

# *Standard*

CNC Turning/Milling Machine

## Conversational CAM 3.11 Instruction Manual

Turning Round

Turning Tool Profiles

Multi-Sided

Indexing

Rope & Barley Twist

Spirals

Surface Planing

Linear Cuts

Tool List

Settings

Legacy Conversational CAM

Revision 3.11

Copyright Legacy Woodworking Machinery  
(800) 279-4570

[www.legacywoodworking.com](http://www.legacywoodworking.com)

[technicalsupport@legacywoodworking.com](mailto:technicalsupport@legacywoodworking.com)

Legacy Woodworking Machinery  
435 W. 1000 N.  
Springville, UT 84663

Content

Warranty and Repair information..... 3

Conversational CAM

- Settings and Start-up for Conversational CAM ..... 4
- Turning Round ..... 5
- Turning Tool Profile..... 6
- Multi Sided..... 7
- Indexing..... 8
- Rope & Barley Twist..... 9
- Spiral ..... 10
- Surface Planing ..... 11
- Linear Cut..... 12
- Tool Setup ..... 13
- Saving, Naming & Moving Code..... 14

Conversational CAM Workbook

- Turning Single Step Program - Round..... 15
- Turning Multiple Step Program
  - Tool Profile ..... 16
  - Indexing..... 17
  - G-code overview ..... 18
- Turning Single Step Program - Rope & Barley ..... 19

Glossary..... 20

G-code Reference Sheet..... 21

# *Standard*

CNC Turning/Milling Machine

Product of:

Legacy Woodworking Machinery

435 W. 1000 N.

Springville, UT 84663

## WARRANTY

Legacy Woodworking Machinery warrants to the original purchaser that its products are free from defects in material and workmanship.

This warranty covers electronic control devices for one year from the date of delivery.

This warranty covers the frame and mechanical devices for three years from the date of delivery.

This warranty does not cover any damage to Legacy products that result from improper installation, accident abuse, misuse, natural disaster, insufficient or excessive electrical supply, abnormal mechanical or environmental conditions, or any unauthorized disassembly, repair or modification.

## Repairs

If product repair or replacement is necessary, the Customer is responsible for all shipping charges, freight, insurance and proper packaging to prevent breakage or damage in transit.

## Customer Support

You can contact the Legacy's customer support department at:

[cncsupport@legacywoodworking.com](mailto:cncsupport@legacywoodworking.com)

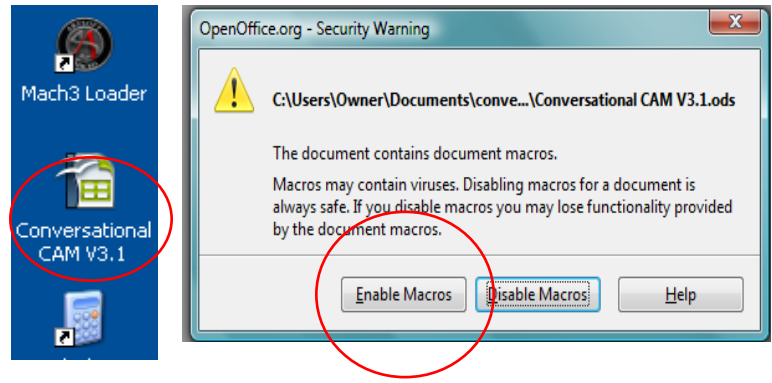
(801) 491-0010

(800) 279-4570

Legacy provides free technical support for the first 90 days to its customers worldwide.

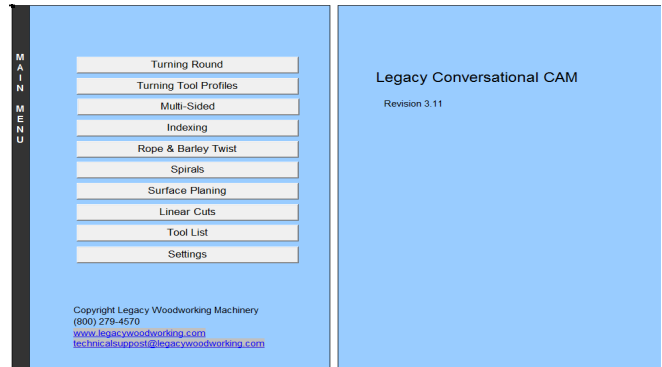
Proudly made in the USA

Conversational CAM is a “Computer Aided Manufacturing” program which uses 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** -



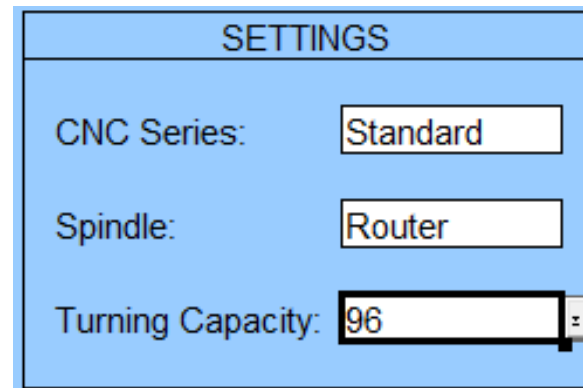
The Main Menu offers a choice of:

- 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: Standard
- Spindle: Router
- Turning Capacity: machine length



**Milling Between Centers Programs**

**Turning Round:** is designed to quickly turn stock round.

**Turning Tool Profiles:** is designed to create a turned element, such as a bead or cove, based on a tool (router bit) profile.

**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 to 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 to multiple start spirals based on pitch.

