INTRODUCTION

Since the beginning of CNC programming many short cuts have become necessary to make processes and programming easier. Just as processes for milling presented in modules 1 and 2 of this course, lathe programming has made improvements as well.

This module teaches how to program Multiple Repetitive Cycles in lathe programming and the advantages of using these cycles. This module teaches the proper use of G70 through G76, with programming examples and videos. The examples in this module is for FANUC 0i controls, for other model controls codes may need to be modified.

3.1 MULTIPLE REPETATIVE CYCLES

There are major programming advantages by the use of Multiple Repetitive Cycles, the programmer will only define the finished shape to be cut for most cycles. The Multiple Repetitive Cycle will contain all the parameters needed for depth of cut, rough stock to leave on surfaces and diameters, and the feed rate, this is a huge advantage to the programmer. Without the use of the Multiple Repetitive Cycles, the programmer will have to program all the rough cuts long hand, rough cuts long hand that will take a minimum of 4 blocks per rough pass. If there are any steps in the contour shape then rough passes will be 6 or more blocks to make one rough contour pass. The program will become very long very fast.

3.1.1 CNC TURNING CENTER: PROGRAMMING EXTERNAL OPERATIONS USING MULTIPLE REPETITIVE CYCLES

In this section we will program *External Lathe Operations* using *Multiple Repetitive Cycles* when possible during the process. The machine to be used is a 2 axis slant bed lathe with a 12 station tool turret. Study the prints in **Figures 1 through 3** that will be used in the programming examples throughout this module.
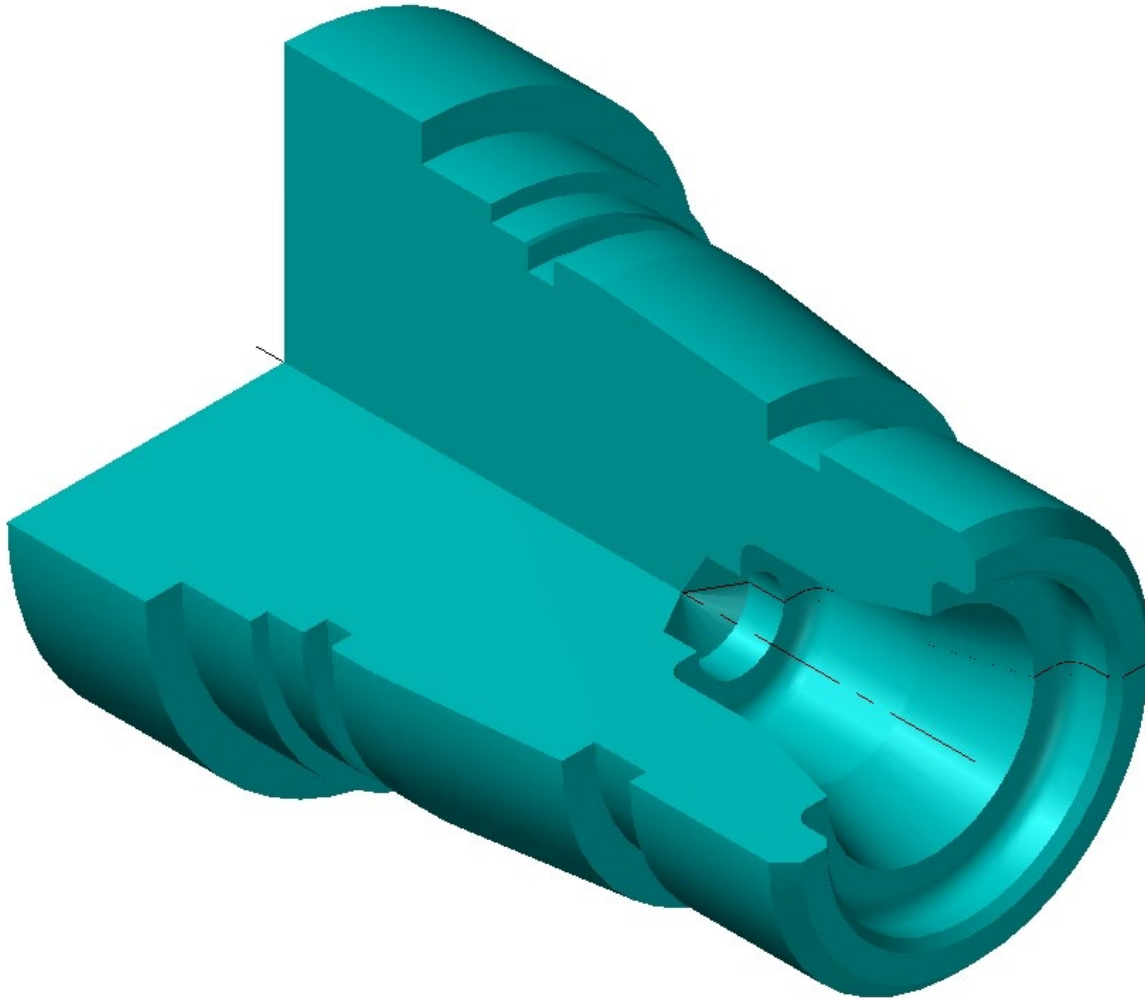
Figure 1, This is a solid CAD (Computer Aided Design) model of the Example Lathe Part. This solid model will help you visualize the part in 3D, and to help you read the 2D part prints shown in figures 2, 3, 4, and 5.
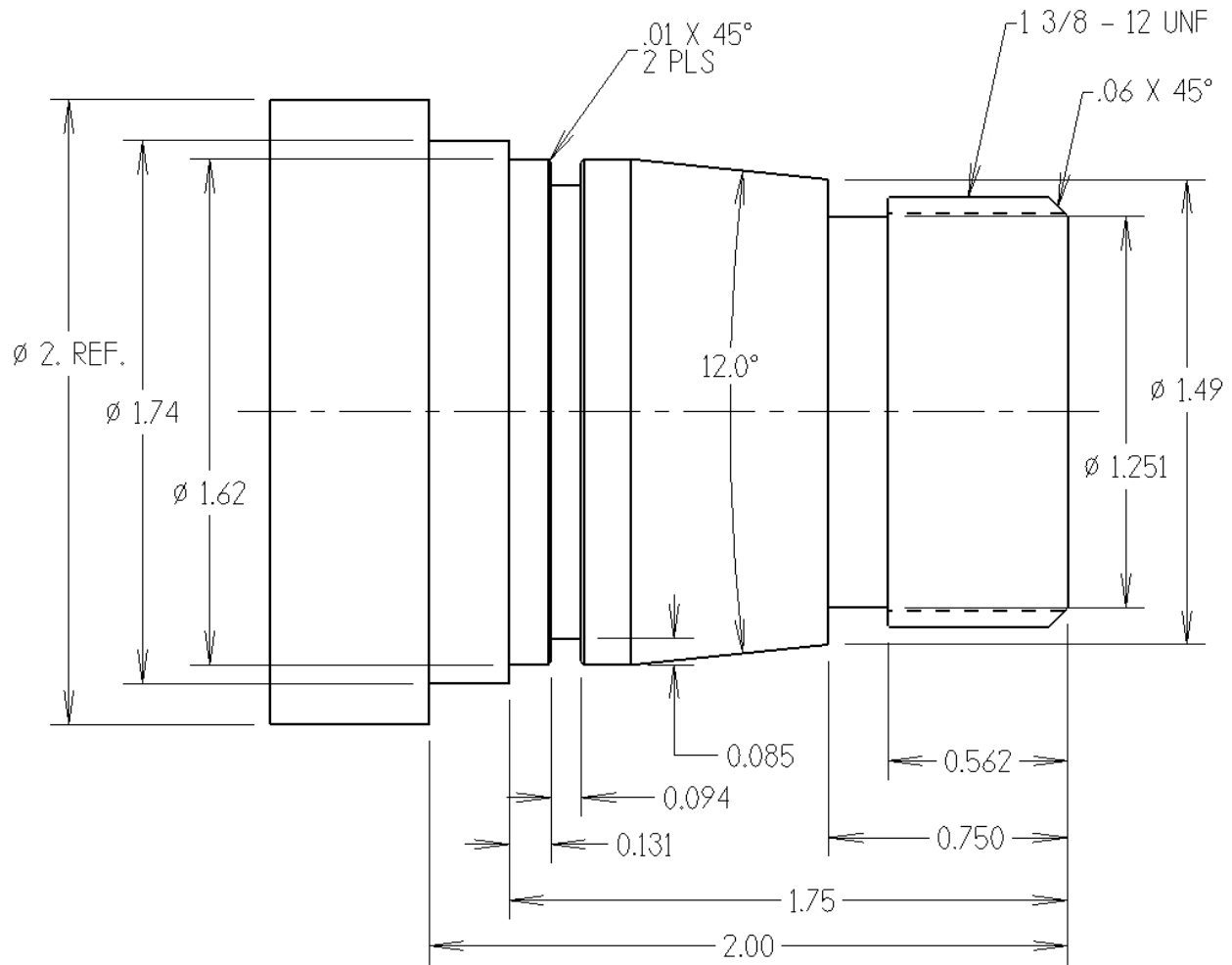
Figure 2 – The figure above is a programming example of a 2D Lathe Part 1 (for external programming, internal not shown) The machine used in programming is a 2-axis slant bed lathe with a 12 station tool turret.

