4

NIMS CNC Lathe Programming & Set-Up

Objectives

Students will be able to:

- Describe the purpose and identify various canned cycles
- Summarize and modify CNC turning programs
- Analyze and edit CNC G-code based programs
- Operate a CNC Lathe by earning Level I NIMS CNC Turning Credential

Orienting Questions

✓ Why are canned cycles beneficial for CNC Programming?

✓ What is the difference between roughing and finishing?

✓ Why is tool-nose radius important to geometry?

✓ Why is NIMS important to Industry?

✓ Is "NIMS" a National or State Accredited Program?

SC ACCELERATE

CLEMSON

Version | Course ID | Rev 1 2013

INTRODUCTION

"CNC Turning Programs" are an arrangement of **"Preparation Codes"** and **"Machining Codes"**. These codes are known as G and M codes; they will control the movement of the machine tool to manufacture the parts. All CNC programs start and end with specific commands so the controller will recognize the stage of the process. This section will give detailed program guide lines that must be followed to complete the operation of the CNC lathe. The programs that will be used to generate the parts during this section will include "**Roughing**" and **"Finishing"** operations. These turning operations will be addressed in this module. These lathe operations will also require **"Tool-nose Compensation"** in order to produce accurate geometry for lines, chamfers, and radii.

4.1 PROGRAM CONFIGURATION

CNC machine controls require an **"Order of Operation"** for all programs. Whether turning or milling, all programs must be written in order from start-up commands to completion commands so the control can interpret the information. These commands are instructions given in G and M code language that the controller can interpret in order to perform the machining operations. These commands are given in blocks of information called **"Word"** commands along with **"Addresses"** followed by numerical values for the movement of X and Z axis.

4.1.1 START-UP CODES

The start-up commands are set forth by the manufacturer to initiate the machine for operation. These start-up commands must be included in the first few lines of each new program. These start-up commands may vary between machine builders, but the final result will be preparing the machine to execute the program to manufacture the part. Typical start-up commands for the lathe will include a **"Home-Positioning"** command, a **"Maximum RPM"** setting, a **"Tool-Call"**, and a **"Cutting Speed"** (Figure 1.). Once these codes are active, positioning can begin to begin stock removal.

O1234;   (PROGRAM NUMBER)

G0 G28 U0.0 W0.0; (RAPID TO MACHINE HOME POSITION)

G50 S2500;   (ESTABLISH MAX RPM)

T0101;   (COMMANDS OFFSET NUMBER 1 AND TOOL NUMBER 1)

G96 S500 M4;   (ESTABLISH CONSTANT SFM, ROTATE SPINDLE CCW)

Figure 1. Start-up Commands

4.1.2 TOOL CALL AND COMPENSATION CODES

CNC lathes equipped with a tool carousel allow the programmer to choose from a variety of tools pre-installed in tool holders within the machine. The programmer will use multiple tool call commands to advance the carousel to a desired tool for specific machining operations during the course of the program. Each tool used must have pre-determined lengths included in an "**Offset Page"** for the X and Z axis which are related to the home position of the machine. These reference numbers are located on the **"Geometry"** Offset Page. These offsets will be activated with the tool call command so the machine can interpret exact positions with relation to the work piece and the machine home position. After inserting the start-up commands, the programmer will then call the first tool and the offset with that tool for the first machining operation. Tool #1 is normally used to establish the **Workshift** for "Z" positioning, this number is recorded on the Workshift Page and is a reference from **Home Position** to the face of the part or "Z" 0.0. All other tools will be referenced to this offset number for "Z" positioning; the advantage of using a "Z" Workshift is that all tools are referenced to Tool 1 in the "Z" position, so if the stock length changes and the workshift value changes, all the other tools are still referenced to Tool #1 and will not need to be changed. The "X" positioning values are independent and do not require a workshift. Each tool is taught as it is positioned in the carousel. Tool offsets can be fine toned by **"Wear Offsets"**, wear offsets are only used when tooling begins to break down and sizing becomes an issue. Tool wear compensates for the breakdown in the tooling and is adjusted to machine the part to proper tolerances.

4.2 STOCK REMOVAL

This section is where the material is removed and shapes of the work piece will take place. Various commands will be used in multiple blocks of information for the stock removal to create the geometry of the work piece. The complexity of the work piece will determine the length of the program. The commands used in each block will be given with an attached value that sends the machine through precise tool paths with specified feed rates in order to generate the profile of the part. The tool paths can be generated using long hand programming or the more commonly used method of using "**Canned Cycles"**. Canned cycles are commands used to perform specific machining operations used to decrease the length and time involved in programming. These canned cycles are G commands used in conjunction with specific letter variables that enable the machine to perform machining operations that are common to everyday machine shop practices.

