

2

CNC Machining Center: Fixed Cycles (Canned Cycles)

Objectives

Students will be able to:

- ✓ Develop a program using "Fixed Cycles" / "Canned Cycles" for machining hole operations on a CNC machining center
- ✓ Analyze the program for errors using the CNC simulator and revise the program as needed
- ✓ Operate and setup the CNC machine, tool setup, set work offsets, NC program input to the CNC control, and part prove out

Orienting Questions

- ✓ What is Fixed Cycles?
- ✓ When is Fixed Cycles used in a program?
- ✓ What are the advantages of using Fixed Cycles?
- ✓ What special rules must be followed when using Fixed Cycles?

Except where otherwise noted, this work was created by Glenn Galloway and is licensed under the Creative Commons Attribution 3.0 Unported License. To view a copy of this license, visit <http://creativecommons.org/licenses/by/3.0/> or send a letter to Creative Commons, 444 Castro Street, Suite 900, Mountain View, California, 94041, USA.

INTRODUCTION

Fixed Cycles are used when programming various types of hole operations, such as; shallow hole drilling, deep hole drilling, spot drilling, counter boring, regular tapping, rigid tapping, various types of boring, and back boring. The use of fixed cycles will make programming of holes easier and shorter than the alternate method of using G1 and G0 to move the tool into the part and back out. This method would make for a very long program if peck drilling was required at each hole location.

2.1 INTRODUCTION TO FIXED CYCLES WITH CAD/CAM SOFTWARE DEMONSTRATION

In this section, Fixed Cycles will be introduced and illustrated with CAM software simulator.

2.1.1 UNDERSTANDING THE FUNCTION OF FIXED CYCLES

If you have manual machining knowledge you will be ahead of the game in understanding the different types of fixed cycles available on CNC machines, and which one is best suited for the particular operation being performed.

For example: if you have ever drilled holes in thick material manually, you probably learned that chips will pack around the drill and cause the drill to break as it is being pushed through the material. The solution is to pull the drill out of the hole to remove chips then back down in the hole to drill deeper into the material. From the various fixed cycles available we would use a Peck Drilling Cycle for this task.

