CNC CONCEPTS, INC.

The Optional Stop

Winter 2001

Tips For Fanuc Control Users From CNC Concepts, Inc.

Issue 50

44 Little Cahill Road Cary, IL 60013 Ph: (847) 639-8847 FAX: (847) 639-8857

The Optional Stop Newsletter is now on our web site!

We've always wanted to make our technical newsletter available to anyone requesting it, but printing and mailing costs have prevented us from freely distributing it. With our website, we have the ability to post this newsletter in Adobe Acrobat (PDF) format, as we're currently doing so with our Instructor Notes newsletter. A recent flurry of cancellations (due to mail contamination scares) has finally pushed us over the edge. From this issue on, we'll be posting The Optional Stop Newsletter on our website and making it available free of charge. This is the last issue that will be mailed to you.

When is each issue available for download?

The Optional Stop is published quarterly with each change in season. You'll be able to download each new issue from the first day of each new season (about March 20, June20, September 20, and December 20). We'll also place a special link on our home page (www.cncci.com) under "Recent updates to this site" as soon as each new issue is available.

Adobe Acrobat Reader

You'll need Adobe Acrobat Reader to view the newsletter. If you don't already have it, you can download the reader from the newsletter download page.

To download

Simply come to our website (www.cncci.com) and click on Books and Publications. Near the bottom of the page under "Publications" click the link for The Optional Stop Newsletter. The current issue (and up to four of the most recent issues) will be available for downloading. Click it to start the download as you would any other PDF file format download. If you have problems downloading, please contact us for help (847-639-8847).

What about my paid subscription?

If you have paid to receive The Optional Stop, unless we hear from you, we will simply continue sending each new issue by mail until your subscription ends. If this is not satisfactory, contact us to receive a refund for the balance of your subscription. We won't be accepting any more renewals or new subscriptions.

M01

A simple way of accounting for time

I've been in many companies where people seem more concerned with accounting for time than they do with making the best use of it. CNC people are often so concerned with monitoring and documenting the time spent on the various tasks they perform that little time is left to actually accomplish anything.

Admittedly, company management must be able to tell what people are doing. They must also be able to account for time spent if accurate quoting is to be done. But the process of accounting for time must be simple, detracting as little as possible from normal responsibilities.

Again, most time accounting done in manufacturing companies is done for the purpose of production control and estimating. And I've seen very elaborate methods used - methods so complicated that they don't make much sense to manufacturing people. Additionally, these accounting methods don't help much when it comes to making improvements in the CNC environment. Managers can be lulled into a false sense of security, since accounted numbers can

In This Issue:

The Optional Stop Newsletter is now on our web site!.....1
A simple way of accounting for time 1
A helical interpolation rough mill custom macro 2
Five axis alternatives 3
Are you breaking taps?.....4
A quick test for tool length.....5
Product corner 5

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. Back issues are available, but a $10.00 charge per issue will apply to any back isues not currently published on our site.
All techniques given in this newsletter are intended to make CNC usage more efficient, easier, and safer. However, CNC Concepts, Inc. can accept no responsibility for the use or misuse of the techniques given.

be manipulated to show just about anything a creative accountant wants to show.

The only two activities

Assuming you have work for your CNC machines (machines don't sit idle due to lack of work), there are only two things happening with each machine. Machines are either in setup or their running production. That's it. There is nothing else. Before you start to disagree, consider our definitions of setup and cycle time.

Setup time is the time it takes to go from making the last workpiece in the most recent production - to making the first good workpiece (efficiently) in the next production run. The entire time a machine is down between production runs (for any reason) is setup time.

Cycle time is the time it takes to complete a production run divided by the number of acceptable workpieces produced.

Based upon these definitions of setup and cycle time, you should easily agree with our point that there are only two activities for CNC machine tools. Admittedly, many activities may be going on during setup have nothing to do with making setups. A setup person may, for example, be busy changing inserts on one machine while another is down for setup. How would the setup person account for this time? If he or she "logs" the time against insert changing, it won't show up as part of setup time, yet the machine in setup is down - and based upon our definition - this time must be considered as part of setup. Of course, anything that can be done to reduce the amount of time a machine is down between production runs will reduce setup time, making the machine more productive, and should be fair game for your setup reduction program.

