

CNC Code Shooter Help



User Manual

Copyright 2018 R. L. and V. L. Ellison

Installation

1. Requirements for Download:

You must have Microsoft Dot-Net Framework 2.0 or later to run this program. The Dot-Net Framework comes standard with Windows. If you don't have it you can download it from:

<http://msdn.microsoft.com/en-us/netframework/default.aspx>. has the .Net Framework 2.0 redistributable package that you need. Download it and install it onto your hard drive. Follow the onscreen instructions to finish with the install.

2. Installation and Set - up:

After you have downloaded the program, you are ready to install. You have a file called: CNCCodeShooter4_** Install.exe. Double click on that file to install the program on your hard drive. Follow the onscreen instructions to finish with the install. After installing you must close down the program that starts from the installer. The program will not function properly that starts from the installer. Start the program from Start -> All programs.

You will have to right click and choose run as administrator to get the program to run or click properties and select run as administrator.

A monitor with a resolution of 768 x 1024 is required.

3. Now you can run the program, it should be in your program files, CNCcodeShooter4_** folder. Select all programs then select CNCcodeShooter4_45 right click and click on Run as Administrator to start the program.

CNC CODE SHOOTER - Help

Basics

The basics for writing a program is to give it a number and description.

Draw the pattern.

Fill out tool information.

Choose a pattern to cut and parameters.

Make G code.

For Lathe

You must draw patterns as you will cut them. For Lathe if you are going to cut from the face to the chuck it has to be drawn that way.

For Mill

You must draw patterns as you will cut them. If you are going to cut

counterclockwise around a pattern it has to be drawn that way.

CNC CODE SHOOTER Mill

<http://www.cnccodeshooter.com>

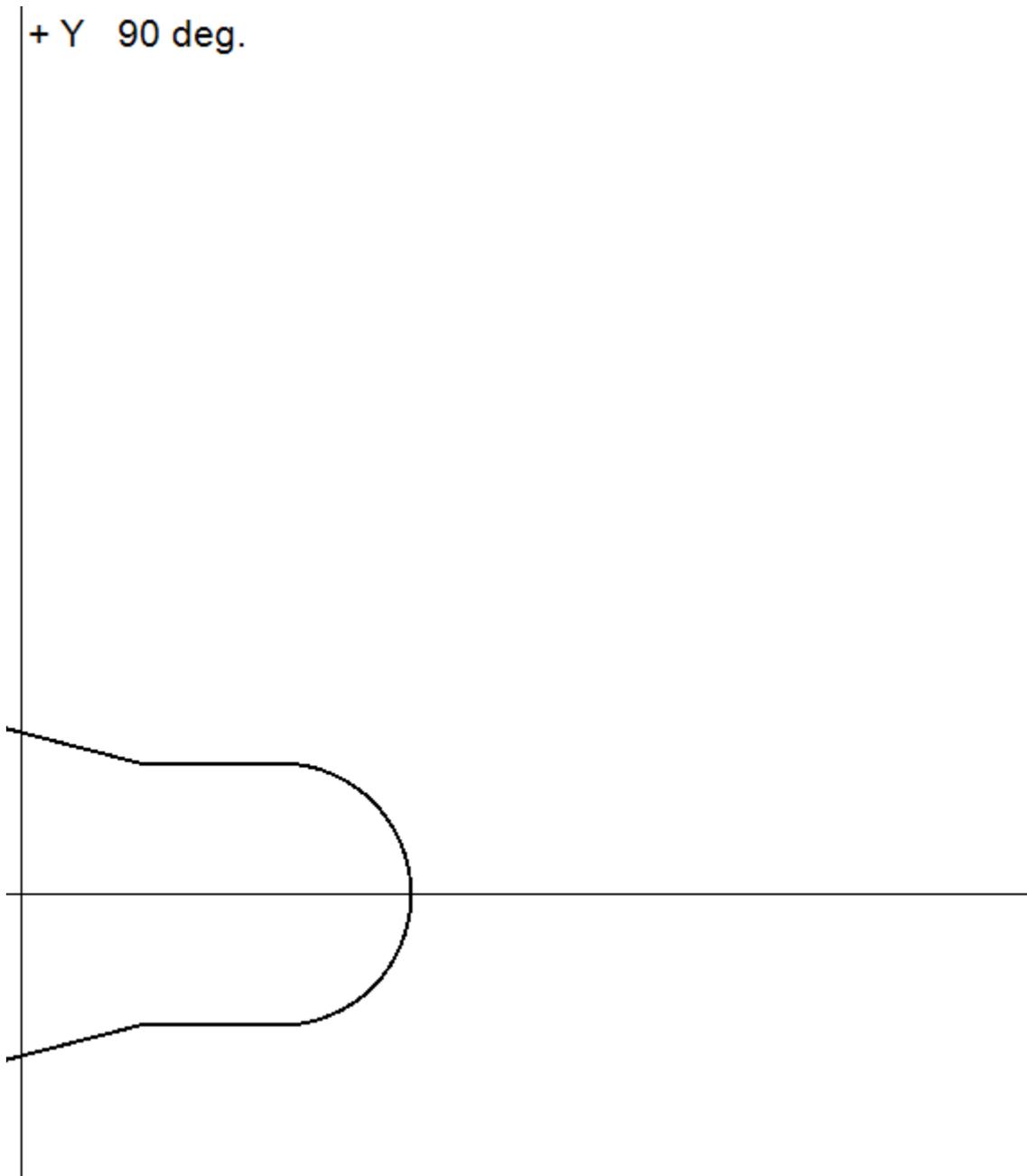
<http://www.cnccodeshooter.com/upload/CodeShooterHelp.pdf>

This Version will not open sessions earlier versions properly.

A few basic precautions that you need to know when using and running the program.

Always use good machining practices. Good visual care is required to make sure that you are creating the correct tool path for your program. Good visual care should be exercised when inputting tool movements to make sure that all clamps and any other fixtures clear the tool path. Always remember to bring your tools to clearance before continuing on to the next cut, this is not an automatic program function so you need to make sure you input the correct numbers as you build your programs.

The Draw Tab



Draw	Code	Program
start X	-0.25	
Start Y	-2.371	
End X	-.25	
End Y	-2.371	
FeedRate	2	
Left or Right side of t		
<input type="button" value="Redraw"/>		
Redraw patterns		
<input type="button" value="Reset All"/>		
Reset all dimensions an		
<input type="button" value="Start Edit"/>		
<input type="button" value="Save Edit"/>		
<pre> 1 Start Pattern # 1 2 line startX=-0.25 s 3 line startX=1.6875 5 Clock startX=3.687 <end=180.0 6 Line startX=3.687 7 Line startX=1.687 8End Pattern # </pre>		

The main difference with auto and instant codeshooter is pro will compute tool path for you on profiles. The draw tab is where you will draw your part. Begin by inputting your pattern description and pressing the start pattern button. Then you input either a line, angle or circular. Keep inputting dimensions until the pattern is complete.

Mouse operation: Right click and drag across the drawing area to rotate the axis, Press the scroll button then scroll to zoom in or out, left click on a point or circle to highlight a point or circle.

Auto uses patterns (multiple lines and arcs in a continuous line) and points and circles to help you cut parts. Points and circles are features and not part of patterns.

The Draw tab can be used for creating dimensional patterns. You must input the lines or arcs in the order you want to machine them. The best way to input radii is to draw your lines then click on add radius click on two connecting lines when they are highlighted input the radius value then the add radius button. The advantages to doing it this way are you don't need to know the x y center points and to change the radius just click on the radius and input the new value then press change radius.

As long as the path is continuous it should generate a proper tool path however if say the fifth line intersects the first it will not take in to account each other. The angles have to be continuous. It only programs in the order they are drawn. The end of one line or radius must be the beginning of the next line or radius. Sometimes if you get a NAN it is because even though the dimension lines intersect the tool path may not. The program will automatically place the end of the current dimension into the start X and Y for the next dimension. The draw tab makes the patterns. You can rotate the axis on the drawing screen by right clicking and dragging across the drawing portion of the screen.

To actually generate a tool path: go to the Programming -> Cut Patterns tab. You must have a tool selected with a tool diameter for the program to generate a tool path. Choose the pattern you wish to run. Choose whether to run just the pattern or you can choose multiple cuts.

Plus or minus is which side of the drawing the tool will start on, You can right click on the plus or minus circles and it will check the appropriate checkbox. On a horizontal line it will start to the Y plus on a Vertical line it will start on the X plus. If the first dimension is a circle minus is to the inside of the circle plus is to the outside of the circle.

Multiple cuts is for when you want to take the pattern in more than one cut. It will move down in Z run the pattern rapid up rapid to the X and Y position move down rerun the pattern until the desired depth is reached.

Also for more than one tool you can change tools then go back to the dimensional tab and rerun the pattern. You can

reset the G code and remake the G code for different tool sizes or to cut on the opposite side.

You can edit the dimensional pattern on the draw tab by selecting which line to edit in the draw tab drop down box then press start edit. The reason for editing the patterns here is to be able to redraw without changing tabs. After making your edits press save edits then redraw. You can also insert a line or delete a line. Select the line to edit before pressing insert, delete or start edit. Remember the end of one line must be the beginning of the next. All other edits are made from the Edit tab.

As always if you find bugs please report them at: support@cncodeshooter.com.

The draw function gives CNC CodeShooter a powerful new tool. First give your pattern a description then click -> start pattern. To generate the proper tool path select whether you want to start on the plus or minus of the dimension choose the feed rate. Plus will start to the left side of the first angle or line being greater than or equal to 45 degrees and less than or equal to 270 degrees. Minus will start on the right side of angles less than 45 degrees and greater than 270. Draw your pattern which can be a combination of lines angles and arcs. Whether you want to draw a line which can also be an angled line an angle or an arc. After filling out the relevant text boxes press the draw button the information will be set and the dimension drawn to the screen. You must draw it in the same order you want to cut it and the first input point is the start point. If you are going to go from a straight line to a radius the tool paths must intersect or you will get NAN (not a number).