2.1.2 EXAMPLE PRINT FOR DEMONSTRATION OF FIXED CYCLES

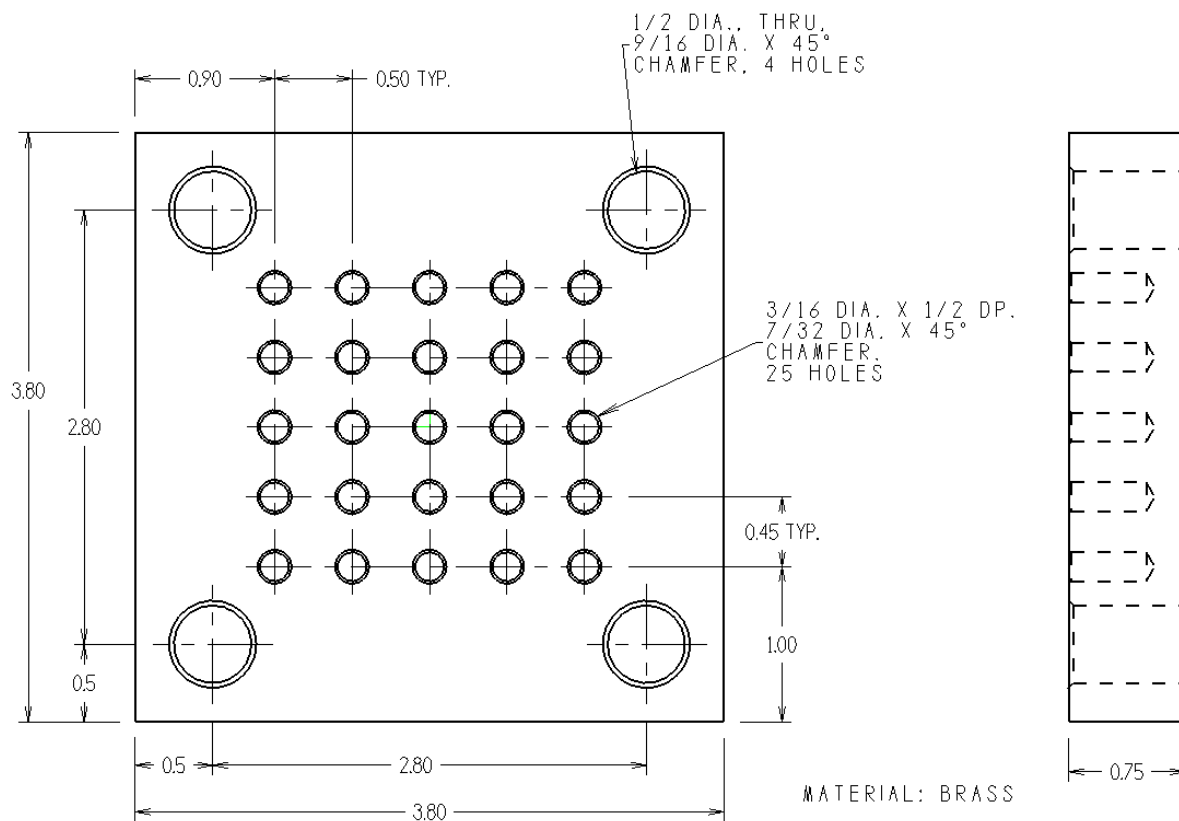


Figure 1 Figure 1 shows that a fixed cycle operation will be used to drill the holes in the material. Note the leader line pointing to the holes specifying size, depth, and number of holes.

2.1.3 GRAPHIC DEMONSTRATION USING CAM SIMULATOR OF FIXED CYCLES

Video 1 (click link below) shows animated tools, spot drilling, and drilling the two sets of holes. Please note: the simulation will not show details like dwell time that is set for spot drills, and the pecking motion as the drill cuts the part. Notice the chamfer around the holes after all tools have run, this is created by the spot drill tool, the chamfer diameter is controlled by the Z depth specified in the fixed cycle.

Video 1: Fixed Cycle Simulation Video (Hold Ctrl key down + Click the link to play)

<https://www.youtube.com/watch?v=-D2vFKO4uDA>

2.2 OVERVIEW OF FIXED CYCLES

Hole operations is a very important and common type of work necessary to produce manufactured parts. Hole operations require repetitive movements in the Z axis at each hole location, the need for fixed cycles became necessary. Fixed cycles are designed to make programming easier and to make the program shorter. There is no limit to the number of holes (new coordinates) that can be programmed after the fixed cycle block.

2.2.1 CODES USED IN FIXED CYCLES

Listed below are most of the codes that are common for most brand CNC controls.

Fixed Cycle G Codes:

G73	Peck drilling cycle (Feed stop with each peck for chip break)
G74	Counter tapping cycle (Left hand threading)
G75	Not for Milling
G76	Fine boring cycle

G73	Peck drilling cycle (Feed stop with each peck for chip break)
G80	Fixed cycle cancel
G81	Drilling cycle (simple drill into part and rapid out)
G82	Drilling cycle, counter boring cycle (same as G81 with added dwell)
G83	Peck drilling cycle (same as G81 with pecking amount added)
G84	Tapping cycle (Right hand threading)
G85 thru G89	Boring cycle (There are various types, feed in rapid out, feed in and out, feed in stop spindle then rapid out, back bore cycle, and others.)
G98	Return to initial point/level in Fixed cycle
G99	Return to the R point/level in Fixed cycle

Other codes used in Fixed Cycles: (Ref. Figure 3, Chart)

R	The distance from the origin in the Z axis, where Z axis motion changes from rapid to feed, it is also used as the distance to retract before moving to the next hole if using G99.
Q	The depth of each Peck until the Z depth is reached
P	

	Dwell amount when the Z axis has reached its depth, specified as a time value with no decimal. Example: P1000= 1 second, P500=1/2 second, P250=1/4 second
F	Cutting feed rate
K	Number of repeats (can be used on the end of any fixed cycle)

2.2.2 ASSESMENT QUIZ 1

Fill in the blank with the proper code.

