Seat No.: _____ Enrolment No._____

GUJARAT TECHNOLOGICAL UNIVERSITY

BE - SEMESTER–VII(OLD) • EXAMINATION – WINTER 2016

Subject Code: 172501 **Date: 29/11/2016**

Subject Name: Computer Aided Manufacturing

Time: 10:30 AM to 01:00 PM **Total Marks: 70**

Instructions:

1. **Attempt all questions.**
2. **Make suitable assumptions wherever necessary.**
3. **Figures to the right indicate full marks.**

Q.1 **(a)** Explain robot configurations with neat sketches. **07**

(b) Write short notes on: (i) Robot specifications (ii) End-effectors in robots. **07**

Q.2 **(a)** Define computer integrated manufacturing (CIM) .Explain the different elements of CIM. **07**

(b) Discuss the salient features of point to point, straight line and contouring CNC systems with neat sketches. **07**

OR

(b) Explain in detail the applications of CNC technology in manufacturing. **07**

Q.3 **(a)** What are advantages of LM guideways? With the help of neat sketches, explain the methods of eliminating backlash in recirculating ballscrews. **07**

(b) Write the manual part programme for the part shown in Figure-1 using single pass canned cycle. Assume that the raw material is cylindrical blank of size Ø 45 mm x 70 mm. Refer to Table -1 for G-codes & M-codes. Use following process plan. **07**

Operation No.	Operation	Tool No.	Cutting Speed (m/min)	Feed Rate (mm/rev)	Depth of Cut (mm)
10	Facing	01	175	0.15	-
20	Rough turning	03	175	0.18	4 (max.)
30	Finish turning	03	200	0.12	1.5(max.)

OR

Q.3 **(a)** What is meant by adaptive control? Explain any one type of adaptive control. What are the advantages of adaptive control? **07**

(b) What are the advantages of sub programming? What care must be taken while using subprogramming facility? List the major operations/functions that can be performed using subprogramming facility. **07**

Q.4 **(a)** Explain Co-ordinate Measuring Machine. **07**

(b) The part shown in the figure-2 requires 10 mm diameter holes in 50 mm thick plate of size 400 mm x 250 mm. Write a part program to drill the holes using deep hole peck drilling cycle. Refer to Table -1 for G-codes & M-codes. Use following process plan. **07**

Operation No.	Operation	Tool	Cutting Speed (m/min)	Feed Rate (mm/min)	Depth of Cut (mm)
10	Drill Ø10 mm	01	30	40	-

OR

Q.4 **(a)** What is parity checking? Explain representation of ISO and EIA numerical control **07**

codes in punched tape.

(b) Explain Do loop and Macro facilities of programming with suitable examples. **07**

Q.5 **(a)** Explain different types of automated guided vehicles. How traffic is controlled in AGV? Explain the safety features in AGV systems. **07**

(b) What are the reasons for using rapid prototyping? Explain the stereolithography in detail with sketches. **07**

OR

Q.5 **(a)** Give advantages, limitations and applications of FMS. **07**

(b) What is the methodology of rapid prototyping? Explain the laminated object manufacturing in detail with sketches. **07**

