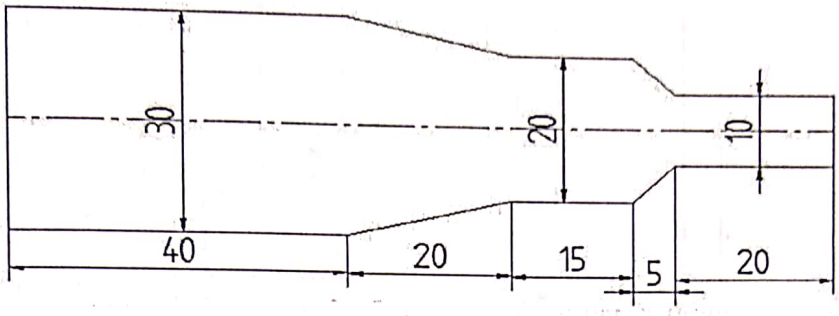
CNC TURNING

Program 01

Write a manual part program for the given part and execute multiple turning and finishing operation.

Objective: To simulate multiple turning and finishing operation on a given billet.



Syntax

G71 U_R_

G71 P_Q_U_W_F_

Where,

U is depth of cut
R is relief

G71 is a multiple turning cycle

P is block number of initial profile
Q is block number of final profile
U, W are finishing allowances in X and Z axis
F is feed rate

Syntax

G70 P_Q_F_

Where,

G70 is finishing cycle
P is block number of initial profile.
Q is block number of final profile.
F is feed rate

CNC Part Program:

%	N3 X20 Z-25
O0001	N4 Z-40
G21 G98	N5 X30 Z-60
G28 U0 W0	N6 Z-100
M06 T0101 (MTJNL 2020K16 R0.8) (OD Turning)	G28 U0 W0
M03 S1000	M06 T0202 (MTJNL 2020K16 R0.4) (OD Finishing)
G00 X30 Z1	M03 S800
G71 U0.5 R1	G70 P1 Q6 F40
G71 P1 Q6 U0.5 W0.5 F40	G28 U0 W0
N1 G01 X10 Z0	M05 M30
N2 Z-20	%

Outcomes:

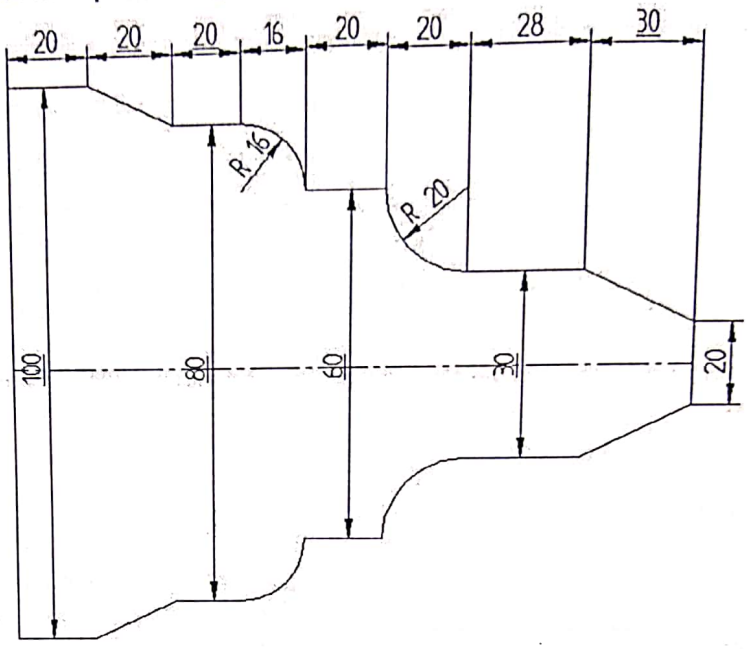
Student will be able to use G and M codes to prepare a part program for virtual machining operations, which is simulated using virtual single point cutting tool.

G-Word.

Program 02

Write a manual part program for the given part and execute multiple turning and finishing operat

Objective: To Simulate multiple turning, finishing operation and parting operation on a given bil



%	N8 X100 Z-154
O0002	N9 Z-174
G21 G98	G28 U0 W0
G28 U0 W0	M06 T0202 (MTJNL 2020K16 R0.4)
M06 T0101 (MTJNL 2020K16 R0.8)	M03 S800
M03 S1000	G70 P1 Q9 F30
G00 X100 Z1	G28 U0 W0
G71 U1 R1	M06 T0303 (20*20, 3.00W, 0.10R, 10 Depth, LH)
G71 P1 Q9 U0.5 W0.5 F40	M03 S500
N1 G01 X20 Z0	G00 X110 Z-174
N2 X30 Z-30	G01 X0
N3 Z-58	G01 X110
N4 G02 X60 Z-78 R20	G28 U0 W0
N5 G01 Z-98	M05 M30
N6 G03 X80 Z-114 R16	%
N7 G01 Z-134	

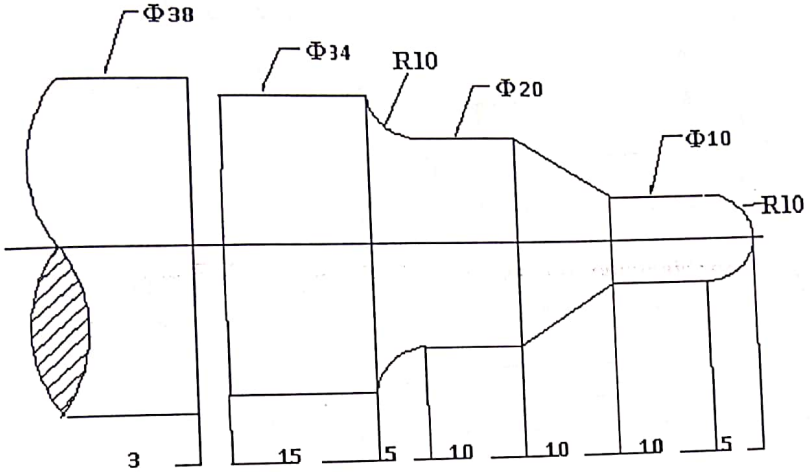
Outcomes:

Student will be able to use G and M codes to prepare a part program for virtual machining operations which is simulated using virtual single point cutting tool and parting tool.

