COMPUTER INTEGRATED MANUFACTURING LAB MANUAL (17MEL77)

DEPARTMENTOFMECHANICALENGINEERING

BAPUJI INSTITUTEOF ENGINEERINGANDTECHNOLOGY

DAVANGERE-577 004

DEPARTMENT OF

MECHANICAL ENGINEERING

COMPUTER INTEGRATED MANUFACTURING LAB MANUAL

2018

(*17MEL77*)

As per VTU Syllabus CBCS scheme for VII Semester

Name : **……**

USN : **……**

Semester:**……** Batch No **…….**

Ravikumar H.N

Faculty Incharge

Mallikarjuna V.K

Instructor

BAPUJI INSTITUTE OF ENGINEERING AND TECHNOLOGY

DAVANGERE- 577 004

VISION OF THE INSTITUTE

To be center of excellence recognized nationally and internationally, in distinctive areas of engineering education and research, based on a culture of innovation and invention.

MISSION OF THE INSTITUTE

BIET contributes to the growth and development of its students by imparting a broad based engineering education and empowering them to be successful in their chosen field by inculcating in them positive approach, leadership qualities and ethical values.

VISION OF THE DEPARTMENT

The department endeavors to be a center of excellence, to provide quality education leading the students to become professional mechanical engineers with ethics, contributing to the society through research, innovation, entrepreneurial and leadership qualities.

MISSION OF THE DEPARTMENT

1. To impart quality technical education through effective teaching- learning process leading to development of professional skills and attitude to excel in Mechanical Engineering.
2. To interact with institutes of repute, to enhance academic and research activities.
3. To inculcate creative thinking abilities among students and develop entrepreneurial skills.
4. To imbibe ethical, environmental friendly and moral values amongst students through broad based education

PROGRAM EDUCATIONAL OBJECTIVES (PEO'S)

1. Enable to understand mechanical engineering systems those are technically viable, economically feasible and socially acceptable to enhance quality of life.
2. Apply modern tools and techniques to solve problems in mechanical and allied engineering streams.
3. Communicate effectively using innovative tools, to demonstrate leadership and entrepreneurial skills.
4. Be a professional having ethical attitude with multidisciplinary approach to achieve self and organizational goals.
5. Utilize the best academic environment to create opportunity to cultivate lifelong learning skills needed to succeed in profession.

PROGRAM SPECIFIC OUTCOMES (PSO'S)

PS01:-Apply the acquired knowledge in design, thermal, manufacturing and interdisciplinary areas for solving industry and socially relevant problems.

PS02:-To enhance the abilities of students by imparting knowledge in emerging technologies to make them confident mechanical engineers.

COMPUTER INTEGRATED MANUFACTURING LAB B.E, VII Semester, Mechanical Engineering [As per Choice Based Credit System (CBCS) scheme]			
Course Code	17MEL77	**CIE Marks**	**40**
Number of Lecture Hours/Week	**03 (1 Hour Instruction+ 2 Hours Laboratory)**	**SEE Marks**	**60**
Total Hours	**40**	**Exam Hours**	**03**
Credits –02			

Course Objectives:

CLO1	To expose the students to the techniques of CNC programming and cutting tool path generation through CNC simulation software by using G-Codes and M-codes
CLO2	To educate the students on the usage of CAM packages and cut part on virtual CNC machine simulator.
CLO3	To make the students understand the importance of automation in industries through exposure to FMS, Robotics, and Hydraulics and Pneumatics.

Part-A

Manual CNC part programming for 2 turning and 2 milling parts. Selection and assignment of tools, correction of syntax and logical errors, and verification of tool path.

CNC part programming using CAM packages. Simulation of Turning, Drilling, Milling operations. 3 typical simulations to be carried out using simulation packages like: Cadem CAM Lab-Pro, Master- CAM.

Program generation using software. Optimize spindle power, torque utilization, and cycle time. Generation and printing of shop documents like process and cycle time sheets, tool list, and tool layouts. Enter program, take tool offsets, cut part in single block and auto mode, measure the virtual part on screen in the virtual CNC machine simulator, for standard CNC control systems FANUC, FAGOR, HAAS and SINUMERIK.

Part B

(Only for Demo/Viva voce)

FMS (Flexible Manufacturing System): Programming of Automatic storage and Retrieval system (ASRS) and linear shuttle conveyor Interfacing CNC lathe, milling with loading unloading arm and ASRS to be carried out on simple components.

(Only for Demo/Viva voce)

Robot programming: Using Teach Pendent & Offline programming to perform pick and place, stacking of objects (2 programs).

Pneumatics and Hydraulics, Electro-Pneumatics: 3 typical experiments on Basics of these topics to be conducted.

After studying this course, students will be able to:

Course Outcomes:

CO1	Understand & write CNC part program for Turning, Facing, Chamfering, Grooving, Step turning, Taper turning, Circular interpolation etc.
CO2	Analyse CNC Mill Part programming for Point to point motions, Line motions, Circular interpolation, Contour motion, Pocket milling- circular, rectangular, Mirror commands etc.
CO3	choose high end CAM packages for machining complex parts and state of art cutting tools and related cutting parameters; optimize cycle time, set up and cut part on anduse Canned Cycles for Drilling, Peck drilling, Boring, Tapping, Turning, Facing,Taper turning Thread cutting etc.
CO4	Simulate Tool Path for different Machining operations of small components using CNC Lathe & CNC Milling Machine. Understand& write programs for Robot control; understand the operating principles of hydraulics, pneumatics and electro pneumatic systems.

Scheme for Examination:

Two Questions from Part A - 60 Marks (30 +30) Viva-Voce - 20 Marks

Total: 80 Marks

DO's

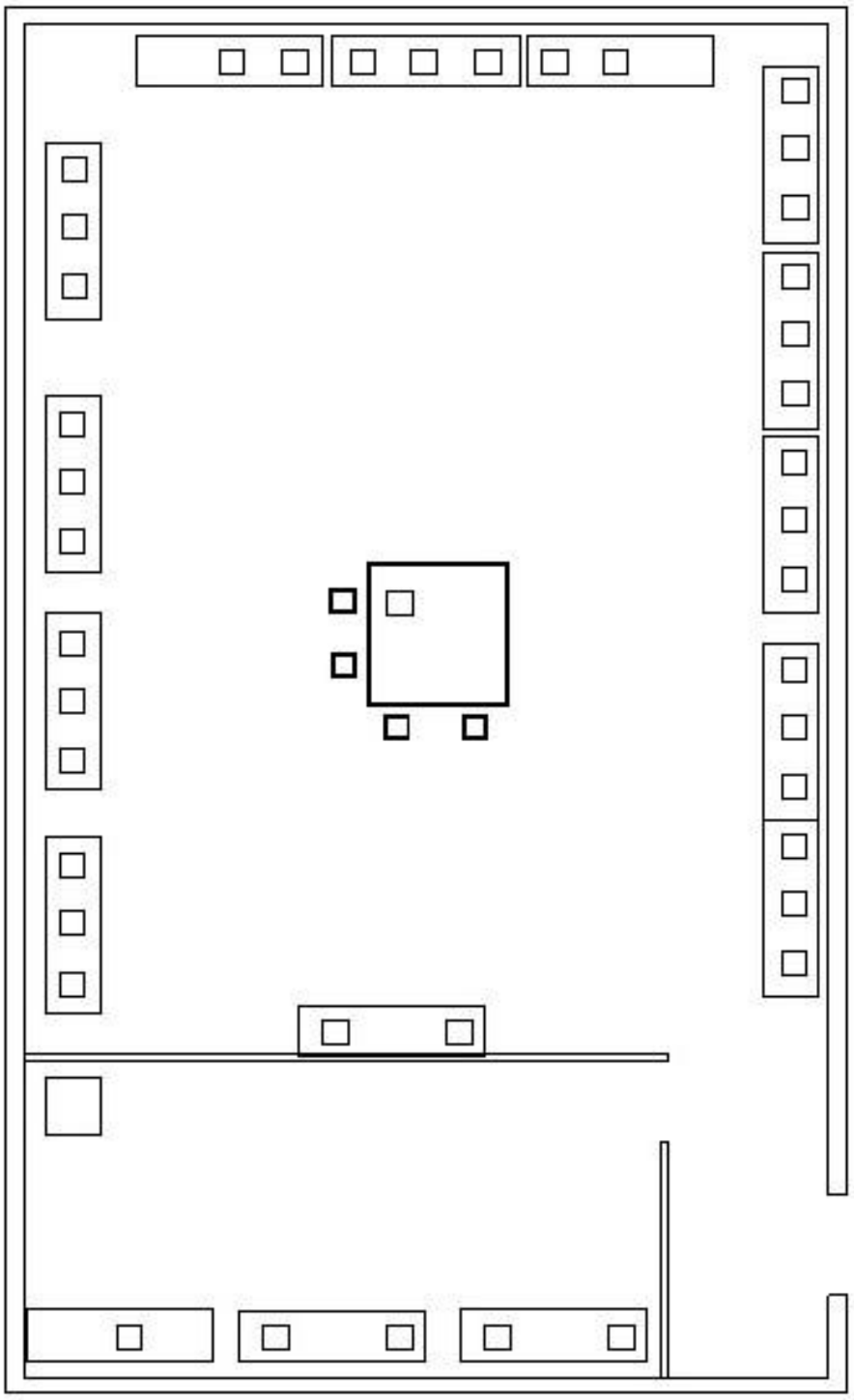
1. Students must always wear uniform and shoes before entering the lab.
2. Proper code of conduct and ethics must be followed in the lab.
3. Windows and doors to be kept open for proper ventilation and air circulation.
4. Check for the electrical connections and inform if any discrepancy found to the attention of lecturer/lab instructor.
5. Performtheexperimentunderthesupervision/guidanceofalecturer/labinstructor only.
6. In case of fire use fire extinguisher/throw the sand provided in the lab.
7. Any unsafe conditions prevailing in the lab can be brought to the notice of the lab incharge.

DONT's

1. Do not touch any system without their prior knowledge,
2. Never overcrowd the laboratory Leave sufficient space for the person to operate the equipment's.
3. Never rest your hands on the system and the display board.

CONTENTS

CIM LAB LAY OUT



CNC PART PROGRAMMING

Software: SwanSoft FANUC 0iT CNC Turning

List of G & M Codes used in SSCNC for Turning Center

GENERALLY USED G-CODES IN TURNING

STANDARD G CODE	FUNCTION
G00	Positioning in Rapid
G01	Straight Interpolation
G02	Circular Interpolation (CW)
G03	Circular Interpolation (CCW)
G04	Dwell
G10	Change in Setting Value
G20	Data Input in Inch
G21	Data Input in metric
G22	Stored Stroke Limit on
G23	Stored Stroke Limit off
G25	Spindle Speed Fluctuation detection OFF
G26	Spindle Speed Fluctuation detection ON
G27	Machine Reference Check
G28	Automatic Machine Reference
G29	Return From Reference
G30	The 2nd Reference return
G31	Skip Function
G32	Threading Process Cycle
G40	Cancel of Compensation
G41	Compensation of the left
G42	Compensation of the right
G65	Call custom Macro
G70	Repeat Cycle - Finishing
G71	Stock Removal in Turning
G72	Stock Removal in Facing
G73	Pattern Repeating Cycle
G74	Peck Drilling in Z Direction
G75	Grooving in X - Direction
G76	Thread Process Cycle
G90	Fixed Process Cycle in Turning
G92	Fixed Process Cycle in Threading
G94	Fixed Process Cycle in Facing
G96	Constant Surface speed
G97	Constant Surface speed cancel
G98	Per min feed
G99	Per revolution feed

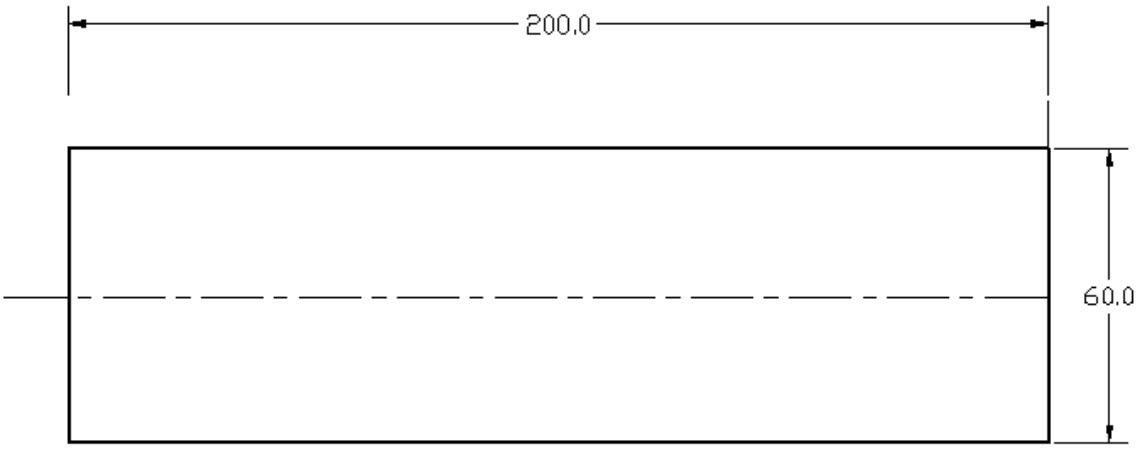
GENERALLY USED M-CODES IN TURNING

STANDARD M CODES	FUNCTION
M00	Unconditional program stop
M01	Conditional program stop
M02	End of Program with return to program start
M03	Spindle Rotation, Clock Wise
M04	Spindle Rotation, Counter Clock Wise
M05	Spindle Stop
M06	Tool Change
M08	Coolant ON
M09	Coolant OFF
M30	End of Program, Reset and return to program beginning
M94	Cancel Mirror Image
M95	X Coordinate Mirror Image
M96	Y Coordinate Mirror Image
M98	Subprogram call
M99	End of Sub Program

List of G & M Codes used in SSCNC for Turning Center

Function	Address	Meaning of address
Program number	O(EIA)/(ISO)	Program number
Block sequence number	N	Sequence number
Preparatory function	G	Sercifies a motion mode (Linear, arc, etc)
Dimension word	X, Z	Command of moving position(absolute type) of each axis
	U, W	Instruction of moving distance and direction(incremental type)
	I, K	Ingredient of each axis and chamfering volume of circulat center
	R	Radius of circle, corner R, edge R
Feed function	F, E	Designation of feedrate and thread lead
Auxiliary function	M	Command of ON/OFF for operating parts of machine
Spindle speed function	S	Designation of speed of main spindle or rotation time of main spindle
Function (Tool)	T	Designation of tool number and tool compensation number
Dwell	P, U, X	Designation of dwell time
Dewignation of program number	P	Designation of calling number of auxiliary program
Designation of sequence No	P, Q	Callling of compound repeat cycle, end number
Number of repetitions	L	Repeat time of auxiliary program
Parameters	A, D, I, K	Parameter at fixed cycle

1. Write a Absolute part program to create the following profile with facing and OD Turning and write comments.



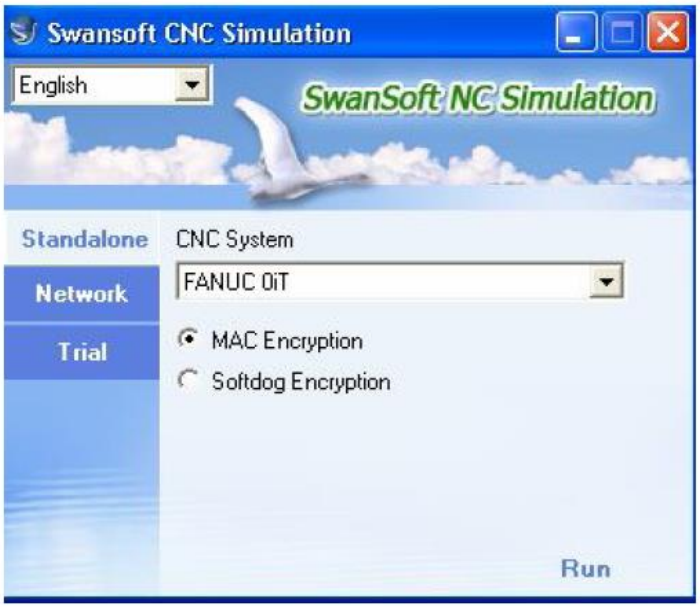
Work piece Information

Bar
Diameter: 70.000
Length: 250.000
Material:08F low-carbon steel

Tool Information

- Work offset at the center
- Tools Management – Select the External turning 35^0 / 90^0 tool.