When you are finished be sure to press -> end pattern.

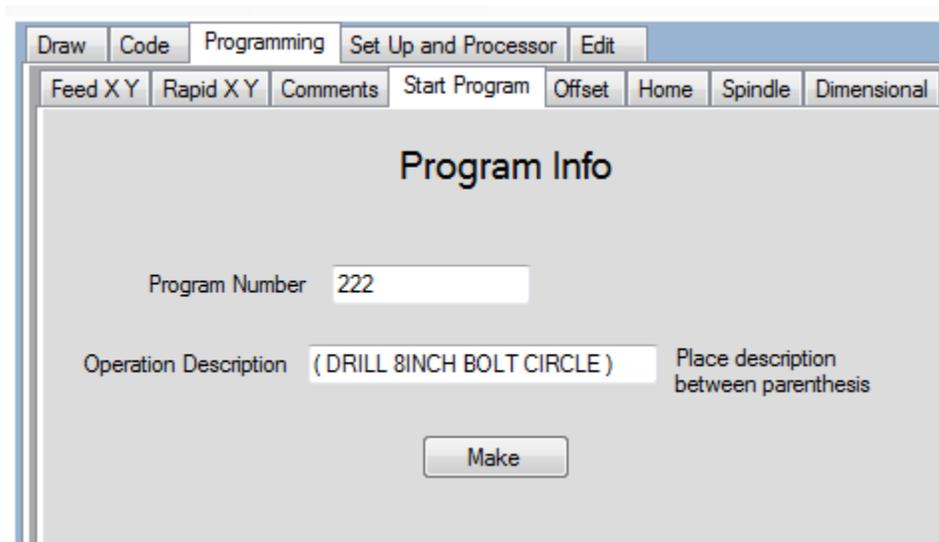
If you make a mistake on the plus or minus or tool diameter you can press reset G- code make changes and press make code button to generate new code. To make an edit select which line number you want to edit and press the start edit button after making changes press the save edit button. After saving changes press redraw to see the modified figure. Remember the end of one dimension must be the start of the next.

Cut circles is for circular interpolating circles. You can choose draw a circle from a point -> highlight a point ; the X and Z will input into the draw circle boxes then input the diameter and press Do It. You can also input the X Y Z and Diameter manually and press enter. once you have a circle drawn you can highlight it by right clicking on it. The X Y and diameter will be input to

the appropriate boxes under the programming-> Cut circle tab.
After you input the rest of the information press Do it.

You can plot points using the Misc -> Bolt circle Tab or on the draw tab: the draw points button (you cannot draw points while you are drawing a pattern).

Start a program



There are different ways to write a program. You can write the whole thing from scratch or import frequently used tool lists or you may have to rewrite an existing program to fix a problem with it.

To save time frequently used tools can be saved to a tool list then the session is saved say as toollist1 or something.

To Open or Save a session, Look on the Start tab for Load Session and Save Session. This will save the contents of the text boxes you have filled out.

To start a program either open the tool list you need or input the tools you will use on the set up tab. Then choose a program number and input it under the Program tab. The program number can be input at anytime it doesn't have to be first but before you make the program you have to input a program number. Program comments are optional. If you are going to mill something and have the tool path figured for you use the draw tab to draw or edit the drawing.

Before you run the first tool you can send the tool information from the setup tab to the tool tab by pressing the send button next to the tool you are going to use on the set up page. Make sure the machine is in a position where the tool changer will clear everything before it changes tools. After each operation you can send each axis home before you change to the next tool. You can change the home position for a machine in the processor tab. You can have multiple processor files for each machine with different home positions as some fixtures might require a special home position.

After all machining programming is done you can view the tool path under the code tab. But before you can make code you have to choose a machine processor file. The mill path line is the center of the tool path so a large diameter tool will have a tool path that is farther from the dimensional lines than a narrow tool. Bolt pattern holes will show the relative hole size if you have input the tool diameter +(TOOL DIAM = 2.00) for example. Otherwise mill path will show a one inch tool or the last read tool. Future versions will show the tool width for milling operations but for now show the tool center line.

If you don't have time to finish a program you can save your work or if you need to make changes you can reload the session. The Tape file (.t file) is saved separately under File->save or Save as or you can open a saved one under File->Open.

To start a new program click on the Program tab. Enter the program number the program will add the Letter O. The next text box is for program description and Set up information. Comments must be enclosed by parenthesis. If you choose a processor file with a yes on \$ and percent those will be added. If not or you choose one that is no they will not be added. This depends on the control you are writing the program for. Click make code to preview the code if it looks right press send to send the code to the program text box. To load the text

boxes with data you can choose load session. This will place data that you have saved in the data input text boxes. If you have to save data press the save session button and data in text boxes will be saved. To save a program go file-> save and save as a .T(tape) file. Save and load session always opens the my document folder it is a good idea to make a Lathe session and mill session folder and lathe programs and mill program folder in your my documents. Or you can make a folder for each program in lathe and mill programs and save your session and tape programs in the same folder. This is helpful if you will have multiple session files for each tape file.

Surface Speed

Surface Footage

Formula

Tool RPM = (surface foot cutting speed * 12) / Tool diameter * PI

Surface Speed = (RPM * Tool Diameter * PI) / 12

1 inch drill at 100 surface feet

$(100 * 12) / (1 * 3.14159) = 382$

Surface Speeds

These are approximate start on the slow side and work up if necessary.

Material	Hardness	Carbide	High Speed Steel
Soft Steel	100	500 - 900	110 - 200
Medium Steel	175 to 220	350	70 - 110
Hard Steel	220 to 300	250	30 - 80
	300 to 425	80 - 130	20 - 50

Bolt Circle

W Z Feed	W Z Rapid	Circle	Bolt O	Drill Tap	G	M	Engrave	#s	Pattern
----------	-----------	--------	--------	-----------	---	---	---------	----	---------

Equal spaced Bolt pattern

Start angle position 0 is
X large moving in a
Counterclockwise direction.
Ylarge is 90
Xsmall is 180
Ysmall is 270.
End position of 360
for a full circle.

Number of Holes

Start Angle

Bolt Circle Diameter

Feed Rate

Rapid to position

Peck Amount

Drill to in Z

X Center

Y Center

G81 or G83

G73 Chip Break

Drill-G81

Peck Drill-G83

Tap G84 (RH)

The bolt Circle tab can be used for equally spaced or non equal bolt circles.

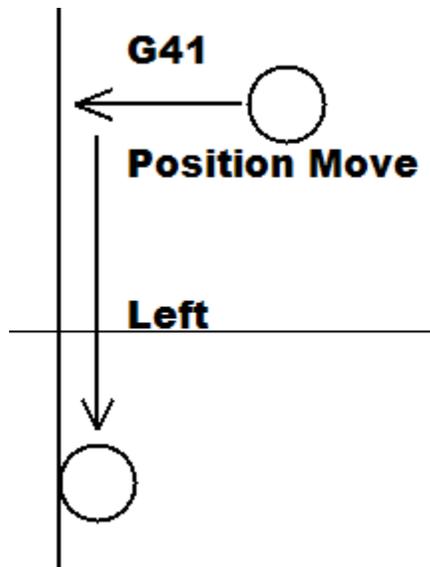
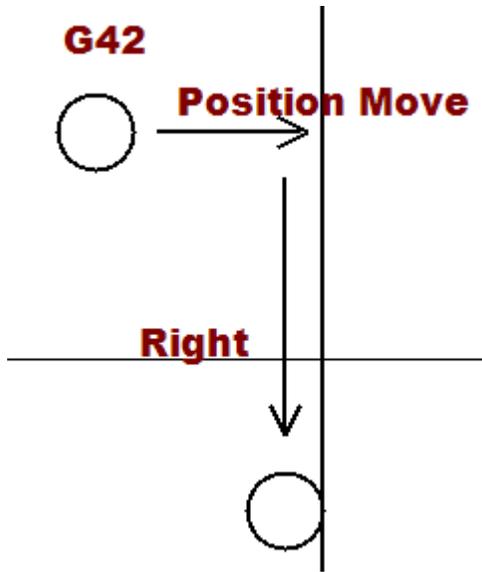
The start angle Zero position is at X plus moving counterclockwise. If the start angle is 45 degrees the hole will be placed in the plus X and plus Y quadrant.

