Slide 1

MATHEMATICS FOR CNC PROGRAMMING

3rd Year
CAD/CAM
ENGINEERING DIVISION

2018

Dr. Tahseen F. Abbas

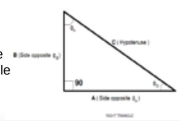
Slide 2

Mathematics for CNC Programming

Many problems in CNC programming involves finding the X and Y coordinates of tool motion based on given length and angle. These tasks can be usually be accomplished by using right-triangle trigonometry.

Determining Sides of Right Triangles:

A right triangle is a three sided figure, one angle is 90 , the longest side of the triangle is called the hypotenuse.



Slide 3

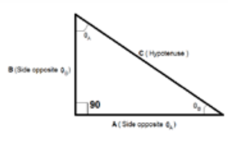
Mathematics for CNC Programming

The relationship between any two sides and the included angle are given by the following general side angle formulas:

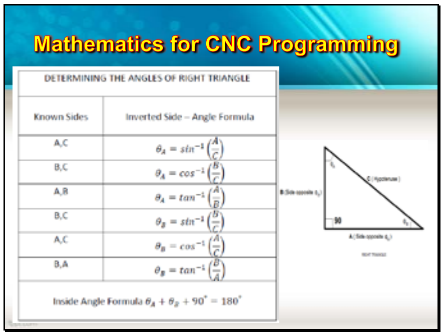
$$\sin(\theta) = \frac{Side\ opposite\ (\theta)}{Hypotenuse}$$

$$\cos(\theta) = \frac{Side\ adjacent\ (\theta)}{Hypotenuse}$$

$$\tan(\theta) = \frac{Side\ opposite\ (\theta)}{Side\ adjacent\ (\theta)}$$

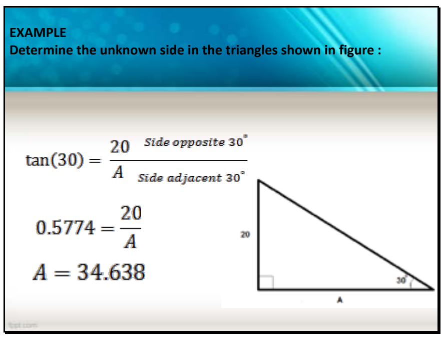


Slide 4

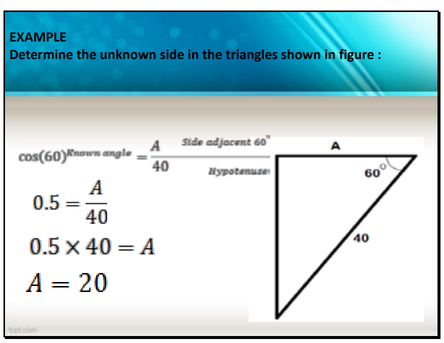


DETERMINING THE ANGLES OF RIGHT TRIANGLE	
Known Sides	Inverted Side – Angle Formula
A,C	$\theta_A = sin^{-1}\left(\frac{A}{C}\right)$
B,C	$\theta_A = cos^{-1}\left(\frac{B}{C}\right)$
A,B	$\theta_A = tan^{-1}\left(\frac{A}{B}\right)$
B,C	$\theta_B = sin^{-1}\left(\frac{B}{C}\right)$
A,C	$\theta_B = cos^{-1}\left(\frac{A}{C}\right)$
B,A	$\theta_B = tan^{-1}\left(\frac{B}{A}\right)$
Inside Angle Formula $\theta_A + \theta_B + 90^{\circ} = 180^{\circ}$	

Slide 5



Slide 6



Slide 7

EXAMPLE

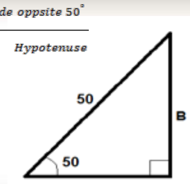
Determine the unknown side in the triangles shown in figure :

$$\sin(50)^{Known\ angle} = \frac{B}{50} \frac{Side\ oppsite\ 50^{\circ}}{Hypotenuse}$$

$$0.766 = \frac{B}{50}$$

$$0.766 \times 50 = B$$

$$B = 38.3$$



Slide 8

Determine the unknown side in the triangle shown in Figure

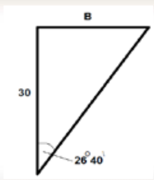
Convert minutes to fractions of degree:

$$26^{\circ}\ 40' \times \frac{1^{\circ}}{60'} = 26.6667^{\circ}$$

$$tan(26.6667) = \frac{B}{30} \frac{Side\ opposite\ 26.6667}{Side\ adjacent\ 26.6667}$$

$$0.5022 = \frac{B}{30}$$

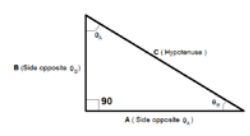
$$B = 15.066$$



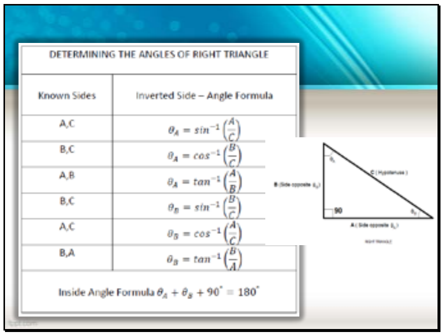
Slide 9

Determining Angles of Right Triangles:

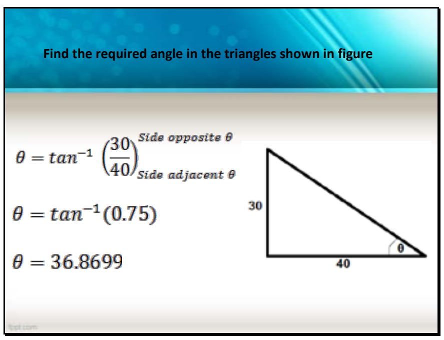
The side angle formulas given before can be inverted to determine the included angle when any two sides of the right triangle are known.



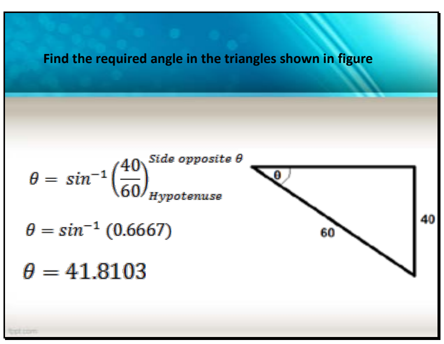
Slide 10



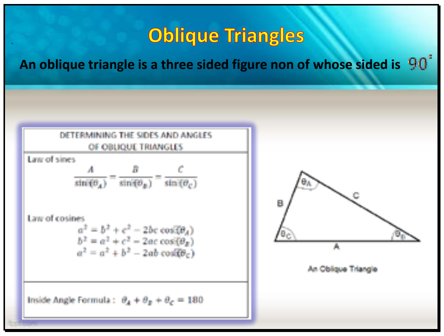
Slide 11



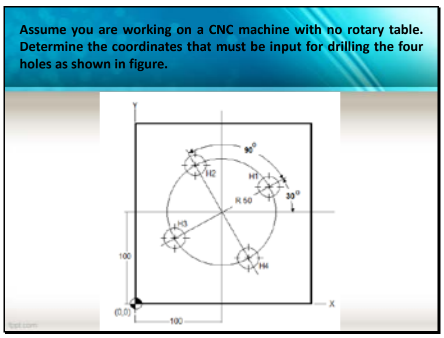
Slide 12



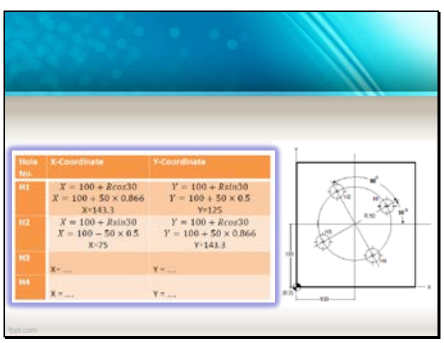
Slide 13



Slide 14

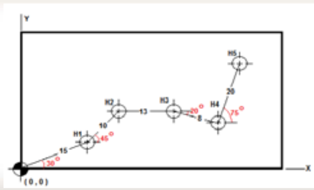


Slide 15



Slide 16

Assume you are working on a CNC machine with no rotary table. Determine the coordinates that must be input for drilling the five holes as shown in figure.



