camInstructor

Computer Numerical Control Programming Workbook

Generic Lathe

Computer Numerical Control Workbook – Generic Lathe
Published by
CamInstructor Incorporated
330 Chandos Crt.
Kitchener, Ontario
N2A 3C2
www.caminstructor.com

Date: September 1, 2010
Author: Matthew Manton and Duane Weidinger
ISBN: 978-1-897466-83-4

National Library of Canada Cataloguing in Publication

To order additional copies of the book contact:
CamInstructor Inc.
330 Chandos Crt, Kitchener, ON, N2A 3C2
Phone 1-877-873-6867
Fax 1-866-741-8421
email sales@caminstructor.com

Printed in Canada

Requirements
Use of the Multi-media CD/DVD requires a computer with speakers, and CD/DVD ROM.
March 25, 2011

TABLE OF CONTENTS

1. AUTOMATIC TOOL CHANGER STANDARD TOOL CAROUSEL 4
2. COMMONLY USED PREPARATORY G CODES 5
3. COMMONLY USED MISCELLANEOUS M CODES 6
4. EXAMPLE OF PROGRAM START-UP LINES 7
5. EXAMPLE OF PROGRAM ENDING LINES 8
6. EXAMPLE OF PROGRAM TOOL CHANGE LINES 8
7. ABSOLUTE & INCREMENTAL POSITIONING 9
    a. **EXERCISE # 7-1 ~ # 7-5**
8. RAPID (G00) AND LINEAR(G01) INTERPOLATION 14
    a. LINEAR INTERPOLATION: **EXERCISE # 8-1 ~ # 8-4** 15
9. DRILL CANNED CYCLE (G81) 19
    a. DRILL CANNED CYCLE: **EXERCISE # 9-1** 20
10. DEEP HOLE PECK DRILL CANNED CYCLE (G83) 22
11. CIRCULAR INTERPOLATION (G02 & G03) : **EXERCISE # 11-1** 23
    a. G02 CIRCULAR INTERPOLATION : **EXERCISE # 11-2** 24
    b. G03 CIRCULAR INTERPOLATION : **EXERCISE # 11-3** 27
    c. CIRCULAR INTERPOLATION: **EXERCISE # 11-4 ~ # 11-7** 29
12. CUTTER COMPENSATION (G40, G41, & G42) 35
13. CNC PROGRAMMING: **EXERCISE # 13** 37
14. CNC PROGRAMMING: **EXERCISE # 14** 40

1 AUTOMATIC TOOL CHANGER STANDARD TOOL CAROUSEL

The CNC Machining Center used in this text is set-up with following tools. All program examples and exercises in this workbook are using the same tools.

Carousel #	Tool Description
1	**O.D. Right Hand Roughing Tool 80°**
2	**O.D. Right Hand Finishing Tool 55°**
3	**Rough Boring Bar Min. Ø0.375**
4	**Rough Boring Bar Min. Ø0.75**
5	**Finish Boring Bar Min. Ø0.375**
6	**Finish Boring Bar Min. Ø0.75**
7	**O.D. Thread Tool**
8	**# 4 Centre Drill**
9	**O.D. Right Hand Groove Tool W 0.125**
10	**O.D. Right Hand Parting Tool W 0.125**
11	**Open Pocket for Variable Tooling**

2 COMMONLY USED PREPARATORY **G** CODES

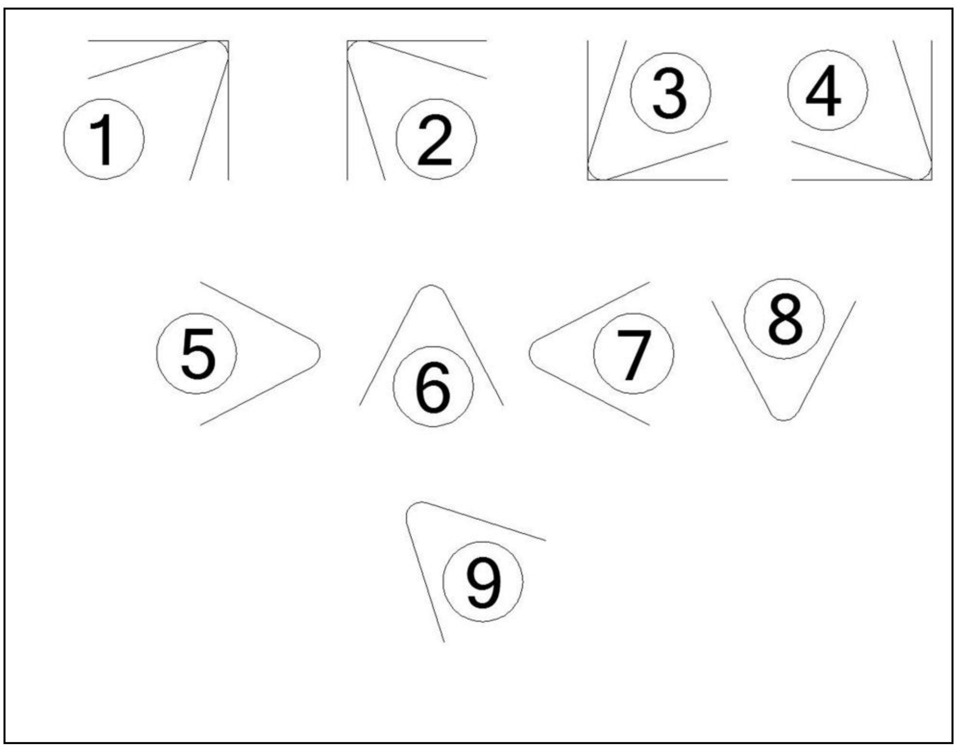
CODE	FUNCTION
G00 *	Rapid traverse motion; Used for non-cutting rapid moves of the machine axis to a location to be machined, or rapid retract moves after cuts have been completed. Maximum rapid motion (I.P.M.) of a CNC Machine will vary on machine model.
G01 *	Linear interpolation motion; Used for actual machining and metal removal. Governed by a programmed feedrate in inches (or mm) per minute. Maximum feed rate (I.P.M.) of a CNC Machine will vary depending on the model of the machine.
G02 *	Circular Interpolation, Clockwise
G03 *	Circular Interpolation, Counterclockwise
G04	Dwell
G18	ZX Plane Selection
G20	Verify Inch Coordinate Positions
G21	Verify Metric Coordinate Positions
G28	Machine Home (Rapid traverse)
G40	Tool Nose Radius Compensation CANCEL *
G41	Tool Nose Radius Compensation LEFT of the programmed path *
G42	Tool Nose Radius Compensation RIGHT of the programmed path *
G50	Max RPM Preset
G52	Local Coordinate system setting
G54-G59	Work Coordinate #1-#6 (Part zero offset location)
G68	Mirror Image for double turrets
G69	Mirror Image CANCEL
G70	Profile Finish Turning fixed cycle
G71	Profile Rough Turning fixed cycle – Z axis direction
G72	Profile Rough Turning fixed cycle – X axis direction
G73	Pattern Repetition cycle
G74	Drilling Cycle
G75	Grooving cycle
G76	Threading cycle

* Programming exercises included

CODE	FUNCTION
G96	Constant Surface Speed (CSS)
G97	Direct RPM Input Mode (cancels CSS mode)
G98	Feed Rate per Minute
G99	Feed Rate per Revolution

As you may have noticed, there are no Incremental or Absolute modes included in the Preparatory Codes. On a CNC turning center or Lathe, the mode is always set to Absolute and diameter, if an Incremental movement is required the letters U or W are used for X or Z respectively.

* Most lathe tools have a radius on the front or cutting edge; it is referred to as Tool Nose Radius. This radius must be compensated for in the calculation of the toolpath much like the cutter radius offset in milling operations, this offset is known as Tool Nose Radius Compensation.



Toolpaths are programmed using the coordinates of the true profile, much like a mill but because there are many possible positions of the tool point called the “command point” we have to enter the position from 1 to 9 in the tool (T) column on the offset page during part set-up.

3 COMMONLY USED MISCELLANEOUS **M** CODES

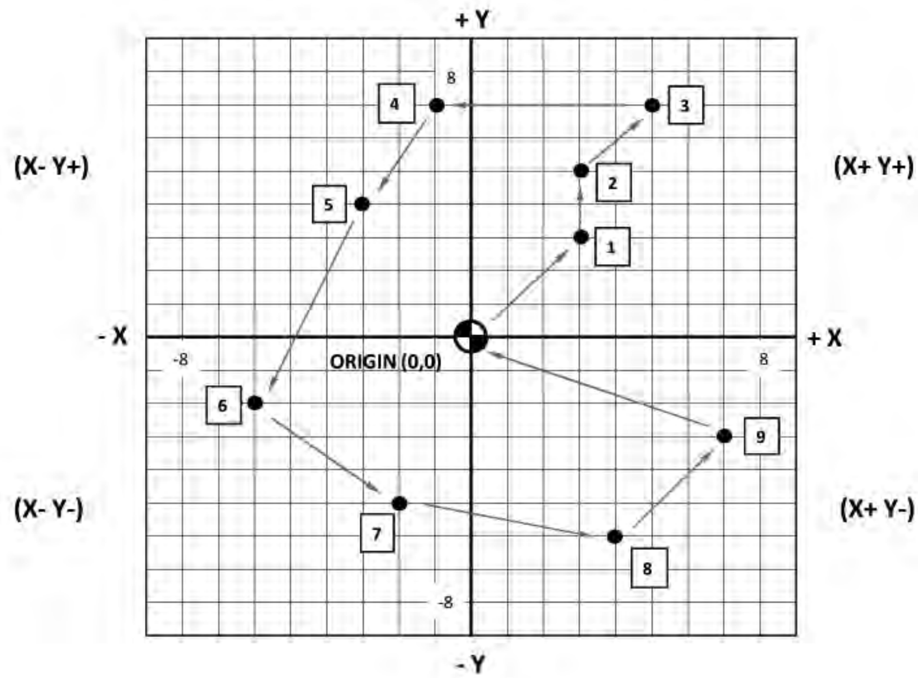
CODE	FUNCTION
M00	The M00 code is used for a Program Stop command on the machine. It stops the spindle, turns off coolant and stops look-a-head processing. Pressing CYCLE START again will continue the program on the next block of the program.
M01	The M01 code is used for an Optional Program Stop command. Pressing the OPT STOP key on the control panel signals the machine to perform a stop command when the control reads an M01 command. It will then perform like an M00. Optional stops are useful when machining the first part to allow for inspection of the part as it is machined.
M03	Starts the spindle CLOCKWISE for most machining. Must have a spindle speed defined. The M03 is used to turn the spindle on at the beginning of program or after a tool change.
M04	Starts the spindle COUNTERCLOCKWISE. Must have a spindle speed defined.
M05	STOPS the spindle. If the coolant is on, the M05 will turn it off.
M08	Coolant ON command.
M09	Coolant OFF command.
M10	Open Chuck
M11	Close Chuck
M12	Tailstock Quill IN
M13	Tailstock Quill OUT
M17	Turret Indexing Forward
M18	Turret Indexing Reverse
M19	Spindle Orientation
M21	Tailstock Forward
M22	Tailstock Backward
M23	Thread Gradual pullout ON
M24	Thread Gradual pullout OFF
M30	Program End and Reset to the beginning of program.
M41	Low Gear selection
M42	Medium Gear selection 1
M43	Medium Gear selection 2
M44	High Gear selection

4 EXAMPLE OF PROGRAM **START-UP LINES**

%	Programs must begin and end with “%” (depending on the type of control.)	
O00023 ;	Letter “O” and up to a five digit program number. Blocks are always terminated by the “;” symbol: End of Block (EOB)	
N10 G20 ;	Nnn - Sequence Number G20 - Verify Inch	Startup Block (Machine Default Setting)
N20 G18 G40 G80 ;	G18 - X,Z Circular Plane Selection G40 – Tool Nose Radius Compensation Cancel G80 - Canned Cycle Cancel	
N30 T0100 M41 ;	T0100 - Tool number #1 to be loaded into the spindle <u>with no offset call</u>. M41 – Select low gear	
N40 G96 S450 M03 ;	G96 - Constant surface speed (spindle will turn at Snnn surface feet per minute regardless of diameter of workpiece) S450 - Cutting speed selection of 450 ft/min. M03 - Starts the spindle in a clockwise direction	
N50 G00 G41 X6.25 Z0.3 T0101 M08 ;	G00 – Rapid feed engagement. G41 – Tool nose radius compensation to the left of the programmed tool path. X6.25 – Tool will rapid to position of 3.125 units from center line of part. Z0.3 – Tool will rapid to position 0.02 units from finished face of part (finished face of part is usually set to Z0). T0101 – Confirms tool #1 and assigns offset #1 M08 – Start coolant pump	

