


## Computer Numerical Control (Part-6)

*6ME4-02: Computer Integrated Manufacturing Systems*

**Tarun Kr. Aseri**  
Asst. Prof.  
Mechanical Engineering  
Engineering College, Ajmer  
Barliya Circle, NH-8,  
Ajmer-305025, India  
Email: tarunaseri[at]ecajmer.ac.in



1

1

## Outcomes

- Advanced preparatory codes
- Mirror (G11-G12-G13)
- Shifting origin (G92)
- Scaling (G72)
- Pattern Rotation (G73)
- Some Canned Cycles

*Tarun K. Aseri*
*Mechanical Engineering Department*

2

## Mirroring

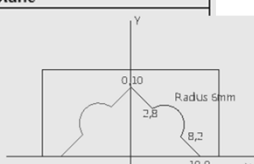
The mirroring command is used when features of components shares symmetry about one or more axes and are also dimensionally identical. By using this code components can be machined using a single set of data and length of programs can be reduced.

- ▶ G10 cancellation of mirroring image
- ▶ G11 Mirror image on X axis
- ▶ G12 Mirror image on Y axis
- ▶ G13 Mirror image on Z axis

*Tarun K. Aseri*
*Mechanical Engineering Department*

3

## Mirroring: Example

00002	Program number
N02 G21	Metric programming
N03 M03 S1000	Spindle start clockwise with 1000rpm
N04 G00 X0.0 Y0.0	Rapid motion towards (0,0)
N05 G00 Z10.0	Rapid motion towards Z=-10 plane
N06 G00 X10.0	
N07 G00 Z0.0	
N08 G01 X8.0 Y2.0	
N09 G03 X2.0 Y8.0 R6.0	
N10 G01 X0.0 Y10.0	
N11 G00 Z10.0	
N12 G00 X0.0 Y0.0	
N13 G11	Mirror image on X axis
N14 G25 N06.12	Repeat lines 6 to 12 (machining of negative X side)
N15 G10	Cancellation of mirror image
N16 M05 M09	Spindle stop and program end

4

### Shifting Origin

G92 code is used to temporarily shift the origin to the reference point specified.

Syntax:  
N?? G92 Xx Yy

*Tarun K. Aseri* Mechanical Engineering Department

5

### Shifting Origin: Example

00003	Program number
N02 G21	Metric programming
N03 M03 S1000	Spindle start clockwise with 1000rpm
N04 G00 X0 Y0	Rapid motion towards (0,0)
N05 G00 Z0.0	Rapid motion towards Z=0 plane
N06 G92 X 10.0 Y 10.0	Point A become new origin
N07 G00 X0.0 Y0.0	Tool movement to point A
N08 G01 X10.0	Machining path A
N09 G01 Y10.0	
N10 G01 X0.0	
N11 G01 Y0.0	
N12 G00 Z10.0	
N13 G92 X 20.0 Y 20.0	Point B become new origin
N14 G00 X0.0 Y0.0	
N15 G00 Z0.0	
N16 G25 N08.11	Repeat lines 8 to 11 (machining of path B)
N17 G00 Z10.0	
N18 M05 M09	Spindle stop and program end

*Department*

6

### Scaling

Scaling function is used to program geometrically similar components with varying sizes.

Syntax: G72 K, where k is the scaling factor.

The scaling command can be cancelled by using the statement G72 K1.0.

*Tarun K. Aseri* Mechanical Engineering Department

7

### Scaling: Example

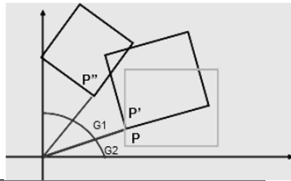
N02 G21	Program number
N03 M03 S1000	Metric programming
N04 G00 X0 Y0	Spindle start clockwise with 1000rpm
N05 G00 Y25.0	Rapid motion towards (0,0)
N06 G00 Z-10.0	
N07 G01 X-25.0	Machining path A
N08 G01 Y-25.0	
N09 G01 X25.0	
N10 G01 Y25.0	
N11 G01 X0.0	
N12 G00 Z10.0	
N13 G00 X0.0 Y0.0	
N14 G72 K2.0	Scaling using factor 2.0
N15 G25 N05.11	Repeat lines 5 to 11 (machining of path B)
N16 G00 Z10.0	
N17 G00 X0.0 Y0.0	
N18 G72 K3.0	Scaling using factor 3.0
N19 G25 N05.11	Repeat lines 5 to 11 (machining of path B)
N20 G00 Z10.0	
N21 G00 X0.0 Y0.0	
N22 M05 M09	Spindle stop and program end

*Mechanical Engineering Department*

8

### Pattern Rotation

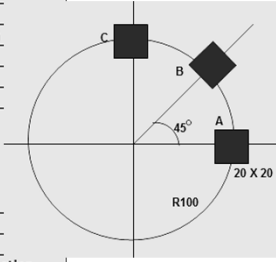
- Pattern rotation is used to obtain a pattern of similar features. G73 code is used to rotate the feature to form a pattern.
- Syntax G73 Aa, where 'a' is the angle of rotation. This command is cumulative, and the angle gets added up on time the program is executed. So all the rotational angle parameters should be cancelled using the code G73.
- The unconditional jump code G25 is used in conjunction with this code to achieve the desired rotation.