.

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

0.025
used for
clearance

end
point

This point will be
be used as the
LAST POINT
of the contour.

This point location will be
used as the start location
for Multiple Repetitive Cycles
(rough facing cycle, and
rough turning).

9 8 7
6 5 4 3 2 1

start
point

Contour path (blue) will be
roughed turned using
G71 Multiple Repetitive Cycle

This point will be used
as the START POINT
of the contour shape.

NOTICE THE GROOVE AND NECK
HAVE BEEN REMOVED FOR NOW.
THE GROOVE AND NECK CANNOT
BE MACHINED WITH THE TOOL
CURRENTLY BEING USED.

ORIGIN

STOCK BEFORE
MACHINING

0.090 Stock,
Will Leave
.02 in Rough
Facing Cycle

Figure 2, The figure above is a programming example of a part drawing for External Operations.

3.1.2 PROGRAM PLANNING FOR EXTERNAL OPERATIONS

Note: A machining plan for each different part you program will vary depending on part features, the plan for this part may look similar to other parts or could be very different in the type of operations needed or tools used for an operation.

After studying the example part prints in **Figure 1, 2and 3**, I have listed below a machining plan as follows:

The part blank will be chucked on the 2.0 REF. diameter, this area will not be part of the outside shape to be programmed. Some tools will be used for more than one operation.

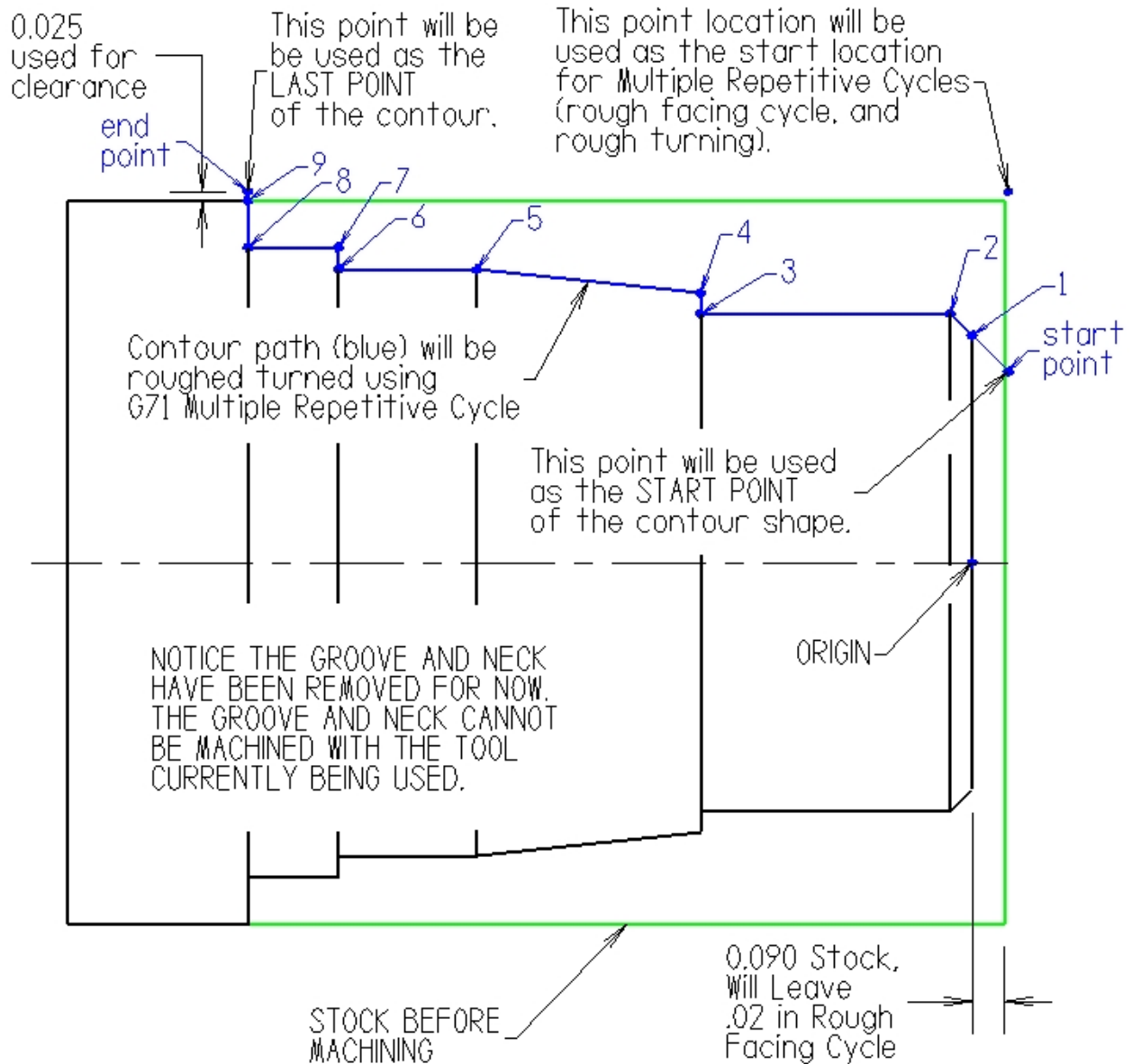|		
Operation 1	Tool 01	Rough Front Face leave .02 inch for finish operation. G72 Cycle
Operation 2	Tool 01	Rough Turn Outside Shape leave .02 on X and Z surfaces, G71 Cycle
Operation 3	Tool 03	Finish Front Face, G code long hand, no cycle
Operation 4	Tool 03	Finish Outside Shape, G70 Cycle
Operation 5	Tool 05	Machine Neck (behind thread)
Operation 6	Tool 05	Machine Groove and 2 chamfers at the groove
Operation 7	Tool 07	Machine 1 3/8 - 12 UNF Threads

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

3.1.3 POINT INDENTIFICATION AND COORDINATE VALUES FOR EXTERNAL OPERATIONS

Using the drawing in **Figures 1,2, and 3,** solve the coordinates and fill in the point chart below**,** answers to follow. These points will be used when programming the facing and external turning operations. Study the notes on the drawing, as we write the program keep these notes in mind.

