

Content

Conversational CAM
Conversational CAM overview .................................................. 3
Turning Round overview .................................................. 4 - 5
Turning Tool Profile overview ................................................. 6
Multi-sided overview .................................................. 7
Indexing overview .................................................. 8
Rope & Barley Twist overview ........................................... 9 - 10
Spiral overview .................................................. 11
Surface Planing overview .................................................. 12
Linear Cut overview .................................................. 13
Tool Setup overview .................................................. 14
Saving, Naming & Moving Code .................................. 15

Conversational CAM Workbook
Turning Round - single step ............................................... 16
Turning Tool Profile with Indexing - multiple step ........... 17 - 19
Rope & Barley Twist - single step ...................................... 20

Fast Start Reference Sheets
Arty Fast Start—Turning .................................................. 21
Arty Fast Start—Flat Stock .................................................. 22
Conversational CAM Overview

Is a “Computer Aided Manufacturing” program which uses specific questions to design turnings and flat milling then produces the G-code needed to run the Legacy CNC milling machine. Conversational CAM was designed by Legacy to streamline the design process of commonly milled components without the use of a CAD (computer aided design) software program to draw the parts. Begin by opening Conversation CAM found on your computer desk top, Start quick launch or toolbar. Enable Macros

The Main Menu offers a choice of:

- Turning Round
- Turning Tool Profiles
- Multi-Sided
- Indexing
- Rope & Barley Twist
- Spirals
- Surface Planing
- Linear cuts
- Tool List
- Settings

From the Main Menu window go to “Settings”

CNC Series: Enter the model you own
Spindle: Choose Spindle or Router
Turning Capacity: Choose the length the matches your machine length.

Milling Between Centers Programs

Turning Round: is designed to quickly turn stock round.

Turning Tool Profiles: is designed to mill tool (router bit) profiles into turned stock, such as beads and coves.

Multi-Sided: is designed to mill straight or tapered multi sided cross sections such as square sections on balusters or table legs.

Indexing: is designed to add indexing components to round stock, such as flutes and reeds.

Rope & Barley Twist: is designed to make single start or multiple start spirals based on classic rope and barley designs. The pitch is determined automatically by the size of the cutter and the diameter of the work piece.

Spirals: is designed for making single start or multiple start spirals based on pitch information you supply.

Flat Stock Programs

Surface Planing: is designed to surface plane flat stock.

Linear Cuts: is designed to add flutes, reeds and bit profile accents to flat stock.

Reference & Setting Screens

Tool List: is a reference screen for tool (router bit) information. You build this list including the name, diameter, plunge depth and part number for your router bits.

Settings: sets your machine model parameters for Conversational CAM
Turning Round  You will use this interface when you wish to turn any part of your work piece round. This program references from the shoulder of the cutter.

Is this the first step of the program? By answering Yes you place a preamble G-code line into your program necessary for running the program. By answering No, no preamble is written. Answer yes if this is the only step in the program or if it will be the first step in a multi process program. Otherwise, answer no.

Is this the last step of the program? By answering Yes, you place a ending G-code process into your program which correctly stops and rewinds the program. By answering No, no ending process in written. Answer yes if this is a single process program or the last step in a multi process program.

Is this the same tool as the previous step? This question will only be available if you are building a multiple process program. If you answer Yes, no tool change will be written into the code. If you answer No, a tool change will be written into the code.

Enter Tool number: This is a reference line. Enter 1 for a single step process or enter the number the bit is in the process. In a four step process enter 1 - 4. You can also view the Tool List screen and access the bit information by clicking the Tool List button. You build the Tool List file, adding bit information as you need.

Enter the spindle RPM: If you have a Spindle, this will set the RPM’s within the program. If you are using a manual Router, this will generate a line of code to remind you to turn on your router and set the RPMs.

Enter the tool diameter: This is the diameter of the cutter you are using.

