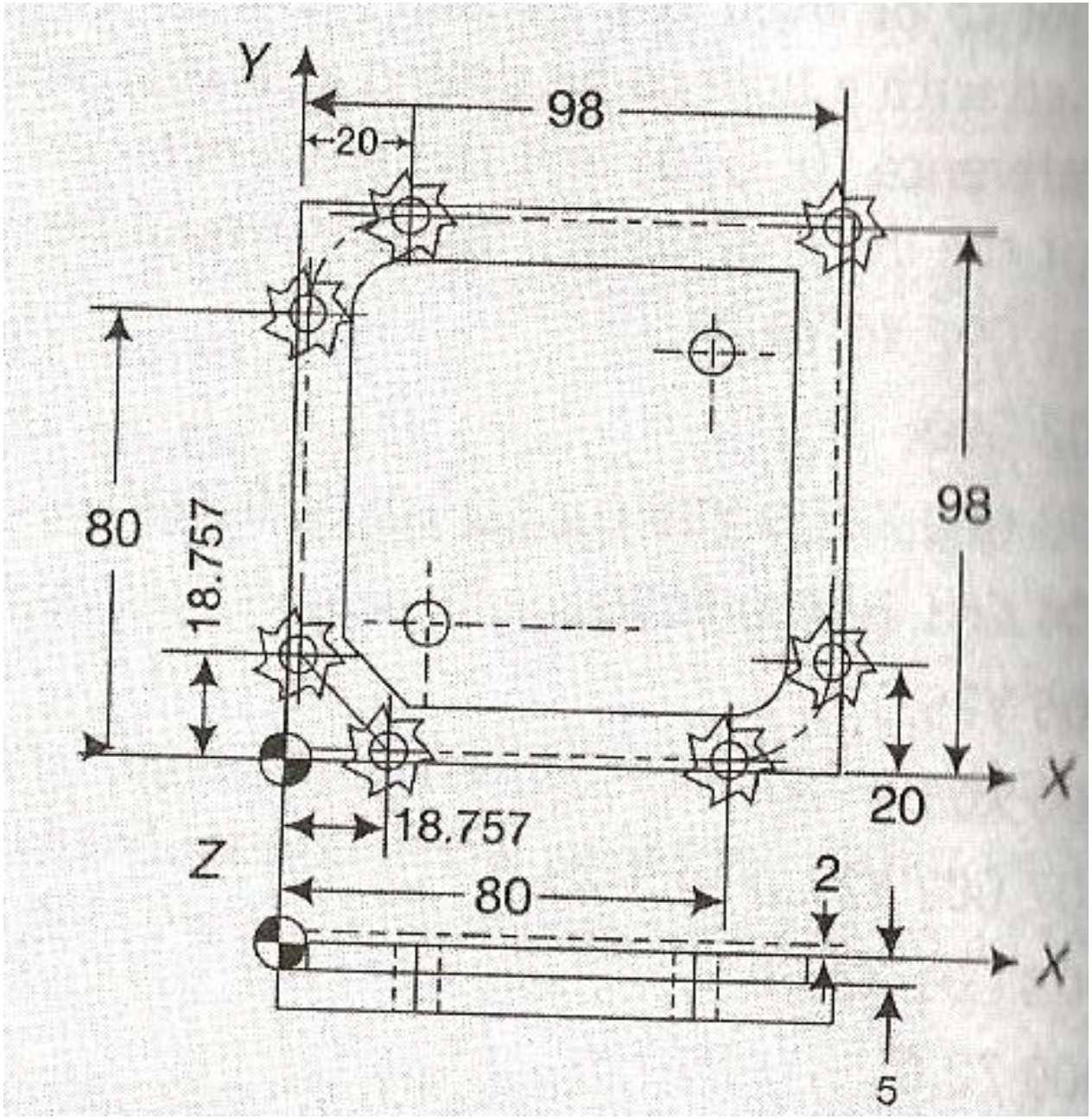
- N001 represents the sequence number of the operation.
- G01 represents linear interpolation
- X12345 will move the table 1.2345 in. in a positive direction along the X axis.
- Y06789 will move the table 0.6789 in. along the Y axis
- M03 Spindle on CW.

Graving Tool

NC part program code	Comments
N001 G21 G90 G92 X-050.0 Y-050.0 Z010.0;	Define origin of axes.
N002 G00 Z-020.0 S1989 M03;	Rapid to cutter depth, turn spindle on.
N003 G01 G94 G42 Y0 D05 F398;	Bring tool to starting y-value, start cutter offset.
N004 G01 X075.0;	Mill lower horizontal edge of part.
N005 G01 X150.0 Y043.02;	Mill angled edge at 35 degrees.
N006 G01 Y070.0;	Mill vertical edge at right of part.
N007 G01 X080.0;	Mill horizontal edge leading to arc.
N008 G17 G02 X050.0 Y100.0 R030.0;	Circular interpolation around arc.
N009 G01 Y125.0;	Mill vertical step above arc.
N010 G01 X0;	Mill top part edge.
N011 G01 Y0	Mill vertical edge at left of part.
N012 G40 G00 X-050.0 Y-050.0 Z010.0 M05;	Rapid move to target point, cancel offset, spindle stop
N013 M30;	End of program, stop machine.

