Go, change the world

Not me But you

National Service Scheme

RV College of Engineering®

REGISTER NOW

UTSARGA MARATHON' 24

5 KM - ₹399/-

10 KM - ₹449/-

10 ACTIVITY POINTS

HELP A CHILD, CHANGE THE WORLD

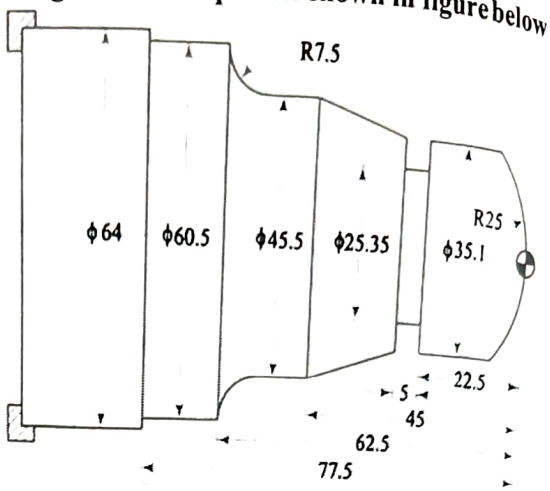
Includes

- T-shirt
- Certificate
- Medal
- Breakfast
- Refreshments

4TH FEBRUARY

nss.rvce | NATIONAL SERVICE SCHEME, RVCE | nss_rvce | National Service Scheme RVCE | NSS RVCE

Exercise No. 01: Write a manual part programming for the component shown in figure below



Operation	Tool	Machining Parameters			Remarks
		Speed (RPM)	Feed (mm/rev)	Depth of cut (mm)	
Rough Turning	Rough	1200	0.2	1	
Finishing Cycle	Lefthand	1500	0.15	0.25	
Grooving	External grooving tool (3mm)	800	0.1	2000 μ	

set in mm/rev

N05 G21 G99 ------ All dimension are set to mm & feed rate is set in mm/rev

N10 G28 U0 W0 ---- Tool moves to home position

N15 M06 T0101 ---- Tool selection: roughing tool is selected

N20 M03 S1200 ---- Spindle is turned on & set at 1200 rpm

N25 M07 ---- Coolant is switched on

N30 G00 X66 Z2 -- Cycle start point is set at (66, 2)

N35 G71 U1 R0.5

N40 G71 P45 Q90 U0.5 W0.2 F0.2 -- Roughing cycle parameters. Depth of cut = 1mm & retract amount is 0.5

N45 G00 X0 --- Finishing allowance in X & Z axis are 0.5 & 0.2. feed rate is 0.2 mm/rev

N50 G01 Z0

N55 G03 X35.5 Z-7.2 R25

N60 G01 X35.1 Z-27.5 -- Body of Program

N65 G01 X45.5 Z-45

N70 G01 X45.5 Z-55

N75 G02 X60.5 Z-62.5 R7.5

N80 G01 X60.5

N85 G01 X64 Z-77.5

N90 G01 X64 Z-95

N95 G28 U0 W0 --- ~~roughing tool~~ goes to home position

N100 M01 Optional Stop

N105 M06 T0202 -- grooving tool (3mm width) is selected from station 2

N110 G00 X36 Z-22.5 Tool is brought to grooving position

N115 [redacted] G75 R0.5 -- grooving parameters with position of groove.