Enter the step over per rotation: This programs the distance the X-axis moves in relation to the A-axis rotation. We recommend .25 for rough cuts and .125 for a smoother finish.

Enter the type of cut: Bottom or Side; match the router bit you are using in this step.

Enter starting Position: This will program where, on your work piece, you want to begin the round turning. For Turning Round the program will place the router bit shoulder at the given starting position, not the center of the bit.

Enter ending position: This will program where, on your work piece, you want to end the round turning. For Turning Round the program will place the router bit shoulder at the given ending position.

Enter the feed rate: The recommended feed rate is 200 - 400 IPM. The feed rate determines how fast the cutter is moving down the work piece.
Conversational CAM

Turning Round continued.

Select the tool touch off method: This will program the type of tool touch off selected: Smart Tool, Tool Offset, Part and Bull Nose. Bull Nose is used with the Standard CNC system only. Go to the Tool Touch Off Method in this manual for more information on tool touch off methods.

Enter the stock diameter: This is the finished diameter of your part. It is recommended that you do not remove more than .25” in a single pass.

Enter the length of the part: This will be made available when using the Bull Nose as the “tool touch off” method. When asked, enter the length of your part.

Enter the plunge depth: Z “0” is referenced from the surface of the part, therefore the plunge depth is the depth of the cut required to finish the part to the desired diameter. It is recommended that you do not remove more than .25” in a single pass.

Note: A red square in the upper right corner of the answer box represents an available help window. These help windows are currently available on the “Turning Round” screen only.

* Once all questions are answered, press the Enter key to generate the G-code
Conversational CAM

**Turning Tool Profiles** interface. This interface is designed to mill router bit profiles into turned stock, such as a bead or cove. This program references the center of the cutter to the X-axis starting position.

**Is this the first step of the program?** Answer yes if this is the only step in the program or if it will be the first step in a multi process program. Otherwise, answer no.

**Is this the last step of the program?** Answer yes if this is a single process program or the last step in a multi process program.

**Is this the same tool as the previous step?** This questions will only be available if you are building a multiple process program.

If you answer Yes, no tool change will be written into the code. If you answer No, a tool change will be written into the program.

**Enter Tool number:** This is a reference line. Enter 1 for a single step process or enter the number the bit is in the process. In a four step process enter 1 - 4.

**Enter the spindle RPM:** If you have a Spindle, this will set the RPM’s within the program. If you are using a manual Router, this will generate a line of code to remind you to turn on your router and set the RPMs.

**Enter the plunge depth:** Enter the whole or partial tool profile depth you wish to cut into the work piece.

**Enter the X axis position of the cut:** This will determine the cutting point along the X-axis in reference to the bit diameter. If you place a cut at two inches, the center of the bit will cut at two inches.

**Select the side to climb mill:** This is important when leaving a shoulder on either end of the cut. If the shoulder is on the left side of the work piece, climb mill left. If the shoulder is on the right side of the work piece climb mill right.

**Enter the feed rate:** The recommended feed rate is 80-120 IPM. The feed rate determines how fast the cutter is moving down the work piece.

**Select the tool touch off method:** This will program the type of tool touch off selected: Smart Tool, Tool Offset, Part and Bull Nose. (Bull Nose is used with the Standard CNC system only).

**Enter finished diameter:** When asked, enter the finished diameter of your part.

**Enter the length of the part:** This will be made available when using the Bull Nose as the “tool touch off” method. When asked, enter the length of your part.
**Conversational CAM**

**Multi Sided** interface is designed to mill straight or tapered multi sided cross sections such as square sections on balusters or table legs. This program references the shoulder of the cutter to the X-axis starting position.

**Is this the first step of the program?** Answer yes if this is the only step in the program or if it will be the first step in a multi process program. Otherwise, answer no.

**Is this the last step of the program?** Answer yes if this is a single process program or the last step in a multi process program.

**Is this the same tool as the previous step?** This question will only be available if you are building a multiple process program.