ACTIVITY 1, SOLVE POINT VALUES AND FILL IN THE POINT CHART

In this exercise solve the points and fill in the point chart below, answers to follow. Refer to **Figures 1 through 3** to calculate the values, remember the X axis values must be the diameter value.

CNC positions are based on the Cartesian Coordinate System, plotting points in x,y,z, and in this case using only x and z coordinates because the module concentrates on Lathe programming. To obtain CNC positions you must have a print with dimensions on it. Diameters would be your "x" locations and Lengths would be your "z" locations. All "z" values should be negative and all "x" values should be positive. Any "r" values would be the radii represented on the drawing.

Lathe Point Chart for External Operations (Refer to Figure 1 through 3)			
Point #	X Value (dia.)	Z Value	R Value
Start Point			
1			
2			
3			
4			

5			
6			
7			
8			
9			
end point			

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

3.1.4 RULES FOR PROGRAMMING MULTIPLE REPETITIVE CYCLES

For a full list of the special rules that must be followed when programming Multiple Repetitive Cycles, the machine manual provided with the purchase of a new CNC machine should be used. FANUC controls offer Type I and Type II cycles. This is determined by a machine control parameters. Type I would require that contour shapes maintain a steady increase or decrease in the part shape. Type II would allow the shape to increase then decrease in size as the tool moves along the path. For example if a diameter being roughed out had a groove along the path, with Type II the tool could dip into the groove, with Type I this would not be allowed.

Here are a few of the common rules to keep in mind that will help you understand tool placement before the Multiple Repetitive Cycle is executed for example and more.

- When programming with G70, G71, G72 or G73, the P and Q words is reference to a block numbers N##, the block number referenced by the P word must have a G00 or G01 on the block or the control will alarm No.65.
- *Sub Programs,* M98/M99 cannot be used during G70 through G73.
- Block numbers used in the program that are referenced with the P word and Q word in G70 through G73 Cycles, must not be repeated in the program.
  When programming G70 through G73, the location the tool is placed in X and Z axis is the place the tool will return to when the cycle is complete. Keep this in mind when programming to avoid the crashing across the part at the end of the cycle. Pay close attention in the example programs where the X and Z axis is located before the cycle is started.

3.1.5 OVEVIEW OF G70 THROUGH G76 CYCLES

The information given in this section will be specific to the FANUC Oi controls and very similar to most controls, the code will need some tweaking for other model controls, refer to machine manuals for specific changes. The programming format for the Multiple Repetitive Cycles below shows all possible codes for each cycle, some codes may be omitted. For example, if the surface speed and tool offset that is currently active, before programming the G71 two blocks, then it is not necessary to define the values on the second G71 block. Other codes on the block cannot be omitted, or the cycle will fail at the machine.

SC ACCELERATE
CLEMSON
CENTER FOR
WORKFORCE DEVELOPMENT

The following information will be used to program the example parts in this module and will be used for the lab project.

Multiple Repetitive Cycles (G70 – G76) programming format

G70 Cycle, Finishing Cycle

> **G70 P**(ns) **Q**(nf)
>
> **ns** = Sequence number of the first block of the finish shape.
>
> **nf** = Sequence number of the last block of the finish shape.

G71 Cycle, Stock Removal in Turning, Roughing Cycle

> **G71 U**(dd) **R**(ee)
> **G71 P**(ns) **Q**(nf) **U**(uu) **W**(ww) **F**(ff) **S**(ss) **T**(tt)
>
> **dd** = Depth of cut per side, will reduce the diameter by 2 times this amount.
>
> **ee** = Pull off amount at the end of each pass.
>
> **ns** = Sequence number of the first block of the finish shape.
>
> **nf** = Sequence number of the last block of the finish shape.
>
> **uu** = Amount of stock on the diameter to leave in the X axis for finishing.
>
> **ww** = Amount of stock to leave on the Z axis for finishing.
>
> **ff** = Feed rate used for roughing cycle.
>
> **ss** = Spindle surface speed for roughing.
>
> **tt** = Tool offset call for roughing.

G72 Cycle, Stock Removal in Facing, Rough Facing

> **G72 W**(dd) **R**(ee)
> **G72 P**(ns) **Q**(nf) **U**(uu) **W**(ww) **F**(ff) **S**(ss) **T**(tt)
>
> **dd** = Depth of cut in the Z axis.

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

ee = Pull off amount at the end of each pass.

ns = Sequence number of the first block of the finish shape.

nf = Sequence number of the last block of the finish shape.

uu = Amount of stock on the diameter to leave in the X axis for finishing.

ww = Amount of stock to leave on the Z axis for finishing.

ff = Feed rate used for roughing cycle.

ss = Spindle surface speed for roughing.

tt = Tool offset call for roughing.

G73 Cycle, Pattern Repeating, Roughing Cycle

G73 U(ii) **W**(kk) **R**(dd)
G73 P(ns) **Q**(nf) **U**(uu) **W**(ww) **F**(ff) **S**(ss) **T**(tt)

ii = Distance and direction of relief in the X axis direction (Radius amount).

kk = Distance and direction of relief in the Z axis direction.

dd = Number of divisions, same as the repetitive count for rough cutting.

ns = Sequence number of the first block of the finish shape.

nf = Sequence number of the last block of the finish shape.

uu = Amount of stock on the diameter to leave in the X axis for finishing.

ww = Amount of stock to leave on the Z axis for finishing.

ff = Feed rate used for roughing cycle.

ss = Spindle surface speed for roughing.

tt = Tool offset call for roughing.

G74 Cycle, End Face or Peck Drilling

SC ACCELERATE
CLEMSON
CENTER FOR
WORKFORCE DEVELOPMENT

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

G74 R(ee)
G74 X(U)____ **Z(W)**____ **P**(ii) **Q**(kk) **R**(dd) **F**(ff)

ee = Amount to pull off at the end of each peck.

X(U) = ABS location of hole when using X or INC amount when using U

Z(W) = ABS final depth with using Z, or INC amount when using W

ii = Movement amount in X direction, without sign.

kk = Depth of each peck without sign, no decimal allowed. Example Q2500 is .2500 peck amount.

dd = Relief amount of the tool at the bottom. Note: the sign of dd is always positive. However, if address X(U) and II are omitted, the relied direction can be specified by the desired sigh.

ff = Feed rate

G75 Cycle, Outer Diameter Groove / Internal Diameter Drilling

G75 R(ee)
G75 X(U)____ **Z(W)**____ **P**(ii) **Q**(kk) **R**(dd) **F**(ff)

ee = Amount to pull off at the end of each peck.