5 EXAMPLE OF PROGRAM **ENDING LINES**

<table>
<tr><td>N200 G00 U-0.05 W-0.05 ;</td><td colspan="2">G00 - Rapid Traverse
U-0.05 – Rapids tool 0.05 incrementally above last X position
W-0.05 – Rapids tool 0.05 incrementally away from last Z position</td></tr>
<tr><td>N210 M05 ;</td><td colspan="2">M05 – Turn off spindle</td></tr>
<tr><td>N220 G28 U0. ;</td><td>G28 - Machine Zero Return
U0 - X axis in the up direction to machine zero</td><td rowspan="2">Send to machine zero Z-axis first to avoid any crash.</td></tr>
<tr><td>N230 G28 W0. ;</td><td>G28 - Machine Zero Return
W0 - Z axis to machine zero</td></tr>
<tr><td>N240 M30 ;</td><td colspan="2">M30 – End of Program and Reset</td></tr>
</table>

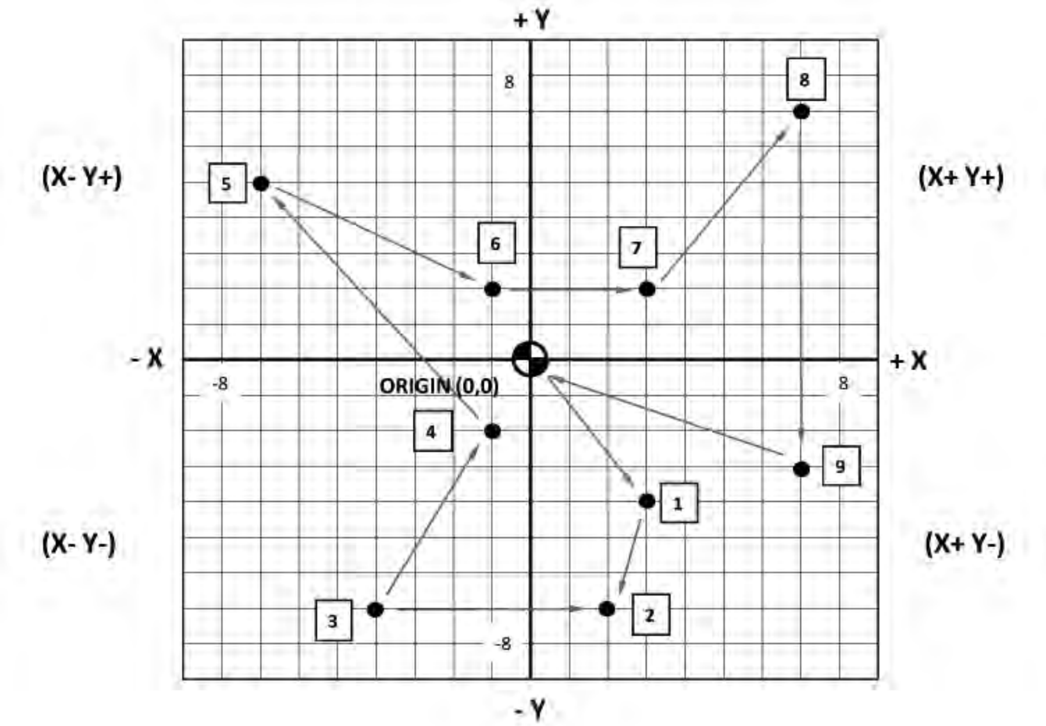


ABSOLUTE PROGRAMMING
All axis motions are based on a fixed zero reference point, known as ABSOLUTE ZERO (part zero).
Each coordinate is in relation to this absolute zero using Cartesian Co-ordinates.

INCREMENTAL PROGRAMMING
All axis motions are based on the distance to the next location.
Each coordinate is based on how far the cutter is to move from start to finish.
For an incremental move in X axis, we use U and for an incremental move in the Z axis we use W. G91 is not used.

STARTING AT THE POINT O (ORIGIN), DESCRIBE THE PATH FROM O THROUGH ALL 9 POINTS AND BACK TO THE POINT O USING ABSOLUTE & INCREMENTAL POSITIONING

ABSOLUTE	X	Z	INCREMENTAL	U	W
O (Origin)			O → 1		
1			1 → 2		
2			2 → 3		
3			3 → 4		
4			4 → 5		
5			5 → 6		
6			6 → 7		
7			7 → 8		
8			8 → 9		
9			9 → O		

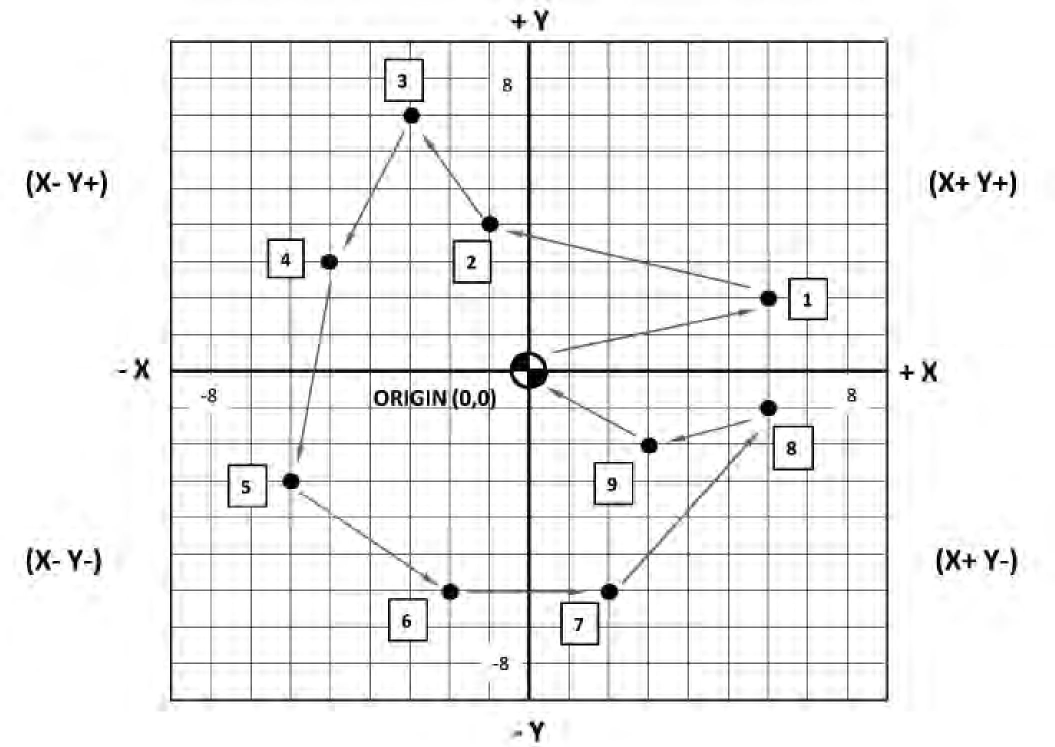


ABSOLUTE PROGRAMMING
All axis motions are based on a fixed zero reference point, known as ABSOLUTE ZERO (part zero).
Each coordinate is in relation to this absolute zero using Cartesian Co-ordinates.

INCREMENTAL PROGRAMMING
All axis motions are based on the distance to the next location.
Each coordinate is based on how far the cutter is to move from start to finish.
For an incremental move in X axis, we use U and for an incremental move in the Z axis, we use W. G91 is not used.

✐***STARTING AT THE POINT O (ORIGIN), DESCRIBE THE PATH FROM O THROUGH ALL 9 POINTS AND BACK TO THE POINT O USING ABSOLUTE & INCREMENTAL POSITIONING***

ABSOLUTE	X	Z	INCREMENTAL	U	W
O (Origin)			O → 1		
1			1 → 2		
2			2 → 3		
3			3 → 4		
4			4 → 5		
5			5 → 6		
6			6 → 7		
7			7 → 8		
8			8 → 9		
9			9 → O		



ABSOLUTE PROGRAMMING

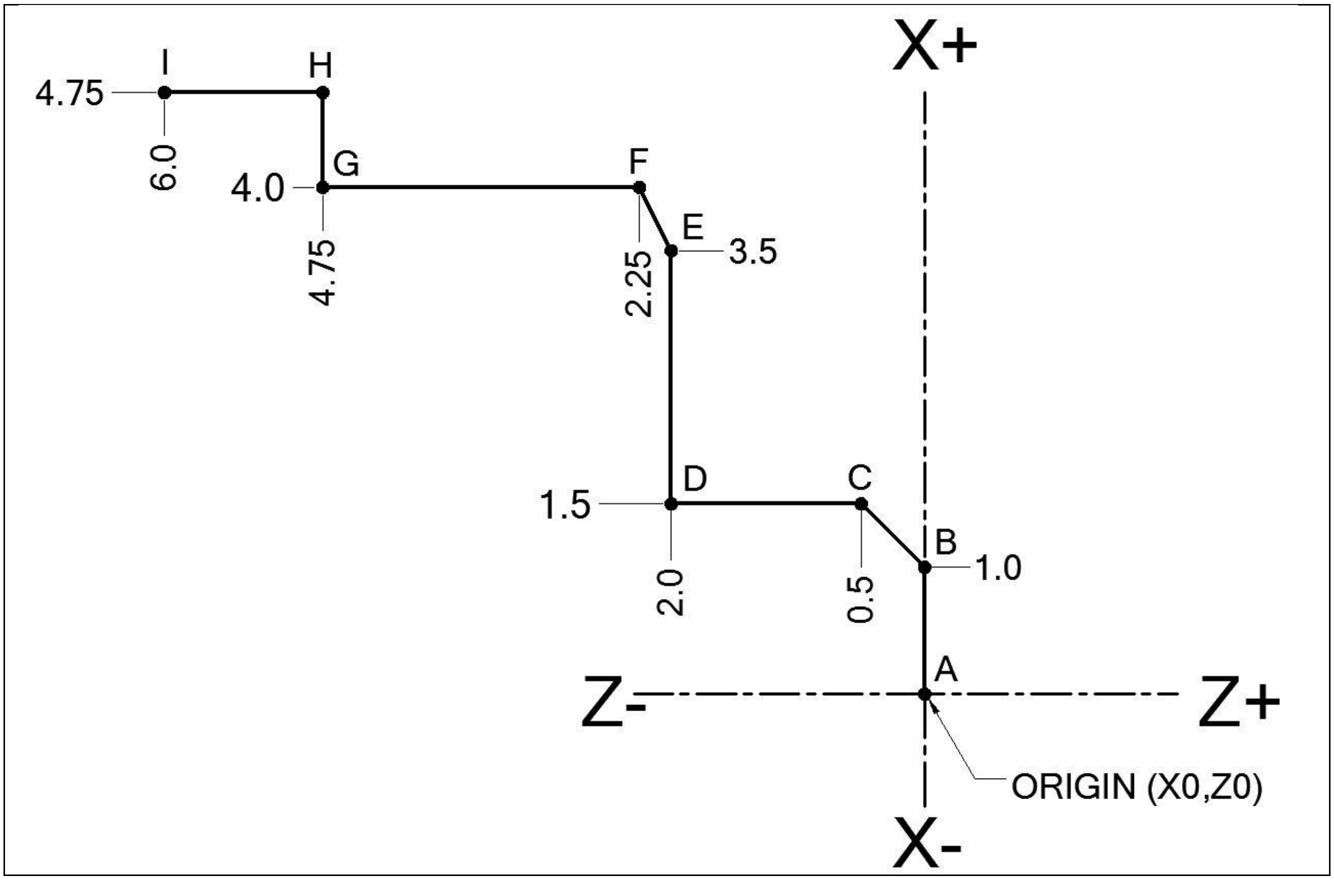
All axis motions are based on a fixed zero reference point, known as ABSOLUTE ZERO (part zero).
Each coordinate is in relation to this absolute zero using Cartesian Co-ordinates.

INCREMENTAL PROGRAMMING

All axis motions are based on the distance to the next location.
Each coordinate is based on how far the cutter is to move from start to finish.
For an incremental move in X axis, we use U and for an incremental move in the Z axis, we use W. G91 is not used.

