

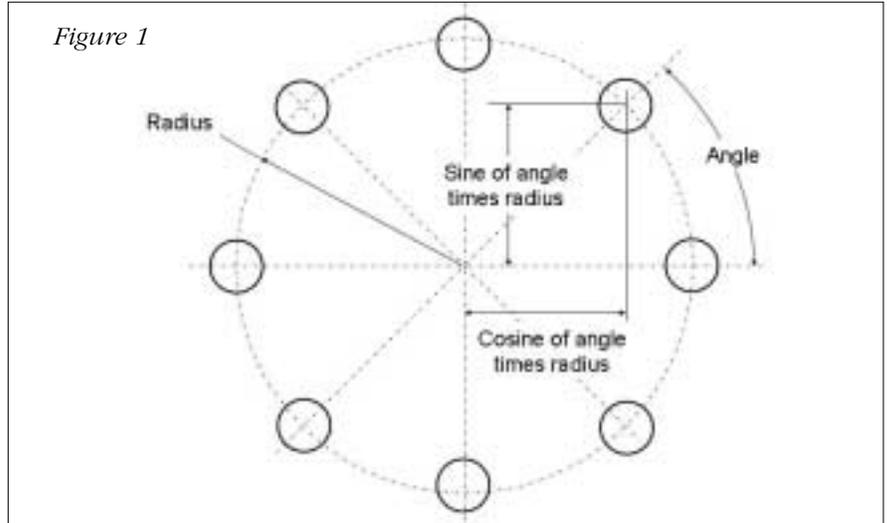
## Using Polar Coordinates for Bolt Circles

When confronted with the need to machine a bolt circle, most machining center manual programmers will use trigonometry to calculate the rectangular coordinates for each hole on the circle in X and Y. And frankly speaking, its not that hard to calculate rectangular coordinates. Figure 1 shows the two needed formulae.

Again, two simple formulae are required. For X coordinates, multiply the cosine of the angle times the radius. For Y coordinates, multiply the sine of the angle times the radius. Then add or subtract the results to or from the bolt circles center position and you have the hole's coordinates.

This method of calculating rectangular coordinates is by far, the most popular way of programming bolt circles for two reasons. First, and as you have seen, its not that hard to do. But second - and more importantly - many CNC controls (especially older ones) don't allow you to do so by any other means. The CNC control must have a feature called polar coordinates in order to program bolt hole circles without having to specify rectangular coordinates. And even some of the more popular current CNC controls (Fanuc) have some pretty severe limitations when you use polar coordinates. For these reasons, you may elect to continue using the rectangular coordinate system to machine bolt circles. But after reading this article, at least you'll know how polar coordinates can be used.

Figure 1



### Words involved with polar coordinates

Two G codes are used with polar coordinates. G16 is used to instate the polar coordinate mode and G15 is used to cancel, placing the machine back in rectangular coordinates mode. G15 is initialized, meaning when you turn the power on, the control automatically instates the rectangular coordinates mode. Some programmers like to include a G15 at the beginning of all programs as part of some safety commands that ensure the machine is in the appropriate modes when the program starts (some odd things will happen if the machine is in polar coordinate mode and you don't know it!)

When in polar coordinate mode, the letter address X no longer specifies a rectangular position along the X axis. Instead, it specifies the radius for the polar coordinate. Y no longer specifies a rectangular position along the Y axis. Instead, it specifies the angle for the polar coordinate. Figure 2 illustrates this.

As you can see, the radius of the bolt circle (X2.5 in our case) remains

the same for the entire circle. Only the angle (specified by Y) changes from hole to hole.

One severe limitation of Fanuc's polar coordinates G15 command is related to the origin for the polar coordinates. Notice from the previous drawing that the center of the pattern