Procedure



Choose the stand alone button

Click on drop down button

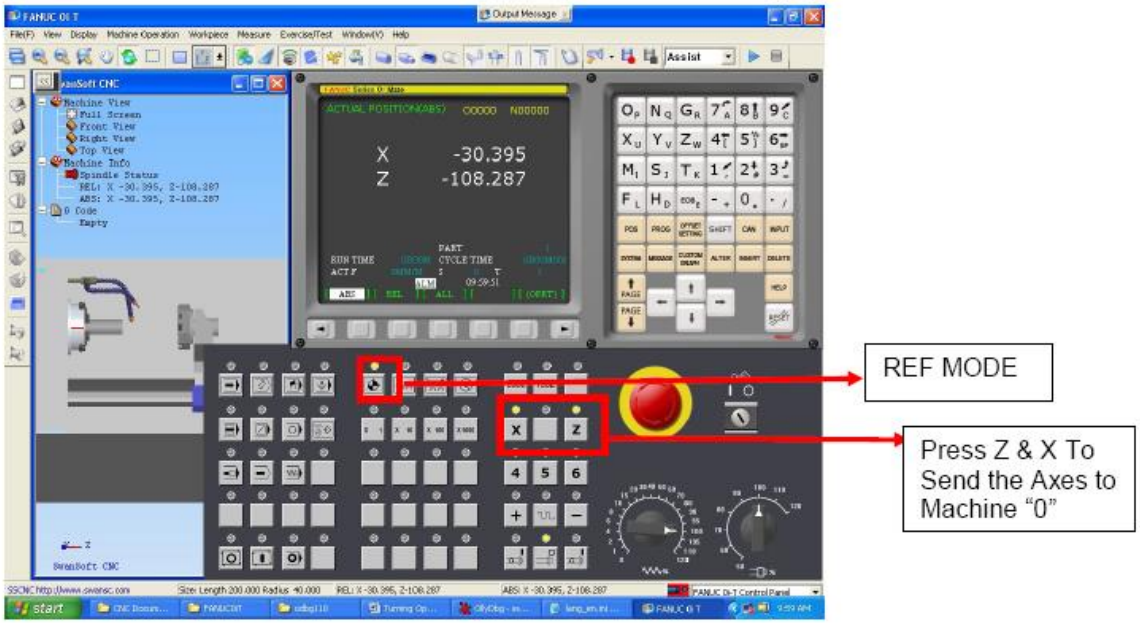
Select Fanuc OiT CNC System

Select MAC Encryption

Go to Run

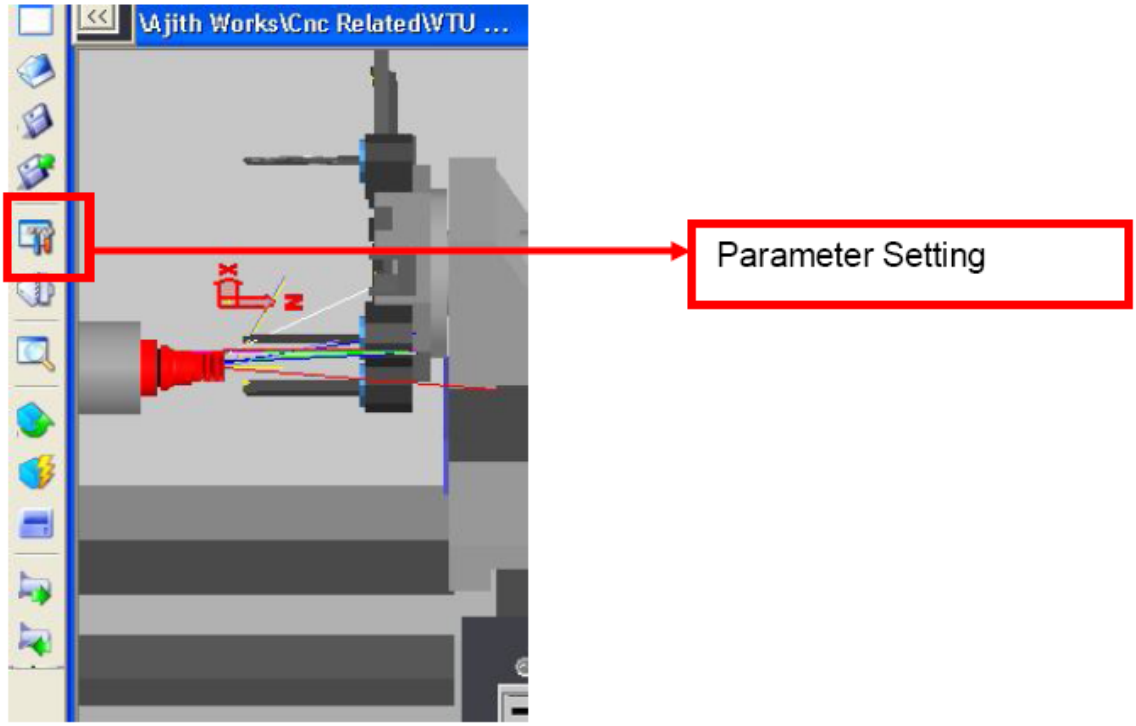
Release the Emergency button

Press write protect off



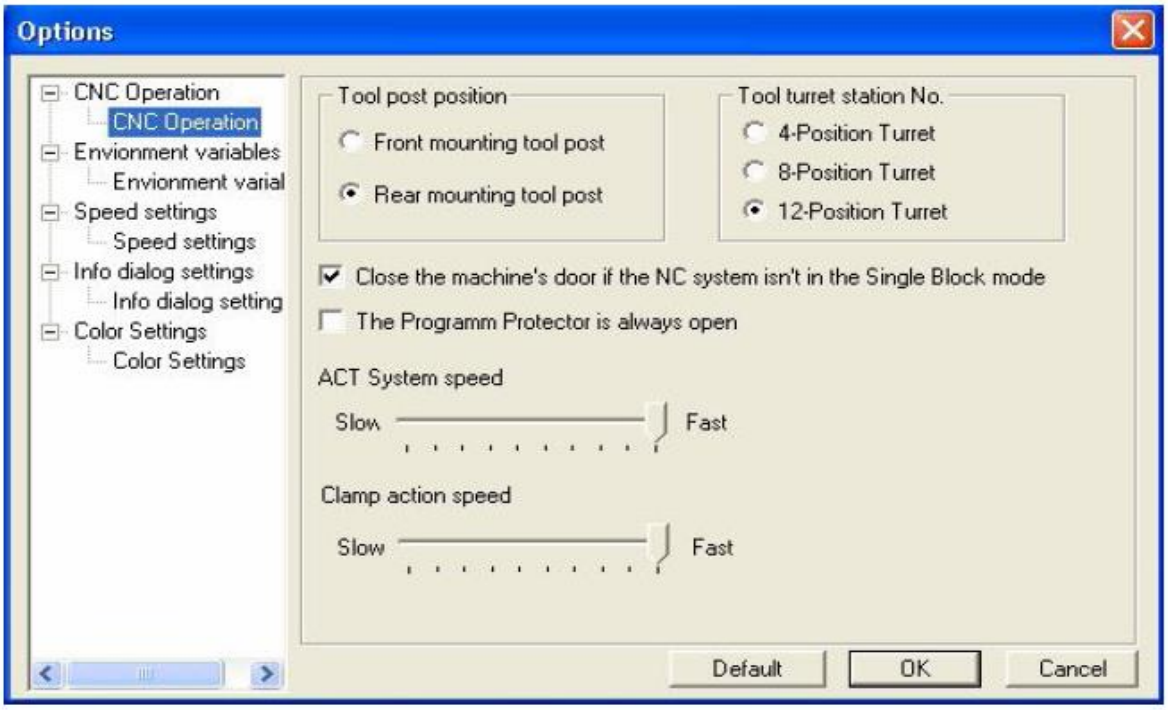
Press Reference and X and Z

Go to Parameter setting

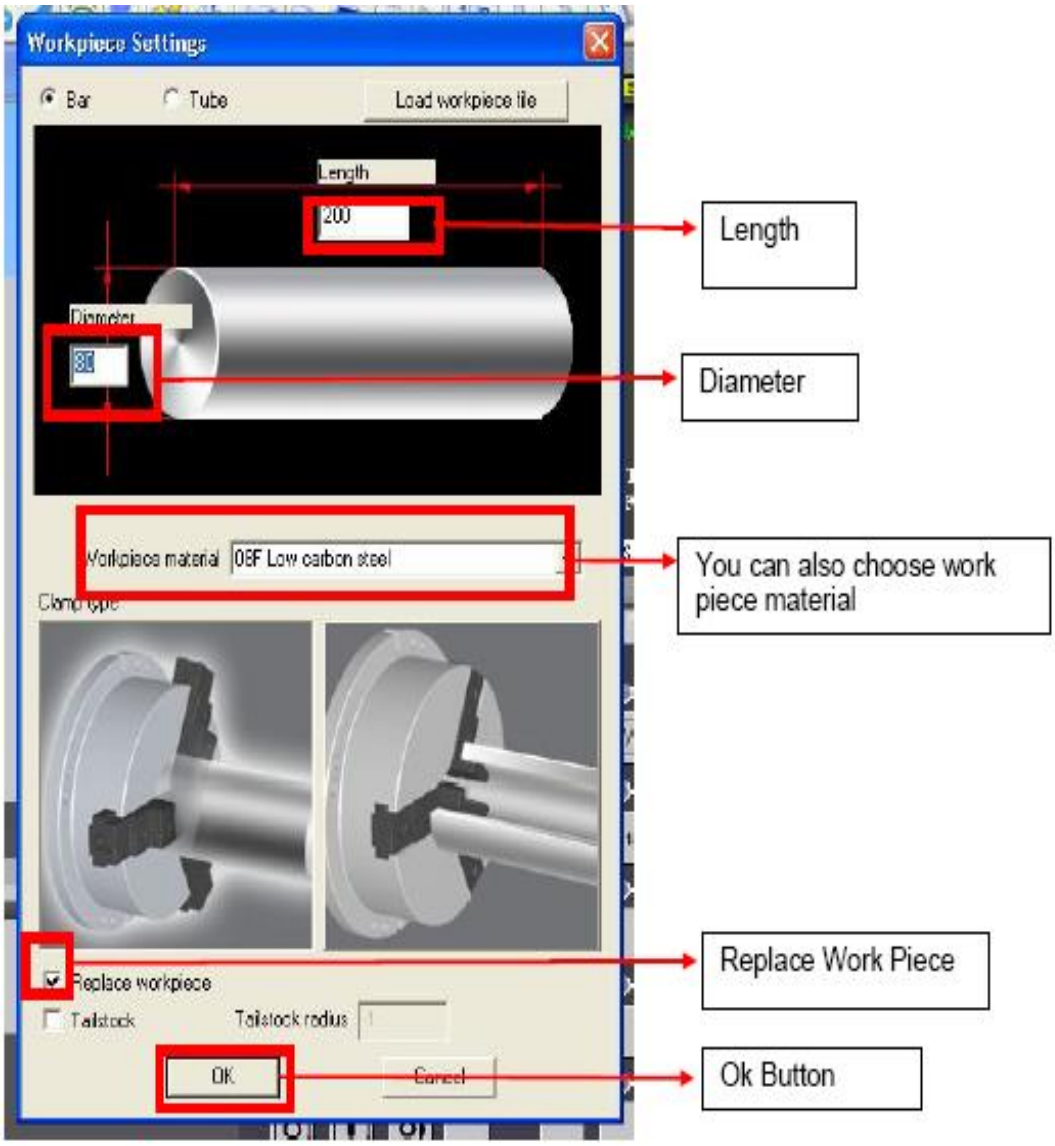


Press parameter setting button to open Parameter setting window.

Select– 4 Position turret and press ok



Workpiece Setting, Go to – Stock size



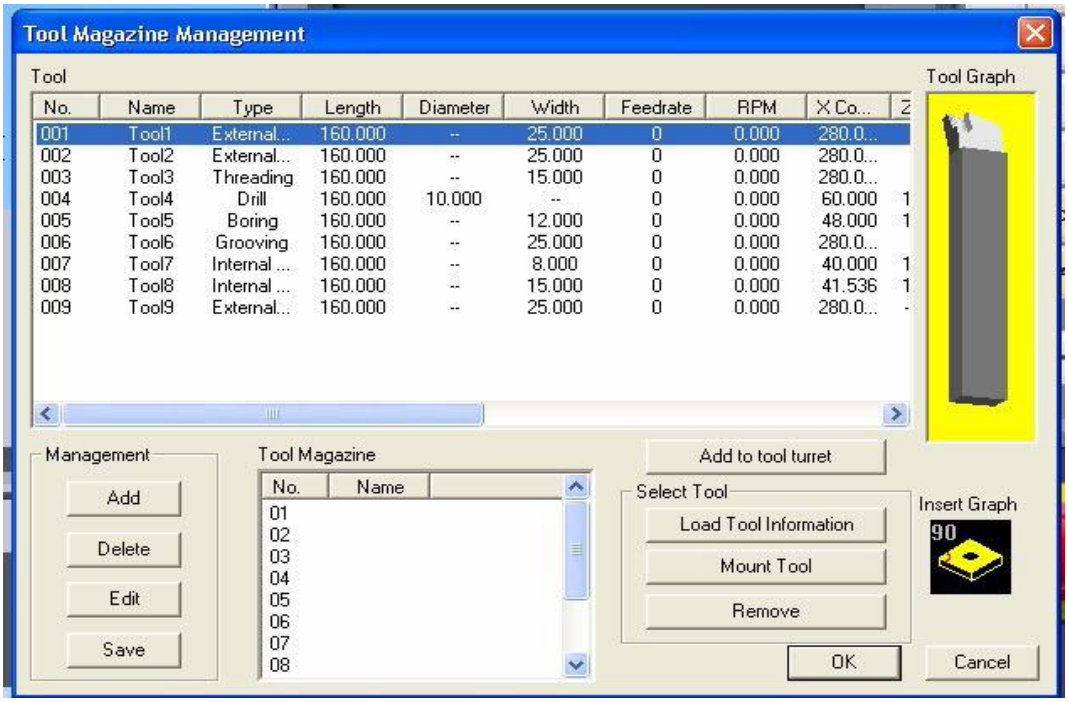
Choose Length of Work Piece =250mm

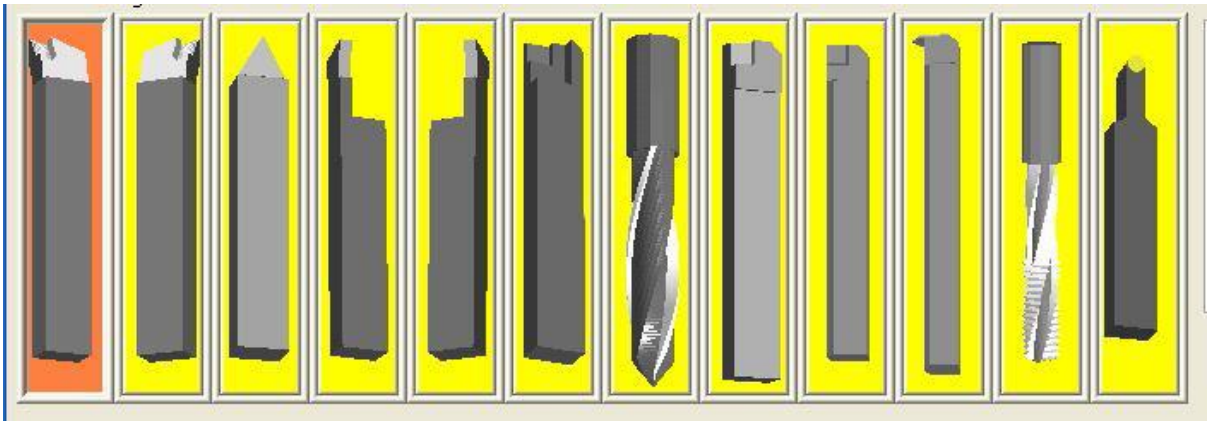
- Choose Diameter of the Work Piece=80mm
- Check Replace work piece button to replace existing work piece .
- Press OK button.

Tool Management:

- Add New Tool
- Edit Tool

- Tool Selection





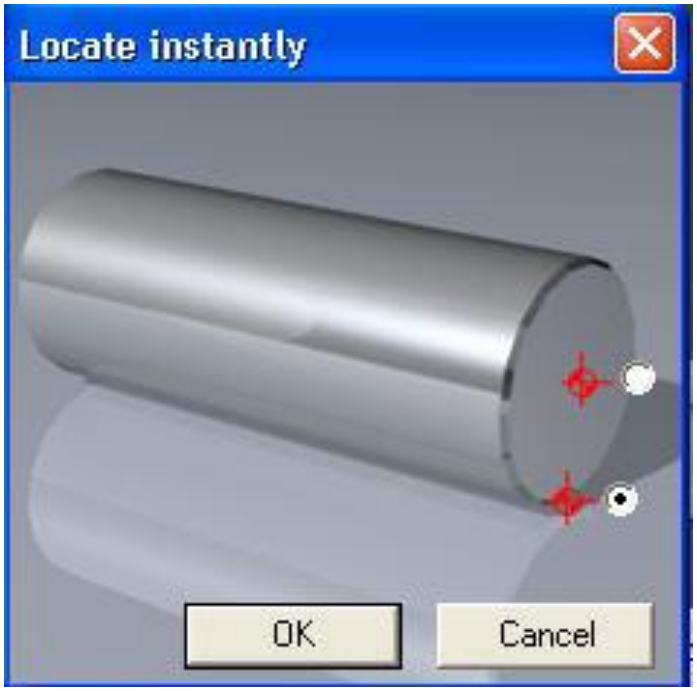
Enter Diameter and Length of the tool in respected places

- Enter Rpm and Feed .
- Enter thickness of the insert.
- Enter Shank and its width.
- Then click OK button to add the toll to Library.

– select the External turning $35/55/90^0$ tool – drag and drop to tool magazine– mount the tool

Workpiece setting – rapid position at center

Press Work set setting Button to view above Menu. To open **Locate instantly window** select Rapid Position button.

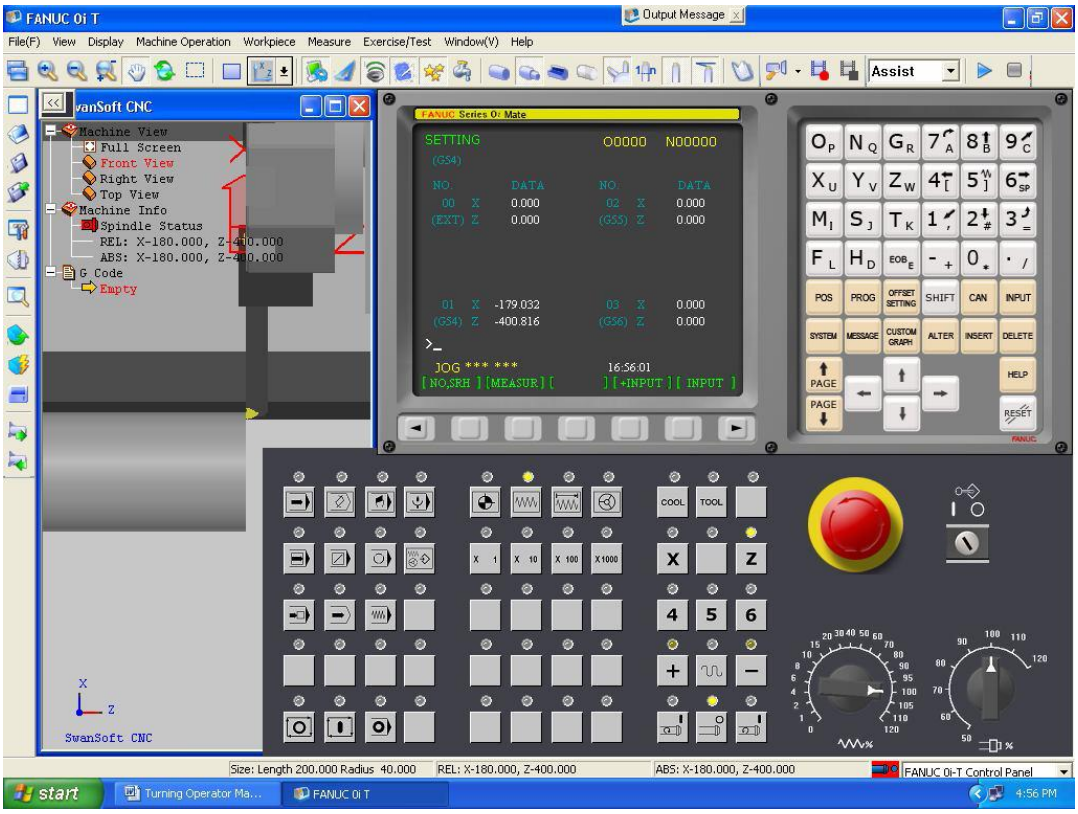


Select the position and press OK button.

Close the machine door

Offset setting – work – X00 – measure --- Z00 – measure

(Do not use this step if X and Z axis home position is zero)

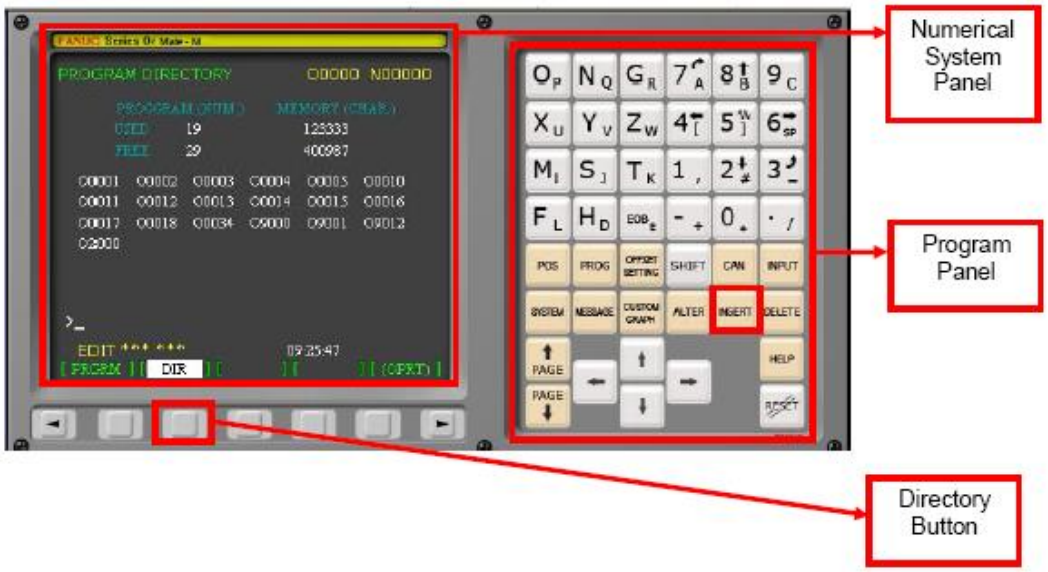


Getting Ready to enter your first program

Before writing program you have to register for memory space.

Press Edit button.

Press PROG Button.



prog – edit – DIR – O0013 (Register your program No. start with alphabet 'O')

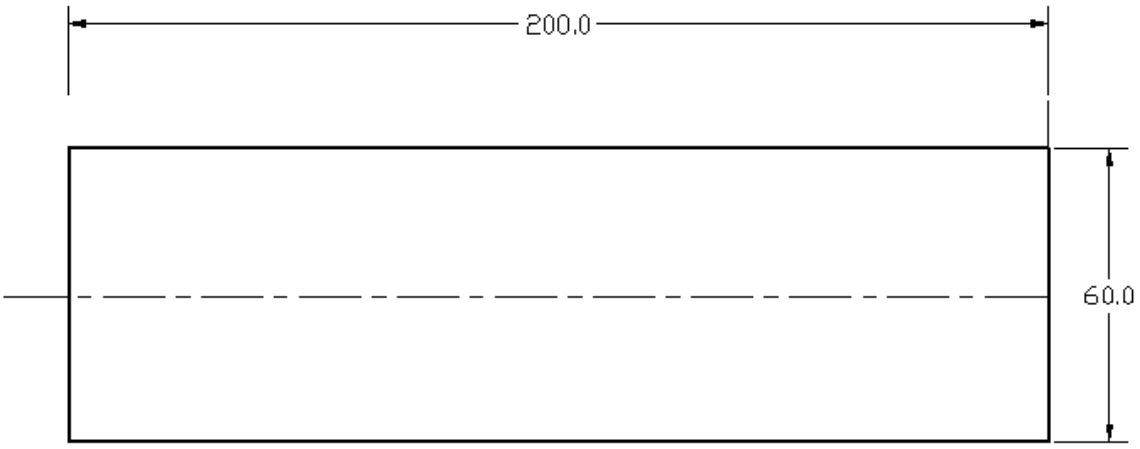
Press **DIR** Button in the Numerical panel to register the program. Type registry number in **Program panel**.

The registry number start with letter **'O'**

Followed by number (**ex: O0009**).

Type the program by using Program panel or by using the computer key board .

In program panel press letter in smaller font press **Shift** then press button for example to type letter p press **Shift** then press **O and P** Button. After entering one line of code press **EOF** button press **Insert** button add this code to numerical control panel.