N120 G75 X25.35 Z-27.5 P2000 Q2000 F0.1 S800 depth of cutter, each cycle feed rate, spindle feed

N125 G28 U0 W0 grooving tool goes to home position

N130 M01 Optional stop

N135 M06 T0303 Finishing tool is selected from station 3 (Type nose)

N140 G00 X66 Z2 cycle start point is set (66,2)

N145 G70 P45 Q90 F0.15 S1500 Finishing cycle parameters with feed rate and spindle speed

N150 G28 U0 W0 Lefthand tool moves to home position

N155 M05 spindle stop

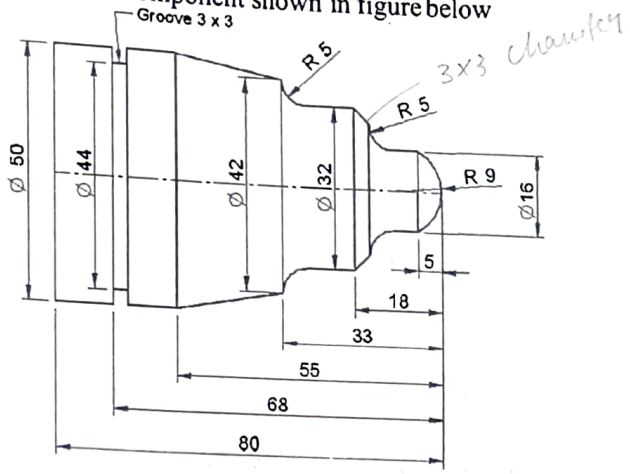
N160 M30 program stop and cursor moves to start block

9 / 10

			CO1	L1
			CO2	L2
			CO3	L3
			CO4	L4

Faculty Signatur...

Exercise No.02: Write a manual part programming for the component shown in figure below



Operation	Tool	Machining Parameters			Remarks
		Speed	Feed	Depth of cut mm	
Rough Turning	Rough	1200	0.2	1	
Finishing cycle	lefthand	1500	0.15	0.25	
Grooving	External grooving tool (3mm)	800	0.1	~~2000 rpm~~	

N05 G21 G99 All dimensions are set in mm & feed rate in mm/rev

N10 G28 U0 W0 Tool moves to home position

N15 M06 T0101 Rough tool is selected in turret station 1

N20 M03 S1200 spindle is turned on

N25 M07 Coolant is turned on

N30 G00 X52 Z2 Cycle start point is set at (52,2)

N35 G71 U1 R0.5 Roughing parameters are set Depth of cut = 1mm, retract amount = 0.5 mm. Finishing allowances in X & Z axis = 0.5 & 0.2. Feed rate is 0.2 mm/rev

N40 G71 P45 Q90 U0.5 W0.2 F0.2

N45 G00 X0

N50 G01 Z0

N55 G03 X16 Z-5 R8

N60 G01 X16 Z-10

N65 G02 X26 Z-15 R5

N70 G01 X32 Z-18

N75 G01 X32 Z-28

N80 G02 X42 Z-33 R5

N85 G01 X50 Z-55

N90 G01 X50 Z-85

Body of Program.

N95 G28 U0 W0 Tool returns to home position

N100 M01 Optional Stop

N105 M06 T0202 grooving tool is selected in turret station 2

N110 G00 X52 Z-65 Cycle start point is set at (52,-65)

groove parameters are set: position of groove, depth of cutter, each cycle feed rate, spindle feed

N115 G75 R0.5

N120 G75 X44 Z-65 P2000 Q0 F0.1 S800

N125 G28 U0 W0 tool moves to home position

N130 M01 Optional Stop

N135 M06 T0303 Finishing (Right hand) tool is selected from station 3

N140 G00 X52 Z2 cycle start point is set (52,2)

