5 NIMS CNC Milling Programming & Set-Up

Objectives

Students will be able to:

- Write and Format CNC milling programs
- Diagram CNC Fanuc Control Programs
- Analyze and edit CNC G-code based programs
- Program and Operate a CNC Mill by earning Level I NIMS CNC Milling Credential

Orienting Questions

- ✓ How do cutter offsets effect CNC programming?
- ✓ What is the difference between cutter compensation left and right?
- ✓ What is the difference between contour machining and point machining?
- ✓ Is "NIMS" a National or State Accredited Program?
- ✓ Why is NIMS important to Industry?

The **bolded/underlined words are key terms…click on the **blue underlined terms** for more information.

Closed Captions and transcripts are available for all videos in this module. Click the CC button at the bottom right of the play menu to turn on closed captioning in the language of your choice. You may also read a full transcript of this video by clicking on the bottom below the play menu.

INTRODUCTION

CNC Milling Programs can be divided into two basic divisions; the "**Preparation Codes"** and the actual "**Machining Codes"**. Each machine tool builder establishes the required G-codes to prep the machine for basic machining operations. A standard CNC milling program may have up to six lines of start-up codes to establish the program. These codes establish code activation and deactivation, positioning, tool selection, height compensation and cutter speed. Once the machine has been prepped and is positioned correctly; machining codes are then activated to begin the stock removal. The stock removal normally falls within two categories; "**Contour Machining**" and "**Point Machining**". Contour machining uses the side of the cutter to remove stock and point machining uses the tip or end of the cutter to remove stock by plunging into the material. Both types of machining require geometry movements and G-coding to determine what type of cutting action is necessary. These machining operations can be activated by a one simple G-Code or a **"Canned-cycle"** that activates a sequence of movements through a G-Code command.

5.1 PROGRAM CONFIGURATION

The Program will consist of G-Codes and M-Codes that governs the machine tool in accordance with standard CNC movements. All Fanuc or G-Code programs will have a program number at the beginning of the program; it will begin with the letter "O" and be followed by a 4-digit number (O3123). Programs may also use **Sequence Numbers** at the start of every program line for a reference. Sequence numbers may increase by 1, 5, 10, or 100, this depends upon the programmer's preference the programmer may also choose not to use sequence numbers at all. It does not affect the outcome of the program. The theory of CNC programming is to produce a program that will machine the part as quickly and accurate as possible.

5.1.1 START-UP CODES

"Start-Up Codes" or "**Preparation Codes"** must be established at beginning of each program to initialize the program. A typical Fanuc milling program (Figure 1.) will have the following codes within the first three lines of the program, G17, G20, G40, G80, and G90. These codes will activate or deactivate certain CNC functions. G17 sets the machining planes to "X" and "Y". G20 establishes the units of movement to be in "inches" vs. "metric". G40 cancels all cutter compensations that may be active. G80 cancels all canned cycles that may active and G90 sets the movement to absolute positioning. The second line of the program normally contains the tool change and tool call command, (M6 T01). The third line of the program will activate a rapid movement to an absolute position that is relative to the "**Work Piece Coordinate**"; this is referred to as the "**WPC**".

```
O3123;

N5 G17 G20 G40 G80 G90;

N10 M6 T01;

N15 G0 G54 X0.0 Y0.0;
```

Figure 1. Start-Up Program

The 3rd program line will be as follows: G0 G54 X0.0 Y0.0; the "X" and the "Y" are subject to change depending upon initial positioning requirements. The WPC codes can range between G54 – 59; six different WPC's can be located within the travel allowances of the machine tool establishing six different absolute positions. G54 – 59 will be our activation codes for establishing absolute positioning in reference to the WPC being used to set our initial origin for our main program. The WPC offset page will list on "X", "Y" and "Z" for G54 through G59; these numbers will establish the distance from the machine home position to the absolute origin point of the part to be machined. This is why G54 – 59 is referred to as the "Work Piece Coordinate". Most WPC's only use a "X" and "Y" reference number to establish location. The "Z" references will be controlled by activating tool lengths using a G43 that we will discuss in the next section. The following print (Figure 2.) demonstrates an example of a vise located on a 20" x 40" mill table. The "X-30.000" and "Y-5.000" dimensions are the actual dimensions that will be recorded on the **"Work"** page establishing the WPC. These are absolute numbers from the "**Home Position**" of the machine to the **"Part Zero Reference".**

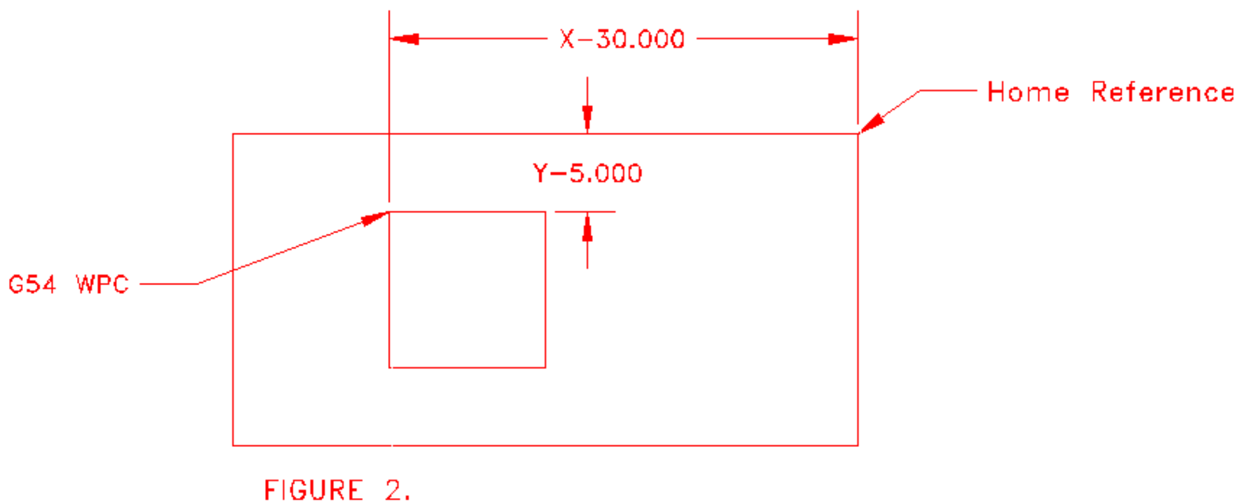


Figure 2. WPC Prints

5.1.2 TOOL CALL AND COMPENSATION CODES

Once the Start-Up codes are established for positioning and tooling selection; a tool length compensation "**G43**" will activate the actual length of the called tool. It will be called as follows: G0 G43 H1 Z1.0 (Figure 3.); G43 activates the length offset number "H1"from the menu offset page by reading offset one. The Menu "**Offset Page**" (Figure 4.) will contain a number that reflects the distance from the end of the tool to the top of the part or the "Z" zero position

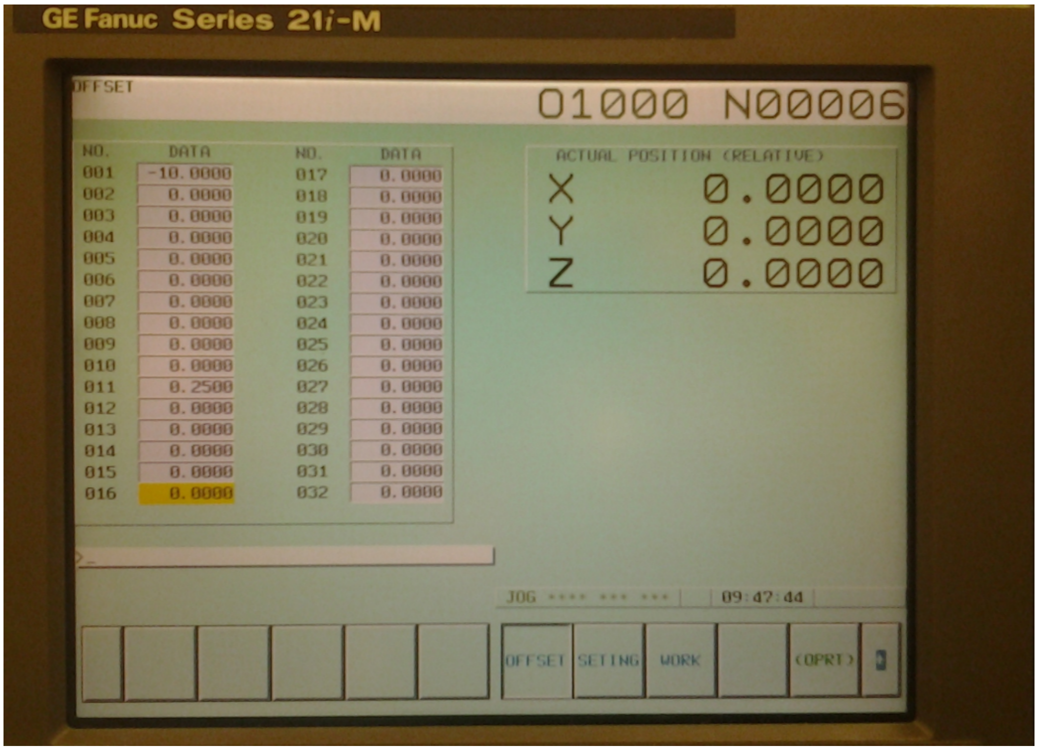
```
N20 G0 G43 H1 Z1.0;

N25 M3 S1000
```