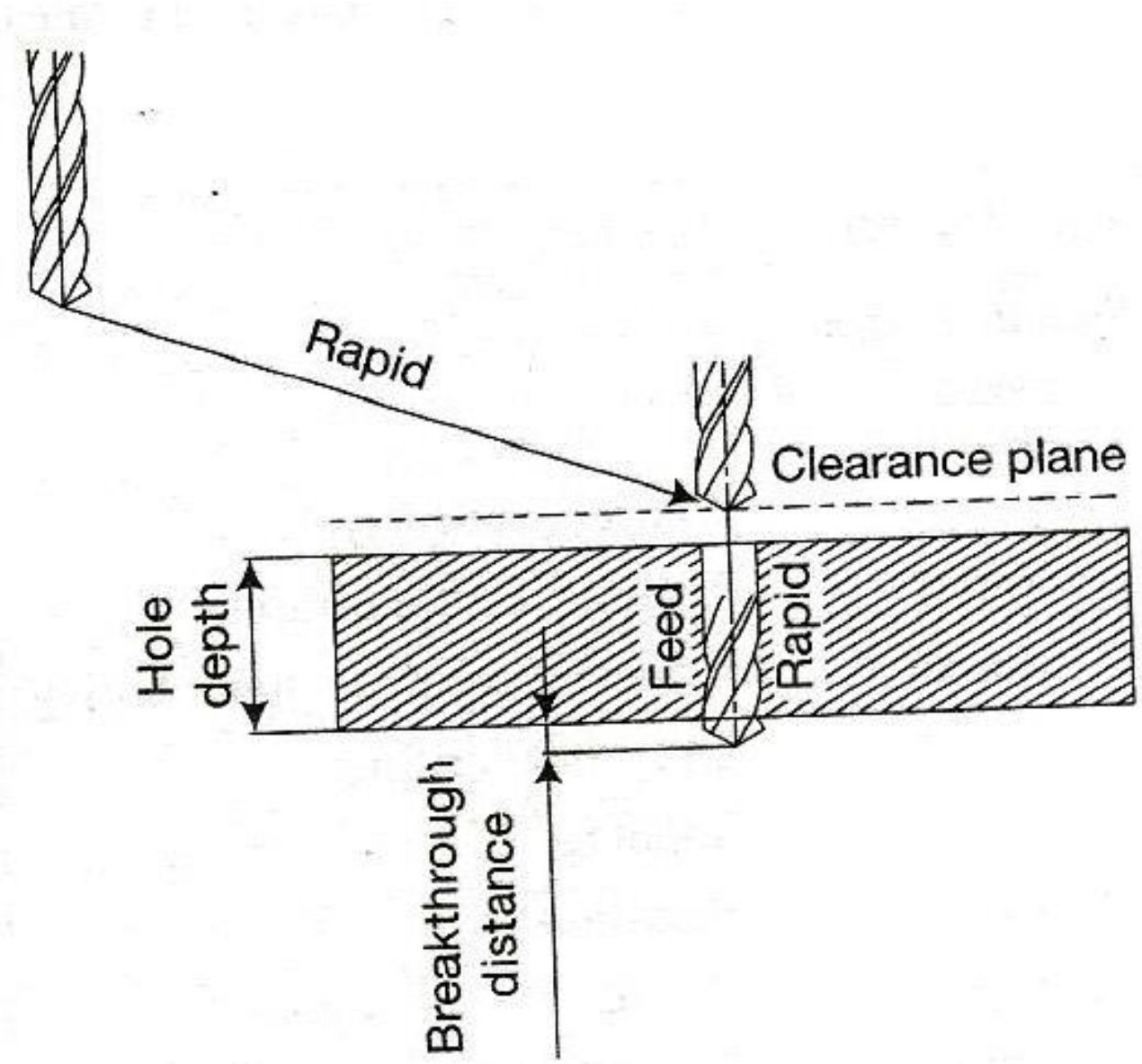

100
10
50
2 holes 8ϕ
50
100
R10
12
10
10
10
12
12
5

TOOL PATH FOR MACHINING THE COMPONENT

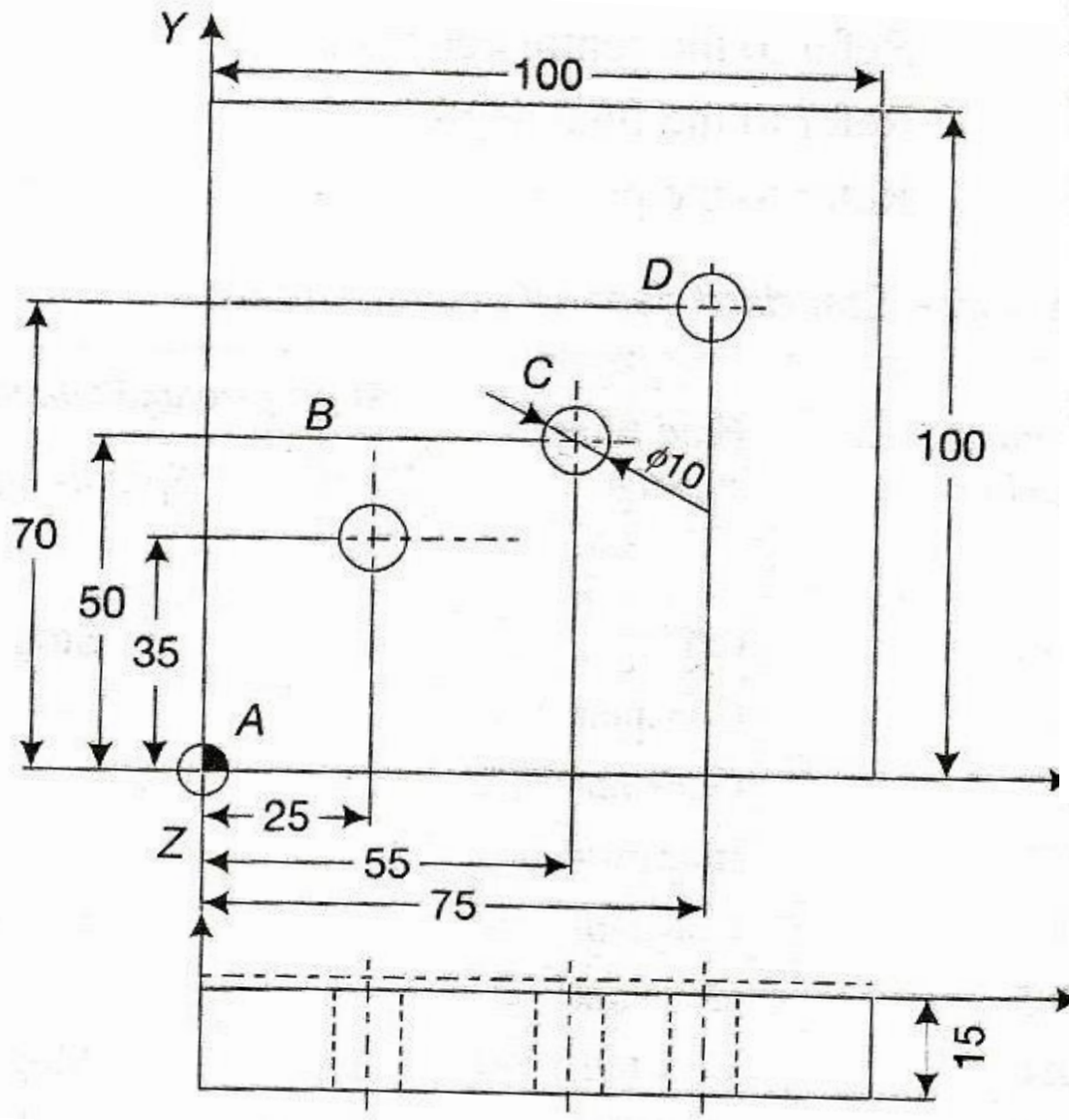


```
%
O1002                                  (Program number)
N010 G71 G92 X0 Y0 Z50                 (Set point)
N015 G90                               (Absolute programming)
N020 T01 S500 M06                      (Tool change speed setting)
N025 G00 Z2.0 M03                      (Rapid move to clearance plane)
N030 G01 Z-5.0 F100                    (Feed to the required depth)
N035 Y80.0 F120                        (Cut along straight line)
N040 G02 X20.0 Y98.0 R20.0 F100        (Circular move)
N045 G01 X98.0                         (Cut along straight line)
N050 Y20                               (Cut along straight line)
N055 G02 X80.0 Y2.0 R18.0              (Circular move)
N060 G01 X18.757                       (Straight line to the intersection point)
N065 X2.0 Y18.757                      (Straight line to the intersection point)
N070 Z2.0 M05                          (Feed to clearance plane stop spindle)
N075 G0 X0 Y0 Z50                      (Rapid to set point)
N080 M02                               (End of program)
```

DRILLING


Rapid
Clearance plane
Hole depth
Feed
Rapid
Breakthrough distance

EXAMPLE FOR CANNED CYCLES




```
N010 G00 X25.0 Y35.0 Z2 *
N015 G01 Z-18.0 F125 *
N020 G00 Z2.0 *
N025 X55.0 Y50.0 *
N035 G01 Z-18.0 F125 *
N040 G00 Z2.0 *
N045 X75.0 Y70.0
N050 G01 Z-18.0 F125 *
N055 G00 Z2.0 *
N065 X0 Y0 Z50 *
```

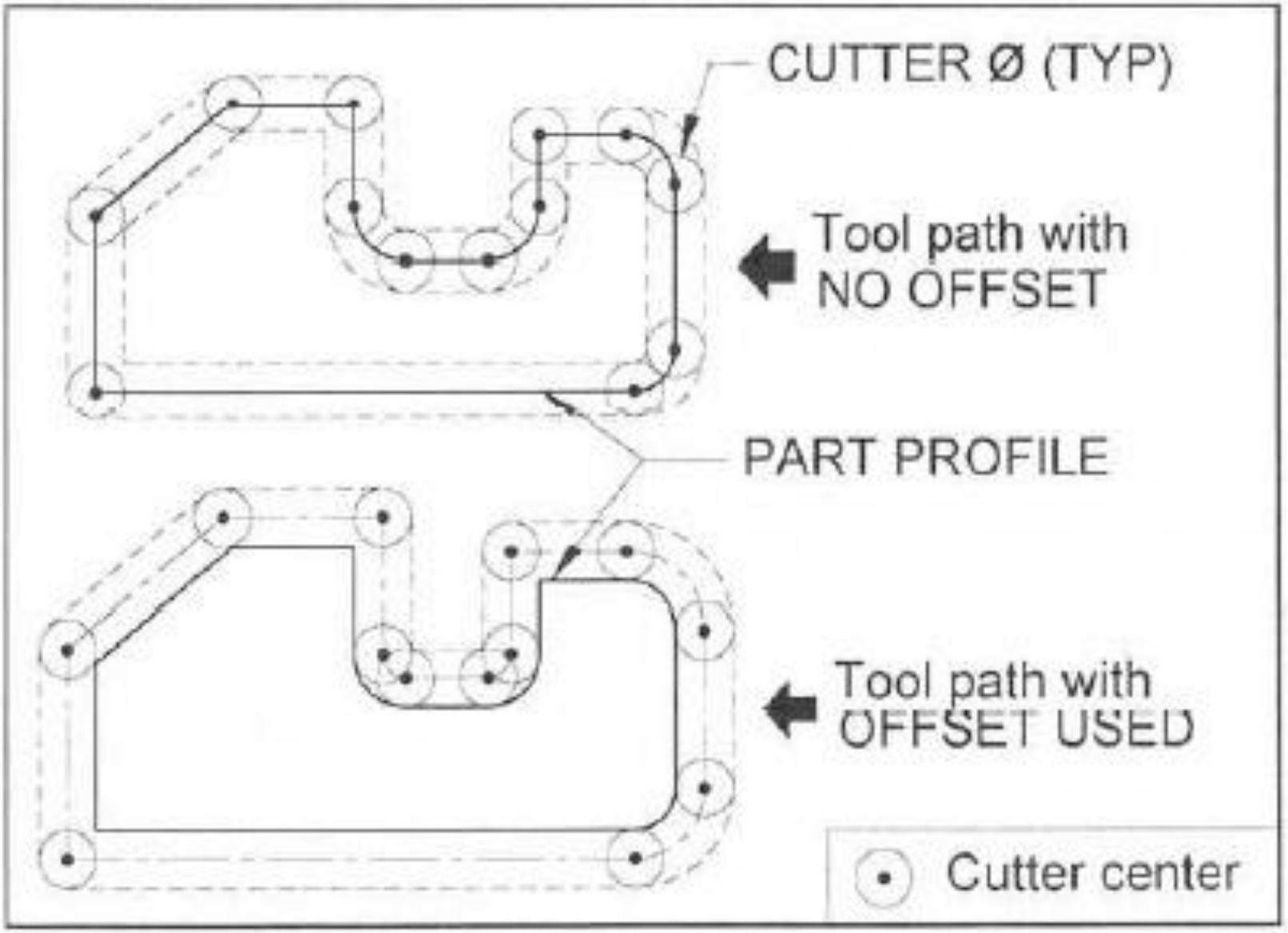
```
N010 G81 X25.0 Y35.0 Z-18.0 R2.0 F125 *
N015 X55.0 Y50.0 *
N020 X75.0 Y70.0 *
N025 G80 X0 Y0 Z50 *
```