Program 03

Write a manual part program for the given part and execute multiple turning, finishing and parting operations.

Objective: To simulate multiple turning, finishing operation and parting operation on a given billet.



%	N7 G01 Z-58
O0003	G28 U0 W0
G21 G98	M06 T0202 (MTJNL 2020K16 R0.4)
G28 U0 W0	M03 S800
M06 T0101 (MTJNL 2020K16 R0.8)	G70 P1 Q7 F30
M03 S1000	G28 U0 W0
G00 X38 Z1	M06 T0303 (20*20, 3.00W, 0.10R, 10 Depth, LH)
G71 U1 R1	M03 S500
G71 P1 Q7 U0.5 W0.5 F40	G00 X39 Z-55
N1 G01 X0 Z0	G01 X0
N2 G03 X10 Z-5 R10	G01 X39
N3 G01 Z-15	G28 U0 W0
N4 X20 Z-25	M05 M30
N5 Z-35	%
N6 G02 X34 Z-40 R10	

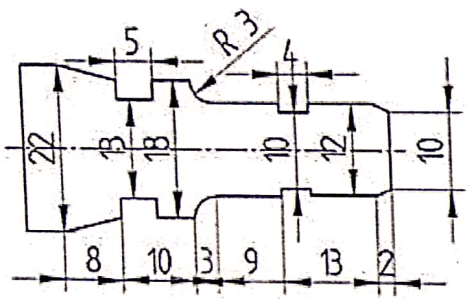
Outcomes:

Student will be able to use G and M codes to prepare a part program for virtual machining operations, which is simulated using virtual single point cutting tool and parting tool.

Program 04

Write a manual part program for the given part and execute multiple turning and finishing operation. Use grooving cycle. Grooving tool width is 3mm.

Objective: To simulate multiple turning, finishing operation and grooving operation on a given bille



G75 Cycle

G75 R_

G75 X_ Z_ P_ Q_ F_

Where

R is the retract amount

X is the diameter upto which the grooving must be done.

Z is the length specifying groove.

P is the pecking increment along x

Q is the stepping distance along z

%	M03 S800
O0001	G70 P1 Q6 F30
G21 G98	G28 U0 W0
G28 U0 W0	M06 T0303
M06 T0101 (MTJNL 2020K16 R0.8)	M03 S600 (16*16, 2.00W, 0.20R, 08Depth, LH)
M03 S1000	G00 X12 Z-15
G00 X22 Z1	G75 R1
G71 U1 R1	G75 X10 Z-11 P2000 Q1000 R1 F30
G71 P1 Q6 U0.5 W0.5 F40	G01 X20
N1 G01 X10 Z0	X18 Z-37
N2 X12 Z-2	G75 R1
N3 Z-24	G75 X13 Z-32 P2000 Q1000 R1 F30
N4 G02 X18 Z-27 R3	G01 X25
N5 G01 X18 Z-37	G28 U0 W0
N6 X22 Z-45	M05 M30
G28 U0 W0	%
M06 T0202 (MTJNL 2020K16 R0.4)	

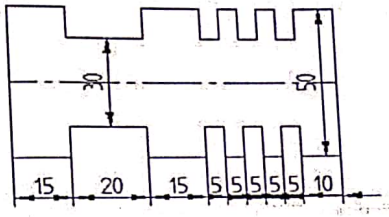
Outcomes:

Student will be able to use G and M codes to prepare a part program for virtual machining operations, which is simulated using virtual single point cutting tool and external grooving tool.

Program 05

Perform multiple turning and finishing operations on a given billet. Use G75 cycle to simulate grooving operation.

Objective: To simulate multiple turning, finishing operation and grooving operation on a given billet.



%	G01 X55
O0005	X50 Z-30
G21 G98	G75 R1
G28 U0 W0	G75 X30 Z-35 P2000 Q1000 R1 F30
M06 T0101 (16*16, 2.00W, 0.20R, 08Depth, LH)	G01 X55
M03 S1000	X50 Z-50
G00 X50 Z-10	G75 R1
G75 R1	G75 X30 Z-70 P2000 Q1000 R1 F30
G75 X30 Z-15 P2000 Q1000 R1 F30	G01 X55
G01 X55	G28 U0 W0
X50 Z-20	M05 M30
G75 R1	%
G75 X30 Z-25 P2000 Q1000 R1 F30	

Outcomes:

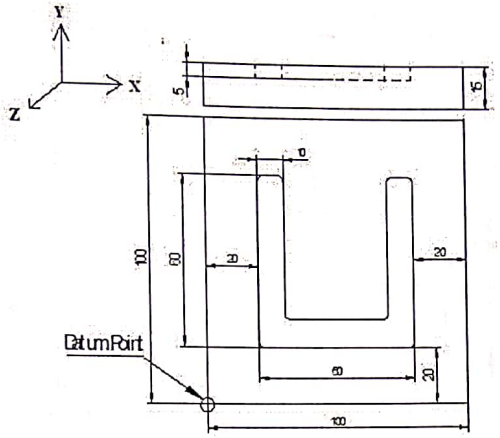
Student will be able to use G and M codes to prepare a part program for virtual machining operations, which is simulated using virtual single point cutting tool and external grooving tool.

CNC MILLING

Program 01

Write a manual part program for Slotting operation for the component as shown in drawing and execute.

Objective: To simulate milling operation on a given sample of material to produce a contour.



