Computer Aided Manufacturing

Dr.S.RAMABALAN,
PRINCIPAL,
E.G.S. PILLAY ENGINEERING COLLEGE,
NAGAPATTINAM.

Cleansing through Breathing

1 min

Recap and review of previous class

5 mins

Computer Aided Manufacturing

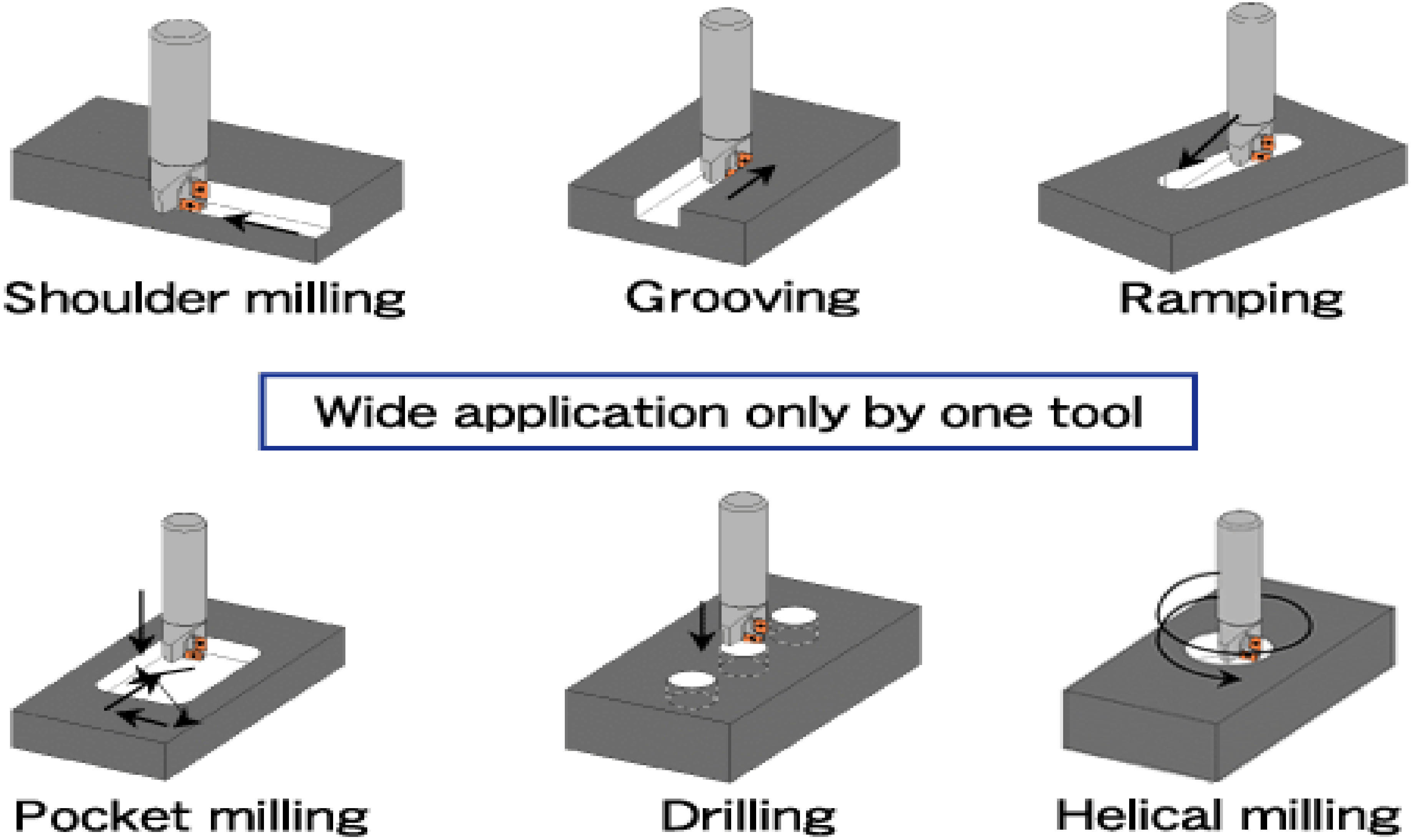
Unit 4

CNC Programming For Machining Centre operations

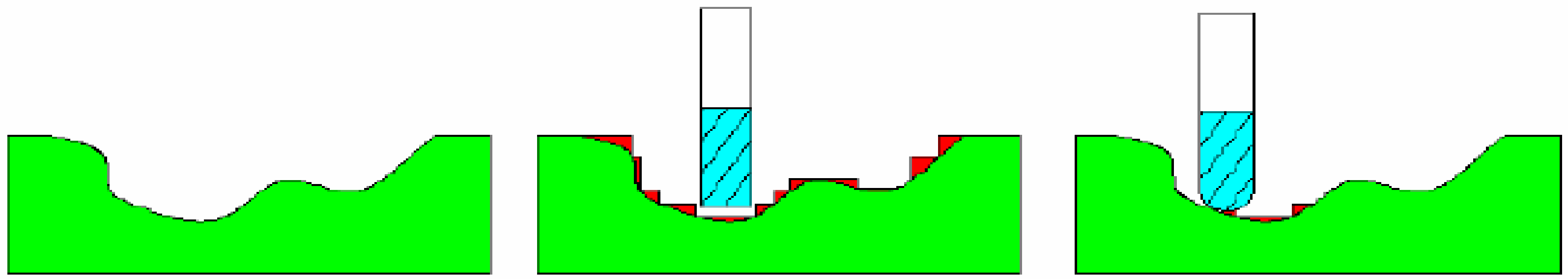
Coordinate system - G & M Codes for machining centre - Programming for FANUC and SIEMENS controller -Machining cycles - Linear and circular interpolation, Contouring, rectangular and circular pocketing, drilling, peck drilling, high speed drilling, Back boring, counter boring and tapping cycle - Cutter diameter compensation -Nomenclature of multi-point cutting tool and tool holder -Tool and work holding devices - Automatic Pallet changer.

CNC Programming – milling

Sequence of operation – Milling



Roughing vs. Finishing



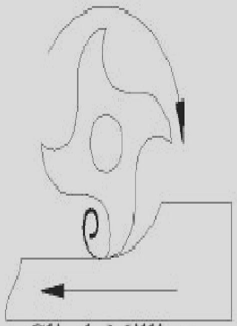
Desired Shape

Rough with Flat End-Tool

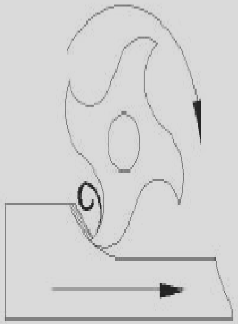
Finish with Ball End-Tool

Milling Direction

Climb milling has many advantages including better surface finish, longer tool life, and the cutter deflects away from the work rather than into it.

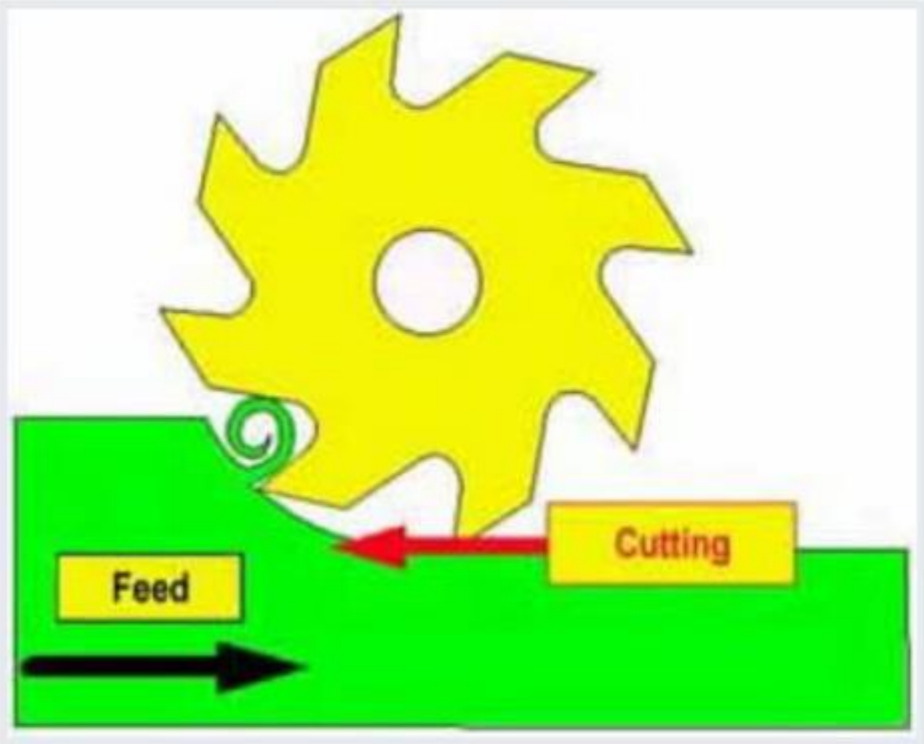


Climb Milling

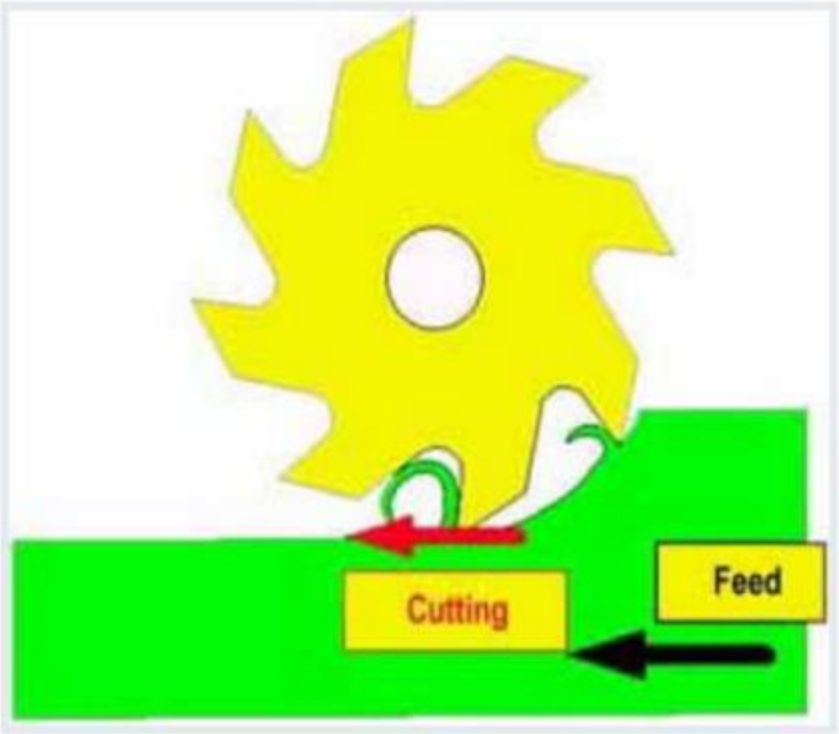


Conventional Milling

Up Milling & Down Milling



Up Milling

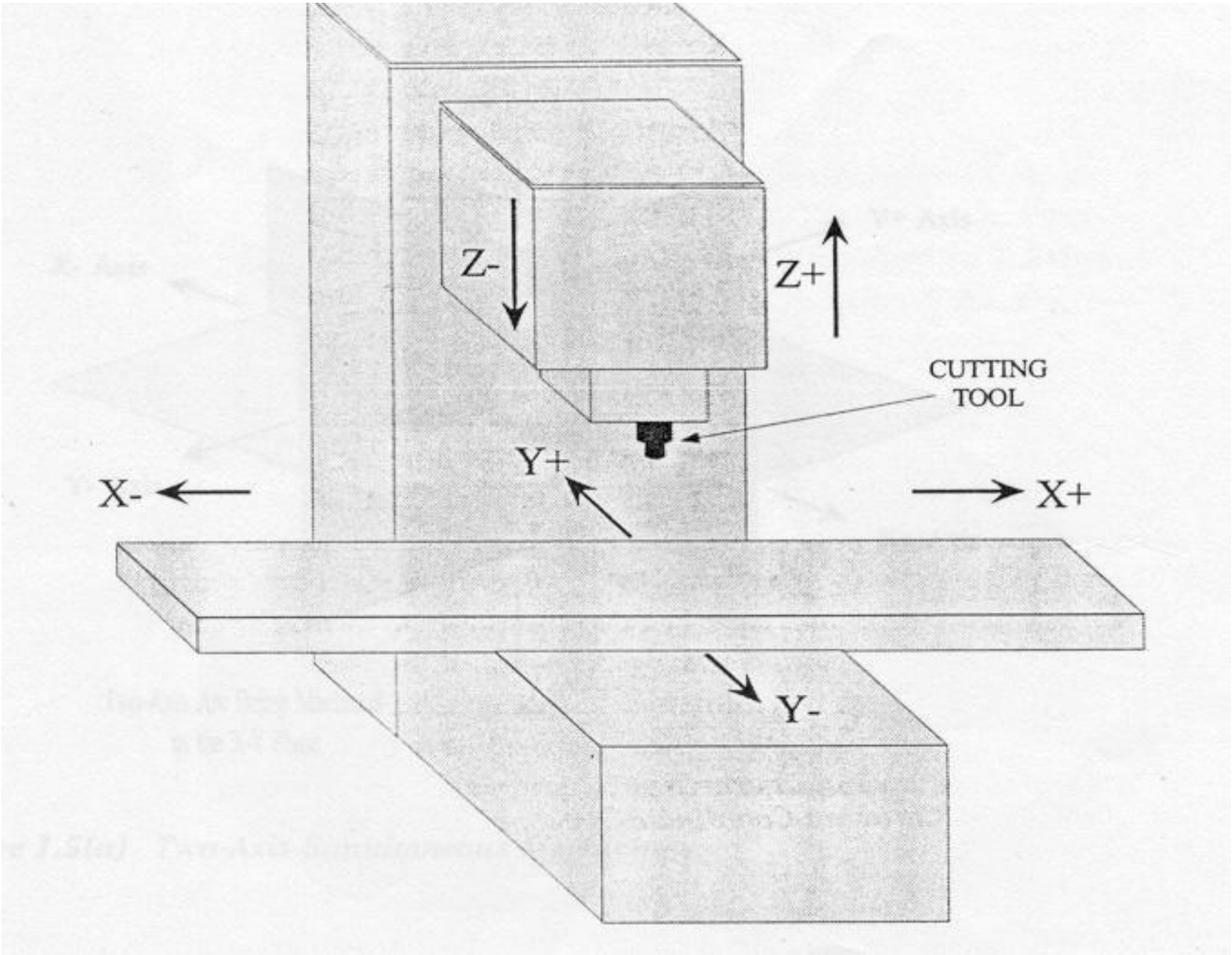


Down Milling

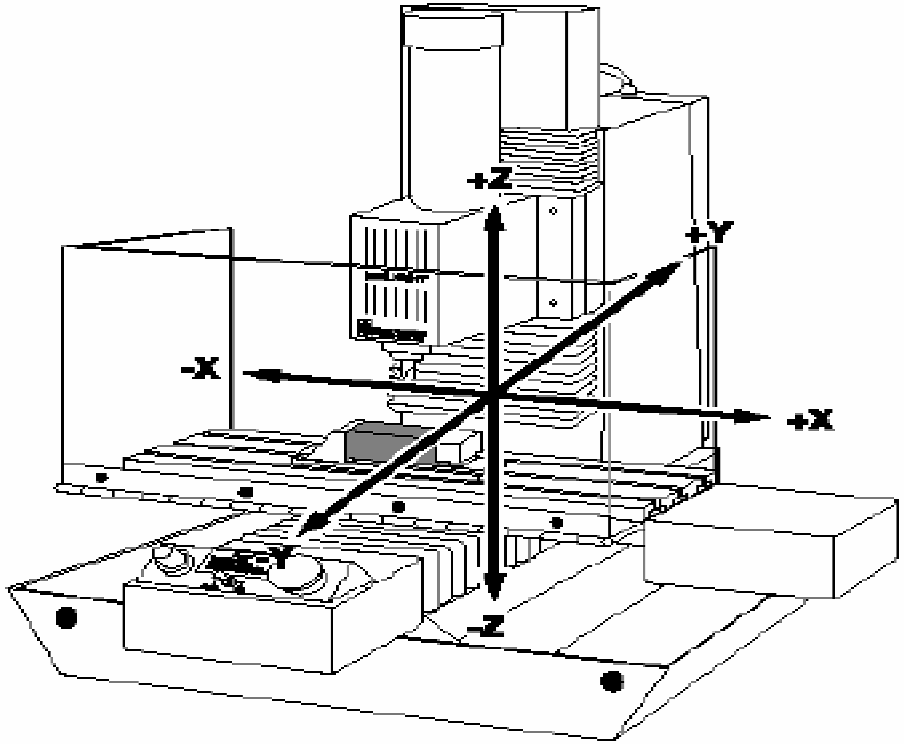
Comparison between Up Milling and Down Milling

SL. NO.	UP MILLING (CONVENTIONAL MILLING)	DOWN MILLING (CLIMB MILLING)
01	Work piece fed in the opposite direction that of the cutter.	Work piece fed in the same direction that of the cutter.
02	Chips are progressively thicker.	Chips are progressively thinner.
03	Strong clamping is required since the cutting force is directed upwards & tends to lift the work piece.	Strong clamping is not required since the cutting force is directed downwards & keep the work piece pressed to the table.
04	Gives poor surface finish, since chips gets accumulated at the cutting zone.	Gives good surface finish, since the chips are thrown away during cutting.
05	Used for hard materials.	Used for soft materials and finishing operations.

Designation of machine axis



Programming for Milling



Program as if the tool cutter moves, not the part

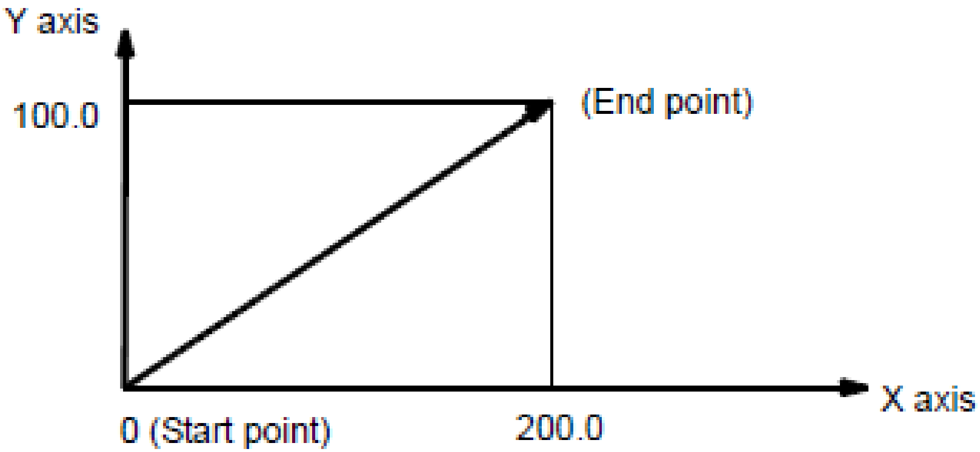
POSITIONING (G00)

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental programming at a rapid traverse rate.

LINEAR INTERPOLATION (G01)

G01 IP_ **F_** ;

(G91) G01X200.0Y100.0F200.0;

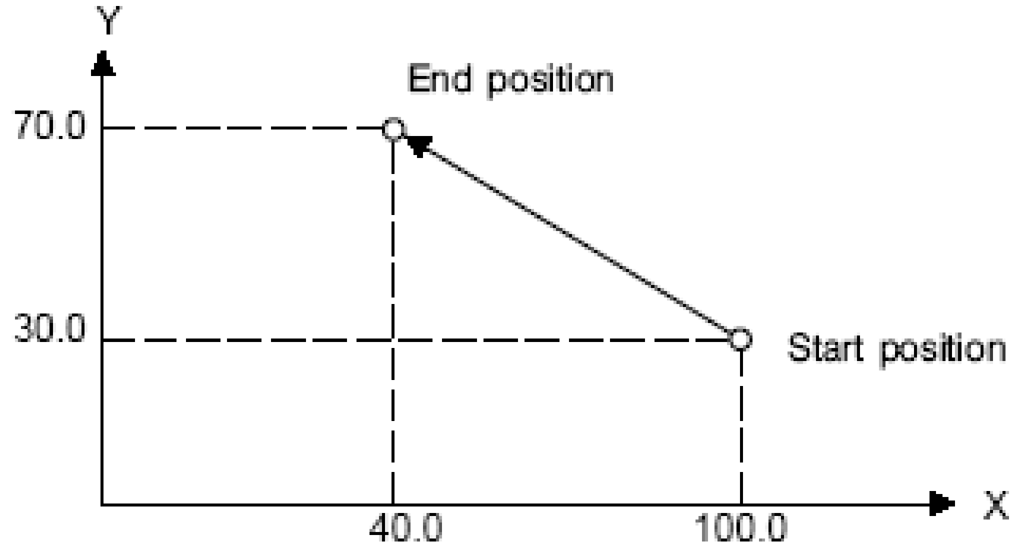


G00	01	Positioning (rapid traverse)	
G01		Linear interpolation (cutting feed)	
G02		Circular interpolation CW or helical interpolation CW	
G03		Circular interpolation CCW or helical interpolation CCW	
G04	00	Dwell, Exact stop	
G10		Programmable data input	
G11		Programmable data input mode cancel	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis
G18		ZpXp plane selection	Yp: Y axis or its parallel axis
G19		YpZp plane selection	Zp: Z axis or its parallel axis
G20	06	Input in inch	
G21		Input in mm	
G28	00	Automatic return to reference position	
G29		Movement from reference position	
G30		2nd, 3rd and 4th reference position return	
G31		Skip function	
G33	01	Threading	
G40	07	Cutter compensation : cancel	
G41		Cutter compensation : left	
G42		Cutter compensation : right	
G43	08	Tool length compensation +	
G44		Tool length compensation -	
G49	08	Tool length compensation cancel	
G52	00	Local coordinate system setting	
G53		Machine coordinate system setting	

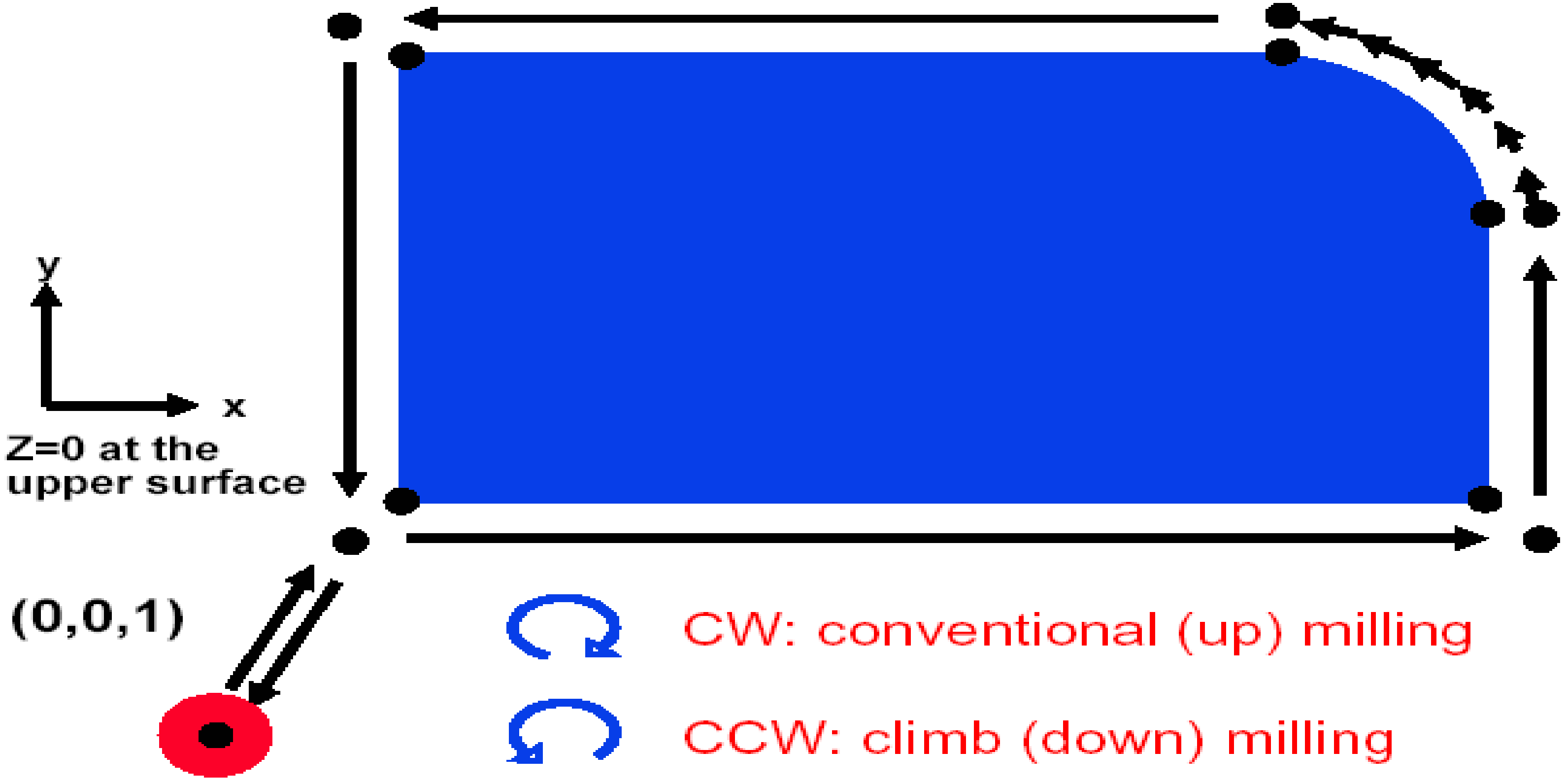
G54	14	Workpiece coordinate system 1 selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G73	09	Peck drilling cycle
G74		Left-handed tapping cycle
G75	01	Plunge grinding cycle (for grinding machine)
G76	09	Fine boring cycle
G77	01	Plunge direct sizing/grinding cycle (for grinding machine)
G78		Continuous-feed surface grinding cycle (for grinding machine)
G79		Intermittent-feed surface grinding cycle (for grinding machine)
G80	09	Canned cycle cancel Electronic gear box : synchronization cancellation
G81	09	Drilling cycle or spot boring cycle Electronic gear box : synchronization start
G82		Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G84.2		Rigid tapping cycle (FS10/11 format)
G84.3		Left-handed rigid tapping cycle (FS10/11 format)
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90	03	Absolute programming
G91		Incremental programming
G94	05	Feed per minute
G95		Feed per revolution
G98	10	Canned cycle : return to initial level
G99		Canned cycle : return to R point level

Absolute and incremental programming

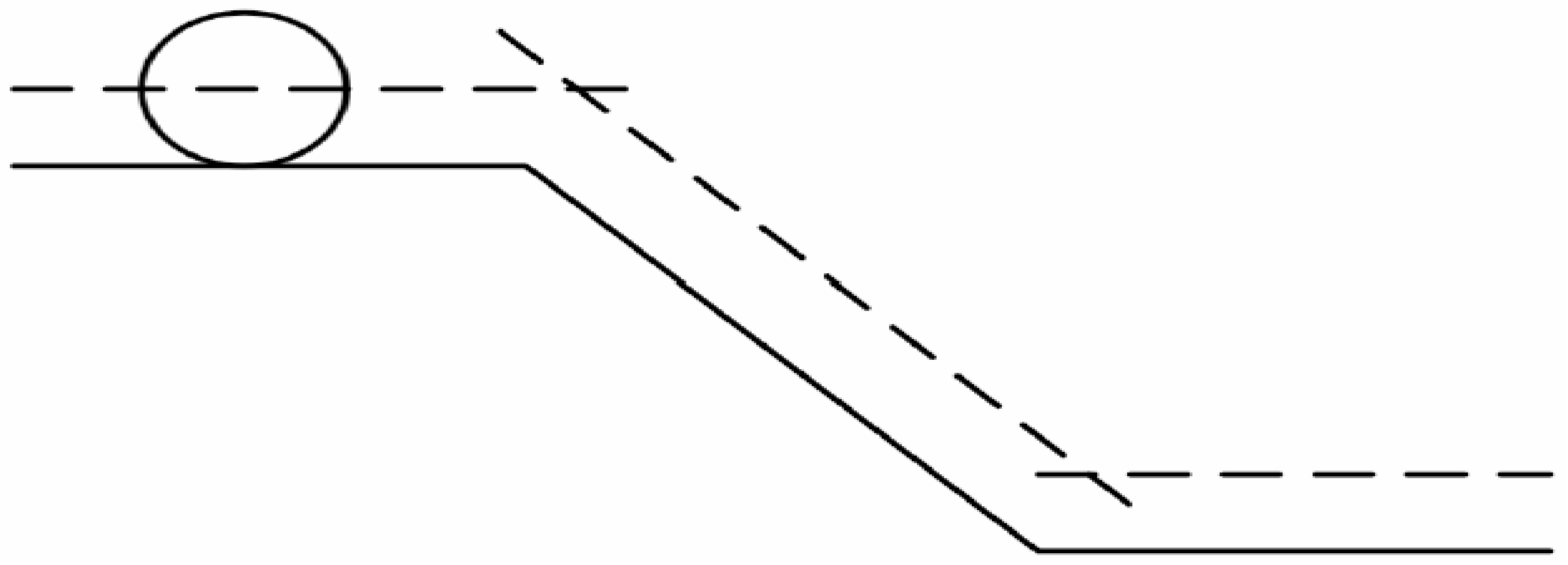
G90 X40.0 Y70.0 ;	Absolute command
G91 X–60.0 Y40.0 ;	Incremental command