4.2.1 G70 & G71 TURNING CODES

Canned cycles are commands commonly used to reduce the length of programs. **"G70"** & "**G71"** are canned cycles used in conjunction with each other that "**Rough"** and "**Finish"** the material to specified diameters and lengths. The commands are given along with letter variables and numerical values that are calculated by the controller to generate the profile of a part. The G71 (Figure 2.) roughing canned cycle is given first and will require two blocks of information: the first block will include the depth of each cut and the tool retract distance after each cut. The second block of information includes the Letter **"P"** and variable number indicating the location of the start of the geometry and this is followed by the letter **"Q"** and a variable number stating the location of the end of the geometry. The **"P"** and **"Q"** represent the sequence numbers following the letter **"N"**. The amount of material to leave on the O.D. for finishing is established by the letter **"U"** and the amount to leave on the lengths for finish cutting is established by the

letter **"W"**. The final code establishes the feed rate of material removal. The G70 (Figure 3.) canned cycle is used for the finish turning operation and will follow the G71 "P" and "Q" start and finish points. The feed

G71 U.05 R.05;	(U=Cutting Depth, R=Retract Distance after each cut)
G71 P1 Q2 U.05 W.005 F.01;	(P1 is the Start of Geometry, Q2 is the End of the Geometry, U.05 is Amount to leave for Finish "X", W.005 is the Amount to leave for Finish "Z" and F.01 is the Roughing Feed Rate)

rate will be lower to provide a smoother surface finish.

G70 P1 Q2 F.008;	(P1 is the Starting point of Geometry; Q2 is the Ending point of the Geometry followed by the Finish Feed Rate)

Figure 2. Stock Removal and Turning Codes

Figure 3. Canned Cycle Finish Turning Cycle

4.2.2 G41 & G42 TOOL NOSE COMPENSATION

Tool nose compensation is activated by a G41 or G42 command; depending upon the direction of tool travel. These codes compensate for the insert tip radius allowing for an accurate machining. G41 activates Tool-nose compensation left and G42 activates Tool-nose compensation right. Typical Tool-nose radii will be either 1/32" or 1/16" of an inch. The following examples (Figure 4.) will demonstrate the proper protocol for machining with tool-nose compensation.
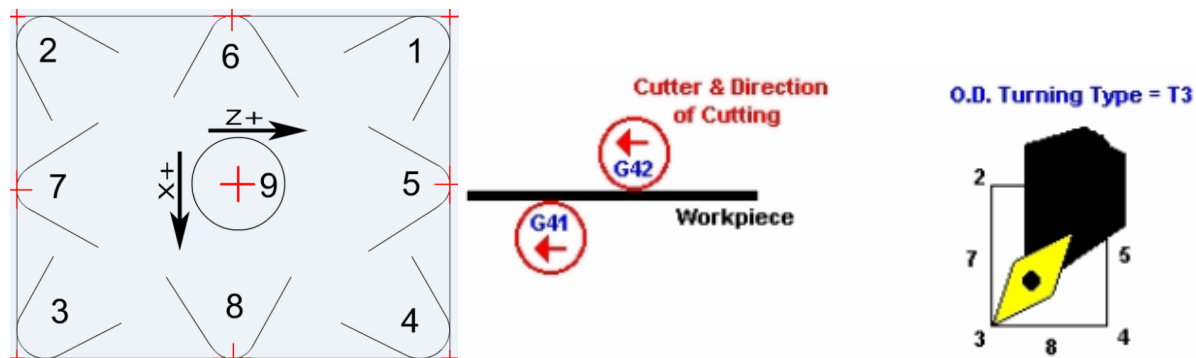
SC ACCELERATE

CLEMSON

Version | Course ID | Rev 1 2013

Figure 4. Tool Nose Compensation

ACTIVITY #1

Match the following codes and definitions listed below.