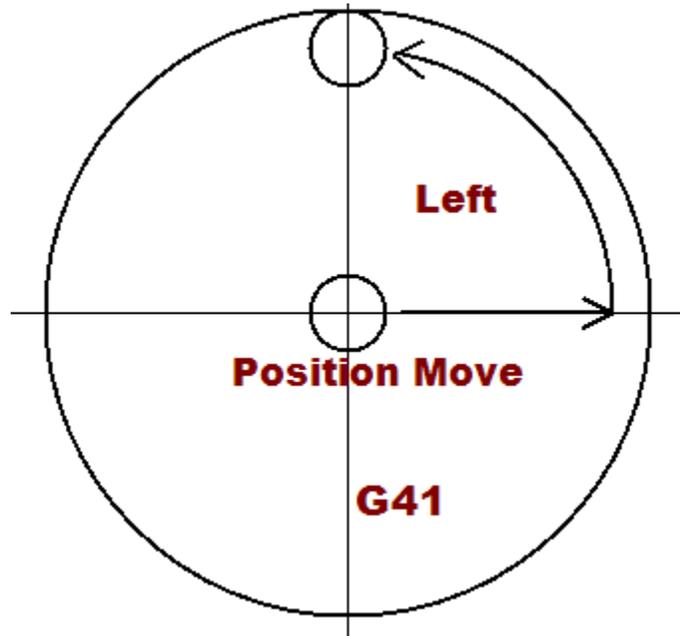
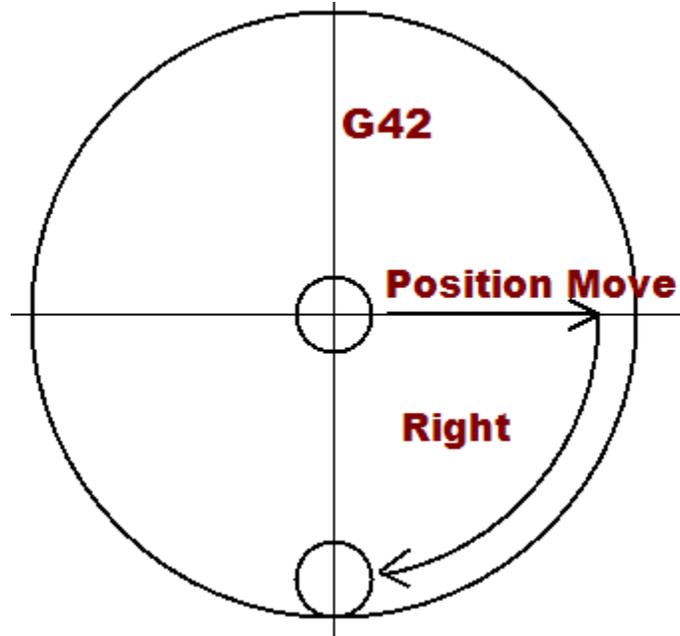
Cutter Compensation

Cutter compensation is very useful for precision milling. You can program a tool path without it but if it doesn't cut to the required size you would have to reprogram.

Compensation allows you to compensate for different size tools or tool pressure. Using cutter compensation will result in a tool path equivalent to the part dimensions. G41 is for left cutter comp G42 is for right cutter comp. G40 is for no cutter comp. The offset values are stored as either D or H offset values. If the control uses H it has to be a different number than the length offset number. It is best to use a one axis move to bring the cutter into position never use an arc move to position the tool. The move right after the position move has to be coordinated with the cutter comp (left for G41 right for G42). While the comp is on all blocks should contain an axis move(No blocks with just an M08 for example).

***If you use the wrong comp (G41 or G42) the tool will position to the wrong side of the tool. It is a good idea to offset the tool up before running the cut to make sure there are no errors.





Circular Interpolation

W Z Feed W Z Rapid Circle Bolt O Drill Tap G M Engrave #s

Circular Interpolate

Start Angle counterclockwise from X Large and Y Center

Degrees of rotation

X Center of Hole

Y Center of Hole

Tool Diameter

Part Radius

Feed Rate (IPM)

Start angle position 0 is Xlarge to counterclockwise. Ylarge is 90 Xsmall is 180 Ysmall is 270. End position of 360 for a full circle.

Direction to Interpolate

Clockwise

Counter Clockwise

Cut inside Circle

Cut outside Circle

To circular interpolate a part you have a choice between Input angles, X large X small, Y large, Y small on start position and finish position so you can cut a full circle use the same start and finish position. Or in 90 degree increments by choosing different start and finish point also you can choose input angle and interpolate various degrees. After choosing direction clockwise and counter clockwise and inside or outside of the circle diameter input the text boxes X center and Y center Tool diameter Part Radius and feed press preview. If the code looks good press send to send the code to the program text box.

*Hole pattern
Drill and Tap
Single Hole Pattern*

W Z Feed	W Z Rapid	Circle	Bolt O	Drill Tap	G	M	Engrave	#s
----------	-----------	--------	--------	-----------	---	---	---------	----

Drill/Tap

Start X position	<input type="text" value="10"/>	G81 or G83 <input type="radio"/> Drill G81 <input checked="" type="radio"/> Peck drill G83 <input type="radio"/> Tap - G84 <input type="radio"/> G73
Start Y position	<input type="text" value="2"/>	
Z depth to drill to	<input type="text" value="-1.5"/>	
Feed Rate	<input type="text" value="2"/>	
Peck Amount	<input type="text" value=".2"/>	<input checked="" type="radio"/> Single Hole <input type="radio"/> Hole Pattern
Rapid to	<input type="text" value="1"/>	
X Increment amount	<input type="text" value="1"/>	Input X increment or Y increment or both the rest of the fields are mandatory.
Y Increment amount	<input type="text" value="1"/>	

Multiple Hole Pattern

Drill and tap can be used for a single hole as in the upper picture or a pattern as in the lower picture.

Engraving

W Z Feed | W Z Rapid | Circle | Bolt O | Drill Tap | G | M | Engrave | #s

Engrave

Z Rapid

X Start

Y start

Z depth

Feedrate

Rotate degrees

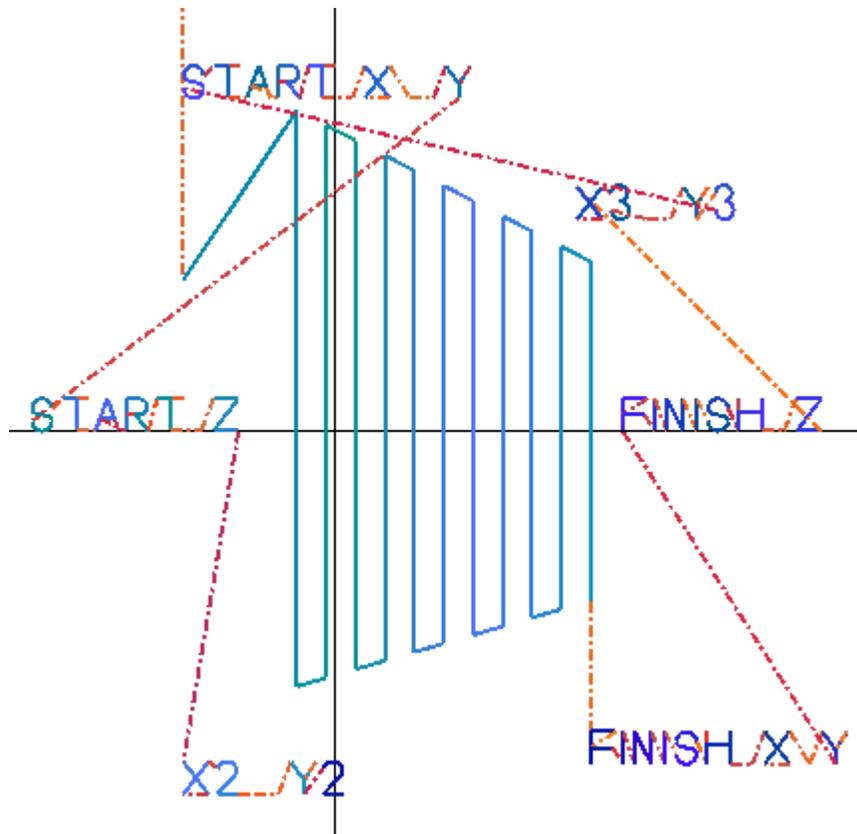
Size

Text

This is a New version.
Be careful and please report
problems to:
Support@CNCCodeShooter.com

To engrave input the text in the bottom text box input Z rapid to feed rate etc. then press preview . After code is generated press send to send the code to the program text box.

Ramp



Ramp is used for milling a back and forth pattern from Start X,Y to X2,Y2 moving in X, Y and Z until Finish X,Y and Z is reached. If no movement in Z is required it will program a flat pattern. The number of cuts must be odd. This is very useful for milling ramps with a ball endmill, where you may increment 2 to 5 thousandths at a time, or milling a back and forth pattern with a regular endmill.

Pattern Milling

W Z Feed	W Z Rapid	Circle	Bolt O	Drill Tap	G	M	Engrave	#s
----------	-----------	--------	--------	-----------	---	---	---------	----

Pattern Milling

Insert pattern here
Line numbers will be added when G code is generated.

Start Z

Finish Z

Z per cut

Feedrate for Z down

```
G00 X0 Y0  
G01 X5 F4.  
Y2.25  
X0
```

This pattern only works in negative Z direction.
(I.E) Finish Z must be smaller than start Z
Z per cut has to be negative also.
Future versions will work in either direction.

Pattern milling is used extensively in CNC Milling. A pattern is milled then the spindle drops down and the pattern is recut until the desired depth is reached. Input the pattern in the pattern text box then select start Z position, depth of cut, Finish Z and Feed rate for the spindle down move. Be sure the spindle is in a position where it can feed down at the end of the pattern.(the last line in the text box) When you press preview the code on the right is produced. In this case 87 lines of code. Press send and the code is sent to the main program editor.

Processor files

Start	Offset	RapWZ	Tool	Home	End	M's	Spindle	4th	Custom	Engrave	Set
Machine File											
Workshift	G54		Baud Rate	4800						Set	
X Home Position	G91 G28 X0.		Parity	EVEN						▼	
Y Home position	G91 G28 Y0.		Data bits	7						After choosing processor press set processor button	
W Home position	G91 G28 W0.		Stop bits	1						Save files to root directory of CNCCodeShooter	
Z home position	G91 G28 Z0.		Flow control	None						Save New File	
length offset code	G43		E or F	F		\$ and %				Use Open button if File is not in Root directory	
G98	Yes		End code	M30		Yes				Open machine file	
Has W Axis	Yes										

Sample processor file. The mill files have a psr extension and can be opened and saved with the CNCCodeShooter. These files are located in the root directory of CNCCodeShooter. Choose the file you want in the drop down box then press set. The drop down box reads file from the root directory of CNC Code Shooter. If you need files from another location press open machine file button. To save files enter the proper information in the text boxes and press the Save or make new file button. Save file in the root of Codeshooter for the drop box to read them.

```
baudrate4800
parityEVEN
stopbits1
databits7
flow controlNone
workshiftG54
EndcodeM30
XHomepositionG91 G28 X0.
hasG98ynYes
ZHomepositionG91 G28 Z0.
EorF
HasWNo
YHomeG91 G28 Y0.
wHomeG91 G28 W0.
```

The first five lines are the communication settings.
The first number is the baud rate
the second is parity,
third is data bits,
fourth is stop bits and
the fifth is flow control.