Milling Direction

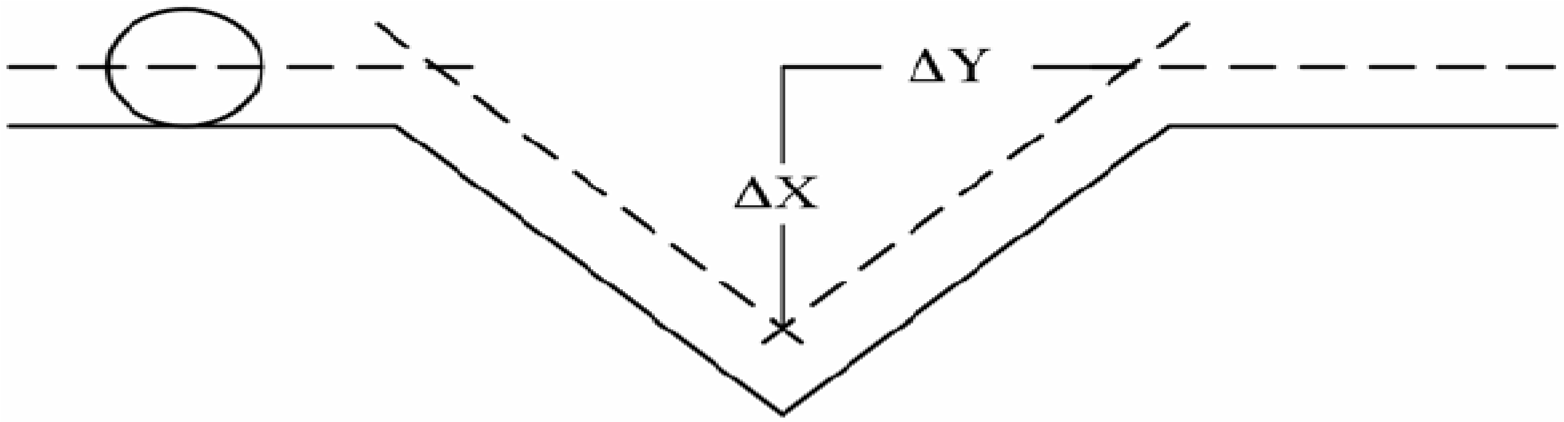


Milling Angles



Centerline paths are parallel to surfaces
Intersections must be calculated

Milling Angles



Multiple ways to find answer
Exploit this to check your answers
If $\Delta x \neq \Delta y$, taper isn't 45º

Programming for Milling

Coordinate Words

X___ Y___ Z___ ; end point coordinates

I___ J___ K___ ; arc center coordinates

U___ V___ W___ ; incremental coordinates in absolute programming mode

Programming for Milling

Circular Interpolation

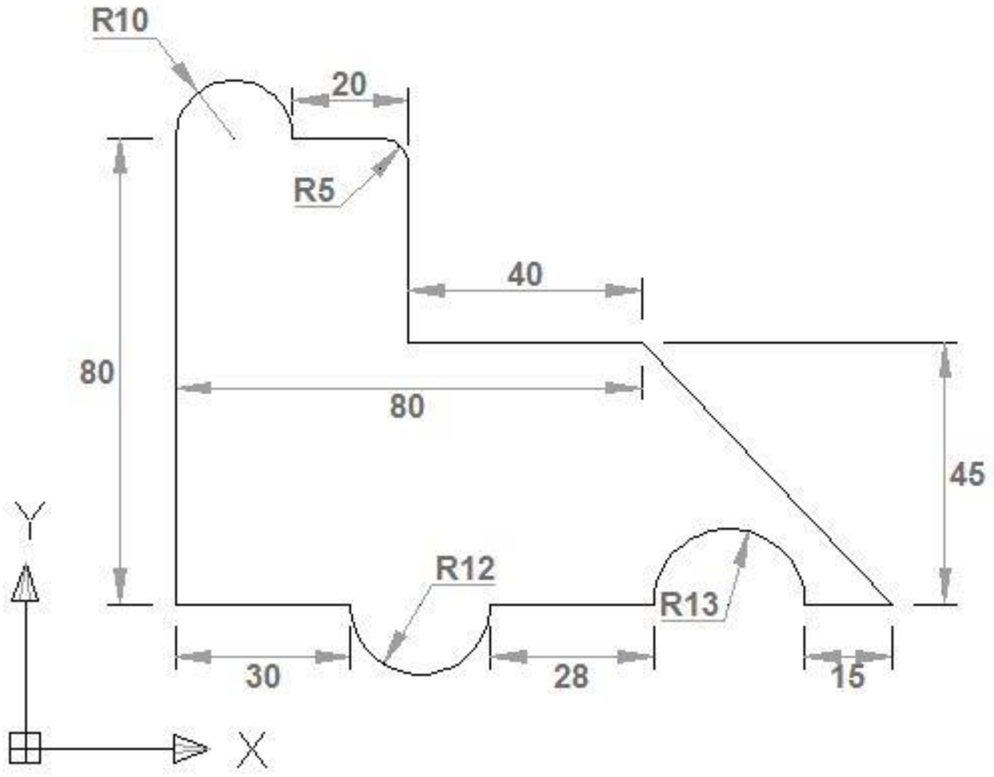
G02/G03 X___ Y___ I___ J___ F___ ;

(X,Y) – end point coordinate, (I, J) – arc center coordinate, F – feedrate

G02 – CW in positive Z direction

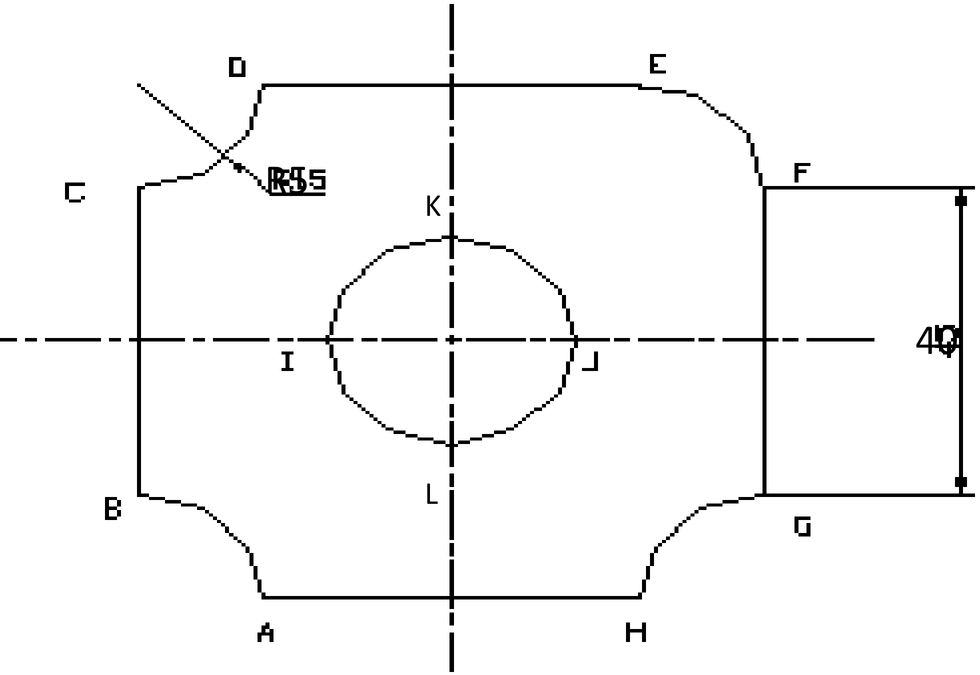
G03 – CCW in positive Z direction

```
G90 G21 M03 S1200
G00 X0 Y0
G01 Z-5
        X30
G03 X54 R12
G01 X82
G02 X108 R13
G01 X123
        X80 Y45
        X40
        Y75
G03 X35 Y80 R5
G01 X20
G03 X0 Y80 R10
G01 Y0
G01 Z10
M05
M30
```



CNCEZ MILLING - I

SPECIMEN DRAWING



A (5,0)   B(0,5) I(10,25)

C(0,45) D(5,50) J(40,25)

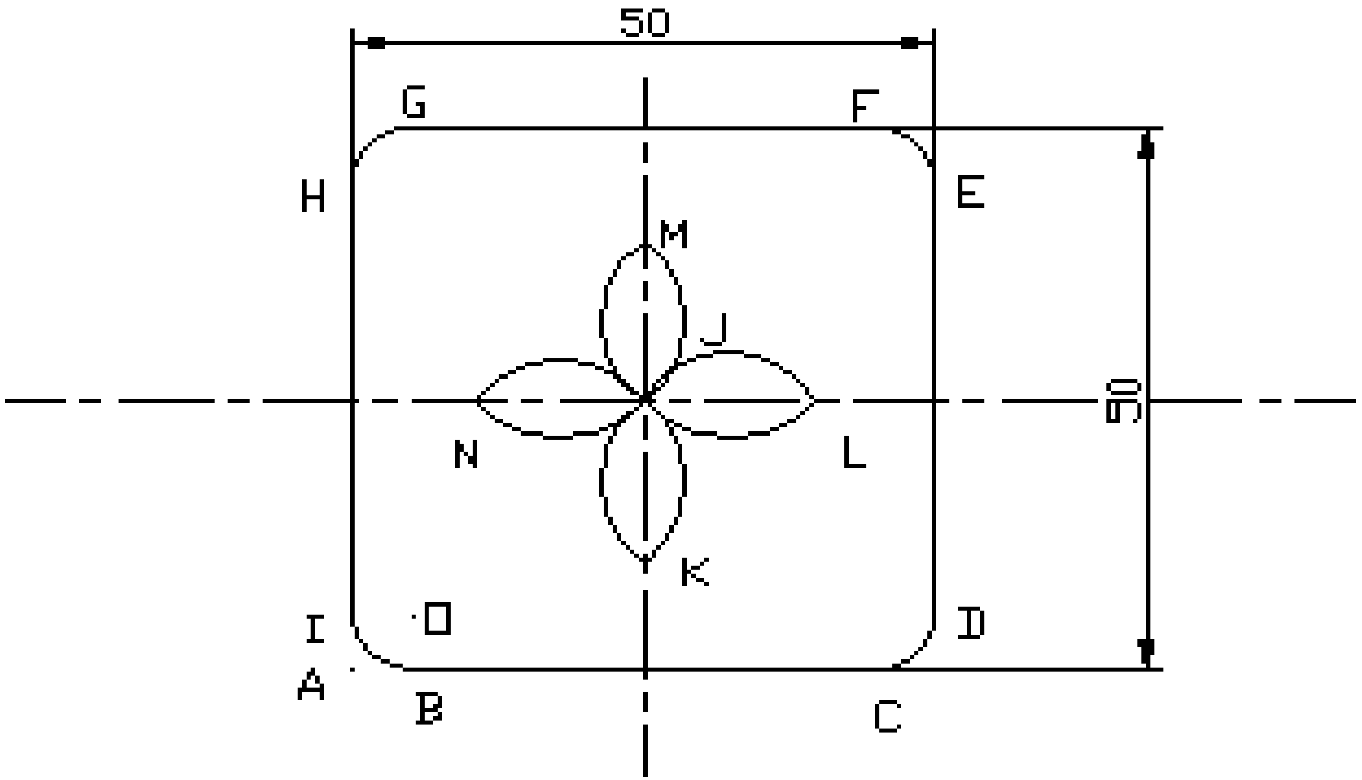
E(45,50) F(50,45) K(25,40)

G(50,5) H(45,0) L(25,10)

Program:

```
:%
: 1005
N05 G90 G21
N10 M06 T01
N15 M03 S1500
N20 G00 X5 Y0 Z5
N25 G01 Z-5 F10
N30 G01 X45
N35 G02 X50 Y5 R5
N40 G01 X50 Y45
N45 G03 X45 Y50 R5
N50 G01 X5 Y50
N55 G02 X0 Y45 R5
N60 G01 X0 Y5
N65 G02 X5 Y0 R5
N70 G01 Z10
N75 G00 X10 Y25
N80 G01 Z-5
N85 G02 X25 Y40 R15
N90 G02 X40 Y25 R15
N95 G02 X25 Y10 R15
N100 G02 X10 Y25 R15
N105 G01 Z10
N110 G00 X0 Y0 Z0
N115 M05
N120 M30
```

CNCEZ MILLING – II
SPENIMEN DRAWING



A(0,0) B(5,0)
C(45,0) D(50,5)
E(50,45) F(45,50)
G(5,50) H(0,45)
I(0,5) J(25,25)
K(25,10) L(40,25)
M(25,40) N(10,25)

All Dimensions Are In mm

Program:

```
:%
: 1006
N05 G90 G21
N10 M06 T01
N15 M03 S1500
N20 G00 X5 Y0
N25 G01 Z-0.5 F10
N30 G01 X45
N35 G03 X50 Y5 R5
N40 G01 X50 Y45
N45 G03 X45 Y50 R5
N50 G01 X5 Y50
N55 G03 X0 Y45 R5
N60 G01 X0 Y5
N65 G03 X5 Y0 R5
N70 G01 Z10
N75 G00 X10 Y25
N80 G01 Z-0.5
N85 G03 X25 Y25 R10.61
N90 G03 X25 Y40 R10.61
N95 G03 X25 Y25 R10.61
N100 G03 X40 Y25 R10.61
N105 G03 X25 Y25 R10.61
N110 G03 X25 Y10 R10.61
N115G03 X25 Y25 R10.61
N120 G03 X10 Y25 R10.61
N125 G01 Z10
N130 G00 X0 Y0
N135 M05 M30
```

Automatic reference position return (G28)

(Example)

N1 G28 X40.0 ;

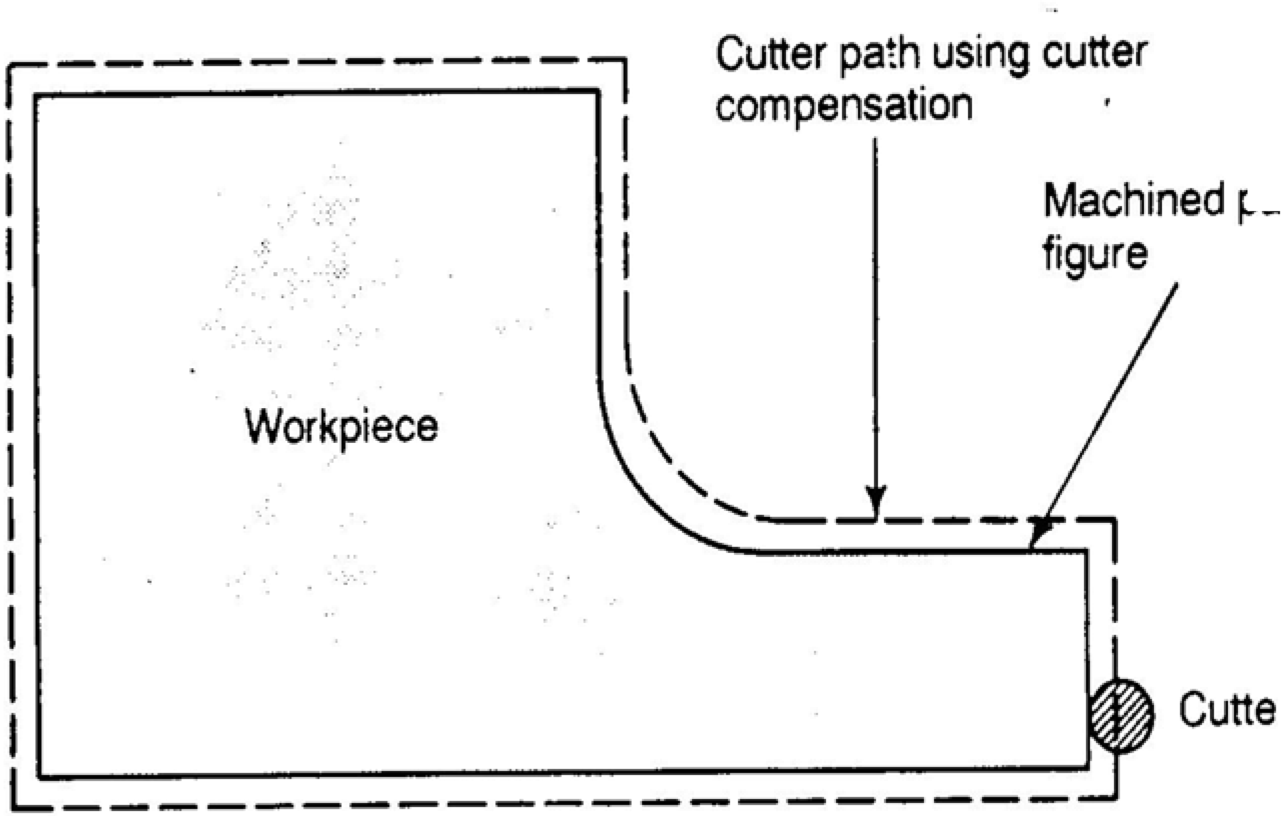
(The tool moves to the reference position along the X-axis and the intermediate position (X40.0) is stored.)

N2 G28 Y60.0 ;

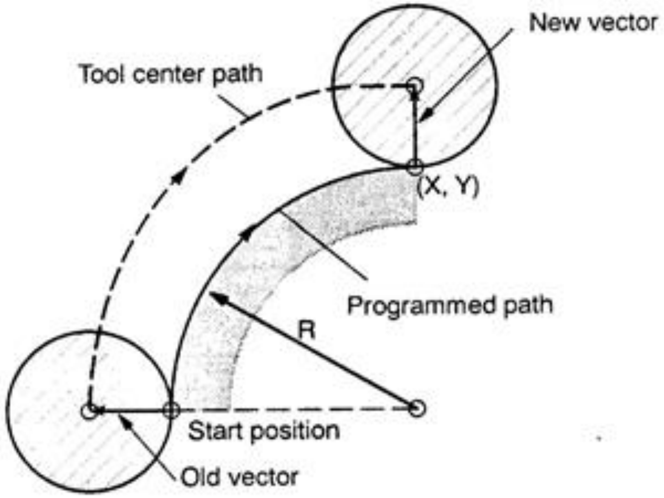
(The tool moves to the reference position along the Y-axis and the intermediate position (Y60.0) is stored.)

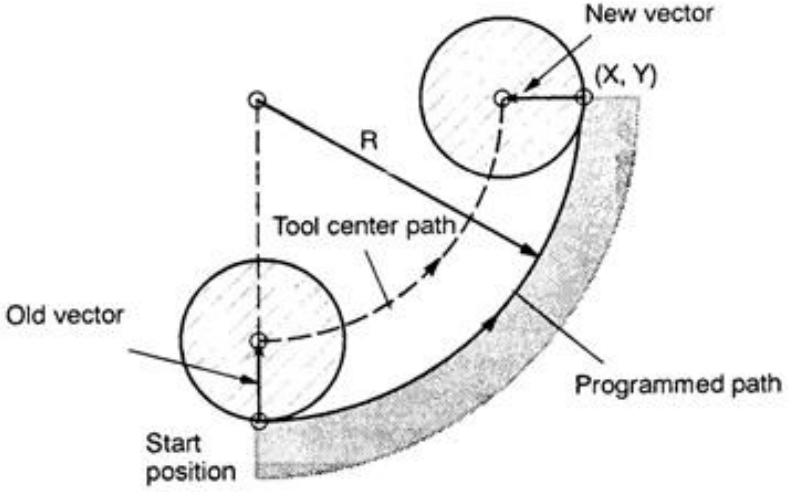
N3 G29 X10.0 Y20.0 ;

Cutter radius compensation



G41 - Cutter radius compensation left

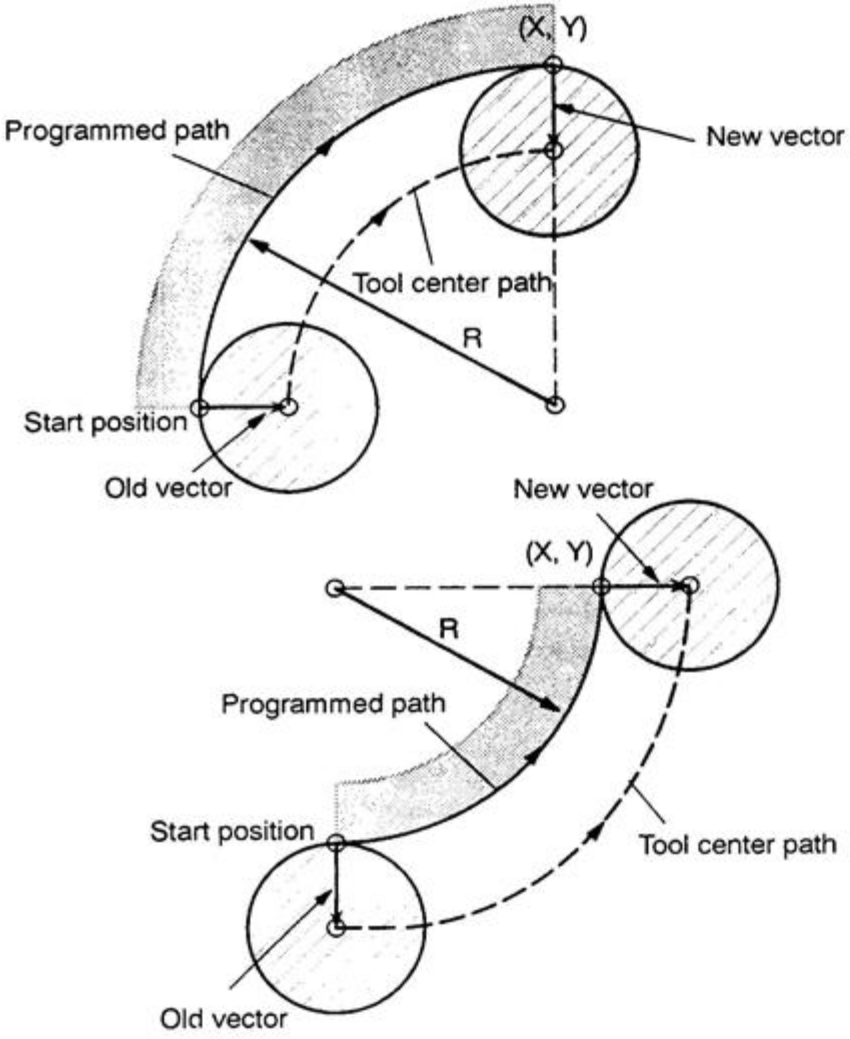




G41 D07;

Here, D specifies the address of offset at which the radius of tool will be mentioned

G42 - Cutter radius compensation right

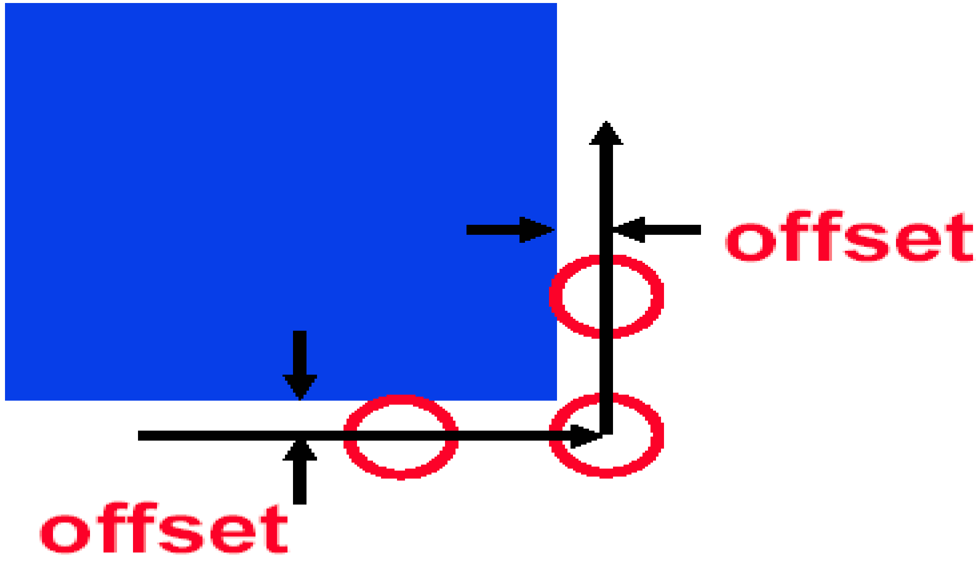


G42 D07;

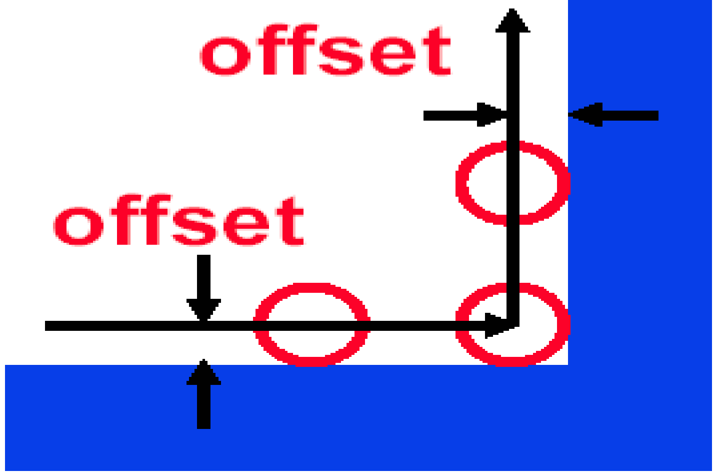
Here, D specifies the address of offset at which the radius of tool will be mentioned

CAM - General Capabilities

cutter offset = 1/2 cutter diameter

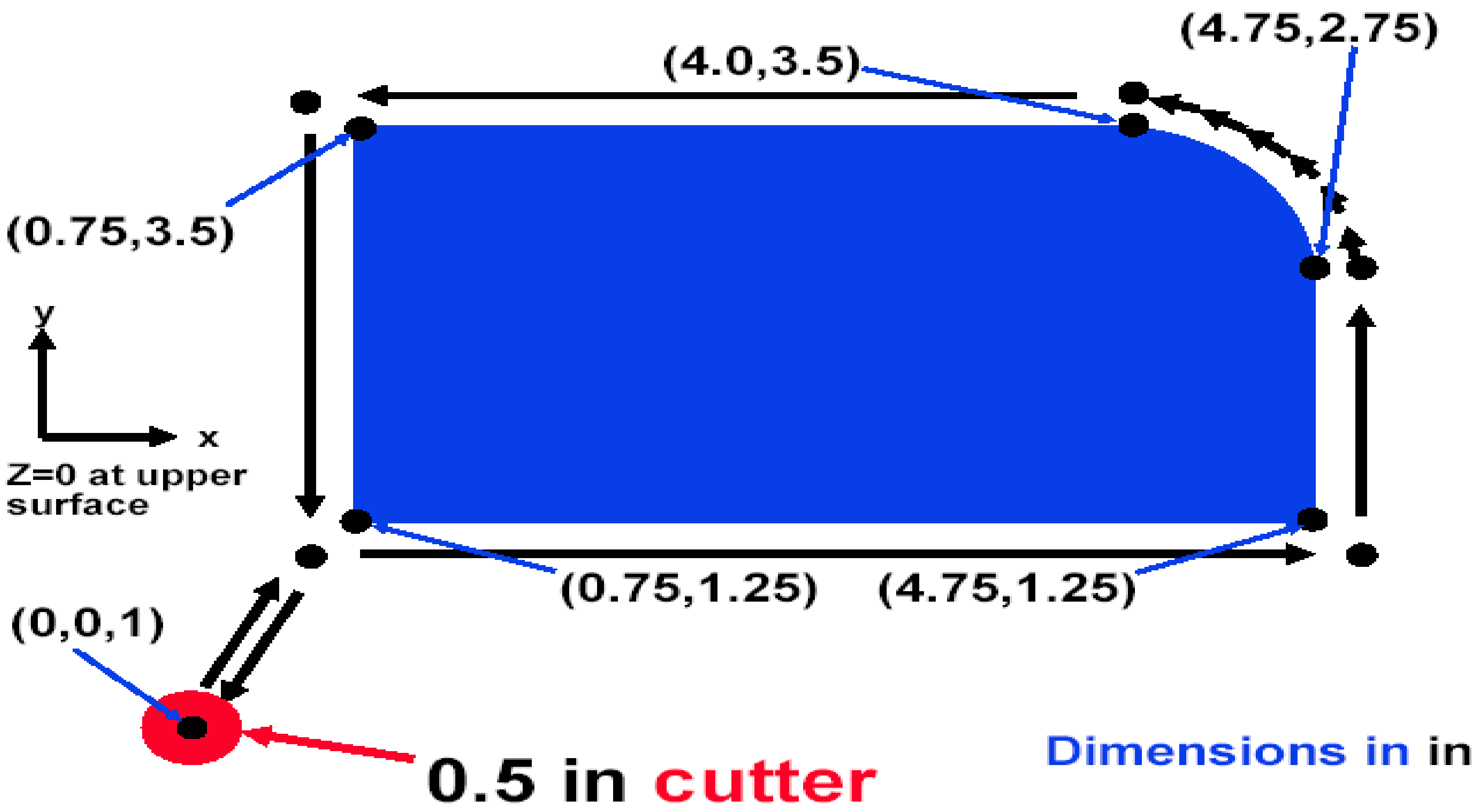


Move past surface
on outside corner.

