

Computing degrees per minute feedrate for rotary axes

Anyone commanding their first feedrate motion (G01) for a rotary axis is in for a rude awakening (machining center or turning center). Unless some higher level programming feature is being used, like cylindrical interpolation on machining centers or polar coordinate interpolation on turning centers, feedrate must be commanded in *degrees per minute* – not inches or millimeters per minute. This is true regardless of whether the motion includes only a rotary axis motion or if the motion includes a rotary motion along with an X, Y, and/or Z motion.

If unaware of this point, most programmers will unwittingly program the feedrate in inches (or millimeters) per minute and then spend hours trying to figure out (at the machine) why the motion rate is inappropriate (in most cases, *much* too slow).

When you really think about it, the CNC control will have no way to determine how fast to rotate the rotary axis if feedrate is specified in inches or millimeters per minute. It does not know the tool's relationship to the center of rotation (unless

you're using cylindrical or polar coordinate interpolation). For this reason, a feedrate in inches or millimeters per minute will have no meaning to the control. This means you must determine and program feedrate for any motion involving a rotary axis in degrees per minute.

Determining degrees per minute feedrate

While there are probably more streamlined formulae to calculate degrees per minute feedrate (and we would be interested in hearing them for publication in a future article), the method we show should help you understand what goes into calculating degrees per minute feedrate as much as how it is actually done. Though similar methods can be developed if you work with the metric system, our examples assume that you work in the inch mode (that you currently program feedrate for linear axis motions in inches per minute).

It can be somewhat difficult to calculate degrees-per-minute feedrate. With our recommended method, you must first know, of course, the feedrate needed in inches per minute. The method we will show to calculate degrees-per-minute feedrate involves calculating the linear length of motion (even though the motion is rotary) and the machining time based on the desired inches per minute feedrate. Only then can the degrees-per-minute feedrate be calculated.

Before we give the actual formulae, consider a simple example to help you visualize what's going on. Say you have to mill a surface involving a 360-degree rotation of the rotary axis. Maybe you're milling a slot or groove around the circumference of a round workpiece. The diameter of this workpiece (at the milling

plane) is 3.25 in diameter. The total milling travel distance will be 10.21 inches (the circumference of a 3.25 in circle [$\pi \times \text{Diameter}$]). Now, consider the inches-per-minute feedrate you would use to mill this slot. Using normal methods (rpm times inches per tooth times number of teeth), say you determine that this cutter should run a 10 inches per minute. Think about it. This motion will take about one full minute to complete (actually 1.021 minutes). Since a full 360-degree rotation will be required, the degrees-per-minute feedrate will be very close to 360 degrees per minute – F360.0! (The actual feedrate will be 352.59 dpm.)

From this example, you should be able to see why our calculations involve determining how much time it will take for the motion to occur. Based upon this time, you can determine how many degrees of rotation

In This Issue:

<i>Computing degrees per minute feedrate</i>	1
<i>Four advantages of constant surface speed</i>	3
<i>Product corner</i>	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 (www.cncci.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.

Come visit our *secure* ON-LINE STORE!

Now you can purchase our CNC training products on-line with a credit card (Visa or MasterCard)! Come visit our recently updated site today!

Products include:

- ☆ CD-rom courses
- ☆ Books & publications
- ☆ Video courses
- ☆ Mini-Vids

www.cncci.com

must occur in one minute. You should also be able to see that degrees per minute feedrate will often be much larger than feedrate specified in inches per minute. And remember – feedrate is modal. If you transition from a motion involving rotary axis motion (large feedrate number) to a motion that involves only a linear axis motion, you must remember to include the (much smaller) inches per minute feedrate word. If you do not, the control will use the very large degrees per minute feedrate as an inches-per-minute feedrate!

Here are the steps we recommend that you take to determine degrees-per-minute feedrate:

1) Determine the length of the motion. While you don't have to be perfect, the closer you come to calculating the actual travel distance, the more precise will be your calculated degrees-per-minute feedrate. Remember that the circumference of a circle (the arc size at the milling plane) is PI times the diameter of the circle. If milling half-way around a circle, simply divide the circumference by two. If milling one-quarter way around the circle, divide the circumference by four. If milling an odd angular amount (say 80 de-

grees), first divide 360 by the angular departure to determine what to divide the circumference by. If the motion involves 80 degrees of rotary departure, for example, divide 360 by 80, to determine what to divide the circumference by (4.5). It is more difficult to determine motion distance if the motion involves rotary and linear axis departure. But again, the more accurate your determination of motion distance, the more accurate your calculated degrees per minute feedrate will be.

2) Determine your desired inches per minute feedrate. Do this just as you do for any other milling operation. Multiply rpm times inches per tooth times number of teeth.