Figure 3. Program Tool Compensation

Figure 3. Start up Codes

The active tool will travel to a “Z” position of 1.0 above the “Z” zero plane by calculating the measured tool length in offset #1. The fifth line will turn on the spindle of the machine tool by using a M3 S1000 format. The “M3” is spindle forward or CW command and “S” sets the RPM of spindle. The speed of



spindle is determined by the material being machined and the type of cutter being used for the operation. Each machine tool will have a limit on the maximum RPM that it can rotate, so this number will vary when factoring the proper speed. The preparation portion of the program has now been completed and is readyfor machining codes.

Figure 4. Off set Code

ACTIVITY #1

Match the following G-codes and M-codes with the correct definitions listed below.

1. G17	_______	A.	Height Compensation
2. G90	_______	B.	Incremental Movement
3. G91	_______	C.	Spindle Speed
4. G54-59	_______	D.	Absolute Movement
5. G43	_______	E.	X & Y Cutter Plane
6. G40	_______	F.	Program End
7. G0	_______	G.	Spindle Direction
8. M3	_______	H.	WPC's
9. M30	_______	I.	Cancel Compensation
10. S1000	_______	J.	Rapid Command

ANSWERS TO ACTIVITY #1

Matching

1. E
2. D
3. B
4. H
5. A
6. I
7. J
8. G
9. F
10. C

5.2 STOCK REMOVAL

"Stock Removal" is the variable of CNC programs; the start-up codes will remain the same throughout basic programs. Stock removal will be ever changing depending upon the job at hand. Stock removal methods will depend upon the amount and shape or geometry of the stock to be machined. Stock removal can be accomplished by several methods or procedures. We have categorized them into two modes of machining, contour machining and point machining. Stock can be machined by using many different types and styles of cutters. The emphasis of this lesson is not on the tooling that is being used to cut the stock, but on the programming that controls the tooling. Proper programming techniques will be the focal point of our study as we survey CNC Mill programming.

5.2.1 CONTOUR MACHINING

"Contour Machining" will be the first type of stock removal to be evaluated. Contour Machining is accomplished when the side of the cutter removes the stock. When the side of the cutter is being used to cut along the geometry of the part, it will be necessary to establish a "**Cutter Compensation Code**". The G-codes will either be **G41** or **G42**, G41 is "**Cutter Compensation Left**" and G42 is "**Cutter Compensation Right**". These two codes will shift the cutter to one side of the geometry or the other depending upon which code is being used and the direction of the tool movement. The amount of the shift will be the radial dimension recorded in the offset menu as called in the program. The following program (Figure 5.) is an example of picking up cutter compensation left (G41) and machining a .500 wide step around a 5.00 square block .25 deep. The shaded areas on Figure 6. through Figure 9. show the progression of the cutting tool moving around the geometry of the cut-out section of the part. An offset of .5 is being called on program line N35 using a Cutter Compensation Left (G41) with a D11 offset page reference shifting the cutter to the left side of the geometry. The value of cutter compensation is being able to adjust the geometry of the part by increasing or decreasing the radial dimension of the cutter without actually changing the dimensions within the program. If a part is oversize, the radial dimension can be decreased allowing the cutter to be shifted toward the geometry. If the part is undersize, the radial dimension can be increased moving the cutter away from the geometry of the part. Once the part has been machined, a Cutter Compensation Cancel (G40) is called on line N60 to deactivate the cutter compensation.

```
O3123;                          (Main Program Number)
N5 G0 G17 G40 G80 G90;          (Start-Up Commands)
N10 M6 T01;                     (Tool Change Command)
N15 G0 G54 X1.0 Y1.0;           (Workpiece Reference establishing Absolute Position)
N20 G0 G43 H1 Z1.0;             (Tool Height established 1” above the part)
N25 M3 S1000;                   (Forward Spindle Speed at 1000 RPM’s)
N30 G0 Z-0.25;                  (Move to -.25 below Z 0.0 to begin contouring)
N35 G1 G41 X-.5 Y0.0 D11 F5.0;  (Linear Feed activating Cutter Comp. Left)
N40 G1 Y-4.5;                   (Move to Absolute Position of Y-4.5)
N45 G1 X-4.5                    (Move to Absolute Position of X-4.5)
N50 G1 Y-.5                     (Move to Absolute Position of Y-.5)
N55 G1 X0.0                     (Move to Absolute Position of X0.0)
N60 G1 G40 X1.0 Y1.0            (Cancel Cutter Compensation)
N65 G0 Z1.0                     (Move to Safe Position in the Z Axis)
N70 G0 G91 G28 Z0.0 Y0.0        (Home Reference move for Y & Z)
N75 M30                         (Program End)
```

