CNC Programming Enhanced Learning System

Mill

By Matthew Manton and Duane Weidinger

camInstructor

CNC Programming Enhanced Learning System - Mill
Published by
CamInstructor Incorporated
330 Chandos Crt.
Kitchener, Ontario
N2A 3C2
www.caminstructor.com

Date: May 1, 2011
Author: Matthew Manton and Duane Weidinger
ISBN: 978-1-897466-88-9

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

Limit of Liability/Disclaimer of Warranty: While the Publisher and Author have used their best efforts in preparing this book, they make no representations or warranties with respect to the accuracy or completeness of the contents of this book and specifically disclaim any implied warranties of merchantability or fitness for a particular purpose. No warranty may be created or extended by representatives. The advice and strategies contained in this book may not be suitable for the readers or users situation. Neither the publisher nor author shall be liable for any damage, loss or any other damages, including but not limited to special, incidental, consequential, or other damages including personal.

Notice
CamInstructor Inc. reserves the right to make improvements to this book at any time and without notice.

Trademarks
All brands are the trademark of their respective owners.

Printed in Canada

Requirements
Use of the Multi-media CD/DVD requires a computer with speakers, and CD/DVD ROM.
September 1, 2011

TABLE OF CONTENTS

LESSON-1 ..ABSOLUTE & INCREMENTAL POSITIONING..1

EXERCISES 1 THROUGH 4 ABSOLUTE & INCREMENTAL.. 5

LESSON-2 .. INTRODUCTION TO CNC CODES 9

AUTOMATIC TOOL CHANGER STANDARD TOOL CAROUSEL 10

COMMONLY USED PREPARATORY G-CODES 11

COMMONLY USED MISCELLANEOUS M-CODES 12

EXAMPLE OF PROGRAM START-UP BLOCKS 13

EXAMPLE OF PROGRAM END BLOCKS 14

EXAMPLE OF PROGRAM TOOL CHANGE BLOCKS 14

RAPID (G00) AND LINEAR (G01) INTERPOLATION 15

CNC PART #1 – SPOT DRILLING SAMPLE PROGRAM... 17

LESSON-3 .. CREATING CNC PRORAMS - CNC PART #1 19

CNC PART #1 – SPOT AND DRILLING SAMPLE PROGRAM1 20

LESSON-4 .. DRILLING USING CANNED CYCLES 23

DRILLING CANNED CYCLES.. 25

CNC PART #1 – SPOT AND DRILLING PROGRAM USING G81 26

CNC PART #1 – WHAT COULD GO WRONG?.. 28

TABLE OF CONTENTS

LESSON-5 .. DRILLING USING CANNED CYCLES 29

CNC PART #2 - SPOT AND DRILLING PROGRAM USING G81.................................... 30

CNC PART #2 - CREATE THE PROGRAM TO SPOT AND DRILL 32

CNC - PART #2 - TYPING UP YOUR PROGRAM USING WINDOWS NOTEPAD........... 34

CNC - PART #2 - BACKPLOTTING 37

CNC - PART #3 - CREATE THE PROGRAM 39

CNC - PART #3 - BACKPLOTTING 43

CNC - PART #4 - CREATE THE PROGRAM 45

LESSON-6 .. STRAIGHT LINE MILLING – LINEAR INTERPOLATION...49

EXERCISE #1 - ABSOLUTE & INCREMENTAL POSITIONING 50

CNC PART #5 – STRAIGHT LINE MILLING SAMPLE PROGRAM 51

CNC PART #6 – STRAIGHT LINE MILLING SAMPLE PROGRAM 54

CNC PART #7 – CREATE THE PROGRAM.. 57

CNC PART #8 – CREATE THE PROGRAM.. 60

LESSON-7 .. CIRCULAR INTERPOLATION..63

CIRCULAR INTERPOLATION EXERCISES 65

CIRCULAR INTERPOLATION SAMPLE PROGRAMS.. 72

LESSON-8 .. CIRCULAR INTERPOLATION..77

CNC PART #9 – CIRCULAR INTERPOLATION CREATE THE PROGRAM 78

CNC PART #10 - CIRCULAR INTERPOLATION CREATE THE PROGRAM 84

TABLE OF CONTENTS

LESSON-9 .. CIRCULAR INTERPOLATION 91

CNC PART #11 - CIRCULAR INTERPOLATION CREATE THE PROGRAM 92

CNC PART #12 - CIRCULAR INTERPOLATION CREATE THE PROGRAM 98

LESSON-10 CUTTER COMPENSATION 105

INTRODUCTION TO CUTTER COMPENSATION 106

CNC PART #13 - CUTTER COMPENSATION CREATE THE PROGRAM 108

CNC PART #14 - CUTTER COMPENSATION CREATE THE PROGRAM 115

APPENDIX 123

EXTRA CNC PROGRAMMING EXERCISES 124

PREPATORY FUNCTIONS – G-CODES 131

MISCELLANEOUS FUNCTIONS – M-CODES 134

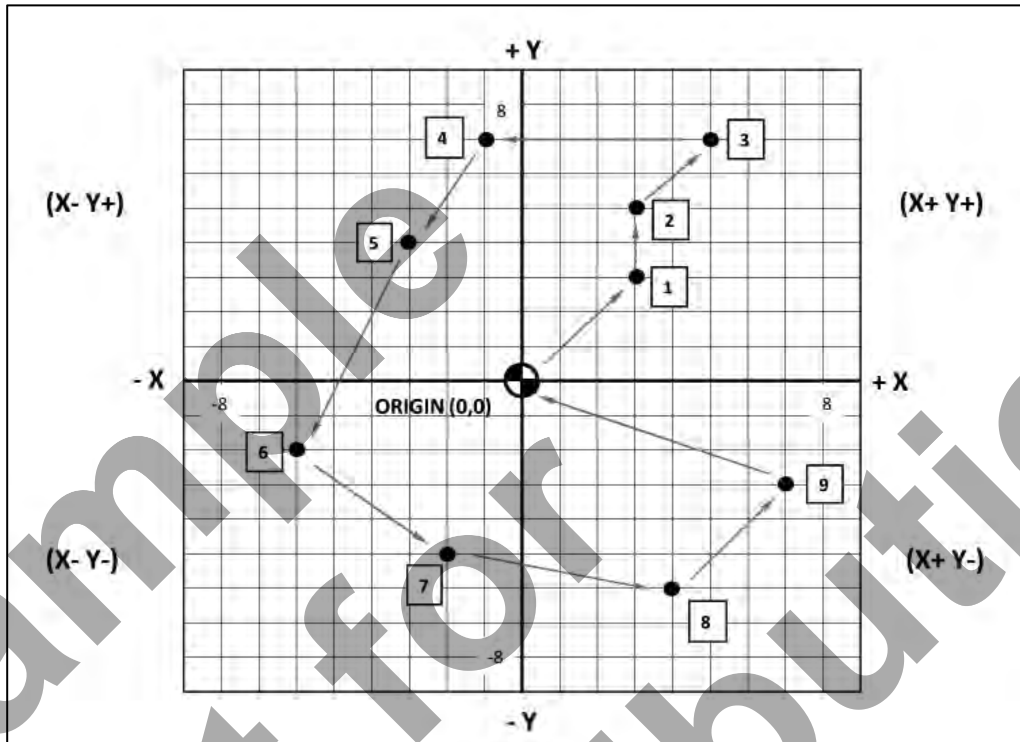
STANDARD DRILL SIZES – INCHES 136

INCH TAP DRILL SIZES 137

METRIC TAP DRILL SIZES 138

NCPLOT INSTALLATION GUIDE 139

CNC PROGRAMMING WORKBOOK



LESSON-1

ABSOLUTE & INCREMENTAL POSITIONING

camInstructor

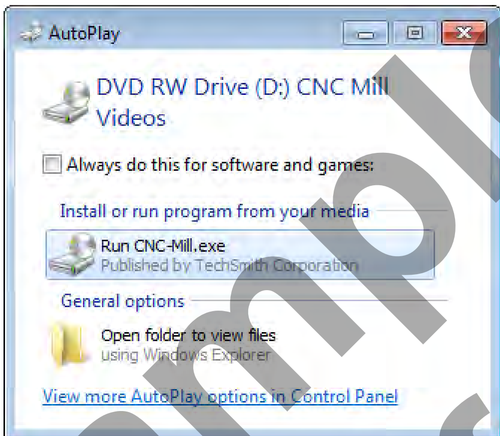
LESSON-1 – Introduction

The **CNC Enhanced Learning System** includes the **CNC Programming Student Workbook** and a **DVD** with the following Videos and support files on it.

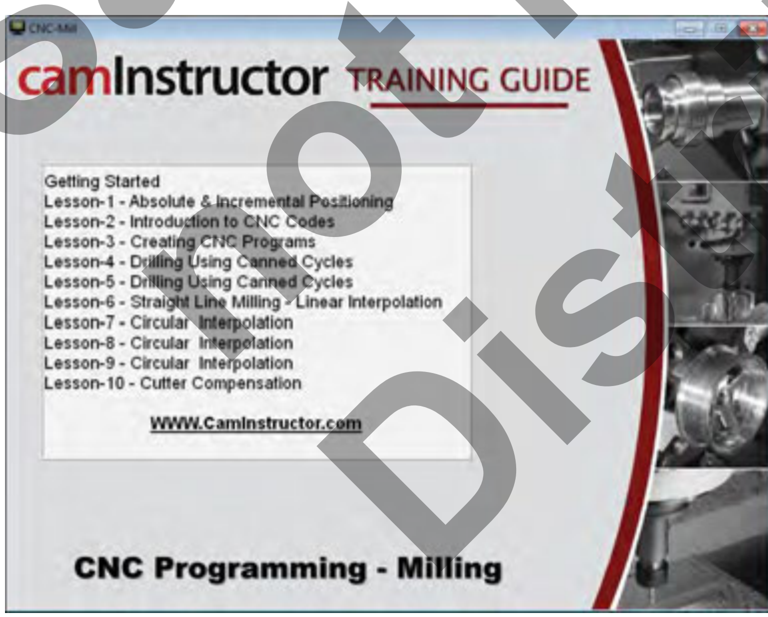
1. Self-Learning Videos
2. NCPlot Installation Software

To view what's on the DVD just follow the instructions below. We encourage you to take a few moments to watch the Getting Started video on the DVD as it provides an overview of how the system works.

Just pop the DVD into your computer, the autorun feature should display the AutoPlay window. Click on the **Run CNC-Mill.exe** file as shown below. Note, if this window is not displayed after putting the DVD into your computer, go to the file manager feature on your computer and select the DVD drive and double click on the **Run CNC-Mill.exe** file.



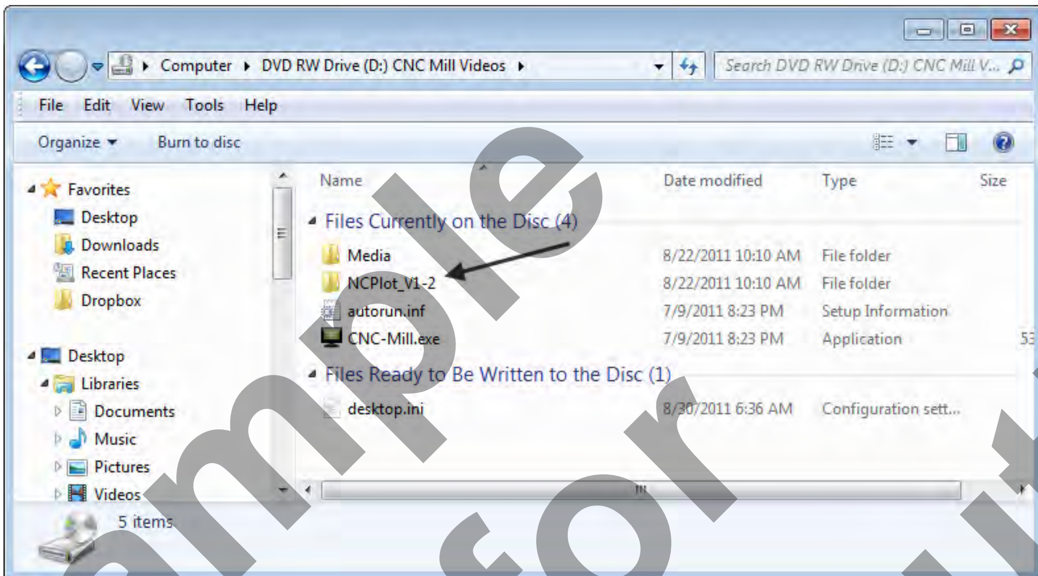
The following Menu Screen should appear;



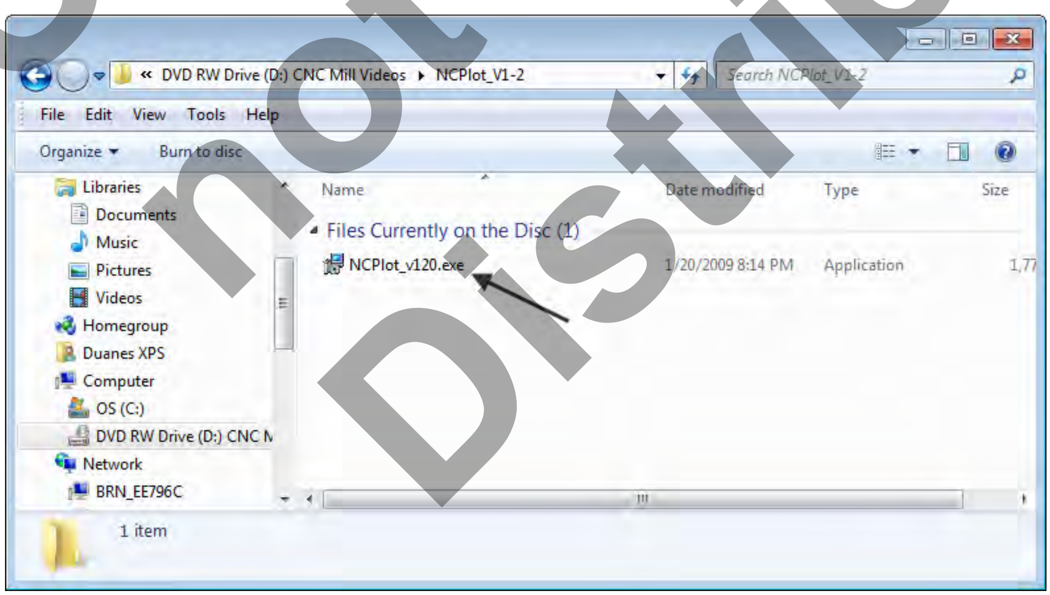
The Menu is your easy access to the Instructional Videos that will guide you through the content and provide you with all the information you need to get through this workbook. You will notice that there are 10 Lessons and a Getting Started link. Each Lesson matches the corresponding lesson in this workbook. Be sure to watch the video first, it will guide you to refer to the workbook.

Your first task is to watch the Getting Started Video. Don't worry about taking notes or filling out anything in the workbook while you watch the Getting Started Video, it is just a preview of what to expect.

The second item on the DVD is the NCPlot installation file. We have provided this to you free of charge so you can install NCPlot onto your computer. NCPlot Software enables you to type in the CNC Code (G Code) and watch what it will do. It is a handy tool to see if your CNC Programs are correct. To access the NCPlot installation file put the DVD into your computer and locate the NCPlot_V1-2 folder as shown below.



Double click on the folder and then double click on the NCPlot_v120.exe file and follow the onscreen instructions.



LESSON-1 – Introduction

Okay let's get started.

Step 1 - Plug in your headphones or make sure your speakers are plugged in and turned on.

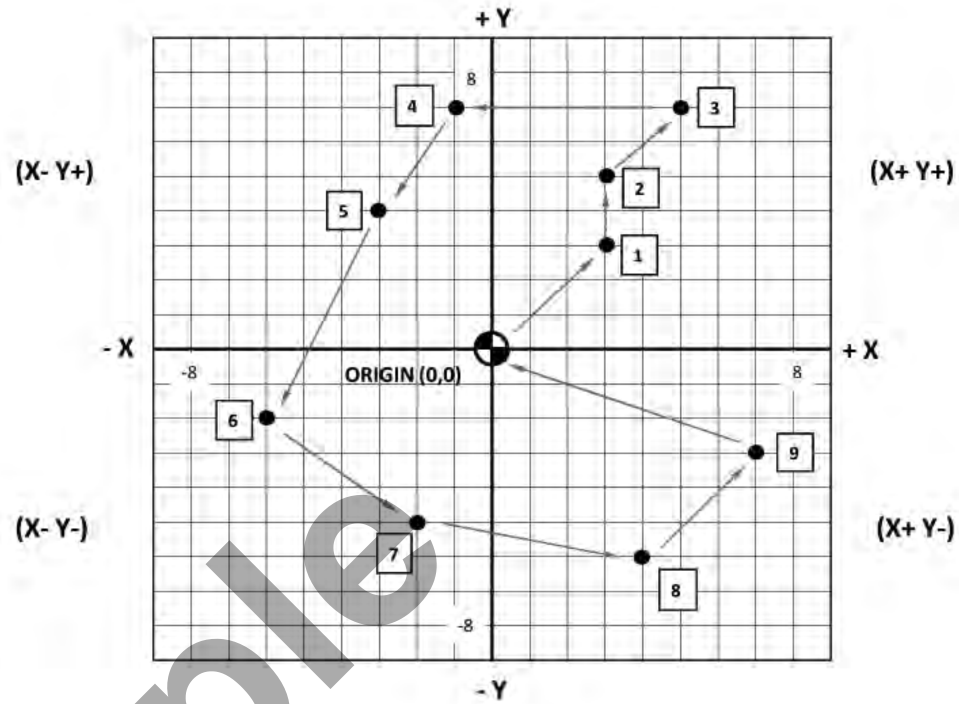
Step 2 - Put the DVD into your computer and launch the menu.

Step 3 - Click on Getting Started and watch the video through to the end. Feel free to pause and rewind the video if you need to watch something again.

Step 4 - Click on Lesson 1 and then click on Lesson-1 – Unit-1, as indicated it is 9 minutes long.

Step 5 - Proceed through the Videos in the proper order and make sure to follow along with the Workbook. Good luck and have fun.

LESSON-1 – EXERCISE #1 - ABSOLUTE & INCREMENTAL POSITIONING

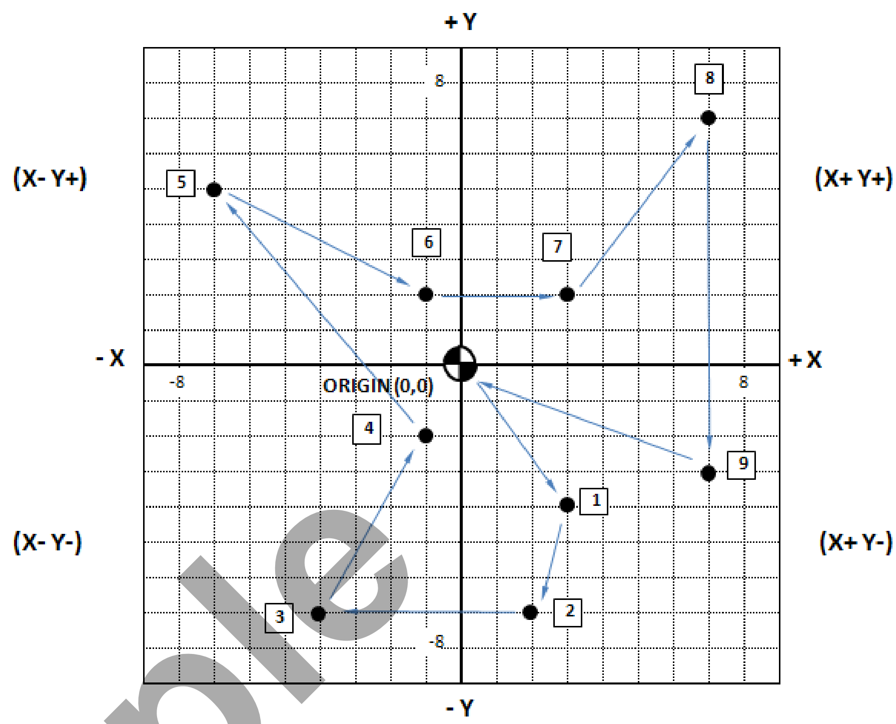


G90 ABSOLUTE PROGRAMMING
All axis motions are based on a fixed zero reference point, known as ABSOLUTE ZERO (part zero).
Each coordinate is in relation to this absolute zero using Cartesian Co-ordinates.

G91 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.

STARTING AT THE POINT O (ORIGIN), DESCRIBE THE PATH FROM O THROUGH ALL 9 POINTS AND BACK TO THE POINT O USING G90 & G91

G90	X	Y	G91	X	Y
O (Origin)	0	0	O → 1	3	3
1	3	3	1 → 2	0	2
2	3	5	2 → 3	2	2
3	5	7	3 → 4	-6	0
4	-1	7	4 → 5	-2	-3
5	-3	4	5 → 6	-3	-6
6	-6	-2	6 → 7	4	-3
7	-2	-5	7 → 8	6	-1
8	4	-6	8 → 9	3	3
9	7	-3	9 → O	-7	3



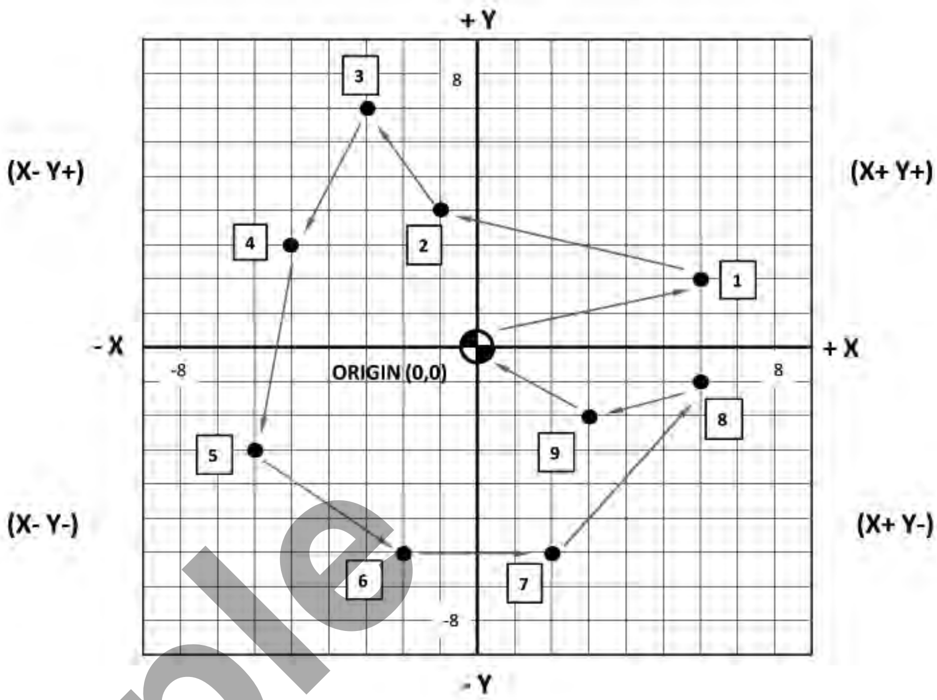
G90 ABSOLUTE PROGRAMMING
All axis motions are based on a fixed zero reference point, known as ABSOLUTE ZERO (part zero).
Each coordinate is in relation to this absolute zero using Cartesian Co-ordinates.

G91 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.

✐ ***STARTING AT THE POINT O (ORIGIN), DESCRIBE THE PATH FROM O THROUGH ALL 9 POINTS AND BACK TO THE POINT O USING G90 & G91***

G90	X	Y	G91	X	Y
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**		

LESSON-1 – EXERCISE #3 - ABSOLUTE & INCREMENTAL POSITIONING



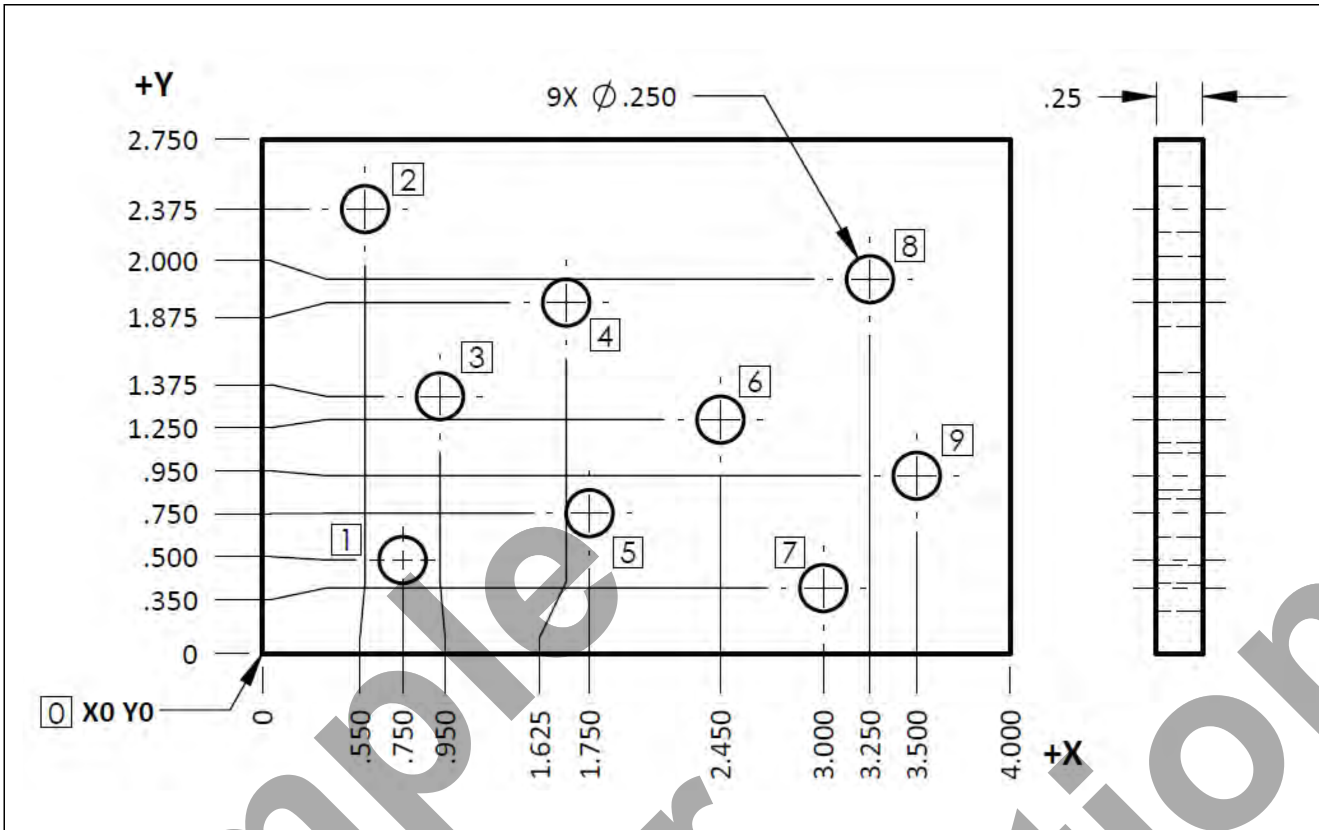
G90 ABSOLUTE PROGRAMMING
All axis motions are based on a fixed zero reference point, known as ABSOLUTE ZERO (part zero).
Each coordinate is in relation to this absolute zero using Cartesian Co-ordinates.

G91 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.

✐ ***STARTING AT THE POINT O (ORIGIN), DESCRIBE THE PATH FROM O THROUGH ALL 9 POINTS AND BACK TO THE POINT O USING G90 & G91***

G90	X	Y	G91	X	Y
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		

LESSON-1 – EXERCISE #4 - ABSOLUTE & INCREMENTAL POSITIONING



✎ ***STARTING AT THE POINT O (ORIGIN), DESCRIBE THE PATH FROM O THROUGH ALL 9 POINTS AND BACK TO THE POINT O USING G90 & G91***

G90	X	Y	G91	X	Y
O (Origin)	**0**	**0**	**O → 1**	**0.750**	**0.500**
1	**0.750**	**0.500**	**1 → 2**	**-0.200**	**1.875**
2	**0.550**	**2.375**	**2 → 3**	**0.400**	**-1.000**
3	**0.950**	**1.375**	**3 → 4**		
4			**4 → 5**		
5			**5 → 6**		
6			**6 → 7**		
7			**7 → 8**		
8			**8 → 9**		
9			**9 → O**		

CNC PROGRAMMING WORKBOOK

CODE	FUNCTION
G00	Rapid traverse motion; This is used for non-cutting rapid moves of the machine axis, or rapid retract moves after cuts have been completed. Maximum rapid motion (I.P.M.) of a CNC Machine will vary dependent on machine model.
G01	Linear interpolation motion; Used for cutting in a straight line under a controlled feedrate. 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
G17	Circular Motion XY Plane Selection
G20	Verify Inch Coordinate Positions
G21	Verify Metric Coordinate Positions
G28	Machine Home (Rapid traverse) G91 is required for rapid move to the G28 reference point.
G40	Cutter Compensation CANCEL

LESSON-2

INTRODUCTION TO CNC CODES

camInstructor

LESSON-2 - INTRODUCTION TO CNC CODES
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 tools and tool numbers listed below.

Carousel #	Tool Description
1	0.125" Diameter Flat End Mill
2	0.250" Diameter Flat End Mill
3	0.375" Diameter Flat End Mill
4	0.500" Diameter Flat End Mill
5	0.750" Diameter Flat End Mill
6	0.375" Diameter Spot Drill
7	0.250" Diameter Drill
8	0.201" Diameter Drill – Number 7 drill
9	0.25"-20 UNC Tap
10	#4 Center Drill

COMMONLY USED PREPARATORY **G** CODES

CODE	FUNCTION
G00	Rapid traverse motion; This is used for non-cutting rapid moves of the machine axis, or rapid retract moves after cuts have been completed. Maximum rapid motion (I.P.M.) of a CNC Machine will vary dependent on machine model.
G01	Linear interpolation motion; Used for cutting in a straight line under a controlled feedrate. 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
G17	Circular Motion XY Plane Selection
G20	Verify Inch Coordinate Positions
G21	Verify Metric Coordinate Positions
G28	Machine Home (Rapid traverse) G91 is required for rapid move to the G28 reference point.
G40	Cutter Compensation CANCEL
G41	Cutter Compensation LEFT of the programmed path
G42	Cutter Compensation RIGHT of the programmed path
G43	Tool Length Compensation
G49	Tool Length Compensation CANCEL
G53	Positions the machine axis relative to Machine Home. It is non modal.
G54	Work Coordinate #1 (Part zero offset location)
G80	Canned Cycle CANCEL
G81	Drill Canned Cycle
G82	Spot Drill Canned Cycle
G83	Peck Drill Canned Cycle
G84	Tapping Canned Cycle
G90	Absolute Programming Positioning
G91	Incremental Programming Positioning
G98	Canned Cycle Initial Point Return
G99	Canned Cycle Rapid (R) Plane Return

COMMONLY USED MISCELLANEOUS **M** CODES

CODE	FUNCTION
M00	The M00 code is used for a Program Stop. The spindle stops and the coolant is turned off. Pressing CYCLE START again will continue 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 used 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. The M05 is used to turn the spindle off at the end of program or before a tool change. If the coolant is on, the M05 will turn it off.
M06	The tool change command along with a tool number will action a tool change. This command will automatically stop the spindle, Z-axis will move up to the machine zero position and the selected tool will be put in the spindle. The coolant pump will turn off right before executing the tool change.
M08	Coolant ON command.
M09	Coolant OFF command.
M30	Program End and Reset to the beginning of program.

Note: Only one “M” code can be used per line. And the M-codes will be the last command to be executed in a line, regardless of where it is located in that line.

EXAMPLE OF PROGRAM **START-UP BLOCKS**

%	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 G00 G17 G40 G49 G80 G90	G00 - Rapid Traverse G17 - X, Y Circular Plane Selection G40 - Cutter Compensation Cancel G49 - Tool Length Compensation Cancel G80 - Canned Cycle Cancel G90 - Absolute Programming	
N30 T8 M06	T8 - Tool number #8 to be loaded into the spindle. M06 - Tool Change	
N40 G00 G90 G54 X1.0 Y1.0 S4000 M03	G00 - Rapid Traverse G90 - Activates control to be in ABSOLUTE. G54 - Selects work coordinate offset system No. 1 X__ - Axis move to initial X position. Y__ - Axis move to initial Y position. S4000 - Spindle speed 4000 RPM for this tool. M03 - Turns the spindle on in a clockwise direction	
N50 G43 H8 Z2. 0	G43 - Tool Length Compensation: Recognizes the tool length offset value stored in the Hnn code offset display register in the offset length display. H8 - Defines to the control the offset register the tool offset value is stored in. **** Tool Length offset # = Tool #*** Z2.0 - Informs the control to move from full spindle retract to this Z value and apply the tool length offset.	

EXAMPLE OF PROGRAM **END BLOCKS**

N200 G00 Z2.0	G00 - Rapid Traverse Z2.0 – Retracts tool to 2.0 above part zero	
N210 M05	M05 – Turn off spindle	
N220 G28 G91 Z0 **** N220 G53 Z0***	G91 - Incremental Programming G28 - Machine Zero Return Z0 - Z axis in the up direction to machine zero	***Send to machine zero Z-axis first to avoid any crash.***
N230 G28 X0 Y0 **** N230 G53 X0 Y0***	G28 - Machine Zero Return X0 - X axis to machine zero Y0 - Y axis to machine zero	****G53 is another way to return to machine zero***
N240 M30	M30 – End of Program and Reset	

EXAMPLE OF PROGRAM **TOOL CHANGE LINES**

N100 G00 Z2.0	Rapid Traverse and Retracts tool to 2.0 above part zero	
N110 M05	M05 – Turn off spindle	
N120 G28 G91 Z0 ; / ****N120 G53 Z0*	Machine Zero Return - Z axis	***Send to machine zero Z-axis first to avoid any crash.***
N130 G28 X0 Y0 / ****N130 G53 X0 Y0*	Machine Zero Return - X, Y axis	
N140 M01	Optional Program Stop	
N150 T9 M06	Tool Change - Tool # 9	
N160 G00 G90 G54 X1.0 Y1.0 S4000 M03	Turn on the spindle and Rapid traverse to X1. Y1.	
N170 G43 H9 Z2.0	Tool Length compensation for Tool #9 (H9)	

*G53 - Positions the machine axis relative to Machine Home. It is non modal.

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 all axis motion up to three axes at once.

This G00 code is modal and causes all the following blocks to be in rapid motion until another Group 01 code is specified. The actual rapid federate is dependent on the machine.

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.

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 up to three axes at once.

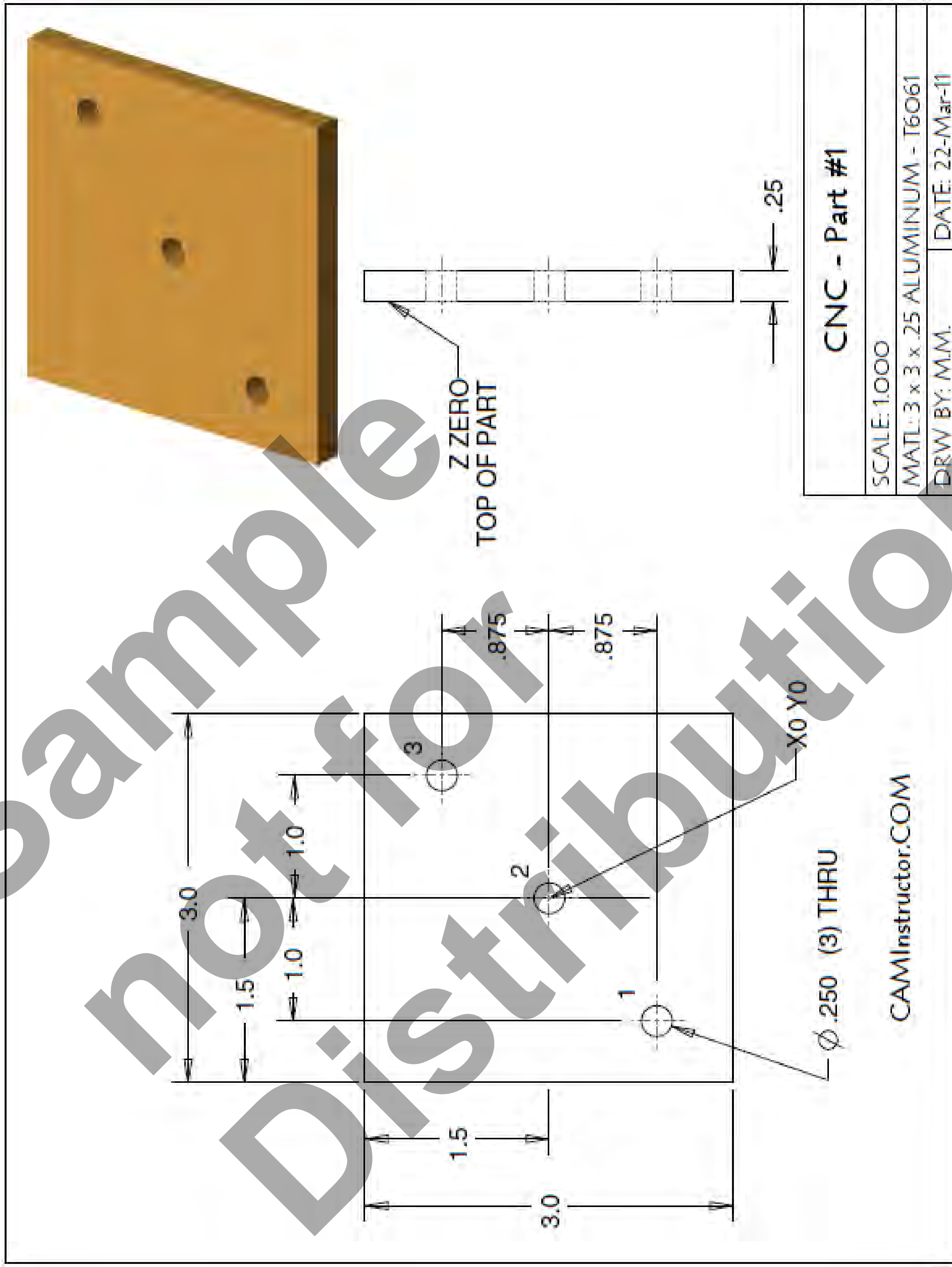
All axes specified will start at the same time and proceed to their destination and arrive simultaneously at the specified feedrate.

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

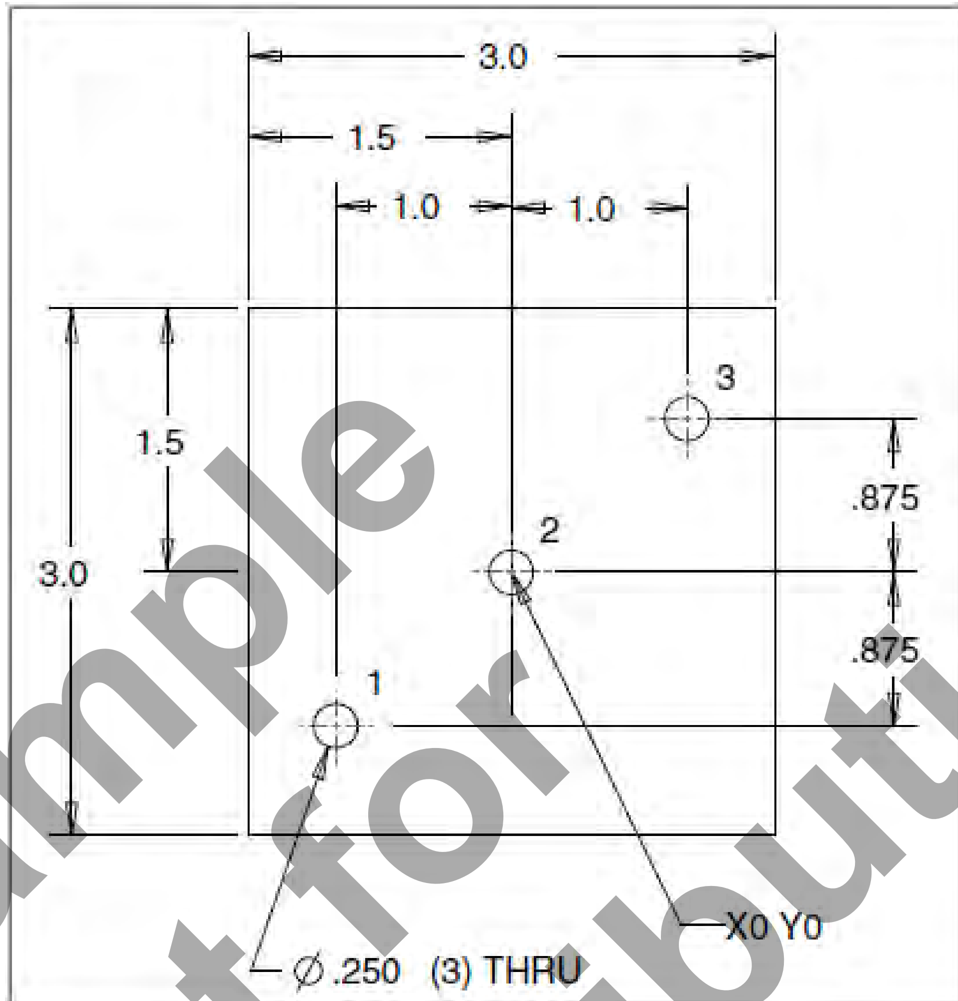
G01 is used for

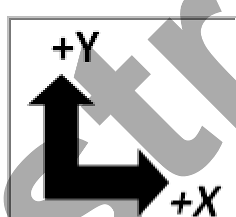
- Drilling a hole
- Machining a slot
- Machining a profile

LESSON-2 - CNC - PART #1



- ***WORK OUT THE X AND Y COORDIANTES FOR HOLES 1,2 AND 3***
  - X0Y0 is at the centre of the part

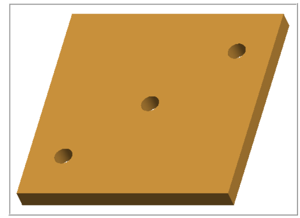




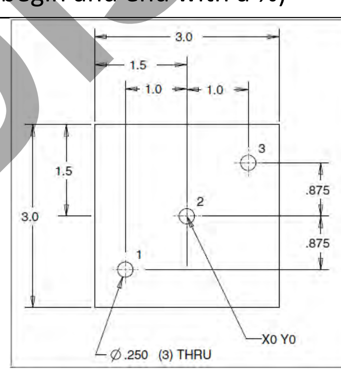
G90	X	Y
1		
2		
3		

LESSON-2 - CNC - PART #1

- ***PROGRAM TO <u>SPOT DRILL THE THREE HOLES ONLY</u> USING A COMBINATION OF G00 AND G01 (CANNED CYCLE DRILL WILL BE USED LATER)***
  - Below is the program to spot drill the three holes with an explanation of each block
  - Use a 0.375” diameter Spot Drill Tool # 6
  - Spindle Speed = 2750 Feed rate = 11 IPM
  - Spot Drill Depth = Z-0.150”
  - X0Y0 is at the centre of the part
  - Z=0 is the top of the part.
  - Information inside the parenthesis ( ) is a comment.
  - The CNC control will ignore all text between the parenthesis



%	(Program must begin and end with a %)
O1	(Program #1 - CNC-PART-1-SPOT DRILLING ONLY)
N10 G20	(Inch programming)
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)	
N40 G00 G90 G54 X-1.0 Y-0.875 S2750 M03 – (Rapid to the X and Y position and turn on the spindle at 2750 RPM)	
N30 T06 M06	(T6-Select tool number 6 to be loaded M06-Activates the tool changer)
N50 G43 H06 Z0.1	(G43 - Activate the tool offset value stored in H06 and rapid to Z0.1)
N60 G01 Z-0.15 F11.0	(Hole #1 - Feed down to Z depth at 11 inches per minute)
N70 G00 Z0.1	(G00- Retract out of hole #1 at rapid to 0.1 above the top of the work piece)
N80 X0 Y0	(G00 is modal - Move at rapid in the X and Y axis to hole #2)
N90 G01 Z-0.15	(Hole #2 - Feed down to Z depth at 11 inches per minute, Feed rate is modal)
N100 G00 Z0.1	(G00- Retract out of hole #2 at rapid to 0.1 above the top of the work piece)
N110 X1.0 Y0.875	(G00 is modal - Move at rapid in the X and Y axis to hole #3)
N120 G01 Z-0.15	(Hole #3 - Feed down to Z depth at 11 inches per minute, Feed rate is modal)
N130 G53 G00 Z0 M05	(G53 – Machine Zero positioning, non modal. Rapid to machine zero in Z, switch spindle off)
N140 G53 X-15.0 Y0	(G53 – Rapid in relation to machine zero X-15.0 and Y0)
N150 M30	(Program end rewind program to the beginning)
%	(Program must begin and end with a %)