1. G70	_____	A. Tool Nose Compensation
2. G71	_____	B. Spindle Direction
3. G41 & G42	_____	C. Home Reference
4. G28	_____	D. Maximum RPM
5. G50	_____	E. Roughing Cycle
6. G96	_____	F. Constant Surface Speed
7. T0101	_____	G. Finishing Cycle
8. M3 & M4	_____	H. Tool Call

ANSWERS TO ACTIVITY #1

Matching
1. G
2. E
3. A
4. C
5. D
6. F
7. H
8. B

(Figure 5.) is an example of turning program that contains all the necessary information to rough and finish a simple part (Figure 6.) while activating tool-nose radius compensation on the finish cycle.

```
O1234;                          (Program Number)

G50 S2500;                      (Establishes Max RPM)

G28 U0.0 W0.0;                  (Takes Machine to Home Position)

T0101   ;                       (Calls Tool 1 and Offset 1)

G96 S500 M4;                    (Establishes Constant SFM, and Rotates Spindle CCW)

G0 X1.1 Z.1;                    (Rapid Move to X Diameter 1.1, Z.1 in Front of Part)

G71 U.05 R.05;                  (U=Cutting Depth, R=Pull away distance after each cut)

G71 P1 Q2 U.05 W.005 F.01;      (P1 is the Start of Geometry, Q2 is the End of the   Geometry,
                                U.05 is Amount to leave for Finish "X",  W.005 is the Amount to
                                leave for Finish "Z" and  F.01 is the Roughing Feed Rate)

N1 G1 X.5;                      (First Point of Geometry)

G1 Z-.250;

G1 X.750;

G1 Z-.750

N2 G1 X1.0                      (Last Point of Geometry)

G0 G28 U0.0 W0.0;               (Rapid back to Home Position)

G50 S2500;                      (Establishes Max. RPM)

T0202   ;                       (Calls Tool 2 and Offset 2)

G96 S600 M4;                    (Establishes Constant SFM, and Rotates Spindle CCW)

G0 G42 X1.1 Z.1;                (Rapid Move to X Diameter 1.1, Z.1 in Front of Part while
                                activating Tool Nose Radius Right)

G70 P1 Q2 F.008;                (P1 is the Starting point of Geometry; Q2 is the Ending point of
                                the Geometry followed by the Finish Feed Rate)

G0 G40 G28 U0.0 W0.0;           (Rapid back to Home Position while cancelling Cutter
                                Compensation)

M30;                            (Ends Program & Rewinds to the beginning of Program)
```

Figure 5. Turning Program

Machine .050

```
O2234;

G50 S2500;

G28 U0.0 W0.0;

T0101;

G96 S500 M4;

G0 X_____ Z_____ ;

_____ _____ _____ ;

_____ _____ _____ _____ _____ _____ ;

N1 _____ _____

_____ _____ ;

_____ _____ _____ _____ ;

_____ _____ ;

_____ _____ ;

_____ _____ _____ _____ ;

_____ _____ ;

_____ _____ ;

_____ _____ _____ _____ ;

_____ _____ ;

_____ _____ _____ _____ ;

N 2 _____ _____ ;

G0 G28 U0.0 W0.0;

G50 S2500;

T0202    ;

G96 S550 M4;

_____ _____ _____ _____ ;

_____ _____ _____ _____ ;

G0 G40 G28 U0.0 W0.0;
```

Figure 6. Finish   Cycle

ACTIVITY #2

Fill in the blanks O2234 (Figure 7.) following part on 8.).   A roughing canned cycle the correct tool called.

on program to machine the print 2234 (Figure and finishing must be used and nose radius must