1. What Fixed Cycle is used for deep hole drilling? _____
2. What code is used to stop or cancel Fixed Cycle that has been started?

3. What Fixed Cycle is used to tap threads in a part? _____
4. What code on a fixed cycle block will cause the tool to pull out to the Initial Level before moving to the next hole? _____
5. What code is used to cause a dwell at the bottom of the hole? _____
6. What Fixed Cycle is the simplest of all the fixed cycles? _____

[Check answers here.](#)

2.2.2 FIXED/CANNED CYCLE APPLICATION CHART

The chart in **Figure 2** shows the difference in the type of tool motion for each Fixed Cycle Code. This will help you choose the proper code to use for each operation.

Fixed Cycle Tool Motion and Application Chart				
G-Code	Z axis motion into the work	Operation when Z axis depth is reached	Z axis motion out of the work	Application
G73	Intermittent feed	N/A	Rapid traverse	High speed peck drilling cycle
G74	Feed	Dwell & reverse spindle	Feed	Left hand tapping cycle
G76	Feed	Spindle stop & orient	Rapid traverse	Fine boring cycle
G80	N/A	N/A	N/A	Cancel
G81	Feed	N/A	Rapid traverse	Drilling cycle
G82	Feed	Dwell	Rapid traverse	Drill, spot drilling, counter boring
G83	Intermittent feed	N/A	Rapid traverse	Peck drilling cycle
G83	Intermittent feed	Dwell	Rapid traverse	Small hole peck drilling cycle
G84	Feed	Dwell & reverse spindle	Feed	Right hand tapping cycle
G85	Feed	N/A	Feed	Boring cycle
G86	Feed	Spindle stop & orient	Rapid traverse	Boring cycle
G87	Feed	Spindle start	Rapid traverse	Back boring cycle
G88	Feed	Dwell & spindle stop	Manual	Boring cycle
G89	Feed	Dwell	Feed	Boring cycle

Figure 2 Fixed Cycle Chart

2.2.3 FIXED CYCLE CODE FORMAT CHART

The chart in **Figure 3** shows all possible codes that can be used with each different fixed cycle code. Other model controls may use a variation from this chart. When writing any program use the machine manual for the machine being programmed.

Fixed/Canned Cycle Code Format Chart	
G-Code	Code format, this is all possible codes allowed on the fixed cycle start block, some maybe omitted
G73	G98/99 G73 X__ Y__ Z__ R__ Q__ F__ K__

G74	G98/99 G74 X__Y__Z__R__P__F__K__
G76	G98/99 G76 X__Y__Z__R__Q__P__F__K__
G80	G80
G81	G98/99 G81 X__Y__Z__R__F__K__
G82	G98/99 G82 X__Y__Z__R__P__F__K__
G83	G98/99 G83 X__Y__Z__R__Q__F__K__
G83	G98/99 G83 X__Y__Z__R__P__Q__F__I__K__
G84	G98/99 G84 X__Y__Z__R__P__F__K__
G85	G98/99 G85 X__Y__Z__R__F__K__
G86	G98/99 G86 X__Y__Z__R__F__K__
G87	G98/99 G87 X__Y__Z__R__Q__P__F__K__
G88	G98/99 G88 X__Y__Z__R__P__F__K__
G89	G98/99 G89 X__Y__Z__R__P__F__K__

Figure 3 Fixed Cycle Code Format Chart

2.2.4 G98 AND G99 CODES USED WITH FIXED CYCLES

G98 and G99 are codes used in fixed cycles to control the Z axis position before moving to the next hole location.

There are two possible positions allowed by the fixed cycle to place the Z axis before moving to the next location.

- The “**Initial Level**”, when specified by G98 at the start of the fixed cycle block
- The “**R Level**”, when specified by G99 at the start of the fixed cycle block

To determine which level to retract the tool too after the final Z axis depth in the fixed cycle has been reached, you must answer this question! Are there obstructions along the path from hole to hole that the tool will need clearance in the Z axis? If the answer is **NO there are not any** obstructions in the path to the next hole, then the Z axis position would be fine to pull up to the R__ plane. By using a G99 at the beginning of the fixed cycle block this will be achieved.