Figure 5.Cutter Compensation Mode

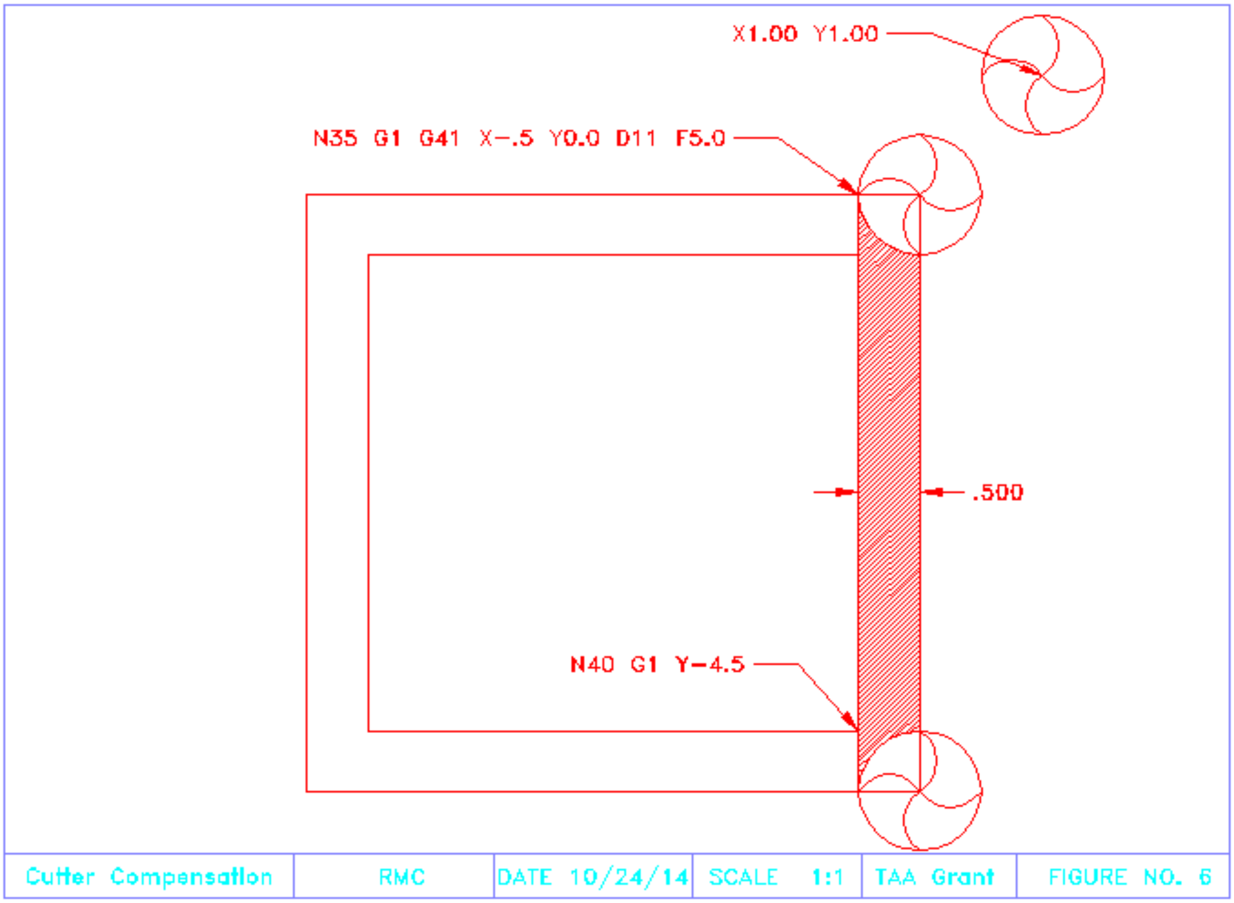


Figure 6. Progression of Cutting Tool

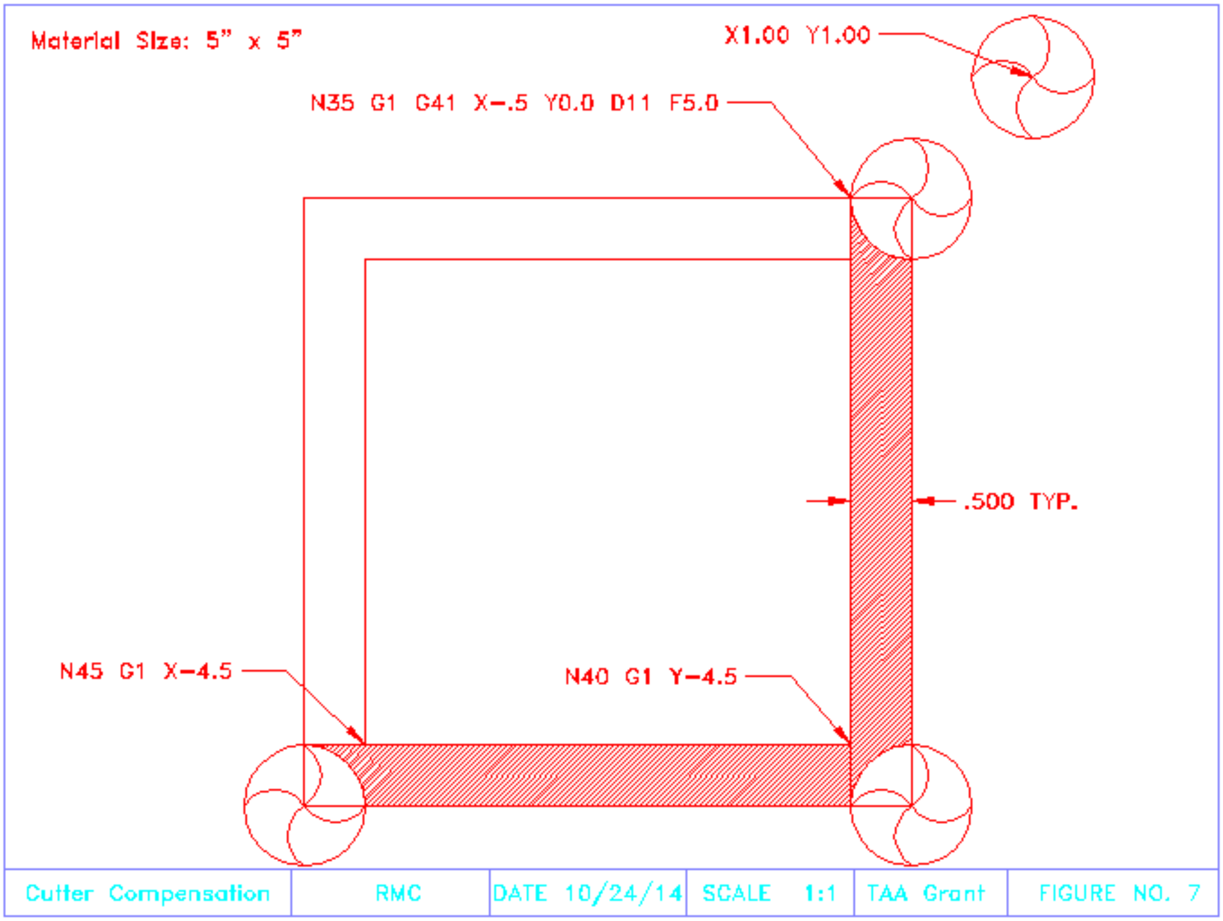


Figure 7. Progression of Cutting Tool

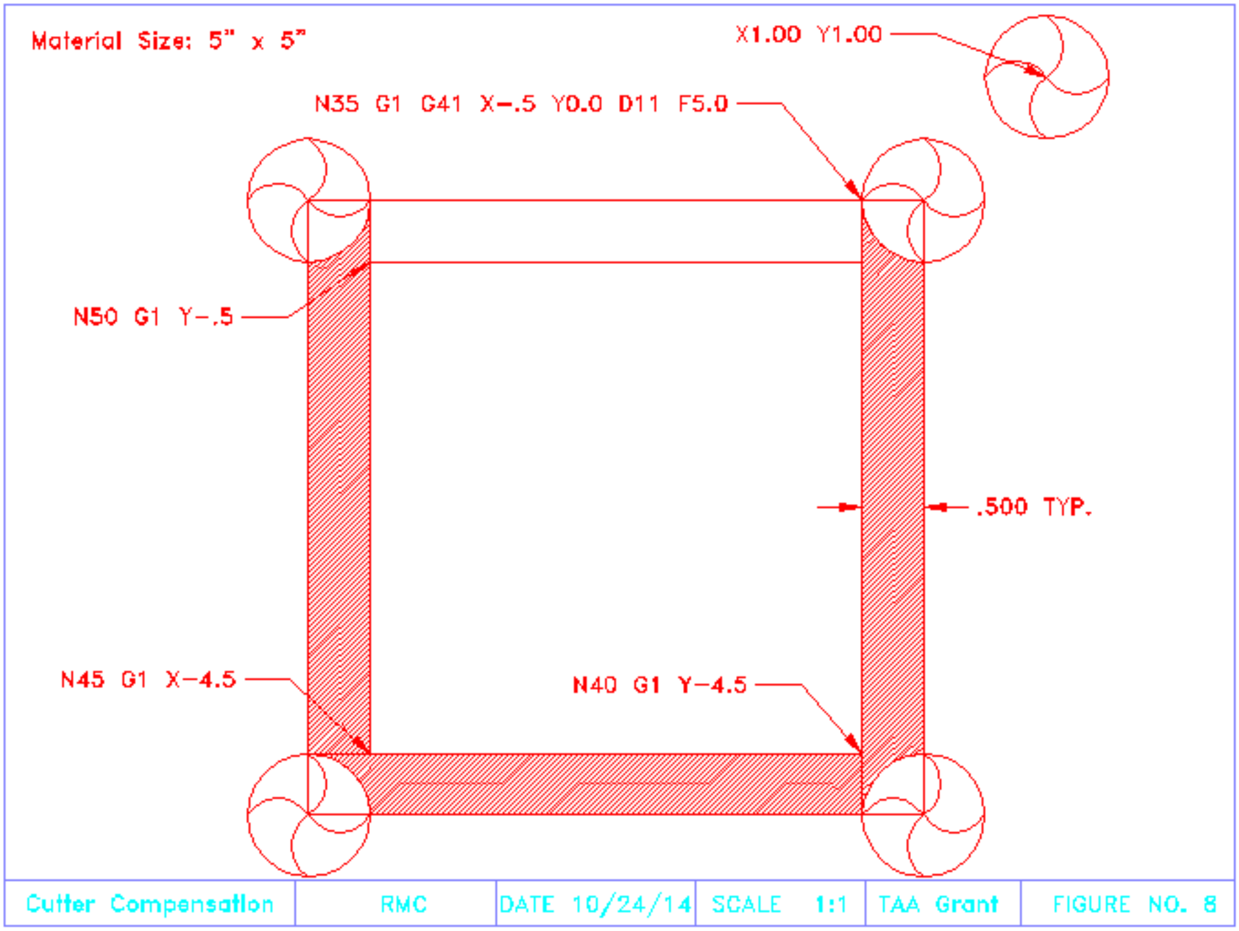


Figure 8. Progression of Cutting Tool

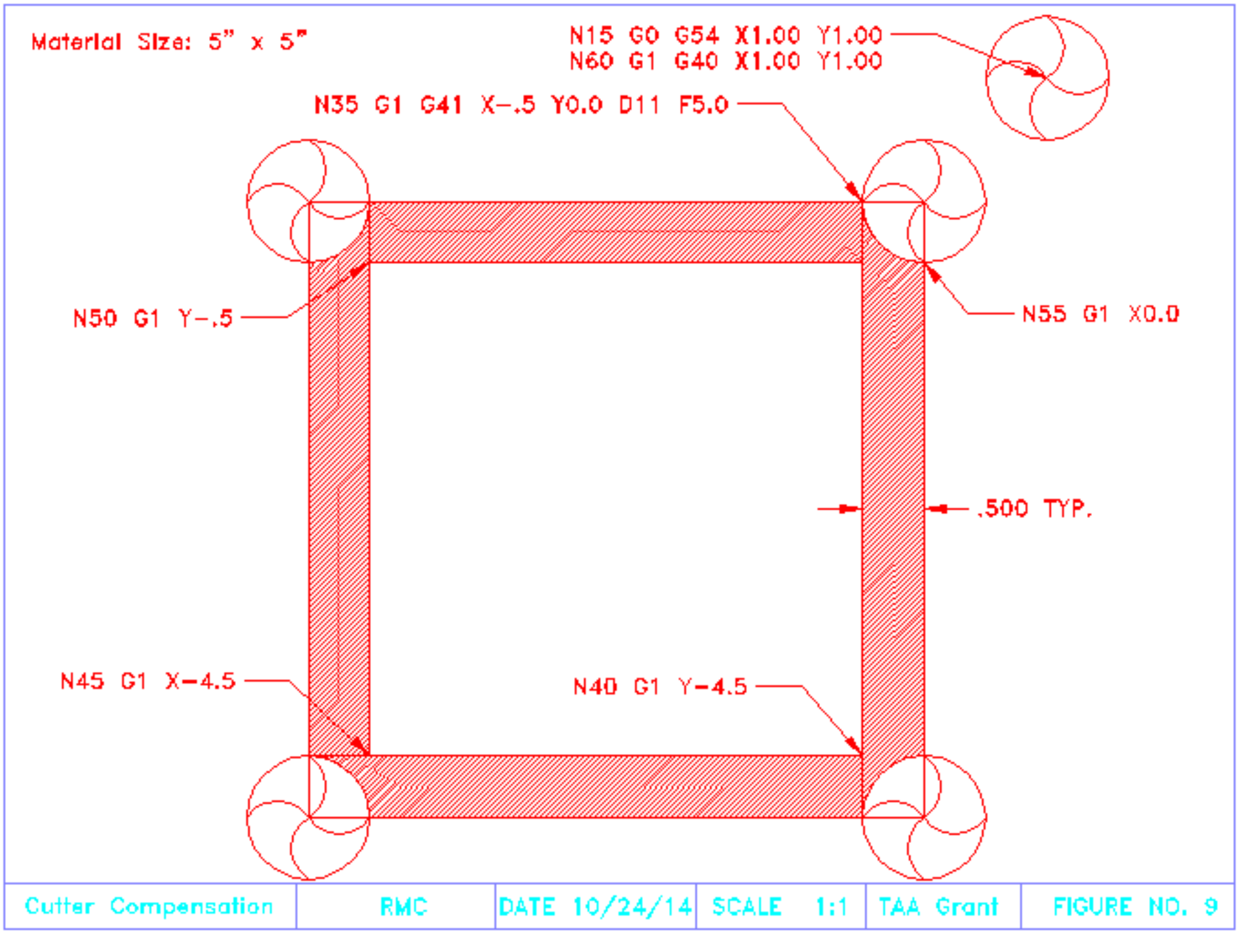


Figure 9. Progression of Cutting Tool

ACTIVITY #2

Fill in the blanks on the following program to machine the exact shape of the previous drawings (Figures 6 - 9); the only difference will be using G42 Cutter Compensation Right instead of G41

O3124;	(Main Program Number)
N5 G0 G17 G40 G80 G90;	(Start-Up Commands)
N10 M6 T01;	(Tool Change Command)
N15 G0 G54 X1.0 Y1.0;	(WPC Establishing Absolute Position)
N20 G0 _______ H1 Z1.0;	(Tool Height established 1” above the part)
N25 M3 S1000;	Forward Spindle Speed at 1000 RPM’s)
N30 G0 Z-0.25;	(Move to -.25 below Z 0.0 to begin contouring)
N35 G1 _______ X_______ Y_______ ________ F5.0;	(Linear Feed activating Cutter Comp. Right)
N40 G1 _________	
N45 G1 _________	
N50 G1 _________	
N55 G1 _________	
N60 G1 ________ X1.0 Y1.0	
N65 G0 Z1.0	(Move to Safe Position in the Z Axis)
N70 G0 G91 G28 Z0.0 Y0.0	(Home Reference move for Y & Z)
N75 M30	(Program End)

Cutter Compensation Left.

ANSWERS TO ACTIVITY #2

O3124;	(Main Program Number)
N5 G0 G17 G40 G80 G90;	(Start-Up Commands)
N10 M6 T01;	(Tool Change Command)
N15 G0 G54 X1.000 Y1.000;	(WPC Establishing Absolute Position)
N20 G0 <u>G43</u> H1 Z1.0;	(Tool Height established 1" above the part)
N25 M3 S1000;	Forward Spindle Speed at 1000 RPM's)
N30 G0 Z-0.25;	(Move to -.25 below Z 0.0 to begin contouring)
N35 G1 G41 <u>X0.0</u> Y<u>-.500</u> <u>D11</u> F5.0;	(Linear Feed activating Cutter Comp. Right)
N40 G1 X<u>-4.500</u>	
N45 G1 Y<u>-4.500</u>	
N50 G1 X<u>-.500</u>	
N55 G1 Y<u>0.0</u>	
N60 G1 <u>G40</u> X1.0 Y1.0	
N65 G0 Z1.0	(Move to Safe Position in the Z Axis)
N70 G0 G91 G28 Z0.0 Y0.0	(Home Reference move for Y & Z)
N75 M30	(Program End)

5.2.2 POINT MACHINING

