IE360: CAD/CAM

Computer Aided Design and Computer Aided Manufacturing

Lecture (11)

Manual Part Programming (cont.)

Dr Ashraf Afifi

Industrial Engineering Department
King Saud University

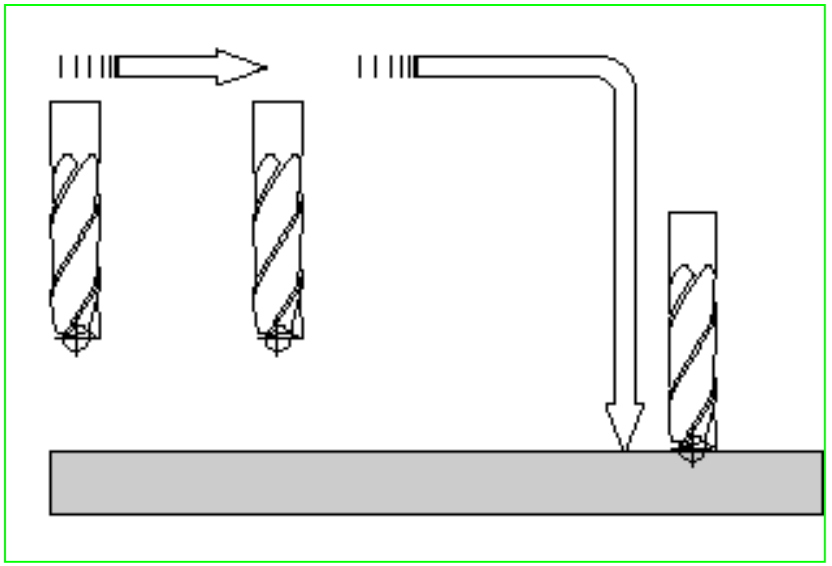
Outline

- Detailed description of G codes
  - G00: Rapid positioning
  - G01: Linear interpolation
  - G02: Circular interpolation (clockwise)
  - G03: Circular interpolation (counterclockwise)
  - G20: Inch units
  - G21: Metric units
  - G90: Absolute positioning
  - G91: Incremental positioning
  - G70: Profile finishing cycle (turning)
  - G71: Profile roughing cycle for turning and boring
  - G74: Peck drilling cycle along the z axis
  - G81: Drilling cycle
  - G98: Return to the initial plane
- Detailed description of M codes
  - M30: Program end, reset to start
  - M03: Spindle rotation normal – clockwise (CW)
  - M04: Spindle rotation reverse – counterclockwise (CCW)
  - M05: Spindle rotation stop
  - M06: Tool change
  - M08: Coolant on
  - M09: Coolant off
- Examples & Exercises

G00: Rapid positioning
Format: N_ G00 X_ Y_ Z_

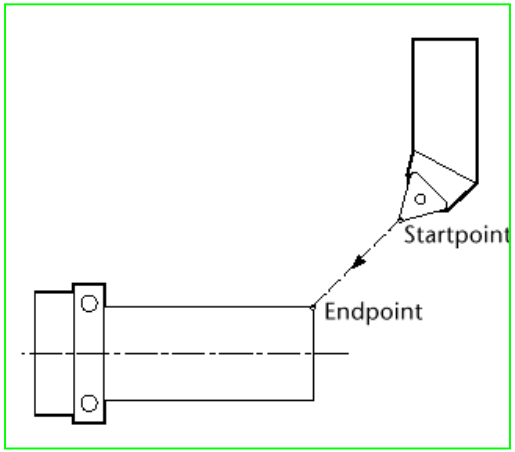
- The G00 command is a ***rapid tool move***, which is used to move the tool linearly from position to position without cutting any material, thus allowing for quick tool positioning.

- It is a modal command, remaining in effect until canceled by another G code command.

