

Time: Two (02) Hours

Total 04 Questions  
Answer all questions

1.

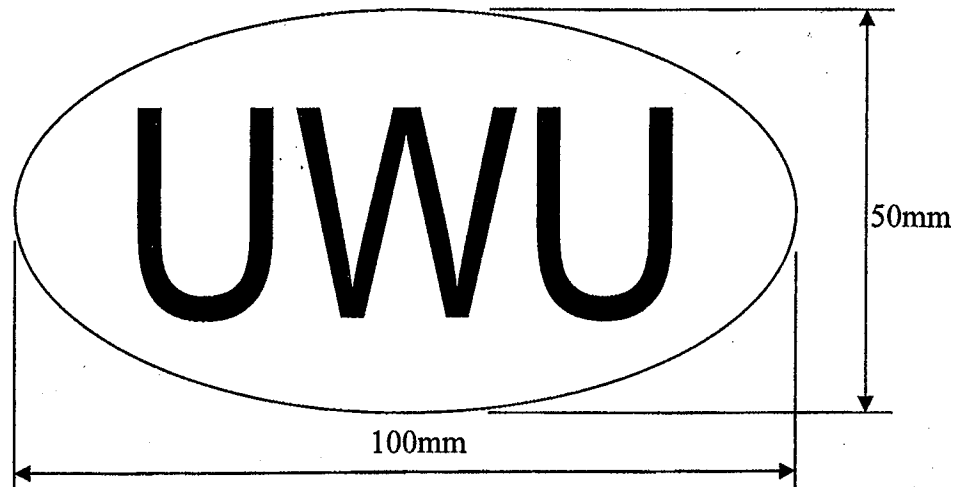


Figure Q01

Figure Q01 above shows a medal. These letters are to be engraved on a CNC machining center. You do not have to machine the outer ellipse. Assume all the missing dimensions and write a G – Code program to cut the three letters. Single line cut is sufficient. List of G – Codes are given at the end of the question paper. Indicate the details of the tool, coordinate systems and the selected dimensions for letters.

(100 marks)

2.

- What are the advantages of sequential product development process?  
(25 marks)
- What can be the reason that some of the sequential product development processes have not yet transformed into integrated development process?  
(25 marks)
- What are the characteristics of the integrated product development process?  
(25 marks)
- What is concurrent engineering?  
(25 marks)

3.

- a. What is the Needs-Metrics Matrix? What is the use of it? (25 marks)
- b. What are the reasons to use a CAD system? (25 marks)
- c. Compare the features used in a CAD system with the features in a CAM system. Explain your answer using drawings. (50 marks)

4.

- a. What is a Variant Process Plan? Explain using an example. (25 marks)
- b. How does a General Process Plan differ from a Variant Process Plan? (25 marks)
- c. Explain three solid modeling representations using a simple object. (50 marks)

M-Code	Function
M0	Program stop.
M2	End of program.
M3	Spindle ON FWD.
M4	Spindle ON REV.
M5	Spindle OFF.
M8	Coolant ON.
M9	Coolant OFF.
M30	Jump to new program.
M98	Call subprogram.
M99	End subprogram.
M100	Mirror image.
M105	Dry-run, all axes.
M106	Dry-run, NO Z axis.
M107	Dry-run off (cancels M105 or M106).

Modal		Non-Modal	
G-Code	Function	G-Code	Function
G0	Positioning-Rapid Traverse	G4	Dwell
G1	Linear Interpolation-Feed	G5	Ellipse
G2	Circular Interpolation-CW	G9	Exact Stop Check
G3	Circular Interpolation-CCW	G28	Return to Machine Home
G22	Stored Stroke Limit ON	G29	Return from Machine Home
G40	Tool Radius Compensation, Cancel	G31	Probe Move
G41	Tool Radius Compensation (Left)	G45	Mold Rotation
G42	Tool Radius Compensation (Right)	G49	Elbow Milling
G53	Work Coordinate System	G62	Automatic Feed Override for Arcs
G59	Modal Corner Rounding	G63	Automatic Feed Override for Arcs Cancel
G60	Modal Corner Rounding Off	G65	User Macro Single Call
G61	Exact Stop Check Mode	G66	User Macro Modal Call
G64	Cutting Mode (Continuous Path ON)	G67	User Macro Modal Call Cancel
G66	User Macro Modal Call	G68	Coordinate System Rotation
G67	User Macro Modal Call Cancel	G73	Draft Pocket Milling Cycle
G68	Coordinate System Rotation	G75	Frame Milling
G70	Inch Programming	G76	Hole Milling Cycle
G71	MM Programming	G77	Circular Pocket Cycle
G72	Axis Scaling	G78	Rectangular Pocket Cycle
G90	Absolute Programming	G79	Bolt Hole Circle Cycle
G91	Incremental Programming	G80	Cancel Modal Drilling