3) Determine how long (in minutes) it will take to make the motion. Time in minutes is equal to motion distance divided by inches-per-minute feedrate. If, for example you determine that motion distance is 5.5 inches and you want to machine at 8.0 inches per minute, it will take 0.6875 minutes (5.5 divided by 8.0) to complete the motion.

4) Determine the feedrate in degrees per minute. Degrees per minute is equal to the amount of ro-

tary axis departure needed for the motion divided by the time it takes to make the motion in minutes (from step 3). Note that you only consider rotary motion. If the motion involves linear axis motion, disregard it. If, for example, the motion involves 80 degrees of rotation, and you've determined (from step 3) that the motion will take 0.5 minutes, the degrees-per-minute feedrate (F word) will be 160.0 - programmed, of course, with F160.0.

Once again, most feedrate specifications in degrees per minute will be much larger than their inches per minute counterparts. The control only looks for rotary axis departures to determine whether to use the specified feedrate as a degrees per minute or inches-per-minute value. If it sees a rotary axis departure in the command, feedrate will be taken in degrees per minute. If no rotary departure is included in the command, (modal) feedrate will be taken in inches per minute. Since degrees-per-minute feedrate specifications are usually much larger than inches per minute specifications, you must remember to include a new feedrate specification (in inches per minute) if the machining operation continues, transitioning from a motion that involves a rotary axis motion to one that does not.



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.

We further divide the key concepts into twenty-eight lessons. Lessons range from under five minutes to just over twenty minutes in length (total course presentation time is five hours fifty-two minutes 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 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

A way to cheat

Because degrees-per-minute feedrate is difficult to calculate, some programmers cheat. They approximate the degrees per minute feedrate and wait until the part is actually being machined. They fine tune the feedrate during machining by adjusting the feedrate override switch. After overriding the feedrate to the desired level, they change the programmed feedrate accordingly. For example, say you program 100 degrees per minute for all motions involving a rotary axis departure. At the machine, when making the cut, maybe you must increase the feedrate override switch to 140% to attain what you consider to be a

good cut. In this case, the feedrate in the program must be changed to 140 degrees per minute. While experienced machinists may be able to develop pretty good cutting conditions using this method, we don't recommend it. With this method, you really won't know what the true inches-per-minute feedrate is.

Machining centers versus turning centers

All points made in this article apply to rotary axes on machining centers as well as turning centers. With machining centers, rotary axes are placed on the table top of vertical machining centers (and specified with either A, B, or C) or they are made an integral part of the table in horizontal machining center (specified with B). With turning centers having a rotary axis (commonly called mill/turn machines), the rotary axis is an integral part of the spindle and specified with the letter address C.

Remember that the reason why you must specify feedrate in degrees per minute has to do with the fact that the control has no idea where the tip of the cutting tool is relative to the center of rotation when you command a rotary axis departure.

But with certain control features, this may not be true. With cylindrical interpolation on machining centers (used to mill around the periphery of a round workpiece), for example, part of the cylindrical interpolation command is to specify the work surface plane relative to the center of rotation. When this programming feature is used, feedrate is specified in inches or millimeters per minute. In similar fashion, when using polar coordinate interpolation on turning centers, you are allowed to specify feedrate in inches or millimeters per minute. (Cylindrical and polar coordinate interpolation are the subjects of articles previously published in *The Optional Stop* newsletter).

M01

The four advantages of constant surface speed


For Fanuc controlled turning centers, G96, of course, specifies constant surface speed mode, while G97 specifies rpm mode. In constant surface speed mode, the spindle speed in rpm is automatically determined by the CNC control based upon the diameter a tool is currently cutting

and the speed specified in surface feet per minute (or meters per minute in metric mode). This mode is only used for single point turning tools (boring bars, turning tools, grooving tools, etc.), when diameters to be machined change substantially throughout the workpiece. There are (at least) four benefits to using constant surface speed mode for appropriate applications.

1) Easier programming

The correct rpm for a cutting tool is based upon this formula: $RPM = 3.82 * SFM / DIAMETER$. If an accurate speed in rpm is to be used, the person calculating the rpm must first determine the tool's appropriate speed in surface feet per minute (note that the formula is different when working in the metric mode). Before a person can even begin to make the calculation, they must know the desired speed in sfm. This speed is published by cutting tool manufacturers and is based upon the material being machined and the material of the cutting tool's cutting edge. When using constant surface speed mode, the programmer simply enters the speed in the program directly in sfm (no need for any calculations). The machine will constantly and automatically perform this calculation, updating the spindle speed in rpm.