CNC Programming Workbook

Lesson-3

Creating CNC Programs - CNC Part #1

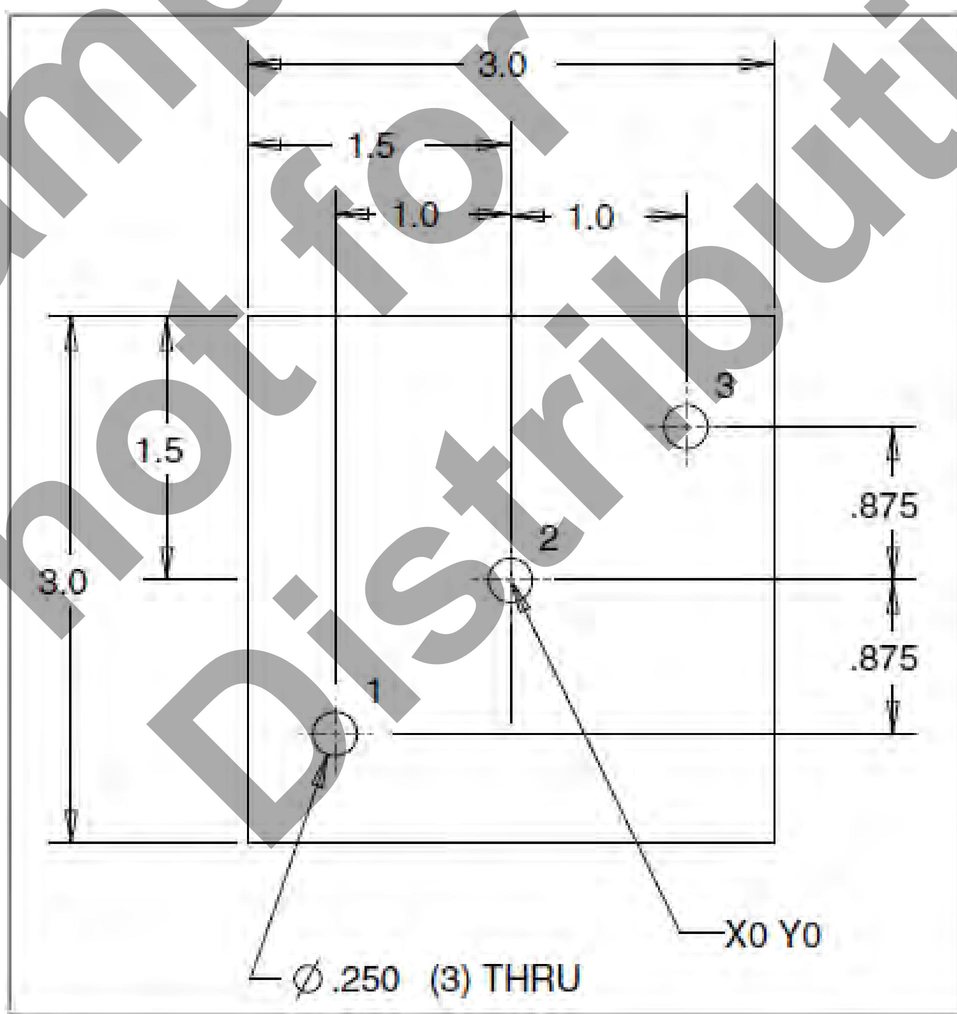
camInstructor

LESSON-3 - CNC - PART #1

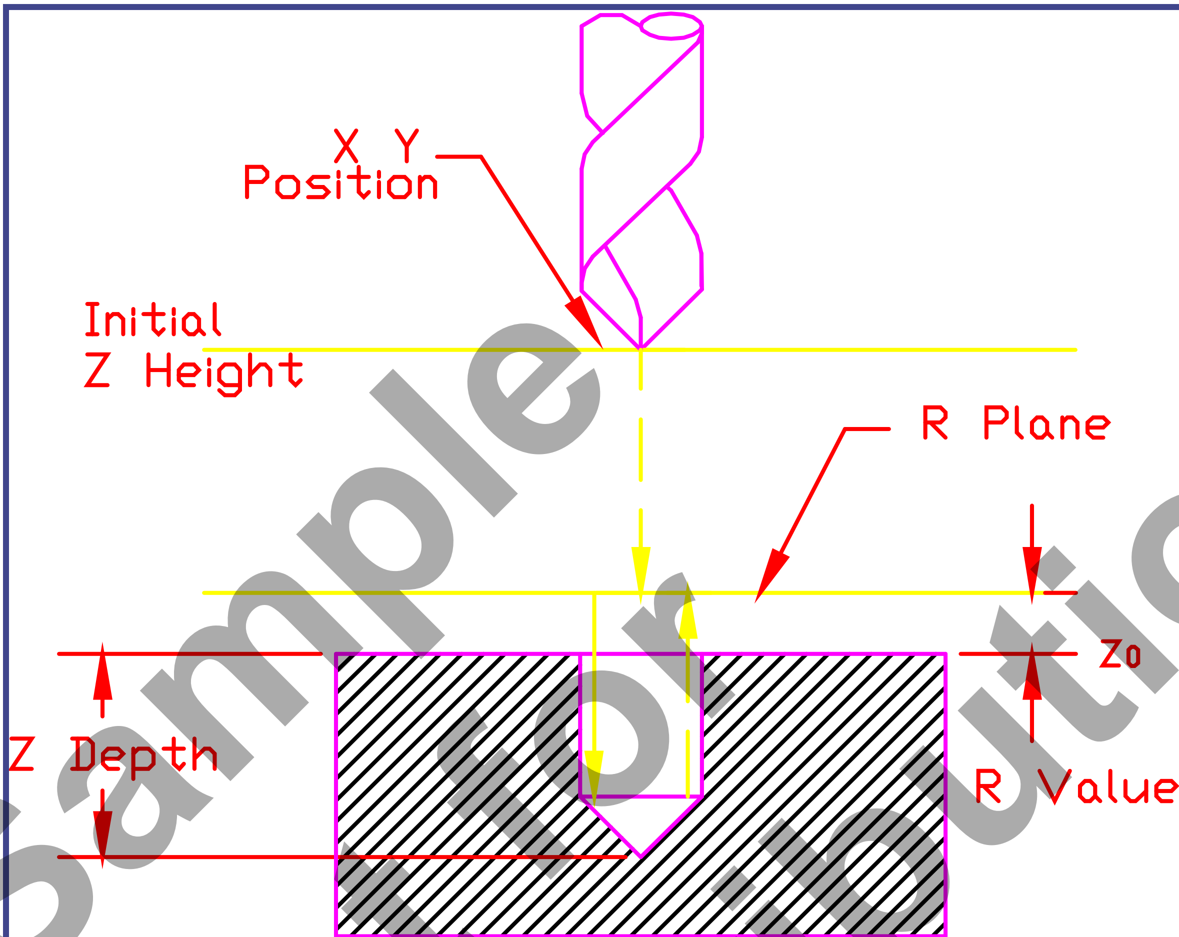
- ***PROGRAM TO <u>SPOT AND DRILL THE THREE HOLES</u> USING A COMBINATION OF G00 AND G01 (CANNED CYCLE WILL BE USED LATER)***
  - Below is the program to spot and drill the three holes with an explanation of each block
  - Use a 0.375” diameter Spot Drill Tool # 6
  - Spot Drill Spindle Speed = 2750 Feed rate = 11 IPM
  - Use a 0.250” diameter Drill Tool # 7
  - 0.250” diameter Drill Spindle Speed = 4500 Feed rate = 15 IPM
  - Spot Drill Depth = Z-0.150”
  - Drill Depth = Z-0.350”
  - X0Y0 is at the centre of the part
  - Z=0 is the top of the part.

%	(Program must begin and end with a %)
O2	(Program #2 - CNC-PART-1-SPOT AND DRILLING)
N10 G20	(Inch programming)
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)	
(SPOT DRILL 0.25” HOLES)	
N30 T06 M06	(T6-Select tool number 6 to be loaded M06-Activates the tool changer)
N40 G00 G90 G54 X-1.0 Y-0.875 S2750 M03	(Rapid to the X and Y position of Hole #1 and turn on the spindle at 2750 RPM)
N50 G43 H06 Z0.1	(G43 - Activate the tool offset value stored in H06 and rapid to Z0.1)
N60 G01 Z-0.15 F11.0	(Hole #1 - Feed down to Z depth at 11 inches per minute)
N70 G00 Z0.1	(G00- Retract out of hole #1 at rapid to 0.1 above the top of the work piece)
N80 X0 Y0	(G00 is modal - Move at rapid in the X and Y axis to Hole #2)
N90 G01 Z-0.15	(Hole #2 - Feed down to Z depth at 11 inches per minute, Feed rate is modal)
N100 G00 Z0.1	(G00- Retract out of hole #2 at rapid to 0.1 above the top of the work piece)
N110 X1.0 Y0.875	(G00 is modal - Move at rapid in the X and Y axis to hole #3)
N120 G01 Z-0.15	(Hole #3 - Feed down to Z depth at 11 inches per minute, Feed rate is modal)
N130 G53 G00 Z0 M05	(G53 – Machine Zero positioning, non modal. Rapid to machine zero in Z, switch spindle off)
N140 G53 X-15.0 Y0	(G53 – Rapid in relation to machine zero X-15.0 and Y0)
(DRILL 0.25” HOLES)	
N160 T07 M06	(T7-Select tool number 7 to be loaded M06-Activates the tool changer)
N170 G00 G90 G54 X-1.0 Y-0.875 S4500 M03	(Rapid to the X and Y position of Hole #1 and turn on the spindle at 4500 RPM)
N180 G43 H07 Z0.1	(G43 - Activate the tool offset value stored in H07 and rapid to Z0.1)
N190 G01 Z-0.35 F15.0	(Hole #1 - Feed down to Z depth at 15 inches per minute through part)
N200 G00 Z0.1	(G00- Retract out of hole #1 at rapid to 0.1 above the top of the work piece)

N210 X0 Y0	(G00 is modal - Move at rapid in the X and Y axis to hole #2)
N220 G01 Z-0.35	(Hole #2 - Feed down to Z depth, at 15 inches per minute, Feed rate is modal)
N230 G00 Z0.1	(G00- Retract out of hole #2 at rapid to 0.1 above the top of the work piece)
N240 X1.0 Y0.875	(G00 is modal - Move at rapid in the X and Y axis to hole #3)
N250 G01 Z-0.35	(Hole #3 - Feed down to Z depth at 15 inches per minute, Feed rate is modal)
N260 G53 G00 Z0 M05	(G53 – Machine Zero positioning, non modal. Rapid to machine zero in Z, switch spindle off)
N270 G53 X-15.0 Y0	(G53 – Rapid in relation to machine zero X-15.0 and Y0)
N270 M30	(Program end rewind program to the beginning)
%	(Program must begin and end with a %)



CNC PROGRAMMING WORKBOOK



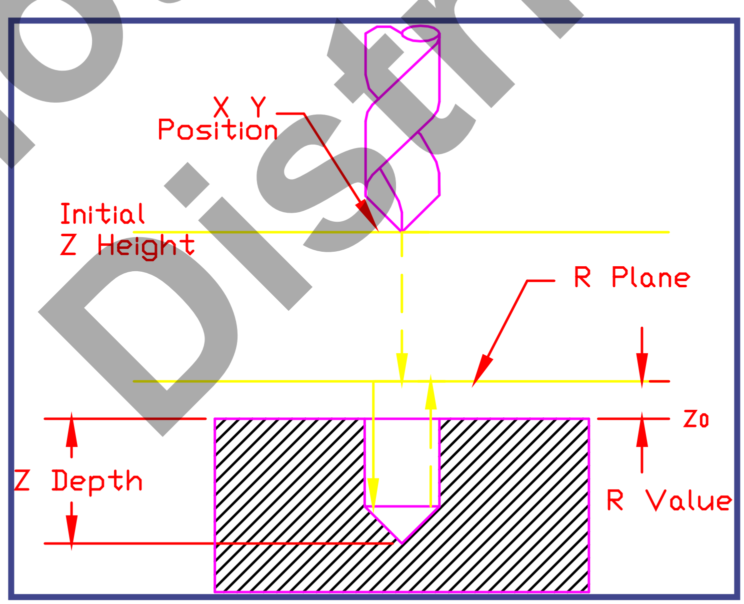
LESSON-4

DRILLING USING CANNED CYCLES

camInstructor

LESSON-4 DRILL CANNED CYCLE **G81**

G80 CANCEL CANNED CYCLE
A canned cycle permits multiple function programming on one block. A canned cycle is canceled with G80.
G81 CANNED CYCLE DRILL
Format: G99 G81 Z-0.625 R0.1 F10. **X** Rapid X location (Optional) **Y** Rapid Y location (Optional) **Z** Z-depth (Feed to Z-depth starting from R Plane) **R** R-Plane (Rapid point to start feeding) **F** Feed rate in inches/min This G code permits the inclusion of multiple axis motions on one block of program. It is used to reduce the length of program. The figure below shows the axis motions that are included with a Canned Cycle Drill. All Z axis motions are in ABSOLUTE with any other axis motions unaffected. *In a canned cycle drill, the cutter moves at rapid to the X and Y, then to a height above the part at rapid rate to the R Plane, which is a point above the work surface. From the R Plane the cutter feeds to the Z-depth at the specified feedrate. When the cutter reaches the Z depth, it retracts at rapid rate to the R Plane.* ***G99*** returns the tool to the R Plane after each hole, **G98** returns the tool to the initial starting plane.



DEEP HOLE PECK DRILL CANNED CYCLE G83

G83 DEEP HOLE PECK DRILL CANNED CYCLE
Format : G99 G83 Z-2.5 Q0.5 R0.1 F10. / G99 G83 Z-2.18 I0.5 J0.1 K0.2 R0.1 F9.

- **X*** Rapid X-axis location
- **Y*** Rapid Y-axis location
- **Z** Z-depth (feed to Z-depth starting from R plane)
- **Q*** Pecking equal incremental depth amount (if I, J and K are not used)
- **I*** Size of first peck depth (if Q is not used)
- **J*** Amount reducing each peck after first peck depth (if Q is not used)
- **K*** Minimum peck depth (if Q is not used)
- **P** Dwell time at Z-depth
- **R** R-plane (rapid point to start feeding)
- **F** Feed rate in inches (mm) per minute

* Indicates optional

This G code is similar to G81 but is used for drilling when the tool must be withdrawn periodically to allow chips to be removed from the hole.

This cycle allows the tool to rapid to the R Plane, feeds towards the Z depth in increments (traversing to the R Plane and back to the point where drilling was interrupted after each increment) until the tool reaches the final Z depth.

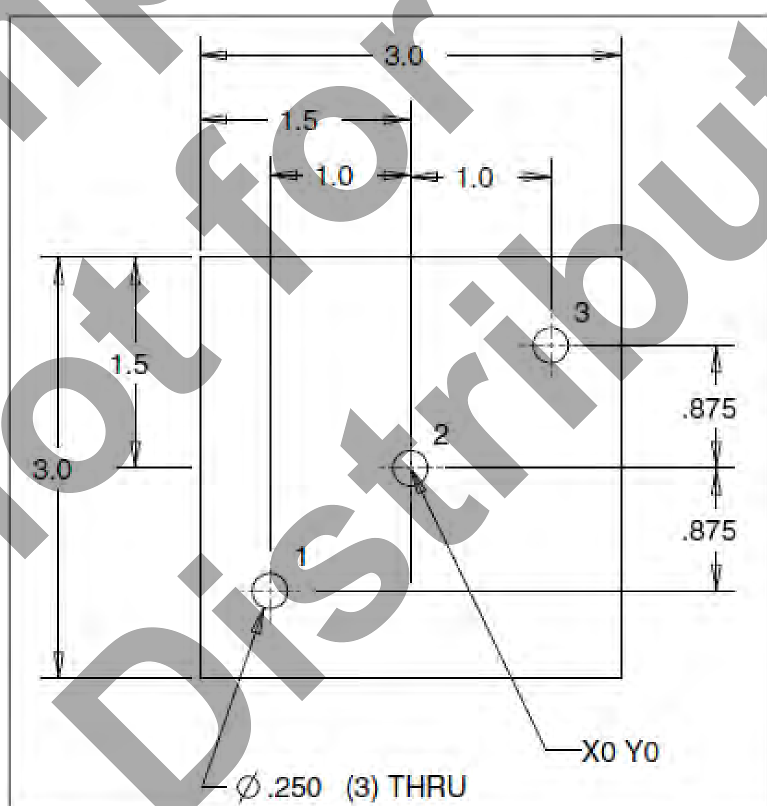
LESSON-4 - CNC - PART #1

- ***CREATE THE PROGRAM TO <u>SPOT AND DRILL THE THREE HOLES</u> USING CANNED CYCLE G81***
  - Use a 0.375” diameter Spot Drill Tool # 6
  - Spot Drill Spindle Speed = 2750 Feed rate = 11 IPM
  - Use a 0.250” diameter Drill Tool # 7
  - 0.250” diameter Drill Spindle Speed = 4500 Feed rate = 15 IPM
  - Spot Drill Depth = Z-0.150”
  - Drill Depth = Z-0.350”
  - X0Y0 is at the centre of the part
  - Z=0 is the top of the part.

%	(Program must begin and end with a %)
O3	(Program #3 - CNC-PART-1-SPOT AND DRILLING USING CANNED CYCLE DRILL G81)
N10 G20	(Inch programming)
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)	
N30 T06 M06	(T6-Select tool number 6 to be loaded M06-Activates the tool changer)
N40 G00 G90 G54 X-1.0 Y-0.875 S2750 M03	(Rapid to the X and Y position of Hole #1 and turn on the spindle at 2750 RPM)
N50 G43 H06 Z0.1	(G43 - Activate the tool offset value stored in H06 and rapid to Z0.1)
N60 G99 G81 Z-0.15 R0.1 F11.0	(Hole #1 – G81 - Feed down to Z depth at 11 inches per minute, and then retract at rapid to Z0.1, this is the R0.1 value. G99 returns the drill tip to the R value after drilling each hole)
N70 X0. Y0.	(Hole #2 - Move at rapid in the X and Y axis to Hole #2. Feed down to Z depth at 11 inches per minute and then retract at rapid to Z0.1)
N80 X1.0 Y.875	(Hole #3 - Move at rapid in the X and Y axis to Hole #3. Feed down to Z depth at 11 inches per minute and then retract at rapid to Z0.1)
N90 G80	(Cancel Canned Cycle Drill)
N100 G53 G00 Z0 M05	(G53 – Rapid to machine zero in Z, switch spindle off)
N110 G53 X-15.0 Y0	(G53 – Rapid in relation to machine zero X-15.0 and Y0)
(DRILL 0.25” HOLES)	
N120 T07 M06	(T7-Select tool number 7 to be loaded M06-Activates the tool changer)
N130 G00 G90 G54 X-1.0 Y-0.875 S4500 M03	(Rapid to the X and Y position of Hole #1 and turn on the spindle at 4500 RPM)
N140 G43 H07 Z0.1	(G43 - Activate the tool offset value stored in H07 and rapid to Z0.1)

LESSON-4 - CNC - PART #1 - Continued

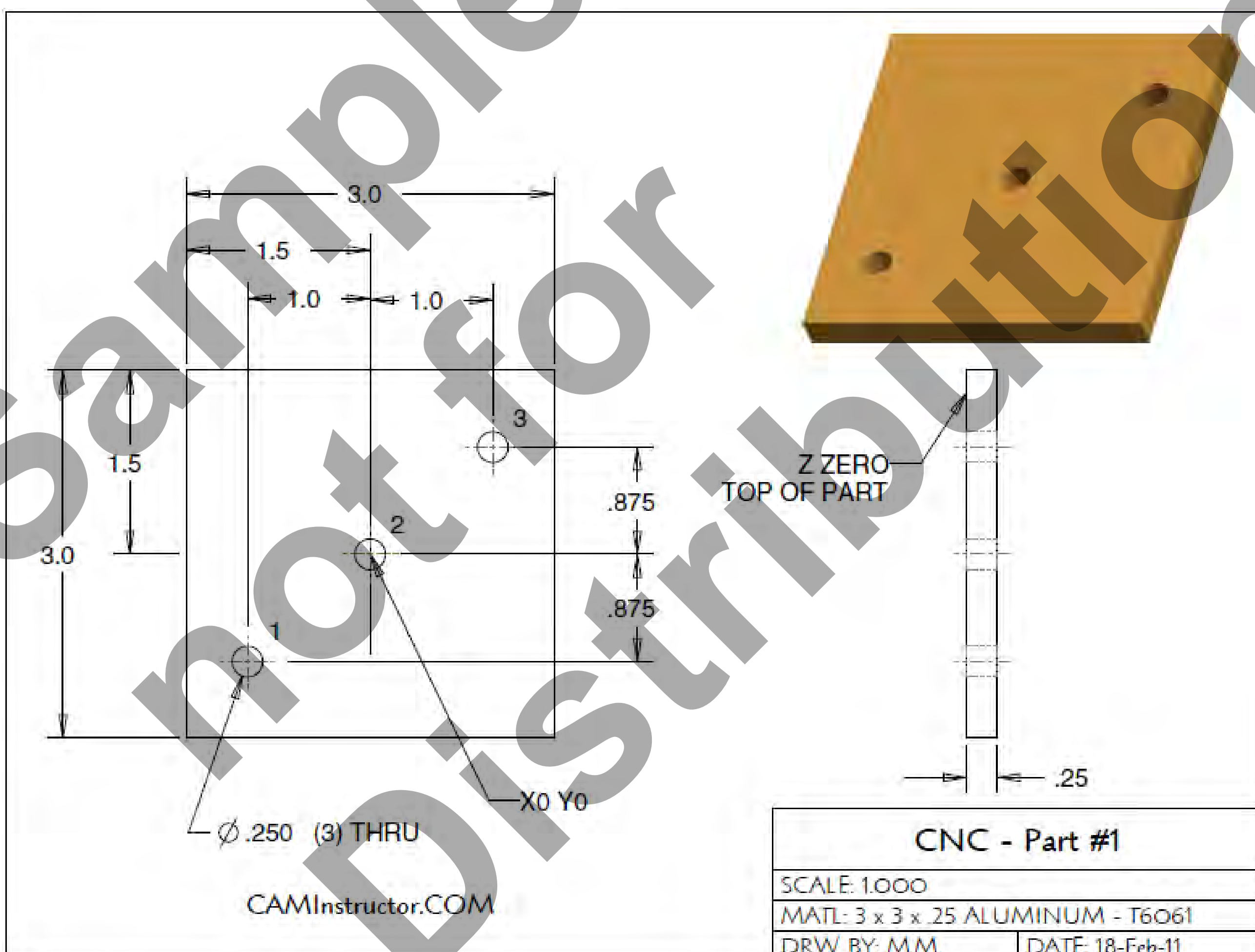
N150 G99 G81 Z-0.35 R0.1 F15.0	(Hole #1 – G81 - Feed down to Z depth at 15 inches per minute, and then retract at rapid to Z0.1, this is the R0.1 value. G99 returns the drill tip to the R value after drilling each hole)
N160 X0. Y0.	(Hole #2 - Move at rapid in the X and Y axis to Hole #2. Feed down to Z depth at 15 inches per minute and then retract at rapid to Z0.1)
N170 X1.0 Y.875	(Hole #3 - Move at rapid in the X and Y axis to Hole #3. Feed down to Z depth at 15 inches per minute and then retract at rapid to Z0.1)
N180 G80	(Cancel Canned Cycle Drill)
N190 G53 G00 Z0 M05	(G53 – Rapid to machine zero in Z, switch spindle off)
N200 G53 X-15.0 Y0	(G53 – Rapid in relation to machine zero X-15.0 and Y0)
N210 M30	(Program end rewind program to the beginning)
%	(Program must begin and end with a %)



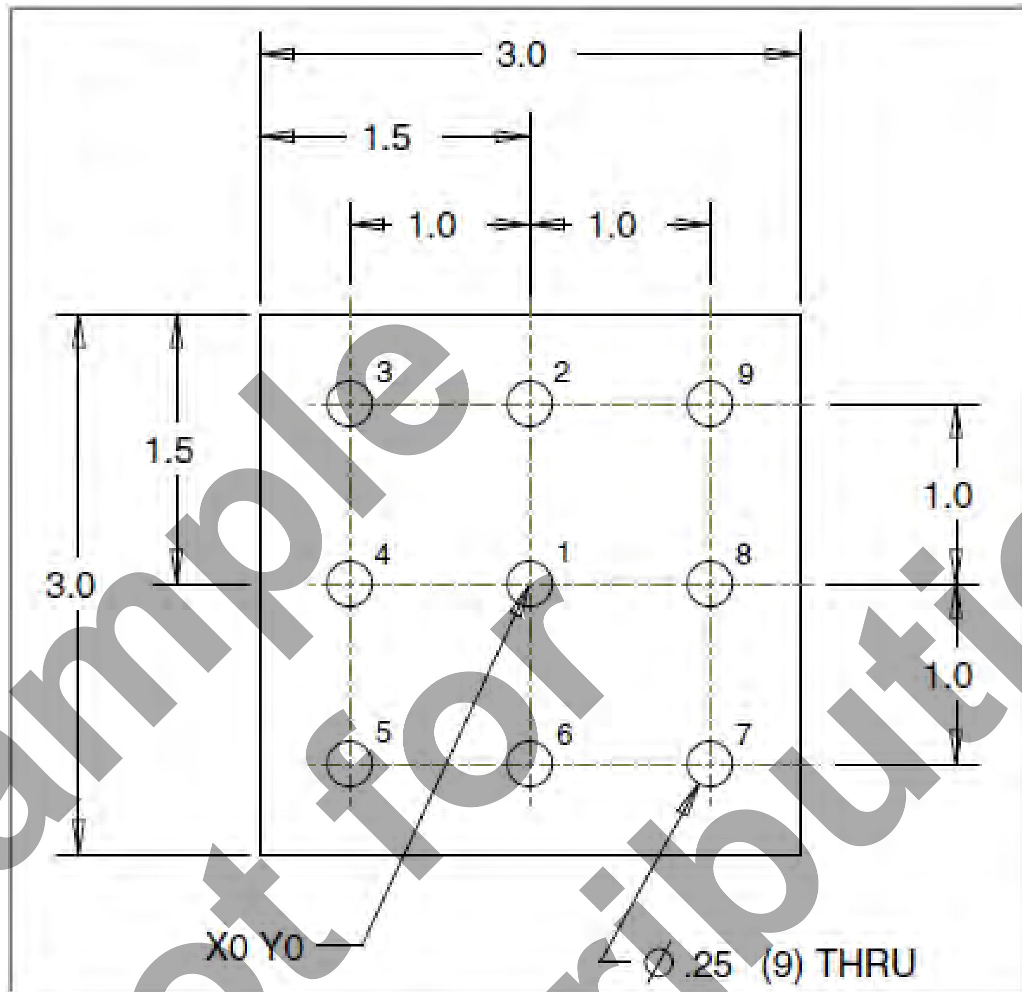
LESSON-4 – WHAT COULD GO WRONG?

- ***IDENTIFY SOME OF THE COMMON PROBLEMS THAT COULD RESULT IN A SCRAPPED PART***

- Do you have X0 Y0 Z0 in the correct position?
- Is the spindle switched on and off at the appropriate time
- Did you use the correct X and Y coordinates for the holes?
- Did you use the correct tool numbers?
- Did you use the correct tool length offset number (H??) for the tool?
- Did you cancel any canned cycles with G80?
- Are the feed-rates correct?
- Is the Z depth in the canned cycle block set to a negative value?
- Is the R value in the canned cycle block set to a positive value?
- What is the difference between Z2 and Z2.0? No decimal point???
- What is the difference between F10 and F10.0? No decimal point???
- What else?



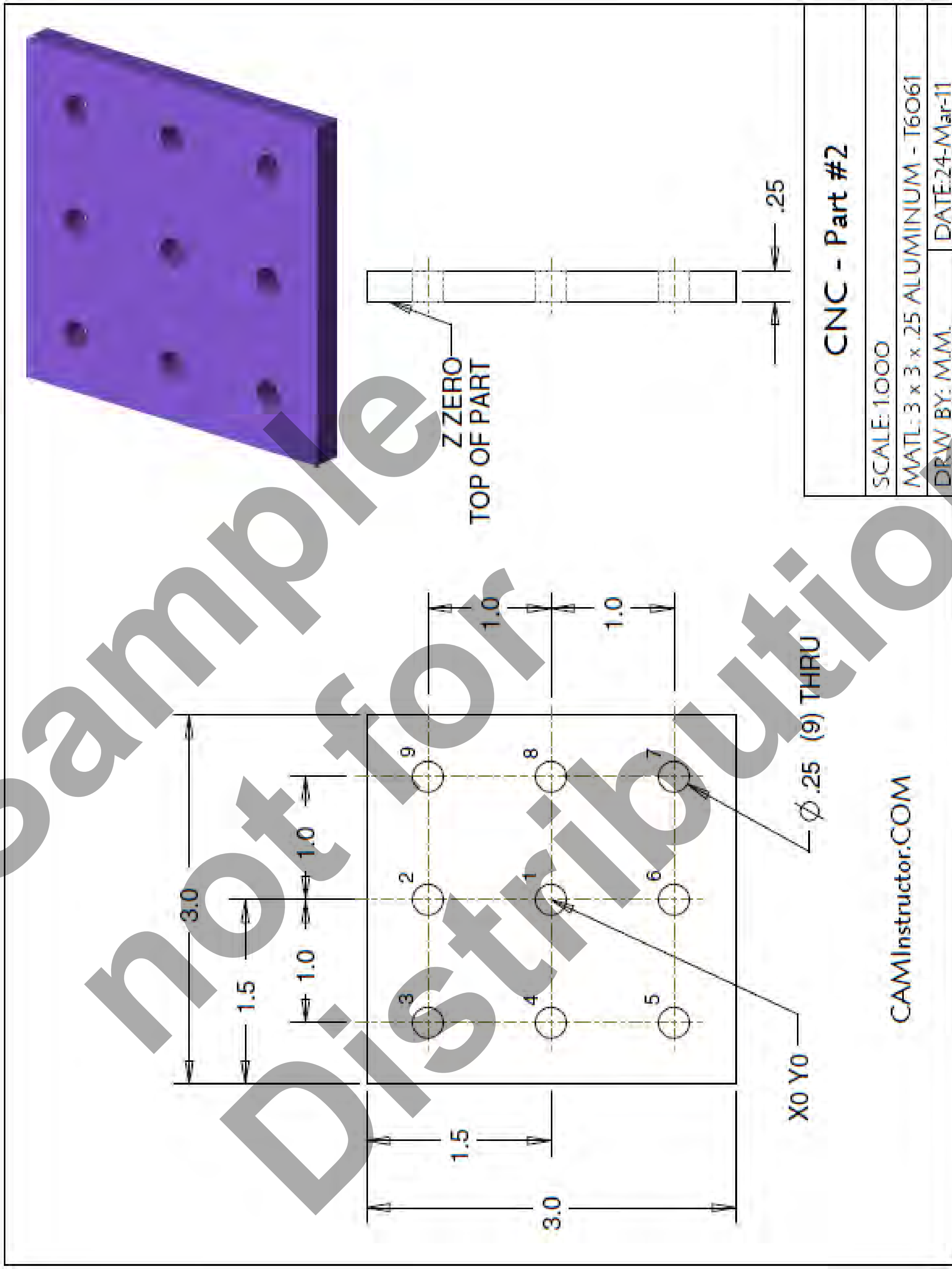
CNC PROGRAMMING WORKBOOK



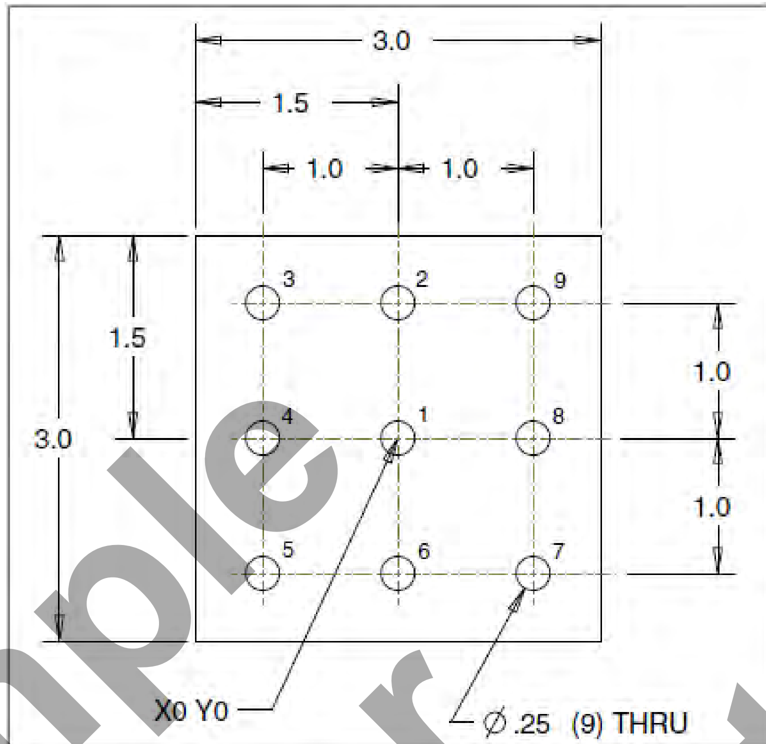
LESSON-5

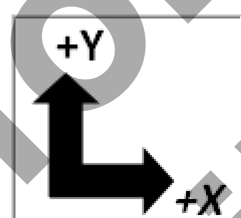
DRILLING USING CANNED CYCLES - CONTINUED

camInstructor


CNC - Part #2
SCALE: 1.000
MATL: 3 x 3 x .25 ALUMINUM - T6061
DRW BY: M.M.
DATE:24-Mar-11
Z ZERO
TOP OF PART
.25
Ø .25 (9) THRU
X0 Y0
CAMInstructor.COM
3.0
1.5
1.0
1.0
1.0
1.0
1.5
3.0
1 2 3 4 5 6 7 8 9

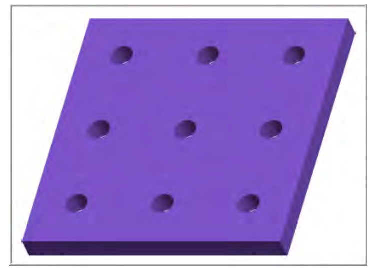
- ***WORK OUT THE ABSOLUTE COORDINATES FOR THE NINE HOLES***
  - X0Y0 is at the centre of the part





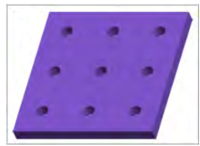
G90	X	Y
1		
2		
3		
4		
5		
6		
7		
8		
9		

- ***CREATE THE PROGRAM TO SPOT AND DRILL THE NINE HOLES USING CANNED CYCLE G81***
  - Use a 0.375” diameter Spot Drill Tool # 6
  - Spot Drill Spindle Speed = 2750 Feed rate = 11 IPM
  - Use a 0.250” diameter Drill Tool # 7
  - 0.250” diameter Drill Spindle Speed = 4500 Feed rate = 15 IPM
  - Spot Drill Depth = Z-0.150”
  - Drill Depth = Z-0.350”
  - X0Y0 is at the centre of the part
  - Z=0 is the top of the part.
  - Type up your program and check it for correctness using NCPlot



%
O4 (Program #4 - CNC-PART-2-SPOT AND DRILLING USING CANNED CYCLE DRILL G81)
N10 G20
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)
N30

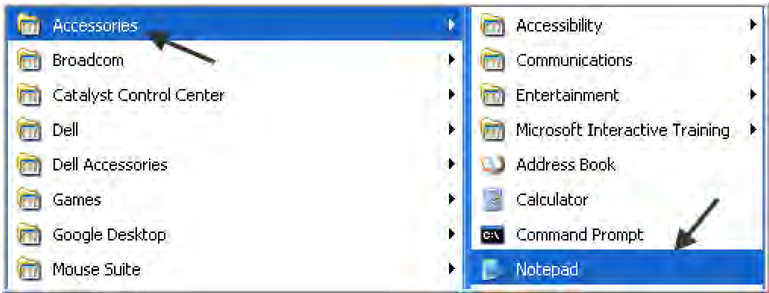
LESSON-5 - CNC - PART #2 - Continued



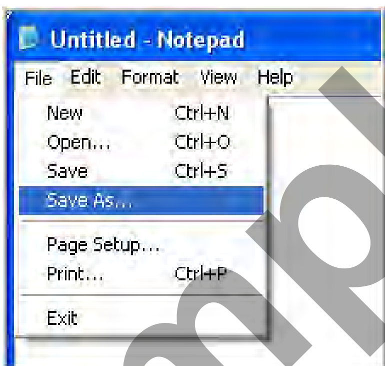
LESSON-5 - CNC - PART #2 – TYPING UP YOUR PROGRAM USING WINDOWS NOTEPAD

- ***Use Windows Notepad to type up your CNC program***

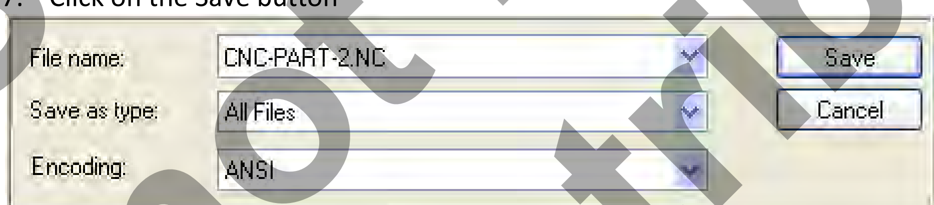
1. Launch Windows Notepad **Start>All Programs>Accessories>Notepad**.



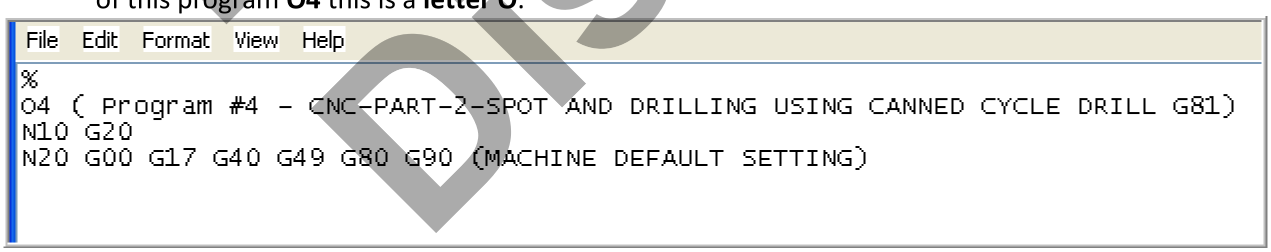
2. Select **File Save As...**



3. **Browse** to where you would like to save this file.
4. Open up the **Save as type** drop down and change to **All files**.
5. **Encoding** should be set to **ANSI**.
6. In the **File name** section enter **CNC-PART-2.NC** This will give this file an extension of **.NC**
7. Click on the Save button

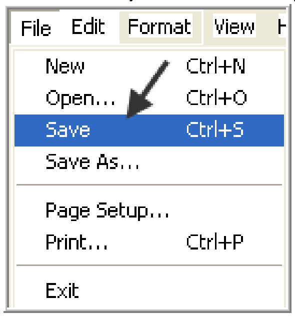


8. Start typing your program, **ALL CAPITALS** for the CNC program codes. Please note on the second line of this program **O4** this is a **letter O**.



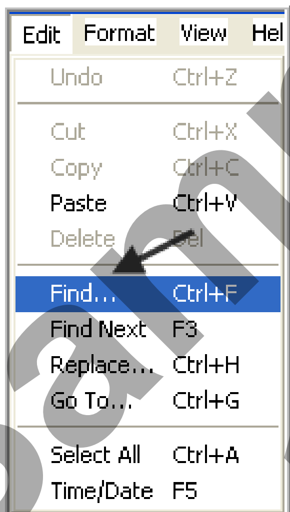
LESSON-5 - CNC - PART #2 – TYPING UP YOUR PROGRAM USING WINDOWS NOTEPAD

9. When you have completed typing your program Save your file, File>**Save** or the shortcut **Ctrl+S**.



- **Now you can check for any Letter O's in your CNC program. Please Note there should not be any letter O's in your CNC program, G00 is "G Zero Zero" not G Letter O!**

10. Select **Edit>Find**.

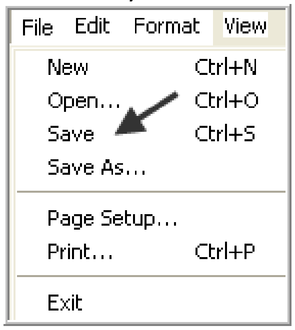


11. Type in the **Letter O** in the **Find what:** space. Now hit the Find Next button. There will be some letter O's in your program, for example the Letter O in the program number at the start of the program and any notes you have in your program enclosed by parenthesis ( ). But for the coding no letter O's.

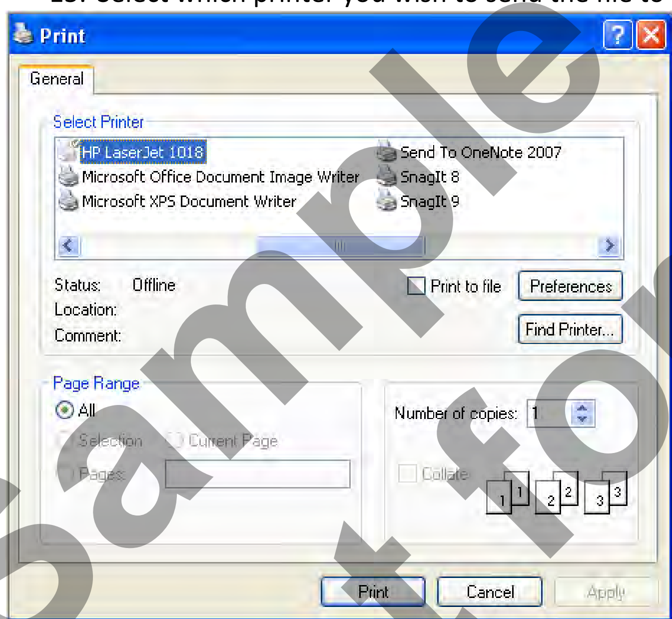
12. I f you do find any letter O's change them to a Zero.

LESSON-5 - CNC - PART #2 – TYPING UP YOUR PROGRAM USING WINDOWS NOTEPAD

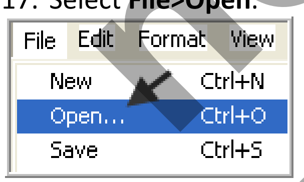
13. When you have checked your program select **File>Save** your file or the shortcut **Ctrl+S**.



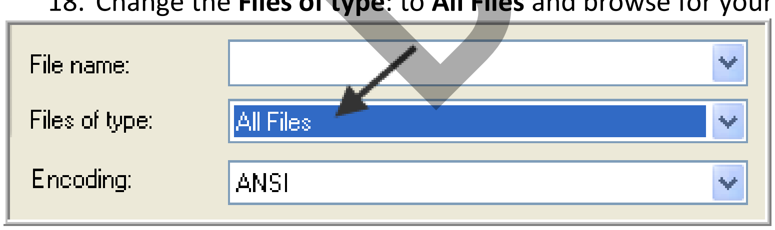
14. You may require a print of your CNC program to do this select **File>Print** or the shortcut **Ctrl+P**.
15. Select which printer you wish to send the file to and then hit the **Print** button



16. To open your CNC program at a later date launch Windows Notepad. **Start>All Programs>Accessories>Notepad**.
17. Select **File>Open**.



18. Change the **Files of type**: to **All Files** and browse for your CNC program.



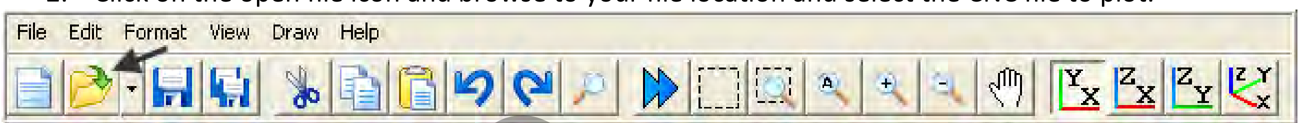
LESSON-5 - CNC - PART #2 - BACKPLOTTING

- ***Use NCPlot to check for correctness***

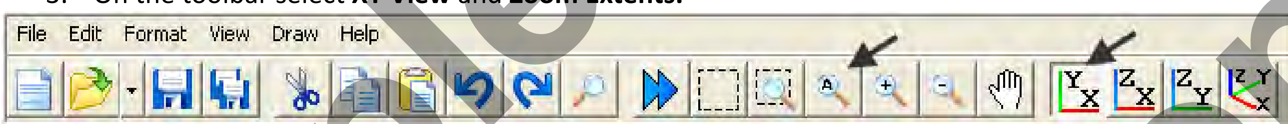
Note: If you do not have NCPlot installed on your computer please go to the last page of the appendix for installation instructions.

1. After typing up your program in Notepad launch the NCPlot application by clicking on the icon on your desktop or **Start>All Programs>NCPlot v1.3> NCPlot v1.3**

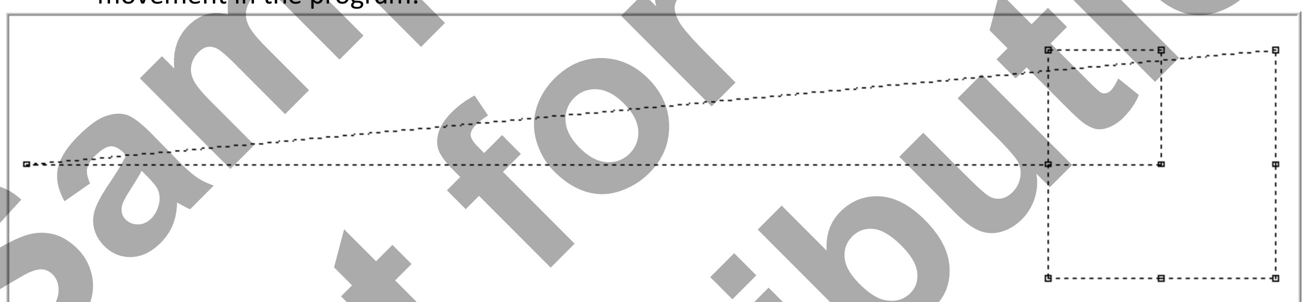
2. Click on the open file icon and browse to your file location and select the CNC file to plot.



