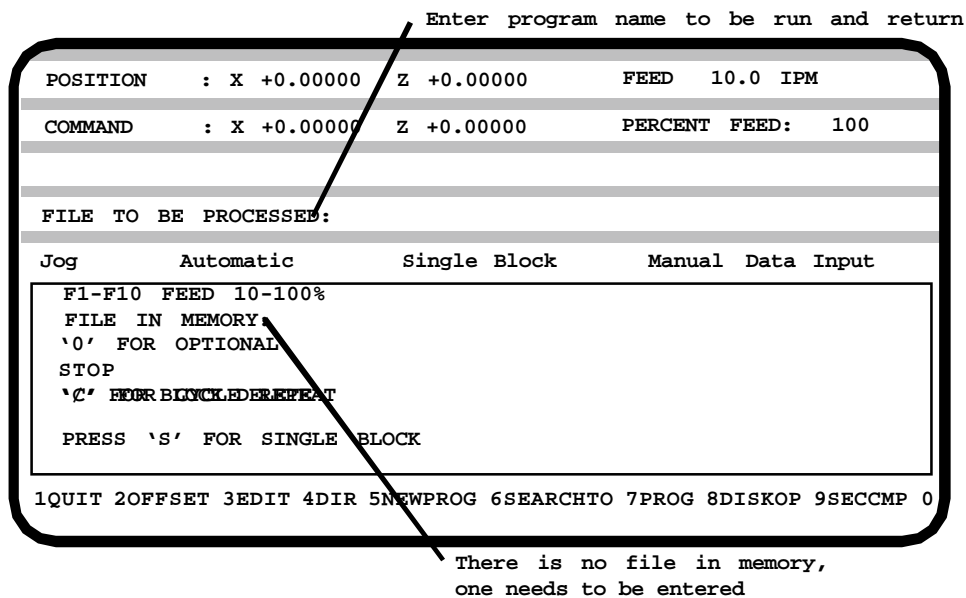


# Automatic Mode

## Running programs

In the Automatic mode the control displays the program that it is currently running. When the control is turned on there is no program selected to run and this space is blank.

Be sure that the tool offsets are correct for the program to be run. If this program is the same as when the control was last shut down, the offsets should still be the same and the program will run without resetting the tools. For example, if you are running a program and shut the control down for the night. When you start up the control the next morning all you have to do is enter the program name in the file to be run once you enter the Automatic mode the next morning. See below. To recall tool offsets from memory refer to F10 in this section.



### 1. Running an existing program

If you have a program saved on the program disk it's name can be entered now and the screen will show that this is the file now in memory. When the cycle start button is pressed the program will be executed.

### 2. Running a different existing program (F5)

Once a program has been selected it stays in memory until it is changed. In the Automatic mode, F5 will delete the currently running program and ask for a new program name. You will notice that after F5 has been pressed the file in memory is blank. If you forget the exact name of the program that you want to run you can press F4 (directory) after F5 has been pressed. This F4 command will list all of your programs on the A: drive, 5-1/4", this is where your programs are stored.

# Automatic Mode

There is a program in memory ready to run

POSITION : X +0.00000 Z +0.00000 FEED 10.0 IPM  
COMMAND : X +0.00000 Z +0.00000 PERCENT FEED: 100

PRESS CYCLE START

Jog Automatic Single Block Manual Data Input

F1-F10 FEED 10-100%  
FILE IN MEMORY: **DEMOPR**  
'0' FOR OPTIONAL STOP  
'/' FOR BLOCK DELETE  
'C' FOR CYCLE REPEAT  
PRESS 'S' FOR SINGLE BLOCK

1QUIT 2OFFSET 3EDIT 4DIR 5NEWPROG 6SEARCHTO 7PROG 8DISKOP 9SECCMP 0

Press F5 to select new program

Program "DEMOPR" is being run

## Program Run - Single step -"S"

It is possible to run the program one line at a time. This is useful when running a new program for the first time. The control displays the next command to be executed before it is run. You can look to see what is going to happen before a mistake is made. To accomplish this get the control into the Automatic mode & input the file to be run. At this point do not push the cycle start yet. Press "S" to activate the single block mode.

POSITION : X +0.00000 Z +0.00000 FEED 10.0 IPM  
COMMAND : X +0.00000 Z +0.00000 PERCENT FEED: 100

PRESS CYCLE START

Jog Automatic Single Block Manual Data Input

F1-F10 FEED 10-100%  
FILE IN MEMORY: **DEMOPR**  
'0' FOR OPTIONAL STOP  
'/' FOR BLOCK DELETE  
'C' FOR CYCLE REPEAT  
PRESS 'S' FOR SINGLE BLOCK

1QUIT 2OFFSET 3EDIT 4DIR 5NEWPROG 6SEARCHTO 7PROG 8DISKOP 9SECCMP 0

Options used in automatic mode

To toggle back to the regular automatic cycle press "A" to turn off the single block mode.

# Automatic Mode

## The "F" keys have the following functions:

F 1	Quit	Go back to the Main menus
F 2	Offset	Adjust tool offsets, correct part size
F 3	Edit	Input and correct programs
F 4	DIR	When no file is in memory this will list all the programs on the user disk
	VER	With a program in memory it will verify and plot the program
F 5	Newprog	This will remove the program from active memory and allow a new one to be entered
F 6	Searchto	This enables the program to be started some place other than the start
F 7	Prog	Runs Calcaid programming system
F 8	Diskop	Disk Operations Erase programs Make a new System disk Make a copy of the user program disk Down and Up load programs from another computer over RS232 Set communication parameters
F 9	Seccmp	Adjust values of secondary tool offsets and TNR compensation values
F10	Sp.fun	Special functions Parts Counter Set value of countdown counter for Continuous cycle Preset feed rate override before starting a program Store tool offset values with program in memory Load tool offset values from memory Set max spindle speed of machine

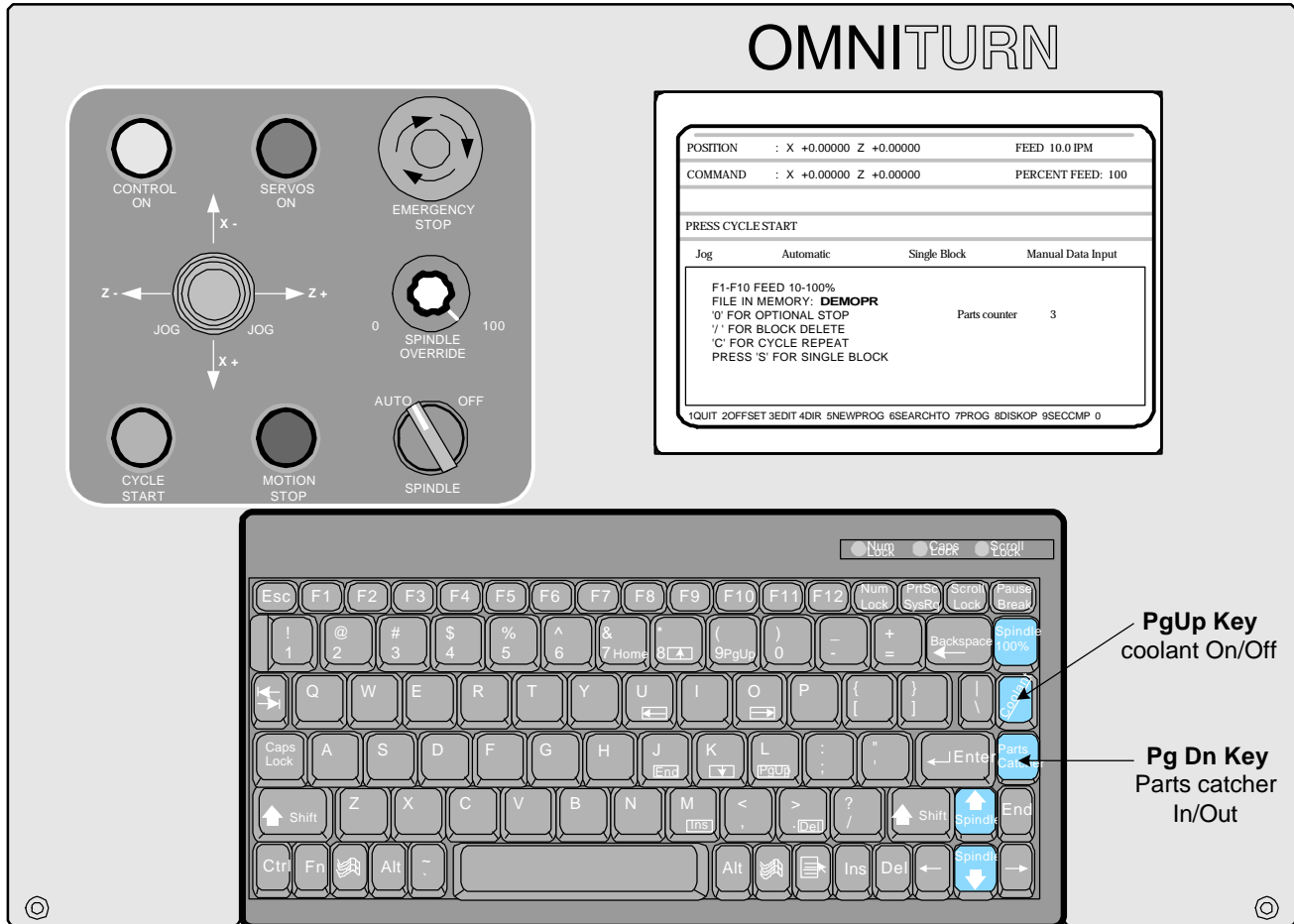
## HOT keys on the keyboard while in the automatic mode

