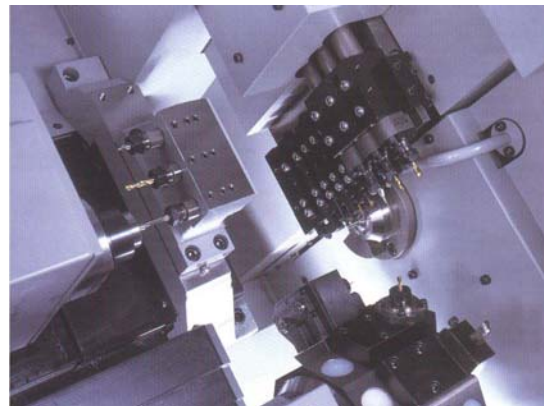
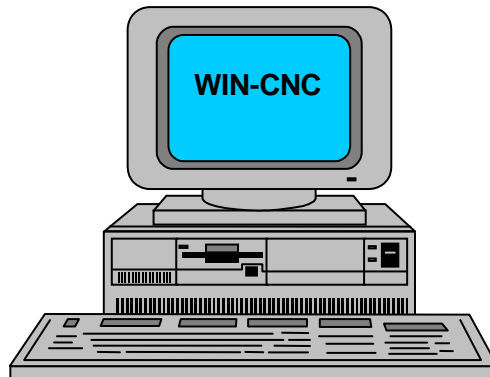


# ***WIN-CNC***

Windows - CNC Communication / Editing Software  
for Cincom Machines



**CITIZEN** The World Leader in CNC Swiss Turning.

**Marubeni Citizen-Cincom Inc.**

Thank you for purchasing this software package. **WIN-CNC** is designed specially for the Citizen Cincom CNC Lathes. **WIN-CNC** will work with most Fanuc controls and also many others.

In most cases all the information you will need is in this manual. The most common problem we have found is with the cable connecting the CNC machine and the computer. Please confirm that the cable wiring is correct with the diagram in this manual on **page 5** before you call.

Thank you again and we hope you enjoy your new CNC communication and editing software package.

Marubeni Citizen-Cincom, Inc.

E-Mail: [software@mctz.com](mailto:software@mctz.com)

Internet: [www.marucit.com/software](http://www.marucit.com/software)

# Table Of Contents

Installation of <b>WIN-CNC</b> .....	Pg. 5
Serial Communications cable configuration .....	Pg. 5
Getting started -First time using <b>WIN-CNC</b> .....	Pg. 6
Using the Files Window .....	Pg. 7

## Main Edit Window

Using the Main Edit Window .....	Pg. 8-22
----------------------------------	----------

### Toolbar Buttons

“Find” Searching for text .....	Pg. 14
“Center-Drill” Feeding depth calculator .....	Pg. 14
“SFM” Calculator.....	Pg. 15
“Thread” Auto programming.....	Pg. 15
“Tap Drill” Drill size calculator .....	Pg. 16
“Radius” Tool nose radius comp calculator for angles .....	Pg. 16
“Calculator” CNC special calculator .....	Pg. 17
“Trig” Right triangle calculator .....	Pg. 17
“Data Pg” Editing the Cincom machining data .....	Pg. 18

### Cincom CNC Programming Help

Make Peck Drill Sub-Program.....	Pg. 19
Make Engrave Sub-Program .....	Pg. 19
CNC Line Numbers.....	Pg. 20
Cut-Off Chamfer .....	Pg. 21
Milling Flats.....	Pg. 21
Compare Files .....	Pg. 22
Convert Files Inch/MM.....	Pg. 22
Program Queue Checking .....	Pg. 23
How to send a file to the CNC machine .....	Pg. 24-26
How to receive a file from the CNC machine .....	Pg. 27-29
Printing a file .....	Pg. 30
How to Change Communication Parameters.....	Pg. 31-32

## **Cincom Machine / Communication Parameters**

L-16 I Yaskawa Motion Pack 110/120 .....	Pg. 33
Fanuc 10T, 2T, 3T, 0T .....	Pg. 33
L-10 VI Fanuc 16TT .....	Pg. 34
B-12 Fanuc .....	Pg. 34
L3-Series V-VII / E-16J/25J Mitsubishi Meldus 320LC/LCV .....	Pg. 35
L5-Series .....	Pg. 36
E-Series II Mitsubishi Meldus 330HLC / 320HLCV .....	Pg. 37
M-20 Mitsubishi Meldus 530HLC .....	Pg. 37

## Installation

Run the **SETUP.EXE** program from within Windows and the files are un-compressed and copied for you. If you have the internet demo version installed then use Control Panel to uninstall WIN-CNC first, then install the new one. No CNC programs will be lost!, but your ID number will change.

**Setup.exe Win95/98/ME/NT/2000** - Assuming your floppy drive is **A:** , from the **Start** button click on **Run**. Now type **A:SETUP** (enter). When the installation is finished, use the **Start** button and **Programs** then click on **WIN-CNC** to start it for the first time. You will now be asked to setup specific Citizen Cincom machines. This will make different folders for each model and also if you choose, will make shortcuts to each on the **desktop**, or in the **Start-Programs-WIN-CNC** menu.  
See - **Getting Started Pg. 6** before using **WIN-CNC**.

### Problems During Install:

If any errors occur during installation, close all other applications, reboot Windows and run setup again. If an error continues please use Windows Scandisk to check each floppy disk. Possibly there was damage in shipping.

## Password

When you run WIN-CNC for the first time, you will be prompted for your registration information. The software will run for 15 days so please fax or send by e-mail the information as soon as possible so we can send you back your unique password. Each copy of WIN-CNC is allowed 3 installs on 3 different PC's. If more are needed, you will need to purchase additional copies. Sorry, we was forced to add this protection! Please read the Password screen and e-mail the data as shown.

## Serial Communication Cable Configuration

computer-25 pin			CNC-25 pin	computer-9 pin			CNC-25 pin
			2-----3				2-----2
			3-----2				3-----3
			7-----7				5-----7
jumper	4-5		4-5	jumper	7-8		4-5
jumper	6-8-20		6-8-20	jumper	1-4-6		6-8-20

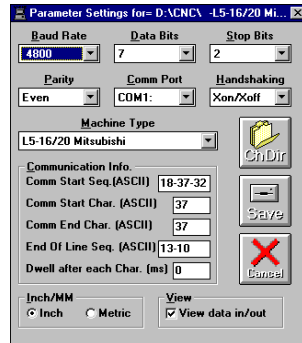
A serial communications cable can be purchased from your nearest computer store, then made to work. You might also need a RS-232 jumper box, Radio Shack PT# 276-1403, and solder it like above. You can make the cable yourself without the jumper box and solder it like above because all that we need is 3 wires, the rest are jumped signals. A null modem cable can now also be used for most CNC machines.

Note: If you use the jumper box from above, configure the cable as a 25pin-25pin, the cable changes to 9pin to 25pin for you.

USB- If all you have is USB ports on a laptop, a USB-Serial adapter is required!

## Getting Started

- 1) To start the program, from the Windows, click on the **Start-Programs-WIN-CNC** icon. To change some of the defaults see **Command Line Options** below. You can also browse through the Windows Explorer and double click on any **“.CNC”** file, **WIN-CNC** will load that file automatically.
- 2) Choose **Param** from the **Main Edit Window drop down menu**. To do so press **Alt-P** or use the mouse. -See page 31, Change Parameters



- 3) Set all the parameters and the machine type to your needs then choose the **Save button**. The communication parameters will be called up every time the currant folder is used. If you have more than one type of machine, make a different folder for each type. Set and **Save** the parameters in each folder for each machine. The installation will create the folders for you but you may have to add some machine types at a later date.

## Command Line Options

There are a few **command line options** that can be used when running **WIN-CNC**.

**WINCNC2K2.EXE c:\cnc\anyfile.cnc /RV=109**

**c:\cnc\anyfile.cnc** -Used to load a file at startup. It must be first on the line.

**/RV=** -This is used to change the Data Pg format for L3-series machines

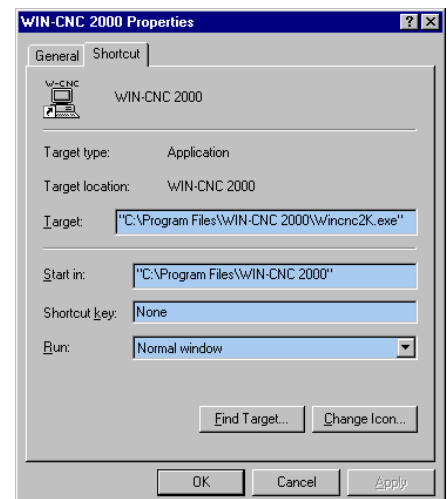
Rom Ver 109 112 115 can be selected.

This is done by changing the properties of the WIN-CNC program Icon and adding the **command line options** to the **Target**. Change **Target** to your needs. example:

**“C:\Program Files\WIN-CNC\WINCNC2K2.EXE /RV=109”**

Use **Explorer** and go to **“\windows\start menu\programs\WIN-CNC”** folder and right click on the **WIN-CNC** icon. Then click on **properties**. Click on **“Short cut”** then change the line **“Target”** like above.

You can also use **Start-Run** and input **WINCNC2K2.EXE /scsetup** this will run the shortcut and folder setup again if you are adding some new machines at a later date.



## Using The Files Window

The **Files Window** is called up when any of the **Main Edit Window** choices are made that need to access the files on the disk. A special file in each folder has machine information and parameters saved in it that help identify each folder as you browse around. Please don't edit or delete the file **WIN-CNC.PRM** from any CNC folder.

1) Just pick a file name by typing the name in the **File Name:** text box, then press (**Enter** or **OK**). You can also press (**Tab**) or **Alt-N** to move to the **Files:** list box, Then use the **arrow keys** and press (**Enter** or **OK**) on your choice.

2) **Drives:** Press (**Tab**) or **Alt-V** to change to the **Drive:** list box, then press the drive letter to change to, example **A** . This will change the directories and files also.

3) **Folders:** Press (**Tab**) or **Alt-D** to change to the **Folder:** list box. Use the **arrow keys** and press (**Enter** or **OK**) on your choice. This will change the files also.

