

Proficiency Test For Machining Center

Name: _____ Date: _____

Section One: General CNC Questions

- 1) The spindle speed for a particular tool in a program is incorrect and you wish to reduce it. The kind of CNC word you must change is:
 A) an F word C) an S word
 B) a G word D) a T word
- 2) You determine that a feedrate for a particular tool must be 5.0 inches per minute (or 5 millimeters per minute in the metric mode). The correct designation in the program would be:
 A) F0.5 C) F5.0
 B) S5.0 D) F0.005
- 3) You are running a proven program (one run before) for the first time in a new setup. You are cautiously allowing the first tool come into its approach position when you determine that the tool is not going where it is supposed to. It is most likely that:
 A) the spindle speed is not correct
 B) the program zero designation is not correct
 C) the dry run switch should be turned on
 D) the programmed coordinates are not correct
- 4) When coordinates going into a program are specified relative to the program zero point, it is called:
 A) the incremental mode
 B) the absolute mode
 C) the rapid mode
 D) the canned cycle mode
- 5) What mode switch position allows programs to be modified?
 A) Jog D) Edit
 B) Auto E) Zero return
 C) MDI (manual data Input)
- 6) What mode switch position allows a program to be run from within the CNC control?
 A) Jog D) Edit
 B) Auto E) Zero return
 C) MDI (manual data Input)
- 7) What mode switch position allows the activation of a single CNC command?
 A) Jog D) Edit
 B) Auto E) Zero return
 C) MDI (manual data Input)
- 8) What mode switch position allows you to send the machine to its reference (home) position?
 A) Jog D) Edit
 B) Auto E) Zero return
 C) MDI (manual data Input)
- 9) What mode switch position lets you move the machine axes manually?
 A) Jog D) Edit
 B) Auto E) Zero return
 C) MDI (manual data Input)
- 10) Which on/off switch works in conjunction with a slash code in the program?
 A) Dry Run D) Single Block
 B) Optional Stop E) Machine Lock
 C) Optional Block Skip/Block Delete
- 11) Which on/off switch works in conjunction with an M01 in the program?
 A) Dry Run D) Single Block
 B) Optional Stop E) Machine Lock
 C) Optional Block Skip/Block Delete
- 12) Which on/off switch makes the control activate one command in the program at a time and then stop?
 A) Dry Run D) Single Block
 B) Optional Stop E) Machine Lock
 C) Optional Block Skip/Block Delete
- 13) The Feedrate Override switch lets the operator control the machine's movement rate during cutting commands.
 True False
- 14) The button used to activate a CNC program is:
 A) Reset D) Cycle Start
 B) Emergency Stop E) Feed Hold
 C) Coolant On
- 15) The button used to temporarily stop the CNC cycle is:
 A) Reset D) Cycle Start
 B) Emergency Stop E) Feed Hold
 C) Coolant On
- 16) The button that will reactivate the CNC program after temporary stoppage is:
 A) Reset D) Cycle Start
 B) Emergency Stop E) Feed Hold
 C) Coolant On
- 17) The button that will clear data in the look-ahead buffer after a temporary program stoppage is:
 A) Reset D) Cycle Start
 B) Emergency Stop E) Feed Hold
 C) Coolant On

General CNC Questions (continued)

18) The button that will actually turn off the power to the machine tool is:

- A) Reset
- B) Emergency Stop
- C) Coolant On
- D) Cycle Start
- E) Feed Hold

19) The display screen mode that lets the operator actually see the CNC program is the:

- A) Offset page
- B) Position page
- C) Alarm page
- D) Program page
- E) Program Check page

20) The display screen mode that lets the operator enter tooling related data is the:

- A) Offset page
- B) Position page
- C) Alarm page
- D) Program page
- E) Program Check page

21) The display screen mode that lets the operator most easily see axis data is the:

- A) Offset page
- B) Position page
- C) Alarm page
- D) Program page
- E) Program Check page

22) The display screen page that will be automatically displayed when the CNC control determines a problem is the:

- A) Offset page
- B) Position page
- C) Alarm page
- D) Program page
- E) Program Check page

23) The display screen page that allows the operator to see a the *distance-to-go* in the current CNC command is the:

- A) Offset page
- B) Position page
- C) Alarm page
- D) Program page
- E) Program Check page

24) Say your machine does not allow manual control of the spindle, yet you must start the spindle during setup. You must:

- A) contact the machine tool builder to have them fix the machine
- B) write a CNC program that starts the spindle at your desired speed
- C) use the MDI mode to activate the spindle
- D) give up. There is no way to start the spindle

25) As part of the machine start up procedure, you must send the machine to its zero return (home) position.