Material – Mild Steel, Operation – Slotting, Billet size–100 x100x15mm and Tool –End mill Φ 10 mm

%
O0001
G17 G21 G90 G98
M06 T01
M03 S1000
G00 X0 Y0 Z50
G43 H01
G00 Z2
G00 X20 Y80
G01 Z-2 F50
X20 Y20
X80 Y20
X80 Y80
G00 Z5
G28 X0 Y0
M05 M30
%

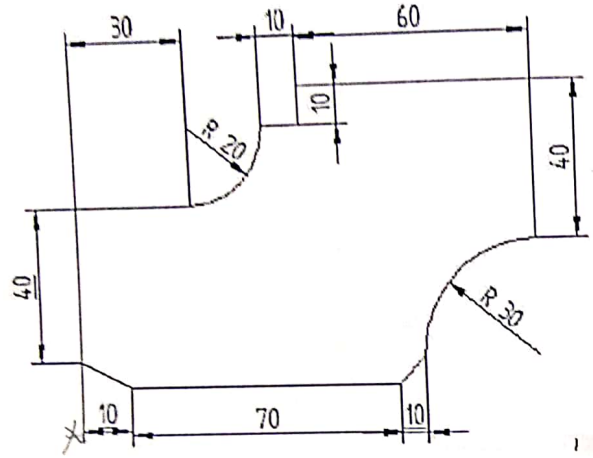
Outcomes:

Student will able to prepare a part program using G and M codes and simulate virtual milling operations using virtual end mill.

Program 02

Write a manual part program for the component as shown in drawing and execute. Thickness of the blo is 10 mm.

Objective: To simulate milling operation on a given sample of material to produce a contour.



Material – Mild Steel, Operation – Milling, Billet size – 120 x 80 x 10mm & Tool –End mill Φ 5mm

%	G01 X120 Y80
O0002	X60 Y80
G17 G21 G90 G98	X60 Y70
M06 T01	X50 Y70
M03 S1000	G02 X30 Y50 R20
G00 X0 Y0 Z50	G01 X0 Y50
G43 H01	X0 Y10
G00 Z2	X10 Y0
G00 X10 Y0	G00 Z5
G01 Z-2 F50	G28 X0 Y0
X80 Y0	M05 M30
X90 Y10	%
G02 X120 Y40 R30	

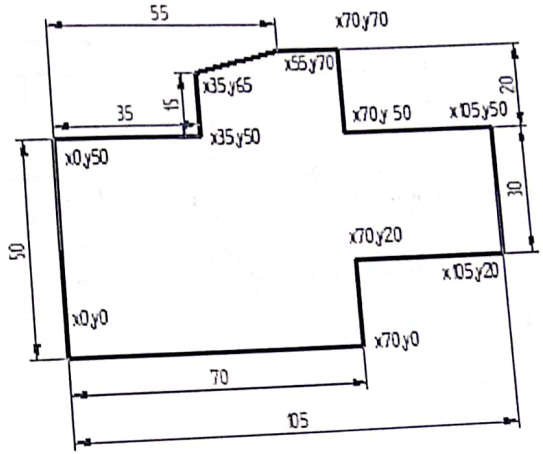
Outcomes:

Student will able to prepare a part program using G and M codes and simulate virtual milling operations using virtual end mill.

Program 03

Write a manual part program for the component as shown in drawing and execute. Thickness of the block is 10 mm.

Objective: To simulate milling operation on a given sample of material to produce a contour.



%	X55 Y70
O0003	X70 Y70
G17 G21 G90 G98	X70 Y50
M06 T01	X105 Y50
M03 S1000	X105 Y20
G00 X0 Y0 Z50	X70 Y20
G43 H01	X70 Y0
G00 Z2	X0 Y0
G01 Z-1 F50	G00 Z5
X0 Y50	G28 X0 Y0
X35 Y50	M05 M30
X35 Y65	%

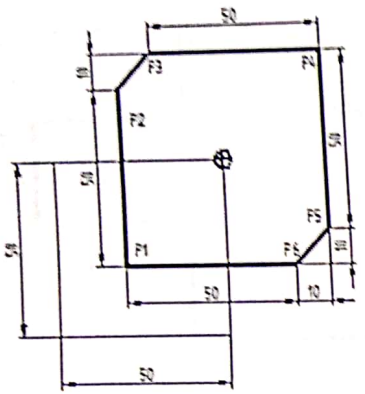
Outcomes:

Student will able to prepare a part program using G and M codes and simulate virtual milling operations using virtual end mill.

Program 04

Write a manual part program for the component as shown in drawing and execute. Thickness of the bl is 10 mm.

Objective: To simulate milling operation on a given sample of material to produce a contour.



	X	Y
P1	-30	-30
P2	-30	20
P3	-20	30
P4	30	30
P5	30	-20
P6	20	-30
P7	-30	-30

Billet size : 100 x100x10 mm
Cutter dia : 6mm

Billet Size: 100×100×10 mm, **Cutter dia:** 5 mm

%	X-30 Y20
O0004	X-20 Y30
G17 G21 G90 G98	X30 Y30
M06 T01	X30 Y-20
M03 S1000	X20 Y-30
G00 X0 Y0 Z50	X-30 Y-30
G43 H01	G00 Z5
G00 Z2	G28 X0 Y0
G00 X-30 Y-30	M05 M30
G01 Z-1 F50	%

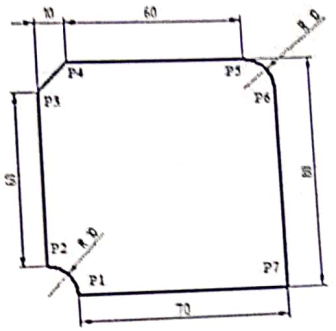
Outcomes:

Student will able to prepare a part program using G and M codes and simulate virtual milling operations using virtual end mill.

Program 05

Write a manual part program for the component as shown in drawing and execute. Thickness of the block is 10 mm.

