Proficiency Test For Turning Center Programming And Operation

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: Turning Center Questions

35) A turning center's diameter controlling axis is its X axis.
❑ True ❑ False

36) Spindle speed for a turning center can be specified in:
❑ A) only RPM
❑ B) only surface feet per minute (SFM)
❑ C) both RPM and SFM
❑ D) inches per revolution

37) Feedrate can be specified in:
❑ A) only feed per revolution (IPR in inch mode)
❑ B) only feed per minute (IPM in inch mode)
❑ C) both feed per minute and feed per revolution
❑ D) RPM

38) For almost all turning centers, program zero must be assigned for every tool. These numbers almost always represent:
❑ A) the distance between program zero and the chuck face
❑ B) the distance between program zero and the machine's zero return (home) position
❑ C) the largest diameter and length the machine can turn
❑ D) the distance from the floor to the spindle centerline

39) Once the program zero assigning values are determined for each tool, they must be placed:
❑ A) in the program at the beginning of each tool in a G50 command (G92 on some machines)
❑ B) in the corresponding geometry offsets
❑ C) in tool wear offsets
❑ D) A or B, depending on the method of program zero assignment

40) You need to index the turret to station number five. However, the machine has no manual buttons and switches to allow turret indexing. You must:
❑ A) contact the machine tool builder to fix the machine
❑ B) write a short program to make the turret index
❑ C) use the MDI mode to command the turret to index using program-like commands
❑ D) you cannot manually activate the turret index on this machine

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, you are worried that tool number three will not stop in its proper approach position, so when the tool comes within about one inch of the workpiece in Z, you press feed hold and check the DISTANCE-TO-GO page on the display screen. Sure enough, the distance to go page reads a distance of -3.5057 inches. It is likely that:
❑ A) the Z axis geometry offset for tool three is incorrect
❑ B) the programmed approach coordinate is incorrect
❑ C) the Z axis work shift value 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 two is a finish turning tool that machines a 3.0000 in +- 0.0005 diameter. After running tool number two and find this diameter to be 3.0020 in. You must
❑ A) increase tool number two's wear offset in X by 0.0020
❑ B) decrease tool number two's wear offset in X by 0.0020
❑ C) increase tool number two's wear offset in Z by 0.0020
❑ D) decrease tool number two's wear offset in Z by 0.0020
❑ E) change the program to make the tool cut 0.0020 in smaller

43) After the problem described in question number 42 is corrected, you must rerun tool number two. You must:
❑ A) rerun the entire program to get to tool number two
❑ B) you cannot rerun tools once they have cut
❑ C) write a small program just for tool number two
❑ D) turn on the optional stop switch, scan to the beginning block of tool number two, and run the program from there

44) You are running the first workpiece in a new setup with a proven program. You notice a very close tolerance on an outside diameter that is to be finish turned by tool number three. You should:
❑ A) simply run the finish turning tool. If the geometry offset and the programmed coordinates are correct, the workpiece will come out right to size.
❑ B) increase tool number three's wear offset by a small amount to force it to leave some excess stock on the diameter. Let tool three cut an measure what it does. Adjust the wear offset accordingly and rerun tool three.
❑ C) there is no way to ensure that this diameter will be machined perfectly for the first workpiece.
❑ D) adjust the Z offset so the tool stays away from the surface being machined.

45) As the finish turning tool described in question 44 continues to machine workpieces, it will eventually begin to wear. What will you have to do several times during the tool's life in order to keep the tool machining the workpiece to the proper diameter?
❑ A) Reduce the tool's wear offset value in X.
❑ B) Reduce the tool's wear offset value in Z.
❑ C) Increase the tool's wear offset value in X.
❑ D) Increase the tool's wear offset value in Z.

46) Eventually the insert in the finish turning tool described in question 44 must be replaced (or indexed). When this is done, you must also:
❑ A) increase the X offset back to its original value
❑ B) decrease the X offset back to its original value
❑ C) increase the Z offset back to its original value
❑ D) decrease the Z offset back to its original value

Proficiency Test For Turning Center Programming And Operation

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: Turning Center Questions

35) A turning center's diameter controlling axis is its X axis.