"Point Machining" will be the next type of machining to be examined. Point machining options can be as follows: Center-Drilling, Drilling, Counter-boring, Reaming, Tapping or any combination listed. The main difference between Contour machining and Point machining will be direction of the cutting motion. Contouring requires a cutter compensation to place the cutter to either the left or right side of the geometry for "X" and "Y" movement. Point machining does not require a left or right compensation, because the cutting motion is vertical movement using the "Z" axis. The point machining operation is programmed to the center of the machine spindle; the size and shape of the tool will govern the geometry of the hole. Point machining can be accomplished by simple G1 movements in the "Z" axis or can be programmed using canned cycles such as G81, G83 or a G84 Tapping Cycle. The selection of the coding sequence will vary from part to part depending upon machining requirements. The simplest method of point machining would be as follows in program O3125 (Figure 10.) This program will drill one hole at coordinate location X-1.0, Y-1.0, .25 deep using Tool #1.

O3125;	(Main Program Number)
N5 G0 G17 G40 G80 G90;	(Start-Up Commands)
N10 M6 T01;	(Tool Change Command)
N15 G0 G54 X-1.0 Y-1.0;	(X and Y establish first hole position)
N20 G0 G43 H1 Z1.0;	(Tool Height established 1" above the part)
N25 M3 S1000;	(Forward Spindle Speed at 1000 RPM's)
N30 G0 Z.1	(Moves tool to .1 above part)
N35 G1 Z-0.25 F5.0;	(Drills hole .25 deep into stock)
N40 G0 Z.1	(Retracts tool to .1 above part)
N45 G0 G91G28 Y0.0 Z0.0	(Home Reference move for Y & Z)
N50 M30	(Program End)

Figure 10. Point Machining

The next example of point machining will be using a "**Drilling Cycle**" G81; the G81 cycle will control the depth of the hole, the retract position for the tool and the feed rate. The first coordinate for the drilled holes can also be located in this line, but normally the coordinate for the first hole is located in the program line previous to the tool change and the remaining hole coordinates follow the G81 line. The following program O3126, (Figure 11.) will be an example for drilling 6 holes according to print (Figure 12.).