If you answer Yes, no tool change will be written into the code. If you answer No, a tool change will be written into the program.

**Enter Tool number:** This is a reference line. Enter 1 for a single step process or enter the number the bit is in the process. In a four step process enter 1 - 4.

**Enter the spindle RPM:** If you have a Spindle, this will set the RPM’s within the program. If you are using a manual Router, this will generate a line of code to remind you to turn on your router and set the RPMs.

**Enter the tool diameter:** This program was designed for surface planing bits only. Enter the diameter of the router bit you are using.

**Enter the number of sides:** This will program the number of surfaces to be milled, the most common being 4, 6 or 8.

**Enter the starting position:** This will program where, on your work piece, you want to begin the milled flat section. For Multi Sided milling the program will place the router bit shoulder at the given starting position, not the center of the bit.

**Enter the ending position:** This will program where, on your work piece, you want to end the milled flat section. For Multi Sided milling the program will place the router bit shoulder at the given ending position.

**Enter the feed rate:** The recommended feed rate is 80-120 IPM. The feed rate determines how fast the cutter is moving down the work piece.

**Select the tool touch off method:** This will program the type of tool touch off selected: Smart Tool, Tool Offset, Part and Bull Nose. (Bull Nose is used with the Standard CNC system only).

**Enter the finished radius:** This is the measurement from the center of your part to any of the finished surfaces. Such as— you want a 3” square, your finished radius is 1.5”.

**Enter the length of the part:** Enter the length of your part.

**Enter the plunge depth:** Enter the amount you want to remove from the surface of the part. Recommended maximum single pass plunge depth is .125
Indexing interface. This interface is designed to add indexing components to round stock, such as flutes and reeds. This program references the center of the cutter to the X-axis starting position.

Is this the first step of the program? Answer yes if this is the only step in the program or if it will be the first step in a multi process program. Otherwise, answer no.

Is this the last step of the program? Answer yes if this is a single process program or the last step in a multi process program.

Is this the same tool as the previous step? This question will only be available if you are building a multiple process program.

If you answer Yes, no tool change will be written into the code. If you answer No, a tool change will be written into the program.

Enter Tool number: This is a reference line. Enter 1 for a single step process or enter the number the bit is in the process. In a four step process enter 1 - 4.

Enter the spindle RPM: If you have a Spindle, this will set the RPM’s within the program. If you are using a manual Router, this is a reminder reference that will appear in the code to turn on your router and set the RPMs.

Enter the tool cutting depth: Depending on the router bit used, the depth will determine the size and shape of your individual cut.

Enter the number of starts: This represents the number of starts for this exact cut located equally around the diameter of your work piece.

Enter the starting position: The starting point for the indexed cut to begin on the work piece, keeping in mind this will measure from the center of the bit.

Enter the ending position: The ending point for the indexed cut to end on the work piece, keeping in mind this will measure from the center of the bit.

Enter the feed rate: The recommended feed rate is 80-120 IPM. The feed rate determines how fast the cutter is moving down the work piece.

Select the tool touch off method: This will program the type of tool touch off selected: Smart Tool, Tool Offset, Part and Bull Nose. (Bull Nose is used with the Standard CNC system only).

Enter finished diameter: When asked, enter the finished diameter of your part.

Enter the length of the part: This will be made available when using the Bull Nose as the "tool touch off" method. When asked, enter the length of your part.
Conversational CAM

**Rope and Barley Twist** interface. You will use this interface to make single start to multiple start spirals based on classic rope and barley designs. The pitch is determined automatically by the size of the cutter, the diameter of the work piece and the number of starts. The classic barley spiral design is usually a single start spiral. The classic rope spiral design is usually a two or three start spiral. This program references the center of the cutter to the X-axis starting position.

**Is this the first step of the program?** Answer yes if this is the only step in the program or if it will be the first step in a multi process program. Otherwise, answer no.