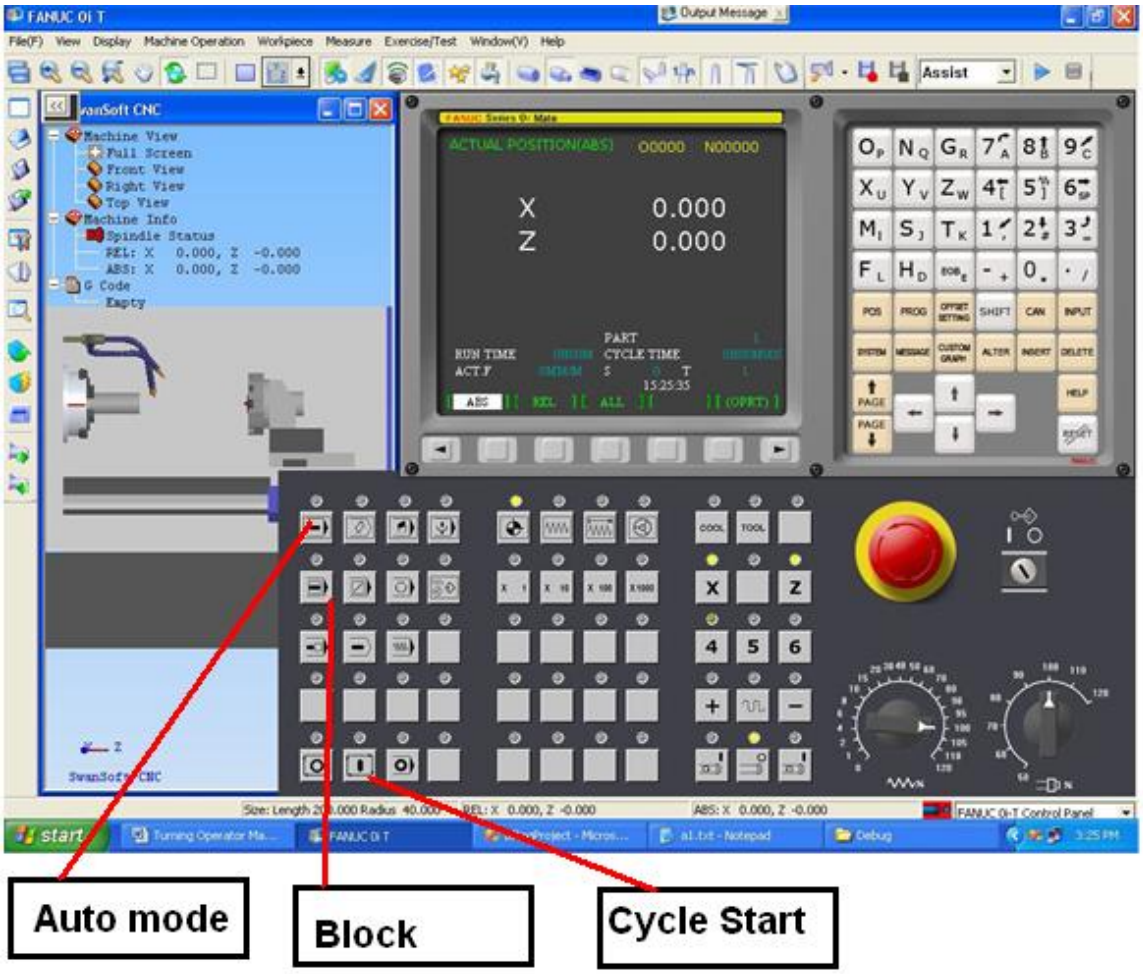
Absolute Method	Comments / Remarks
O0012	Program Number
G54	Work co-ordinate system
T0100	Tool T01 selection
S400M03	Spindle to rotate at 400 RPM in Clockwise direction
M08	Coolant on
G00X00Z00t0101F2.5	Tool T01 Positions in Rapid traverse
G00X71	1 mm away from the set coordinate
G01X0Z-1F0.25	Lanier Interpolation (G01) Movements in X & Z Directions to create the paths. The feed is at 0.25mm/rev. Table feed = 0.25 x 400 rpm = 45mm/min
G00X71	
G01Z-5	
G01X0	
G00X71Z-5	
G01X65	
G01Z-205	
G00X66Z-5	
G01X60	
G01Z-205	
G00X61	
G28U0W0	Go Back to tool change position in rapid (m/c 00)
M09	Coolant off
M05	Spindle off
M30	Program End

After entering the program

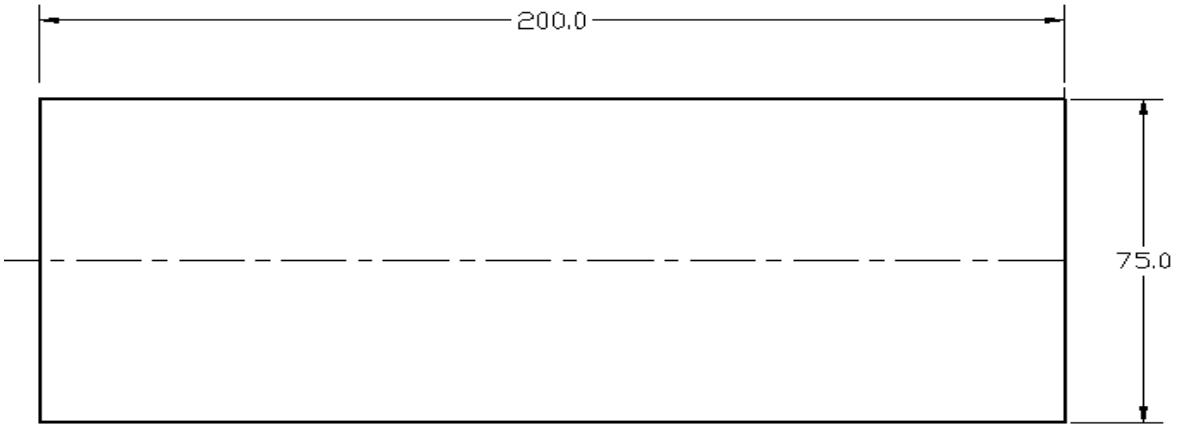
Go to Auto mode

Select block by block

Cycle Start


Auto mode
Block
Cycle Start

2. Write a part program to create the following profile with facing and OD Turning. Absolute, Incremental and Incremental with using canned cycle and write the comments of the each line program .



Work piece Information

Bar
Diameter: 80.000
Length: 250.000
Material:08F low-carbon steel

Tool Information

- Work offset at the center
- Tools Management – Select the External turning 35^0 / 90^0 tool.

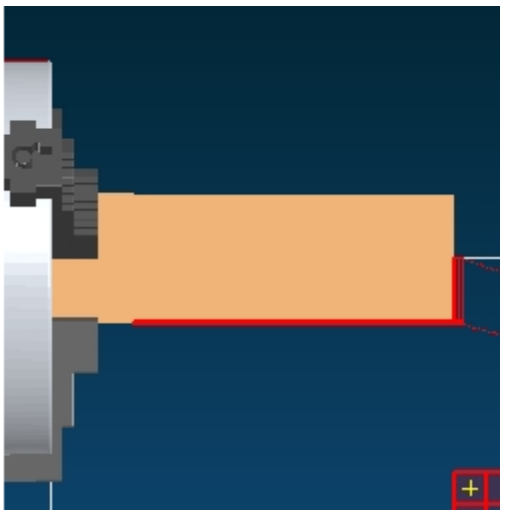
Procedure

1. Relese the Emergency button
2. Press Reference and X and Z
3. Press write protect off
4. Go to Parameter setting – 4 Position turret
5. Tools Management – select the External turning 35/55/90^0 tool – drag and drop to tool magzine– mount the tool
6. Workpiece setting – stock size – Ø80, 250 length
7. Workpiece setting – rapid position at center
8. Close the machine door
9. Offset setting – work – X00 – measure --- Z00 – measure

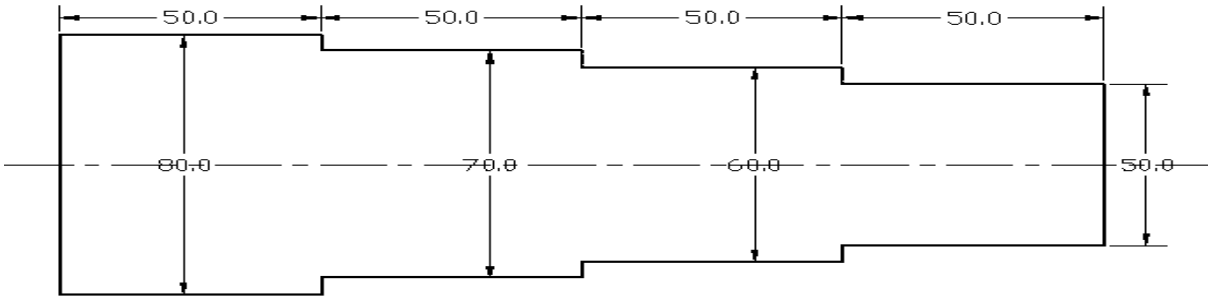
   (Do not use this step if X and Z axis home position is zero)
10. prog – edit – DIR – O0013 (Register your program No. start with alphabet 'O')

CNC Codes

Absolute Method	Incremental	Incremental with using canned cycle
O0012 G54 T0100 S400M03M08 G00X00Z00F2.5 G00X81 G01X0Z-1F0.25 G00X81 G01Z-5 G01X0 G00X81Z-5 G01X77.5 G01Z-205 G00X78.5Z-5 G01X75 G01Z-205 G00X76 G28U0W0 M09 M05 M30	O0012 G54 T0100 S400M03M08 G00X00Z00F2.5T0101 G00U81 G01W-2.5F0.25 G01U-81 G00U81 G01W-5 G01U-81 G00U81 G00W5 G01U-3.5 G01W-200 G00U1W205 G01U-3.5 G01W-205 G00U1 G28U0W0 M09 M05 M30	O0012 G54 T0100 S400M03M08 G00X00Z00F2.5T0101 G00U81 G94U-81W-2F0.25 W-4 W-5 G90U-3W-200F0.25 U-4 U-5 G28U0W0 M09 M05 M30



3. Write a part program to create the following profile with facing, OD Turning and Step Turning write the comments of the each line program .



Workpiece Information

Bar
Diameter: 80.000
Length: 250.000
Material:08F low-carbon steel

Tool Information

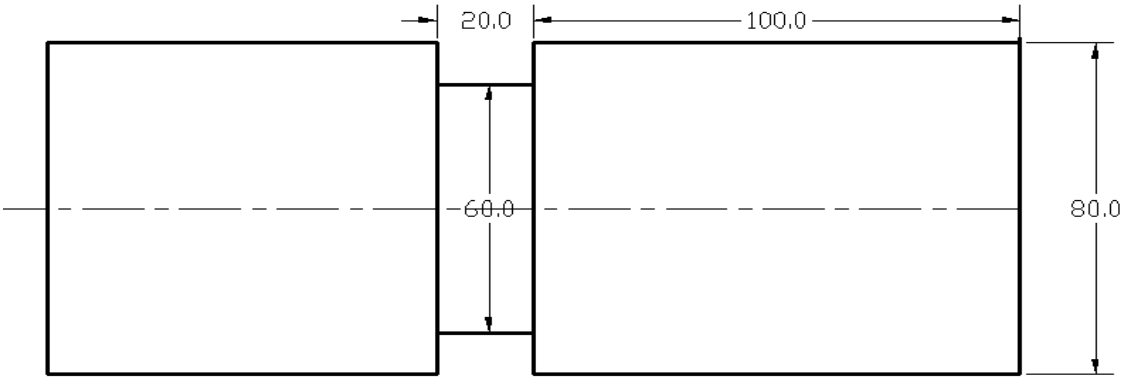
- Work offset at the center
- Tools Management – Select the External turning 35^0 tool.

CNC Codes

Absolute Method	Incremental	Incremental with using canned cycle
O0017 G54 T0100 S500M03M08T0101 G00X00Z00F0.25 G00X50 G01X50Z-50F0.26 G01X60 G01Z-100 G01X70 G01Z-160 G01X80 G28U0W0 M09 M05 M30	O0017 G54 T0100 S500M03M08T0101 G00X00Z00F0.25 G00U50 G01W-50F0.26 U10 W-50 U10 W-50 U10 G28U0W0 M09 M05 M30	O0017 G54 T0100 S500M03M08T0101 G00X00Z00F0.25 G00U80 G90U-2W-150F0.25 U-4 U-6 U-8 U-10 G90U-12W-100F0.25 U-14 U-16 U-18 U-20 G90U-22W-50F0.25 U-24 U-26 U-28 U-30 G28U0W0 M09 M05 M30

4. Write a part program to create the following profile with Step Turning (under cut)

write the comments of the each line program .



Workpiece Information

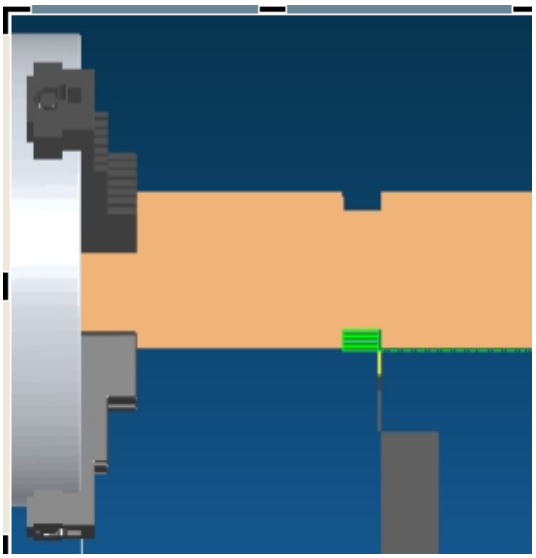
Bar
Diameter: 80.000
Length: 200.000
Material:08F low-carbon steel

Tool Information

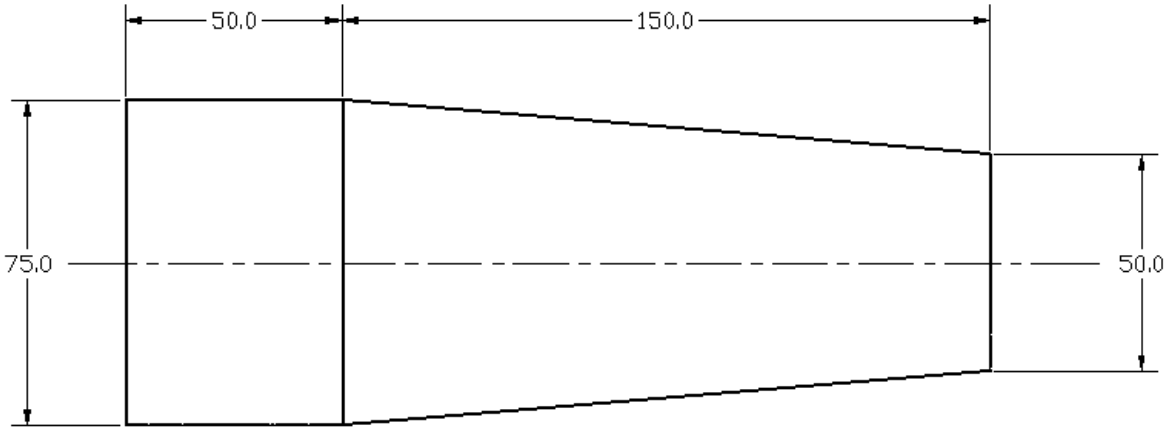
- Work offset at the center
- Tools Management – Select the Groveeing tool with thickness of 1mm and that edge also taken into consideration. (move 1mm and give 19mm cutting)

CNC Code

O0022
G56T0200S400M03M08
G00X00Z00F0.25
U81
W-100
W-1
G90U-2.5W-19F0.25
U-5
U-7.5
U-10
U-12.5
U-15
U-17.5
U-20
M09M05M30
%



5. Write a part program to create the following Facing OD Turning and Taper profile using G90 and G94 fixed cycle with facing, Taper Turning write the comments of the each line program ..



Work piece Information: Bar

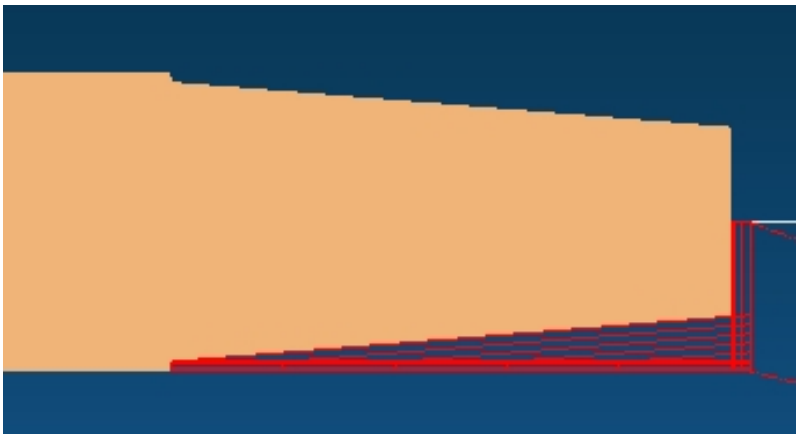
Diameter: 80.000 Length: 250.000 Material:08F low-carbon steel

Tool Information

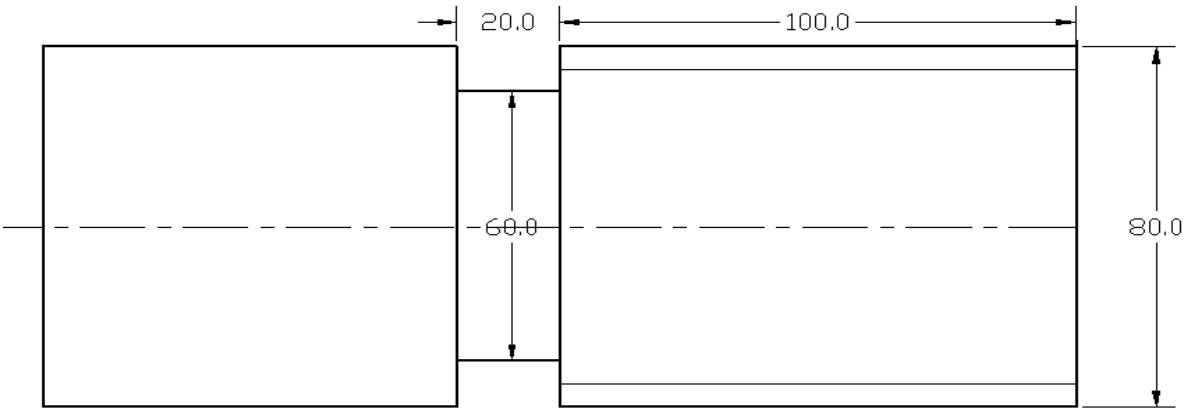
- Work offset at the center
- Tools Management – select the External turning 35^0 tool – drag and drop tool.

CNC Codes

Absolute Method	Incremental
O0017 G54 T0100 S500M03M08T0101 G00X00Z00F0.25 G00X80 G94X0Z-2F0.25 Z-4 Z-5 G90X78Z-155F0.25 X76 X75 Z-35R-2.5 Z-65R-5 Z-95R-7.5 Z-125R-10 Z-155R-12.5 G28U0W0 M09 M05 M30	O0017 G54 T0100 S500M03M08T0101 G00X00Z00F0.25 G00U80 G94U-80W-2F0.25 W-4 W-5 G90U-2W-155F0.25 U-4 U-5 W-35R-2.5 W-65R-5 W-95R-7.5 W-125R-10 W-155R-12.5 G28U0W0 M09 M05 M30



5. Write a part program to create the following Under Step Cutting and Thread Cutting profile using G90 and G92 fixed cycle write the comments of the each line program .



Work piece Information:

Bar
Diameter: 80.000
Length: 200.000
Material:08F low-carbon steel

Tool Information

- Work offset at the center
- Tools Management –
  First Select the Grooving tool with thickness of 1mm and that edge also taken into consideration. (move 1mm and give 19mm cutting)

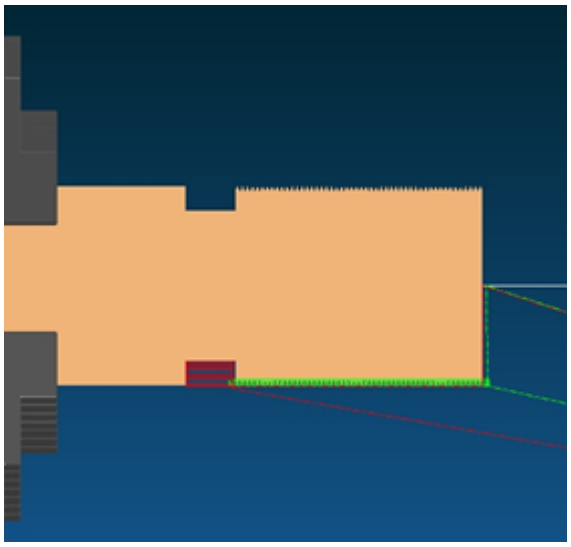
  Second select the Thread cutting tool Insert parameter Length 2.5 insert thickness and insert diameter 12 tool – drag and drop tool.

  Pitch = 0.061 x diameter

CNC Code

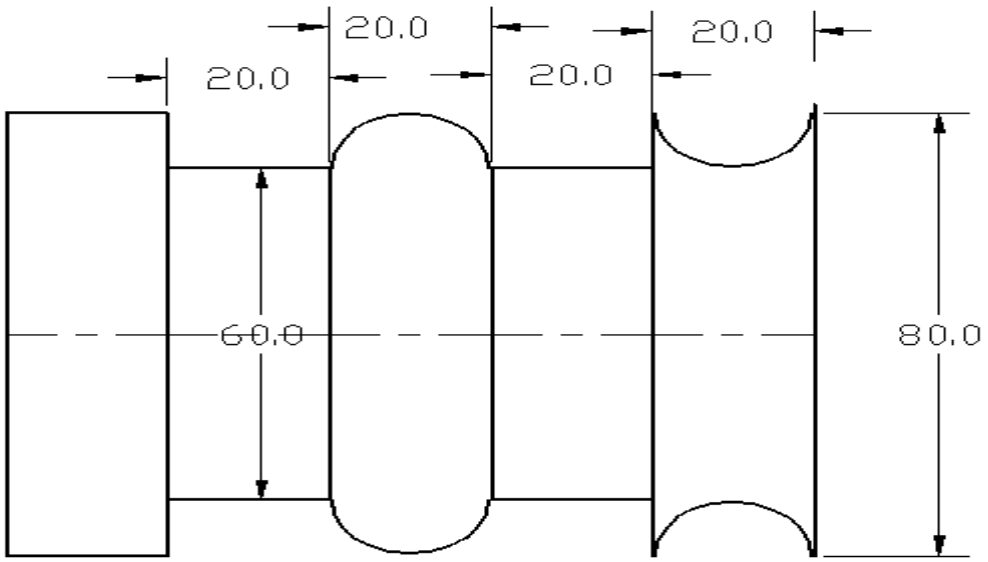
O0022
G54T0100S400M03M08
G00X00Z00F0.25
U81
W-100
W-1
G90U-2.5W-19F0.25
U-5
U-7.5
U-10
U-12.5
U-15
U-17.5
U-20
G00U2

```
G28U0W0
G55
T0200S400M03M08
G00X0Z0F0.25
G00U81W1
G92U-1W-105F0.25
U-3
U-5
G28U0W0
M09M05M30
%
```



6. Write a part program to create the following Under Step Cutting and Concave,

Convex Cutting profile using G90 and G02 circular cycle write the comments of the each line program ..