X(U) = ABS location of hole when using X or INC amount when using U

Z(W) = ABS final depth with using Z, or INC amount when using W

ii = Movement amount in X direction, without sign.

kk = Depth of each peck without sign, no decimal allowed. Example Q2500 is .2500 peck amount.

dd = Relief amount of the tool at the bottom. Note: the sign of dd is always positive. However, if address X(U) and ii are omitted, the relied direction can be specified by the desired sigh.

ff = Feed rate

G76 Cycle, Multiple Thread Cutting

G76 P(mm)(rr)(aa) **Q**(d Min) **R**(d)

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

G76 X(U) Z(W) R(ii) **P**(kk) **Q**(dd) **F**(ff)

mm = Number of spring passes

rr = Pull off direction,00 will be a x axis move, 10 will be X & Z =.1

aa = Tool infeed angle (60)

Example of the P word in the first G76 block: P010060 this will be one spring pass, pull up in X at the end of each pass, and 60 degree thread angle. No decimal allowed.

d Min = Minimum pass depth, after the first pass, this is the amount per side. No decimal allowed, example Q0020 this would equal .0020 minimum pass depth.

d = Finish pass amount, (R0) No decimal allowed.

X(U) = Root diameter of thread, last pass threading dia.

Z(W) = Threading length in Z axis, final Z axis value.

ii = Difference of thread radius for tapered thread. Use 0 for straight threads (R0)

kk = Height of thread, single side depth. No decimal allowed, example P0455 is .0455 single side thread depth.

dd = Depth of cut for 1st cut only, radius value. No decimal allowed, example Q0150 is .0150 depth of cut first pass.

ff = Feed rate, thread lead.

3.1.6 WRITING THE PROGRAM FOR EXTERNAL OPERATIONS

Study the program below and reference to information and rules in section 3.1.4 and 3.1.5 (above) to help you understand the codes in the Multiple Repetitive Cycles. Pay close attention to the colored areas in red and blue.

%
O0001 (MTT 252 MOD 3 LATHE EXAMPLE EXTERNAL OPERATIONS)

```
G20
( TOOL - 1 OFFSET - 1 )
(  ROUGH FACE )
G0 T0100
G18
G97 S1000 M03
G0 X2.05 Z.1 T0101
G50 S1500
G96 S600
( STOCK REMOVAL IN FACING CYCLE )
G72 W.025 R.025
G72 P1 Q2 U0. W.02 F.008
( P1 AND Q2 IN THE SECOND G72 IS A REFERANCE TO BLOCK NUMBERS N1 AND N2 BELOW )
( BLOCK N1 THROUGH N2 DEFINES THE FINISH SHAPE AND FINISH FEED RATE, ALSO )
( THIS CYCLE CAN CUT CONTOURED FACES IF NEEDED, DOES NOT HAVE TO BE FLAT )
N1 G1 Z0. F.004
X-.032
N2 Z.1
( ROUGH OUTSIDE )
Z.1
X2.05
( STOCK REMOVAL IN TURNING CYCLE )
G71 U.05 R.05
G71 P10 Q20 U.04 W.02 F.008
( P10 AND Q20 IS REFERANCED TO BLOCK NUMBERS N10 AND N20 BELOW )
( BLOCK N10 THROUGH N20 DEFINES THE FINISH SHAPE AND FINISH FEED RATE )
N10 G0 X1.055
G1 G42 X1.255 Z0. F.004
X1.375 Z-.06
Z-.75
X1.49
X1.62 Z-1.3684
Z-1.75
X1.74
Z-2.
X2.
N20 G40 X2.05
G0 Z.1


Continued

G28 U0. W0. M05
M01
```

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

```
(TOOL - 3 OFFSET - 3)
( FINISH FACE )
G0 T0300
G97 S1000 M03
G0 X1.425 Z0. T0303
G50 S1500
G96 S600
G1 X-.032 F.005
G0 Z.1
( FINISH TURN OUTSIDE )
X2.05
( FINISHING CYCLE )
G70 P10 Q20
( P10 AND Q20 IS REFERANCED TO BLOCK NUMBERS N10 AND N20 ABOVE )
G0 Z.1
G28 U0. W0. M05
M01
(TOOL - 5 OFFSET - 5)
(OD GROOVE - NARROW  INSERT - GC-4094)
G0 T0500
G97 S1000 M03
G0 X1.55 Z-.75 T0505
G50 S1500
G96 S600
( GROOVE CYCLE )
G75 R.15
G75 X1.251 Z-.656 P9000   Q3000   R0. F.008
 ( NOTE: WHEN A GROOVE SHAPE HAS CHAMFERS )
( THEN G75 WILL NOT WORK LONG HAND PROGRAMMING WOULD BE REQUIRED )
(NEXT RAPID TO SECOND GROOVE WITH CHAMFERS)
G0 X1.7685
Z-1.5316
( THIS GROOVE HAS CHAMFERS, THEREFORE G75)
(CANNOT BE USED, LONG HAND PROGRAMMING IS USED INSTEAD)
G1 X1.6271 Z-1.6023
X1.5941 Z-1.6188
X1.45
G0 X1.7685
Z-1.7062
G1 X1.6271 Z-1.6355
X1.5941 Z-1.619
X1.45
G0 X1.7685
G28 U0. W0. M05
```

Continued

```
M01
(TOOL - 7 OFFSET - 7)
( 1 3/8-12 THREAD )
G0 T0700
G97 S400 M03
G0 X1.575 Z.2163 T0707
( MULTIPLE THREAD CUTTING CYCLE )
G76 P010029 Q0. R0.
G76 X1.2848 Z-.7 P0451 Q0175 R0. F.08333
( ALL THREAD DATA IS DEFINED BY THE PAIR OF BLOCKS G76 )
G28 U0. W0. M05
M30
%
```

ACTIVITY 2, PROGRAMMING REVIEW

Review question about the program on the pervious pages.
Multiple choice, circle the best answer

1. In the G71 cycle, what are the codes P10 and Q20 used for in the program?

    a. Stock amount to leave in X and Z axis

    b. Reference to blocks N10 and N20

    c. How many cuts to make in the X and Z axis

    d. Sub program call numbers

2. In the threading cycles what roll does the P0451 code play?

    a. Rough amount for threading

    b. Diameter of thread

    c. Thread pitch amount

    d. Height of thread, single side depth

3. Which cycle was used to groove the neck (first groove)?

    a. G72

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

  b. G71

  c. G75

  d. G70

4. What is the depth of cut amount in the Z axis in the facing cycle?

  a. .0025

  b. .025

  c. .050

  d. .250

5. What code or codes control the stock amount to leave on diameters and faces in the roughing cycle?

  a. U and W

  b. F

  c. P and Q

  d. U and R

3.1.7 PROGRAM SIMULATION AND VIDEO FOR EXTERNAL OPERATIONS

You will notice the simulation is just that, a simulation, the part will not really be turning and speeds will not be accurate during simulation. The second video will show real time stock removal at the CNC machine. Be sure to notice what operations are taking place with each different tool, and other details of the videos. There is an activity after the videos.