Tarun K. Aseri Mechanical Engineering Department

9

### Pattern Rotation: Example

00001	Program number	
N02 G21	Metric programming	
N03 M03 S1000	Spindle start clockwise with 1000rpm	
N04 G00 X0 Y0	Rapid motion towards (0,0)	
N05 G00 X90.0		
N06 G00 Z-10.0		
N07 G01 Y10.0		
N08 G01 X110.0		
N09 G01 Y-10.0	Machining path A	
N10 G01 X90.0		
N11 G01 Y0.0		
N12 G00 Z10.0		
N13 G00 X0.0 Y0.0		
N14 G73 A45.0	Rotation of axis 45 degree in <i>ccw</i> direction	<p style="text-align: center;">Engineering Department</p>
N15 G25 N05.13	Repeat lines 5 to 13 (machining of path B)	
N16 G73 A45.0	Rotation of axis 45 degree in <i>ccw</i> direction	
N17 G25 N05.13	Repeat lines 5 to 13 (machining of path B)	
N18 G73	All the rotation cancelled	
N19 M05 M09	Spindle stop and program end	

10

### Canned Cycles

A canned cycle is a preprogrammed sequence of events / motions of tool / spindle stored in memory of controller. Every canned cycle has a format. Canned cycle is modal in nature and remains activated until cancelled. Canned cycles are a great resource to make manual programming easier. Often underutilized, canned cycles save time and effort.

Tarun K. Aseri Mechanical Engineering Department

11


### Machining a Rectangular pocket

- This cycle assumes the cutter is initially placed over the center of the pocket and at some clearance distance (typically 0.100 inch) above the top of the pocket. Then the cycle will take over from that point, plunging the cutter down to the "peck depth" and feeding the cutter around the pocket in ever increasing increments until the final size is attained. The process is repeated until the desired total depth is attained.
- The overall length and width of the pocket, rather than the distance of cutter motion, are programmed into this cycle.

Tarun K. Aseri Mechanical Engineering Department

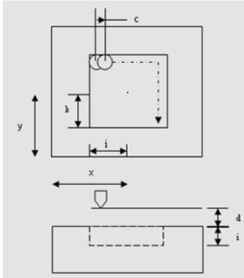
12

### Machining a Rectangular pocket



- The syntax is : G87 Xx Yy Zz Ii Jj Kk Bb Cc Dd Hh Ll Ss (This G code is entirely controller specific and the syntax may vary between controller to controller).


x,y - Center of the part  
 z - Distance of the reference plane from top of part  
 i - Pocket depth  
 j,k - Half dimensions of the target geometry (pocket)  
 b - Step depth  
 c - Step over  
 d - Distance of the reference plane from top of part  
 h - Feed for finish pass  
 l - Finishing allowance  
 s - Speed



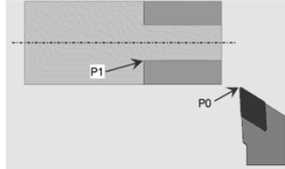
Tarun K. Aseri
Mechanical Engineering Department

13

### Turning Cycles: Lathe




- The G80 command will make the tool move in a series of rectangular paths cutting material axially until the tool tip reaches target point P1 where the cycle ends as shown in figure 31.5. Cutting movements will be at the cutting feed rate. All other movements will be at rapid traverse rate.
- The syntax is G80 Xx Zz Ff



Tarun K. Aseri
Mechanical Engineering Department

14

### Roughing Cycle: Lathe



- In roughing cycle, the final finishing cycle profile is used to perform the roughing operation for the higher material removal rate.
- The syntax is G81 Pp Qq Uu Ww Dd Ff Ss


Where, p - start of cycle block no  
 q - Finish of cycle

$\left. \begin{matrix} u \\ w \end{matrix} \right\}$  Finishing allowance in x and y directions  
 d - Depth of cut

Tarun K. Aseri
Mechanical Engineering Department

15

### Summary



- Advanced preparatory codes
  - Mirror (G11-G12-G13) and its cancellation
  - Shifting origin (G92)
  - Scaling (G72)
  - Pattern Rotation (G73)
- Canned Cycles

Tarun K. Aseri
Mechanical Engineering Department

16



17