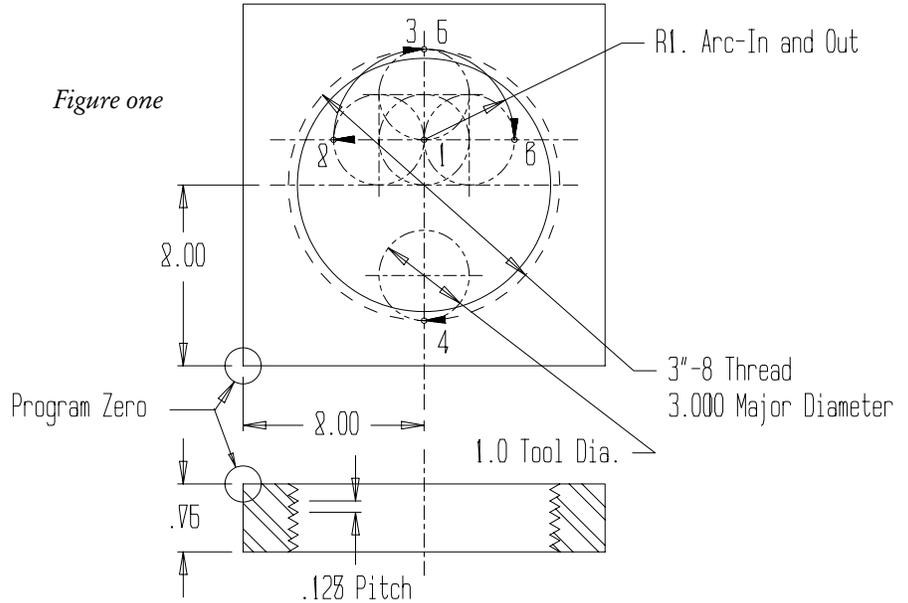


A custom macro for thread milling (internal thread)

This internal thread milling example utilizes a thread milling cutter that can machine the entire thickness of the workpiece in one pass around the hole. It drives the milling cutter in a clockwise (conventional milling) direction from top to bottom to machine the thread. To keep it simple, the approach and retract arc are 1/4 of a circle (ninety degree arcs). Figure one shows an example workpiece along with the milling cutter's tool path.

While it is not the intention of article text to teach you how to thread mill (you can find discussions of thread milling in past issues of The Optional Stop), in order to understand this application you must know that most CNC controls utilize a simple G02 or G03 for helical motion. The control can easily determine the difference between helical motion and circular motion. If a Z axis departure is included in the G02 or G03 command, the control assumes helical interpolation is being performed. The actual value of the Z departure is directly related to the portion of a full circle being machined in XY. For a quarter (90 degree) circle, as is needed during the approach and retract, the Z departure must be one quarter of the pitch. For half a circle, the Z axis must depart half the pitch.

Since this is a user-created canned cycle application, we will use the G65 command in the main program to pass letter address arguments to the thread milling custom macro. Figure two shows the arguments to be used in the calling command.



Here is a portion of the program that includes a call statement to the thread milling custom macro.

```

O0006 (Program number)
.
.
(Machine hole to get ready for thread
milling)
.
N150 T04 M06 (Place thread milling
cutter in spindle)
N155 G90 G54 S400 M03 T05 (Select
absolute mode, coordinate system,
start spindle, get next tool ready)
N160 G00 X2.0 Y2.0 (Rapid over to
center of hole)
N165 G43 H04 Z.1 (Rapid up to
workpiece in Z)
N170 G65 P1006 X2.0 Y2.0 Z0.75
Q0.125 T1.0 M3.0 A1.0 F5.0 (Call cus-
tom macro and mill thread)
N175 G91 G28 Z0 M19 (Return to tool
change position)
N180 M01 (Optional stop)
.
.

```

Argument assignments used in the custom macro are as follows. X:#24, Y:#25, Z:#26, Q: #17, T:#20, M:#13, A:#1, F:#9.