- True
- False

26) Proven CNC programs (those run before) require no program verification procedures.

- True
- False

27) There is always some way to adjust offsets to ensure that every tool in every program will machine with a little excess stock,

meaning (if the program is correct) there is never an excuse to scrap the first workpiece in a production run.

- True
- False

28) How often should you check the way lubrication levels on your CNC machine?

- A) Once every hour
- B) Once every week
- C) Once every shift
- D) Once every month

29) While there is never a good excuse for a crash, if you do have a mishap and your machine crashes, you should:

- A) check visually to see that there is no damage and continue running production
- B) immediately notify your supervisor
- C) fix any damage and continue running production
- D) stop running production for the balance of your shift

30) As you are verifying a new program that has never run before, you find what you consider to be a serious mistake in the program's motions. You should:

- A) do your best to fix the mistake and continue
- B) notify your supervisor so the programmer can be contacted
- C) assume that the program has been well checked by the programmer and run the program as it is

31) When having a problem of almost any kind, it is often helpful to be able to look at a list of all currently instated commands. The display screen mode that allows this is:

- A) Program
- B) Program Check
- C) Offset
- D) Position

32) A program stop is encountered (M00) and the machine stops. However, you are not sure why the program stop is in the program. You should:

- A) press cycle start to resume the program's execution
- B) call up the PROGRAM display screen mode and see if there is a message telling you what to do near the M00
- C) turn on single block to cautiously step through the next few commands

33) While running a new program, the machine goes into alarm state and shows alarm number 41 with the message "Overcutting will occur during cutter radius compensation." You should:

- A) restart the program and try again
- B) turn on single block and cautiously run the next few commands
- C) consult the alarm list in the operation manual to further diagnose the alarm

34) After diagnosing an alarm it is possible that you must:

- A) change a tool offset value
- B) edit commands in the program
- C) press the reset button to cancel the alarm
- D) all of the above