3. On the toolbar select **XY View** and **Zoom Extents.**



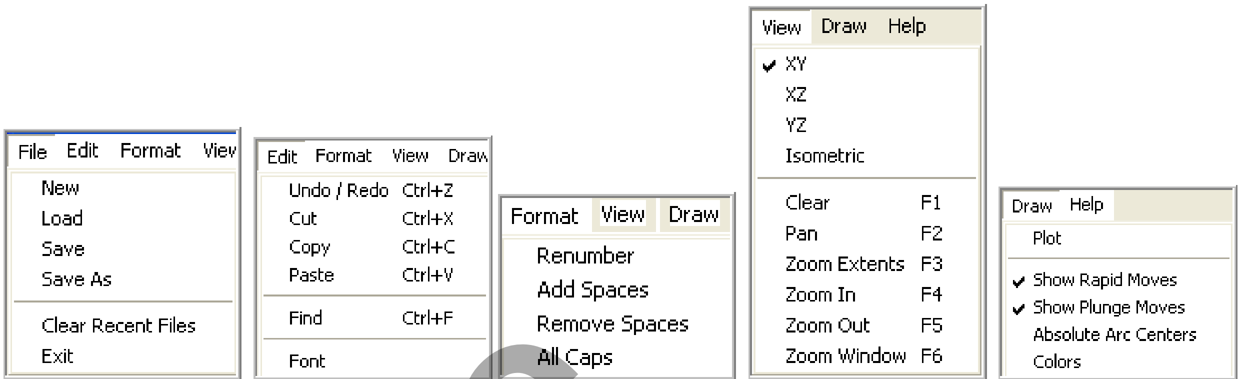
4. You plot should appear as below. The point on the left of the screen shot below is the G53 X-15.0 Y0 movement in the program.



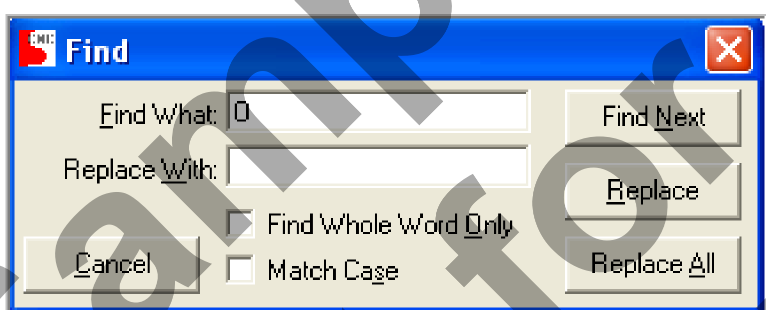
5. On the toolbar select **ZX View** and **Zoom Extents.**

6. On the toolbar select **XY View .**

7. Review and experiment with the various pull down menu options:



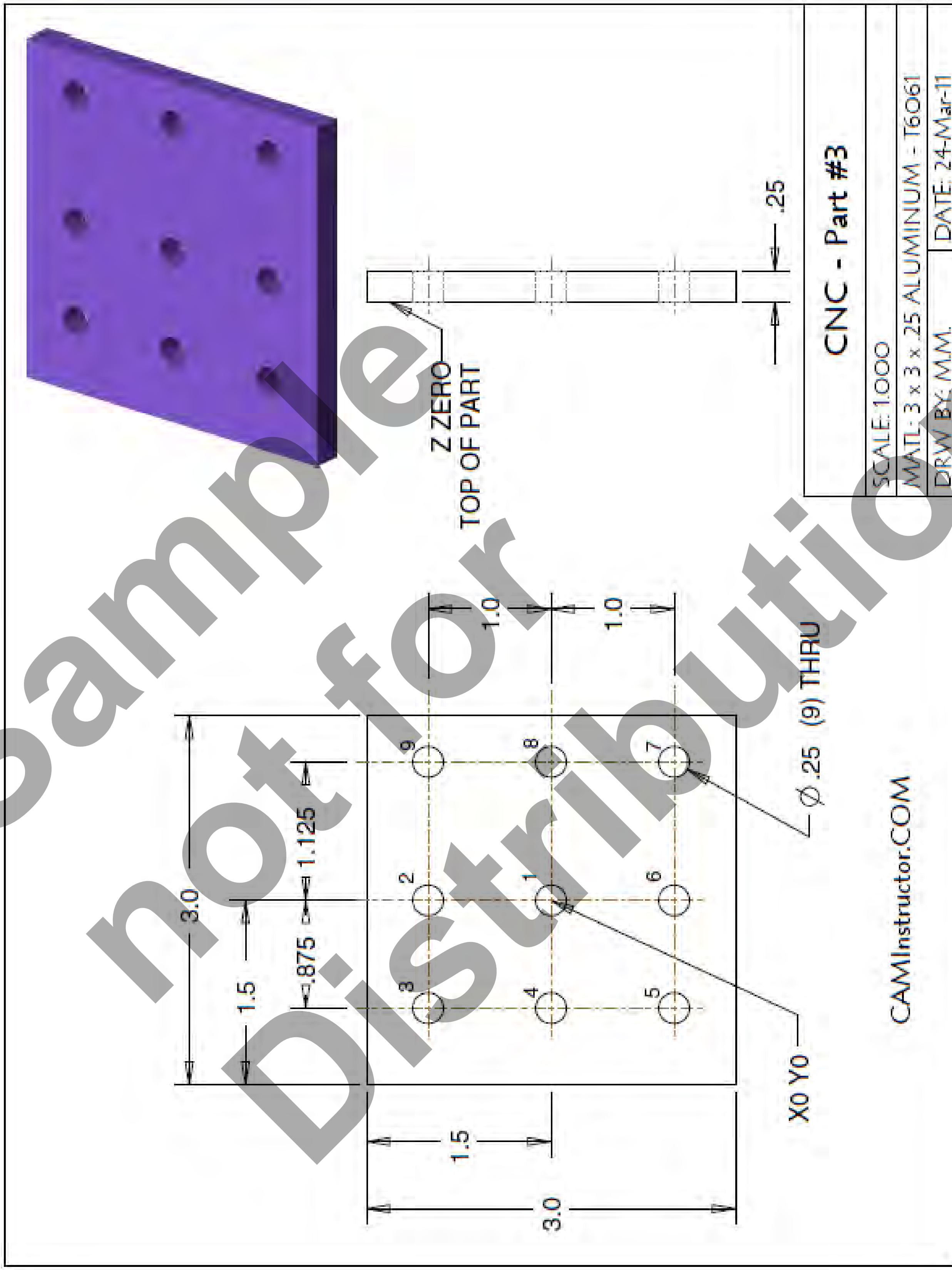
8. Press the function **F6** on your keyboard to action a Zoom Window. Zoom in on part of your plot.
9. Select **Ctrl+F** on your keyboard to open the **Find** dialog box. In the Find What section type in the **letter O** and then hit the **Find Next** button to see if there are any letter O's in the G and M codes.



10. You can use NCPlot instead of Windows Notepad to type up your CNC program.

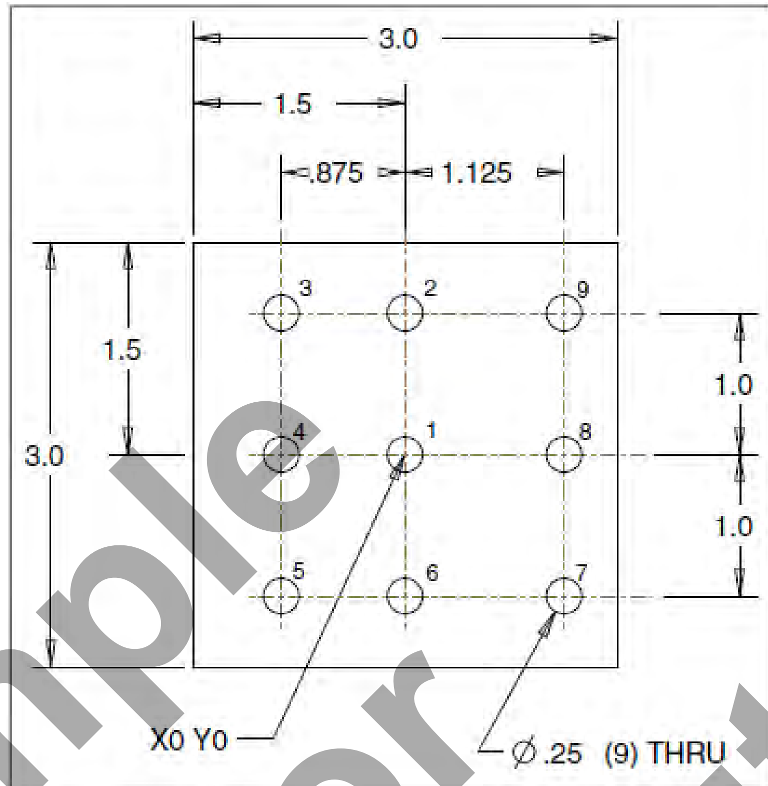
LESSON-5 - CNC - PART #3

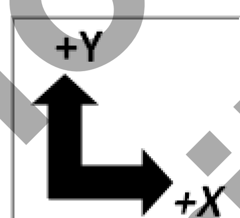
- ***PART #3 IS SIMILAR TO PART #2 BUT HAS DIFFERENT DIMENSIONS FOR THE HOLE CENTRES***



LESSON-5 - CNC - PART #3

- ***WORK OUT THE ABSOLUTE COORDINATES FOR THE NINE HOLES***
  - X0Y0 is at the centre of the part

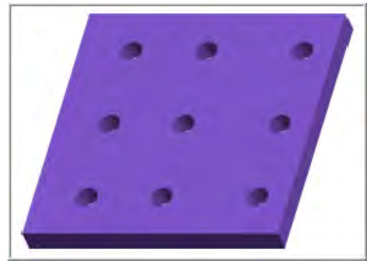




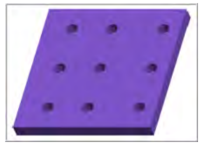
G90	X	Y
1		
2		
3		
4		
5		
6		
7		
8		
9		

LESSON-5 - CNC - PART #3

- ***CREATE THE PROGRAM TO <u>SPOT AND DRILL THE NINE HOLES</u> USING CANNED CYCLE G81***
  - Use a 0.375” diameter Spot Drill Tool # 6
  - Spot Drill Spindle Speed = 2750 Feed rate = 11 IPM
  - Use a 0.250” diameter Drill Tool # 7
  - 0.250” diameter Drill Spindle Speed = 4500 Feed rate = 15 IPM
  - Spot Drill Depth = Z-0.150”
  - Drill Depth = Z-0.350”
  - X0Y0 is at the centre of the part
  - Z=0 is the top of the part.
  - Type up your program and check it for correctness using NCPlot



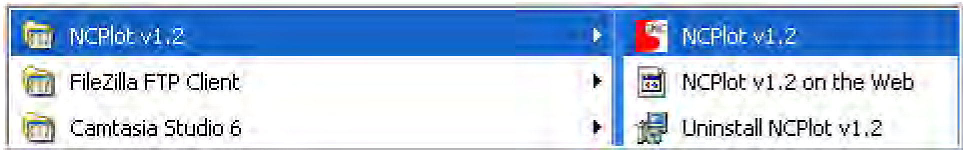
%
O5 (Program #5 - CNC-PART-3 - SPOT AND DRILLING USING CANNED CYCLE DRILL G81)
N10 G20
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)
N30



LESSON-5 - CNC - PART #3 - BACKPLOTTING

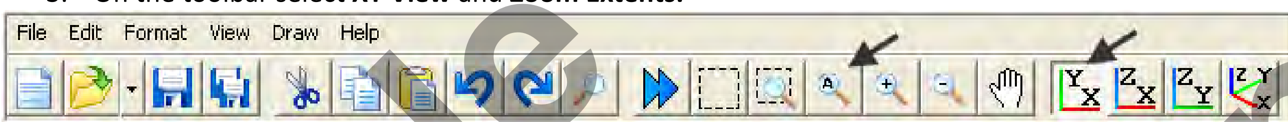
- ***Use NCPlot to check for correctness***

1. After typing up your program in Notepad launch the NCPlot application by clicking on the icon on your desktop or **Start>All Programs>NCPlot v1.3> NCPlot v1.3**

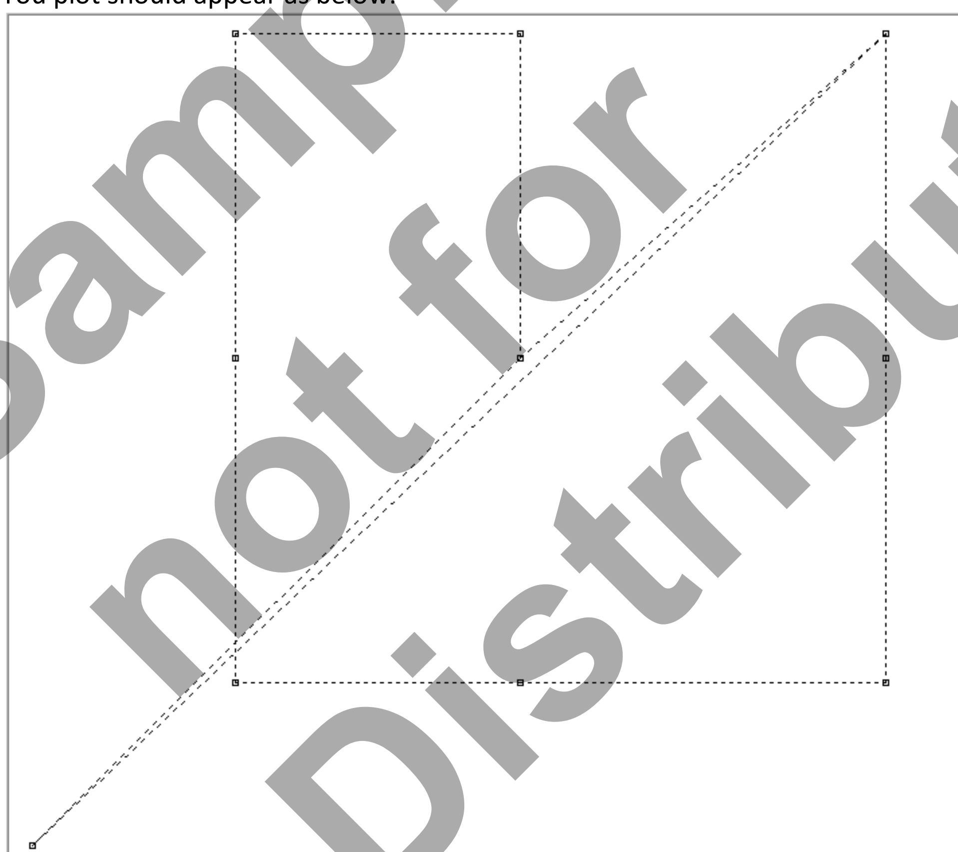


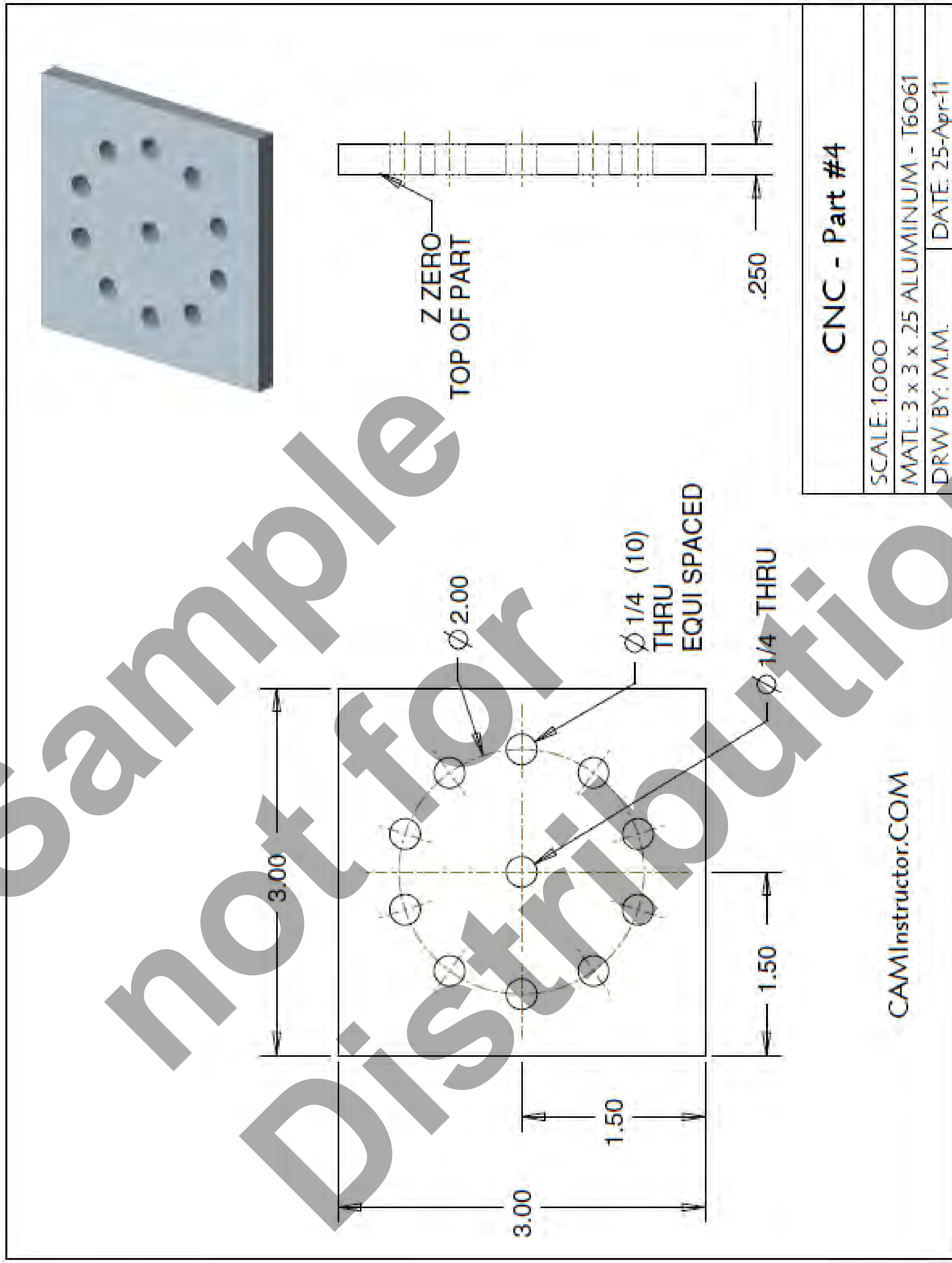
2. Click on the open file icon and browse to your file location and select the CNC file to plot.

3. On the toolbar select **XY View** and **Zoom Extents.**



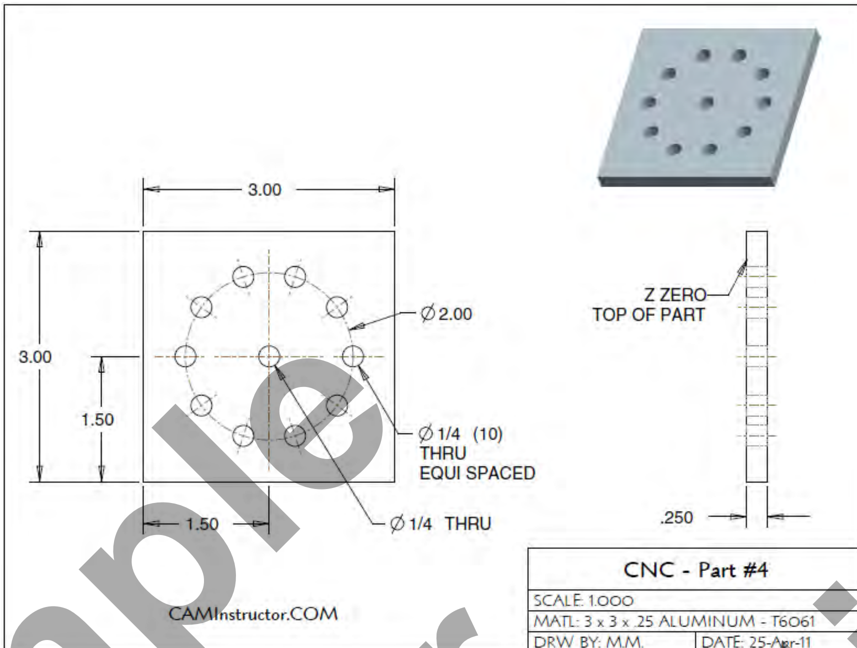
4. You plot should appear as below.

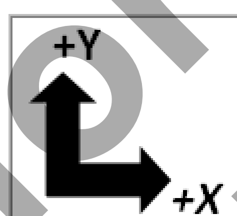



Z ZERO
TOP OF PART
.250
⌀2.00
⌀1/4 (10)
THRU
EQUI SPACED
⌀1/4 THRU
3.00
1.50
1.50
3.00
CAMInstructor.COM
CNC - Part #4
SCALE: 1.000
MATL: 3 x 3 x .25 ALUMINUM - T6061
DRW BY: M.M.
DATE: 25-Apr-11

LESSON-5 - CNC - PART #4

- ***WORK OUT THE ABSOLUTE COORDINATES FOR THE ELEVEN HOLES***
  - X0Y0 is at the centre of the part
  - Use Trigonometry to work out the center positions of the holes or draw the part up on a CAD system and then identify the center positions of each hole.





G90	X	Y
1	0	0
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		

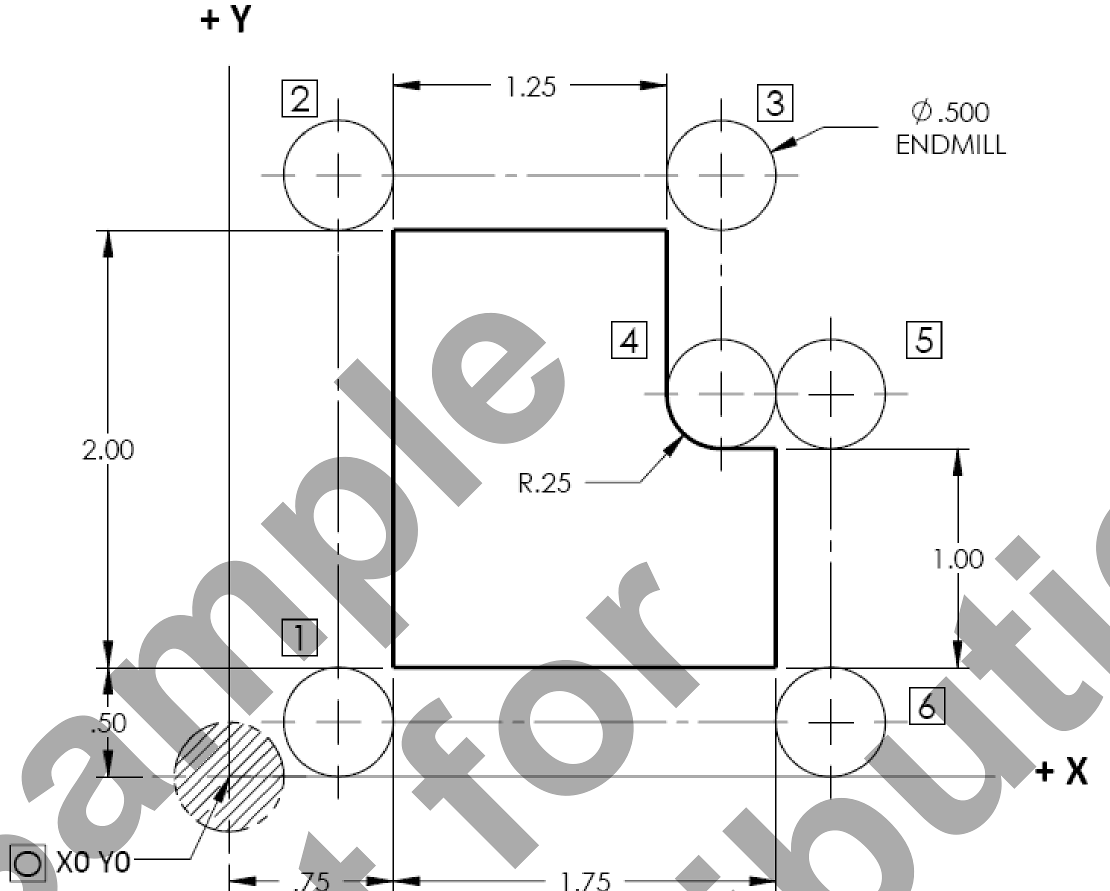
LESSON-5 - CNC - PART #4

- ***CREATE THE PROGRAM TO <u>SPOT AND DRILL THE ELEVEN HOLES</u> USING CANNED CYCLE G81***
  - Use a 0.375” diameter Spot Drill Tool # 6
  - Spot Drill Spindle Speed = 2750 Feed rate = 11 IPM
  - Use a 0.250” diameter Drill Tool # 7
  - 0.250” diameter Drill Spindle Speed = 4500 Feed rate = 15 IPM
  - Spot Drill Depth = Z-0.150”
  - Drill Depth = Z-0.350”
  - X0Y0 is at the centre of the part
  - Z=0 is the top of the part.
  - Type up your program and check it for correctness using NCPlot

%
O99 (CNC-PART-4 - SPOT AND DRILLING USING CANNED CYCLE DRILL G81)
N10 G20
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)
N30

LESSON-5 - CNC - PART #4 - Continued

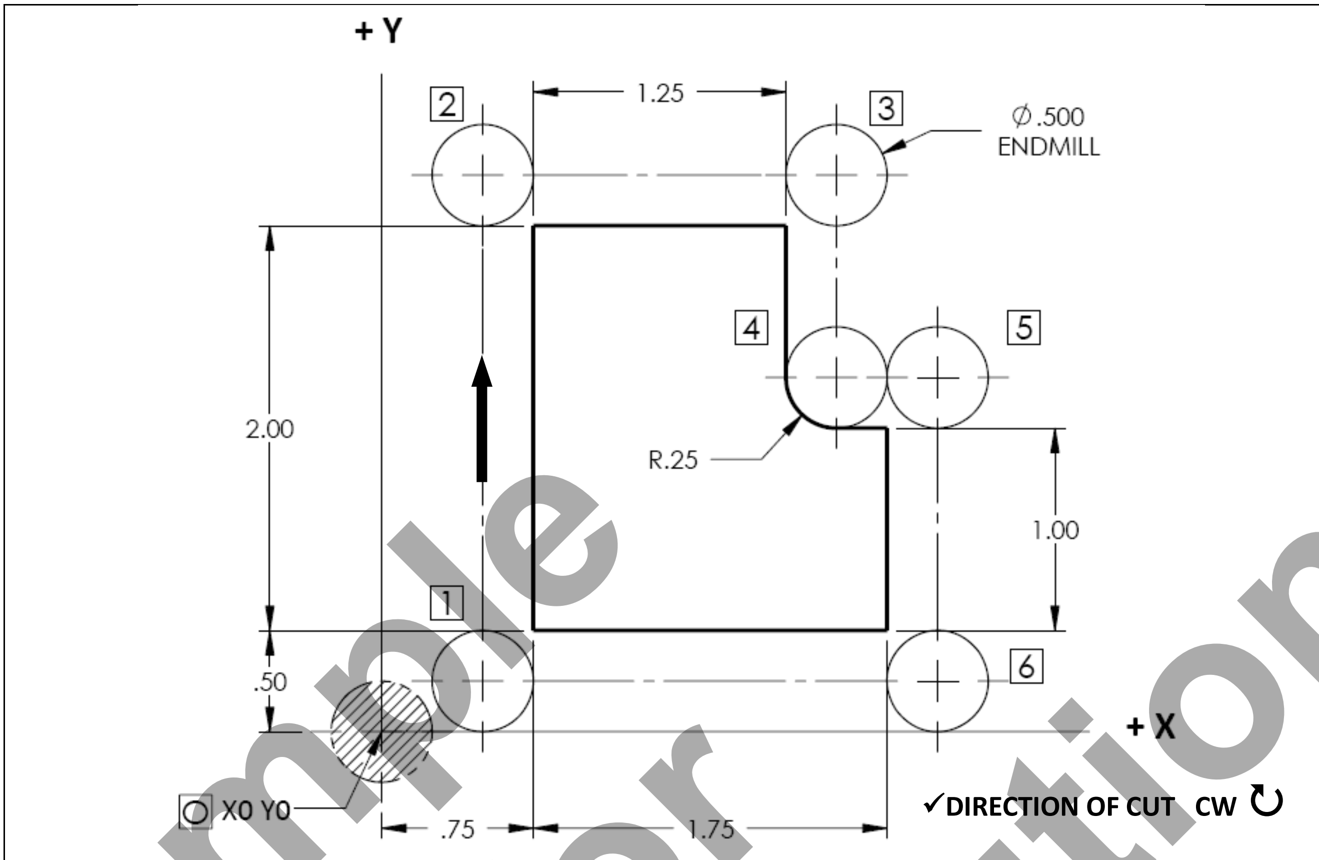
CNC PROGRAMMING WORKBOOK



LESSON-6

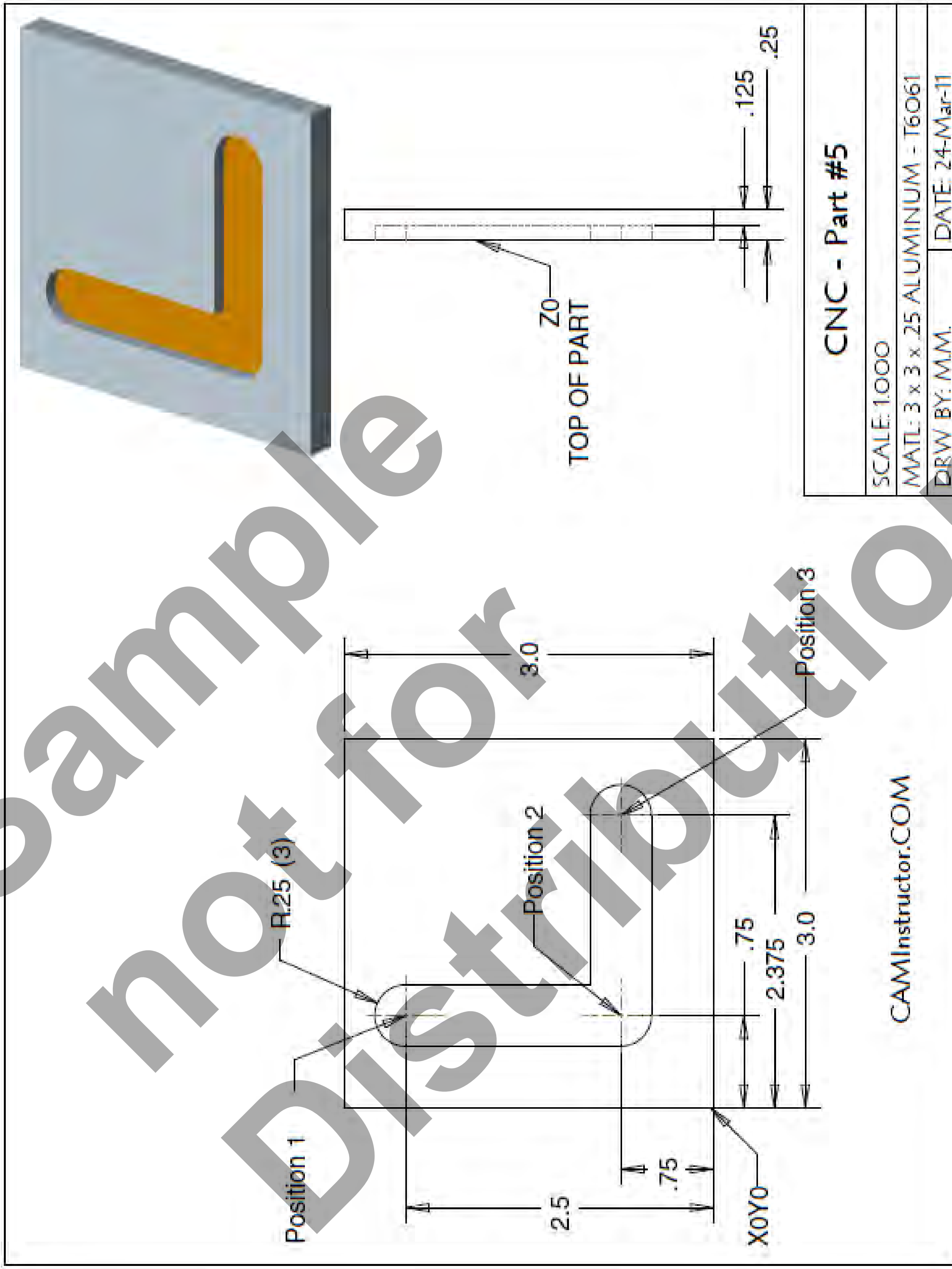
STRAIGHT LINE MILLING – LINEAR INTERPOLATION

LESSON-6 – STRAIGHT LINE MILLING
LESSON-6 – EXERCISE #1 - ABSOLUTE & INCREMENTAL POSITIONING



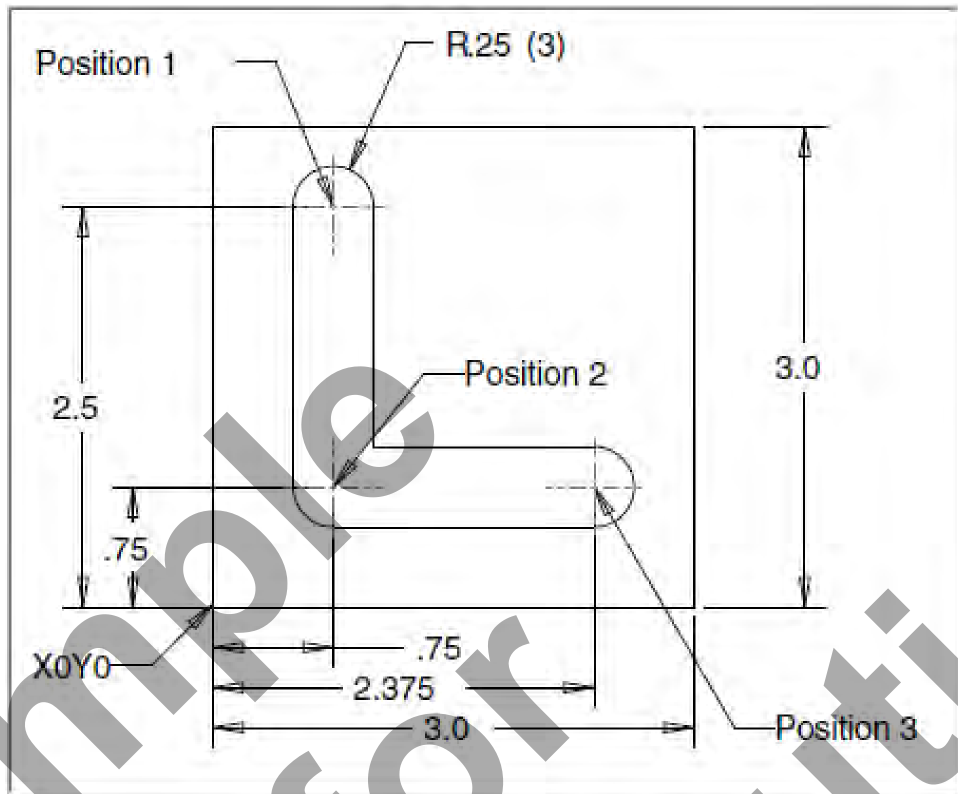
STARTING AT THE POINT O (ORIGIN), DESCRIBE THE ENDMILL PATH FROM O THROUGH ALL THE POINTS AND BACK TO THE POINT O USING G90 & G91. CUTTER DIAMETER = 0.5" RADIUS = 0.25'

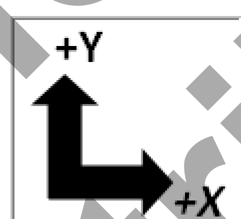
G90	X	Y	G91	X	Y
O	0	0	O → 1	0.5	0.25
1	0.5	0.25	1 → 2	0	2.5
2	0.5	2.75	2 → 3		
3			3 → 4		
4			4 → 5		
5			5 → 6		
6			6 → 1		
1			1 → O		
O					


CNC - Part #5
SCALE: 1.000
MATL: 3 x 3 x .25 ALUMINUM - T6061
DRW BY: M.M.
DATE: 24-Mar-11
Z0
TOP OF PART
.125
.25
3.0
3.0
2.375
.75
2.5
.75
R.25 (3)
Position 1
Position 2
Position 3
X0Y0
CAMInstructor.COM

LESSON-6 - CNC - PART #5

- ***WORK OUT THE ABSOLUTE COORDINATES FOR POSITION 1, 2 AND 3***
  - X0Y0 is at the lower left corner of the part
  - These X and Y coordinates will be used to machine the L shaped slot

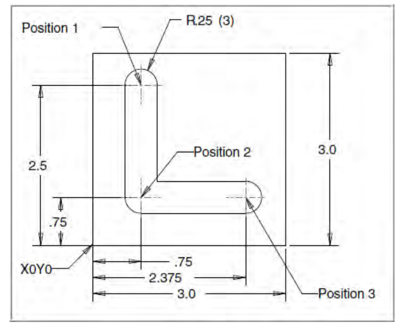




G90	X	Y
1		
2		
3		

LESSON-6 - CNC - PART #5

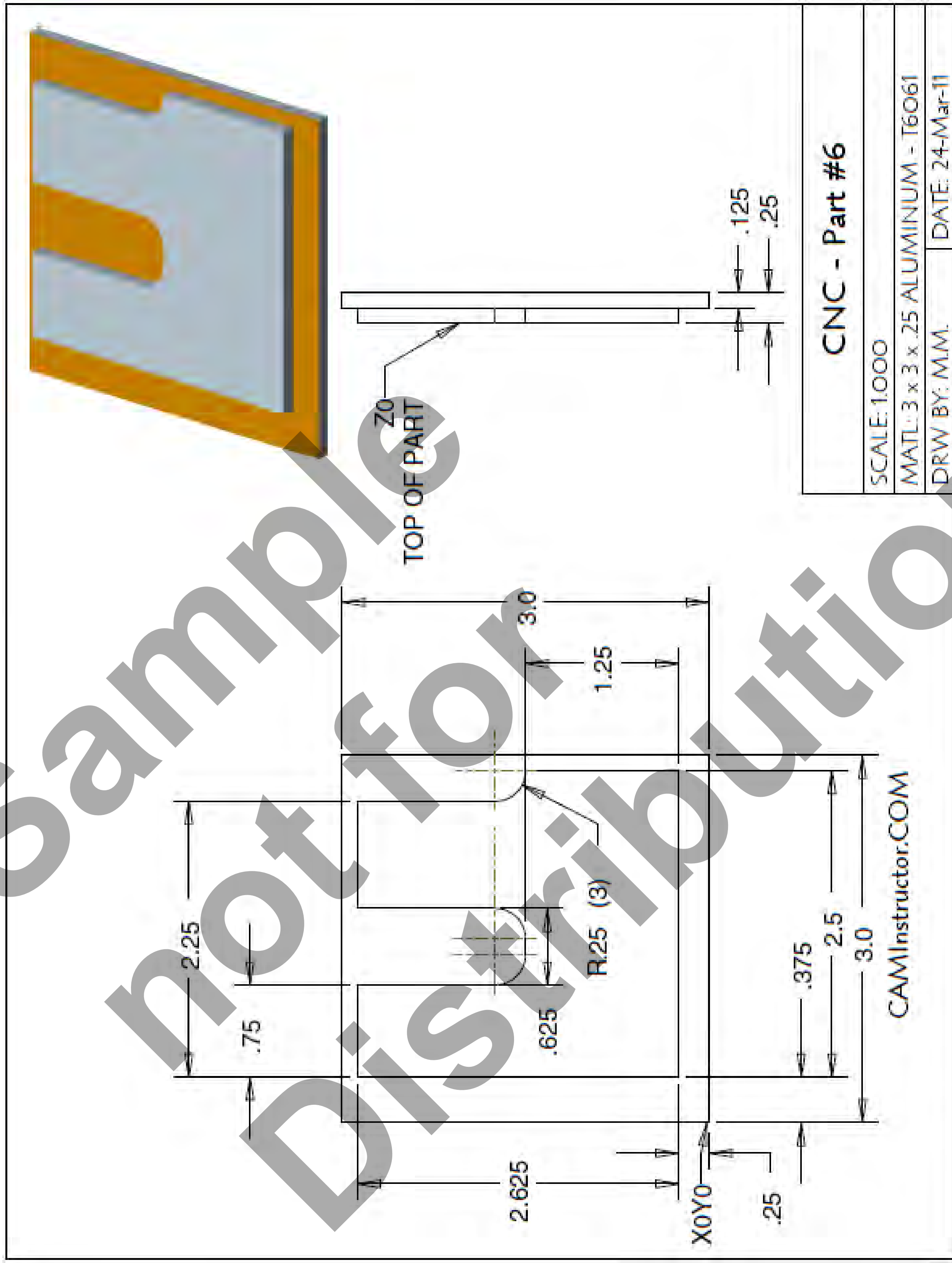
- ***PROGRAM TO MACHINE THE "L SHAPED" SLOT***
  - Use a 0.5" diameter End Mill Tool # 4
  - Speed = 3050 Feed rate =20 IPM
  - X0Y0 is at the lower left corner of the part
  - Z=0 is the top of the part.
  - The slot depth is 0.125"
  - Enter the part at Position 1 and sink to depth using linear interpolation G01
  - Then move to Position 2 and finally Position 3



%	(Program must begin and end with a %)
O6	(Program #6 – Part #5 – STRAIGHT LINE MILLING)
(T4 - 1/2 FLAT ENDMILL - H4)	
N10 G20	(**Inch programming)**
N20 G00 G17 G40 G49 G80 G90	(MACHINE DEFAULT SETTING)
N30 T4 M6	(T4-Select tool number 4 to be loaded M06-Activates the tool changer)
N40 G00 G90 G54 X0.75 Y2.5 S3050 M3	(Rapid to the X and Y to Position #1 and turn on the spindle at 3050 RPM)
N50 G43 H04 Z0.1	(G43 - Activate the tool offset value stored in H04 and rapid to Z0.1)
N70 G1 Z-0.125 F10.	(Position #1 - Feed down to Z depth at 10 inches per minute)
N80 Y0.75 F20.0	(Move to Position #2 - at 20 inches per minute)
N90 X2.375	(Move to Position #3 - at 20 inches per minute)
N100 Z0.1 F10.0	(Retract out of the part at feedrate to 0.1 above the top of the work piece)
N120 M05	(Spindle off)
N130 G00 G91 G28 Z0	(G28 – Machine Zero positioning. Rapid to machine zero in Z)
N140 G28 X0 Y0	(G28 – Rapid in relation to machine zero X0 and Y0)
N150 G90	(RETURN TO ABSOLUTE PROGRAMMING)
N160 M30	(Program end rewind program to the beginning)
%	(Program must begin and end with a %)

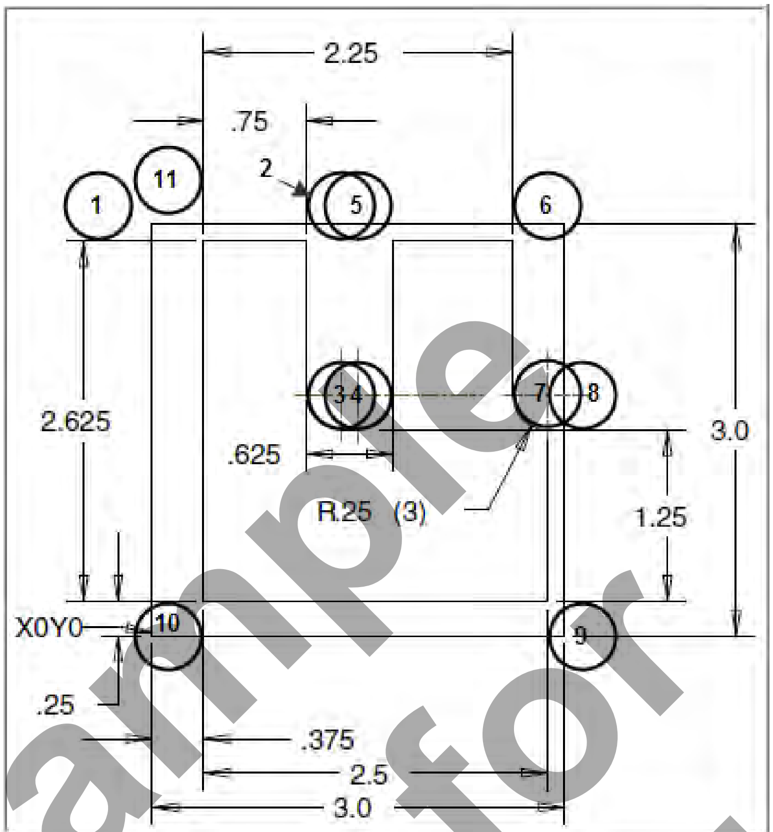
In earlier programs we used **G53** to return the machine to coordinates in relation to the Machine Zero (home position). **G28** is another code that will accomplish this. **G28** is a more common way to send the machine to machine zero, it will work on many different types of CNC machines.