In similar fashion, an operator may be in the middle of a production run on a CNC machine when he or she is called away to perform some other function for the company (possibly performing secondary operations on another machine). Again, how is this time accounted for? If the operator logs the time against the secondary operation, it won't show up as part of cycle time (or production run time) for the machine that sits idle. Of course, anything that shortens the time it takes to complete a production run will improve cycle time, and should be fair game for your cycle time reduction program.

Which method is best?

We're not trying to encourage you to change your accounting methods. But we are trying to expose potential limitations. Our simplistic accounting method work best when your people are trying to make improvements. It provides a very narrow focus for anyone working to improve.

M01

A helical interpolation rough mill custom macro

by Scott Lowe of Maha USA

I came up with this macro because we do a lot of hole making through helical interpolation. This macro makes it more efficient to write such programs and you are also less prone to make data entry mistakes because this task has been lessened considerably. This is being done using custom macro B on a Fanuc 16M control.

Here is the custom macro.

```
:7100(HELICAL INTERPOLATION
  ROUGH MILL)
(10-15-01)
#24=#5041 (X AXIS CURRENT POSI-
  TION SYSTEM VARIABLE)
#25=#5042 (Y AXIS CURRENT POSI-
  TION SYSTEM VARIABLE)
#30=[#1-#2]/2 (SUBTRACT CUTTER DI-
  AMETER FROM HOLE DIAMETER/2)
IF[#3EQ2]THEN#32=42 (CONDITIONAL
  BRANCH STATEMENT ESTABLISHING
  CUTTER COMP RIGHT)
IF[#3EQ3]THEN#32=41 (CONDITIONAL
  BRANCH STATEMENT ESTABLISHING
  CUTTER COMP LEFT)
G90G0Z3. (RAPID TO THREE MILLIME-
  TERS ABOVE PART)
G90G1Z0.F#9 (FEED TO Z ZERO)
G90G1G#32D#7X[#30+#24]F#9 (FEED
  RADIALLY OUT TO SPECIFIED HOLE
  SIZE)
#31=#6+0
#33=0
N10IF[#33LE#26]GOTO20 (CONDI-
  TIONAL BRANCH USED FOR REPETI-
  TION)
```

```
G90G#3I-#30G91Z-#6 (ONE FULL HELI-
  CAL REVOLUTION)
#33=#33-#31
GOTO10
N20G90G#3I-#30 (ONE FULL REVOLU-
  TION TO CLEAN UP BOTTOM OF
  HOLE)
G90G1X[#24-#30/4] (FEED OFF)
G90G0Z#18 (RAPID OUT OF HOLE)
G40D0
M99
```

Here is an example of a calling command in the cutting program. Note that I load the tool, start the spindle, make a prior move in XY, and instate tool length compensation prior to calling the macro. Also, I use the metric mode, so all values are specified in millimeters.

```
G65 P7100 A38.0 B19.05 C3.0 D4.0
  F1400.0 R20.0 Z-20.0 K2.0
```

Here is what each variable in the call statement represents:

```
A(#1) = HOLE DIAMETER
B(#2) = CUTTER DIAMETER
C(#3) = CUTTER FEED DIRECTION
  (3=COUNTERCLOCKWISE, 2=CLOCK-
  WISE)
D(#7) = CUTTER COMP OFFSET
  NUMBER
F(#9) = FEEDRATE IN MM PER MINUTE
R(#18) = CLEARANCE PLANE
Z(#26) = DEPTH OF HOLE(ALWAYS A
  NEGATIVE NUMBER)
K(#6) = Z DEPTH PER ONE FULL HELICAL
  REVOLUTION
```

Note that this custom macro will cause multiple passes to be made until the depth per one full helical revolution (K) is equal to the depth of hole (Z). Z must be evenly divisible by K. Also, this custom macro requires that you're machining into the Z0 surface. Simple modifications would allow you to machine into any Z surface.

I establish variable #31 by adding zero to variable #6(address K). Variable #33 starts off at zero. As part of the repetition process, I take variable #33 and subtract variable #31 from it, coming up with the current Z end point for the helical move. From the example above, variable #31 equals #6 (2.0mm) plus zero. When I subtract #31 from #33, variable #33 becomes negative 2mm. With repetition, #33 becomes increasingly more negative (by 2.0 mm increments in this example. In block N10, if #33, is less than or equal to #26 (depth of hole) then the program will jump to block N20. Number #33 keeps accumulating depth until the condition in block N10 is met, in which case the loop is exited.