Section Two: Machining Center Questions

- 35) Spindle speed for machining centers is always specified in:
- A) surface feet or meters per minute
 - B) RPM
 - C) inches per minute
 - D) inches per revolution
- 36) When viewing a vertical machining center from the front, the X axis is left/right, Y is fore/aft, and Z is up/down.
- True False
- 37) Many machining center setups require manual measurements to determine the program zero assignment values. These values almost always represent:
- A) are the combined lengths of all tools
 - B) the distance between the program zero point and the machine's zero return position in
 - C) the distance between program zero and the table top
 - D) the distance from the tool tip to the spindle nose
- 38) Once the program zero designation values are determined, they must be placed:
- A) in the program's G92 command
 - B) in the corresponding fixture offset
 - C) in the tool length compensation offsets
 - D) in the cutter radius compensation offsets
 - E) A or B, depending upon the method of program zero assignment
- 39) You need to manually make a tool change to place tool number one in the spindle. However, the machine has no manual buttons or switches for making tool changes. You must:
- A) contact the machine tool builder to fix the machine
 - B) write a short program to make the tool change
 - C) use the MDI mode to command the tool change using program-like commands
 - D) you cannot manually activate the tool changer with this machine
- 40) You are running the first workpiece with a proven program (one that has run before). The first four tools cut just fine. However, the fifth tool (an end mill machining a pocket) does not cut quite deep enough. It is most likely that:
- A) the program zero assignment is incorrect
 - B) the tool length compensation offset for tool five is not correct
 - C) the cutter radius compensation offset for tool five is not correct
 - D) the program coordinates for tool five are incorrect
- 41) You are verifying a new program, cautiously allowing each tool to approach the workpiece using single block and dry run. Tools one and two run just fine. However, as tool three gets within about one inch of the work surface, you are worried that it may not stop in its correct position, so you press feed hold and check the DISTANCE-TO-GO page on the display screen. Sure enough, the distance to go reads a value of Z-3.5435 inches. It is likely that:
- A) tool length compensation is not instated for this tool
 - B) the tool length compensation offset value is incorrect
 - C) the programmed Z approach coordinate for tool three is incorrect
 - D) any of the above
- 42) You are running the first workpiece with a proven program (one that has run before). Tool number four is a milling cutter that machines a pocket. The pocket is supposed to be 0.5000 in \pm 0.001 deep. However, after tool four mills the pocket, you find the pocket to be 0.4970 deep. You must:
- A) change the program to make the endmill plunge 0.003 in deeper
 - B) increase the tool length compensation value by 0.003 in
 - C) decrease the tool length compensation value by 0.003 in
 - D) increase the program zero designation Z value by 0.003 in
- 43) After correcting the problem given in question 42, you must re-run tool number four. You must:
- A) turn on the optional stop switch, scan to the proper pick-up block for tool four, and run the program from there
 - B) run the entire program from the beginning to get to tool number four
 - C) you cannot rerun tools once they have cut
- 44) A milling cutter in a proven program is machining a circular counter-bored hole using circular interpolation. The counter-bored hole size is supposed to be 3.0000 in, but when you measure the hole, you find it to be 2.9960. You must:
- A) change the programmed coordinates for the milling cutter
 - B) increase the cutter radius compensation value for the milling cutter by 0.002 in
 - C) decrease the cutter radius compensation value for the milling cutter by 0.002 in
 - D) the programmer must fix this kind of problem
- 45) You are running the first workpiece with a proven program. Tool number five is a milling cutter machining a pocket to a very tight depth tolerance. You should:
- A) simply run the milling cutter. If the tool length compensation value, the program Z coordinates, and the program zero assignment in Z are correct, the pocket will come out right to size
 - B) increase the tool length compensation value for tool five by a small amount. This will force excess stock to be left in the pocket. After machining, check the pocket depth and adjust the tool length compensation offset value accordingly and rerun the tool.
 - C) there is no way to ensure that the pocket is machined to the correct depth

Proficiency Test For Machining Centers

Name: _____ Date: _____

Section One: General CNC Questions

- 1) The spindle speed for a particular tool in a program is incorrect and you wish to reduce it. The kind of CNC word you must change is:
 A) an F word C) an S word
 B) a G word D) a T word
- 2) You determine that a feedrate for a particular tool must be 5.0 inches per minute (or 5 millimeters per minute in the metric mode). The correct designation in the program would be:
 A) F0.5 C) F5.0
 B) S5.0 D) F0.005
- 3) You are running a proven program (one run before) for the first time in a new setup. You are cautiously allowing the first tool come into its approach position when you determine that the tool is not going where it is supposed to. It is most likely that:
 A) the spindle speed is not correct
 B) the program zero designation is not correct
 C) the dry run switch should be turned on
 D) the programmed coordinates are not correct
- 4) When coordinates going into a program are specified relative to the program zero point, it is called:
 A) the incremental mode
 B) the absolute mode
 C) the rapid mode
 D) the canned cycle mode
- 5) What mode switch position allows programs to be modified?
 A) Jog D) Edit
 B) Auto E) Zero return
 C) MDI (manual data Input)
- 6) What mode switch position allows a program to be run from within the CNC control?
 A) Jog D) Edit
 B) Auto E) Zero return
 C) MDI (manual data Input)
- 7) What mode switch position allows the activation of a single CNC command?
 A) Jog D) Edit
 B) Auto E) Zero return
 C) MDI (manual data Input)
- 8) What mode switch position allows you to send the machine to its reference (home) position?
 A) Jog D) Edit
 B) Auto E) Zero return
 C) MDI (manual data Input)
- 9) What mode switch position lets you move the machine axes manually?
 A) Jog D) Edit
 B) Auto E) Zero return
 C) MDI (manual data Input)
- 10) Which on/off switch works in conjunction with a slash code in the program?
 A) Dry Run D) Single Block
 B) Optional Stop E) Machine Lock
 C) Optional Block Skip/Block Delete
- 11) Which on/off switch works in conjunction with an M01 in the program?
 A) Dry Run D) Single Block
 B) Optional Stop E) Machine Lock
 C) Optional Block Skip/Block Delete
- 12) Which on/off switch makes the control activate one command in the program at a time and then stop?
 A) Dry Run D) Single Block
 B) Optional Stop E) Machine Lock
 C) Optional Block Skip/Block Delete
- 13) The Feedrate Override switch lets the operator control the machine's movement rate during cutting commands.
 True False
- 14) The button used to activate a CNC program is:
 A) Reset D) Cycle Start
 B) Emergency Stop E) Feed Hold
 C) Coolant On
- 15) The button used to temporarily stop the CNC cycle is:
 A) Reset D) Cycle Start
 B) Emergency Stop E) Feed Hold
 C) Coolant On
- 16) The button that will reactivate the CNC program after temporary stoppage is:
 A) Reset D) Cycle Start
 B) Emergency Stop E) Feed Hold
 C) Coolant On
- 17) The button that will clear data in the look-ahead buffer after a temporary program stoppage is:
 A) Reset D) Cycle Start
 B) Emergency Stop E) Feed Hold
 C) Coolant On