O3126;	(Main Program Number)
N5 G0 G17 G40 G80 G90;	(Start-Up Commands)
N10 M6 T01;	(Tool Change Command)
N15 G0 G54 X-1.250 Y-2.50;	(X and Y establish first hole position)
N20 G0 G43 H1 Z1.0;	(Tool Height established 1" above the part)
N25 M3 S1000;	(Forward Spindle Speed at 1000 RPM's)
N30 G0 Z.1	(Moves tool to .1 above part)
N35 G81 R.100 Z-0.25 F5.0;	(Drilling Cycle Command)
N40 X-1.75	(2^{nd} Hole Position)
N45 X-2.250	(3^{rd} Hole Position)
N50 X-2.75	(4^{th} Hole Position)
N55 X-3.250	(5^{th} Hole Position)
N60 X-3.75	(6^{th} Hole Position)
N65 G0 G80 Z1.00	(G80 Cancels Drilling Cycle)
N45 G0 G91G28 Y0.0 Z0.0	(Home Reference move for Y & Z)
N50 M30	(Program End)

Figure 11. Drilling Cycle

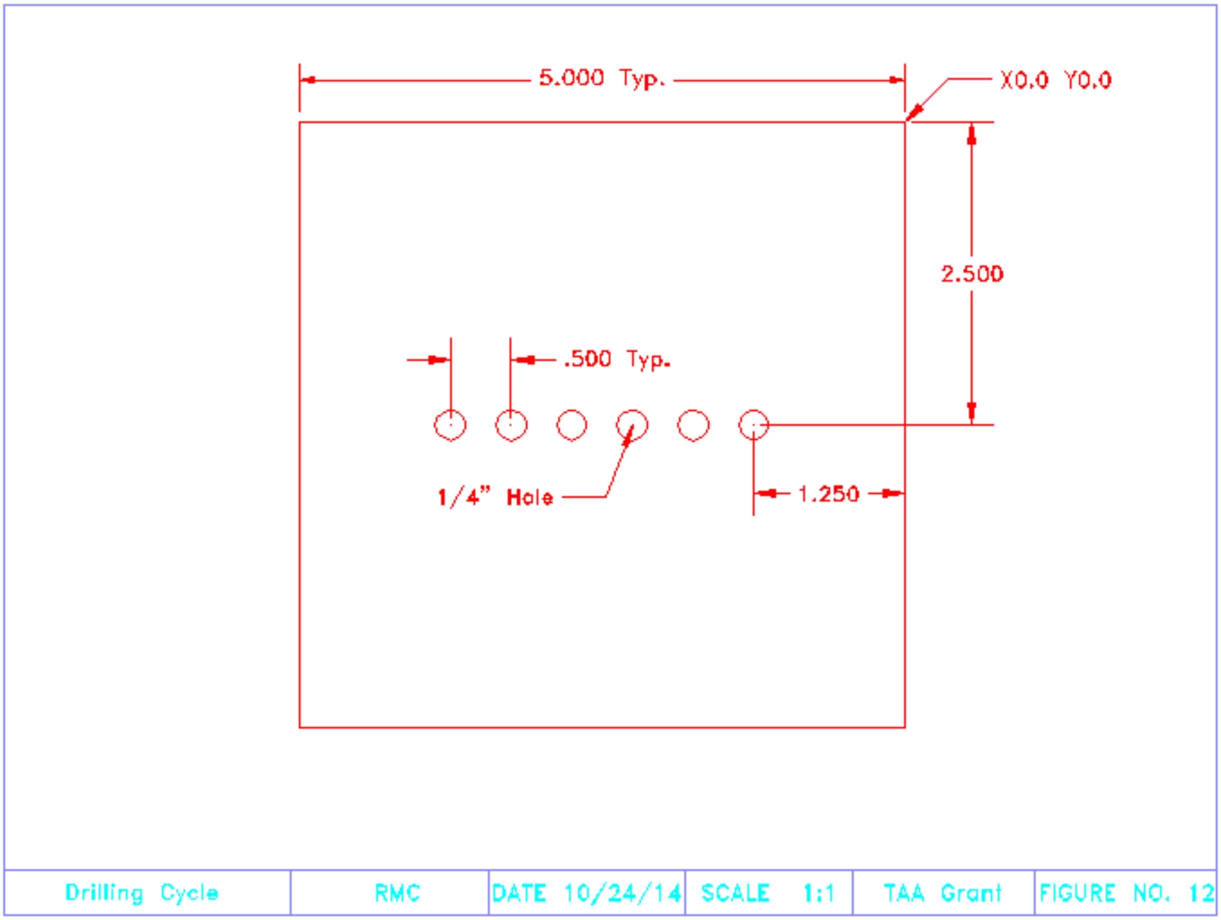


Figure 12. Drilling Cycle

The only difference between a G81 and G83 command will be the drilling action. A G81 command will be a straight move to the set depth of the "Z", a G83 command is known as a "**Peck Drilling Cycle**". This cycle is designed to break the chip by retracting in and out of the hole. The increment of the drill depth is controlled by the letter "Q" in the G83 drilling cycle. A sample peck drilling format would be as follows: G83 Z-1.00 R.100 Q.250 F5.0; the hole would be 1.00 deep, the drill depth increment would be .250 with a retract point of .100.

ACTIVITY #3

Fill in the blanks on the following program to drill the holes .5 deep as located on Print (Figure 13.). Use a standard drilling cycle.

O3127;	(Main Program Number)
N5 G0 G17 G40 G80 G90;	(Start-Up Commands)
N10 M6 T01;	(Tool Change Command)
N15 G0 G54 X_______ Y_______;	(Position Hole #1)
N20 G0 _______ H1 Z1.0;	(Tool Height established 1" above the part)
N25 M3 S1000;	Forward Spindle Speed at 1000 RPM's)
N30 G0 Z0.1;	(Move to Z.100 to begin Drilling Cycle)
N35 G______ Z______ R______ F5.0;	
N40 _________	(Position Hole #2)
N45 _________	(Position Hole #3)
N50 _________ __________	(Position Hole #4)
N55 _________	(Position Hole #5)
N60 _________	(Position Hole #6)
N65 G0 G _______ Z1.0	
N70 G0 G91 G28 Z0.0 Y0.0	(Home Reference move for Y & Z)
N75 M30	(Program End)

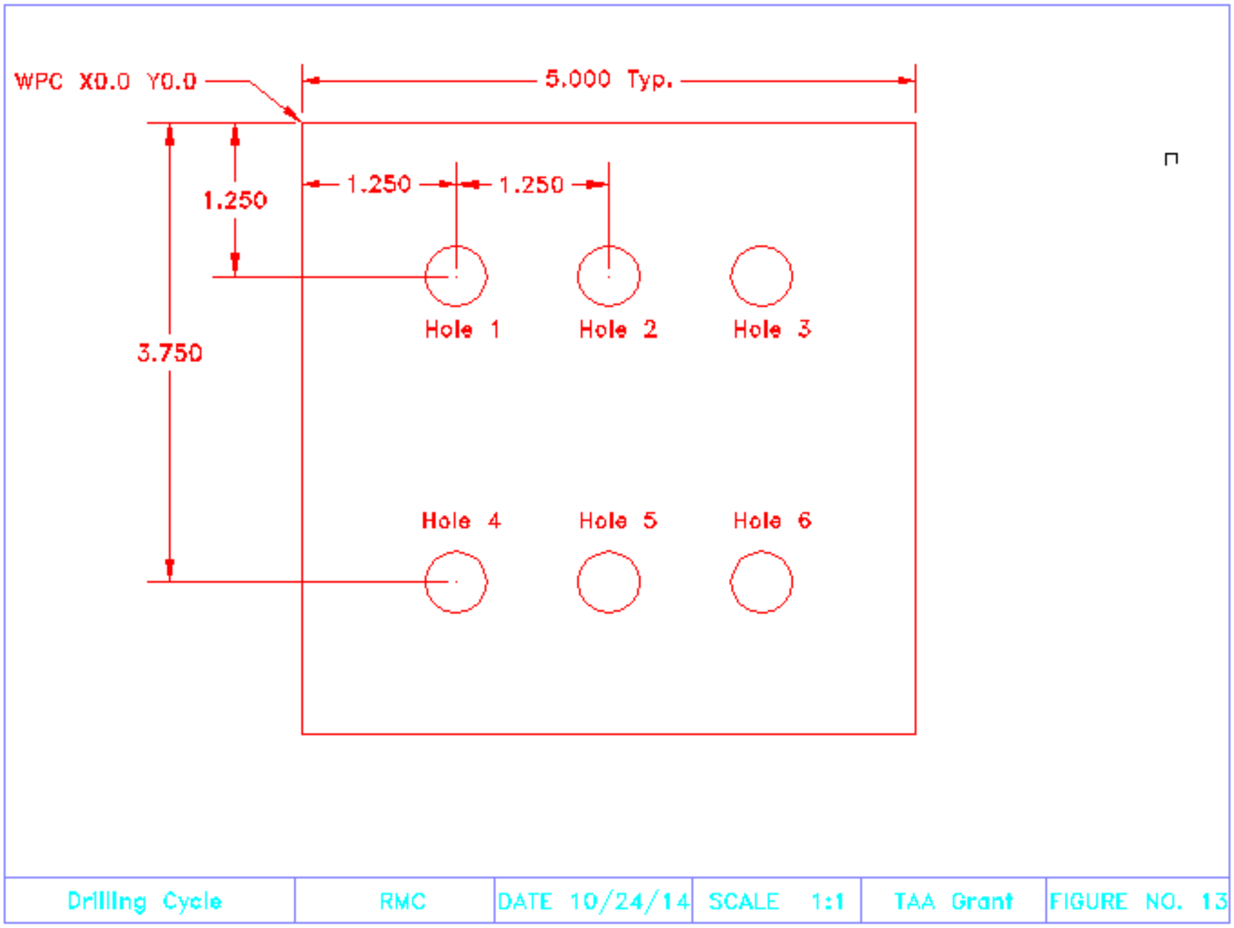


Figure 13. Drilling Cycle

ANSWERS TO ACTIVITY #3

O3127;	(Main Program Number)
N5 G0 G17 G40 G80 G90;	(Start-Up Commands)
N10 M6 T01;	(Tool Change Command)
N15 G0 G54 X1.25 Y-1.25;	(Position Hole #1)
N20 G0 G43 H1 Z1.0;	(Tool Height established 1" above the part)
N25 M3 S1000;	Forward Spindle Speed at 1000 RPM's)
N30 G0 Z0.1;	(Move to Z.100 to begin Drilling Cycle)
N35 G81 Z-.500 R.100 F5.0;	(Standard Drilling Cycle)
N40 X2.500	(Position Hole #2)
N45 X3.750	(Position Hole #3)
N50 X1.250 Y-3.75	(Position Hole #4)
N55 X2.500	(Position Hole #5)
N60 X3.750	(Position Hole #6)
N65 G0 G80 Z1.0	(Drill Cycle Cancel)
N70 G0 G91 G28 Z0.0 Y0.0	(Home Reference move for Y & Z)
N75 M30	(Program End)