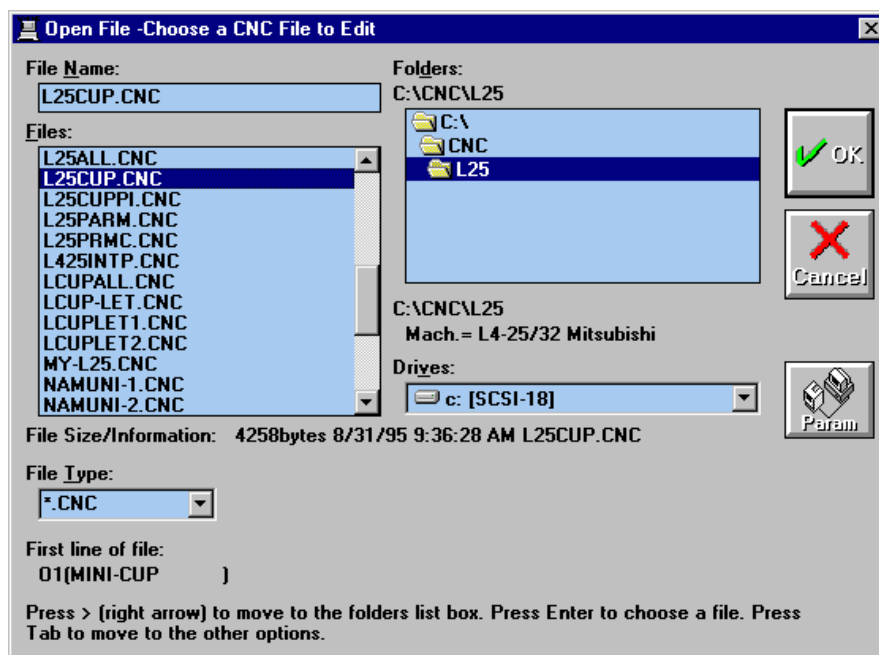
-You can also search by pressing the first letter of a file or folder to move the menu bar.

-The default file extension is **.CNC** . Using this extension enables the first line of the file the menu bar is currently on to be displayed for better identification. You can show only the files you like by changing the **File:** text box to **\*.\*** or **\*.CNC;\*.TXT** or change **FileType:**

-Default File Name Example...

TESTFILE. = TESTFILE.  
TESTFILE = TESTFILE.CNC

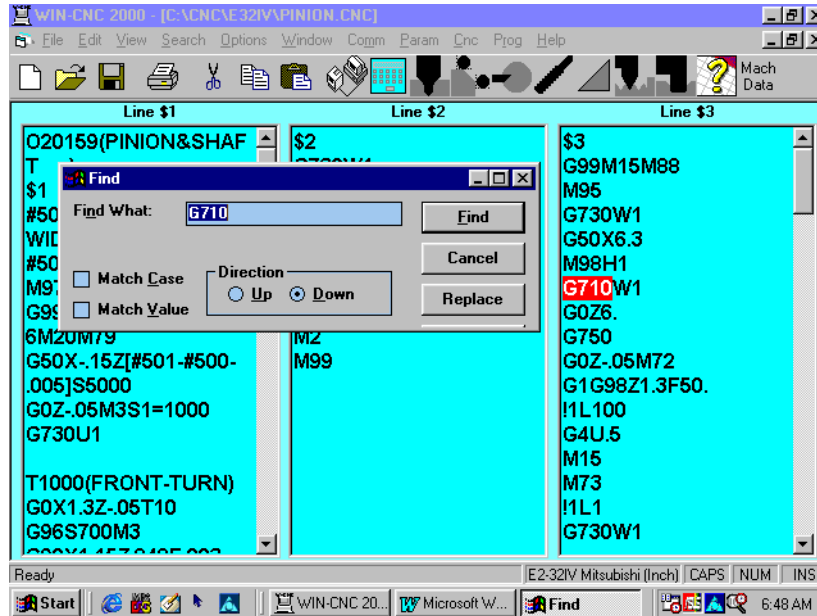
If you input (.) then the default is not used, you are specifying the extension yourself. If the (.) was left out then the default of **.CNC** will be added automatically for you. This can be changed on the **Options menu** by changing the default file spec. See page 11 to change the default extension.



## Main Edit Window

The benefit of this software package is the ability to edit 3 program columns at one time. For example the **E/M-Series** machines have **\$1-\$2-\$3** and the **L-Series V-VII** has **\$1-\$2**

To get started, simply start typing. You can press **Alt** to access the **drop down menu**, then the **highlighted** letter of each **menu choice**, or use the **arrow keys** to maneuver around. Or press the **Left Mouse Button** on your choice to access the menu bar. Following are examples of all the editors abilities.



## Drop down Menu Choices

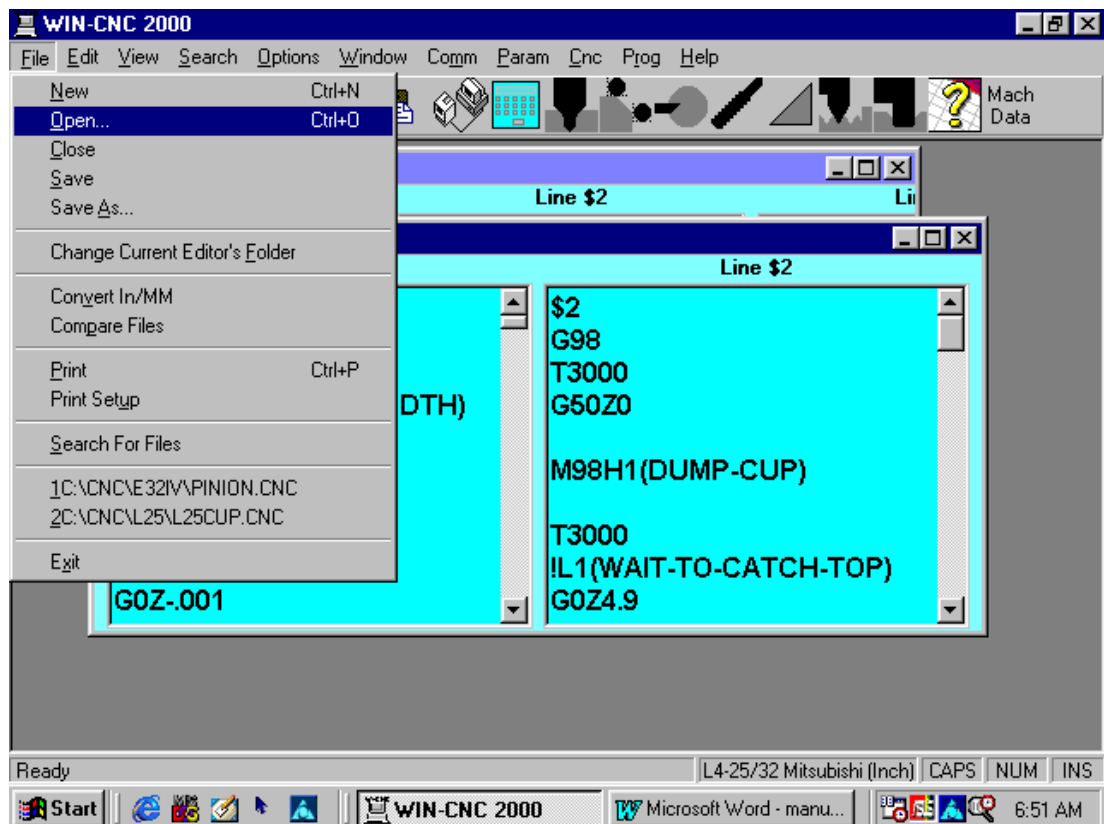
<u>F</u> ile	Alt-F
<u>N</u> ew	Alt-F N
<u>O</u> pen	Alt-F O
<u>S</u> ave	Alt-F S
<u>P</u> rint	Alt-F P
Print Set <u>u</u> p	Alt-F U
<u>S</u> ee <u>k</u> Files	Alt F E
<u>1</u> - <u>5</u>	Alt F 1-5
<u>E</u> xit	Alt-F X

**New** This will prompt you to choose the folder for the New file and open a new edit window. Then it will call the **Data Page Window** so you don't forget to set the Machining data. After setting the data, the common CNC code is added for you to save extra typing.

**Open** This will call the **Files Window**, for you to choose a file name to load from disk into the editor. (see Pg. 7 **Using The Files Window**)

## Drop down Menu Choices (cont.)

- Save** This will call the **Files Window** for you to choose a file name to save your CNC program to disk. If the file is already there then you are prompted to overwrite it or not. (see Pg. 7 **Using The Files Window**)
- Print** Print the current program loaded in the editor. The print format is unique to each Cincom model. One, two, or three columns of text. If a single line program is used then you can choose 1-3 columns for the single line program to be printed side by side to save paper.
- Print Setup** First choose which printer to use then you can set the left margin if needed to put your programs in a 3 ring binder. You can also set the printed font style and size. You can use **Send Text to Printer** to save time printing on dot matrix printers.(not so common any more)
- Seek Files** The **Seek Window** will be called. Input a file name or use wildcards like "\*.cnc" then press **Search**. Double click on the file found and it will be called into the editor.
- 1 C:\CNC\L20V\SAMPLE.CNC** Fast load one of the last files edited.  
This will save you the time looking for the file in the Files window.
- Exit** If the program has not been saved and was edited then you are prompted to save the file first, otherwise **WIN-CNC** is terminated.