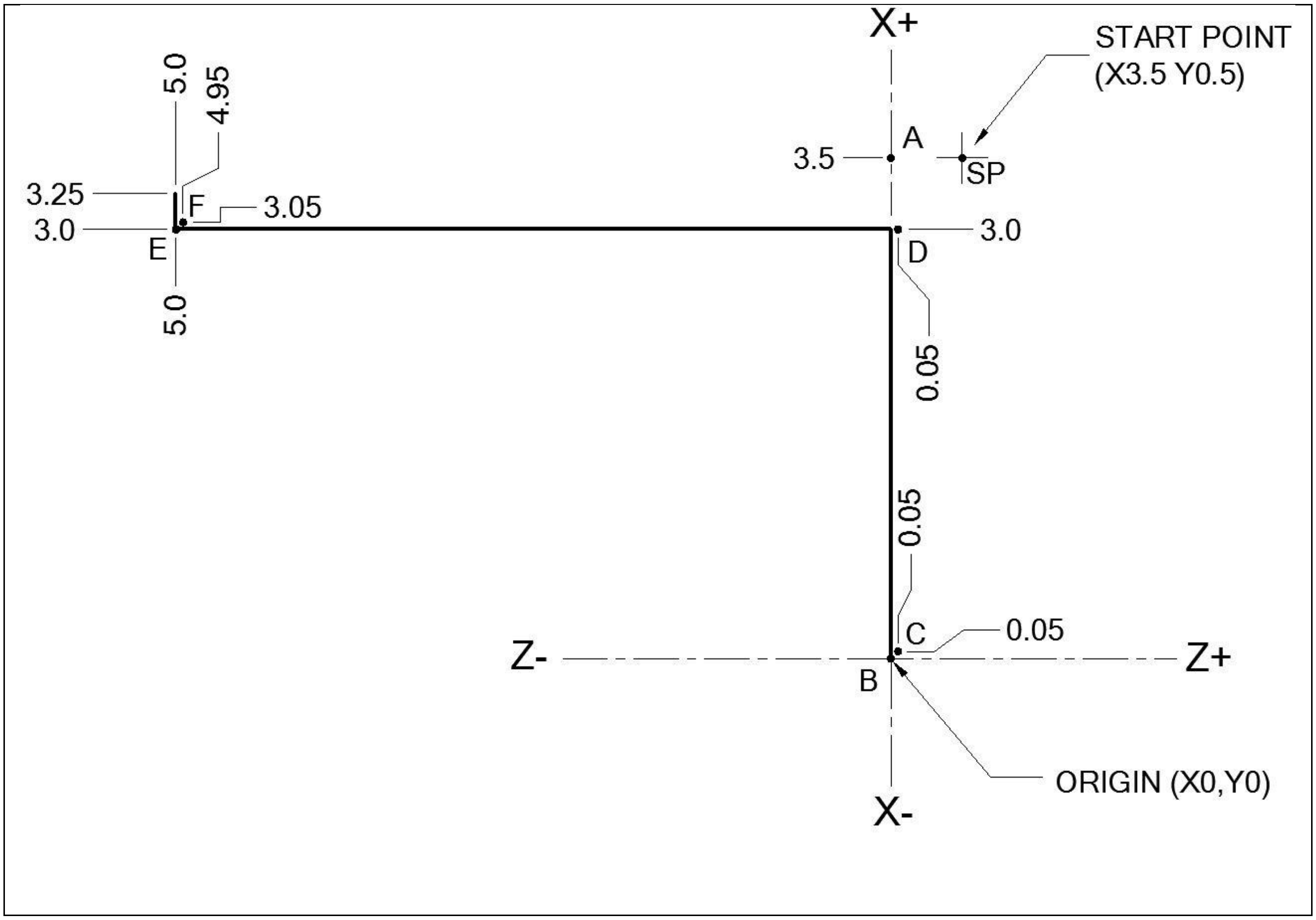
STARTING AT THE POINT O (ORIGIN), DESCRIBE THE PATH FROM O THROUGH ALL 9 POINTS AND BACK TO THE POINT O USING ABSOLUTE & INCREMENTAL POSITIONING

ABSOLUTE	X	Z	INCREMENTAL	U	W
O (Origin)			O → 1		
1			1 → 2		
2			2 → 3		
3			3 → 4		
4			4 → 5		
5			5 → 6		
6			6 → 7		
7			7 → 8		
8			8 → 9		
9			9 → O		



STARTING AT THE POINT A (ORIGIN), DESCRIBE THE TOOLPATH THROUGH ALL THE POINTS USING ABSOLUTE & INCREMENTAL POSITIONING

ABSOLUTE	X	Z	INCREMENTAL	U	W
A			**A → B**		
B			**B → C**		
C			**C → D**		
D			**D → E**		
E			**E → F**		
F			**F → G**		
G			**G → H**		
H			**H → I**		
I					



BEGIN AT START POINT SP (X2.5, Z0.5), DESCRIBE THE PATH FROM SP THROUGH POINTS A-F AND BACK TO POINT SP, USING ABSOLUTE & INCREMENTAL POSITIONING

ABSOLUTE	X	Z	INCREMENTAL	U	W
SP (START POINT)			SP → 1		
A			A → B		
B			B → C		
C			C → D		
D			D → E		
E			E → F		
F			F → SP		

8 RAPID **G00** AND LINEAR **G01** INTERPOLATION

G00 RAPID TRAVERSE

This code is used for rapid motion of the cutter in air to traverse from one position to another as fast as possible. This code will work for both axis motions at once.

This G00 code is modal and causes all the following blocks to be in rapid (up to 1000 in./min.) motion until another Group 01 code is specified.

Generally, rapid motions "will not" be in a straight line. All the axes specified are moved at the maximum speed and will not necessarily complete each axis move at the same time. It activates each axis drive motor independently of each other and, as a result, the axis with the shortest move will reach its destination first. So ***you need to be careful of any obstructions to avoid with this type of rapid move.***

- G00 is used when you are positioning the cutter in 'fresh air'.
- Retracting from a hole you have drilled.
- Rapid traverse is not used when cutting the part.
- Used incorrectly, rapid traverse will break a cutter very easily and possibly remove the part from the chuck.

G01 LINEAR INTERPOLATION

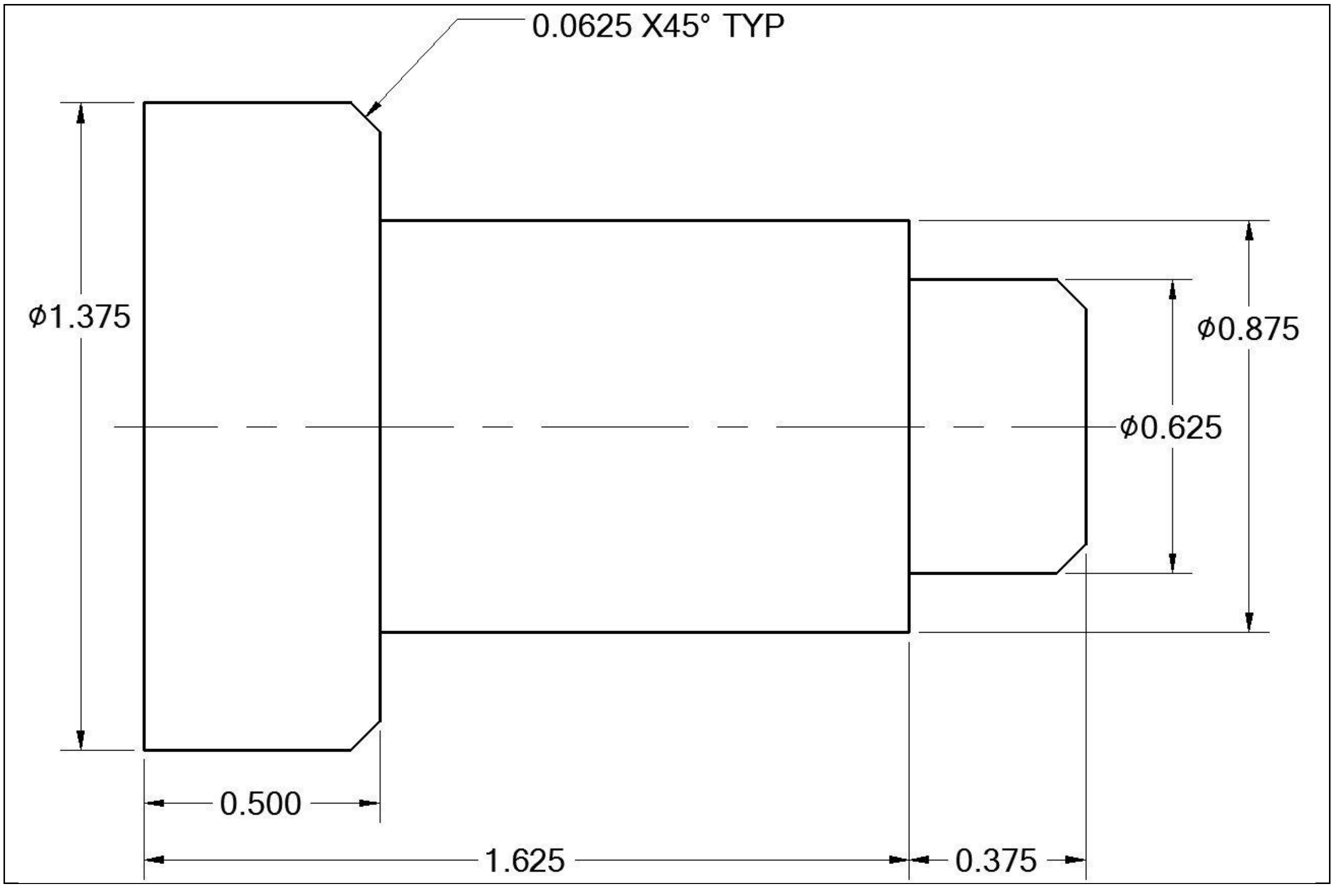
This G code provides for straight line (linear) motion with programmed feedrate for all axis motions from point to point. Motion can occur with both axes at once.

All axes specified will start at the same time and proceed to their destination and arrive simultaneously at the specified feedrate.

To program a feedrate, the F command is used. The F command is modal and may be specified in a previous block.

G01 is used for

- Drilling a hole
- Turning a diameter
- Machining a profile I.D and O.D.
- Grooving I.D and O.D.