### In This Issue:

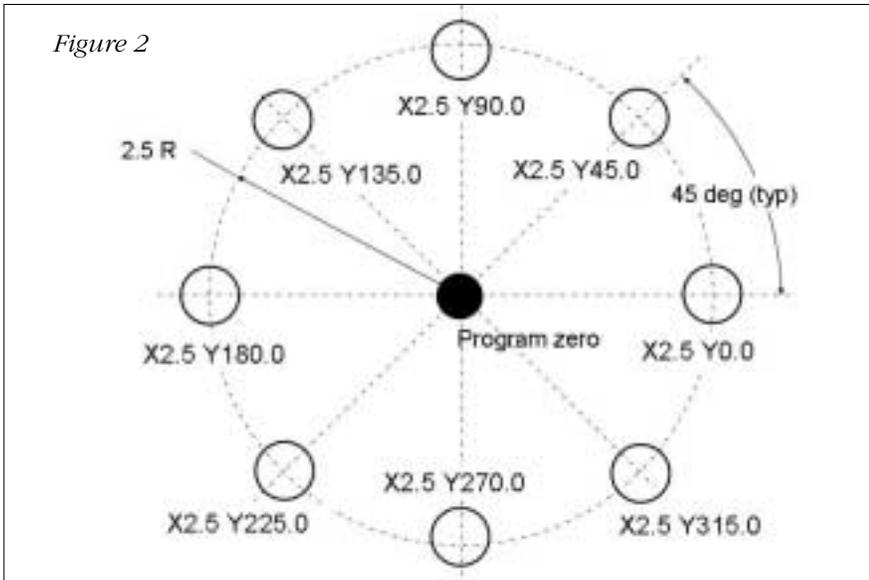
<i>Using polar coordinates.....</i>	<i>1</i>
<i>How do you use a tool pre-setter.....</i>	<i>4</i>
<i>Product corner .....</i>	<i>5</i>

### 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.cnci.com](http://www.cnci.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 2



happens to be the program zero point for the program. So Fanuc is simply assuming that the origin for polar coordinates is the program zero point - which is rarely the case with bolt circles (unless the bolt circle is around a ring or flange - and the center of the ring is the program zero point for the program). This is yet another reason many programmers shy away from polar coordinates - it's not very feasible for most applications since the origin for polar coordinates must be the program zero point for the program.

### Using G52 to shift the program zero point

Called temporary shift of program zero, the G52 command allows you to temporarily place the program zero point at the center of your bolt circle. X and Y within the G52 command tell the control the distance from the true program zero point (probably set with fixture offsets) to the center of the bolt circle - which is of course, the origin for the polar coordinate system. For example, consider the following command.

N050 G52 X3.0 Y4.0 (Temporarily shift program zero to the center of bolt circle)

This command will shift the program zero point from its original position to a location of 3.0 in X and 4.0 in Y. The control will continue to use this location as program zero until another G52 command is given.

With every G52 command, you must specify the distances in X and Y from the original program zero point to the new one. So if you have several bolt hole patterns to machine, you'll have several G52 commands - all specifying the distance from the original program zero point to the center of each bolt circle.

When you are finished machining bolt circles, you must remember to set the program zero point back to its original position. To do so, include the following command in your program.

N105 G52 X0 Y0 (Reset program zero to original position)

### Example program

Here is an example program that uses polar coordinates to machine three bolt circles and uses G52 to shift program zero to the center of each bolt circle. Figure 3 shows the workpiece to be machined.

```
O0001 (Program number)
N005 T01 M06 (Place drill in spindle)
N010 G90 G54 S1000 M03 T02 (Select abs. mode, coordinate system, start spindle, get next tool ready)
N015 G52 X4.0 Y4.0 (Shift program zero to center of first bolt circle)
N020 G00 G16 X2.0 Y0 (Move over first hole on bolt circle)
N025 G43 H01 Z0.1 (Instate tool length comp., move just above hole)
N030 G81 X2.0 Y0 R0.1 Z-0.75 F4.0 (Machine first hole of first bolt circle)
N035 Y45.0 (Second hole)
N040 Y90.0 (Third hole)
N045 Y135.0 (Fourth hole)
N050 Y180.0 (Fifth hole)
N055 Y225.0 (Sixth hole)
N060 Y270.0 (Seventh hole)
N065 Y315.0 (Eighth hole)
N070 G80 (Cancel cycle)
N075 G52 X8.0 Y8.0 (Shift program zero to center of second bolt circle)
N080 G81 X2.0 Y0 R0.1 Z-0.75 F4.0 (Machine first hole of second bolt circle)
N085 Y45.0 (Second hole)
```

**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.

**Setup reduction CD-rom course**

When machines are in setup, they are not producing. Anything you can do to reduce setup time will improve the productivity of your CNC machines. In this **3 hour, 47 minute course**, we'll expose many techniques to help you reduce setup time and cost for the two most popular types of CNC machine tools.