## Drop down Menu Choices (cont.)

<b>E</b> dit		<b>Alt-E</b>	
<b>U</b> ndo	Ctrl+Z	<b>Alt-E U</b>	
<b>C</b> ut	Ctrl+X	<b>Alt-E T</b>	
<b>C</b> opy	Ctrl+C	<b>Alt-E C</b>	see page 13 for examples
<b>P</b> aste	Ctrl+V	<b>Alt-E P</b>	
<b>S</b> elect All		<b>Alt-E S</b>	

**Undo** -Undo the last edit. If you hit the delete key by accident? Only one level of undo is available.

**Cut** -Deletes the current selected text and stores it in the clipboard.

**Copy** -Stores the current selected text in the clipboard.

**Paste** -Inserts the clipboard contents at the current location.

**Select All** -Selects all the text in the current editor.

<b>V</b> iew		<b>Alt-V</b>	
<b>F</b> onts		<b>Alt-V F</b>	
Edit <b>B</b> ackground Color		<b>Alt-V B</b>	
<b>W</b> indow Background Color		<b>Alt-V W</b>	
Line <b>\$1</b>	F1	<b>Alt-V 1</b>	
Line <b>\$1</b> Min/Max	Ctrl+F1		
Line <b>\$2</b>	F2	<b>Alt-V 2</b>	
Line <b>\$2</b> Min/Max	Ctrl+F2		
Line <b>\$3</b>	F3	<b>Alt-V 3</b>	
Line <b>\$3</b> Min/Max	Ctrl+F3		

**Fonts** -Displays a list of fonts to choose from. Also the font color can be changed.

**Edit Back Color** -Choose the background colors for the text in the editor.

**Window Back Color** -Choose the background colors for the area behind the text boxes.

**Line \$1-\$3** -Press F1-F3 to move around to \$1-\$2-\$3

**Line \$1-\$3 Min/Max** -Press Ctrl+(F1-F3) to maximize the editor for better viewing. You can also click the mouse on the label **Line \$** above the editor.

## Drop down Menu Choices (cont.)

<u>S</u> earch	<b>Alt-S</b>
<u>F</u> ind	<b>Alt-S F</b>
Find <u>N</u> ext Shift+F3	<b>Alt-S N</b>

**Find** -This will call up the find window to locate text. You can also select text, then start find and the selected text will default to the search text. A great tool added was search by value. So if you searched for "X.5" and your program had "X0.5" the search would not find it. If used "**Match Value**" the text X.5 and X0.5 would be the same.

<u>O</u> ptions	<b>Alt-O</b>
<u>T</u> oolbar	<b>Alt-O T</b>
Default File <u>E</u> xtension	<b>Alt-O E</b>
Default File <u>S</u> pec	<b>Alt-O S</b>

**Toolbar** -Hides/Shows the toolbar icons. This could be used to have more viewing area of your programs.

**Default File Extension** -When **WIN-CNC** is loaded \*.**CNC** is used as default. When saving files and no extension is used then this extension is added to the file name automatically for you. See page 7 for details.

**Default File Spec** -When the files window is loaded \*.**CNC** is used as default for the types of files to be listed. You can show only the files you want so the operators don't edit the wrong files. Example \*.**CNC**;\*.**TXT** would show only files that end with TXT and CNC. All others are not listed.

<u>W</u> indow	<b>Alt-W</b>
<u>C</u> ascade	<b>Alt-W C</b>
<u>T</u> ile	<b>Alt-W T</b>
<u>A</u> rrange Icons	<b>Alt-W A</b>
<u>1</u> C:\WIN-CNC\Program1.CNC	

**Cascade** -Arranges all the open edit windows in rows to be able to find any hidden behind.

**Tile** -Arranges all the open edit windows to be sized to fit the screen at one time, to see the entire window of each.

**Arrange Icons** -Lines up all the minimized Child windows of WIN-CNC in a row at the bottom of WIN-CNC.

**1 C:\WIN-CNC\Program1.CNC** -All the current windows are listed so you can choose the one you want easily from the bottom of the menu.

## Drop down Menu Choices (cont.)

Param

**Alt-P**

**Param** -Open the communication parameters menu. See page 31,  
**Change Parameters Menu**, for details.

Comm

**Alt-m**

**Communication** -Calls up the communication window to send / receive files.  
See page 24-29

Cnc

**Alt-C**

Prog

**Alt-R**

(same as the Toolbar buttons but accessible by the menu)

Make Peck Drill Sub Program

Make Engrave Sub Program

CNC Line Numbers

**(Toolbar buttons)** -Same as Toolbar but using the keyboard.

**Make Peck Drill Sub** -This is used to write a sub program that will peck the drill  
as needed, if you don't have any drill cycles on the CNC. See page 19

**Make Engrave Sub** -This is used to write a sub program that will engrave  
letters using any axis letters, linear or polar. See page 19

**CNC Line Numbers** -This is used to insert sequence numbers in a program.  
Also to remove sequence numbers as needed. See page 20

Help

**Alt-H**

Access the Help menu

M-G Codes

**Alt-H M**

Call the M-G code Help Window

Editor Help

**Alt-H E**

Call the editor Help Window

**M-G Codes** -Calls the help file for the current machine type. You can browse  
around or search to a specific G or M code. You can also copy examples  
to your program.

**Editor Help** -Shows the commands for the editor on the screen.

## Special Key Combinations

**Ctrl-Left Arrow** = Move the cursor one word to the left  
**Ctrl-Right Arrow** = Move the cursor one word to the right  
**F1-F2-F3** = Switch between \$1-\$2-\$3 programs  
**Ctrl+F1-F2-F3** = Expand the current window to the full screen  
**Home / End** = Move the cursor to the Start / End of the current line  
**PgUp / PgDn** = Display the previous / next page  
**Ctrl-E** = Information about combination center drill countersinks  
**Ctrl-D** = To view or change the Machining Data Page (not all machines)  
**Ctrl-F** = To search for some characters in a program  
**Ctrl-M** = M-G code help screen. Only for certain machines  
**Ctrl-S** = SFM and RPM information  
**Ctrl-H** = Shows help screen with all the above key strokes  
**Ctrl-A** = Calculator for trig. or any function  
**Ctrl-T** = Thread programming G76/G92/G32  
**Ctrl-L** = Tap Drill Information  
**Ctrl-G** = Triangle info. Trig  
**Ctrl-R** = Radius Compensation help  
**Ctrl-C** = Copy text to clipboard  
**Ctrl-V** = Paste text from clipboard to editor

## Block Operations - Copy / Delete Text

### Copy Blocks of Text

- 1) Move the cursor to the first character to be copied.
- 2) Press and hold down the **Shift Key**.
- 3) While holding the **Shift Key** down, press the **Arrow Keys** and your text will be highlighted or **Marked**. Or drag the mouse to highlight.
- 4) Now you can release the **Shift Key**. The text is still **Marked**. Press **Ctrl** key and hold it down then press **C** key to copy the **Marked** text into a temporary buffer.
- 5) Move the cursor to the place you would like to copy the text to.
- 6) Press and hold the **Ctrl Key** again and press the **V Key**. The text in the buffer is now inserted into your program.

### Delete Blocks of Text

- 1) Move the cursor to the first character to be **Deleted**.
- 2) Press and hold down the **Shift Key**.
- 3) While holding the **Shift Key** down press the **Arrow Keys** and your text will be highlighted or **Marked**. Or drag the mouse to highlight.
- 4) Now you can release the **Shift Key**. The text is still **Marked**. Press the **Delete Key** to remove the **Marked** text.

## Main Edit Command Buttons

I have put together a group of tools used to help you write your CNC programs without accessing all different notes of your own. Some are unique to the Swiss type machines but others are just nice to have and save a lot of time programming. Each one is explained below.

-Any of the tools that have some form of answer, you don't need to remember it. When each window is closed the answer is displayed in the lower left of the **Main Edit Window's status bar**.

-If you use metric programming and use inch threads or inch center drills etc., then the inch data is converted to metric for you. Or metric data is converted to inch.

## Find

Click on the menu **Search** and then **Find**. Use **Find** to find or search for text in your program. If you are in \$1 then **Find** will only look in \$1 for the text. The **Find** window will always stay on top while in use. You can also search by value so "X.5" would find "X0.5".

## Center Drill Info.

Click on **Cen-Drill** to call up the Center Drill chart. Input the hole size to drill and **Tab** to the next text box. Set either the chamfer amount or the chamfer diameter and press **Tab** again. The Center drills you can use will be displayed and also how deep to feed, to get the desired chamfer.

Cen#	Body	Pilot-An	Pilot-Dia	Pilot-Len	Ch-An	Depth
#00	.125	118	.025	.025	60	
#0	.125	118	.0313	.0313	60	
#1	.125	118	.0496	.0496	60	
#2	.1875	118	.078	.078	60	
#3	.25	118	.1094	.1094	60	=.2597
#4	.3125	118	.125	.125	60	=.2665
#5	.4375	118	.1875	.1875	60	=.2936
#6	.5	118	.2188	.2188	60	=.3072
#7	.625	118	.25	.25	60	
#8	.75	118	.3125	.3125	60	

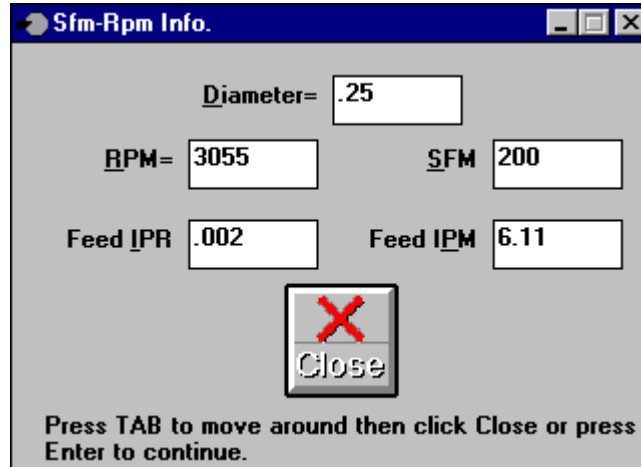
To change the above data, edit the file CEN-INFO.TXT

The file **CEN-INFO.TXT** has the center drill data stored in it. If you have a special type of center that you use as your standard, you can edit **CEN-INFO.TXT** and add it to the list. You can use metric or inch centers and the data is converted for you.