In This Issue:

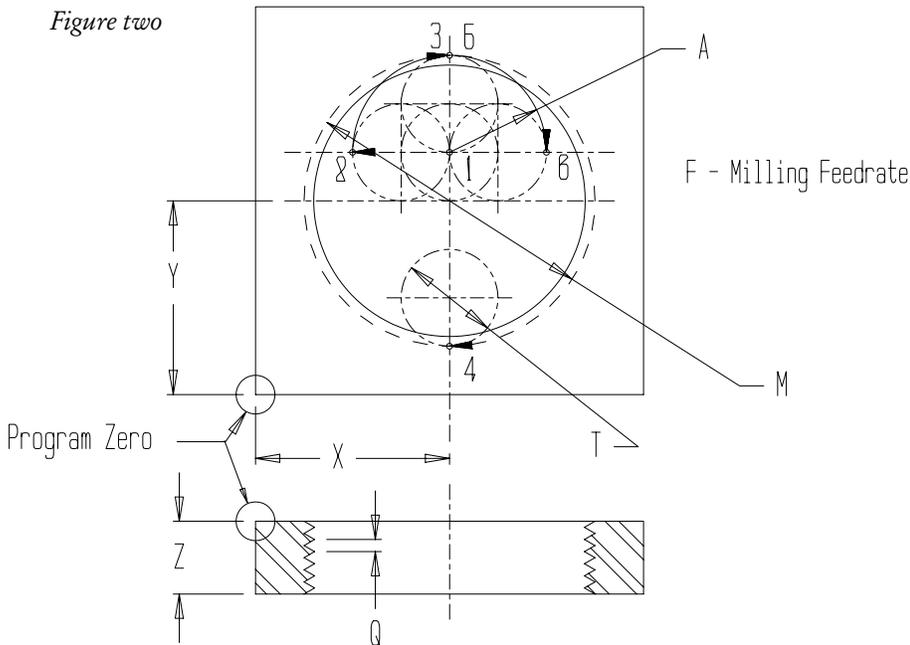
<i>A custom macro for thread milling internal threads</i>	1
<i>Programming format for turning centers</i>	2
<i>Using G52 to temporarily shift program zero</i>	3
<i>Product corner</i>	4

Subscription Information:

The Optional Stop is published quarterly by CNC Concepts, Inc. and is distributed free of charge to people downloading it from our website (www.cncconcepts.com). Back issues are available, but a \$10.00 charge per issue will apply to any back issues not currently published on our site. Back issues 1-50 are available for a total of \$199.00.

All techniques given in this newsletter are intended to make CNC usage more efficient, easier, and/or safer. However, CNC Concepts, Inc. can accept no responsibility for the use or misuse of the techniques given.

Figure two



Custom macro to mill thread:

```
O1006 (Program number)
#20 = #20/2 (Set #20 to tool radius instead of diameter)
#101 = -[#26 + 0.1] (Initial Z position)
G00 X#24 Y[#25 + #13/2 - #1] (Rapid to center of approach radius, point 1)
Z#101 (Rapid to beginning Z position)
#101 = #101 - #17/4 (Calculate Z position at end of approach radius)
G01 X[#24 - #1 + #20] F[#9 * 5] (Fast feed to beginning of approach radius, point 2)
```

```
G02 X#24 Y[#25 + #13/2 - #20] Z#101 R[#1 - #20] F#9 (Circular approach to beginning of thread, point 3)
#101 = #101 - #17/2 (Calculate Z position half way around thread)
G02 Y[#25 - #13/2 + #20] Z#101 R[#13/2 - #20] (Mill half way around thread, point 4)
#101 = #101 - #17/2 (Calculate Z position at end of thread)
G02 Y[#25 + #13/2 - #20] Z#101 R[#13/2 - #20] (Mill balance of thread, point 5)
```

```
#101 = #101 - #17/4 (Calculate Z position at end of retract radius)
G02 X[#24 + #1 - #20] Y[#25 + #13/2 - #1] Z#101 R[#1 - #20] (Retract to point 6)
G00 X#24 (Return to center of approach radius)
Z0.1 (Rapid to clearance position above workpiece)
M99 (end of custom macro)
```

Again, this custom macro causes the thread milling cutter to make one pass around the thread (machining the entire thread in one pass) while milling in a conventional milling manner in a downward Z direction.

M01

Programming format turning centers

Fanuc (in their ultimate wisdom) maintains two different programming styles among their control models. These two different programming methods are most related to multiple repetitive cycles and subprogramming (though there may be other times when variations occur).