C	Continuous	The program will run continuously -This toggles on / off
O	Optional stop	This makes the M01 act as a stop program command -This toggles on / off
/	Block delete	With this active the control will skip over program lines starting with "/"
A	Automatic	The program will run from start to finish with one cycle start
S	Single block	The program will run one line at a time with each press of the cycle start
F1-F10		Feed rate overrides. The function keys will adjust feed rates (only while program is in motion)
Pg Up		Coolant on/off (M08/09)
Pg Dn		Parts catcher Out/In (M25/26)

## M function keyboard controls

Toggle M functions on and off with keyboard controls.

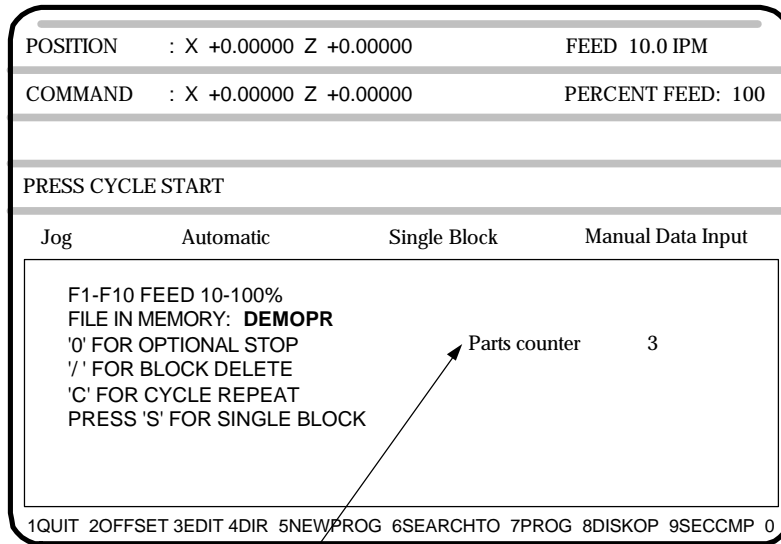
- Press the key once to turn the function on, Press again to turn it off
- Works only in Jog or Automatic mode



OmniTurn front panel Keyboard

# Automatic Mode

## Parts counter - P



The parts counter is turned on in F10

**To turn on a PARTS COUNTER** first press F10, then Press P while you have this screen open. There will be no effect here. When you go to the Automatic mode main screen there will be a counter on the screen. The counter will count up each time a M30 or M02 is executed. To clear the counter come back into the F10 screen and press P again. See notes at the end of this chapter on Function keys, F10.

## Program run -Optional stop activation - M01 -"O"

Optional stops can be put into the program, M01 This stop command is one that can be skipped over. To turn the optional stops on go to Automatic mode, once the program is selected and before the program is run, press "O". This will cause the program to stop like a M00. To get past the stop, press "cycle start". To turn the optional stop off, press "O" again. Uses for the optional stop:

Insert an M01 after a G92 statement for a new tool. This will help when running a new program to be sure that the tool offsets have been entered correctly. Once the program is tested you can turn off the stop and let the program run automatically.

Have an M01 at the beginning of a program that is going to use an automatic bar feed or parts loader. This way, you can have the optional stop activated when you are setting up the machine. Once the cycle and program are proven correct, you can turn off the stop and let the machine run automatically.

## Program run - Cycle repeat -"C"

This is useful for automatic bar feeder or automatic loader operations. When this is activated the program will automatically go back to the beginning of the program after a M30 is encountered and run the program again. The program will continue to run until it is stopped or the continuous counter is set. (see F10 in automatic mode). To turn it off, press "C" again.

# Automatic Mode

## / - Block Delete

POSITION	: X +0.00000 Z +0.00000	FEED 10.0 IPM	
COMMAND	: X +0.00000 Z +0.00000	PERCENT FEED: 100	
PRESS CYCLE START			
Jog	Automatic	Single Block	Manual Data Input
F1-F10 FEED 10-100% FILE IN MEMORY: DEMOPR '0' FOR OPTIONAL STOP '/' FOR BLOCK DELETE <b>BLOCK DELETE ACTIVE</b> 'C' FOR CYCLE REPEAT PRESS 'S' FOR SINGLE BLOCK			
1 QUIT 2 OFFSET 3 EDIT 4 DIR 5 NEW PROG 6 SEARCH TO 7 PROG 8 DISK OP 9 SECCMP 0			

The block delete is used to by pass lines of the program. Put the forward slash 'P' at the beginning of a line. When you want to skip the line just press the "P" key in the automatic mode. Then the **Block Delete Active** text will show up on the screen. This is commonly put on the line with the coolant on command. This way you can turn the coolant off (skip over the coolant on line) by activating the Block Delete.

### Creating a new program

There are a number of ways to create a new program. Here are a few:

**-Use the text edit in OmniTurn.** First a new program name has to be created. This is done by going into the Automatic mode and typing in the new name when the control asks "FILE TO BE PROCESSED". After the RETURN key is hit the control will answer "FILE NOT FOUND, PRESS ANY KEY TO CONTINUE". By doing this you have accomplished two things.

1. If there was already a program with the name you just entered, the control would now be ready to run it. If this is the case, then you would have to select a new name or change the program of the existing one already there.
2. If there was no other program that had the new name then there was one created and loaded into the text editor. Once the new name is entered into the text editor, press F3 to enter the editor. The text editor will ask "PRESS F1 TO CREATE A NEW FILE, ESC TO ABORT". After pushing F1 the editor will provide a new blank screen to enter your program.

**-CAM system off line,** transfer a file via floppy or RS-232. Once they are on the OmniTurn program disk they can be run like any other existing program. Please refer to the section in DOS notes on the format.

**-RS-232 or Disk transfer.** Manually enter a program in a text editor on another computer, transfer as above. Once they are on the OmniTurn program disk, they can be run like any other existing program. Please refer to the section in DOS notes on the format. **-Use Calcaid in OmniTurn.** See the section on using Calcaid.

# Automatic Mode

## Function Keys

On the top of the keyboard is a group of "F" keys. These are used differently throughout the control software. Notations are made on the screen to help the operator remember how the keys are being used with the different sections of software. Care should be taken to remember that these keys change depending on the "Mode" the control is in. Following will be the description of how the Function keys are used in the Automatic mode.

### Function Keys - Automatic Mode - Program not in process

Following are the definitions of the function keys when the control is in the automatic mode and the program is not in motion.

F1 Exit Automatic mode, go to main screen

Pressing the F1 key will exit you from the Automatic mode. This is necessary to get to any of the other modes, ie. Jog or MDI.