## Main Edit Command Buttons (cont.)

### SFM

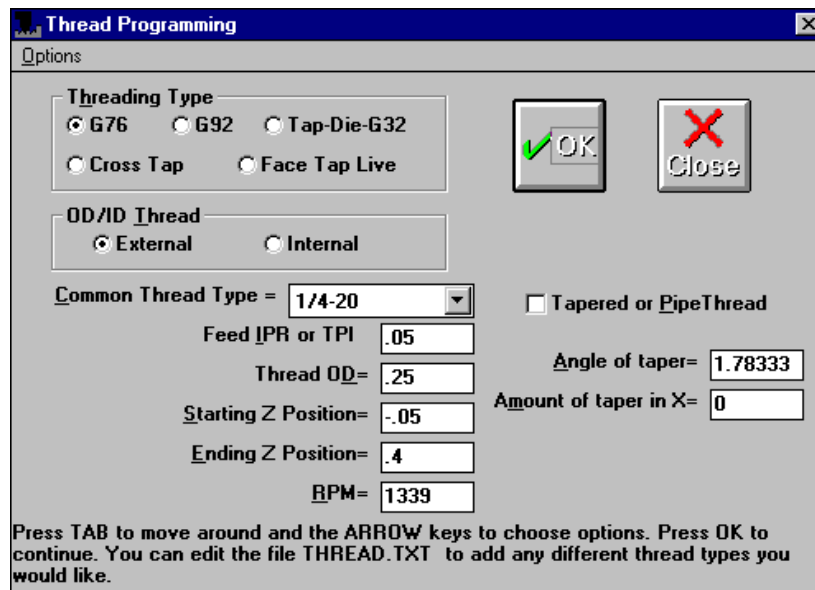
Click on **SFM**. Input the diameter and **Tab** to the next data. The data calculates after every key press. If you are using Metric then SFM = SMM



The screenshot shows a dialog box titled "Sfm-Rpm Info." with a blue title bar. It contains several input fields: "Diameter=" with the value ".25", "RPM=" with "3055", "SFM" with "200", "Feed IPR" with ".002", and "Feed IPM" with "6.11". Below these fields is a "Close" button with a red X icon. At the bottom, there is a text instruction: "Press TAB to move around then click Close or press Enter to continue."

### Threading

Click on **Thread**. This will help you input the code into your program to cut threads. Choose the thread type and G-Code to use. Then fill in the rest of the data. When finished, the code will be inserted into your program for you. If your thread type is not in **common thread** then set the thread type to **non common thread** and set the data manually.



The screenshot shows a dialog box titled "Thread Programming" with a blue title bar. It has an "Options" tab. The "Threading Type" section has radio buttons for "G76", "G92", "Tap-Die-G32", "Cross Tap", and "Face Tap Live". The "OD/ID Thread" section has radio buttons for "External" and "Internal". The "Common Thread Type" is set to "1/4-20" in a dropdown menu. There are input fields for "Feed IPR or TPI" (.05), "Thread OD" (.25), "Starting Z Position" (-.05), "Ending Z Position" (.4), and "RPM" (1339). A checkbox for "Tapered or PipeThread" is unchecked. The "Angle of taper" is set to "1.78333" and "Amount of taper in X" is set to "0". There are "OK" and "Close" buttons. At the bottom, there is a text instruction: "Press TAB to move around and the ARROW keys to choose options. Press OK to continue. You can edit the file THREAD.TXT to add any different thread types you would like."

The file **THREAD.TXT** can be edited and you can add any special threads that you cut often. Just look at the notes in the **THREAD.TXT** file. If you have metric threads the data will be converted for you to run in inch, or inch to metric.

## Main Edit Command Buttons (cont.)

### Tap Drill Info.

Click on **Tap Drill**. This is a fast way to get the drill size for taps. You can choose from common taps or you can enter your own data. Just press **Calculate** to get the drill size. As with the other tools you can edit **TAPDRILL.TXT** to add your own common taps. Metric and Inch are converted for you.

The screenshot shows a window titled "Tap Drill Size". It has two columns of "Common Tap Sizes". The left column lists sizes like 2-64, 3-48, 4-40, 5-40, 6-32, and 6-40. The right column lists metric sizes like M1.6-.35, M1.8-.35, M2-.4, M2.2-.45, M2.5-.45, M3-.5, M3.5-.6, M4-.7, and M4.5-.75. The "6-32" size is selected in the left column. To the right of these columns are input fields for "DIAM." (showing ".138"), "TPI or Pitch" (showing ".03125"), and "% OF THD" (showing "75"). Below these fields, it says "Tap Drill Size = .1076in / 2.732mm". On the far right, there is a "Tap Type" section with two radio buttons: "Cut Tap" (which is selected) and "Roll Tap". At the bottom right is a "Close" button with a red X icon. At the bottom of the window, there is a text instruction: "Press TAB to move around. Input DIAM and TPI. Press Enter or DBLCLICK on Common Tap to set DIAM,TPI for you. Edit the file TAPDRILL.TXT to change the common tap sizes."

### Radius

Click on **Radius**. This is used to help you compensate for the radius of the tool when cutting angles. Input the radius and angle of cut and the amount of compensation is calculated for you.

Ex. R=.0156 Angle=45. =U-.0183 W.0091

#### No Comp

G1Z0  
X.4  
X.5 Z.05  
Z.3

#### With Comp

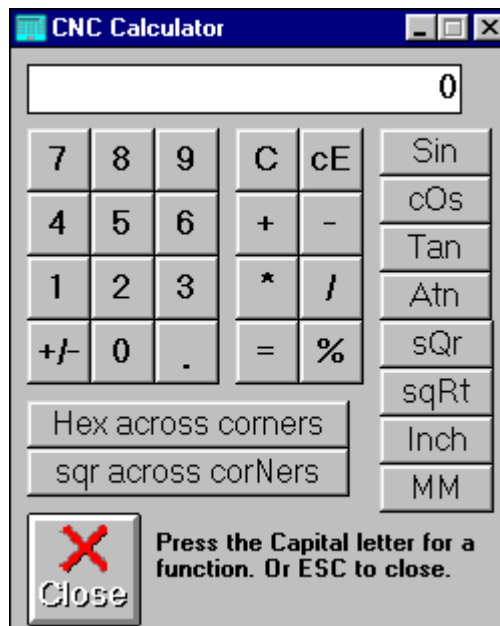
G1Z0  
X.3817(.4-.0183)  
X.5 Z.0591(.05+.0091)  
Z.3

The screenshot shows a window titled "Tool Nose Radius". It has input fields for "Tool Nose Radius=" (showing ".0156") and "Angle from X=" (showing "45"). Below these fields, it says "Radius Comp = U-.0183 W.0091". At the bottom left is a "Close" button with a red X icon. On the right side, there is a diagram showing two circles representing tool profiles. The top circle is larger and represents the tool without compensation. The bottom circle is smaller and represents the tool with compensation. The diagram shows the relationship between the tool radius, the angle of cut, and the resulting compensation values U and W.

## Main Edit Command Buttons (cont.)

### Calc

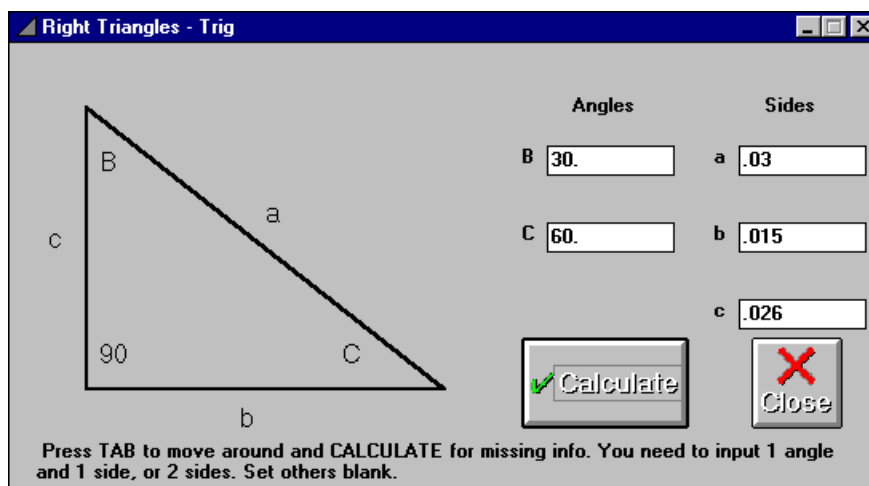
Click on **Calc**. This is a simple calculator that can also use Trig functions. Your answer is displayed when the window is closed. I also added "Hex across corners" Ex-.5 hex stock is .5774 across the corners.



### Trig

Click on **Trig**. Use Trig to get your angles and programming numbers. Always delete the data from the text boxes that you are not using, then you need to have any two sets of data and click on Calculate.

1 side and 1 angle  
2 sides

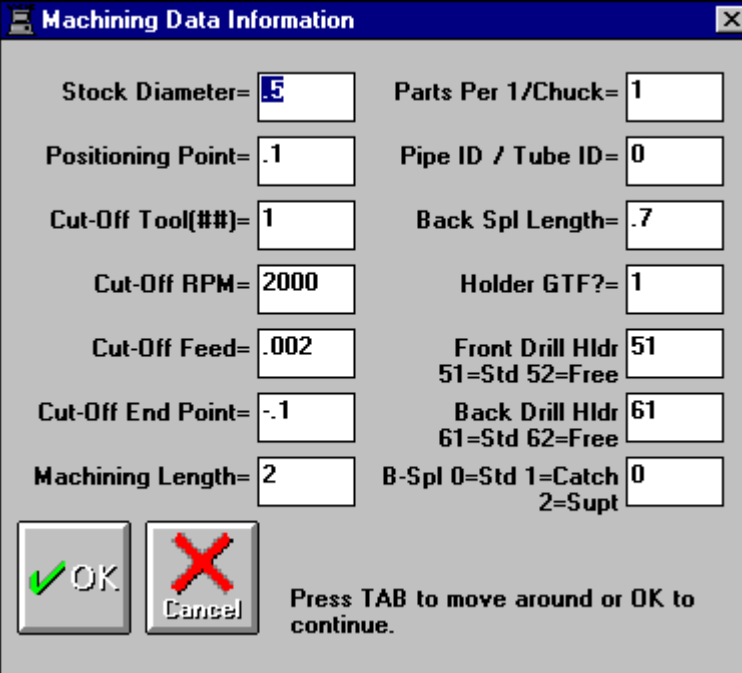


## Main Edit Command Buttons (cont.)

### Machining Data Page

The **Machining Data Page** is used for the Citizen machines with the Mitsubishi control and the B12. It is the special data used by the control to operate, like the **Stock Size** and **Machining Length** etc. This special data is added to the bottom of the program when it is output through the RS232 and is formatted in a special way. The data has to go back in to the machine the exact same way. This is an easy way to edit the data within **WIN-CNC**.