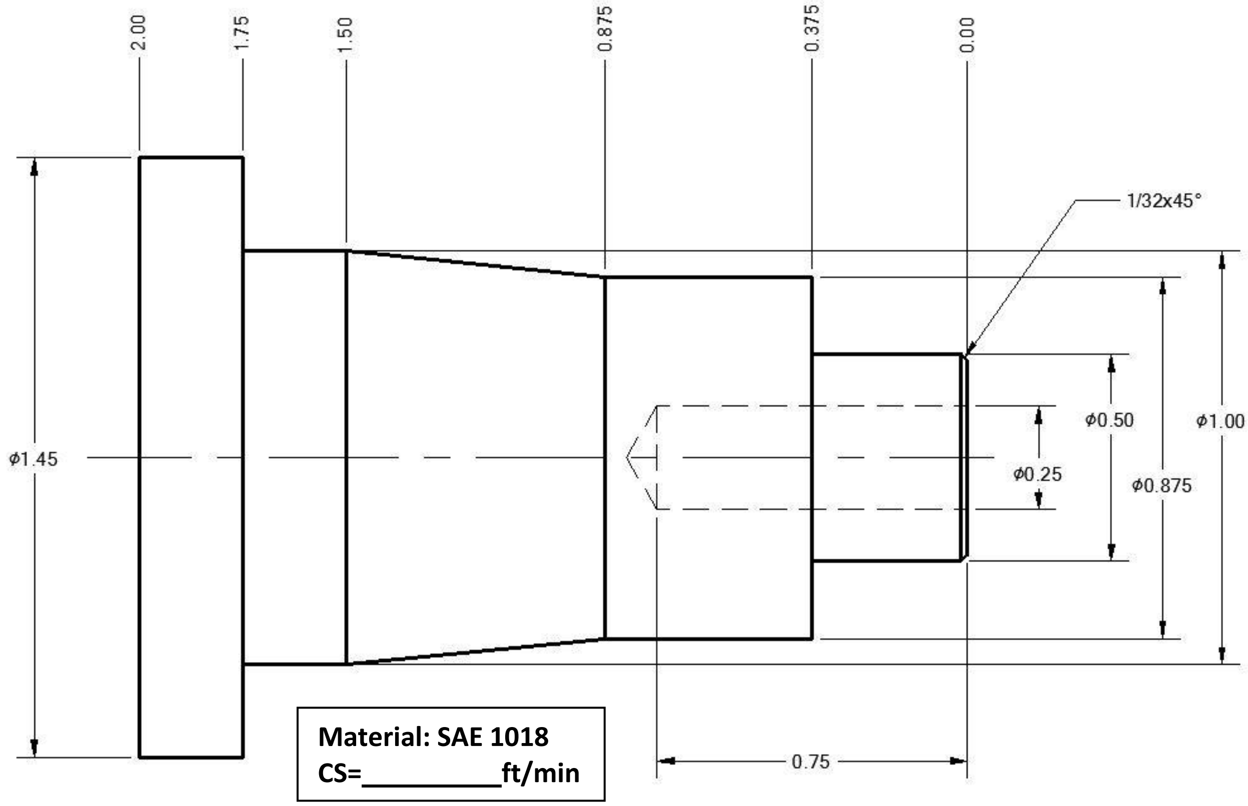


- ***LINEAR TURNING EXERCISE #8-1 (CREATE A ROUGHING TOOLPATH)***
  - Program a rough turning toolpath
  - X0 is centerline of the part, Z0 is the front face (far right) of the part
  - All X positions are diameter, all Z positions past the front face are Z-
  - To create a roughing toolpath, the front face is skimmed flat (faced), only to leave a small amount of material for a finish pass.
  - The tool is then retracted in X&Z (U&W) a small amount (0.05) then rapids to the X diameter of the first Z feed across the rough OD of the part.
  - The tool is once again retracted in X&Z (U&W) then rapids back to a safe position in Z, then brought to the next position in X diameter for the next feed across and so on.
  - For this project, take a maximum cut of 0.25 off the diameter, leave 0.04 on the diameter and 0.005 on all faces for a finish pass to be programmed in the next exercise.
  - The 0.0625 x 45° chamfers will be added in the finishing toolpath

```
%
O00081 ;                    (PROGRAM NAME, ROUGH TURNING EXERCISE)
N1 G20   ;                  (VERIFY INCH MODE)
N3 G40 G80 G99              (SAFETY LINE WITH FEED AS INCH\REV.)
N5 T0100 M41                (TOOL CALL AND GEAR RANGE)
N7 G50 S4000 ;              (SET MAX. SPEED AT 4000 RPM, CALL TOOL #2 NO OFFSETS)
N9 G97 S500 M03 ;           (START SPINDLE 500 RPM CLOCKWISE ROTATION)
N11 G00 G41 X___Z___ T0101 M08  ;(TOOL NOSE RADIUS OFFSET, SAFE POSITION, LARGER DIAMETER THAN
             ROUGH MATERIAL, SAFE DISTANCE FROM FRONT FACE OF ROUGH PART, COOLANT ON)
N13 G96 S300 ;              (CONSTANT SURFACE SPEED ENGAGED AT 300 SFM)
N15 Z_____                  (RAPID TO 0.01 FROM FRONT FACE OF THE PART, NON-CUTTING MOVE)  ;
N17 G_____ X0 F15.0  ;      (FEED TOOL TO FACE TO CENTERLINE OF PART, FEEDRATE=15.0" / MIN. )
N19 G____ Z.1  ;            (RAPID RETRACT)  ;
N21 X____  ;                (MOVE TO THE FIRST Z AXIS CUTTING POSITION)  ;
N23 G01 Z-____              (FEED TO FULL LENGTH OF PART, 15.IPM FEEDRATE REMAINS IN EFFECT)  ;
N25 U___ W-___              (RETRACT OFF PART IN FEED MODE)  ;
N27 G00 Z_____              (RAPID TO SAFE POSITION IN FRONT OF PART)  ;
N29 X____                   (RAPID TO NEXT CUTTING DEPTH)  ;
N31 G____ Z-1.625           (FEED TO FULL LENGTH OF PART, 15.IPM FEEDRATE REMAINS IN EFFECT)  ;
N33 U.1 W-.1                (RETRACT OFF PART IN FEED MODE)  ;
N35 G____ Z_____            (RAPID TO SAFE POSITION IN FRONT OF PART)  ;
N37 X____                   (RAPID TO NEXT CUTTING DEPTH)  ;
N39 G01 Z-____              (FEED TO SECOND STEP LENGTH, 15.IPM FEEDRATE REMAINS IN EFFECT)
N41 U___ W-___              (RETRACT OFF PART IN FEED MODE)  ;
N43 G40 G___ X____ Y____ T0100      (CANCEL TOOL NOSE RADIUS OFFSET, RAPID TO ORIGINAL START
                                    POSITION)  ;
N45 G28 Z0. ;
N47 G28 X0. ;
N49 M30                     (PROGRAM END)  ;
%
```

- ***PROFILE TURNING EXERCISE #8-2 (CONTOUR THE FINISH PROFILE)***
    - Tool will be a carbide insert with a 0.032" tool nose radius (Tool # 2)
    - Start contour from X0 Z0.

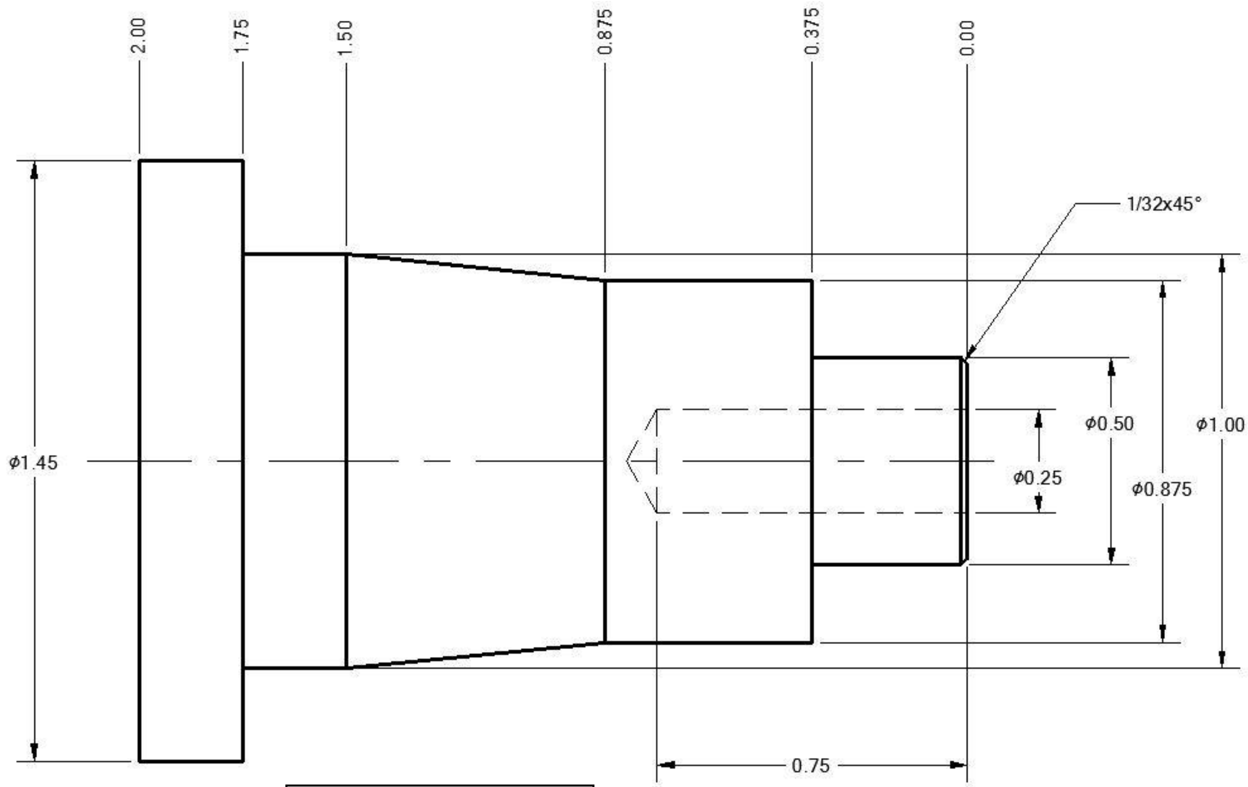
%
O00082 ;
N10 G20 ;
N20 G40 G80 G99 (MACHINE DEFAULT SETTING) ;
N30



Material: SAE 1018
CS=__________ft/min

Tool #? () – ROUGH AND FINISH TOOLPATH	
Spindle Speed RPM = 4xCS/D=	Feed = in/min
Depth of Cut =	Start from the top of the slot

%
O00083;



Material: Al. 6061
CS=__________ft/min

Tool #8 (#4 CENTRE DRILL) – FACE MUST BE CENTRE DRILLED BEFORE DRILLING
Tool #11 (Ø 0.25 DRILL) – DRILL HOLE 0.75 DEEP

Spindle Speed RPM = 4xCS/D=	Feed = in/min

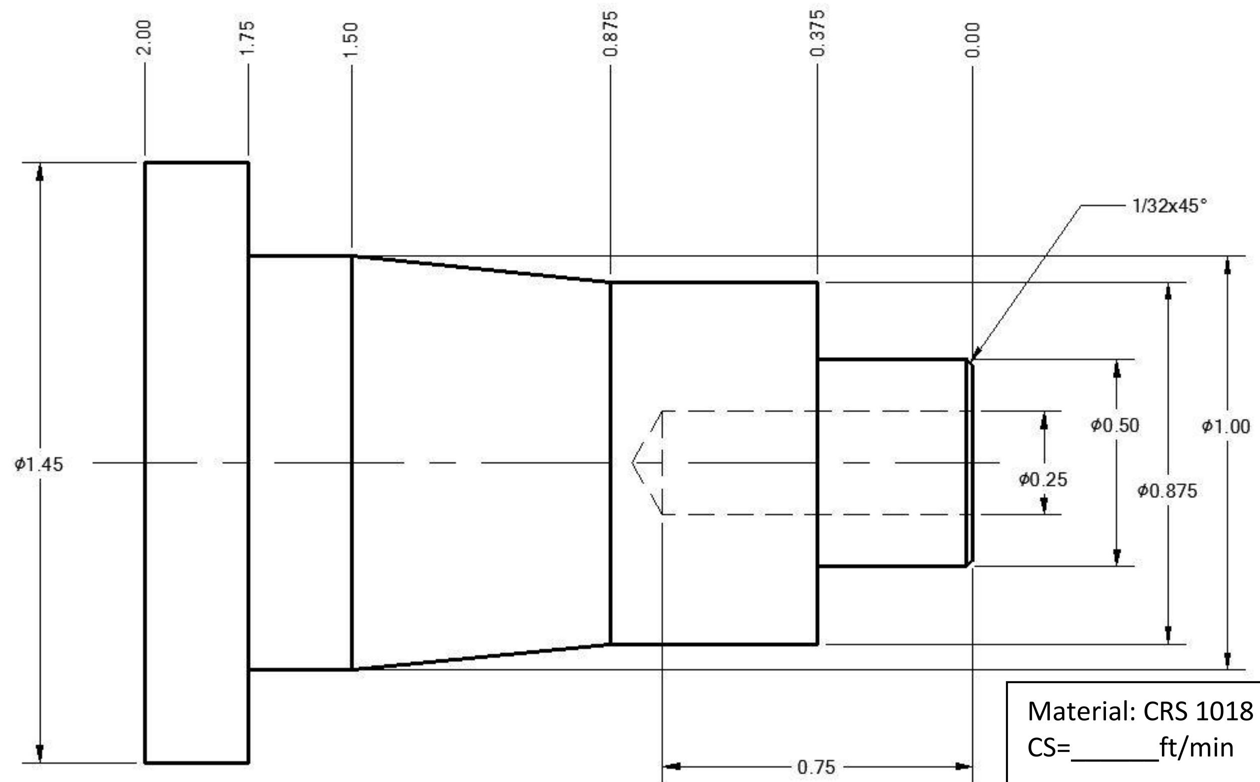
```
%
O00084 ;                    (PROGRAM NAME, CENTRE DRILL AND DRILLING EXERCISE)
N1 G20   ;                  (VERIFY INCH MODE)
N3 G40 G80 G99 ;            (SAFETY LINE WITH FEED AS INCH\REV.)
N5 T0800 M41 ;              (TOOL CALL NO OFFSETS AND GEAR RANGE)
N7 G50 S____ ;              (SET MAX. SPEED)
N9 G97 S____ M03 ;          (START SPINDLE, CLOCKWISE ROTATION)
N11 G00 X___Z___ T0808 M08 ;    (X CENTRE OF PART, Z SAFE DISTANCE FROM FRONT FACE OF PART, COOLANT ON)
N13 Z_____                  (RAPID TO 0.05 FROM FRONT FACE OF THE PART, NON-CUTTING MOVE)
N15 G____ Z- 0.269 F15.0 ;(FEED TOOL TO C'DRILL DEPTH, FEEDRATE=15.0" / MIN.)
N17 G____ Z.1 ;             (RAPID RETRACT)
N19 G___ X____ Y____ T0800;     (RAPID TO ORIGINAL START POSITION)
N21 G28 Z0. ;                   (SEND TOOL TO HOME POSITION IN Z AXIS)
N23 G28 X0. ;                   (SEND TOOL TO HOME POSITION IN X AXIS)
N25 M01 ;                       (OPTIONAL STOP)
N27 T1100 M41  ;            (TOOL CALL NO OFFSETS AND GEAR RANGE)
N29 G50 S____ ;             (SET MAX. SPEED)
N31 G97 S____ M03 ;         (START SPINDLE,CLOCKWISE ROTATION)
N33 G00 X___ Z___ T1111 M08 ;   ( X CENTRE OF PART, Z SAFE DISTANCE FROM FRONT FACE OF PART, COOLANT ON)
N35 Z______                     (RAPID TO 0.05 FROM FRONT FACE OF THE PART, NON-CUTTING MOVE)
N37 G______ Z-_____ F15.0 ;     (FEED TOOL TO DRILL DEPTH, FEEDRATE=15.0" / MIN. )
N39 G______ Z.1 ;               (RAPID RETRACT)
N41 G40 G___ X_____ Y_____ T0800 ;(RAPID TO ORIGINAL START POSITION)
N43 G28 Z0. ;                   (SEND TOOL TO HOME POSITION IN Z AXIS)
N45 G28 X0. ;                   (SEND TOOL TO HOME POSITION IN X AXIS)
N47 M30 ;                       (PROGRAM END)
%
```

9 TURNING CANNED CYCLES **G71, G72**

G71-G72 ROUGH TURNING CYCLE
A canned cycle, which permits multiple function programming in one code, is very helpful to the programmer for ease of programming and more compact programs. The G71 cycle allows for rough turning in the Z- direction (towards the chuck) The G72 cycle allows for rough facing in the X- direction (towards the centerline of the part)
G71 CANNED CYCLE
Format: G71 U___ R___ **G71 P___ Q___ U___ W___ F___S___** First G71 block **U** Depth of roughing cut **R** Amount of retract after each cut Second G71 block **P** First block number of finish contour **Q** Last block number of finish contour **U** Amount of stock left for finish operation (diameter) X axis **W** Amount of stock left on all faces for finish operation Z axis **F** Feed rate in inches or mm /rev. **S** Spindle speed in ft or m /min.
G72 CANNED CYCLE
Format: G72 W___ R___ **G72 P___ Q___ U___ W___ F___S___** First G71 block **U** Depth of roughing cut **R** Amount of retract after each cut Second G71 block **P** First block number of finish contour **Q** Last block number of finish contour **U** Amount of stock left for finish operation (diameter) X axis **W** Amount of stock left on all faces for finish operation Z axis **F** Feed rate in inches or mm /rev. **S** Spindle speed in ft or m /min.