**Flat Stock Programs**

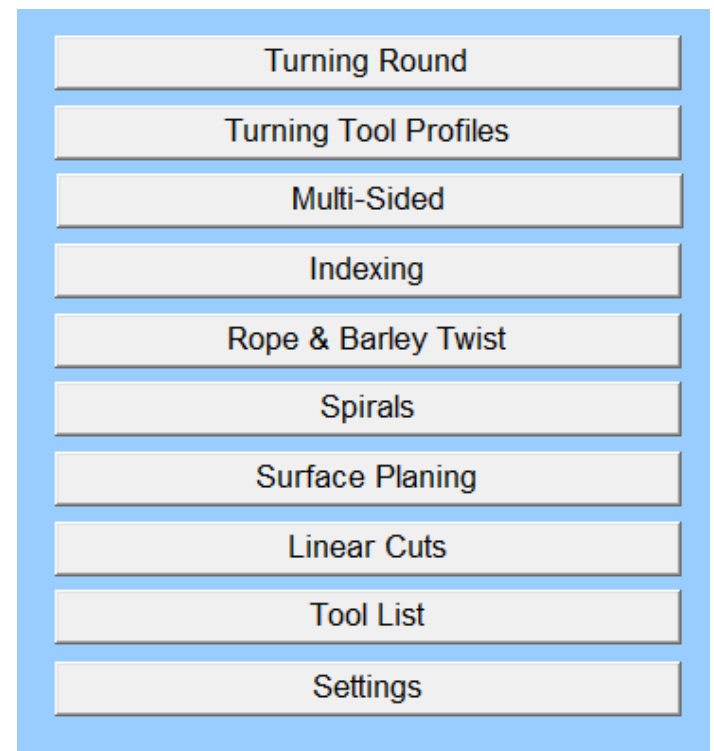
**Surface Planing:** is designed to surface plane flat stock.

**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:** is the for setting up your machine type.



**Turning Round:** You can use this interface when you wish to turn any part of your work piece round. This program references the shoulder of the cutter to the X-axis starting position.

**Is this the first step of the program?** By answering Yes you place a preamble G-code process 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 a number that helps reference the bit assigned to this process.

**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 diameter:** This is the diameter of the router bit you are using.

**Enter the step over per rotation:** This programs how far the X-axis moves for each A-axis rotation and determines the surface smoothness for this cut.

**Enter the type of cut:** Bottom or Side; this is determined by the bit you are using.

**Enter starting Position:** Will program where, on your work piece, you want to begin the round turning.

**Enter ending position:** Will program where, on your work piece, you want to end the round turning.

**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.

Smart Tool Touch off method

*(Optional upgrade for Standard CNC systems)*

**Enter the stock diameter:** Is the finished diameter.

Part Touch Off method

**Enter the plunge depth:** The machine has no reference to Z-0 in this process. . Enter the cut plunge depth in relation to the surface of the part.

Bull Nose touch off

*(Recommended touch off method for turning on a Standard CNC)*

**Enter the stock diameter:** Is the finished diameter.

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

The screenshot shows the 'TURNING ROUND' interface with a vertical title bar on the left. The main area contains a list of questions and input fields. At the top, there are three buttons: 'Main Menu', 'Clear Fields', and 'Copy Code'. The questions are: 'Is this the first step of the program?', 'Is this the last step of the program?', 'Is this the same tool as the previous step?', 'Enter the tool number.', 'Enter the spindle RPM.', 'Enter the tool diameter.', 'Enter the step over per rotation.', 'Enter the type of cut.', 'Enter the starting position.', 'Enter the ending position.', 'Enter the feed rate.', 'Select the tool touch off method.', 'Enter the stock diameter.', and 'Enter the length of the part.' Each question has a corresponding input field. A 'Tool List' button is located next to the 'Enter the tool number.' field. A red square in the upper right corner of the first input field indicates an available help window.

This close-up shows the 'Select the tool touch off method.' dropdown menu with 'Part' selected. Below it are input fields for 'Enter the plunge depth.' and 'This field is not used.'

This close-up shows the 'Select the tool touch off method.' dropdown menu with 'Bull Nose' selected. Below it are input fields for 'Enter the stock diameter.' and 'Enter the length of the part.'

The screenshot shows a table titled 'Recommended Maximum Router Bit RPM' with columns for 'Bit Dia' and 'RPM'. The data is as follows:

Bit Dia	RPM
0.00" - 1.00"	24,000
1.00" - 2.00"	19,000
2.00" - 2.50"	16,000
2.50" - 3.00"	13,000

**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

This is the overview for the **Turning Tool Profiles** interface. This interface is designed to create a turned element, such as a bead or cove, based on a router bit profile. 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 a number that helps reference the bit assigned to this process.

**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 reminder code 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.

**Enter the stock diameter:** Is the finished diameter.

**Enter the plunge depth:** The machine has no reference to Z-0 in this process. . Enter the cut plunge depth in relation to the surface of the part. diameter.

**Enter the length of the part:** Will be available only if Bull Nose is selected for touch off method. Enter the length of the part.