Work piece Information

Bar
Diameter: 80.000
Length: 200.000
Material:08F low-carbon steel

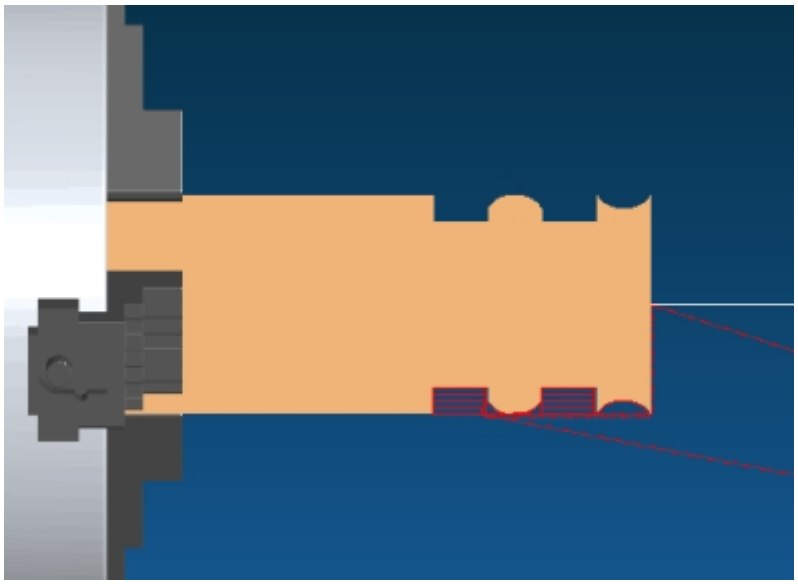
Tool Information

- Work offset at the center
- Tools Management – First Select the Grooving tool with thickness of 1mm and that edge also taken into consideration. (move 1mm and give 19mm cutting)

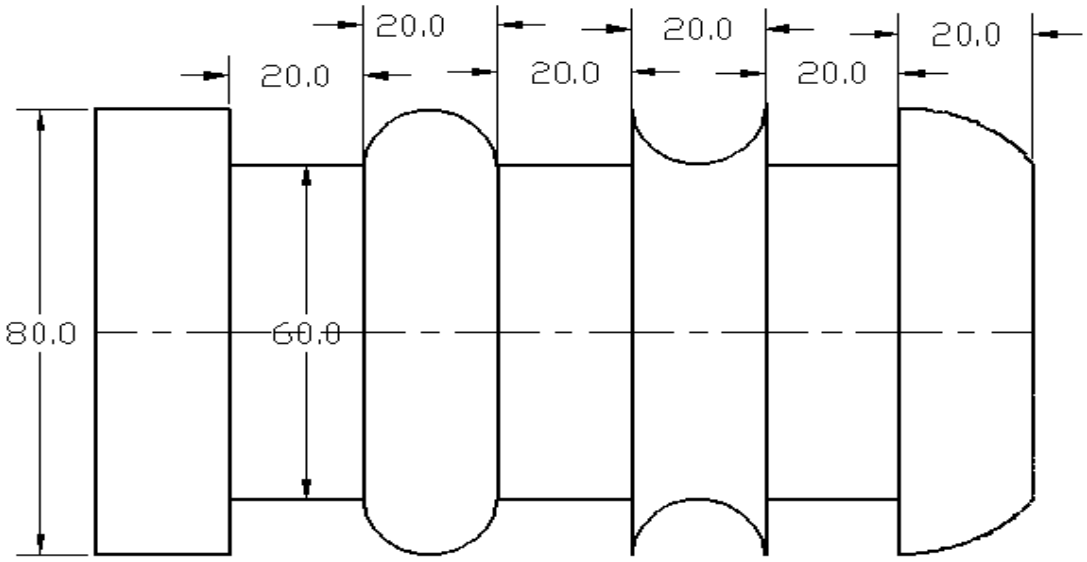
CNC Code

```
O0023
G54
T0100M03M08S400
G00X0Z0F025
G00U81
W-21
G90U-5W-19F0.25
U-10
U-15
U-20
G00W-40
G90U-5W-19F0.25
U-10
U-15
U-20
```

```
G00
U2
W60
U-2
G02U-10W-10R10
G02U10W-9R10
G00
W-20
U-11
G03U10W-10R10
G03U-10W-11R10
G00W-1U11
G28U0W0
M09M05M30
%
```



7. Write a part program to create the following Under Step Cutting and Concave ,Convex Cutting, using G90 and G02 and G03 circular cycle write the comments of the each line program .



Work piece Information

Bar

Diameter: 80.000

Length: 200.000

Material:08F low-carbon steel

Tool Information

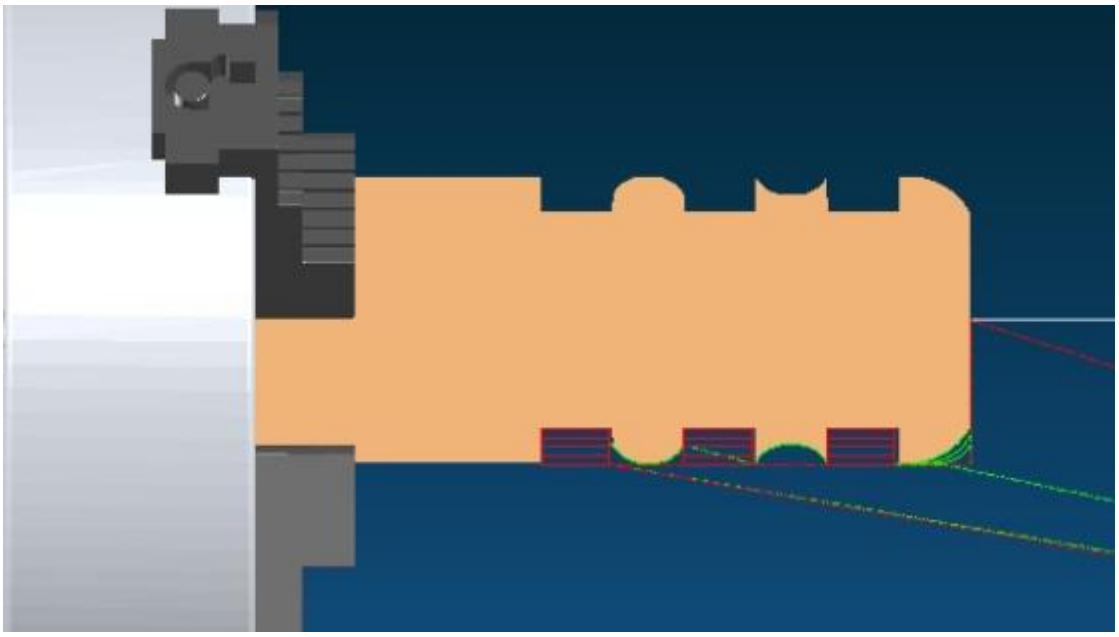
- Work offset at the center
- Tools Management

Select the Grooving tool with thickness of 1mm.for step turning and single point cutting tool 35° angle length 15mm, insert thickness 3mm for convex and concave

CNC Code

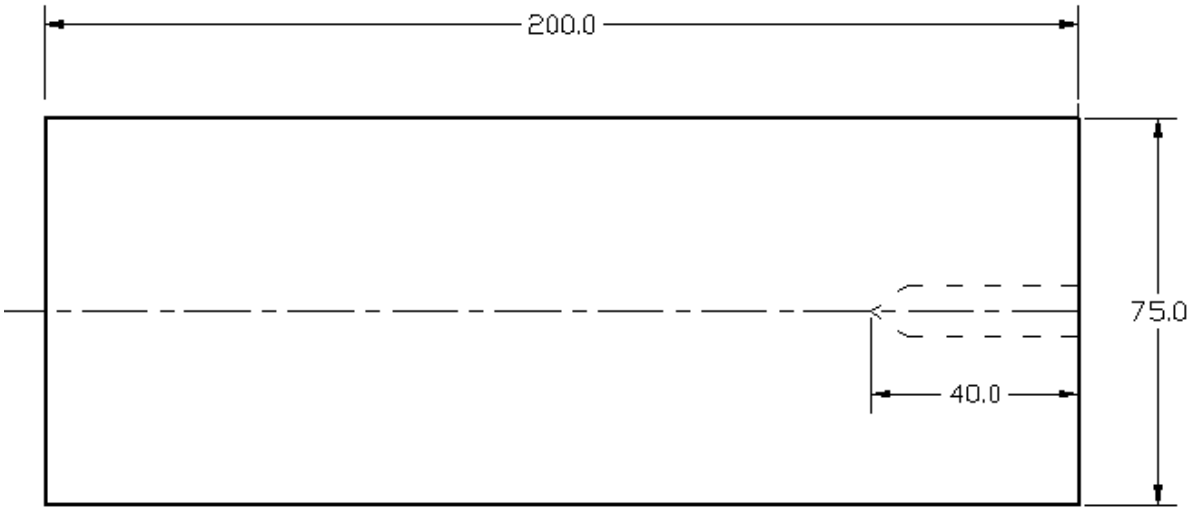
```
O0024
G54
T0100M03M08
X0Z0
G00U81
W-21
G90U-5W-19F0,25
U-10
U-15
U-20
```

```
G00W-40
G90U-5W-19F0.25
U-10
U-15
U-20
G00W-40
G90U-5W-19F0.25
U-10
U-15
U-20
G28U0W0
G55
T0200
G00X0Z-10
G02U-10W10R10
G00U11
W-15
U-1
G02U-15W15R15
G00U16
W-20
U-1
G02U-20W20R20
G00U21
W-40
U-1
G02U-10W-10R10
G02U10W-10R10
G00U1
W-20U-11
G03U10W-10R10
G03U-10W-10R10
G00W-1
U11
G28U0W0
M05M09M30
%
```



8. Write a part program to create the following peck drilling operation.

usingG74 and Q peck distance in microns.



Work piece Information

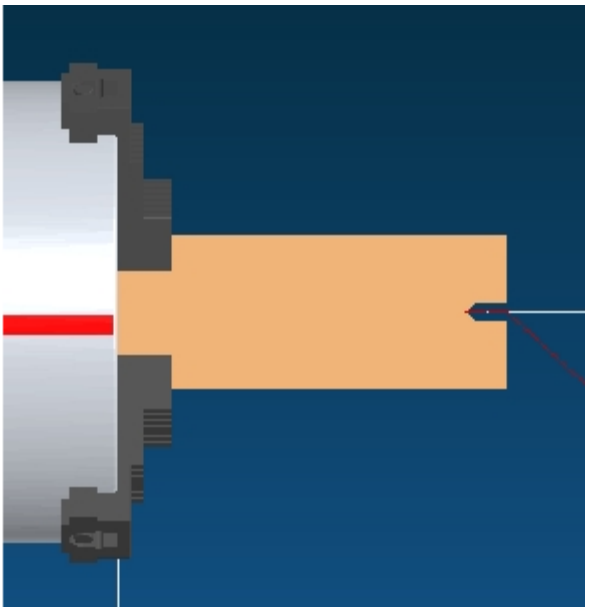
Bar
Diameter: 75
Length: 200
Material:08F low-carbon steel

Tool Information

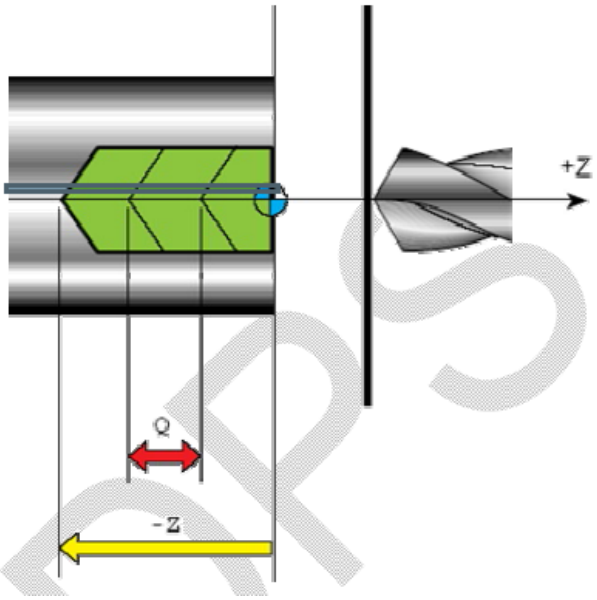
- Work offset at the center
- Tools Management – Select the drilling tool with diameter 10mm. Q is pecking distance in microns.

CNC Code

O0024
G54
T0100M03M08
X0Z0
G74R5
G74W-40Q10000F0.5
G28U0W0
M09M05M30
%

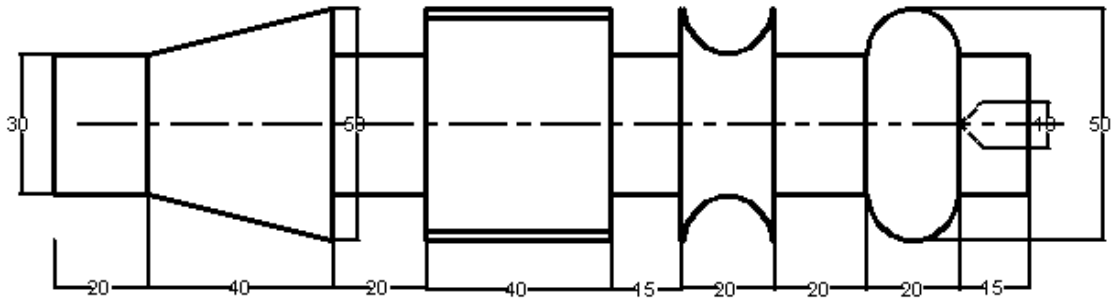


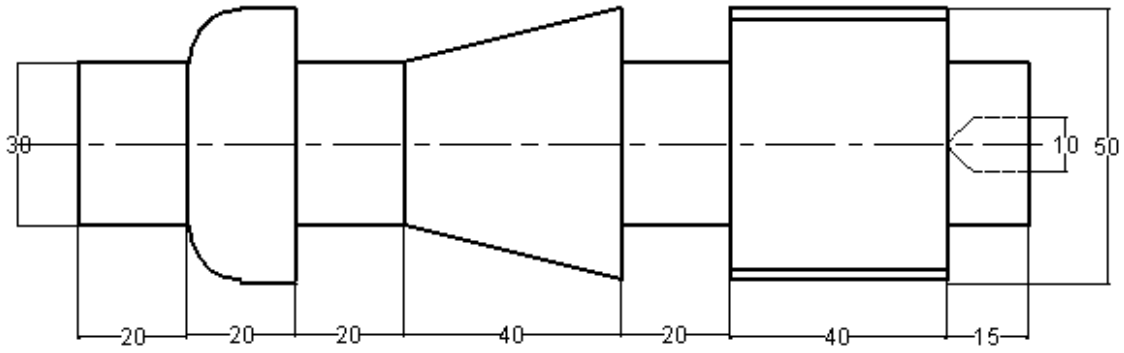
- Q is pecking distance in microns.



Exercise :

9.Using Swansoft write a part program to create the following profile with Facing, Turning, Step Turning, Drilling, Concave/Convex and Thread Cutting M50 x 3mm pitch, choose a work piece of size 55 diameter 250 length.





OPERATING PROCEDURE OF MASTER CAM

Step 1

The usual modification is that is done after initializing the master cam is the alteration of the available screen ie modification of grid size and as per the requirement of user screen, configure, selection grid. Select grid size (configure

→ grid) The dimensioning requirement is usually selected ie Metric or English etc.

Step 2

Once the screen and the dimensioning adjustments are done the basic required part geometry of the object is created by using the create command.

Create command: Rectangle or Circle or Line or Fillet.

Step 3

After defining the tool parameters and the machining operations parameter, the job setup is done. The job is usually defined in terms of length width and thickness (X, Y, Z). The selection of material for the job is done in jobsetup

Step 4

After the required part geometry is created, the tool path are defined on the selected geometry depending upon the milling operation.

Tool path → contour → save → chain → done

Tool path → pocket → save → area or chain or polygon → done

Once the tool path are defined, press done, tool parameter screen will appear where in the required tool diameter is entered or the tool can be selected from the tool manager.

Similarly the above step is carried out for all operations the contour parameter such as clearance, retract, feed, top of stock, depth aregiven.

Step 5

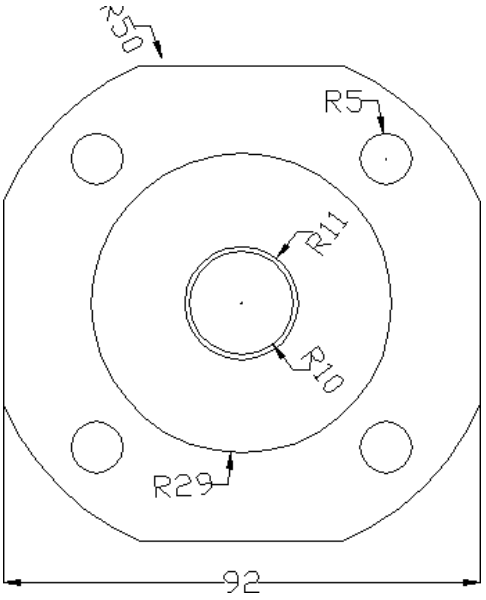
Completion of the above setup leads to operation manager where in all the operation are selected by select all, paths are regenerated by Regenerate path, and by Verify, machining operation is done. Before machining, configuration is verified where in use of job setup values are highlighted.

Step 6

Press post button to the get the NC program

Exercise 1

Aim : To machine the model as per thesketch.



Procedure:

Step 1

The usual modification is that is done after initializing the master cam is the alteration of the available screen ie modification of grid size and as per the requirement of user screen,

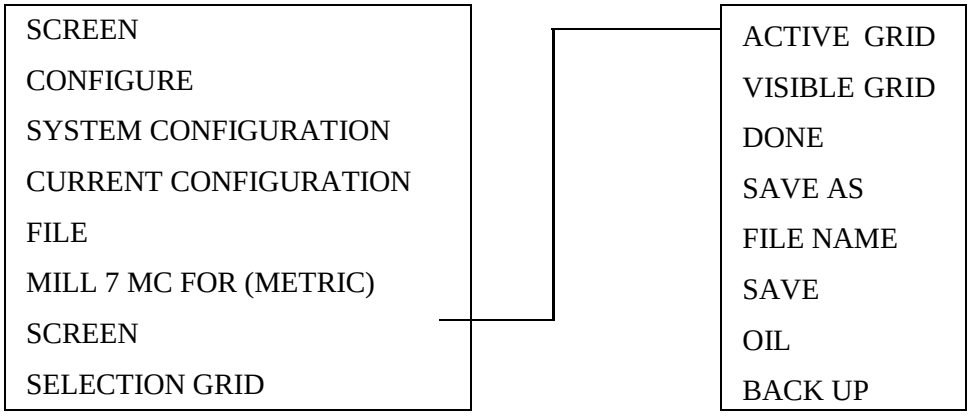
Choose-mainmenu → Screen → Configure → Current Configuration File

Select- Mill9.Mcfg(metric)

Choose-mainmenu → Screen → Nextmenu → Sel.Grid

Select-Active Grid, Visible Grid, Spacing x =1 and y =1, Grid Size= 110,

Origin x=0,y=0 and choose suitable for Grid.



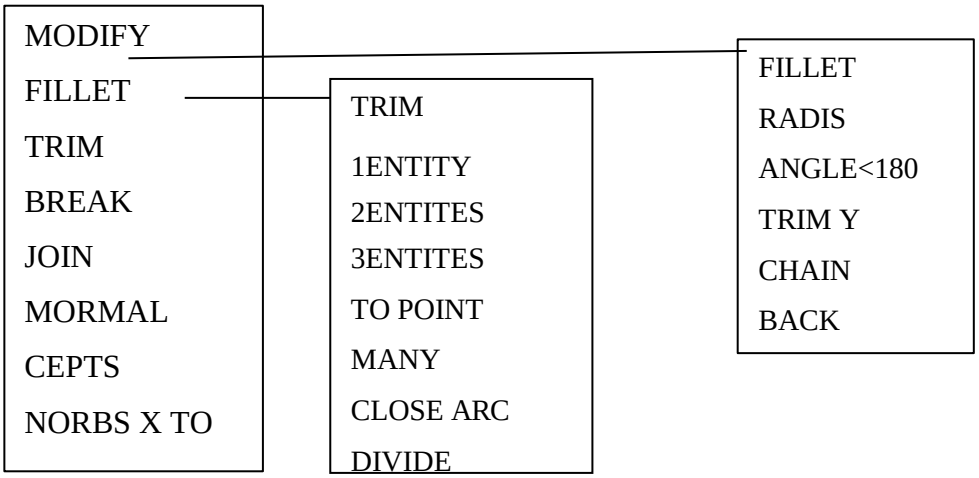
Step 2

1. To create inner and outercircles.
   - Choose-mainmenu → Create → Arc → circpt+dia.
   - Typediameter100 → Enter
   - Selectorigin
   - Press Esc. And reselectcircpt+dia
   - Enter diameter58
   - Selectorigin
   - Press Esc. And reselectcircpt+dia
   - Repeat same procedure for diameter 20 and22mm
   - xi. Press Esc. to exit circle function
   - Choose it screenmenu.

2. To create construction of rectangle /Square
   - Choose-mainmenu → Create → Rectangle → 1 point
   - Enter width 92 and height 92 (bcz. given figure is square) pressOK
   - Selectorigin
   - Press Esc to exit linefunction

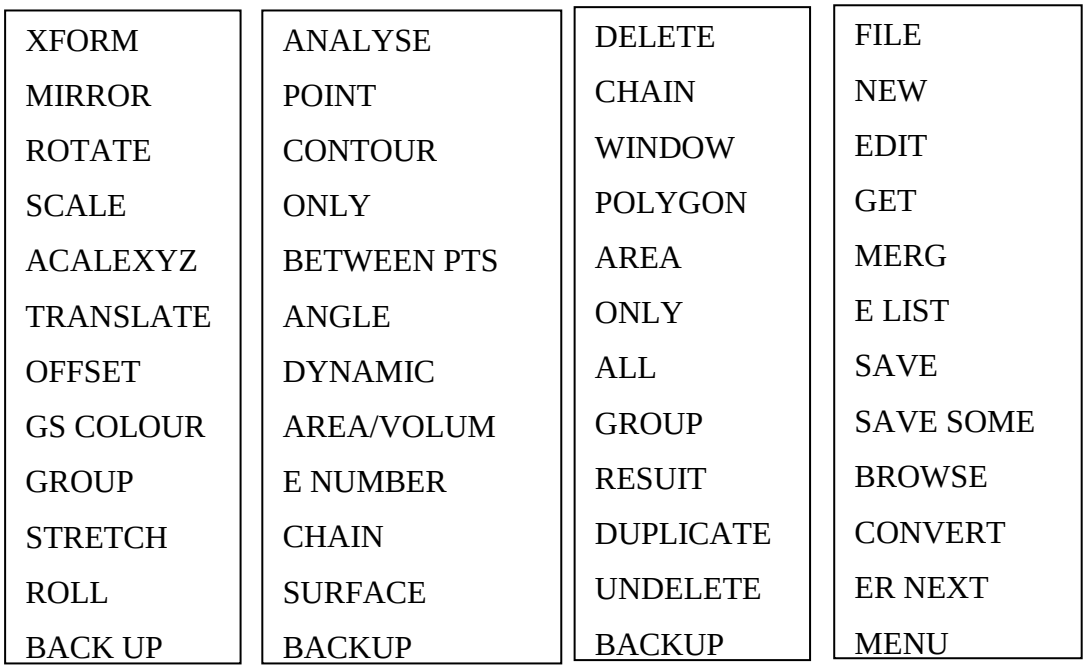
3. Trimming unwanted portion oflines

- Choose-mainmenu → Modify → Trim → 3 entities
- Select 1 entity as vertical line of square inside thecircle
  2 entity as Horizontal line of square inside the circle 3
  entity as arc of the circle inside thesquare.