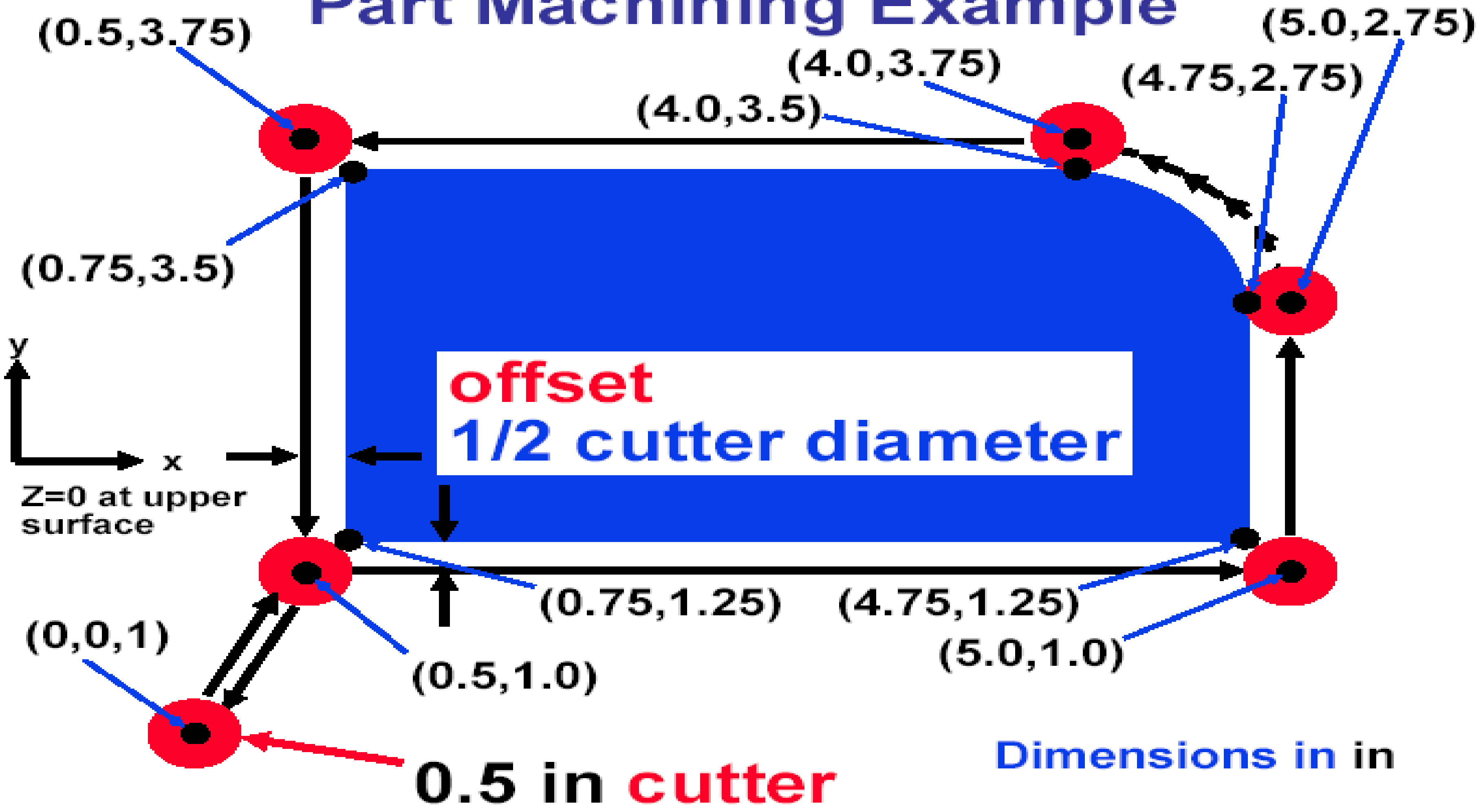


Move to surface
on inside corner.

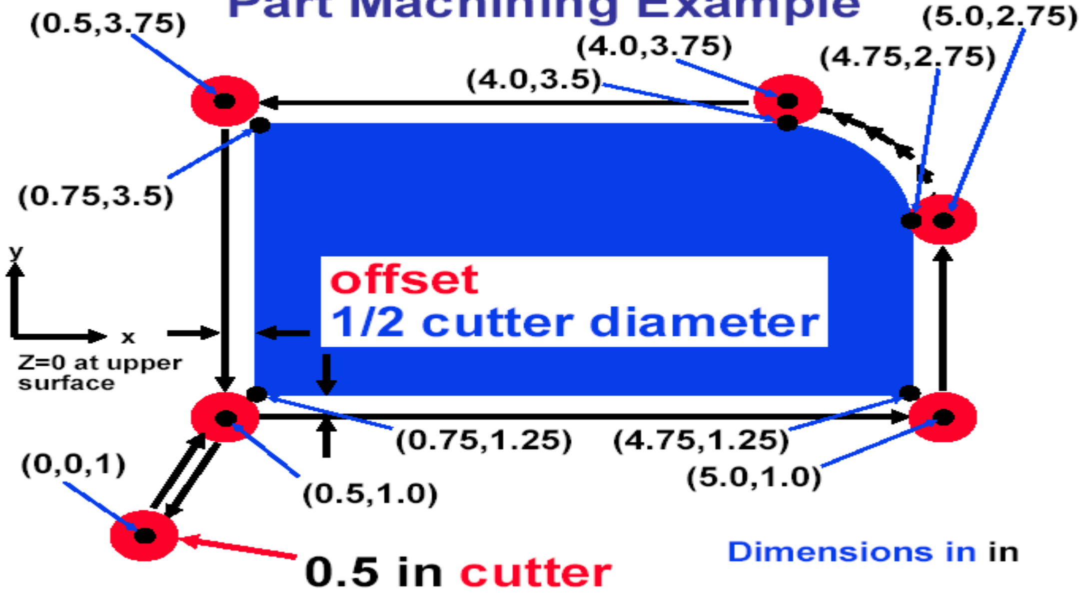
Part Machining Example



Part Machining Example



Part Machining Example



NC Program Example

```
N001 G90                              ;absolute r
N002 M03 S1000                        ;spindle on, 1000 RPM

N003 G00 X0.5 Y1.0 Z0.05              ;rapid traverse
N004 G01 X0.5 Y1.0 Z-0.6 F10          ;advance into part (-z)

N005 G01 X5.0 Y1.0 Z-0.6 F10          ;cut in +x direction
N006 G01 X5.0 Y2.75 Z-0.6 F10         ;cut in +y direction
N007 G03 X4.0 Y3.75 I4.0 J2.75 F10    ;ccw circular interp.(90º)
N008 G01 X0.5 Y3.75 Z-0.6 F10         ;cut in -x direction
N009 G01 X0.5 Y1.0   Z-0.6 F10        ;cut in -y direction

N010 G01 X0.5 Y1.0 Z0.0 F10           ;retract out of part (+z)
N011 G00 X0.0 Y0.0 Z0.0               ;rapid traverse

N012 M05                              ;spindle off
N013 M30                              ;end of program
```

NC Program Decoding (on machine)

previous block

N003 G00 X0.5 Y1.0 Z0.05

no change in X,Y

N004 G01 X0.5 Y1.0 Z-0.6 F10

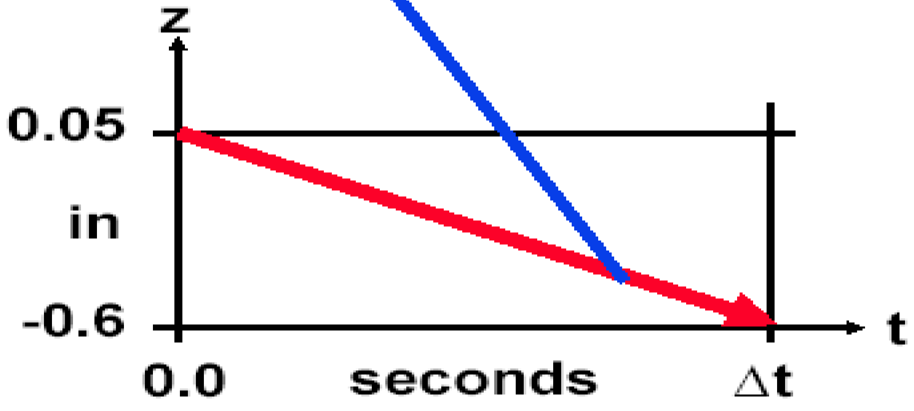
step (block) #4 in process

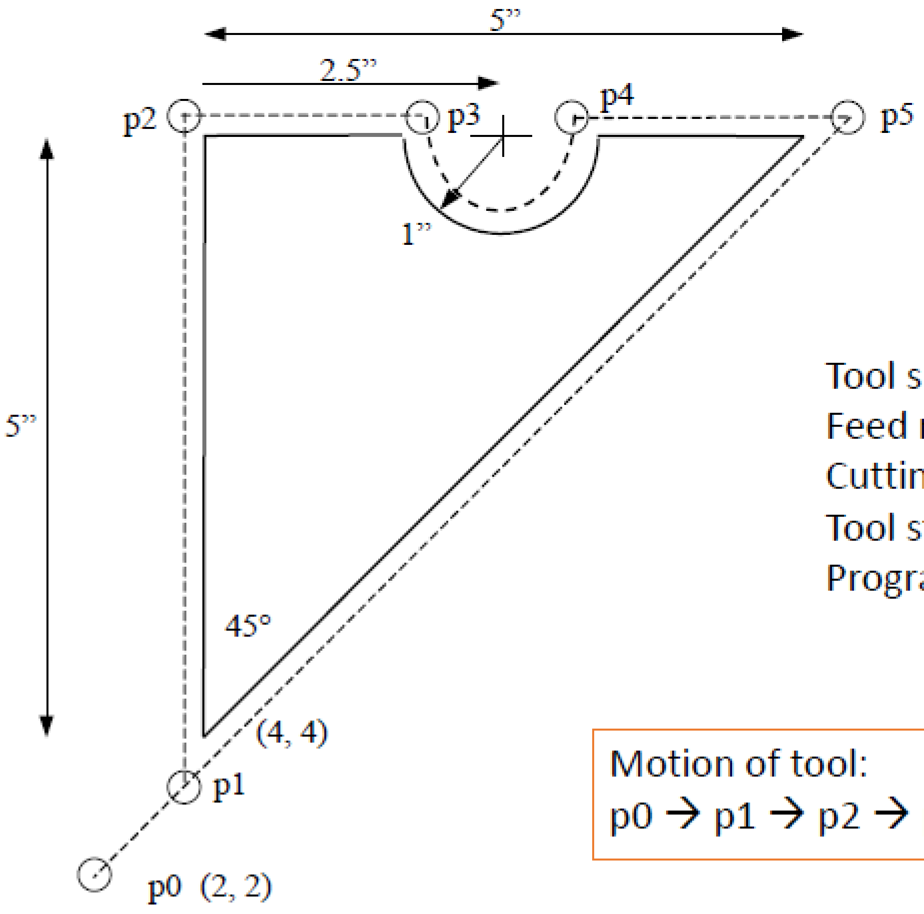
linear interpolation

10 in/min feedrate

$$\Delta t = 0.65\text{in}\ \frac{\text{min}}{10\ \text{in}}$$

$$\Delta t = 3.9\ \text{sec}$$

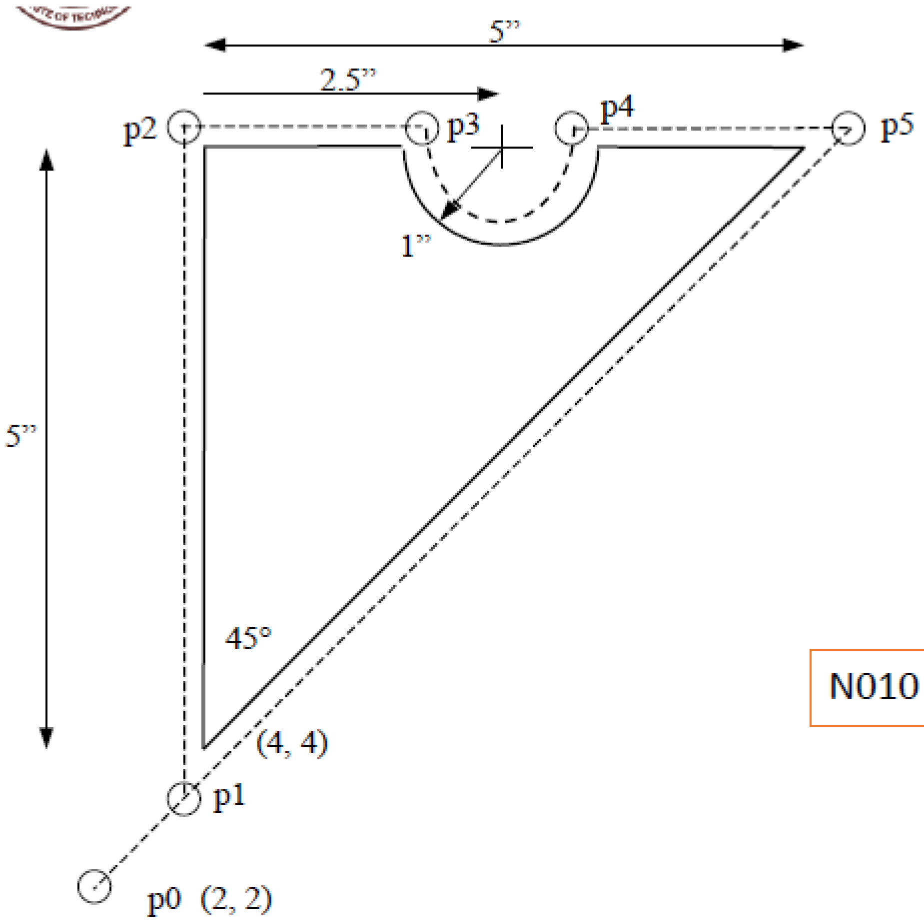




Tool size = 0.25 inch,
Feed rate = 6 inch per minute,
Cutting speed = 300 rpm,
Tool start position: 2.0, 2.0
Programming in inches

Motion of tool:
p0 → p1 → p2 → p3 → p4 → p5 → p1 → p0

1. Set up the programming parameters



Programming in inches

Use absolute coordinates

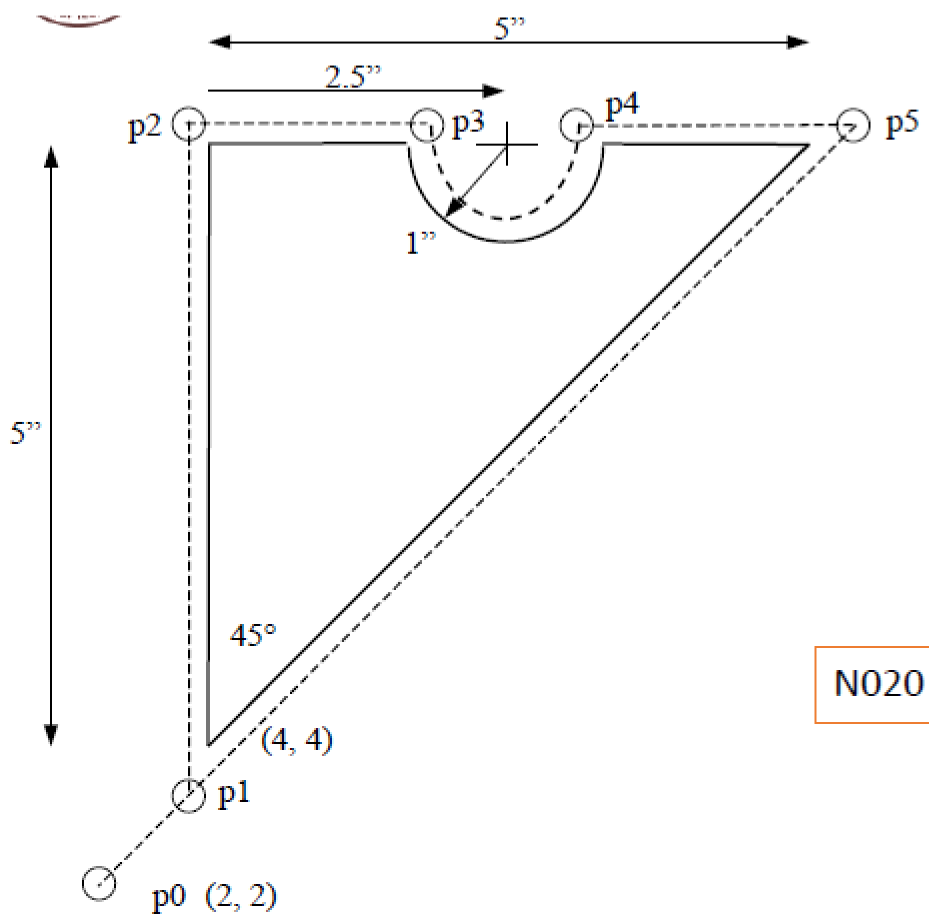
Feed in ipm

N010 G70 G90 G94 G97 M04

Spindle speed in rpm

Spindle CCW

2. Set up the machining conditions



Machine moves in XY-plane

Use full-circle interpolation

Feed rate

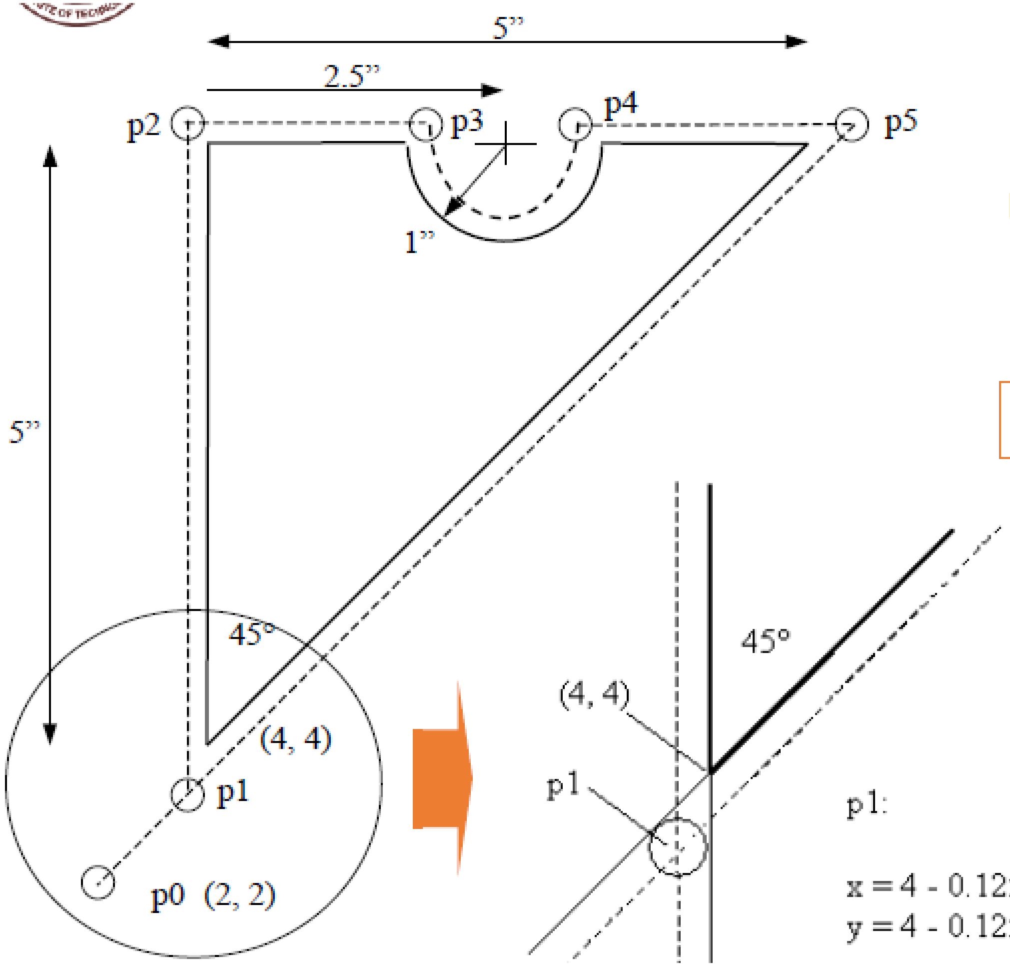
Spindle speed

```
N020 G17 G75 F6.0 S300 T1001 M08
```

Tool no.

Flood coolant ON

3. Move tool from p0 to p1 in straight line



Linear interpolation

target coordinates

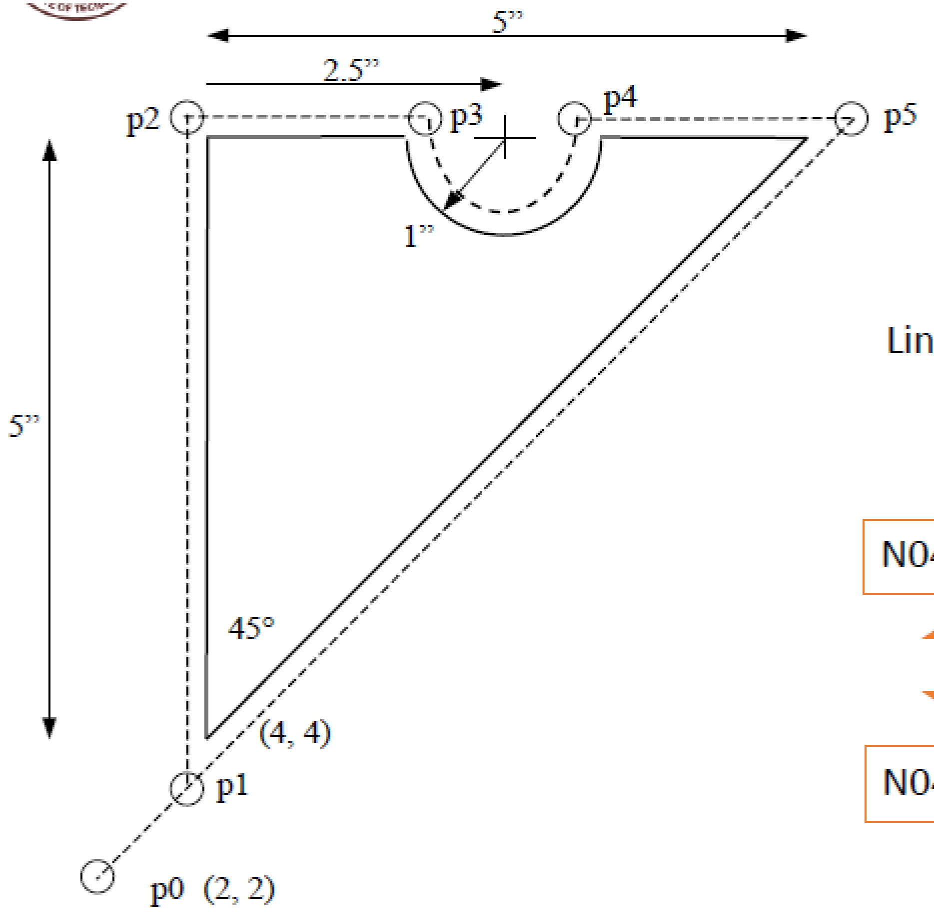
N030 G01 X3.875 Y3.698

p1:

$x = 4 - 0.125 = 3.875$

$y = 4 - 0.125/\tan 22.5 = 3.698$

4. Cut profile from p1 to p2



Linear interpolation

target coordinates

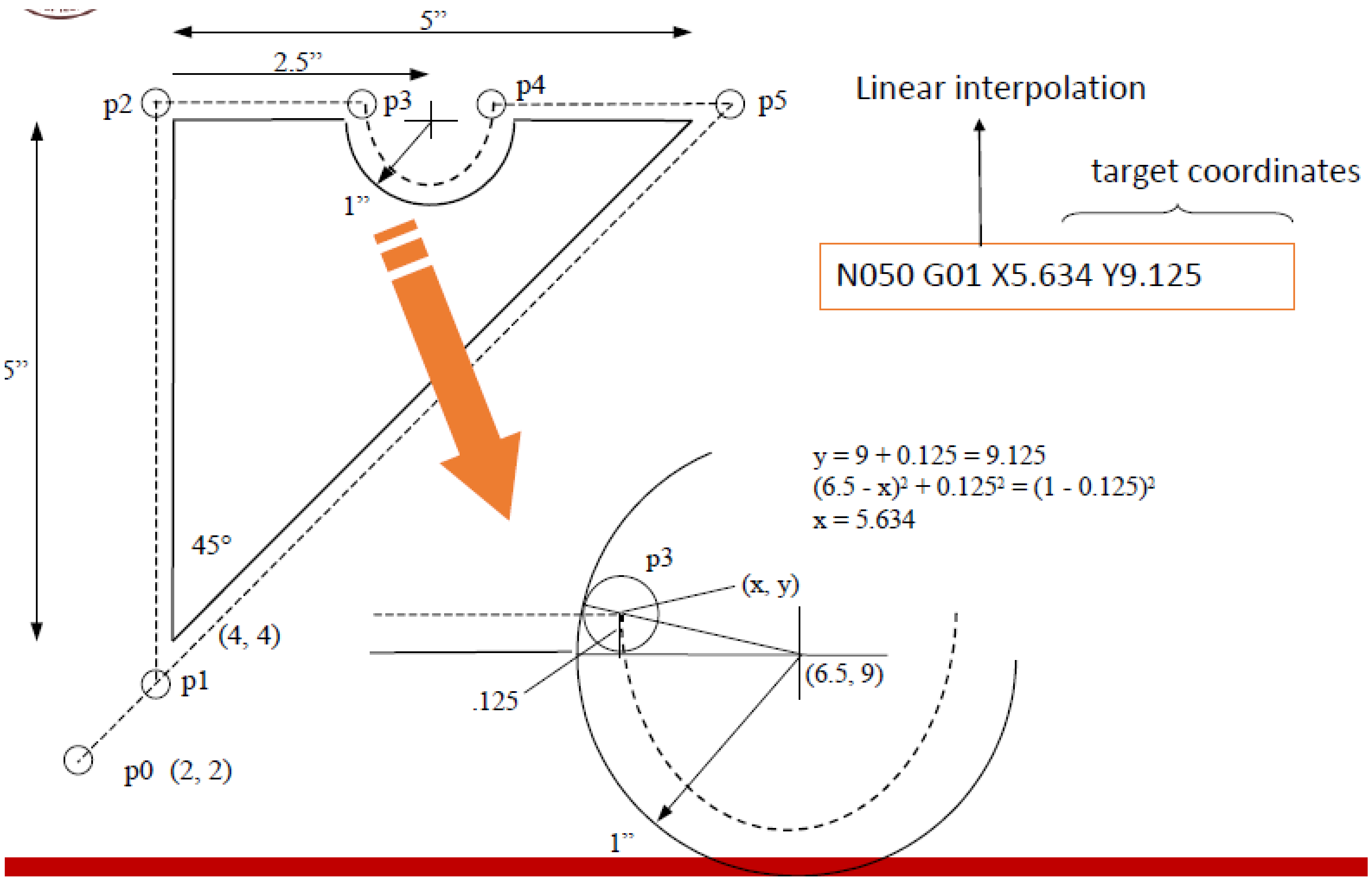
N040 G01 X3.875 Y9.125

or

N040 G01 Y9.125

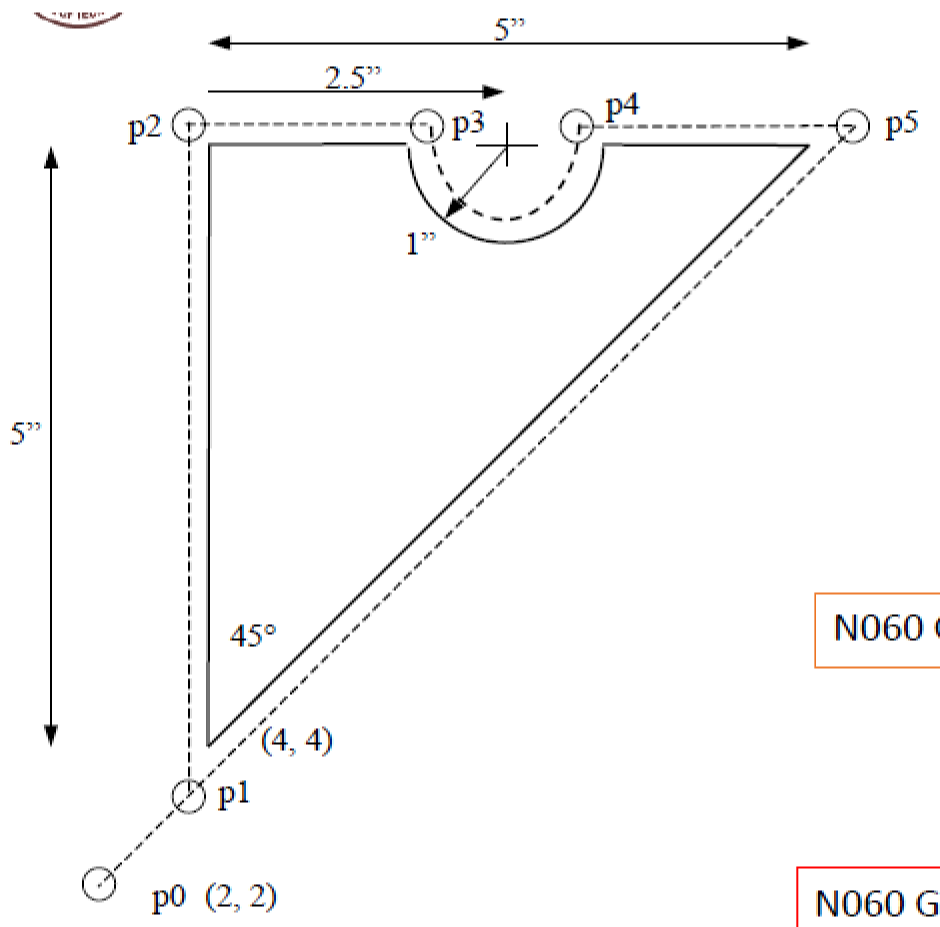
X-coordinate does not change ➔ no need to program it