This problem is caused by the fact that Fanuc does not place a full-character keyboard on all of their control models. With some controls (like many OT controls), certain important letter addresses are missing. For example, letter addresses D and L are not available on some control models.

For most controls, Fanuc allows a multiple repetitive cycle to be commanded in one block. Here is a G71 command as it is specified on 6T, 10T, and 15T controls:

```
N020 G71 P025 Q055 U0.040 W0.005
D1000 F0.010
```

Letter address D is used to specify the depth-of-cut per roughing pass. Most controls do not allow a decimal point with the D word. But again, most OT controls do not have a D word. So Fanuc breaks up the G71 command into two blocks. A U word in the first G71 command specifies the depth of cut. Here is an example.

```
N015 G71 U0.1
```



Two new computer based training courses!

❖ Setup Reduction For CNC
❖ Cycle Time Reduction For CNC

Affordable courseware helps you improve productivity!

Setup & cycle time defined

During any CNC machine's usage, there are really only two activities. Either machines are in setup or they are running production. Setup time is the total time the machine is down between production runs. Cycle time is the time it takes to complete a production run divided by the number of good workpieces produced.

CDR-STR ~ Setup reduction CD-rom course..... \$239.00
CDR-CTR ~ Cycle time reduction CD-rom course..... \$239.00

CNC CONCEPTS, INC.

N020 G71 P025 Q055 U0.040 W0.005
F0.010

With this format, at least you can specify the depth of cut with a decimal point.

Advantages of the two-block multi-repetitive cycle format

There are a few advantages of the two-block format. Generally speaking, you can specify more information in the program than you can with the one-block format. With G71, for example, you can specify the retract amount with an R word in the first G71 block (how far the tool will back away after each pass). For the one-block format, this value is not programmable and must be specified with a parameter setting.

Similar advantages occur with the G76 threading cycle command. With the two-block format, you are allowed to specify thread chamfering amount, number of spring passes, minimum depth per pass, and final pass depth right in the program. With the one-block format, these values are all set by parameters and cannot be programmed (unless you use the G10 command to change parameters).

Sub-programming differences

Controls that allow one-block entry for canned cycles use the letter ad-

dress L to specify the number of executions for sub-programs. For example, the command

N050 M98 P1000 L5

specifies that program O1000 will be executed five times. Machines that require the two-block canned cycles don't allow the letter address L. Instead, they require that the number of executions be specified as part of the P word. The command

N050 M98 P0051000

calls program O1000 and executes it five times. Note that the first three digits of the P word specify the number of executions. To call program O1000 and execute it once, the command

N050 M98 P1000

is used for all Fanuc control models.

Specifying format style

As you may have experienced, programs written using one of these format styles cannot be run in a control that requires the other (without modification). While modifications are minor, there may be a way to eliminate the need to change programs from one machine to another. Go to the setting page on your controls. Look for a parameter called FORMAT (not all control models have it). If the control has this parameter, you'll be able to select the one-block or two-block entry

method for canned cycles. You may be able to make all of your turning centers compatible in this regard.

M01

Using G52 to temporarily shift program zero on machining centers

Program zero is, of course, the location from which all positions in your program are specified when you're working in the absolute mode (G90). There are times when you must machine identical workpiece attributes in several locations (consider, for example, five identical pockets). If you work from one program zero point, these identical workpiece attributes will require substantially different CNC commands in the absolute mode.

For this reason, many programmers use the incremental mode to specify the motions for identical workpiece attributes, storing them in a sub-program and executing the sub-program once for each attribute. While this works, programming in the incremental mode is cumbersome, time-consuming, and error-prone. If your machine allows G52, there's a much better way.

G52 allows you to specify a temporary shift of program zero. An X, Y, and/or Z word in the G52 command will tell the control how far to shift in each axis. Note that these values are specified from the original program zero point, so if you have consecutive G52s, each must be specified from the original program zero point. And when you're finished, you must remember to cancel the G52 by including X0, Y0, and/or Z0 in the G52 command. Here's a quick example that shows the technique.

Main program:

O0001 (Program number)

N005 T01 M06 (Place tool in spindle)

N010 G54 G90 S500 M03 (Select coordinate system, abs mode and start spindle)



New computer based training course! CD-ROM disk: over 6 hours!