Finishing parameters are set

N145 G70 P45 Q90 F0.15 S1500 feed rate & spindle speed

N150 G28 U0 W0 tool returns to home position

N155 M05 Spindle Stop

N160 M30 cursor Stop & cursor moves to start block

10

10

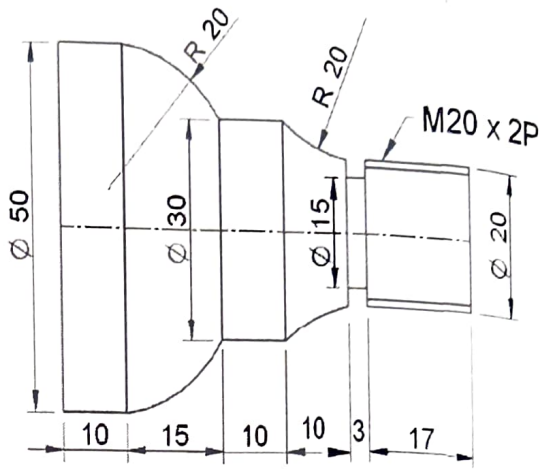
	Diagram: 1M Procedure:2M Writing Program: 5M Simulation the program and print out: 2M	10 M	CO1 CO2 CO3 CO4	L1 L2 L3 L4

Faculty Signature with Date

12/2/24

Exercise No. 03: Write a manual part programming for the component shown in figure below

Show threading calculations here.



Operation	Tool	Machining parameters			Remarks
		Speed	Feed	Depth of Cut	
Rough Turning	Rough	1200	0.2	1	
Finishing Cycle	Left hand	1500	0.15	0.25	
Grooving Cycle	External grooving tool	600	0.1	2000 μ	
Threading	External threading tool	300	2	200 μ	

N05 G21 G99 All dimensions are set in mm & feed rate in mm/rev

N10 G28 U0 W0 tool moves to home position

N15 M06 T0101 Rough tool is selected from station 1

N20 M03 S1200 Spindle start & speed is set at 1200 rpm

N25 M01 Coolant is switched on

N30 G00 X52 Z2 Cycle start point is set at (52,2)

N35 G71 U1 R0.5 Roughing parameters are set. Depth of cut = 1mm retract amount = 0.5

N40 G71 P45 Q70 U0.5 W0.2 F0.2 Finishing allowance in X & Z axis is 0.5 & 0.2. Feed rate is 0.2 mm/rev

N45 G00 X0 Z0

N50 G01 X120 Z-20

N55 G02 X30 Z-30 R20

N60 G01 X30 Z-40

N65 G03 X50 Z-55 R20

N70 G01 Z-65

Body of program

N75 G28 U0 W0 Tool moves to home position

N80 M01 Optional Stop

N85 M06 T0202 Finishing (left hand) tool is selected from station 2

N90 G00 X52 Z2 cycle start point is set (52,2)

N95 G70 P45 Q70 F0.15 S1500 Finishing parameters are set spindle speed & feed rate

N100 G28 U0 W0 Tool returns to home position

N105 M01 Optional Stop

N110 M06 T0303 grooving tool (3mm width) is selected from station 3

N115 G00 X22 Z-17 cycle start point is set (22,-17)

grooving parameters are set. position of groove,

N120 G75 R0.5 depth of cutter, each cycle feed rate & spindle speed

N125 G75 X15 Z-17 P2000 Q0 F0.1 S600

N130 G28 U0 W0 tool moves to home position

N135 M01 Optional Stop

N140 M06 T0404 threading tool is selected

N141 M03 S300

N145 G00 X20 Z10 threading position is set to (20,10)