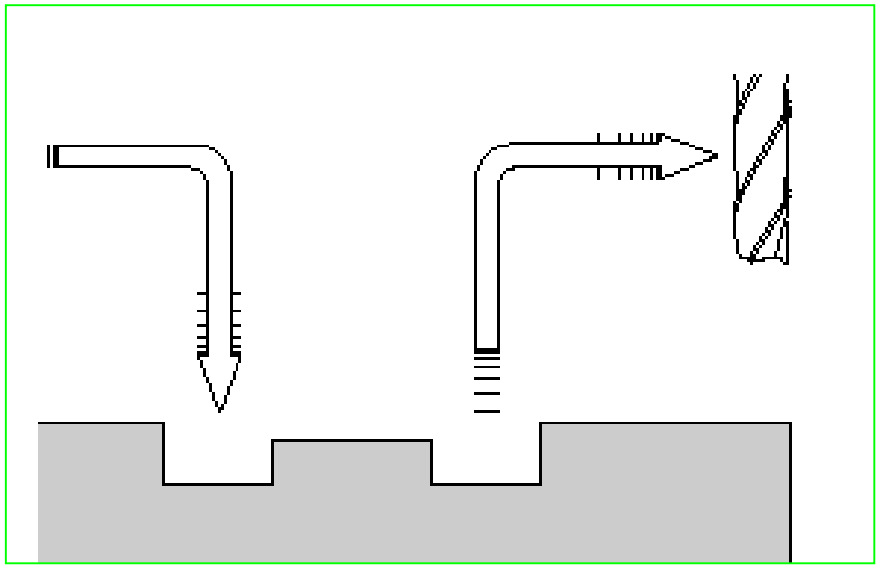


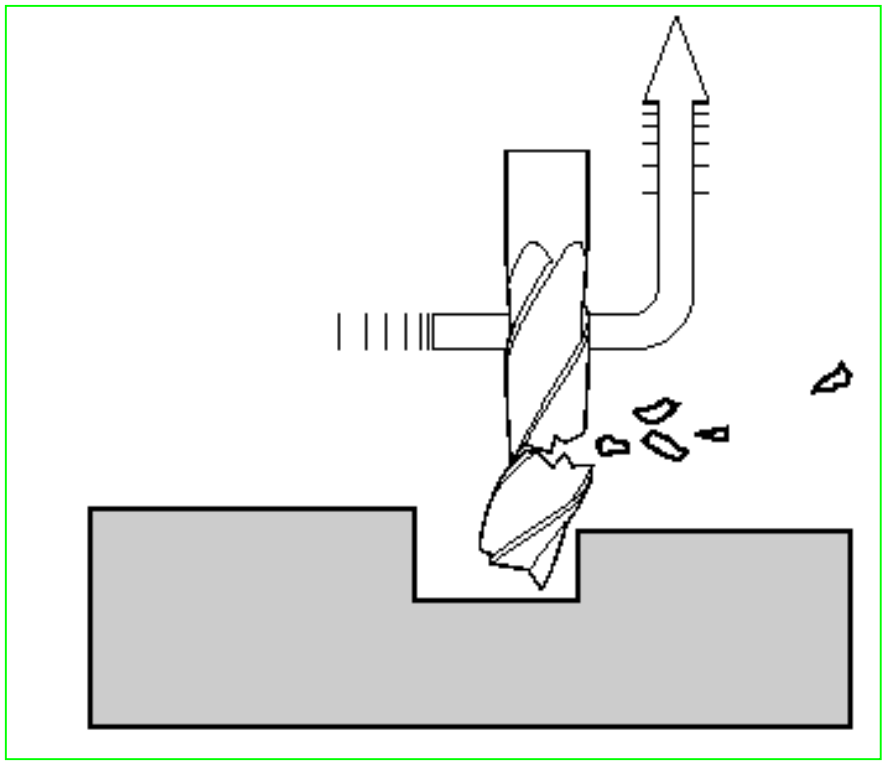
- **Example:** N15 G00 X1.8 Z0.0

In this example, the tool positions in rapid mode from its present location to a point at X1.8, Z0.0.



- The G00 command should have two distinct movements to ensure that vertical moves are always separate from horizontal moves.
  - In a typical rapid move toward the part, the tool first moves horizontally in the XY plane. Then, it feeds down in the Z axis direction.
  - When moving out of the part, the tool first goes up in the Z axis direction, then laterally in the XY plane.
- As the opposite diagram shows, if the above rules are not followed, an accident can result.





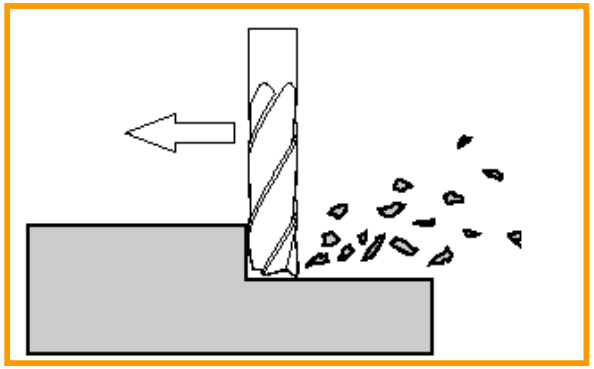
G01: Linear interpolation

Format: N_ G01 X_ Y_ Z_ F_

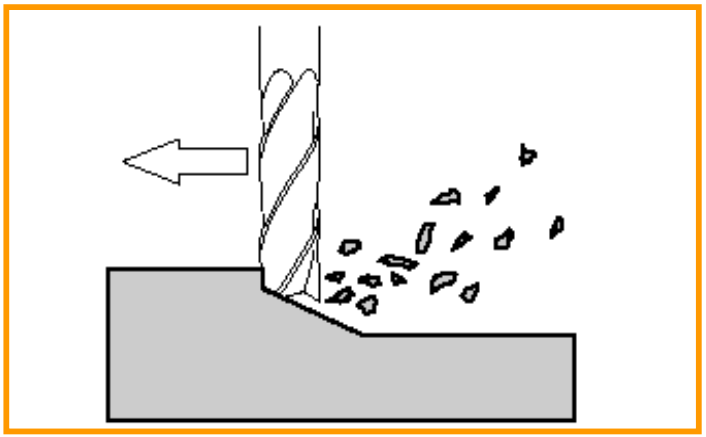
- The G01 command is specifically used for the linear removal of material from a workpiece, in any combination of the X, Y, or Z axes. The machine tool follows a linear trajectory.