SC ACCELERATE
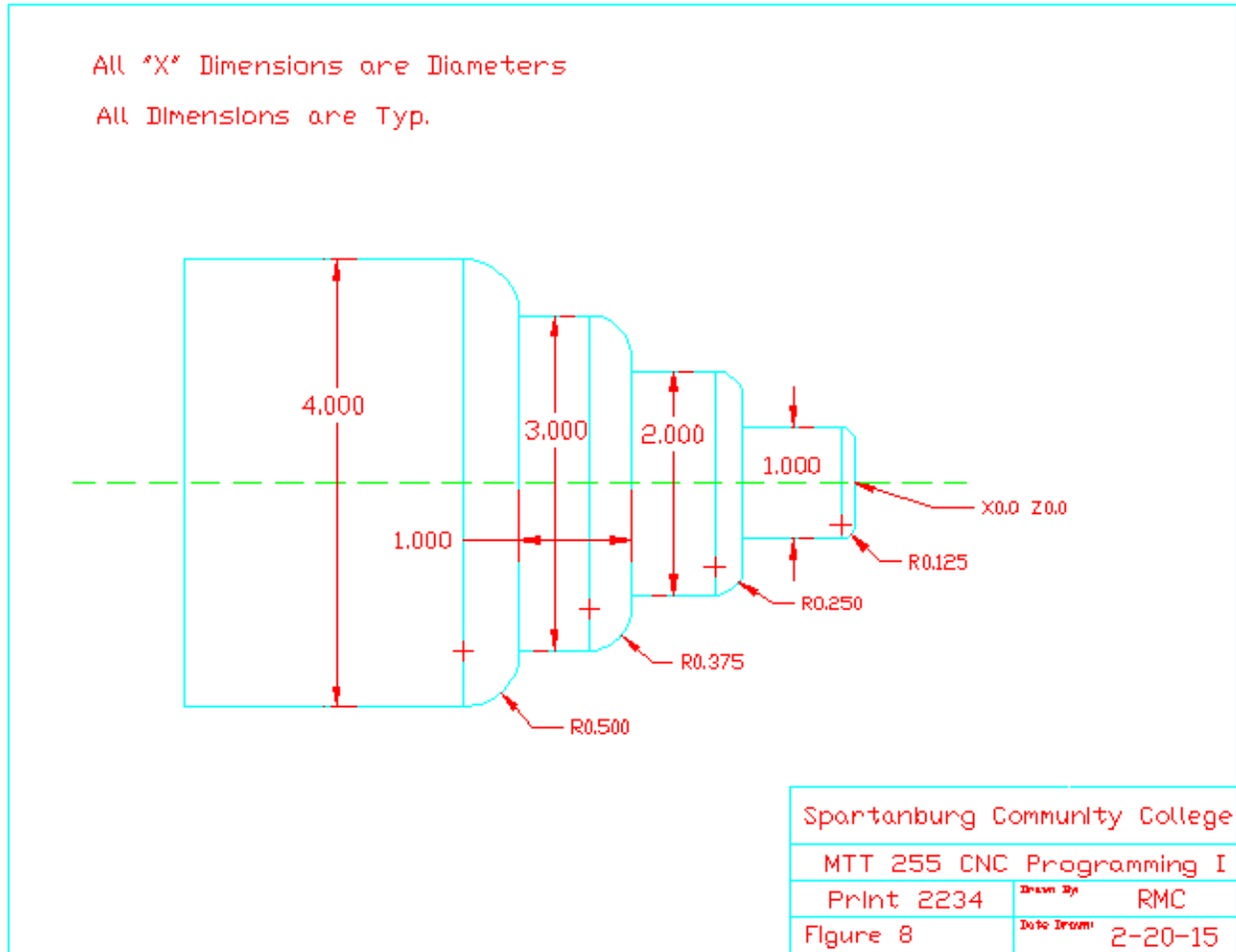CLEMSON

Figure 7. Program O2234

Figure 8. Print 2234

ANSWERS TO ACTIVITY #2

```
O2234;

G50 S2500;

G28 U0.0 W0.0;

T0101;

G96 S500 M4;

G0 X4.1 Z.1;

G71 U.05 R.05;

G71 P1 Q2 U.05 W.005 F.01;

N1 G1 X.75;

G1 Z0.0;

G3 X1.0 Z-.125 R.125;

G1 Z-1.0;

G1 X1.5;

G3 X2.0 Z-1.25 R.25;

G1 Z-2.0;

G1 X2.25;

G3 X3.0 Z-2.375 R.375;

G1 Z-3.0;

G3 X4.0 Z-3.5 R.5;

N2 G1 X4.1;

G0 G28 U0.0 W0.0;

G50 S2500;

T0202     ;

G96 S550 M4;

G0 G42 X4.1 Z.1;

G70 P1 Q2 F.008;

G0 G40 G28 U0.0 W0.0;

M30;
```

4.3 NIMS TURNING 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 Turning Set-Up & Operations. The credential will require the machining of a physical project (Figure 9.) and an online exam. The actual part will need to be machined to industry standards meeting the requirements of the Performance Assessment Worksheet (Figure 10.).The online exam will be a multiple choice format focusing on basic CNC Turning knowledge.

4.3.1 NIMS TURNING SET-UP & OPERATIONS PROJECT

Introduction: The following activity will require all turning knowledge acquired in this module; this process will include setting up and programming the CNC lathe to machine the **actual part** according to the specifications listed on NIMS Print (Figure 9.).

Materials and methods: The material will be 2.50 Diameter x 4.875 long aluminum stock being held with bored soft jaws.