**Is this the last step of the program?** Answer yes if this is a single process program or the last step in a multi process program.

**Is this the same tool as the previous step?** This question will only be available if you are building a multiple process program.

If you answer Yes, no tool change will be written into the code. If you answer No, a tool change will be written into the program.

**Enter Tool number:** This is a reference line. Enter 1 for a single step process or enter the number the bit is in the process. For example, in a four step process enter 1 - 4.

**Enter the spindle RPM:** If you have a Spindle, this will turn on and set the spindle at this RPM. If you are using a manual Router, this is will generate a line of code to remind you to turn on your router and set the RPMs.

**Enter the tool diameter:** The diameter of the cutter you are using. When referenced to the finished diameter and the number of starts this information will help determine the pitch of the spiral.

**Enter the tool cutting depth:** Depending on the cutter used, the depth will determine the size and shape of your individual cut. When doing a spiral a bead is usually formed by cutting the full router bit profile height.

**Enter the number of starts:** The number of starts for the spiral design. When referenced to the tool diameter and finished diameter this information will help determine the pitch of the spiral.

**Enter the starting position:** Where do you want the spiral cut to begin on the work piece, keeping in mind this will measure from the center of the bit.

**Enter the ending position:** Where do you want the spiral cut to end on the work piece, keeping in mind this will measure from the center of the bit.
Rope and Barley Twist continued.

Select the direction of the spiral: You have a choice of Right of Left. Right is clockwise, Left is counter clockwise.

Enter the feed rate: The recommended feed rate is 80-120 IPM. The feed rate determines how fast the cutter is moving down the work piece.

Select the tool touch off method: This will program the type of tool touch off selected: Smart Tool, Tool Offset, Part and Bull Nose. (Bull Nose is used with the Standard CNC system only).

Enter finished diameter: When asked, enter the finished diameter of your part.

Enter the length of the part: This will be made available when using the Bull Nose as the ”tool touch off” method. When asked, enter the length of your part.

Note: This program has been designed to cut the spiral in two passes. The first pass will move down the work piece removing 70% of the cutting depth. The return pass will complete the pass to the full cutting depth.

When milling spirals, it is beneficial to start and end spirals on a cove. The cove can be programmed in the “Turning Tool Profiles” using the same bit used to cut the spiral.

To achieve the pineapple cut, repeat this process twice, exactly the same, except change the spiral direction.
**Conversational CAM Overview**

**Spirals** interface. This interface is designed for making single start to multiple start spirals based on pitch. This program references the center of the cutter for the X-axis cutting position. This program references the center of the cutter.

**Is this the first step of the program?** Answer yes if this is the only step in the program or if it will be the first step in a multi process program. Otherwise, answer no.

**Is this the last step of the program?** Answer yes if this is a single process program or the last step in a multi process program.

**Is this the same tool as the previous step?** This question will only be available if you are building a multiple process program. If you answer Yes, no tool change will be written into the code. If you answer No, a tool change will be written into the program.

**Enter Tool number:** This is a reference line. Enter 1 for a single step process or enter the number the bit is in the process. For example, in a four step process enter 1-4.

**Enter the spindle RPM:** If you have a Spindle, this will set the RPM’s within the program. If you are using a manual Router, this is will generate a line of code to remind you to turn on your router and set the RPMs.

**Enter the spiral pitch:** This enables you to determine the pitch of the spiral design. Pitch = the length the bit travels down the X-axis to one A-axis rotation.

**Enter the tool cutting depth:** Depending on the router bit used, the depth will determine the size and shape of your individual cut. When doing a spiral a bead is usually formed by cutting the full router bit profile height.

**Enter the number of starts:** The number of starts for the spiral design.

**Enter the starting position:** Where do you want the spiral cut to begin on the part, keeping in mind this will measure from the center of the bit.

**Enter the ending position:** Where do you want the spiral cut to end on the part, keeping in mind this will measure from the center of the bit.