- The G01 is modal and requires a user variable **feedrate** (designated by the letter F followed by a number).

- Linear interpolation, or straight-line feed moves, on the flat XY plane (no Z values are specified).

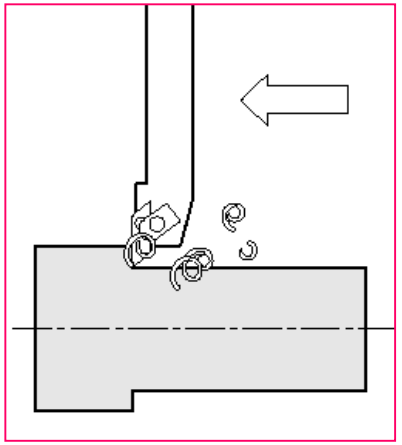


- Diagonal feed moves are a result of a G01 command, where two or more axes are used at once.



- **Example:** G01 Z-2.5 F0.01

In this example, the tool cuts a straight line from its present location to a point at Z-2.5 at a rate of 0.01 ipr.



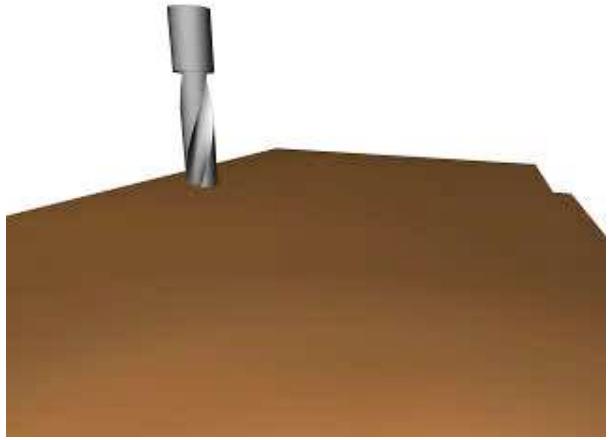
G02: Circular interpolation (clockwise)

Format: N_ G02 X_ Y_ Z_ I_ J_ K_ F_ Or N_ G02 X_ Y_ Z_ R_ F_

G03: Circular interpolation (counterclockwise)

Format: N_ G03 X_ Y_ Z_ I_ J_ K_ F_ Or N_ G03 X_ Y_ Z_ R_ F_

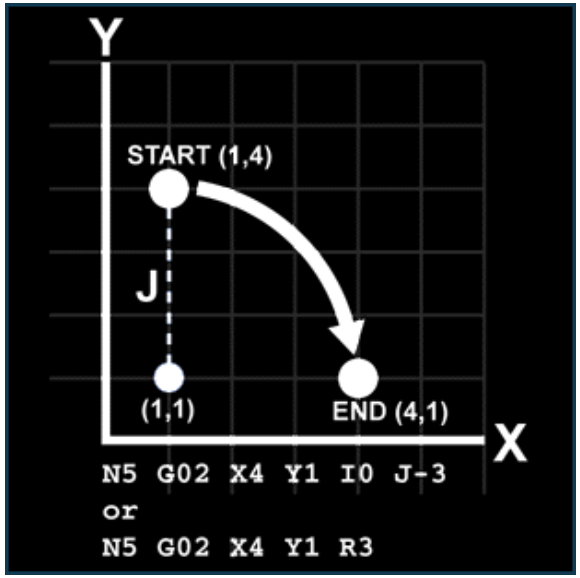
- Circular interpolation is more commonly known as radial (or arc) feed moves.
- The G02 and G03 commands are respectively used for all clockwise and counterclockwise radial feed moves, whether they are quadratic arcs, partial arcs, or complete circles, as long as they lie in any one plane.
- The G02 and G03 commands are modal and are subject to a user-definable feedrate.



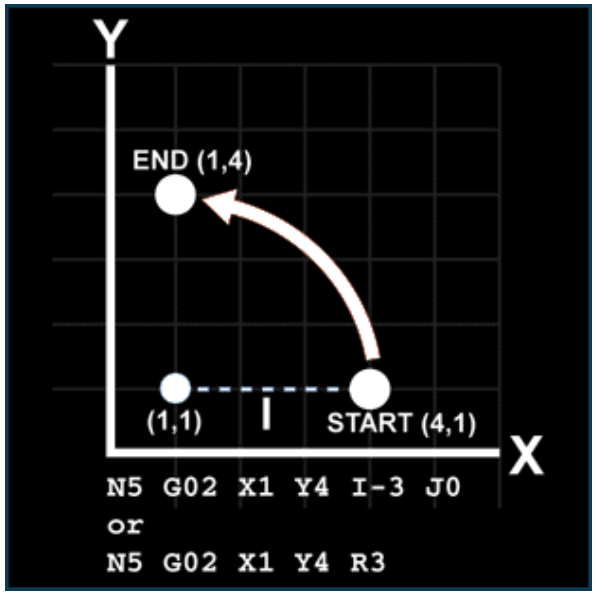
- The G02 and G03 commands require an endpoint and a radius in order to cut the arc.