	Main Menu	Clear Fields	Copy Code
TURNING TOOL PROFILES	Is this the first step of the program?		<input type="text" value=""/>
	Is this the last step of the program?		<input type="text" value=""/>
	Is this the same tool as the previous step?		<input type="text" value=""/>
	Enter the tool number.	<input type="button" value="Tool List"/>	<input type="text" value=""/>
	Enter the spindle RPM.		<input type="text" value=""/>
	Enter the plunge depth.		<input type="text" value=""/>
	Enter the X axis position of the cut.		<input type="text" value=""/>
	Select the side to climb mill.		<input type="text" value=""/>
	Enter the feed rate.		<input type="text" value=""/>
	Select the tool touch off method.		<input type="text" value=""/>
	Enter the finished diameter.		<input type="text" value=""/>
	Enter the length of the part.		<input type="text" value=""/>

Select the tool touch off method.	<input type="text" value="Smart Tool"/>
Enter the stock diameter.	<input type="text" value=""/>
This field is not used.	<input type="text" value=""/>

Select the tool touch off method.	<input type="text" value="Part"/>
Enter the plunge depth.	<input type="text" value=""/>
This field is not used.	<input type="text" value=""/>

Select the tool touch off method.	<input type="text" value="Bull Nose"/>
Enter the stock diameter.	<input type="text" value=""/>
Enter the length of the part.	<input type="text" value=""/>

**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 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 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

	Main Menu	Clear Fields	Copy Code
<b>MULTI-SIDED</b>	Is this the first step of the program?		<input type="text" value=""/>
	Is this the last step of the program?		<input type="text" value=""/>
	Is this the same tool as the previous step?		<input type="text" value=""/>
	Enter the tool number.	<input type="button" value="Tool List"/>	<input type="text" value=""/>
	Enter the spindle RPM.		<input type="text" value=""/>
	Enter the tool diameter.		<input type="text" value=""/>
	Enter the number of sides.		<input type="text" value=""/>
	Enter the starting position.		<input type="text" value=""/>
	Enter the ending position.		<input type="text" value=""/>
	Enter the feed rate.		<input type="text" value=""/>
	Select the tool touch off method.		<input type="text" value=""/>
	Enter the finished radius.		<input type="text" value=""/>
	Enter the length of the part.		<input type="text" value=""/>

Select the tool touch off method.	<input type="text" value="Smart Tool"/>
Enter the finished radius.	<input type="text" value=""/>
This field is not used.	<input type="text" value=""/>

Select the tool touch off method.	<input type="text" value="Part"/>
Enter the plunge depth.	<input type="text" value=""/>
This field is not used.	<input type="text" value=""/>

Select the tool touch off method.	<input type="text" value="Bull Nose"/>
Enter the finished radius.	<input type="text" value=""/>
Enter the length of the part.	<input type="text" value=""/>

This is the overview for the **Indexing** interface. This interface is designed to add indexing components or 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 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 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 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 the 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:** Where do you want the cut to being 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 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 to find the diameter center point of a work piece.

**Enter finished diameter:** Enter the finished diameter. This program is assuming the part is already turned round to the finished diameter - the indexing depth will be calculated from this diameter.

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

	Main Menu	Clear Fields	Copy Code
INDEXING	Is this the first step of the program?		<input type="text" value=""/>
	Is this the last step of the program?		<input type="text" value=""/>
	Is this the same tool as the previous step?		<input type="text" value=""/>
	Enter the tool number.	<input type="button" value="Tool List"/>	<input type="text" value=""/>
	Enter the spindle RPM.		<input type="text" value=""/>
	Enter the tool cutting depth.		<input type="text" value=""/>
	Enter the number of starts.		<input type="text" value=""/>
	Enter the starting position.		<input type="text" value=""/>
	Enter the ending position.		<input type="text" value=""/>
	Enter the feed rate.		<input type="text" value=""/>
	Select the tool touch off method.		<input type="text" value=""/>
	Enter the finished diameter.		<input type="text" value=""/>
	Enter the length of the part.		<input type="text" value=""/>
	Select the tool touch off method.		
Enter the finished diameter.			<input type="text" value=""/>
This field is not used.			<input type="text" value=""/>
Select the tool touch off method.			<input type="text" value="Part"/>
This field is not used.			<input type="text" value=""/>
This field is not used.			<input type="text" value=""/>
Select the tool touch off method.			<input type="text" value="Bull Nose"/>
Enter the finished diameter.			<input type="text" value=""/>
Enter the length of the part.			<input type="text" value=""/>



This is the overview for the **Rope and Barley** interface. You will use this interface 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, 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 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. 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 a reminder reference that will appear in the code to turn on your router and set the RPMs.

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

**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:** 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 being 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.

**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 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 to find the diameter center point of a work piece.

**Enter finished diameter:** Enter the finished diameter. This program is assuming the part is already turned round to the finished diameter - the indexing depth will be calculated from this diameter.

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

	Main Menu	Clear Fields	Copy Code
R O P E & B A R L E Y  T W I S T	Is this the first step in the program?		<input type="text" value=""/>
	Is this the last step in the program?		<input type="text" value=""/>
	Is this the same tool as the previous step?		<input type="text" value=""/>
	Enter the tool number.	<input type="button" value="Tool List"/>	<input type="text" value=""/>
	Enter the spindle RPM.		<input type="text" value=""/>
	Enter the tool diameter.		<input type="text" value=""/>
	Enter the tool cutting depth.		<input type="text" value=""/>
	Enter the number of starts.		<input type="text" value=""/>
	Enter the starting position.		<input type="text" value=""/>
	Enter the ending position.		<input type="text" value=""/>
	Select the direction of the spiral.		<input type="text" value=""/>
	Enter the feed rate.		<input type="text" value=""/>
	Select the tool touch off method.		<input type="text" value=""/>
	Enter the finished diameter.		<input type="text" value=""/>
	Enter the length of the part.		<input type="text" value=""/>
Select the tool touch off method.			<input type="text" value="Part"/>
This field is not used.			<input type="text" value=""/>
This field is not used.			<input type="text" value=""/>
Select the tool touch off method.			<input type="text" value="Bull Nose"/>
Enter the finished diameter.			<input type="text" value=""/>
Enter the length of the part.			<input type="text" value=""/>

**Note:** This program has been designed to cut the spiral in two passes. The first pass will move down the work piece at 70% of the cutting depth. The return pass will 100% of the cutting depth.