CNC Machining Center Programming, Setup, and Operation

Affordable courseware for CNC machining centers!

A very popular CNC machine type!

CNC machining centers are among the most popular types of CNC machine tools. Most companies that have any CNC machines have at least one. Unfortunately, companies are finding it more and more difficult to find and hire qualified CNC people. Many are realizing that they must provide extensive training to new hires and provide at least some continuing training to established employees.

A proven method

This affordable courseware makes it possible to train CNC people from scratch. While we assume the student has some basic machining practice experience, we assume nothing about their previous CNC skills. Using our proven *key concepts approach*, we bring students up to speed gradually – constantly building upon previously presented information – and we stress the reasons *why* things are done as importantly as *how* they're done. Six of the ten key concepts are most related to programming, and four are related to setup and operation.

24 lessons!

We further divide the key concepts into twenty-four lessons. Lessons range from under five minutes to just over twenty minutes in length (total course presentation time is just over six hours on one CD-rom).

The most popular control!

All examples are shown in the format for the most popular control in the industry – the Fanuc control. Note that many control manufacturers claim to be Fanuc-compatible.

What you get!

The CD-rom disk is jam-packed with over six hours of information about CNC machining centers. By itself, it makes formidable training tool. It's price is \$149.00.

When purchased with the workbook answer combination (\$30.00), you'll be able to confirm that you truly understand the material (24 exercises, one for each lesson).

When purchased with the optional student manual (\$60.00), you'll have a way to easily review information after you finish the course.

If all items are purchased (total: \$239.00), we include a one-year subscription to our newsletter, The Optional Stop.



The price for unlimited training? ... \$149.00 (courseware only)

Companion manual: \$60.00, workbook/answer combination: \$30.00

Contact CNC Concepts, Inc. (847-639-8847) to order!

N015 G00 X1.0 Y1.0 (Move to first position in XY)
 N020 G43 H01 Z0.1 (Move to first Z position)
 N025 G52 X1.0 Y1.0 (Shift coordinate system)
 N030 M98 P1000 (Call subprogram – motions specified from new origin)
 N035 G52 X4.0 Y1.0 (Shift coordinate system)
 N040 M98 P1000 (Call subprogram – motions specified from new origin)
 N045 G52 X7.0 Y1.0 (Shift coordinate system)
 N050 M98 P1000 (Call subprogram – motions specified from new origin)
 N055 G52 X0 Y0 (Reset original program zero point)

Subprogram

O1000 (Subprogram)
 N1 G00 X0 Y0 (Move to center of pocket in XY)
 N2 G01 Z-0.15 F2.0 (Plunge into pocket)
 N3 X0.25 (Start milling 0.5 square pocket)
 N4 Y0.25
 N5 X-0.25
 N6 Y-0.25
 N7 X0.25
 N8 Y0
 N9 X0
 N10 G00 Z0.1 (Retract in Z)
 N11 M99 (End of subprogram)

In the main program, notice that we shift the coordinate system in line N025. The center of the 0.5 square pocket to be milled is the origin for the shifted coordinate system. Then we

call the subprogram in line N030. The subprogram moves to the pocket center (we're already there for the first pocket) and then machines the pocket. Note that all coordinates in the subprogram are specified from the center of the pocket in XY.

When the subprogram ends (N11), the control returns to the main program to the command after the calling M98, which in our case is another G52 command to shift the program zero point to the center of the next pocket to be milled. Then the subprogram is called again. We repeat this once more for the third pocket. Note that you must remember to reset the original program zero point when you're done using the temporary shift. We do this in line N055 of the main program.

One last point. Remember that each G52 is taken from the original program zero point. One common mistake is made when programmers try to shift program zero from the current shift point. Again, each G52 is specified from the same location (the original program zero point). This is why the command G52 X0 Y0 shifts program zero back to its original position.

M01

Product Corner

On-line CNC Courses

Mike Lynch has been teaching CNC-related courses and seminars for over twenty years. If you work around CNC machines, you or someone you know has probably attended one of his excellent presentations. Now we're bringing Mike's courses to the Internet. Just about everything done in his face-to-face courses will be done in the on-line courses - and you'll be able to attend from the comfort of your own home or office!