F1 Tool offset screen, used to modify tool offsets

This function key brings up the screen to adjust the tool offsets. Tool offsets are used to correct the starting location of the tools, and they will effect the finished part dimensions. These values are created when the tools are setup in the jog mode. When the F2 key is pressed the screen will then ask what tool number you want to adjust. The distances shown are the amount needed to travel from the Home position to the offset location. See below:

1	X: +0.86480 Z: -1.25340	17	X: +0.00000 Z: +0.00000
2	X: +1.65025 Z: -1.99200	18	X: +0.00000 Z: +0.00000
3	X: +2.91130 Z: -0.93885	19	X: +0.00000 Z: +0.00000
4	X: +0.00000 Z: +0.00000	20	X: +0.00000 Z: +0.00000
5	X: +0.00000 Z: +0.00000	21	X: +0.00000 Z: +0.00000
6	X: +0.00000 Z: +0.00000	22	X: +0.00000 Z: +0.00000
7	X: +0.00000 Z: +0.00000	23	X: +0.00000 Z: +0.00000
8	X: +0.00000 Z: +0.00000	24	X: +0.00000 Z: +0.00000
9	X: +0.00000 Z: +0.00000	25	X: +0.00000 Z: +0.00000
10	X: +0.00000 Z: +0.00000	26	X: +0.00000 Z: +0.00000
11	X: +0.00000 Z: +0.00000	27	X: +0.00000 Z: +0.00000
12	X: +0.00000 Z: +0.00000	28	X: +0.00000 Z: +0.00000
13	X: +0.00000 Z: +0.00000	29	X: +0.00000 Z: +0.00000
14	X: +0.00000 Z: +0.00000	30	X: +0.00000 Z: +0.00000
15	X: +0.00000 Z: +0.00000	31	X: +0.00000 Z: +0.00000
16	X: +0.00000 Z: +0.00000	32	X: +0.00000 Z: +0.00000

OFFSET NUMBER:  
Press Esc to exit offset adjustment screen

In the example above you see the offset screen with three tools being used. When it is necessary to correct a tool offset, enter the amount of change that is required. As an example, we will assume that tool 2 in the above example is a turning tool and is cutting apart .001" too large. So enter the offset change of -.001" for the X Diameter offset. When this value is entered you will notice that the total value of X has changed. This addition does not have to be done by the operator.

# Automatic Mode

After selecting a number and pressing Return the screen will ask

1	X: +0.86480 Z: -1.25340	17	X: +0.00000 Z: +0.00000
2	X: +1.65025 Z: -1.99200	18	X: +0.00000 Z: +0.00000
3	X: +2.91130 Z: -0.93885	19	X: +0.00000 Z: +0.00000
4	X: +0.00000 Z: +0.00000	20	X: +0.00000 Z: +0.00000
5	X: +0.00000 Z: +0.00000	21	X: +0.00000 Z: +0.00000
6	X: +0.00000 Z: +0.00000	22	X: +0.00000 Z: +0.00000
7	X: +0.00000 Z: +0.00000	23	X: +0.00000 Z: +0.00000
8	X: +0.00000 Z: +0.00000	24	X: +0.00000 Z: +0.00000
9	X: +0.00000 Z: +0.00000	25	X: +0.00000 Z: +0.00000
10	X: +0.00000 Z: +0.00000	26	X: +0.00000 Z: +0.00000
11	X: +0.00000 Z: +0.00000	27	X: +0.00000 Z: +0.00000
12	X: +0.00000 Z: +0.00000	28	X: +0.00000 Z: +0.00000
13	X: +0.00000 Z: +0.00000	29	X: +0.00000 Z: +0.00000
14	X: +0.00000 Z: +0.00000	30	X: +0.00000 Z: +0.00000
15	X: +0.00000 Z: +0.00000	31	X: +0.00000 Z: +0.00000
16	X: +0.00000 Z: +0.00000	32	X: +0.00000 Z: +0.00000

**X DIAMETER ADJUSTMENT:**  
Press Esc to exit offset adjustment screen

Now enter the value of change-(ie: -.001) and press Return. The value of X will update and then ask you about Z.

1	X: +0.86480 Z: -1.25340	17	X: +0.00000 Z: +0.00000
2	X: +1.64975 Z: -1.99200	18	X: +0.00000 Z: +0.00000
3	X: +2.91130 Z: -0.93885	19	X: +0.00000 Z: +0.00000
4	X: +0.00000 Z: +0.00000	20	X: +0.00000 Z: +0.00000
5	X: +0.00000 Z: +0.00000	21	X: +0.00000 Z: +0.00000
6	X: +0.00000 Z: +0.00000	22	X: +0.00000 Z: +0.00000
7	X: +0.00000 Z: +0.00000	23	X: +0.00000 Z: +0.00000
8	X: +0.00000 Z: +0.00000	24	X: +0.00000 Z: +0.00000
9	X: +0.00000 Z: +0.00000	25	X: +0.00000 Z: +0.00000
10	X: +0.00000 Z: +0.00000	26	X: +0.00000 Z: +0.00000
11	X: +0.00000 Z: +0.00000	27	X: +0.00000 Z: +0.00000
12	X: +0.00000 Z: +0.00000	28	X: +0.00000 Z: +0.00000
13	X: +0.00000 Z: +0.00000	29	X: +0.00000 Z: +0.00000
14	X: +0.00000 Z: +0.00000	30	X: +0.00000 Z: +0.00000
15	X: +0.00000 Z: +0.00000	31	X: +0.00000 Z: +0.00000
16	X: +0.00000 Z: +0.00000	32	X: +0.00000 Z: +0.00000

**Z ADJUSTMENT:**  
Press Esc to exit offset adjustment screen

Enter the amount of change in Z and then press Return.

To correct another tool, enter the tool number now and press Return. **To exit the tool offset** correction screen press ESC and press return. This will tell the control that you are done and bring you back to the Automatic mode.

# Automatic Mode

## Notes:

1. The control will allow you to clear the offsets by pressing C (for clear). Please only do this when you have had experience with the control and understand what you are doing. Clearing offsets can cause you to crash tools if it is done incorrectly!
2. The smallest offset changes are:  
.00005" in Z  
.0001" in X, (this is equal to .00005" on the radius)
3. For Tool Offset changes of more than .02" the control will ask the operator if this is correct. If not it will ask you to re-enter the correction This is a safety feature to ensure that you do not put in a large correction in error, ie 1" instead of .001"
4. If you have no change to a offset value just press Return without inputting a value. The control assumes that you want Zero change.

# Automatic Mode

## Program Editing -F3

**F3** Edit, On screen text editor, used to change existing programs, or enter new ones

The editor is a full function text editor. In the OmniTurn you will be using only a small part of the capability of the editor. In the following description the most basic functions. If you want to learn more follow the instructions given in the HELP screens. (F1 while in the editor is active)

### Starting the editor:

The editor is accessed from the Automatic mode by pressing F3 at any time. The program listed as "FILE IN MEMORY" will be activated. If you want to work on a program you have to make it the active program. When you enter the Automatic mode it asks "FILE TO BE PROCESSED", type the file name that you want to edit and press RETURN. When the file name appears as the file in memory press F3. If you have to correct the program that is currently running just press F3.

- It will ask if you want to make a Backup copy. Making these backup copies is not required. If you are new to PC's and DOS it is suggested that you make the backup copy so that if you loose the work that you have created it can be brought back.
- Either press "ESC" for no backup or F1 to create the backup file

### The editor can also be used to enter new programs.

- Get to the Automatic mode
- If there is a program in file memory press F5
- Enter the new program name when prompted: "FILE TO BE PROCESSED"
- If the control does not tell the program is not found then you are using a name that already exists. Either pick a new name or plan on erasing the program that already exists with that name.
- Press any key to continue
- Press F3 to enter the editor
- The editor will ask if you really intend to create a new program, press F1 if you do, if not press "ESC"