Objective: To simulate milling operation on a given sample of material to produce a contour.



Billet size : 100x100x10 mm
Cutter dia : 8mm

Billet Size: 100×100×10 mm, **Cutter dia:** 5 mm

%	G01 X10 Y80
O0004	X20 Y90
G17 G21 G90 G98	X80 Y90
M06 T01	G02 X90 Y80 R10
M03 S1000	G01 X90 Y10
G00 X0 Y0 Z50	X20 Y10
G43 H01	G00 Z5
G00 Z2	G28 X0 Y0
G00 X20 Y10	M05 M30
G01 Z-2 F50	%
G03 X10 Y20 R10	

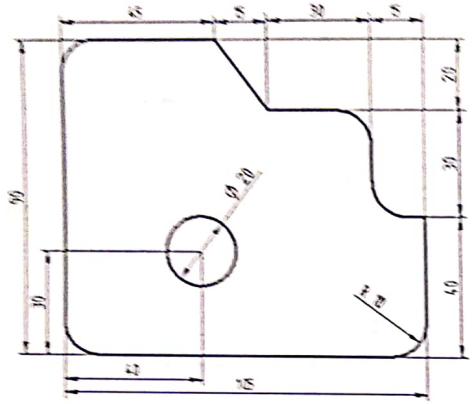
Outcomes:

Student will able to prepare a part program using G and M codes and simulate virtual milling operations using virtual end mill.

Program 06

Write a manual part program for Slotting operation for the component as shown in drawing and execute. Thickness of the block is 10 mm.

Objective: To simulate pocket milling operation on a given sample of material.



%	G03 X100 Y40 R10
O0004	G01 X105 Y40
G17 G21 G90 G98	X105 Y10
M06 T01	G02 X95 Y0 R10
M03 S1000	G01 X10 Y0
G00 X0 Y0 Z50	G02 X0 Y10 R10
G43 H01	G00 Z5
G00 Z2	G28 U0 W0
G00 X0 Y10	M06 T02
G01 Z-2 F50	M03 S1200
X0 Y80	G00 X40 Y30
G02 X10 Y90 R10	G01 Z-10 F120
G01 X45 Y90	G00 Z5
X60 Y70	G28 U0 W0
X80 Y70	M05 M30
G02 X90 Y60 R10	%
G01 X90 Y50	

Outcomes:

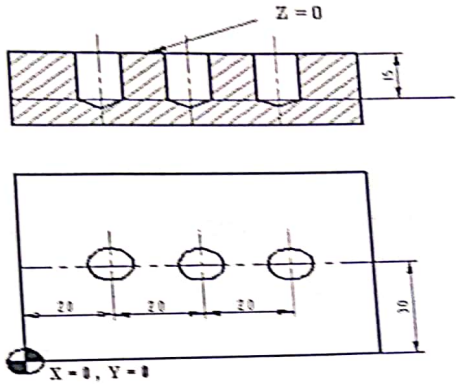
Student will able to prepare a part program using G and M codes and simulate virtual milling operation using virtual end mill.

CNC DRILLING

Program 01

Simulate drilling operations on a given sample of material.

Objective: To simulate drilling operations on a given sample of material.



3 HOLES OF 10 mm DIA

%
O0001
G17 G21 G90 G98
M06 T01
M03 S1000
G00 X0 Y0 Z50
G43 H01
G00 X20 Y30
G82 Z-15 F120
X40 Y30
X60 Y30
G80 G28 X0 Y0
M05 M30
%

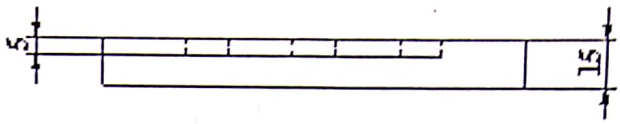
Outcomes:

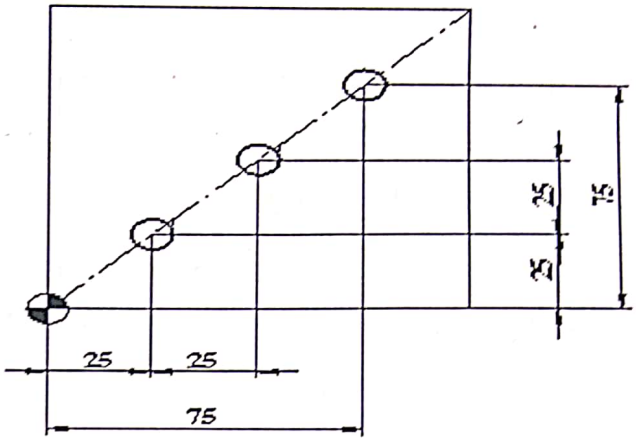
Students will be able to prepare a part program and simulate multiple drilling operations on a given billet using virtual drill bit.

Program 02

Simulate drilling operations on a given sample of material.

Objective: To simulate drilling operations on a given sample of material.






```
%
O0002
G17 G21 G90 G98
M06 T01
M03 S1000
G00 X0 Y0 Z50
G43 H01
G00 X25 Y25
G81 Z-5 F120
X50 Y50
X75 Y75
G80 G28 X0 Y0
M05 M30
%
```

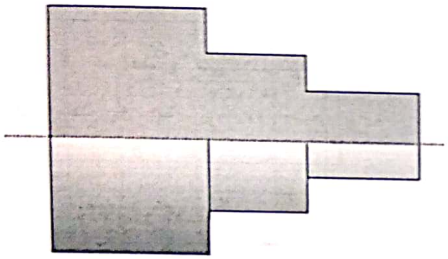
Outcomes:

Students will be able to prepare a part program and simulate multiple drilling and slotting operation a given billet using virtual drill bit and end mill.