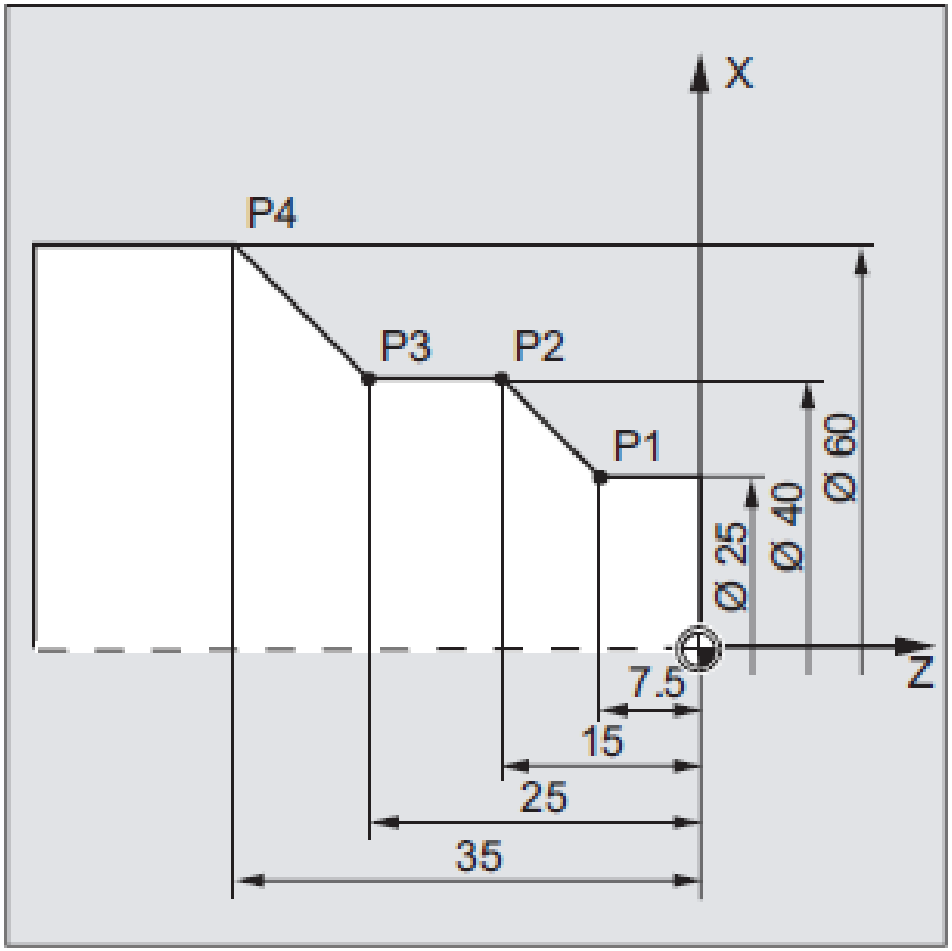
SIMPLIFIED PART PROGRAM USING CANNED CYCLED ‘G’ Code

STANDARD CANNED CYCLE MOTIONS

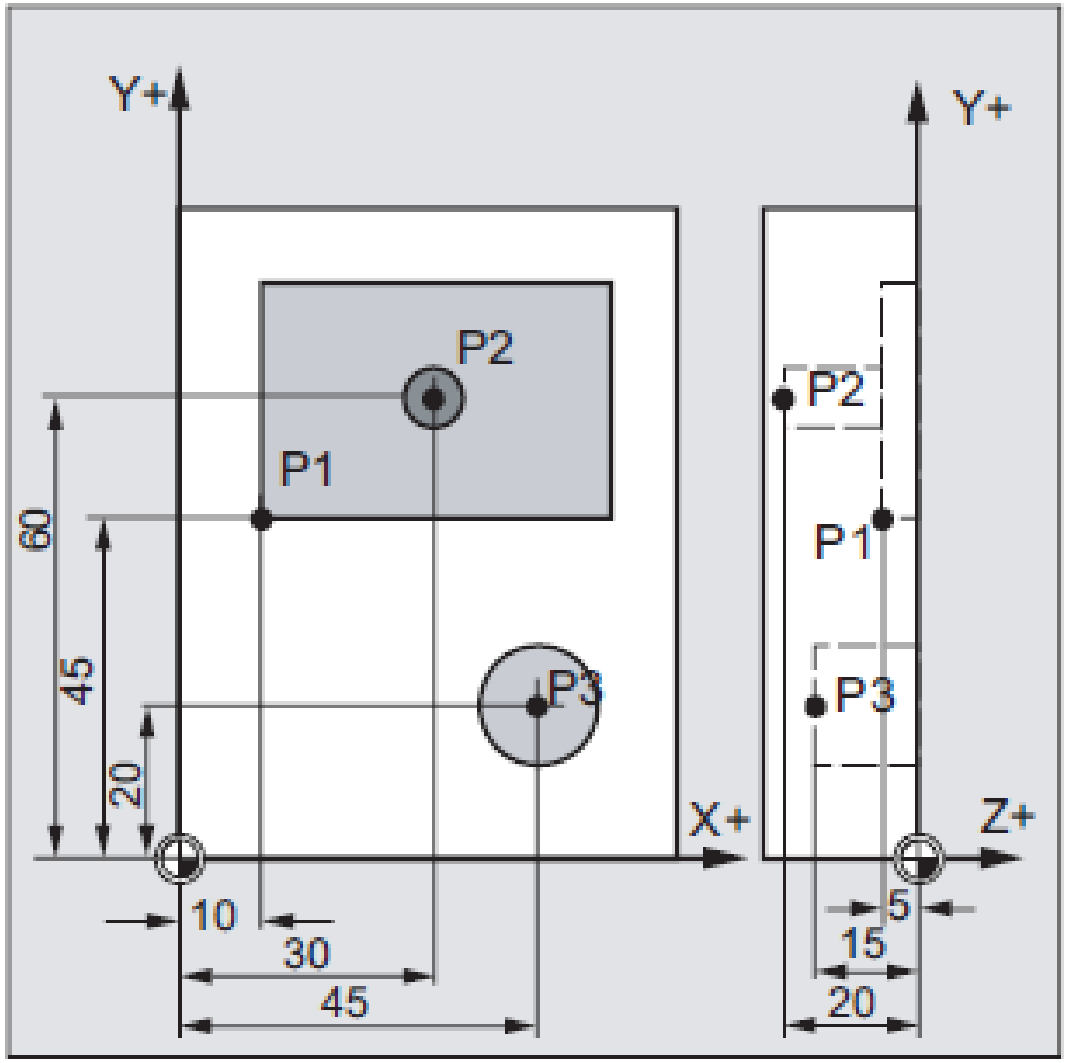
Canned cycle number	*Feed from surface*	*At programmed depth (end of feed point)*			*Used for*
		Dwell	*Spindle speed*	*Spindle return motion*	
G80	Off	—	Stop	—	Cancel canned cycle
G81	Constant	—	—	Rapid	Drilling, centre drilling
G82	Constant	Yes	—	Rapid	Counter sinking, counter boring
G83	Intermittent	—	—	Rapid	Deep-hole drilling
G84	Constant	—	Reverse	Feed	Tapping
G85	Constant	—	—	Feed	Reaming
G86	Constant	—	Stop	Rapid	Boring
G87	Constant	—	Stop	Manual	Multiple boring
G88	Constant	Yes	Stop	Manual	Boring
G89	Constant	Yes	—	Feed	Boring