Video 1, (ctrl + click link) http://youtu.be/ne6iV6mWcxI was taken using CAD/CAM software simulation.

Video 2, (ctrl + click link) http://youtu.be/PXG4Bs2KsYw was taken at the CNC Lathe running the program shown in 1.1.5 above. This video shows Multiple Repetitive Cycles G70, G71, G72, G75 and G76.

ACTIVITY3

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

1. In video 2, the groove tool required will require the spindle to rotate in which direction? Note: This could have been edited at the machine during prove out to match available tools at the time.
    a. CW M03
    b. CCW M04
    c. CSS G96
    d. None of these
2. What stock amount was left by the roughing face tool for finishing later with the next tool?
    a. .02
    b. .025
    c. .05
    d. None of these
3. The feed rate of F.01 in the G 71 roughing cycle is used for?
    a. Rough and finish cuts
    b. Rough cuts
    c. Finish cuts
    d. None of these

3.1.8 FURTHER STUDY

Other Related Video Links, below are some links on YouTube that use Multiple Repetitive Cycles.

Related Video Link 1 https://www.youtube.com/watch?v=kRMmYxOZ6mU

Related Video Link 2 https://www.youtube.com/watch?v=jOgkdnbrQYM

3.2 CNC TURNING CENTER: PROGRAMMING INTERNAL OPERATIONS USING MILTIPLE REPETITIVE CYCLES

In this section we will program Internal Lathe Operations using Multiple Repetitive Cycles G70, G71, and G74, for this process.

3.2.1 LATHE PART PRINT WITH ADDED INTERNAL MACHINING

Study the prints in **Figure 4 and 5** below that will be used in the following programming example. This is the same example part print that was used for the External Operations above with the additional Internal work. The prints in **Figure 4 and 5** will only show the internal area added to the part.

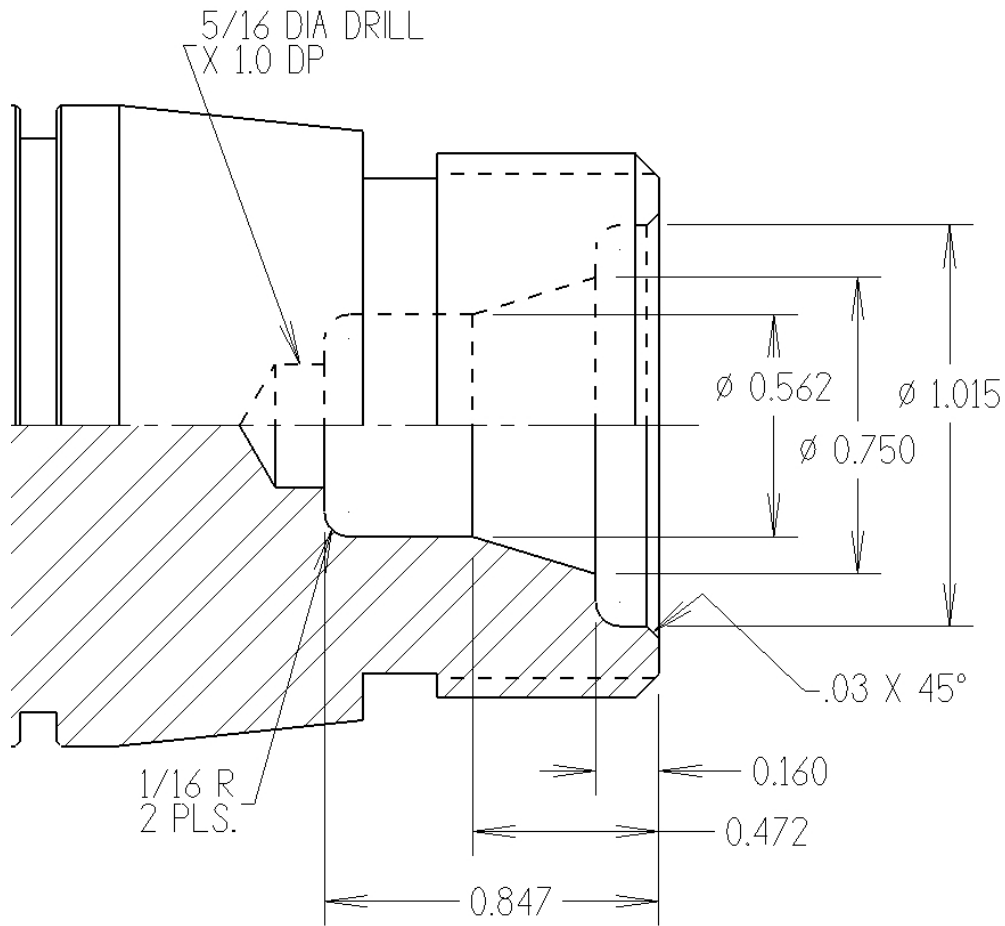**SC ACCELERATE** | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

Figure 4, Lathe Part Print 2 (for internal programming)
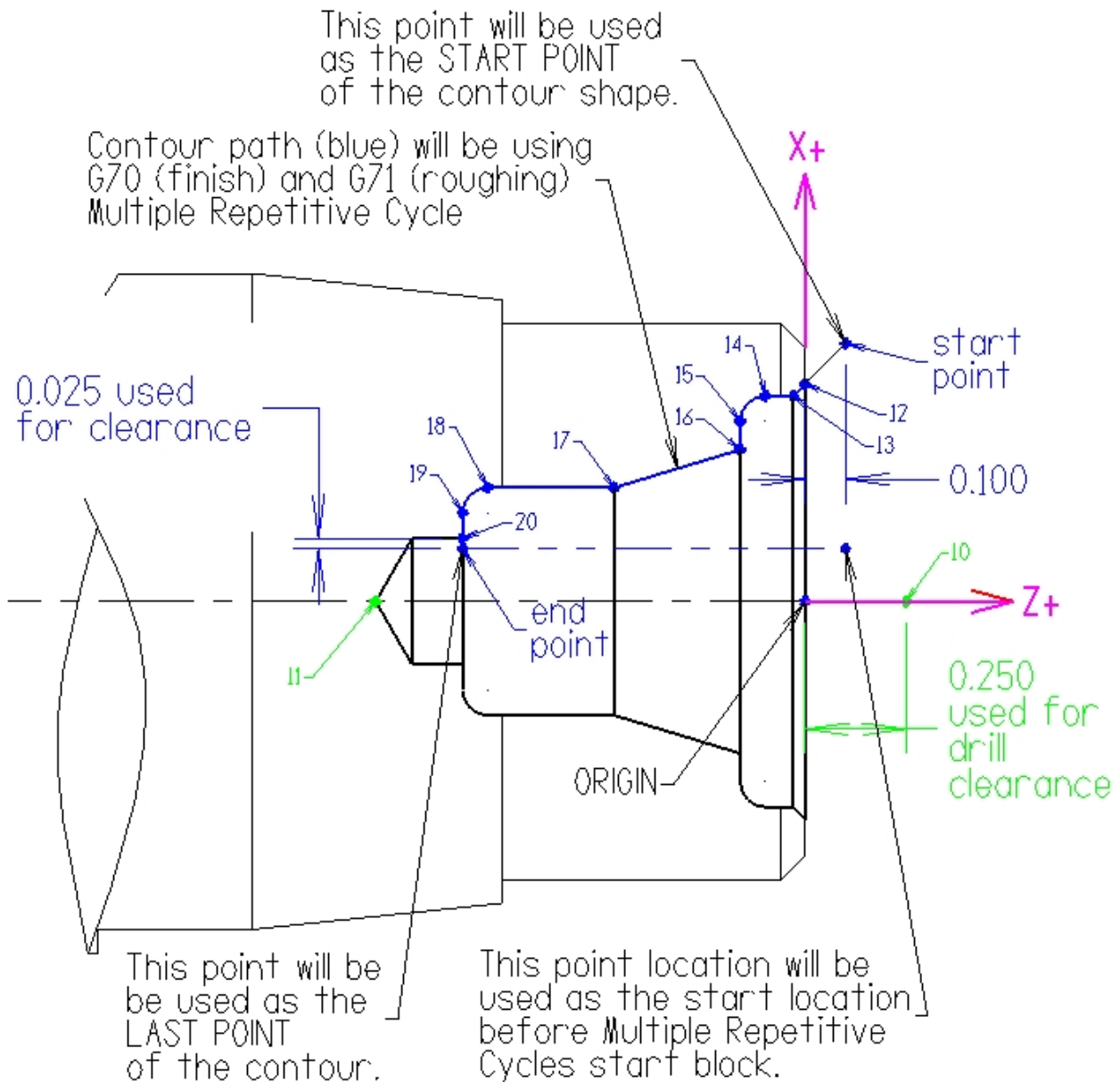
Figure 5: Part drawing used to identify points needed for Internal Operations.