### Exiting the editor and saving corrections made

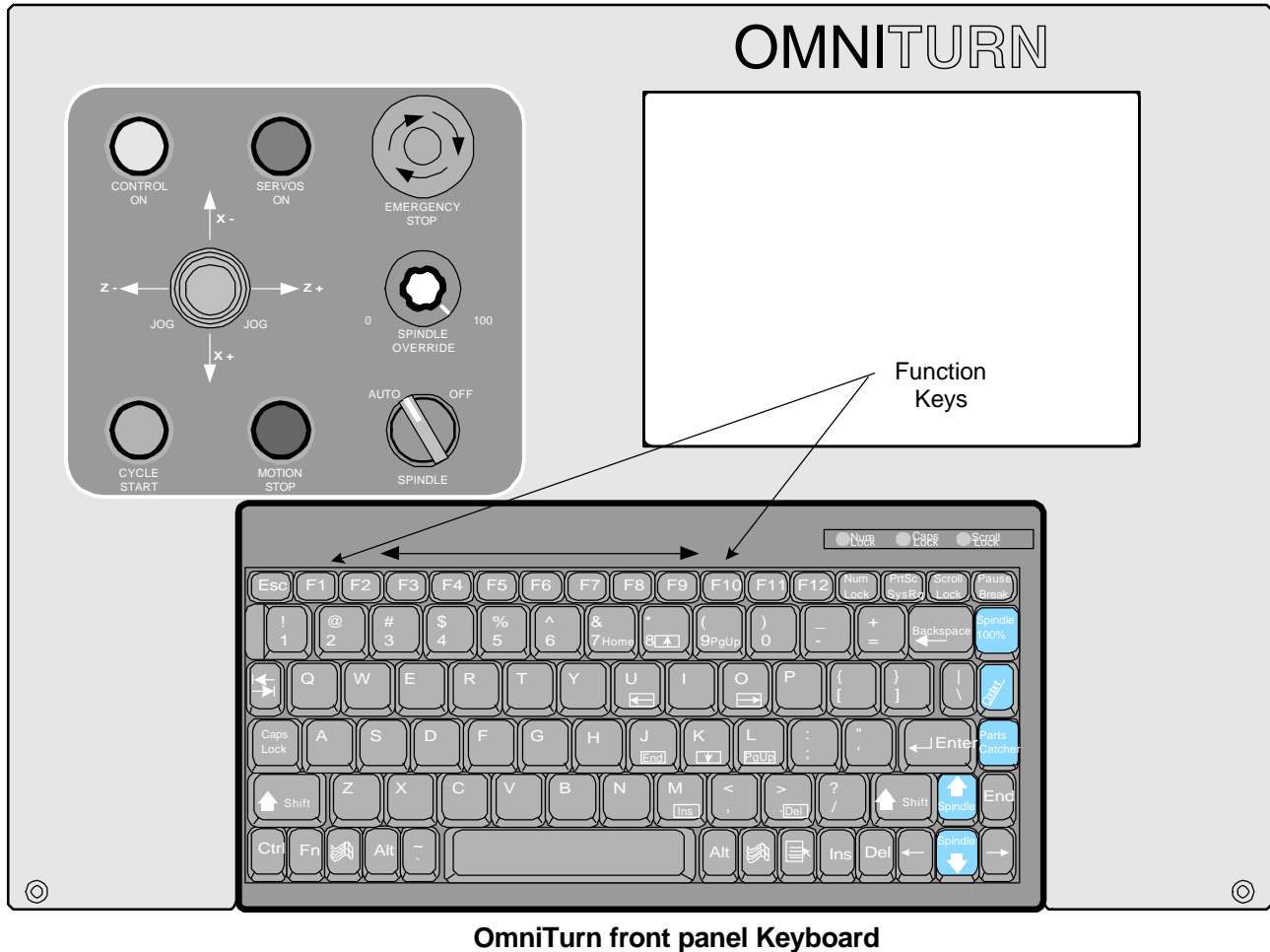
- Press F1 -this is for HELP
- Press F2 -this will exit the editor and save the corrections that have been made

### Exiting the editor and NOT saving the corrections made

- Press F1 -this is for HELP
- Press "ESC" -this will halt the automatic saving function of the exit routine
- Press F2 -this will exit the editor without saving the corrections

# Automatic Mode

## The Keyboard



OmniTurn front panel Keyboard

**Number keys:** Use the number keys across the top of the keyboard. The number keys on the right side are not active as numbers unless set with the number lock key. (located above the #7 key)

**Cursor keys:** These are the number keys on the right side of the keyboard. They will move the cursor around the screen as required.

Pgup: will move the cursor one screen back per hit

Pgdn: will move the cursor one screen forward per hit

End: will move the cursor to the end of the line it is on

Ins: will add spaces in front of the cursor location

Del: will delete the character at the cursor

Back Space: will erase the character to the left of the cursor

### If you want to add lines to a program:

- Move the cursor to where you need to add the line.
- Type the new text and enter a RETURN if needed

### Correcting existing text:

- move the cursor to the text to be corrected
- type the new text in
- delete the text being removed

# Automatic Mode -Notes on the Editor

The text editor can help you manipulate and modify your programs. The editing functions are:

## **F4 -Block delete**

This function is used to erase sections of the program. To use this function:

- move the cursor to the beginning of the code that you want to erase
- press F4 to start the text marking
- move the cursor to the end of the text that you want to erase. The text that you are going to erase is now being marked.
- if you want to  
the code and cancel the delete command press F5
- to complete the erase press F4 again

## **F3 -Block copy**

This function is used to copy sections of the program.

- move the cursor to the beginning of the code that you want to copy
- press F3 to start the text marking
- move the cursor to the end of the text that you want to copy. The text that you are going to copy is now being marked.
- if you want to unmark the code and cancel the copy command press F5
- to end the marking of the code to be copied press F3 again.
- then move the cursor to the location that you want the code copied to, then press F3
- you can copy the code more than once, move the cursor again and press F3 again
- to end the marking press F5

## **F9 -Find, F10 -Replace**

This function is used to find text in the program and then replace it with new text. An example for using this is to change the rapid travel feed rates. Sometimes it is advantageous to lower the rapid feed rates when you are working with a new program and expensive tooling. Before running the program for the first time you could change the F300 moves to F100. Run the program and prove it out, then change the F100's back to F300 for production

- go to the text editing screen
- move the cursor to the beginning of the program
- press F9, at the top of the screen you will see FIND ". " type the text that you want to replace. In the case of the example you could type F300
- press F10, at the top of the screen you will see REPLACE ". " type the text that you want to put in place of the found text. In the case of the example you could type F100
- press RETURN get back to the edit mode
- press F10 to find and replace, repeat pressing the F10 key until you have gone through the entire program

## **Renaming programs**

In the editor you can duplicate a program and give it a new name. This is very useful! IE: if you want to make changes to an existing program and still have the original to fall back to if the changes don't work the way you want, or if you have a family of parts that use the same basic program with only minor changes. With renaming you could make a custom program for each part without having to type the entire program again.

- make the program active in the automatic mode
- press F3 to go to the editor
- press ESC to bring the text to the screen, then press F1 to get to the HELP screen
- at the HELP screen press F5 for RENAME. You will see the original file name at the top of the screen with B: in front of the name. An example of this would be "B: FILENAME". Use the arrow keys to move the cursor to the right of the B:. Then type the new name of the file and delete the old one. The new name could be "B:NEWFILE".
- after the name has been typed press RETURN. This will take you back to your program.
- when you are done working with the program, save and exit the editor as you normally would, press F1, then F2. Now you will still have the old program, and a new one.

# Verification -F4

## Verification (F4)

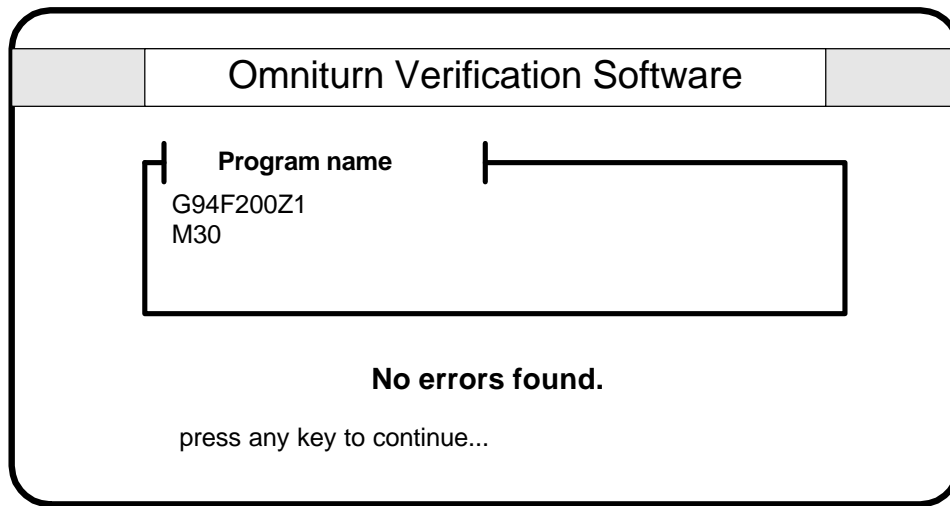
This function will check the current program. The verification does a few things automatically:

1. Check syntax and look for possible errors, then allow corrections
2. List all the tools used  
List all secondary offsets used and their current values
3. Show estimated cycle time
4. Show graphically the tool path created by the G code program

### Running Verification

To use the verification software you must first make your program active in the Automatic mode. Once the program is active you will notice that the F4 indication on the bottom of the screen changes to VER. Press F4 and the verification software will be called up.