The sixth is work shift depending on the machine. Example work shift would be G54 for most controls.

G55 is optional work shift. G92 is preset used on older machines. (ie. if you don't call out G54 the machine defaults to G50). (Okumas use G15P1 or G15P?) Check your control Manuel for proper settings.

The seventh is end code either M30 or M02.

The eight X Home Position on most machines use G28 or G30 then the axis and 0. Okumas use G30P1 or G30P2 the last number can change so that the machine can have more than one home position.

The ninth is drill code some machines require a G98 for return level. (For Okumas leave this empty). If the drill code requires G98 input Yes if it is null (ie it doesn't use G98) make it No.

The tenth is Z Home Position on most machines use G28 or G30 then the axis and 0. Okumas use G30P1 or G30P2 the last number can change so that the machine can have more than one home position.

The eleventh is E or F for feed code.

The twelfth is has W if the machine has a W axis.

The thirteenth is Y Home Position on most machines use G28 or G30 then the axis and

0. Okumas use G30P1 or G30P2 the last number can change so that the machine can have more than one home position.

The fourteenth is W Home Position on most machines use G28 or G30 then the axis and 0. Okumas use G30P1 or G30P2 the last number can change so that the machine can have more than one home position.

The fifteenth is offset length code usually G43 Okuma controls use G56

Check your machine manual for proper settings.

Send and Receive Programs Using the send tab

The screenshot shows the 'CNC Code Shooter Send and Receive' application window. The title bar indicates the file path: C:\Users\Randall\Documents\MillPrograms\pragma111.t. The interface is split into two main sections.

Left Panel (Code List): A list of G-code lines from O111 to N47. The code includes comments like '(RIGHT SIDE WHEEL BLOCK)', '(MILL POCKET AND PROFILE)', '(TOOL DIAM =1.25)', and '(ROUGH OUT POCKET)'. The code lines are: O111 (RIGHT SIDE WHEEL BLOCK), O111 (MILL POCKET AND PROFILE), N1 G00 G17 G40 G90, N2 G54, (T01 ROUGHER MILL), (ROUGH OUT POCKET), (TOOL DIAM =1.25), N3 G28 G40 G00 G91 G17 G80 Z0 G54, N4 G00 G90, N5 G00 M06 T1, N6 M03 S400, N7 G00 G43 Z10.0 H01, N8 G00 X-1., N9 G00 Y.5, N10 G00 Z0.1, N11 G01 Z-0.15 F0.02, N12 G01 X4.3 F0.01, N13 G01 X4.3 Y0.5 F0.01, N14 G02 X4.3 Y-0.5 I0.0 J-0.5 F0.01, N15 G01 X-1.0 F0.01, N16 G00 Y1.0, N17 G01 X4.3 Y1.0 F0.01, N18 G02 X4.3 Y-1.0 I0.0 J-1.0 F0.01, N19 G01 X-1.0 F0.01, N20 G01 Z-0.3 F0.02, N21 G00 Y.5, N22 G01 X4.3 F0.01, N23 G01 X4.3 Y0.5 F0.01, N24 G02 X4.3 Y-0.5 I0.0 J-0.5 F0.01, N25 G01 X-1.0 F0.01, N26 G00 Y1.0, N27 G01 X4.3 Y1.0 F0.01, N28 G02 X4.3 Y-1.0 I0.0 J-1.0 F0.01, N29 G01 X-1.0 F0.01, N30 G01 Z-0.45 F0.02, N31 G00 Y.5, N32 G01 X4.3 F0.01, N33 G01 X4.3 Y0.5 F0.01, N34 G02 X4.3 Y-0.5 I0.0 J-0.5 F0.01, N35 G01 X-1.0 F0.01, N36 G00 Y1.0, N37 G01 X4.3 Y1.0 F0.01, N38 G02 X4.3 Y-1.0 I0.0 J-1.0 F0.01, N39 G01 X-1.0 F0.01, N40 G01 Z-0.6 F0.02, N41 G00 Y.5, N42 G01 X4.3 F0.01, N43 G01 X4.3 Y0.5 F0.01, N44 G02 X4.3 Y-0.5 I0.0 J-0.5 F0.01, N45 G01 X-1.0 F0.01, N46 G00 Y1.0, N47 G01 X4.3 Y1.0 F0.01.

Right Panel (Settings): A configuration area for communications. It includes a 'Use Processor' button and a 'Choose a configuration file' dropdown set to 'Quantum.psr'. Below this are dropdown menus for Baud Rate, Data Bits, Parity, and Stop Bits. A 'Flow Control' section has radio buttons for 'XOnXoff', 'RTS', and 'No Handshake' (which is selected). Other settings include ReadTimeout = 9500, BaudRate = 4800, Parity = Even, DataBits = 7, Stop Bits = One, and Hand Shake = None. A 'COM1 is open' status is shown. At the bottom, there is a 'Send Status' label and a text input field.

For Communication settings.

You can use a processor file or set the settings manually. If you choose a processor file press use Processor after selecting processor file from the drop down box. After setting the communication settings manually or choosing the processor files choose the com port(com port 1, 2, 3 etc) and press Save Settings. If the settings are not set right you have to close the port by pressing the Close Port button before resetting them.

Most Controllers require a \$ (dollar sign) before the program number and a % (percent sign) before the first block and a % (percent sign) after the M30 or last block. Refer to the machine manual to verify each controllers settings on the particular machine that you are operating.

To set manually

First you will select the com port, parity, data bits, baud rate, flow control and stop bits.

Then press save settings button. It should say that the com port is open.

Most new computers do not have a RS 232 connection so a USB to RS 232 adapter has to be used. These can be found at Fry's, Most new computers do not have a RS 232 connection so a USB to RS 232 adapter has to be used. These can be found at Fry's Radio Shack and other places.

Radio Shack and other places.

Software Handshake Cable - How to make a cable with a 9 - pin connection on one side to a 25 - pin connection on the other side. This is a typical machine to computer connection cable. We sell the cables already made ranging from 20 feet - 50 feet, shielded or regular.

1. Cross signals SD (serial transmit data) and RD (serial receive data) with each other.
2. Connect signal SG(signal ground) to SG (signal ground).
3. Jump together signals RTS (request to send) and CTS (clear to send) at both ends of the cable.
4. Jump together signals DTR (data terminal ready), DSR (data set ready), and DCD (data carrier detect) at both ends of the cable.

Below is a snapshot of a program created by CNC Code Shooter Mill. You will be able to create a program from beginning to the end with CNCCodeShooter.

Begin your program with the start tab, setting your program number, the tool description, and the work shift. Next you will click the button Make Code and preview it in the preview box. If the code is good then click the send button to send it over to the text editor to start creating your program.

Renumber your program to your own sequence of numbers for your own program or use the default setting.

This is just an example . Every machine will have different configurations and will be slightly different depending on specific settings that are required by different machines. Refer to the machine manual for the specific settings required by individual machines.(example: the baud rate, or the parity, etc.)

```
$
:00001 Flange Adapter (Program Description)
(More Description)
%
#4 CENTER DRILL (Tool Description)
N1 G90 G80 G40 G00 G17 (Cancel Codes)
N2 S1000 M03 (spindle 1000rpm / forward)
G15P1 (Work shift)
N3 G00 X2.1213 Y2.1213 (1st position move)
N4 G56 Z1.0 H1 M08 (work shift)
N5 G81 Z-.28 R.1 F2.0 M54 (drill Cycle)
N6 Z-2.1213
N7 Y-2.1213
N8 X2.1213
N9 X3.2476 Y1.875
N10 X0 Y-3.75
N11 X-3.2476 Y1.875
N12 Y-1.875
N13 X0 Y-3.75
N14 X3.2476 Y-1.875
N15 G80 G00 Z1.0
N16 G91 G30P1 Z0 M09
N17 G90 M01
N18 T2 M06
(17/32 DIA. DRILL)
N28 G90 G80 G40 G00 G17
N29 S539 M03
N30 G00 X2.1213 Y2.1213
```

```
N31 G56 Z1.0 H2 M08
N32 G83 Z-.9596 R.1 Q.1 F4.3
N33 X-2.1213
N34 Y-2.1213
N35 X2.1213
N36 G80 G00 Z1.0
N37 G91 G30P1 Z0 M09
N38 G90 M01
N39 T3 M06
(27/64 DIA. DRILL)
N49 G90 G80 G40 G00 G17
N50 S679 M03
N51 G00 X3.2476 Y1.875
N52 G56 Z1.0 H3 M08
N53 G83 Z-.9268 R.1 Q.1 F5.0
N54 X0 Y3.75
N55 X-3.2476 Y1.875
N56 Y-1.875
N57 X3.2476 Y-1.875
N58 G80 G00 Z1.0
N59 G91 G30P1 Z0 M09
N60 G90 M01
N61 T1 M06
N62 M30
%
```

A word about rapid travel.

Rapid travel does not always rapid point to point.

Most Fanuc machines rapid at a forty five degree angle until one axis reaches it's destination then the other axis travels in a straight line until it reaches it's destination.

As always be careful running a CNC machine.

CNC CODE SHOOTER LATHE

<http://www.cnccodeshooter.com>

<http://www.cnccodeshooter.com/upload/CodeShooterHelp.pdf>

Basics

The basics for writing a program is to give it a number and description.

Draw the pattern.

Fill out tool information.

Choose a pattern to cut and parameters.