Requirements for successful completion: This lab will require all safety procedures to be followed to complete the manufactured part. Tool lengths and offset set-ups including proper programming format, graphic evaluation coupled with first piece single-block application will be required. The machined part will match the print specifications. Once the physical part has been inspected by the sponsor and approved, the part will be inspected by two Met-Tec members to verify the dimensions. Once the inspection process is complete, an online exam will be taken to earn the NIMS Level I Turning Credential.
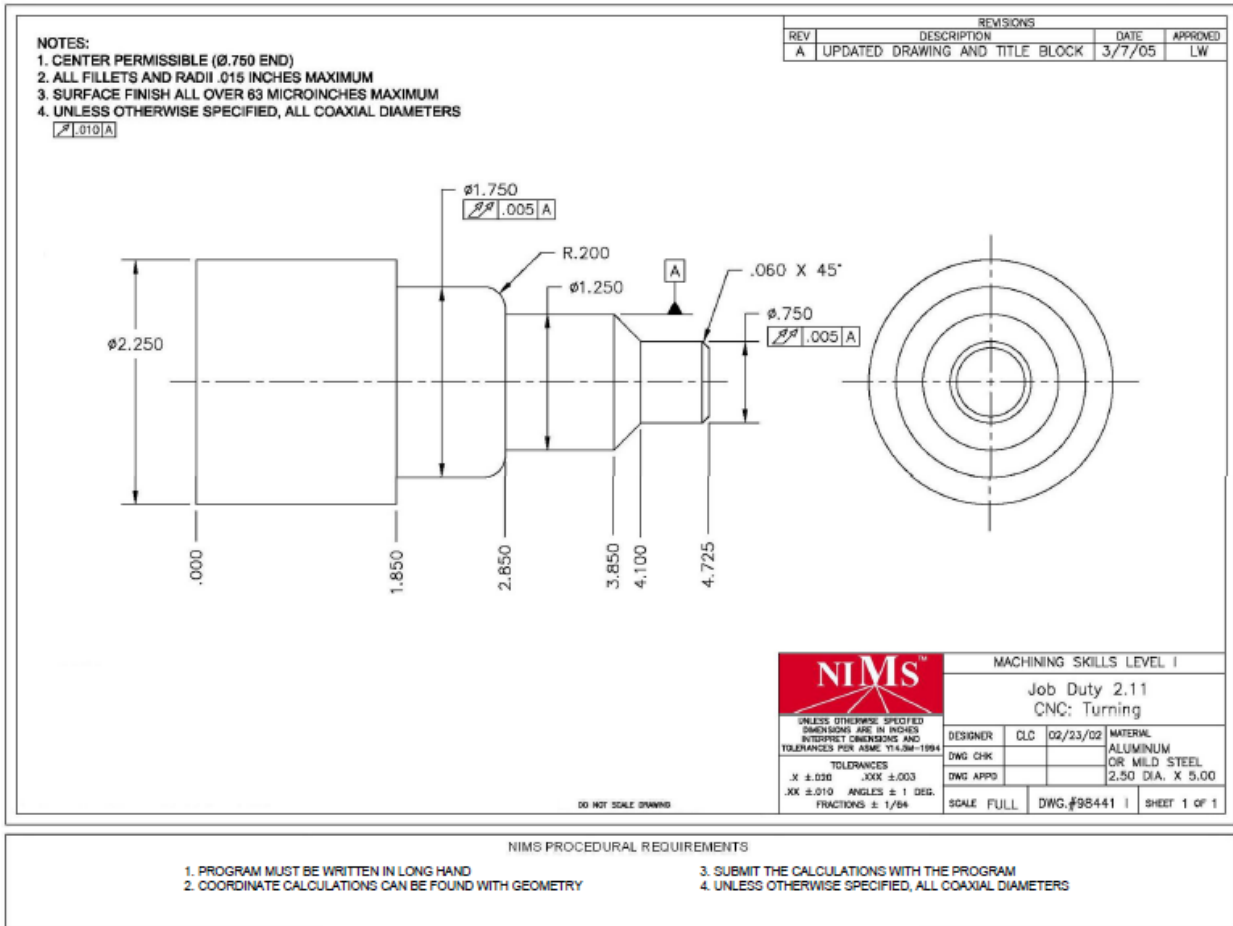
Figure 9. NIMS Print

Performance Standards
CNC Turning

Material
Aluminum or mild steel.

Duty
- Set up, program and operate a CNC lathe or turning center and manufacture a part within tolerance.
- Work from a process sheet.
- Understand the x, 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 all standards given on lathe operations (2.9) to develop a simple part (with linear and circular Interpolations).

Accuracy Level: Match the requirements of the part print.

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

Tooling: CNC lathe or turning center and computer workstation

Material as per print

Tooling as appropriate

Measuring instruments as needed

Reference: Operation process sheet

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 Assessment Worksheet
CNC Turning

INSTRUCTIONS: Rate the candidate's performance for the CNC Turning job according to the criteria below. The checklist below represents only a listing of criteria to be evaluated. It is not a sequence of process steps or a process plan for making the part. For each item, check the box under Pass or Fail accordingly.