### 1. Syntax and program check.



The first operation performed by this section is the checking of syntax (is the program grammatically correct) and a screen for possible gross errors (like trying to move Z -50). If there are no errors or comments then the screen will appear as above.

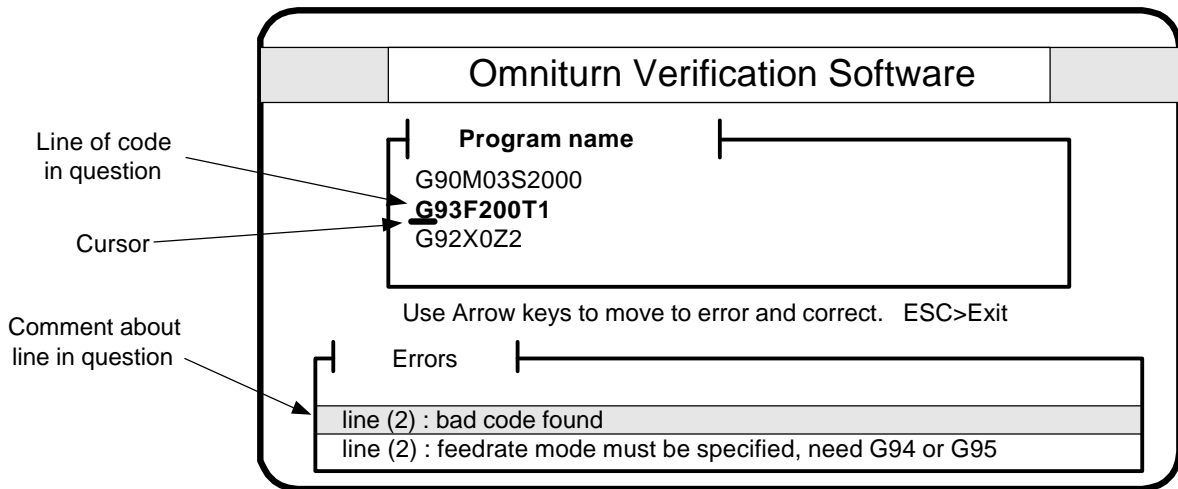
### Error comments and corrections

### The error comments are just that, comments.

There are times that the "errors" it finds are moves that you intended to do. Use the verification software as a guide, not a rule. When the software was developed it was intended to try and screen out all sorts of possible errors, so this causes it to catch more good lines. Make corrections to your program with care.

If the verification finds lines of your program that it does not like it will show the line of code in the upper box, highlighted. In the lower box is a description of the possible problem.

# Verification -F4



## Making corrections:

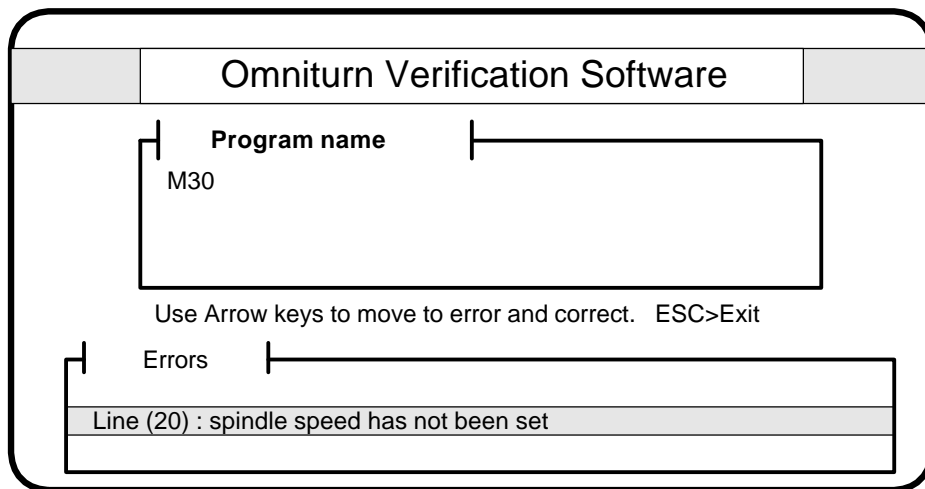
If you find an error in your program it is possible to correct it now. Use the left / right arrow keys to move the cursor to the point in question. In the above example the G93 should be G94. Arrow over to the 3 and then type a 4. When you are done with the correction you can go on to the next comment. When you are done with all the corrections the checker will go back and run the check one more time.

## Going to the next comment:

If there are more than a few comments about the program you can use the up and down arrows to move to new comments.

## Missing spindle speeds:

If the program does not have a spindle speed the OmniTurn will comment. This is because a spindle speed is needed to estimate the cycle time. If you do not want to add a spindle speed to the program you can force a cycle time estimate in the verification when it asks for a spindle speed.



Exiting Syntax checking: Press ESC.

## 2. Listing tools and offsets

The next screen is a list of the tools used and some information about them:

# Verification - F4

Omniturn Verification Software					
Program name					
Tool	Description	G92	G92Z		
T1	(LH Turning tool 008 radius)	0.10800	0.20800		
T2	(LH Threading tool)	0.00000	0.20000		
Use Arrow keys to move to error and correct. ESC>Exit					
Secondary offsets use					
Offset	X	Z	Offsets	X	Z
No Secondary offsets specified.					

## This screen shows a lot of information:

- It lists the tools to be used with their G92 starting points. This makes setting up a program easier. No longer do you have to look through long programs to find the tools and their starting values.
- There is also a description of the tool. The verification software will show any comment that appears on the same line with the tool call in the program. In the above program the are comments on the line:

```
T1 (LH Turning tool 008 radius)
G92X.108Z.208
```

It is good practice to show the type of tool, the size of the radius used, and which direction it points

- The secondary offsets are also shown. The current value of these offsets are listed. To make changes to these offsets you have to go back to the automatic mode and then F9.

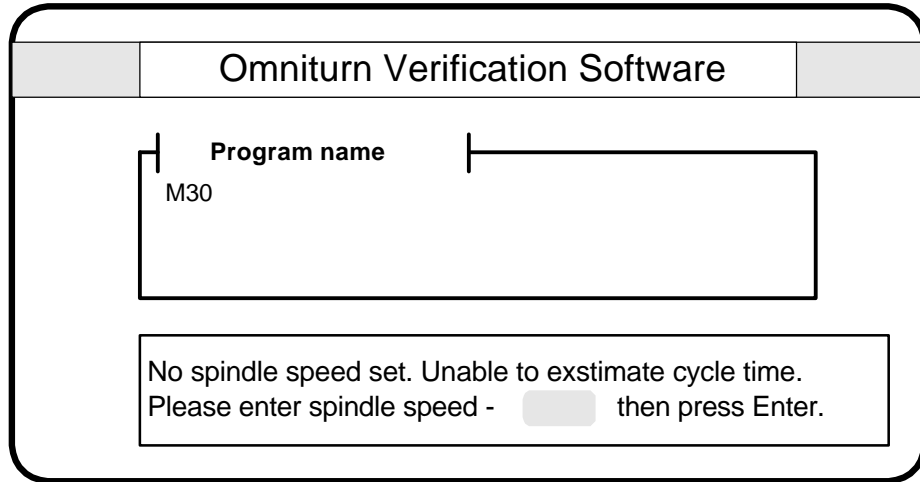
## 3. Estimating cycle time

Now the cycle time is estimated.

Omniturn Verification Software	
Cycle time	
Estimated cycle time for this part is 30 seconds.	
Press any key to continue...	