5. Cut profile from p2 to p3



N050 G01 X5.634 Y9.125

6. Cut along circle from p3 to p4



circular interpolation, CCW motion

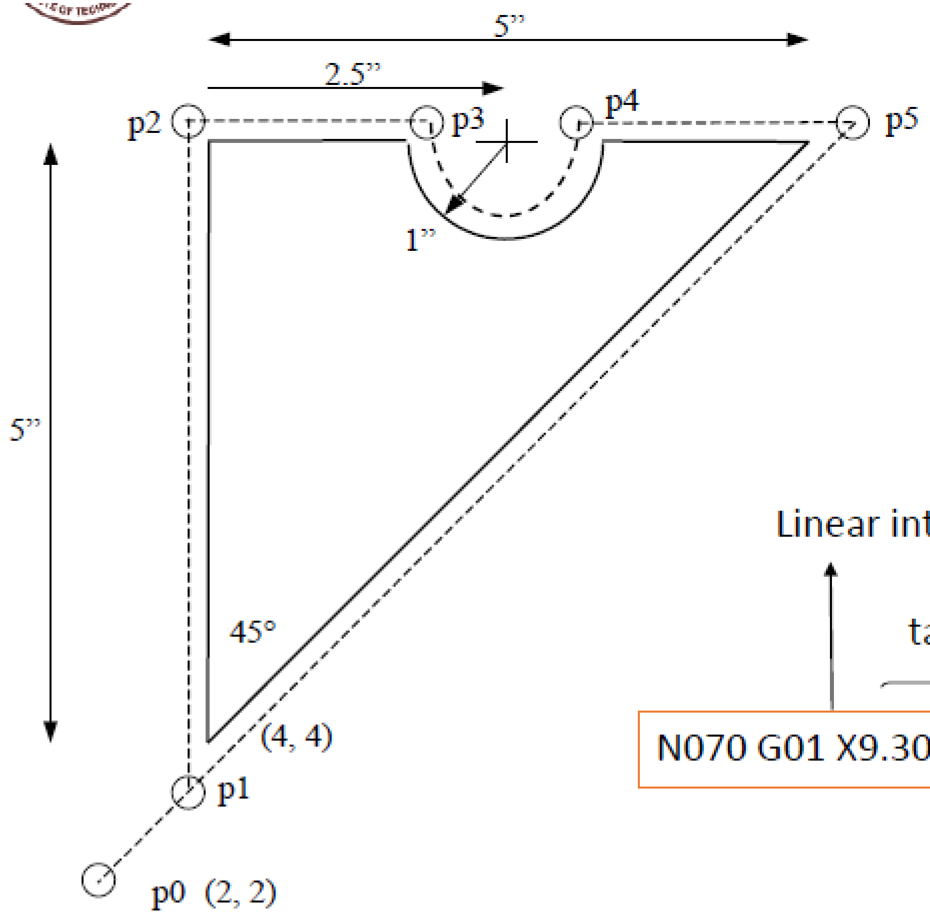
target coordinates

N060 G03 X7.366 Y9.125 I6.5 J9.0

coordinates of center of circle

N060 G03 X7.366 Y9.125 I0.866 J-0.125

7. Cut from p4 to p5

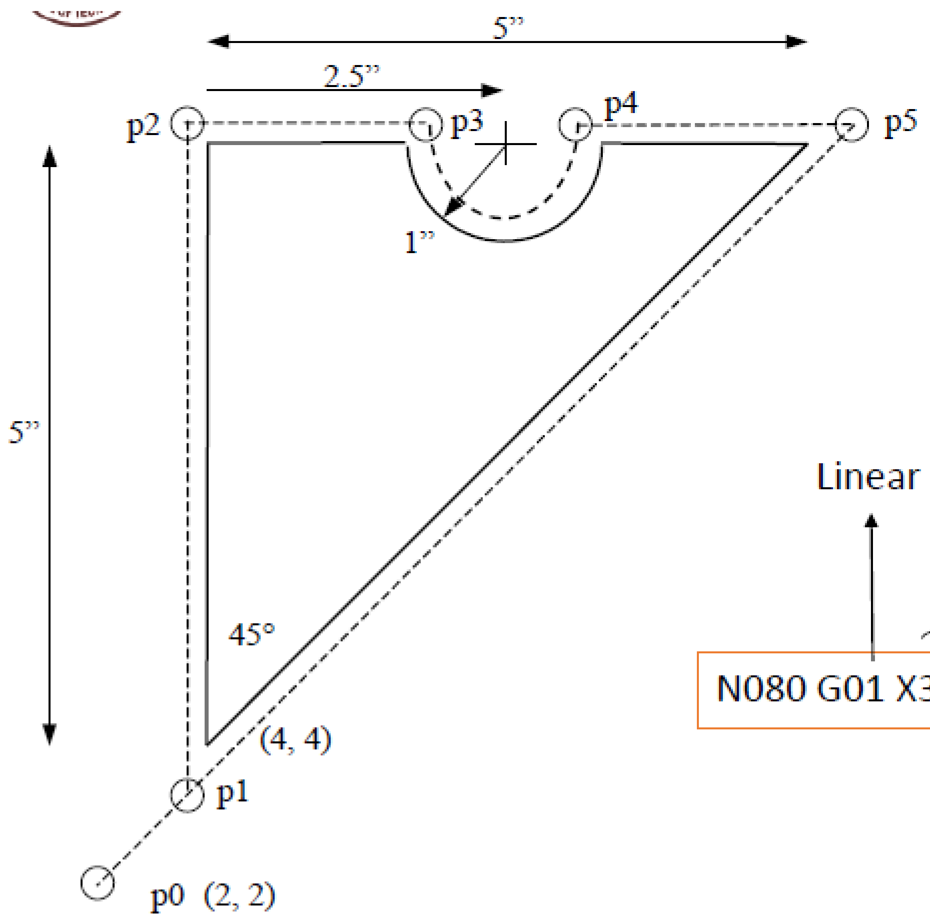


Linear interpolation

target coordinates (Y is unchanged)

N070 G01 X9.302

8. Cut from p5 to p1

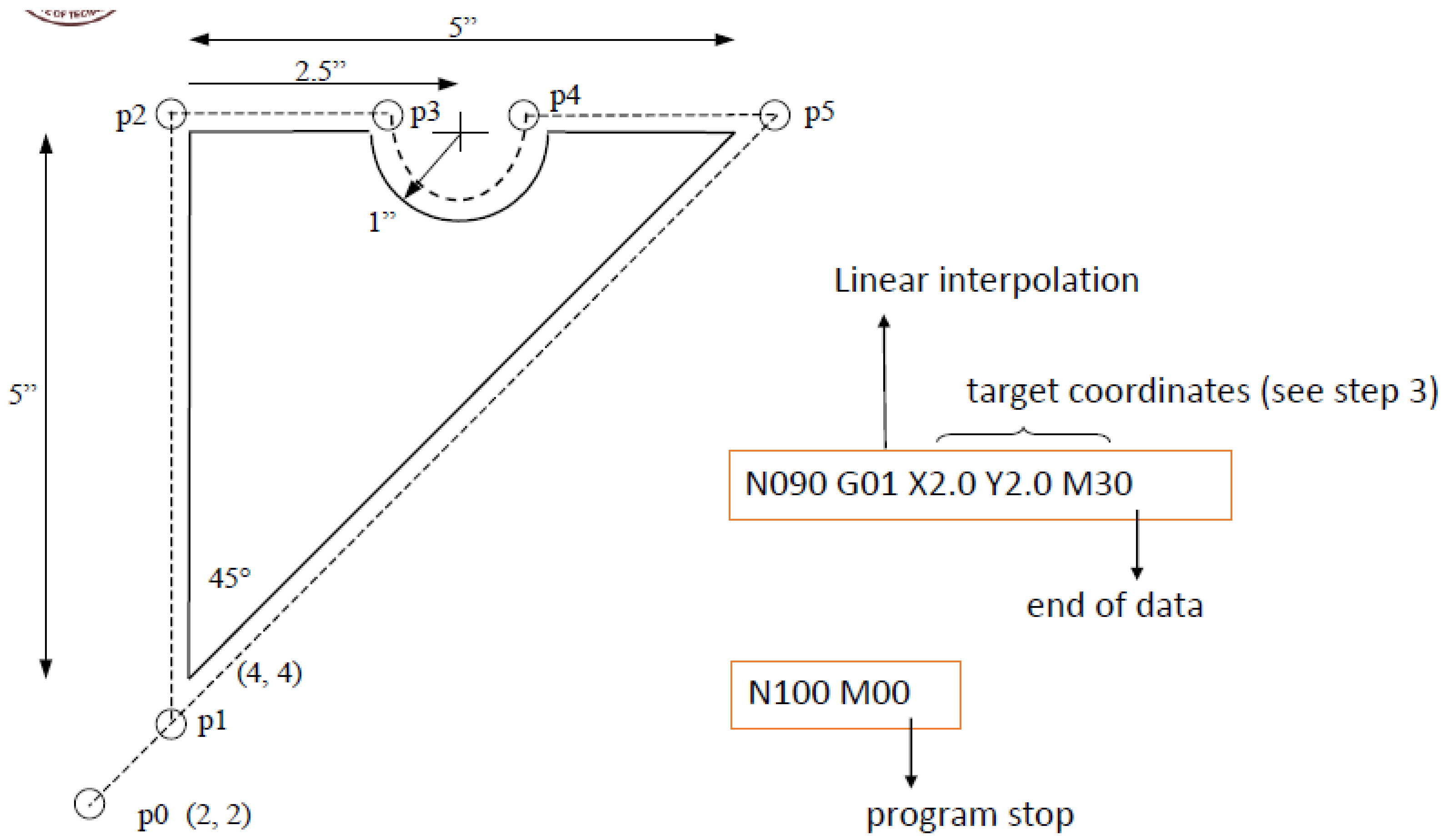


Linear interpolation

target coordinates (see step 3)

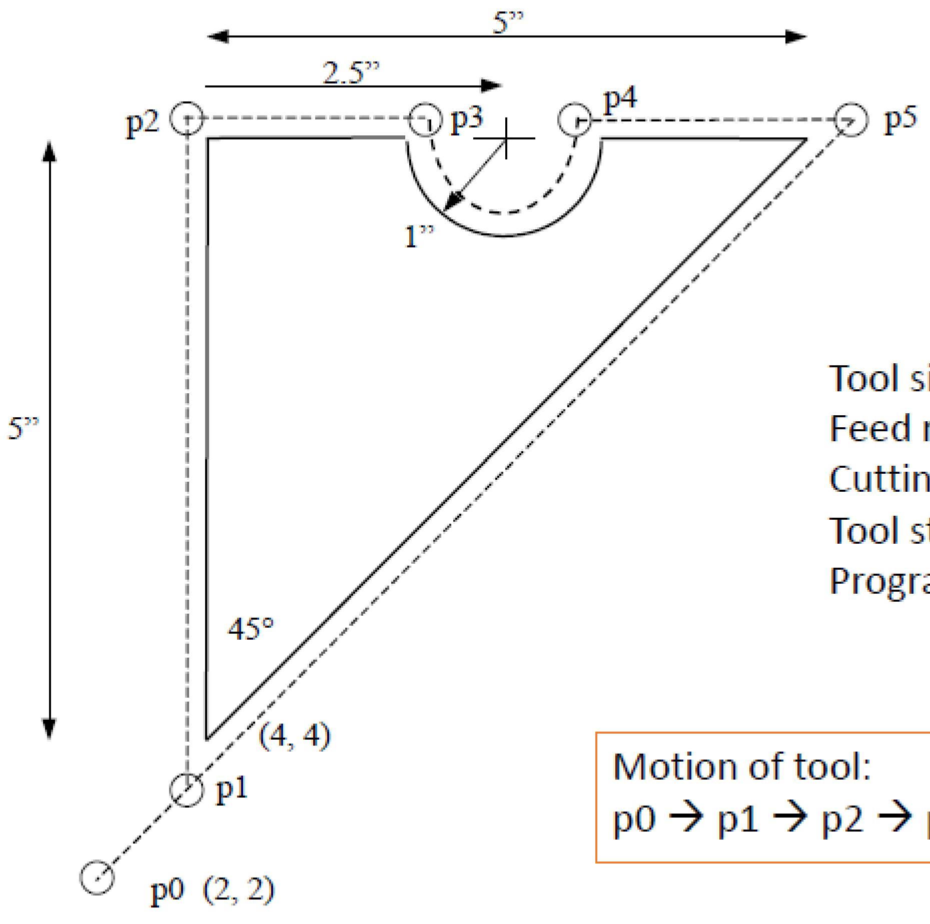
N080 G01 X3.875 Y3.698

9. Return to home position, stop program



10. Complete program

```
N010 G70 G90 G94 G97 M04
N020 G17 G75 F6.0 S300 T1001 M08
N030 G01 X3.875 Y3.698
N040 G01 X3.875 Y9.125
N050 G01 X5.634 Y9.125
N060 G03 X7.366 Y9.125 I0.866 J-0.125
N070 G01 X9.302
N080 G01 X3.875 Y3.698
N090 G01 X2.0 Y2.0 M30
```

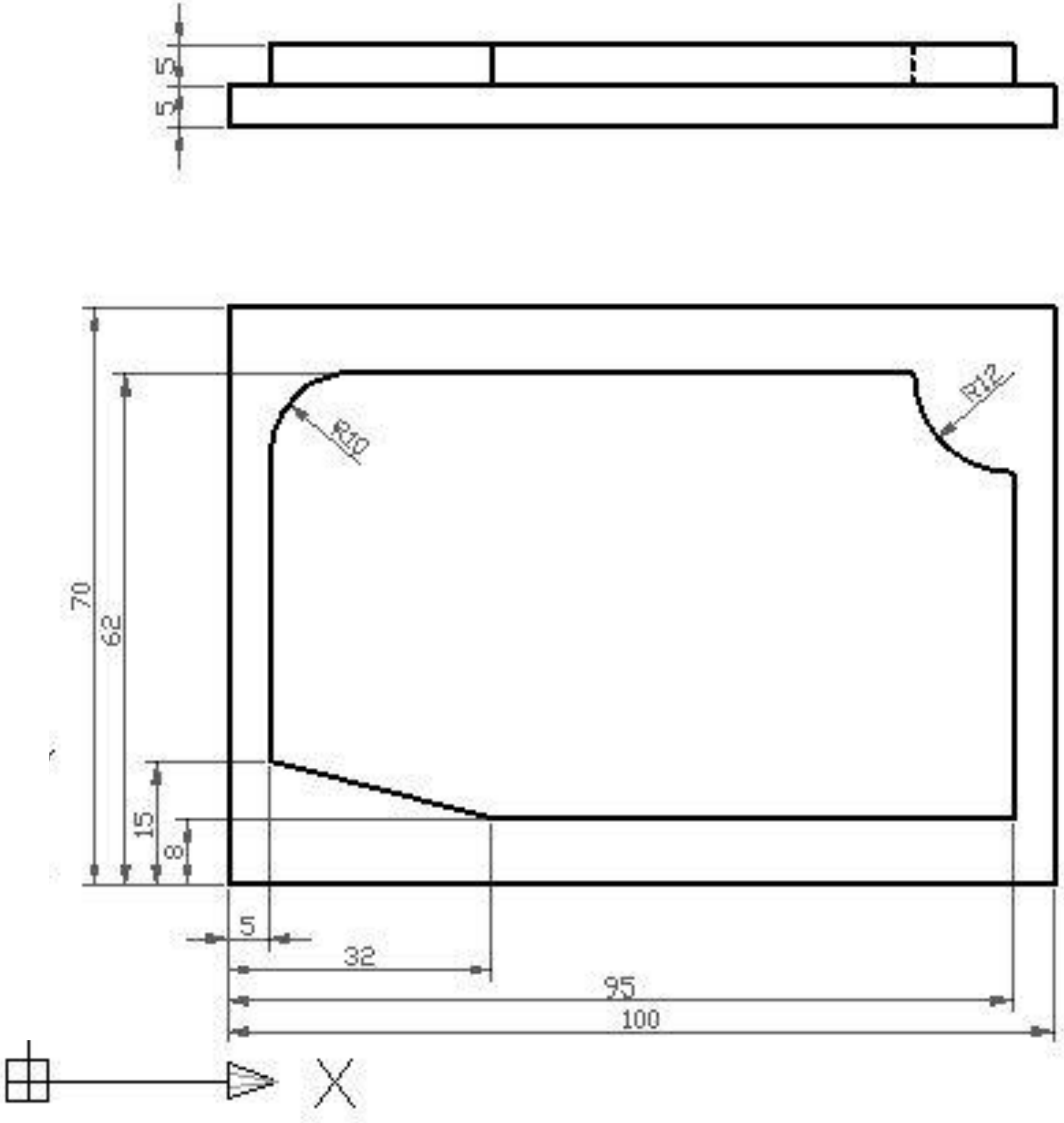


Tool size = 0.25 inch,
Feed rate = 6 inch per minute,
Cutting speed = 300 rpm,
Tool start position: 2.0, 2.0
Programming in inches

Motion of tool:
p0 → p1 → p2 → p3 → p4 → p5 → p1 → p0

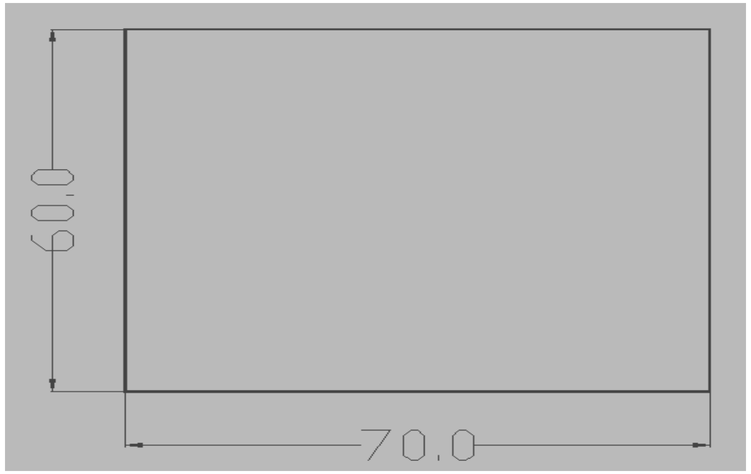
CNC Mill Program (G41 Cutter Radius Compensation Left)

N10 T2 M3 S447 F80
N20 G0 X112 Y-2
N30 Z-5
N40 G41
N50 G1 X95 Y8 M8
N60 X32
N70 X5 Y15
N80 Y52
N90 G2 X15 Y62 I10 J0
N100 G1 X83
N110 G3 X95 Y50 I12 J0
N120 G1 Y-12
N130 G40
N140 G0 Z100 M9
N150 X150 Y150
N160 M30

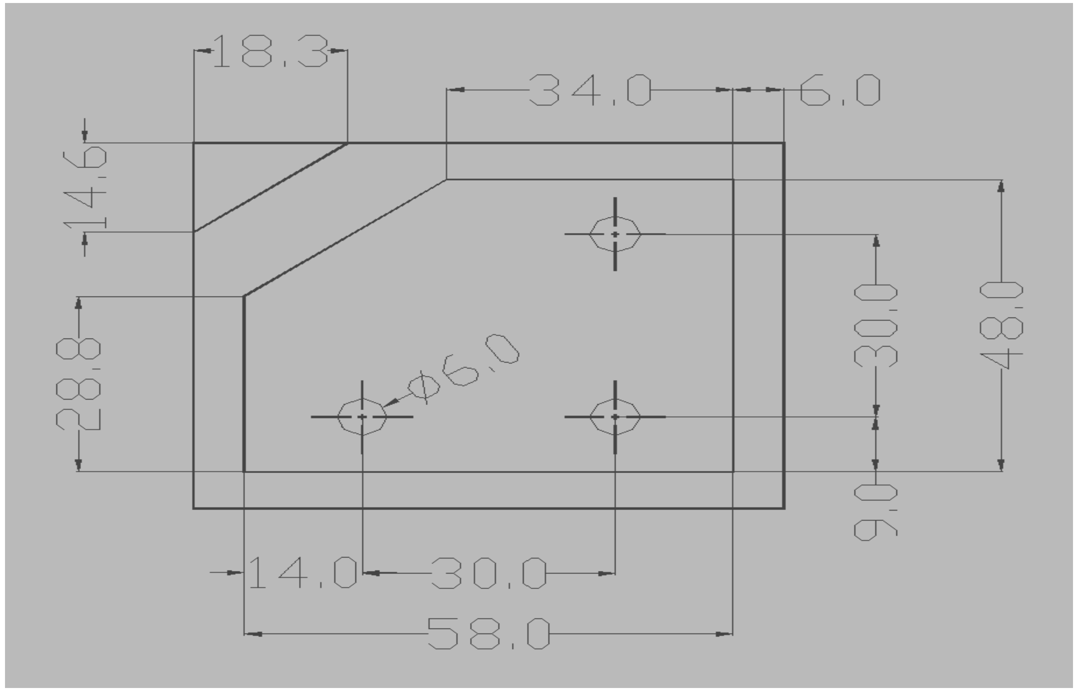


Programming Example (Siemens Controller)

Raw Material



Finished Part



Program Interpretation (Siemens controller)

G55 X200 Y80

Setting the datum to the lower left corner of the work piece

Program Interpretation

G55 X200 Y80
Program 1

Program Identification Number

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1

N001 Sequence Number
M06   Tool Change (End Mill with   Diameter=12mm
T1   Tool Number

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400

Start rotating the spindle clockwise with 400 rpm

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150

Go to Safe Position with feed 150mm/min

Program Interpretation

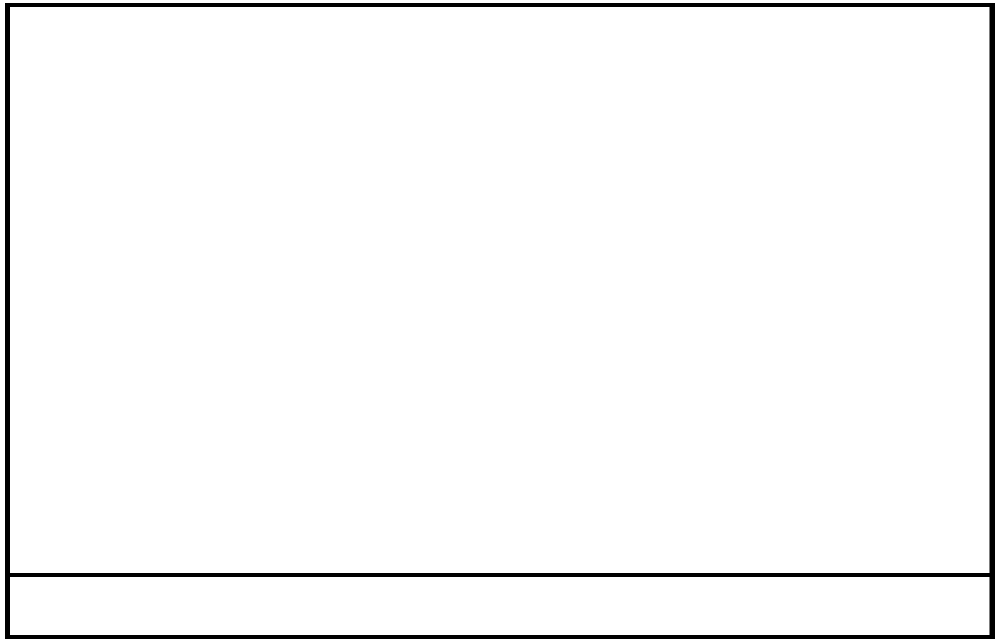
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X0 Y0 Z-0.5 ZFeed 150

Lower the end mill to determine the depth of cut

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X0 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75

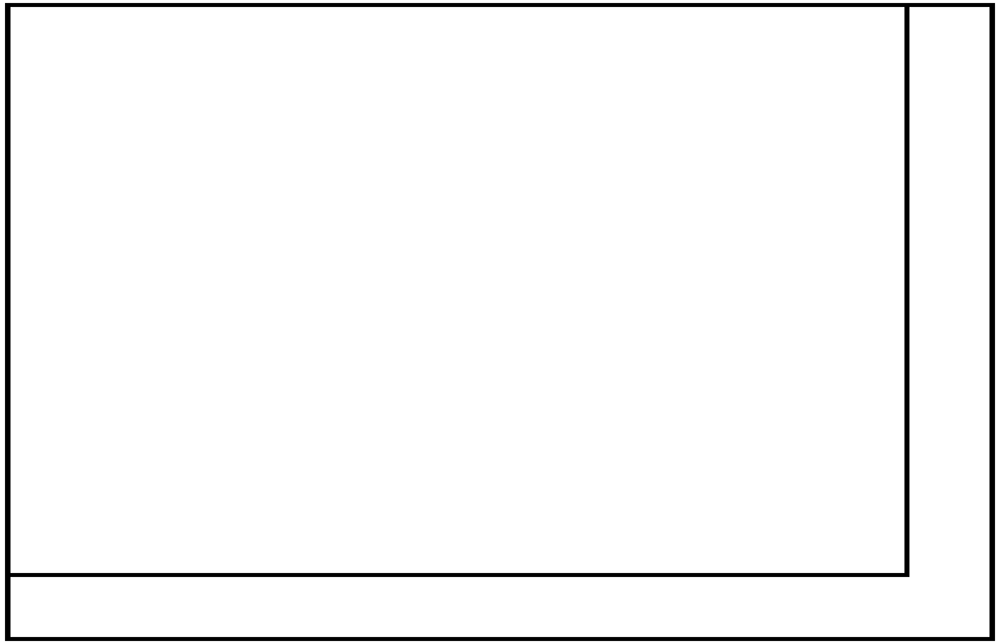
Move from the lower left corner of the work piece to the right lower one cutting with feed=75mm/min



Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X0 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75

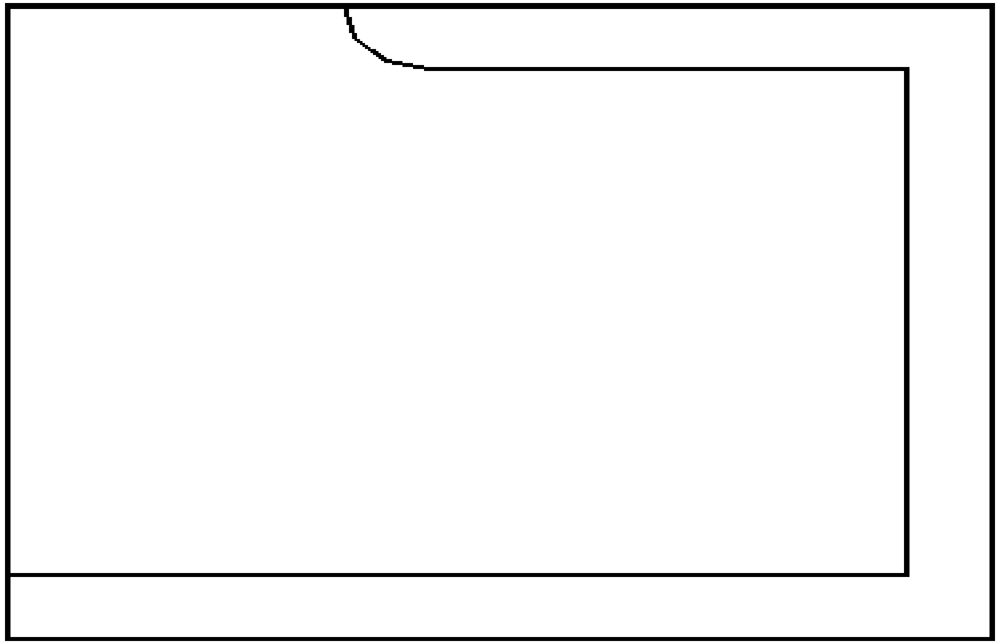
Move from the lower left corner of the work piece to the right lower one cutting with feed=75mm/min



Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X0 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75

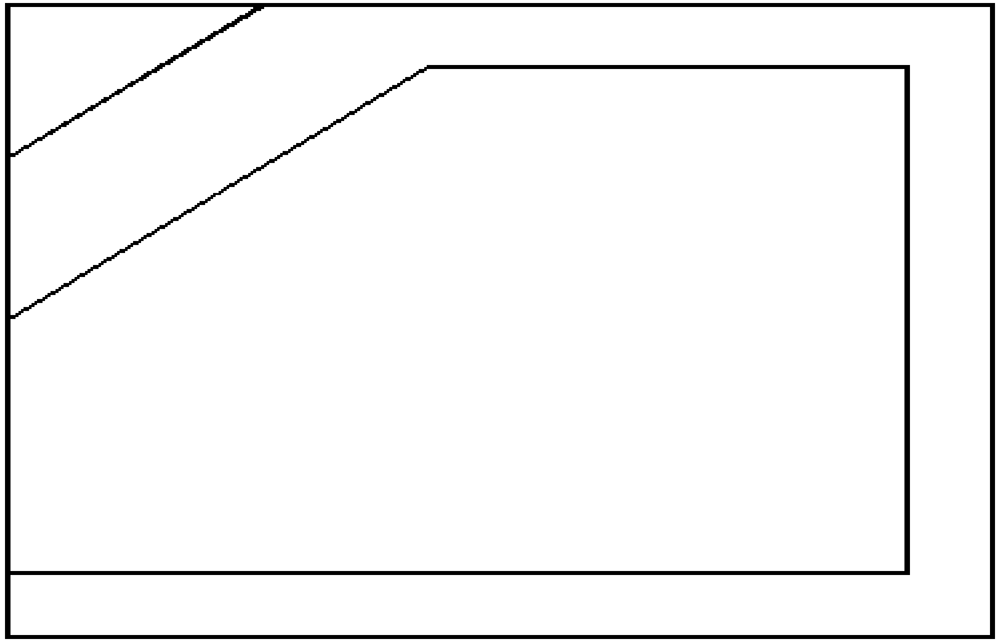
Cutting the horizontally up to X=30



Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X0 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75

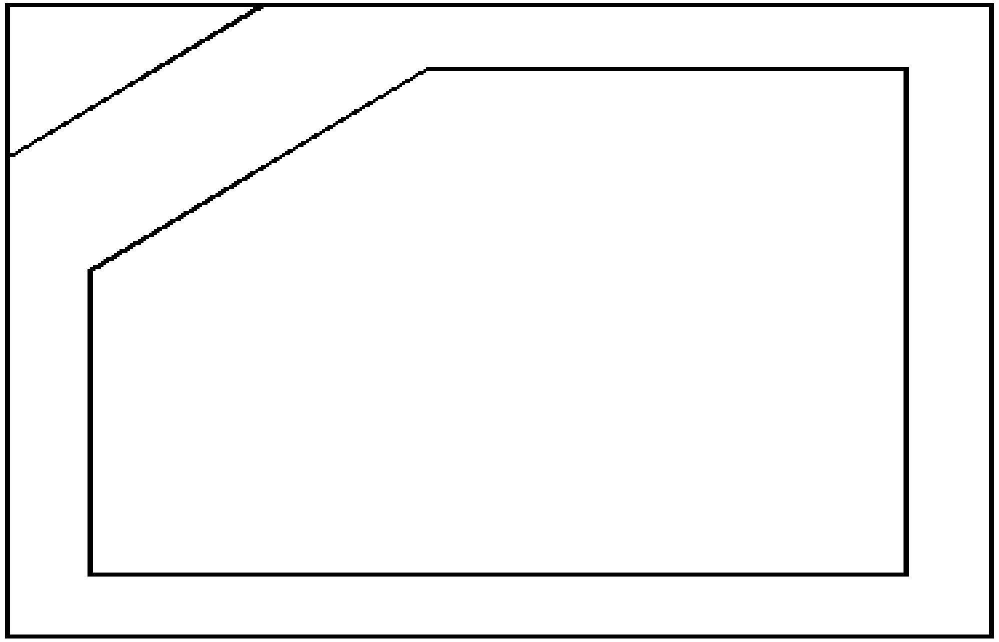
Cutting to X=0 & Y=40



Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X0 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75

Complete the countering



Program Interpretation

```
G55 X200 Y80
Program 1
N001  M06 T1
N002  M03 rpm 400
N003  G01 X0 Y0 Z0 XYFeed 150
N004  G01 X0 Y0 Z-0.5 ZFeed 150
N005  G01 X70 Y0 Z-0.5 XYFeed 75
N006  G01 X70 Y60 Z-0.5 XYFeed 75
N007  G01 X30 Y60 Z-0.5 XYFeed 75
N008  G01 X0 Y40 Z-0.5 XYFeed 75
N009  G01 X0 Y0 Z-0.5 XYFeed 75
N010  G81 R3 E9 N7 Z-0.5
```

Repeat 7 times blocks from N003 to N009 with incremental offset of Z=-0.5

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X0 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
N011 M05

Spindle Off

Program Interpretation

```
G55 X200 Y80
Program 1
N001  M06 T1
N002  M03 rpm 400
N003  G01 X0 Y0 Z0 XYFeed 150
N004  G01 X0 Y0 Z-0.5 ZFeed 150
N005  G01 X70 Y0 Z-0.5 XYFeed 75
N006  G01 X70 Y60 Z-0.5 XYFeed 75
N007  G01 X30 Y60 Z-0.5 XYFeed 75
N008  G01 X0 Y40 Z-0.5 XYFeed 75
N009  G01 X0 Y0 Z-0.5 XYFeed 75
N010  G81 R3 E9 N7 Z-0.5
N011  M05
N012  M02
```

End Program

Program Interpretation

Tool Change

Changing the tool

Program Interpretation

Tool Change

G55 X200 Y80

Setting the datum to the lower left corner of the work piece

Program Interpretation

Tool Change
G55 X200 Y80
Program 2

Program Identification Number

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2

N001 Sequence Number
M06   Tool Change (Drill with Diameter=6mm
T2   Tool Number

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400

Start rotating the spindle clockwise with 400 rpm

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150

Go to Safe Position with feed 150mm/min

Program Interpretation

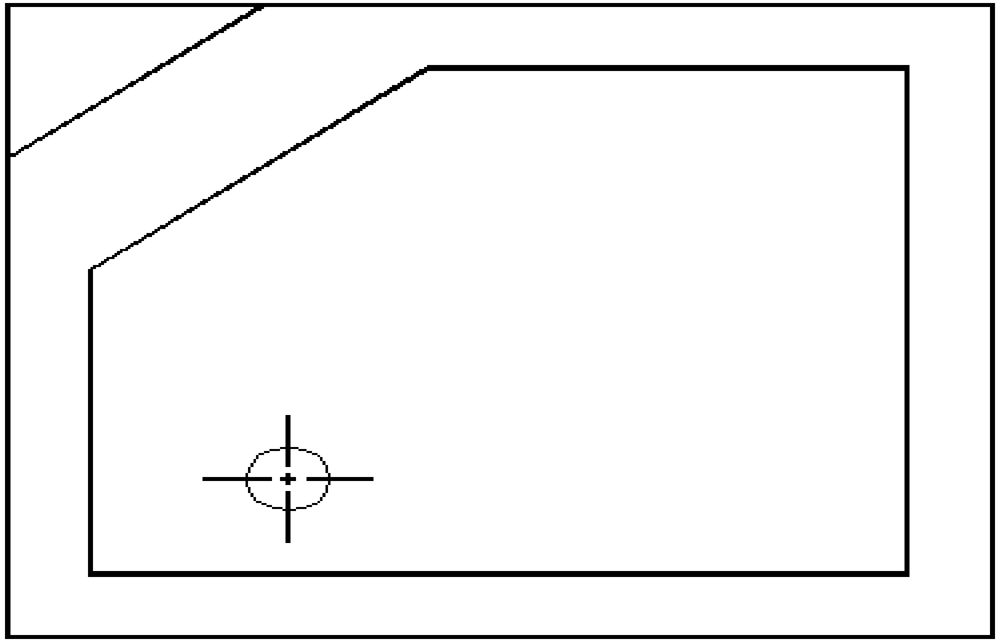
Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150

Stop above the center of the first hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75

Start Drill the first hole



Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150

Retract to a position above the hole

Program Interpretation

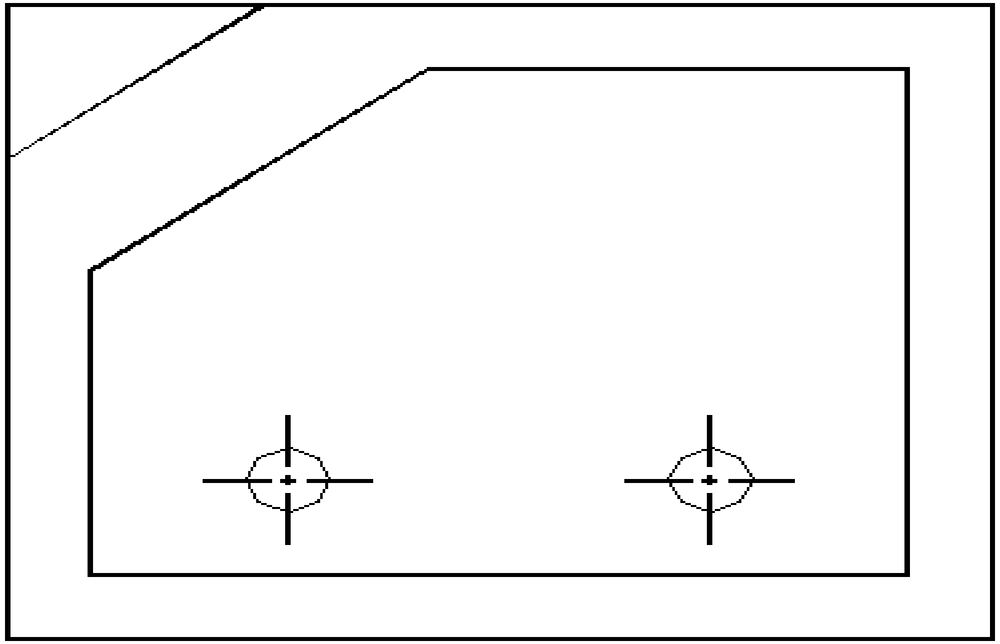
Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150

Stop above the center of the second hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75

Drill the second hole



Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150

Retract to a position above the second hole

Program Interpretation

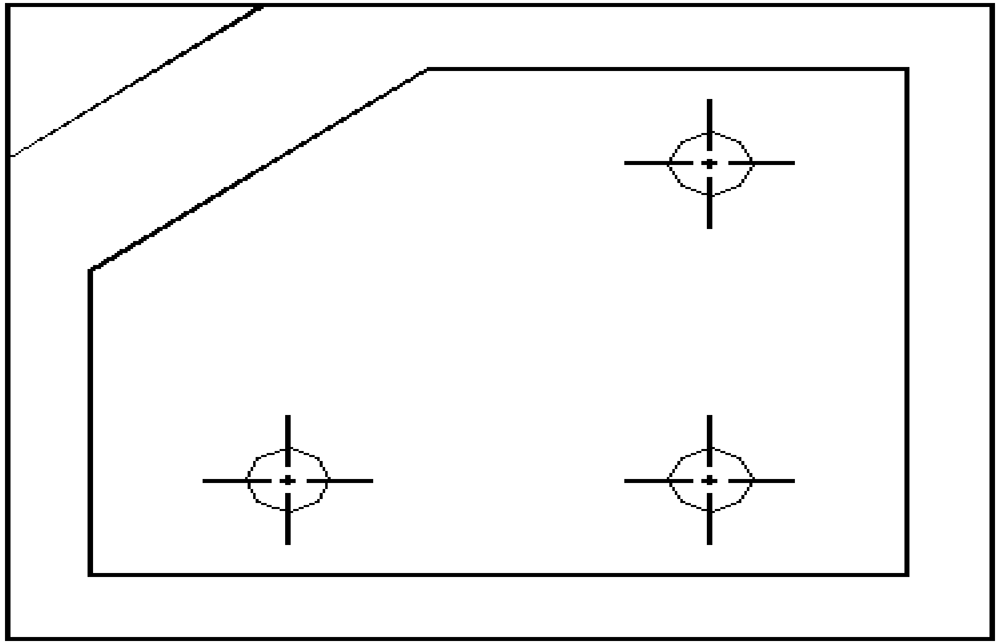
Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150

Stop above the center of the third hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75

Drill the third hole



Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150

Retract to a position above the third hole

Program Interpretation

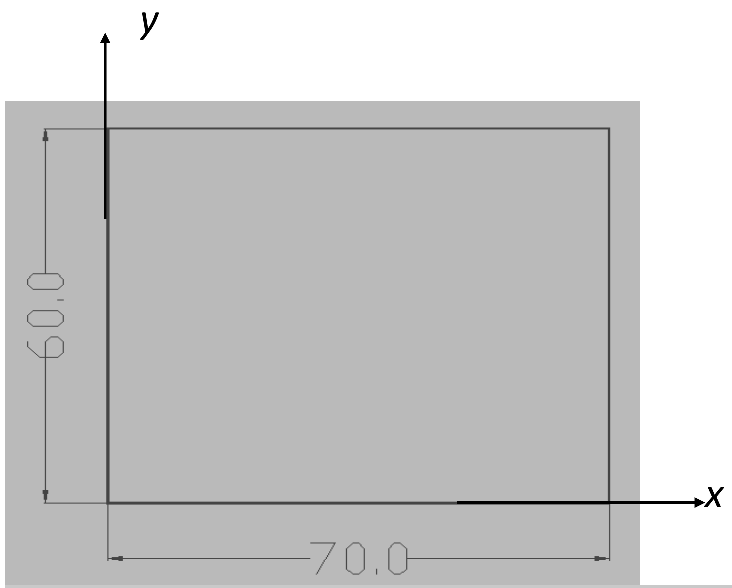
Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05

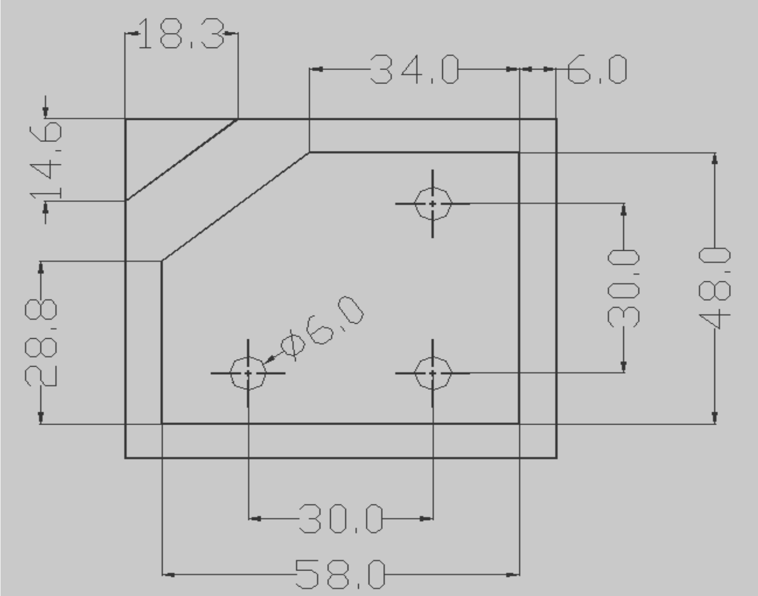
Spindle off

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05
N014 M02
End Program

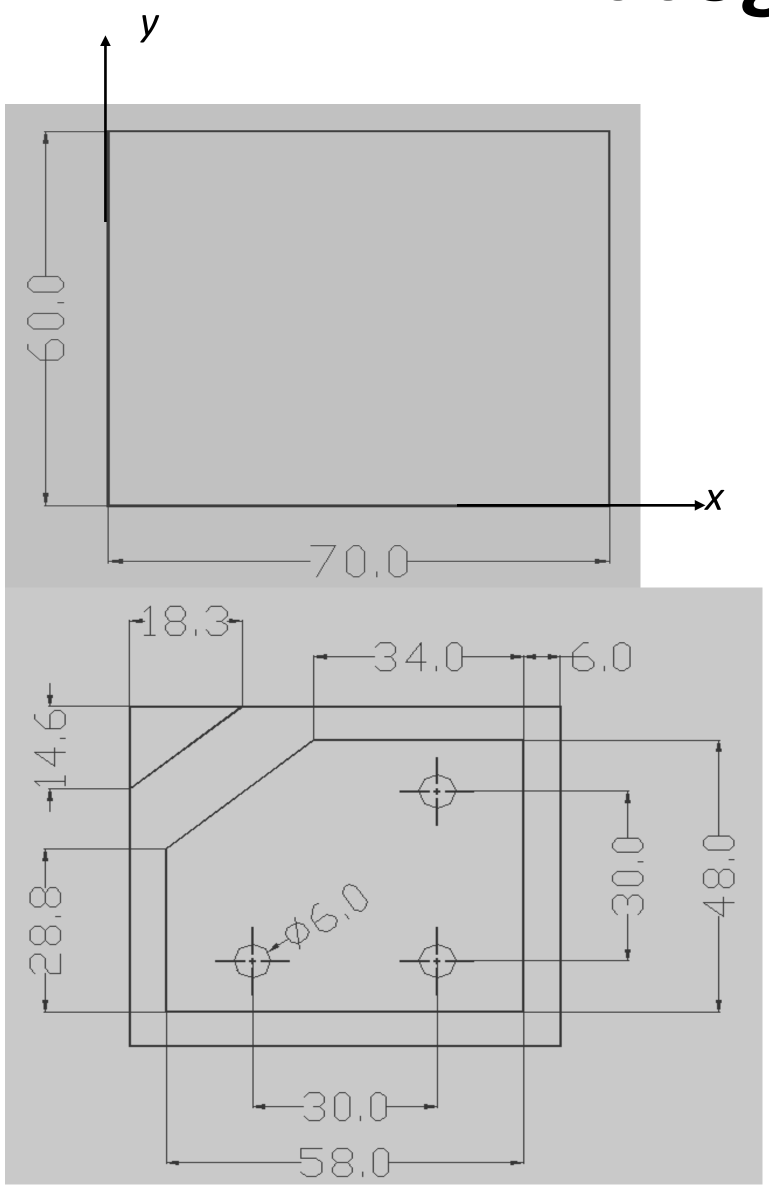
Programming Example (Siemens controller)





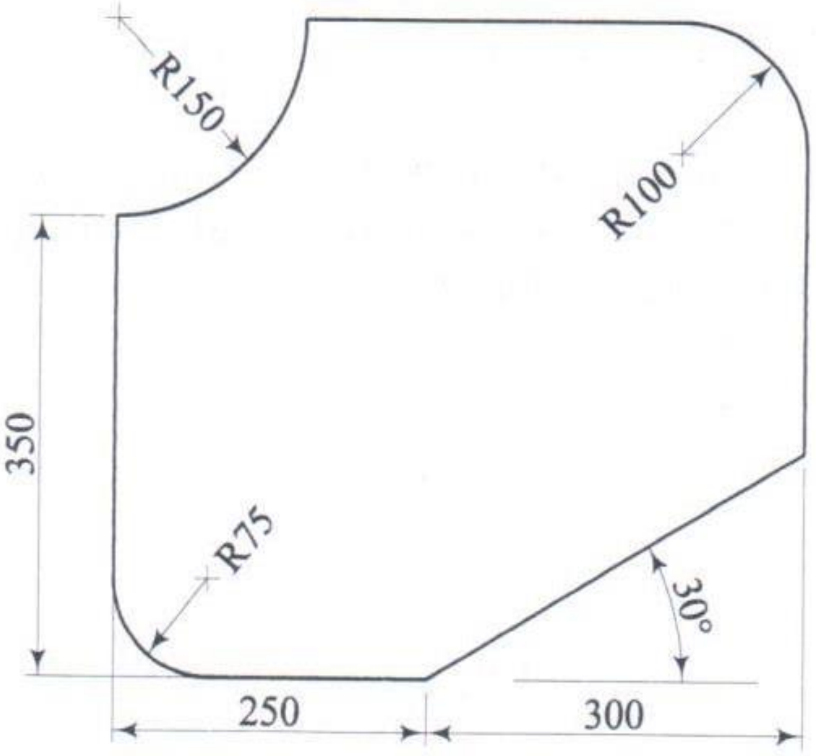
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X0 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
N011 M05
N012 M02

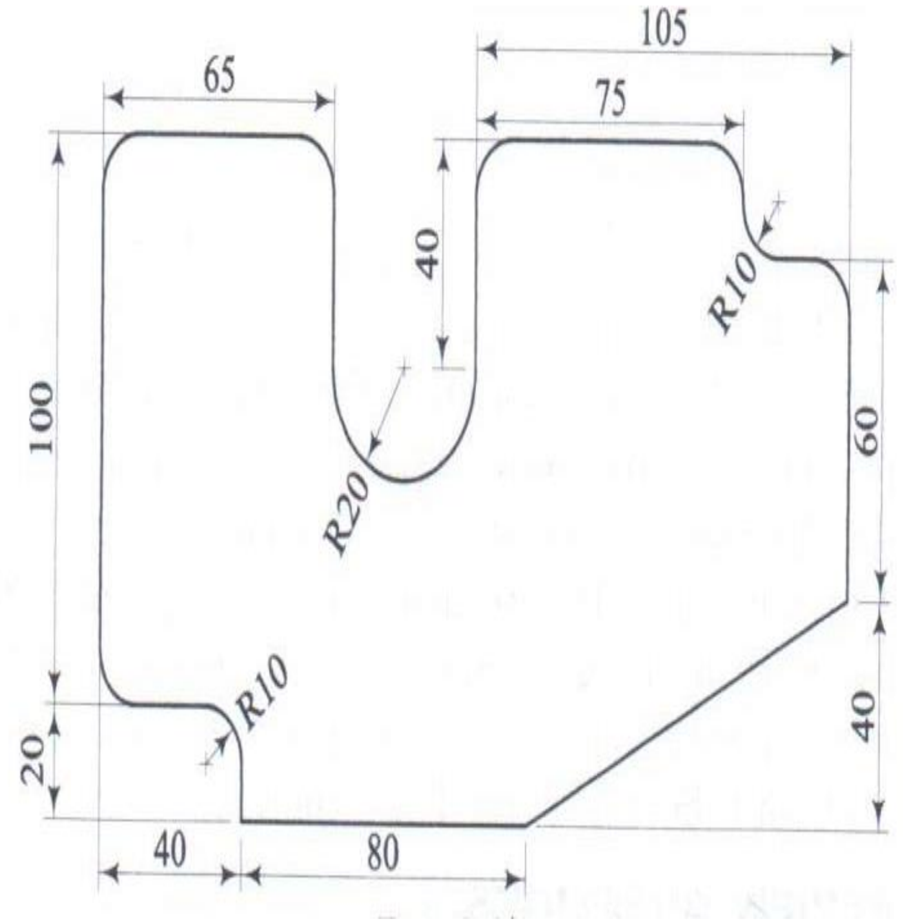
Programming Example



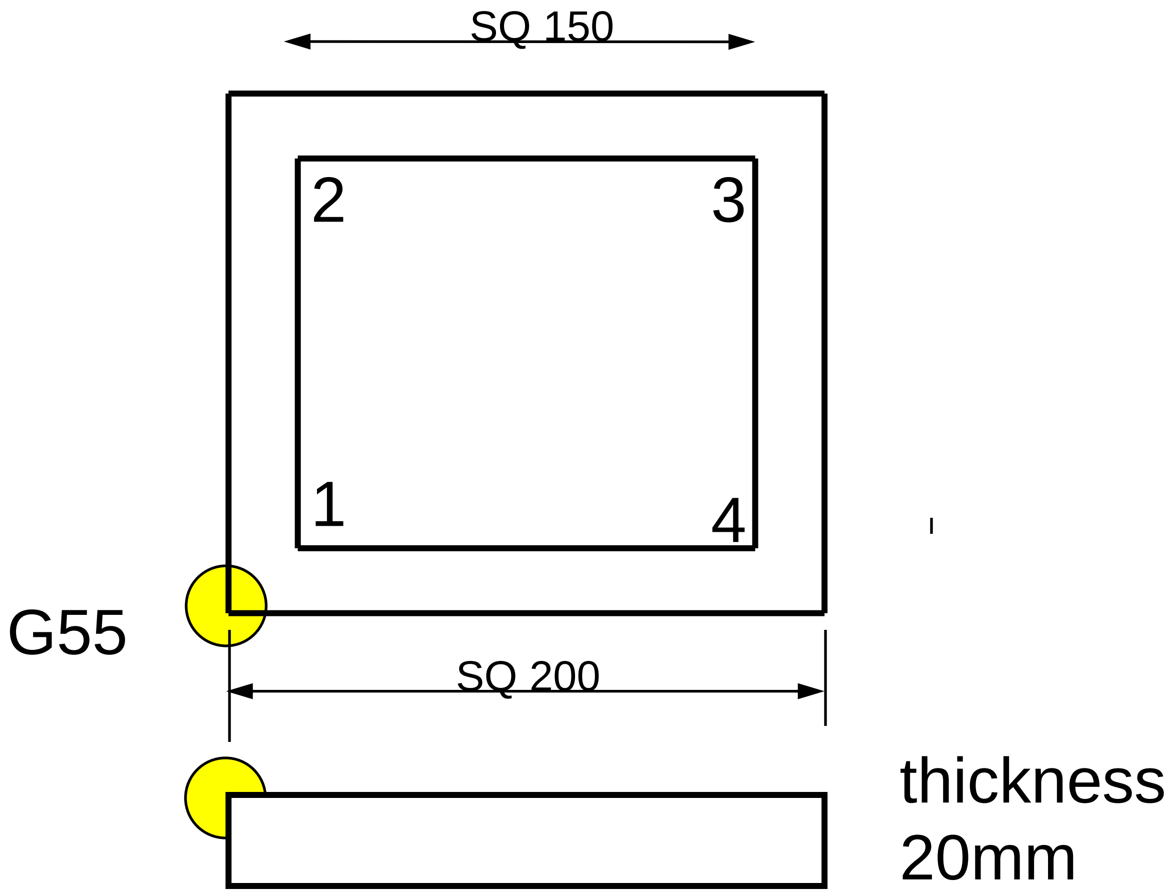
Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X0 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05
N014 M02