- To find the radius, simply measure ***the center of the tool at the start of the arc to the center of the arc***. This radius is written in terms of the X and Y distances. To avoid confusion, these values are assigned variables, called I and J, respectively.

- G02 and G03 can also be specified by entering the X and Y endpoints and then R for the radius. However, the R value is limited to a maximum movement of 90°.



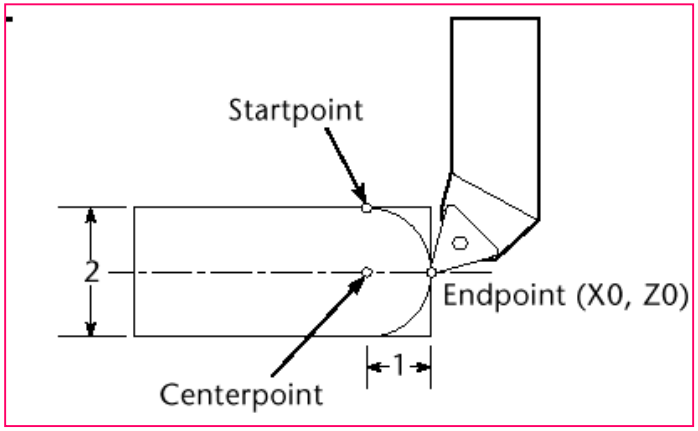
G02: Circular interpolation (CW)



G03: Circular interpolation (CCW)

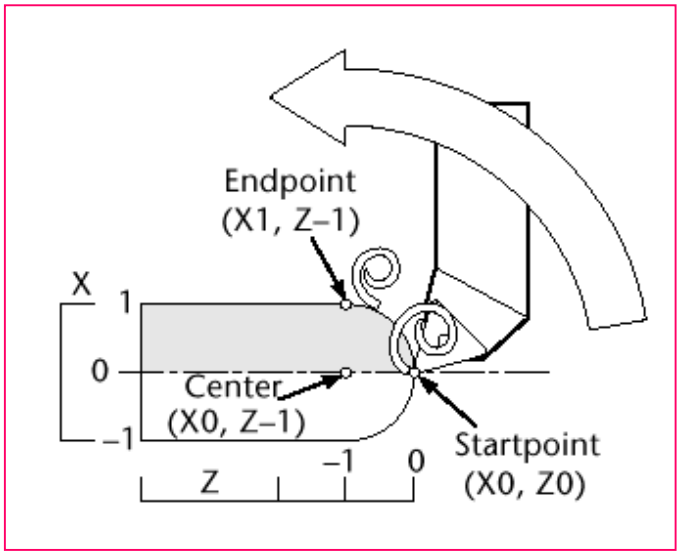
- G02 Example (turning):
  - N05 G01 X1.0 Z-1.0 F0.012
  - N10 G02 X0.0 Z0.0 I-1.0 K0.0
  - Or N10 G02 X0.0 Z0.0 R1.0

In this example, the tool cuts a **clockwise** arc from its present position to (X0, Z0) at a feedrate of 0.012 ipr.



- G03 Example (turning):
  - N10 G01 X0.0 Z0.0 F0.012
  - N15 G03 X1.0 Z-1.0 I0.0 K-1.0

In this example, the tool cuts a **counterclockwise** arc from its present position to (X1, Z-1) at a feedrate of 0.012 ipr.



G20: Inch units Format: N_ G20

G21: Metric units Format: N_ G21

- The G20 command defaults the system to inch units.
  - When a program is being run and the G20 command is encountered, all coordinates are stated as **inch** units.
  - This command is usually found at the beginning of a program. However, on some controllers it can be used to switch from metric units in the middle of a program.
- The G21 command defaults the system to metric units.
  - When a program is being run and the G21 command is encountered, all coordinates are stated as **millimeter** units.
  - This command is usually found at the beginning of a program. However, it can be used to switch between metric and inch units in the middle of a program.

G90: Absolute positioning Format: N_ G90

G91: Incremental positioning Format: N_ G91

- The **G90** command defaults the system to accept all coordinates as absolute coordinates.

- These coordinates are measured from a fixed origin (X0, Y0, Z0) and expressed in terms of X, Y, and Z distances.

- The **G91** command defaults the system to accept all coordinates as incremental, or relative, coordinates.