Part-B CAPSTURN

Problem – 1

For the figure shown, perform the turning operation by CAPSTURN process



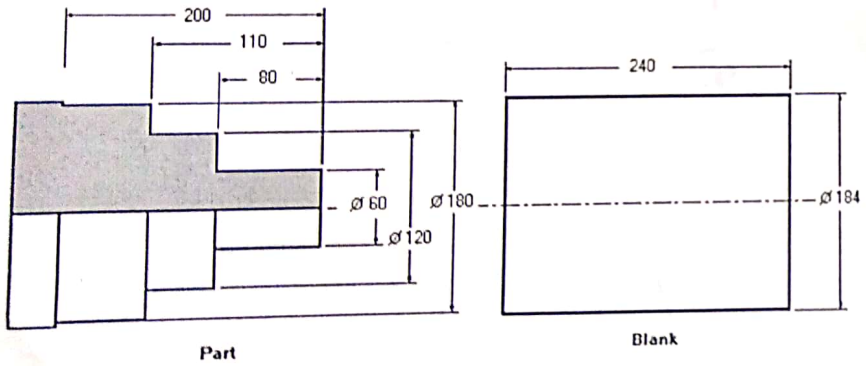
Objectives:

Creating part and blank shapes

Rough and finish turning

Viewing tool path

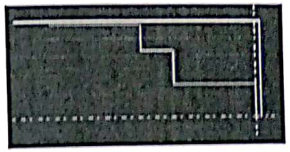
Part drawing



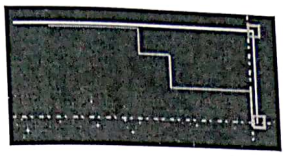
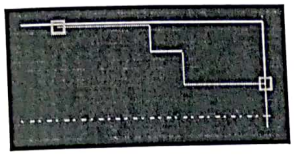
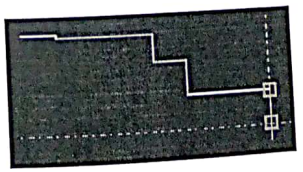
All dimensions are in millimeters

	Start a new job If you have just started CAPSturn, Select Create a new part in the startup dialog box and click on OK. If you have already started CAPSturn, select File → New
	Work setup The Work setup dialog box appears automatically when you start a new part. **Setup data 1** Select mm for Units, select any Work piece material, and enter this data, leaving the rest unaltered: Jaw position absolute Z = -210 Clearance Z = 5 Clearance OD = 200 **Setup data 2** Leave the default data unaltered. **Documentation** Click on the Documentation tab and enter suitable data. All the data except the Remarks will appear in the NC program and other documents like the cycle time

	sheet. You can omit any or all it. Anything that you do not enter appears as a blank in the documents. Click on OK to exit from the Work setup dialog box.
	**Save the File** Select File → Save. Enter the File name as Shaft. The file will now be saved as Shaft.CPT.
	Values can be entered as fractions. E.g., 7.5 can be entered as 7+1/2.
	Click on the Geometry button at the bottom left of the screen.
	**Draw the part** Select Draw → Quick shaft. At the prompt Specify the start point enter 0,0 and press Enter. At the prompt Select the direction click on the arrow pointing vertically up. At the prompt Specify X coordinate enter 60 and press Enter. At the prompt Specify Z coordinate enter –80 and press Enter. Enter the following co-ordinates in sequence: 120 -110 180 -200 • Click the right mouse button and then select Done.

	**Define the part** Through this step you are telling CAPSturn "This set of lines is the part". Select Draw → Define Part. At the prompt Select the first object click on the line shown. Click on Yes at the prompt Create part shape?
	**Define the blank** Select Draw → Define Blank. Enter the following data and click on OK: Length = 240 Diameter = 184 Start Z = 5
	If the blank shape is a complex shape and not a plain cylinder, Draw its shape. In the blank definition screen click on Select contour and then select the contour. Through this step you are telling CAPSturn "This set of lines is the blank".
	**Machining** Click on the Machining button at the bottom left of the screen. We will now do the following machining operations:

	Rough facing using the Plain face operation Rough turning using the Contour turn operation Finish facing using the Finish face operation Finish turning using the Finish turn operation
	To perform a machining operation in CAPSturn: Select the operation Select the tool Select the area to be machined Specify the machining parameters You will find this sequence of steps repeated for every operation.
	**Rough facing** **Select the operation** Select Machining → Turning → Plain face **Select the tool** Click on New tool. Click on OK to accept the default tool that CAPSturn has already selected. **Select the area to be machined**

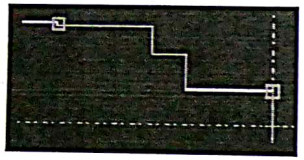
	When you are prompted for the Start X and End X click on the points shown, on the BLANK. Specify the machining parameters, and in the operation dialog box click on OK to accept the default data.
 	**Rough turning** Select Machining → Turning → Contour turn. In the Select an existing tool / new tool dialog box click on the tool selected earlier for facing and click on OK. When you are prompted for the Start point and End point of machining click on the points shown, on the PART. In the operation dialog box click on OK to accept the default data.
 	**Finish facing** Select Machining → Turning → Finish face. Click on New tool. Click on OK to accept the default tool that has been selected. For the Start point and End point of machining click on the points shown, on the PART. Enter this data in the operation dialog box, leaving the rest unaltered. Overshoot X = -0.5. This is 25 % more than the tool nose radius.
	Finish turning

Select Machining → Turning → Finish turn.

In the Select an existing tool / new tool dialog box click on the tool selected earlier for finish facing and click on OK.

For the Start point and End point of machining click on the points shown, on the PART.

In the operation dialog box click on OK to accept the default data.



Save the file.

Select a machine

Select Machine.

Click on a machine from the list.

Click on Select.

Work setup, Geometry definition, and Machining are independent of any machine. Functions like NC program and cycle time generation require the machine to be selected since they use the machine's mechanical data like feed rate limits, spindle speed limits and tool change time.