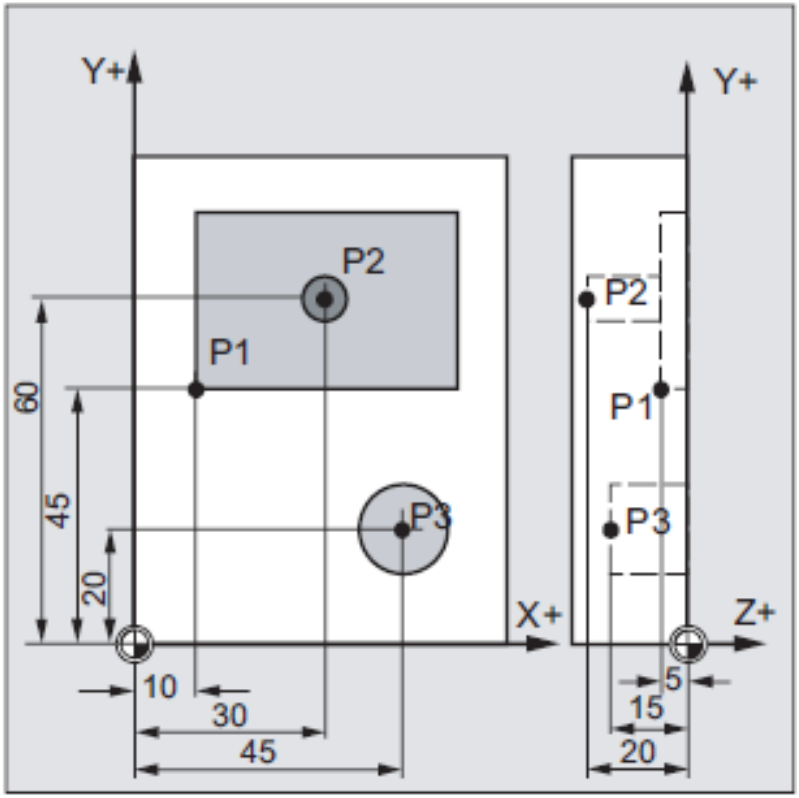
CUTTER RADIUS COMPENSATION





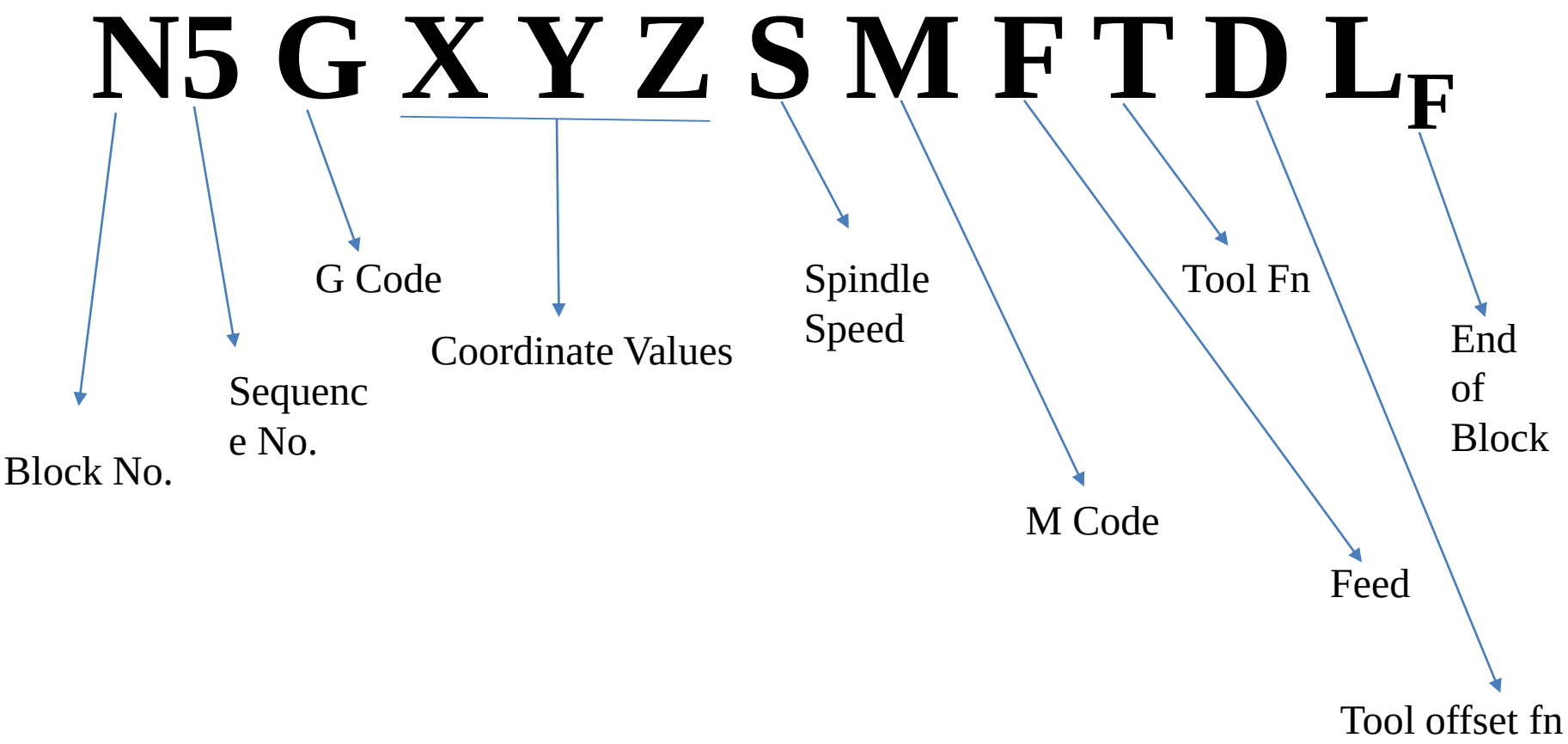
Position	Coordinates
P1	X25 Z-7.5
P2	X40 Z-15
P3	X40 Z-25
P4	X60 Z-35


Y+
P2
P1
60
45
20
P3
X+
10
30
45
Y+
P2
P1
P3
Z+
5
15
20



Position	Coordinates
P1	X10 Y45 Z-5
P2	X30 Y60 Z-20
P3	X45 Y20 Z-15

PROGRAM STRUCTURE


N5 G X Y Z S M F T D L F
Block No.
Sequenc e No.
G Code
Coordinate Values
Spindle Speed
M Code
Feed
Tool Fn
Tool offset fn
End of Block

G Codes – Preparatory functions

G code		Description
Group 1		
G00 [1)]	1	Rapid traverse
G01	2	Linear movement
G02	3	Circle/helix in clockwise direction
G03	4	Circle/helix in counterclockwise direction
G33	5	Thread cutting with constant lead
Group 2		
G17 [1)]	1	XY plane
G18	2	ZX plane
G19	3	YZ plane
Group 3		
G90 [1)]	1	Absolute programming
G91	2	Incremental programming

G94 [1]	1	Feedrate in [mm/min, inch/min]
G95	2	Revolutional feedrate in [mm/rev, inch/rev]
Group 6		
G20 [1]	1	Inch input system
G21	2	Metric input system
Group 7		
G40 [1]	1	Deselection of cutter radius compensation
G41	2	Compensation left of contour
G42	3	Compensation to right of contour
Group 8		
G43	1	Positive tool length compensation on
G44	2	Negative tool length compensation on
G49 [1]	3	Tool length compensation off

Group 9		
G73	1	Deep-hole drilling cycle with chip breakage
G74	2	Left tapping cycle
G76	3	Fine drill cycle
G80 [1)]	4	Cycle off
G81	5	Drilling cycle counterboring
G82	6	Countersink drilling cycle
G83	7	Deep-hole drilling cycle with chip removal
G84	8	Right tapping cycle
G85	9	Drilling cycle
G86	10	Drilling cycle, retraction with G00
G87	11	Reverse countersinking
G89	12	Drilling cycle, retraction with machining feedrate