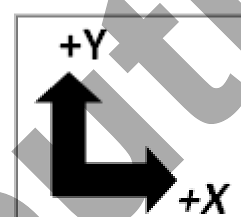
As you can see above at block **N130**, G28 is activated in **G91 incremental mode**, and then at block N150 the program is returned to **G90 absolute mode**.


CNC - Part #6
SCALE: 1.000
MATL: 3 x 3 x .25 ALUMINUM - T6061
DRW BY: M.M.
DATE: 24-Mar-11
CAMInstructor.COM
Z0
TOP OF PART
.125
.25
3.0
1.25
2.25
.75
.625
R.25 (3)
.375
2.5
3.0
2.625
X0Y0
.25

LESSON-6 - CNC - PART #6

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE CONTOUR***
  - X0Y0 is at the lower left corner of the part
  - Use a 0.5” diameter End Mill Tool # 4
  - Start at X-0.5 Y3.125 and machine the contour in a clockwise direction – climb milling



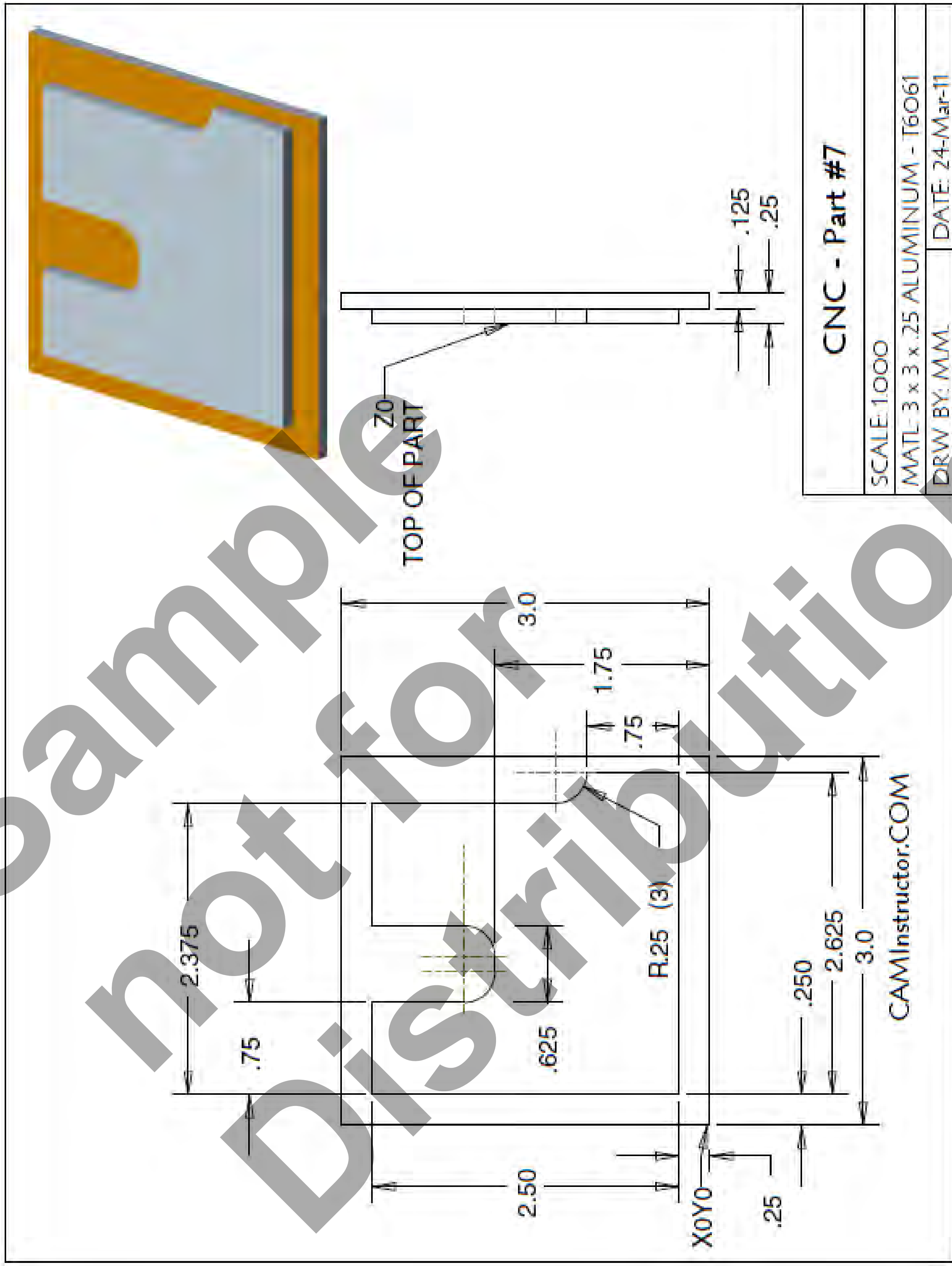


G90	X	Y
1	-0.5	3.125
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		

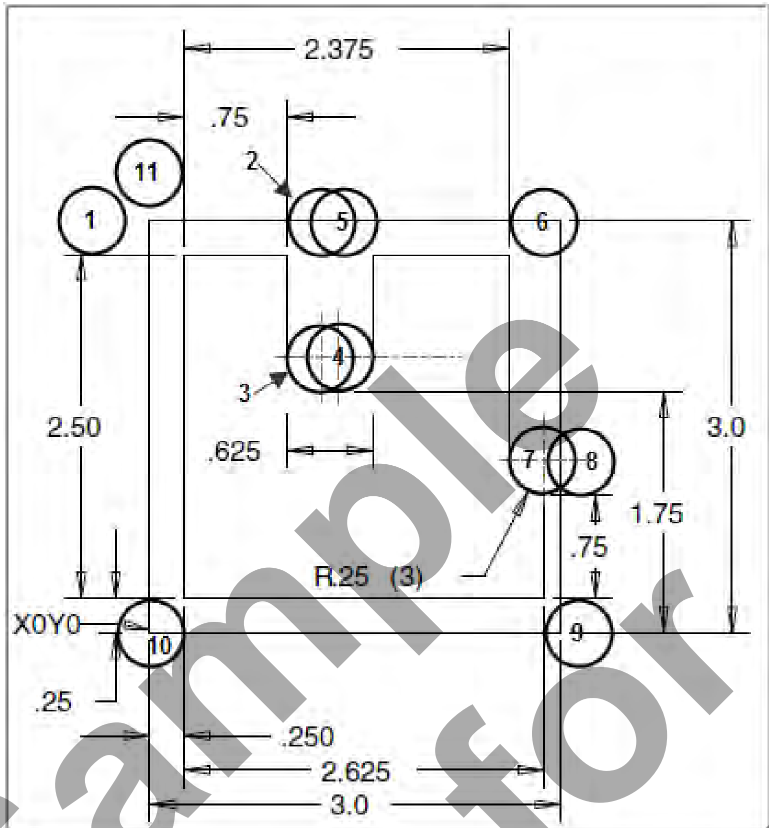
LESSON-6 - CNC - PART #6

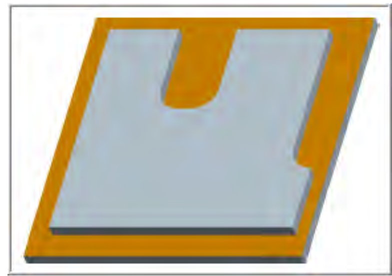
- ***PROGRAM TO MACHINE THE CONTOUR***
  - Use a 0.5” diameter End Mill Tool # 4
  - Speed = 3050 Feed rate =20 IPM
  - X0Y0 is at the lower left corner of the part
  - Z=0 is the top of the part.
  - Machine the contour at a depth is 0.125”

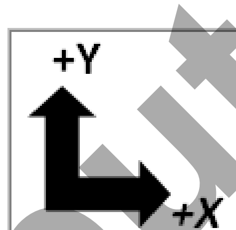
%	(Program must begin and end with a %)
O8	(Program #8 - CNC-PART-6 - STRAIGHT LINE MILLING)
(T4 - 1/2 FLAT ENDMILL - H4)	
N10 G20	(**Inch programming)**
N20 G00 G17 G40 G49 G80 G90	(MACHINE DEFAULT SETTING)
N30 T4 M6	(T4-Select tool number 4 to be loaded M06-Activates the tool changer)
N40 G00 G90 G54 X-0.5 Y3. 125 S3050 M3	(Rapid to the X and Y to start Position #1 and turn on the spindle at 3050 RPM)
N50 G43 H04 Z0.1	(G43 - Activate the tool offset value stored in H04 and rapid to Z0.1)
N60 G1 Z-0.125 F10.	(Position #1 - Feed down to Z depth at 10 inches per minute)
N70 X1.375 F20.0	(Move to Position #2 - at 20 inches per minute)
N80 Y1.75	(Move to Position #3)
N90 X1.5	(Move to Position #4)
N100 Y3.125	(Move to Position #5)
N110 X2.875	(Move to Position #6)
N120 Y1.75	(Move to Position #7)
N130 X3.125	(Move to Position #8)
N140 Y0	(Move to Position #9)
N150 X0.125	(Move to Position #10)
N160 Y3.375	(Move to Position #11)
N170 Z0.1 F10.0	(Retract out of the part at feedrate to 0.1 above the top of the work piece)
N180 M05	(Spindle off)
N190 G00 G91 G28 Z0	(G28 – Machine Zero positioning. Rapid to machine zero in Z)
N200 G28 X0 Y0	(G28 – Rapid in relation to machine zero X0 and Y0)
N210 G90	(RETURN TO ABSOLUTE PROGRAMMING)
N220 M30	(Program end rewind program to the beginning)
%	(Program must begin and end with a %)


Z0
TOP OF PART
.125
.25
3.0
1.75
.75
2.375
.75
.625
R.25 (3)
2.50
.250
2.625
3.0
X0Y0
.25
CAMInstructor.COM
CNC - Part #7
SCALE: 1.000
MATL- 3 x 3 x .25 ALUMINUM - T6061
DRW BY: M.M.
DATE: 24-Mar-11

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE CONTOUR***
  - X0Y0 is at the lower left corner of the part
  - Use a 0.5" diameter End Mill Tool # 4
  - Start at X-0.5 Y3.0 and machine the contour in a clockwise direction – climb milling



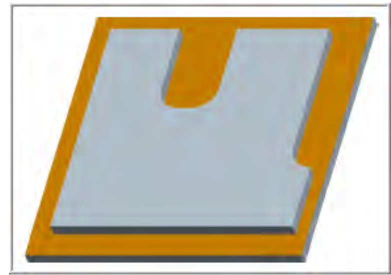




G90	X	Y
1	-0.5	3.0
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		

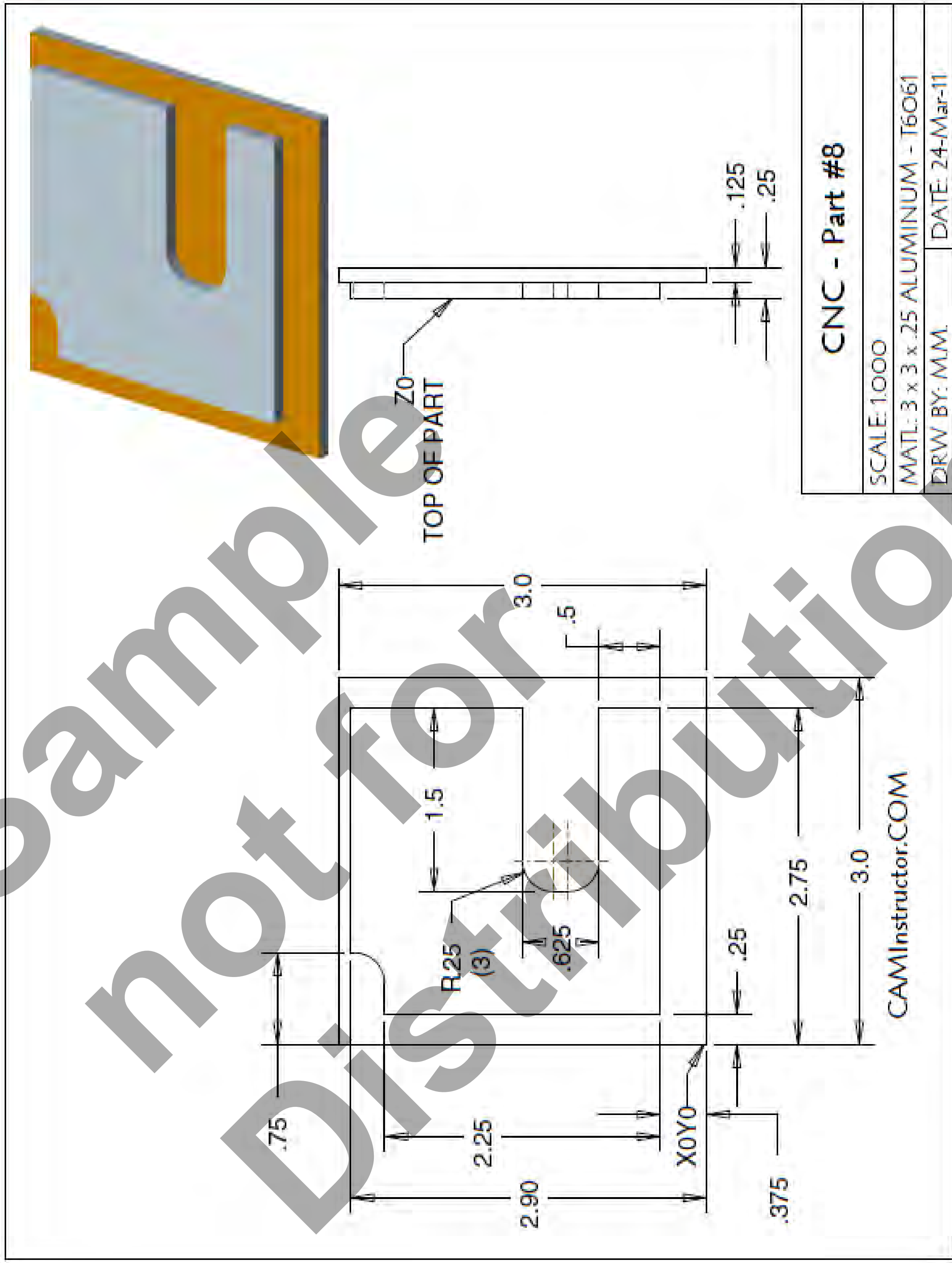
LESSON-6 - CNC - PART #7

- ***CREATE THE PROGRAM TO MACHINE THE CONTOUR***
  - Use a 0.5” diameter End Mill Tool # 4
  - Speed = 3050 Feed rate =20 IPM
  - X0Y0 is at the lower left corner of the part
  - Z=0 is the top of the part.
  - Machine the contour at a depth is 0.125”
  - Start at X-0.5 Y3.0 and machine the contour in a clockwise direction – climb milling
  - Type up your program and check it for correctness using NCPlot



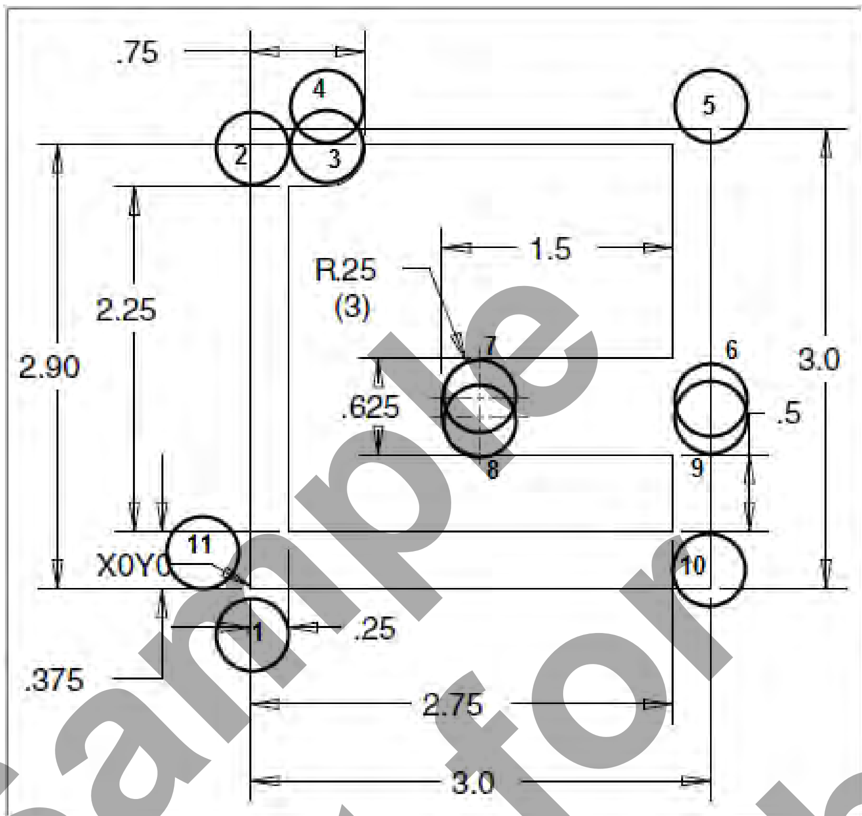
%
O009 (Program #9 - CNC-PART-7 - STRAIGHT LINE MILLING)
N10 G20
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)
N30

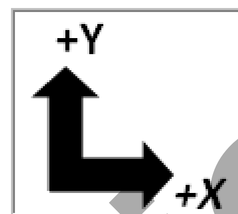
LESSON-6 - CNC - PART #8



LESSON-6 - CNC - PART #8

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE CONTOUR***
  - X0Y0 is at the lower left corner of the part
  - Use a 0.5" diameter End Mill Tool # 4
  - Start at X0 Y-0.375 and machine the contour in a clockwise direction – climb milling





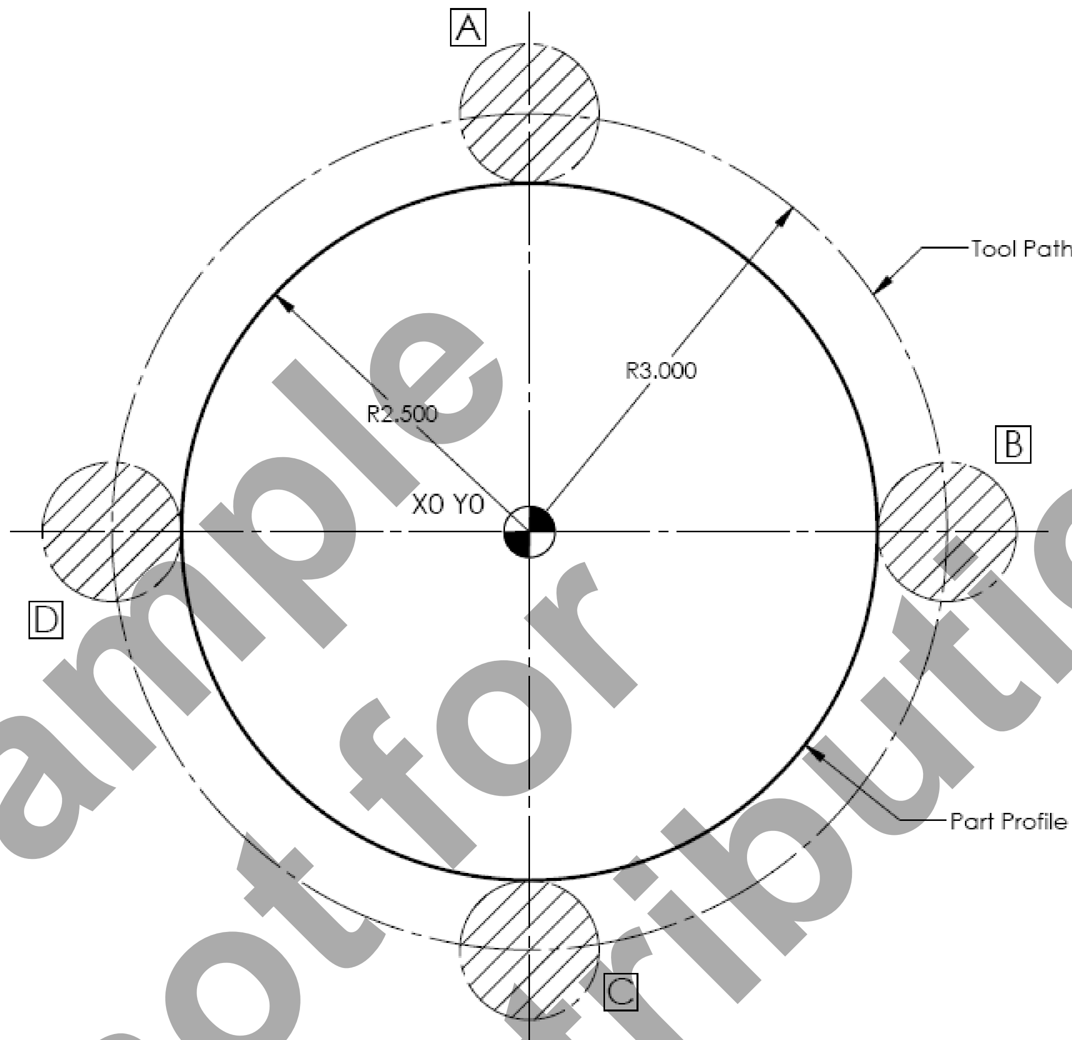
G90	X	Y
1	0	-0.375
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		

LESSON-6 - CNC - PART #8

- ***CREATE THE PROGRAM TO MACHINE THE CONTOUR***
  - Use a 0.5” diameter End Mill Tool # 4
  - Speed = 3050 Feed rate =20 IPM
  - X0Y0 is at the lower left corner of the part
  - Z=0 is the top of the part.
  - Machine the contour at a depth is 0.125”
  - Start at X-0.5 Y3.0 and machine the contour in a clockwise direction – climb milling
  - Type up your program and check it for correctness using NCPlot

%
O10 (Program #10 - CNC-PART- 8 - STRAIGHT LINE MILLING)
N10 G20
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)
N30

CNC Programming Workbook



Lesson-7

Circular Interpolation

camInstructor

LESSON-7 - CIRCULAR INTERPOLATION - G02 & G03

- ***In the next series of circular interpolation exercises you will explore how to machine arcs and complete circles.***
- ***G02 and G03 allow the machining of circles and arcs***

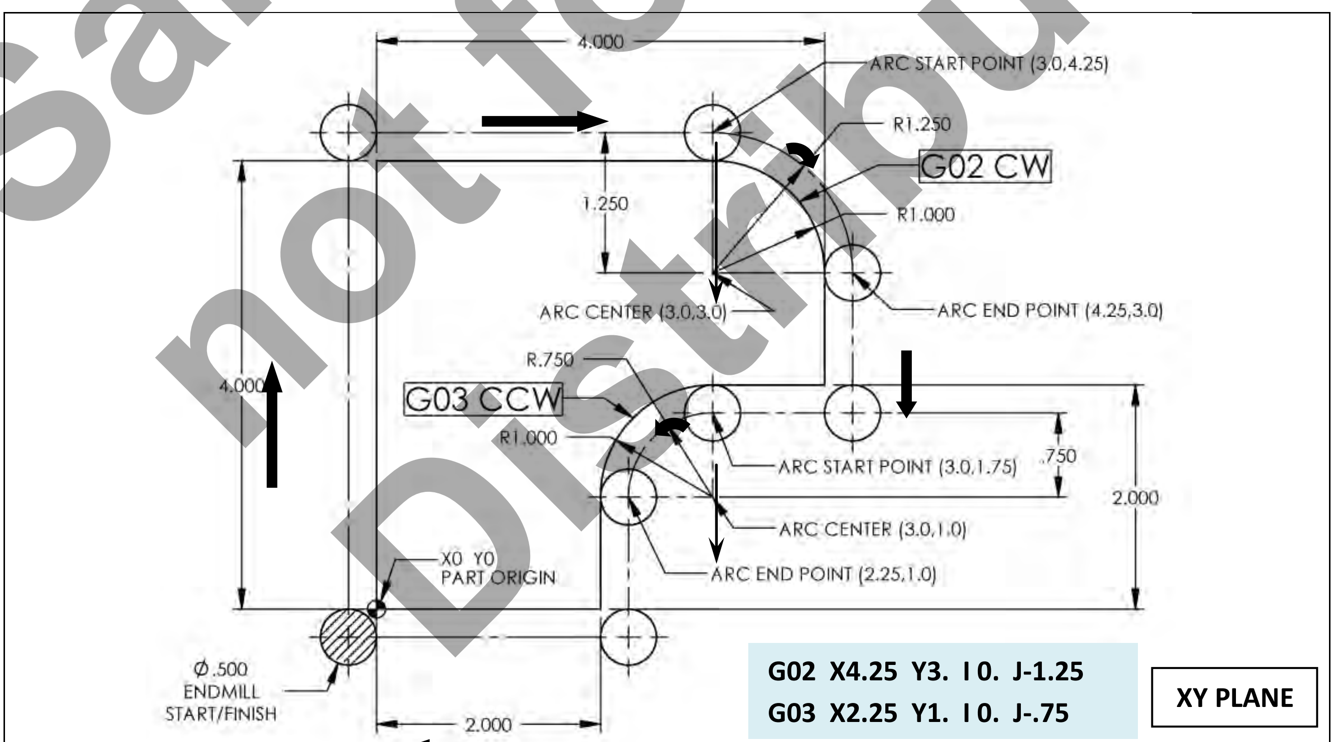
When the machine is required to move in a straight line under a controlled federate, linear interpolation is used G01. When it is necessary to machine in a circular motion in any plane (XY, YZ, XZ) circular interpolation is used G02 and G03.

All circular interpolation moves are defined using three pieces of information.

1. ***DIRECTION OF TRAVEL: CLOCKWISE G02, COUNTER CLOCKWISE G03***
2. ***PROGRAMMED END POINT OF THE ARC***
3. ***ARC CENTER: INCREMENTAL DISTANCE FROM START POINT TO ARC CENTER (I, J, K OR R FOR RADIUS, I,J AND K ARE NOT USED)***

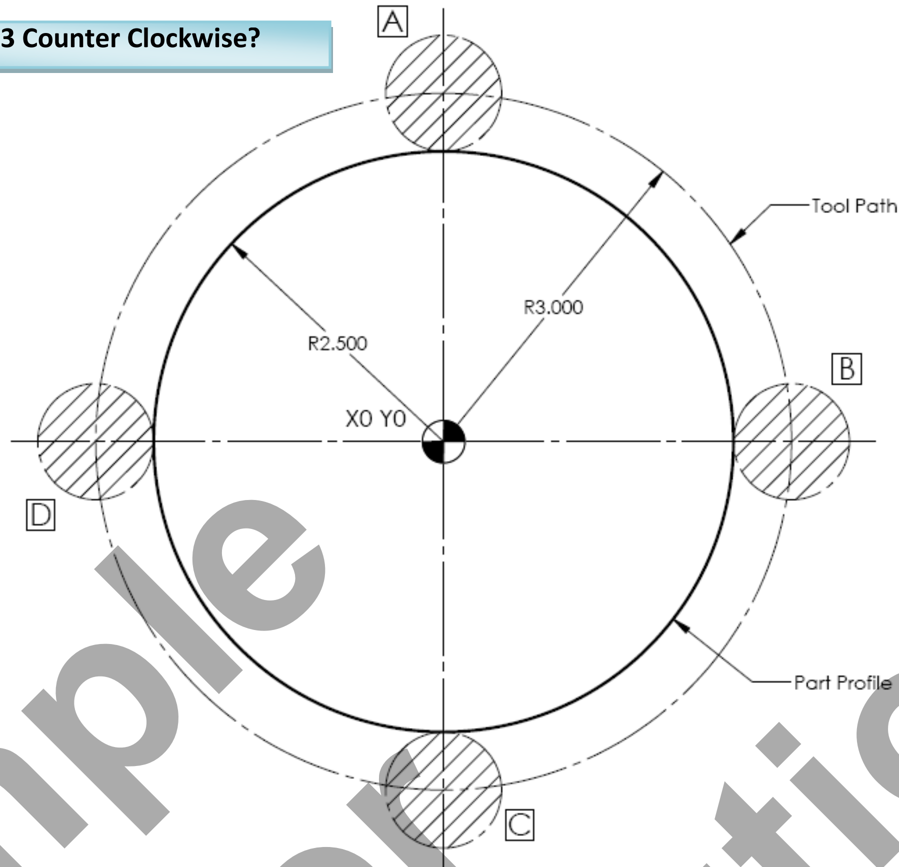
- ***When trying to figure out a circular interpolation move answer these three questions:***

  I. What is the direction of travel, clockwise or counterclockwise – G02 or G03?
  II. Where is the programmed end point?
  III. What is the incremental distance from the start of the arc to the center of the arc being machined – I and J values?



G02 & G03 Circular Interpolation Format

G02 -Clockwise or G03 Counter Clockwise?



What are the circular interpolation movements G02 or G03?

1. A to B Circular Interpolation (CW) **G02**

2. A to C Circular Interpolation (CW) **G02**

3. D to C Circular Interpolation (CCW) ?

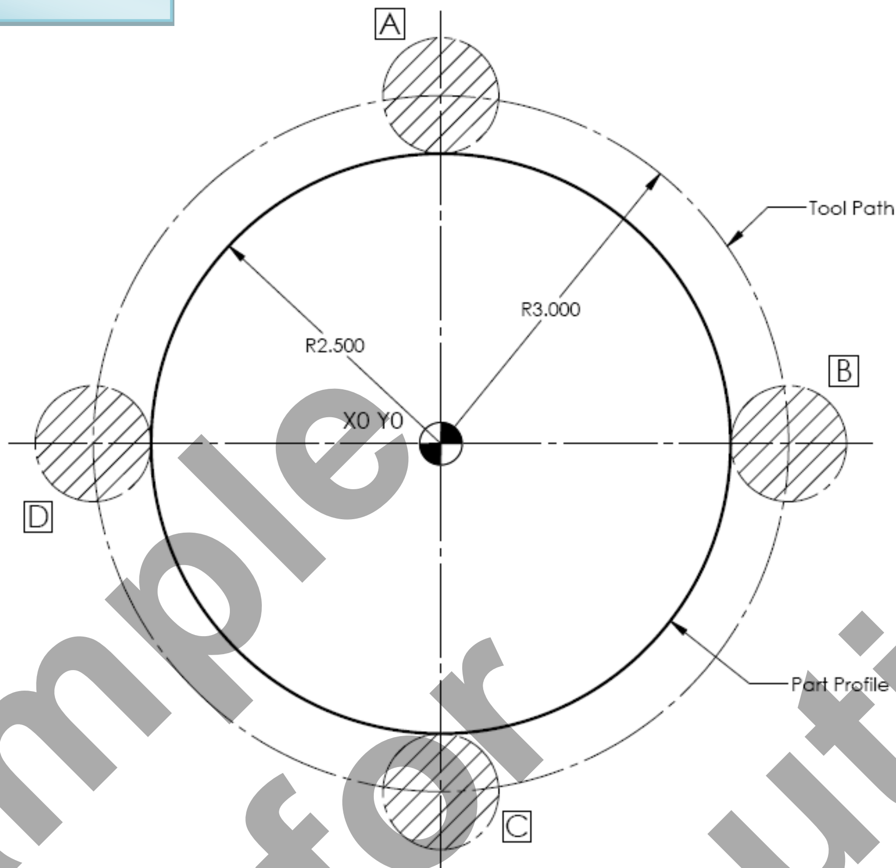
4. D to B Circular Interpolation (CCW) ?

5. A to A Circular Interpolation (CW) ?

CIRCULAR INTERPOLATION - **PROGRAMED END POINT**

1.0” Diameter End Mill

Radius = 0.5



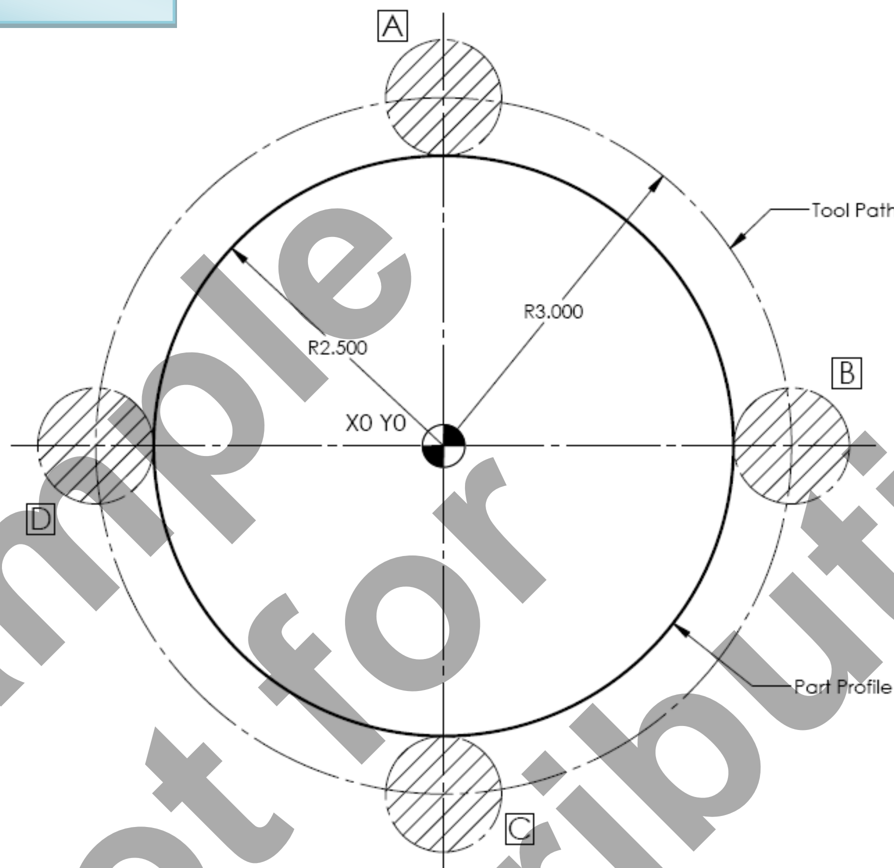
1. What are the coordinates at the center of the cutter for A, B, C, and D

A: **X0 Y3.0**	B: **X Y**
C: **X Y**	D: **X Y**

- ***I , J and K Values are measured from the tool start to the center of the arc***
- ***I , J and K values are INCREMENTAL***
- ***I= X Axis   J = Y Axis   K= Z Axis***

1.0” Diameter End Mill

Radius = 0.5

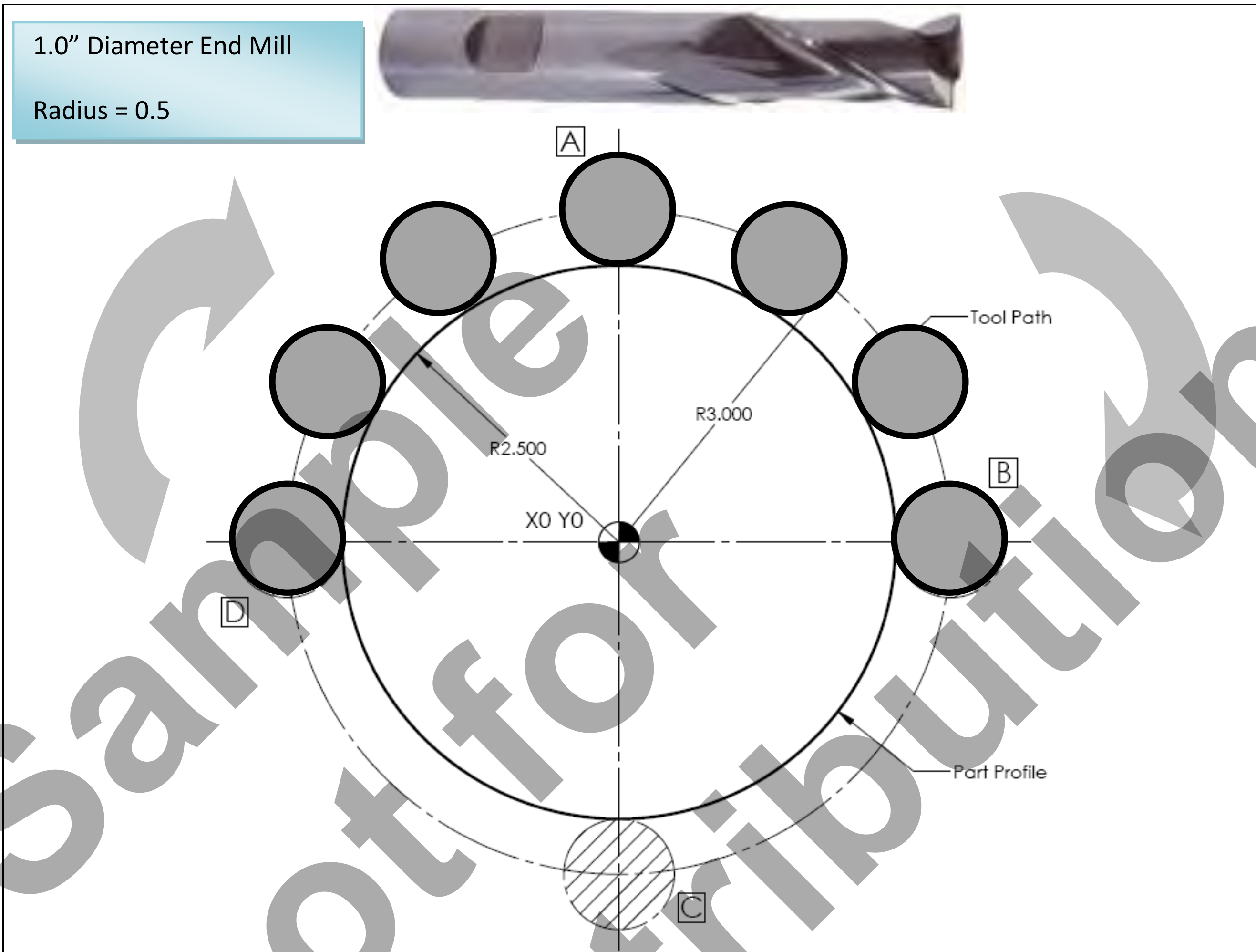


1. What are the **incremental** distances from the start of cut to the center of the programmed arc?

- **I =** The Incremental distance along the **X axis** from the start of the arc to the center of the programmed arc
- **J =** The Incremental distance along the **Y axis** from the start of the arc to the center of the programmed arc

A to the centre of the circle: **I0 J-3.0**	B to the centre of the circle: **I J**
C to the centre of the circle: **I J**	D to the centre of the circle: **I J**

- ***What does the block of code look like moving from D to B clockwise?***
  - ***G02 X3.0 Y0 I3.0 J0***
- ***What does the block of code look like moving from A to D counterclockwise?***
  - ***G03 X-3.0 Y0 I-3.0 J0***



- ***When trying to figure out a circular interpolation move answer these three questions:***
  I. What is the direction of travel, clockwise or counterclockwise – G02 or G03?
  II. Where is the programmed end point?
  III. What is the incremental distance from the start of the arc to the center of the arc being machined – I and J values?
- ***Work out the following circular interpolation blocks***

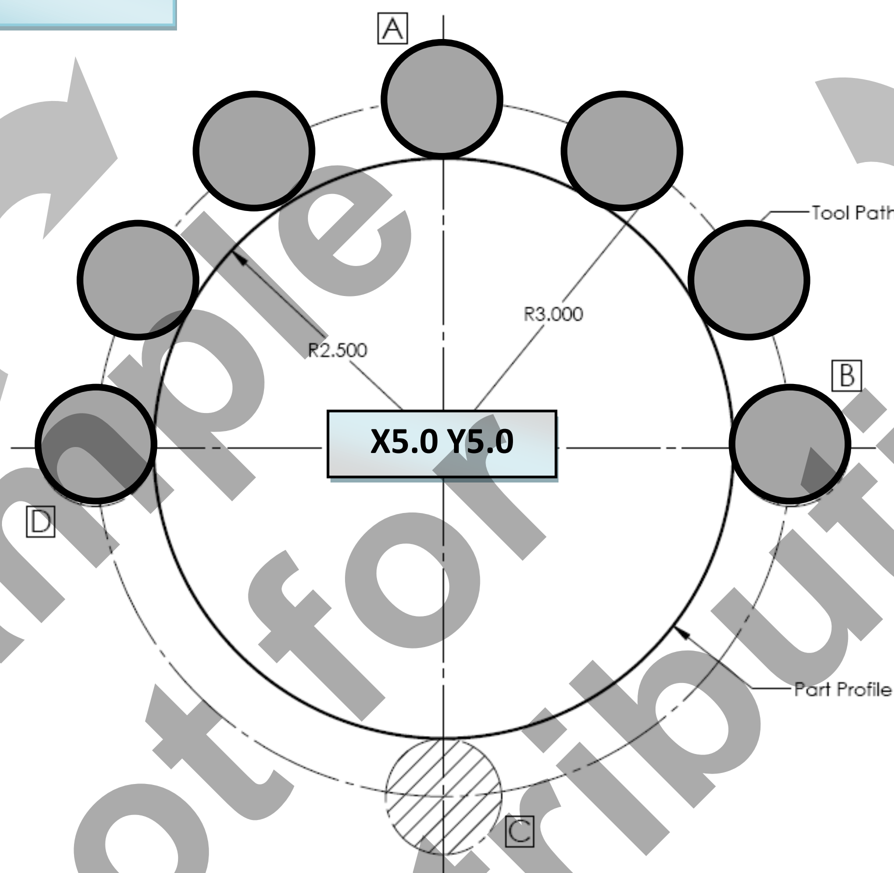
C to D Clockwise:	**D to C** Counterclockwise:
A to A Clockwise:	**B to C** Counterclockwise:

LESSON-7 - CIRCULAR INTERPOLATION - G02 & G03

- ***The center of the circle has now been changed to X5.0 Y5.0***
- ***What does the block of code look like moving from D to B clockwise?***
  - ***G02 X8.0 Y5.0 I3.0 J0***
- ***What does the block of code look like moving from A to D counterclockwise?***
  - ***G03 X2.0 Y5.0 I-3.0 J0***

1.0" Diameter End Mill

Radius = 0.5



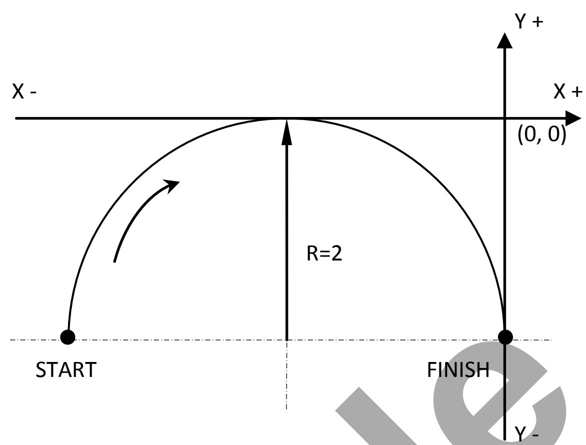
- ***When trying to figure out a circular interpolation move answer these three questions:***
  I. What is the direction of travel, clockwise or counterclockwise – G02 or G03?
  II. Where is the programmed end point?
  III. What is the incremental distance from the start of the arc to the center of the arc being machined – I and J values?
- ***Work out the following circular interpolation blocks***

C to D Clockwise:	**D to C** Counterclockwise:
A to A Clockwise:	**B to C** Counterclockwise:

To cut a complete circle of 360°, you do not need to specify an end point X, Y, or Z. Just program I, J, or K to define the centre of the circle.