- **View tool path**

- Click on the Toolpath button at the bottom left of the screen.

Select Toolpath → Start.

The mode can be solid or line mode. Simulation can be till the next motion, next tool, next operation or end of the program.

The simulation can also be viewed in Single step mode. The tool moves one step at every click of the mouse or space bar on the keyboard.

Use the Zoom and Pan functions for closer inspection of the tool path.

Generate NC program

Click on NC program on the menu bar.

Click on OK in the dialog box that appears.

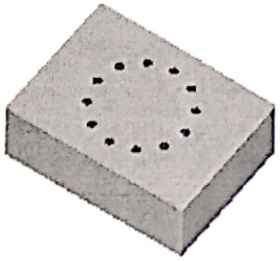
Click on Edit to view the NC program.

Select File → Exit to exit from the editor. The NC program is saved as a text file with a .PRG extension in the folder where CAPSturn is installed.

CAPSMILL

Problem – 1

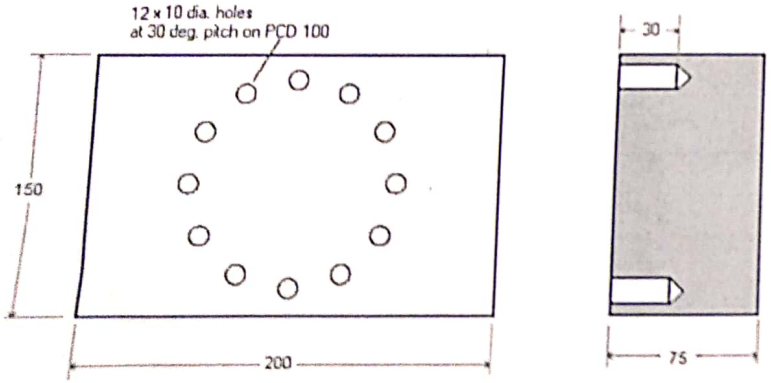
For the figure shown, perform the milling operation by CAPSMILL process



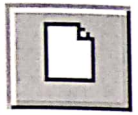
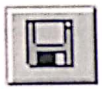
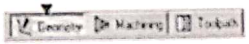
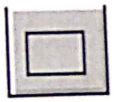
Objectives:

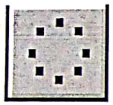
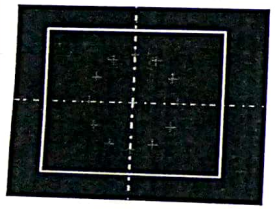
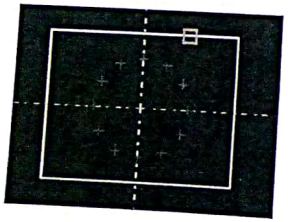
Creating part and blank shapes
Performing face milling, drilling and reaming operations
Viewing tool path
Generating NC program

Part drawing

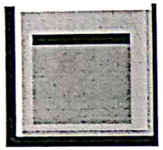


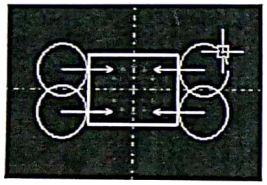
All Dimensions are in mm

	**Start a new part** If you have just started CAPSmill, Select Create a new part in the startup dialog box and click on OK. If you have already started CAPSmill, select File → New.
	**Work setup** The Work setup dialog box appears automatically when you start a new part. **Setup data** • Enter the following data, leaving the rest unaltered: For Units select 'MM'. For Work piece material select any material. **Documentation** • Click on the Documentation tab and enter suitable data. All the data except the Remarks will appear in the NC program and in other documents like the cycle time sheet. You can omit any or all of them. Any field that you do not enter appears blank in the shop documents generated by the program. • Click on OK to exit from the Work setup dialog box.
	**Save the File** • Select File → Save. • Enter the File name as 'Plate'. The file will now be saved as Plate.cpm. All job files that you create in CAPSmill will be stored with the extension, dot cpm.
	• Click on the Geometry button at the bottom left of the screen.
	**Draw blank** **Draw rectangle** • Select Draw → Shapes → Rectangle (Center – Length – Width). • Enter the following data: Length = 200

	Breadth = 150 Corner radius = 0 Rotation angle = 0 • Click on OK. • At the prompt Specify the center point of the rectangle, enter the coordinates 0,0 through the keyboard and press the Enter key or Done.
	**Draw part** **Draw points for 10mm diameter holes** • Select Draw → Point pattern → Points on Circle. • Enter the following data and click on OK. Diameter = 100 Angle of first hole = 0 Number of holes = 12 • At the prompt Enter the center point for this pattern enter 0, 0 through the keyboard and press Enter.
	**Zoom picture** • Select View →Zoom all to expand the view. The part has now been defined; your drawing should appear as shown.
	**Define blank shape** • Select Draw → Define blank. Click on Select contour button in the dialog box that appears. • At the prompt select a contour click on the 200 x 150 rectangle drawn earlier. • Enter the following data in the dialog box that appears and click on OK. Blank thickness = 80.0 Z coordinate at bottom = -75.0

	The Minimum and Maximum X, Y coordinates are picked up automatically from the selected rectangle. Leave them unaltered.
	The blank definition is optional. If it is not defined, it is automatically assumed as a rectangular block surrounding the area that is cut.
	Steps in performing a machining operation: Select the operation Select the tool Specify the area to be machined Specify the machining parameters You will find this sequence of steps repeated for every operation
	**Machining** Get into the machining mode by clicking on the Machining button at the bottom left of the screen.





Face milling

Select the operation

- Select Machining > Milling > Face mill.

Select the tool

- Click on New tool.
- In the Select tool type list, select a face mill type with a square shoulder.
- In the Select tool list, click on the 125.0mm. dia. face mill.
- Click on OK.