4. Copy and rotate thearc

- Choose-mainmenu → Xform → Rotate
- Select anywhere onarc.
- Choosedone.
- Selectorigin.
- Select copy, enter the values. No of steps 3 and angle is90^0
- Choose OK in the dialogue box remaining slots forcreated.



5. Trimming unwanted portion of lines(Square)
   - Choose-mainmenu → Modify → Trim → 3entities
   - Select 1 Entity vertical line of square inside thearc
     2 Entity Horizontal line of square inside the arc
     3 Entity arc inside thesquare.

6. To create constructionlines
   - Choose-mainmenu → Create → Line → Polar
   - Enter the first co-ordinateorigin
   - Enter angle in degree45^0
   - Enter the line length45
   - Press Esc to exit linefunction

7. To create inner 10 mm diametercircles
   - Choose- mainmenu → Create → Arc → circpt +dia.
   - Type diameter10 → Enter
   - Select end point of inclinedline.
   - Press Esc. And reselectcircpt+dia

8. Copy and rotate theCircle
   - Choose-mainmenu → Xform → Rotate
   - Select anywhere on 10 mm diameter circle
   - Choosedone.
   - Selectorigin.
   - Select copy, enter the values. No of steps 3 and angle is90^0
   - Choose OK in the dialogue box remaining 3 circles forcreated.

Now your drawing is ready for operations

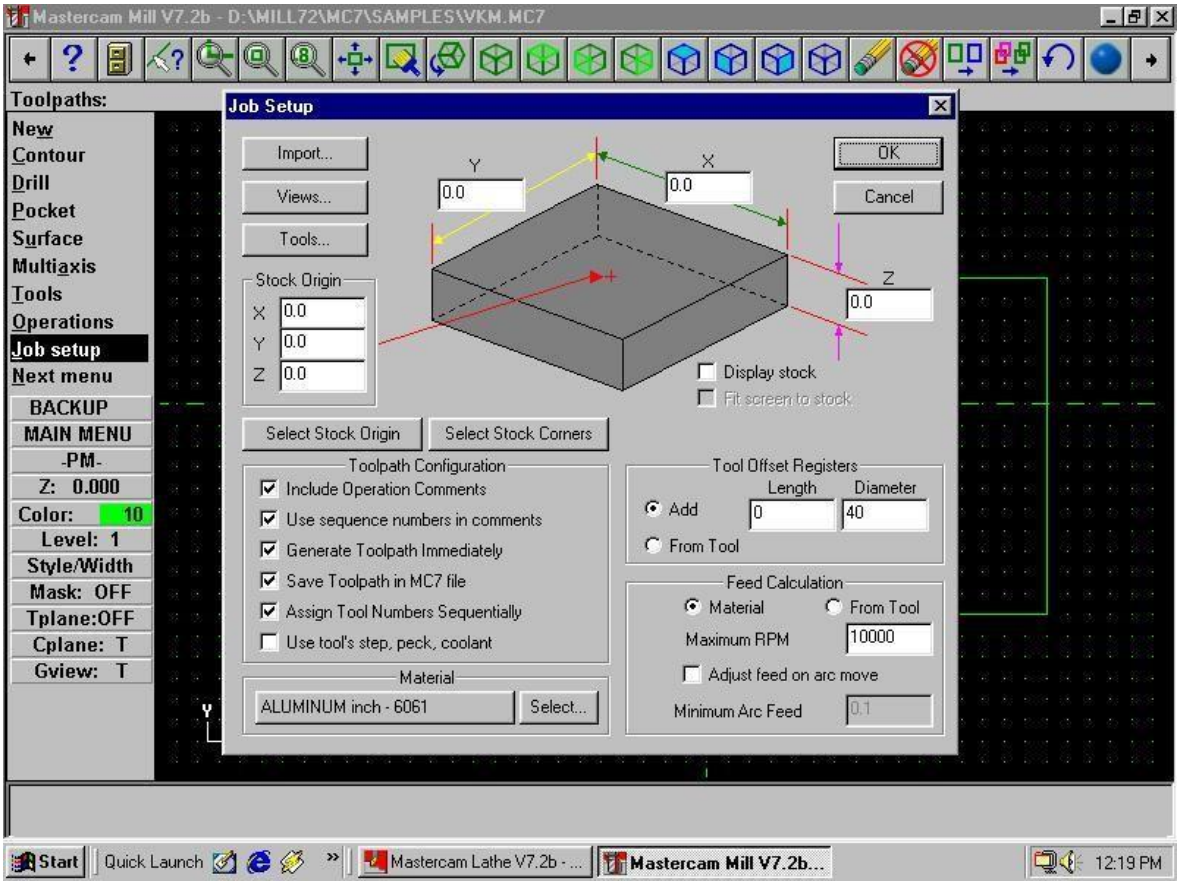
The following operations to be conduct Using the geometry

- **Facing**
- **Contour**
- **Pocketing**
- **Drilling**

Step 3

1. Make job setup for givengeometry
   - Choose-mainmenu → Toolpath → Job setup
   - Enter the X =110 Y=110 and Z=55 mmrespectively
   - Select Display Stack and Fit toScreen.
   - Outside the drawing doted boundary red line isdisplayed

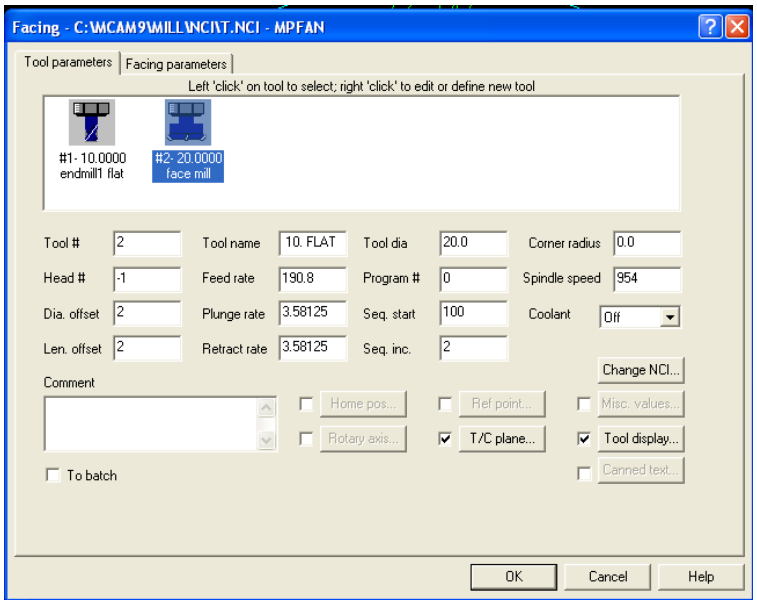
2. Another option for jobsetup
   - Choose-mainmenu → Toolpath → Jobsetup
   - Select stock origin (0,0)
   - select the stockcorners.
   - Enter the value of Z = 55 (+ value).
   - Select the display stock checkbox.
   - Select fit stock checkbox.
   - Choose OK the stock should be enclosed by red dottedline.

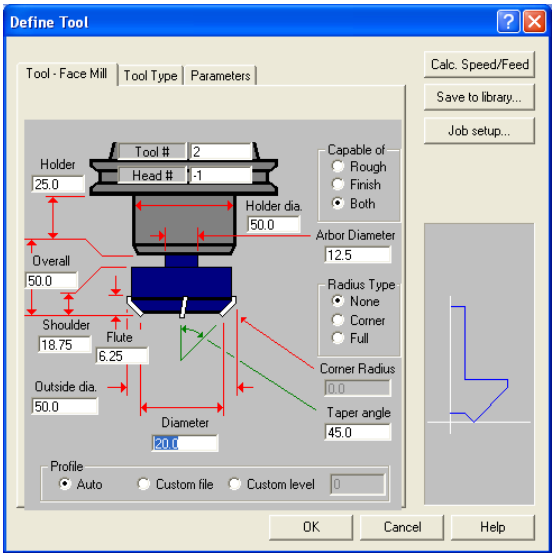


Step 4

1. Create tool path forFacing

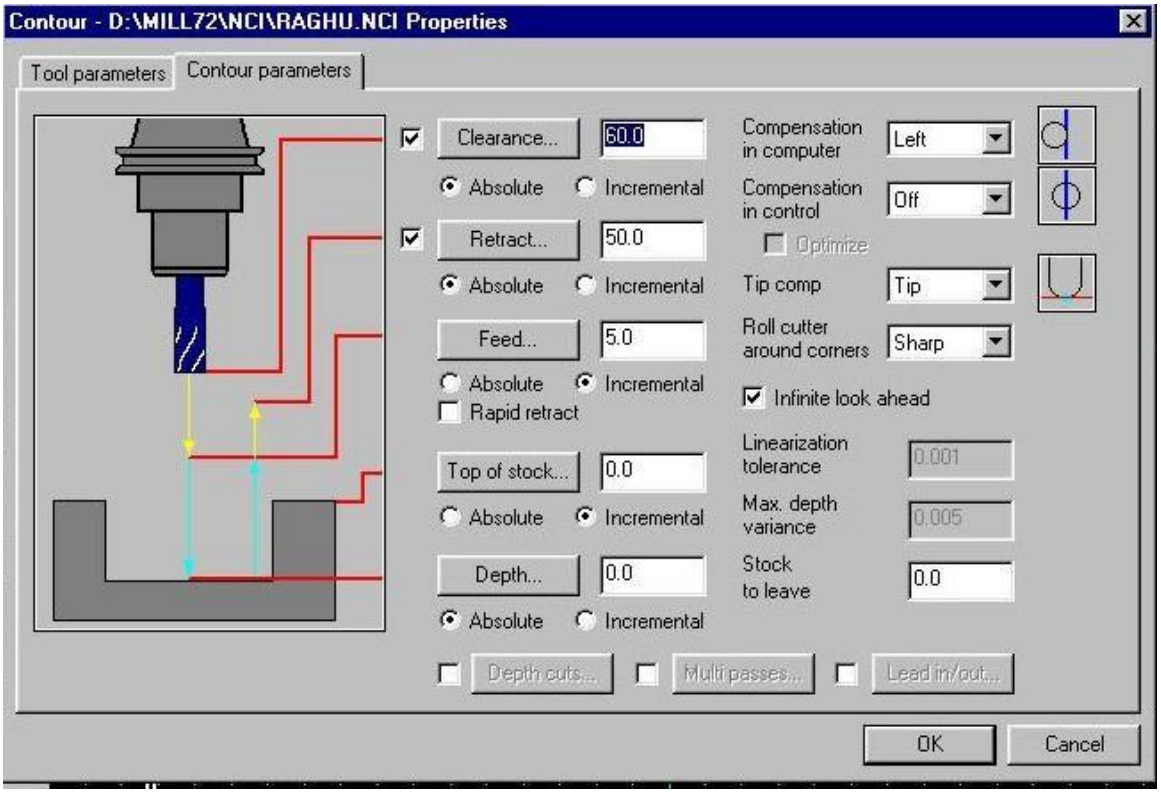
- Choose-mainmenu → Toolpath → Face
- Select start point for the chain at periphery of thefigure.
- Choosedone.
- Right click in the tool display area and select a 10 mm flat endmill
- from the toollibrary.
- Right click on the tool display , go to tool type select face milltool,
- Select the facing parameters. Give Depth of cut -2mm (-negative)
- Select depth Cuts, Give Rough Cut 3, Finishing cut 1No. Finish Step0.5
- Choose OK twice in the tool path should be showingfigure.





1. Create tool path forContour

- Choose-mainmenu → Toolpath → Contour
- Select start point for the chain at position1.
- Choosedone.
- Right click in the tool display area and select a 10 mm flat end mill from the toollibrary.
- Select the contour parameters , Give Depth of cut -53mm (- negative), Use Multi passes, Depth Cuts and Lead in/outoptions.
- Choose OK twice in the tool path should be showingfigure.



3. Create tool path for pocket 1

- Choose-mainmenu → Toolpath → Pocket
- Select the 58 mm diameter circle.
- Choosedone.
- Right click in tool display area and select 8mm flat end mill from toollibrary.
- Enter pocketing parameters and note that the depth shouldbe negative say-10.
- Choose OK the tool path should look like the picturepattern.

4 Create tool path for pocket 2

- Choose-mainmenu → Toolpath → Pocket
- Select first 22 mm diametercircle.
- Choosedone.
- Right click in tool display area and select 5mm flat end mill from toollibrary.
- Enter pocketing parameters and note that the depth should be negative-12.
- Choose OK the tool path should look like the picturepattern.

6. Create tool path for pocket 3

- Choose-mainmenu → Toolpath → Pocket
- Select first 20 mm diametercircle.
- Choosedone.
- Right click in tool display area and select 3mm flat end mill from toollibrary.
- Enter pocketing parameters and note that the depth shouldbe negative say-32.
- Choose OK the tool path should look like the picturepattern.

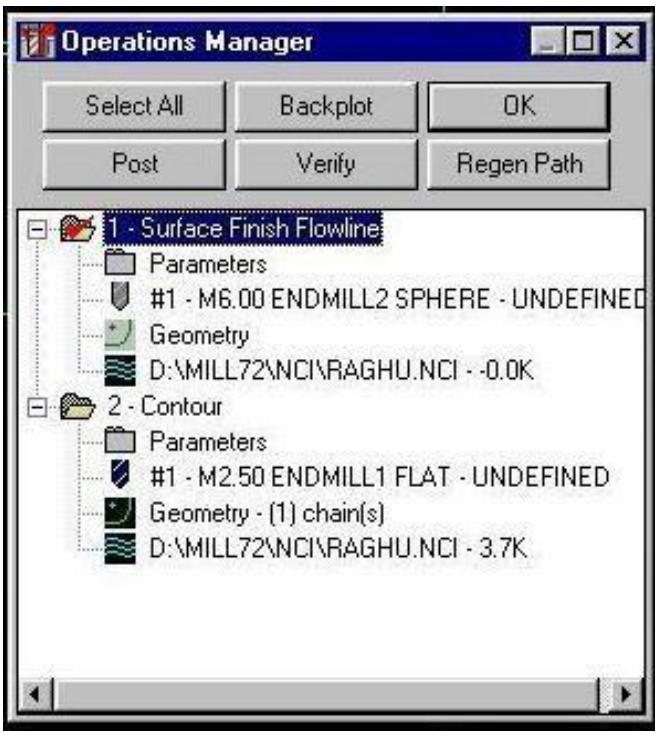
6. Create tool path forDrilling

- Choose-mainmenu → Toolpath → Drill
- Select Entities Choose 10mm diameter circles one byone.
- Choose done. Tool path is generated, Choose once againdone
- Right click in the tool display area and select a 10 mm drill mill from the tool library.
- Select the drill parameters and note that the depth should be negative say-60.
- Choose OK twice in the tool path should be showingfigure.

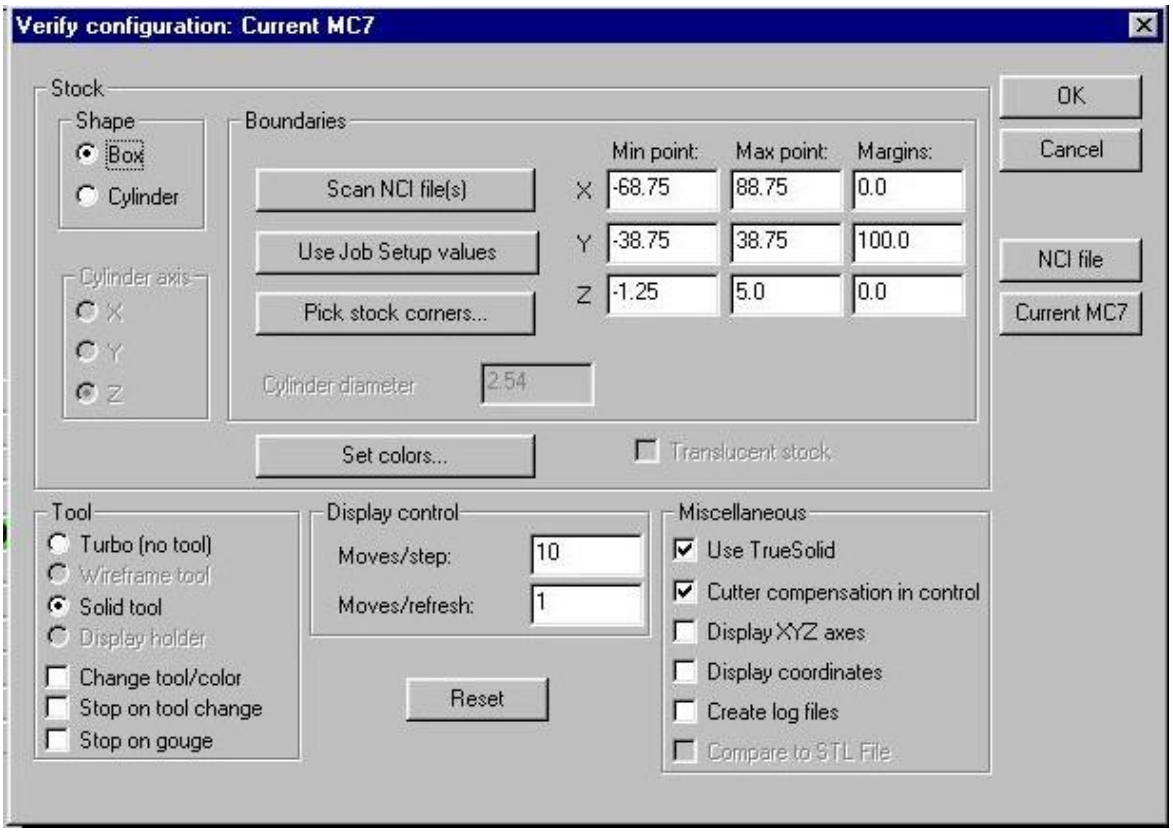
Step 5

Tool path Generator

- i. Choose - mainmenu → Operation
- A dialogue box appears select all regeneratepath
- When the tool path generation completes the dialogue boxdisplay
- choose verify, a deluge box is displayed select run (machine) item.

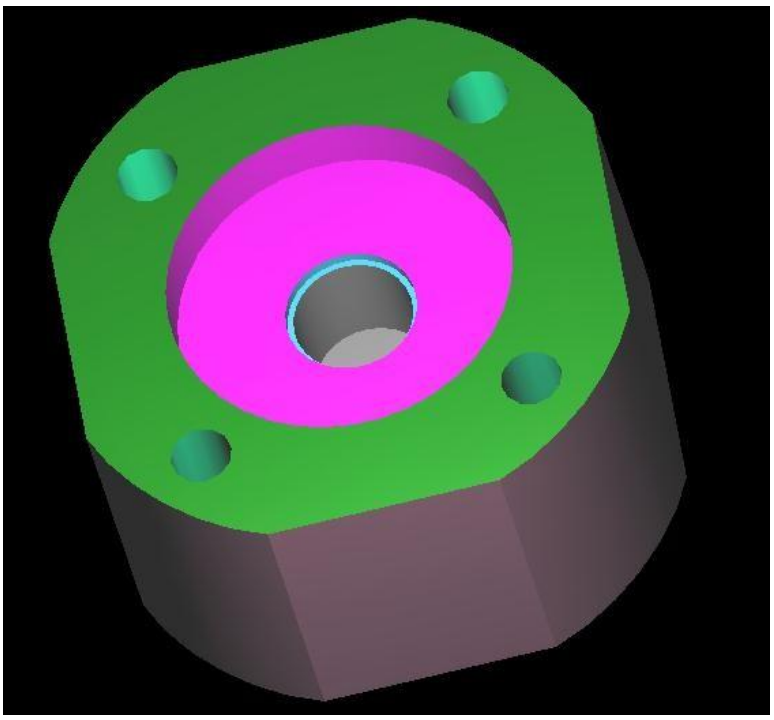


Operation management of Tool Path



Configuration of Tool path

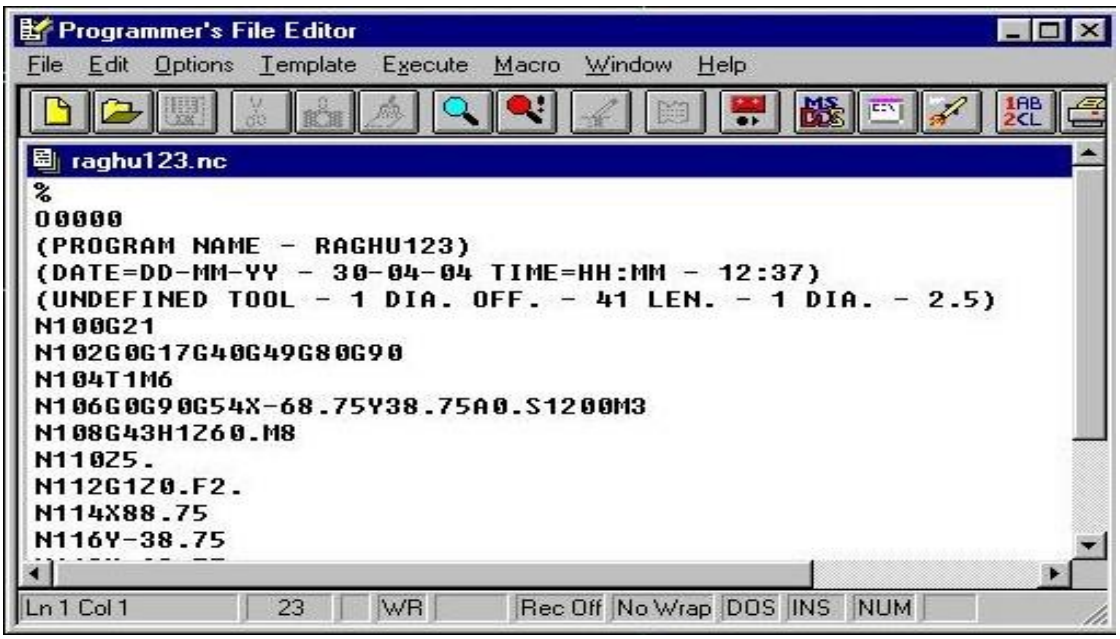
If your model is square/Rectangle go to shape, select box , suppose your model is Cylindrical go to shape, select cylinder



Step 6

Post processor NC Program

- Choose-mainmenu → Operation
- A dialogue box appears select all regenerate path, choosepost
- Show path (desktop) for saving post processor Notepadfile.