**Select the direction of the spiral:** You have a choice of Right or Left. Right is clockwise, Left is counter clockwise.

**Enter the feed rate:** The recommended feed rate is 80-120 IPM. The feed rate determines how fast the cutter is moving down the part.

**Select the tool touch off method:** This will program the type of tool touch off selected: Smart Tool, Tool Offset, Part and Bull Nose. (Bull Nose is used with the Standard CNC system only).

**Enter finished diameter:** When asked, enter the finished diameter of your part.

**Enter the length of the part:** This will be made available when using the Bull Nose as the “tool touch off” method. When asked, enter the length of your part.
Conversational CAM

**Surface Planing** interface. This interface is designed to surface plane flat stock using a surface planing or bottom cleaning bit. This program references the shoulder of the cutter to the X-axis starting position.

**Is this the first step of the program?** Answer yes if this is the only step in the program or if it will be the first step in a multi process program. Otherwise, answer no.

**Is this the last step of the program?** Answer yes if this is a single process program or the last step in a multi process program.

**Is this the same tool as the previous step?** This question will only be available if you are building a multiple process program.

If you answer Yes, no tool change will be written into the code. If you answer No, a tool change will be written into the program.

**Enter Tool number:** This is a reference line. Enter 1 for a single step process or enter the number the bit is in the process. For example, in a four step process enter 1 - 4.

**Enter the spindle RPM:** If you have a Spindle, this will set the RPM’s within the program. If you are using a manual Router, this is will generate a line of code to remind you to turn on your router and set the RPMs.

**Enter the tool diameter:** This is the diameter of the cutter you are using.

**Enter the depth of cut:** This is the depth you wish to be removed from the surface of the part. The depth can vary from 0 to .25”.

**Enter the width:** This is the width of the part.

**Enter the X-axis starting position:** This will establish the starting point for the milling pass. A negative number will move the cutter off the wood.

**Enter the X-axis ending position:** This will determine the ending point of the milling pass. It is recommended that you move the cutter off the part. Therefore - cutter diameter + part length = X-axis ending position.

**Enter the feed rate:** The recommended feed rate is 180-200 IPM. The feed rate determines how fast the cutter is moving down the work piece.

**Select the tool touch off method:** This will program the type of tool touch off selected: Smart Tool, Tool Offset, Part and Bull Nose. Bull Nose is used with the Standard CNC system only.
Conversational CAM Overview

**Linear Cuts** interface. This interface is designed for making linear tool paths along your workpiece such as flutes and reeds. As well as router bit profiles for creating moulding work. The program references the center of the cutter for the X & Y axis starting positions.

*Is this the first step of the program?* Answer yes if this is the only step in the program or if it will be the first step in a multi process program. Otherwise, answer no.

*Is this the last step of the program?* Answer yes if this is a single process program or the last step in a multi process program.

*Is this the same tool as the previous step?* This question will only be available if you are building a multiple process program.

If you answer Yes, no tool change will be written into the code. If you answer No, a tool change will be written into the program.

**Enter Tool number:** This is a reference line. Enter 1 for a single step process or enter the number the bit is in the process. For example, in a four step process enter 2, 3 or 4.

**Enter the spindle RPM:** If you have a Spindle, this will set the RPM’s within the program. If you are using a manual Router, this is will generate a line of code to remind you to turn on your router and set the RPMs.

**Enter the tool cutting depth:** The will create the programs cutting depth for the router bit you are using.

**Enter the number of parallel cuts:**

**Enter the Y-axis spacing:** When making multiple cuts, this will program the spacing between multiple cut along the Y-axis. This measurement will be from center of cut to center of cut.

**Enter the Y-axis starting position:** Programs the starting position on the Y-axis. The Y-axis spacing is calculated from this position.

**Enter the X-axis starting position:** Programs the starting point on the X-axis. This starting point will be at the center of the router bit cut.