ROUGH TURNING CANNED CYCLE EXERCISE

EXERCISE # 9-1



Material: CRS 1018
CS=______ft/min

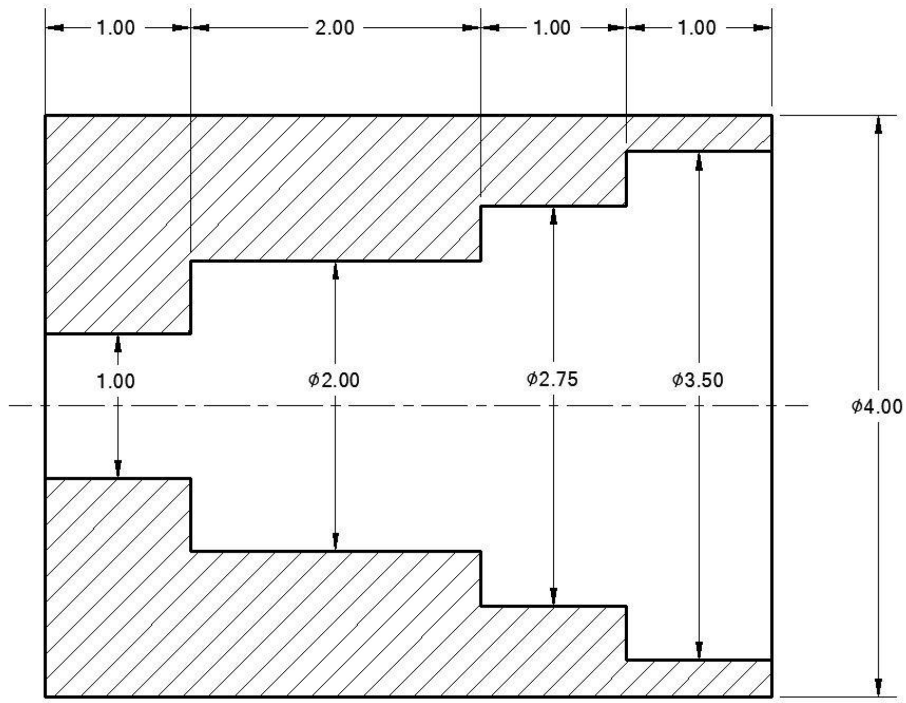
1. Create A Rough Turning Toolpath Using The Above Drawing
2. Use G71 (Roughing Towards The Chuck In Z Axis)
3. Leave 0.02"/Side For Finishing Later

1. Spindle Speed RPM = 4xCS/D=_________	1. Feed =________in/min
1. Depth of Cut =_______	

%
O00091;

ROUGH BORING CANNED CYCLE EXERCISE

EXERCISE # 9-2



Material: CRS 1018
CS=______ft/min

1. Create A Rough Boring Toolpath Using The Above Drawing
2. Use G71 (Roughing Towards The Chuck In Z Axis)
3. Leave 0.02"/Side For Finishing Later

1. Spindle Speed RPM = 4xCS/D=_________	1. Feed =_________in/min
1. Depth of Cut =________	

%
O00092;

ROUGH BORING CANNED CYCLE EXERCISE (CONT'D)

EXERCISE # 9-2

G70 FINISH TURNING/BORING CANNED CYCLE

Format : G70 P__ Q__ F__ S__

P= First block number of the finish contour
Q= Last block number of the finish contour
F= Cutting feedrate for the finishing (overrides the feed in roughing contour)
S= spindle speed (overrides speed in roughing contour)

This canned cycle is used after the roughing canned cycle is finished. It does not have to be run directly after the roughing cycle but can be run in the same main program. The start and finish blocks of the original definition of the profile that was used in the rough cycle are used to define the contour of the finish cycle. It is recommended that the same start point is used for both rough and finish cycles to ensure safe toolpaths of both operations.

TYPICAL O.D. FINISH CANNED CYCLE

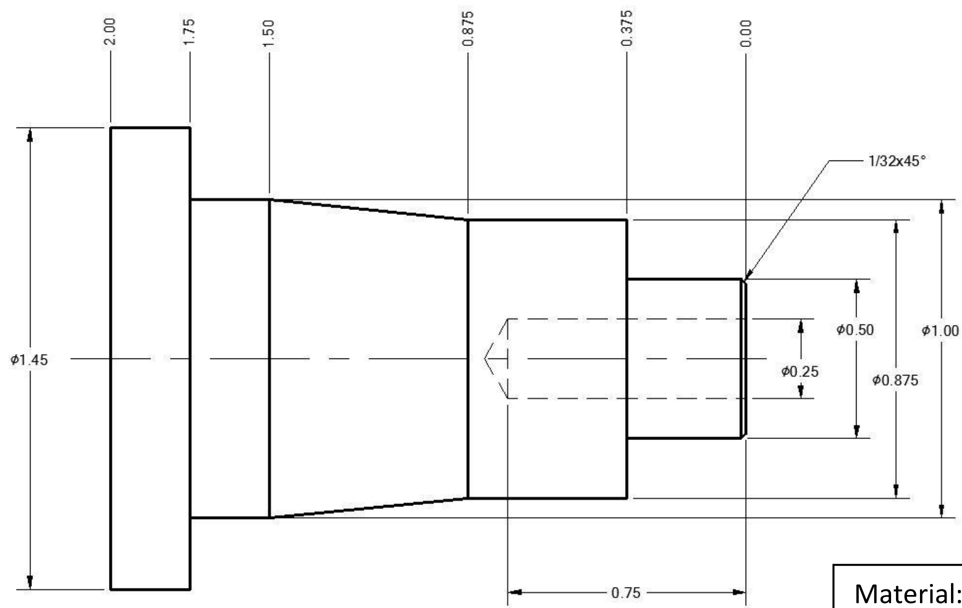
```
N37 T0500 M42                 (OD FINISH TOOL & GEAR SEL.)
N38 G96 S500 M03              (CSS. SPEED)
N39 G42 X__ Z__ T0505 M08     (CUTTER COMPENSATION & START POS.)
N40 G70 P11 Q19 F12.0         (CALL LINES FOR FINISH COORDS)
N41 G00 G40 X__ Z__ T0100
N42 M01
```

TYPICAL I.D. FINISH CANNED CYCLE

```
N43 T0700     M42             (ID FIN TOOL & GEAR SEL.)
N44 G96 S475 M03              (CSS. SPD)
N45 G00 G41 X__ Z__ T0707 M08 (CUTTER COMPENSATION & START POS.)
N46 G70 P27 Q34 F12.0         (CALL LINES FOR FIN. COORDS)
N47 G00 G40 X__ Z__ T0700
N48 M01
```

FINISHING CANNED CYCLE **G70**

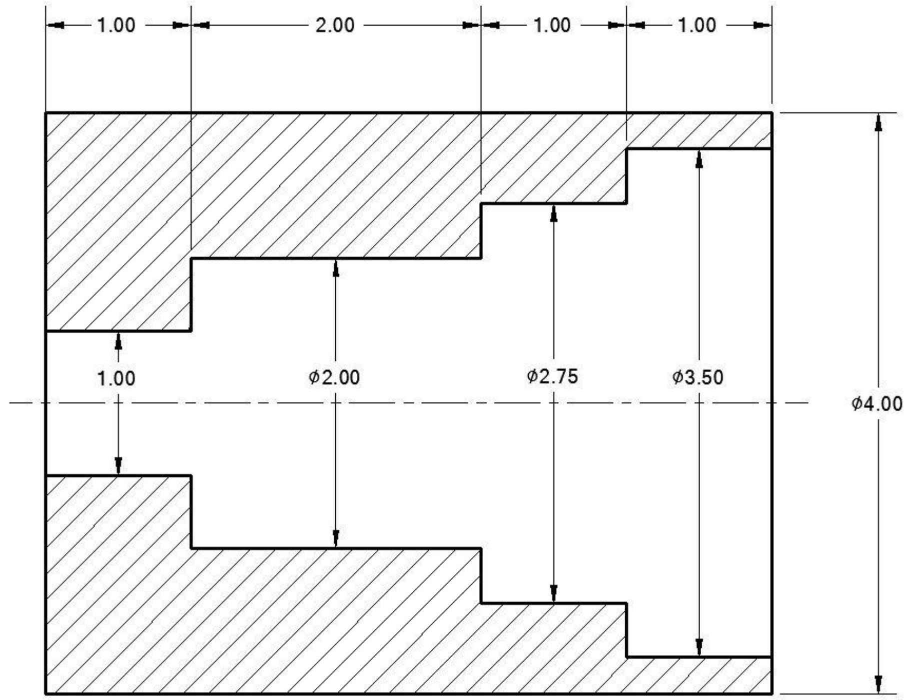
EXERCISE # 10-1



Material: CRS 1018
CS=______ft/min

1. Create A Finish Turning Toolpath Using The Rough Toolpath From Exercise # 9-1
2. Use G70 (Finish Canned Cycle)
3. Use A Finish Feedrate Of 0.01”/Rev. And A Max. Speed Of 2000 Rpm

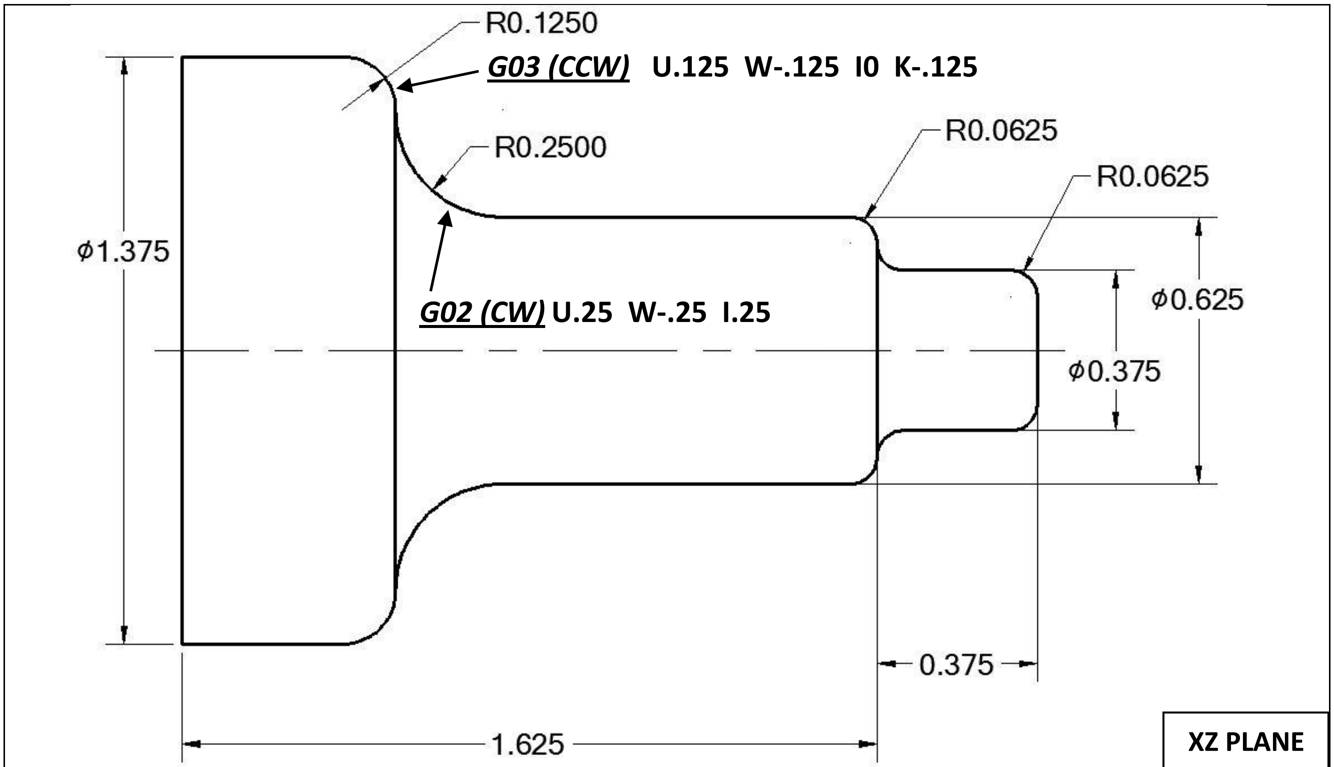
%
O00101;



1. Create A Finish Boring Toolpath Using The Rough Toolpath From Exercise # 9-2
2. Use G70 (Finish Canned Cycle)
3. Use A Finish Feedrate Of 0.01"/Rev. And A Max. Speed Of 2000 Rpm

%
O00102;

11 CIRCULAR INTERPOLATION **G02** & **G03**



G02 & G03 Circular Interpolation Format			
G02 Clockwise Direction	U.25 W-.25 Arc End Point (optional incremental)	I 0.25 K0 Incremental Distance from the tool start point to the center of arc	F15. Feed rate 15 in/min
G03 Counter Clockwise	U.125 W-.125 Arc End Point (optional incremental)	I 0. J-.125 Incremental Distance from the tool start point to the center of arc	F15. Feed rate 15 in/min

When the machine is required to move in a straight line under a controlled federate, linear interpolation is used (G01). When it is necessary to travel in the circular motion in any plane (XY, YZ, XZ) circular interpolation is used (G02, G03).