The final example of a point machining cycle will be a "**Tapping Cycle**" **G84**; this operation is also known by the term "**Rigid Tapping**". The hole or holes to be tapped must be prepped before the tapping operation can be performed. The preparation consist of drilling the hole to the correct drill tap size and countersinking the lead in portion of the hole with a chamfer as specified on the print. Once these operations are completed; the Tapping Cycle can be programmed. The following programming example O3128 (Figure 14.) will be an example of a typical Tapping Cycle tapping the holes on print (Figure 13.). The program will only consist of the tapping cycle portion following the tool change. We will assume that the tapped holes have been drilled and countersunk to begin the threading process.

```
O3128: (Tapping Program)
;
;
N100 M6 T03;                          (Tool Change Command)
N105 G0 G54 X1.25 Y-1.25;             (Position Hole #1)
N110 G0 G43 H3 Z1.0;                  (Tool Height established 1" above the part)
N115 G0 Z0.500;                       (Move to Z.500 to begin Tapping Cycle)
N120 M29 S300;                        (Establishes Solid Tapping Mode at 300 RPM's)
N125 G84 G98 Z-.500 R.100 P1000 F23.077;   (Standard Tapping Cycle)
(G84 – Solid Tapping Mode)
(G98 – Establishes Return to initial "Z" before moving to next position)
("Z" – Tapping Depth from top of part, Always a negative number)
("R" – Reference Point above part to begin Tapping Cycle)
("P" – Dwell or pause time at the bottom of tapped hole before spindle is reversed)
(Tap Feed Rate: Calculated from Spindle Speed and Tap Pitch)
(Tap Feed Rate Example: ½-13 Example➔ "Pitch x Spindle Speed" (.0769 x 300) = 23.077 in/min.)
N130 X2.500                           (Position Hole #2)
N135 X3.750                           (Position Hole #3)
N140 X1.250 Y-3.75                    (Position Hole #4)
N145 X2.500                           (Position Hole #5)
N150 X3.750                           (Position Hole #6)
N155 G0 G80 Z1.0                      (Tapping Cycle Cancel)
N160 G0 G91 G28 Z0.0 Y0.0             (Home Reference move for Y & Z)
N165 M30                              (Program End)
```

Figure 14. Tapping Cycle

ACTIVITY #4

Fill in the blanks on the following program lines to tap ¼-20 hole at 500 RPM .375 deep.

N120 _______ _______;

N125 _______ _______ _______ ________ _______ ________;

ANSWERS TO ACTIVITY #4

N120 M29 S500;

N125 G84 G98 Z-.370 R.100 P1000 F25.00;

5.3. NIMS MILLING SET-UP & OPERATIONS EXAM

The National Institute for Metalworking Skills (**NIMS**) is a nationally recognized accrediting agency providing quality standards for the metalworking industry. NIMS main goal is to improve the quality of training programs to build a skilled workforce to provide career opportunities for all candidates. If a program or institution wants to obtain NIMS accreditation, it must go through a three-step process including a **Self-Study**, an **On-Site Evaluation** and then all instructors must have **Individual Credentials** for the areas that they instruct. Once a site has been approved, the accreditation is for a five-year period and may be renewed to maintain NIMS certification every five years. A NIMS certified site will have ability to provide guidance and instruction in all areas of accreditation for enrolled students. For this particular section of this module, the student will prepare and attempt to earn Level I Credential in CNC Milling Set-Up & Operations. The credential will require the machining of a physical project (Figure 15.) and an online exam. The actual part will need to be machined to industry standards meeting the requirements of the Performance Assessment Worksheet (Figure 16.).The online exam will be a multiple choice format focusing on basic Milling CNC knowledge.

5.3.1. NIMS MILLING SET-UP & OPERATIONS PROJECT

Introduction: The following activity will require all milling knowledge acquired in this module; this process will include setting up and programming the CNC mill to machine the actual part according to the specifications listed on NIMS Print (Figure 15.).

Materials and methods: The material will be 2.50 x 3.50 x .75 thick aluminum stock being held in a standard vise.

Requirements for successful completion: This lab will require all safety procedures to be followed to complete the manufactured part. WPC and offset set-ups including proper programming format, graphic evaluation coupled with first piece single-block application. The machined part will match the print specifications. Once the physical part has been inspected by the sponsor and approved by the Met-Tec; an online exam will be taken to earn the NIMS Level I Milling Credential.

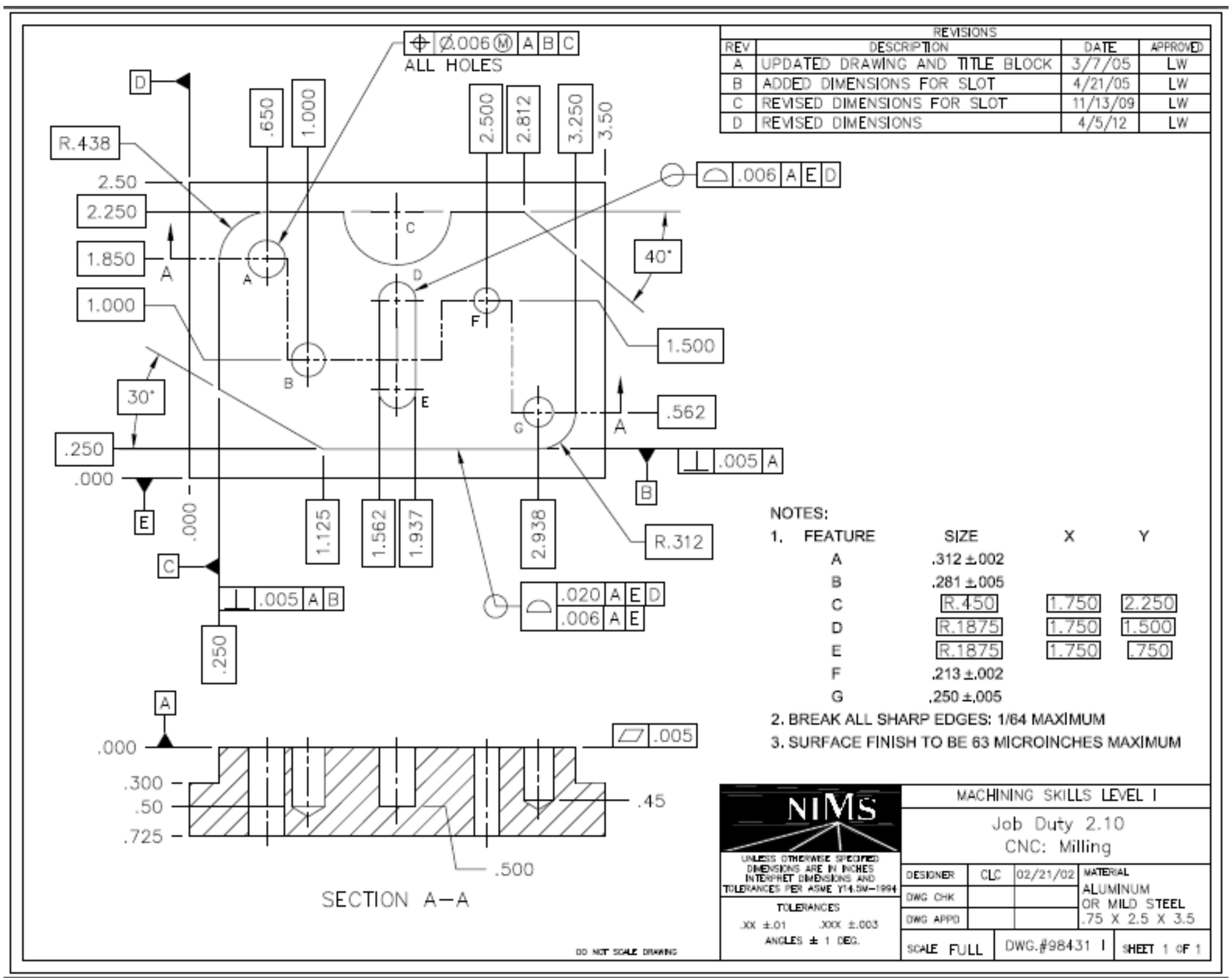


Figure 15. Milling Set-up & Operations Exam

Performance Standards
CNC Milling

Material

Aluminum or mild steel.

