Seat No.: _____ Enrolment No._____

GUJARAT TECHNOLOGICAL UNIVERSITY

B. E. - SEMESTER – VII • EXAMINATION – WINTER 2012

Subject code: 171903 Date: 01/01/2013

Subject Name: Computer Integrated Manufacturing

Time: 10.30 am - 01.00 pm Total Marks: 70

Instructions:

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

Q.1	**(a)**	What are the basic components of numerical control system Draw and discuss function of each component.	**04**
	(b)	Explain the term "Work Volume "with respect to robots.	**03**
	(c)	Enlist benefits of flexible manufacturing system.	**03**
	(d)	Process rationalization and standardization is a benefit derived from CAPP. Explain.	**04**
Q.2	**(a)**	Why is part classification and coding required in GT. Explain OPTIZ system of coding .	**07**
	(b)	Explain with neat sketch AS/RS system used in FMS	**07**
		OR	
	(b)	Explain the types of flexibilities in FMS and discuss the factors on which these flexibilities depend.	**07**
Q.3	**(a)**	List the various models of CIM. Explain anyone in detail.	**07**
	(b)	Explain with neat diagram retrieval type CAPP.	**07**
		OR	
Q.3	**(a)**	What is CIM. Discuss the components of CIM.	**07**
	(b)	Explain with neat diagram generative type CAPP.	**07**
Q.4	**(a)**	Draw neat sketches and explain various robot configurations.	**07**
	(b)	Discuss various applications of robots.	**07**
		OR	
Q.4	**(a)**	Explain the commonly used robot programming languages giving examples.	**07**
Q.4	**(b)**	Draw a vertically articulated robot. Explain its working showing work envelope, degree of freedom and discuss its applications.	**07**
Q.5	**(a)**	What are canned cycles? What is the difference between a canned cycle and subroutine ? Discuss how a canned cycle can be useful in writing a part program .	**07**
	(b)	Write a part program for component shown in Figure.1.	**07**
		OR	
Q.5	**(a)**	What are the different geometric statements used to write part program using APT? Discuss each statement with a suitable example	**07**
	(b)	Explain with the help of part programming , the concept of subroutine and do loops for programming the component shown in Figure.2	**07**

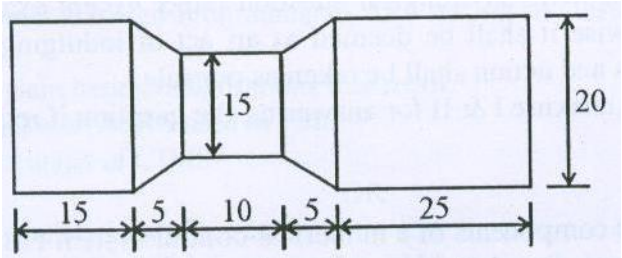

Figure 1
Dimensions are in mm.

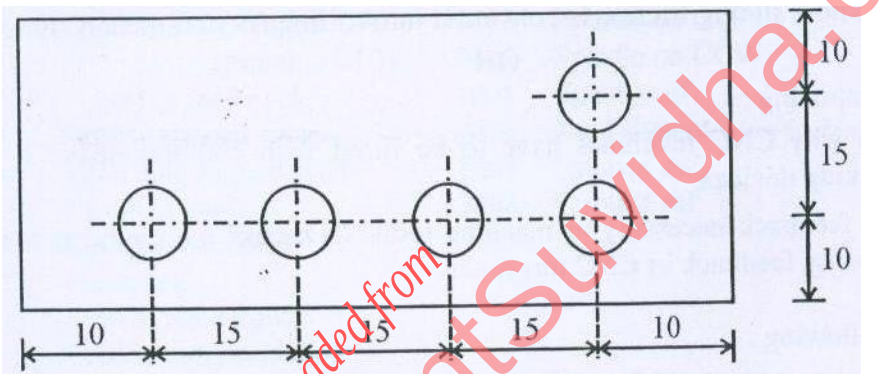

Figure 2
Dimensions are in mm.

Annexture - (I)

MISCELLANEOUS (M) FUNCTIONS

Following is a list of miscellaneous functions used in the milling and turning examples in this text. Other M functions common to General Numeric and FANUC controllers are also listed.

M00—Program stop.
M01—Optional stop.
M02—End of program (rewind tape).
M03—Spindle start clockwise.
M04—Spindle start counterclockwise.
M05—Spindle stop.
M06—Tool change.
M08—Coolant on.
M09—Coolant off.
M13—Spindle on clockwise, coolant on (on some systems).
M14—Spindle on counterclockwise, coolant on.
M17—Spindle and coolant off (on some systems).
M19—Spindle orient and stop.
M21—Mirror image X axis.
M22—Mirror image Y axis.
M23—Mirror image off.
M30—End of program, memory reset.
M41—Low range.
M42—High range.
M48—Override cancel off.
M49—Override cancel on.
M98—Jump to subroutine.
M99—Return from subroutine.

Annexture - (II)

PREPARATORY FUNCTIONS (G CODES)

Following is a list of preparatory functions used in CNC milling examples in this text. Other codes commonly used on General Numeric controllers are also listed.

G00—Rapid traverse positioning.
G01—Linear interpolation (feedrate movement).
G02—Circular interpolation clockwise.
G03—Circular interpolation counterclockwise.
G04—Dwell.
G10—Tool length offset value.
G17—Specifies X/Y-plane.
G18—Specifies X/Z plane.
G19—Specifies Y/Z plane.
G20—Inch data input (on some systems).
G21—Metric data input (on some systems).
G22—Safety zone programming.
G23—Cross through safety zone.
G27—Reference point return check.
G28—Return to reference point.
G29—Return from reference point.
G30—Return to second reference point.
G40—Cutter diameter compensation cancel.
G41—Cutter diameter compensation left.
G42—Cutter diameter compensation right.
G43—Tool length compensation positive direction.
G44—Tool length compensation negative direction.
G45—Tool offset increase.
G46—Tool offset decrease.
G47—Tool offset double increase.
G48—Tool offset double decrease.
G49—Tool length compensation cancel.
G50—Scaling off.
G51—Scaling on.
G73—Peck drilling cycle.
G74—Counter tapping cycle.
G76—Fine boring cycle.
G80—Canned cycle cancel.
G81—Drilling cycle.
G82—Counter boring cycle.
G83—Peck drilling cycle.
G84—Tapping cycle.
G85—Boring cycle (feed return to reference level).
G86—Boring cycle (rapid return to reference level).
G87—Back boring cycle.
G88—Boring cycle (manual return).
G89—Boring cycle (dwell before feed return).
G90—Specifies absolute positioning.
G91—Specifies incremental positioning.
G92—Program absolute zero point.
G98—Return to initial level.
G99—Return to reference (R) level.