N150 G76 P051060 Q100 R0.15

N155 G76 X17.548 Z-17 P1226 Q200 F2

N160 G28 U0 W0 tool returns to home position

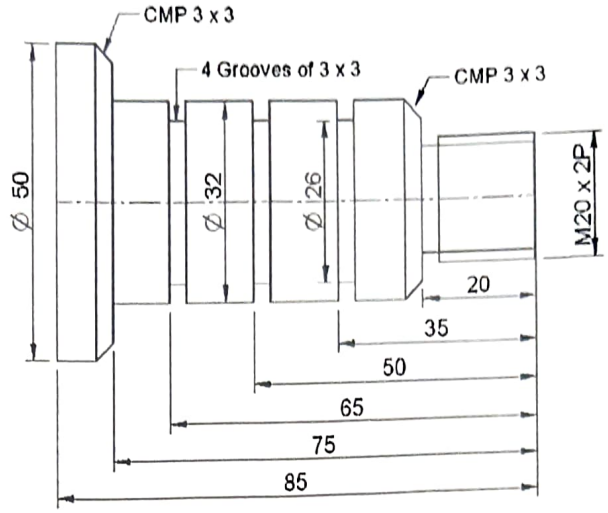
N165 M05 Spindle Stop

N170 M30 cursor stops & cursor returns to start block

8/10

	Diagram: 1M Procedure:2M Writing Program: 5M Simulation the program and print out: 2M	10 M	CO1 CO2 CO3 CO4	L1 L2 L3 L4

Exercise No. 04: Write a manual part programming for the component shown in Figure below



Operation	Tool	Machining Parameters			Remarks
		Speed	Feed	Depth of Cut	
Rough Turning	Rough	1200	0.2	1	
Finishing Cycle	Tejprand	1500	0.15	0.25	
Grooving	External grooving tool	600	0.1	2000 μ	
Threading	External threading tool	300	2	200 μ (Inch) 100 (Min)	

N05 G21 G99 All dimension set in mm & feed rate in mm/rev

N10 G28 U0 W0 tool moves to home position

N15 M06 T0101 Rough tool is selected from station 1

N20 M03 S1200 Spindle on & speed set to 1200 rpm

N25 M07 Coolant on

N30 G00 X52 Z2 cycle start position is set to (52, 2)

N35 G71 U1 R0.5 Roughing parameters: depth of cut = 1mm retract amount 0.5 : Finishing allowance in X & Z axis = 0.5 & 0.2 feed rate = 0.2 mm/rev

N40 G71 P45 Q80 U0.5 W0.2 F0.2

N45 G00 X20 Z0

N50 G01 Z-20

N55 G00 X26

N60 G01 X32 Z-23

N65 G01 Z-75

N70 G00 X44

N75 G01 X50 Z-78

N80 G01 Z-85

Body of Program

N85 G28 U0 W0 Tool returns to home position

N90 M01 Optional Stop

N95 M06 T0202 Finishing tool is selected from station 2

N100 G00 X52 Z2 cycle start position is set to (52, 2)

N105 G70 P45 Q80 F0.15 S1500 Finishing parameters: feed rate & spindle speed

N110 G28 U0 W0 tool returns to home position

N115 M01 Optional stop

N120 M06 T0300 Grooving tool is selected from 3rd station

N125 G00 X33 Z-20 grooving position is set (33,-17)

N130 G75 X17 Z-20 P2000 Q0 F0.1 S600 grooving parameters

position, depth of cut, distance b/w grooves

N135 G00 X33 Z-35 feed rate & spindle speed

N140 G75 R0.5 Relief clearance is set to 0.5

N145 ~~G00 X33 Z-62~~ G75 X26 Z-35 P2000 Q0 F0.1 S600

position, depth of cut, distance b/w grooves

N150 ~~G00~~ X33 Z-65 feed rate & feed speed

N155 G75 R0.5 Relief clearance is set to 0.5

N160 G75 X26 Z-65 P2000 Q0 F0.1 S600

N165 G28 U0 W0 tool returns to home position

N170 M01 Optional Stop

N175 M06 T0404 Threading tool is selected from station 4

N176 M03 S300

N180 G00 X20 Z10 threading position is set (20,10)