G70: Profile finishing cycle (turning)

Format: N_ G70 P_ Q_F _

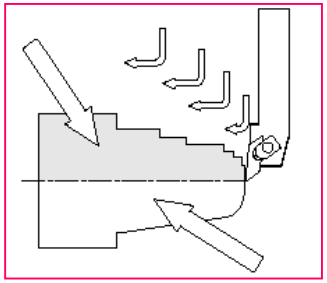
P: Start block, Q: End block, and F: Feedrate

G71: Profile roughing cycle for turning and boring

Format: N_ G71_ P_ Q_ U_ W_ D_ F _

P: Start block, Q: End block, U: Amount of stock to be left for finishing in X, W: Amount of stock to be left for finishing in Z, D: Depth of cut for each pass in thousandths, and F: Feedrate for rough pass

- The G71 command turns the part to the diameter specified in the program.
- The cutting moves are parallel to the Z axis. The upper half of the profile shows the G71 command and how it "steps" down to size. The lower half of the profile shows how the G70 command cleans up with a finishing pass.



- Example: N35 G71 P40 Q100 U.025 W.025 D625 F0.012
  - In this example, the MCU reads the data between N40 and N100 and removes material according to the defined profile. The cycle has a depth of cut of 1/16 inch for each pass at 0.012 ipr

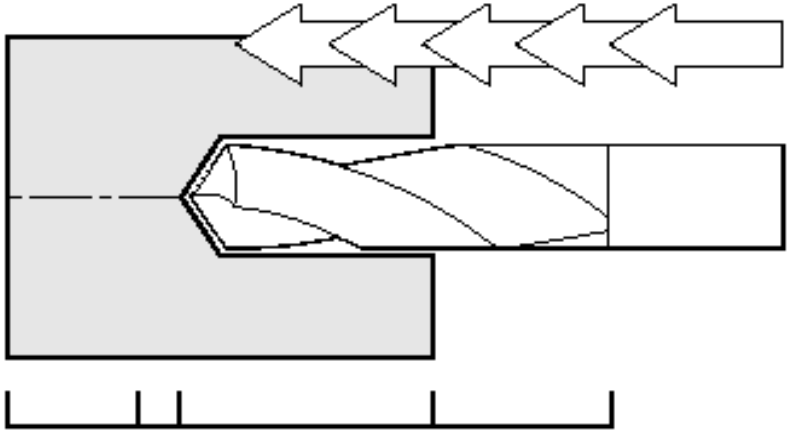
G74: Peck drilling cycle along the z axis

Format: N_ G74 X0 Z_ K_ F_

X0: X always 0, Z: Total depth, K: Peck depth, and F: Feed rate

- The G74 command executes a peck drilling cycle with automatic retracts and incremental depths of cut.

- During a peck drilling cycle, the tool feeds in to the peck distance or depth of cut, then retracts a small pre-determined distance, which is the chip-breaking process, and then feeds to the next peck, which takes the tool deeper. This process is repeated until the final Z depth is reached.



- Example: N20 G74 X0 Z-1.0 K0.125 F0.015

In this example, a hole is peck drilled to a total depth of 1 in, using 0.125 inch for the depth of each peck.

G81: Drilling cycle

Format: N_ G81 X_ Y_ Z_ R_ F_

- The G81 command invokes a drill cycle at specified locations.
- This cycle can be used for bolt holes, and drilled patterns, among other tasks.
- The G81 command is modal and so remains active until overridden by another move command or canceled by the G80 command.

G98: Return to the initial plane

Format: N_ G98

- The G98 command forces the tool to return to the Z initial plane in a drilling operation.
- This forces the tool up and out of the workpiece.

Detailed description of M codes

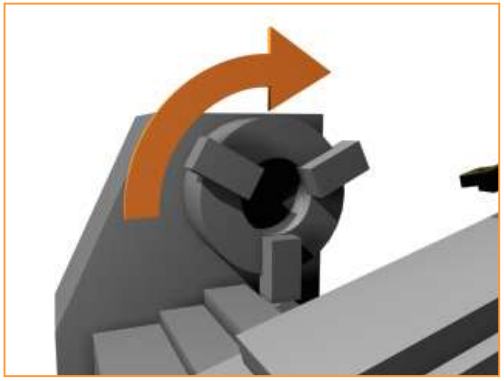
M30: Program end, reset to start Format: N_ M30

➢ The M30 is used to end the program and reset it to the start. In other words, no more program commands follow it.

M03: Spindle rotation normal – clockwise (CW) Format: N_ S_ M03
M04: Spindle rotation reverse – counterclockwise (CCW) Format: N_ S_ M04

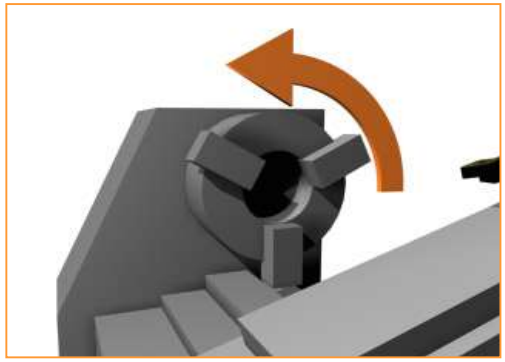
➢ The M03 command switches the spindle on in a clockwise rotation.