G96	1	Constant cutting rate on
G97 [1)]	2	Constant cutting rate off

M Codes – Miscellaneous Functions

M00 – PROGRAMME STOP UNCONDITIONAL

M01 – PROGRAMME STOP OPTIONAL

M02 – END OF PROGRAMME

M03 – SPINDLE ROTATION CW

M04 – SPINDLE ROTATION CCW

M05 – SPINDLE STOP

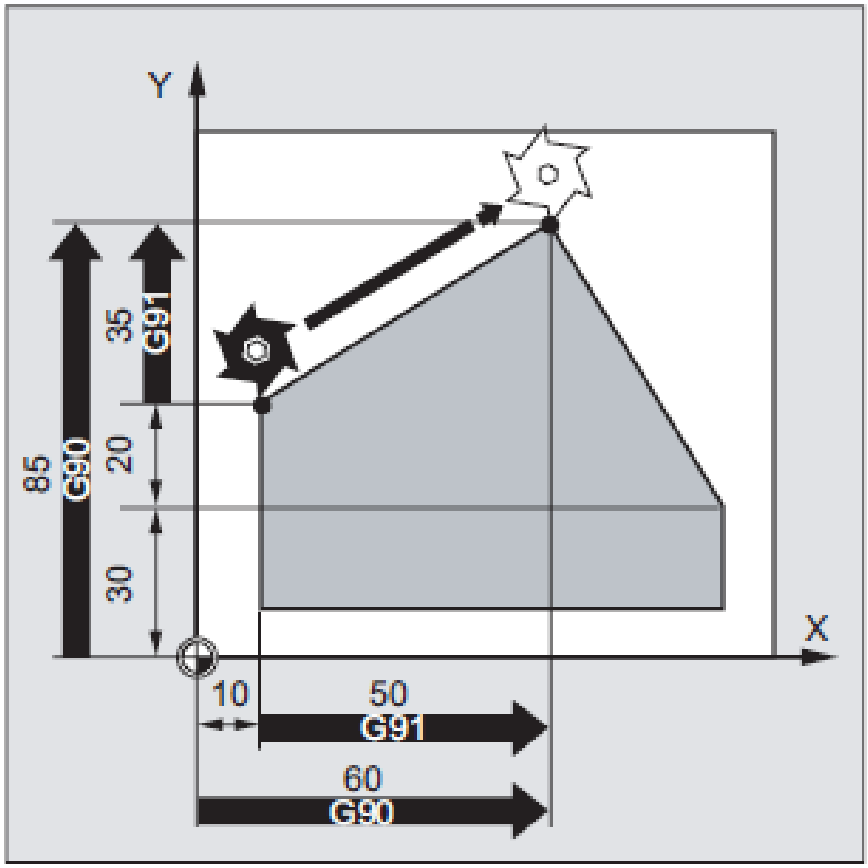
M08/09 – COOLANT ON/OFF

M17 – SUBROUTINE END

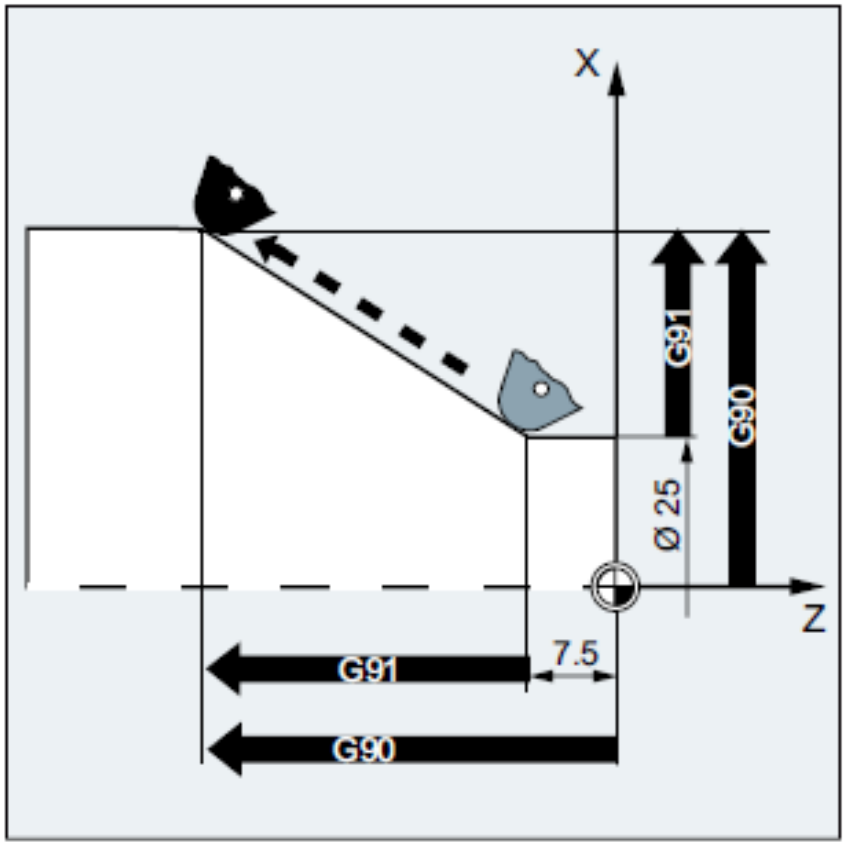
M19 – SPINDLE ORIENTATION

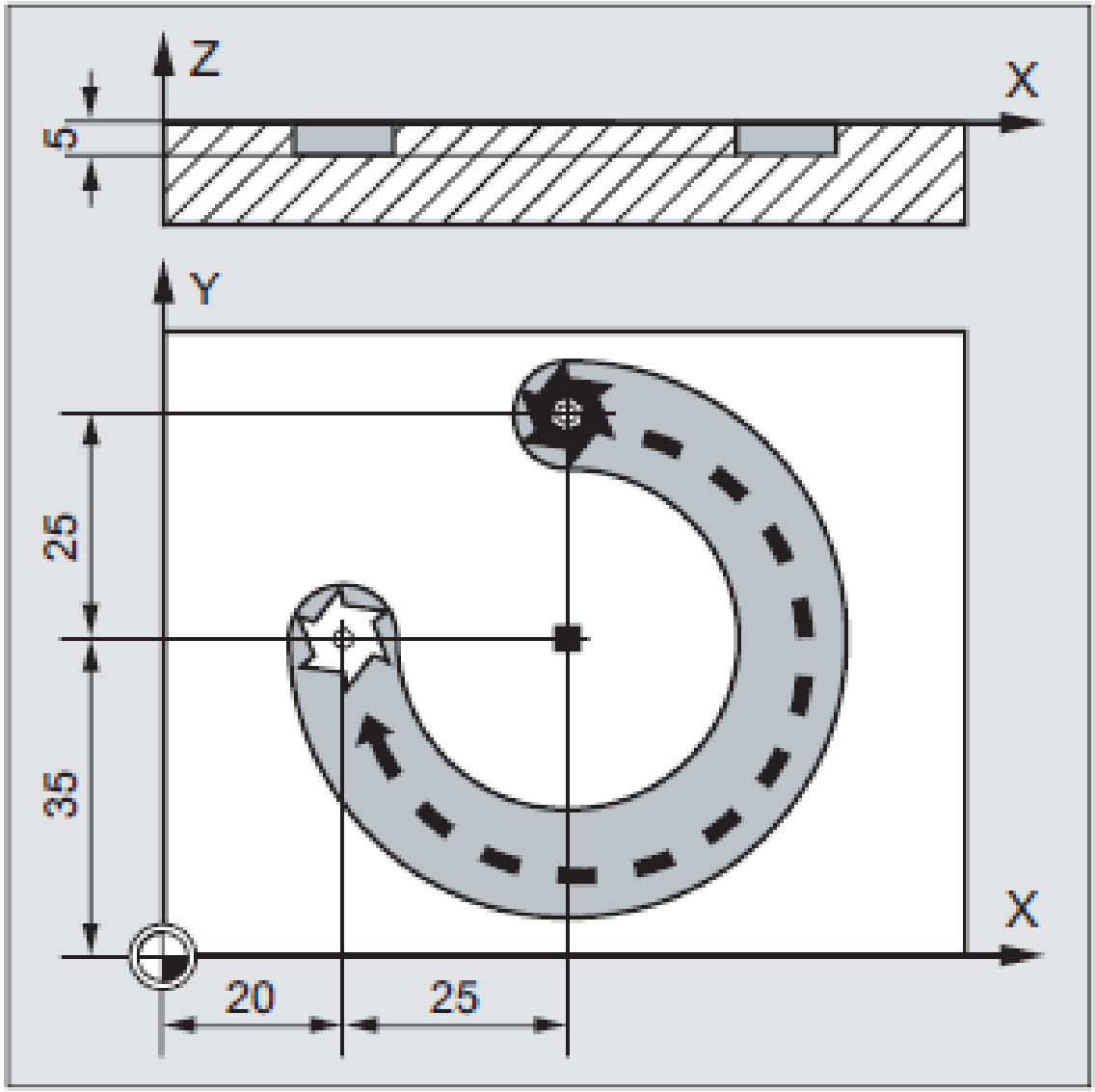
M30 – END OF PROGRAMME

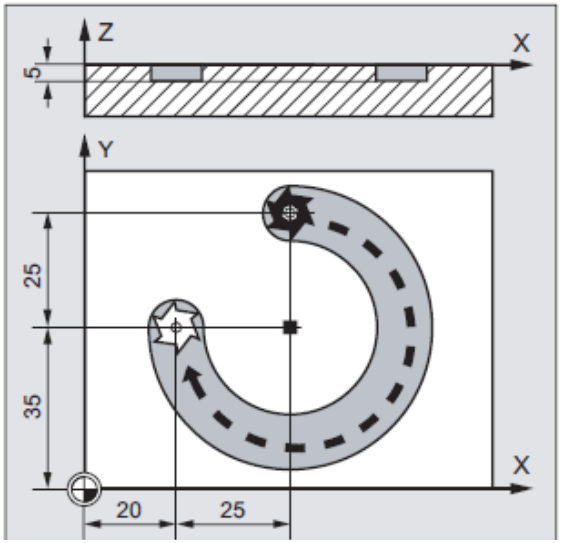
Milling:



Turning:




Z
X
5
Y
25
35
X
20
25



Program code	Comment
N10 G90 G0 X45 Y60 Z2 T1 S2000 M3	; Absolute dimension input, in rapid traverse to position XYZ, tool selection, spindle on with clockwise direction of rotation
N20 G1 Z-5 F500	; Linear interpolation, feed of the tool.
N30 G2 X20 Y35 I0 J-25	; Clockwise circular interpolation, circle end point in absolute dimensions, circle center point in incremental dimensions.
N40 G0 Z2	; Traverse
N50 M30	; End of block

Absolute/incremental dimensioning (G90, G91)

Inch/metric input (G70/G71) or (G20/G21)

N10 T1 M6 ; Loading of tool T1

N20 G96 S100 LIMS=2500 ;

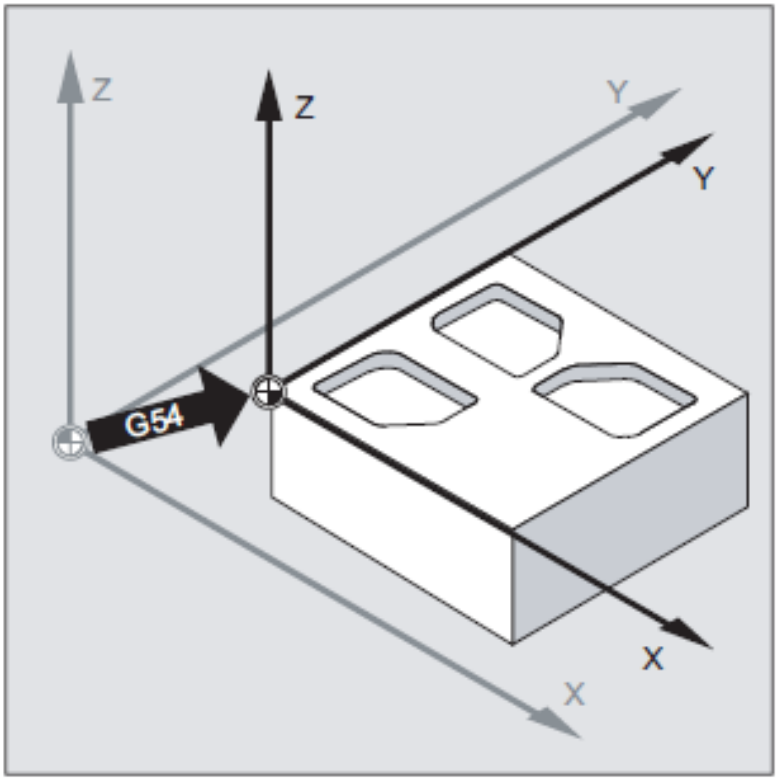
Explanation

Constant cutting rate = 100 m/min,

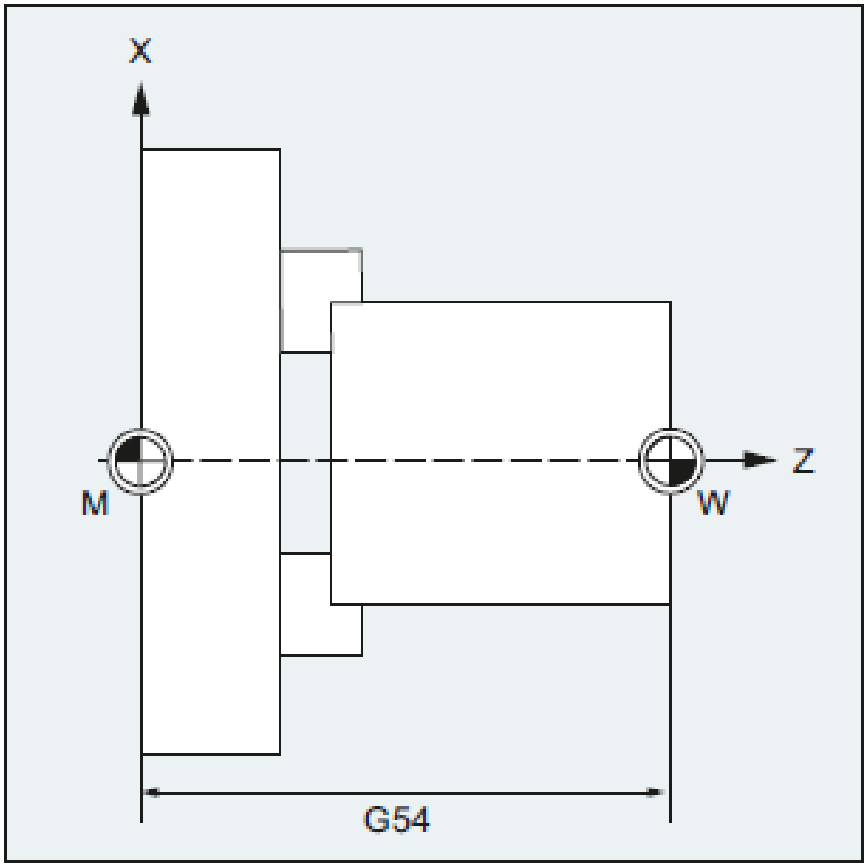
max. speed 2,500 rpm

Settable zero offset – G54

Milling:

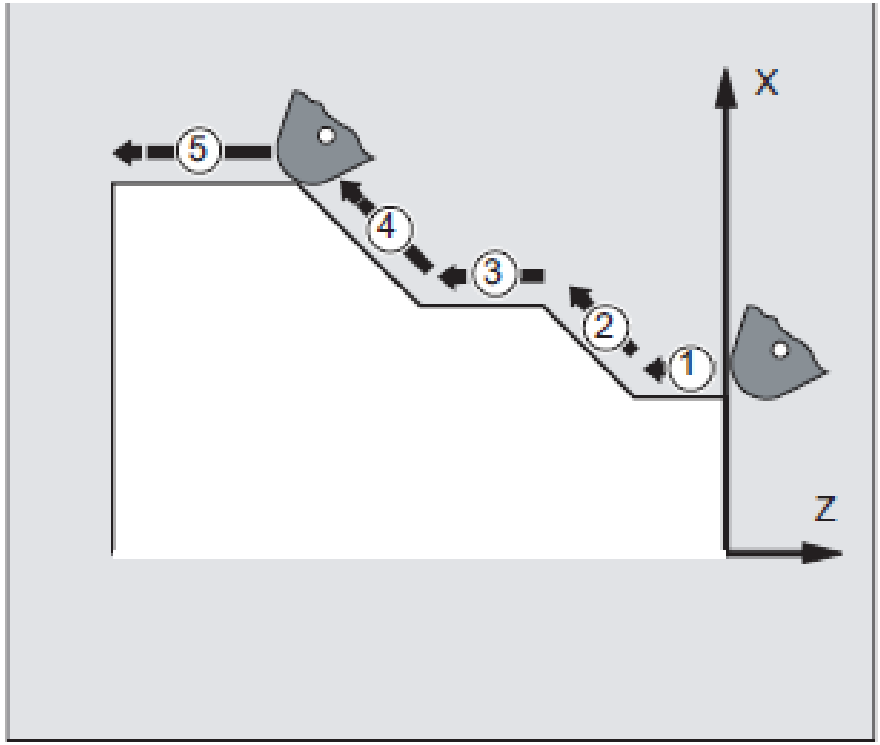


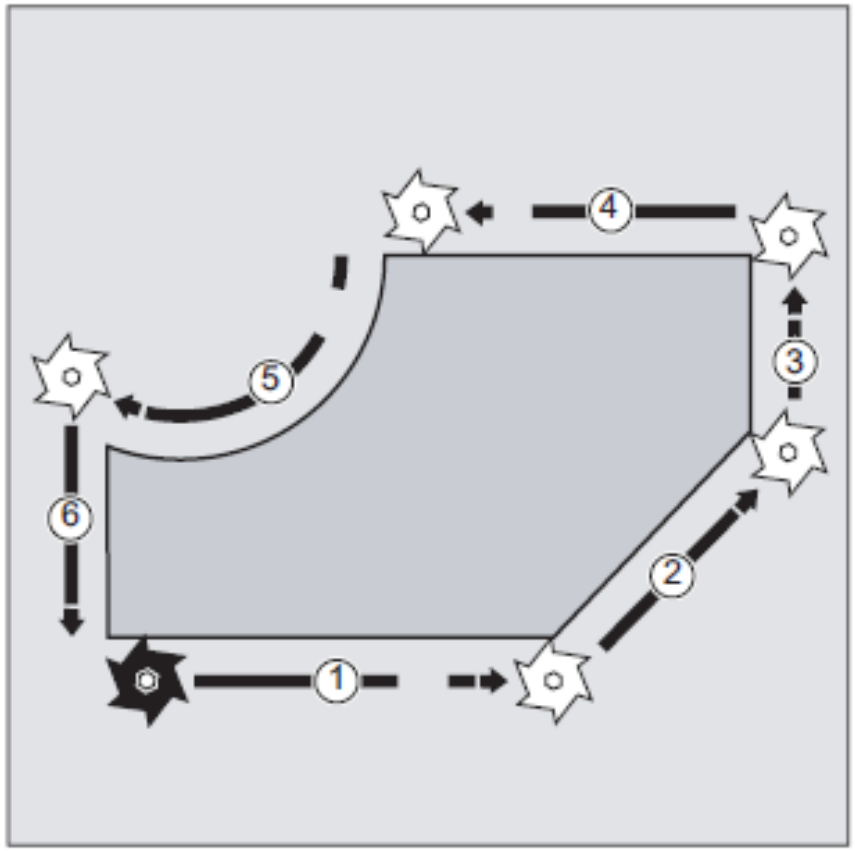
Turning:



EXTENDED WORD ADDRESSES

ANG = to assign exterior angle
AR = apex angle
AC = absolute coordinate
AP = polar angle
CIP = circular interpolation
CT = circular tangent
CHF = chamfer (general use)
CHR = chamfer (base distance)
CR = radius for circular interpolation
CYCLE = machining cycle
GOTO = jump function
IC = incremental coordinate
LIMS = upper spindle speed limitation
MSG() = message (max 65 characters)
RND = rounding
RP = polar radius
SPOS = spindle position in degrees