The velocity at which the tool is moving is controlled by the feed rate (F) command.

All circular interpolation moves are defined and machined by programming in three pieces of information into the control.

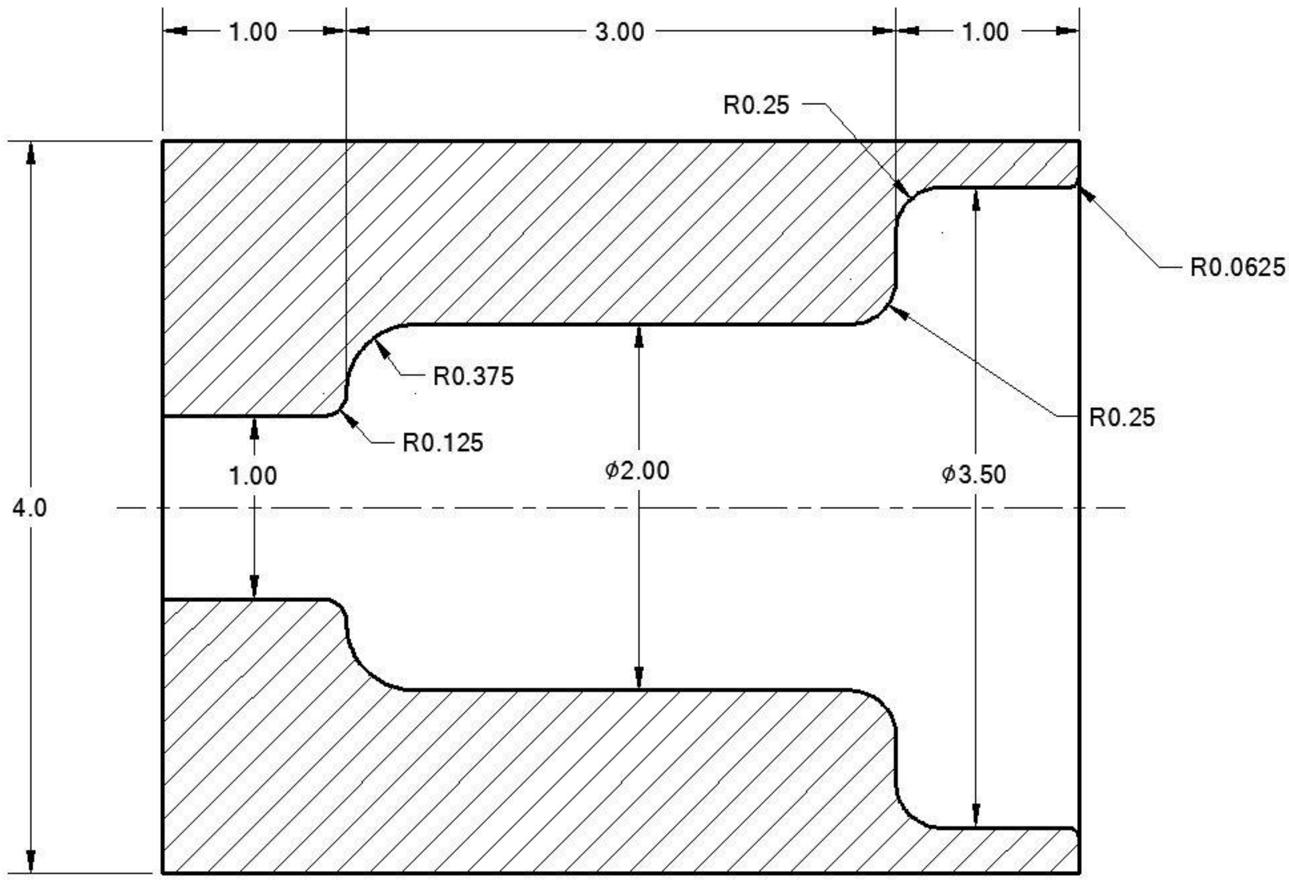
1. ***DIRECTION OF TRAVEL: CLOCKWISE G02, COUNTER CLOCKWISE G03***
2. ***ARC END POINT: X AXIS, Z AXIS***
3. ***ARC CENTER: INCREMENTAL DISTANCE FROM START POINT TO ARC CENTER (I, J, K)***

CIRCULAR INTERPOLATION EXERCISE *TURNING*

EXERCISE # 11-1

- ***Contour the profile as shown on page 31.***
- ***Material: Aluminum 6061 & Use appropriate Spindle Speed and Feedrate.***

%
O00111;



I = *INCREMENTAL DISTANCE* ALONG THE X-AXIS
FROM THE *TOOL START POINT* TO THE *CENTER OF ARC* (ARC PIVOT POINT)

J = *INCREMENTAL DISTANCE* ALONG THE Y-AXIS
FROM THE *TOOL START POINT* TO THE *CENTER OF ARC* (ARC PIVOT POINT)

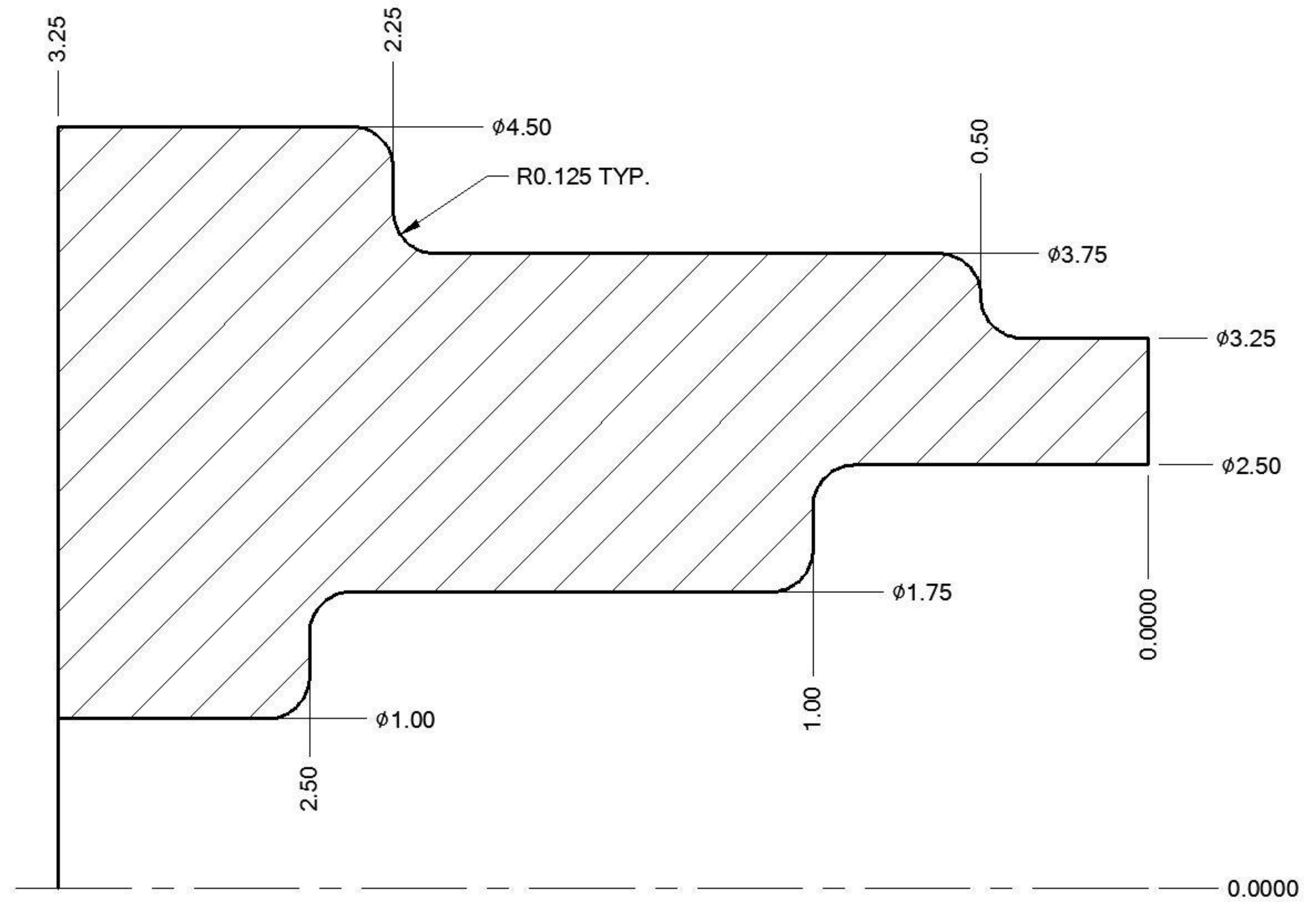
K = *INCREMENTAL DISTANCE* ALONG THE Z-AXIS
FROM THE *TOOL START POINT* TO THE *CENTER OF ARC* (ARC PIVOT POINT)

CIRCULAR INTERPOLATION EXERCISE *BORING*

EXERCISE # 11-2

- ***Contour the profile as shown on previous page.***
- ***Material: Aluminum 6061 & Use appropriate Spindle Speed and Feedrate.***
- ***I and K are used for Circular Interpolation on a lathe, I and J are used on a mill***

%
O00112;



G02/G03

I = *INCREMENTAL DISTANCE* ALONG THE X-AXIS
FROM THE *TOOL START POINT* TO THE *CENTER OF ARC* (ARC PIVOT POINT)

K = *INCREMENTAL DISTANCE* ALONG THE Z-AXIS
FROM THE *TOOL START POINT* TO THE *CENTER OF ARC* (ARC PIVOT POINT)

THIS DRAWING IS A PROFILE OF ONLY HALF OF THE PART. ALL X VALUES ARE IN DIAMETER (Ø), THE LINE MARKED 0.000 AT THE BOTTOM OF THE PART REPRESENTS THE CENTERLINE OF THE ROUND MATERIAL (X0). AND THE LINE MARKED 0.000 AT THE RIGHT REPRESENTS THE FRONT OF THE PART (Z0). Z0 IS USUALLY FOUND AT THE FRONT OF THE PART SO THAT ANY MOVE INTO THE MATERIAL IN Z WILL BE A NEGATIVE VALUE.