****

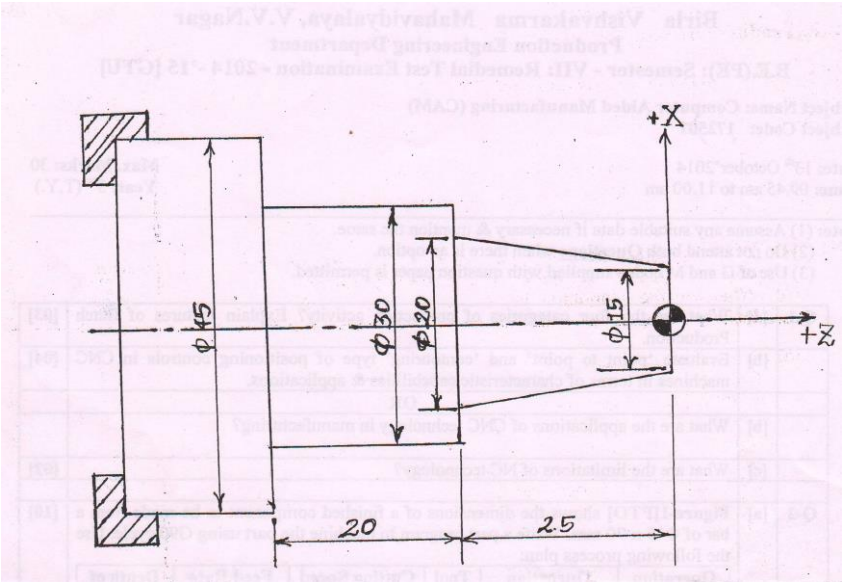

Figure – 1

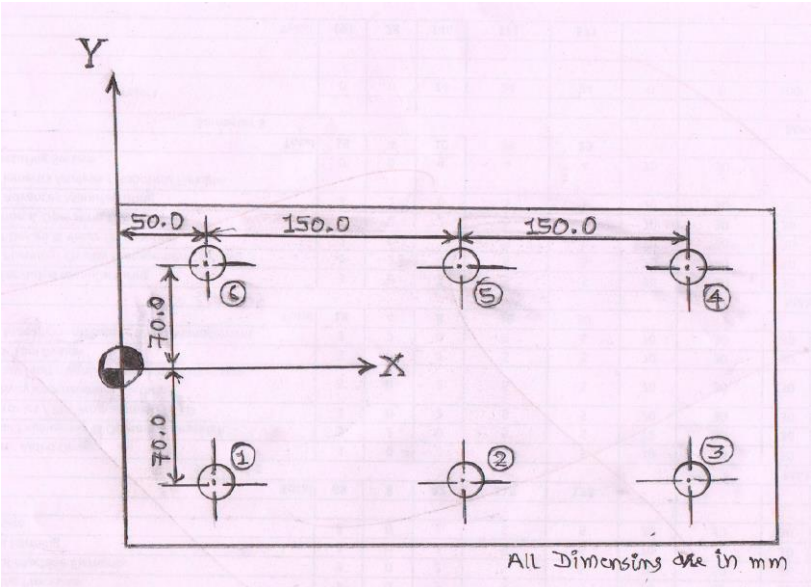

Figure – 2

Table – 1

NC/CNC Manual Part Programming Codes

Turning Center FANUC OT Controller		Machining Center FANUC OM Controller	
G00	Positioning in Rapid	G00	Positioning in Rapid
G01	Linear Interpolation	G01	Linear Interpolation
G02	Circular Interpolation (CW)	G02	Circular Interpolation (CW)
G03	Circular Interpolation (CCW)	G03	Circular Interpolation (CCW)
G04	Dwell	G04	Dwell
G20	Inch Units Programming	G17	XY Plane
G21	Metric Units Programming	G18	XZ Plane
G28	Automatic return to reference point	G19	YZ Plane
G29	Automatic return from reference point	G20	Inch Units Programming
G40	Cutter compensation Cancel	G21	Metric Units Programming
G41	Cutter compensation Left	G28	Automatic return to reference point
G42	Cutter compensation Right	G29	Return from Zero Return Position
G43	Tool Length Compensation (Plus)	G40	Cutter compensation Cancel
G44	Tool Length Compensation (Minus)	G41	Cutter compensation Left
G49	Tool Length Compensation Cancel	G42	Cutter compensation Right
G50	Maximum spindle speed in rpm	G43	Tool Length Compensation (Plus)
G96	Constant surface speed ON	G44	Tool Length Compensation (Minus)
G97	Constant surface speed OFF	G49	Tool Length Compensation Cancel
G98	Feed rate, per min	G90	Absolute Positioning/ Programming
G99	Feed rate, per rev.	G91	Incremental Positioning/ Programming
		G94	Feed rate, per min
		G95	Feed rate, per rev.
		G98	Return to initial point level
		G99	Return to R point level
	Canned Cycles		**Canned Cycles**
G90	Single Pass Turning Cycle	G73	High Speed Peck Drilling Cycle
G92	Single Pass Threading Cycle	G74	Left hand Tapping Cycle
G94	Single Pass Facing Cycle	G76	Fine Boring Cycle
G70	Finishing Cycle	G80	Cancel Canned Cycles
G71	Turning Cycle	G81	Spot Drilling Cycle
G71	Multi-pass Rough Turning Cycle	G82	Counter Boring Cycle
G72	Multi-Pass Rough Facing Cycle	G83	Deep Hole Peck Drilling Cycle
G73	Pattern Repeating	G84	Right hand Tapping cycle
G74	Grooving in Z-axis	G85	Boring Cycle
G75	Grooving in X-axis		
G76	Multi-Pass Thread Cutting		

Miscellaneous Codes

M00	Program Stop	M06	Tool Change
M01	Optional Program Stop	M98	Subprogram Call
M02	Reset	M99	Subprogram End
M03	Spindle On/start Clockwise	M70	Mirror On – X
M04	Spindle On/start Counterclockwise	M71	Mirror On – Y
M05	Spindle Stop	M80	Mirror Off – X
M08	Coolant On	M81	Mirror Off – Y
M09	Coolant Off	M30	Program End & Rewind
M30	Program End & Rewind	M00 to M09	– same as Turning Center codes