Work out the following circular interpolation blocks

PROGRAM CIRCULAR INTERPOLATION USING ABSOLUTE AND INCREMENTAL IN DIRECTION SHOWN



ABSOLUTE

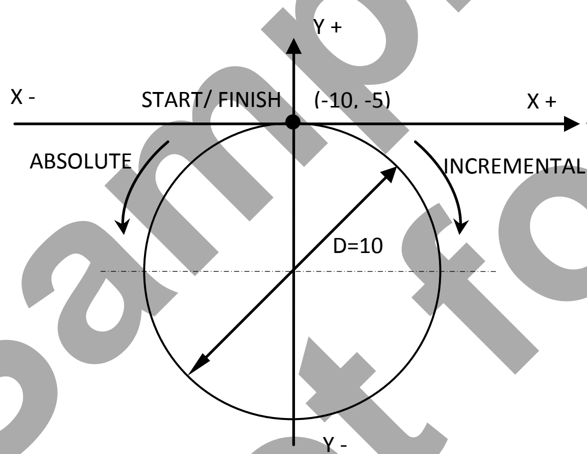
START POINT	X	Y
END POINT	X	Y

G	X	Y	I	J

INCREMENTAL

START POINT	X	Y
END POINT	X	Y

G	X	Y	I	J



ABSOLUTE (COUNTER CLOCKWISE DIRECTION)

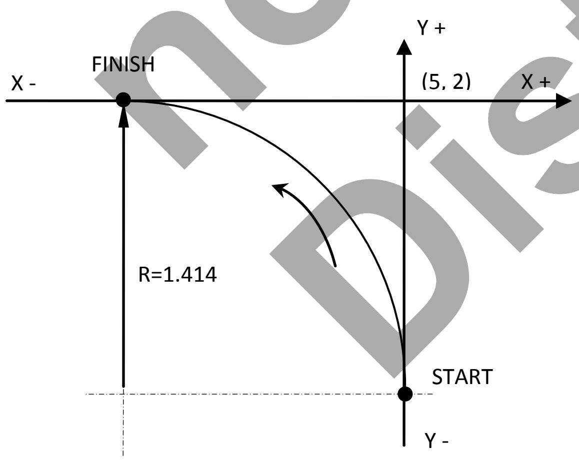
START POINT	X	Y
END POINT	X	Y

G	X	Y	I	J

INCREMENTAL (CLOCKWISE DIRECTION)

START POINT	X	Y
END POINT	X	Y

G	X	Y	I	J



ABSOLUTE

START POINT	X	Y
END POINT	X	Y

G	X	Y	I	J

INCREMENTAL

START POINT	X	Y
END POINT	X	Y

G	X	Y	I	J

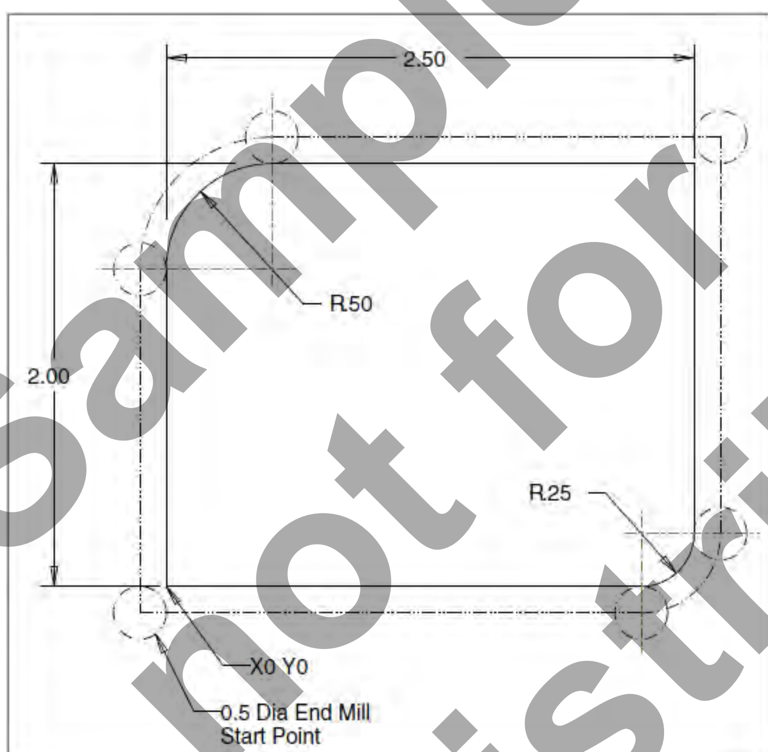
LESSON-7 – CIRCULAR INTERPOLATION G02

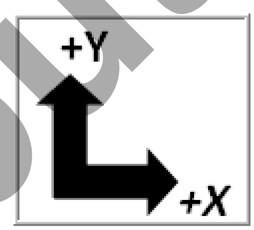
Review the CNC program below that machines the contour.

- The cutter being used is a 0.5” diameter end mill.
- X0 Y0 is the lower left corner of the part.
- Start position lower left corner, machines clockwise.
- The blocks of CNC code are made up of G01 and a G02 moves.
- Note that the G02 move to machine the 0.5” and 0.25” radius can use either **I and J** values or **R** for the radius.

All circular interpolation moves are defined using three pieces of information.

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



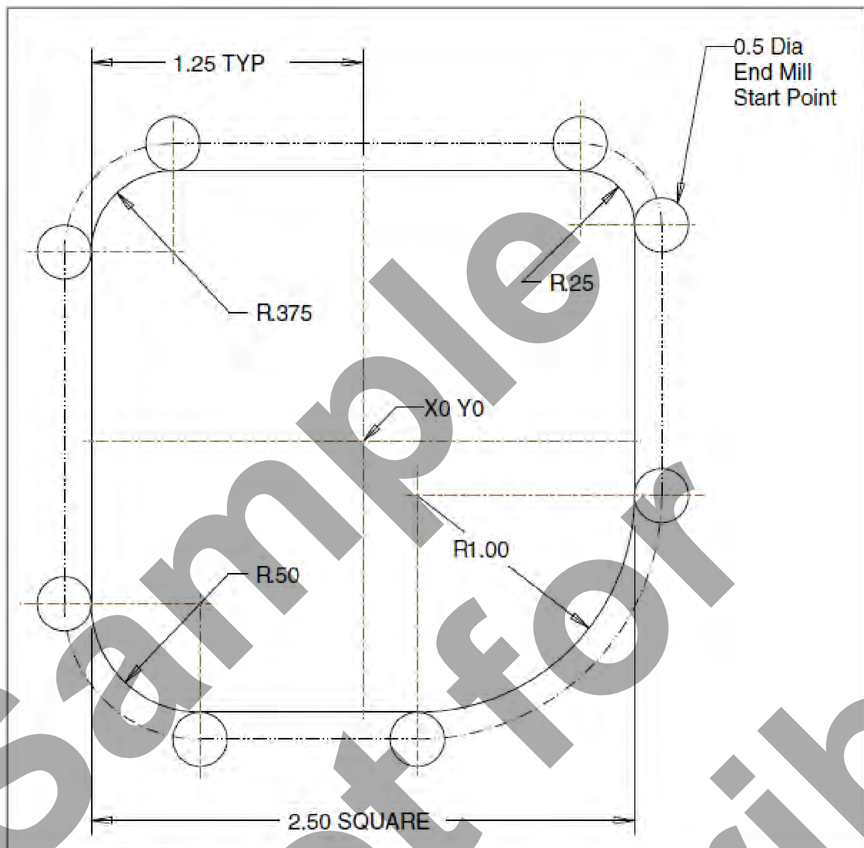


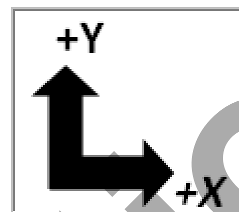
N160 G1 X-.25 Y-.25 (start position) **N170 Y1.5** **N180 G2 X.5 Y2.25 I.75 J0. (OR R0.5)** **N190 G1 X2.75** **N200 Y.25** **N210 G2 X2.25 Y-.25 I-.5 J0. (OR R0.25)** **N220 G1 X-.25**	**I =** The Incremental distance along the **X axis** from the start of the arc to the center of the programmed arc **J =** The Incremental distance along the **Y axis** from the start of the arc to the center of the programmed arc

LESSON-7 – CIRCULAR INTERPOLATION G02

Review the CNC program below that machines the contour.

- The cutter being used is a 0.5" diameter end mill.
- X0 Y0 is the center of the part.
- Start position is the upper right, machines clockwise.
- The blocks of CNC code are made up of G01 and a G02 moves.
- Note that the G02 move to machine the radii can use either **I and J** values or **R** for the radius.





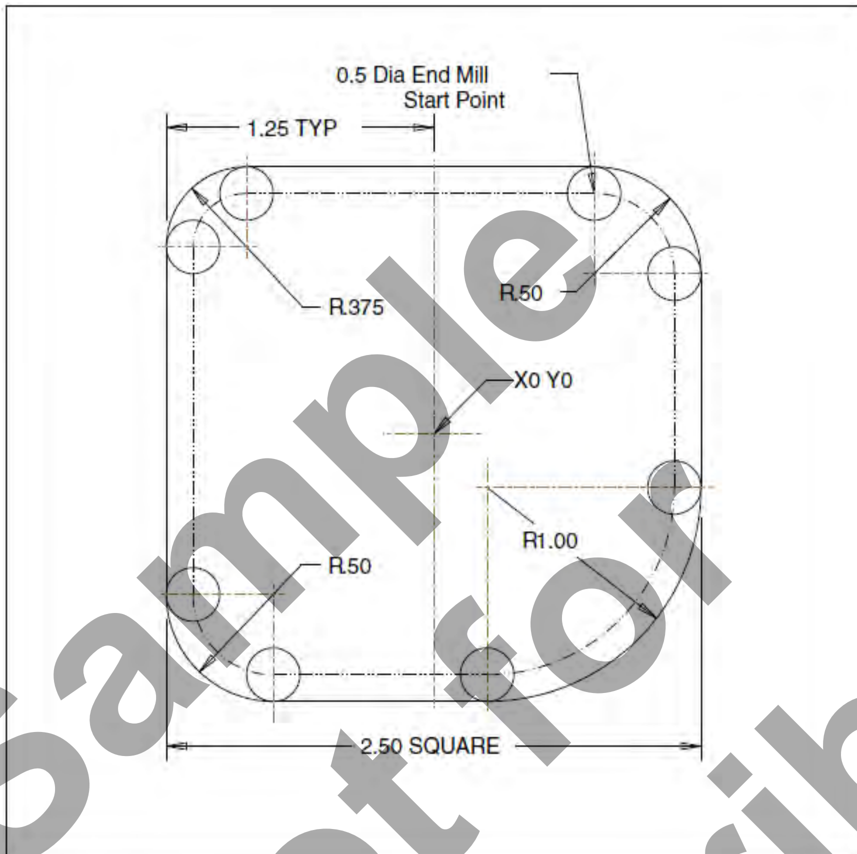
Using I and J Values	Using R for Radius Values
N160 G1 X1.5 Y1.0 (start position)	**N160 G1 X1.5 Y1.0 (start position)**
N170 Y-.25	**N170 Y-.25**
N180 G2 X.25 Y-1.5 I-1.25 J0.	**N180 G2 X.25 Y-1.5 R1.25**
N190 G1 X-.75	**N190 G1 X-.75**
N200 G2 X-1.5 Y-.75 I0. J.75	**N200 G2 X-1.5 Y-.75 R.75**
N210 G1 Y.875	**N210 G1 Y.875**
N220 G2 X-.875 Y1.5 I.625 J0.	**N220 G2 X-.875 Y1.5 R.625**
N230 G1 X1.	**N230 G1 X1.**
N240 G2 X1.5 Y1. I0. J-.5	**N240 G2 X1.5 Y1. R.5**

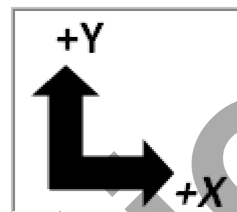
To cut a complete circle of 360°, you do not need to specify an end point X, Y, or Z. Just program I, J, or K to define the centre of the circle.

LESSON-7 – CIRCULAR INTERPOLATION G03

Review the CNC program below that machines the contour around the inside of the pocket.

- The cutter being used is a 0.5” diameter end mill.
- X0 Y0 is the center of the part.
- Start position is the upper right, machines contour counter clockwise.
- The blocks of CNC code are made up of G01 and a G03 moves.
- Note that the G03 move to machine the radii can use either **I and J** values or **R** for the radius.

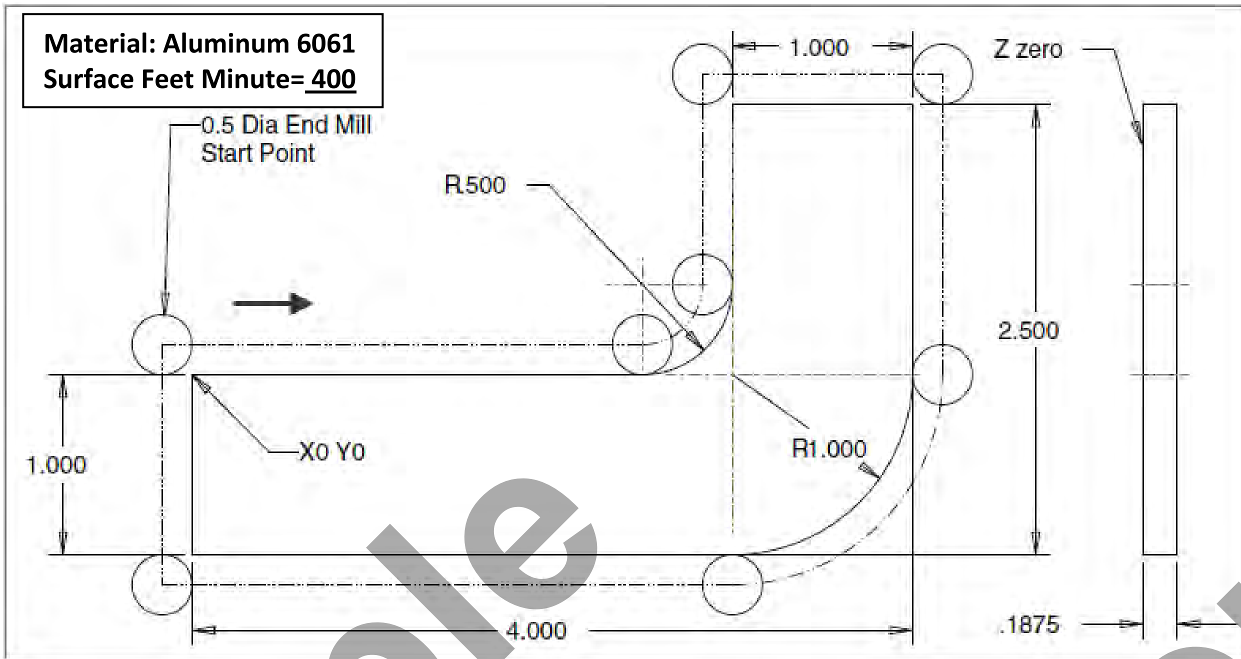




Using I and J Values	**Using R for Radius Values**
N40 G0 G90 G54 X.75 Y1. A0. S2500 M3	**N40 G0 G90 G54 X.75 Y1. A0. S2500 M3**
N50 G43 H4 Z.35	**N50 G43 H4 Z.35**
N60 G1 Z.125 F15.	**N60 G1 Z.125 F15.**
N70 X-.875	**N70 X-.875**
N80 G3 X-1. Y.875 I0. J-.125 (0.375 Radius)	**N80 G3 X-1. Y.875 R.125 (0.375 Radius)**
N90 G1 Y-.75	**N90 G1 Y-.75**
N100 G3 X-.75 Y-1. I.25 J0. (0.5 Radius)	**N100 G3 X-.75 Y-1. R.25 (0.5 Radius)**
N110 G1 X.25	**N110 G1 X.25**
N120 G3 X1. Y-.25 I0. J.75 (1.0 Radius)	**N120 G3 X1. Y-.25 R.75 (1.0 Radius)**
N130 G1 Y.75	**N130 G1 Y.75**
N140 G3 X.75 Y1. I-.25 J0. (0.5 Radius)	**N140 G3 X.75 Y1. R.25 (0.5 Radius)**
N150 G0 Z.35	**N150 G0 Z.35**

LESSON-7 – CIRCULAR INTERPOLATION G02 & G03

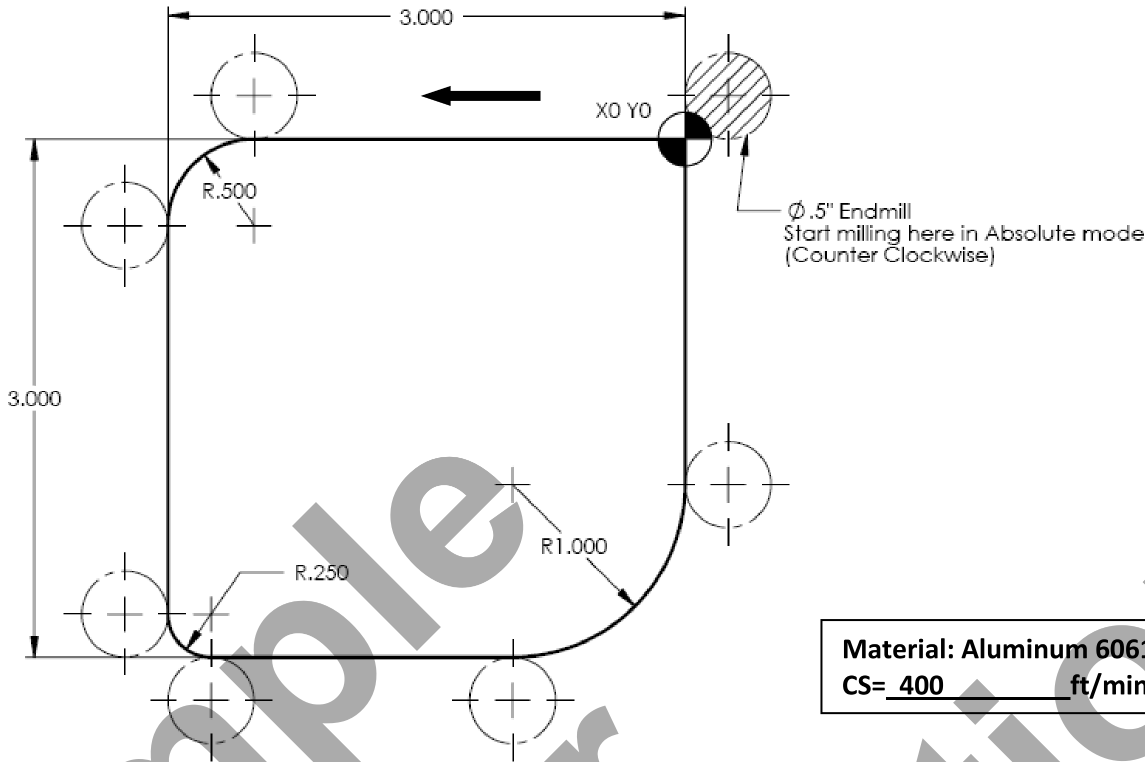
Review the program to machine the contour shown below.



Tool #4 (Ø.500" Flat End Mill) – Mill the profile	
Spindle Speed RPM = 3.82xSFM/D= 3.82x400/0.5 =3056	Feed Per Tooth (FPT) = 0.003 – 2 Flute Cutter Feed Per Minute = FPT x 2 Flutes x RPM = 18.336
Depth of Cut = Z-0.200	Start from the top left corner

%	
O11 (G02-G03-EXAMPLE)	N60 X4.25
N10 G20	N65 Y0.
N15 G0 G17 G40 G49 G80 G90	N70 G2 X3. Y-1.25 I-1.25 J0.
N20 T4 M6	N75 G1 X-.25
N25 G0 G90 G54 X-.25 Y.25 S3056 M3	N80 Y.25
N30 G43 H4 Z2.	N85 G0 Z2.
N35 Z.1	N90 M5
N40 G1 Z-.2 F18.	N95 G91 G28 Z0.
N45 X2.5	N100 G28 X0. Y0.
N50 G3 X2.75 Y.5 I0. J.25	N105 M30
N55 G1 Y1.75	%

Create the program to machine the contour shown below.

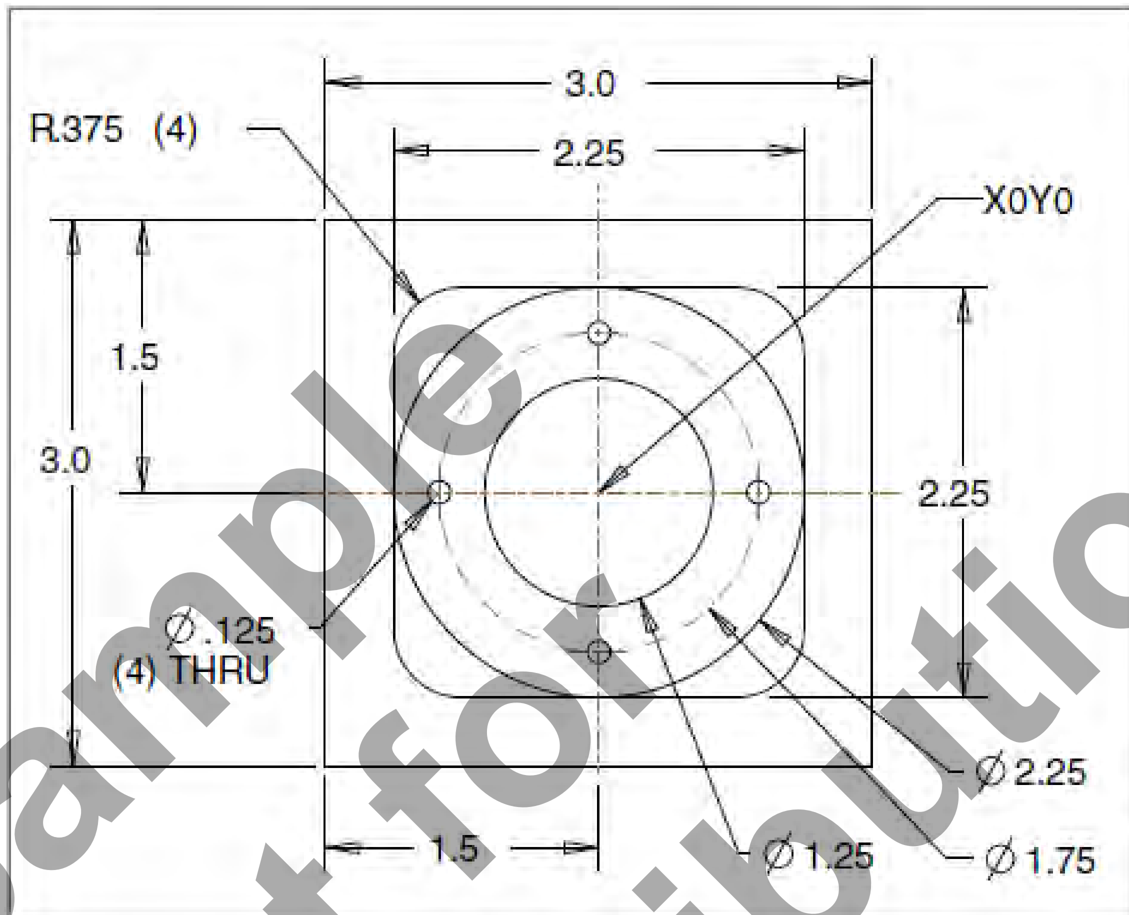


Material: Aluminum 6061
CS=_400__________ft/min

Tool #4 (Ø.500" Flat End Mill) – Mill the profile	
Spindle Speed RPM = 3.82xSFM/D= 3.82x400/0.5 =3056	Feed Per Tooth (FPT) = 0.003 – 2 Flute Cutter Feed Per Minute = FPT x 2 Flutes x RPM = 18.336
Depth of Cut = Z-0.150	Start from the top right hand corner

%
O12

CNC PROGRAMMING WORKBOOK

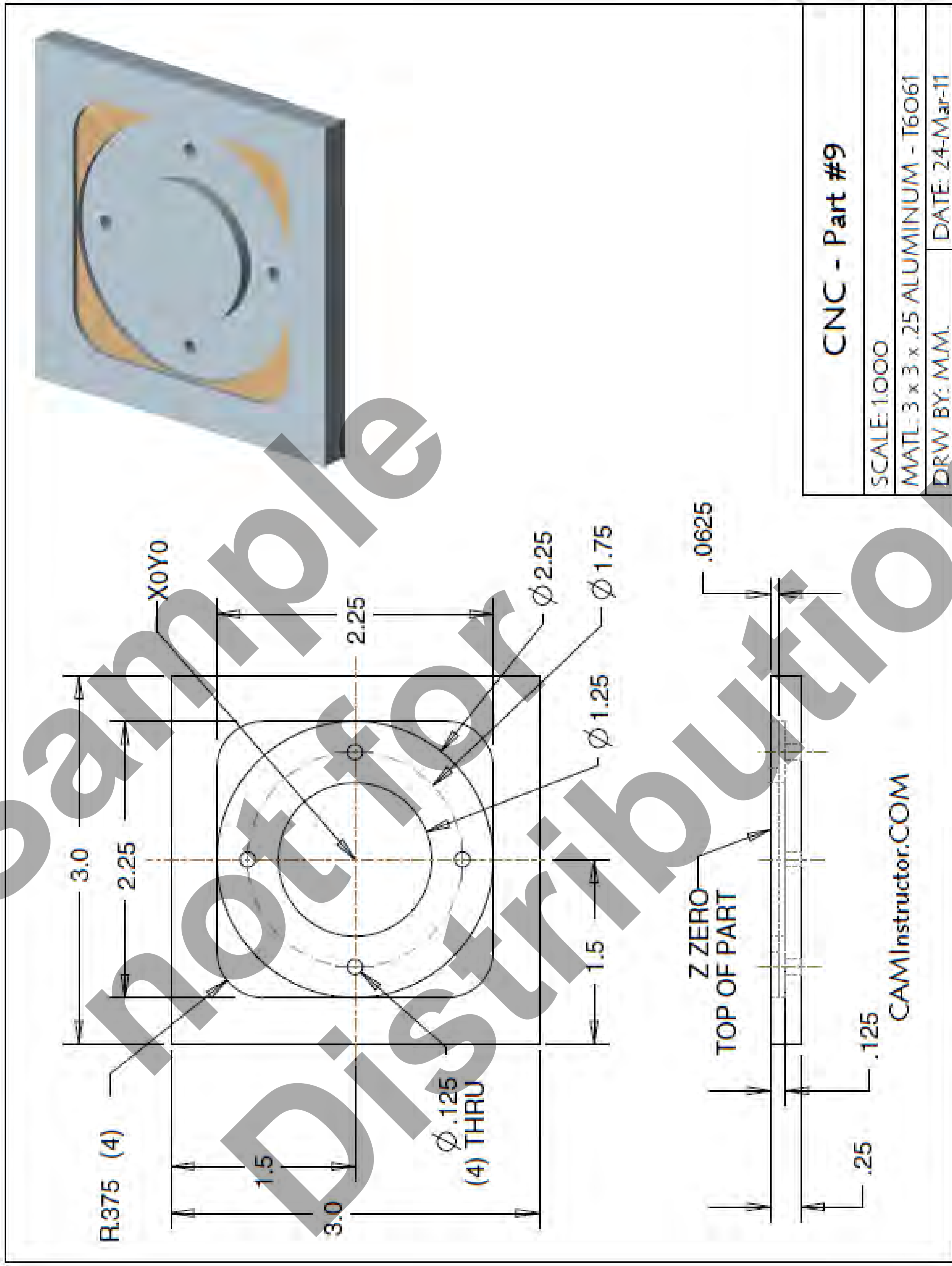


MILL-LESSON-8

CIRCULAR INTERPOLATION - CONTINUED

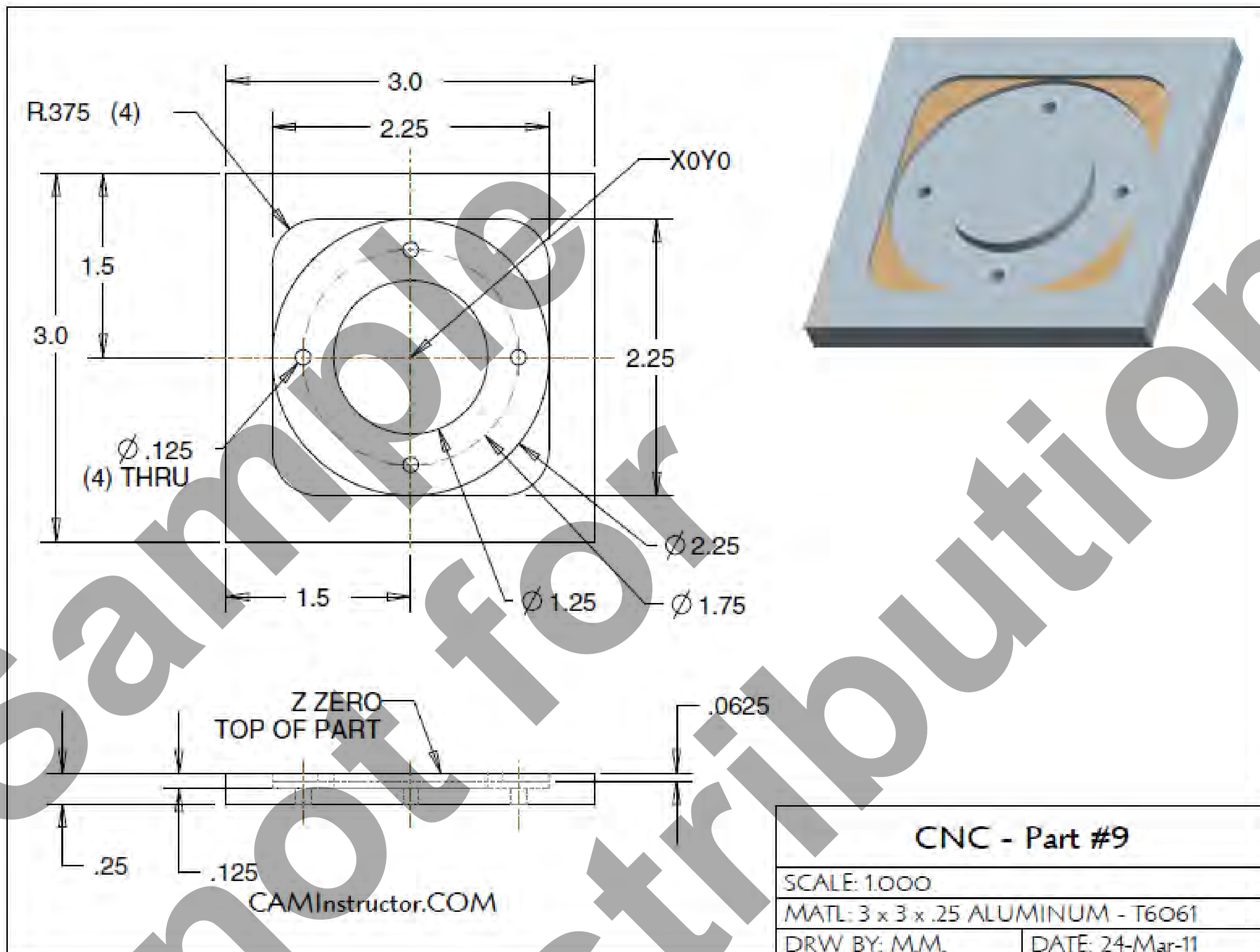
camInstructor

LESSON-8 - CNC - PART #9



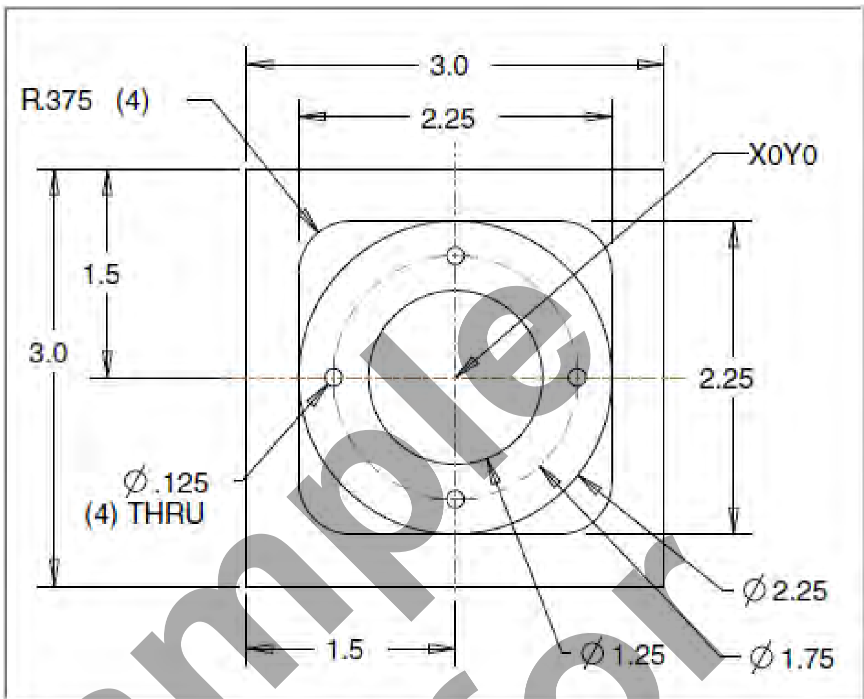
LESSON-8 - CNC - PART #9

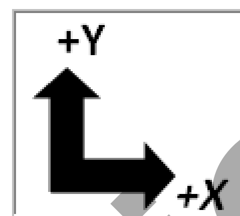
- ***The Machining Process***
- ***Ø.500" Endmill - Tool # 4 - 3056 RPM Feedrate 18 IPM***
  - Machine the 2.25" square with the 0.375" corner radii - 0.0625" depth
  - Machine the circular 2.25" and 1.25"diameter circular profile - 0.125" depth
- ***Ø.125" Endmill - Tool # 1- 5000 RPM Feedrate 7 IPM***
  - Drill the Ø .125" holes through the part (4 places)
  - **Note:** The Ø.125" Endmill is designed for center cutting machining. No center drilling or pilot hole is required.



LESSON-8 - CNC - PART #9

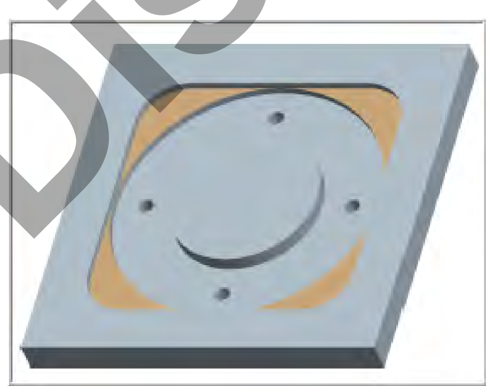
- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART***
  - X0Y0 is at the center of the part
  - Use a 0.5” diameter End Mill Tool # 4 and a 0.125” diameter End Mill Tool # 1
  - Climb mill the inside contours





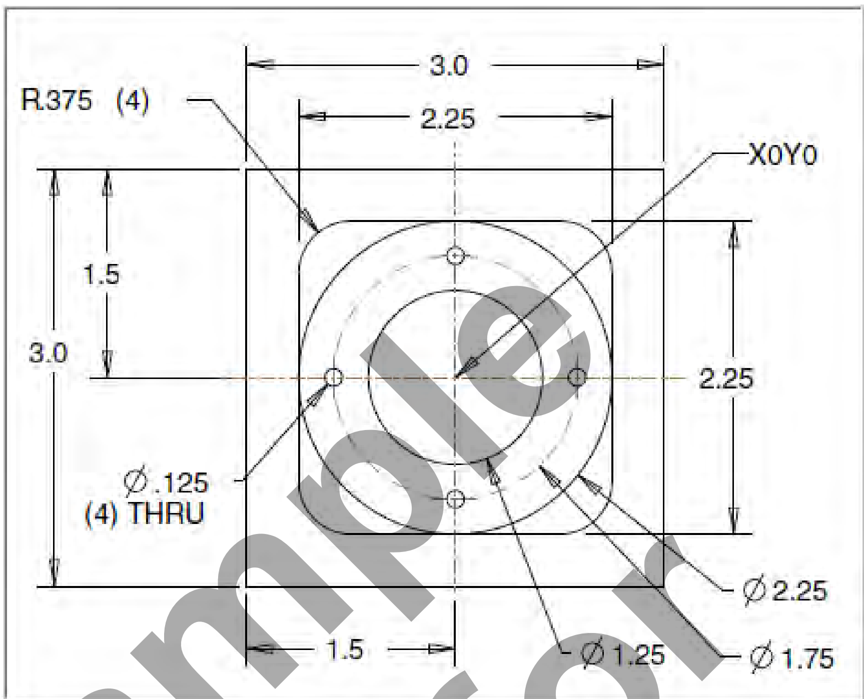
- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE FOUR HOLES***

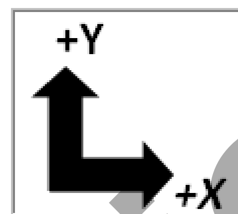
G90	X	Y
1		
2		
3		
4		



LESSON-8 - CNC - PART #9

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART***
  - X0Y0 is at the center of the part
  - Use a 0.5” diameter End Mill Tool # 4 and a 0.125” diameter End Mill Tool # 1
  - Climb mill





- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS OF THE 0.5” DIAMETER END MILL***

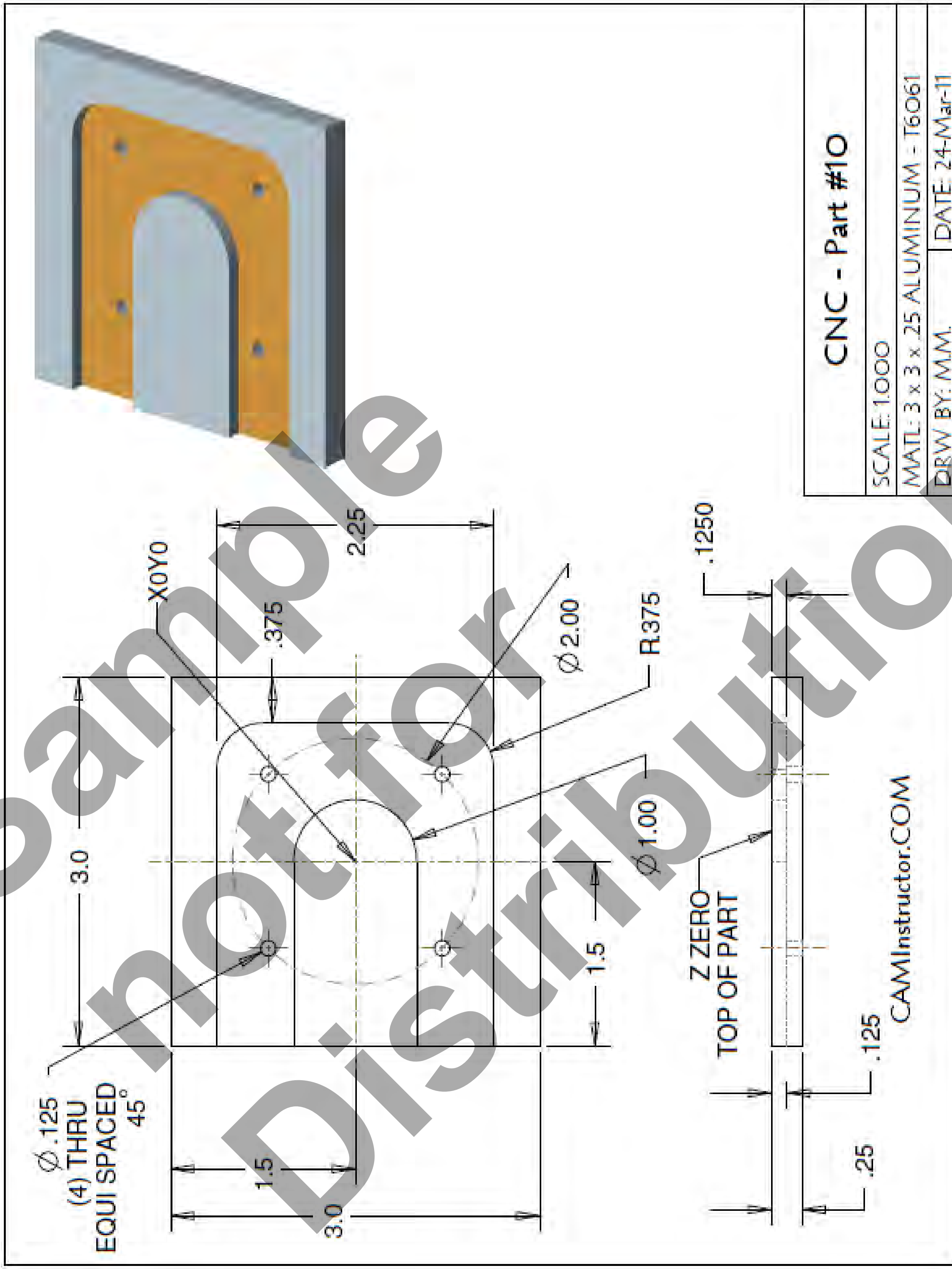
G90	X	Y
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		

LESSON-8 - CNC - PART #9

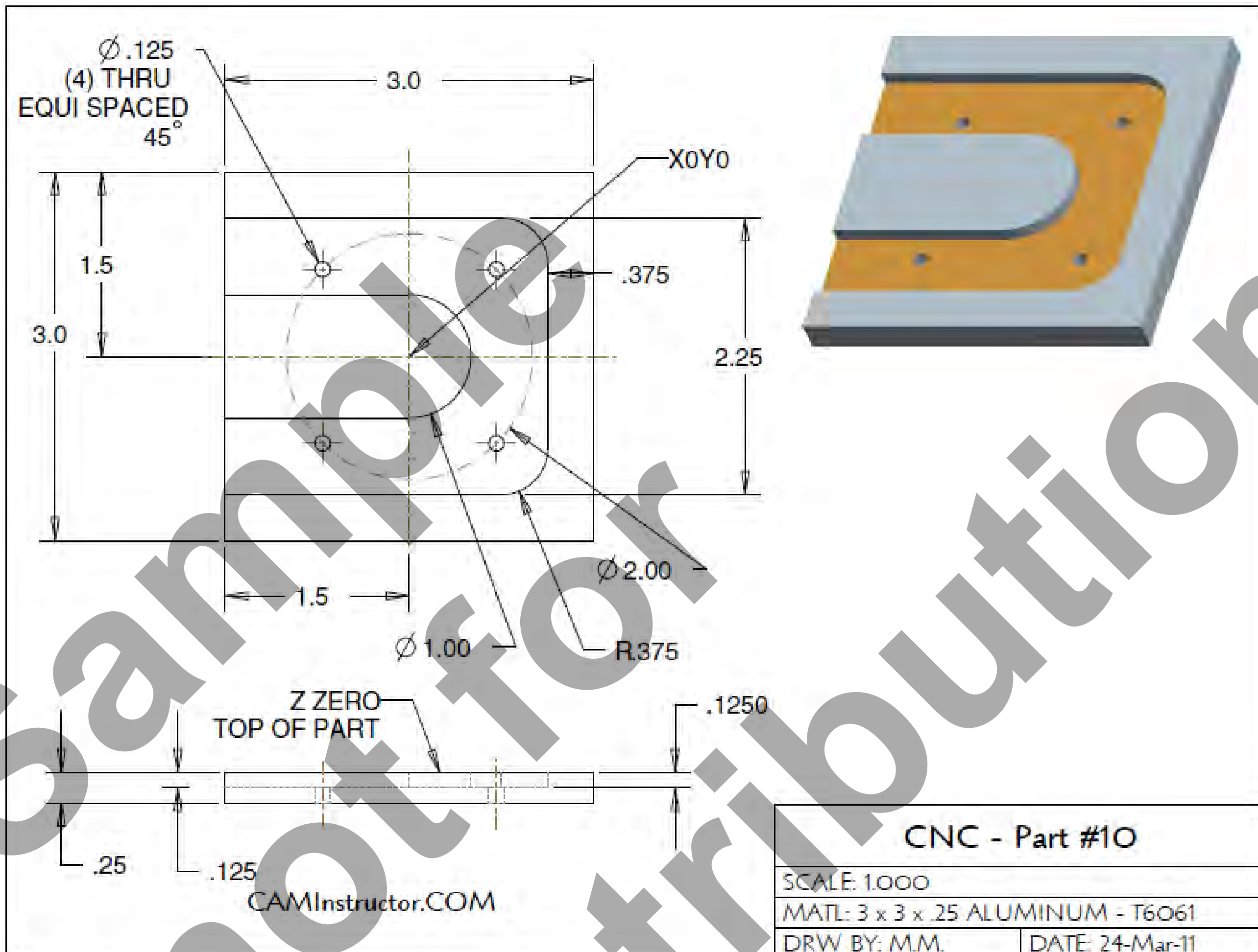
- ***CREATE THE PROGRAM TO MACHINE THE PART***
  - Use a 0.5” diameter End Mill Tool # 4
  - Speed = 3050 Feed rate =20 IPM
  - Use a 0.125” diameter End Mill Tool # 1
  - Speed = 5000 Feed rate =7 IPM
  - X0Y0 is at the center of the part
  - Z=0 is the top of the part.
  - Climb mill
  - Type up your program and check it for correctness using NCPlot

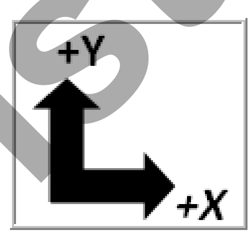
%
O13   (CNC PART #9)
N10   G20
N20   G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)
N30

LESSON-8 - CNC - PART #9 - Continued


CNC - Part #10
SCALE: 1.000
MATL: 3 x 3 x .25 ALUMINUM - T6061
DRW BY: M.M.
DATE: 24-Mar-11
X0Y0
.375
2.25
Ø2.00
R.375
.1250
Ø1.00
Z ZERO
TOP OF PART
3.0
1.5
Ø.125
(4) THRU
EQUI SPACED
45°
.125
.25
CAMInstructor.COM

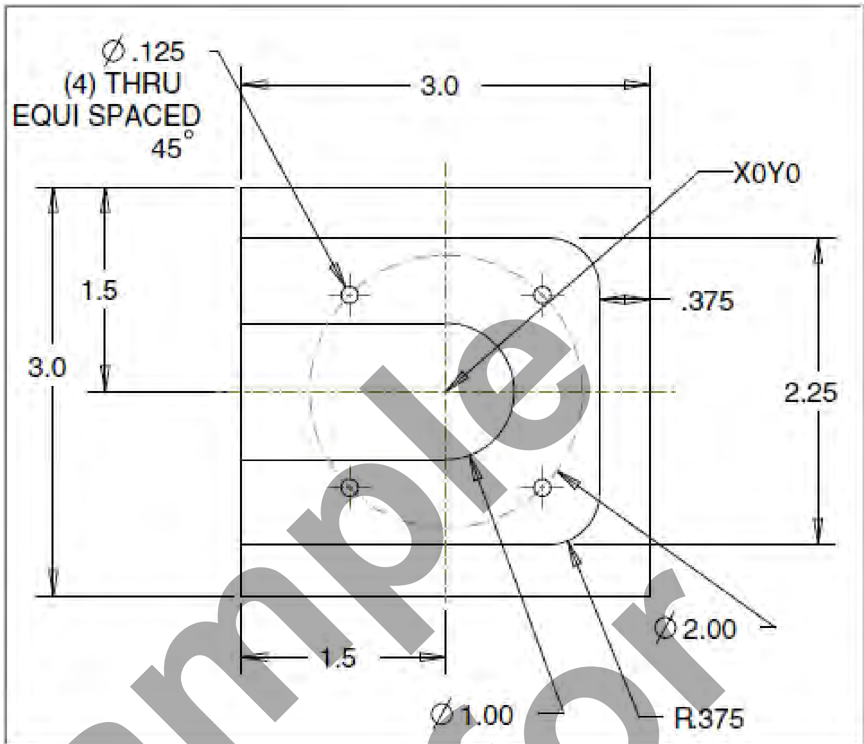
- ***The Machining Process***
- ***Ø.500" Endmill - Tool # 4 - 3056 RPM Feedrate 18 IPM***
  - Machine the open slot with the 0.375" fillet radii - 0.125" depth
- ***Ø.125" Endmill - Tool # 1 - 5000 RPM Feedrate 7 IPM***
  - Drill the Ø .125" holes through the part (4 places)
  - **Note:** The Ø.125" Endmill is designed for center cutting machining. No center drilling or pilot hole is required.