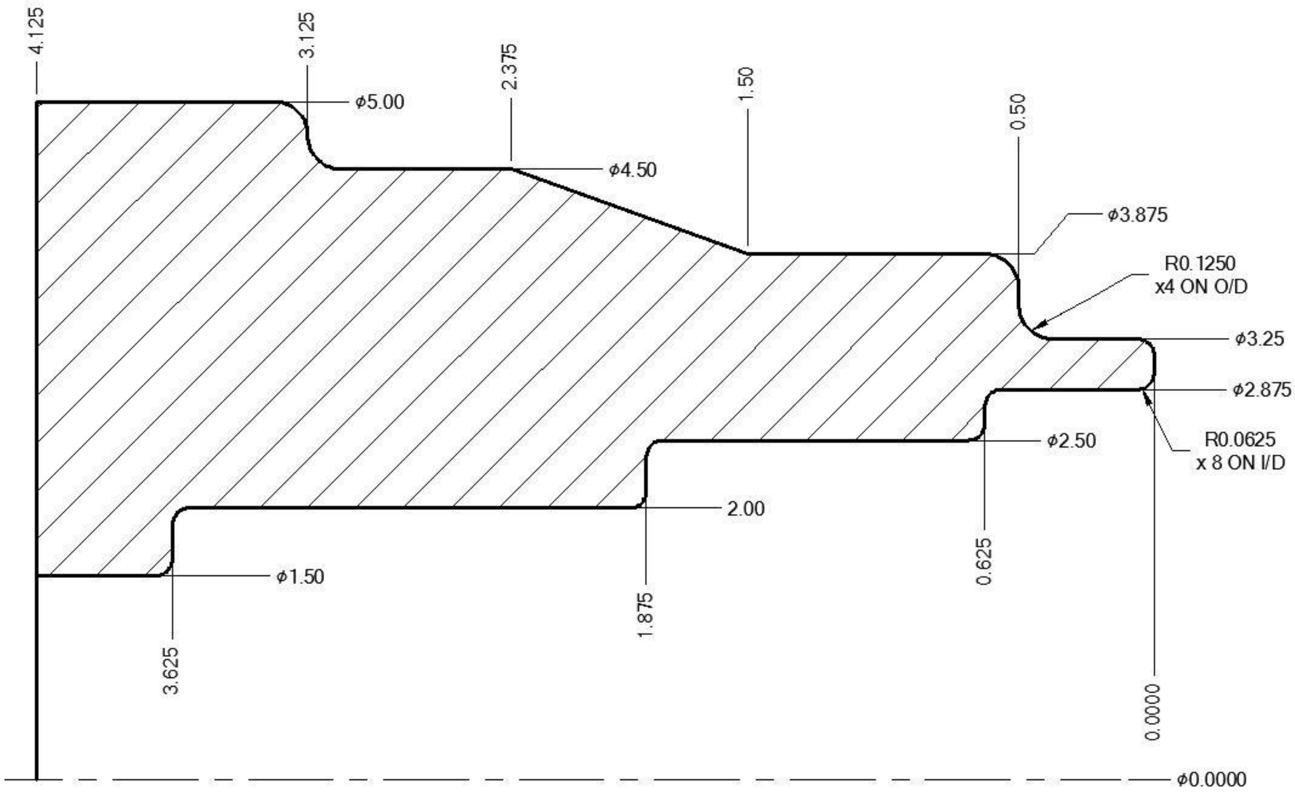
THE PROGRAM WILL START AT THE MACHINE HOME POSITION, MOST MACHINES WILL ALSO DO A TOOL CHANGE AT THE MACHINE HOME. WHEN PROGRAMMING THE COUNTOUR OF A PART, A "SAFE POSITION" TO START THE PROGRAM SHOULD BE DETERMAINED. IN THIS CASE A POSITION OF X4.75, Z0.25 WOULD BE SUITABLE.

G02/G03 EXERCISE - TURNING AND *BORING*

EXERCISE # 11-3

- ***Contour the profile as shown on the previous page.***
- ***Material: Aluminum 6061 & Use appropriate Spindle Speed and Feedrate.***
- ***I and K are used for Circular Interpolation on a lathe, I and J are used on a mill.***
- ***Start profile from X4.75, Z0.25 then rapid to X3.25, Z0.1 and begin with OD toolpath.***

%
O00113;



G02/G03

I = *INCREMENTAL DISTANCE* ALONG THE X-AXIS
FROM THE *TOOL START POINT* TO THE *CENTER OF ARC* (ARC PIVOT POINT)

K = *INCREMENTAL DISTANCE* ALONG THE Z-AXIS
FROM THE *TOOL START POINT* TO THE *CENTER OF ARC* (ARC PIVOT POINT)

THIS DRAWING IS A PROFILE OF ONLY HALF OF THE PART. ALL X VALUES ARE IN DIAMETER (Ø), THE LINE MARKED 0.000 AT THE BOTTOM OF THE PART REPRESENTS THE CENTERLINE OF THE ROUND MATERIAL (X0). AND THE LINE MARKED 0.000 AT THE RIGHT REPRESENTS THE FRONT OF THE PART (Z0). Z0 IS USUALLY FOUND AT THE FRONT OF THE PART SO THAT ANY MOVE INTO THE MATERIAL IN Z WILL BE A NEGATIVE VALUE.

THE PROGRAM WILL START AT THE MACHINE HOME POSITION, MOST MACHINES WILL ALSO DO A TOOL CHANGE AT THE MACHINE HOME. WHEN PROGRAMMING THE COUNTOUR OF A PART, A "SAFE POSITION" TO START THE PROGRAM SHOULD BE DETERMAINED. IN THIS CASE A POSITION OF X5.5, Z0.5 WOULD BE SUITABLE.

G02/G03 EXERCISE - TURNING AND *BORING*

EXERCISE # 11-4

- ***Contour the profile as shown on the previous page.***
- ***Material: Aluminum 6061 & Use appropriate Spindle Speed and Feedrate.***
- ***I and K are used for Circular Interpolation on a lathe, I and J are used on a mill.***
- ***Start profile from X5.25, Z0.25 then rapid to a suitable position to create the front OD fillet.***

%
O00114;

12 G40, G41, & G42 TOOL NOSE COMPENSATION

When a program is created it is done so using the insert's command or reference point (see figure 12a below). **Tool Nose Compensation** is used to offset the tool by a distance that will bring the cutting edge of the insert to the proper position in relation to the specific radius of the insert being used. The radius of the tool must be input into the controller and it will calculate the proper offset known as Tool Nose Compensation.

G40 CUTTER COMPENSATION CANCEL

G40 will cancel the G41 or G42 cutter compensation commands that are in effect at the time.

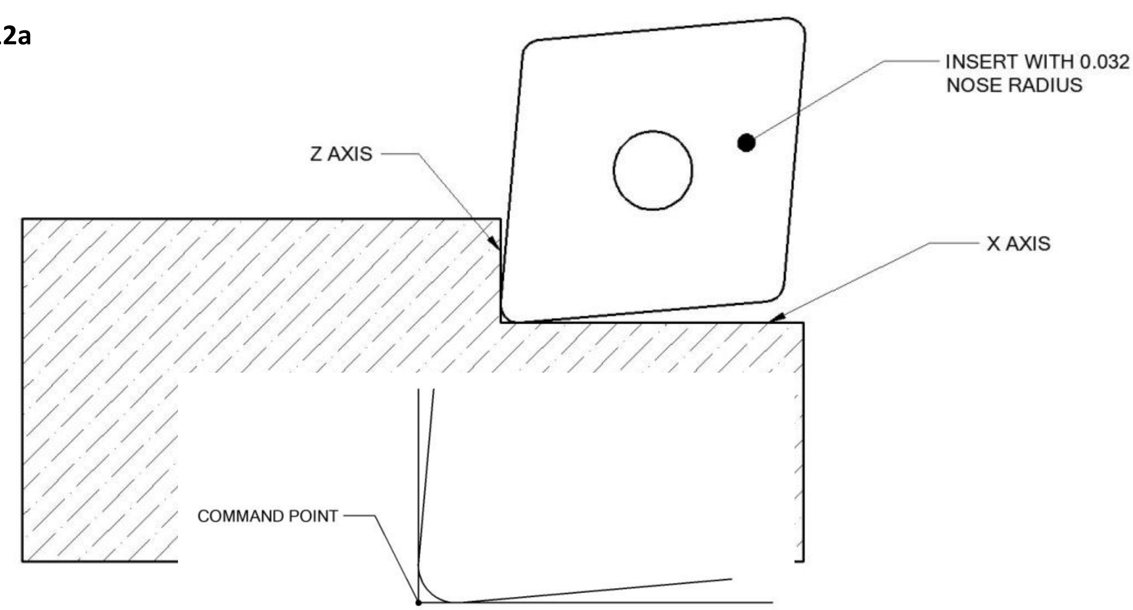
G41 CUTTER COMPENSATION LEFT (BORING)

G41 will select cutter compensation to the **LEFT** of the contouring direction; generally G41 is used for boring. The tool is compensated for the radius of the tool tip. The value of the compensation (tool radius) must be entered in the controller registry during set-up.

G42 CUTTER COMPENSATION RIGHT (TURNING)

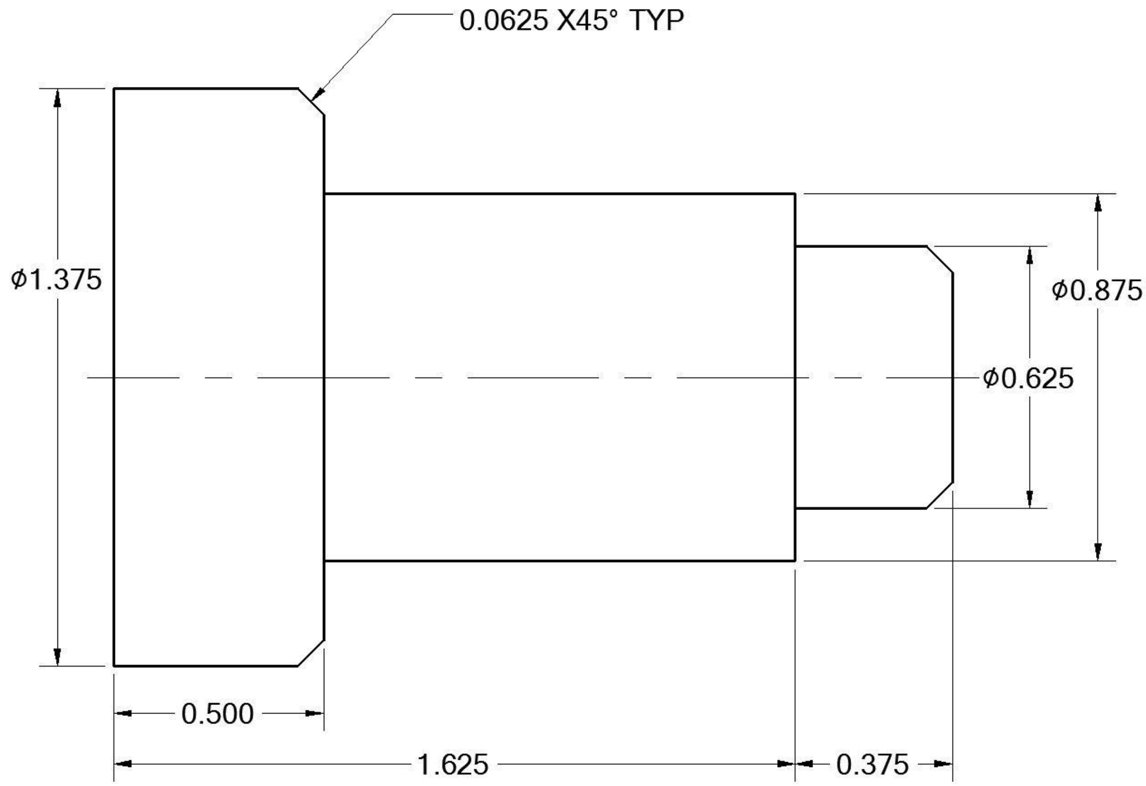
G42 will select cutter compensation to the **RIGHT** of the contouring direction; generally G42 is used for turning. The tool is compensated for the radius of the tool tip. The value of the compensation (tool radius) must be entered in the controller registry during set-up.

Figure 12a



13 PROGRAMMING USING G42

EXERCISE #13



EXAMPLE OF PROGRAMMING WITH TOOL NOSE RADIUS OFFSET

```
%
O00013
N1 G20...

N52 T0200                         (TOOL CALL FOR FINISH TURN)
N53 G96 S450 M03                  (SETTING OF SPEED CLOCKWISE)
N54 G00 G42 X1.5 Z0.5 T0202 M08  (RAPID TO SAFE POSITION APPLY T.N.R.O.)
N55 X0.375 Z0.0625                (RAPID TO START POSITON OF CHAMFER)
N56 G01 X0.625 Z-0.0625 F0.008    (START FINAL PROFILE FEED 0.008"/REV)
N57 Z-0.375
N58 X0.875
N59 Z-1.5
N60 X1.25
N61 X1.375 Z-1.5625
N62 Z-2.0
N63 U0.05                       (INCREMENTAL X PULLOUT)
N64 G00 G40 X1.5 Z0.5 T0200     (RAPID TO SAFE POSITION, CANCEL T.N.R.O.)
N65 M01
```

NOTE: T.N.R.O. = Tool Nose Radius Offset

The initial tool call is without the tool offset number, when the tool nose radius offset is called then the tool offset is called, to include any size offsets with the T.N.R.O.