**Enter the X-axis ending position:** Programs the ending point on the X-axis. This ending point will be at the center of the router bit cut.

**Enter the feed rate:** The recommended feed rate is 80-120 IPM. The feed rate determines how fast the cutter is moving down the work piece.

**Select the tool touch off method:** This will program the type of tool touch off selected: Smart Tool, Tool Offset, Part and Bull Nose. (Bull Nose is used with the Standard CNC system only).
Conversational CAM Overview

**Tool Setup** reference screen. This reference screen allows you to make a personalized bit library for use while writing Conversational CAM programs. **THIS IS A REFERENCE LIBRARY ONLY!**

The Tool List has been designed in a simple spreadsheet format. If you double click on any cell you can change the information contained in that cell.

**Tool Number:** Is a numeric list, but can be changed to meet your personal format.

**Router Bit Number:** We used the part number from the company we order our router bits from for easy reordering. You may reference any information you would like.

**Cutting Diameter:** Enter diameter of the router bit.

**Profile Height:** This is the cutting height of the profile for the router bit.

   **Measuring the profile height:**
   You will need a set of calipers and a flat block approximately 2 x 3 inches in size. Lay the router bit on a flat work surface. Slide the block up to the router bit cutting tip until they are touching. Align the bit and block parallel. Measure the space between the block and the router bit shoulder with the calipers.

   ![Diagram of measuring profile height](Diagram)

   This is not a perfect measurement but it a good starting point.

**Height Offset:** Is the difference between one tool and another tool in reference to a reference tool. We recommend that you use Smart Tool, if available, or touch off the Part or Bull Nose.

**Description:** Enter the description of your router bit.

**Main Menu:** will take you back to the main menu from any page.

**Clear Fields:** will clear all question fields on the page you are currently viewing.

**Copy Code:** will copy the code to the windows clipboard.
Conversational CAM

Saving, naming and moving G-code from Conversation CAM to the OM5 Control Software.

When you finish answering all the available questions a hard return will create the G-code. Once the G-code has been generated—click the “Copy Code” button. This will copy the code to the Windows clip board.

Open “Notepad”
Found on the quick launch icon toolbar at the bottom left of your computer screen
Or
On the Windows Start menu.
A Notepad shortcut icon has also be placed on your desktop.

Select “Edit” from the Notepad menu bar to drop down the Edit menu. Do not use the “Edit” menu from the Conversational CAM program.

Select “Paste” - the code stored on the Windows clip board should appear in the Notepad screen.

You can also right click your mouse to bring up the “paste” option.

Once the G-code is pasted onto the Notepad screen, go to:
File then Save.

Save to My Documents in the G-code folder.
Close Notepad
This file is now saved on the hard drive and is available for downloading into the OM5 Control Software.
**Turning Round** workbook. This will take you step by step through turning round. You will need a 1.5” x 10” piece of stock and the 1.25” Surface Planing Bit (Magnate Bit #2704). Begin by mounting the stock into your machine, using one of the supplied Index Hubs. This program will mill a shoulder 1” from the headstock end of the stock. You should always leave enough material at the Headstock end to avoid hitting the mounting screws.

In Conversational CAM—go to the Turning Round screen. Write a program following the outline to the right.

- Is this the first step of the program? Yes
- Is this the last step of the program? Yes
- Enter the tool number: 1
- Enter the spindle RPM: 19000
- Enter the tool diameter: 1.25
- Enter the step over per rotation: 0.25
- Enter the type of cut: Bottom
- Enter the starting position: 1
- Enter the ending position: 10
- Enter the feed rate: 200
- Select the tool touch off method: Smart Tool
- Enter the stock diameter: 1.5

This is the finished stock diameter
This field is not used
- Press Enter -

Your code should generate on the right side of the screen and should match the code pictured below.

Click the “Copy Code” button at the top of the questions field.

Open Note book—Paste and Save to G-code file.