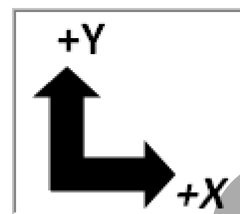




LESSON-8 - CNC - PART #10

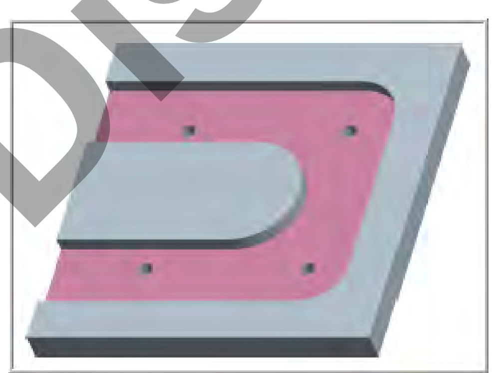
- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART***
  - X0Y0 is at the center of the part
  - Use a 0.5” diameter End Mill Tool # 4 and a 0.125” diameter End Mill Tool # 1





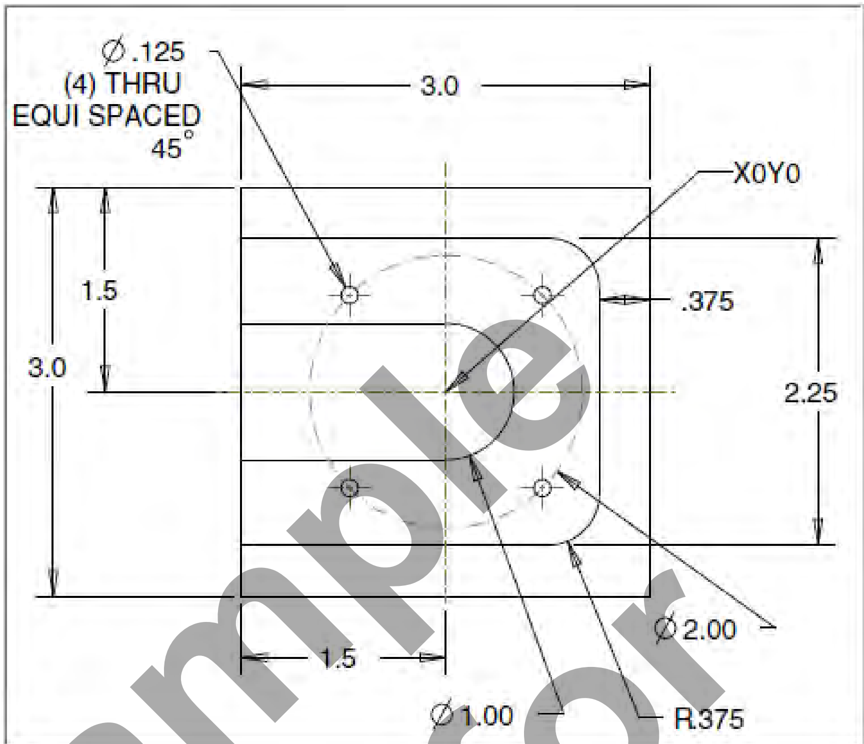
- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE FOUR HOLES***

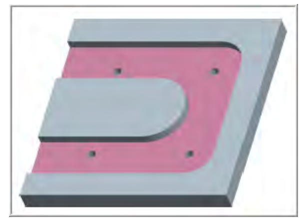
G90	X	Y
1		
2		
3		
4		

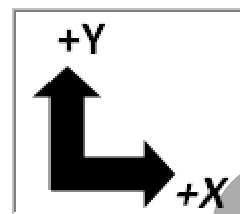


LESSON-8 - CNC - PART #10

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART***
  - X0Y0 is at the center of the part
  - Use a 0.5” diameter End Mill Tool # 4 and a 0.125” diameter End Mill Tool # 1
  - Climb mill



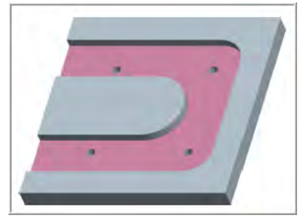




- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS OF THE 0.5’ DIAMETER END MILL***

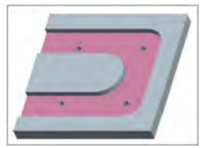
G90	X	Y
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		

- ***CREATE THE PROGRAM TO MACHINE THE PART***
  - Use a 0.5” diameter End Mill Tool # 4
  - Speed = 3050 Feed rate =20 IPM
  - Use a 0.125” diameter End Mill Tool # 1
  - Speed = 5000 Feed rate =7 IPM
  - X0Y0 is at the center of the part
  - Z=0 is the top of the part.
  - Climb mill
  - Type up your program and check it for correctness using NCPlot

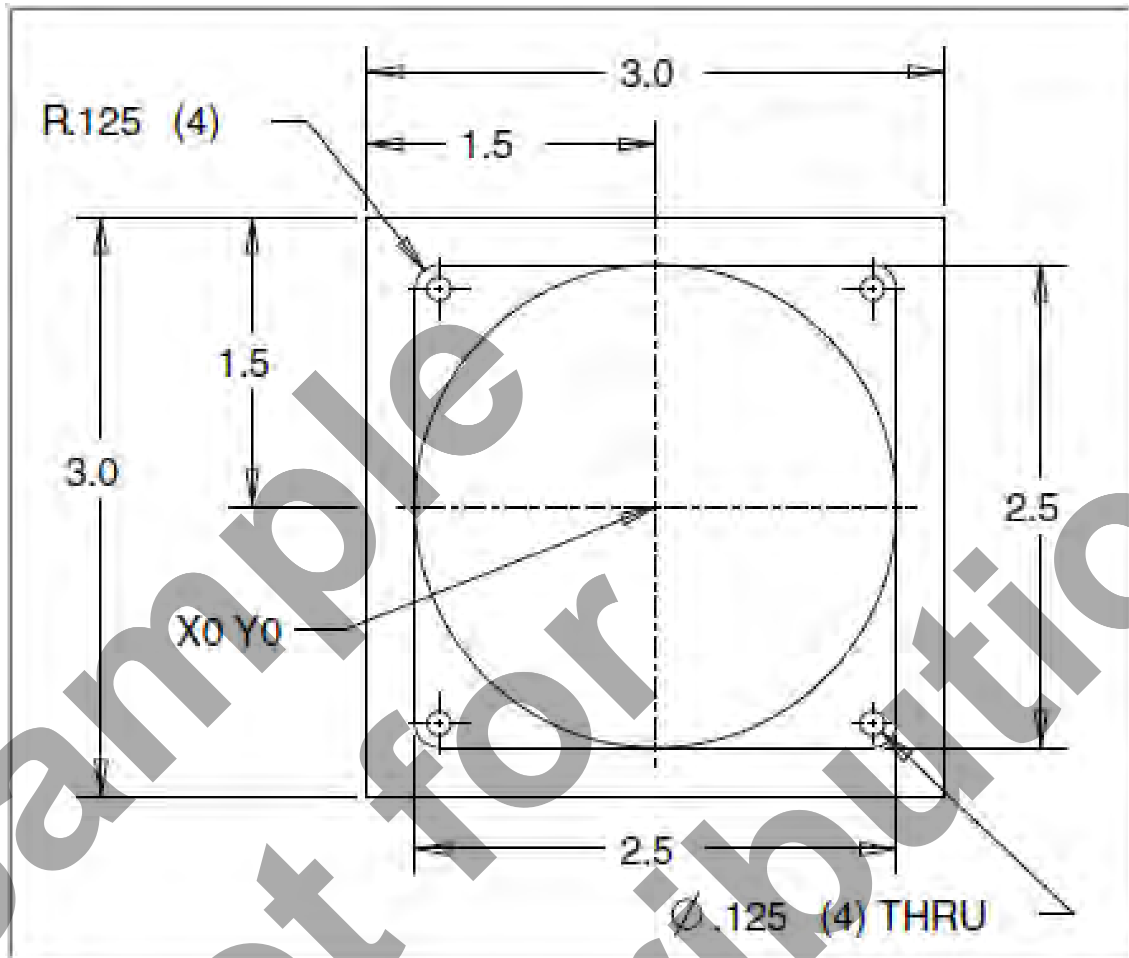


%
O003
N10   G20
N20   G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)
N30

LESSON-8 - CNC - PART #10



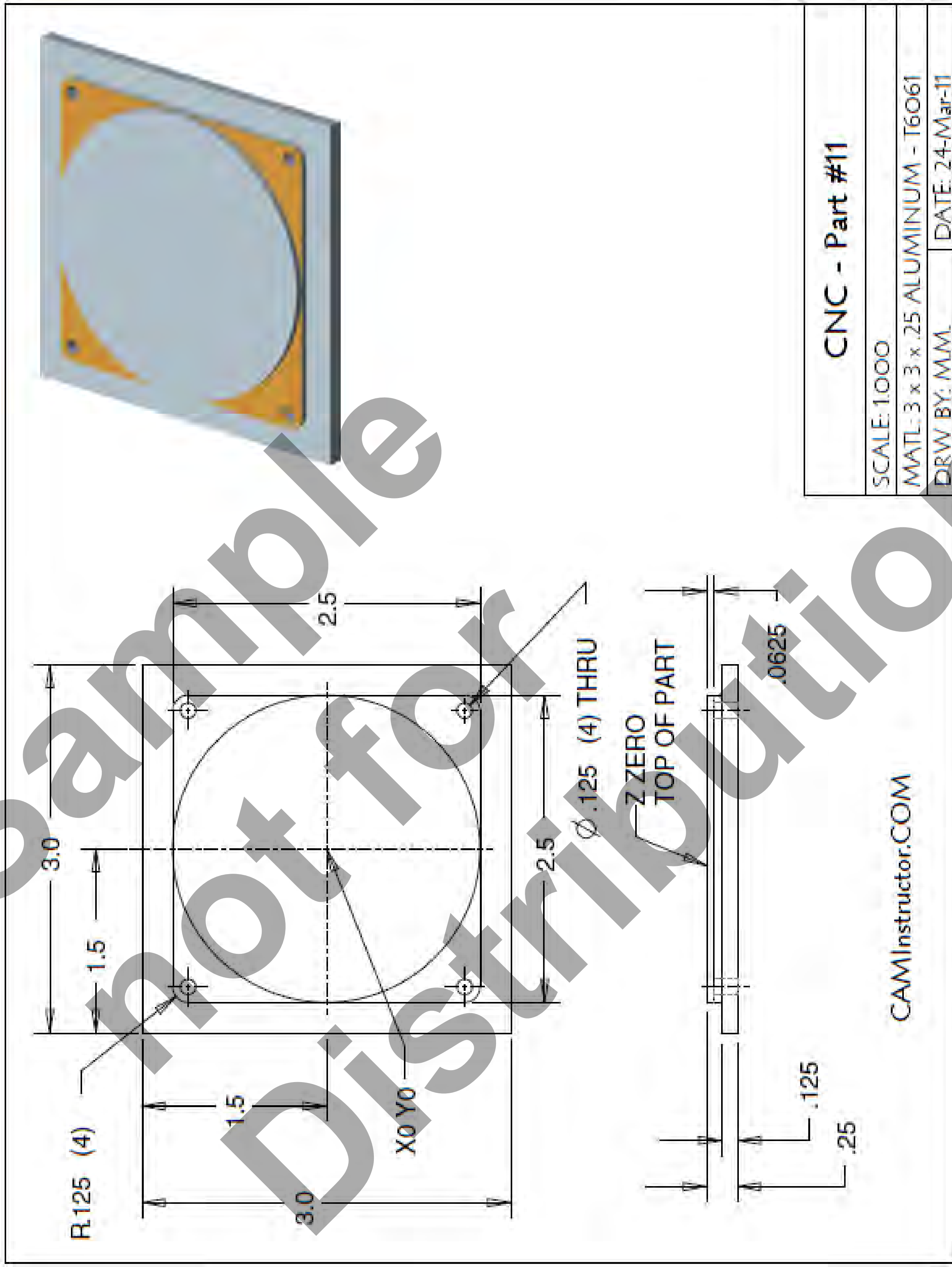
CNC PROGRAMMING WORKBOOK

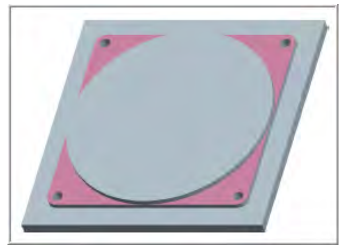


LESSON-9

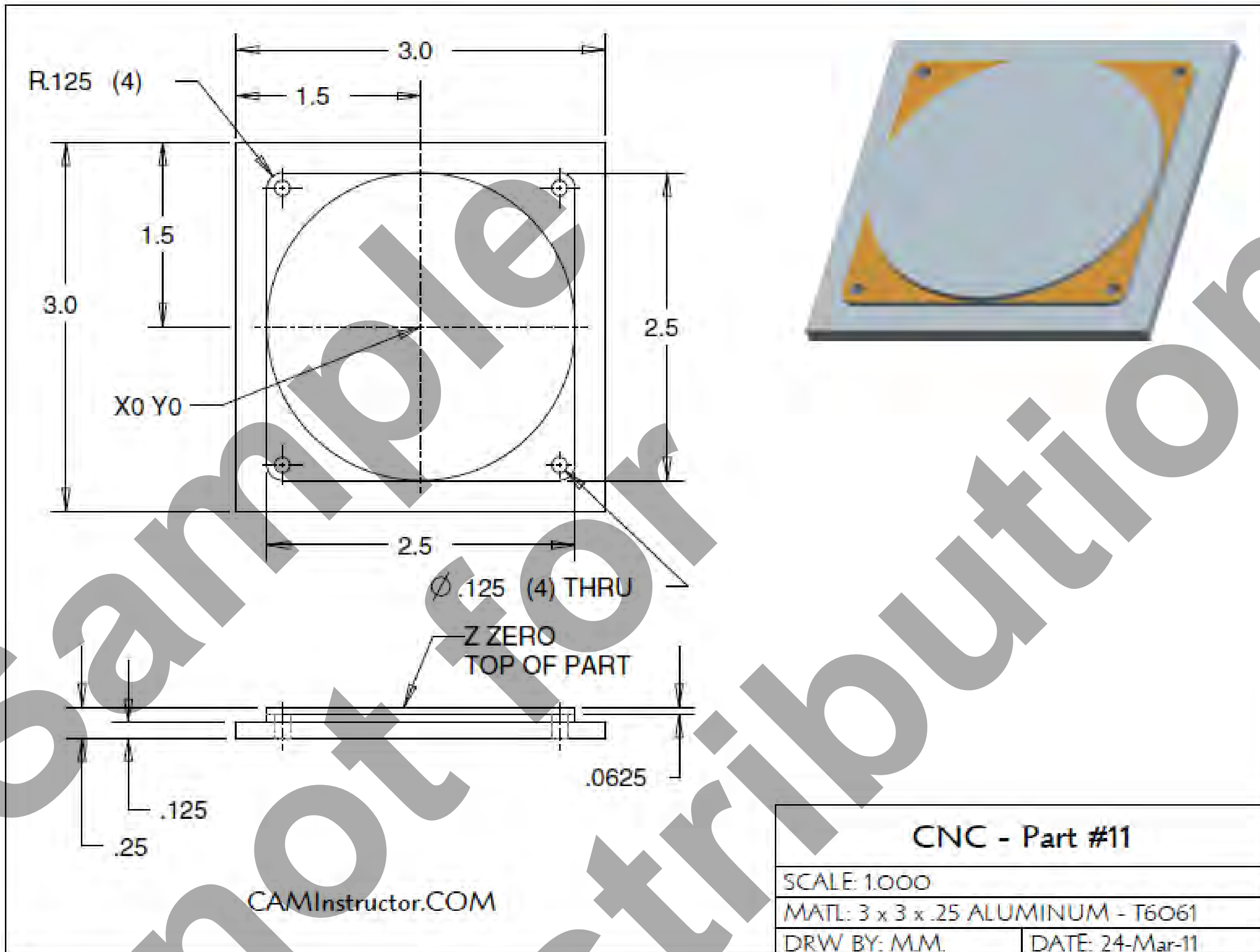
CIRCULAR INTERPOLATION - CONTINUED

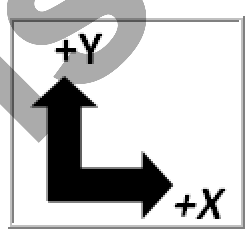
camInstructor


CNC - Part #11
SCALE: 1.000
MATL: 3 x 3 x .25 ALUMINUM - T6061
DRW BY: M.M.
DATE: 24-Mar-11
2.5
3.0
1.5
2.5
Ø.125 (4) THRU
Z ZERO
TOP OF PART
.0625
R.125 (4)
1.5
X0 Y0
3.0
.125
.25
CAMInstructor.COM

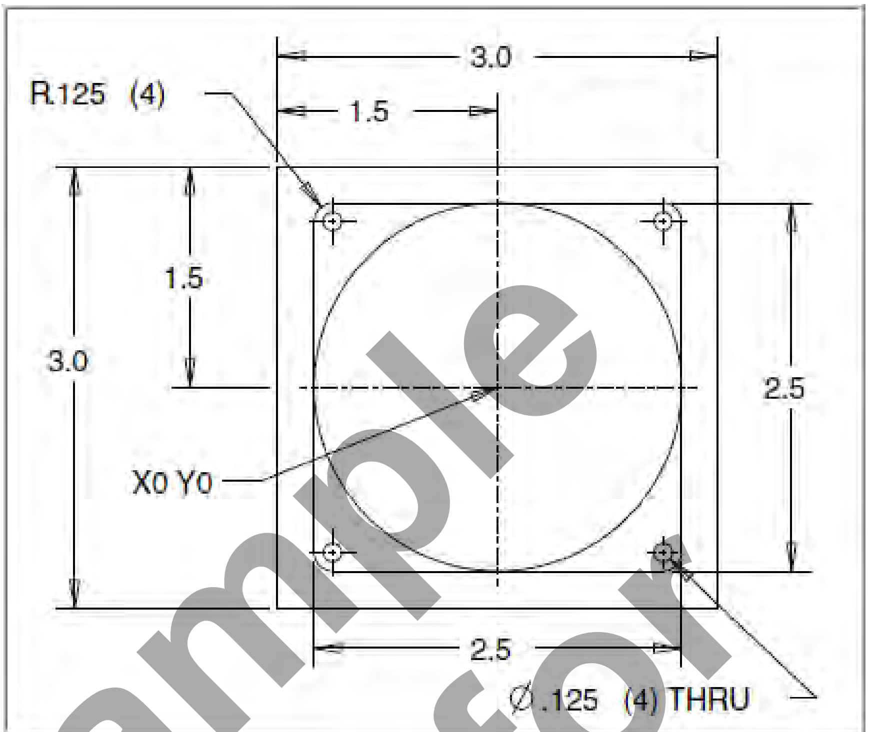


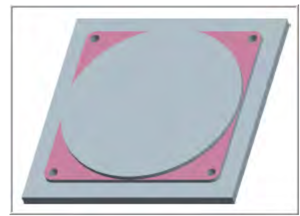
- ***The Machining Process***
- ***Ø.500" Endmill - Tool # 4***
  - Machine the profile with the .125" radii at Z-.125" deep – 1 Cut
  - Machine the circular 2.5" diameter circular profile – 1 Cut
- ***Ø.125" Endmill - Tool # 1***
  - Drill the Ø .125" holes through the part (4 places)
  - **Note:** The Ø.125" Endmill is designed for center cutting machining. No center drilling or pilot hole is required.

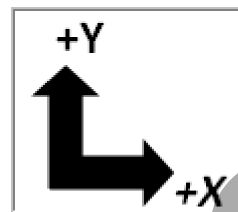




- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART***
  - X0Y0 is at the center of the part
  - Use a 0.5” diameter End Mill Tool # 4 and a 0.125” diameter End Mill Tool # 1

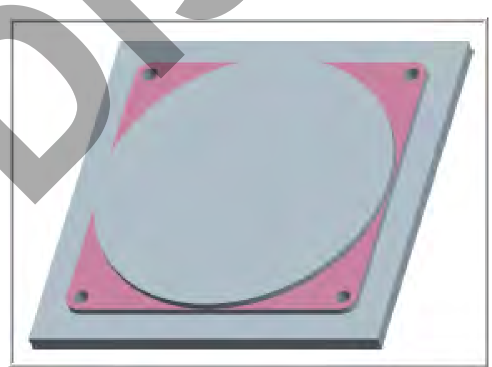




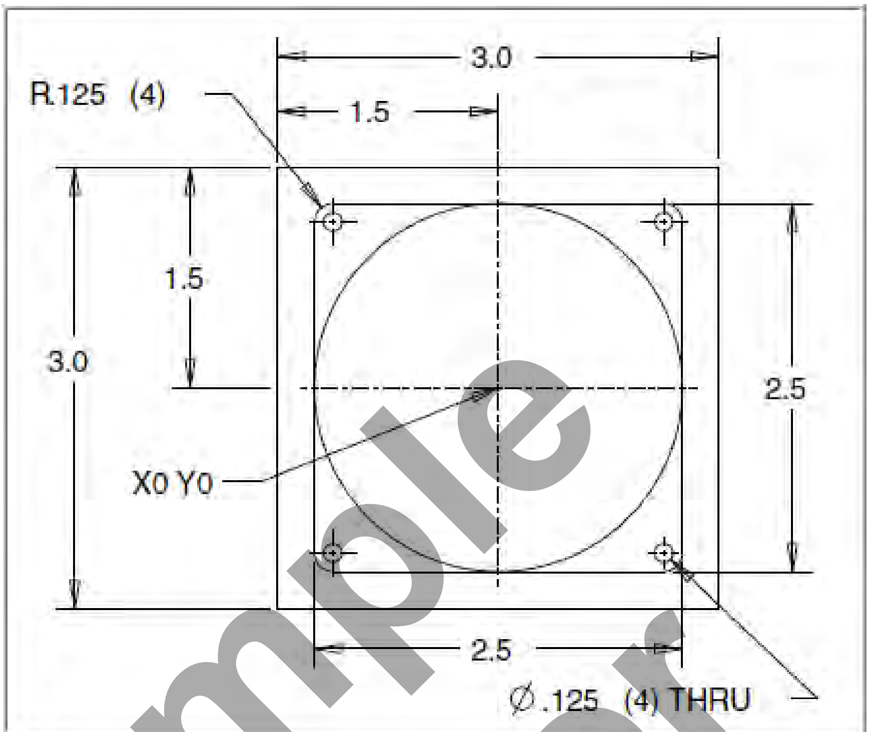


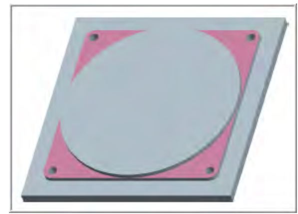
- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE FOUR HOLES***

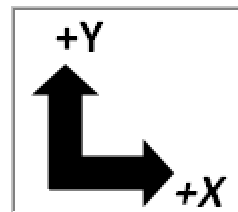
G90	X	Y
1		
2		
3		
4		



- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART***
  - X0Y0 is at the center of the part
  - Use a 0.5” diameter End Mill Tool # 4 and a 0.125” diameter End Mill Tool # 1
  - Climb mill



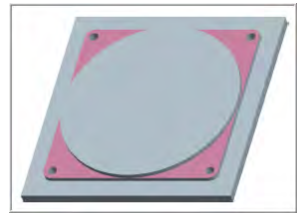




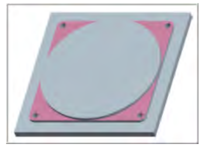
- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS OF THE 0.5’ DIAMETER END MILL***

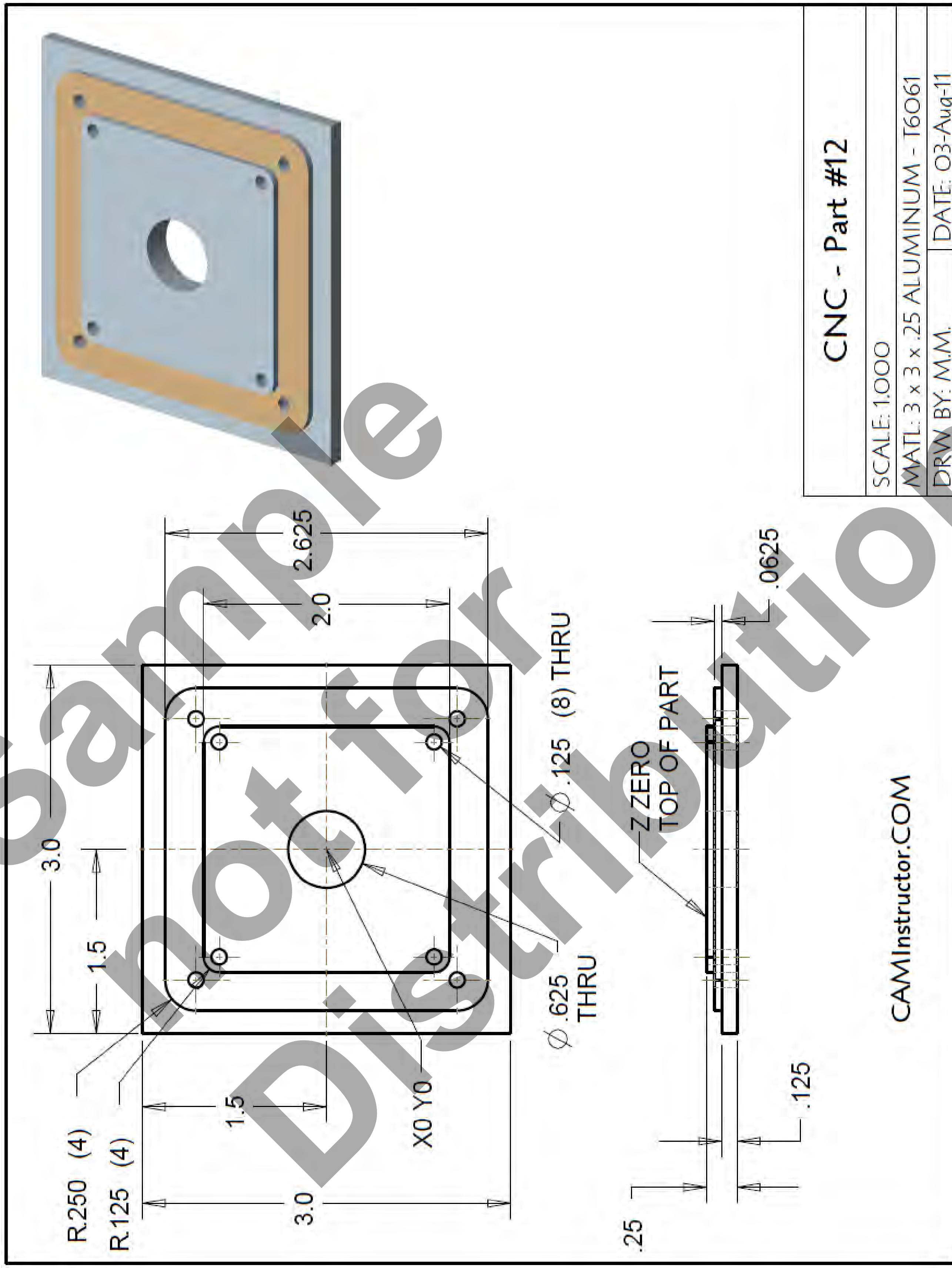
G90	X	Y
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		

- ***CREATE THE PROGRAM TO MACHINE THE PART***
  - Use a 0.5” diameter End Mill Tool # 4
  - Speed = 3050 Feed rate =20 IPM
  - Use a 0.125” diameter End Mill Tool # 1
  - Speed = 5000 Feed rate =7 IPM
  - X0Y0 is at the center of the part
  - Z=0 is the top of the part.
  - Climb mill
  - Type up your program and check it for correctness using NCPlot

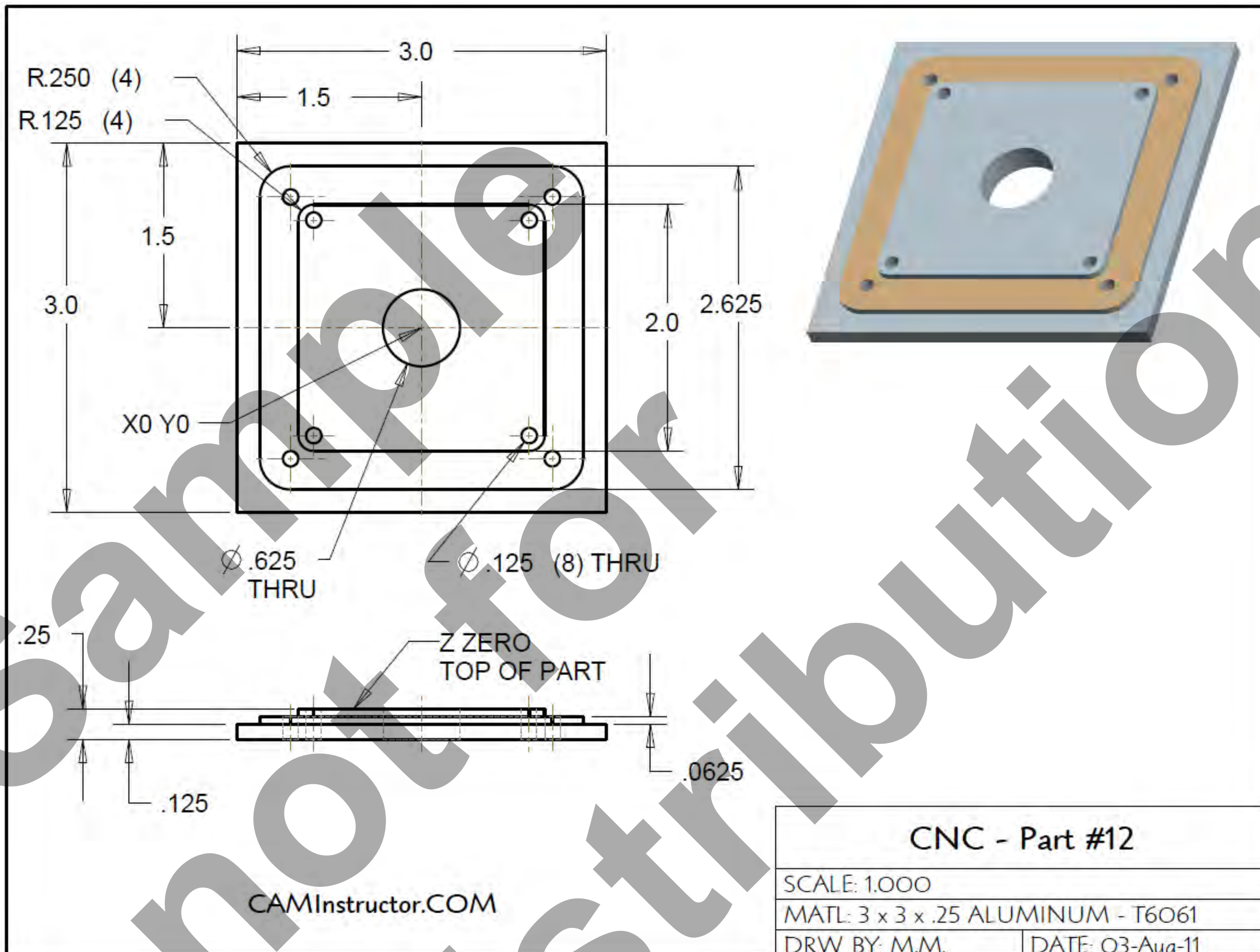


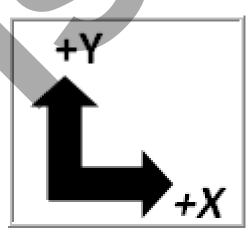
%
O003
N10 G20
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)
N30




CNC - Part #12
SCALE: 1.000
MATL: 3 x 3 x .25 ALUMINUM - T6061
DRW BY: M.M.
DATE: 03-Aug-11
2.625
2.0
3.0
1.5
R.250 (4)
R.125 (4)
Ø .125 (8) THRU
Ø .625 THRU
X0 Y0
Z ZERO
TOP OF PART
.0625
.125
.25
CAMInstructor.COM

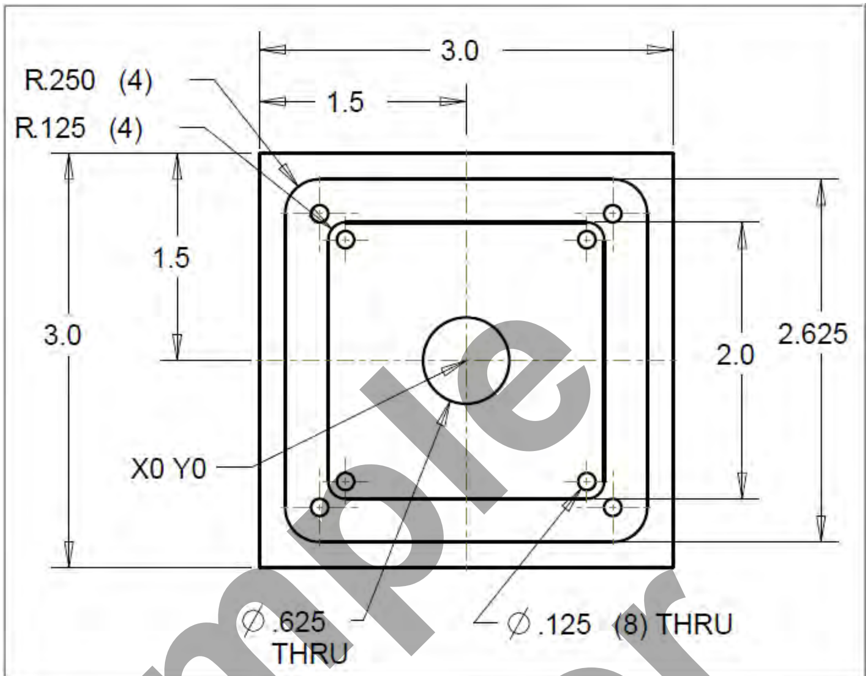
- ***The Machining Process***
- ***Ø.500" Endmill - Tool # 4- 3050 RPM Feedrate 20 IPM***
  - Machine the profiles with the .125" and .25 radii.
  - Machine the circular 0.625" diameter through hole. Sink to depth at center and use circular interpolation to finish the bore.
- ***Ø.125" Endmill - Tool # 1- 5000 RPM Feedrate 7 IPM***
  - Drill the Ø .125" holes through the part (8 places)
  - **Note:** The Ø.125" Endmill is designed for center cutting machining. No center drilling or pilot hole is required.

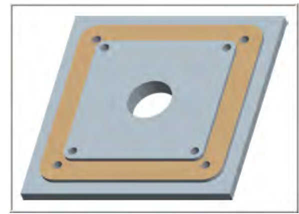


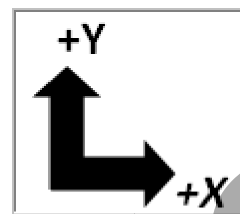


LESSON-9 - CNC - PART #12

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART***
  - X0Y0 is at the center of the part
  - Use a 0.5” diameter End Mill Tool # 4 and a 0.125” diameter End Mill Tool # 1





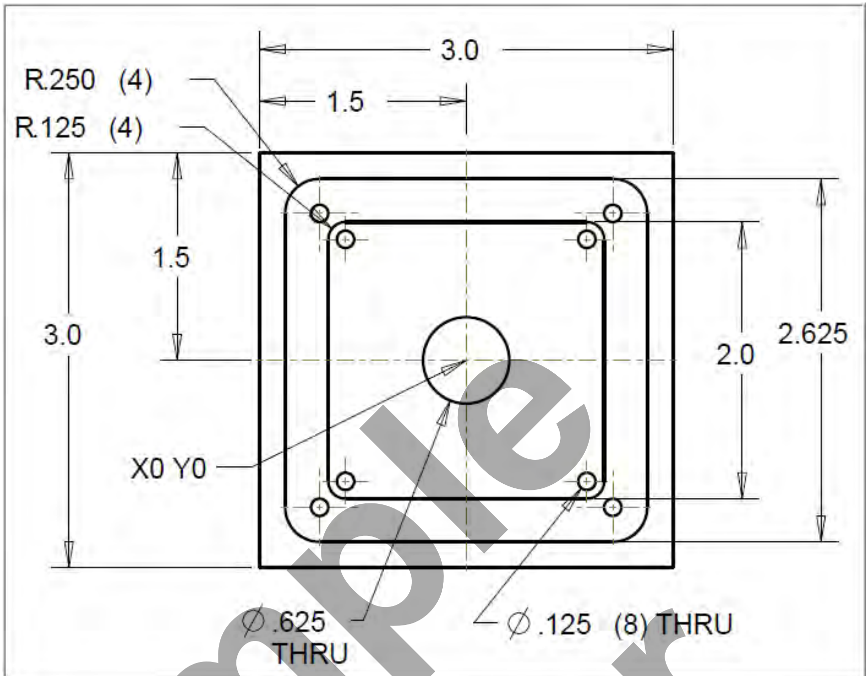


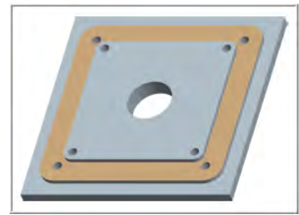
- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE HOLES***

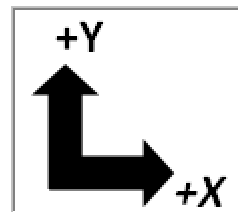
G90	X	Y
1	0	0
2		
3		
4		
5		
6		
7		
8		
9		

LESSON-9 - CNC - PART #12

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART***
  - X0Y0 is at the center of the part
  - Use a 0.5” diameter End Mill Tool # 4 and a 0.125” diameter End Mill Tool # 1
  - Climb mill





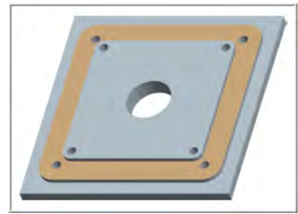


- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS OF THE 0.5’ DIAMETER END MILL***

2.625” Square Coordinates			2.0” Square Coordinates		
G90	**X**	**Y**	**G90**	**X**	**Y**
1			**1**		
2			**2**		
3			**3**		
4			**4**		
5			**5**		
6			**6**		
7			**7**		
8			**8**		
9			**9**		

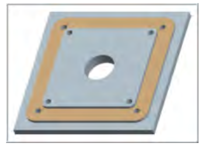
LESSON-9 - CNC - PART #12

- ***CREATE THE PROGRAM TO MACHINE THE PART***
  - Use a 0.5" diameter End Mill Tool # 4
  - Speed = 3050 Feed rate =20 IPM
  - Use a 0.125" diameter End Mill Tool # 1
  - Speed = 5000 Feed rate =7 IPM
  - X0Y0 is at the center of the part
  - Z=0 is the top of the part.
  - Climb mill
  - Type up your program and check it for correctness using NCPlot

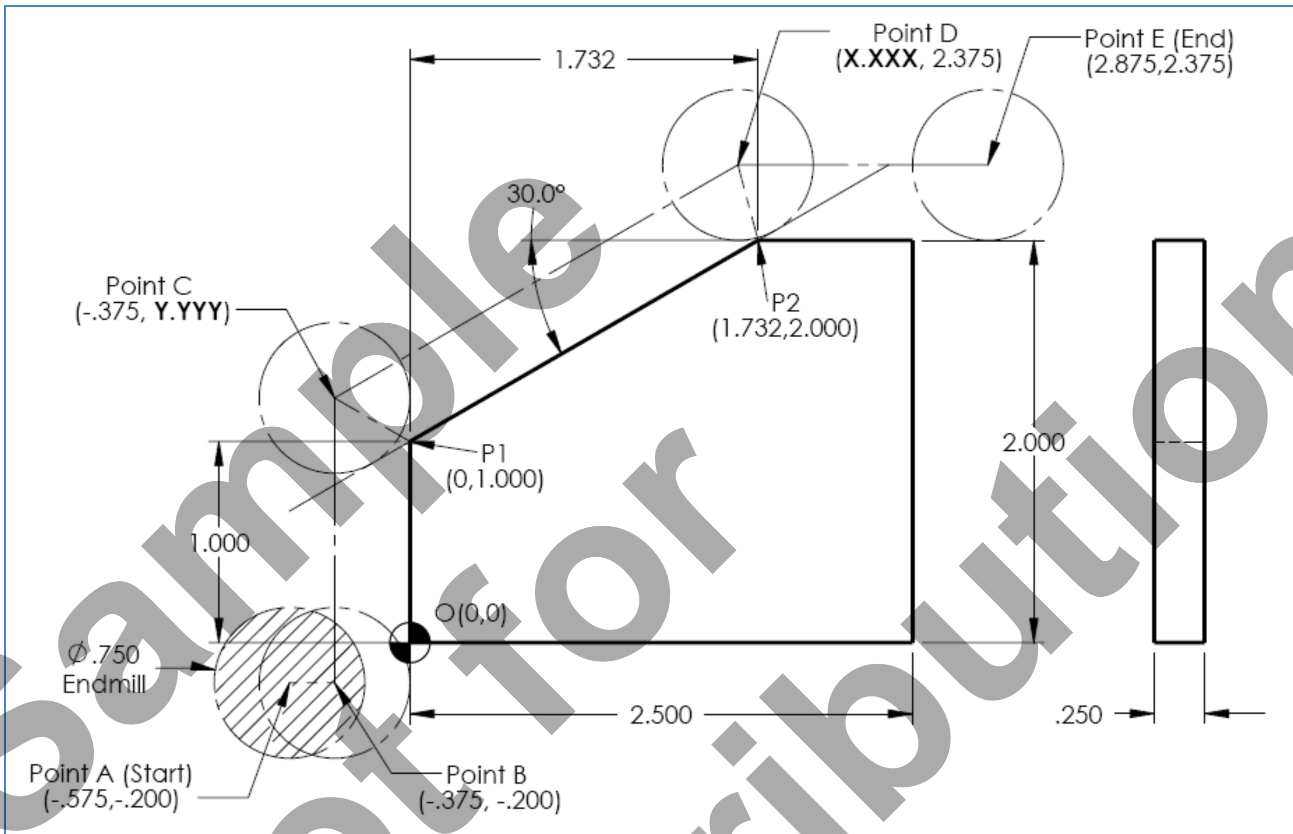


%
O003
N10 G20
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)
N30

LESSON-9 - CNC - PART #12 - Continued



CNC PROGRAMMING WORKBOOK



LESSON-10

CUTTER COMPENSATION

camInstructor

LESSON-10 - G40, G41, & G42 CUTTER COMPENSATION

Cutter Compensation is used to offset the center of the cutter and move the cutter either to the left or right the distance of the cutter radius. When cutting angled geometry, substantial computations are required to determine the center of the cutter. Using Cutter Compensation, you can program the part as if the center of the cutter will be travelling along the geometry.

G40 CUTTER COMPENSATION CANCEL

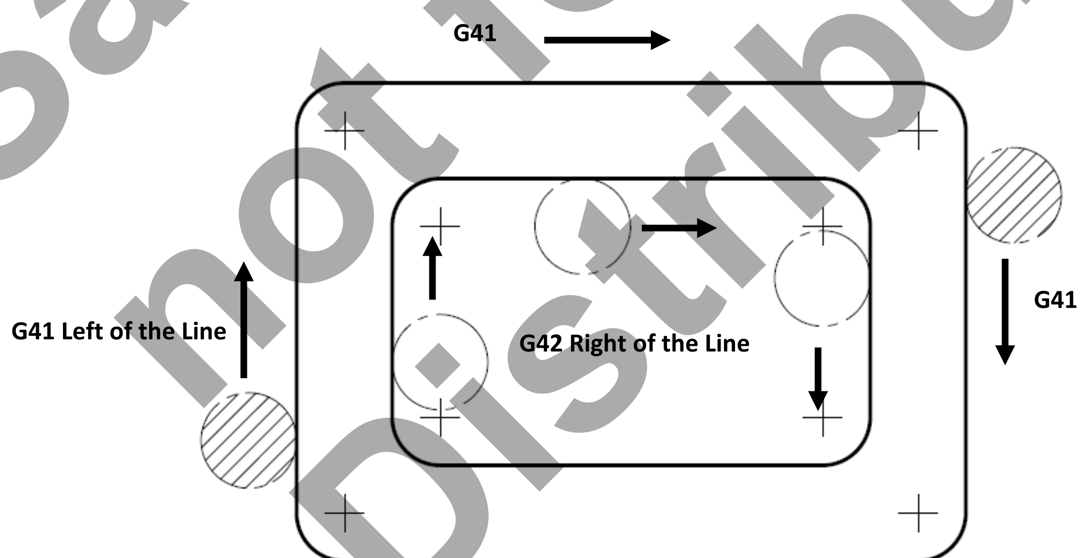
G40 will cancel the G41 or G42 cutter compensation commands.