Result: The required geometry is created.

Post processor program Exercise 1

```
%
O0000
(PROGRAM NAME - 1)
(DATE=DD-MM-YY - 27-11-07 TIME=HH:MM - 10:05)
(4. FLAT ENDMILL TOOL - 2 DIA. OFF. - 42 LEN.- 2 DIA. - 4.)
N100G21
N102G0G17G40G49G80G90
/N104G91G28Z0.
/N106G28X0.Y0.
/N108G92X0.Y0.Z0.
N110T2M6
N112G0G90X-32.249Y62.A0.S50M5
N114G43H2Z20.M8
N116Z5.
N118G1Z-6.667F2.2
N120Y57.
N122G3X-22.249Y47.R10.
N124G1X22.249
N126G2X47.Y22.249R52.
N128G1Y-22.249
N130G2X22.249Y-
47.R52.N132G1X-22.249
N134G2X-47.Y-22.249R52.
N136G1Y22.249
N138G2X-22.249Y47.R52.
N140G3X-
16.527Y56.038R10.N142X-
17.489Y60.317R10.
N144G1X-19.628Y64.836
N146G0Z10.
N148X-32.249Y62.
N150Z5.
N152G1Z-15.333
N154Y57.
N156G3X-22.249Y47.R10.
N158G1X22.249
N160G2X47.Y22.249R52.
N162G1Y-22.249
N164G2X22.249Y-
47.R52.N166G1X-22.249
N168G2X-47.Y-22.249R52.
N170G1Y22.249
N172G2X-22.249Y47.R52.
N174G3X-
16.527Y56.038R10.N176X-
17.489Y60.317R10.
N178G1X-19.628Y64.836
N180G0Z10.
N190G3X-22.249Y47.R10.
N192G1X22.249
N194G2X47.Y22.249R52.
N196G1Y-22.249
N198G2X22.249Y-
47.R52.N200G1X-22.249 N202G2X-
47.Y-22.249R52.
```

N204G1Y22.249
N206G2X-
22.249Y47.R52.
N208G3X-
16.527Y56.038R10.
N210X-
17.489Y60.317R10.
N212G1X-
19.628Y64.836
N214G0Z10.
N216X-
32.249Y6
2.
N218Z5.
N220G
1Z-
32.667
N222Y
57.
N224G3X-
22.249Y47.R10.
N226G1X22.249
N228G2X47.Y22.
249R52.N230G1Y
-22.249
N232G2X22.249Y
-
47.R52.N234G1X-
22.249 N236G2X-
47.Y-22.249R52.
N238G1Y22.249
N240G2X-
22.249Y47.R52.
N242G3X-
16.527Y56.038R10.
N244X-
17.489Y60.317R10.
N246G1X-
19.628Y64.836
N248G0Z10.
N250X-
32.249Y6
2.
N252Z5.
N254G
1Z-
41.333
N256Y
57.
N258G3X-
22.249Y47.R10.
N260G1X22.249
N262G2X47.Y22.
249R52.N264G1Y
-22.249
N266G2X22.249Y
-
47.R52.N268G1X-22.249
N270G2X-47.Y-22.249R52.
N272G1Y22.249
N274G2X-22.249Y47.R52.
N276G3X-
16.527Y56.038R10.N278X-
17.489Y60.317R10.
N280G1X-19.628Y64.836
N282G0Z10.
N284X-32.249Y62.
N286Z5.
N288G1Z-50.
N290Y57.
N292G3X-22.249Y47.R10.
N294G1X22.249
N296G2X47.Y22.249R52.
N298G1Y-22.249

N300G2X22.249Y-47.R52.
N302G1X-22.249
N304G2X-47.Y-22.249R52.
N306G1Y22.249
N308G2X-22.249Y47.R52.
N310G3X-
16.527Y56.038R10.N312X-
17.489Y60.317R10.
N314G1X-19.628Y64.836
N316G0Z20.
N318M5
N320G91G28Z0.M9
N322M01
(6. FLAT ENDMILL TOOL - 1 DIA. OFF. -
41 LEN.- 1 DIA. - 6.)
N324T1M6
N326G0G90X-15.384Y-20.65A0.S50M5
N328G43H1Z20.M8
N330Z5.
N332G1Z-5.333F1.5
N334G2X-25.313Y0.R26.438
N336X-1.125Y24.188R24.188
N338X23.063Y0.R24.188
N340X1.125Y-21.938R21.938
N342X-20.813Y0.R21.938
N344X-1.125Y19.688R19.688
N346X18.563Y0.R19.688
N348X1.125Y-17.438R17.438
N350X-16.313Y0.R17.438
N352X-1.125Y15.188R15.188
N354X14.063Y0.R15.188
N356X1.125Y-12.938R12.938
N358X-11.813Y0.R12.938
N360X-1.125Y10.688R10.688
N362X9.563Y0.R10.688
N364X1.125Y-8.438R8.438
N366X-7.313Y0.R8.438
N368X-1.125Y6.188R6.188
N370X5.063Y0.R6.188
N372X1.125Y-3.938R3.938
N374X-2.813Y0.R3.938
N376X-1.125Y1.688R1.688
N378X.563Y0.R1.688
N380G0Z10.
N382X26.
N384Z5.
N386G1Z-5.333
N388G3X0.Y26.R26.
N390X-
26.Y0.R26.N392X0.
Y-26.R26.
N394X26.Y0.R26.
N396G0Z10.
N398X-15.384Y-20.65
N400Z5.
N402G1Z-12.667
N404G2X-25.313Y0.R26.438

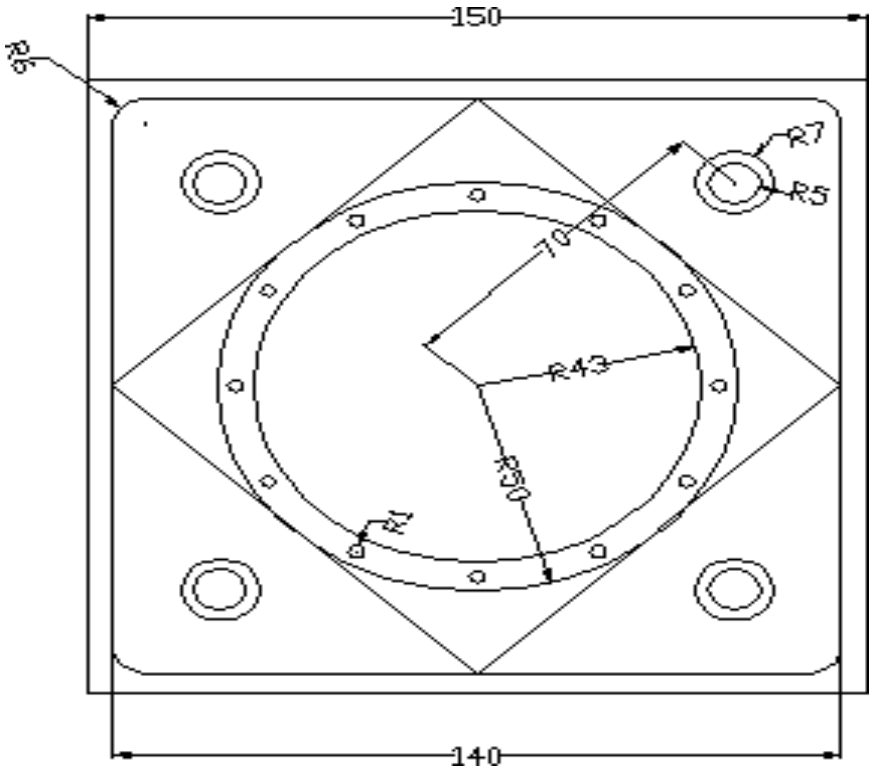
N406X-
1.125Y24.188R24.188
N408X23.063Y0.R24.188
N410X1.125Y-
21.938R21.938 N412X-
20.813Y0.R21.938 N414X-
1.125Y19.688R19.688
N416X18.563Y0.R19.688
N418X1.125Y-
17.438R17.438 N420X-
16.313Y0.R17.438 N422X-
1.125Y15.188R15.188
N424X14.063Y0.R15.188
N426X1.125Y-
12.938R12.938 N428X-
11.813Y0.R12.938 N430X-
1.125Y10.688R10.688
N432X9.563Y0.R10.688
N434X1.125Y-
8.438R8.438 N436X-
7.313Y0.R8.438 N438X-
1.125Y6.188R6.188
N440X5.063Y0.R6.188
N442X1.125Y-
3.938R3.938 N444X-
2.813Y0.R3.938 N446X-
1.125Y1.688R1.688
N448X.563Y0.R1.688
N450G0Z10.
N452X26.
N454Z5.
N456G1Z-
12.667
N458G3X0.Y26.R
26. N460X-
26.Y0.R26.N462X
0.Y-26.R26.
N464X26.Y0.R26.
N466G0Z10.
N468X-15.384Y-
20.65 N470Z5.
N472G1Z-20.
N474G2X-
25.313Y0.R26.438 N476X-
1.125Y24.188R24.188
N478X23.063Y0.R24.188
N480X1.125Y-
21.938R21.938 N482X-
20.813Y0.R21.938 N484X-
1.125Y19.688R19.688
N486X18.563Y0.R19.688
N488X1.125Y-
17.438R17.438 N490X-
16.313Y0.R17.438 N492X-
1.125Y15.188R15.188
N494X14.063Y0.R15.188
N496X1.125Y-12.938R12.938
N498X-11.813Y0.R12.938
N500X-1.125Y10.688R10.688
N502X9.563Y0.R10.688
N504X1.125Y-8.438R8.438
N506X-7.313Y0.R8.438
N508X-1.125Y6.188R6.188
N510X5.063Y0.R6.188
N512X1.125Y-3.938R3.938
N514X-2.813Y0.R3.938

N516X-1.125Y1.688R1.688
N518X.563Y0.R1.688
N520G0Z10.
N522X26.
N524Z5.
N526G1Z-20.
N528G3X0.Y26.R26.
N530X-
26.Y0.R26.N532X0.
Y-26.R26.
N534X26.Y0.R26.
N536G0Z10.
N538Z20.
N540X-6.047Y9.487
N542Z5.
N544G1Z-6.
N546G2X-1.125Y10.688R10.688
N548X9.563Y0.R10.688
N550X1.125Y-8.438R8.438
N552X-7.313Y0.R8.438
N554X-1.125Y6.188R6.188
N556X5.063Y0.R6.188
N558X1.125Y-3.938R3.938
N560X-2.813Y0.R3.938
N562X-1.125Y1.688R1.688
N564X.563Y0.R1.688
N566G0Z10.
N568X11.5
N570Z5.
N572G1Z-
6.N574G3X0.Y11.5R11
.5 N576X-
11.5Y0.R11.5
N578X0.Y-11.5R11.5
N580X11.5Y0.R11.5
N582G0Z10.
N584X-6.047Y9.487
N586Z5.
N588G1Z-14.
N590G2X-1.125Y10.688R10.688
N592X9.563Y0.R10.688
N594X1.125Y-8.438R8.438
N596X-7.313Y0.R8.438
N598X-1.125Y6.188R6.188
N600X5.063Y0.R6.188
N602X1.125Y-3.938R3.938
N604X-2.813Y0.R3.938
N606X-1.125Y1.688R1.688
N608X.563Y0.R1.688
N610G0Z10.
N612X11.5
N614Z5.
N616G1Z-14.
N618G3X0.Y11.5R11.5
N620X-11.5Y0.R11.5 N622X0.Y-
11.5R11.5 N624X11.5Y0.R11.5

N626G0Z10.
N628X
-
6.047Y
9.487
N630Z
5.
N632G1Z-22.
N634G2X-
1.125Y10.688R10.6
88
N636X9.563Y0.R1
0.688
N638X1.125Y-
8.438R8.438
N640X-
7.313Y0.R8.438
N642X-
1.125Y6.188R
6.188
N644X5.063
Y0.R6.188
N646X1.125
Y-
3.938R3.938
N648X-
2.813Y0.R3.9
38 N650X-
1.125Y1.688R
1.688
N652X.563Y
0.R1.688
N654G0Z10.6
56X11.5
N658Z5.
N660G1-
22.N662G
3X0.Y11.
5R11.5
N664X-
11.5Y0.R
11.5
N666X0.
Y11.5R1.
5
N668X11.
5Y0.R115
N670G0Z
10.
N672X
-
6.047Y
9.487
N6745.
N676G1Z-30.
N678G2X-
1.125Y10.688R10.688
N680X9.563Y0.R10.688
N682X1.125Y-8.438R8.438
N684X-7.313Y0.R8.438
N686X-1.125Y6.188R6.188
N688X5.063Y0.R6.188
N690X1.125Y-3.938R3.938
N692X-2.813Y0.R3.938
N694X-1.125Y1.688R1.688
N696X.563Y0.R1.688
N698G0Z10.
N700X11.5
N702Z5.
N704G1Z-
30.N706G3X0.Y11.5R1
1.5 N708X-
11.5Y0.R11.5
N710X0.Y-11.5R11.5
N712X11.5Y0.R11.5
N714G0Z10.
N716Z20.
N718G98G81X28.284Y28.284Z-50.R5.F1.5
N720X-28.284
N722Y-28.284
N724X28.284
N726G80
N728M5
N730G91G0G28Z0.M9
N732G90
N734M30

Exercise 2

Aim : To machine the model as per thesketch.



Procedure:

Step 1

The usual modification is that is done after initializing the master cam is the alteration of the available screen ie modification of grid size and as per the requirement of user screen,

Choose-mainmenu → Screen → Configure → Current Configuration File

Select- Mill9.Mcfg(metric)

Choose-mainmenu → Screen → Nextmenu → Sel.Grid

Select-Active Grid, Visible Grid, Spacing x =1 and y =1, Grid Size= 110,

Origin x=0,y=0 and choose suitable for Grid.

Step 2

1. To create inner and outercircles.
   - Choose-mainmenu → Create → Arc → circpt +dia.
   - Typediameter100 → Enter
   - Selectorigin
   - Press Esc. And reselectcircpt+dia
   - Enter diameter85
   - Selectorigin
   - Press Esc. And reselectcircpt+dia
   - Repeat same procedure for diameter 93mm
   - Selectorigin
   - Press Esc. to exit circle function Press Esc. And reselectcircpt+dia
   - Repeat same procedure for pitch circle diameter 3mm
   - Selectquadrant
   - Press Esc. to exit circlefunction
   - Choose it screenmenu.

2. Copy and rotate the 3 mmcircle
   - Choose-mainmenu → Xform → Rotate
   - Select anywhere on 3mmcircle
   - Choosedone.
   - Selectorigin.
   - Select copy, enter the values. No of steps 11 and angle is30^0
   - Choose OK in the dialogue box remaining slots forcreated.

3. To create construction of rectangle /Square
   - Choose-mainmenu → Create → Rectangle → 1 point
   - Enter width 150 and height 150 (bcz. given figure is square) pressOK
   - Selectorigin,
   - Press Esc.
   - Enter width 140 and height 140 (bcz. given figure is square) pressOK
   - Selectorigin
   - Press Esc to exit Rectangle function.

4. To create construction oflines
   - Choose-mainmenu → Create → Line → Endpoint
   - Enter the first co-ordinate Midpoint of horizontal line of 140mmSquare
   - Enter the Second co-ordinate Midpoint of Vertical line of 140mmSquare
   - Enter the same method for other threesides
   - Press Esc to exit linefunction

5. To create construction ofline
   - Choose-mainmenu → Create → Line → Polar
   - Enter the first co-ordinateorigin
   - Enter angle in degree45^0
   - Enter the line length70
   - Press Esc to exit linefunction

6. To create inner 10 mm and 15 mm diametercircles
   - Choose-mainmenu → Create → Arc → circpt+dia.
   - Typediameter10 → Enter
   - Select end point of inclinedline.
   - Press Esc. And reselectcircpt+dia
   - Choose-mainmenu → Xform → Offset
   - Choose operation = copy, No. of steps =1 & Offset distance= 2.5mm
   - Select 10 mm diacircle
   - Click outside the circle (Direction Inside or Outside)
   - ThenoK

7. Copy and rotate theCircle
   - Choose-mainmenu → Xform → Rotate
   - Select anywhere on 10 & 15 mm diametercircle
   - Choose done. Selectorigin.
   - Select copy, enter the values. No of steps 3 and angle is90^0
   - Choose OK in the dialogue box remaining 3 circles forcreated.

Now your drawing is ready for operations

The following operations to be conduct Using the geometry

Facing, Contours, Pocketing and Drilling

Step 3

1. Make job setup for givengeometry
   - Choose-mainmenu → Toolpath → Jobsetup
   - Enter the X =155 Y=155 and Z=65 mmrespectively
   - Select Display Stack and Fit toScreen.
   - Out side the drawing doted boundary red line isdisplayed

2. Another option for jobsetup
   - Choose-mainmenu → Toolpath → Job setup
   - Select stock origin (0,0)
   - select the stockcorners.
   - Enter the value of Z = 65 (+ value).
   - Select the display stock checkbox.
   - Select fit stock checkbox.
   - Choose OK the stock should be enclosed by red dottedline.

Step 4

1. Create tool path forFacing
   - Choose-mainmenu → Toolpath → Face
   - Select start point for the chain at periphery of thefigure.
   - Choosedone.
   - Right click in the tool display area and select a 10 mm flat endmill
   - from the toollibrary.
   - Right click on the tool display , go to tool type select face milltool,
   - Select the facing parameters. Give Depth of cut -3mm (-negative)
   - Select depth Cuts, Give Rough Cut 3, Finishing cut 1No. Finish Step0.5
   - Choose OK twice in the tool path should be showingfigure.

2. Create tool path for Contour 1,2 &3

- Choose-mainmenu → Toolpath → Contour
- Select 100mm diametercircle.
- Choosedone.
- Right click in the tool display area and select a 12 mm flat end mill from the toollibrary.
- Select the contour parameters , Give Depth of cut -3mm - negative, (Cumulative ie 3+3=6mm)
- Use Multi passes and Depth Cutsoptions.
- Choose OK twice in the tool path should be showingfigure.
- Repeat same procedure for other contoursfor
- Rambo's Square with depth 3 mm, (Cumulative ie 3+3+3=9mm)
- Filleted Square with depth 3 mm, (Cumulative ie 3+3+3+3=12mm)

Create tool path for Contour4

- Choose-mainmenu →Toolpath → Contour
- Select start point for the chain at position 1. (outer periphery of the square)
- Choosedone.
- Right click in the tool display area and select a 5 mm flat end mill from the toollibrary.
- Select the contour parameters, Give Depth of cut -65mm (- negative), Use Multi passes, Depth Cuts and Lead in/outoptions.
- Choose OK twice in the tool path should be showingfigure.

3. Create tool path for pocket1

- Choose-mainmenu → Toolpath → Pocket
- Select the 86 mm diametercircle.
- Choosedone.
- Right click in tool display area and select 8mm flat end mill fromtool
- library.
- Enter pocketing parameters and note that the depth shouldbe negative say-65.

- Choose OK the tool path should look like the picture pattern. 4

Create tool path for pocket2

- Choose-mainmenu → Toolpath → Pocket
- Select first 15 mm diametercircle.
- Choosedone.
- Right click in tool display area and select 3mm flat end mill from toollibrary.
- Enter pocketing parameters and note that the depth shouldbe negative say-15.
- Choose OK the tool path should look like the picturepattern.