Make G code.

For Lathe

You must draw patterns as you will cut them. For Lathe if you are going to cut from the face to the chuck it has to be drawn that way.

For Mill

You must draw patterns as you will cut them. If you are going to cut counterclockwise around a pattern it has to be drawn that way.

A few basic precautions that you need to know when using and running the program.

Always use good machining practices. Good visual care is required to make sure that you are creating the correct tool path for your program. Good visual care should be exercised when inputting tool movements to make sure that all clamps and any other fixtures clear the tool path. Always remember to bring your tools to clearance before continuing on to the next cut, this is not an automatic program function so you need to make sure you input the correct numbers as you build your programs.

Tool simulation is new as of version 3.8.

The program will show some tools : To show tools, input at the tool description: (LH TNMG M04),(LH TNMG),(RH CNMG),(RH CNMG M04),(RH DNMG),(RH DNMG M04),(RH TNMG),(RH TNMG M04)(RH VNMG),(RH VNMG M04),(OD THREADING TOOL),(ID THREADING TOOL),(FACE GROOVE TOOL),(OD GROOVE TOOL),(OD GROOVE TOOL),(BACK FACE GROOVE TOOL) or (LH BAR TNMG) these are the only tools accepted they must be in Caps. M4 or M04 will show the reverse of the tool as if it were upside down.

Processor files

Machine File

Workshift Baud Rate

X Home Position Parity

Z home position Data bits After choosing processor
press set processor button

1 or 2 line G71 and G72 Stop bits

End code Flow control

E or F Save files to root directory of CNCCodeShooter Use Open button if
File is not in Root directory

I or R

Sample processor file. The lathe files have a lsr extension and can be opened and saved with the CNCCodeShooter. These files are located in the root directory of CNCCodeShooter. Choose the file you want in the drop down box then press set. The drop down box reads file from the root directory of CNC Code Shooter. If you need files from another location press open machine file button. To save files enter the proper information in the text boxes and press the Save or make new file button. Save file in the root of Codeshooter for the drop box to read them.

```
baudrate4800
parityEVEN
stopbits1
databits7
flow controlNone
workshiftG54
EndcodeM30
XHomepositionX15.
oneortwo2
ZHomepositionZ10.
EorF
IorR
```

The first five lines are the communication settings.

The first number is the baud rate

the second is parity,

third is data bits,

fourth is stop bits and

the fifth is flow control.

The sixth is work shift depending on the machine. Example work shift would be G54 for most controls.

G55 is optional work shift. G92 is preset used on older machines. (ie. if you don't call out G54 the machine defaults to G50). (Okumas use G15P1 or G15P?) Check your control Manual for proper settings. The seventh is end code either M30 or M02. The eighth X Home Position on most machines use G28 or G30 then the axis and 0. Okumas use G30P1 or G30P2 the last number can change so that the machine can have more than one home position. The ninth is whether the machine uses one line or two line G71 and G72 code. The tenth is Z Home Position on most machines use G28 or G30 then the axis and 0. Okumas use G30P1 or G30P2 the last number can change so that the machine can have more than one home position. The eleventh is E or F for feed code. The twelfth is I or R for taper moves most machine use I. Check your machine information Check your machine manual for proper settings.

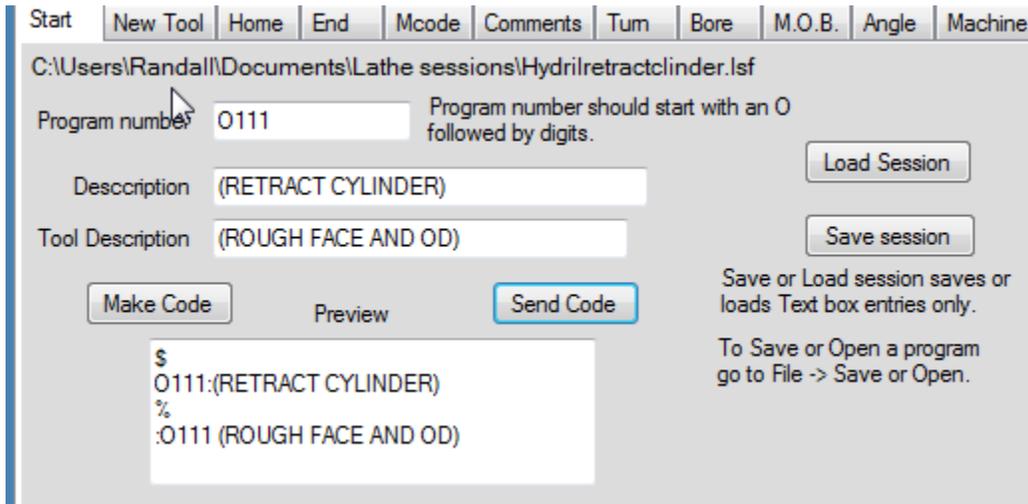
Data Input

Data input boxes usually require a numbered input. Letters such as Z or X will be added for you. Tool number and offset require an integer while position moves can be in decimal format. If the program prompts you, "You must enter a value" you have probably left a value out or it is in the incorrect format. Z10.5 would be wrong because the Z will be added by the program.

A common mistake is two decimal points the program will also prompt you, "You must enter a value," as this is in the incorrect format. On position moves if it is to an even number say 10 inches if you input 10 the program will output 10.0 as this is the format most machines require.

Check your machine manual for proper settings.

Start a New Program



Start | New Tool | Home | End | Mcode | Comments | Turn | Bore | M.O.B. | Angle | Machine

C:\Users\Randall\Documents\Lathe sessions\Hydrilretractcylinder.lsf

Program number: 0111 Program number should start with an O followed by digits.

Description: (RETRACT CYLINDER)

Tool Description: (ROUGH FACE AND OD)

Buttons: Load Session, Save session, Make Code, Preview, Send Code

Save or Load session saves or loads Text box entries only.
To Save or Open a program go to File -> Save or Open.

```
$
O111:(RETRACT CYLINDER)
%
:O111 (ROUGH FACE AND OD)
```

To start a new program click on the start tab. Enter the program number preceded by the Letter O not a zero. The next Text two text boxes are for program description and Set up information. Comments must be enclosed by parenthesis. If you choose a processor file with a yes on \$ and percent those will be added. If not or you choose one that is no they will not be added. This depends on the control you are writing the program for. Click make code to preview the code if it looks right press send to send the code to the program text box. To load the text boxes with data you can choose load session. This will place data that you have saved in the data input text boxes. If you have to save data press the save session button and data in text boxes will be saved. To save a program go file-> save and save as a .T(tape) file. Save and load session always opens the my document folder it is a good idea to make a Lathe session and mill session folder and lathe programs and mill program folder in your my documents. Or you can make a folder for each program in lathe and mill programs and save your session and tape programs in the same folder. This is helpful if you will have multiple session files for each tape file.

Set up and New Tool

Rapid	Feed	Radius	Drill	G Code	#s	G76	G92	Setup	Face	Groove	Calc
Operation Description											
				Tool #	Tool Description			Machining Description			
				0101	<input type="button" value="Send"/>	CNMG432			ROUGH FACE AND		
Customer	Sample			0505	<input type="button" value="Send"/>	3" DRILL					
Tape #	O111			0707	<input type="button" value="Send"/>	2" BORING BAR CI			ROUGH BORE		
Part number				0303	<input type="button" value="Send"/>	CNMG 432			FINISF FACE AND (
Revision				0909	<input type="button" value="Send"/>	2" BAR CNMG432			FINISH BORE		
<input type="button" value="Now"/> Date					<input type="button" value="Send"/>						
Description	For Demo				<input type="button" value="Send"/>						
Machine	Hwa Cheon				<input type="button" value="Send"/>						
Operation #					<input type="button" value="Send"/>						
<input type="button" value="Send Page"/>					<input type="button" value="Send"/>						
Start	New Tool	Home	End	Mcode	Comments	Turn	Bore	M.O.B.	Angle	Machine	
Comments (T0505 3" DRILL)						<input type="radio"/> T0101 <input type="radio"/> T0505 <input type="radio"/> T0909 <input type="radio"/> T0202 <input type="radio"/> T0606 <input type="radio"/> T1010 <input type="radio"/> T0303 <input type="radio"/> T0707 <input type="radio"/> T1111 <input type="radio"/> T0404 <input type="radio"/> T0808 <input checked="" type="radio"/> existing					
More comments ()											
Stop Spindle	M05		<input type="button" value="Preview"/>								
Max spindle speed	G02 X9.2125 Z		Max RPM 150		<input type="button" value="Send Code"/>						
New Tool #	T0505										
Start Spindle	M03 G96	350	Surface Footage	<input type="radio"/> G97 <input checked="" type="radio"/> G96		Spindle <input checked="" type="radio"/> Forward <input type="radio"/> Reverse		Gear <input type="radio"/> M38 <input type="radio"/> M39 <input type="radio"/> M40 <input type="radio"/> M41 <input type="radio"/> M42			
M03 is forward M04 is Reverse G96 is constant surface speed S is Surface feet in G96 S is RPM in G97 G97 is fixed spindle speed Use G97 for threading											

It is best to save tool lists with save session then you can open a session and have the tool list ready with less typing. Once the tool list is ready by pressing send on the button next to the selected tool it will send the information to the new tool tab. To find spindle speed go to the #s tab their is a chart for surface speeds and an RPM calculator. G96 is constant surface speed ie the spindle will speed up as the diameter gets smaller. G97 is constant RPM use G97 for threading. G50 S? sets max RPM

this should be set according to the work size. A heavy part or long part would bet set slower than a small, short part

Rapid Feed Radius Drill G Code #s G76 G92 Setup Face Groove Calc

Set Start Box in one hundred increments

1 100 200 300 400 500 600 700 800 900

First Block will be 10 unless changed here Renumber will start with the number in this box.