G41 CUTTER COMPENSATION LEFT

G41 will action cutter compensation left. The tool is moved to the left of the programmed path to compensate for the radius of the tool. A **Dnn** must also be programmed to select the correct tool size from the DIAMETER/RADIUS offset display register.

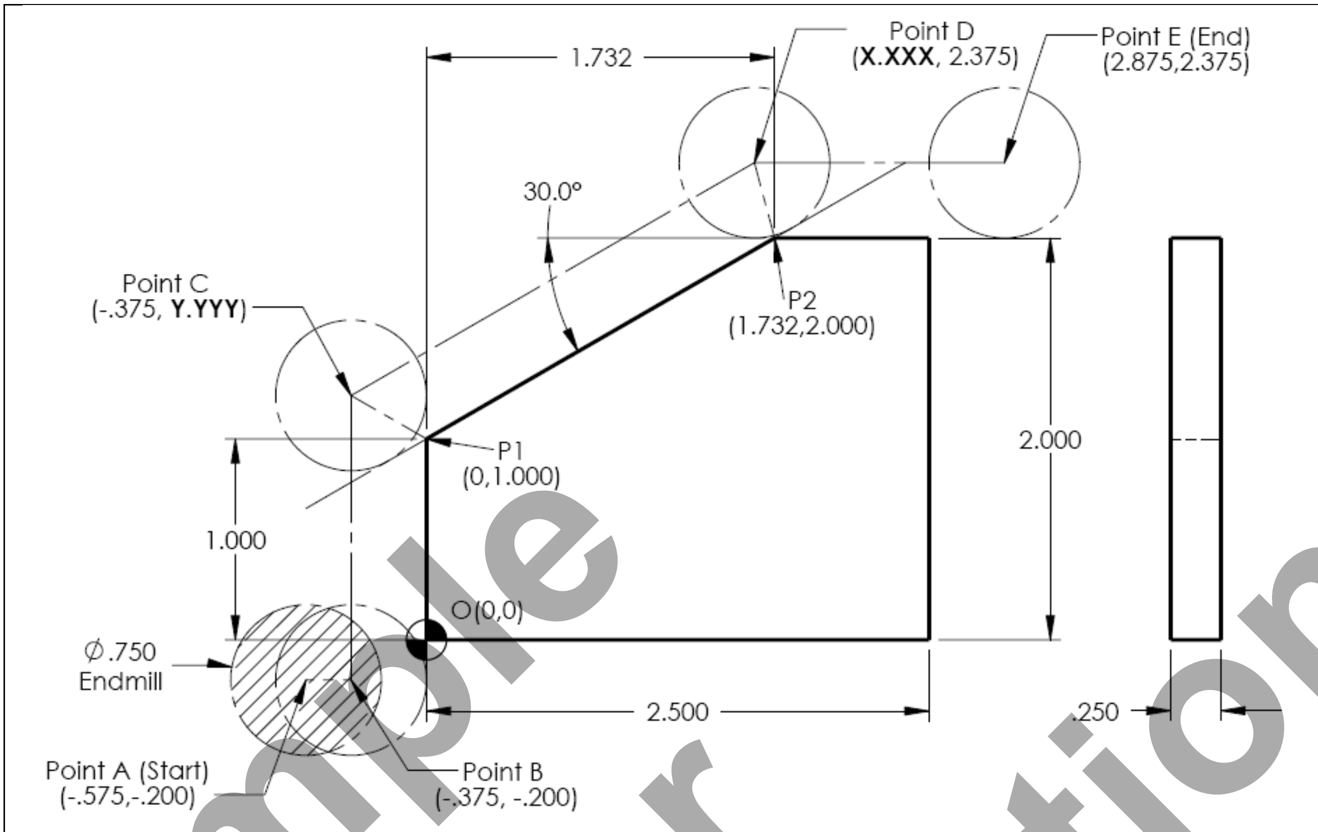
G42 CUTTER COMPENSATION RIGHT

G42 will action cutter compensation right. The tool is moved to the right of the programmed path to compensate for the size of the tool.



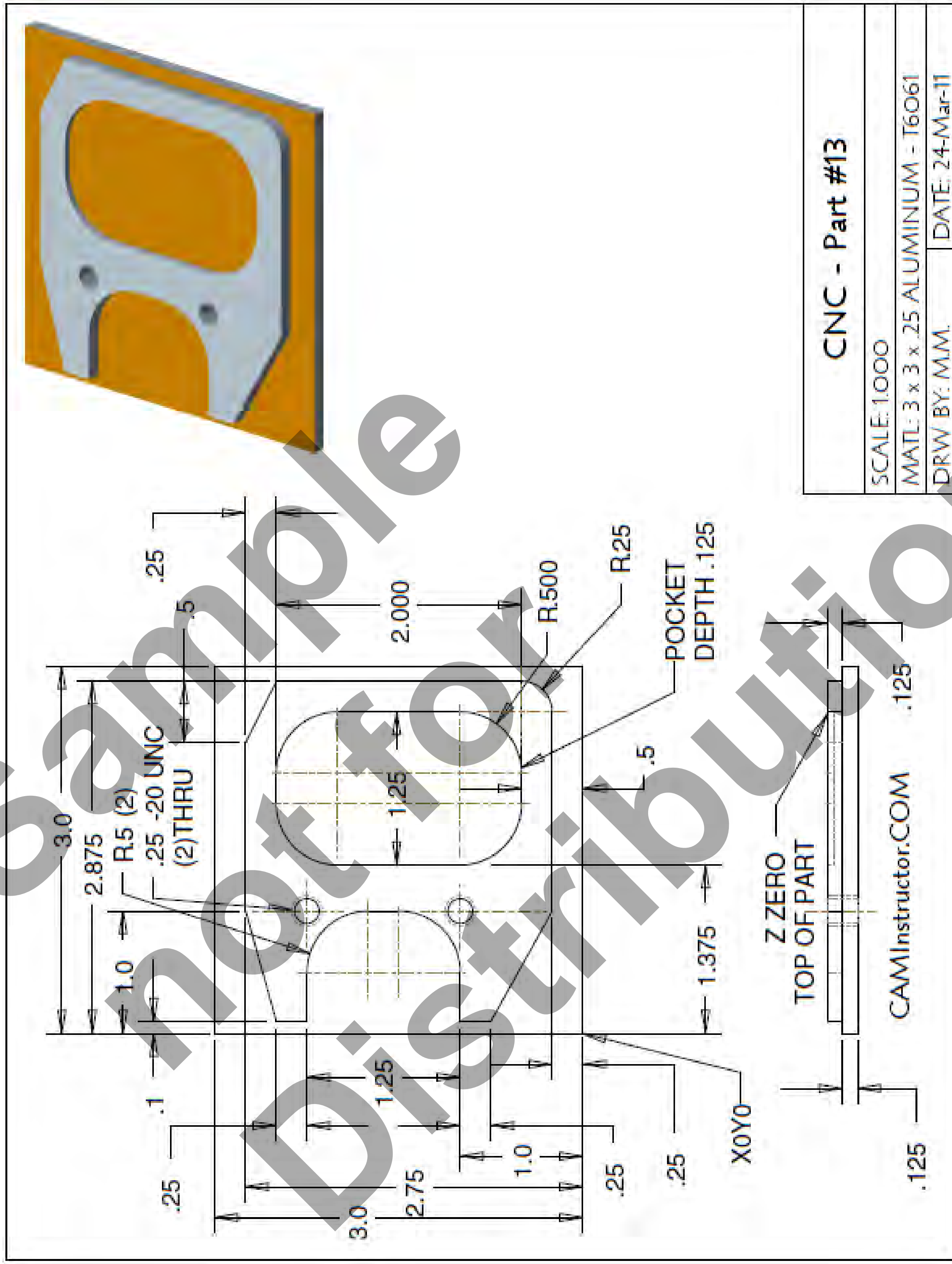
G41 : Tool is moved to the left of the profile (e.g. Outside Profile shown as above)
G42 : Tool is moved to the right of the profile (e.g. Inside Profile shown as above)

LESSON-10 - **G40, G41, & G42** CUTTER COMPENSATION



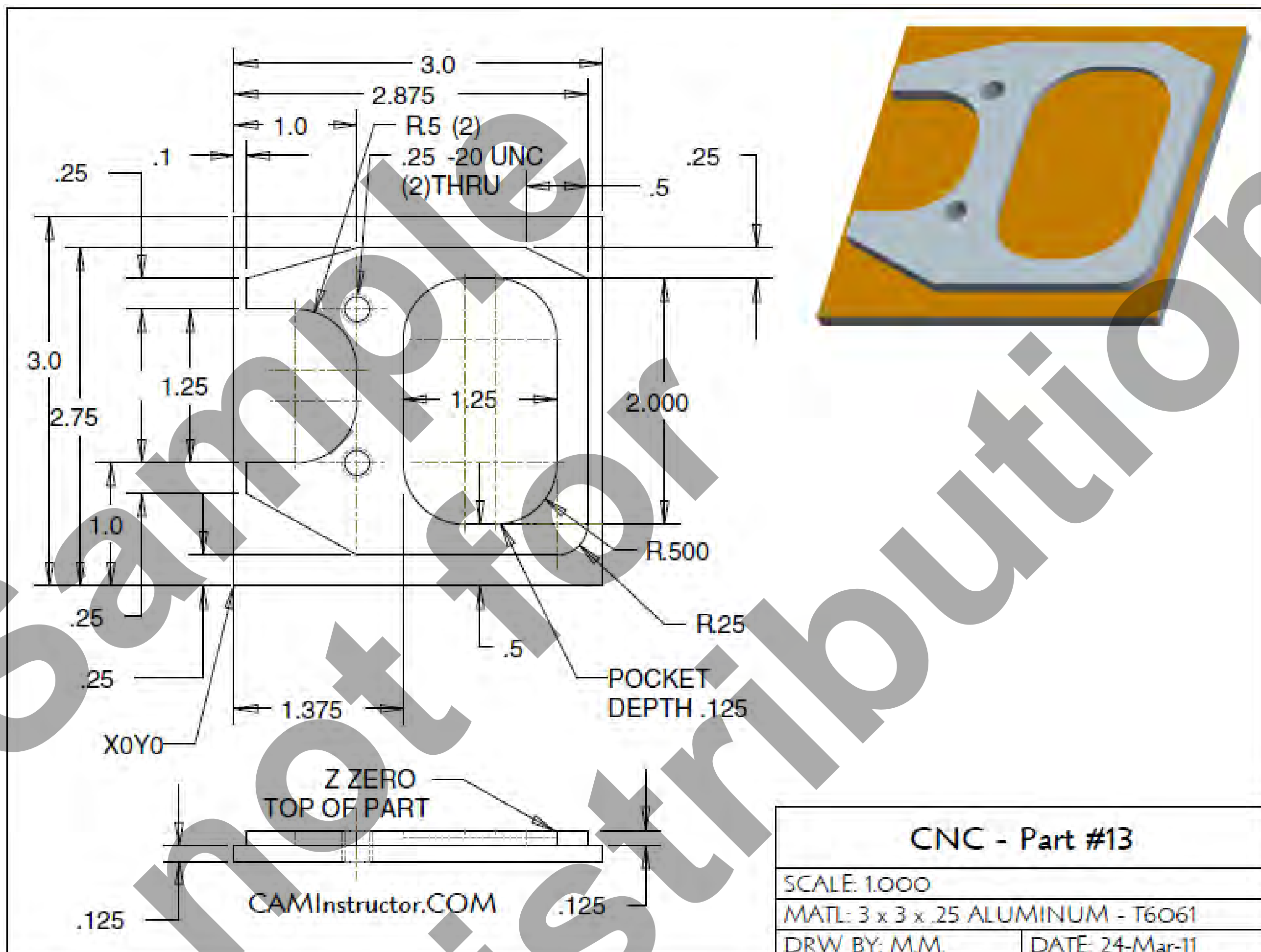
Without Cutter Compensation	Using Cutter Compensation
%	%
O10011	O10012
N1 T12 M06 (3/4" Flat Endmill)	N1 T12 M06 (3/4" Flat Endmill)
N2 G90 G54 G00 X-.575 Y-.200 S1500 M03	N2 G90 G54 G00 X-.575 Y-.200 S1500 M03
N3 G43 H12 Z2.	N3 G43 H12 Z2.
N4 Z.1	N4 Z.1
N5 G01 Z-.25 F10.	N5 G01 Z-.25 F10.
N6 X-.375 (Point B)	**N6 G41 X0. D12** (Origin)
N7 Y?.???? (Point C)	N7 Y1.0 (P1)
N8 X?.???? Y2.375 (Point D)	N8 X1.732 Y2.0 (P2)
N9 X2.875 (Point E)	N9 X2.875
N10 M05	**N10 G40 M05**
N11 G00 Z2.	N11 G00 Z2.
N12 G91 G28 Z0. (Machine Home Z-Axis)	N12 G91 G28 Z0. (Machine Home Z-Axis)
N13 G28 X0. Y0. (Machine Home X,Y-Axis)	N13 G28 X0. Y0. (Machine Home X,Y-Axis)
N14 M30	N14 M30
%	%

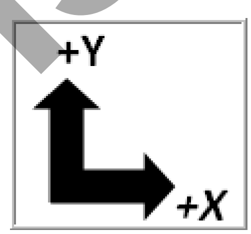
The Cutter Compensation (G41) will reflect the actual part geometry (O, P1, P2..) instead of Cutter Centers (Point B,C,D & E)


CNC - Part #13
SCALE: 1.000
MAT'L: 3 x 3 x .25 ALUMINUM - T6061
DRW BY: M.M.
DATE: 24-Mar-11
CAMInstructor.COM
POCKET
DEPTH .125
Z ZERO
TOP OF PART
X0Y0
.25 -20 UNC
(2)THRU
R.5 (2)
R.500
R.25
2.000
1.25
1.375
2.875
3.0
2.75
1.0
.125
.25
.5
.1

LESSON-10 - CNC - PART #13

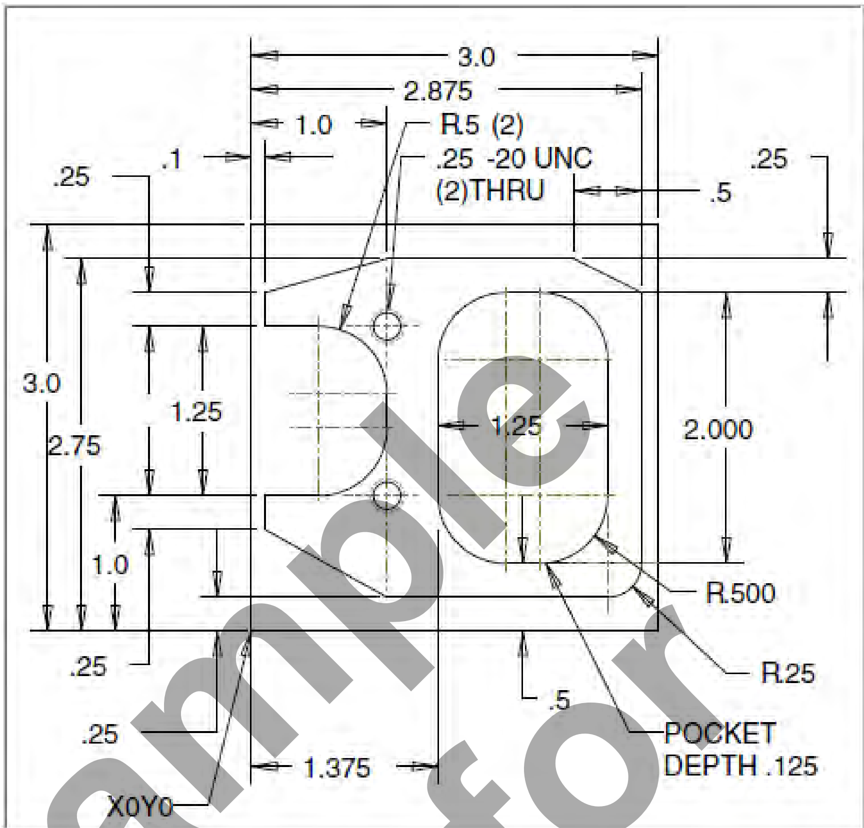
- ***The Machining Process***
- ***Ø.750" Endmill - Tool # 6***
  - Machine the profile and pocket using cutter compensation
  - 0.750" diameter end mill Spindle Speed = 2100 Feed rate = 25 IPM
- ***Ø.375" Spot Drill - Tool # 6***
  - Spot Drill Spindle Speed = 2750 Feed rate = 11 IPM
- ***Ø.201" Drill - Tool # 8***
  - 0.201" diameter Drill Spindle Speed = 4500 Feed rate = 15 IPM
- ***Ø.25"-20 UNC Tap - Tool # 9***
  - 0.250-20 UNC Tap Spindle Speed = 1000 Feed rate = 50 IPM

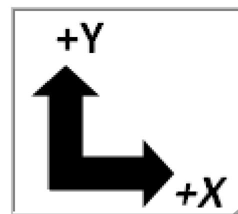




LESSON-10 - CNC - PART #13

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART***
  - X0Y0 is at the lower left corner of the part



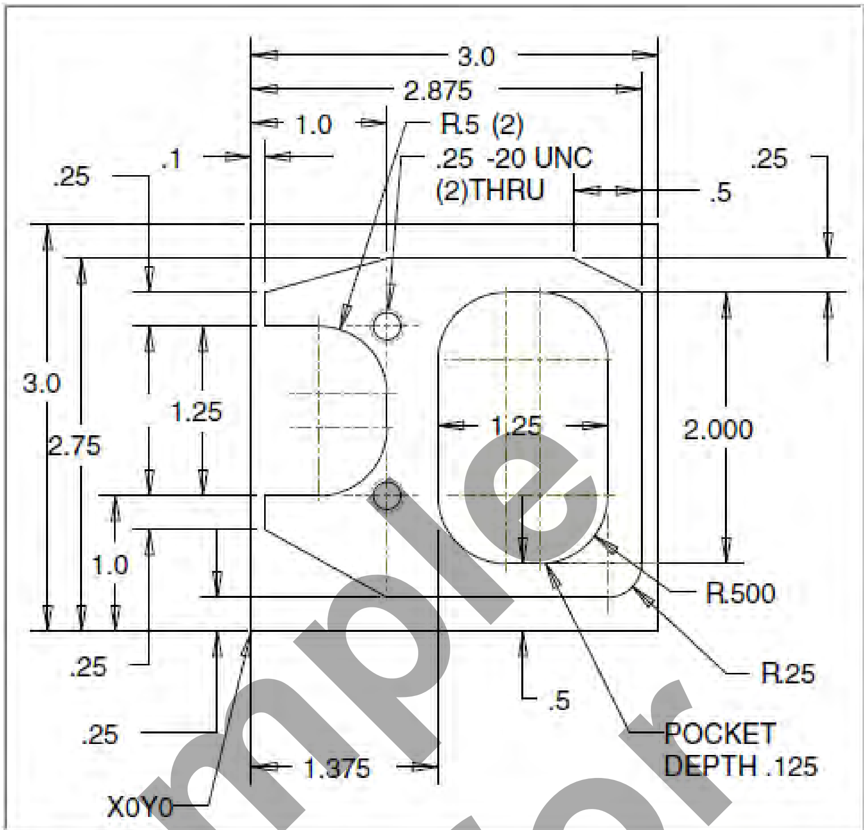


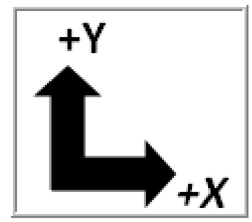
- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE TWO HOLES***

G90	X	Y
1		
2		

LESSON-10 - CNC - PART #13

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART - CONTOUR***
  - X0Y0 is at the lower left of the part



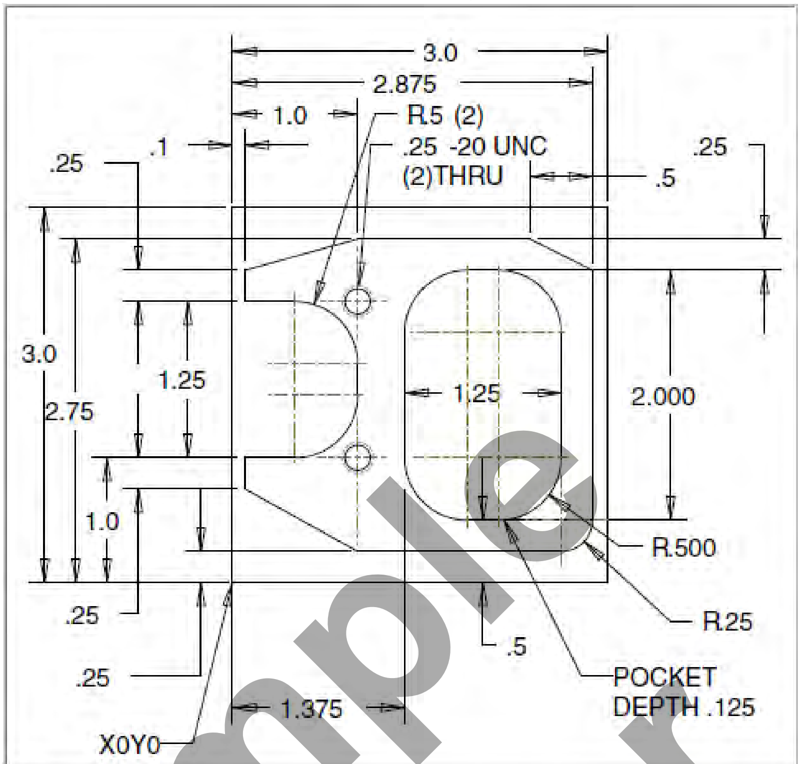


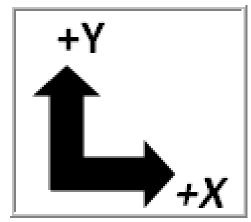
- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS OF THE 0.75' DIAMETER END MILL TO MACHINE THE CONTOUR***

G90	X	Y	G90	X	Y
1			12		
2			13		
3			14		
4			15		
5			16		
6			17		
7					
8					
9					
10					
11					

LESSON-10 - CNC - PART #13

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART - POCKET***
  - X0Y0 is at the lower left of the part





- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS OF THE 0.75' DIAMETER END MILL TO MACHINE THE POCKET***

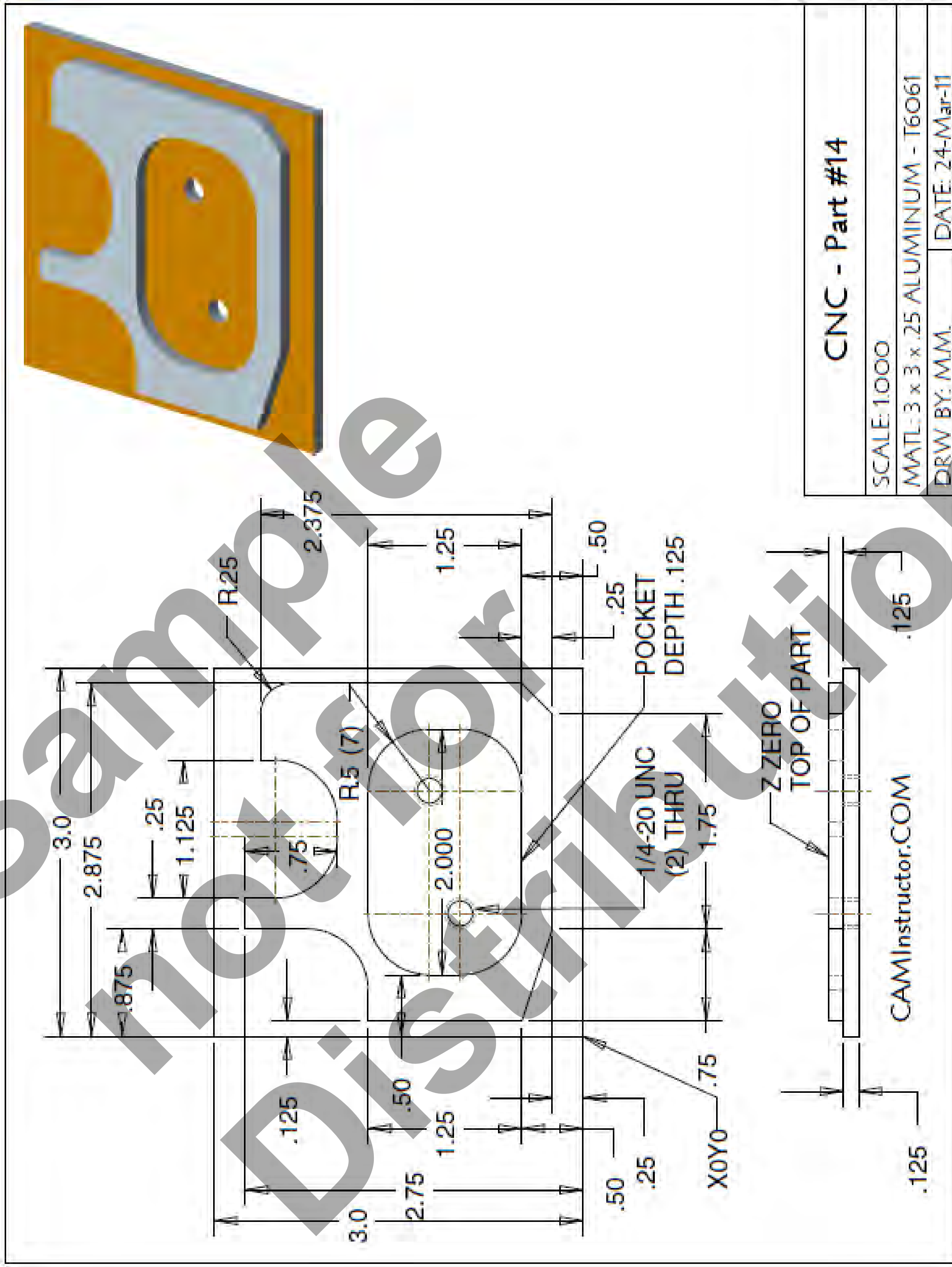
G90	X	Y
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		

LESSON-10 - CNC - PART #13

- ***CREATE THE PROGRAM TO MACHINE THE PART***
- ***The Machining Process***
- ***Ø.750" Endmill - Tool # 6***
  - Machine the profile and pocket using cutter compensation
  - 0.750" diameter end mill Spindle Speed = 2100 Feed rate = 25 IPM
- ***Ø.375" Spot Drill - Tool # 6***
  - Spot Drill Spindle Speed = 2750 Feed rate = 11 IPM
- ***Ø.201" Drill - Tool # 8***
  - 0.201" diameter Drill Spindle Speed = 4500 Feed rate = 15 IPM
- ***Ø.25"-20 UNC Tap - Tool # 9***
  - 0.201" diameter Drill Spindle Speed = 1000 Feed rate = 50 IPM

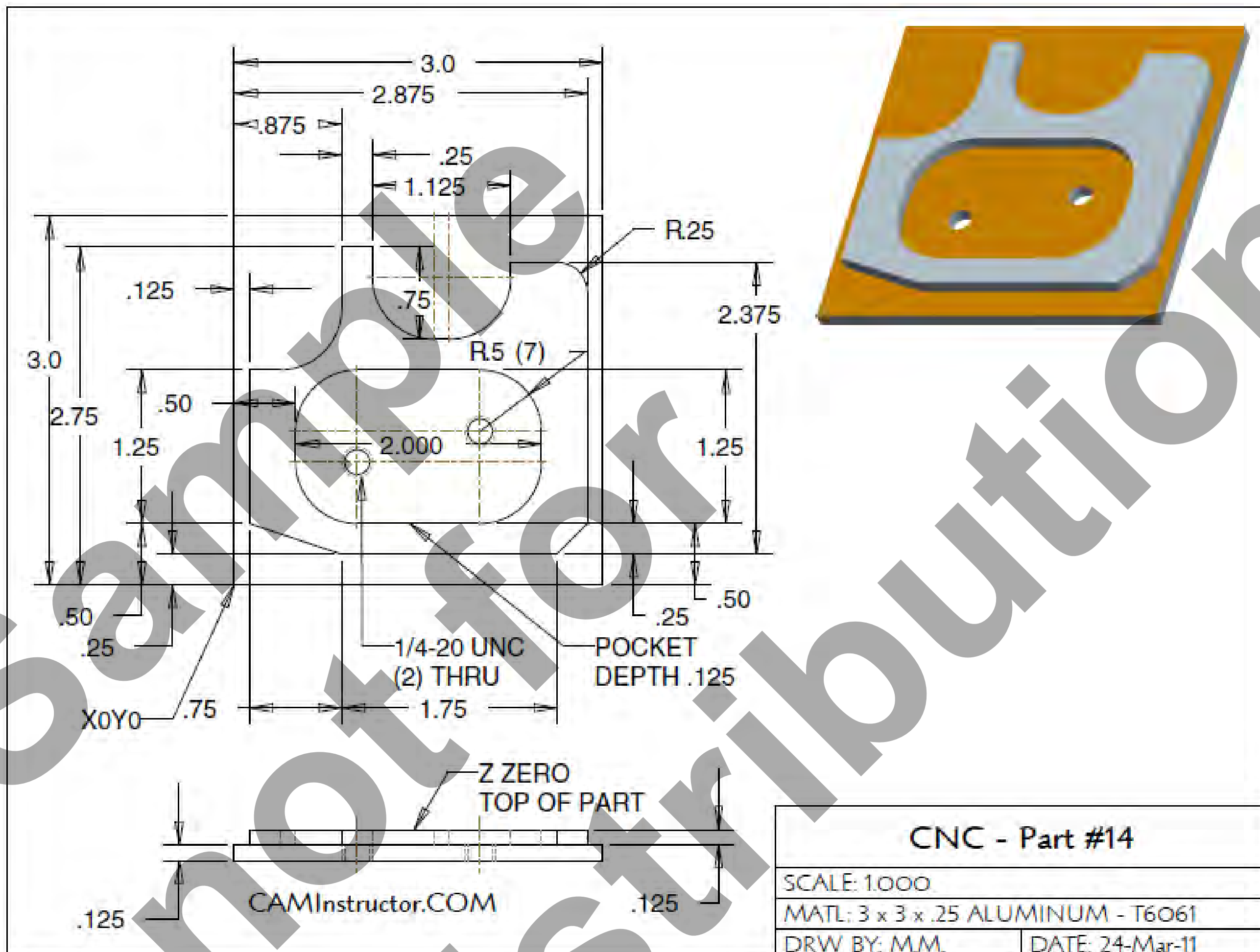
%
O888
N10 G20
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)
N30

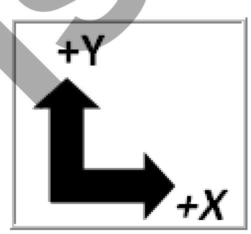
LESSON-10 - CNC - PART #13 - Continued


CNC - Part #14
SCALE: 1.000
MATL: 3 x 3 x .25 ALUMINUM - T6061
DRW BY: M.M.
DATE: 24-Mar-11
R.25
R.5 (7)
POCKET
DEPTH .125
1/4-20 UNC
(2) THRU
Z ZERO
TOP OF PART
X0Y0
CAMInstructor.COM

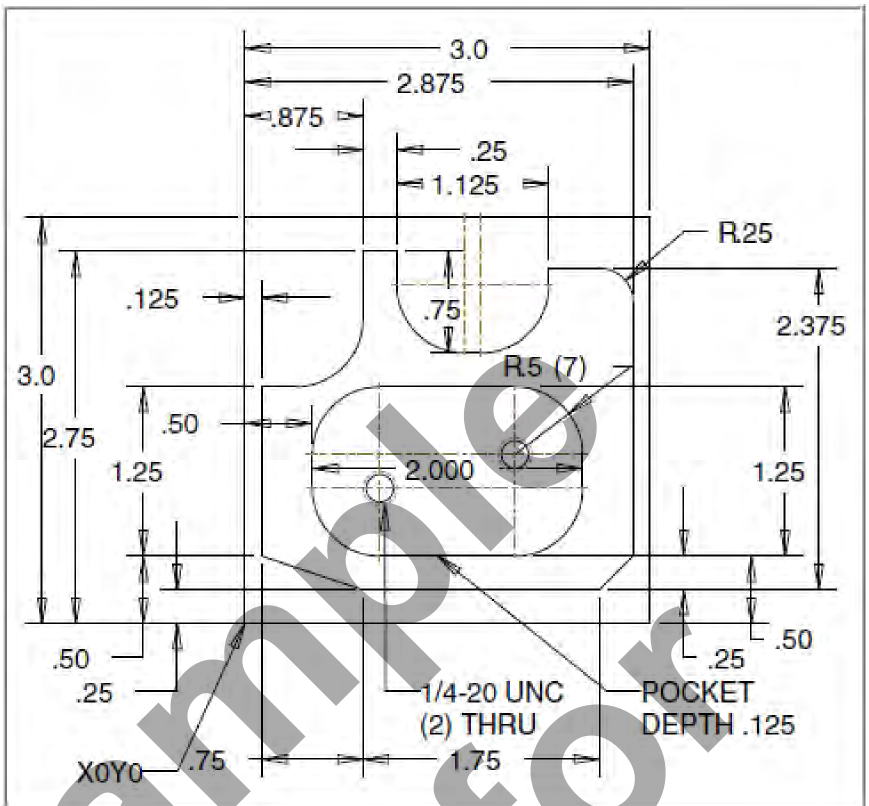
LESSON-10 - CNC - PART #14

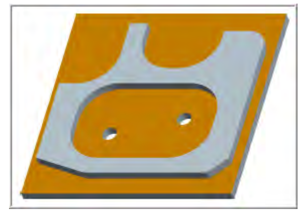
- ***The Machining Process***
- ***Ø.750" Endmill - Tool # 6***
  - Machine the profile and pocket using cutter compensation
  - 0.750" diameter end mill Spindle Speed = 2100 Feed rate = 25 IPM
- ***Ø.375" Spot Drill - Tool # 6***
  - Spot Drill Spindle Speed = 2750 Feed rate = 11 IPM
- ***Ø.201" Drill - Tool # 8***
  - 0.201" diameter Drill Spindle Speed = 4500 Feed rate = 15 IPM
- ***Ø.25"-20 UNC Tap - Tool # 9***
  - 0.201" diameter Drill Spindle Speed = 1000 Feed rate = 50 IPM

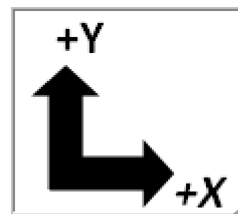




- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART***
  - X0Y0 is at the lower left of the part

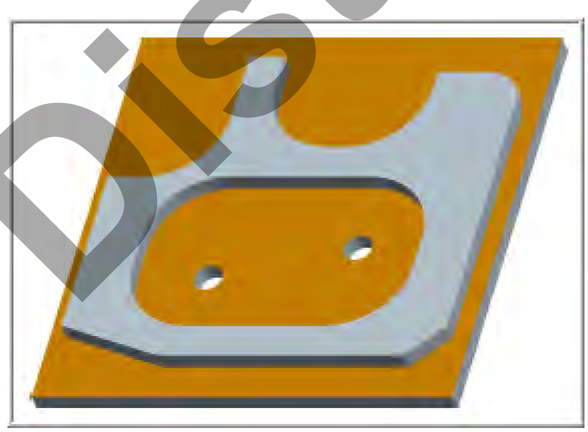






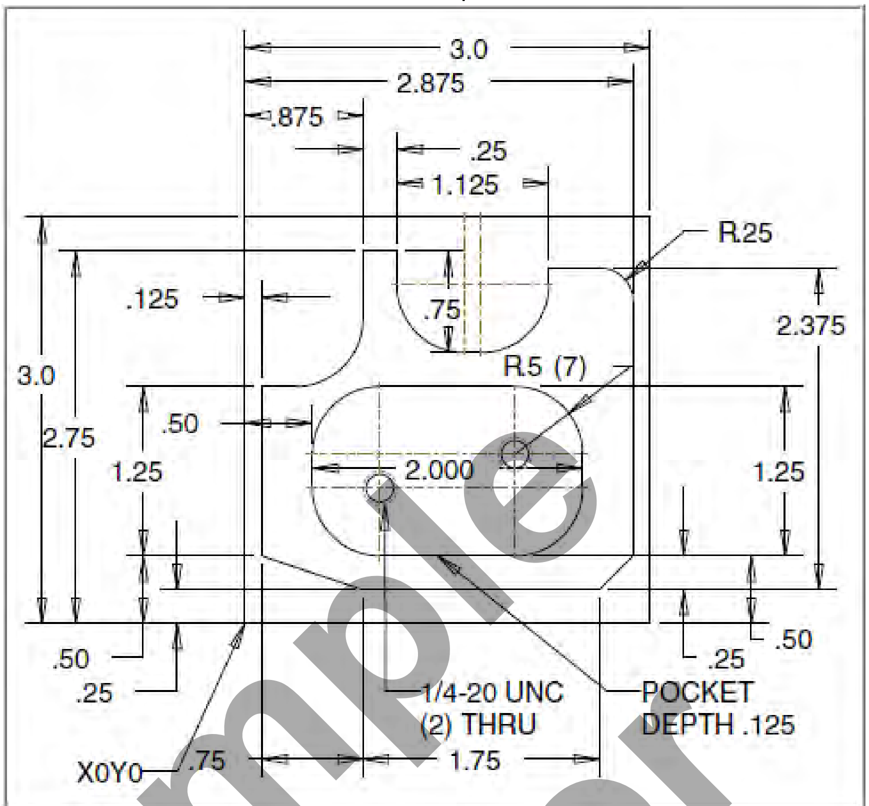
- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE TWO HOLES***

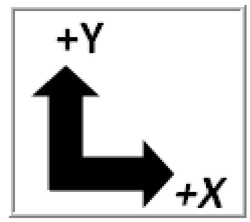
G90	X	Y
1		
2		



LESSON-10 - CNC - PART #14

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART - CONTOUR***
  - X0Y0 is at the lower left of the part

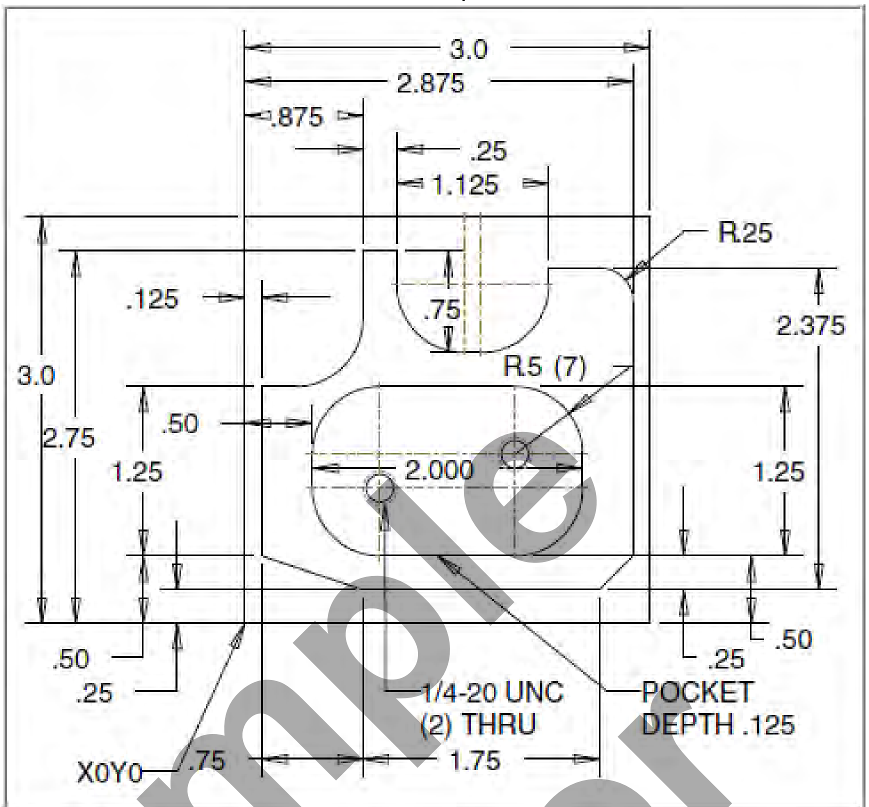


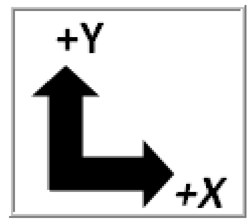


- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS OF THE 0.75' DIAMETER END MILL TO MACHINE CONTOUR***

G90	X	Y	G90	X	Y
1			**12**		
2			**13**		
3			**14**		
4			**15**		
5			**16**		
6			**17**		
7					
8					
9					
10					
11					

- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS TO MACHINE THE PART - POCKET***
  - X0Y0 is at the lower left of the part

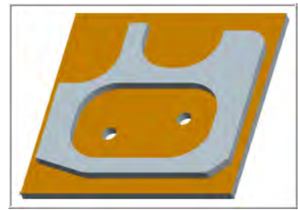




- ***WORK OUT THE ABSOLUTE X AND Y COORDINATES FOR THE VARIOUS POSITIONS OF THE 0.75' DIAMETER END MILL TO MACHINE POCKET***

G90	X	Y
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		

- ***CREATE THE PROGRAM TO MACHINE THE PART***
- ***The Machining Process***
- ***Ø.750" Endmill - Tool # 6***
  - Machine the profile and pocket using cutter compensation
  - 0.750" diameter end mill Spindle Speed = 2100 Feed rate = 25 IPM
- ***Ø.375" Spot Drill - Tool # 6***
  - Spot Drill Spindle Speed = 2750 Feed rate = 11 IPM
- ***Ø.201" Drill - Tool # 8***
  - 0.201" diameter Drill Spindle Speed = 4500 Feed rate = 15 IPM
- ***Ø.25"-20 UNC Tap - Tool # 9***
  - 0.201" diameter Drill Spindle Speed = 1000 Feed rate = 50 IPM



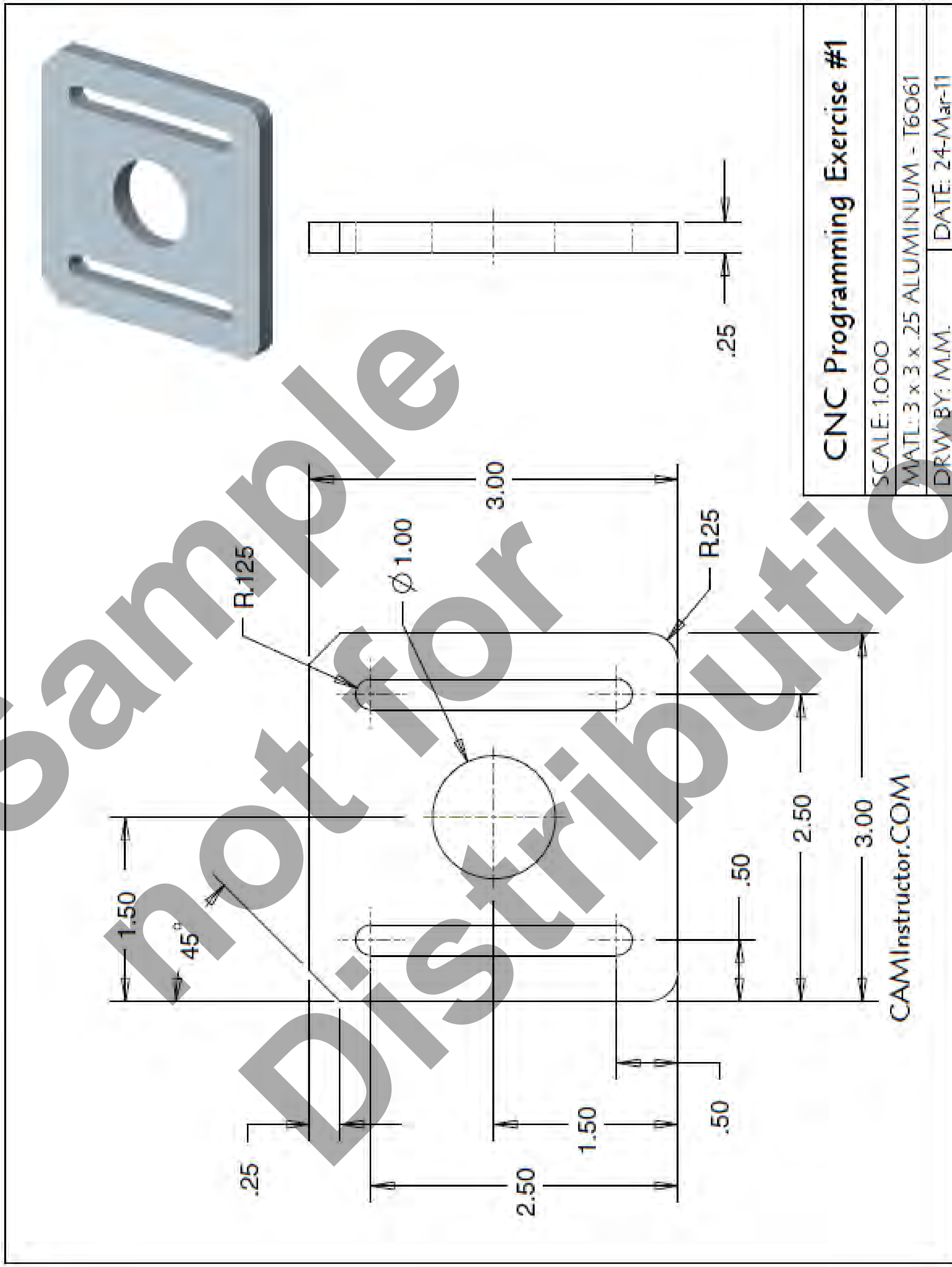
%
O003
N10 G20
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)
N30