threading parameters are

N185 G76 P05 1060 Q100 R0.15 set

N190 G1.76 X 17.548 Z-17 P1226 Q200 F2

N195 G28 U0 W0 tool returns to home position

N200 M05 Spindle stop

N205 M30 Cursor Stop & Cursor returns to start

N161 G00 X33 Z-75 position, depth of cut, block

N162 G75 R0.5 feed rate & spindle speed, relief

163 G75 X26 X-75 P2000 Q0 F0.1 S600

89

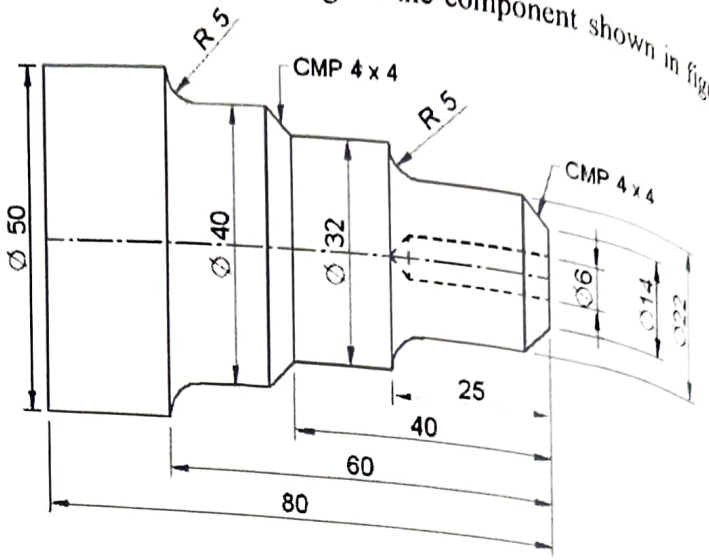
10

	Diagram: 1M Procedure:2M Writing Program: 5M Simulation the program and print out: 2M	10 M	CO1 CO2 CO3 CO4	L1 L2 L3 L4

Faculty Signature with Date

Keshav 12/2/24

Exercise No.05: Peck Drilling - Write a manual part programming for the component shown in figure below



Operation	Tool	Machining Parameters			Remarks
		Speed	Feed	Depth of Cut	
Rough Turning	Rough	1200	0.2	1	
Finishing Cycle	Lefthand	1500	0.15	0.25	
Drilling	Drill (dia 6mm)	600	0.15	2000μ	

N05 G21 G99 All dimensions are set to mm & feed rate in mm/min

N10 G28 U0 W0 tool moves to home position

N15 M06 T0101 Rough tool is selected from station 1

N20 M03 S1200 Spindle is switched on & speed is set at 1200 rpm

N25 M07 Coolant is switched on.

N30 G00 X52 Z2 cycle start position is set (52,2)

N35 G71 U1 R0.5 Roughing parameters are set Depth of cut 1mm retract amount = 0.5. Finishing allowance in X & Z axis is 0.5 & 0.2

N40 G71 P45 Q85 U0.5 W0.2 F0.2 Feed rate = 0.2 mm/rev

N45 G00 X14 Z0

N50 G01 X22 Z-4

N55 G01 X22 Z-20

N60 G02 X32 Z-25 R5 } Body of program.

N65 G01 X32 Z-40

N70 G01 X40 Z-44

N75 G01 X40 Z-55

N80 G02 X50 Z-60 R5

N85 G01 X50 Z-80

N90 G28 U0 W0 Tool moves to home position

N95 M01 Optional stop

N100 M06 T0202 Finishing (left hand) tool is selected from station 2

N105 G00 X52 Z2 cycle start point is set (52,2)

Finishing parameters are set

N110 G70 P45 Q85 F0.15 S1500 feed rate & spindle speed.

N115 G00 U0W0 tool returns to home position.

N120 M01 Optional stop

N125 M06 T0505 Drilling tool (6mm dia) is selected from station 5

N130 G00 X0 Z2 drilling start point is set (0,2)

N135 G74 R0.5 drilling parameters are set, drilling depth, feed rate, spindle speed and relief amount at end of cut

N140 G74 X0 Z-25 Q2000 F0.15 S600

N145 G28 U0W0 tool returns to home position

N150 M05 Spindle Stop

N155 M30 Cursor Stop & Cursor returns to start block

10

10

	Diagram: 1M Procedure:2M	10 M	CO1 CO2	L1 L2