Slide 17

Hole No.	Hole Coordinates	
H1	$X_1 = 15\cos 30$ $Y_1 = 15\sin 30$	X1= 12.99 Y1 = 7.5
H2	$X_2 = X_1 + 10\cos 45$ $Y_2 = Y_1 + 10\sin 45$	X2 =20.061 Y2 = 14.57
H3	$X_3 = X_2 + 13$ $Y_3 = Y_2$	X3 = 33.061 Y3 = 14.57
H4	$X_4 = X_3 + 8\cos 20$ $Y_4 = Y_3 - 8\sin 20$	X4 = 40.518 Y4 = 11.834
H5	$X_5 = X_4 + 20\cos 75$ $Y_5 = Y_4 + 20 sin 75$	X5 = 45.69 Y5 = 31.153

Slide 1

FANDAMNTAL OF CUTTING TOOLS IN CNC MILLING PROCESS

Dr. Tahseen F. Abbas

Slide 2

CUTTING TOOLS

Anatomy of a Milling Tool

Milling is done using a cylindrical milling tool mounted in a milling tool holder that is then mounted in the tool spindle on the machine.

Slide 3

TYPES OF CUTTING TOOLS

A wide range of tool types and configurations are available for CNC milling machines.

The most commonly used tools are:

- End mills (Flat, Ball, Chamfer), Face mill
- Corner Rounding tools
- Slot Tools
- Center Drill
- Twist Drill
- Tap
- Reamer
- Counterbore

Slide 4

End Mills

End mills can be center cutting and non-center cutting. As their name implies, center cutting end mills have cutting edges on both the end face of the cutter and the sides. Center cutting end mills are essential for plunge milling.

Slide 5

End Mills

Non-center cutting end mills have cutting edges only on the sides and are used only for side milling. These tools are identified by a small hole at the center.

Slide 6

End Mills

Ball end mills produce a radius at the bottom of pockets and slots. Ball end mills are used for contour milling, shallow slotting, contour milling and pocketing applications

BALL END MILL

Slide 7

Flutes

•Two Flute: Has the greatest amount of flute space, allowing for more chip carrying capacity in softer materials. Used primarily in slotting and pocketing of non-ferrous materials like aluminum .

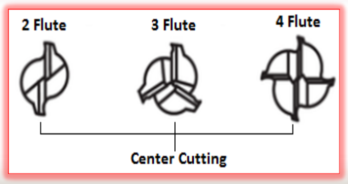
•Three Flute: Allows for better part finish in harder materials. The three flutes provide for greater strength and the ability to pocket and slot both ferrous and non-ferrous materials.

•Four Flute/Multiple Flute: Ideal for finish milling. The extra flutes allow for faster feed rates to produce a much finer finish than two or three flute tools. However, the reduced flute space may cause problems with chip removal.

Slide 8

Flutes

The most common flute numbers for general milling operations are two (better space for chip ejection) and four (better surface finish).



Slide 9

End Mill Materials

End mills are made out of either cobalt steel alloys (known as high speed steel, or HSS), or from tungsten carbide in a cobalt lattice (shortened to "carbide").

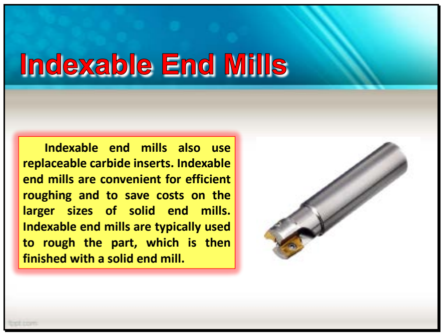
•High Speed Steel (HSS): **HSS is used for general purpose milling of both ferrous and non-ferrous materials.**

•Cobalt: **Cobalt is an M42 tool steel with an 8% cobalt content. Cobalt is more expensive but provides better wear resistance and toughness than HSS. Metal removal rates and finish are better than HSS.**

•Solid Carbide: **Carbide is considerably harder, more rigid, and more wear resistant than HSS. Carbide is used primarily in finishing applications.**

Slide 10

Slide 11



Slide 12

Slide 13

Drills

Spot Drills

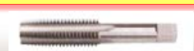
Spot drills are special purpose drills with very short flutes, they can precisely spot a hole. The goal is use the spot drill to make a little dimple in the workpiece that keeps the twist drill from walking so that the hole winds up in the right place.

Slide 14

Taps and Reamers

Reamers : are used to enlarge an existing hole to a precise tolerance and to add a high quality surface finish. Reamers require a hole be drilled first that is fairly close to the final size so that the reamer actually removes relatively little material. Reamers ensure a hole has an accurate diameter, roundness, and good surface finish

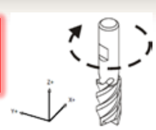
Taps are used to cut internal threads of a specific size and pitch. Like reamers, a tap requires a hole be drilled first to the size of the minor diameter.



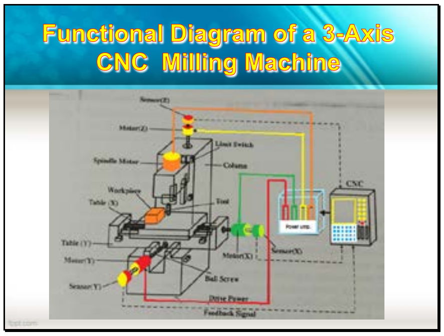
Slide 15

Tool Rotation Direction

All tools (except left---handed taps) rotate clockwise (M03) when viewed from the machine spindle looking down at the part.



Slide 16


Functional Diagram of a 3-Axis
CNC Milling Machine
Sensor(Z)
Motor(Z)
Limit Switch
Spindle Motor
Column
Workpiece
Table (X)
Tool
CNC
Table (Y)
Motor(Y)
Motor(X)
Sensor(X)
Sensor(Y)
Ball Screw
Drive Power
Feedback Signal

Slide 1

Basic Motion Commands

Dr. Tahseen F. Abbas
3rd Year CAD/CAM Eng.
Lec.5
10 Slides

Slide 2

G00 – Rapid traverse

When the tool is moving to a position preparatory to executing a cutting motion or when it is moving to the tool change position, the motion is a essentially a waste of time and is executed as fast as possible. The motion is called Rapid traverse.

Format

G00 X... Y... Z...

X, Y, Z = coordinates of destination point

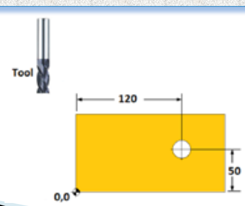
The block consists of the rapid traverse command G00 followed by the destination coordinates.

Slide 3

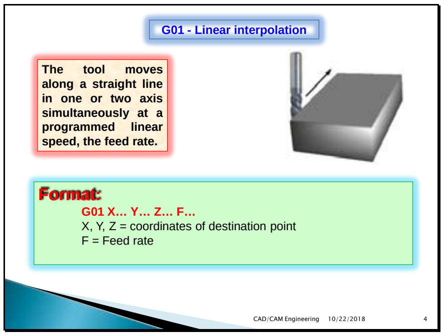
Ex.