SC ACCELERATE   MTT 252

Open Text   CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

3.2.2 PROGRAM PLANNING FOR INTERNAL OPERATIONS

After studying the revised prints, **Figures 4 and 5**, I have listed below a machining plan as follows:
We will continue to add to the previous list of operations and pick the next available tool numbers.

Operation 8	Tool 09	Drill 5/16 dia X 1. deep, G74 Peck Drilling Cycle
Operation 9	Tool 11	Rough Bore Inside Shape leave .02 on X and Z surfaces, G71 Roughing Cycle
Operation 10	Tool 11	Finish Bore Inside Shape, G70 Finishing Cycle

3.2.3 POINT VALUES FOR INTERNAL OPERATIONS

ACTIVITY 4, SOLVE POINT VALUES AND FILL IN THE POINT CHART

Using the drawing in **Figures 4 and 5,** solve the coordinates and fill in the point chart below**,** answers to
follow. These points will be used when programming the facing and outside turning operations. Study the
notes on the drawing, as we write the program and keep the notes in mind, remember the X axis values
must be the diameter value.

Lathe Point Chart using **Figure 4 and 5**			
Point #	X Value (dia.)	Z Value	R Value
10			
11			
Start Point			
12			
13			
14			
15			
16			
17			
18			

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

19			
20			
end point			

3.2.4 G71 CYCLE APPLIED TO INTERNAL OPERATIONS

The G70 and G71 cycle is the same cycle that was used for roughing and finishing the outside contour of the example part. When we programmed these cycles for external turning the tool was located at a point in front of the stock and just above the bar stock diameter. To use these same cycles for internal work we must first drill a hole for the boring tool. We will be working from the opposite direction than when working on external operations. With this in mind, the boring bar must be located before the start of the cycle, in front of the stock, and just below the drill diameter. The block numbers used to reference P(ns) and Q(nf) must be numbers that have not been used in the program already. The U(ww) in the second G76 block must be negative when roughing from small diameter to large diameter.

Multiple Repetitive Cycles (G70, G71, and G74) programming format

G70 Cycle, Finishing Cycle

G70 P(ns) **Q**(nf)

ns = Sequence number of the first block of the finish shape.

nf = Sequence number of the last block of the finish shape.

G71 Cycle, Stock Removal in Turning, Roughing Cycle

G71 U(dd) **R**(ee)
G71 P(ns) **Q**(nf) **U**(uu) **W**(ww) **F**(ff) **S**(ss) **T**(tt)

dd = Depth of cut per side, will reduce the diameter by 2 times this amount.

ee = Pull off amount at the end of each pass.

ns = Sequence number of the first block of the finish shape.

nf = Sequence number of the last block of the finish shape.

uu = Amount of stock on the diameter to leave in the X axis for finishing. (must be negative with programming internal contours)

ww = Amount of stock to leave on the Z axis for finishing.

ff = Feed rate used for roughing cycle.

ss = Spindle surface speed for roughing.

tt = Tool offset call for roughing.

G74 Cycle, End Face or Peck Drilling

G74 R(ee)
G74 X(U)___ **Z(W)**___ **P**(ii) **Q**(kk) **R**(dd) **F**(ff)

ee = Amount to pull off at the end of each peck.

X(U) = ABS location of hole when using X or INC amount when using U

Z(W) = ABS final depth with using Z, or INC amount when using W

ii = Movement amount in X direction, without sign.

kk = Depth of each peck without sign, no decimal allowed. Example Q2500 is .2500 peck amount.

dd = Relief amount of the tool at the bottom. Note: the sign of dd is always positive. However, if address X(U) and II are omitted, the relied direction can be specified by the desired sigh.

ff = Feed rate

3.2.5 WRITING THE PROGRAM FOR INTERNAL OPERATIONS

Study the program below and reference the information in section 1.2.4 (above) and 1.1.4 (above) to help you understand the codes used in the Multiple Repetitive Cycles.

```
%
O0002 ( MTT 252 MOD 3 LATHE EXAMPLE INTERNAL OPERATIONS )
G20
```

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

```
( TOOL - 9 OFFSET - 9 )
( DRILL 5/16 HOLE )
G0 T0900
G97 S1200 M03
G0 X0. Z.25 T0909
Z.1
( END FACE PECK DRILLING CYCLE )
G74  R.05
G74  Z-1.0659 Q2500 F.01
( ALL DRILLING DATA IS DEFINED BY THE PAIR OF BLOCKS G74 )
G0 Z.25
G28 U0. W0. M05
M01
```

Continued

```
(TOOL - 11 OFFSET - 11)
( ROUGH BORE INSIDE )
G0 T1100
G97 S1500 M03
G0 X.2625 Z.1 T1111
G50 S1500
G96 S600
( STOCK REMOVAL IN TURNING CYCLE )
G71 U.05 R.05
G71 P100 Q200 U-.04 W.02 F.005
( P100 AND Q200 IS REFERANCED TO BLOCK NUMBERS N100 AND N200 BELOW )
( BLOCK N100 THROUGH N200 DEFINES THE FINISH SHAPE AND FINISH FEED RATE )
N100 G0 X1.275
G1 G41 X1.075 Z0.0 F.004
X1.015 Z-.03
Z-.0975
G3 X.89 Z-.16 R.0625
G1 X.75
X.562 Z-.472
Z-.7845
G3 X.438 Z-.847 I-.0625
G1 X.3125
N200 G1 G40 X.2625
G0 Z.1
( FINISH BORE )
G50 S1500
```

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

(FINISHING CYCLE)
G70 P100 Q200
(**P100 AND Q200** IS REFERANCED TO BLOCK NUMBERS **N100 AND N200** ABOVE)
G0 Z.1
G28 U0. W0. M05
M30
%

3.2.6 PROGRAM SIMULATION AND VIDEO FOR INTERNAL OPERATIONS

Video 3, "ctrl + click link" Video 3 was taken using CAD/CAM software simulation.