One common reason why setup people and operators on the shop floor continue to use rpm mode - even though it is easier to use sfm mode - is that they have a background in manual machining. With manual lathes, there is no such thing as constant surface speed. Everything must be done in rpm. When a manual machinist is exposed to constant surface speed on a CNC turning center for the first time, the spindle will probably appear to be running much too fast (most manual machinists cannot run at optimum spindle speeds for obvious safety reasons). Frankly speaking, many manual machinists determine spindle speed in rpm by the seat of their pants, caus-



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!



ing the spindle to run at a speed that looks good. Indeed, I've talked to many that never use the rpm formula given earlier - and some that have never heard of it. Many continue using the seat-of-the-pants approach even on CNC turning centers when it is safe to run optimum spindle speeds.

2) Consistent workpiece finish

As long as you work inches- (or millimeters-) per revolution feedrate mode (G99 on a Fanuc control), witness marks left on the workpiece by the cutting tool will remain consistent throughout the cutting tool's machining of the workpiece if you use constant surface speed. Since the feedrate is tied to spindle speed, as the spindle speeds up and slows down in rpm, so will the feedrate in inches per minute. Again, this causes the tool to create a consistent finish throughout the workpiece.

3) Optimum tool life

Since the spindle is constantly running at the appropriate speed in sfm, tool life will be at it's maximum. When compromising speed as must be done when working in rpm mode, the spindle seldom runs at the appropriate rpm, and tool life may suffer.

4) Optimum cycle time

Again, feedrate is directly tied to spindle rpm if you're working in per revolution feedrate mode with constant surface speed. The faster the spindle runs, the faster the tool will machine. So when you're working in constant surface speed mode, cycle time is inversely proportional to spindle speed in rpm. The faster the spindle runs, the shorter the cycle time. Since most machinists working exclusively in rpm mode tend to error on the side of caution (commonly choosing an rpm that is much slower than it must be), cycle time will be longer than it should be.

A few cautions

Out of round workpieces - If you machine a lot of castings or forgings (or any other kind of workpiece that doesn't run true in the spindle), you must be very careful with the constant surface speed mode. One of the reasons why manual machinists have trouble getting used to constant surface speed mode is that they cannot predict what rpm the spindle will be running. And if, for instance, you program the cutting tool to go to the spindle center in X, as is done when facing a workpiece to center, the spindle will run up to its maximum speed in the current spindle range. If working in the high spindle range, the spindle will run up to its maximum. For round, true running workpieces, this is acceptable. But if workpieces are at all out of round, the vibration set up by running too fast can be very dangerous. It can cause the workpiece to be thrown from the workholding device. Note that there is a special function that lets you set a temporary maximum rpm (G50 with Fanuc). But you must remember to use it (and know the maximum rpm a workpiece can be rotated without vibration).

Poor workholding setups - Since speed in rpm is difficult to determine when working in constant surface speed mode, it should only be used when making adequate setups. Another reason why manual machinists tend to prefer working in rpm mode (and running at slower than optimum speeds) is that the workholding devices they've used on manual lathes (commonly manual three-jaw chucks) could not apply the gripping force necessary to run at optimum spindle speed. Most CNC turning centers use hydraulic three-jaw chucks that can supply the gripping force needed to run at maximum speeds. But if you're workholding device cannot grip with adequate force, of course, you cannot run at optimum speeds.

Cycle time waster - While one of the benefits of constant surface speed is improved efficiency, it must

be programmed wisely. If you work exclusively in the constant surface speed mode, the spindle will always be rotating at the appropriate speed - matching the diameter the cutting tool would be cutting. While this is great during each tool's cutting operation, consider what happens during tool changing. The cutting tool will rapid to a tool change position (usually a large diameter in X). During this motion, the spindle will slow down. Depending upon your spindle drive system's response time, it will likely take longer for the spindle to slow down than for the rapid motion to occur. If it does, cycle time will be wasted. The same is true dur-

CNC TRAINING

Fanuc Users!

New CD-rom Courses!

No other training media offers the interactivity of software based training. Click on a lesson title, and you're shown the topics for that lesson. Click on a topic to see & hear content for that topic. When you've studied all topics in a lesson, do an exercise and check it. And then - on to the next lesson!

► **Two Basic CNC Courses:**

- ✕ Machining center programming, setup & operation.....**\$239.00**
- ✕ Turning center programming, setup & operation.....**\$239.00**

► **Three Advanced Courses:**

- ✕ Advanced techniques with basic CNC features.....**\$189.00**
- ✕ Setup reduction for CNC machining & turning centers.....**\$239.00**
- ✕ Parametric Programming for CNC machining & turning centers...**\$795.00**

CNC Mini-Vids!