When milling spirals, it is beneficial to start and end on a cove.

To achieve the pineapple cut, just repeat this process twice, except change the spiral direction making one Left and one Right.

**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 for 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. 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 a reminder reference that will appear in the code 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 number of starts:** The number of starts for the spiral design.

**Enter the starting position:** Where do you want the spiral cut to being 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.

**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 to find the diameter center point of a work piece.

**Enter finished diameter:** Enter the finished diameter. This program is assuming the part is already turned round to the finished diameter - the indexing depth will be calculated from this diameter.

**Enter the length of the part:** Enter the length of the part

	Main Menu	Clear Fields	Copy Code
<b>S P I R A L S</b>	Is this the first step in the program?		<input type="text" value=""/>
	Is this the last step in the program?		<input type="text" value=""/>
	Is this the same tool as the previous step?		<input type="text" value=""/>
	Enter the tool number.	<input type="text" value=""/>	<input type="text" value=""/>
	Enter the spindle RPM.		<input type="text" value=""/>
	Enter the spiral pitch.		<input type="text" value=""/>
	Enter the tool cutting depth.		<input type="text" value=""/>
	Enter the number of starts.		<input type="text" value=""/>
	Enter the starting position.		<input type="text" value=""/>
	Enter the ending position.		<input type="text" value=""/>
	Select the direction of the spiral.		<input type="text" value=""/>
	Enter the feed rate.		<input type="text" value=""/>
	Select the tool touch off method.		<input type="text" value=""/>
	Enter the finished diameter.		<input type="text" value=""/>
	Enter the length of the part.		<input type="text" value=""/>

Select the tool touch off method.

Enter the finished diameter.

*This field is not used.*

Select the tool touch off method.

*This field is not used.*

*This field is not used.*

Select the tool touch off method.

Enter the finished diameter.

Enter the length of the part.

**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. This pro-

**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. 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: Part is recommended as the tool touch off method for Standard CNC machines.

	Main Menu	Clear Fields	Copy Code
SURFACE PLANING	Is this the first step in the program?		<input type="text" value=""/>
	Is this the last step in the program?		<input type="text" value=""/>
	Is this the same tool as the previous step?		<input type="text" value=""/>
	Enter the tool number.		<input type="text" value=""/> Tool List
	Enter the spindle RPM.		<input type="text" value=""/>
	Enter the tool diameter.		<input type="text" value=""/>
	Enter the depth of cut.		<input type="text" value=""/>
	Enter the width.		<input type="text" value=""/>
	Enter the X-axis starting position.		<input type="text" value=""/>
	Enter the X-axis ending position.		<input type="text" value=""/>
	Enter the feed rate.		<input type="text" value=""/>
	Select the tool touch off method.		<input type="text" value=""/>

This is the overview for the **Linear Cuts** interface. This interface is designed for making linear tool paths along your work piece 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 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. 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 a reminder reference that will appear in the code 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 to find the diameter center point of a work piece.

	Main Menu	Clear Fields	Copy Code
<b>L I N E A R  C U T S</b>	Is this the first step in the program?		<input type="text" value=""/>
	Is this the last step in the program?		<input type="text" value=""/>
	Is this the same tool as the previous step?		<input type="text" value=""/>
	Enter the tool number.	<input type="button" value="Tool List"/>	<input type="text" value=""/>
	Enter the spindle RPM.		<input type="text" value=""/>
	Enter the tool cutting depth.		<input type="text" value=""/>
	Enter the number of parallel cuts.		<input type="text" value=""/>
	Enter the Y-axis spacing.		<input type="text" value=""/>
	Enter the Y-axis starting position.		<input type="text" value=""/>
	Enter the X-axis starting position.		<input type="text" value=""/>
	Enter the X-axis ending position.		<input type="text" value=""/>
	Enter the feed rate.		<input type="text" value=""/>
	Select the tool touch off method.		<input type="text" value=""/>

This is the overview for the **Tool Setup** reference screen. This reference screen allows you to make a personalized bit library for use while writing Conversational

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

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

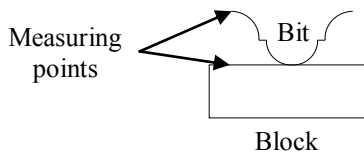
**Router Bit Number:** We have used the part number from the company we order our router bit from for easy reordering. You may reference any information you would like.

**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 should with the calipers.



This is not a perfect measurement but it is 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 Bull Nose or Part.

**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.

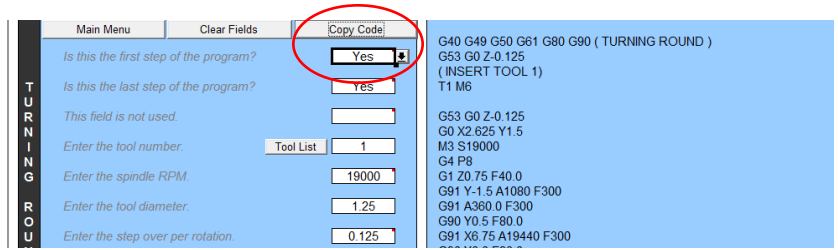
**Copy Code:** will copy the code to the windows clipboard.