We begin by presenting preliminary information, including justification issues and resources you have available to reduce setup time. We then present principles of setup reduction. We show two task types related to setup, three general ways to reduce setup time, and four steps to implementing setup reduction. Finally, we show countless specific techniques you can apply to reduce setup time in the approximate order that setups are made. 135 page manual is included

**Cycle time reduction CD-rom course**

During production runs, machines are supposed to be producing. Yet there may be activities occurring that are not very productive. In this **six hour course**, we'll show many techniques that will help you keep machines in cycle for as great a percentage of time as possible.

We begin by presenting preliminary information that will help you understand how to improve machine utilization. We then present principles of cycle time reduction, including the two tasks types related to running production, the three ways to reduce cycle time, and the four steps to reducing cycle time. Finally, we show countless specific techniques you can apply to reduce cycle time in the approximate order that production runs are completed. 140 page manual is included.

**For CNC machining & turning centers**

These courses address the two most popular forms of metal-cutting CNC machine tools - machining centers and turning centers. When appropriate, we separate presentations for the purpose of clarity. However, since you may be interested in only one machine type or the other, a few presentations during workpiece sizing and dull tool replacement are duplicated for both machine types.

**Pricing:**

**CDR-STR ~ Setup reduction CD-rom course..... \$239.00**

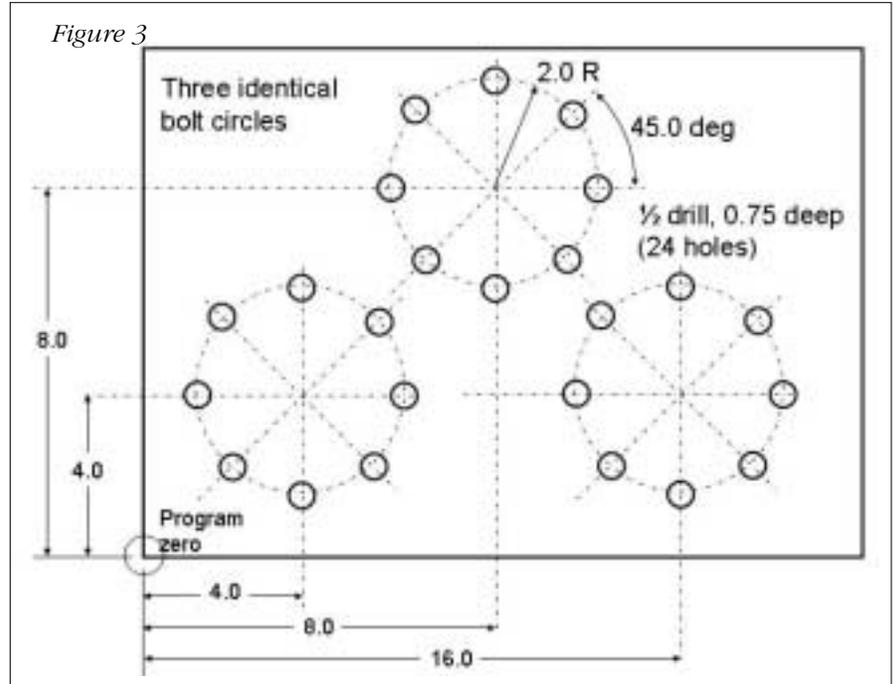
**CDR-CTR ~ Cycle time reduction CD-rom course..... \$239.00**

**CNC CONCEPTS, INC.**

N090 Y90.0 (Third hole)  
 N095 Y135.0 (Fourth hole)  
 N100 Y180.0 (Fifth hole)  
 N105 Y225.0 (Sixth hole)  
 N110 Y270.0 (Seventh hole)  
 N115 Y315.0 (Eighth hole)  
 N120 G80 (Cancel cycle)  
 N125 G52 X16.0 Y4.0 (Shift program zero to center of third bolt circle)  
 N130 G81 X2.0 Y0 R0.1 Z-0.75 F4.0 (Machine first hole of third bolt circle)  
 N135 Y45.0 (Second hole)  
 N140 Y90.0 (Third hole)  
 N145 Y135.0 (Fourth hole)  
 N150 Y180.0 (Fifth hole)  
 N155 Y225.0 (Sixth hole)  
 N160 Y270.0 (Seventh hole)  
 N165 Y315.0 (Eighth hole)  
 N170 G80 G15 (Cancel cycle, cancel polar coordinates)  
 N175 G52 X0 Y0 (Reset program zero to original position)  
 N180 G91 G28 Z0 M19 (Return to tool change position, orient spindle)  
 N185 M01 (Optional stop)  
 N190 \_

### A few notes about the program

1) Note that we had to cancel the canned cycle (G80) between bolt circles. This is because G81 is modal - and will machine a hole in every command until G80 - including the G52 command that shifts the program zero point. Without the cancellation between bolt circles, the last



hole of the first and second bolt circles will be machined twice.