5. Create tool path for Drilling 1 (10 mm dia)

- Choose-mainmenu → Toolpath → Drill
- Select Entities Choose 10mm diameter circles one byone.
- Choose done. Tool path is generated, Choose once againdone
- Right click in the tool display area and select a 10 mm drill mill from thetool library.
- Select the drill parameters and note that the depth should be negative-68.
- Choose-Esc

Create tool path for Drilling 2 (03 mm dia)

- Toolpath Drill
- Select Entities Choose 3 mm diameter circles one byone.
- Choose done. Tool path is generated, Choose once againdone
- Right click in the tool display area and select a 3 mm drill mill from the tool library.
- Select the drill parameters and note that the depth should be negative-30.
- Choose OK twice in the tool path should be showingfigure

Step 5

Tool path Generator

- Choose-mainmenu → Operation
- A dialogue box appears select all , regeneratepath
- When the tool path generation completes the dialogue boxdisplay
- chooseverify,

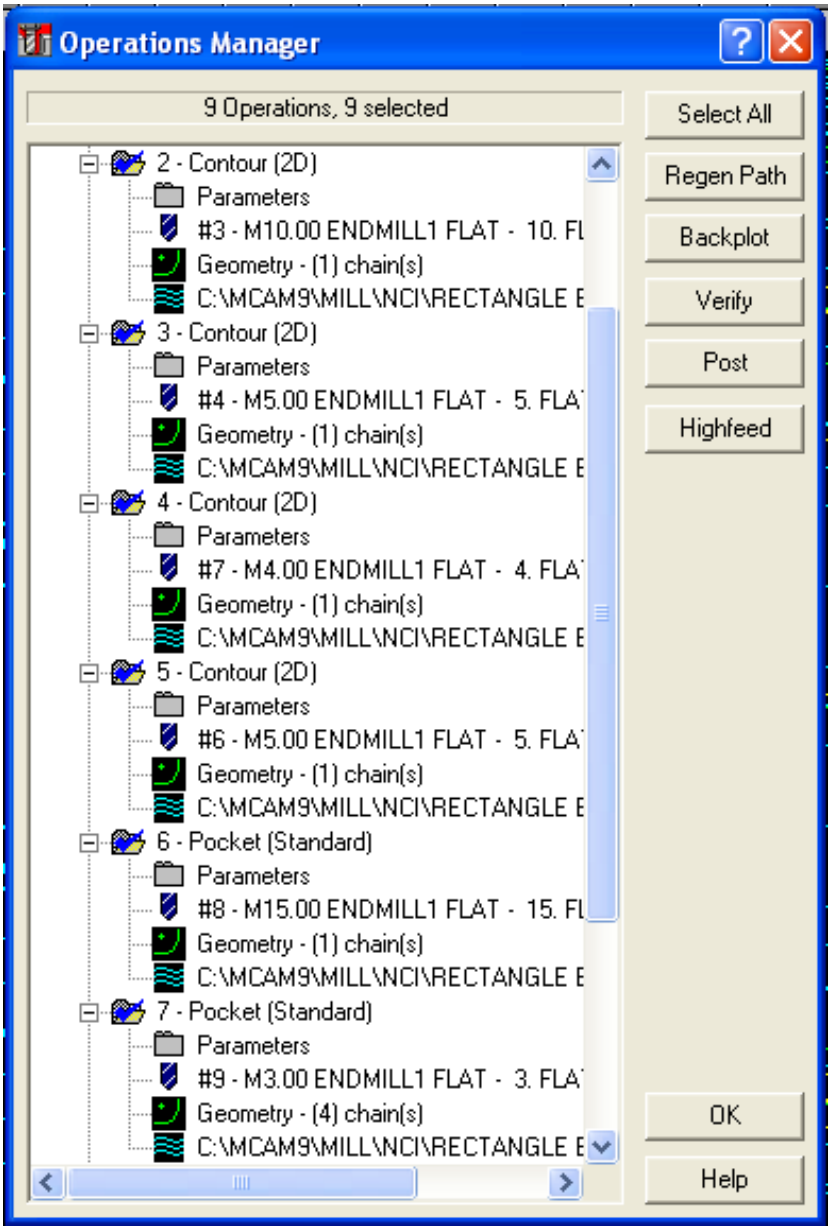
- Verifytrue solid deluge box is displayed select run (machine) item.
- Verify true solid deluge box go to Configure

  Select Job Box orCylinder,

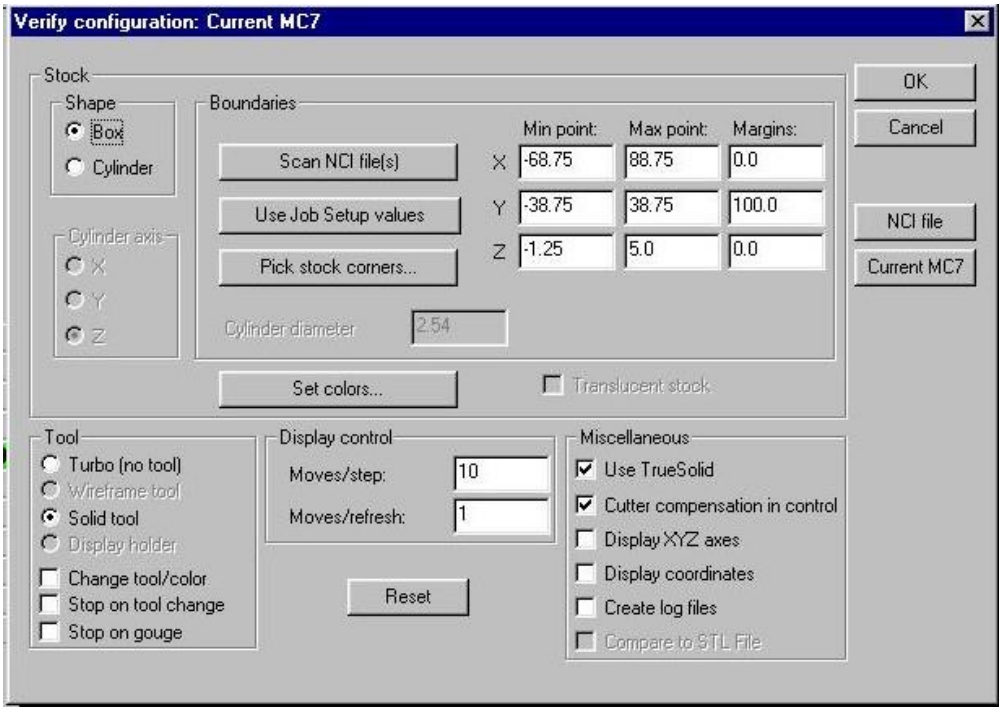
  Select use job setup values for present file operations

  In Display control, use Simulation speed / qualityvariations

  In miscellaneous select use true solid & Cutter compensation in control In Toll select Solid tool.

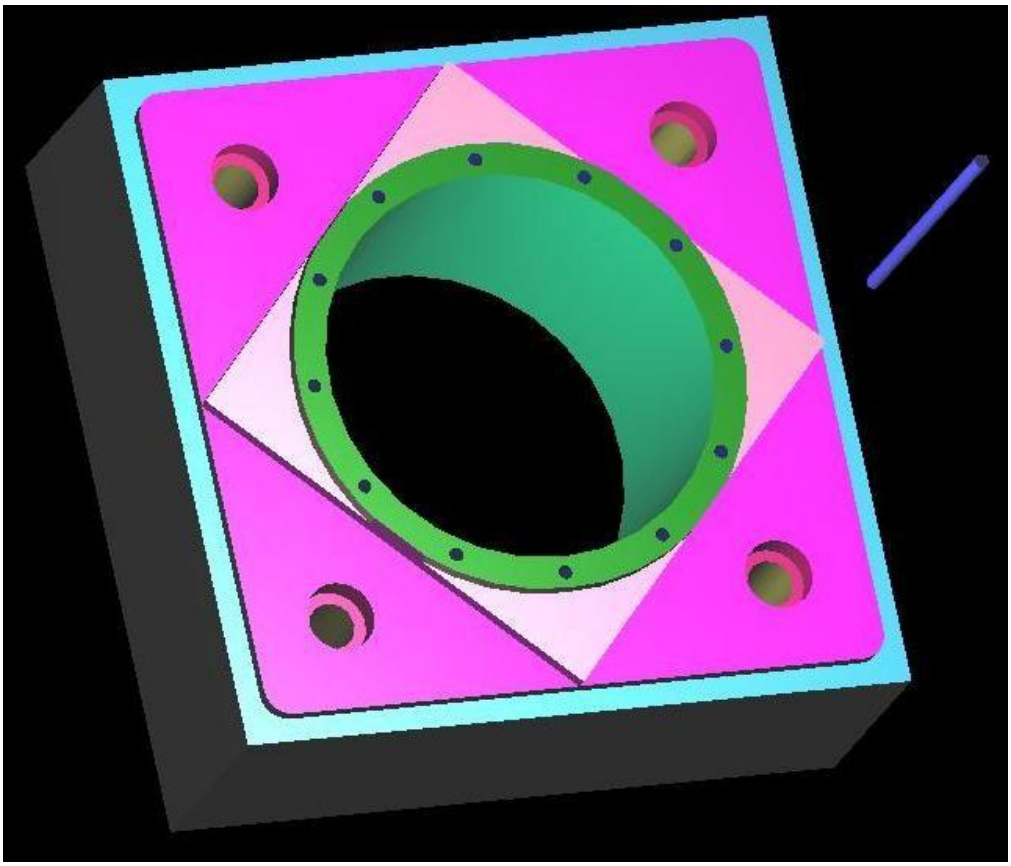


Operation management of Tool Path



Configuration of Tool path

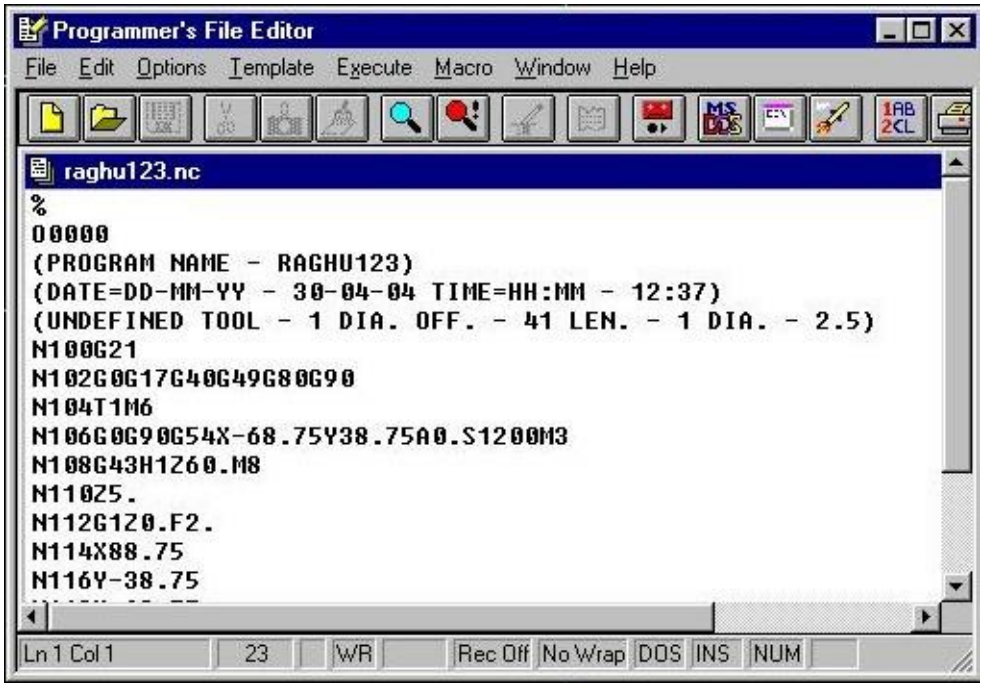
If your model is square/Rectangle go to shape, select box , suppose your model is Cylindrical go to shape, select cylinder



Step 6

Post processor NC Program

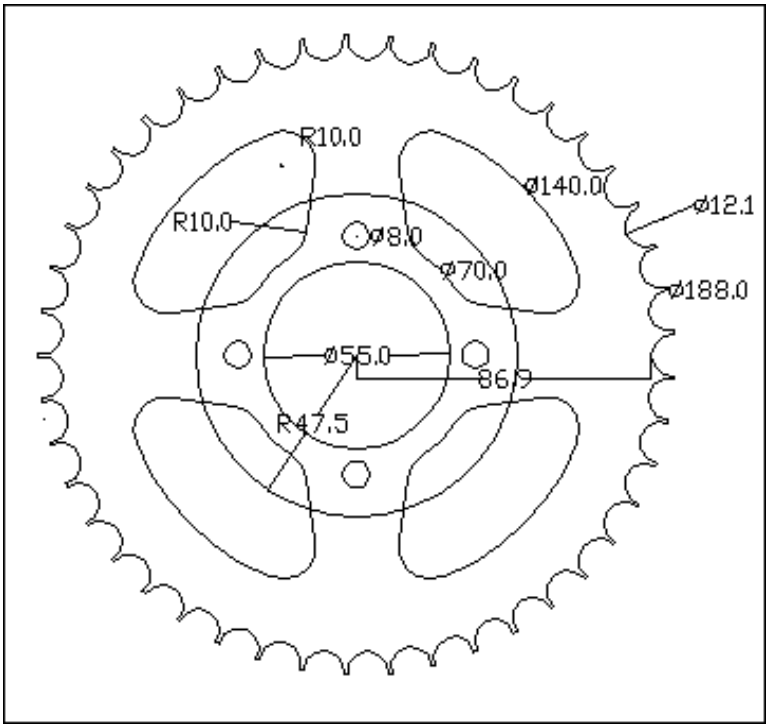
- Choose-mainmenu → Operation
- A dialogue box appears select all regenerate path, choosepost
- Show path (desktop) for saving post processor Notepadfile.



Exercise No.3

SPROCKET

Aim : To machine the model as per thesketch.



Procedure:

Step 1

The usual modification is that is done after initializing the master cam is the alteration of the available screen ie modification of grid size and as per the requirement of user screen,

Choose-mainmenu → Screen → Configure → Current Configuration File

Select- Mill9.Mcfg(metric)

Choose-mainmenu → Screen → Nextmenu → Sel. Grid

Select-Active Grid, Visible Grid, Spacing x =1 and y =1, Grid Size= 200, Origin x=0,y=0 and choose suitable for Grid.

Step 2

To create inner and outer circles.

- Choose-mainmenu → Create → Arc → circpt +dia.

- Typediameter188 → Enter
- Selectorigin
- Press Esc. And reselectcircpt+dia
- Enter diameter174
- Selectorigin
- Press Esc. And reselectcircpt+dia
- Repeat same procedure for pitch circle diameter 12mm
- Selectquadrant
- Press Esc. to exit circlefunction
- Choose it screenmenu.
- Create arc 3points
- First point – intersection of 12mm & 188 mm diacircles.
- Second point – quadrant of 174 mmcircle
- Third point is again opposite side of First point – intersection of 12mm & 188 mm diacircles.
- Delete 12 mmcircle

Copy and rotate the 3 point arc

- Choose-mainmenu → Xform → Rotate
- Select anywhere on 3 pointarc
- Choosedone.
- Selectorigin.
- Select copy, enter the values. No of steps 44 and angle is $360/45^0$
- Choose OK in the dialogue box remaining arcs forcreated.

Trimming unwanted portion of teeth

- Choose-mainmenu → Modify → Trim → 3entities
- Select 1 Entity Select 3 point arc inside the 188 mm dia circle 2 Entity adjacent 3 point arc inside the 188 mm diacircle 3 Entity 188 mm dia circle in between the two 3point arcs.

Copy and rotate the 188 mm dia circle in between the two 3point arcs

- Choose-mainmenu → Xform → Rotate

- Select anywhere on arc in between the two 3pointarcs
- Choosedone.
- Selectorigin.
- Select copy, enter the values. No of steps 44 and angle is$360/45^0$
- Choose OK in the dialogue box remaining arcs forcreated.
- Choose-mainmenu → Create Arc → circpt +dia.
- Typediameter140 → Enter
- Selectorigin
- Repeat same procedure for diameter 70mm
- Selectorigin

To create construction ofline

- Choose-mainmenu → Create → Line → multiline
- Enter the first point isorigin
- Enter the Second point is quadrant of 140 mm diacircle.
- Press Esc to exit linefunction
- Go to xform offset select the line offset distance10mm
- Repeat the same by 15mm
- Choose-mainmenu → Create → Line → multiline
- Enter the first point is intersection of 140mmcircle and 10 mm offsetline
- Enter the Second point is intersection of 70 mm circle and 15 mm offset line.
- Delete both offsetlines
- Go to fillet select radius 10mm
- Enter the first inclinedline
- Enter the Second 140mm diacircle.
- Same steps repeat for 70mm diacircle

To create construction of line

- Choose-mainmenu → Create → Line → Polar
- Enter the first co-ordinateorigin
- Enter angle in degree45^0
- Enter the line length 145

Trimming unwanted portion of 140/70 dia circle

- Choose-mainmenu → Modify → Trim → 3entities
- Select 1 Entity Select 140 mm diacircle
  2 Entity Select 70 mm diacircle
  3 Entity 45^0 line in between 140 and 70 diacircles.

Mirror

- Choose-mainmenu → Xform → Mirror
- Select chain, anywhere onfillet/line/arc
- End here, done , Mirror aboutline
- Choosedone.

Copy and rotate the mirrored item

- Choose-mainmenu → Xform → Rotate
- Select chain, Endhere,
- Choosedone.
- Selectorigin.
- Select copy, enter the values. No of steps 3 and angle is90^0
- Choose OK in the dialogue box remaining arcs forcreated.

To create inner 55 mm and 95 mm diametercircles

- Choose-mainmenu → Create → Arc → circpt+dia.
- Typediameter 55 → Enter
- Go tobackup,
- circpt +dia95mm
- Press Esc., and reselectcircpt+dia
- ThenoK

To create inner 08mm diameter circles

- Choose-mainmenu → Create → Line → Polar
- Enter the first co-ordinateorigin
- Enter angle in degree0^0
- Enter the line length40

- Press esc to exit line function

- Choose-mainmenu → Create → Arc → circpt +dia 8mm enter End point go and Select of theline

Copy and rotate the Circle

- Choose-mainmenu → Xform → Rotate
- Select anywhere on 8 mm diametercircle
- Choose done. Selectorigin.
- Select copy, enter the values. No of steps 3 and angle is90^0
- Choose OK in the dialogue box remaining 3 circles forcreated.

Now your drawing is ready for operations

The following operations to be conduct using the geometry

Facing, Contours, Pocketing and Drilling

Step 3

1. Make job setup for givengeometry
   - Choose-mainmenu → Toolpath → Jobsetup
   - Enter the X =190 Y=190 and Z= 11 mmrespectively
   - Select Display Stack and Fit toScreen.
   - Out side the drawing doted boundary red line isdisplayed

2. Another option for jobsetup
   - Choose-mainmenu → Toolpath → Job setup
   - Select stock origin (0,0)
   - Select the stock corners.
   - Enter the value of Z = 11 (+ value).
   - Select the display stock checkbox.
   - Select fit stock checkbox.
   - Choose OK the stock should be enclosed by red dottedline.

Step 4

Create tool path for Facing

- Choose-mainmenu → Toolpath → Face
- Select start point for the chain at periphery (Teeth) of thefigure.
- Choosedone.
- Right click in the tool display area and select a 10 mm flat end mill from the toollibrary.
- Right click on the tool display , go to tool type select face milltool,
- Select the facing parameters. Give Depth of cut -3mm (-negative)
- Select depth Cuts, Give Rough Cut 2, Finishing cut 1No. Finish Step1
- Choose OK twice in the tool path should be showingfigure.

Create tool path for Contour

- Choose-mainmenu → Toolpath → Contour
- Select start point for the chain at periphery (Teeth) of thefigure.
- Choosedone.
- Right click in the tool display area and select a 3 mm flat end mill from the toollibrary.
- Select the contour parameters , Give Depth of cut -8 mm - negative, (Cumulative ie 3+8=11mm)
- Use 3 Multi passes with spacing 2 mm and Depth Cuts options. Select depth Cuts, Give Rough Cut 3, Finishing cut 2 No. Finish Step1
- Choose OK twice in the tool path should be showingfigure.

Create tool path for pocket 1,2,3,4 and 5

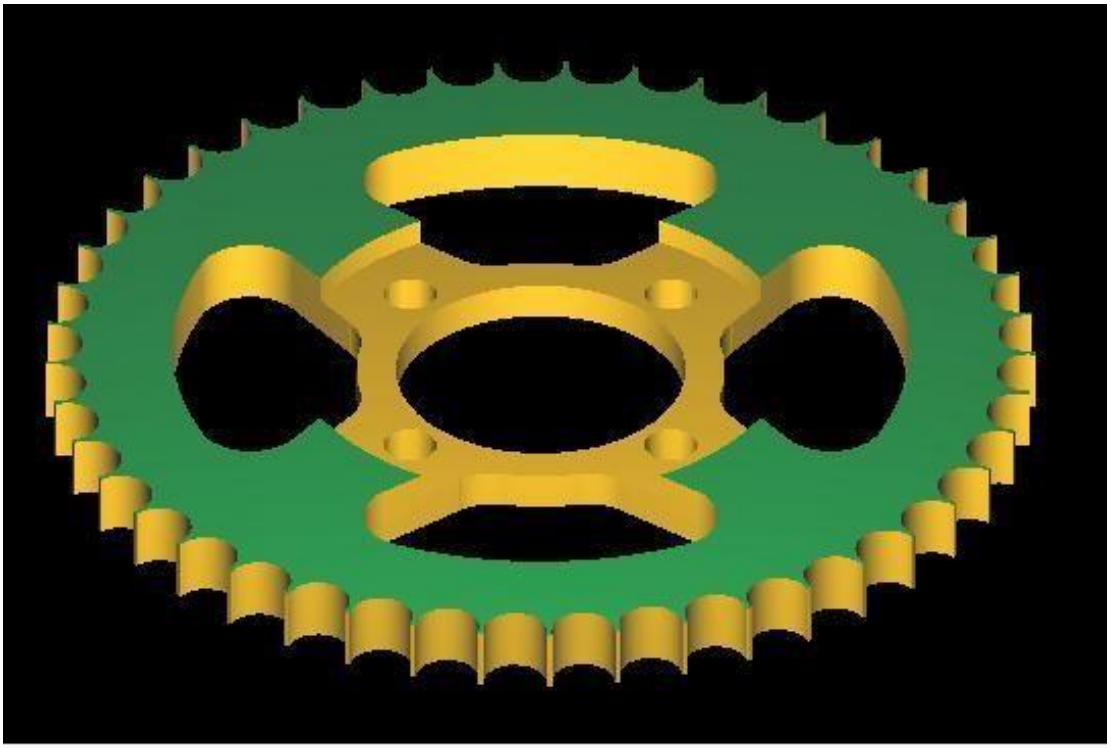
- Choose-mainmenu → Toolpath → Pocket
- Select the 55 mm diameter circle, another 4 slotspacket.
- Choosedone.
- Right click in tool display area and select 5 mm flat end mill fromtool
- library.
- Enter pocketing parameters and note that the depth shouldbe negative say-8.
- Choose OK the tool path should look like the picturepattern.

Create tool path for pocket 95 mm dia

- Choose-mainmenu → Toolpath → Pocket
- Select first 95 mm diametercircle.
- Choosedone.
- Right click in tool display area and select 10mm flat end mill from toollibrary.
- Enter pocketing parameters and note that the depth shouldbe negative say--6.
- Choose OK the tool path should look like the picturepattern.

Create tool path for Drilling 1 (08 mm dia)

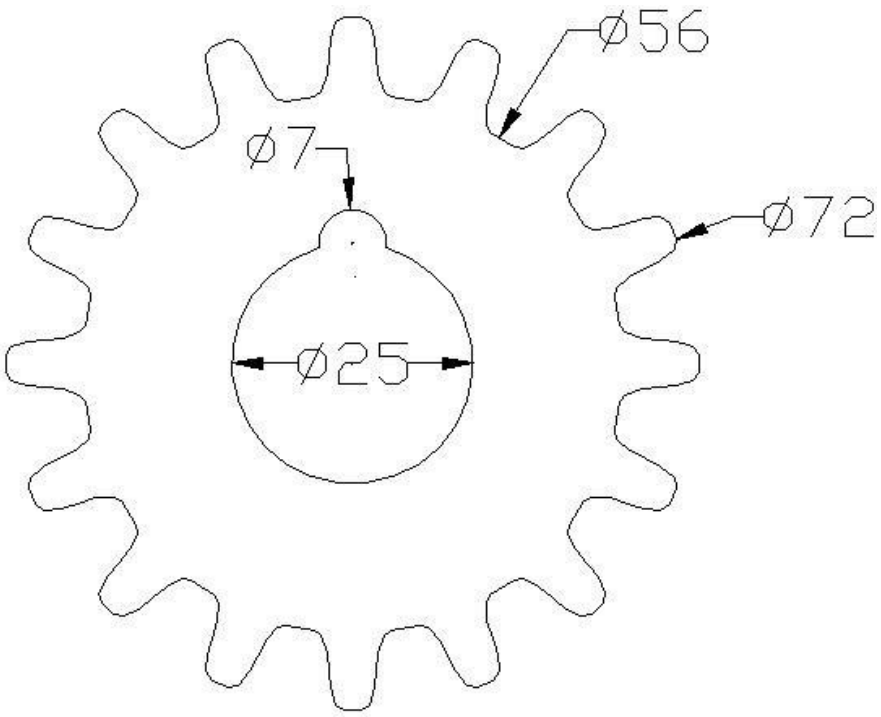
- Choose-mainmenu → Tool path → Drill
- Select Entities Choose 8 mm diameter 4 circles one byone.
- Choose done. Tool path is generated, Choose once againdone
- Right click in the tool display area and select a 8 mm drill mill from the tool library.
- Select the drill parameters and note that the depth should be negative -10.
- Choose-Esc



Exercise No. 4

Gear

Aim : To machine the model as per thesketch.