 ■ True ❑ False

36) Spindle speed for a turning center can be specified in:
 - ❑ A) only RPM
 - ❑ B) only surface feet per minute (SFM)
 - ■ C) both RPM and SFM
 - ❑ D) inches per revolution

37) Feedrate can be specified in:
 - ❑ A) only feed per revolution (IPR in inch mode)
 - ❑ B) only feed per minute (IPM in inch mode)
 - ■ C) both feed per minute and feed per revolution
 - ❑ D) RPM

38) For almost all turning centers, program zero must be assigned for every tool. These numbers almost always represent:
 - ❑ A) the distance between program zero and the chuck face
 - ■ B) the distance between program zero and the machine's zero return (home) position
 - ❑ C) the largest diameter and length the machine can turn
 - ❑ D) the distance from the floor to the spindle centerline

39) Once the program zero assigning values are determined for each tool, they must be placed:
 - ❑ A) in the program at the beginning of each tool in a G50 command (G92 on some machines)
 - ❑ B) in the corresponding geometry offsets
 - ❑ C) in tool wear offsets
 - ■ D) A or B, depending on the method of program zero assignment

40) You need to index the turret to station number five. However, the machine has no manual buttons and switches to allow turret indexing. You must:
 - ❑ A) contact the machine tool builder to fix the machine
 - ❑ B) write a short program to make the turret index
 - ■ C) use the MDI mode to command the turret to index using program-like commands
 - ❑ D) you cannot manually activate the turret index on this machine

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, you are worried that tool number three will not stop in its proper approach position, so when the tool comes within about one inch of the workpiece in Z, you press feed hold and check the DISTANCE-TO-GO page on the display screen. Sure enough, the distance to go page reads a distance of -3.5057 inches. It is likely that:
 - ❑ A) the Z axis geometry offset for tool three is incorrect
 - ❑ B) the programmed approach coordinate is incorrect
 - ❑ C) the Z axis work shift value 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 two is a finish turning tool that machines a 3.0000 in +- 0.0005 diameter. After running tool number two and find this diameter to be 3.0020 in. You must
 - ❑ A) increase tool number two's wear offset in X by 0.0020
 - ■ B) decrease tool number two's wear offset in X by 0.0020
 - ❑ C) increase tool number two's wear offset in Z by 0.0020
 - ❑ D) decrease tool number two's wear offset in Z by 0.0020
 - ❑ E) change the program to make the tool cut 0.0020 in smaller

43) After the problem described in question number 42 is corrected, you must rerun tool number two. You must:
 - ❑ A) rerun the entire program to get to tool number two
 - ❑ B) you cannot rerun tools once they have cut
 - ❑ C) write a small program just for tool number two
 - ■ D) turn on the optional stop switch, scan to the beginning block of tool number two, and run the program from there

44) You are running the first workpiece in a new setup with a proven program. You notice a very close tolerance on an outside diameter that is to be finish turned by tool number three. You should:
 - ❑ A) simply run the finish turning tool. If the geometry offset and the programmed coordinates are correct, the workpiece will come out right to size.
 - ■ B) increase tool number three's wear offset by a small amount to force it to leave some excess stock on the diameter. Let tool three cut an measure what it does. Adjust the wear offset accordingly and rerun tool three.
 - ❑ C) there is no way to ensure that this diameter will be machined perfectly for the first workpiece.
 - ❑ D) adjust the Z offset so the tool stays away from the surface being machined.

45) As the finish turning tool described in question 44 continues to machine workpieces, it will eventually begin to wear. What will you have to do several times during the tool's life in order to keep the tool machining the workpiece to the proper diameter?
 - ■ A) Reduce the tool's wear offset value in X.
 - ❑ B) Reduce the tool's wear offset value in Z.
 - ❑ C) Increase the tool's wear offset value in X.
 - ❑ D) Increase the tool's wear offset value in Z.

46) Eventually the insert in the finish turning tool described in question 44 must be replaced (or indexed). When this is done, you must also:
 - ■ A) increase the X offset back to its original value
 - ❑ B) decrease the X offset back to its original value
 - ❑ C) increase the Z offset back to its original value
 - ❑ D) decrease the Z offset back to its original value