Start Number 1 Renumber Remove Line Numbers

Increment Amount 1 Add Block Numbers

Set Start=1 Increment=1 Remove Spaces

Set Start=10 Increment=5 Set Start=10 Increment=10 Remove all spaces in program.

Approximate Surface Speeds

Material	Hardness	Carbide	HSS
soft steel	100	500 - 900	110 - 200
medium steel	180 - 220	350	70 - 110
Hard steel	220 - 300	250	30 - 80
	300 - 400	80 - 130	20 - 50

Font and Font Color Change text in text box

Tab Box Color Change group box color.

Find RPM Part Diameter Surface Footage RPM

Surface Footage 4 250 Calculate RPM 238.7

Surface Footage

Formula

Part RPM = (surface foot cutting speed * 12) / Part diameter * PI

Surface Speed = (RPM * Part Diameter * PI) / 12

1 inch drill at 100 surface feet

$(100 * 12) / (1 * 3.14159) = 382$

Surface Speeds

These are approximate start on the slow side and work up if necessary.

Material	Hardness	Carbide	High Speed Steel
Soft Steel	100	500 - 900	110 - 200
Medium Steel	175 to 220	350	70 - 110
Hard Steel	220 to 300	250	30 - 80
	300 to 425	80 - 130	20 - 50

Rapid X and Z

Rapid	Feed	Radius	Drill	G Code	#s	G76	G92	Setup	Face	Groove	Calc	
Rapid X and Z								X4.	X16.			
Be careful when rapid traveling in two axis. Most machines do not rapid point to point. It will rapid at a forty five degree angle untill one axis reaches it's destination,then the other axis will continue in a straight line untill it reaches it's destination.								X 6.	X18.			
Input X, Z or both Axis.								X 8.	X20.			
X position to Rapid to	<input type="text"/>	X Rapid									X10.	X 24.
	<input type="text"/>	X and Z									X 12.	X 30.
Z Position to Rapid to	<input type="text"/>	Z Rapid									X14.	X 40.
Code preview	<input type="text"/>	Coolant On									Z0.	Z 1.0
	<input type="text"/>	Coolant Off									Z 0.1	Z2.
<input type="button" value="Send"/>									Z .25	Z 10.		
								Z .5	Z 20.			

The rapid tab is for bringing the axis into location. There are X and Z commonly used positions on the right. For specific numbers use the text boxes. When using the text boxes enter the position only not the axis letter. The program will add the axis letter. If the message box says you must enter a value the most likely problem is inserting two decimal points or something that does not read as a number (a letter or symbol) check the text boxes for a mistake and try again. The coolant on is most often used when bringing the tool up to the part usually when the machine rapids to 1 inch off the face turn coolant on. When the machine rapids an inch off the part and the operation is finished turn the coolant off.

The main difference with pro and instant codeshooter is pro will compute tool path for you on dimensions. The draw tab is where you will draw your part. Begin by inputting your pattern description and pressing the start pattern button. Then you input either a line, angle or circular. Keep inputting dimensions until the pattern is complete.

The best way to input radii is to draw your lines then click on add radius click on two connecting lines when they are highlighted input the radius value then the add radius button. The advantages to doing it this way are you don't need to know the x y center points and to change the radius just click on the radius and input the new value then press change radius.

The Draw tab can be used for generating tool paths from dimensional patterns. You must input the lines or arcs in the order you want to machine them. As long as the path is continuous it should generate a proper tool path. The angles have to be continuous. It only programs in the order they are drawn. Draw the cut as you want to cut it. A separate cut requires a separate pattern. The end of one line or radius must be the beginning of the next line or radius. The program will automatically place the end of the current dimension into the start X and Z for the next dimension. The draw tab make the patterns.

Choose whether it is Od or Id.

To actually run them go to the dimensional tab. You must have a tool selected with a tool radius for the program to generate a tool path. Choose the pattern you wish to run. Choose whether to run just the pattern or you can choose canned cycle.

Canned cycle is for when you want to take the pattern in more than one cut. On a canned cycle the start position should be larger than the biggest Od or smaller than the smallest Id. When the tool retract it will rapid to the start position.

Also for more than one tool you can change tools then go back to the dimensional tab and rerun the pattern.

You can edit the dimensional pattern on the draw tab by selecting which line to edit in the draw tab drop down box then press start edit. The reason for editing the patterns here is to be able to redraw without changing tabs. After making your edits press save edits then redraw. You can also insert a line or delete a line. Select the line to edit before pressing insert, delete or start edit. Remember the end of one line must be the beginning of the next. All other edits are made from the Edit

tab.

This is newly released version if you find bugs please report them at: support@cncodeshooter.com.

The draw function gives CNC CodeShooter a powerful new tool. First give your pattern a description then click -> start pattern. To generate the proper tool path select whether you want to start on the plus or minus of the dimension choose the feed rate. Draw your pattern which can be a combination of lines angles and arcs. Whether you want to draw a line which can also be an angled line an angle or an arc. After filling out the relevant text boxes press the draw button the information will be set and the dimension drawn to the screen. You must draw it in the same order you want to cut it and the first input point is the start point. If you are going to go from a straight line to a radius the tool paths must intersect or you will get NAN (not a number).

When you are finished be sure to press -> end pattern.

If you make a mistake on the plus or minus or tool diameter you can press reset G- code make changes and press make code button to generate new code. To make an edit select which line number you want to edit and press the start edit button after making changes press the save edit button. After saving changes press redraw to see the modified figure. Remember the end of one dimension must be the start of the next.

Linear Feed

The image shows a software interface for controlling a CNC machine's linear feed. At the top, there is a navigation bar with tabs: Rapid, Feed, Radius, Drill, G Code, #s, G/6, G92, Setup, Face, Groove, and C. The 'Feed' tab is selected. Below the navigation bar, the title 'Linear Feed' is centered. A message reads: 'You can also use the custom boxes in comments tab to do multiple lines or the turn or bore tabs. Input X, Z or both Axis.' The interface contains three input fields: 'X position to Feed to' on the left, 'Feed Rate' in the center, and 'Z Position to feed to' on the right. Below the 'X position' field is a button labeled 'X Feed only'. Below the 'Z position' field is a button labeled 'Z Feed only'. Below the 'Feed Rate' field is a button labeled 'Feed X and Z'. Below these fields is an 'Output' section with a text box and a button labeled 'Send Code to Editor'.

The feed tab is where the real work is done on CNCs. The cutting axis turn the metal off the part. When using the text boxes enter the position only not the axis letter. The program will add the axis letter. If the message box says you must enter a value the most likely problem is inserting two decimal points or something that does not read as a number (a letter or symbol) check the text boxes for a mistake and try again.

Radius help

Rapid | Feed | Radius | Drill | G Code | #s | G76 | G92 | Setup | Face | Groove | Calc

Full Radius | Partial Radius

Check known position buttons first

Tool Nose Radius .031

X Part Start Position 8.8035

Z Part Start Position

X Finish Part Position

Z Finish Part Position -4.375

Radius .03

Feed Rate (IPR) .01

Known X Start
 Known X finish

Choose style of Radius
 Corner Radius
 Fillet Radius

Direction to Interpolate
 Clockwise
 Counter Clockwise

X start is larger than X finish
 X start is smaller than X finish

Z start is larger than Z finish
 Z start is smaller than Z finish

Clockwise or counterclockwise is for a slant bed machine. Flat beds will compensate. All boxes must be checked.

Help Preview Code G01 X8.8035 Z-4.314 F0.01
Send Code G03 X8.9255 Z-4.375 R0.061 F0.01

The Radius tab is for turning Radius on parts. Normally you will know either X start or X Finish position choose the proper radio button for this then choose outside or fillet radius then direction to interpolate this is determined on a slant bed machine.

You should know either starting X position or finishing X position.
If you know the starting X position you should know the finish Z position.
If you know the finish X position you should know the starting Z position.
After clicking on the known X position two input boxes will disappear depending on which button you checked.
Check if the X starting position is larger or smaller than the finish X position.
Check if the starting Z position is larger or smaller than the finish Z position.
If you are cutting towards the chuck it is minus. If you are moving towards the tail stock it is plus.
A corner radius can be on an ID or OD so can a fillet Radius.
Basically a fillet radius is the radius from an OD to a face or a bore to a face.

A corner radius is from an OD face to a larger OD.

An ID corner radius is from a face to a smaller bore.

Clockwise is G02 Counterclockwise is G03 these are figured from on a slant bed machine.

The tool upside down and the machine in forward (M03). Flatbed machines should compensate.

Enter tool nose radius, print Radius size, desired feed rate and positions.

If you know starting X and finish Z it will calculate finish X and start Z and vice versa.

If you are using a flatbed machine it will be the opposite. If the X start position is larger or smaller than X finish choose the correct button and the same for Z start and finish. Fill out the text boxes with your values and press preview. If it looks right send the code to the program text box.

Angles

Start	New Tool	Home	End	Mcode	Comments	Turn	Bore	M.O.B.	Angle	Machine
-------	----------	------	-----	-------	----------	------	------	--------	-------	---------

Angle	<input type="text" value="45"/>
Starting Z	<input type="text" value="0"/>
Ending Diameter	<input type="text" value="12"/>
Diameter of angle at face	
Length	<input type="text" value=".25"/>
Feedrate	<input type="text" value=".01"/>
Tool nose Radius	<input type="text" value=".03"/>

Choose Angle or Chamfer

- Chamfer: Known face OD , angle and length
- Known Face, length and diameter at face
- Known start X endX and Face

- Starting diameter is smaller than ending diameter
- Starting diameter is larger than ending diameter

- End Z is smaller than Start Z
- End Z is larger than Start Z

Preview

<code>G01 X11.4649 Z0.0 F0.01</code>
<code>G01 X12.0 Z-0.2676 F0.01</code>

The angle tab is for working with angles or chamfers. A chamfer you usually know the face location. The finish O.D. and the chamfer angle and length. The other choices are for angles. The next one is if you know the diameter at the face and the angle and length of the angle.