G00 X120.0 Y50.0 Z10.0

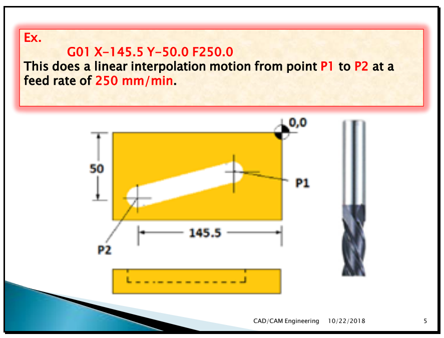
This moves the tool at rapid from its current position to the center of the hole.



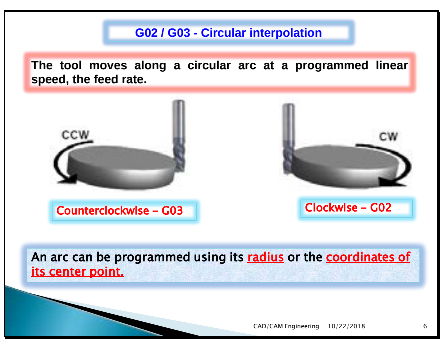
Slide 4



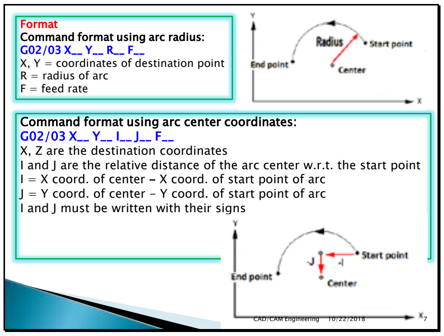
Slide 5



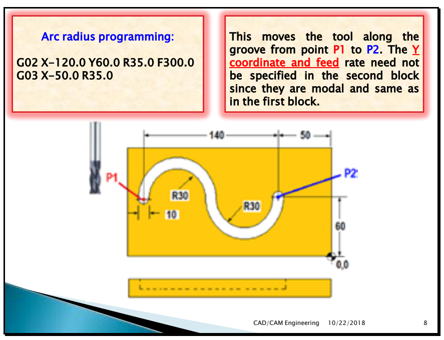
Slide 6



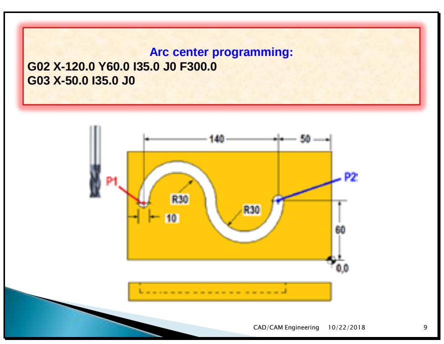
Slide 7



Slide 8



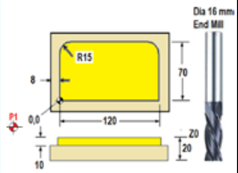
Slide 9



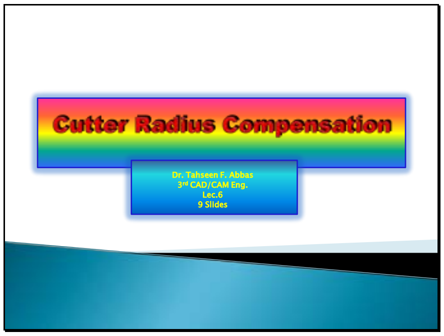
Slide 10

The tool starts at point P1 (X-50.0,Y-15.0) and goes around the part counterclockwise. It then goes back to P1. Offset number 23 has the tool dia. value of 16 mm.

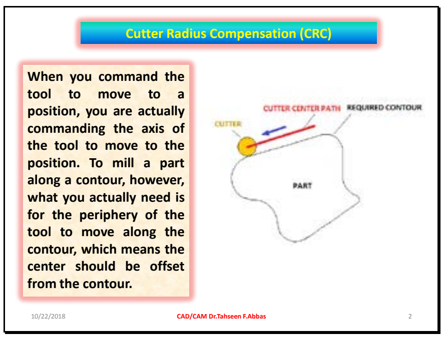
```
-----
-----
G01 G42 D23 X0 Y0 F380.0
X120.0
Y55.0
G03 X105.0 Y70.0 R15.0
G01 X 15.0
G03 X0 Y55.0 R15.0
G01 Y-10.0
G40 X-50.0 Y-15.0
-----
-----
```



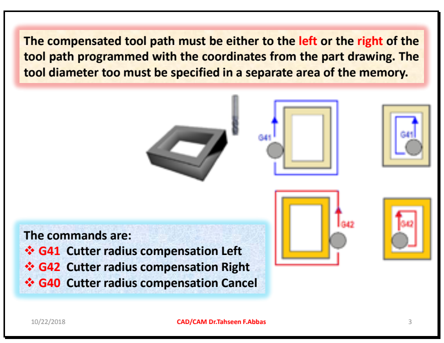
Slide 1



Slide 2

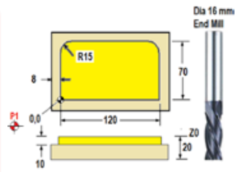


Slide 3



Slide 4

G01 G42 D21 X0 Y0 F380.0
X120.0
Y55.0
G03 X105.0 Y70.0 R15.0
G01 X 15.0
G03 X0 Y55.0 R15.0
G01 Y-10.0
G40 X-50.0 Y-15.0



The tool starts at point P1 (X-50.0,Y-15.0) and goes around the part counterclockwise. It then goes back to P1. Offset number 21 has the tool dia. value of 16.

Slide 5

Tool Length Compensation

Tools used in machining a part are of different lengths. It would be extremely tedious to write the program with these lengths taken into consideration.

Format
G00 / G01 G43 Hnn

G43 is the length compensation command.

H is the tool offset number.

The tool length compensation must be activated with the first motion after every tool change.

Example
G00 G43 H14

Slide 6

Program Structure

Start
The first line is the % character.

The second line is the program number, written as Onnnn.
E.g., O1960 means program number 1960.

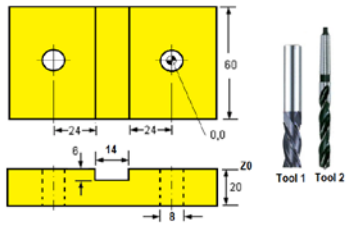
End
The last but one line is the program end command (M02 or M30).

Block numbers
Block numbers add clarity to the program. They are written as N

Slide 7

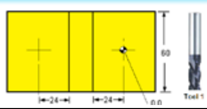
Sample program

This sample program is a simple full program that does a drilling operation followed by a grooving operation.



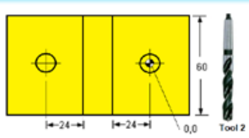
Slide 8

Program	Block Explanation
%	Program start character
O1961	Program number 1961
G00 G91 G28 Y0 Z0	Move to position away from part for tool change
T01 M06	Tool change to tool number 1 (14 dia. End mill)
S500 M03	Spindle speed 500 RPM, CW
G00 X-31.0 Y-40.0 M08	Move at rapid to position for milling, coolant ON
G43 H1 Z-3.0	Rapid to depth for first cut
G90G01 Y40.0 F350.0	Cut 1
G00 Z-6.0	Rapid to depth for second cut
G01 Y-40.0	Cut 2
G00 Z2.0 M05	Rapid above part and spindle OFF
M09	Coolant OFF
G00 G91 G28 Y0 Z0	Rapid to tool change position and spindle OFF



Slide 9

T02 M06	Tool change to tool number 2 (8mmDrill)
S1400 M03	Spindle speed 1400 RPM, CW
G00 X0 Y0 M08	Rapid to 1st hole position, coolant ON
G43 H2 Z3.0	Rapid above part
G01 Z-23.0 F200.0	Feed into 1st hole
G00 3.0	Rapid out of hole
G00 X-62.0 Y0	Rapid to 2nd hole position
G01 Z-23.0 F200.0	Feed into 2nd hole
G00 Z3.0	Rapid out of hole
M05	Spindle OFF
G00 G91 G28 Y0 Z0 M09	Rapid to tool change position and coolant OFF
M02	Program end



Slide 1

Circular Interpolation Using "I" and "J"

Slide 2

G02 Circular Interpolation Using "I" and "J"

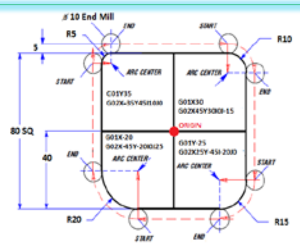
The "I"and "J" values are INCREMENTAL distances from where the tool starts cutting the arc (START POINT) to the ARC CENTER.

only one of the I, J is specified, the others are assumed to be zero.