PROGRAMMING EXERCISE

EXERCISE #13-1

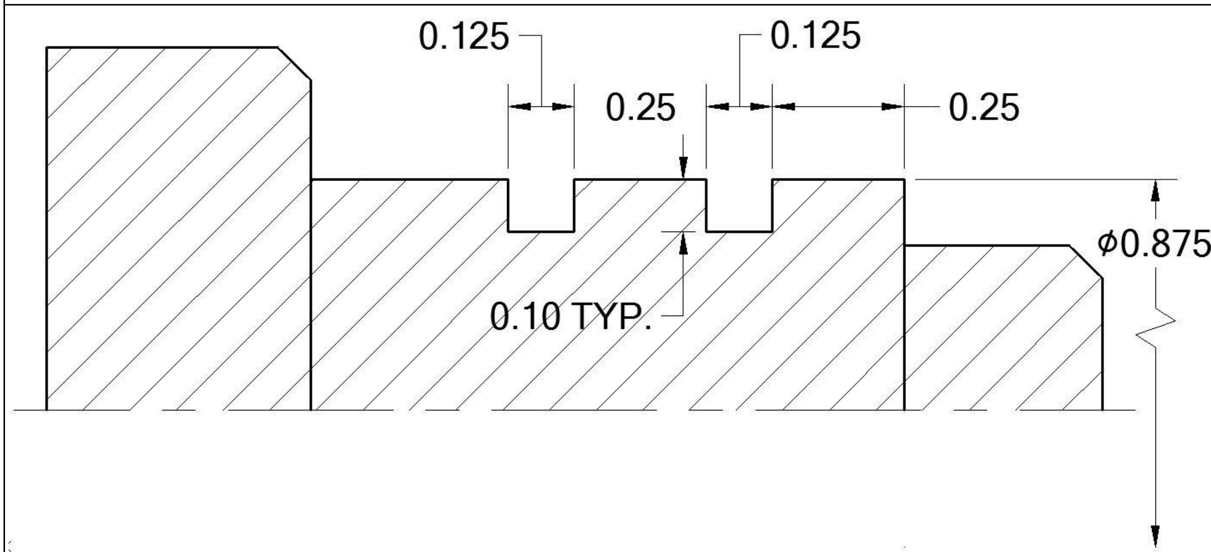
- ***Edit exercise 11-3 to now include Tool Nose Radius Offset for both turning and boring.***

GROOVING

When a circular slot or groove is needed in a part a special shaped tool may be needed. A grooving tool is that tool. It comes in many shapes and sizes but usually has one thing in common, which is that it feeds straight in along the X axis and plunges into the part. A grooving tool is not made for turning but to plunge into the part and make a groove around the part.

PARTING-OFF

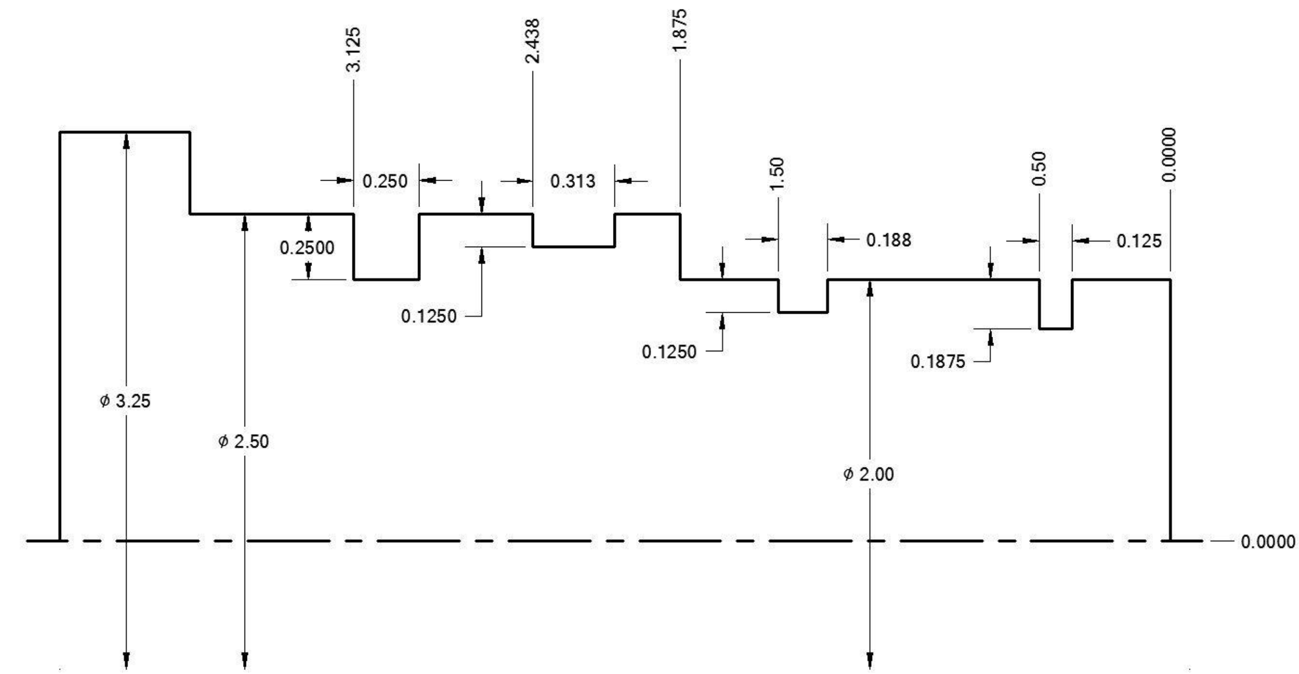
Parting-off is very similar to grooving but instead of stopping at a required depth of groove, the part-off tool is able to go right to the center of the part to allow the finished part to fall away from the chuck of the lathe. The design of the tool may be similar but the intent is very different.



EXAMPLE OF PROGRAMMING A GROOVING OPERATION

```
O0001400(TYPICAL GROOVING EXAMPLE)

N1 G20
...
N90 T0900 M42                 (TOOL CALL & GEAR SELECTION)
N91 G97 S650 M03              (SET SPEED AND TURN CLOCKWISE)
N92 G00 X1.0 Z0.25 T0909 M08  (RAPID TO SAFE POSITION & PICK-UP OFFSETS)
N93 Z-0.375                   (RAPID TO GROOVE POSITION)
N94 G01 X0.675 F0.004         (FEED TO GROOVE DEPTH)
N95 G04 X0.5                  (DWELL 0.5 SECOND)
N96 X1.0 F0.05                (FAST FEED OUT)
N97 G00 W-0.375               (INCREMENTAL RAPID TO NEXT LOCATION)
N98 G01 X0.675 F0.004         (FEED TO GROOVE DEPTH)
N99 G04 X0.5                  (DWELL 0.5 SECOND)
N100 X1.0 F0.05               (FAST FEED OUT)
N101 G00 X1.0 Z0.25 T0900 M09 (RAPID TO SAFE POSITION CANCEL OFFSETS)
N102 M01                      (OPTIONAL STOP)
```

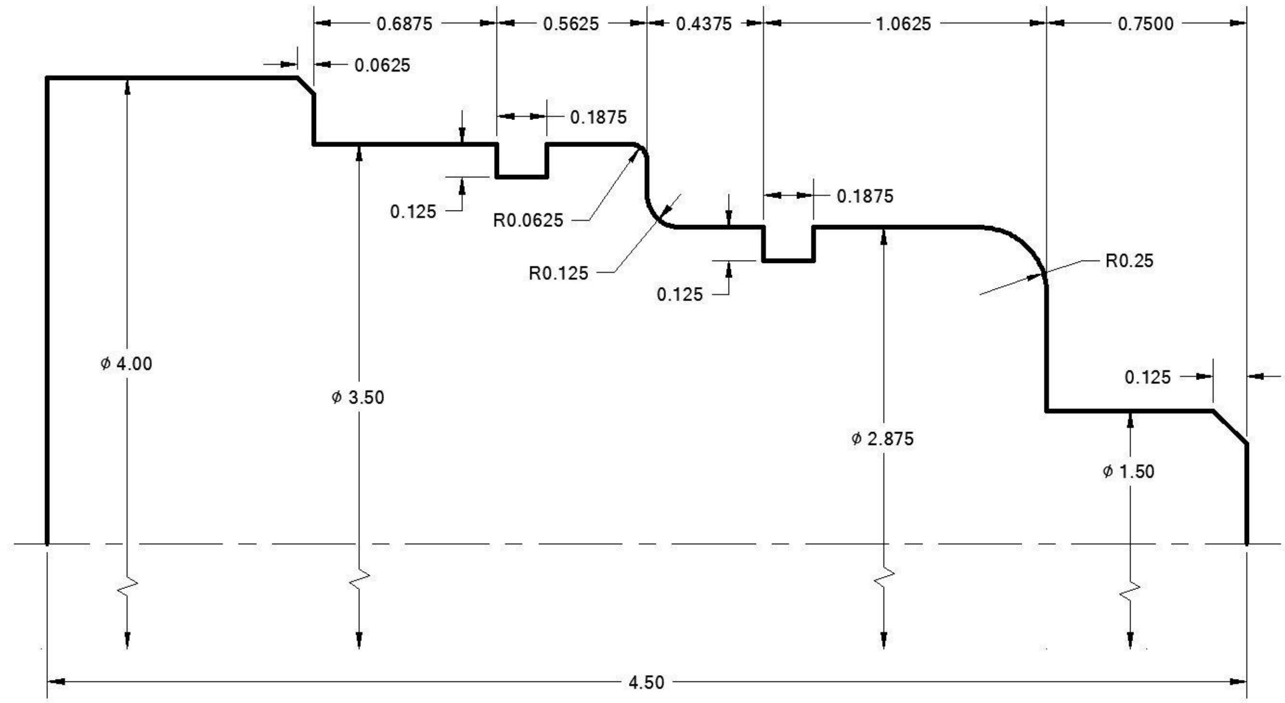


1. **Create Grooving Toolpath Using The Above Drawing**
2. **The Grooving Tool Is 0.125” Wide**
3. **Go To Finish Depth With Every Plunge**
4. **The Drawing Represents Only Half Of The Round Part**

%
O0001401;

Material: Aluminum 6061
CS=______________ft/min

XZ PLANE



1. Tool #1 (OD Roughing Tool) – Create A Toolpath To Rough The Above Profile	
Spindle Speed RPM = 4xCS/D=	Feed = in/min
Depth of Cut =0.1/side	Use a G71 Canned Cycle
2. Tool #2 (OD Finishing Tool) – Finish The OD Using The Profile Created In The First Operation	
Spindle Speed RPM = 4xCS/D=	Feed = in/min
Depth of Cut =0.02/side	Use a G70 Canned Cycle
3. Tool #9 (0.125 Right Hand Groove Tool) – Machine Both Grooves By Plunging In The Middle Of The Groove, Then Retract And Move Over To The Left And Right To Finish	
Spindle Speed RPM = 4xCS/D=	Feed = in/min
Start in the middle of groove	

CNC PROGRAMMING EXERCISE (CONTINUED)

EXERCISE #14-2

%
O00014;

CNC PROGRAMMING EXERCISE (CONTINUED)

EXERCISE #14-2

Computer Numerical Control Programming Work Book

Generic Lathe

About CamInstructor

CamInstructor is the one-stop shop for Mastercam Training Products and was created to serve the Mastercam community. It is the work of Matthew Manton and Duane Weidinger who are both dedicated to bringing you the best Mastercam Training Products available. Their goal is to offer a wide variety of Mastercam learning materials that appeal to your particular teaching or learning style.

About the Authors

Matthew Manton is a licensed Tool & Die Maker with 15 years experience in the Tool & Die Trade. He has a B.Ed. and is a Teacher of Tool & Die and CAD/CAM at George Brown College in Toronto, Ontario where he has taught for over 20 years. Matthew is a Certified Distance Education Instructor and has been teaching Mastercam for the past 10 years.

Duane Weidinger is a licensed Machinist with over 10 years experience in the machining trade. He also holds a Degree in Education with over 15 years experience teaching a combination of CNC Machining and CAD/CAM Programming.

Available Book Titles:

Training Guides include:
Book and DVD of Videos including Mastercam Demo
Mastercam Training Guide - Mill 2D
Mastercam Training Guide - Mill 3D
Mastercam Training Guide - Mill 2D&3D
Mastercam Training Guide - Lathe
Mastercam Training Guide - Multi-Axis
Mastercam Training Guide - Solids
Mastercam Training Guide - Wire
Mastercam Training Guide - Teacher Kit

Combo Books
Mastercam Training Guide – Mill 2D/Lathe Combo
Mastercam Training Guide – Mill 2D&3D/Lathe Combo

Online Training/Network License:

Premium Bundle: Mill 2D, Mill 3D, Lathe, Solids, Wire, Multi-Axis and Teacher Kit
Available for online use or can be installed on the school server
and licensed to students.

camInstructor
330 Chandos Crt. Kitchener, ON N2A 3C2
phone: 1-877-873-6867 fax: 1-866-741-8421
email sales@caminstructor.com • www.caminstuctor.com

9 781897 466834