M01

Five axis alternatives

by Dennis Griffon of Tri-Tech Precision Products, Inc.

There are currently three alternatives to achieve 4th and 5th axis machining capabilities:

1. Five-axis Machining Center
2. Tilting / Rotary Tables
3. Spindle Head Attachments

I thought your readers might benefit from my insights about these alternatives.

Five-axis Machining Center - There are currently more than fifteen (15) original- equipment manufacturers of five-axis machining centers sold in the U.S. These manufacturers include all of the biggest names in manufacturing technology such as: Cincinnati Milacron, Giddings & Lewis (parent of Fadal Engineering), Haas Automation, Deckle-Maho, Nicholas-Correa, SNK, Okuma, Toshiba and Mitsubishi. The majority of these machining centers are large, powerful, accurate and expensive with prices averaging well above $500,000. The least expensive 5-axis machining centers on the market today have a starting price of approximately $350,000. The chief draw-back of these 'lower cost' 5-axis machining centers is their range of motion which is generally limited to (+/- 35□). Work pieces that require a steeper angle of cut must be manually repositioned and restarted. Also, these machines are somewhat less rigid than their 3-axis counterparts.

Rotary Tables - Several manufacturers now produce rotary or tilt-rotary tables which can be mounted to the bed of a (3-axis) CNC machine. The table will hold and rotate a part at various angles to provide machine access to many sides of the part. Rotary tables have been available for many years and are used extensively by small and large machine shops around the world. Simple rotary tables are available to

provide 4th-axis capability to nearly any 3-axis machine at a price ranging from $5,000 to $35,000. Tilt-rotary tables can provide 4th and 5th axis capability with a price range of $35,000 to $75,000. The main draw-back of tilt-rotary tables is that they are limited to small parts that can be attached to the table itself. Even the largest tilt-rotary tables can reduce the working envelope of the host machine by 60% to 70%.

Spindle Head Attachments fall into three categories -

a) *Indexable head attachments* such as the Tech-Nara, Gerardi or Benz can be mounted to the spindle of most 3-axis machining centers. The indexable head must be manually repositioned to various angles of as much as 90 degrees in increments of one degree. The repositioned head can then access all points of the host machine's working envelope. The indexable head attachment is not programmable and therefore cannot perform simultaneous 5-axis machining. The head requires manual intervention and another machine pass for each new angle required and cannot be set for complex angles or fractions of a degree. The head is designed for only light machining and is not suited for hard metals or significant metal removal. In spite of these limitations more than 3,000 of these attachments are sold annually with list prices in excess of $10,000.

b) Programmable - Two U.S. companies (Dayton Machine Tool and Bertsche Engineering Corp.) and at least one German company manufacture and sell a programmable head for attachment to large 3-axis milling machines. These heads are large, fairly accurate and expensive. Because of their size and weight they cannot be mounted onto any of the small to mid-sized machining centers and are reserved for only the largest gantry and bridge type milling machines. Each installation requires custom fabrication and fitting. The price range of these head attachments is $175,000 to $350,000 plus the cost of custom fabrication and installation..

c) My company's Tri-Tech Model 5412 head attachment is fully programmable and can be used to convert nearly any 3-axis machining center to a true, simultaneous 5-axis machining center. The M5412 has a wide range of motion including tilt capability of +/- 90□ and continuous 360□ rotary motion. Because the M5412 is a spindle attachment, it can access all points of the machining center's work envelope. The M5412 is portable and can be mounted, in thirty (30) minutes, with no modification to the host machine. At nearly 150 pounds, the M5412 is a heavy duty attachment yet compact enough to attach to many smaller to mid-size machines.

While the "low priced" 5-axis machining centers are fairly rigid, they cannot match the rigidity of their 3-axis counterparts for significant metal removal. The M5412 allows the user to perform all significant metal removal using the heavy duty 3-axis machining center then mount the M5412 attachment

M01

Are you breaking taps?

Editor's note: This article was written by a person that did not provide a name from a company called Function First Fabricating. If this person provides his or her name, we'll gladly include it in our next issue.

If your machine does not have a spindle encoder, you should be using a "floating" tapping holder. The holder will extend or compress to make up for mismatches in speed and feed.