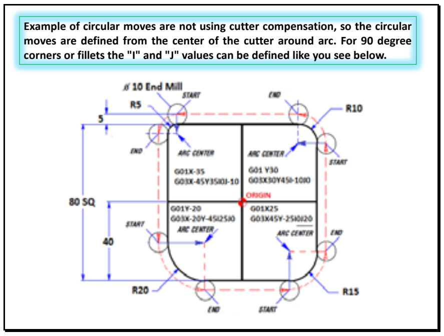
Use of I,or J is the only way to cut a complete 360 degree arc; in this case, the starting point is the same as the ending point and no X,or Y is needed. To cut a complete circle of 360 degrees ,you do not need to specify an ending point X, or Y just program I, or J to define the center of the circle.

Slide 3

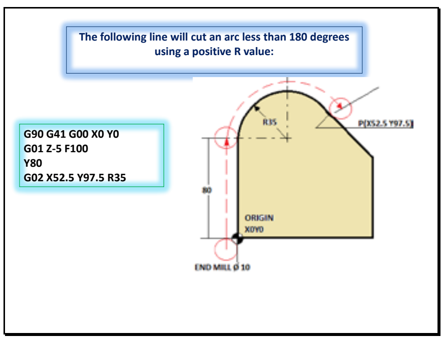
NOTE: Example of circular moves are not using cutter compensation, so the circular moves are defined from the center of the cutter around arc. For 90 degree corners or fillets the "I" and "J" values can be defined like you see below



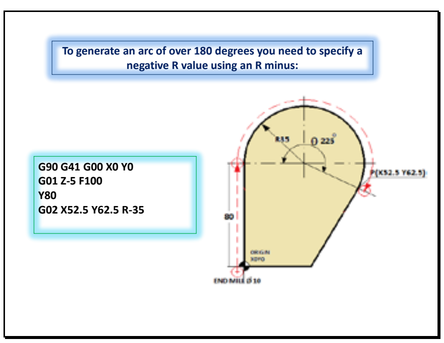
Slide 4



Slide 5



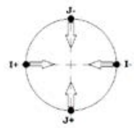
Slide 6



Slide 7

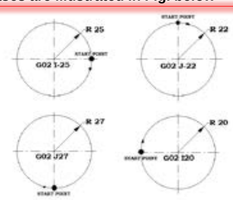
Fill Circular Movement

By selecting a pole point of a circle and using an incremental line of program to create a full circle all values on this line program will have a zero value except for the I or J axis on the appropriate pole axis which will represent the radius to be produced



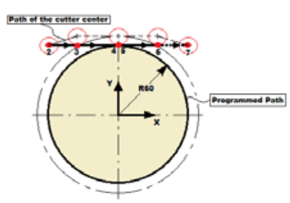
Slide 8

Since the I and J are already incremental the G91 is active on the X and Y values only. If starting
From a pole axis the only axis that need programming is the pole axis that represent the radius
Using clockwise G02 the line of a circle program for different cases are illustrated in Fig. below



Slide 9

EXAMPLE: Use cutter diameter compensation to machine the outside of the circle in figure below.

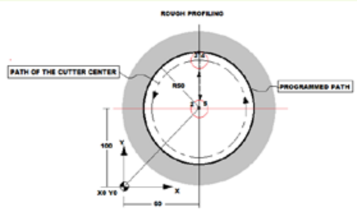


Slide 10

Command	Meaning
G90 G0 X-60 Y60 S400	Rapid to position 2
G01 Z-5 M03 F200	Plunge to 5 . spindle on CW
G41 X-25 D20	Ramp on the left of upward tool motion on next move to 3 . offset tool by radius value in address D20
G01 X0	Cut to 4
G02 X0 Y60 J-60	Cut arc to position 5
G01 X25	Cut to position 6
G00 G40 X60	Ramp off to the next move to 7

Slide 11

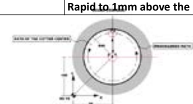
The inside circle shown in figure is to be machined via cutter diameter compensation. Use this teqnique for executing operation 1 (rough profiling) and operation 2 (finish profiling)



Slide 12

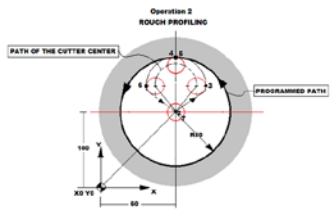
Program for operation 1- Rough Profiling-

Command	Meaning
G90 G00 X60 Y100 S700	Rapid to position(2)
G01 Z-5 M03 F250	Plunge to 5mm. Spindle on (CW)
G41 Y150 F200 D20	Ramp on tom the left of upward tool motion on next move to Y150 (3). Offset tool by radius value in address D20
G03 X60 Y150 I0 J-50	Cut arc to (4)
G00 G40 Y100	Ramp off to the next move to Y100 (5)
Z1	Rapid to 1mm above the part



Slide 13

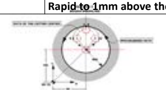
FINISHING



Slide 14

Program for operation 2- Finishing-

COMMAND	MEANING
G90 G00 X60 Y100 S500	Rapid to position(2)
G01 Z-5 M03 F200	Plunge to 5mm. Spindle on (CW)
G41 X90 Y120 D30	Ramp to the left of upward tool motion on next move to Y120 (3). Offset tool by radius value in address D30
G03 X60 Y150 R30	Cut R30 arc to (4)
I0 J-50	Cut R50 arc to (5)
X40 Y120 R30	Cut R30 arc to (6)
G00 G40 X60 Y100	Ramp off to the next move to X60 Y100 (7)
Z1	Rapid to 1mm above the part



Slide 1

MILLING CENTER
Tangential Entries and Exits

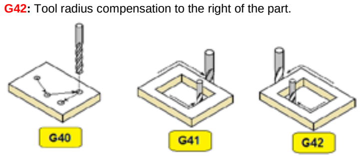
Assist. Prof. Dr.Tahseen F. Alani
CAD/CAM Engineering
2014-2015
17 Slides

Slide 2

Tool radius compensation

Tool compensation may be applied in two different ways depending on the programming direction.

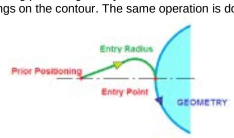
- **G40:** Cancellation of tool radius compensation
- **G41:** Tool radius compensation to the left of the part.
- **G42:** Tool radius compensation to the right of the part.



Slide 3

Tangential Entries and Exits.

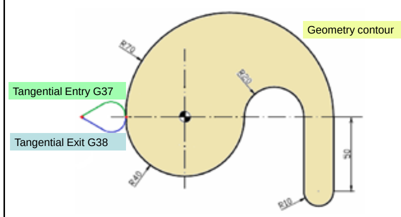
The purpose of these functions is for the tool not to enter the contour in a straight line, but describing a particular radius in order to approach the starting point tangentially. This is done to avoid possible markings on the contour. The same operation is done to exit.



- A tangential entry consists of a linear interpolation with tool radius compensation and a circular interpolation to enter the contour.
- The distance between the previous point and the entry point must never be smaller than twice the diameter of the tool.
- The entry radius must never be smaller than the tool diameter.