Video 4, "ctrl + click link" Video 4 was taken at the CNC Lathe running the program shown in 1.2.5 above. This video shows Multiple Repetitive Cycles G70, G71, and G74.

Suggested Activity

ACTIVITY BASED ON VIDEO 3 AND 4

Match the correct letter to the blanks to finish the statements.

The _____ is defined by two blocks.
The retract amount between pecks is set by the _____.
Final Z axis depth, feed rate, and peck amount is controlled by the _____.
The _____ and _____ in the second G71 block are used to define the finish part shape.
To leave stock for finishing, the U word must be _____ in the second G71 block.
The _____ is only one block of code.
The P and Q words used in the G70 Cycle reference the block numbers used in the _____ Roughing Cycle.

a. second G74 block
b. P word
c. negative
d. R word
e. G70 Cycle
f. G74 Cycle
g. Q word
h. G71

SC ACCELERATE
CLEMSON
CENTER FOR
WORKFORCE DEVELOPMENT

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

3.2.7 FURTHER STUDY

Other Related Video Link, below is a link on YouTube that uses Multiple Repetitive Cycles.

Related Video Link 3, "ctrl + click link" https://www.youtube.com/watch?v=KSRY7wYKHEQ

MAJOR CONCEPTS

KEY CONCEPTS WHEN PROGRAMMING WITH MULTIPLE REPETITIVE CYCLES

- Multiple Repetitive Cycles is used to make programming of repetitive cuts for facing, turning, grooving, and threading easier to program and makes the program much shorter is size.
- When programming rough cuts for turning (external) and boring (internal), the G71 cycle can make complex roughing cuts easy by only programming the finish shape to be used for roughing and finishing.
- Threading can be defined with two blocks using G76 that will cut the thread all the way to finish size. When using other codes like G32, each thread pass takes one block to position to cut depth and then the G32 to make one pass, this could take around 20 to 30 blocks or more of code to do what can be done with two G76 blocks.
- For lab projects the final prove out will be done in the lab and will determine if the program is ready for production by using a CNC simulator before going to the CNC machine in the lab. During the lab exercise you will edit tool number codes and spindle rotation codes as needed. Once the adjustments have been make to the program and the prove out is complete, then the job is released to production.

KEY TERMS

SC ACCELERATE

CLEMSON
CENTER FOR
WORKFORCE DEVELOPMENT

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

External Lathe Operations – *work performed on the external area of the part including the front face. This will include operations, front facing, turning the outside shape, grooves on the outside diameter and face, threading on the outside diameter.*

Internal Lathe Operations - *work performed on the inside area of the part. This will include operations, hole operations (spot drill, drill, ream, tap, etc), turning the inside shape, grooves on the inside diameter, threading on the inside diameter.*

Multiple Repetitive Cycles – *Special features programmed into the CNC control that will aid in programming. The cycles make the programming process faster and make editing easier.* Using the cycles will shorten the program thus saving memory space in the CNC control.

Sub Programs – *portions of a program that is repetitive may be removed from the original program and stored in the control. These sub programs can be called by the main program as often as needed by the main program.*

LABS

PHYSICAL LAB USING A 2 AXIS TURNING CENTER

- Lab exercises will be done at the CNC machine using a 2 axis turning center, if not available a CNC simulator can be used but will not be able to fully complete the lab exercise.

-Materials needed for the lab is the CNC program, 2" diameter wax bar stock 4" long, and a list of the tools needed for the setup process.

-Requirements for successful lab completion will be to write the CNC program, run a program simulation, make necessary edits, setup and prove out a CNC program on the CNC machine using Multiple Repetitive Cycles.

-Programming instructions for the lab exercise using Print 3 in Figure 10:

-Write a CNC program using Multiple Repetitive Cycles when possible for the following lathe operations. The part will be chucked on the 2.0 REF. diameter, this diameter should not be included in the turning operation.

- Rough face
- Rough turn outside
- Finish face
- Finish turn outside
- Groove neck

- Thread external threads
- Drill center hole
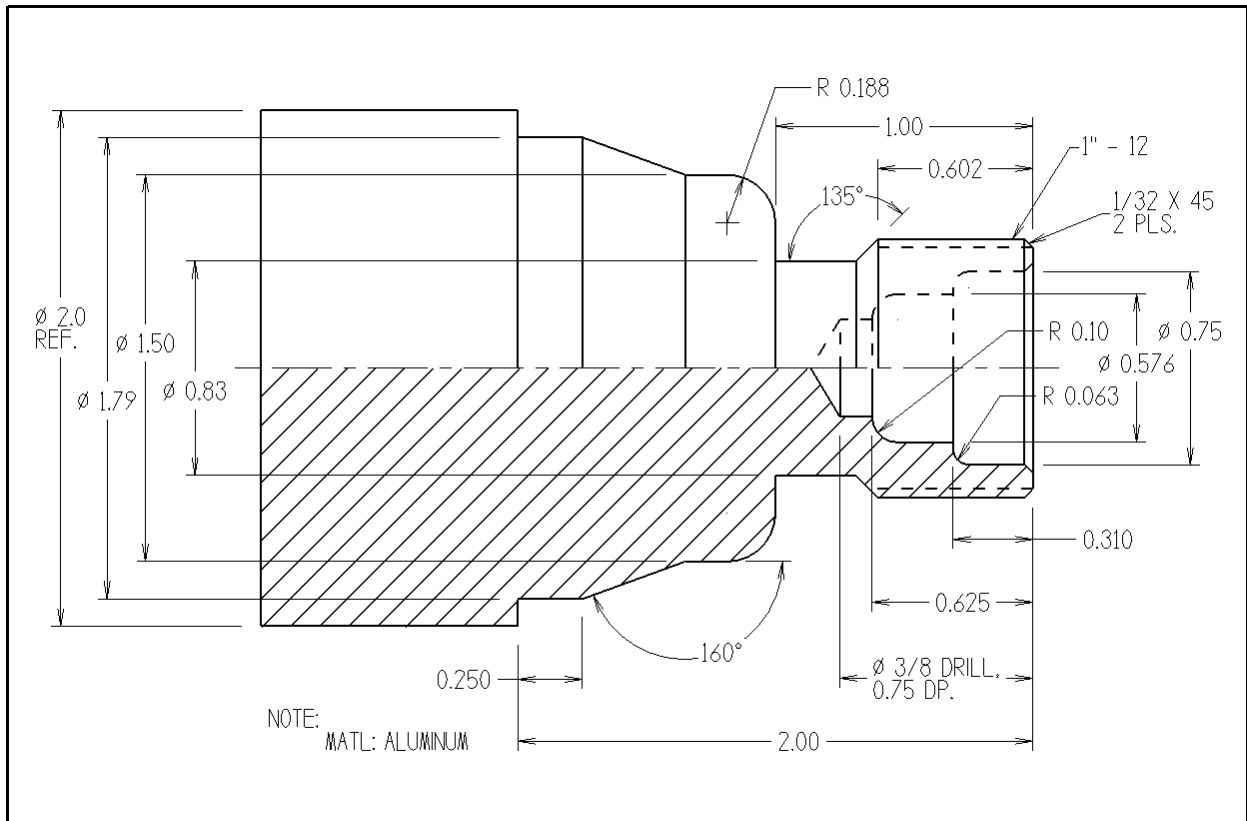- Rough bore inside
- Finish bore inside

Figure 6, Lathe Part Print 3 (for lab project)

ASSESSMENT

True or False: Read the following questions and determine whether the statement is true or false.