We're currently offering three courses:

- Machining Center Programming (\$69.00)
- Turning Center Programming (\$69.00)
- Parametric Programming (\$89.00)

Lesson structure is similar in all classes. Each lesson includes course text (that you can print) and a PowerPoint slide show to present the material for study. Once you've completed the lesson and are confident that you understand the material, there's an on-line test to take. Grading is immediate and automatic - so you'll see the results right away. Some lessons also require that you do a programming assignment. You turn in your assignment by email or fax to be graded. Grading is not automatic and will take a day or two. Other lesson activities (not included in all lessons) include announcements, class polls, and surveys.

All classes are on-going, meaning you can enroll and begin at any time. They're self-paced, so you can go through the course material at your own speed. If you're a motivated person, these on-line courses will work well. And if you have problems, your instructor, Mike Lynch, is just an email away!

You can find much more information about these classes on our website (www.cncci.com). Check them out today!

M01



New computer based training course! CD-ROM disk: 5.8 hours!
CNC Turning Center Programming, Setup, and Operation
Affordable courseware for CNC turning centers!
28 lessons!

A very popular CNC machine type!
 CNC turning centers are among the most popular types of CNC machine tools. Most companies that have any CNC machines have at least one. Unfortunately, companies are finding it more and more difficult to find and hire qualified CNC people. Many are realizing that they must provide extensive training to new hires and provide at least some continuing training to established employees.

A proven method
 This affordable courseware makes it possible to train CNC people from scratch. While we assume the student has some basic machining practice experience, we assume nothing about their previous CNC skills. Using our proven *key concepts approach*, we bring students up to speed gradually – constantly building upon previously presented information – and we stress the reasons *why* things are done as importantly as *how* they're done. Six of the ten key concepts are most related to programming, and four are related to setup and operation.

The most popular control!
 All examples are shown in the format for the most popular control in the industry – the Fanuc control. Note that many control manufacturers claim to be Fanuc-compatible.

What you get!
 The CD-rom disk is jam-packed with 5 hours, 52 minutes of information about CNC turning centers. By itself, it makes formidable training tool. It's price is \$149.00. When purchased with the workbook answer combination (\$30.00), you'll be able to confirm that you truly understand the material (28 exercises, one for each lesson). When purchased with the optional student manual (\$60.00), you'll have a way to easily review information after you finish the course.

If all items are purchased, we include a one-year subscription to our newsletter, The Optional Stop.

The price for unlimited training? ... \$149.00 (courseware only)
 Companion manual: \$60.00, workbook/answer combination: \$30.00



The Optional Stop Newsletter Enclosed!

Mailroom: PAID SUBSCRIPTION ENCLOSED. Please assure delivery to addressee. Thank You!

Products That Address The Needs Of CNC Technology Today!

Use this handy form to request more information about our products.

CNC Video Courses:

- Machining Center Programming And Operation
- Turning Center Programming And Operation
- Conversational Turning Center Programming & Op
- Custom Macro Programming
- Four Axis Turning Center Programming

CD-rom Courses:

- Machining Center Programming, Setup, & Operation
- Turning Center Programming, Setup, & Operation
- Setup Reduction for CNC
- Cycle Time Reduction for CNC
- Parametric Programming
- CNC Router Programming, Setup, & Operation
- Advanced Techniques With Basic CNC Features

CNC Mini-Vids:

- Related To Selected CNC Topics Like Setup Time Reduction, Cycle Time Reduction, Probing

CNC Course Curriculumms For Instructors:

- Machining Center Programming And Operation
- Turning Center Programming And Operation
- CNC Setup Reduction for CNC
- CNC Router Programming And Operation
- Maximizing CNC Utilization (NEW!)

If other than above:

Name: _____

Company Name: _____

Address: _____

City, State, Zip: _____

CNC Publications:

- Machining Center Self Study Course
- Turning Center Self Study Course
- CNC Router Self Study Course
- Managing CNC Operations
- CNC For Machining
- CNC Advanced Techniques
- CNC Accessory Devices
- The Optional Stop Subscription & Back Issues

CNC In-Plant Training:

- Basic CNC Courses
- CNC Tune-Up - For your *experienced* People



Check Out Our Website!

<http://www.cncci.com/>

Information about each of our products is on line!