	Tool Number	Router Bit Number	Cutting Diameter	Profile Height	Height Offset	Description	
T O O L  L I S T	0	0	0	0		Reference Tool	
	1	2704	1.25	0.5		Surface Planing	
	2	3952	2	0.435		Classic Spiral	
	3	7581	1	0.15		Rope Moulding	
	4						
	5						
	6						
	7						
	8						
	9						
	10						
	11						
	12						
	13						
	14						
	15						
	16						
	17						
	18						
	19						
	20						
	21						
	22						
	23						
	24						
	25						
	26						
	27						
	28						
	29						
	30						
	31						
32							

Main Menu	Clear Fields	Copy Code
Is this the first step of the program?		<input type="checkbox"/>

Naming, saving and moving G-code from Conversation CAM to the OM5 Control Software.

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



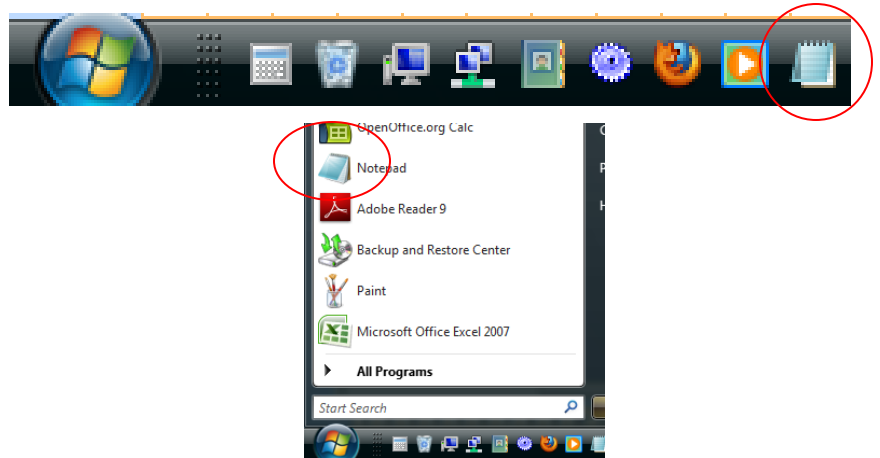
Open “Notepad”

Found on the quick launch icon toolbar at the bottom left of your computer screen

Or

On the Windows Start menu.

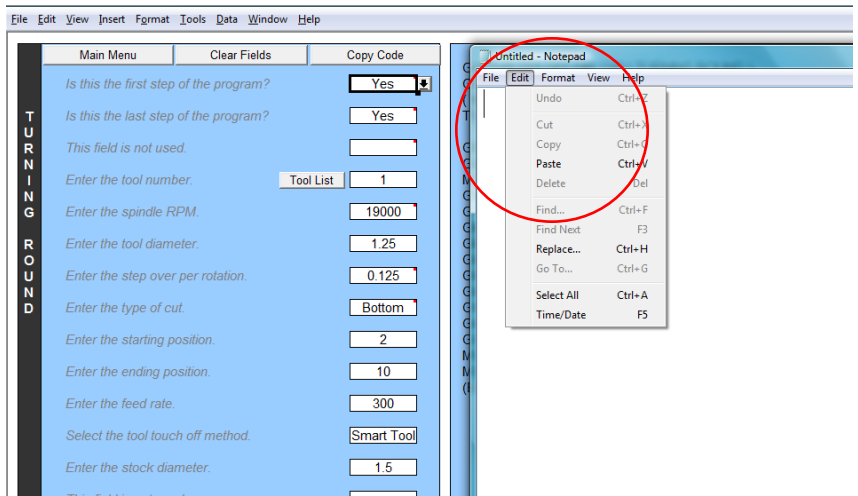
A Notepad shortcut icon has also be placed on your desktop.



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.

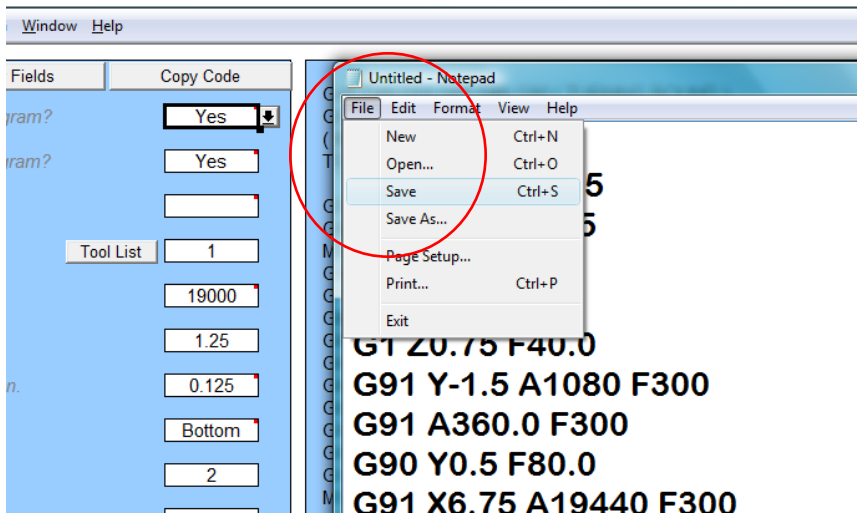


Once the G-code is on the Notepad screen, go to:  
File then Save.

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 Single Step Program

Fill out the form using these parameters:

Blank: 10" x 1.5"  
1.25" Bottom Cleaning Bit  
Single Process program

This is the first step of the program—Yes  
This is the last step of the program—Yes  
N/A  
The tool number is 1  
The spindle RPM is 19000  
The tool diameter is 1.25  
The step over per rotation is 0.25  
The type of cut is bottom  
The starting position is 1.0  
The ending position is 10  
The feed rate is 200  
The touch off method is Bull Nose  
The stock diameter is 1.5  
The stock length is 10"

Hitting the “enter” button will generate the code onto the right side of the screen.