To look at the data just click on **Data Pg** and the data will be converted to simple text boxes that you can edit. When this window unloads all the data is converted again and added to the bottom of the program for you.



Machining Data Information	
Stock Diameter=	5
Positioning Point=	.1
Cut-Off Tool(##)=	1
Cut-Off RPM=	2000
Cut-Off Feed=	.002
Cut-Off End Point=	-.1
Machining Length=	2
Parts Per 1/Chuck=	1
Pipe ID / Tube ID=	0
Back Spl Length=	.7
Holder GTF?=	1
Front Drill Hldr 51=Std 52=Free	51
Back Drill Hldr 61=Std 62=Free	61
B-Spl 0=Std 1=Catch 2=Supt	0

OK Cancel Press TAB to move around or OK to continue.

\*\*\* Note for L3-16/20 Mitsubishi users if running in Inch \*\*\*

There are currently three types of Citizen software for the L3-Series machines that format the **Machining Data** differently between versions. Basically the decimal point is moved around. The default for WIN-CNC is **Citizen ROM version 115** . To use others as default see **Command Line Options** on page 6

**RV109** -Used on all machines with 3600RPM P.O. spindle

**RV112** -Used for a short time and had some decimal point differences from RV109

**RV115** -Used on all type VII machines you must set Mitsubishi param **Bit Select #7 bit 7** to a "1" to use with WIN-CNC.

## Make Peck-Drill Sub-Program

Most peck drill cycles are designed for drilling normal holes. As you know, or will soon find out, we often have to drill very small holes, very deep, and into very tough materials. This part of the software has it's own menu like most of the other functions.

- 1) Press **Alt-r** for the **Prog** menu, press **D** to call up the **Peck Drill Window**
- 2) Use the **Tab Key** or the mouse, and to set all the information in the edit boxes.
- 3) Choose the **Make Peck Sub button** and the **Files Window** (see Pg. 7) is called up. Input a file name and the software makes a **CNC Sub Program** for the drilling. To use this **CNC Sub-Program**, send the file to the CNC machine (see Pg. 23) and use it in your CNC program like this...

Main program	Sub Program #
<b>M98 P1234</b>	<b>O1234</b>

The best way to peck drill is to use **Macro Programming** and design a **Peck-Drill Macro** special for your parts. This is easy to change right at the machine to solve pecking problems. If you have a **L-Series V-VII or E-M Series** machines and use the **User Macro Option**, and you would like to try this out, let me know and I'll send you a copy of a macro program that works very similar to the peck drill sub in this software package.

Program No. 0 ___ 4 digit	1111	Amount of 3rd and all pecks	.015
Peck with X Y or Z axis	Z	Clearance away to start feed	.02
Starting position for drill	0	Retreat pos. out of hole	-.05
Ending position for drill	3.	Amount of P type dwell 0=none	500
Amount of 1st peck	.05	Dwell after how many pecks	5
Amount of 2nd peck	.03	G1 feed rate for each peck	.0005
		Feed rate out of hole 0=rapid	0

Close Make Peck Sub

## Make Engraving Sub Program

- 1) Press **Alt-r** for the **Prog** menu, press **E** to call up the **Engrave Window**
- 2) Use the **Tab Key** or the mouse, and to set all the information in the edit boxes.
- 3) Press **Make CNC Sub** and a CNC program is created.

-To designate the type of axis there is a special format used... **ZWR** or **CHP** or **YVD**  
-The format is the **Absolute axis name, Incremental axis name, Type of axis.**

The type of axis can be **"R"** for radius, **"D"** for diameter, or **"P"** for polar.

## Make Engraving Sub Program (cont.)

Example settings...

**L16/20** engraved so you can read the letters from the operator's view on a flat.

Axis for Height = <b>YVD</b>	Direction from top of letter to bottom Height = <b>+</b>
Axis for Width = <b>ZWR</b>	Direction from left of letter to right Width = <b>+</b>
Axis for Depth = <b>XUD</b>	Direction into cut = <b>-</b>

**M20** engraved so you can read the letters from the operator's view on a flat.

Axis for Height = <b>YVR</b>	Direction from top of letter to bottom Height = <b>+</b>
Axis for Width = <b>ZWR</b>	Direction from left of letter to right Width = <b>+</b>
Axis for Depth = <b>XUD</b>	Direction into cut = <b>-</b>

**E-32** engraved so you can read the letters from the operator's view using "C" axis.

Axis for Height = <b>CHP</b>	Direction from top of letter to bottom Height = <b>-</b>
Axis for Width = <b>ZWR</b>	Direction from left of letter to right Width = <b>-</b>
Axis for Depth = <b>XUD</b>	Direction into cut = <b>-</b>

## CNC Line Numbers

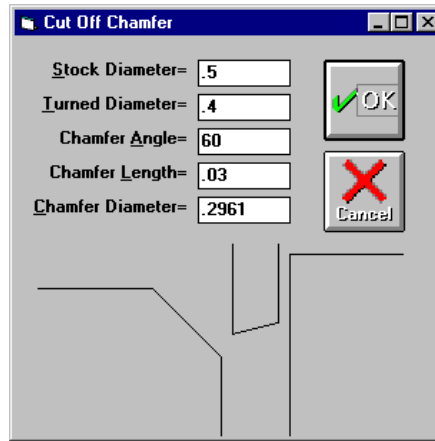
- 1) Press **Alt-C** for the **CNC** menu, press **N** to call up the **Line Numbers Window**
- 2) Use the **Tab Key** or the mouse, and to set all the information in the edit boxes.
- 3) Press **OK** and a CNC program is altered.

The CNC codes currently known to need line numbers are...

M98 H      M99 P      GOTO      G7? P Q

## Cut-Off Chamfer

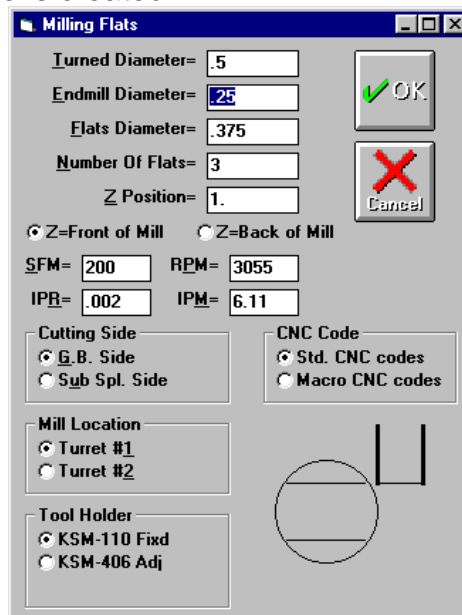
- 1) Press **Alt-r** for the **Prog** menu, press **C** to call up the **Cut-Off Chamfer Window**
- 2) Use the **Tab Key** or the mouse, and to set all the information in the edit boxes.
- 3) Press **OK** and the CNC code is created.



This was made just to make the programming of the chamfer at the cut-off easy. Anyone can do it but this is faster. Simply input the data and then paste the code into your program. This will plunge first then do the chamfer and cut-off.

## Mill Flats Programming

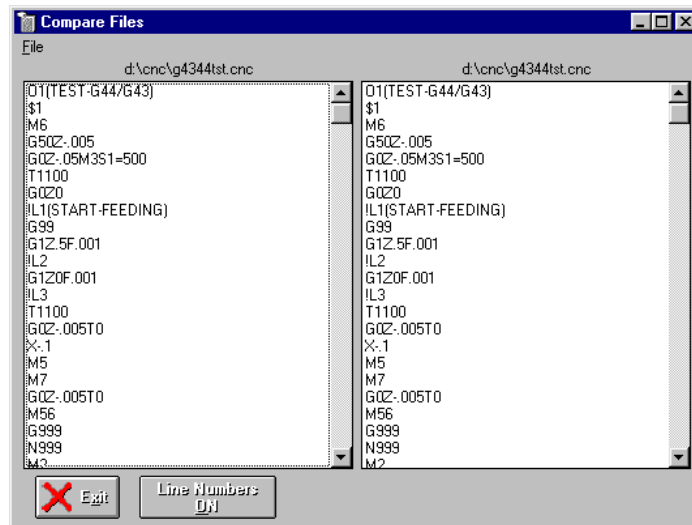
- 1) Press **Alt-r** for the **Prog** menu, press **M** to call up the **Milling Flats Window**
- 2) Use the **Tab Key** or the mouse, and to set all the information in the edit boxes.
- 3) Press **OK** and the CNC code is created.



This was developed just to save time. Anyone can program flats but this makes it much easier and faster. I used macro programming to save memory in the CNC but some machines don't have macro so you can use standard CNC codes also. Just answer the questions and paste the CNC code into your program.

## Compare CNC Files

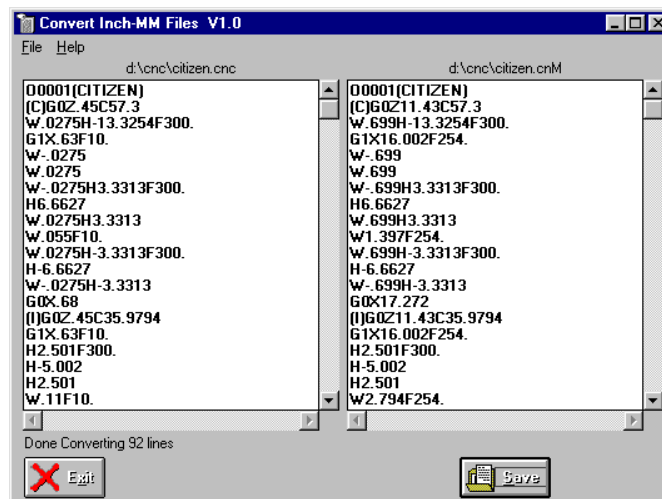
- 1) Press **Alt-F** for the **File** menu, press **p** to call up the **Compare Window**
- 2) Press **Alt-F** for the **File** menu again in the Compare Window
- 3) Choose the old and new files to compare.