General CNC Questions (continued)

18) The button that will actually turn off the power to the machine tool is:

- A) Reset
- B) Emergency Stop
- C) Coolant On
- D) Cycle Start
- E) Feed Hold

19) The display screen mode that lets the operator actually see the CNC program is the:

- A) Offset page
- B) Position page
- C) Alarm page
- D) Program page
- E) Program Check page

20) The display screen mode that lets the operator enter tooling related data is the:

- A) Offset page
- B) Position page
- C) Alarm page
- D) Program page
- E) Program Check page

21) The display screen mode that lets the operator most easily see axis data is the:

- A) Offset page
- B) Position page
- C) Alarm page
- D) Program page
- E) Program Check page

22) The display screen page that will be automatically displayed when the CNC control determines a problem is the:

- A) Offset page
- B) Position page
- C) Alarm page
- D) Program page
- E) Program Check page

23) The display screen page that allows the operator to see a the *distance-to-go* in the current CNC command is the:

- A) Offset page
- B) Position page
- C) Alarm page
- D) Program page
- E) Program Check page

24) Say your machine does not allow manual control of the spindle, yet you must start the spindle during setup. You must:

- A) contact the machine tool builder to have them fix the machine
- B) write a CNC program that starts the spindle at your desired speed
- C) use the MDI mode to activate the spindle
- D) give up. There is no way to start the spindle

25) As part of the machine start up procedure, you must send the machine to its zero return (home) position.

- True
- False

26) Proven CNC programs (those run before) require no program verification procedures.

- True
- False

27) There is always some way to adjust offsets to ensure that every tool in every program will machine with a little excess stock,

meaning (if the program is correct) there is never an excuse to scrap the first workpiece in a production run.

- True
- False

28) How often should you check the way lubrication levels on your CNC machine?

- A) Once every hour
- B) Once every week
- C) Once every shift
- D) Once every month

29) While there is never a good excuse for a crash, if you do have a mishap and your machine crashes, you should:

- A) check visually to see that there is no damage and continue running production
- B) immediately notify your supervisor
- C) fix any damage and continue running production
- D) stop running production for the balance of your shift

30) As you are verifying a new program that has never run before, you find what you consider to be a serious mistake in the program's motions. You should:

- A) do your best to fix the mistake and continue
- B) notify your supervisor so the programmer can be contacted
- C) assume that the program has been well checked by the programmer and run the program as it is

31) When having a problem of almost any kind, it is often helpful to be able to look at a list of all currently instated commands. The display screen mode that allows this is:

- A) Program
- B) Program Check
- C) Offset
- D) Position

32) A program stop is encountered (M00) and the machine stops. However, you are not sure why the program stop is in the program. You should:

- A) press cycle start to resume the program's execution
- B) call up the PROGRAM display screen mode and see if there is a message telling you what to do near the M00
- C) turn on single block to cautiously step through the next few commands

33) While running a new program, the machine goes into alarm state and shows alarm number 41 with the message "Overcutting will occur during cutter radius compensation." You should:

- A) restart the program and try again
- B) turn on single block and cautiously run the next few commands
- C) consult the alarm list in the operation manual to further diagnose the alarm

34) After diagnosing an alarm it is possible that you must:

- A) change a tool offset value
- B) edit commands in the program
- C) press the reset button to cancel the alarm
- D) all of the above

Section Two: Machining Center Questions

- 35) Spindle speed for machining centers is always specified in:
- A) surface feet or meters per minute
 - B) RPM
 - C) inches per minute
 - D) inches per revolution
- 36) When viewing a vertical machining center from the front, the X axis is left/right, Y is fore/aft, and Z is up/down.
- True False
- 37) Many machining center setups require manual measurements to determine the program zero assignment values. These values almost always represent:
- A) are the combined lengths of all tools
 - B) the distance between the program zero point and the machine's zero return position in
 - C) the distance between program zero and the table top
 - D) the distance from the tool tip to the spindle nose
- 38) Once the program zero designation values are determined, they must be placed:
- A) in the program's G92 command
 - B) in the corresponding fixture offset
 - C) in the tool length compensation offsets
 - D) in the cutter radius compensation offsets
 - E) A or B, depending upon the method of program zero assignment
- 39) You need to manually make a tool change to place tool number one in the spindle. However, the machine has no manual buttons or switches for making tool changes. You must:
- A) contact the machine tool builder to fix the machine
 - B) write a short program to make the tool change
 - C) use the MDI mode to command the tool change using program-like commands
 - D) you cannot manually activate the tool changer with this machine
- 40) You are running the first workpiece with a proven program (one that has run before). The first four tools cut just fine. However, the fifth tool (an end mill machining a pocket) does not cut quite deep enough. It is most likely that:
- A) the program zero assignment is incorrect
 - B) the tool length compensation offset for tool five is not correct
 - C) the cutter radius compensation offset for tool five is not correct
 - D) the program coordinates for tool five are incorrect
- 41) You are verifying a new program, cautiously allowing each tool to approach the workpiece using single block and dry run. Tools one and two run just fine. However, as tool three gets within about one inch of the work surface, you are worried that it may not stop in its correct position, so you press feed hold and check the DISTANCE-TO-GO page on the display screen. Sure enough, the distance to go reads a value of Z-3.5435 inches. It is likely that:
- A) tool length compensation is not instated for this tool
 - B) the tool length compensation offset value is incorrect
 - C) the programmed Z approach coordinate for tool three is incorrect
 - D) any of the above
- 42) You are running the first workpiece with a proven program (one that has run before). Tool number four is a milling cutter that machines a pocket. The pocket is supposed to be 0.5000 in \pm 0.001 deep. However, after tool four mills the pocket, you find the pocket to be 0.4970 deep. You must:
- A) change the program to make the endmill plunge 0.003 in deeper
 - B) increase the tool length compensation value by 0.003 in
 - C) decrease the tool length compensation value by 0.003 in
 - D) increase the program zero designation Z value by 0.003 in
- 43) After correcting the problem given in question 42, you must rerun tool number four. You must:
- A) turn on the optional stop switch, scan to the proper pick-up block for tool four, and run the program from there
 - B) run the entire program from the beginning to get to tool number four
 - C) you cannot rerun tools once they have cut
- 44) A milling cutter in a proven program is machining a circular counter-bored hole using circular interpolation. The counter-bored hole size is supposed to be 3.0000 in, but when you measure the hole, you find it to be 2.9960. You must:
- A) change the programmed coordinates for the milling cutter
 - B) increase the cutter radius compensation value for the milling cutter by 0.002 in
 - C) decrease the cutter radius compensation value for the milling cutter by 0.002 in
 - D) the programmer must fix this kind of problem
- 45) You are running the first workpiece with a proven program. Tool number five is a milling cutter machining a pocket to a very tight depth tolerance. You should:
- A) simply run the milling cutter. If the tool length compensation value, the program Z coordinates, and the program zero assignment in Z are correct, the pocket will come out right to size
 - B) increase the tool length compensation value for tool five by a small amount. This will force excess stock to be left in the pocket. After machining, check the pocket depth and adjust the tool length compensation offset value accordingly and rerun the tool.

