EML2322L – MAE Design and Manufacturing Laboratory

CNC Mill Assessment

Name: _____ **Grader's Initials: _____**

Description: This assessment gages your understanding of the material covered in the CNC Mill Training Resources guide. **Correct any T/F questions which are FALSE** and ask Mike questions you have regarding this material.

1. A good quality drawing must be made of the part that includes appropriate tolerances for each feature, notes about which surfaces need finishing, how good the finishes must be, and allowable (maximum) fillet sizes on relevant features. T / F
2. You must complete a setup sheet for the machine you plan to use to manufacture your part. T / F
3. HSS tools are approximately the same cost as WC tools. T / F
4. WC tools are tougher than HSS tools. T / F
5. Once endmills reach ½″ in diameter, they are typically strong enough to cut anything we need to, and at that point larger tools just cost more money without much gain in strength / stiffness. T / F
6. Always select the largest L:D ratio possible for peak productivity, tool life, and surface finish. T / F
7. Do not use more than 5 flutes when full slotting in non-ferrous materials like aluminum. T / F
8. Roughing tools are stronger than finishing tools because they have generous fillets or chamfers on their cutting tips and serrated edges to break chips into smaller pieces for improved evacuation. T / F
9. WC can withstand approximately 2.5X more heat and is 2.5X stiffer than HSS alloys. T / F
10. HSS can only be ground so sharp, while WC can be honed sharper than HSS. T / F
11. TiN and TiAlN coatings are intended for cutting aluminum, as they reduce galling. T / F
12. A 45° helix tool is less likely to get pulled out of an ER-32 collet than a 35° helix tool. T / F
13. Endmills are available with flat ends, ball ends, and convex radii corners. T / F
14. If cutting a feature that requires a longer endmill, use a normal length tool first and then switch to the longer tool(s) as necessary. T / F
15. All endmills are center-cutting. T / F
16. Feeding an endmill too slowly is as bad for it as feeling it too quickly. T / F
17. If you cut twice as deep, the forces are twice as large. This means you must be more careful to ensure the part is clamped securely when taking deeper axial cuts with the side of an endmill, even if only removing a small amount of material. T / F
18. Do not open the tooling cabinets (to see what's available for use or to remove a tool) without first asking Mike's permission, as most of the tools belong to DML and the rest are under Mike's supervision so they last for more than one use. T / F

19. NEVER touch the tapered portion of a toolholder, as doing so causes corrosion that permanently degrades its precision. T / F

20. Always wipe off the taper with a clean rag, remove any corrosion from the taper using a piece of Scotch-Brite, spray a light coating of WD-40 on the freshly cleaned taper, and place a small dab of grease on the pull stud bulb. T / F

21. Always select the stiffest toolholder available that provides adequate working clearance. T / F

22. When installing tools in ER-style toolholders, always load the collet into the collet chuck BEFORE installing the collet nut onto the toolholder and tightening it to the correct torque or you will damage the collet, nut, and toolholder. T / F

23. When installing a toolholder into the CNC, always rotate the spindle so the toolholder engagement tangs are closest to the operator and visually check that they engage their mating slots in each toolholder BEFORE releasing the toolholder clamping button. T /F

24. Be VERY CAREFUL when inserting a toolholder into the spindle TO NOT bump the pull stud into the side of the precision ground taper. T / F

25. Probe each tool after you finish loading all the tools necessary to make your part. T / F

26. Failure to unload tools, return tools to their appropriate containers, and return toolholders/collets to their respective carts will result in suspension of CNC use privileges. T / F

27. Helical ramping is better than pre-drilling. T / F

28. High efficiency milling places large axial forces on the cutting tool and workpiece. T / F

29. Using a long tool for an entire pocket (deeper than 2×D) dramatically reduces the metal removal rate because of the large reductions in spindle speed and feedrate required to not destroy the tool. T / F

30. Reamers should be run at twice the speed and half the feedrate of a comparable size drill. T / F

31. Reamers should be retracted with the spindle off to preserve the finish and mitigate bell-mouthing of the hole entrance. T / F

32. Reduce calculated spindle speed by 25% for every tool diameter over 2×D cutting depth and only increase your spindle speed after verifying adequate cutting tool stiffness and chip evacuation. T / F

33. When ready to test a program, offset the Z-axis height value stored in the relevant work offset register by one inch (or more) in the NEGATIVE Z direction. T / F

34. For a first part run, set the SPINDLE SPEED and FEEDRATE overrides to 100% and 80%. T / F

35. For a first part run, set the RAPID override to 50% anytime you are within 6″ of the part. T / F

36. If ANYTHING is changed in the program, you must re-prove (i.e. perform another dry run of) the associated portion(s) of the program and tool(s) involved. T / F

37. It is okay to run the CNC while talking to or with another person as long as you are careful. T / F

38. ALWAYS remove the tool from the spindle before cleaning the machine. T / F

39. ALWAYS leave a tool in the spindle overnight, but never in carousel pocket #1. T / F

40. Calculate the speeds and feeds for the following tools when using a good quality CNC machine based on the information presented on the TA SPEEDS & FEEDS CALCULATIONS page. Show your work for each calculation on the space provided or on the backside of this page.

End Mills							
Tool Diameter [in]	**Tool Material**	**No. of Flutes**	**Part Material**	**Surface Speed [SFM]**	**Spindle Speed [RPM]**	**Chip Load [in/tooth]**	**Feed Rate [in/min]**
0.5	HSS	3	Aluminum				
0.5	WC	3	Aluminum				
0.5	WC	6	Mild Steel (.2-.3C)				
0.5	WC	6	Titanium				

Drills							
Tool Diameter [in]	**Tool Material**	**Depth**	**Part Material**	**Surface Speed [SFM]**	**Spindle Speed [RPM]**	**Feed Rate [in/rev]**	**Feed Rate [in/min]**
0.5	HSS	< 3xD	Aluminum				
0.5	HSS	< 3xD	Mild Steel (.2-.3C)				
0.5	HSS	> 3xD	Mild Steel (.2-.3C)				
0.5	HSS	< 3xD	Titanium				

Reamers							
Tool Diameter [in]	**Tool Material**	**-**	**Part Material**	**Surface Speed [SFM]**	**Spindle Speed [RPM]**	**Feed Rate [in/rev]**	**Feed Rate [in/min]**
0.5	HSS	-	Aluminum				
0.5	HSS	-	Mild Steel (.2-.3C)				
0.5	HSS	-	Titanium				