2) Note that only the Y value (angle) changes from hole to hole. There is no need to repeat the redundant radius (X word) in each command.

3) Note that the actual commands to machine each bolt circle are identical. These commands could, of course, be placed in a subprogram and the subprogram can be called

three times. This will minimize the length of the program.

### Does your control allow you to specify the origin?

Again, most Fanuc controls force you to use the G52 command and keep relocating the program zero point in order to use polar coordinates. However, there are control manufacturers out there that do allow you to specify the origin for the polar coordinate system right in the polar coordinate command (though it may not be a G16). Having this ability has some pretty interesting implications.

First of all, when machining bolt circles, you won't have to cancel the canned cycle between bolt circles. And of course, you won't have to reassign the program zero point for each bolt circle. But very interestingly, you'll also be able to use polar coordinates for other positioning motions that would normally require some trigonometry. Consider the drawing shown in Figure 4.

If using the rectangular coordinate system to contour mill this workpiece, of course, you will need to do some trigonometry, at least for points 4 and 5. However, with the

**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).

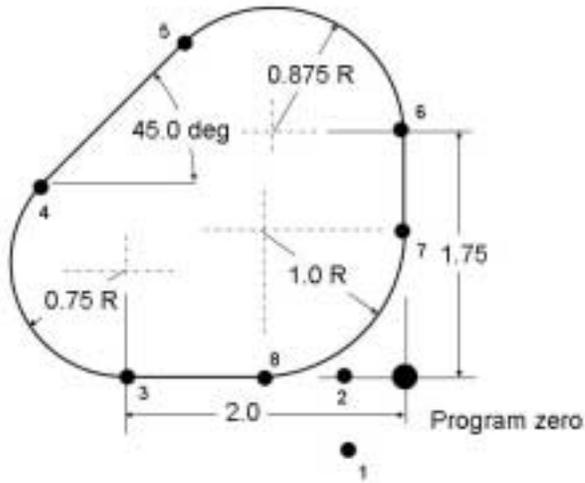
**The most popular control!**  
 All examples are shown in the format for the most popular control in the industry - the Fanuc control. Note that many control manufacturers claim to be Fanuc-compatible.

**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!

**CNC**  
CONCEPTS, INC.

Figure 4



polar coordinate system (and if you can specify the origin for polar coordinates within the polar coordinate command), this contour can be programmed manually without needing any trig!

Remember that polar coordinates are simply being used to specify the end point for positioning movements. While being done in a different manner than with the rectangular coordinate system, polar coordinates can be used in conjunction with any motion command, including G00, G01, G02, and G03. Again, were just using another method to

specify end points for motion commands.

For our example, well say G16 still instates and G15 still cancels the polar coordinate mode. Within the G16 command, say X and Y are now used to specify the origin of the polar coordinate system (which makes more sense) and that Q specifies the radius and A specifies the angle for the end point. Remember, if your control allows this, you'll have to reference your control manufacturer's manual to determine the related letter addresses for origin, radius, and angle, as well as the G codes for polar coordinates. Were simply giving one

possibility. Consider the following program.

```
O0001 (Program number)
N005 T01 M06 (Place 1.0 inch milling
cutter in spindle)
N010 G90 G54 S700 M03 T02 (Select
abs. mode, coordinate system, start
spindle, and get the next tool ready)
N015 G00 X-0.5 Y-0.6 (Rapid to point
1)
N020 G43 H01 Z-0.6 (Rapid to work
surface in Z)
N025 G41 G01 D31 Y0 F5.0 (Instate
cutter comp to point 2)
N030 G16 X-2.0 Y0.75 A270.0 Q0.75
(Instate polar coordinates, feed to
point 3)
N040 G02 A135.0 R0.75 (Circular
move to point 4)
N045 G01 X-0.875 Y1.75 A135.0
Q0.875 (Feed to point 5)
N050 G02 A0 R0.875 (Circular move to
point 6)
N055 G01 X-1.0 Y1.0 A0 Q1.0 (Feed to
point 7)
N060 G02 A270.0 R1.0 (Circular move
to point 8)
N065 G00 G16 Z0.1 (Rapid out in Z,
cancel polar coordinates)
N070 G40 (Cancel cutter comp.)
N075 G91 G28 Z0 M19 (Return to tool
change position, orient spindle)
N080 M01 (Optional stop)
```

With most controls that allow this, all words related to polar coordinates are modal. So in line N040, for example, only the changing value (the angle) must be included in the command. The origin in XY as well as the radius remain the same.