Duty

- Set up, program, and operate a CNC mill or machining center and manufacture a part within tolerance
- Work from a process sheet and part print.
- Understand the x, y, z Cartesian coordinate system.
- Create a tool set up sheet.
- Understand fundamental machine processing, feeds and speed, and select simple part.

Performance Standard

Write a program at the machine or off line. Setup the machining operation and perform standards given on mill operations (2.10) to develop a simple part (with linear and circular interpolations).

Accuracy Level: Match the requirements of the part print. 63 Ra microinch finish

Assessment Equipment and Material:

Workstation: A standard workbench, a CNC mill with continuous path capability on 2½ axes.

Material: A part matching the material requirements of the part print, material: cold rolled steel.

Tooling: A 6" milling vise or greater, screws, studs, nuts, washers, and clamps sufficient to secure the vise, or the part to the table. Assorted parallels, ball peen, and composition hammers, assorted cutters and cutter adapters fitted to the machine spindle, files, magnetic base for indicators, soft jaws for the vise and assorted cutters.

Measuring Inst: Required micrometers, combination set, dial indicator, 6" rule, a 6" vernier, dial, or electronic caliper, adjustable parallels, edge finder, appropriate tools for determining squareness, and surface finish comparison standards.

Reference: Machinery's Handbook, operator's manual of the machine tool.

Performance Project – CNC Milling **Evaluation Criteria**		Pass	Fail
1. Overall Dimensions Length 3.50 ± .010 Width 2.50 ± .010 Thickness .725 ± .003	Pass = within tolerance Fail = out of tolerance	☐ ☐ ☐	☐ ☐ ☐
2. Profile tolerance within limits Position ± .006 Depth .300 ± .003	Pass = within tolerance Fail = out of tolerance	☐ ☐	☐ ☐
3. Hole A Position ± .006 Diameter ± .002	Pass = within tolerance Fail = out of tolerance	☐ ☐	☐ ☐
4. Hole B Position ± .006 Diameter .281 ± .005 Depth .500 ± .010	Pass = within tolerance Fail = out of tolerance	☐ ☐ ☐	☐ ☐ ☐
5. Hole F Position ± .006 Diameter ± .002	Pass = within tolerance Fail = out of tolerance	☐ ☐	☐ ☐
6. Hole G Position ± .006 Diameter ± .005 Depth .45 ± .010	Pass = within tolerance Fail = out of tolerance	☐ ☐ ☐	☐ ☐ ☐
7. Slot D-E Position ± .006 Width .312 ± .002 Depth .500 ± .003	Pass = within tolerance Fail = out of tolerance	☐ ☐ ☐	☐ ☐ ☐
8. Break all sharp edges .015 max.	Pass = within tolerance Fail = out of tolerance	☐	☐
9. Surface finish 63 Ra microinches min.	Pass = within tolerance Fail = out of tolerance	☐	☐
END OF CNC MILLING EVALUATION			

Figure 16. CNC Milling Performance Project

5.3.2 NIMS ONLINE EXAM PREPARATION

The Machinery's Handbook will also be the only reference material allowed to be accessed while taking the online NIMS Milling examination. Proper use and knowledge of the Machinery's Handbook will be critical for passing the exam. Topics of study will be similar to that which appeared on the NIMS Turning exam. Shop safety, measurement, print reading, formulas, and CNC coding will also be the areas of concentration. The majority of the exam will focus on basic CNC knowledge. General sample milling questions that may be included are listed in (Figure 17.).

1) The width of a milled slot, hole diameter or counterbore depth can best be measured with a/an:
   a) Outside micrometer
   b) Dial test indicator
   c) Inside micrometer
   d) Vernier caliper

2) A right-hand end mill rotates in the spindle and when viewed from the front end (cutting edges).
   a) Counterclockwise, clockwise
   b) Clockwise, clockwise
   c) Clockwise, counterclockwise
   d) Counterclockwise, counterclockwise

3) Feed rates for milling operations are expressed as
   a) Inches per revolution
   b) Feet per minute
   c) Meters per millimeter
   d) Inches per minute

4) Which of the following types of end mills is capable of plunge cutting a hole?
   a) Three fluted gashed end mill
   b) Two fluted end mill
   c) Four fluted gashed end mill
   d) Four fluted center cutting end mill
   e) Both b and d are capable of starting a hole

5) Which of the following is the proper method for milling a 0.375-inch slot on a vertical milling machine?
   a) Use a 0.375-inch diameter end mill and take the cut in one pass
   b) Use a 0.250-inch diameter end mill and take two passes each overlapping one another
   c) Use a 0.125-inch diameter end mill and take three passes each overlapping one another
   d) Use a 0.250-inch diameter end mill for a roughing cut and a 0.375-inch diameter mill for a finishing cut.

Figure 17. Sample Milling Questions

The following formulas can be used to solve the following problems for calculating **Inches per Minute Feed rate**, **Inches per Revolution** and **Surface Speed**. (Figure 18.) Contains examples of word problems with the correct data inserted into the proper formula with the outcomes.

Inches Per Minute Feedrate

IPM = # of Flutes x Chip Load x RPM

Inches Per Revolution

IPR = # of Flutes x Chip Load

Surface Speed

$$SFM = \frac{RPM \times Dia.}{4}$$

1) Calculate the feed in inches per minute for 01 tool steel with a cutting speed of 55 SFM. The HSS end mill has a diameter of 0.875 inches with four flutes. The chip load per tooth is 0.008 inches.
a) 8.05 IPM
b) 601.6 IPM
c) 7.86 IPM
d) 296.51 IPM

Inches Per Minute Feedrate

$$8.05 = 4 \times .008 \times 251$$

2) A CNC milling machine uses a feed rate given in inches per revolution (IPR). What is the feed in inches per revolution for a four fluted end mill with a recommended chip load per tooth of 0.014 inches?
a) 0.014 IPR
b) 0.028 IPR
c) 0.042 IPR
d) 0.056 IPR

Inches Per Revolution

$$.056 = 4 \times \text{Chip } .014$$

3) What is the correct SFM for a ¾" diameter end mill machining 01 tool steel at 267 RPM?
a) 50 SFM
b) 266.7 SFM
c) 133.3 SFM
d) 150 SFM

Surface Speed

$$50 = \frac{267 \times .75}{4}$$

Figure 18. Word Problem Examples

ACTIVITY #5

Answer the following practice questions from the Machinery's Handbook.

1. Referring to the Machinery Handbook, what is the minimum pitch diameter for an external thread 3/8-16 UNC-2A?

2. Referring to the Machinery Handbook, what is the tap drill size for a ½-13 UNC-2B with 75% thread engagement?

3. What is the largest square that can be machined from a 3.0 diameter piece of stock?

4. Calculate the RPM's for a drill operation with a cutting speed of 150 SFM using a 3/8" diameter spilt-point drill with a feed rate of .005 IPR.

5. Find the TPF for a workpiece with a large diameter of 4.0 and a small diameter of 3.0 with a taper length of 6.0 inches.

6. What is the time required to make three turning passes on a piece of stainless steel 8.0 inches long at 400 RPM's with a .008 feed rate?

7. What is the TPF for a piece 7.5 inches long with a .060 taper per inch?

8. What are the hole sizes for a 1.125 diameter using an RC3 fit?

9. What is the counter-bore diameter for ½ SHCS?

10. What is the normal clearance size for a ½ bolt?

11. What is the included angle of a standard flat head screw?

12. What is the bore range for 1-8 UNC-2B thread?

13. What is the formula for calculating the “pitch” of a thread?

14. What is the formula for calculating the basic diameter of a numbered screw?

15. What is the formula for calculating tap drill size?

ANSWERS TO ACTIVITY #5

Answers
1. .3287
2. 27/64 or .4219
3. 2.121
4. 1600 RPM's
5. 2.00
6. 7.5
7. .096
8. 1.125 – 1.1258
9. 13/16
10. 17/32 or .531
11. 82 Degrees
12. .865 - .890
13. 1 / TPI
14. .06 + (.013 x # of Screw)
15. Major Dia. - Pitch

MAJOR CONCEPTS

KEY CONCEPTS

- CNC Milling Programs can be divided into two basic divisions known as **"Preparation Codes"** and "**Machining Codes"**. Preparation Codes will basically follow the same format time after time. Machining Codes will vary depending upon the operations.
- Stock Removal falls within two categories known as "**Contour Machining**" and "**Point Machining**". Contour Machining uses the side of the cutter by establishing "**Cutter Compensation".** Cutter Compensation is the off-setting of the actual cutter by referring to G41 or G42 command. **"Point Machining"** uses the tip or end of the cutter to remove stock. Common point machining practices include drilling, tapping, and counter boring.
- The "**Work Piece Coordinate**" establishes an absolute reference position from home zero to the part reference zero. The "**WPC**" reference is required in all milling programs. The WPC codes G54-59 can establish multiple part positions on the same machining plane.