	Main Menu	Clear Fields	Copy Code
TURNING ROUND	<i>Is this the first step of the program?</i>		<input type="text" value="Yes"/>
	<i>Is this the last step of the program?</i>		<input type="text" value="Yes"/>
	<i>This field is not used.</i>		<input type="text"/>
	<i>Enter the tool number.</i>	Tool List	<input type="text" value="1"/>
	<i>Enter the spindle RPM.</i>		<input type="text" value="19000"/>
	<i>Enter the tool diameter.</i>		<input type="text" value="1.25"/>
	<i>Enter the step over per rotation.</i>		<input type="text" value="0.25"/>
	<i>Enter the type of cut.</i>		<input type="text" value="Bottom"/>
	<i>Enter the starting position.</i>		<input type="text" value="1"/>
	<i>Enter the ending position.</i>		<input type="text" value="10"/>
	<i>Enter the feed rate.</i>		<input type="text" value="200"/>
	<i>Select the tool touch off method.</i>		<input type="text" value="Bull Nose"/>
	<i>Enter the stock diameter.</i>		<input type="text" value="1.5"/>
	<i>Enter the length of the part.</i>		<input type="text" value="10"/>

Copy and paste the code into Notepad. Save the file and then open the file in Mach3.

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

Take the time to check your code line by line. If there are any 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 Mach3.

```
G40 G49 G50 G61 G80 G90 ( TURNING ROUND )
G53 G0 Z-0.125
G0 X11.5 Y-0.5
M0 ( INSERT AND TOUCH OFF TOOL 1 )
G92 Z0.7
G53 G0 Z-0.125
G0 X1.625 Y1.5
M0 ( TURN ON THE ROUTER AT 19000 RPM )
G4 P1
G1 Z0.75 F40.0
G91 Y-1.5 A1080 F200
G91 A360.0 F200
G90 Y0.5 F80.0
G91 X7.75 A11160 F200
G90 Y0.0 F80.0
G91 A360.0 F200
G90
G53 G0 Z-0.125
M0 ( TURN OFF THE ROUTER )
M30
(End of the program)
```

# Conversation CAM Workbook

## Turning Tool Profile

### Multiple Step Program—Step 1

Fill out the form using these parameters:  
 Blank - 1.5" x 10" (you can use the round blank from the turning Round work sheet)  
 2" Classic Spiral Bit  
 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 Part
- N/A
- N/A

Hitting the “enter” 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 are any 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.

	Main Menu	Clear Fields	Copy Code
TURNING TOOL PROFILES	Is this the first step of the program?		<input type="button" value="Yes"/>
	Is this the last step of the program?		<input type="button" value="No"/>
	This field is not used.		<input type="text"/>
	Enter the tool number.	<input type="button" value="Tool List"/>	<input type="text" value="2"/>
	Enter the spindle RPM.		<input type="text" value="19000"/>
	Enter the plunge depth.		<input type="text" value="0.435"/>
	Enter the X axis position of the cut.		<input type="text" value="5"/>
	Select the side to climb mill.		<input type="button" value="Left"/>
	Enter the feed rate.		<input type="text" value="80"/>
	Select the tool touch off method.		<input type="button" value="Part"/>
	This field is not used.		<input type="text"/>
	This field is not used.		<input type="text"/>

Copy and paste the code into Notepad. Save the file and then open the file in Mach3.

```
G40 G49 G50 G61 G80 G90 ( TURNING TOOL PROFILE )
G53 G0 Z-0.125
G0 X5 Y0.0
M0 ( INSERT AND TOUCH OFF TOOL 2 )
G92 Z0.0
G0 Z1.0
G0 X5 Y0.0
M0 ( TURN ON THE ROUTER AT 19000 RPM )
G4 P1
G1 Z0.125 F40
G91 Z-0.56 A-1800 F80
G91 A-360 F80
G90
G53 G0 Z-0.125
M0 ( TURN OFF THE ROUTER )
```



# Conversation CAM Workbook

## Indexing

### Multiple Step Program—Step 2

Fill out the form using these parameters:  
 Blank - 1.5" x 10" - You will be using the part from  
 the "Turning Tool Profile" work sheet.  
 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 Part  
 N/A  
 N/A

Hitting the "enter" button will generate the code  
 onto the right side of the screen.

	Main Menu	Clear Fields	Copy Code
INDEXING	<i>Is this the first step of the program?</i>		<input type="text" value="No"/>
	<i>Is this the last step of the program?</i>		<input type="text" value="Yes"/>
	<i>Is this the same tool as the previous step?</i>		<input type="text" value="Yes"/>
	<i>This field is not used.</i>		<input type="text" value="Tool List"/>
	<i>Enter the spindle RPM.</i>		<input type="text" value="19000"/>
	<i>Enter the tool cutting depth.</i>		<input type="text" value="0.07"/>
	<i>Enter the number of starts.</i>		<input type="text" value="6"/>
	<i>Enter the starting position.</i>		<input type="text" value="5"/>
	<i>Enter the ending position.</i>		<input type="text" value="1.5"/>
	<i>Enter the feed rate.</i>		<input type="text" value="200"/>
	<i>Select the tool touch off method.</i>		<input type="text" value="Part"/>
	<i>This field is not used.</i>		<input type="text" value=""/>
	<i>This field is not used.</i>		<input type="text" value=""/>

Copy and paste the code into Notepad. Save the file and then open the file in Mach3.