If the answer to the question is, **YES there are** obstructions along the path to the other holes then a G98 should be used instead of a G99. G98 will allow the tool to retract to the initial level after completing the final Z depth. The initial level is established by the last programmed “Z axis value” before the fixed cycle is started.

Example with G99:

The last known Z value before the fixed cycle block is “Z.2”, this will establish the initial level. Since G99 has been used, they retract before going to the next hole will be to the “R level” of “R.1”. After all holes have been drilled the Z axis will return to the initial level after the G80 has been read.

```
T3 M6 (1/2 DRILL)
G0 G90 G54 X-1.4 Y-1.4 S2000 M3
```

G43 H3 Z1.

Z.2 (INITIAL LEVEL DEFINED, LAST Z LOCATION BEFORE THE FIXED CYCLE START)

G99 G81 X-1.4 Y-1.4 Z-.9502 R.1 F25. (START FIXED CYCLE, SPOT DRILL)

X1.4

Y1.4

X-1.4

G80 (FIXED CYCLE CANCEL)

Example with G98:

The last known Z value before the fixed cycle block is “Z.75”, this will establish the “Initial level”. Since G98 has been used, the retract Z position before going to the next hole will be to the “Initial level”. The tool will be at the initial level before moving to the next hole location, this will allow clearance for obstructions along the path. The initial level can be whatever amount is needed for clearance. The “R Level” will still be used in the fixed cycle as the “feed plane”, and for pull out amount after each peck in the peck drilling cycle.

T3 M6 (1/2 DRILL)

G0 G90 G54 X-1.4 Y-1.4 S2000 M3

G43 H3 Z1.

Z.75 (INITIAL LEVEL DEFINED, LAST Z LOCATION BEFORE THE FIXED CYCLE START)

G98 G81 X-1.4 Y-1.4 Z-.9502 R.1 F25. (START FIXED CYCLE, SPOT DRILL)

X1.4

Y1.4

X-1.4

G80 (FIXED CYCLE CANCEL)

2.2.5 LOOKING AT THE CNC PROGRAM WITH FIXED CYCLES

We will now look at the CNC program for the same part example used thus far. This will be the same program used in the CAM video demonstration. The tool list used in the program is shown below.

- Tool 1 ----- 3/4" Dia. Spot drill ----- 4 holes
- Tool 2 ----- 3/8" Dia. Spot drill ----- 25 holes
- Tool 3 ----- 1/2" Dia. Drill ----- 4 holes
- Tool 4 ----- 3/16" Dia. Drill ----- 25 holes

Notice the **dark blue** text and descriptions in the program below is the key areas to study.

```
%
O7002(MTT 252 MOD FIXED CYCLES)
( T1 | 3/4 SPOTDRILL | H1 )
( T2 | 3/8 SPOTDRILL | H2 )
( T3 | 1/2 DRILL | H3 )
( T4 | 3/16 DRILL | H4 )
N102 G0 G20 G17 G40 G80 G90
N104 T1 M6 (3/4 SPOT DRILL TOOL)
```

Continued:

```
N106 G0 G90 G54 X-1.4 Y1.4 S2500 M3 (LOCATE OVER FIRST HOLE TO BE SPOT
DRILLED)
N108 G43 H1 Z1.
N110 Z.2 (INITIAL LEVEL DEFINED, LAST Z LOCATION BEFORE FIXED CYCLE
START)
(SPOT DRILL 4 HOLES)
N112 G99 G82 X-1.4 Y1.4 Z-.281 R.1 P.25 F30. (START FIXED CYCLE, SPOT DRILL)
N114 X1.4 (THE CYCLE WILL REPEAT AT EACH NEW LOCATION UNTIL
CANCELED)
N116 Y-1.4
N118 X-1.4
N120 G80 (FIXED CYCLE CANCEL)
N122 Z1.
N124 M5
N126 G91 G28 Z0.
```