The next choice if you know the face or start location of the angle and start and end X. Then choose if the starting diameter is larger or smaller than the ending diameter. Finally choose whether the start Z position is larger or smaller than the ending Z position. After you have filled out the text boxes on the left, Angle, starting Z, Tool nose radius and feed rate etc press preview and if the code looks correct press send to send the code to the program text box.

G76 Thread Help

Rapid	Feed	Radius	Drill	G Code	#s	G76	G92	Setup	Face	Groove	Calc
-------	------	--------	-------	--------	----	-----	-----	-------	------	--------	------

Straight or Tapered		G76 thread cycle	
<input type="radio"/> Straight thread		Do not use constant surface speed on threads. Use G97 not G96.	
<input type="radio"/> Tapered thread		Make sure X start position is bigger than biggest diameter on OD thread. Make sure it is smaller than smallest ID.	
Taper	<input type="text"/>	Per foot	
X Rapid to position	<input type="text"/>	11	
Z Rapid to position	<input type="text"/>	.1	Make sure you start far enough away from the thread in Z for the machine to synchronize the lead.
Z location to thread to	<input type="text"/>	-3	X start diameter is major OD for external thread. Largest diameter for external tapered thread. Minor diameter for ID thread. Smallest diameter for ID tapered thread.
X start threading diameter	<input type="text"/>	11	
X diameter to thread to.	<input type="text"/>	10.875	Root diameter for external, largest root for Od tapered. Major diameter for ID, smallest for ID tapered.
Depth of cut	<input type="text"/>	.009	in thousandths (ie .005)
Threads per Inch	<input type="text"/>	8	Preview <input type="button" value="Preview"/>
Lead in Angle	Up to 60 for V thread Up to 29 for Acme	<input type="text"/>	0 <input type="button" value="Send"/> <input type="button" value="Help"/>

The G76 thread cycle is used for threading using even cuts until it reaches final thread diameter. It can be used for either straight or tapered threads. X position to rapid to is the position at which it will start the cycle. When the tool pulls out it will return to this diameter so for a tapered thread it must be larger than the largest diameter for an ID thread smaller than the smallest diameter. X starting diameter is the position of the first cut. Z rapid to position is the start of the thread cycle and must be far enough away from the part to stabilize the cut before it reaches the part. Consult your machine manual for details.

G92 Thread Help

G92 Thread Cycle
Always use G97
(Constant spindle speed for threading)

Major Diameter

Minor Diameter

Maximum cut in thousandths

Minimum cut in thousandths

Threads per Inch

X Start position

Z Start position

Z Stop position

Reduce cuts by

OD
 ID

Straight or Tapered
 Straight thread
 Tapered thread
Taper per foot

Help Preview Send

```
N1 G00 X10.9 Z0.0
N2 G92 Z-3.0 F.125 X10.869
N3 X10.858
N4 X10.8475
N5 X10.8375
N6 X10.828
N7 X10.819
N8 X10.8105
N9 X10.8025
N10 X10.8
```

The first X position move or rapid to position should be larger than the major diameter of an external thread or smaller than the minor diameter of an internal thread.

The first Z position move or rapid to position should be far enough from thread for the machine to synchronize the lead and spindle check your machine instructions for a safe lead position.

Major diameter should be the Outside diameter of external thread or the largest diameter of an external tapered thread.

Major diameter on an Internal thread should be the thread root or the dimension to thread to.

On an internal tapered thread it is the root diameter at the smallest diameter to thread to

Minor diameter on an external thread is the root diameter or the diameter to thread to.

On an external tapered thread it is the root diameter at the largest part of the taper.

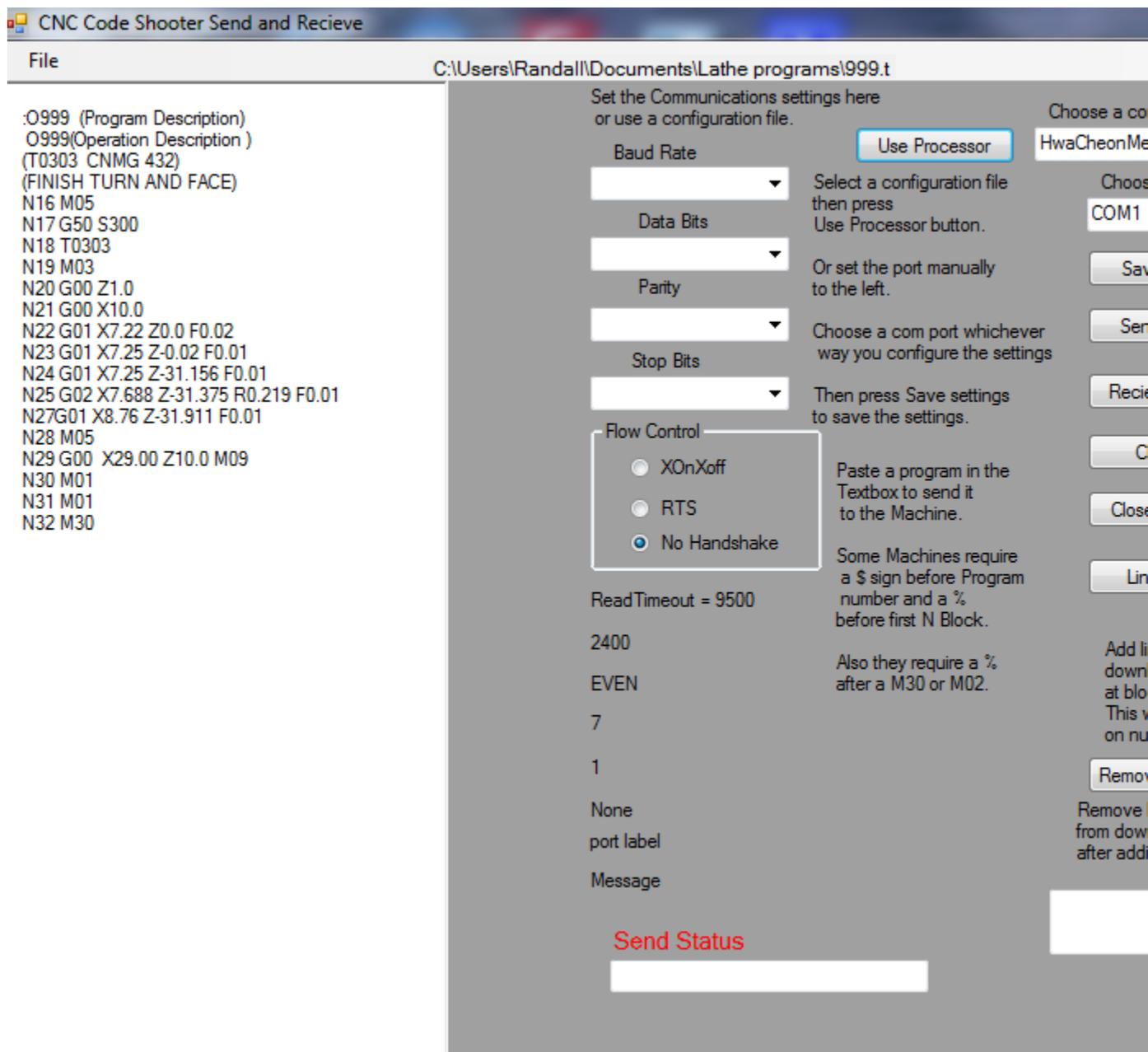
Minor diameter on an internal thread is the bore diameter.

On an internal tapered thread it is the smallest diameter of the taper.

The I value on an external tapered thread should be a negative number.

The I value on an internal tapered thread should be a positive number.

Machine Communications



For Communication settings.

Send and Receive a program using the send tab

You can use a processor file or set the settings manually. If you choose a processor file press use Processor after selecting processor file from the drop down box. After setting the

communication settings manually or choosing the processor files choose the com port(com port 1, 2, 3 etc) and press Save Settings. If the settings are not set right you have to close the port by pressing the Close Port button before resetting them.

Most Controllers require a \$ (dollar sign) before the program number and a % (percent sign) before the first block and a % (percent sign) after the M30 or last block. Refer to the machine manual to verify each controllers settings on the particular machine that you are operating.

To set manually
First you will select the com port, parity, data bits, baud rate, flow control and stop bits.
Then press save settings button. It should say that the com port is open.

Most new computers do not have a RS 232 connection so a USB to RS 232 adapter has to be used. These can be found at Fry's, Radio Shack and other places.

Software Handshake Cable - How to make a cable with a 9 - pin connection on one side to a 25 - pin connection on the other side. This is a typical machine to computer connection cable. We sell the cables already made ranging from 20 feet - 50 feet, shielded or regular.

1. Cross signals SD (serial transmit data) and RD (serial receive data) with each other.
2. Connect signal SG(signal ground) to SG (signal ground).
3. Jump together signals RTS (request to send) and CTS (clear to send) at both ends of the cable.
4. Jump together signals DTR (data terminal ready), DSR (data set ready), and DCD (data carrier detect) at both ends of the cable.

Machine Side	DB - 25 pin	DB - 9 pin	Computer Side
--------------	-------------	------------	---------------

SD (pin #2) connect to RD (pin #2)
RD (pin #3) connect to SD (pin #3)
SG (pin #7) connect to SG (pin #5)

DB - 25 pin	DB - 9 pin
RTS (pin #4) jumper to CTS (pin #5) to CTS (pin #8)	RTS (pin #7) jumper
DSR (pin #6) jumper to DCD (pin #8) and to DTR (pin #4) and	DCD (pin #1) jumper

jumper to DTR (pin #20)

jumper to DSR (pin #6)

Or you could order a cable online at: www.cnccodeshooter.com.