Compare was added to be able to find differences in files but on a very simple basis. Only whole lines are compared and the left side will be RED and the right side will be GREEN with the different text.

## Convert CNC Files Inch/MM

- 1) Press **Alt-F** for the **File** menu, press **v** to call up the **Convert Window**
- 2) Press **Alt-F** for the **File** menu again in the Convert Window
- 3) Choose the file to be converted and click on **Save**.



Most users will run in Inch or Metric but not both because Citizen/Mitsubishi machines need parameter changes to change over. For the few that do change over their programs need to be changed. This will do that part of the process easily. Choose **Help About** to see which codes are converted.

# Program Que Checking

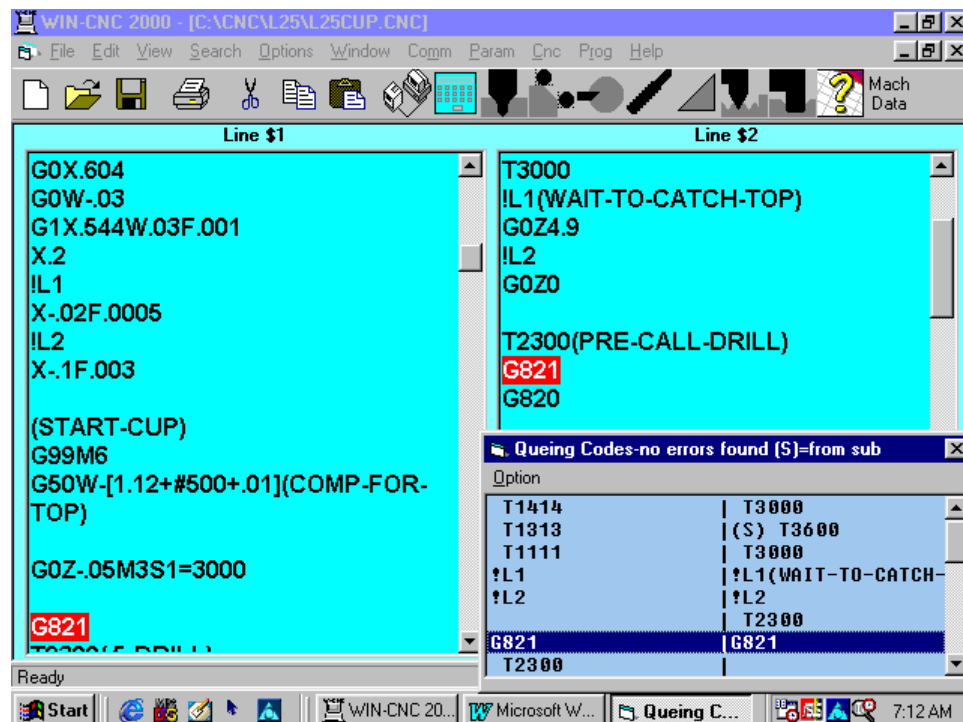
I added this feature to make your life easier when it you have a complex program with many queuing codes and many machining pattern changes. Sometimes the most time troubleshooting a job at the machine ends up being a queuing statement was wrong or missing! When you are done with your program just start this feature and a window will appear showing all the "T" codes, along with any Citizen special codes like G700's, G800's, !L1!L2 etc. This will also follow the path of the program so when a sub program call is used "M98P1234" you may get a message, "You have a M98P sub call. This will be skipped and may not report the queues properly." Most programmers don't use queuing codes in the subs but if you do, you must check them manually.

Codes like "H98H1" and "GOTO100" are read and check just like running on the CNC. When a code was called from the "M98H1" the windows will show an (S) next to the statement so you can tell it was in the sub.

Just to confirm you used proper "T" codes and didn't forget to turn on an offset or to cancel offsets, the list shows all "T" codes. This can be turned On/Off by the "Option" menu.

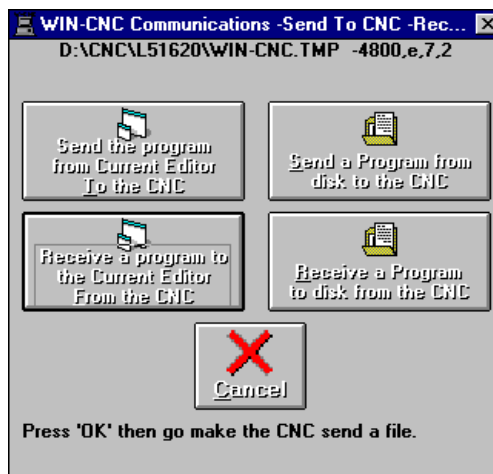
For a quick flow of the program just cursor down through the codes and the editor will jump around to show you where these codes are in each system. If you find a problem, this makes it much easier to see when all 2 or 3 lines are pointing to the problem area simultaneously.

- 1) Press **Alt-R** for the **File** menu, press **q** to call up the **Queuing Window** or just click on the menu "**Prog**" then "**Check Program Queuing**"



## Send A File To The Machine

- 1) Choose **Comm** from the **Main Edit Window drop down menu** by pressing **Alt-M** then you can either send the current file in the editor to the CNC or you can send a different file from disk to the CNC .
- 2) The communication parameters and machine type are loaded from a special file saved in every directory. If you have 2 or more different machines, you only pick the file you want to send to the machine and the software makes any necessary changes special for that machine. (See **Change Parameters** Pg. 31).
- 3) The **Communication Window** is now active. Press the **Send button** first, then go to the machine and press the keys needed to make the machine ready to receive a program. (If there is a problem during communication press the **Cancel button** and you are returned to the **Main Edit Window**.)



## To Send To L-16 I Yas 110/120

- 1) At the L-16, go to the **Edit Mode**.
- 2) Press the **IN** key on the L-16 and the program is sent from the computer. (Only one file can be sent at one time. Sorry but it is a limitation of the machine control.)

## To Send To L-Series V-VII / E/M-Series Mit

- 1) At the machine go to the **DATA SCREEN**. Press the **INPUT** soft key on the **CRT panel**.
- 2) Input the program type, **1**, then cursor to the right DATA field and leave it empty.
- 3) Press the **INPUT** key on the **main panel** and the program is sent from the computer. The Program # will appear in the DATA field during communication.

Example      #**(1)**      DATA(    **\_** )      **INPUT**

## Send A File To The Machine (cont.)

### To Send To B12 Fanuc

- 1) At the machine go to the **EDIT SCREEN**. Press the **I/O** key on the **CRT panel**. Now press the **INPUT** key on the **CRT panel**.
- 2) Press the **CAN** key on the **main key pad** to clear the data field and leave it empty.
- 3) Press the **INPUT** key on the **main key panel** and the program is sent from the computer. You will see the word **INPUT** flashing, indicating the CNC is receiving the program.

Example      (1234) press **CAN**      DATA(    )    press **INPUT**

### To Send To a L-10 VI Fanuc 16TT

- 1) Go to the **EDIT MODE** of the Fanuc.
- 2) Press the \$1 key
- 3) Press the **READ** soft key, then the **EXEC** soft key and the first program is sent from the computer to \$1. To find **READ** press the right soft key.
- 4) Press the \$2 key
- 5) Press the **READ** soft key, then the **EXEC** soft key and the second program is sent from the computer to \$2. To find **READ** press the right soft key.

### To Send To a Fanuc 0T,3T,2T

- 1) Go to the **EDIT MODE** of the Fanuc.
- 2) Press the **INPUT** key and the program is sent from the computer.

### To Send To a Fanuc 10T

- 1) Go to the **EDIT MODE** at the Fanuc. Press the **Right Soft Key** on the CRT to change the menus until you see **READ**.
- 2) Press **READ** then press **ALL** and the program or programs are sent from the computer.

## Send A File To The Machine (cont.)

### To Send To L5-16/20 Mitsubishi

- 1) At the machine go to the **EDIT SCREEN**. Press the **List** key on the **CRT panel** then press **I/O**. Now press the cursor keys to go to **INPUT Communications** key and press **Input** on the **main panel**.
- 2) Select what area to send from the **I/O Contents** combo box by using the **cursor** and pressing **Input**. Usually the **machining programs** is already selected.
- 3) Select the **I/O Target** combo box by using the **cursor** and pressing **Input**. Usually the **work area** is already selected.
- 4) Press **Input** to get past the **Program No.** list box because the file name-number will come in them the Comm port
- 5) Select a device setting, **Communications Set** by using the **cursor** then press **Input**. Usually use the device called **MCC** will be used. After you choose one it will be the default. This is what sets the parameters for communication.
- 6) Select which port to use from the **Input/Output port** combo box by using the **cursor** and pressing **Input**. Usually the **Com1** is already selected.
- 7) Now press **Input** once more on the **[Input]** command button and communication is started.
- 8) If more than one program is needed make the PC ready again and press **Input** again to start communication again. You don't need to go back to the beginning
- 9) Press **Esc** to clear the screen.

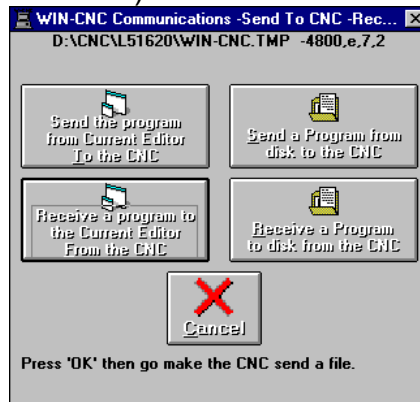
## Receive A File From The Machine

- 1) Choose **Comm** from the **Main Edit Window drop down menu** by pressing **Alt-M** then you can either put the data from the CNC in the editor, or you can put the data into a file on the disk.

If **saving to a file** then input a new file name in the **File text box** or choose another file name from the **File list**, to overwrite. Press the **OK button** or (**Enter**).

The communication parameters and machine type are loaded from a special file saved in every directory. If you have 2 or more different machines, you only pick the file you want to receive from the machine and the software makes any necessary changes special for that machine. (See **Change Parameters** Pg. 31).