N128 M01
N130 G0 G20 G17 G40 G80 G90
N132 T2 M6 (3/8 SPOT DRILL TOOL)
N134 G0 G90 G54 X-1. Y-.9 S3500 M3 (LOCATE OVER FIRST HOLE TO BE SPOT
DRILLED)
N136 G43 H2 Z1.
N138 Z.2 (INITIAL LEVEL DEFINED, LAST Z LOCATION BEFORE FIXED CYCLE
START)
(SPOT DRILL 25 HOLES)
N140 G99 G82 X-1. Y-.9 Z-.109 R.1 P.5 F30. (START FIXED CYCLE, SPOT DRILL)
N142 X-.5 (THE CYCLE WILL REPEAT AT EACH NEW LOCATION UNTIL
CANCELED)
N144 X0.
N146 X.5
N148 X1.
N150 Y-.45
N152 X.5
N154 X0.
N156 X-.5
N158 X-1.
N160 Y0.
N162 X-.5
N164 X0.
N166 X.5
N168 X1.
N170 Y.45
N172 X.5
N174 X0.
N176 X-.5
N178 X-1.
N180 Y.9
N182 X-.5
N184 X0.

Continued:

N186 X.5
N188 X1.

N190 G80 (FIXED CYCLE CANCEL)
N192 Z1.
N194 M5
N196 G91 G28 Z0.
N198 M01
N200 G0 G20 G17 G40 G80 G90
N202 T3 M6 (1/2 DRILL)
N204 G0 G90 G54 X-1.4 Y-1.4 S2000 M3 (LOCATE OVER FIRST HOLE TO BE SPOT
DRILLED)
N206 G43 H3 Z1.
N208 Z.2 (INITIAL LEVEL DEFINED, LAST Z LOCATION BEFORE FIXED CYCLE
START)
N210 G99 G81 X-1.4 Y-1.4 Z-.9502 R.1 F25. (START FIXED CYCLE, SPOT DRILL)
N212 X1.4(THE CYCLE WILL REPEAT AT EACH NEW LOCATION UNTIL
CANCELED)
N214 Y1.4
N216 X-1.4
N218 G80 (FIXED CYCLE CANCEL)
N220 Z1.
N222 M5
N224 G91 G28 Z0.
N226 M01
N228 G0 G20 G17 G40 G80 G90
N230 T4 M6 (3/16 DRILL)
N232 G0 G90 G54 X-1. Y-.9 S3000 M3 (LOCATE OVER FIRST HOLE TO BE SPOT
DRILLED)
N234 G43 H4 Z1.
N236 Z.2 (INITIAL LEVEL DEFINED, LAST Z LOCATION BEFORE FIXED CYCLE
START)
(DRILL 25 HOLES)
N238 G99 G83 X-1. Y-.9 Z-.5563 R.1 Q.1 F25. (START FIXED CYCLE, SPOT DRILL)
N240 X-.5(THE CYCLE WILL REPEAT AT EACH NEW LOCATION UNTIL
CANCELED)
N242 X0.
N244 X.5
N246 X1.
N248 Y-.45
N250 X.5
N252 X0.

N254 X-.5
N256 X-1.
N258 Y0.
N260 X-.5
N262 X0.
N264 X.5
N266 X1.

Continued:

N268 Y.45
N270 X.5
N272 X0.
N274 X-.5
N276 X-1.
N278 Y.9
N280 X-.5
N282 X0.
N284 X.5
N286 X1.
N288 G80 (FIXED CYCLE CANCEL)
N290 Z1.
N292 M5
N294 G91 G28 Z0.
N296 G28 Y0.
N298 M30
%

2.2.6 VIDEO DEMONSTRATION OF FIXED CYCLES

Video 2 (click link below), was taken at the CNC Machining Center. This video shows the dwell time when the tool reaches the programmed Z depth, and the pecking action during machining, which is not shown in the CAM simulation video. The dwell time has been set to a value that is longer than normal so you can see the tool dwell.