Procedure:

Step 1

The usual modification is that is done after initializing the master cam is the alteration of the available screen ie modification of grid size and as per the requirement of user screen,

Choose-mainmenu → Screen → Configure → Current Configuration File

Select- Mill9.Mcfg(metric)

Choose-mainmenu → Screen → Nextmenu → Sel. Grid

Select-Active Grid, Visible Grid, Spacing x =1 and y =1, Grid Size= 100,

Origin x=0,y=0 and choose suitable for Grid.

Step 2

To create inner and outer circles.

- → Choose-mainmenu → Create Arc circpt +dia.

- Typediameter 72 → Enter
- Selectorigin
- Press Esc. And reselectcircpt+dia
- Go tobackup
- And reselectcircpt+dia
- Enter diameter56
- Selectorigin
- Press Esc. And reselectcircpt+dia
- Then create one pitch enter circle for diameter 64mm
- Selectorigin
- Press Esc. And reselectcircpt+dia
- Repeat same procedure for pitch circle diameter 7mm
- Select quadrant of 25 mmdia
- Press Esc. to exit circlefunction

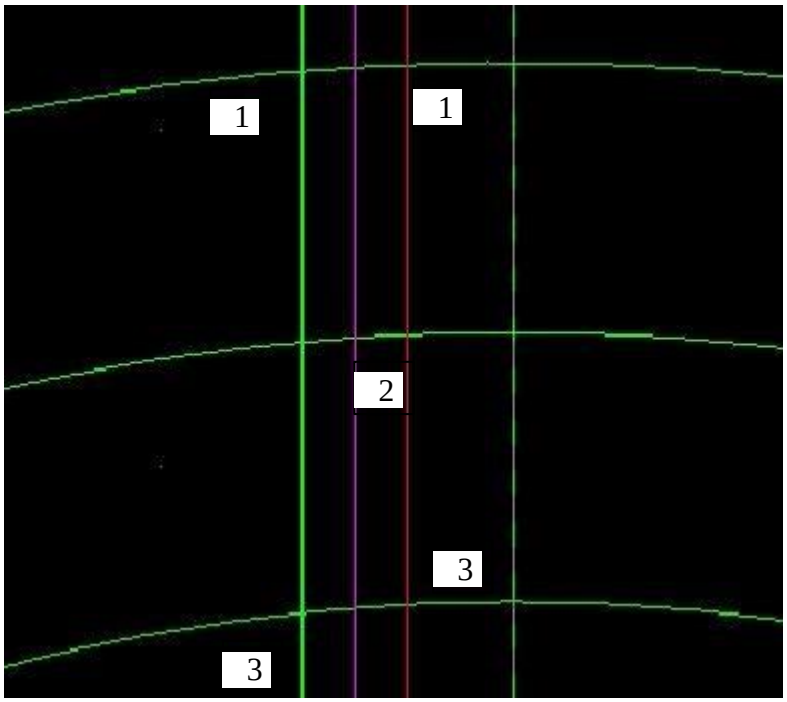
To create construction ofline

- Choose-mainmenu → Create → Line → multiline
- Enter the first point isorigin
- Enter the Second point is quadrant of 72 mm diacircle.
- Press Esc to exit linefunction
- Go to xform offset select the line offset distance1.5mm
- Repeat the same by 3.0mm
- Again go to xform offset select the line offset distance 0.75 mm (inbetween line 1.5 – 3)

Choose, menu, create,Arc

- Create arc 3points

1



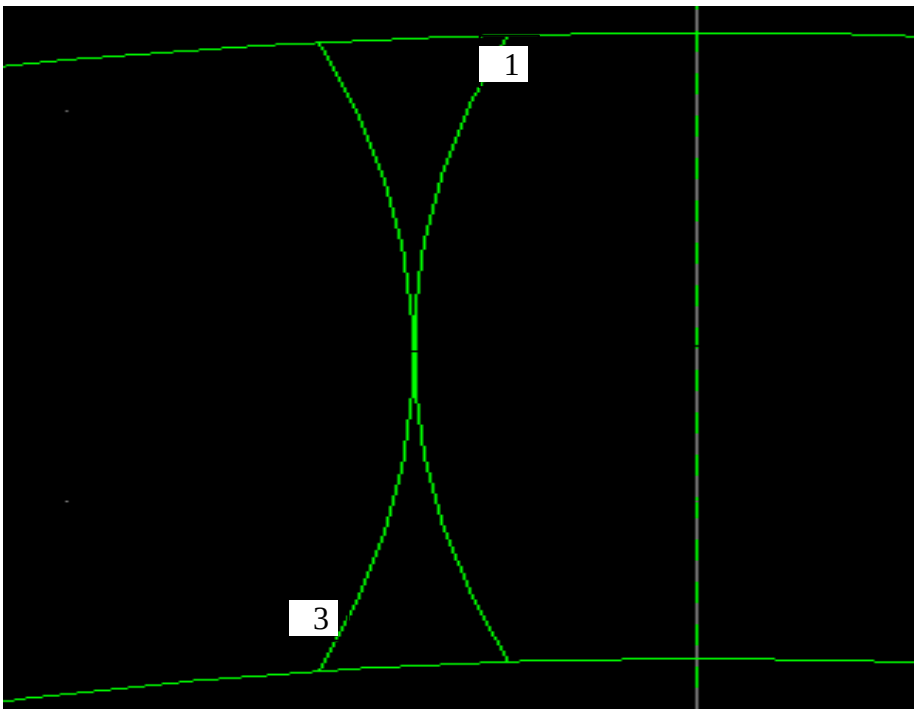
- First point – intersection of 72mm circle & offset line3mm.
- Second point – intersection of 64mm circle & 1.5 +0.75 mm offset linemm.
- Third point - intersection of 56 mm circle & offset line3mm.

Same procedure is repeat opposite side

Delete all the vertical line and 64 mm diameter circle

Trimming unwanted portion of teeth

- Choose-mainmenu → Modify → Trim → 2entities
- Select 1 Entity Select 3 point arc (No.3) inside the 56 mm diacircle
  2 Entity adjacent 3 pointarc(No.1) inside the 72 mm diacircle



Fillet

- Choose-mainmenu → Create → Fillet
- Radius, take 0.5 mm,enter
- Select item 1 arc and dia 72circle
- Repeat the procedure item 3arc and dia 56circle

Mirror

- Choose-mainmenu → Xform → Mirror
- Select four 4 entities one by one 1) fillet, 2) item No 3 arc, 3) item No 1 arc and 4)fillet
- End here, done , Mirror about line or yaxis
- Choosedone.

Trimming unwanted portion of teeth

- Choose-mainmenu → Modify → Trim → 2 entities

  Select 1 Entity Select fillet of theteeth

  2 Entity Select 72 mm dia circle inside portion of teeth

Copy and rotate the gear teeth

- Choose-mainmenu → Xform → Rotate
- Select gearteeth
- Choosedone.
- Selectorigin.
- Select copy, enter the values. No of steps 15 and angle is$360^0/16$
- Choose OK in the dialogue box remainingteeths.

Trimming unwanted portion of teeth

- Choose-mainmenu → Modify → Trim → 3 entities

  Select 1 Entity Select fillet of the teeth (dia56)

  2 Entity Select opposite fillet of the teeth (dia56)

  3 Entity Select 56 mm dia circle in between twofillet

Copy and rotate the dedandam arc ie 56 dia circle

- Choose-mainmenu → Xform → Rotate
- Select 56 mm dia circle in between twofillet
- Choosedone.
- Selectorigin.
- Select copy, enter the values. No of steps 15 and angle is$360^0/16$
- Choose OK in the dialogue box remainingteeths.

Now your drawing is ready for operations

The following operations to be conduct using the geometry

Facing, Contours, Pocketing and Drilling

Step 3

1. Make job setup for givengeometry
   - Choose-mainmenu → Toolpath → Jobsetup
- Enter the X =75 Y= 75 and Z= 33 mmrespectively
- Select Display Stack and Fit toScreen.
- Out side the drawing doted boundary red line isdisplayed

Step 4

Create tool path for Facing

- Choose-mainmenu → Toolpath → Face
- Select start point for the chain at periphery (Teeth) of thefigure.
- Choosedone.
- Right click in the tool display area and select a 10 mm flat end mill from the toollibrary.
- Right click on the tool display , go to tool type select face milltool,
- Select the facing parameters. Give Depth of cut -3mm (-negative)
- Select depth Cuts, Give Rough Cut 2, Finishing cut 1No. Finish Step1
- Choose OK twice in the tool path should be showingfigure.

Create tool path for Contour

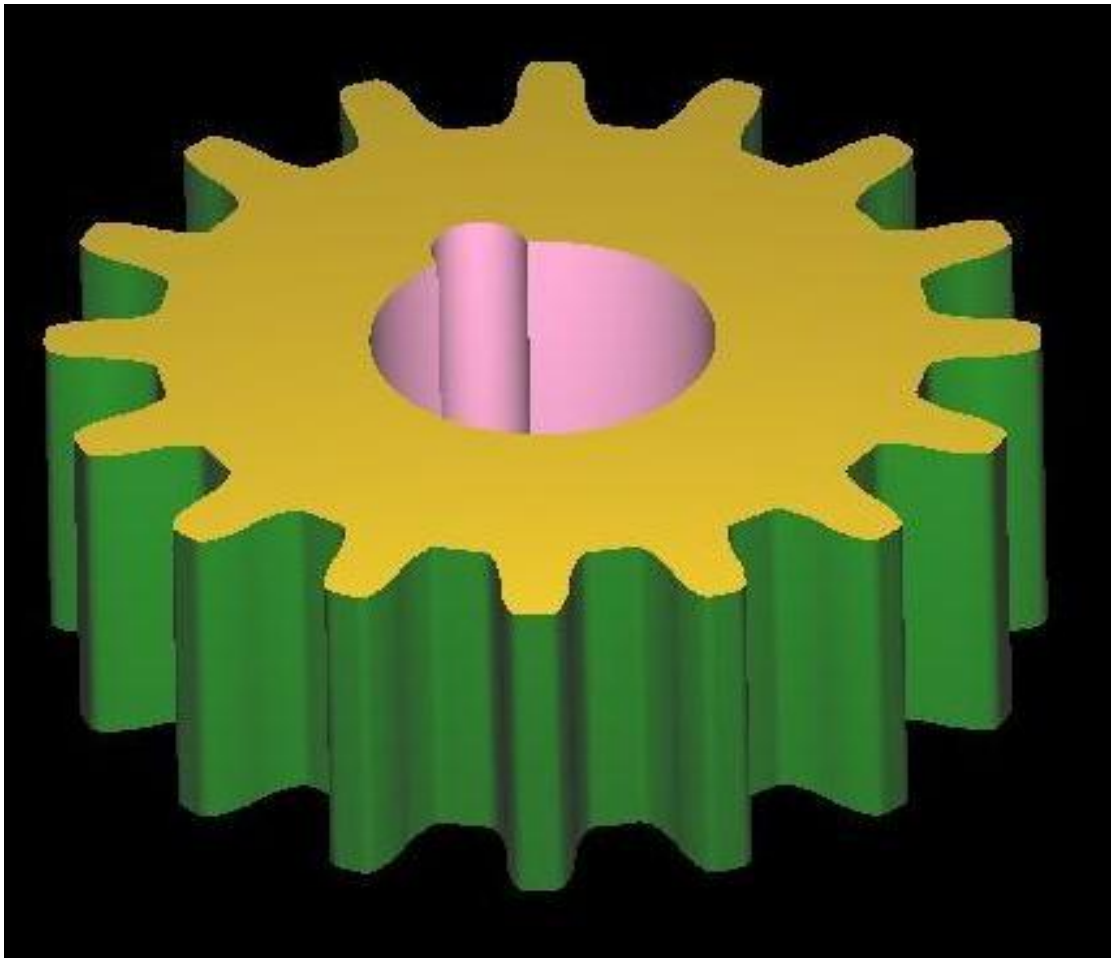
- Choose-mainmenu → Toolpath → Contour
- Select start point for the chain at periphery (Teeth) of thefigure.
- Choosedone.
- Right click in the tool display area and select a 2 mm flat end mill from the toollibrary.
- Select the contour parameters , Give Depth of cut -33 mm - negative, (Cumulative ie 3 + 30 = 33mm)
- Use 5 Multi passes with spacing 1 mm and Depth Cuts options. Select depth Cuts, Give Rough Cut 4, Finishing cut 3 No. Finish Step2
- Choose OK twice in the tool path should be showingfigure.

Create tool path for pocket

- Choose-mainmenu → Toolpath → Pocket
- Select the 25 mm diameter circle,
- Choosedone.
- Right click in tool display area and select 5 mm flat end mill from toollibrary.
- Enter pocketing parameters and note that the depth shouldbe negative say-33.
- Choose OK the tool path should look like the picturepattern.

Create tool path for Drilling 1 (07 mm dia)

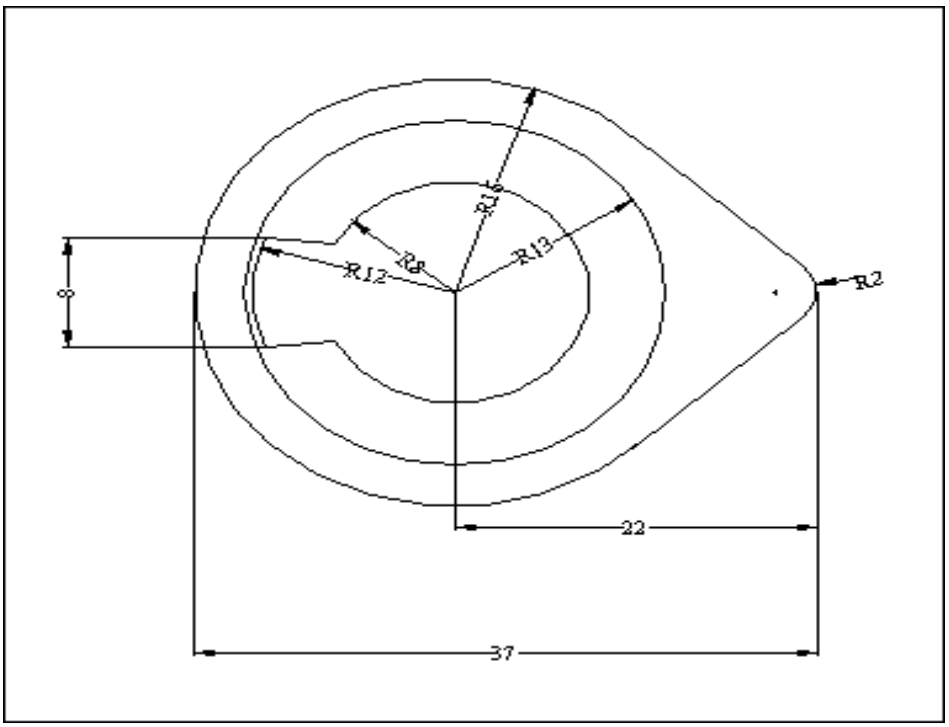
- Choose-mainmenu → Toolpath → Drill
- Select Entities Choose 8 mmdiameter.
- Choose done. Tool path is generated, Choose once againdone
- Right click in the tool display area and select a 7 mm drill mill from the tool library.
- Select the drill parameters and note that the depth should be negative -35.
- Choose-Esc

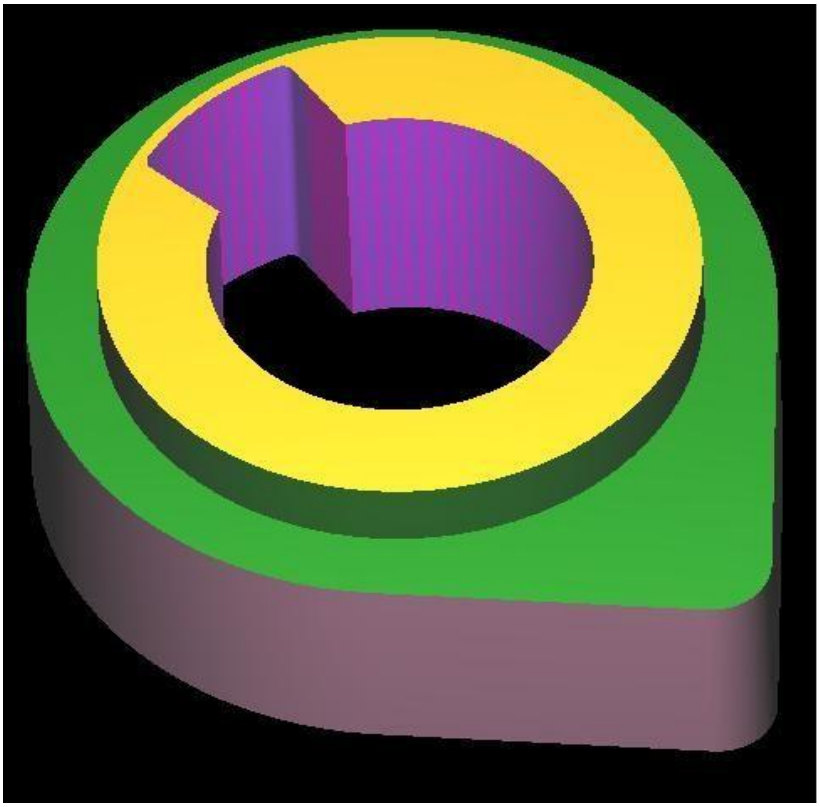


Exercise 5

CAM

Aim : To machine the model as per thesketch.

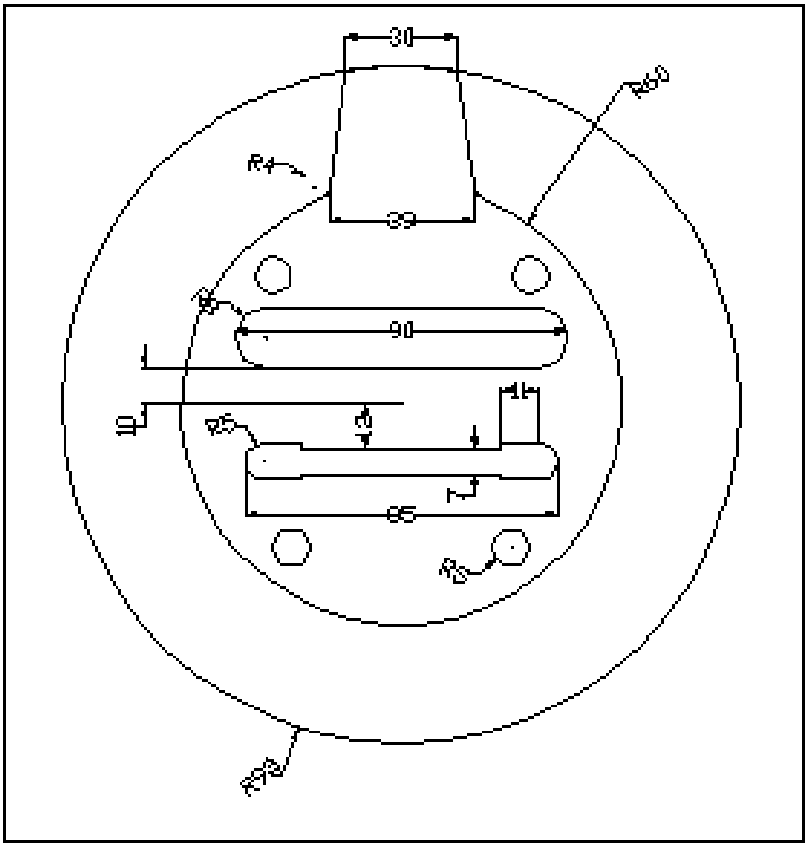


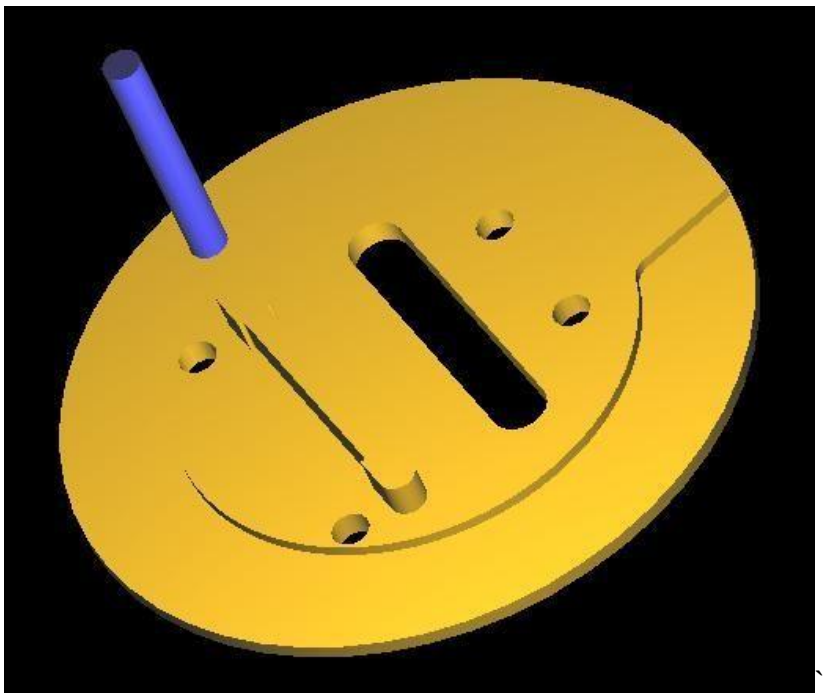


Exercise No. 6

Plate

Aim : To machine the model as per thesketch.





Exercise No. 7

Gear