➢ The spindle speed is designated by the S letter address, followed by the spindle speed in revolutions per minute.



➢ The M04 command switches the spindle on in a counterclockwise rotation.

➢ The spindle speed is designated by the S letter address, followed by the spindle speed in revolutions per minute.



M05: Spindle rotation stop Format: N_ M05

- The M05 command turns the spindle off.
- Although other M codes turn off all functions (for example, M00 and M01), this command is dedicated to shutting the spindle off directly.
- The M05 command appears at the end of a program.

M06: Tool change Format: N_ T_ M06

- The M06 command halts all program operations for a tool change.
- It is actually a two-fold command.
  - First, it stops all machine operations - for example, the spindle is turned off and oriented for the tool change, and all axes motion stops - so that it is safe to change the tool.
  - Second, it actually changes the tool.

M08: Coolant on Format: N_ M08

- The M08 command switch on the coolant flow.

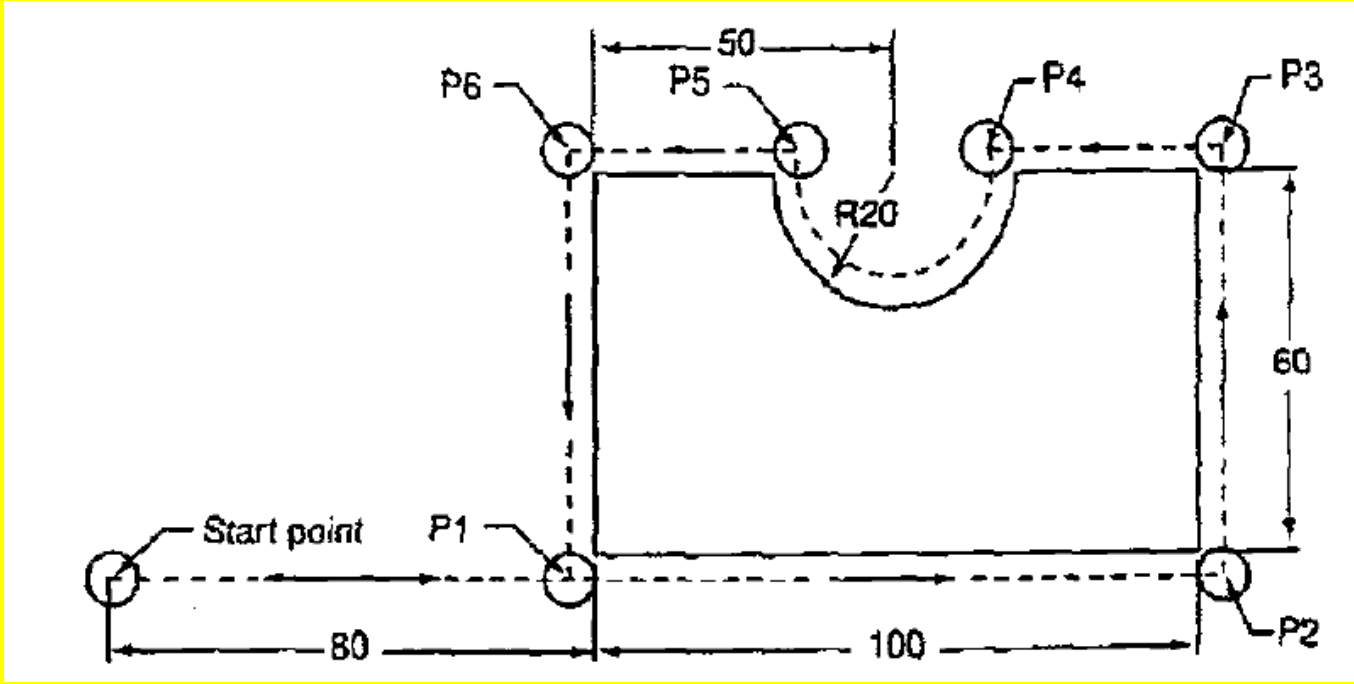
M09: Coolant off Format: N_ M09

- The M09 command shuts off the coolant flow.

- The coolant should be shut off prior to tool changes or when you are rapiding the tool over long distances.

Examples

➢ Example 1: Write a part program to mill the edge of the plate shown in the following figure. The dimensions are in millimeters. A cutter of 10 mm diameter named T01 is selected for this job. A constant machining feed rate of 350 mm/min and a rapid traverse feed rate of 950 mm/min are used. A constant spindle speed rate of 717 rpm is used. The coolant should be on.