Move to the OM5 Control Software and run the “Set Machine Coordinates” if it has not already been run.

**OM5 - Work Offsets screen**
- Enter 1.5 in the Rotation Feed Rate box

**OM5 - Tool Setup screen** - Turn Smart Tool A On

**OM5 - Run Machine screen** - Load Code
- Find your saved code
- Begin the program - Cycle Start

The program will bring the spindle forward for a router bit change. Click Cycle Start again to begin the Smart Tool touch off. The program will begin the milling process.
Fill out the form using these parameters:
Blank - 1.5” x 10” (you can use the round blank from the turning Round work sheet)
Classic Plunge Router Bit 1.5”
1st step of a multiple step program

This is the first step of the program
This is NOT the last step of the program
N/A
The tool number is 2
The spindle RPM is 19000
The plunge depth is 0.435
The X-axis position of the cut is 5
The side to climb mill is the left
The feed rate is 80
The tool touch off method is smart tool
The finished diameter is 1.5
N/A

Pressing the enter or tab button will generate the code onto the right side of the screen.

The code generated should look like the example code to the right.

Take the time to check your code line by line. If there is a difference, review the questions above, make the needed corrections, hard return (Enter) to make the changes to the code screen.

Copy the code into notepad.

Because this is a multiple step program, you will not save the code at this time.
Fill out the form using these parameters:
Blank - 1.5” x 10” - You will be using the part from the “Turning Tool Profile” worksheet.
2” Classic Spiral Bit
Last step of a multiple step program

This is NOT the first step of the program
This is the last step of the program
This is the same tool as the previous step
N/A
The spindle RPM is 19000
The cutting depth is 0.07
The number of starts is 6
The starting position is 5
The ending position is 1.5
The feed rate is 200
The touch off method is smart tool
The finished diameter is 1.5
N/A

Pressing the enter or tab button will generate the code onto the right side of the screen.

The code generated should look like the example code to the right.

Take the time to check your code line by line. If there is a difference, review the questions above, make the needed corrections, hard return (Enter) to make the changes to the code screen.

Copy the code into notepad.

It is important to remember when adding code to notepad with a multi process program, code is pasted at the end of the code and above any sub-routine code.

Save your code into the hard drive file Desktop—Documents—G-code giving your code a descriptive name.

Your code is ready to use in the OM5 Control Software.
As you paste the second code into notepad, you
should have a code that looks like the example to the
right. Note the “Turning Tool Profile” code listed at
the beginning and the “Indexing” code listed sec-
don. If your code differs from what is shown, check
your question screens and code placement.

When two or more steps are programmed together,
some basic code can be removed to help the program
run more efficiently. These excess codes generally
are commands that either pause the machine or turn
the Spindle on or off. They can be deleted.

M5 at the end of the “Turning Tool Profile” code
section turns the spindle off, which is not necessary
because we are not making a tool change.
G53 G0 Z-0.125 moves the Z axis to –0.125 posi-
tion on the machine coordinates. This code is not
needed because the Z-axis was already raised at the
end of the “Turning Tool Profile” section.
M3 S19000 turns the spindle on, but it is already on.
G4 P8 is a pause in the program to wait for the spin-
dle to reach the given RPMs.
Fill out the form using these parameters:
Blank: 10” x 1.5” - use the rounded, fluted piece already cut from the previous programs.
Rope Moulding Bit 1” diameter
Single step program

This is the first step of the program
This is the last step of the program
N/A
The tool number is 3
The spindle RPM is 21000
The tool diameter is 1
The tool cutting depth is 0.15
The number of starts is 3
The starting position is 5
The ending position is 10
The direction of the spiral is left
The feed rate is 200
The touch off method is smart tool
The finished diameter is 1.5
N/A

Pressing the enter or tab button will generate the code onto the right side of the screen.

The code generated should look like the example code to the right.

Take the time to check your code line by line. If there is a difference, review the questions above, make the needed corrections, hard return (Enter) to make the changes to the code screen.