One-hour (approx.) videos on selected CNC topics.
\$179.00 each

- ✕ Setup time reduction for CNC
- ✕ Cycle time reduction for CNC
- ✕ Introduction to probing
- ✕ Machining center cutting operations
- ✕ Introduction to Fanuc parameters
- ✕ Live tooling on turning centers
- ✕ Rotary device programming

For complete information about these products (and more) come visit our web site! www.cncci.com

CNC CONCEPTS, INC. 44 Little Cahill Road Cary, IL 60013 Ph: 847-639-8847 Fax: 847-639-8857 Email: lynch@cncci.com Internet: www.cncci.com

ing each tool's approach. As the tool rapids to a smaller diameter, the spindle speed will increase in rpm. This constant increase and decrease in rpm not only wastes cycle time, it wastes electricity and causes undue wear-and-tear on the spindle drive system. To program around this problem, most programmers will temporarily switch to the rpm mode just before the rapid motion to the tool change position, using a speed in rpm that is appropriate for the next tool's first position. In this way, the wasted change in spindle speed will not occur, and when the next tool is in position, the spindle will be running at its correct rpm. At this point the constant surface speed mode is re-selected with the proper speed in sfm.

Minimal diameter changes - If you are machining but one diameter on a workpiece, or if there is but a small difference from one diameter being machined to the next (say, under 1 inch of diameter change), constant surface speed will not help you much. In this case, many programmers will calculate the appropriate speed in rpm and run the machine in rpm mode. This also eliminates the cycle time wasting problems caused by constant surface speed (just mentioned).

M01

Product corner

Instructors! New update for Machining Center Curriculum!

If you are currently teaching with our machining center curriculum, be sure to request the free update. We've improved the curriculum in four ways:

1) Presentation links slide in every lesson

We've made each lesson interactive. Instead of having to find the slide that begins each topic on your own, we've added a special links

slide in each lesson. This lets you start/continue your presentation at the start of any major topic in the lesson.

Each underlined topic on this slide is a text link that, if clicked, will send the presentation to the chosen topic within the lesson. To get back to the presentation links page, simply click the return button in the lower left corner of any slide.

As you're preparing to teach, you may decide not to include certain topics in your presentation. Perhaps you want to skip them for now and come back to them later. In this case, when you get to the topic you wish to skip, simply click on the return button to get back to the presentation links page. Then click on the topic after the one you wish to skip.

2) Instructor notes manual

This version of the curriculum still includes the set of slides containing audio guidance, and we think you'll agree that it makes the easiest way to gain an understanding of how to present each topic throughout the course. While we feel this audio guidance is extremely helpful, we've received several comments from instructors saying they want a way to prepare to teach without having to have a computer available – and that they'd like a way to make notes of their own as they present the course.

The Instructor Notes manual contains information that parallels what is done with audio guidance, giving you another way to prepare to teach. You can now do so without having a computer. However, we do not include every slide in this manual (there are over 6,500 slides in the curriculum). Instead, we've picked key slides at transition points and provided a short description of how to present the topic. Even so, this manual is 644 pages long!

For the update, this manual is included on disk one in Adobe Acrobat format, so you can make copies (be ready for a lot of printing!). Or you can purchase it from us at an additional price (\$75.00).

3) Fly-in text boxes

Regardless of how much preparation you do, it will be impossible to memorize every presentation included in this lengthy curriculum. Again, there are over 6,500 slides. It can be especially difficult during transition points, when the presentation is shifting gears. If you don't know what's coming up (as you're teaching), you can look ill-prepared in front of your students. For this reason, most slides now include fly-in text boxes to help you stay on track. Here is an example. The text box in the upper right corner will fly in from the right as soon as this slide is displayed.

Sometimes the fly-in text box will keep you from forgetting to make a key point. Other times it will help at a transition point to remind you about where the presentation is going. In many cases, what is in the text box can be used as your first sentence for the slide, helping you quickly dig into the material.

4) Review slide presentation

We cannot stress enough the importance of reviewing information you have presented earlier in the class. In the review slide presentation, we begin each key concept with the key concept name and a list of lessons related to the key concept. This lets you re-stress the importance of each key concept. We then include a slide for each lesson in the key concept with bullet points to help you remember the topics that need reviewing. Actually, these bullet point slides are identical to the presentation links slide (discussed earlier) in each lesson, except they are all together in one handy presentation. If students are having problems with a topic, simply call up the lesson's presentation and go to the presentation links page. Then click on the topic to present it again.

A hard copy of this presentation is included.

M01

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: _____

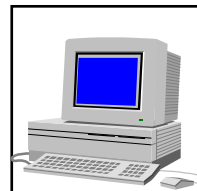
City, State, Zip: _____

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!