Notice that any kind of motion command can be specified - and that there is no need to do any trigonometry to determine the location of points four and five.

**M01**

## How do you use a tool pre-setter?

Measuring tool length compensation values for a machining center off line is a great way to minimize the time a machining center is down between production runs (reducing setup time). Someone can be assem-



**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



bling and measuring tools for up-coming jobs while the machine is running production. This does, of course, assume you have enough tooling, someone available to perform the related tasks, and time during the production run to perform these tasks.

Remember that there are two ways to use tool length compensation. With one method, the distance from each tool tip down to the Z axis program zero point is placed into the tool length compensation offset (a large negative value) and the Z axis program zero assignment value (fixture offset Z) is always set to zero. This tends to be a popular method for vertical machining centers, and especially in job shops and tool rooms. But with this method, tool length compensation value measurement is almost always done at the machine, while the machine is down between production runs. And these measurements are taken by the machines setup person.

If you're going to be measuring tool lengths off line, you almost have to use the second method. While there are ways to measure tools off line with this method, the process gets pretty cumbersome.

With the second method, and again, this is the method we recommend for measuring tools off line, the distance from the spindle nose (at the Z axis zero return position) to the Z axis program zero point is stored in the program zero assignment (fixture offset Z) register. This will be a large negative value. The distance from the tool tip to the spindle nose (the tool's length) is placed in the tool length compensation offset (as a positive value).

Note that this method does not force you to measure tool length compensation values off line. It can still be done at the machine. First, touch the spindle nose to a flat block (a gauge block is recommended). Second, reset the Z axis position display to zero. Third, back the spindle off and load a tool. Fourth, bring the

tool tip to the block. The tool's length will be shown on the Z axis display.

To measure off line, of course, requires some kind of measuring tool. Parlec, Kennemetal, and other manufacturers can supply them. Purchased tool length measuring devices are great (but somewhat expensive) since they have easy-to-read digital readouts and can be quickly calibrated. However, you can easily make your own tool length measuring device using a simple height-gauge.

With whichever device you use to measure tool lengths off line, you must take into consideration the small gap that occurs between the spindle nose and the flange of the tool holder. The best way to do so is to make or buy a tool holder pot that locates the tool holder (at a tool bench) in the same way that the spindle locates the tool in the machine. This way, the gap will be the same for each device, and you'll be able to zero the height-gauge at the face of the tool pot.

M01

---

### Product corner

---

#### Two new eBooks!

***Fanuc Quick Reference for Machining Centers***

***Fanuc Quick Reference for Turning Centers***

***only \$39.00 each***

**T**hese unique e-Books can be loaded on your **Palm handheld** computer to provide instant access to just about anything you need to know about using your Fanuc controlled machining or turning center. Put it in your pocket or keep it in your toolbox. When a question comes up about machine/control usage, the answer will be just a few taps away!

All information is provided using a series of text links - just like those you find in Internet websites. Any underlined text is a link. Tap any link to be shown the page for the link. Programming links include G codes, M codes, and letter addresses (these are CNC words like N, G, X, Y, Z, R, etc.). Operation links include buttons & switches and operation procedures.

---

#### **A must-have for CNC people!**

Consider the time it takes to search through programming and operation manuals to find the information you seek. When you do finally find the information you seek, you're met with the cryptic descriptions offered by most machine and control manufacturers. Quick Reference for Machining Centers will save you time and effort. Understandable answers to your questions will be but a few taps away! Order yours today!

---

#### **View it on your handheld!**

Fanuc Quick Reference eBooks are displayed with *Palm Reader*. Palm Reader is a free software program supplied by . Just in case you didn't get Palm Reader with your Palm handheld, and to keep you from having to download it from Palm Digital Media, we include Palm Reader on the CD with your e-book.

---

#### **View it on your desktop PC!**

Fanuc Quick Reference eBooks can also be displayed on your desktop PC with *Palm Reader for Windows*, another freeware program from Palm Digital Media. We include it on the CD with your e-book. Palm Reader for Windows works in much the same way that Palm Reader works on your handheld. This gives you the ability to view your e-book where-ever and when-ever you want! Another free version of Palm Reader (for Macintosh) is available from Palm Digital Media.

**Call now! 847-639-8847**

**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!