Slide 4

Programming Example



Slide 5

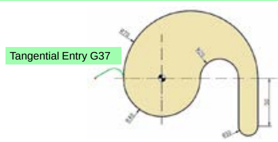
The structure of a program for contouring any geometry is always the following:

Header

```
G0 Z100            ; Safety position.
T10 D10            ; Call to the Ø 10 mm tool.
S2000 M03          ; Start the spindle clockwise (M03).
```

Enter the contour

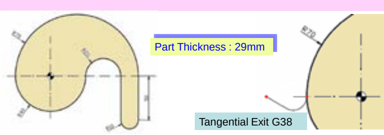
```
X-70 Y0                          ; Position before the entry.
G43 Z0                           ; Z down movement to the surface.
N1 G01 G91 Z-5 F100              ; Penetration step.
G90 G37 R10 G42 X-40 Y0 F100     ; Tangential entry and tool radius compensation.
```



Slide 6

Geometry

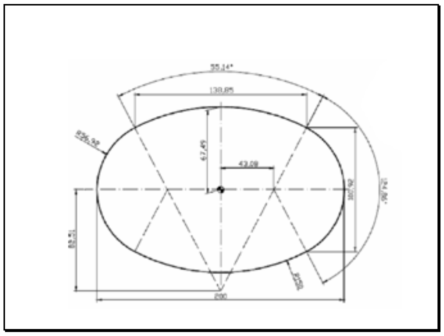
```
G03 X40 Y0 R40
G02 X80 Y0 R20
G01 X80 Y-50
G03 X100 Y-50 R10
G01 X100 Y0
G03 G38 R10 X-40 Y0 R70          ; Tangential exit.
N2 G1 G40 X-70 Y0                ; Return to starting point without compensation
```



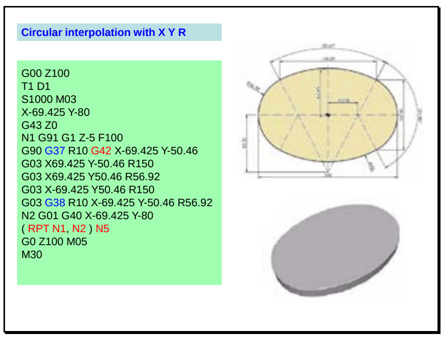
Repeat down movements.

```
(RPT N1,N2)N5    ; Repeat down movements five times
G0 Z100          ; Starting plane.
M30              ; End of program.
```

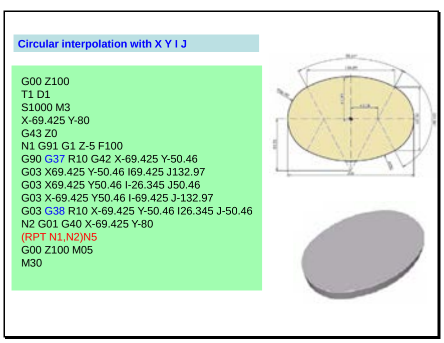
Slide 7



Slide 8



Slide 9



Slide 10

MILLING CENTER
FUNCTIONAL PROGRAMMING

Lecture 6

Assist. Prof. Dr.Tahseen F. Alani
CAD/CAM Engineering
2014-2015
16 Slides

Slide 11

Mirror image (G10/G11/G12/G13)

The mirror image cycle is usually generated to repeat the whole program in different areas of the part with respect to the symmetry axes.

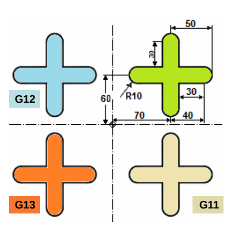
- G10: Mirror image cycle cancellation.
- G11: Mirror on X axis.
- G12: Mirror on Y axis.
- G13: Mirror on origin.

When working with "Mirror image" or "Coordinate rotation", the movement after these functions must be programmed in absolute coordinates (G90).

Slide 12

```
T10 D10
M06
G43 G0 Z100
X0 Y0 S1000 M3
N1 X30 Y30
Z0
N2 G01 G91 Z-5 F500 S1000
G90 G37 R10 G41 X60 Y30
G01 Y50
X30
G02 X30 Y70 R10
G01 X60
Y100
G02 X80 Y100 R10
G01 Y70
X110
G02 X110 Y50 R10
G01 X80
Y20
G02 X60 Y20 R10
G01 G38 R10 X60 Y30
N3 G01 G40 X30 Y30
(RPT N2,N3)N3
G00 Z100
N4 X0 Y0
G11
(RPT N1,N4)
G10G12
(RPT N2,N3)
G10G13
(RPT N2,N3)
G10
M30
```

Mirror image

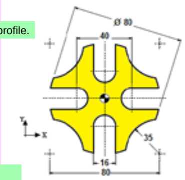


Part Thickness : 20 mm

Slide 13

Mirror image

```
N10 G00 X50 Y0 Z10 S800 T0202 M03
N20 G42 X39.192 Y8
N30 G01 Z-5 F200
N40 G90 …          ; Beginning of the profile.
N50 G02 …
N60 G03..
N70 G01…
N80 G02…
N90 G01…
N100 G03 …
N110 G02 …
120 G03 …
N130 G01…
N140 G02…
N150 G01…          ; End of the profile.
N160 G11: G12:  G13
N170 (RPT N40, N150)
180 G10
N190 G01 Z10
N200 G00 G40 X50 Y0 Z30 M05
N210 M30
```



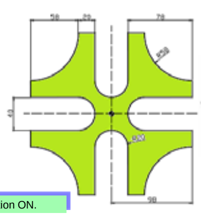
Slide 14

Coordinate rotation G73.. Q..

The program for this part uses coordinate (pattern) rotation taking advantage of the fact that the part is symmetrical in all its quadrants. We program only a forth of the contour and then rotate the coordinate axes 90º taking the center point (part zero) as the rotation point. Since this function acts after it is defined, after programming the rotation, we repeat the programmed quadrant using the RPT instruction as often as the number of quadrants remaining. The rotation function is cancelled with another **G73** but without a rotating angle.

Slide 15

```
T10 D10
M06
G43 G0 Z100
X125 Y0 S1500 M03
Z0
N3 G01 G91 Z-5 F500 S1000
G01 G90 G42 X98 Y20 F1000
N1 X98 Y40
G02 X40 Y98 R58
G01 X20 Y98
X20 Y40
G02 X-20 Y40 R20
G01 Y98
N2 G73 Q90          ; Rotation ON.
(RPT N1,N2)N3
G73                 ; Rotation OFF.
N4 G01 G40 X125 Y0
(RPT N3,N4)N5
G0 Z100 M05
M30
```



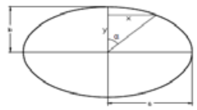
Part Thickness : 30mm

Slide 16

PARAMETRIC PROGRAMMING: Ellipse

The ellipse program has many variations depending on the position of the ellipse and on whether it is a full ellipse or it is going to end at a particular angle.

To calculate the XY positions, it uses the values of the semi minor axis and semi major axis with the sine and cosine formulae.



Formula of the ellipse

$X = a \cdot \sin \alpha$

$Y = b \cdot \cos \alpha$

Slide 17

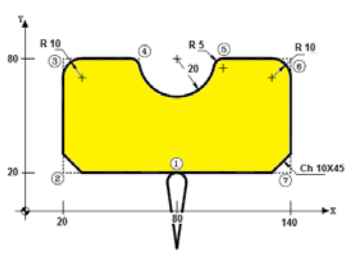
Program

```
(P100 = 0)                ; Starting angle.
(P101 = 360)              ; Final angle.
(P102 = 0.5)              ; Angular step
(P103 = 100)              ; Semi major axis (X).
(P104 = 50)               ; Semi minor axis (Y).

T01 D01
G00 G43 Z100 S2000 M3
Y P104
Z5
G01 Z0 F100
G01 Z-5 F100
N1 (P120 = SIN P100 * P103, P121 = COS P100 * P104)
N2 G01 X P120 Y P121 F500
(P100 = P100 + P102)
(IF P100 LT P101 GOTO N1)
(P100 = P101)
(RPT N1, N2)
G00 Z100 M05
M30
```

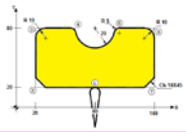
Slide 18

PROFILE EDITOR: Example 1



Slide 19

PROFILE EDITOR Example 1



Profile 1

Profile definition without rounding, chamfers or tangential exit.