Exercises

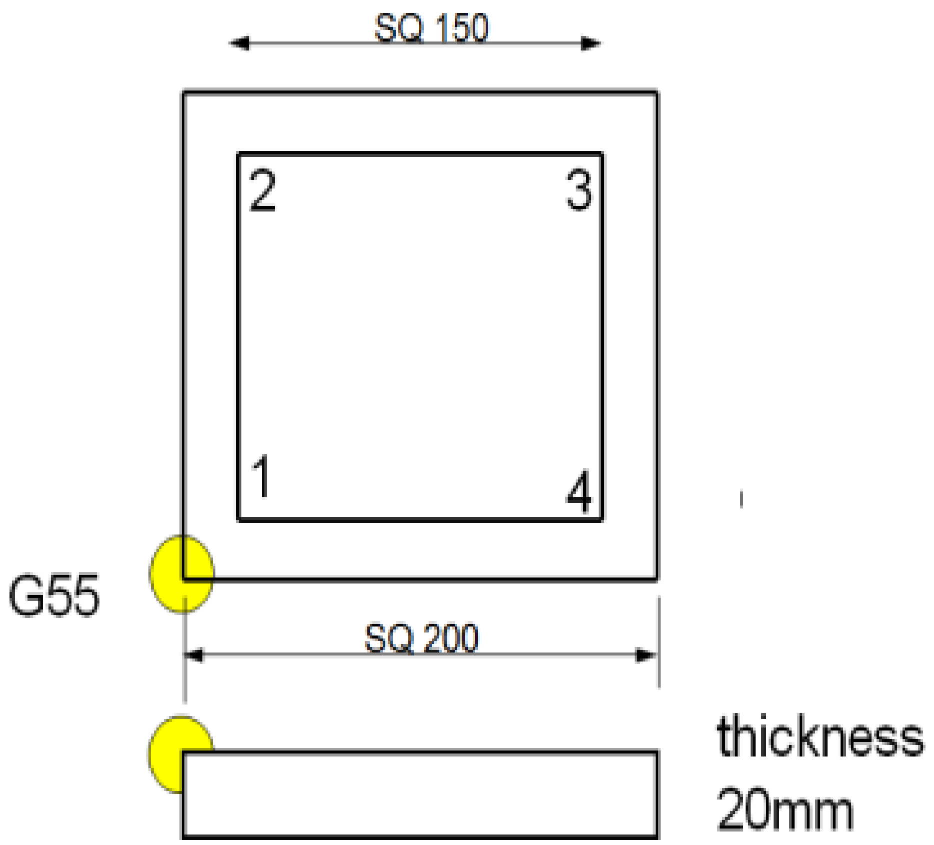

R150
R100
350
R75
30°
250
300


105
65
75
40
R10
100
R20
60
R10
40
20
40
80

Programming Examples



- **NC Part Program**
- O1986;
- N10 G21 G94;
- N20 G91 G28 Z0.0;
- N30 T01 M06;
- N40 G90 **G55** G00 X-25.0 Y20; G55- Setting Machine reference datum
- N50 G43 H01 Z50; G43- Cutter length compensation+
- N60 S1000 M03;
- N70 G01 Z-10 F400;
- N80 G41 D01 X20.0 Y20; G41 – Cutter compensation left
- N90 X180.0;
- N100 Y180.0;
- N110 X20.0;
- N120 Y20.0;
- N130 X-25.0 Y-25;
- N140 G40 G00 Z100.0; G40 – Cutter compensation cancel
- N150 G91 G28 Z0.0;
- N160 G91 G28 X0 Y0;
- N170 M05;
- N180 M30;



CNC Pocket Milling Program Example – Peck Milling

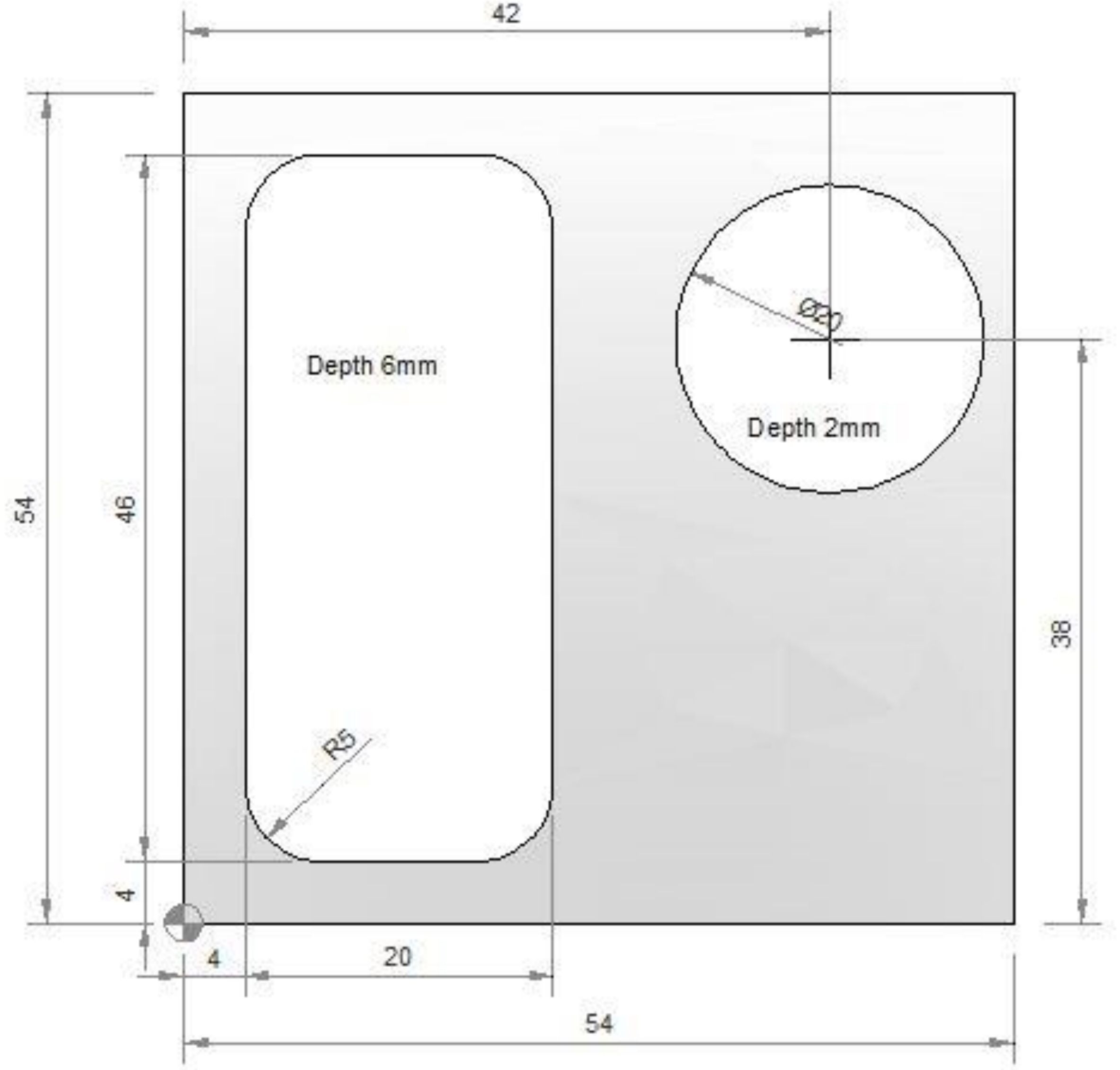
Main Program

Milling cutter diameter: 10mm

```
N05 G55
N10 M6 T2 H3 G43
N15 M3 S1000 F60
N20 G0 X9 Y9 Z1
N25 G1 Z0
N30 M98 P030035
N35 G0 Z1 G90
N40 X42 Y38
N45 G1 Z-2 F30
N50 X47 F300
N55 G3 X47 Y38 I-5 J0
N60 G0 Z100
 N65 G49
N70 M30
```

Subprogram

```
O0035
N05 G1 Z-2 G91 F30
N10 X10 F100
N15 Y36
N20 X-10
N25 Y-36
N30 M99
```



Explanation

M98 P030035

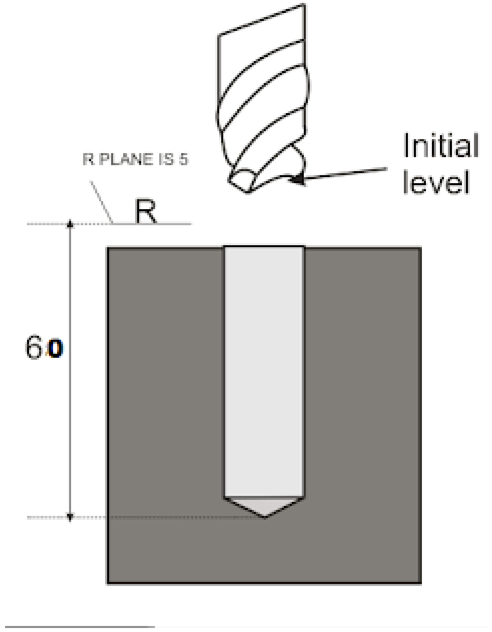
this code mean call Subprogram No. 0035 three times.

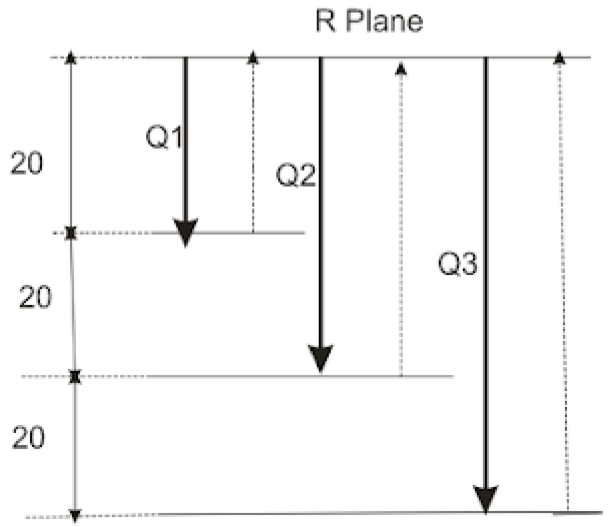
G43- Tool length compensation +

G49- Tool length compensation cancel

Fanuc G73 high speed peck drilling cycle program

we have to know which type of drilling is done more in depth. for deep drilling is considered as DEPTH/ DIA => 5
The benefits of peck drilling reduce cycle time. In G73 peck drilling after each drill, tool retract only 1 mm. This drilling cycle is used mostly drill soft materials like; Aluminium






```
O4231
N10  M06 T06 ;
N20  G90 G80 G17 G00 G54 X0 Y0 ;
N30  G43 Z100 H11 ;
N40  M03 S1500 ;
N50  M08 ;
N60  G99 G73 Z-55 R5 Q20 F300 ;
N70  G98 G80 G00 Z100 ;
N80  M05 M09 M30 ;
```

DESCRIPTION OF PROGRAM

N10- Tool change command , select tool no. 6
N20- Absolute co-ordinate command , cancel canned cycle command , selection of XY plane, rapid command, work coordinate for tool positioning at X0 and Y0.
N30- Tool height offset compensation command , where tool is 100 along Z axis , tool height code H11.
N40- Spindle on clockwise , speed is 1500 rpm .
N50- Coolant ON .
N60- Return to R-plane in canned cycle , Peck drilling cycle command , Depth of drill is 55 , R- plane distance is 5 , depth of each cut is 20(incremental), feed rate per minute is 300 .
N70- Tool is return at initial position , cancel canned cycle , rapid command where tool is 100 mm up along z axis.
N80- Spindle off , coolant off , main program end .

Fanuc G81 drilling cycle program with operation repeats

why we can do operation repeat ?

Following fig. program ; we used K4 command in N60 . The reason is , we can see in the following fig. there is 4 hole drilling on similar distance on X30 and Y0 . Therefore we do not have to need specify every time XY place . In these program in N60 only we used G91 and K4 command .

G81 X_ Y_ Z_ R_ F_ K_ ;

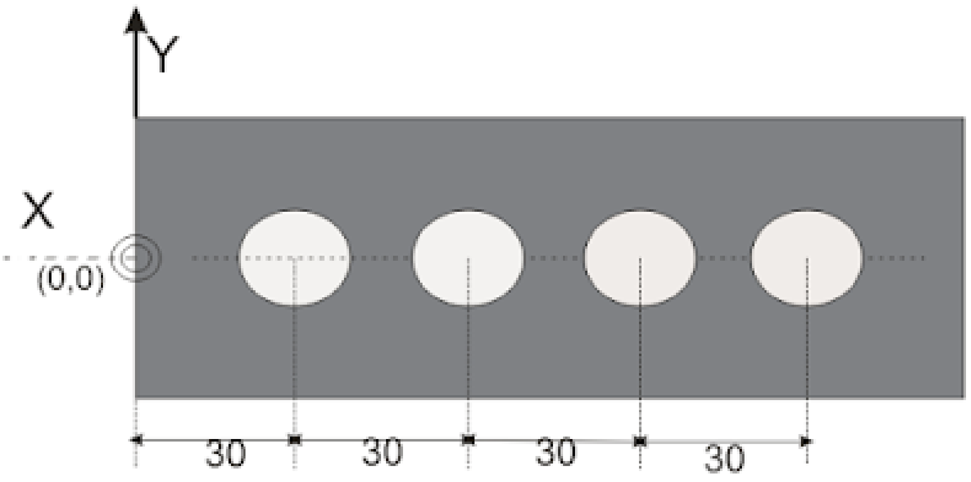
Where , **XY-** Position of hole

Z- Depth of operation perform

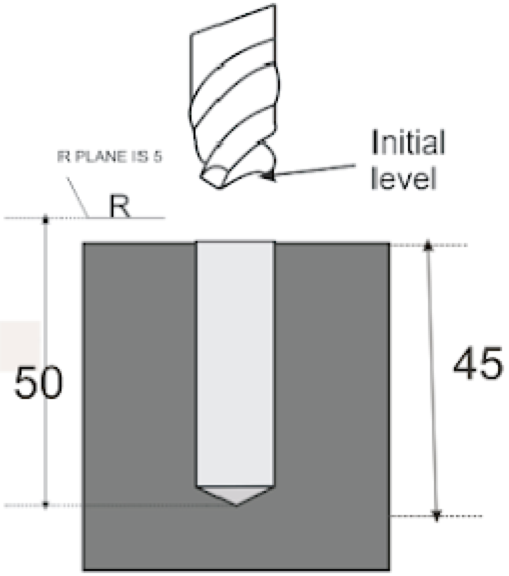
R- R plane position.

F- cutting feed rate

K- no of times operation repeats.



```
O5124
N10   M06 T06 ;
N20   G90 G80 G17 G00 G54 X0 Y0 ;
N30   G43 Z100 H4 ;
N40   M03 S1500 ;
N50   M07 ;
N60   G99  G91 G81 X30 Y0 Z-45 R5  K4 F120  ;
N70   G98 G90 G80 G00 Z100 ;
N80   M05 M09 M30 ;
```



DESCRIPTION OF PROGRAM

N10- Tool change command , select tool no. 6

N20- Absolute co-ordinate command , cancel canned cycle command , selection of XY plane, rapid command, work coordinate for tool positioning at X0 and Y0.

N30- Tool height offset compensation command , where tool is 100 along Z axis , tool height code H4.

N40- Spindle on clockwise , speed is 1500 rpm .

N50- Coolant ON .

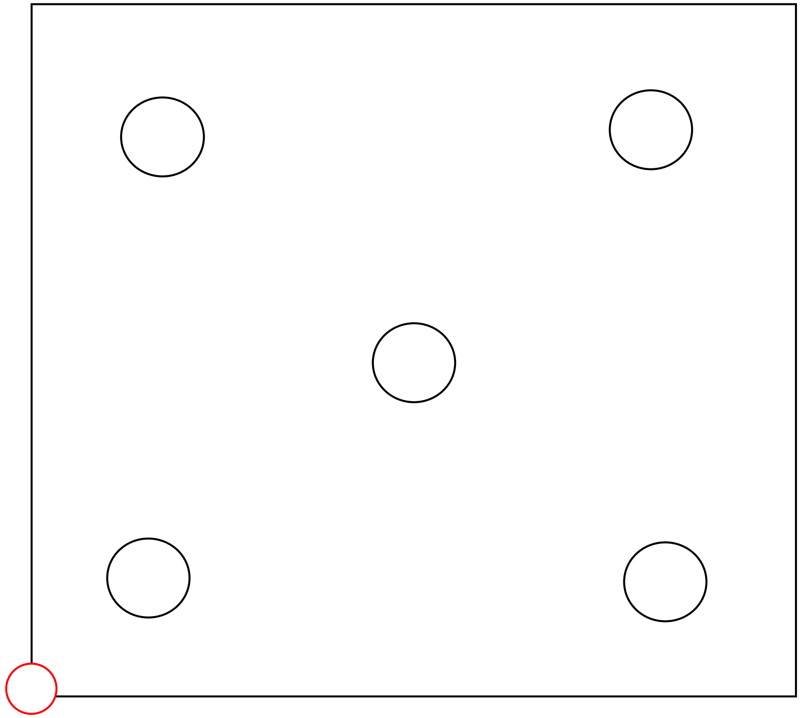
N60- Return to R-plane in canned cycle , Incremental command, Drilling cycle command, first position is X30 and Y0 Depth of drilling is 45 , R-plane distance is 5 ,feed rate per minute is 120 .

After that second drilling position is X30 from current position and Y0 because we provide incremental command and no of repetition K4 (repeating operation), similarly reaming operation will perform.

N70-Tool return initial position ,Absolute co-ordinate command(cancel incremental command), cancel canned cycle , rapid command , where tool along Z axis is 100 .

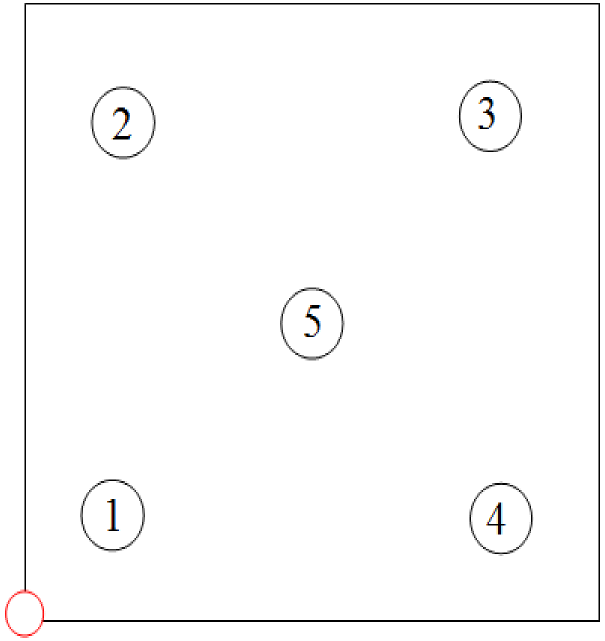
N80- Spindle off , coolant off, Main program end .

Programming Example



BLANK SIZE
100*100*20
DIA. 8, FIVE HOLES
HOLE1 (20,20)
HOLE2 (20,80)
HOLE3 (80,80)
HOLE4 (80,20)
HOLE5 (50,50)

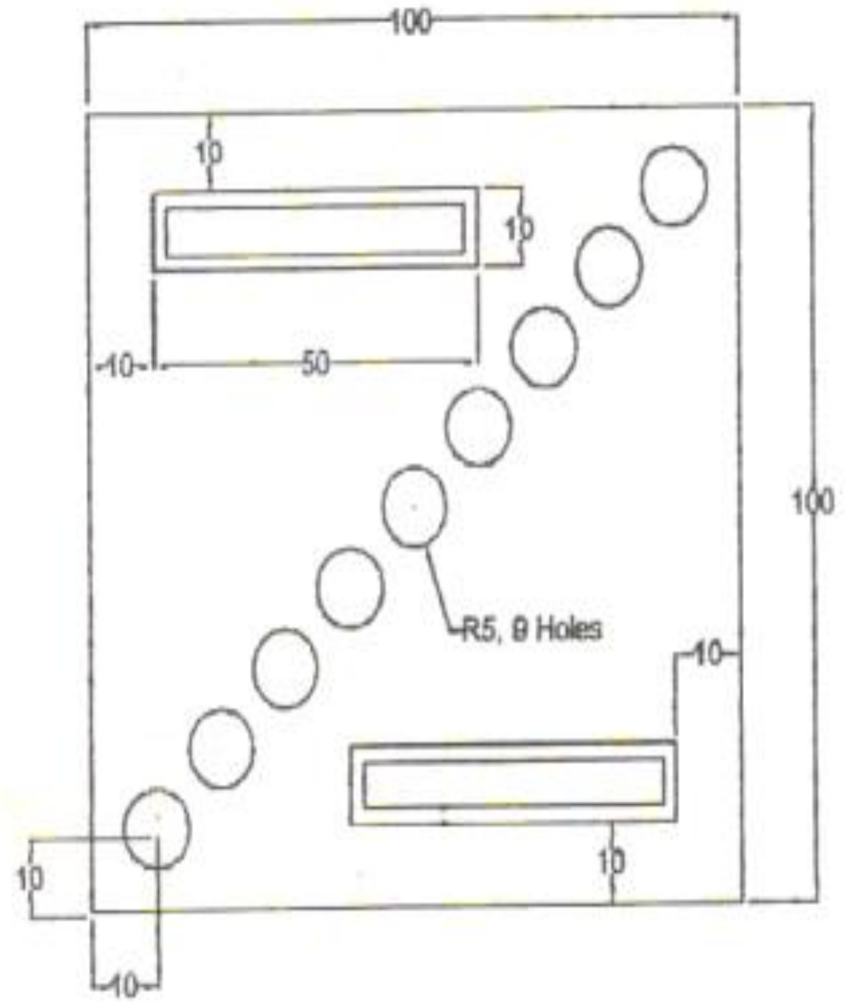
- **NC Part Program**
- O1975;
- N10 G21 G94;
- N20 G91 G28 Z0.0;
- N30 T01 M06;
- N40 G90 G54 G00 X20.0 Y20.0;
- N50 G43 H01;
- N60 S1200 M03 Z0.0;
- **N70 G99 G81 X20.0 Y20.0 Z-23.0 R5.0 F300;**
- N80   Y80.0;
- N90   X80.0;
- N100   Y20.0;
- N110   X50.0 Y50.0;
- N120 G80 G00 Z100.0 M05;
- N130 G91 G28 Z0;
- N140 G28 X0 Y0;
- N150 M30;



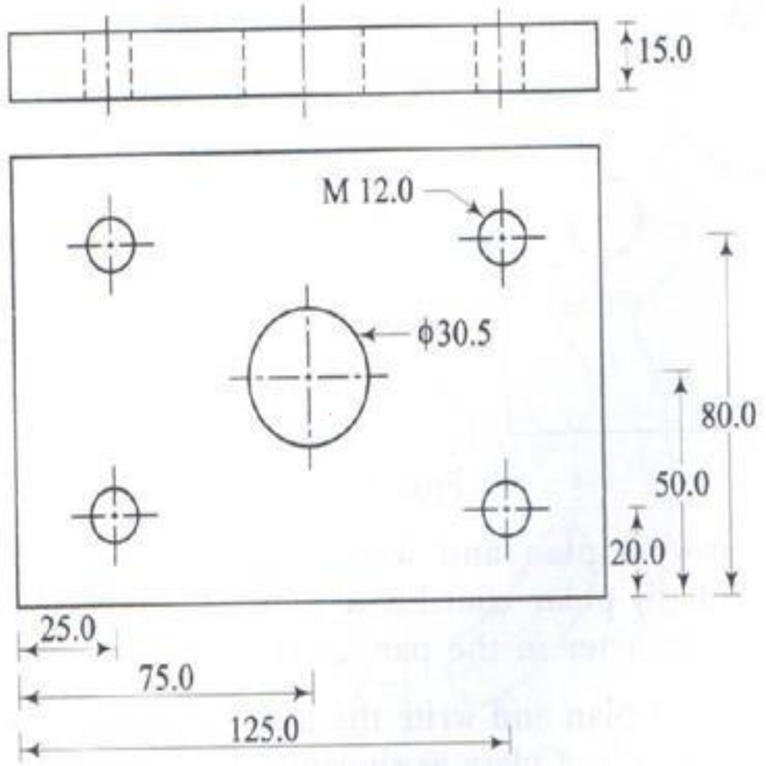
BLANK SIZE
100*100*20
DIA. 8, FIVE HOLES

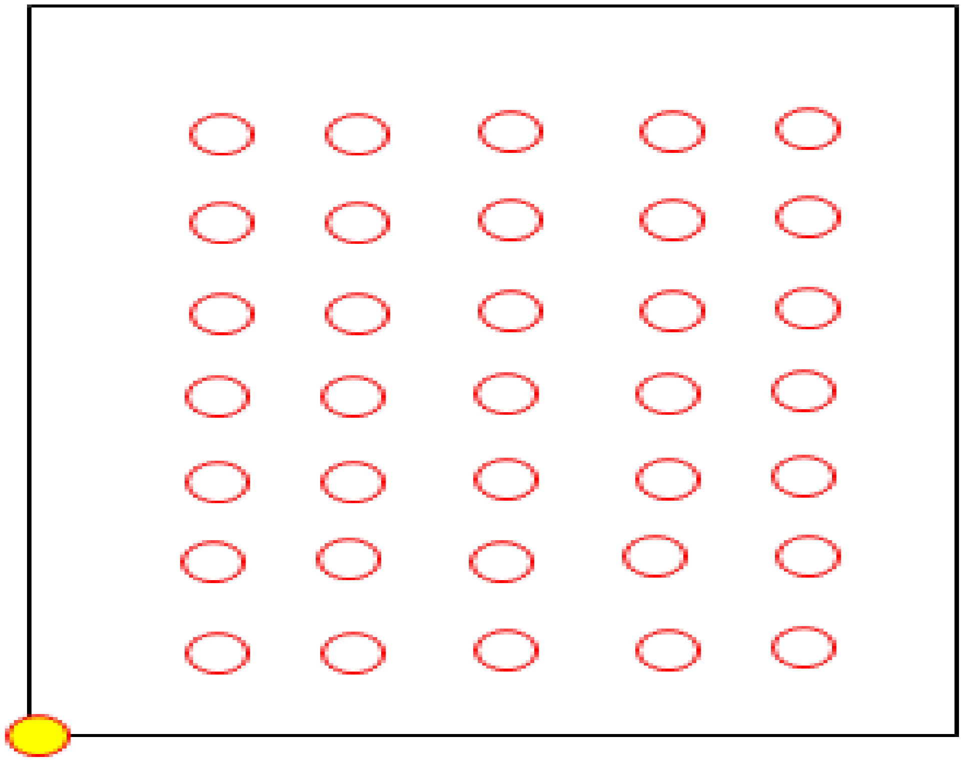
HOLE1	(20,20)
HOLE2	(20,80)
HOLE3	(80,80)
HOLE4	(80,20)
HOLE5	(50,50)

- O1975;
- N10 G21 G94;
- N20 G91 G28 Z0.0;
- N30 T01 M06;
- N40 G90 G54 G00 X10.0 Y10.0;
- N50 G43 H01;
- N60 S1200 M03 Z50;
- N70 G99 G81 X10.0 Y10.0 Z-5 R5.0 F300;
- N80 G91 X10 Y10 K8;
- N90 G00 Z50 M05;
- N100 G91 G28 Z0.0;
- N110 G91 G28 X0.0 Y0.0;
- N120 T02 M06;
- N130 G90 G54 G00 X40.0 Y10.0;
- N140 G43 H02;
- N150 S1200 M03 Z50;
- N160 G00 Z5 ;
- N170 G01 Z-3 F60;
- N180 X90 ;
- N190 Y20;
- N200 X40;
- N210 Y10;
- N220 G00 Z50;
- N230 G00 X10 Y80 ;
- N240 G00 Z5;
- N250 G01 Z-3 F60;
- N260 X60;
- N270 Y90;
- N280 X10;
- N290 Y80;
- N300 G00 Z50 M05;
- N310 G91 G28 Z0;
- N320 G28 X0 Y0:
- N330 M30;



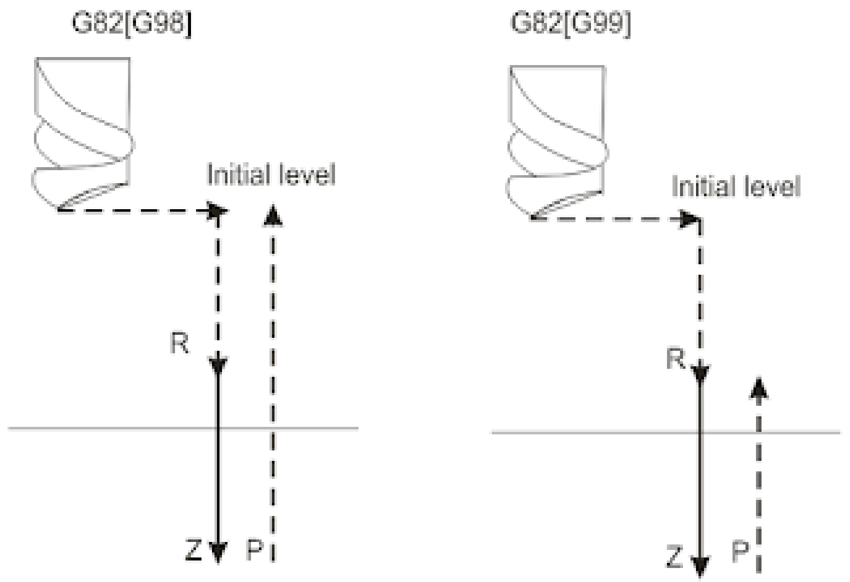
Exercises





Fanuc G82 counter boring cycle / Drilling cycle

G82 cycle is used for normal drilling . Cutting feed is performed to the bottom of the hole .At the bottom , a dwell is performed , the tool is retracted. This cycle is used to drill hole more accurately with respect to depth.