Remember, NIMS requires that all specifications must be met within the allowable tolerance limits. If the part does not meet all specifications, the candidate/trainee must correct or redo the project.

Performance Project – CNC Turning Evaluation Criteria		Pass	Fail
1. 1.850 Length	Pass = within tolerance Fail = out of tolerance		
2. 2.850 Length	Pass = within tolerance Fail = out of tolerance		
3. 3.850 Length	Pass = within tolerance Fail = out of tolerance		
4. 4.100 Length	Pass = within tolerance Fail = out of tolerance		
5. 4.725 Length	Pass = within tolerance Fail = out of tolerance		
6. .750	Pass = within tolerance Fail = out of tolerance		
7. 1.250	Pass = within tolerance Fail = out of tolerance		
8. 1.750	Pass = within tolerance Fail = out of tolerance		
9. 2.250	Pass = within tolerance Fail = out of tolerance		
10. Runnouts .005	Pass = within tolerance Fail = out of tolerance		
11. Surface finish 63 Ra microinches min.	Pass = within tolerance Fail = out of tolerance		
12. Unless otherwise specified, all coaxial diameters .010	Pass = within tolerance Fail = out of tolerance		
13. Radius .200	Pass = within tolerance Fail = out of tolerance		
14. Chamfer .06 X 45°	Pass = within tolerance Fail = out of tolerance		
END OF CNC TURNING EVALUATION			

Figure 10. Performance Evaluation CNC Turning

Version | Course ID | Rev 1 2013

4.3.2 NIMS ONLINE EXAM PREPARATION

The **"Machinery's Handbook"** will also be the only reference material allowed to be accessed while taking the online NIMS CNC Turning examination. Proper use and knowledge of the Machinery's Handbook will be critical for passing the exam. Shop safety, measurement, print reading, formulas, and CNC coding will be the areas of concentration. The majority of the exam will focus on basic CNC knowledge. Being able to locate and properly reference the material will be the key to successfully completing the exam. The online exams have a time limit of 90 minutes to complete. General sample questions that may be included are listed in (Figure 11.).

1) The safest way to remove chips from a lathe or milling machine is to use a:
   - a) Air hose and high pressure nozzle
   - b) Glove covering the operator's hand
   - c) Shop rag soaked in solvent
   - d) Brush

2) The most important factor in selecting measuring tools for inspection is:
   - a) Access to measuring equipment
   - b) Process capability of the part in assembly
   - c) Tolerancing of dimensions to inspect
   - d) Verification of the sampling plan

3) Apply the *Machinery's Handbook* to determine the hole size for a Class FN2 force fit if the shaft diameter is 1.625 inches.
   - a) 1.6181 inches
   - b) 1.6275 inches
   - c) 1.6203 inches
   - d) 1.6238 inches

4) Referring to the *Machinery's Handbook,* what is the minimum pitch diameter for an external thread 7/16-14 UNC - 2A?
   - a) 0.3862 inches
   - b) 0.3909 inches
   - c) 0.385 inches
   - d) 0.3876 inches

5) Referring to the *Machinery's Handbook,* what is the tap drill for a 5/16-18 UNC thread with a 75% thread engagement?
   - a) 0.272-inch diameter drill
   - b) 0.257-inch diameter drill
   - c) 0.2983-inch diameter drill
   - d) 0.201-inch diameter drill

Figure 11. Sample Questions

The following formulas (Figure 12.) can be used to solve the following problems for calculating **Length of Taper**, **Revolutions per Minute** and **Cutting Time.** (Figure 13.) contains examples of word problems with the correct data   inserted into the proper formula with the   outcomes.

$$\text{Taper Per Foot}$$

$$TPF = \frac{(D - d)}{\text{Length of Taper}} \times 12$$

$$\text{Revolutions Per Minute}$$

$$RPM = \frac{CS \times 4}{\text{Diameter}}$$

$$\text{Cutting Time}$$

$$\text{Time} = \frac{L}{FN}$$

Figure 12. Formulas

1) Applying the formula below, find the taper per foot for a workpiece with a large diameter of 3.5 inches and a small diameter of 2.75 inches with a taper length of 5.5 inches.
   a) 3.5 inches
   b) 1 .64 inches
   c) 3.27 inches
   d) 1.75 inches

Taper Per Foot

$$1.64 = \frac{(3.5 - 2.75)}{5.5} \times 12$$

2) Calculate the rpm for a drilling operation with a cutting speed of 150 SFM applying a 1/4" diameter drill with a feed rate of 0.005 inches per revolution.
   a) 2400 RPM
   b) 7500 RPM
   c) 4800 RPM
   d) 3750 RPM