Starting point	X1: 80	Y1: -20
Straight	X2: 80	Y2: 20
Straight	X2: 20	Y2: 20
Straight	X2: 20	Y2: 80
Straight	X2: 60	Y2: 80

Counterclockwise arc X2: 100 Y2: 80 XC: 80 YC: 80 R: 20

Straight	X2: 140	Y2: 80
Straight	X2: 140	Y2: 20
Straight	X2: 80	Y2: 20
Straight	X2: 80	Y2: -20

Slide 20

Definition of rounding, chamfers and tangential entry and exit.

Select the CORNERS option and define:

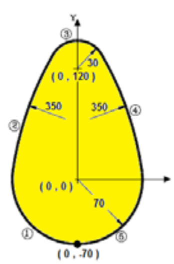
Tangential entry	Select point "1"	Assign radius = 5
Chamfer	Select point "2"	Assign size = 10
Rounding	Select point "3"	Assign radius = 10
Rounding	Select point "4"	Assign radius = 5
Rounding	Select point "5"	Assign radius = 5
Rounding	Select point "6"	Assign radius = 10
Chamfer	Select point "7"	Assign size = 10
Tangential exit	Select point "1"	Assign radius = 5

Press [ESC] to quit the CORNERS option.

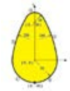
End of editing
Press the **FINISH + SAVE PROFILE** The CNC quits the profile editing mode and the shows the :
ISO-coded program that has been generated.

Slide 21

PROFILE EDITOR: Example 2



Slide 22



Profile 2

Profile definition

Starting point	X1: 0	Y1: -70		
Clockwise arc (1)	XC: 0	YC: 0	R: 70	
Clockwise arc (2)			R: 350	Tang: Yes
Clockwise arc (3)	XC: 0	YC: 120	R: 30	Tang: Yes

The CNC shows all the possible solutions for section 2. Select the correct one.

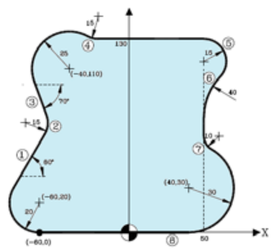
Clockwise arc (4)		R: 350	Tang: Yes
Clockwise arc (5)	X2: 0 Y2: -70 XC: 0 YC: 0	R: 70	Tang: Yes

The CNC shows all the possible solutions for section 4. Select the correct one.

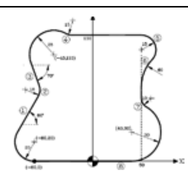
End of editing
Press the **FINISH + SAVE PROFILE** The CNC quits the profile editing mode and the shows the :
ISO-coded program that has been generated.

Slide 23

PROFILE EDITOR: Example 3



Slide 24



Profile 3

Profile definition

Starting point	X1: -60 Y1: 0		
Clockwise arc	XC: -60 YC: 20	R: 20	
Straight (1)	Ang: 60		Tang: Yes

The CNC shows all the possible solutions for section 1. Select the correct one.

Counterclockwise arc (2)		R: 15	Tang: Yes

The CNC shows all the possible solutions for section 2. Select the correct one.

Straight (3)	Ang: -70		Tang: Yes
Clockwise arc	XC: -40 YC: 110	R: 25	Tang: Yes

The CNC shows all the possible solutions for section 3. Select the correct one.

Slide 25

Counterclockwise arc (4) R: 15 Tang: Yes

Straight Y2: 130 Ang: 0 Tang: Yes

The CNC shows all the possible solutions for section 4. Select the correct one.

Clockwise arc (5) XC: 50 R: 15 Tang: Yes

The CNC shows all the possible solutions for section 5. Select the correct one.

Counterclockwise arc (6) R: 40 Tang: Yes

Straight X2: 50 Ang: 270 Tang: Yes

The CNC shows all the possible solutions for section 6. Select the correct one.

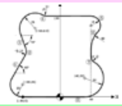
Counterclockwise arc (7) R: 10 Tang: Yes

Clockwise arc XC: 40 YC: 30 R: 30 Tang: Yes

The CNC shows all the possible solutions for section 7. Select the correct one.

Straight (8) X2: -60 Y2: 0 Tang: Yes

The CNC shows all the possible solutions for section 8. Select the correct one.



End of editing

Press the **FINISH + SAVE PROFILE** The CNC quits the profile editing mode and the shows :
the **ISO-coded program** that has been generated.

Slide 1

Fixed Drilling Canned Cycle

G80 G81 G83

((G98 , G99))

Computer Aided Manufacturing
Dr. Tahseen F. Abbas
Lectuer No.2 3rd Year
CAD/CAM Engineering 2014

Slide 2

G80 Fixed Canned Cycle Cutting Mode Cancel

Command Format: G80

Argument Instruction:

This command is to cancel the fixed canned cycle cutting mode of G81~G89.

Besides G80, movement commands G00, G01, G02 and G03 can also be used to cancel fixed canned cycle cutting mode.

Slide 3

Program Sample

```
G17 G90 G00 G54 X0. Y0.;
Z100.;
G99 G81 X0. Y0. Z-20. R10. K1 F100.;
G80; ---------------------- (G81 cycle cancel)

G17 G90 G00 G54 X0. Y0.;
Z100.;
G99 G81 X0. Y0. Z-20. R10. K1 F100.;
G00 Z100.;--------------- (G81 cycle cancel)
```

Slide 4

G81 Drilling Cycle

Command Format:

G81 X… Y… Z… R… K… F… ;

Argument Instruction:

- X… Y… Coordinate of hole position (mm).
- Z… Coordinate of hole bottom (mm).
- R… Coordinate of R point (retraction point) mm.
- K… Times of iteration.
- F… Feedrate (G94 mm/min) (G95 mm/rev).

Slide 5

Action Instruction

1. Fast position to hole position (X, Y, yet maintain the original height of tool);

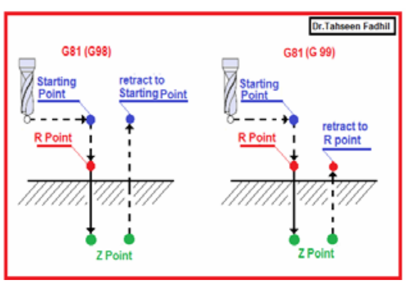
2. Fast position to the coordinate of R point (R);

3. Cut to the hole bottom position (Z) with specified cutting feedrate and spindle speed;

4. In G98 mode, fast return to the starting point; In G99 mode, fast return to R point;

5. If K is specified (> 1) , repeat steps 2~4 until reach specified drilling iteration times; otherwise procedure ends;

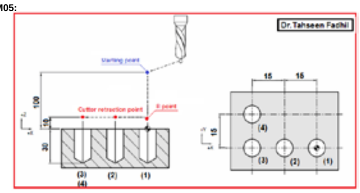
Slide 6



Slide 7

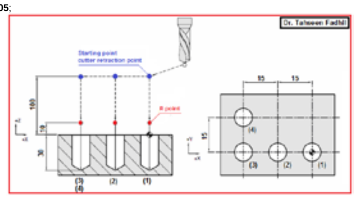
Program Sample

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G81 X0. Y0. Z-30. R10. K1 F100.; ---   (1)
X-15.;--------------------------------------------   (2)
X-30.;--------------------------------------------   (3)
X-30. Y15.; --------------------------------------   (4)
G80 G91 G28 X0. Y0. Z0.;
M05:
```



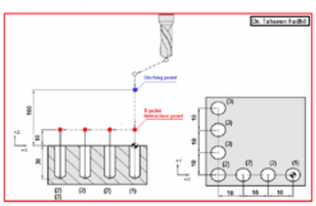
Slide 8

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.
G98 G81 X0. Y0. Z-30. R10. K1 F100.; --   (1)
X-15.;--------------------------------------------   (2)
X-30.;--------------------------------------------   (3)
X-30. Y15.; --------------------------------------   (4)
G91 G80 G28 X0. Y0. Z0.;
M05;
```