- 2) The **Communication Window** is now active. Press the **Receive button** first, then go to the machine and press the keys needed to make the machine send a program. (If there is a problem during communication press the **Cancel button** and you are returned to the **Main Edit Window**.)



## To Receive From L-16 I Yas 110/120

- 1) At the L-16, go to the **Edit Mode**.
- 2) Search to the program you want to send at the L-16 by pressing **O1234** and then press the **Right Cursor Key**. Your program is now displayed on the L-16 CRT.
- 3) Press the **OUT** key on the L-16 and the program is sent to the computer. (Only one program can be sent at one time. Sorry but it is a limitation of the machine control.)

## To Receive From L-Series V-VII / E/M-Series Mit

- 1) At the machine go to the **DATA SCREEN**. Press the **OUTPUT** soft key on the CRT panel.
- 2) Input the program type, **1**, then cursor to the right DATA field, type in your prog #.
- 3) Press the **INPUT** key on the main panel and the program is sent to the computer.

Example      #(1)      DATA(    1234)      INPUT

## Receive A File From The Machine (cont.)

### To Receive From a B12 Fanuc

- 1) At the machine go to the **EDIT SCREEN**. Press the **I/O** key on the **CRT panel**. Now press the **OUTPUT** key on the **CRT panel**.
- 2) Press the program # (**1234**) or use the cursor key to choose the program.
- 3) Press the **INPUT** key on the **main key panel** and the program is sent from the computer. You will see the word **OUTPUT** flashing, indicating the CNC is sending the program to the computer.

### To Receive From a L-10 VI Fanuc 16TT

- 1) Go to the **EDIT MODE** of the Fanuc.
- 2) Press the \$1 key
- 3) Press the program # **O1234**, then press the **PUNCH** soft key, then the **EXEC** soft key and the program is sent to the computer to \$1. To find **PUNCH** press the right soft key.
- 4) Press the \$2 key
- 5) Press the program # **O1234**, then press the **PUNCH** soft key, then the **EXEC** soft key and the program is sent from the computer to \$2. To find **PUNCH** press the right soft key.

### To Receive From a Fanuc 0T,3T,2T

- 1) Go to the **EDIT MODE** of the Fanuc.
- 2) Press the program #**O1234**, then press the **OUT/START(green key)** and the program is sent to the computer. Press **O-9999** to send all programs.

### To Receive From a Fanuc 10T

- 1) Go to the **EDIT MODE** at the Fanuc.
- 2) Press the program #**O1234**, then press the **Right Soft Key** on the CRT to change the menus until you see **PUNCH**.
- 3) Press **PUNCH** and the program is sent to the computer.

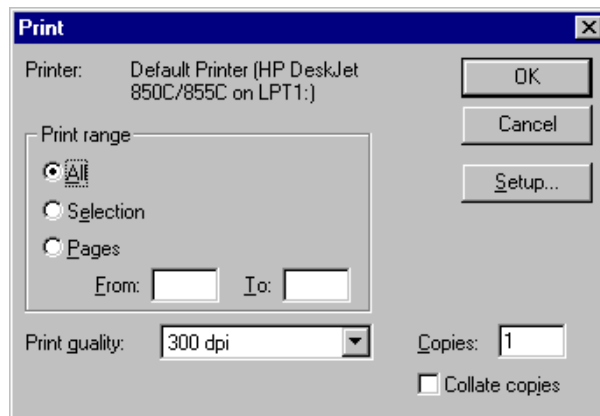
## Receive A File From The Machine (cont.)

### To Receive From a L5-16/20 Mitsubishi

- 1) At the machine go to the **EDIT SCREEN**. Press the **List** key on the **CRT panel** then press **I/O**. Now press the cursor keys to go to **Output Communications** key and press **Input** on the **main panel**.
- 2) Select what area to send from the **I/O Contents** combo box by using the **cursor** and pressing **Input**. Usually the **machining programs** is already selected.
- 3) Select the **I/O Target** combo box by using the **cursor** and pressing **Input**. Usually the **work area** is already selected.
- 4) Input the file number or press **Input** to get past the **Program No.** edit box.
- 5) If the file was not given above then choose it from the list using the **cursor** and pressing **Input**.
- 6) Select a device setting, **Communications Set** by using the **cursor** then press **Input**. Usually use the device called **MCC** will be used. After you choose one it will be the default. This is what sets the parameters for communication.
- 7) Select which port to use from the **Input/Output port** combo box by using the **cursor** and pressing **Input**. Usually the **Com1** is already selected.
- 8) Now press **Input** once more on the **[Output]** command button and communication is started.
- 9) Press **Esc** to clear the screen.

## Printing A File

- 1) Choose **File** from the drop down menu by pressing **Alt-F** then press **P**
- 2) The **Print Window** is called up. Press the **OK button** to print the current file in the editor.



The software first reads the parameters from another file in the directory named **WIN-CNC.PRM**. Once this is done..

-If you have a **E or M -Series machine** chosen, the file will be printed in 3 columns \$1-\$2-\$3 just like it would look at the machine.

-If you have a **L-Series V-VII machine** chosen, the file will be printed in 2 columns \$1-\$2 just like it would look at the machine.

-If you have **Fanuc** or **Yaskawa machine** chosen, then you are prompted to print the file in 1 or 2 or 3 columns. Using 3 columns saves paper!

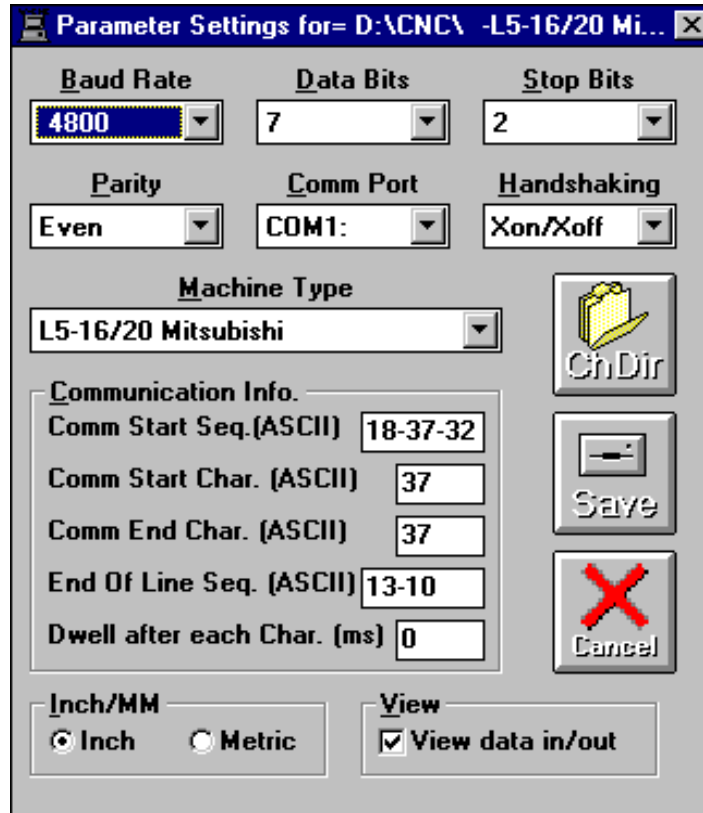
-To change the font and size of the printed CNC program, use "Print Setup" from the "File" menu.

-If you use Explorer you can drag a ".CNC" file onto the Print Manager, or it's icon, and the file will print, then exit **WIN-CNC** automatically. You don't need to load **WIN-CNC**.

## Change Parameters Menu

Every time you run **WIN-CNC**, a file named **WIN-CNC.PRM** is examined to determine the machine type and communication parameters that are set up for the current DOS directory you are in. This will enable you to have many different CNC machines setup in different DOS directories and the software then knows the difference from one CNC machine and another. Every time you change directories **WIN-CNC.PRM** is examined again in the new directory.

- 1) Choose **Param** from the **Main Edit Window** drop down menu by pressing **Alt-P** .  
The **Comm Param** window is now called up.



The image shows a dialog box titled "Parameter Settings for= D:\CNC\ -L5-16/20 Mi...". It contains several settings for a CNC machine:

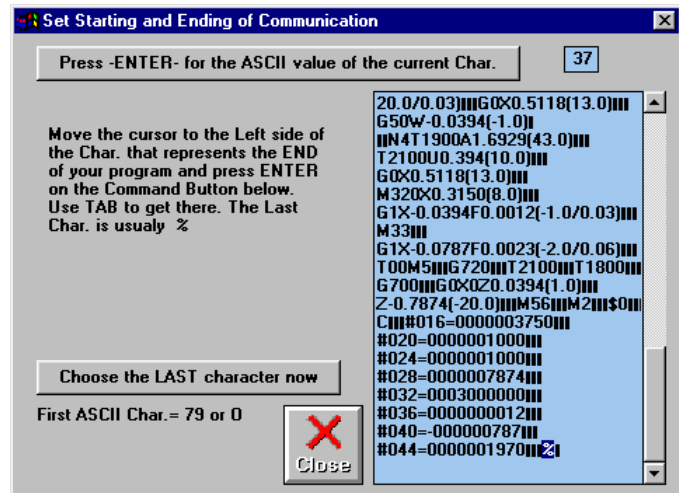
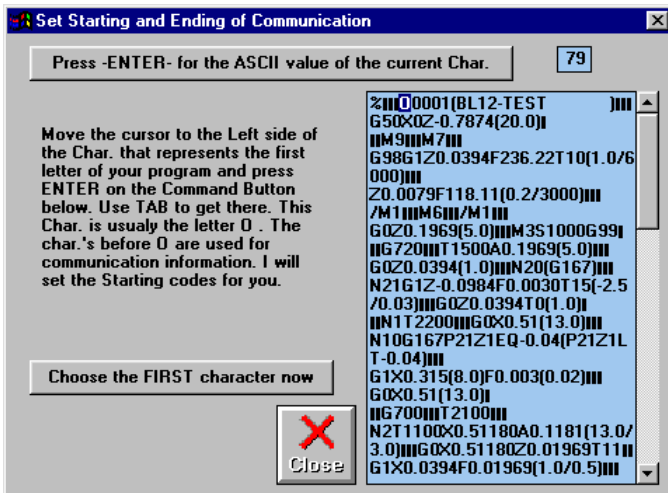
- Baud Rate:** 4800 (dropdown)
- Data Bits:** 7 (dropdown)
- Stop Bits:** 2 (dropdown)
- Parity:** Even (dropdown)
- Comm Port:** COM1: (dropdown)
- Handshaking:** Xon/Xoff (dropdown)
- Machine Type:** L5-16/20 Mitsubishi (dropdown)
- Communication Info.:**
  - Comm Start Seq. (ASCII): 18-37-32
  - Comm Start Char. (ASCII): 37
  - Comm End Char. (ASCII): 37
  - End Of Line Seq. (ASCII): 13-10
  - Dwell after each Char. (ms): 0
- Inch/MM:** ☒ Inch ☐ Metric
- View:** ☒ View data in/out

On the right side of the dialog box, there are three buttons: "ChDir" (with a folder icon), "Save" (with a floppy disk icon), and "Cancel" (with a red X icon).