Video 2: Fixed Cycle Machine Video Demonstration (Hold Ctrl key down + Click the link to play) <http://youtu.be/9CTw9m70-QQ>

SUMMARY OF MAJOR CONCEPTS

Fixed Cycles is used to make programming hole operations easier. Fixed Cycles is used to program various hole operations using Point-To-Point machining. When programming with Fixed Cycles, this will greatly reduce the size of the program to save memory in the CNC control. Fixed Cycles are also known as Canned Cycles, in this module the term Fixed Cycle will be used. In the early days of NC machines before CNC machines became available, programs were punched into a paper or mylar tape. The punched tape was used to drive the NC machine. Programmers stored programs, cycles used for doing various types of hole operations. These were stored in cans. The can could be pulled out and the tape spliced into any other program tape so that the cycle could be used over and over. This is how the nick name canned cycles was created. Now that it is CNC and punched tapes are no longer used for CNC machines, the name fixed cycles was a better description for this type of operation. Some people will still use the term canned cycles.

All “CNC Controller” brands offer fixed cycles, they are generally G codes that start with G73-G74 and G80-G89, and these are the most common fixed cycle codes. Most control manufacturers offer extra cycles to meet the special needs for the job. Fixed Cycles are used in programming Point-To-Point machining operations. This type of machining operations are used in the machining of hole operations such as drilling, boring, tapping, and other various types of hole operations.

Programming hole operations become an easy task when using fixed cycles due to the fact that parameters for drilling, peck drilling, tapping, boring, and others, are defined on one block. Once the cycle is started multiple holes can be machined that have the same operation performed at each hole location with just a new “X” and/or “Y” axis value. This will cause the controller to repeat all the same functions at each new location until commanded to cancel the fixed cycle.

The controller must analyze the program or error using the CNC simulate and revise the program if needed. There are a few other observations and decisions that must be made when programming with fixed cycles:

- 1- As the tool travels from one hole to the next hole, clearance for the tool path must be noted. The clearance for the Z axis will be determined.
- 2- The type of fixed cycle that is best for the operation to be performed must be determined.
- 3- Holes that are exactly the same size, depth, and are on the same Z plane can be grouped together for a cycle.

KEY TERMS

Dwell A time value used to allow the tool to remain at the final Z depth. A pause in the Z axis will allow the tool to continue to remove all the material at the final depth. Used in counter boring, spot drilling, or center drilling etc.

Fixed/Canned Cycle A sequence of actions that is executed by a single block of code and can be repeated as needed then canceled.

Initial Level The Z height that is established before the start of the fixed cycle. This is also the level the tool will retract to when using a G98 on the fixed cycle start block. This initial level is used to gain enough tool clearance for obstructions as the tool moves from one location to the next.

Peck A method used in Peck Drilling to pull chips from the hole before continuing deeper. The amount of desired peck in the cycle is specified with the Q word. This is used for small holes and deep holes to keep chips from packing around the drill in the holes causing the tool to break.

Point-To-Point A method of programming where the machining is only performed at the location specified. The tool is not in contact with the work during motion to the next hole location. Machining will take place at each location programmed until canceled.

R Level The Z height that the tool will feed into the work piece. This is also the level the tool will retract to when using a G99 on the fixed cycle start block.

LAB

LAB 1. PHYSICAL LAB USING A 3 AXIS MACHINING CENTER

- Lab exercise will be done at the CNC machine using a 3 axis machining center, or if not available a CNC simulator can be used for a partial lab exercise.

-Materials needed are the CNC program, 4" x 4" wax block, and spot drill and drill tools. Tools will be loaded into the machine and tool lengths set, program entered into the CNC machine, wax block loaded into the machine vise, and the part origin set (G54).

-Requirements for successful lab completion will be to program, setup and prove out a CNC program using Fixed Cycles.

-Programming instructions for the lab exercise using **Print 2 in Figure 4:**

Write a CNC program using Fixed/Canned Cycles for the following operations.

- Spot drill, drill, and bore one .875 hole
- Spot drill, drill, and tap three 10-32 holes
- Spot drill and drill four 1/4" diameter holes