## Verification -F4

If the program does not have a spindle speed you are now given a chance to input a single speed for estimating the cycle time:



Omniturn Verification Software

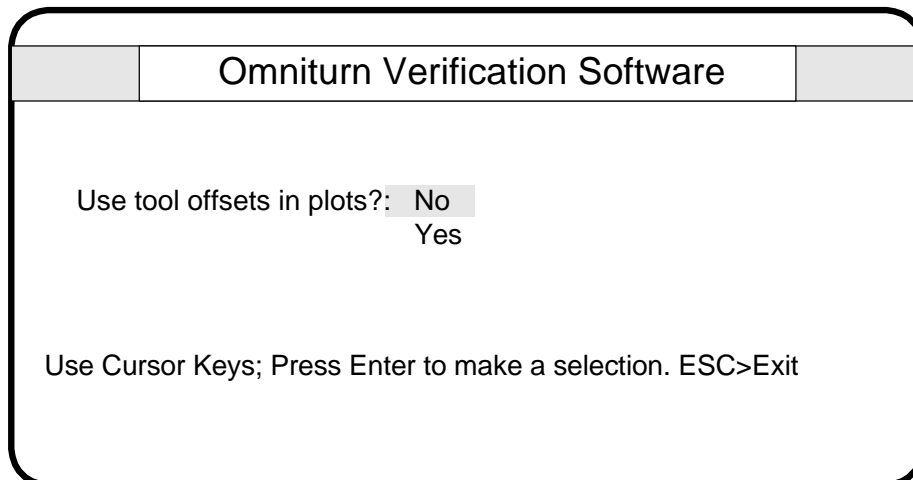
Program name  
M30

No spindle speed set. Unable to estimate cycle time.  
Please enter spindle speed -  then press Enter.

### 4. Tool path graphics

Before you show the graphics you have to choose to show them with or without the tool offset locations. If you are running this software on your OmniTurn and the tools have all been set, then enter yes. If they have not been set or you are running the software off-line, then say no. Use the Up / Down arrow keys to highlight your selection. Then press Enter.

Running the graphics with tool offset locations helps pick up possible collision conditions between tool paths and the work piece.



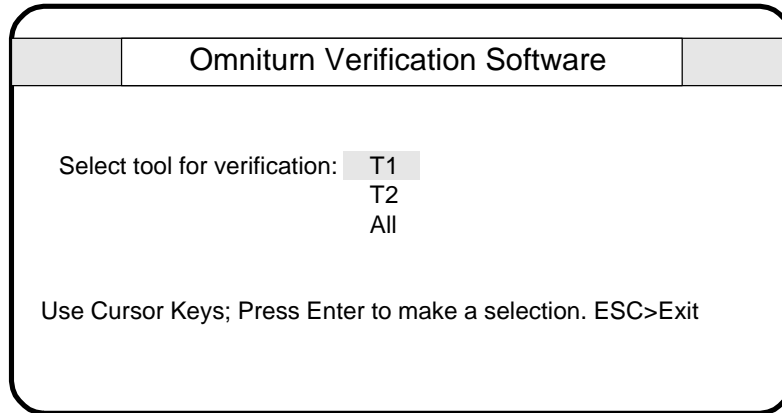
Omniturn Verification Software

Use tool offsets in plots?: No  
Yes

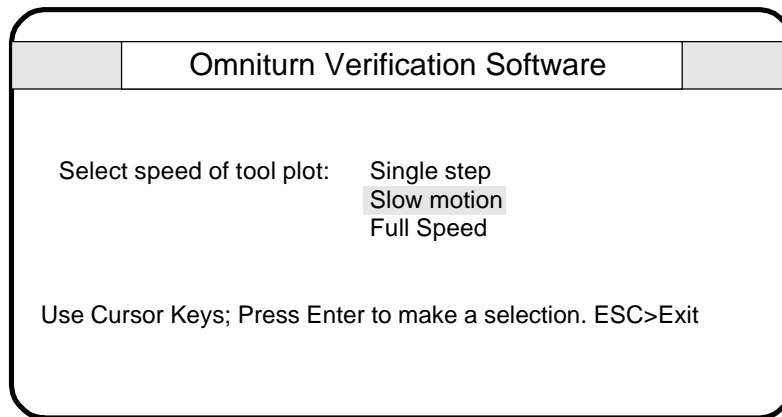
Use Cursor Keys; Press Enter to make a selection. ESC>Exit

# Verification -F4

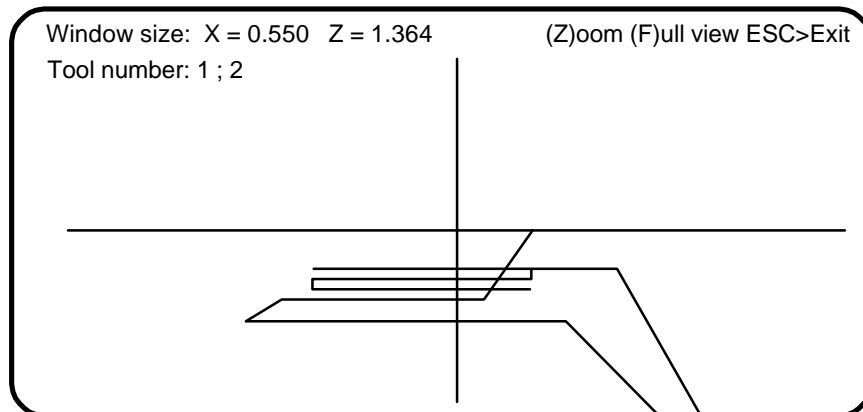
Then select the tool that you want to see. Or all. Use the Up / Down arrow keys again. Then press Enter.



Select the speed of the graphics display. If you choose single step you can advance through the program using the Enter to go to the next move.



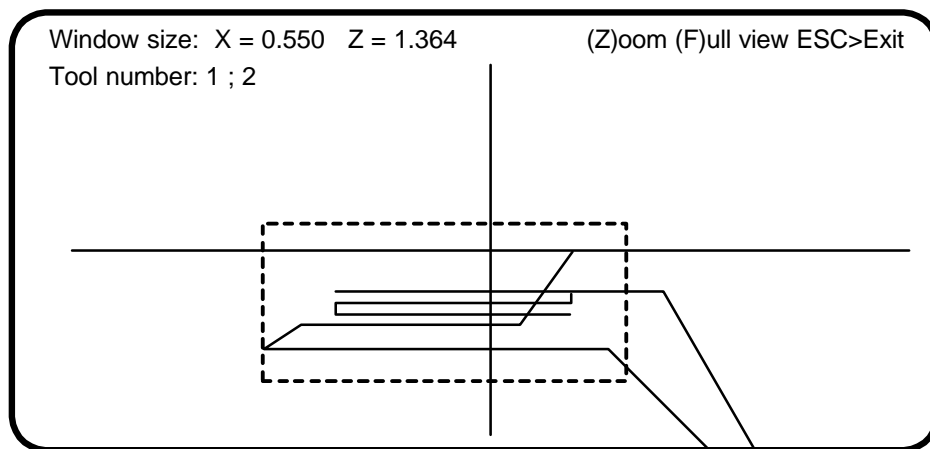
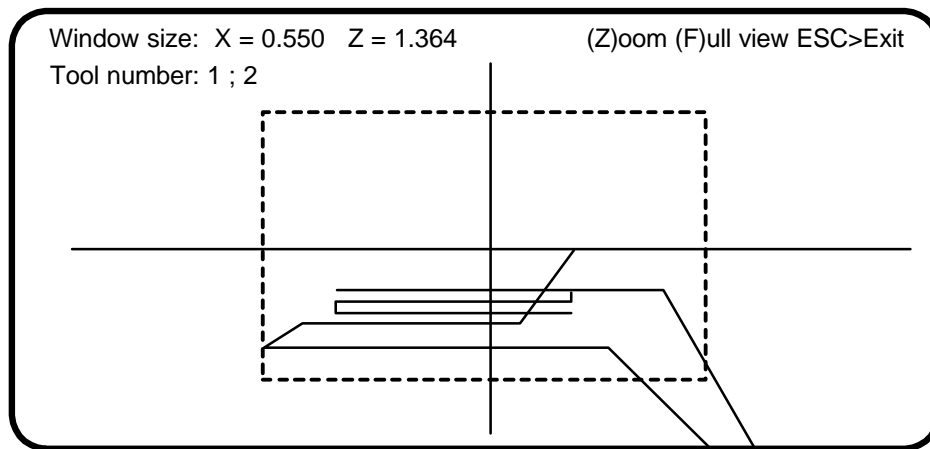
Then the graphics are shown:



**Zooming** You can look closer at a section of the graphics by zooming in. Press the Z key. A window locator will appear. If you want to get much closer than one zoom, press it more than once, the window will get smaller and smaller.