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

Take the time to check your code line by line. If  
 there are any 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 Mach3.

```
( INDEXING )
G0 Z1.0
G0 X5 Y0.0
M0 ( TURN ON THE ROUTER AT 19000 RPM )
G4 P1
G1 Z1.0 F60
M98 P100 Q6
G90
G53 G0 Z-0.125
M0 ( TURN OFF THE ROUTER )
M30
(End of the program)

O0100
G90
G1 Z-0.0525 F20.0
G1 X1.5 F200
G1 Z-0.07 F20.0
G1 X5 F200
G1 Z1.0 F60.0
G91 A60 F200
G90
M99
```

# Conversation CAM Workbook

## Multiple Step Program

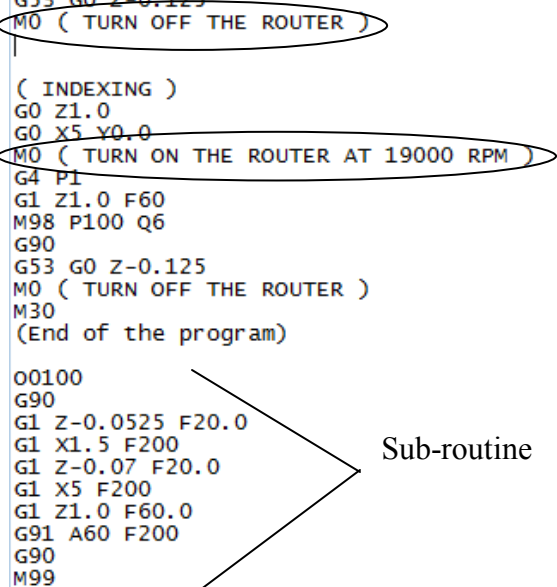
### Paste-Name-Save

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 second. If your code differs from what is shown, check your question screens and code placement.

**M0** at the end of the “Turning Tool Profile” code section reminds you to turn off the router, which is not necessary because we are not making a tool change.

```
G40 G49 G50 G61 G80 G90 ( TURNING TOOL PROFILE )
G53 G0 Z-0.125
G0 X11.5 Y0.0
M0 ( INSERT AND TOUCH OFF TOOL 2 )
G92 Z0.7
G53 G0 Z-0.125
G0 X5 Y0.0
M0 ( TURN ON THE ROUTER AT 19000 RPM )
G4 P1
G1 Z0.875 F40
G91 Z-0.56 A-1800 F80
G91 A-360 F80
G90
G53 G0 Z-0.125
M0 ( TURN OFF THE ROUTER )
( INDEXING )
G0 Z1.0
G0 X5 Y0.0
M0 ( TURN ON THE ROUTER AT 19000 RPM )
G4 P1
G1 Z1.0 F60
M98 P100 Q6
G90
G53 G0 Z-0.125
M0 ( TURN OFF THE ROUTER )
M30
(End of the program)

00100
G90
G1 Z-0.0525 F20.0
G1 X1.5 F200
G1 Z-0.07 F20.0
G1 X5 F200
G1 Z1.0 F60.0
G91 A60 F200
G90
M99
```



Note that sub-routines are listed after M30 or the “End of the program” line. This sub-routine represents the indexed flutes, which will run 6 times in this program.

# Conversation CAM Workbook

## Rope & Barley Twist

### Single Step Program

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 Part

N/A

N/A

Hitting the "enter" button will generate the code onto the right side of the screen.

	Main Menu	Clear Fields	Copy Code
R O P E & B A R L E Y  T W I S T	Is this the first step in the program?		<input type="button" value="Yes"/>
	Is this the last step in the program?		<input type="button" value="Yes"/>
	This field is not used.		<input type="text"/>
	Enter the tool number.	<input type="button" value="Tool List"/>	<input type="text" value="3"/>
	Enter the spindle RPM.		<input type="text" value="21000"/>
	Enter the tool diameter.		<input type="text" value="1"/>
	Enter the tool cutting depth.		<input type="text" value="0.15"/>
	Enter the number of starts.		<input type="text" value="3"/>
	Enter the starting position.		<input type="text" value="5"/>
	Enter the ending position.		<input type="text" value="10"/>
	Select the direction of the spiral.		<input type="text" value="Left"/>
	Enter the feed rate.		<input type="text" value="200"/>
	Select the tool touch off method.		<input type="text" value="Part"/>
	This field is not used.		<input type="text"/>
This field is not used.		<input type="text"/>	
Copy and paste the code into Notepad. Save the file and then open the file in Mach3.			

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

Take the time to check your code line by line. If there are any 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 Mach3.

```
G40 G49 G50 G61 G80 G90 ( ROPE OR BARLEY TWIST )
G53 G0 Z-0.125
G0 X5 Y0.0
M0 ( INSERT AND TOUCH OFF TOOL 3 )
G92 Z0.0
G0 Z1.0
G0 X5 Y0.0
M0 ( TURN ON THE ROUTER AT 21000 RPM )
G4 P1
G1 Z1.0 F40
M98 P0200 Q3
G90
G53 G0 Z-0.125
M5
M30
(End of program)

O0200
G90
G1 Z-0.1125 F20.0
G91 X5 A600 F200
G90 Z-0.15 F20
G91 X-5 A-600 F200
G90 Z1.0 F60
G91 A120 F200
G90
M99
```

**Bull Nose:** The Bull Nose center is a 1.4”diam. x 1.25” Delrin center designed to be placed over the point of the tailstock shaft. The Legacy Bull Nose is a pointed extension to the tailstock shaft that has been manufactured to an exact thickness to guarantee accurate Z-axis “0” settings when willing between centers.

and

Bull Nose is a tool touch off method used with the Standard CNC OM5 Control Software specifically designed to accurately set the Z-axis home position for A-axis turning. When choosing “Bull Nose” as a touch off method, a Bull Nose is placed on the tailstock shaft and the cutter is brought down to the Bull Nose surface to establish Z “0” for the work offset.