If you do encounter a bug or just have a suggestion something that you may need, please e-mail us your request and we will do our best to fix the situation as soon as possible. E-Mail address is: mail to: support@cnccodeshooter.com.

Below is a snapshot of a lathe program. You will be able to create a program from beginning to the end with CNCCodeShooter. Begin your program with the start tab, setting your program number, the tool description, and the work shift. Next you will click the button Make Code and preview it in the preview box. If the code is good then click the send button to send it over to the text editor to start creating your program. Renumber your program to your own sequence of numbers for your own program or use the default setting.

This is just an example . Every machine will have different configurations and will be slightly different depending on specific settings that are required by different machines. Refer to the machine manual for the specific settings required by individual machines. (example: the baud rate, or the parity, etc.)

```
$
:00576 (ADAPTER)
(1ST OP)
%
(ROUGH AND FINISH FACE TRIGON INSERT R=.031)
N1 G30 U0 W0 (HOME CODE)
N2 G50 X10.741 Z13.289 S900 M42 (X,Z PRESET SPINDLE CLAMP 900RPM)
N3 G00 T0101 (TOOL 1 OFFSET 1)
N4 G96 S350 M03 (SPINDLE FORWARD 350 CONSTANT SURFACE SPEED)
N5 G00 X10.0 Z.1 (RAPID TO POSITION)
N6 X2.0
N7 G72 P8 Q9 U0 W0 D0350 F.008 (CANNED FACE CYCLE)
N8 G00 Z0.0
N9 G01 X0.0 F.008
N10 Z.1
N11 G30 U0 W0 T0100 (HOME CODE, CANCEL OFFSET)
N12 M01 (OPTIONAL STOP)

(#3 CENTER DRILL)
N13 G30 U0 W0 (HOME CODE)
N14 G50 X13.654 Z11.964 S900 M42 (PRESET, RPM CLAMP, GEAR CHANGE)
N15 G00 T0303 (TOOL CHANGE AND OFFSET)
N16 G97 S763 M03 (FIXED RPM, 763 RPM, SPINDLE FORWARD)
N17 G00 X10.0 Z.2 M08 (RAPID TO, COOLANT ON)
N18 X0.0
N19 G00 Z.1
N20 G74 Z-.185 K.3 F.006 (DRILL CYCLE)
N28 G00 Z1.0 M09 (CANCEL DRILL CYCLE AND RAPID AWAY FROM FACE,
COOLANT OFF)
N29 G30 U0 W0 T0300 (HOME CODE, CANCEL OFFSET)
N30 M05
N31 M01

(.843 TWIST DRILL)
```

N32 G30 U0 W0 (HOME CODE)
N33 G50 X13.542 Z9.625 S600 M42 (PRESET, RPM CLAMP, GEAR CHANGE)
N34 G00 T0505
N35 G97 S236 M03 (FIXED RPM, SPINDLE FORWARD)
N36 G00 X10.0 Z.1 M08
N37 X0.0
N38 G74 Z-3.47 K.2 F.005
N39 G00 Z1.0 M09
N40 M05
N41 G30 U0 W0 T0500 (HOME CODE, CANCEL OFFSET)
N42 M01

(ROUGH AND FINISH OD TRIGON INSERT R=.031)

N43 G30 U0 W0 (HOME CODE)
N44 G50 X10.741 Z13.289 S900 M42 (PRESET, RPM CLAMP, GEAR CHANGE)
N45 G00 T0101
N49G96 S350 M03 (SPINDLE FORWARD 350 CONSTANT SURFACE SPEED)
N50 G00 X10.0 Z.1 M08 (RAPID TO POSITION, COOLANT ON)
N51 G00 X2.0
N52 G00 Z.2 (POSITION TOOL)
N53 G71 P54 Q60 U0 W0 D0500 F.01(CANNED TURN CYCLE LEAVE .0)
N54 G01 X1.1
N55 G01 Z0.0 F.005
N56 G03 X1.36 Z-.13 R.13
N57 G01 Z-.720
N58 X1.685
N59 X1.71 Z-.735
N60 G01 Z-.937
N61 X2.
N62 G00 Z.2
N63 G00 X10.0 Z.2 M09 (COOLANT OFF)
N64 G30 U0 W0 T0100 (HOME CODE, CANCEL OFFSET)
N65 M01

(GROOVE OD)

N70 G30 U0 W0
N71 G50 X10.525 Z13.618 S900 M42 (PRESET, RPM CLAMP, GEAR CHANGE)
N72 G00 T1111
N73 G96 S350 M03
N74 G00 Z1.0 M08
N75 X2.0 Z-.4
N76 X1.5
N77 G01 X1.125 F.001
N78 X1.5 F.004
N79 Z-.355
N80 X1.125 F.001
N81 Z-.398
N82 F.006 X1.5
N83 G00 Z1.0 M09
N84 G30 U0 W0 T1100 (HOME CODE, CANCEL OFFSET)

N85 M30

%

A word about rapid travel.

Rapid travel does not always rapid point to point.

Most Fanuc machines rapid at a forty five degree angle until one axis reaches it's destination then the other axis travels in a straight line until it reaches it's destination.

As always be careful running a CNC machine.

Canned Cycles

G70 Finishing Cycle Canned cycle

G71 Turning Cycle Canned cycle

One line G71

P - Pickup block Q - Quit block U - X stock W - Z stock D - Depth

F - feed S - Spindle speed

Some controls use a two line code some use a one line code. Check your machine manual to see which type your machine requires.

A two line example

G71 U.1 R.03(U depth of cut per side R Retract amount)

G71 P111 Q133 U.01 W.003 F.016(P Pickup Block number Q Quit Block number U leave stock in X W leave stock in Z F feedrate)

The Feedrate on the G71 line overrides subsequent feed while in canned cycles.

On finish cycle G70 it will use the feedrate on individual lines.

G72 Facing Cycle Canned cycle

Some controls use a two line code some use a one line code. Check your machine manual to see which type your machine requires.

A two line example

G72 W.1 R.03(W depth of cut R Retract amount)

G72 P111 Q133 U.01 W.003 F.016(P Pickup Block number Q Quit Block number U leave stock in X W leave stock in Z F feedrate)

The Feedrate on the G72 line overrides subsequent feed while in canned cycles. On finish cycle G70 it will use the feedrate on individual lines.

G73 Pattern Repeat

G74 Drilling Cycle

one block

X (u) groove diameter finish

Z (w) Z last peck depth of hole

I - depth of cut

K - distance of cut

D - relief of last cut - Zero for face grooves

F - feedrate

G74 2 block

first block R= return amount

(U) X - same Z (w) - same P - depth no sign

Q - peck amount no sign ?R - relief zero for face groove?

F - feedrate ?S - spindle speed?

Q = .5605 = .5605

G74 R 0.03 - return amount

G74 Z - 4.0 Q 1250 F.010

G75 Grooving Cycle

G75 1 block

U (x) finish diameter W (z) finish position Z

W (z) finish position

I - depth of cut

K - distance

D - relief amount (zero for or don't use for face groove)

F - feedrate (per revolution)

S - spindle speed (55)

G75 2 block

1st block R - return amount

X (u) finish diameter

Z (w) finish Z position

P - depth of cut

Q - distance between grooves

R - return amount

F - feedrate (per revolution)

S - Spindle Speed

If Z, W, Q and K are not used will cut X axis only

G74 & G75 can use incremental or absolute

can use automatic return

return is for 2 block only

elec is set by machine parameter if return amount is used in 2 block and relief.

if return and relief amount are both used in 2 block X determines the valve

R- sets the relief

Single Groove

G75 X3.8 I.08 F.006

Multiple Grooves

G75

G00 X 4.01 Z.625

G75 X3.75 Z 1.125 K .25 F.007

G76 Threading Cycle

G76 2 block

G76 P Q R

G76 X Z P Q F

1st block P - six digit of 3 pairs

1st Z - # of finish cuts no decimal (00-99)

3 - 4 - # of leads pullout 0 - 9.9 lead

5 - 6 - < of thread 00, 29, 30, 55, 60 only

Z - add block

X - finish diameter - incremental first to last difference

Z - finish

R - X difference per side

P - Thread depth no decimal

Q - 1st cut depth

F - Feedrate ie thread pitch

Single block

T 0600 - M41

G97 S 400 M 03

G00 X 5.01 Z -.3 T 0606 M 08 (start)

```
G76 X4.875 Z 2.1 IO K.0311 D 0110
A 60 P 1 F.125
G00 X 18. Z 5.0 M 09
```

2 block

```
T 0600 M41
G97 S 400 M 07
G00 X 5.01 Z .3 T 0606 (start)
G76 P 011060 Q 006 R .0005
G76 X 4.875 Z -3.0 P 0625
Q 0150 F .1250
G00 X 18. Z 5.0 M 09
```

G80 Canned Cycle Cancel

G83 Face Drilling Cycle

G84 Face Tapping Cycle

G86 Face Boring Cycle

G90 Absolute Positioning

Movement in relation to a fixed point (ie X0 Y0)

G91 Incremental Positioning

(This is rarely used on lathes) movement from current position

G91 X1.01 would move 1.01 in X axis from wherever the machine currently is

G92 OD Thread Cutting Cycle

G94 - Face Turning Cycle

A word about rapid travel.

Rapid travel does not always rapid point to point.

Most Fanuc machines rapid at a forty five degree angle until one axis reaches it's destination then the other axis travels in a straight line until it reaches it's destination.

As always be careful running a CNC machine.

If you do encounter a bug or just have a suggestion something that you may need, please e-mail us your request and we will do our best to fix the situation as soon as possible.

E-Mail address is: rellison@cncodeshooter.com