Move the window over the area of the screen you want to zoom. Use the arrow keys to move it around.

## Verification -F4



### Notes about the graphics:

The graphics will show the path created by the G code in the program. If you have tool nose compensations created in Calcaid the graphics will be showing the path of the center of the tool nose radius. So the location of paths from tool to tool may overlap incorrectly since the tool nose radiuses will be different for each tool.

# Automatic Mode

**F4** Directory, list all of the programs on the disk -in use when a program is not active

This function key is not always active. Once you have selected a program to run and there is an active file in memory this function is deactivated. In you have yet to select a current program this key will bring up a list of programs available on the program disk. This will list all of the programs you have stored on the A: disk, (the 3 1/2" floppy). Once you have reviewed the programs available press "Esc" to return to the Automatic mode screen.

**F 5** Different file, used to change the running program to a new one

This function key is used to change the program that is in the current file and allow the operator to enter in a new program name to be run. After F5 is pressed the control will ask "FILE TO BE PROCESSED". Type the new name in and Return. Then press the cycle start and the new program will run. Be sure that the tool offsets are set before running the new program. This key is not always active. If there is no active file in memory F5 does not appear.

**F 6** Search to, used to start the program at another location other than the beginning

This function key will allow you to start the program at a point other than the beginning. This is very useful for running new programs and skipping over sections of program that you do not have to check. It is intended for skipping to a tool change, this is an easily noted beginning section of code. F6 can also be used with programs that have line numbers.

After the F6 command is pushed (while in the Automatic mode) the screen will ask "SEARCH TO?"

POSITION	: X +0.00000 Z +0.00000	FEED 10.0 IPM	
COMMAND	: X +0.00000 Z +0.00000	PERCENT FEED: 100	
SEARCH TO ?			
Jog	Automatic	Single Block	Manual Data Input
F1-F10 FEED 10-100% FILE IN MEMORY: DEMOPR 'O' FOR OPTIONAL STOP '/' FOR BLOCK DELETE 'C' FOR CYCLE REPEAT PRESS 'S' FOR SINGLE BLOCK			
1QUIT 2OFFSET 3EDIT 4DIR 5NEWPROG 6SEARCHTO 7PROG 8DISKOP 9SECCMP 0			

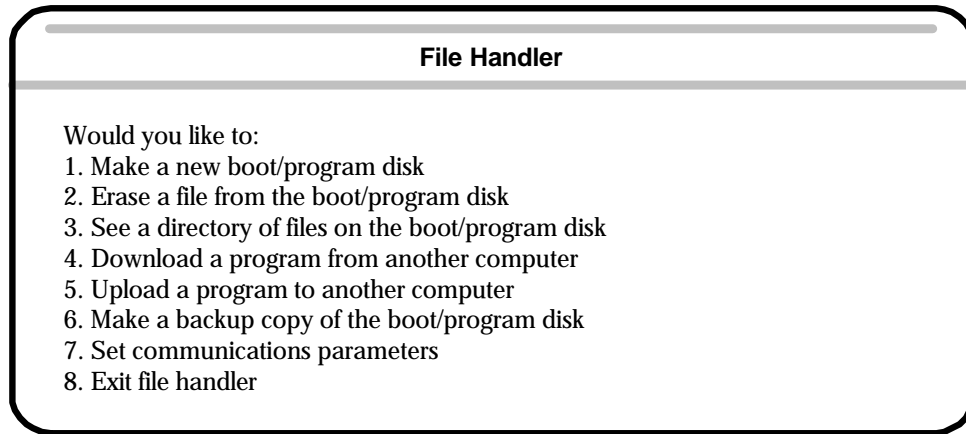
After typing in the text to search to press the Return key. Then press the cycle start button to start the slide.

As an example, if you are want to skip to Tool #2 type in T2. The control will skip the code before this line and start the program with the T2 command. If you are using an OmniTurn with spindle control be sure that after your tool changes you have a spindle on (M03 or M04) and an S command. The F6 - Search to does not read the previous commands and you will have to be sure that the spindle is running. If you use line numbers in your program, it is possible to skip to these instead of the tool changes.

# Automatic Mode

- F7** Programming system, Calls up the Calcaid programming system See the section on Calcaid. Chapters 9 and 10 in this manual.
- F 8** Diskop, Calls up a list of Disk operations

The F8 key will call up the following screen:



By entering a number you will be prompted through the requested task. Listed below is a description of each option.

1. This will help make a new boot/program disk, the larger of the two disks. It stores all of your programs.
2. This will prompt you through erasing programs that you no longer need.
3. This will list all of your programs on the larger disk.
4. This will prompt you through downloading programs from another computer. This is used if you write programs on another computer and want to transfer it to the OmniTum. You will have to run a cable between the RS-232 ports on both computers. (See Section 7 for cable configuration) The communications parameters are set by option 7 on this screen.
5. This will send programs from the OmniTum to another computer via the RS-232 port. (See Section 7 for cable configuration)
6. This will copy the boot/program disk completely. All of your programs will be duplicated. Take this disk away from the machine and store it in a safe place that is dry, dark (no direct sunlight), cool (room temp.), and no magnets near by. This should be done after you have written and tested a program. If there is a problem with the disk in the OmniTum you can replace it with your backup and continue uninterrupted operation.
7. This will display the communications parameters and allow you to change them. The communications parameter options are:  
Baud rate (2400, 1200, 300)  
Parity (o,e,n)  
Data bits (7,8)  
Stop bits (1,2)

# Automatic Mode

8. This will exit the file handling screen and bring you back to the Automatic mode screen.

## F 9 Secondary tool offset screen, used to modify secondary tool offsets

This function will call up the secondary offset table. There are 32 offsets available and 32 tool nose radius compensation offsets. Please refer to the section on secondary offsets for their use. Notice that this differs from the offset table screen in that almost all of the offset values are set to 0.00000. Secondary offsets are corrected like offsets.

1	X: +0.00000 Z: +0.00000 R: 0.00000	17	X: +0.00000 Z: +0.00000 R: 0.00000
2	X: +0.00000 Z: +0.00000 R: 0.00000	18	X: +0.00000 Z: +0.00000 R: 0.00000
3	X: +0.00000 Z: +0.00000 R: 0.00000	19	X: +0.00000 Z: +0.00000 R: 0.00000
4	X: +0.00000 Z: +0.00000 R: 0.00000	20	X: +0.00000 Z: +0.00000 R: 0.00000
5	X: +0.00000 Z: +0.00000 R: 0.00000	21	X: +0.00000 Z: +0.00000 R: 0.00000
6	X: +0.00000 Z: +0.00000 R: 0.00000	22	X: +0.00000 Z: +0.00000 R: 0.00000
7	X: +0.00000 Z: +0.00000 R: 0.00000	23	X: +0.00000 Z: +0.00000 R: 0.00000
8	X: +0.00000 Z: +0.00000 R: 0.00000	24	X: +0.00000 Z: +0.00000 R: 0.00000
9	X: +0.00000 Z: +0.00000 R: 0.00000	25	X: +0.00000 Z: +0.00000 R: 0.00000
10	X: +0.00000 Z: +0.00000 R: 0.00000	26	X: +0.00000 Z: +0.00000 R: 0.00000
11	X: +0.00000 Z: +0.00000 R: 0.00000	27	X: +0.00000 Z: +0.00000 R: 0.00000
12	X: +0.00000 Z: +0.00000 R: 0.00000	28	X: +0.00000 Z: +0.00000 R: 0.00000
13	X: +0.00000 Z: +0.00000 R: 0.00000	29	X: +0.00000 Z: +0.00000 R: 0.00000
14	X: +0.00000 Z: +0.00000 R: 0.00000	30	X: +0.00000 Z: +0.00000 R: 0.00000
15	X: +0.00000 Z: +0.00000 R: 0.00000	31	X: +0.00000 Z: +0.00000 R: 0.00000
16	X: +0.00000 Z: +0.00000 R: 0.00000	32	X: +0.00000 Z: +0.00000 R: 0.00000

Secondary offset number:  
Press C to clear all offsets:  
Press Esc to exit offset adjustment screen

First: Select a secondary offset number

Next: Use the return key to enter past the X and Z inputs.