Copy the code into notepad.

Save your code into the hard drive file Desktop—Documents—G-code giving your code a descriptive name.

Your code is ready to use in the OM5 Control Software.
1. Power on to main CNC Tower.
2. Power on to computer.
3. Open Legacy CNC Control Interface by launching the Mach3 Loader found on your desktop, Start menu or toolbar. Choose your machine from the Session Profile screen.
4. Click the flashing red “System Reset” button.
5. Click “Set Machine Coordinates”.
6. Click the “Turning” button and set the “Stock Diameter” display to the diameter of your work piece.
7. Turn on “Smart Tool”.
8. Securely mount your work blank in the machine between centers (A axis).
9. Open Conversational CAM V3.11 from the desktop, Start menu or toolbar.
10. Enable Macros
11. From the Main Menu, choose one of the turning CAM process you wish to use.
12. Answer questions according to your work piece design. Choose “Smart Tool” for the tool touch off method.
13. Generate G-code
14. Click “Copy Code”
15. Open notepad and “paste” code to screen by right clicking the mouse and clicking “paste” OR going to Edit and “paste” on the notepad menu.
16. Name and save this notepad code to the G-code folder in My Documents. Use a name that is descriptive and will help you identify your part. Your program has now been saved to the hard drive and is ready to be accessed by the Legacy CNC Control Interface.
17. From the Legacy CNC Control screen click “load code”. You should find the code in the G-code folder found in My Documents.
18. Load the code by double clicking the file name or choosing “open”.
19. With your code loaded—Click “Cycle Start”
20. The program will begin and prompt a tool change—insert the correct tool.
21. After the tool change, click “Cycle Start” to continue the program through to the end following prompts for tool changes.
CNC Fast Start Reference Sheet
Flat Stock Milling

1. Power on to main CNC Tower.
2. Power on to computer.
3. Open Legacy CNC Control Interface by launching the Mach3 Loader found on your desktop, Start menu or toolbar. Choose your machine from the Session Profile screen.
4. Click the flashing red “System Reset” button.
5. Click “Set Machine Coordinates”.
6. Make sure the Turning button is off, “Current Offset” should read 1.
7. Securely mount your work blank in the machine.
8. Place a center finding tool or pointed tool into the Spindle/Router.
9. Set your “Work Offset Position” for X and Y by manually moving the Spindle over your work piece to the XY origin (page 15)
10. Zero the “Work Offset Display” for X and Y.
11. Turn “Smart Tool” on.
12. Place the Mobile Smart Pad flat on your work piece, plug it into the Stationary Smart pad. Center the spindle above the pad.
13. Click “Smart Tool Setup”. The machine will automatically run the “Smart Tool” sequence. Follow prompts to remove the Mobile Pad. Remove pad. Click OK to finish the routine.
14. Open Conversational CAM V3.11 from the desktop, Start menu or toolbar - Enable Macros.
15. From the Main Menu, choose the type of flat stock milling you wish to design.
16. Answer the questions according to your work piece design choosing “Smart Tool” for the tool touch off method.
17. Generate code.
18. Click “Copy Code” at the top of the screen.
19. Open notepad and “paste” code to notepad screen by right clicking the mouse and clicking “paste” OR going to Edit and “paste” on the notepad menu.
20. Name and save this notepad text file to the G-code folder in My Documents. Use a name that is descriptive and will help you identify your part. Your program has now been saved to the hard drive and is ready to be loaded by the Legacy CNC Control Interface.
21. From the Legacy CNC Control screen click “load code”. You should find the code in the G-code folder found in My Documents.
22. Load the code by double clicking the file name or choosing “open”.
23. With your code loaded—Click “Cycle Start”
24. The program will begin and prompt a tool change—insert the correct tool.
25. After the tool change, click “Cycle Start” to continue the program through to the end following prompts for tool changes.