Specify the area to be machined

- At the prompt Select a contour click on the rectangle, which was defined as the blank earlier.
- Select Rectangle-Center point, length and breadth from the select entity list and click on done.

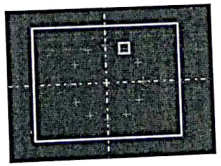
Specify the machining parameters

A machining parameters dialog box appears with default values.

- Enter 5 for Material thickness.
- Click on OK.
- At the prompt Select a circle to specify the machining start point click on the top right circle.

While defining the blank you specified the blank thickness as 80.0 and Z coordinate at bottom as -75.0 The extra 5.0 material is removed in face milling, and the top of the face milled surface of the part becomes Z0 after this.

Default values and cutting parameters are automatically selected from the database for the selected tool type, tool material and work piece material.



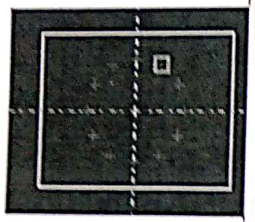
Drilling

- Select Machining → Hole → Drilling.
- Click on New tool. In the Tool type box expand the 'Drill' list and select 'Twist drill'.
- Select a 9.8mm dia. drill and click on OK.
- At the prompt Select the points to be machined click on any point on the point pattern, click on Done.

Since the hole is part of a pattern, all the points in the pattern get selected.

- Select No for the prompt Do you want to omit any holes?
- For Work surface Z enter 0, and for Hole depth enter 32.0.
- Click on OK.

For quick search, type the first numeral of the tool size you are searching.



Reaming

- Select Machining → Hole → Reaming
- Click on New tool. In the Tool type box expand the 'Reamer' list and select 'Reamer'.
- Click on a 10.0mm dia. reamer, HSS tool material and Click on OK.
- At the prompt Select the points to be machined click on any point on the point pattern, click on Done.
- Select No for the prompt Do you want to omit any holes?
- For Work surface Z enter 0 and for Hole depth enter 30.0 Click on OK.

Save the file.

Select a machine

Whatever you did so far (Work setup, Geometry definition, Machining) is independent of any machine.

Further functions like the NC program, cycle time, etc. require the machine to be selected since they use the machine's mechanical data like feed rate and spindle speed limits, spindle power, etc.

Select Machine.

Click on a machine from the list.

Click on Select.

View tool path

- Click on the Toolpath button at the bottom left of the screen.
- Click on to simulate till the end.
- Select to start simulation.
- After viewing the simulation try out the other simulation options:

Modes: Solid or line mode
Views: Isometric, XY, YZ or XZ.
Simulate till: Next motion, next tool, next operation or end of the program.

NC program

- Click on NC program on the menu bar. Click on OK in the dialog box that appears.
- Click on Edit to view the NC program.
- Select File → Exit to exit from the editor.

Power graph

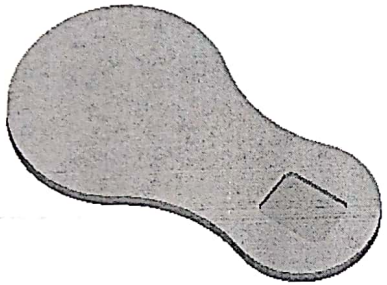
- Select Documents → Power.

To see the exact spindle power for an operation, move the cursor over its power bar.
If you think the spindle is being under or over utilized in any operation, you can change the operation's cutting parameters.
Double click on the power bar for the operation. Its cutting parameters are displayed.
When you change the parameters and exit from the dialog box, the new spindle power is displayed.

Click on Close to exit. If you have changed the cutting parameters, you can now choose to either retain the new values or discard them.

Problem – 2

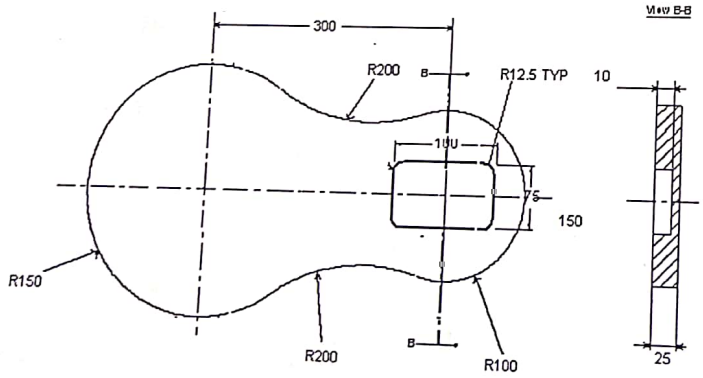
For the figure shown, perform the milling operation by CAPSMILL process

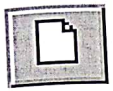
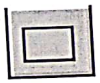


Objectives:

Face milling
Pocket milling
Side milling

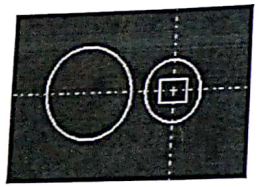
Part drawing



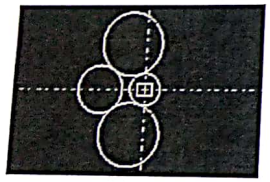
	**Start a new part** **Work setup** • For Units select 'MM'. • For Work piece material select any material from the list. Leave the rest of the data unaltered. • Enter suitable data in the Documentation page. Save the file with a suitable name.
	**Draw the part** Switch to the Geometry mode.
	**Draw the rectangular pocket** • Select Draw → Shapes → Rectangle (Center – Length – Width) • Enter the following parameters: Length = 100.0 Width = 75.0 Corner radius = 12.5 • Click on OK • At the prompt Specify the center point of the rectangle, enter the coordinates 0,0 through the keyboard and press the Enter key or Done.