➢ Answer:

```
%                                                    (Program start flag)
:1001                                                (Program number 1001)
N01 G91                                              (use incremental dimensions)
N02 G21                                              (select metric unit)
N03 G00 X0.0 Y0.0 Z40.0 T01 M06                      (load the tool above the start
                                                      point by40 millimeters)
N04 G01 X65.0 Y0.0 Z-55.0 F950 S717 M03              (move rapidly towards P1 at
        traverse feed rate of 950 mm/min and set spindle rotation CW at 717
        rpm. Also, bring the cutter down to its appropriate place)
N05 G01 X10.0 F350 M08                               (approach P1 at the
        machining feed rate of 350 mm/min and start the coolant)
N06 G01 X110.0                                       (move the tool from P1 to P2)
N07 G01 Y70.0                                        (move the tool from P2 to P3)
N08 G01 X-40.0                                       (move the tool from P3 to P4)
```

N09 G02 X-30.0 Y0.0 Z0.0 I-15.0 J-5.0 (move the tool from P4 to P5 clockwise along a circular arc. I and J specify the incremental $\pm x$ and $\pm y$ distance from the center of the tool at the start of the arc to the center of the arc, respectively)

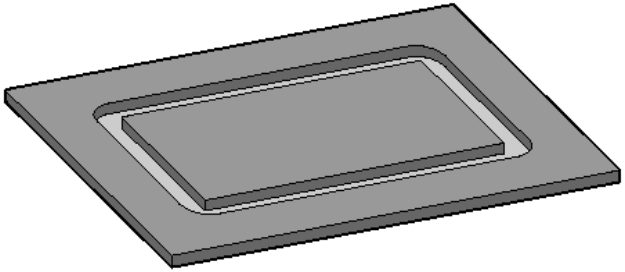
N10 G01 X-40.0 (move the tool from P5 to P6)

N11 G01 Y-70.0 (move the tool from P6 to P1)

N12 G01 X-75.0 Y0.0 Z55.0 F950 M09 (move the tool from P1 to the start point and turn off both the spindle and coolant)

N13 M30

➢ Example 2: This program introduces you to the Cartesian coordinate system and absolute coordinates. Only single-axis, linear-feed moves show the travel directions of the X, Y, and Z axes.



Workpiece Size: X5, Y4, Z1
Tool: Tool #3, 3/8" End Mill
Tool Start Position: X0, Y0, Z1 (Relative to workpiece)

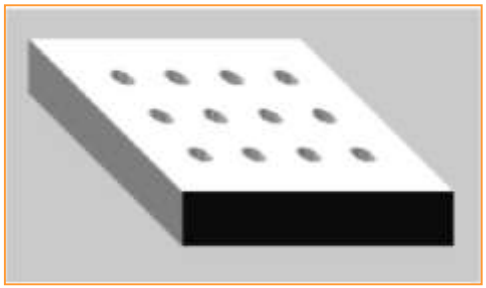
```
%
:1001
N5 G90 G20
N10 T3 M06
N15 S1200 M03
N20 G00 X1 Y1
N25 Z0.125
N30 G01 Z-0.125 F5
N35 X4 F20
N40 Y3
N45 X1
N50 Y1
N55 G00 Z1
N60 X0 Y0
N65 M05
N70 M30
```

➢ Example 3: This program introduces arcs: G02 (clockwise) and G03 (counterclockwise). These are all simple quarter quadrant arcs with a 1inch radius.

Workpiece Size: X5, Y4, Z1
Tool: Tool #2, 0.25" Slot Mill
Tool Start Position: X0, Y0, Z1 (Relative to workpiece)

```
%
:1003
N5 G90 G20
N10 T2 M06
N15 S1200 M03
N20 G00 X0.5 Y0.5
N25 Z0.25
N30 G01 Z-0.25 F5
N35 G02 X1.5 Y1.5 I1 J0 F10
N40 X2.5 Y2.5 R1
N45 X3.5 Y1.5 I0 J-1
N50 X4.5 Y0.5 R1
N55 G01 Y1.5
N60 G03 X3.5 Y2.5 R1
N65 X2.5 Y3.5 I-1 J0
N70 X1.5 Y2.5 R1
N75 X0.5 Y1.5 I0 J-1
N80 G01 Y0.5
N85 G00 Z1
N90 X0 Y0
N95 M05
N100 M30
```

➢ Example 4: This program involves a simple drilling cycle with a defined retract plane. **Once the G-code for the drill cycle has been executed, only the X and/or Y location of the remaining holes need to be defined .**