Then: Enter the value of tool nose compensation, IE .007 and then press ESC

### Changing a TNR compensation value:

When you put a value in the R offset table it writes over the old value. So if you have a number already in the register that you want to use and it is not the value needed, all you have to do is enter the correct value. As an example if you have a value of .032 in the offset and you want to change it to .008, just enter the new value. You do not have to clear the register first. If you want to correct a value slightly, you must enter in the final value needed. ie: if you have .007 and want to increase it by .001 you must enter .008. **Do not enter -.001**

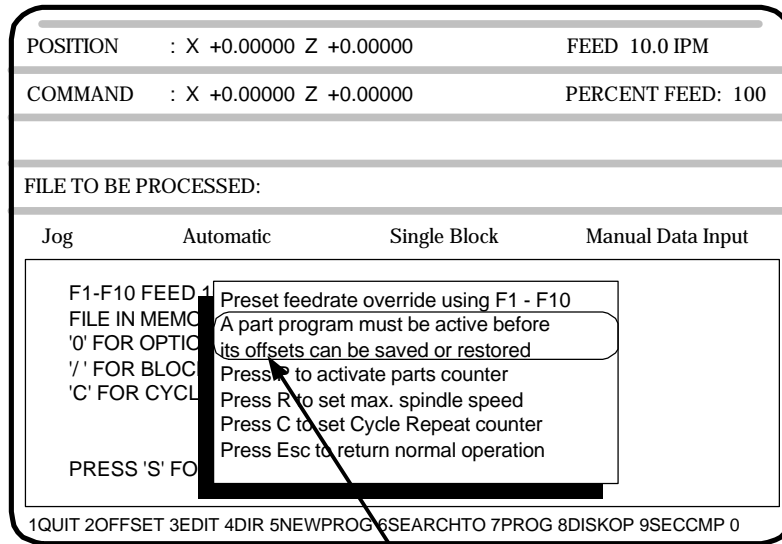
### Clearing secondary offsets to Zero.

It is possible to clear all of the secondary offsets by pressing C when asked for a secondary offset number. This will set the entire table to zeros.

Individual offsets can be set to zero by pressing C when asked to enter a correction amount.

# Automatic Mode

**F 10** Special Function, used to call up a list of special functions.



If a program is active the prompt will be:  
PRESS L TO LOAD OFFSETS  
PRESS S TO SAVE OFFSETS

This screen will allow you to:

- **Save and recall tool offset tables.**

If you are using a tooling system that allows you to remove and replace tooling exactly this function is useful to you. Also if you use the same tool set for a number of different programs. When you save your program it is possible to save the tool offsets as well. The saved offsets are put on the A: disk with the programs.

The screen above is shown if you do not have a part program active. Offsets can not be saved or recalled. If you have a part program active the control will allow you to save and load the offsets from memory:

- L** -press L to load the offsets from memory
- S** -press S to save the offsets to the disk

- **Modify the feedrate :**

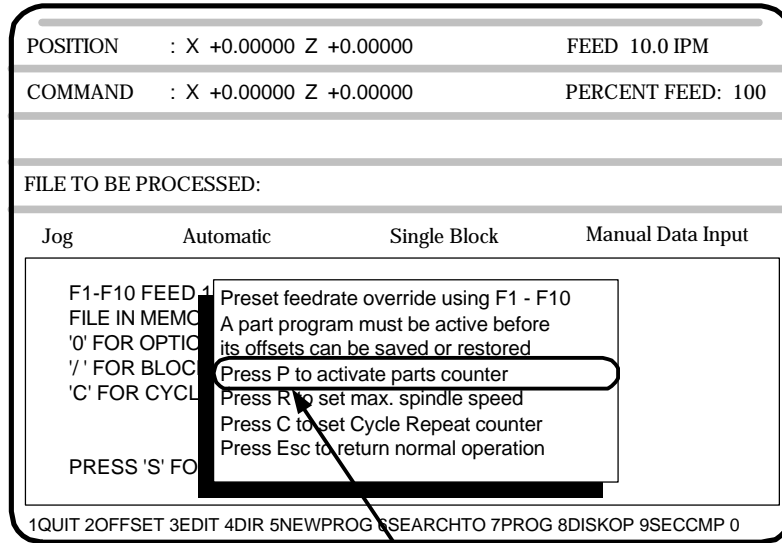
To be used in a program before you start running it. This eliminates the need to race to the function keys after you press the cycle start. When you are running a program for the first time you might want to lower the feedrate to only 20% by pressing F2 so that you can watch the motion of the tool before you cut material.

- **Set the number of:**

Cycles the Automatic mode will run before stopping when you set the cycle repeat to "C". This is good for use with a barfeed. As an example you could set the machine up and tell it you need 20 pieces. The OmniTurn will make the required amount and then stop.

- If you have the infinitely variable spindle speed control this will let you tell the control what spindle speed the machine is set at. That way the control will output the speed requested in your program without having to figure any ratios out. This is covered in greater detail with the documentation on the option.

# Automatic Mode



Press P now, it will turn the counter on in the Automatic mode

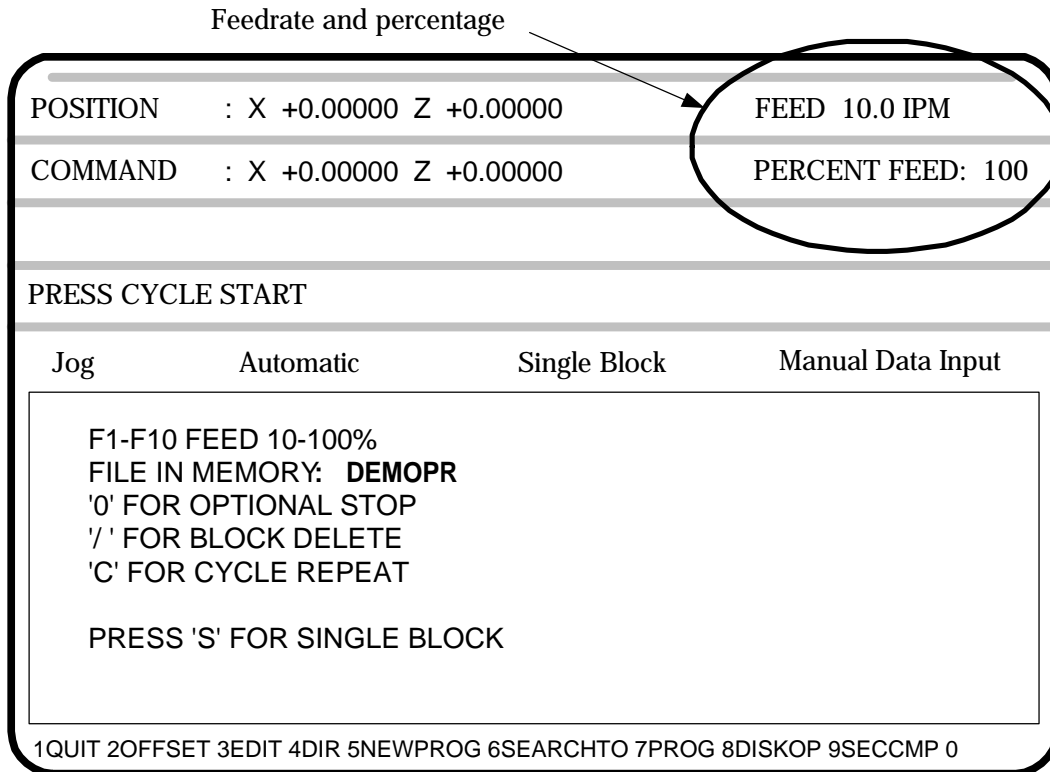
• **Turn on a PARTS COUNTER:** Press P while you have this screen open. There will be no effect here. When you go to the Automatic mode main screen there will be a counter on the screen. The counter will count up each time a M30 or M02 is executed. To clear the counter come back into the F10 screen and press P again.

# Automatic Mode

## Feedrate override

### Function Keys - Automatic Mode - Program in process

When the program is running it is possible to change the feed rates. The function keys will select a percentage of the original feedrate. F1 = 10%, F2 = 20%, ..... F10 = 100%. IE if you push F1 while the program is running the feedrate will drop to 10% of what ever you have set in the program. The feedrate and what percentage of that feedrate is being run is displayed on the automatic mode screen on the upper right corner of the screen.



If you want to change the feedrate before you press cycle start select F10. This will allow you to preload a percentage before the program is started.