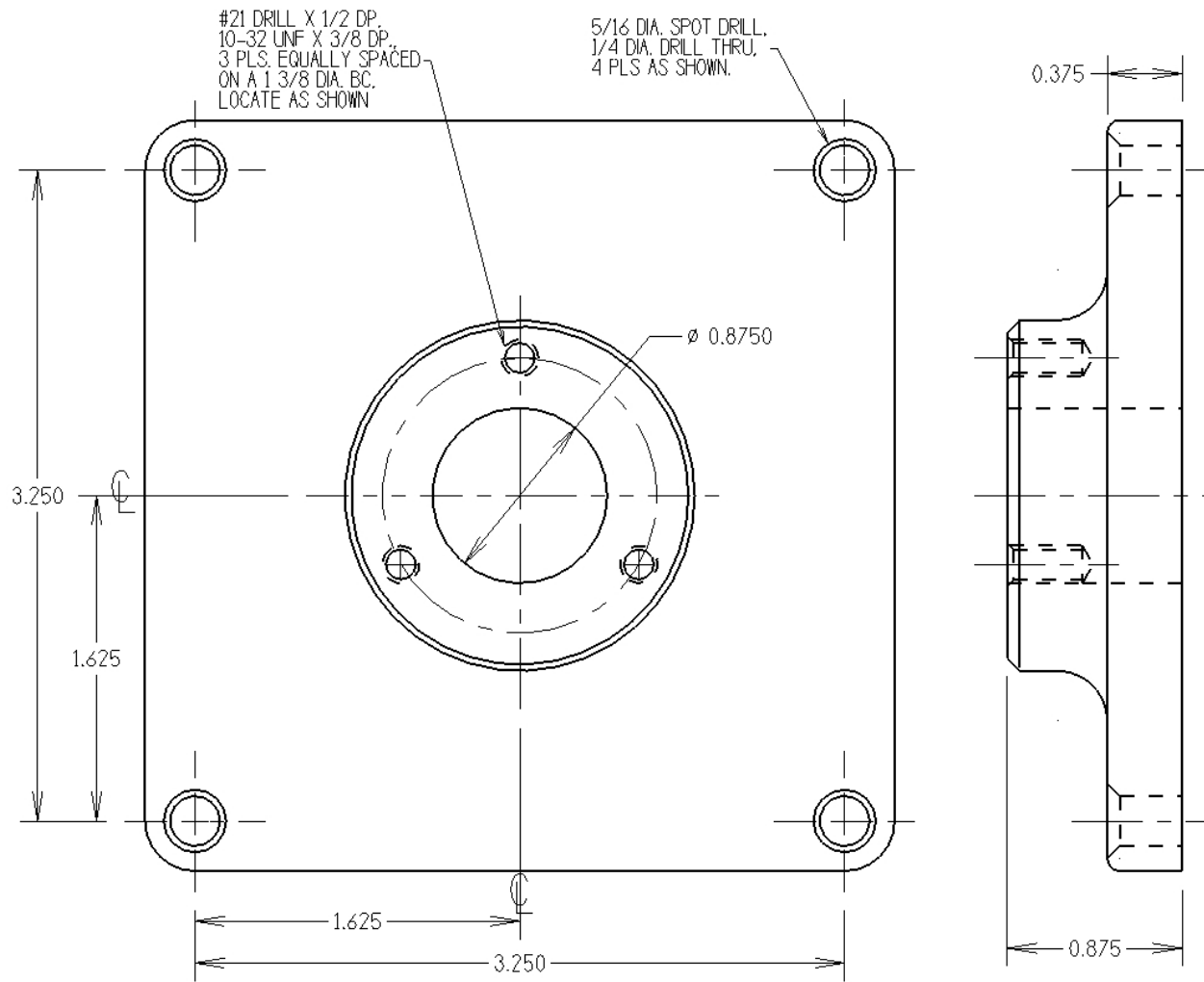


Figure 4 Print

ASSESSMENT**MULTIPLE CHOICE, TEST 1**

Instructions: Circle the best answer.

1. When programming with Fixed Cycles, what type of programming is used?
 - a. Contouring
 - b. Point-To-Point
 - c. High Speed Machining
 - d. Automatic Slot Mill

2. Programming with Fixed Cycles will allow:
 - a. Shorter programs
 - b. Easier manual program writing
 - c. Editing one place will affect all holes for that cycle
 - d. All of the above

3. Fixed Cycles is canceled by programming what code?
 - a. G40
 - b. M00
 - c. G80
 - d. M80

4. The Initial level is the location set by:
 - a. Programming a new tool
 - b. The Z axis location before starting the Fixed Cycle
 - c. The number of holes to be drilled
 - d. The type of cycle being used

5. How many holes (new coordinates) can be programmed using Fixed Cycles to perform the operation?
 - a. Infinite amount
 - b. 50
 - c. 100
 - d. 999

[Check answers here.](#)

ANSWER YES OR NO, TEST 2

Instructions: Check Yes or No.