**Conversational CAM:** 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.

**Delrin:** A self lubricating high grade plastic.

**Feed rate:** Is the speed the cutter moves past the part, measured in IPM “inches per minute”. When using the A-axis, the feed rate is the speed the part moves past the cutter. Guidelines: larger diameter of cutter = slower feed rate. Deeper the cut = slower feed rate. If the bit chatters, slow down the feed rate speed up the RPMs. Recommended feed rates for the Standard CNC machines are 40 - 120 IPM.

**G-code:** are functions in the Numerical control programming language. The G-codes are the codes that position the tool and do the actual work, as opposed to M-codes, that manage the machine; T for tool-related codes, and S and F are tool-Speed and tool-Feed. The programming language of Numerical Control (NC) is sometimes informally called G-code. But in actuality, G-codes are only a part of the NC-programming language that controls NC and CNC machine tools.

**OM5 Control Software:** The OM5 software has been custom designed by Legacy as an interface to Mach3 and is the control software used to run all Legacy CNC milling machines. The OM5 software was designed as a user friendly screen interface to help utilize the unique milling attributes of the Legacy CNC mills.

**Part:** Refers to the work stock placed in the Legacy Mill.

and

Is a tool touch off method used in Conversational CAM where the cutter is brought down to the surface of the part to establish the cutter length in reference to the machine Z axis “0” position.

**RPM—Rotations Per Minute:** Recommended Maximum Router Bit RPM

Bit Diameter	RPM
0.00” - 1.00”	21,000
1.00” - 2.00”	19,000
2.00” - 2.50”	16,000
2.50” - 3.0”	13,000

**Smart Tool:** Is a tool touch off method designed by Legacy to set the Z-axis height offset in relation to the part surface and router bit length. This eliminates the hassles of setting up tool tables with height offsets. To use Smart Tool, you must have the Smart Tool programming and Stationary Pad installed on your machine as well as the Mobile Pad. When choosing Smart Tool in a Conversational CAM program, code will be generated to run the Smart Tool touch off procedure. Smart Tool must be turned on in the OM5 software program to run Smart Tool successfully.

**Stepper Motor:** Is the type of motor used to drive the axes on Legacy Mills. These are Open Loop impulse motors which measure movement through electrical impulses traveling through a magnetic field. These are a low maintenance, easy to use motor with no internal gear system. If you run into a physical barrier these motors will not be damaged, but you will hear a ratcheting sound. You will need to reference your machine if this happens.

**Surface Planing:** To smooth or level the uneven surface of a work piece by using a flat bottom bit to accurately remove material.

**Tool Offset:** Is a tool touch off method used in the Conversational CAD program. Using a tool library based on the length of each tool in reference to a standard “reference tool” length. This will allow the operator to assign a number and saved tool data on individual tools.

**Tool Touch Off Methods:** These are the methods available for measuring the router bit length in relationship to the Z “0”. The four tool touch off methods used are Smart Tool, Tool Offset, Part and Bull nose.

G-codes are the codes that position the tool and do the actual work.

M-codes manage the machine

G0: Rapid positioning  
 G1: Linear interpolation  
 G2: Clockwise circular/helical interpolation  
 G3: Counterclockwise circular/helical interpolation  
 G4: Dwell  
 G10: Coordinate system origin setting  
 G12: Clockwise circular pocket  
 G13: Counterclockwise circular pocket  
 G15/G16: Polar Coordinate moves in G0 and G1  
 G17: XY plane select  
 G18: XZ plane select  
 G20/G21: Inch/Millimetre unit  
 G28: Return home  
 G30: Return home  
 G31: Straight probe  
 G40: Cancel cutter radius compensation  
 G41/G42: Start cutter radius compensation left/right  
 G43: Apply tool length offset (plus)  
 G49: Cancel tool length offset  
 G50: Reset all scale factors to 1.0  
 G51: Set axis data input scale factors  
 G52: Temporary coordinate system offsets  
 G53: Move in absolute machine coordinate  
 G54: Use fixture offset 1  
 G55: Use fixture offset 2  
 G56: Use fixture offset 3  
 G57: Use fixture offset 4  
 G58: Use fixture offset 5  
 G59: Use fixture offset 6  
 G61/G64: Exact stop/Constant Velocity mode  
 G68/G69: Rotate program coordinate  
 G70/G71: Inch/Millimetre unit  
 G80: Cancel motion mode  
 G90: Absolute distance mode  
 G91: Incremental distance mode  
 G92: Offset coordinates and set parameters  
 G93: Inverse time feed mode  
 G94: Unite per minute  
 G98: Rapid Height by Z height  
 G99: Rapid Height by R height

M0: Program stop  
 M1: Optional program stop  
 M2: Program end  
 M3/M4: Rotate spindle clockwise/counterclockwise  
 M5: Stop spindle rotation  
 M6: Tool change  
 M30: Program end and rewind  
 M47: Repeat program from first line  
 M98: Call subroutine  
 M99: Return from subroutine/repeat

S codes are related to the tool Speed

F codes are related to the tool Feed

T codes are tool related and represent the tool number.

H codes are tool height related and represent the height offset listed in the tool library for the tool listed. Example: H6 = height offset for Tool 6