G82 X_ Y_ Z_ R_ P_ F_ K_;

Where , **XY-** Position of hole

Z- Depth of operation perform

R- R plane position

P- Dwell time when tool reach at bottom

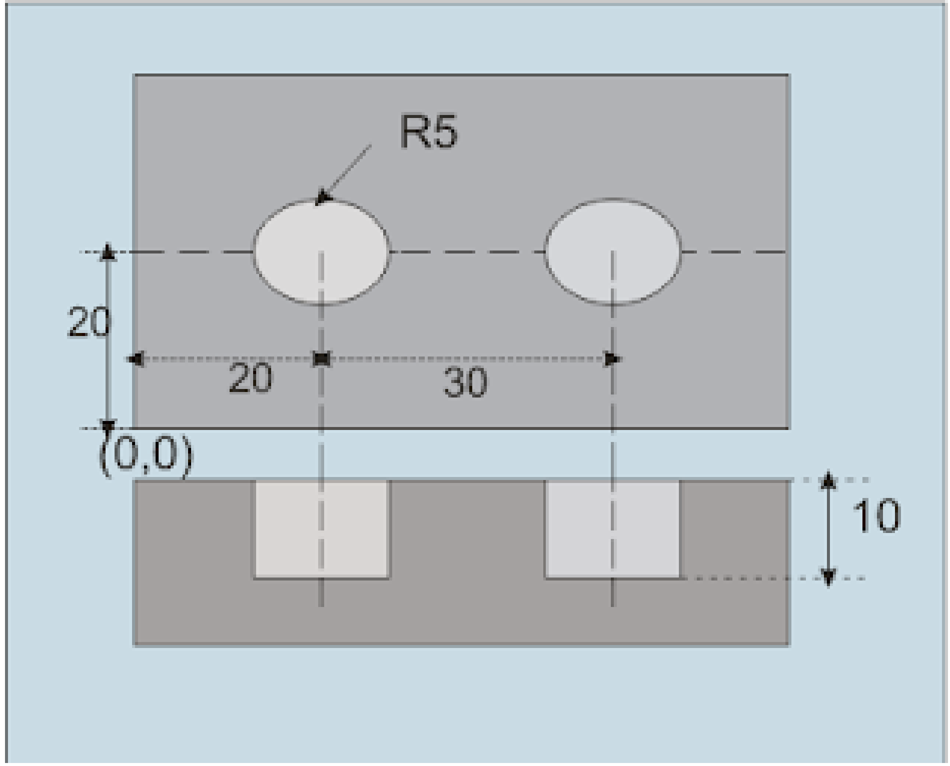
F- cutting feed rate

K- no of times operation repeats.

Operation :- First tool positioning at XY axis , after the tool rapidly traverse up to R-plane level , after that tool is start drilling operation . When the bottom of the tool has been reached, a dwell is performed. Then tool is retracted rapidly.

O5124

```
N10   M06 T04 ;
N20   G90 G80 G17 G00 G54 X0 Y0;
N30   G43 Z100 H4 ;
N40   M03 S1500 ;
N50   M07 ;
N60   G99 G82 X20 Y20 Z-10 R5  P1000 F120 ;
N70   X50 Y20 ;
N80   G98 G80 G00 Z100 ;
N90   M05 M09 M30 ;
```



DESCRIPTION OF PROGRAM

N10- Tool change command , select tool no. 4

N20- Absolute co-ordinate command , cancel canned cycle command , selection of XY plane, rapid command, work coordinate for tool positioning at X0 and Y0.

N30- Tool height offset compensation command , where tool is 100 along Z axis , tool height code H4.

N40- Spindle on clockwise , speed is 1500 rpm .

N50- Coolant ON .

N60- Return to R-plane in canned cycle , counter drilling command ,first drilling position is X20 and Y20 Depth of drilling is 10 , R- plane distance is 5 ,dwell time is 1 sec ,   feed rate is 120.

N70- Second drill position where X 50 and Y20 and drilling depth is 10.

N80-Tool return initial position , cancel canned cycle , rapid command , where tool along Z axis is 100 .

N90-Spindle off , coolant off, Main program end .

Fanuc G83 peck drilling cycle program

G83 peck drilling cycle perform intermittent cutting feed to the bottom of the hole while removing shaving from the hole.

G83 X_ Y_ Z_ R_ Q_ P_ F_ K_

Where , **XY-** Position of hole

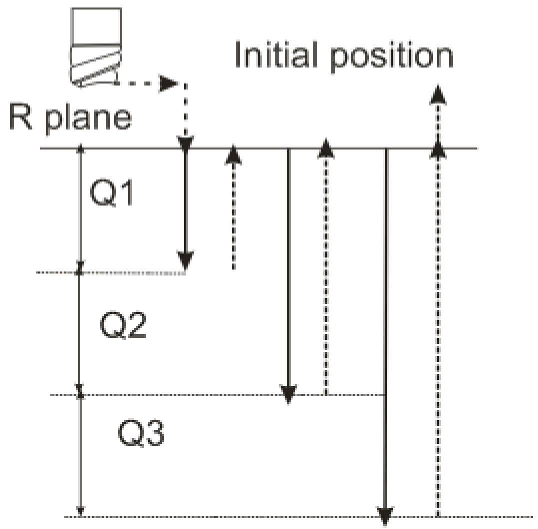
Z- Depth of operation perform

R- R plane position.

Q- depth of each cut

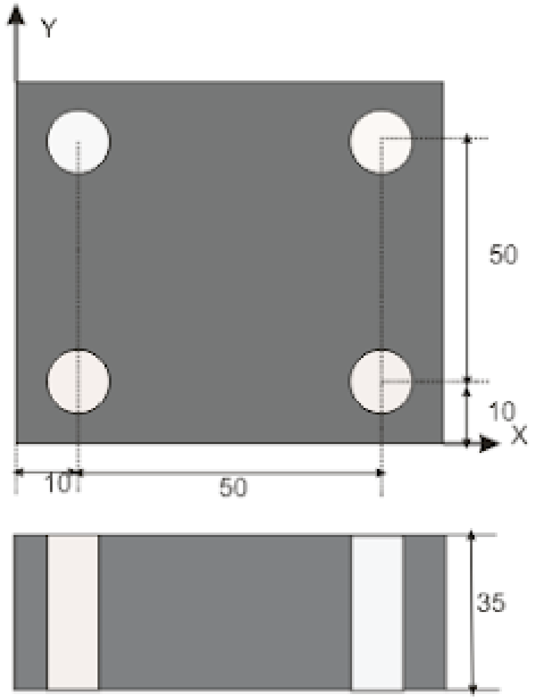
F- cutting feed rate

K- no of times operation repeats.



Operation:- In above fig Q represent depth of each cut . Cutting depth is always incremental value (suppose first cut is 20 mm then tool take second cut is increment by 20mm it means total depth is 40 mm). After each cut tool return at R plane , when operation is end tool return at initial position .

```
O5124
N10   M06 T05 ;
N20   G90 G80 G17 G00 G54 X0 Y0 ;
N30   G43 Z100 H4 ;
N40   M03 S1000 ;
N50   M07 ;
N60   G99 G83 X10 Y10 Z-40 R5 Q10 F75
N70   X60 Y10 ;
N80   X10 Y60 ;
N90   X60 Y60 ;
N100 G98 G80 G00 Z100 ;
N110 M05 M09 M30 ;
```



DESCRIPTION OF PROGRAM

N10- Tool change command , select tool no. 5

N20- Absolute co-ordinate command , cancel canned cycle command , selection of XY plane, rapid command, work coordinate for tool positioning at X0 and Y0.

N30- Tool height offset compensation command , where tool is 100 along Z axis , tool height code H4.

N40- Spindle on clockwise , speed is 1000 rpm .

N50- Coolant ON .

N60- Return to R-plane in canned cycle , Peck drill command ,first drilling position is X10 and Y10 Depth of boring is 40(from R-plane) , R- plane distance is 5 , each cut is 10 mm , feed rate per minute is 75 .

N70- Second drilling position is X60 and Y10;

N80- Third drilling position is X10 and Y60;

N90- Fourth drilling position is X60 and Y60;

N100- Tool return initial position , cancel canned cycle , rapid command , where tool along Z axis is 100 .

N110- Spindle off , coolant off, Main program end .

G84 Tapping Cycle Example CNC Program

Fanuc G84 cycle performs tapping. In this tapping cycle, when the bottom of the hole has been reached, the spindle is rotated in the reverse direction.

G84 X_ Y_ Z_ R_ P_ F_ K_

Where , **XY-** Position of hole

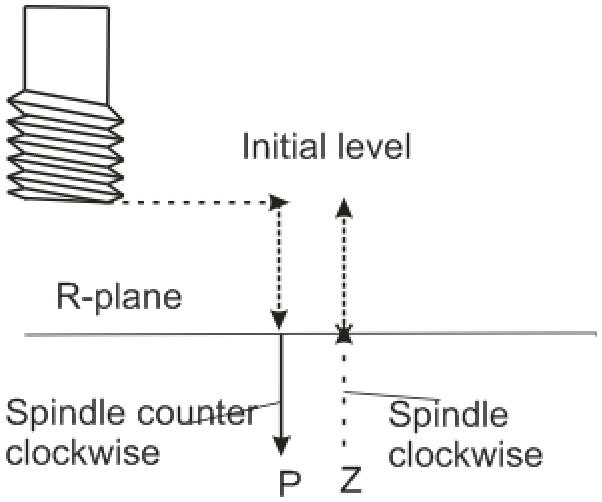
Z- Depth of operation perform from retraction plane

R- R plane position.(retraction plane)

P- Dwell time

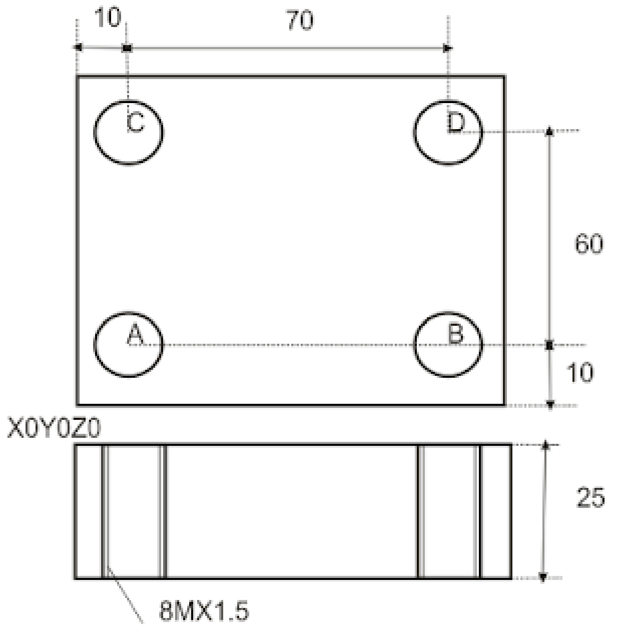
F- cutting feed rate

K- no of times operation repeats.



Operation:- Tapping is performed by rotating the spindle clockwise. When the bottom of the hole has been reached, the spindle is rotated in the reverse direction for retraction. This operation creates threads.

```
O5124
N10   M06 T07 ;
N20   G90 G80 G17 G00 G54 X0 Y0 ;
N30   G43 Z100 H1 ;
N40   M03 S1000 ;
N50   M07 ;
N60   G99 G84 X10 Y10 Z-30 R5 P300 F1.25 ;  [A]
N70   X80 Y10 ;                              [B]
N80   X10 Y70 ;                              [C]
N90   X80 Y70 ;    [D]
N100 G98 G80 G00 Z100 ;
N110 M05 M09 M30 ;
```



DESCRIPTION OF PROGRAM

N10- Tool change command , select tool no. 7

N20- Absolute co-ordinate command , cancel canned cycle command , selection of XY plane, rapid command, work coordinate for tool positioning at X0 and Y0.

N30- Tool height offset compensation command , where tool is 100 along Z axis , tool height code H1.

N40- Spindle on clockwise , speed is 1000 rpm .

N50- Coolant ON .

N60- Return to R-plane in canned cycle , tapping cycle command ,first tapping position is X10 and Y10 Depth of tapping is 30(from R-plane) , R-plane distance is 5 , dwell time 300 ,   feed rate is 1.25 .[A]

N70- Second tapping position is X80 and Y10; [B]

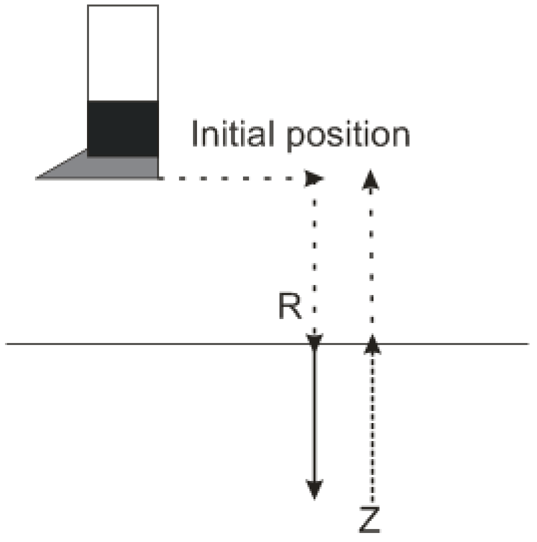
N80- Third tapping position is X10 and Y70; [C]

N90- Fourth tapping position is X80 and Y70; [D]

N100- Tool return initial position , cancel canned cycle , rapid command , where tool along Z axis is 100 .

N110- Spindle off , coolant off, Main program end .

Fanuc G85/G86 boring cycle



Fanuc G85/G86 cycle is used to bore a hole .

G85/G86 X_ Y_ Z_ R_ F_ K_

Where , **XY-** Position of hole

Z- Depth of operation perform from retraction plane

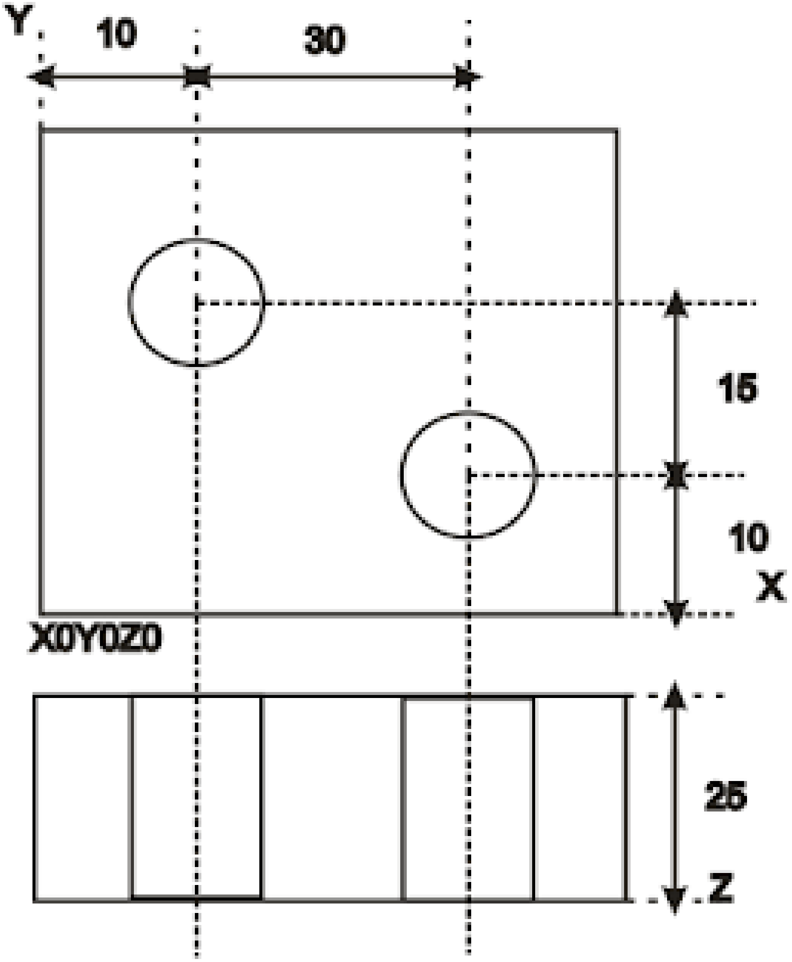
R- R plane position.(retraction plane)

F- cutting feed rate

K- no of times operation repeats.

Operation:- First positioning of tool at XY plane. After that tool rapidly traverse up to R plane , then boring operation start from R plane to point Z. After that tool rapidly return to initial position.

```
O5130
N10   M06 T08 ;
N20   G90 G80 G17 G00 G54 X0 Y0 ;
N30   G43 Z100 H1 ;
N40   M03 S1000 ;
N50   M07 ;
N60   G99 G86 X10 Y25 Z-30 R5  F100 ;
N70   X40 Y10 ;
N80   G98 G80 G00 Z100 ;
N90   M05 M09 M30 ;
```



DESCRIPTION OF PROGRAM

N10- Tool change command , select tool no. 8

N20- Absolute co-ordinate command , cancel canned cycle command , selection of XY plane, rapid command, work coordinate for tool positioning at X0 and Y0.

N30- Tool height offset compensation command , where tool is 100 along Z axis , tool height code H1.

N40- Spindle on clockwise , speed is 1000 rpm .

N50- Coolant ON .

N60- Return to R-plane in canned cycle , Boring cycle command ,first boring position is X10 and Y25 Depth of tapping is 30(from R-plane) , R- plane distance is 5 , feed rate is 100 .

N70- Second boring position is X40 and Y10; [

N80- Tool return initial position , cancel canned cycle , rapid command , where tool along Z axis is 100 .

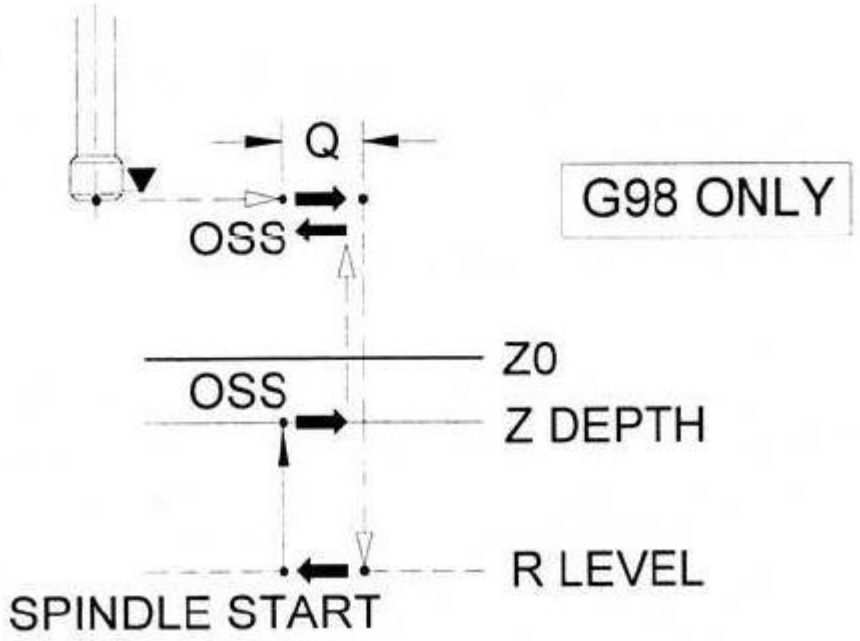
N90- Spindle off , coolant off, Main program end .

G87 Back Boring cycle

The G87 Back Boring cycle is a special cycle, its practical usage is limited due to the special tooling and setup requirements. Use the G87 Cycle only if the total costs can be justified economically. The boring bar must be set very carefully, it must be preset to match the diameter required for backboring, its cutting point must be set in the spindle oriented mode, facing the opposite direction than the shift direction.

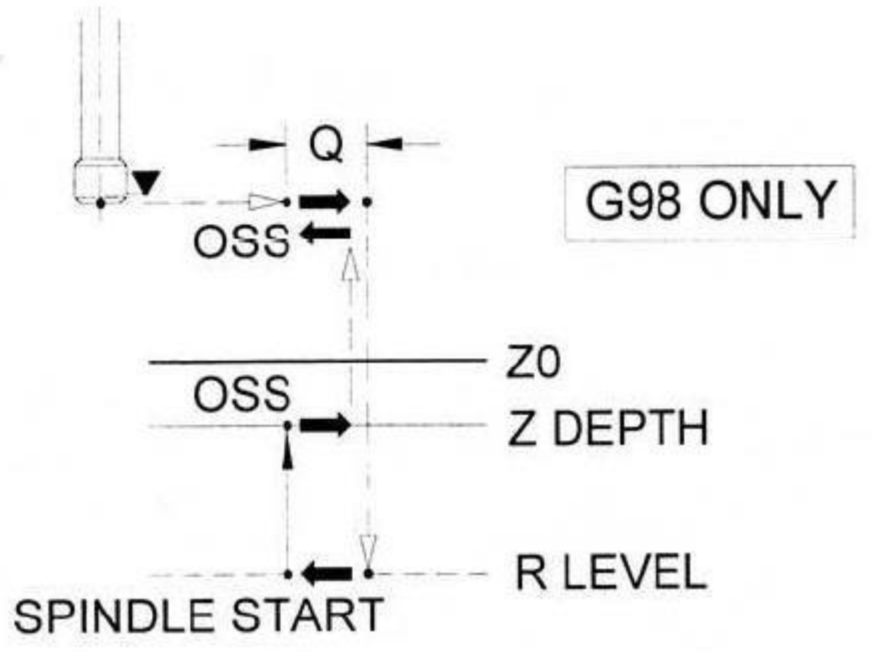
Code line for G87 Back Boring cycle:
There are two program formats available for the G87 back boring canned cycle. Unfortunately G99 is never used with the G87 cycle.
First one using the Q, which is commonly used:
N100 G98 G87 X… Y… R… Z… Q… F…
Second one using I and J:
N100 G98 G87 X… Y… R… Z… I… J… F…

Diagram for G87 Back Boring cycle:



Steps for the G87 Back Boring cycle:

1. Rapid motion to XY position of the hole position.
2. Spindle Rotation Stop.
3. Spindle Orientation.
4. Shift OUT (OSS) by the Q value or shift by the amount and direction of I and J.
5. Rapid motion to the R level, i.e., to the bottom of the hole position.
6. Shift IN (OSS)by the Q value or shift back in the opposite direction of I and J.
7. Spindle rotation ON (M03).
8. Feedrate motion to the depth in Z.
9. Spindle rotation STOP.
10. Spindle orientation.
11. Shift OUT (OSS) by the Q value or shift by the amount and direction of I and J.
12. Rapid motion to the Initial level, i.e., to the top of the hole position.
13. Shift (OSS) IN by the Q value or shift back in the opposite direction of I and J.
14. Spindle rotation ON.



Milling Tools

HSS Flat Endmills

4-flute

3-flute

2-flute

Carbide Insert Endmills

Milling Tools (Cont'd)

HSS Ball Endmills

4-flute

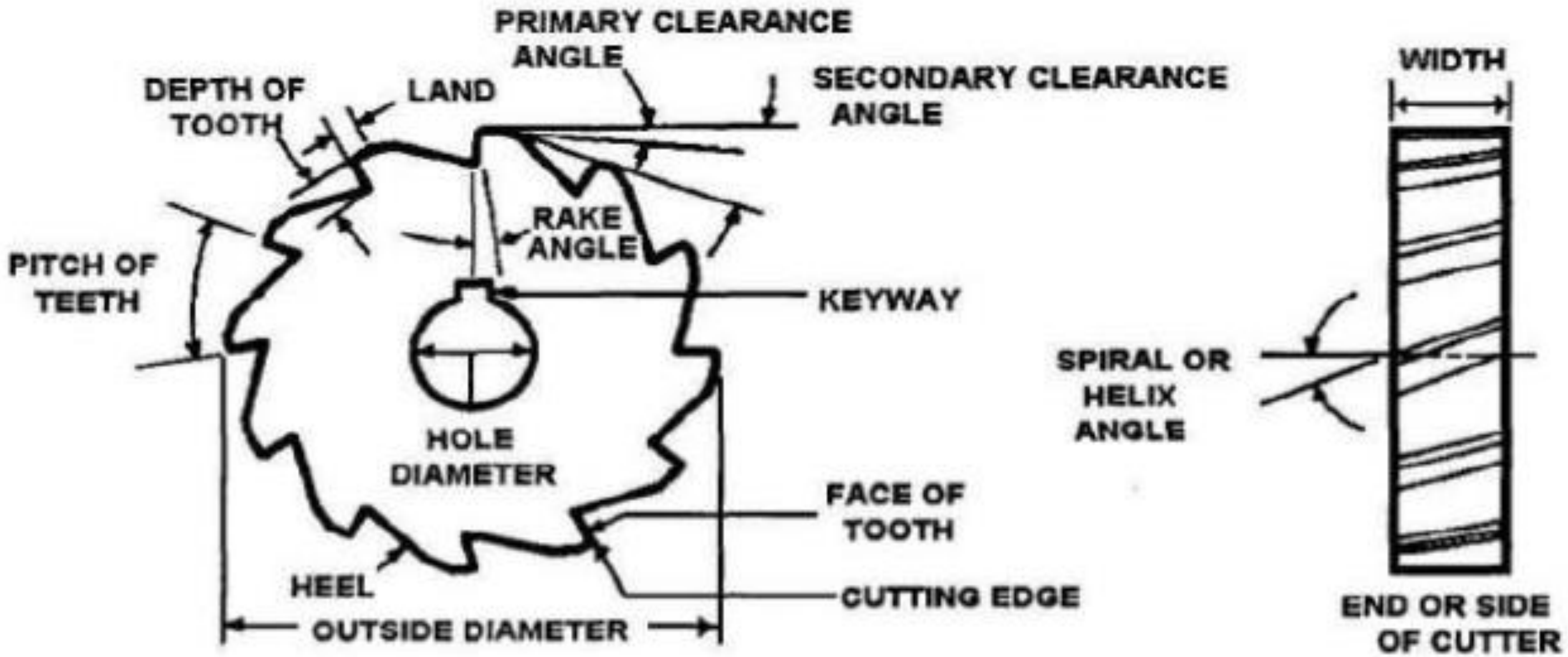
3-flute

2-flute double ended

Carbide Insert Ball Endmills

Milling cutter nomenclature

NOMENCLATURE OF MILLING CUTTER



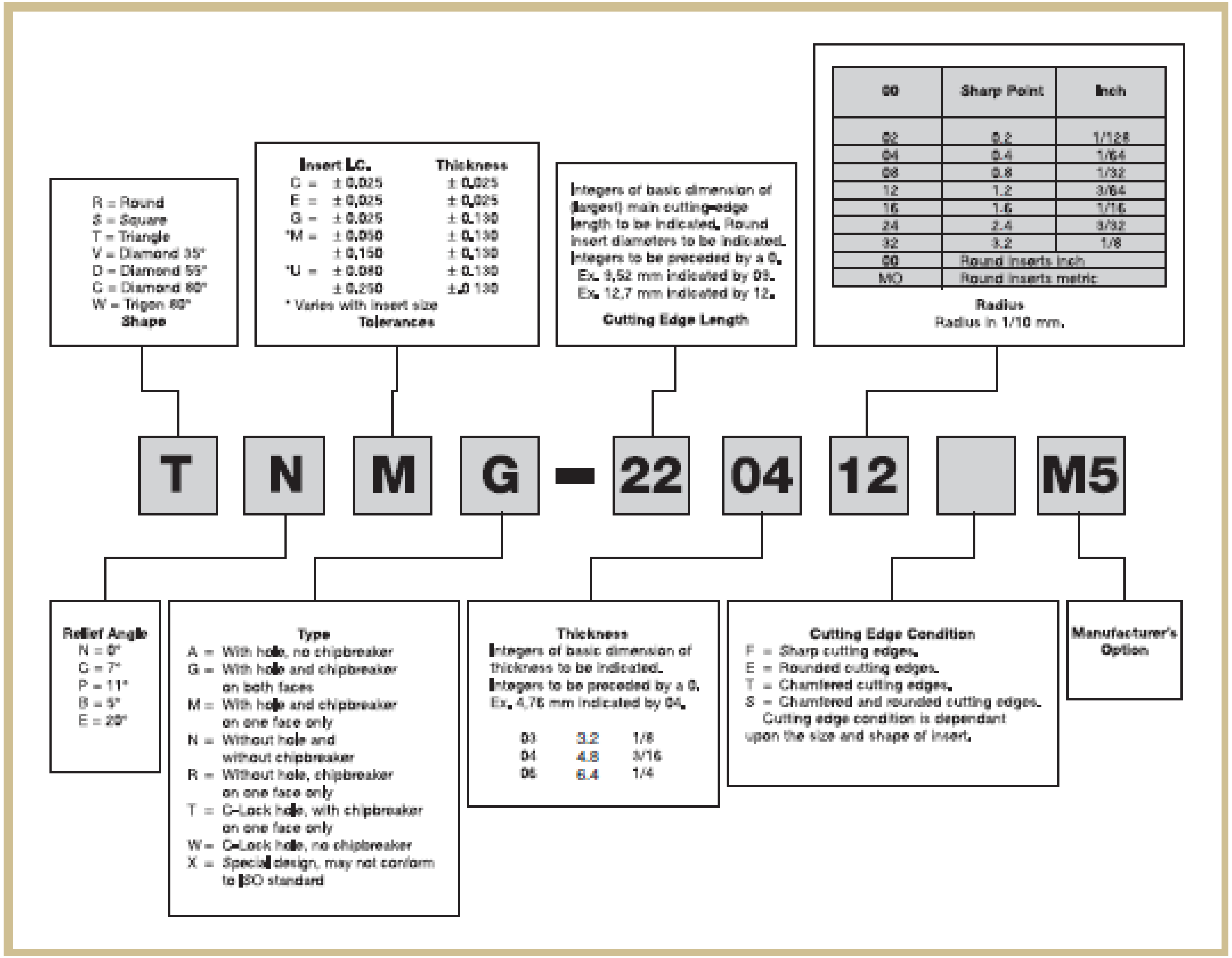


FIGURE 42—Shown here is the code key for milling-machine insert designations. **(Courtesy of Carboloy Inc., www.carboloy.com)**