1. **True or False** The most common cycle used for rough turning and rough boring is G71, however it is possible to use G72 if desired.

2. **True or False** If the machine control has enough memory to hold programs that are written long hand, then there is no advantage to use Multiple Repetitive Cycles.

3. **True or False** Each CNC program can only contain one roughing cycle and one finish cycle, if the part requires external and internal work where the roughing and finish cycle needs to be used again, then it must be placed in a sub program.

4. **True or False** Tool nose radius compensation is not active in the roughing cycles.

5. **True or False** The "P" word in the first G76 block sets spring passes, pull off direction, and tool infeed angle.

Multiple Choice: Read the following questions or statements and select the best answer.

1. What feed rate will be active when the G70 cycle is executed?

   a) by the last active feed rate given before the start of the cycle
   b) by the feed rate given on the G70 cycle block
   c) by the feed rate given on the G71 cycle block
   d) by the feed rate given between block **P**(ns) **Q**(nf) that define the part shape

2. What code/word, in the two G71 blocks, controls the depth of cut for each roughing pass?

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

a) by the "R" word
b) by the "U" word in the second G71 block
c) by the "U" word in the first G71 block
d) by the "W" word in the second block

3. What code/word, in the peck drilling cycle, controls the amount of each peck?

a) by the "Q" word
b) by feed rate times the final Z depth amount
c) by the "R" word
d) by the Z depth amount divided by the drill diameter

4. What must be changed, when using G71 cycle for rough boring vrs rough turning, that controls the stock to leave in the X axis direction?

a) start and stop location must be inside the drill diameter (X axis)
b) the "U" word must be negative
c) change the "P" and "Q" words for the bore shape
d) all of the above

5. What Multiple Repetitive Cycle is use to cut threads?

a) G74
b) G76
c) G70
d) G96

SC ACCELERATE
CLEMSON
CENTER FOR
WORKFORCE DEVELOPMENT

CRITICAL THINKING QUESTIONS

1. <u>Directed Paraphrasing</u>: In your own words create an original definition or explanation of the use of the G71 roughing cycle presented in this module.

2. <u>Short Summary</u>: Create a short summary of the advantages of using Multiple Repetitive Cycles over long hand cut by cut G code programming in 3 sentences or less that answer most or all of the following questions; "When and where multiple repetitive cycles are used, how is multiple repetitive cycles implemented in the program and why is multiple repetitive cycles needed for programming?"

ANSWER KEY

ANSWERS TO ACTIVITY 1

Lathe Point Chart Answers			
Point #	X Value (Dia.)	Z Value	R Value
start point	1.055	0.10	
1	1.255	0.0	
2	1.375	-0.06	
3	1.375	-0.75	
4	1.49	-0.75	
5	1.62	-1.3684	
6	1.62	-1.75	
7	1.74	-1.75	
8	1.74	-2.00	
9	2.00	-2.00	

end point	2.05	-2.00	

ANSWERS TO ACTIVITY 2

1. b
2. d
3. c
4. b
5. a

ANSWERS TO ACTIVITY 3

The ____f.____ is defined by two blocks.
The retract amount between pecks is set by the ___d.____.
Final Z axis depth, feed rate, and peck amount is controlled by the ___a.___.
The ___b.___ and ___g.___ in the second G71 block are used to define the finish part shape.
To leave stock for finishing, the U word must be ___c.___ in the second G71 block.
The ___e.____ is only one block of code.
The P and Q words used in the G70 Cycle reference the block numbers used in the ___h.___ Roughing Cycle.

ANSWERS TO ACTIVITY 4

Lathe Point Chart Answers for **Activity 4**			
Point #	X Value (dia.)	Z Value	R Value
10	0.0	0.25	
11	0.0	-1.062	
Start Point	1.275	0.1	
12	1.075	0.0	
13	1.015	-0.03	
14	1.015	-0.0975	
15	0.890	-0.16	0.0625

SC ACCELERATE
CLEMSON
CENTER FOR
WORKFORCE DEVELOPMENT

16	0.75	-0.16	
17	0.562	-0.472	
18	0.562	-0.785	
19	0.438	-0.847	0.0625
20	0.3125	-0.847	
end point	0.2625	-0.847	

ANSWERS TO ASSESMENTS

Answer Key for Assessments

True or False	Multiple Choice
1. True 2. False 3. False 4. True 5. True	1. d 2. c 3. a 4. d 5. b

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

CRITICAL THINKING ANSWERS

1. The G71 cycle can make complex roughing cuts easy by only programming the finish shape to be used for roughing and finishing

2. Multiple Repetitive Cycles are used to make programming of repetitive cuts for facing, turning, grooving, and threading easier to program and shorter in size. It contains parameters needed for depth of cut, rough stock to leave on surfaces and diameters, and the feed rate. Without the use of the Multiple Repetitive Cycles, the programmer will have to program all the rough cuts long hand, rough cuts long hand that will take a minimum of 4 blocks per rough pass.

SC ACCELERATE
CLEMSON
CENTER FOR
WORKFORCE DEVELOPMENT

SC ACCELERATE | **MTT 252**

Open Text | **CNC Setup and Operations (Hybrid)**

Module# 3

CNC Lathe Operations

ATTRIBUTION TABLE

Author/s	Title	Source	License
Glenn Galloway	Figure 1	h	Creative Commons Attribution 3.0 Unported License
Glenn Galloway	Figure 2	h	Creative Commons Attribution 3.0 Unported License
Glenn Galloway	Figure 3	h	Creative Commons Attribution 3.0 Unported License
Glenn Galloway	Figure 4	h	Creative Commons Attribution 3.0 Unported License
Glenn Galloway	Figure 5	h	Creative Commons Attribution 3.0 Unported License
Glenn Galloway	Figure 6	h	Creative Commons Attribution 3.0 Unported License
Glenn Galloway	Video 1	http://youtu.be/ne6iV6mWcxI	Creative Commons Attribution 3.0 Unported License
Glenn Galloway	Video 2	http://youtu.be/PXG4Bs2KsYw	Creative Commons Attribution 3.0 Unported License
Allen Wrench	Related Video Link 1	https://www.youtube.com/watch?v=kRMmYxOZ6mU	Creative Commons Attribution 3.0 Unported License
buschbwbusch	Related Video Link 2	https://www.youtube.com/watch?v=jOgkdnbrQYM	Creative Commons Attribution 3.0 Unported License
Glenn Galloway	Video 3	http://youtu.be/iD75xLHyPKI	Creative Commons Attribution 3.0 Unported License

SC ACCELERATE | MTT 252

Open Text | CNC Setup and
Operations (Hybrid)

Module# 3

CNC Lathe
Operations

Author/s	Title	Source	License
Glenn Galloway	Video 4	https://youtu.be/wBblqQLp790	Creative Commons Attribution 3.0 Unported License
Tensor Cadcam	Related Video Link 3	https://www.youtube.com/watch?v=KSRY7wYKHEQ	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.