- 2) Choose the Baud Rate, Data Bits, Machine type, and the rest, one at a time. If you have a different machine than is listed then see details below...
- 3) Choose the **Save button...**, this saves all the parameters in file **WIN-CNC.PRM**
- 4) You are now back to the **Main Edit Window**.  
See pages 33-37 for the CNC's common parameter settings.

# Troubles Communicating With Non Cincom Machines

- 1) Choose the Machine Name **Other**, and input your own name for this machine.
- 2) Now set the Communication port, Baud rate, Parity, Word length, and Stop bits, and Hand-shake, one at a time.
- 3) Now choose **Set parameters by examining a file..** , this will receive a file from the CNC machine and examine it's contents, to determine the rest of the special parameters listed in the menu. You can also set all of these by yourself if you know what they should be.
- 4) Do what it says on the screen and send a CNC program from the machine to the computer.
- 5) Press **Cancel button** when the CNC is finished sending. Choose the start and end of the program by moving the cursor in the text and pressing the command button. The parameters are now set.



- 6) Choose the **Save button...** , this will save all the parameters in file **WIN-CNC.PRM**
- 7) You are now put back into the **Main Edit Window**.

Common settings are...

Machine Name .....	Other (any name)
Communications Port .....	COM1:
Baud Rate .....	4800
Parity .....	Even
Data Bits .....	7
Stop Bits .....	2
Hand Shake .....	XonXoff
Start Sequence -ASCII .....	18-37-13-10 (DC2,%,LF,CR)
Char. before program starts -ASCII .....	37 (%)
Program End Character -ASCII .....	37 (%)
End Of Line Sequence -ASCII .....	13-10 (LF,CR)

## L-16I Yaskawa MM 110/120

Check these parameters...

"X" means leave whatever it was.

bits	7	6	5	4	3	2	1	0	
<b>1004</b>	<b>X</b>	<b>X</b>	<b>1</b>	<b>1</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>X</b>	(end of line LF)
<b>1009</b>	<b>0</b>	<b>1</b>	<b>0</b>	<b>1</b>	<b>0</b>	<b>1</b>	<b>0</b>	<b>1</b>	(2 SB 4800 BD)

## Fanuc 0T Parameters

Setting Parameters Page...

EIA/ISO= 1 (ISO)

I/O= 0

Check these parameters...

"X" means leave whatever it was.

bits	7	6	5	4	3	2	1	0
<b>002</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>0</b>	<b>X</b>	<b>1</b>
<b>012</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>0</b>	<b>X</b>	<b>1</b>
<b>552</b>	<b>10</b>	(4800 BD)						
<b>553</b>	<b>10</b>	(4800 BD)						

## Fanuc 3T,2T Parameters

Setting Parameters Page...

EIA/ISO= 1 (ISO)

I/O= 0

Check these parameters...

"X" means leave whatever it was.

bits	7	6	5	4	3	2	1	0
<b>005</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>0</b>	<b>X</b>	<b>1</b>
<b>014</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>0</b>	<b>X</b>	<b>1</b>
<b>068</b>	<b>4800</b>	(4800 BD)						
<b>069</b>	<b>4800</b>	(4800 BD)						

## Fanuc 10T Parameters

Check these parameters...

"X" means leave whatever it was.

bits	7	6	5	4	3	2	1	0
<b>0000</b>	<b>X</b>	<b>X</b>	<b>X</b>	<b>0</b>	<b>0</b>	<b>X</b>	<b>X</b>	<b>X</b>
<b>5110</b>	<b>1</b>							
<b>5111</b>	<b>2</b>	(2 SB)						
<b>5112</b>	<b>10</b>	(4800 BD)						
<b>5120</b>	<b>1</b>							
<b>5121</b>	<b>2</b>	(2 SB)						
<b>5122</b>	<b>10</b>	(4800 BD)						

## L-10 VI Fanuc 16TT Parameters

Setting Parameters Page...

EIA/ISO= 1 (ISO)  
I/O Channel= 0

Check these parameters...

"X" means leave whatever it was.

bits	7	6	5	4	3	2	1	0
0100	X	X	X	X	1	X	X	X
0101	1	0	0	0	0	0	0	1 (2 SB)
0102	0	(channel)						
0103	10	(4800 BD)						

## B-12 Fanuc Parameters

To view parameters press the **EDIT** key ,then the **I/O** soft key, and then **PARM** soft key.

**IN/OUT PARAM 1P**      <PORT>      <DEVICE>      (DEVICE can = 0-4)

#1 DATA IN	1	1	:
#2 DATA OUT	1	1	:

-----  
**IN/OUT PARAM 2P**

#	<0>	<1>	<2>	<3>	<4>
1 BAUD RATE		2	(4800 BD)		
2 STOP BITS		1	(2 SB)		
3 PARITY BIT		0			
4 EVEN PARITY		1			
5 DATA LENGTH		3			
6 HARD CONT		1			
7 DC CODE CONT		1			
8 DC CODE OUT		1			
9 TV CHECK		0			
10 EOB OUT		1	(LF)		

-----  
**IN/OUT PARAM 3P**

#	<0>	<1>	<2>	<3>	<4>
11 LINE FEED		0			
12 TIME OUT		500			
13 DATA CODE		1			
14 DC3 CODE		1			

-----  
(Fanuc Side) Setting Parameters Page...

EIA/ISO= 1(ISO)

Used for param/offset input/output

I/O Channel= 0

0100	0	0	0	0	1	0	0	0	
0101	0	0	0	0	0	0	0	1	(2 SB)
0102	0	(channel)							
0103	10	(4800 BD)							
0134	0	0	0	0	0	0	0	0	
0135	0	0	0	0	0	0	0	0	
0161	0	0	0	0	0	0	0	0	

# L3-Series V-VII / E16J/25J Mitsubishi Meldus 320

## Parameters

To view these parameters press the **DATA** soft key ,then the **MENU** soft key, and then **PARM** soft key.

---

### [I/O BASE PARM]

IN/OUT 6. 1/4

#	<PORT>	<DEVICE>
1 DATA IN	0	1 :MCC
2 DATA OUT	0	1 :MCC

(DEVICE can = 0-4)

---

### [ I/O DEVICE PARM(1) ]

IN/OUT 6. 2/4

#	<0>	<1>	<2>	<3>	<4>
1 DEVICE NAME		MCC			
2 BAUD RATE		2			(4800 BD)
3 STOP BITS		3			(2 SB)
4 PARITY EFFECTIVE		0			
5 EVEN PARITY		1			
6 CHR. LENGTH		3			
7 TARMINATER TYPE		1			
8		00			
9		00			
10 REWIND CODE		00			
11 HAND SHAKE		3			
12 DC CODE PARITY		1			

---

### [ I/O DEVICE PARM(1) ]

IN/OUT 6. 3/4

#	<0>	<1>	<2>	<3>	<4>
1 DC2/DC4 OUTPUT		3			
2 CR OUTPUT		1			
3 EIA OUTPUT		0			
4 TITLE FEED OUT		0			
5 FEED CHR.		0			
6 PARITY V			0		
7 TIME-OUT SET		999			
8 DATA ASCII		0			
9					
10					
11					
12					

---

## **L5-Series Mitsubishi Meldus 600 Parameters**

- 1) At the machine go to the **EDIT SCREEN**. Press the **List** key on the **CRT panel** then press **I/O**. Now press the cursor keys to go to **Communications Parameter** key and press **Input** on the **main panel**.
- 2) At the **Input/Output** combo box use the **cursor** keys and press **Input** on the device you want to change. Usually **MCC**.
- 3) Set all the data to...

Device Name:	Any name or MCC
Baud Rate:	4800
Stop Bit:	2
Parity:	Even
Character Length:	7
Hrdware Flow:	Invalid
Xon/Xoff:	Valid
DC Code output:	Output
TV Check:	Off
Eob Out:	CR LF
Feed No.:	0
Time Out:	500
Data Code:	ISO
DC3 Code:	0x13

# E/M-Series Mitsubishi Meldus 330/320/530 Parameters

To view these parameters press the **DATA** soft key ,then the **MENU** soft key, and then **PARM** soft key.

---

## [I/O BASE PARM]

IN/OUT 6. 1/2

#	<PORT>	<DEVICE>
1 DATA IN	1	1 :MCC
2 DATA OUT	1	1 :MCC
(DEVICE can = 0-4)		

---

## [ I/O DEVICE PARM(1) ]

IN/OUT 6. 2/2

#	<0>	<1>	<2>	<3>	<4>
1 DEVICE NAME		MCC			
2 BAUD RATE		2			(4800 BD)
3 STOP BITS		3			(2 SB)
4 PARITY EFFECTIVE		0			
5 EVEN PARITY		1			
6 CHR. LENGTH		3			
7 TARMINATER TYPE		1			
8		00			
9		00			
10 REWIND CODE		00			
11 HAND SHAKE		3			
12 DC CODE PARITY		1			
13 DC2/DC4 OUTPUT		3			
14 CR OUTPUT		1			
15 EIA OUTPUT		0			
16 TITLE FEED OUT		0			
17 FEED CHR.		0			
18 PARITY V			0		
19 TIME-OUT SET		999			
20-28		0			

---