KEY TERMS

CNC Milling Programs: The body of code that controls the movement of the machine tool.

Preparation Codes: General Start-Up codes required to establish the CNC Program.

Machining Codes: CNC codes required to for stock removal.

Contour Machining: Removing stock with the side of the cutter.

Point Machining: Removing stock with the tip of the cutter.

Canned Cycle: Activates a sequence of movements.

Sequence Numbers: Reference numbers at the beginning of each program line.

WPC: Work Piece Coordinate

Work: The offset page were the WPC's are located.

Home Position: The absolute machine zero position.

Part Zero Reference: Term used to describe the location of the WPC.

Offset Page: The page were tool lengths and diameters are recorded.

Stock Removal: The process of removing stock by machining.

G43: Tool Height Compensation.

G41 & G42: Cutter Compensation Left and Right.

Drilling Cycle: G81 code that controls drilling parameters.

Peck Drilling Cycle: G83 code that controls pecking drilling parameters.

Tapping Cycle: G84 code that activates Rigid Tapping Cycle.

G80: Deactivates Canned Drilling and Tapping Cycles.

MODULE REINFORCEMENT

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

1. WPC establishes "Part Reference Zero". T or F

2. Contour Machining is activated by G43.	T	or	F
3. The code for activating Compensation Left is G41.	T	or	F
4. The code for activating Compensation Right is M42.	T	or	F
5. G81 and G83 are Drilling Cycles.	T	or	F
6. G80 may be a Preparation Code.	T	or	F
7. Sequence Numbers are mandatory in CNC programs.	T	or	F
8. Point Machining operations do not use canned-cycles.	T	or	F
9. Rigid Tapping may be referred to as Solid Tapping.	T	or	F
10. Contouring and Point machining are both methods of stock removal.	T	or	F

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

1. Select the correct line sequence for proper a Start-Up line.
   A. G17 G20 X1.5 Y1.5
   B. G17 G80 G40 G54
   C. G17 G20 G40 G80
   D. G17 G20 G40 G80 G90
2. Select the correct line sequence for establishing a tool call.
   A. M3 T01
   B. M6 TO1
   C. M6 T0101
   D. M6 T01
3. Select the correct line sequence for establishing a tool height call.
   A. G0 G41 H1 Z1.0
   B. G0 G43 H5 Z1.0
   C. G0 G43 H1 X1.0
   D. G0 G42 H1 Z1.0
4. Select the correct line sequence for establishing G41.
   A. G0 G41 X0.0 Y0.0 D11;
   B. G1 G41 X0.0 Y0.0 Z0.0 D11;
   C. G1 G41 X0.0 Y0.0 D11 F10.0;
   D. G1 G41 D11 F10.0;
5. Select the correct code for Rigid Tapping.
   A. G80
   B. G81
   C. G83
   D. G84
6. Select the correct code for Peck Drilling.
   A. G18
   B. G83
   C. G43
   D. G42

7. Select the correct Tapping Formula for calculating feed rate.
   A. Pitch x RPM = in/min.
   B. TPI x RPM = in/min.
   C. TPI x Pitch = in/min.
   D. None of the Above
8. Select the correct line sequence for a Drilling Cycle.
   A. G81 Z.5 R.100 Q.100 F5.0
   B. G83 Z-.5 R.100 F5.0
   C. G84 Z-.5 R.100 Q.100 F5.0
   D. None of the Above

9. Select the correct line sequence for a Peck Drilling Cycle.
   A. G81 Z.5 R.100 Q.100 F5.0
   B. G83 Z-.5 R.100 F5.0
   C. G84 Z-.500 R.100 Q.100 F5.0
   D. None of the Above
10. Select the correct line sequence for a Tapping Cycle.
   A. G84 G98 Z-.500 R.100 P1000 F23.077
   B. G84 G99 Z-.500 R.100 P1000 F23.077
   C. G84 G98 Z-.500 R.100 Q1000 F23.077
   D. None of the Above

Answer Key

True or False	Multiple Choice
1. T	1. D
2. F	2. D
3. T	3. B
4. F	4. C
5. T	5. D
6. T	6. B
7. F	7. A
8. F	8. D
9. T	9. D
10. T	10. A

DISCUSSION PROMPTS

LIST THE DIFFERENCES BETWEEN CONTOUR AND POINT MACHINING.

In Module 5, we machined geometry using contour and point machining methods. What are the different functions and purposes of both methods for removing stock?

CUTTER COMPENSATION LEFT AND RIGHT

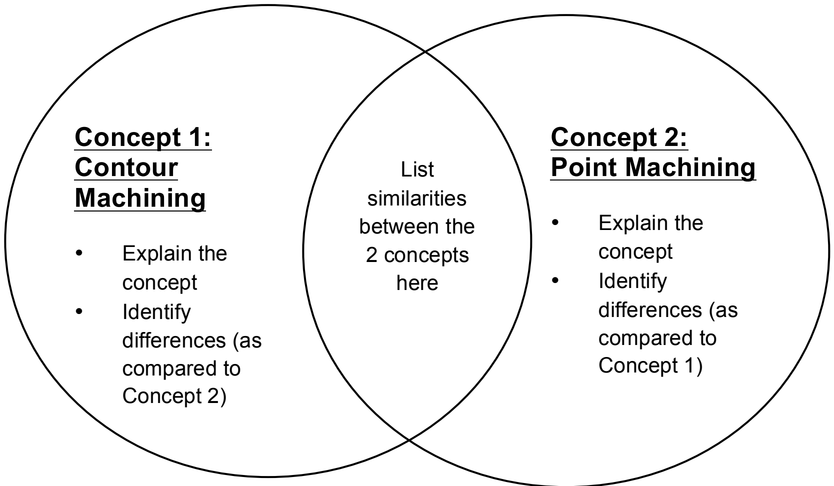
In Module 5, we used cutter compensation left and right, what are the advantages of using cutter compensations?

DRILLING AND TAPPING CYCLES

In Module 5, we used drilling and tapping cycles, what are the advantages of these canned cycles?

CRITICAL THINKING

	Concept #1 **G81 Drilling Cycle**	**Concept #2** **G83 Peck Drilling Cycle**
Define or explain each concept		
Explain how the concepts are similar		
Explain how each concept is different with respect to specific attributes		



- Explain Contour and Point Machining techniques used for stock removal.

ATTRIBUTION TABLE

Author/s	Title	Source	License
Mark Cramer	*Figure 1. Start-up Program*	*Module author*	CC BY-SA 4.0
Mark Cramer	*Figure 2. WPC Print*	*Module author, developed using AutoCad software.*	CC BY-SA 4.0
Mark Cramer	*Figure 3. Start-up Codes*	*Module Author*	CC BY-SA 4.0
Mark Cramer	*Figure 4. Offset Page*	*Module author*	CC BY-SA 4.0
Mark Cramer	*Figure 5. Cutter Compensation Mode*	*Module author*	CC BY-SA 4.0
Mark Cramer	*Figures 6.-9.* *Compensation Prints*	*Module author, developed using AutoCad software.*	CC BY-SA 4.0
Mark Cramer	*Figure 10. Point Machining*	*Module author*	CC BY-SA 4.0
Mark Cramer	*Figure 11-13. Drilling Cycle*	*Module author*	CC BY-SA 4.0
Mark Cramer	*Figure 14.Tapping Cycle*	*Module author*	CC BY-SA 4.0

Mark Cramer	*Figure 15. Drilling & Tapping Cycle*	*Module author, developed using AutoCad software.*	CC BY-SA 4.0
Mark Cramer	*Figure 16. CNC Milling Performance Project*	*Module author*	CC BY-SA 4.0
Mark Cramer	*Figure* 17. Sample Milling Questions	*Module author*	CC BY-SA 4.0
Mark Cramer	*Figure* 18. *Word Problem Examples*	*Module author*	CC BY-SA 4.0

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