Revolutions Per Minute

$$2400 = \frac{150 \times 4}{.250}$$

3) The time required to make three turning passes over a piece of stainless steel 8.5 inches in length at 350 rpm with a 0.006" feed is approximately minutes.
   a) 26.7 minutes
   b) 12.1 minutes
   c) 10.7 minutes
   d) 4.1 minutes

Cutting Time

$$4.04 = \frac{8.5}{.006 \times 350}$$

$$4.04 \times 3 \text{ Passes} = 12.1$$

Figure 13. Word Problems

ACTIVITY #3

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

1. Referring to the Machinery Handbook, what is the maximum 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 ½-20 UNC-2B with 75% thread engagement?

3. What is the largest square that can be machined from a 2.0 diameter piece of stock?

4. Calculate the RPM's for a drill operation with a cutting speed of 90 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 4.5 and a small diameter of 3.5 with a taper length of 6.0 inches.

6. What is the time required to make two turning passes on a piece of stainless steel 8.0 inches long at 400 RPM's with a .012 feed rate?

7. What is the TPF for a piece 7.5 inches long with a .090 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 3/4 SHCS?

10. What is the normal clearance size for a 5/8 bolt?

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

12. What is the bore range for 1-12 UNF-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 #3

Answers
1. .3331
2. # 7 / .201
3. 1.414
4. 917 RPM's
5. 2.0
6. 3.33
7. 144
8. 1.125 – 1.126
9. 1.1875
10. 21/32 or .656
11. 90 Degrees
12. .910 - 928
13. 1 / TPI
14. .06 + (.013 x # of Screw)
15. Major Dia. - Pitch

MAJOR CONCEPTS

KEY CONCEPTS

- CNC Turning 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 "**Rough Machining**" and "**Finish Machining**". Rough Machining cycles are designed to remove as much stock as possible. Finish Machining is designed to accurately complete the part.
- **"Tool Nose Radius Compensation"** enables accurate machining of angles and radii. G41 and G42 are the two G codes used to accurately detail the tool nose radius compensation for the cutting insert.
- **"Point Machining"** uses the tip or end of the cutter to remove stock. Common point machining practices include drilling, tapping, and counter boring.

KEY TERMS

CNC Turning 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.

Rough Machining: G71 mode removing the bulk of stock.

Finish Machining: G70 controlling the final machining pass to complete contour.

Order of Operations: Sequence of program arrangement.

Canned Cycle: Activates a sequence of movements.

Sequence Numbers: Reference numbers at the beginning of certain program lines.

Workshift: Controls "Z" positioning to establish tool lengths.

Home Position: The absolute machine zero position.

Offset Page: The page were tool offsets for "X" and "Z" are recorded.

Stock Removal: The process of removing stock by machining.

G41 & G42: Left and Right Tool Nose Radius Compensation.

G40: Cancels Tool Nose Radius Compensation.

T0101: Tool and Offset Call.

G50: Maximum RPM Spindle Speed.

G96: Constant Surface Speed.

NIMS: National Institute for Metalworking Skills

ASSESSMENT

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

1. Worshift establishes "Part Reference Zero" for "X".	T	or	F
2. Rough Machining is activated by G70.	T	or	F
3. The code for activating Tool Nose Compensation Left is G41.	T	or	F
4. The code for activating Tool Nose Compensation Right is M42.	T	or	F
5. G40 cancels cutter compensation.	T	or	F
6. G50 may be a Preparation Code.	T	or	F
7. Sequence Numbers are mandatory in canned cycles.	T	or	F
8. The "P" and "Q" refer to sequence numbers.	T	or	F
9. Wear offsets are used for initial set-up.	T	or	F
10. Offsets for "X" are independent references.	T	or	F

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

1. Select the correct line sequence for setting Maximum RPM.
    A. GO G40 S2500
    B. G40 M3 S2500
    C. G50 S2500
    D. None of the Above
2. Select the correct line sequence for establishing a tool call.
    A. T0101
    B. TO1O1
    C. T01
    D. M6 T01
3. Select the correct line sequence for establishing a G70.
    A. G70 P1 Q2 F.008
    B. G70 N1 Q2 F.008
    C. G70 P1 Q2 F.120
    D. None of the Above
4. Select the correct line sequence for establishing G42.
    A. G0 G42 X1.1 Z.1
    B. G0 G41 X1.1 Z.1
    C. G1 G42
    D. None of the Above
5. Select the correct G-code for Roughing.
    A. G80
    B. G71
    C. G70
    D. G84

6. Select the correct code for Spindle CW.
    A. G03
    B. G02
    C. M3
    D. M4
7. Select a correct roughing feedrate.
    A. .012 IPM
    B. .012 IPR
    C. .012 RPM
    D. None of the Above
8. Select the correct line sequence for establishing constant surface speed.