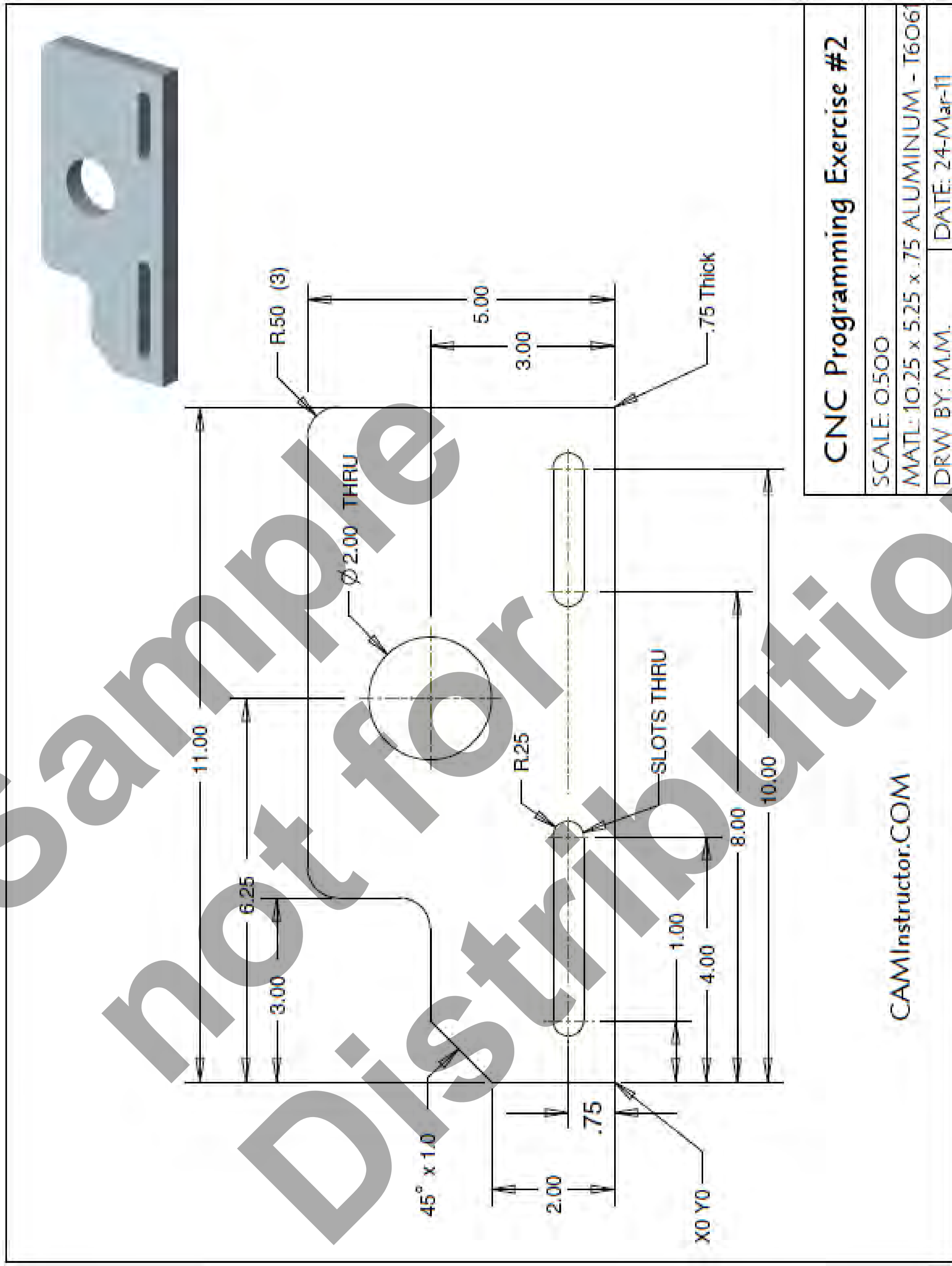
CNC PROGRAMMING WORKBOOK

APPENDIX

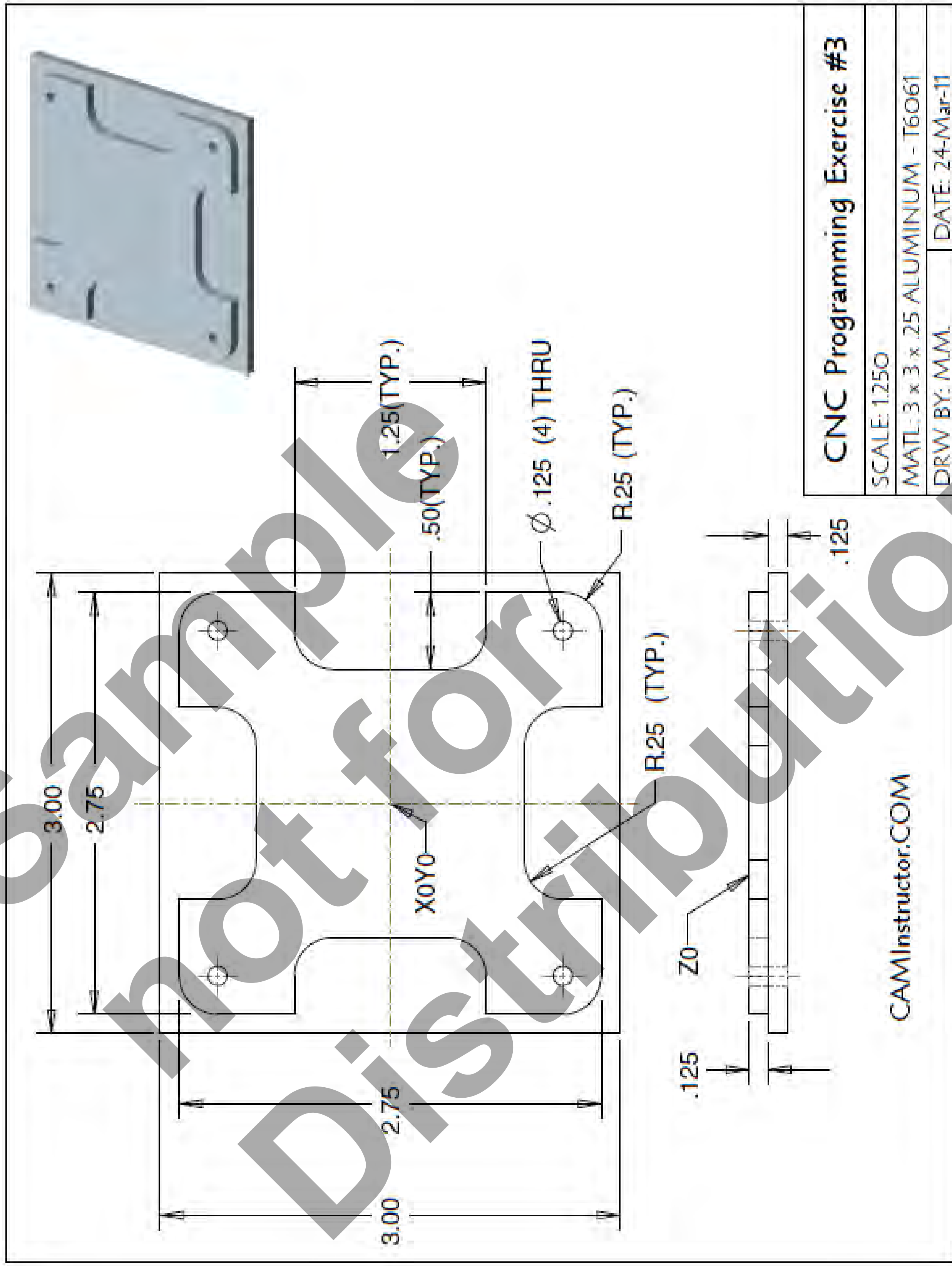
camInstructor

Exercise #1

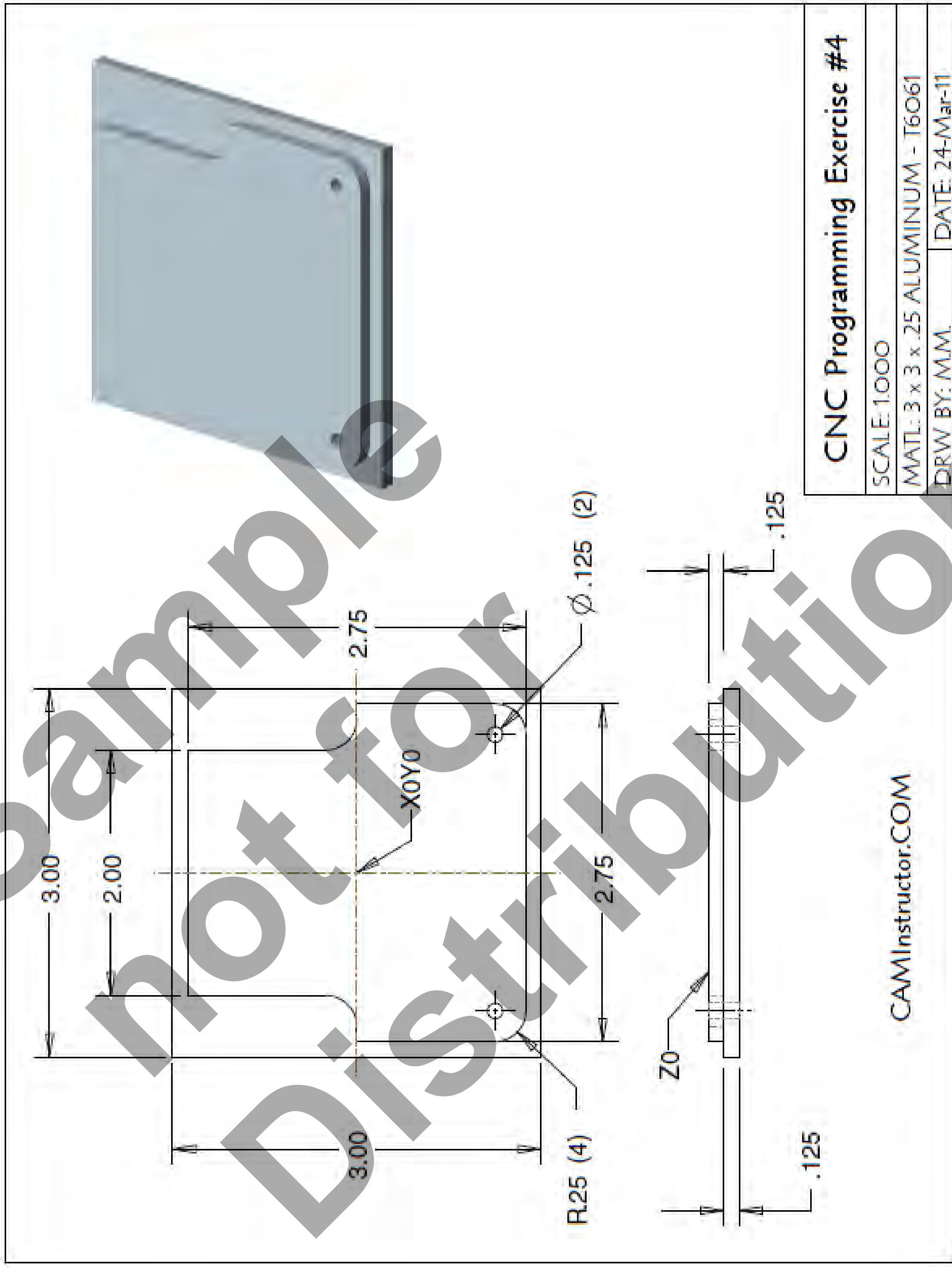

CNC Programming Exercise #1
SCALE: 1.000
MATL: 3 x 3 x .25 ALUMINUM - T6061
DRWN BY: M.M.
DATE: 24-Mar-11
CAMInstructor.COM
3.00
R.125
⌀1.00
R.25
.25
1.50
45°
2.50
3.00
.50
1.50
.50
.25
2.50


CNC Programming Exercise #2
SCALE: 0.500
MATL: 10.25 x 5.25 x .75 ALUMINUM - T6O6
DRW BY: M.M.
DATE: 24-Mar-11
CAMInstructor.COM
R.50 (3)
Ø2.00 THRU
R.25
SLOTS THRU
.75 Thick
X0 Y0
45° x 1.0
5.00
3.00
11.00
6.25
3.00
1.00
4.00
8.00
10.00
.75
2.00

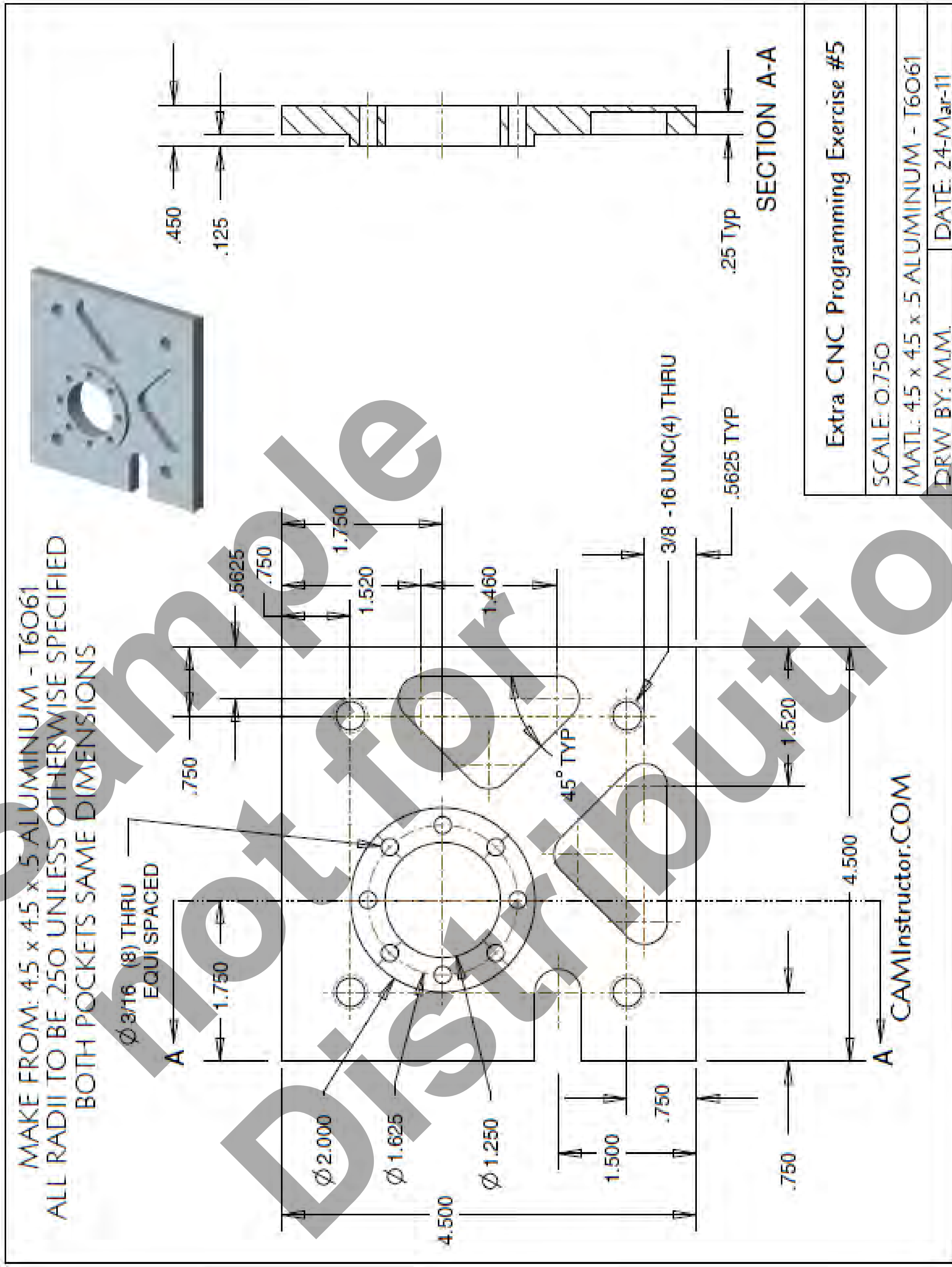
Extra CNC Programming Exercise #3



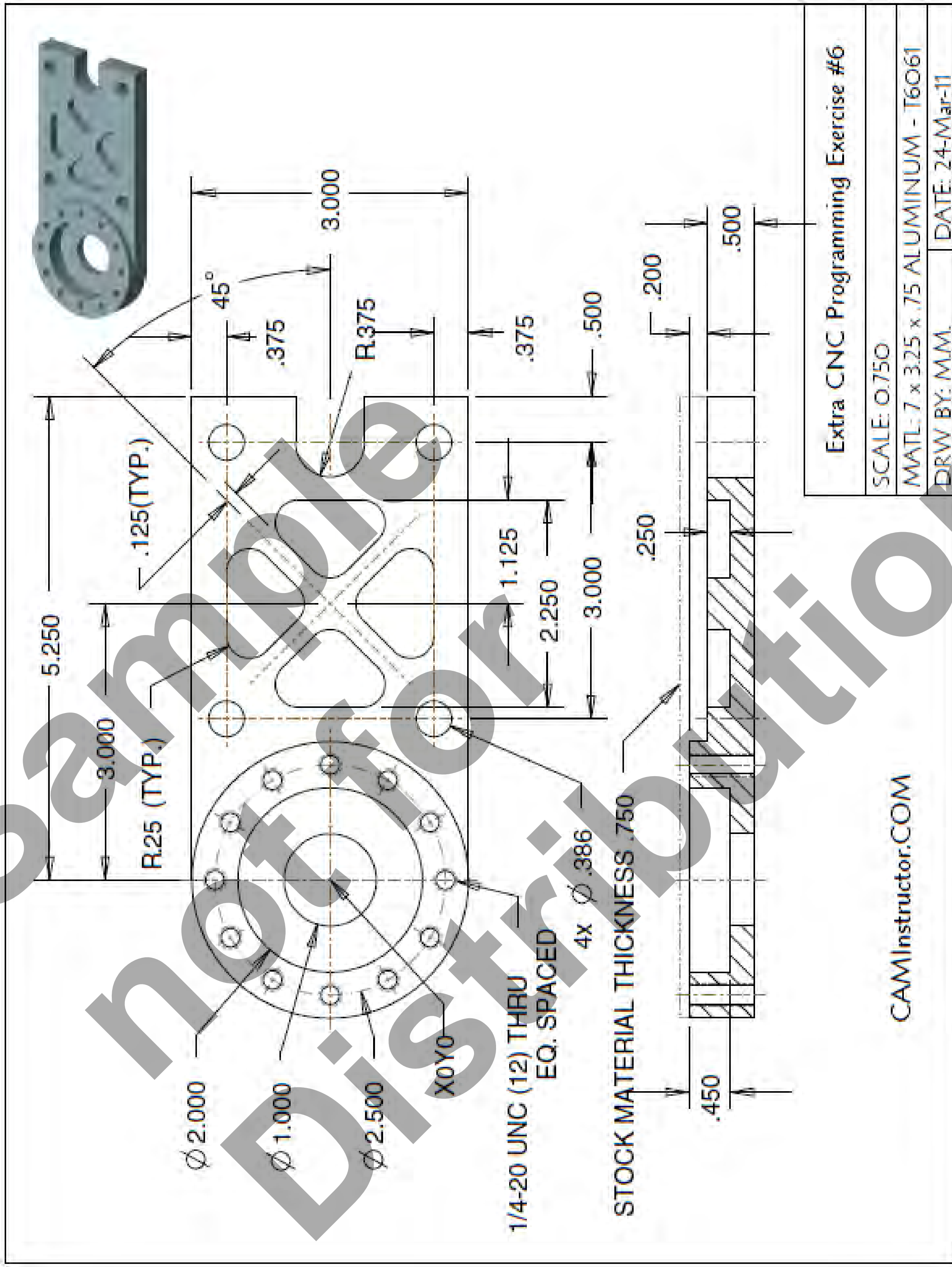
Extra CNC Programming Exercise #4



Extra CNC Programming Exercise #5



Extra CNC Programming Exercise #6

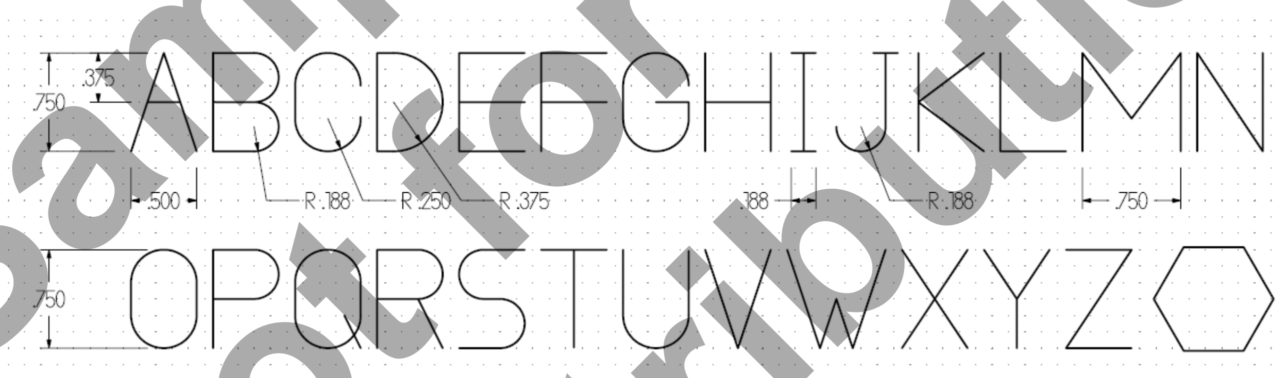


Extra CNC Programming Exercise #7

Instructions:

1. Create your own design.
2. The material size: 6” x 1.5” x .125” Aluminum
3. The part is held in the vise.
4. Locate the part flush with the left hand side of the vise jaw.
5. X0 Y0 is the top left hand corner of the material.
6. Z0 is the top of the material.
7. Center Drill is used as an engraving tool to machine the letters.
8. Spindle Speed: 5000 rpm
9. Feedrate: 12 in/min
10. Depth of Cut : -0.025”
11. Center Drill 2 mounting holes using Canned Cycle
12. Depth of Cut: -0.25
13. Minimum of five letters
14. You can use the suggested letter shapes & size or create your own lettering design.

Suggested Lettering:



Appendix – Preparatory Functions – G-Codes

G00 RAPID POSITIONING MOTION
G01 LINEAR INTERPOLATION MOTION
G02 CIRCULAR INTERPOLATION MOTION - CLOCKWISE
G03 CIRCULAR INTERPOLATION MOTION - COUNTECLOCKWISE
G04 DWELL
G09 EXACT STOP
G10 PROGRAMMABLE OFFSET SETTING
G12 CW CIRCULAR POCKET MILLING
G13 CCW CIRCULAR POCKET MILLING
G17 CIRCULAR MOTION XY PLANE SELECTION (G02 or G03)
G18 CIRCULAR MOTION ZX PLANE SELECTION (G02 or G03)
G19 CIRCULAR MOTION YZ PLANE SELECTION (G02 or G03)
G20 VERIFY INCH COORDINATE POSITIONING
G21 VERIFY METRIC COORDINATE POSITIONING
G28 MACHINE ZERIO RETURN THRU REF. POINT
G29 MOVE TO LOCATION THROUGH G28 REF. POINT
G31 FEED UNTIL SKIP FUNCTION
G35 AUTOMATIC TOOL DIAMETER MEASUREMENT
G36 AUTOMATIC WORK OFFSET MEASUREMENT
G37 AUTOMATIC TOOL LENGTH MEASUREMENT
G40 CUTTER COMPENSATION CANCEL G41/G42/G141
G41 2D CUTTER COMPENSATION, LEFT (X, Y, D)
G42 2D CUTTER COMPENSATION, RIGHT (X, Y, D)
G43 TOOL LENGTH COMPESATION POSITIVE (H, Z)
G44 TOOL LENGTH COMPENATION NEGATIVE (H, Z)

G47 TEXT ENGRAVING (X, Y, Z, R, I, J, P, E, F)
G49 TOOL LENGTH COMPENSATION CANCEL G43/G44/G143)
G50 SCALING G51 CANCEL
G51 SCALING (X, Y, Z, P)
G52 WORK OFFSET COORDINATE POSITING
G52 GLOBAL WORK COORDINATE OFFSET SHIFT
G52 GLOBAL WORK COORDINATE OFFSET SHIFT
G53 MACHINE COORDAINTE POSITIONING, NON-MODAL (X, Y, Z, A, B)
G54 WORK OFSET COORDIANTE POSITIONING #1
G55 WORK OFSET COORDIANTE POSITIONING #2
G56 WORK OFSET COORDIANTE POSITIONING #3
G57 WORK OFSET COORDIANTE POSITIONING #4
G58 WORK OFSET COORDIANTE POSITIONING #5
G59 WORK OFSET COORDIANTE POSITIONING #6
G60 UNI-DIRECTIONAL POSITIONING (X, Y, Z, A, B)
G61 EXACT STOP, MODAL (X, Y, Z, A, B)
G64 EXACT STOP G61 MODE CANCEL
G65 MACRO SUB-ROUTINE CALL
G68 ROATION (G17, G18, G19, X, Y, Z, R)
G69 ROTATION G68 CANCEL
G70 BOLT HOLE CIRCLE with a CANNED CYCLE (I, J, L)

G71 BOLTHOLEARC with a CANNED CYCLE (I, J, K, L)
G72 BOLT HOLES ALONG AN ANGLE with a CANNED CYCLE (I, J, L)
G73 HIGH SPEED PECK DRILL CANNED CYCLE (X, Y, A, B, Z, I, J, K, Q, P, R, L, F)
G74 REVERSE TAPPING CANNED CYCLE (X, Y, A, B, Z, J, R, L, F)
G76 FINE BORING CANNED CYCLE (X, Y, A, B, Z, I, J, P, Q, R, L, F)
G77 BACK BORE CANNED CYCLE (X, Y, A, B, Z, I, J, Q, R, L, F)
G80 CANCEL CANNED CYCLE
G81 DRILL CANNED CYCLE (X, Y, A, B, Z, R, L, F)
G82 SPOT DRILL/COUNTERBORE CANNED CYCLE (X, Y, A, B, Z, P, R, L, F)
G83 PECK DRILL CANNED CYCLE (X, Y, A, B, Z, I, J, K, Q, P, R, L, F)
G84 TAPPING CANNED CYCLE (X, Y, A, B, Z, J, R, L, F)
G85 BORE IN, BORE OUT CANNED CYCLE (X, Y, A, B, Z, R, L, F)
G86 BORE IN, STOP, RAPID OUT CANNED CYCLE (X, Y, A, B, Z, R, L, F)
G87 BORE IN AND MANUAL RETRACT CANNED CYCLE (X, Y, A, B, Z, R, L, F)
G88 BORE IN, DWELL, MANUAL RETRACT CANNED CYCLE (X, Y, A, B, Z, P, R, L, F)
G89 BORE IN, DWELL, BORE OUT (X, Y, A, B, Z, P, R, L, F)
G90 ABSOLUTE POSITIONING
G91 INCREMENTAL POSITIONING
G92 GLOBAL WORK COORDINATE SYSTEM SHIFT (FANUC) (HAAS)
G92 SET WORK COORDINATE VALUE (YA SNAC)
G93 INVERSE TIME FEED MODE ON
G94 INVERSE TIME FEED MODE OFF/FEED PER MINUTE ON
G98 CANNED CYCLE INITIAL POINT RETURN
G99 CANNED CYCLE “R” PLANE RETURN

Appendix – Miscellaneous Functions – M-Codes

M00 PROGRAM STOP
M01 OPTIONAL PROGRAM STOP
M02 PROGRAM END
M03 SPINDLE ON CLOCKWISE
M04 SPINDLE ON COUTERCLOCKWISE
M05 SPINDLE STOP
M06 TOOL CHANGE
M08 COOLANT ON
M09 COOLANT OFF
M19 ORIENT SPINDLE (P, R)
M21-M28 OPTIONAL USER M CODE INTERFACE WITH M-FIN SIGNAL
M30 PROGRAM END AN RESET
M31 CHIP AUGER FORWARD
M32 CHIP AUGER REVERSE
M33 CHIP AUGER STOP
M34 COOLANT SPIGOT POSITION DOWN, INCREMENT
M35 COOLANT SPIGOT POSITION UP, DECREMENT
M36 PALET PART READY
M39 ROTATE TOOL TURRET
M41 SPINDLE LOW GEAR OVERRIDE
M42 SPINDLE HIGH GEAR OVERRIDE
M50 EXECUTE PALLET CHANGE

M51-M58 OPTIONAL USER M CODE SET
M59 OUTPUT RELAY SET (N)
M61-M68 OPTIONAL USER M CODE CLEAR
M69 OUTPUT RELAY CLEAR (N)
M75 SET G35 OR G136 REFERENCE POINT
M76 CONTROL DISPLAY INACTIVE
M77 CONTROL DISPLAY ACTIVE
M78 ALARM IF SKIP SIGNAL FOUND
M79 ALARM IF SKIP SIGNAL NOT FOUND
M80 AUTOMATIC DOOR OPEN
M81 AUTOMATIC DOOR CLOSE
M82 TOOL UNCLAMP
M83 AUTO AIR JET ON
M84 AUTO AIR JET OFF
M86 TOOL CLAMP
M88 COOLANT THROUGH SPINDLE ON
M89 COOLANT THROUGH SPINDLE OFF
M93 AXIS POS CAPTURE START (P, Q)
M94 AXIS POS CAPTURE STOP
M95 SLEEP MODE
M96 JUMP IF NO SIGNAL (P, Q)
M97 LOCAL SUB-PROGRAM CALL (P, L)
M98 SUB-PROGRAM CALL (P, L)
M99 SUB-PROGRAM/ROUTINE RETURN OR LOOP
M109 INTERACTIVE USER INPUT (P)

Standard Drill Sizes - Inches

Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.
80 =	.0135	43 =	.089	8 =	.199	25/64 =	.3906
79 =	.0145	42 =	.0935	7 =	.201	X =	.397
1/64 =	.0156	3/32 =	.0938	13/64 =	.2031	Y =	.404
78 =	.016	41 =	.096	6 =	.204	13/32 =	.4063
77 =	.018	40 =	.098	5 =	.2055	Z =	.413
76 =	.020	39 =	.0995	4 =	.209	27/64 =	.4219
75 =	.021	38 =	.1015	3 =	.213	7/16 =	.4375
74 =	.0225	37 =	.104	7/32 =	.2188	29/64 =	.4531
73 =	.024	36 =	.1065	2 =	.221	15/32 =	.4688
72 =	.025	7/64 =	.1094	1 =	.228	31/64 =	.4844
71 =	.026	35 =	.110	A =	.234	1/2 =	.500
70 =	.028	34 =	.111	15/64 =	.2344	33/64 =	.5156
69 =	.0292	33 =	.113	B =	.238	17/32 =	.5313
68 =	.031	32 =	.116	C =	.242	35/64 =	.5469
1/32 =	.0313	31 =	.120	D =	.246	9/16 =	.5625
67 =	.032	1/8 =	.1250	1/4 (E) =	.250	37/64 =	.5781
66 =	.033	30 =	.1285	F =	.257	19/32 =	.5938
65 =	.035	29 =	.136	G =	.261	39/64 =	.6094
64 =	.036	28 =	.1405	17/64 =	.2656	5/8 =	.625
63 =	.037	9/64 =	.1406	H =	.266	41/64 =	.6406
62 =	.038	27 =	.144	I =	.272	21/32 =	.6563
61 =	.039	26 =	.147	J =	.277	43/64 =	.6719
60 =	.040	25 =	.1495	K =	.281	11/16 =	.6875
59 =	.041	24 =	.152	9/32 =	.2813	45/64 =	.7031
58 =	.042	23 =	.154	L =	.290	23/32 =	.7188
57 =	.043	5/32 =	.1563	M =	.295	47/64 =	.7344
56 =	.0465	22 =	.157	19/64 =	.2969	3/4 =	.750
3/64 =	.0469	21 =	.159	N =	.302	49/64 =	.7656
55 =	.052	20 =	.161	5/16 =	.3125	25/32 =	.7813
54 =	.055	19 =	.166	O =	.316	51/64 =	.7969
53 =	.0595	18 =	.1695	P =	.323	13/16 =	.8125
1/16 =	.0625	11/64 =	.1719	21/64 =	.3281	53/64 =	.8281
52 =	.0635	17 =	.173	Q =	.332	27/32 =	.8438
51 =	.067	16 =	.177	R =	.339	55/64 =	.8594
50 =	.070	15 =	.180	11/32 =	.3438	7/8 =	.875
49 =	.073	14 =	.182	S =	.348	57/64 =	.8906
48 =	.076	13 =	.185	T =	.358	29/32 =	.9063
5/64 =	.0781	3/16 =	.1875	23/64 =	.3594	59/64 =	.9219
47 =	.0785	12 =	.189	U =	.368	15/16 =	.9375
46 =	.081	11 =	.191	3/8 =	.375	61/64 =	.9531
45 =	.082	10 =	.1935	V =	.377	31/32 =	.9688
44 =	.086	9 =	.196	W =	.386	63/64 =	.9844

Inch Tap Drill Sizes

INCH SIZES - NATIONAL COARSE UNC

TAP SIZE	DRILL SIZE
#1-64	#53
#2-56	#51
#3-48	5/64"
#4-40	#43
#5-40	#39
#6-32	#36
#8-32	#29
#10-24	#25
#12-24	#17
1/4-20	#7
5/16-18	F
3/8-16	5/16
7/16-14	U
1/2-13	27/64
9/16-12	31/64
5/8-11	17/32
3/4-10	21/32
7/8-9	49/64
1"-8	7/8
1-1/8-7	63/64
1-1/4-7	1-7/64
1-1/2-6	1-11/32
1-3/4-5	1-35/64
2"-4-1/2	1-25/32

INCH SIZES - NATIONAL FINE UNF

TAP SIZE	DRILL SIZE
#0-80	3/64"
#1-72	#53
#2-64	#50
#3-56	#46
#4-48	#42
#5-44	#37
#6-40	#33
#8-36	#29
#10-32	#21
#12-28	#15
1/4-28	#3
5/16-24	I
3/8-24	Q
7/16-20	W
1/2-20	29/64
9/16-18	33/64
5/8-18	37/64
3/4-16	11/16
7/8-14	13/16
1"-14	15/16
1-1/8-12	1-3/64
1-1/4-12	1-11/64
1-1/2-12	1-27/64
1-3/4-12	1-43/64
2"-12	1-59/64

Metric Tap Drill Sizes

METRIC COARSE SIZES

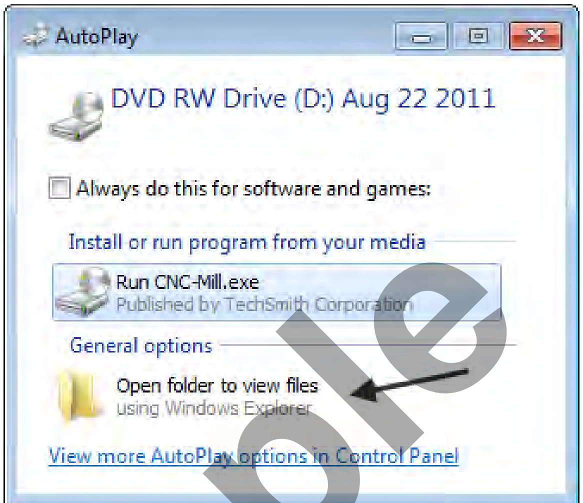
TAP SIZE	DRILL SIZE
1mm x .25	.75mm
1.1 x .25	.85
1.2 x .25	.95
1.4 x .3	1.1
1.6 x .35	1.25
1.7 x .35	1.3
1.8 x .35	1.45
2 x .4	1.6
2.2 x .45	1.75
2.5 x .45	2.05
3 x .5	2.5
3.5 x .6	2.9
4 x .7	3.3
4.5 x .75	3.7
5 x .8	4.2
6 x 1	5
7 x 1	6
8 x 1.25	6.8
9 x 1.25	7.8
10 x 1.5	8.5
11 x 1.5	9.5
12 x 1.75	10.2
14 x 2	12
16 x 2	14
18 x 2.5	15.5
20 x 2.5	17.5
22 x 2.5	19.5
24 x 3	21
27 x 3	24
30 x 3.5	26.5

METRIC FINE SIZES

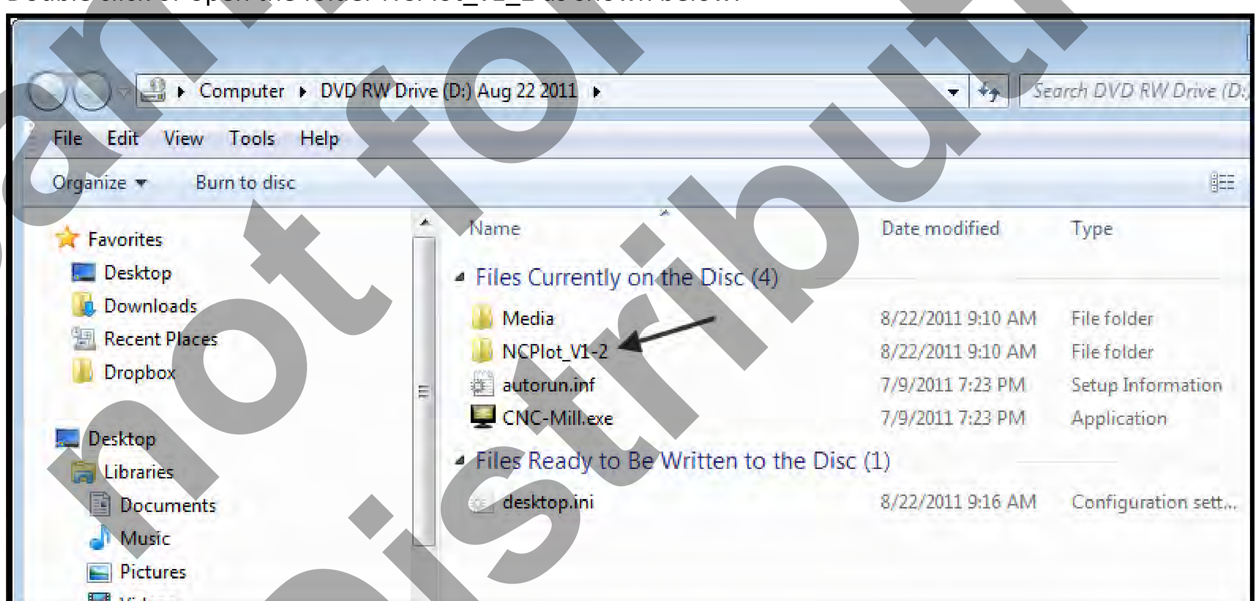
TAP SIZE	DRILL SIZE
4 mm x .35	3.6mm
4 x .5	3.5
5 x .5	4.5
6 x .5	5.5
6 x .75	5.25
7 x .75	6.25
8 x .5	7.5
8 x .75	7.25
8 x 1	7
9 x 1	8
10 x .75	9.25
10 x 1	9
10 x 1.25	8.8
11 x 1	10
12 x .75	11.25
12 x 1	11
12 x 1.5	10.5
14 x 1	13
14 x 1.25	12.8
14 x 1.5	12.5
16 x 1	15
16 x 1.5	14.5
18 x 1	17
18 x 2	16
20 x 1	19
20 x 1.5	18.5
20 x 2	18
22 x 1	21
22 x 1.5	20.5
22 x 2	20
24 x 1.5	22.5
24 x 2	22
26 x 1.5	24.5
27 x 1.5	25.5
27 x 2	25
28 x 1.5	26.5
30 x 1.5	28.5
30 x 2	28

Installation Instructions for Workbook users:

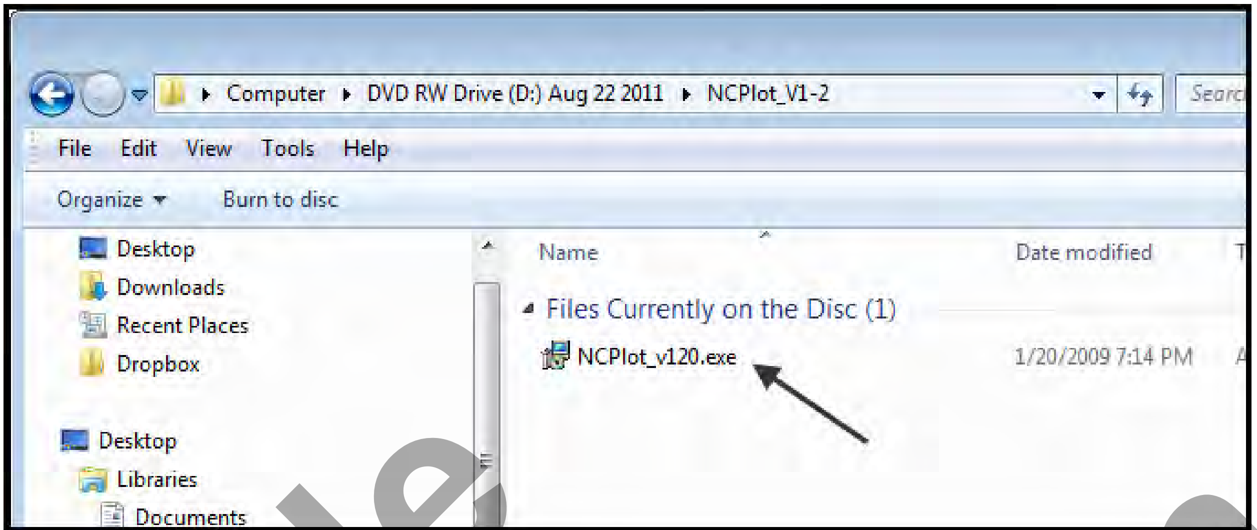
1. Locate the DVD/CD that came with the workbook (fixed to the back cover).
2. Insert the disc into the DVD/CD ROM tray of your computer.
3. When the AutoPlay window is displayed select Open folder to view files as shown below:



4. Double click or open the folder NCPlot_v1_2 as shown below:

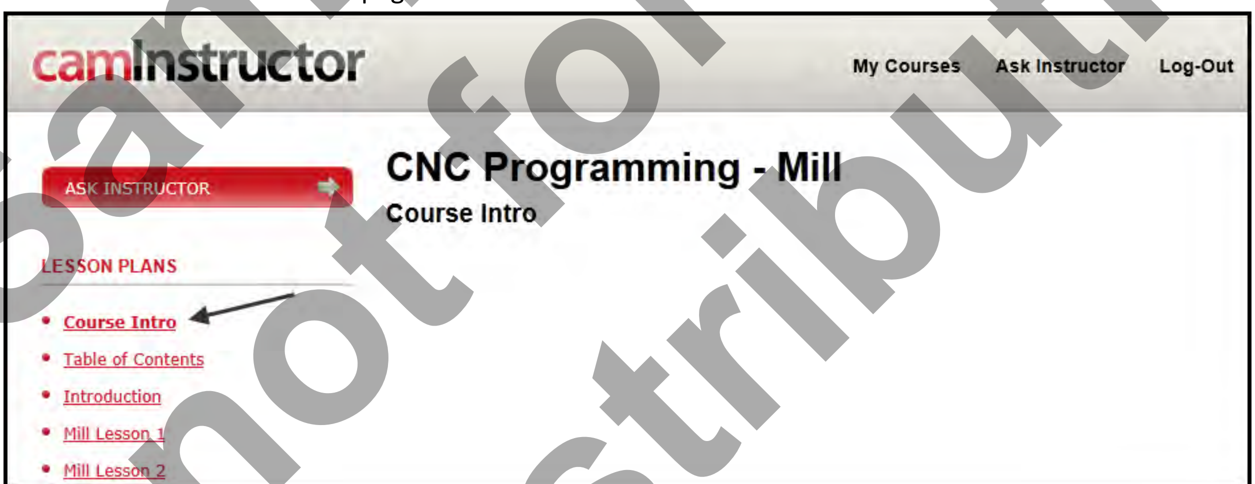


5. Double-click the NCPlot.exe file as shown below. This will start the installation process. Follow the instructions on the screen to complete the installation.



Installation Instructions for Online Course users:

1. Go to the Course Intro page of the Online Course as shown below:



2. Locate the "What you need to complete this course" section on this page and click on the NCPlot link to download the NCPlot installation file. Make sure you save the file to a place on the hard drive that you can be easily located.
3. Once the file has downloaded locate it and double-click the NCPlot.zip file to extract the installation files onto your hard drive. Make sure you extract the files into a new folder on your hard drive and remember where the folder is located.
4. Open the folder the files were extracted to and double click on NCPlot.exe.
5. Follow the instructions on the screen.