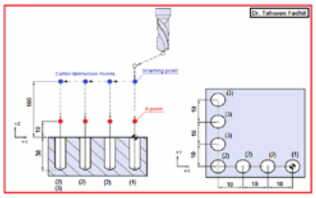
Slide 9

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G81 X0. Y0. Z-20. R10. K1 F100.; --   (1)
G91 X-10. K3; --------------------------------   (2)
Y10. K3; ----------------------------------------   (3)
G91 G80 G28 X0. Y0. Z0.;
M05;
```



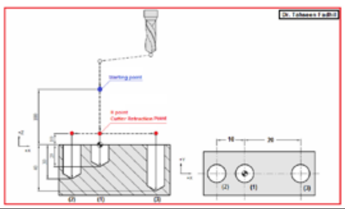
Slide 10

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G81 X0. Y0. Z-20. R10. K1 F100.; ---  (1)
G91 X-10. K3; -------------------------------------  (2)
Y10. K3; ------------------------------------------  (3)
G91 G80 G28 X0. Y0. Z0.;
M05;
```



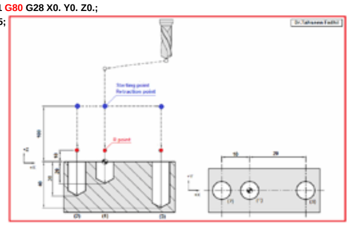
Slide 11

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G81 X0. Y0. Z-20. R10. K1 F100.; -1)
X-10. Z-30.; --------------------------------  (2)
X20. Z-40.; ---------------------------------  (3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



Slide 12

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G81 X0. Y0. Z-20. R10. K1 F100.; --  (1)
X-10. Z-30.; --------------------------------------  (2)
X20. Z-40.; ---------------------------------------  (3)
G91 G80 G28 X0. Y0. Z0.;
M05;
```



Slide 1

Fixed Canned Cycle

G80 G81 G83

((G98,G99))

Computer Aided Manufacturing
Dr. Tahseen F. Abbas
Lectuer No.3 3rd Year
CAD/CAM Engineering 2014

Slide 2

G83 Peck Drilling Cycle

Command Format

G83 X.. Y.. Z .. R .. Q .. K .. F .. ;

Argument Instruction

X ..Y.. Coordinate of hole position (mm).
Z .. Coordinate of hole bottom (mm).
R .. Coordinate of R point (i.e. retraction point) (mm).
Q .. Cutting feed per time (mm).
K .. Times of iteration.
F .. Feedrate (G94 mm/min) (G95 mm/rev).

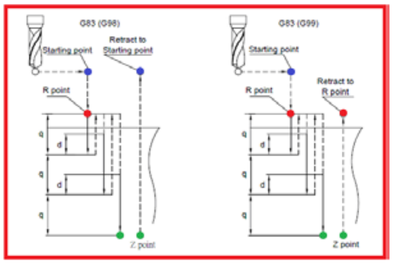
Slide 3

Action Instruction (taking G17 plane for example)

1. Fast position to hole position (X, Y, yet maintain original tool height);
2. Fast position to the coordinate of R point (R);
3. Peck drill for a feed according to specified feedrate and spindle speed;
4. Fast return to R point;
5. Fast position to a certain height away from the last manufacturing point,
6. Cutting feed
7. Fast return to R point ;
8. Repeat steps 5~7 until cutting to the hole bottom;
9. In G98 mode, fast return to the starting point; In G99 mode, fast return to R point;
10. If K is specified (> 1), repeat steps 2~9 until obtaining specified drilling repetion times; otherwise procedure ends;

Slide 4

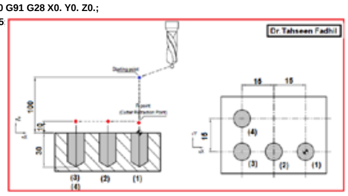
Illustration



Slide 5

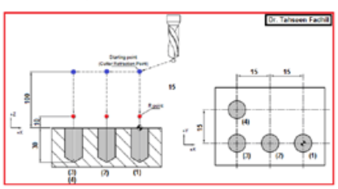
Program Sample

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G83 X0. Y0. Z-30. R10. Q4 K1 F100.;-   (1)
X-15.;------------------------------------------   (2)
X-30.;------------------------------------------   (3)
X-30. Y15.; -----------------------------------   (4)
G80 G91 G28 X0. Y0. Z0.;
M05
```



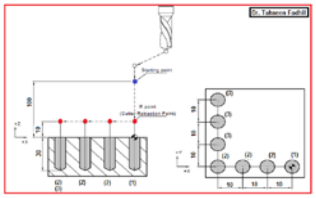
Slide 6

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G83 X0. Y0. Z-30. R10. Q4 K1 F100.;-   (1)
X-15.;------------------------------------------   (2)
X-30.;------------------------------------------   (3)
X-30. Y15.; -----------------------------------   (4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



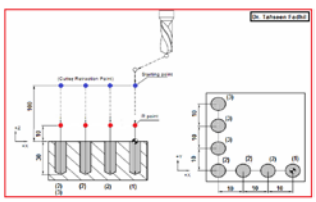
Slide 7

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G83 X0. Y0. Z-20. R10. Q4 K1 F100.;--   (1)
G91 X-10. K3; ------------------------------------   (2)
Y10. K3; -----------------------------------------   (3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



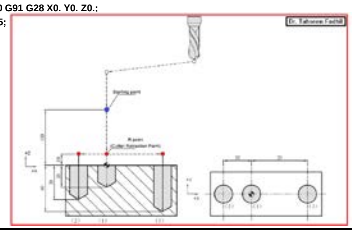
Slide 8

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G83 X0. Y0. Z-30. R10. Q4 K1 F100.;---  (1)
G91 X-10. K3; ------------------------------------   (2)
Y10. K3; -------------------------------------------   (3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



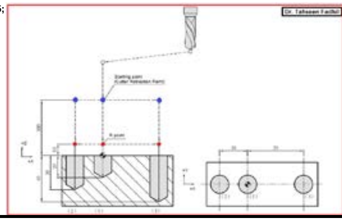
Slide 9

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G83 X0. Y0. Z-20. R10. Q4 K1 F100.;--   (1)
X-10. Z-30.; ---------------------------------------   (2)
X20. Z-40.; ----------------------------------------   (3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



Slide 10

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G83 X0. Y0. Z-20. R10. Q4 K1 F100.;--   (1)
X-10. Z-30.; ----------------------------------------   (2)
X20. Z-40.;-------------------------------------------   (3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



Slide 1

Right Hand Screw Thread Tapping Cycle

G84

Lecture FOUR

Computer Aided Manufacturing

Dr. Tahseen F. Abbas

CAD/CAM Engineering 2014

Slide 2

G84 Right-Handed Screw Thread Tapping Cycle

Command Format:

G84 X .. Y .. Z .. R .. P .. K .. F.. ;

Argument Instruction

- **X** ..**Y** .. Coordinate of hole position (mm).
- **Z** .. Coordinate of hole bottom (mm).
- **R** .. Coordinate of R point (i.e. retraction point) (mm).
- **P** .. Dwell time at hole bottom (1/1000 sec), minimum unit, and decimal times are not allowed.
- **K** .. Times of iteration.
- **F** .. Cutting feedrate (**G94** mm/min) (**G95** mm/rev).