A. G97 S550 M3
B. G96 S550 M4
C. G95 M3 S550
D. None of the Above

9. Select the correct line sequence for Home Positioning
    A. G28 U0.0 W0.0
    B. G28 X0.0 Z0.0
    C. G28 X0.0 Y0.0 Z0.0
    D. None of the Above
10. Select the correct line sequence for a Radius move.
    A. G1 X1.0 Z-.125 R.125
    B. G2 X1.0 Z-.125 R.090
    C. G3 X1.0 Z-.250 R.250
    D. None of the Above

Answer Key

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

DISCUSSION PROMPTS
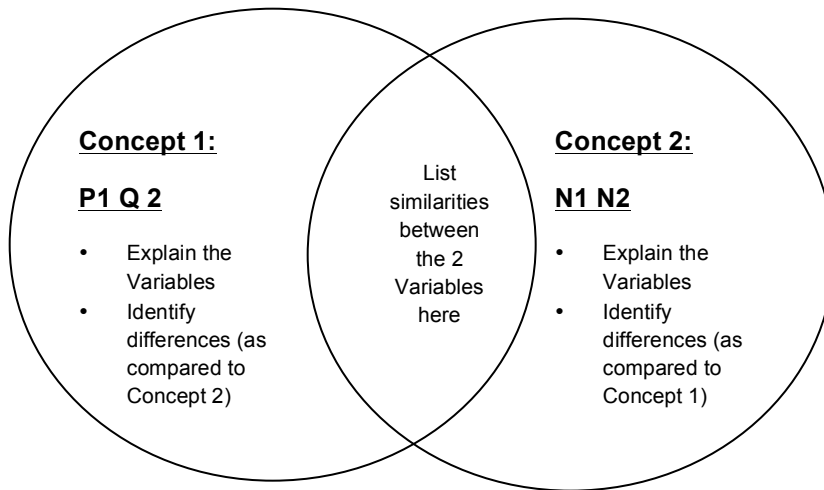
ROUGHING AND FINISHING CYCLES

In Module 4, we machined geometry using roughing and finishing methods. What are the different functions and purposes of both methods for removing stock?

CUTTER COMPENSATION LEFT AND RIGHT

In Module 4, we used Tool Nose Compensation left and right, what are the advantages of using tool nose compensations?

CRITICAL THINKING

	Concept #1 G71 Roughing Cycle	Concept #2 G70 Finish Cycle
Define or explain each concept		
Explain how the concepts are similar		
Explain how each concept is different with respect to specific attributes		

Concept 1:

P1 Q 2

- Explain the Variables
- Identify differences (as compared to Concept 2)

List similarities between the 2 Variables here

Concept 2:

N1 N2

- Explain the Variables
- Identify differences (as compared to Concept 1)

- Explain Variables used for references.

SC ACCELERATE

CLEMSON

Version | Course ID | Rev 1 2013

ATTRIBUTION TABLE

Author/s	Title	Source	License
Mark Cramer	Figure 1. Start-up Commands	Module Author	CC BY-SA 4.0
Mark Cramer	Figure 2. Stock Removal and Turning Codes	Module Author	CC BY-SA 4.0
Mark Cramer	Figure 3. Canned Cycle Finish Turning Cycle	Module Author	CC BY-SA 4.0
Created and Maintained by the Real World Software Developers and Machinists at www.KentechInc.com	Figure 4. Tool Nose	Kipware.blogspot.com	CC BY-SA 4.0
Mark Cramer	Figure 5. Turning Program	Module Author	CC BY-SA 4.0
Mark Cramer	Figure 6. Finish Cycle G70/G71/G42 Print	Module Author, developed using AutoCad software.	CC BY-SA 4.0
Mark Cramer	Figure 7. Program O2234	Module Author	CC BY-SA 4.0
Mark Cramer	Figure 8. 2234 Print	Module Author, developed using AutoCad software.	CC BY-SA 4.0
National Institute for Metalworking Skills, Inc.	Figure 9. NIMS Print	NIMS-SKILLS.org	CC BY-SA 4.0
National Institute for Metalworking Skills, Inc.	Figure 10. NIMS Evaluation	NIMS-SKILLS.org	CC BY-SA 4.0