1. Can a group of holes that are the same diameter be drilled using one Fixed Cycle?

- Yes
 No

2. Can a group of holes that are all the same diameter, and one or more of the group has a different depth, be drilled using one Fixed Cycle?

- Yes
 No

3. Can one hole be drilled using Fixed Cycle Programming?

- Yes
 No

4. Can Fixed Cycles be programmed when using Sub Programs?

- Yes
 No

5. Can multiple Fixed Cycles be programmed in one CNC program?

- Yes
 No

[Check answers here.](#)

CRITICAL THINKING

Choose one or two formative assessments to include here:

1. One Sentence Summary: Describe in 1 sentence, the X,Y, and Z axis motion, for the G81 drilling cycle that has been executed for two holes. Be sure to include answers to some of the following questions in your sentence; “Who does what to whom, when, where, how, and why”
2. Sequencing Events: Describe the sequence of operations for the following hole operations, what hole operations can be grouped together if any, what tool can be used to do multiple groups of holes. Try to think about making the program run time as short as possible. What cycle would you pick for each tool and why?

Three tools will be required for this task; Tool 1= 1/2 diameter spot drill tool, Tool 2= 1/4 diameter drill, Tool 3= 3/8 diameter drill.

3 holes 1/4 diameter X 3/4 deep with a .28 diameter chamfer

2 holes 1/4 diameter X 1 1/8 deep with a .28 diameter chamfer

5 holes 3/8 diameter X 1 1/2 deep with a .41 diameter chamfer

[Answers to Critical Thinking.](#)

FURTHER STUDY

The video links below show some related drilling operations on CNC machines.

[Canned Cycles Video.](#)

[G82 Canned Cycle for Center Drilling](#)

[Canned cycles G82 & G83 Center Drilling & Peck Drilling](#)

[Peck with a G83](#)

ASSESSMENT

ANSWERS TO ACTIVITIES

Answer Key

<u>Test 1</u> Multiple Choice	<u>Test 2</u> Yes or No	<u>Quiz 1</u> Yes or No
1. b	1. Yes	1. Yes
2. d	2. No	2. No
3. c	3. Yes	3. Yes
4. b	4. Yes	4. Yes
5. a	5. Yes	5. Yes
Back to Test 1	Back to Test 2	Back to Quiz 1

ANSWERS TO CRITICAL THINKING ACTIVITIES

1. With the G81 cycle the X and Y axis will locate to position then the Z axis will move to the initial level, then to the R plane, then feed to depth specified by Z value, then the Z axis will retract out of the hole and move to the second hole where all will be repeated.
2. The first operation, spot drill five 1/4 diameter holes to a chamfer diameter of .28, stop the cycle and restart and spot drill five 3/8 diameter holes to a chamfer diameter of .41.

The second operation, drill three 1/4 diameter holes to 3/4 deep, stop the cycle and restart with the new depth of 1 1/8.

The third operation, drill five 3/8 diameter holes to .41 deep.

[Back to Critical Thinking section.](#)

2.8 ATTRIBUTION TABLE

Author/s	Title	Source	License
Glenn Galloway	Video 1 and 2	Glenn Galloway YouTube	Creative Commons Attribution 3.0 Unported License
Glenn Galloway	Figure 1 through 4	Glenn Galloway	Creative Commons Attribution 3.0 Unported License

This workforce solution was funded by a grant awarded by the U.S. Department of Labor’s Employment and Training Administration. The solution was created by the grantee and does not necessarily reflect the official position of the U.S. Department of Labor. The Department of Labor makes no guarantees, warranties, or assurances of any kind, express or implied, with respect to such information, including any information on linked sites, and including, but not limited to accuracy of the information or its completeness, timeliness, usefulness, adequacy, continued availability or ownership.