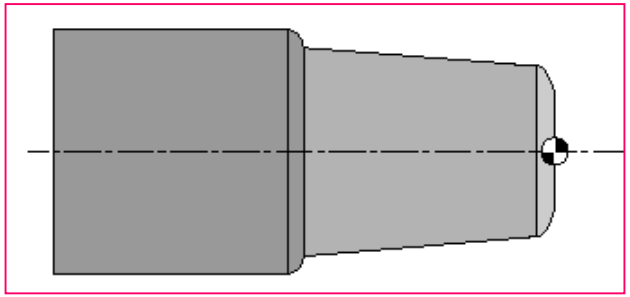
Workpiece Size: X5, Y4, Z1
Tool: Tool #7, 1/2" HSS Drill
Tool Start Position: X0, Y0, Z1 (Relative to workpiece)

```
%
:1005
N5 G90 G20
N10 T7 M06
N15 S1000 M03
N20 G00 X1 Y1
N25 Z0.25
N30 G98 G81 X1 Y1 Z-0.25 R0.25 F3
N35 Y2
N40 Y3
N45 X2
N50 Y2
N55 Y1
N60 X3
N65 X4
N70 Y2
N75 Y3
N80 X3
N85 Y2
N90 G00 Z1
N95 X0 Y0
N100 M05
N105 M30
```

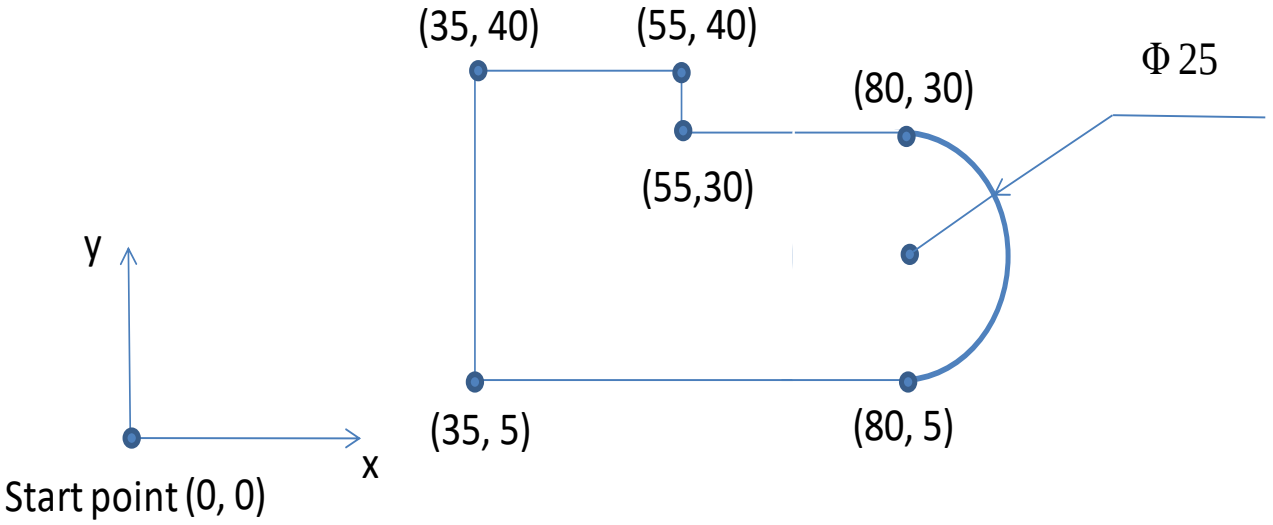
➢ Example 5: This program introduces you to the G71 turning cycle and G70 finishing cycle commands.

Workpiece Size: 4" Diameter by 2" Length
Tool: Tool #1, Right-hand Turning Tool
Tool Start Position: X2, Z3

```
%
:1003
N5 G90 G20
N10 T01
N15 M06
N20 G00 X2.1 Z0.05
N25 G71 P30 Q50 U0.025 W0.005 D625 F0.012
      N30 G01 X1 Z0
      N35 G03 X1.5 Z-0.25 I0 K-0.25
      N40 G01 X1.75 Z-2
      N45 G03 X2 Z-2.125 I0 K-0.125
      N50 G01 X2.2
N55 G70 P30 Q50 F0.006
N60 G00 X2 Z3
N65 M05
N70 M30
```



➢ **Exercise 3:** Write a part program manually to mill the edge of the plate shown below. The dimensions are in millimeters. Assume that the raw material, a rectangular plate, has a thickness of 15 mm and that its bottom face has the z coordinate of 0. Assume that the tool, named T03, diameter is 10 mm, the machining feed rate is 250 mm/min, the rapid traverse feed rate is 850 mm/min, and the spindle speed rate is 700 rpm. The coolant should be on.



➢ **Exercise 4:** Write a part program manually to machine the part shown below. The dimensions are all in inches and the tool start position is at the point (X2, Z3). Use the following machining parameters and tools:

1. Roughing – spindle speed of 500 rpm and feedrate of 0.012 ipr, T02
2. Finishing – spindle speed of 700 rpm and feedrate of 0.006 ipr, T03
3. Drilling – drill speed of 500 rpm and a feedrate of 0.004 ipr, T04

The depth of cut for each roughing pass is 0.0625 inch and the amount of stock to be left for finishing on the X and Z axes are 0.05 and 0.004 inches, respectively. The hole is peck drilled to a total depth of 1 inch using 0.125 inch for the depth of each peck.