Tool holder Selection – Milling

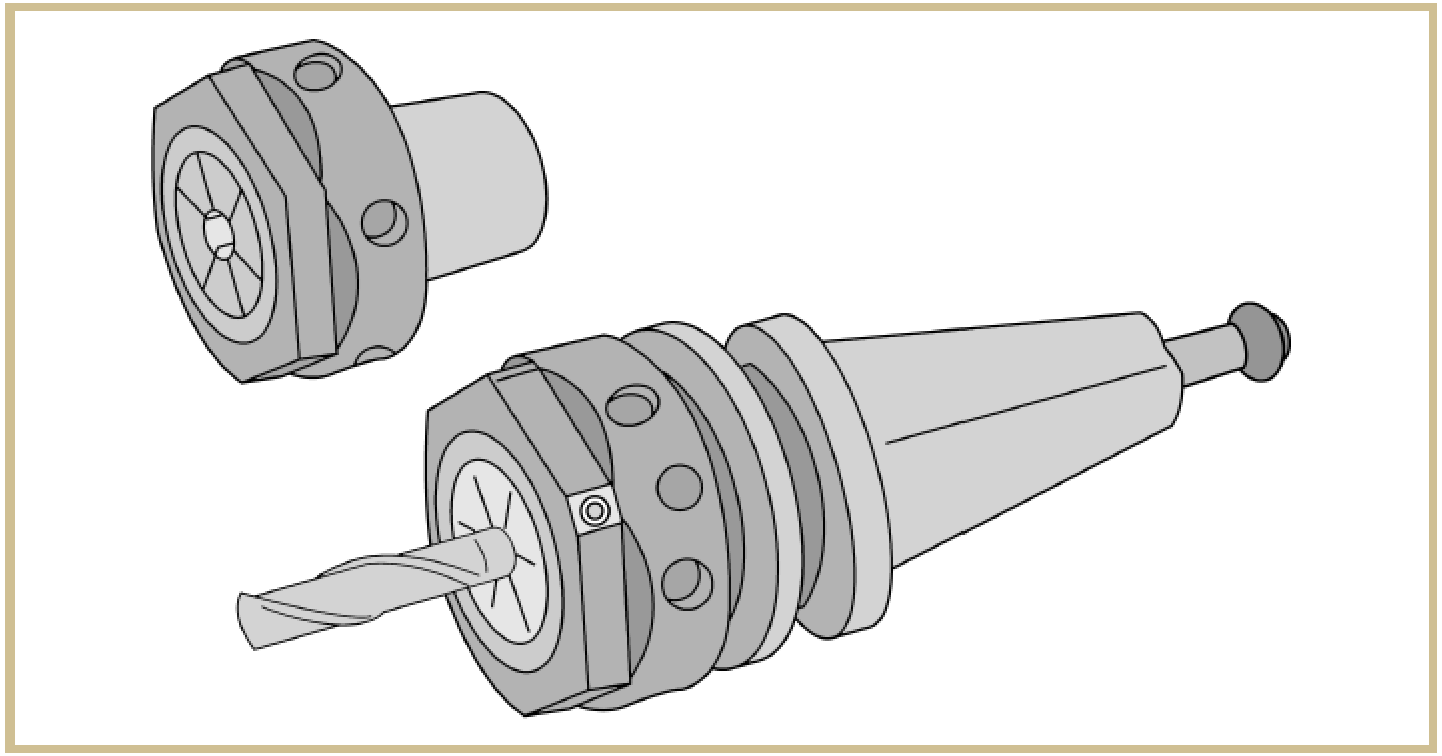
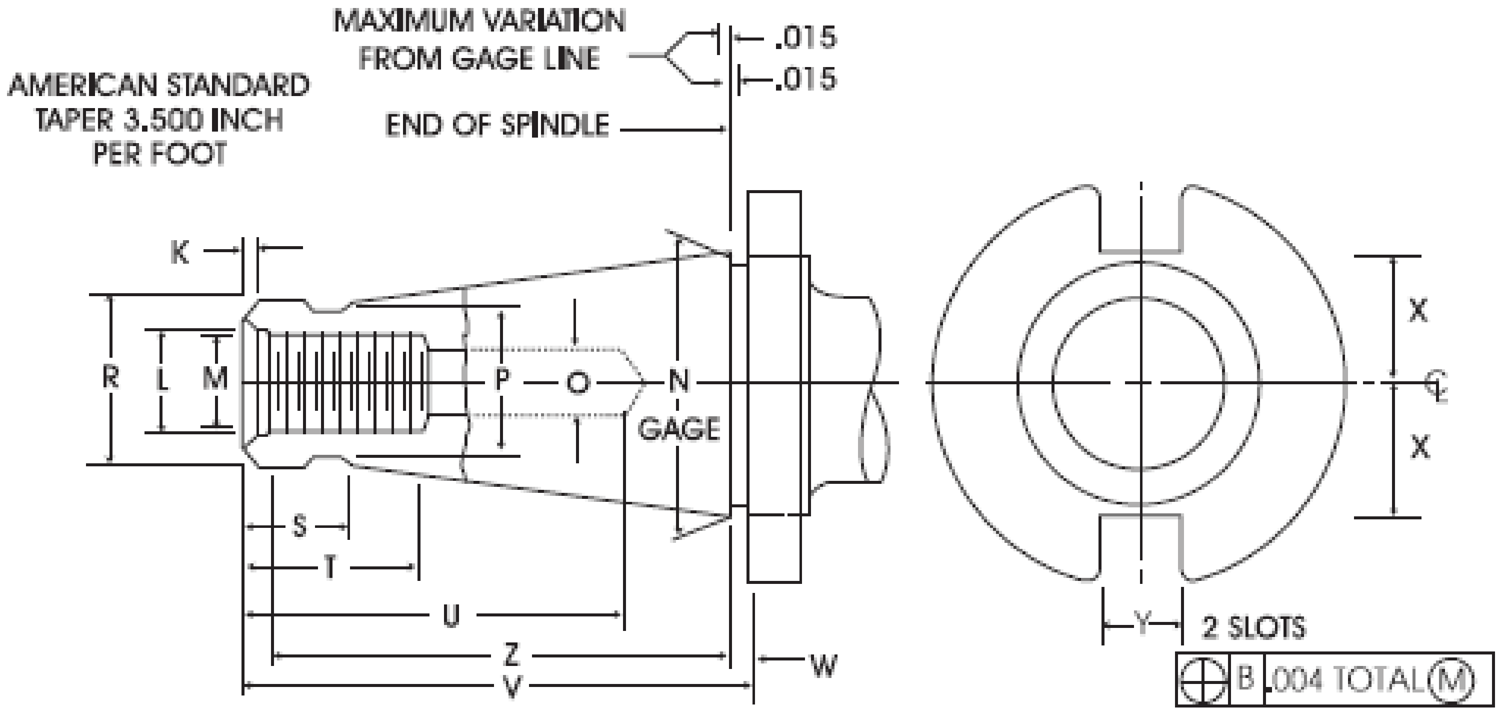


FIGURE 38—The spindles of milling machines have standard dimensions to ensure compatibility with different manufacturers' products. These tapers are a self-releasing type so that tools can be rapidly changed by automatic toolholder changes.



N- GAGE DIAMETER OF TAPER
O- TAP DRILL SIZE FOR DRAW-IN-THREAD
P- DIAMETER OF NECK
M- SIZE OF THREAD FOR DRAW-IN-BOLT UNC-2B
R- PILOT DIAMETER
S- LENGTH OF PILOT
T- MINIMUM LENGTH OF USABLE THREAD
U- MINIMUM DEPTH OF CLEARANCE HOLE
V- DISTANCE FROM REAR OF FLANGE TO END OF ARBOR
W- CLEARANCE OF FLANGE FROM GAGE DIAMETER
X- TOOL SHANK CENTERLINE TO DRIVING SLOT
Y- WIDTH OF DRIVING SLOT
Z- DISTANCE FROM GAGE LINE TO BOTTOM OF C-BORE
K- DEPTH OF 60° CENTER
L- DIAMETER OF C–BORE

FIGURE 39—Toolholders have standard dimensions to ensure proper fit within spindles of machine tools. **(Copyright © ANSI 1997-2000, ANSI B5.18-1972, R1991. Source: *Machinery's Handbook,* 25th Edition, page 920, Table 2, Essential Dimensions of American National Standard Tool Shanks for Milling Machines)**

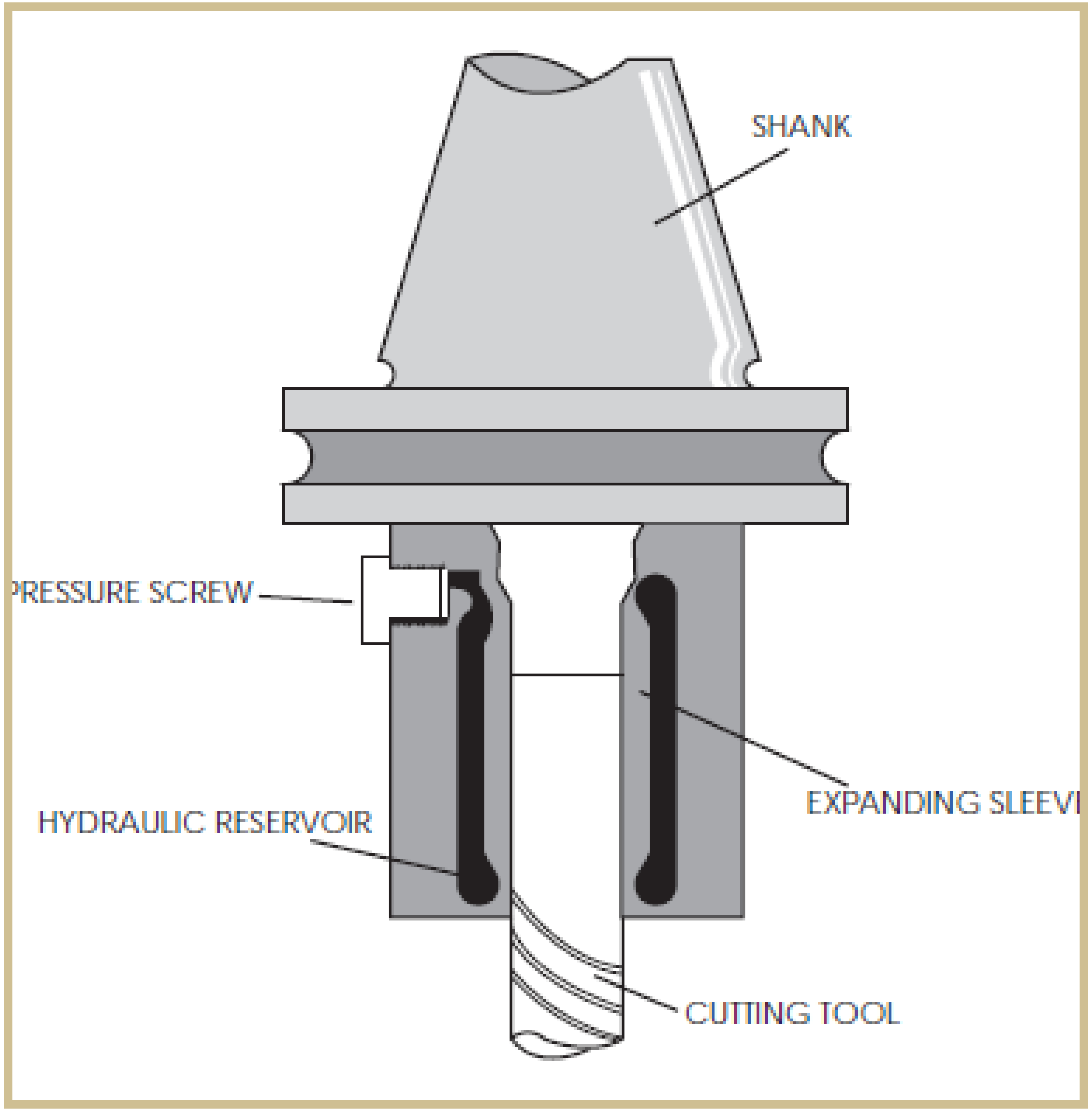


FIGURE 48—Hydraulic toolholders provide high clamping forces, good concentricity, and few unbalances. Clamping forces are generated by a piston acting on a hydraulic reservoir within the end of the toolholder that forces an expanding sleeve to grip the cutting tool.

FIGURE 49—High centrifugal forces in the rotating spindle actually cause the inside diameter of the spindle to increase slightly. This change can cause the toolholder to be drawn farther into the spindle, causing a change in the Z-axis position and also making removal of the tool difficult when the spindle is stopped.

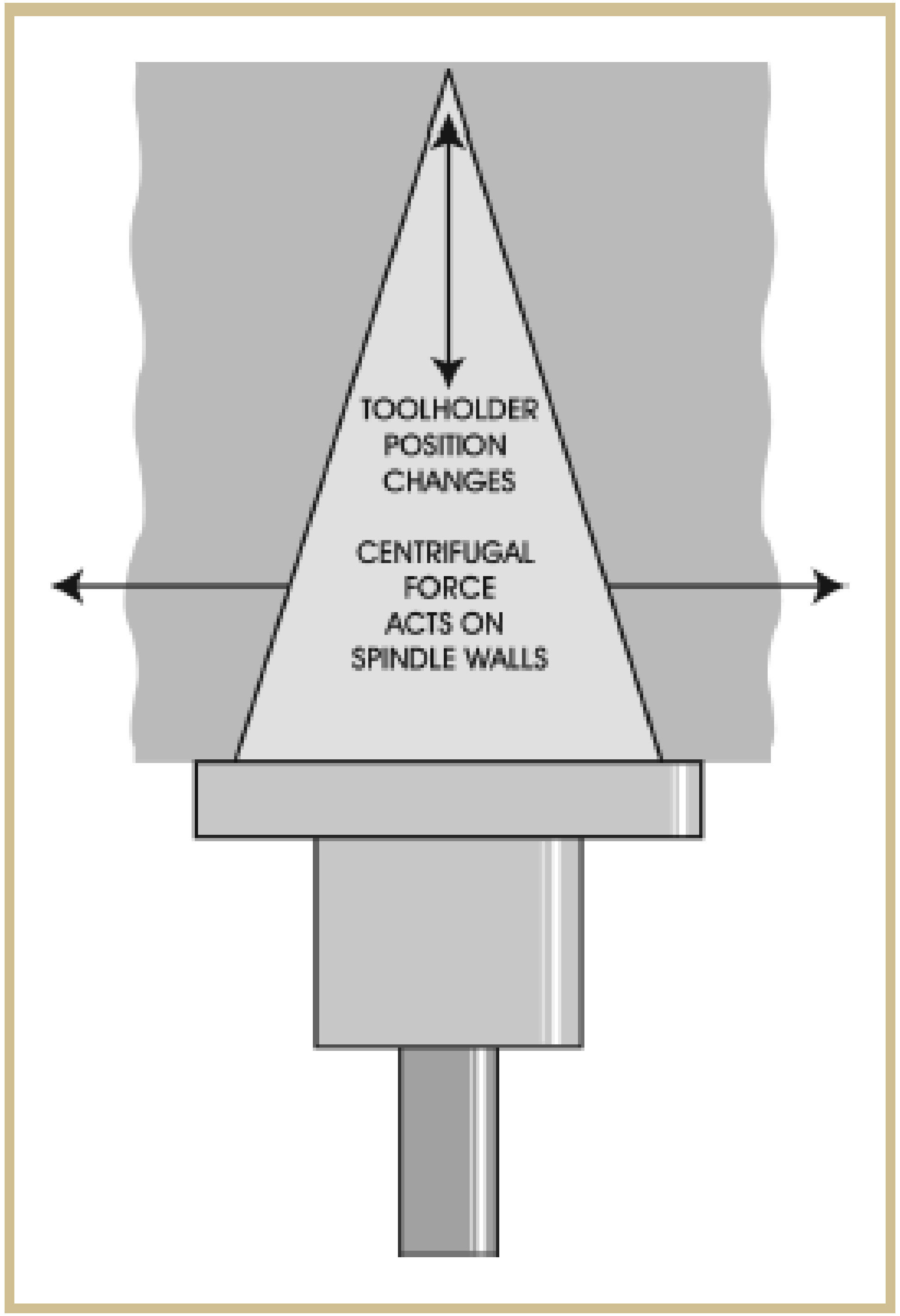
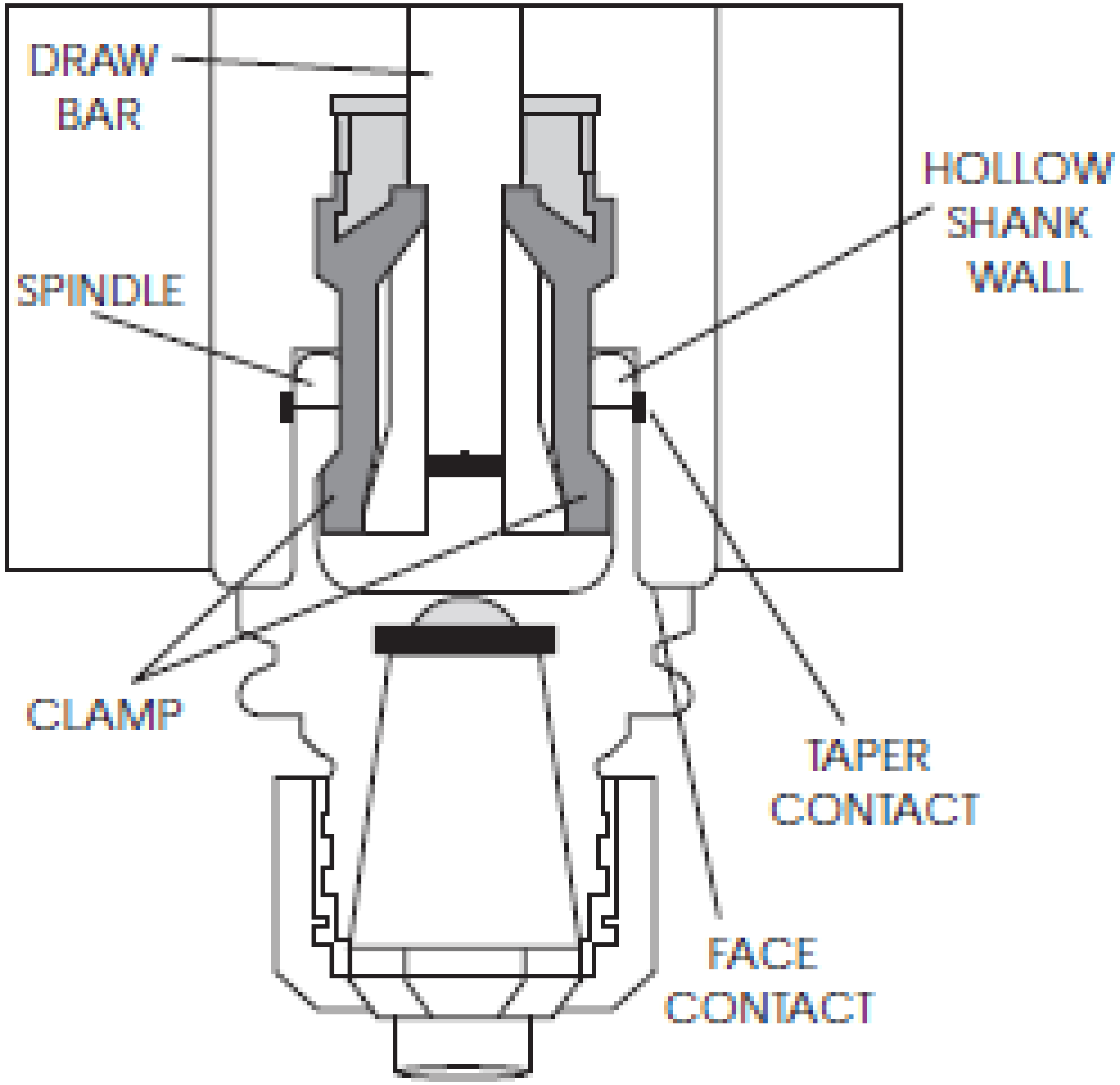
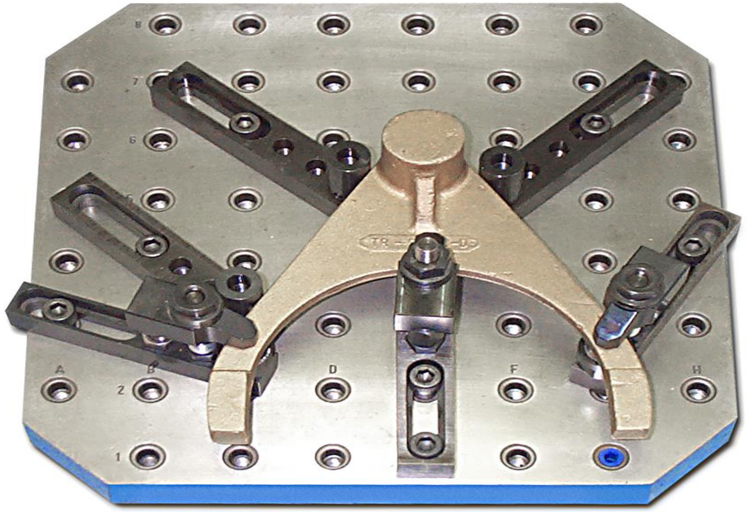
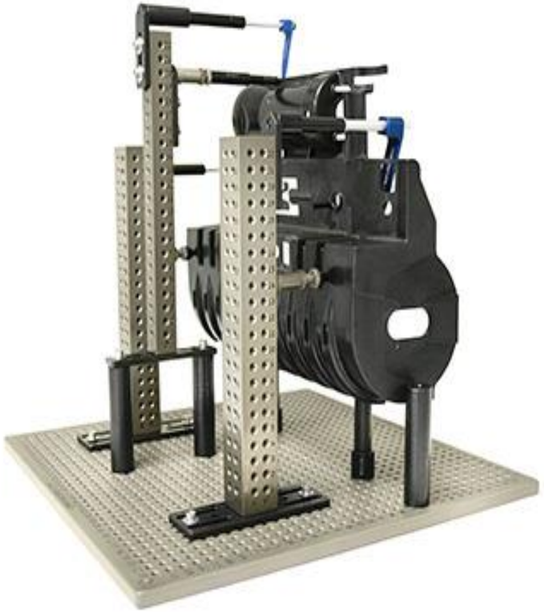


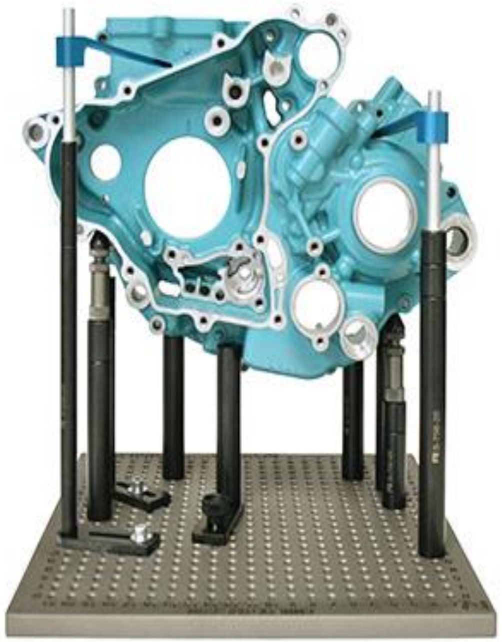
FIGURE 50—The HSK system, which uses a hollow shank and taper, is a type of spindle system for machine tools.



Work Holding Device - Milling







Equipment

Pallets

- The workpiece is placed on a pallet (module) which can be oriented in different directions by the machine

Automatic Pallet Changers

- When the workpiece is finished, automatic pallet changers remove it and replace it with another workpiece

Pallets

Example of a part mounted on a pallet

Courtesy Toth Industries

Pallet Changers

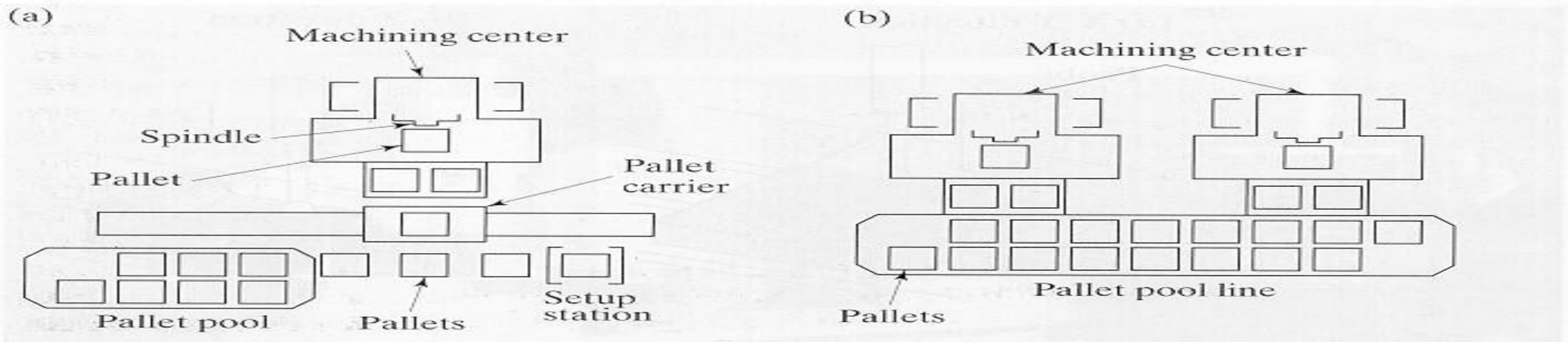


FIGURE 24.4 (a) Schematic illustration of the top view of a horizontal-spindle machining center showing the pallet pool, set-up station for a pallet, pallet carrier, and an active pallet in operation (shown directly below the spindle of the machine). (b) Schematic illustration of two machining centers with a common pallet pool. Various other arrangements are possible in such systems. *Source*: Hitachi Seiki Co., Ltd.

Discussion

10 mins