The spindle takes longer to reverse directions than the feed. A machine with a spindle encoder will recognize this, and adjust the feed to match (rigid tapping). If you don't have an encoder on your spindle, a floating tap holder is the way to go. I have heard, and seen people get away without one by running very low spindle speeds. But if you tap many holes at all, the special holder is worth the money.

If you go with the floater, you can kick your spindle speeds back up to normal. Just remember a couple things:

(1) The spindle is still going to "coast down", even more so with higher spindle speeds. On most of our machines this translates into about 1 turn for every 100rpm. Program the depth a little short to keep from coasting too deep, especially on blind holes.

(2) Since the holder can extend, it's possible for the tap to have not made it completely out of the hole before the machine moves to the next location (snap). Use a larger R-plane to help avoid this.

A couple of the controls we own have features to compensate while "flexible" tapping. Fadal can use a P-code to adjust feed on the way out of the hole. P5, for example, reduces the feedrate by 5% on the way out. Incon allows a D-code to dwell (feed) at the bottom to wait for the spindle to catch up.

If you have the spindle encoder, nothing beats rigid tapping.

M01

A quick test for tool length

by Bob McRoberts

In appreciation for the help you've given me, I would like to offer you an idea for your newsletter.

The problem:

I have a Mori-Seiki SH-403 Horizontal Machining Center. On a specific job, the tool length is critical. If the tool length was too short, the spindle face would collide with the tombstone. Therefore, I need to "test" the tool length to ensure that it is long enough. I do so with custom macro B. Here is a few lines of the program that should give your readers the idea.

```
N1(T1   SPOT DRILL)
IF[#11001 LT 4.5]GOTO 902  (checks T1
  tool length for 4.5" min.)
'
' (body of program)
'
G0 G80 G91 G28 Z0.0 M9 (home ma-
  chine in Z axis)
GOTO 901
N902 #3000=1 (TOOL TOO SHORT)
N901 M1
```

On my machine, the #11001 system variable returns the currently instated tool length compensation value. This variable number may be different on other controls.

Product Corner: New CD-rom Course: Setup reduction for CNC machining & turning centers!

Get the most from your CNC machines!

During any CNC machine's usage, there are really only two activities. Machines are either 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.

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 comprehensive 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 basic premises that will help you understand how to improve machine utilization. Included are presentations on justification issues and resources you have available to reduce setup time.

We then present principles of setup reduction; principles that can be applied to any form of production equipment. 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 - from preparation to make setups through first workpiece inspection! In each presentation, we include printable handouts (63 handouts totaling over 120 pages) to reinforce your understanding of presented techniques. A 135 page companion manual is also included.

Price: $239.00 includes 3 hours, 47 minutes of presentation, 135-page companion manual, & 63 printable handouts (over 120 pages) in Adobe Acrobat format. Adobe Acrobat Reader is included on the CD.

Courseware only: $189.00

Special offer! Until January 31, 2002, you can get the full course (CD & manual) for $199.00! You can purchase the CD only for $149.00! Order today!

New Instructor's Curriculum for Setup Reduction!

Here's another ready-made CNC curriculum to help you teach CNC courses!

Paralleling our CD-rom course for setup reduction, this curriculum helps you teach live courses on setup reduction. Instructor materials include slide shows with and without audio guidance (totaling over 400 slides each), instructions, two written exercises (with answers), setup reduction planning forms, and suggestions for group activities. Student materials include a 135-page manual.

As with all of our curriculums, **instructor's materials are free** with your first order of student manuals (minimum 20 to qualify for free instructor materials).

Download free samples on our website or request free samples disk.

You can see free samples for both the CD-rom course and the instructor's curriculum for setup reduction on our website!

www.cncci.com

CNC CONCEPTS, INC.
44 Little Cahill Road
Cary, IL 60013
Ph: (847) 639-8847
Fax: (847) 639-8857

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
- ❑ Parametric Programming
- ❑ CNC Router Programming, Setup, & Operation
- ❑ Advanced Techniques With Basic CNC Features

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 Mini-Vids:

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

CNC In-Plant Training:

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

CNC Course Curriculums 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!)

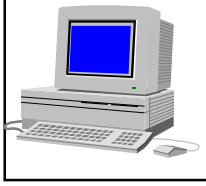
If other than above:

Name: _____

Company Name: _____

Address: _____

City, State, Zip: _____



Check Out Our Website!
http://www.cncci.com/
Information about each of our products is on line!