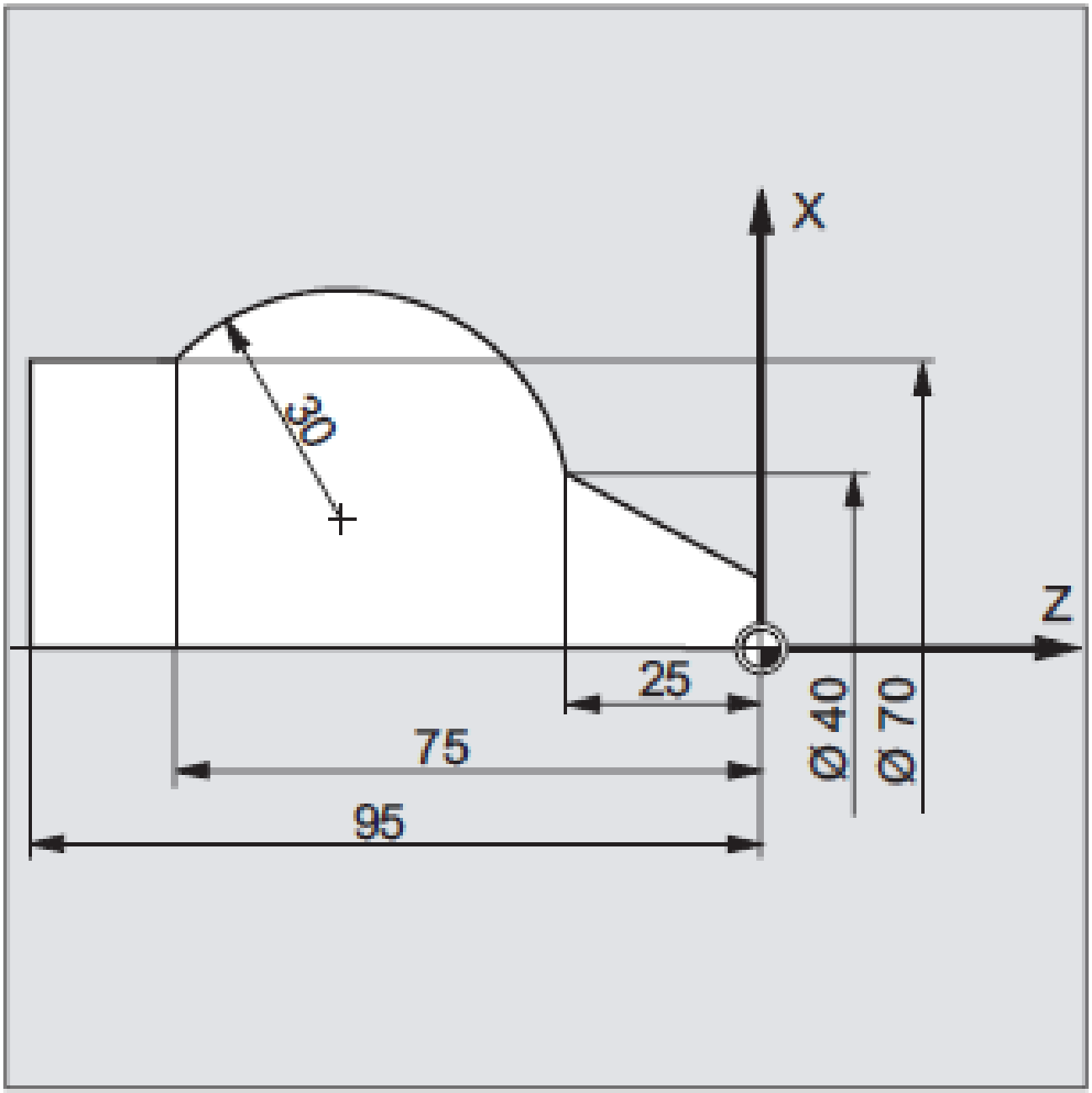


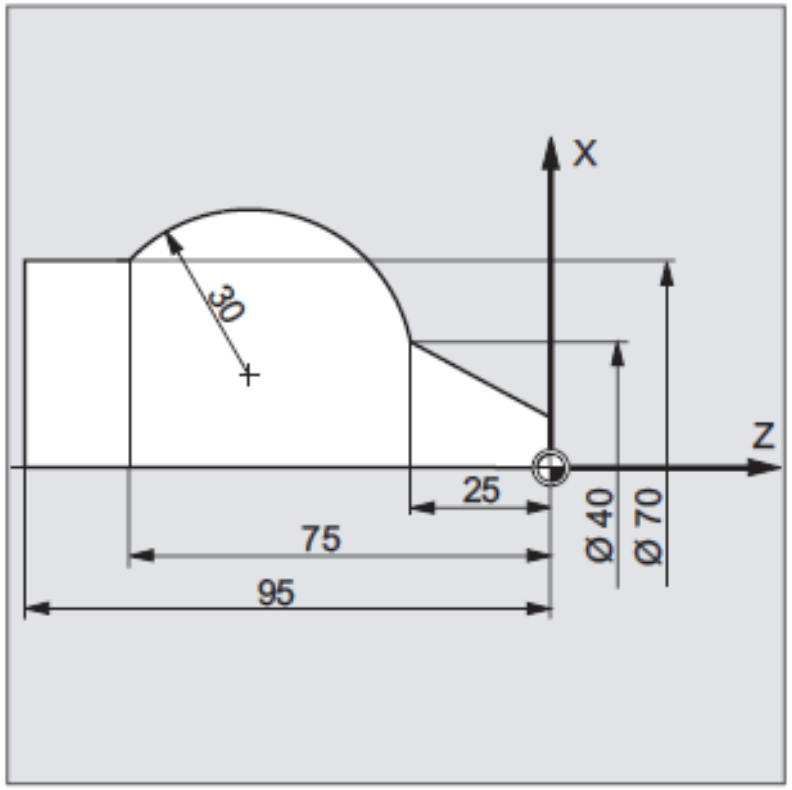


TURNING

MILLING

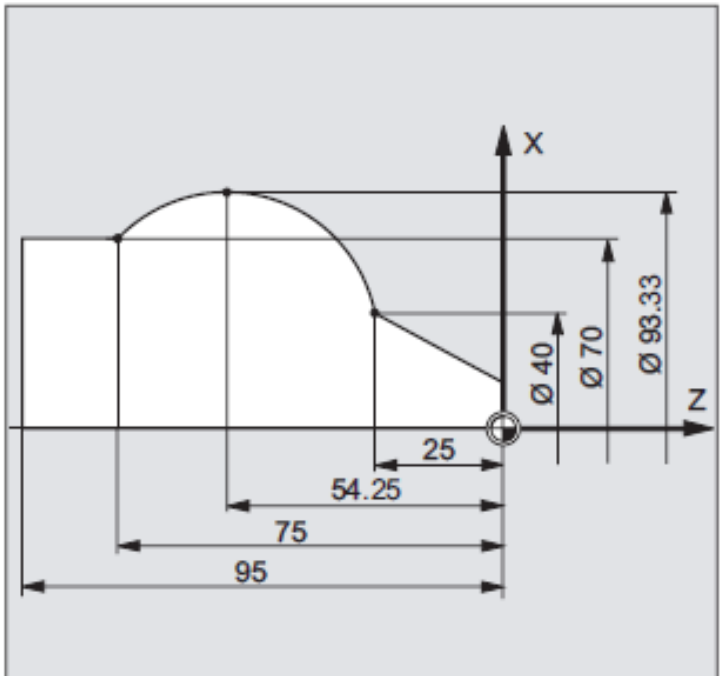
EXAMPLE



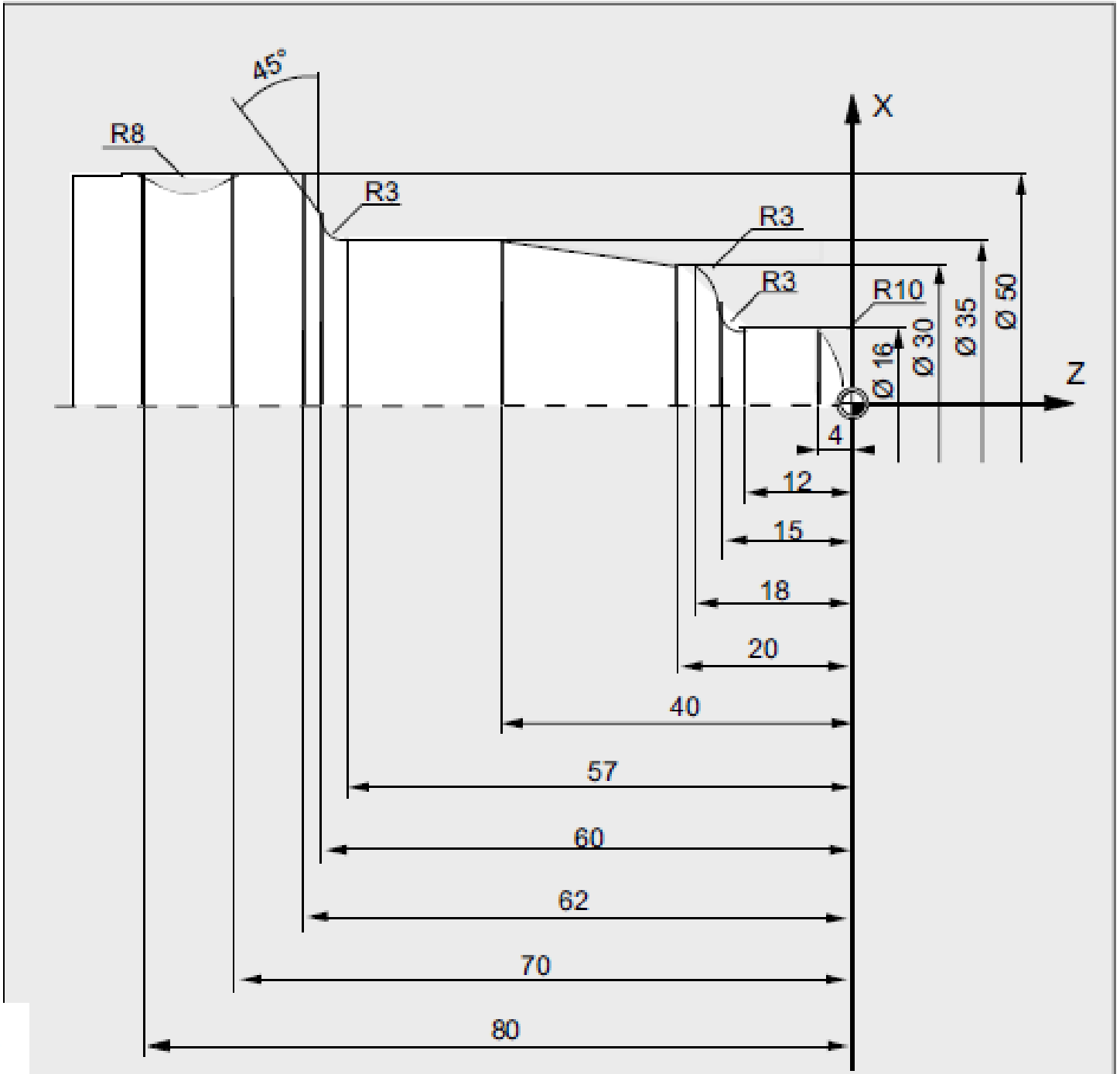


Program code

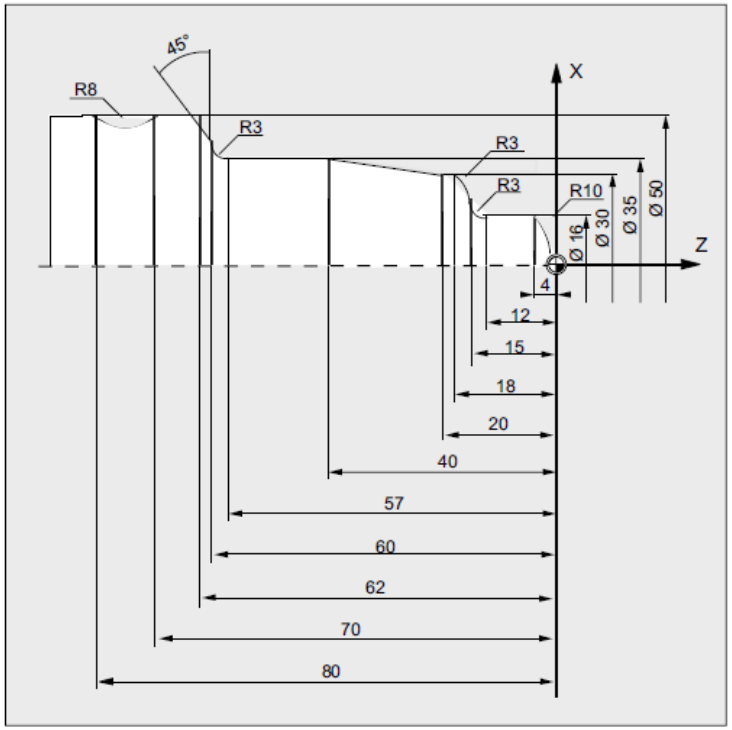
```
...
N125 G1 X40 Z-25 F0.2
N130 G3 X70 Z-75 CR=30
N135 G1 Z-95
...
```




```
N130 CIP X70 Z-75 I1=93.33 K1=-54.25
N135 G1 Z-95
```



Top view



Program code	Comment
N5 G0 G53 X280 Z380 D0	; Starting point
N10 TRANS X0 Z250	; Zero offset
N15 LIMS=4000	; Speed limitation (G96)
N20 G96 S250 M3	; Select constant cutting rate
N25 G90 T1 D1 M8	; Select tool selection and offset
N30 G0 G42 X-1.5 Z1	; Set tool with tool radius compensation
N35 G1 X0 Z0 F0.25	
N40 G3 X16 Z-4 I0 K-10	; Turn radius 10
N45 G1 Z-12	
N50 G2 X22 Z-15 CR=3	; Turn radius 3
N55 G1 X24	
N60 G3 X30 Z-18 I0 K-3	; Turn radius 3
N65 G1 Z-20	
N70 X35 Z-40	
N75 Z-57	
N80 G2 X41 Z-60 CR=3	; Turn radius 3
N85 G1 X46	
N90 X52 Z-63	

```
N95 G0 G40 G97 X100 Z50 M9          ; Deselect tool radius compensation and approach
                                      tool change location
N100 T2 D2                          ; Call tool and select offset
N105 G96 S210 M3                    ; Select constant cutting rate
N110 G0 G42 X50 Z-60 M8             ; Set tool with tool radius compensation
N115 G1 Z-70 F0.12                  ; Turn diameter 50
N120 G2 X50 Z-80 I6.245 K-5         ; Turn radius 8
N125 G0 G40 X100 Z50 M9             ; Retract tool and deselect tool radius
                                      compensation
N130 G0 G53 X280 Z380 D0 M5         ; Approach tool change location
N135 M30                            ; End of program
```