Slide 3

Action Instruction (taking G17 plane for example)

1. Fast position to hole position (X, Y, yet main tain original tool height);
2. Fast position to the coordinate of R point (R);
3. Tapping begins, spindle rotates clockwisely.
4. Cut to the hole bottom position (Z) with specified cutting feedrate
5. Spindle stops; if P is specified, dwell at the hole bottom for specified time;

6. Spindle rotates reversely, cut to R point with specified cutting feedrate

7. Tapping ends, spindle stops; If P is specified, dwell at R point

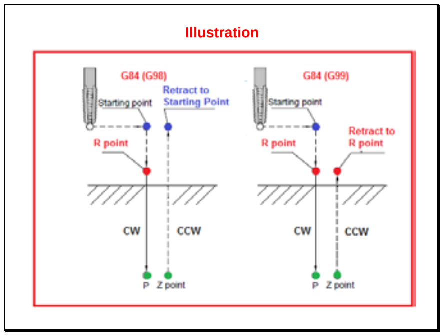
8. In G98 mode, fast return to the starting point; In G99 mode, fast return to R

9. If K is specified (> 1), repeat steps 2~8 until obtaining specified drilling repetition times; otherwise procedure ends;

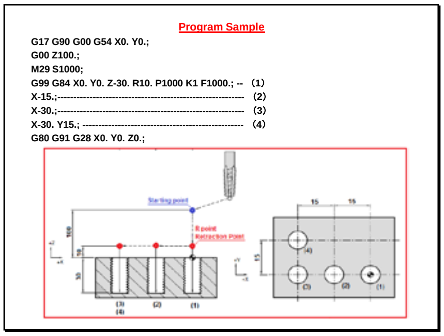
10. In G91 mode, argument R specifies the distance between R point and the starting point; argument Z specifies the distance between hole bottom position and R point; if K is specified (> 1), after each tapping process, the hole will do incremental offset according to specified X, Y and then continue next tapping process.

11. In G94 mode, cutting feedrate (F) is "rotation speed (S) × thread pitch (PITCH)"; In G95 mode, cutting feedrate (F) is "thread pitch (PITCH)".;

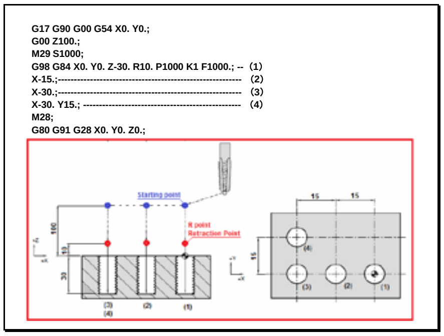
Slide 4



Slide 5

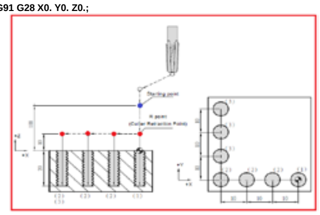


Slide 6



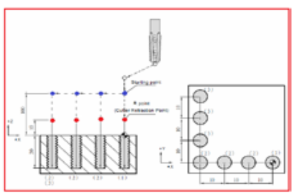
Slide 7

```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G99 G84 X0. Y0. Z-20. R10. P1000 K1 F1000.; --   (1)
G91 X-10. K3; -------------------------------------------   (2)
Y10. K3; --------------------------------------------------   (3)
M28;
G80 G91 G28 X0. Y0. Z0.;
```



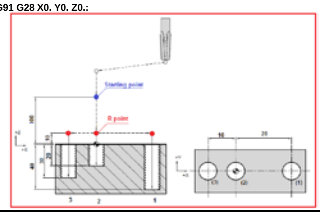
Slide 8

```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G98 G84 X0. Y0. Z-20. R10. P1000 K1 F1000.; -----   (1)
G91 X-10. K3; ----------------------------------------------   (2)
Y10. K3; -----------------------------------------------------   (3)
M28;
G80 G91 G28 X0. Y0. Z0.;
```



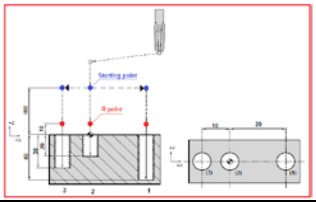
Slide 9

```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G99 G84 X0. Y0. Z-20. R10. P1000 K1 F1000.; ---   (1)
X-10. Z-30.; -----------------------------------------------   (2)
X20. Z-40.;--------------------------------------------------   (3)
M28;
G80 G91 G28 X0. Y0. Z0.:
```



Slide 10

```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G98 G84 X0. Y0. Z-20. R10. P1000 K1 F1000.; ----  (1)
X-10. Z-30.; ---------------------------------------------------   (2)
X20. Z-40.;-----------------------------------------------------   (3)
M28;
G80 G91 G28 X0. Y0. Z0.;
```



Slide 1

MILLING CENTER PROGRAMMING WITH SUBPROGRAMS

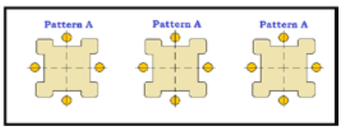
Lecture 22

Assist. Prof. Dr.Tahseen F. Alani
CAD/CAM Engineering
2014-2015
7 Slides

Slide 2

Subprogram concept:

The programmer will often encounter a sequence of machining instructions that must be repeated. The programmer could code the instructions to machine each pattern individually for the part shown in fig.



In this case the subprogram would be called three times to machine pattern A

Slide 3

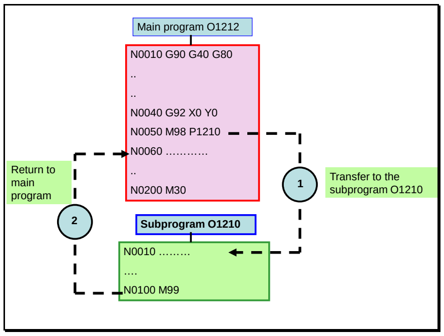
Format of the Subprogram:

Subprograms are repeated according to the following format:

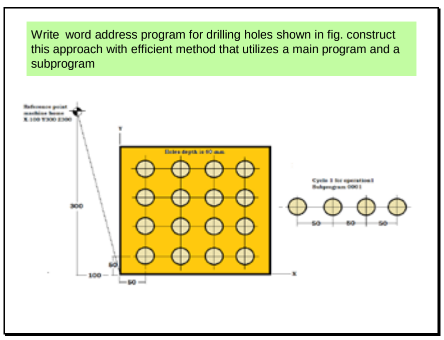
```
O XXXX
Block 1
Block 2
..
..
..
Block n
M99
```

M99 is the last statement in the subprogram and signals are return to the main program

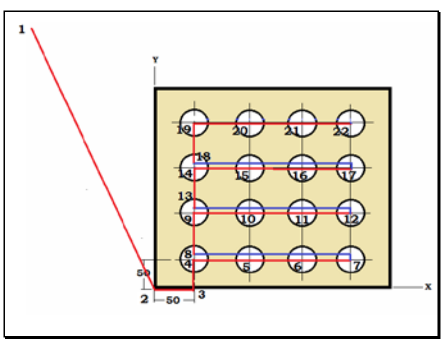
Slide 4



Slide 5



Slide 6



Slide 7

Word address command Main Program	Word address command Subprogram
O1616	O0001
N0010 G90 G21 G40 G80	N0010 G90 X50
N0020 T0101 M06	N0020 G91 Y50
N0030 G00 G90 X0 Y0 Z0 S2000 M03	N0030 G90 G83 Z-60 R1 Q1 F10
N0040 G43 Z0.1 M08	N0040 M98 P0002 L3
M0050 M98 P0001 L4	N0050 G80
N0060 G80	N0060 M99
N0070 G00 G90 Z1.0 M05	O0002
N0080 M09	N0010 G91 X50
N0090 X0 Y0 Z0	N0020 M99
N0100 M30	