Draw R100.0 and R150.0 circles

- Select Draw →Circle → Center - radius.
- At the prompt Specify the center point enter 0, 0 and press Enter.
- At the prompt Specify the radius enters 100 and press Enter.
- Similarly draw a circle with radius 150.0 at center point – 300.0, 0.



- Select View → Zoom all.
- The screen now looks like this.

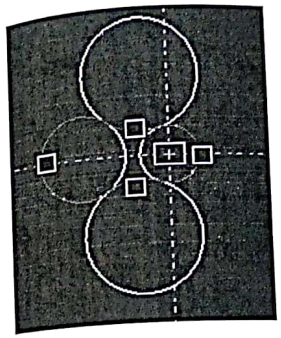


Draw R 200.0 circles tangential to these two circles

- Select Draw →Circle → Tangent – tangent – radius.
- At the prompts Select the object and Select next object, click on the upper halves of the circles.
- At the prompt Specify the Radius enter 200.0.

Similarly draw another tangential R 200.0 circle, this time by clicking on the lower halves of the circles.

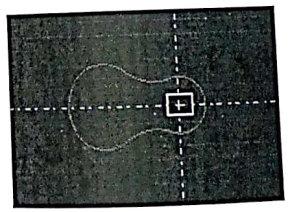
- Select View →Zoom all, to get all the objects within the screen.



Define the contour

Construct the required contour using these circles.

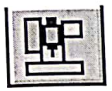
- Select Draw → Define contour → Interactive contour.
- At the prompt Specify the first object click on one of the circles.
- Click on the other circles at the locations shown each time the prompt Select the next object appears.
- Click on Yes at the prompt Do you want to save it as a contour?



Define the blank

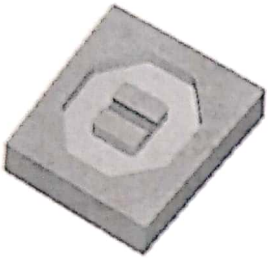
- Select Draw → Define blank.
- Click on Select contour and click anywhere on the contour that we defined earlier.
- Enter the data shown below and click on OK.
  Blank thickness = 30.0
  Z coordinate at bottom = - 25.0
- Save the file.

	**Machining operations** • Switch to the Machining mode
	**Face milling** Select the Face mill operation. • Select an appropriate face mill. • Select the blank contour as the area to be milled. • In the operation dialog box, for Process select Roughing, for Work surface Z enter 0, for Material thickness enter 5.0
	**Pocket milling** • Select the Pocket milling operation. • Select a 32.0 dia. end mill. • Select the small rectangle as the area for pocket milling. • In the operation dialog box, for Process select Roughing, for Work surface Z enter 0, for Pocket depth enter 10.0, for Side allowance after roughing enter 0.5.

	**Side milling – mill corners** • Select the Side mill operation. • Select a 25.00 dia. end mill. • Select the small rectangle as the machining area. • At the prompt Specify the machining start point click anywhere on the contour. • At the prompt Mill the whole contour? Click Yes. • At the prompt Select a circle to specify the side for milling click on the inside circle. • In the operation dialog box, for Process enter Side finish, for Work surface Z enter 0, for Material width enter 0.5, for Material depth enter 10.0. • In the Tool entry / exit tab, for Entry distance and Exit distance enter 5.0.
	**Select a machine**
	**View tool path** Switch to the Toolpath mode.
	**View NC program and other documents** • Generate and see the power graph, NC program, cycle time, tools list and tool layout sheet.

Problem – 3

For the figure shown, perform the milling operation by CAPSMILL process

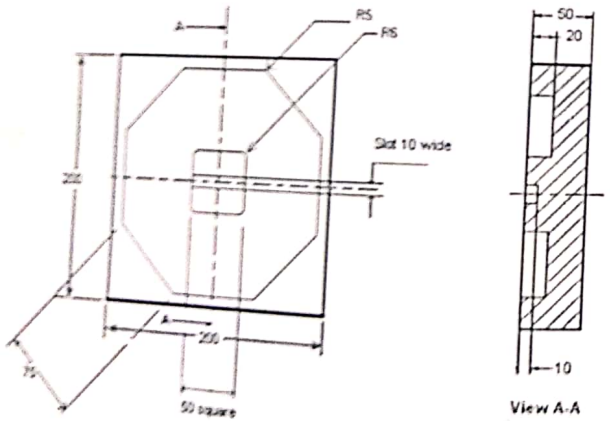


Objectives:

Milling a pocket with an island.

Milling a slot.

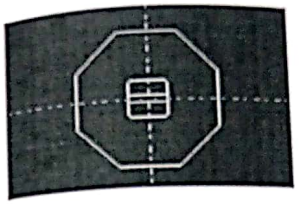
Part drawing



Start a new file

Define the work setup

Save the file



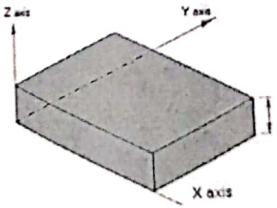
Draw part

- Draw the octagon using Draw → Shapes → Polygon.
- Draw the square using Draw → Shapes →Rectangle.
- Draw the slot using Draw → Shapes →Slot.

The part looks as shown in the figure.

Shapes drawn in CAPSmill through its standard drawing functions like rectangles, holes, polygons, pattern of holes are ready for machining and need not be defined as contours.

If you construct a part or a blank shape using lines, arcs and circles, you have to chain them as contours.



Draw blank

Define blank

- Draw the square for the blank.
- Define it as a blank with a suitable thickness and Z coordinate

	**Machine part** **Mill the pocket** • Check the prompt Do you want to specify islands ? and click on OK. • Select 50 X 50 rectangle for the prompt select islands • Press the right mouse button and select Done. **Mill the slot** • Select Machining → Milling → Slot mill.
	**View tool path** • Switch to the Toolpath mode.
	**View NC program and shop documents** • Generate and see the power graph, NC program, cycle time, tools list and tool layout sheet.